

ECLIPSE

Version 2014.1



ECLIPSE
ECLIPSE reservoir simulation software

Technical Description

Schlumberger

Proprietary notice

Copyright (c) 1982 - 2014 Schlumberger. All rights reserved. Reproduction or alteration without prior written permission is prohibited, except as allowed under applicable law.

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

Trademarks & service marks

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

Table of Contents

1	Introduction	1
	ECLIPSE features	2
	ECLIPSE options	11
	Ancillary programs	17
	Additional programs	18
2	Adjoint gradient screening	22
	Keywords	22
	Output	24
	Restrictions	25
	Formulation	25
3	Alkaline model	28
	Alkaline model 1	28
	Using alkaline model 1	31
	Summary of keywords	31
	Example problem	32
4	API tracking	37
	The API tracking facility	37
5	Aquifer modeling facilities	40
	Numerical aquifer	40
	Carter-Tracy aquifer	42
	Fetkovich aquifer	44
	Constant flux aquifer	47
	Constant head aquifers	48
	Multi-component analytic aquifers	48
	Summary of keywords	48
6	Asphaltene option	51
	Asphaltene formation: three-component model	52
	Deposition - entrainment	54
	Asphaltene formation: Pre-2011.1 model	55
	Asphaltene characterization and precipitation	56
	Asphaltene flocculation-dissociation	58
	Asphaltene deposition	59
	<i>Plugging control</i>	60

Asphaltene damage (both deposition models)	60
Oil viscosity	61
Wettability change	62
General comments on viscosity behavior	63
List of keywords	63
Outputs	64
<i>Asphaltene reaction number</i>	65
Example	65
Converting a pre-2011.1 model	66
7 Brine tracking	68
The Brine Model	68
Brine conservation equation	68
Multi-component brine model	69
The PVT properties of brine	72
Low salinity option	73
<i>Low salinity option with end-point scaling</i>	75
<i>Low salinity and surfactant options</i>	79
Setting the initial conditions	88
Wells, rivers and aquifers	89
Tracer related options	89
The polymer flood option	89
The surfactant flood option	90
<i>Multi-component brine and surfactant</i>	90
Solving the conservation equations	90
Summary of keywords	91
Examples	93
8 CO₂ injection: geological storage, EOR and ECBM	97
Geological storage	97
CO ₂ STORE	98
Using CO ₂ STORE	98
CO ₂ SOL	99
Using CO ₂ SOL	100
CO ₂ SOL/H ₂ SSOL/GASSOL	101
Using GASSOL	101
GASWAT	102
Using GASWAT for CO ₂ storage issues	103
Boundary conditions	103
Note on conversions for salinity units	104
9 CO₂ storage	108
The CO ₂ STORE option	108

Using the CO ₂ storage model	112
Thermal CO₂STORE	113
10 CO₂/H₂S solution in the aqueous phase	115
Water component properties	116
11 Chemical reactions	118
Formulation	118
Reaction rate	119
Reactions involving solid components	120
Equilibrium deviation reaction rates	121
Tracer dependent reaction rates	122
Component conservation equations	123
Energy conservation equation	123
Non-uniqueness	124
Reaction rate units	124
Stability of thermal runs	124
Output	124
Chemical reaction keywords	125
Examples	126
12 Coal bed methane model	129
Adsorption/diffusion models in ECLIPSE 100	131
Adsorption models in ECLIPSE 300	132
Time dependent diffusion in ECLIPSE 300	133
Instant sorption in ECLIPSE 300	135
Rock compaction	135
Modeling compositional effects with ECLIPSE 100	136
Using the ECLIPSE 100 Coal Bed Methane Model	138
Single gas mode	138
Two gas mode - compositional effects	139
<i>Solvent properties and adsorption data</i>	139
<i>Equilibration</i>	140
Solvent injection	140
Summary data	140
Summary of keywords (ECLIPSE 100)	141
Examples ECLIPSE 100	143
Basic test ECLIPSE 100	143
CO ₂ injection example ECLIPSE 100	146
Summary of keywords (ECLIPSE 300)	150
Using the ECLIPSE 300 coal bed methane model	152
Grid data	152
Properties data	153

Equilibration	154
<i>Time-dependent sorption</i>	154
Output and restart	155
Summary keywords	155
Examples	155
13 Compatibility between ECLIPSE 100 specialist extensions	156
Compatibility information	156
14 Convergence reports	160
ECLIPSE 100 Report	160
Output of cells with convergence problems	163
ECLIPSE 300 Report	164
Solution change target (SCT)	165
Throughput target (TPT)	165
Time truncation error (TTE)	165
Minimum step codes	166
Maximum decrease factors	166
Output of cells with convergence problems	167
Example	167
15 Diffusion	168
Diffusion coefficients: ECLIPSE 100	168
Diffusion coefficients: ECLIPSE 300	172
Relationship between the two diffusion models	173
Relationship between the diffusion coefficients	173
Component flow	174
Thermal diffusion and hydrostatic equilibrium calculations	175
The cross phase diffusion option (ECLIPSE 300)	175
The cross phase diffusion option (ECLIPSE 100)	176
Diffusivity calculations	177
Block center diffusivity calculations	177
Corner point diffusivity calculations	179
Radial diffusivity calculations	179
Matrix-fracture diffusivity calculations	181
Using the molecular diffusion option	181
Summary of keywords	182
16 Dual porosity	184
Transmissibility calculations	185
Block to block transmissibility calculations	186
Recovery mechanisms	188
Standard gravity drainage model (GRAVDR)	189
Alternative gravity drainage model (GRAVDRM)	192

Vertical discrete matrix gravity drainage model	193
Diffusion	194
Viscous displacement (ECLIPSE 100)	194
Facilities specific to dual porosity runs	196
Restrictions on dual porosity runs	198
Keyword summary	199
Transfer functions	200
Modifying the recovery versus time response (ECLIPSE 100)	201
Scaling the fracture relative permeability (ECLIPSE 100)	202
Solution of the linear equations	202
Calculation of fractional volumes for gravity drainage	204
Variation of the sigma factor with drainage process	206
Rock volume and cell activation in dual porosity thermal calculations	206
Integrated capillary pressure option	207
Discretized matrix model (ECLIPSE 100)	209
Matrix sub-grid	210
Initialization, solution method and efficiency	212
Reporting the matrix solution	213
Restrictions	213
Discretized matrix model (ECLIPSE 300)	213
Single medium conductive fractures (ECLIPSE 300)	214
The single medium approach	214
Using the conductive fractures model	215
Compatibility	216
17 Environmental tracers	217
The environmental tracer model	217
Treatment of adsorption	218
Treatment of decay	218
Treatment of diffusion	219
Partitioned tracers	219
Using the environmental tracer option	221
Summary of keywords	223
Example problem	224
18 Equations of state	231
Two-parameter equations of state	231
Flash calculations	233
Viscosity evaluation	234
The Lorentz-Bray-Clark method	234
The modified Lorentz-Bray-Clark method	235

The Pedersen method	236
Three parameter equations of state	236
Use of the equation of state for hydrocarbon mixtures	237
Calculation process	238
Calculation of phase states	239
Fast flash option	240
19 Equations of state regions	241
Reservoir equation of state	241
Surface equation of state	242
20 File handling in ECLIPSE	244
File internal format	245
GRAF files	246
21 Flux boundary conditions	248
Using flux boundary conditions	248
Single flux region	248
Multiple flux regions	250
Treatment of wells	250
Drainage regions	251
Aquifers	251
Restart runs	251
Pressure boundary conditions (ECLIPSE 100)	252
No-flow boundary conditions (ECLIPSE 100)	252
Local grid refinement and coarsening	253
Reading ECLIPSE 100 flux files into ECLIPSE 300	253
Thermal option	253
Current limitations	254
Summary of keywords	255
22 Foam model	257
ECLIPSE 100 foam model	257
Gas mobility reduction	259
Using the ECLIPSE 100 foam model	263
The ECLIPSE 300 foam model	265
Summary of keywords	267
Example data set for ECLIPSE 100	269
Example data set for ECLIPSE 300	275
23 Formulation of the equations	283
Description of equations	283
Variable sets	284
ECLIPSE 100	284

ECLIPSE 300	286
The fully implicit method	287
The IMPES and AIM methods	287
The CFLLIMIT option for AIM and IMPES runs	288
Instability in IMPES	289
Convergence criteria for residual equations	290
ECLIPSE 100	290
ECLIPSE 300	291
Flows	292
ECLIPSE 100	292
ECLIPSE 300	293
<i>Transport coefficients</i>	295
<i>Transport coefficients for drainage and imbibition</i>	295
Reservoir densities	295
Newton iteration of the non-linear residual	296
24 Gas field operations model	298
Description of the facilities	299
Seasonality profile generation	299
Swing factor and DCQ adjustment	300
<i>Interpretations of the swing requirement</i>	301
<i>Calculating the DCQ value</i>	303
Control and output of sales gas rate	303
Estimation of delivery capacity	304
Automatic compressors	305
<i>Simple and extended compressor options with GASFCOMP</i>	306
Specifying production rates for the second pass using GASFTARG and GASFDECR	307
Applying separate gas supply contracts to individual groups	307
Defining multiple contract groups	307
Simulating with multiple contract groups	308
Automatic compressors with multiple contract groups	309
Using the gas field operation model	309
Specifying the wells	310
Setting up the network	311
Group control instructions	311
Advancing the simulation	312
Examining the results	314
Restrictions	314
Summary of keywords	315
25 Gas lift optimization	317
Optimizing gas lift to individual wells	317

Optimizing gas lift within a group	319
Optimizing gas lift within a network	320
Treatment of wells that can only flow under gas lift	321
Using the gas lift optimization facility	323
Summary of keywords	326
26 Gas plants and NGLS	327
The recovery plant option	328
27 GASSOL solution in the aqueous phase	330
Aqueous phase properties	331
28 GASWAT option	333
Equation of state modifications	334
The conventional Peng-Robinson equation of state	334
Modifications for the vapor-aqueous system	335
Calorific values	337
Salinity	337
Relative permeability tables	338
Initialization	338
Summary items	338
29 Gas calorific value control	339
Adjusting guide rates to control mean calorific value	340
Action when calorific value target cannot be met	342
Controlling the energy production rate	343
Using the facility	345
Defining the calorific value	345
Controlling the mean calorific value	346
Calculating guide rates on the basis of calorific value	346
Controlling the energy production rate	346
Economic limits	347
Use with the gas field operations model	347
Satellite groups	347
Energy injection	347
Output	348
Restrictions	348
Summary of keywords	348
30 Geomechanics	349
Geomechanics external to ECLIPSE	349
The VISAGE workflow in ECLIPSE	350
Summary of keywords	351
Geomechanics internally within ECLIPSE Compositional	351

Elastic stress equations	352
<i>Gridding</i>	353
<i>Fluid component conservation, rock volume conservation and volume balance equations</i>	353
<i>Interaction between the fluid and the rock</i>	353
<i>Variables and solution of the equations</i>	354
<i>Boundary conditions and initialization</i>	354
<i>Plasticity</i>	354
<i>User Interface</i>	355
<i>Initial in-situ stress</i>	355
Using the coupled geomechanics model	356
Summary of keywords	357
31 Gi pseudo-compositional model	361
<i>Using the Gi pseudo-compositional model</i>	364
<i>Summary of keywords</i>	364
<i>Example problem</i>	365
32 Gradient option	371
<i>Example</i>	374
<i>Data file overview</i>	375
RUNSPEC section overview	375
Summary section overview (ECLIPSE 100 only)	376
<i>Gradient calculation performance keywords</i>	376
<i>Special extensions</i>	377
<i>Output from the Gradient option</i>	380
<i>Gradient option formulation</i>	388
Obtaining gradients in ECLIPSE	388
Regression in SimOpt using ECLIPSE gradients	389
Explicit gradients (ECLIPSE 100 only)	390
<i>Instantaneous gradients</i>	390
<i>Summary of keywords</i>	391
Parameter specification keywords	392
Parameter modifier keywords	393
SUMMARY keywords	395
Miscellaneous keywords	395
<i>Example problem (ECLIPSE 100 only)</i>	395
33 Group production rules	403
<i>Drilling new wells</i>	403
<i>Opening new connections (re-perforating)</i>	404
<i>Reducing the THP limit</i>	405
<i>Changing VFP table numbers (retubing)</i>	406

Changing well ALQ values (applying lift)	406
Activating compressors	407
The effect of closing wells during drilling and workovers	408
Using production rules with the network option	408
34 Hysteresis	410
Using the hysteresis option	410
Relative permeability hysteresis in the non-wetting phase	411
Carlson's method for relative permeability hysteresis	413
Killough's method for relative permeability hysteresis	413
Jargon's method for relative permeability hysteresis	414
<i>Differences in hysteresis treatment in three-phase cases</i>	414
Consistency requirements	415
Relative permeability hysteresis in the wetting phase	415
Modeling oil wet systems	419
Capillary pressure hysteresis	419
Water capillary pressure	419
Gas capillary pressure	422
35 Hysteresis in WAG floods	423
Using the WAG hysteresis option	423
Conventional and WAG hysteresis	424
WAG hysteresis saturation regions	426
Non-wetting phase model (Gas)	429
Two-phase model	429
Dynamic trapped saturation	431
Three-phase model	431
Modification to the residual oil	433
Wetting phase model (Water)	434
36 Independent reservoir regions	437
Using the option	437
Merging two or more corner point grids	437
Examples	438
37 Initializing the study	440
Equilibration	440
Basic hydrostatic equilibration	440
Equilibration regions	441
Contact and datum depths	441
Density calculations	442
<i>Black oil composition with respect to depth</i>	442
<i>Initial composition with respect to depth</i>	442

Calculating the initial conditions	444
<i>Consistency and overlapping transition zones</i>	445
<i>Heavy oil</i>	445
Accurate fluids in place calculation	445
The threshold pressure option	446
The quiescence option	446
Reservoir temperature	447
Phase pressures in ECLIPSE 300	448
Equilibration keywords	448
Enumeration	449
Specifying the initial state of a run in ECLIPSE 300	450
Enumeration keywords	451
Mixed hydrostatic equilibration	452
Single phase states in EoS compositional models	453
38 Licenses	454
License Overview	454
Licenses used by specific simulators	455
Which keywords invoke which features?	455
Special Cases	457
Open ECLIPSE	457
Parallel	457
Block Parallel	457
Multiple Realization	458
Lagrangian Tracer Solver	458
Listing of available licenses	458
License compatibility	459
2014.1 license retirements	459
39 Local grid refinement and coarsening	460
Local grid refinement	460
Radial refinement in a single column	460
Radial refinement in a box of four columns (ECLIPSE 100)	462
Placing wells in local grids	463
Cartesian refinements	464
Defining local grid block sizes	465
Defining local grid rock properties	466
Amalgamating Cartesian local grids (ECLIPSE 100)	467
Editing local grids	469
Grid and solution reports	470
Tuning	470
Timing reports	470
Vertical equilibrium (ECLIPSE 100)	470
Good practice in designing LGRs	471

Dual, multiple and triple porosity	471
Unstructured local grid refinements	471
Region data and end point scaling	472
Switching refinements on and off (ECLIPSE 100)	472
Summary files	473
Transmissibility calculations	474
Timesteps	475
Restrictions on the use of local grids	476
Treatment of wells in local grids	477
Parallel load balancing with LGRs	479
Gravity and grid effects	479
Geometry and grid data in LGRs	479
Transmissibility data	479
Pore volumes	480
Radial refinements	480
Alphabetical list of GRID section keywords allowed within a local grid	482
Local grid coarsening	484
Implementation	485
Upscaling	486
Automatic refinement (ECLIPSE 100)	488
Treatment of wells in automatically refined grids	489
Current restrictions	490
Summary of keywords	490
Example LGR problem	493
40 Miscible flood modeling	499
Model formulation	499
Relative permeability model	500
Effect of water saturation	500
Viscosity model	501
Density model	502
Todd and Longstaff mixing parameter	503
The control of miscibility with pressure	503
Control of numerical diffusion in miscible flow simulation	504
Using the ECLIPSE miscible flood option	506
Summary of keywords	507
41 Mixed water relative permeability modeling	509
42 Multiphase flash	511
Using multiphase	511
Multiphase keywords	511

43 Multi porosity	513
Using the multi porosity option	513
Matrix-matrix transmissibility and diffusivity	515
Matrix discretization (NMATOPTS)	515
<i>Coal in multi porosity</i>	517
Triple porosity	517
<i>Transmissibilities</i>	518
PROPS, REGION and SOLUTION sections	518
Output	519
Examples	519
Limitations	519
44 Multisegment wells	520
The multisegment well model	521
Segment structure	521
Inflow performance	523
The pressure loss calculation	525
The top segment	526
External sources and sinks	526
Secondary wells	526
Looped flow paths	527
Pressure drops from VFP tables	530
The drift flux slip model	532
Model formulation for gas-liquid slip	533
The profile parameter	534
The drift velocity	535
Three-phase flow and oil-water slip	537
Friction	539
Acceleration pressure loss	539
Downhole separators (ECLIPSE 100)	540
Thermal multisegment well model	544
Conductive heat transfer	544
<i>Heat transfer along the well</i>	545
<i>Heat transfer between the well and the reservoir or to an external temperature</i>	545
<i>Heat transfer from the tubing to the casing</i>	547
Using the multisegment well model	548
Defining the flow model	549
Modifying pressure drops and segment properties	550
Workovers	550
Heat loss	550
Output	551
Performance	552
Restrictions	553

Converting friction wells to multisegment wells	554
Summary of keywords	555
45 Network option	558
Using the standard network option	562
Keywords required	565
The extended network model	567
Automatic compressors	570
Automatic chokes	570
Well and group efficiency factors	572
Using the extended network model	573
The network model in ECLIPSE 300	575
Summary of keywords	577
46 Nine-point schemes	579
The nine-point scheme based on diagonal NNC transmissibilities	579
47 Non-Darcy flow	582
Forchheimer models	584
Velocity-dependent relative permeabilities	586
Capillary number model	587
Alternative model for capillary number effects	589
48 Non-Newtonian fluid flows	592
Simulation of Non-Newtonian fluid flows	592
49 Optimization	596
Non-linear gradient-based optimization	596
Adjoint gradient calculations	599
Optimizer search direction	601
Line search	602
Reservoir optimization	603
Guidelines on using the reservoir optimization feature	605
Checking your modified model	608
Tracking the progress and results of the optimizer	609
Summary of keywords (ECLIPSE 300)	611
Example	611
50 Parallel option	613
Activating a parallel run	613
Partitioning	614
ECLIPSE 100	614
<i>Options to improve performance</i>	614
<i>Efficiency and performance estimation</i>	615

ECLIPSE 300	616
Running LGRs in parallel (ECLIPSE 100)	618
Performance bottlenecks	618
Memory scaling	619
Output (ECLIPSE 100)	619
Summary of keywords	620
51 Petro-elastic model	622
Assumptions	622
The petro-elastic model	623
Level 1: Impedances and velocities	623
Level 2: Effective bulk/shear modulus and density	624
Level 3: Rock properties and fluid properties	625
<i>Effective porosity</i>	625
<i>Effective fluid density</i>	626
<i>Effective bulk modulus of the fluid</i>	626
<i>Effective bulk modulus of the minerals</i>	627
<i>Effective shear modulus of the minerals</i>	627
<i>Effective bulk modulus of the frame</i>	627
<i>Effective shear modulus of the frame</i>	627
Level 4: Individual fluid phase properties	628
<i>Bulk modulus of oil</i>	628
<i>Bulk modulus of water</i>	628
<i>Bulk modulus of gas</i>	629
Implementation of the petro-elastic model	630
Summary of keywords	631
52 Polymer flood model	633
The ECLIPSE 100 polymer flood simulation model	634
Treatment of fluid viscosities	636
Treatment of polymer adsorption	637
Treatment of permeability reductions and dead pore volume	637
Treatment of the non-Newtonian rheology	638
<i>Model 1 description</i>	639
<i>Model 3 description</i>	641
Temperature effects	644
Temperature-sensitive popping polymer microgels	644
The polymer flood model and wells	645
Using the polymer flood model	646
Restrictions	647
The ECLIPSE 300 polymer flood model	648
Treatment of fluid viscosities	648
Treatment of polymer adsorption	648
Restrictions	649

Summary of ECLIPSE 100 keywords	649
Summary of ECLIPSE 300 keywords	652
ECLIPSE 100 examples	654
Example 1	654
Example 2	658
ECLIPSE 300 examples	661
Example 1	661
Example 2	667
Example 3	673
53 Potential calculations	680
Phase potentials	680
Well potentials	682
54 Recovery mechanisms	684
Solution gas drive	685
Water drive	686
Summary output	687
Details of the recovery calculation	687
55 Reservoir coupling	689
Coupling and synchronization	691
Group control	692
Flowlines and surface networks	693
Reservoir coupling files	694
Current restrictions	695
Using the reservoir coupling facility	696
The master reservoir	696
The slave reservoirs	698
Reading a reservoir coupling file in the master reservoir	700
Driving a slave reservoir with a reservoir coupling file	700
Using the gas field operations model in coupled reservoirs	701
Sharing licenses with the slaves	702
Reservoir coupling file structure	703
Summary of keywords	707
56 Restarts	708
Fast restarts from a SAVE file	708
Flexible restarts from a data file	709
Requesting RESTART files	710
SCHEDULE section data not stored on the RESTART file	711
Notes	712

57 Rivers	713
The river model	714
The Saint-Venant equations	714
Coupling of river and groundwater flow	715
<i>Model one</i>	715
<i>Model 2</i>	717
Using the river option	718
Summary of keywords	718
Example: A simple river	721
58 Rock compaction	724
Tabulated compaction curves	724
Output	727
Water induced compaction (ECLIPSE 100)	728
Palmer-Mansoori rock model	729
Compositional coal swelling/shrinkage model (ECLIPSE 300)	730
59 Saturation functions	732
Saturation table keywords	733
Saturation table lookup	735
Tolerance for critical saturations	736
Water saturation properties	736
Gas saturation properties	737
Three phase gas relative permeability models	738
Oil saturation properties	739
Three phase oil relative permeability models	741
<i>Default formula</i>	741
<i>Stone's first model (modified)</i>	742
<i>Stone's first model exponent</i>	746
<i>Stone's second model (modified)</i>	747
Three phase relative permeability models	747
Baker method	747
IKU method	748
ODD3P method	750
<i>ODD3P model definition</i>	753
<i>ODD3P saturation functions</i>	753
<i>End-point scaling</i>	756
<i>Capillary pressure calculation control</i>	756
<i>End-point scaling calculation control</i>	759
<i>Saturation direction indicators</i>	759
<i>Reporting facilities</i>	760
Table end points	760

Consistency requirements	761
Near critical oil and gas relative permeabilities	762
60 Saturation table scaling	766
Scaling of capillary pressure functions	768
Scaling of relative permeability functions	768
Miscellaneous points	772
Combined grid-block and temperature-dependent table end-points	774
Output of scaled end-points	775
The initial fluid mobility correction	776
End-point variations through a transition zone	778
Leverett J-function	779
Matching initial water distribution	780
Consistency requirements	781
Example of end-point scaling	782
With multiple tables	782
61 Shale gas	784
Simulating shale gas reservoirs	784
Time-dependent sorption	784
Instant sorption	786
Coal and pore volumes	786
Examples	787
62 Solid phase	788
Summary of keywords	788
Solid properties	790
Solid phase properties	790
Solid saturations	791
Adsorption model of deposition and erosion	792
Adsorption of dissolved water components	794
Permeability reduction caused by plugging	795
Solid wettability effects	795
Transport of suspended solids	796
Initial solids	796
Chemical reactions	797
Solid production	798
63 Solution of the linear equations	799
Material balance	799
Nested factorization preconditioning	800
Initialization for nested factorization preconditioner	802
Solution procedure for nested factorization	802

Orthomin	803
Convergence criteria	804
Treatment of wells	805
Nested factorization for LGRs	806
The ECLIPSE 300 WARP linear solver	807
WARP sub-grids	808
WARP solution procedure for sub-grids	810
Generalized minimum residual method (GMRES)	811
The Constrained Pressure Residual (CPR) Preconditioner	812
64 Solvent model	815
Model formulation	817
Relative permeability model: miscible case	818
Transition between miscible and immiscible cases	818
Pressure effect on relative permeability and capillary pressure miscibility	820
Effect of water saturation	820
Viscosity model	821
Density model	822
Pressure effect on viscosity and density miscibility	823
Choice of the mixing parameter	824
Using the solvent model	824
Control of numerical diffusion in miscible flow simulation	825
Summary of keywords	827
Example	828
65 Surface tension effects	834
Miscibility	834
Relative permeability interpolation parameter	835
Capillary pressure interpolation parameter	835
Evaluation of surface tensions	836
Use of the MISCTNUM option	837
66 Surfactant model	838
The simulation model	838
Relative permeability model	839
Capillary pressure	841
Water PVT properties (ECLIPSE 100)	842
Treatment of adsorption (ECLIPSE 100)	842
Modeling changes in wettability due to surfactant adsorption (ECLIPSE 100)	843
Saturation function interpolation (ECLIPSE 300)	851
Using the ECLIPSE 100 surfactant model	852
Using the ECLIPSE 300 surfactant model	853
Summary of ECLIPSE 100 keywords	855

Summary of ECLIPSE 300 keywords	857
ECLIPSE 100 example	859
Example 1	859
Example 2	863
ECLIPSE 300 example	869
Example 1	869
Example 2	874
67 Temperature option	881
ECLIPSE 100 and 300 differences	881
Thermal properties	882
Viscosity modification	882
Thermal conduction	883
Heat losses/gains	883
Equilibrium state	883
Setting the initial conditions	884
Wells	884
Solution output	884
Summary output	884
Restrictions	885
Convergence issues	885
Example data set	885
68 Tensor permeability	886
Discretization	887
Orthogonality error	889
Restrictions	890
69 Thermal option	891
Formulation	892
Primary variables	892
Conservation equations	892
Thermodynamic equilibrium condition	893
Total mole fractions	894
Thermal flash	894
Fluid volume	895
Inter-block flows	895
Well flows	897
Heat conduction	897
Solid phase	897
Solution method	898
Thermal properties	899

Rocks	899
<i>Rock heat capacity</i>	900
<i>Thermal conductivity</i>	900
<i>Thermal transmissibility</i>	901
<i>Definition of multiplier for Heat Conduction</i>	901
<i>Dual porosity or dual permeability</i>	902
<i>Hysteretic rock compaction</i>	902
Relative permeability curves	902
Live oil	902
<i>K-values</i>	903
<i>Density</i>	904
<i>Enthalpy</i>	908
<i>Viscosity</i>	915
<i>Example</i>	917
Dead oil	919
<i>Density</i>	919
<i>Viscosity</i>	919
<i>Example</i>	920
Water and steam	920
<i>K-values</i>	920
<i>Water density</i>	921
<i>Steam density</i>	922
<i>Energies and enthalpies</i>	923
<i>Water viscosity</i>	924
<i>Steam viscosity</i>	924
Combustion	924
Foamy oil	924
Thermal features	926
Initialization	927
<i>Explicit initialization</i>	927
<i>Equilibration</i>	928
<i>Mixed initialization</i>	929
Aquifers	929
Heaters	929
Well inflow	930
Water, steam and gas injection	930
Steam trap	932
Steam production rate	932
Dual porosity	932
Multisegment wells	933
Temperature dependent well cutback	933
Temperature range checking	933
Thermal output	933
Convergence criteria	933
Timestepping criteria	934

Tuning thermal models for performance	937
Current limitations	937
Converting a blackoil data set to thermal	938
Live oil fluid properties	939
Volatility types	939
K-values for live components	939
Oil phase density	940
Gas phase density	940
Oil phase viscosity	941
Gas phase viscosity	941
Specific heats	941
Other data	941
Water properties	941
Initial conditions	941
Well properties	942
Example	942
Summary of keywords	942
Examples	950
70 Timing reports	953
ECLIPSE 100 timing Reports	953
ECLIPSE 300 timing reports	955
71 Total compressibility checks	957
72 Transmissibility calculations	960
Cell ordering and geometry	960
Transmissibility calculations	961
Cartesian transmissibility calculations	961
Block center transmissibility calculations	961
OLDTRANR transmissibility calculations	963
Corner point transmissibility calculations	964
HALFTRAN transmissibility calculations (ECLIPSE 100)	965
Radial transmissibility calculations	966
R-transmissibility	966
Azimuthal transmissibility	967
Vertical transmissibility	968
Variations on the transmissibility calculation (ECLIPSE 100)	969
73 Tracer tracking	972
Control of numerical diffusion (ECLIPSE 100)	973
Lagrangian tracer solver (ECLIPSE 300)	974
Using the ECLIPSE tracer tracking facility	975
Tracking original fluids	976

Usage	977
Fluid in place reports	978
Tracer structure	978
The normalization option	979
Alternative production reporting	979
Following the tracers	980
SUMMARY section keywords	980
Example	981
Delumping and lumping	982
Diffusion and dispersion	983
Treating passive tracers as non-linear	983
74 Units	984
Conversion factors	986
75 Velocity calculations	989
Velocity concepts	989
Velocity definitions	990
ECLIPSE 100 velocity calculations	991
Capillary number	991
Asphaltene deposition and entrainment	994
76 Vertical equilibrium	995
Using the VE option	996
The cell saturation distribution	996
VE relative permeabilities and capillary pressures	1000
Residual flow model	1001
Miscellaneous points	1003
Compressed vertical equilibrium option	1004
Using the option	1004
Breaking the compressed VE cells	1005
Tabular description of the cell	1005
Use with local grid refinement	1006
77 Water components	1007
Water component properties	1007
Converting PVTW data to water component data	1009
Converting PVTWSALT data to water component data	1010
Effective salinity calculations	1010
Limitations	1012
Summary of keywords	1013
Examples	1015

78 Wellbore friction option	1016
Calculating the frictional pressure loss	1016
Using the wellbore friction option	1018
Multilateral wells	1019
Workovers	1023
Graphical presentation of friction effects in horizontal wells	1023
Multilateral friction well plots in GRAF	1026
Production flow rates from upstream connections	1027
Summary of keywords	1028
Example	1028
79 Well inflow performance	1034
Inflow performance relationship in ECLIPSE 100	1034
Inflow performance relationship in ECLIPSE 300	1035
The connection transmissibility factor	1035
The flow-dependent skin factor	1038
The phase mobilities	1040
Special inflow equations	1041
Russell-Goodrich	1041
Dry gas pseudo-pressure	1041
Generalized pseudo-pressure in ECLIPSE 100	1041
Generalized pseudo-pressure in ECLIPSE 300	1042
<i>Velocity dependent relative permeability effects</i>	1045
Injecting connections	1046
High mobility injection wells	1047
The wellbore head term	1048
The productivity index	1048
80 Well modeling facilities	1051
Well completions	1051
Well controls and limits	1053
Economic limits	1054
Automatic cutback of flow rate	1056
Efficiency factors	1056
Automatic opening of wells and well connections	1057
Group and field control facilities	1058
Group production control options	1058
Economic limits and cutback limits	1063
Group injection control options	1063
Sales gas production control	1067
Pressure maintenance	1069
Calculating flow targets for wells under group control	1069

Workover and drilling rig constraints	1070
Vertical flow performance	1070
Well name and well list template matching	1072
A RUNSPEC section: pre-95A fixed record structure	1074
Detailed description of RUNSPEC records	1075
B ECLIPSE Pre-2002A integer controls	1085
RPTPROPS	1086
Examples	1088
RPTGRID	1089
Examples	1093
RPTRST	1094
Examples	1097
RPTSCHED	1098
Examples for ECLIPSE 100	1106
Example for ECLIPSE 300	1106
RPTSOL	1107
Examples	1113
C Retired keywords	1114
LSCRITL	1115
PSEUDOS	1116
WCONINJ	1117
D Bibliography	1120

1

Introduction

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

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

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

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

About this manual

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

For information on licensing see "[Licenses](#)".

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.

ECLIPSE features

Free format input

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

Phase and Fluid options

ECLIPSE 100

ECLIPSE 100 can be used to simulate one, two or three-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. 71]. The Redlich-Kwong, Soave-Redlich-Kwong, Peng-Robinson and Zundkevitch-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](#)" for further information.

ECLIPSE 300

In [GASWAT](#) mode the solution of hydrocarbon components in the water phase is permitted. See "[GASWAT Option](#)" for further information.

Geometry options

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

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

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

See "[Transmissibility Calculations](#)" for further information.

Fully implicit technology (Black Oil)

ECLIPSE uses the fully-implicit method to provide stability over long timesteps. 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](#)" for further information.

Adaptive Implicit and IMPES (Compositional)

ECLIPSE 300

In a compositional model where the number of components and hence equations to be solved is greater than say 5 or 6, the cost of using a fully implicit technique can become prohibitive in both memory and CPU time. In ECLIPSE 300 this problem is tackled by using an 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](#)" 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](#)".

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 tri-diagonal 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 calculates their transmissibilities. If you are using the Local Grid Refinement option (see "[Local grid refinement](#)"), 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 two-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 and Vertical Equilibrium options for example) is deduced from data provided in the [GRID](#) section. Space is saved by not storing unnecessary data for inactive cells.

Vertical equilibrium

ECLIPSE 100

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

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

See "[Vertical Equilibrium](#)" 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](#)" 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](#)" 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](#)").

See "[Hysteresis](#)" 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](#)" 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 sulfur 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](#)" for further information

API tracking

ECLIPSE 100

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

See "[API Tracking](#)" 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](#)" for further information.

Temperature model

ECLIPSE 100

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

See the "[Temperature option](#)" 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](#)" 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](#)" for further information.

Individual well controls

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

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

Production wells can also be subject to an additional class of ‘economic’ constraints. A producer is automatically shut-in or re-completed if its oil, gas or liquid rate falls below an economic limit. If its water cut, gas-oil ratio or water-gas ratio exceeds the specified upper limit, the well is worked over or shut-in. The rate of doing workovers can be limited by rig availability. Water cut, gas-oil ratio and water-gas ratio limits, and minimum economic rate limits, can also be applied to individual connections, which is closed if their limits are violated. In addition there is an option to limit the water cut, gas-oil ratio or water-gas ratio from a well by successively reducing its production rate each time the limit is violated. In ECLIPSE 100 wells subject to a minimum tubing head pressure limit can have their productivity boosted by automatically switching on artificial lift when their oil or liquid flow rates fall below a specified limit, subject to constraints on the group or field capacity for artificial lift. Individual wells can be ‘flagged’ to make the simulation run stop if they are shut-in for any reason, or another well can be opened automatically. The run can also be made to stop if all the producers in the field, or in one or more nominated groups, are shut-in. Wells that are shut-in can be tested periodically to see if they can flow economically again.

See "[Well controls and limits](#)" 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 workover 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](#)" 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](#)" 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 timestep to inject the surplus gas.

See "[Sales gas production control](#)" 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](#)" 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. Multisegment Wells (see "[Multisegment wells](#)") or the ECLIPSE 100 Wellbore Friction option (see "[Wellbore friction](#)") 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 timesteps. 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](#)" 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](#)" for further information.

Aquifers

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

See "[Aquifer Modeling Facilities](#)" for further information.

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,
- Multisegment wells.
- Thermal option

The special extensions may be purchased separately as required.

Polymer flood model

ECLIPSE 100

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

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

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

See "[Polymer flood model](#)" 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 subsea 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 multilevel 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 "[Network option](#)" 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 ([item 25](#) in the OPTIONS keyword) to reduce computing costs in sensitivity studies.

See "[Flux boundary conditions](#)" 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 auto-refinement facility may be used to refine an existing global model as the reverse process to grid coarsening.

See "[Local grid refinement and coarsening](#)" for further information.

Wellbore friction

ECLIPSE 100

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

See "[Wellbore friction option](#)" for further information.

Gas field operations model

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

- Using a seasonal profile of monthly multipliers to the mean rate or DCQ (Daily Contracted Quantity) to set the production target,
- Adjusting the DCQ each contract year to obey a swing factor requirement,
- Control and output of sales gas production,
- Estimation of delivery capacity at each timestep, 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.

ECLIPSE 100

See "[Gas field operations model](#)" for further information.

Solvent model

ECLIPSE 100

The Solvent Model is a four-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 hydrocarbons. 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 "[Solvent model](#)" for further information.

Gi pseudo-compositional model

ECLIPSE 100

The Gi pseudo-compositional model is a four-equation model which improves the modeling of volatile oil and gas condensate systems, without moving to a fully compositional description. The Gi parameter is a measure of the amount of dry gas that has contacted the original hydrocarbons. The saturated R_s and R_v values and the B_o and B_g of the oil and gas are supplied to ECLIPSE as functions of both pressure and Gi. For example the saturated R_v value of the gas will reduce with Gi; thus the gas is able to contain less vaporized oil. This effect overcomes one of the problems associated with black oil models, namely that of predicting overoptimistic 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](#)" for further information.

Surfactant model

ECLIPSE 100

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

the surfactant concentration on the water viscosity, and the adsorption of surfactant onto the rock, can be treated by the model.

See "[Surfactant model](#)" for further information.

Gas lift optimization

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

See "[Gas lift optimization](#)" 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](#)" for further information.

Environmental tracer

ECLIPSE 100

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

See "[Environmental tracers](#)" for further information.

Coal bed methane

ECLIPSE 100

The Coal bed methane option allows the modeling of gas production (methane) from coal seams. Gas present in the coal matrix diffuses into a fracture system and can be produced using traditional oil field technology. The option models the diffusion process and the flow of water and gas within the fracture system. It is also possible to model enhanced recovery mechanisms using a third conservation equation to represent an injected gas, typically CO₂.

See "[Coal bed methane model](#)" for further information.

Foam model

ECLIPSE 100

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

See "[Foam model](#)" for further information.

Reservoir coupling

ECLIPSE 100

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

See "[Reservoir coupling](#)" for further information.

Gas calorific value control

ECLIPSE 100

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

See "[Gas calorific value control](#)" for further information.

Multisegment wells

The multisegment well model provides a detailed description of fluid flow in the well bore. The facility is specifically designed for horizontal and multilateral 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 precalculated VFP tables, which can potentially offer greater accuracy and provide the ability to model chokes and other flow control devices. The Multisegment 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 "[Multisegment wells](#)" 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 option runs in fully implicit mode. The production system model is the multilevel hierarchy in ECLIPSE 300, with an implicit crossflow 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](#)" for further information.

Ancillary programs

Ancillary programs available with ECLIPSE are described below.

VFPi

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

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

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

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

GRAF

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

Typical uses of GRAF are:

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

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 file to generate color displays of saturations and pressures. You can step forwards or backwards through color sequences creating animated displays of water floods and gas coning events for example.

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 wide range of plotters and printers.

Additional programs

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

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 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 MPI, which is provided with ECLIPSE.

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

FloGrid

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

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

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

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

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

PVTi

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

SCAL

SCAL is a tool to help you effectively use laboratory derived relative permeability and capillary pressure measurements in reservoir simulation. The program has facilities to read in laboratory data, perform quality control such as curve smoothing, group data according to lithological parameters and end-point values, transform the laboratory data into rock curves suitable for ECLIPSE, and automatically assign these curves to grid cells on the basis of rules which you set up (for example, as a function of porosity / permeability / lithological parameters). The output consists of a series of [INCLUDE](#) files, for both the [PROPS](#) and [REGIONS](#) sections. The program has facilities for 3D visualization of simulation grids and three-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 Multisegment Well Model (see "[Multisegment wells](#)"). 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 [COMPSEG / COMPSEGL](#) keywords describing the multisegment 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.

ECLIPSE Office

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

SimOpt

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

PlanOpt

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

FrontSim

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

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

2

Adjoint gradient screening

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

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

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

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

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

The adjoint method for gradient calculation is also used in the optimization of well control settings to maximize production from a simulation model (see "[Optimization](#)").

The adjoint production optimization method cannot be used at the same time as this adjoint gradient screening option.

The sections that follow give an overview of the keywords necessary to use this option. The output from this option is described as well as a list of the restrictions and limitations of its use. A description of the formulation is also provided.

Keywords

In the RUNSPEC section, the [AJGRADNT](#) keyword must be specified to activate the option.

Specifying the objective functions

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

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

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

Specifying the parameters

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

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

Example

```

SCHEDULE
AJGPARAM
'PERMXY' 'PORO' /
AJGWELLS
'P1' 'WBHP' /
'P2' 'WBHP' /
'I1' 'WBHP' /
/
RPTRST
BASIC=2 /
WELSPECS
'P1' 'GRUP' 1 1 3000.0 'OIL' 1* 'NO' /
'P2' 'GRUP' 1 5 3000.0 'OIL' 1* 'NO' /
'I1' 'GRUP' 5 1 3000.0 'WATER' 1* 'NO' /
/
COMPDAT
'P1' 1 1 1 1 'OPEN' 0 1* 0.2 1* 0.0 1* 'Z' /
'P2' 1 5 1 1 'OPEN' 0 1* 0.2 1* 0.0 1* 'Z' /
'I1' 5 1 1 1 'OPEN' 0 1* 0.2 1* 0.0 1* 'Z' /
/
WCONPROD
'P*' 'OPEN' 'LRAT' 3* 100 1* 1* /
WCONINJE
'I1' 'WAT' 'OPEN' 'RATE' 400 1* 1*
/
TSTEP
50*5.0 /
END

```

Output

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

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

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

Example

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

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

...
Lines skipped here
...
'ENDAJG'          0 'MESS'
```

The new records appended to the restart file are:

- AJGRADNT - the header marking the start of the section
- AJGNAMES - a character array consisting of blocks of four fields containing the following information
 - Name of the sensitivity (internally assigned by ECLIPSE)
 - Name of the well
 - Name of the dynamic parameter on the well

- Name of the static grid property
- AJGPARA - an integer array containing the following information for the record that appears next in the file
 - Report step number
 - Well number
 - Parameter number
 - Number of sensitivity values that will be output
 - Reserved for future use
- AJGFN 1 etc. - a real array containing one sensitivity value for every active grid cell
- ENDAJG - the header marking the end of the adjoint gradient section.

Restrictions

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

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

Formulation

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

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

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

$$dW = \left(\frac{\partial W}{\partial p} \right) \cdot dp + \left(\frac{\partial W}{\partial X} \right) \cdot dX \quad \text{Eq. 2.1}$$

Remembering that the flow equations still have to be honored:

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

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

$$\frac{dW}{dp} = \left\{ \left(\frac{\partial W}{\partial p} \right) + \Psi \cdot \left(\frac{\partial R}{\partial p} \right) \right\} + \left\{ \left(\frac{\partial W}{\partial X} \right) + \Psi \cdot \left(\frac{\partial R}{\partial X} \right) \right\} \cdot \frac{dX}{dp} \quad \text{Eq. 2.3}$$

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

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

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

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

So, at each time level, k , the Lagrange multipliers can be calculated by solving the “adjoint” equations, equations 2.4 and 2.5:

$$\frac{\partial W_k}{\partial X_k} + \Psi_{k,k} \cdot \frac{\partial R_k}{\partial X_k} = 0 \quad \text{Eq. 2.4}$$

$$\Psi_{k,m} \cdot \frac{\partial R_m}{\partial X_m} + \Psi_{k,m+1} \cdot \frac{\partial R_{m+1}}{\partial X_m} = 0 \quad \text{Eq. 2.5}$$

where $m = 1, \dots, k-1$.

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

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

3

Alkaline model

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

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

Alkaline flooding is usually more efficient if the acid content of the reservoir oil is relatively high.

Note: In ECLIPSE 300, it is possible to model the effect of alkaline on water-oil surface tension when Surfactant option is activated. This can be achieved by specifying ALK as a water component type, see "[Water component keywords](#)", and specifying keyword [ALSURFST](#) in the PROPS section.

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

The alkaline model described in the following sections provides a simplified model that does not take into account the in-situ surfactant creation and the phase behavior:

Model 1

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

Alkaline model 1

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

The alkaline conservation equation

The alkaline is assumed to exist only in the water phase. To specify the concentration of alkaline in a water injector, refer to the [WALKALIN](#) keyword. The distribution of the injected alkaline is modeled by solving a conservation equation:

$$\frac{d}{dt} \left(\frac{VS_w C_a}{B_r B_w} \right) + \frac{d}{dt} \left(V\rho_r C_a^a \frac{1 - \varphi}{\varphi} \right) = \sum \left[\frac{T k_{rw}}{B_w \mu_s \text{eff}} (\delta P_w - \rho_w g D_z) \right] C_a + Q_w C_a \quad \text{Eq. 3.1}$$

where

V is the block pore volume

S_w is the water saturation

C_a denotes the alkaline concentration

B_w, B_r are the water and rock formation volume respectively

ρ_w, ρ_r denote the water and rock density respectively

C_a^a denotes the adsorbed alkaline concentration

φ denotes the porosity

Σ denotes the sum over neighboring cells

T is the transmissibility

k_{rw} is the water relative permeability

$\mu_s \text{eff}$ denotes the effective viscosity of the salt

P_w is the water pressure

g is the gravity acceleration

D_z is the cell center depth

Q_w is the water production rate.

By default, the alkaline concentrations are updated at the end of a timestep after the inter-block phase flows have been determined.

If the alkaline effects of the flow properties are very strong, the alkaline concentrations can be solved at each Newton iteration so that the tracer concentrations are updated simultaneously with the phase equations. To do this, set item 126 of the OPTIONS keyword to 1.

If the [ECLMC](#) keyword is found in the RUNSPEC section, the default solution is to update the alkaline concentration at each Newton iteration. The new alkaline concentrations are then used to update the flow properties for the subsequent iterations.

Treatment of adsorption

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

- either a lookup table of adsorbed alkaline as a function of alkaline concentration using the [ALKADS](#) keyword

- or by a generic analytical adsorption model using the [ADSORP](#) keyword. This model enables the adsorption to depend on rock permeability as well.

If desorption is prevented (refer to the item 1 of the [ALKROCK](#) keyword), then the adsorbed alkaline concentration may not decrease with time. If desorption is allowed, then each grid block retraces the adsorption isotherm as the alkaline concentration falls in the cell.

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

Alkaline effect on water-oil surface tension

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

$$\sigma_{wo} = \sigma_{wo}(C_{surf})A_{st}(C_{alk}) \quad \text{Eq. 3.2}$$

where

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

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

Alkaline effect on surfactant/polymer adsorption

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

$$V\rho_r C_{s,p}^a \frac{1-\varphi}{\varphi} A_{ad}(C_{alk}) \quad \text{Eq. 3.3}$$

where

V is the pore volume of the cell

ρ_r is the mass density of the rock (see the [SURFROCK](#) / [PLYROCK](#) keywords)

$C_{s,p}^a$ is the surfactant/polymer adsorbed concentration obtained from the [SURFADS](#) / [PLYADS](#) / [PLYADSS](#) / [ADSORP](#) keywords

φ is the porosity

$A_{ad}(C_{alk})$ is the adsorption multiplier at alkaline concentration (refer to the [ALSURFAD](#) / [ALPOLADS](#) keywords).

When the alkaline is assumed to be adsorbed irreversibly, by preventing desorption in the [ALKROCK](#) keyword, its effect on the surfactant/polymer adsorption is assumed to be irreversible as well. Using the previous notations, the mass of adsorbed surfactant or polymer then becomes:

$$V\rho_r C_{s,p}^a \frac{1-\varphi}{\varphi} A_{ad}(C_{alk\ max}) \quad \text{Eq. 3.4}$$

where

$C_{alk\ max}$ is the maximum alkaline concentration reached in the block, corresponding to the actual effect on surfactant/polymer adsorption.

Note: If surfactant and polymer are used in the same run the **PLYROCK** keyword will supersede the **SURFROCK** keyword.

Using alkaline model 1

This model is activated by specifying the **ALKALINE** keyword in the RUNSPEC section. The polymer flood model or the surfactant flood model should be active as well.

The presence of at least one of the following keyword in the PROPS section is obligatory.

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

The concentration of the injected alkaline in a water injector is specified using the **WALKALIN** keyword.

For the list of output associated with the alkaline model, refer to the SUMMARY and SCHEDULE sections below.

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

Summary of keywords

RUNSPEC section

- **ALKALINE** activates the Alkaline model

PROPS section

Restriction	Alkaline Model 1 keyword	Description
Only if the surfactant option is activated.	ALSURFST	tables of oil/water surface tension as a function of alkaline concentration.
Only if the surfactant option is activated.	ALSURFAD	tables of surfactant adsorption as a function of alkaline concentration.
Only if the polymer option is activated.	ALPOLADS	table for polymer adsorption as a function of alkaline concentration.
	ALKADS	table for alkaline adsorption function as a function of alkaline concentration
	ADSORP	analytical adsorption isotherm (alternative to ALKADS)
	ALKROCK	alkaline-rock adsorption/desorption properties

SUMMARY section

Refer to [Alkaline SUMMARY output controls](#) in the *ECLIPSE Reference Manual*.

SCHEDULE section

Keyword	Description
RPTSCHED	Controls output from the SCHEDULE section to the print file. Mnemonic ALKALINE outputs the alkaline concentrations and the alkaline adsorption when active. Outputs are: <ul style="list-style-type: none">• ALKALINE: alkaline concentration• ALKADS: alkaline concentration adsorbed• ALKMAX: maximum alkaline concentration adsorbed
WALKALIN	Specifies the alkaline concentration for a water injector. This concentration can be set as a UDA (user defined argument).

Example problem

This example mentions only those sections of an ECLIPSE data file that apply to alkaline data input and output.

RUNSPEC

```
RUNSPEC
TITLE
Example Alkaline and Surfactant Case.

DIMENS
```

```

      5      5      1   /
NONNC
OIL
WATER
SURFACT
ALKALINE
FIELD

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

```

PROPS

```

PROPS

SWFN
-- Table 1 - Immiscible curves - SATNUM
  .25      .0      4.0
  .7       1.0      .0
/
-- Table 2 - Miscible curves - SURFNUM
  .25      .0      4.0
  .99      1.0      .0
/
SOF2
-- Table 1 - Immiscible curves - SATNUM
  .3000    .0000
  .7600  1.0000
/
-- Table 2 - Miscible curves - SURFNUM
  .0100    .0000
  .7600  1.0000
/
PVTW
 4000.0   1.0   3.03E-06   .5   0.0   /
 4000.0   1.0   3.03E-06   .5   0.0   /
PVDO
  .0      1.0      2.0
 8000.0   .92      2.0
/
-- Table 2 defaulted
/
ROCK
 4000.0           .30E-05   /
 4000.0           .30E-05   /
DENSITY
 52.0000  64.0000   .04400   /
 52.0000  64.0000   .04400   /
-- Water Viscosity vs surfactant concentration tables.
SURFVISC

```

```

-- Surfactant      Water viscosity at 4000 psi
-- concentration
-- LB/STB          CENTIPOISE
  0.0              1.0
  0.1              1.1
  0.2              1.2
  0.8              1.3  /
  0.0              0.5
  0.1              0.6
  0.2              0.7
  0.8              0.8  /

-- Surfactant Adsorption tables
SURFADS
-- Surfactant      Adsorbed saturation
-- concentration
-- LB/STB          LB/LB
  0.0              0.00000
  0.1              0.00005
  0.2              0.00005
  1.0              0.00005
/
-- Table 2 defaulted
/
-- Alkaline adsorption
ALKADS
-- Alkaline         Adsorbed
-- concentration    saturation
-- LB/STB
  0.0              0.00000
  0.2              0.00002
  1.0              0.00005 /
/
-- Alkaline adsorption properties
ALKROCK
-- De-adsorption model
  1                  /
/
-- Alkaline multipliers for Surfactant Adsorption
ALSURFAD
-- alkaline         adsorption
-- concentration   multiplier
-- LB/STB
  0.0              1.00000
  0.1              0.99995
  0.2              0.99500
  1.0              0.50000 /
/
-- Water/Oil surface tension vs Surfactant concentration
SURFST
-- Surfactant      Water/Oil Surface Tension
-- concentration
-- LB/STB          LB/IN
  0.0              100.0
  0.1              50.0
  0.2              10.0
  0.5              0.0  /
/
-- Water/Oil surface tension multipliers vs alkaline concentration
ALSURFST
-- Alkaline         Water/Oil Surface Tension
-- concentration   multiplier
-- LB/STB
  0.0              1.0
  0.1              0.9
  0.2              0.9
  0.5              0.9  /
/
-- Capillary de-saturation curve
SURFCAPD
-- LOG10(Capillary Number)   Miscibility
--                                Function

```

```

      -9          0.0
      2          0.0
      5          1.0
     10         1.0 / 
/
-- Extra rock properties.
SURFROCK
-- De-adsorption model      Rock mass
-- indicator                 Density
-- 1 - reversible
-- 2 - irreversible
    1                      LB/RB
    1                      1000 /
    1                      1000 /

```

SUMMARY

```
SUMMARY
```

```
CTFRALK
```

```
'P' 5 5 1 /
```

```
/
```

```
FTPRAALK
```

```
FTPTALK
```

```
FTIRALK
```

```
FTITALK
```

```
WTPTALK
```

```
'P' /
```

```
WTITALK
```

```
'I' /
```

```
BTCNFALK
```

```
1 1 1 /
```

```
2 2 1 /
```

```
5 5 1 /
```

```
/
```

```
BTADSAALK
```

```
1 1 1 /
```

```
2 2 1 /
```

```
5 5 1 /
```

```
/
```

```
FTPRSUR
```

```
WTPRSUR
```

```
'P' /
```

```
WTPTSUR
```

```
'P' /
```

```
WTITSUR
```

```
'I' /
```

```
BTCNFSUR
```

```
1 1 1 /
```

```
2 2 1 /
```

```
5 5 1 /
```

```
/
```

```
FTIPTSUR
```

```
FTIPFSUR
```

```
RTIPTSUR
```

```
1 /
```

```
RTIPFSUR
```

```
1 /
```

```
BTIPTSUR
```

```
1 1 1 /
```

```
/
```

```
FTADSUR
```

```
RTADSUR
1 /
```

```
BTADSUR
1 1 1 /
/
```

```
BTCASUR
1 1 1 /
/
```

SCHEDULE

```
SCHEDULE
RPTSCHED
```

```
ALKALINE
```

```
/
```

```
WELSPECS
```

```
'I' 'G' 1 1 4000 'WAT' /
'P' 'G' 5 5 4000 'OIL' /
/
```

```
COMPDAT
```

```
'I' 'OPEN' 1 1 1 1 'OPEN' 0 .0 1.0 /
'P' 'OPEN' 5 5 1 1 'OPEN' 0 .0 1.0 /
/
```

```
WCONPROD
```

```
'P' 'OPEN' 'BHP' 5* 3999.0 /
/
```

```
WCONINJ
```

```
'I' 'WAT' 'OPEN' 'RATE' 200.0 /
/
```

```
-- Start injecting the Surfactant Slug
```

```
WSURFACT
```

```
'I' 0.8 /
/
```

```
-- Start injecting the Alkaline
```

```
WALKALIN
```

```
'I' 0.8 /
/
```

```
TSTEP
```

```
2*100
/
```

```
-- Water chase
```

```
WSURFACT
```

```
'I' 0.0 /
/
```

```
WALKALIN
```

```
'I' 0.0 /
/
```

```
TSTEP
```

```
5*100 /
/
```

```
END
```

4

API tracking

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

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

Without the API tracking facility, the presence of different types of oil in the reservoir could be handled with the aid of PVT region numbers. For instance, 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. So, for example, 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 timestep 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 timestep to update the oil surface density in each grid block, in order to model the mixing of the different oil types.

The API tracking facility is activated by specifying the [API](#) keyword 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 API tracking facility

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 keywords [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](#)". 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 overrides 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. To accomplish this, 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 indicates 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 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, which is similar to runs that do not use API Tracking. For instance, 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** and **SWAT** for example), 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 timestep. When a converged solution for the timestep 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 timestep.

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.

The [APILIM](#) keyword can be used to monitor the numbers of grid blocks with API values that are outside specified limits and additionally to constrain the API values to within these limits.

API tracking in wells

The oil surface density in each well is calculated at the end of the timestep 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. (If required, the pre-98A treatment of wellbore hydrostatic head, which uses the well’s average oil surface density, can be restored by setting [item 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. In other words, the API is interpreted as a property of the oil component rather than the liquid oil phase. However, it is the API in the liquid oil phase that is used in subsequent calculations involving the surface density of oil.

By using [item 58](#) of the [OPTIONS](#) keyword this behavior can be modified. When the item 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. However, some 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.

5

Aquifer modeling facilities

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The models available are:

- Numerical aquifers
- Carter-Tracy aquifers
- Fetkovich aquifers
- Constant flux aquifers
- Constant head aquifers
- Multi-component aquifers

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

The other types of aquifer, classed as ‘analytic aquifers’, are represented by computed source terms in the reservoir grid cells with which they connect.

A fuller description of these numerical and analytical aquifers is provided in the following sections.

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, and PVT and saturation table numbers) may be specified explicitly using keyword [AQUNUM](#) in the GRID section, or if the properties of 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 only information concerning the shape of the numerical aquifer is given by the cross-sectional area, length and depth.

The aquifer should be connected to a face of the reservoir using the keyword AQUCON. The first cell of the aquifer declared in the keyword AQUNUM connects to the specified faces of the reservoir by non-neighbor connections. The set of cells defining the aquifer are connected together (possibly by non-neighbor connections) in the order specified in AQUNUM. The aquifer cells are isolated from the grid except for these

connections. In dual porosity runs, the aquifer should be placed in the lower half (that is the fracture part) of the grid.

The aquifer **pore volume**, for simulation grid cell i , is calculated from the data entered in the **AQUNUM** keyword using the equation:

$$\text{PORV}_i = \text{PORO}_i \cdot \text{LENGTH}_i \cdot \text{AREA}_i \quad \text{Eq. 5.1}$$

where

PORV_i is the pore volume of the aquifer cell i

PORO_i is the porosity of the aquifer cell i

LENGTH_i is the length of the aquifer cell i

AREA_i is the area of the aquifer cell i .

The porosity of the aquifer cell is taken from the grid block if it is defaulted in the **AQUNUM** keyword. Any values of **MULTPV** entered in the **GRID** or **EDIT** sections will not apply to the numerical aquifer cells, and any edits to **PORV** in the **EDIT** section for numerical aquifer cells are ignored.

The **transmissibility** between two aquifer cells is given by:

$$\text{TR} = \frac{\text{CDARCY}}{1/T_i + 1/T_j} \quad \text{Eq. 5.2}$$

with

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

where

PERMX_i is the permeability of aquifer cell i

XSECT_i is the cross-sectional area

LENGTH_i is the length of the aquifer cell i .

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).

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

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**. The use of a multiplier may be considered in order to aid history matching, for instance.

ECLIPSE 100

Transmissibility values are calculated using the **OLDTRAN** block centered method (see "[Transmissibility calculations](#)").

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 [AQUDIMS](#) keyword in the RUNSPEC section.

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):

$$T_c = \frac{\mu_w \varphi C_t r_o^2}{k_a c_1} \quad \text{Eq. 5.3}$$

where

μ_w is the viscosity of water in the aquifer

φ is the aquifer porosity

C_t is the total (rock + water) compressibility

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

k_a is the aquifer permeability

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):

$$\beta = c_2 h \theta \varphi C_t r_o^2 \quad \text{Eq. 5.4}$$

where

h is the aquifer thickness

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

φ is the aquifer porosity

C_t is the total (rock + water) compressibility

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

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 using the following expression:

$$t_D = \frac{t}{T_c} \quad \text{Eq. 5.5}$$

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

$$p_{a0} - \bar{p} = \frac{Q_a}{\beta} \text{PI}_D(t_D) \quad \text{Eq. 5.6}$$

where

Q_a is the aquifer inflow rate

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

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

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

$$\bar{Q}_{ai} = \alpha_i \{ a - b [p_i(t + \Delta t) - p_i(t)] \} \quad \text{Eq. 5.7}$$

where

$$a = \frac{1}{T_c} \left\{ \frac{\beta \Delta p_{ai} - W_a(t)(\text{PI}'_D(t + \Delta t))_D}{\text{PI}'_D(t + \Delta t)_D - t_D \text{PI}'_D(t + \Delta t)_D} \right\} \quad \text{Eq. 5.8}$$

and

$$b = \frac{\beta}{T_c [\text{PI}'_D(t + \Delta t)_D - t_D \text{PI}'_D(t + \Delta t)_D]} \quad \text{Eq. 5.9}$$

where

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

PI_D' is the derivative of PI_D with respect to t_D

α_i is the area fraction for each connection.

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

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

where

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

m_i is an aquifer influx coefficient multiplier.

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

The aquifer properties (compressibility, porosity, initial pressure, depth and radius for example) should be defined using the keyword [AQUCT](#), and the aquifer connections to one or more faces of the reservoir should be made using the keywords [AQUANCON](#) and [AQANCONL](#). The latter two keywords help to distinguish between connections to a global grid cell or a local grid cell.

ECLIPSE 100

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

The aquifer should also be given a table number for the influence function table that it will 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.

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](#).

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

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.

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

Fetkovich aquifer

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

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

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

The aquifer inflow is modeled by the equation:

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

where

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

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

J is the specified Productivity Index of the aquifer

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

p_a is the pressure in the aquifer at time t

p_c is the capillary pressure

p_i is the water pressure in a connecting grid block i

ρ is water density in the aquifer

d_i is the depth of grid block i

d_a is the datum depth of the aquifer.

ECLIPSE 100

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

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

$$\alpha_i = \frac{m_i A_i}{\sum m_i A_i} \quad \text{Eq. 5.11}$$

where

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

m_i is an aquifer influx coefficient multiplier.

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

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

$$W_a = C_t V_{w0} (p_{a0} - p_a) \quad \text{Eq. 5.12}$$

where

W_a is the cumulative total influx from the aquifer

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

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

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

ECLIPSE 100

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, which has the dimension of time, is given by:

$$T_c = \frac{C_t V_{w0}}{J} \quad \text{Eq. 5.13}$$

where

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

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

J is the specified Productivity Index of the aquifer.

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

$$\bar{Q}_{ai} = \alpha_i J (p_a - p_i + \rho g (d_i - d_a)) \left(\frac{1 - e^{(-\Delta t / T_c)}}{\Delta t / T_c} \right) \quad \text{Eq. 5.14}$$

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

At the end of each timestep, the aquifer's cumulative total influx is incremented and its pressure is updated using equation 5.12.

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

The aquifer properties (compressibility, porosity, initial pressure, depth and productivity index for example) should be defined using the keyword [AQUFETP](#), and the aquifer connections to one or more faces of the reservoir should be made through the keywords [AQUANCON](#) and [AQANCONL](#). The latter two keywords help to distinguish between connections to a global grid cell or a local grid cell.

ECLIPSE 100

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

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. 38].

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.

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

For Tracer Tracking runs, the initial concentrations of water phase tracers in the aquifer may be supplied using the keyword [AQANTRC](#).

For the Brine Tracking option, the initial salt concentration may be supplied using [AQUFET](#) (ECLIPSE 100 only) or [AQUFETP](#).

Constant flux aquifer

A constant flux aquifer has its water influx rate specified directly by the user, instead of being calculated by an analytic aquifer model. For the purpose of dimensioning, among other things, 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:

$$Q_{ai} = F_a \cdot A_i \cdot m \quad \text{Eq. 5.15}$$

where

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

F_a is the aquifer flux, entered by the user

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

m is the aquifer influx multiplier, entered by the user.

The aquifer should be defined using the [AQUFLUX](#) keyword in the SOLUTION section, and the aquifer connections to one or more faces of the reservoir should be made via the keywords [AQUANCON](#) and [AQANCONL](#). The latter two keywords help to distinguish between connections to a global grid cell or a local grid cell.

To achieve time-dependency, the flux may be modified during simulation by reentering the [AQUFLUX](#) keyword in the SCHEDULE section.

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

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

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

Constant head aquifers

A constant head aquifer can be defined in the SOLUTION section by specifying the [AQUCHWAT](#) keyword for a water aquifer, and by the [AQUCHGAS](#) keyword for a gas aquifer.

The aquifer connections to one or more faces of the reservoir should be made via the keywords [AQUANCON](#) and [AQANCONL](#). These keywords help to distinguish between connections to a global grid cell or a local grid cell.

To achieve time-dependency, the conditions may be modified during simulation by reentering the AQUCHWAT or AQUCHGAS keywords in the SCHEDULE section.

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

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

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

For tracer tracking runs, the concentrations of water- or gas-phase tracers in the aquifer may be supplied using the keyword [AQANTRC](#).

Multi-component analytic aquifers

ECLIPSE 300

Constant head gas aquifers defined by the [AQUCHGAS](#) keyword can be composed of any components that can be present in the gas phase.

Refer to the [AQUCHGAS](#) keyword description for more information.

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

Refer to:

- "Analytic water aquifers" for the CO2STORE and GASWAT options,
- "Aquifer properties" for the CO2SOL option,
- "Aquifer properties" for the COMPW option.

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

Summary of keywords

RUNSPEC

- [AQUDIMS](#) sets dimensions for aquifers

GRID

Restriction	Keyword	Description
	AQUCON	specifies connection data for numerical aquifers
	AUCT	specifies the property data for Carter-Tracy aquifers
E100 only	AQUNNC	set non-neighbor connection values explicitly for numerical aquifers.
	AQNUM	assigns a numerical aquifer to a block.
	INIT	send aquifer connection data to the INIT file: Aquifer connection arrays AQUIFERN, AQUIFERA and AQUIFERG are written to the INIT file depending on whether numerical or analytic aquifers are present.
	RPTGRID	controls output from the GRID section to the print file: Use AQUCON and AQNUM to output numerical aquifer connections and definitions respectively

PROPS

Restriction	Keyword	Description
E300 only	AUCT	specifies the property data for Carter-Tracy aquifers
	AQUTAB	influence function tables for Carter-Tracy aquifers.
E100 only	RPTPROPS	controls output from the PROPS section to the print file: AQUTAB outputs Carter-Tracy influence tables.

SOLUTION

Restriction	Keyword	Description
	AQANCONL	specifies local grid refinement connection data for analytic aquifers
E100 only	AQANNC	sets connection values explicitly for analytic aquifers.
	AQANTRC	Sets initial tracer concentrations for analytic aquifers.
E300 only	AQSTREAM	sets the composition of the injected gas stream for an AQUCHGAS aquifer; or sets the composition of an AQUCHGAS aquifer; or sets the composition of the GASWAT water analytic aquifer.
E300 only	AQSTREAW	sets the composition of a multi-component water analytic aquifer
E100 only	AQUALIST	sets up lists of aquifers for output
	AQUANCON	specifies connection data for analytic aquifers
	AQUCHGAS	specifies the property data for constant head/pressure gas aquifers
	AQUCHWAT	specifies the property data for constant head/pressure water aquifers
	AUCT	specifies the property data for Carter-Tracy aquifers
E100 only	AQUFET	specification data for Fetkovich aquifers
	AQUFETP	specifies the property data for Fetkovich aquifers
	AQUFLUX	specifies a constant flux aquifer
E100 only	RAINFALL	specifies a monthly varying flux rate for a constant flux aquifer.

Restriction	Keyword	Description
	RPTSOL	<p>controls output from the SOLUTION section to the print file:</p> <p>To output analytic aquifer data such as Fetkovich, Carter-Tracy, Constant Flux or Constant Head use the AQUFET, AQUFETP, AQUCT, AQUFLUX, AQUANCON or AQANCONL arguments.</p> <p>ECLIPSE 100: If set to 2: data for each aquifer-grid block connection will also be printed (input using AQUANCON).</p> <p>ECLIPSE 300: Data for each aquifer-grid block connection will be printed using the argument. Note: AQUFET applies only for ECLIPSE 100.</p> <p>Note: AQANCONL applies only for ECLIPSE 300</p>

SUMMARY

Refer to the following sections in the *ECLIPSE Reference Manual*.

- [Analytic aquifer quantities](#)
- [Numerical aquifer quantities](#)

SCHEDULE

Restriction	Keyword	Description
	AQSTREAM	sets the composition of the injected gas stream for an AQUCHGAS aquifer; sets the composition of an AQUCHGAS aquifer; or sets the composition of a GASWAT water analytic aquifer
E300 only	AQSTREAW	sets the composition of a multi-component water analytic aquifer.
	AQUCHGAS	specifies the property data for constant head/pressure gas aquifers
	AQUCHWAT	specifies the property data for constant head/pressure water aquifers
	AQUCT	specifies the property data for Carter-Tracy aquifers
E100 only	AQCWFAC	changes the property for all constant head aquifers
	AQUFETP	specifies the property data for Fetkovich aquifers
	AQUFLUX	specifies a constant flux aquifer
E100 only	RAINFALL	specifies a monthly varying flux rate for a constant flux aquifer.
E300 only	RPTPRINT	controls output of printed reports. Use item 13 of keyword to report aquifer flows.
	RPTSCHED	<p>controls output from the SCHEDULE section to the print file:</p> <p>Use AQUCT to output the Carter-Tracy aquifer status or AQUFET, AQUFETP to output the Fetkovich aquifer status.</p> <p>Use AQUCHWAT, AQUCHGAS and AQUFLUX to output constant head/pressure or flux aquifer information.</p> <p>Note: AQUFET applies only to ECLIPSE 100.</p>

6

Asphaltene option

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The purpose of this option is to model asphaltene formation and its impact on reservoir performance. There are a number of asphaltene models currently in existence, but there is still no consensus about the characterization of asphaltene behavior. The asphaltene option implements a simplified but practical model that provides users with sufficient flexibility to simulate a wide range of processes associated with asphaltene behavior.

The asphaltene modeling option is activated with the `ASPHALT` keyword in the RUNSPEC section.

Asphaltene formation

Given that the terminology of these processes in the literature may be different, here we use the following terms:

- precipitation: where asphaltene particles (“fines”) appear in the fluid
- dissolution: the reverse of precipitation, where fines dissolve back into the oil
- flocculation: where the precipitated fines aggregate to form bigger particles, (“flocs”)
- dissociation: the reverse of flocculation; where flocs split up into fines
- deposition: where flocs leave the fluid and attach to the reservoir rock
- entrainment: the reverse of deposition, where deposited asphaltene becomes suspended in the fluid.

Asphaltene damage

Asphaltene precipitation and deposition can alter the reservoir flow in a number of ways. It is possible to model the effects of asphaltene on the following reservoir and fluid properties:

- porosity
- permeability
- viscosity
- wettability

Pre-2011.1 model

A modified asphaltene model was introduced in the 2011.1 version, called the three-component model, since it uses separate components to represent the precipitation, flocculation and deposition states of the asphaltene. The pre-2011.1 model is still available, and both are described in this chapter. The main differences are:

- The three-component model simplifies the precipitation model by allowing only one component to represent the asphaltene in the oil phase (precipitate).
- The three-component model uses an extra solid component to model deposited asphaltene. This leads to some differences in the way the deposition terms are calculated. There are also slight differences in the way asphaltene deposition data is entered in keywords [ASPDEPO](#), [ASPLCRT](#) and [ASPWETF](#). The solid (deposition) component does not take part in the flash, and the solid properties need to be specified.

The value of the solid density in the three-component model is important as it affects the reservoir pressure. If it is less than or greater than the equivalent density of the precipitate in the oil phase, then as asphaltene is deposited the reservoir pressure will increase or decrease to preserve the fluid volume.

In the pre-2011.1 model, deposited asphaltene is still considered as part of the oil phase, so the oil properties do not change as asphaltene is deposited. In the three-component model, deposited asphaltene is removed from the oil phase, leading to a lighter oil and increased volatility. This in turn will affect the reservoir pressure if gas comes out of solution.

- The three-component model no longer uses the pre-2011.1 asphaltene permeability damage model, keyword [ASPKDAM](#). Instead a solid-dependent mobility multiplier can be used.

Asphaltene formation: three-component model

Characterization

Three special components can be defined using the [CATYPE](#) keyword:

- A precipitation component. This component represents the oil components that can become asphaltene. A fraction of the component concentration is identified as being precipitated fines.
- A floc component.
- A deposited asphaltene component. This component must also be a solid component, see "[Solid Phase](#)". This component is only required if asphaltene deposition is modeled.

Precipitation - dissolution

The asphaltene precipitation component can be either dissolved in oil or precipitated as a fine. The [ASPREWG](#) or [ASPPW2D](#) keyword can be used to define the maximum amount of the asphaltene that can be dissolved W_{lim} , which is expressed as the weight of the precipitation component in oil as a percentage of the total weight of oil. This can be expressed as a limit on the mole fraction of the precipitation component dissolved in oil:

$$x_{lim} = \frac{W_{lim}}{100} \cdot \frac{m_{woil}}{m_{wp} \cdot x_p} \quad \text{Eq. 6.1}$$

where $m_{woil} = \sum m_{wi} \cdot x_i$ is the molecular weight of the oil.

If the mole fraction of the precipitation component is less than this value, $x_p < x_{lim}$, the precipitation component will be totally dissolved and there will be no fines. If the mole fraction of the precipitation component exceeds this value, $x_p > x_{lim}$, the dissolved and fine fractions of the precipitation component are given by:

$$x_{pd} = x_{lim}$$

$$x_{pf} = x_p - x_{lim}$$

The fines take part in the flocculation process.

Flocculation - dissociation

Flocculation and dissociation are modeled by two reactions. In the flocculation reaction the fines are converted into the flocculation component, with a rate given by:

$$R_f = V_f \cdot r_f \cdot \frac{m_{wp}}{m_{woil}} \cdot C_{pf} \quad \text{Eq. 6.2}$$

where

$V_f = (1-S_s) \cdot V_p$ is the fluid volume

V_p is the pore volume

S_s is the solid saturation, representing the fraction of the pore volume filled with asphaltene deposit

$C_{pf} = S_o \cdot B_o \cdot x_{pf}$ is the concentration of the fines in the oil phase

S_o is the oil saturation

B_o is the oil molar density

r_f is the flocculation rate specified by the [ASPFLRT](#) keyword.

In the dissociation reaction, the flocculation component is converted into fines, with a rate given by:

$$R_{dis} = V_f \cdot r_{dis} \cdot C_f \quad \text{Eq. 6.3}$$

where

$C_f = S_o \cdot B_o \cdot x_f$ is the concentration of the flocs in the oil phase

r_{dis} is the dissociation rate specified by the [ASPFLRT](#) keyword.

Deposition - entrainment

Deposited asphaltene is modeled by a solid component. Deposition is modeled by a pair of reactions between the asphaltene floc component and the deposited asphaltene component.

Deposition

The first (forward) reaction models adsorption and plugging. The rate at which asphaltene floc is converted to deposit is given by:

$$R_d = \alpha \cdot V_f \cdot C_f + \gamma_2 \cdot \bar{F}_o \cdot L \cdot (C_f - C_{fcr})^+ \quad \text{Eq. 6.4}$$

where

α is the adsorption or static deposition coefficient, specified by the [ASPDEPO](#) keyword

γ_2 is the plugging coefficient, specified by the fifth item of the [ASPDEPO](#) keyword

$C_f = S_o \cdot B_o \cdot x_f$ is the concentration of the flocs in the oil phase (flowing flocs)

C_{fcr} is the critical concentration of flocs for plugging to occur, specified by the [ASPLCRT](#) keyword

\bar{F}_o is the average oil flow rate

L is the length of the flow path

and the equation uses the notation

$$(x)^+ = \max(x, 0)$$

Equation 6.4 can be written in terms of \bar{U}_o , the oil phase average velocity:

$$R_d = V_f \cdot [\alpha \cdot C_f + \gamma_2 \cdot \bar{U}_o \cdot (C_f - C_{fcr})^+] \quad \text{Eq. 6.5}$$

where

$$\bar{F}_o \cdot L = \bar{U}_o \cdot V_f$$

Plugging only occurs if the floc concentration exceeds the critical floc concentration $C_f > C_{fcr}$ and if the volume fraction of deposited asphaltene, represented by the solid saturation S_s , exceeds a critical value $S_s \geq S_{cr}$, where C_{fcr} and S_{cr} are specified by the [ASPLCRT](#) keyword. The half-life of the reaction gives an indication of how large a timestep the simulator can take. Ignoring the C_{fcr} term in equation 6.5, the half-life of the deposition reaction is

$$T_{1/2} = \ln / [\alpha + \gamma_2 \cdot \bar{U}_o] \quad \text{Eq. 6.6}$$

Entrainment

The second (backward) reaction models entrainment. The rate at which the deposited asphaltene is converted to the floc component is given by:

$$R_e = V_f \cdot \beta \cdot (U_o - U_{cr})^+ \cdot S_o \cdot S_s \quad \text{Eq. 6.7}$$

where

β is the entrainment coefficient, specified by the [ASPDEPO](#) keyword

U_{cr} is the critical velocity for entrainment, specified by the [ASPDEPO](#) keyword.

The average oil flow rate \bar{F}_o and average oil velocity \bar{U}_o are both calculated at the previous timestep. First the total oil flow rates in each direction (F_{ox} , F_{oy} and F_{oz}) and flow areas (A_x , A_y and A_z) are calculated by summing the flows and areas on opposite faces, and including any non-neighbor connection contributions:

$$F_{on} = \sum \left| T \cdot \frac{k_{ro}}{\mu_o} \cdot \Delta P \right| \quad \text{Eq. 6.8}$$

$$A_n = \sum A \quad \text{Eq. 6.9}$$

Here the flow is calculated from the inter-cell transmissibility T and pressure difference ΔP , and the cell oil relative permeability k_{ro} and viscosity μ_o . The average velocity in each direction is given by:

$$U_{on} = F_{on} / A_n \quad \text{Eq. 6.10}$$

The average flow is determined by taking the average over the N dimensions:

$$\bar{F}_o = (F_{ox} + F_{oy} + F_{oz}) / (2N) \quad \text{Eq. 6.11}$$

The phase average velocity \bar{U}_o is calculated using the root mean square of the direction velocities:

$$\bar{U}_o = \sqrt{(U_{ox}^2 + U_{oy}^2 + U_{oz}^2)} \cdot \frac{C}{\phi \cdot (1-S_s) \cdot S_o} \quad \text{Eq. 6.12}$$

where

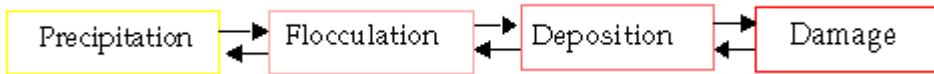
ϕ is the porosity

C is a conversion factor which is used to convert volumes from reservoir barrels to cubic feet if the simulation is in field units.

These flow and velocity calculations only take into account the flows between grid blocks. Well terms are not included - except through their effect on the flow into the grid block. This is discussed further in "[Velocity calculations](#)".

Asphaltene formation: Pre-2011.1 model

Asphaltene modeling is decomposed into different stages:



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

Asphaltene characterization and precipitation

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

Asphaltene as weight percentage

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

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

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

Note that the temperature can be used as a precipitation triggering variable only in thermal runs.

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

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

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

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

$$w_{adis} = \frac{\sum_{l=i}^j m_{wl} x_l}{\sum_{l=1}^N m_{wl} x_l} \quad \text{Eq. 6.13}$$

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

$$w_{apre} = \frac{m_{wk}x_k + \sum_{l=i}^j m_{wl}x_l}{\sum_{l=1}^N m_{wl}x_l} \quad \text{Eq. 6.14}$$

where

m_{wl} is the molecular weight of component l

x_l is the liquid fraction of component l

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

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

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

$$asp_{lim}(P) = \frac{\sum_{l=i}^j m_{wl}x_l^a}{\sum_{l=1}^N m_{wl}x_l} \quad \text{Eq. 6.15}$$

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

$$x_l^a = asp_{lim}(P) \frac{\sum_{l=1}^N m_{wl}x_l}{\sum_{l=i}^j m_{wl}} \quad \text{Eq. 6.16}$$

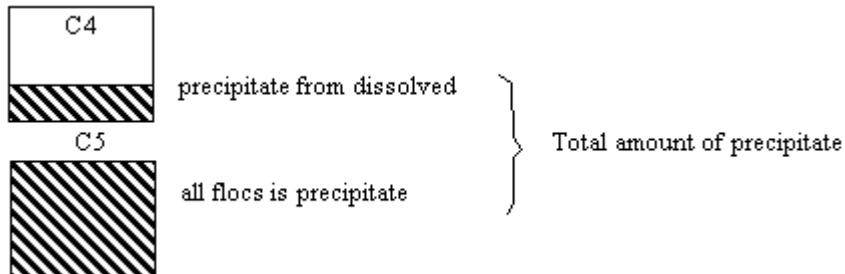
Hence the fraction of precipitate from the dissolved asphaltene writes:

$$E = \frac{\sum_{l=i}^j m_{wl}(x_l - x_l^a)}{\sum_{l=1}^N m_{wl}x_l} \quad \text{Eq. 6.17}$$

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

An example

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



Asphaltene flocculation-dissociation

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

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

$$R_a = \frac{\partial C_a}{\partial t} = r_{ia} C_i - r_{ai} C_a \quad \text{Eq. 6.18}$$

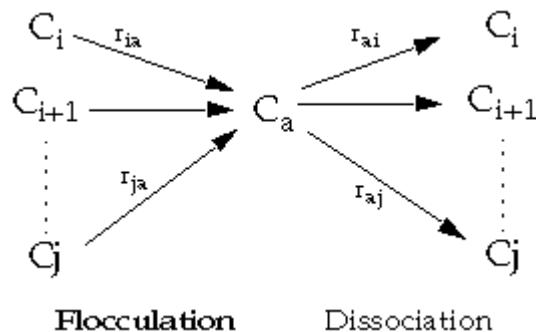
where

r_{ia} is the flocculation rate coefficient of the fines

and r_{ai} is the dissociation rate coefficient of the flocs.

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

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



The maximum number of flocculation is set using argument 20 of [TABDIMS](#) keyword. The default value is 5.

Data input

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

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

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

Asphaltene deposition

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

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

The deposition in the flow direction i is modeled as follows

$$\frac{\partial \varepsilon_i}{\partial t} = \frac{\alpha}{d} \varphi C_a + \gamma_1 |F_{oi}| C_a - \beta(|U_{oi}| - U_{cr})^+ \varepsilon_i \quad \text{Eq. 6.19}$$

where

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

α is the adsorption or static deposition coefficient

d is the dimension of the problem (1, 2 or 3)

φ is the current porosity (time t)

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

γ_1 is the plugging coefficient

F_{oi} is the oil Darcy flux

β is the entrainment coefficient

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

U_{cr} is the user input critical velocity.

The overall volume fraction deposit is the sum of the deposits in each direction i :

$$\varepsilon = \sum_{i=1}^d \varepsilon_i \quad \text{Eq. 6.20}$$

Assuming a very small binary interaction between the flocs component and the other components, the volumetric flocs concentration is taken as:

$$C_a = Z_a = n_a / (\sum_{i=1,ncmps} n_i) \quad \text{Eq. 6.21}$$

In the pre-2010.2 version, the flocs mole fraction that deposits is taken as: $n_a^{dp} = \varepsilon n_a^f$, and the flocs mole fraction that remains in solution is corrected to: $n_a^f = n_a^f - n_a^{dp}$.

The concentration of the flowing part of the flocs is calculated as: $C_a = \frac{n_a^f S_o x_a}{b_o}$

where

b_o is the oil molar density,

S_o the oil saturation and

x_a the molar fraction of the flocs in the liquid phase.

In some cases, this approach tends to overestimate the deposition level. Item 242 of the OPTIONS3 keyword can be used to revert to this pre-2010.2 behavior. Note also that item 231 of OPTIONS3 can be used to revert to the pre-2010.1 deposition behavior.

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

Plugging control

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

$$\frac{\partial \varepsilon_i}{\partial t} = \frac{\alpha}{d} \varphi C_a + \gamma_1 |F_{oi}| (C_a - C_{acr})^+ - \beta (|U_{oi}| - U_{cr})^+ \varepsilon_i \quad \text{Eq. 6.22}$$

Basically the coefficient γ is set as:

$$\gamma_1 = \begin{cases} \gamma_1 & \text{if } \varepsilon > \varepsilon_{cr} \text{ and/or } C_a > C_{acr} \\ 0 & \text{if not} \end{cases} \quad \text{Eq. 6.23}$$

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

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

$$r \sim \sqrt{\frac{K}{\phi}} \rightarrow r_{cr} \sim r_0 \left(1 - \frac{\varepsilon_{cr}}{\phi_0}\right)^{\frac{\delta-1}{2}} \quad \text{Eq. 6.24}$$

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

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

Asphaltene damage (both deposition models)

Asphaltene damage represents the effects of asphaltene on the reservoir and reservoir flow.

Porosity

Asphaltene deposition reduces the fluid volume in the reservoir. In the solid deposition model this effect is modeled by the solid saturation. In the pre-2011.1 model the porosity is reduced by asphaltene deposition:

$$\varphi = \varphi_0 - \int_0^t \frac{\partial \varepsilon}{\partial t} dt \quad \text{Eq. 6.25}$$

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

Permeability

In the three-component deposition model, solid is deposited and the effects of deposition on permeability can be modeled by using a solid-dependent mobility multiplier. See keyword [SOLIDMMS](#) or [SOLIDMMC](#).

In the pre-2011.1 model the effects of deposition on permeability can be modeled in one of two ways, using the [ASPKDAM](#) keyword.

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

$$\frac{K}{K_0} = \left(1 - \frac{\varepsilon}{\varphi_0}\right)^\delta \quad \text{Eq. 6.26}$$

where δ is a user input (based on core experiment data).

To use this model the user needs to specify PORS in the second argument of [ASPHALTE](#) keyword, and provide the exponent with the [ASPKDAM](#) keyword.

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

Oil viscosity

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

Using the [ASPVISO](#) keyword in conjunction with item 3 of the [ASPHALTE](#) keyword, the user can model the viscosity alteration as a function of the mass fraction of precipitate in the oil C_p . This includes both the fines and the floc component:

$$C_p = \frac{m_{wp} \cdot x_{pf} + m_{wf} \cdot x_f}{m_{woil}}$$

Three different models are available:

- Generalized Einstein model (one parameter)

$$\frac{\mu}{\mu_0} = 1 + aC_p \quad \text{Eq. 6.27}$$

where the default for a is $a = 2.5$, and μ_0 is the oil viscosity at $C_p = 0$. The user can specify the coefficient a using [ASPVISO](#).

- Krieger and Dougherty model (two parameter)

$$\frac{\mu}{\mu_0} = \left(1 - \frac{C_p}{C_{p0}}\right)^{-\eta} C_{p0} \quad \text{Eq. 6.28}$$

where:

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

C_{p0} is the concentration for maximum packing.

The user can specify the coefficient η and the maximum concentration C_{p0} using [ASPVISO](#).

- Lookup table that gives the oil viscosity multiplier as a function of the mass fraction of the asphaltene precipitate. The first column is the volume fraction and the second column is the viscosity multiplier. The number of rows can be set in item 19 of [TABDIMS](#) keyword.

Depending on the model chosen, the [ASPVISO](#) keyword takes the appropriate number of entries.

Wettability change

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

It is assumed that the wettability can be determined from the water-oil capillary pressure values: if the water-oil capillary pressure is zero or negative, the grid cell is assumed to be oil-wet and there will be no wettability change.

If the grid cell is water-wet, that is it has a positive water/oil capillary pressure, the relative permeabilities (entered using, for example, [SWFN](#) and [SOF3](#)) are combined with the oil-wet saturation functions entered using the [ASPKROW](#) keyword as follows:

1. The weighting factor F is determined as a function of the volume fraction of asphaltene deposit (S_s or ε), provided using the [ASPWETF](#) keyword.

2. The critical oil in water and water saturations are scaled

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

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

where S_{ocra} and S_{ocri} are the critical oil in water saturations from the [ASPKROW](#) table and [SOF3](#) table respectively, and S_{wcra} and S_{wcri} are the critical water saturations from the [ASPKROW](#) table and [SWFN](#) table respectively.

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

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

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

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

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

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

General comments on viscosity behavior

Viscosity behavior

Oil viscosity changes are due to two competing effects:

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

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

List of keywords

RUNSPEC section

- [ASPHALTE](#) activates the asphaltene option. It takes three arguments that select the precipitation model, the permeability damage model and the oil viscosity model.

PROPS section

Restriction	Keyword	Description
	ASPP1P	sets the variable asphaltene precipitation is tabulated against: pressure P, or molar fraction Z, or temperature T.
	ASPREGW	sets the table for the dissolution-precipitation asphaltene model based on percentage molar weight of asphaltene content as a function of one variable: P or T or Z.
	ASPP2P	sets the variables asphaltene precipitation is tabulated against: (P,T) or (P,Z) or (T,Z).
	ASPPW2D	sets the 2D table for the dissolution-precipitation asphaltene model based on percentage molar weight of asphaltene content as a function of two variables: (P,T) or (P,Z) or (T,Z).
Pre-2011.1 model only	ASPFLOC	sets the precipitating components range and the flocs component number.
	ASPFLRT	sets the flocculation-dissociation rates.
	ASPDEPO	sets the deposition parameters.

Restriction	Keyword	Description
Pre-2011.1 model only	ASPKDAM	sets the permeability damage model: either an exponent if the permeability is correlated to the porosity or a table of permeability multiplier as a function of volume fraction of asphaltene deposit
	ASPVISO	sets the oil viscosity model parameters
	ASPLCRT	sets the critical values for the volume fraction of asphaltene deposit and the critical value for flocs concentration for plugging control, and how to use these critical values (AND or OR).
	ASPKROW	sets the water and oil wet relative permeability as a function of water saturation for the wettability change due to asphaltene deposition.
	ASPWETF	sets the wettability weighting factor between the water-wet and oil-wet relative permeability as a function of the volume fraction of asphaltene deposit.
	CATYPE	identifies the components for precipitation, flocculation and deposition. It is a replacement for ASPFLOC

Outputs

The following arguments are associated with the [RPTRST](#) / [RPTSCHED](#) keywords for the ASPHALTENE option.

Restriction	Keyword	Description
	AREAC	Asphaltene reaction rates.
Pre-2011.1 model only.	ASPADS	Asphaltene cumulative adsorption volume fraction. The volume in these reported values is considered to be the grid block volume (bulk volume).
Pre-2011.1 model only.	ASPDOT	Asphaltene cumulative net deposition volume fraction.
Pre-2011.1 model only.	ASPENT	Asphaltene instantaneous entrainment volume fraction.
Pre-2011.1 model only.	ASPFLO	Oil phase velocities used in the entrainment calculations. The mnemonics output to the restart file are VELOILI+, VELOILJ+, and VELOILK+ depending on the dimension of the problem.
Three-component model only.	ASPFLT	Average oil flow rate at the previous timestep.
	ASPFRD	Asphaltene fraction precipitated and dissolved.
Pre-2011.1 model only.	ASPKDM	Asphaltene permeability damage multiplier.
	ASPLIM	Asphaltene precipitate fraction limit.
Pre-2011.1 model only.	ASPLUG	Asphaltene instantaneous plugging volume fraction
	ASPRET	Asphaltene precipitate fraction including flocs
	ASPREW	Asphaltene precipitate fraction
Three-component model only	ASPVEL	Average oil velocity at the previous timestep.
	ASPVOM	Asphaltene oil viscosity multiplier

For information on the SUMMARY keywords refer to [Asphaltene SUMMARY output controls](#) in the *ECLIPSE Reference Manual*.

In pre-2011.1 versions of the simulator, the asphaltene flocculation rates could be reported using the ASPFL/BASPFL keywords. These keywords are no longer available - instead use the AREAC/BAREAC keywords.

Asphaltene reaction number

The BAREAC summary keyword is followed by a number of records, each containing four values, the I, J, K values of the cell block and the reaction number:

```
BAREAC
1 1 1 1 /
1 1 1 2 /
/
```

Pre-2011.1 model

In the pre-2011.1 model, pairs of reactions are used to model flocculation and dissociation. The reaction numbers have the following meanings:

- 2N-1 the rate of flocculation for the Nth precipitating component
- 2N the rate of dissociation for the Nth precipitating component

Three-component model

In the three-component model, pairs of reactions are used to model flocculation and dissociation and optionally deposition and entrainment. Only one component is allowed to precipitate so the reaction numbers have the following meanings:

1. the rate of flocculation
2. the rate of dissociation
3. the rate of deposition (if used)
4. the rate of entrainment (if used)

Example

In the RUNSPEC section, switch on asphaltene deposition model and the solid model (if required):

```
ASPHALTE
WEIGHT /
SOLID
```

In the PROPS section, specify the hydrocarbon and asphaltene components. For example if components 6, 7 and 8 are the precipitation, floc and deposition components:

```
CNAMES
C1      C3      C6      C10     C15      C20      ASPH      DEPO  /
-- Asphaltene component type
CATYPE
 5*      PREC      FLOC      DEP /
-- Make deposition component a solid
CVTYPE
 7*      SOLID /
-- ... define asphaltene concentration limits
ASPREWG
-- pres    %_wt
 1000.0    30.0
```

```

10000.0    100.0 /
-- ... asphaltene floc rates
ASPFLRT
-- C20 <->ASPH
 0.001
 0.0001 /
-- ... asphaltene deposition
ASPDEPO
-- adsorp plug entrain Vcr
 5.0E-6 0.0 0.00 1.0E6 /
-- ... asphaltene damage ratio
SOLIDMMS
-- deposit mult
 0.0    1.0
 1.0    0.5 /
-- ... asphaltene viscosity change
ASPVISO
-- vfrac mult
 0.00  1.00
 1.00  1.00 /

```

Converting a pre-2011.1 model

Pre-2011.1 asphaltene data sets can be converted to the three-component solid deposition model. Assuming the pre-2011.1 model has only one precipitating component, the conversion is straightforward:

1. Add the `SOLID` keyword in the `RUNSPEC` section.
2. Increase the number of components by one in the `COMP` keyword.
3. Add the `CVTYPE` keyword in the `PROPS` section and define the new last component as a solid.
4. Add the `CATYPE` keyword in the `PROPS` section and define the precipitation, flocculation and deposition components. The precipitation and flocculation components are the same as those defined in the `ASPFLOC` keyword, and the deposition component is the new solid component. Remove the `ASPFLOC` keyword.
5. Add properties for the new solid component. The fluid properties can be copied from the flocculation component, although many of these will not be used since the solid component will not be considered in the flash. The deposition component molecular weight `MW` should be the same as the flocculation component molecular weight if mass is to be conserved. Solid properties for the deposition component can be defined using the appropriate keywords. See "[Solid keywords](#)". The solid component density `SDREF` is important as it will affect the volume balance and hence the cell pressure as asphaltene is deposited.
6. The two models use different plugging coefficients. The pre-2011.1 model uses γ_1 , specified by item 2 of the `ASPDEPO` keyword, while the three-component model uses γ_2 , specified by item 5 of the same keyword. The coefficient can be converted using $\gamma_2 = A \cdot \gamma_1$ where A is an average cell face area.
7. The two models use different measures for asphaltene deposit. The pre-2011.1 model uses the volume fraction in the cell ε , whereas the solid model uses the solid saturation S_s , that is the volume fraction in the pore $S_s = \varepsilon / \phi$. The ε values used in keywords must be converted to solid saturations:
 - replace ε_{cr} with S_{cr} in argument 1 of keyword `ASPLCRT`.
 - replace ε values with S_s values in the first column of `ASPWETF`, if the keyword is used.
8. Replace the `ASPKDAM` keyword, if used, with the `SOLIDMMS` keyword. If the `ASPKDAM` data is supplied as a table then the same data can be used for `SOLIDMMS`, except the first column must be

converted from ε values to S_s values. If the ASPKDAM data is supplied as an exponent δ then an equivalent SOLIDMMMS table must be created with columns S_s and $(1-S_s)^\delta$.

9. Replace the pre-2011.1 summary keywords.

Pre-2011.1 model	Three-component model
ASPADS/ BASPADSASPLUG/ BASPLUG	Adsorption and plugging are considered together as deposition. The volume fraction of adsorbed and plugging asphaltene are not calculated. Instead the rate of deposition can be found from the third reaction reported by AREAC/ BAREAC.
ASPDOT/BASPDOT	Use the solid keywords to report the amount of deposition SSOLID/ BSSOLID.
ASPENT /BASPENT	The volume fraction of entrained asphaltene is not calculated. Instead the rate of deposition can be found from the fourth reaction reported by AREAC/ BAREAC.
ASPFL0	Use keyword ASPFLT to report the flow rate and ASPVEL to report the average velocity. In addition the summary vectors BASPFLT and BASPVEL can be used.
ASPKDM/BASPKDM	Use the solid keywords to report the change in mobility that is due to asphaltene damage: SMMULT/BSMMULT.

Table 6.1: Summary keywords

7

Brine tracking

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

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

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

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

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

The Brine Tracking facility is activated by the keyword [BRINE](#) in the RUNSPEC section, and the low salinity option is activated by the [LOWSALT](#) in the RUNSPEC as well.

ECLIPSE 100 only

The multi-component brine option is activated by the keyword [BRINE](#) in conjunction with the keyword [ECLMC](#) in the RUNSPEC section.

The Brine Model

Brine conservation equation

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

$$\frac{d}{dt} \left(\frac{VS_w C_s}{B_w} \right) = \Sigma \left[\frac{Tk_{rw}}{B_w \mu_s \text{eff}} (\delta P_w - \rho_w g D_z) \right] C_s + Q_w C_s \quad \text{Eq. 7.1}$$

where

ρ_w denotes the water density

Σ denotes the sum over neighboring cells

C_s denotes the salt concentration in the aqueous phase

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

D_z is the cell center depth.

B_w is the water formation volume

T is the transmissibility

k_{rw} is the water relative permeability

S_w is the water saturation

V is the block pore volume

Q_w is the water production rate

P_w is the water pressure

g is the gravity acceleration

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

Multi-component brine model

ECLIPSE 100 only

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

Other salinity effects (for example, the low salinity option discussed below) do not use the multi-component brine model, and they are not active when the multi-component brine model is active. Certain combinations are explicitly banned; in particular, the `SALTNODE` keyword and other keywords depending on it will not work with `ECLMC`.

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

$$\frac{d}{dt} \left(\frac{VS_w(C_{++} + C_{++}^S)}{B_w} + VC_{++}^r \right) = \Sigma \left[\frac{Tk_{rw}}{B_w \mu_w} (\delta P_w - \rho_w g D_z) \right] C_{++} + Q_w C_{++} \quad \text{Eq. 7.2}$$

$$\frac{d}{dt} \left(\frac{VS_w C_-}{B_w} \right) = \Sigma \left[\frac{Tk_{rw}}{B_w \mu_w} (\delta P_w - \rho_w g D_z) \right] C_- + Q_w C_- \quad \text{Eq. 7.3}$$

$$C_+ = C_- - C_{++} \quad \text{Eq. 7.4}$$

where

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

C_+ denotes the equivalent molar concentration of monovalent cations

C_- denotes the equivalent molar concentration of anions

C_{++}^s denotes the concentration of divalent cations associated with surfactant

C_{++}^r denotes the concentration of divalent cations associated with rock

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

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

Salt name	Chemical formula	Molecular weight [kg/kg-M]
Sodium Chloride	NaCl	58.44
Potassium Chloride	KCl	74.55
Calcium Chloride	CaCl ₂	110.98
Magnesium Chloride	MgCl ₂	95.21
Sodium Carbonate	Na ₂ CO ₃	105.99
Potassium Carbonate	K ₂ CO ₃	138.20
Calcium Carbonate	CaCO ₃	100.09
Magnesium Carbonate	MgCO ₃	84.31
Sodium Sulfate	Na ₂ SO ₄	142.04
Potassium Sulfate	K ₂ SO ₄	166.97
Calcium Sulfate	CaSO ₄	136.14
Magnesium Sulfate	MgSO ₄	120.37

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

Cation exchange

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

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



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

$$\frac{(C_{++})(C_+^r)^2}{(C_{++})(C_+)^2} = \beta^r q^r, \quad \text{Eq. 7.7}$$

$$\frac{(C_{++})(C_+^s)^2}{(C_{++})(C_+)^2} = \beta^s C^s, \quad \text{Eq. 7.8}$$

where

q^r denotes the rock cation exchange capacity

C^s denotes the equivalent molar concentration of surfactant

β^r denotes the cation exchange constant for rock

β^s denotes the cation exchange constant for surfactant

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

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

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

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

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

where

C_{++}^{inj} is the maximum injection concentration and

`THIONX` is a relative threshold concentration, which can be set using the `TUNING` keyword. The default value for `THIONX` is 10^{-2} .

The PVT properties of brine

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

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

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

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

ECLIPSE 100

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

$$B_w(P, C_s) = \frac{B_w(P_{ref}, C_s)}{1 + X + (X^2/2)} \quad \text{Eq. 7.9}$$

where $X = C(P - P_{ref})$

$$(B_w(P, C_s)\mu_w)(P, C_s) = \frac{B_w(P_{ref}, C_s)\mu_w(P_{ref}, C_s)}{1 + Y + (Y^2/2)} \quad \text{Eq. 7.10}$$

where $Y = (C - C_v)(P - P_{ref})$

ECLIPSE 300

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

$$\mu_w(P, C_s) = \frac{\mu_w(P_{ref}, C_s)}{1 + Y + (Y^2/2)} \quad \text{Eq. 7.11}$$

where $Y = -C_v(P - P_{ref})$

Low salinity option

ECLIPSE 100 only

This option provides a mechanism for modeling the salinity dependence of the oil and water relative permeabilities and the water-oil capillary pressure as functions of the salt concentration. The gas relative permeability and the gas-oil capillary pressure are assumed to be independent of salinity. This option is selected by specifying the [LOWSLT](#) keyword in the RUNSPEC section. The [SATNUM](#) and [LWSLTNUM](#) keywords are used to define the high and low salinity saturation regions respectively. (The [LSNUM](#) keyword may be used as an abbreviated alias for the [LWSLTNUM](#) keyword.)

The salinity dependence is modeled as a two stage process. Firstly, the oil and water saturation (horizontal) end-points are interpolated between the high and low table values. Secondly, this set of interpolated saturation end-points is used to look up the high and low salinity saturation functions in the high and low salinity saturation tables using the associated high and low relative permeability and capillary pressure (vertical) end-points. The resultant looked-up values of high and low salinity saturation functions are then combined using a further interpolation. Therefore, the saturation function table lookups are performed using scaled saturation end-points; that is, two-point, horizontal end-point scaling is applied even when end-point scaling has not been specified via the [ENDSCALE](#) keyword.

Given two sets of saturation functions, one for high salinity and one for low salinity, the oil and water table saturation end points are interpolated firstly according to:

$$\begin{aligned} S_{wco}^i &= F_1 S_{wco}^l + (1-F_1) S_{wco}^h \\ S_{wcr}^i &= F_1 S_{wcr}^l + (1-F_1) S_{wcr}^h \\ S_{wmax}^i &= F_1 S_{wmax}^l + (1-F_1) S_{wmax}^h \\ S_{owcr}^i &= F_1 S_{owcr}^l + (1-F_1) S_{owcr}^h \\ S_{ogcr}^i &= F_1 S_{ogcr}^l + (1-F_1) S_{ogcr}^h \end{aligned} \quad \text{Eq. 7.12}$$

where

F_1 is a weighting function which is tabulated as a function of the salt concentration and is defined in the second column of the [LSALTFNC](#) keyword.

S_{wco}	is the saturation table derived connate water saturation.
S_{wcr}	is the saturation table derived critical water saturation.
S_{wmax}	is the saturation table derived maximum water saturation.
S_{owcr}	is the saturation table derived critical oil saturation in water.
S_{ogcr}	is the saturation table derived critical oil saturation in gas.

superscripts h, l, i denote the high, low and interpolated table saturation end-points respectively.

Lower case subscripts and superscripts are used here to signify the saturation table derived end-points.
Upper case versions are used subsequently to signify scaled end-points.

The high and low water-oil capillary pressures and water and oil relative permeabilities are looked up in the high and low salinity saturation tables by applying two-point saturation (horizontal) end-point scaling:

$$\begin{aligned}
 P_{cow}^h &= f(S_w, S_{wco}^i, S_{wmax}^i, P_{cowmax}^h) \\
 P_{cow}^l &= f(S_w, S_{wco}^i, S_{wmax}^i, P_{cowmax}^l) \\
 k_{rw}^h &= f(S_w, S_{wcr}^i, S_{wmax}^i, k_{rwmax}^h) \\
 k_{rw}^l &= f(S_w, S_{wcr}^i, S_{wmax}^i, k_{rwmax}^l) \\
 k_{rog}^h &= f(S_{og}, S_{ogcr}^i, S_{omax}^i, k_{romax}^h) \\
 k_{rog}^l &= f(S_{og}, S_{ogcr}^i, S_{omax}^i, k_{romax}^l) \\
 k_{row}^h &= f(S_{ow}, S_{owcr}^i, S_{omax}^i, k_{romax}^h) \\
 k_{row}^l &= f(S_{ow}, S_{owcr}^i, S_{omax}^i, k_{romax}^l)
 \end{aligned} \tag{Eq. 7.13}$$

where

f	signifies the end-point scaling operation.
S_w	is the grid block water saturation.
S_{ow}	is the oil saturation for oil relative permeability to water.
S_{og}	is the oil saturation for oil relative permeability to gas.
S_{omax}	is the maximum oil saturation., $S_{omax} = (1 - S_{wco}^i - S_{gco}^h)$
S_{gco}	is the saturation table derived connate gas saturation.
k_{rwmax}	is the saturation table derived maximum water relative permeability.
k_{romax}	is the saturation table derived maximum oil relative permeability.
P_{cowmax}	is the saturation table derived maximum oil-water capillary pressure.

For the default three-phase oil relative permeability model, the oil saturations used to look up the oil relative permeabilities are:

$$\begin{aligned} S_{ow} &= (1 - S_w - S_g) \\ S_{og} &= (1 - S_w - S_g) \end{aligned} \quad \text{Eq. 7.14}$$

For the Stone 1 and Stone 2 three-phase oil relative permeability models, the oil saturations used to look up the oil relative permeabilities are:

$$\begin{aligned} S_{ow} &= (1 - S_w) \\ S_{og} &= (1 - S_g - S_{wco}^i) \end{aligned} \quad \text{Eq. 7.15}$$

Finally, the relative permeabilities and capillary pressure are interpolated according to:

$$\begin{aligned} k_{rw}^i &= F_1 k_{rw}^l + (1-F_1)k_{rw}^h \\ k_{row}^i &= F_1 k_{row}^l + (1-F_1)k_{row}^h \\ k_{rog}^i &= F_1 k_{rog}^l + (1-F_1)k_{rog}^h \\ P_{cow}^i &= F_2 P_{cow}^l + (1-F_2)P_{cow}^h \end{aligned} \quad \text{Eq. 7.16}$$

where

S_g is the grid block gas saturation.

F_2 is a weighting function which is tabulated as a function of the salt concentration and is defined in the third column of the LSALTFNC keyword. (Note that this defaults to the second column if the third column is defaulted.)

The gas relative permeability and gas-oil capillary pressure are assumed to be independent of salinity and are looked up directly in the high salinity gas saturation table. If the current grid block gas saturation exceeds the maximum gas saturation defined in the high salinity table, the values of gas relative permeability and gas-oil capillary pressure are assumed to remain constant at their corresponding maximum values.

Low salinity option with end-point scaling

End-point scaling (which is specified using the [ENDSCALE](#) keyword) may be applied to both the high and low salinity saturation functions.

For high salinity end-point scaling, the set of keywords [SWL](#), [SWLPC](#), [SWCR](#), [SWU](#), [SGL](#), [SGLPC](#), [SGCR](#), [SGU](#), [SOWCR](#), [SOGCR](#), [KRW](#), [KRWR](#), [KRG](#), [KRGR](#), [KRO](#), [KRORW](#), [KRORG](#), [PCOW](#) and [PCOG](#) is available and applies to the tables defined by [SATNUM](#). In addition, for the high salinity tables, the scaled end-points may also be specified as a function of depth using the keywords [ENPTVD](#), [ENKRVD](#) and [ENPCVD](#).

For the low salinity end-point scaling, the reduced set of keywords [LSWL](#), [LSWLPC](#), [LSWCR](#), [LSWU](#), [LSOWCR](#), [LSOGCR](#), [LKRW](#), [LKWR](#), [LKRO](#), [LKROW](#), [LKROG](#) and [LPCW](#) is available and applies to the set of tables defined by [LWSLTNUM](#) (or [LSNUM](#)). The low salinity set corresponds to the high salinity set of keywords, except that the end-points associated with scaling the gas relative permeability and the gas-oil capillary pressure are omitted because these are assumed to be independent of salinity.

When end-point scaling has been specified, the salinity dependence is modeled in a similar manner to that described above except that the table derived end-points are replaced by the values from their

corresponding end-point scaling arrays. (Where the end-point scaling arrays have been omitted from the input data, the end-point array values are defaulted to their corresponding table end-point values).

Given two sets of saturation functions, one for high salinity and one for low salinity, the scaled oil and water saturation end-points are interpolated firstly according to:

$$\begin{aligned} S_{WL}^I &= F_1 S_{WL}^L + (1-F_1) S_{WL}^H \\ S_{WLPC}^I &= F_1 S_{WLPC}^L + (1-F_1) S_{WLPC}^H \\ S_{WCR}^I &= F_1 S_{WCR}^L + (1-F_1) S_{WCR}^H \\ S_{WU}^I &= F_1 S_{WU}^L + (1-F_1) S_{WU}^H \\ S_{OWCR}^I &= F_1 S_{OWCR}^L + (1-F_1) S_{OWCR}^H \\ S_{OGCR}^I &= F_1 S_{OGCR}^L + (1-F_1) S_{OGCR}^H \end{aligned} \quad \text{Eq. 7.17}$$

where

F_1 is a weighting function which is tabulated as a function of the salt concentration, and is defined in the second column of the [LSALTFNC](#) keyword.

S_{WL} is the scaled minimum water saturation for relative permeability.

S_{WLPC} is the scaled minimum water saturation for oil-water capillary pressure.

S_{WCR} is the scaled critical water saturation.

S_{WU} is the scaled maximum water saturation.

S_{OWCR} is the scaled critical oil saturation in water.

S_{OGCR} is the scaled critical oil saturation in gas.

superscripts H , L , I denote the high, low and interpolated scaled saturation end-points respectively.

Lower and upper case subscripts and superscripts are used here to signify the table derived and the scaled end-points respectively.

The capillary pressure and relative permeabilities are looked up in the high and low salinity saturation tables applying the set of scaled end-points shown below. For capillary pressure, only two-point scaling is employed:

$$\begin{aligned} P_{COW}^H &= f(S_W, S_{WLPC}^I, S_{WU}^I, P_{CW}^H) \\ P_{COW}^L &= f(S_W, S_{WLPC}^I, S_{WU}^I, P_{CW}^L) \end{aligned} \quad \text{Eq. 7.18}$$

For the relative permeabilities, two-point or three-point end-point scaling is employed according to whether item 1 of the [SCALECRS](#) keyword has been set to NO or YES respectively. (Two-point scaling is selected by default if this keyword is omitted.) For two-point scaling:

$$\begin{aligned}
K_{RW}^H &= f(S_W, S_{WCR}^I, S_{owcr}^h, S_{WU}^I, K_{RW}^H, K_{RWR}^H) \\
K_{RW}^L &= f(S_W, S_{WCR}^I, S_{owcr}^l, S_{WU}^I, K_{RW}^L, K_{RWR}^L) \\
K_{ROG}^H &= f(S_{OG}, S_{OGCR}^I, S_{gcr}^h, S_{OMAX}, K_{RO}^H, K_{RORG}^H) \\
K_{ROG}^L &= f(S_{OG}, S_{OGCR}^I, S_{gcr}^h, S_{OMAX}, K_{RO}^L, K_{RORG}^L) \\
K_{ROW}^H &= f(S_{OW}, S_{OWCR}^I, S_{wcr}^h, S_{OMAX}, K_{RO}^H, K_{RORW}^H) \\
K_{ROW}^L &= f(S_{OW}, S_{OWCR}^I, S_{wcr}^l, S_{OMAX}, K_{RO}^L, K_{RORW}^L)
\end{aligned} \tag{Eq. 7.19}$$

For three-point scaling, the scaled critical water saturation end-point is taken from the interpolated value or the respective high and low salinity values according to whether the default or the Stone 1 or 2 three-phase oil relative permeability models has been selected:

$$\begin{aligned}
K_{RW}^H &= f(S_W, S_{WCR}^I, S_{OWCR}^I, S_{WU}^I, K_{RW}^H, K_{RWR}^H) \\
K_{RW}^L &= f(S_W, S_{WCR}^I, S_{OWCR}^I, S_{WU}^I, K_{RW}^L, K_{RWR}^L) \\
K_{ROG}^H &= f(S_{OG}, S_{OGCR}^I, S_{GCR}^H, S_{OMAX}, K_{RO}^H, K_{RORG}^H) \\
K_{ROG}^L &= f(S_{OG}, S_{OGCR}^I, S_{GCR}^H, S_{OMAX}, K_{RO}^L, K_{RORG}^L)
\end{aligned} \tag{Eq. 7.20}$$

Default three-phase oil relative permeability model:

$$\begin{aligned}
K_{ROW}^H &= f(S_{OW}, S_{OWCR}^I, S_{WCR}^I, S_{OMAX}, K_{RO}^H, K_{RORW}^H) \\
K_{ROW}^L &= f(S_{OW}, S_{OWCR}^I, S_{WCR}^I, S_{OMAX}, K_{RO}^L, K_{RORW}^L)
\end{aligned} \tag{Eq. 7.21}$$

Stone 1 and 2 three-phase oil relative permeability models:

$$\begin{aligned}
K_{ROW}^H &= f(S_{OW}, S_{OWCR}^I, S_{WCR}^H, S_{OMAX}, K_{RO}^H, K_{RORW}^H) \\
K_{ROW}^L &= f(S_{OW}, S_{OWCR}^I, S_{WCR}^L, S_{OMAX}, K_{RO}^L, K_{RORW}^L)
\end{aligned} \tag{Eq. 7.22}$$

where

- f signifies the end-point scaling operation.
- S_W is the grid block water saturation.
- S_{OW} is the oil saturation for oil relative permeability to water.
- S_{OG} is the oil saturation for oil relative permeability to gas.
- S_{wcr} is the saturation table derived critical water saturation.
- S_{owcr} is the saturation table derived critical oil in water saturation.
- S_{OMAX} is the scaled maximum oil saturation, $S_{OMAX} = (1 - S_{WL}^I - S_{GL}^H)$.
- S_{GL} is the scaled minimum gas saturation.
- S_{GCR} is the scaled critical gas saturation.

- S_{gcr} is the saturation table derived critical gas saturation.
- K_{RW} is the scaled maximum water relative permeability.
- K_{RWR} is the scaled water relative permeability at residual oil saturation.
- K_{RO} is scaled maximum oil relative permeability.
- K_{ROrg} is the scaled oil relative permeability at residual gas saturation.
- K_{ROW} is the scaled oil relative permeability at residual water saturation.
- P_{CW} is the scaled maximum oil-water capillary pressure.

Noting that:

1. For the scaling of both the high and low salinity saturation tables, specifically, for the scaling of the relative permeabilities of oil to gas and water, only the high salinity minimum and critical scaled gas saturations are used.
2. For two-point scaling, the positioning of the relative permeability at residual saturation of the displacing phase is determined by the two-point scaled table value of the displacing critical phase saturation whereas, for three-point scaling, this is determined by the scaled end-point of the displacing phase critical saturation.
3. For the default three-phase oil relative permeability model, the critical water saturation used in the scaling of the high and low salinity oil relative permeabilities to water is based upon the interpolated value, whereas for the Stone 1 and 2 three-phase oil relative permeability models, the scaling is based upon the respective high and low salinity values.

For the default three-phase oil relative permeability model, the oil saturations used to look up the oil relative permeabilities are:

$$\begin{aligned} S_{OW} &= (1 - S_W - S_G) \\ S_{OG} &= (1 - S_W - S_G) \end{aligned} \quad \text{Eq. 7.23}$$

For the Stone 1 and Stone 2 three-phase oil relative permeability models, the oil saturations used to look up the oil relative permeabilities are:

$$\begin{aligned} S_{OW} &= (1 - S_W) \\ S_{OG} &= (1 - S_G - S_{WCO}) \end{aligned} \quad \text{Eq. 7.24}$$

Finally, the relative permeabilities and capillary pressure are interpolated according to:

$$\begin{aligned} K_{RW}^I &= F_1 K_{RW}^L + (1-F_1) K_{RW}^H \\ K_{ROW}^I &= F_1 K_{ROW}^L + (1-F_1) K_{ROW}^H \\ K_{ROG}^I &= F_1 K_{ROG}^L + (1-F_1) K_{ROG}^H \\ P_{COW}^I &= F_2 P_{COW}^L + (1-F_2) P_{COW}^H \end{aligned} \quad \text{Eq. 7.25}$$

where

F_1 is a weighting function which is tabulated as a function of the salt concentration, and is defined in the second column of the `LSALTFNC` keyword.

F_2 is a weighting function which is tabulated as a function of the salt concentration, and is defined in the third column of the `LSALTFNC` keyword. (Note that this defaults to the second column if not specified.)

The gas-oil capillary pressure and gas relative permeability are assumed to be independent of salinity. For capillary pressure two-point scaling is employed:

$$P_{COG} = f(S_G, S_{GLPC}^H, S_{GU}^H, P_{CG}^H) \quad \text{Eq. 7.26}$$

For gas relative permeability, depending upon the value of item 1 of the `SCALECRS` keyword, either two-point scaling:

$$K_{RG} = f(S_G, S_{GCR}^H, S_{ogcr}^h, S_{GU}^H, K_{RG}^H, K_{RGR}^H) \quad \text{Eq. 7.27}$$

or three-point scaling is employed:

$$K_{RG} = f(S_G, S_{GCR}^H, S_{OGCR}^H, S_{GU}^H, K_{RG}^H, K_{RGR}^H) \quad \text{Eq. 7.28}$$

where

S_G is the grid block gas saturation.

S_{GLPC} is the scaled minimum gas saturation for capillary pressure.

S_{GU} is the scaled maximum gas saturation.

S_{ogcr} is the saturation table derived critical oil saturation in gas.

P_{CG} is the scaled maximum oil-gas capillary pressure.

K_{RG} is the scaled maximum gas relative permeability.

K_{RGR} is the scaled gas relative permeability at residual oil saturation.

Hence only the high salinity table and high salinity scaled end-points are used for modeling the gas saturation functions. If the current grid block gas saturation exceeds the maximum high salinity scaled gas saturation, the values of gas relative permeability and gas-oil capillary pressure are assumed to remain constant at their corresponding maximum values.

Low salinity and surfactant options

If, in addition to the `LOWSLT` option, the `SURFACT` option has been specified, the high and low salinity immiscible saturation functions are combined in a similar manner to that described above and then combined with the miscible saturation functions. The miscible saturation functions are assumed to be independent of salinity. The `SURFNUM` keyword is used to define the miscible saturation regions and the `SATNUM` and `LWSLTNUM` (or `LSNUM`) keywords are used to define the immiscible high and low salinity saturation regions respectively. In this case, however, the initial interpolation of the saturation (horizontal) end-points and the subsequent interpolation of the vertical end-points following the table look-up involve a two stage process:

Given two sets of immiscible saturation functions, one for high salinity and one for low salinity, the immiscible oil and water table saturation end-points are interpolated firstly according to:

$$\begin{aligned} S_{wco}^{imm} &= F_1 S_{wco}^l + (1-F_1) S_{wco}^h \\ S_{wcr}^{imm} &= F_1 S_{wcr}^l + (1-F_1) S_{wcr}^h \\ S_{wmax}^{imm} &= F_1 S_{wmax}^l + (1-F_1) S_{wmax}^h \\ S_{owcr}^{imm} &= F_1 S_{owcr}^l + (1-F_1) S_{owcr}^h \\ S_{ogcr}^{imm} &= F_1 S_{ogcr}^l + (1-F_1) S_{ogcr}^h \end{aligned} \quad \text{Eq. 7.29}$$

and these are then combined with the miscible saturation end-points according to:

$$\begin{aligned} S_{wco}^i &= F_3 S_{wco}^{mis} + (1-F_3) S_{wco}^{imm} \\ S_{wcr}^i &= F_3 S_{wcr}^{mis} + (1-F_3) S_{wcr}^{imm} \\ S_{wmax}^i &= F_3 S_{wmax}^{mis} + (1-F_3) S_{wmax}^{imm} \\ S_{owcr}^i &= F_3 S_{owcr}^{mis} + (1-F_3) S_{owcr}^{imm} \\ S_{ogcr}^i &= F_3 S_{ogcr}^{mis} + (1-F_3) S_{ogcr}^{imm} \end{aligned} \quad \text{Eq. 7.30}$$

where

F_3 is tabulated as a function of the capillary number (expressed in terms of its logarithm base ten), and corresponds to the second column of the [SURFCAPD](#) keyword.

superscripts h, l denote the immiscible high and low table saturation end-points respectively.

superscripts imm, mis denote the immiscible interpolated and miscible table saturation end-points respectively.

superscript i denotes the interpolation between immiscible and miscible table saturation end-points.

The immiscible high and low salinity capillary pressures and relative permeabilities are looked up in the high and low salinity saturation tables by applying two-point saturation (horizontal) end-point scaling using the interpolated saturation end-points:

$$\begin{aligned}
P_{cow}^h &= f(S_w, S_{wco}^i, S_{wmax}^i, P_{cowmax}^h) \\
P_{cow}^l &= f(S_w, S_{wco}^i, S_{wmax}^i, P_{cowmax}^l) \\
k_{rw}^h &= f(S_w, S_{wcr}^i, S_{wmax}^i, k_{rwmax}^h) \\
k_{rw}^l &= f(S_w, S_{wcr}^i, S_{wmax}^i, k_{rwmax}^l) \\
k_{rog}^h &= f(S_{og}, S_{ogcr}^i, S_{omax}, k_{romax}^h) \\
k_{rog}^l &= f(S_{og}, S_{ogcr}^i, S_{omax}, k_{romax}^l) \\
k_{row}^h &= f(S_{ow}, S_{owcr}^i, S_{omax}, k_{romax}^h) \\
k_{row}^l &= f(S_{ow}, S_{owcr}^i, S_{omax}, k_{romax}^l)
\end{aligned} \tag{Eq. 7.31}$$

The miscible relative permeabilities are looked up in the miscible tables by applying two-point saturation (horizontal) end-point scaling using the interpolated saturation end-points:

$$\begin{aligned}
k_{rw}^{mis} &= f(S_w, S_{wcr}^i, S_{wmax}^i, k_{rwmax}^{mis}) \\
k_{rog}^{mis} &= f(S_{og}, S_{ogcr}^i, S_{omax}, k_{romax}^{mis}) \\
k_{row}^{mis} &= f(S_{ow}, S_{owcr}^i, S_{omax}, k_{romax}^{mis})
\end{aligned} \tag{Eq. 7.32}$$

The immiscible high and low salinity relative permeabilities and capillary pressure are interpolated according to:

$$\begin{aligned}
k_{rw}^{imm} &= F_1 k_{rw}^l + (1-F_1)k_{rw}^h \\
k_{row}^{imm} &= F_1 k_{row}^l + (1-F_1)k_{row}^h \\
k_{rog}^{imm} &= F_1 k_{rog}^l + (1-F_1)k_{rog}^h \\
P_{cow}^{imm} &= F_2 P_{cow}^l + (1-F_2)P_{cow}^h
\end{aligned} \tag{Eq. 7.33}$$

The oil-water capillary pressure is taken directly from the interpolated immiscible term:

$$P_{cow}^i = P_{cow}^{imm} \tag{Eq. 7.34}$$

The miscible and immiscible water relative permeability and oil relative permeability to water are interpolated according to:

$$\begin{aligned}
k_{rw}^i &= F_3 k_{rw}^{mis} + (1-F_3)k_{rw}^{imm} \\
k_{row}^i &= F_3 k_{row}^{mis} + (1-F_3)k_{row}^{imm}
\end{aligned} \tag{Eq. 7.35}$$

For the default three-phase oil relative permeability model, the oil relative permeability to gas is interpolated between the miscible and the immiscible values:

$$k_{rog}^i = F_3 k_{rog}^{mis} + (1-F_3)k_{rog}^{imm} \tag{Eq. 7.36}$$

whereas, for the Stone 1 and 2 three-phase oil relative permeability models, this is taken directly from the immiscible value:

$$k_{rog}^i = k_{rog}^{imm} \tag{Eq. 7.37}$$

The gas saturation functions are looked up in the high salinity immiscible table; that is, the gas relative permeability and gas-oil capillary pressure are assumed to exhibit no dependence upon salinity or surfactant.

When end-point scaling is specified via the `ENDSCALE` keyword, the interpolations follow a similar pattern to that described above in the absence of end-point scaling but noting that no end-point scaling is applied to the miscible saturation tables. Hence given two sets of immiscible saturation functions, one for low salinity and one for high salinity, the scaled saturation end points are interpolated firstly according to:

$$\begin{aligned} S_{WL}^{IMM} &= F_1 S_{WL}^L + (1 - F_1) S_{WL}^H \\ S_{WLPC}^{IMM} &= F_1 S_{WLPC}^L + (1 - F_1) S_{WLPC}^H \\ S_{WCR}^{IMM} &= F_1 S_{WCR}^L + (1 - F_1) S_{WCR}^H \\ S_{WU}^{IMM} &= F_1 S_{WU}^L + (1 - F_1) S_{WU}^H \\ S_{OWCR}^{IMM} &= F_1 S_{OWCR}^L + (1 - F_1) S_{OWCR}^H \\ S_{OGCR}^{IMM} &= F_1 S_{OGCR}^L + (1 - F_1) S_{OGCR}^H \end{aligned} \quad \text{Eq. 7.38}$$

and these are then combined with the miscible table saturation end-points according to:

$$\begin{aligned} S_{WL}^I &= F_3 S_{wco}^{mis} + (1 - F_3) S_{WL}^{IMM} \\ S_{WLPC}^I &= F_3 S_{wco}^{mis} + (1 - F_3) S_{WLPC}^{IMM} \\ S_{WCR}^I &= F_3 S_{wcr}^{mis} + (1 - F_3) S_{WCR}^{IMM} \\ S_{WU}^I &= F_3 S_{wmax}^{mis} + (1 - F_3) S_{WU}^{IMM} \\ S_{OWCR}^I &= F_3 S_{ower}^{mis} + (1 - F_3) S_{OWCR}^{IMM} \\ S_{OGCR}^I &= F_3 S_{ogcr}^{mis} + (1 - F_3) S_{OGCR}^{IMM} \end{aligned} \quad \text{Eq. 7.39}$$

where

superscripts ,*H*, *L* denote the immiscible high and low scaled saturation end-points respectively.

superscripts *IMM*, *MIS* denote the immiscible interpolated and miscible scaled saturation end-points respectively.

superscript *I* denotes the interpolation between immiscible scaled and miscible saturation end-points.

The capillary pressures and relative permeabilities are looked up in the high and low salinity saturation tables applying the set of scaled end-points shown below. For capillary pressure, only two-point scaling is employed:

$$\begin{aligned} P_{COW}^H &= f(S_W, S_{WLPC}^I, S_{WU}^I, P_{CW}^H) \\ P_{COW}^L &= f(S_W, S_{WLPC}^I, S_{WU}^I, P_{CW}^L) \end{aligned} \quad \text{Eq. 7.40}$$

For the immiscible relative permeabilities, either two-point or three-point end-point scaling is employed according to whether item 1 of the `SCALECRS` keyword has been set to NO or YES respectively. For two-point scaling the immiscible saturation functions are looked up in the high and low salinity saturation tables applying the set of scaled end-points shown below:

$$\begin{aligned}
K_{RW}^H &= f(S_W, S_{WCR}^I, S_{owcr}^h, S_{WU}^I, K_{RW}^H, K_{RWR}^H) \\
K_{RW}^L &= f(S_W, S_{WCR}^I, S_{owcr}^l, S_{WU}^I, K_{RW}^L, K_{RWR}^L) \\
K_{ROG}^H &= f(S_{OG}, S_{OGCR}^I, S_{gcr}^h, S_{OMAX}, K_{RO}^H, K_{RORG}^H) \\
K_{ROG}^L &= f(S_{OG}, S_{OGCR}^I, S_{gcr}^h, S_{OMAX}, K_{RO}^L, K_{RORG}^L) \\
K_{ROW}^H &= f(S_{OW}, S_{OWCR}^I, S_{wcr}^h, S_{OMAX}, K_{RO}^H, K_{RORW}^H) \\
K_{ROW}^L &= f(S_{OW}, S_{OWCR}^I, S_{wcr}^l, S_{OMAX}, K_{RO}^L, K_{RORW}^L)
\end{aligned} \tag{Eq. 7.41}$$

For three-point scaling the immiscible water relative permeability and the oil relative permeability to gas are looked up in the high and low salinity saturation tables applying the set of scaled end-points shown below:

$$\begin{aligned}
K_{RW}^H &= f(S_W, S_{WCR}^I, S_{OWCR}^I, S_{WU}^I, K_{RW}^H, K_{RWR}^H) \\
K_{RW}^L &= f(S_W, S_{WCR}^I, S_{OWCR}^I, S_{WU}^I, K_{RW}^L, K_{RWR}^L) \\
K_{ROG}^H &= f(S_{OG}, S_{OGCR}^I, S_{GCR}^H, S_{OMAX}, K_{RO}^H, K_{RORG}^H) \\
K_{ROG}^L &= f(S_{OG}, S_{OGCR}^I, S_{GCR}^H, S_{OMAX}, K_{RO}^L, K_{RORG}^L)
\end{aligned} \tag{Eq. 7.42}$$

For the immiscible oil relative permeability to water and for three-point scaling, the critical water saturation end-point is taken from the interpolated salinity value for the default three-phase oil relative permeability model:

$$\begin{aligned}
K_{ROW}^H &= f(S_{OW}, S_{OWCR}^I, S_{WCR}^I, S_{OMAX}, K_{RO}^H, K_{RORW}^H) \\
K_{ROW}^L &= f(S_{OW}, S_{OWCR}^I, S_{WCR}^I, S_{OMAX}, K_{RO}^L, K_{RORW}^L)
\end{aligned} \tag{Eq. 7.43}$$

but from the respective high and low salinity values for the Stone 1 and 2 three-phase oil relative permeability models:

$$\begin{aligned}
K_{ROW}^H &= f(S_{OW}, S_{OWCR}^I, S_{WCR}^H, S_{OMAX}, K_{RO}^H, K_{RORW}^H) \\
K_{ROW}^L &= f(S_{OW}, S_{OWCR}^I, S_{WCR}^L, S_{OMAX}, K_{RO}^L, K_{RORW}^L)
\end{aligned} \tag{Eq. 7.44}$$

The miscible relative permeabilities are looked up in the miscible tables by applying two-point saturation (horizontal) end-point scaling using the interpolated saturation end-points together with the table derived values of maximum relative permeabilities:

$$\begin{aligned}
K_{RW}^{MIS} &= f(S_W, S_{WCR}^I, S_{WMAX}^I, k_{rwmax}^{mis}) \\
K_{ROG}^{MIS} &= f(S_{OG}, S_{OGCR}^I, S_{OMAX}, k_{romax}^{mis}) \\
K_{ROW}^{MIS} &= f(S_{OW}, S_{OWCR}^I, S_{OMAX}, k_{romax}^{mis})
\end{aligned} \tag{Eq. 7.45}$$

The immiscible high and low salinity relative permeabilities and capillary pressure are interpolated according to:

$$\begin{aligned}
 K_{RW}^{IMM} &= F_1 K_{RW}^L + (1-F_1)K_{RW}^H \\
 K_{ROW}^{IMM} &= F_1 K_{ROW}^L + (1-F_1)K_{ROW}^H \\
 K_{ROG}^{IMM} &= F_1 K_{ROG}^L + (1-F_1)K_{ROG}^H \\
 P_{COW}^{IMM} &= F_2 P_{COW}^L + (1-F_2)P_{COW}^H
 \end{aligned} \tag{Eq. 7.46}$$

The oil-water capillary pressure is taken directly from the interpolated immiscible term:

$$P_{COW}^I = P_{COW}^{IMM} \tag{Eq. 7.47}$$

The miscible and immiscible water relative permeability and oil relative permeability to water are interpolated according to:

$$\begin{aligned}
 K_{RW}^I &= F_3 K_{RW}^{MIS} + (1-F_3)K_{RW}^{IMM} \\
 K_{ROW}^I &= F_3 K_{ROW}^{MIS} + (1-F_3)K_{ROW}^{IMM}
 \end{aligned} \tag{Eq. 7.48}$$

For the default three-phase oil relative permeability model, the oil relative permeability to gas is interpolated between the miscible and the immiscible values:

$$K_{ROG}^I = F_3 K_{ROG}^{MIS} + (1-F_3)K_{ROG}^{IMM} \tag{Eq. 7.49}$$

whereas, for the Stone 1 and 2 three-phase oil relative permeability models, this is taken directly from the immiscible value:

$$K_{ROG}^I = K_{ROG}^{IMM} \tag{Eq. 7.50}$$

The gas-oil capillary pressure and gas relative permeability are assumed to be independent of salinity. For capillary pressure two-point scaling is employed:

$$P_{COG} = f(S_G, S_{GLPC}^H, S_{GU}^H, P_{CG}^H) \tag{Eq. 7.51}$$

For gas relative permeability, depending upon the value of item 1 of the `SCALECRS` keyword, either two-point scaling:

$$K_{RG} = f(S_G, S_{GCR}^H, S_{ogcr}^h, S_{GU}^H, K_{RG}^H, K_{RGR}^H) \tag{Eq. 7.52}$$

or three-point scaling is employed:

$$K_{RG} = f(S_G, S_{GCR}^H, S_{OGCR}^H, S_{GU}^H, K_{RG}^H, K_{RGR}^H) \tag{Eq. 7.53}$$

Hence only the high salinity table and high salinity scaled end-points are used for modeling the gas saturation functions.

Low salinity option and surfactant option with wettability dependence

If, in addition to the `LOWSALT` option, the `SURFACTW` option has been specified, two sets of high and low salinity functions are modeled, one corresponding to oil-wet and the other to water-wet. These are each processed and combined to form a composite immiscible set which is then combined with the miscible saturation functions. The miscible saturation functions are assumed to be independent of salinity. The `SURFNUM` keyword is used to define the miscible saturation regions, the `SATNUM` and `LWSLTNUM` (or `LSNUM`) keywords are used to define the immiscible high and low salinity oil-wet saturation regions respectively and the `SURFWNUM` (or `HWSNUM`) and `LSLTWNUM` (or `LWSNUM`) keywords are used to define the immiscible high and low salinity oil-wet saturation regions respectively.

The oil-wet and water-wet high and low salinity saturation functions are each combined in the manner described above for the **LOWSALT** plus **SURFACT** options. However, the interpolation process applied to the end-points involves an additional interpolation according to the adsorbed surfactant concentration. Using the saturation table derived critical water saturation as an example, the horizontal interpolation process prior to table lookup of the saturation functions consists of:

$$S_{wcr}^{imm-ow} = F_1 S_{wcr}^{ow-l} + (1-F_1) S_{wcr}^{ow-h} \quad \text{Eq. 7.54}$$

$$S_{wcr}^{imm-ww} = F_1 S_{wcr}^{ww-l} + (1-F_1) S_{wcr}^{ww-h}$$

$$S_{wcr}^{imm} = F_4 S_{wcr}^{imm-ow} + (1-F_4) S_{wcr}^{imm-ww}$$

$$S_{wcr}^i = F_3 S_{wcr}^{mis} + (1-F_3) S_{wcr}^{imm}$$

where

F_4 is tabulated as a function of the adsorbed surfactant concentration and corresponds to the second column of the **SURFADDW** keyword.

superscripts *ow-h*, *ow-l* denote the oil-wet immiscible high and low table saturation end-points respectively.

superscripts *ww-h*, *ww-l* denote the water-wet immiscible high and low table saturation end-points respectively.

superscripts *imm-ow*, *mis-ow* denote the immiscible oil-wet and water-wet interpolated table saturation end-points respectively.

superscript *i* denotes the interpolation between immiscible and miscible table saturation end-points.

Following table look-up, the relative permeabilities and capillary pressures are combined in a similar manner to that described above. Hence using water relative permeability as an example, the vertical interpolation process following table lookup of the saturations functions consists of:

$$k_{rw}^{imm-ow} = F_1 k_{rw}^{ow-l} + (1-F_1) k_{rw}^{ow-h} \quad \text{Eq. 7.55}$$

$$k_{rw}^{imm-ww} = F_1 k_{rw}^{ww-l} + (1-F_1) k_{rw}^{ww-h}$$

$$k_{rw}^{imm} = F_4 k_{rw}^{imm-ow} + (1-F_4) k_{rw}^{imm-ww}$$

$$k_{rw}^i = F_3 k_{rw}^{mis} + (1-F_3) k_{rw}^{imm}$$

Hence apart from these additional interpolation steps, the interpolation process is similar to that for when only one pair of immiscible high and low salinity saturation functions are being modeled. Similar treatments apply for the oil-water capillary pressure and for the oil relative permeabilities, noting that for the Stone 1 and 2 three-phase oil relative permeability model, only the oil-wet immiscible oil relative permeability gas is employed. If end-point scaling is specified, the modeling is similar to that described for the low salinity option and surfactant option without wettability dependence (**LOWSALT** and **SURFACT**) except that two pairs of high and low salinity scaled end-points are used.

See "[Surfactant model](#)" for additional information.

Miscible and immiscible low and high salinity saturation regions and end-points

The following combinations of saturation regions are available:

1. If only the `LOWSALT` option is specified, the `SATNUM` and `LWSLTNUM` keywords are used to define the high and low salinity saturation function table numbers respectively.
2. If the `LOWSALT` and `SURFACT` options are specified, the `SATNUM` and `LWSLTNUM` keywords are used to define the high and low salinity saturation function table numbers respectively and the `SURFNUM` keyword is used to define the miscible saturation function table numbers.
3. If the `LOWSALT` and `SURFACTW` options are specified, the `SATNUM` and `LWSLTNUM` keywords are used to define the oil-wet high and low salinity saturation function table numbers respectively and the `SURFWNUM` and `LSLTWNUM` keywords are used to define the water-wet high and low salinity saturation function table numbers respectively and the `SURFNUM` keyword is used to define the miscible saturation function table numbers.
4. If the only the `SURFACT` option is specified, the `SATNUM` and `SURFNUM` keywords are used to define the immiscible and miscible saturation function table numbers respectively. This does not involve the low salinity option but is included here for completeness. See "[Surfactant model](#)" for further information.
5. If the only the `SURFACTW` option is specified, the `SATNUM` and `SURFWNUM` keywords are used to define the immiscible oil-wet and water-wet saturation function table numbers respectively and the `SURFNUM` keyword is used to define the miscible saturation function table numbers. This does not involve the low salinity option but is included here for completeness. See "[Surfactant model](#)" for further information.
6. The `LSNUM`, `LWSNUM` and `HWSNUM` keywords may be used as abbreviated aliases for the `LWSLTNUM`, `LSLTWNUM` and `SURFWNUM` keywords respectively.

This is summarized below:

Options	Saturation Regions
<code>LOWSALT</code>	<code>SATNUM</code> and <code>LWSLTNUM</code> (or <code>LSNUM</code>)
<code>LOWSALT</code> and <code>SUFRACT</code>	<code>SURFNUM</code> , <code>SATNUM</code> and <code>LWSLTNUM</code> (or <code>LSNUM</code>)
<code>LOWSALT</code> and <code>SUFRACTW</code>	<code>SURFNUM</code> , <code>SATNUM</code> and <code>LWSLTNUM</code> (or <code>LSNUM</code>) and <code>SURFWNUM</code> (or <code>HWSNUM</code>) and <code>LSLTWNUM</code> (or <code>LWSNUM</code>)
<code>SURFACT</code>	<code>SURFNUM</code> and <code>SATNUM</code>
<code>SURFACTW</code>	<code>SURFNUM</code> , <code>SATNUM</code> and <code>SURFWNUM</code> (or <code>HWSNUM</code>)

For the immiscible saturation regions, the following sets of end-point scaling arrays are available:

<code>SATNUM</code>	<code>LWSLTNUM</code> (or <code>LSNUM</code>)	<code>SURFWNUM</code> (or <code>HWSNUM</code>)	<code>LSLTWNUM</code> (or <code>LWSNUM</code>)
High Salinity Oil-Wet	Low Salinity Oil-Wet	High Salinity Water-Wet	Low Salinity Water-Wet
<code>SWL</code>	<code>LSWL</code>	<code>HWSWL</code>	<code>LWSWL</code>
<code>SWLPC</code>	<code>LSWLPC</code>	<code>HWSWLPC</code>	<code>LWSWLPC</code>
<code>SWCR</code>	<code>LSWCR</code>	<code>HWSWCR</code>	<code>LWSWCR</code>
<code>SWU</code>	<code>LSWU</code>	<code>HWSWU</code>	<code>LWSWU</code>

SATNUM	LWSLTNUM (or LSNUM)	SURFWNUM (or HWSNUM)	LSLTWNUM (or LWSNUM)
High Salinity Oil-Wet	Low Salinity Oil-Wet	High Salinity Water-Wet	Low Salinity Water-Wet
SGL			
SGLPC			
SGCR			
SSGU			
SOWCR	LSOWCR	HWSOWCR	LWSOWCR
SOGCR	LSOGCR	HWSOGCR	LWSOGCR
KRW	LKRW	HWKRW	LWKRW
KRWR	LKRWR	HWKRWR	LWKWR
KRG			
KRGR			
KRO	LKRO	HWKRO	LWKRO
KRORW	LKRORW	HWKRORW	LWKRORW
KRORG	LKRORG	HWKRORG	LWKRORG
PCW	LPCW	HWPCW	LWPCW
PCG			

Hysteresis

The low salinity option does not currently include relative permeability hysteretic effects in the water and oil phases where the dependence on salinity is being modeled; however, if hysteresis has been specified for the gas phase, this will be modeled according to the value of item 2 in the [EHYSTR](#) keyword.

Output of scaled end-point tables

If output of the scaled saturation functions to the DBG file is requested via the [EPSDEBUG](#) or [EPSDBG5](#) keywords, these will be reported for all the saturation tables involved in the high and low salinity calculations. However, with the exception of the high salinity set associated with the [SATNUM](#) keyword, the scaled gas saturation functions will be omitted because they are not used in these salinity and surfactant dependent calculations.

Two-phase water-oil models

Two phase water-oil models are modeled in a similar manner to that described above for three-phase models except that only the two-phase water relative permeability, two phase oil relative permeability to water and the water-oil capillary pressure are employed.

Version 2014.1 extensions

Prior to the 2014.1 version end-point scaling was only available for the high salinity saturation functions and both the oil-wet and water-wet sets employed the same set of end-points. For the 2014.1 version, the following changes have been made:

1. Only the high salinity (oil-wet) saturation tables are scaled using the set of end-point scaling keywords consisting of: [SWL](#), [SWLPC](#), [SWCR](#), [SWU](#), [SGL](#), [SGLPC](#), [SGCR](#), [SGU](#), [SOWCR](#), [SOGCR](#), [KRW](#), [KRWR](#), [KRG](#), [KRG](#), [KRO](#), [KRORW](#), [KRORG](#), [PCW](#) and [PCG](#).
2. The low salinity (oil-wet) saturation tables are scaled using the set of end-point scaling keywords consisting of [LSWL](#), [LSWLPC](#), [LSWCR](#), [LSWU](#), [LSOWCR](#), [LSOGCR](#), [LKW](#), [LKWR](#), [LKRO](#), [LKROW](#), [LKORG](#) and [LPCW](#).
3. The high salinity (water-wet) saturation tables are scaled using the set of end-point scaling keywords consisting of [HWSWL](#), [HWSWLPC](#), [HWSWCR](#), [HWSWU](#), [HWSOWCR](#), [HWSOGCR](#), [HWKRW](#), [HWKRWR](#), [HWKRO](#), [HWKRORW](#), [HWKRORG](#) and [HWPCW](#).
4. The low salinity (water-wet) saturation tables are scaled using the set of end-point scaling keywords consisting of [LWSWL](#), [LWSWLPC](#), [LWSWCR](#), [LWSWU](#), [LWSOWCR](#), [LWSOGCR](#), [LWKRW](#), [LWKWR](#), [LWKRO](#), [LWKROW](#), [LWKORG](#) and [LWPCW](#).
5. The [LNSUM](#), [LWSNUM](#) and [HWSNUM](#) keywords have been introduced as abbreviated aliases for the [LWSLTNUM](#), [LSLTWNUM](#) and [SURFWNUM](#) keywords respectively.

In addition, for the 2014.1 version, changes of behavior may arise when compared to previous versions because of the following differences between this version and prior versions:

1. Prior to the 2014.1 version, the high and low scaling involving the gas saturation end-points utilizes the corresponding high and low gas saturation values. For the 2014.1 version, the scaling of the high and low salinity saturation functions involving the gas saturation end-points (both with and without end-point scaling being specified via the [ENDSCALE](#) keyword) utilizes only the high salinity values. This may lead to differences between the 2014.1 and prior versions in cases where the high and low minimum or critical gas saturation values are different, in particular, when three-point scaling has been specified.
2. Prior to the 2014.1 version, in some cases when three-point scaling has been specified, the low salinity scaling utilizes the minimum rather than the critical values of water saturation. This has been rectified for the 2014.1 version.
3. Prior to the 2014.1 version, if end-point scaling has been specified, three-point end-point scaling is disabled for the two-phase water-oil models whilst looking up the water and oil saturation functions

The version prior to 2014.1 may be selected by setting [item 225](#) of the [OPTIONS](#) keyword to 1.

Setting the initial conditions

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

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

Multi-component brine

ECLIPSE 100 only

When the multi-component brine model is activated the initial mass concentration of each salt specified in the [BRINE](#) keyword in the [RUNSPEC](#) section must be entered. For initialization by equilibration the [SALTVD](#) keyword supplies a table of salt mass concentrations versus depth with one column of data for each salt. For initialization by enumeration the [SALT](#) keyword supplies the initial salt concentration of each salt for every grid block.

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

Wells, rivers and aquifers

ECLIPSE 100 only

The salinity of injected water can be specified independently for each water injector using the [WSALT](#) keyword (the argument(s) can be used as a user defined argument). The default salt concentration is zero.

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

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

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

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

Tracer related options

ECLIPSE 100 only

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

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

The polymer flood option

ECLIPSE 100 only

The polymer flood model (see “[Polymer flood model](#)”), activated using the option [POLYMER](#) in the RUNSPEC section, allows you to investigate the effect of varying brine concentrations on the efficiency of the polymer flood. The study of the interaction of brine with the polymer solution is, however, optional within the model.

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

The polymer adsorption can be set to depend on the brine concentration and on the rock permeability using the [ADSORP](#) keyword with POLYMER as an argument. Polymer adsorption can also be set to depend on brine concentration using the keywords [PLYADSS](#) and [ADSALNOD](#).

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

Multi-component brine and polymer

ECLIPSE 100 only

The multi-component brine model ([ECLMC](#) keyword in the RUNSPEC section) cannot currently be used with the polymer/brine flood model (that is, the combination of polymer and brine using the [SALTNODE](#) keyword; ECLMC can be used with the basic polymer flood model).

The surfactant flood option

ECLIPSE 100 only

The surfactant model (see "[Surfactant model](#)"), activated using the option [SURFACT](#) in the RUNSPEC section, allows you to investigate the effect of varying brine concentrations on the efficiency of the surfactant flood.

The study of the interaction of brine with the surfactant solution is, however, optional within the model. For models which have the surfactant and brine options activated, the oil-water surface tension of a fully mixed surfactant solution can be entered as a function of the salt concentration. For this, the keywords [ESSNODE](#) and [SURFSTES](#) can be used to enter two-dimensional tables for the oil-water surface tension of the solution as a function of surfactant and salt concentrations. The surfactant adsorption can be set to depend on the brine concentration and on the rock permeability using the [ADSORP](#) keyword with SURFACT as an argument.

Multi-component brine and surfactant

ECLIPSE 100 only

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

$$C_{SE} = \frac{C_-}{1 - \alpha^s X^s}, \quad X^s = \frac{C_{++}^s}{C^s} \quad \text{Eq. 7.56}$$

where

C_{SE} denotes the surfactant effective salinity

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

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

The effective salinity calculation is activated using keyword SURFESAL in the PROPS section, which also specifies the effective salinity parameter, α^s , in equation 7.56. This salinity is then used in the surfactant adsorption isotherms specified in the [ADSORP](#) keyword with SURFACT as an argument. Furthermore, this salinity as well as the surfactant concentration can be used to update the water-oil surface tension if such option is requested (using keywords SURFSTES and ESSNODE).

Solving the conservation equations

ECLIPSE 100 only

For the multi-component brine model the ion components are solved as active tracers. The ion concentrations determine the effective salinities, which in turn have a strong impact on surfactant-water-oil phase behavior and surfactant/polymer adsorption, and hence affect the phase flows. By default tracers are solved only at the end of a timestep once the equations for phases have converged. In this way the salinity is treated explicitly in the calculations which may lead to numerical instability if the timesteps are large and

the dependence on salinity is strong. For this reason active tracers can now be solved in each Newton iteration so that the tracer concentrations are updated simultaneously with the phase equations.

When the [ECLMC](#) keyword is present in the RUNSPEC section the multi-component brine equations are solved in each Newton iteration. The standard behavior of solving the brine equations at the end of a timestep can be restored by setting item 126 of the OPTIONS keyword to 2. When solved in each Newton iteration the brine concentration used when calculating effective salinities is computed as a weighted average of the concentration at time t and the concentration at time $t + \Delta t$ corresponding to the current iteration:

$$C = (1-\theta)C(t) + \theta C(t + \Delta t) \quad \text{Eq. 7.57}$$

where

θ is a weighting factor.

$\theta = 0$ corresponds to the standard, explicit treatment of brine concentration, and $\theta = 1$ corresponds to a fully coupled approach. The value of θ can be set in item 13 in the second record of the [TUNING](#) keyword (Default value: 1).

Summary of keywords

RUNSPEC

Restriction	Keyword	Description
	BRINE	activates the brine tracking option
E100 only	LOWSALT	activates the low salinity option
E100 only	ECLMC	activates multi-component brine

GRID

ECLIPSE 100 only keyword	Description
IONROCK	specify the rock ion exchange capacity
RPTGRID	controls output from the GRID section to the print file: Use argument IONROCK to output rock ion exchange capacity.

PROPS

Restriction	Keyword	Description
	BDENSITY	specify the brine density
E100 only	LSALTFNC	specify the low salinity fraction as function of the salt concentration in the grid block.
	PVTWSALT	specify water PVT functions with salt present

Restriction	Keyword	Description
	RPTPROPS	controls output from the PROPS section to the print file: Use arguments BDENSITY and PVTWSALT to output brine properties to the print file.
E100 only	IONXROCK	activate cation exchange on rock surfaces and specify cation exchange constant for rock.
E100 only	IONXSURF	activate cation exchange on surfactant micelles and specify cation exchange constant for surfactant and equivalent molecular weight of surfactant.
E100 only	SURFESAL	specify parameter for effective salinity calculation with respect to surfactant phase behavior.
E100 only	PLYESAL	specify parameter for effective salinity calculation with respect to polymer properties.
E100 only	DSPDEINT	interpolation of dispersion tables by water density.
E100 only	SWL, SWLPC, SWCR, SWU, SGL, SGLPC, SGCR, SGU, SOWCR, SOGCR, KRW, KRWR, KRG, KRGR, KRO, KRORW, KRORG, PCW and PCG	Immiscible (oil-wet) high salinity scaled end-points.
E100 only	LSWL, LSWLPC, LSWCR, LSWU, LSOWCR, LSOGCR, LKRW, LKRWR, LKRO, LKRORW, LKRORG and LPCW	Immiscible (oil-wet) low salinity scaled end-points.
E100 only	HWSWL, HWSWLPC, HWSWCR, HWSWU, HWSOWCR, HWSOGCR, HKRW, HWKRWR, HKRO, HWKRORW, HWKRORG and HWPCW	Immiscible water-wet (high salinity) scaled end-points.
E100 only	LWSWL, LWSWLPC, LWSWCR, LWSWU, LWSOWCR, LWSOGCR, LKRW, LWKRWR, LKRO, LWKRORW, LWKRORG and LWPCW	Immiscible water wet low salinity scaled end-points.

REGIONS

Restriction	Keyword	Description
	SATNUM	specify the high salinity table number for each block. For the Surfactant option, this corresponds to the immiscible high salinity oil-wet tables.
E100 only	LWSLTNUM (or LSNUM)	specify the low salinity table number for each block. For the Surfactant option, this corresponds to the immiscible low salinity oil-wet tables.

Restriction	Keyword	Description
E100 only	SURFWNUM (or HSWNUM)	For the Surfactant option, this corresponds to the immiscible high salinity water-wet tables.
E100 only	LSLTWNUM (or LWSNUM)	For the Surfactant option, this corresponds to the immiscible low salinity water-wet tables.

SOLUTION

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

SUMMARY

Refer to [Brine option quantities](#) in the *ECLIPSE Reference Manual*.

SCHEDULE

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

Examples

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

RUNSPEC

```

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

```

PROPS

```

PROPS

-- Rock properties
ROCK
4000.0 0.000008 /

-- Fluid densities at surface conditions
DENSITY
45 64 0.09 /

-- PVT data for oil
PVDO
400 1.012 1.160
1200 1.0040 1.164
2000 0.9960 1.167
2800 0.9880 1.172
3600 0.9802 1.177
4400 0.9724 1.181
5200 0.9646 1.185
5600 0.9607 1.190
/

-- PVT data for water with salt present
PVTWSALT
3000.0 /
0.0 1.000 3.0E-6 1.00 1.0E-6
10.0 0.8 3.0E-6 1.00 1.0E-6 /


-- Saturation functions
SWFN
0.20 0.0 0.0
0.40 0.1 0.0
0.80 0.5 0.0
1.0 1.0 0.0
/


SOF2
0.2 0.0
0.4 0.12
0.6 0.35
0.7 0.68
0.8 1.0
/

```

SOLUTION

```

SOLUTION

-- Equilibration data
EQUIL
5270  4000.0  5270.0  0.0  400.0  0.0  0  0  5  /

-- Salt concentration versus depth table
SALTVD
5000.0  9.0
5500.0  9.0  /


-- Aquifer data
AQUFETP
1  5400  1*  2.0E9  3.0E-5  540.96  1  10.0  /
/


AQUCT
2  5400.0  1*  20.0  0.10  3.0E-5  2400.0  140.0  4.8  1  1  9.5  /
/


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

```

SUMMARY

```

SUMMARY

-- Field salt production rates and totals
FSPR
FSPT

-- Field salt injection rates and totals
FSIR
FSIT

-- Field salt production and injection concentrations
FSPC
FSIC

-- Field salt in place
FSIP

-- Group salt production and injection rates
GSPR
'G'  /
GSIR
'G'  /

-- Well salt production and injection rates
WSPR
'PRODUCER'  /
WSIR
'INJECTOR'  /

-- Block salt concentration
BSCN
1 1 1  /
/

```

SCHEDULE Example

SCHEDULE section example

```

SCHEDULE

-- Well specification data
WELSPCS
    'PRODUCER'    'G'     1   1   5200   'OIL'    /
    'INJECTOR'    'G'    14   1   5400   'WAT'    /
/

COMPDATA
    'PRODUCER'    1   1   1   10   'OPEN'   1   -1   0.5   /
    'INJECTOR'    14   1   5   10   'OPEN'   1   -1   0.5   /
/

WCONPROD
    'PRODUCER'    'OPEN'   'ORAT'   200.0  4*   1500.0   /
/

GCONINJE
    'FIELD'      'WAT'    'VREP'   3*   0.7   /
/

WCONINJE
    'INJECTOR'    'WAT'    'OPEN'   'GRUP'   /
/

-- Salt concentration for injection well
WSALT
    'INJECTOR'    1.0   /
/

-- Timestep data
TSTEP
1.0  9.0  90.0  100.0  100.0  100.0  100.0  250.0  250.0  1000.0  1000.0   /
END

```

8

CO₂ injection: geological storage, EOR and ECBM

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

One of the major issues of CO₂ injection problems is the chemical reactivity between the supercritical fluid (referred to as the gas phase in this document) and the liquid phase. There are several facilities available in ECLIPSE 300, depending on the site and operational conditions that need to be modeled.

Geological storage

In Aquifers

Using the [CO2STORE](#) option, three phases are considered: a CO₂-rich phase (labeled gas phase), an H₂O-rich phase (labeled liquid phase) and a solid phase.

This option gives accurate mutual solubilities of CO₂ in water ($x\text{CO}_2$) and water in the CO₂-rich phase ($y\text{H}_2\text{O}$). Salts are described as components of the liquid/solid phase. To allow for a solid phase the [SOLID](#) option should be used.

The [GASWAT](#) option provides a gas/aqueous phase equilibrium method. The liquid mole fraction $x\text{CO}_2$ is accurately predicted; however, the gas phase mole fraction $y\text{H}_2\text{O}$ is less accurate. The gas composition is not restricted to CO₂/H₂O, as other gases and their solubility in water can be included.

In Oil Depleted Reservoirs and EOR (Enhanced oil recovery)

The option best suited for this case is [CO2SOL](#). Water is not considered in the oil or gas phase, but the solubility of CO₂ in the water phase is accounted for. The gas/oil composition is not restricted to CO₂ but only CO₂ is considered soluble in water.

In Gas Depleted Reservoirs

This case can be modeled using the [GASWAT](#) option by adding gas components, such as CH₄ and H₂S for example.

ECBM

The [COAL](#) option provides a diffusive flow and sorption model for Enhanced Coal Bed Methane recovery (ECBM). A full description of this option is provided in "[Coal bed methane model](#)".

Since the [CO2STORE](#), [CO2SOL](#) and [GASWAT](#) options estimate the fluid properties differently, they cannot be used together.

CO2STORE

The mutual solubilities of CO₂ and H₂O are calculated to match experimental data for typical CO₂ storage conditions: typically 12-250 degrees C and up to 600 bars. They are calculated following the procedure given by Spycher and Pruess [\[Ref. 102\]](#) and [\[Ref. 103\]](#), based on fugacity equilibration between water and a CO₂ phase. Water fugacity is obtained by Henry's law, while CO₂ fugacity is calculated using a modified Redlich-Kwong equation of state.

See "[CO₂ storage](#)".

- Salt precipitation is enabled by the use of the [SOLID](#) option. If the [SOLID](#) option is not used, salts are assumed to stay in the liquid phase.
- The gas density is obtained by an accurately tuned cubic equation of state.
- The brine density is first approximated by the pure water density and then corrected for salt and CO₂ effects by Ezrokhi's method [\[Ref. 133\]](#).
- The CO₂ gas viscosity is calculated from [\[Ref. 117\]](#) and [\[Ref. 37\]](#).

Using CO2STORE

RUNSPEC section

The option switched on using the [CO2STORE](#) keyword. The number of components is specified by the [COMPS](#) keyword. Five components are currently allowed: CO₂, H₂O, NaCl, CaCl₂ and CaCO₃.

PROPS section

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

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

Initial state constructed by equilibration

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

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

SOLUTION section

The [EQUIL](#) keyword is used for initial state equilibration.

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

SCHEDULE section

The composition of the injected gas can be specified using the keyword [WINJGAS](#).

Mole fractions x_{CO_2} and y_{H_2O} are output by writing XMF and YMF in [RPTSCHED](#).

Note: See the note about salt precipitation in CO₂ Storage "[Compatibility](#)".

CO2SOL

[CO2SOL](#) is a three-phase compositional option for simulating CO₂ flooding in oil reservoirs (EOR, storage in oil depleted reservoirs). Water is not allowed to dissolve in the gas or oil phases, CO₂ can be present in the three phases, and other components in the oil and gas phases.

See "[CO₂/H₂S solution in the aqueous phase](#)".

CO₂ partitioning between the oil and gas phases is calculated by a fugacity equilibration method. A cubic equation of state is used to model oil and gas phase densities and fugacities.

The amount of CO₂ dissolved in water, and other aqueous phase properties, are computed using solubility data entered by the user with either the [SOLUBILI](#), [SOLUBILS](#) or [SOLUBILT](#) keywords or defaulted following Chang et al. [Ref. 18]. When using SOLUBILI or SOLUBILT the salting-out effect is taken into account by correcting solubility in pure water with the Chang et al. correlation for brine [Ref. 18]. The SOLUBILS keyword can be used to describe a more complex salt dependency using the [BRINE](#) tracking. The SOLUBILT keyword can be used to describe a more complex temperature dependency. To obtain details on the solubility data used in a simulation the SOLU mnemonic may be specified in the [RPTPROPS](#) keyword.

Molecular diffusion between gases can be modeled using the [DIFFUSE](#) option. See also "[Diffusion](#)".

It is possible to set initial dissolved CO₂ as a function of depth using the [RSWVD](#) or [RSW](#) keywords. This initial solution honors this value unless it exceeds the saturated CO₂ concentration in which case the saturated value is used.

Notes

- The initial CO₂ concentration may not be in exact equilibrium with the hydrocarbon phase. In such a case the initial flash slightly modifies the input values.
- The first line of the solubility table defined using the [SOLUBILI](#) keyword represents water at standard conditions, when the dissolved CO₂ amount is at a minimum. The simulator does not assume that the amount of CO₂ in water at the lowest pressure is zero; if you want to specify it as such then you should set it to zero in the first line of the table.
- Instantaneous equilibrium is assumed between the oil, gas and water phases.

Using CO2SOL

RUNSPEC section

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

The [DIFFUSE](#) option allows for the molecular diffusion of gases. See also "[GASWAT Option](#)".

The brine tracking option is activated by the [BRINE](#) keyword.

PROPS section

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

Fluid component properties can be generated from the PVT *i* database. The component 'CO2' must be present in [CNAMES](#).

The initial dissolved CO₂ may be specified as a function of depth using the [RSWVD](#) keyword.

The [SOLUBILI](#) keyword allows the user to enter a solubility data set as a function of reservoir pressure. The [SOLUBILT](#) keyword is similar to SOLUBILI but allows the user to enter a temperature dependent table. For both keywords, the table will be automatically corrected for the salinity effect following Chang et al. [Ref. 18] if either the [SALINITY](#) or [SALINITR](#) keyword has been used. In this case, the temperature used in that correction will be taken from the [RTEMP](#) keyword in the case of SOLUBILI and from table itself for SOLUBILT. The density of water at surface condition can be adjusted using the [DENSITY](#) or [GRAVITY](#) keywords.

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

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

The coefficients for molecular diffusion must be set with [DIFFCGAS](#) if the [DIFFUSE](#) option is used.

If CO₂ appears as liquid, it will be labelled as oil phase. In such a case, make sure that the [SOF3](#) keyword (relative permeability for CO₂ liquid) is appropriate.

Initial state constructed by equilibration

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

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

SOLUTION section

The [EQUIL](#) keyword is used for equilibration.

- Initial state set by enumeration

- The oil phase composition is set with [XMF](#), the gas phase composition with [YMF](#), water composition with [AMF](#) and the total oil and gas composition with [ZMF](#). Water content in CO₂ can be defined by [RSW](#) or [RSWVD](#).

SCHEDULE section

The composition of the injected gas can be specified using the keyword [WINJGAS](#).

The aqueous mole fractions a CO₂ and a H₂O can be output by adding the argument AMF in [RPTSCHED](#).

CO2SOL/H2SSOL/GASSOL

[CO2SOL](#) / [H2SSOL](#) is a three-phase compositional option for simulating CO₂ and H₂S flooding in oil reservoirs (EOR, storage in oil depleted reservoirs).

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

See "[CO₂/H₂S solution in the aqueous phase](#)" and "[GASSOL solution in the aqueous phase](#)".

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

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

Molecular diffusion between gases can be modeled using the [DIFFUSE](#) option. See also "[Diffusion](#)".

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

Notes

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

Using GASSOL

RUNSPEC section

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

The [DIFFUSE](#) option allows for the molecular diffusion of gases. See also "[GASWAT option](#)".

PROPS section

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

Fluid component properties can be generated from the PVT *i* database. The dissolved gas components must be present in [CNAMES](#).

The initial concentrations of dissolved gases may be specified as a function of depth using the [RSWVD](#) keyword.

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

The coefficients for molecular diffusion must be set with [DIFFCGAS](#) if the [DIFFUSE](#) option is used.

Initial state constructed by equilibration

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

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

SOLUTION section

The [EQUIL](#) keyword is used for equilibration.

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

SCHEDULE section

The composition of the injected gas can be specified using the keyword [WINJGAS](#).

The aqueous mole fractions of dissolved gas components can be output by adding the mnemonic AMF in [RPTSCHE](#).

GASWAT

The [GASWAT](#) option uses a modified Peng-Robinson equation of state in order to model gas/aqueous phase equilibria.

GASWAT allows a multi-component gas phase. CO₂, N₂, and H₂S solubilities in water are treated with the Peng Robinson equation of state (EOS) modified by Soreide and Whitson. However, solubilities of other gases like methane are treated by the original Peng Robinson EOS by default. "[GASWAT option](#)" provides an exhaustive description of this feature.

Molecular diffusion between gases can be modeled using the [DIFFUSE](#) option. See "[Diffusion](#)".

Using GASWAT for CO₂ storage issues

RUNSPEC section

This option is switched on using the [GASWAT](#) keyword. The number of components is specified by the [COMPS](#) keyword. Water has to be defined as a component; salts are not components. The [WATER](#) keyword should not be present, as H₂O is a component specified in COMPS.

The Diffusion option allows for molecular diffusion of gases.

PROPS section

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

Salinity can be set up by the [SALINITY](#) keyword or on a regional basis with the [SALINITR](#) keyword

The coefficients for molecular diffusion must be set with [DIFFCGAS](#) if the Diffusion option is used.

Initial state constructed by equilibration

If an initial gas-water contact exists within the reservoir you may specify either the vapor or the aqueous phase composition at the contact, either [ZMFVD](#) or [COMPVD](#) can be used.

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

SOLUTION section

The [EQUIL](#) keyword is used for initial state equilibration.

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

SCHEDULE section

The composition of the gas injected can be specified using the keyword [WINJGAS](#).

Mole fractions xCO₂ and yH₂O can be output by writing arguments XMF and YMF in [RPTSCHED](#).

Boundary conditions

Several facilities are available.

- Numerical aquifers contain only a water phase. For **CO2STORE** and **GASWAT** cases, make sure that no supercritical CO₂ (CO₂ rich phase being labeled as gas phase) reaches the boundary. Likewise for the liquid CO₂ for a **CO2SOL** run.
- For **CO2STORE** and **GASWAT** cases, analytic aquifers can be used to let the CO₂ rich phase leave the reservoir. For this, use **AQUCHWAT**, **AQUCHGAS** and **AQSTREAM** and ensure that the composition of the gas aquifer specified by **AQUCHGAS** is as close as the composition of the gas reaching the boundary.

Note on conversions for salinity units

As the various CO₂ options use different frameworks within ECLIPSE, units for salinity input differ.

Salt concentrations in **XMF** and **YMF** must be entered in mole fraction. The **SALINITY** keyword unit is molality (gm-M/kg of solvent). Salinity in **SOLUBILS** must be expressed in kg/m³ of brine.

Conversions between these different units are described below.

Molality to mole fraction

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

where:

mole fraction of solvent in the aqueous phase:

$$x_{\text{solvent}} = \frac{n_{\text{solvent}}}{n_{\text{total}}} = \frac{n_{\text{solvent}}}{n_{\text{solvent}} + \sum_{i \neq \text{solvent}} n_i}$$

$$\text{molality of component } i: \text{molality}_i = \frac{n_i}{m_{\text{solvent}}} \text{ [gm-M/kg]},$$

M_{solvent} is the molecular weight of solvent [gm/gm-M]

Thus:

$$\text{molality}_{\text{solvent}} = 1000 / M_{\text{solvent}} \text{ [gm-M/kg]},$$

In most of the cases, the solvent is H₂O. As $M_{H_2O} = 18.014$ gm/gm-M, $\text{molality}_{H_2O} = 55.5$ gm-M/kg.

Concentration (mg/L) to mole fraction

Chemical analysis usually gives concentration in mg/L (=10⁻³kg/m³) and the initial composition of the brine/water in ECLIPSE must be entered in mole fraction.

Concentrations in mg/L can be converted to mole fraction through:

$$x_i = \frac{\left(\frac{c_i}{M_i}\right)}{\sum_j \left(\frac{c_j}{M_j}\right)} = \frac{[i]}{\sum_j [j]}$$

where

[i] denotes the molarity of component i in gm-M/L,

c_i the concentration in mg/L and

M_i the molecular weights in mg/gm-M.

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

Example

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

Species	Concentration (mg/L)	Species	Concentration (mg/L)
Na^+	90110	HCO_3^-	45
Ca^{2+}	1760	I^-	1
Mg^{2+}	2230	HS^-	nb
K^+	470	CO_3^{2-}	0
NH_4^{2+}	0	OH^-	0
Fe^{2+}	0	TDS (Total Dissolved Solid)	250000
Cl^-	141470	pH	5.3
SO_4^{2-}	6600	Water Density at reservoir conditions (kg/m ³)	1164
Br^-	64		

Table 8.1: Brine sample analysis

ECLIPSE CO2STORE

Targeted ion concentrations (for ECLIPSE CO2STORE option):

$$c_{\text{Na}^+} = 90.11 \text{ gm/L}$$

$$c_{\text{Ca}^{2+}} = 1.76 \text{ gm/L}$$

$$c_{\text{Cl}^-} = 141.47 \text{ gm/L}$$

For one liter of brine:

$$m_{\text{H}_2\text{O}} = \text{reservoir brine density} - \text{TDS} = 1164 - 250 = 914 \text{ gm}$$

$$n_{\text{H}_2\text{O}} = \frac{m_{\text{H}_2\text{O}}}{M_{\text{H}_2\text{O}}} = \frac{914}{18.014} = 50.738 \text{ gm-M}$$

$$n_{\text{Na}^+} = \frac{90.11}{23} = 3.919 \text{ gm-M}$$

$$n_{\text{Ca}^{2+}} = \frac{1.76}{40} = 0.044 \text{ gm-M}$$

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

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

	Sodium (Na)	Chlorine (Cl)	Sodium Chloride (NaCl)
Initial	$n_{Na^{+0}}$	$n_{Cl^{-0}}$	0
At t time n_t	$n_{Na^{+0}} - \zeta$	$n_{Cl^{-0}} - \zeta$	ζ
Final n_f	0	$n_{Cl^{-f}} = n_{Cl^{-0}} - n_{Na^{+0}}$	$n_{Na^{+0}}$

where ζ is the reaction advancement.

Here $n_{NaCl(aq)} = n_{Na^{+0}}$ because Na^+ is the limiting factor.

For $Ca^+ + 2Cl^- \Leftrightarrow CaCl_2$:

	Calcium (Ca)	Chlorine (Cl)	Calcium Chloride ($CaCl_2$)
Initial	$n_{Ca^{2+0}}$	$n_{Cl^{-f}}$	0
At t time n_t	$n_{Ca^{2+0}} - \psi$	$n_{Cl^{-f}} - 2\psi$	ψ
Final n_f	0	$n_{Cl^{-f}} - 2n_{Ca^{2+0}}$	$n_{Ca^{2+0}}$

where ψ is the reaction advancement.

Here $n_{CaCl_2(aq)} = n_{Ca^{2+0}}$ because Ca^{2+} is the limiting factor.

The ionic species are reconciled into neutral component in ECLIPSE. The neutral components in this case are $NaCl$ and $CaCl_2$. These are formed from the ionic species in order of increasing solubility. Starting with the salt with lowest solubility, in this case $NaCl$, the limiting ionic species is Na^+ , that is, 3.92 gm-M of $NaCl$ can be formed, leaving $(4.00 - 3.92) = 0.08$ gm-M/L of Cl^- for the formation of 0.04 gm-M/L of $CaCl_2$. It should be noted that the water analysis in this case is consistent with the Ca^{2+} concentration.

$$n_{total} = n_{NaCl(aq)} + n_{CaCl_2(aq)} + n_{H_2O} = 54.701 \text{ gm-M}$$

$$x_{NaCl} = \frac{n_{NaCl(aq)}}{n_{total}} = \frac{3.919}{54.701} = 0.0716$$

$$x_{CaCl_2} = \frac{n_{CaCl_2(aq)}}{n_{total}} = \frac{0.044}{54.701} = 0.0008$$

$$x_{H_2O} = \frac{n_{H_2O}}{n_{total}} = \frac{46.408}{54.701} = 0.9276$$

Note: In this example, the molality of salt you have to input for the keyword [SALINITY](#) (as used in the CO2SOL and GASWAT options) is:

$$\text{for one liter of solution, } \text{molality}_{\text{salt}} = \frac{\sum_{\text{salt}} n_i}{\text{mass}_{H_2O}} = \frac{3.963}{0.916} = 4.34 \text{ gm-M/kg}$$

Molality to mass per volume unit (kg/m³)

$$\text{Mass per volume unit} = \frac{d}{B} \times \frac{\text{molality} \times 10^{-3} \times M}{1 + \text{molality} \times 10^{-3} \times M}$$

where

M = salt molecular weight (typically 58.44 kg/kg-M for NaCl)

B = brine volume factor

d = surface brine density

CO₂ storage

<input type="checkbox"/>	ECLIPSE 100
<input checked="" type="checkbox"/>	ECLIPSE 300
<input type="checkbox"/>	SPECIAL

In ECLIPSE 300 there are several options available in order to study the storage of CO₂ under various conditions. This chapter describes the [CO2STORE](#) option, for CO₂ - H₂O systems with salts. In addition the following options are available.

- The [CO2SOL](#) option, typically used to study CO₂ storage in oil reservoirs - water is not allowed in the oil/gas phase.
- The [GASWAT](#) option, general equation of state for gas-water systems.
- The [COAL](#) option for coal bed methane reservoirs.

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

Note: In thermal mode the CO2STORE option can be used with any number of components and include a liquid oil phase, this is described in [Thermal CO2STORE](#).

The CO2STORE option

For the [CO2STORE](#) option three phases are considered, a CO₂ rich phase, an H₂O rich phase and a solid phase. The CO₂ rich phase is labeled the gas phase while the H₂O rich phase is labeled the aqueous phase (liquid phase). The mutual solubilities of CO₂ and H₂O are calculated to match experimental data for CO₂ - H₂O systems under typical CO₂ storage conditions: 12-100 °C and up to 600 bar. The model is also applicable at elevated temperatures as detailed in [\[Ref. 103\]](#), up to 250 °C. Fluid components are specified with the [COMPS](#) keyword. Allowed component names are currently CO2, H2O and the salts NACL, CACL2 and CACO3.

Component name	Phases
CO2	Aqueous/Gas
H2O	Aqueous/Gas
NACL	Aqueous/Solid
CACL2	Aqueous/Solid

Component name	Phases
CAC03	Aqueous/Solid

Table 9.1: Components that can be used with the CO2STORE option

The option is similar to the [GASWAT](#) option, but the salt components of the aqueous or solid phase are tracked as part of the fluid system. Also the partitioning of the component H₂O in the gas phase has been matched accurately. Fluid properties are assigned internally based on the names; this means that a minimum of user data is needed in order to supply the fluid properties.

Phase splitting

The partitioning of CO₂ and H₂O in the aqueous and gas phase follows the procedure given by Spycher and Pruess, [\[Ref. 102\]](#) and [\[Ref. 103\]](#). The salts are assumed to stay in the aqueous phase unless the [SOLID](#) option is activated.

With the [SOLID](#) option the components NaCl, CaCl₂ and CaCO₃ can be present in both the aqueous phase and the solid phase according to the following equilibrium reactions:

- H₂O ⇌ H⁺ + OH⁻
- CO₂ + H₂O ⇌ HCO₃⁻ + H⁺
- HCO₃⁻ ⇌ CO₃²⁻ + H⁺
- CaCl⁺ ⇌ Ca₂⁺ + Cl⁻
- NaCl(s) ⇌ Na⁺ + Cl⁻
- CaCl₂(s) ⇌ CaCl⁺ + Cl⁻
- CaCO₃(s) ⇌ Ca²⁺ + CO₃²⁻

If only NaCl is present a simplified procedure can be used and the maximum NaCl solubility in the aqueous phase is calculated by [\[Ref. 86\]](#).

$$WtNaCl = 26.218 + 0.0072T + 0.000106T^2$$

where T is the temperature in °C.

It is recommended that the [SOLIDMMS](#) keyword is also used in order to model the decrease in mobility as a function of the solid saturations. The gas and aqueous saturations are normalized to be between zero and unity, so the two phase water and gas relative permeability curves can be used. However it should be noted that the effective fluid volume is given by

$$FV = (1-S)PV$$

where PV is the pore volume and S is the solid saturation. It is possible to specify a target solid saturation change for each timestep by using item 21 of [TSCRIT](#). A small value will usually enhance the stability.

Activity coefficient models

The CO₂ and H₂O activity are computed following Spycher and Pruess, [\[Ref. 102\]](#) and [\[Ref. 103\]](#). This is done using Margules expressions:

$$\ln(\gamma_{H2O}) = (A_M - 2A_M x_{H2O}) x_{CO2}^2$$

$$\ln(\gamma_{CO2}) = 2A_M x_{CO2} x_{H2O}^2 e^{(\gamma_{CO2}')}$$

where:

$$A_M = 0 \quad (T \leq 100^\circ C)$$

$$A_M = a \cdot (T-100) + b \cdot (T-100)^2 \quad (T > 100^\circ C)$$

In these expressions, a and b are regressed parameters and x_{CO2} and x_{H2O} denote the aqueous phase mole fractions. The CO₂ activity coefficient is modified for salting out effects by γ_{CO2}' .

There are two models that can be applied in order to compute γ_{CO2}' :

- The Rumpf et. al 1994 model as detailed in [Ref. 102], applicable up to 160 °C.
- The model given in [Ref. 103], applicable up to 250 °C.

To choose the model, use the keyword [ACTCO2S](#).

Density

The gas density is obtained by a cubic equation of state tuned to accurately give the density of the compressed gas phase, following Spycher and Pruess, [Ref. 102]. A modified Redlich-Kwong equation of state is used, where the attraction parameter is made temperature dependent:

$$P = \left(\frac{RT_K}{V-b_{mix}} \right) \cdot \left(\frac{a_{mix}}{T_K^{1/2} V(V+b_{mix})} \right)$$

where V is the molar volume, P is the pressure, T_K the temperature in Kelvin, R is the universal gas constant and a_{mix} and b_{mix} are the attraction and repulsion parameters. For more information see also [Ref. 103].

The transition between liquid CO₂ and gaseous CO₂ will lead to rapid density changes for the CO₂-rich phase. The simulator uses a narrow transition interval between the liquid state and gaseous state.

For the brine density we first compute the density of pure water. The default model uses an analytical form of the water density as detailed by *The International Association for the Properties of Water and Steam* (IAPWS-IF97). In order to use the pre-2010.1 water density calculations as detailed in [Ref. 62], item 232 in OPTIONS3 can be put to unity. The Ezrokhi's method is then applied to calculate the effect of salt and CO₂ as described in [Ref. 133].

Viscosity

The CO₂ gas viscosity is calculated from [Ref. 117] and [Ref. 37].

Aqueous speciation

It is possible to output the different ions and chemical components in the aqueous phase by adding AQSP to the [RPTRST](#) or [RPTSCHED](#) keyword. Similarly the pH is output by adding AQPH. Additionally, the SUMMARY keywords BAQSP and BAQPH can be used.

Names of the species considered are given below

Aqueous species	
aH2O	Pure water
aCO2	Carbon dioxide
aH+	Hydrogen ion
aOH-	Hydroxide ion
aCO3-2	Carbonate ion
aHCO3-1	Bicarbonate ion
aCl-	Chloride ion
aNa+	Sodium ion
aCa++	Calcium ion
aCaCl+	Calcium monochloride ion

Table 9.2: Aqueous Species

Note: As an example, the total amount of aqueous NaCl is given by the molar density, named MLSC followed by the component index, while the speciation outputs the ions a NA⁺ and a CL⁻.

pH calculation

The pH is calculated as

$$pH = -\log(\zeta_H \cdot m_H)$$

where

ζ_H is the activity coefficient of the H⁺ ion and

m_H is the molality of the H⁺ ion.

Analytic water aquifers

An analytic water aquifer defined by [AQUCHWAT](#), [AQUCT](#), [AQUFETP](#) or [AQUFLUX](#) contains only the "H₂O" component. It is possible to modify the aquifer composition using the [AQSTREAM](#) keyword. Any component is allowed to leave or enter the reservoir. Additionally, the aquifer water properties are calculated as described above.

If item 131 of the OPTIONS3 keyword is set to 1, this reverts to the pre-2007.2 behavior where the water entering / leaving the reservoir is made up entirely of the "H₂O" component. If item 119 of OPTIONS3 is set to 1, this reverts to the pre-2007.1 behavior where the aquifer water properties were calculated using the water PVT tables. See "[Aquifer modeling facilities](#)".

Using the CO₂ storage model

RUNSPEC section

The option is invoked by the [CO2STORE](#) keyword. If a solid phase is considered the [SOLID](#) keyword must be specified. The capillary pressure mode is selected by default so that the capillary pressure is added to the gas phase pressure. To subtract the capillary pressure from the aqueous phase pressure use item 8 of the [OPTIONS3](#) keyword. To run with a saturation weighted capillary pressure option the keyword [FORMOPTS](#) can be used with the mnemonic PCSPLIT. For runs with large capillary pressure it is recommended to set item 8 of [OPTIONS3](#), so the system pressure is the gas phase pressure. Also note that the [WATER](#) keyword should not be used with the [CO2STORE](#) option.

PROPS section

The fluid components are set by specifying the names in the [COMPS](#) keyword.

The water relative permeability is specified using the [WSF](#) keyword, while the gas relative permeability and the gas-water capillary pressure are entered using the [GSF](#) keyword.

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

If an initial gas-water contact exists within the reservoir you may specify either the vapor or the aqueous phase composition at the contact ([ZMFVD](#) or [COMPVD](#) may be used).

To enable molecular diffusion, the keywords [DIFFCWAT](#) and [DIFFCGAS](#) should be used. Cross phase diffusion can be included by the keywords [DIFFCWG](#) and [DIFFCGW](#). Note that without the cross phase diffusion option no diffusive flow is computed between two neighbor cells that are single phase, one being aqueous and the other gas. The interaction with the formation water could therefore be slightly under predicted without including the cross phase diffusion term. For more information see "[Diffusion](#)".

SOLID properties need to be assigned in the usual way. The component reference density is assigned using the keyword [SDREF](#). The [SOLIDMMS](#) keyword can be used to specify mobility multipliers.

For temperatures above 160 °C an alternative activity coefficient model for the aqueous CO₂ should be considered using the keyword [ACTCO2S](#).

SUMMARY section

The keywords FWCD, FGCDI and FGCDM can be used to output the field total molar amount of CO₂. RWCD, RGCDI and RGCDM can be used to output the same quantities on a regional basis.

Field	Region	Description
FWCD	RWCD	Dissolved in water
FGCDI	RGCDI	In an immobile gas phase
FGCDM	RGCDM	In a mobile gas phase

Notes

- Quantities such as FGIP, FGIPL and FGIPG depend on the state of the CO₂ at standard conditions. This means that if all the CO₂ injected is in a dissolved state these quantities are

zero. The field molar amount injected for a given component can be obtained using the keywords FCMIT and FCMIP.

- Also the aqueous phase is considered as “hydrocarbon” when calculating the “hydrocarbon weighted” field pressure reported by FPR and FPRH. For a comparison with a simulation involving an oil phase such as [CO2SOL](#), the “pore volume weighted” pressure returned by FPRP can be used. Additionally, setting item 151 of the OPTIONS3 keyword to 1 ensures that only the gas phase is treated as “hydrocarbon”.

SOLUTION section

The [EQUIL](#) keyword is used for equilibration. An initial solid saturation and corresponding composition can be assigned by the keywords [SSOLID](#) and [SMF](#). Some care must be taken as this might not be in equilibrium. The initial flash will however put the minerals in equilibrium with the formation water.

SCHEDULE section

To specify a CO₂ injector the keyword [WINJGAS](#) can be used to define the composition of the injector. The composition of a water injector can also be specified by using this keyword. However, to specify the composition of an injected water stream, the keywords [WELLSTRW](#) and [WINJW](#) are available with the [CO2STORE](#) option. When using the [SOLID](#) option it is possible to specify a target solid saturation change by using item 21 of [TSCRIT](#).

If the adaptive implicit solution technique (the [AIM](#) option) is used, the [AIMCON](#) keyword can be used in order to increase the number of implicit cells allowing large stable timesteps. Typically this will be the case at the end of a simulation when CO₂ injection has stopped.

Compatibility

The [CO2STORE](#) option is currently not compatible with the multisegment well option.

Note: Salt precipitation is enabled by the use of the [SOLID](#) option. If the [SOLID](#) option is not used, salts are assumed to stay in the liquid phase. For such cases some residual water might become more and more concentrated, with salt resulting in unphysically dense water. However, this water will typically become immobile, so the overall simulation is only marginally affected. Typically this will only occur near a CO₂ injection well, due to injection of dry CO₂.

The model applies a maximum NaCl concentration for the CO₂ solubility model, corresponding to the value of salting out while other salts are limited to a fixed value of 5 Molar in the CO₂ solubility model

Thermal CO₂STORE

The [CO2STORE](#) option can be used with the [THERMAL](#) option for live oil and gas. The fluid properties are defined similarly to a standard thermal case, but the CO₂ component is allowed to dissolve in the aqueous phase. Using the [CO2STORE](#) option together with [THERMAL](#) it is assumed that the temperature range of the reservoir is limited to 12-250 °C.

If no oil K-values are supplied the phases assumed is as for a standard CO₂STORE case, a aqueous water rich phase and a CO₂ rich gas phase. The density of the CO₂ rich phase and the water vapor is computed as for the standard CO₂STORE option. Additional components are taken as water components unless the user supply water K-values for these components or define them as gas only components. This is done by the

keywords [KWTABTn](#) and [CVTYPE](#). Note that the CO₂ and H₂O component is allowed to be defaulted when using the CO₂STORE option. If this is done the usual computations for the solubilities of CO₂ and H₂O will be applied.

In THERMAL it is possible to define the H₂O components as part of the components defined in [CNAMES](#), or if left out it will be added as the last component of the system. If the H₂O component is specified it is also possible for H₂O to be in the oil phase. However this is only allowed if oil K-values are supplied using [KVTABTn](#), where you need to supply the split of H₂O between oil and vapor.

Note: With thermal CO₂STORE, the aqueous mole fractions are named [AMF](#), while [XMF](#) refers to the oil phase mole fractions.

Density and viscosity

The density of the aqueous phase is computed as described by the keyword [DENAQQA](#), equation [27.3](#)) and [VISCAQA](#), equation [27.5](#). Default values will be assigned for CO₂/CH₄, NACL/CACL₂/CACO₃ component. The vapor phase density is computed as for a standard thermal case, but the CO₂ and H₂O contribution to the phase specific volume is computed internally.

Similarly for vapor phase viscosity the CO₂ and H₂O contribution when calculating the phase viscosity are computed internally unless a component specific value for the viscosity is assigned.

Enthalpy

The aqueous enthalpy is computed by the parameters given in the keywords [SPECHW1](#) and [SPECHW2](#).

For the gas phase the usual parameters given for a thermal live oil run should be supplied, note that for CO₂ the parameters defining the enthalpies for both an oil and a vapor should be given even if not an oil phase is considered.

The H₂O molar enthalpy in the vapor phase is computed internally.

Relative permeability

The relative permeability need to be of the three phase type. If not an oil phase is considered, the oil relative permeability need to be assigned dummy values.

Note: [WSF](#) and [GSF](#) cannot be used to specify the relative permeabilities.

Initializing the study

The typical conditions is that the reservoir is water. It is possible to assign the initial aqueous mole fractions by the [AMFVD](#) keyword when using the [EQUIL](#) keyword. If the reservoir is initialized by enumeration the aqua mole fractions are assigned by the [AMF](#) keyword.

Compatibility

The thermal CO₂STORE option is currently not compatible with the multisegment well option.

10

CO₂/H₂S solution in the aqueous phase

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The CO₂ and H₂S solution algorithm allows carbon dioxide and/or hydrogen sulfide to dissolve in the aqueous phase. This is switched on using respectively the [CO2SOL](#) and/or [H2SSOL](#) keyword in the RUNSPEC section. The basic model is a fugacity function for aqueous CO₂ and H₂S which is constructed to match solubility data and which takes the form:

$$f_{CO_2}^A = P a_{CO_2} \varphi_{CO_2}(P) \quad \text{Eq. 10.1}$$

$$f_{H_2S}^A = P a_{H_2S} \varphi_{H_2S}(P)$$

The phase equilibrium between the aqueous CO₂ or H₂S and the hydrocarbon phases is then defined by the conditions that the fugacity values are equal. It is also possible to construct a CO₂ and H₂S aqueous phase Gibbs energy contribution.

The water phase properties may be entered using the [SOLUBILI](#) keyword. Alternatively, a default set of data is provided for CO₂:

- The CO₂ solubility is calculated from the correlations of Chang, Coats and Nolen [Ref. 18].
- The effects of salt and CO₂ on water density are calculated using Ezrokhi's method [Ref. 133]
- Water compressibility and viscosity are taken from the [PVTW](#) data.

For H₂S no default values exist.

To obtain details of the solubility data used in a simulation the SOLU mnemonic may be specified with the [RPTPROPS](#) keyword.

The function $\varphi(P)$ is constructed by considering a pure CO₂ or H₂S/aqueous mixture. The gas phase fugacity is obtained using the equation of state.

The initial dissolved CO₂ or H₂S solubility as a function of depth may be specified using the [RSWVD](#) keyword. This initial solution honors this value unless it exceeds the saturated CO₂ concentration in which case the saturated value is used.

Notes

- The initial CO₂ or H₂S concentration may not be in exact equilibrium with the hydrocarbon phases and initial flash modifies the input values slightly.
- The first line of the solubility table defined using the [SOLUBILI](#) keyword represents water at standard conditions, when the dissolved CO₂ or H₂S amount is at a minimum. The simulator does not assume that the amount of CO₂ or H₂S in water at the lowest pressure is zero; if you want to specify it as such then you should set it to zero in the first line of the table. The data of this first line can be set to obtain a satisfying density of water with a small amount of dissolved CO₂ or H₂S.

For further information see "[CO2SOL](#)".

Water solubilities of both CO₂ and H₂S

If both components are made soluble this should be entered by the keywords [SOLUCO2H](#) and [SOLUH2SC](#) specifying the solubility of CO₂ in the presence of H₂S and the solubility of H₂S in the presence of CO₂. Please note that no default values exist in this case. If only CO₂ or H₂S is made soluble the keyword [SOLUBILI](#) is used to specify either CO₂ or H₂S as the component reacting with the aqueous phase.

Note: The CO₂ component is always regarded as the first component followed by the H₂S component when entering composition dependent data with [RSW](#) or [AMF](#).

Water component properties

After computing the CO₂ aqueous mole fraction using the phase equilibrium algorithm mentioned previously, the saturated formation volume factor FVF_{sat} , the water compressibility C_{sat} , the water viscosity μ_{sat} and the water saturation pressure P_{sat} can be calculated using the CO₂-saturated water properties tables given by either [SOLUBILI](#), [SOLUBILS](#) or [SOLUBILT](#) using linear interpolation.

The formation volume factor FW at pressure P is calculated by:

$$\frac{1}{FVF} = \frac{1}{FVF_{sat}} \cdot (1 + X_c + 0.5X_c^2)$$

where the compressibility term X_c is given by

$$X_c = c_{sat} \cdot (P - P_{sat})$$

Density

The reservoir density of water as given by the mnemonic [DENW](#) of [RPTRST](#) or [RPTSOL](#) is given by:

$$\frac{\rho_s}{MW} \cdot \frac{MW_r}{FVF} \quad \text{Eq. 10.2}$$

where

ρ_s is the surface density, specified with either keywords [DENSITY](#), [GRAVITY](#) or [SOLUBILS](#).

MW is the molecular weight of pure water (18.015 kg/kg-M).

MW_r is the molecular weight of water in the reservoir calculated using the aqueous mole fraction of CO₂ (AMF_{CO_2}) and water and their molecular weights ($AMF_{CO_2} \cdot MW_{CO_2} + (1-AMF_{CO_2}) \cdot MW$). The molecular weight of CO₂ is specified using the keyword [MW](#).

FVF is the formation volume factor calculated as explained earlier.

Viscosity

The viscosity of water at pressure P is given by

$$\frac{\mu_{sat}}{(1 + Y_c + 0.5Y_c^2)} \quad \text{Eq. 10.3}$$

where the term Y_c is given by

$$Y_c = -c_V \cdot (P - P_{sat})$$

where

c_V is the “viscosibility”, specified with keyword [PVTW](#)

Aquifer properties

An analytic water aquifer defined by [AQUCHWAT](#), [AQUCT](#), [AQUFETP](#) or [AQUFLUX](#) contains only the “WATER” component. Any water component is allowed to leave the reservoir. If item 131 of the OPTIONS3 keyword is set to 1, this reverts to the pre-2008.1 behavior where the water entering or leaving the reservoir is made up entirely of the “WATER” component. Additionally, the water properties of the analytic aquifers are calculated as described above. If the item 119 of OPTIONS3 is set to 1, this reverts to the pre-2008.1 behavior where the water PVT tables were used. See "[Aquifer modeling facilities](#)".

11

Chemical reactions

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

This chapter describes how chemical reactions are modeled in the ECLIPSE 300 simulator. Chemical reactions can only be used in fully implicit runs. They can be used in blackoil runs and compositional runs (EoS, K-value and thermal).

The keywords for chemical reactions can be used to model a number of other processes, for example:

- combustion
- biodegradation
- decay of radioactive tracers
- non-equilibrium reactions
- foam decay and adsorption.

Formulation

The implementation of chemical reactions in ECLIPSE follows [Ref. 24].

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

$$\sum (S_{Rri} \cdot C_i) \rightarrow \sum (S_{Pri} \cdot C_i) \quad \text{Eq. 11.1}$$

where:

S_{Rri} is the stoichiometric coefficient for the reactant C_i ,

S_{Pri} is the stoichiometric coefficient for the product C_i .

The stoichiometric coefficients are defined by the [STOREAC](#) and [STOPROD](#) keywords. The phase of the reactants can be specified by the [REACPHA](#) keyword.

Reaction rate

The reaction rate R_r for reaction r in a grid cell is given by an Arrhenius equation:

$$R_r = V_b \cdot A_r \cdot e^{(-E_r)/(RT)} \cdot \prod c_{ri}^{n_{ri}} \quad \text{Eq. 11.2}$$

where:

V_b is the bulk volume (rock volume plus pore volume),

A_r is a reaction rate constant, defined by [REACRATE](#),

E_r is the activation molar energy, defined by [REACACT](#),

R is the gas constant,

T is the temperature which may be limited using [REACLIMS](#),

n_{ri} is the index or order of the component dependent term, defined by [REACCORD](#).

Component concentration

The terms c_{ri} represent the component concentration in the reacting phase ([REACCPHA](#)). If the reactant is in the oil phase:

$$c_{ri} = \theta \cdot b_o \cdot S_o \cdot x_i \quad \text{Eq. 11.3}$$

If the reactant is in the gas phase:

$$c_{ri} = \theta \cdot b_g \cdot S_g \cdot y_i \quad \text{Eq. 11.4}$$

If the reactant is water in the water phase:

$$c_{ri} = \theta \cdot b_w \cdot S_w \cdot a_i \quad \text{Eq. 11.5}$$

If the reactant is in all phases:

$$c_{ri} = \theta \cdot m_i \quad \text{Eq. 11.6}$$

Here

θ is the porosity,

b_o , b_g and b_w are the molar densities of oil, gas and water;

S_o , S_g and S_w are the oil, gas and water saturations;

x_i , y_i , a_i and m_i are the mole fractions in the oil, gas and water 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 [11.4](#):

$$c_{ri} = P \cdot y_i \quad \text{Eq. 11.7}$$

where P is the pressure and can be limited using the [REACLIMS](#) keyword.

If the reactant is in the solid phase, [11.6](#) can be used - since at present a solid component cannot appear in any other phase.

Component concentration order

The order n_{ri} of the component term in equation [11.2](#) should be non-negative for all reactants. The order for a reactant can be set to zero, in which case the reaction rate is independent of the concentration of that component; however, this is not recommended, unless keyword [EQLDREAC](#) is also used. If the order of a component dependent term is non-zero, and that component is not a reactant, then it acts as a catalyst or inhibitor.

To improve stability when simulating zero-order reactions, the reaction rate is multiplied by the scaling factor $m / (m + s)$ where m is the molar density of the reactant and s is given by item 4 of keyword [REACOPS](#). Please note that the default value of s (1) is an experimental value and may be adjusted to ensure that the scaling factor $m / (m + s)$ tends to zero with the concentration of the reactant.

Porosity order

In equations [11.3](#) - [11.6](#), the component terms c_{ri} represent the concentrations per cell volume. In ECLIPSE it is more natural to consider the concentrations per pore volume:

$$c'_{ri} = c_{ri} / \theta$$

The reaction rate, equation [11.2](#) can then be rewritten as

$$R_r = V_p \cdot \theta^{n_{rp}} \cdot A_r \cdot e^{(-E_r/(RT))} \cdot \prod c'_{ri}^{n_{ri}} \quad \text{Eq. 11.8}$$

where:

V_p is the pore volume, $V_p = \theta \cdot V_b$

n_{rp} is the index or order of the porosity term given by $n_{rp} = \sum_i n_{ri} - 1$

Here the sum is over all reaction components **except** those where the rate is defined by the partial pressure [11.7](#). If necessary the porosity order can be modified using the [REACPORD](#) keyword.

Reactions involving solid components

In simulations with solid components the fluid saturations are modified to include the solid saturation \hat{S}_s :

$$\hat{S}_o + \hat{S}_g + \hat{S}_w + \hat{S}_s = 1$$

Normalized fluid saturations can then be defined

$$S_o = \hat{S}_o / (1 - \hat{S}_s)$$

$$S_g = \hat{S}_g / (1 - \hat{S}_s)$$

$$S_w = \hat{S}_w / (1 - \hat{S}_s)$$

Equations [11.3](#) - [11.5](#) are modified, for example [11.3](#) becomes:

$$c_{ri} = \theta \cdot b_o \cdot \hat{S}_o x_i = \theta \cdot b_o \cdot S_o \cdot x_i \cdot (1 - \hat{S}_s)$$

Equation 11.8 then becomes

$$R_r = V_p \cdot \theta^{n_{rp}} \cdot (1 - \hat{S}_s)^{n_{rs}} \cdot A_r \cdot e^{(-E_r/(RT))} \cdot \prod c_{ri}^{n_{ri}}$$

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, equation 11.7 or where the reactant is in all phases, equation 11.6. If necessary the values of n_{rs} , can be modified using the [REACCORD](#) keyword.

Problems may occur in the matrix inversion if the solid saturation is close to 1. To prevent this, the solid dependent term in the reaction rate is replaced by

$$\left(1 - \frac{\hat{S}_s}{\hat{S}_{s\max}}\right)^{n_{rs}}$$

The value of $\hat{S}_{s\max} \leq 1$ can be defined using the [REACSMAX](#) keyword. As the solid saturation approaches the limit $\hat{S}_{s\max}$, the reaction rate tends to zero.

Equilibrium deviation reaction rates

The reaction rate can contain extra terms that depend on the deviation from an equilibrium value. Equation 11.2 is replaced with:

$$R_r = V_b \cdot A_r \cdot e^{(-E_r/(RT))} \cdot \prod c_{ri}^{n_{ri}} \cdot \Pi D_{mijk} \quad \text{Eq. 11.9}$$

The additional term ΠD_{mijk} is the product of one or more terms of the form:

$$D_{mijk} = \text{sign}(d_{mijk}) \cdot |d_{mijk}|^n \quad \text{Eq. 11.10}$$

The equilibrium deviation terms d_{mijk} are given by one of the following models:

$$d_{1ijk} = \theta \cdot b_o \cdot S_o \cdot (x_i - y_j / K_k) \quad \text{Eq. 11.11}$$

$$d_{2ijk} = \theta \cdot b_g \cdot S_g \cdot (K_k \cdot x_i - y_j) \quad \text{Eq. 11.12}$$

$$d_{3ijk} = \theta \cdot b_w \cdot S_w \cdot (a_i - 1 / K_k) \quad \text{Eq. 11.13}$$

$$d_{4ijk} = \theta \cdot (F_k(a_i) - C_a) \quad \text{Eq. 11.14}$$

In these equations x_i is a component oil mole fraction, y_j is a component gas mole fraction, a_i is a water component mole fraction and C_a is the adsorbed solid concentration. The [EQLDREAC](#) keyword should be used to define the values of m , i , j , k and n , where:

m determines which of equations 11.11 - 11.14 is used,

i is the first component number,

j is the second component number,

k is either

- the number of the equilibrium k-value, K_k , defined by the [EQLDKVCR](#) keyword
- the number of the equilibrium table F_k , defined by the [EQLDTAB](#) keyword

n is the order of the deviation term in equation 11.10.

The equilibrium position is given by $x_i = y_j / K_k$, $a_i = 1 / K_k$ or $C_a = F_k(a_i)$. The equilibrium rate term D_{mijk} is zero when the components i and j are in equilibrium, and can be positive or negative or both. The reaction stoichiometry should be set so that component i reacts to form component j for positive reaction rates. The direction of the reaction can be restricted to forwards, in which case equation 11.10 is replaced by:

$$D_{mijk} = (\max(0, d_{mijk}))^n \quad \text{Eq. 11.15}$$

or backwards, in which case equation 11.10 is replaced by:

$$D_{mijk} = -(\max(0, -d_{mijk}))^n \quad \text{Eq. 11.16}$$

The component numbers i and j can be set to 0:

- If the component number i is set to zero, the oil mole fraction x_i is replaced by a value of 1 in equations 11.11 and 11.12, and the water mole fraction a_i is replaced by a value of 1 in equations 11.13 and 11.14.
- If the component number j is set to zero, the gas mole fraction y_i is replaced by a value of 1 in equations 11.11 and 11.12. j should be set to zero if equation 11.13 or equation 11.14 is used.

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 values that are defined by the [EQLDKVCR](#) keyword.

The equilibrium deviation terms D_{mijk} 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 equation 11.9 becomes

$$R_r = V_b \cdot A_r \cdot e^{(-E_r/(RT))} \cdot \prod D_{mijk} \quad \text{Eq. 11.17}$$

This reaction type is primarily intended for use with “inert” components, such as non-volatile or incondensable components in thermal simulations, or solid or water components. If it is used with live components, then there is a possibility that the reaction will compete with the flash.

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. Equation 11.2 is replaced with:

$$R_r = V_b \cdot A_r \cdot e^{(-E_r/(RT))} \cdot \prod c_{ri}^{n_{ri}} \cdot \prod T_{rj} \quad \text{Eq. 11.18}$$

The tracer multiplier terms T_{rj} are calculated explicitly (at the previous timestep) as

$$T_{rj} = (\theta \cdot t_j \cdot c_c)^{n_{rj}} \quad \text{Eq. 11.19}$$

where

t_j is the tracer concentration

c_c is the concentration of the carrier component c in the tracer phase p_j

n_{tj} is the order of the tracer term.

If the tracer phase is oil then:

$$c_c = b_o \cdot S_o \cdot x_c \quad \text{Eq. 11.20}$$

If the tracer phase is gas:

$$c_c = b_g \cdot S_g \cdot y_c \quad \text{Eq. 11.21}$$

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

$$c_c = b_w \cdot S_w \quad \text{Eq. 11.22}$$

If the tracer is in all phases:

$$c_c = m_c \quad \text{Eq. 11.23}$$

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 **RPTSCHE**.

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:

$$R_{fl} = \frac{d}{dt} (V_p m_{fl}) - \sum_r [(S_{Pri} - S_{Rri}) \cdot R_r] = 0 \quad \text{Eq. 11.24}$$

where the flow terms between cells and to and from wells have been ignored for simplicity, see "Formulation of the equations".

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:

$$R_e = \frac{d}{dt} (V_b e) - \sum_r [H_r \cdot R_r] = 0 \quad \text{Eq. 11.25}$$

where:

V_b is the bulk volume,

H_r is the reaction molar 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, equation 11.24 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 molar enthalpy H_r should also be multiplied by N since N reactions produce N times as much energy. This ensures the energy balance equation, equation 11.25 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.

Reaction rate units

The units of the rate constant A_r are not fixed but are chosen to ensure that the units of the reaction rate R_r defined by 11.2 are correct when used in equation 11.24. 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 per unit volume R_r are:

- kg-M/day (METRIC)
- lb-M/day (FIELD)
- gm-M/hr (LAB)
- kg-M/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 [REACCRT](#) keyword for the temperature used in the Arrhenius term in equation 11.2:

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

and the oil saturation used in equation 11.3:

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

Output

The reaction rate equation 11.2 can be output to the summary file or the RESTART file for graphical display or to the print file.

To output the reaction rate to the summary file the mnemonics BREAC, FREAC and FREAT can be used in the SUMMARY section.

BREAC outputs the reaction rate for specified cells and specified reactions.

FREAC outputs the reaction rate for the whole field for specified reactions.

FREAT outputs the reaction total since the beginning of the simulation for the whole field for specified reactions.

If the reaction number is defaulted in keywords FREAC or FREAT, then data are output for all reactions.

For example:

```
BREAC
-- I J K R
 3 4 5 1 / -- Rate for reaction 1 in cell (3,4,5)
 3 4 5 2 / -- Rate for reaction 2 in cell (3,4,5)
 6 7 8 2 / -- Rate for reaction 2 in cell (6,7,8)
/
FREAC
1 2 / -- Reaction rates summed over all cells for reactions 1 and 2
FREAT
/ -- Reaction totals summed over all cells for all reactions
```

To output the reaction rate to the RESTART file the REAC mnemonic can be used with the [RPTRST](#) keyword. The reaction rate is printed for each cell and each reaction. For example:

```
RPTRST
REAC/
```

To output the reaction rate to the print file the REAC mnemonic can be used with the [RPTSCHED](#) keyword. The reaction rate is printed for each cell and each reaction. For example:

```
RPTSCHED
REAC/
```

Chemical reaction keywords

RUNSPEC

- [REACTION](#) Number of reactions

PROPS

Restriction	Keyword	Description
	EQLDKVCR	K-values for equilibrium deviation reactions
	EQLDREAC	Equilibrium deviation reaction rates
	EQLDTAB	Equilibrium values for water-solid deviation reactions
	REACACT	Reaction activation molar energy
Thermal option.	REACENTH	Reaction molar enthalpy

Restriction	Keyword	Description
	REACCORD	Reaction rate component order
	REACLIMS	Limits on pressures and temperatures used to calculate rates
	REACOPS	Options and convergence criteria
	REACPHTA	Reactant phase
	REACPORD	Reaction rate porosity term order
	REACRATE	Reaction rate constant
Solid option	REACSMAX	Reaction rate solid saturation limit.
Solid option.	REACSORD	Reaction rate solid order.
	REACTRAC	Tracer dependent reaction rate terms
	STOPPROD	Stoichiometric coefficients of products
	STOREAC	Stoichiometric coefficients of reactants

SUMMARY

Field	Block	Description
FREAC	BREAC	Reaction rate
FREAT		Reaction total

SCHEDULE

Restriction	Keyword	Description
Solid option or Thermal option.	REACCRIT	Reaction criteria

The RPTRST and RPTSCHED arguments are:

Keyword	Description
TREACM	Reaction rate for each cell
REAC	Tracer dependent reaction rate multiplier for each cell

Examples

Example 1: Combustion

[Ref. 24] gives a thermal example with 5 components plus water:

Component	Name
C1	$\text{C}_{32}+\text{C}_{47}$
C2	C_{20}

Component	Name
C3	C ₆ +C ₈ +C ₁₂
C4	O ₂
C5	N ₂ -CO _X
W	H ₂ O

Table 11.1: Example names for components

Reaction	Stoichiometry
1	C1 + 45.17C4 -> 37.46 C5 + 29.71 W
2	C2+25.08 C4 -> 20.8 C5 + 16.5 W
3	C3 + 11.66 C4 -> 9.65 C5 + 7.67 W

Table 11.2: Example stoichiometry

Reaction	Rate constant	Activation Molar Energy (Btu/lb-M)	Reaction Molar Enthalpy (Btu/lb-M)
1	3.4054 x 10 ¹⁰	32785	16000
2	0.28164 x 10 ¹⁰	32785	16000
3	0.4035 x 10 ¹⁰	32785	16000

Table 11.3: Example chemical reaction parameters

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

STOPPROD
-- 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

STOPPROD
-- 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)

EQLDKVR
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

```

12

Coal bed methane model

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The naturally occurring methane within coal seams can in some cases be economically produced using oil field technology. In these “Coal Bed Methane” projects, wells are drilled into the coal seam to produce gas. Considerable reserves of this “unconventional” natural gas are present in many parts of the world.

The coal beds are naturally fractured systems with the gas adsorbed into the coal matrix. Primary production occurs by initially de-watering the natural fractures and hence reducing the pressure in the fracture system. The reduced pressure in the fractures allows gas desorption from the surface of the coal to the fracture. Gas diffuses from the bulk of the coal towards the fracture surface.

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

In ECLIPSE 100 it is possible to introduce a second gas, known as a solvent, for enhanced Coal Bed Methane projects, while in ECLIPSE 300 a full compositional treatment is possible.

CAUTION: You should be aware that certain entities, corporations, individuals or organizations may or do hold applicable patents with regard to enhanced recovery of coal bed gases through injection of both inert and non-inert gases. You are strongly encouraged to investigate if any proposed field applications are subject to these patents.

Single porosity model

In ECLIPSE 300 it is possible to activate an instant sorption model that can be used in single porosity mode. With this model the amount of adsorbed gas is immediately in equilibrium with the free gas phase.

Dual porosity model

The dual porosity model consists of two interconnected systems representing the coal matrix and the permeable rock fractures. To model such systems, two simulation cells are associated with each block in the geometric grid, representing the coal matrix and fracture volumes of the cell. In ECLIPSE, the properties of these blocks may be independently defined. In an ECLIPSE dual porosity run, 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.

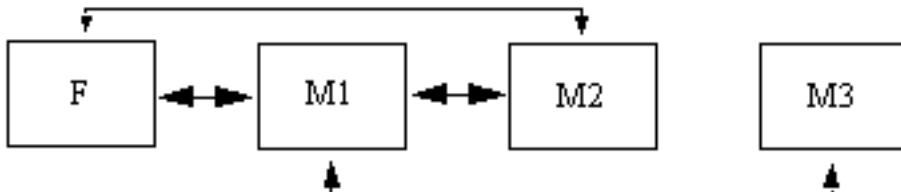
The [DUALPERM](#) option should only be used if the instant adsorption model in ECLIPSE 300 is activated ([CBMOPTS](#)) or if the time-dependent adsorption model is used with multi-porosity. With the time-dependent sorption model, one of the submatrix porosities will then represent the pore volume of the matrix having conductivity to neighbor matrix grid blocks, while grid cells assigned to be coal will get a zero transmissibility.

Note: In a coal bed methane model the pore volume of the matrix cells has a different interpretation than for an ordinary dual porosity run; it gives the coal volume of the cell, using the time-dependent sorption model. By default the porosity is set to unity minus the porosity of the fracture. The cell bulk volume multiplied by the porosity then equals the coal volume. It is possible to use the keyword [ROCKFRAC](#) instead of the porosity to set the coal volume in ECLIPSE 300.

Note: By default the input permeabilities in the fracture are scaled by the porosities. To turn this off use the keyword [NODPPM](#).

Triple porosity model

In ECLIPSE 300, the triple porosity model, [TRPLPORO](#), can be activated together with the Coal Bed Methane option. The triple porosity model offers an extra connection between the second submatrix (M2) and the fracture (F) and, if using four porosities, from the “isolated” third submatrix (M3) to the first submatrix (M1) as shown in the diagram below.



This enables the modeling of conventional flow in micro pores or mini fractures within the coal matrix together with diffusion of coal gas as formulated by the time-dependent sorption model. Different setups are possible depending on the setting of adsorbed gas in the submatrix. For the transmissibility and/or diffusivity between the M2-F connection, [SIGMAV](#) values are allowed to be assigned to the fracture grid cells. Note that to model dual permeability, the connection transmissibilities could be set by [PERMMF](#) instead of using the matrix [PERMX](#) values, but K-layer indices in the [PERMMF](#) case only run over matrix porosities while for the triple porosity model the [SIGMAV](#) values include the fracture porosity. For more information see "[Triple Porosity](#)" and also consider the example data set [CBMTRPL.DATA](#).

Multi porosity model

In ECLIPSE 300 a multi porosity model can be activated, enabling a detailed study of the transient behavior in the matrix. See "[Multi porosity](#)".

Adsorption/diffusion models in ECLIPSE 100

The adsorbed concentration on the surface of the coal is assumed to be a function of pressure only, described by a Langmuir isotherm. The Langmuir isotherm is input as a table of pressure versus adsorbed concentrations. Different isotherms can be used in different regions of the field. See keyword [COALNUM](#).

The diffusive flow of gas from the coal matrix is given by:

$$F_g = \text{DIFFMF} \cdot D_c \cdot (\text{GC}_b - \text{GC}_s) \quad \text{Eq. 12.1}$$

where

F_g = gas flow

DIFFMF = matrix-fracture diffusivity

D_c = diffusion coefficient

GC_b = bulk gas concentration

GC_s = surface gas concentration.

GC_s is a function of fracture pressure given by the [LANGMUIR](#) table. The diffusion coefficient D_c is supplied using the [DIFFCOAL](#) keyword.

The diffusivity is given by:

$$\text{DIFFMF} = \text{DIFFMMF} \cdot \text{VOL} \cdot \sigma \quad \text{Eq. 12.2}$$

where

DIFFMMF is the multiplying factor input using [DIFFMMF](#) (default = 1.0)

VOL is the cell coal volume

σ is the factor to account for the matrix-fracture interface area per unit volume.

Sigma is a shape factor which accounts for the matrix fracture interface area per unit volume of coal matrix. For pseudo-steady-state flow in the coal matrix, the gas desorption time (time) is

$$\tau = \frac{1}{\sigma \cdot D_c \cdot \text{DIFFMMF}}$$

If the surface gas concentration is greater than the bulk gas concentration, gas may be re-adsorbed into the coal. In this case the flow of gas is treated slightly differently, as shown below. It is also possible to control the rate of re-adsorption and, in the limiting case, to prevent re-adsorption.

$$F_g = \text{DIFFMF} \cdot D_c \cdot S_g \cdot \text{RF} \cdot (\text{GC}_b - \text{GC}_s) \quad \text{Eq. 12.3}$$

where

S_g is the gas saturation in the fracture

RF is the re-adsorption factor, input using the [DIFFCOAL](#) keyword (default = 1.0).

Gas in the fracture will not contact the entire surface area of the coal if water is also present in the fracture. Hence the flow of gas into the coal is reduced by S_g to represent the reduced contact area.

The RF factor can be used to control the rate of re-adsorption. Setting RF = 0 prevents gas re-adsorption.

Adsorption models in ECLIPSE 300

The sorption models that can be used are:

- Table input, the adsorption capacities for each component are input by the [LANGMUIR](#) keyword.
- The Extended Langmuir isotherm. See the keyword [LANGMEXT](#).

Table input

Using the keyword [LANGMUIR](#), the adsorption capacities of each component are given versus pressure. Table lookup is done using the partial pressure, in order to determine the adsorption capacity of each component. The first row of the table is recommended to be at zero (partial) pressure, defining the minimum values to be zero. Thus if the component is not present the adsorption capacity is set to zero.

The partial pressures, used to look up the adsorption capacities, are scaled with corresponding [LANGMPL](#) values if this keyword is present.

Extended Langmuir isotherm

The Extended Langmuir isotherm, [Ref. 5], can be used to describe the coal sorption for the different components. See keyword [LANGMEXT](#).

The adsorption capacity is a function of the pressure and the free gas phase composition. For each component, two parameters need to be input: the Langmuir volume constant V_i and the Langmuir pressure constant P_i . These parameters are typically determined from experiments. Different isotherms can be used in different regions of the field. See keyword [COALNUM](#).

The multi-component adsorption capacity is calculated by:

$$L(p, \alpha, y_1, y_2, \dots)_i = \theta \frac{P_s}{RT_s} \left(V_i \frac{\frac{y_i \alpha p}{P_i}}{1 + \sum_{j=1}^{nc} \frac{y_j \alpha p}{P_j}} \right)$$

where

θ = Scaling factor

P_s = Pressure at standard conditions

R = Universal gas constant

T_s = Temperature at standard conditions

V_i = Langmuir volume constant for component i

P_i = Langmuir pressure constant for component i

y_i = Hydro carbon mole fraction in gas phase for component i

p = Pressure

α = Pressure multiplier as given by [LANGMPL](#)

For the special case of a single component, the Extended Langmuir isotherm is identical to the usual Langmuir isotherm giving the storage capacity as a function of pressure only:

$$L(p, \alpha) = \theta \frac{P_s}{RT_s} \left(V \frac{\frac{\alpha p}{P}}{1 + \frac{\alpha p}{P}} \right) \quad \text{Eq. 12.4}$$

where V is the maximum storage capacity for the gas, referred to as the Langmuir volume constant, and P is the Langmuir pressure constant. The constants used in the Extended Langmuir formulation can hence be estimated from a series of single-component gas experiments.

Since the Langmuir volume constant V is given as surface volume over coal weight, we translate this to moles over coal weight using the ideal gas law (with the term $\frac{P_s}{RT_s}$).

It is possible to scale the adsorption capacity by a factor θ for each cell in the grid with the [LANGMULT](#) keyword. Typically this can be used to account for differences in ash or moisture contents. To apply a different scaling for each component the keyword [LANGMULC](#) can be used.

Note that [LANGMULT](#) and [LANGMULC](#) cannot be used together.

Time dependent diffusion in ECLIPSE 300

The diffusive flow between the matrix and the fracture is given by

$$F_i = DIFFMF \cdot D_{c,i} \cdot S_g \cdot RF_i \cdot (m_i \cdot \rho_c L_i) \quad \text{Eq. 12.5}$$

where

m_i = Molar density in the matrix coal.

$DIFFMF$ = Matrix fracture (or multi porosity) diffusivity.

ρ_c = Rock density (Coal density), specified using [ROCKDEN](#).

$D_{c,i}$ = Diffusion coefficient (coal) component i, specified using [DIFFCBM](#).

RF_i = Readsorption factor component i, specified using [RESORB](#).

S_g = Gas saturation. For desorption, a value of unity is used.

$\rho_c L_i$ = Equilibrium molar density of adsorbed gas, specified using [LANGMUIR](#) or specified using [ROCKDEN](#) for the coal density ρ_c with [LANGMEXT](#) for L_i

As can be seen by the flow equation and as noted in the above table the density of the coal needs to be specified, as this is multiplied by the Langmuir adsorption capacity. The term $\rho_c L_i$ gives the equilibrium adsorbed molar density. The coal density ρ_c is entered with the keyword ROCKDEN. When using the **LANGMUIR** keyword to specify the sorption, the coal density is not used as the input is then given as a concentration (i.e. $\rho_c L_i$ is entered directly).

For adsorption, the area of the gas in contact with the coal is accounted for by scaling with the gas saturation. (It is possible to use a value of unity instead with item 93 of the OPTIONS3 keyword)

The matrix-fracture diffusivity is given by

$$\text{DIFFMF} = \text{DIFFMMF} \cdot \text{VOL} \cdot \sigma \quad \text{Eq. 12.6}$$

where

DIFFMMF is the multiplying factor input using **DIFFMMF** (default = 1.0)

VOL is the coal volume

σ is the factor to account for the matrix-fracture interface area per unit volume.

Often the component's sorption time is a quantity that is easier to obtain than the diffusion coefficients. For desorption we write the flow as

$$F_i = \frac{\text{VOL}}{\tau_i} \cdot (m_i - \rho_c L_i) \quad \text{Eq. 12.7}$$

where

$$\tau_i = \frac{1}{D_{c,i} \cdot \text{DIFFMMF} \cdot \sigma}$$

is called the sorption time. The parameter controls the time lag before the released gas enters the coal fracture system. The sorption times are given by the diffusion coefficients, DIFFCBM, and the matrix-fracture interface area, **SIGMA**, together with the multiplying factor DIFFMMF. If the sorption times are known a value of unity can be assigned to σ and DIFFMMF. The diffusion coefficients can then be assigned to the reciprocal of the sorption times.

The time dependent diffusive flow can be used with

- Dual porosity
- Multi porosity

It is possible to have connections between coal cells having the same coal region numbers. For connections between cells having different adsorption properties it is assumed that a pore volume cell is placed in between.

Diffusion between matrix-matrix cells

Using the multi-porosity option, a coal cell can connect to a matrix pore volume cell. The diffusion between two cells I and J is computed as:

$$F_i^{I \rightarrow J} = \text{DIFF}^{IJ} \cdot D_{c,i} \cdot S_g \cdot RF_i \cdot (m_i^I - \rho_c L_i^J)$$

where we assume I to be the coal cell. Note that the innermost sub-matrix grid cell is used when calculating the diffusivity.

$$\text{DIFF}^{IJ} = \text{DIFFMMF}^I \cdot \text{VOL}^I \cdot \sigma^I$$

where the index I corresponds to the index of the sub-matrix grid-cell I. If cell I corresponds to a pore volume cell, the volume used is the pore volume. The diffusivity multipliers and sigma factors are specified by the keywords `DIFFMMF` and `SIGMAV`, while the volumes are input by the appropriate `ROCKFRAC` or `PORO` keywords.

For pre-2012.1 versions of ECLIPSE, the volume used also corresponds to the coal volume for inner matrix sub-grid cells. To obtain this behavior, use item 278 in the `OPTIONS3` keyword. For some combinations of the coal options, as for example with the Triple Porosity option, this might still be useful. The sigma factor then needs to be altered to account for the two connections from the coal volume.

Diffusion between coal cells

Using the multi-porosity option, a coal cell can connect to another coal cell. This is allowed if the grid cells have the same coal region number, thus having the same sorption properties. The potential for diffusion between two coal cells I and J are computed by using the molar density, m_i^I and m_i^J , in the coal volumes

$$F_i^{I \rightarrow J} = \text{DIFF}^{IJ} \cdot D_{c,i} \cdot (m_i^I - m_i^J)$$

The innermost sub-matrix grid cell is used when calculating the diffusivity.

$$\text{DIFF}^{IJ} = \text{DIFFMMF}^I \cdot \text{VOL}^I \cdot \sigma^I$$

Instant sorption in ECLIPSE 300

If the sorption time approaches zero, the adsorbed gas will be in instant equilibrium with the free gas phase. The keyword `CBMOPTS` is used to activate the instant sorption model.

The instant sorption model can be used with

- Single porosity
- Dual porosity
- Multi-porosity

With this model every cell is assumed to contain a pore volume, the remaining fraction of the bulk volume being taken as the coal volume. For dual and multi-porosity models this is usually not appropriate as each pore system will add up the rock volume. In this case an average rock volume over the pore systems is calculated and partitioned among the connected pore systems. The default behavior can be changed by using the keyword `ROCKFRAC` to specify the fraction of the bulk volume of each simulation grid cell making up the coal volume. The sum of the porosities and the rock fractions over the connected pore systems should usually be unity.

Rock compaction

There are several models available in order to model the compression and expansion of the pore volumes and the effect on permeability for a coal bed methane reservoir.

A Palmer-Mansoori model ([Ref. 78]) is available using the `ROCKPAMA` keyword.

Options for the ROCKPAMA and [ROCKPAME](#) models can be activated by using [CBMOPTS](#) items 4-9.

For ROCKPAMA, this includes modeling the sorption induced strain term as a function of the actual adsorbed gas concentration, while by default this is estimated using an instant adsorption assumption. This option in CBMOPTS is important if the diffusion process of the sorption process is so that the standard Palmer-Mansoori model would predict swelling/shrinkage due to the instant adsorption assumption, without the support of gas exchange. For cases where the standard Palmer-Mansoori model predicts a reduction in pore volume for increased pressure, this needs to be supported by gas adsorbing into the coal.

It is possible to tabulate the sorption induced strain term versus the coal gas concentration using the keywords [TPAMEPS](#) and [TPAMEPSS](#). This also means that ROCKPAMA can be used for multi-components.

For the rock models ROCKPAMA and ROCKPAME it is possible to output the number of passes through the rebound pressure for each grid cell. The rebound pressure is when the pressure derivative of the pore volume changes sign. This output is made possible using [NPMREB](#) with [RPTSCHED](#) and [RPTRST](#), and activating the output using CBMOPTS item 4.

ECLIPSE 300

Another extension for multi-component gases is available by specifying the ROCKPAME keyword. This enables the modeling of coal swelling and shrinkage as a function of both composition and pressure, but assumes an instant adsorption process for the sorption induced strain term.

For more information see "[Rock compaction](#)".

The effect of compaction is only modeled in the pore volume. The coal volume of a simulation cell and its adsorbed contents are related to the surface gas volume per unit coal volume. For example, if [ROCKTAB](#) or [ROCKTABH](#) keyword is defined, the pore volume multiplier (as a function of block pressure – same for fracture and matrix cell) is not applied to the matrix volume of coal cells.

Modeling compositional effects with ECLIPSE 100

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 CO₂ 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 CO₂ 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:

- Langmuir isotherm for pure gas, typically methane.
The table defines the gas surface concentration versus pressure.
- Langmuir isotherm for pure solvent, CO₂ say.
The table defines the solvent surface concentration versus pressure.
- Table of the relative adsorption of gas and solvent as a function of the composition of the fracture gas.

Then:

Surface concentration of gas = $C_g(P) \cdot F_g(f_g)$

Surface concentration of solvent = $C_s(P) \cdot F_s(f_g)$

Where

$C_g(P)$ is the adsorption of pure gas at pressure P ([LANGMUIR](#) keyword)

$C_s(P)$ is the adsorption of pure solvent at pressure P ([LANGSOLV](#) keyword)

F_g is the gas relative adsorption ([COALADS](#) keyword)

F_s is the solvent relative adsorption

f_g is the fraction of gas in the fracture = $S_g / (S_g + S_{\text{solv}})$

Adsorption model 2: COALPP keyword

The second adsorption model is based on the partial pressures of the component gases. Again, three input tables are required:

- Langmuir isotherm for pure gas, typically methane.
The table defines the gas surface concentration versus pressure.
- Langmuir isotherm for pure solvent, CO₂ for example.
The table defines the solvent surface concentration versus pressure
- 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

f_g is the fraction of gas in the fracture = $S_g / (S_g + S_{\text{solv}})$

Note: Either the [COALADS](#) or COALPP keyword must be specified when the secondary recovery scheme, that is coal and solvent, is used.

Using the ECLIPSE 100 Coal Bed Methane Model

Single gas mode

The model is activated by specifying the keyword [COAL](#) in the RUNSPEC section. The dual porosity model should also be activated with the RUNSPEC keyword [DUALPORO](#), and the program should be run in gas-only or gas-water mode ([OIL](#) may not be specified in RUNSPEC).

Grid data

The first $\text{NDIVIZ}/2$ layers of the model represent the coal matrix, and the second $\text{NDIVIZ}/2$ 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:

Geometry

Either [DX](#) / [DY](#) / [DZ](#) or [ZCORN](#) / [COORD](#)

Porosity

This is used to calculate the coal volume, and is taken to represent the fraction of the volume occupied by coal. (The porosity can be defaulted, in which case it is set to 1 - fracture porosity.)

Net-to-Gross

Optional, defaults to 1.0

Coal region number

[COALNUM](#). This is optional and defaults to 1 for all the matrix cells. Note that for a matrix cell given a zero value, the cell is taken as a normal pore volume cell and porosities and permeabilities need to be assigned. The number of coal regions should be set by item 6 of [REGDIMS](#) (default =1).

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:

Keyword	Description
LANGMUIR	Inputs tables of surface gas concentration versus pressure.
DIFFCOAL	Inputs the diffusion coefficient and the re-adsorption factor.

It is possible to scale the tables of surface gas concentration versus pressure on a cell-by-cell basis using the [MLANG](#) keyword. This keyword is optional, and if it is omitted the raw table data is used.

Equilibration

If the model is equilibrated using the [EQUIL](#) keyword, no extra data is required. In this case the initial coal gas concentration is set to the equilibrium concentration at the prevailing fracture pressure. Two optional keywords are available to set the initial coal gas concentration manually:

Keyword	Description
GASCONC	Inputs the coal gas concentration on a block-by-block basis.
GCVD	Inputs the coal gas concentration as a function of depth.

Care should be taken when using non-equilibrium starts. ECLIPSE prints a warning if the coal gas concentration exceeds the equilibrium value in any grid block.

For enumerated starts, 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:

Keyword	Description
FCGC	Field coal gas concentration.
RCGC	Region coal gas concentration.
BCGC	Block coal gas concentration.
BPORVMOD	Block pore volume multiplier, rock compaction models.
BPERMMOD	Block permeability multiplier, rock compaction models.

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:

Keyword	Description
SDENSITY	Solvent surface density
PVDS	Solvent PVT tables
SSFN	Gas/solvent relative permeability data for flow in the fractures.

The adsorption process is described by two extra keywords:

Keyword	Description
LANGSOLV	Langmuir isotherm for pure solvent
COALADS or COALPP	Relative adsorption data for gas and solvent.

The `LANGSOLV` data can be scaled using the `MLANGSLV` keyword in a manner analogous to the `MLANG` keyword.

Either the `COALADS` keyword or the `COALPP` keyword can be used to specify the relative adsorption data. If the `COALADS` keyword is used, the relative adsorption is specified as a table of relative adsorption factors. The alternative `COALPP` keyword inputs a table of pressure factors for the partial pressure model.

Equilibration

By default the initial solvent fraction is taken to be zero. It is possible to specify an initial solvent fraction using one of two keywords:

Keyword	Description
<code>SOLVFRAC</code>	Inputs the initial solvent fraction in the fracture system. This keyword should be used when the initial conditions are specified by equilibration
<code>SSOL</code>	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` or `COALPP` data. Two optional keywords are available to set the initial coal solvent concentration manually:

Keyword	Description
<code>SOLVCONC</code>	Inputs the coal solvent concentration on a block-by-block basis
<code>SCVD</code>	Inputs the coal solvent concentration as a function of depth.

Notes

- 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.
- 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.
- The connate gas saturation is specified by starting the gas saturation table (`SGFN`) at S_{gco} , and ending the water saturation table (`SWFN`) at $1.0 - S_{gco}$.

It is also possible to initialize the model by enumeration and to include a solvent saturation. This is achieved using the `SSOL` keyword in the `SOLUTION` section.

Solvent injection

Solvent can be injected by specifying a gas injection well. A separate keyword `WSOLVENT` can then be used to specify the fraction of the gas flow that is solvent.

Summary data

Refer to [Coal bed methane model](#) in the *ECLIPSE Reference Manual*.

Summary of keywords (ECLIPSE 100)

RUNSPEC

Restriction	Keyword	Description
	CBMOPTS	Options related to the coal bed methane model
	COAL	Specifies that the coal bed methane model is to be used.
Required for dual porosity runs. Do not use DUALPERM	DUALPORO	Specifies a dual porosity run.
Must be used to activate the solvent model if you are using the two gas mode.	SOLVENT	Activates solvent model.

GRID

Keyword	Description
COALNUM	Assign coal regions.
DIFFMMF	Modify the Matrix-Fracture diffusivity.
ROCKFRAC	Specify rock (coal) fractions of the bulk volume on a cell by cell basis.
RPTGRID	Control output from the GRID section: COALV and PORVF output pore volumes of the coal and fluid parts of grid cells.
RPTGRIDL	Control output from the GRID section for local refined grids: COALV and PORVF output pore volumes of the coal and fluid parts of grid cells.
RPTINIT	Control output from the GRID and EDIT sections to INIT file: COALV and PORVF output pore volumes of the coal and fluid parts of grid cells.
SIGMA	Specify the dual porosity SIGMA factor.
SIGMAV	Specify the SIGMA factor on a cell by cell basis.

PROPS

Keyword	Description
COALADS	Specify the gas/solvent relative adsorption data.
COALPP	Specify the gas/solvent partial pressure data.
DIFFCOAL	Define the gas diffusion data.
LANGMPL	Tables of pressure scaling factors for the adsorption capacity for each grid cell.
LANGMUIR	Tables of coal surface gas concentration.
LANGSOLV	Tables of coal surface solvent concentration.
MLANG	Maximum surface gas concentration.
MLANGSLV	Maximum surface solvent concentration.

Keyword	Description
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.
TPAMEPS	Sorption induced strain values for the ROCKPAMA model.
TPAMEPSS	Sorption induced solvent strain values for the ROCKPAMA model.

SOLUTION

Keyword	Description
GASCONC	Initial coal gas concentration.
GCVD	Initial coal gas concentration versus depth.
RPTSOL	Controls the output from the SOLUTION section: <ul style="list-style-type: none"> 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

Refer to [Coal bed methane](#) in the *ECLIPSE Reference Manual*.

Note: 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

Keyword	Description
RPTRST	The restart file will contain the arrays "COALGAS", "COALSOLV", "GASSATC", "MLANG" and "MLANGSLV" whose meanings are documented in the <i>ECLIPSE File Formats Reference Manual</i>

Keyword	Description
RPTSCHE	Control the output from the SCHEDULE section: <ul style="list-style-type: none"> 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 ECLIPSE 100

Basic test ECLIPSE 100

RUNSPEC

Example of a basic ECLIPSE 100 test in the RUNSPEC section:

```
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 /
 3   1   0   0   0   1 /
WELLDIMS
 2   13   1   2 /
NUPCOL
 4 /
START
 26 'JAN' 1983 /
NSTACK
 20 /
FMTOUT
FMTIN
```

GRID

Example of a basic ECLIPSE 100 test in the GRID section:

```
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 /
'PERMY'   500000 /
-- 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

Example of a basic ECLIPSE 100 test in the PROPS section:

```
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

Example of a basic ECLIPSE 100 test in the REGIONS section:

```
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

Example of a basic ECLIPSE 100 test in the SOLUTION section:

```
SOLUTION =====
EQUIL
4000 3959 4000 0 0 0 0 0 0 /
RPTSQL
'PRES' 'SWAT' 'SGAS' 'FIP=2' 'EQUIL' 'GASCONC' /
```

SUMMARY

Example of a basic ECLIPSE 100 test in the SUMMARY section:

```
SUMMARY =====
FPR
FGPR
FWPR
FCGC
RCGC
1 2 3 /
BCGC
2 2 1 /
/
RUNSUM
SEPARATE
```

SCHEDULE

Example of a basic ECLIPSE 100 test in the SCHEDULE section:

```
SCHEDULE =====
RPTSCHE
'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* 200.000 0.000000 1* 0.000000000 /
/
TSTEP
1.0 9.0 90.0 9*100 6*1000
/
END
```

CO₂ injection example ECLIPSE 100

RUNSPEC

ECLIPSE 100 CO₂ injection example in the RUNSPEC section:

```
RUNSPEC =====
TITLE          CO2 Injection Example
DIMENS
 8   8   2 /
DUALPORO
WATER
GAS
SOLVENT
COAL
FIELD
EQLDIMS
 1 100   2   1   20 /
TABDIMS
 1   1   20   20   2   5 /
 2   1   0   0   0   1 /
WELLDIMS
 2  13   1   2 /
NUPCOL
 4 /
START
 26 'JAN' 1983 /
NSTACK
 20 /
```

GRID

ECLIPSE 100 CO₂ injection example in the GRID section:

```
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

ECLIPSE 100 CO₂ injection example in the PROPS section:

```

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

ECLIPSE 100 CO₂ injection example in the SOLUTION section:

```

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

ECLIPSE 100 CO₂ injection example in the SUMMARY section:

```

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

ECLIPSE 100 CO₂ injection example in the SCHEDULE section:

```

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'   /
/
COMPDATA
'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*       50.000   0.000000   1*   0.000000000/
/
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* 50.000 0.000000 1* 0.000000000/
/
WCONINJ
'I' 'GAS' 'SHUT' 'RATE' 100000.0 /
/
NEXTSTEP
0.1 /
TSTEP
1.0 9.0 90.0 900.00 3*1000
/
END

```

Summary of keywords (ECLIPSE 300)

RUNSPEC section

Keyword	Description
COAL	Specifies that the coal bed methane model is to be used.
CBMOPTS	Options for coal bed methane
DUALPORO	Specifies a dual porosity run
NMATRIX	Define the number of matrix cells (multi-porosity option)

GRID section

Keyword	Description
DIFFMMF	Modify the matrix-fracture or matrix-matrix diffusivity.
SIGMA	Input the dual/multi-porosity SIGMA factor.
SIGMAV	Input the SIGMA factor on a cell by cell basis.
COALNUM	Assign coal regions
COALNUMR	Assign coal regions for matrix sub cells

Keyword	Description
CRNDENS	Input rock (coal) density by coal region number
ROCKDEN	Input rock (coal) density.
ROCKFRAC	Input rock (coal) fractions of the bulk volume on a cell by cell basis.
RPTGRID	Control output from the GRID section: COALV and PORVF output pore volumes of the coal and fluid parts of grid cells.
RPTGRIDL	Control output from the GRID section for local refined grids: COALV and PORVF output pore volumes of the coal and fluid parts of grid cells.
RPTINIT	Control output from the GRID and EDIT sections to INIT file: COALV and PORVF output pore volumes of the coal and fluid parts of grid cells.

PROPS section

Keyword	Description
LANGMEXT	Input the Langmuir volume and pressure constant for each component.
LANGMPL	Inputs a pressure scaling factor for the adsorption capacity for each grid cell.
LANGMUIR	Input the Langmuir table for each component.
DIFFCBM	Input gas diffusion data.
RESORB	Input gas readsorption factors.
LANGMULC	Input a scaling factor for the adsorption capacity for each grid cell and each component.
LANGMULT	Input a scaling factor for the adsorption capacity for each grid cell.
ROCKPAMA	Palmer-Mansoori rock compaction model.
ROCKPAME	Modified Palmer-Mansoori model made dependent on composition for coal swelling and shrinkage.
RPTPROPS	Controls output from the PROPS section. DIFFCBM outputs diffusion data. LANGMEXT outputs the extended Langmuir isotherm data. LANGMUIR outputs the Langmuir table data.
TPAMEPS	Component sorption induced strain values for the Palmer-Mansoori model
ROCKPAMA	This is an alternative to ROCKPAME.

SOLUTION section

Keyword	Description
GASADCO	Specify the initial concentration of adsorbed coal gas.
GASADEC	Specify the initial equilibrium concentration of adsorbed coal gas
GASCCMP	Specify the component used by GASCONC and GASSATC.
GASCONC	Specify the initial concentration of adsorbed coal gas for one component
GASSATC	Specify the initial equilibrium concentration of adsorbed coal gas for one component.
SORBFrac	Scale initial adsorbed gas.

Keyword	Description
SORBPRES	Specify Langmuir sorption pressure.
RPTSOL	Controls output from the SOLUTION section. CGAS outputs the adsorbed gas concentration as surface volume / rock volume. The output array names are CGAS1, ..., CGASN (up to the number of components).

SUMMARY section

Refer to [Miscellaneous well and group quantities](#) in the *ECLIPSE Reference Manual*.

If methane is component number one, the methane production rate can be output by specifying:

```
FCWGPR
1 /
```

SCHEDULE section

Keyword	Description
WINJGAS	Specify nature of injection gas.
RPRKPAME	Report rock compaction multipliers (ROCKPAME) versus pressure for a given cell.
RPTSCHED / RPTRST	CGAS outputs the adsorbed gas concentration as surface volume / rock volume. The output array names are CGAS1, ..., CGASN (up to the number of components).

Using the ECLIPSE 300 coal bed methane model

The model is activated by specifying the keyword [COAL](#) in the RUNSPEC section. The dual porosity model should also be activated with the RUNSPEC keyword [DUALPORO](#), and the number of coal regions should be set by item 6 of [REGDIMS](#) (default =1). For multi porosity cases, the keyword [NMATRIX](#) specifies the number of matrix cells per cell (see "[Multi porosity](#)"). It is possible to choose between two types of adsorption model: instant and time dependent. This is done by using the [CBMOPTS](#) keyword. For the instant adsorption model it is also possible to specify that no water flow is allowed in and out of a coal cell by using the [CBMOPTS](#) keyword.

Note: The coal bed methane option is currently not compatible with the Thermal option or the ECLIPSE 300 black oil option.

Grid data

Using the time dependent adsorption model and dual porosity, the first $NDIVIZ/2$ layers of the model represent the coal matrix and the second $NDIVIZ/2$ layers represent the fracture system. The input follows what is described for ECLIPSE 100 above, except:

- It is strongly recommended to use the keyword [ROCKFRAC](#) to set the coal volumes instead of specifying porosities when using the time dependent sorption model.
- The density of the coal must be supplied by the [ROCKDEN](#) keyword unless the [LANGMUIR](#) keyword is used.

- It is possible to set coal regions with the [COALNUM](#) keyword; the default is that all matrix cells are assigned to coal region number one.
- Using the multi-porosity option, the [NMATOPTS](#) keyword can be used to generate the pore volumes and coal volumes for the matrix sub-cells. This also generates [SIGMA](#) factors automatically for the partitioned matrix sub-grid cells based on the initial input sigma factor. Refer to "[Multi porosity](#)" and "[Shale gas](#)" for the definition of the coal and pore volume when doing matrix discretization.

Properties data

Gas and water PVT, EOS and relative permeability data need to be supplied in the usual way. In addition, the following keywords must be supplied:

Restriction	Keyword	Description
Required	LANGMEXT	Defines the Langmuir isotherm for each component and each coal region.
Required	LANGMUIR	Alternative to LANGMEXT , this keyword defines the Langmuir isotherm for each component and each coal region by table.
Required	DIFFCBM	Inputs the diffusion coefficients (not used by the instant sorption model)

If the [GASWAT](#) option is used water and gas relative permeability curves are entered by [WSF](#) and [GSF](#). It is also possible to use the three phase relative permeability curves [SWFN](#), [SGFN](#) and [SOF3](#). However, in the case where there is no oil present, you must still enter an oil relative permeability curve. Optionally the [FACTLI](#) or [ISGAS](#) keywords can also be used in order to make sure that a single-phase hydrocarbon is not labeled as an oil phase during the simulation. Note that components predicted to be in an oil phase do not adsorb on the coal surface.

Note: If an [OIL](#) phase is present then using the option to interpolate the relative permeabilities around the critical point should be considered. This can be enabled using the argument KRMIX in the third item of the keyword [CBMOPTS](#). The default behavior is not to perform the interpolation which is equivalent to using the [NOMIX](#) keyword.

The adsorbed coal gas is always reported as gas, where the density of the adsorbed gas is set to a liquid-like density.

Keyword	Description
RESORB	Inputs the re-adsorption factors (default=1.0)
LANGMULC	Inputs a scaling factor for the adsorption capacity for each grid cell and each component.
LANGMULT	Inputs a scaling factor for the adsorption capacity for each grid cell.
ROCKPAMA	Palmer-Mansoori rock compaction model.
ROCKPAME	Modified Palmer-Mansoori model made dependent on composition for coal swelling and shrinkage.

Equilibration

Time-dependent sorption

The matrix coal gas is always initialized according to the input matrix properties. This means that the input mole fractions and pressure are used to set the gas contents according to the Langmuir adsorption capacity.

An equilibrium is obtained if the initial specified composition of the fractures equals the input matrix composition and the system has free gas. In addition the pressure of the fracture must be level or higher than the input matrix pressure. If the fractures are fully water filled, an equilibrium mole fraction and sorption pressure is calculated internally.

Please note that the resulting adsorbed mole fractions generally are not equal to the input mole fractions.

When equilibration is done by [EQUIL](#) the fractures are equilibrated in the usual way. The initial coal gas molar densities are calculated from the prevailing matrix pressure and composition as specified by [COMPVD](#) or [ZMFVD](#).

A different sorption pressure can be specified by the keyword [SORBPRES](#). Entering pressure values which are lower than the computed matrix pressure will give a lower gas content. It is also possible to model undersaturated coal by scaling the calculated coal gas by a factor (see the keyword [SORBFrac](#)).

It is also possible to input the initial coal gas concentration by using the keywords [GASADCO](#) and/or [GASADEC](#). The first specifies the concentration of the gas for each grid cell and each component. The second will scale the input Langmuir adsorption capacity so that the input concentration is matched at the given pressure and composition. It should be noted that the components need to be present, otherwise it is not possible to match the input concentrations. It is however possible to first input a composition used for scaling and thereafter use [GASADCO](#) to set the final concentration. It is also possible to select one component to use to obtain one scaling value by using the keyword [GASSATC](#). In this case, all of the components use the scaling value obtained for this specific component.

Note that [GASSATC](#), [GASADEC](#), [LANGMULT](#) or [LANGMULC](#) cannot be used together.

An enumerated start is possible in the usual way. The input needed is:

- Mole fraction values for the gas phase ([YMF](#)) and oil phase ([XMF](#)). If no oil phase is present then values of zero (0) should be entered for XMF.
- Pressure.
- Gas and water saturations.

The matrix gas mole fractions and the pressure are then used to initialize the matrix coal gas.

Several equilibrium regions can be used in order to specify different conditions for the matrix and the fractures.

Instant sorption

For the instant sorption model the adsorbed gas content is calculated from the free gas phase composition and pressure for each simulation cell. If water is present the pressure in the pore space and the composition given by [COMPVD](#) or [ZMFVD](#) is used to determine the amount of adsorbed gas. [SORBPRES](#) and [SORBFrac](#) can currently not be used with the instant sorption model.

Output and restart

For the time-dependent model the adsorbed molar densities are output in the usual way as molar densities in the coal volumes.

Using the instant desorption model the adsorbed molar density AMSC is also output, where MLSC are the molar densities in the pore volumes (AMSC is only output when a flexible restart file has been requested)

If component-specific scaling is applied, this is output as arrays MLANG1, ..., MLANGn. (Do not include LANGMULT values. If used, this is output to the INIT file as LANGMULT.)

The adsorbed gas concentration, reported as surface volume over rock volume, can be output by [RPTRST](#) with CGAS. The output array names are CGAS1,..., CGASN (up to the number of components).

Summary keywords

The SUMMARY keywords are:

Keyword	Description
FCWGPR	Production rate for a specific component (given in surface volume by assuming an ideal gas)
FCWGPT	Production total for a specific component (given in surface volume by assuming an ideal gas)
WYMF	Well mole fraction for a specific component

Examples

The following data sets come with the ECLIPSE installation:

File	Description
ECBM2.DATA	Enhanced coal bed methane example
ECBM2I.DATA	Instant sorption and single porosity
ECBM4.DATA	ECBM with coal shrinkage and swelling effects
CBMTRPL.DATA	Coal bed methane example using three porosities
CBM_E300_ROCKPAMA_TPAM.DATA	Coal bed methane example with Palmer-Mansoori rock compaction using TPAMEPS.

Table 12.1: Coal bed methane example files

13

Compatibility between ECLIPSE 100 specialist extensions

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL

ECLIPSE 100 contains 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 specialist extensions 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 their compatibility has been assigned a lower priority for development.

This chapter summarizes the combinations of ECLIPSE 100 specialist extensions 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.

Note: The term "Special Option" is used in the ECLIPSE manual to denote a feature which requires a separate license to run. The extensions described in this chapter historically all required such a license; however, since the 2014.1 release the licensing scheme for ECLIPSE has been simplified and a number of these extensions no longer require a separate license. Please refer to "[Licenses](#)" for full details.

Compatibility information

Coal bed methane model

The coal bed methane model is incompatible with:

- The foam 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. For this reason these combinations are banned. However, the option may still be useful in cases where the other fluid is contained within the [USEFLUX](#) region. If this is the case, the ban may be lifted using [item 232](#) in OPTIONS. 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 gas field operations model.
- Gas lift optimization.
- The Gi pseudo-compositional model.

Foam model

The foam model is incompatible with:

- The coal bed methane model.

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:

- Flux boundary conditions.
- Local grid refinement.
- Multisegment wells.
- The wellbore friction option.

Gradient option

The gradient option is incompatible with:

- Reservoir coupling.

When run with local grid refinement, only in-place refinements are allowed.

Local grid refinement and coarsening

The local grid refinement and coarsening option is incompatible with:

- Environmental tracers (if the tracer is diffusing).
- The Gi pseudo-compositional model.

In addition, the auto refinement option cannot be run with:

- Multisegment wells.
- The wellbore friction option.

Multisegment wells

Multisegment wells are incompatible with:

- The Gi pseudo-compositional model.
- The auto refinement option in local grid refinement.
- The solvent model.
- The wellbore friction option.

Note also that multisegment wells cannot be used with the temperature option in ECLIPSE 100.

Network option

The network option has no restrictions on its use with other options.

Parallel options

Parallel options cannot be used with:

- The gas field operations model.
- The `VISCD` keyword.

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:

- Multisegment 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:

- Relative permeability hysteresis in the oil and water phases.

- The coal bed methane model.

Wellbore friction option

The wellbore friction option is incompatible with:

- The Gi pseudo-compositional model.
- The auto refinement option in local grid refinement.
- Multisegment wells.
- The Solvent model.

Note also that the Wellbore friction option cannot be used with the miscible flood and brine tracking options in ECLIPSE 100.

14

Convergence reports

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 timesteps are selected. The form of this output varies significantly between the two simulators.

ECLIPSE 100 Report

Mnemonics SUMMARY and NEWTON in the [RPTSCHED](#) keyword control output of reports showing the convergence of the non-linear iterations and a brief summary of the state of the reservoir after each timestep (including non-report times). A sample of the sort of output produced is shown below. The report is copied to the debug file if switch 1of [DEBUG](#) is greater than zero.

```

IT= 0  CNV      CELL      MAT BAL    DPRESS    DSWAT     DSGAS
OIL .04562( 6, 10, 1) 1.0-002    .00 .00000   .00000
WAT .00000( 10, 10, 1) 1.6-010    .00 .00000   .00000
GAS .04780( 1, 1, 1) 6.6-003    .00 .00000   .00000
  LIMIT=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, 1) -2.5-006  -233.55   .00006   .00000
GAS 3.80303( 10, 10, 1) -6.8-004 -170.60   .00013   .01982
  LIMIT= 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
  LIMIT= 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
  LIMIT= 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](#)" 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	Description
State 1	there is no liquid hydrocarbon phase. R _v , the oil-gas ratio in the gas phase, is a variable.
State 2	there are both liquid and gaseous hydrocarbon phases. S _{gas} , the saturation of the gaseous phase, is a variable.
State 3	there is no gaseous hydrocarbon phase. R _s , the gas-oil ratio in the liquid phase, is a variable.

Table 14.1: Solution states

Once the non-linear equations have been solved, ECLIPSE prints a brief summary of the state of the reservoir at the end of the timestep. On the first line (beginning STEP) are the timestep number, the current time, the length of the timestep just finished, and an argument which indicates the reason why that timestep length was selected. Possible values of this argument are:

INIT Either the 1st timestep in the run, or a maximum value for the next timestep was set using keyword [TUNING](#) or [NEXTSTEP](#).

TRNC Timestep was set in order to keep estimate of time truncation errors within specified targets.

MINF Timestep was determined by the minimum permitted ratio between one timestep and the next (default 0.3 - see [TUNING](#)). In the PRT file this will usually be followed by the mnemonic (in brackets) that would have been used had this restriction not taken precedence.

MAXF Timestep was determined by the maximum permitted ratio between one timestep and the next (default 3.0 - see [TUNING](#)).

MINS Timestep is the minimum permitted length (default 0.1 DAYS, see [TUNING](#)). In the PRT file this will usually be followed by the mnemonic (in brackets) that would have been used had this restriction not taken precedence.

MAXS Timestep is the maximum permitted length (default 365 DAYS, see [TUNING](#)).

- REPT Timestep was set so as to end at the next report time.
- HALF Timestep is half the interval to the next report time (if timestep would otherwise be larger, but not big enough to reach report time, ECLIPSE reduces the timestep to half the remaining interval).
- CHOP If the non-linear equations do not converge, a new timestep 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).
- THRPT Timestep was determined by the maximum throughput ratio THRUPT (see TUNING).
- SATM Timestep 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 Timestep chosen so that the maximum expected pressure change in any cell is TRGDPR, set in the TUNING keyword.
- PCHP Timestep 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 Timestep was determined by the network balancing error controls set in items 6-8 in keyword NETBALAN. This applies only to the Network Option.
- EMTH Timestep was set so as to end at the end of the month. This applies only to the GOR Penalty option (see keyword WGOPEN) and the Gas Field Operations Model.
- LGRC Timestep 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 Timestep 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 Timestep was set to coincide with a well cycling on/off time, or was restricted when a cycling well turned on (see keyword WCYCLE).

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

$$\text{TIME TRNC ERR} = dS - \frac{dS_p}{dt_p} dt \quad \text{Eq. 14.1}$$

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.

Output of cells with convergence problems

It is possible to output information to the RESTART file providing details of cells that are causing convergence problems. At every non-linear iteration the worst cells have their counter increased by one. ECLIPSE 100 outputs two sets of mnemonics to the RESTART file. The first set provides the worst convergence cells as measured by the magnitude of the residual of every equation, whereas the second set provides the worst cells based on the magnitude of the Newton increments for every variable.

This option is activated by specifying the argument 'CONV=*integer*' in keyword [RPTRST](#). By default, this 'integer' value is set to 1, so that the single worst cell in the run is focused on. Modifying this 'integer' value alters the number of worst cells.

Depending on the phases present in the model and the options used, different arguments are output to the RESTART file. Specifying 'CONV=*integer*' in keyword RPTRST outputs all relevant arguments for the particular model. The possible arguments are:

Argument	Description
CNV_OIL	Residual of oil equation
CNV_WAT	Residual of water equation
CNV_GAS	Residual of gas equation
CNV_PLY	Residual of polymer equation (POLYMER option)
CNV_SAL	Residual of brine equation (BRINE option when used with polymer)
CNV_SOL	Residual of solvent equation (SOLVENT option)
CNV_GGI	Residual of Gi Pseudo-compositional Model (GIMODEL option)
CNV_DPRE	Pressure increment
CNV_DWAT	Water saturation increment
CNV_DGAS	Gas saturation increment / R_v increment / R_s increment
CNV_DPLY	Polymer concentration increment
CNV_DSAL	Brine concentration increment
CNV_DSOL	Solvent concentration increment
CNV_DGGI	Dry injection gas parameter (Gi) increment

Note: The active variables for each grid block depend on the solution state (see table 14.1). CNV_DGAS therefore denotes a composite argument reporting the cells with the largest changes in gas saturation, oil-gas ratio in the gas phase, or gas-oil ratio in the liquid phase.

For models with local grid refinements, the worst converging cells are evaluated separately in the global grid and in each local grid. This is because ECLIPSE 100 by default solves local grids separately using local timestepping. For each grid (global and local) the worst converging cells have their counter increased by 1 after each iteration. The number entered in the CONV argument in RPTRST is weighted by the ratio of the number of active cells in the local grid to the total number of active cells including global host cells to determine how many local grid cells are output.

ECLIPSE 300 Report

The simulator outputs information concerning convergence and timesteps to the log file (and echoes this to the PRINT file). A section of this might take the following form:

	Time	Step	GOR	WCT	OPR	WPR	GPR	FPR	WIR	GIR	It
Init;	10.0	10.0	6.46	.026	959.3	26.57	6200	3530.3	340.0	4700	2
SCT ;	25.0	15.0	6.48	.027	955.7	26.76	6200	3524.1	340.0	4700	1
SCT ;	41.0	16.0	6.51	.027	950.9	26.98	6200	3518.1	340.0	4700	1
SCT ;	59.0	18.0	6.54	.027	946.8	27.23	6200	3511.7	340.0	4700	1
SCT ;	80.0	21.0	6.57	.028	942.5	27.51	6200	3504.4	340.0	4700	1
SCT ;	105.0	25.0	6.61	.028	937.6	27.79	6200	3495.8	340.0	4700	1
SCT ;	136.0	31.0	6.65	.029	932.0	28.19	6200	3485.1	340.0	4700	1
SCT ;	175.0	39.0	6.70	.030	925.2	28.73	6200	3471.9	340.0	4700	1
SCT ;	225.0	50.0	6.76	.031	917.0	29.42	6200	3455.2	340.0	4700	1
Max ;	285.0	60.0	6.83	.032	907.0	30.18	6200	3435.6	340.0	4700	1
HRep;	325.0	40.0	6.92	.032	895.8	30.25	6200	3422.9	340.0	4700	1
Rep ;	365.0	40.0	6.97	.033	888.3	30.40	6200	3410.1	340.0	4700	1

The step codes at the left of each line contain the reason for each step. The full set is:

ADA	: Maximum increase factor for Appleyard timestep selector. The number of Newton iterations will be appended to this step code. If this number is greater than 9 a '+' will be used.
CFL	: Stability limited
EFF+	: Timestep chosen by efficient timestep selection method (ZIPPY2)
EnC	: Energy Change (thermal)
HRep	: Half step to report
Init	: Initial step
Max	: Maximum step
MaxW	: Maximum step after well modification
MDF	: Max decrease factor
MIF	: Max increase factor
Min	: Minimum step
NITE	: Number of Iterations
NVLP	: Appleyard timestep selector based on envelope function
PCT	: Pressure change target
Redo	: Redoing the timestep due to a chop
Reduc	: Reduced step
Rep	: Step to report
SAC	: Saturation Change (thermal)
SCT	: Solution change target
SST	: Solid solution change target (CO2STORE option)
Spec	: Specified next step
TEC	: Temperature Change (thermal)
TPT	: Throughput target
TTE	: Time Truncation Error
WAG	: Step to WAG change

Solution change target (SCT)

Since our primary solution variables are molar densities, it is natural to use these to control the convergence. However, large saturation changes may occur for very small changes in composition, due to the problems of near critical states mentioned in "Saturation functions", so the use of an effective saturation is preferred. Consider the expression:

$$\Delta S' = \frac{dU^T}{dm_c} \cdot \Delta m_c \quad \text{Eq. 14.2}$$

where

$$U^T = \frac{m_t L}{b_o^m} + \frac{m_t V}{b_g^m} + \frac{m_w}{b_w^m} \quad \text{Eq. 14.3}$$

where

U^T is the fluid volume per unit pore volume

$\Delta S'$ is thus the change in fluid volume per unit pore volume for a given molar density change, that is the effective saturation change for the molar density change.

In all cases the maximum solution change over a 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 timestep allowed. All these elements of the timestepping 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 timestepping 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,

$$\Delta X_a = X^{T+\Delta T} - X^T \quad \text{Eq. 14.4}$$

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

$$\Delta X_e = \frac{dX}{dT} \cdot \Delta T \quad \text{Eq. 14.5}$$

where $\frac{dX}{dT}$ is the change per unit time for the previous step.

The time truncation error (TTE) is then estimated from the value of

$$\left| \Delta X_a - \Delta X_e \right| \cdot \frac{dU^T}{dX} \quad \text{Eq. 14.6}$$

where $\frac{dU^T}{dX}$ is used to normalize into dimensionless units of order 1.

Note that if the solution is changing at a constant rate, ΔX_e equals ΔX_a , and there is no TTE limitation.

Minimum step codes

Minimum step codes may start with the two letters Mn followed by an abbreviated version of one of the above - for example MnTP - where the second part indicates what constraint would have applied if the minimum step had not been imposed.

MnNC	:	Minimum timestep, No Convergence
MnSC	:	Minimum timestep due to Solution change
MnTP	:	Minimum timestep due to Throughput change
MnPC	:	Minimum timestep due to Pressure change
MnTT	:	Minimum timestep due to Time Truncation error
MnTE	:	Minimum timestep due to Temperature change (thermal)
MnEn	:	Minimum timestep due to Energy change (thermal)
MnSA	:	Minimum timestep 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 timestep 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 timestep 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 timestep allows the simulator to accurately calculate events that are occurring on a smaller time scale than the original timestep 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, counters are increased by one to indicate the cells that have the largest volume balance residual equations and pressure solution updates. This diagnostic feature can be activated via the 'CONV=integer' mnemonic of the [RPTRST](#) keyword. By default, this 'integer' value is set to 10, which implies that the worst 10 cells in the run are focused on. Modifying this 'integer' value alters the number of cells being monitored. These counter values are output with the arguments CONV_VBR and CONV_PRU respectively.

ECLIPSE 300 also outputs the argument CONV_NEW which records the number of non-linear iterations required by each cell in order to satisfy the solution change convergence criteria at the last timestep.

These 3D convergence reports can prove useful by enabling problem cells in a run to be pin-pointed. The values are best interpreted relative to one another rather than as absolute numbers.

Example

If 'CONV=40' is used with the [RPTRST](#) keyword, then the three records CONV_VBR, CONV_PRU and CONV_NEW are output to the RESTART files during the simulation. If a given cell reports a value of 1000 for the CONV_VBR data then this implies that the cell has been in the top 40 cells with the largest volume balance residual for 1000 non-linear iterations in the simulation so far. Similarly for the CONV_PRU data and the pressure solution update. If a cell reports a value of 3 for the CONV_NEW data then this implies that at the previous timestep that particular cell satisfied the solution change convergence criteria on the 3rd non-linear iteration.

15

Diffusion

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.

Diffusion coefficients: ECLIPSE 100

ECLIPSE 100 allows diffusion of both gas and oil components within both the oil and gas phases. For example, in a gas condensate model with no oil phase present, gas diffuses from a low R_v region to a high R_v region, and vaporized oil flows in the opposite direction from the high R_v to the low R_v region (R_v is the vaporized oil-gas ratio). Consider a gas component in the gas phase. The moles of gas and oil within the gas phase are given by:

$$M_{gas} = \frac{S_g \cdot \rho_g}{B_g \cdot MW_g} \quad \text{Eq. 15.1}$$

$$M_{oil} = \frac{R_v \cdot S_g \cdot \rho_o}{B_g \cdot MW_o} \quad \text{Eq. 15.2}$$

hence the gas total molar concentration c is

$$c_g = \frac{S_g}{B_g} \left(\frac{\rho_g}{MW_g} + \frac{R_v \cdot \rho_o}{MW_o} \right) = \frac{S_g \cdot \rho_g}{B_g \cdot MW_g} (1 + R_v F) \quad \text{Eq. 15.3}$$

The gas mole fraction is

$$x_g = \frac{\rho_g / MW_g}{\rho_g / MW_g + R_v \rho_o / MW_o} = \frac{1}{1 + R_v F} \quad \text{Eq. 15.4}$$

with

$$F = \frac{\rho_o \cdot MW_g}{\rho_g \cdot MW_o} \quad \text{Eq. 15.5}$$

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

The diffusive flow of gas at surface conditions F_{gg} is given by:

$$F_{gg} = \frac{MW_g \cdot J_g \cdot A \cdot \varphi}{\rho_g} \quad \text{Eq. 15.6}$$

For more information see [Ref. 90]. The gas-in-gas diffusion coefficient D_{gg} is given by:

$$J_g = -c_g \cdot D_{gg} \cdot \frac{dx_g}{dl} \quad \text{Eq. 15.7}$$

with

$$dx_g = -\frac{F}{(1 + R_v F)^2} dR_v$$

Substituting gives the diffusive flow of gas:

$$F_{gg} = \frac{S_g}{B_g} \cdot \frac{F}{1 + R_v F} \cdot D_{gg} \cdot \frac{dR_v}{dl} \cdot A \cdot \varphi \quad \text{Eq. 15.8}$$

where

J_g is the molar flux of gas per unit area

$\frac{dx_g}{dl}$ is the mole fraction gradient of gas

A is the area of flow

φ is the porosity

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

$$F_{gg} = \frac{S_g}{B_g} \cdot \frac{F}{(1 + R_v F)} \times D_{gg} \times \frac{(R_{vj} - R_{vi})}{d} \times A \times \varphi$$

or

$$F_{gg} = -\frac{S_g}{B_g} \cdot \frac{F}{1 + R_v F} \cdot D_{gg} \cdot (R_{vi} - R_{vj}) \cdot T_d \quad \text{Eq. 15.9}$$

where

T_d is the diffusivity = $A\varphi / d$

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 $\text{MIN}(S_{gi}, S_{gj})$

B_g is taken as $(B_{gi} + B_{gj}) / 2$

R_v is taken as $(R_{vi} + R_{vj}) / 2$

For the oil-in-gas diffusive flow relationship F_{og} :

$$F_{og} = \frac{\text{MW}_o \cdot J_o \cdot A \cdot \varphi}{\rho_o}$$

and the oil-in-gas diffusion coefficient D_{og} is given by:

$$J_o = -c_g \cdot D_{og} \cdot \frac{dx_o}{dl}$$

$$c_g = \frac{S_g}{B_g} \cdot \frac{\rho_g}{\text{MW}_g} (1 + R_v F)$$

$$x_o = \frac{R_v F}{1 + R_v F}$$

$$dx_o = \frac{F}{(1 + R_v F)^2} dR_v$$

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

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

where

J_o is the molar flux of oil per unit area

$\frac{dx_o}{dl}$ is the mole fraction gradient of oil

For the oil-in-oil diffusive flow relationship F_{oo} :

$$F_{oo} = \frac{\text{MW}_o \cdot J_o \cdot A \cdot \varphi}{\rho_o}$$

and the oil-in-oil diffusion coefficient D_{oo} is given by:

$$J_o = -c \cdot D_{oo} \cdot \frac{dx_o}{dl}$$

$$c = \frac{S_o}{B_o} \cdot \frac{\rho_o}{\text{MW}_o} \left(\frac{R_s + F}{F} \right)$$

$$x_o = \frac{F}{F + R_s}$$

$$dx_o = - \frac{F}{(F + R_s)^2} dR_s$$

$$F_{oo} = \frac{S_o}{B_o} \cdot \frac{1}{(F + R_s)} \times D_{oo} \times \frac{(R_{sj} - R_{si})}{d} \times A \times \varphi$$

$$F_{oo} = - \frac{S_o}{B_o} \times \frac{1}{(F + R_s)} \times D_{oo} \times (R_{si} - R_{sj}) \times T_d$$

For the gas-in-oil diffusive flow relationship F_{go} :

$$F_{go} = \frac{\text{MW}_g \cdot J_g \cdot A \cdot \varphi}{\rho_g}$$

and the gas-in-oil diffusion coefficient D_{go} is given by:

$$J_g = -c_o \cdot D_{go} \cdot \frac{dx_g}{dl}$$

$$c_o = \frac{S_o}{B_o} \cdot \frac{\rho_o}{\text{MW}_o} \left(\frac{F + R_s}{F} \right)$$

$$x_g = \frac{R_s}{F + R_s}$$

$$dx_g = \frac{F}{(F + R_s)^2} dR_s$$

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

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

["Diffusivity calculations"](#) describes the calculation of the diffusivities in more detail.

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. 90]:

$$J_i = -cD_i \frac{\partial x_i}{\partial d} \quad \text{Eq. 15.10}$$

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

$$J_i = -cD_i^a x_i \frac{1}{RT} \frac{\partial}{\partial d} [\mu_i - M_i g (h - h_0) + M_i D_i^T \ln(T)] \quad \text{Eq. 15.11}$$

where:

J_i is the molar flux of component i per unit area per unit time,

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 chemical potential of component i , given by as constant,

μ_i

$$\mu_i = \mu_{i0} + RT \ln(f_i) \quad \text{Eq. 15.12}$$

where:

μ_{i0} is the reference chemical potential,

f_i is the component fugacity.

Therefore, there are two possible ways of specifying diffusion coefficients:

- Use normal diffusion coefficients D_i defined by keywords **DIFFCOIL** (**DIFFCWAT** for GASWAT or CO2STORE option) and **DIFFCGAS**.
- Use activity corrected diffusion coefficients D_i^a defined by keywords **DIFFAOIL** and **DIFFAGAS**. Thermal diffusion coefficients D_i^T can optionally be defined by keywords **DIFFTOIL** and **DIFFTGAS**.

Relationship between the two diffusion models

Equation 15.10 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 equation 15.11. 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, equation 15.11 can be written as:

$$J_i = -cD_i^a x_i \frac{\partial}{\partial d} \ln(f_i) \quad \text{Eq. 15.13}$$

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

$$\frac{\partial}{\partial d} \ln(f_i) = \frac{\partial}{\partial \ln(x_i)} \ln(f_i) \cdot \frac{\partial}{\partial x_i} \ln(x_i) \cdot \frac{\partial x_i}{\partial d} = \frac{\partial}{\partial \ln(x_i)} \ln(f_i) \cdot \frac{1}{x_i} \cdot \frac{\partial x_i}{\partial d} \quad \text{Eq. 15.14}$$

and we have ignored terms $\partial \ln(f_i) / \partial x_j$. equation 15.13 can then be written as:

$$J_i = -\left(c \cdot D_i^a \cdot \frac{\partial}{\partial \ln(x_i)} \ln(f_i) \cdot \frac{\partial x_i}{\partial d} \right) \quad \text{Eq. 15.15}$$

Comparing equations 15.10 and 15.15, D_i^a is given by:

$$D_i^a = \frac{D_i}{\frac{\partial \ln(f_i)}{\partial \ln(x_i)}} \quad \text{Eq. 15.16}$$

D_i^a is therefore the activity-corrected diffusion coefficient of [Ref. 90].

Equivalence at low pressures

At low pressures, the two coefficients are equal, as $f_i \rightarrow x_i P$, and the predicted flows become the same, as $\partial \ln(f_i) / \partial d \rightarrow \partial \ln(x_i) / \partial d \rightarrow (1/x_i) \cdot (\partial x_i / \partial d)$.

Component flow

In a porous medium, with both oil and gas present, the molar concentration includes the saturation, S and porosity, φ , so that

$$J_i = J_{io} + J_{ig} \quad \text{Eq. 15.17}$$

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

$$J_{io} = -\left(\varphi \cdot S_o \cdot b_o^m \cdot D_{io}^a \cdot x_i \cdot \frac{1}{RT} \cdot \frac{\partial}{\partial d} (RT \ln f_i - M_i Gh + M_i D_{io}^T \ln T)\right) \quad \text{Eq. 15.18}$$

where b_o^m is the molar density of oil, or expressed in terms of normal diffusion coefficients:

$$J_{io} = -\left(\varphi \cdot S_o \cdot b_o^m \cdot D_{io} \cdot \frac{\partial x_i}{\partial d}\right) \quad \text{Eq. 15.19}$$

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:

$$F_i^{diff} = F_{io}^{diff} + F_{ig}^{diff} \quad \text{Eq. 15.20}$$

where:

$$F_{io}^{diff} = T_D D_{io}^a \left(\frac{x_i S_o b_o^m}{RT} \right)_u \Delta (RT \ln f_i - M_i Gh + M_i D_{io}^T \ln T) \quad \text{Eq. 15.21}$$

$$F_{ig}^{diff} = T_D D_{ig}^a \left(\frac{y_i S_g b_g^m}{RT} \right)_u \Delta (RT \ln f_i - M_i Gh + M_i D_{ig}^T \ln T) \quad \text{Eq. 15.22}$$

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

In terms of normal diffusion coefficients:

$$F_{io}^{diff} = T_D D_{io} (S_o b_o^m)_u \Delta x_i \quad \text{Eq. 15.23}$$

$$F_{ig}^{diff} = T_D D_{ig} (S_g b_g^m)_u \Delta y_i \quad \text{Eq. 15.24}$$

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:

$$T_D = \frac{\varphi A}{d} \quad \text{Eq. 15.25}$$

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", 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 15.11, the diffusive flow will be zero when

$$\frac{\partial}{\partial d} [\mu_i - M_i gh + M_i D_i^T \ln(T)] = 0 \quad \text{Eq. 15.26}$$

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 equation 15.26 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 300)

As well as diffusion of oil and gas components within the oil and gas phases, diffusion of components between the phases may be important.

In ECLIPSE 300 two models are available:

- The activity driven diffusion. The diffusion between cells are driven by the chemical potential, thus resulting in diffusion over phase boundaries. The diffusion coefficients for the cross phase diffusion are set by combining the diffusion coefficients given by **DIFFAOIL** and **DIFFAGAS** so $D_{OG}^a = \sqrt{D_o^a \times D_g^a}$. The activity driven diffusion option is not compatible with the **CO2STORE** option
- The molecular driven diffusion. The cross phase diffusion coefficients are set by the keywords **DIFFCOG** and **DIFFCGO**. The fluids are mixed on the border of each cell and the resulting composition is used to set up the potential for molecular diffusion of each component. This model can be used by EOS runs and the **CO2STORE** option. For **GASWAT** or the **CO2STORE** option the water-gas and gas-water diffusion coefficients are set by the keywords **DIFFCWG** and **DIFFCGW**. The molecular diffusion option will construct a border composition by mixing the composition of neighbor cells. This means that the option generally will require more computational time than the activity driven diffusion option.

Note: The activity driven diffusion option can be used with the GASWAT option.

The molecular cross-phase diffusion option

As an example we consider a cell I containing oil and a cell J containing gas. An artificial border composition is constructed by $\tilde{m}_i = (bo \cdot x_i)_I + (bg \cdot y_i)_J$

where

bo is the oil molar density

x_i is the oil mole fraction of component i,

bg is the gas molar density

y_i is the gas mole fraction of component i

The border pressure and temperature is set to $\tilde{P} = \frac{P_I + P_J}{2}$, $\tilde{T} = \frac{T_I + T_J}{2}$

The corresponding oil and gas mole fractions and densities of the artificial border cell is denoted \tilde{x}_i , \tilde{y}_i , \tilde{b}_o and \tilde{b}_g . The cross phase diffusive flow between the oil in cell I and the gas in cell J for component i is given by $F_{cross} = FX_{og} + FX_{go} + FY_{og} + FY_{go}$

$$FX_{og} = Diff \times Dog_i \times \min(So_I, Sg_J) \times bo \times \max(\tilde{x}_i - \tilde{x}_i, 0)$$

$$FX_{go} = Diff \times Dgo_i \times \min(So_I, Sg_J) \times \tilde{b}_o \times \min(\tilde{x}_i - \tilde{x}_i, 0)$$

$$FY_{og} = Diff \times Dog_i \times \min(So_I, Sg_J) \times \tilde{b}_g \times \max(\tilde{y}_i - y_i, 0)$$

$$FY_{go} = Diff \times Dgo_i \times \min(So_I, Sg_J) \times bg \times \min(\tilde{y}_i - y_i, 0) \text{ where}$$

So_I and Sg_J is the oil and gas saturation in cell I and J,

$Diff$ is the diffusivity

Dog_i , Dgo_i are the cross phase oil-gas and gas-oil diffusion coefficients.

It is possible to activate the molecular cross phase diffusion option without also specifying phase specific diffusion coefficients.

The cross phase diffusion option (ECLIPSE 100)

As well as diffusion of oil and gas components within the oil and gas phases, diffusion of components directly from the gas phase to the oil phase may be important.

For example, if in a dual porosity system the fracture contains gas and the matrix mainly oil, then gas is expected to diffuse from the fracture to the matrix and oil from the matrix to the fracture. This process can be modeled using the cross phase diffusion option.

The cross phase diffusion process is assumed to be limited by the slower diffusion within the oil. Hence the diffusive flow takes the form of the gas-in-oil and oil-in-gas diffusion:

$$F_{gx} = \frac{S_c}{B_o} \cdot \frac{F}{(F + R_{sj})} \cdot D_{gx} \cdot (R'_{si} - R_{sj}) \cdot T_d \quad \text{Eq. 15.27}$$

$$F_{ox} = \frac{S_c}{B_o} \cdot \frac{1}{(F + R_{sj})} \cdot D_{ox} \cdot (R'_{si} - R_{sj}) \cdot T_d$$

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 $\text{MIN}(S_{oi} + S_{gi}, S_{oj} + S_{gj}) - \text{MIN}(S_{oi}, S_{oj}) - \text{MIN}(S_{gi}, S_{gj})$

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

$$T_{D_{xI}} = \frac{M_{D_{xI}} A F_{dip}}{B} \quad \text{Eq. 15.28}$$

where:

$T_{D_{x_I}}$ 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:

$$A = \frac{(D_{x_J} D_{y_I} D_{z_I} R_{ntg_I} + D_{x_I} D_{y_J} D_{z_J} R_{ntg_J})}{(D_{x_I} + D_{x_J})} \quad \text{Eq. 15.29}$$

$$B = \frac{\left(\frac{D_{x_I}}{\varphi_I} + \frac{D_{x_J}}{\varphi_J}\right)}{2} \quad \text{Eq. 15.30}$$

$$F_{\text{dip}} = \frac{D_h^2}{D_h^2 + D_v^2} \quad \text{Eq. 15.31}$$

with:

$$D_h^2 = \left(\frac{D_{x_I} + D_{x_J}}{2}\right)^2 \quad \text{Eq. 15.32}$$

$$D_v^2 = (d_I - d_J)^2 \quad \text{Eq. 15.33}$$

where

d denotes depth

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:

$$T_{D_{x_I}} = \frac{M_{D_{x_I}}}{\frac{1}{T_I} + \frac{1}{T_J}} \quad \text{Eq. 15.34}$$

with:

$$T_I = \frac{\varphi_I R_{nt} g_I \mathbf{A} \cdot \mathbf{D}_I}{\mathbf{D}_I \cdot \mathbf{D}_I} \quad \text{Eq. 15.35}$$

where

$$\mathbf{A} \cdot \mathbf{D}_I = A_x D_{x_I} + A_y D_{y_I} + A_z D_{z_I} \quad \text{Eq. 15.36}$$

and

$$\mathbf{D}_I \cdot \mathbf{D}_I = D_{x_I}^2 + D_{y_I}^2 + D_{z_I}^2 \quad \text{Eq. 15.37}$$

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 similarly D_{x_I} , D_{y_I} and D_{z_I} 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 averages. 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:

$$T_{D_{r_I}} = \frac{M_{D_{r_I}} F_{\text{dip}}}{\frac{1}{T_I} + \frac{1}{T_J}} \quad \text{Eq. 15.38}$$

with:

$$T_I = \frac{\varphi_I R_{ntg_I} D_{\theta_I} D_{z_I}}{D_{1p}} \quad \text{Eq. 15.39}$$

$$D_{1p} = \frac{R_1^2}{(R_2^2 - R_1^2)} \ln(R_1/R_2) + \frac{1}{2} \quad \text{Eq. 15.40}$$

and

$$T_J = \frac{\varphi_J R_{ntg_J} D_{\theta_J} D_{z_J}}{D_{2m}} \quad \text{Eq. 15.41}$$

$$D_{2m} = \frac{R_3^2}{(R_3^2 - R_2^2)} \ln\left(\frac{R_3}{R_2}\right) - \frac{1}{2} \quad \text{Eq. 15.42}$$

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

$$T_I = \frac{2\varphi_I R_{ntg_I} D_{z_I} \ln\left(\frac{R_2}{R_1}\right)}{D_{\theta_I}}$$

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

$$T_I = \frac{\varphi_I D_{\theta_I} (R_2^2 - R_1^2)}{D_{z_I}}$$

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:

$$T_D = \text{DIFFMMF} \cdot \varphi_m \cdot V \cdot \sigma \quad \text{Eq. 15.43}$$

where

φ_m is the matrix cell porosity

V is the matrix cell bulk volume

σ is a factor to account for the matrix-fracture interface area per unit bulk volume.

Kazemi [Ref. 61] has proposed the following form for σ :

$$\sigma = 4 \left(\frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2} \right) \quad \text{Eq. 15.44}$$

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 $\text{NDIVIZ}/2$ 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:

- 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 overwritten 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

The diffusion coefficients and average phase molecular weights have to be supplied with the [DIFFC](#) keyword in the PROPS section. Separate gas-in-oil and oil-in-oil diffusion coefficients need to be specified for the cross phase diffusion model. If these are not supplied, no cross phase diffuse will occur.

ECLIPSE 300

Diffusion coefficients can be entered using [DIFFCOIL](#), [DIFFCGAS](#), [DIFFCWAT](#), [DIFFCOG](#), [DIFFCGO](#), [DIFFCWG](#), [DIFFCGW](#) or [DIFFAOIL](#) and [DIFFAGAS](#).

ECLIPSE 100

In dual porosity runs, the fracture-to-fracture diffusion calculation can be turned off using the [DIFFDP](#) keyword.

Summary of keywords

RUNSPEC

- [DIFFUSE](#) Enables molecular diffusion

GRID

Restriction	Keyword	Description
	DIFFMMF	Matrix-fracture diffusivity multipliers
	DIFFMR	Diffusivity multipliers in the radial direction
E100 only	DIFFMR-	Diffusivity multipliers in the negative radial direction.
	DIFFMTHT	Diffusivity multipliers in the theta direction
E100 only	DIFFMTH-	Diffusivity multipliers in the negative theta direction.
	DIFFMX	Diffusivity multipliers in the X-direction
E100 only	DIFFMX-	Diffusivity multipliers in the negative X-direction.
	DIFFMY	Diffusivity multipliers in the Y-direction
E100 only	DIFFMY-	Diffusivity multipliers in the negative Y-direction.
	DIFFMZ	Diffusivity multipliers in the Z-direction
E100 only	DIFFMZ-	Diffusivity multipliers in the negative Z-direction.
E100 only	MULTREGD	Multiplies the diffusivity between flux or MULTNUM regions.

EDIT

Restriction	Keyword	Description
	DIFFR	Diffusivities in the radial direction
	DIFFTHT	Diffusivities in the theta direction
	DIFFX	Diffusivities in the X-direction
	DIFFY	Diffusivities in the Y-direction
	DIFFZ	Diffusivities in the Z-direction
E100 only	MULTREGD	Multiplies the diffusivity between flux or MULTNUM regions.

PROPS

Restriction	Keyword	Description
E300 only	DIFFAGAS	Activity corrected gas diffusion coefficients.
E300 only	DIFFAOIL	Activity corrected oil diffusion coefficients
E100 only	DIFFC	Molecular diffusion data for each PVT region.
E300 only	DIFFCGAS	Gas diffusion coefficients.
E300 only	DIFFCOIL	Oil diffusion coefficients.
E300 only	DIFFCWAT	Water diffusion coefficients (GASWAT or CO2STORE option).
E300 only	DIFFCWG	Water-Gas cross phase diffusion coefficients (GASWAT or CO2STORE option).
E300 only	DIFFCGW	Water-Gas cross phase diffusion coefficients (GASWAT or CO2STORE option).
E300 only	DIFFCOG	Oil-Gas cross phase diffusion coefficients.
E300 only	DIFFCGO	Gas-Oil cross phase diffusion coefficients.
E100 only	DIFFDP	Restricts molecular diffusion in dual porosity runs.
E300 only	DIFFTGAS	Thermal diffusion coefficients for gas.
E300 only	DIFFTOIL	Thermal diffusion coefficients for oil.

SCHEDULE

- [DIFFMMF](#) Matrix-fracture diffusivity multipliers

16

Dual porosity

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

In a naturally fractured (for example carbonate) reservoir, fluids exist in two interconnected systems:

- The rock matrix, which usually provides the bulk of the reservoir volume
- The highly permeable rock fractures.

To model such systems in ECLIPSE, 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. This procedure is referred to as “dual porosity” modeling.

If the matrix blocks are linked only through the fracture system, this is considered to be 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. However, 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.

In a dual porosity or dual permeability run of ECLIPSE, the number of layers in the Z-direction should be doubled. ECLIPSE associates the first half of the grid (the first $\text{NDIVIZ} / 2$ 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.

For local grid refinements, the $\text{NDIVIZ} / 2$ rule is applied to the number of refined cells (item 10 in [CARFIN](#)) in the LGR. The position of the LGR in the host grid (CARFIN items 6 and 7) is specified by its location in the matrix; it will automatically be duplicated in the fracture.

ECLIPSE 300

Note that for ECLIPSE 300, if the multiporosity option is used (see “[Multi porosity](#)” and keyword [NMATRIX](#)) then the first NDIVIZ/N layers will be the matrix blocks, where N is the number of porosities. References to $\text{NDIVIZ} / 2$ in this chapter should be checked against the multi porosity documentation if there is more than one matrix porosity.

ECLIPSE 100

However, for the “[Discretized matrix model \(ECLIPSE 100\)](#)” (“Russian Doll model”) the additional blocks are all internal to the ECLIPSE calculation, and the matrix in the grid is always $\text{NDIVIZ}/2$ layers regardless of the number of matrix subdivisions. This is because the properties of the matrix sub-cells cannot be specified independently, but are all copies of the same matrix cell data.

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:

$$TR = CDARCY \cdot K \cdot V \cdot \sigma \quad \text{Eq. 16.1}$$

where by default:

K is taken as the X-direction permeability of the matrix blocks (unless [PERMMF](#) is also specified; see below),

V is the matrix cell bulk volume (note that this is not the pore volume, having no porosity factor), and

σ is a factor of dimensionality $LENGTH^2$, to account for the matrix/fracture interface area per unit volume, that is the size of the blocks in the matrix volume.

The default use of the X-direction permeability can be overridden by using the matrix-fracture permeability specified by the [PERMMF](#) keyword. Note that for one-dimensional (Y- or Z-direction) cases in the global grid with X-direction extension in an LGR, the X-direction permeability will be set to zero in all grids and [PERMMF](#) must be used to enable dual porosity NNCs to be generated. (This is unlikely in real simulations, but many test problems are affected by this.)

Kazemi [[Ref. 61](#)] has proposed the following form for σ :

$$\sigma = 4 \left(\frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2} \right) \quad \text{Eq. 16.2}$$

where l_x , l_y and l_z are typical X, Y and Z dimensions of the blocks of material making up the matrix volume. (l_x , l_y and l_z are thus not related to the simulation grid dimensions). Alternatively, as σ acts as a multiplier on the matrix-fracture coupling, it may simply be treated as a history matching parameter.

σ can be specified as a single value for the whole field using the keyword [SIGMA](#) 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 specified values of the permeabilities of cells in the fracture system are multiplied by the fracture porosities to yield effective fracture permeabilities. If the [NODPPM](#) keyword is used then the specified permeability is used directly, 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:

$$TR = CDARCY \cdot V \cdot \left(\frac{f_x \cdot K_x}{l_x^2} + \frac{f_y \cdot K_y}{l_y^2} + \frac{f_z \cdot K_z}{l_z^2} \right) \quad \text{Eq. 16.3}$$

where:

$f_x f_y f_z$ are input in LTOSIGMA

$K_x K_y K_z$ are the matrix permeability values

$l_x l_y l_z$ are the matrix block dimensions input using LX, LY and LZ.

If the dual porosity but not the dual permeability option is selected, the matrix blocks have no transmissibilities between them. If dual porosity and dual permeability is chosen, the matrix blocks have their normal transmissibilities.

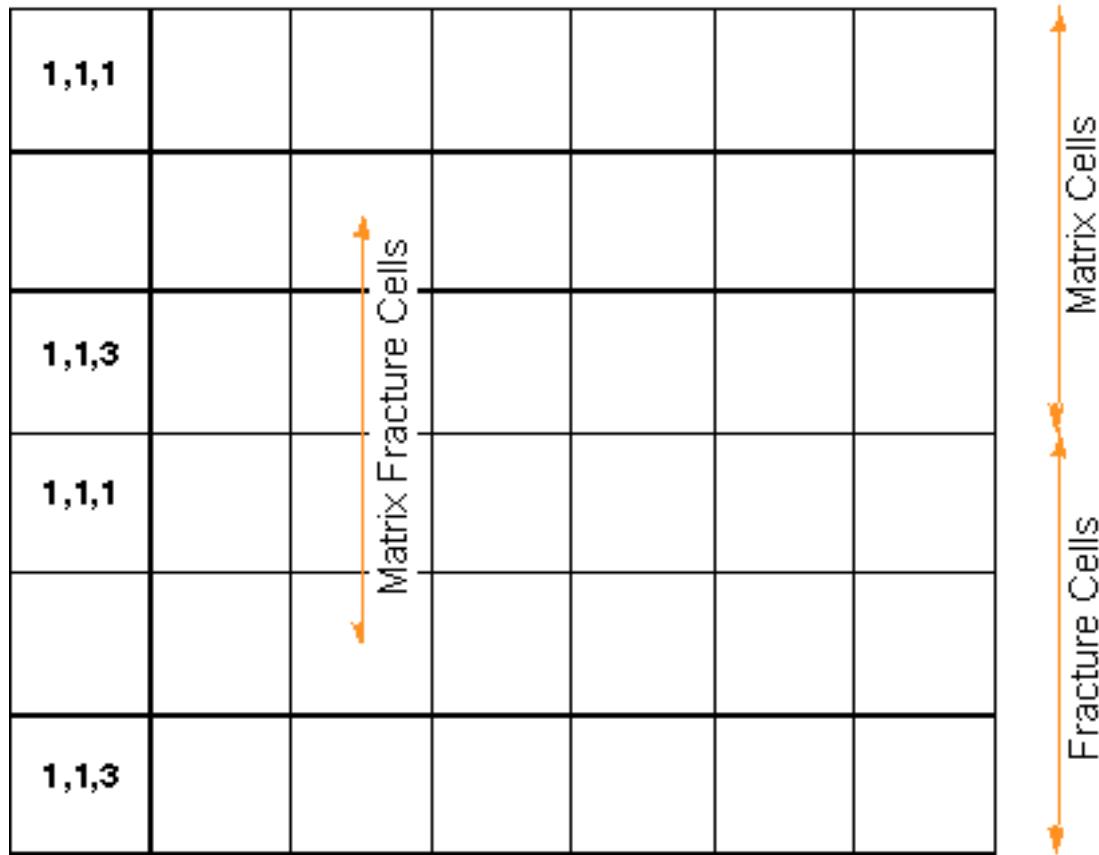


Figure 16.1. A simple, dual porosity, dual permeability system.

Block to block transmissibility calculations

It is now also possible to represent re-imbibition from a fracture cell by the matrix cell below it. In this case the transmissibility between the matrix and fracture cells is a conventional spatial one, since the two cells are not superimposed on one another, and the properties of the two cells contribute to the transmissibility in exactly the same way as for the transmissibility between two neighboring fracture cells, or two neighboring matrix cells. See "[Transmissibility calculations](#)" for the details of this calculation. This modeling is most appropriate when the physical matrix block size (l_z as used in the Kazemi formula for the usual matrix-fracture transmissibility multiplier represented by the `SIGMA` and `SIGMAV` keywords) is comparable to the size of the grid cell rather than significantly different from it, so that the environment of a lower matrix physical block also comprises neighboring fracture cells rather than just its own fracture cell.

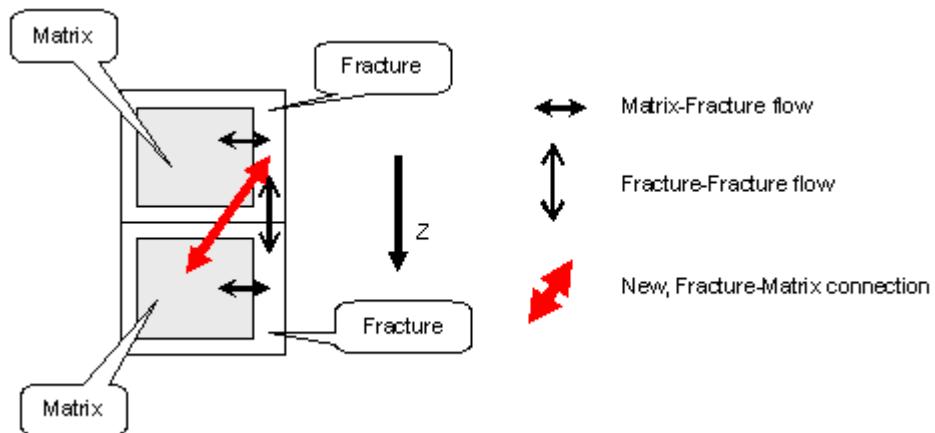


Figure 16.2. Physical view of the matrix and fracture cells for the block to block connection

Figure 16.2 shows this new transmissibility on a picture of the two physical cells (that is the matrix and fracture cells occupy the same physical location.) This figure appears to show the lower matrix disconnected from the upper fracture by its own surrounding fracture, but this is only for presentation purposes: there will always be some contact between the upper fracture and lower matrix. This is schematically represented in figure 16.3, where the lower matrix and fracture are shown as single adjacent blocks, but the upper matrix is split in two, one half abutting the lower matrix (a disconnection in dual porosity but a connection in dual permeability) and one half abutting the lower fracture (not represented in the model, but conceptually possible). The upper fracture contacts both lower fracture and lower matrix.

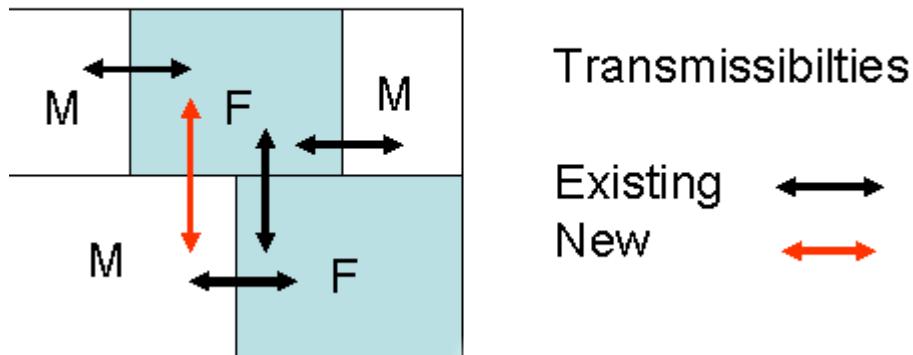


Figure 16.3. Schematic view of the contacts between the upper and lower cells of the two porosity systems

The extent of the contact between the upper fracture and lower matrix is a variable of the model, and hence a new multiplier is applied to the transmissibility between these cells, to represent this effective contact area between the matrix and the fracture, which will be different from the contact area between the two vertically neighboring fracture cells, or two vertically neighboring matrix cells. This contact area multiplier is specified by the keywords **BTOBALFA** or **BTOBALFV**; the former specifies a single value for the whole grid, while the latter specifies it cell by cell. The presence of these keywords also activates these additional transmissibilities, which will otherwise not be present.

Note: Block to block connections cannot be made when a multi porosity model is specified ([NMATRIX](#) or [TRPLPORS](#)).

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 is full of gas.

Gravity imbibition/drainage

Fluid exchange between the fracture and matrix due to gravity is modeled in one of two ways. One method is to consider each of the matrix and fracture cells as separately in vertical equilibrium, and then to calculate additional potential due to differences in contact heights between the matrix and fracture. There are two options for this, which are activated by including either of the keywords [GRAVDR](#) or [GRAVDRM](#) (described in the next two sections) in the RUNSPEC section. This method leaves the number of porosities in the grid as 2.

The other method (described in the section following) uses a modification of the Discrete Matrix model (ECLIPSE 100) or multi porosity model (ECLIPSE 300), and uses N matrix porosities, where N is user-

defined, to create a vertical stack of finely spaced matrix cells which describe the distribution of properties within a single block of matrix material (with height as defined by the `DZMTRX` keyword). This option is not available with dual permeability runs, which should use a more detailed discretization of the whole grid; the advantage of discretizing only the matrix is that its equations can be eliminated from the solution before the linear solution is performed, so that no additional time is spent in that part of the code.

The `DZMTRX` keyword is also required for the `GRAVDR` or `GRAVDRM` models, and a non-zero value should be given for it in the `GRID` section in all gravity drainage cases.

Standard gravity drainage model (`GRAVDR`)

The illustration shows a typical block of matrix material containing oil and water. The fractional height of the water table in the fracture is X_W and the fractional height of the water displacement front in the block of matrix material is X_w .

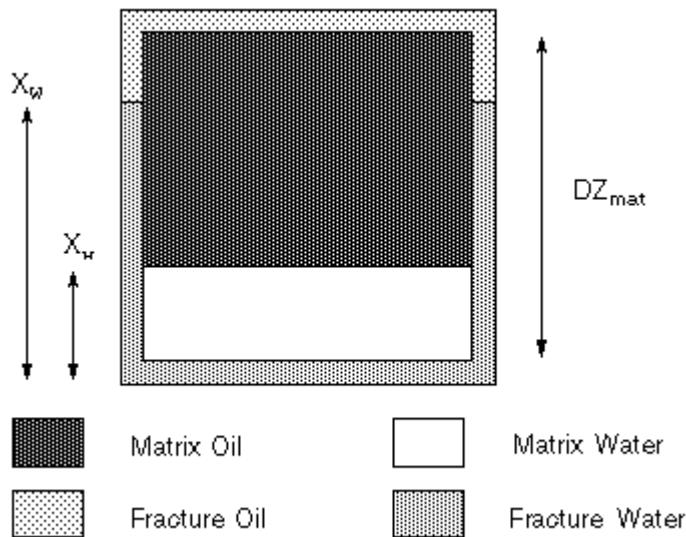


Figure 16.4. A typical block of matrix material containing oil and water

The pressure difference due to gravity alone is

$$\Delta P = DZ_{mat}(X_W - X_w)(\rho_w - \rho_o)g \quad \text{Eq. 16.4}$$

where

ρ_w is the density of water

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

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

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

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

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

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

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

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

$$F_g = TR \cdot GMOB \times \left(P_{of} - P_{om} + d_{fm} \rho_g g + P_{cogf} - P_{cogm} + \frac{DZ_{mat}(X_G - X_g)(\rho_o - \rho_g)g}{2} \right) \quad \text{Eq. 16.5}$$

$$F_o = TR \cdot OMOB \left(P_{of} - P_{om} + d_{fm} \rho_o g - \frac{DZ_{mat}(X_G - X_g)(\rho_o - \rho_g)g}{2} \right) \quad \text{Eq. 16.6}$$

where

TR is the transmissibility between the fracture and matrix cells,

GMOB is the gas mobility in the (upstream) fracture cell,

OMOB is the oil mobility in the (upstream) fracture cell,

P_{of} is the oil phase pressure in the fracture cell,

P_{om} is the oil phase pressure in the matrix cell,

d_{fm} is the difference in depth between the fracture and matrix cells (usually zero),

ρ_g is the density of gas at reservoir conditions,

ρ_o is the density of oil at reservoir conditions,

g is the acceleration due to gravity,

P_{cogf} is the capillary pressure of gas in the fracture cell (normally zero)

P_{cogm} is the capillary pressure of gas in the matrix cell.

Note: The gravity drainage head $DZ_{mat}(X_G - X_g)(\rho_o - \rho_g)g$ 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") 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 item 11 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

In 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:

$$\sigma_p^{\text{eff}} = f_p \cdot \sigma + (1-f_p) \cdot \sigma_{gd} \quad \text{Eq. 16.7}$$

where p =oil, gas, water, and

$$f_p = \frac{|\Delta P_p|}{|\Delta P_p| + |\Delta P_{pgd}|} \quad \text{Eq. 16.8}$$

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

$$\Delta P_{pij} = P_{pi} - P_{pj} + P_{pci} - P_{pcj} - g \bar{\rho}_p \Delta d \quad \text{Eq. 16.9}$$

where

(P_{pi}, P_{pj}) are the cell pressures,

(P_{pci}, P_{pcj}) are the cell capillary pressures,

Δd is the height difference between the cells,

g is the acceleration due to gravity

$\bar{\rho}_p$ is the saturation weighted average density of phase p

ΔP_{gd} is the gravity drainage term, which depends on the phase:

$$\Delta P_o = -\frac{1}{2} (\Delta P_{gd}^{og} + \Delta P_{gd}^{ow}) \quad \text{Eq. 16.10}$$

$$\Delta P_g = \frac{1}{2} \Delta P_{gd}^{og} \quad \text{Eq. 16.11}$$

$$\Delta P_w = \frac{1}{2} \Delta P_{gd}^{ow} \quad \text{Eq. 16.12}$$

$$\Delta P_{gd}^{og} = g \times DZ_{mat} (\bar{\rho}_o - \bar{\rho}_g) (X_{ig} - X_{jg}) \quad \text{Eq. 16.13}$$

$$\Delta P_{gd}^{ow} = g \times DZ_{mat} (\bar{\rho}_w - \bar{\rho}_o) (X_{iw} - X_{jw}) \quad \text{Eq. 16.14}$$

where X_g and X_w are the fractions of the cell containing mobile gas and water respectively. These are estimated from the cell saturation using a vertical equilibrium model. If the gravity drainage term is dominant, $f_p \rightarrow 0$ and $\sigma_{\text{eff}} \rightarrow \sigma_{gd}$.

ECLIPSE 100

Note that in ECLIPSE 100, the **SIGMAGD** keyword has a different function, and is used to differentiate oil-gas and oil-water mechanisms. See "[Variation of the sigma factor with drainage process](#)".

Alternative gravity drainage model (GRAVDRM)

An alternative gravity drainage model is available based on the formulation proposed by Quandalle and Sabathier in [Ref. 89].

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 wettability systems, the two transmissibilities give a better match to a finely gridded matrix block model.

The matrix fracture flow is given by:

$$F = F_h + F_{up} + F_{down} \quad \text{Eq. 16.15}$$

as shown in the following figure:

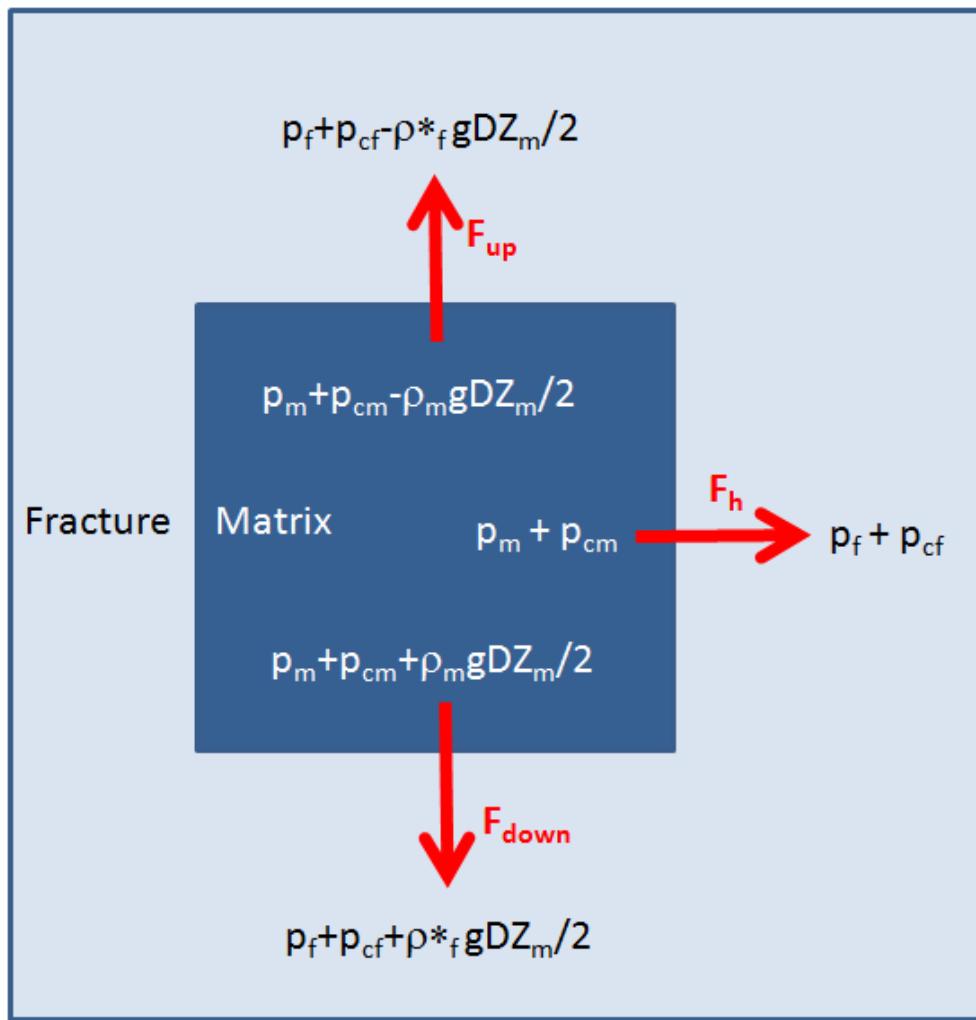


Figure 16.5. Flows and potentials in the GRAVDRM model

The horizontal flow is:

$$F_h = TR_{sigma} \cdot MOB_h \cdot (P_f + P_{cf} - P_m - P_{cm}) \quad \text{Eq. 16.16}$$

where:

TR_{sigma} is the transmissibility based on the **SIGMA** keyword (that is defined by equation 16.1 with σ values from the **SIGMA** keyword, and K described as in the text following equation 16.1). Note that there is no switching between the **SIGMA** and **SIGMAGD** keywords in this model: there is no effective σ as in equation 16.7 for ECLIPSE 300, or switching between **SIGMA** and **SIGMAGD** as described for ECLIPSE 100 in the **SIGMAGD** keyword description.

MOB_h is the phase (upstream) mobility. See MOB_v below for the definition of upstream.

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:

$$F_{down} = TR_{sigmaggd} \cdot MOB_{down} \cdot (P_f + P_{cf} - P_m - P_{cm} + (\rho_m - \rho_f^{star}) \cdot g \cdot (DZ_{mat})/2) \quad \text{Eq. 16.17}$$

$$F_{up} = TR_{sigmaggd} \cdot MOB_{up} \cdot (P_f + P_{cf} - P_m - P_{cm} - (\rho_m - \rho_f^{star}) \cdot g \cdot (DZ_{mat})/2) \quad \text{Eq. 16.18}$$

where:

$TR_{sigmaggd}$ is the transmissibility based on the **SIGMAGD** keyword

MOB_v (where $v = up$ or $down$) is the mobility of the fluid in the upstream cell for the relevant phase. The upstream cell is determined from the sign of the potential difference (which is the whole parenthesized expression in either equation 16.16, equation 16.17 or equation 16.18: this includes the gravitational potential energy term. The upstream cell has the higher total potential).

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 **LTONSIGMA** keyword in conjunction with the **LX**, **and **keywords.****

Vertical discrete matrix gravity drainage model

The third alternative gravity drainage model uses the discrete matrix model (ECLIPSE 100) or multi porosity model (ECLIPSE 300), both of which provide a number of matrix porosities connected together in a chain, with the outermost connected to the fracture. If the **NMATOPTS** keyword is set to the value

VERTICAL, then this chain of matrix porosities will be interpreted as a vertical stack of matrix cells connected within a single block of matrix material of height DZMATRIX. This value of the keyword will also connect all submatrix porosities to the fracture. The number of porosities used will still be defined by the [NMATRIX](#) keyword.

When these options are activated, the physical picture of the fracture cell and its associated matrix cells is as shown in figure 16.6. Each matrix cell sees the same fracture cell conditions. To prevent this limitation reducing the accuracy of the solution unacceptably, the properties of the fracture cell are altered for each NNC describing the matrix-fracture links to agree better with the property variations with height which would be experienced if the fracture were similarly discretized. This alteration takes into account the change with height of gravitational potential and any phase discontinuities (which are described in the same way as for the [GRAVDR](#) model).

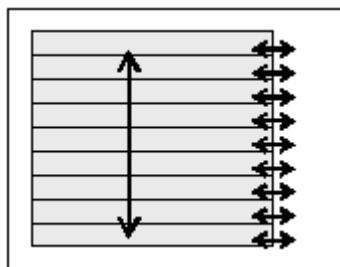


Figure 16.6. The discretized matrix material block within a fracture cell

Note: The connections to the fracture are shown explicitly; the discrete matrix model connections are shown as a single arrow.

ECLIPSE 300

In ECLIPSE 300, the input requirements for multi porosity give greater flexibility in the variation of properties at different heights of the matrix, but it is recommended that all the porosities are given the same property values.

Diffusion

Molecular diffusion of gas and oil between the matrix and fracture may be a significant production mechanism from the matrix. The ECLIPSE diffusion option is described in "[Diffusion](#)".

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. 41]).

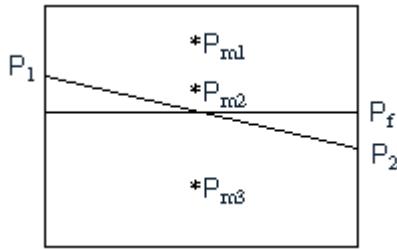


Figure 16.7. A representative matrix block in a fracture potential gradient

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,

$$F = \frac{T}{2} \cdot M_m (P_{m1} - P_1) + \frac{T}{2} \cdot M_m (P_{m1} - P_2) \quad \text{Eq. 16.19}$$

where:

T is the matrix-fracture transmissibility

M_m is the matrix mobility.

Hence

$$F = T \cdot M_m (P_{m1} - P_f) \quad \text{Eq. 16.20}$$

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 (for example P_{m3}) then the flow is from fracture to matrix, and as above can be expressed as:

$$F = T \cdot M_f (P_f - P_{m3}) \quad \text{Eq. 16.21}$$

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

$$F = \frac{T}{2} \cdot M_f \cdot (P_1 - P_{m2}) - \frac{T}{2} \cdot M_m \cdot (P_{m2} - P_2) \quad \text{Eq. 16.22}$$

This can be written as

$$F = \frac{T}{2} \cdot M_f \cdot \left(P_f + \frac{(P_1 - P_2)}{2} \right) \cdot P_{m2} - \frac{T}{2} \cdot M_m \cdot \left(P_{m2} - P_f + \frac{(P_1 - P_2)}{2} \right) \quad \text{Eq. 16.23}$$

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

$$F = \frac{T}{2} \cdot M_f \cdot (P_f - P_{m2}) - \frac{T}{2} \cdot M_m \cdot (P_{m2} - P_f) + \frac{T}{4} \cdot G \cdot L \cdot (M_f - M_m) \quad \text{Eq. 16.24}$$

The matrix-fracture flows are modified in a similar manner for all three directions.

The fracture potential gradient, G, is estimated from the potential differences to the surrounding cells.

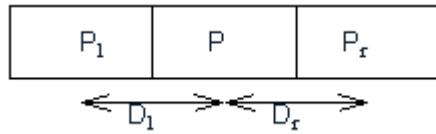


Figure 16.8. Estimating the potential differences to the surrounding cells

$$G = \frac{|P_l - P|}{2D_l} + \frac{|P - P_r|}{2D_r} \quad \text{Eq. 16.25}$$

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.

Note: Only certain keywords can be copied to the fracture in this way. Check the DPGRID keyword for the complete list

Use of FLUXNUM or MULTNUM or OPERNUM

The use of these keywords differs between ECLIPSE 100 and ECLIPSE 300.

ECLIPSE 100

Only the matrix cells need to be input. ECLIPSE 100 will copy these values to the fracture cells automatically. If both matrix and fracture cells are given, a warning message would be output complaining about too much data.

ECLIPSE 300

Both matrix and fracture cells should be provided. [FLUXNUM](#) and [MULTNUM](#) use with [MULTREGT](#) or [MULTREGP](#), will automatically copy all matrix cells to the corresponding fracture cells, so dummy fracture values could be input. For [OPERNUM](#) use, and other FLUXNUM and MULTNUM uses, there is no copying of data from the matrix to fracture cells.

Partially fractured reservoirs

In a dual-porosity single-permeability run, the simulation model may be specified to be dual porosity only in part of the reservoir, using the keyword [DPNUM](#). ECLIPSE treats the rest of the reservoir as in a conventional single porosity run. In this case, the single porosity cells are taken to be matrix cells for input purposes, and any data input for fracture cells in the single porosity region is ignored.

Diffusion

The Molecular Diffusion option can be limited to calculate only the matrix/fracture diffusive terms, by using the [DIFFDP](#) keyword. Here we are making the assumption that diffusion in the fracture system is negligible compared to the fracture flows. The matrix/fracture diffusivity can be modified using the [DIFFMMF](#) keyword in the GRID and SCHEDULE sections.

Modifying the recovery versus time response

ECLIPSE 100

The shape of the recovery versus time response can be modified by changing the matrix relative permeability curves. ECLIPSE provides a simplified method of changing the input relative permeabilities using only a single parameter. Typically this is useful when matching a dual porosity cell to a finely gridded single porosity model. The facility is described in more detail in a subsequent section and is activated with the [DPKRMOD](#) keyword.

Integrated capillary pressure option

This option automatically generates a modified capillary pressure function with the aim of improving the prediction of final recovery from a matrix cell. The option is described in detail in a subsequent section and is activated with the [INTPC](#) keyword.

Calculating SIGMA from the matrix dimensions

ECLIPSE 100

If the Viscous Displacement option has been used, the representative matrix block sizes will have to be entered using the [LX](#), [LY](#) and [LZ](#) keywords. It is possible to calculate the [SIGMA](#) value from these block dimensions by specifying the [LTOSIGMA](#) keyword. Note that if this keyword is used then any input SIGMA values are ignored.

Restrictions on dual porosity runs

The following restrictions apply to dual porosity ([DUALPORO](#)) runs (except in single porosity regions specified using [DPNUM](#)), but **not** to dual porosity, dual permeability ([DUALPERM](#)) runs.

- Wells connect only to fracture cells - not to matrix cells.
- Non-neighbor connections (keyword [NNC](#)) may not be used with matrix cells. The internal connection of each matrix cell to its appropriate fracture cell is made automatically by ECLIPSE.
- Each active matrix cell must connect with an active fracture cell.

In single porosity regions, there are no active fracture cells. Within these regions, data is only required for the matrix cells; any data for the fracture cells are ignored. Wells should connect to matrix blocks within single porosity regions.

For a coal bed methane model, the [DUALPERM](#) option should only be used if the instant adsorption model in ECLIPSE 300 is activated ([CBMOPTS](#)) or if the time-dependent adsorption model is used with Multi-Porosity. With the time-dependent sorption model, one of the submatrix porosities will then represent the pore volume of the matrix having conductivity to neighbor matrix grid blocks, while grid cells assigned to be coal will get a zero transmissibility.

Keyword summary

Restriction	Keyword	Description
GRID keyword	BTOBALFA	Initiates block to block connections and specifies a single contact area multiplier.
GRID keyword	BTOBALFV	Initiates block to block connections and specifies contact area multipliers.
Switch in RUNSPEC	DUALPORO	Initiates the dual porosity model.
Switch in RUNSPEC	DUALPERM	Initiates the dual permeability model.
Switch in RUNSPEC	GRAVDR	Requests alternative gravity drainage model
Switch in RUNSPEC	GRAVDRM	Requests alternative gravity drainage model
RUNSPEC keyword	NMATRIX	In ECLIPSE 100, requests the discretized matrix model. In ECLIPSE 300, requests the equivalent discretized matrix if the NMATOPTS option is used - otherwise see " Multi porosity ".
SIGMA or SIGMAV required	SIGMA	Matrix-to-fracture coupling factor for the entire grid. Matrix-to-fracture coupling factor for the current box.
SIGMA or SIGMAV required	SIGMAV	Matrix-to-fracture coupling multiplier for the entire grid.
SIGMA or SIGMAV required	MULTSIG	Matrix-to-fracture coupling multiplier for the current box.
DZMTRX or DZMTRXV required if gravity drainage is active.	MULTSIGV	Specifies the typical block height for gravity drainage for the entire grid.
DZMTRX or DZMTRXV required if gravity drainage is active	DZMTRX	Specifies the typical block height for gravity drainage for the current box.
E100 only Optional when the viscous displacement option is active	DZMTRXV	Specifies the representative X dimension of the matrix block.
E100 only Optional when the viscous displacement option is active	LX	Specifies the representative Y dimension of the matrix block.
E100 only Optional when the viscous displacement option is active	LY	Specifies the representative Z dimension of the matrix block.
E100 only Optional when the viscous displacement option is active	LZ	Restricts the molecular diffusion calculation to just the matrix-fracture flows.
Optional	DIFFDP	Enables user to enter grid data for the matrix cells only.
Optional	DPGRID	Identifies the extent of the regions in which the single porosity model may be applied in a dual porosity run.

Restriction	Keyword	Description
E100 only Optional if the hysteresis option is active	DPNUM	Imbibition table region for matrix-fracture flows.
E100 only Optional	IMBNUMMF	K_r region for matrix-fracture flows.
Optional	KRNUMMF	Specifies that permeabilities in the fracture cells are not to be multiplied by the fracture porosity.
Recommended	NODPPM	To specify different saturation region to the matrix and fracture cells.
Optional	SATNUM	In E100: Matrix-to-fracture coupling for oil-gas gravity drainage, for the entire grid. If this is used, SIGMA is the coupling for oil-water gravity drainage In E300: Alternative matrix-fracture coupling for all phases for gravity-drainage dominated potentials.
Optional	SIGMAGD	SIGMAGD by cell for the current box.
Optional	MULSGGD	Matrix-to-fracture multiplier for SIGMAGD, for the entire grid.
Optional	MULSGGDV	Matrix-to-fracture multiplier for SIGMAGD by cell, for the current box.
Optional	INTPC	Invokes the integrated capillary pressure option
E100 only Optional	DPKRMOD	Modifies oil K_r curve for matching fine grid single porosity recovery vs. time, and allows scaling of the fracture relative permeability.
E100 only Optional if the viscous displacement option is active	LTOSIGMA	Requests that the SIGMA value is to be calculated from the LX, LY and LZ data.
Optional if the discretized matrix option is active	NMATOPTS	Specifies the geometry type of the matrix and the size of the outer matrix cell.
E300 only Optional	PERMMF	Permeability for Matrix-Fracture Coupling.
E300 only Optional	MULTMF	Multiplier for Matrix-Fracture Permeability.
Optional	DIFFMMF	Multiplier for Matrix-Fracture Diffusivities.

Table 16.1: Summary of dual porosity keywords

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_{cogn} to match the observed data or match to a finely gridded single porosity model.

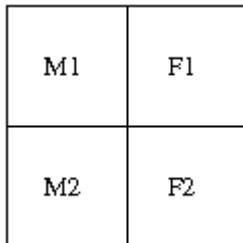


Figure 16.9. A simple reservoir grid used to simulate a gas/oil transfer function

Modifying the recovery versus time response (ECLIPSE 100)

The eventual recovery of oil from a matrix block is governed by the final balance between the gravity head and the capillary pressure. The initial rate of drainage is governed by the transmissibility between the matrix and fracture, which in turn is controlled by the shape factor SIGMA. The ‘shape’ of the recovery curve between the end points can be modified by changing the relative permeability function for the matrix-fracture flow.

This can be done by modifying the relative permeability curves directly. However, when matching a finely gridded single porosity model or experimental results it is often simpler to modify a single parameter to change the ‘shape’ of the recovery versus time response. The [DPKRMOD](#) keyword applies a quadratic modification to the oil relative permeability to achieve this aim.

The modification factor is not a physical quantity and should be regarded only as a tuning parameter to match a dual porosity block with a finely gridded single porosity model.

The scaling for oil-in-water relative permeability is as follows.

For the case where m_w (the modification factor set in DPKRMOD) is greater than 0.0:

$$K_r = \bar{K}_r + M(K_r(s) - \bar{K}_r) \quad \text{Eq. 16.26}$$

where

\bar{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 $M = 4.0m_w K_r(s)^2 - 4.0m_w K_r(s) + 1$

For the case where 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.

Given a saturation s , a new saturation s' is calculated such that:

$$\bar{s} + M(s' - \bar{s}) = s \quad \text{Eq. 16.27}$$

where

\bar{s} is $(\text{SOWCR} + 1.0 - \text{SWCO}) / 2$

M is the modification function, which is given by $M = 4.0m_w K_r(s')^2 - 4.0m_w K_r(s') + 1$

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:

$$K_{rf}(S_w) = K_{rf}^{\text{tab}}(S_w) \cdot K_{rm \max} / K_{rf\max} \quad \text{Eq. 16.28}$$

where

$K_{rf}^{\text{tab}}(S_w)$ is the fracture relative permeability looked up using the input table

$K_{rm \max}$ is the matrix relative permeability at $S_w = 1.0 - S_{owcr}$

$K_{rf\max}$ is the maximum fracture relative permeability

S_{owcr} is the critical oil saturation to water.

The other phases are treated in an analogous manner. In the gas case the relative permeability is scaled to the matrix value at $S_g = 1.0 - S_{ogcr}$ and in the oil case at $S_o = 1.0 - \max(S_{crg}, S_{crw})$.

Solution of the linear equations

The linear equations associated with dual porosity/permeability simulations can be written in the form

$$Ax + Uy = R_m$$

$$Lx + By = R_f$$

where

A is the usual banded matrix for the matrix cells

B is the banded matrix for the fracture cells

R_m 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

$$(B - L \cdot A^{-1} U)y = R_f - L \cdot A^{-1} R_m \quad \text{Eq. 16.29}$$

$$x = A^{-1}(R_m - Uy) \quad \text{Eq. 16.30}$$

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 three-phase run, the diagonal entries of the Jacobian matrix are then 6×6 matrices of the form.

$$D = \begin{bmatrix} D_m & U \\ L & D_f \end{bmatrix} \quad \text{Eq. 16.31}$$

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](#)"). Using the notation of that chapter, an approximation, B , is constructed to the full Jacobian matrix.

$$A = D + L1 + U1 + L2 + U2 + L3 + U3 \quad \text{Eq. 16.32}$$

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

$$\begin{aligned} B &= (P + L3)P^{-1}(P + U3) \\ P &= (T + L2)T^{-1}(T + U2) \\ T &= (G + L1)G^{-1}(G + U1) \\ G &= (\Gamma + L)\Gamma^{-1}(\Gamma + U) \end{aligned} \quad \text{Eq. 16.33}$$

where Γ is a diagonal matrix (whose elements are all 3×3 matrices in the three-phase case), and G has the same structure as D .

ECLIPSE 100

If item 60 of the OPTIONS keyword is set greater than zero, the less efficient dual permeability solver for pre-97A versions of ECLIPSE is restored. If you encounter problems with your simulation run you may also wish to use this option.

Calculation of fractional volumes for gravity drainage

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

$$X_{wi} = \frac{S_{wi} - S_{wco}}{1 - S_{cohy} - S_{wco}} \quad \text{Eq. 16.34}$$

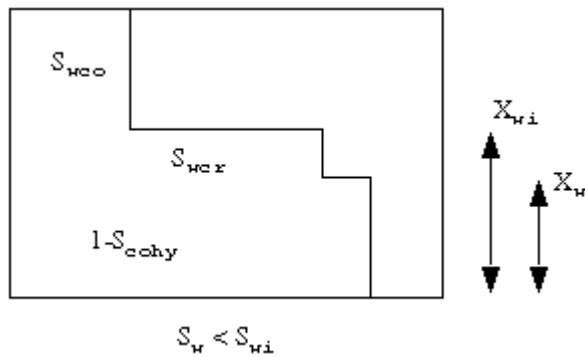


Figure 16.10. Initial fractional volume of water below the water contact

where

S_{wi} is the initial water saturation

S_{wcr} is the critical water saturation

S_{wco} is the connate water saturation

S_{cohy} is the connate hydrocarbon saturation

$= S_{ocow}$ (oil/water systems)

$= S_{ocow} + S_{gco}$ (oil/water/gas systems)

$= S_{gco}$ (gas/water systems)

S_{crhy} is the critical hydrocarbon saturation.

If $S_w > S_{wi}$ then

$X_w = 1$ when

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

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

If $S_w < S_{wi}$ then

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

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

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

$$X_{gi} = \frac{S_{gi} - S_{gco}}{1 - S_{lco} - S_{gco}} \quad \text{Eq. 16.35}$$

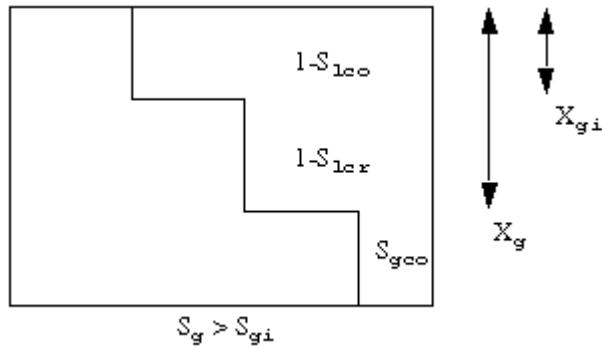


Figure 16.11. Initial fractional volume of gas above the gas contact

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_{ogow} (oil/gas systems)

= $S_{ogow} + S_{wco}$ (oil/water/gas systems)

= S_{wco} (gas/water systems)

S_{lcr} is the critical liquid saturation

If $S_g > S_{gi}$ then

$$X_g = 1 \text{ when } S_g = S_{g \text{ max}} = X_{gi}(1 - S_{lco}) + (1 - X_{gi})(1 - S_{lcr})$$

$$X_g = (S_g - X_{gi}(S_{lcr} - S_{lco}) - S_{gco}) / (1 - S_{lcr} - S_{gco}) \text{ when } S_g < S_{g \text{ max}}$$

If $S_g < S_{gi}$ then

$$X_g = 0 \text{ when } S_g = S_{g \text{ min}} = X_{gi}S_{gcr} + (1 - X_{gi})S_{gco}$$

$$X_g = (S_g - X_{gi}(S_{gcr} - S_{gco}) - S_{gco}) / (1 - S_{lco} - S_{gcr}) \text{ when } S_g > S_g \text{ min}$$

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 three-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.

ECLIPSE 300

Note that in ECLIPSE 300, the [SIGMAGD](#) keyword has a different function, as described in "[Calculation of fractional volumes for gravity drainage](#)".

Rock volume and cell activation in dual porosity thermal calculations

Rock volume

In single porosity calculations, the rock volume is simply the cell volume less the pore volume. In dual porosity calculations, this leads to a double counting of rock volume, as the cell volume is in reality common to all the cells of different porosity sharing the same location. To prevent this, the rock volume, defined as the average cell volume for all porosities less the total pore volume in all cells in that location, is shared between the porosities. Each cell receives the same rock volume as its pore volume. Any remaining rock volume is given to the matrix. If there is not enough rock volume to achieve this, the rock volume is shared in proportion to the cell pore volume.

As a special case, if a cell in one porosity has ACTNUM=2 (see below), meaning that the cell is rock only, then the remaining rock is shared equally among all the porosities for that cell location.

As a special case, if a cell in one porosity has ACTNUM=3 (see below), meaning that the cell is pore-volume only, then the pore volume assumed is the physical volume divided by the number of porosities. If an asymmetric pore volume distribution is required, the net-to-gross value can be adjusted as a weighting factor.

If this option does not achieve the desired rock distribution, the user can override it with the [ROCKSPLV](#) keyword.

ACTNUM

The [ACTNUM](#) keyword allows the user to deactivate cells. In thermal calculations, the user can also deactivate either the fluids or the rock to give an energy-only or flow-only calculation in that cell.

In dual porosity, total deactivation of the matrix causes deactivation of the fracture, and the other way round. But rock-only or flow-only deactivation only affects the porosity specified. The specification of rock-only or flow-only cells does not affect the distribution of rock among the porosities, as described above; however, the rock or pore volume calculated may be set to zero by the ACTNUM flag.

There is no automatic default for synchronizing ACTNUM rock-only or flow-only deactivations with the rock distribution. It is suggested that the user takes advantage of the possibility to deactivate using porosity (keyword [PORO](#)) or pore volume (keyword [PORV](#)) and [ROCKSPLV](#) to control deactivation through the volume sizes on a per cell, per porosity basis.

If a cell is rock-only (ACTNUM=2) then the rules for apportioning rock volume are modified; see above. If a cell is pore-volume only (ACTNUM=3) then the rules for apportioning pore volume are modified; see above.

Integrated capillary pressure option

When a gravity drainage option is in use, the final recovery from the matrix is determined by the balance between the capillary pressure and gravity forces. The final saturation distribution in the matrix block can be thought of as a transition zone where at each height the capillary pressure is $P_c = \Delta \rho \cdot g \cdot h$.

Within the dual porosity matrix representation, it is desirable that the final recovery should be equal to the final recovery from the continuous matrix medium. This can be achieved by modifying the capillary pressure curves in the matrix to take account of the fact that the rock P_c curves need to be integrated over the matrix block height to calculate the average saturation.

The integrated P_c option is activated by using the [INTPC](#) keyword in the PROPS section. The appropriate modified capillary pressure curves are written to the PRINT file for inspection if either the SWFN or SGFN mnemonics are used in the [RPTPROPS](#) keyword.

The integration procedure for the gas-oil capillary pressure is as follows:

We start with a piecewise 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.

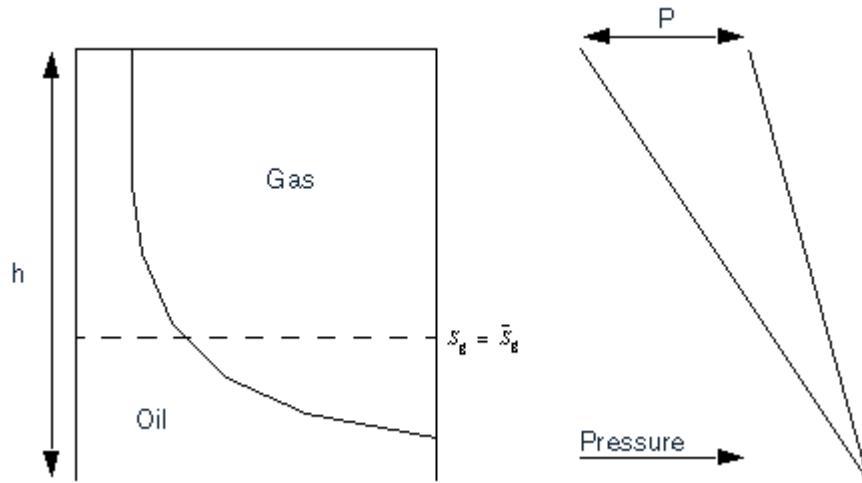


Figure 16.12. Integrated capillary pressure

The average gas saturation \bar{S}_g in the matrix block up to the height h equivalent to $p_c = P$ is given by

$$\bar{S}_g = \frac{1}{P} \cdot \int_0^P P_c^{-1}(p) dp \quad \text{Eq. 16.36}$$

We can now set up a modified capillary pressure function such that the capillary pressure at a saturation \bar{S}_g is equal to the gravity drainage head for a block of height h and saturation \bar{S}_g .

The gravity drainage model accounts for the gravity head within a grid block, by assuming vertical equilibrium. (Note that if the gravity drainage option is not used the gravity head within a grid block is ignored.)

For the gravity drainage model selected by [GRAVDR](#), the gravity head P_{gdr} is proportional to the mobile oil saturation, assuming the fracture is filled with gas:

$$P_{gdr} = \frac{S_t - \bar{S}_g}{S_t} \cdot \Delta\rho \cdot g \cdot h \quad \text{Eq. 16.37}$$

where

S_t is 1-SWCO-SOGR

SWCO is the connate water saturation

SOGCR is the critical oil-in-gas saturation

$\Delta\rho$ is the difference between the gas and oil densities

g is the acceleration due to gravity

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 \bar{S}_g the capillary pressure is exactly equal to P_{gdr} and hence the equilibrium saturation will be at (and only at) $S_g = \bar{S}_g$.

The constructed capillary pressure function P_c' should have $P_c'(\bar{S}_g) = P_{gdr}$, and thus

$$P_c'(\bar{S}_g) = \frac{S_t - \bar{S}_g}{S_t} \cdot P \quad \text{Eq. 16.38}$$

where P is the true capillary pressure at the top of the block when the equilibrium saturation is \bar{S}_g . This can be calculated by looking up the table of \bar{S}_g versus P given by Equation 16.36, which is stored in ECLIPSE as a table.

For the alternative gravity drainage model selected by [GRAVDRM](#), the gravity head P_{gdr} is proportional to $P/2$, assuming the fracture is filled with gas. This condition requires that oil is not re-infiltrating into the matrix, and this possibility should be excluded using the GRAVDRM keyword.

Notes

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

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

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

- At other saturations ($S \neq \bar{S}$) a potential difference generates the usual flow to and from the block.
- 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 subdivides 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 one dimensional grid system. Each matrix grid cell could be regarded as a “Russian Doll” of nested matrix sub-cells.

Matrix sub-grid

The matrix subdivision 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. If NMATOPTS is omitted, then provided NMATRIX is present, the default values from NMATOPTS will be assumed.

First an overview of the discretization is given for the linear and non-linear geometry types, and then a more detailed view of the volume and transmissibility calculations will be provided.

Overview of Linear geometry

If the geometry is assumed to be 1D then the SIGMA factor and hence the matrix-fracture transmissibility are assumed to represent 1D flow.

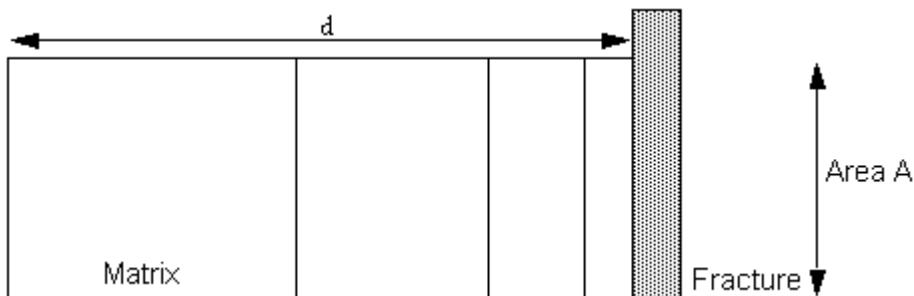


Figure 16.13. Nested matrix sub-cells in linear geometry

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.

Overview of cylindrical and spherical geometries

In the 2D (Cylindrical) and 3D (Spherical) flow options the geometry is taken to be Cartesian to enable simple mapping from the input matrix-fracture transmissibility.

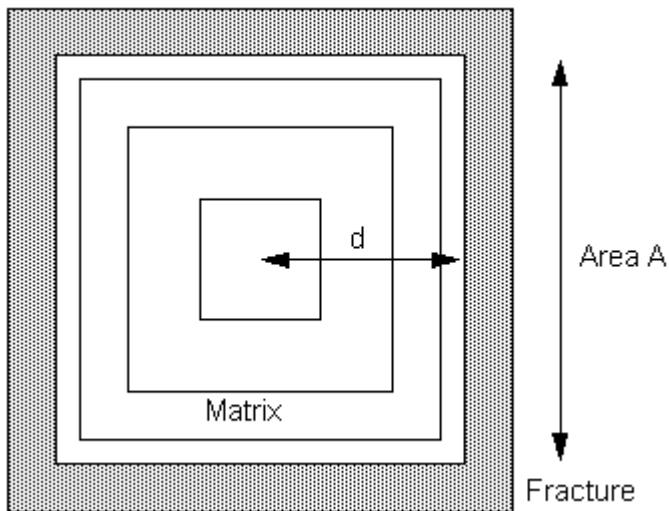


Figure 16.14. Nested matrix sub-cells in cylindrical and spherical geometry

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.

Volume and transmissibility calculations details

Let the n^{th} sub-matrix cell have volume v_n , where $n=1$ is the sub-cell nearest the fracture, and v_N is the sub-cell furthest away. N is the value input by keyword [NMATRIX](#). Then:

$$V_m = \sum_n v_n$$

where V_m is the volume of the whole matrix region.

The outermost sub-cell volume v_1 is determined by the fraction of the fracture pore volume specified by item 2 of the [NMATOPTS](#) keyword. The sizes of the remaining sub-cells are then functions of the factor

$$f = \left(\frac{X}{\xi_1} \right)^{\frac{1}{N-1}}$$

where

X is the distance of the center of the matrix from the fracture surface, and

ξ_1 is the width of the outermost sub-cell.

For a LINEAR assumed geometry (NMATOPTS item 1), the distances X and ξ_1 are assumed proportional to the volumes V_m and v_1 respectively. For CYLINDRICAL or SPHERICAL geometries, these distances are

calculated from the geometry of a cylindrical or spherical shell with inner and outer radii X and $X-\xi_1$, using the known volumes V_m and v_1 for the whole cylinder/sphere and the shell.

The factor f gives the distance ξ_n from the matrix-fracture surface to the inner surface of the remaining ($n > 1$) interior sub-cells by

$$\xi_n = f^{n-1} \xi_1$$

that is, the logarithms of the ξ_n scale linearly.

As with ξ_1 , when the assumed geometry is CYLINDRICAL or SPHERICAL, the sub-cells are cylindrical or spherical shells around the center of the matrix, and $X-\xi_n$ is the inner radius of the shell. The shell volume is determined from the inner and outer radii, with the same constant of proportionality as used to determine X and ξ_1 in the first place. For a LINEAR assumed geometry the sub-cells are assumed to be slabs parallel to the matrix-fracture surface, arranged from the surface to the center of the matrix, and the volumes are assumed proportional to the distances. The overall effect of this is to obey the relationship

$$\frac{(X^D - (X-\xi_1)^D)}{X^D} = \frac{v_1}{V_m}$$

The transmissibilities between the sub-cells are then determined by the area of the surface between them and the distance between the sub-cell centers (where for CYLINDRICAL and SPHERICAL geometries, cell center means the radius halfway between the cell's two surfaces, and for LINEAR geometries, the position halfway between the cell's two surfaces). Thus for any geometry, the center of cell n ($n > 1$) is at

$$c_n = \frac{1}{2}(\xi_{n-1} + \xi_n)$$

and the area of the interface between cells n and $n+1$ is

$$A_n = (X-\xi_n)^{D-1}$$

where D is the dimensionality (LINEAR=1, CYLINDRICAL=2, SPHERICAL=3).

Then the transmissibility of the original matrix-fracture connection is scaled to the sub-cell to sub-cell connection by the factor

$$G_n = \left(\frac{A_n}{A_X} \right) \left(\frac{2X}{c_n - c_{n-1}} \right)$$

where A_X is the area associated with overall matrix size X .

For the transmissibility between the outer matrix sub-cell and the fracture, the factor G_1 is $2X / \xi_1$

Initialization, solution method and efficiency

The internal sub-matrix cells cannot be explicitly initialized in the way that the normal matrix cells can, and so the [EQUI](#) keyword must be used to perform an initialization.

The matrix sub-grid forms a series of separate 1D flow systems connected through the outer sub-cells only to the corresponding fracture cells. In the solution matrix structure these flows lead to a set of tri-diagonals connected to the fracture system. This structure is used in the pre-conditioning step by eliminating the tri-diagonals before the fracture system is solved, effectively reducing the problem to a set of flow equations

for the fracture only. Once the fracture equations have been solved a back substitution gives the matrix sub-grid solution.

Clearly, discretizing the matrix increases the total number of cells in the model. However, the efficiency of the above procedure ensures that increasing the number matrix sub-cells will have a significantly smaller effect on the run time than a similar increase in the number of simulation grid blocks.

Reporting the matrix solution

The solution for separate sub-cells (or rings) of the matrix can be output as arrays over the simulation grid by setting any of the first 5 data items in [RPTSCHE](#) to 2. If an individual sub-cell solution is required as a function of time, the following summary keywords are available:

BPR, BRS, BRV, BOSAT and BWSAT.

The matrix sub-cell is specified by appending its ring number to the end of the keyword, for example:

```
BOSAT7
 1 1 2 /
 /
```

The ring number starts from 1 for the outer sub-cell and increases towards the center of the matrix cell. Note that ring 1 is equivalent to the keyword without a ring number and represents the matrix sub-cell adjacent to the fracture.

Restrictions

The option cannot at present be used with the following:

- gravity drainage models ([GRAVDR](#), [GRAVDRM](#))
- Local grid refinement option
- Dual permeability model
- Parallel option

and must use the following to initialize the sub-matrix cells

- [EQUIL](#)

Discretized matrix model (ECLIPSE 300)

The discretized matrix model can be invoked in ECLIPSE 300 in a similar way to ECLIPSE 100. However there are some differences in the setup as the ECLIPSE 300 model is based on the multi-porosity options described in "[Multi porosity](#)". The [DIMENS](#) keyword needs to include all porosities in the Z-direction.

The model generates new [SIGMAV](#) values for the connections between the matrix sub-grid cells, following the geometric partitioning as described for ECLIPSE 100. The input primary (outer) matrix block volumes, pore volumes and rock volumes, are partitioned between the matrix sub-grid cells. Both transmissibilities and diffusivities are then generated from the new [SIGMA](#) values, the block volumes, pore volumes and rock volumes (the rock volumes are only used if the rock contains adsorbed gas). Permeabilities, for the transmissibilities as given by [PERMX](#) or [PERMMF](#), are by default copied from the primary matrix grid cells if unspecified.

The geometrical partitioning is done in the same way as for ECLIPSE 100 for LINEAR, RADIAL and SPHERICAL discretization methods. In addition the UNIFORM and VERTICAL methods are available in ECLIPSE 300. For information, see the [NMATOPTS](#) keyword.

The volumes of the partitioned outer matrix grid cells, that connect to the fracture grid cells, can be set either

- to match a fraction of the fracture pore volume using NMATOPTS item 3 set to FPORV, or
- to match a fraction of the matrix bulk volume using NMATOPTS item 3 set to MBLKV.

When the feature is used with coal or shale gas, the definition of volumes with adsorbed gas needs to be considered. For information on this, see "[Shale Gas](#)". It is recommended that the keyword [ROCKFRAC](#) is used to set the input rock fractions and that the [PORO](#) keyword is used to set the input porosity fractions. This makes it easier to switch between the instant adsorption model and the time-dependent sorption model, since the inputs for setting rock and pore volumes are compatible. Only the input from the primary matrix grid cells are used to generate the partitioned matrix volumes.

Further information about the ECLIPSE 300 matrix discretization model can be found in "[Multi porosity](#)".

Single medium conductive fractures (ECLIPSE 300)

An alternative dual porosity formulation in ECLIPSE 300 is the Single Medium Conductive Fractures development. This is based on a generalization of the technique described in van Lingen et al, 2001 [[Ref. 116](#)].

The technique allows the incorporation of the effects of conductive fractures into a single medium model (that is not explicitly using a dual system of porosities or permeabilities) by modification of grid properties and the generation of pseudo tables. It is focused on modeling fractures on scales that are too large to be homogenized using the traditional approach.

For further details of the technique, please refer to [[Ref. 116](#)]. The sections below assume that the reader has familiarity with the paper, and outline the approach taken in ECLIPSE 300, including extensions and generalizations on the technique described in the paper, and how to use the ECLIPSE 300 development.

The single medium approach

For a given input fracture swarm distribution map, and associated fracture properties, there are three subsequent main areas of implementation:

- Grid property modifications
- Pseudo generation
- Well PI modifications

The implementation of the technique in these areas is described in the following paragraphs.

Grid property modifications

Firstly grid block properties are modified to take account of the physical void introduced by the fractures. The porosities are increased by proportional volume averaging, and permeabilities flow-averaged using Darcy's Law, either arithmetically (parallel to fracture) or harmonically (perpendicular to the fracture).

The approach taken in ECLIPSE 300 differed to that in the paper in

- the full use of all cell dimensions (rather than assuming square cells)
- the calculation of directional permeability enhancements (parallel and perpendicular components)
- allowing for vertical permeability enhancement
- allowing for fracture volumes in porosity modifications.

One further modification is to recognize fractures being defined in a zigzag rasterized format. When the path of a fracture changes direction by 90 degrees within one cell, the effective length of the fracture is halved. This attempts to model the intention that the fracture does not traverse the cell in question twice, but rather somehow ‘turns a corner’.

Pseudo generation

The saturation curves of the fractures and affected grid blocks are combined under the assumption that oil is displaced from fractures by either gas or water before any oil is displaced from the matrix volume of a grid block, which requires that any capillary imbibition is dominated by viscous and gravity effects. Under this assumption, the saturation tables are averaged by determining end point saturations from mass conservation, and end point relative permeabilities by Darcy's Law arithmetic averaging of effective phase flows. The original fracture tables are then used in a scaled form during the fracture invasion by gas or water, and scaled matrix tables are used for matrix invasion. Notice that (as in the paper) the fracture tables are assumed straight lines; no attempt is made to combine the full shape of the curve with that of the matrix.

Since the permeabilities and saturation tables used potentially vary on a grid block basis, the above could feasibly result in one set of pseudo tables for each affected cell. To reduce the dimensionality, a grouping technique is used that averages the pseudo end points used over cells with common attributes, these being saturation region, permeability value, and the number of contained fractures. This extended the paper's treatment which assumed that layers represented the saturation regions.

Note: Capillary curves are not included in this treatment, and are merely extrapolated at constant value from the matrix end-point values.

Well PI

By default, wells completed in grid blocks containing conductive fractures will experience enhanced flows due to the increased effective permeability, due to both the permeability and relative permeability changes. [Ref. 116] argues that this only arises if the fracture actually cuts the well bore in a connection cell. Because of this, you are required to specify whether or not a well connection cell is to be modified by this treatment by indicating whether it is cut by fractures or not. If not, then the well PI is modified so as to reverse the changes in permeability, and the connection cell saturation table reverts to that of the original matrix block. You are also given the chance to enter a PI multiplier to indicate perhaps some degree of connection with conductive faults.

Using the conductive fractures model

The conductive fractures model is activated by specification of the [SCFDIMS](#) keyword in the RUNSPEC section, which also sets up dimensioning defaults for reading in the fracture details, and also defaults for the pseudo grouping algorithms used.

The conductive fractures themselves should be specified using the [CONDFRAC](#) keyword, one for each fracture being modeled. This specifies the path of fracture planar segments through the grid blocks, the effective aperture, fracture permeability, and saturation table number to use. The format is similar to the [FAULTS](#) keyword, but note that the fractures are assumed to pass through a block, whereas faults run alongside blocks. For this reason, when a ‘corner’ is detected in the fracture, the contribution from the two intersecting segments is halved, to reflect the fact that the corner is formed notionally at the center of the cell, rather than around the edges.

The well intersection behavior is controlled using the [WELLCF](#) keyword.

Compatibility

The feature has been explicitly compatible with the restart mechanism, and will run in parallel mode. It is currently restricted to family (ii) saturation table keywords - SWFN, SGFN and SOF2/SOF3.

The development is known not to be compatible with the following options:

- radial grids
- analytic saturation functions
- geomechanics
- MPFA
- Up-front memory
- LGRs

17

Environmental tracers

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

The environmental tracers option is an extension to the ECLIPSE 100 passive tracer tracking model; it is not available in ECLIPSE 300. The option enables the modeling of contaminants and other substances as they flow within a host water, oil or gas phase.

The passive tracer tracking models give you the ability to track up to 50 tracer substances within a single model. The environmental tracer option extends the modeling to account for adsorption of the tracer on to the bulk rock, for decay of the tracer over time, and for molecular diffusion of the tracer. It is possible to model adsorption, decay and diffusion within a single tracer.

The environmental tracer model

As for passive tracers, the flow of an environmental tracer through the porous medium is assumed to have no influence on the flow of the water and hydrocarbon phases or on the flow of other tracers.

Environmental tracers are therefore solved, in a similar fashion to passive tracers, at the end of a timestep after the oil, water and gas equations have converged.

For an environmental tracer present in a single phase the governing equation is:

$$\frac{d}{dt}\left(\frac{VSC}{B}\right) + \frac{d}{dt}\left(V\rho_r C^a \frac{1-\Phi}{\Phi}\right) = \Sigma \left[\frac{Tk_r}{B\mu} (\delta P - \rho g D_z) C + DFD_c S \delta C \right] + QC - V \frac{S}{B} \lambda C \quad \text{Eq. 17.1}$$

where

S denotes the host phase saturation

C denotes the flowing tracer concentration

C^a denotes the adsorbed tracer concentration

ρ_r denotes the mass density of the rock formation

φ denotes the porosity

ρ denotes the host phase density

- μ denotes host phase viscosity
- D_z denotes the cell center depth.
- B denotes the host phase formation volume factor
- T denotes the transmissibility
- k_r denotes the host phase relative permeability
- V denotes the block pore volume
- Q denotes the host phase production rate
- P denotes the host phase pressure
- g denotes the acceleration due to gravity
- D_c denotes the tracer diffusion coefficient
- DF denotes the diffusivity
- λ denotes the tracer decay constant
- Σ denotes the sum over neighboring cells

Treatment of adsorption

Adsorption is treated as an instantaneous effect in the model. The effect of tracer adsorption is to strip tracer from the leading edge of a tracer front. Desorption effects may occur as the tracer concentration decreases.

The adsorption model can handle both stripping and desorption effects. You specify an adsorption isotherm, which tabulates the saturated rock adsorbed concentration versus the local tracer concentration in solution. As an alternative to tabulating, the isotherm coefficients can be supplied for an analytical isotherm which allows for dependencies of adsorption on rock permeability.

There are currently two adsorption models which can be selected. The first model ensures that each grid cell retraces the adsorption isotherm as the tracer concentration rises and falls in the cell. The second model assumes that the adsorbed tracer concentration on the rock may not decrease with time, and hence does not allow for any desorption. More complex models of the desorption process may be implemented if required.

To ensure equilibrium, the initial input tracer concentrations can either partition between the fluid and solid, thereby conserving the total input concentration in the fluid, or can be honored by adding adsorbed tracer concentration to the rock. The choice of initialization model is specified in item 3 of the keyword [TRROCK](#).

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 equation 17.1:

$$F_t = DF \cdot D_c S (C_{ci} - C_{cj}) \quad \text{Eq. 17.2}$$

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

$$DF = \frac{A \cdot \varphi}{d} \quad \text{Eq. 17.3}$$

where

A denotes the interface area between the two cells

d denotes the distance between the cell centers

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

In a system where there is significant convection of the tracer, molecular diffusion is likely to be a relatively small effect. Numerical dispersion associated with the discrete approximation of the flow equation is likely to dominate any molecular dispersion. However, the diffusion model is useful when the convection is small. In the limiting case of zero permeability, the only mechanism for flow is by diffusion; this kind of situation may occur when modeling a contaminant initially stored in concrete.

Partitioned tracers

Partitioned tracers allow ECLIPSE 100 to solve a range of problems that could not be tackled otherwise. For example, if marked gas is injected into a reservoir it is possible that the marker may dissolve in the water. The partitioned tracer option can model this behavior. With a single phase tracer in, say, the gas phase, ECLIPSE solves a conservation equation for the total amount of tracer in a grid block taking into account the inflow and outflow of gas, equation 17.1. With a partitioned tracer one or more additional phases can be nominated (in the [TRACER](#) keyword) into which the tracer may also dissolve.

Note: The partitioned tracer functionality can be used independently of adsorption, decay and diffusion. Hence, it is possible to have a standard passive tracer which partitions into multiple phases.

Standard partitioned tracer

A standard partitioned tracer can exist in two phases which are specified in the [TRACER](#) keyword. For input and output purposes these two phases are regarded as being the ‘free’ and ‘solution’ phases, so that the conventional ECLIPSE 100 keywords for two-phase tracers can be used. The ‘free’ phase can be thought of as the reference phase for the tracer.

If we use subscripts ‘f’ and ‘s’ to denote the free and solution phases, respectively, the governing equation for a standard environmental partitioned tracer is:

$$\begin{aligned} & \frac{d}{dt} \left(V \left[\frac{S_f}{B_f} C_f + \frac{S_s}{B_s} C_s \right] \right) + \frac{d}{dt} \left(V \rho_r C^a (C_s) \frac{1 - \Phi}{\Phi} \right) \\ &= \Sigma [F_f + F_s] + Q_f C_f + Q_s C_s - V \frac{S_s}{B_s} \lambda C_s \end{aligned} \quad \text{Eq. 17.4}$$

$$\begin{aligned} F_f &= \frac{Tk_f}{B_f \mu_f} (\delta P - \rho_f g D_z) C_f \\ F_s &= \frac{Tk_{rs}}{B_s \mu_s} (\delta P - \rho_s g D_z) C_s + DFD_c S_s \delta C_s \end{aligned} \quad \text{Eq. 17.5}$$

The concentration of the tracer in the solution phase is not independent of the concentration in the free phase. The concentrations are related through the partitioning functions:

$$K(P) = \frac{C_s}{C_f} \quad \text{Eq. 17.6}$$

where the functions $K(P)$ are entered in tabular form in the [TRACERKP](#) keyword and are required for each grid block. The particular table to be used is specified with the [TRKPFxxx](#) keyword, which is input in the REGIONS section. A table number is also required for use within the wellbore, and this is specified in item 5 of the TRACER keyword.

As shown in equations 17.4 and 17.5, when a standard partitioned tracer is used in conjunction with adsorption, decay and diffusion, it is assumed that these processes happen only in the solution phase. For example, the adsorption function, $C^a(C_s)$, in equation 17.4 is evaluated based on the tracer concentration in the solution phase.

Multi-partitioned tracers

The multi-partitioned tracer option is an extension and generalization of standard partitioned tracers. A multi-partitioned tracer, which is defined by specifying MULT in the item 4 of the [TRACER](#) keyword, can partition into any number of phases and have phase specific adsorption, decay and diffusion parameters.

The governing equation for a generalized, multi-partitioned tracer is:

$$\frac{d}{dt} \left(V \sum_j \frac{S_j}{B_j} C_j \right) + \frac{d}{dt} \left(V \rho_r C^a(C) \frac{1 - \Phi}{\Phi} \right) = \sum_j F_j + \sum_j Q_j C_j - V \sum_j \frac{S_j}{B_j} \lambda_j C_j \quad \text{Eq. 17.7}$$

$$F_j = \frac{Tk_{rj}}{B_j \mu_j} (\delta P - \rho_j g D_z) C_j + DFD_{cj} S_j \delta C_j \quad \text{Eq. 17.8}$$

where the summation over j indicates the phases in which the tracer is present, typically (a subset of) oil, water and gas.

As for standard partitioned tracers, the concentrations of the tracer in different phases are related through pressure dependent partitioning functions. For a multi-partitioned tracer these functions are entered in tabular form in the [TRACERKM](#) keyword. The reference phase is taken as the phase specified in the second item of the TRACER keyword. TRACERKM then specifies partitioning functions relating the concentration in each of the solution phases to the concentration in the reference phase. For example, for a tracer partitioning into oil, water and gas with water as the reference phase, TRACERKM holds two columns of data specifying the functions:

$$K_{ow}(P) = \frac{C_o}{C_w} \quad K_{gw}(P) = \frac{C_g}{C_w} \quad \text{Eq. 17.9}$$

Decay and diffusion can be modeled in all phases with phase specific decay and diffusion constants. For adsorption, the concentration to use when evaluating the adsorption function can be specified in the sixth item of the TRACER keyword. The options are to use either one of the concentrations in the phases that the tracer partitions into, or to use the overall, pore-volume-averaged, concentration defined by:

$$C = \sum_j \frac{S_j}{B_j} C_j \quad \text{Eq. 17.10}$$

Initialization of partitioned tracers

The initial partitioned tracer concentrations are input in tabular form using either of the keywords [TBLK](#) or [TVDP](#). The keyword TBLK specifies the initial concentrations of a tracer in each grid block, whereas the keyword TVDP specifies tables of initial tracer concentration versus depth. Only the tracer concentration for the free (reference) phase is specified in the SOLUTION section. The initial free phase tracer concentrations are then updated using the formula

$$C_f = \frac{C_f^{ini} FIP_f}{\sum_j K_{jf}(P) FIP_j} \quad \text{Eq. 17.11}$$

where

FIP_j is the fluid in place for phase j and

$K = 1$ for $j = f$.

The concentration of the solution phase(s) is then calculated from the relationship $C_j / C_f = K_{jf}(P)$ which has been input using the [TRACERKP](#) or [TRACERKM](#) keyword. In this way the concentration specified in the SOLUTION section is distributed between the phases whilst honoring the partitioning functions.

Injection and production

The tracer concentration in the free and solution phase in the wellbore is calculated depending on whether the well is of production or injection type. If the well is injecting, the concentration of the free phase is taken from the keyword [WTRACER](#) and the concentration of the solution phase(s) is then calculated using the relationship $C_j / C_f = K_{jf}(P)$, where P is taken to be the pressure in the wellbore. If the well is producing, the concentration in the solution and free phases are calculated by summing the concentrations in each of the well connections. In this case the connection concentrations of the free and solution phases are taken to be the grid block values.

Note: It is possible to combine the partitioned tracer functionality with the Surfactant Model, thus allowing surfactant to partition into more than one phase. This is done by specifying a tracer called ‘SUR’ which ECLIPSE recognizes as being the surfactant. For a surfactant, adsorption is calculated based on the water phase concentration, that is the ‘free’ phase.

Using the environmental tracer option

A passive tracer is set up in the normal manner using the following ECLIPSE 100 keywords to specify the tracer data:

Section	ECLIPSE 100 only keyword	Description
RUNSPEC	TRACERS	Specify the number of tracers in each phase (NOTRAC, NWTRAC, NGTRAC) and the number of environmental tracers (NETRAC).

Section	ECLIPSE 100 only keyword	Description
PROPS	TRACER	Specify the host phase of each tracer.
REGION	TNUM	Specify the region numbers to be used for initializing the concentration of tracer in each grid block and specify link cells to tables of initial tracer concentration with respect to depth.
SOLUTION	TBLK or TVDP	Specify the initial tracer concentration as either the numerical value or as tables of depth versus tracer concentration.
SCHEDULE	WTRACER	Inject tracer through wells.
SCHEDULE	RIVTRACE	Inject tracer in rivers.

Table 17.1: Keywords used to specify environmental tracer data

To activate the environmental tracer option, the parameter NETRAC in keyword TRACERS in the RUNSPEC section should be set greater than zero. NETRAC should be set equal to (or greater than) the number of tracers within the model that either adsorb, decay or diffuse; it need not be set equal to the total number of tracers in the model, if not all of them adsorb, decay or diffuse.

To activate the partitioned tracer option the keyword [PARTTRAC](#) must be entered in RUNSPEC section.

The adsorption, decay and diffusion data is supplied with the keywords [TRADS](#), [TRDCY](#) and [TRDIF](#) respectively. These keywords require the tracer name to be appended to the keyword to indicate which tracer the data applies to. For adsorption, the [ADSORP](#) keyword may be used, with the tracer name specified as an argument, as an alternative to TRADS. The ADSORP keyword supplies coefficients for a generalized Langmuir isotherm with permeability dependence. For standard partitioned tracers, the data entered in TRDIF and TRDCY apply to diffusion and decay in the solution phase. For multi-partitioned tracers, data specific to each phase can be entered in TRDIF and TRDCY.

If the adsorption model is required, the rock mass density needs to be specified by using the [TRROCK](#) keyword. The grid block masses can be output by using the 'ROCKMASS' mnemonic in the [RPTTSOL](#) keyword. The TRROCK keyword is also used to specify the adsorption model and the initial distribution of adsorbed tracer. For multi-partitioned tracers the concentration to be used in adsorption calculations can be specified by the sixth item of the TRACER keyword.

A passive tracer can be thought of as a dimensionless marker of the host fluid. However, in the environmental tracer case it may be convenient to ascribe a unit to the tracer. A general unit name can be given to a tracer using the third item in the TRACER keyword. Note that the adsorption data (TRADS or ADSORP keyword) should be based on this unit.

The diffusivity values can be output by setting mnemonics 'DIFFX', 'DIFFY' and 'DIFFZ' in the [RPTGRID](#) keyword. Note that the diffusivity associated with non-neighbor connections can be found in the non-neighbor connection list generated by the RPTGRID mnemonic 'ALLNNC'.

The tables of tracer adsorption functions, tracer decay and tracer rock properties can be output to the print file by setting the mnemonics 'TRADS', 'TRDCY' and 'TRROCK' in the [RPTPROPS](#) keyword.

The quantity of the tracer that has decayed or been adsorbed can be output to the print file by setting either of the mnemonics 'TRADS' or 'TRDCY' in the [RPTSCHED](#) keyword. Two families of SUMMARY keywords are available to output the adsorption and decay: FTADS, RTADS, BTADS and FTDCY, RTDCY, BTDCY. As with other tracer SUMMARY section keywords, the tracer name should be appended on to the keyword.

Tracer in place reports can be output to the print file by specifying the 'FIPTR' mnemonic in RPTSCHED. For a standard partitioned tracer the amount of the tracer in the free and solution phases will be reported. For a multi-partitioned tracer the amount reported in the 'solution' phase is the sum over all solution phases.

Restrictions

At present the adsorption, decay and diffusion options are only available for single phase and partitioned tracers. It is not possible to model adsorption, decay or diffusion for tracers existing in a phase that can dissolve in another phase. For example, it is not possible to model a decaying tracer in the gas phase of a model in which the gas can dissolve in the oil.

Single phase environmental tracers employing adsorption and decay can use either the standard tracer solver method or the “cascade” or TVD algorithms, which limit numerical diffusion (for the latter two, specify 'DIFF' in item 5 of the RUNSPEC section keyword [TRACERS](#) and for the TVD algorithm use the [TRACTVD](#) keyword). However, if a tracer uses the diffusion model it cannot use the cascade algorithm. Hence, even if 'DIFF' is set in TRACERS, the default solution method is employed for any tracers with the diffusion option active. TRACTVD can be used with diffusion (and dispersion), but not when LGRs are present.

All partitioned tracers are solved using the standard tracer solver.

If the adsorption option is active, the tracer equations are non-linear. Because of this non-linearity, it is likely that the cascade method is faster than the default method, providing, of course the tracer does not employ the diffusion option as well.

Environmental tracers cannot be introduced in a restart run.

If a cell is defined with zero pore volume in the GRID section, then making it active by changing the pore volume in the EDIT section using [PORV](#), is not allowed with the environmental tracer option.

Summary of keywords

RUNSPEC section

Keyword	Description
TRACERS	Specify dimensions and options for tracers.
EQLDIMS	Specify dimensions for tracer concentration versus depth tables used for equilibration.
PARTTRAC	Enables and dimensions the partitioned tracer option.

PROPS section

Keyword	Description
RPTPROPS	Controls output from the PROPS section: TRACER outputs passive tracer names. TRADSxxx, TRROCK and TRDCYxxx output environmental tracer properties.
TRACER	Define the tracers.
TRACERKP	Define the partitioning functions K(P) for partitioned tracers (max two phases).
TRACERKM	Define the partitioning functions K(P) for partitioned tracers (any no. of phases).
TRADS	Specify adsorption data (tabulated isotherm).
ADSORP	Specify adsorption data (coefficients for analytical isotherm).
TRDCY	Specify decay data.

Keyword	Description
TRDIF	Specify diffusion data.
TRROCK	Specify rock data for adsorption.

REGIONS section

Keyword	Description
RPTREGS	Controls output from the REGIONS section: TNUM outputs tracer versus depth region numbers.
TNUM	Specify tracer concentration regions.
TRKPFxxx	Specify partitioning function regions for tracer xxx.

SOLUTION section

Keyword	Description
RPTSOL	Controls output from the SOLUTION section: FIPTR outputs tracer fluid in place reports ROCKMASS outputs rock mass TBLK outputs initial grid block tracer concentrations TVDP outputs initial tracer concentration versus depth tables.
TBLK	Specify initial tracer concentrations.
TVDP	Specify initial tracer concentration versus depth.

SUMMARY section

Refer to [Environmental tracer SUMMARY output controls](#) in the *ECLIPSE Reference Manual*.

SCHEDULE section

Keyword	Description
RPTSCHE	Controls output from the SCHEDULE section: FIPTR outputs tracer fluid in place reports TBLK outputs initial grid block tracer concentrations TRADS or TRDCY output tracer adsorption and decay.
WTRACER	Set tracer concentration in injection wells.
RIVTRACE	Set tracer concentration for injection into rivers.

Example problem

The tracer names are:

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 section

```
GRID =====
EQUALS
'DX'    10    /
'DY'    100   /
'DZ'    10    /
'PERMX' 1000  /
'PERMY' 1000  /
'PORO'  0.25  /
'TOPS'   2000  1 10  /
'TOPS'   2005  11 20  /
'TOPS'   2010  21 30  /
'TOPS'   2015  31 40  /
'TOPS'   2020  41 50  /
'TOPS'   2025  51 60  /
'TOPS'   2030  61 70  /
'TOPS'   2035  71 80  /
'TOPS'   2040  81 90  /
'TOPS'   2055  91 100 /

-- Zero permeability for tracer ZZZ
--
'PERMX' 0.0  41 50  2 2  1 1  /
'PERMY' 0.0  41 50  2 2  1 1  /
/
```

PROPS

```

PROPS      =====
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  

--  

TRDCYYY  

15.0  /  

--  

-- Tracer diffusion data for tracer ZZZ  

--  

TRDIFZZZ  

0.3   /  

--  

-- Rock data required for tracer adsorption  

--  

TRROCK  

1 2650  /  

2 2650  /  

--  

-- Output of Environmental 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
FTDCYYY

RTADSYYY
2 /

BTADSYYY
2 1 1 /
/

BTDCYYY
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'  'SURFBULK'  '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
```

18

Equations of state

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

Two-parameter equations of state

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. 71] and [Ref. 25].

The generalized form of such an equation of state is:

$$Z^3 + E_2 Z^2 + E_1 Z + E_0 = 0 \quad \text{Eq. 18.1}$$

with:

$$E_2 = (m_1 + m_2 - 1)B - 1 \quad \text{Eq. 18.2}$$

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

$$E_0 = -[AB + m_1 m_2 B^2(B + 1)] \quad \text{Eq. 18.3}$$

The coefficients m_1 and m_2 depend upon the equation used. Refer to table 18.1 for details.

Equation of State	m1	m2
Redlich-Kwong	0	1

Equation of State	m1	m2
Soave-Redlich-Kwong	0	1
Zudkevitch-Joffe	0	1
Peng-Robinson	$1 + \sqrt{2}$	$1 - \sqrt{2}$

Table 18.1: The dependence of m1 and m2 on the equation of state

The cubic equation for the Z-factor may be solved to obtain Z-factors for the liquid and vapor phases. In general, three solutions are obtained.

The distinction between the liquid and the vapor phase is made by choosing the smallest root as the Z-factor for the liquid phase, and the largest root as the Z-factor for the vapor phase.

Fugacity coefficients are calculated using:

$$\ln(f_i/(px_i)) = -\ln(Z-B) + \frac{A}{(m_1-m_2)B} \left(\frac{2\Sigma_i}{A} - \frac{B_i}{B} \right) \ln\left(\frac{Z+m_2B}{Z+m_1B}\right) + \frac{B_i}{B}(Z-1) \quad \text{Eq. 18.4}$$

where:

$$\Sigma_i = \sum_j A_{ij} x_j \quad \text{Eq. 18.5}$$

$$A = \sum_{j=1}^n \sum_{k=1}^n x_j x_k A_{jk} \quad \text{Eq. 18.6}$$

$$B = \sum_{j=1}^n x_j B_j \quad \text{Eq. 18.7}$$

$$A_{jk} = (1-\delta_{jk})(A_j A_k)^{\frac{1}{2}} \quad \text{Eq. 18.8}$$

and δ_{jk} are binary interaction coefficients, normally between hydrocarbons and non-hydrocarbons.

These four equations express the mixing laws used in all the equations of state.

The variables are defined by the following equations:

$$A_j = \Omega_a(T, j) \frac{P_{rj}}{T_{rj}^2} \quad \text{Eq. 18.9}$$

$$B_j = \Omega_b(T, j) \frac{P_{rj}}{T_{rj}} \quad \text{Eq. 18.10}$$

$\Omega_a(T, j)$ and $\Omega_b(T, j)$ in equations 18.9 and 18.10 respectively are functions of the acentric factor w_j and the reduced temperature T_{rj} .

For Redlich-Kwong:

$$\Omega_a(T, j) = \Omega_{a_o} T_{rj}^{-1/2}$$

$$\Omega_b(T, j) = \Omega_{b_o}$$

For Soave-Redlich-Kwong:

$$\Omega_a(T, j) = \Omega_{a_0} [1 + (0.48 + 1.574w_j - 0.176w_j^2)(1 - T_{rj}^{1/2})]^2$$

$$\Omega_b(T, j) = \Omega_{b_0}$$

For Zudkevitch-Joffe:

$$\Omega_a(T, j) = \Omega_{a_0} F_{aj}(T) T_{rj}^{-1/2}$$

$$\Omega_b(T, j) = \Omega_{b_0} F_{bj}(T)$$

For Peng-Robinson:

$$\Omega_a(T, j) = \Omega_{a_0} [1 + (0.37464 + 1.54226w_j - 0.26992w_j^2)(1 - T_{rj}^{1/2})]^2$$

$$\Omega_b(T, j) = \Omega_{b_0}$$

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,

$$\Omega_a(T, j) = A + B \cdot T^{1/2} + CT \quad \text{Eq. 18.11}$$

This expansion is used if [TEMPVD](#) is specified, so that each cell and well has its own temperature.

The normal PR form can be optionally modified for a large acentric factor, using the factor $(0.379642 + 1.48503w_j - 0.164423w_j^2 + 0.016666w_j^3)$ rather than $(0.37464 + 1.53226w_j - 0.2699w_j^2)$ for $w_j > 0.49$. This correction is invoked by use of the [PRCORR](#) keyword.

Ω_{a_0} and Ω_{b_0} are constants depending upon the equation of state, as shown in table 18.2.

Equation of state	Ω_{a_0}	Ω_{b_0}
RK, SRK, ZJ	0.4274802	0.08664035
PR	0.457235529	0.07796074

Table 18.2: The dependence of PR constants on the equation of state

These default values may be overwritten if required, by using the [OMEGA_A](#) and [OMEGA_B](#) 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.

ECLIPSE 300 obtains the required variations of component saturation pressure and liquid density with temperature by using the correlations of Reidel, and Gunn and Yamada. These correlations require the input of the normal boiling point and the reference liquid density at a specified reference temperature.

Flash calculations

For a thermodynamic system to be in equilibrium, it is further required that the fugacities in the liquid and vapor phases must be equal for each component:

$$f_{iL} = f_{iV} \quad \text{Eq. 18.12}$$

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

$$f_i = f_i(T, p, x_i) \quad \text{Eq. 18.13}$$

The fugacities can be calculated directly from an equation of state.

Equilibrium constants (otherwise known as K-values) for each component can be defined as:

$$K_i = \frac{y_i}{x_i} \quad \text{Eq. 18.14}$$

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

$$x_i = \frac{z_i}{1 + (K_i - 1)V} \quad \text{Eq. 18.15}$$

$$y_i = \frac{K_i z_i}{1 + (K_i - 1)V} \quad \text{Eq. 18.16}$$

If K-values are used to specify inter-phase equilibria, these two equations are used to specify the liquid and vapor compositions directly, with K-values obtained by table look-up as functions of pressure.

Viscosity evaluation

In compositional mode the phase viscosity values are obtained using either the Lorentz-Bray-Clark method or the Pedersen et al. method.

The Lorentz-Bray-Clark method

By default the Lorentz-Bray-Clark method is used [Ref. 70]. The viscosity μ for each phase is given by

$$[(\mu - \mu^\circ) \cdot \xi + 0.0001]^{1/4} = \sum_{i=1}^5 a_i \cdot b_r^{i-1} \quad \text{Eq. 18.17}$$

μ° and ξ are functions of the composition x_i , the component molecular weights, critical pressures and critical temperatures.

The coefficients a_i are set equal to the values given in. [Ref. 70]. These values can be modified by using the **LBCCOEF** keyword.

The reduced molar densities b_r are given by

$$b_r = \frac{b_p}{b_c} \quad \text{Eq. 18.18}$$

where the phase volumes b_p are calculated from the pressure and compressibility (and volume shift if used), and the critical molar density b_c for each phase. The critical molar density is defined by:

$$\frac{1}{b_c} = V_c = \sum x_i \cdot V_{c,i} \quad \text{Eq. 18.19}$$

The critical molar volumes for each component $V_{c,i}$ can be specified directly by either of the keywords **VCRIT** or **VCRITVIS**, or can be calculated from critical z-factors entered by the keywords **ZCRIT** or **ZCRITVIS**.

The modified Lorentz-Bray-Clark method

A modified form of the Lorentz-Bray-Clark method can also be used:

$$[(\mu - \mu^\circ) \cdot \xi + 0.0001] = \left(\sum_{i=1}^5 a_i \cdot b_r^{i-1} \right)^4 \cdot e^{\left(\frac{b_r}{b_{max,r} - b_r} \right)} \quad \text{Eq. 18.20}$$

This modified equation is intended to improve the modeling of the behavior of heavy oils. It was requested by a client and has not been published.

In this equation the viscosity increases exponentially as the reduced molar density b_r approaches a maximum $b_{max,r}$. For small values of b_r , such that $b_r \ll b_{max,r}$, the modified Lorentz-Bray-Clark equation has the same form as the original equation, Equation 18.17. The maximum reduced molar density can be expressed in terms of a maximum molar density b_{max}

$$b_{max,r} = \frac{b_{max}}{b_c}$$

which in turn can be expressed in terms of the minimum component volumes $V_{min,i}$

$$\frac{1}{b_{max}} = V_{min} = \sum x_i \cdot V_{min,i} \quad \text{Eq. 18.21}$$

The minimum component volumes $V_{min,i}$ are entered using the keyword **VCOMPACT**. If the **VCOMPACT** keyword is found the modified Lorentz-Bray-Clark method will be used instead of the Lorentz-Bray-Clark method.

When entering $V_{min,i}$ data it is important to ensure that the actual molar density never exceeds the maximum, so that

$$b_p < b_{max} \quad \text{Eq. 18.22}$$

If the molar densities do exceed this value then a large constant value is used for the viscosity.

A simplified version of the modified Lorentz-Bray-Clark method can be used if the minimum component volumes $V_{min,i}$ are set proportional to the critical component volumes

$$V_{min,i} = \frac{1}{b_0} \cdot V_{c,i} \quad \text{Eq. 18.23}$$

In this case the maximum reduced molar density in equation 18.20 will be a constant:

$$b_{max,r} = b_0 \quad \text{Eq. 18.24}$$

Example

In this example of the simplified version of the modified Lorentz-Bray-Clark method, we wish to set the reduced molar density $b_{max,r}$ to a constant value of 4.776. Given values of VCRIT for each of the three components, the three VCOMPACT values are obtained by dividing each VCRIT value by 4.776.

VCRIT	99.20	93.90	148.30	/
VCOMPACT	20.77	19.66	31.05	/

The Pedersen method

If the Pedersen *et al.* viscosity calculation, [Ref. 80], is required instead of the default method, then the **PEDERSEN** keyword should be added to the PROPS section. The calculation is based on a modification of the corresponding states principle in which the mixture viscosity μ is related to a reference viscosity μ_o by terms involving properties of the mixture (denoted with a subscript m) and the reference material (denoted with a subscript o). The properties are the critical temperature T_c , critical pressure P_c , molecular weight M_w and rotational coupling factor α :

$$\mu(P, T) = \left(\frac{T_{cm}}{T_{co}} \right)^{-1/6} \cdot \left(\frac{P_{cm}}{P_{co}} \right)^{2/3} \cdot \left(\frac{M_{wm}}{M_{wo}} \right)^{1/2} \cdot \left(\frac{\alpha_m}{\alpha_o} \right) \cdot \mu_o(P_o, T_o)$$

The reference material is taken to be methane. Pedersen [Ref. 80] gives formulas for the mixture properties, and the reference pressure P_o and reference temperature T_o . The mixture molecular weight is given by:

$$M_{wm} = c_1 \cdot 1.304 \times 10^{-4} \cdot (\bar{M}_w^{c_2 \cdot 2.303} - \bar{M}_n^{c_2 \cdot 2.303}) + \bar{M}_n$$

where \bar{M}_w and \bar{M}_n are the weight average and number average mole weights. The rotational coupling factors are given in terms of the reduced density ρ_{rm} by:

$$\alpha_m = 1 + c_5 \cdot \rho_{rm}^{c_3} \cdot M_{wm}^{c_4}$$

$$\alpha_o = 1 + c_6 \cdot \rho_{rm}^{c_3}$$

The coefficients c_1, \dots, c_6 can be defined by the **PEDTUNE** and **PEDTUNER** keywords. Note that the default value of c_6 is related to the default values of c_4 and c_5 by

$$c_6 = c_5 \cdot M_{wo}^{c_4}$$

Three parameter equations of state

The traditional weakness of the two-parameter equations of state (EoS), such as the Peng-Robinson, Redlich-Kwong for example, 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:

$$V_{\text{mol},p} = V_{\text{mol},p}^{\text{EoS}} - \sum_{i=1}^N z_i c_i \quad \text{Eq. 18.25}$$

where

p = (liquid, vapor) represents the phase of the system

$V_{\text{mol},p}^{\text{EoS}}$ 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 a set of dimensionless shift parameters, s_i , by:

$$s_i = \frac{c_i}{b_i} \quad \text{Eq. 18.26}$$

where:

$$b_i = \frac{\Omega_{b,i} RT_{ci}}{p_{ci}} \quad \text{Eq. 18.27}$$

The shift parameters are entered in the dimensionless form using the keyword `SSHIFT`.

Use of the equation of state for hydrocarbon mixtures

A typical oil is composed of many millions of components. The equation of state has one value of A and one value of B to model the phase behavior of this mixture.

For example, the PR EoS is defined as follows:

$$P = \frac{RT}{V-B} - \frac{A}{V(V+B)+B(V-B)} \quad \text{Eq. 18.28}$$

In a typical reservoir simulation model, 4-8 components and pseudo-components are used to define the mixture. Each component has a value of T_c , P_c , V_c or Z_c , ω , Ω_a , Ω_b and binary interaction coefficients, δ_{ij} (that increase or decrease the forces between component pairs), that must be specified in the fluid characterization process.

These parameters are for pure components, such as Methane or Nitrogen, and are usually considered fixed. Thus, you must alter the remaining component parameters so that the resulting calculated phase behavior matches the experimental data. This phase behavior matching process is carried out with the PVT *i* program.

Calculation process

This section outlines the calculation process used by ECLIPSE 300 to determine phase compositions, and the amount of liquid and vapor present in a grid block, at equilibrium.

The ECLIPSE 300 calculation process utilizes linear, non-linear (Newton) and flash iterations to obtain a converged solution. 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.

The remainder of this section describes the iteration process used to flash the mixture in each grid block, to determine the molar volumes and the phase compositions 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. These values can be estimated from the values used in the previous iteration, using equation 18.37, or, when no previous values exist, from Wilson's formula:

$$K_i = \frac{e^{\left[5.37(1+\omega_i)\left(1-\frac{1}{T_{ri}}\right)\right]}}{P_{ri}} \quad \text{Eq. 18.29}$$

Then, given z_i and K_i , ECLIPSE 300 solves the Flash Equation to obtain the molar fraction of vapor, V.

The flash equation solved by ECLIPSE 300 is:

$$g(V) = \sum_{i=1}^{n_c} \frac{z_i(K_i - 1)}{1 + V(K_i - 1)} = 0 \quad \text{Eq. 18.30}$$

ECLIPSE 300 solves this equation to obtain the mole fractions of each component in the liquid and vapor phases:

$$x_i = z_i / [1 + V(K_i - 1)], \quad i = 1, 2, \dots, n_c \quad \text{Eq. 18.31}$$

$$y_i = (K_i z_i) / [1 + V(K_i - 1)], \quad i = 1, 2, \dots, n_c \quad \text{Eq. 18.32}$$

Using these mole fractions, the Peng-Robinson equation of state can then be expressed as a cubic equation of the compressibility factor, Z:

$$Z^3 - Z^2(1-b) + Z(a-3b^2 - 2b) - (ab - b^2 - b^3) = 0 \quad \text{Eq. 18.33}$$

where

Z is defined by $Z = \frac{PV}{RT}$ and the two remaining parameters are defined by:

$$a = \frac{AP}{R^2 T^2} \quad \text{Eq. 18.34}$$

$$b = \frac{BP}{RT} \quad \text{Eq. 18.35}$$

The solution of this 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 this 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 used to determine if a grid block is in equilibrium (if ε is very small),

$$\sum_{i=1}^{n_c} \left(\frac{f_{iL}}{f_{iV}} - 1 \right)^2 < \varepsilon \quad \text{Eq. 18.36}$$

If equilibrium is not achieved, then new estimates 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:

$$K_i^{NEW} = K_i^{OLD} \cdot \left[\frac{f_{iL}}{f_{iV}} \right] \quad \text{Eq. 18.37}$$

After equilibrium is achieved, ECLIPSE 300 begins the next non-linear iteration.

Calculation of phase states

A cell in a compositional simulation may be either water-filled, or in a single-phase or two-phase hydrocarbon state. The single-phase states may possess a saturation pressure, usually lower than the current pressure, at which a second hydrocarbon phase appears. For temperatures above the cricondentherm however, such a pressure does not exist, and the cell composition leads to a single-phase hydrocarbon at all pressures.

The appearance of a two-phase state is detected by application of the Michelsen stability test [Ref. 74]. 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. 32]. This is a minimization method which uses an approximate Hessian that is constrained to be symmetric and positive definite. In the early stages of the iteration an accelerated successive substitution method, which uses a two-point GDEM technique [Ref. 29], is effective.

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 and accuracy of the method for a particular data set.

19

Equations of state regions

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

It is possible to set up more than one equation of state description, and associate these with sets of cells using the [EOSNUM](#) keyword in the REGIONS section. The most common application of this is a study containing two or more disconnected reservoirs, coupled through the production system.

Define NR as the number of reservoir equations of state, and NS as the number of surface equations of state. These are specified in [TABDIMS](#), and both default to 1. If a value is entered for NR, this becomes the default for NS. There is a limit of 10 on both NR and NS.

Reservoir equation of state

For each reservoir equation of state (EoS), ECLIPSE 300 expects information corresponding to the following keywords:

Restriction	Keyword
	RTEMP or TEMPVD
	EOS
	TCRIT
	PCRIT
	VCRIT or ZCRIT
optional - default VCRIT or ZCRIT	VCRITVIS or ZCRITVIS
	MW
	ACF
	OMEGAA
	OMEGAB
optional - default 0	SSHIFT
ZJ equation only	TBOIL
ZJ equation or Thermal	TREF
ZJ equation or Thermal	DREF or GREF
Thermal only	CREF

Restriction	Keyword
Thermal only	PREF
Thermal only	THERMEX1
optional - miscible only	PARACHOR
	BIC
Thermal only	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

This information 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 must have the same names. The most common practice is to alter the properties of the heavy components when, for example, performing regressions to a number of reservoir fluids.

Having defined the EoS properties to be associated with the EoS regions, they are associated with cells using [EOSNUM](#), which acts like [SATNUM](#). For example, in the case of a 35 x 14 x 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, there are the same number of surface equations of state as there are 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 overwritten using the following keywords:

Restriction	Keyword
	EOSS
	TCRITS
	PCRITS
	VCRITS or ZCRITS
	EOSS
	MWS
	ACFS
	OMEGAAS
	OMEGABS
optional - default 0	SSHIFTS
ZJ equation only	TBOILS
ZJ equation or Thermal	TREFS
ZJ equation or Thermal	DREFS or GREFS
Thermal only	PREFS
	BICS
parachors and viscosity values are not employed in surface calculations	RTEMP

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 (NRth) 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 is 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 the 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](#).

20

File handling in ECLIPSE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

File	Purpose	Contents
PRINT file	Main printer output	RPT... keywords
DEBUG file	Debug output	DEBUG keyword
MSG file	Message output (XML format)	MSGFILE keyword
GRID file	Grid geometry file	NOGFF and GRIDFILE keyword)
SAVE file	Non-recurrent data for fast restarts	SAVE keyword)
INITIAL file	Initialization file for GRAF	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	WRFTPPLT keyword
RSSPEC file	Restart index file	RSSPEC keyword in the RUNSPEC section
EDESC file	Report step communication file for VISAGE external geomechanics workflow.	
VDESC file	Stress step communication file for the VISAGE external geomechanics workflow.	
HMD file	Gradients file for SimOpt	HMDIMS in the RUNSPEC section). This keyword is used in the ECLIPSE 100 gradient option.

File	Purpose	Contents
GRFT file	RFT gradients for SimOpt	GRADRFT in the SCHEDULE section). This keyword is used in the ECLIPSE 100 gradient option.

Table 20.1: Files produced by ECLIPSE for a simulation run

The production of all of these files, except for the main printer output and VDESC files, may be controlled by the user with the keywords indicated.

The EDESC and VDESC files are produced by runs which use the VISAGE external geomechanics workflow. This workflow is activated by the [VISAGE](#) keyword in the RUNSPEC section. These small formatted files contain information on either all report dates (in the case of EDESC), or just the report dates at which stress step output has been generated (in the case of VDESC). It is intended to be processed by the VISAGE Coupler application which controls the ECLIPSE-VISAGE interaction.

The contents, and hence the size, of the printer output file are controlled using the [RPTGRID](#), [RPTPROPS](#), [RPTREGS](#), [RPTSCHED](#), [RPTSMRY](#), [RPTSOL](#) and [RPTRST](#) keywords.

In addition, you can choose whether input and output files are to be formatted or unformatted, (see the RUNSPEC section keywords [FMTIN](#) and [FMTOUT](#)), and whether RESTART and SUMMARY files are to be unified (see the RUNSPEC section keywords [UNIFIN](#) and [UNIFOUT](#)). You should be aware of which types of file it is appropriate to use on your system. Some guidelines are given here.

Formatted files are standard 80 character card image files, which may be straightforwardly transferred between different computers. All the files generated by the ECLIPSE programs can be converted from formatted to unformatted files or vice-versa using the utility program Convert.

In general, unformatted non-unified files should be used, giving efficient use of disk space and allowing you to examine output as the simulation progresses. However, it may be necessary to use formatted files if the simulation is performed on one type of computer but restarts or graphics 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: 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 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 timestep. 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 NOGFF 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 and saturations for example) for all the cells at each report step.
ECLIPSE 100	The file content and report frequency may be controlled with the keyword RPTRST .
ECLIPSE 300	The solutions to be written must be chosen using the RPTRST keyword in the SOLUTION or SCHEDULE sections; otherwise the file is empty.

21

Flux boundary conditions

x	ECLIPSE 100
x	ECLIPSE 300
	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

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 NMFIPIR 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:

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

Several flux regions may be defined and all relevant flows stored in 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 timestep in ECLIPSE 100. The oil, water, gas, voidage, wet gas and component well rates are written to the flux file at each timestep 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 timestep 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 item 25 in 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 item 25 is set > 1, then location of the main production well within each drainage region is identified by a negative region number. [FLUXNUM](#) treats negative numbers as if they were positive.

Aquifers

Any aquifer data in the reduced run is processed in the usual way. Numerical aquifers declared outside the [USEFLUX](#) region are ignored in the reduced run (the aquifer support is accounted for by the flows across the flux boundary generated by the [DUMPFLUX](#) run). When determining aquifer connections in the reduced run, no extra connections are generated across the flux boundary when the flux boundary is enclosed within the box defined in [AQUCON](#), [AQUFET](#) or [AQUANCON](#). Hence no new aquifer connections are made in the reduced run that were not in the base run, and support from aquifer connections outside the reduced flux region is accounted for by fluxes in the reduced run.

Restart runs

The reduced run can use flux information from a full field run that consists of a sequence of restarts. The [USEFLUX](#) keyword in the reduced run should specify the root name of the last ECLIPSE run in the full field restart sequence. The FLUX files from all the restarts must exist and either be all formatted or all unformatted. Keyword [FMTIN](#) should be specified in the RUNSPEC section if the files are formatted.

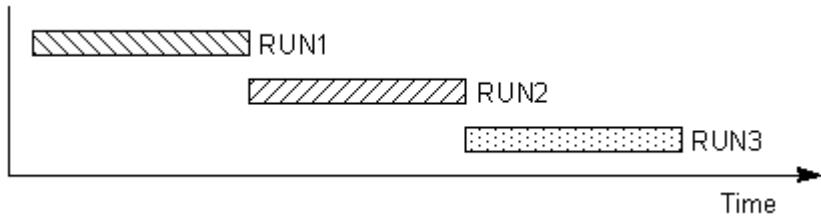


Figure 21.1. A sequence of restarts that together make a full field run

The reduced run `SMALL.DATA` includes the `USEFLUX` keyword as follows:

```
.....  
USEFLUX  
'RUN3'  
/.....
```

This run uses files `RUN1.FLUX`, `RUN2.FLUX` and `RUN3.FLUX` to read flux information for the time sequence represented by *'s. The number of restarts in the sequence is limited to 20.

It is possible to restart a reduced run from the corresponding full field run by specifying the full field root name in the `RESTART` keyword in the usual manner.

Note: The periods covered by the restarts in the sequence should not overlap.

ECLIPSE 300

Note: When doing a reduced run from a restart, the `GRID` file from the restart run must also be present in order for the simulation to initialize correctly.

Pressure boundary conditions (ECLIPSE 100)

Instead of using the flows of each phase from the full field run as boundary conditions on the reduced run, an alternative treatment using pressures and saturations is available (see the `FLUXTYPE` keyword). The pressures, saturations, R_s and R_v values in the ‘halo’ cells surrounding each flux region are stored in the `FLUX` file from the full field run. In the reduced run, ECLIPSE 100 calculates the flows across the flux boundary using the pressures and saturations in the halo cells surrounding the reduced flux region.

When the total flux across the boundary of the reduced region does not alter appreciably, the default flow boundary condition is adequate. This is the case, for example, when the flux option is used to examine the effect of changing the well data in the reduced region when the flux boundary is wholly an inflow boundary or wholly an outflow boundary and the distribution of fluids on the flux boundary does not alter appreciably in the reduced run. When more severe changes are made to the flow regime and fluid distribution, the pressure boundary condition should be used to allow the phases to redistribute 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 and coarsening

ECLIPSE 100

Within ECLIPSE 100, refined grids and coarsenings must not be placed at flux boundaries in a [DUMPFLUX](#) run.

ECLIPSE 100

In a reduced flux [USEFLUX](#) run, local grids and coarsenings 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 or coarsened cells and the flux boundary.

ECLIPSE 300

In a reduced flux [USEFLUX](#) run, local grids cannot straddle or be placed at boundaries of the reduced flux region (defined by [FLUXREG](#)). However, they can be placed within flux regions provided that there is at least one cell between the refined cells and the flux boundary. Coarsenings cannot straddle boundaries, but may be placed at them.

Refined grids or coarsenings 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 or coarsenings 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. When running reduced flux [USEFLUX](#) runs in parallel, amalgamations may not be used outside of the [USEFLUX](#) region.

Reading ECLIPSE 100 flux files into ECLIPSE 300

Flux runs are usually run by ECLIPSE 300 and use a [FLUX](#) file generated by ECLIPSE 300. It is possible to read an ECLIPSE 100 [FLUX](#) file. An ECLIPSE 100 [FLUX](#) file contains flows in terms of stock tank oil and gas volumes, and you must then also enter [XSTC](#) and [YSTC](#), the stock tank oil and gas compositions, in order that the program can generate component flows.

It is not possible to use an ECLIPSE 100 [FLUX](#) file to run a thermal reduced data.

Thermal option

ECLIPSE 300

The flux boundary option can be used for 2 different scenarios:

1. Full field model is a Thermal model, and the reduced model is also a Thermal model
2. Full field model is either an ECLIPSE 300 Blackoil model or a Compositional model, and the reduced run is a Thermal model.

For option 1, a thermal reduced run can be carried out from a thermal base run. In this case, the energy flows across the boundary are written out to the [FLUX](#) file in the full field run.

For option 2, a thermal reduced run can be carried out from a compositional or an ECLIPSE 300 blackoil base case. In this case the energy flows across the boundary are calculated in the reduced run.

The conductive term is defined as $C_{ei} = -\sum_n \Psi_{ni} \cdot A \cdot (T_i - T_n)$,

where

Ψ_{ni} is the heat transmissibility between cells n and i . This term is calculated as follows:

$$\Psi_{ni} = \alpha \times Y_{ni}$$

where γ_{ni} is the transmissibility between cells n and i taken from the base run, and

α is the heat transmissibility factor, defined using the [THFLXTF](#) keyword.

The Λ term is the saturation dependent transmissibility multiplier of the cell inside the active flux region.

The convective term is defined as $F_{ei} = \sum_n \gamma_{ni} \sum_p M_p^e \cdot dP_{pni}$,

where

M_p^e is the energy generalized mobility, and is calculated as follows:

$$M_p^e = H_p \cdot k_{rp}(S_p) \cdot \frac{b_p}{\mu_p}.$$

Referring to the above form, the $k_{rp}(S_p) \cdot \frac{b_p}{\mu_p}$ part is taken from the base run, and the phase enthalpy H_p is calculated in the cell inside the active flux region during the reduced run.

To enable a thermal reduced run from a non-thermal base run, the [FLUXOPTS](#) keyword should be used in the base run and set to YES. This will allow ECLIPSE 300 to provide all the essential information to perform the reduced thermal simulation.

Current limitations

- [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, surfactant, tracers, API and brine tracking without polymer, the solvent model and other features involving additional conservation equations. The keywords [POLYMER,BRINE](#), [SURFACTANT](#), [SOLVENT](#), [ALKALINE](#), or [TRACER](#) have been banned in normal usage. The ban can be lifted by using item 232 in OPTIONS. In this case the user is required to ensure that the effects of these keywords are local to the USEFLUX region; no checking of this will be done by the simulator.
- ECLIPSE 100*
- The pressure boundary conditions do account for solvent, API and brine tracking and the relevant keywords for these noted above are not normally banned for such boundary conditions.
- Gas Field Operations*
- The flux boundary option should not be used in conjunction with automatic DCQ reduction in the gas field operations model.
- Parallel Option*
- Using DUMPFLUX, the WARP linear solver cannot be used, instead the pre-2002A linear solver is activated.
 - The nine-point schemes ([NINEPOIN](#) keyword) cannot be used when the flux boundary option is used.
- ECLIPSE 300*
- The multi-component water model ([COMPW](#)) cannot be used when the flux boundary option is used.

Summary of keywords

RUNSPEC

- [REGDIMS](#) Item 4 specifies the maximum number of flux regions in the run.

GRID

Restriction	Keyword	Description
	ADDREG	Adds a constant to the specified array in a flux region.
	COPY	Copies data from one array to another.
	DUMPFLUX	Makes a full field run write a FLUX file.
	EQUALREG	Sets an array to a constant in a flux region.
	FLUXNUM	Identifies extent of each flux region.
	FLUXREG	Identifies which flux regions are active.
	FLUXTYPE	Specifies the type of flux boundary condition.
	MULTIREG	Multiplies an array by a constant in a given flux region.
E100 only	RPTGRID	Controls on output from the GRID section: FLUXNUM outputs flux region numbers.
	USEFLUX	Specifies the FLUX file used for a reduced run.
E100 only	USENOFLO	Allows a USEFLUX data set to be run without a FLUX file.
E300 only	XSTC	Specifies oil composition at standard conditions.
E300 only	YSTC	Specifies gas composition at standard conditions.
E300 only	THFLXTF	Specifies the heat transmissibility or flow transmissibility ratio T

Options keywords

Restriction	Keyword	Description
E100 only	OPTIONS	<p>Activates special program options</p> <p>item 25: Drainage regions are written to the DEBUG file.</p> <p>item 65: Remove certain error messages about wells connecting in multiple flux regions, or for local grids adjacent to flux boundaries in a DUMPFLUX run.</p> <p>item 232: Removes the ban on EOR/tracer functionality with DUMPFLUX and USEFLUX runs. In this case, the user must ensure that on EOR/tracer data is needed from outside the flux boundary.</p>
E300 only	OPTIONS3	<p>Activates special program options</p> <p>item 69: Remove certain error messages about wells connecting in multiple flux regions in a DUMPFLUX run.</p>

Restriction	Keyword	Description
E300 only	FLUXOPTS	YES: Makes a full-field run output necessary information for a thermal reduced run.

22

Foam model

x	ECLIPSE 100
x	ECLIPSE 300
	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 breakthrough of injected gas or to reduce the production of gas cap.

A foam is generated by adding a surfactant to an aqueous phase, and passing a gas through the surfactant to generate a stable dispersion of gas bubbles in the liquid. The foam can be transported with the gas flow into the reservoir. The major beneficial effect of the foam is to reduce the mobility of the gas. Note that in foam flooding the reduction of the interfacial tension (between oil and water) is not a significant effect. The reduction of gas mobility typically depends on a range of factors including pressure and shear rate.

The foam stability has a major effect on the usefulness of foam injection. Typically the foam suffers from adsorption on to the rock matrix, decay over time, and enhanced decay in the presence of water.

The physics of the foam flooding process is in general very complex. For example, when foam bubbles form in a porous medium the bubble size typically fills the pore size of the rock matrix. These bubbles tend not to move until they are compressed (hence reducing their size) by applying a higher pressure. Then in turn more bubbles are generated at the new higher pressure, but with the original bubble size.

ECLIPSE 100

The ECLIPSE 100 foam model does not attempt to model the details of foam generation and collapse. Foam is modeled as tracer which may be transported with either the gas or the water phase with account taken of adsorption on to the rock surface and decay over time. Two models are available to describe the effect of foam on the gas mobility: a tabular model and a functional model.

ECLIPSE 300

The ECLIPSE 300 foam model is similar to the ECLIPSE 100 foam model. A water component is used to model the foam surfactant. The ECLIPSE 300 foam model does not have specific keywords for decay or adsorption. Instead the chemical reaction model can be used to model the decay of the surfactant, or to convert the surfactant into a solid component. The ECLIPSE 100 functional model of the gas mobility can be used.

ECLIPSE 100 foam model

Foam conservation equation

Although foam is essentially a mixture of gas, water and surfactant, ECLIPSE models it as an effective concentration of surfactant transported in either the gas or the water phase. Hence the foam concentration can be thought of as the surfactant concentration existing in foam form. The distribution of the injected foam is therefore solved by a conservation equation:

$$\begin{aligned}
& \frac{d}{dt} \left(\frac{VS_g C_f}{B_r B_g} \right) + \frac{d}{dt} \left(V\rho_r C_f^a \frac{1-\varphi}{\varphi} \right) \\
&= \Sigma \left[\frac{T k_{rg}}{B_g \mu_g} M_{rf} (\delta P_g - \rho_g g D_z) \right] C_f + Q_g C_f - \lambda(S_w, S_o) V C_f
\end{aligned} \tag{Eq. 22.1}$$

as a tracer in the gas phase with decay, or as a tracer in the water phase with decay:

$$\begin{aligned}
& \frac{d}{dt} \left(\frac{VS_w C_f}{B_r B_w} \right) + \frac{d}{dt} \left(V\rho_r C_f^a \frac{1-\varphi}{\varphi} \right) \\
&= \Sigma \left[\frac{T k_{rw}}{B_w \mu_w} (\delta P_w - \rho_w g D_z) \right] C_f + Q_w C_f - \lambda(S_w, S_o) V C_f
\end{aligned} \tag{Eq. 22.2}$$

where

C_f	denotes the foam concentration
ρ_w, ρ_g	denotes the water and gas density respectively
Σ	denotes the sum over neighboring cells
C_f^a	denotes the adsorbed foam concentration
μ_w, μ_g	denotes the water and gas viscosity respectively
D_z	is the cell center depth.
B_r, B_w, B_g	is the rock, water and gas formation volume respectively
T	is the transmissibility
k_{rw}, k_{rg}	is the water and gas relative permeability respectively
S_w, S_g	is the water and gas saturation respectively
V	is the block pore volume
Q_w, Q_g	is the water and gas production rate respectively
P_w, P_g	is the water and gas pressure respectively
λ	is the rate decay parameter function of oil and water saturation
M_{rf}	is the gas mobility reduction factor described below
g	is the acceleration due to gravity
φ	is the porosity.

By default, the foam concentrations are updated fully implicitly at the end of a timestep after the inter-block phase flows have been determined. If the foam effects on the flow properties are very strong, the foam concentrations can be solved in each Newton iteration so that the tracer concentrations are updated

simultaneously with the phase equations by setting item 126 of the OPTIONS keyword to 1. The new foam concentrations are then used to update the flow properties for the subsequent iterations.

The foam is assumed to exist only in either the gas or the water phase and either equation 22.1 or equation 22.2 only is solved. This is specified using the [FOAMOPTS](#) keyword, which is also used to specify which form of gas mobility reduction model is to be used.

Adsorption

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

- either a look-up table of adsorbed foam as a function of the flowing foam concentration using the [FOAMADS](#) keyword
- or by a generic analytical adsorption model using the [ADSORP](#) keyword with FOAM as an argument. This model enables the adsorption to depend on rock permeability as well.

The mass adsorbed foam on to the rock is given by:

$$V \cdot \left(\frac{1-\varphi}{\varphi} \right) \cdot \rho_r \cdot C_f^a \quad \text{Eq. 22.3}$$

and the mass density of the rock ρ_r should be supplied using the second argument of [FOAMROCK](#) keyword. If desorption is prevented using the first argument of the [FOAMROCK](#) keyword, then the adsorbed foam concentration may not decrease with time. If desorption is allowed then each grid block retraces the adsorption isotherm as the foam concentration rises and falls in the cell.

Foam decay

Foam effectiveness will typically reduce over time, even in conditions very favorable to foam stability. This reduction in effectiveness may be accelerated in the presence of water or oil. The reduction in foam effectiveness over time is modeled by foam decay $\lambda(S_w, S_o)$. The half-life of the decay can be a function of both oil and water saturation. If the decay half-life is a function of both oil and water saturation, the foam is assumed to decay with the minimum half-life. The corresponding data is entered using [FOAMDCYW](#) and [FOAMDCYO](#).

Gas mobility reduction

The foam modifies the gas mobility by way of a simple multiplier supplied as a function of foam concentration (that is the effective surfactant concentration). The mobility modification is applied explicitly; that is, the modification associated with the conditions at the end of each iteration or timestep is applied at the subsequent iteration or timestep.

Gas mobility reduction models

Two methods of modeling the gas mobility reduction M_{rf} due to foam are available:

Tabular model:

For this model the gas mobility reduction factor is modeled using a set of tables which specify the mobility reduction as tabulated functions of foam concentration ([FOAMMOB](#) keyword), pressure ([FOAMMOBP](#) keyword) and shear ([FOAMMOBS](#) keyword) from which the net mobility reduction factor is calculated.

Functional form:

For this model, the gas mobility reduction factor is modeled in terms of a set of functions which represent the individual reduction factors due to surfactant concentration ([FOAMFSC](#) keyword), oil saturation ([FOAMFSO](#) keyword), water saturation ([FOAMFSW](#) keyword) and capillary number ([FOAMFCN](#) keyword). These are combined multiplicatively with a reference mobility reduction factor ([FOAMFRM](#) keyword) to determine the net mobility reduction factor.

These two methods are selected using the [FOAMOPTS](#) keyword. This tabular gas mobility reduction model is available with either gas or water specified as the transport phase. However, the functional gas mobility reduction model is only available if water is specified as the transport phase. The formulation of these two models is described below.

Gas mobility reduction – tabular model

For the tabular model, the gas mobility reduction factor M_{rf} is expressed in the form:

$$M_{rf} = (1 - M_{rf}^{cp})M_v(V_g) + M_{rf}^{cp} \quad \text{Eq. 22.4}$$

$$M_{rf}^{cp} = (1 - M_c(C_f))M_p(P) + M_c(C_f) \quad \text{Eq. 22.5}$$

where

$M_v(V_g)$ is the mobility reduction modifier due to gas velocity ([FOAMMOBS](#) keyword).

$M_c(C_f)$ is the gas mobility reduction modifier due to foam concentration ([FOAMMOB](#) keyword).

$M_p(P)$ is the mobility reduction modifier due to pressure ([FOAMMOBP](#) keyword).

P is the oil phase pressure.

V_g is the gas velocity (see below).

The gas velocity is calculated according to the expression:

$$V_g = B_g \left(\frac{F_g}{\varphi \cdot A} \right) \quad \text{Eq. 22.6}$$

where

F_g is the gas flow rate in surface units.

B_g is the gas formation volume factor.

φ is the average porosity of the two cells.

A is the flow area between the two cells.

The gas mobility reduction modifiers due to pressure and gas velocity are optional and will tend to increase the gas mobility as compared to the mobility reduction due to foam concentration alone.

Gas mobility reduction - functional model

For the functional model, the mobility reduction factor M_{rf} is expressed in the form [Ref. 65] and [Ref. 20]:

$$M_{rf} = \frac{1}{1 + (M_r \cdot F_s \cdot F_w \cdot F_o \cdot F_c)} \quad \text{Eq. 22.7}$$

where

M_r is the reference mobility reduction factor.

F_s is the mobility reduction factor component due to surfactant concentration.

F_w is the mobility reduction factor component due to water saturation.

F_o is the mobility reduction factor component due to oil saturation.

F_c is the mobility reduction factor component due to gas velocity (capillary number).

The reference mobility reduction factor M_r is typically in the range of 1 to 100 and corresponds to the normalized resistance to flow for a minimum bubble size in the absence of factors that increase bubble size. It is specified using the [FOAMFRM](#).

Each mobility reduction factor may optionally be specified or omitted according to whether or not the associated keyword is present. However, at least one of these components must be specified for the mobility reduction factor to be operational.

The mobility reduction factor F_s represents the dependence upon foam (effective surfactant) concentration and is expressed as:

$$F_s = \left(\frac{C_s}{C_s^r} \right)^{e_s} \quad \text{Eq. 22.8}$$

where

C_s is the effective surfactant concentration.

C_s^r is the reference surfactant concentration.

e_s is an exponent which controls the steepness of the transition about the point

$$C_s = C_s^r.$$

The significance of F_s is determined by the reference surfactant concentration above which the presence of surfactant becomes significant in the creation of foam. For low surfactant concentrations $C_s < C_s^r$ (weak foam), the value of F_s will be less than 1 and will tend to 0 as the surfactant concentration decreases to 0. Conversely, for high surfactant concentrations $C_s > C_s^r$ (strong foam), the value of F_s will be greater than 1 and will increase with increasing surfactant concentration. The steepness and curvature of F_s versus C_s is determined by the

value of e_s . The rate of change about the point where $C_s = C_s^r$ will increase with increasing e_s . The reference surfactant concentration and the exponent term are specified using the [FOAMFSC](#) keyword.

The mobility reduction factor F_w represents the dependence upon water saturation and is expressed as:

$$F_w = 0.5 + \frac{a \tan[f_w \cdot (S_w - S_w^1)]}{\pi} \quad \text{Eq. 22.9}$$

where

S_w is the water saturation.

S_w^1 is the limiting water saturation below which the foam ceases to be effective.

f_w is the weighting factor which controls the sharpness in the change in mobility.

The gas mobility increases sharply as water saturation decreases towards this limiting value. The limiting water saturation and the weighting factor are specified using the [FOAMFSW](#) keyword.

The mobility reduction factor F_o represents the dependence upon oil saturation and is expressed as:

$$F_o = \left(\frac{S_o^m - S_o}{S_o^m} \right)^{e_o} \quad \{S_o \leq S_o^m\} \quad \text{Eq. 22.10}$$

$$F_o = 0.0 \quad \{S_o > S_o^m\} \quad \text{Eq. 22.11}$$

where

S_o is the oil saturation.

S_o^m is the maximum oil saturation above which the foam ceases to be effective.

e_o is an exponent which controls the steepness of the transition about the point
 $S_o = S_o^m$

The maximum value of F_o is 1 and occurs when $S_o = 0.0$. For values of $S_o < S_o^m$, the value of F_o will decrease with increasing S_o . The rate of decrease is controlled by the value of the exponent e_o . For values of $S_o \geq S_o^m$, the value of F_o will be 0 and will completely eliminate any gas mobility reduction irrespective of the values of the other multiplicative factors. The maximum oil saturation and the exponent term are specified using the [FOAMFSO](#) keyword.

The mobility reduction factor F_c represent the dependence upon capillary number and is expressed as:

$$F_c = \left(\frac{N_c^r}{N_c} \right)^{e_c} \quad \text{Eq. 22.12}$$

where

N_c is the capillary number.

N_c^r is the reference capillary number.

e_c is an exponent which controls the steepness of the transition about the point

$$N_c = N_c^r.$$

This component controls the gas mobility for shear-thinning in the low foam quality region. The reference capillary number and the exponent term are specified using the [FOAMFCN](#) keyword. The capillary number is a dimensionless parameter which provides a measure of the ratio of viscous to capillary forces and is calculated according to the expression:

$$N_c = C_N \frac{\| K \cdot \nabla P \|}{\sigma_{wg}} \quad \text{Eq. 22.13}$$

where

K is the rock permeability.

P is the pressure.

σ_{wg} is the gas-water interfacial tension defined by the [FOAMFST](#) keyword.

C_N is the conversion factor which depends upon the units used.

The calculation of the capillary number is described in "[Capillary number](#)".

The mobility reduction factor is only calculated for grid blocks that contain foam with a concentration greater than or equal to the minimum concentration specified by item 3 of the [FOAMFSC](#) keyword.

Using the ECLIPSE 100 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 $1/(\text{poro} * \text{area})$ values used to calculate the gas velocity.

Foam transport, adsorption and decay

The optional [FOAMOPTS](#) keyword may be used to select the transport phase for the foam and the method of modeling gas mobility reduction due to foam. If this keyword is not specified, the transport phase will default to gas and the tabular model for gas mobility reduction will be used.

Restriction	Foam transport, adsorption and decay keywords	Description
	FOAMOPTS	Foam transport phase and mobility reduction method options.
Optional	FOAMADS	Foam adsorption data
	FOAMROCK	Foam-rock properties
Optional	FOAMDCYW	Foam decay data as a function of water saturation
Optional	FOAMDCYO	Foam decay data as a function of oil saturation

Gas mobility reduction - tabular model

If the tabular method of modeling gas mobility reduction as a function of either or both pressure and shear is selected, there is one obligatory keyword in the PROPS section to describe the reduction in gas mobility as a function of foam concentration. In addition there are two optional keywords which can be used to control the gas mobility.

Restriction	Gas mobility tabular model keywords	Description
Required	FOAMMOB	Tables of the gas mobility reduction factor as a function of foam concentration
Optional	FOAMMOBP	Gas mobility reduction as a function of pressure
Optional	FOAMMOBS	Gas mobility reduction as a function of shear

Gas mobility reduction - functional model

The functional method of modeling gas mobility can only be used if the foam transport phase is water. If the functional method is selected, the following keywords are available in the PROPS section to describe the reduction in gas mobility in the presence of foam. The Gas mobility functional model keywords:

Restriction	Gas mobility functional model keywords	Description
Required	FOAMFSC	Foam mobility reduction dependence on foam surfactant concentration
Optional	FOAMFRM	Foam mobility reference reduction factor
Optional	FOAMFCN	Foam mobility reduction dependence on capillary number
Obligatory if the FOAMFCN keyword specified	FOAMFST	Gas-water interfacial tension as a function of foam surfactant concentration
Optional	FOAMFSO	Foam mobility reduction dependence on oil saturation
Optional	FOAMFSW	Foam mobility reduction dependence on water saturation

With the exception of the FOAMFSC keyword, which must be specified if the functional model of gas mobility reduction is selected, and the FOAMFST keyword, which must be specified if the FOAMFCN keyword is specified, all of these keywords are optional.

The Foam Model's PROPS section data can be written to the PRINT file by using the FOAM mnemonic in the [RPTPROPS](#) keyword.

The injected foam concentration of a water or gas injector is specified using the [WFOAM](#) keyword in the SCHEDULE section.

Output controls

Output to the PRINT file can be controlled by using the following arguments in the [RPTSCHED](#) and [RPTSOL](#) keywords:

Argument	Description
FOAM	Foam concentration for each grid block.
FIPFOAM	A foam balance sheet for the field and each region.
FOAMADS	Foam currently adsorbed onto the rock.
FOAMDCY	The decay half life and the total decayed foam.
FOAMMOB	The gas mobility reduction factor. Note that, for the tabular model of gas mobility reduction, the gas mobility reduction includes the pressure dependent effect but not the shear effect, because the shear effect is calculated from the local velocities.
WELLS=2	Generates a field/group/well/connections summary of the foam injection and production.

The list of SUMMARY keywords has been extended to output foam-specific data. These keywords follow the passive tracer format with a keyword concatenated with the tracer name. Here foam is identified with the tracer name FOA. For example, the keyword for the field total foam production would be FTPRFOA.

Restrictions

If a cell is defined with zero pore volume in the GRID section, then making it active by changing the pore volume in the EDIT section using [PORV](#), is not allowed with the foam option.

The ECLIPSE 300 foam model

Transport

In the ECLIPSE 300 Foam Model, surfactant is transported in the water phase as a separate water component. See "[Water components](#)". The surfactant component properties (such as molecular weight) must be specified using the relevant water component keywords. Water components that behave as foam surfactants are defined using the [CWTYPE](#) keyword.

Adsorption

Foam adsorption and desorption can be modeled using chemical reactions. See "[Chemical Reactions](#)", "[Solid Phase](#)" and "[Adsorption of dissolved water components](#)".

The equilibrium between the waterborne surfactant component and the adsorbed solid component occurs when the concentrations are such that the two reaction rates are equal. If the reaction rates are linear functions of the concentrations, then the adsorbed (solid) concentration will be proportional to the surfactant concentration times the ratio of the adsorption and desorption reaction rates R_a / R_d .

An example of using reactions to model adsorption is given in "[Example data set for ECLIPSE 300](#)".

Decay

Foam decay can be modeled using a chemical reaction to convert the surfactant component to water. For example, a foam with a half life of $T_{1/2}$ days can be modeled with a reaction in which the foam surfactant decays to water at a rate of $(\ln 2) / T_{1/2}$ times the surfactant concentration.

It is not possible to vary the reaction rate as a function of the oil or water saturations directly, as done in the ECLIPSE 100 using the [FOAMDCYO](#) and [FOAMDCYW](#) keywords. However, it is possible to make the reaction rate depend on the concentration of a particular component (for example water) using the [REACCORD](#) keyword, or on the concentration of a tracer using the [REACTRAC](#) keyword. See "[Chemical reactions](#)" for more details of chemical reactions.

An example of using reactions to model foam decay is given in "[Example data set for ECLIPSE 300](#)".

Gas mobility reduction

ECLIPSE 100 gas mobility function keywords (see [22.1](#)) can be used to specify the gas mobility multiplier for the ECLIPSE 300 Foam Model. The surfactant concentration is calculated by summing the mass of all surfactant components in the cell and dividing by the water volume at surface conditions. The gas-water surface tension is then calculated the [FOAMFST](#) keyword. This is then used to determine the capillary number, which can be used to modify the mobility with the [FOAMFCN](#) keyword. The capillary number is calculated in different ways in the two simulators. This is discussed further in "[Capillary number](#)".

Comparison with ECLIPSE 100

If the surfactant component is given the same properties as the water component it will act as a tracer, as in the ECLIPSE 100 model. The foam volume will not be modeled, but its effect on the gas mobility will be. The ECLIPSE 300 Foam Model should therefore behave in a similar manner to the ECLIPSE 100 Foam Model, when that is used with water-phase transport and the functional gas mobility model (keyword [FOAMOPTS](#) items 1 and 2 set to WATER and FUNC).

Output controls

The relevant water component output mnemonics and keywords (and reaction / solid keywords if used) can be used to output information on the surfactant component. In addition the following foam mnemonics / keywords can be used:

Description	RPTSOL / RPTSCHED / RPTRST arguments	SUMMARY keyword
Surfactant concentration	FOAM	BFOAM
Gas-water surface tension	FOAMST	BFOAMST
Capillary number	FOAMCNM	BFOAMCNM

Description	RPTSOL / RPTSCHED / RPTRST arguments	SUMMARY keyword
The gas mobility reduction factor	FOAMMOB	BFOAMMOB

Table 22.1: Output controls

Summary of keywords

RUNSPEC (ECLIPSE 100)

- [FOAM](#) Activates the Foam Model.

GRID (ECLIPSE 100)

Keyword	Description
RPTGRID	Controls output from the GRID section. The FOAM argument outputs a map of the 1/(poro*area) values used to calculate the gas velocity for the tabular model of mobility reduction.

PROPS

Restriction	Keyword	Description
E100 only	ADSORP	Analytical adsorption isotherm (alternative to FOAMADS).
E100 only	FOAMADS	Foam adsorption data.
E100 only	FOAMDCYO	Foam decay data (as a function of oil saturation).
E100 only	FOAMDCYW	Foam decay data (as a function of water saturation).
	FOAMFCN	Foam mobility reduction functional dependence upon capillary number.
	FOAMFRM	Foam mobility reference reduction factor.
	FOAMFSC	Foam mobility reduction functional dependence upon foam surfactant concentration.
	FOAMFSO	Foam mobility reduction functional dependence upon oil saturation.
	FOAMFST	Gas-water interfacial tension as a function of foam surfactant concentration.
	FOAMFSW	Foam mobility reduction functional dependence upon water saturation.
E100 only	FOAMMOB	Tables of the gas mobility reduction factor as a function of foam concentration.
E100 only	FOAMMOBP	Gas mobility reduction as a function of pressure.
E100 only	FOAMMOBS	Gas mobility reduction as a function of shear.
E100 only	FOAMOPTS	Foam transport phase and mobility reduction method options.
E100 only	FOAMROCK	Foam-rock properties.
E100 only	RPTPROPS	Controls output from the PROPS section (FOAM argument for foam properties).

SUMMARY

Refer to [Foam SUMMARY keywords](#) in the *ECLIPSE Reference Manual*.

SOLUTION

Restriction	Keyword	Description
E100 only	SFOAM	Initial foam concentrations.
E100 only	TVDP	Initial foam concentrations with depth.
E100 only	TNUM	Foam concentration regions.
	RPTSQL	Output of data specific to the Foam Model

Restriction	RPTSQL argument	Description
E100 only	FIPFOAM	Output of foam fluid in place reports: 1: gives a report for the whole field. 2: in addition, gives a report for each fluid-in-place region.
	FOAM	Output of grid block foam concentration.
E100 only	FOAMADS	Output of grid block foam adsorption.
	FOAMCNM	Output of logarithm base ten of grid block capillary number if the functional model of mobility reduction has been selected and the FOAMFCN keyword specified.
E100 only	FOAMDCY	Output of grid block foam decay.
	FOAMMOB	Output of gas mobility factor due to foam.
E100 only	FOAMST	Output of gas-water surface tension.

Note: The output of the gas mobility reduction factor for the tabular model will include the effects of pressure dependence ([FOAMMOBP](#)) but not the shear effect ([FOAMMOBS](#)), as the shear effect is calculated for each flow separately.

SCHEDULE

Restriction	Keyword	Description
E100 only	RPTSCHE	Controls output from the SCHEDULE section. Argument FOAM outputs the foam concentrations and the foam adsorption when active.
E100 only	WFOAM	Foam concentration for injection wells (can be used as a User Defined Argument (UDA)).

Restriction	RPTSCHE argument	Description
E100 only	FIPFOAM	Output of foam fluid in place reports: 1: gives a report for the whole field. 2: in addition, gives a report for each fluid-in-place region.
	FOAM	Output of grid block foam surfactant concentration.

Restriction	RPTSCHED argument	Description
E100 only	FOAMADS	Output of grid block foam adsorption.
	FOAMCNM	Output of logarithm base ten of grid block capillary number if the functional model of mobility reduction has been selected and the FOAMFCN keyword specified.
E100 only	FOAMDGY	Output of grid block foam decay.
	FOAMMOB	Output of gas mobility factor due to foam.
E300 only	FOAMST	Output of gas-oil surface tension.

Note: The output of the gas mobility reduction factor for the tabular model will include the effects of pressure dependence ([FOAMMOBP](#)) but not the shear effect ([FOAMMOBS](#)), as the shear effect is calculated for each flow separately.

Example data set for ECLIPSE 100

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 ADSORPTION 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 DATA AS 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
--
RPTSQL
'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
WELSPCS
  '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
WCONINJE
  '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

```

Example data set for ECLIPSE 300

The "Example data set for ECLIPSE 100" can be modified to run with ECLIPSE 300. A new well is added to inject the foam surfactant in the water phase. The functional model is used for the gas mobility. Two components are added, to represent the foam surfactant in the water phase and that adsorbed to the rock.

RUNSPEC

The RUNSPEC section is modified to include the water components, solid model and number of reactions. In addition the number of wells is increased:

```

RUNSPEC
TITLE
  Simple FOAM model example
DIMENS

```

```

10 10 3 /
OIL
WATER
GAS
SOLID           Solid component used to model adsorbed foam
BLACKOIL
COMPSOL
3 /           Oil, gas and one solid component
COMPW
2 /           Water and foam surfactant components
REACTION
2 1 1* 1* 1 / Foam decay, adsorption/desorption
FIELD

EQLDIMS
1 100 10 1 1 /

TABDIMS
2 1 16 12 2 12 1 1 /

REGDIMS
2 1 0 0 0 1 /

WELLDIMS
3 1 1 3 /

NUPCOL
4 /

START
19 'APR' 1994 /

NSTACK
24 /

FMTOUT

```

GRID

The GRID section is the same as the ECLIPSE 100 example:

```

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, PORE 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 has a number of changes from the ECLIPSE 100 example:

- Solid component properties for adsorbed foam are specified
- Water component properties for water and the foam surfactant are specified
- The functional gas mobility reduction model is used
- A reaction is used to model foam decay
- Two reactions are used to model foam adsorption and desorption

```
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      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  /  

/  

--Solid component properties  

CNAMES          OIL GAS SOLID /  

CVTYPE          1* 1* SOLID /  

MW              1* 1* 18.015 /  

SDREF            1* 1* 62.964 /  

--Water component properties  

CWTYPE          1*      SURFF / Water component 2 is foam surfactant  

MW              18.015 18.015 /  

PREFW            4014.7 4014.7 / PVTW item 1  

DREFW            62.964 62.964 / (DENSITY item 2) / (PVTW item 2)  

CREFW            3.13D-6 3.13D-6 / PVTW item 3  

VREFW            0.31 0.31 / PVTW items 4 and 5  

--  

--Foam decay, adsorption and desorption  

--  

STOREAC  

--OIL  GAS  SOLID  WAT  SURF  

  0.0  0.0  0.0    0.0  1.0 / Decay      SURF->WATER  

  0.0  0.0  1.0    0.0  0.0 / SOLID<->SURF  

STOPROD

```

```

--OIL  GAS  SOLID  WAT  SURF
  0.0  0.0  0.0    1.0  0.0 /      Decay          SURF->WATER
  0.0  0.0  0.0    0.0  1.0 /      SOLID<->SOLID
REACRATE
--DECAY      ADSO
  0.000231 1.0 / Foam decay with constant half life
--                                = 3000 days
EQLDREAC
1 3 3 5 1 BOTH 1 / Rate proportional to SOLID - EQLDTAB_1(SURF)
EQLDTAB
  0.0      0.00000
  0.001    0.22065
  1.000    0.22065 /
REACCORD
--OIL  GAS  SOLID  WAT  SURF
  0.0  0.0  0.0    0.0  1.0 / Foam decay proportional to SURF conc
  0.0  0.0  0.0    0.0  0.0 /
--
--
-- GAS MOBILITY REDUCTION
--
FOAMFRM
1.1 /
1.1 /

FOAMFSC
0.001 0.1 1.0e-6 /
0.001 0.1 1.0e-6 /

FOAMFST
0       10.0
0.01   5.0
0.05   0.1/
0       11.0
0.01   6.0
0.05   0.2 /
FOAMFCN
1E-7 0.1 /
1E-7 0.1 /

--
-- OUTPUT CONTROLS FOR PROPS DATA
--
RPTPROPS
'FOAM' /

```

REGIONS

The REGIONS section is unchanged:

```

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 has been modified to include initial water composition. The initial solids are defaulted:

```

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   /
-- VARIATION OF INITIAL WATER COMPOSITION WITH DEPTH
--
-- DEPTH    WAT    SURF
WMFVD       8200    1.0 0.0
            8500    1.0 0.0   /
-- OUTPUT CONTROLS
--
RPTSQL      'PRESSURE' 'SWAT' 'SGAS' 'FOAM' /

```

SUMMARY

The simulator specific SUMMARY vectors are changed:

```

SUMMARY
--REQUEST PRINTED OUTPUT OF SUMMARY FILE DATA
RUNSUM

-- 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
/

BSSAT
10 10 3 /

```

```

1 1 1 /
/
BFOAM
10 10 3 /
1 1 1 /
/
BREAC
10 10 3 1 /
10 10 3 2 /
10 10 3 3 /
1 1 1 1 /
1 1 1 2 /
1 1 1 3 /
/

```

SCHEDULE

The SCHEDULE section has been modified to reflect the different foam injection method. Water alternating with gas (WAG) is injected. The water contains a foam surfactant to reduce the mobility of the following gas:

```

SCHEDULE
-- CONTROLS ON OUTPUT AT EACH REPORT TIME
RPTSCHED
'PRESSURE' 'SWAT' 'SGAS' 'SOIL' 'FOAM' 'WELLS=2' 'NEWTON=2' 'FIPFOAM=2'
'FOAMMOB' /

RPTRST
'BASIC=3' 'FIP' 'FREQ=2' /


-- WELL SPECIFICATION DATA
--
-- WELL      GROUP LOCATION BHP   PI
-- NAME      NAME     I   J   DEPTH DEFN
WELSPCS
'PRODUCER' 'G'    10 10    8400 'OIL'  /
'GINJ'      'G'    1   1    8335 'GAS'  /
'WINJ'      'G'    1   1    8335 'WAT'  /
/


-- SPECIFY THE FOAM INJECTOR
--
WELLSTRW
  WATER 1.0 0.0 /  PURE WATER
  FOAM 0.9 0.1 /  10% SURFACTANT IN WATER
/
WINJW
  WINJ STREAM FOAM /  SET WELL WINJ TO USE THE FOAM STREAM
/


-- 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 /
'GINJ'      1   1 1 1 'OPEN' 1   -1  0.5 /
'WINJ'      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 /
/

```

```

-- INJECT FOAM
--
-- WELL    INJ   OPEN/ CNTL   FLOW
-- NAME    TYPE  SHUT  MODE   RATE
WCONINJE
    'GINJ'   'GAS' 'SHUT' 'RATE'  100000  1* 10000 /
    'WINJ'   'WAT' 'OPEN' 'RATE'   65000  1* 10000 /
/
TSTEP
 30 /

-- INJECT GAS
--
-- WELL    INJ   OPEN/ CNTL   FLOW
-- NAME    TYPE  SHUT  MODE   RATE
WCONINJE
    'GINJ'   'GAS' 'OPEN' 'RATE'  100000  1* 10000 /
    'WINJ'   'WAT' 'SHUT' 'RATE'   65000  1* 10000 /
/
-- INJECT FOAM
--
-- WELL    INJ   OPEN/ CNTL   FLOW
-- NAME    TYPE  SHUT  MODE   RATE
WCONINJE
    'GINJ'   'GAS' 'SHUT' 'RATE'  100000  1* 10000 /
    'WINJ'   'WAT' 'OPEN' 'RATE'   65000  1* 10000 /
/
TSTEP
 30 /

-- INJECT GAS
--
-- WELL    INJ   OPEN/ CNTL   FLOW
-- NAME    TYPE  SHUT  MODE   RATE
WCONINJE
    'GINJ'   'GAS' 'OPEN' 'RATE'  100000  1* 10000 /
    'WINJ'   'WAT' 'SHUT' 'RATE'   65000  1* 10000 /
/
TSTEP
 70 /

END

```

23

Formulation of the equations

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 timesteps in the simulation. Both fully implicit and IMPES may be used in the same run by using keywords [DIMPES](#) or [DIMPLICIT](#) at appropriate places in the SCHEDULE section.

ECLIPSE 300

The default solution procedure is [AIM](#) for all compositional runs and [FULLIMP](#) for all black oil or thermal runs. The default solution procedure may be changed by using one of the keywords [AIM](#), [FULLIMP](#) or [IMPES](#) in the RUNSPEC section.

Description of equations

The non-linear residual, R_{fl} , for each fluid component in each grid block at each timestep is:

$$R_{fl} = \frac{\delta M}{\delta t} + F + Q \quad \text{Eq. 23.1}$$

where

dM is the mass, per unit surface density, accumulated during the current timestep, δt

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:

$$R_{fu} = f_l - f_v = 0 \quad \text{Eq. 23.2}$$

where f_l and f_v are the component fugacities in the liquid and vapor phases respectively.

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

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;

$$R = \begin{bmatrix} R_o \\ R_w \\ R_g \end{bmatrix} \quad X = \begin{bmatrix} P_o \\ S_w \\ S_g \text{ or } R_s \text{ or } R_v \end{bmatrix} \quad \text{Eq. 23.3}$$

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

$$\frac{dR_i}{dX_j} = \begin{bmatrix} \frac{dR_o}{dP_o} & \frac{dR_o}{dS_w} & \frac{dR_o}{dS_g} \\ \frac{dR_w}{dP_o} & \frac{dR_w}{dS_w} & \frac{dR_w}{dS_g} \\ \frac{dR_g}{dP_o} & \frac{dR_g}{dS_w} & \frac{dR_g}{dS_g} \end{bmatrix}_{ij} \quad \text{Eq. 23.4}$$

The mass change during the timestep, dt , is then proportional to

$$dM = M_{t+dt} - M_t \quad \text{Eq. 23.5}$$

with

$$M = PV \begin{bmatrix} \frac{S_o}{B_o} + \frac{R_v S_g}{B_g} \\ \frac{S_w}{B_w} \\ \frac{S_g}{B_g} + \frac{R_s S_o}{B_o} \end{bmatrix} \quad \text{Eq. 23.6}$$

where

PV is the pore volume

B_o is the oil formation volume factor

B_w is the water formation volume factor

B_g is the gas formation volume factor

R_s is the solution gas/oil ratio

R_v is the vapor oil/gas ratio.

When S_g is zero the solution variable becomes R_s (undersaturated oil) and when S_o is zero the solution variable becomes R_v (undersaturated gas). Terms in the Jacobian are adjusted in accordance with the change of variable. No approximations are made in evaluating the Jacobian in ECLIPSE 100. Great care is taken to compute all derivatives to ensure quadratic convergence of Newton's method.

Hydrocarbon states are:

1. State 1: Gas only $R_s = 0, S_g = 1 - S_w,$

variables are $P_o, S_w, R_v.$

2. State 2: Gas & oil, $R_v = R_{v \text{ sat}}, R_s = R_{s \text{ sat}},$

variables are P_o, S_w, S_g

3. State 3: Oil only, $R_v = 0, S_g = 0,$

variables are $P_o, S_w, R_s.$

The number of cells in each state, together with the number of cells changing state during a Newton iteration (NTRAN), are printed in the summary of each Newton iteration.

Note: Cells almost entirely full of water are in state 2 by default. Therefore the solution variable R_s is set to $R_{s \text{ sat}}$. A transition from state 2 to state 3 may occur in such blocks, for example due to the influx of undersaturated oil. The cell's R_s value is then limited to not exceed the average GOR of inflows from neighboring cells as this will probably be a better estimate than $R_{s \text{ sat}}$.

ECLIPSE 300

The primary solution variables, used for cells in all phase states, are $X = (P, m_1, \dots, m_{N_c}, m_w)$

where

m_1, \dots, m_{N_c} 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

$$R_m = PV - FV = 0 \quad \text{Eq. 23.7}$$

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

$$FV = PV \left(\frac{m_t L}{b_o^m} + \frac{m_t V}{b_g^m} + \frac{m_w}{b_w^m} \right) \quad \text{Eq. 23.8}$$

with the total hydrocarbon molar density given by

$$m_t = \sum_c m_c \quad \text{Eq. 23.9}$$

where

L and V are the liquid and vapor mole fractions, and

b_o^m, b_g^m and b_w^m are the molar densities of oil, gas and water.

The form of the Jacobian matrix condition in the compositional case is then

$$J = \frac{dR_f}{dX}$$

which can be depicted in terms of the primary solution variables (P, m_w, m_c for each hydrocarbon component c) and residual equations (R_w, R_c and R_m) for the fully implicit method as

$$J = \begin{matrix} & P & m_w & m_1 & \dots & m_{N_c} \\ \text{Water} & x & x & x & \dots & x \\ \text{Comp 1} & x & x & x & \dots & x \\ \dots & x & x & x & \dots & x \\ \text{Comp} N_c & x & x & x & \dots & x \\ \text{Volumebal} & x & x & x & \dots & x \end{matrix} \quad \text{Eq. 23.10}$$

The residual equation $R(X) = 0$ is solved for X using Newton's method.

In two phase blocks further secondary solution variables are required. These are the $J_i = \log(K_i)$ values for each component, and the vapor mole fraction V. In terms of these variables the equal fugacity residual condition may be expressed as:

$$R_{fu} = J - H_t + H_v \quad \text{Eq. 23.11}$$

where

$$H_t = \ln\left(\frac{f_t}{P_x}\right) \quad \text{Eq. 23.12}$$

$$H_v = \ln\left(\frac{f_v}{P_y}\right) \quad \text{Eq. 23.13}$$

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

$$R_n = \sum_i z_i \frac{(K_i - 1)}{1 + V(K_i - 1)} = 0 \quad \text{Eq. 23.14}$$

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

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

$$R_{fu}(X_f) = 0 \quad \text{Eq. 23.15}$$

In [IMPSAT](#) mode we have three additional residuals to define the saturation variables X_o , X_g , X_w :

$$R_p = PV(X - S_p(P, m)) = 0 \quad \text{Eq. 23.16}$$

where $S_p(P, m)$ is the fractional volume occupied by phase p .

The fully implicit method

The fully implicit method is the default for black oil runs in both ECLIPSE 100 and ECLIPSE 300. It is not recommended for compositional runs where there are often too many components to be able to use the fully implicit method efficiently.

Newton's method is used to solve the non-linear residual equations $R(X) = 0$. Several iterations may be required to reduce the residuals to a 'sufficiently small' value. In ECLIPSE 100, two measures are used to define the meaning of 'sufficiently small'. These are the material balance error and the maximum saturation normalized residual.

The IMPES and AIM methods

The IMPES (IMplicit Pressures Explicit Saturation) 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.

$$R = \frac{M_{t+dt} - M_t}{dt} + F(P_{t+dt}, S_t) + Q(P_{t+dt}, S_t) \quad \text{Eq. 23.17}$$

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 200 psi. 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.

AIM

ECLIPSE 300

The Adaptive IMplicit method ([AIM](#)) is a compromise between the fully implicit and IMPES procedures. Cells with a high throughput ratio are chosen to be implicit for stability and to obtain large 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

ECLIPSE 300

This option is based on the paper "IMPES Stability: The Stable Step" by K.H. Coats [Ref. 26]. The reader is referred there for all technical points. By default, AIM and IMPES runs choose the implicit cells/timestep 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 timestep selection and implicit cell choice on a much more rigorous footing based on the maximum **stable** timestep allowed in the **explicit** cells. This timestep 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.

- items 2 to 4 of the CFLLIMIT keyword can be altered to allow timesteps that are theoretically unstable
- 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 23.1 water is flowing from cell A to cell B. Water levels at time t are shown by the dark shading. Because the saturation in cell B is below S_{wcr} at time t , water cannot flow out of cell B. During the 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.

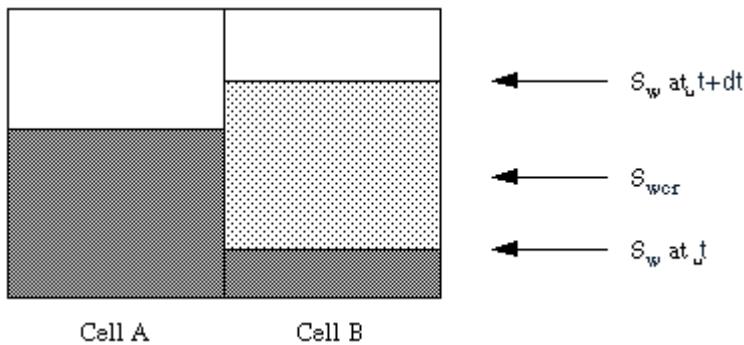


Figure 23.1. The most common instability using IMPES

This type of instability may limit the efficiency of the IMPES method. The problem is reduced in some simulators (but not in ECLIPSE) by automatically rescaling the input relative permeability curves to equally spaced saturation intervals. This is not done in ECLIPSE because:

- you may wish the data to be honored precisely
- sharp relative permeability contrasts are handled efficiently by the fully implicit method
- table look-up is equally efficient either way (contrary to popular belief)
- the user has the option of inputting ‘smooth’ k_r curves

Convergence criteria for residual equations

The Newton solver aims to reduce the residuals $R(X)$ to zero. In ECLIPSE 100 the convergence criteria are primarily based on material balance and maximum residual checks on R . In ECLIPSE 300, the convergence criteria are primarily based on the solution change Δx being ‘sufficiently small’ in the Newton iteration.

ECLIPSE 100

Material balance

If the residuals are summed over all cells in the reservoir, the flow terms F cancel, because the flow out of one cell is always equal and opposite in sign to the corresponding flow into its neighboring cell. Thus the sum of the residuals for each phase or component corresponds to the net mass accumulation within the reservoir less the net influx through wells. This is the material balance error. For a three-component system we have

$$\begin{aligned}\sum_i (R_o)_i &= \sum_i \left(\frac{dM_o}{dt} \right)_i + \sum_i (Q_o)_i \\ \sum_i (R_w)_i &= \sum_i \left(\frac{dM_w}{dt} \right)_i + \sum_i (Q_w)_i \\ \sum_i (R_g)_i &= \sum_i \left(\frac{dM_g}{dt} \right)_i + \sum_i (Q_g)_i\end{aligned}\quad \text{Eq. 23.18}$$

where

\sum_i refers to the sum over all reservoir cells and

$(R_o)_i$ is the oil residual in cell i

In ECLIPSE 100 the material balance errors are converted to meaningful, problem independent, numbers by scaling to equivalent field saturation values:

$$\begin{aligned}MB_o &= \bar{B}_o dt \left\{ \left(\sum_i (R_o)_i \right) / \left(\sum_i (\text{PV})_i \right) \right\} \\ MB_w &= \bar{B}_w dt \left\{ \left(\sum_i (R_w)_i \right) / \left(\sum_i (\text{PV})_i \right) \right\} \\ MB_g &= \bar{B}_g dt \left\{ \left(\sum_i (R_g)_i \right) / \left(\sum_i (\text{PV})_i \right) \right\}\end{aligned}\quad \text{Eq. 23.19}$$

where \bar{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 1.0E-7. MB values are printed out in the summary of each Newton iteration. Conventional material balance accounts in conventional units can also be printed at each report time.

Normalized residuals

Although great care is taken to ensure that material balance errors are unusually small in ECLIPSE 100, this is not considered to be a sufficiently rigorous test of convergence. The second test is obtained by computing the maximum saturation normalized residuals

$$\begin{aligned} \text{CNV}_o &= \bar{B}_o dt \cdot \text{MAX}_i \left| \frac{(R_o)_i}{(\text{PV})_i} \right| \\ \text{CNV}_w &= \bar{B}_w dt \cdot \text{MAX}_i \left| \frac{(R_w)_i}{(\text{PV})_i} \right| \\ \text{CNV}_g &= \bar{B}_g dt \cdot \text{MAX}_i \left| \frac{(R_g)_i}{(\text{PV})_i} \right| \end{aligned} \quad \text{Eq. 23.20}$$

where MAX_i is the maximum value over all cells in the reservoir.

By converting each convergence error to an equivalent saturation value we can attach sensible limits to the CNV numbers, which are considered to have converged if they are all less than 0.001.

The convergence criteria can be altered by the [TUNING](#) keyword. However, the default values are most often sufficient and it is **not** recommended that they should be changed. Loosening the convergence tolerances in a difficult run often results in increased run times.

ECLIPSE 300

The convergence criteria in ECLIPSE 300 are based on solution change targets. Since the primary solution variables X are molar densities, it is natural to use these to control the convergence. However, large saturation changes may occur for very small changes in composition, so the use of an effective saturation is preferred. Consider the expression:

$$\Delta S' = \frac{du^T}{dm_c} \cdot \Delta m_c \quad \text{Eq. 23.21}$$

where

$$u^T = \frac{m_t L}{b_o^m} + \frac{\dot{m}_t V}{23b_g^m} + \frac{m_w}{b_w^m}$$

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 tighten down the convergence.

Flows

The flow rates between neighboring cells are computed for each simulator as described below.

ECLIPSE 100

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

$$F_{ni} = T_{ni} \begin{bmatrix} \frac{k_{ro}}{B_o \mu_o} & 0 & \frac{R_v k_{rg}}{B_g \mu_g} \\ 0 & \frac{k_{rw}}{B_w \mu_w} & 0 \\ \frac{R_s k_{ro}}{B_o \mu_o} & 0 & \frac{k_{rg}}{B_g \mu_g} \end{bmatrix}_u \times \begin{bmatrix} dP_{oni} \\ dP_{wni} \\ dP_{gni} \end{bmatrix} \quad \text{Eq. 23.23}$$

where

$$\begin{aligned} dP_{oni} &= P_{on} - P_{oi} - \rho_{oni} g (D_n - D_i) \\ dP_{wni} &= P_{wn} - P_{wi} - \rho_{wni} g (D_n - D_i) \\ &= P_{on} - P_{oi} - \rho_{wni} g (D_n - D_i) - P_{cown} + P_{cowi} \\ dP_{gni} &= P_{gn} - P_{gi} - \rho_{gni} g (D_n - D_i) \\ &= P_{on} - P_{oi} - \rho_{gni} g (D_n - D_i) + P_{cogn} - P_{cogi} \end{aligned}$$

T_{ni} is the transmissibility between cells n and i,

k_r is the relative permeability (k_{ro} is the relative permeability of oil etc.),

μ is the viscosity (μ_w is the viscosity of water etc.),

dP is the potential difference (dP_{gni} is the gas potential difference between cells n and i),

ρ is the fluid density (ρ_{oni} is the density of oil at the interface between cells n and i),

g is the acceleration due to gravity

(0.0000981 in metric units, 0.00694 in field units and 0.000968 in lab units),

D is the cell center depth.

The subscript u indicates that the fluid mobilities are to be evaluated in the upstream cell (cell n if dP_{ni} is positive, cell i if dP_{ni} is negative). The upstream calculation applies separately for each equation (oil, water, gas) so that, for example, oil may flow from cell i to cell n while water flows from cell n to cell i.

The net flow rate from cell i into neighboring cells is obtained by summing over the neighboring cells,

$$F_i = \sum_n F_{ni}.$$

The rate of flow into a production well from cell i is

$$Q_i = -T_{wi}(P_{oi} - H_{iw} - P_{bh}) \begin{bmatrix} \frac{k_{ro}}{B_o \mu_o} + \frac{R_v k_{rg}}{B_g \mu_g} \\ \frac{k_{rw}}{B_w \mu_w} \\ \frac{k_{rg}}{B_g \mu_g} + \frac{R_s k_{ro}}{B_o \mu_o} \end{bmatrix}_i \quad \text{Eq. 23.24}$$

where

T_{wi} is the well connection transmissibility factor

H is the hydrostatic head correction

P_{bh} is the bottom hole pressure

Well terms are discussed in detail in a separate chapter: see "Well inflow performance".

ECLIPSE 300

The flow rate of component c embedded in a phase p ($p=o,w,g$) into cell i from a neighboring cell n, F_{pni}^c , is

$$F_{pni}^c = T_{ni} M_p^c dP_{pni} \quad \text{Eq. 23.25}$$

where

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

$$M_p^c = x_p^c k_{rp} (S_p) \frac{b_p^m}{\mu_p} \quad \text{Eq. 23.26}$$

where

x_p^c is the mole fraction of component c in phase p

k_{rp} is the relative permeability of phase p

S_p is the saturation of phase p

b_p^m is the molar density of phase p

μ_p is the viscosity of phase p

dP_{pni} is the potential difference of phase p between cells n and i, given by

$$dP_{pni} = P_{pn} - P_{pi} - \rho_{pni} G (D_n - D_i) \quad \text{Eq. 23.27}$$

or

$$dP_{pni} = P_n - P_i + P_{cpn} - P_{cpi} - \rho_{pni} g (D_n - D_i) \quad \text{Eq. 23.28}$$

where

P_{cp} is the capillary pressure for phase p

ρ_p is the mass density of phase p

g is the acceleration due to gravity

(0.0000981 in metric units, 0.00694 in field units and 0.000968 in lab units)

D is the cell center depth and

T_{ni} is the transmissibility between cells n and i.

The fluid mobilities M_p^c are evaluated in the upstream cell for each phase p separately (cell n if dP_{pni} is positive, cell i if dP_{pni} is negative). The upstream calculation applies separately for each phase (oil, water, gas) so that, for example, oil may flow from cell i to cell n while water flows from cell n to cell i.

The net flow rate of component c from cell i into neighboring cells is obtained by summing over all phases over the neighboring cells, $F_i^c = \sum_n F_{pni}^c$.

For a component c existing in both oil and gas phases with liquid and vapor mole fractions x_c and y_c , the component mobilities in oil and gas can then be written as

$$M_o^c = x_c k_{ro}(S_w, S_g) \frac{b_o^m}{\mu_o} \quad \text{Eq. 23.29}$$

and

$$M_g^c = y_c k_{rg}(S_g) \frac{b_g^m}{\mu_g} \quad \text{Eq. 23.30}$$

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

$$Q_{pi}^c = -T_{wi}(P_{pi} - H_{iw} - P_{bh}) M_{pi}^c \quad \text{Eq. 23.31}$$

where

T_{wi} is the well connection transmissibility factor

H is the hydrostatic head correction

P_{bh} is the bottom hole pressure

M_{pi}^c is the generalized mobility of component c in phase p

See "Well inflow performance" for further information.

Transport coefficients

The component mobilities can be modified in ECLIPSE 300 by including transport coefficients, [Ref. 11]. The transport coefficients α_p^c act as multipliers to the mobility. Thus equation 23.26 becomes:

$$F_{pni}^c \quad \text{Eq. 23.32}$$

where

$\alpha_p^c = \alpha_p^c(z^t)$ is the transport coefficient given by the [ALPHA](#) keyword

z^t is the mole fraction of component t defined by the [TRCOEF](#) keyword.

Transport coefficients for drainage and imbibition

When hysteresis is present it is possible in ECLIPSE 300 to modify the component mobilities depending on whether drainage (decreasing wetting phase saturation) or imbibition (increasing wetting phase saturation) is occurring. The component mobilities are modified by introducing transport coefficients which act as multipliers to the mobility. The transport coefficients for the drainage and imbibition process are specified separately and denoted by $\alpha_p^{c,d}$ and $\alpha_p^{c,i}$ respectively.

Thus equation 23.26 becomes:

$$M_p^c = \begin{cases} \alpha_p^{c,d} x_p^c k_{rp}^d \frac{b_p^m}{\mu_p} & \text{if drainage process} \\ \alpha_p^{c,i} x_p^c k_{rp}^i \frac{b_p^m}{\mu_p} & \text{if imbibition process} \end{cases}$$

where

$\alpha_p^{c,d} = \alpha_p^{c,d}(z^t)$ is the transport coefficient given by the [ALPHAD](#) keyword

$\alpha_p^{c,i} = \alpha_p^{c,i}(z^t)$ is the transport coefficient given by the [ALPHAI](#) keyword

z^t is the mole fraction of component t defined by the [TRCOEF](#) keyword.

Reservoir densities

ECLIPSE 100

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

$$\begin{aligned} \rho_o &= (\rho_{so} + CR_s \rho_{sg}) / B_o \\ \rho_g &= (C \rho_{sg} + R_v \rho_{so}) / B_g \end{aligned} \quad \text{Eq. 23.33}$$

with $C = 1$ for metric and lab units and $C = 178.1076$ in field units.

The reservoir density at the interface between cells n and i is computed as the average of the reservoir densities in cells n and i

$$\begin{aligned}\rho_{oni} &= (\rho_{on} + \rho_{oi})/2 \\ \rho_{gni} &= (\rho_{gn} + \rho_{gi})/2\end{aligned}\quad \text{Eq. 23.34}$$

ECLIPSE 300

In the compositional case, the reservoir densities are obtained from

$$\rho_p = MW_p b_p^m \quad \text{Eq. 23.35}$$

where

MW_p is the average molecular weight of the phase p .

The reservoir density at the interface between cells n and i is computed as the saturation weighted average of the reservoir densities in cells n and i

$$\rho_{pni} = \frac{(\rho_{pn} S_{pn} + \rho_{pi} S_{pi})}{(S_{pn} + S_{pi})} \quad \text{Eq. 23.36}$$

Newton iteration of the non-linear residual

Given the non-linear residual $R = R_{fl}$ and the solution variables X , we wish to solve the non-linear equation $R(X) = 0$. In each iteration of the non-linear equations the sequence of operations is:

- Given current solution X , obtain $R(X)$, then find a correction Δx such that

$$R(X + \Delta x) \sim R(X)J \cdot \Delta x = 0, \text{ where } J \text{ is the Jacobian } \frac{d}{dX}R(X).$$

- To do this, solve the linear equations

$$J \cdot \Delta x = R(X) \quad \text{Eq. 23.37}$$

or

$$\Delta x = J^{-1}R(X) \quad \text{Eq. 23.38}$$

- 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

$$\Delta x = J^{-1} \cdot R'_{fl} \quad \text{Eq. 23.39}$$

In the **IMPES** case, the matrix J^{-1} has a special structure, in that band terms only appear for pressure derivatives.

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

$$J'_D = \begin{matrix} P & S_w & m_1 & \dots & m_N \\ \text{Water} & x & x & 0 & \dots & 0 \\ \text{Compl} & x & 0 & x & \dots & 0 \\ \dots & \dots & \dots & \dots & \dots & \dots \\ \text{CompN}_c & x & 0 & 0 & \dots & x \\ \text{Volumebal} & x & x & x & \dots & x \end{matrix} \quad \text{Eq. 23.40}$$

$$J'_B = \begin{matrix} P & S_w & m_1 & \dots & m_N \\ \text{Water} & x & 0 & 0 & \dots & 0 \\ \text{Compl} & x & 0 & 0 & \dots & 0 \\ \dots & \dots & \dots & \dots & \dots & \dots \\ \text{CompN}_c & x & 0 & 0 & \dots & 0 \\ \text{Volumebal} & 0 & 0 & 0 & \dots & 0 \end{matrix} \quad \text{Eq. 23.41}$$

Thus there is one column, the diagonal, and the bottom row in the diagonal case, and one column only in the band case.

By subtracting a multiple of each line from the last, all terms in the molar balance equations $R_c = 0$ may be

zeroed except the first, $\frac{dR'_m}{dP}$. This is a banded pressure equation, which may be solved using the nested factorization linear solver (see "[Solution of the linear equations](#)").

The remaining elements of X may then be found by back-substitution.

ECLIPSE 100

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

$$J'_D = \begin{matrix} P & S_w & X_3 \\ \text{Water} & x & x & 0 \\ \text{Oil} & x & 0 & x \\ \text{Gas} & x & 0 & x \end{matrix} \quad \text{Eq. 23.42}$$

$$J'_B = \begin{matrix} P & S_w & X_3 \\ \text{Water} & x & 0 & 0 \\ \text{Oil} & x & 0 & 0 \\ \text{Gas} & x & 0 & 0 \end{matrix} \quad \text{Eq. 23.43}$$

24

Gas field operations model

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:

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

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 $DCQ \times$ 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) \times 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 operation 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 timestep, and can be output to the SUMMARY file.

Automatic compressor operation

ECLIPSE 100

When the gas field operations model is used in conjunction with the network option, gas compressors in the pipelines can be switched on and off automatically as required. The compressors may consume gas at a specified rate for power, which will be added to the gas consumption rate whenever the compressors are operating. The simplest option has two states: either “compression on” or “compression off”. A more versatile option switches on a series of compressors in a predefined sequence, and allows each compressor to have up to five levels of compression.

Description of the facilities

Seasonality profile generation

The engineer enters 12 profile factors, one for each month starting with January, using the keyword [SWINGFAC](#). The simulation is then advanced one contract period at a time, using the keyword [GASYEAR](#) or [GASPERIO](#) instead of [TIME](#), [TSTEP](#) or [DATES](#). The contract period is assumed to start on the current date at which GASYEAR or GASPERIO is read. ECLIPSE flags an error if this is not the first day of a month. During the contract period, the monthly sales gas production rate target for the field is set equal to the DCQ multiplied by the month’s profile factor. At the end of the contract period, ECLIPSE writes an output report and dump a RESTART file if required.

When the profile factor, and hence the production rate, changes from one month to the next, ECLIPSE automatically limits the timestep to the end of the month. Thus ECLIPSE models the different flow rate periods individually, rather than taking longer timesteps with an average rate. There is also an option to limit the timestep 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, timesteps 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 reset 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:

$$ACQ = 365.25 \times DCQ \quad \text{Eq. 24.1}$$

The number 365.25 is the average number of days in a year, with a leap year every fourth year.

In order to produce the required ACQ for a given DCQ, the profile factors should be normalized such that the sum over all months of {profile factor x number of days in month} = 365.25, with February allocated 28.25 days. If this "normalization requirement" is obeyed, then when leap years are taken into account every fourth year, the average annual production at a constant DCQ is DCQ x 365.25, which corresponds with the definition of ACQ given above.

When the [SWINGFAC](#) keyword is entered, ECLIPSE checks that the profile factors obey this normalization requirement, and issue a warning if they do not.

Operations and reports during the contract period

The keyword [GASYEAR](#) or [GASPERIO](#) advances the simulation through one or more complete contract periods. Well operations can be performed in the usual manner at the end of a contract period, and a printed report is generated at the end of each contract period depending on the [RPTSCHE](#)D 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 keyword 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 cannot 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 $DCQ \times SWING$ 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 $DCQ \times$ profile to $DCQ \times$ swing for an instant at any time in the contract year.”

During the first pass, the field is produced with a target rate of $DCQ \times PROFILE$ for the whole contract year. In addition, the delivery capacity is calculated and compared against a target of $DCQ \times SWING$. 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 \times swing factor for the previous timestep.)

This interpretation ensures the field is able to produce its ACQ (that is, DCQ \times profile for each month), and has the potential to increase its production rate to DCQ \times 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 \times SWING, but only until it has produced its ACQ ($= 365.25 \times DCQ$). ECLIPSE simulates through the contract year until the cumulative sum of {monthly swing factor \times number of days in the month} equals 365.25. When accumulating this sum, February is allocated 28.25 days. If the target of DCQ \times 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 \times 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 \times sum over months from start of contract year to end of specified month of {profile factor \times number of days in month}

\div sum over all months in the contract year of {profile factor \times 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 \times 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 \times profile to DCQ \times 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 \times 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 underestimate. 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 underestimates cannot be corrected. Thus, to obtain an accurate answer, the amount by which the DCQ can decrease from one iteration to the next should be constrained. You can supply a limiting reduction factor (item 5 in keyword [GASYEAR](#), item 6 in [GASPERIO](#)). If at the end of the first pass the DCQ has to be reduced to a fraction that is smaller than the limiting reduction factor, it is reduced only by the limiting reduction factor and the first pass is repeated again. The process continues until either the DCQ reduction lies within the limiting factor, or the maximum number of iterations has been reached ([GASYEAR](#) item 7, [GASPERIO](#) item 8) at which the DCQ reduction is no longer restricted. The nearer the reduction factor is to 1.0 the more accurate is the result, but at the expense of requiring more iterations of the first pass to converge it. Supplying an “anticipated reduction factor” ([GASYEAR](#) item 6, [GASPERIO](#) item 7) could help reduce the number of iterations required to converge the DCQ, by bringing the starting value nearer to the final value. Before the decline from plateau has begun, this factor is ignored. But once the DCQ has been reduced automatically below its initial value, the anticipated reduction factor is applied to the DCQ at the start of each contract period. Of course, you ought to have a reasonable idea of the rate of DCQ reduction before using this option. An anticipated reduction that is too fast results in unduly pessimistic DCQs, since ECLIPSE does not increase the DCQ automatically. A warning message is printed if the DCQ has not been reduced by more than the anticipated rate at the end of a contract period, as this would imply that the anticipated rate is too fast.

Control and output of sales gas rate

ECLIPSE 100

In ECLIPSE 100 the sales gas rate is defined as the **gross** gas production from the formation, **minus** the gas consumption, **plus** any gas imports. Gas can be consumed or imported at a specified rate at any group, as

requested using the keyword [GCONSUMP](#). Gas can also be consumed by an automatic compressor when it is operating (see "Automatic compressors").

ECLIPSE 300

In ECLIPSE 300 the sales gas rate is defined as the **gross** gas production from the formation, **minus** the gas consumption. Gas can be consumed as fuel using the keyword [GRUPFUEL](#). Gas may only be imported via a gas producing satellite group (see keyword [GSATPROD](#)). Note that advanced and make-up gas (see keywords [GADVANCE](#) and [GINJGAS](#)) is only available for reinjection 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 keywords:

FGSR Field Gas Sales Rate

GGSR Group Gas Sales Rate

FGST Field Gas Sales Total (cumulative)

GGST Group Gas Sales Total (cumulative)

The keywords 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 reinjecting the surplus gas. It is not appropriate to do this when running the gas field operation model.

Estimation of delivery capacity

At the start of each timestep, 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 [GASFDEL](#)) 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 timestep after the first entry of the [SWINGFAC](#) keyword (which effectively turns this facility on), irrespective of whether [GASYEAR](#), [GASPERIO](#), [DATES](#), [TIME](#) or [TSTEP](#) is used to advance the simulation. When the option for automatic DCQ adjustment is in use (see "Swing factor and DCQ adjustment") 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 keywords FGDC, GGDC and WGDC.

The delivery capacity calculated at each timestep is not exact; in fact it is somewhat optimistic. There are two reasons for this. Firstly, it is calculated at the start of the timestep, 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 timestep. Thus at the end of the timestep (when the delivery capacity is in fact reported) it is actually smaller than the value calculated at the start of the timestep. 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 buildup 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 timestep: 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 timestep.

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.
- When using the network option, the delivery capacity is calculated with the network balanced at the capacity flow rates. The potential is calculated using the THP limits obtained from the nodal pressures of the network balanced at the required offtake rate. Thus the potentials do not account for the increased wellhead pressures that would result when the flows increase to capacity.

Automatic compressors

ECLIPSE 100

The network option can be used with the gas field operations model to take account of the pipeline pressure losses in the gathering system (see "[Network option](#)"). 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 operation 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 keyword GMCPL that writes the current compression level of the compressor in each group's pipeline to the SUMMARY file. The compression level is represented in the SUMMARY file by a number between 0 (for no compression) and 5 (for the highest allowed level).

The extended compressor option requires a small amount of extra storage, which must be reserved by setting the first switch in the [GASFIELD](#) keyword in the RUNSPEC section to YES.

Specifying production rates for the second pass using GASFTARG and GASFDECR

ECLIPSE 100

The [GASFTARG](#) keyword can be used to specify a rate at which the field is to be produced during the second pass of the Gas Field Operations Model, independent of the calculated DCQ.

The GASFTARG keyword is used in conjunction with [SWINGFAC](#) keyword. During the first pass of the model the DCQ will be calculated using the swing factors provided. However, on the second pass the field is produced at the lesser of $DCQ \times PROFILE$ and the rate specified in the GASFTARG keyword. For months defaulted in the GASFTARG keyword the field is produced according to the PROFILE factors.

The [GASFDECR](#) keyword enables the field gas production rate for the second pass to be adjusted through monthly decrements. On the second pass the field is then produced at $DCQ \times PROFILE - DECREMENT$.

The GASFTARG and GASFDECR keywords are restricted to single contract simulations.

Applying separate gas supply contracts to individual groups

The gas field operations model by default applies a single set of swing factors and profile factors to the FIELD group at the top of the grouping hierarchy. There is, however, an option to allow individual groups to have their own independent gas supply contracts. Each "contract group" has its own set of swing factors and profile factors, and its own DCQ which ECLIPSE can adjust independently to meet its swing requirement.

Defining multiple contract groups

ECLIPSE 100

In order to use the multiple contract groups facility, the flag ' YES ' must be set in item 1 of the RUNSPEC section keyword [GASFIELD](#). This reserves the extra storage needed by various extensions of the gas field operations model. This step is not necessary in ECLIPSE 300.

To define multiple contract groups, use the keyword [GSWINGF](#) instead of the [SWINGFAC](#) keyword to activate the gas field operations model. The GSWINGF keyword is used to identify the contract groups and to give each one its own set of swing and profile factors. (In comparison, the SWINGFAC keyword reads in a single set of swing and profile factors, and applies it to the FIELD group.)

The set of contract groups must obey the following restrictions:

- One contract group must not be subordinate to another contract group.
- The set of contract groups must cover the whole field, that is, every open production well must be subordinate to a contract group.
- There must be no gas consumption or, for ECLIPSE 100, import at any node downstream of a contract group (otherwise the contract group's sales gas targets would not be set correctly).

Simulating with multiple contract groups

The keyword [GASYEAR](#) or [GASPERIO](#) is still used to advance the simulator through an integral number of contract periods. ECLIPSE assumes that all contract groups start their contract periods at the same time, and the contract period length is the same for each group. However, each contract group generally has a different DCQ. So item 2 in keyword [GASYEAR](#) or item 3 in [GASPERIO](#) (the initial DCQ) is ignored, and instead the initial DCQ of each contract group must be entered with the keyword [GDCQ](#). The remaining items in keyword [GASYEAR](#) or [GASPERIO](#) are applied to all contract groups.

When you set the initial DCQ of each contract group with keyword [GDCQ](#), you should state whether the DCQ is variable or must remain fixed at the specified value. Variable DCQs are adjusted automatically as necessary to obey the group's swing requirement, depending on the option selected in item 3 of keyword [GASYEAR](#) or item 4 of [GASPERIO](#). Fixed DCQs remain at their specified values until reset in a subsequent [GDCQ](#) keyword.

Fixed-DCQ contract groups

If gas produced by other operators is fed into the same network that you use, you may wish to determine the DCQ which your group is able to supply against assumed production profiles from the other operators' groups. In this situation, you could model the other operators' groups as fixed-DCQ contract groups, and your own group as a variable-DCQ contract group. The group DCQ values can be written to the [SUMMARY](#) file with the [SUMMARY](#) section mnemonic [GGDCQ](#) (Group Gas DCQ).

When there is only one variable-DCQ group in the problem, the simulation process is exactly the same as for the single contract group case described in Section 2, except that the swing requirement and DCQ adjustment is applied to the variable DCQ group instead of the FIELD, and gas from the other fixed-DCQ groups also enters the network. All the options in item 3 of the [GASYEAR](#) keyword or item 4 of [GASPERIO](#) are available for the variable-DCQ group.

The flow targets applied to the fixed-DCQ groups are equivalent to the target rate applied to the variable-DCQ group. In the first pass of the contract period each group has a target rate equal to its $DCQ \times$ Swing Factor; or its $DCQ \times$ Profile Factor if the 'PRO' option is selected. In the second pass of the contract period, the flow target is its $DCQ \times$ 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 $DCQ \times$ 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 $DCQ \times$ 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 keyword 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 $DCQ \times$ 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 $DCQ \times$ swing factor for the whole contract period in the first pass

- NO

For no automatic DCQ reduction.

The other options (ACQ and PRO for example) 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 $DCQ \times$ Profile.

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 operation model

RUNSPEC section

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 shown in the following table.

E100	E300	Field	Group	Description
x	x	FGSR	GGSR	Field/Group Gas Sales Rate
x	x	FGST	GGST	Field/Group Gas Sales Total (cumulative)
x	x	FGDC	GGDC	Field/Group Gas Delivery Capacity
x	x	FGDCQ	GGDCQ	Field/Group Gas DCQ
x			GMCPL	Group Multi-level Compressor Level.
x	x		GPR	Group or node PPressure in network
x			GPRDC	Group or node PPressure in network at Delivery Capacity
x			GPRFP	Group or node PPressure in network from end of First Pass
x			GGPRNBFP	Gas flow rate along Group's or node's outlet branch in network, from end of First Pass.

The other useful gas field operations keywords are:

E100	E300	Keyword	Description
x	x	NGOPAS	Number of iterations to converge DCQ in first pass
x		WGPRFP	Well Gas Production Rate from end of First Pass
x		WTHPFP	Well Tubing Head Pressure from end of First Pass.
x		WBHPFP	Well Bottom Hole Pressure from end of First Pass.

Remember that the keywords FGPR, GGPR, FGPT and GGPT refer to the **gross** gas produced from the formation, without allowing for consumption. Note also that the mnemonics WGPRFP, WTHPFP, WBHPFP, GPRFP and GGPRNBFP do not refer to continuous quantities but rather the values of these quantities at the end of the first pass.

Specifying the wells

Specify the wells in the usual way with `WELSPECS`, `COMPDAT` and `WCONPROD` (and `WCONINJE` for any water injectors). The optional keywords (for example `WECON`, `WEFAC`) are not mentioned explicitly in this example.

If there is a multilevel 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 operation model is in use, the network is balanced at the beginning of every timestep.

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  /
/
```

ECLIPSE 300

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 timestep, and any automatic compressors are turned on and off as required.

If you wish to disable all rate limits for wells and groups when calculating the delivery capacity, the keyword [GASFDELC](#) must be used to indicate this requirement. The delivery capacity is then the sum of the well production rates when operating at their BHP or THP limits, minus any gas consumption.

```
GASFDELC
2 /
/
```

To use features 1 and 2, that is, seasonality profile generation and DCQ adjustment, the keyword [GASYEAR](#) or [GASPERIO](#) must be used instead of [TIME](#), [TSTEP](#) or [DATES](#). This advances the simulation through an

integral number of contract periods, starting at the current date (which must be the first day of a month). GASYEAR may be used if the contract period is a year, otherwise GASPERIO must be used. At the end of each contract period, reports are printed and restarts are written as requested with RPTSCHED or RPTPRINT. Reports can also be printed during the contract period if an Annual Scheduling File has been set up requesting them. For example, to request a printed report for the 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 timestep 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' /
```

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 timestep as usual (only during the second pass, of course, if the automatic DCQ adjustment option is on).

The best way to plot the results in GRAF is in the form of histograms. Figure 24.1 shows the results of the example discussed above. The contract year starts on 1st September, and the initial DCQ is 150,000 Mscf/day. With this DCQ, the annual production profile fits inside the delivery capacity decline curve for the first two years. During the third year, however, the field cannot deliver at the winter peak rate. ECLIPSE has to reduce the DCQ to 96,000 Mscf/day to obey the swing factor requirement. For the fourth year, the DCQ is reduced to 59,000 Mscf/day. The DCQ reduction in the third and fourth years turns out to be very substantial, and for greater accuracy the engineer would be recommended to apply a DCQ reduction limit or use the accelerated iteration scheme with a suitable convergence tolerance.

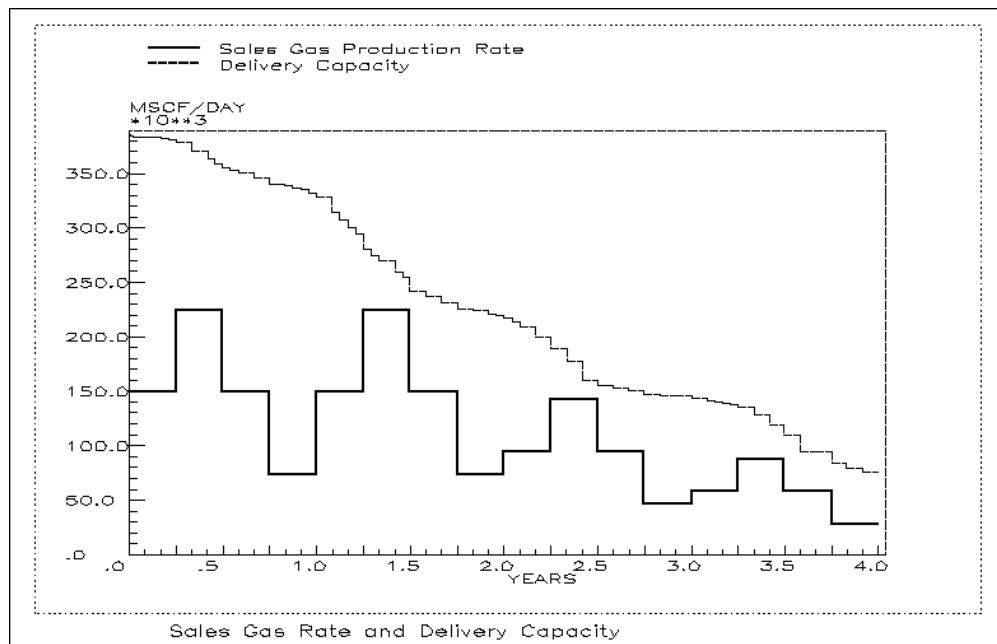


Figure 24.1. Sales Gas Production Rate and Delivery Capacity

Restrictions

- Do not use the keywords **GCONSOLE**, **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](#)”) 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

Restriction	Keyword	Description
E100 only	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

Refer to [Gas field operations SUMMARY keywords](#) in the *ECLIPSE Reference Manual*.

SCHEDULE

Restriction	Keyword	Description
E100 only	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
Required only if there is an annual scheduling file.	GASEND	Denotes end of keywords to go into the annual scheduling file.
E100 only	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 "), automatic compressors must be defined with keyword NETCOMPA instead.
	GASFDEL	Specifies the definition of the delivery capacity. If this keyword is omitted, the default definition is used, in which all group and well limits are honored except those on the group and field gas production rate.
	GASMONTH	Denotes the times within each year that operations on the annual scheduling file should be performed, and requests printed reports. Omit this keyword if there is no annual scheduling file (see GASBEGIN and GASEND).
	GASPERIO	Advances the simulation through one or more complete contract periods, with the field sales gas target set equal to the DCQ multiplied by the monthly Profile Factor. Offers the option of automatic DCQ reduction to obey the Swing Requirement. TIME , TSTEP or DATES may be used instead, but the field target rate would have to be set manually each month and there would be no option of automatic DCQ reduction.

Restriction	Keyword	Description
	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.
E100 only	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
E100 only	NETCOMPA	Defines automatic compressors in the extended network model. use this instead of GASFCOMP if you are using the extended network mode
E100 only	OPTIONS	Item 34 prevents wells being drilled automatically in the second 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 second pass production target.
E100 only	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: <ul style="list-style-type: none"> • Calculation of the delivery capacity • Automatic compressor switching • Control of sales gas rate instead of gross production rate.
E100 only	GASFDTARG	Specifies a gas rate at which the field is to be produced in the second pass of the gas field operations model.
E100 only	GASFDECR	Specifies a decrement to the gas rate at which the field is to be produced in the second pass of the gas field operations model.

25

Gas lift optimization

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

The gas lift optimization facility is a special extension to both ECLIPSE 100 and ECLIPSE 300. It determines how much lift gas to allocate to each well in order to meet well, group or field production targets. If production targets cannot be met, it determines how to make the best use of the existing lift gas resources by allocating lift gas preferentially to the wells that can make the best use of it.

The gas lift optimization facility can solve the following problems:

- 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?

ECLIPSE 100

- Optimizing gas lift within a simple network

If the wells flow into a subsea 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 timestep, 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

For a well under THP control, as the supply of lift gas to it is gradually increased, the production rate at first increases due to the reduced density of the mixture in the tubing. However, as the lift gas supply is

increased further, friction pressure losses in the tubing become more important, and the production rate peaks and then starts to decrease (see figure 25.1). For an individual well with no constraints other than a THP limit, with an unlimited free supply of lift gas, the optimum lift gas injection rate is the value at the peak (point A).

In reality, of course, lift gas injection is never free. Compression costs can be expressed as a cost per unit rate of lift gas injection (for example, dollars/day per Mscf/day). This must be balanced against the value of the extra amount of oil produced. Thus there is a “minimum economic gradient” of oil production rate versus lift gas injection rate, at which the value of the extra amount of oil produced by a small increase in the lift gas injection rate is equal to the cost of supplying the extra amount of lift gas. The optimum lift gas injection rate is then somewhat lower than the peak value, at the point on the curve where its gradient equals the minimum economic gradient (point B).

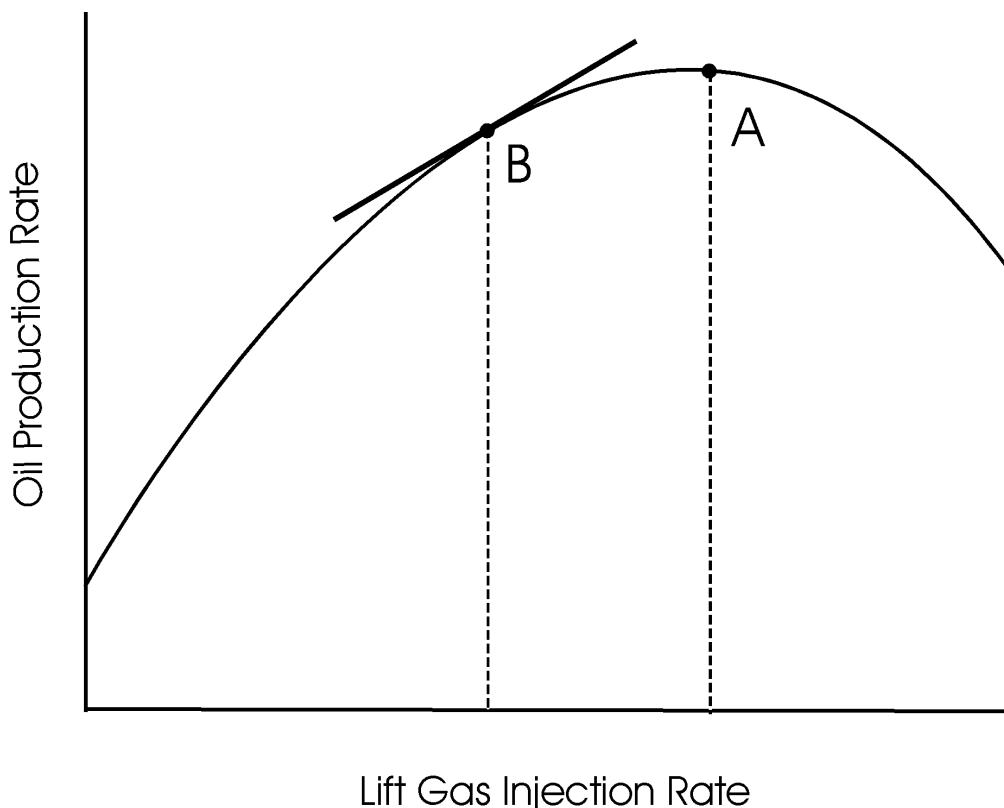


Figure 25.1. Optimum Lift Gas Injection Rate

For flexibility, each well can be assigned 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 than that indicated by the minimum economic gradient. For example, if the well is subject to 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. This is accomplished using the keyword `LIFTOPT`. A typical increment size could be 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. (In general, however, the computing time for gas lift optimization is only an important consideration when the Network option is being used.)

Each time ECLIPSE performs a gas lift optimization, it must decide whether lift gas increments must be added to, or subtracted from, each well. Taking the current lift gas injection rate of a well, ECLIPSE calculates what the well would produce if the lift gas injection rate were increased by one increment, and also what it would produce if the rate were reduced by one increment. The production rates are adjusted if necessary to obey any rate or BHP limits that the well may be subject to. From this information, ECLIPSE calculates the well's "weighted incremental gradient" and its "weighted decremental gradient" according to the formula:

$$\text{Gradient} = \left\{ \frac{\alpha_w \cdot \Delta Q_o}{(\Delta Q_{gli} + \alpha_g \cdot \Delta Q_g)} \right\} \quad \text{Eq. 25.1}$$

where:

α_w is the weighting factor for the preferential allocation of lift gas.

α_g is the gas production rate weighting factor.

ΔQ_o is the increment or decrement in oil production rate.

ΔQ_g is the increment or decrement in gas production rate.

ΔQ_{gli} is the gas lift increment or decrement.

By default, the gas production rate weighting factor α_g is defined to be 0.0 such that, in general, the gradients are calculated according to the reduced formula:

$$\text{Gradient} = \left\{ \frac{\alpha_w \cdot \Delta Q_o}{\Delta Q_{gli}} \right\} \quad \text{Eq. 25.2}$$

However, a positive value of α_g may be employed to discourage the allocation of lift gas to wells which are producing more gas. The gas rate term in the denominator is subject to the constraint $\alpha_g \cdot \Delta Q_g \geq 0.0$. If $\alpha_g \cdot \Delta Q_g \gg \Delta Q_{gli}$ the gradient becomes essentially an inverse GOR term. The weighting factors α_w and α_g are specified via items 4 and 6 respectively of the [WLIFTOPT](#) keyword.

These gradients are updated each time an increment is added or removed. ECLIPSE adds lift gas increments to the well as long as its weighted incremental gradient exceeds the minimum economic gradient, subject to any limits on the lift gas supply.

Optimizing gas lift within a group

If a group has any production rate constraints, and/or a limit on its total rate of lift gas supply, ECLIPSE allocates lift gas preferentially to the wells that gain the most benefit from it. Lift gas increments are allocated in turn to the well that currently has the largest weighted incremental gradient. The procedure takes account of any limits on the group production rate or lift gas supply applied to any level of group, including the FIELD level group.

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 allocate 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, at which point any surplus lift gas increments are removed from the group. The group has surplus lift gas if it exceeds any production rate limits or a lift gas supply limit, or contains any wells that have a weighted decremental gradient less than the minimum economic gradient. Lift gas increments are removed in turn from the well that currently has the smallest weighted decremental gradient, until there is no surplus lift gas in the group.

The final stage of the optimization process involves looking to see whether any lift gas increments could profitably be added anywhere in the field. ECLIPSE looks for the well with the largest weighted incremental gradient that does not exceed its own lift gas supply limit, and does not belong to a group that exceeds a rate limit or a lift gas supply limit. (Note that this means that, in general, no further lift gas will be allocated once a field rate target is achieved. However, selected wells may be allocated further lift gas in order to continue optimizing the oil rate if item 7 of the [WLIFTOPT](#) keyword is set to YES, for wells which belong to groups which are under gas target control.) If the well's weighted incremental gradient exceeds the minimum economic gradient, it receives another increment of lift gas. The process continues until no well can profitably receive any more lift gas increments.

The whole optimization process is efficient because it starts from the previous optimized distribution of lift gas. If the previous lift gas distribution is still the optimum, then no lift gas increments need be added or subtracted. This is more efficient than building the lift gas distribution up from zero at each timestep.

If wells have efficiency factors to take account of regular downtime (see keyword [WEFAC](#)), their lift gas injection rates are multiplied by their efficiency factors when they are added to the group lift gas supply rate. This is consistent with the summation of flow rates for wells with downtime, and preserves the ratio of production rate to lift gas injection rate.

If groups have efficiency factors to model synchronized downtime of their subordinate wells (see keyword [GEFAC](#)), their lift gas injection rates are multiplied by their efficiency factors when they are added to the lift gas supply rate of the parent group.

Optimizing gas lift within a network

ECLIPSE 100

The problem becomes much more complicated when the group belongs to a production network, with a flowline to a higher level group (see "[Network option](#)"). Here, the wells' THP limits increase as the group's production rate increases. Adding a small amount of 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 is required to flow through the flowline together 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 subsea manifold connected to the surface by a flowline. The choice of increment size is important also. A small increment size increases the required computation time because 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 magnitude of the resulting change in the flow rate is comparable with the specified tolerance in the network balancing iterations. It is recommended that the [NETBALAN](#) keyword is used to decrease the network convergence tolerance by a factor of 10 (item 2), and to increase the number of 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, to retain the well as a candidate for gas lift, if required, with the possibility of allocating lift gas to it at a later time if it becomes economical to do so, 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 will die 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 simply to initiate flow whereas five increments would provide the best ratio of oil production to lift gas injection (see figure 25.2). The well's incremental gradient would be set equal to this ratio (line O - A), to be compared with the incremental and decremental gradients of the other wells and, if this gradient is large enough, the well is given five increments of lift gas to start it flowing (position A).

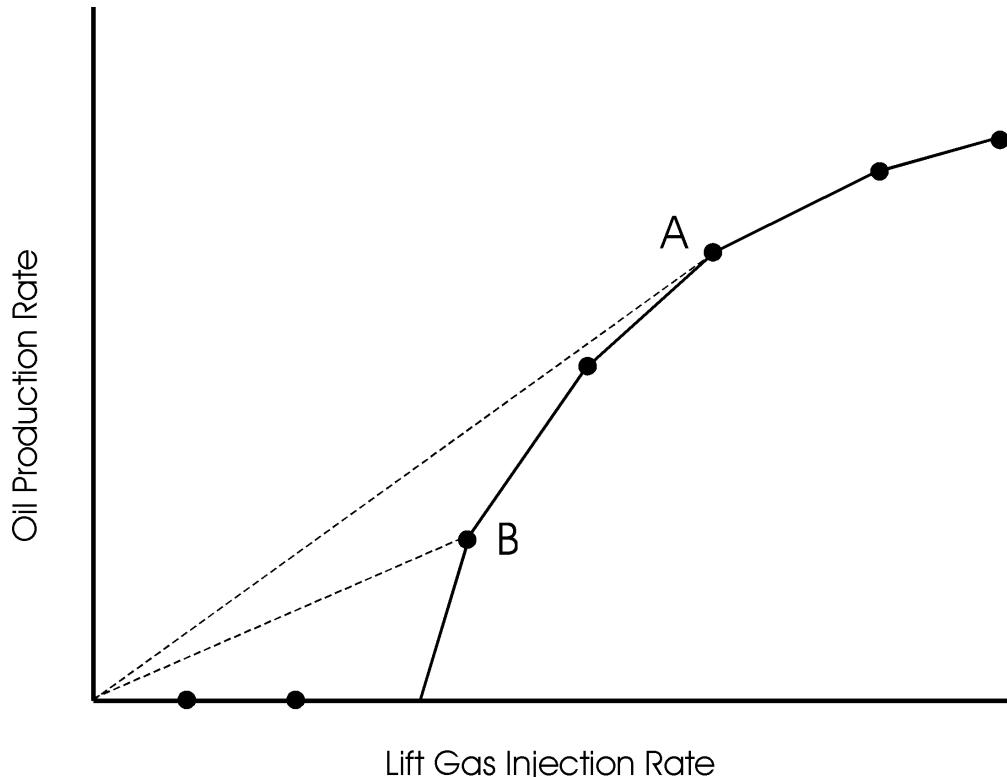


Figure 25.2. Incremental gradient calculation

Thereafter the well is treated as a normal well for optimization, with single increments being added or subtracted as necessary. However, if the well is only just able to flow at a particular rate of gas lift, that is, if the removal of another increment would cause the well to die, the decremental gradient is calculated in a special way. In the current example the well can flow with three lift gas increments, but not with two. If one increment is subtracted the well does not flow but it has two remaining increments allocated which are wasted. Hence if the lift gas rate is decreased it is expedient to 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 level group) 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, to allocate lift gas to keep a well flowing, if required, regardless of whether lift gas is needed to achieve a group or field production rate target, assign a weighting factor greater than 1.0 (for example 1.01) together with a negative value for the minimum lift gas rate.

Note: There is a possible conflict when an economic limit is applied to a well (for example using the [WECON](#) keyword) that has been instructed to be kept flowing in the manner described above using gas lift optimization. When the well's flow becomes too low, it may be shut and hence will not be available to take part in the allocation of gas lift. Take care to avoid conflicting economic limits if a well is to be kept alive by means of gas lift optimization.

Using the gas lift optimization facility

The VFP tables (keyword [VFP_iPROD](#)) 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 "VFP_i User Guide" for more details.

Gas lift effects are modeled by interpolating the VFP table with an ALQ value equal to the current lift gas injection rate. The ALQ values in each table must span the expected range of lift gas injection rates for the well, as extrapolation of the tables can give unrealistic behavior.

ECLIPSE 100

In ECLIPSE 100, lift gas injection rates lying in between tabulated ALQ values are handled by linear interpolation, by default, like the other parameters in the table. However, there is an option to apply cubic spline interpolation to the ALQ values. This has the advantage of making the gradient of well performance with respect to the lift gas injection rate vary smoothly across the range of ALQ values. With linear interpolation, the gradient can change abruptly at the tabulated ALQ points, which may bias the optimized lift gas distribution towards these values. Spline interpolation is enabled with the keyword [VFPTABL](#).

LIFTOPT keyword

The gas lift optimization facility is activated with the keyword [LIFTOPT](#), where the lift gas increment size and the minimum economic gradient are specified. The keyword can also be used to adjust the frequency with which the gas lift is optimized. The default is every timestep. 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 timestep, 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 would otherwise die without gas lift. For wells to receive their minimum lift gas allocation even if no gas lift is necessary to achieve a group or field production rate target, if required, set their weighting factors to a value greater than 1.0. A weighted gas production rate term may be included in the denominator of the gradient calculation by setting the gas rate weighting factor to a non-zero value and, if required, selected wells may be designated to continue receiving lift gas after a gas rate group target has been achieved in order to continue optimization of the oil rate.

GLIFTOPT keyword

The keyword [GLIFTOPT](#) can optionally be used to set limits on the lift gas supply and the total gas rate (produced gas + lift gas) for individual groups. Remember that the lift gas supply to a group equals the sum of the lift gas injection rates of its subordinate wells or groups multiplied by their efficiency factors.

Group production rate limits

Group production rate limits are set with either keyword [GCONPROD](#) or [GCONPRI](#) as usual, depending on which method of group control is required. (Note that any gas rate limits set with these keywords refer to gas produced from the reservoir, and do not include the lift gas.) Lift gas is allocated if a group or the field cannot reach its oil production rate target without it. Subject to any limits on the lift gas supply or total gas rate set in keyword [GLIFTOPT](#), increments are added until the oil production rate target is just exceeded and 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 timestep, the reservoir conditions may change sufficiently over the timestep such 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 timestep may fall somewhat short of this by the end of the timestep. Conversely, a group's water rate limit honored in the first iteration may be exceeded by the final iteration of the timestep. 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 requires the rate limit to be honored by the gas lift optimization process. However, the rate limits are honored more exactly if optimizations are performed at each of the first [NUPCOL](#) iterations of every timestep; this is permissible provided that any network present is not balanced less frequently.

Use with group production rules

ECLIPSE 100

In ECLIPSE 100, gas lift optimization can be used in combination with the group production rules (see "[Group Production Rules](#)" 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 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 timestep.

Use with the network option

gas lift optimization can be used with standard or extended networks in ECLIPSE 100 (see "[Network Option](#)"). When there is a production network, gas lift can only be optimized when the network is being balanced. If the network is only balanced in the first iteration of the timestep, gas lift can only be optimized for conditions at the beginning of each timestep. However, if the network is balanced in each of the first `NUPCOL` Newton iterations of every timestep (by setting a negative balancing interval in keyword `NETBALAN`), then the gas lift distribution can be optimized with this frequency also. Note that the computation time taken by each optimization is much greater if a network is present and increases sharply with the square of the number of wells in the simulation. In order to reduce the computation time, it is recommended that item 2 (convergence tolerance) of the `NETBALAN` keyword is decreased by an order of magnitude and item 3 (maximum number of iterations) is increased.

When setting up the network with keyword `GRUPNET`, item 6 can be used 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 one 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 set to zero. It is important to ensure that the range of gas fraction (GFR) values in the branch's VFP table encompasses the anticipated range of operating lift gas rates and produced gas rates in order to avoid the need for extrapolation.
- 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, it is important to ensure that the range of `ALQ` values in the VFP table encompasses the anticipated operating range in order to avoid 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 that are applied to each well, group and the entire field are output as part of the production report at each reporting time. These rates can also be written to the `SUMMARY` file by specifying the `SUMMARY` section keywords `WGLIR`, `GGLIR` and `FGLIR` (Well/Group/Field Gas Lift Injection Rate).

The incremental gradients of selected wells (that is, the increase in oil production rate achieved by allocating an additional increment of gas lift to the well, divided by the sum of increment size and a weighted gas production rate term) can also be written to the `SUMMARY` file by using the keyword `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 nor as flexible as the gas lift optimization option. The equivalent keywords for gas lift optimization are `GLIFTOPT` and `WLIFTOPT`.

Gas Lift Optimization is not compatible with networks in ECLIPSE 300 (see "[Network option](#)"), and will be disabled if a standard or extended network is detected.

The flux boundary conditions facility should not be used with gas lift optimization, as no account will be taken of lift gas supplied to wells outside the flux boundary.

Summary of keywords

SUMMARY

Refer to [Well and group gas flows](#) in the *ECLIPSE Reference Manual*.

Note: In the presence of gas lift optimization, the produced gas reported (for example using the FGPR, GGPR and WGPR summary section keywords) does not include produced lift gas, only produced formation gas.

SCHEDULE

Restriction	Keyword	Description
	GLIFTOPT	Sets limits on group lift gas supply.
	LIFTOPT	Activates gas lift optimization and sets some governing parameters.
E100 only	VFPPROD	Inputs VFP tables.
E100 only	VFPTABL	Selects the method of interpolating the ALQ variable in VFP tables.
	WLIFTOPT	Sets gas lift optimization data for wells.

26

Gas plants and NGLS

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

A gas plant table may be used instead of a flash calculation for a separator stage (see item 9 of keyword [SEPCOND](#)). In almost all cases, a single stage only will be required. Gas plant tables are defined with keyword [GPTABLE](#), [GPTABLEN](#) or [GPTABLE3](#). The table defined with GPTABLE specifies the oil recovery fraction (the number of moles of liquid output per mole of feed) for each component as a function of the ‘heavy’ mole fraction of the feed. The components that comprise the heavy fraction are also defined in the table.

The GPTABLEN keyword should be used instead of GPTABLE if NGL (natural gas liquid) output is also required. The table defined with GPTABLEN specifies both the oil recovery fractions and the NGL recovery fractions for each component as a function of the heavy mole fraction of the feed. The [DNGL](#) keyword should also be used to set the partial densities in the NGL stream.

The GPTABLE3 keyword defines a table with recovery fractions for oil, NGL and gas for each component as a function of the heavy mole fraction of the feed. Use this keyword instead of GPTABLE or GPTABLEN if it is required that, for one or more components, the sum of the oil, NGL and gas recovery fractions is not unity. You can use this keyword to reduce or exclude one or more components in the post separator oil, NGL or gas streams. The DNGL keyword should also be used to set the partial densities in the NGL stream.

The way in which the gas plant table splits the input streams is described below:

Suppose the input composition to the gas plant is z , and the oil, NGL and gas recovery factors are R_o , R_{NGL} and R_g . Then the moles of each component in the oil, NGL and gas streams are:

$$m_{o_i} = R_{o_i} z_i \quad \text{Eq. 26.1}$$

$$m_{\text{NGL}_i} = R_{\text{NGL}_i} z_i \quad \text{Eq. 26.2}$$

$$m_{g_i} = R_{g_i} z_i \quad \text{Eq. 26.3}$$

Where R_g is either taken directly from GPTABLE3, or calculated from the requirement that the factors sum to unity when using GPTABLE or GPTABLEN.

To obtain the oil, NGL and gas compositions, the streams are normalized:

$$x_i = \frac{m_{o_i}}{\sum m_{o_i}} \quad \text{Eq. 26.4}$$

$$n_i = \frac{m_{\text{NGL}_i}}{\sum m_{\text{NGL}_i}} \quad \text{Eq. 26.5}$$

$$y_i = \frac{m_{g_i}}{\sum m_{g_i}} \quad \text{Eq. 26.6}$$

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

$$L = \sum_i m_{o_i} \quad \text{Eq. 26.7}$$

$$N = \sum_i m_{\text{NGL}_i} \quad \text{Eq. 26.8}$$

$$V = \sum_i m_{g_i} \quad \text{Eq. 26.9}$$

In order to define a volume rate of NGLs, the molar density must be obtained. For this we use Amagat's law which states that the mixture molar volume is the mole-fraction weighted sum of the component molar volumes:

$$V_{\text{mix}} = \sum_i n_i V_i \quad \text{Eq. 26.10}$$

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

$$V_i = \frac{Mw_i}{\rho_i} \quad \text{Eq. 26.11}$$

Production control

Production wells may be made subject to NGL production limits or set NGL production rate targets using the [WCONPROD](#), [WCONHIST](#) and [WELTARG](#) keywords.

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 composed 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 reinjection.

The volume of NGL produced is obtained using Amagat's law, adding the 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:

FNRR NGL recovery volume rate

FNRT NGL recovery volume total

FNRM NGL recovery molar rate

FGRR gas remaining volume rate

FGRT gas remaining volume total

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'.

27

GASSOL solution in the aqueous phase

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The GASSOL solution algorithm allows a specified number of gas components to dissolve in the aqueous phase. This option is selected using the [GASSOL](#) keyword in the RUNSPEC section. The basic model consists of fugacity functions for dissolving hydrocarbons in the aqueous phase. These are constructed to match the supplied solubility data and take the form:

$$f_i^A = x_i^A P \varphi_i^A(P, T) \quad \text{Eq. 27.1}$$

where x_i^A is the saturated mole fraction from the solubility data for component i and $\varphi_i^A(P, T)$ is the fugacity coefficient for component i in the aqueous phase at pressure P and temperature T . The phase equilibrium between the aqueous hydrocarbons and the hydrocarbon phases is defined by the condition that the fugacity values are equal. Using the equation of state with a sample pure “gas” composition, the corresponding vapor fugacity f_i^V is computed. The aqueous fugacity coefficient for component i is then given by:

$$\varphi_i^A(P, T) = \frac{f_i^A}{P x_i^A} \quad \text{Eq. 27.2}$$

It is also possible to construct the aqueous hydrocarbon component Gibbs energy contributions.

The solubility properties of the hydrocarbon components in the aqueous phase may be entered as a function of pressure using the [SOLUAQA](#) keyword. The data for one or more soluble components may be defaulted, whereupon a default set of solubilities is generated internally as a function of pressure. It is strongly recommended, however, that appropriate values are supplied via the [SOLUAQA](#) keyword to ensure sufficient accuracy in the modeling of the solubility behavior.

The solubility data used in the simulation, which may be a combination of input and default values, can be reported by via the SOLU mnemonic of the [RPTPROPS](#) keyword.

The initial dissolved hydrocarbons solubility may be specified as a function of depth using the [RSWVD](#) keyword.

Notes

- The initial dissolved hydrocarbon concentrations may not be in exact equilibrium with the hydrocarbon oil and gas phases. In addition, the initial flash may also modify the input values slightly.
- The [SOLUAQA](#) keyword employs two records per component to specify the solubility data. The first record consists simply of the component number, which should correspond with numerical index of the component in the sequence of components defined by the [CNAMES](#) keyword. The second record consists of a table of solubility as a function of pressure.
- When using the [SOLUAQA](#) keyword, the solubilities of one or more dissolving hydrocarbon components may optionally be defaulted. In such cases, a default solubility table is generated by applying the GASWAT framework with the component properties specified in the PROPS section. The calculations assume a feed composition comprising an equal split between water and the component whose properties are being calculated for a range of pressures that is defined internally.
- If the solubilities for all of the dissolving hydrocarbon components are defaulted, the default tables will consist of twenty pressure nodes per table. If the solubilities for only some of the dissolving hydrocarbon components are defaulted, the number of pressure nodes in the defaulted tables will be equal to the maximum number of pressure nodes found in the non-defaulted tables. It is strongly recommended that appropriate input values are supplied rather than relying upon the internally generated default values in order to ensure sufficient accuracy in the modeling of the aqueous behavior of the hydrocarbons.

Aqueous phase properties

Once the aqueous hydrocarbon mole fractions have been determined as described above the water density and viscosity can be calculated using Ezrokhi's method [Ref. 133] together with the density and viscosity coefficients specified using the [DENAQA](#) and the [VISCAQA](#) keywords respectively.

Density

The water density ρ is calculated using Ezrokhi's method [Ref. 133] and a correlation of the form:

$$\log 10(\rho) = \log 10(\rho_0(P, T)) + \sum_i A_i(T) w_i \quad \text{Eq. 27.3}$$

where

w_i is the weight fraction of the non-water component i and

$\rho_0(P, T)$ is the density of pure water.

The coefficients $A_i(T)$ are given by:

$$A_i(T) = a_{0,i} + a_{1,i} T + a_{2,i} T^2 \quad \text{Eq. 27.4}$$

where the temperature T is specified in Celsius.

The three regression coefficients for each component $a_{0,i}$, $a_{1,i}$ and $a_{2,i}$ are specified by the [DENAQA](#) keyword.

The reservoir density of aqueous phase can be output using the mnemonic DENW with the [RPTRST](#) or [RPTSQL](#) keywords.

Viscosity

The water viscosity μ is calculated using Ezrokhi's method [Ref. 133] and a correlation of the form:

$$\log 10(\mu) = \log 10(\mu_0(P, T)) + \sum_i B_i(T) w_i \quad \text{Eq. 27.5}$$

where

w_i is the weight fraction of the non-water component i and

$\mu_0(P, T)$ is the viscosity of pure water.

The coefficients $B_i(T)$ are given by:

$$B_i(T) = b_{0,i} + b_{1,i} T + b_{2,i} T^2 \quad \text{Eq. 27.6}$$

where the temperature T is specified in Celsius.

The three regression coefficients for each component $b_{0,i}$, $b_{1,i}$ and $b_{2,i}$ are specified by the [VISCAQA](#) keyword.

Aquifer properties

An analytic water aquifer defined by [AQUCHWAT](#), [AQUCT](#), [AQUFETP](#) or [AQUFLUX](#) contains only the pure water component. Any water component is allowed to leave the reservoir. Additionally, the water properties of the analytic aquifers are calculated as described above.

28

GASWAT option

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The GASWAT option in ECLIPSE 300 provides a method of modeling gas phase/aqueous phase equilibria using an equation of state. The Peng Robinson equation of state is modified following the suggestions of Soreide and Whitson [Ref. 101] to obtain accurate gas solubilities in the aqueous phase.

Note: The GASWAT option is two-phase only. By using it, the gas and water phases are implicitly requested. It is not possible to model the oil phase while using this option. If the [GASWAT](#) keyword is present, then the [OIL](#), [WATER](#) and [GAS](#) keywords are not necessary and will be ignored.

The temperature of the reservoir may vary with depth, and this is reflected in the equation of state treatment, in which temperature coefficients are stored to be combined with the temperature of each cell. A global salinity value may be entered to correct the default equation of state modifications for brine concentration. The modified equation of state coefficients may also be overridden by the user.

The GASWAT option also allows molar calorific values to be entered under the [CALVAL](#) or [CALVALR](#) keywords. Gas qualities, calorific production rates and calorific production totals for wells and groups are calculated and reported. A group can be given a quality target using the [GQUALITY](#) keyword - quality is here defined as calorific value per mole of gas. The group control algorithm will attempt to meet this quality target by distributing production between wells of better and worse quality.

The reservoir may be equilibrated with a single phase present or with an initial gas-water contact. In the latter case the condition that the two phases are in equilibrium at the contact is honored - you specify the composition of one of the phases and the program obtains the composition of the other from the equilibrium condition.

A number of new summary items have been added so that calorific production rates, calorific production totals and quality values may be obtained for wells, groups and the field.

The water analytic aquifers are dealt with in the same way as in the CO2STORE option. Please refer to "[Analytic water aquifers](#)".

The determination of single phase as gas or water use a simplified criteria. If the temperature is less than the pseudo critical temperature, as computed by the Li criteria, see keyword [FACTLI](#), and the phase contains mostly H₂O, the phase is labeled water otherwise gas. It is possible to activate the pre-2008.1 behavior by setting [item 146](#) to 6 in the OPTIONS3 keyword. Also the flash will treat H₂O as the most heavy component unless [item 125](#) in OPTIONS3 is set to -1.

Note that the default density and viscosity calculations for water with solutes may be replaced by Ezrokhi's method, the latter also used in conjunction with [CO2STORE](#) and [GASSOL](#). This is done by adding the

keywords [DENAQA](#) and [VISCAQA](#) respectively for density and viscosity, and specifying the Ezrokhi temperature coefficients for each component.

Equation of state modifications

The conventional Peng-Robinson equation of state

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

$$A = \sum_{jk} x_j x_k A_{jk} \quad \text{Eq. 28.1}$$

where x are component mole fractions and

$$A_{jk} = (1 - k_{jk})(A_j A_k)^{\frac{1}{2}} \quad \text{Eq. 28.2}$$

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

$$A_j = \frac{\Omega_j P_{rj}}{T_{rj}^2} \quad \text{Eq. 28.3}$$

The reduced critical temperatures and pressures are $T_r = T / T_c$ and $P_r = P / P_c$. We define reduced A-coefficients by extracting this temperature and pressure dependence:

$$A_j^r = \frac{A_j^r \cdot P}{T^2} \quad \text{Eq. 28.4}$$

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

$$A_j^r = \frac{\Omega_{oj} T_{cj}^2 \alpha_j}{P_{cj}} \quad \text{Eq. 28.5}$$

with

$$\alpha_j^{\frac{1}{2}} = 1 + f(w_j) \left(1 - T_{rj}^{\frac{1}{2}} \right) \quad \text{Eq. 28.6}$$

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

$$f(w_j) = 0.37464 + 1.54226w_j - 0.26996w_j^2 \quad \text{Eq. 28.7}$$

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

$$A_j^r = \frac{\Omega_{oj} T_{cj}^2 \left(p_j + q_j T^{\frac{1}{2}} \right)^2}{P_{cj}} \quad \text{Eq. 28.8}$$

where $p_j = 1 + f(w_j)$ and $q_j = -f(w_j) \Big|_{T_c^{\frac{1}{2}}}$.

Then:

$$A_{rjk} = [1 - k_{jk}] \left[\frac{\Omega_{oj} T_{cj}^2}{P_{cj}} \right]^{\frac{1}{2}} \left[\frac{\Omega_{ok} T_{ck}^2}{P_{ck}} \right]^{\frac{1}{2}} \left[p_j p_k + (p_j q_k + p_k q_j) T^{\frac{1}{2}} + q_j q_k T \right] \quad \text{Eq. 28.9}$$

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

$$A_{1jk} = (1 - k_{jk}) \left(\frac{\Omega_{oj} T_{cj}^2}{P_{cj}} \right)^{\frac{1}{2}} \left(\frac{\Omega_{ok} T_{ck}^2}{P_{ck}} \right)^{\frac{1}{2}} (p_j p_k) \quad \text{Eq. 28.10}$$

$$A_{2jk} = (1 - k_{jk}) \left(\frac{\Omega_{oj} T_{cj}^2}{P_{cj}} \right)^{\frac{1}{2}} \left(\frac{\Omega_{ok} T_{ck}^2}{P_{ck}} \right)^{\frac{1}{2}} (p_j q_k + p_k q_j) \quad \text{Eq. 28.11}$$

$$A_{3jk} = (1 - k_{jk}) \left(\frac{\Omega_{oj} T_{cj}^2}{P_{cj}} \right)^{\frac{1}{2}} \left(\frac{\Omega_{ok} T_{ck}^2}{P_{ck}} \right)^{\frac{1}{2}} [q_j q_k] \quad \text{Eq. 28.12}$$

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

$$A_{jk} = \left(A_{1jk} + A_{2jk} T^{\frac{1}{2}} + A_{3jk} T \right) \frac{P}{T^2} \quad \text{Eq. 28.13}$$

Modifications for the vapor-aqueous system

If the Peng-Robinson equation is used in the above form for a gas-aqueous system the predicted saturated mole fractions of gases dissolved in water are hopelessly wrong - predicted solubilities are typically 10-7 mole/mole, whilst experimental values are nearer 10^{-3} mole/mole.

Soreide and Whitson propose two changes:

- The expression for $\alpha^{\frac{1}{2}}$ above is modified for the water component. It is replaced by an expression with a more complex temperature dependence:

$$\alpha^{\frac{1}{2}} = 1 + aq_1(1 - aq_2 T_r) + aq_3(T_r^{-3} - 1) \quad \text{Eq. 28.14}$$

with default values:

$$aq_1 = 0.4530$$

$$aq_2 = 1 - 0.0103 c_s^{1.1}$$

$$aq_3 = 0.0034$$

c_s is the brine concentration expressed as a molality. (Molality is given by $[1000 \times (\text{moles of salt}) / (\text{mass of water})]$, where moles and mass are expressed in consistent units such as kg-M and kg respectively. 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:

$$\alpha^{\frac{1}{2}} = p + qTr^{\frac{1}{2}} + rTr + sTr^{(-3)} \quad \text{Eq. 28.15}$$

- Different binary interaction coefficients are used for the gas and aqueous phases. This would cause problems near the critical point, but for a water-dominated system the critical temperature is in the region of $T_{cw} = 647\text{K}$, which should be well above the area of interest.

Soreide and Whitson propose adding a temperature dependence to the aqueous phase binary interaction coefficients:

$$k_{jw}^a = bq_1 + bq_2 T_{rj} + bq_3 T_{rj}^2 \quad \text{Eq. 28.16}$$

with default values:

$$bq_1 = A_0(1 + S_0 c_s)$$

$$bq_2 = A_1(1 + S_1 c_s)$$

$$bq_3 = A_2(1 + S_2 c_s)$$

where:

$$A_0 = 1.112 - 1.7369 w_j^{-0.1}$$

$$A_1 = 1.1001 + 0.836 w_j$$

$$A_2 = -0.15742 - 1.0988 w_j$$

and:

$$S_0 = 0.017407$$

$$S_1 = 0.033516$$

$$S_2 = 0.011478$$

Special values are used for binary interaction coefficients between the aqueous phase and N₂, CO₂ and H₂S:

$$k_{N_2w}^a = -1.70235 \left(1 + 0.025587 c_s^{\frac{3}{4}}\right) + 0.44338 \left(1 + 0.08126 c_s^{\frac{3}{4}}\right) T_{rN_2} \quad \text{Eq. 28.17}$$

$$k_{co_2w}^a = -0.31092 \left(1 + 0.15587 c_s^{0.7505}\right) + (0.23580) \left(1 + 0.17837 c_s^{0.979}\right) T_{rCO_2} - 21.2566 e^{(-6.7222 T_{rCO_2}) c_s} \quad \text{Eq. 28.18}$$

$$K_{H_2Sw}^a = -0.20441 + 0.23426 T_{rH_2S} \quad \text{Eq. 28.19}$$

These fit into the general quadratic form, apart from the exponential term in the CO₂ 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_{ijk} may then be simply assembled with the cell temperature to build up the equation of state A-coefficient.

The program automatically uses both these modifications in GASWAT mode. The water component is identified as that with name 'H2O' - this must be present. Nitrogen, carbon dioxide and hydrogen sulfide, if present, should be identified with the names 'N2', 'CO2' and 'H2S' respectively. Keywords such as **SALINITY**, **SALINITR**, **AQCOEF**, **BICAQ1**, **BICAQ2** and **BICAQ3** are entered once, and need not be specified for each equation of state region or separately for reservoir and surface conditions. The gas binary interaction coefficients are obtained from the usual user-input data.

Calorific values

Units

These are entered in the PROPS section under the **CALVAL** or **CALVALR** keywords. The metric units are kJ/kg-M. For conversion purposes $1 \text{ m}^3 = 0.04165935 \text{ kg-M}$ at normal standard conditions (compare with "[Conversion for an ideal gas \(Z=1\)](#)"). Well and group quality is reported in kJ/kg-M. Calorific rates and totals are in kJ/day and kJ respectively. The equivalent field units are Btu/lb-M 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:

- The group has more than one son under group control
- 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, for instance in kg-M, and m_w the mass of the water, for instance in kg. This may be related to the usual definition in terms of parts per million by mass:

$$c_s^{ppm} = A \cdot c_s / (1 + B \cdot c_s) \quad \text{Eq. 28.20}$$

$$c_s = c_s^{ppm} / (A - B \cdot c_s^{ppm}) \quad \text{Eq. 28.21}$$

with $A = 10^3 M_{\text{NaCl}}$, $B = 10^{-3} M_{\text{NaCl}}$, $M_{\text{NaCl}} = 58.44 \text{ kg/kg-M}$

The salinity variation of the equation of state coefficients is not treated on a cell by cell basis - a single value is entered using the [SALINITY](#) keyword in the PROPS section. If it is not entered the salinity value is assumed to be zero.

Relative permeability tables

The water relative permeability is input using the [WSF](#) keyword, while the gas relative permeability and the gas-water capillary pressure are entered using the [GSF](#) keyword. These keywords are analogous to the [SOF2](#) and [SGFN](#) keywords.

The saturation tables can be output to the PRINT file by entering the WSF or 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 overridden using the tenth argument of EQUIL.

Note: 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

Refer to [Gas calorific value control option](#) in the *ECLIPSE Reference Manual*.

DENSITY values are not used in the GASWAT option: both gas and water density values are obtained from the equation of state.

Note as well that the water phase is considered as “hydrocarbon” when calculating the “hydrocarbon weighted” field pressure reported by FPR and FPRH. For a comparison with a simulation involving an oil phase such as [CO2SOL](#), the “pore volume weighted” pressure returned by FPRP can be used. Additionally, setting [item 151](#) in OPTIONS3 to 1 ensures that only the gas phase is treated as “hydrocarbon”.

Gas calorific value control

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

The gas calorific value control option is only available in ECLIPSE 100; it is not available in ECLIPSE 300, but the GASWAT option in that simulator (see "[GASWAT option](#)") 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/sm³ (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-M (METRIC), Btu/lb-M (FIELD), J/gm-M (LAB). Conversion between the two definitions of quality can be performed assuming ideal gas behavior:

- 23.6903 sm³/kg-M (METRIC)
- 0.379506 Mscf/lb-M (FIELD)
- 23690.3 scc/gm-M (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

$$Q_{pG} = \sum_{W \in i} Q_{pW} + M_G \sum_{W \in g^+} G_{pW} + M_G \alpha \sum_{W \in g^-} G_{pW} \quad \text{Eq. 29.1}$$

where:

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

Q_{pW} is well W's production rate of the controlled phase p

G_{pW} is well W's guide rate of the controlled phase p

M_G is the guide rate multiplier for group G

α is the guide rate adjustment factor for the set $W \in g^-$

$W \in i$ represents the set of subordinate wells operating under their individual control modes

$W \in g^+$ represents the set of subordinate wells on group control whose calorific values exceed the target

$W \in g^-$ represents the set of subordinate wells on group control whose calorific values are less than the target.

The group's guide rate multiplier is obtained from equation 29.1 by setting

$$Q_{pG} = Q_{pG}^{\text{targ}} \quad \text{Eq. 29.2}$$

where

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

This gives

$$M_G = \frac{Q_{pG}^{\text{targ}} - \sum_{\substack{W \in i \\ W \in g^+}} Q_{pW}}{\sum_{\substack{W \in i \\ W \in g^-}} G_{pW} + \alpha \sum_{\substack{W \in i \\ W \in g^-}} G_{pW}} \quad \text{Eq. 29.3}$$

The guide rate adjustment factor α is calculated from the condition that the mean calorific value of the group's gas production must be equal to a specified target,

$$\frac{\sum_{\substack{W \\ W}} C_{gW} Q_{gW}}{\sum_{\substack{W \\ W}} Q_{gW}} = C_{gG}^{\text{targ}} \quad \text{Eq. 29.4}$$

where:

Q_{gW} is well W's gas production rate

C_{gW} is the calorific value of the gas produced by well W

C_{gG}^{targ} is the target mean calorific value for group G.

The flow rate of a well under group control depends upon its guide rate and the group's guide rate multiplier,

$$\begin{aligned} Q_{gW} &= M_G G_{pW} (Q_g / Q_p)_W && \text{for } (W \in g^+) \\ Q_{gW} &= \alpha M_G G_{pW} (Q_g / Q_p)_W && \text{for } (W \in g^-) \end{aligned} \quad \text{Eq. 29.5}$$

where

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

Substituting equation 29.5 into equation 29.4 yields

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

Simultaneous solution of equations 29.1 and 29.6 yields the guide rate adjustment factor α

$$\alpha = \frac{a \sum_{W \in g^+} (C_{gW} - C_{gG}^{\text{targ}}) G_{pW} \left(\frac{Q_g}{Q_p} \right)_W - b \sum_{W \in g^+} G_{pW}}{a \sum_{W \in g^-} (C_{gG}^{\text{targ}} - C_{gW}) G_{pW} \left(\frac{Q_g}{Q_p} \right)_W + b \sum_{W \in g^-} G_{pW}} \quad \text{Eq. 29.7}$$

where:

$$a = Q_{pG}^{\text{targ}} - \sum_{W \in i} Q_{pW} \quad \text{Eq. 29.8}$$

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

Once α has been obtained from equation 29.7, the group's guide rate multiplier M_G can be obtained from equation 29.3. α of course must be constrained to be positive. A negative solution for α implies that the target mean calorific value cannot be achieved at the target flow rate. When both M_G and α are known, the group controlled wells can be given their production rate targets. If a well cannot achieve its rate target, it drops out of group control and becomes independently controlled (at its own rate or pressure limit), and the calculations for M_G and α must be repeated.

If the group G has subordinate groups that have been given group guide rates, then these groups and their guide rates will be dealt with as if they were wells in the above equations. The wells subordinate to these groups with guide rates are not processed as individual entities. Essentially, the subscript W in the above equations refers to the set of wells or groups with guide rates that can come under direct control of group G.

Action when calorific value target cannot be met

In order to meet the mean calorific value target, the set of wells under group control must include at least one well producing gas with a calorific value greater than the target, and at least one producing with less than the target calorific value. ECLIPSE can take certain remedial actions when there is insufficient high or low calorific value gas available.

Not enough high calorific value gas

If the group is meeting its production rate target, but the numerator of equation 29.7 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 equation 29.7 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 the 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

$$E_G = \sum_{W \in i} E_W + M_G \sum_{W \in g^+} G_{eW} + M_G \alpha \sum_{W \in g^-} G_{eW} \quad \text{Eq. 29.9}$$

where:

E_W is $C_{gW}Q_{gW}$, the energy production rate from the well

G_{eW} is

$$C_{gW}G_{gW} \quad \text{Eq. 29.10}$$

is 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 timestep.

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 equation 29.9 by equating E_G with the target value. If there is **no** simultaneous control of the mean calorific value, the group's guide rate multiplier is

$$M_G = \frac{E_G^{\text{targ}} - \sum_{W \in i} E_W}{\sum_{W \in g} G_{eW}} \quad \text{Eq. 29.11}$$

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

$$M_G = \frac{E_G^{\text{targ}} - \sum_{W \in i} E_W}{\sum_{W \in g^+} G_{eW} + \alpha \sum_{W \in g^-} G_{eW}} \quad \text{Eq. 29.12}$$

When the controlled quantity is energy, equation 29.6 becomes

$$M_G \sum_{W \in g^+} \left(1 - \frac{C_{gG}^{\text{targ}}}{C_{gW}}\right) G_{eW} - \alpha M_G \sum_{W \in g^-} \left(\frac{C_{gG}^{\text{targ}}}{C_{gW}} - 1\right) G_{eW} = \sum_{W \in i} (C_{gG}^{\text{targ}} - C_{gW}) Q_{gW} \quad \text{Eq. 29.13}$$

Simultaneous solution of equation 29.9 and 29.13 gives the guide rate adjustment factor required to meet the target mean calorific value

$$\alpha = \frac{e \sum_{W \in g^+} \left(1 - \frac{C_{gG}^{\text{targ}}}{C_{gW}}\right) G_{eW} - b \sum_{W \in g^-} G_{eW}}{e \sum_{W \in g^-} \left(\frac{C_{gG}^{\text{targ}}}{C_{gW}} - 1\right) G_{eW} + b \sum_{W \in g^+} G_{eW}} \quad \text{Eq. 29.14}$$

where:

$$e = E_G^{\text{targ}} - \sum_{W \in i} E_W \quad \text{Eq. 29.15}$$

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

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 $\Delta Q_g W C_g W$, where $\Delta Q_g W$ 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](#)" 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.

```
WCALVAL
-- Well      Calorific
-- name     value
'PRODA*'   30000.0 /
'PRODB*'   26000.0 /
/
```

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:

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/sm³ ([METRIC](#)), Btu/Mscf ([FIELD](#)), J/scc ([LAB](#)). (See "[Tracer tracking](#)" 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 [GUIDEDECAL](#) 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}

$$\beta_W = (1 + A C_{gW})^B \quad \text{Eq. 29.16}$$

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 [GUIDEDECAL](#), 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"). 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.

Refer to [Gas calorific value control option](#) in the *ECLIPSE Reference Manual*.

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](#)"), 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

Refer to [Gas calorific value control option](#) in the *ECLIPSE Reference Manual*.

SCHEDULE section

Keyword	Description
CALTRAC	Specifies which tracer is to be equated with the calorific value.
DCQDEFN	When using the Gas Field Operations Model, use this to tell ECLIPSE whether the DCQ targets refer to energy or gas rate.
GCALECON	Allows minimum economic energy rates and mean calorific values to be defined for groups and the field.
GCONCAL	Specifies the target calorific value for a group or the field.
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.

30

Geomechanics

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

Geomechanics provides improved ability to simulate compaction processes that are not available with the standard rock compaction option (see "[Rock compaction](#)"). This gives improved ability to predict subsidence and compaction in a reservoir, and also improved modeling of fluid flow through stress dependent permeability and Biot's constant.

Two separate geomechanics workflows are available:

- ["Geomechanics external to ECLIPSE"](#)
- ["Geomechanics internally within ECLIPSE Compositional"](#)

Note: It is anticipated that the capability offered by the internal geomechanics option will be replaced and extended by the external coupling of ECLIPSE to VISAGE through the use of Petrel RE. The internal geomechanics option is currently on maintenance only and is not available commercially.

Geomechanics external to ECLIPSE

The geomechanics workflow which is external to ECLIPSE uses the Schlumberger VISAGE rock mechanics simulator [\[Ref. 94\]](#) to process the results of ECLIPSE, together with rock and fluid elastic and plastic properties, in order to calculate the stresses and strains which will occur in the reservoir due to the changes in pressure, saturation (or composition) and temperature. It can modify the pore volume and permeability (or transmissibility) of the model due to these calculated stresses and strains.

In the VISAGE workflow, there are three main styles of interaction with ECLIPSE:

- One-way coupling

The results of an ECLIPSE simulation (as stored in the restart files) are used to calculate the stresses and strains. There is no feedback from the computations in VISAGE to the inputs of the ECLIPSE simulation, so the ECLIPSE simulation can be run completely independently of the VISAGE simulation.

- Explicit coupling

At certain times in the simulation (known as stress times), the ECLIPSE run is stopped, and the results from the restart files are processed within the VISAGE simulator. After the VISAGE simulation at the current stress time is complete, the ECLIPSE run is restarted from this time but with certain properties

such as pore volume and permeability (or transmissibility) modified based on the stresses and strains computed by VISAGE.

- Implicit coupling

The ECLIPSE run is stopped at a stress step report time, and the results in the restart files are processed in VISAGE. The ECLIPSE run is restarted from the previous stress step report time with modified pore volume and permeability (or transmissibility) and the process repeated until the change in pore volume calculated by ECLIPSE over the period agrees with that computed by VISAGE.

The VISAGE workflow in ECLIPSE

From 2008.2 onwards, features have been added to the ECLIPSE 100 and ECLIPSE 300 simulators to assist the VISAGE application in this external geomechanics workflow. The [VISAGE](#) keyword activates these workflow features.

1. The internal geomechanics workflow is not compatible with the VISAGE workflow. For this reason, all internal geomechanics keywords and restart summary mnemonics are ignored by ECLIPSE if the [VISAGE](#) keyword is present.
2. The list of report dates at which external geomechanics calculations are required is specified in the [VISDATES](#) keyword. A list of available report dates from a simulation is requested by running the VISAGE workflow in initialization mode, this is achieved by setting item 1 of the [VISOPTS](#) keyword to YES.
 - a. In initialization mode, the model is run as though [NOSIM](#) is present and ECLIPSE writes out a file containing information about all report steps that would be performed in the simulation. This file has the same root name as the simulation, but with an [EDESC](#) extension (for example, when running a simulation model `NAME . DATA`, this file will be `NAME . EDESC`). Also in initialization mode, all requests for stress steps are ignored; and [GRID](#), [INIT](#) and time zero restart files are output.
3. ECLIPSE can be made to exit at a stress step report time (once the restart files have been written) by setting item 2 of the [VISOPTS](#) keyword to YES. This feature is turned off by default but it must be turned on in explicit or implicit coupling in the VISAGE workflow.
4. When the field pressure at a report time has changed by more than the relative tolerance (specified in item 4 of the [VISOPTS](#) keyword) since the last stress step report time, then an additional stress step will be performed at that report time - this is called a dynamic stress step. Note that this feature must be turned off when repeating the calculations over a stress step interval in the implicit coupling, and any dynamic stress steps which have been performed must be added to the [VISDATES](#) list. The dynamic stress step mechanism is turned off by default, but is turned on by setting item 3 of the [VISOPTS](#) keyword to YES.

Note: The VISAGE suite of products can only handle a maximum of 99 stress steps in its workflow. If you request more than 99 stress steps in the [VISDATES](#) keyword, or if a dynamic stress step would be triggered after 99 stress steps have already occurred, then ECLIPSE will exit with an error.

5. In order to guarantee that restart files are generated at all of the stress step report times, any [FREQ](#) mnemonic specification in the [RPTRST](#) keyword is ignored and [FREQ=1](#) (that is, output a restart file at every report step) is enforced by default. If a smaller amount of data is required in the restart files, the frequency of output can be reduced by
 - a. specifying the [RPTRST BASIC](#) mnemonic as 3, 4 or 5,

- b. setting the FREQ mnemonic to the desired output frequency and
- c. setting item 6 of the VISOPTS keyword to YES.

Note: When modifying the restart output frequency, you must ensure that there will still be restart information available at all required stress step dates.

6. In the VISAGE workflow, the minimum requirement for restart file content is that it can be used to restart the run. For this reason, the RPTRST BASIC=2 mnemonic is enforced when the VISAGE keyword is present. In addition, the RPTRST ROCKC mnemonic is enforced so that the modified pore volume and transmissibility multiplier are present in the restart files. By default, any other property requests will be ignored. If additional properties are required in the restart file, these must be specified in the RPTRST keyword, and item 1 of the VISOPTS keyword must be set to YES.
7. Whenever a new stress step is encountered, ECLIPSE writes a list of all stress steps performed to a small file with the same root name as the simulation, but with a VDESC extension (for example, when running a simulation model NAME.DATA, this file will be NAME.VDESC). This information allows the external geomechanics simulator to know which of the requested stress steps have been performed, and also whether any dynamic stress steps have been performed.
 - a. At the beginning of the workflow, this VDESC file needs to be emptied (if it already exists from a previous run). ECLIPSE is instructed to empty this file when the VISAGE initialization is requested by setting item 1 of the VISOPTS keyword to YES.

Summary of keywords

RUNSPEC

- [VISAGE](#) Activate the VISAGE geomechanics workflow

SOLUTION

- [VISDATES](#) Specify VISAGE geomechanics stress step dates
- [VISOPTS](#) Specify VISAGE geomechanics workflow options

Geomechanics internally within ECLIPSE Compositional

Note: It is anticipated that the capability offered by the internal geomechanics option will be replaced and extended by the external coupling of ECLIPSE to VISAGE through the use of Petrel RE. The internal geomechanics option is currently on maintenance only and is not available commercially.

Some features of coupling the geomechanics rock stress calculation with the fluid flow include:

- Flexibility
 - Stress, traction and displacement boundary conditions can be set on external as well as internal surfaces.
 - 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.

A finite element method (FEM) is used to calculate the stress. The finite element method retains second order accuracy when the grid is skewed which can arise, for example, when using Corner-Point geometry. Most of the simulator technology is available when running the coupled stress calculation such as the ability to run in black oil, compositional or thermal modes and the full use of well model features.

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

There is a cap model available for plastic deformation including both a shear failure and a yield in compression. The cap model has a failure surface and an elliptical yield cap that closes the open space between the failure surface and the hydrostatic axis. The yield cap expands in the stress space according to a specified hardening rule.

Initial in-situ stress field induced from overburden plus gravity, or tectonic boundary stresses, or thermal effects can be simulated separated or as a whole.

For this release, there are a number of important restrictions in the finite element option. Radial gridding is not allowed. There is no ability to run parallel calculations. Dual porosity is not available. These are discussed in "[Restrictions](#)".

Elastic stress equations

A preliminary publication that outlines some of the basic ideas discussed below can be found in [\[Ref. 108\]](#) of the bibliography. A more recent publication can be found in [\[Ref. 109\]](#).

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

$$\begin{aligned} \frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} &= 0 \\ \frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{zy}}{\partial z} &= 0 \\ \frac{\partial \sigma_z}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \rho g &= 0 \end{aligned} \quad \text{Eq. 30.1}$$

Here ρ is the rock density or a combination of rock and fluid reservoir density, and g is the acceleration due to gravity constant. The elastic normal stresses σ and shear stresses τ can be expressed in terms of strains, ε and γ , as:

$$\begin{aligned} \sigma_x &= 2G\varepsilon_x + \lambda(\varepsilon_x + \varepsilon_y + \varepsilon_z) - \alpha P - (2G + 3\lambda)\alpha_T(T - T_r) \\ \sigma_y &= 2G\varepsilon_y + \lambda(\varepsilon_x + \varepsilon_y + \varepsilon_z) - \alpha P - (2G + 3\lambda)\alpha_T(T - T_r) \\ \sigma_z &= 2G\varepsilon_z + \lambda(\varepsilon_x + \varepsilon_y + \varepsilon_z) - \alpha P - (2G + 3\lambda)\alpha_T(T - T_r) \end{aligned} \quad \text{Eq. 30.2}$$

$$\begin{aligned} \tau_{xy} &= G\gamma_{xy} \\ \tau_{yz} &= G\gamma_{yz} \\ \tau_{zx} &= G\gamma_{zx} \end{aligned} \quad \text{Eq. 30.3}$$

Note that stresses as defined in equation 30.2 include the pore pressure and Biot constant and are, therefore, total stresses. Constants G , also known as the shear modulus, and λ are Lame's constants. They are functions of Young's modulus, E , and Poisson's ratio, ν :

$$G = \frac{E}{2(1+\nu)}, \quad \text{Eq. 30.4}$$

$$\lambda = \frac{Ev}{(1-2\nu)(1+\nu)} \quad \text{Eq. 30.5}$$

Strains $\varepsilon_{x,y,z}$ are defined in terms of displacements in the x, y, z directions, namely u , v and w . Thus

$$\begin{aligned} \varepsilon_x &= \frac{\partial u}{\partial x} & \varepsilon_y &= \frac{\partial v}{\partial y} & \varepsilon_z &= \frac{\partial w}{\partial z} \\ \gamma_{xy} &= \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} & \gamma_{xz} &= \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} & \gamma_{xy} &= \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \end{aligned} \quad \text{Eq. 30.6}$$

These strains do not form a tensor since the shear strains γ_{xy} , γ_{yz} , γ_{zx} are not multiplied by a factor of 1/2. However, stresses as defined above in equation 30.2 do form a tensor. Within the simulator, the rock force is calculated in units of Newtons, lbf, or dynes depending on either METRIC, FIELD or LAB units are used for the simulation.

Gridding

The finite element stress calculation use the same standard corner point or Cartesian simulation grid as the flow equations.

For the finite element stress calculation, a variational or Galerkin form of the above equations is set up where each equation in equation 30.1 is multiplied by a component of a test function. This is then summed over all components and integrated over the domain of three dimensional grids. Basis functions are piecewise tri-linear. In all unknown rock displacements are defined at the nodes or corner-points of the grid. This is discussed further in [Ref. 69].

Fluid component conservation, rock volume conservation and volume balance equations

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

$$V^{n+1}(1-\varphi^{n+1}) = V^n(1-\varphi^n) \quad \text{Eq. 30.7}$$

where V is the bulk volume of the grid block, φ is the porosity. This is equivalent to the relation $\delta\varphi = -(1-\varphi)\delta\varepsilon_b$. A second porosity-stress relation is available and is discussed in the GEOMECH keyword.

Fluid component mass conservation assumes the usual form and is discussed elsewhere (see Aziz and Settari [Ref. 8], and also Quandalle and Sabathier [Ref. 89]).

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

$$\sigma_T = \sigma_{eff} - \alpha P \quad \text{Eq. 30.8}$$

where

σ_T is the total stress,

σ_{eff} is the rock effective stress,

α is Biot's constant and

P is the pore (fluid) pressure.

Tensile stresses are positive, compressive are negative. Note that this sign convention holds also when defining tabular functions of permeability and Biot constant versus principal stress. You may also choose to define compressive stresses as positive. This can be set with item 6 of the keyword [GEOMECH](#).

Variables and solution of the equations

Variables used in the fluid flow simulator are moles of all fluid components per unit pore volume, energy per unit bulk volume if thermal simulation is specified, pressure of one of the fluid phases. Some of these variables are eliminated before entering the solver. In particular, one of the fluid molar densities is eliminated using the volume balance equation. Variables used in the stress calculation are rock displacements in the x, y, z directions, and porosity. The linear solver used to solve the system of stress equations is an algebraic multi-grid solver, [Ref. 110].

Boundary conditions and initialization

Either stress or displacement boundary conditions can be chosen. Each cell of each side of the simulation grid, that is X-/X+ (or equally I-/I+), Y-/Y+ (or J-/J+), and Z-/Z+ (or K-/K+), can have one of the above boundary conditions prescribed. Internal boundaries (cell faces adjacent to internal inactive cells) can also have a load or displacement boundary condition applied.

Note: All boundary conditions cannot be stress based, as this is an ill-posed problem for the displacements solution. If some stress boundaries are set, then part of the reservoir must have some displacements imposed.

Here, + means the outer normal direction of this boundary face is the same as the axis, while - refers to the outer normal direction that is different from the axis. For example, if X axis is from left to right, then X- means the outer normal direction of this boundary face is to the left, X+ refers to the one has right outer normal direction. To set boundaries conditions the user can specify:

- Traction boundary conditions set with the keywords [GMTRABC](#).
- Displacement boundary conditions set with the keywords [GMDISBC](#).
- Principal stress boundary conditions set with the keyword [GMPSTBC](#).

Boundary conditions in local grid refinements can be set with keywords [GMTRABCL](#), [GMDISBCL](#) or [GMPSTBCL](#).

Plasticity

The keyword [GEOYLDF](#) allows classical yield functions to be set in specified regions of the grid. Either Drucker-Praeger (default) or Mohr-Coulomb failure criteria can be chosen. All of these are defined with a cohesion and angle of internal friction. Hardening laws and parameters can also be set.

Another plastic deformation cap model that can simulate both shear failure and compaction cap is available through keyword [GEOYCAP](#). The cap is defined with a cohesion, angle of internal friction, and material uniaxial tensile strength. The main features of the cap model include a failure surface F_s :

$$F_s = \Gamma + \sqrt{J_2} + C e^{(B I_1)} - A = 0 \quad \text{Eq. 30.9}$$

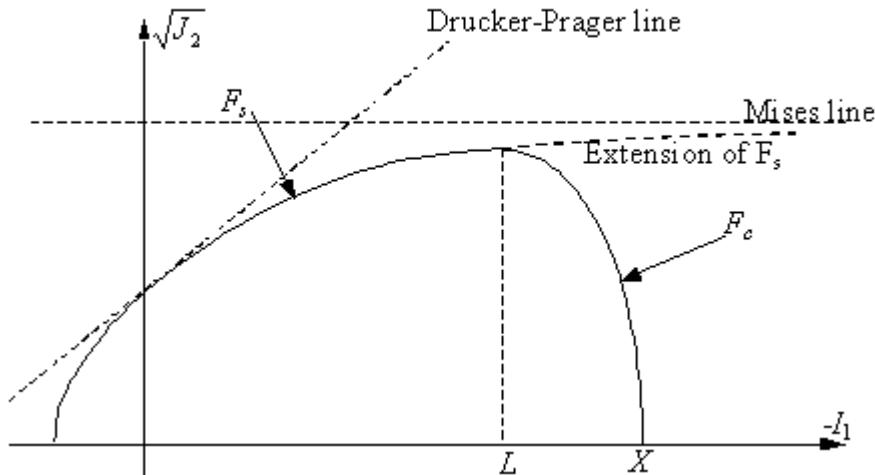
and an elliptical yield cap F_c :

$$F_c = R^2 \Gamma^2 J_2 + (I_1 - L)^2 - (X - L)^2 = 0 \quad \text{Eq. 30.10}$$

that closes the open space between the failure surface and the hydrostatic axis. Here

$$A = \frac{1}{\sqrt{3}} \Gamma \sigma_s \quad B = \frac{\alpha}{\left(\frac{1}{\sqrt{3}} \sigma_s - k\right) \Gamma} \quad C = \left(\frac{1}{\sqrt{3}} \sigma_s - k\right) \Gamma \quad \text{Eq. 30.11}$$

Several material properties are required to input to define these cap surfaces.



Arbitrary geomechanical regions in the grid can be set using the keyword [YLDNUM](#). Parameters and yield functions are then defined in each region. For example, a region can be set around a particular well.

User Interface

A Young's modulus, Poisson ratio and rock density must be entered for each grid block in the GRID section. A Biot constant may also be entered but, if not, is defaulted to unity. Stress, traction and displacement boundary conditions are entered in both the SOLUTION and SCHEDULE sections. If a plastic calculation (finite element only) is required, a cohesion, angle of internal friction and hardening parameters if desired in different regions of the grid are input in the PROPS section. Further discussion of this option is found in the [GEOYLDF](#) keyword.

Initial in-situ stress

Initial in-situ stresses induced from overburden stress plus gravity, or tectonic boundary condition or thermal effects can be calculated separately or as a whole.

For gravity-induced stresses, rock density is set in [ROCKDEN](#), and overburden traction is set by [OVERBDI](#) in the SOLUTION section. Tectonic loadings are defined from [TECBDI](#) in the SOLUTION section.

Using the coupled geomechanics model

CAUTION: The geomechanic rock stress (force) balance equations that are implemented in ECLIPSE 300 are based on the assumption of small strains. Maximum strains should not exceed 5% during the life of the simulation. This restriction applies to all work with this option.

A partly-coupled geomechanics calculation in ECLIPSE 300 is selected using the [GEOMECH](#) keyword in the RUNSPEC section.

- 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 stress is updated using the latest fluid (pore) pressures.

The other choice is:

- to update the rock stress at every timestep.
- The third item is choice of the calculation method (obsolete: the supported method is the finite element FE one).

Defining the rock mechanical parameters

In the GRID section, several keywords are used to set the rock mechanical properties.

The Young's modulus, Poisson ratio and rock density are set with the keywords [YOUNGMOD](#), [POISSONR](#) and [ROCKDEN](#), respectively. A value of YOUNGMOD and POISSONR is needed for each grid block. Rock density is used to calculate the rock weight contribution to the gravity force as accounted for in the vertical rock force balance equation.

Interaction between the rock and fluid is determined by the Biot constant. A keyword [BIOTC](#) sets this for each grid. If not set, then this will default to 1.0 and a message is printed in the output (.PRT) file informing the user of this default. It is also possible to define a multiplier of this constant as a function of principal stress. See the next section.

When running in thermal mode, a coefficient of linear thermal expansion for each grid can be set with the [GLTHEX](#) keyword.

Modifying permeability as a function of stress

A permeability multiplier as a tabular function of maximum and minimum principal stress can be input for each geomechanics region. (See [YLDNUM](#) and the first argument of [GEODIMS](#)). The total number of geomechanics regions is set with the first argument of GEODIMS, and a region number for each grid cell is then defined with the keyword [YLDNUM](#).

Similarly, a multiplier for the Biot constant can be set as a function of maximum and minimum principal stress. A table can be defined with the keyword [BIOTCTAB](#) in each geomechanics regions of the simulation.

For both of these multipliers, the multipliers are calculated after every rock stress update if partly coupled, or explicitly at the end of each timestep if fully coupled.

Stress boundary conditions

Boundary conditions for the geomechanics rock force (stress) balance are determined with keywords [GMDISBC](#), [GMTRABC](#), or [GMPSTBC](#) in the SOLUTION and SCHEDULE section. A boundary condition can be set on the I+ (X+), I- (X-), J+ (Y+), J- (Y-) and K+ (Z+) or K- (Z-) boundaries (faces). Note that K- is considered to be the top of the reservoir, K+ the bottom.

If no boundary condition is entered for a particular face, then that face is assigned a zero-normal-displacement boundary condition. A user-defined boundary condition is either a constant rock displacement or a load (applied stress) on this surface. A load can be supplied as a traction using the GMTRABC keywords or with known principal stress vectors with the keyword GMPSTBC.

Input file

The remaining part of the ECLIPSE 300 input file with geomechanics remains exactly the same as before.

Performance

If partly coupled with periodic stress updates, then the simulation proceeds until either a report or an end of timestep, when an update to the rock stress is requested. At this time the rock force balance equations and porosity-stress relationship are solved. These use the fluid (pore) pressures from the standard simulation and the input Biot constant to determine the total rock stress. Once this calculation is complete, any updates to geometric terms, or stress dependent permeability/Biot constant multipliers are carried out. The reservoir simulation then proceeds with updated permeabilities and porosities until the next geomechanical stress update.

Restrictions

The geomechanics functionality which is available internally within ECLIPSE 300 has the following restrictions.

- Dual porosity calculations are not available.
- Radial local grid refinement is not available.
- Unstructured grids (either global or local) are not available.
- Parallel is not available.
- Radial gridding is not available.
- Coordinates-defined local grid refinement is not available.

Summary of keywords

RUNSPEC

Keyword	Description
GEOMECH	Selects the geomechanics option, allows choices on partly coupled solution of the rock stress equations, stress calculation type, porosity-stress relation, initialization
GEODIMS	Sets the number of geomechanics regions in the simulation, sets table dimensions, in terms of the number of rows and columns, for the PERMSTAB and BIOTCTAB keywords.
SCDPDIMS	Sets dimensions of scale damage tables defined with keyword SCDATABG .

GRID section

Keyword	Description
BIOTC	Sets the Biot constant for each grid block
GLTHEX	Sets a coefficient of linear thermal expansion for each grid block
POISSONR	Sets the Poisson ratio for each grid block.
PROPFLOW	Defines parameters to be used with the proppant flowback feature
ROCKDEN	Sets the rock density for each grid block.
YOUNGMOD	Sets the Young's modulus for each grid block.

PROPS section

Keyword	Description
BIOTCTAB	Defines a multiplier for the Biot constant, this being a two-dimensional tabular function of maximum and minimum principal stress. A table is entered for each geomechanics region.
GEOYLDIF	Defines parameters for the plastic yield function in each geomechanics region.
GEOYCAP	Defines parameters for a shear failure surface, a yield cap in compression, and hardening laws in each geomechanics region.
PERMSTAB	Defines a permeability multiplier, this being a two-dimensional tabular function of maximum and minimum principal stress. A table is entered for each geomechanics region
SCDATABG	Defines a scale damage table to be used together with the proponent flowback feature, keyword PROPFLOW

REGIONS section

- [YLDNUM](#) Defines a geomechanics region number for each grid block.

SOLUTION section

Keyword	Description
DISPBC	Initial displacement velocity (mixed) boundary conditions
DISPBCL	Initial displacement velocity (mixed) boundary conditions in an LGR
GMDISBC	Initial displacement (Dirichlet) boundary condition
GMDISBCL	Initial displacement (Dirichlet) boundary condition in an LGR
GMTRABC	Initial traction (Neumann) boundary condition
GMTRABCL	Initial traction (Neumann) boundary condition in an LGR
GMPSTBC	Initial principal Stress boundary conditions
GMPSTBCL	Initial principal Stress boundary conditions in an LGR
OVERBDI	Overburden stresses for initial gravity induced stress field
OVERBDIL	Overburden stresses for initial gravity induced stress field in an LGR
PSTRBC	Initial principal Stress boundary conditions

Keyword	Description
PSTRBCL	Initial principal Stress boundary conditions in an LGR
STRESBC	Permits the application of a load or a velocity (rigid) boundary condition on all external and internal surfaces of the simulation grid.
STRESBCL	Initial displacement and traction boundary conditions in a local grid refinement
TECBDI	Initial tectonic boundary condition for initial tectonic induced stresses
TECBDIL	Initial tectonic boundary condition for initial tectonic induced stresses in an LGR
TRACTBC	Initial traction (Neumann) boundary conditions
TRACTBCL	Initial traction (Neumann) boundary conditions in a local grid refinement
RPTRST	Controls output for GRAF or RESTART files.

The solution reporting keywords and arguments are:

Restriction	Reporting keyword	Description
	COHESION	Cohesion from plastic calculation
E300 only	DROCKDIS	Delta rock displacements or change from initial conditions; this produces output with the names DROCKDIX, DROCKDIY, DROCKDIZ.
E300 only	EFFSTRES	Rock stresses: this produces arrays with the names EFFSTRXX, EFFSTRYY, EFFSTRZZ, EFFSTRXY, EFFSTRXZ, EFFSTRYZ.
	GENPLSTN	Generalized plastic strain from plastic calculation
	GENPSTRS	Generalized plastic strain multiplied by stiffness matrix
E300 only	GOSTRES	Effective rock stresses induced by gravity and overburden
E300 only	PGOSTRES	Total stresses induced by gravity and overburden
E300 only	TCSTRES	Effective stresses induced by initial tectonic stress boundary conditions
	PTCSTRES	Principal stresses induced by initial tectonic stress boundary conditions
	THSTRES	Effective stresses induced by initial thermal effects
	PTHSTRES	Principal stresses induced by initial thermal effects
	PLASDISP	Plastic displacement
E300 only	PRSTRESS	Principal stresses; this produces magnitude output with the names PRSTRES1, PRSTRES2, PRSTRES3, and directions(l1,m1,n1), (l2,m2,n2), (l3,m3,n3)
E300 only	ROCKDISP	Rock displacements; this produces output with the names ROCKDISX, ROCKDISY, ROCKDISZ
E300 only	STRAIN	Strain tensor: this produces arrays with the names STRAINXX, STRAINYY, STRAINZZ, STRAINXY, STRAINXZ, STRAINYZ
E300 only	TOTSTRES	Total stress tensor: this produces arrays with the names TOTSTRXX, TOTSTRYY, TOTSTRZZ, TOTSTRXY, TOTSTRXZ, TOTSTRYZ
E300 only	DEFFSTRS	Delta effective stresses = change from initial conditions; this produces output with the names DEFFSTXX, DEFFSTYY, DEFFSTZZ, DEFFSTYZ, DEFFSTXZ, DEFFSTXY

Restriction	Reporting keyword	Description
E300 only	DTOTSTRS	Delta total stresses = change from initial conditions; this produces output with the names DTOTSTXX, DTOTSTYY, DTOTSTZZ, DTOTSTYZ, DTOTSTXZ, DTOTSTXY
E300 only	DPRSTRES	Delta principal stresses = change from initial conditions; this produces output with the names DPRSTRS1, DPRSTRS2, DPRSTRS3 followed by their respective directions
E300 only	DSTRAIN	Delta strain components = change from initial conditions; this produces output with the names DSTRANXX, DSTRANYY, DSTRANZZ, DSTRANYZ, DSTRANXZ, DSTRANXY

SCHEDULE section

Keyword	Description
DISPBC	Displacement (Dirichlet) boundary conditions
DISPBCL	Displacement (Dirichlet) boundary conditions in an LGR
GMDISBC	Displacement (Dirichlet) boundary condition
GMDISBCL	Displacement (Dirichlet) boundary condition in an LGR
GMTRABC	Traction (Neumann) boundary condition
GMTRABCL	Traction (Neumann) boundary condition in an LGR
GMPSTBC	Principal Stress boundary conditions
GMPSTBCL	Principal Stress boundary conditions in an LGR
PSTRBC	Principal Stress boundary conditions
PSTRBCL	Principal Stress boundary conditions in an LGR
RPTSCHED	Controls output to the print file
STRESBC	Permits the application of a load or a velocity (rigid) boundary condition on all external and internal surfaces of the simulation grid.
STRESEBCL	Displacement and traction boundary conditions in a local grid refinement
TRACTBC	Traction (Neumann) boundary conditions
TRACTBCL	Traction (Neumann) boundary conditions in a local grid refinement
WSCTAB	Links a well to a particular scale damage table as defined with SCDATABG This feature is part of the PROPFLOW option.

31

Gi pseudo-compositional model

x	ECLIPSE 100
	ECLIPSE 300
	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, [Ref. 27]. 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.

A simple active gas tracking calculation is performed to determine the cumulative amount of dry injection gas (Gi), which has contacted the oil in each grid cell. Gi can then be used as the correlating parameter for determining the fluid property changes at constant pressure. An extra variable Gi has therefore to be stored in each grid cell.

The tracking calculation is performed in a novel way. Suppose f_{dr} denotes the fraction of invading gas (with $R_v=R_{v,inj}$). Then the effective R_v of the invading gas is:

$$R_{v,inv} = f_{dr} \cdot R_{v,inj} + (1 - f_{dr})R_{v,cell} \quad \text{Eq. 31.1}$$

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

$$f_{dr} = \frac{R_{v,inv} - R_{v,cell}}{R_{v,inj} - R_{v,cell}} \quad \text{Eq. 31.2}$$

If the well injects dry gas ($R_{v,inj} = 0$), this expression reduces to $f_{dr} = 1 - R_{v,inv} / R_{v,cell}$.

The cumulative injection gas flowing into each grid cell (gi) can then be defined as:

$$Gi = \int \frac{f_{dr} F_g}{HCPV} dt \quad \text{Eq. 31.3}$$

where

F_g is the surface flow rate of gas,

$HCPV$ is the hydrocarbon pore volume of the cell.

The saturated 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 $R_{v,sat}$ and $R_{s,sat}$ with Gi that is the most significant and should be ignored. For example it is likely that $R_{v,sat}$ 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 one dimensional 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 gas entering 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; this component is regarded as being in equilibrium with the gas already present in the cell. The second 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:

$$f_{dr} \cdot F_g = \left(1 - \frac{R_{v,inv}}{R_{v,cell}}\right)F_g \quad \text{Eq. 31.4}$$

If the grid cell contains a well completion injecting dry gas, there is an additional term to represent the dry gas inflow from the well:

$$f_{dr} \cdot Q_g = \left(1 - \frac{R_{v, inj}}{R_{v, cell}}\right) Q_g \quad \text{Eq. 31.5}$$

where

f_{drw} is the dry gas fraction of the well flow,

Q_g is the well gas flow rate.

Now the change in Gi from the definition is:

$$\delta Gi = \frac{f_{dr} F_g + f_{drw} Q_g}{HCPV} \delta t \quad \text{Eq. 31.6}$$

$$\delta Gi = \frac{(R_v - R_{v, up}) F_g \delta t}{R_v \cdot \varphi \cdot V \cdot (1 - S_w)} + \frac{(R_v - R_{v, inj}) Q_g \delta t}{R_v \cdot \varphi \cdot V \cdot (1 - S_w)} \quad \text{Eq. 31.7}$$

where

R_v is the vaporized oil-gas ratio,

$R_{v, up}$ is the R_v in the upstream cell,

$R_{v, inj}$ 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 $VOL \rightarrow 0$, $\delta t \rightarrow 0$ the differential equation for Gi in one dimension becomes:

$$\frac{d}{dt} Gi = \frac{F_g}{\varphi \cdot R_v A (1 - S_w)} \frac{dR_v}{dx} \quad \text{Eq. 31.8}$$

where

A is the flow area.

Data generation procedure

The model requires two-dimensional tables (P, Gi) to be entered for the saturated R_s , R_v , B_o and B_g data tables spanning the range of pressures and Gi parameter values encountered in the simulation. Dependence of the phase viscosities on Gi is assumed to be negligible. Variations of the surface oil and gas gravities with Gi are also ignored.

The new PVT tables are generated by an extension to the standard constant volume depletion procedure usually used to produce black oil PVT data. Instead of simply generating black oil tables by depleting the PVT cell, we now need to deplete the PVT cell to a pressure below the dew point and then inject dry gas into the cell in a series of steps. At each step the mixture is flashed to thermodynamic equilibrium and the excess gas withdrawn until the mixture occupies the same volume before the gas injection step. The new

values for R_v , R_s , B_o , and B_g are calculated to correspond with the updated value of Gi . The procedure is repeated injecting gas at a lower pressure — in each case the liquid should completely evaporate to yield the full saturated PVT curves.

It is assumed in the model that the data generated by the above method is sufficiently path independent to adequately represent the compositional effects occurring during gas injection (that is, no matter how a given point is reached in (P, Gi) space, the R_v , R_s , B_o , and B_g values will be the same as that used in the generating procedure).

Note: The Gi model with $Gi=0$ will not always behave as a black oil model without the GI option in it. According to [EQ 31.3], in a Gi Model, there will be a finite amount of Gi generation whenever there exists the combination of a finite fractional gas flux into a grid cell and a finite gas flow rate in the grid cell. Furthermore, Gi accumulates over time. In a black oil model without GI option, this does not happen.

Using the Gi pseudo-compositional model

The model should only be used 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 PVTO and PVTG 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 sat, R_v sat, B_g sat and B_o 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

The keywords are:

- [GIMODEL](#) Activates the Gi pseudo-compositional model.

PROPS section

Keyword	Description
BGGI	Variation of saturated gas FVF with pressure and Gi.
BOGI	Variation of saturated oil FVF with pressure and Gi.
GIALL	Variation of saturated properties with pressure and Gi.
GINODE	Gi nodal values.
RPTPROPS	Controls output from the PROPS section. Arguments GINODE and so on output 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

- [BGI](#) Block Gi value

SCHEDULE section

Keyword	Description
RPTSCHED	Controls output from the SCHEDULE section. Argument GI outputs the current Gi values.

Example problem

This problem illustrates the use of the option. The grid is one 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 =====
NOGFF

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 /  
  
RPTSQL  
'PRES'   'SOIL'   'SWAT'   'SGAS'   'RS'    'RV'    'FIP'  
'EQUIL'  'RSVD'  'GI'   /
```

SCHEDULE

```
SCHEDULE  ======  
WELSPCS  
'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 /  
/  
  
RPTSCHE  
'PRES'   'SOIL'   'SWAT'   'SGAS'   'RS'    'RV'    'RESTART=2'  
'FIP'    'WELLS=2'  'SUMMARY=2'  'CPU=2'  
'NEWTON=2' /  
  
TSTEP  
10*20.0 /  
  
END
```

32

Gradient option

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 are usually set using [HMMULTXX](#) on the various gradient regions, set in [HMxxxxxx](#), unless otherwise specified.

- X-direction transmissibility multipliers input through parameter HMMULTX applied to region HMTRANX
- Y-direction transmissibility multipliers input through parameter HMMULTY applied to region HMTRANY
- Z-direction transmissibility multipliers input through parameter 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 parameter HMMULTPV applied to region HMPORVM
- Analytic aquifer strengths through keywords [HMMLCTAQ](#) applied to [HMAQUCT](#) (Carter Tracy), and [HMMLFTAQ](#) applied to [HMAQFET](#) (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 parameter HMMLTPX, applied to region as in HMPERMX.
- Y-direction permeability multipliers input through parameter HMMLTPY, applied to region as in HMPERMY.
- Z-direction permeability multipliers input through parameter HMMLTPZ, applied to region as in HMPERMZ.

- X and Y-direction permeability multipliers input through `HMMMLTPXY`, 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 `HMMMRREGT` applied to `HMMULRGT`.
- Multipliers on the well productivity index using keyword `HMWPIMLT`.
- Multipliers on the well connection transmissibility factor and skin using the keyword `HMMLTWCN` applied to the connections specified in `HMWELCON`.

All the above parameters may be used with ECLIPSE 100. In this release of ECLIPSE 300 only pore volume, transmissibility and permeability parameters are available.

The derivatives are written to a file (extension .HMD) which can be interpreted by SimOpt.

The data contained within the .HMD file is generally encoded and is meaningless unless the SimOpt program is used. A facility for unencoded gradients is also available and is controlled by the licensing system. Please contact your ECLIPSE account manager for details.

Output from the gradient option

Gradients of observable well quantities (for example, WBP, WOPR) are written to the HMD file at each 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 ECLIPSE account manager for details. The arguments available in the GRADRESV keyword are:

Argument	Description
SWAT	Water saturation
SGAS	Gas saturation
SOIL	Oil saturation
RS	Dissolved gas-oil ratio

Argument	Description
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 for which gradients are calculated in the [RUNSPEC](#) keyword [HMDIMS](#).

You should supply region tables in the [REGIONS](#) section using keywords [HMPORVM](#), [HMTRANX](#) etc. to delimit the property parameter regions on the simulation grid.

ECLIPSE 100 only

Alternatively, for faults which have been specified using the [FAULTS](#) keyword, property parameters may be requested using the [HMFAULTS](#) keywords in the [GRID](#) section.

ECLIPSE 100 only

Gradients for analytic aquifer properties may be requested by using the keywords [HMAQUFET](#) or [HMAQUCT](#) in the [SOLUTION](#) section, whereas gradients for numerical aquifer properties may be requested by using the keyword [HMAQUNUM](#) in the [GRID](#) section.

ECLIPSE 100 only

Gradients for rock compaction and compressibility parameters can be obtained by specifying [HMROCK](#) and [HMROCKT](#) in the [PROPS](#) section.

ECLIPSE 100 only

To obtain gradients of the solution with respect to inter-region transmissibility multipliers declared using keyword [MULTREGT](#) in the [GRID](#) section, the [HMMULRGT](#) keyword can be used in the [GRID](#) section.

ECLIPSE 100 only

To obtain gradients of the solution with respect to well connection transmissibility factor and skin multipliers declared using keyword [HMMLTWCN](#) in the [SOLUTION](#) section, the [HMWELCON](#) keyword can be used in the [SOLUTION](#) section.

For each gradient property parameter, ECLIPSE calculates derivatives of well-flow quantities (as specified with the [GRADWELL](#) keyword) for all active wells. This information is written to the file `ROOT.HMD`, where `ROOT` is the base name of the run.

ECLIPSE 100 only

Additional information can be written out to the `ROOT.HMD` file if requested; for example, refer to the [GRADRESV](#) keyword.

ECLIPSE 100 only

If a satisfactory match has been obtained using modifiers based on the gradient analysis, then the model may be used for prediction without the overhead in calculating the derivatives, simply by specifying the [NOHMD](#) (or [NOHMO](#)) keyword in the [SCHEDULE](#) section (which switches off all gradient calculations). This is the mechanism for a ‘evaluation’ run in SimOpt.

Current restrictions

- ECLIPSE 100 only* • 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 multisegment 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

Example 1

To set three transmissibility multiplier parameters (with NX=3 NY=3 NZ=3), set the number of parameters, gradient regions and subregions using RUNSPEC keyword [HMDIMS](#)

```
.....  
HMDIMS  
1 1 3 /  
.....
```

and set appropriate keywords in the REGIONS section

```
.....  
HMTRANZ  
3*0 3*1 3*0 /  
COPY  
'HMTRANZ' 'HMTRANX' /  
/  
EQUALS  
.....
```

```
'HMTRANY' 9*1 /
/
```

Example 2

To set one pore volume multiplier parameter (with NX=3 NY=3 NZ=3), set the number of parameters, gradient regions and subregions using RUNSPEC keyword HMDIMS

```
HMDIMS
1 1 1 /
```

and set appropriate keyword in the REGIONS section

```
HMPORVM
3*0 3*1 3*0 /
```

Data file overview

RUNSPEC section overview

The following keywords are specific to the gradient option

Keyword	Description
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:

Keyword	Description
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:

Keyword	Description
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
HMSUMLINS	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 AUCT, AQUFETP or AQNUM (this may be defaulted to refer to all aquifers,

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 HMAQNUM 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 arguments are:

Keyword	Description
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:

Argument	Description
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 /
/
```

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](#)"). 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)

1. Current time
2. Last timestep used

Subsequent records are written in the order shown in the following figure:

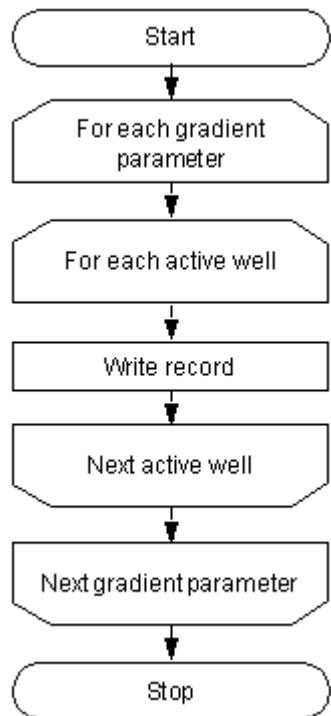


Figure 32.1. Order for writing records

Record 'NAMES'

This record consists of 5 items (CHARACTER*8)

1. Well name
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
3. Local grid name or blank for a well in the global grid
4. Status of the well:
'PRODUCER' or 'INJECTOR'
5. Status of the well: 'OPEN' or 'STOPPED' or 'SHUT'

Note: Item 2: Only REGIONS section keyword HMPORVM is available at present.

Record 'HMHEADI'

This record consists of three items (INTEGER)

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 HMAQNUM - 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

2. The type of Gradient parameter:

1 HMTRANX

2 HMTRNXY

32 HMTRANY

4 HMTRANZ

5 HMSIGMA

ECLIPSE 300

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
 - 306 HMKRG
 - 307 HMKRO
 - 309 HMKRWR
 - 310 HMKRGR
 - 311 HMKRORG
 - 312 HMKRORW
 - 313 HMSWLP
 - 314 HMSGLPC
 - 315 HMPCW
 - 316 HMPCG
 - 1041 HMROCK
 - 1042 HMROCKT
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

1. Oil rate at surface conditions
UNITS: sm³/day (METRIC), stb/day (FIELD), scc/hr (LAB)

ECLIPSE 300

2. Water rate at surface conditions
UNITS: sm³/day (METRIC), stb/day (FIELD), scc/hr (LAB)

ECLIPSE 300

3. Gas rate at surface conditions
UNITS: sm³/day (METRIC), Mscf/day (FIELD), scc/hr (LAB)

ECLIPSE 300

4. Water cut
UNITS: sm³/sm³ (METRIC), stb/stb (FIELD), scc/scc (LAB)

ECLIPSE 300

5. Gas Oil ratio
UNITS: sm³/sm³ (METRIC), Mscf/stb (FIELD), scc/scc (LAB)
6. 1 point pressure average (see [WPAVE](#) and [WWPAVE](#))
UNITS: barsa (METRIC), psia (FIELD), atma (LAB)
7. 4 point pressure average (see [WPAVE](#) and [WWPAVE](#))
UNITS: barsa (METRIC), psia (FIELD), atma (LAB)
8. 5 point pressure average (see [WPAVE](#) and [WWPAVE](#))
UNITS: barsa (METRIC), psia (FIELD), atma (LAB)
9. 9 point pressure average (see [WPAVE](#) and [WWPAVE](#))
UNITS: barsa (METRIC), psia (FIELD), atma (LAB)

This can be changed: see the comments below on [GRADWELL](#).

Record 'VALUES'

This record consists of a variable number of items (DOUBLE PRECISION).

This record contains the values of the well quantities referred to in the 'DERIVS' record. This is only output once for each well at each step, so if there are several Gradient parameters, the values are only output for the first parameter. The reported rates are production rates, so a negative sign indicates injection of the pertinent phase. When a well is shut, the 'DERIVS' and 'VALUES' records are not calculated and may be reported as zeroes.

ECLIPSE 100 only

As writing all this information can be quite voluminous, the amount of output may be configured using the [RPTHMD](#) keyword (to supply output only for working producers, etc.) or by the [RPTHMW](#) keyword (to switch off output for nominated wells), or only at report times, etc.

This data is used by SimOpt. Generally, the derivatives are unformatted and encoded (though the 'VALUES' record is not encoded). The 'CODESTAT' record outputs whether the file is encoded or not. SimOpt must be used to interpret the derivatives. However, the output of unencoded derivative information is controlled by the password mechanism for ECLIPSE software - please contact your ECLIPSE account manager for more information.

The following records provide version control:

- ECLIPSE 100 only

The records E100VER, E100INFO and E100ORIG provide version control. E100VER is the version of ECLIPSE; E100INFO lists the date and time at which the HMD file was created together with the dataset root name; and E100ORIG reports the date and time at which the current copy of ECLIPSE was built.

ECLIPSE 300 only

- The records E300VER, E300INFO and E300ORIG provide version control. E300VER is the version of ECLIPSE; E300INFO lists the date and time at which the HMD file was created together with the dataset root name; and E300ORIG reports the date and time at which the current copy of ECLIPSE was built.
- The convert utility may be used to convert from unformatted to formatted if required; the formatted HMD file has the extension FHMD (UNIX) or FHM (PC). The RUNSPEC keyword **FMTHMD** requests a formatted Derivatives file.

Alternative output of well quantities

The above default quantities loaded into the 'DERIVS' and 'VALUES' records may be inappropriate. The **GRADWELL** keyword allows you to specify a list of mnemonics (such as WBP, WWCT and WBHP) 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 Open ECLIPSE (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 ' 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 and densities for example. These may be used to form an objective function in conjunction with a Petro-elastic Model in order to compare predicted seismic response. Such a case might be the calculation of seismic impedance.

Furthermore, it is now possible to request output of gradients of pressure-wave impedance and shear-wave impedance, if the appropriate keywords have been specified in the dataset to utilize the Petro-elastic Model inside the simulator. The information output is controlled by the password mechanism for ECLIPSE software - please contact your ECLIPSE account manager for more information.

Output of derivatives of grid block pressures and saturations for example, 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:

1. Parameter name
 2. Keyword name associated with parameter
 - ‘ GRADZONE ’ for regional parameters defining gradzones
 - ‘ FAULTS ’ for faults
 - ‘ AUCTS ’/‘ AQUFET ’ for aquifers
 - ‘ SATNUM ’ relative permeability shape parameters
 - ‘ PVTNUM ’/‘ SATNUM ’/‘ ROCKNUM ’ for rock parameters
 - ‘ MULTNUM ’ for inter-region transmissibility multipliers
 3. Feature name: Name of fault for fault parameters
 - Blank for regional parameters
 - Blank for all other parameters
 4. Parameter type. This is described under HMHEADI.
 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
 6. Associated phase. This is a pointer internal to ECLIPSE.
 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 formulation

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](#)”).

This can be written as

$$R(X_{t+dt}, X_t, \lambda) = \frac{M_{t+dt}(X_{t+dt}, \lambda) - M_t(X_t, \lambda)}{dt} + F_{t+dt}(X_{t+dt}, \lambda) + Q_{t+dt}(X_{t+dt}, \lambda) = 0 \quad \text{Eq. 32.1}$$

where

the subscripts t and $t + dt$	denote quantities at the current and advanced 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,
λ	is any gradient parameter,
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 equation 32.1 throughout with respect to λ yields

$$0 = \frac{dR}{d\lambda} = \frac{\partial R}{\partial \lambda} + \frac{\partial R}{\partial X_{t+dt}} \cdot \frac{\partial X_{t+dt}}{\partial \lambda} + \frac{\partial R}{\partial X_t} \cdot \frac{\partial X_t}{\partial \lambda} \quad \text{Eq. 32.2}$$

which can be written as

$$J \cdot \frac{\partial X_{t+dt}}{\partial \lambda} = -\frac{\partial R}{\partial X_t} \cdot \frac{\partial X_t}{\partial \lambda} - \frac{\partial R}{\partial \lambda} \quad \text{Eq. 32.3}$$

where the Jacobian $J = \partial R / \partial X_{t+dt}$ is the same as that used to solve the reservoir equations, so no additional matrix setup is required. Therefore, for each gradient parameter λ selected, a set of linear

equations, equation 32.3 must be solved at each simulator timestep to determine the gradient $\frac{\partial X_{t+dt}}{\partial \lambda}$. When there are several gradient parameters, each requires the solution of the same set of linear equations, equation 32.3 but with different right hand sides. The right hand sides contain two terms. The first is the product of $\partial R / \partial X_t$, which is independent of λ , and the previously calculated gradient at the old timestep, $\partial X_t / \partial \lambda$.

For a fully implicit discretization the second term in the right hand side of equation 32.3 is given by:

$$\frac{\partial R}{\partial \lambda} = \frac{1}{dt} \left(\frac{\partial M_{t+dt}}{\partial \lambda} - \frac{dM_t}{d\lambda} \right) + \frac{\partial F_{t+dt}}{\partial \lambda} + \frac{\partial Q_{t+dt}}{\partial \lambda} \quad \text{Eq. 32.4}$$

Evaluation of equation 32.4 depends on the parameter λ , and additional code is required for each new parameter. If, for example λ is a pore volume multiplier, then it would appear linearly in the accumulation terms, and not appear in the flow or well terms. Therefore equation 32.4 could be evaluated as:

$$\frac{\partial R}{\partial \lambda} = \frac{1}{dt} \left(\frac{\partial M_{t+dt}}{\partial \lambda} - \frac{dM_t}{d\lambda} \right) = \frac{1}{\lambda} \frac{(M_{t+dt} - M_t)}{dt} \quad \text{Eq. 32.5}$$

where the right hand side has already been calculated in evaluating the residual equation 32.1.

Regression in SimOpt using ECLIPSE gradients

The solution $\partial X / \partial \lambda$ of equation 32.4 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 and well flow rates. 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 and HMKRW, in [HMxxxxxx](#)). During a gradient-based minimization loop, a sequence of multiplicative modifiers λ_i , $i = i, \dots, v$ may be applied to the base dataset whose effect is cumulative, so that after v iterations, the cumulative modifier is

$$\Lambda_n = \prod_{i=1}^n \lambda_i$$

If the regression requires gradients with respect to the total modifier Λ_n , then

$$\frac{\partial X}{\partial \Lambda_n} = \frac{\partial X}{\partial (\Lambda_{n-1} \lambda_n)} = \frac{1}{\Lambda_{n-1}} \cdot \frac{\partial X}{\partial \lambda_n}$$

In SimOpt, the regression operates on the parameter Λ_n and the sensitivities provided by ECLIPSE are divided by Λ_n at each regression iteration. For regression loops such as SimOpt relying on the cumulative parameters Λ_n in this way, it is suggested that the log transform $\lambda = \ln(\lambda)$ is applied.

ECLIPSE 100 only

For parameters whose cumulative modifiers are additive (such as HMSWCR and HMSOWCR, in [HMxxxxxx](#)), the cumulative modifier would be

$$\Lambda_n = \sum_{i=1}^n \lambda_i$$

in which case no modification is required because

$$\frac{\partial X}{\partial A_n} = \frac{\partial X}{\partial (A_{n-1} + \lambda_n)} = \frac{\partial X}{\partial \lambda_n}$$

For any objective function $G = G(X, \lambda)$, used during regression, then

$$\frac{dG}{d\lambda} = \sum_t \frac{\partial G(X_t, \lambda)}{\partial X_t} \cdot \frac{\partial X_t}{\partial \lambda} + \frac{\partial G}{\partial \lambda}$$

so once $\partial X_t / \partial \lambda$ is known then regression can be performed for any objective function G .

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.

Instantaneous gradients

The instantaneous gradient facility provides an alternative set of derivatives to those generated for history matching purposes. Recall the history matching philosophy in which derivatives of various solution quantities at the current timestep are computed with respect to variations in a set of parameters at time zero. In contrast, the instantaneous gradient option computes derivatives of solution quantities at the current timestep with respect to variations in the parameters at the current timestep. The instantaneous gradient option harnesses the existing ECLIPSE gradient code with the exception that the gradient data need not be propagated forward in time. As a result the instantaneous gradient option can be switched on and off during the course of a simulation via the [GUPFREQ](#) keyword.

The aim of the instantaneous gradient feature is to provide information on the sensitivity of a well to its, and other wells', control modes. For example, if the well P1 is under bottom hole pressure control the gradient of the oil flow rate with respect to the bottom hole pressure can be requested. The calculation of such a gradient combines two factors, the response of the well to a variation in its own bottom hole pressure, and the response of the reservoir to the variation in the well's bottom hole pressure. Since the response of the reservoir is part of the gradient calculation the interaction between wells can also be studied. For example, the relationship between the water flow rate of well P1 and the water injection rate of well I1. However, it should be noted that a number of approximations are made within the gradient calculation that can reduce the accuracy of the calculation when considering the influence of one well upon another.

The instantaneous gradient options are activated via the [GDIMS](#) keyword in the RUNSPEC section. Since the gradient output is available as summary data an unencoded gradient license is required. In the summary section the gradient of the oil, water, gas flow rates and the bottom hole pressure with respect to a gradient parameter can be requested with the keywords WOFWCn, WWFWCn, WGFWCn and WBHWCn, where n is the parameter number required. The gradient parameters are defined by specifying wells using the [GWRTWCV](#) keyword in the SCHEDULE section. Wells selected as gradient parameters must be under either oil, water, gas rate or bottom hole pressure control since the response of the group hierarchy is not included in the gradient calculation. In addition the feature is restricted to the standard well model in unrefined grids.

Example usage

Specify two instantaneous gradient parameters in the RUNSPEC section.

```
GDIMS
2 /
```

Request gradient output in the SUMMARY section. The first item requests that the gradient of the oil flow rate for wells P1 and P2 with respect to the instantaneous gradient parameter 1. The second item that the water flow rate for wells P1 and I1 with respect to the instantaneous gradient parameter 2.

```
WOFWC1
P1 P2 /
WWFWC2
P1 I1 /
```

Specify the instantaneous gradient parameters in the SCHEDULE section. Define the first parameter to be the control mode of well P1, the second to be the control mode of well I1. Compute the gradients at the next timestep only. The values of the requested instantaneous gradient summary mnemonics will be frozen at the last computed value.

```
GWRTWCV
P1 / -- gradient parameter 1
I1 / -- gradient parameter 2
/
GUPFREQ
NEXT /
```

At a later point in the simulation re-specify the instantaneous gradient parameters and compute the gradients for all subsequent timesteps.

```
GWRTWCV
P2 / -- gradient parameter 1
I2 / -- gradient parameter 2
/
GUPFREQ
ALL /
```

Summary of keywords

RUNSPEC section

Keyword	Description
FMTHMD	Declares that the HMD file is to be formatted.
GDIMS	Specifies dimensions for the instantaneous gradient option.
HMDIMS	Specifies dimensions for the gradient option.
RPTHMD	Specifies controls on the output to the HMD file.
UNCODHMD	Requests that the HMD file contains unencoded derivatives.

Parameter specification keywords

GRID section

Keyword	Description
HMFAULTS	Specifies gradients for faults.
HMMULRGT	Specifies inter-region transmissibility parameters.
HMAQNUM	Specifies gradients for numerical aquifer parameters.

PROPS section

- [HMROCK](#) Specifies rock compressibility parameters.
- [HMROCKT](#) Specifies rock compaction parameters.

REGIONS section

[HMxxxxxx](#) Specifies gradient parameters. The following options apply to the [HMxxxxxx](#) keyword.

Restriction	Argument	Description
	HMTRANX	Specifies X-direction transmissibility parameters.
	HMTRANY	Specifies Y-direction transmissibility parameters.
	HMTRANZ	Specifies Z-direction transmissibility parameters.
	HMTRNXY	Specifies X- and Y-direction transmissibility parameters.
	HMTRANR	Specifies R-direction transmissibility parameters.
	HMTRANT	Specifies theta-direction transmissibility parameters.
	HMPORVM	Specifies pore volume parameters.
E100 only	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.

Restriction	Argument	Description
	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.
	HMSGGLPC	Specifies lower gas saturation for capillary pressure parameter.
	HMPCW	Specifies maximum water capillary pressure parameter.
	HMPCG	Specifies maximum gas capillary pressure parameter.

SOLUTION section

Keyword	Description
HMAQUCT	Specifies gradients for Carter-Tracy aquifers.
HMAQUFET	Specifies gradients for Fetkovich aquifers.
HMWELCON	Specifies gradients for well connections

SCHEDULE section

Keyword	Description
GUPFREQ	Specifies update frequency for the instantaneous gradient option.
GWRTWCV	Specifies parameters for the instantaneous gradient option.
HMWPIMLT	Specifies gradients with respect to well productivity index.

Parameter modifier keywords

The cumulative multipliers for gradient parameters are given by the [HMMULTxx](#) keyword. The following table shows which section they can be used in together with the usage of the other parameter modifier keywords.

Section	Keyword	Description
GRID and EDIT	HMMULTFT	Specifies modifiers for fault transmissibilities.
GRID	HMMULTSG	Specifies multipliers on dual porosity sigma factor.
GRID and EDIT	HMMULTPV	Specifies pore volume modifiers.
GRID and EDIT	HMMULTX	Specifies X-direction transmissibility modifiers.
GRID and EDIT	HMMULTY	Specifies Y-direction transmissibility modifiers.
GRID and EDIT	HMMLTXY	Specifies XY-direction transmissibility modifiers.
GRID and EDIT	HMMULTZ	Specifies Z-direction transmissibility modifiers.
GRID and EDIT	HMMULTX-	Specifies negative X-direction transmissibility modifiers.

Section	Keyword	Description
GRID and EDIT	HMMULTY-	Specifies negative Z-direction transmissibility modifiers.
GRID	HMMREGT	Specifies inter-region transmissibility modifiers.
GRID	HMMULTR	Specifies R-direction transmissibility modifiers.
GRID and EDIT	HMMULTTH	Specifies theta-direction transmissibility modifiers.
GRID	HMMLTPX	Specifies X-direction permeability modifiers.
GRID	HMMLTPY	Specifies Y-direction permeability modifiers.
GRID	HMMLTPZ	Specifies Z-direction permeability modifiers.
GRID	HMMLTPXY	Specifies XY-direction permeability modifiers.
EDIT	HMMULTTH	Specifies theta-direction transmissibility modifiers.
GRID	HMMULTSG	Specifies multipliers on dual porosity sigma factor.

PROPS section

[HMPROPS](#) Section heading for property modifiers. The following options apply to the HMPROPS keyword.

Argument	Description
HASWCR	Specifies critical water saturation modifiers.
HASOWCR	Specifies critical oil-in-water saturation modifiers.
HASOGCR	Specifies critical oil-in-gas saturation modifiers.
HASWL	Specifies connate water saturation modifiers.
HMKRW	Specifies maximum water relative permeability modifiers.
HMKRG	Specifies maximum gas relative permeability modifier.
HMKRO	Specifies maximum oil relative permeability modifier.
HMKRWR	Specifies maximum displacing water relative permeability modifier.
HMKRGR	Specifies maximum displacing gas relative permeability modifier
HMKRORG	Specifies maximum displacing oil-in-gas relative permeability modifier.
HMKRORW	Specifies maximum displacing oil-in-water relative permeability modifier.
HASWLPC	Specifies modifier on lower water saturation used for water capillary pressure.
HASGLPC	Specifies modifier on lower gas saturation used for gas capillary pressure.
HMPCW	Specifies modifier on water capillary pressure.
HMPCG	Specifies modifier on gas capillary pressure.

SOLUTION section

Keyword	Description
HMMLCTAQ	Specifies modifiers for Carter-Tracy aquifers.
HMMLFTAQ	Specifies modifiers for Fetkovich aquifers.
HMMLTWCN	Specifies modifiers for well connections

SUMMARY keywords

Refer to [Simulator performance keywords](#) in the *ECLIPSE Reference Manual*.

Miscellaneous keywords

Keyword	Description
GRADRESV	Specifies derivatives of solution quantities to be output to the RESTART file.
GRADRFT	Specifies derivatives of RFT quantities to be output to the RFT file.
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 [PVTD](#) 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 [PVTO](#) 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 and gradient region for example.

```
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 1 2 1 2 1 2 1 2  
1 2 1 2 1 2 1 2 1  
2 1 2 1 2 1 2 1 2  
1 2 1 2 1 2 1 2 1  
2 1 2 1 2 1 2 1 2  
1 2 1 2 1 2 1 2 1  
2 1 2 1 2 1 2 1 2  
1 2 1 2 1 2 1 2 1  
2 1 2 1 2 1 2 1 2  
1 2 1 2 1 2 1 2 1  
2 1 2 1 2 1 2 1 2  
1 2 1 2 1 2 1 2 1  
2 1 2 1 2 1 2 1 2  
  
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
'PVNUM'  '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'   'DEN0'  '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 timestep 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.

33

Group production rules

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

This chapter gives details of the implementation of the [PRORDER](#) keyword actions, which are performed by ECLIPSE 100 whenever a group fails to meet its target production rate set in keyword [GCONPROD](#). This feature is currently not available in ECLIPSE 300.

The available actions are:

1. Drill new wells from the drilling queue
2. Open new connections on AUTO in existing wells
3. Reduce each well's THP limit to a specified lower value
4. Change VFP table numbers (for example to model a retubing operation)
5. Change well ALQ values (that is apply a fixed amount of artificial lift)
6. Switch on, or increase power of, compressors linked to the group. (This option is only available if the Extended Network model is in use - see "[The extended network model](#)").

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 [WDRILTIME](#) item 2) and all drilling rigs available to its group (keyword [GRUPRIG](#)) are already occupied for the whole of the current timestep
- 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 timestep.

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 timestep 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 [WELSMIN](#). 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 timestep, 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 “[Network option](#)”), 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](#)"). This offers a much more sophisticated means of optimizing 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 timestep, 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 timestep.

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](#)"). 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 multilevel. Single-level compressors switch directly to the new ALQ value when they are

turned on. Multilevel 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 multistage 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 multilevel 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 timestep if all available drilling (or workover) rigs are occupied for the whole timestep.

By default, a well starts flowing at the beginning of the timestep 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 timestep 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 timestep 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 “[Network option](#)”), 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.

34

Hysteresis

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

This chapter describes the hysteresis option. This option enables different saturation functions to be specified 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 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 end-points 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 instead follows a scanning curve generated by the hysteresis model employed. The way in which this hysteresis process is treated in ECLIPSE follows the techniques described by Aziz and Settari [Ref. 8], Killough [Ref. 64], Carlson [Ref. 16], Land [Ref. 66] and Jargon (this hysteresis method is unpublished).

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](#), or the directional variants, are 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](#), or the directional variants, 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, capillary pressures, or both.

The keyword `EHYSTR` in the PROPS section is specific to hysteresis. This keyword 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.

Features of the ECLIPSE hysteresis model include:

- The reservoir is always equilibrated using the drainage curves.
- If the Hysteresis option is used in a non-VE run with a `COMP RP` or `COMP VE` 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 `COMP DAT` and `COMP IMB`, and also with the `WELL COMP` keyword.
- Capillary pressure hysteresis is always applied using the Killough method [Ref. 64].

ECLIPSE 100

ECLIPSE 100

The hysteresis option cannot be used in combination with the vertical equilibrium option in the same run.

The hysteresis option cannot be used in combination with the surfactant model.

Relative permeability hysteresis may be modeled by the Carlson [Ref. 16] or Killough [Ref. 64] methods but not the method of Jargon.

Relative permeability hysteresis in the non-wetting phase

A typical pair of relative permeability curves for a non-wetting phase is shown in figure 34.1. The curve 1 to 2 represents the user-supplied drainage relative permeability table, and the curve 2 to 3 represents the user-supplied imbibition relative permeability table. (Note that the non-wetting phase saturation increases from right to left in this diagram). The critical saturation of the imbibition curve, S_{ncri} , is greater than that of the drainage curve, S_{ncrd} . The two curves **must** meet at the maximum saturation value, S_{nmax} .

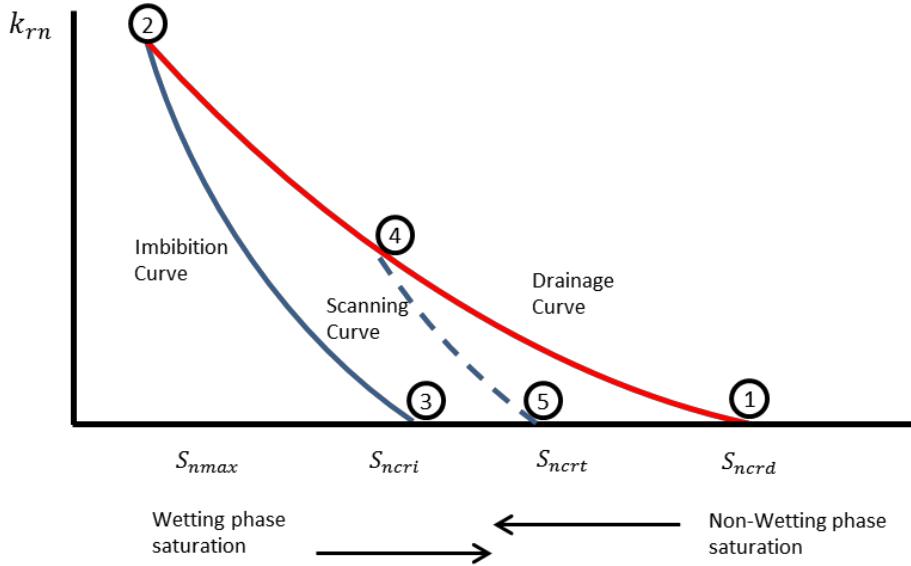


Figure 34.1. A typical pair of relative permeability curves for a non-wetting phase

The primary drainage curve is for a process which starts at the maximum possible wetting phase saturation, S_{wmaxd} . (This value will depend upon the end points of the saturation tables specified using the [SATNUM](#) keyword.) If the wetting phase saturation decreases to S_{wmin} , this primary drainage curve is used.

In a similar way, if the initial saturation is S_{wmin} , and the wetting phase saturation increases to S_{wmaxi} , the imbibition table data will be used. (The maximum wetting phase saturation which can be reached, S_{maxi} , 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. 16] or Killough's method [Ref. 64]. 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 or Killough's method or J. Jargon's method (unpublished). 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

$$S_{ncrt} = S_{ncrd} + \frac{S_{hy} - S_{ncrd}}{1 + C(S_{hy} - S_{ncrd})} \quad \text{Eq. 34.1}$$

where

$$C = \frac{1}{S_{ncri} - S_{ncrd}} - \frac{1}{S_{n \max} - S_{ncrd}} \quad \text{Eq. 34.2}$$

(Killough's formulae have been adapted to allow for non-zero values of S_{ncrd} .)

The relative permeability for a particular saturation S_n on the scanning curve is

$$K_{rn}(S_n) = \frac{K_{rni}(S_{norm})K_{rnd}(S_{hy})}{K_{rnd}(S_{n \max})} \quad \text{Eq. 34.3}$$

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

$$S_{norm} = S_{ncri} + \frac{(S_n - S_{ncri})(S_{n \max} - S_{ncri})}{S_{hy} - S_{ncrt}} \quad \text{Eq. 34.4}$$

With Killough's method S_{ncrt} will always lie between S_{ncrd} and S_{ncri} . But if the drainage and imbibition curves are made to coincide, the scanning curve will not necessarily follow this combined curve, except at its end points.

ECLIPSE 300

If the [DRAINAGE](#) keyword is specified, the scanning curve will be forced always to lie below the drainage curve.

The Killough method described above has been modified slightly to overcome an un-physical behavior when S_{hy} is small. As S_{hy} tends to S_{ncrd} in equation 34.1, tends to S_{ncrd} , which leads to an infinite imbibition relative permeability gradient. This in turn can cause convergence problems, while having little material effect on the simulation results. Therefore equation 34.1 has been modified as follows:

$$S_{ncrt} = S_{ncrd} + \frac{S_{hy} - S_{ncrd}}{A + C(S_{hy} - S_{ncrd})} \quad \text{Eq. 34.5}$$

where

$$A = 1 + a \cdot (S_{n \max} - S_{hy}) \quad \text{Eq. 34.6}$$

The parameter a is input through the fourth data item in the [EHYSTR](#) keyword and defaults to 0.1.

Jargon's method for relative permeability hysteresis

ECLIPSE 300

In this model the trapped saturation S_{ncrt} is constructed by moving the drainage critical saturation towards the imbibition critical saturation by the same fraction that the hysteresis saturation has moved towards the maximum non-wetting saturation

$$S_{ncrt} = S_{ncrd} + \frac{(S_{ncri} - S_{ncrd})(S_{hy} - S_{ncrd})}{(S_{nmax} - S_{ncrd})} \quad \text{Eq. 34.7}$$

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 is determined

$$R(X) = \frac{K_{ri}(S_n)}{K_{rd}(S_n)} \quad \text{Eq. 34.8}$$

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

$$X = \frac{S_n - S_{ncrd}}{S_{nmax} - S_{ncrd}} \quad \text{Eq. 34.9}$$

Clearly X lies between 0 and 1 and R goes from 0 at $X = 0$ to 1 at $X = 1$, where the two curves meet. If the drainage curve is always above the imbibition curve, R also must lie between 0 and 1.

The scanning relative permeability value is then constructed by multiplying the drainage curve value by the function R , where R is evaluated at an X -value which reflects the fractional distance of the current saturation between the drainage curve end point and the hysteresis saturation

$$K_{rn}(S_n) = R(X_s)K_{rnd}(S_n) \quad \text{Eq. 34.10}$$

$$X_s = \frac{S_n - S_{ncrd}}{S_{hy} - S_{ncrd}} \quad \text{Eq. 34.11}$$

This can also be written as

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

where

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

Differences in hysteresis treatment in three-phase cases

ECLIPSE 300

The treatment of relative permeability hysteresis differs between the Stone's method and the default option.

In the [STONE2](#) case, K_{row} is obtained at $S_n = 1 - S_w$, equal to the hydrocarbon saturation in the block. It is thus natural to take the maximum historical hydrocarbon saturation as S_{nhy} and to apply hysteresis to 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 applied 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_{nmax} . Clearly also $S_{ncri} > S_{ncrd}$ is required.

ECLIPSE 100

ECLIPSE 100 checks these consistency relationships if the `HYSTCHCK` keyword is included in the `PROPS` section.

Relative permeability hysteresis in the wetting phase

There is an option to use the Killough model for wetting phase hysteresis. Otherwise the same curve is used to obtain the wetting phase relative permeability in both drainage and imbibition processes and for which either the drainage or the imbibition curve can be selected. You make these modeling choices using item 2 of the `EHYSTR` keyword.

A typical pair of wetting phase relative permeability curves suitable for the Killough model are shown in figure 34.2. The curve 1 to 2 represents the user-supplied drainage relative permeability table, and the curve 2 to 3 represents the user-supplied imbibition relative permeability table. The two curves **must** meet at the connate saturation ($S_{wco} = 1 - S_{nmax}$). The maximum saturation on the imbibition curve is $1 - S_{ncri}$.

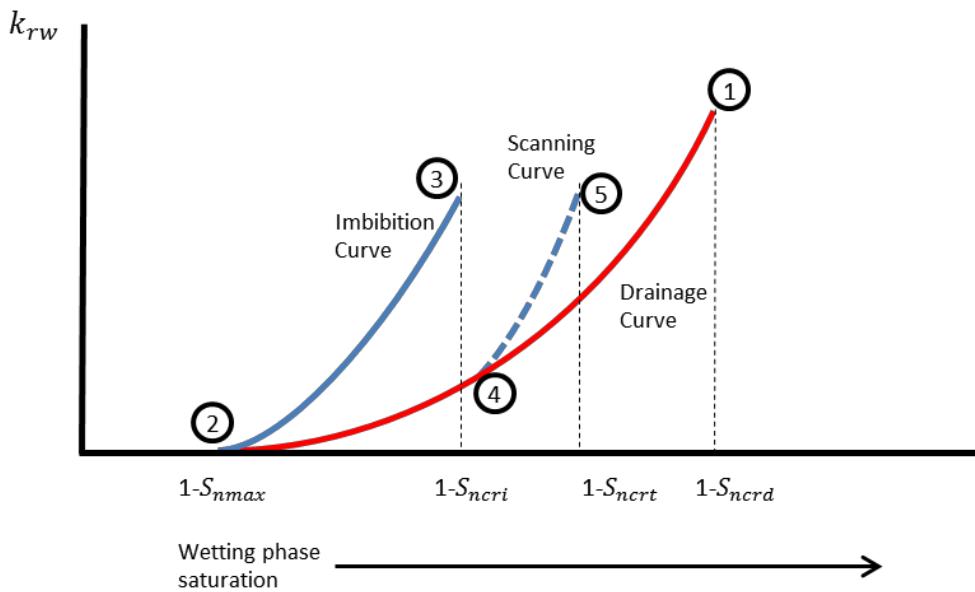


Figure 34.2. A typical pair of relative permeability curves for a wetting phase

An initial drainage process would follow the drainage curve (point 1 to point 2). An imbibition process starting at point 2, $S_w = S_{wco} = 1 - S_{nmax}$, follows the bounding imbibition curve (point 2 to point 3). Point

3, $S_w = 1 - S_{ncri}$, is the maximum wetting phase saturation that can be reached starting from S_{wco} , since the trapped non-wetting phase saturation is S_{ncri} .

An imbibition process that starts from an intermediate saturation (point 4) will follow a scanning curve (point 4 to point 5). The saturation at point 4 is $S_w = 1 - S_{hy}$, where S_{hy} is the maximum non-wetting phase saturation reached. The maximum saturation that can be reached on the scanning curve (point 5) is $S_w = 1 - S_{ncrt}$, where S_{ncrt} is the trapped critical saturation of the non-wetting phase, as defined in the previous section.

If a further drainage process begins from any point on the scanning curve, the same scanning curve is retraced until point 4 is reached, where the drainage curve is rejoined.

Killough's method for calculating the scanning curves uses some of the quantities derived in the previous section for the non-wetting phase. The trapped critical non-wetting phase saturation S_{ncrt} is determined for the particular value of S_{hy} . The wetting phase relative permeability at the complementary saturation is calculated, thus fixing the position of point 5 via

$$K_{rw}(1 - S_{ncrt}) = K_{rwd}(1 - S_{ncrt}) + (K_{rwi}(1 - S_{ncri}) - K_{rwd}(1 - S_{ncri})) \left(\frac{S_{ncrt} - S_{ncrd}}{S_{ncri} - S_{ncrd}} \right)^A \quad \text{Eq. 34.12}$$

where the exponent A is a curvature parameter entered in item 3 of the keyword **EHYSTR**

K_{rwd} and K_{rwi} represent the wetting phase relative permeability values on the bounding drainage and imbibition curves respectively.

The relative permeability for a particular saturation S_w on the scanning curve is given by

$$K_{rw}(S_w) = K_{rwd}(1 - S_{hy}) + \frac{(K_{rw}(1 - S_{ncrt}) - K_{rwd}(1 - S_{hy}))K_{rwi}(1 - S_{norm})}{K_{rwi}(1 - S_{ncri})} \quad \text{Eq. 34.13}$$

where S_{norm} is the function of $S_n = 1 - S_w$ defined in the non-wetting phase hysteresis section.

In some cases it has been observed that the path of the wetting phase relative permeability scanning curve is not contained within the region formed by the drainage and imbibition bounding curves. An example of this behavior can be observed in figure 34.3. Typically, the scanning curve, which is constructed from a transformed replica of the imbibition curve, will depart initially from this region if its initial gradient is less than that of the drainage curve at the point of departure. This tends to occur when the initial gradient of the imbibition curve is itself small.

If Item 13 of the **EHYSTR** keyword is set to 1, the scanning curve will be constructed from a reduced portion of the imbibition curve in order to mitigate this effect. For cases where the scanning curve would have otherwise remained within the region of the bounding curves this correction has very little effect. This modification is only applied to the wetting phase relative permeability hysteresis scanning curves, that is, when item 2 of the **EHYSTR** is set to 4 or 7. In extreme cases where very low initial gradients are being modeled, this modification may still result in scanning curves which depart marginally from within the region of the bounding curves.

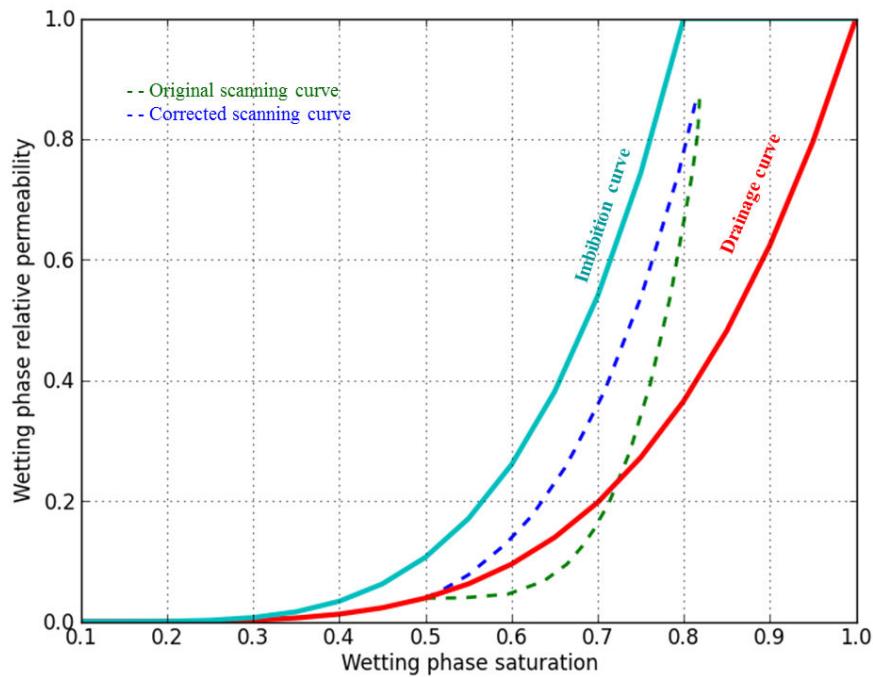


Figure 34.3. Correction of the Killough scanning curve for the wetting phase relative permeability

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 in figure 34.2).

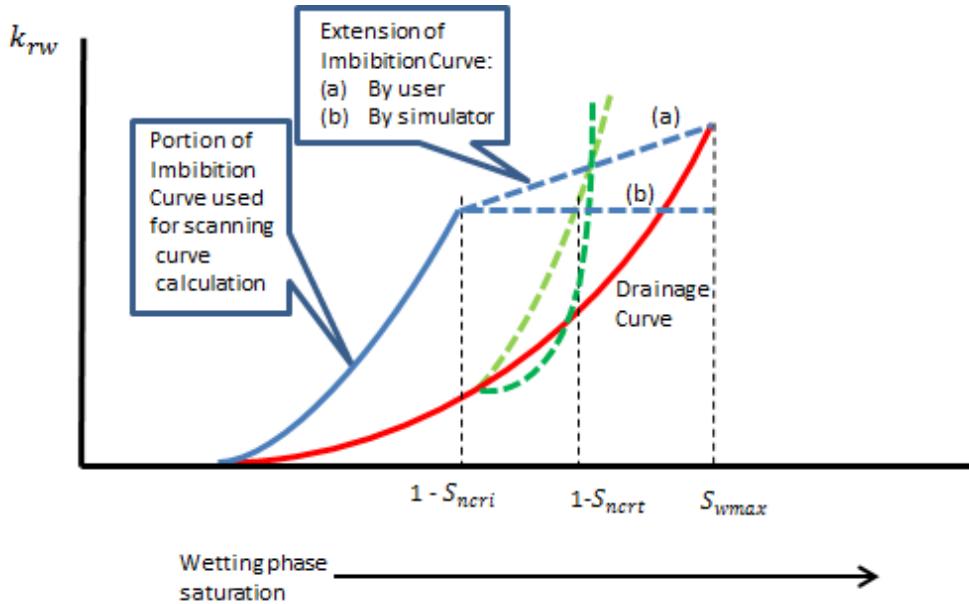


Figure 34.4. Overshoot of the Killough scanning curve for the wetting phase relative permeability

When modeling relative permeability hysteresis in the wetting phase using the Killough hysteresis model, it is important to note that the construction of the hysteresis wetting phase scanning curve is based upon the properties of the non-wetting phase critical saturations in addition to the wetting phase saturation functions. The target point for the scanning curve is determined by the non-wetting phase trapped critical saturation (point 5 in figure 34.2) and occurs at a wetting phase saturation of $1-S_{ncrt}$ for oil relative permeability to water and water relative permeability and at $1-S_{ncrt} - S_{wco}$ for oil relative permeability to gas. For example, for a water-wet system and for water relative permeability hysteresis, only the portions of the water relative permeability imbibition and drainage curves up to water saturations of $1-S_{ocri}$ and $1-S_{ocrd}$ respectively are used for constructing the scanning curve. If the imbibition and drainage curves extend beyond these saturations, for example, to form a closed loop at the maximum attainable saturation, the scanning curve may deviate beyond the region bounded by the drainage and imbibition curves as the wetting phase saturation approaches its maximum attainable value. This is illustrated in figure 34.4 where only the portion of the imbibition curve up $1-S_{ncri}$ has been used for constructing the wetting phase scanning curve, in accordance with Killough model. However, in curve (a), the imbibition curve has been extended in the input data table to the maximum wetting phase saturation to form a closed loop with the drainage curve. On the other hand, if the last saturation entry in the table were $1-S_{ncri}$, internally the saturation function would be extended horizontally for saturations up to S_{wmax} , as shown in curve (b). In the case of figure 34.4, both scanning curves, (the original Killough model (dark green)), and the scanning curve correction for Killough model (light green)), appear to deviate outside the region bounded by the imbibition curve whereas, for Killough model, this extension of the imbibition curve beyond $1-S_{ncri}$ can arguably be considered an artefact because this portion of the imbibition curve is not used by the Killough model for constructing the wetting phase scanning curve.

Note: The Carlson hysteresis model (and also the Killough hysteresis model if the wetting phase scanning curve correction has been specified) use inverse lookup of the imbibition curve to construct the scanning curve. For these cases the imbibition curve (following scaling if end-point scaling has been specified)

should be monotonically increasing. Otherwise the inverse lookup will be poorly defined and undesired results may occur during the calculation of the scanning curves.

Modeling oil wet systems

It is possible to apply the non-wetting phase hysteresis models (Carlson, 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 S_w is decreasing, the IMBNUM curves when S_w 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 saturation function data must satisfy certain relationships if the Killough wetting phase model is applied to oil (option 7, item 2 of **EHYSTR** keyword). In these cases it is required that $S_g \text{ max} = 1.0 - S_{oco}$ and also $S_w \text{ max} = 1.0 - S_{oco}$, while $S_o \text{ max}$ can be equal to 1.0, where S_{oco} is the **connate** oil saturation.

Note: One has to be aware that if $S_{oco} > 0$ then $S_w \text{ max} < 1$ 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 is applied 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 below.

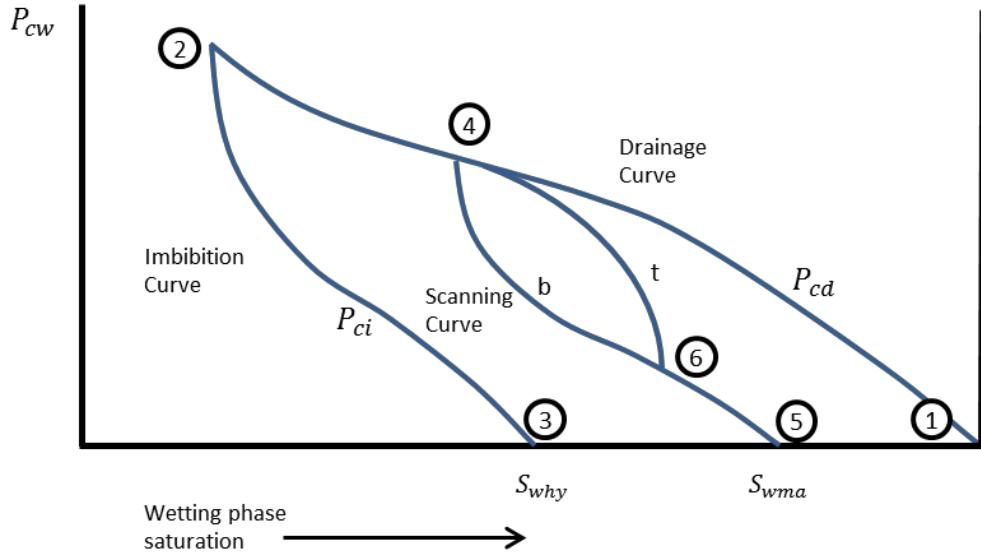


Figure 34.5. The water capillary pressure case

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 point 5. This curve is formed as a weighted average of the drainage and imbibition curves, starting with 100% of the drainage curve at point 4. The form used is that suggested by Killough, namely

$$P_c = P_{cd} + F(P_{ci} - P_{cd}) \quad \text{Eq. 34.14}$$

with

$$F = \left(\frac{1}{S_w - S_{why} + E} - \frac{1}{E} \right) / \left(\frac{1}{S_{wma} - S_{why} + E} - \frac{1}{E} \right) \quad \text{Eq. 34.15}$$

in which

E is a curvature parameter of the order of 0.1.

S_{why} is the water saturation at the hysteresis reversal point 4, that is the minimum historical water saturation in the cell

S_{wma} is the maximum water saturation attainable allowing for the trapped non-wetting phase saturation (that is at $S_n = S_{ncrt}$).

S_{wma} is $1 - (S_{ocr} - S_{gcrt})$, the critical gas saturation reflecting the trapping saturation due to relative permeability hysteresis (depending on the choice of Carlson's Killough's or Jargon's method).

This occurs when $S_w \rightarrow S_{why}$, $F \rightarrow 0$ and $P_{cw} \rightarrow P_{cwd}$ and when $S_w \rightarrow S_w^{\max}$, $F \rightarrow 1$ and $P_{cw} \rightarrow P_{cwi}$. The curvature parameter E can be specified by item 1 of the `EHYSTR` keyword.

The imbibition capillary pressure P_{ci} in equation 34.14 is obtained by looking up the capillary pressure on the imbibition curve after scaling between the end-points S_{wcri} and S_{wma} , where S_{wcri} is the critical water saturation on the imbibition curve.

Suppose that a second reversal, back to a drainage process, occurs at point 6, rather than continuing to S_{wma} . Unlike the case of relative permeability hysteresis, this does not necessarily re-traverse the scanning curve (marked by a ‘b’ in the above diagram), but may follow a new drainage to imbibition to drainage scanning curve that returns to the point 4, marked by a ‘t’ in the above diagram. The form of this is:

$$P_c = P_{ci} + G(P_{cd} - P_{ci}) \quad \text{Eq. 34.16}$$

with

$$G = \left(\frac{1}{S_{dep} - S_w + E} - \frac{1}{E} \right) / \left(\frac{1}{S_{dep} - S_{why} + E} - \frac{1}{E} \right) \quad \text{Eq. 34.17}$$

in which

S_{why} the water saturation at the hysteresis reversal point 4

S_{dep} is the departure saturation, the point at which $G = 0$.

In ECLIPSE, the default is for the imbibition scanning curve ‘b’ described by equation 34.14 to be **retraversed** when this reversal back to drainage occurs.

ECLIPSE 100

To obtain the **new** scanning curve ‘t’ described by equation 34.16 for secondary drainage, item 6 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

$$P_c = P_{ci} + F(P_{cd} - P_{ci}) \quad \text{Eq. 34.18}$$

with

$$F = \left(\frac{1}{S_{why} - S_w + E} - \frac{1}{E} \right) / \left(\frac{1}{S_{why} - S_{wcrd} + E} - \frac{1}{E} \right) \quad \text{Eq. 34.19}$$

in which

S_{why} is the water saturation at the hysteresis reversal point. In this case it is the minimum historical water saturation in the run.

S_{wcrd} is the critical water saturation on the drainage curve, which is the same for the scanning curves

When a second reversal back to imbibition occurs at a new departure point S_{dep} on the imbibition to drainage scanning curve, a second scanning curve is then followed during further imbibition from S_{dep} to S_{why} .

Gas capillary pressure

Similar expressions are used for gas capillary pressure hysteresis. In this case the gas saturation is the non-wetting saturation, and drainage to imbibition scanning curves return to the trapped critical gas saturation.

Consider the gas capillary pressure case shown below.

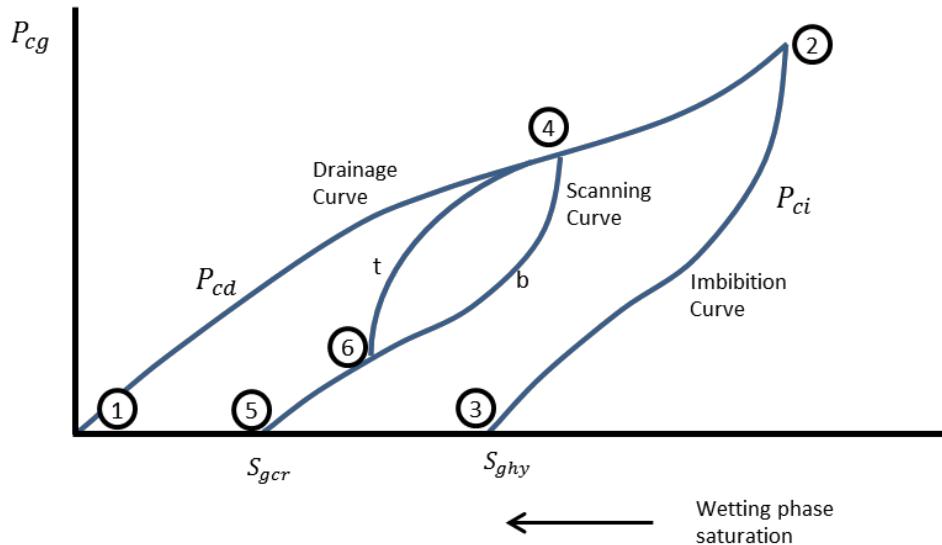


Figure 34.6. The gas capillary pressure case

Analogously to the case of P_{cw} ,

$$P_c = P_{cd} + F(P_{ci} - P_{cd}) \quad \text{Eq. 34.20}$$

with

$$F = \left(\frac{1}{S_{ghy} - S_g + E} - \frac{1}{E} \right) / \left(\frac{1}{S_{ghy} - S_{gcr} + E} - \frac{1}{E} \right) \quad \text{Eq. 34.21}$$

in which

E is a curvature parameter of the order of 0.1.

S_{ghy} is the gas saturation at the hysteresis reversal point 4

S_{wma} is the maximum water saturation attainable allowing for the trapped non-wetting phase saturation (that is, at $S_n = S_{ncrt}$)

when $S_w \rightarrow S_{why}$, $F \rightarrow 0$ and $P_{cw} \rightarrow P_{cwb}$

and when $S_w \rightarrow S_w^{\max}$, $F \rightarrow 1$ and $P_{cw} \rightarrow P_{cwi}$.

35

Hysteresis in WAG floods

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

This chapter describes the WAG (Water Alternating Gas) hysteresis option.

The hysteresis models described in "[Hysteresis](#)" 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 three-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 three-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 [\[Ref. 16\]](#) and [\[Ref. 66\]](#).

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 [RPTSCHE](#) 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.

In ECLIPSE 100, WAGHYSTR cannot be specified in a restart run unless it has been specified in the base run and any intermediate restart runs.

In ECLIPSE 300, WAGHYSTR can be specified in a restart run without having been specified in a previous base or restart run. However, this facility should only be used after carefully considering the efficacy of such an approach, for example, in situations where the restart containing the first WAGHYSTR keyword corresponds to the commencement of a WAG injection process, this may be an appropriate course of action. However, the results for simulations where WAG hysteresis has been modeled from the outset will be different from those where WAG hysteresis is commenced at some subsequent time thereafter because the hysteretic treatments prior to this time will be different. Moreover, the saturation history used by the WAG hysteresis model will only have been initiated upon the first occurrence of the WAGHYSTR keyword. This saturation history will also be lost if the WAGHYSTR keyword is not specified in all successive restarts following its first occurrence.

In ECLIPSE 300, from 2014.1 onwards, an alternative WAG hysteresis model is available. This is selected by including the keyword [WAGHALT](#) in the RUNSPEC Section. This alternative model seeks to replicate more closely the behavior of the ECLIPSE 100 WAG hysteresis model, both in terms of the change in saturation direction decisions and in the modeling of conventional hysteresis for those phases not subject to WAG hysteresis. The restart records written and read by this alternative WAG hysteresis model are not compatible with the default model; if the WAG hysteresis model is changed in a restart run, the previous WAG saturation history will not be preserved nor utilized.

It is important to note that the modeling of a WAG injection process does not of itself mandate that WAG hysteresis should also be modeled. If reservoir core measurements or simulations indicate significant hysteretic characteristics, it may be advisable firstly to establish which of the conventional or WAG hysteresis effects in the gas and or water phases are more significant before deciding which hysteresis model to select. Conventional hysteresis can be modeled in conjunction with WAG hysteresis and will be applied to the oil phase and also the gas and/or water phases (if these are not subject to WAG hysteresis) according to the options selected using the [EHYSTR](#) keyword.

Conventional and WAG hysteresis

There are some important differences between the specification of the relative permeability saturation functions for conventional and WAG hysteresis. For conventional hysteresis, the drainage and imbibition data are used to describe the two-phase hysteretic behavior of oil-water and gas-oil capillary pressure, water relative permeability, gas relative permeability and oil relative permeability to water and gas. For WAG plus conventional hysteresis, the bulk of this two-phase drainage and imbibition data is used in a similar way but with the following differences:

- The gas relative permeability data referred to by the drainage saturation table indices (SATNUM) are assumed to be representative of the two-phase drainage (increasing gas saturation) gas relative

permeability to oil in the presence of connate water only and are used as a basis for constructing all the associated scanning curves. With the exception of the critical gas saturation, the gas relative permeability data referred to by the imbibition saturation table indices (`IMBNUM`) are not used for modeling WAG hysteresis. However, they may still be required for conventional hysteresis according to the hysteresis model selected.

- The water relative permeability data referred to by the drainage saturation table indices (`SATNUM`) represent the two-phase water relative permeability to oil in the absence of gas. For conventional hysteresis, this column of data defines the two-phase drainage water relative permeability. If WAG hysteresis is applied to the water phase, this curve will be used for both drainage and imbibition processes in the absence of gas.
- The water relative permeability data referred to by the imbibition table indices (`IMBNUM`) represent the three-phase water relative permeability, that is, the water relative permeability to oil in the presence of gas, measured or modeled at the maximum gas saturation encountered during the gas injection cycle of the WAG process. For conventional hysteresis, this column of data defines the two-phase water imbibition relative permeability.

In general, the more significant difference between conventional and WAG hysteresis is the impact of successive water and gas injection cycles upon the trapped gas and hence upon the gas phase rather than the water phase relative permeability. When modeling WAG hysteresis in the water phase, careful consideration should be given to the suitability of the conventional two-phase drainage and imbibition relative permeability data for reuse with WAG hysteresis. This is complicated by the different characteristics of these saturation functions for these different types of hysteresis. For conventional hysteresis and a water-wet model, the imbibition water relative permeability curve typically exhibits a higher relative permeability value than the drainage curve at a given water saturation. However, for WAG hysteresis, the three-phase water relative permeability curve typically exhibits a lower relative permeability than the two-phase curve at a given water saturation.

Given that the selection of the WAG hysteresis option for the water phase implies presumably that the three-phase effects associated with the WAG injection process are more significant than the effects associated with conventional hysteresis, the two-phase drainage and imbibition water relative permeability data need to be adapted for use with WAG hysteresis. For conversion from conventional drainage to WAG hysteresis two-phase water relative permeability data, the possible approaches include:

- Using the existing drainage data unchanged. If the drainage and imbibition water relative permeability data do not exhibit significant hysteresis, this may be a convenient approach.
- Using some form of averaged drainage and imbibition data. If the drainage and imbibition water relative permeability data exhibit significant hysteresis, this be a suitable approach given that this data will be used for both the drainage and imbibition processes in the absence of the gas phase.

For conversion from conventional imbibition to WAG hysteresis three-phase water relative permeability data, there is no simple approach based upon substitution or adaptation of existing data. The underlying premise of WAG hysteresis in the water phase is that the water relative permeability is lower following a gas flood than when initially only water and oil is present. Hence the three-phase WAG hysteresis data should ideally be obtained from measurements or modeling of water relative permeability in the presence of the gas phase.

One seemingly possible solution to the difference between conventional and WAG hysteresis for water relative permeability would appear to be provided by interchanging the water relative permeability columns of data in the pairs of tables associated with the drainage and imbibition saturation table indices. However, this is a difficult process to accomplish because only single columns of data in pairs of otherwise unchanged tables are involved and the table saturation nodes may be different. Moreover, the suitability of such an approach is questionable given that if the drainage and imbibition two-phase data columns are

interchanged neither will be necessarily representative of the WAG hysteresis two-phase or three-phase water relative permeability preceding or following a gas injection cycle respectively.

It should be noted that if WAG hysteresis is applied only to the gas phase, the existing conventional hysteresis relative permeability saturation functions can be used unmodified.

When selecting WAG hysteresis, consideration should also be given to the form of conventional hysteresis being applied to the phases not subject to WAG hysteresis. For example, in a water-wet system, if the Killough hysteresis model is applied to both the wetting and non-wetting phases and if oil is wetting to gas, the hysteresis scanning curves for the relative permeability of oil to gas (wetting phase) will be driven by the properties of the two-phase drainage and imbibition gas relative permeability (non-wetting phase). Hence it will still be necessary to specify both suitable drainage and imbibition gas relative permeability data.

For the modeling of capillary pressure hysteresis in WAG injection processes, there is limited information in the published literature pertaining to how the characteristics differ if at all from those of conventional hysteresis. In ECLIPSE capillary pressure hysteresis, if selected, is modeled using the Killough hysteresis model irrespective of whether conventional or WAG hysteresis is selected.

WAG hysteresis saturation regions

For various reservoir modeling strategies, simulations may involve comparisons between conventional and WAG hysteresis. When changing from conventional to WAG hysteresis or vice versa, the use of the drainage (`SATNUM`) and imbibition (`IMBNUM`) saturation tables for both conventional and WAG hysteresis requires that the water relative permeability data associated with the imbibition saturation tables and possibly also the drainage saturation tables are changed. Alternatively separate sets of tables can be maintained such that only the drainage and saturation table indices are changed. In order to facilitate the process of changing between conventional and WAG hysteresis, two additional saturation region numbers have been introduced for representing separately the WAG hysteresis saturation functions. The keywords `WH2NUM` and `WH3NUM` can be used to specify the table numbers for the WAG hysteresis two-phase and three-phase water relative permeability functions respectively. This provides a mechanism for the coexistence of both conventional two-phase and WAG hysteresis saturation functions.

The `WH2NUM` and `WH3NUM` keywords are optional; if these are not specified, the region numbers associated with these keywords will default to those defined for the `SATNUM` and `IMBNUM` keywords respectively. For the saturation tables associated with the `WH2NUM` and `WH3NUM` keywords, only the water relative permeability functions are utilized and the remaining data may be defaulted. In cases where the conventional two-phase water relative permeability data are considered suitable for use as the WAG-hysteresis two-phase relative permeability functions, the `WH2NUM` keyword can be omitted.

Where the `WH2NUM` or `WH3NUM` keywords have been specified, the minimum, critical and maximum water saturations should be equal for the corresponding `SATNUM` and `WH2NUM` table indices and for the corresponding `IMBNUM` and `WH3NUM` table indices. If one or more of these conditions are not satisfied, warning messages will be issued. Irrespective of whether or not the `WH2NUM` or `WH3NUM` keywords have been specified, warning messages will also be issued for grid blocks where the WAG hysteresis designated three-phase water relative permeability function is greater than the WAG hysteresis designated two-phase water relative permeability function at one or more table saturation nodes.

These additional optional water relative permeability functions can be considered essentially as additional columns of the associated `SATNUM` and `IMBNUM` saturation tables. If the end-point scaling option has been specified, the conventional and WAG hysteresis relative permeability will both be subject to the same horizontal (saturation) scaling using the keywords `SWL`, `SWCR` etc. for the drainage and two-phase curves and using the keywords `ISWL`, `ISWCR` etc. for the imbibition and three-phase curves. However, the vertical

(relative permeability) scaling for the two-phase and three-phase water relative permeability data is applied separately via the additional keywords `KRWH`, `KRWRH` and `IKRWH`, `IKRWHR` respectively.

If the `WH2NUM` or `WH3NUM` keywords have been specified and the saturation table numbers differ from those specified via the corresponding `SATNUM` and `IMBNUM` keywords respectively, the two-phase and three-phase water relative permeability data will be used instead of the drainage and imbibition relative permeability data for all water relative permeability calculations in those saturation regions for which WAG hysteresis has been applied to the water phase. This is specified via item 5 of the `WAGHYSTR` keyword on a per saturation region basis.

For grid blocks for which the saturation table numbers specified by the `SATNUM` and `WH2NUM` keywords are the same, if end-point scaling has been specified, the values of `KRWH` and `KRWRH` will be ignored if specified. If these saturation table numbers are not the same, the values of `KRWH` and `KRWRH` will apply to the water relative permeability table values specified via `WH2NUM`. Similarly, for grid blocks for which the saturation table numbers specified by the `IMBNUM` and `WH3NUM` keywords are the same, if end-point scaling has been specified, the values of `IKRWH` and `IKRWHR` will be ignored if specified. If these saturation table numbers are not the same, the values of `IKRWH` and `IKRWHR` will apply to the water relative permeability table values specified using `WH3NUM`.

A simple example of the use of the `WH2NUM` and `WH3NUM` keywords is shown below for the family II set of saturation functions. The gas and oil saturation functions associated with the `WH2NUM` and `WH3NUM` keywords are not used and can be defaulted in their entirety. For the water saturation functions only the saturation and water relative permeability columns are used and the remaining data may be defaulted subject to the specification of the minimum number of entries required for valid table entry. A similar approach may also be employed for the Family I set of saturation tables.

```

TITLE
Example using WH2NUM and WH3NUM with Family II Saturation Functions

DIMENS
10 1 1 /
...
SATOPTS
HYSTER
/
TABDIM3
4 ...
/
PROPS
...
EHYSTR
1* 2
/
...
WAGHYSTR
4* YES /
/
/
SGFN
... / -- Table 1 drainage (conventional hysteresis)
... / -- Table 2 imbibition (conventional hysteresis)
/ -- Table 3 unused
/ -- Table 4 unused
SOF3
... / -- Table 1 drainage (conventional hysteresis)
... / -- Table 2 imbibition (conventional hysteresis)
/ -- Table 3 unused
/ -- Table 4 unused
SWFN
... / -- Table 1 drainage (conventional hysteresis)
... / -- Table 2 imbibition (conventional hysteresis)
... / -- Table 3 two-phase krw only used for WAG hysteresis
... / -- Table 4 three-phase krw only used for WAG hysteresis

```

```

REGIONS
...
SATNUM
10*1 /
IMBNUM
10*2 /
WH2NUM
10*3 /
WH3NUM
10*4 /
...

```

A more complex example of the use of [WH2NUM](#) and [WH3NUM](#) is shown below for a model consisting of five layers each containing three grid blocks. End-point scaling has been specified and WAG hysteresis has been applied to the water phase in SATNUM regions (and hence layers for this model) 1, 2, 3 and 4 but not 5. The [WAGHYSTR](#) keyword single row table numbers 6 to 20 are ignored because there are no SATNUM regions associated with these table numbers. The usage of WH2NUM and WH3NUM for each layer and their associated vertical scaling end-points is summarized below:

1. WH2NUM is equal to SATNUM and WH3NUM is equal to IMBNUM for this layer and hence the two-phase and three-phase water relative permeability data used for WAG hysteresis will be taken from SATNUM and IMBNUM respectively. If KRW, KRWR, IKRW or IKRWR have been specified, they will be ignored when applying water relative permeability vertical scaling because the arrays KRW, KRWR, IKRW or IKRWR will be used if specified.
2. WH2NUM is not equal to SATNUM and WH3NUM is not equal to IMBNUM for this layer and hence the two-phase and three-phase water relative permeability data used for WAG hysteresis will be taken from WH2NUM and WH3NUM respectively. If KRW, KRWR, IKRW or IKRWR have been specified, their values will be used when applying water relative permeability vertical scaling and the values of KRW, KRWR, IKRW and IKRWR will be ignored if specified.
3. WH2NUM is equal to SATNUM but WH3NUM is not equal to IMBNUM for this layer and hence the two-phase and three-phase water relative permeability data used for WAG hysteresis will be taken from SATNUM and WH3NUM respectively. If specified, KRW and KRWR will be used for the two-phase water relative permeability vertical scaling but IKRW and IKRWR will be used for the three-phase water relative permeability vertical scaling.
4. WH2NUM is not equal to SATNUM but WH3NUM is equal to IMBNUM for this layer and hence the two-phase and three-phase water relative permeability data used for WAG hysteresis will be taken from WH2NUM and IMBNUM respectively. If specified, KRW and KRWR will be used for the two-phase water relative permeability vertical scaling but IKRW and IKRWR will be used for the three-phase water relative permeability vertical scaling.
5. WH2NUM is not equal to SATNUM and WH3NUM is not equal to IMBNUM for this layer. However, WAG hysteresis has not been applied to the water phase for this layer and hence this layer will be subject to conventional hysteresis according to the items specified via the [EHYSTR](#) keyword and WH2NUM, WH3NUM, KRW, IKRWR and IKRW will be ignored.

```

TITLE
Example using WH2NUM and WH3NUM with EPSDEBUG

DIMENS
3 1 5 /
ENDSCALE
/
SATOPTS
HYSTER
/
TABDIMS
20...
/

```

```

PROPS
...
EHYSTR
1* 2
/
EPSDEBUG
1 3 1 1 1 4
/
WAGHYSTR
4* YES /
4* YES /
4* YES /
4* YES /
4* NO /
/ Table 6
...
/ Table 20
...
REGIONS
...
SATNUM
3*1 3*2 3*3 3*4 3*5 /
IMBNUM
3*6 3*7 3*8 3*9 3*10 /
WH2NUM
3*1 3*12 3*3 3*14 3*15 /
WH3NUM
3*6 3*17 3*18 3*9 3*20 /
...

```

If end-point scaling has been specified, the reporting of the scaled tables via the `EPSDEBUG` or `EPSDBGS` keyword will reflect the use of `WH2NUM` and `WH3NUM` for the grid blocks selected for reporting in the manner described above. Specifically, if WAG hysteresis has been applied to the water phase:

6. If `WH2NUM` is not equal to `SATNUM`, the drainage report will indicate not only the drainage table number but also the two-phase table number from which the water relative permeability data will be taken in place of the drainage table water relative permeability data. In addition, `KRWH` and `KRWRH` will be used instead of `KRW` and `KRWR`.
7. If `WH3NUM` is not equal to `IMBNUM`, the imbibition report will indicate not only the imbibition table number but also the three-phase table number from which the water relative permeability data will be taken in place of the imbibition table water relative permeability. In addition, `IKRWH` and `IKRWRH` will be used instead of `IKRW` and `IKRWR`.

Non-wetting phase model (Gas)

Two-phase model

The gas phase model is based on the theory developed by Land and Carlson. The two-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 the following figure).

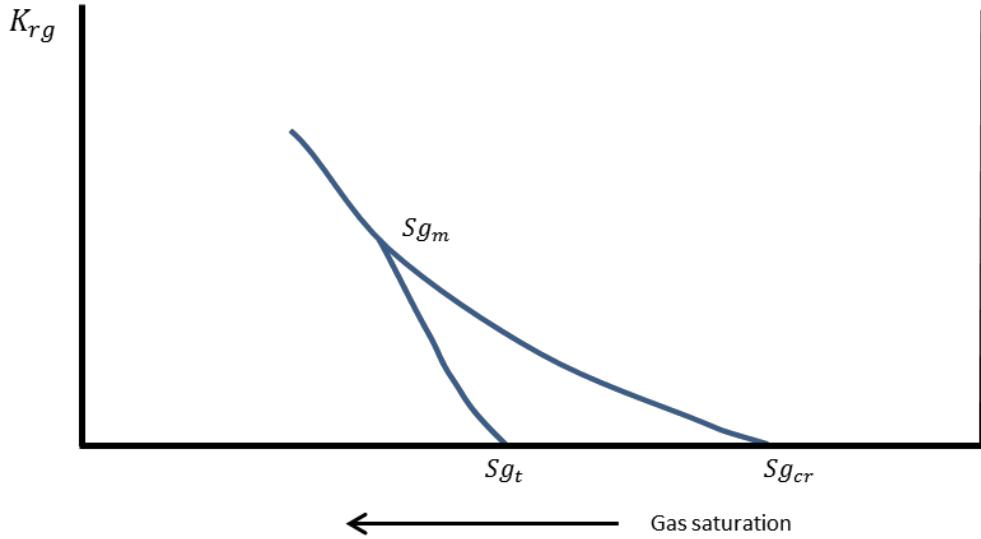


Figure 35.1. A typical drainage process followed by an imbibition process

Consider a drainage process reaching a maximum gas saturation Sg_m followed by an imbibition process leading to a trapped gas saturation Sg_{trap} . The trapped gas saturation Sg_{trap} is given by

$$Sg_{trap} = Sg_{cr} + \frac{(Sg_m - Sg_{cr})}{(1 + C \cdot (Sg_m - Sg_{cr}))} \quad \text{Eq. 35.1}$$

where

C is Land's parameter, input with the `WAGHYSTR` keyword, or specified by the transition parameter as discussed later.

Note: As for the generic hysteresis calculation, the Killough method has been modified slightly to overcome non-physical behavior when Sg_m is small. As Sg_m tends to Sg_{cr} in Equation 35.1, Sg_{trap} tends to Sg_{cr} , 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. Equation 35.1 has been modified as follows:

$$Sg_{trap} = Sg_{cr} + \frac{Sg_m - Sg_{cr}}{A + C(Sg_m - Sg_{cr})}$$

where $A = 1 + a \cdot (Sg_{max} - Sg_m)$, where Sg_{max} is the maximum attainable gas saturation. The parameter a is input through the fourth data item in the `EHYSTR` keyword and defaults to 0.1.

The gas relative permeability on the scanning curve is given by:

$$K_{rg}(Sg) = K_{rg}^{drain}(Sg_f) \quad \text{Eq. 35.2}$$

where:

$$Sg_f = Sg_{cr} + \frac{1}{2} \left\{ \left(Sg - Sg_{trap} \right) + \sqrt{\left(Sg - Sg_{trap} \right)^2 + \frac{4}{C} \left(Sg - Sg_{trap} \right)} \right\} \quad \text{Eq. 35.3}$$

This transformation typically has an infinite gradient dSg_f / dSg when Sg is close to Sg_{trap} . As the input drainage relative permeability functions are represented as piece-wise linear functions, the final relative permeability function also has an infinite gradient. As this causes convergence problems the transformation Sg to Sg_f is modified to a linear function near to the trapped gas saturation. The proportion of the transformation that departs from the pure Carlson equation can be controlled using the sixth data item in the [WAGHYSTR](#) keyword.

Note: In order to make a section of the imbibition curve linear to prevent an infinite gradient in the final relative permeability curve, the Carlson mapping algorithm is applied and the Land's parameter C in

equations 35.1 and 35.3 can be replaced by the transition parameter C^{trans} defined as

$$C^{trans} = \frac{1}{S_{ncri} - S_{ncrd}} - \frac{1}{S_{n\ max} - S_{ncrd}}$$

If the gas saturation increases following an imbibition process, the gas relative permeability follows the imbibition curve provided that the model remains in two-phase mode, that is if the displacing phase is oil. The criterion for the model to remain in two-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 via 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 35.3 can be interpreted as the free gas saturation for a given saturation state. Hence the trapped gas saturation is given by

$$Sg_{td} = Sg - Sg_f \quad \text{Eq. 35.4}$$

Three-phase model

The gas relative permeability follows a secondary drainage curve when the gas saturation increases and where the water saturation exceeds the connate value at the turnaround point. A typical secondary drainage curve is illustrated in the following figure.

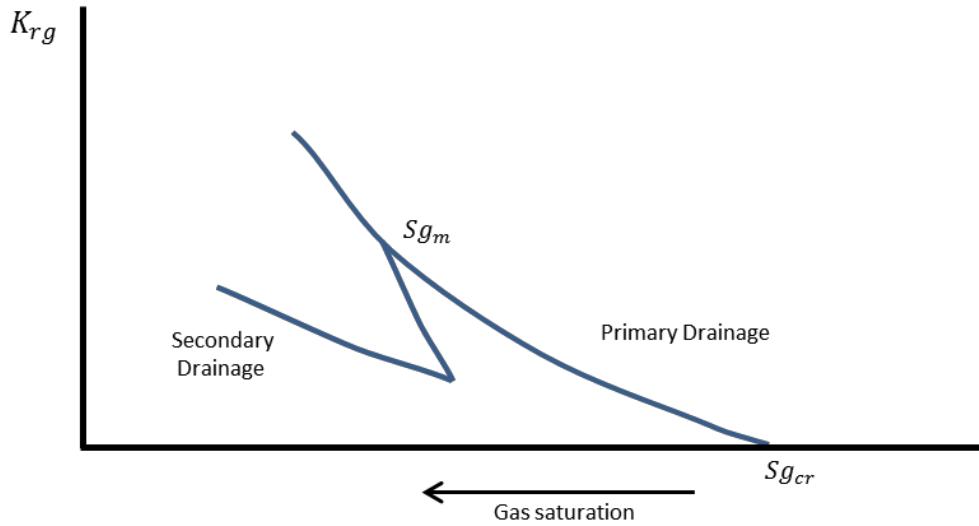


Figure 35.2. Schematic diagram showing a typical secondary drainage curve

These secondary drainage curves are calculated using the following equation

$$K_{rg}^{drain} = [K_{rg}^{input} - K_{rg}^{input}(Sg^{start})] \cdot \left[\frac{Sw_{co}}{Sw^{start}} \right]^{\alpha} + [K_{rg}^{imb}(Sg^{start})] \quad \text{Eq. 35.5}$$

where:

K_{rg}^{drain} is the calculated secondary drainage relative permeability as a function of Sg

K_{rg}^{input} is the input relative permeability at Sg

$K_{rg}^{input}(Sg^{start})$ is the input relative permeability at the gas saturation at the start of the secondary drainage curve

Sw_{co} is the connate water saturation

Sw^{start} is the water saturation at the start of the secondary drainage curve.

$K_{rg}^{imb}(Sg^{start})$ is the relative permeability at the start of the secondary drainage process (that is the K_r at the end of the imbibition curve)

α is the reduction exponent input in the [WAGHYSTR](#) keyword.

Subsequent imbibition curves are treated in an analogous way to an imbibition process from the primary drainage curve. The saturation is transformed using the Carlson equation, [35.3](#) and looked up on the secondary drainage curve. The trapped gas saturation is taken as the maximum historical trapped gas saturation, not the trapped gas saturation for this cycle.

If the transformed saturation Sg_f falls below the saturation at the start of the secondary drainage process, the secondary drainage curve cannot be used. In this case the transformation is modified and the primary drainage curve is used:

$$K_{rg}(Sg) = K_{rg}^{drain}(Sg_f^m) \quad \text{Eq. 35.6}$$

where Sg_f^m is given by:

$$Sg_f^m = Sg_{cr} \text{ where } Sg_f \leq Sg_{trap} \quad \text{Eq. 35.7}$$

$$Sg_f^m = \left\{ (Sg_f - Sg_{cr}) \cdot \frac{(Sg^{up} - Sg_{gcr})}{(Sg^{start} - Sg_{trf})} \right\} + \left\{ Sg_{cr} - Sg_{trf} \cdot \frac{(Sg^{up} - Sg_{gcr})}{(Sg^{start} - Sg_{trf})} \right\} \quad \text{Eq. 35.8}$$

$$Sg_f > Sg_{trap}$$

where:

$$Sg_{trf} = (Sg_{trap} - Sg_{gcr}) \quad \text{Eq. 35.9}$$

Sg^{start} is the gas saturation at the start of the secondary drainage curve

Sg^{up} is the gas saturation on the drainage curve corresponding to the imbibition gas relative permeability at the start of the cycle.

ECLIPSE 100

In some cases it has been observed that the gas relative permeability falls to near zero when the primary drainage curve is used to construct the secondary or subsequent drainage cycle in the manner described above. This can be prevented by using a modified version of equation 35.8 in which $(Sg^{start} - Sg_{trf})$ is replaced by $(Sg^{start} - Sg_{trap})$ in the denominators:

$$Sg_f^m = \left\{ (Sg_f - Sg_{cr}) \cdot \frac{(Sg^{up} - Sg_{gcr})}{(Sg^{start} - Sg_{trap})} \right\} + \left\{ Sg_{cr} - Sg_{trf} \cdot \frac{(Sg^{up} - Sg_{gcr})}{(Sg^{start} - Sg_{trap})} \right\} \quad \text{Eq. 35.10}$$

This modified model may be selected by setting item 197 of the OPTIONS keyword to 1.

In cases where, for one or more successive cycles of gas injection, the maximum gas saturation attained for the cycle does not exceed the maximum historical gas saturation attained on a previous cycle, the trapped gas saturation Sg_{trap} will not increase but remain constant at its value reached on the previous cycle experiencing the maximum historical gas saturation. Similarly the dynamic trapped gas saturation Sg_{td} will increase to the value of the trapped gas saturation and remain at this value until the gas saturation next exceeds the maximum historical gas saturation.

Note: The WAG (Water Alternating Gas) hysteresis option can be applied to calculate the gas trapped saturation during the CO₂ storage process. Although the WAG hysteresis option was initially developed for a three-phase system, it is also applicable in the GASWAT and CO₂STORE options, which describe a two-phase system. In the GASWAT and CO₂STORE options, the three-phase constitutive equations for WAG hysteresis are applied to a two-phase system.

Modification to the residual oil

If the STONE 1 three-phase oil relative permeability model is employed, (see "Stone's first model (Modified)") it is optionally 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.

$$SOM_{mod} = SOM - (a \cdot Sg_{td}) \quad \text{Eq. 35.11}$$

where

SOM is the minimum residual oil saturation

Sg_{td} is the dynamic trapped gas saturation given by equation 35.4

a is input as item 8 of the `WAGHYSTR` keyword.

Wetting phase model (Water)

The wetting phase model is based on the observation that the water relative permeability curve measured following a gas flood shows significantly less mobility. An example of the two relative permeability curves is shown in the following figure.

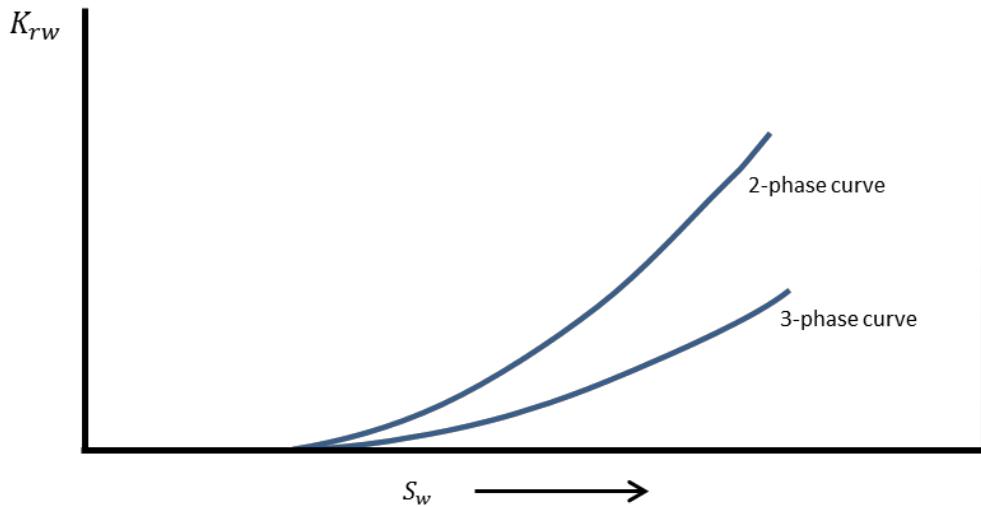


Figure 35.3. Two-phase and three-phase relative permeability curves

The two-phase curves are taken to be the drainage curves (`SATNUM`) and the three-phase curves to be the imbibition curves (`IMBNUM`). Note that these curves are not strictly drainage and imbibition but apply to the two-phase and three-phase cases respectively.

For an imbibition process, with S_w increasing, the relative permeability function used is interpolated between the two-phase and three-phase curves using the following equation

$$K_{rw}^{imb} = K_{rw2} \cdot \left(1 - \frac{Sg_{start}}{Sg_{max}}\right) + K_{rw3} \cdot \left(\frac{Sg_{start}}{Sg_{max}}\right) \quad \text{Eq. 35.12}$$

where:

K_{rw}^{imb} is the calculated imbibition relative permeability

K_{rw2} is the two-phase relative permeability at S_w

K_{rw3} is the three-phase relative permeability at S_w

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 the following figure.

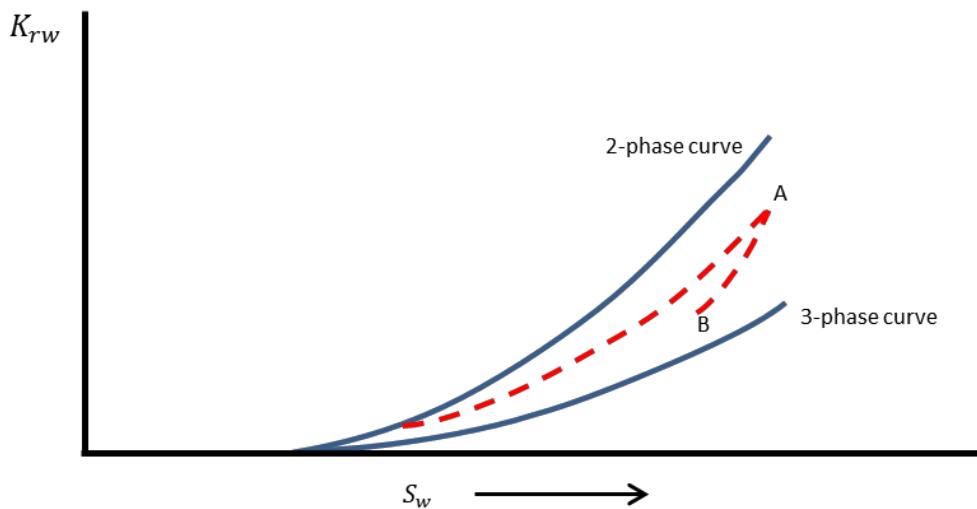


Figure 35.4. Subsequent drainage relative permeability

An initial imbibition process may reach point A. The subsequent drainage process in this case moves to point B, on a curve interpolated between the original imbibition curve and the input three-phase curve.

The drainage curve moves either side of the imbibition curve depending on the prevailing gas saturation relative to the gas saturation at the start of the drainage process Sg^{sdrain} .

If $Sg > Sg^{sdrain}$ then

$$K_{rw}^{drain} = K_{rw}^{imb} \cdot \left(1 - \frac{Sg - Sg^{sdrain}}{Sg_{max} - Sg^{sdrain}}\right) + K_{rw3} \cdot \left(\frac{Sg - Sg^{sdrain}}{Sg_{max} - Sg^{sdrain}}\right) \quad \text{Eq. 35.13}$$

If $Sg < Sg^{sdrain}$ then

$$K_{rw}^{drain} = K_{rw}^{imb} \cdot \left(1 - \frac{Sg^{sdrain} - Sg}{Sg^{sdrain}}\right) + K_{rw2} \cdot \left(\frac{Sg^{sdrain} - Sg}{Sg^{sdrain}}\right) \quad \text{Eq. 35.14}$$

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 S_w

$K_{rw\ 2}$ is the input two-phase curve at S_w

$K_{rw\ 3}$ is the input three-phase curve at S_w

Subsequent secondary drainage processes follow an interpolated curve in much the same manner as the primary imbibition, but using a modified Sg^{start} arranged such that the curve runs through point B.

36

Independent reservoir regions

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, and the reservoirs are not connected by aquifers), the linear equations solved by ECLIPSE for the two reservoirs are independent. The solution of the two sets of linear equations in general require different numbers of linear iterations.

The Independent reservoir option allows ECLIPSE to utilize this potential efficiency gain.

Note: This option is not compatible with the parallel option.

Using the option

1. Set NRFREG (the maximum number of independent reservoir regions) in item 3 of the RUNSPEC section keyword [REGDIMS](#).
2. Use the [ISOLNUM](#) keyword in the GRID section to establish the separate regions.
There must be at least one active cell allocated to each reservoir region between 1 and NRFREG . Any cells given a reservoir region number of 0 or > NRFREG are treated as inactive.
The ISOLNUM data can be written out to the debug file for an existing model by using the [RPTISOL](#) keyword in the GRID section. The ISOLNUM array can be subsequently added to the dataset.
3. The ISOLNUM array can be output to the PRINT file using the mnemonic ISOLNUM in the [RPTGRID](#) keyword.
4. If the mnemonic NEWTON in the [RPTSCHED](#) keyword is activated, a separate summary line for each reservoir is printed at each Newton iteration.

Merging two or more corner point grids

Corner point grids are usually supplied for the entire grid from a pre-processing program such as PETREL. However, if a number of separate models have been run independently and are later required to be run as a

single model, there are a number of facilities in ECLIPSE 100 to enable the corner point grids to be input separately.

If the separate grids are to be placed side by side in the X or Y directions, the **COORD** and **ZCORN** keywords can be used in conjunction with a **BOX** keyword to input the data for a portion of the field. Note that the grids should be separated by a row of inactive cells to prevent the coordinate lines overlapping. Also any inactive parts of the merged model should have dummy COORD and ZCORN values input; the cells can then be made inactive using the **ACTNUM** keyword. See "[Example 1](#)".

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](#)".

Examples

Example 1

To merge a 2*4*1 grid with a 3*2*1 grid side by side in the X direction.

```
RUNSPEC section:
DIMENS
 6 4 1 /
GRID section:
BOX
 1 2 1 4 1 1 /
COORD.... for the first grid.
ZCORN....
BOX
 4 6 1 2 1 1 /
COORD.... for the second grid.
ZCORN....
EQUALS
 'ACTNUM' 0 3 3 1 4 1 1 /   inactivate the row between the grids.
 'ACTNUM' 0 4 6 3 4 1 1 /   inactivate the redundant region of
 /                           the grid.
```

Example 2

To merge a 2*4*1 grid with a 3*2*1 grid, ‘stacked’ in the Z direction.

```
RUNSPEC section:
DIMENS
 3 4 3 /
NUMRES
 2 /
GRID section:
COORDSYS
 1 2 /
 3 3 /
RESVNUM
 1 /
BOX
 1 2 1 4 1 1 /
COORD.... for the first grid.
ZCORN....
RESVNUM
```

```
2 /
BOX
1 3   1 2   3 3   /
COORD....      for the second grid.
ZCORN.....
EQUALS
'ACTNUM' 0  1 3  1 4  2 2 / inactivate the layer between the grids.
'ACTNUM' 0  3 3  1 4  1 1 / inactivate the redundant region of
'ACTNUM' 0  1 3  3 4  3 3 / the grid.
/
```

37

Initializing the study

The initial reservoir conditions can be defined in one of four ways:

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

- The first method assumes the reservoir is in hydrostatic equilibrium. See [Basic hydrostatic equilibration](#).

The other three methods are all forms of non-equilibrium initialization:

- The initial conditions can be read from a RESTART file generated by an earlier run. See ["Restarts"](#).
- The initial conditions can be set explicitly. See ["Enumeration"](#).
- The initial conditions can be calculated by a combination of equilibration and enumeration. See ["Mixed hydrostatic equilibration"](#).

Thermal option only

In compositional equation of state (EoS) models, determining whether a single phase is liquid or vapor can be problematic. This issue is discussed in ["Single phase states in EoS compositional models"](#).

Further information on initializing:

- ["Using the ECLIPSE tracer tracking facility"](#) for details on initializing tracers in ECLIPSE.
- ["Using the polymer flood model"](#) for details on initializing the Polymer Flood Model in ECLIPSE.
- ["Injecting solvent into the reservoir"](#) for details on initializing the Solvent Model in ECLIPSE.

Equilibration

Basic hydrostatic equilibration

For a reservoir containing a single phase fluid, the vertical pressure gradient can be determined from the fluid density. If the reservoir pressure is given at a datum depth, then the fluid density can be calculated at that depth and the pressure in the reservoir can be determined by marching up (and down) the reservoir in small steps, recalculating the density at each step.

The algorithm is more complicated when more than one phase is present. Contact depths (where two phases meet) must be provided, together with the capillary pressure at these depths. In addition sufficient information must be provided to allow the densities of all phases to be calculated as described in ["Density calculations"](#). Starting at the datum depth, the hydrostatic pressure for the “datum” phase can be calculated by marching up and down the reservoir. Pressures in the other phases can then be determined at the contact depths, and then the hydrostatic pressures can be determined throughout the reservoir by marching up and

down again. Once the phase pressures are known, phase saturations can be determined at each depth so that the hydrostatic pressure variation is balanced by the capillary pressure between the phases. Further details are given in "[Calculating the initial conditions](#)".

When two or more phases are present, the algorithm assumes that water is heavier than oil which is heavier than gas. If this condition does not hold, then the algorithm will not work properly and the results will not be as expected. Failure of the condition can be a sign that the fluid has not been properly characterized, or, if heavy oil is being modeled, that the oil may in fact be heavier than water. For more information, see "[Heavy oil](#)".

The hydrostatic equilibration provides tables of pressure and saturation as a function of depth. The equilibration can be performed on a finer grid than that used for the simulation, in which case the resulting saturations are 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; see "[The quiescence option](#)".

The aim of equilibration is to set up a static initial configuration in which the phases present are in equilibrium and in which the inter-block flows are zero. Successful equilibration can be verified by running the simulation without wells to check that no flows occur.

Equilibration regions

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 (NTEQUL) is specified by using the [EQLDIMS](#) keyword in the RUNSPEC section. Cells are associated with these regions using the [EQLNUM](#) keyword. Cells in a given equilibration region should have the same pressure table for their PVT properties, but they can use different saturation tables.

Cells in different equilibration regions should be non-communicating, otherwise they may not be in equilibrium. See "[The threshold pressure option](#)".

Contact and datum depths

For each equilibration region, the fluid contact depths, the capillary pressures at these depths and the pressure at a given datum depth are specified using the [EQUIL](#) keyword.

The datum depth for the pressure can usually lie anywhere in the equilibration region. However, in black oil runs, if dissolved gas or vaporized oil data are required but defaulted, then the datum depth must correspond to the gas-oil 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.

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.

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.

Density calculations

Data describing the fluid as a function of depth need to be provided to allow the equilibration algorithm to calculate the fluid density and hence the hydrostatic pressure gradient. The data required depends on the fluid model and is described separately in the sections "[Black oil composition with respect to depth](#)" and "[Initial composition with respect to depth](#)". Unless defaulted, tables must be entered for each [EQLNUM](#) region. The maximum number of depth values in any such table must be set using the keyword [EQLDIMS](#) in the RUNSPEC section.

In gas-water problems, there is no need to enter any data concerning the nonexistent oil phase.

Black oil composition with respect to depth

Dissolved gas

In black oil models containing dissolved gas, the dissolved gas concentration R_s or the bubble point pressure P_b may be entered as a function of depth using the keywords [RSVD](#) and [PBVD](#) respectively. A table must be entered for each [EQLNUM](#) region unless the data is defaulted altogether. At any position in the reservoir, the R_s value must be less than or equal to the saturated R_s value at the local pressure.

In runs with dissolved gas, if both the RSVD and PBVD keywords are omitted, the dissolved gas concentration in undersaturated oil is set everywhere to be equal to the saturated R_s value at the gas-oil contact. In this case, the datum depth for the equilibration must be set at the at the gas-oil contact, so that the saturated R_s value can be calculated there.

Vaporized oil

In gas condensate problems, the vaporized oil concentration R_v or the dew point pressure P_d may be entered as a function of depth using the keywords [RVVD](#) and [PDVD](#) respectively. A table must be entered for each [EQLNUM](#) region unless the data is defaulted altogether. At any position in the reservoir, the R_v value must be less than or equal to the saturated R_v value at the local pressure.

In runs with vaporized oil, if both the RVVD and PDVD keywords are omitted, the vaporized oil concentration in undersaturated gas is set everywhere to be equal to the saturated R_v value at the gas-oil contact. In this case, the datum depth for the equilibration must be set at the gas-oil contact, so that the saturated R_v value can be calculated there.

Initial composition with respect to depth

ECLIPSE 300

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 for setting up the initial compositions, which usually vary with depth:

1. For single hydrocarbon phase initial states, such as condensates above the dew point, or oils above the bubble point, or for "supercritical" reservoirs in which a smooth transition exists between the oil and gas properties and the saturation pressure is 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 then sets the phase

identification method to produce a phase change at this depth. For a condensate field the gas-oil 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.

After the calculations described in cases 2 and 3 above, the oil composition is known below the gas-oil contact, and the vapor composition is known above the gas-oil contact. Oil may be present above the contact, and vapor below, depending on the capillary pressure. Therefore it is necessary to determine the composition of these phases:

- If there is any vapor below the contact then it will be in equilibrium with the oil below the contact. Therefore a bubble point calculation is performed on the oil below the contact. If the reservoir pressure is below the oil bubble point then the vapor composition can be set equal to the composition of the vapor in equilibrium with the oil at the reservoir pressure. If however the reservoir pressure is above the oil bubble point then the vapor composition will be set equal to the bubble point vapor composition. In this case the oil and vapor will not be in equilibrium, and as a result - if vapor is in fact present at these depths - the algorithm will fail to match the required compositions and saturations.
- If there is any oil above the contact then it will be in equilibrium with the vapor above the contact. Therefore a dew point calculation is performed on the vapor above the contact. If the reservoir pressure is above the vapor dew point then the oil composition can be set equal to the composition of the oil in equilibrium with the vapor at the reservoir pressure. If however the reservoir pressure is below the vapor dew point then the oil composition will be set equal to the dew point oil composition. In this case the oil and vapor will not be in equilibrium, and as a result - if oil is in fact present at these depths - the algorithm will fail to match the required compositions and saturations.

The calculated saturation pressures, and oil and vapor compositions and phase saturations can be output as a function of depth by specifying the SOLVD mnemonic with the RPTSOL keyword. The compositions and saturations will match the actual values in the reservoir if the oil and vapor are in equilibrium. However they will fail to match when the oil and vapor are not in equilibrium which can occur if the reservoir pressure is above the oil bubble point or below the vapor dew point. In these cases the specified composition (COMPVD or ZMFVD) should be adjusted so that the fluid is two phase at the reservoir pressures.

The three cases above correspond to the initialization type specified by the tenth argument of the EQUIP keyword. In addition, the eleventh 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.

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 (P_w , P_o , P_g), and composition (or R_s and R_v for black oil) against depth.

The second stage interpolates this table and uses the saturation functions to obtain the fluid conditions in each grid block in the region.

In the first stage, depth points for the internal tables are spaced equally throughout the region. The number of depth points 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.

ECLIPSE 300

Depth table data can be reported using the SOLVD mnemonic in the [RPTSQL](#) keyword.

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 phase pressures (P_w , P_o , P_g) and composition (or R_s and R_v) in each grid block. Different interpolation methods can be used to improve the accuracy, see "[Accurate fluids in place calculation](#)".

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

$$P_{cow}(S_w) = P_o - P_w \quad \text{Eq. 37.1}$$

If $P_o - P_w$ exceeds the highest capillary pressure value in the SWFN/SWOF table (corresponding to the lowest saturation value $S_{w\min}$), the water saturation is set equal to $S_{w\min}$.

If $P_o - P_w$ is less than the lowest capillary pressure in the table (corresponding to the highest saturation value $S_{w\max}$), the water saturation is set equal to $S_{w\max}$ 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

$$P_{cog}(S_g) = P_g - P_o \quad \text{Eq. 37.2}$$

If $P_g - P_o$ is less than the lowest capillary pressure value in the table (corresponding to the lowest gas saturation value $S_{g\min}$), the gas saturation is set equal to $S_{g\min}$.

If $P_g - P_o$ exceeds the highest capillary pressure in the table (corresponding to the highest gas saturation value $S_{g\max}$), the gas saturation is set equal to $S_{g\max}$ and the oil pressure is adjusted to follow the gas pressure gradient.

Consistency and overlapping transition zones

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 $S_{gmax} = 1 - S_{wmin}$ (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 $S_w = 1$ below the water transition zone). The consistency requirements between saturation tables are detailed in "[Saturation functions](#)". If the consistency requirements are not met, $S_{gmax} < 1 - S_{wmin}$, then not only will $S_o > 0$ above the gas transition zone, it may even increase. If the transition zones overlap, then the water saturation will decrease above the gas-oil contact. The gas saturation will increase until it reaches its maximum value. Above this point, as the water saturation continues to decrease, the oil saturation will increase.

In cases where the water-oil and gas-oil transition zones closely overlap, equilibration 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. 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

$$P_{cow}(S_w) + P_{cog}(S_g = 1 - S_w) = P_g - P_w \quad \text{Eq. 37.3}$$

Heavy oil

ECLIPSE 300 Thermal

If oil is heavier than water (based on the composition provided at the datum depth) and if the water-oil contact lies above the reservoir, then the reservoir will be initialized with heavy oil with the datum pressure being interpreted as the pressure of the oil phase. A non-zero value for item 217 in [OPTIONS3](#) can be used to restore the pre-2010.1 behavior for thermal heavy oil equilibration.

Accurate fluids in place calculation

There is a choice of methods to calculate the initial fluid in place values (the initial oil, water and gas saturations) in each grid block. These are selected using the ninth argument of the [EQUIL](#) keyword, which specifies the number of sub-layers (N) used to obtain the initial saturations.

Center-point equilibration (N=0)

The simulator sets the fluid saturations in each grid block according to the conditions at its center. This is the fastest method, and yields a stable initial state since the phase pressure differences in the simulation are also taken between cell centers. But it is the least accurate method, particularly in cases where the fluid contact passes through large grid blocks.

Horizontal fine grid equilibration (N<0)

The top and bottom halves of each grid block are each divided into -N layers of equal thickness, and the saturations are determined locally in each layer. The phase saturations for the block are set equal to the average of the saturations in each layer. This option provides a more accurate calculation of the fluids in place, but may yield a solution that is not completely stable in the initial state.

Tilted block fine grid equilibration (N>0)

This option is similar to the previous one, 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 of the grid blocks 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 generally 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 for example). 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 option

If the reservoir contains two or more communicating equilibration regions having different pressures or contact depths, fluid flows between these regions immediately the simulation begins. The ‘quiescence switch’ only produces local steady state conditions within each separate equilibration region. Global quiescence, however, can be achieved by using the threshold pressure facility. The facility prevents flow between adjacent equilibration regions from occurring until the potential difference exceeds a specified threshold value. Thereafter the potential difference for flow across equilibration region boundaries is reduced by the appropriate threshold value.

The Threshold Pressure facility is enabled by including the switch [THPRES](#) in keyword [EQLOPTS](#) in the RUNSPEC section. Each interface between adjacent equilibration regions then has to have its own threshold pressure. If the threshold pressure is also irreversible (the switch [IRREVERS](#) in keyword [EQLOPTS](#) in the RUNSPEC section), each interface has two threshold pressures, one for flow in each direction. The threshold pressure values are set using the keyword [THPRES](#). Values can be defaulted to the maximum potential difference between communicating cells across the interface (that is, just sufficient to prevent flow occurring across the interface before the reservoir is perturbed).

The quiescence option

ECLIPSE 100

When using the fine grid equilibration algorithm to achieve a more accurate fluids-in-place calculation, the initial conditions may no longer be in hydrostatic equilibrium for each phase. This is because the

saturations are now taken as average values over a grid block rather than the values at cell centers. The values of pressure in the fine grid equilibration are still taken from block center values.

The Quiescence option, set using the [EQLOPTS](#) keyword, 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

$$P_{wat} = P - P_{cow} + PMODW \quad \text{Eq. 37.4}$$

and

$$P_{gas} = P + P_{cog} + PMODG \quad \text{Eq. 37.5}$$

The phase pressure modifications $PMODW$ and $PMODG$ are determined to achieve quiescence at initial conditions and are then applied throughout the simulation. These can be thought of as frozen pseudo-capillary pressure modifications. The quiescent pressures are constructed from the initial tables of phase pressure versus depth, p_{oil} , p_{gas} and p_{wat} over the depth of the field. Then, in the absence of capillary pressures, $p_{oil} = p_{gas}$ at the oil-water contact and $p_{oil} = p_{gas}$ at the gas-water contact.

The (oil phase) pressure in each grid block center, P is modified by

$$P = \max(P, p_{wat} - (\tilde{\rho}_{wat} - \tilde{\rho}_{oil})H/2, p_{gas} - (\tilde{\rho}_o - \tilde{\rho}_g)H/2) \quad \text{Eq. 37.6}$$

where

H is the height of a cell, and

ρ denotes the phase gravity density.

The water and gas phase pressure modifications are then determined from

$$PMODW = sign(p_{wat} - P + P_{cow}) \times \min(|p_{wat} - P + P_{cow}|, (\tilde{\rho}_w - \tilde{\rho}_o)H/2) \quad \text{Eq. 37.7}$$

and

$$PMODG = sign(P + P_{cog} - p_{gas}) \times \min(|P + P_{cog} - p_{gas}|, (\tilde{\rho}_o - \tilde{\rho}_g)H/2) \quad \text{Eq. 37.8}$$

These phase pressure modifications are chosen such that the water phase pressure P_{wat} approximately follows the hydrostatic water pressure curve p_{wat} in the presence of mobile water, and likewise so that the gas phase pressure P_{gas} approximately follows the hydrostatic gas pressure curve p_{gas} in the presence of mobile gas.

Note that even for a model with zero capillary pressures, the phase pressure modifications is not necessarily zero. The phase pressure modifications can be reported using the mnemonic ‘QUIESC’ in the [RPTSOL](#) keyword.

Reservoir temperature

The reservoir temperature must be specified in two different types of simulations:

- Compositional cases, where the temperature is constant in each cell (though it can vary with depth), but is needed to determine the fluid properties.
- Thermal or Temperature option cases, where the temperature can change with time. The initial temperature must be specified to determine the initial energy in the reservoir.

ECLIPSE 300	In compositional cases, the constant reservoir temperature can be specified by the RTEMP or TEMPVD keywords in the PROPS section.
ECLIPSE 300 Thermal	In Thermal cases, the initial reservoir temperature can be set using the RTEMPVD keyword in the SOLUTION section. The PROPS section keywords RTEMP or TEMPVD can be used as an alternative.
Temperature option	In Temperature option cases, the initial reservoir temperature can be set using the RTEMPVD keyword in the SOLUTION section, or the RTEMP and RTEMPA keyword in both the PROPS and SOLUTION sections. In ECLIPSE 100, the keywords RTEMP and RTEMPA are accepted in the PROPS section only if the EQUIL keyword is defined in the SOLUTION section. Non-equilibrium initializations require these keywords to appear in the SOLUTION section only.

Phase pressures in ECLIPSE 300

By default, oil and gas phase pressures are derived from the pressure solution variable P using the formula

$$P_{oil} = P - P_{cog} \times \left(\frac{S_g}{S_o + S_g} \right) \quad \text{Eq. 37.9}$$

and

$$P_{gas} = P + P_{cog} \times \left(\frac{S_o}{S_o + S_g} \right) \quad \text{Eq. 37.10}$$

where

P_{cog} is the gas-oil capillary pressure,

S_o and S_g are the oil and gas saturations respectively.

This P_c splitting technique is performed to enable better convergence and property smoothness above the critical point. It may be disabled using keyword [FORMOPTS](#), option NOPCSPT. When reporting initialization data using the [RPTRST](#) or [RPTSOL](#) keywords, the pressures are defined as

[PRESSURE](#) is the solution variable P which is derived from the phase pressures calculated during the [EQUIL](#) hydrostatic equilibration and reported with the SOLVD option in RPTSOL.

[POIL](#) is the oil phase pressure P_{oil} defined in equation 37.9

[PGAS](#) is the gas phase pressure P_{gas} defined in equation 37.10

Note: The P_c splitting technique applies only to the ECLIPSE 300 compositional mode.

Equilibration keywords

RUNSPEC

- [EQLDIMS](#) Dimensions of equilibration tables
- [EQLOPTS](#) Options for equilibrations

REGIONS

- [EQLNUM](#) Equilibration region numbers

SOLUTION

Restriction	Keyword	Description
	COMPVD	Composition versus depth table
	EQUIL	Datum depth and pressure, contact depth and capillary pressures, equilibration parameters
	PBVD	Bubble point P_b versus depth table for black oil
	PDVD	Dew point p_d versus depth table for black oil
	RSVD	R_s versus depth table for black oil
Temperature Option	RTEMPA	Initial temperature
Temperature or Thermal Options	RTEMPVD	Initial temperature versus depth table
	RVVD	R_v versus depth table for black oil
	TVDP	Tracer versus depth table
E300 only	WMFVD	Water composition versus depth table.
E300 only	ZMFVD	Composition versus depth table.

Enumeration

In some cases it may be required to start a study from a point in existing production, where a restart cannot be used. For example, when starting a compositional simulation using the results of a black oil model. As flow patterns have developed, this cannot be done by equilibration. Enumeration allows the initial solution to be specified directly. Three sets of data are required for all simulations:

- The pressure:
[PRESSURE](#), or, for ECLIPSE 100, [PRVD](#).
- Saturations:
[SGAS](#) and [SWAT](#)
- Composition:
For a black oil simulation with dissolved gas, either [PBUB](#) or [RS](#).
For a black oil simulation with vaporized oil, either [PDEW](#) or [RV](#).
For a compositional simulation with two hydrocarbon phases, both [XMF](#) and [YMF](#).
For a compositional simulation with only a single hydrocarbon phase, [ZMF](#).

The above three sets of data are sufficient for most simulations. However, special options may require additional data. For example:

- ECLIPSE 300 • Hydrocarbon components in water:

[AMF](#)

- ECLIPSE 300 • Water components in water:

[WMF](#)

- ECLIPSE 300 • Solid model

Solid composition:

[SMF](#)

- ECLIPSE 300 • The temperature:

[TEMP I](#)

There are several alternative ways of entering this data:

- Like GRID and EDIT section keywords, all the enumeration keywords may also be set using [EQUALS](#), [ADD](#) and [MULTIPLY](#) applied to boxes of cells.
- Data can be read directly from a flexible 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.

- ECLIPSE 300 • Instead of [XMF](#) and [YMF](#), tables of composition with respect to pressure can be entered 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. This should help ensure that the chosen liquid and vapor are close to phase equilibrium and avoid large transient effects that occur when the initial phase are not in equilibrium. This is discussed further in "[Specifying the initial state of a run in ECLIPSE 300](#)".

Specifying the initial state of a run in ECLIPSE 300

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 , $i = 1, \dots, N_c$). The condition that the liquid and vapor be in equilibrium, in the compositional case, is:

$$f_{iL}(P, x) = f_{iV}(P, y) \quad \text{Eq. 37.11}$$

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:

$$z_i = x_i L + y_i V \quad \text{Eq. 37.12}$$

with

$$L = \frac{S_o b_o^m}{S_o b_o^m + S_g b_g^m} \quad \text{Eq. 37.13}$$

where b_o^m and b_g^m are the oil and gas molar densities.

As long as equation 37.11 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 equation 37.11 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 inter phase 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 undersaturated. 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 inter phase equilibrium is observed for any specified cell pressures.

Enumeration keywords

PROPS

The following keywords can only be used in ECLIPSE 300.

- [XMFVP](#) Liquid composition versus pressure table
- [YMFVP](#) Vapor composition versus pressure table

SOLUTION

Restriction	Keyword	Description
E300 only	AMF	Initial water compositions
E300 only	GETSOL	Obtain initial solution array from a restart file.

Restriction	Keyword	Description
	PBUB	Initial bubble point pressures
	PDEW	Initial dew point pressures
	PRESSURE	Initial pressures
E100 only	PRVD	Initial pressures versus depth tables.
	RS	Initial solution gas-oil ratios
	RV	Initial vapor oil-gas ratios
	SGAS	Initial gas saturations
E300 only	SMF	Initial solid compositions. T
E300 only	SOIL	Initial oil saturations
E300 only	SWAT	Initial water saturations
E300 only	TEMPI	Initial temperatures.
E300 only	XMF	Initial oil compositions.
E300 only	YMF	Initial gas compositions
	ZMF	Initial oil compositions

Mixed hydrostatic equilibration

Thermal

An additional initialization method has been added for Thermal simulations. This option should be triggered using items 1 and 2 of the [EQUIL](#) keyword, together with saturation arrays ([SOIL](#), for instance) and the [NEI](#) keyword.

For each equilibration region, the datum depths and the pressure at a given datum depth are specified in order to determine the initial pressure map for the simulation model. The number of depth points in this table is set using the keyword [EQLDIMS](#) in the RUNSPEC section. The depth points are spaced equally throughout the region. The software algorithm sets up an internal table of pressures against depths. Between each pair of depth points in the table, the pressure gradient is calculated iteratively using a density (assuming an oil phase) consistent with the average pressure within the depth step. The pressure values are calculated by stepping from the datum depth up to the top and down to the bottom of the equilibration region. The pressure and temperatures are then determined in each grid block, in the equilibration region, when the internal table is interpolated to obtain these values at the grid block center depth.

The [NEI](#) keyword provides a sample hydrocarbon composition for each equilibration region which is needed to determine non-equilibrium initialization composition tables internally by generating consistent oil and gas compositions for each cell. This is achieved by flashing to the cell pressure - effectively a constant composition expansion. The simulator tries to match the oil, water and gas saturations in each equilibration region, and if the specified compositions cannot match the phase saturations, then the components that have the highest gas phase mole fraction have their compositions adjusted.

If the [NEI](#) sample provided is only for an oil composition, then the pressure calculated for any gas phase present will not be defined efficiently. An alternative method is to use the [XMFVP](#) / [YMFVP](#) or [XMF](#) / [YMF](#) keywords.

A table of depth values against initial pressure and temperature values will be automatically output to the PRINT file under the header "MH_EQUIL" (Pressure with depth (Mixed Hydrostatic Equilibration)).

This initialization method should not be mixed with the standard equilibration method, otherwise an error message will be issued.

Single phase states in EoS compositional models

ECLIPSE 300

Distinguishing whether a single phase fluid is a liquid or a vapor is not always straightforward, and is sometimes arbitrary, for example if the fluid is supercritical, $P > P_c$ or $T > T_c$. If the fluid properties (for example density, viscosity) are determined correctly, then it might be thought that determining whether the fluid is liquid or vapor is only a matter of labelling / reporting and has no effect on the simulation. However, the label is important, since it effects wells controlled by liquid or gas rates.

To determine a single phase state at (P, T) correctly, the bubble point $P_{bub}(T)$, dew point $P_{dew}(T)$ or critical point (P_c, T_c) may be needed. These cannot be calculated quickly, and to do so for every flash would slow down the simulation significantly. Therefore an approximate critical temperature is calculated for the mixture using the Li formula:

$$T_c^{mix} = F_{Li} \cdot \frac{\sum_j T_{cj} \cdot V_{cj} \cdot z_j}{\sum_j V_{cj} z_j}$$

where

z_j is the mole fraction of component j in the fluid,

T_{cj} is the critical temperature of component j , given by the [TCRIT](#) keyword,

V_{cj} is the critical volume of component j , given by the [VCRIT](#) keyword,

F_{Li} is a tuning factor.

Single phase states are then labelled as liquid if $T_c^{mix} > T$ and gas if $T_c^{mix} < T$.

The tuning factor F_{Li} can be set using the [FACTLI](#) keyword. Alternatively the simulator will calculate a value of F_{Li} to ensure where possible that any single phase cell in the reservoir has the correct user-specified liquid or vapor state.

38

Licenses

License Overview

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

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

The FLEX license checking procedure checks for the availability of an appropriate license for an ECLIPSE option when a keyword activating the feature is encountered. In most cases, the activating keyword is in the RUNSPEC section, so is encountered at the beginning of the run.

Activating keywords for some features, however, may not be encountered until the SCHEDULE section, and perhaps even part way through a run. For example, the Standard Network option is activated by the [GRUPNET](#) keyword, and the Gas Field Operations Model may not be activated until the [SWINGFAC](#) keyword is read. If an appropriate license is not available when a particular feature is activated, the run stops.

A DATACHECK license is available, which allows ECLIPSE to run a dataset in NOSIM mode to check for data errors without taking up a full license.

ECLIPSE 100

To avoid the run stopping part way through, it is possible to reserve all the required licenses at the beginning of the run, using the [LICENSES](#) keyword in the RUNSPEC section to identify the features in the run for which licenses will be required.

Note: The [LICENSES](#) keyword is used by the ECLIPSE launchers to work out which licenses are used, except for parallel licenses, in order to implement the license aware job scheduling for ECLIPSE 100 and ECLIPSE 300.

This chapter provides information about the licenses used by specific simulators. A description of the keywords required to invoke these features is also given.

There are a number of special cases which will be dealt with separately:

- Open ECLIPSE
- Parallel
- Block Parallel
- Multiple Realization (MR).

Licenses used by specific simulators

In most cases, the licenses being used by the simulators are quite straight forward to work out. A base `eclipse` license is checked out and, depending on other factors, additional licenses may also be used.

Primarily it is the keywords in the dataset that determine which licenses are required. Standard usage is described below along with some exceptions and special cases.

ECLIPSE 100 uses a base `eclipse` license. A more complex example involving reservoir coupling is as follows:

It is possible to couple several ECLIPSE models together. See "[Reservoir Coupling](#)". If a dataset is coupled to others, it will contain the `SLAVES` keyword. In this case, the total license requirement is the same as if all the datasets had been run concurrently, with the additional requirement for a `rescoupling` license. For example, if the following datasets are coupled:

- A master dataset which only requires an `eclipse` license.
- A slave which requires an `eclipse`, `lgr` and a `networks` license.
- Another slave that requires an `eclipse`, `lgr` and a `multisegwells` license.

The total license usage is 3x `eclipse`, 2x `lgr`, 1x `networks`, 1x `multisegwells` and 1x `rescoupling`.

ECLIPSE 300 typically requires a `compositional` and an `eclipse` license. Again, there are several possibilities.

- If the ECLIPSE 300 dataset contains the `THERMAL` keyword, then ECLIPSE 300 will only check out an `eclipse` and a `THERMAL` license.
- If the user has an `e300` license available it will be used instead of an `eclipse` and a `compositional` license. However, the `e300` license is no longer sold.
- If the dataset contains the `BLACKOIL` keyword, a user can run ECLIPSE 300 in blackoil mode with an `eclipse` license and there is no need for a `compositional` license. Similarly if the dataset contains the `TEMP` keyword, the user can run ECLIPSE 300 in blackoil mode with an `eclipse` license and there is no need for a `THERMAL` license.

Which keywords invoke which features?

The following table below shows the feature names, the keywords that invoke the functionality and the license key name. Where two license names appear, the simulator will attempt to check out the first one and if this fails it will attempt to check out the second.

License Feature Name	Keywords	License Key	E100	E300
Compositional	COMPS	compositional		x
Datacheck	NOSIM	datacheck	x	x
ECLIPSE SynSize	PEDIMS, GRADRESV	eclipse4d	x	
Well Bore Friction (MAINTENANCE MODE)	FRICTION	friction multisegwells	x	x

License Feature Name	Keywords	License Key	E100	E300
Lagrangian Tracer Solver	LAGTRAC	fs_streamtrace frontsim		x
Gas Field Operations	GASFIELD, GASPERIO, GASYEAR, SWINGFAC	gasfield	x	x
Gas Lift Optimization	GLIFTOPT, LIFTOPT, WLIFTOPT	gaslift	x	x
Geomechanics	GEOMECH	geomechanics		x
Local Grid Refinement / Coarsening	LGR, CARFIN, RADFIN, AUTOREF, COARSEN, AUTOCOAR	lgr	x	x
Multiple Realization	MULTREAL is generated by a pre-processor	multiple_realisation parallel	x	x
Multisegment wells	WSEGDIMS, WSEGSOLV	multisegwells	x	x
Networks	NETWORK, GNETINJE, GRUPNET	networks	x	x
Open ECLIPSE Runtime License	Called by controller	openeclipse	x	x
Parallel ECLIPSE	PARALLEL	parallel multiple_realisation	x	x
Proppant flowback in a fracture	PROPFLOW	proppantflow		x
Reservoir Coupling	DUMPCUPL, GRUPMAST, GRUPSLAV, RCMASTS, SLAVES, USECUPL	rescoupling	x	
Reservoir Optimization	RESOPT	resopt		x
Sand Management Advisor	WELLSMA	sma		x
Thermal	THERMAL	thermal		x
Unencoded Gradients	FMTHMD, GDIMS, HMDIMS, RPTHMD, UNCODHMD	unencodedhmd	x	x
Water Services	RIVRDIMS, SAMG and summary vectors reporting hydraulic heads.	waterservices	x	

Table 38.1: Feature names, keywords that invoke the functionality and the license key name

Special Cases

Open ECLIPSE

Open ECLIPSE enables ECLIPSE 100 and ECLIPSE 300 to be controlled by, and communicate interactively with, other applications. Although ECLIPSE is primarily used in a batch standalone mode controlled solely by the contents of the input data file, there are many situations in which it would be advantageous to have ECLIPSE controlled by another software application.

For instance, there may be a requirement to have ECLIPSE tightly coupled to a surface gathering system model, or a specialized production optimization application. It could also be useful to couple ECLIPSE to an interactive controller, to allow you to view the current status of the simulation and make well management decisions during the course of the run. In addition to the licenses that ECLIPSE usually checks out, the simulator will also request an `openeclipse` license.

Parallel

The parallel option allows the simulation of a single data set to be distributed across a number of processors. This allows large simulations to be carried out in a shorter time than would normally be the case.

Both ECLIPSE 100 and ECLIPSE 300 are optimized to provide the shortest time to solution, and consequently tailor the number of domains, and hence the linear solver, to the number of available processors.

This has a number of consequences. Firstly, the results obtained on different numbers of processors will agree to engineering accuracy, not to machine accuracy. Secondly, the scalability is suboptimal, as the linear solver becomes less efficient as the number of processors increases.

Running ECLIPSE in parallel requires the same set of licenses as running in serial, plus a number of `parallel` or `multiple_realisation` licenses. The total number of these licenses is equal to the first argument of the `PARALLEL` keyword.

For a simulation requiring n parallel options, first the simulator attempts to acquire `nparallel` options. If unsuccessful and if the `parallel` option does not exist, the simulator attempts to acquire `nmultiple_realisation` options. If the `parallel` option exists, the simulator attempts to acquire as many `parallel` options as possible. Assuming that the simulator has acquired `nparallel` options, it will then attempt to acquire $n-npmultiple_realisation$ options.

The number of parallel licenses required is independent of how the model is run. As an example, say an ECLIPSE model is split into 8 domains, which could be run in a number of configurations. For instance, 1 computer with 8 processors, or 2 computers running 4 processes each, or 8 computers running 1 process each. For all these cases, the model is split into 8 domains, so 8 parallel licenses would be required. If the model was split into 16 domains, then 16 parallel licenses would be required.

Block Parallel

The Block Parallel license is activated by the `BPARA` keyword. This is an alternative license option for parallel simulations with up to eight parallel domains. In such cases, a single Block Parallel license can be used instead of the (up to) eight parallel licenses that would normally be required. To use this option the `PARALLEL` keyword must also be present, with item 1 (the number of domains) set to a value in the range 2 to 8. If more than eight domains are requested the simulation will stop with an error message. The Block Parallel license feature is not compatible with Multiple Realization or the Reservoir Coupling option.

Multiple Realization

Multiple Realization (MR) licensing was implemented as a way to lower the cost of running multiple similar versions of the same model.

Firstly, the user must generate MR datasets as they cannot be made by hand. Examples of programs able to generate MR datasets include MEPO and Petrel. The dataset differs from a standard dataset in that it includes the `MULTREAL` keyword.

Following the `MULTREAL` keyword is a UID (Unique Identifier), which is a long string of numbers and a word ('yes' or 'no'). All MR datasets generated from the same base case at the same time will have the same UID. The word, Yes or No, tells ECLIPSE whether to continue if MR licenses are not available. This is however not yet implemented in the generating applications.

When the first of a group with a single UID starts running, the standard set of licenses that would be required for that job is checked out. In addition, an MR license or, if not available, a parallel license is checked out as well. When the second and subsequent jobs from the same group start, they each only check out an MR license, or if not available, a parallel license. Even if the first job finishes, the standard set of licenses are kept checked out until all running jobs from that group finish.

Parallel and reservoir coupling are excluded from this licensing scheme, and if any dataset includes these features, they will be checked out as normal.

The example in figure 38.1 shows 6 MR jobs running through a system that can handle 3 jobs at a time.

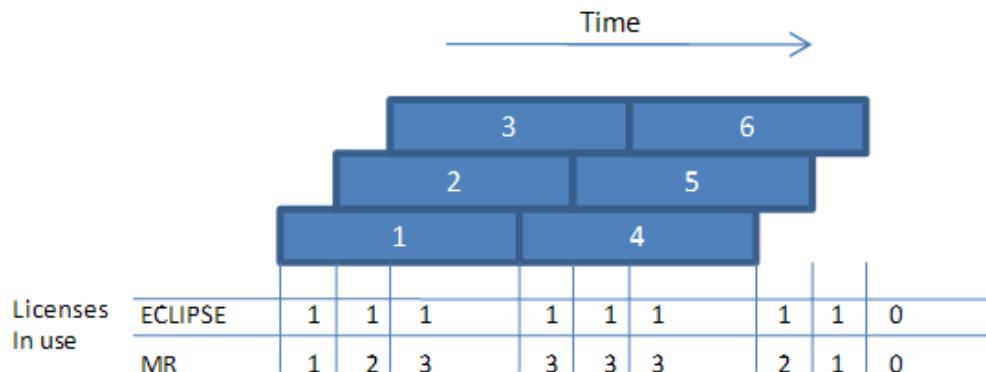


Figure 38.1. Example shows 6 MR jobs running through a system that can handle 3 jobs at a time

Lagrangian Tracer Solver

The Lagrangian tracer solver feature uses a FrontSim streamline (`fs_streamtrace`) license in the first instance. However, if no `fs_streamtrace` license is available then a full `frontsim` license will be taken instead. A Petrel locked FrontSim license will not activate this feature.

Listing of available licenses

The list of available licenses is not output by default. This output can be enabled by setting the `LISTLICENSES` keyword to TRUE in the configuration file.

License compatibility

License compatibility						
Keywords in deck			License availability			Result
m ways parallel?	Block parallel	Multiple realization	eclipse	bparallel	parallel	
PARALLEL	BPARA	MULTREAL				
$m \leq 8$	Y	N	1	1		Pass and run case
$m \leq 8$	Y	N	1	0		Error and exit
Any	Y	Y				Error and exit
$m > 8$	Y					Error and exit
Any	N	Follows parallel and multiple realization logic				

Table 38.2: License compatibility for parallel, block parallel and multiple realization

2014.1 license retirements

The 2014.1 release of ECLIPSE allows a number of options to be used without the need for a special license where previous releases had required one. These options are as follows:

- CO₂ Storage
- Coal Bed Methane Model
- Environmental Tracers
- Flux Boundary Conditions
- Foam Model
- Gas Calorific Value Control
- Polymer Flood Model
- Gi Pseudo-Compositional Model
- Solvent Model
- Surfactant Model

Local grid refinement and coarsening

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 auto-refinement facility may be used to refine an existing global model as the reverse process to grid coarsening.

Local grid refinement

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 $6 \times 4 \times 18 = 432$ cells. The [INRAD](#) keyword specifies the inner radius of the innermost block of the radial refinement.

This creates a refinement that looks like the following figure.

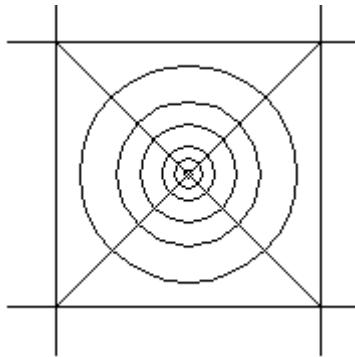


Figure 39.1. Single column radial grid refinement

Alternatively it is possible to use a two-dimensional model with NTHETA =1 that looks like the following figure.

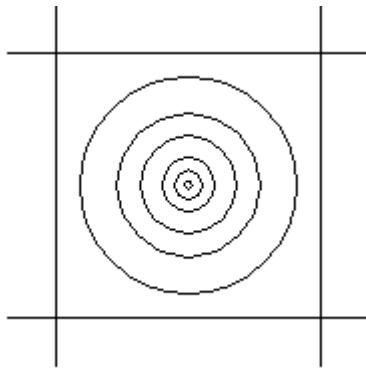


Figure 39.2. Single column radial grid refinement with NTHETA=1

By default, non-neighbor connections are generated automatically within the local grid to complete the circle connecting angular segment ITTHETA =1 to ITTHETA =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 $6 \times 8 \times 18 = 864$ cells. The [INRAD](#) keyword specifies the inner radius of the innermost block of the radial refinement.

This creates a refinement which looks like the following figure.

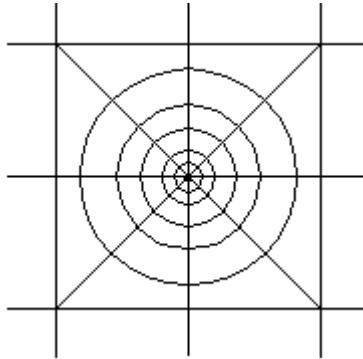


Figure 39.3. Radial refinement in a four column box

Alternatively, setting [NTHETA=4](#) creates a refinement which looks like the following figure.

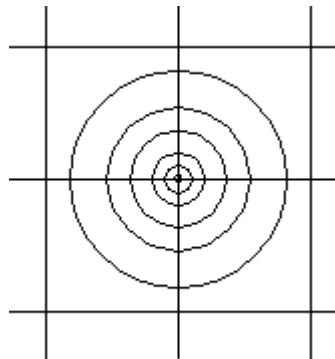


Figure 39.4. Radial refinement in a four cell box with $NTHETA=4$

By default, non-neighbor connections are generated automatically within the local grid to complete the circle connecting angular segment $ITHETA=1$ to $ITHETA=8$, unless circle completion is specifically disallowed by specifying ‘[INCOMP](#)’ in item 3 of the [COORDSYS](#) keyword entered between the [RADFIN4](#) and [ENDFIN](#) keywords. The [ENDFIN](#) keyword tells ECLIPSE to finish reading GRID data for the local

grid and to continue reading GRID data for the global grid. The permitted values of NTHETA (the number of angular segments) are 4 or 8. NWMAX is the maximum number of wells in the refinement. If NWMAX is not specified it is defaulted to 1, which is the normal value in a radial refinement.

To reserve memory for the refinement, set MAXLGR=1 and MAXCLS=864 in the keyword [LGR](#) in the RUNSPEC section, for example:

```
LGR
--MAXLGR  MAXCLS
 4      864 /
```

Placing wells in local grids

To place a well in a local grid, use keywords [WELSPECL](#) and [COMPDATL](#) instead of [WELSPECS](#) and [COMPDAT](#). WELSPECL is similar to WELSPECS except that there is an additional item - the name of the local grid - between the group name and the I-J location. COMPDATL has exactly the same items as COMPDAT. In these keywords the I, J and K locations of the wellhead and the connecting cells refer to the cell coordinates in the local grid system. The I and J locations of the connecting cells must be entered explicitly in COMPDATL, and should not be defaulted as they can be in COMPDAT.

To complete well PRODUCER in layers 9 to 11 of the refined grid named 'SOUTH', in all four theta sections of the innermost block:

```
WELSPECL
-- WELL -- GROUP -- LOCAL GRID -- LOCATION -- BHP -- PREF
-- NAME -- NAME -- NAME -- I J -- DEPTH -- PHASE
'PRODUCER' 'G' 'SOUTH' 1 1 8060 'OIL' /
/
COMPDATL
-- WELL -- LOCAL -- LOCATION -- OPEN -- SAT -- CONN -- WELL
-- NAME -- GRID IX JY K1 K2 -- SHUT -- TAB -- FACT -- DIAM
'PRODUCER' 'SOUTH' 1 1 9 11 'OPEN' 0 -1 0.4 /
'PRODUCER' 'SOUTH' 1 2 9 11 'OPEN' 0 -1 0.4 /
'PRODUCER' 'SOUTH' 1 3 9 11 'OPEN' 0 -1 0.4 /
'PRODUCER' 'SOUTH' 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 \(ECLIPSE 100\)](#)").

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 the following figure (in the areal direction).

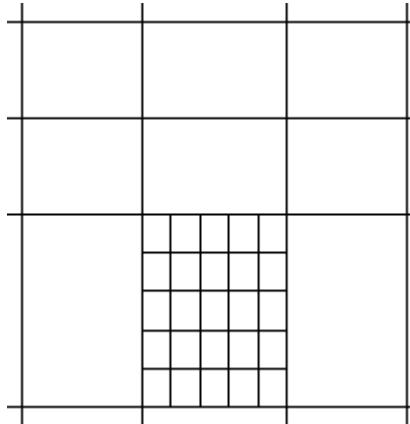


Figure 39.5. Cartesian grid refinement

It is recommended that you insert [ENDFIN](#) between successive [RADFIN](#), [RADFIN4](#) and [CARFIN](#) keywords. The primary purpose of [ENDFIN](#) is to revert ECLIPSE to reading data for the global grid system.

Note that the [CARFIN](#) keyword is designed to refine a general box, rather than just a single column of cells as in [RADFIN](#) or a box of four columns 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:

```
WELSPEC1
-- WELL -- GROUP -- LOCAL GRID -- LOCATION -- BHP -- PREF
-- NAME -- NAME -- NAME -- I J -- DEPTH -- PHASE
'HORIZ'   'G'       'LGR1'     1 3       8060   'OIL'   /
/
```

```

COMPDATL
-- WELL -- LOCAL -- LOCATION --- OPEN - SAT - CONN - WELL - KH - S - D - PEN
-- NAME -- GRID -- IX JY K1 K2 -- SHUT - TAB - FACT - DIAM - - - - DRN
'HORIZ' 'LGR1' 1 3 3 3 'OPEN' 0 -1 0.4 3* 'X' /
'HORIZ' 'LGR1' 2 3 3 3 'OPEN' 0 -1 0.4 3* 'X' /
'HORIZ' 'LGR1' 3 3 3 3 'OPEN' 0 -1 0.4 3* 'X' /
'HORIZ' 'LGR1' 4 3 3 3 'OPEN' 0 -1 0.4 3* 'X' /
'HORIZ' 'LGR1' 5 3 3 3 'OPEN' 0 -1 0.4 3* 'X' /
/

```

Defining local grid block sizes

Most GRID section keywords may be inserted between [CARFIN](#), [RADFIN](#) or [RADFIN4](#) and [ENDFIN](#) to alter refined grid block properties from those of the host cells. Alternatively, grid data for local grids may be input using the [REFINE](#) keyword if the local grid has previously been defined using [CARFIN](#), [RADFIN](#) or [RADFIN4](#).

If [DX](#), [DY](#) or [DZ](#) are specified for the local grid then ECLIPSE checks for consistency with the dimensions of the global host cells. Local grid block sizes can also be defined in terms of fractions of the host grid using the following keywords, which are especially useful if the global grid has been defined in corner geometry:

[NXFIN](#), [NYFIN](#), [NZFIN](#) Define number of local cells in each host cell

[HXFIN](#), [HYFIN](#), [HZFIN](#) Define ratios for splitting the host cell into local blocks.

For example, to define a local grid of 5 layers in a host of 2 layers:

```

CARFIN
'LGR2' 3 5 1 2 4 5 6 4 5 /
NZFIN
3 2 /
HZFIN
1.0 2.0 3.0
2.0 1.0 /

```

This gives 3 local layers in the top host layer, with DZ values in the ratio 1:2:3 and 2 local layers in the bottom host layer, with DZ values in the ratio 2:1. For example, if the host layer thicknesses are 60 ft. and 30 ft., local DZ values are 10 ft., 20 ft., 30 ft., 20 ft., 10 ft.

This option could be used to set up a logarithmic refinement, for example for a Cartesian refinement:

```

HXFIN
1.0 10.0 100.0 /

```

Note that the numbers used in the [HXFIN](#), [HYFIN](#), [HZFIN](#) data could correspond to actual [DX](#), [DY](#), [DZ](#) values, but these need not fit exactly in the host cell, as they are adjusted automatically. If no ratios are given, the default is to split the host cell in equal proportions. Data may be defaulted for some or all host cells, but data must be complete if given for a particular host cell. For example, to divide a host grid of 3 layers into 7 local cells, with splitting ratios defined for the middle layer:

```

NZFIN
3 2 2 /

```

```
HZFIN
3* 1.0 2.0 2* /
```

The keywords HZFIN and NZFIN can also be used in radial refinements declared by RADFIN or RADFIN4, to define the vertical splitting. INRAD should always be specified for radial refinements. If OUTRAD is specified after RADFIN or RADFIN4 then its value applies to the inner radius of the outermost cell of the local grid, because the outermost cell assumes the odd shape required to fit into the Cartesian host cell. If DR or DRV values are specified after RADFIN or RADFIN4 then the sum of INRAD and the DR values must be less than the DX and DY values of the host cells so that the refinement fits within the host cell dimensions. An incomplete set of DR or DRV values may be provided for a local grid and any missing data for the outer rings is completed automatically as a geometric series (ECLIPSE 100), or logarithmic series (ECLIPSE 300) to fit within the host block. Rather than supplying the DR values directly, you can instead define relative sizes for the DR values using the HRFIN keyword:

```
RADFIN
--NAME I1 I2 J1 J2 K1 K2 NR NT NZ
'LGR3' 6 6 4 4 1 5 5 4 20 /
INRAD
0.25 /
OUTRAD
3000.0 /
HRFIN
1.0 2.0 3.0 4.0 /
```

This gives *NR* - 1 linearly increasing DR values scaled such that their sum is equal to OUTRAD. HRFIN could be useful when the default logarithmic distribution creates tiny cells near the well bore, which can lead to convergence problems.

If the global grid is corner-point, then the ZCORN and COORD keywords may be used between the CARFIN / ENDFIN and RADFIN / ENDFIN pairs for example 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 and transmissibility multipliers, for example, are copied to local grid cells from the global host cells. In most cases this is adequate, but it is possible to define properties directly for some or all of the cells within a local grid by specifying grid arrays between the CARFIN / ENDFIN, RADFIN / ENDFIN, RADFIN4 / ENDFIN or REFINE / ENDFIN pairs. When the end of the GRID section is reached, any values that have not been explicitly defined for local cells are then copied from the global host cells. For example:

```
CARFIN
-- NAME I1-I2 J1-J2 K1-K2 NX NY NZ NWMAX
'NORTH' 5 10 5 10 1 4 18 18 8 5 /
EQUALS
'PORO' 0.3 1 18 1 18 1 4 /
'PERMZ' 50.0 1 18 1 18 1 4 /
'NTG' 0.9 1 18 1 18 1 8 /
/
MULTIPLY
'PORO' 0.5 6 13 6 13 1 2 /
'NTG' 0.2 6 13 6 13 1 2 /
```

```
/  
ENDFIN
```

This defines the porosities only in the top four layers on the local grid, and these are subsequently modified for the top two layers with the [MULTIPLY](#) keyword. The porosities for cells in the bottom four layers are copied from their host cells in the global grid.

Since by default ECLIPSE does not copy any properties from host cells until the end of the GRID section is reached, it is not permitted to modify any property for cells within the local grid (by using the [ADD](#), [COPY](#), [MULTIPLY](#) or [OPERATE](#) keywords) without previously defining the property for those cells.

This behavior can be changed by specifying the [LGRCOPY](#) keyword in the RUNSPEC, GRID, EDIT, PROPS or REGIONS section, depending on the simulator used. This will cause global values to be copied ‘just in time’ to the corresponding local grid cells. There are limitations on the use of this associated with the order of the keywords in the data file - for instance, global values of DX must have been set before the LGR operations are carried out in order for the correct global parent cell to be allocated to each LGR cell. If DX is redefined for the global cells after an operation requiring the LGRCOPY functionality to be used has been invoked, this will trigger an error. Similar limitations apply to all keywords which may affect which global cell a particular LGR cell is physically located in.

Special care should be taken with the [MINPV](#) keyword. It is likely that a suitable minimum pore volume for the global grid will not be suitable for much smaller local grid cells. A more suitable value should be specified for them using [MINPVV](#).

Amalgamating Cartesian local grids (ECLIPSE 100)

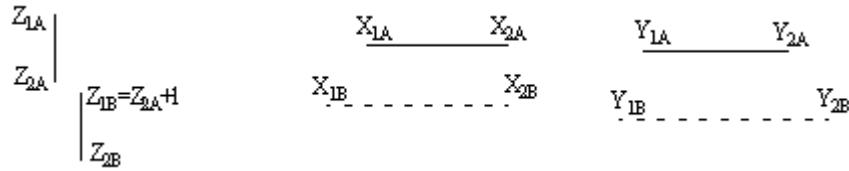
ECLIPSE 300 allows local grids to be placed anywhere in the reservoir provided they do not overlap. In ECLIPSE 100 a local grid cannot be placed next to another local grid.

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 zigzag pattern instead of a single large refinement.

Local grids within the same amalgamation are allowed to touch, but must not overlap. Several geometrical configurations allow amalgamation. Using the notation of the drawing below we have:

- Z-direction (A above B): If $Z_{1B} = Z_{2A} + 1$ then an amalgamation is defined if $X_{1B} \leq X_{2A}$ and $X_{1A} \leq X_{2B}$ and $Y_{1B} \leq Y_{2A}$ and $Y_{1A} \leq Y_{2B}$. This case corresponds to the drawing below. Swap indices A and B for the case B above A.
- X-direction (B on the right side of A): $X_{1B} = X_{2A} + 1$ then an amalgamation is defined if $Y_{1B} \leq Y_{2A}$ and $Y_{1A} \leq Y_{2B}$ in the case where NNCs and NEWTR are set. If no NNCs or NEWTR is set, then the following condition on the Z-coordinates is used: $Z_{1B} \leq Z_{2A}$ and $Z_{1A} \leq Z_{2B}$. For B on the left of A swap indices A and B.
- Y-direction: same as X-direction.

Note that this test is done for all pairs of active LGRs where a Flux Boundary ([USEFLUX](#)) is not used. This includes active and inactive cells.



Any valid Cartesian grids may be amalgamated, even if they do not touch. ECLIPSE treats an amalgamation as a single entity by lumping together all its constituent local grids, with non-neighbor connections between any adjacent grids in the amalgamation. The amalgamation is handled in the same way that ECLIPSE treats a single local grid; thus the 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, [WEOPENL](#) should be used instead of [WEOPEN](#) to open or shut wells or well connections; and [WFRICTNL](#) must be used instead of [WFRICTN](#) for the ECLIPSE 100 Wellbore Friction option. The '-L' versions of the keywords each contain an extra data item to specify which local grid the following connection belongs to. Either keyword from each pair may be used for wells completed in a single local grid.

For example, consider defining a deviated well in neighboring local grids 'WEST' and 'EAST'. The GRID section might look like:

```

CARFIN
-- NAME    I1-I2   J1-J2   K1-K2  NX  NY  NZ  NWMAX--
'WEST'    3 4     2 3     1 2     8   4   4   1   /
CARFIN
-- NAME    I1-I2   J1-J2   K1-K2  NX  NY  NZ  NWMAX--
'EAST'    4 5     4 5     1 2     8   4   4   1   /
AMALGAM
'EAST'  'WEST'  /
/

```

This generates a composite refinement which looks like the following figure (in the areal direction).

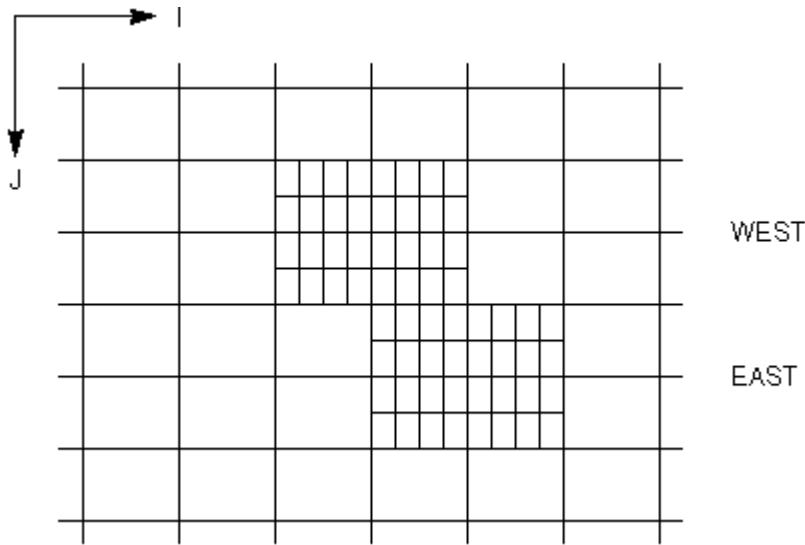


Figure 39.6. A composite refinement

A well may be specified in the SCHEDULE section in the following way:

```

WELSPECL
-- WELL -- GROUP -- LOCAL GRID -- LOCATION -- BHP -- PREF
-- NAME -- NAME -- NAME -- I J -- DEPTH -- PHASE
'DEV'      'G'      'EAST'      2 2      8000      'OIL'   /
/
COMPDATM
-- WELL - LOCAL - LOCATION - OPEN - SAT - CONN - WELL - KH - S - D - PEN -
-- NAME - GRID - I J K1 K2 - SHUT - TAB - FACT - DIAM - DRN -
'DEV'    'WEST'   6 3 3 3 'OPEN'   0   -1   0.3   3*   'X'   /
'DEV'    'WEST'   6 4 2 2 'OPEN'   0   -1   0.3   3*   'X'   /
'DEV'    'EAST'   2 1 2 2 'OPEN'   0   -1   0.3   3*   'X'   /
'DEV'    'EAST'   2 2 1 1 'OPEN'   0   -1   0.3   3*   'X'   /
/

```

Editing local grids

It is possible to modify data for local grids in the EDIT section by inserting modifications between a **REFINE** / **ENDFIN** pair, just as for the global grid, in order to modify pore volume, depth and transmissibility data. This is not recommended for general use and should be used with care. For example:

```

REFINE
'NORTH'  /
EQUALS
'PORV'   1000.0  1 18 1 18 1 8 /
'TRANZ'   0.0    1 18 1 18 4 4 /
/
ENDFIN

```

edits PORV and TRANZ within the local grid NORTH.

If the pore volume is edited for any cells in the global grid that have been refined, then ECLIPSE carries over the modification to local cells by applying an additional pore volume multiplier to the local grid, which is defined as the ratio of edited to unedited pore volumes for the global host cells. Similarly, when transmissibilities are edited on the global grid, ECLIPSE imposes an additional transmissibility multiplier to the local grids which is determined in the same manner. This additional transmissibility multiplier applies to regular as well as to non-neighbor connections within the local grids, and also to connections between the local and global grids.

Grid and solution reports

Just as [WELSPECL](#) and [COMPDATL](#) (or [COMPDATM](#)) replace [WELSPECS](#) and [COMPDAT](#) for refined wells, so [RPTGRIDL](#) replaces [RPTGRID](#) in the GRID data section grid reports for all local grids. RPTGRID may also be used in the normal way to produce global grid reports.

ECLIPSE 100

Refined cells have the decimal point replaced by R in reports output to the PRT file using the RPTGRID, [RPTSOL](#), and [RPTSCHED](#) keywords.

Tuning

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 timestep in all local grids.

Timing reports

ECLIPSE 100

Extra timing arguments have been added to the timing reports. These reports are initiated by using argument 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).

Good practice in designing LGRs

When considering a grid refinement it is good practice to conduct sensitivity studies comparing alternative grids. This allows the engineer to understand the choices made between execution speed and accuracy of results. It is particularly important to ensure adequate resolution of areas where solution variables are likely to vary rapidly, such as oil/water or gas/water contacts in the region of a well. When determining these areas consider the connectivity between cells around an LGR; for example, transmissibility between cells in the vertical direction can be larger than other directions due to the aspect ratio of cells and consequently full depth LGRs may be more important than wider areal extent.

Dual, multiple and triple porosity

In dual, multiple and triple porosity runs, the global coordinates specified in [CARFIN](#) and [RADFIN](#) refer to the (outer, in the case of multi-porosity) matrix blocks only. For example, in dual porosity, in an 8*8*6 global grid, to refine the matrix cell at (8, 8, 2) together with its corresponding fracture cell at (8, 8, 5) use;

```
CARFIN
--
      I1 I2 J1 J2 K1 K2 NX NY NZ NW
'CARF882' 8 8 8 8 2 2 5 5 10 3 /
ENDFIN
```

The refinement has 5*5*5 matrix cells and 5*5*5 fracture cells. For multiple and triple porosity, the same principle applies, namely that the global grid z- or k-extents (K1 and K2, data items 6 and 7) are always defined for the matrix grid only, but the size of the local grid in the z- or k-direction (NZ, data item 10) covers all porosities. To find its size in one porosity only (for example in the matrix) it must be divided by NPOROS, which is NMATRIX+1, where NMATRIX is the number of matrix porosities (for multi porosity, it is data item 1 of keyword [NMATRIX](#), or it is 1 for dual porosity), or the number of matrix and vug porosities (for triple porosity, it is data item 1 of keyword [TRPLPORS](#)). Conversely, NZ must be divisible exactly by NPOROS.

ECLIPSE 100

If the keyword [DPNUM](#) has been used to specify regions of the reservoir for which a conventional single porosity model is to be used in a dual porosity run, then the local grid may not straddle different DPNUM regions. A local grid in a dual porosity run must either lie entirely within a dual porosity region or entirely within a single porosity region. Within an LGR amalgamation, all the constituent local grids must be in the same DPNUM region.

Unstructured local grid refinements

Unstructured (PEBI) grids can now be run as local grid refinements. The [EXTFIN](#) keyword can be used to specify an unstructured local grid refinement. The [EXTHOST](#) keyword can be used to specify the global host cells for an unstructured local grid refinement, or similarly, the [EXTREPGL](#) keyword can be used to specify the replaced global cells for an unstructured local grid refinement. Finally, the [TRANGL](#) keyword can be used to specify the global-local connections and associated transmissibilities for an unstructured local grid refinement.

Between any `EXTFIN/ENDFIN` bracket, either `EXTHOST` or `EXTREPGGL` 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` and `PVTNUM` for example) 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 [TIME](#), [TSTEP](#) or [DATES](#) keyword is read in the restart run. In order to continue to model the well, a new counterpart well should be defined in the local grid (with a different name) using [WELSPECL](#) and [COMPDATL](#) or [COMPDATM](#), which has all the attributes of the existing shut well. ECLIPSE then ignores the existing global well, and an error condition results if the well is ever to be reopened. This procedure is not recommended in general.

ECLIPSE 300

Flexible restarts can be used in ECLIPSE 300 to add LGRs on a restart run, given that no LGRs existed in the base run. The new LGRs should be added to the GRID section (as normal). No other keywords are required. The new LGRs are turned on at the next 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 are 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 timesteps. In addition SUMMARY file quantities are output at the local timesteps; 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](#)"), there is little point in dumping SUMMARY files for the local timesteps since the global and local timesteps are all synchronized.

The local timestep 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 timestep 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](#)”) 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

$$\text{TRANN}_{lg} = \frac{\text{CDARCY} \cdot \text{TMLTX}_l \cdot \text{DIPC}}{B} \quad \text{Eq. 39.1}$$

where

CDARCY is Darcy's constant in the appropriate units

TMLTX_l is the X+ transmissibility multiplier for global cell g

DIPC is the dip correction

and

$$B = \frac{1}{2A_l} \cdot \left(\frac{\text{DX}_g}{\text{PERMX}_g} + \frac{\text{DX}_l}{\text{PERMX}_l} \right) \quad \text{Eq. 39.2}$$

The area is given by

$$A_l = \text{DY}_l \cdot \text{DZ}_l \cdot \text{NTG}_l \quad \text{Eq. 39.3}$$

The dip correction is given by

$$\text{DIPC} = \frac{\text{DHS}}{\text{DHS} + \text{DVS}} \quad \text{Eq. 39.4}$$

with

$$\text{DHS} = \left(\frac{\text{DX}_l + \text{DX}_g}{2} \right)^2 \quad \text{Eq. 39.5}$$

and

$$\text{DVS} = (\text{DEPTH}_l - \text{DEPTH}_g)^2 \quad \text{Eq. 39.6}$$

Note: The convention used for transmissibility multipliers: For connections in the X-, Y-, Z- directions, the global value for the adjoining block is used. For connections in the X+, Y+, Z+ directions, the local grid block value is used.

Transmissibilities between local cells in adjacent amalgamated local grids are calculated from the usual formulae when the [NEWTRAN](#) method is used. When [OLDTRAN](#) or [OLDTRANR](#) is used, connections are possible between local cells on faces of adjacent refined cells in the global grid. The interface areas used for the calculation account for partial overlap. These overlap areas are obtained by projecting the

refinements on the faces of adjacent refined cells in the global grid onto the unit square; then the proportion of cell areas in each overlap is applied as a multiplier to the actual local face areas in the OLDTRAN formula.

A report on transmissibilities between the local and global grids and between any amalgamated local grids can be obtained by using the mnemonic ALLNNC (switch 24) in [RPTGRIDL](#).

Treatment of transmissibility multipliers in local grids

It is possible to enter transmissibility multipliers [MULTX](#), [MULTY](#) and [MULTZ](#) for example 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 [MULTTHHT](#) 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 for example, 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 the 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*
 - Separate local grid refinements must not be placed in adjacent cells of the global grid, unless the refinements belong to the same amalgamation.
 - Partially overlapping LGRs are not allowed.
 - Keyword [NINENUM](#) is not compatible with the local grid refinement option.
- ECLIPSE 100*
 - 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.
 - It is not currently possible to coarsen within a local grid.
- ECLIPSE 100 only*
 - 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*
 - Adjoining radial LGRs are allowed.
 - 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).

- 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.
- 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](#)).
- Numerical aquifers cannot be attached to local grids.
- The diffusion options (molecular diffusion and environmental tracers) are not available in local grid runs.
- The [NNC](#) or [EDITNNC](#) keywords for the global grid should not refer to non-neighbor connections connecting refined cells in the global grid.
- It is not possible to use the grid coarsening option when restarting a run.
- Local grids cannot be added to a radial 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](#), [WEOPEN](#), [WEOPENL](#), [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](#), [WEOPENL](#), [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](#), [WEOPEN](#), [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 timestep. 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 timestep. 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 for example, 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

- The keyword [GCONTOL](#) cannot be used in local grid refinement runs.

ECLIPSE 100

- Keyword [WLIMTOL](#) only works for economic limit checks on wells in active local grids. It is ignored in all other limit violation checks.

ECLIPSE 100

- In parallel runs, LGRs cannot be switched on (using the [LGRON](#) keyword) after they have been switched off.

ECLIPSE 100

- In parallel runs, wells cannot be declared inside of LGRs (using the [WELSPECL](#) keyword) after LGRs have been switched off.

Other problems with local grid wells

- 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 unrefined 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

- 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 timestep. 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 timestep. 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.

Gravity and grid effects

For grid cells in an LGR that connect to global grid cells at very different depths, gravity effects might cause the solution to look unusual. For example if the local grid is defined through all the layers, gravity effects might cause water to accumulate at the bottom layer of the LGR, while this is not seen if the LGR is removed. The reason is that there is a grid effect due to the difference in depth of the bottom layer of the local grid and the depth of the connecting global grid cells.

Geometry and grid data in LGRs

Transmissibility data

In general, if grid data are not specified for a local refinement (by inserting keywords between [CARFIN](#), [RADFIN](#) or [RADFIN4](#) and [ENDFIN](#)) then they are derived from the properties of the host cells. You should note the following conventions used for default local permeability and transmissibility data.

Permeabilities

Unless permeabilities are explicitly defined for a local radial grid, global [PERMX](#) data are copied to [PERMR](#) and global [PERMY](#) data are copied to [PERMTHT](#). If the PERMX and PERMY values are different, a warning is issued by ECLIPSE.

Transmissibility multipliers

Transmissibility multipliers ([MULTX](#), [MULTY](#), [MULTZ](#)) are applied to the X+, Y+ and Z+ faces of each grid block. For a local refinement, multipliers can be copied from the host grid only if the positive face of the local cell coincides with a host cell face; otherwise, a default value of 1.0 is used. If this is inappropriate, transmissibility multipliers should be explicitly defined for the local grid.

Note: For connections between local and global cells, the transmissibility multipliers for the adjoining global block are used for flows in the X-, Y- or Z- directions. The local grid block multipliers are used for flows in the X+, Y+ or Z+ directions.

Pore volumes

The pore volume of a refined global cell may differ from the sum of the pore volumes of the local cells which it contains, either because the local porosities and net-to-gross ratios differ from the values for the host cell or because of discrepancies in geometry.

To avoid significant volumetric errors for radial grids, the geometric volumes of the outer refined blocks are adjusted to fit the host cell. However, local porosities and NTG values are not modified, so differences in global and local pore volumes may still occur.

After computing the local pore volumes, ECLIPSE replaces the pore volume of the host cell with the sum of the refined pore volumes. If a difference of more than 1% is detected, a warning message is issued. Pore volume reports are produced after any modifications have been made.

Active cells

The local cells identification, specified using the [ACTNUM](#) keyword, is inherited from the parent global cell. This property can be overwritten by inserting the ACTNUM keyword between [CARFIN](#), [RADFIN](#), or [RADFIN4](#) and [ENDFIN](#) only if the value for the global cell is 1.

Radial refinements

1. A local radial grid may be defined in either a single column of cells (using [RADFIN](#)), or in a box consisting of four columns of cells (using [RADFIN4](#)). The geometry of the local grid may be defined using one of the following methods:
 - a. Refinements in the R-direction may be defined by user-given [DR](#) values. In the Z-direction, refinements can be specified by [NZFIN](#) / [HZFIN](#) data. Missing data are calculated automatically, so that a radial grid can be defined by a minimum set of one [RADFIN](#) or [RADFIN4](#) keyword and one [INRAD](#) keyword.
 - b. In a block-centered host, [DZ](#) values may be input directly, with radii defined as for the case (a). Use of user-defined [DTHETA](#) values is not recommended, as the local angles are usually restricted by the host cell geometry.
 - c. In a corner-point host, a local radial grid may be defined by corner-point coordinates. Note that the local [COORD](#) data are expected in cylindrical coordinates, relative to an origin at the center of the refinement. (Note: Cartesian local [COORD](#) data are defined relative to the global grid origin.)
2. A local radial refinement defined through [RADFIN](#) can only be inserted in a single column of host cells, with [NTHETA](#) = 1 or [NTHETA](#) = 4. Such a radial grid defined by user-specified corner points (in a corner-point host) must be 3-dimensional (that is, [NTHETA](#) = 4).

A local radial refinement defined through [RADFIN4](#) can only be inserted in a box of four columns of host cells, with [NTHETA](#) = 4 or [NTHETA](#) = 8. Such a radial grid defined by user-specified corner points (in a corner-point host) must be 3-dimensional (that is, [NTHETA](#) = 8).

A local radial grid cell must fit inside a single host cell.

3. The center of a radial refinement defined through [RADFIN](#) is assumed to be at the geometric center of the top host block.

The center of a radial refinement defined through [RADFIN4](#) is assumed to lie at the central node of the top of the refined box.

4. The sides of the inner radial cells are assumed to be vertical.
5. The innermost radius must be defined by [INRAD](#).

6. The radius for the inside of the outer local block can be defined by [OUTRAD](#). Otherwise it is calculated as $RMAX / 3.0$ where RMAX is the maximum radius of a circle contained within the radial refinement.
7. The user may provide some or all DR values. Any missing radii are completed as a geometric series (ECLIPSE 100), as follows:

No of radii = NR

No of radii defined by user = NU

Radius of inside of outer local block = RREF = R_{NR-1}

Last radius defined by user = RU

Then for $i = NU + 1$ to $NR-2$

$$R_{i+1} = FAC \cdot R_i$$

$$\text{where } FAC = \left(\frac{RREF}{RU} \right)^{1/(NR-NU-1)}$$

8. For local transmissibility calculations using RADFIN, the outer radius is taken as the area equivalent radius of the host grid block, based on the top area of the host cell. When RADFIN4 is used, the equivalent radius is based on the top area of the four host cells. For volumetrics and connections to the global grid, the sides of the local grid are taken from the coordinate lines for the host.

Note that the equivalent radii are shown in the output reports for the local COORD values.

9. The figure below shows the numbering convention used for connecting local radial grids to the host when RADFIN is used:

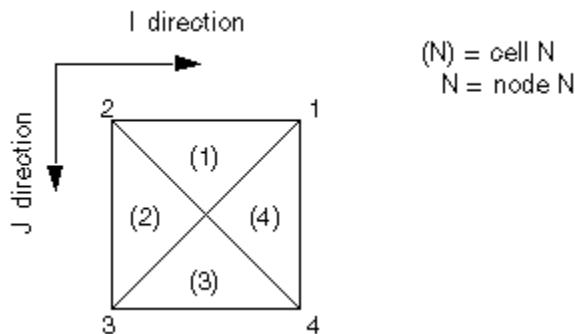


Figure 39.7. The numbering convention used by RADFIN for connecting radial LGRs to host cells

The radial cells are numbered so that Theta increases in an anti-clockwise direction, starting from the quadrant corresponding to the Y- face of the global cell. If necessary, the local coordinates are rotated to align the radial cell with its host.

Reported COORD data will show the corrected Theta values.

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 the following figure.

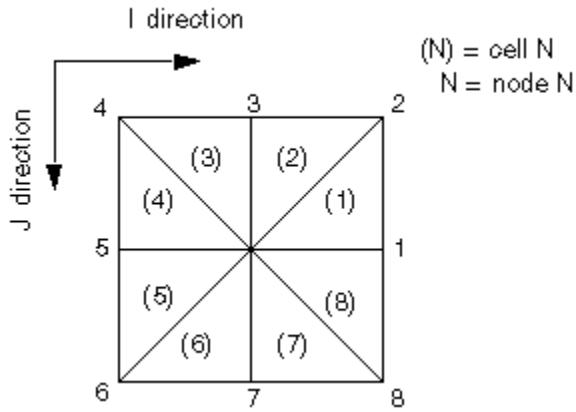


Figure 39.8. The numbering conventions used by RADFIN4 for connecting radial LGRs to host cells

In the above case, Theta increases anti-clockwise in (I,J) space, but there is of course no reason why the coordinate system should not be right handed, in which case theta increases clockwise.

10. Transmissibility calculations

For flow between local radial blocks, the standard ECLIPSE method based on pressure equivalent radii is used to compute transmissibilities.

Flow between the outer local block and the adjoining host block is treated as linear, but with the transmissibility center for the local block taken at the pressure equivalent radius. With radii computed as described earlier, and host cells which are approximately square in cross-section, this method corresponds to the approach described by Pedrosa and Aziz [Ref. 81], and Ewing et al. [Ref. 35].

Alphabetical list of GRID section keywords allowed within a local grid

Refer to the *ECLIPSE Reference Manual* for a further description of each keyword.

Keyword	Description
ACTNUM	Identifies active grid blocks.
ADD	Adds specified constants to specified arrays in the current box.
ADDZCORN	Adds a constant to the cell corner point depths array ZCORN.
BOX	Redefines 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.

Keyword	Description
DIFFMTHT	Theta direction diffusivity multipliers for current input box.
DIFFMX	X+ direct ion 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	Vertical dimension of a typical block of matrix material, for the entire grid.
DZMTRXV	Vertical dimension of a typical block of matrix material, for the current box.
DZNET	Grid block net thicknesses for the current box.
ENDBOX	Redefines 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	Imports 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.

Keyword	Description
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 and 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	Overrides principal solver directions within a local grid.
THCGAS	Gas phase thermal conductivities for the current box.
THCOIL	Oil phase thermal conductivities for the current box.
THCONR	Specifies rock and fluid thermal conductivities for the current box.
THCROCK	Rock thermal conductivities for the current box.
THCSOLID	Solid phase thermal conductivities for the current box.
THCWATER	Water phase thermal conductivities for the current box.
TOPS	Depths of top faces of grid blocks for the current box.
VEFIN	Controls the Vertical Equilibrium model within a local grid.
ZCORN	Depths of grid block corners.

Local grid coarsening

Grid Coarsening is the complement of local grid refinement. Cells may be amalgamated to reduce the total number of active cells in the global grid.

ECLIPSE 100

To reserve memory for grid coarsening, in keyword [LGR](#) in the RUNSPEC section set MCOARS equal to the number of coarsening cells (amalgamations), for example:

```

LGR
--MAXLGR   MAXCLS   MCOARS
    0        0        48   /

```

Because grid coarsening generates non-neighbor connections, the keyword [NONNC](#) must not be specified in the RUNSPEC section.

Coarsenings are specified in the GRID data section using keyword [COARSEN](#). For example, with NDIVIX =10, NDIVIY=10 and NDIVIZ=6 we can use:

```

COARSEN
-- I1 I2 J1 J2 K1 K2 NX NY NZ --
    1 10  1  4  1  6  5  2  3  /
    1  4  5 10  1  6  2  3  3  /
/

```

The first coarsening reduces the number of active cells from $10*4*6 = 240$ to $5*2*3 = 30$. The eight cells at (1,1,1), (2,1,1), (1,2,1), (2,2,1), (1,1,2) (2,1,2), (1,2,2), (2,2,2) are amalgamated into a single cell for simulation. The second coarsening reduces the number of active cells from $4*6*6 = 144$ to $2*3*3 = 18$, a reduction in the number of active cells by a factor of 8. Thus the number of amalgamated cells is MCOARS = $30+18 = 48$.

NX, NY, and NZ are the number of coarsened cells in each direction within each coarsening set. If values are not provided then default values are set to 1. Specified values of NX must be an exact factor of $I2 - I1 + 1$, and similarly for NY and NZ.

Coarsening sets must not overlap one another, but they may join together as in the above example. Thus the coarsening facility is extremely flexible, allowing the engineer to build a highly complex grid. Local grid refinements cannot be placed within or adjacent to a grid coarsening, that is the local grids may only be applied to non-coarsened cells.

Implementation

Grid coarsenings are set for the duration of a run and may not be switched on or off at different times during a run. The amalgamations are computed automatically by ECLIPSE which also computes the appropriate non-neighbor connections between different coarsenings and between normal cells and coarsened cells.

Within each amalgamation, a “representative cell” is chosen in the center of the amalgamation. All cells within the amalgamation apart from the representative cell are made inactive. The representative cell is given the properties (dimensions, permeability and pore volume for example) 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 reactivated 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.

ECLIPSE 100

Just as refined cells have the decimal point replaced by R in solution reports (specified using [RPTGRID](#), [RPTSOI](#), or [RPTSCHED](#)), so coarsened cells have the decimal point replaced by C.

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 defined in the following sections.

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

$$TOPS_c = \frac{\sum_f PV_f \cdot T\bar{OPS}_f}{PV_c}$$

where $T\bar{OPS}$ 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

$$TRANX_c = \sum_{JK} \left[\frac{1}{\sum_I \left[\frac{1}{TRANX_f} \right]} \right]$$

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 $\sum_{J=J1}^{J2} \sum_{K=K1}^{K2}$, and in the I-direction $\sum_{I=IL}^{IU}$,

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 and SOWCR, are taken from the values in the representative cell. No relative permeability upscaling is done here; take care not to define coarse cells too large, in order to encompass wide relative permeability variations between different rock types.

Any well completed in a coarse cell has its completions nominally moved to the representative cell (for reporting purposes) and the connection factor and drawdown are calculated using the upscaled quantities DX_c , DY_c , DZ_c , Kx_c , Ky_c , Kz_c and $DEPTH_c$. This means that in each coarse cell with a completion on the fine grid, the well is assumed to penetrate fully through the coarse cell in the appropriate direction of penetration.

Note: If the connection factor, Kh or r_o (pressure equivalent radius) is given in [COMPDAT](#), then this is not upscaled from fine to coarse grids.

Faults in coarsened regions

Where x- or y-faults exist in coarsened regions, these are represented by non-neighbor connections between cells which are physically adjacent despite having different z-coordinates. When coarsening is defined such that the fault lies between two original cells which will subsequently be in the same column of coarse cells, these connections are no longer sensible, as they connect cells in the z-direction where the geometry of the situation requires a connection in the x- or y-direction.

In this case, the ends of the non-neighbor connection are recalculated so that the connection is still made across the fault in the correct direction. This is geometrically equivalent to moving the fault to the edge of the new coarsened cell.

The transmissibility of the new non-neighbor connection is calculated exactly as for the upscaling example above, except that one of the transmissibilities in the sum will be the old non-neighbor connection transmissibility. Diffusivity and thermal transmissibility, if appropriate, are recalculated in exactly the same way.

Note: There can be only one fault per direction in any one coarsened cell. So a coarsened cell can contain both an x-fault and a y-fault, but not two x-faults or two y-faults.

Automatic refinement (ECLIPSE 100)

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 $NRX \times NRY \times NRZ$ cells. Each grid cell is simply refined uniformly by the factors NRX, NRY, NRZ in each dimension. It is possible to coarsen an auto-refined grid so as to achieve auto-refinement only in a part of the grid.

The base dataset should first be run without auto-refinement to check for any errors. Error and consistency checking of the dataset with auto-refinement is performed with respect to the base grid, but any output files such as GRID, RESTART or FLUX files, refer to the auto-refined grid.

Treatment of GRID section data in automatically refined grids

Data for the auto-refined 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 auto-refined grid is simply inherited from host cells in the base grid.

By default, data for the multiplier keywords, for example [MULTX](#), is also inherited from host cells in the base grid. Therefore in the auto-refined grid, [MULTX\(IB, JB, KB\)](#) applies to all auto-refined 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](#), for example, will be applied only between auto-refined cells on the X+ interface between these base grid cells.

If item 4 of the [AUTOREF](#) keyword is set to 2, then [MULTX](#), for example, assumes its default behavior in the auto-refined grid, and additional multipliers, for example [SMULTX](#), may also be entered, which only apply between auto-refined cells on the X+ interface between base grid cells.

Any numerical aquifers will be represented equivalently in the auto-refined grid by the central auto-refined 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 auto-refined. The keyword [AUTOCOAR](#) may be used to specify coarsening boxes in the base grid which are not auto-refined; this allows regions of the auto-refined 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 auto-refined grid in an appropriate manner. Edited pore volumes in the base grid are applied to auto-refined 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 auto-refined 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 auto-refined grid from the host cells in the base grid. This applies also to end-points specified in the PROPS section.

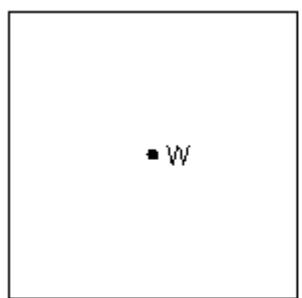
Treatment of SUMMARY section data in automatically refined grids

Any grid cell numbers or completion location indices for the base grid entered in the SUMMARY section are converted to the central auto-refined cell within the base cell. The SUMMARY and RSM files exhibit this conversion.

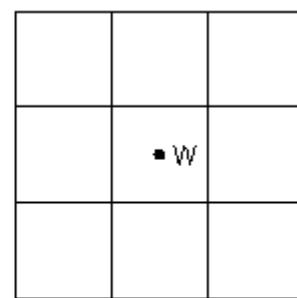
Treatment of wells in automatically refined grids

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 the following week in the case $NRX = NRY = 3$:



Well position in base grid



Well position in automatically refined grid

Figure 39.9. Well positioning in automatically refined grids

Similarly, a horizontal connection (COMPDAT item 13 = 'X' or 'Y') in the base grid is replaced by NRX or NRY connections in the central X or Y direction row of the base cell.

Note: The above rule is inappropriate for a well located at a corner or an edge of a base grid cell; ECLIPSE still places it in the central column or row of refined cells.

If the connection factor and Kh are not explicitly defined using keyword COMPDAT in the base dataset, then ECLIPSE calculates the connection factors within the auto-refined grid in the usual way.

If Kh is defined in keyword COMPDAT then its value in the auto-refined 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 auto-refined grid is then computed from the default formula on the auto-refined 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

- Local grid refinements may not be used with automatic refinement.
- The NNC and EDITNNC keywords are not available with automatic refinement.
- Wellbore friction and multisegment wells may not be used with automatic refinement.
- The nine-point schemes are incompatible with auto-refinement, and it is not possible to re-specify transmissibility multipliers within the SCHEDULE section when using auto-refinement.
- Radial local grid refinements are assumed vertical: see item 7 of "Radial refinements". Non-vertical refinements are not forbidden, as historically this has been an acceptable workflow. However, the code does assume that the central axis of the refinement is vertical, and, if the refinement is highly slanted, this may cause the axis to be outside of the refinement at its base, and so invert some cells. Users are advised to heed the warnings emitted in this case, and treat them as errors where possible

Summary of keywords

Common keywords

Section	Keyword	Description
EDIT, GRID, PROPS, REGIONS, SOLUTION, SCHEDULE	ENDFIN	Terminates data for a local grid.
EDIT, GRID, PROPS, REGIONS, SOLUTION, SCHEDULE	REFINE	Indicates that subsequent data refers to a named local grid.

RUNSPEC

Restriction	Keyword	Description
E100 only	AUTOREF	Sets options for auto-refinement.
	LGR	Sets options and dimensions for local grid refinement and coarsening.
E300 only	NOAIMLGR	Suppresses making all LGR cells implicit in AIM runs.

GRID section

Restriction	Keyword	Description
E100 only	AUTOCOAR	Is analogous to the COARSEN keyword and may be used in auto-refined grids.
E100 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.
	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.
E100 only	OUTRAD	Inner radius of the outer annulus in a radial LGR.
E100 only	QMOBIL	Controls the mobile end point correction in an LGR.
	RADFIN	Sets up a radial LGR.
E100 only	RADFIN4	Sets up a 4 column radial LGR.
	RPTGRIDL	controls grid section output for LGRs
	TRANGL	Specifies the global-local connections and associated transmissibilities for an unstructured local grid refinement.

Restriction	Keyword	Description
E100 only	SMULTX , SMULTY , SMULTZ	Alternative transmissibility multipliers for auto-refinement.
E100 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.

SOLUTION section

- [GETGLOB](#) Read only the global restart data.

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

Restriction	Keyword	Description
E300 only	COMPAGHL	Sets the gravity head in local grid completion cells.
	COMPDATL	Defines well completions in an LGR.
E300 only	COMPKRIL	Sets injection relative permeabilities in local grid completion cells.
	COMPLMPL	Lumps well completions for automatic workovers.
E300 only	COMPMBIL	Sets injection mobilities in local grid completion cells.
E100 only	COMPRL	Scales saturations for local grid well connections.
E100 only	COMPVEL	Sets well connection depth data in local grids.
E100 only	LGRFREE	Switches to local timestepping.
E100 only	LGRLOCK	Switches to global timestepping.
E100 only	LGROFF	Switches a local grid refinement off
E100 only	LGRON	Switches a global grid on.
E100 only	LSCRITS	Sets WARP linear solver convergence criteria for sub-grids.
E100 only	NEXTSTPL	Sets the next local timestep.
E100 only	TUNINGL	Sets simulator control parameters for all local grids.
E100 only	TUNINGS	Sets simulator control parameters for specific local grids.
	WELMOVEL	Moves a well into an LGR at a restart.
	WELOPENL	Shuts or reopens wells or well connections in an LGR.
	WELSPECL	Defines a well in an LGR.
E100 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.

Options keyword (ECLIPSE 100)

Switch	Description
32	controls timestep chops due to wells dying under THP control
37	replaces local grid solution states with those of the global host cell.
46	controls the interpolation of MINPV or MINPVV to LGRs.
51	controls the first local grid solution following a global timestep.
52	controls the use of corner point depths in VE runs.
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 9 x 3 x 3 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 three-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     Pgco
 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      Viscosity
 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
-- Ppub   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:

```

WELSPCS
-- Name Group I J Dbh Phase
  'INJ' 'G'   9 5 3725 'WAT' /
/

```

Specify connections of producer in local grid:

```

COMPDATA
-- Name LOCAL GRID I J K1-K2 Op/Sh SatTab ConFact Diam KH S D Dirn
  'PROD' 'LGR1'   3 2 2 2 'OPEN' 2*          0.375 3* 'X' /
  'PROD' 'LGR1'   4 2 2 2 'OPEN' 2*          0.375 3* 'X' /
  'PROD' 'LGR1'   5 2 2 2 'OPEN' 2*          0.375 3* 'X' /
  'PROD' 'LGR1'   6 2 2 2 'OPEN' 2*          0.375 3* 'X' /
  'PROD' 'LGR1'   7 2 2 2 'OPEN' 2*          0.375 3* 'X' /
/

```

Specify connections of injector in global grid:

```

COMPDATA
-- 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 /
/

```

Specify timesteps and reports:

```

TSTEP
  4*100 /
RPTSCHE
  'PRES' 'SWAT' 'SGAS' 'FIP' 'WELLS=2'
  'SUMMARY=2' 'CPU=2' 'WELSPCS' /

```

```
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
```

40

Miscible flood modeling

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. 111]).

The ECLIPSE 100 solvent model also provides a four-component extension of the black oil model to enable modeling of reservoir recovery mechanisms in which injected fluids are miscible with the hydrocarbons 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 "[Solvent model](#)" 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](#)" for details. This chapter refers only to the Todd-Longstaff model adopted in ECLIPSE 100.

Model formulation

The model we shall consider is a three-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.

In this miscible flood model, the injected gas is identified as a miscible agent (solvent) with the oil. The flow equations are a special case of the standard three-phase, three-component flow (water, oil and gas) where the gas-oil capillary pressure is set to zero because of the miscibility $P_{cog} = 0$.

A non-zero capillary pressure between the water and hydrocarbon phases, $P_{cnw}(S_w)$, which depends on the water phase saturation is however possible. The component pressures are then given by:

$$P_g = P_o \quad \text{Eq. 40.1}$$

$$P_w = P_o - P_{cnw}(S_w) \quad \text{Eq. 40.2}$$

Relative permeability model

The injected gas and reservoir oil are considered to be miscible components of the hydrocarbon (non-wetting) phase. The flow is two-phase in character and two-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 $S_n = S_o + S_g$ 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 gas relative permeabilities are then computed as saturation weighted fraction that is:

$$k_{rg} = \left(\frac{S_g - S_{gc}}{S_n - S_{gc} - S_{or}} \right) k_{rn}(S_n) \quad \text{Eq. 40.3}$$

$$k_{ro} = \left(\frac{S_o - S_{or}}{S_n - S_{gc} - S_{or}} \right) k_{rn}(S_n) \quad \text{Eq. 40.4}$$

It is possible to modify the straight line miscible relative permeabilities using the [MSFN](#) keyword. In this case the miscible gas and oil relative permeabilities are:

$$k_{rg} = M_{krsg} \left(\frac{S_g - S_{gc}}{S_n - S_{gc} - S_{or}} \right) k_{rn}(S_n) \quad \text{Eq. 40.5}$$

$$k_{ro} = M_{kro} \left(\frac{S_o - S_{or}}{S_n - S_{gc} - S_{or}} \right) k_{rn}(S_n) \quad \text{Eq. 40.6}$$

where the M_{krsg} and M_{kro} functions are supplied with the [MSFN](#) keyword. The saturations supplied in this keyword are normalized and honor the end-point supplied in the [SORWMIS](#) and [SGCWMIS](#) keywords. The critical oil-to-water saturation S_{or} is entered with the [SORWMIS](#) keyword and the critical oil to gas saturation S_{gc} is supplied with the [SGCWMIS](#) keyword.

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 oil in-place 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:

$$S_o^* = \text{MAX}(S_o - S_{or}, 0.0) \quad \text{Eq. 40.7}$$

where S_o is the oil saturation as above. 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.

Note: Equations 40.5, 40.6 and 40.7 can produce a discontinuity in the oil and gas 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 may experience non-linear convergence problems if this situation arises, but it issues a warning message during data checking.

Viscosity model

The effective oil and miscible gas viscosities follow the Todd-Longstaff model. They are given by:

$$\mu_{o \text{ eff}} = \mu_o^{1-\omega} \mu_m^\omega \quad \text{Eq. 40.8}$$

$$\mu_{g \text{ eff}} = \mu_g^{1-\omega} \mu_m^\omega \quad \text{Eq. 40.9}$$

The mixture viscosity μ_m is given by a 1/4-power fluid mixing rule

$$\frac{1}{\mu_m}^{1/4} = \frac{S'_g}{S'_n} \left(\frac{1}{\mu_g} \right)^{1/4} + \frac{S'_o}{S'_n} \left(\frac{1}{\mu_o} \right)^{1/4} \quad \text{Eq. 40.10}$$

which can be rewritten as

$$\mu_m = \frac{\mu_o \mu_g}{\left(\frac{S'_g}{S'_n} \cdot \mu_o^{1/4} + \frac{S'_o}{S'_n} \cdot \mu_g^{1/4} \right)^4} \quad \text{Eq. 40.11}$$

where the mobile saturations are defined by:

$$S'_o = S_o - S_{or}$$

$$S'_g = S_g - S_{gc}$$

$$S'_n = S'_o + S'_g$$

For a fully mixed state, that is $\omega = 1$, the effective viscosities are equal to the mixture viscosities, $\mu_{o \text{ eff}} = \mu_{g \text{ eff}} = \mu_m$. This case models a large dispersed oil-gas zone. The Todd-Longstaff model treats this case as a local unit mobility ratio displacement.

For a fully segregated case, that is $\omega = 0$, the effective viscosities are equal to the ‘pure’ phase viscosities, $\mu_{o \text{ eff}} = \mu_o$, $\mu_{g \text{ eff}} = \mu_g$. Such a case corresponds to a local high adverse mobility ratio displacement and models a negligibly thin oil-gas dispersed zone.

The mixing parameter approach allows the case of a partial mixing zone to be modeled by choosing an intermediate value of ω .

Density model

The treatment of effective oil and gas densities 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 as follows. First, the partially mixed or effective viscosities are calculated using equations 40.8, 40.9 and 40.11. The value of each effective component viscosity is then used to yield an effective saturation fraction to be used in oil and gas density calculations:

$$\left(\frac{S_o}{S_n}\right)_{oe} = \frac{\mu_o^{1/4}(\mu_{o\text{ eff}}^{1/4} - \mu_g^{1/4})}{\mu_o^{1/4} \cdot (\mu_{o\text{ eff}}^{1/4} - \mu_g^{1/4})} \quad \text{Eq. 40.12}$$

$$\left(\frac{S_o}{S_n}\right)_{ge} = \frac{\mu_o^{1/4}(\mu_{g\text{ eff}}^{1/4} - \mu_g^{1/4})}{\mu_g^{1/4} \cdot (\mu_{g\text{ eff}}^{1/4} - \mu_g^{1/4})} \quad \text{Eq. 40.13}$$

The effective oil and gas densities ($\rho_{o\text{ eff}}$, $\rho_{g\text{ eff}}$) are now computed from the effective saturations fractions in expressions in equations 40.12 and 40.13 and the pure component densities (ρ_o , ρ_g) using the following formulae:

$$\rho_{o\text{ eff}} = \rho_o \left(\frac{S_o}{S_n} \right)_{oe} + \rho_g \left[1 - \left(\frac{S_o}{S_n} \right)_{oe} \right] \quad \text{Eq. 40.14}$$

$$\rho_{g\text{ eff}} = \rho_o \left(\frac{S_o}{S_n} \right)_{ge} + \rho_g \left[1 - \left(\frac{S_o}{S_n} \right)_{ge} \right] \quad \text{Eq. 40.15}$$

For the fully mixed case, that is $\omega = 1$, the effective saturations are equal:

$$\left(\frac{S_o}{S_n}\right)_{oe} = \left(\frac{S_o}{S_n}\right)_{ge} = \frac{\mu_o^{1/4}(\mu_m^{1/4} - \mu_g^{1/4})}{\mu_m^{1/4} \cdot (\mu_o^{1/4} - \mu_g^{1/4})}$$

and the effective densities are therefore equal, $\rho_{o\text{ eff}} = \rho_{g\text{ eff}} = \rho_m$.

For the unmixed, fully segregated case, that is $\omega = 0$, the effective saturations simplify to

$$\left(\frac{S_o}{S_n}\right)_{oe} = 1, \left(\frac{S_o}{S_n}\right)_{ge} = 0$$

and the effective densities are equal to the “pure” phase densities $\rho_{o\text{ eff}} = \rho_o$, $\rho_{g\text{ eff}} = \rho_g$.

The model allows you to specify a different surface density for the solvent gas to that for the solution gas. In equations 40.14 and 40.15, the pure solvent gas density at reservoir conditions, ρ_g , is computed from the solvent gas surface density input with the **SDENSITY** keyword. The oil density at reservoir conditions, ρ_o , is calculated from the stock tank oil and solution gas surface densities.

Note that equations 40.12 and 40.13 are singular for unit mobility ratio displacements. In that case, where $\mu_o = \mu_g$, the effective densities are computed as follows. First a mixture density is defined as:

$$\rho_m = \rho_o \left(\frac{S_o}{S_n} \right) + \rho_g \left(\frac{S_g}{S_n} \right) \quad \text{Eq. 40.16}$$

Then:

$$\rho_{o\ eff} = (1-\omega)\rho_o + \omega\rho_m \quad \text{Eq. 40.17}$$

$$\rho_{g\ eff} = (1-\omega)\rho_g + \omega\rho_m \quad \text{Eq. 40.18}$$

where ω is the Todd and Longstaff parameter input with the [TLMIXPAR](#) keyword.

Todd and Longstaff 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 gas. 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 gas saturation increase behind the gas front. Todd-Longstaff accounted for the effects of viscous fingering in two dimensional 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):

$$\frac{1}{B_g} = \frac{1}{B_{gm}}M_p + \frac{1}{B_{gi}}(1-M_p) \quad \text{Eq. 40.19}$$

$$\frac{1}{B\mu_g} = \frac{1}{B\mu_{gm}}M_p + \frac{1}{B\mu_{gi}}(1-M_p) \quad \text{Eq. 40.20}$$

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:

$$B_{gm} = \frac{\rho_{g,s}}{\rho_{g\text{ eff}}} \quad \text{Eq. 40.21}$$

where

$\rho_{g,s}$ = surface density of solvent gas (**SDENSITY** keyword)

$\rho_{g\text{ eff}}$ = effective solvent density in equations 40.15 or 40.18.

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 .

Because the miscible capillary pressure is zero, the capillary pressure is treated as follows:

$$P_{cog} = P_{cog\text{ (immisc)}}(1-M_p) \quad \text{Eq. 40.22}$$

Control of numerical diffusion in miscible flow simulation

The relative permeability of the oil and solvent gas components in the Todd-Longstaff model are specified by equations 40.3 and 40.4 or 40.5 and 40.6. 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 gas. The two-point upstream projection technique is illustrated in the following figure.

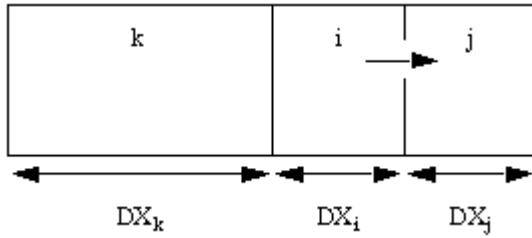


Figure 40.1. The two-point upstream projection technique

We require to compute the flow of component C between cells i and j , which we take to be aligned along the X-axis. Suppose that cell i is the upstream cell for the flow of component C between i and j . Let cell k be the second upstream cell to cell j and K_{ck} , K_{ci} and K_{cj} denote the component relative permeabilities in cells k , i and j respectively. DX_k , DX_i and DX_j are the X-direction cell sizes.

For single point upstream weighting, the flow rate of component C across the $i-j$ cell interface is $F_{ij,c}$, where:

$$F_{ij,c} = T_{ij} \frac{K_{ci}}{B_{ci} \mu_{ci}} DP_{ij,c} \quad \text{Eq. 40.23}$$

where

T_{ij} is the transmissibility between cells i and j

$DP_{ij,c}$ 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 40.23 by a linearly extrapolated relative permeability, K_{ci}^* , obtained from cells i and k ,

$$K_{ci}^* = \frac{2DX_i + DX_k}{DX_i + DX_k} K_{ci} - \frac{DX_i}{DX_i + DX_k} K_{ck} \quad \text{Eq. 40.24}$$

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:

$$\text{MIN}(K_{ci}, K_{cj}) < K_{ci}^* < \text{MAX}(K_{ci}, K_{cj}) \quad \text{Eq. 40.25}$$

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

- Cell k lies outside the reservoir mesh (that is the $k-i$ cell boundary is an edge of the reservoir).
- Cell k is inactive.
- The $k-i$ cell boundary contains a zero transmissibility multiplier.
- 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 [MISCTNUM](#) 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 MISCTNUM region data can be obtained using the mnemonic MISCTNUM 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 MISCTNUM 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 MISCTNUM 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 MISCTNUM region number of zero. For those cells that are allocated a positive MISCTNUM 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 $k_m(S_n)$ in equations [40.4](#) and [40.3](#)). 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](#). 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 gas-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 [40.1](#) and [40.2](#) with end points implied from equations [40.3](#) and [40.4](#) or [40.5](#) and [40.6](#).

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 gas-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 gas 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 [DRS DT](#) refers to any immiscible zones in the grid. Its default value in these regions will be infinity. In the miscible zones, the value of [DRS DT](#) 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 example, for cumulative totals, inter-region gas flows and gas-in-place. 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.

Summary of keywords

RUNSPEC

- [MISCIBLE](#) Initiates the mixing calculation.

PROPS

Keyword	Description
MSFN	Miscible gas/oil saturation functions.
PMISC	Pressure dependent miscibility tables.
RPTPROPS	Controls output from the PROPS section. Argument PMISC outputs the miscibility function and the pressure miscibility function.

Keyword	Description
SDENSITY	Solvent density at surface conditions.
SGCWMIS	Miscible critical gas saturation tables.
SORWMIS	Miscible residual oil saturation tables.
TLMIXPAR	Todd-Longstaff mixing parameter.

REGIONS

Keyword	Description
MISCTNUM	Miscibility region numbers.
RPTREGS	Controls output from the REGIONS section. Argument MISCTNUM outputs miscibility region numbers.

SOLUTION

Keyword	Description
RPTSOL	Controls output from the SOLUTION section. Use as for gas phase output.

SUMMARY

No specific keyword is required for the miscible option. The outputs are those available for the gas phase.

SCHEDULE

- [RPTSCHE](#) Controls output from the SCHEDULE section. Use as for gas phase output.

41

Mixed water relative permeability modeling

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

An alternative model, that takes into account the dissolution of gases, may be used to calculate the water relative permeability during a [CO2SOL](#), [H2SSOL](#) or [GASSOL](#) simulation. This optional mixing model is activated using the keyword [MIXWPERM](#).

In this case, the standard water relative permeability is corrected to take into account the hydrocarbon moles dissolved in the water phase, using the expression:

$$\tilde{K}_w = \frac{m_w K_w + \sum_i (m_i)_{Aq} [\alpha_i(P, T) K_o + \beta_i(P, T) K_g]}{m_w + \sum_k (m_k)_{Aq}}$$

in which

\tilde{K}_w denotes the new water relative permeability obtained following the mixing calculation,

K_w denotes the standard water relative permeability prior to mixing,

K_o and K_g denote, respectively, the oil and gas phase permeabilities,

m_w represents the water component molar density,

$(m_k)_{Aq}$ and $(m_i)_{Aq}$ denote, respectively, the aqueous molar density of components k and i defined by [CNAMES](#), with indices k and i each spanning all of the soluble hydrocarbon components.

Finally, denoting pressure and temperature (the latter in degrees Celsius) using P and T respectively, the oil and gas activity functions for component i are represented, respectively, by:

$$\alpha_i(P, T) = a_{i1}(P) + a_{i2}(P)T + a_{i3}(P)T^2$$

$$\beta_i(P, T) = b_{i1}(P) + b_{i2}(P)T + b_{i3}(P)T^2$$

The coefficients a_{i1} , a_{i2} and a_{i3} are interpolated, respectively, from columns 2, 3 and 4 of the [MIXACTO](#) keyword input table for component i .

Similarly, coefficients b_{i1} , b_{i2} and b_{i3} are interpolated, respectively, from columns 2, 3 and 4 of the [MIXACTG](#) keyword input table for component i .

Note that the sum $\alpha_i + \beta_i$ must not exceed unity for the given pressure and temperature, for any component i . This ensures that the new water relative permeability following mixing is bounded by unity. If $\alpha_i + \beta_i$ exceeds unity, the simulation will stop and the activity function coefficients must be reduced accordingly.

42

Multiphase flash

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

Multiphase components can be used in EoS runs for asphaltene prediction. These components can appear in the solid, liquid or gas phases.

Using multiphase

The [SOLID](#) keyword must be used if multiphase components are present.

Multiphase components

The simulator can be used to predict when asphaltene precipitation first occurs. At the end of each timestep a multiphase flash is performed to determine whether a stable asphaltene phase exists. The multiphase 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 [SOLID](#) keyword is used, and if the heaviest hydrocarbon is an aromatic (see the [HYDRO](#) keyword).

Multiphase keywords

RUNSPEC

Keyword	Description
MULTIPHA	Requests the multiphase flash which can be used to predict asphaltene precipitation.
SOLID	Requests Solids option.

PROPS

- [HYDRO](#) Define multiphase components.

SUMMARY section keywords

Refer to [Grid block solid quantities](#) in the *ECLIPSE Reference Manual*.

SCHEDULE section keywords

Solid properties that can be output with RPTRST, RPTSCHED and RPTSOL

Keyword	Description
BSOL	Solid reservoir molar densities.
DENS	Solid reservoir density.
SSOLID	Solid saturations.

43

Multi porosity

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

In ECLIPSE 300 it is possible to set up a multi porosity model as an extension to the dual porosity model. We then have the fracture system plus a given number of matrix pore systems.

There are two ways to supply the grid properties for the matrix sub cells:

- Using the keyword [NMATOPTS](#) the matrix sub cells are assigned pore volumes as described in the ["Discretized matrix model \(ECLIPSE 100\)"](#). The grid is geometrically partitioned as a “Russian Doll” of nested matrix sub-cells. This enables grid data to be supplied only for the dual porosity part of the grid and the nested matrix sub-grid cells data are generated automatically. New [SIGMAV](#) factors are automatically generated from the input [SIGMA/SIGMAV](#) factors from the primary matrix. Other data such as permeabilities can still be set by the user, although this is not recommended. By default, if these are not specified, the input primary matrix values are used to populate the sub-cell values.
- Input over the grid. The data is supplied for the entire simulation grid, similarly to the dual porosity model, but with grid data such as permeabilities also for the matrix-matrix connections.

In either case the number of pore systems is input using the [NMATRIX](#) keyword. This general setup gives a variety of possibilities for studying the transient behavior in the matrix. In particular the option can be used for.

- Shale gas reservoirs. See ["Shale gas"](#).
- Coal bed methane. See ["Adsorption models in ECLIPSE 300"](#)

The special case of triple porosity systems to represent vuggy carbonates uses the same underlying multi porosity model, but restricts the number of porosities to 3 (fracture, matrix, vugs with fracture and matrix connections) or 4 (with additional “isolated” vugs connected only to the matrix). This model is activated using the [TRPLPORS](#) keyword, which also specifies if isolated vugs are to be used. The data are supplied for the entire simulation grid, as with the multi porosity option (2) above. This model is described in more detail in the Triple Porosity section below. Triple porosity is also available in ECLIPSE 100 but the general multi-porosity feature is **not** available.

Using the multi porosity option

RUNSPEC section

The number of simulation cells in the Z-direction, as specified by the [DIMENS](#) keyword, needs to be a multiple of the number of pore systems. The number of pore systems is equal to the

number of matrix pore systems, set by the [NMATRIX](#) keyword, increased by one to account for the fracture system ([NPOROS =NMATRIX+1](#)).

The [DUALPORO](#) keyword is also required. If a dual permeability calculation is to be made where matrix to matrix connections are also used, then the [DUALPERM](#) keyword is also required. For multi-porosity, this will create a multi-permeability model - the sub-matrices will also be connected together. In most cases, this may not be the required solution, as the primary job of the sub-matrices is usually to model transients in the matrix-to-fracture flow.

Note: It is important to note that the ECLIPSE 300 model requires the full grid dimensions to be set by [DIMENS](#), unlike the discretized matrix model in ECLIPSE 100.

GRID section

In order to be compatible with standard dual porosity input, the first [NDIVIZ](#) / [NPOROS](#) layers are associated with the outermost matrix cells connecting to the fractures. The next layers are associated with the fracture cells. Thereafter the matrix sub cells follow as illustrated in the figure below.

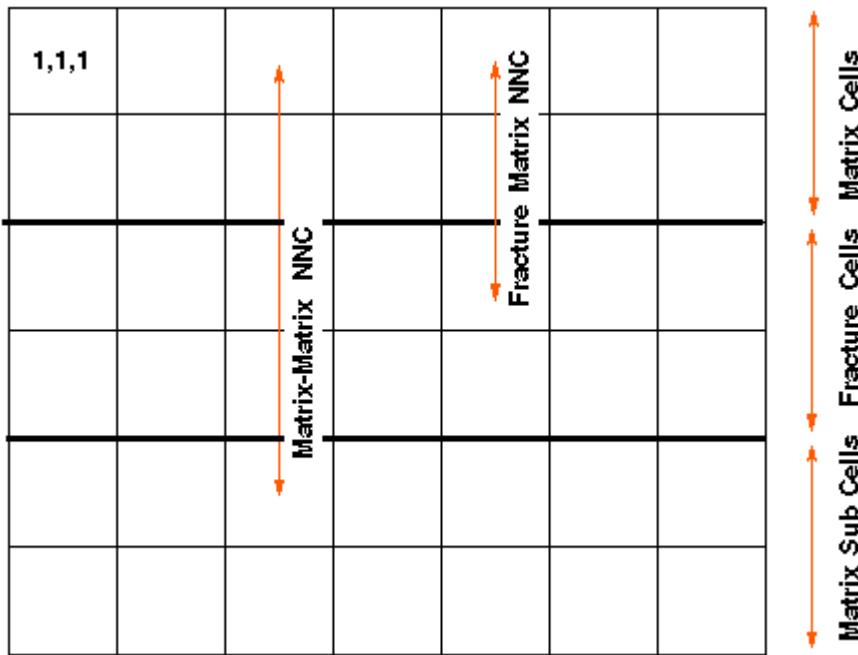


Figure 43.1. Multi porosity example having three pore systems.

The connections between the porosity systems are automatically generated. It is possible to supply the grid for the outermost primary matrix grid cells only, and using the keyword [DPGRID](#) the fracture and sub matrix grid are copied from the outermost matrix grid if they are not specified. Please note that [SIGMAV](#) values should be specified for all the matrix subgrid cells, including when [DPGRID](#) is used.

By default, properties such as porosities and permeabilities together with [SIGMAV](#) should be input for the sub-matrix grid-cells. In a multi-permeability model, it is possible to alter the transmissibility between the sub-grid cells using the [MULTX](#), [MULTY](#) and [MULTZ](#) multipliers.

This can, for example, be useful in order to disable communication between some of the matrix sub-grid cells.

Matrix-matrix transmissibility and diffusivity

Each pore system is connected as described above.

The transmissibilities between the sub-grid cells are then calculated similar to the matrix-fracture transmissibilities as described in "[Dual Porosity](#)". Note that the innermost matrix grid cell properties are used to set up the transmissibilities and diffusivities.

The permeability factor used between the subgrid porosities can be set, similarly to dual porosity, by the keyword `PERMMF`.

For two grid cells where inner matrix cell I connects to the outer grid cell J (which could be matrix or fracture), the transmissibility is given as

$$TR_{IJ} = CDARCY \cdot K_I \cdot VB_I \cdot \sigma_K$$

where

K_I is taken as the X-direction permeability of the matrix blocks (unless `PERMMF` is also specified),

VB_I is the matrix cell bulk volume (note that this is not the pore volume, having no porosity factor), and

σ_K is a factor of dimensionality $LENGTH^{-2}$, to account for the matrix-matrix interface area per unit volume.

Note that the cell index for σ_K has been written as K rather than I . When I and J are both matrix cells, then K is the *outer* matrix cell J . However, when outer cell J is the fracture, then K is the inner cell: that is, K is the outermost matrix. The result of this is to have all the sigma values in contiguous locations in the first $N - 1$ porosity locations in the `SIGMAV` keyword input. The innermost `SIGMAV` porosity (porosity $N = \text{NMATRIX} + 1$) is not used.

The diffusivity is similarly given as

$$DIFF_{IJ} = DIFFMMF_I \cdot VOL_I \cdot \sigma_K$$

where

$DIFFMMF_I$ is the diffusivity multiplier, and

VOL_I is the matrix cell pore volume or coal volume (coal bed methane option)

Matrix discretization (`NMATOPTS`)

When the `NMATOPTS` option is invoked, the matrix sub-grid cell properties are copied from the outer primary matrix grid-cells. Only the bulk volume, porosities and/or rock fraction of the primary matrix grid cells are used to generate the matrix sub-grid cells' new volumes. The pore volume and transmissibility/diffusivity follow what is described in the "[Discretized matrix model \(ECLIPSE 100\)](#)". However ECLIPSE 300 does not generate the transmissibilities directly, but generates new sigma (`SIGMAV`) values following the geometric partitioning of the matrix block, and updates the block volumes, pore volumes and rock volumes of the matrix grid cells. In addition, the ECLIPSE 300 model can be used with `UNIFORM`

partitioning. The pore volumes are evenly distributed between the matrix sub-grid cells and the transmissibilities are calculated following a 1D computation where the distance DX of the matrix cells is assumed to be partitioned as $DX / NMATRIX$.

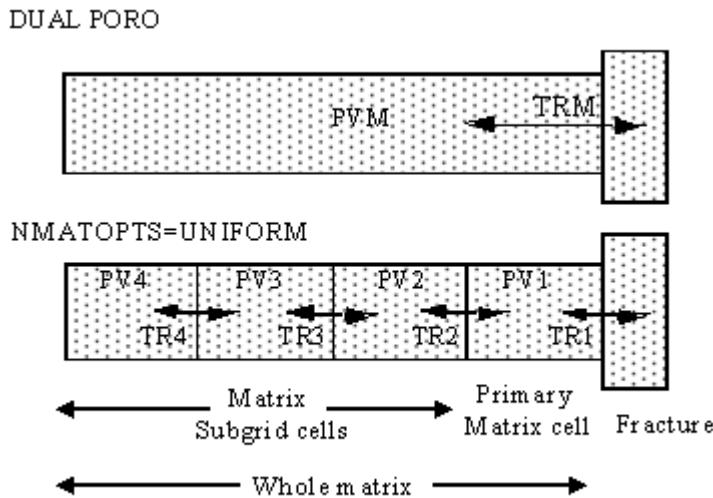


Figure 43.2. Structure of the matrix discretization and nomenclature

In the figure above, the uniform partitioning is used with $NMATRIX=4$. Each block is partitioned uniformly and sigma values generated for the connections. The pore volume of the primary matrix cell is divided into $PV1 = PV2 = PV3 = PV4 = \frac{PVM}{4}$. The assumptions for the UNIFORM partitioning is the same as for the LINEAR except that each matrix block is uniformly partitioned. The available methods with NMATOPTS are:

NMATOPS	Assumption for initial matrix-fracture transmissibility/diffusivity	Partitioning method
UNIFORM	1D flow over the distance d and through area A	Partitions the distance d uniformly
LINEAR	1D flow over the distance d and through area A	Logarithmic growth factor of the distances
RADIAL	2D flow, four times flow, over a distance d and through area A	Logarithmic growth factor of the distances
SPHERICAL	3D flow, six times flow, over a distance d and through area A	Logarithmic growth factor of the distances
VERTICAL	For gravity drainage. A depth assigned to the sub-matrix grid cells.	Partitioned uniformly (vertically)

Table 43.1: Available methods with NMATOPTS

Based on the method, a new internal grid is generated, representing the initial matrix-fracture connection. The distance "d" is partitioned resulting in new bulk volumes, pore volumes and rock volumes. Based on the initial sigma factor assigned to the primary matrix grid, new sigma factors are calculated and populated in the matrix grid.

As an example consider the UNIFORM partitioning above and then how to calculate new sigma values.

Assume that we have a unit m^3 cube, $BV=1m^3$. The distance $d = 1m$ and the area $A=1 m^2$. Partitioning d into $1/4m$ gives the flow over $1/4m$ for each connection through an area A . Since “ d ” and “ A ” are already embedded in the initial sigma value, we multiply and get $(\sigma * 4)$ for the connection. This needs to be scaled to apply to the new block volumes of the cells and so we get the factor $(\sigma * 4) * BV / (BV/4)$ for the new sigmas, that is $(\sigma * 16)$.

For more information about the discretization, see the "[Discretized matrix model \(ECLIPSE 100\)](#)".

Coal in multi porosity

When using the [NMATOPTS](#) matrix discretization feature with coal or shale gas, the definition of coal volumes needs to be considered. See "[Coal bed methane model](#)" and for the partitioning into the matrix subgrid cells see "[Shale gas](#)". Only the input of the primary matrix sub-grid cells is needed when generating the sub-cells' coal and pore volumes.

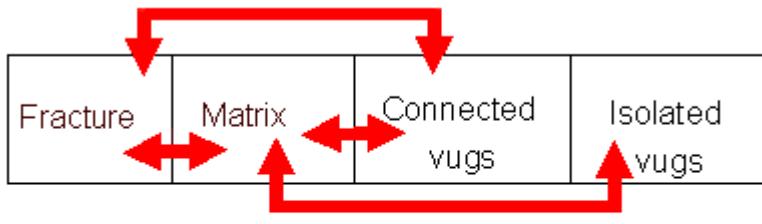
In general it is recommended to have all sub-grid cells defined as coal and/or use the instant sorption method when doing a matrix discretization. Having nonhomogeneous properties means that the number of matrix cells and the placement of pore, coal volume becomes important when considering the response from the matrix. For specialized models, the general multi-porosity option can be used with user-defined sigma values within the sub-matrix grid.

Triple porosity

When the [TRPLPORS](#) keyword is used, only 3 or 4 porosities are allowed. The matrix and fracture porosities are specified as porosities 1 and 2 as usual. Porosity 3 is used to represent vugs (larger voids in the rock which are not part of the fracture system) which are connected to the matrix in the same way as the fracture is connected to the matrix, and additionally to the fracture through micro fracture connections. If used, porosity 4 represents additional vugs with connection to the matrix only (isolated vugs).

Unlike the standard multi porosity connections, a porosity in triple porosity may be connected to more than 2 other porosities, as the matrix can be connected to the fracture and two different vug porosities (isolated and connected). There is also a cyclic path in the porosity system, formed by the matrix, fracture and connected vug porosities. These relationships are illustrated in the figure below. Note that the logical order, with the fracture as the outer porosity, is shown, but in the grid description the fracture is still the second porosity.

As for the multi porosity model, the [DUALPORS](#) and, where needed, the [DUALPERM](#), keywords are required. The input allows for the possibility of a triple permeability model, where the vug porosities form a connected continuum. However, in most if not all cases, only a dual permeability model will be required for triple porosity. To make the vug cells connected only to the local matrix and, for connected vugs, to the local fracture, you should zero the permeability or add permeability multipliers of size zero to the appropriate permeability for the other vug-matrix, vug-fracture or vug-vug connection. The convention for assigning cell permeabilities to triple porosity connections is noted in the Transmissibilities section below.



Triple porosity for fractured matrix with vugs
(including cyclic flow paths)

Figure 43.3. Triple porosity connectivity

Note: `NMATRIX` and `NMATOPTS` should not be used with `TRPLPORO`

Transmissibilities

In the current version of the input for this model, transmissibilities between vugs porosities and the matrix and fracture porosities are still described by Kazemi-type multipliers using the `SIGMA` and `SIGMAV` keywords. However, the `LTOSIGMA` keyword cannot be used and the interpretation of the `SIGMA` multiplier as an inverse squared matrix length is not applicable. In particular the connection between fracture and connected vugs is through micro fractures, and the connection between the matrix and vugs is through the typical inter-vug distance in the matrix and the size of the vugs. The `SIGMA` values used for all connections to vugs, whether through micro fractures from the fracture, or through matrix rock from the matrix, should honor this difference from standard matrix-fracture connection values.

An exact Kazemi-type equivalent to the `LTOSIGMA` input cannot be provided. However, for matrix-vug sigmas, a useful rule of thumb is that the product $V\sigma$ can be replaced by $\beta A/L$ where A is the cross-sectional area of the vug, L is half the distance between two vugs, and β is the fraction of total volume occupied by the vugs. There is no simple formula to represent the connected-vug/fracture sigma, and you should consider general transmissibility theory for the micro fractures characterizing this connection.

Input values for `SIGMAV` are applied to porosities in the following way:

- matrix/fracture connection sigma values are input through the matrix cells;
- fracture/connected-vug sigma values are input through the fracture cells;
- matrix/connected-vug sigma values are input through the connected-vug cells (third porosity);
- matrix/isolated-vug sigma values are input through the isolated-vug cells (fourth porosity, if present).

PROPS, REGION and SOLUTION sections

This follows the input as for a standard single or dual porosity model. However, simulation grid cell data must be specified on a cell-by-cell basis, regardless of the use of `NMATOPTS`.

An exception is the regional data such as `SATNUM`, `EQLNUM` and `FIPNUM`. If no value is supplied these are copied from the primary matrix grid cell.

Output

Summary output follows the usual syntax, where block solutions can be obtained by specifying a cell index corresponding to the matrix sub-cell.

On the restart file the sub-cell matrix solutions are only output if a flexible restart file is requested.

Output to the PRT file using the [RPTGRID](#), [RPTSOL](#) and [RPTSCHED](#) keywords will output the requested quantity for all cells.

Examples

The following data sets come with the ECLIPSE installation:

File	Description
MPORO.DATA	Multi porosity example, gas-water case
SHALEGAS1.DATA	Shale gas

Table 43.2: Multi porosity example files

Limitations

The multi porosity option cannot currently be used with:

- Coarsening
- Block to block connections ([BTOBALFA](#) or [BTOBALFV](#))
- The triple porosity option cannot currently be used with parallel in ECLIPSE 100.
- Local grid refinements when using the discretized matrix model.

44

Multisegment wells

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

The multisegment 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 multilateral 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. 55].

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 one-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 precalculated VFP tables, which can potentially offer greater accuracy and provide the ability to model chokes.

Benefits

The multisegment 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 multilateral topology. Data input has been designed for ease of use with multilateral 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 multiphase flow. Use of the drift flux model or precalculated 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 multisegment 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 downhole 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 multilateral 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 multisegment well model

Segment structure

A multisegment 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 wellbore. A multilateral well has a series of segments along its main stem, and each lateral branch consists of a series of one or more segments that connects at one end to a segment on the main stem (figure 44.1). It is possible, if required, for lateral branches to have sub-branches; this may be useful when modeling certain inflow control devices as part of the network of segments.

The segment network for each well may thus have any number of ‘generations’ of branches and sub-branches, but it must conform to gathering-tree topology. Segments within the gathering tree topology can be connected in order to form loops - this facility is described in “[Looped flow paths](#)”.

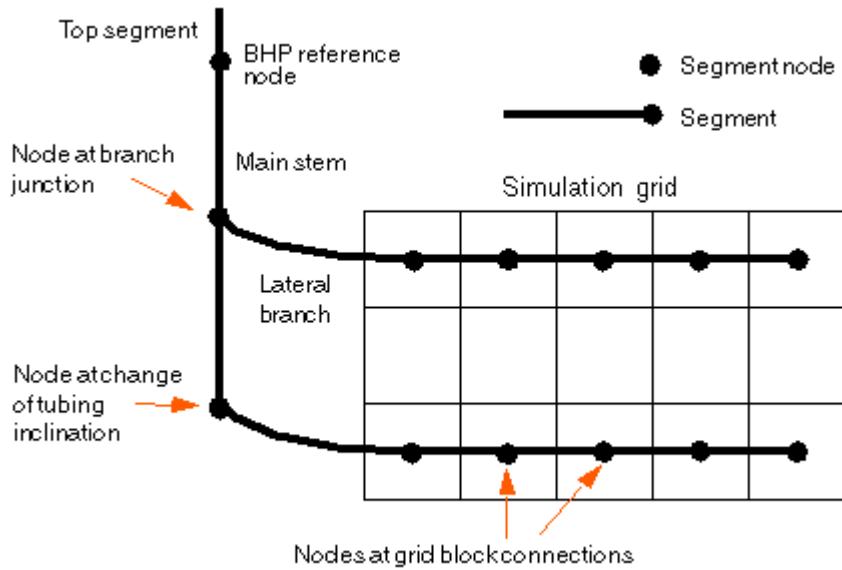


Figure 44.1. A multilateral, multisegment well

Each segment consists of a **node** and a **flow path** to its parent segment's node. (We use the term ‘flow path’ here rather than ‘branch’, as the latter term is reserved for lateral branches in multilateral wells, each of which may contain several segments.) A segment’s node is positioned at the end that is furthest away from the wellhead (figure 44.2). Each node lies at a specified depth, and has a nodal pressure which is determined by the well model calculation. Each segment also has a specified length, diameter, roughness, area and volume. The volume is used for wellbore storage calculations, while the other attributes are properties of its flow path and are used in the friction and acceleration pressure loss calculations. Also associated with each segment’s flow path are the flow rates of oil, water and gas, which are determined by the well model calculation.

Note: The top segment is a special case. It does not have a parent segment so no pressure drop is calculated along this segment’s flow path. Hence there is no need to specify diameter, roughness and area for this segment. The equation for the pressure drop along the flow path is replaced by a boundary condition appropriate to the well’s mode of control. See “[The top segment](#)” for details.

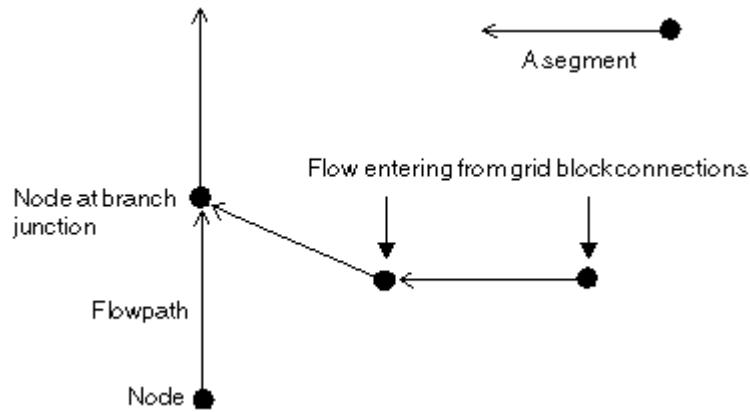


Figure 44.2. Well segments

A segment node must be positioned at each branch junction. Flow from the formation through grid-block-to-well connections also enters the well at segment nodes. A segment (or to be more precise, its node) can accept flow from any number of grid block connections (figure 44.3). If you wish to have a separate segment node for each connection (for greatest accuracy) then it is best to position the nodes to lie at the center of their corresponding grid block connections. But it is also possible to reduce the number of segments by allocating two or more grid block connections to each segment. In this case it best to position each segment node to lie at the center of the perforated interval that is allocated to the segment. However, it is not possible to allocate a single grid block connection to more than one segment.

ECLIPSE allocates each grid block connection to the segment whose node lies nearest to it. If necessary, you can override this by explicitly specifying the segment to which the connection should be allocated in the [COMPSEGS](#) (or [COMPSEGL](#)) keyword.

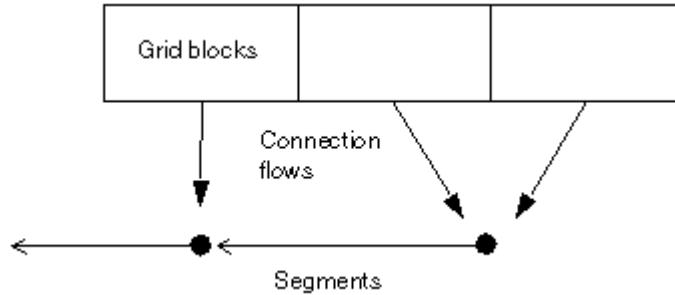


Figure 44.3. Allocating connection flows to segments

It is also possible to position segment nodes at intermediate points along the wellbore, 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

$$q_{pj} = T_{wj} M_{pj} (P_j + H_{cj} - P_n - H_{nc}) \quad \text{Eq. 44.1}$$

where

q_{pj} is the volumetric flow rate of phase p in connection j at stock tank conditions in ECLIPSE 100 or reservoir conditions in ECLIPSE 300. The flow is taken as positive from the formation into the well, and negative from the well into the formation.

T_{wj} is the connection transmissibility factor.

M_{pj} is the phase mobility at the connection. If the node is injecting fluid into the formation, the ratio of the phase mobilities reflects the mixture of phases in the segment

P_j is the pressure in the grid block containing the connection.

H_{cj} is the hydrostatic pressure head between the connection's depth and the center depth of the grid block.

P_n is the pressure at the associated segment's node n.

H_{nc} is the hydrostatic pressure head between the segment node n and the connection's depth.

Note the two components of the hydrostatic head, H_{cj} and H_{nc} . In multisegment wells the connection depths are not forced to be equal to the grid block center depths as they are in wells (Figure 44.4). They may be set independently in keyword `COMPSEGS` (or `COMPSEGL`), or defaulted to allow ECLIPSE to calculate them from the segment depths and their position within the segment. The hydrostatic head H_{nc} between a connection and its corresponding segment node is calculated implicitly in the usual way from the density of the mixture in the segment. The hydrostatic head H_{cj} between the connection and the grid block center is calculated from an average of the fluid densities in the grid block.

ECLIPSE 100

The average is weighted according to the fluid relative permeabilities, to give a smooth variation as the fluid in the block is displaced by another phase. This average density is calculated explicitly at the beginning of each timestep. If the use of explicit quantities causes oscillations from one timestep to the next, they can be damped by setting item 63 of the `OPTIONS` keyword, which averages the head term with its value at the previous timestep.

ECLIPSE 300

The average is weighted according to the implicit fluid saturations, to give a smooth variation as the fluid in the block is displaced by another phase.

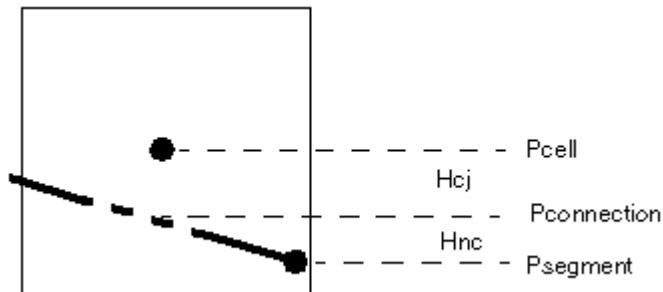


Figure 44.4. Hydrostatic head components

The pressure loss calculation

The following options are available for calculating the pressure drop

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](#)"). The acceleration pressure drop across the segment is the difference between the velocity heads of the inflowing and outflowing mixture (see "[Acceleration pressure loss](#)").

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 countercurrent 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.

Another built-in model calculates the pressure drop across an inflow control device that is placed around a section of the tubing and diverts the inflowing fluid from the adjacent part of the formation through a sand screen and then into a spiral constriction before it enters the tubing. See keyword [WSEGSICD](#) for further details.

There is also a built-in model that corresponds to an ‘autonomous’ inflow control device. This works in a similar way to the ‘spiral’ inflow control device. See keyword [WSEGAI](#) 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 multiphase 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 countercurrent 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 wellbore 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 Multisegment Well Model may simply not converge.

A situation where it may be useful to model the whole wellbore 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 wellbore 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 kickoff process for example. Water import may be used to take account of the water used to power a downhole hydraulic pump if this is injected into the wellbore.

ECLIPSE 300

There is also an option to remove fluid from a segment into an external sink, at a rate that is a function of the segment's pressure. This facility is intended for use with the ECLIPSE 300 Thermal option to model recirculating wells.

Secondary wells

ECLIPSE 300

As an extension to the facilities afforded by the keyword [WSEGEXSS](#), further controls may be exerted over well segments by defining one or more segments to be secondary wells. This is accomplished using the keyword [WSEGWELL](#) which associates a specified well segment with a secondary well name. Once this association has been established, the secondary well keywords [SCONPROD](#), [SCONINJE](#) and [STEST](#) may be employed to control the production from and injection into the segments. The keywords [WSEGEXSS](#) and [WSEGWELL](#) are mutually exclusive for a given well segment. With the exception of the top segment, any segment in the well topology may be associated with a secondary well name.

There are a number of possible applications for this facility. For example, a dual-tubing well may be modeled in terms of a multisegment well where the conventional (primary) well can be used to represent one tubing and a well segment can be used to represent the other tubing by defining this as a secondary well. There are, however, a number of factors to consider when constructing this form of model. For example, if the primary well shuts, the secondary well will also shut. Moreover, the multisegment well topology can become complex if both secondary producers and injectors are modeled via secondary wells. In addition, the controls available with a secondary well are more limited than those available with a conventional well.

The fluid production and injection rates for secondary wells may be specified in terms of a target rate plus one or more limits. The limits include the minimum operating pressure for secondary producers (item 10 of the [SCONPROD](#) keyword) and the maximum operating pressure for secondary injectors (item 8 of the [SCONINJE](#) keyword). In some cases, depending upon the multisegment well topology, it may be conveniently an additional pressure constraint to an alternative segment to that nominated as the secondary well. For example, for a producer, it may be more appropriate to specify a minimum operating pressure for the secondary tubing segment whose depth is similar to the reference depth for the primary well BHP than for the secondary well segment itself with a surface outlet.

The alternative pressure constraint is optional and is specified in terms of an alternative segment number and associated pressure limit (items 12 and 13 respectively of the [SCONPROD](#) keyword for producers and items 13 and 14 respectively of the [SCONINJE](#) keyword for injectors). For secondary producers, the calculation of the molar rate required to provide the specified target fluid rate utilizes the minimum operating pressure of the secondary well segment (item 10 of the [SCONPROD](#) keyword). If the pressure in the secondary segment reduces to within a few percent of this minimum operating pressure, in some circumstances fluctuations in the secondary production rate can occur. In such cases, items 12 and 13 of the [SCONPROD](#) keyword may be used to define a cutoff pressure for the secondary well segment by specifying item 12 as the secondary well segment number and item 13 as the cutoff pressure. In this case a suggested value for item 13 is 10% higher than the value for item 10.

Looped flow paths

The gathering tree topology required by multisegment wells normally prevents the formation of looped flow paths within the well itself. However, it is possible to add looped flow paths as follows:

- The basic topology of the well must be specified in the normal manner using the [WELSEGS](#) keyword. One or more pairs of nodes of different segments must be made physically coincident. (In practice, this just means that the segment nodes should be specified as being at the same true vertical depth in the [WELSEGS](#) keyword.) Note that this in itself is not sufficient to form a looped flow path and these node pairs will still act as though they are unconnected in any way.
- Each such pair of segment nodes must be “linked” using the [WSEGLINK](#) keyword. This is the key step that instructs each pair of segment nodes to communicate and form looped flow paths. At least one of these segments must be at the end of a branch.

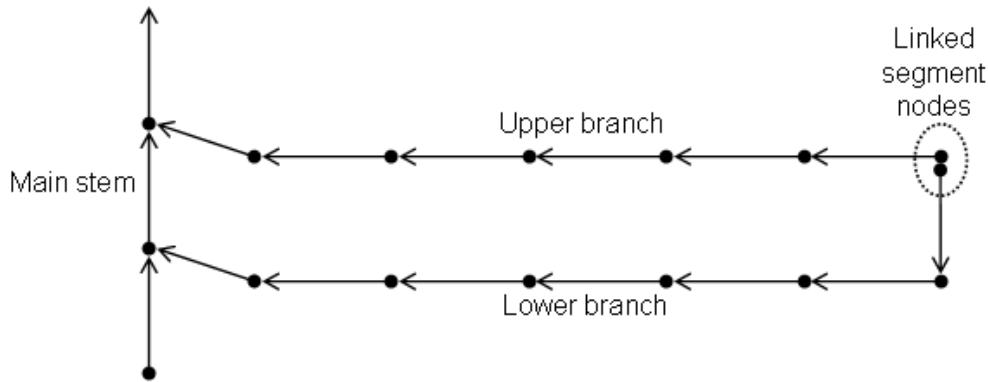


Figure 44.5. Simple looped multisegment well

A pair of linked segment nodes is shown in the previous figure. This link forms a looped flow path between the upper and lower branch in this multisegment well. Flow from the lower branch can pass into the main stem of the well directly or via the upper branch, and flow from the upper branch can pass into the main stem directly or via the lower branch. The pressure variations in the upper and lower branches will determine the direction of the flow.

The pressure drop equations for the segments whose nodes are linked are unaffected, but the material balance equations for one of the segments are replaced instead by equations that enforce equality of the solution (pressure, flowing fraction of water and flowing fraction of gas) at the linked segment nodes.

Handling of linked segment properties

In the [WSEGLINK](#) keyword, the first segment of each pair specified is known as the **primary linked segment**, while the second segment of the pair is known as the **secondary linked segment**. If the first segment of the pair is at the end of a branch, but the second is not, then the first segment will be treated as the secondary linked segment instead. It is the secondary linked segment that has its material balance equations replaced by the solution equality equations.

- If connection flows or external source /sink flows are specified for both the linked segments (using either the [COMPSEGS](#) or [COMPSEGL](#) keywords), then these flows will all be assigned to the primary linked segment (as it is the only one of the pair that has the material balance equations which incorporate these terms).
- Any volume associated with the secondary linked segment will be allocated as additional volume in the primary linked segment.
- part from the replacement of the material balance equations by linked segment node solution equality equations in the secondary linked segment, each of these segments is exactly the same as any other segment used to define a multisegment well. Hence the pressure drop equation in either of the linked segments can be modified in any of the ways described in "[The pressure loss calculation](#)".

Example application: Modeling an unpacked annulus

As an example of an application of looped flow paths within multisegment wells, consider the modeling of flow in an unpacked annulus.

One branch of the well can be used to model the tubing and may contain multiple segments, each in direct communication with its immediate neighbors. This tubing branch is shown as a dashed line in figure 44.6.

Another branch of the well can be used to model the annulus; the segments on this branch are also in direct communication with their immediate neighbors. This annulus branch is shown as a dotted line in figure 44.6. Flow only enters the well via connections at annulus segments.

Flow can move freely along the annulus and pass into the tubing via control valves at intervals along the well. Without the ability to model looped flow paths, the flow from a certain annulus segment would have been constrained so that it would have to flow into the tubing using a specific valve. However, due to the presence of looped flow paths, flow in an annulus segment is free to pass through any of the valves into the tubing.

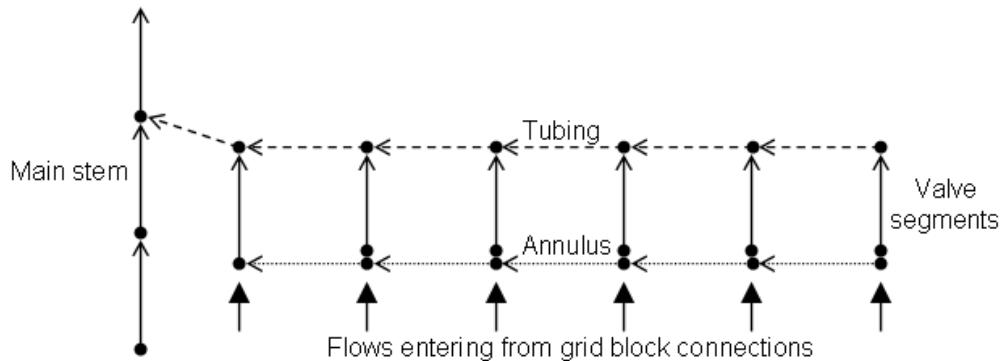


Figure 44.6. Schematic of a multisegment well with an unpacked annulus

Impact of looped flow paths on the well linear equations

From [Ref. 55], in the absence of such linked segment nodes, the linear system arising from the well equations can be solved very efficiently due to the special structure of the well matrix. However, as soon as loops are added, this special structure is lost and an alternative method for solving the well system must be used. ECLIPSE uses an iterative method called GMRES [Ref. 93] to solve the well linear system arising from the presence of loops, with an incomplete LU factorization pre-conditioner used to accelerate the convergence. If required, the parameters that control the GMRES solver and the pre-conditioner can be modified using the `WSEGSOLV` keyword.

ECLIPSE 100

If the pre-conditioned GMRES solver fails to converge or encounters serious problems then the system will be solved again using a pivoted Gaussian elimination, hereafter referred to as a “direct” solution. This is an expensive method both in memory and CPU cycles, but is likely to succeed should GMRES fail. Warning messages will be issued that a direct solution had been used; the user should alter to eliminate these messages if possible. The Gaussian elimination `WSEGSOLV` can be suppressed by setting item 182 in the `OPTIONS` keyword; in this case the un converged GMRES result will be accepted.

Restrictions

- One of the segments linked using the `WSEGLINK` keyword must be at the end of a branch.
- It is not permissible for the secondary linked segment to have zero effective absolute roughness. That is, item 8 in the `WELSEGS` record defining the secondary linked segment properties must be non-zero.
- A segment cannot be both a secondary link segment and a labyrinth inflow control device with non-positive configuration number (specified using the `WSEGLABY` keyword). If the need for this particular situation arises, the specification of the primary and secondary link segments can be reversed in the `WSEGLINK` keyword.

- The top segment of the well cannot be part of a loop so it cannot appear in a row of the `WSEGLINK` keyword.

Pressure drops from VFP tables

VFP tables offer the most flexible, and potentially the most accurate, means of determining the pressure drop across each segment. Interpolating the pressure drop from a table is also considerably faster than calculating it from a multiphase 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 of 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). 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. 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](#)”), 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 (item 8 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. However, multiphase 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 multiphase 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

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 multisegment wells ought to have the following attributes:

Simplicity

The calculation has to be done many times in each segment at every timestep. 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 multisegment 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

$$v_g = C_0 j + v_d \quad \text{Eq. 44.2}$$

where

v_g is the flow velocity of the gas phase, averaged across the pipe area

C_0 is the profile parameter (or distribution coefficient) resulting from the velocity and gas concentration profiles

j is the volumetric flux of the mixture

v_d is the drift velocity of the gas.

The volumetric flux of the mixture is same as the average mixture velocity, and is the sum of the gas and liquid superficial velocities

$$j = v_{sg} + v_{sl} = \alpha_g v_g + (1-\alpha_g)v_l \quad \text{Eq. 44.3}$$

where α_g is the gas volume fraction, averaged across the pipe area.

The average liquid flow velocity is thus

$$v_l = \frac{1-\alpha_g}{1-\alpha_g} C_0 j - \frac{\alpha_g}{1-\alpha_g} v_d \quad \text{Eq. 44.4}$$

Several of the one-dimensional slip correlations take the general form of equation 44.2. The task here is to develop relationships for C_0 and v_d that are simple, continuous and differentiable, and which extend into the countercurrent flow regime. The derivation of these relationships is described in [Ref. 97]. The relationships contain some adjustable parameters that can be tuned to fit observations. In [Ref. 97] the parameter values are optimized to fit a series of experiments performed in 15 cm (6 in) diameter tubing. In [Ref. 96] the optimization process was improved and the model was extended to allow for the observation that the presence of free gas can disrupt the slip between oil and water, depending on the pipe inclination. The default values of the parameters in the model in ECLIPSE are those of the original implementation

described in the subsequent sections of this chapter, but you may request ECLIPSE to use the optimized parameters from [Ref. 97] or [Ref. 96] 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. 135] have shown that the value of C_0 can range from 1.0 to 1.5. 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. 3], and Hasan and Kabir [Ref. 49]. 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

$$C_0 = \frac{A}{[1 + (A-1)\beta^*]^2} \quad \text{Eq. 44.5}$$

The coefficient A is defaulted to 1.2, but can be set by you to any value not less than 1.0. The purpose of the term β^* is to make C_0 reduce to 1.0 at high values of α_g and j . It is evaluated from the formula

$$\beta^* = \frac{(\beta-B)}{(1-B)} \quad \text{subject to } 0 \leq \beta^* \leq 1 \quad \text{Eq. 44.6}$$

β is a parameter that approaches 1.0 as α_g approaches 1.0, and also as the mixture velocity approaches a particular value. We choose the velocity of the onset of the annular flow regime to be the velocity at which profile slip vanishes. The transition to annular flow occurs when the gas superficial velocity v_{sg} reaches the 'flooding' value that is sufficient to prevent the liquid from falling back against the gas flow, v_{sgf} . An expression for v_{sgf} is given in the next section, with equation 44.12. We thus choose the following expression for β

$$\beta = \max\left(\alpha_g, F_v \frac{\alpha_g | j |}{v_{sgf}}\right) \quad \text{Eq. 44.7}$$

The parameters B and F_v are user-definable constants. B represents the value of the gas volume fraction or the flooding velocity fraction at which the profile parameter C_0 will start to reduce from the value A . It defaults to 0.3. F_v is a multiplier on the flooding velocity fraction. It defaults to 1.0, but the profile flattening can be made more or less sensitive to the velocity by respectively increasing or decreasing its value.

The default values for the parameters A and B can be changed by requesting either of the optimized parameter sets from [Ref. 97] or [Ref. 96] with the keyword `WSEGDFMD`. Note that in the SHI-04 model there is no profile slip, $A = 1.0$, making the value of B irrelevant.

There is a limit on the maximum value that the parameter B may have. Intuitively we would expect the gas superficial velocity $v_{sg} = \alpha_g v_g$ to increase with both α_g and j . Thus we require

$$\frac{\partial}{\partial \alpha_g}(\alpha_g C_0) > 0 \text{ and } \frac{\partial}{\partial j}(jC_0) > 0 \quad \text{Eq. 44.8}$$

Both these criteria will be satisfied if

$$B < \frac{2-A}{A} \quad \text{Eq. 44.9}$$

With the default value of $A = 1.2$ the constraint is $B < 0.6667$.

The drift velocity

The derivation of the expression for the drift velocity v_d is described in [Ref. 97]. The expression is derived by combining data on the limits of countercurrent 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

$$v_g(v_l = 0) = \frac{v_d}{1-\alpha_g C_0} \quad \text{Eq. 44.10}$$

As α_g approaches zero, the gas velocity should approach the rise velocity of a single bubble, for which Harmathy recommended the value of $1.53v_c$ where v_c is the characteristic velocity

$$v_c = \left[\frac{\sigma_{gl} g (\rho_l - \rho_g)}{\rho_l^2} \right]^{1/4} \quad \text{Eq. 44.11}$$

where

σ_{gl} is the gas-liquid interfacial tension,

g is the acceleration due to gravity,

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 Makkennery obtained the relation

$$v_g(v_l = 0) = K_u \left(\frac{\rho_l}{\rho_g} \right)^{1/2} v_c \quad \text{Eq. 44.12}$$

where K_u is the ‘critical Kutateladze number’, which is related to the dimensionless pipe diameter

$$D^* = \left[\frac{g(\rho_l - \rho_g)}{\sigma_{gl}} \right]^{1/2} D \quad \text{Eq. 44.13}$$

by the function in table 44.1.

D^*	K_u
≤ 2	0

D^*	K_u
4	1.0
10	2.1
14	2.5
20	2.8
28	3.0
≥ 50	3.2

Table 44.1: The relationship between the dimensionless pipe diameter and the critical Kutateladze number

To interpolate between these two extremes we make use of the flooding curve described by Wallis to define the limit of the countercurrent 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

$$v_d = \frac{(1-a_g C_0) C_0 K(a_g) v_c}{\sqrt{\frac{\rho_g}{\rho_l} a_g C_0 + 1-a_g C_0}} \quad \text{Eq. 44.14}$$

where

$$K(a_g) = 1.53 / C_0 \quad \text{when } a_g \leq a_1$$

$$K(a_g) = K_u \quad \text{when } a_g \geq a_2$$

and a linear interpolation of these values when $a_1 < a_g < a_2$. The default values of a_1 and a_2 are set to 0.2 and 0.4 respectively, to match data used by Zuber and Findlay to demonstrate the transition from the bubble flow regime. Their default values can be changed by requesting either of the optimized parameter sets from [Ref. 97] or [Ref. 96] with the keyword [WSEGDFMD](#).

The formulation described above is intended to model the behavior of gas-liquid mixtures in a vertical pipe. To scale the drift velocity for inclined flow it is multiplied by an inclination factor

$$v_d(\theta) = m(\theta)v_d|_{\theta=0} \quad \text{Eq. 44.15}$$

where θ is the angle of inclination from the vertical and $v_d|_{\theta=0}$ is given by equation 44.14. By default ECLIPSE uses the inclination factor given by Hasan and Kabir [Ref. 48]

$$m(\theta) = (\cos \theta)^{0.5}(1 + \sin \theta)^2 \quad \text{Eq. 44.16}$$

Hasan and Kabir present the formula as valid for oil/water flow for $\theta < 70^\circ$, but we shall apply this for all inclination angles. For horizontal segments the drift velocity becomes zero.

Item 2 of keyword [WSEGDFMD](#) allows you to override this default formula for the inclination factor. You may input your own table of m vs. θ with keyword [WSEGDFIN](#). Note that if you supply your own table it is not written to the RESTART file, so it must be present in all restarted data sets. Alternatively, you can ask ECLIPSE to use a built-in table that reproduces the SHI-03 model described in [Ref. 97]. This is the recommended option if you select the SHI-03 model's parameter set in item 1 of keyword [WSEGDFMD](#). Note that the built-in table has $m > 1$ for vertical pipes, since it was necessary to rescale v_d at all values of the inclination angle, including $\theta = 0$, to fit the experimental observations. If you opt for the SHI-04 model described in [Ref. 96] you are recommended to select the corresponding relation for the inclination factor, which is

$$m(\theta) = 1.85(\cos \theta)^{0.21}(1 + \sin \theta)^{0.95} \quad \text{Eq. 44.17}$$

Note that this also has $m > 1$ for vertical pipes.

Three-phase flow and oil-water slip

The drift flux model described above was formulated for two-phase gas-liquid mixtures. For three-phase mixtures flowing in the wellbore, 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

$$\sigma_{gl} = \frac{\alpha_o \sigma_{go} + \alpha_w \sigma_{wg}}{\alpha_o + \alpha_w} \quad \text{Eq. 44.18}$$

σ_{go} and σ_{wg} are the gas-oil and water-gas interfacial tensions. They can either be supplied as tabulated functions of pressure with the keywords **STOG** and **STWG** respectively (in ECLIPSE 100), or calculated internally from built-in correlations obtained from Beggs [Ref. 13]. In ECLIPSE 100 the reservoir fluid temperature is unknown and is arbitrarily set to 160 °F (71.1 °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. 48] to model the slip between oil and water in the liquid phase. This takes the form of

$$v_o' = C_0' v_l + v_d' \quad \text{Eq. 44.19}$$

They suggest a value of 1.2 for the profile parameter, which becomes unity when oil is the continuous phase at $\alpha_o > 0.7$. We implement this as a continuous function of the holdup fraction of oil in the liquid phase, $\alpha_{ol} = \alpha_o / (\alpha_o + \alpha_w)$

$$\begin{aligned} C_0' &= A && \text{when } (\alpha_{ol} \leq B1) \\ C_0' &= 1 && \text{when } (\alpha_{ol} \geq B2) \\ C_0' &= A - (A - 1) \left(\frac{\alpha_{ol} - B1}{B2 - B1} \right) && \text{when } (B1 < \alpha_{ol} < B2) \end{aligned} \quad \text{Eq. 44.20}$$

The values of the coefficients default to $A = 1.2$, $B1 = 0.4$ and $B2 = 0.7$. A condition equivalent to equation 44.9, that the oil superficial velocity should increase with the oil holdup fraction, requires

$$B1 < (2 - A)B2 \quad \text{Eq. 44.21}$$

The oil-liquid drift velocity is calculated as

$$v_d' = 1.53 v_c' (1 - \alpha_{ol})^2 \quad \text{Eq. 44.22}$$

where

$$v_c' = \left[\frac{\sigma_{ow} g (\rho_w - \rho_o)}{\rho_w^2} \right]^{1/4} \quad \text{Eq. 44.23}$$

σ_{ow} is the oil-water interfacial tension. This can either be supplied as a tabulated function of pressure with the keyword `STOW` (in ECLIPSE 100) or calculated internally as the difference between the values of σ_{go} and σ_{wg} obtained from the above correlations. Hasan and Kabir state that the drift velocity should become zero when oil is the continuous phase, but we keep the form of equation 44.20 because when $\alpha_{ol} = 0.7$ the drift velocity will have diminished by an order of magnitude from its value at $\alpha_{ol} = 0$.

If you select the SHI-03 or SHI-04 model in item 1 of keyword `WSEGDFMD`, ECLIPSE will use a different formulation for oil-water slip based on the results described in or respectively. In these experiments no profile slip was observed between the oil and water phases, and accordingly the parameter A in equation 44.21 is set equal to 1.0. The parameters $B1$ and $B2$ are redundant since C_0' is always 1.0. In place of equation 44.15 the oil-liquid drift velocity is calculated as

$$v_d' = 1.53 v_c' (1 - \alpha_{ol})^{n'} \quad \text{Eq. 44.24}$$

where n' is set equal to 0.95 or 1.0 respectively.

The oil-water drift velocity in inclined segments is scaled in a similar way as described in the previous section for the gas-liquid drift velocity, using an inclination factor $m(\theta)$ as shown in equation 44.16. By default ECLIPSE uses Hasan and Kabir's relation [Ref. 48] as shown in equation 44.15. 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. 97]. This is the recommended option if you select the optimized set of model parameters in keyword `WSEGDFMD`. Note that the built-in table has different inclination factors for gas-liquid and oil-water drift velocities. If you opt for the SHI-04 model described in [Ref. 96] you are recommended to select the corresponding relation for the inclination factor, which is

$$m(\theta) = 1.07 \cos \theta + 3.23 \sin 2\theta - 2.32 \sin 3\theta \quad \text{Eq. 44.25}$$

This relation is valid up to 88° from the vertical. Between 88° and 90° (horizontal) ECLIPSE will extrapolate $m(\theta)$ linearly to zero.

The effect of gas on oil-water slip

The presence of free gas entrained in the liquid can disrupt the slip between the oil and water phases, as [Ref. 96] concludes. It proposes that the inclination factor $m(\theta)$ for oil-water slip should be multiplied by a factor $m_g(\theta, \alpha_g)$ that describes this effect as a function of the pipe inclination and the gas volume fraction:

$$\begin{aligned} m_g(\theta, \alpha_g) &= 1 - \frac{\alpha_g}{a_3(\theta)} && \text{when } \alpha_g < a_3 \\ m_g(\theta, \alpha_g) &= 0 && \text{when } \alpha_g > a_3 \end{aligned} \quad \text{Eq. 44.26}$$

The oil-water drift velocity decreases to zero as the gas volume fraction increases from zero to the value a_3 and remains at zero at all higher values of α_g . The cutoff value a_3 is a function of the pipe inclination. At angles between 0 and 70 degrees from the vertical only a small amount of gas is required to eliminate the drift between oil and water, while for near horizontal flows gas has a much smaller effect (due to segregation from the liquid). The following equation approximates the observations of [Ref. 96]

$$a_3(\theta) = 0.017 e^{\theta^{3.28}} \quad \text{Eq. 44.27}$$

where θ_r is the angle from the vertical in radians ($= \pi\theta^\circ / 180$).

You can include the gas effect in the calculation of the drift velocity by selecting the SHI-04 option in item 3 of keyword WSEGDFMD. You are recommended to select this option whenever you select the SHI-04 model in item 1. However, the SHI-04 gas effect option may be used with any of the drift flux models available in item 1, but note that it will not reduce the profile slip which is present in the ORIGINAL model where $A = 1.2$.

Friction

The calculation of the friction pressure loss is based on the formulation used in the correlation of Hagedorn and Brown [Ref. 45]

$$\delta P_f = \frac{C_f f L w^2}{A^2 D \rho} \quad \text{Eq. 44.28}$$

where

f is the Fanning friction factor

L is the length of the segment

w is the mass flow rate of the fluid mixture through the segment

A is the segment's area of cross-section for flow

D is the segment's diameter

ρ is the in-situ density of the fluid mixture

C_f is a units conversion constant

2.679E-15 (METRIC), 5.784E-14 (FIELD), 1.523E-13 (LAB), 2.644E-15 (PVT-M).

The Fanning friction factor is calculated using the same method as in the Wellbore Friction Option (see "Calculating the frictional pressure loss"), but with the Reynolds number formulated as

$$R_e = \frac{C_r D w}{A \mu} \quad \text{Eq. 44.29}$$

C_r is a units conversion constant

0.01158 (METRIC and PVT-M), 0.01722 (FIELD), 0.02778 (LAB).

The viscosity μ is calculated as an average of the phase viscosities weighted according to their volume fraction.

The above formulation becomes equivalent to the homogeneous model used in the Wellbore Friction option when there is no slip between the phases.

Acceleration pressure loss

The acceleration pressure loss across a segment is the difference between the velocity head of the mixture flowing across the segment's outlet junction and the velocity heads of the mixture flowing through all its inlet junctions (its main inlet and any side branches connecting with the segment).

$$\delta P_a = H_{v\text{out}} - \sum_{\text{inlets}} H_{v\text{in}} \quad \text{Eq. 44.30}$$

The velocity head of the mixture flowing through a junction is

$$H_v = \frac{0.5C_f w^2}{A^2 \rho} \quad \text{Eq. 44.31}$$

For the outlet junction flow, A is the cross-section area of the segment. For inlet junction flows, A is the maximum of the cross-section areas of the segment and the inlet segment. Thus the acceleration pressure loss will give the Bernoulli effect at restrictions but no pressure recovery at expansions. Note also that, by including the full velocity head of any inflow from side branches, we are assuming that the inflow is directed into the segment with no momentum loss.

Inflow of reservoir fluid into the segment through grid block connections is **not** included in the inflow velocity head. The acceleration pressure loss thus takes account of the need to accelerate this fluid from zero velocity to the segment's outflow velocity.

Downhole separators (ECLIPSE 100)

Downhole separators are available in the ECLIPSE 100 version of the multisegment well model.

A downhole 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 downhole separator

A segment is designated as a downhole 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 downhole pump is required to maintain the flow through the separator and reinject 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 downhole water separator, there is a simple device model with an additional term to limit the carryover 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 44.7 shows schematically a multisegment well with a downhole separator extracting water to reinject into the formation. The segments show the direction of positive flow (towards the wellhead) to illustrate that the offtake branch has negative flow.

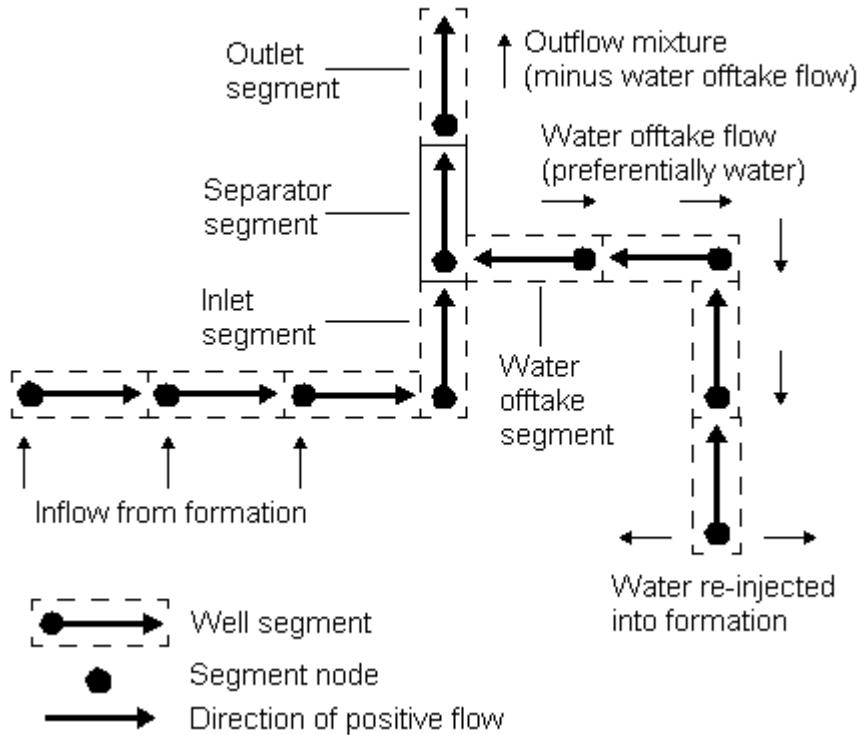


Figure 44.7. Segment diagram for a well with a downhole water separator

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

$$\begin{aligned}\alpha_w^{of} &= E + (1 - E)\alpha_w^{sep} \\ \alpha_g^{of} &= (1 - E)\alpha_g^{sep} \\ \alpha_o^{of} &= (1 - E)\alpha_o^{sep}\end{aligned}\tag{Eq. 44.32}$$

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} ,

$$\begin{aligned} E &= E_{max} && \text{when } \alpha_w^{sep} > \alpha_{lim} \\ E &= \frac{\alpha_w^{sep}}{\alpha_{lim}} E_{max} && \text{when } \alpha_w^{sep} < \alpha_{lim} \end{aligned} \quad \text{Eq. 44.33}$$

E_{max} and α_{lim} are user-defined constants for the offtake, specified in keyword [WSEGSEP](#).

The local volumetric flow rate of each free phase through the offtake is proportional to its holdup fraction α^{of} . Free oil (and free gas) may contain dissolved gas (and vaporized oil) according to the R_s (and R_v) of the phase within the separator segment. The flows at surface conditions are thus

$$\begin{aligned} Q_o^{of} &= Aj(\alpha_o^{of} b_o^{sep} + R_v^{sep} \alpha_g^{of} b_g^{sep}) \\ Q_w^{of} &= Aj\alpha_w^{of} b_w^{sep} \\ Q_g^{of} &= Aj(\alpha_g^{of} b_g^{sep} + R_s^{sep} \alpha_o^{of} b_o^{sep}) \end{aligned} \quad \text{Eq. 44.34}$$

where

- A is the area of the offtake,
- j is the volumetric flux of the mixture (that is its flow velocity)
- 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

$$\begin{aligned} \alpha_g^{of} &= E + (1-E)\alpha_g^{sep} \\ \alpha_w^{of} &= (1-E)\alpha_w^{sep} \\ \alpha_o^{of} &= (1-E)\alpha_o^{sep} \end{aligned} \quad \text{Eq. 44.35}$$

where

$$\begin{aligned} E &= E_{max} && \text{when } \alpha_g^{sep} > \alpha_{lim} \\ E &= \frac{\alpha_g^{sep}}{\alpha_{lim}} E_{max} && \text{when } \alpha_g^{sep} < \alpha_{lim} \end{aligned} \quad \text{Eq. 44.36}$$

Behavior of the model

An oil/water separation system powered by electric submersible pumps is described in [Ref. 14]. A pump is necessary to boost the pressure to reinject the water into the formation. Typically, the pump is placed either at the separator inlet to boost the feed pressure (a ‘push-through 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 carryover 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 reinjected along with the water. The choice of pump power depends on whether the primary objective is to reinject 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 reinjecting some of the produced oil).

ECLIPSE offers a number of ways of automatically cutting back the pressure differential from an overpowered pullthrough pump to maintain optimum separation:

- Use the built-in device model for a pullthrough pump for a downhole 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 carryover 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 keyword SPPOW.
- Represent the pump pressure versus flow characteristics by a VFP table (using keyword [WSEGTABLE](#)). 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 carryover.
- Represent the pump pressure versus flow characteristics by a VFP table (using keyword [WSEGTABLE](#)) 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.

```

VFP PROD
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 non linearities 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](#)").

Thermal multisegment well model

For the 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 [\[Ref. 107\]](#).

Energy inflow

The flow of energy between a grid block and its associated segment's node is given by the inflow performance relationship

$$q_{ej} = T_{wj} \cdot \sum_p M_{epj} \cdot (P_j + H_{cj} - P_n - H_{nc}) \quad \text{Eq. 44.37}$$

where

M_{epj} is the energy mobility in each phase at the connection. If the node is injecting fluid into the formation, the ratio of the phase mobilities reflects the mixture of phases in the segment.

The other terms in this equation are the same as used in the flow equation, equation [44.37](#).

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

$$Q = -K \cdot A \cdot \frac{\partial T}{\partial X} \quad \text{Eq. 44.38}$$

where:

K is the thermal conductivity and

A is the cross-sectional area.

This formula is used to calculate heat conduction along the well.

For a length of cylindrical annulus, equation 44.38 can be integrated over the radius to give

$$Q = \frac{L}{R} \cdot (T_i - T_o) \quad \text{Eq. 44.39}$$

where

$$R = \frac{1}{2\pi} \cdot \frac{1}{K} \cdot \ln \frac{r_o}{r_i} \quad \text{Eq. 44.40} \quad \text{the specific thermal resistance of the annulus}$$

L is the pipe length

T_i is the temperature at the inner radius

r_i and T_o is the temperature at the outer radius r_o .

The resistance of a thin annulus is calculated by substituting $r_o = r_i + \Delta r$ into equation 44.40 to give

$$R = \frac{1}{2\pi} \cdot \frac{\Delta r}{K} \cdot \frac{1}{r_i} \quad \text{Eq. 44.41}$$

Heat transfer along the well

Equation 44.38 is used to model heat transfer along the well. You can specify a thermal conductivity and a cross-sectional area using the `WELSEGS` keyword. 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

Equation 44.39 is used to calculate heat conduction from the well to the formation. For heat losses from a segment to a reservoir grid, heat transfer occurs from a bulk flow in the segment across a pipe wall, insulation, annulus, casing wall, cement and altered formation to the unaltered formation. Heat loss from a segment to rocks outside the simulation grid can be modeled in exactly the same way if you specify the external temperature.

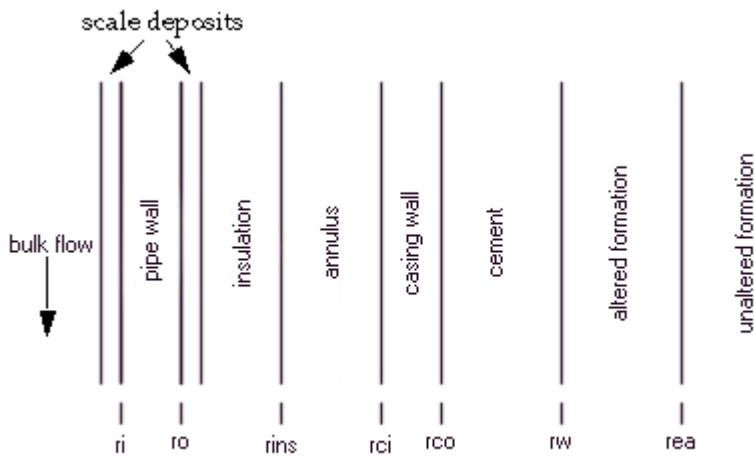


Figure 44.8. Heat conduction between a well and the formation

The specific thermal resistance for these cases is defined by summing terms similar to equations 44.40 and 44.41. See Prats [Ref. 87].

$$\begin{aligned}
 R_h = & \frac{1}{2\pi} \left[\frac{1}{h_f \cdot r_i} + \frac{1}{h_{P_i} \cdot r_i} + \frac{\ln(r_o / r_i)}{\lambda_P} + \dots \right. \\
 & \dots + \frac{1}{h_{P_o} \cdot r_o} + \frac{\ln(r_{ins} / r_o)}{\lambda_{ins}} + \frac{1}{h_{rc_an} \cdot r_{ins}} + \dots \\
 & \dots + \frac{\ln(r_{co} / r_{ci})}{\lambda_P} + \frac{\ln(r_w / r_{co})}{\lambda_{cem}} + \frac{\ln(r_{ea} / r_w)}{\lambda_{ea}} + \frac{f(t_D)}{\lambda_E} \left. \right]
 \end{aligned}$$

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. The remaining parameters are illustrated in figure 44.8 and are defined:

r_i is the inner radius of the pipe

r_o is the outer radius of the pipe

r_{ins} is the external radius of the insulation

r_{ci} is the inner radius of the casing

r_{co} is the outer radius of the casing

r_w is the wellbore radius

r_{ea} is the radius of the altered earth zone

The calculated value for the specific thermal resistance, R_h , is entered using the [WSEGHEAT](#) keyword.

Heat transfer from the tubing to the casing

Equation 44.38 is used to calculate heat conduction losses from a segment to another segment. This case is only applicable when the multisegment 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, as illustrated in the following figure.

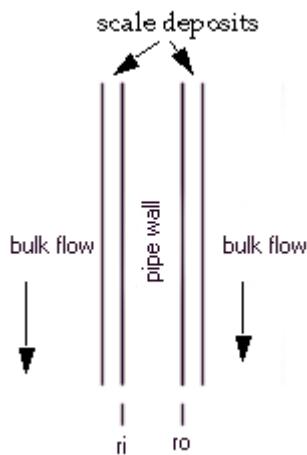


Figure 44.9. Heat conduction between the tubing and the casing

The specific thermal resistance for this case 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

r_i , r_o , h_f , h_{P_i} and λ_P retain their previous definitions and:

h_{P_o} is the heat transfer coefficient across any deposits of dirt or scale on the outer wall.

The calculated value for the specific thermal resistance, R_h , is entered using the [WSEGHEAT](#) keyword.

Using the multisegment well model

If the run contains any multisegment wells, the dimensioning data must be supplied in the RUNSPEC section keyword [WSEGDIMS](#). A run cannot contain both multisegment wells and friction wells; any friction wells must be converted to multisegment wells (see below).

Defining multisegment wells

A multisegment well, like any other well, must first be introduced with the [WELSPCS](#) keyword. The connection data must also be specified in the usual way with the [COMPDAT](#) (or [COMPDTL](#)) keyword. The well must then be defined as a multisegment 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 multisegment 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 multilateral 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 multisegment 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 (see figure 44.10), since the flow is modeled as entering the well segment at its node.

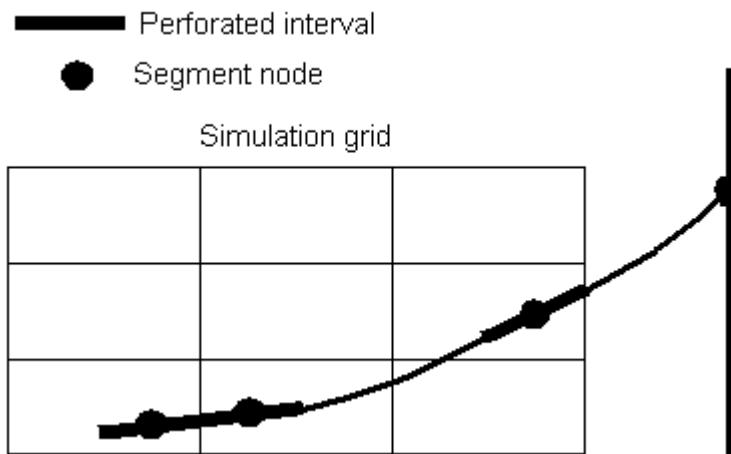


Figure 44.10. Segment node positions with one segment per grid block connection

In multisegment 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](#)").

ECLIPSE 100

It is possible to define the initial conditions (the pressure and the fluid contents) in each segment manually using keyword `WSEGINIT`. This overrides the standard initialization that ECLIPSE performs, which depends in the type of well and the conditions in the connecting grid blocks.

If required, looped flow paths within multisegment wells can be specified using the `WSEGLINK` keyword. See "[Looped flow paths](#)" for further information.

Defining the flow model

The `WELSEGS` keyword also defines the default flow model for the well: homogeneous flow or the drift flux model. When the segments are first defined, they adopt the flow model specified in `WELSEGS`. 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](#)"). You can change the model definition, which dictates the default values of these parameters for all well segments, using keyword `WSEGDFMD`. If you wish to give some of the parameters different values from their model defaults, you can do so by **subsequently** entering the `WSEGDFPA` keyword. You can set selected parameters for individual segments with keyword `WSEGFMOD`. The drift flux model also requires values of the interfacial tensions; these may either be input as functions of pressure using tables with keywords `STOG`, `STOW` and `STWG` (in ECLIPSE 100) or calculated from built-in correlations (see "[Three-phase flow and oil-water slip](#)").

Individual segments may be designated to have their pressure drops determined from VFP tables, using the keyword `WSEGTABL` (see "[Pressure drops from VFP tables](#)"). The tables must be structured as production well VFP tables, and must have previously been entered with the keyword `VFPPROD`.

Specific segments may also be designated to represent

- a flow limiting valve by using the keyword [WSEGFLIM](#)
- or a general sub-critical valve by using keyword [WSEGVALV](#)
- or a ‘labyrinth’ flow control device by using the keyword [WSEGLABY](#)
- or a ‘spiral’ flow control device by using the keyword [WSEGSICD](#)
- or an ‘autonomous’ flow control device by using the keyword [WSEGAICD](#).

ECLIPSE uses built-in formulae to model the pressure losses across these devices.

Modifying pressure drops and segment properties

The [WSEGMULT](#) keyword applies a scaling factor to the frictional pressure drop along specified segments. This could be used, for example, to reduce the flow from a branch by imposing a high frictional pressure drop in the segment nearest the main stem. It could also be used to account for pressure loss in bends, if their ‘equivalent length’ is known, as they increase the effective length of the segment for the frictional pressure drop.

Optionally, the pressure drop multiplier can be made to vary with the water-oil ratio and gas-oil ratio of the mixture flowing through the segment. A variable multiplier may be applied, for example, to model the operation of an adjustable flow control device whose purpose is to choke back production from high WOR or GOR regions.

The [WSEGPROP](#) keyword can be used to alter the geometric properties of a segment. The segment’s length, however, cannot be altered because the grid block connections are located by their overall distance down the tubing.

Workovers

When a workover with an action '+CON' is performed on a multisegment 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 multilateral wells, the branch topology is taken into account so that only branches subordinate to the worst-offending connection are closed off.

Heat loss

ECLIPSE 300 Thermal

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 multisegment well source terms, the keyword [WSEGEXSS](#) can be used to specify the temperature of imported gas.

Thermal contact lengths

Heat transfer occurs to or from a segment node. Since each segment node is at the end of a segment, the heat transfer to and from each node represents half the heat transfer in the two segments either side of the node. For heat transfer between a completion and a segment the thermal contact length is the length of well within the completion cell. This should be specified using the COMPSEGS keyword. For heat transfer between segments, or between a segment and an external temperature, the thermal contact length is the length of well closest to the segment node.

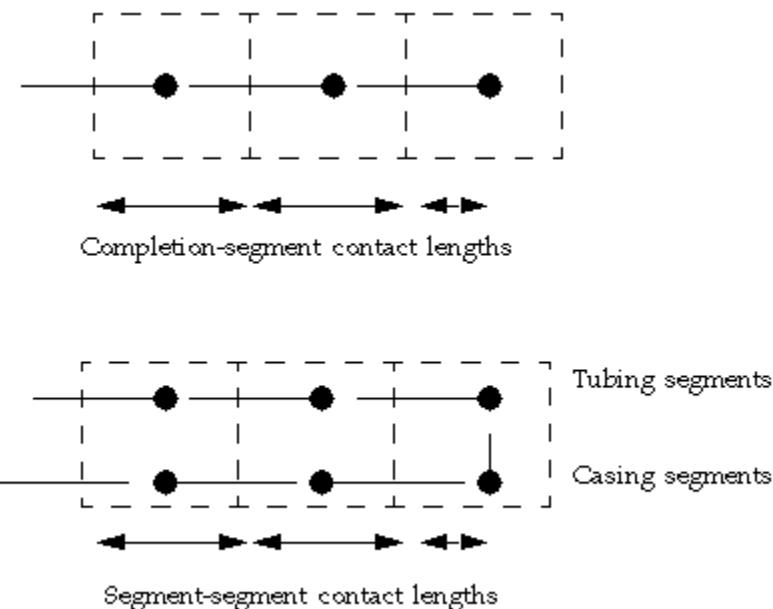


Figure 44.11. Thermal contact lengths

Output

ECLIPSE 100

The segment structure and connection locations are written to the PRINT file in tabular form if the RPTSCHED mnemonic WELSPCS is set. It is advisable to check your data by examining this output, as an error can easily be made while entering complicated multilateral well data.

At each report time, when well reports are requested (RPTSCHED mnemonic WELL), 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 multisegment wells (see "[Graphical presentation of friction effects in horizontal wells](#)"). GRAF recognizes the presence of multisegment wells, and produces the same series of prompts as it does for friction wells.

More detailed information on the flowing conditions inside multisegment wells can be written to the RFT file at specified times (see the [WRFTPPLT](#) 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 multisegment 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 multisegment 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 multisegment wells (see "Graphical presentation of friction effects in horizontal wells"). GRAF recognizes the presence of multisegment wells, and produces the same series of prompts as it does for friction wells. Note that the SUMMARY section keywords for connection flows differ from the corresponding ECLIPSE 100 keywords.

Performance

If the multisegment well contains looped flow paths (see the WSEGLINK keyword), then a different solver is used for the well linear equations with each well iteration. The WSEGSOLV keyword can be used to modify the behavior of this looped well linear solver if required.

ECLIPSE 100

Multisegment wells can require substantially more well iterations to converge than standard wells, especially when being opened or while switching into or out of crossflow. By default, each well is allowed up to MXWSIT iterations to converge, which is set using keyword TUNING (or TUNINGL or TUNINGS). 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 timestep. For multisegment 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. Examples include item 30, which governs methods aimed at improving convergence by chopping the increment in all the well’s solution variables and item 63, which can 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 multisegment 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 WVPEXP).

ECLIPSE 300

Multisegment wells can require more well iterations to converge than standard wells require. There is a CPU expense if the simulator is run in compositional mode with many multisegment wells and many segments per well. To gain the best performance, it is recommended that only a few wells be run as multisegment 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. See keyword [CVCRIT](#) for more details.

The complexity of the branches does not significantly affect performance of the multisegment wells. Neither does the number of completions that are connected.

See also "[The top segment](#)".

If a multisegment 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 item 34 in the OPTIONS3 keyword 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 crossflowing. Firstly, the multisegment 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 item 34 of OPTIONS3 is set to 10000 then all multisegment wells use the above.

Restrictions

ECLIPSE 100

Multisegment wells cannot at present be used in conjunction with the following facilities:

- The wellbore friction option
- The temperature option
- The solvent model
- The Gi model
- Automatic refinement in the LGR option

In addition, it is strongly recommended that multisegment wells are not used in conjunction with the following facilities:

- The [COMPFLSH](#) keyword.

In both the conventional well and the multisegment well models, the inflowing fluid is transformed to flash conditions when it enters the wellbore through the connections to the reservoir grid. To calculate the fluid properties (for example for the hydrostatic head) a back transformation is applied based on the overall inflow; this is consistent with the treatment of the well as a “mixing tank” and is thus appropriate for the conventional well model. With a multisegment well, this back transformation (based on the overall inflow) is still applied, even though the local fluid properties may vary with position in the well. Thus the local fluid properties are not as accurately represented as they should be for a discretized wellbore model such as a multisegment well. These fluid properties influence the hydrostatic and frictional pressure gradients, the out flowing mixture properties when crossflow occurs, and slip (when using the drift-flux model).

ECLIPSE 300

Multisegment 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](#))
- flash transformation facility ([COMPFLSH](#)).

Converting friction wells to multisegment wells

ECLIPSE 100

The keywords [WFRICSEG](#) and [WFRICSGL](#) provide an additional way of defining a multisegment 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 "Wellbore friction option") into datasets that use multisegment 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 multisegment 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](#)([WFRICTNL](#)).

The [WFRICSEG](#) keyword does not offer the flexibility of [WELSEGS](#) and [COMPSEGS](#) in defining multisegment 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 multilateral 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 [WSSEGDIAMS](#). For example, a well consisting of a main stem and a single lateral branch will be converted to have three branches: the upper part of the main stem, the lateral branch and the section of the main stem below the branch junction.

Converting a dataset from using friction wells to using multisegment wells

1. Remove the [FRIC](#) keyword from the RUNSPEC section, replacing it with the [WSEGDI](#)MS keyword.
 - a. The maximum number of multisegment wells in the run is the same as the maximum number of friction wells.
 - b. The maximum number of segments per well is 1 + the maximum number of grid block connections per well.
 - c. The maximum number of branches per well should allow for the ‘extra branches’ created by ECLIPSE in the main stem below lateral branch junctions
2. Change all [WFRIC](#)TN keywords to [WFRIC](#)SEG. In LGR runs, any [WFRIC](#)TNL keywords should be changed to [WFRIC](#)SGL. The keyword data should remain unchanged.
3. Optionally the [SUMMARY](#) section can include the mnemonics associated with multisegment wells (SOFR for example)

When ECLIPSE converts a friction well into a multisegment well, it prints out its equivalent WELSEGS and COMPSEGS (or COMPSEGL) keyword data in the PRINT file, at the place where [WFRIC](#)TN (or [WFRIC](#)TNL) 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

- [WSEGDI](#)MS Sets array dimensions for multisegment wells.
- [WSEGSO](#)LV Sets parameters for the multisegment well iterative linear solver.

PROPS section

Restriction	Keyword	Description
E100 only (used in the drift flux model).	STOG	Oil-gas surface tension versus pressure
E100 only (used in the drift flux model).	STOW	Oil-water surface tension versus pressure
E100 only (used in the drift flux model).	STWG	Water-gas surface tension versus pressure

SUMMARY section

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 multiphase 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  /
/
```

Refer to the following sections in the *ECLIPSE Reference Manual*.

- [Multisegment wells](#)
- [Secondary group and well quantities](#)

SCHEDEULE section

Restriction	Keyword	Description
	COMPSEGL	Defines the location of multisegment well completions in a local grid.
	COMPSEGS	Defines the location of completions in a multisegment well.
E300 only	CVCRIT	Items 23 - 26 set convergence criteria for multisegment wells. T
E100 only	OPTIONS	<p>Item 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.</p> <p>Item 63 controls the calculation of the hydrostatic head between each connection of a multisegment well and its corresponding grid block.</p> <p>Item 77 restores back-compatibility with the pre-2000A drift flux model calculations involving oil-water slip and inclined tubing.</p>
E300 only	OPTIONS3	Item 34 may help convergence in difficult situations involving crossflow.
E300 only	SCONPROD	Secondary well production controls.
E300 only	SCONINJE	Secondary well injection controls

Restriction	Keyword	Description
E300 only	STEST	Secondary well reopen test control
	WELSEGS	Defines the segment structure of a multisegment well.
E100 only	WFRICSEG	Converts WFRICTN keyword data for a friction well into data for a multisegment well.
E100 only	WFRICSGL	Converts WFRICTNL keyword data for a friction well into data for a multisegment well.
	WSEGAICD	Defines segments to represent an ‘autonomous’ inflow control device.
	WSEGDFIN	Inputs a table of inclination scaling factors versus deviation angle for the drift flux slip model
	WSEGDFMD	Defines the drift flux slip model
	WSEGDFPA	Changes parameter values in drift flux slip model.
	WSEGEXSS	Specifies fluid import/removal through an external source/sink.
	WSEGFLIM	Defines segments to represent a flow limiting valve.
	WSEGFMOD	Specifies the multiphase flow model and related parameters.
Thermal only	WSEGHEAT	Specifies heat loss from a segment.
E100 only	WSEGINIT	Sets initial pressure and fluid contents of well segments.
E100 only	WSEGITER	Activates the robust iteration scheme for solving multisegment wells.
	WSEGLABY	Defines segments to represent a ‘labyrinth’ inflow control device.
	WSEGLINK	Defines the chord segment links for specifying looped flow paths
	WSEGMULT	Imposes constant or variable multipliers for segment frictional pressure drops.
	WSEGPROP	Modifies the properties of individual well segments.
E100 only	WSEGPULL	Defines a pullthrough pump for a downhole water separator.
E100 only	WSEGSEP	Defines a segment to represent a downhole separator.
	WSEGSICD	Defines segments to represent a ‘spiral’ inflow control device.
	WSEGTABL	Designates segments to have their pressure drops calculated from a VFP table.
	WSEGVALV	Defines segments to represent a sub-critical valve.
E300 only	WSEGWELL	Define the association between segment number and secondary well name.

45

Network option

<input checked="" type="checkbox"/>	ECLIPSE 100
<input checked="" type="checkbox"/>	ECLIPSE 300
<input checked="" type="checkbox"/>	SPECIAL

The network option is designed to provide variable tubing head pressure (THP) limits to groups of wells, which depend on the groups' flow rates according to a set of pipeline pressure loss relationships. The option calculates the well THP limits dynamically by balancing the flow rates and pressure losses in the network.

Consider a group of wells flowing into a common manifold which is connected by a pipeline to an outlet at a fixed pressure (figure 45.1). For example, the group may represent the wells flowing into a particular subsea completion manifold, and the fixed pressure outlet may be a separator or stock tank. The pressure in the manifold will vary with the overall production rate of the group, according to the relationship between the pressure loss through the pipeline and the flow rate, water cut and GOR of the group. This relationship is represented schematically in figure 45.2 by the pipeline curve. However, the group's production rate will itself vary with the manifold pressure if at least one of its wells is on THP control, since the manifold pressure is applied to each well as a THP limit. The combined production rate of the wells versus the THP limit is represented by the wellbore curve in figure 45.2. The actual manifold pressure and flow rate of the group will be given by the intersection of these two curves.

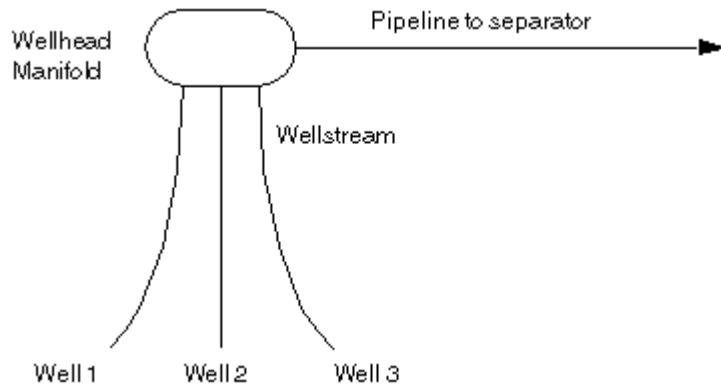


Figure 45.1. Common wellhead manifold and pipeline

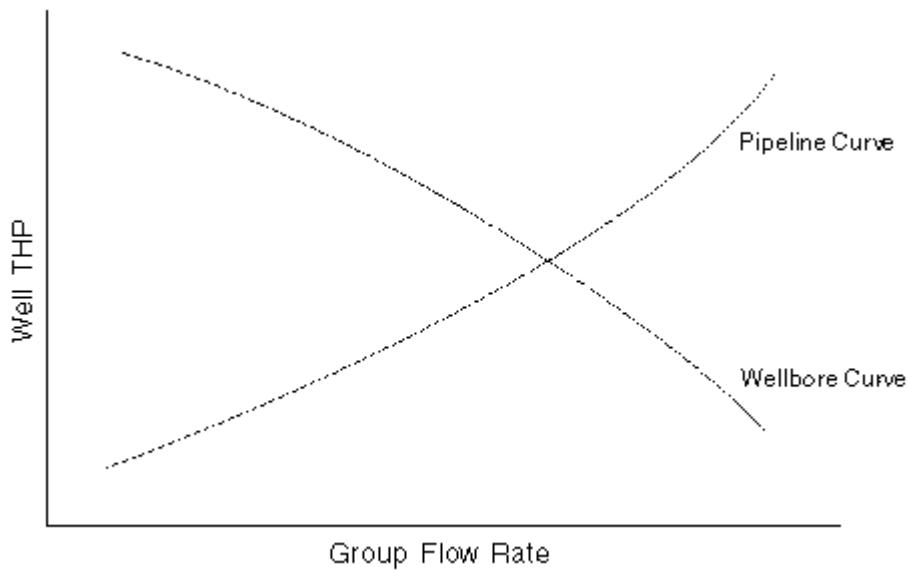


Figure 45.2. Well THP versus group flow rate for wellstream and pipeline

Using the ECLIPSE multilevel 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 overwritten by the calculated values.) The nodal pressure of each group is printed in the well reports in the THP column, and can also be written to the SUMMARY file using the mnemonic GPR.

For example, consider the grouping tree shown in Figure 45.3. The groups on the bottom level (GR-A1 for example) may be subsea completion manifolds, connected to their respective platforms PLAT-A and PLAT-B by pipelines. The two platforms can be declared as fixed-pressure nodes, with pressures set equal to their separator pressures. Alternatively the FIELD can be given a fixed pressure, and the two platforms connected to it by additional pipelines. It is not essential for the network to cover the whole grouping tree. For example group GR-A2 can be isolated from the network (by not giving it a pipeline connection to PLAT-A), and THP limits for its wells set or defaulted independently.

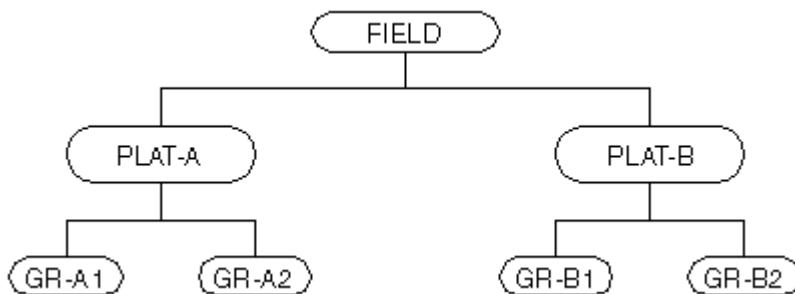


Figure 45.3. An example network

Note that the network structure coincides with the grouping hierarchy built using the [GRUPTREE](#) keyword. If you require the network to have a different topology from the group control hierarchy, then you must use the extended network model which is described in a later section (see "[The extended network model](#)").

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 subsea 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 (unchocked) 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 subsea 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](#)".) 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 if 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](#)".) 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 (ALQ), 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 reset 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, which decreases PLAT-A's nodal pressure, consequently reducing the nodal pressures in the subordinate groups and boosting their wells' production.

ECLIPSE 100

Additional, more versatile, facilities for automatic pump or compressor switching are available when using the extended network model, which is described in a later section (see "[Automatic compressors](#)").

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 looking 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 then cannot be also used to represent a pump or compressor, and the pipeline cannot contain any automatic pump/compressor.

Injection network

In addition to the production network, a separate network can also be defined for water or gas injection. Each network exists independently of the other. A particular group may be a node in both networks, if required, but in this case the group will have two independent nodal pressures, one for each network to which it belongs.

Pumps can be modeled in the injection network flow lines, but the artificial lift quantity cannot be varied.

As an example of an injection network, consider a pump that takes water at a fixed inlet pressure and pumps it into a manifold supplying a number of water injectors. The outlet pressure of the pump, and thus the pressure in the manifold, varies with the total injection rate. The injection wells operate under THP control, with the THP equal to the pressure in the manifold. Using the grouping tree shown in figure 45.3, let the injection pump be situated on platform PLAT-B, and the injection wells belong to group GR-B2. Declare PLAT-B as a fixed-pressure node in the water injection network, with its 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](#)").

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 also possible for subordinate groups 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 *i* 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. This 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 timestep, 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 timestep 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 timestep, and to specify the ‘balancing interval’ for subsequent timesteps. Subsequent balancing calculations are performed at the beginning of each timestep 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 timestep. 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 timestep. By the end of the timestep, 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 timestep or interval between balancing calculations, the greater the balancing error will be at the end of the timestep. ECLIPSE 100 checks the balancing error in each branch of the network at the end of each timestep, 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 timestep. The THP target of wells is also checked against the parent node pressure before the network is considered balanced.

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 timestep. (The value of NUPCOL may be set using either keyword [NUPCOL](#) or keyword [GCONTROL](#)). This option, in general, results in much smaller balancing errors at the end of the timestep, as the reservoir conditions at each successive Newton iteration more closely resemble the converged conditions at the end of the timestep. However, the option may cause the timesteps 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 timestep. (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 the largest branch balancing error, in the NETBALAN keyword. ECLIPSE 100 attempts to adjust the timestep length so as to keep the balancing error at approximately its target value, subject of course to any other constraints on the timestep. Timesteps limited by the target balancing error can be identified by the mnemonic ‘NETW’ printed in the timestep information line. If the maximum permitted balancing error is exceeded, ECLIPSE 100 chops the timestep and rebalance the network. Since timestep 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 timestep in which a well dies. You can set a minimum timestep size to prevent a large reduction in step size when wells die.

Sub-sea completion manifolds

ECLIPSE 100

When a 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 a 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 timestep, 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 timestep 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 corresponds to 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](#)"). 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:

Restriction	Keyword	Description
	VFPPROD	Enter the VFP tables for the production wells and the production network pipelines.
	VFPINJ	Enter the VFP tables for the injection wells and the injection network pipelines.
	WELSPECS	Enter the specification data for all wells.
	COMPDAT	Specify the connection data for all wells.
	WCONPROD	Set the control targets and limits for the production wells. THP limits should not be set for wells belonging to a group which is a node in the production network, as they will be determined at each balancing calculation.
	WCONINJE	Set the control targets and limits for the injection wells. THP limits should not be set for wells belonging to a group which is a node in the injection network, as they will be determined at each balancing calculation.
	GRUPTREE	Construct the grouping tree, if there is more than one level of groups below the FIELD. The example below shows the GRUPTREE records for the network in Figure 45.3.
	GRUPNET	Define the production network. For example, in Figure 45.3, let PLAT-A and PLAT-B have fixed pressures of 300 psia and the bottom level groups be sub-sea completion manifolds connected to the platforms by pipelines.

Restriction	Keyword	Description
	GNETINJE	Define an injection network. For example, in Figure 45.3, let PLAT-B have a fixed pressure of 14.7 psia and GR-B2 contain the water injectors. The water injection pipeline between these two nodes is represented by injection VFP table number 2, which includes the pump performance characteristics.
	GCONPROD	You can, optionally, use this keyword to set production rate targets and limits for any group. For example, the FIELD has an oil production target of 50,000 stb/day, while the two platforms have water handling limits of 10,000 stb/day. Because the four well-groups are manifold groups subordinate to a group under rate control, they should be given guide rates. Here they are each required to produce the same amount of liquid, so their liquid guide rates are set to the same nominal value - say 100.
E100 only	GNETPUMP	Use if automatic pump/compressor switching is required in the production network. For example, the pipeline from GR-B1 to PLAT-B will be interpolated with an artificial lift quantity of 50 if the oil production rate of GR-B1 falls below 500 stb/day.
	NETBALAN	Use of this keyword is optional. It sets the balancing interval, convergence tolerances and iteration limits, and instructions to limit the balancing error by reducing the timestep size. In the example below, we set a negative balancing interval so that ECLIPSE 100 balances the network in each of the first NUPCOL Newton iterations of every timestep (ECLIPSE 300 always does this).
E100 only	WAITBAL	Optional keyword to prevent Production Rule actions being performed to maintain group production targets until the network is balanced (ECLIPSE 300 always prevents this happening).
	WNETDP	Optional keyword to apply an additional fixed pressure drop between a well's tubing head and its group's node in the network.

Table 45.1: SCHEDULE section keywords

Examples

[GRUPTREE](#) example:

```
GRUPTREE
'GR-A1' 'PLAT-A' /
'GR-A2' 'PLAT-A' /
'GR-B1' 'PLAT-B' /
'GR-B2' 'PLAT-B' /
/
```

[GRUPNET](#) example:

```
GRUPNET
-- GROUP    FIXED      VFP    PUMP    MANIFOLD      INCLUDE    ALQ=
-- NAME    PRESSURE    TAB    POWER    GROUP?      LIFT GAS?    DENS?
'PLAT-*'   300  /
'GR-A1'     1*       1     1*     'YES'    /
'GR-A2'     1*       2     1*     'YES'    /
'GR-B1'     1*       3     1*     'YES'    /
'GR-B2'     1*       3     1*     'YES'    /
/
```

Note: In ECLIPSE 300, the manifold groups must have VFP table numbers of 9999, so a set of intermediate groups must be declared whose outlet pipelines have their pressure drops determined by the VFP tables 1, 2 and 3

GNETINJE example:

```
GNETINJE
-- GROUP   NETWORK  FIXED      VFP
-- NAME    PHASE    PRESSURE   TAB
'PLAT-B' 'WATER'  14.7      /
'GR-B2'  'WATER'  1*        2  /
/
```

GCONPROD example:

```
GCONPROD
-- GROUP  CNTL    OIL    WATER   GAS    LIQU   LIMIT   AVAIL   GUIDE   G.R.
-- NAME   MODE    RATE   RATE    RATE   RATE    ACTION   FLD    RATE   PHASE
'FIELD' 'ORAT'  50000   /      'PLAT*' 'NONE'  1*     10000  2*      'RATE'   /      'GR*'   'NONE'  5*      'YES'    100   'LIQ'   /
/
```

GNETPUMP example:

```
GNETPUMP
-- GROUP    RATE    PHASE    NEW VFP    NEW LIFT   NEW GAS
-- NAME    LIMIT    TAB NO    QUANTITY  CONSUMPTION
'GR-B1'  500     'OIL'    1*       50        /
/
```

NETBALAN example:

```
NETBALAN
-1.0 /
/
```

WAITBAL example:

```
WAITBAL
'YES' /
/
```

WNETDP example:

```
WNETDP
-- WELL      PRESSURE
-- NAME      DROP
'PROD1'    20.0  /
'PROD2'    25.0  /
/
```

The extended network model

The extended network model contains several enhancements over the Standard model for production networks:

- 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 trunk line at different points. This is shown schematically in figure 45.4. GA1, GB1, GA2 and GB2 are all well-groups feeding into the trunk line at nodes N1, N2, N3 and N4 respectively. Let us assume that FIELD is the fixed pressure node.

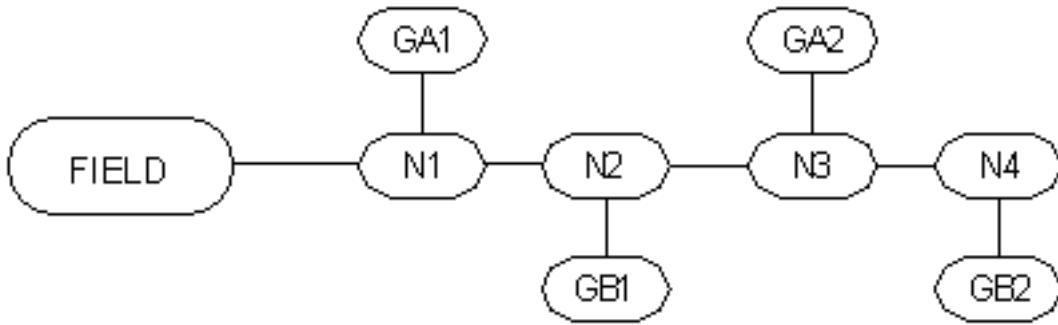


Figure 45.4. The network structure

Now suppose that groups GA1 and GA2 belong to Company A whilst groups GB1 and GB2 belong to Company B. Clearly both companies may need to apply targets or limits to their own total production rate. This would be achieved by defining a group hierarchy with `GRUPTREE` as shown in figure 45.5, which is different from the network structure.

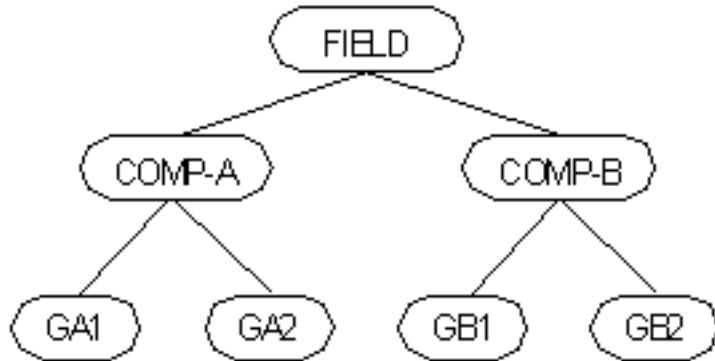


Figure 45.5. The group control hierarchy

The group control hierarchy should be constructed as usual with keyword `GRUPTREE` and the relevant production targets set with `GCONPROD` or `GCONPRI`. But by using the Extended Network Model, a separate network structure can be constructed with keywords `BRANPROP` and `NODEPROP`. The network consists of a gathering tree structure of nodes and branches. The nodes at the bottom level of the tree are “source nodes”, which inject flow into the network. The source nodes must be well-groups or satellite groups (or master groups in Reservoir Coupling master runs), and all source nodes should correspond to groups in the GRUPTREE hierarchy; in ECLIPSE 100 the corresponding nodes and groups must have the same name in both structures but this is not necessary in ECLIPSE 300. The remaining network nodes and GRUPTREE groups need not coincide. The top node of the network tree (the terminal node) must be a fixed pressure node.

In the example above, GA1, GA2, GB1 and GB2 are common to both the network structure and the GRUPTREE hierarchy. Also the fixed pressure node corresponds to the FIELD group (although this is not compulsory). But nodes N1, N2, N3 and N4 only appear in the network structure, and groups COMP-A and COMP-B only appear in the group control hierarchy.

The network may consist of two or more separate gathering trees, in which case each tree must have its own fixed pressure terminal node at the top. It is possible for subordinate groups also to be fixed-pressure nodes; these form ‘sub-networks’ whose pressures are independent of the main network but whose flows will be added into it. For example, in the network shown in figure 45.4 the nodes FIELD and N3 could both be fixed-pressure nodes. Node N3 then acts as a source into node N2 of the main network, while keeping its own pressure independent of the pressures in the main network.

A consequence of separating the group control hierarchy from the network structure is that the location of any gas consumption or import must be identified in both structures. ECLIPSE must know where consumed gas is extracted from the network (to compute downstream pressure losses correctly), and also where it is extracted from the GRUPTREE hierarchy (to perform the group production accounting). Accordingly the `GCONSUMP` keyword in ECLIPSE 100 has an additional item (Item 4) to identify the node at which the gas is extracted from or added to the extended network structure. Alternatively the keyword `NCONSUMP` can be used to define the consumption at a network node, and optionally to assign the consumption to a particular group. The corresponding keyword in ECLIPSE 300 for removing gas from the network is `NGASREM`.

Automatic compressors

ECLIPSE 100

Automatic compressors or pumps are defined with keyword [NETCOMPA](#) instead of [GNETPUMP](#). Each is located at a specified branch of the network. They offer a much wider range of features than the GNETPUMP compressors of the standard network model. Their principal features are:

Activation by group target

Each compressor (or pump) is turned on whenever a nominated group fails to meet 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 support 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 an 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.

Multilevel compression

Multilevel 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 while rebalancing 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](#)").

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 [GCONEENG](#) 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 subsea 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, that is it is governed by item 3 of NETBALAN and **not** item 5. Whilst it is clearly dependent upon the relative sizes of the tolerances in items 2 and 4 of NETBALAN, in general substantially more iterations are needed if there are chokes in the network than if there were not. You should be prepared to increase item 3 of NETBALAN accordingly.

Note: Convergence problems may arise if a group is given a very small rate target, either manually or as a result of a relatively small guide rate. This is particularly true if the low flow required from the wells under that group results in some of them being unable to operate under THP control.

Well and group efficiency factors

When wells have efficiency factors set in keyword **WEFAC**, by default each well contributes its 'time-average' production rate to the network flows, that is its flow rate multiplied by its efficiency factor. The network branch pressure losses are thus calculated from the time-average rate sums of the subordinate wells. This is equivalent to the way in which the group production rates are summed, and is most appropriate when the wells go down individually for short 'random' periods.

But if you wish to model the 'peak' pressure losses in the network, when all the wells are flowing at the same time, then it is possible to specify that the network branch flows should reflect the full flow rates of each well. By setting Item 3 of keyword **WEFAC**, it is possible to specify for each well individually whether its production rate should or should not be multiplied by its efficiency factor when adding its contribution to the network flows. The **group** flow rates, however, are still the time-average sum of the well flows.

ECLIPSE 100

If all the wells in a group have their downtimes synchronized, so that the group either flows with its full rate or not at all, it is more appropriate to apply an efficiency factor to the group (with keyword [GEFAC](#)) rather than the individual wells. If there is a network node with the same name as this group, by default the group's efficiency factor is applied to this node and its flow rate is multiplied by the efficiency factor when adding the node's flow rate to the flow in its parent node. The pressure drop in a node's outlet branch is calculated from the node's flow rate. Thus the node's outlet branch and the branches upstream of the node reflect the full flow rates, while the branches downstream of its parent node reflect the time-average flows. (This is equivalent to the Standard Network model's treatment of group efficiency factors.) However, by setting item 3 of keyword GEFAC it is possible to specify that the group's full flow rates should be used for calculating all downstream pressure losses, by not assigning the group's efficiency factor to the corresponding network node. If there is no network node with the same name as the group, it is possible to apply an efficiency factor to another appropriate network node using keyword [NEFAC](#).

This flexibility is only available when using the Extended Network model, since network node and branch flows are accounted separately from the group flows. In the Standard Network option the branch flows are always the same as the reported group flow rates.

Using the extended network model

Below, we summarize how the use of the Extended Network model differs from that of the Standard model. Other keywords that are not mentioned here (for example [NETBALAN](#)) are common to both models.

RUNSPEC section

To enable the extended version of the network model, the keyword [NETWORK](#) must be entered in the RUNSPEC section. It has two items of data: the maximum number of nodes in the network and the maximum number of branches in the network. So, for the network in Figure 45.4 we might use

```
-----  
NETWORK  
9 8 /  
-----
```

SUMMARY section

The same keywords apply to both the standard and extended network models. When used with the Extended version, names supplied to these keywords must correspond to network nodes instead of [GRUPTREE](#) groups.

Construct the network with BRANPROP and NODEPROP instead of GRUPNET

The network structure is defined in the SCHEDULE section by its branches. Each branch is identified by its inlet node and outlet node. The branches must first be defined with the keyword [BRANPROP](#). This also sets the branch properties, that is the VFP table number and ALQ value. An example of how the production network in Figure 45.4 may be constructed is:

```
-----  
BRANPROP  
-- Inlet   Outlet     VFP      ALQ      ALQ=  
-- node     node      table    value    dens?  
'GB2'    'N4'        3 /  
'GA2'    'N3'        2 /  
'GB1'    'N2'        1 /  
'GA1'    'N1'        9999 /  
'N4'     'N3'        4 /  
'N3'     'N2'        4 /  
-----
```

```
'N2'      'N1'          5 /
'N1'      'FIELD'        6 /
/
```

After the network structure has been defined with BRANPROP, the properties of the network nodes must be defined with keyword NODEPROP. Only the fixed pressure node(s), and other nodes with non-default properties need to be specified here. In the example in figure 45.4, the fixed-pressure node corresponds to the FIELD group and is thus named 'FIELD'. Also let us assume that the source node GA1 corresponds to an automatic choke inlet (or manifold group in ECLIPSE 100) and there is no gas lift. The NODEPROP keyword is then:

```
NODEPROP
-- Node      Fixed      Choke inlet or      Add      Choke control
-- name     pressure    manifold group?    lift gas?   group
  'FIELD'    200.0      /                   /
  'GA1'      1*         'Y'      /
```

Define any gas consumption or import with either GCONSUMP or NCONSUMP

ECLIPSE 100

Any gas consumption or import should be declared in both the network and the group hierarchy. This can be done with either keyword GCONSUMP or NCONSUMP. GCONSUMP declares consumption/import at a group and allows you to specify the network node where this should be applied as well. NCONSUMP declares consumption at a network node and allows you to specify the group where this should be applied as well. For a constant consumption rate, the two keywords are equivalent and it does not matter which you use. However, there are some differences between the two keywords:

- NCONSUMP allows only a constant consumption rate, while GCONSUMP also allows a consumption rate that is a fraction of the group's production, and also a gas import rate.
- The two keywords have different defaults for assigning consumption to the other structure. If consumption is declared at a source group in GCONSUMP, by default it will also be assigned to the corresponding source node of the network. However, if consumption is declared at a source node in NCONSUMP, by default it will **not** be assigned to the corresponding group.
- NCONSUMP may only be used with the Extended Network model. GCONSUMP may be used with either the Extended or Standard models.

Remove any fuel gas from the network with NGASREM

ECLIPSE 300

Any fuel gas consumed at a group (see keyword GRUPFUEL) by default still remains present in the network flow and contributes to the downstream branch pressure drops. But it is possible to remove a corresponding amount of gas from the network at a particular node by using the NGASREM keyword. The gas removal maybe defined as a specified rate or a fraction of the gas flow through the node.

Define any automatic compressors with NETCOMPA

ECLIPSE 100

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.

```
NETCOMP
-- 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 NETCOMP to stay on permanently). They are turned on automatically again as soon as they are needed.

Defining a choke

In figure 45.4 suppose we wish to control the flow of the FIELD group using a choke. This is done by introducing a new node, NCHOKE, to represent the choke inlet, between the nodes N1 and FIELD. 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 timestep. (The value of NUPCOL may be set using keyword [NUPCOL](#) or keyword [GCONTOL](#).) During any subsequent iterations that are required to converge the timestep, the nodal pressures and well THP limits remain unchanged. Thus the balancing error at the end of the timestep reflects only the changes in reservoir conditions that have taken place after the NUPCOLth 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 timestep; either of these options would increase the balancing error and so they are not implemented in ECLIPSE 300.

The default convergence criterion (item 2 of keyword [NETBALAN](#)) is substantially tighter in ECLIPSE 300 because the balancing calculation is more advanced. The branch flows and pressure losses are solved in an inner iteration loop nested inside the iterations that balance the network with the reservoir. The inner loop is converged to a tolerance that is a factor of 10 tighter than the tolerance for the network/reservoir coupling iterations.

The total number of network/reservoir coupling iterations performed over each timestep is reported in the timestep summary table in the column headed [NIT](#) .

Another difference from 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](#)"). However, there are some differences between the two simulators:

- ECLIPSE 100 uses a ‘subsea 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 subsea 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 subsea 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

Keyword	Description
NETWORK	Indicates that the extended network model is to be used, and sets array dimensioning data. The keyword should not be entered if the ‘standard’ network model is to be used for the production network.

SUMMARY section

The following keywords control output of data specific to the network option.

When using the extended network model, the list of names following these keywords must correspond to network nodes instead of [GRUPTREE](#) groups.

Refer to [Network option](#) in the *ECLIPSE Reference Manual*.

SCHEDULE section

Restriction	Keyword	Description
	BRANPROP	Defines the branches in an extended network.
E100 only	COMPOFF	Switches off automatic compressors defined by NETCOMPA in the extended network model or GASFCOMP in the gas field operations model.

Restriction	Keyword	Description
E100 only	GNETDTP	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 network model injection network.
E100 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.
E100 only	NCONSUMP	Specifies gas consumption rates at extended network nodes.
E100 only	NEFAC	Specifies efficiency factors for extended network nodes.
	NETBALAN	Sets parameters for the network balancing calculation.
E100 only	NETCOMPA	Defines automatic compressors in the extended network.
E300 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
E100 only	OPTIONS	<p>Item 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.</p> <p>Item 57 invokes an alternative procedure for handling automatic choke calculations when a low flow target has been set.</p>
E300 only	OPTIONS3	<p>Item 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.</p> <p>Item 51 controls the frequency of recalculating the well potentials, which depend on the THP limits obtained from the network.</p> <p>Item 63 prevents extrapolation of the VFP tables for the branch pressure drop.</p>
E100 only	WAITBAL	Prevents automatic drilling and other production rule actions being performed until the network is balanced.
E100 only	WNETCTRL	Selects whether well THP limits or flow rate limits should be held constant after the network has been balanced.
	WNETDPP	Applies an additional fixed pressure drop between the tubing head of a well and the corresponding network node of its group.

46

Nine-point schemes

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The default discretization in ECLIPSE employs a five-point discretization in two dimensions or a seven point discretization in three dimensions. However, when the grid is non-orthogonal, or the flow is not aligned with the principal directions of the grid, then the solution accuracy may be compromised by this discretization. ECLIPSE uses a nine-point scheme to address this issue.

There are two nine-point schemes in ECLIPSE that address grid non-orthogonality errors and grid orientation effects only.

The [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](#)" for more details. The two nine-point schemes should not be used together.

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. 131]. The standard five-point transmissibilities are first obtained from data entered in the GRID and EDIT sections, and these are then modified to obtain the nine-point transmissibilities.

To derive the nine-point scheme, we start by splitting each transmissibility in the usual five-point scheme into two half-transmissibilities:

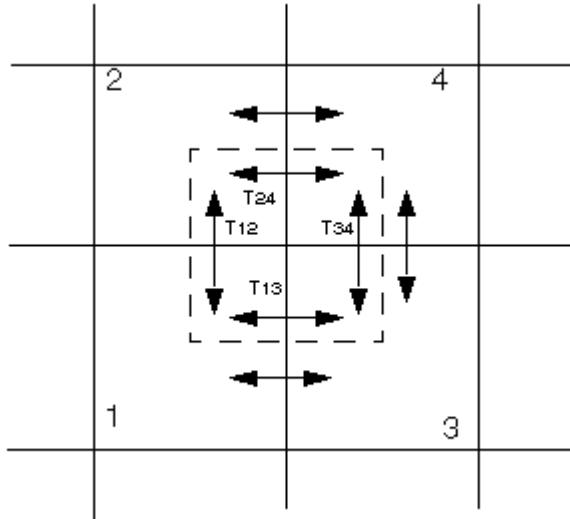


Figure 46.1. Splitting each five-point transmissibility into two half-transmissibilities

Considering just the transmissibility values ($T_{12}T_{24}T_{13}T_{34}$), we wish to replace these by the equivalent system ($t_{12}t_{13}t_{23}t_{24}t_{34}t_{14}$). This will introduce a diagonal coupling between cells 1 and 4 and cells 2 and 3.

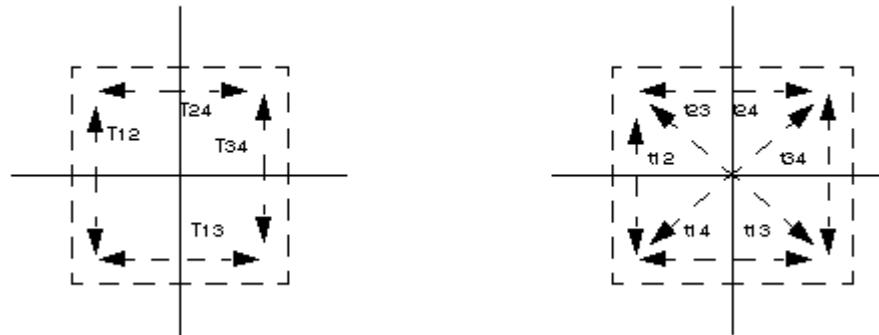


Figure 46.2. Deriving the nine-point transmissibilities using diagonal coupling between cells

We wish the values of ($t_{12}t_{13}t_{23}t_{24}t_{34}t_{14}$) to preserve the total x and y transmissibilities for the pattern, so that

$$T_{13} + T_{24} = t_{13} + t_{24} + t_{23} + t_{14}$$

$$T_{12} + T_{34} = t_{12} + t_{34} + t_{23} + t_{14}$$

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

$$t_{14} = t_{23}$$

A choice for t_{14} 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. 98] we use

$$t_{13} = t_{24} = \frac{1}{3} \left[\frac{(T_{13} + T_{24})(T_{12} + T_{34})}{(T_{13} + T_{24} + T_{12} + T_{34})} \right]$$

The total x and y transmissibility conditions above then define the values of the combinations $s_x = t_{13} + t_{24}$ and $s_y = t_{12} + t_{34}$, for example

$$s_x = t_{13} + t_{24} = T_{13} + T_{24} - \frac{2}{3} \left[\frac{(T_{13} + T_{24})(T_{12} + T_{34})}{(T_{13} + T_{24} + T_{12} + T_{34})} \right]$$

Finally, the ratio of t_{13} and t_{24} is chosen to be the same as the ratio of the original values T_{13} and T_{24} ,

$$t_{13} = \frac{s_x T_{13}}{(T_{13} + T_{24})}$$

$$t_{24} = \frac{s_x T_{24}}{(T_{13} + T_{24})}$$

A similar relation defines t_{12} and t_{34} . 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 can be a problem.

The option is invoked by using the [NINEPOIN](#) keyword in the RUNSPEC section. The nine-point transmissibilities will then by default be calculated across the entire grid. In ECLIPSE 100, by using the [NINENUM](#) keyword in the GRID section, you can choose to apply the scheme only in nominated regions of the reservoir. Note that the [NINENUM](#) keyword is not compatible with the Local Grid Refinement option.

It should be noted that the [NINEPOIN](#) option is specifically designed to account for grid orientation effects only 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 in the RUNSPEC section, as follows:

```
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.

47

Non-Darcy flow

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The non-Darcy feature we consider here is the Forchheimer correction which takes into account the inertia effects due to high velocity that may occur in high permeability regions, such as fractures. The Forchheimer parameter can be input by using either the `VDFLOW` or `VDFLOWR` keywords in the `PROPS` section. For a homogeneous reservoir the model writes (cgs units):

$$\frac{\partial P}{\partial x} = \left(\frac{\mu}{Kk_r A} \right) q + \beta \rho \left(\frac{q}{A} \right)^2 \quad \text{Eq. 47.1}$$

where:

- q is the volumetric flow rate
- K is the rock permeability
- k_r is the relative permeability
- A is the area through which flow occurs
- μ is the dynamic fluid viscosity
- ρ is the fluid density
- β is the Forchheimer parameter
- $\frac{\partial P}{\partial x}$ is the pressure gradient normal to the area

The Forchheimer parameter β is the user input in Forchheimer units: 1 Forch = 1 atm.s².gm⁻¹. An alternative unit for β can be obtained by noting that 1 atm = 1.01325 bars and 1 bar = 10⁶ gm.cm⁻¹.s⁻², therefore 1 Forch = 1 atm.s².gm⁻¹ = 1.01325.10⁶ cm⁻¹.

Typical values in cm⁻¹ are rather large for engineering purposes and the Forchheimer (Forch) is used in all unit sets. Dake [Ref. 30] shows a typical value of $\beta = 10^7$ cm⁻¹ = 9.86 Forch at $K = 60$ mD.

For other unit sets (FIELD, METRIC, LAB and PVT) two constants C_1 and C_2 are introduced as

$$\frac{\partial P}{\partial x} = \frac{1}{C_1} \left(\frac{\mu}{Kk_r A} \right) q + \frac{C_2}{C_1} \beta \rho \left(\frac{q}{A} \right)^2 \quad \text{Eq. 47.2}$$

When considering the flow between two grid blocks i and n we use transmissibility. For the sake of simplicity, we apply block-centered transmissibility (OLDTRAN) here to demonstrate. Thus:

$$q = C_1 F_{ND} T_{in} \frac{k_r}{\mu} \Delta P_{in} \quad \text{Eq. 47.3}$$

where:

$$T_{in} = \frac{A}{\frac{\Delta X_i}{2K_i} + \frac{\Delta X_n}{2K_n}} \quad \text{Eq. 47.4}$$

and where the **non-Darcy flow factor** is given by:

$$F_{ND} = \frac{1}{1 + C_2 \rho \frac{T_{in} k_r}{A^2 \mu} \frac{1}{2} (\beta_i \Delta X_i + \beta_n \Delta X_n) q} \quad \text{Eq. 47.5}$$

Note that the F_{ND} factor is dimensionless and gives a quantitative indication of the deviation of the flow behavior from Darcy's law for which $F_{ND} = 1$.

Unit conversion factors

Denoting by a star index other unit sets and taking advantage of some variables being expressed in the same units whatever the unit system (such as the viscosity and the β factor), the constants C_1 and C_2 are obtained as:

$$C_1 = \frac{\Delta P_{cgs}}{\Delta P^*} \frac{\Delta X^*}{\Delta X_{cgs}} \frac{K_{cgs}}{K^*} \frac{A_{cgs}}{A^*} \frac{q^*}{q_{cgs}} \quad \text{Eq. 47.6}$$

$$C_2 = \frac{\rho_{cgs}}{\rho^*} \frac{q_{cgs}}{q^*} \left(\frac{\Delta X_{cgs}}{\Delta X^*} \right)^2 \frac{K_{cgs}}{K^*} \left(\frac{A^*}{A_{cgs}} \right)^2 \quad \text{Eq. 47.7}$$

The following table gives the numerical values of these unit conversion factors.

	Metric	Field	Lab	PVT-Metric
C_1	0.00852702	0.00112712	3.6	0.00864
C_2	1.1574074E-9	3.17277463E-8	2.77777778E-7	1.1574074E-9

Table 47.1: Unit conversion factors $C1$ and $C2$

Construction of flows

From the equation for F_{ND} we have two coupled equations to solve for each phase p :

$$F_{ND} = \frac{1}{1 + C_2 \rho_p \frac{T_{in}}{A^2} \frac{k_{rp}}{\mu_p} \frac{1}{2} (\beta_i \Delta X_i + \beta_n \Delta X_n) q} \quad \text{Eq. 47.8}$$

and

$$q = C_1 F_{ND} T_{in} \left(\frac{k_{rp}}{\mu_p} \right) \Delta P_{pin} \quad \text{Eq. 47.9}$$

Substituting q into F_{ND} we obtain a quadratic equation for F_{ND}

$$BF_{ND}^2 + F_{ND} - 1 = 0 \quad \text{Eq. 47.10}$$

that can be solved to give (the negative root being ignored)

$$F_{ND} = \frac{-1 + \sqrt{1 + 4B}}{2B} \quad \text{Eq. 47.11}$$

where for simplicity we put

$$B = C_2 C_1 \rho_p \left(\frac{T_{ni}}{A} \right)^2 \left(\frac{k_{rp}}{\mu_p} \right)^2 \frac{1}{2} (\beta_i \Delta X_i + \beta_n \Delta X_n) \Delta P_{pni} \quad \text{Eq. 47.12}$$

Assumption and simplifications

As the Forchheimer correction is expected to be significant only in regions of high velocity, it is assumed that those regions have a constant rock permeability K and constant porosity φ . Under these assumptions the B -factor can be simplified to

$$B = C_2 C_1 \beta \rho_p \left(\frac{T_{ni} K}{A} \right) \left(\frac{k_{rp}}{\mu_p} \right)^2 \Delta P_{pni} \quad \text{Eq. 47.13}$$

Using the fact that in ECLIPSE the factor unit C_1 is included in the transmissibility ($T_{ni} = C_1 T_{ni}$), we obtain the final form for the B -factor used in ECLIPSE 300

$$B = C_2 \beta \rho_p \left(\frac{T_{ni} K}{A} \right) \left(\frac{k_{rp}}{\mu_p} \right)^2 \Delta P_{pni} \quad \text{Eq. 47.14}$$

The values of F_{ND} and $B / (\Delta P_{pni})$ for gas and oil may be examined using the mnemonics FFOR (G, O) and BFOR (G, O) in the [RPTSOL](#) or [RPTSCHE](#) keywords.

Note: In the current version, ECLIPSE 100 reports FFORG and BFORG, that is for the gas phase only.

Forchheimer models

In the above discussion the Forchheimer parameter (β) is a constant supplied by the user using the keywords [VDFLOW](#) or [VDFLOWR](#). This parameter may alternatively be calculated from one of two models implemented in ECLIPSE 300. The choice of model is made using the [VELDEP](#) keyword in the RUNSPEC section, with the model parameters supplied using the [VDKRG](#) and [VDKRO](#) keywords in the PROPS section.

ECLIPSE 300 only For each phase p , the models are:

Model 1:

$$\beta_p = \frac{a_p}{\varphi^{b_p} S_p^{c_p} (K k_{rvp})^{d_p}} \quad \text{Eq. 47.15}$$

Model 2:

$$\beta_p = \beta_{dp} S_p^{c_p} (K k_{rvp})^{d_p} \quad \text{Eq. 47.16}$$

where S_p is the normalized saturation of the pth phase, which is related to the true saturation $S_p^{(3P)}$ by:

$$S_p = \frac{S_p^{(3P)}}{1 - S_w} \quad \text{Eq. 47.17}$$

where S_w is the water saturation.

The set of parameters (a_p, b_p, c_p, d_p) or (β_{dp}, c_p, d_p) are, **and must be**, experimentally determined. There appears to be no real physical basis for the porosity and saturation dependency in equation 47.15 and saturation dependency in equation 47.16 other than in the original work done on the Forchheimer effect. These dependencies may be ignored by setting the values of (b_p, c_p) in the VDKRO and/or VDKRG keywords in the PROPS section to zero.

The values of β for gas and oil may be examined using the mnemonics BTFOR (G, O) in the RPTSOL or RPTSCHEK keywords.

Forchheimer parameter based well D-factors

If a Forchheimer parameter for the gas phase is calculated using one of the models specified via the VELDEP keyword, a D-factor for each production well connection is also calculated.

Note: For ECLIPSE 100, a D-factor is **not** calculated, by default, if a constant Forchheimer parameter is supplied using the keywords VDFLOW or VDFLOWR. However, if item 218 of the OPTIONS keyword is set to 2, the D-factor calculated via either the VDFLOW or VDFLOWR keywords will be applied to wells except those for which the WDFAC or WDFACCOR keywords have been specified or for well completions for which item 12 of the COMPDAT keyword (or item 13 of the COMPDATL keyword) has been specified.

Note: For ECLIPSE 300, a D-factor is **not** calculated if a constant Forchheimer parameter is supplied using the keywords VDFLOW or VDFLOWR.

For ECLIPSE 300, D-factors calculated via the VELDEP keyword override D-factors supplied using the keywords COMPDAT or WDFAC. If the WDFACCOR keyword is specified, however, then the value calculated from its correlation overrides the value calculated from equation 47.18. Calculated D-factors for well connections may be written to the summary file using the mnemonic CDFAC.

The equation for the Forchheimer parameter dependent D-factor is as follows:

$$D = \alpha \cdot \beta_g \cdot \frac{K_e}{h} \cdot \frac{1}{r_w} \cdot \frac{\gamma_g}{\mu_g} \quad \text{Eq. 47.18}$$

where:

α is a constant, the value of which depends on the units system (see table 47.2)

- β_g is the Forchheimer parameter for gas (in Forchheimer units, Forch)
- K_e is the effective permeability of the connected grid block (generally the geometric mean of permeabilities involved)
- h is the length of the connection
- r_w is the wellbore radius
- γ_g is the relative density of produced or injected gas at surface conditions with respect to air at standard temperature and pressure
- μ_g is the gas viscosity in the connection grid block

	Metric	Field	Lab	PVT-Metric
Unit	day/sm ³	day/Mscf	hr/scc	day/sm ³
Value	2.24460 x 10 ⁻¹⁰	6.83352 x 10 ⁻⁸	5.41375 x 10 ⁻¹¹	2.25533 x 10 ⁻¹⁰
Pre-2006.1 value	2.70863 x 10 ⁻¹⁸	7.67000 x 10 ⁻¹⁷	6.50072 x 10 ⁻²³	2.70863 x 10 ⁻¹⁸

Table 47.2: Unit-dependent constants for the Forchheimer parameter-dependent skin-factor

The values of the unit-dependent constant were changed in 2006.1. The pre-2006.1 values are still available for back-compatibility. To revert to the pre-2006.1 values, set item 96 of OPTIONS3 keyword to 2 or 3.

Note that equation 47.18 uses an approximation based on the assumption that the pressure equivalent radius, r_0 , of the well is much greater than the wellbore radius. This may not be a good approximation with smaller grid sizes. Equation 47.19, which does not use this approximation, can be used by setting item 96 of the OPTIONS3 keyword to 1 or 3.

$$D = \alpha \cdot \beta_g \cdot \frac{K_e}{h} \cdot \left(\frac{1}{r_w} - \frac{1}{r_0} \right) \cdot \frac{\gamma_g}{\mu_g} \quad \text{Eq. 47.19}$$

Velocity-dependent relative permeabilities

At high gas velocities, in addition to the Forchheimer correction, it is also possible to take into account more complex effects by introducing a velocity dependence of the relative permeabilities [Ref. 52]. The model may make the oil and gas relative permeabilities functions of Capillary Number (CN) according to the user-specified switches set using the [VELDEP](#) keyword in the RUNSPEC section. The properties of the velocity-dependent relative permeabilities for the oil and gas are set with the keywords [VDKRO](#), [VDKRG](#) and [VDKRC](#).

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 practice. 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.

The normal engineering practice is to combine IFT and velocity through to form the Capillary Number; several alternative definitions are available for the Capillary Number and three of these are described below. Where flow velocities are high, turbulence effects as predicted by the Forchheimer model become

significant and a model for this effect has been developed as part of this overall scheme. These models do not work with black oil runs since they are intended for gas condensate systems.

Capillary number model

The CN model has two effects on the gas and oil relative permeabilities, namely as the CN increases:

- It reduces the residual saturations
- It changes the relative permeability from the user-specified (immiscible) saturation curves towards an internally-generated miscible curve.

The CN is calculated from one of three alternate models. For phase p = (oil, gas)

Capillary number model 1

$$N_{cp}^{(1)} = \frac{v_g \mu_g}{\sigma} \quad \text{Eq. 47.20}$$

Capillary number model 2

$$N_{cp}^{(2)} = \frac{K k_{rvp} \Delta P_p}{\sigma L} \quad \text{Eq. 47.21}$$

Capillary number model 3

$$N_{cp}^{(3)} = (2\phi S_p K k_{rvp})^{1/2} \frac{\Delta P_p}{\sigma} \quad \text{Eq. 47.22}$$

where:

S_p is the normalized or HCPV saturation of hydrocarbon phase p , which is related to the grid block phase saturation $S_p^{(3P)}$ according to: $S_p = \frac{S_p^{(3P)}}{1-S_w}$

ΔP_p is the pressure drop of hydrocarbon phase p in the direction of flow

μ_g is the gas viscosity

v_g is the gas velocity in the direction of flow

σ is the gas-oil surface tension

k_{rvp} is the CN-modified relative permeability of hydrocarbon phase p (more details below)

The CN-modified relative permeabilities k_{rvp} are calculated at the previous timestep. This is discussed in more detail below.

Note: Capillary number model 1 for the oil and gas phases depends only on the gas velocity and gas viscosity. This apparent discrepancy has been verified by experiment.

Given a phase the CN calculated from equations 47.20, 47.21 or 47.22, and Normalized Capillary Number (NCN) are defined by:

$$N_{cnp} = \frac{N_{cbp}}{N_{cp}} \quad \text{Eq. 47.23}$$

where N_{cbp} is the phase Base Capillary Number (BCN). The phase BCN is a lower threshold value below which the CN has no effect on the phase relative permeabilities. As with all the parameters in this and the Forchheimer models, the phase BCN should be determined experimentally for the reservoir and fluid of interest. If this data is not available, it may be estimated using the following procedure.

Base capillary number estimation

Generally, the rock or base relative permeability curves used to describe the flow of gas and oil within the reservoir will have been generated at or near ambient conditions. Ideally, the phase CNs will have been measured from these laboratory analyses and are reported to you; in practice, 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 [$\sim 3.5 \times 10^{-5}$ m/s]. This value for gas velocity along with the low pressure gas-oil surface tension and gas viscosity can be used in equation 47.20 to estimate the oil and gas BCN in the absence of experimental data.

The effect of CN on residual saturation

Experimental evidence suggests that as the CN rises above the BCN, the residual saturation is reduced. In the CN model implemented here, this effect is modeled through the use of the saturation scaling parameter, X_p , where:

$$X_p = 1 - e^{(-m_p N_{cnp})} \quad \text{Eq. 47.24}$$

where m_p is, **and must be**, an experimentally determined parameter. Therefore, given a phase residual saturation S_{rbp} determined from routine Special Core Analysis (SCAL), the effect of the CN dependency is to change the residual saturation to:

$$S_{rbp} \rightarrow X_p S_{rbp} \quad \text{Eq. 47.25}$$

Note if the m_p -parameter is set to zero, it has the effect of setting that phase residual saturation to zero regardless of the value of the phase CN.

The effect of CN on relative permeability curves

The miscible or straight-line relative permeability curve is constructed from:

$$k_{rmp} = \frac{\left(\frac{S_p}{(1-S_{wco})} - \frac{X_p S_{rbp}}{(1-S_{wco})} \right)}{\left(1 - \frac{X_p S_{rbp}}{(1-S_{wco})} \right)} \quad \text{Eq. 47.26}$$

where S_{wco} is the connate water saturation. This can be simplified to the form:

$$k_{rmp} = \frac{(S_p - X_p S_{rbp})}{((1 - S_{wco}) - X_p S_{rbp})} \quad \text{Eq. 47.27}$$

The user-input base relative permeability curves, k_{rbp} , are modified because of the change in residual saturation through equation 47.27. The CN-modified relative permeability curve for phase p is now calculated from:

$$k_{rvp} = N_{cnp}^{1/n_p} k_{rbp} + (1 - N_{cnp}^{1/n_p}) k_{rmp} \quad \text{Eq. 47.28}$$

where n_p is given by:

$$n_p = n_{1p} S_p^{n_{2p}} \quad \text{Eq. 47.29}$$

and (n_{1p}, n_{2p}) are, **and must be**, experimentally determined parameters.

Explicit calculation of capillary number dependency

From equations 47.20, 47.21 and 47.22, we see that the CN is a function of the phase relative permeability k_{rvp} : note equations 47.20 and 47.21 are effectively equivalent assuming the second expression is only applied to the gas phase. However, from equations 47.24, 47.26 and 47.28, we see k_{rvp} is a function of CN.

It is possible to write this dependency as a non-linear equation that can be solved in a fully implicit scheme and it is planned to do this in a future release. For now, however, we calculate the phase CN and NCN at the end of the last 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.

Alternative model for capillary number effects

An alternative model for CN effects, based on the work of Whitson, Fevang and Saevareid [Ref. 128], is also available. This model is designed for near wellbore flows exhibiting condensate ‘blockage’ type behavior in a gas condensate reservoir. It may be activated via item 5 of the **VELDEP** keyword in the RUNSPEC section.

This model is more straightforward than the above in that:

- There is no effect on residual saturations
- There is no lower threshold or base capillary number value. (Below which the CN has no effect on the gas relative permeabilities.)
- It depends on only two parameters, both of which have default values.

In the implementation in ECLIPSE 300, the CN-modified gas relative permeability, k_{rg} , is an admixture of a straight-line miscible relative permeability, $k_{rg} M$, and the user-input immiscible, rock relative permeability, $k_{rg} I$; this admixture is controlled by a CN dependent transition function, f_I :

$$k_{rg} = f_I k_{rgI} + (1 - f_I) k_{rgM} \quad \text{Eq. 47.30}$$

The calculation of the miscible relative permeability is discussed later. The transition function depends on the gas capillary number, N_{cg} , and is given by:

$$f_I = \frac{1}{(\alpha \cdot N_{cg})^n + 1} \quad \text{Eq. 47.31}$$

where

$$\alpha = \alpha^0 / k_{rg} \quad \text{with} \quad k_{rg}^- = \frac{k_{rgM} + k_{rgI}}{2} \quad \text{Eq. 47.32}$$

and α^0 is a constant depending only on rock properties:

$$\alpha^0 = \frac{\alpha_c^0}{\sqrt{K \cdot \varphi}} \quad \text{Eq. 47.33}$$

where K is the rock permeability and φ is the porosity.

The gas capillary number is calculated from model 1, equation 47.20, using a pore gas velocity:

$$v_{pg} = \frac{v_g}{\varphi \cdot (1 - S_w)} \quad \text{Eq. 47.34}$$

This model depends on two parameters: the exponent n in equation 47.31 and the coefficient α_c^0 in equation 47.33. These parameters are defaulted to 0.65 and 1.0E4 respectively, with the keyword **VDKRGC** permitting you to override these values.

Calculation of the miscible gas relative permeability

In the well inflow generalized pseudo-pressure option, at each pseudo-pressure below the saturation pressure, the model iterates to find a gas saturation such that the ratio of the **immiscible** mobilities (that is, using immiscible relative permeability) of oil and gas is equal to the ratio of the total liquid moles (L) to total vapor moles (V). Hence:

$$\frac{k_{roI} b_o}{\mu_o} \frac{\mu_g}{k_{rgI} b_g} = \frac{L}{V} \quad \text{Eq. 47.35}$$

The ratio of the immiscible relative permeabilities is calculated at this gas saturation,

$$R = k_{rgI} / k_{roI} \quad \text{Eq. 47.36}$$

Note that, in block-to-block flow calculations, the gas saturation is available so k_{rgI} and k_{roI} are calculated directly at that saturation in order to calculate the ratio R .

The miscible gas relative permeability is calculated from

$$k_{rgM} = k_{rg}^0 \left\{ \frac{1}{1 + (1/R)} \right\} \quad \text{Eq. 47.37}$$

where k_{rg}^0 is the gas relative permeability at $S_g = 1 - S_{wco}$ (and S_{wco} is the connate water saturation).

Pre-2007.1 behavior

Pre-2007.1, the miscible gas relative permeability was calculated using a straight line curve of the form, equation 47.26 but with no change in the residual saturation (that is, X_g has a value of unity). This behavior can be restored (instead of using equation 47.37) by setting item 99 of OPTIONS3 to a non-zero value.

Calculation of the oil relative permeability

The calculation of the oil relative permeability depends on the value of item 5 in the **VELDEP** keyword.

- If VELDEP item 5 has a value of 1, then

$$k_{ro} = f_{oI} k_{roI} + (1-f_{oI}) k_{roM}$$

where the miscible oil relative permeability is of the form given by equation 47.26 but with no change in the residual saturation (that is, X_o has a value of unity), and the immiscible-miscible transition function, f_{oI} , is the same as that used in the original model for capillary number effects in oil flows (see equation 47.28) so

$$f_{oI} = N_{cnP}^{1/n_p}$$

The calculation of f_{oI} requires n_{1o} , n_{2o} and N_{cbo} (and the selection of the capillary number model using VELDEP item 1). Hence it increases the number of parameters required by the alternative model from 2 to 5.

- If VELDEP item 5 has a value of 2, then

$$k_{ro} = k_{rg} / R$$

with R calculated using equation 47.36. This is the preferred method for achieving the desirable property that the alternative model only depends on two parameters.

Pre-2007.1 behavior of the alternative model

Pre-2007.1, the alternative model did not use equation 47.35, but instead found a gas saturation such that the ratio of the **modified** mobilities of oil and gas is equal to the ratio of the total liquid moles (L) to total vapor moles (V). That is

$$\frac{k_{ro} b_o}{\mu_o} \frac{\mu_g}{k_{rg} b_g} = \frac{L}{V}$$

where

b_p is the phase molar density and

μ_p is the phase viscosity.

This pre-2007.1 behavior can be recovered by setting item 99 of OPTIONS3 to a non-zero value.

Notes

- This alternative model for CN effects in gas flows cannot be used at the same time as the original CN model for **gas** flows. If VELDEP keyword item 5 is set to 1, then the alternative model can be used in conjunction with the original model for CN effects in **oil** flows. If VELDEP keyword item 5 is set to 2, then it cannot be used with the original model. The selection of models is controlled by the VELDEP keyword.
- Both CN and Forchheimer effects may be included in the generalized pseudo pressure option that calculates a blocking factor due to near well condensate dropout (see "Special inflow equations"). This is controlled by the PICOND keyword.

48

Non-Newtonian fluid flows

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

Simulation of non-Newtonian fluid flows requires a significant change to the traditional Darcy flow equation. Indeed the rheology of the fluid, which can be described by an apparent or effective viscosity, may be a non-linear function of the flow rate and may as well depend on the rock properties such as the permeability and the porosity. In order to remain as close to the Darcy form as we can, we introduce a multiplier B that modifies the mobility of the fluid depending the cell properties and local flow rate. The solution of the non-linear equation is obtained by local Newton iteration backed up by a secant method.

The multiphase Darcy equation is therefore modified as follows:

$$Q = AK(k_r / \mu)B\Delta P \quad \text{Eq. 48.1}$$

where B is **positive** and may be a complex function involving grid properties, flow rate or pressure drop, and the rheology of the fluid (models are provided below):

$$B = B(d, K, \varphi, Q, |\Delta P|, n, \tau, \delta, \beta) \quad \text{Eq. 48.2}$$

The rheology (n, τ, δ, β) is set to depend on the polymer concentration (used as a tracer) that allows having a distributed rheology in the reservoir.

As B is dependent on the flow rate, we need to solve the flow between two adjacent cells and the flow at the connections between the wells and the reservoir.

Simulation of Non-Newtonian fluid flows

Flow between two cells in the reservoir

The flow between two adjacent cells I and j in the reservoir in a given direction is the solution of the following flow equation:

$$\Delta P_{ij} = \frac{\mu_{ij}}{k_{rij}} \left[\frac{d_i}{AK_i} \frac{Q}{B_i(\dots, Q, \dots)} + \frac{d_j}{AK_j} \frac{Q}{B_j(\dots, Q, \dots)} \right] \quad \text{Eq. 48.3}$$

where k_{rij} / μ_{ij} is the upstream mobility. This equation is solved numerically using a Newtonian algorithm backed up with a secant method. The solution of this equation leads to:

$$Q = \begin{cases} \frac{k_{rij}}{\mu_{ij}} T^* \Delta P_{ij} & \text{if } (B_i > 0) \text{ and } (B_j > 0) \\ 0 & \text{otherwise} \end{cases} \quad \text{Eq. 48.4}$$

where the new transmissibility T^* is

$$T^* = \frac{1}{\frac{d_i}{AK_i} \frac{Q}{B_i(\dots, Q, \dots)} + \frac{d_j}{AK_j} \frac{Q}{B_j(\dots, Q, \dots)}} \quad \text{Eq. 48.5}$$

Flow at the well connection

For the flow between the well and the reservoir grid block, we assume that there is no preferred direction for the non-Newtonian fluid. Under that simplification we keep the connection factor unchanged, so the well flow rate writes:

$$Q = T_w M B \Delta P \quad \text{Eq. 48.6}$$

where

T_w is the well connection factor

M is the mobility

B is the non-Newtonian modifier, function of the flow rate Q

ΔP is the pressure drop (draw)

Equation 48.6 is also solved by local Newtonian iteration backed up with a secant method to find the flow rate and therefore the right modifier value.

Note: The assumption that there is no preferred flow path at the connection between the well and the grid block containing the well may become questionable if the degree of anisotropy (ratio between the permeabilities in the plane containing the completion) becomes significant. Because of the shear thinning effect the injectivity can become direction dependent.

Unit conversion

Equations 48.1 and 48.6 are only valid when expressed in cgs units. In addition to the unit factor conversion used for Newtonian fluids, we need to introduce another coefficient to take into account the conversion for the B factor. For other units systems used in ECLIPSE we therefore introduce a coefficient C_u such as:

$$C_u = \frac{B_{cgs}}{B^*} \quad \text{Eq. 48.7}$$

where

B_{cgs} is the non-Newtonian modifier expressed in cgs units and

B^* is expressed in other units.

Non-Newtonian model: Herschel-Bulkley fluids

Herschel-Bulkley fluids are described by the following non-Newtonian modifier

$$B = \begin{cases} \frac{\left(1 - \frac{d\alpha\tau}{|\Delta P|}\right)}{\frac{1}{12}\left(9 + \frac{3}{n}\right)^n(72\delta K)^{(1-n)/2}} \left(\frac{|Q|}{A}\right)^{1-n} & \text{if } \left(\frac{d\alpha\tau}{|\Delta P|}\right) < 1 \\ 0 & \text{otherwise} \end{cases}$$

where:

Q is the volumetric flow rate

K is the rock permeability

n is the power index $0 < n \leq 1$

A is the area through which flow occurs

d is the distance between cells

τ is the yield stress

α is a weighing factor for the yield. $\alpha = \sqrt{\frac{\varphi}{2}} \sqrt{\frac{\delta}{K}}$, with the porosity φ

δ is the tortuosity

ΔP is the pressure difference between cells

As mentioned earlier the rheological parameters n , τ , δ are function of the polymer concentration to allow spatial distribution of the fluid properties. The unit conversion C_u is now also dependent on n

$$C_u = \left(\frac{Q_{cgs}}{Q^*}\right)^{1-n} \left(\frac{A^*}{A_{cgs}}\right)^{1-n} \left(\frac{K_{cgs}}{K^*}\right)^{(1-n)/2} \quad \text{Eq. 48.8}$$

	Metric	Field	Lab
C_u (HB.)	$\left(\frac{\sqrt{10}}{86.4}\right)^{1-n}$	$\left(\frac{541.16844955}{8640}\right)^{1-n}$	$\left(\frac{\sqrt{10}}{6}\right)^{1-n}$

Table 48.1: Units of conversion, Herschel-Bulkley fluids

Usage

This model applies only to aqueous phase (water and polymer). To activate the model the following keywords are required:

Section	Keyword	Description
RUNSPEC	POLYMER	activate the polymer flood option
RUNSPEC	NNEWTF	activate the Herschel-Bulkley model and provides some table sizes
	FHERCHBL	provides the look up tables for the rheological parameters as a function of the polymer concentration.

49

Optimization

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

This chapter describes the optimization facilities within ECLIPSE.

Several aspects of an engineering study involve the optimization of an ECLIPSE model. Increasingly, these optimization tasks are being incorporated into software programs, often using the simulator as a 'black box' function evaluation in an outer optimization loop. Optimization methods are readily divided into gradient and non-gradient methods. Methods that do not use gradients of the objective function with respect to model parameters, such as simulated annealing, are typically only applied to greatly simplified models, as they require a great many runs of the simulator. However, even methods that do use gradient information can be prohibitively expensive if the gradients are derived numerically, from multiple runs of the 'black box'.

ECLIPSE production profile gradients can be calculated much more efficiently from within the simulator itself (see "[Gradient option](#)"). The SimOpt interactive application successfully makes use of these ECLIPSE gradients in an external gradient-based optimization loop, as an aid in the history matching phase of a study.

Optimization capabilities have also been built into ECLIPSE 300, which use gradients of a general objective function rather than the production profiles. The gradients are calculated using the Adjoint technique, which has significant performance advantages when the number of parameters is large. Production constraints are handled using a Lagrangian formulation of the problem, which also provides an enormous speed up compared to treatments external to ECLIPSE.

The sections that follow give a technical description of the various aspects of the optimization workflow environment within ECLIPSE 300. This is followed by some guidelines on the use of the optimization feature, a tabulated list of optimization keywords, and subsequently some examples of their use in the ECLIPSE 300 reservoir optimization option.

Non-linear gradient-based optimization

Optimization structure

The central task of optimization is to maximize some objective function, J , by variation of a set of control parameters, λ . An example optimization problem is that of reservoir optimization, where J can be simply the total oil produced and λ are the controlling well rates during a simulation run.

This optimization task is invariably iterative. For gradient-based optimization, this can be represented schematically as shown in the following figure.

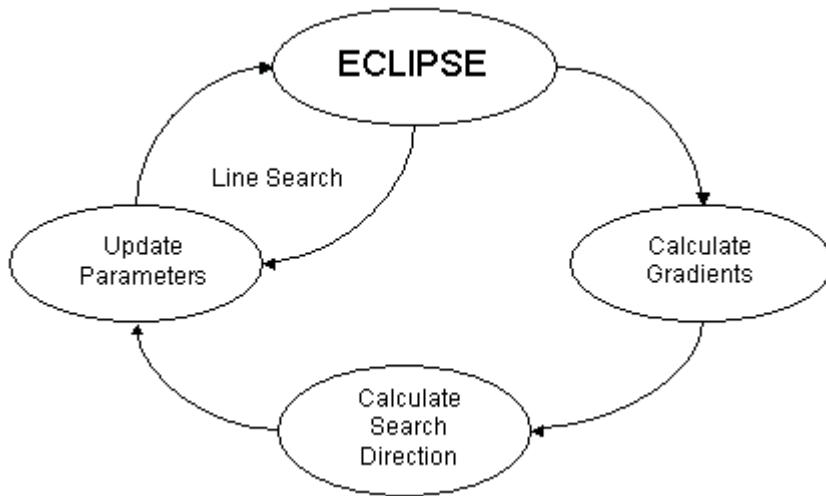


Figure 49.1. Schematic representation of gradient based optimization

This iterative structure is based on obtaining a search direction, s , from a set of gradients of the objective function with respect to the control parameters, $dJ/d\lambda$. A line search is performed using this initial search direction to find the optimum step size at which point the line search is terminated and a new direction calculated. This loop is repeated until one (or more) termination criteria are met. These termination criteria include lower limits on the relative changes of J or of λ .

This optimization loop structure has been implemented in ECLIPSE 300, and is controlled by keyword definitions given in the OPTIMIZE section of the ECLIPSE 300 data file.

Impact of constraints

The presence of constraints within an optimization greatly increases the complexity of the problem [Ref. 100]. Constraints may vary from simple bounds on parameter values to full non-linear functional inequality relationships between parameters and ECLIPSE outputs, for example, the well and group production constraints.

The former are dealt with relatively easily, by a restriction of the step during a line search. However, the latter require modifications to the search direction.

Inequality constraints are most frequently found in the form $f(x, y) \leq K$, which may be rewritten as:

$$f(x, y) - K = V \leq 0 \quad \text{Eq. 49.1}$$

A constraint of this form is said to become “active” when the inequality becomes an equality, that is when $V = 0$. Otherwise the constraint is said to be “inactive”.

A simple example that illustrates the impact of constraints is shown in the following figure.

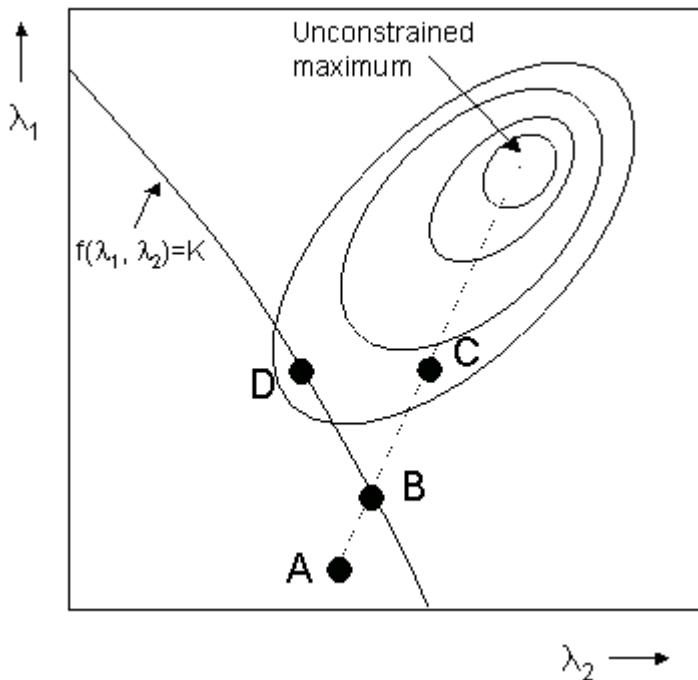


Figure 49.2. An Example of constrained optimization.

This diagram shows the objective function contour map for a problem with two variables, λ_1 and λ_2 , in which a nonlinear inequality constraint function $f(\lambda_1, \lambda_2) \leq K$ divides the parameter space into feasible (bottom left) and infeasible (top right) regions. From a starting position at A using a search direction indicated by the dashed line, a line search would have to terminate at B (to avoid violating the constraint). From B, an unconstrained search direction would point towards C, which violates the constraint. The direction we require is towards the constrained maximum at D.

The Lagrangian formulation is a well established mechanism for combining an objective function with equality constraints. For the example above, we form the following Lagrangian function: [Ref. 100]

$$L(\underline{\lambda}) = J(\underline{\lambda}) - \Phi V(\underline{\lambda}) \quad \text{Eq. 49.2}$$

For suitable values of the Lagrange multiplier, Φ , unconstrained optimization of the Lagrangian simultaneously achieves constrained optimization of the original objective function.

The Lagrangian formulation relies on being able to determine which inequality constraints are active, that is which are currently an equality constraint. As the above illustration indicated, a specialized line search algorithm can be used for just such a purpose. However, one potential problem of this approach is that it may dominate the optimization convergence - at worst we may end up following the Simplex algorithm of linear programming, with one iteration per constraint.

"[Adjoint gradient calculations in ECLIPSE](#)" shows how we use the Lagrangian formulation to construct the optimization gradients using the Adjoint method.

Adjoint gradient calculations

After a simulation run, the optimization workflow starts with the calculation of the gradients of the objective function with respect to the chosen parameters. These gradients must take into account the equality constraints imposed by the fluid flow calculations solved in ECLIPSE.

At each simulator timestep, ECLIPSE forces the fully implicit residual in each simulator grid block to be zero (see "Formulation of the Equations").

CAUTION: The adjoint gradient calculations require a fully implicit formulation of the equations. If you try to use the reservoir optimization feature without this type of formulation, the run will be aborted.

This cell residual constraint equation may be written as

$$\mathbf{R}(\mathbf{X}^{n+1}, \mathbf{X}^n) = \frac{\mathbf{M}(\mathbf{X}^{n+1}) - \mathbf{M}(\mathbf{X}^n)}{dt} + \mathbf{F}(\mathbf{X}^{n+1}) + \mathbf{Q}(\mathbf{X}^{n+1}) = 0 \quad \text{Eq. 49.3}$$

where

the superscripts $n+1$ and n denote the current and previous timestep respectively, and

- | | |
|----------------------------|---|
| \mathbf{X}^n | are the solution variables (pressure and molar densities for example) at time n |
| $\mathbf{M}(\mathbf{X}^n)$ | is the fluid-in-place in each simulator grid block at time n , |
| $\mathbf{F}(\mathbf{X}^n)$ | is the net flow rate into neighboring grid blocks at time n , and |
| $\mathbf{Q}(\mathbf{X}^n)$ | is the net flow rate into wells at time n . |

Note that the objective function is a function of both the control parameters λ and the simulation primary variables X ; that is $J = J(X, \lambda)$.

A Lagrangian formulation of the objective function is used in order to comply with the constraints imposed by the cell residuals,

$$L(\mathbf{X}, \lambda) = J(\mathbf{X}, \lambda) - \Phi \cdot \mathbf{R}(\mathbf{X}, \lambda) \quad \text{Eq. 49.4}$$

Note that when all the residuals are satisfied ($R = 0$) the Lagrangian is simply the objective function. The vector of Lagrange multipliers on the residuals, Φ , is known as the “adjoint vector”.

We are interested in forming the total parameter derivatives of this Lagrangian:

$$\frac{dL}{d\lambda} = \frac{\partial L}{\partial \lambda} + \frac{\partial L}{\partial \mathbf{X}} \cdot \frac{\partial \mathbf{X}}{\partial \lambda} \quad \text{Eq. 49.5}$$

for which we require the partial Lagrangian derivatives:

$$\frac{\partial L}{\partial \mathbf{X}} = \frac{\partial J}{\partial \mathbf{X}} - \left(\Phi \cdot \frac{\partial \mathbf{R}}{\partial \mathbf{X}} \right) \quad \text{Eq. 49.6}$$

and

$$\frac{\partial L}{\partial \lambda} = \frac{\partial J}{\partial \lambda} - \left(\Phi \cdot \frac{\partial \mathbf{R}}{\partial \lambda} \right) \quad \text{Eq. 49.7}$$

as well as the solution derivatives $\frac{\partial \underline{X}}{\partial \underline{\lambda}}$.

Since $R = 0$, we are free to choose our vector of Lagrange multipliers in any way we choose.

As there is an implicit dependence of the solution variables on the control parameters, $\underline{X} = \underline{X}(\underline{\lambda})$, the natural choice of $\underline{\Phi}$ is that which sets equation 49.6 to zero. This removes the need to calculate the solution derivatives. Hence we solve the adjoint set of linear equations:

$$\underline{\Phi} \cdot \frac{\partial R}{\partial \underline{X}} = \frac{\partial J}{\partial \underline{X}} \quad \text{Eq. 49.8}$$

for the adjoint vector $\underline{\Phi}$. The dimension of the adjoint vector for each time period is that of the normal fluid flow Jacobian (number of grid cells times number of simulation variables).

As an example, scoping equation 49.8 into four distinct time periods, it may be written as:

$$[\underline{\Phi}^1 \ \underline{\Phi}^2 \ \underline{\Phi}^3 \ \underline{\Phi}^4] \begin{bmatrix} A^{11} & 0 & 0 & 0 \\ A^{21} & A^{22} & 0 & 0 \\ 0 & A^{32} & A^{33} & 0 \\ 0 & 0 & A^{43} & A^{44} \end{bmatrix} = \left[\frac{\partial J}{\partial \underline{X}^1} \frac{\partial J}{\partial \underline{X}^2} \frac{\partial J}{\partial \underline{X}^3} \frac{\partial J}{\partial \underline{X}^4} \right] \quad \text{Eq. 49.9}$$

where the A matrices are derivatives of the fluid flow residuals with respect to solution variables, given by

$$A^{mn} = \frac{\partial \underline{R}^m}{\partial \underline{X}^n} \quad \text{Eq. 49.10}$$

where m and n are time indices. By multiplying out $A^{mn} = \frac{\partial \underline{R}^m}{\partial \underline{X}^n}$, equation 49.9 term by term the general equation, which must be solved at any given time, index n is

$$\underline{\Phi}^n A^{nn} = \frac{\partial J}{\partial \underline{X}^n} - \underline{\Phi}^{n+1} A^{n+1,n} \quad \text{Eq. 49.11}$$

This involves the value of the adjoint vector at time period, $n+1$. Since the only independent equation is the last one (for which we can solve for) then the adjoint $\underline{\Phi}^4$ equation, equation 49.11 must be solved backwards in time.

With the adjoint vector, $\underline{\Phi}$, chosen as the solution of this system, the Lagrangian gradients we require are:

$$\frac{dL}{d\underline{\lambda}} = \frac{\partial J}{\partial \underline{\lambda}} - \left(\underline{\Phi} \cdot \frac{\partial \underline{R}}{\partial \underline{\lambda}} \right) \quad \text{Eq. 49.12}$$

Note: The effort of calculating these gradients is dominated by the solution for the adjoint vector, $\underline{\Phi}$, which is independent of the type or number of parameters. This makes the adjoint method very efficient for large number of parameters compared to other gradient calculation techniques.

Introducing Additional Constraints

In the reservoir simulator the fluid flow residual equations ($\underline{R} = 0$) are constraints, which must always be observed as the objective function is optimized. Optionally, you may impose additional constraints on the system, for example limiting the production of a certain phase from a group of wells.

The Lagrangian approach allows us to readily incorporate these additional equality constraint equations, $\mathbf{V} = \mathbf{0}$, by the introduction of further Lagrange multipliers, Ψ :

$$L(\mathbf{X}, \underline{\lambda}) = J(\mathbf{X}, \underline{\lambda}) - \Phi \cdot \mathbf{R}(\mathbf{X}, \underline{\lambda}) - \Psi \cdot \mathbf{V}(\mathbf{X}, \underline{\lambda}) \quad \text{Eq. 49.13}$$

The Lagrangian derivatives given in equations 49.5 to 49.7 above may be extended to include derivatives of the additional constraint functions. This allows us to form an extended adjoint system of equations, from which we obtain both Φ and Ψ . The Lagrangian gradient equations, 49.6 and 49.7, then become:

$$\frac{dL}{d\mathbf{X}} = \frac{\partial J}{\partial \mathbf{X}} - \left(\Phi \cdot \frac{\partial \mathbf{R}}{\partial \mathbf{X}} \right) - \left(\Psi \cdot \frac{\partial \mathbf{V}}{\partial \mathbf{X}} \right) \quad \text{Eq. 49.14}$$

$$\frac{dL}{d\underline{\lambda}} = \frac{\partial J}{\partial \underline{\lambda}} - \left(\Phi \cdot \frac{\partial \mathbf{R}}{\partial \underline{\lambda}} \right) - \left(\Psi \cdot \frac{\partial \mathbf{V}}{\partial \underline{\lambda}} \right) \quad \text{Eq. 49.15}$$

Notice that we have retained the special identity of the cell residual equations, as distinct from all other constraints. This allows us to solve for the Lagrange multipliers using modifications to the existing simulation fluid flow matrix A , rather than having to construct an augmented system.

In order to solve for Ψ , we arbitrarily set $\frac{\partial L}{\partial \underline{\lambda}_i} = 0$ in equation 49.15 for as many parameters, index i , as there are active constraints. Consequently, equation 49.15 only applies to those parameters not used in determining Ψ . This effectively divides the parameter set into 'free' and 'dependent' subsets: $\underline{\lambda} = (\underline{\lambda}_f, \underline{\lambda}_d)$, where the free parameters are those for which we have Lagrangian gradients, and the dependent parameters are those which 'depend' on the free parameters using the constraint equations. This has important implications for the optimization workflow, as discussed in "[Optimizer search direction](#)".

Optimizer search direction

Given the Lagrangian gradients, $dL / d\underline{\lambda}$, the optimization workflow then proceeds by calculating a search direction, s .

However, in the presence of equality constraints other than the cell residual equations, we do not have Lagrangian gradients for dependent parameters, $\underline{\lambda}_d$. The usual course of action is to determine the 'free' search direction, s_f , update the free parameter values, $\underline{\lambda}_f$, and then invert the constraint equations to update $\underline{\lambda}_d$. Unfortunately, since the constraint equations are most generally non-linear, this inversion is iterative, and expensive, since each function evaluation requires a simulation run.

Instead, changes in the dependent parameters are determined from changes in the free parameters using a linearization of the perturbed constraint equations:

$$\frac{\partial \mathbf{R}}{\partial \mathbf{X}} \cdot \Delta \mathbf{X} \frac{\partial \mathbf{R}}{\partial \underline{\lambda}_d} \cdot \Delta \underline{\lambda}_d + \frac{\partial \mathbf{R}}{\partial \underline{\lambda}_f} \cdot \Delta \underline{\lambda}_f = 0 \quad \text{Eq. 49.16}$$

$$\frac{\partial \mathbf{V}}{\partial \mathbf{X}} \cdot \Delta \mathbf{X} \frac{\partial \mathbf{V}}{\partial \underline{\lambda}_d} \cdot \Delta \underline{\lambda}_d + \frac{\partial \mathbf{V}}{\partial \underline{\lambda}_f} \cdot \Delta \underline{\lambda}_f = 0 \quad \text{Eq. 49.17}$$

With $\Delta \underline{\lambda}_f = s_f$, we solve this system for $s_d = \Delta \underline{\lambda}_d$, thus obtaining the full search direction. In solving the above, our boundary conditions are that $\Delta \mathbf{X} = 0$ at time zero, and so these equations must be solved forwards in time.

The modified workflow for determining the optimization search direction now consists of two distinct parts.

- First the search direction for the free parameters is determined during the backwards equation solution.

- Then the free parameter search direction is used to determine the search direction for the set of dependent parameters in a forward solution of the system.

Note: In this release, the calculation of the search direction from the gradients is performed using either a steepest descent or a conjugate gradient algorithm. The conjugate gradient algorithm requires slightly more computational work and storage. Due to the presence of the constraints, our experience is that there is not much difference in the convergence rate of the two methods, although the conjugate gradient algorithm does appear to give a more stable convergence path.

Line search

Given a search direction, s , the standard line search algorithm is to determine some step size multiplier, α , which maximizes the objective function value. The control parameters are updated using the algorithm $\lambda = \lambda + \alpha s$. The following figure illustrates the problem.

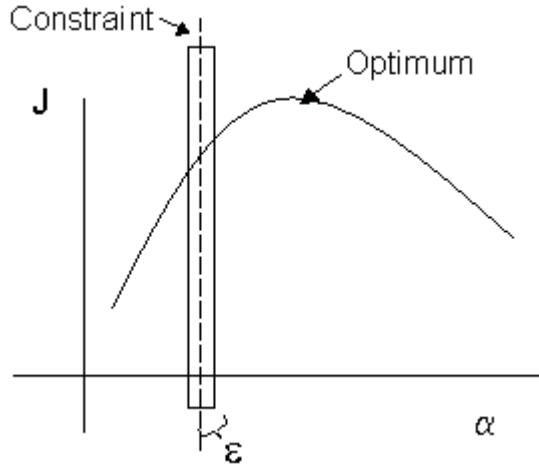


Figure 49.3. Line search

With no preferred scale, a line search may take many iterations to find the optimum position. However, along the way, we may encounter, or jump over, a new constraint. An indication of the position of the nearest constraint to the starting position of a line search can be obtained by expanding the constraint inequalities to give:

$$V_i(\mathbf{X} + \alpha_i \Delta \mathbf{X}, \underline{\lambda}_f + \alpha_i \Delta \underline{\lambda}_f) = V_i(\mathbf{X}, \underline{\lambda}) + \alpha_i \left(\frac{\partial V_i}{\partial \mathbf{X}} \cdot \Delta \mathbf{X} \right) + \alpha_i \left(\frac{\partial V_i}{\partial \underline{\lambda}_f} \cdot \Delta \underline{\lambda}_f \right) \leq 0 \quad \text{Eq. 49.18}$$

Clearly, the closest constraint (that which would be first violated if an unrestricted amount of the search direction were used) is that for which the value of α is smallest when the inequality becomes an equality constraint. This may be obtained by rearranging equation 49.18, from which:

$$\alpha_i = -V_i \left| \left(\frac{\partial V_i}{\partial \underline{\lambda}_f} \cdot \Delta \underline{\lambda}_f + \frac{\partial V_i}{\partial \mathbf{X}} \cdot \Delta \mathbf{X} \right) \right| \quad \text{Eq. 49.19}$$

This sets a useful guide for the line search algorithm in the presence of constraints, and in practice can save many iterations of the line search.

As a further aid to convergence of the overall optimization loop, the tolerance used in determining the proximity of a constraint, ε , can be specified to potentially ‘catch’ several new constraints at once.

Reservoir optimization

The ECLIPSE 300 reservoir optimization option allows you to optimize production by variation of well-based parameters. The option is triggered by the `RESOPT` keyword in the RUNSPEC section. This activates the processing of the OPTIMIZE section of a data set, in which you can specify particular forms of the objective function, parameters and constraints appropriate to the reservoir optimization task. These are described below.

The objective function

The reservoir optimization objective function is specified using the `OPTFUNC` keyword. This keyword permits specification of a very general function of the form:

$$J = \sum_i C_i \sum_{t_1(i)}^{t_2(i)} d_i(t) f_i(t) \Delta t \quad \text{Eq. 49.20}$$

where

$f_i(t)$ denotes a produced or injected fluid rate (for example, field oil production rate),

t_1 and t_2 are restart indexes defining the time range of interest,

C_i is a function-dependent weighting coefficient (whereby for example water production could be penalized), and d_i is a discount factor, given by:

$$d_i(t) = (1 + r_i)^{-N(t)} \quad \text{Eq. 49.21}$$

for annual discount rate r_i and simulation time $N(t)$ in fractional years.

The time limits appearing on the second summation allow functions to be included in the total objective function at different times with different weights, and different discount factors.

Control parameters

Parameters are specified using the `OPTPARS` keyword. In this release, the available parameters are:

- Well bottom hole pressures
- Well oil and gas production target rates
- Well oil, water and gas injection target rates
- The segment cross sectional area of a multisegment well

As an example, consider the bottom hole pressure at which a well is operated as a parameter. In general the value of this parameter varies over the course of a simulation, resulting in many control parameters for this well, one for each period of time. A general parameter is therefore denoted λ_p^t , where p identifies a specific control parameter (for example, bottom hole pressure for a particular well), and t scopes the parameter in time.

Parameters are created using the time axis defined by restart files. Thus, to vary well rates quarterly, restart files must be requested at three month intervals in the data set. For m user specified parameters, and with the number of restart files defined as n , the complete vector of control parameters for the problem is thus:

$$\underline{\lambda} = [[\lambda_1^1 \ \lambda_1^2 \ \dots \ \lambda_1^n] \ [\lambda_2^1 \ \lambda_2^2 \ \dots \ \lambda_2^n] \ \dots \ [\lambda_m^1 \ \lambda_m^2 \ \dots \ \lambda_m^n]] \quad \text{Eq. 49.22}$$

This vector is reported in the PRINT file during an optimization run.

Note: In reservoir optimization, the total number of control parameters can be very high. For example, in a large study with perhaps 1000 wells under optimizer control, and with settings to be changed four times a year over a twenty-five year period, there are a total number of 105 control parameters. Thus an efficient gradient calculation scheme, such as the adjoint method, is critically important.

Production constraints

Production inequality constraints are specified using the [OPTCONS](#) keyword. The Reservoir Optimization feature allows you to specify constraints in the general form:

$$f(\underline{X}, \underline{\lambda}) \leq K \quad \text{Eq. 49.23}$$

where

\underline{X} are the reservoir solution variables,

$\underline{\lambda}$ is the vector of control parameters,

K is a constant, and

f is a fluid production or injection rate, from either a group or a well.

You may also specify the time period for which the constraint is to apply, defaulted to the entire simulation period.

File output

The reservoir optimization option writes output to the ECLIPSE PRINT file. For each iteration of the optimization procedure, the following are reported:

- Objective function value
- Parameter values
- Constraint values
- A list of constraints which are currently active
- Optimizer search direction
- Line search details.

As an option the results of each optimization iteration may be output to separate summary and restart files for visualization by ECLIPSE Office. This is achieved by using the [RPTOPT](#) keyword mnemonic SEPARATE.

Optimizer performance

Parameter selection

It is strongly recommended that, from the available selection of optimization parameters, you try the bottom hole pressure parameters first.

The reason for this is that production rates are insensitive to well target rates whenever a well falls onto bottom hole pressure control. With bottom hole pressures as parameters, the production rates are only insensitive when a well is shut, which happens less frequently, and thus optimizations performed with such parameters are usually more successful.

Adjoint gradient accuracy

In general the most accurate gradients are obtained when the adjoint functions are integrated over small timesteps. The adjoint solution is generated using restart files, so it is best to request frequent restarts.

In principle, the obvious timesteps to use are those used in the forward simulation. Unfortunately, the variation of timesteps between optimizer iterations makes this unsuitable for our purposes, unless the parameters are defined on a separate time axis. Thus, for example, the adjoint functions could be integrated at the timestep level, whilst the parameters are defined at some coarser scale. The presence of the constraint functions precludes this as a practical technique; since with fewer parameters than constraint functions, the adjoint system is under determined, making gradient calculations impossible.

Parameter bounds

It is quite important to set realistic parameter bounds for the optimization, using the parameter definition under the `OPTPARS` keyword. This prevents unphysical (and unacceptable) values from occurring, and also prevents the simulator from going into tricky convergence regions.

Termination criteria

It is important to set the optimizer tolerances appropriately using the `OPTTUNE` keyword. These have been set with broadly reasonable defaults but, if they are changed, then care should be taken not to set them too tight - for example, by allowing termination only when the objective function is changing by too small a fraction.

Guidelines on using the reservoir optimization feature

This section gives you some guidelines on how to use the reservoir optimization feature.

Note: The starting point for the use of the reservoir optimization feature is that you should have an ECLIPSE 300 model that runs without any problems.

Screening your model for suitability

The first thing you should do is check your model for its compatibility with the requirements of the Reservoir Optimization feature.

Features that the model must include

It should be fully implicit.

It should have reasonably frequent report step output; monthly report output is a good rule-of-thumb.

The adjoint method in the reservoir optimization feature uses report steps, as opposed to timesteps, to calculate the required gradients, so the gradient calculation will lose accuracy if the report steps are too far apart.

Features that the model should not include

- The use of the parallel option ([PARALLEL](#) keyword).
- The use of fast restarts ([LOAD](#) keyword).
- The use of local grid refinements ([CARFIN](#), [EXTFIN](#) or [RADFIN](#) keywords).
- The use of grid coarsening ([COARSEN](#) keyword).
- The use of the sector modelling option ([SECTOR](#) keyword).
- The use of the thermal option ([THERMAL](#) keyword).
- AIM, IMPES or IMPSAT formulations.
- Any well controls which will compete against the optimizer, for example [ACTION](#) keywords, [WECON](#).

If necessary (and possible) you need to decide if it is possible for you to modify your model to meet these requirements. If you change your model, make sure that it can still be run successfully in ECLIPSE 300 and that these changes do not affect the results of the simulation dramatically.

If your model is large, then it may be useful to construct a coarse version so that you can determine if the Reservoir Optimization feature gives useful results, before attempting the use the feature on the detailed model.

Deciding the optimization problem you want to solve

You need to decide what objective you are trying to achieve, for example, maximizing discounted oil production total while minimizing water production total.

You also need to decide what constraints (if any) you need to impose while trying to achieve this objective, for example, the water cut must never rise above 10% in certain wells and groups.

Finally you must decide which well control parameters you want to use to try to achieve your constrained objective.

- Well target production/injection rates are only good for optimization purposes as long as the well stays on target production. Once the well falls off target rate control there will be no response to changing the target. In this situation, WBHP is a good control parameter as there will always be a response while the well is open.
- For smart well optimization, the use of valve parameters controlling the cross-sectional area of a valve segment ([WSEGVALV](#)) is encouraged.
- Typically the calculation of adjoint gradients (sensitivities) will take a time which is less than, but of the order of, the time of the normal simulation.
- Each simulation performed with the control parameters obtained by a model update from the adjoint gradients will take a similar time, and as this is a gradient-based optimization, the use of a line-search algorithm means that there may be several simulations before an optimum is achieved using a given search direction.

The use of constraints and also simple bounds on parameters help to constrain the line-search step size and reduce the number of simulation runs required. Clearly however the overall time to run the optimizer can become a concern. You can control the overall run time as follows:

Now that you have decided your optimization problem, there are a couple of issues for you to consider related to the time it takes for your model to run.

- Control on the maximum number of optimization iterations (gradient evaluations) and simulations is provided using the `OPTDIMS` keyword.
- If the optimization problem (that is the objective function, parameters and constraints) is restricted to a particular sector of the model, then you should consider using a model of just that sector to reduce the time required for each simulation.
- If your model has a long history period, and the optimization problem (that is the objective function, parameters and constraints) are only specified in the prediction part of the schedule, then you should consider using a restart file to reduce the time required for each simulation.

Modifying your model for reservoir optimization

To specify your reservoir optimization problem in the model, you need to:

1. Add the `RESOPT` keyword to the RUNSPEC section to request the Reservoir Optimization
2. Add the `OPTIMIZE` section keyword before your `END` keyword
3. Add the `OPTFUNC` keyword to the `OPTIMIZE` section to specify your objective function
4. Add the `OPTCONS` keyword to the `OPTIMIZE` section to specify your production/injection constraints.
 - a. You should ensure that none of these constraints are violated at the start of the optimization because the optimizer must start from a feasible position honoring all the constraints.

ECLIPSE 300 will check for this when processing the `OPTCONS` keyword, but it is a good idea to look at the values of constraint quantities in the forward simulation before adding the `OPTCONS` keyword.

For example, if the maximum FWPR in the forward simulation is $2000\text{sm}^3/\text{day}$, then setting an optimization constraint of $1000\text{sm}^3/\text{day}$ is invalid. The starting point for simulation will violate constraints and there is no mechanism to return to a feasible parameter space honoring the constraint.

5. Add the `OPTPARS` keyword to the `OPTIMIZE` section to specify your control parameters.
 - a. You should try to make sure that your initial parameter values lie within the upper and lower limits set by this keyword.

There is an initial feasibility check in ECLIPSE 300 to ensure that the parameter values in the forward simulation are within these limits, though it is a good idea to look at the range of parameter values in the forward simulation before specifying the limits.

6. Add the `OPTDIMS` keyword to the `OPTIMIZE` section to specify how many optimization iterations and how many simulations are allowed in the Reservoir Optimization feature.

Tip: Initially these limits should both be set to one so that you can perform a simple sensitivity analysis for parameter screening.

7. Add the [RPTOPT](#) keyword to the OPTIMIZE section and request SEPARATE output.

This uses more hard-disk space than non-separate output but does allow tracking of the progress of the parameters, constraints and objective function components fully. A separate summary file will be created for each optimization iteration, and these production profiles may be visualized independently in ECLIPSE Office.

8. As you would in a normal simulation, add summary and reporting output requests for any simulation results you want to track.
 - a. If a production constraint has been added in the OPTIMIZE section it is good practice to add the mnemonic for this constraint to the SUMMARY section so that you can track whether the constraint is violated in each simulation.

Note: The method used for calculating the adjoint gradients relies on restart files (containing the well and grid block solutions) being generated at each report time during the simulation. Any [RPTRST](#) keyword mnemonics that turn off the required reporting will be ignored.

Checking your modified model

Now that you have specified the Reservoir Optimization problem in the model, and you have indicated that you just want one gradient calculation performed and no optimization (using the [OPTDIMS](#) keyword), you should run the model in ECLIPSE 300 and examine the content of the PRT file.

1. Look for general errors and warnings, in ECLIPSE 300 these always start with the strings “@-- Error” and “@--Warning”. There should be no errors introduced if the steps above have all been followed.
 - This should tell you if any constraints are violated at your starting point. If this is the case, you will get the message:

```
.....@--Error at 150.0000 Days 4 May 1988
@ Optimization initial point is infeasible.....
```

The optimizer must stay within the feasible parameter space at all times (except when backtracking from a constraint). If you are violating any constraints, then you either have to change your starting point to make it feasible or you have to change your constraint.

- This will also tell you if your initial parameter values are outside their lower and upper bounds. If this is the case, you will get a message similar to:

```
.....@--Error at 150.0000 Days 4 May 1988
@ Optimization Parameter      5 is out of range
@ Parameter Name : WBHP
@ Parameter Domain: P1
@ Restart Index : 5
@ Current parameter value is 250.000
@ Lower limit =0.100000E+03, Upper Limit =0.200000E+03
@ Change parameter range using the OPTPARS keyword.....
```

If you have initial parameter values that are outside their lower and upper bounds, then you must either change your initial parameter values, or you must change your parameter bounds.

2. Look at the sensitivity of your objective to the parameter values. This is indicated by the magnitude of the search direction reported for the parameters. These are reported in a table immediately after the text:

=Optimization Parameter Update-----

You need to decide whether there is a reasonable response of the objective function to the parameters you specified. (The definition of a reasonable response is largely problem dependent, the expected parameter changes should not all be too close to zero.) If there is not a reasonable response, then you should reconsider your parameter choice as the optimizer will not be able to make much progress with ineffective parameters.

If you need to change the parameters or constraints for any of the reasons given in this section, then you will need to run the Reservoir Optimization again to check the new optimization problem you have specified.

Tracking the progress and results of the optimizer

If you have a feasible starting point, and you are happy with your choice of parameters, then you should change the limits on the number of iterations and simulations given in the `OPTDIMS` keyword. The new values you choose will be largely dependent on the time your model takes to simulate.

Tip: A reasonable choice for the first optimization attempt is a limit of ten on each number.

Then you run the simulation model with the Reservoir Optimization feature. Essentially, this performs a sequence of simulations for you, calculating new gradient directions and performing line-searches when deemed necessary by the optimizer.

You can check the progress of the optimizer in the PRT file. At the end of each simulation, the following information is output:

```
=====
Optimization Iteration: 1 Simulation No: 1
Current Objective Function value: 2702118.0
Prev Valid Objective Function value: 1986265.8
Best Valid Objective Function value: 2702118.0 (at Itn: 1 Sim: 1)
Tolerance in Objective Function : 0.1000E-05
=====
```

This output tells you how many simulations and gradient calculations have been performed so far. It also tells you the current and previous objective function values, the best valid objective function (that is, the largest value for which no constraints have been violated) and the iteration and simulation at which it occurred.

During line-searches, it is possible for the optimizer to step too far in the search direction and hence violate a constraint. If this happens, you will see text such as:

```
Constraints have been overshot using Alpha = 512.00000
Backtracking Alpha to honor constraints.
Proceeding with new Line Search Alpha value = 259.17128
```

The status of any active constraints are reported in a table under the text:

=Optimization: Constraints=====

This table lists the constraints that we are sufficiently close to so that we can consider we are actually on the constraint (that is, have an activity ratio with an absolute value of less than unity).

Active Constraint	Report Index	Current Value	Limiting Value	Activity Ratio
FWPRFIELD	14	0.1443E+03	0.1500E+03	-0.763
FWPRFIELD	15	0.1501E+03	0.1500E+03	+0.019
FWPRFIELD	16	0.1555E+03	0.1500E+03	+0.737

At each simulation, the parameter values (and the update in these values from the previous simulation) are output in a table after the text:

```
=Optimization Parameter Update=====
```

At the end of the optimization, the parameter values that were used in the simulation that gave the best objective function and did not violate the constraints are given in a table after the text:

```
=Optimization Parameters For Best Valid Iteration=====
```

The reason that the optimizer terminated is reported at the end of the PRT file:

```
=====
```

```
Optimization Terminating:
```

This will be one of the following:

1. Objective function has converged

This means that the relative change in the objective function over the last two simulations is less than the tolerance specified in the [OPTTUNE](#) keyword

2. Objective function has zero gradient

This means that the average gradient of the objective function with respect to the control parameters is less than the tolerance specified in the [OPTTUNE](#) keyword

3. Control parameters have converged

This means that the average relative parameter change over the control parameters is less than the tolerance specified in the [OPTTUNE](#) keyword

4. No control parameters are active

5. Maximum number of optimization iterations reached.

This means that we have reached the limit on the number of optimization iterations specified in the [OPTDIMS](#) keyword

6. Maximum number of optimization simulation reached

This means that we have reached the limit on the number of optimization iterations specified in the [OPTDIMS](#) keyword

If you ran the optimizer using the SEPARATE mnemonic in the [RPTOPT](#) keyword, then the Reservoir Optimization produces separate results files for each iteration in the optimization. These results files have names of the form BASE_01.xxx, BASE_02.xxx and so on where BASE.DATA is the base data-set, and _01, _02 and so on label the optimization iteration. You can track the progress of the objective function, constraints and parameters at each optimization iteration by plotting their equivalent summary vectors (if you have requested these) in Results Viewer in ECLIPSE Office. If you intend to view the results in this way, then you should also specify the CLRFILES mnemonic in the [RPTOPT](#) keyword so that

the results files from any previous optimizations are removed before starting this optimization, otherwise the presence of these old results files can cause confusion.

[Ref. 31] gives some examples of reservoir optimization studies.

Summary of keywords (ECLIPSE 300)

RUNSPEC

- [RESOPT](#) Activate the Reservoir Optimization License.

SCHEDULE section

- [RPTRST](#) Set the restart output frequency using BASIC and FREQ

OPTIMIZE section

Keyword	Description
OPTCONS	Specify the optimization constraint functions
OPTDIMS	Specify the key optimization dimensions
OPTFUNC	Specify the optimization objective function
OPTLOAD	Hot-start a reservoir optimization simulation
OPTOPTS	Specify the optimization options (for example which optimizer to use)
OPTPARS	Specify the optimization control parameters
OPTTUNE	Define tuning for the optimization (for example, tolerances)
RPTOPT	Request the style of optimization output

Miscellaneous

- Item 77 of the OPTIONS3 keyword - compute and output constraint function gradients

Example

The following is the OPTIMIZE section of a reservoir optimization example data set.

The Reservoir Optimization license has been activated by the presence of the [RESOPT](#) keyword in the RUNSPEC section. Restart files have been requested every calendar month.

```
-----
OPTIMIZE
-----
OPTDIMS
-- Maximum no.  Maximum no.
-- Iterations   Simulations
      30          100 /
OPTTUNE
```

```

-- Constraint Tolerance
0.05 /

OPTFUNC
-- O.F.          O.F.
-- Component    Domain
'FOPT'        'FIELD' /
/

OPTPARS
-- Parameter   Parameter   Lower   Upper
-- Mnemonic    Domain     Limit   Limit
'WBHP'        'P1'       100.0   350.0 /
'WBHP'        'P2'       100.0   350.0 /
/

OPTCONS
-- Constraint   Constraint   Constraint   Constraining
-- Mnemonic     Domain      Type       Value
'FOPR'         'FIELD'    '<'       3015 /
'FGPR'         'FIELD'    '<'       665000 /
/

```

In this example:

- The objective is to maximize the field oil production total (specified using the `OPTFUNC` keyword).
- This is subject to limits of 3015 SM³/DAY on the field oil production rate and 665000 SM³/DAY on the field gas production rate (specified using the `OPTCONS` keyword).
- These constraints are considered to be met if the simulated value lies within 1 part in 20 of the limiting value (specified using the `OPTTUNE` keyword).
- The optimizer tries to achieve this objective by varying the bottom hole pressure in wells P1 and P2 between 100 and 300 BARSA at each reporting interval over the whole time period of the simulation (specified using the `OPTPARS` keyword).
- The optimizer is allowed a maximum of 30 gradient simulations or 100 forward simulations (whichever happens first) to try to achieve this objective (specified using the `OPTDIMS` keyword).

50

Parallel option

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 activated by using 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 multi-core computers or clusters with high performance interconnects. 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

Parallel versions ECLIPSE 100 and ECLIPSE 300 are fundamentally different in two respects. Firstly, the well model calculations of ECLIPSE 100 are not done in parallel. This has a minor impact on the scalability of ECLIPSE 100. The second and more important difference is in the approach used in the making the linear solver run in parallel. ECLIPSE 100 uses a modified nested factorization approach, which maintains full coupling across the grid at each linear iteration. ECLIPSE 300 uses a domain decomposition approach.

ECLIPSE 100 restricts you to a 1-dimensional domain decomposition, but maintains full coupling of the solution across the reservoir at each linear iteration, at the expense of increasing the work required per linear iteration by 1.3. We would however not expect the number of linear iterations to increase significantly as we increase the number of domains. ECLIPSE 100 parallel runs are restricted to three dimensional datasets.

ECLIPSE 300 allows a 2-dimensional decomposition and requires no extra work per linear iteration, at the cost of breaking the coupling across the reservoir at each linear iteration. We would therefore expect the number of linear iterations to increase with the number of domains. (Note that each domain is coupled to its neighbor only at each linear iteration, but the reservoir is fully coupled at the level of each Newton iteration.)

In both ECLIPSE simulators a balance has to be struck between increasing the amount of work per linear iteration and increasing the number of linear iterations.

ECLIPSE 100

ECLIPSE 100 partitions the reservoir either in the x or the y direction, whichever is the outer solver direction. It automatically divides the reservoir into domains with approximately equal numbers of active cells.

The number of planes allocated to each domain can be modified using the [DOMAINS](#) keyword in the [GRID](#) section. Since each domain is processed by an individual processor, it is advantageous to have broadly similar numbers of active cells within each domain. The partitions of the reservoir and the solver directions can be output to the Print file by including the [DOMAINS](#) mnemonic in [RPTGRID](#), or graphically using the domains output to the [INIT](#) file.

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.

- 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:

- 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.
- 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 made to run in parallel. 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.
- 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 [RPTSCHE](#)D, 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
      WRRES 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

```

PARALLEL
16 'S' /

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.

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.

$$S = GAM + TREE \quad \text{Eq. 50.1}$$

Serial fraction:

Theoretical parallel CPU time:

$$\left(\frac{1-S}{NP} + \frac{S}{NP - 1} \log_2(NP) \right) CPU_{serial} \quad \text{Eq. 50.2}$$

NP is the number of processors (usually = number of domains)

where

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 bandwidth limitations of the message passing system. In the shared memory case these include the startup cost of the parallel loops and the bottleneck 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 overridden 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 below. Note that this keyword also contains partitions for the 2 and 3 processor cases.

Process	Cells
1	1 to 40
2	41 to 60
3	61 to 80
4	81 to 120

Table 50.1: The division of cells between processors using the PSPLITX keyword.

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 run again.

To examine the partition of a particular run, the PART mnemonic can be placed in the [RPTRST](#) keyword, and then plotted from the RESTART file using any standard package (for example, GRAF, FloViz). This is useful to determine if any wells cross partition boundaries.

Note the WARP linear solver splits each parallel partition into a number of sub-grids, which are connected through NNCs; see "[The ECLIPSE 300 WARP linear solver](#)" for a detailed description.

Running LGRs in parallel (ECLIPSE 100)

Parallel domain decomposition runs can include local grids. These are defined in the normal way. By default, in ECLIPSE 100 each local grid is treated as a complete entity and is allocated to one of the slave processes. This will usually be the most efficient approach if there is a large number of local grids; if we have only a few LGRs, however, there will be a load imbalance. Each slave process can handle any number of local grids. Each LGR can be allocated manually to a specific process using the [IHOST](#) keyword.

Alternatively, ECLIPSE 100 can allocate the LGRs automatically. While doing this ECLIPSE 100 attempts to balance the number of active refined cells.

In ECLIPSE 300 the refined cells inherit their host processor from the global grid. Consequently, LGRs are split across processors. If there is a large number of LGRs it can be difficult to achieve a good load balance.

Performance bottlenecks

There are several possible reasons for performance bottlenecks.

- 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 [RPTRST](#) 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 one 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 timestepped LGRs can destroy this load balancing as these are processed in completely different ways. Also, locally timestepped 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

ECLIPSE 100 and ECLIPSE 300 allocate memory as needed using dynamic memory management. Therefore, the [MEMORY](#) keyword is not needed anymore.

The memory resources will reduce as the number of processors (as specified in the [PARALLEL](#) keyword) increases. This is a consequence of the individual domains becoming smaller as the number of processors increases. Thus less storage per domain is needed. How the memory requirements scale as the number of processors used for a particular job increases is very case-dependent, and is not linear. All the well and tabular data is stored simultaneously by all processes. Consequently, severely over-dimensioning the maximum number of wells or completions, for example, will waste memory on each process with a parallel job. This could become expensive as the number of processors used in a parallel job is increased.

Output (ECLIPSE 100)

When running the parallel option in ECLIPSE 100, output may appear in a different file location than in serial. For each processor, additional files are created to store the output from that processor. Additionally, terminal output for processors other than the master are written to a separate log file. All of these files have the same file types as the main output files but have the processor number appended after the file root and before the file type. No additional files are created for processor number 1, the master processor. An example of the files output in parallel is:

File type	Main output file	Output file for processor n
Print	BASE.PRT	BASE.n.PRT
Debug	BASE.DBG	BASE.n.DBG
Message	BASE.MSG	BASE.n.MSG
Log	Output to terminal	BASE.n.LOG

Table 50.2: Example of the files output in ECLIPSE 100 when using the parallel option.

At the end of a successful run, output files other than the master are deleted. The additional files are retained if either a bug or error occurs or if item 2 of the [PARAOPTS](#) keyword is set to 1. Output controlled

by keywords beginning RPT may also not be output in parallel unless item 8 of the PARAOPTS keyword is set to 1.

Messages, comments, warnings, errors and bugs from all processors are included in the main output files in parallel. In the main output files, the number of the processor that produced a message is included in the message header. For example:

```
@@--MESSAGE ON PROCESSOR 1 AT TIME      0.0    DAYS   (1-JAN-2000):
@      THE MEMORY REQUIRED TO PROCESS THE SOLUTION SECTION IS
@          1679 MBYTES  PROCESSOR 1
@          1329 MBYTES  PROCESSOR 2
@          1284 MBYTES  PROCESSOR 3
@          1261 MBYTES  PROCESSOR 4
```

If identical messages are repeated simultaneously across processors then the message will only be output once in the main output files. If a very large number of messages are output in short succession then the parallel message buffer may become full. If this happens, the buffer size will be increased automatically. If insufficient memory is available then the code will exit. This issue can be avoided by reducing the maximum number of messages output using the [MESSAGES](#) keyword

Summary of keywords

RUNSPEC

Restriction	Keyword	Description
E300 only	NPROCX	Defines the number of processors to be used in the x-direction.
E300 only	NPROCY	Defines the number of processors to be used in the y-direction.
	PARALLEL	Initializes the Parallel option.
E300 only	PDIVX	Defines the partitioning in the x-direction.
E300 only	PDIVY	Defines the partitioning in the y-direction.
E300 only	PSPLITX	Specifies multiple domain decomposition in the x-direction.
E300 only	PSPLITY	Specifies multiple domain decomposition in the y-direction.

GRID (ECLIPSE 100)

ECLIPSE 100 only keyword	Description
DOMAINS	Defines the partitioning of the grid.
IHOST	Allocates LGRs to specific processes.
PARAOPTS	Permits some fine tuning of the simulator.
RPTGRID	Controls output from the GRID section. The DOMAINS argument outputs the domain split.

SOLUTION section (ECLIPSE 300)

ECLIPSE 300 only keyword	Description
RPTRST	Controls solution output for GRAF and flexible restarts. Argument PART outputs domain partitions
RPTSOL	Controls output from the SOLUTION section. Argument PART outputs domain partitions.

51

Petro-elastic model

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 grid cell.

Note: You must have an `eclipse4d` license in order to access this feature. See "[Licenses](#)" for more information.

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 grid cell 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).
- There is a homogeneous mineral modulus.
- The rock is isotropic.
- 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

Most of the theory of this model (that is the assumptions and equations) can be found in [Ref. 72].

The petro-elastic model can be decomposed into four levels. These are:

- "Level 1: Impedances and velocities"
- "Level 2: Effective bulk/shear modulus and density"
- "Level 3: Rock properties and fluid properties"
- "Level 4: Individual fluid phase properties"

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 and so on.

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 grid cell

I_S the impedance to shear waves for a particular simulation grid cell

ν the Poisson’s ratio for a particular simulation grid cell

These are defined as

$$I_P = \rho V_P \quad \text{Eq. 51.1}$$

$$I_S = \rho V_S \quad \text{Eq. 51.2}$$

$$\nu = \frac{\frac{1}{2}V_P^2 - V_S^2}{V_P^2 + V_S^2} \quad \text{Eq. 51.3}$$

where

ρ is the effective density of all the material (fluids, minerals, frame) in the grid cell

V_P is the effective pressure wave velocity in the simulation grid cell

V_S is the effective shear wave velocity in the simulation grid cell

These effective velocities are calculated by

$$V_P = \sqrt{\frac{K + (4G/3)}{\rho}} \quad \text{Eq. 51.4}$$

$$V_S = \sqrt{\frac{G}{\rho}} \quad \text{Eq. 51.5}$$

where

K is the effective bulk modulus of the simulation grid cell

G is the effective shear modulus of the simulation grid cell

Level 2: Effective bulk/shear modulus and density

Effective bulk modulus

The effective bulk modulus of a simulation grid cell is a combination of the effects of the fluids, the frame and the minerals (for example, sand, shale, silt) in the grid cell. The combination is performed using the Gassmann fluid substitution equation. This relies on the assumptions listed in "[Assumptions](#)".

The following form of Gassmann's equation is used in the petro-elastic model:

$$K = K_{frame} + \frac{\left(1 - \frac{K_{frame}}{K_{minerals}}\right)^2}{\frac{\varphi_{eff}}{K_{fluid}} + \frac{(1-\varphi_{eff})}{K_{minerals}} - \frac{K_{frame}}{(K_{minerals})^2}} \quad \text{Eq. 51.6}$$

where

K_{frame} is the effective bulk modulus of the dry rock frame for this grid cell

$K_{minerals}$ is the effective bulk modulus of the minerals in this grid cell

K_{fluid} is the effective bulk modulus of the pore fluid in this grid cell

φ_{eff} is the effective porosity of this grid cell

Effective shear modulus

With the assumption that the fluids in the grid cell are not too viscous, then the overall shear modulus is just the shear modulus of the dry rock frame, that is:

$$G = G_{frame} \quad \text{Eq. 51.7}$$

where

G_{frame} is the shear modulus of the dry rock frame for this grid cell

Effective density

The effective density in a grid cell is given by the effective porosity weighted average of the fluid density and the mineral density in that grid cell, that is:

$$\rho = \varphi_{eff} \cdot \rho_{fluid} + (1-\varphi_{eff}) \cdot \rho_{minerals} \quad \text{Eq. 51.8}$$

where

$\rho_{minerals}$ is the effective density of the minerals in this grid cell

ρ_{fluid} is the effective density of the fluids in this grid cell

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 grid cell is based on the porosity at the reference pressure, φ_0 , in the grid cell 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.

$$\varphi_{eff} = \varphi_0 \left(1 + X + \frac{X^2}{2} \right) \quad \text{Eq. 51.9}$$

where

$$X = C(P - P_{ref}) \quad \text{Eq. 51.10}$$

and

C is the rock compressibility for the grid cell

P is the pressure in the grid cell

P_{ref} is the rock compressibility reference pressure for the grid cell

In each grid cell, 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.

$$\varphi_{eff} = \varphi_0 C(P)$$

where

$C(P)$ is the rock compaction pore volume multiplier for the grid cell (this is specified as tables of multipliers against pressure in the [ROCKTAB](#) keyword)

P is the pressure in the grid cell

Effective fluid density

The effective fluid density is the saturation weighted average of the individual fluid component densities, that is,

$$\rho_{fluid} = S_o \rho_o + S_w \rho_w + S_g \rho_g \quad \text{Eq. 51.11}$$

where

$S_{o/w/g}$ is the saturation of oil/water/gas in the grid cell

$\rho_{o/w/g}$ is the density of oil/water/gas in the grid cell

These quantities are all available within the simulator for each grid cell.

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:

$$\frac{1}{K_{fluid}} = \frac{S_o}{K_o} + \frac{S_w}{K_w} + \frac{S_g}{K_g} \quad \text{Eq. 51.12}$$

where

$K_{o/w/g}$ is the bulk modulus of the oil/water/gas in the grid cell

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:

$$K_{fluid} = (K_{liquid} - K_g)(1 - S_g)^e + K_g \quad \text{Eq. 51.13}$$

where

e is the “patchy saturation” exponent

K_{liquid} is the combined bulk modulus of the liquid phases, given by

$$\frac{S_o + S_w}{K_{liquid}} = \frac{S_o}{K_o} + \frac{S_w}{K_w} \quad \text{Eq. 51.14}$$

The patchy saturation exponent is supplied directly as an input for each petro-elastic region as part of the Petro-elastic Model definition in the simulation model. This equation was first presented in [Ref. 15].

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 grid cell, that is

$$K_{minerals} = C_1 \varphi_{eff} + C_0 \quad \text{Eq. 51.15}$$

where

C_0 is the zeroth order coefficient of effective porosity

C_1 is the first order coefficient of effective porosity

The zeroth and first order coefficients of effective porosity are supplied directly as an input for each petro-elastic region as part of the petro-elastic model definition in the simulation model.

Effective shear modulus of the minerals

The effective shear modulus of the minerals, $G_{minerals}$, is supplied directly as an input for each petro-elastic region as part of the petro-elastic model definition in the simulation model.

Effective bulk modulus of the frame

The effective bulk modulus of the dry rock frame is given by:

$$(K_{frame})^\alpha = C_K (K_{minerals})^\alpha + \sum_{i=1}^7 \left\{ C_{K,i}(P) \cdot \varphi_{eff}^i \right\} \quad \text{Eq. 51.16}$$

where

α is a specified exponent

C_K is the coefficient of the $K_{minerals}$ term in the calculation of K_{frame}

$C_{K,i}(P)$ are specified tabular functions of pressure

α is usually -1 or +1 and specifies whether the dry rock frame and mineral bulk moduli are related harmonically or arithmetically.

Both α and the tabular functions are supplied directly as an input for each petro-elastic region as part of the Petro-elastic Model definition in the simulation model.

Effective shear modulus of the frame

The effective shear modulus of the dry rock frame is given by

$$(G_{frame})^\alpha = C_G (G_{minerals})^\alpha + \sum_{i=1}^7 \left\{ C_{G,i}(P) \cdot \varphi_{eff}^i \right\} \quad \text{Eq. 51.17}$$

where

α is the same exponent as specified in equation 51.16

C_G is the coefficient of the $G_{minerals}$ term in the calculation of G_{frame}

$C_{G,i}(P)$ are specified tabular functions of pressure

Both α and the tabular functions are supplied directly as an input for each petro-elastic region as part of the petro-elastic model definition in the simulation model.

Level 4: Individual fluid phase properties

The calculations for the bulk moduli of the oil, water and gas in a grid cell are based on the empirical equations given in [Ref. 72].

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 grid cell is given by

$$K_o = \rho_o V_o^2 \quad \text{Eq. 51.18}$$

where

ρ_o is the density of oil (gm/cc) in the grid cell

V_o is the velocity of sound in oil in the grid cell given (in m/s) by

$$V_o = 2096 \sqrt{\frac{\rho_{o,R}}{2.6 - \rho_{o,R}}} - (3.7 \cdot T) + (4.64 \cdot P) + 0.0115 \cdot \left\{ \left(4.12 \sqrt{\frac{1.08}{\rho_{o,R}}} - 1 \right) \cdot T \cdot P \right\} \quad \text{Eq. 51.19}$$

where

T is the temperature ($^{\circ}\text{C}$) in the grid cell. 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 grid cell

$\rho_{o,R}$ is the density of oil (gm/cc) in the grid cell at reference conditions (15.6°C and atmospheric pressure). This is calculated by first correction for temperature effects

$$\rho_{o,P} = \rho_o \left\{ 0.972 + 3.81 \times 10^{-4} (T + 17.78)^{1.175} \right\} \quad \text{Eq. 51.20}$$

and then correcting for pressure effects using

$$\rho_{o,P} = \rho_{o,R} + (0.00277P - 1.71 \times 10^{-7}P^3)(\rho_{o,R} - 1.15)^2 + 3.49 \times 10^{-4}P$$

Bulk modulus of water

The bulk modulus of water in a grid cell is given by

$$K_w = \rho_w V_w^2 \quad \text{Eq. 51.21}$$

where

V_w is the velocity of sound waves in water in the grid cell given (in m/s) by

$$V_w = V_{pw} + C_S \cdot S + (780 - 10P + 0.16P^2) \cdot S^{1.5} - 1820S^2 \quad \text{Eq. 51.22}$$

where

S is the salinity of the water in the grid cell (in ppm/106, that is, as a fraction of 1)

V_{pw} is the velocity of sound waves in pure water in the grid cell

Again, as this is an empirical formula, P must be in MPa. S is supplied directly as an input for each petro-elastic region as part of the Petro-elastic Model definition in the simulation model, while C_S and V_{pw} are given by

$$C_S = 1170 - 9.6T + 0.055T^2 - 8.5 \times 10^{-5}T^3 + 2.6P - 0.0029TP - 0.0476P^2$$

where

T is in °C, and

$$V_{pw} = \sum_{i=1}^4 \left\{ \sum_{j=0}^3 (w_{ij} T^i P^j) \right\} \quad \text{Eq. 51.23}$$

where

$w_{00} = 1402.85$	$w_{02} = 3.437 \times 10^{-3}$	Eq. 51.24
$w_{10} = 4.871$	$w_{12} = 1.739 \times 10^{-4}$	
$w_{20} = -0.04783$	$w_{22} = -2.135 \times 10^{-6}$	
$w_{30} = 1.487 \times 10^{-4}$	$w_{32} = -1.455 \times 10^{-8}$	
$w_{40} = -2.197 \times 10^{-7}$	$w_{42} = 5.230 \times 10^{-11}$	
$w_{01} = 1.524$	$w_{03} = -1.197 \times 10^{-5}$	
$w_{11} = -0.0111$	$w_{13} = -1.628 \times 10^{-6}$	
$w_{21} = 2.747 \times 10^{-4}$	$w_{23} = 1.237 \times 10^{-8}$	
$w_{31} = -6.503 \times 10^{-7}$	$w_{33} = 1.327 \times 10^{-10}$	
$w_{41} = 7.987 \times 10^{-10}$	$w_{34} = -4.614 \times 10^{-13}$	

Bulk modulus of gas

The bulk modulus of gas in a grid cell is given (in MPa) by

$$K_g = \frac{P\gamma}{1 - \left(\frac{P_r}{Z} \cdot f \right)} \quad \text{Eq. 51.25}$$

where

$$GRAV = \frac{\rho_{\text{gas}|\text{surface}}}{\rho_{\text{air}|\text{surface}}}$$

$$P_r = \frac{P}{4.892 - (0.04048 \cdot GRAV)}$$

$$T_r = \frac{T + 273.15}{94.72 + (170.75 \cdot GRAV)}$$

$$f = c \cdot d \cdot m + a$$

$$Z = (a \cdot P_r) + b + (c \cdot d)$$

$$a = 0.03 + 0.00527(3.5 - T_r)^3$$

$$b = 0.642T_r - 0.007T_r^4 - 0.52$$

$$c = 0.109(3.85 - T_r)^2$$

$$d = e \left\{ - \left[0.45 + 8 \left(0.56 - \frac{1}{T_r} \right)^2 \right] \frac{P_r^{1.2}}{T_r} \right\}$$

$$m = -1.2 \left\{ 0.45 + 8 \left(0.56 - \frac{1}{T_r} \right)^2 \right\} \frac{P_r^{0.2}}{T_r}$$

$$\gamma = 0.85 + \frac{5.6}{P_r + 2} + \frac{27.1}{(P_r + 3.5)^2} - 8.7e^{\{-0.65(P_r + 1)\}}$$

GRAV is available within the simulator.

Again, since this is an empirical formula, *P* is in MPa, and *T* is in °C.

Implementation of the petro-elastic model

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 equation 51.22

T the temperature in equation 51.19, equation 51.22, and equation 51.25 (this may be available elsewhere in the simulation model but the petro-elastic temperature is specified separately)

ρ_{minerals} the effective density of the minerals in equation 51.8

*C*₁ the first order coefficient of effective porosity in the calculation of the effective bulk modulus of the minerals in equation 51.15

C_0	the zeroth order coefficient of effective porosity in the calculation of the effective bulk modulus of the minerals in equation 51.15
C_K	is the coefficient of the $K_{minerals}$ term in the calculation of K_{frame} in equation 51.16
C_G	the coefficient of the $G_{minerals}$ term in the calculation of G_{frame} in equation 51.17
$G_{minerals}$	the effective shear modulus of the minerals in equation 51.17
α	the exponent in the calculation of the effective bulk/shear modulus of the dry rock frame in equation 51.16 and 51.17
e	the “patchy saturation” exponent in equation 51.13 (only used if Wood’s Law with Brie et al is requested). Switch to select between Generalized Wood’s Law and Wood’s Law with Brie et al in the calculation of the effective bulk modulus of the fluid
$C_{K,i}(P)$	the tabular functions of pressure in the calculation of the effective bulk modulus of the dry rock frame in equation 51.16
$C_{G,i}(P)$	the tabular functions of pressure in the calculation of the effective shear modulus of the dry rock frame in equation 51.17

Outputs

The output from the Petro-elastic Model depends on which simulated seismic-derived properties are requested (from the list of P-wave impedance, S-wave impedance and Poisson’s ratio). When a simulated seismic-derived property is requested, it is output as a value on each active grid cell 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"](#)) and request gradients. For further information on requesting gradients, see ["Gradient Option"](#).

Summary of keywords

RUNSPEC

Keyword	Description
PEDIMS	Sets the number of petro-elastic regions in the simulation, sets table dimensions, in terms of the number of rows, for the PEKTABx and PEGTABx keywords.

REGIONS section

Keyword	Description
PENUM	Specifies the petro-elastic region number for each grid cell in the simulation model.

PROPS section

Keyword	Description
PECOEFS	Specifies a variety of fluid and rock properties, and also options to be used in the petro-elastic calculations, for each petro-elastic region in the simulation model.
PEKTABx	Specifies the frame bulk modulus functions for each petro-elastic region in the simulation model.
PEGTABx	Specifies the frame shear modulus functions for each petro-elastic region in the simulation model.

SOLUTION/SUMMARY section

Keyword	Description
RPTRST	Controls output of properties (including simulated seismic-derived properties) to RESTART files. The Petro-elastic Model specific arguments for this keyword are shown below.
GRADRESV	Specifies which gradients of solution data are output to the RESTART files. The Petro-elastic Model specific arguments for this keyword are shown in below.

Note: For each of these properties, you must also request the DEN item in the RPTRST keyword.

Restriction	Keyword	Description
E100 only	RPTRST	<ul style="list-style-type: none"> ACIP Output gradient of pressure-wave impedance ACIS Output gradient of shear-wave impedance POIS Output gradient of Poisson's ratio
E100 only	GRADRESV	<ul style="list-style-type: none"> ACIP Output gradient of pressure-wave impedance ACIS Output gradient of shear-wave impedance POIS Output gradient of Poisson's ratio

52

Polymer flood model

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The main objective of polymer injection during water flooding of oil reservoirs is to decrease the mobility of the injected water. This decrease results in a more favorable fractional flow curve for the injected water, leading to a more efficient sweep pattern and reduced viscous fingering. Some 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 with the polymer concentration). 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 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. In ECLIPSE 100 these effects are modeled by means of a mixing parameter applied to the viscosity terms in the fluid flow equations. In ECLIPSE 300 these effects are currently not modeled; it is assumed that the polymer is fully mixed in the water phase.

Certain polymer solutions, such as those of the PA type (Polyamides), 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 pre-flush with fresh water to reduce the exposure of the polymer solution to the reservoir brine. Both ECLIPSE 100 and ECLIPSE 300 models allow you to investigate the effect of varying brine concentrations on the efficiency of the polymer flood. The brine effect is, however, optional.

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 100 allows you to either neglect the desorption effect or to allow the adsorption isotherm to be retraced after passage of the polymer slug. ECLIPSE 300 allows you to model the effects of adsorption and desorption by combining the Polymer Flood Model with the Solid Model.

ECLIPSE 100 considerations

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 100 model assumes that the change is negligible.

The effect of temperature variations on the behavior of the polymer solution can be optionally modeled.

In core flooding experiments, it is often observed that injected polymer slugs break through to producers earlier than tracer slugs (for example, NaCl). The polymer fluid velocity is higher than that of the tracer fluid within the porous medium and this is due to the fact that only a fraction of the total pore space is available to the polymer fluid. As the inaccessible pore space to the polymer fluid increases, the effective polymer velocity through the rock increases and leads to a faster breakthrough of polymer. This chromatographic effect is modeled by assuming that the dead pore space is constant for each rock type:

- and either does not exceed the corresponding irreducible water saturation (pre 2008.2).
- or independent of the water saturation (post 2008.2 included)

The rheology of polymer solutions is not simple. At low flow rates the viscosity of the solution is approximately constant and depends only on the concentration of polymer in the solution. At higher flow rates the solution viscosity reduces in a reversible (elastic) manner. At even higher velocities the large polymer molecules begin to break up, and the viscosity reduction becomes irreversible (plastic). The effects tend to be greatest in the vicinity of injection wells where the fluid velocity is greatest, and so is the shear rate.

The ECLIPSE 100 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 blackoil equations are therefore used to describe the hydrocarbon phases in the model.

Modification is required to the standard aqueous (water) equation and additional equations are needed to describe the flow of polymer and brine within the finite difference grid. The water, polymer and brine equations used in the model are as follows:

$$\frac{d}{dt} \left(\frac{VS_w}{B_r B_w} \right) = \Sigma \left[\frac{Tk_{rw}}{B_w \mu_{w,eff} R_k} (\delta P_w - \rho_w g D_z) \right] + Q_w \quad \text{Eq. 52.1}$$

$$\frac{d}{dt} \left(\frac{V^* S_w C_p}{B_r B_w} \right) + \frac{d}{dt} \left(V \rho_r C_p \frac{1 - \varphi}{\varphi} \right) = \Sigma \left[\frac{Tk_{rw}}{B_w \mu_{p,eff} R_k} (\delta P_w - \rho_w g D_z) \right] C_p + Q_w C_p \quad \text{Eq. 52.2}$$

$$\frac{d}{dt} \left(\frac{VS_w C_n}{B_r B_w} \right) = \Sigma \left[\frac{Tk_{rw} C_n}{B_w \mu_{s,eff} R_k} (\delta P_w - \rho_w g D_z) \right] + Q_w C_n \quad \text{Eq. 52.3}$$

pre-2008.2

$$V^* = V \left(1 - \frac{S_{dpv}}{S_w} \right) \quad \text{Eq. 52.4}$$

post-2008.2

$$V^* = V (1 - S_{dpv}) \quad \text{Eq. 52.5}$$

where

S_{dpv} denotes the dead pore space within each grid cell

C_p^a denotes the polymer adsorption concentration

ρ_r denotes the mass density of the rock formation

φ denotes the porosity

ρ_w denotes the water density

Σ denotes the sum over neighboring cells

R_k denotes the relative permeability reduction factor for the aqueous phase due to polymer retention

C_p, C_n denote the polymer and salt concentrations respectively in the aqueous phase

$\mu_{a\text{eff}}$ denotes the effective viscosity of the water ($a=w$), polymer ($a=p$) and salt ($a=s$).

D_z is the cell center depth.

B_r, B_w are the rock and water formation volumes

T is the transmissibility

k_{rw} is the water relative permeability

S_w is the water saturation

V is the block pore volume

Q_w is the water production rate

P_w is the water pressure

g is the acceleration due to gravity

The model makes the assumption that the density and formation volume factor of the aqueous phase are independent of the polymer and salt concentrations. The polymer solution, reservoir brine and the injected water are represented in the model as miscible components in the aqueous phase, where the degree of mixing is specified through the viscosity terms in the conservation equations.

The principal effects of polymer and salt on the flow of the aqueous phase are represented by equations 52.1 to 52.4 above. The fluid viscosities ($\mu_w\text{ eff}$, $\mu_p\text{ eff}$, $\mu_s\text{ eff}$) 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 52.2. The adsorption term requires the user to specify the adsorption isotherm, $C_p a$ for each rock type. The effect of pore blocking and adsorption on the aqueous phase relative permeability is treated through the term, R_k , which requires the input of a residual resistance factor for each rock type.

The equations solved for the polymer model are a discretized form of the differential equations 52.1 - 52.4. In order to avoid numerical stability problems that could be triggered by strong changes in the aqueous phase properties over a timestep (resulting from large changes in the local polymer/salt concentrations) a fully implicit time discretization is used, which is therefore free from this type of instability.

Treatment of fluid viscosities

The viscosity terms used in the fluid flow equations contain the effects of a change in the viscosity of the aqueous phase due to the presence of polymer and salt in the solution. However, to incorporate the effects of physical dispersion at the leading edge of the slug and also the fingering effects at the rear edge of the slug the fluid components are allocated effective viscosity values that are calculated using the Todd-Longstaff technique.

Consider the effective polymer viscosity term. The viscosity of a fully mixed polymer solution as an increasing function of the polymer concentration in solution ($\mu_m (C_p)$) must be input. The viscosity of the solution at the maximum polymer concentration also needs to be specified and denotes the injected polymer concentration in solution (μ_p). The effective polymer viscosity is taken to be:

$$\mu_{p,\text{eff}} = (\mu_m (C_p))^\omega \cdot \mu_p^{1-\omega} \quad \text{Eq. 52.6}$$

where

ω is the Todd-Longstaff mixing parameter.

The mixing parameter is useful in modeling the degree of segregation between the water and the injected polymer solution. If $\omega = 1$ then the polymer solution and water are fully mixed in each.

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):

$$\mu_{w,e} = (\mu_m (C_p))^\omega \cdot \mu_w^{1-\omega} \quad \text{Eq. 52.7}$$

In order to calculate the effective water viscosity to be inserted into equation 52.1, the total water equation is written as the sum of contributions from the polymer solution and the pure water. The following expression then gives the effective water viscosity to be inserted into equation 52.1:

$$\frac{1}{\mu_{w,\text{eff}}} = \frac{1 - \bar{C}}{\mu_{w,e}} + \frac{\bar{C}}{\mu_{p,\text{eff}}} \quad \text{Eq. 52.8}$$

$$\bar{C} = \frac{C_p}{C_{p,\text{max}}} \quad \text{Eq. 52.9}$$

where

\bar{C} is the effective saturation for the injected polymer solution within the total aqueous phase in the cell.

If the brine option is active, the above expressions still hold for the effective polymer and water viscosity terms. The injected salt concentration needs to be specified in order to evaluate the maximum polymer solution viscosity, μ_p . The effective salt component viscosity to be used in equation 52.3 is set equal to the effective water viscosity.

Note that the effective viscosities are computed at the pressure of the oil phase.

Treatment of fluid viscosities in the well

The multisegment well and friction well models both include frictional pressure drops which depend upon fluid viscosity. Polymer bearing water will by default have a viscosity as defined by equation 52.9, where the Todd-Longstaff mixing parameter ω is assigned to the well using the [WELSPECS](#) or [WELSPEC1](#) keyword. There is a further option to control this viscosity; which is using the keyword [WPOLYRED](#) to set the polymer reduction factor α . The water phase viscosity in the wellbore is then calculated as

$$\mu_w = \alpha\mu_{w\text{ eff}} + (1-\alpha)\mu_w \quad \text{Eq. 52.10}$$

Polymer and water are assumed to travel with the same velocity in the wellbore.

Treatment of polymer adsorption

Adsorption is treated as an instantaneous effect in the model. The effect of polymer adsorption is to create a stripped water bank at the leading edge of the slug. Desorption effects may occur as the slug passes. The isotherm adsorption can be specified as:

- either a look-up table of adsorbed polymer as a function of polymer concentration using the [PLYADS](#) keyword if there is no salinity dependence of the adsorption, or the [PLYADSS](#) keyword if there is. For salinity dependence, the [ADSALNOD](#) keyword is used for the salinity values, not the [SALTNODE](#) keyword.
- or by a generic analytical adsorption model using the [ADSORP](#) keyword with POLYMER as an argument. This model enables the adsorption to depend on the salinity and rock permeability.

Note that the multi-component brine model ([ECLMC](#) keyword in the RUNSPEC section) cannot currently be used with the polymer/brine flood model.

If desorption is prevented (item 4 of the [PLYROCK](#) keyword) then the adsorbed polymer concentration may not decrease with time. If desorption is allowed then each grid block retraces the adsorption isotherm as the polymer concentration rises and falls in the cell.

Treatment of permeability reductions and dead pore volume

The adsorption process causes a reduction in the permeability of the rock to the passage of the aqueous phase and is directly correlated to the adsorbed polymer concentration. In order to compute the reduction in rock permeability, you are required to specify the residual resistance factor (RRF) for each rock type (item 2 of the [PLYROCK](#) keyword). The actual resistance factor can then be calculated:

$$R_k = 1.0 + (\text{RRF}-1.0) \frac{C_p^a}{C_p^{a\text{ max}}} \quad \text{Eq. 52.11}$$

The value of the maximum adsorbed concentration, $C_p^{a\max}$, depends on the rock type and needs to be specified as item 5 of the [PLYROCK](#) keyword. This value must be non-zero.

The dead pore volume (item 1 of the [PLYROCK](#) keyword) must also be specified for each rock type. It represents the amount of total pore volume in each grid cell that is inaccessible to the polymer solution. The effect of the dead pore volume within each cell is to cause the polymer solution to travel at a greater velocity than inactive tracers embedded in the water. This chromatographic effect is modeled by assuming that the dead pore space is constant for each rock type:

- and either does not exceed the corresponding irreducible water saturation (pre 2008.2)
- or is independent of the water saturation (post 2008.2 included).

If required, the RRF can be set on a cell-by-cell basis using the [PLYKRRF](#) keyword. Any cells for which it is not set will use the value for their rock type given by item 2 of the [PLYROCK](#) keyword.

If the temperature option is specified, then a temperature dependence can be provided for the RRF using the [PLYTRRF](#) keyword.

Similarly, the value of the maximum adsorbed concentration can be set on a cell-by-cell basis using the [PLYCAMP](#) keyword. Any cells for which it is not set will use the value for their rock type given by item 5 of the [PLYROCK](#) keyword.

Treatment of the non-Newtonian rheology

You can choose from three models to take into account some of the non-Newtonian rheological behavior reported for polymer solutions.

Model 1

This model targets the shear thinning or thickening of polymer that has the effect of changing the polymer viscosity at higher flow rates. It assumes that shear rate is proportional to the flow viscosity. This assumption is not valid in general, as for example, a given flow in a low permeability rock will have to pass through smaller pore throats than the same flow in a high permeability rock, and consequently the shear rate will be higher in the low permeability rock. However, under the consideration of a single type of reservoir rock, this assumption can be reasonable.

This model is activated by the [PLYSHEAR](#) keyword. Note that shear thinning can cause runaway positive feedback due to the loop: increased speed causes decreased viscosity causes increased speed. It may be easier to control the shear-thinning effect by using Model 3 below. Shear thickening may exacerbate convergence problems in wells where crossflow is disallowed. Please refer to the keyword description for more details. Note that crossflow in wells is now allowed with polymer.

Model 2

The second model is the Herschel-Bulkley model that can be used to model shear thinning and thickening as well as yield stress, dependent on polymer concentration. Refer to "[Non-Newtonian fluid flows](#)" for more details.

Model 3

The third model is similar to Model 1 but instead uses a logarithmic look-up calculation in order to update the water phase and polymer flow properties in the presence of shear thinning/thickening. This model is activated by the [PLYSHLOG](#) keyword. The shear multipliers can be specified either as a function of water phase/polymer flow velocity or as function of water phase/polymer flow shear rate; keyword [SHRATE](#) is required for the latter.

Model 1 description

In Model 1, the velocity corresponding to the given water/polymer flow is calculated as:

$$V_i = B_w \cdot \frac{F_i}{\varphi A(S_w - S_{wcr})} \quad \text{Eq. 52.12}$$

where

F_i is the water/polymer ($i = w, p$ respectively) 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 the two cells

S_w is the water saturation

S_{wcr} is the critical water saturation.

The shear effect is skipped in those cases where the mobile water saturation is less than or equal to zero.

Note:

Note that the pre-2014.1 velocity calculation is given by:

$$V_i = B_w \cdot \frac{F_i}{\varphi A} \quad \text{Eq. 52.13}$$

and may be activated by setting OPTIONS keyword [item 228](#) to 1.

The reduction in each of the effective component viscosities is assumed to be reversible as a function of the relevant velocity. The resulting shear water and shear polymer viscosities are calculated, respectively, as:

$$\mu_{w,sh} = \mu_{w,eff} Z_w^*, \quad \mu_{p,sh} = \mu_{p,eff} Z_p^* \quad \text{Eq. 52.14}$$

where

$\mu_{w,sh}$ is the shear water viscosity following shear thinning/thickening

$\mu_{p,sh}$ is the shear polymer viscosity following shear thinning/thickening

$\mu_{w,eff}$ is the effective water viscosity (see equation [52.8](#))

$\mu_{p,eff}$ is the effective polymer viscosity (see equation [52.6](#))

Z_w^* is the shear factor applied to the effective water viscosity and is calculated based on the water phase flow velocity prior to shear thinning/thickening

Z_p^* is the shear factor applied to the effective polymer viscosity and is calculated based on the polymer flow velocity prior to shear thinning/thickening

and each shear factor is constructed in the form

$$Z = \frac{1 + (P - 1)M}{P} \quad \text{Eq. 52.15}$$

where

P is the viscosity multiplier assuming no shear effect (entered using the [PLYVISC](#), [PLYVISCS](#), [PLYVISCT](#) or [PLYVSCST](#) keywords)

M is the shear thinning/thickening multiplier supplied in the [PLYSHEAR](#) keyword.

Note: For $M = 1$, or no shear effect, we recover the effective component viscosities, and for $M = 0$, or maximum shear thinning, the shear viscosities are

$$\mu_{w,sh} = \frac{\mu_{w,eff}}{P}, \quad \mu_{p,sh} = \frac{\mu_{p,eff}}{P} \quad \text{Eq. 52.16}$$

which corresponds to the minimum viscosity that can be obtained. If the polymer concentration is zero ($P=1$) we recover the effective component viscosities which, in that particular case, equal the pure water viscosity.

With the above definitions, there is a local implicit relationship between the water phase velocity through a grid block face and its associated velocity-dependent water phase viscosity in the presence of polymer. The equation for this relationship is

$$V_{i,sh} = V_{i,0} \left(\frac{\mu_{i,eff}}{\mu_{i,sh}(V_{i,sh})} \right) \quad \text{Eq. 52.17}$$

where

$V_{i,sh}$ is the shear-modified velocity associated with the water phase/polymer ($i = w, p$ respectively) flow,

$V_{i,0}$ is the Darcy velocity without shear calculated from equation 52.12 (or from equation 52.13 if [OPTIONS](#) keyword item 228 is set to 1),

and the velocity dependence of the shear-modified viscosity is explicitly shown. This equation is solved by an appropriate root finding method. Once the solution of this equation is available for a given grid block face, the sheared velocity at that location is known along with the additional derivatives that it gives rise to; these modified values and derivatives then enter the global implicit calculation for reservoir flows for the current Newton iteration.

The well inflows are treated in a manner analogous to the treatment of block to block flows. The viscosity of the polymer solution flowing into the well is calculated assuming a velocity at a representative radius from the well. The representative radius is taken to be:

$$R_r = \sqrt{R_w R_a} \quad \text{Eq. 52.18}$$

where

R_w is the well bore radius (taken from diameter input in [COMPDAT](#))

R_a is the area equivalent radius of the grid block in which the well is completed.

In the present version of ECLIPSE 100, the radial inflow equation is not integrated over distance from the well to account for the local viscosity reduction due to the local velocity.

Model 3 description

Model 3 is implemented via two approaches, as detailed below. Note that the representative radius for the well inflows described above for Model 1 also applies in each approach below.

Approach I: Based on velocity

In this first approach, for a specified set of reference conditions comprising polymer concentration, salinity and temperature, the shear thinning/thickening multipliers, Z^{ref} , are specified as a function of water phase/polymer flow velocity, V , this time using the [PLYSHLOG](#) keyword.

As in Model 1, the velocities corresponding to the water and polymer flows are calculated according to equation [52.12](#) (or according to equation [52.13](#) if OPTIONS keyword item 228 is set to 1).

The resulting shear viscosities are again obtained using equation [52.14](#) where the calculation of each shear thinning/thickening multiplier (multiplier Z_w^* for the water phase flows and multiplier Z_p^* for the polymer flows) comprises the following series of steps:

- For each velocity in [PLYSHLOG](#), calculate a corresponding viscosity multiplier using

$$Z(V) = \frac{1 + (P-1)M^{ref}(V)}{P} \quad \text{Eq. 52.19}$$

where

M^{ref} is calculated by applying the reverse of equation [52.19](#) and using the reference data provided in [PLYSHLOG](#), and

P is the viscosity multiplier interpolated at the given conditions using the relevant polymer viscosity keyword (one of [PLYVISC](#), [PLYVISCS](#), [PLYVISCT](#), [PLYVSCST](#)).

- Take the natural logarithm of each viscosity multiplier calculated above and the natural logarithm of each corresponding velocity and construct the piecewise linear function through the resulting set of data points:

$$f : \ln V \mapsto \ln Z \quad \text{Eq. 52.20}$$

- Find where the relationship between the unsheared and sheared velocities satisfies the input shear data obtained experimentally, this done by locating the point of intersection of the above piecewise linear function and the straight line represented by:

$$\ln V + \ln Z = \ln V_0 \quad \text{Eq. 52.21}$$

in which V_0 is the Darcy velocity corresponding to the water phase/polymer flow calculated from equation [52.12](#) (or equation [52.13](#) if OPTIONS keyword item 228 is set to 1) without any shear effects. This intersection provides the shear-modified velocity for the given water phase/polymer flow and the relevant shear thinning/thickening multiplier.

Note that the shear data specified via the [PLYSHLOG](#) keyword must satisfy the constraint

$$\frac{d\ln M^{\text{ref}}}{d\ln V} > -1 \quad \text{Eq. 52.22}$$

which henceforth ensures a unique point of intersection. In view of equation 52.14, the reservoir flow rates in equation 52.1 and equation 52.2 are modified accordingly together with the associated derivatives, that is, the unsheared flows are multiplied by the reciprocal of the relevant shear multiplier calculated as per above. A similar transformation is likewise applied to the unsheared well component mobilities.

Approach II: Based on shear rate

In this second approach, the shear multiplier is specified instead as a function of water phase/polymer flow shear rate. Once again the shear data is specified using the `PLYSHLOG` keyword, for a specified set of reference conditions. Keyword `SHRATE` must also be specified in order to activate this option. In this case, the first column in each `PLYSHLOG` table corresponds to the shear rate, γ .

This approach follows Approach I very closely, but with the following modifications:

- The calculation of shear multiplier in equation 52.19 is done for each shear rate (as opposed to velocity) entered in the `PLYSHLOG` table.
- Take the natural logarithm of each shear rate in `PLYSHLOG` and take the natural logarithm of the corresponding viscosity multiplier calculated via equation 52.19. Construct the piecewise linear function through the resulting set of data points:

$$f : \ln \gamma \mapsto \ln Z \quad \text{Eq. 52.23}$$

- Locate the point of intersection of the above piecewise linear function and the straight line represented by:

$$\ln \gamma + \ln Z = \ln \gamma_0 \quad \text{Eq. 52.24}$$

in which γ_0 is the shear rate corresponding to the given inter-block water phase/polymer flow prior to shear thinning/thickening and is calculated as follows,

$$\gamma_0 = C_\gamma V_0 \sqrt{\frac{\varphi S_w}{K k_{rw}}} \quad \text{Eq. 52.25}$$

where:

V_0 is the unsheared inter-block velocity calculated from equation 52.12 (or from equation 52.13 if `OPTIONS` keyword item 228 is set to 1)

K is the absolute permeability

k_{rw} is the water relative permeability associated with the flow from the upwind cell

S_w is the water saturation in the upwind cell

φ is the average porosity of the two cells and

C_γ is the shear rate constant specified via the `SHRATE` keyword.

At a well connection, the transformation 52.25 is applied once again to convert, this time, from the velocity in the connection to the corresponding shear rate. Corresponding grid block quantities are used in order to carry out this default conversion which therefore replicates that applied in the reservoir flow calculation.

Note:

The pre-2014.1 conversion from velocity to shear rate used for reservoir flows is given by

$$\gamma_0 = C_\gamma V_0 \sqrt{\frac{\varphi}{KS_w k_{rw}}} \quad \text{Eq. 52.26}$$

and the pre-2014.1 conversion from velocity to shear rate for well connections is given by

$$\gamma_0 = \frac{C_\gamma V_0^{wc}}{D^{wc}} \quad \text{Eq. 52.27}$$

in which V_0^{wc} and D^{wc} represent, respectively, the unsheared velocity and the diameter in the given wellbore connection. Note that conversions 52.26 and 52.27 may be activated in conjunction with the new default velocity calculation (by setting OPTIONS keyword item 228 to 2) or activated in conjunction with the pre-2014.1 velocity calculation (by setting OPTIONS keyword item 228 to 3). The possible options and corresponding conversions are summarized in the following table.

	Default	OPTIONS item 228 set to 1	OPTIONS item 228 set to 2	OPTIONS item 228 set to 3
Conversions applied	Equations 52.12 and 52.25	Equations 52.13 and 52.25	Equations 52.12, 52.26 and 52.27	Equations 52.13, 52.26 and 52.27

Table 52.1: Options available for converting between (i) flow rate and velocity, and (ii) velocity and shear rate for models using the SHRATE keyword

The shear effect is skipped in those cases where a conversion from velocity to shear rate is not possible due to zero denominator terms associated with 52.25 and 52.26.

As with Approach I, the input PLYSHLOG data must satisfy the constraint

$$\frac{d\ln M^{ref}}{d\ln \gamma} > -1 \quad \text{Eq. 52.28}$$

which yields a unique point of intersection. Again, the unsheared reservoir flow rates and unsheared well component mobilities are modified by multiplying through by the reciprocal of the relevant shear thinning/thickening factor. The associated derivatives are modified in line with this shear effect.

Cases where the shear effect is omitted

Shear thinning/thickening is not applied in those cases where:

- The value of the unsheared velocity (or shear rate) is less than the minimum velocity (or shear rate) specified in PLYSHLOG.
- There is no significant effect due to the polymer concentration (case where $P = 1$)

In such cases, the shear viscosities are equal to the effective component viscosities, that is, $Z_w^* = Z_p^* = 1$.

Temperature effects

Various effects of temperature on the polymer flood model are considered. First, the viscosity of the polymer solution is now allowed to vary with temperature. This is analogous to the variation with salinity specified by [PLYVISCS](#), and two new keywords implement this: [PLYVISCT](#), which provides tables of temperature variation only as a function of polymer concentration, and [PLYVSCST](#), which includes both temperature and salinity variations. All these keywords (and [PLYVISC](#), which just considers the effect of polymer concentration) provide a multiplier for the given conditions which modifies the pure water viscosity specified in [PVTW](#).

Note that a temperature effect on the pure water viscosity can be specified with [WATVISCT](#), which is not a multiplier but a replacement viscosity value. The data input in [PVTW](#) is used only to provide a multiplier for the pressure variation (viscosibility) in this case. A warning is given if the new polymer viscosity keywords for temperature variation are used with [WATVISCT](#), but the cumulative effect of both specifications is allowed. It is for the user to decide which combination of inputs best fits the model.

Second, the polymer can degrade due to thermal effects. This is modeled as a half-life of the reduction of polymer concentration, and is input as a table of half-life against temperature by the keyword [PLYDHFLF](#). The equation 52.2 is modified by the addition of the term

$$Q_{HL} = -\alpha \left(V^* S_w C_p + \frac{1-\varphi}{\varphi} V \rho_r C_p^a \right) \quad \text{Eq. 52.29}$$

on the right-hand side, where α is related to the half-life T_H by $\alpha = \ln(2)/T_H$.

Thirdly, the adsorption of the polymer can also be temperature dependent. In this case, multiple occurrences of the [PLYADS](#) or [PLYADSS](#) keywords specifying the adsorbed concentration can be used. Each occurrence refers to a different temperature, which is specified by the use of the [PLYATEMP](#) keyword.

Finally, the residual resistance factor of the adsorbed polymer (keywords [PLYROCK](#) and [PLYKRRF](#)) can be made temperature dependent using the [PLYTRRF](#) keyword.

The temperature model must be activated with the keyword [TEMP](#) for these keywords to function.

See also [Ref. 22]

Temperature-sensitive popping polymer microgels

Various technologies (such as Bright Water™: see [Ref. 88] and [Ref. 39]) now exist to inject polymer microgels which have low impact on viscosity and residual resistance factor at low injection temperatures, but, when heated to reservoir temperature, “pop” (expand their molecular size considerably) to produce much higher solution viscosities and adsorbed material residual resistance factors. This allows them to block high permeability streaks (“thief zones”) and ensure better sweep efficiency.

In ECLIPSE 100, these two effects are modeled by separate keywords. Temperature-dependent polymer solution viscosity is represented by the two keywords [PLYVISCT](#), which provides tables of temperature variation only as a function of polymer concentration, and [PLYVSCST](#), which includes both temperature and salinity variations. To model the temperature sensitivity of these microgels, the multipliers in either keyword should increase markedly between injection and reservoir temperatures. The temperature dependence of the residual resistance factor is represented by the [PLYTRRF](#) keyword, which multiplies the residual resistance factor of the adsorbed polymer (keywords [PLYROCK](#) and [PLYKRRF](#)) by a factor dependent on the temperature. Again, the multiplier value should increase markedly between injection and reservoir temperatures.

The temperature model must be activated with the keyword [TEMP](#) for these keywords to function. This model will also allow rock conduction of heat from the reservoir rock to the cooler injected fluid and it is important that the conductivity ([THCONR](#)) is large enough to heat the polymer solution as it passes through the formation, for the gel-popping representation to work.

It should be noted that, while the microgel “popping” behavior is expected to be irreversible, the current modeling described here has more limited irreversibility. The adsorbed polymer should be prevented from returning to solution by setting item 4 of keyword [PLYROCK](#) to the value 2. The viscosity of the “popped” microgel cannot be prevented from returning to its low-temperature value if the solution is later cooled - which may happen by mixing with water or other polymer solution injected later in the process. It is hoped that the blocking effect of the initial viscosity increase will act to decrease such mixing and retain the blocking effect to later times.

For the temperature dependence of the residual resistance factor, there are additional factors governing how the irreversibility of the popping effect is represented. In principle, the cell containing the adsorbed polymer can be heated and cooled multiple times. As the cell heats, existing adsorbed polymer can expand further according to the temperature, as represented by the [PLYTRRF](#) data. If the cell is re-cooled by further injectant, and more polymer is adsorbed, then there will be at least two populations of adsorbate in the cell with two different RRF values. ECLIPSE 100 provides two approximations to this effect. In the first, the re-cooling effect is ignored, and the polymer is assumed to respond to the maximum temperature experienced by the cell while polymer is present. This allows for initial cooling by the injectant, while minimizing the memory usage required to represent adsorbate at multiple RRF values. This option uses only the [PLYTRRF](#) keyword.

The second approximation allows the existence of more than one adsorbate RRF state. As an example, in a given grid cell, polymer adsorbed at the start of injection, and subsequently heated by the reservoir, is assigned the RRF value of the reservoir temperature. Polymer injected later and adsorbed in the same cell at a cooler temperature is assigned the RRF value of the injection temperature. The total RRF value for that cell is a weighted average of the two different RRFs, with the amounts of adsorbed polymer providing the weights. The temperatures are specified by ranges between the values of the [PLYTRRF](#) table, and in this case the RRF multipliers are assumed constant in a given temperature range rather than being piece-wise linear.

This approximation is activated by the [PLYTRRFA](#) keyword, which also specifies the number of temperature intervals used. In this case, a separate adsorbed polymer concentration is stored for each cell and each interval. More intervals can be specified, to represent partial expansion of the microgel, but this will increase the amount of memory used. There is a trade-off between any performance degradation due to this extra memory compared to the improved representation of the polymer history, just as there is a trade-off between using piece-wise constant multipliers for the intervals in this option, compared with the piecewise linear representation in the previous option.

In this case, the part of the adsorbed polymer whose current RRF value corresponds to the nnnth temperature interval of the [PLYTRRF](#) table can be viewed using the mnemonic CABINnnn, where nnn has a format between 001 and 999 but must not exceed the [PLYTRRFA](#) argument value.

The polymer flood model and wells

Prior to the 2012.1 release, the polymer flood option was not compatible with crossflowing wells. From 2012.1, the polymer concentration is solved as an implicit variable in the well model. Brine concentration is also solved this way for cases with polymer and brine. This allows the standard well model to calculate polymer and brine crossflows. Pre-2012.1 behavior may be recovered using [OPTIONS](#) keyword [item 176](#). Polymer and brine are solved as tracers within the water phase rather than as a separate phase and travel with the same velocity as the water phase.

Multisegment and friction wells

If the wellbore friction Model or multisegment well model are used, then water viscosity may be calculated taking polymer concentration into account. The effective viscosity of the water/polymer mixture is calculated as described above in equation 52.10. The Todd-Longstaff mixing parameter ω and maximum polymer and brine concentrations are determined by the polymer mixing table number defined for the well. See the [WELSPECS](#) and [WELSPECL](#) keywords for details of how to specify the table number. The water viscosity may be further tuned using the [WPOLYRED](#) keyword.

Note: The effect of non-Newtonian rheology is not included in the wellbore.

Using the polymer flood model

The option is activated by the keyword [POLYMER](#) in the RUNSPEC section. If salt-sensitivity for polymer is required, this can be set by adding the keyword [BRINE](#) to the RUNSPEC section. By default the polymer model is not salt-sensitive.

The mixing parameter data is obligatory and should be provided using the keyword [PLMIXPAR](#). The maximum number of mixing parameter regions is set using the parameter [NPLMIX](#) in the keyword [REGDIMS](#). The associated mixing region for each grid cell is specified using the [PLMIXNUM](#) keyword. The mixing region for each well is specified using [WELSPECS](#) or [WELSPECL](#) as appropriate. The maximum polymer and salt concentrations to be used in calculating the effective fluid component viscosities are entered under the keyword [PLYMAX](#).

The viscosity of a fully mixed polymer solution needs to be defined as function of the solution concentration. This should be done using the keyword [PLYVISC](#). If the salt-sensitive option is activated (BRINE specified), then the keywords [SALTNODE](#) and [PLYVISCS](#) should be used to enter the two-dimensional table for the viscosity of the solution.

The polymer adsorption data should be entered using either with the keyword [PLYADS](#), with the [PLYADSS](#) keyword accompanied by the keyword [ADSALNOD](#), or with the [ADSORP](#) keyword with POLYMER as argument in the PROPS section. When the ADSORP keyword is used, the polymer adsorption can be set to depend on the salinity and rock permeability. In this case, with the standard brine option, the salt's mass concentration is directly used in the analytic model, and therefore the units of the coefficients need to be interpreted accordingly.

Note that the multi-component brine model ([ECLMC](#) keyword in the RUNSPEC section) cannot currently be used with the polymer/brine flood model.

Other polymer-rock parameters such as the rock mass density used in the adsorption calculation, the dead pore volume and the residual resistance factor are input using the keyword [PLYROCK](#).

If required, individual cell values can be set for the rock mass density, the residual resistance factor, and the maximum adsorbed polymer using [PLYRMDEN](#), [PLYKRRF](#) and [PLYCAMAX](#) respectively.

Non-Newtonian rheology can be activated by setting one of the following:

- The [PLYSHEAR](#) keyword in the PROPS section for model 1.
- Opting for the Herschel-Bulkley model as described in "Non-Newtonian fluid flows".
- The [PLYSHLOG](#) keyword in the PROPS section for model 3 (approach I).
- The [PLYSHLOG](#) and [SHRATE](#) keywords in the PROPS section for model 3 (approach II).

If Model 1 is activated, the shear thinning/thickening data consists of tables of viscosity reduction as a function of local velocity. The values of $1/(\phi \cdot A)$ used to calculate the velocity can be printed out using the mnemonic POLYMER in the [RPTGRID](#) keyword. The representative radius used in the well inflow calculation is output with the well connection data.

If model 3 (approach I) is activated, the shear thinning/thickening data consists of tables of shear multipliers as a function of local velocity, and these correspond to a specified set of reference conditions. If model 3 (approach II) is activated, the multipliers are specified as a function of local shear rate instead.

Note however that if non-Newtonian rheology is not an important effect, it should be omitted as the calculation of the velocity dependent viscosity can significantly increase the CPU requirements for the run.

When the brine option is activated in conjunction with polymer, the initial salt concentrations are specified with either the [SALT](#) keyword (if the enumeration method is used for initialization) or the [SALTVD](#) keyword (if the [EQUIL](#) keyword is used). The initial polymer concentration and the initial adsorbed polymer concentration are both automatically set to zero by default; however the [SPOLY](#) keyword can be used to override this.

If the model contains analytic aquifers, the initial salt concentration in each aquifer should be specified using the appropriate keywords [AQUCT](#), [AQUFET](#) or [AQUFETPP](#). Since the aquifer volume is large, the polymer concentration is assumed to be always zero - any polymer entering the aquifer due to backflow is adsorbed by the aquifer rock.

If the standard brine is used, the polymer and salt well injection concentration should be set using the keyword [WPOLYMER](#) in the SCHEDULE section.

Note that the multi-component brine model ([ECLMC](#) keyword in the RUNSPEC section) cannot currently be used with the Polymer/Brine Flood Model (that is, the combination of polymer and brine using the [SALTNODE](#) keyword; ECLMC can be used with the basic Polymer Flood Model).

Restrictions

The IMPES solution algorithm cannot currently be used with the Polymer option.

If a cell is defined with zero pore volume in the GRID section, then making it active by changing the pore volume in the EDIT section using [PORV](#), is not allowed with the polymer option.

When using [RESTART](#) with the polymer flood option the base case must have neither polymer nor brine, or it must have the same polymer and brine choices as the restart. This is in part due to the fact that brine is solved as a tracer in the absence of polymer, but as a fully implicit variable in the presence of polymer. If a base case with brine alone is intended for use with a restart using polymer, then the [POLYMER](#) keyword must be added to the base case as well. The following table shows the compatible base cases and restart cases.

	No POLYMER or BRINE (Base case)	POLYMER (Base case)	BRINE (Base case)	POLYMER and BRINE (Base case)
No POLYMER or BRINE (Restart case)	Compatible	Not compatible	Not compatible	Not compatible
POLYMER (Restart case)	Compatible	Compatible	Not compatible	Not compatible

	No POLYMER or BRINE (Base case)	POLYMER (Base case)	BRINE (Base case)	POLYMER and BRINE (Base case)
BRINE (Restart case)	Compatible	Not compatible	Compatible	Not compatible
POLYMER and BRINE (Restart case)	Not compatible	Not compatible	Not compatible	Compatible

Table 52.2: Polymer and Brine restart compatibility

The ECLIPSE 300 polymer flood model

In ECLIPSE 300 polymer is modeled as a water component as described in "[Water components](#)". At least one water component must be identified as a polymer ([CWTYPE](#)) and can be input into the reservoir by defining the injection water composition ([WINJW](#)).

In contrast with the ECLIPSE 100 polymer flood model, in ECLIPSE 300 the polymer is assumed to be fully mixed with the water and to move at the same velocity as the water phase.

Polymer can be adsorbed onto the rock surface. For ECLIPSE 300 a solid component can be used to represent adsorbed polymer, and adsorption and polymer can be modeled using a chemical reaction. See "[Chemical Reactions](#)" and "[Solid Phase](#)".

Treatment of fluid viscosities

The viscosity terms used in the fluid flow equations contain the effects of the viscosity change of the water phase due to the presence of polymer and any other water component defined in the model as described in "[Water components](#)". This is, the water phase viscosity is calculated as a linear combination of the viscosity of each water component as specified in equation [77.3](#).

The variation of the water phase viscosity with polymer concentration can be specified using the keyword [PLYVISC](#). In this case, the water phase viscosity is given by $m(C_p)\mu_w$ where the multiplier m is a function of polymer concentration C_p as specified in [PLYVISC](#) and μ_w is calculated using equation [77.3](#).

In salt sensitive studies, the variation of the viscosity of the water phase with polymer concentration can be modeled using the keyword [PLYVISCS](#). The water phase viscosity is given by $m(C_p, E_s)\mu_w$ where the multiplier m is a function of polymer concentration C_p and effective salinity E_s as specified by the keywords [PLYVISCS](#) and [ESPNODE](#). The effective salinity is calculated as a function of the water component concentrations as defined by the keyword [SALTEFF](#)

The viscosity of each water component at their reference pressure can be defined with the keywords [VREFW](#) and [PREFW](#). However, for the polymer model it is advised that these parameters are set to those of water so that the polymer concentration effect of the water phase viscosity is only defined by the keywords [PLYVISC](#) or [PLYVISCS](#).

Treatment of polymer adsorption

Adsorbed polymer can be modeled using the [SOLID](#) model. The [COMPS](#) or [COMPSOL](#) keyword should be used to set the number of non-water components (including solids). The [CVTYPE](#) keyword should be used

to identify the solid component. The properties of adsorbed polymer are defined using the solid keywords, see "[Solid PROPS keywords](#)".

The polymer absorption and desorption processes can be modeled using chemical reactions, see "[Chemical Reactions](#)". The **REACTION** keyword should be used to specify the number of chemical reactions to be simulated. Different chemical reactions are available to model the absorption of polymer. In particular, dissolution processes can be modeled using "[Equilibrium deviation reaction rates](#)", see "[Adsorption of dissolved water components](#)".

In ECLIPSE 100 absorption is treated as an instantaneous process. In ECLIPSE 300 this effect could be modeled by setting a large constant reaction rate with the keyword **REACRATE**. This might have a significant impact in convergence.

Restrictions

The following features of the ECLIPSE 100 polymer flood model are currently not available in the ECLIPSE 300 polymer flood model:

- The effect of Todd-Longstaff mixing parameter to model the physical dispersion at the leading edge of the slug and the fingering effects at the rear edge of the slug.
- The effect of temperature variations on the behavior of the polymer solution.
- The effect of reduced relative permeability of the polymer solution.
- The effect of dead pore volume on increasing the velocity of the polymer relative to the water phase velocity.
- The effect of non-Newtonian rheology.

Summary of ECLIPSE 100 keywords

RUNSPEC

Keyword	Description
BRINE	Enables the Polymer Salt-sensitivity option.
REGDIMS	Item 10 sets the maximum number of mixing parameter regions.
POLYMER	Enables the Polymer Flood Model.

GRID

Keyword	Description
RPTGRID	Controls output from the GRID section. The POLYMER argument outputs a map of the $1/(poro*area)$ values used for shear thinning calculations.

PROPS

Keyword	Description
ADSALNOD	Salt concentration nodes for polymer solution adsorption (by saturation region).
PLYADS	Polymer adsorption isotherms without salinity dependence.
PLYADSS	Polymer adsorption isotherms with salinity dependence.
ADSORP	Analytical adsorption isotherms with salinity and permeability dependence.
PLYMAX	Polymer/salt concentrations for mixing calculations.
PLYROCK	Specifies the polymer-rock properties.
PLYSHEAR	Polymer shear thinning/thickening data.
PLYSHLOG	Polymer shear thinning/thickening data (alternative logarithmic model).
SHRATE	Activate the shear rate form of the PLYSHLOG shear thinning/thickening option.
PLYVISC	Polymer solution viscosity function.
PLYVISCS	Polymer/salt solution viscosity function.
PLYVISCT	Polymer solution viscosity function with temperature
PLYVSCST	Polymer/salt solution viscosity function with temperature.
PLYDHFLF	Polymer half life for thermal degradation.
RPTPROPS	Controls output from the PROPS section, including arguments for controlling the properties output for the Polymer Flood Model.
SALTNODE	Salt concentration nodes for polymer solution viscosity (by pressure region).
TEMPNODE	Temperature nodes for polymer solution viscosity.
PLMIXPAR	Todd-Longstaff mixing parameter.
PLYKRRF	Residual resistance factor (RRF) of individual grid blocks.
PLYTRRF	Temperature dependence of the RRF by region.
PLYTRRFA	Number of temperature intervals activating the second mode of PLYTRRF.
PLYCAMAX	Maximum polymer adsorption value of individual grid blocks.
PLYRMDEN	Mass density of individual grid blocks.
PLYATEMP	Temperature specifications for temperature dependence of polymer adsorption tables PLYADS or PLYADSS

REGIONS

The keywords are:

- [PLMIXNUM](#) Polymer mixing region numbers.

SOLUTION

Keyword	Description
RPTSQL	Controls output from the SOLUTION section. You can set the following arguments: <ul style="list-style-type: none"> FIPPLY outputs polymer/salt fluid in place reports. PBLK outputs initial grid block polymer concentration values. PLYADS outputs the current adsorbed polymer concentration on the formation rock. RK outputs the water relative permeability reduction factor. SALT outputs the initial grid block salt concentration values SALTVTD outputs the tables of initial salt concentration versus depth.
SALT	Initial salt concentrations.
SALTVTD	Salt concentration versus depth for equilibration.
SPOLY	Initial polymer concentrations.

SUMMARY

Refer to [ECLIPSE 100 Polymer flood SUMMARY output controls](#) in the *ECLIPSE Reference Manual*.

SCHEDULE

Keyword	Description
WELSPECS	Polymer mixing region number for wells.
WELSPECL	Polymer mixing region number for wells.
WSEGINIT	Initial segment polymer and brine concentrations for multisegment wells.
RPTSCHE	Controls output from the SCHEDULE section. You can set the following arguments: <ul style="list-style-type: none"> FIPSALT: polymer/salt fluid in place reports. PBLK: polymer concentration values. PLYADS: adsorbed polymer concentration on the formation rock. RK: water relative permeability reduction factor. SALT: salt concentration values.

Keyword	Description
RPTRST	<p>Argument <code>VISC</code> allows to output extra viscosities when Polymer is active. You can obtain the following output:</p> <ul style="list-style-type: none"> • <code>EPVIS</code>: effective polymer viscosity (Polymer option). • <code>EMVIS</code>: effective mixture (water/polymer) viscosity (Polymer option). • <code>EWV_POL</code>: effective water viscosity (Polymer option). • <code>EWV_SAL</code>: effective water viscosity due to salt concentration (Brine option). • <code>PLADALK</code>: polymer adsorption multiplier as a function of alkaline concentration. • <code>SFIPPLY/SFIPPOL</code>: polymer in place (mass) • <code>RRFFAC</code>: residual resistance factor. • <code>TRXBYRRF</code>: I-direction transmissibility divided by the residual resistance factor. • <code>TRYBYRRF</code>: J-direction transmissibility divided by the residual resistance factor. • <code>TRZBYRRF</code>: K-direction transmissibility divided by the residual resistance factor. • <code>SHWVISI, SHWVISJ, SHWVISK</code>: shear viscosity of the water/polymer solution due to shear thinning/thickening in the positive I-, J- and K-directions, respectively (<code>PLYSHEAR</code> and <code>PLYSHLOG</code> keywords). • <code>POLYMAX</code>: maximum polymer concentration. • <code>RRFTMAX</code>: RRF maximum temperature. • <code>POLYMER</code>: polymer concentration.
WPOLYMER	Sets polymer/salt concentrations for injection wells. The arguments can be used as User Defined Arguments (UDAs).
WPOLYRED	Sets the polymer/water mixture viscosity reduction parameter.

Summary of ECLIPSE 300 keywords

The polymer component and water properties must be defined by the water component keywords. See "[Water component keywords](#)".

A solid component can be used to represent adsorbed polymer. See "[Solid Phase](#)".

Adsorption and desorption could then be modeled using a pair of chemical reactions. See "[Chemical Reactions](#)".

PROPS

The keywords are:

Keyword	Description
ESPNODE	Effective salinity nodes.
PLYOPTS	Polymer model options.

Keyword	Description
PLYVISC	Polymer solution viscosity function.
PLYVISCS	Polymer/salt solution viscosity function.
SALTEFF	Set of coefficients to compute the effective salinity.
RPTPROPS	Controls output from the PROPS section. Arguments: <ul style="list-style-type: none">• PLYVISC: echoes the polymer viscosity table specified by the PLYVISC keyword.• PLYVISCS: echoes the polymer viscosity table specified by the PLYVISCS keyword.

SOLUTION

Keyword	Description
RPTSOL	Controls output from the SOLUTION section. Arguments: <ul style="list-style-type: none">• POLY: polymer concentration values.• POLYVM: polymer solution viscosity multiplier.• ESALTP: effective salinity values.
RPTRST	Arguments: <ul style="list-style-type: none">• POLY: polymer concentration values.• POLYVM: polymer solution viscosity multiplier.• ESALTP: effective salinity values.

SUMMARY

Refer to [ECLIPSE 300 Polymer flood SUMMARY output controls](#) in the *ECLIPSE Reference Manual*.

Note that "Water component", "Solid Phase" and "Chemical Reactions" summary vectors may be used to provide additional output.

SCHEDULE

Keyword	Description
PLYVISC	Polymer solution viscosity function
PLYVISCS	Polymer solution viscosity function.

Keyword	Description
RPTSCHE	Controls output from the SOLUTION section. Arguments: <ul style="list-style-type: none">• POLY: polymer concentration values.• POLYVM: polymer solution viscosity multiplier.• ESALTP: effective salinity values.
RPTRST	Arguments: <ul style="list-style-type: none">• POLY: polymer concentration values.• POLYVM: polymer solution viscosity multiplier.• ESALTP: effective salinity values.

ECLIPSE 100 examples

Example 1

This is a 1-D example modeling water injection into oil at adverse mobility ratios. To aid recovery, a polymer slug is injected into the reservoir some time after the primary water drive. Water injection is resumed after the polymer injection period. The performance of the polymer drive is assumed not be salt sensitive in this study.

RUNSPEC

```
RUNSPEC =====
TITLE
Polymer flood model test problem - oil/water/polymer

DIMENS
15    1    1    /
OIL
WATER
POLYMER
METRIC

TABDIMS
1    1    50    2    2    20    1    20    /
REGDIMS
2    1    0    0    0    1    /
WELLDIMS
2    1    1    2    /
START
1 'JAN' 1983    /
FMTOUT
FMTIN
```

GRID

```
GRID      =====
EQUALS
'DX'     100    /
'DY'     100    /
'DZ'     20     /
'PERMX'  100    /
'PERMY'  100    /
'TOPS'   3990.0 /
'PORO'   0.5    /
/
RPTGRID
'PORO'  'PORV'  /
```

PROPS

```
PROPS      =====
SWFN
  .20    .0    4.0
  .7     0.7   2.0
  1.0    1.0   0.0
/
SOF2
  .3000  .0000
  .8000  1.0000
/
PVTW
  .0    1.0   3.03E-06  .5   0.0 /
PVDO
  .0    1.0    5.0
  8000.0  .92   5.0
/
ROCK
  4000.0   .30E-05 /
DENSITY
  52.0000  64.0000   .04400 /
```

Define the viscosity of the polymer solution as a function of the polymer concentration. Note that the function values refer to the multiplication factor for the pure water viscosity to obtain the true polymer solution viscosity.

```
PLYVISC
  0.0   1.0
  7.0   20.0 /
```

Define the polymer-rock properties. A series of entries is required for each rock region. These values represent the inaccessible pore volume, the residual resistance factor, the rock mass density, the adsorption index and the maximum polymer concentration, to be used in the calculation of the actual resistance factor.

```
PLYROCK
  0.15  2.67  1000.0  2  0.0035 /
```

Define the polymer adsorption properties. A table of values is required for each rock region. Each table represents the equilibrium concentration of polymer adsorbed on the rock and the concentration of polymer in the surrounding aqueous phase.

```
PLYADS
 0.0  0.000
 2.0  0.0015
 8.0  0.0025 /
```

Mixing parameter to indicate the degree of segregation between the polymer solution and the water at the trailing edge of the slug. In this case the polymer and water in each grid cell are fully mixed. A value of the mixing parameter should be entered for each mixing region.

```
PLMIXPAR
 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, the 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'   /
/
COMPDATA
'I'      ' 1  1  1  1 'OPEN'   0  .0  1.0 /
'P'      '15 1  1  1 'OPEN'   0  .0  1.0 /
/
WCONPROD
'P'  'OPEN'  'LRAT'  3* 1000.0 /
/
WCONINJ
'I'  'WAT'  'OPEN'  'RATE'  1000.0 /
/
```

Initial water flood

Specify polymer concentration in the injection stream of well 'I'. The second figure refers to the injection salt concentration and is ignored in this run.

```
WPOLYMER
'I'  0.0  0.0 /
/
TSTEP
1.0 99.0 2*100.0 /
```

Polymer slug in fresh water

Specify the polymer concentration in the injection stream of well 'I'.

```
WPOLYMER
'I'  5.0  0.0  /
/
TSTEP
5*100.0 /
```

Chase water period

Specify the polymer concentration in the injection stream of well 'I'.

```
WPOLYMER
'I'  0.0  0.0  /
/
TSTEP
10*100.0  /
END
```

Example 2

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.

```

PLMIXPAR
  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

```
PLMIXNUM
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

```

ECLIPSE 300 examples

Example 1

A polymer slug is injected into the reservoir and 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
Test for Polymer in E300
FIELD
DIMENS
30 30 10/
WATER

```

```

OIL
GAS
COMPW
2/
COMPS
6 /
TABDIMS
1 1 50 50 1 20 /
WELLDIMS
2 10 1 /
START
21 'DEC' 2013 /

```

GRID

```

GRID =====
INIT
GRIDFILE
1 1 /
TOPS
900*2000/
DX
9000*20/
DY
9000*20/
DZ
9000*10/
PORO
9000*0.2/
PERMX
9000*100/
PERMY
9000*100/
PERMZ
9000*100/

```

PROPS

```

PROPS =====
EOS
PR/
PRCORR
RTEMP
160/
STCOND
60 14.7/
CNAMES
C1 C3 C6 C10 C15 C20 /

```

```

TCRIT
343.0
665.7
913.4
1111.8
1270.0
1380.0 / 

PCRIT
667.8
616.3
436.9
304.0
200.0
162.0 / 

ZCRIT
0.290
0.277
0.264
0.257
0.245
0.235 / 

MW
16.04
44.10
86.18
149.29
206.00
282.00 / 

ACF
0.013
0.1524
0.3007
0.4885
0.6500
0.8500 / 

BIC
0.0
0.0 0.0
0.0 0.0 0.0
0.05 0.005 0.0 0.0
0.05 0.005 0.0 0.0 0.0 / 

-- Rock fluid properties
SWFN
0.2 0 45.0
0.2899 0.0022 19.03
0.3778 0.0180 10.07
0.4667 0.0607 4.90
0.5556 0.1438 1.8
0.6444 0.2809 0.5
0.7000 0.4089 0.05
0.7333 0.4855 0.01
0.8222 0.7709 0.0
0.9111 1.0000 0.0
1.00 1.0000 0.0 / 

SGFN
0.00 0.000 0.0
0.05 0.000 0.0
0.0889 0.001 0.0
0.1778 0.010 0.0
0.2667 0.030 0.001
0.3556 0.05 0.001
0.4444 0.10 0.03
0.5333 0.20 0.8
0.6222 0.35 3.0
0.65 0.39 4.0
0.7111 0.56 8.0
0.80 1.0 30.0 /

```

```

SOF3
0.00 0.0 0.0
0.0889 0.0 0.0
0.1500 0.0 0.0
0.1778 0.0 0.0110
0.2667 0.0 0.0370
0.3 0.0 0.0560
0.3556 0.0123 0.0878
0.4444 0.0835 0.1715
0.5333 0.2178 0.2963
0.6222 0.4153 0.4705
0.7111 0.6769 0.7023
0.80 1.0 1.0 /

ZMFVD
1000.0 0.5 0.03 0.07 0.2 0.15 0.05
10000.0 0.5 0.03 0.07 0.2 0.15 0.05 /

DENSITY
1* 62.4 1* /

ROCK
75.0 5.0E-04 /

```

Define the water component properties. Here there are two water components: fresh water and polymer.

```

-- Water and polymer properties
WNAMES
PURE POLYMER /

CWTYPE
1* POLY /

CREFW
3.E-8 3.E-8 /

MW
18 1E6 /

PREFW
2000 2000 /

VREFW
1.2 1.2 /

```

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   1
1   2.25
2   5.5
3   7.75
4   10  /

```

```

PLYOPTS
2.E-6 /

```

The polymer properties can be echoed using keyword **RPTPROPS**.

```

RPTPROPS
'PLYVISC'  /

```

SOLUTION

```
SOLUTION =====
PRESSURE
9000*2000/

SWAT
9000*0.3/

SGAS
9000*0/

WMF
9000*1 9000*0/

XMF
9000*0.5 9000*0.03 9000*0.07 9000*0.2 9000*0.15 9000*0.05/

YMF
9000*0 9000*0 9000*0 9000*0 9000*0 9000*0/
```

The polymer properties can be reported using the [RPTRST](#) and the [RPTSOL](#) keywords.

```
RPTRST
'BASIC=2' POLY POLYVM /
RPTSOL
POLY POLYVM /
```

SUMMARY

```
SUMMARY =====
FWMIR
/
FWMIT
/
FWMPR
/
FWMPT
/
FAMF
/
WWMPR
PROD /
/
WWMIR
INJ /
/
BAMF
 1 1 5 /
 5 5 5 /
10 10 5 /
/
BMLST
 1 1 5 /
 5 5 5 /
10 10 5 /
/
BMLSC
 1 1 5 /
 5 5 5 /
10 10 5 /
/
BVWAT
 1 1 5 /
 5 5 5 /
10 10 5 /
/
```

```

BPOLY
 1 1 5 /
 5 5 5 /
10 10 5 /
/
BPOLYVM
 1 1 5 /
 5 5 5 /
10 10 5 /
/
ALL
RUNSUM

```

SCHEDULE

```

SCHEDULE =====
RPTSCHED
MLSC MWAT POLY POLYVM /

RPTRST
BASIC=3 FREQ=1 MLSC MWAT POLY POLYVM /

WELLSPEC
INJ FIELD 1 1 3500 1* 0 /
PROD FIELD 30 30 3500 1* 0 /
/

COMPDAT
INJ 1* 1* 1 10 OPEN /
PROD 1* 1* 1 10 OPEN /
/

WCONINJE
INJ WATER OPEN RATE 10000 /
/
-- Definition of the injection streams
WELLSTRW
FRESH 1.0 0.0 /
POLYM 0.9999998 2E-7 /
/
-- Injection of polymer slug
WINJW
'I*' STREAM POLYM /
/
DATES
25 DEC 2013 /
/
WCONPROD
PROD OPEN BHP 1* 1* 1* 1* 1* 200 /
/
DATES
1 FEB 2014 /
/
-- Update the properties of the polymer
PLYVISC
0   1
1   2.0
2   5.2
3   7.4
4   9.5 /
/
DATES
1 MAR 2014 /
1 APR 2014 /

```

```

1 MAY 2014 /
1 JUN 2014 /
/
WCONINJE
INJ WATER OPEN RATE 10000 /
/
-- Resume fresh water injection
WINJW
'I*' STREAM FRESH /
/
DATES
1 JUL 2014 /
1 AUG 2014 /
1 SEP 2014 /
1 OCT 2014 /
1 NOV 2014 /
1 DEC 2014 /
/
END

```

Example 2

A polymer slug is injected into the reservoir and water injection is resumed after the polymer injection period. In this study the injected polymer is salt sensitive.

RUNSPEC

```

RUNSPEC =====
TITLE
Test for Polymer in E300 with salinity effects
FIELD
DIMENS
30 30 10/
WATER
OIL
GAS
COMPW
4/
COMPS
6 /
TABDIMS
1 1 50 50 1 20 /
WELLDIMS
2 10 1 /
START
21 'DEC' 2013 /

```

GRID

```

GRID =====
INIT

```

```

GRIDFILE
1 1 /

TOPS
900*2000/

DX
9000*20/

DY
9000*20/

DZ
9000*10/

PORO
9000*0.2/

PERMX
9000*100/

PERMY
9000*100/

PERMZ
9000*100/

```

PROPS

```

PROPS =====

EOS
PR/

PRCORR

RTEMP
160/

STCOND
60 14.7/

CNAMES
C1 C3 C6 C10 C15 C20 /

TCRIT
343.0
665.7
913.4
1111.8
1270.0
1380.0 /

PCRIT
667.8
616.3
436.9
304.0
200.0
162.0 /

ZCRIT
0.290
0.277
0.264
0.257
0.245
0.235 /

MW
16.04

```

```

44.10
86.18
149.29
206.00
282.00 /

ACF
0.013
0.1524
0.3007
0.4885
0.6500
0.8500 /

BIC
0.0
0.0 0.0
0.0 0.0 0.0
0.05 0.005 0.0 0.0
0.05 0.005 0.0 0.0 0.0 /

-- Rock fluid properties
SWFN
0.2 0 45.0
0.2899 0.0022 19.03
0.3778 0.0180 10.07
0.4667 0.0607 4.90
0.5556 0.1438 1.8
0.6444 0.2809 0.5
0.7000 0.4089 0.05
0.7333 0.4855 0.01
0.8222 0.7709 0.0
0.9111 1.0000 0.0
1.00 1.0000 0.0 /

SGFN
0.00 0.000 0.0
0.05 0.000 0.0
0.0889 0.001 0.0
0.1778 0.010 0.0
0.2667 0.030 0.001
0.3556 0.05 0.001
0.4444 0.10 0.03
0.5333 0.20 0.8
0.6222 0.35 3.0
0.65 0.39 4.0
0.7111 0.56 8.0
0.80 1.0 30.0 /

SOF3
0.00 0.0 0.0
0.0889 0.0 0.0
0.1500 0.0 0.0
0.1778 0.0 0.0110
0.2667 0.0 0.0370
0.3 0.0 0.0560
0.3556 0.0123 0.0878
0.4444 0.0835 0.1715
0.5333 0.2178 0.2963
0.6222 0.4153 0.4705
0.7111 0.6769 0.7023
0.80 1.0 1.0 /

ZMFVD
1000.0 0.5 0.03 0.07 0.2 0.15 0.05
10000.0 0.5 0.03 0.07 0.2 0.15 0.05 /

DENSITY
1* 62.4 1* /

ROCK
75.0 5.0E-04 /

```

Define the water component properties. Here there are four water components: fresh water, two salts and polymer.

```
-- Water and polymer properties
WNAMES
PURE NaCl CaCO3 POLYMER /

CWTYPE
1* 1* 1* POLY /

CREFW
3.E-8 3.E-8 3.E-8 3.E-8 /

MW
18 58.44 100.0869 1E6 /

PREFW
2000 2000 2000 2000 /

VREFW
1.2 1.2 1.2 1.2 /
```

Define the effective salinity as the concentration of water components two and three:

```
-- Effective salinity
SALTEFF
-- model    effective      a_0  a_1  a_2  a_3  a_4  b_0  b_1  b_2  b_3  b_4
-- type     salinity type   POLY      1       0.0  0.0  1.0  1.0  0.0  1.0  0.0  0.0  0.0  /

```

Define the viscosity of the polymer solution as a function of the polymer concentration and effective salinity:

```
ESPNODE
0
10
20/

PLYVISCS
0 1
1.05
1.1 /
1 2.25
2
1.75 /
2 5.5
5
4.5 /
3 7.75
7
6.3 /
4 10
9.2
8.1 /
/
PLYOPTS
2.E-6 /
```

The polymer properties can be echoed using keyword **RPTPROPS**.

```
RPTPROPS
'PLYVISCS' /
```

SOLUTION

```

SOLUTION =====

PRESSURE
9000*2000/

SWAT
9000*0.3/

SGAS
9000*0/

WMF
9000*0.995 9000*0.005 9000*0 9000*0/

XMF
9000*0.5 9000*0.03 9000*0.07 9000*0.2 9000*0.15 9000*0.05/

YMF
9000*0 9000*0 9000*0 9000*0 9000*0 9000*0/

```

The polymer properties can be reported using the [RPTRST](#) and the [RPTSOL](#) keywords.

```

RPTRST
'BASIC=2' POLY POLYVM ESALTP /

RPTSOL
POLY POLYVM ESALTP /

```

SUMMARY

```

SUMMARY =====

FWMIR
/
FWMIT
/
FWMPR
/
FWMPT
/
FAMF
/
WWMPR
PROD /
/
WWMIR
INJ /
/
BAMF
 1 1 5 /
 5 5 5 /
10 10 5 /
/
BMLST
 1 1 5 /
 5 5 5 /
10 10 5 /
/
BMLSC
 1 1 5 /
 5 5 5 /
10 10 5 /
/
BVWAT
 1 1 5 /
 5 5 5 /
10 10 5 /
/

```

```

BPOLY
 1 1 5 /
 5 5 5 /
10 10 5 /
/
BPOLYVM
 1 1 5 /
 5 5 5 /
10 10 5 /
/
BESALTP
1 1 5 /
5 5 5 /
10 10 5 /
/
ALL
RUNSUM

```

SCHEDULE

```

SCHEDULE =====
RPTSCHE
MLSC MWAT POLY POLYVM ESALTP/

RPTRST
BASIC=3 FREQ=1 MLSC MWAT POLY POLYVM ESALTP/

WELLSPEC
INJ FIELD 1 1 3500 1* 0 /
PROD FIELD 30 30 3500 1* 0 /
/

COMPDAT
INJ 1* 1* 1 10 OPEN /
PROD 1* 1* 1 10 OPEN /
/

WCONINJE
INJ WATER OPEN RATE 10000 /
/

-- Definition of the injection streams
WELLSTRW
FRESH 1.0 0.0 0.0 0.0/
BRINE 0.99 0.005 0.005 0.0 /
POLYM 0.9899998 0.005 0.005 2E-7 /
/

-- Injection of Brine
WINJW
'I*' STREAM BRINE /
/

DATES
25 DEC 2013 /
/

-- Injection of Polymer Slug
WINJW
'I*' STREAM POLYM /
/

WCONPROD
PROD OPEN BHP 1* 1* 1* 1* 1* 200 /
/

DATES
1 FEB 2014 /
1 MAR 2014 /

```

```

1 APR 2014 /
1 MAY 2014 /
1 JUN 2014 /
/

WCONINJE
INJ WATER OPEN RATE 10000 /
/

-- Resume fresh water injection
WINJW
'I*' STREAM FRESH /
/

DATES
1 JUL 2014 /
1 AUG 2014 /
1 SEP 2014 /
1 OCT 2014 /
1 NOV 2014 /
1 DEC 2014 /
/

END

```

Example 3

A salt-sensitive polymer slug is injected into the reservoir and water injection is resumed after the polymer injection period. Polymer adsorption is also included in the model.

RUNSPEC

```

RUNSPEC =====

TITLE
Test for Polymer in E300 with salinity and adsorption effects

FIELD

DIMENS
30 30 10 /

WATER

OIL

GAS

SOLID

COMPW
4 /

COMPS
7 /

FULLIMP

REACTION
1 1 1* 1* 1 /

TABDIMS
1 1 50 50 1 20 /

WELLDIMS
2 10 1 /

START

```

```
21 'DEC' 2013 /
```

GRID

```
GRID      =====
INIT
GRIDFILE
1 1 /
TOPS
900*2000/
DX
9000*20/
DY
9000*20/
DZ
9000*10/
PORO
9000*0.2/
PERMX
9000*100/
PERMY
9000*100/
PERMZ
9000*100/
```

PROPS

```
PROPS      =====
EOS
PR/
PRCORR
RTEMP
160/
STCOND
60 14.7/
CNAMES
C1 C3 C6 C10 C15 C20 APOLY/
CVTYPE
6* SOLID /
TCRIT
343.0
665.7
913.4
1111.8
1270.0
1380.0
1380.0 /
PCRIT
667.8
616.3
436.9
```

```

304.0
200.0
162.0
162.0 / 

ZCRIT
0.290
0.277
0.264
0.257
0.245
0.235
0.235 / 

MW
16.04
44.10
86.18
149.29
206.00
282.00
1E5 / 

ACF
0.013
0.1524
0.3007
0.4885
0.6500
0.8500
0.8500 / 

ZI
0.5 0.03 0.07 0.2 0.15 0.05 0/ 

BIC
0.0
0.0 0.0
0.0 0.0 0.0
0.05 0.005 0.0 0.0
0.05 0.005 0.0 0.0 0.0
0.0 0.0 0.0 0.0 0.0 0.0 / 

-- Rock fluid properties
SWFN
0.2 0 45.0
0.2899 0.0022 19.03
0.3778 0.0180 10.07
0.4667 0.0607 4.90
0.5556 0.1438 1.8
0.6444 0.2809 0.5
0.7000 0.4089 0.05
0.7333 0.4855 0.01
0.8222 0.7709 0.0
0.9111 1.0000 0.0
1.00 1.0000 0.0 / 

SGFN
0.00 0.000 0.0
0.05 0.000 0.0
0.0889 0.001 0.0
0.1778 0.010 0.0
0.2667 0.030 0.001
0.3556 0.05 0.001
0.4444 0.10 0.03
0.5333 0.20 0.8
0.6222 0.35 3.0
0.65 0.39 4.0
0.7111 0.56 8.0
0.80 1.0 30.0 / 

SOF3
0.00 0.0 0.0
0.0889 0.0 0.0
0.1500 0.0 0.0

```

```

0.1778 0.0 0.0110
0.2667 0.0 0.0370
0.3 0.0 0.0560
0.3556 0.0123 0.0878
0.4444 0.0835 0.1715
0.5333 0.2178 0.2963
0.6222 0.4153 0.4705
0.7111 0.6769 0.7023
0.80 1.0 1.0 /

ZMFVD
1000.0 0.5 0.03 0.07 0.2 0.15 0.05 0
10000.0 0.5 0.03 0.07 0.2 0.15 0.05 0 /

DENSITY
1* 62.4 1* /

SDREF
6* 970.0 /

ROCK
75.0 5.0E-04 /

```

Define the water component properties. Here there are four water components: fresh water, two salts and polymer.

```

-- Water and polymer properties
WNAMES
PURE NaCl CaCO3 POLYMER /

CWTYPE
1* 1* 1* POLY /

CREFW
3.E-8 3.E-8 3.E-8 3.E-8 /

MW
18 58.44 100.0869 1E6 /

PREFW
2000 2000 2000 2000 /

VREFW
1.2 1.2 1.2 1.2 /

```

Define the effective salinity as the concentration of water components two and three:

```

-- Effective salinity
SALTEFF
-- model    effective      a_0  a_1  a_2  a_3  a_4  b_0  b_1  b_2  b_3  b_4
-- type    salinity type
POLY      1           0.0  0.0  1.0  1.0  0.0  1.0  0.0  0.0  0.0  0.0  /

```

Define the viscosity of the polymer solution as a function of the polymer concentration and effective salinity:

```

ESPNODE
0
10
20/

PLYVISCS
0   1
     1.05
     1.1   /
1   2.25
2
     1.75  /
2   5.5
5

```

```

        4.5   /
3    7.75
    7
6.3   /
4    10
9.2
8.1   /
/
PLYOPTS
2.E-6 /

```

The polymer properties can be echoed using keyword [RPTPROPS](#).

```

RPTPROPS
'PLYVISCS' /

```

The polymer adsorption properties are defined using [EQLDREAC](#) keyword model 4. This provides a reversible reaction between the solid component (component 7 in this example) and the POLYMER water component (component 11 in this example).

```

--Adsorption reactions
STOREAC
-- C1 C3 C6 C10 C15 C20 APOLY W NaCl CaCO3 POLY
 0 0 0 0 0 0      0 0 0 0 1           / Adsorbed <-> Dissolved Polymer
STOPROD
-- C1 C3 C6 C10 C15 C20 APOLY W NaCl CaCO3 POLY
 0 0 0 0 0 1      0 0 0 0 0           / Adsorbed <-> Dissolved Polymer
REACRATE
10 /
EQLDREAC
1 4 11 0 1 BOTH 1 /

EQLDTAB
--WMF_Poly Ca
 0.0      0.0
 5E-8     0.01
 1E-7     0.01 /

```

SOLUTION

```

SOLUTION =====
PRESSURE
9000*2000/

SWAT
9000*0.3/

SGAS
9000*0/

WMF
9000*0.995 9000*0.005 9000*0 9000*0/

XMF
9000*0.5 9000*0.03 9000*0.07 9000*0.2 9000*0.15 9000*0.05/

YMF
9000*0 9000*0 9000*0 9000*0 9000*0 9000*0/

RPTRST
'BASIC=2' POLY POLYVM ESALTP /

RPTSOL
POLY POLYVM ESALTP /

```

SUMMARY

```

SUMMARY =====
FWMIR
/
FWMIT
/
FWMPR
/
FWMPT
/
FAMF
/
WWMPR
PROD /
/
WWMIR
INJ/
/
BAMF
 1 1 5 /
 5 5 5 /
10 10 5 /
/
BMLST
 1 1 5 /
 5 5 5 /
10 10 5 /
/
BMLSC
 1 1 5 /
 5 5 5 /
10 10 5 /
/
BVWAT
 1 1 5 /
 5 5 5 /
10 10 5 /
/
BPOLY
 1 1 5 /
 5 5 5 /
10 10 5 /
/
BPOLYVM
 1 1 5 /
 5 5 5 /
10 10 5 /
/
BESALTP
1 1 5 /
5 5 5 /
10 10 5 /
/
BREAC
1 1 5 /
5 5 5 /
10 10 5 /
/
ALL
RUNSUM

```

SCHEDULE

```

SCHEDULE =====
RPTSCHED

```

```

MLSC MWAT POLY POLYVM ESALTP/
RPTRST
BASIC=3 FREQ=1 MLSC MWAT POLY POLYVM ESALTP/

WELLSPEC
INJ FIELD 1 1 3500 1* 0 /
PROD FIELD 30 30 3500 1* 0 /
/

COMPDAT
INJ 1* 1* 1 10 OPEN /
PROD 1* 1* 1 10 OPEN /
/

WCONINJE
INJ WATER OPEN RATE 10000 /
/

-- Definition of the injection streams
WELLSTRW
FRESH 1.0 0.0 0.0 0.0/
BRINE 0.99 0.005 0.005 0.0 /
POLYM 0.9899998 0.005 0.005 2E-7 /
/

-- Injection of Brine
WINJW
'I*' STREAM BRINE /
/

DATES
25 DEC 2013 /
/

-- Injection of Polymer Slug
WINJW
'I*' STREAM POLYM /
/

WCONPROD
PROD OPEN BHP 1* 1* 1* 1* 1* 200 /
/

DATES
1 FEB 2014 /
1 MAR 2014 /
1 APR 2014 /
1 MAY 2014 /
1 JUN 2014 /
/

WCONINJE
INJ WATER OPEN RATE 10000 /
/

-- Resume fresh water injection
WINJW
'I*' STREAM FRESH /
/

DATES
1 JUL 2014 /
1 AUG 2014 /
1 SEP 2014 /
1 OCT 2014 /
1 NOV 2014 /
1 DEC 2014 /
/

END

```

53

Potential calculations

Phase potentials

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

ECLIPSE 100

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, the SUMMARY file and 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,

$$\begin{aligned}\Phi_{\text{oil}} &= P_{\text{oil}} - \rho_o g (D - \text{DATUM}) \\ \Phi_{\text{wat}} &= P_{\text{wat}} - \rho_w g (D - \text{DATUM}) \\ \Phi_{\text{gas}} &= P_{\text{gas}} - \rho_g g (D - \text{DATUM})\end{aligned}\quad \text{Eq. 53.1}$$

where

- ρ_o = PVT region average oil density
- ρ_w = PVT region average water density
- ρ_g = PVT region average gas density
- D = depth of the cell
- DATUM = datum depth
- g = acceleration due to gravity

ECLIPSE 300

The phase potential of a cell is calculated by using the block density rather than the PVT region average density.

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 datum depth can be provided for each fluid-in-place region using the keywords [DATUMR](#) or [DATUMRX](#), in which case the value specified in any DATUM keyword is ignored.

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

$$\bar{\Phi}_p = \frac{\sum \Phi_p \cdot PV \cdot S_p}{\sum PV \cdot S_p} \quad \text{Eq. 53.2}$$

ECLIPSE 100

Setting OPTIONS keyword [switch 31](#) non-zero and divisible by two, weights the potential by pore volume rather than phase pore volume.

ECLIPSE 300

Setting OPTIONS3 keyword switch 86 to 1 weights the potential by hydrocarbon pore volume rather than by phase pore volume.

The PVT region average density is always weighted by phase saturation, so that

$$\bar{\rho}_p = \frac{\sum PV \cdot S_p \cdot \rho_p}{\sum PV \cdot S_p} \quad \text{Eq. 53.3}$$

where the sum is over a PVT region, and PV is the reference pore volume. When the phase saturation is zero within a PVT region (for example, oil phase saturation for a gas condensate above the dew point pressure), a pore volume weight is used instead to obtain the average density which is only used to report what the potential would be if the phase occupied the PVT region.

Initial contact corrected potential (ECLIPSE 100)

ECLIPSE 100 also calculates a representative potential, where the pressure is corrected to the datum depth using the water density below the initial WOC, the gas density above the initial GOC, and the oil density between the two initial contacts.

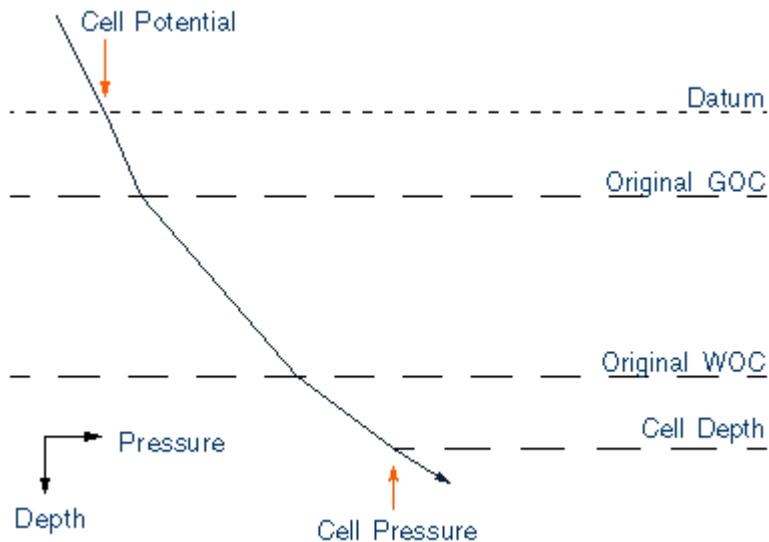


Figure 53.1. Calculating the initial contact corrected potential

This representative potential can be thought of as the potential of the phase occupying the region at initial equilibrium conditions.

Output of potentials

The potentials can be output using the following keywords.

ECLIPSE 100

To the PRINT file:

```
RPTSQL
'POTO' 'POTW' 'POTG' 'POTC' /
RPTSCHED
'POTO' 'POTW' 'POTG' 'POTC' /
```

ECLIPSE 300

To the PRINT file:

```
RPTSQL
'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 "Network Option") 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](#)).
- Wells can be closed automatically when their potential production or injection rates fall below an economic limit (see keywords [WECON](#) and [WECONINJ](#)).
- 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](#)).
- 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 keywords:

Field	Group	Well	Connection	Description
FOPP	GOPP	WOPP	COPP	Oil Potential Production rate
FOPI	GOPI	WOPI	COPI	Oil Potential Injection rate
FWPP	GWPP	WWPP	CWPP	Water Potential Production rate
FGPP	GWPI	WWPI	CWPI	Water Potential Injection rate
FWPI	GGPP	WGPP	CGPP	Gas Potential Production rate
FGPI	GGPI	WGPI	CGPI	Gas Potential Injection rate

In ECLIPSE 300 well potentials can be written to the SUMMARY file, using the keywords:

Well	Description
WOPP	Oil Potential Production rate
WWPP	Water Potential Production rate
WWIP	Water Potential Injection rate
WGPP	Gas Potential Production rate
WGIP	Gas Potential Injection rate

The field and group potentials are the sum of their subordinate wells' potentials. The connection potentials are the individual connection flow rates when the well is operating at its potential.

54

Recovery mechanisms

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:

$$\text{Total oil in place} = \text{PV} \cdot S_o \cdot \frac{1}{B_o} \quad \text{Eq. 54.1}$$

$$S_o = 1 - S_w - S_g$$

where

PV is the pore volume

S_o is the oil saturation

B_o is the oil formation volume factor

S_w is the water saturation

S_g is the gas saturation

The change in oil in place can be attributed as follows:

$$\begin{aligned} DC &= -d(\text{PV}) \cdot S_o \frac{1}{B_o} \\ DW &= \text{PV} \cdot d(S_w) \frac{1}{B_o} \\ DG &= \text{PV} \cdot d(S_g) \frac{1}{B_o} \\ DE &= \text{PV} \cdot S_o \cdot d\left(\frac{1}{B_o}\right) \end{aligned} \quad \text{Eq. 54.2}$$

where

- DC is the production due to rock compaction
- DW is the production due to water influx
- DG is the production due to gas influx
- DE is the production due to oil expansion
- $d(PV)$ is the change in pore volume over the timestep
- $d(S_w)$ is the change in water saturation over the timestep
- $d(S_g)$ is the change in gas saturation over the timestep
- $d \frac{1}{B_o}$ is the change in formation volume factor over the timestep.

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:

$$\begin{aligned} \text{DG} &= \text{DG-DE} \\ \text{DE} &= 0 \end{aligned} \quad \text{Eq. 54.3}$$

The oil production from each mechanism (DC, DW, DG, DE) is summed in each fluid-in-place region and also summed over timesteps. A table of production by each recovery mechanism can be output by using the RECOV mnemonic in the [RPTSCHE](#) 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 subdivided 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:

$$\begin{aligned} DS &= PV \cdot F_s \cdot d(S_g) \cdot \frac{1}{B_o} \\ DF &= PV \cdot (1-F_s) \cdot d(S_g) \cdot \frac{1}{B_o} \end{aligned} \quad \text{Eq. 54.4}$$

where

DS is the production due to solution gas

DF is the production due to free gas

F_s is the fraction of S_g that was originally solution gas.

Water drive

A similar procedure is available to distinguish production from two different water sources, for example injected water and aquifer water. Again, a tracer is used to track (say) the aquifer water, and the production is subdivided into production associated with ‘traced’ water and production associated with the remaining water.

To use this facility the following steps must be taken:

1. Increase the number of water tracers by one in the **TRACERS** keyword in the **RUNSPEC** section.
2. Add a tracer called ‘WTR’ in the **PROPS** section. The special name ‘WTR’ **must** be used to enable the recovery mechanism to recognize the tracer.

```
TRACER
'WTR' 'WAT' /
/
```

3. Initialize the tracer in the **SOLUTION** section in the usual way using the **TBLK** or **TVDP** keywords, and set the tracer concentration for any analytic aquifers using the **AQANTRC** keyword.

The initial tracer concentration should be either 1.0 for ‘traced’ water, or 0.0 for the rest.

Two new oil production values associated with the traced water and the remaining water will be calculated as follows:

$$\begin{aligned} \text{DWT} &= \text{PV} \cdot F_t \cdot d(S_w) \cdot \frac{1}{B_o} \\ \text{DWR} &= \text{PV} \cdot (1-F_t) \cdot d(S_w) \cdot \frac{1}{B_o} \end{aligned} \quad \text{Eq. 54.5}$$

where

- DWT is the production due to traced water
- DWR is the production due to remaining water
- F_t is the fraction of S_w that is ‘traced’ water.

Summary output

The oil production attributable to each mechanism, and the corresponding fraction of the total oil production, can also be output to the **SUMMARY** file.

Refer to [Oil recovery mechanism](#) in the *ECLIPSE Reference Manual*.

Details of the recovery calculation

ECLIPSE solves the oil conservation equation:

$$d(\text{OIP}) = \text{PV}^{t+dt} \cdot S_o^{t+dt} \cdot \left[\frac{1}{B_o} \right]^{t+dt} - \text{PV}^t \cdot S_o^t \cdot \left[\frac{1}{B_o} \right]^t \quad \text{Eq. 54.6}$$

Some care is needed to associate the production mechanisms such that the total oil produced equals the change in oil-in-place, $d(\text{OIP})$. Below we show how the production associated with each recovery mechanism is calculated.

Let:

$$\begin{aligned} \text{PV}^{t+dt} &= \text{PV}^t + d(\text{PV}) \\ S_o^{t+dt} &= S_o^t + d(S_o) \\ \left[\frac{1}{B_o} \right]^{t+dt} &= \left[\frac{1}{B_o} \right]^t + d\left[\frac{1}{B_o} \right] \\ d(\text{OIP}) &= [\text{PV} + d(\text{PV})][S_o + d(S_o)]\left[\frac{1}{B_o} + d\left(\frac{1}{B_o} \right) \right] - \left[\text{PV} \cdot S_o \cdot \frac{1}{B_o} \right] \end{aligned} \quad \text{Eq. 54.7}$$

Expanding the conservation equation,

$$\begin{aligned} d(\text{OIP}) &= d(\text{PV}) \cdot S_o^t \cdot \left[\frac{1}{B_o} \right]^t + d(\text{PV}) \cdot S_o^t \cdot d\left[\frac{1}{B_o} \right] \\ &\quad + \text{PV}^t \cdot d(S_o) \cdot \left[\frac{1}{B_o} \right]^t + d(\text{PV}) \cdot d(S_o) \cdot \left[\frac{1}{B_o} \right]^t + \text{PV}^t \cdot S_o^t \cdot d\left[\frac{1}{B_o} \right] \\ &\quad + \text{PV}^t \cdot d(S_o) \cdot d\left[\frac{1}{B_o} \right] + d(\text{PV}) \cdot d(S_o) \cdot d\left[\frac{1}{B_o} \right] \end{aligned} \quad \text{Eq. 54.8}$$

The terms in this equation that are associated with the various production mechanisms are:

$$\begin{aligned}\text{Rock Compaction} &= \text{DC} = -d(\text{PV}) \cdot S_o^t \cdot \left[\frac{1}{B_o} \right]^{t+dt} \\ \text{Saturation Change} &= \text{DS} = -\text{PV}^{t+dt} \cdot d(S_o) \cdot \left[\frac{1}{B_o} \right]^t \\ \text{Oil Expansion} &= \text{DE} = -\text{PV}^t \cdot S_o^{t+dt} \cdot d\left[\frac{1}{B_o}\right] - d(\text{PV}) \cdot d(S_o) \cdot d\left[\frac{1}{B_o}\right]\end{aligned}\quad \text{Eq. 54.9}$$

Now, $d(S_o) = -d(S_w) - d(S_g)$, giving the final breakdown of the production:

$$\begin{aligned}\text{Rock Compaction} &= \text{DC} = -d(\text{PV}) \cdot S_o^t \cdot \left[\frac{1}{B_o} \right]^{t+dt} \\ \text{Water Influx} &= \text{DW} = \text{PV}^{t+dt} \cdot d(S_w) \cdot \left[\frac{1}{B_o} \right]^t \\ \text{Gas Influx} &= \text{DG} = \text{PV}^{t+dt} \cdot d(S_g) \cdot \left[\frac{1}{B_o} \right]^t \\ \text{Oil Expansion} &= \text{DE} = -\text{PV}^t \cdot S_o^{t+dt} \cdot d\left[\frac{1}{B_o}\right] - d(\text{PV}) \cdot d(S_o) \cdot d\left[\frac{1}{B_o}\right]\end{aligned}\quad \text{Eq. 54.10}$$

When the pressure falls below the bubble point the oil expansion term will usually be negative, as the oil production will then be by solution gas drive. In this case the oil expansion term (DE) is added to the gas influx term (DG):

If gas saturation is greater than zero then:

$$\begin{aligned}\text{DG} &= \text{DG} + \text{DE} \\ \text{DE} &= 0.0\end{aligned}\quad \text{Eq. 54.11}$$

55

Reservoir coupling

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL

The reservoir coupling facility is a special extension to ECLIPSE 100; it is not available in ECLIPSE 300. It provides a means of coupling a number of separate ECLIPSE 100 simulation models to account for constraints on overall production and injection rates, and optionally the sharing of a common surface network.

The individual simulation models are run as separate processes, each having its own standard ECLIPSE data file. One model is chosen to be the “master”, and the other models are defined as “slaves”. The slave processes are activated by the master, and run in synchronization with it. The master process imposes production and injection constraints on the slaves to meet overall targets. On suitable systems, the slave processes may be run in parallel with each other on separate workstations. Communication between the processes is handled by the message-passing system MPI, which must be present on the system.

A typical situation in which the reservoir coupling facility would be useful is as follows. Imagine you are responsible for an area that contains a number of separate independent reservoirs. For each reservoir, you have a separate simulation model, which you have used to history match the reservoirs independently. The simulation models may have different characteristics. For example, some may be three-phase while others are two-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 three-phase if each phase was active in one or more of the reservoir models, and if one of them used the VE 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 and scheduling data for example, will be entered in the normal way in its data file. Alternatively, all the reservoirs to be simulated may be represented by slave reservoirs. In this case, the master reservoir should just be a dummy reservoir for example a single grid block), containing no wells but having the group hierarchy down to the master groups.

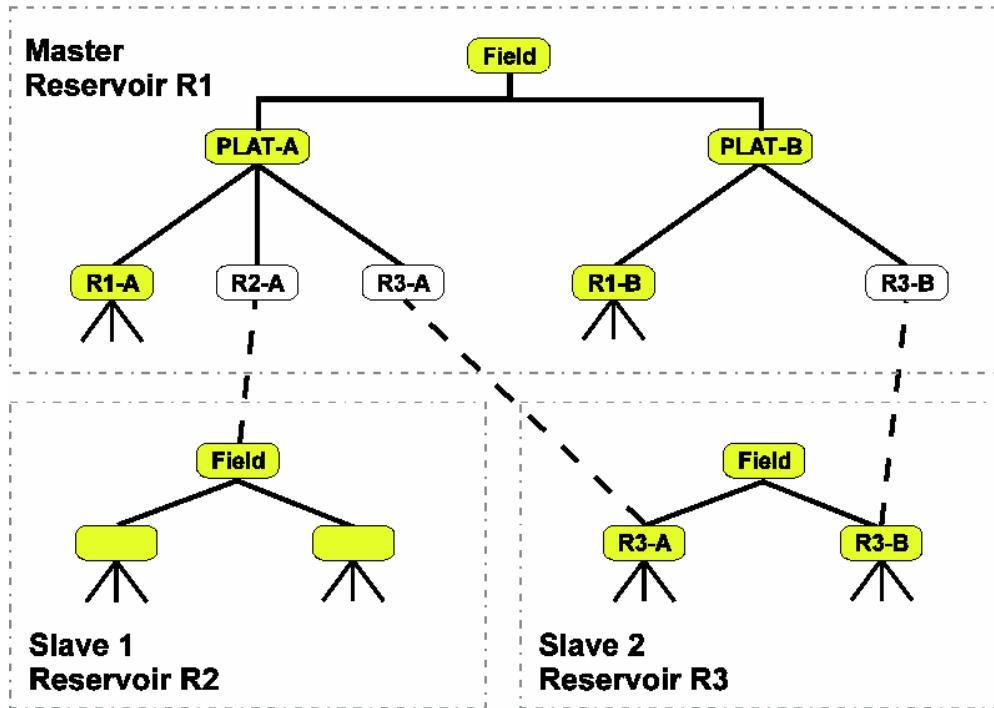


Figure 55.1. Three coupled reservoirs; one is the master

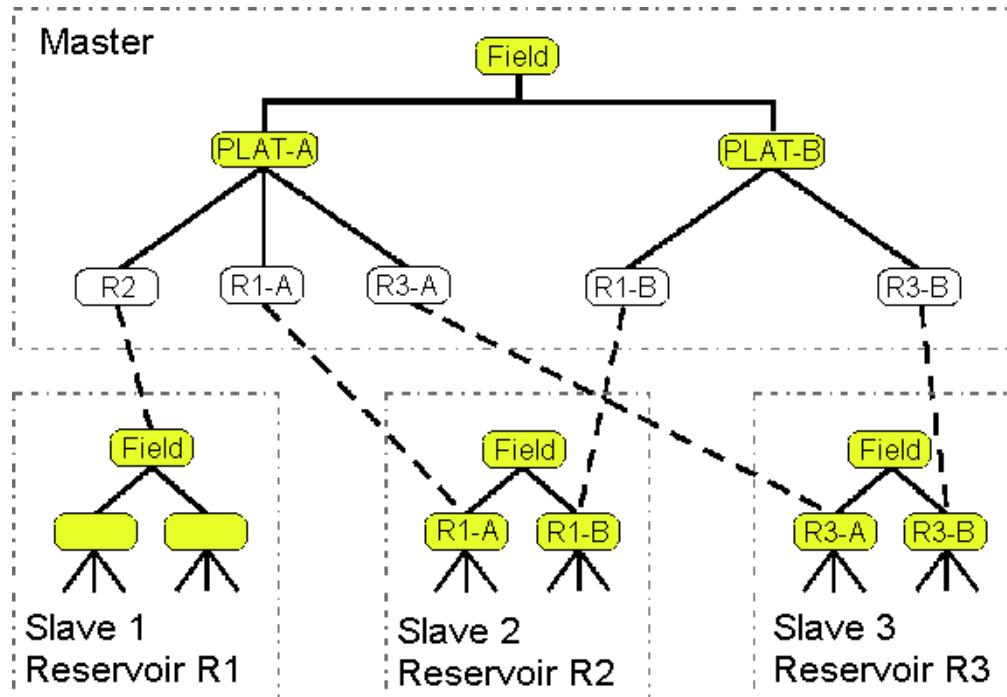


Figure 55.2. Three reservoirs coupled by a dummy reservoir master

Figures 55.1 and 55.2 illustrate these two cases applied to three coupled reservoirs. In figure 55.1, one of these reservoirs R1 is chosen as the master. Its own wells are under groups R1-A and R1-B. The other two reservoirs are represented as slaves. The master reservoir controls the slave reservoirs through the groups R2, R3-A and R3-B. These are declared as master groups in the master reservoir, and do not have any subordinate wells. The master group R2 is coupled to the FIELD group in the slave reservoir R2. The master groups R3-A and R3-B are coupled to the level 1 groups of the same name in the slave reservoir R3. In the slave reservoir R2, the FIELD group is declared as a slave group, coupled to group R2 in the master reservoir. In the slave reservoir R3, the groups R3-A and R3-B are declared as slave groups, coupled to the groups of the same name in the master reservoir. This example shows that a given slave reservoir may be coupled to the master by more than one group, which need not necessarily belong to the same parent group in the master reservoir.

Figure 55.2 shows the same set of reservoirs, but this time all three of them are represented as slave reservoirs. The master reservoir is a dummy reservoir with no wells. It contains the group hierarchy down to the master groups. Note that groups R1-A and R1-B are now master groups, being coupled to slave groups of the same name in reservoir R1 which is now a slave reservoir.

Coupling and synchronization

At the start of the master run, the master enquires the start date of each slave run. A slave may, if necessary, be set to start later than the master. In this case the slave will be “dormant”, not contributing any production or injection, until its start date is reached in the master run.

The events that couple the master and the slave runs are synchronized with the timesteps of the master run. Note that these are the **actual** timesteps, 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 timestep size in the master reservoir (keyword **TUNING**).

For each timestep in the master run:

1. The master must first decide the length of the timestep. This is done in the usual way, but subject to two additional constraints:
 - a. The timestep 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. Timesteps limited in this way are identified by the timestep control mnemonic **SLVR** in the timestep summary written to the Print file.
 - b. The timestep 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 timestep. (See keywords **GRUPMAST** and **RCMASTS**.) Timesteps limited in this way are identified by the timestep control mnemonic **SLVC** in the timestep summary written to the Print file.
2. At the beginning of the timestep, 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 timestep. 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 timestep, and tells the master.
4. The master completes its own timestep, assuming that the production and injection rates of the slave groups remain constant over the timestep.
5. When the master has completed its timestep, all the slave reservoirs are advanced to the end of the master's timestep, using the rate targets allocated to the slave groups as constraints. The slaves may take one or more of their own timesteps to reach this time. The timestep that is limited to reach this time is identified by the timestep control mnemonic **MAST** in the timestep 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 timestep.
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 timestep. 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 timestep.

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 timestep, but the flows are reported at the **end** of the timestep. If the flow rate of a slave group changes over the length of the master's timestep, 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 timesteps. There is an option to limit the master run's timesteps 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 "[Network option](#)"). 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](#)"). 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 timestep (which is the default in keyword [NETBALAN](#)). Thus all networks will be balanced simultaneously at each timestep of the master process. At each iteration of the network balancing calculation in the master:

1. The master sends the nodal pressures of the master groups to the respective slave groups in the slave reservoirs.
2. The slave reservoirs balance their own networks, using the nodal pressures passed from the master as the “fixed” pressures of the slave groups.
3. The slave processes send back to the master the flows of the slave groups at the current conditions.
4. The master sums the flows up to its FIELD level, and updates the pressures in the nodes of its network in the usual way.

Note that at each balancing iteration in the master run, the slaves balance their own networks to convergence.

If the slave reservoirs take more than one timestep to advance to the end of the master timestep, 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 timestep.

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 timestep of the master run, the following information is written to this file for each master group:

1. The group’s coupling constraints. These include the rate target applied to the group by the master run. Other rate limits are included, to prevent the group increasing its flow sufficiently to violate any rate limits imposed on higher level groups, assuming the flows of all other groups remain constant. If the group is part of a pipeline network, its network nodal pressure is also included.
2. The group’s potential flow rates. These are the flow rates that the coupled slave group’s subordinate wells would achieve, at the current grid block conditions, if all rate limits were ignored and only their THP and BHP limits were observed. The potentials may be used to determine guide rates for group control.
3. The group’s production and injection rates. These are the rates that the coupled slave groups have achieved, averaged over the length of the master run’s current timestep. 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:

- To substitute for one or more of the slave reservoirs in the master run. The flow rates of the slave groups will be read from the file and incorporated into the master groups' flows.

This would allow the master to be run with a reduced number of coupled slaves. In fact, if all the slaves were replaced by the data on the Reservoir Coupling file, the master could be run on its own without requiring MPI to activate the slave runs.

Obviously, the slave group flows read from the file cannot be influenced by any changes made to the coupled system. So be careful about making changes that would have significant repercussions on the slave reservoirs. Use of the reservoir coupling file for this purpose has similarities with the flux boundary conditions option.

- To drive a slave reservoir simulation on its own, supplying the rate (and perhaps network pressure) constraints that were originally dictated by the master. MPI is not required for this task.

At a time corresponding to the start of each timestep 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 timestep. For additional flexibility, the user can choose to ignore limits for particular produced or injected phases. For each produced and injected phase, there is an option to take the rate constraints as specified in the slave's data file, or to take the rate constraints from the Reservoir Coupling file, or to take the minimum of the two constraints. (See keyword [GRUPSLAV](#).)

A set of SUMMARY section keywords is provided to output the rate limits the master groups are subject to, as a function of time. This offers a useful way of displaying the rate limits imposed by the reservoir coupling file.

When a Reservoir Coupling file is used for either of the two purposes described above, the timesteps of the simulation are limited so as not to overshoot the timesteps of the original master run that produced the file. At these times the flow rates and the constraints on the file may change. Timesteps limited in this way are identified by the timestep control mnemonic MAST in the timestep 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

- 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.
- 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 timestep.
- 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](#)"). 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](#)").

- The master timestep 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 timestep.
- 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.
- 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.
- Automatic compressors in an extended network (see keyword [NETCOMPA](#)) cannot respond to falling production from a group in a different reservoir. For example, compressors in a slave reservoir cannot be activated by a group in the master reservoir failing to meet its GCONPROD target.

Using the reservoir coupling facility

The reservoir coupling facility uses the MPI message passing system to send messages and data between the master and the slave runs.

Reservoir coupling using MPI

To run a coupled reservoir simulation MPI must first be installed on your system. Refer to the "ECLIPSE Suite Installation Guide" for instructions on how to install MPI.

By using MPI as the message passing system slave reservoirs may be run either as serial processes or in parallel. To run a slave reservoir in parallel the number of processors on which it is to be run must be specified in the master data set using the [SLAVES](#) keyword. The [PARALLEL](#) keyword should also appear in the slave data set with the number of processors on which the slave is to be run as the first argument.

It is not possible to run a master reservoir in parallel.

Starting a reservoir coupling simulation using MPI is achieved using the parallel ECLIPSE macro, `@mpieclipse`. The total number of processors to be used by the master reservoir and combined slave reservoirs should be provided as the argument to `@mpieclipse`.

for example, `@mpieclipse -procs X`

The master reservoir

The master reservoir is represented by a standard ECLIPSE data file. It may, if required, be a restart from a previous run. It must have as active phases all the phases that are active in any of the slave runs. It must contain the group hierarchy down to the master groups that couple it to the slaves, and of course the hierarchy must include its own well groups if it has any. The group hierarchy is defined in the usual way with keyword [GRUPTREE](#).

The master does not necessarily have to represent a real reservoir. It may just be a “dummy reservoir” whose purpose is to couple the slave reservoirs through its production and injection constraints (and

optionally a pipeline network). In this case, the data file should still contain enough information to run ECLIPSE. At least one grid block is necessary. Fluid properties should be provided as required for its active phases. No wells should be defined, but the group hierarchy must be defined down to the master groups.

The [SLAVES](#) keyword must be used in the [SCHEDULE](#) section to activate the coupled slave runs. The keyword data specifies the machine on which each slave process is to be run, and the directory where their data files are located. For example, for figure 55.1:

```
SLAVES
-- slave    datafile      machine      directory
-- name     root        hostname      of data file
'RES-R2'  'base'      'rios'        '/usr/models/res2' /
'RES-R3'  'base'      'sg-indigo'   '/usr/models/res3' /
/
```

If you wish the master run to create a Reservoir Coupling file containing the rates and constraints of all the master groups, enter the keyword [DUMPCUPL](#):

```
DUMPCUPL
'U' /
```

After the group hierarchy has been set up with keyword [GRUPTREE](#), the master groups must be identified with keyword [GRUPMAST](#). The keyword also specifies the slaves to which the master groups are coupled, and sets a limiting fraction for flow rate change in a single timestep. For example, for figure 55.1:

```
GRUPMAST
-- master      slave      associated      limiting
-- group      reservoir    slave        rate change
-- name       name        group        fraction
'R2'        'RES-R2'   'FIELD'      0.1  /
'R3-A'      'RES-R3'   1*          0.1  /
'R3-B'      'RES-R3'   1*          0.1  /
/
```

If limiting rate change fractions have been set for the master groups, it is advisable to set a minimum length to which the timestep 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 timestep.

```
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 timestep (which is the default).

If you wish the master run to stop when a slave run finishes for any reason, set its master group’s end-run flag in item 8 of keyword [GECON](#). Do **not** set any of the other GECON limits, as the master groups effectively contain no wells, and the master run cannot control workovers in the slaves. Economic limits can, of course, be set in the slave data files.

Set the report times or dates in the usual way. For easier synchronization it is best to use [DATES](#), especially if the slave runs start on different dates. If this is the case, it is also advisable to use the [DATE](#) keyword in the [SUMMARY](#) section. Aim to set common report dates in each coupled simulation. This is especially important for restarts, as the coupled simulations can only be restarted correctly from [RESTART](#) files written on the same date.

The slave reservoirs

The slave reservoirs are represented by standard ECLIPSE data files. These may be restarts from previous runs. You should ensure that they are set up to start at the required date. If they are restart runs, ensure also that the correct file name and report number are entered in the [RESTART](#) keyword.

A slave run may be set to start at a date later than the start of the master run. In this case the slave will be “dormant”, having no production or injection, until its start date is reached in the master run. Note that the simulation time will not be the same as in the master run, so it is best to use the [DATES](#) keyword to specify report times, and also to include the [DATE](#) keyword in the [SUMMARY](#) section to present the results by calendar date rather than simulation time. An error message will be issued if the slave run is set to start earlier than the master run.

The only additional keyword required in a slave run’s data file is [GRUPSLAV](#). This identifies the slave groups, and the associated master groups to which they are coupled. You can also set instructions to ignore any rate limits set in the slave run’s data file, or take the minimum of these limits and the limits imposed by the master run. For example, for slave reservoir R2 in Figure 55.1, using the default option of observing only the rate limits communicated from the master for all phases:

```
GRUPSLAV
-- slave assoc oil wat/liq gas resv oil wat gas
-- group master prod prod prod prod inj inj inj
-- name group limits limits limits limits limits limits 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 overwrite 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 timestep (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 timesteps 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 timestep size throughout the set of coupled reservoirs. So it is best to have a common set of report dates. You will certainly need to synchronize the report dates on which RESTART files are to be written, if you wish subsequently to restart the coupled simulation.

Reading a reservoir coupling file in the master reservoir

A reservoir coupling file can be used to take the place of one or more of the slave reservoirs. Instead of activating their slave runs, ECLIPSE reads their flow rates from this file. To use this option, make the following changes to the master reservoir's data file:

- Add the keyword [USECUPL](#), to specify the name and format of the Reservoir Coupling file to be read. If the run spans a sequence of Reservoir Coupling files generated by a series of restarts of the master run, enter the name of the last file in the sequence.
- Remove the particular slave reservoir's data record from the [SLAVES](#) keyword. If there are no more slave reservoirs to be actively coupled to the master, the [SLAVES](#) keyword should be removed.
- In keyword [GRUPMAST](#), the master groups associated with the substituted slave reservoir must have their slave reservoir name in item 2 defaulted.

For example, to substitute slave reservoir R3 in figure 55.1 with a Reservoir Coupling file written by the run 'master.data':

```
USECUPL
'master'  'U'  /
SLAVES
--   slave    datafile      machine      directory
--   name     root        hostname     of data file
--
--   'RES-R2'  'base'       'rios'        '/usr/models/res2'  /
/
GRUPMAST
--   master      slave      associated      limiting
--   group      reservoir    slave        rate change
--   name       name       group        fraction
--   'R2'        'RES-R2'   'FIELD'      0.1  /
--   'R3-A'      1*          1*          /
--   'R3-B'      1*          1*          /
/
```

Driving a slave reservoir with a reservoir coupling file

A slave run may be performed on its own, instead of being activated by the master run. It will read the Reservoir Coupling file to obtain the constraints that the master run originally applied to its slave groups over the course of the simulation. MPI is not required for this option.

To use this option, set up the slave's data file as though it were going to be activated by the master, but in addition include the keyword [USECUPL](#) as described above. The slave groups are identified with the keyword [GRUPSLAV](#), and you can choose whether to take their rate constraints from the Reservoir Coupling file, the slave's own data file, or the minimum of both.

You must set up the slave groups for production control (with keyword [GCONPROD](#) for guide rate control or keyword [GCONPRI](#) for prioritization), and for injection control of the injected phases (with keyword [GCONINJE](#)). Reinjection and voidage replacement targets applied in the master reservoir to groups superior to the master groups are translated into surface rate and reservoir volume rate targets respectively at the slave groups.

If the master and slave reservoirs are coupled by a pipeline network, the nodal pressures of the slave groups that were set by the master in the original coupled run will also be read from the Reservoir Coupling file. The slave groups must of course be defined as fixed-pressure nodes in keyword [GRUPNET](#).

A useful way of monitoring the production and injection targets 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. Note that the value of these SUMMARY quantities will differ between master and slave groups. The value reported in the slave group is the overall target applied owing to the combination of all limits on the group. The value reported by the same SUMMARY quantities for the master group will be the rate limit set in the GCONPROD, GCONPRI or GCONINJE keywords.

As an example, to drive slave reservoir R3 in figure 55.1 from the Reservoir Coupling file written by the run ‘master.data’, but imposing an additional limit on the gas production:

```

USECUPL
'master'  'U' /
GRUPSLAV
-- slave    assoc   oil      wat/liq  gas      resv    oil      wat      gas
-- group    master   prod     prod     prod     inj     inj     inj
-- name     group   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](#)”) 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.

- 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.
- 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 timesteps will be limited to one month, regardless of the setting of GASYEAR item 4 or GASPERIO item 5.
- 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.
- All automatic compressors (see keyword [GASFCOMP](#)) must be situated in the master reservoir. Any in the slave reservoirs will be ignored.

- 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.
- Reservoir Coupling files should only be used with care. They store the master groups' flows during the second pass of the contract year, when producing at DCQ x profile factor. But the flows during the first pass (when producing at DCQ x swing factor) are **not** recorded. Thus if you replace a slave reservoir by a reservoir coupling file, its production rate in the first pass will be the same as in the second pass. This will only be correct when the swing factors are the same as the profile factors. A Reservoir Coupling file cannot be used to drive a slave on its own when it is using the gas field operations model.

Sharing licenses with the slaves

There was a major change in the licensing model used with Reservoir Coupling in 2005A. Prior to 2004A a reservoir coupling job would check out only one `eclipse` license and one feature license no matter how many `SLAVES` there were.

From 2005A this policy was modified so that each `SLAVE` requires an ECLIPSE license as well as the master. Each `SLAVE` also requires one feature license for every feature it uses.

The following example indicates how works:

Master

`SLAVE 1` uses `LGR` and `Multisegwells`

`SLAVE 2` uses `LGR` and `networks`

`SLAVE 3` uses `networks`

Prior to 2004A the following licenses would have been required to run this model:

- 1 `eclipse`
- 1 `lgr`
- 1 `network`
- 1 `multisegwells`
- `rescoupling`

Since 2005A this is:

- 4 `eclipse`
- 2 `lgr`
- 2 `network`
- 1 `multisegwells`
- 1 `rescoupling`

Reservoir coupling file structure

The reservoir coupling file structure is similar to the structure of other files that ECLIPSE reads and writes. This is described in "[File handling in ECLIPSE](#)". 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 timestep of the master run, the following records are written:

```
'TIMESTEP' 2 'DOUB'
```

followed by the start and end time of the timestep, 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 and GGPRS for example)
- 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 timestep.

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 **CTRLMODE** 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'
 0.000000000000D+00 0.10000000000000D+01
'GRUPNAME'         2 'CHAR'
'B3-A'            'B3-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

'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

'POTNS' 11 'DOUB'

```

```

0.00000000000000D+00 0.00000000000000D+00 0.00000000000000D+00
0.00000000000000D+00 0.00000000000000D+00 0.00000000000000D+00
0.00000000000000D+00 0.00000000000000D+00 0.00000000000000D+00
0.00000000000000D+00 0.00000000000000D+00

'ENDSTEP'          0 'MESS'
.....
```

Summary of keywords

SUMMARY section

Refer to [Reservoir coupling facility](#) in the *ECLIPSE Reference Manual*.

SCHEDULE section

Keyword	Description
DEBUG	Switch 39 turns on debug from the systems annex for MPI calls. Switch 42 turns on debug from the reservoir coupling facility.
DUMPCUPL	Instructs the master run to write a reservoir coupling file, containing the flows and applied constraints of the master groups.
GRUPMAST	Must be used in the master run to identify the master groups and to indicate which slaves they are associated with.
GRUPSLAV	Must be used in each slave run to identify the slave groups and their associated master groups in the master run, and to set the filter flags that determining which rate constraints are to be imposed on them
RCMASTS	Sets a minimum length for timesteps 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

56

Restarts

There are two methods of restarting an ECLIPSE run:

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

- A fast restart, which 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 recalculated. The SAVE keyword must be used in RUNSPEC section of the original run to tell ECLIPSE 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.
- A flexible restart, which reads a complete data file. This is the slower method, as the data must be processed again (for example the transmissibilities are recalculated). However, it does allow 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 are possible.

Both methods require the time-dependent data (for example grid block solution variables and the status of wells) to be stored at one or more times in the original simulation run. By default, the time-dependent data is stored in the RESTART file. 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. See "[Requesting RESTART files](#)".

Fast restarts from a SAVE file

ECLIPSE 100

Restarting a run from a SAVE is usually quicker to initialize than a restart from a complete data file. This method can be used for simple restarts where only the SCHEDULE section data is to be modified.

The procedure is as follows:

1. In the base run, use the `SAVE` keyword to instruct ECLIPSE to create a SAVE file, and use the `RPTSCHED` or `RPTRST` keywords to control the frequency with which the solution data is to be written to the RESTART file.
2. Set up the data file for the restart run, consisting of a `LOAD` keyword, a `RESTART` keyword, the `SUMMARY` and `SCHEDULE` section data.

For example,

```
LOAD
  BASE /
  RESTART
  BASE 11 /
  SUMMARY
  INCLUDE
```

```
SUMMARY.DAT /
SCHEDULE
...
...
```

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 - fast restarts cannot be performed from SAVE files produced by earlier versions.

ECLIPSE 100

Note: In ECLIPSE 100, for a parallel fast restart run the [PARALLEL](#) keyword should not be present in the restart data file since the record is read from the SAVE file instead. However, a parallel fast restart run must be executed on the same number of processors as the base run.

ECLIPSE 100

Note: Fast restarts are not supported in with the [PARALLEL](#) option in ECLIPSE 100 runs containing local grid refinements

ECLIPSE 300

Note: The use of fast restarts is not compatible with the [PARALLEL](#) option in ECLIPSE 300.

- a. In the SCHEDULE section, unless keyword [SKIPREST](#) has been entered, delete the well and timestepping 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 the restart time as required.
3. Restarting from time-dependent data stored in the SAVE file:
 - a. 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 /
```

- b. 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. It 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
 - a. 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 /
```
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 and transmissibilities for example.

It is possible also to do a flexible restart from time-dependent restart information stored in 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 requests that the time dependent data for the restart is taken from the unformatted SAVE file generated by the base run.

Requesting RESTART files

By default, the restart data written at each report time will be stored in separate RESTART files. The files' suffix 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 to read a Unified RESTART file, the keyword [UNIFIN](#) must be present in the RUNSPEC section - otherwise ECLIPSE will look 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 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](#)" for more details on file formats.

ECLIPSE 100

To create RESTART files in ECLIPSE 100, from which a later run can restart, the procedure is:

1. Set either the mnemonic [BASIC](#) in the [RPTRST](#) keyword > 0, or set the mnemonic [RESTART](#) in the [RPTSCHE](#) keyword > 0.

- a. If either mnemonic 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).
 - b. If the mnemonic = 2 (or = 1 also for unified files), Restart files are created at every report time until the switch is reset. None are deleted.
 - c. Giving the mnemonic BASIC in the RPTRST keyword a value > 2 results in the automatic generation of Restart files at specific intervals of time.
2. A restart file can also be generated at time 0.0 days, containing the initial solution, either by setting the mnemonic RESTAR in the RPTSOL keyword to 2, or by setting the mnemonic BASIC to 2 in keyword RPTRST in the SOLUTION section. There must be at least one TIME, TSTEP or DATES keyword present to generate the time zero restart file.
 3. 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.
 4. If you want the time-dependent restart data to be appended to the SAVE file, use the SAVE mnemonic 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.

ECLIPSE 300

In ECLIPSE 300, use keyword RPTRST to control the output and contents of the RESTART file.

SCHEDULE section data not stored on the 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, OPTIONS3)
- Annual scheduling files in the ECLIPSE gas field operations model extension for ECLIPSE 100 (keywords GASBEGIN, GASEND and any keywords in between)
- The keywords SLAVES, DUMPCUPL and USECUPL in the ECLIPSE Reservoir Coupling Facility extension for ECLIPSE 100.
- Well PI scaling tables (keyword PIMULTAB).
- Scale deposition and damage tables (keywords SCDPTAB and SCDACTAB).
- The inclination factor table for the drift flux model in multisegment wells (keyword WSEGDFIN).
- Gas plant tables (keywords GPTABLE, GPTABLEN and GPTABLE3)
- The TUNING keyword.

ECLIPSE 300

- Non-neighbor connection transmissibility. This is stored in the INIT file, but changes made to it in the schedule section, for instance using the MULTX keyword, are not carried over for restarts. Any restart for a run which changes non-neighbor transmissibilities in the schedule section must be made before the transmissibilities are changed.

Note: These keywords must be present in the SCHEDULE section of the restart run, if they are needed in the run.

Notes

The following data cannot be changed at a restart:

- The grid dimensions NDIVIX, NDIVIY, NDIVIZ, and the number of active cells.
- The radial/Cartesian switch.
- The phases present and the permitted mass transfers
- (OIL, WATER, GAS, dissolved GAS, vaporized OIL).
- The number of components present.
- The unit convention.
- The vertical equilibrium options, keyword VE in RUNSPEC.
- The type of well model, for existing wells.
- 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.
- 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.
- A warning message is issued if the RESTART file was not generated by the current version of ECLIPSE. In general, using flexible restarts, ECLIPSE 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 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.

Rivers

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

This chapter describes how rivers are modeled in the ECLIPSE 100 simulator. Within ECLIPSE it is possible to model the complex interaction between surface river networks and subsurface aquifers for both saturated and unsaturated flows. This is achieved by computing a solution to the one-dimensional Saint-Venant equations, which model open channel flow, at each aquifer timestep. The Saint-Venant equations are coupled fully implicitly with the groundwater equations and in this manner ECLIPSE is able to provide an accurate solution.

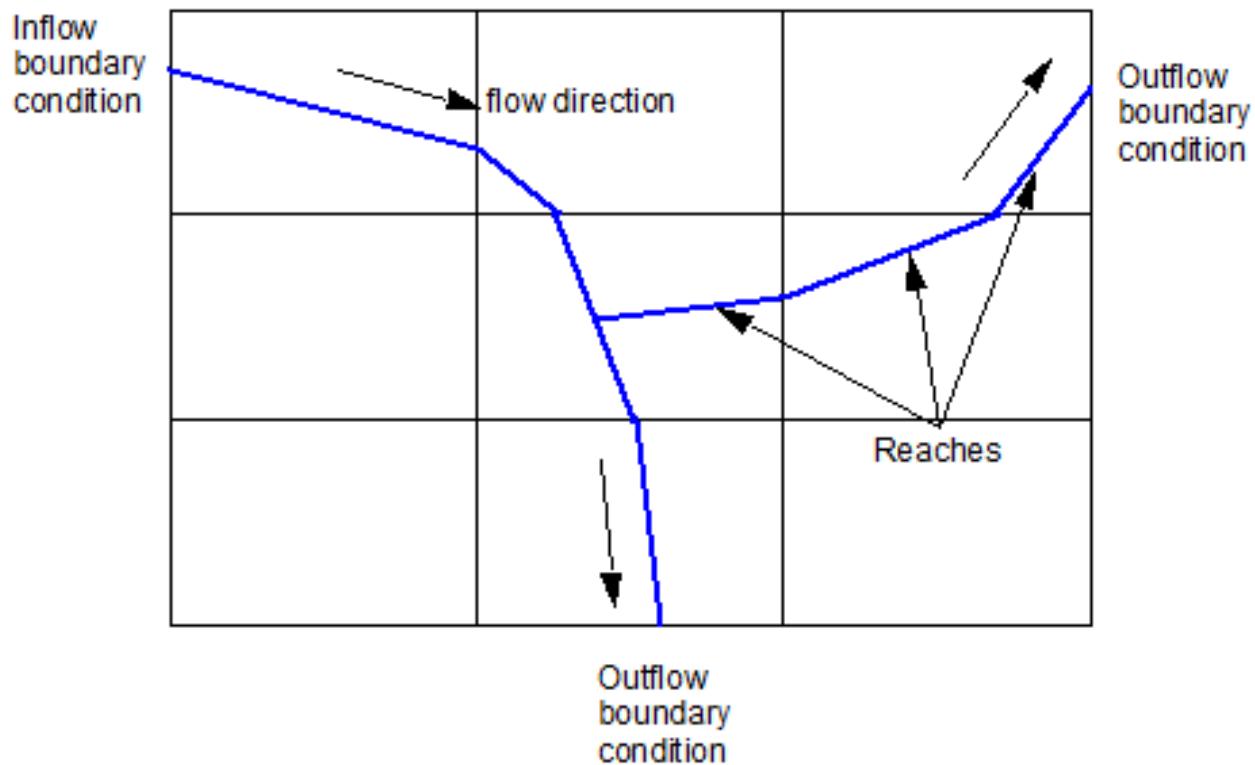


Figure 57.1. Figure showing a simple river network illustrating the river reach structure

The river model

The Saint-Venant equations

The ECLIPSE river model is a tool to handle coupling of surface river networks and subsurface groundwater flows. This is achieved by computing a solution to the one-dimensional Saint-Venant equations at each simulation timestep. It is possible to simulate either transient or steady-state behavior in the river. The steady-state option partially addresses the difference in time scale between the fast river channel dynamics and the slow groundwater dynamics, and focuses on the exchange of flow between the river and groundwater.

The following form of the Saint-Venant equations, with the flux Q and cross-sectional area A as independent variables, are solved

$$\frac{\partial A}{\partial t} + \frac{\partial Q}{\partial x} = E \quad \text{Eq. 57.1}$$

$$\frac{\partial Q}{\partial t} + \frac{\partial}{\partial x} \left(\frac{Q^2}{A} + gI_1 \right) = gI_2 + gA(S_0 - S_f) \quad \text{Eq. 57.2}$$

where

Q is the flux across the cross sectional area A

E is the river exchange term

x is the distance along the river

t is time

A is the cross sectional area

g is the acceleration due to gravity

I_1 is the hydrostatic pressure force term given by $I_1 = \int_0^{h(x,t)} (h-\eta) b(x, \eta) d\eta$ where

$b(x, \eta) = \frac{\partial A(x, \eta)}{\partial \eta}$ and $h(x, t)$ is the water depth.

I_2 is the pressure force term due to longitudinal width variations given by $I_2 = \int_0^{h(x,t)} (h-\eta) \frac{\partial b(x, \eta)}{\partial x} d\eta$

S_0 is the natural river slope $S_0 = \frac{\partial z}{\partial x}$

S_f is the frictional slope $S_f = \frac{Q | Q | n^2}{A^2 \Pi^2 R^{4/3}}$, where n is the Manning friction coefficient, R is the hydraulic radius and Π is a constant depending on the unit system being used.

There are three simplified forms of the Saint-Venant equations which may be used instead of the full time-dependent Saint-Venant equations for the river model. The first simplified form is the steady-state Saint-Venant equations:

$$\frac{\partial Q}{\partial x} = E \quad \text{Eq. 57.3}$$

the second simplified form which can be used is the diffusive wave form:

$$\frac{\partial Q}{\partial x} = E \quad \text{Eq. 57.4}$$

$$\frac{\partial I_1}{\partial x} = I_2 + A[S_0 - S_f] \quad \text{Eq. 57.5}$$

and the third simplified form is the kinematic wave form:

$$\frac{\partial Q}{\partial x} = E \quad \text{Eq. 57.6}$$

$$S_0 - S_f = 0 \quad \text{Eq. 57.7}$$

These three sets of equations are derived from the full Saint-Venant equations under certain simplifying assumptions. The kinematic wave equations are derived by making the assumption that the hydrostatic pressure force term and the net momentum flux are constant along the length of the river. The diffusive wave equations are derived from the Saint-Venant equations by making the assumption that the net momentum flux of the river is constant along the length of the river. The steady-state Saint-Venant equations are derived assuming the solution is independent of time.

ECLIPSE river solves the river equations in a number of stages. The first is the pre-solve stage, where a simplified form of the equation is solved individually for each river reach by Newton iteration. This provides an initial estimate of the river solution for each river reach. During a second solution stage, ECLIPSE solves the entire river system (that is all rivers in a given model) by computing a converged implicit solution coupled with the groundwater system. The user can request to solve each river by specifying one of the four solution methods, which are based on solving either the kinematic, diffusive, steady-state or the full Saint-Venant system of equations.

Coupling of river and groundwater flow

The river model computes the flow of water between a river and the underlying groundwater using an infiltration equation. In ECLIPSE there is a choice of two infiltration equations which may be used.

Model one

The first model, which is the default, computes the flow E between the river and the groundwater in the underlying aquifer using the equation

$$E = J\alpha[P_{\text{aquifer}} - P_{\text{river}} - H] \quad \text{Eq. 57.8}$$

where

J is the reach productivity index (PI)

α is the contact area between the river channel and the river grid connection

P_{aquifer} is the pressure in the aquifer

P_{river} is the pressure in the river

H is the river pressure head term between the aquifer and river elevation.

The pressure in the river is calculated using the expression

$$P_{\text{river}} = P_{\text{ref}} + \rho_{\text{sw}} gh \quad \text{Eq. 57.9}$$

where

ρ_{sw} is the water surface density in the river and

P_{ref} is a given reference pressure.

The pressure in the aquifer takes into account capillary pressure P_{cow} and is given by

$$P_{\text{aquifer}} = P - P_{\text{cow}}.$$

The pressure head term H is calculated using the expression

$$H = \rho g (Z_{\text{aquifer}} - Z_{\text{river}}) \quad \text{Eq. 57.10}$$

where

ρ is the density of the fluid in the aquifer

g is the acceleration due to gravity

Z_{aquifer} is the elevation of the aquifer

Z_{river} is the elevation of the river.

When the aquifer is not fully saturated with water (the aquifer is unconfined) the fluid density ρ in the aquifer is calculated using the expression

$$\rho = \frac{\sum_i k_{ri} \rho_i}{\sum_i k_{ri}} \quad \text{Eq. 57.11}$$

$i = o, g, w$

where

k_{ri} is the relative permeability and

ρ_i is the reservoir density of phase i , respectively.

When the aquifer is confined, the fluid density is simply taken to be the water phase density. It should be noted that the infiltration equation for the unconfined case reduces to that of the confined case when the aquifer is fully saturated.

The contact area α between the river channel and the aquifer may be a function of the water depth in the river h , however when the contact area α is constant the reach productivity index (PI) J is related to the streambed conductance C as follows

$$J = \frac{C}{\rho g \alpha}.$$

The use of a streambed conductance implies the presence of a leaky streambed layer with a given thickness.

Model 2

The second model of infiltration uses the equation

$$E = C (H_{\text{aquifer}} - H_{\text{river}}) \quad \text{Eq. 57.12}$$

where

C is the river bed conductance

H_{aquifer} is the hydraulic head in the aquifer

H_{river} is the hydraulic head in the river.

The term riverbed conductance is defined to be

$$C = \frac{KwL}{m} \quad \text{Eq. 57.13}$$

where

K is the hydraulic conductivity of the river

w width of the river reach

L is the length of the river

m is the thickness of the riverbed deposits.

In this model the hydraulic head in the river is simply defined to be

$$H_{\text{river}} = Z_{\text{ref}} - Z_{\text{river}} + m + h \quad \text{Eq. 57.14}$$

where

Z_{ref} is the reference elevation

Z_{river} is the elevation of the river bottom

h is the water height in the river.

The hydraulic head in the aquifer is calculated using the expression

$$H_{\text{aquifer}} = Z_{\text{ref}} - Z_{\text{aquifer}} + \frac{(P_{\text{aquifer}} - P_{\text{ref}})}{\rho_w g} \quad \text{Eq. 57.15}$$

where

Z_{aquifer} is the elevation of the aquifer

P_{aquifer} is the pressure in the aquifer

P_{ref} is a given reference pressure

ρ_w is the water density in the aquifer

g is the acceleration due to gravity.

The flow between the river and aquifer is no longer dependent on the hydraulic head in the aquifer when the hydraulic head in the aquifer is below the river bed bottom elevation. In this case the hydraulic head in the aquifer is taken to be the river bed elevation.

Using the river option

The River option in ECLIPSE is enabled by specifying the keyword [RIVERSYS](#) together with other related keywords. The RIVERSYS keyword defines the name of the river, the names of the river branches, the solution method to be used and the upstream/downstream boundary conditions.

The structure of each river is specified by data input using the [COMPRIV](#), [REACHES](#) and [GRDREACH](#) keywords in the SCHEDULE section. The keyword COMPRIV is used to input a list of grid blocks that are connected to the river. Note that rivers cannot be connected to cells in local grid refinements. (It is possible to connect a river reach to an inactive grid block, but in this case the exchange flow between the river and the aquifer is not computed.) The REACHES keyword is used to define the reach structure of the river and also the required parameters for each reach such as Manning's roughness coefficient. The GRDREACH keyword is used to define the location of grid blocks that connect to a given river reach. Other information about how the river connects to the grid, such as the contact area and the productivity index (PI), is also specified with this keyword.

The river cross-section and the upstream/downstream boundary conditions to be applied are defined using [RIVRXSEC](#), [MASSFLOW](#) and [DEPTHHTAB](#) keywords in the PROPS section.

The keyword [RIVRDIMS](#), which is specified in the RUNSPEC section, is used to describe the dimensions of the river data to be used in the run.

Summary of keywords

RUNSPEC section keywords

- [RIVRDIMS](#) River dimension data and activates river option

PROPS section keywords

Keyword	Description
DEPTHHTAB	Tables of river depth data for downstream boundary conditions
MASSFLOW	Tables of river massflow data for upstream boundary conditions
QHRATING	Tables containing relationships between massflow and depth for downstream boundary conditions
RBEDCONT	Tables of riverbed/grid-block contact area data
RIVRXSEC	Tables of river channel profile data

SOLUTION section keywords

- [RIVERSYS](#) Defines the branch structure and boundary conditions for a river

SUMMARY section keywords

List of SUMMARY section keywords

Segment	Description	Metric units	Field units	Lab units
SRRFLOW	Reach flux through cross-sectional area at current time	m ³ /day	ft ³ /day	cc/hr
SRRAREA	Reach area at current time	m ²	ft ²	cm ²
SRRDEPTH	Reach water depth at current time	m	ft	cm
SRREXCH	Reach exchange flow between river and grid at current time	m ³ /day	ft ³ /day	cc/hr
SRRHEAD	Reach hydraulic head at current time	m	ft	cm
SRRFRODE	Reach Froude number			
SRRVEDER	Reach Vedernikov number			
SRRQR	Reach exchange flow between river and grid over timestep	m ³	ft ³	cc
SRRQT	Reach cumulative exchange flow between river and grid over all time	m ³	ft ³	cc
SRBQR	Branch exchange flow between river and grid over timestep	m ³	ft ³	cc
SRBQT	Branch cumulative exchange flow between river and grid over all time	m ³	ft ³	cc
SRTQR	River total exchange flow between river and grid over timestep	m ³	ft ³	cc
SRTQT	River total cumulative exchange flow between river and grid over all time	m ³	ft ³	cc
SRTFR	Reach tracer flow rate	Tracer unit/day	Tracer unit/day	Tracer unit/hr
SRTFC	Reach tracer concentration	Tracer unit/sm ³	Tracer unit/stb	Tracer unit/scc
SRSFR	Reach brine flow rate	kg/day	lb/day	gm/hr
SRSFC	Reach brine concentration	kg/sm ³	lb/stb	gm/scc

Table 57.1: SUMMARY keywords

Connection	Description	Metric units	Field units	Lab units
CRREXCH	Exchange flow rate between river and grid	m ³ /day	ft ³ /day	cc/hr
CRRPROT	Total water production	m ³	ft ³	cc

Connection	Description	Metric units	Field units	Lab units
CRRINJT	Total water injection	m ³	ft ³	cc

Table 57.2: Flow rate and total water keywords

SCHEDULE section keywords

Keyword	Description
COMPRIV	Specifies the grid blocks which connect to a river
DEBUG	Item 76 controls the output of debug information for rivers. This keyword may also be placed in any other section
GRDREACH	Defines how the river reaches connect to the grid block connections
REACHES	Defines the reaches within a river
RIVDEBUG	Allow debug output for rivers
RIVERSYS	Defines the branch structure and boundary conditions of a river
RIVRPROP	Allow modification of river reach properties
RIVSALT	Allow salt injection in rivers
RIVTRACE	Allow tracers injection in rivers

List of [WRFTPPLT](#) arguments

Argument	Description
READDEPTH	Reach depth
REALENST	Reach length
REALENEN	Reach cumulative distance from river mouth
REAXCORD	X-coordinate of reach node
REAYCORD	Y-coordinate of reach node
REACHFLX	Reach flux
REACHARE	Reach cross-sectional area
REACHDEP	Reach river water depth
REACHEXG	Reach exchange between river and grid
REAFRODE	Reach Froude number
REACHNBR	Reach number
REACHBRN	Reach branch number
REACHNXT	Outlet reach number
BRNST	First reach in river branch
BRNEN	Last reach in river branch
BRANNAME	River/branch name

Table 57.3: RFT, PLT and river data output using WRFTPPLT keyword

List of [RPTSCHED](#) arguments:

Argument	Description
WELSPECS	Output the river reach and connection SCHEDULE section data to the print file.

Table 57.4: RPTSCHED keyword output

Example: A simple river

The following dataset extract outlines the keywords required to simulate a simple river in ECLIPSE 100.

This example shows how to implement a single river channel with upstream and downstream boundary conditions prescribed through the use of mass flow and depth tables respectively. Note that all the data input is in [METRIC](#) units.

The [RIVRDIMS](#) keyword describes the dimensions of the river data to be used in the run and needs to be added to the RUNSPEC section. In this example RIVRDIMS specifies that the model will include one river consisting a single branch which can be made up of a maximum of ten reaches. The number of reaches specified in the table should always be one greater than the number input

The RIVRDIMS keyword also specifies the maximum number of [RIVRXSEC](#), [RBEDCONT](#), [DEPTHHTAB](#), [MASSFLOW](#) and [QHRATING](#) tables that can be included in the run and the maximum number of rows of data expected in the tables. For example, in the current instance a maximum of one massflow table consisting of a maximum of two rows of data can be specified. It should be noted that the number of reaches per river should always be set one greater than the number of reaches input in the [REACHES](#) keyword as ECLIPSE always reserves the first reach for internal use. It should also be noted that ECLIPSE uses additional reaches to accommodate the upstream boundary conditions. As a consequence, there is no flow between these reaches and the grid.

```
RIVRDIMS
--RIVER REACH BRANCH GRID RIVRXSEC RBEDCONT DEPTHHTAB MASSFLOW QHRATING
    1    10      1   10      1     2 1    1 2    1 2 0 2 /
```

The keyword RIVRXSEC is used to describe the cross-section of the river as a function of the depth and is parameterized by the depth, wetted perimeter and the cross-sectional area of the river. In this example the keyword has been used to describe a river with a simple box channel cross-section which is 10 m wide and 5m high. The RBEDCONT keyword is used to define the contact area of a river with a grid block as a function of time. The MASSFLOW and DEPTHHTAB tables are used to define the upstream massflow and downstream depth boundary conditions as functions of time. The following keywords need to be added to the PROPS section for this simple example:

```
RIVRXSEC
    0          10          0
    5.0        20.0        50.0
/
```

```
RBEDCONT
0.0 100.0
5.0 100.0
/
```

```
MASSFLOW
1 JAN 2007 00:00 10.0
4 MAR 2007 00:00 10.0
/
```

```
DEPTHHTAB
1 JAN 2007 00:00 1.0
```

```
4 MAR 2007 00:00 1.0
/
```

In the SCHEDULE section the keyword **RIVERSYS** is used to name the river, prescribe the type of equation to be solved and define which type of boundary conditions will be imposed on the river. This keyword may alternatively be specified in the SOLUTION section. In the dataset extract below the river has been called 'MYRIVER' and it has been specified that the kinematic wave equations will be solved. The upstream and downstream boundary conditions will be prescribed as a function of time through the use of the keywords **MASSFLOW** and **DEPTHTAB** set in the **PROPS** section.

The general structure of the river is defined with the use of the keywords **COMPRIV**, **REACHES** and **GRDREACH**. The keyword **COMPRIV** defines how the river is connected to the grid. In this example, the keyword is used to define a straight line of nine grid blocks which are connected to the river. The **REACHES** keyword is then used to specify the reaches structure of the river. The position of the river mouth, the length of each reach and the depth change along the river reach along with some other parameters such as Manning's roughness coefficient are also set here for each river reach. The keyword **GRDREACH** is then finally used to define how the river reaches defined in **REACHES** are connected to the grid blocks defined in **COMPRIV**. Other parameters such as the productivity index (PI) are set here.

The following keywords need to be added to the SCHEDULE section:

```
RIVERSYS
'MYRIVER' KINEMATI /
1 MYRIVER DOWNSTREAM DEPTH TABLE=1 UPSTREAM MASSFLOW TABLE=1 /
/
```

```
COMPRIV
'MYRIVER' 5 9 1 /
'MYRIVER' 5 8 1 /
'MYRIVER' 5 7 1 /
'MYRIVER' 5 6 1 /
'MYRIVER' 5 5 1 /
'MYRIVER' 5 4 1 /
'MYRIVER' 5 3 1 /
'MYRIVER' 5 2 1 /
'MYRIVER' 5 1 1 /
/
```

```
REACHES
'MYRIVER' 450.0 900.0 0.0 0.0 INC /
2 2 1 MYRIVER 100 0.0 1 0.22e-01 2* /
3 3 2 MYRIVER 100 0.0 1 0.22e-01 2* /
4 4 3 MYRIVER 100 0.0 1 0.22e-01 2* /
5 5 4 MYRIVER 100 0.0 1 0.22e-01 2* /
6 6 5 MYRIVER 100 0.0 1 0.22e-01 2* /
7 7 6 MYRIVER 100 0.0 1 0.22e-01 2* /
8 8 7 MYRIVER 100 0.0 1 0.22e-01 2* /
9 9 8 MYRIVER 100 0.0 1 0.22e-01 2* /
10 10 9 MYRIVER 100 0.0 1 0.22e-01 2* /
/
```

```
GRDREACH
'MYRIVER' /
5 9 1 MYRIVER 2* 1* 'J' 1* 100 0 1.2e-2 3* /
5 8 1 MYRIVER 2* 1* 'J' 1* 100 0 1.2e-2 3* /
5 7 1 MYRIVER 2* 1* 'J' 1* 100 0 1.1e-2 3* /
5 6 1 MYRIVER 2* 1* 'J' 1* 100 0 1.2e-2 3* /
5 5 1 MYRIVER 2* 1* 'J' 1* 100 0 1.2e-2 3* /
5 4 1 MYRIVER 2* 1* 'J' 1* 100 0 1.1e-2 3* /
5 3 1 MYRIVER 2* 1* 'J' 1* 100 0 1.2e-2 3* /
5 2 1 MYRIVER 2* 1* 'J' 1* 100 0 1.2e-2 3* /
5 1 1 MYRIVER 2* 1* 'J' 1* 100 0 1.1e-2 3* /
/
```

In the summary section it is possible to output time-dependent information about the river. In this example the water depth (SRRDEPTH), exchange flow (SRREXCH) and Froude number (SRRFRODE) are output for the fifth reach of 'MYRIVER'. The branch flow (SRBQR) is also output for the first branch in the river.

```
-- River reach water depth
SRRDEPTH
  MYRIVER 5 /
/
-- River reach exchange flow rate
SRREXCH
  MYRIVER 5 /
/
-- River reach Froude number
SRRFRODE
  MYRIVER 5 /
/
-- Branch exchange flow over timestep
SRBQR
  MYRIVER 1 /
/
```

58

Rock compaction

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 ("Tabulated compaction curves")
- The ability to modify the transmissibility as a function of pressure
- A hysteretic model to allow partial reflation
- A cyclic hysteretic model to allow compaction and dilation
- A water-induced compaction model ("Water induced compaction (ECLIPSE 100)")
- The Palmer-Mansoori rock model (Coal Bed Methane option only) ("Palmer-Mansoori rock model")
- The ability to modify the transmissibility as a function of porosity.

Tabulated compaction curves

The compaction of the rock can be modeled as a general function of pressure using the [ROCKTAB](#) keyword. In the case where a grid block pressure increases, the compaction can either be assumed to be reversible or irreversible.

In some cases these two refraction 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 refraction curves can be input to describe the reversible refraction behavior from each starting pressure. For pressures below the starting pressure, the behavior is only partially reversible, given by the deflation curve.

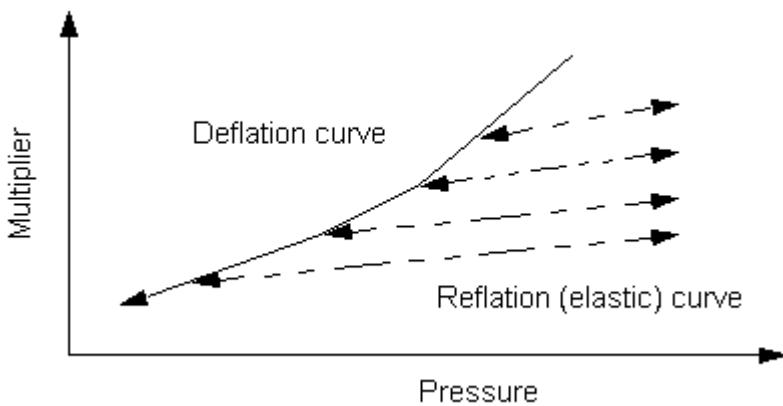


Figure 58.1. Rock Compaction: Hysteresis option

A second hysteretic option can be used to model cyclic dilation and recompaction. This model is based on Beattie, Boberg and McNab (1991). The **ROCKTABH** keyword is again used to input curves that describe the reversible (or elastic) behavior. The reversible region is bounded at low pressures by the deflation (or compaction) curve and at high pressures by a dilation curve which describes partially reversible expansion. This option can be selected using the **ROCKCOMP** keyword.

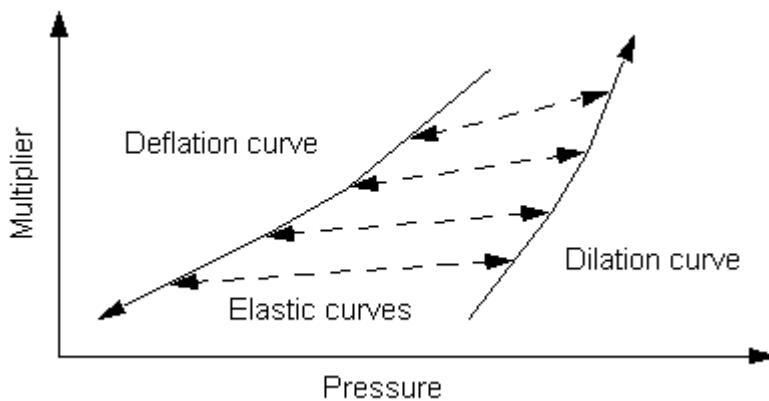


Figure 58.2. Rock Compaction: Boberg Hysteresis option

Initially it is assumed the reservoir rock is fully compacted. The initial pore volume and transmissibility multipliers will be set to the deflation curve value at the cell pressure. In general these multipliers will not be equal to one, so the initial pore volume and transmissibility will differ from the values entered in the data file. Alternatively, the rock compaction multipliers can be tabulated as functions of pressure change (ΔP), and the **ROCKOPTS** keyword can be used to store the initial pressure (see below). The initial multipliers will then be set to the deflation curve values at $\Delta P = 0$, which can be set to 1.

If the initial reservoir rock is not fully compacted, pore volume and transmissibility values can be set by finding an elastic curve with pore volume multiplier equal to one at the initial cell pressure. In this case the pore volume will be the same as the value entered in the data file, although the transmissibility may still differ. This option can be selected using the **ROCKOPTS** keyword.

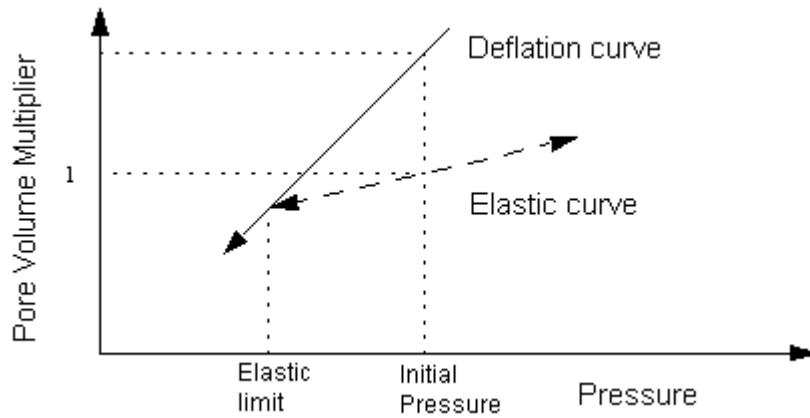


Figure 58.3. Initial multipliers

Transmissibility modification

In addition to the compaction (the reduction in the pore volume), it is possible to modify the transmissibility as a function of pressure.

ECLIPSE 100

At present the transmissibility modification is treated explicitly, that is the modification due to a change in pressure only affects the subsequent timestep.

If a transmissibility multiplier is specified with the [ROCKTAB](#) or [ROCKTABH](#) keywords, the well connection transmissibility factor, grid block transmissibilities and NNCs are modified accordingly.

ECLIPSE 300

The generalized mobility terms are updated for grid cells and well connections. Similar functionality, but based on the corresponding changes in porosity, is also available by specifying the keywords [ROCKTRMX](#), [ROCKTRMY](#) and [ROCKTRMZ](#).

In dual porosity runs, it is also possible to specify pressure tables of the sigma multiplier in order to modify the matrix-fracture coupling as a result of rock compaction. Keywords [ROCKTSIG](#) or [ROCKTHSG](#) provide the input of irreversible/reversible and hysteretic tables, respectively.

Overburden pressure

A reservoir simulator typically works in terms of pressure, as pressure is the natural variable when considering the flow. However, it is often better to consider the compaction as a function of the effective stress on the rock, which is the overburden pressure minus the fluid pressure. The [ROCKTAB](#), [ROCKTABH](#) or [ROCK2D](#) and [ROCK2DTR](#) keywords tabulate the compaction multipliers against the effective fluid pressure instead $P_{eff\ fluid} = P_{fluid} - \text{Overburden}$.

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 $-P_{eff\ fluid}$.

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 made equivalence to the initial reservoir pore volume in ECLIPSE by setting the second item of the [ROCKOPTS](#) keyword to ‘ STORE ’.

With the rock compaction model (keywords [ROCKTAB](#), [ROCKTABH](#)), setting the second item in [ROCKOPTS](#) to ‘ STORE ’ and tabulating the [ROCKTAB](#) and [ROCKTABH](#) tables against effective pressure (which is zero at equilibrated conditions) treats the initial PORO and PORV as reservoir conditions.

Transmissibility as a function of porosity

In order to define a model which can be used to specify the dependence of transmissibility on porosity, refer to items 4 and 5 of the [ROCKCOMP](#) keyword. There are two different options which can be used:

- EXP Transmissibility is as exponential function of porosity:

$$\frac{\kappa}{\kappa_0} = e^{\left[\kappa_{\text{mult}} \left(\frac{\varphi - \varphi_0}{1 - \varphi_0} \right) \right]}$$

where

κ_0 is the initial transmissibility,

φ_0 is the initial porosity and

κ_{mult} is the user-defined multiplier factor defined by keywords, [ROCKTRMX](#), [ROCKTRMY](#) and [ROCKTRMZ](#) for a particular direction. This model is based on the general approach as described by Lebel (2002) [Ref. 83].

- CZ Transmissibility is a function of porosity via the Carmen-Kozeny type formula:

$$\frac{\kappa}{\kappa_0} = \frac{\varphi}{\varphi_0} \tau \left[\left(\frac{1 - \varphi_0}{1 - \varphi} \right) \right]$$

where τ is an integer defined in item 5 of the [ROCKCOMP](#) keyword.

Items 4 and 5 of the [ROCKCOMP](#) keyword are available for use with the [ROCKTAB](#) and [ROCKTABH](#) keywords. They are currently not available for use with the [ROCK](#) keyword.

Output

Both the modified pore volumes and the transmissibilities can be output the Print and RESTART files.

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 arguments used in the print and RESTART files are:

Argument	Description
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.

Argument	Description
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:

- 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.

- A full two-dimensional table of pore volume multipliers as a function of both pressure and water saturation:

$$V = V_G \cdot M(P, Sw_{max} - Sw_{ini})$$

where $M(P, S)$ is a two-dimensional table entered using the [ROCK2D](#) and [ROCKWNOD](#) keywords.

The transmissibility modification is handled in a similar manner. If method 1 is used to input the compaction data, the transmissibility modification is the product of the pressure and saturation multipliers. If method 2 is used, transmissibility modifiers can be entered using an analogous two dimensional table with the [ROCK2DTR](#) keyword.

Palmer-Mansoori rock model

Note: This is only available in the coal bed methane option (see the [COAL](#) keyword).

The Palmer-Mansoori rock model ([Ref. 78]) adjusts the pore volumes, $V(P)$, at the current pressure, P , using

$$V(P) = V(P_0) \left\{ 1 + \frac{c_m}{\varphi_0} (P - P_0) + \frac{\varepsilon_l}{\varphi_0} \left(\frac{K}{M} - 1 \right) \left(\frac{\beta P}{1 + \beta P} - \frac{\beta P_0}{1 + \beta P_0} \right) \right\}$$

where:

φ_0 is the initial porosity

P_0 is the initial pressure

$V(P_0)$ is the pore volume at the initial pressure

K	is the bulk modulus
M	is the constrained axial modulus
γ	is the grain compressibility
β, ε_l	are parameters of the match between the Langmuir curve and the change in volumetric strain due to matrix shrinkage
f	is a fraction between 0 and 1
$c_m = \frac{g}{M} \left[\frac{K}{M} + f - 1 \right]^\gamma$	where g is a geometric factor (typical in the range 0–1), related to the orientation of the fracture system, see [Ref. 136].

Following Palmer-Mansoori, assuming that permeability varies with porosity as

$$\frac{k}{k_0} = \left(\frac{\varphi}{\varphi_0} \right)^n$$

where the exponent n is typically 3, then the transmissibility multiplier is taken as

$$\left\{ 1 + \frac{c_m}{\varphi_0} (P - P_0) + \frac{\varepsilon_l}{\varphi_0} \left(\frac{K}{M} - 1 \right) \left(\frac{\beta P}{1 + \beta P} - \frac{\beta P_0}{1 + \beta P_0} \right) \right\}^n$$

This model is activated using the `PALM-MAN` option on item 1 of the `RUNSPEC` section keyword `ROCKCOMP`. The parameters for the model are set in the `PROPS` section keyword `ROCKPAMA`.

ECLIPSE 300

Cells not connected to coal cells only apply the compression term to the pore volume and permeability multipliers.

Compositional coal swelling/shrinkage model (ECLIPSE 300)

A modified form of the Palmer-Mansoori rock model is available by using the `ROCKPAME` keyword. A similar approach is described in [Ref. 73]. An important difference is that the model described here will not predict volume changes for undersaturated coal. Each component of the system can be assigned Langmuir curve parameters β_k, ε_k , making the swelling or shrinkage of the coal dependent on composition. The strain of one component is computed as if the coal gas is instantly in equilibrium following the extended Langmuir formula for adsorption capacity

$$e_k = \frac{\varepsilon_k \beta_k a_k P}{1 + \sum \beta_j a_j P}$$

where $a_k = L_k / (\sum L_j)$ represent the adsorbed mole fraction of component k , P the pressure and L_k are the adsorption capacities of each component (see keyword `LANGMEXT`).

The total strain is then computed as

$$E = \sum e_k$$

Using this in the Palmer-Mansoori equation the pore volume, $V(P, Y)$, at the current pressure, P , and gas phase composition $Y = (y_1, y_2, \dots, y_{nc})$ are adjusted using

$$V(P, Y) = V_0 \left\{ 1 + \frac{c_m}{\varphi_0} (P - P_0) + \frac{1}{\varphi_0} \left(\frac{K}{M} - 1 \right) (E - E_0) \right\}$$

where:

φ_0 is the initial porosity

P_0 is the initial pressure

V_0 is the pore volume at the initial pressure and composition

K is the bulk modulus

M is the constrained axial modulus

γ is the grain compressibility

β_k, ε_k are parameters of the match between the Langmuir curve and the change in volumetric strain due to matrix shrinkage for component k.

f is a fraction between 0 and 1

the term c_m is given as $c_m = \frac{1}{M} \left[\frac{K}{M} + f - 1 \right] \gamma$

For undersaturated coal where the adsorbed gas is not changing the formula is modified so the strain is constant. The strain used then corresponds to the state where the gas would start to desorb.

Note also that the [ROCKPAME](#) keyword only apply to regions with cells connected to coal cells as assigned by [COALNUM](#).

Note: The rock compaction models takes the input porosities as the porosities at reservoir conditions. In areas without coal different rock compaction models can be used but some care must be taken so the input porosities are treated consistently for the entire model.

Saturation functions

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

Saturation functions are used to describe properties that depend on the phase saturations, namely the relative permeabilities and capillary pressures. The phase saturations are linked by the relationship:

$$S_o + S_g + S_w = 1 \quad \text{Eq. 59.1}$$

In two-phase systems there is only one independent saturation and the properties are one dimensional functions. For example, in an oil-water system $S_g = 0$ and properties are a function of either S_w or $S_o = 1 - S_w$. Data for these one dimensional functions are entered as tables (keywords [SWOF](#) and [SGOF](#) / [SLGOF](#) or the alternative family [SWFN](#), [SGFN](#) and [SOF2](#)).

In three-phase systems there are two independent saturations and properties can be either one or two dimensional functions. A number of ways of modelling three phase properties are available:

- Capillary pressures and the water and gas relative permeabilities are treated as one dimensional functions, but the oil relative permeability is treated as a two dimensional function by interpolating the oil-gas and oil-water relative permeabilities. Data is entered as one dimensional tables using either of the families of the two phase keywords, except that [SOF3](#) is needed instead of [SOF2](#) to describe the two oil relative permeabilities. Interpolation of the oil relative permeability is performed using either the default method or one of Stone's methods.
- Two dimensional tables [SOF32D](#), [SGF32D](#) and [SWF32D](#) can be entered for any (or all) of the phase relative permeabilities. The capillary pressures are described either by using two dimensional tables [PCG32D](#), [PCW32D](#) (ECLIPSE 100) or by using the one dimensional keywords (ECLIPSE 300).
- The Baker methods can be used to describe any (or all) of the phase relative permeabilities as two dimensional functions. Data is entered as one dimensional tables using the keywords [SOF3](#), [SGFN](#) (or [SGF3](#)) and [SWFN](#) (or [SWF3](#)).

ECLIPSE 300

The methods described above can be applied to individual phases, and different methods can be used for each phase. Two further methods are available that apply to all phases:

ECLIPSE 300

- The IKU three-phase method. Data is entered as one dimensional tables using the keywords [SOF3](#), [SGF3](#) and [SWF3](#)

ECLIPSE 300

- The ODD3P three phase method. Data is entered as one dimensional tables using the keywords [SOF3](#), [SGF3](#) and [SWF3](#).

Saturation table keywords

All the saturation functions may be entered in multi-table form; tables are associated with cells using the [SATNUM](#) keyword and its hysteresis counterparts.

One dimensional tables (for two or three phases systems)

There are two families of keywords that define one-dimensional 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.

The keywords for the first family of tables are:

Keyword	Description
SWOF	Sets water relative permeability $k_{rw}(S_w)$, oil relative permeability in water $k_{row}(S_w)$, and water-oil capillary pressure $P_{cow}(S_w)$ as a function of water saturation. This can be used for two or three-phase systems with both water and oil.
SGOF	Sets gas relative permeability $k_{rg}(S_g)$, oil relative permeability in gas at connate water $k_{rog}(S_g)$, and oil-gas capillary pressure $P_{cog}(S_g)$ as a function of gas saturation. Either SGOF or SLGOF can be used for two or three-phase systems with both gas and oil.
SLGOF	Sets gas relative permeability $k_{rg}(S_l)$, oil relative permeability in gas at connate water $k_{rog}(S_l)$, and oil-gas capillary pressure $P_{cog}(S_l)$ as a function of liquid saturation $S_l = S_o + S_w = 1 - S_g$. Either SGOF or SLGOF can be used for two or three-phase systems with both gas and oil.

Table 59.1: Family (i) saturation table keywords

The keywords for the second family of tables are:

Restriction	Keyword	Description
	SWFN	Sets water relative permeability $k_{rw}(S_w)$ and capillary pressure $P_{cow}(S_w)$ as a function of water saturation. This can be used for two or three-phase systems with water.
	SGFN	Sets gas relative permeability $k_{rg}(S_g)$ and capillary pressure $P_{cog}(S_g)$ as a function of gas saturation. This can be used for two or three-phase systems with gas.
	SOF2	Sets oil relative permeability $k_{ro}(S_o)$ as a function of oil saturation. This can be used for two-phase systems with oil.
	SOF3	Sets relative permeability of oil in water $k_{row}(S_o)$, and oil in gas $k_{rog}(S_o)$ at the connate water saturation, as a function of oil saturation. This can be used for three-phase systems with oil.

Restriction	Keyword	Description
E100 only	SGWFN	Sets the gas $k_{rg}(S_g)$ and water $k_{rw}(S_g)$ relative permeabilities and the gas-water capillary pressure $P_{cgw}(S_g)$ for two-phase systems with only gas and water present as a function of gas saturation $S_g = 1 - S_w$. (This keyword is optional, as the gas and water saturation functions could alternatively be supplied 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.)

Table 59.2: Family (ii) saturation table keywords

Two dimensional tables (for three phase systems)

For three phase systems, two dimensional tables can be used to describe saturation properties. The keywords are:

Restriction	Keyword	Description
	SOF32D	Sets the oil relative permeability $k_{ro}(S_w, S_g)$ in a three-phase system as a function of both water saturation and gas saturation.
	SWF32D	Sets the water relative permeability $k_{rw}(S_o, S_g)$ in a three-phase system as a function of both oil saturation and gas saturation.
	SGF32D	Sets the gas relative permeability $k_{rg}(S_o, S_w)$ in a three-phase system as a function of both oil saturation and water saturation.
E100 only	PCW32D	Sets the water/oil capillary pressure $P_{cow}(S_o, S_g)$ in a three-phase system as a function of both oil saturation and gas saturation.
E100 only	PCG32D	Sets the gas/oil capillary pressure $P_{cog}(S_o, S_w)$ in a three-phase system as a function of both oil saturation and water saturation.

Table 59.3: Two dimensional saturation tables

One dimensional tables for three phase models (Baker, IKU and ODD3P)

This is required for three-phase systems which use the Baker (see [BAKER1](#) and [BAKER2](#) keywords), IKU (see [IKU3P](#) keyword) or the ODD3P (see the [ODD3P](#) keyword) relative permeability models.

The keywords are:

Restriction	Keyword	Description
E300 only	SWF3	Sets water relative permeability to oil $k_{rwo}(S_w)$ and gas $k_{rwg}(S_w)$ and capillary pressure $P_{cow}(S_w)$ as a function of water saturation.
E300 only	SGF3	Sets gas relative permeability to oil $k_{rgo}(S_g)$ and water $k_{rgw}(S_g)$ and capillary pressure $P_{cog}(S_g)$ as a function of gas saturation.

Restriction	Keyword	Description
E300 only	SOF3	Sets relative permeability of oil in water $k_{row}(S_o)$, and oil in gas $k_{rog}(S_o)$ at the connate water saturation, as a function of oil saturation.

Table 59.4: One dimensional tables for three phase models

Saturation table compatibility

Sufficient data must be provided to allow the simulators to calculate relative permeabilities and capillary pressures for all phases present. In two-phase systems, the one dimensional tables must be used. In general, either family can be used, however family (ii) must be used if running

ECLIPSE 100

- Miscible Flood

ECLIPSE 100

- Solvent options

ECLIPSE 100

- Two-phase gas-water systems

Keywords from the different families must not be mixed in the same run.

In three-phase systems, both one and two dimensional tables can be used. If one dimensional tables (of either family) are supplied, then two dimensional interpolation methods, such as those of Stone can be used.

ECLIPSE 300

Other models for three phase systems, Baker, IKU and ODD3P require specific tables: [SOF3](#), [SGF3](#) and [SWF3](#).

If a two-dimensional table [SOF32D](#) is used to describe the oil relative permeability $k_{ro}(S_w, S_g)$, either one or two dimensional tables can be used to describe the gas and water relative permeabilities. If the family (i) one dimensional tables ([SWOF](#) / [SGOF](#)) are used, then the one dimension oil relative permeabilities in these tables ($k_{row}(S_w)/k_{rog}(S_g)$) will be ignored.

If two-dimensional tables are used to describe the water or gas relative permeability, then capillary pressure data can be supplied as follows:

- ECLIPSE 100

Two-dimensional tables ([PCG32D](#) / [PCW32D](#))

ECLIPSE 300

- One dimensional table, in which case the one dimensional relative permeability in the table will be ignored.

Saturation table lookup

Saturation values need not be specified at equal intervals, and are used in the specified form, rather than being interpolated onto regular intervals.

Default values represented by 1* may be inserted as required in one or more rows for columns 2 onwards for the one-dimensional table keywords [SWFN](#), [SWF3](#), [SWOF](#), [SGFN](#), [SGWFN](#), [SGF3](#), [SGOF](#), [SLGOF](#), [SOF2](#) and [SOF3](#). Where default values have been entered, the actual values will be calculated by linear interpolation using the nearest non-default neighbors and substituted into the table.

However, where only one non-default value is entered in an entire column or where default values are entered at the start or end of the tables, the interpolation behavior for ECLIPSE 100 and ECLIPSE 300 can differ. For example, for the SGOF keyword, if the capillary pressure column is defaulted throughout, the

values are replaced with 0.0 throughout by both simulators. However, if only one value is not defaulted and this is not the first value in the table, in ECLIPSE 100, the first value is set to 0.0, linear interpolation is performed to replace any default values up to the defined value, and all subsequent values are assigned to the defined value. In contrast, in ECLIPSE 300, all defaulted values in the column are set equal to the single non-defaulted value. It is recommended, therefore, that when defaults are entered into a table, the table is reported using the [RPTPROPS](#) keyword and the resultant values inspected to verify the intended interpolation behavior.

Tolerance for critical saturations

Critical saturations are assigned from the saturation tables by establishing the highest value of saturation for which the associated relative permeability is zero. In practice, the exceeding of a threshold (tolerance) value is used to test whether the relative permeability value is zero or non-zero. For ECLIPSE 100 and 300 the default thresholds are 1×10^{-6} and 1×10^{-20} respectively. These default values may be modified using the [TOLCRIT](#) keyword.

Water saturation properties

One dimensional tables

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 water 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
...		
...		

(Note that ECLIPSE can handle zero gradients in the capillary pressure versus saturation, as shown in the above example between the S_w values 0.2 & 0.22.)

$S_{w\max}$ The maximum water saturation value in the table.

The equilibration calculation sets the water saturation to this value in grid blocks that lie below the water transition zone.
In the example shown above, $S_{w\max} = 1.0$, which results in the water zone being fully saturated with water.

In an **oil-wet system**, in order to use correctly the Killough wetting phase model on the oil phase (option 7, item 2 of [EHYSTR](#) keyword), $S_{w\max}$ should equal $1.0 - S_{oco}$, S_{oco} being the connate oil saturation.

Two dimensional tables

A full 2-dimensional table of water relative permeability versus oil and gas saturations can be entered in three-phase runs using the [SWF32D](#) keyword.

When using [SWF32D](#) the connate water is defined as $S_{wco} = 1.0 - S_{o,max}$.

Keyword [PCW32D](#) can be used in conjunction with [SWF32D](#) to enter the water/oil capillary pressure if required.

Gas saturation properties

One dimensional tables

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 gas 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 gas transition zone). In this respect S_{gco} is the **connate** gas saturation.

ECLIPSE 100

Normally in ECLIPSE 100 $S_{gco} = 0.0$, as in the example above.

ECLIPSE 300

In ECLIPSE 300 $S_{gco} = 0.0$.

$S_{g\ max}$ The maximum gas saturation value in the table. The equilibration calculation sets the gas saturation to this value in grid blocks that lie above the gas transition zone. Normally $S_{g\ max} = 1.0 - S_{wco}$, as in the example above, with $S_{wco} = 0.22$ as in the first [SWFN](#) table example in this chapter.

In an **oil-wet system**, in order to use correctly the Killough wetting phase model on the oil phase (option 7, Item 2 of [EHYSTR](#) keyword), $S_{g\ max}$ should equal $1.0 - S_{oco}$, S_{oco} being the connate oil saturation.

Two dimensional tables

A full 2-dimensional table of gas relative permeability versus oil and water saturations can be entered using the [SGF32D](#) keyword, in three-phase runs.

ECLIPSE 100

Keyword [PCG32D](#) can be used in conjunction with [SWF32D](#) to enter the gas/oil capillary pressure if required.

Three phase gas relative permeability models

A number of formulae are available for calculating the two-dimensional three-phase gas relative permeability at particular oil and water saturations, from the input one-dimensional relative permeabilities:

ECLIPSE 300

- Default method

ECLIPSE 300

- Baker 1 (can also be used for oil and water)
- ECLIPSE 300*
- Baker 2 (can also be used for oil and water)
- ECLIPSE 300*
- IKU (used for all phases)
- ECLIPSE 300*
- ODD3P (used for all phases)

The formulae are not needed if the [SGF32D](#) keyword has been used to input the three-phase oil relative permeability as a two-dimensional table versus oil and gas saturations (see above).

Default formula

ECLIPSE 300

The gas relative permeability is given by

$$k_{rg} = \frac{S_o k_{rgo} + (S_w - S_{wco}) k_{rgw}}{S_o + S_w - S_{wco}} \quad \text{Eq. 59.2}$$

The formula is essentially the same as the default formula for oil in "[Three phase oil relative permeability models](#)".

Here

k_{rgo} is the gas relative permeability for a system with oil, gas and connate water

k_{rgw} is the gas relative permeability for a system with gas and water only

k_{rgo} is calculated by lookup of the one-dimensional gas relative permeability table. The value is modified if the [MISCIBLE](#) option is used. The value is further modified if the default HCSCAL option is used. For more information, see [FORMOPTS](#).

The k_{rgw} calculation depends on whether the [NOMIX](#) keyword has been specified. If it has, the value is calculated by lookup of the one-dimensional gas relative permeability table. If [NOMIX](#) has not been specified, the oil and gas relative permeability tables are interpolated. See "[Near critical oil and gas relative permeabilities](#)" for details.

Oil saturation properties

One dimensional tables

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 two-phase run).

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 relative 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 $1 - S_{wco} - S_{oco}$ as determined from the SWFN table. The two oil relative permeabilities should be the same at $S_{o \max}$ (both represent a case with $S_o = S_{o \max}$, $S_w = S_{wco}$ and $S_g = 0$). ECLIPSE reports an error if this condition is not satisfied.
In an oil-wet system , normally $S_{o \max} = 1.0$.	

Two dimensional tables

A full 2-dimensional table of oil relative permeabilities versus water and gas saturations can be input using the [SOF32D](#) keyword, in three-phase runs. 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										
--SGAS	0.22	0.27	0.32	0.37	0.42	0.47	0.52	0.57	0.72	0.77	0.78 /

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	0.000 /	
0.15	0.215	0.103	0.069	0.036	0.017	0.003	0.002	0.001	0.000	0.000 /	
0.20	0.100	0.065	0.031	0.015	0.002	0.002	0.001	0.000	0.000	0.000 /	
0.25	0.060	0.025	0.014	0.002	0.001	0.001	0.000	0.000	0.000	0.000 /	
0.30	0.020	0.012	0.001	0.001	0.001	0.000	0.000	0.000	0.000	0.000 /	
0.35	0.010	0.001	0.001	0.001	0.000	0.000	0.000	0.000	0.000	0.000 /	
0.40	0.000	0.000	0.000	0.000	0.000	0.000	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 number of formulae are available for calculating the two dimensional three-phase oil relative permeability at particular water and gas saturations, from the input one dimensional relative permeabilities of oil in water and oil in gas and connate water:

- Default method
 - Stone 1 (modified)
 - Stone 2 (modified)
- ECLIPSE 300 • Baker 1 (can also be used for gas and water)
- ECLIPSE 300 • Baker 2 (can also be used for gas and water)
- ECLIPSE 300 • IKU (used for all phases)
- ECLIPSE 300 • ODD3P (used for all phases)

The formulae are not needed if the [SOF32D](#) keyword has been used to input the three-phase oil relative permeability as a two-dimensional table versus water and gas saturations (see above).

Default formula

The default model for the three-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 (for example poor conditioning and negative values).

The default model assumed by ECLIPSE is shown in figure 59.1. The oil saturation is assumed to be constant and equal to the block average value, S_o , throughout the cell. The gas and water are assumed to be completely segregated, except that the water saturation in the gas zone is equal to the connate saturation, S_{wco} . The full breakdown, assuming the block average saturations are S_o , S_w and S_g (with $S_o + S_w + S_g = 1$) is as follows:

In a fraction $S_g / (S_g + S_w - S_{wco})$ of the cell (the gas zone),

the oil saturation is S_o

the water saturation is S_{wco}

the gas saturation is $S_g + S_w - S_{wco}$

In a fraction $(S_w - S_{wco}) / (S_g + S_w - S_{wco})$ of the cell (the water zone),

the oil saturation is S_o

the water saturation is $S_g + S_w$

the gas saturation is 0

The oil relative permeability is then given by

$$k_{ro} = \frac{S_g k_{rog} + (S_w - S_{wco}) k_{row}}{S_g + S_w - S_{wco}} \quad \text{Eq. 59.3}$$

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 .

For the default model, the values of $k_{rog}(S_o)$ and $k_{row}(S_o)$ are looked up as functions of the oil saturation $S_o = 1 - S_w - S_g$. Both two-phase oil relative permeability functions are tabulated as functions of oil saturation in the input data either directly if the Family (ii) set of keywords is used or indirectly via internal tabular conversion if the Family (i) set of keywords is used.

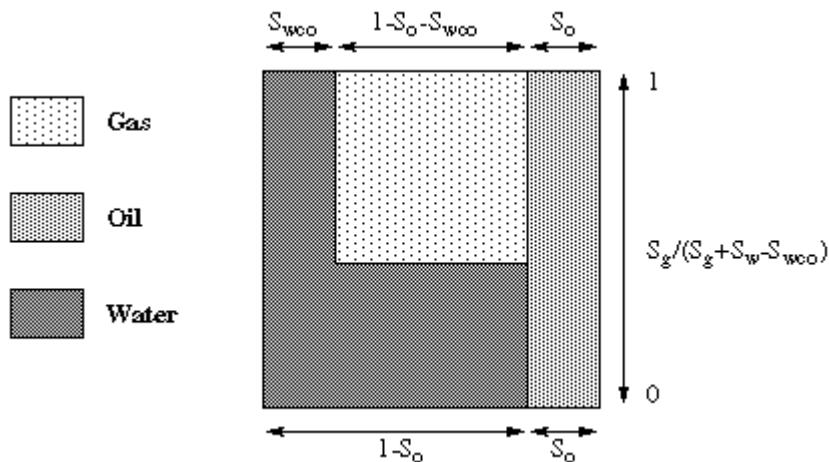


Figure 59.1. The default three-phase oil relative permeability model assumed by ECLIPSE

Stone's first model (modified)

A modified version of Stone's first three-phase oil relative permeability model [Ref. 106], summarized in [Ref. 8], is available with keyword `STONE1` in the PROPS section. The formula is

$$k_{ro} = k_{rocw} S S_o F_w F_g \quad \text{Eq. 59.4}$$

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

$$SS_w = (S_w - S_{wco}) / (1 - S_{wco} - S_{om}) \text{ when } S_w > S_{wco}$$

$$SS_g = S_g / (1 - S_{wco} - S_{om})$$

where

S_o , S_w and S_g represent the 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

k_{row} denotes the oil relative permeability for a system with oil and water only

k_{rocw} denotes the oil relative permeability in the presence of connate water only

S_{om} is the minimum residual oil saturation

For Stone's first model, the values of $k_{rog}(S_{og})$ and $k_{row}(S_{ow})$ are looked up as two-phase saturation functions where $S_{og} = 1 - S_{wco} - S_g$ and $S_{ow} = 1 - S_w$. Both two phase oil relative permeability functions are tabulated as functions of oil saturation in the input data either directly if the family (ii) set of keywords is used or indirectly via internal tabular conversion if the family (i) set of keywords is used.

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 the following figure:

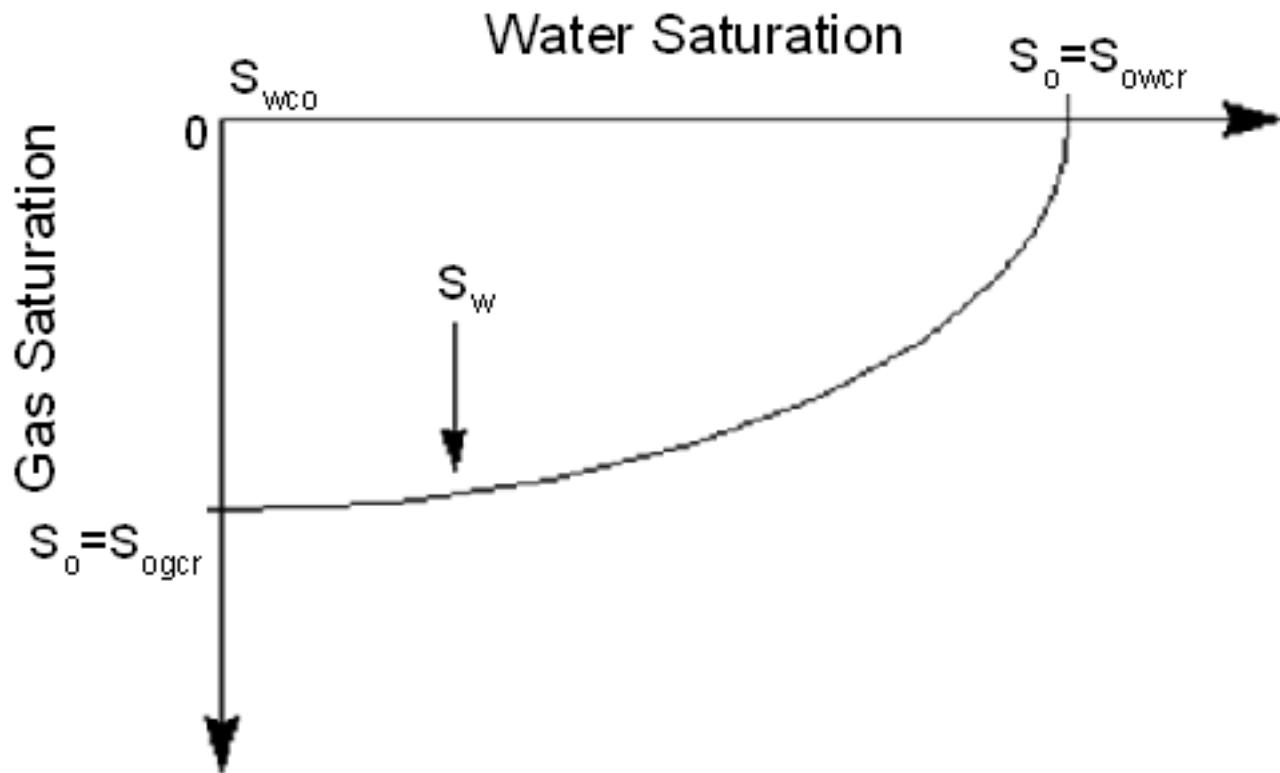


Figure 59.2. Calculating the minimum residual oil saturation from the water saturation

If the SOMGAS keyword is used, S_{om} is interpolated from the gas saturation as shown in the following figure:

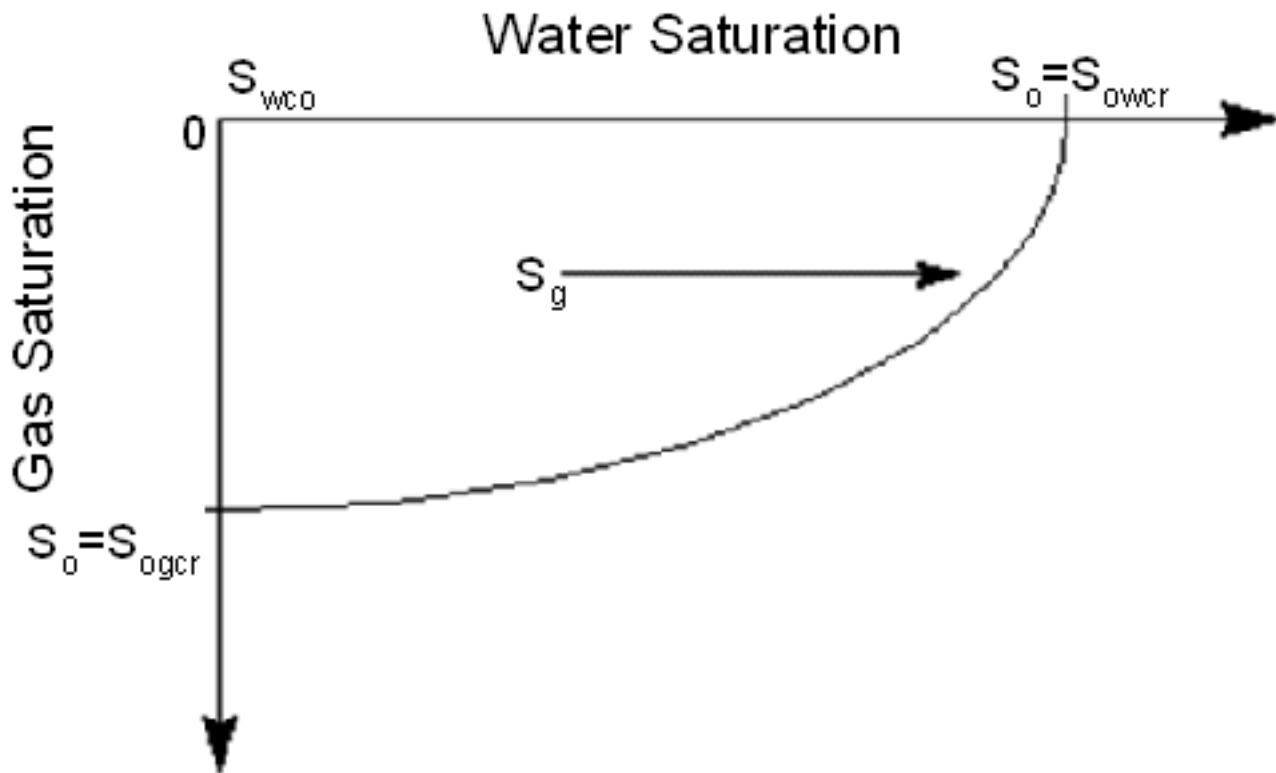


Figure 59.3. Calculating the minimum residual oil saturation from the gas saturation

If the hysteresis option is active, the drainage and imbibition S_{om} values are taken from the [SATNUM](#) and [IMBNUM](#) tables. For example in the SOMWAT case:

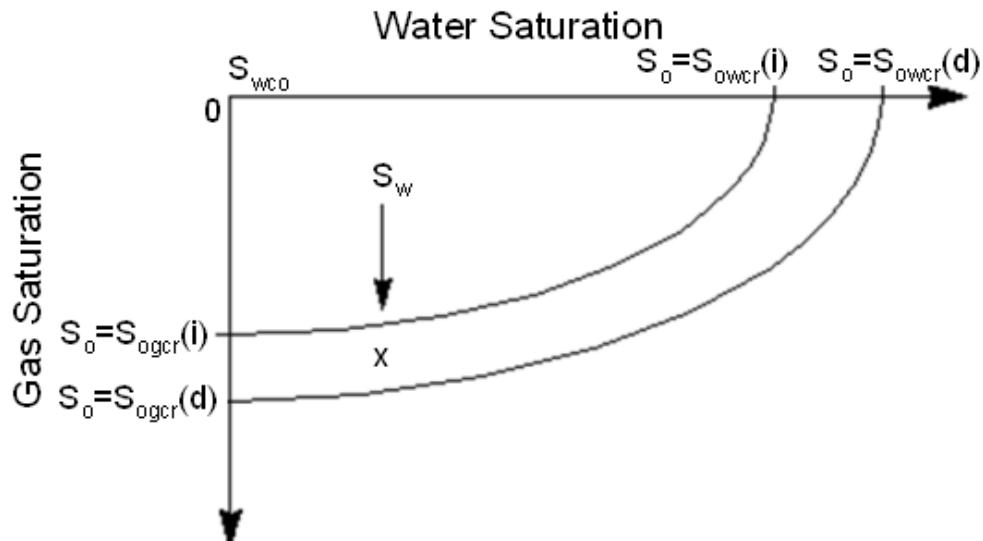


Figure 59.4. Calculating the minimum residual oil saturation for drainage and imbibition with hysteresis

The S_{om} value (point x in the picture above) is calculated as follows:

$$S_{om} = S_{om}^{(i)} + F \cdot (S_{om}^{(d)} - S_{om}^{(i)}) \quad \text{Eq. 59.5}$$

$$F = F_{sw} F_w + (1-F_{sw}) F_g \quad \text{Eq. 59.6}$$

$$F_{sw} = \frac{S_w - S_{wco}}{1 - S_{wco} - S_{owcr}^{(h)}} \quad \text{Eq. 59.7}$$

$$F_w = \frac{S_{owcr}^{(i)} - S_{owcr}^{(h)}}{S_{owcr}^{(i)} - S_{owcr}^{(d)}} \quad \text{Eq. 59.8}$$

$$F_g = \frac{S_{ogcr}^{(i)} - S_{ogcr}^{(h)}}{S_{ogcr}^{(i)} - S_{ogcr}^{(d)}} \quad \text{Eq. 59.9}$$

$S_{owcr}^{(i)}$ is the critical oil-to-water imbibition

$S_{owcr}^{(d)}$ is the critical oil-to-water drainage

$S_{owcr}^{(h)}$ is the critical oil-to-water actual

$S_{ogcr}^{(i)}$ is the critical oil-to-gas imbibition

$S_{ogcr}^{(d)}$ is the critical oil-to-gas drainage

$S_{ogcr}^{(h)}$ is the critical oil-to-gas actual

The calculation of three phase oil relative permeability using Stone's method 1 is enabled by means of the **STONE1** keyword in the **PROPS** section of the input data. Tabulated output of the three phase oil relative permeability values is controlled by the mnemonic **SOF** in the **RPTPROPS** keyword.

ECLIPSE 300

If **STONEPAR** is used, this is obtained as a function of the A and B parameters:

$$S_{om} = S_{ogcr} X_g^A + S_{owcr} (1 - X_g)^A + B S_{gm}^2 X_g (1 - X_g) \quad \text{Eq. 59.10}$$

where:

$$S_{gm} = 1 - S_{wco} - S_{ogcr} \quad \text{Eq. 59.11}$$

Stone's first model exponent

The formula for Stone's first model may be rearranged such that an exponent may be applied to the combination of saturation terms:

$$k_{ro} = \beta (k_{row} k_{rog}) / k_{rocw} \quad \text{Eq. 59.12}$$

where

$$\beta = \left(\frac{SS_o}{(1-SS_w) \cdot (1-SS_g)} \right)^\eta \quad \text{Eq. 59.13}$$

This formulation is due to Hustad and Holt [Ref. 59]. The β term may be interpreted as a variable that varies between zero and one for low and high oil saturations respectively. If the exponent η is equal to 1.0, the formula corresponds to Stone's first model. Values above 1.0 cause the oil isopersms to spread within the saturation space, whereas values below 1.0 have the opposite effect. The value of the exponent may be used, therefore, to match the predicted oil recovery to the observed data and may be specified using the **STONE1EX** keyword.

Stone's second model (modified)

A modified version of Stone's second three phase oil relative permeability model [Ref. 105], summarized in [Ref. 8] can be used. The formula is

$$k_{ro} = k_{rocw} \left[\left(\frac{k_{row}}{k_{rocw}} + k_{rw} \right) \left(\frac{k_{rog}}{k_{rocw}} + k_{rg} \right) - k_{rw} - k_{rg} \right] \quad \text{Eq. 59.14}$$

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

k_{rocw} is the oil relative permeability in the presence of connate water only

Both sets of two-phase oil relative permeability functions are tabulated as functions of oil saturation in the input data either directly if the Family (ii) set of keywords is used or indirectly via internal tabular conversion if the Family (i) set of keywords is used. For Stone's second model, the values of $k_{rog}(S_{og})$ and $k_{row}(S_{ow})$ are looked up as two-phase saturation functions where $S_{og} = 1 - S_{wco} - S_g$ and $S_{ow} = 1 - S_w$.

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.

Three phase relative permeability models

Baker method

ECLIPSE 300

Baker's two interpolation methods [Ref. 9] can be used to determine three-phase relative permeability. The methods are selected by using the **BAKER1** or **BAKER2** keywords in the PROPS section. The Baker methods can be applied to any (or all) phases.

The keyword **BAKER1** is used to perform linear interpolation based on the relations:

$$k_{ro} = \frac{(S_g - S_{gc})K_{rog} + (S_w - S_{wc})K_{row}}{(S_g - S_{gc}) + (S_w - S_{wc})} \quad \text{Eq. 59.15}$$

$$k_{rg} = \frac{(S_o - S_{oc})K_{rgo} + (S_w - S_{wc})K_{rgw}}{(S_o - S_{oc}) + (S_w - S_{wc})} \quad \text{Eq. 59.16}$$

$$k_{rw} = \frac{(S_o - S_{oc})K_{rwo} + (S_g - S_{gc})K_{rwg}}{(S_o - S_{oc}) + (S_g - S_{gc})} \quad \text{Eq. 59.17}$$

The keyword `BAKER2` is used to perform linear interpolation based on the relations:

$$k_{ro} = \frac{S_g K_{rog} + S_w K_{row}}{S_g + S_w} \quad \text{Eq. 59.18}$$

$$k_{rg} = \frac{S_o K_{rgo} + S_w K_{rgw}}{S_o + S_w} \quad \text{Eq. 59.19}$$

$$k_{rw} = \frac{S_o K_{rwo} + S_g K_{rwg}}{S_o + S_g} \quad \text{Eq. 59.20}$$

$K_{rog}(S_o)$ and $K_{row}(S_o)$ are looked up from the `SOF3` table.

$K_{rgo}(S_g)$ and $K_{rgw}(S_g)$ are looked up from the `SGF3` table.

$K_{rwo}(S_w)$ and $K_{rwg}(S_w)$ are looked up from the `SWF3` table.

This linear interpolation may be shown geometrically on a ternary diagram similar to figure 59.5.

IKU method

ECLIPSE 300

The IKU model, which is due to Hustad [Ref. 58] and [Ref. 60], is an alternative three-phase relative permeability model to those of Stone and is selected by specifying the `IKU3P` keyword in the PROPS section. This model applies to all three phases symmetrically and data are entered for water and gas with respect to the other two phases using the `SWF3` and `SGF3` keywords. Normalized saturations S_{in} are used to look up, in normalized tables, the pairs of two-phase relative permeabilities $k_{rij}(S_{in})$ and $k_{rik}(S_{in})$ where $i, j, k = g, o, w$ subject to the constraint that $i \neq j \neq k$. These are then combined to calculate the three-phase relative permeability k_{ri} . The normalized saturation S_{in} is given by:

$$S_{in} = \frac{S_i - S_{imn}}{S_{imx} - S_{imn}} \quad \text{Eq. 59.21}$$

where

S_i is the grid block saturation for phase i

S_{imn} and S_{imx} is given by:

$$S_{imn} = \frac{S_j S_{irj} + S_k S_{irk} + S_{irj} S_{irk} (S_i - 1)}{S_j (1 - S_{irk}) + S_k (1 - S_{irj})} \quad \text{Eq. 59.22}$$

$$S_{imx} = \frac{S_j S_{kri} + S_k S_{jri} + S_{jri} S_{kri} (S_i - 1)}{S_j S_{kri} + S_k S_{jri}} \quad \text{Eq. 59.23}$$

where

S_j and S_k are the grid block saturations for phases j and k respectively and

S_{irj} , S_{irk} , S_{jri} , S_{kri} , S_{krj} are the residual saturations of phases i to j , i to k , j to i , k to j and j to k respectively and S_{jrk}

The normalized table look-up for $k_{rij}(S_{in})$ is accomplished by normalizing the tabulated data for $k_{rij}(S_i)$ between the lower limit S_{irj} and the upper limit $1-S_{jri}$ such that $S_i = S_{irj}$ and $S_i = 1-S_{jri}$ correspond to 0 and 1 respectively. The value of $k_{rij}(S_{in})$ is then determined for the supplied value of S_{in} . A similar process is employed to calculate $k_{rik}(S_{in})$, $k_{rji}(S_{jn})$, $k_{rjk}(S_{jn})$, $k_{rki}(S_{kn})$ and $k_{rkj}(S_{kn})$.

Each pair of two-phase relative permeabilities is then combined to calculate the three-phase relative permeability:

$$k_{ri} = \frac{S_j}{S_j + S_k} \cdot k_{rij} + \frac{S_k}{S_k + S_j} \cdot k_{rik} \quad \text{Eq. 59.24}$$

and similarly for k_{rj} and k_{rk} . This formulation is symmetrical and is applied to all three phases.

This calculation process may be illustrated for the case of relative permeability to oil k_{ro} with the aid of a ternary diagram (as illustrated in the following diagram):

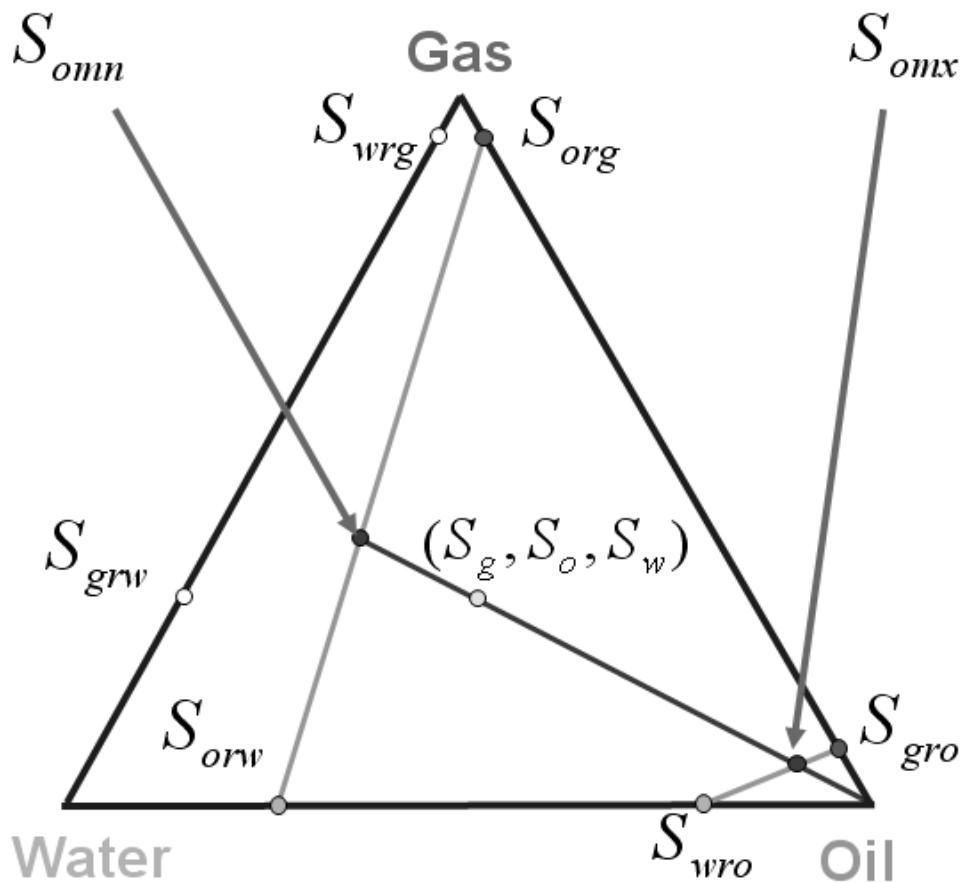


Figure 59.5. Ternary diagram showing the linear interpolation used to calculate Somn and Somx

It is assumed that a three-phase saturation condition exists as depicted by the point denoted by (S_g, S_o, S_w) in the center of the ternary diagram. A line is drawn through this point and the oil apex. This line meets the line constructed by joining the grid block points S_{orw} and S_{org} at the minimum saturation to oil flow S_{omn} and the line constructed by joining the grid block points S_{wro} and S_{gro} at the maximum saturation to oil flow S_{omx} . These oil saturation extrema are calculated according to:

$$S_{omn} = \frac{S_w S_{orw} + S_g S_{org} + S_{orw} S_{org} (S_o - 1)}{S_w (1 - S_{org}) + S_g (1 - S_{orw})} \quad \text{Eq. 59.25}$$

$$S_{omx} = \frac{S_w S_{gro} + S_g S_{wro} + S_{wro} S_{gro} (S_o - 1)}{S_w S_{gro} + S_g S_{wro}} \quad \text{Eq. 59.26}$$

using values pertaining to the grid block, and the normalized oil saturation S_{on} for the grid block is calculated according to:

$$S_{on} = \frac{S_o - S_{omn}}{S_{omx} - S_{omn}} \quad \text{Eq. 59.27}$$

The two-phase relative permeabilities $k_{row}(S_{on})$ and $k_{rog}(S_{on})$ are determined from table look-up where $k_{row}(S_o)$ has been normalized to unity extent between the table values for S_{orw} and $1 - S_{wro}$ and $k_{rog}(S_o)$ has been normalized to unity extent between the table values for S_{org} and $1 - S_{gro}$. Finally k_{ro} is calculated from:

$$k_{ro} = \frac{S_w}{S_w + S_g} \cdot k_{row} + \frac{S_g}{S_g + S_w} \cdot k_{rog} \quad \text{Eq. 59.28}$$

k_{rg} and k_{rw} are calculated in a similar fashion.

ODD3P method

ECLIPSE 300

The ODD3P method provides a coupled hysteretic model for three-phase relative permeabilities and capillary pressures. This method is due to Hustad [Ref. 56] and extends the IKU method [Ref. 60] described above. A summary of the ODD3P method is presented below. A full description of this method may be found in [Ref. 56]. It is recommended that this reference is read as a precursor to using this model.

Whereas the Stone-type models are based upon water-wet systems, the ODD3P method is more general with respect to wettability in that it uses pairs of two-phase relative permeability data for each of the three phases, with the wettability property being implied via the supplied input data.

The primary motivation for the development of the ODD3P method has been the difficulty in reproducing measured core flooding behavior using reservoir simulators with the traditional models of three-phase relative permeability [Ref. 56]. The ODD3P method also ensures consistency (continuous functions) of the three-phase properties at the two-phase and single-phase boundaries with the respective two-phase properties. The saturation profiles observed in core flooding simulations have been found to be strongly dependent upon the capillary forces and the boundary conditions at the core inlet and outlet. In addition, the relative permeability has been found to depend not only upon fluid saturation but also upon saturation history. The requirements for the ODD3P method, therefore, include relative permeability and capillary pressure values which are saturation history dependent. Hence the properties assigned to the relative permeability and capillary pressure for a particular set of three-phase saturations should not be unique but instead depend upon the saturation history leading up to the current saturation condition.

Problems have also been found to occur when modelling three-phase systems if a first-contact miscible injection process causes an oil-water system to become a gas-water system due to the labelling of the

hydrocarbon phase. This can result in discontinuities in relative permeabilities and capillary pressures. A further motivation for the ODD3P method, therefore, has been to ensure that relative permeability and capillary pressure are consistent and continuous for the flow processes which transition from any two-phase state to another two-phase state via a three-phase state.

In this model an important distinction is made between the end-point saturations and residual saturations [Ref. 56]. The end-point saturation is defined as the lowest value of saturation for which the phase is continuous and also corresponds to the saturation where the relative permeability is zero. In contrast, the residual saturation is defined as the saturation at which the phase becomes immobile. This condition may arise for different values of capillary pressure according to the pressure gradient of the phase or how high or low a capillary pressure is achieved by the flow conditions. The relative permeability need not be zero at the residual saturation, but for a phase not to flow, the phase pressure gradients must be zero. On the basis of these definitions, therefore, the residual saturation is greater than or equal to the end-point saturation.

End-point and residual saturation data will typically incorporate information about the processes which have been subjected to upscaling. For this model, the end-point saturation applied to a simulation model grid block may be related to the upscaled processes by including a representation of the sweep efficiency of the grid block using a Land-type formulation [Ref. 66].

Overview

The ODD3P method provides a three-phase relative permeability model which is based upon the saturation functions of the contributing three pairs of two-phase relative permeability saturation functions. This method extends the normalized saturation approach employed by the IKU method [Ref. 60], which is described above. In the ODD3P method, relative permeability and capillary pressure hysteresis effects are incorporated together with grid-block end-point scaling which can optionally include miscible effects.

Instead of the traditional specification of drainage and imbibition saturation functions and the explicit selection of the wetting phase, three sets of saturation functions are employed which are based upon the following ideas:

- Originally, in terms of geological time, formations which currently contain hydrocarbon fluids contained only water. Over a period of millions of years, hydrocarbon fluids gradually penetrated these formations until a point of equilibrium was reached between water, oil and gas. A set of primary saturation functions is used to represent this initial state of the reservoir and the initial production of hydrocarbon fluids. These also define the initial departure of the reservoir from its equilibrium conditions.
- When the formation is exploited commercially, an initial process of sometimes unassisted oil and gas production is usually followed by injection to maintain recovery. Typically the injection process will consist of alternating water and gas (WAG) injection which is one mechanism for improving oil recovery. At some point the saturation behavior will cease to follow the primary saturation functions and enter a hysteresis loop. Two sets of hysteresis saturation functions are used to model the hysteresis behavior experienced by the reservoir when fluid saturations increase and decrease as a function of the cyclical injection processes.

The hysteresis curves are specified in terms of increasing and decreasing saturation directions instead of the more usual notation of drainage and imbibition. For example, the relative permeability to water for the water-oil system is expressed both in terms of the increasing and decreasing water saturation. This corresponds to processes which result in the water saturation increasing and decreasing respectively with respect to the oil saturation. Conversely, the relative permeability to oil for the oil-water system is expressed in terms of increasing and decreasing oil saturation with respect to the water saturation.

The use of primary saturation functions which are distinct from the secondary and tertiary hysteresis saturation functions permits the modeling of the relative permeability behavior where, for example, the decreasing saturation characteristics of relative permeability are different for primary and tertiary

processes. One complexity of this model is establishing the appropriate criteria for the relative permeabilities and capillary pressures to transition from the primary to the hysteresis saturation functions.

Three-phase formulation

The three-phase relative permeabilities and capillary pressures are based upon pairs of representative two-phase properties which are combined through a saturation weighting scheme. Each phase property is made dependent upon its own normalized saturation in order to ensure consistency [Ref. 60]. Zero relative permeabilities are ensured at and below the end-point saturations. The gas-water data is represented by the oil-water data in order to avoid the difficulties associated with representing the hydrocarbon-water system when gas and oil are miscible. This is especially important for first-contact miscible floods in which phase labeling changes between gas and oil require that the functions of relative permeability and capillary pressure remain continuous. This approach also avoids the potential interpolation complexities associated with the use of separate saturation functions tabulated against different sets of saturation values.

Unlike the choice of the different Stone methods for the selection of three-phase relative permeability models and the separate selection of the different methods for modelling hysteresis effects, for example, Jargon or Killough, the ODD3P method incorporates implicitly both three-phase relative permeability and capillary pressure hysteresis. In addition, no explicit specification of the wetting phase is required. Instead, the wetting phase is implicit in the characteristics of the tables of relative permeability and capillary pressure expressed as functions of gas, oil and water saturation for the primary, secondary hysteresis and tertiary hysteresis processes.

Saturation process modes (directions)

The primary, secondary hysteresis and tertiary hysteresis processes refer to the direction of the phases' saturations. The directions of the gas, oil and water saturations are defined with respect to the change in their respective grid-block saturations. The relative permeability and capillary pressure data are based upon three pairs of two phase data such that, for each two-phase pair, $S_i = 1 - S_j$ and $S_k = 0$ where the subscripts i, j and k denote the three phases subject to the constraint that $i \neq j \neq k$. The saturation directions for each phase may be summarized as follows:

Primary

S_w decreasing

S_g increasing

$S_o = 1 - S_g$ or $S_o = 1 - S_w$

Secondary hysteresis

S_w increasing

S_g decreasing

$S_o = 1 - S_g$ increasing

$S_o = 1 - S_w$ decreasing

Tertiary hysteresis

S_w decreasing

S_g increasing

$S_o = 1 - S_g$ decreasing

$$S_o = 1 - S_w \text{ increasing}$$

For a given saturation process mode, the gas saturation direction will be in the opposing direction to that of the water saturation. For the increasing and decreasing hysteresis saturation directions, the oil saturation direction will be in the opposing direction to the water or gas saturation against which it has been measured. For example, for the secondary hysteresis process, gas saturation is decreasing; therefore, if the oil saturation is measured or modeled against gas saturation, the measurements for increasing oil saturation will be used for the secondary hysteresis process.

Availability and limitations

The ODD3P model is currently only available for ECLIPSE 300 compositional models using an equation of state solution method.

ODD3P model definition

The ODD3P three-phase relative permeability and capillary pressure hysteresis model is selected by specifying the [ODD3P](#) keyword in the PROPS section. The use of the default three-phase relative permeability model and the other permeability models [STONE](#) (an alias for [STONE2](#)) and [STONE1](#), [STONE2](#), [IKU3P](#) and ODD3P are mutually exclusive.

ODD3P saturation functions

Saturation function keywords

The relative permeabilities and capillary pressures are expressed as a function of gas, oil and water saturation using the [SGF3](#), [SOF3](#) and [SWF3](#) keywords respectively. In general, three sets of these tables will be required per saturation region corresponding to the primary, secondary hysteresis and tertiary hysteresis saturation processes.

In order to avoid discontinuities due to phase labelling changes in miscible models, the gas-water relative permeability data are replaced by the oil-water relative permeability data. Therefore, the relative permeability to water data for a water-oil system k_{rwo} (column 2 of the [SWF3](#) keyword) are used to represent the relative permeability to water for a gas-water system k_{rgw} (column 3 of the [SWF3](#) keyword). Similarly, the relative permeability to oil data for an oil-water system k_{row} (column 2 of the [SOF3](#) keyword) are used to represent the relative permeability to gas for a gas-water system k_{rgw} (column 3 of the [SGF3](#) keyword). It follows that the data in column 3 of the [SWF3](#) and [SGF3](#) keywords may be defaulted, subject to the caveat that sufficient information is specified in terms of initial values of 0.0 and final values of 1.0 in order to infer the residual and maximum saturations to be applied to the gas-water and water-gas systems. An example of the specification of these three keywords is shown below in abbreviated form.

SGF3			
	Sg	krgo(Sg)	krgw(Sg)
--	0.00000	0.000000E+00	0.000000E+00
	0.00010	0.000000E+00	0.000000E+00
	0.00034	0.141600E-13	1*
	0.00049	0.610900E-13	1*
	...		
	0.83000	0.502886E+00	1*
	0.91000	0.726646E+00	1*
	0.95000	0.865329E+00	0.100000E+01
	0.96000	0.900000E+00	0.100000E+01
/			

SOF3			
	So	krow(So)	krog(So)
--	0.00000	0.000000E+00	0.000000E+00

```

0.04000 0.140000E-02 0.000000E+00
0.09000 0.315000E-02 0.383300E-06
0.17000 0.595000E-02 0.455400E-04
0.20000 0.700000E-02 0.333655E-03
...
0.82000 0.670000E+00 0.360146E+00
0.85000 0.700000E+00 0.433916E+00
0.87000 0.700000E+00 0.483097E+00
0.95000 0.700000E+00 0.765334E+00
0.99767 0.700000E+00 0.987923E+00
0.99922 0.700000E+00 0.995944E+00
0.99951 0.700000E+00 0.997451E+00
0.99966 0.700000E+00 0.998230E+00
0.99990 0.700000E+00 0.100000E+01
/

```

```

SWF3
--      Sw        krwo(Sw)        krwg(Sw)        Pcw0(Sw)
0.05000 0.000000E+00 0.000000E+00 0.986923E+00
0.15000 0.000000E+00 1*           0.986923E+00
0.18000 0.150000E-02 1*           0.493462E-01
...
0.80000 0.500000E+00 1*           0.102640E-01
0.99990 0.999750E+00 0.100000E+01 0.888300E-02
1.00000 0.100000E+01 0.100000E+01 0.888231E-02
/

```

In practice it is envisaged that the relative permeability and capillary pressure data will be obtained either from laboratory measurements of cores or from network simulation models where grain size distributions and pore wettabilities will be representative of the reservoir being modeled. The data are obtained from three discrete sets of two phases measurements or simulations, that is, in the absence of the third phase. In particular it should be noted that, for the [SOF3](#) keyword, the relative permeability to oil for an oil-gas system should be defined in the absence of connate water.

Saturation function region numbers

The association between saturation tables and saturation region numbers follows the usual pattern. However, whereas in the traditional hysteresis models, the drainage and imbibition saturation tables are associated with saturation regions using the keywords [SATNUM](#) and [IMBNUM](#) respectively, for the ODD3P method, the keywords [PSTNUM](#), [ISTNUM](#) and [DSTNUM](#) are used to define the tables associated with the primary, secondary hysteresis and tertiary hysteresis saturations respectively. Internally, the [SATNUM](#) and [IMBNUM](#) keywords are aliases of the [PSTNUM](#) and [ISTNUM](#) keywords respectively and may be used as alternatives. This may be useful when converting existing data sets to work with the ODD3P model. However, there is no equivalent internal alias for the [DSTNUM](#) keyword. It is recommended, therefore, that for the purposes of clarity the matching trio of [PSTNUM](#), [ISTNUM](#) and [DSTNUM](#) keywords should be used wherever possible.

The saturation region keyword [PSTNUM](#) refers to the primary saturation tables, [ISTNUM](#) refers to the tables containing the secondary hysteresis saturation functions, and [DSTNUM](#) refers to the tables containing the tertiary hysteresis saturation functions. The selection of the appropriate combination of saturation functions for a given process direction is made internally according to the current saturations and the history of the saturation directions thus far in the simulation.

The [PSTNUM](#), [ISTNUM](#) and [DSTNUM](#) keywords are optional but are required if hysteresis is to be modeled. These may be used to define the entire set of grid blocks or some subset via the usual set of array assignment and manipulation operators. For grid blocks for which one or more of these keywords have not been specified, the following defaulting process will be applied:

- [PSTNUM](#): Primary saturation function region number defaults to 1.
- [ISTNUM](#): Secondary hysteresis saturation function region number defaults to [PSTNUM](#).

- DSTNUM: Tertiary hysteresis saturation function region number defaults to ISTNUM.

Although the ODD3P method is implicitly hysteretic, hysteresis may be switched off for a given grid block by defining the values of ISTNUM and DSTNUM to be equal. In addition, the modeling of separate primary, secondary and tertiary processes may be switched off by specifying the values of ISTNUM and DSTNUM not only to be equal to each other but also equal to the value of PSTNUM. The only constraint upon the values of PSTNUM, ISTNUM and DSTNUM for a given grid block, therefore, is that if PSTNUM is equal to ISTNUM or DSTNUM, then ISTNUM and DSTNUM must also be equal. Failure to comply with this constraint will cause an error to be reported during the checking of the input data.

Saturation function data constraints

Because of the complexity of the ODD3P model, there is a number of specific input data constraints associated with the specification of the saturation function data for this model. These are summarized below.

- For a specified pair of hysteresis curves, the hysteresis value of the end-point saturation S_{irj} is taken as the lower value of the two S_{irj} values for the secondary and tertiary hysteresis curves, where S_{irj} is defined to be the maximum tabulated saturation value in S_i for which k_{rij} is zero, with S_i measured in the increasing sense. In cases where the secondary and tertiary hysteresis values of S_{irj} are found to be different, which is permissible, a warning is issued. This is to identify possible mismatches between the pairs of curves specified for secondary and tertiary hysteresis operations.
- Each pair of hysteresis curves is required to form a closed hysteresis loop in the region enclosed between their bounding residual saturations. This may be expressed as follows:
 - For the increasing and decreasing relative permeability functions $k_{rij}S_i \text{ inc}$ and $k_{rij}S_i \text{ dec}$, the values should be equal at S_{irj} and $1-S_{jri}$.
 - Similarly for the increasing and decreasing relative permeability functions $k_{rji}S_j \text{ inc}$ and $k_{rji}S_j \text{ dec}$, the values should be equal at S_{jri} and $1-S_{irj}$.
 - Corresponding constraints apply to the increasing and decreasing hysteresis oil-water and gas-oil capillary pressures.

If the above conditions are not satisfied, appropriate warning messages will be issued. Failure to comply with these conditions will lead to inconsistent or unpredictable results.

- In general, it is expected that the relative permeability functions will be monotonically increasing over the range of the closed loop hysteresis curves. Similarly, it is expected that the capillary pressures will vary monotonically over this range. Although these conditions are not mandatory warning messages will be issued if they are not met.
- The first saturation table entries for the secondary and tertiary hysteresis curves are checked for equal initial values of saturation and a warning is issued if this test is not satisfied. This is to identify possible mismatches between pairs of curves specified for secondary and tertiary hysteresis operations.
- For the primary saturation [SGF3](#) tables, the initial gas saturation value should be 0.0. Similarly for the primary [SWF3](#) tables, the final water saturation value should be 1.0. In most cases, for the primary tables, negative values of capillary pressure are not required, and a warning is issued if negative values are encountered.
- In some cases, the primary capillary pressure curve may intersect the secondary hysteresis or tertiary hysteresis capillary pressure curves; such intersections do not prevent correct operation of the model.

For the 2014.1 version, warnings are no longer issued if any such conditions are encountered. However, these warnings may be reinstated by setting item 307 of the OPTIONS3 keyword to 1 in the PROPS or prior sections.

- If the secondary and tertiary hysteresis capillary pressure curves intersect within the hysteresis saturation extent, the hysteresis modelling will fail and this will be reported as an error condition during the input data checking.
- There are no constraints on the intersection of the primary, secondary hysteresis and tertiary hysteresis relative permeability curves.

End-point scaling

For the ODD3P method, when the `ENDSCALE` keyword has been specified in the RUNSPEC Section, end-point scaling is accomplished by applying scaling via grid-block end-point saturations using the PROPS Section keywords `PSORG`, `PSGRO`, `PSORW`, `PSWRO`, `PSGRW` and `PSWRG` for the primary saturation functions and the corresponding keywords `HSORG`, `HSGRO`, `HSORW`, `HSWRO`, `HSGRW` and `HSWRG` for the hysteresis saturation functions. Only one set of hysteresis end-point scaling keywords is required because the hysteresis saturation functions form closed loop curves. For a given grid block, the following constraints apply to the specification of these data values:

Residual Saturation	Primary Constraint	Hysteresis Constraint	Primary-Hysteresis Constraint
SORW		$HSORW > 0.0$	$PSORW \leq HSORW$
SWRO	$PSWRO > 0.0$	$HSWRO > 0.0$	$PSWRO = HSWRO$
SORG	$PSORG > 0.0$	$HSORG > 0.0$	$PSORG = HSORG$
SGRO	$PSGRO \leq 0.001$	$HSGRO > 0.0$	$PSGRO < HSGRO$
SGRW		$HSGRW > 0.0$	$PSGRW < HSGRW$
SWRG	$PSWRG > 0.0$	$HSWRG > 0.0$	$PSWRG = HSWRG$

Table 59.5: End-Point Scaling Constraints

If `ENDSCALE` has not been specified, the grid-block end-point values will be taken from those of the appropriate saturation table. Similarly if `ENDSCALE` has been specified but not all end-point values have been defined for each grid block using the above keywords, the unspecified values will default to those of the appropriate saturation table. The above constraint checking is performed after any defaults have been applied. Failure to comply with the above input data constraints will lead to warning messages.

Capillary pressure calculation control

The calculation of the capillary pressure is controlled by the PROPS section table keywords `PCODD3P`, `PCODD3PG` and `PCODD3PW` which should be specified for each saturation region. In practice, only those tables which are associated with the primary saturation region numbers are used; those associated with the secondary and tertiary hysteresis region numbers may be defaulted. The `PCODD3P` keyword consists of a single row of seven columns of data per saturation region. These are defined as follows:

1. Capillary pressure calculation type. This provides thirteen functional relationships for the calculation of capillary pressure which are numbered 0 to 12.
2. The constant δ_g in the gas saturation dependent capillary pressure weighting function.
3. The first exponent α_g in the gas saturation dependent capillary pressure weighting function.

4. The second exponent β_g in the gas saturation dependent capillary pressure weighting function.
5. The constant δ_w in the water saturation dependent capillary pressure weighting function.
6. The first exponent α_w in the water saturation dependent capillary pressure weighting function.
7. The second exponent β_w in the water saturation dependent capillary pressure weighting function.

The data specified in columns 2 to 7 are only required if the functional relationship type is between 1 and 8. For these types, the capillary pressure weighting functions $H(S_g)$ and $F(S_w)$ are calculated as shown below:

$$H(S_g) = \frac{\delta_g S_g^{\alpha_g}}{S_g^{\alpha_g} + (1-S_g)^{\beta_g}} \quad \text{Eq. 59.29}$$

and

$$F(S_w) = \frac{\delta_w S_w^{\alpha_w}}{S_w^{\alpha_w} + (1-S_w)^{\beta_w}} \quad \text{Eq. 59.30}$$

The **representative** capillary pressures, that is, the values obtained from table look-up using the normalized saturations are corrected according to the following equations:

$$P_{cow} = \hat{P}_{cow} + H \cdot R \quad \text{Eq. 59.31}$$

$$P_{cgo} = \hat{P}_{cgo} + F \cdot R \quad \text{Eq. 59.32}$$

$$R = \hat{P}_{cgw} - \hat{P}_{cgo} - \hat{P}_{cow} \quad \text{Eq. 59.33}$$

where the circumflex denotes the **representative** (IFT scaled) capillary pressures. The corrected capillary pressures may be made functions of one or two pairs of saturations as shown below:

Saturation Dependence	Oil-Water System	Gas-Oil System	Gas-Water System
1	$\hat{P}_{cow}(S_o)$	$\hat{P}_{cgo}(S_g)$	$\hat{P}_{cgw}(S_g)$
2	$\hat{P}_{cow}(S_w)$	$\hat{P}_{cgo}(S_o)$	$\hat{P}_{cgw}(S_w)$

Table 59.6: Saturation functions

The calculation control parameter defined above in column 1 of the `PCODD3P` keyword selects the functional relationship type to calculate the capillary pressures as shown below:

Type	Calculation of oil-water and gas-oil capillary pressures
0	$P_{cow} = 0.0, P_{cgo} = 0.0$
1&2	$P_{cow} = \hat{P}_{cow}(S_o) + H \cdot R, P_{cgo} = \hat{P}_{cgo}(S_g) + F \cdot R$
3&4	$P_{cow} = \hat{P}_{cow}(S_o) + H \cdot R, P_{cgo} = \hat{P}_{cgo}(S_o) + F \cdot R$
5&6	$P_{cow} = \hat{P}_{cow}(S_w) + H \cdot R, P_{cgo} = \hat{P}_{cgo}(S_g) + F \cdot R$
7&8	$P_{cow} = \hat{P}_{cow}(S_w) + H \cdot R, P_{cgo} = \hat{P}_{cgo}(S_o) + F \cdot R$

Type	Calculation of oil-water and gas-oil capillary pressures
9	$P_{cow} = \hat{P}_{cow}(S_o), P_{cgo} = \hat{P}_{cgw}(S_w) - \hat{P}_{cow}(S_o)$
10	$P_{cow} = \frac{S_g}{S_g + S_o} \hat{P}_{cgw}(S_w) + \frac{S_o}{S_g + S_o} \hat{P}_{cow}(S_w), P_{cgo} = \frac{S_w}{S_o + S_w} \hat{P}_{cgw}(S_g) + \frac{S_o}{S_o + S_w} \hat{P}_{cgo}(S_g)$
11	$P_{cow} = \frac{S_g}{S_g + S_o} \hat{P}_{cgw}(S_w) + \frac{S_o}{S_g + S_o} \hat{P}_{cow}(S_w), P_{cgo} = \hat{P}_{cgo}(S_g)$
12	$P_{cow} = \frac{S_g [S_o [\hat{P}_{cgw}(\tilde{S}_w) - \hat{P}_{cgo}(\tilde{S}_{wg})] + S_w [\hat{P}_{cgw}(\tilde{S}_{ww}) - \hat{P}_{cgw}(\tilde{S}_{wg})]] + S_o (S_o + S_w) \hat{P}_{cow}(\tilde{S}_{ww})}{S_o (S_g + S_o + S_w)}$ $P_{cgo} = \left[\frac{S_o}{(S_o + S_w)} + \frac{S_g S_w}{(S_o + S_w)(S_g + S_o + S_w)} \right] \cdot \hat{P}_{cgo}(\tilde{S}_g) +$ $\left[\frac{S_w}{(S_o + S_w)} + \frac{S_g S_w^2}{S_o (S_o + S_w)(S_g + S_o + S_w)} \right] \cdot \hat{P}_{cgw}(\tilde{S}_g) -$ $\left[\frac{S_g S_w}{(S_g + S_o + S_w)} \right] \cdot \hat{P}_{cgw}(\tilde{S}_w) - \left[\frac{S_w}{(S_g + S_o + S_w)} \right] \cdot \hat{P}_{cow}(\tilde{S}_w)$

Table 59.7: Functional relationship types for the calculation of capillary pressures

For functional relationship types 1 to 8, R is calculated as shown below:

Type	Calculation of value R
1	$R = \hat{P}_{cgw}(S_g) - \hat{P}_{cgo}(S_g) - \hat{P}_{cow}(S_o)$
2	$R = \hat{P}_{cgw}(S_w) - \hat{P}_{cgo}(S_g) - \hat{P}_{cow}(S_o)$
3	$R = \hat{P}_{cgw}(S_g) - \hat{P}_{cgo}(S_o) - \hat{P}_{cow}(S_o)$
4	$R = \hat{P}_{cgw}(S_w) - \hat{P}_{cgo}(S_o) - \hat{P}_{cow}(S_o)$
5	$R = \hat{P}_{cgw}(S_g) - \hat{P}_{cgo}(S_g) - \hat{P}_{cow}(S_w)$
6	$R = \hat{P}_{cgw}(S_w) - \hat{P}_{cgo}(S_g) - \hat{P}_{cow}(S_w)$
7	$R = \hat{P}_{cgw}(S_g) - \hat{P}_{cgo}(S_o) - \hat{P}_{cow}(S_w)$
8	$R = \hat{P}_{cgw}(S_w) - \hat{P}_{cgo}(S_o) - \hat{P}_{cow}(S_w)$

Table 59.8: Functional relationship types and the calculation of R

The functional relationship types 9, 10 and 11 correspond to cases where R is equal to 0.0.

Capillary pressure functional dependence calculation control parameter options 9, 10 and 11 correspond to cases where R is equal to 0.0. For options 10, 11 and 12 the saturation weightings are obtained from the grid block saturations S_i . Option 12 incorporates saturation weighting based upon all three grid block saturations (S-Weighted Model). This model has been developed by Statoil ASA see [Ref. 104] and results obtained from this model are described in [Ref. 57] and compared with alternative models above.

For ultimate flexibility the capillary pressure weighting functions $H(S_g)$ and $F(S_w)$ may be defined explicitly using the keywords `PCODD3PG` and `PCODD3PW` which tabulate these functions against gas and water saturation respectively. Either of these keywords may optionally be specified. If the `PCODD3PG` keyword is specified, the values defined in columns 2, 3 and 4 of the `PCODD3P` keyword will be ignored.

Similarly, if the PCODD3PW keyword is specified, the values defined in columns 5, 6 and 7 of the PCODD3P keyword will be ignored.

End-point scaling calculation control

The end-point scaling calculation is controlled by the PROPS section table keyword EPSODD3P. This keyword provides control over the application of a sweep efficiency component to the process-dependent end-point saturations. The following Land formulation [Ref. 56] is used for this purpose and calculates a modified grid-block end-point saturation S_{irj} according to the expression:

$$S_{irj} = \frac{S_i^{max}}{1 + \frac{S_i^{max}}{1 + S_{irj}^p} - \frac{S_j^{max}}{1 - S_{jri}^p}} \quad \text{Eq. 59.34}$$

where

S_i^{max} is the maximum grid block saturation and

S_{irj}^p and S_{jri}^p are the process-dependent grid-block end-point saturations and

$i \neq j$, and S_i^{max} are subject to the constraint $S_i^{max} \leq 1 - S_{jri}^p$

The Land formulation may be applied individually to any or all of the oil, gas and water saturations.

Saturation direction indicators

During the calculation process, six saturation process direction indicators are employed to monitor the switching of the process directions from primary to hysteresis and between increasing and decreasing saturations. These six saturation indicators are specified per grid block and may possess one of four values:

1. Using primary saturation functions with switching to hysteresis not permitted.
2. Using primary saturation functions with switching to hysteresis permitted.
3. Using decreasing saturations.
4. Using increasing saturations.

Internally, these saturation direction indicators are assigned an initial value of 1. When, during the course of the simulation, the grid-block saturations come within the hysteresis extent as defined by the hysteresis grid-block end-points, the indicators are assigned a value of 2. Subsequently when a saturation process direction change occurs these indicators are assigned a value of 3 or 4 corresponding to whether the saturation direction is increasing or decreasing respectively. Thereafter, the saturation direction indicators will switch between the values of 3 and 4 in response to changes in the phase's saturation direction.

In some special cases, it may be helpful to initialize these saturation direction indicators to values other than 1, for example, when the results of some experiment or some other simulation provide details for the initial saturation states. Under these circumstances, it is possible to initialize these saturation direction pointers to values in the range of 1 to 4 on a per grid-block basis using the REGION Section keywords **SDROW**, **SDRWO**, **SDROG**, **SDRG0**, **SDRGW** and **SDRWG**. When only partial array specifications are provided for these arrays, the unspecified portions of the arrays will be initialized to 1. Considerable care should be exercised when using this facility. In general, the initial is at ion of these arrays to non-default values will not be necessary and these arrays need not be specified in the input data.

Reporting facilities

In view of the complexity of the ODD3P method, a considerable number of array reporting facilities are provided both as solution arrays and for output to the Summary file on a per grid-block basis. The arrays which are provided for output are listed below with the characters i and j signifying gas G , oil, O and water W , noting that $i \neq j$. The following grid-block arrays are considered to be useful for the general monitoring of the calculation process.

- $KR_{i,j,R}$: Representative relative permeabilities.
- $SDR_{i,j}$: Saturation process direction indicators.
- $S_{i,NRM}$: Normalized saturations.
- $S_{i,R,B}$: Residual saturations.

When specifying the required outputs in the input data file, the characters i and j should be replaced with the specific phase pairs which are required. For output on a per grid-block basis as Summary vectors, the array names above should be pre-appended with a B and defined in the SUMMARY section for the selected grid-block indices.

Table end points

In the equilibration calculation (see keyword [EQUIL](#)) the water and gas saturation values above and below the transition zones are determined from the maximum and minimum saturation values in the water and gas saturation tables. The oil saturation values are thus defined consequentially from these values. The initial saturations in each zone are shown in the following figure.

	$S_g = S_{g,max}$
GAS ZONE	$S_w = S_{w,co}$
	$S_o = 1 - S_{g,max} - S_{w,co}$
<hr/>	
GAS - OIL TRANSITION ZONE	
	$S_g = S_{g,co}$
OIL ZONE	$S_w = S_{w,co}$
	$S_o = 1 - S_{g,co} - S_{w,co}$
<hr/>	
OIL-WATER TRANSITION ZONE	
	$S_g = S_{g,co}$
WATER ZONE	$S_w = S_{w,max}$
	$S_o = 1 - S_{g,co} - S_{w,max}$
<hr/>	

Figure 59.6. Initial saturations for each zone

In the case where the two-dimensional alternatives of the saturation functions are used ([SWF32D](#), [SGF32D](#), [PCW32D](#), [PCG32D](#)) the equilibration process is done as follows, based on the depth of the cell:

GAS	$S_w = S_{wco}$ $S_o = 0 \text{ or from } PCG32D(TZ)$ $S_g = 1 - S_o - S_{wco}$
OIL	$S_g = 0$ $S_o = 1 - S_{wco} \text{ or from } PCW32D(TZ)$ $S_w = 1 - S_o$
WATER	$S_g = 0$ $S_o = 0 \text{ or from } PCW32D(TZ)$ $S_w = 1 - S_o$

Figure 59.7. Equilibration process based on the depth of the cell

Note: This equilibration process does not handle the situation where the water zone overlaps with the gas zone.

Consistency requirements

The oil, water and gas saturation tables, for each particular saturation table number region, must obey certain consistency requirements. These are detailed below.

- $S_g \text{ max}$ must not exceed $1 - S_{wco}$

If this condition is violated, the gas saturation in the gas cap will be reset to $1 - S_{wco}$, to prevent a negative oil saturation.

Normally, if there is no oil in the gas cap, $S_g \text{ max} = 1 - S_{wco}$.

- S_{gco} must not exceed $1 - S_w \text{ max}$

If this condition is violated, the gas saturation in the water zone will be re-set to $1 - S_w \text{ max}$, 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 $1 - S_w \text{ max} = 0$.

- $S_o \text{ max}$ must equal $1 - S_{wco} - S_{gco}$
- $k_{row}(S_o \text{ max})$ must equal $k_{rog}(S_o \text{ max})$
- $k_{rw}(S_w = 0) = k_{rg}(S_g = 0) = k_{row}(S_o = 0) = k_{rog}(S_o = 0) = 0$.

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:

- $S_g \text{ max}$ must equal $1 - S_{oco}$.

- $S_o \text{ max}$ is normally equal to 1.0.
- $S_w \text{ max}$ must equal $1-S_{oco}$.

Mobile fluid requirements

ECLIPSE 100

In order to ensure that at least one fluid remains mobile in the three-phase ternary diagram, ECLIPSE 100 also checks for the following:

- $S_{owcr} + S_{wcr} < 1$
- $S_{ogcr} + S_{gcr} + S_{wco} < 1$

ECLIPSE issues a warning if these conditions are not satisfied. When **SCALECRS** is used, this would lead to an error because in an oil-water run, for example, K_{rw} would be scaled between S_{wcr} and $1-S_{owcr}$ that is a null range if the first condition is not satisfied.

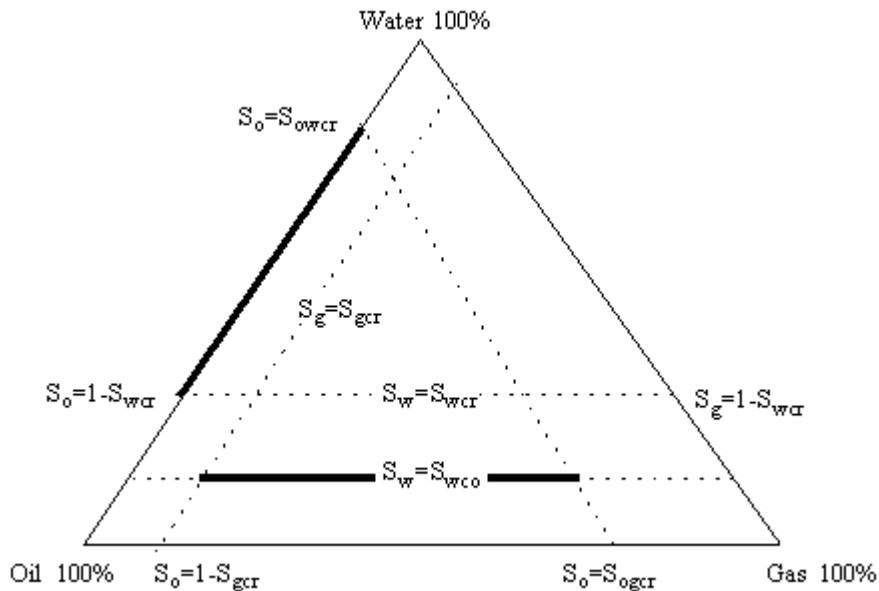


Figure 59.8. Ternary diagram showing mobile fluid end-points

Near critical oil and gas relative permeabilities

ECLIPSE 300

In compositional mode it is possible for a cell to make a direct transition from an oil to a gas, without passing through a series of intermediate saturations. This could result in a jump in the relative permeability. To prevent this, the oil and gas relative permeabilities are interpolated, as described below. This interpolation is not used or required when **ISGAS** is specified. It will be disabled if **NOMIX** is used.

Consider a system with mobile hydrocarbon, initially regarded as an oil, but under gas injection. The effect of gas injection is to move the phase envelope to the left, and the system critical temperature can pass through the reservoir temperature.

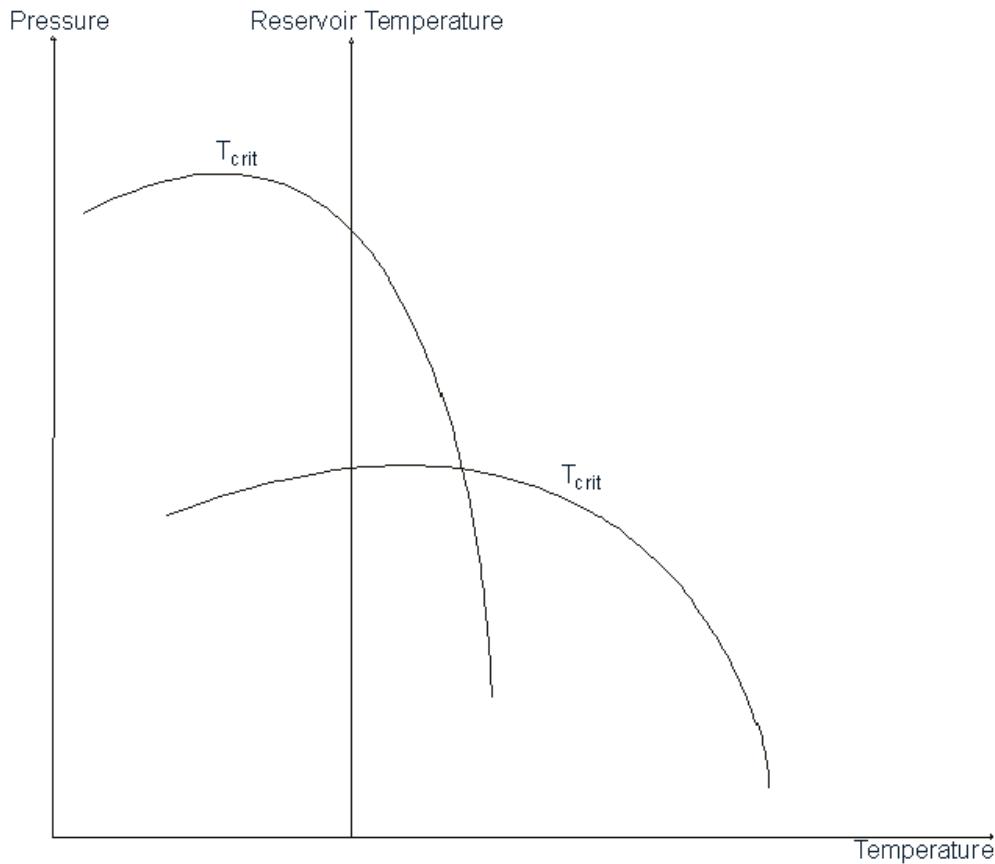


Figure 59.9. The effect of gas injection

There must be continuity between the hydrocarbon relative permeability, K_{ro} when the system is an oil and K_{rg} when the system is a gas, for arbitrary S_w .

To do this we define a water-hydrocarbon relative permeability for single phase hydrocarbon states. Oil-gas and gas-oil relative permeabilities are only relevant if both oil and gas are present. This water-hydrocarbon K_{rh} must tend to the user entered K_{ro} when the composition is oil-like, and to K_{rg} when it is gas-like.

To set up a variable to interpolate between these two values, we define the pseudo-critical temperature T^{crit} which is given by the Li correlation as:

$$T^{\text{crit}} = \frac{\sum_j T_{cj} \cdot V_{cj} \cdot z_j}{\sum_j V_{cj} z_j} \quad \text{Eq. 59.35}$$

Then define f as:

$$f = \frac{T^{\text{crit}}}{T^{\text{res}}} \quad \text{Eq. 59.36}$$

f will clearly be 1 if the reservoir temperature T^{res} is the critical temperature, but becomes greater than 1 for an oil ($T^{\text{crit}} > T^{\text{res}}$) and less than 1 for a gas. If we choose values at which the system is to be regarded completely as an oil or a gas, such as $f_o = 1.25$ and $f_g = 0.75$ (these values may be user defined by the **LILIM** keyword), then an interpolating function may be defined as

$$E = \frac{f-f_g}{f_o-f_g} \quad \text{Eq. 59.37}$$

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

$$K_{rhw} = EK_{row}^u + (1-E)K_{rgw}^u \quad \text{Eq. 59.38}$$

where K_{row}^u and K_{rgw}^u 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_{row}^u and K_{rgw}^u 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:

$$K_{ro} = gK_{rhw}(S_w) + (1-g)K_{rog}(S_o) \quad \text{Eq. 59.39}$$

$$g = \frac{S_w - S_{wc}}{S_w - S_{wc} + S_g} \quad \text{Eq. 59.40}$$

$$K_{rg} = hK_{rhw}(S_w) + (1-h)K_{rgo}(S_g) \quad \text{Eq. 59.41}$$

$$h = \frac{S_w - S_{wc}}{S_w - S_{wc} + S_o} \quad \text{Eq. 59.42}$$

At connate water $S_w = S_{wc}$, the interpolation parameters defined by equations 59.40 and 59.41 are zero, $g = h = 0$, and the oil and gas relative permeabilities become

$$K_{ro} = K_{rog}(S_o) \quad \text{Eq. 59.43}$$

$$K_{rg} = K_{rgo}(S_g) \quad \text{Eq. 59.44}$$

Therefore, to ensure there is no discontinuity when oil becomes gas, or vice versa, it is necessary for the oil and gas relative permeabilities to be the same at the connate water saturation. This can be accomplished using similar interpolation to that used in equation 59.38:

$$K_{rog} = EK_{rog}^u + (1-E)K_{rgo}^u \quad \text{Eq. 59.45}$$

$$K_{rgo} = EK_{rog}^u + (1-E)K_{rgo}^u \quad \text{Eq. 59.46}$$

where K_{rog}^u and K_{rgo}^u are the user-input oil and gas relative permeabilities.

This second scaling of the relative permeabilities at the connate water is controlled by the [FORMOPTS](#) keyword options `HCSCAL` and `NOHCSCAL`.

These equations can be simplified for the gas relative permeability, since a single user-defined table is defined for both gas-in-oil and gas-in-water $K_{rgo}^u = K_{rgw}^u = K_{rg}^u$. Therefore equations 59.41, 59.38 and 59.46 can be written as

$$K_{rg} = E(hK_{row}^u + (1-h)K_{rog}^u) + (1-E)K_{rg}^u \quad \text{Eq. 59.47}$$

60

Saturation table scaling

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The ECLIPSE saturation table end-point scaling option provides a mechanism for redefining 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. Rescaling is applied separately to the relative permeability and capillary pressure curves. In general, rescaling consists of a **two-point** transformation, where two saturation values in the tables are 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 usually eight, and in some cases ten, saturation table end-points which may be identified:

Keyword	Description
SWL	The connate water saturation. This is the smallest water saturation entry in a water saturation table.
SWCR	The critical water saturation. This is the highest water saturation for which the water is immobile.
SWU	The maximum water saturation. This is the largest water saturation entry in a water saturation table.
SGL	The connate gas saturation. In ECLIPSE 100 this is the smallest gas saturation entry in a gas saturation table. In ECLIPSE 300 this is set to zero by default irrespective of the values specified in the saturation functions; if item 103 of the OPTIONS3 keyword is set to a non-zero value, non-zero connate gas saturations are permitted.
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.

Keyword	Description
SGWCR	The scaled critical gas-in-water saturation. This is the highest gas saturation for which the gas is immobile in a gas-water system.
SWGCR	The scaled critical water-in-gas saturation. This is the highest water saturation for which the water is immobile in a water-gas system.

Table 60.1: Saturation table end points

ECLIPSE 300

Note: The SGWCR and the SWGCR saturation table end-points are only available in ECLIPSE 300. They are only used by the [IKU3P](#) three-phase relative permeability model and by the [BAKER1](#) and the [BAKER2](#) three-phase relative permeability models where the Baker gas model (SGWCR) or water model (SWGCR) has been specified.

The saturation end-points taken from the saturation tables in the input data file are referred to as the unscaled saturation end points.

The end-point scaling option enables new values to be defined for any of the above 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 set of end-points is applicable in two phase runs. The new values defined using the end-point scaling option are referred to as the scaled end-points. When relative permeability and capillary pressure values need to be computed at a particular saturation, a linear transformation is used to determine the equivalent saturation to be used for look-up in the unscaled table.

For example, in a grid cell whose water saturation is SW , scaled connate water saturation is $SWCO$, scaled maximum water saturation is SWU , which belongs to a saturation region whose unscaled connate water is S_{wco} and whose unscaled maximum water saturation is S_{wmax} , the water capillary pressure P_{cow} is evaluated in the saturation table at the saturation, S_w' , where

$$S_w' = S_{wco} + \frac{(SW - SWCO)(S_{wmax} - S_{wco})}{SWU - SWCO}$$

so that P_{cow} is evaluated by lookup in the input capillary pressure table using

$$P_{cow}(SW) = P_{cow}(S_w')(table)$$

In addition it is possible to scale the relative permeability and capillary pressure values using the keywords [KRW](#) and [PCW](#) for example.

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](#)

Note: If one or more end-point scaling arrays are not specified, the values are inferred from their associated table values. For [SGU](#) this can lead to differences between ECLIPSE 100 and ECLIPSE 300 in cases where the maximum gas saturation table entry is less than $(1 - S_{wco} - S_{ogcr})$. This is because ECLIPSE 100 uses the maximum gas saturation table entry value whereas ECLIPSE 300 calculates the maximum gas saturation according to $(1 - S_{wco} - S_{ogcr})$.

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 of the **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:

$$P_c = P_{ct} \frac{PCW}{P_{cm}} \quad \text{Eq. 60.1}$$

where

P_{ct} capillary pressure from the table

P_{cm} maximum P_c in the table - at $S_w = S_{wco}$, connate water (this is the entry capillary pressure in the table)

PCW maximum P_c from the PCW data.

Scaling of relative permeability functions

There are two options available for the scaling of relative permeabilities. In the default case, the scaling process preserves relative permeabilities at two saturation nodes. The following end points are assumed for each phase relative permeability:

K_{rw} SWCR and SWU

K_{rg} SGCR and SGU

K_{row} SOWCR and (1.0-SWL-SGL)

K_{rog} SOGCR and (1.0-SWL-SGL)

An alternative scaling of relative permeabilities can be invoked by using the **SCALECRS** keyword in the PROPS section. The second form of scaling preserves the relative permeabilities at three saturation nodes. In three-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 two-phase mobile region. Convergence problems can arise in certain cases when using this form of scaling if the intermediate end point begins to approach the maximum saturation for one of the phases. For example, in an oil-water problem in which the residual oil saturation to water tends to zero at a certain depth, the saturation (1.0-SOWCR) begins to approach SWU and this can in turn lead to a discontinuity in the water relative permeability as $S_w \rightarrow S_{w \max}$.

ECLIPSE 100

You may limit this sharpness by using the `SCALELIM` keyword which sets an upper limit on the value of (1.0 - SOWCR) to be used in scaling the water relative permeability function. The `SCALELIM` keyword is optional and at present can only be applied to water relative permeabilities.

For example, we consider the scaling of the water relative permeability function $K_{rw}(SW)$ at a water saturation SW where the scaled and unscaled critical water saturations are $SWCR$ and S_{wcr} , scaled and unscaled oil-in-water critical saturations are $SOWCR$, and S_{owcr} , scaled and unscaled connate gas saturations are SGL and S_{gco} , and scaled and unscaled maximum water saturations are SWU and S_{wmax} .

With the two point scaling method, the water relative permeability $K_{rw}(SW)$ is then evaluated in the input saturation table at the new saturation S_w' , where

$$S_w' = S_{wcr} + \frac{(SW - SWCR)(S_{wmax} - S_{wcr})}{SWU - SWCR} \quad \text{Eq. 60.2}$$

so that K_{rw} is evaluated by lookup in the input table using

$K_{rw}(SW) = K_{rw}(S_w')(table)$ for $SWCR \leq SW \leq SWU$. For $SW \leq SWCR$ then $K_{rw}(SW) = 0$ and for $SW \geq SWU$ then $K_{rw}(SW) = K_{rwmax}(table)$.

With the three point scaling method, if SR is the displacing critical saturation of the associated phase saturation (SR),

$SR = 1-SOWCR-SGL$ 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:

$$SWCR \leq SW \leq SR$$

$$S_w' = S_{wcr} + \frac{(SW - SWCR)(S_r - S_{wcr})}{SR - SWCR} \quad \text{Eq. 60.3}$$

$$SR \leq SW \leq SWU$$

$$S_w' = S_r + \frac{(SW - SR)(S_{wmax} - S_r)}{SWU - SR} \quad \text{Eq. 60.4}$$

and also for

$$SW \leq SWCR \quad \text{Eq. 60.5}$$

then

$$K_{rw}(SW) = 0 \quad \text{Eq. 60.6}$$

, and for

$$SW \geq SWU \quad \text{Eq. 60.7}$$

then

$$K_{rw}(SW) = K_{rwmax}(\text{table}) \quad \text{Eq. 60.8}$$

The two-point 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 [KWRW](#), [KGRG](#), [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 [KWRW](#) 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:

$$K_{rw} = K_{rw}(\text{table}) \left(\frac{\text{KRW (grid block)}}{K_{rw \max}(\text{table})} \right) \quad \text{Eq. 60.9}$$

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.

$SR = 1\text{-SOWCR-SGL}$ in water/oil or gas/oil/water runs

$SR = 1\text{-SGCR}$ in gas/water runs

Hence the two cases are:

$$SWCR \leq SW \leq SR$$

$$K_{rw}(S) = K_{rw}(S') \frac{KRWR(\text{grid block})}{K_{rw}(S_r)(\text{table})} \quad \text{Eq. 60.10}$$

$$SR \leq SW \leq SWU$$

$$K_{rw}(S) = KRWR + \frac{(K_{rw}(S')(\text{table}) - K_{rw}(S_r)(\text{table}))}{(K_{rw\max}(\text{table}) - K_{rw}(S_r)(\text{table}))} (KRW - KRWR) \quad \text{Eq. 60.11}$$

If the value of $K_{rw\max} = K_{rw}(SR)$, a linear function is assumed between KRWR and KRW.

The two-point scaling method is illustrated in the following figure:

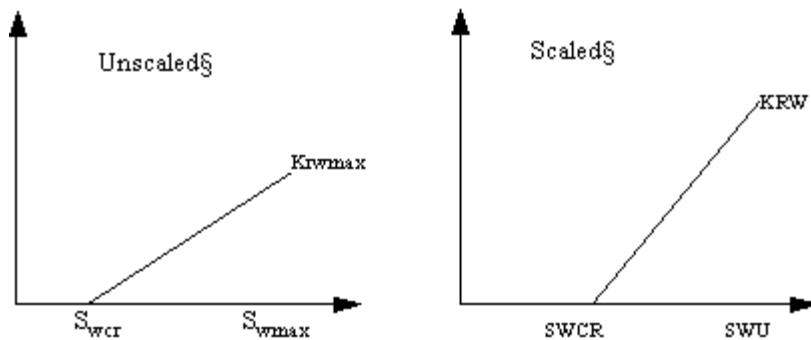


Figure 60.1. Two-point scaling

and the three-point scaling method is illustrated in the following figure:

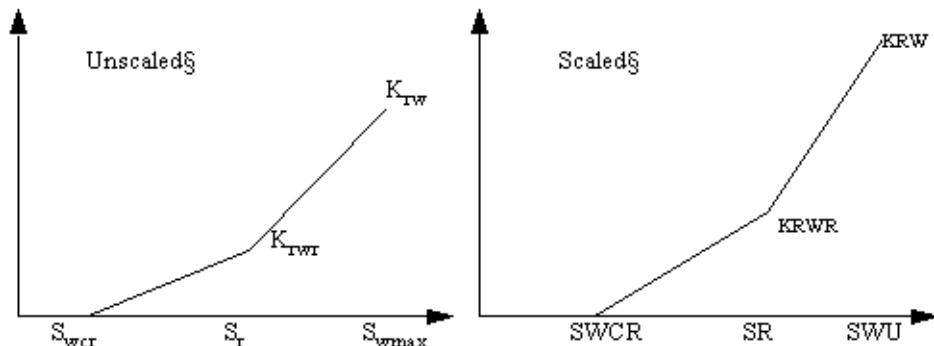


Figure 60.2. Three-point scaling

The [KRWR](#), [KGRR](#), [KGRG](#), [KRORW](#), [KRORG](#) keywords and their directional and hysteretic specializations can be used to set the relative permeabilities at the critical/residual saturation for both two-point scaling ([SCALECRS](#) keyword item 1 set to NO, which is the default if this keyword is not specified) and for the alternative three-point scaling ([SCALECRS](#) keyword item 1 set to YES). However, the impact of these keywords upon the resultant scaling will typically be different for the two-point and three-point scaling methods.

For the two-point scaling of water relative permeability, the value specified by KRWR will be preserved at the table saturation $S_w = 1 - S_{sowcr} - S_{geo}$ but noting that this value will be modified by the two-point horizontal scaling applied to S_w if $S_{wcr} \neq SWCR$ and/or if $S_{wmax} \neq SWU$ because of the resultant two-point transformation in saturation space.

In contrast for the three-point scaling, the value specified by KRWR will be preserved at the transformed saturation $SW = 1 - SOWCR - SGL$. Corresponding scaling differences between two and three-point scaling apply also to KGRR, KRORW and KRORG.

Considerable care should be exercised when using the critical/residual vertical scaling end-points. For example, for two-point scaling, if $S_{wmax} = 1 - S_{sowcr} - S_{geo}$, it will not be possible to preserve both the values specified by KRWR and KRW at the same point in saturation space. Similarly, for three-point scaling, if $SWU = 1 - SOWCR - SGU$ it will not be possible to preserve both values specified by KRWR and KRW at the same point in saturation space.

In such cases, in general, the value specified by KRW will be selected; however, due to small differences in notionally equal floating point values, the value specified by KRWR may be selected instead. It is recommended that the [EPSDEBUG](#) or [EPSDBG5](#) keyword should be used to view the tables following the application of end-point scaling.

ECLIPSE 100

For the two-point scaling method ([SCALECRS](#) keyword item 1 set to NO, which is the default if this keyword is not specified), if the unscaled critical/residual and maximum relative permeabilities are equal, the scaled critical/residual relative permeability will be set equal to the scaled maximum relative permeability irrespective of whether this value has been specified. For example, if in the input unscaled water saturation functions, $k_{rwr} = k_{rwmax}$, where k_{rwr} is the value of $k_{rw}(S_w)$ at $S_w = 1 - S_{sowcr} - S_{geo}$ and k_{rwmax} is the value of $k_{rw}(S_w)$ at $S_w = S_{wmax}$, the value of KRWR will be assumed to be equal to that of KRW.

Miscellaneous points

The end-point scaling option is activated using the keyword [ENDSCALE](#) in the RUNSPEC section. The directional scaling options are activated by specifying the appropriate arguments of ENDSCALE.

The end point saturations may be supplied in one of two ways:

- by cell, using the keywords [SWL](#), [SWCR](#), [SWU](#), [SGL](#), [SGCR](#), [SGU](#), [SOWCR](#), [SOGCR](#); or,
- by depth, using the [ENPTVD](#) keyword.

The cell keywords need not specify data for the whole field - any values that are not set do not cause an error, (as for [PORO](#) or [DX](#)), but imply that they are to be obtained from the default values for the cell, or from ENPTVD, if present.

If SWL, SOGCR and ENPTVD are used, values taken from ENPTVD are used to fill in any data that is not set, as long as the endpoint table number is not 0. End point table numbers are set in the REGIONS section, using [ENDNUM](#), and default to 1, but a value of 0 may be entered.

The overall process is thus:

1. If keywords SWL and SOGCR are present, use the value
2. If not, and ENPTVD is present with non-zero ENDNUM, use this
3. Otherwise, use defaults obtained from tables.

The values of the end-point arrays SWL and SOGCR may be printed out for each cell using [RPTSOL](#) with the appropriate mnemonic, and output for plotting with GRAF using [RPTRST](#).

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 [FILEEPS](#) 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](#)).

Note: When directional scaling is used, the isotropic form must also be supplied if scaling is required in equilibration and well block flows.

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 overridden by any subsequent end points defined using the COMPRP keyword.

ECLIPSE 100

In the present version of ECLIPSE 100 the [SWLPC](#) and [SGLPC](#) values cannot be supplied using the [ENPTVD](#) depth type keywords.

ECLIPSE 100

The end-point scaling option should not be used at present with the vertical equilibrium option.

ECLIPSE 100

The unscaled and scaled maximum oil saturations used for the two-point end-point scaling of the oil relative permeability to water and gas are calculated according to $S_{omax} = 1 - S_{wco}$ and $SOU = 1 - SWL$ respectively. These can be modified to include connate gas; that is, $S_{omax} = 1 - S_{wco} - S_{gco}$ and $SOU = 1 - SWL - SGL$ respectively by setting item 205 of the OPTIONS keyword to 1.

ECLIPSE 300

The saturation table end-points can depend on temperature in a thermal simulation using the keywords [ENKRVVT](#), [ENPCVVT](#) and [ENPTVVT](#).

ECLIPSE 300

The saturation table end-points can depend on composition using the keywords [ENKRVC](#), [ENPCVC](#) and [ENPTVC](#).

Combined grid-block and temperature-dependent table end-points

ECLIPSE 300

For thermal models, it is possible to combine the grid-block end-points, which are temperature independent, with the temperature-dependent table end-points. In general, for the End-point scaling option, the grid-block end-points are obtained from the end-point arrays, for saturation, relative permeability and capillary pressure end-points respectively. The end-point arrays are specified using keywords such as [SWL](#), [SWCR](#), [SWU](#), [KRW](#) and [PCW](#), possibly in combination with the depth-dependent end-point tables [ENPTVD](#), [ENKRVD](#) and [ENPCVD](#). The defaulted values are furnished from the end-points obtained from the saturation tables.

If the [ENDSCALE](#) keyword is specified with item 5 set to a value of 1 or 2, it is possible either to supersede or combine respectively these values with those specified in the temperature-dependent end-point tables [ENPTVVT](#), [ENKRVVT](#) and [ENPCVVT](#) for saturation, relative permeability and capillary pressure end-points respectively.

If item 5 is set to 0 or defaulted, the use of grid block end-points and temperature-dependent table end-points is mutually exclusive. This is the only option available for versions of ECLIPSE 300 prior to the 2011.1 version and is the default option for the 2011.1 version.

If item 5 is set to 1, a temperature dependent end-point, where specified, will supersede the grid block value.

If item 5 is set to 2, a temperature dependent table end-point, where specified, will be combined with the grid block value such that the resultant temperature dependent grid block end-point $Ep_{gb}(T)$ is given by:

$$Ep_{gb}(T) = Ep_t(T) \times \frac{Ep_{gb}}{Ep_t(T_1)} \quad \text{Eq. 60.12}$$

where

Ep is used to signify any of the saturation, relative permeability or capillary pressure end-points

$Ep_t(T)$ is the end-point value obtained from the temperature-dependent table for the grid-block temperature T

Ep_{gb} is the temperature-independent grid-block end-point value

$Ep_t(T_1)$ is the end-point value from the temperature-dependent table at temperature T_1 which corresponds to the first temperature entry in the table.

This form of scaling ensures that the original grid-block end-point is honored at the lowest temperature in the table.

Any end-point in any of the temperature-dependent end-point tables may be defined to be temperature independent by specifying the values in all rows of the appropriate column to be default values (1*). The consequence of this defaulting will depend upon the value of item 5 of the [ENDSCALE](#) keyword. If this value is 0, the combination of grid-block and temperature-dependent table end-points is not permitted; if

only the temperature-dependent tables are specified, the defaults will be supplied from the saturation table end-points. However, if item 5 of the `ENDSCALE` keyword is 1 or 2, the defaults will be supplied from the grid-block end-points, except for the relative permeabilities at critical and residual saturations. If these are defaulted, these will be scaled according to the current values of the maximum relative permeabilities. The value of item 5 is ignored for non-thermal models.

Output of scaled end-points

ECLIPSE 300

When the saturation end-point scaling depends on temperature or composition (`ENPTVT` and `ENPTVC` keywords), the scaled values can be output using the following keywords and mnemonics.

Refer to [Processed scaled end-points](#) in the *ECLIPSE Reference Manual*.

To the PRINT file:

```
RPTSQL
PSWL PSWCR PSWU PSGL PSGCR PSGU PSOWCR PSOGCR /
RPTSCHEDE
PSWL PSWCR PSWU PSGL PSGCR PSGU PSOWCR PSOGCR /
```

To the SUMMARY file

Block keyword	Description
BPSWL	Block scaled minimum water saturation.
BPSWCR	Block scaled critical water saturation.
BPSWU	Block scaled maximum water saturation.
BPSGL	Block scaled minimum gas saturation.
BPSGCR	Block scaled critical gas saturation.
BPSGU	Block scaled maximum gas saturation.
BPSOWCR	Block scaled critical oil-in-water saturation.
BPSOGCR	Block scaled critical oil-in-gas saturation.

To the RESTART file:

```
RPTRST
PSWL PSWCR PSWU PSGL PSGCR PSGU PSOWCR PSOGCR /
```

When the relative permeability end-point scaling depends on temperature or composition (`ENKRVT` and `ENKRCV` keywords), the scaled values can be output using the following keywords and mnemonics.

To the PRINT file:

```
RPTSQL
PKRO PKRG PKRW PKRGR PKRWR PKRORG PKRORW /
RPTSCHEDE
PKRO PKRG PKRW PKRGR PKRWR PKRORG PKRORW /
```

To the SUMMARY file

Block keyword	Description
BPKRG	Block scaled gas relative permeability at maximum gas saturation.

Block keyword	Description
BPKRGR	Block scaled gas relative permeability at residual oil saturation.
BPKRO	Block scaled oil relative permeability at maximum oil saturation.
BPKRORG	Block scaled oil relative permeability at critical gas saturation.
BPKRORW	Block scaled oil relative permeability at critical water saturation.
BPKRW	Block scaled water relative permeability at maximum water saturation.
BPKRWR	Block scaled water relative permeability at residual oil saturation.

To the RESTART file:

```
RPTRST
PKRO PKRG PKRW PKRGR PKRWR PKRORG PKRORW /
```

When the capillary pressure end-point scaling depends on temperature or composition ([ENPCVT](#) and [ENPCVC](#) keywords), the scaled values can be output using the following keywords and mnemonics.

To the PRINT file:

```
RPTSOL
PPCG PPCW /
RPTSCHE
PPCG PPCW /
```

To the SUMMARY file

Block keyword	Description
BPPCG	Block scaled maximum gas-oil capillary pressure.
BPPCW	Block scaled maximum water-oil capillary pressure.

To the RESTART file:

```
RPTRST
PPCG PPCW /
```

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 the following figure.

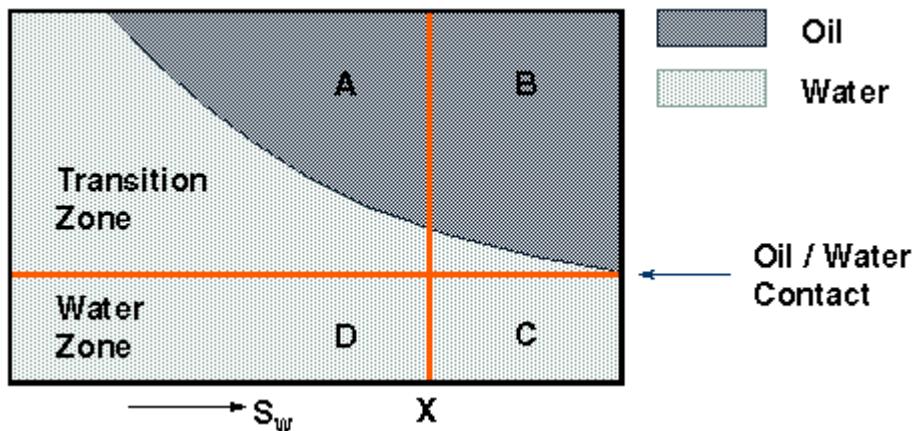


Figure 60.3. Water saturation versus depth for a typical grid cell that intersects the oil-water transition zone

The water saturation at the value X is 1.0-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 ($i=1,\dots,N$) 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

$$M_{pi} = 0 \text{ if } S_{pi} < S_{p,cr,i}$$

$$M_{pi} = V_i(S_{pi} - S_{p,cr,i}) \text{ if } S_{pi} > S_{p,cr,i}$$

where V_i is the pore volume of layer i

- The total mobile phase volume, M_p , in each cell is then determined using $M_p = \sum_i M_{pi}$. The total phase volume, $T_p = \sum_i V_i S_{pi}$.
- The average phase saturation in the cell is $S_p' = T_p / (\sum_i V_i)$ and the average mobile phase saturation is $M_p' = M_p / (\sum_i V_i)$.
- For cells in 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. 60.13

- The remaining cells in the grid contain either mobile or immobile fluid throughout their depth. In this case

$$S'_{p,cr,i} = (\sum_i V_i S_{p,cr,i}) / (\sum_i V_i) \quad \text{Eq. 60.14}$$

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 keywords **SOWCR**, **SOGCR**, **SWCR** and **SGCR** (items 24-27) in the **RPTSOL** keyword (the output is headed by keywords **PSOWCR**, **PSOGCR**, **PSWCR** and **PSGCR**).

End-point variations through a transition zone

In some reservoirs it is necessary to model a depth variation of the critical saturation of one phase such that the critical saturation equals the phase saturation providing the phase saturation is less than a threshold (See figure 60.4). Although the **ENPTVD** keyword could be used to model the profile, a more convenient method, which also gives the correct initial mobile fluids-in-place, is to initially specify a constant critical saturation with depth and set $S'_{p,cr,i} = S'_{p,M'} - M'_{p,i}$ regardless of the phase mobility in each cell. This calculation can be invoked by using the **TZONE** keyword in the **PROPS** section.

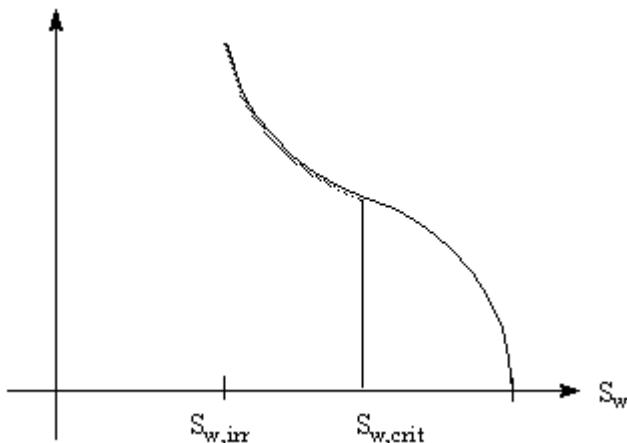


Figure 60.4. Modeling a depth variation of critical saturation

To use the facility and achieve the result shown, specify the keyword **ENDSCALE** in the **RUNSPEC** section, turn on the second data item in the **TZONE** keyword, and set the critical water saturation to $S^*_{w,crit}$. Output of the processed end points used in the simulation can be requested with the mnemonics **SOWCR**, **SOGCR**, **SWCR** and **SGCR** (switches 24-27) in the **RPTSOL** keyword at all depths.

The TZONE keyword allows control of the calculation for each phase separately; if the water switch is set true then the critical water saturation will be corrected to the initial water saturation where this is below the input critical saturation.

Leverett J-function

The Leverett J-function is a dimensionless group that allows the capillary pressure function to be correlated with the rock properties.

$$J(S_w) = \frac{P_c}{\sigma} \left(\frac{K}{\varphi} \right)^{\frac{1}{2}} \quad \text{Eq. 60.15}$$

where

P_c capillary pressure

σ surface tension

K rock permeability

φ rock porosity.

If either the [JFUNC](#) or the [JFUNCR](#) keyword is used in the GRID section then the capillary pressure is calculated as follows:

$$P_c = F \cdot J(S) \quad \text{Eq. 60.16}$$

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.

F Scaling Factor given by:

$$F = \sigma \cdot \left(\frac{\varphi}{K} \right)^{\frac{1}{2}} \cdot U_{\text{const}} \quad \text{Eq. 60.17}$$

where

σ Surface tension - specified in the [JFUNC](#) or the [JFUNCR](#) keyword

φ cell porosity

K cell permeability

U_{const} constant depending on the unit system employed:

= 0.318316 (METRIC units, pressure in bars, perm in mD)

= 4.61678 (FIELD units, pressure in psi, perm in mD)

= 0.314153 (LAB units, pressure in atm, perm in mD)

= 0.314153 (PVT-M units, pressure in atm, perm in mD)

The permeability direction to be used in the calculation may be specified using item 6 of the [JFUNC](#) or the [JFUNCR](#) keywords. If this is defaulted, the permeability will be calculated from $K = (K_x + K_y)/2$, unless the model is a cross section, in which case K will be selected from the appropriate horizontal permeability. For a one-dimensional model, the permeability value K is obtained from the permeability corresponding to the direction of the row or column of cells. In addition, irrespective of the permeability direction specified:

- if the numbers of cells in the x and y directions are both equal to 1, the z-direction permeability values will be used
- if the number of cells in the x direction only is equal to 1, the y-direction permeability values will be used
- if the number of cells in the y direction only is equal to 1 the x-direction permeabilities will be used.

The [JFUNC](#) or the [JFUNCR](#) keyword in the [GRID](#) section causes the scaling factor F to be computed from the grid data and stored in the [PCW](#) or [PCG](#) array. The scaling factor can be output by using the mnemonic [ENDPT](#) in the [RPTPROPS](#) keyword.

ECLIPSE 100

Note that, if the surface tension is required to be a function of pressure, the pressure dependent capillary pressure option ([STOW](#) and [STOG](#) keywords) can be used to modify the surface tension. In this case:

$$P_c = J(S) \cdot \sigma \cdot \frac{\sigma(P)}{\sigma_{\text{ref}}} \cdot \left(\frac{\varphi}{K}\right)^{\frac{1}{2}} \cdot U_{\text{const}} \quad \text{Eq. 60.18}$$

where $\sigma(P)$ and σ_{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:

$$P_{cw}(S_w \text{ init}) = P_{\text{oil}} - P_{\text{wat}} \quad \text{Eq. 60.19}$$

for the appropriate equilibration region, where:

P_{oil} is the oil phase pressure

P_{wat} is the water phase pressure

$S_w \text{ init}$ is the required water saturation from [SWATINIT](#)

If $P_{\text{oil}} - P_{\text{wat}} < 0$, the [PCW](#) value is left unchanged and the equilibrated S_w value will be the maximum water saturation, [SWU](#). The phase pressures P_{wat} and P_{gas} are determined by equilibration using [EQUIL](#) and the supplied PVT data. See [OPTIONS](#) keyword [item 74](#) and [OPTIONS3](#) keyword [item 143](#) for a different scaling cases.

If the water distribution and the input capillary pressure curves are very inconsistent, the scaling required may be extreme. The scaled maximum capillary pressure can be output to the [PRINT](#) file using the mnemonic [SWATINIT](#) (item 30) in the [RPTSOL](#) keyword, and is automatically output to the [INIT](#) file.

These arrays should be checked to ensure that the scaling is physically reasonable. The [PPCWMAX](#) keyword could be used to limit the maximum capillary pressure scaling.

If the [SWATINIT](#) keyword is used with the J-function option ([JFUNC](#) or the [SWATINITJFUNC](#) keyword) then in regions of the field where 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](#)". The same requirements apply to the scaled endpoints.

- SGU must not exceed 1.0 - SWL.

If this condition is violated, the gas saturation in the gas cap is reset to 1.0 - SWL, to prevent a negative oil saturation. Normally, if there is no oil in the gas cap, SGU = 1.0 - SWL.

- SGL must not exceed 1.0 - SWU

If this condition is violated, the gas saturation in the water zone is reset 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 $K_{rw \text{ max}}$ 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

$\text{SR} = 1 - \text{SOWCR}(\text{grid block}) - \text{SGL}(\text{grid block})$ in water/oil or gas/oil/water runs

$\text{SR} = 1 - \text{SGCR}(\text{grid block})$ 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:

- $\text{SOWCR} + \text{SWCR} < 1.0$
- $\text{SOGCR} + \text{SGCR} + \text{SWL} < 1.0$

ECLIPSE issues a warning if these conditions are not satisfied. When [SCALECRS](#) is used, this would lead to an error because in an oil-water run, for example, K_{rw} would be scaled between [SWCR](#) and 1.0 - [SOWCR](#), which is a null range if the first condition is not satisfied.

Note: In some cases when consistency warning messages involving SOWCR and SOGCR are reported by ECLIPSE, these are referred to using the alternative names SOCRW and SOCRG respectively. (It should be noted that these are not accepted as alternative keyword names.)

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 /

61

Shale gas

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

Shale gas reservoirs often have a matrix porosity system where the transient behavior in the matrix becomes important. Some of the gas might be adsorbed on the surface of the shale and some exists as a free gas in the matrix pore structure. In order to model such reservoirs the dual/multi porosity option can be used together with the Coal Bed Methane Model for adsorbed gas on the rock formation. See "["Multi porosity"](#)" and "["Adsorption models in ECLIPSE 300"](#)"

There are two models for flow of adsorbed gas into a simulation cell.

- Instant Sorption Model.
The amount of adsorbed gas is in instant equilibrium with the free gas composition and pressure.
- Time-dependent Sorption Model.

To choose between the Time-dependent or Instant Sorption Models the keyword `CBMOPTS` is used.

Simulating shale gas reservoirs

Grid cells with adsorption

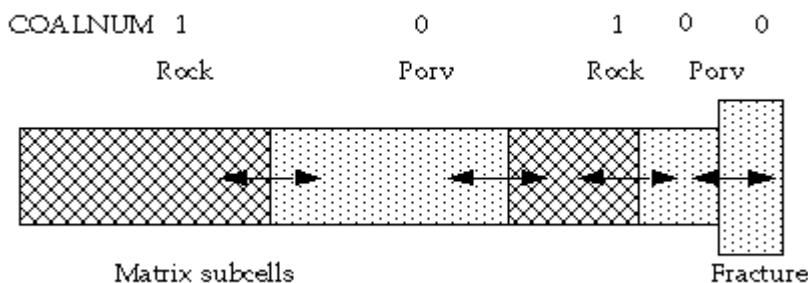
In order to specify cells having an adsorbed gas the `COALNUM` keyword is used for both methods. It is possible to redefine the coal region numbers by the keyword `COALNUMR`.

Time-dependent sorption

For the time-dependent method a simulation cell either contains free gas in a pore space or adsorbed gas in the rock. The rock is represented by one simulation cell and the pore volume by a connecting simulation cell. Darcy flow through a rock cell is not permitted. The usual pore volume of a cell now represents the rock volume where the imaginary micro pore space flow is accounted for by a diffusive flow equation. A cell having a non-zero coal region number as set by `COALNUM` hence needs to specify a "porosity" value that corresponds to a rock fraction value or using the keyword `ROCKFRAC`. For cells having a zero coal region number the porosity value correspond to the pore volume fraction.

Note: It is recommended that the `ROCKFRAC` keyword is used to set the rock volume and not the porosity keyword `PORO`.

If pore volume-pore volume connections exists a permeability value also need to be input in order to compute the transmissibility between the matrix subgrid cells. This is usually done by [PERMX](#).



Matrix discretization (NMATOPTS) and time-dependent sorption

Using the time-dependent method the primary matrix subgrid cells are used to define both the total pore volume and the total coal volume that is partitioned. The primary matrix subgrid cells need to be defined as coal by `COALNUM` in order to tell that the matrix contains adsorbed gas. To redefine the matrix subgrid cells with a coal region number the keyword `COALNUMR` should be used.

With the time-dependent sorption model, the diffusive flow of adsorbed gas can be studied in detail using the matrix discretization option. The coal region numbers need to be the same for all connected matrix sub-grid cells. Mixing pore volume and coal volume with the option is not generally recommended as the interpretation of the resulting response from the matrix, which might include Darcy flow, needs to be carefully considered. Usually in such cases, the basic multi-porosity model should be used where the connection transmissibilities/diffusivities are controlled by user input of the sigma factors. Alternatively, the instant adsorption model can be used if appropriate. Darcy flow then controls the gas flowing towards the fractures within the pore space of the matrix.

The matrix discretization option generates sigma factors for the connections. Thus for pore-volume to pore-volume connections, the permeabilities together with the generated sigma factor are used to generate the transmissibilities in a similar way as for dual porosity. It is possible to specify permeabilities individually for the different pore systems. If unspecified, the input permeability given to the primary matrix cell is used. The transmissibility would then be constructed according to the block volume of the sub-matrix grid cell and the permeability of the sub-matrix grid cell. Note that such a transmissibility must be viewed as an internal connection within the matrix block. The transport will be diffusive through a cell connected with a non-zero coal region number.

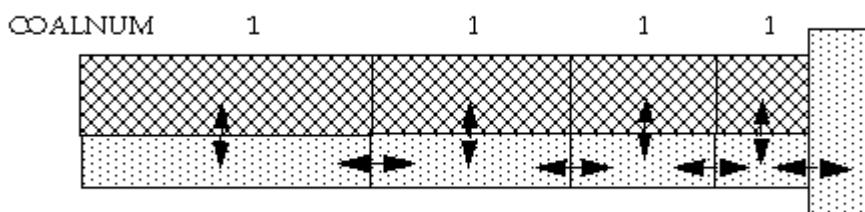
For more information about the diffusive flow see "Coal bed methane model".

The second item of NMATOPTS specifies a fraction of either the fracture pore volume or a fraction of the matrix block volume to be used for the first primary matrix grid cell. The method is selected by the third item of NMATOPTS. With the time-dependent method, it is usually better to specify a fraction of the matrix block volume since the first matrix sub-grid cell is not necessarily a pore volume cell. Similarly if all the cells in the matrix are defined as rock cells (cells with a coal region number) there is no pore volume represented in the matrix. This means that if a non-zero pore volume is assigned to the primary matrix block on input, the volume will be neglected. Warnings are issued if this is the case. It should also be noted that if the third item of NMATOPTS is put to FPORV and there is no pore volume within the matrix, the partitioning method switches to use the MBLKV method for this matrix grid cell. Warnings are issued if this is the case. Any rock volume that is assigned to cells that are not coal cells will not alter the simulation results. Using the FPORV method with NMATOPTS, the first matrix block will get a size that will match the specified pore volume under the assumption that all the matrix blocks have a pore volume. However, if

only some of the sub-matrix cells have a pore volume, a normalization of the input porosity is applied to these cells in order to match the input matrix grid cells total pore volume. As an example, if only one sub-matrix grid cell is a pore volume cell, the entire matrix pore volume is assigned to this sub-matrix grid cell.

Instant sorption

For the instant desorption model the pore volume has the usual interpretation, and the adsorption desorption process is represented as a source sink term into the matrix pore system. It should be noted that the coal volume of each matrix subgrid cell is computed from the average grid cell volume of each coal cell, where the pore volume of all the connected matrix and fracture grid cells are subtracted. If [ROCKFRAC](#) is specified for a non-coal cell, this volume is also subtracted from the coal volume. The coal volume is then distributed evenly between the coal cells.



Matrix discretization (NMATOPTS) and instant sorption

In combination with the multi porosity option the matrix subgrid cells are assigned pore volumes and rock/coal volumes automatically if the [NMATOPTS](#) keyword is present. The primary (outer) matrix pore and rock volume is partitioned among the matrix subgrid cells. Note that the pore volume of the fractures are subtracted from the rock volume. The primary matrix subgrid cells need to be defined as coal by [COALNUM](#) in order to tell that the matrix contains adsorbed gas. To redefine the matrix subgrid cells with a coal region number the keyword [COALNUMR](#) should be used. Only the pore volume and coal volumes as defined by the primary matrix sub-grid cells are used to generate the matrix sub-grid cells coal and pore volumes.

Note that having different adsorption properties for the different subcells also means that the partitioning of the volumes between the subcells becomes important.

It is possible to assign a rock density for each coal region number by the keyword [CRNDENS](#) in order to assign different density to the matrix subgrid cells.

Coal and pore volumes

If the [ROCKFRAC](#) keyword is used we have the following pore volume and coal volume definitions for the primary matrix sub-grid cells:

Sorption model	Pore volume primary matrix subgrid, PV_m	Coal volume primary matrix subgrid, CV_m
Instant	$V_m \varphi_m \times NTG_m$	$V_m r_m$
Time-dependent	$V_m \varphi_m \times NTG_m$	$V_m r_m$

The following table shows the pore volume and coal volume definitions for the primary matrix subgrid cells if [ROCKFRAC](#) is not used.

Sorption model	Pore volume primary matrix subgrid, PV_m	Coal volume primary matrix subgrid, CV_m
Instant	$V_m \varphi_m \times NTG_m$	$V_m - PV_m - PV_f$
Time-dependent	$V_m - CV_m - PV_f$	$V_m \varphi_m \times NTG_m$

where

V_m, V_f Bulk volume, matrix or fracture

φ_m, φ_f Porosity, matrix or fracture

r_m Rock fraction value in matrix

NTG_m, NTG_f Net to gross, matrix or fracture

Examples

The following data sets come with the ECLIPSE installation:

File	Description
MPORO.DATA	Gas water case (matrix discretization)
SHALEGAS1.DATA	Shale gas
SHALEGAS1T.DATA	Shale gas with time-dependent sorption

Table 61.1: Shale gas example files

62

Solid phase

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

Solid components can be used with the Blackoil, Equation of State (EoS), K-value, or Thermal options. Within the reservoir, these components remain in the solid phase throughout the simulation. However, solid components can be converted to fluid components through a chemical reaction. A solid component could be used, for example, to simulate production and take up of coke during combustion.

Solids can be either suspended in the oil phase, or deposited 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 are deposited on the formation reduce the pore volume available for flow. Permeability multipliers can be used to model this effect.

Summary of keywords

RUNSPEC section

- [SOLID](#) Requests solids option

GRID section

- [ROCKDEN](#) Rock density

PROPS section

Restriction	Keyword	Description
	CVTYPE	Define solid phase components
	CVTYPES	Define solid phase components at surface conditions
Reactions only	REACSMAX	Reaction rate solid saturation limit
Reactions only	REACCORD	Reaction rate solid order
	SCREF	Solid compressibility
	SDREF	Solid density at reference conditions

Restriction	Keyword	Description
	SOLIDADS	Solid adsorption table
	SOLIDMMC	Solid mobility multiplier (as a function of adsorbed concentration)
	SOLIDMMS	Solid mobility multiplier (as a function of adsorbed saturation)
	SOLWTAB	Solid wettability interpolation parameter (as a function of adsorbed concentration)
	SOLWW	Weight factor for wetting-solid concentration
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

REGIONS section

- [SOLWNUM](#) Solid wettability region numbers

SOLUTION section

Keyword	Description
SMF	Initial solid molar fraction
SSOLID	Initial solid saturation

In addition, solid specific mnemonics can be used with the [RPTSOL](#) keyword. These are described in the SCHEDULE keywords section.

SUMMARY section

Solid properties and production data can be output to the summary file.

Refer to [Grid block solid quantities](#), and [Solid production totals](#) in the *ECLIPSE Reference Manual*.

SCHEDULE section

Keyword	Description
BSOL	Solid reservoir molar densities
DENS	Solid reservoir density
HSOL	Solid enthalpies (Thermal option)
SSOLID	Solid saturations
SSFRAC	Suspended solids fraction
SOLADS	Adsorbed solid concentration (mass of solid per mass of rock), requires ROCKDEN
SOLADW	Adsorbed wetting-solid concentration (mass of solid per mass of rock), requires ROCKDEN

Keyword	Description
SMMULT	Solid mobility multiplier
SOLWET	Solid wettability interpolation parameter

Solid properties

Solid phase

The [SOLID](#) keyword must be used if solid components are present.

The number of components (including solid components) must be specified using the [COMPS](#) keyword, or the [COMPSOL](#) keyword if it is a [BLACKOIL](#) simulation.

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

$$b_s = \frac{N_c}{\sum_{c=1}^{N_c} x_s^c \cdot V_{sol}^c}$$

where

x_s^c is the mole fraction of component c in the solid phase.

The component molar solid phase volume V_{sol}^c is given by:

$$V_{sol}^c = \frac{MW^c}{\rho^c}$$

where MW^c is the molecular weight of component c given by the [MW](#) keyword, and the component solid phase density ρ^c is given by:

$$\rho^c = \frac{\rho_{ref}^c}{(1 + C_{T1}(T - T_{ref}))(1 - C_p(P - P_{ref}))}$$

The reference density ρ_{ref}^c is defined by the [SDREF](#) keyword

the standard temperature and pressure, T_{ref} and P_{ref} are defined by keywords [STREF](#) and [SPREF](#) (or by [STCOND](#))

the thermal expansion coefficient C_{T1} is defined by the [STHERMX1](#) keyword, and

the component isothermal compressibility C_p is defined by the [SCREF](#) keyword

Trapped gas

Thermal only

When modeling foamy oil it may be necessary to model a component that has a gas-like compressibility in the solid phase (see "[Foamy oil](#)"). This can be done using keywords [CCTYPE](#). In this case the component molar solid volume is given by the gas law

$$V_{sol}^c = \frac{Z^c RT}{P} \quad \text{Eq. 62.1}$$

where R is the gas constant and the compressibility terms Z^c can be set for each component using the [ZFACTOR](#) keyword.

Keyword [CCTYPES](#) can be used to set the compressibility at surface conditions.

Specific heats

Thermal only

The specific heat C_{sol}^c of a component c in the solid phase is given by

$$C_{sol}^c = C_{sol1}^c + C_{sol2}^c \cdot (T - T_{st})$$

where

the coefficients are defined by keywords [SPECHS](#) and [SPECHT](#)

T_{st} is the standard temperature defined by [STCOND](#)

the specific heat must always be positive.

Solid saturations

When solids are present the pore volume V_p is taken up by both fluids and solids

$$V_p = V_f + V_s = V_p \cdot (V_{flu} + V_{sol})$$

where the specific fluid volume is given by

$$V_{flu} = V_{oil} + V_{gas} + V_{wat}$$

Normalized fluid saturations are defined as fractions of the fluid volume

$$S_o = \frac{V_{oil}}{V_{flu}}, S_g = \frac{V_{gas}}{V_{flu}}, S_w = \frac{V_{wat}}{V_{flu}}$$

The normalized saturations satisfy the equation:

$$S_o + S_g + S_w = 1$$

Saturations can also be defined as fractions of the pore volume:

$$\hat{S}_s = V_{sol}$$

$$\hat{S}_o = V_{oil} = V_{flu} \cdot S_o = (1 - \hat{S}_s) \cdot S_o$$

$$\hat{S}_g = V_{gas} = V_{flu} \cdot S_g = (1 - \hat{S}_s) \cdot S_g$$

$$\hat{S}_w = V_{wat} = V_{flu} \cdot S_w = (1 - \hat{S}_s) \cdot S_w$$

These modified saturations satisfy the equation:

$$\hat{S}_o + \hat{S}_g + \hat{S}_w + \hat{S}_s = 1$$

The solid saturation \hat{S}_s and the multiplier $1 - \hat{S}_s$ are used to calculate permeability reduction and initial conditions.

Adsorption model of deposition and erosion

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

$$C_s = f_s \cdot W_s \quad \text{Eq. 62.2}$$

where

f_s is the fraction of solids in suspension

W_s is the total weight of solids per unit pore volume.

The adsorbed solid concentration C_a is defined as the weight of solids per weight of rock:

$$C_a = (1 - f_s) \cdot W_s / W_R \quad \text{Eq. 62.3}$$

where

W_R is the weight of rock per unit pore volume:

$$W_R = \rho_R \cdot (1 - \theta) / \theta \quad \text{Eq. 62.4}$$

θ is the porosity and

ρ_R is the rock density given by the [ROCKDEN](#) keyword.

Keyword [SOLIDADS](#) is used to define an adsorption function:

$$C_a = C_a(C_s) \quad \text{Eq. 62.5}$$

which allows equations 62.2, 62.3 and 62.4 to be solved for the suspended solids fraction f_s .

There are two special cases

- If the SOLIDADS keyword is not used, it is assumed that all solids are adsorbed, $f_s = 0$.
- If the SOLIDADS keyword is used, but the rock density is zero, then it is assumed that all solids are in suspension, $f_s = 1$.

Output of adsorbed solid concentration and suspended solids fraction

The adsorbed solid concentration C_a and suspended solids fraction f_s can be output to the print file or restart file by using the mnemonics SOLADS and SSFRAC with the RPTRST, RPTSCHED and RPTSQL keywords. Values in individual grid blocks can be output to the summary file by using the keywords BSOLADS and 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

/

Adsorption of dissolved water components

In the previous section, the physical processes of deposition and erosion are modeled by adsorption functions. Adsorption can also be used to model the chemical process where a dissolved solid comes out of solution and is adsorbed to the rock. In this case a water component is used to represent the dissolved solid and a solid component is used to represent the component in the adsorbed state.

A reversible chemical reaction is used to model the dissolution process. See "[Equilibrium deviation reaction rates](#)".

The equilibrium value can be set in two different ways:

- Using the [EQLDKVCR](#) keyword to specify an equilibrium k-value for the water component.
- Using the [EQLDTAB](#) keyword to specify the equilibrium value for the adsorbed solid as a function of the water mole fraction of the dissolved component.

The reaction can be one way, to model just adsorption, or just desorption, or reversible, to model both processes simultaneously.

If the reaction rate is set to a high value, the adsorption / desorption process will be close to instantaneous, but this may be harder to converge. In this case a smaller reaction rate may improve the run, although it will take longer to reach equilibrium.

If the water component and solid component occupy different volumes, the adsorption reaction will change the fluid + solid volume which will lead to a pressure change to compensate. This can be avoided by setting similar values for the water and solid reference densities and compressibilities (see for example [DREFW](#) and [SDREF](#)).

In ECLIPSE 100, keywords such as [ALKADS](#), [FOAMADS](#), [PLYADS](#) and [SURFADS](#) define an adsorbed concentration as a function of a dissolved concentration $C_a = C_a(C_d)$. This data can be converted for use with [EQLDTAB](#). The water mole fraction of the dissolved components is given by:

$$a_d = \frac{MW_w}{\rho_{ws}} \cdot \frac{C_d}{MW_d}$$

where

MW_w and MW_d are the molecular weight of the water mixture and the dissolved component (which are the same if the dissolved component is treated as a tracer), and

ρ_{ws} is the surface density of water.

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 [RPTSQL](#) keywords. Values of the mobility multiplier in individual grid blocks can be output to the summary file by using the keyword [BSMMULT](#).

Solid wettability effects

Adsorbed solids can change the wettability of the rock. Two sets of saturation functions can be specified representing different wettability conditions. An interpolation parameter can be defined as a function of the adsorbed wetting-solid concentration using the [SOLWTAB](#) keyword. The concentration of wetting-solid is calculated by

$$C_{wa} = (1 - f_s) \cdot [\sum w^c \cdot m^c \cdot MW^c] / W_R$$

Here w^c is a wettability weight factor given by the [SOLWW](#) keyword. A value of $w^c = 0$ means the solid component has no effect on wettability, while a value of $w^c = 1$ means the full weight of the component is included in the wettability calculation.

The saturation functions corresponding to low adsorbed solids are identified using the [SATNUM](#) keyword, and the saturation functions corresponding to high adsorbed solids are identified using the [SOLWNUM](#) keyword. In this model the adsorbed solids only affect oil and water saturation functions (relative permeability and capillary pressure). The gas saturation functions are determined by the SATNUM functions alone.

This model cannot be used with other saturation function options, such as hysteresis, [ODD3P](#) and [IKU3P](#). However it can be used with the oil-water miscibility model described in the "[Surfactant Model](#)".

Output of the solid wettability data

The wetting-solid concentration, C_{wa} , and the solid wettability interpolation parameter can be output to the print file or restart file by using the mnemonics SOLADW and SOLWET with the [RPTRST](#), [RPTSCHE](#)D and [RPTSO](#)L keywords. Values of the concentration and interpolation parameter in individual grid blocks can be output to the summary file by using the keywords BSOLADW and BSOLWET.

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 - that is 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:

- defined explicitly by the user
- calculated by the simulator from equilibrium conditions
- 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

$$m^c = (1 - \hat{S}_s) \cdot m_{fl}^c + \hat{S}_s \cdot b_s \cdot x_s^c \quad \text{Eq. 62.6}$$

where

\hat{S}_s is the initial solid saturation, defined by keyword [SSOLID](#)

x_s^c is the initial solid molar fraction, defined by keyword [SMF](#)

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 equation [62.6](#) becomes:

$$m^c = m_{fl}^c$$

Initial porosity

The terms $(1 - \hat{S}_s)$ and \hat{S}_s in equation [62.6](#) represent the fractions of the pore-volume initially containing fluid and solid. The initial pore volume, which is usually calculated from the porosity θ should therefore include both a fluid volume and a solid volume:

$$\theta = \frac{V_f + V_s}{V_f + V_s + V_r}$$

where V_r is the rock volume.

In some circumstances the measured porosity θ' may only take into account the initial fluid volume

$$\theta' = \frac{V_f}{V_f + V_s + V_r} = \theta \cdot (1 - \hat{S}_s)$$

If these values are used to define the porosity ([PORO](#)) then the initial pore volumes should in theory be multiplied by a factor $1 / (1 - \hat{S}_s)$ using [MULTPV](#). However if the initial solid saturations are small or zero this term can be neglected.

Initial energy

Thermal only

In thermal runs the initial energy density e is calculated from:

$$V_b \cdot e = V_p \cdot ((1 - \hat{S}_s) \cdot e_{fl} + \hat{S}_s \cdot b_s \cdot e_s) + V_r \cdot e_r \quad \text{Eq. 62.7}$$

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](#)". 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](#) and [REACSMAX](#) keywords.

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 specified in [Grid block solid quantities](#) and [Solid production totals](#) in the *ECLIPSE Reference Manual*.

63

Solution of the linear equations

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The linear equations in ECLIPSE 100 are solved iteratively using Orthomin [Ref. 118]. The default preconditioner for the ORTHOMIN iterations is Nested Factorization. Alternatively, if the [CPR](#) keyword is entered, the Constrained Pressure Residual (CPR) preconditioner can be used instead. At each iteration, Orthomin minimizes the sum of the squares of the residuals by orthogonalizing 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 method for solving the linear equations in ECLIPSE 300 is similar to ECLIPSE 100, except that the GMRES [Ref. 93] algorithm is employed instead of ORTHOMIN. In ECLIPSE 300 the GMRES iterations can be preconditioned by nested factorization, which is the default, or by the CPR preconditioner.

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 three-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

$$r = b - Ay = (B - A)y \quad \text{sum} = \sum_{i=1}^{n \text{ cells}} r_i \quad \text{Eq. 63.1}$$

This sum is zero if B is chosen such that

$$\text{colsum}(B) = \text{colsum}(A)$$

where ($\text{colsum}A$) 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 ($\text{rowsum}A$) 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. 122].

Nested factorization preconditioning

A typical matrix for a simple three-dimensional system is shown in equation 63.2 which shows the structure of the Jacobian matrix for a simple 3*2*3 problem.

$$A = \begin{bmatrix} D & U_1 & . & U_2 & . & . & U_3 & . & . & . & . & . & . & . & . & . \\ L_1 & D & U_1 & . & U_2 & . & . & U_3 & . & . & . & . & . & . & . & . \\ . & L_1 & D & . & . & U_2 & . & . & U_3 & . & . & . & . & . & . & . \\ L_2 & . & . & D & U_1 & . & . & . & . & U_3 & . & . & . & . & . & . & . \\ . & L_2 & . & L_1 & D & U_1 & . & . & . & . & U_3 & . & . & . & . & . & . \\ . & . & L_2 & . & L_1 & D & . & . & . & . & . & U_3 & . & . & . & . & . \\ L_3 & . & . & . & . & . & D & U_1 & . & U_2 & . & . & U_3 & . & . & . & . \\ . & L_3 & . & . & . & . & L_1 & D & U_1 & . & U_2 & . & . & U_3 & . & . & . \\ . & . & L_3 & . & . & . & L_1 & D & U_1 & . & U_2 & . & . & U_3 & . & . & . \\ . & . & . & L_3 & . & . & L_2 & . & L_1 & D & U_1 & . & . & . & U_3 & . & . \\ . & . & . & . & L_3 & . & . & L_2 & . & L_1 & D & U_1 & . & . & . & U_3 & . & . \\ . & . & . & . & . & L_3 & . & . & L_2 & . & L_1 & D & . & . & . & . & U_3 & . \\ . & . & . & . & . & . & L_3 & . & . & . & . & D & U_1 & . & U_2 & . & . \\ . & . & . & . & . & . & . & L_3 & . & . & . & L_1 & D & U_1 & . & U_2 & . \\ . & . & . & . & . & . & . & . & L_3 & . & . & L_1 & D & . & . & U_2 & . \\ . & . & . & . & . & . & . & . & . & L_3 & . & L_2 & . & . & D & U_1 & . \\ . & . & . & . & . & . & . & . & . & . & L_3 & . & L_2 & . & L_1 & D & U_1 \\ . & . & . & . & . & . & . & . & . & . & . & L_3 & . & L_2 & . & L_1 & D & U_1 \end{bmatrix} \quad \text{Eq. 63.2}$$

ECLIPSE chooses direction 1 automatically to correspond to the direction of highest transmissibility (usually the Z direction). Thus L1 and U1 usually correspond to derivatives of flows between cells in the Z direction. The inner solver direction may be overridden using the `SOLVDIRS` keyword. This may be useful, for example, if it is known in advance that the principal flow is in the X direction.

Three-dimensional finite difference systems give rise to sparse linear equations with banded matrices equation (63.2) such as

$$A \equiv D + L1 + U1 + L2 + U2 + L3 + U3 \quad \text{Eq. 63.3}$$

where

D is a diagonal matrix

L1, L2 and L3 are lower bands

U1, U2 and U3 are upper bands

The elements of D, L1, L2, L3, U1, U2, U3 are 2×2 matrices for implicit two-phase systems (for example oil/water, oil/gas, gas/water) and 3×3 matrices for implicit three-phase systems. In the [IMPEs](#) case these elements are scalars.

L2 and U2 connect cells in direction 2; L3 and U3 connect cells in direction 3. In Nested Factorization, an approximation, B, is constructed by the following nested sequence of factorizations

$$\begin{aligned} B &= (P + L3)P^{-1}(P + U3) = P + L3 + U3 + L3 \cdot P^{-1} \cdot U3 \\ P &= (T + L2)T^{-1}(T + U2) = T + L2 + U2 + L2 \cdot T^{-1} \cdot U2 \\ T &= (G + L1)G^{-1}(G + U1) = G + L1 + U1 + L1 \cdot G^{-1} \cdot U1 \end{aligned} \quad \text{Eq. 63.4}$$

where

G is a diagonal matrix and

P^{-1} , T^{-1} and G^{-1} are the inverse of PT and G respectively.

It follows that

$$\begin{aligned} B &= G + L1 + U1 + L2 + U2 + L3 + U3 + L1G^{-1}U1 + L2T^{-1}U2 + L3P^{-1}U3 \\ &= A + G - D + L1G^{-1}U1 + L2T^{-1}U2 + L3P^{-1}U3 \end{aligned} \quad \text{Eq. 63.5}$$

The diagonal matrix G is computed using

$$0 = G - D + L1G^{-1}U1 + \text{Colsum}(L2T^{-1}U2 + L3P^{-1}U3) \quad \text{Eq. 63.6}$$

This choice of G ensures that **colsum**(B) = **colsum**(A) which, in turn, ensures that there are no material balance errors in the solution of the linear equations.

Alternatively, the preconditioner may be constructed such that **rowsum**(B) = **rowsum**(A), thus summing the pressure terms in A by rows to ensure accurate solution of the pressure equation in **IMPES** or **AIM** cases. This technique is described in [Ref. 122].

Off band elements, corresponding to non-neighbor connections, arising in the treatment of, for example, faults, local grid refinement and completing the circle in 3D radial studies, destroy the simple structure of the bands $L1$, $U1\dots$, 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

$$G = D - L1G^{-1}U1 - \text{colsum}(L2T^{-1}U2) - \text{colsum}(L3P^{-1}U3)$$

The calculation proceeds one cell at a time; thus when G^{-1} is known for a cell, we can calculate the contribution $L1G^{-1}U1$ to the value of G in the next cell. When G^{-1} is known on a line, we can calculate the contribution of $\text{colsum}(L2T^{-1}U2)$ to G on the next line. When G is known on a plane, we can calculate the contribution of $\text{colsum}(L3P^{-1}U3)$ 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 , that is:

$$G = D - L1G^{-1}U1 - \text{rowsum}(L2T^{-1}U2) - \text{rowsum}(L3P^{-1}U3)$$

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

$$(P + L3)(I + P^{-1}U3)y = r \quad \text{Eq. 63.7}$$

using

$$y = P^{-1}(r - L3y)$$

and

$$y = y - P^{-1}U_3y \quad \text{Eq. 63.8}$$

Equation 63.7 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 L_3y involves the ‘known’ solution, y , on the previous plane. Similar considerations apply to the solution of equation 63.8 that is performed in reverse order starting with the last plane and sweeping backwards one plane at a time.

During the solution of equations 63.7 and 63.8 we must compute vectors of the form $z = P^{-1}q$ on each plane. This involves solving equations like

$$(T + L_2)(I + T^{-1}U_2)z = q \quad \text{Eq. 63.9}$$

using

$$z = T^{-1}(q - L_2z)$$

and

$$z = z - T^{-1}U_2z \quad \text{Eq. 63.10}$$

Equation 63.9 is solved in a forward sweep through the plane, one line at a time. Similarly equation 63.10 is solved by sweeping backwards one line at a time.

Finally, we note that during the solution of equations 63.9 and 63.10 we must compute vectors of the form $w = T^{-1}v$ on each plane. This involves solving the tri-diagonal equation

$$(G + L_1)(I + G^{-1}U_1)w = v \quad \text{Eq. 63.11}$$

using

$$w = G^{-1}(v - L_1w) \quad \text{Eq. 63.12}$$

and

$$w = w - G^{-1}U_1w \quad \text{Eq. 63.13}$$

Equations 63.12 and 63.13 are solved by sweeping first forwards, then backwards through the cells in a line. This may be solved as a block tri-diagonal in the intermediate nesting if a method exists of inverting the line tri-diagonal equations. This may be done simply, using the Thomas algorithm, and so the entire system may be solved. Computing time required to calculate a new search direction is reported under the mnemonic SRCH in the ECLIPSE timing reports.

Orthomin

The Orthomin procedure used by ECLIPSE can be summarized in the steps below:

1. Compute the initial solution

$$y = B^{-1}b, \quad x \rightarrow y \quad \text{Eq. 63.14}$$

2. Compute the initial residual

$$r = b - Ay \quad \text{Eq. 63.15}$$

- If the column 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.
3. Test for convergence and escape if converged
 4. Find a new search direction

$$y = B^{-1}b \quad \text{Eq. 63.16}$$

5. Orthogonalize y to the stack of previous search directions (z)

$$y = y - \frac{(y \cdot z)}{(z \cdot z)z} \quad \text{Eq. 63.17}$$

6. Find the optimum step length, s , which minimizes the saturation normalized sum of the squares of the residuals ($r \cdot r$):

$$s = \frac{r \cdot q}{q \cdot q} \quad \text{Eq. 63.18}$$

$$q = A \cdot y \quad \text{Eq. 63.19}$$

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 $\text{MIN}(0.1|b|_{\max}, 0.001r_{\text{top}}, \text{XXXLCV})$. 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 column constraint makes the initial residuals quite large. Values of the step length, s , and the orthogonalizations coefficients (beta) are also printed in the debug output. The perfect step is $s = 1.0$, but this is seldom achieved. If the seventh **DEBUG** parameter is set to 2 then the sum of the absolute un-normalized residuals is printed corresponding to the input residual, b , and the final residual, $r = b - Ax$. This is the ‘residual from solution check’. If the seventh **DEBUG** parameter is set to 3 then residual maps are printed for each phase at each iteration.

ECLIPSE 300

The convergence criterion on the linear solver is a reduction in RMS of the residual, so that convergence is achieved at the nth iteration if

$$\frac{\|R\|_2}{\|R^o\|_2} < TOL \quad \text{Eq. 63.20}$$

where

$$\|R\|_2 = \sqrt{\frac{\sum_{i=1}^N \sum_{c=1}^{N_{ci}} \left(\frac{R_{i,c}}{PV_i} \right)^2}{N}} \quad \text{Eq. 63.21}$$

with

R = linear residual

R^o = initial linear residual

$R_{i,c}$ = residual for equation c in grid cell i

PV_i = pore volume for cell i

N = total number of active cells

N_{ci} = number of equations for cell i

TOL = RMS convergence tolerance specified in **CVCRIT** (default 1.0E-5).

Treatment of wells

The strongly coupled fully-implicit treatment of wells in ECLIPSE 100 extends the linear equations to include the well variables, w , corresponding to the bottom hole flowing pressure, and well bore flowing fractions. Thus each well, like each cell, has three variables in the three phase case. Including the well terms, the linear equations now become

$$\begin{aligned} Ax + Cw &= r \\ Rx + Dw &= u \end{aligned} \quad \text{Eq. 63.22}$$

where

u is the well residuals,

and if there are N grid blocks and L wells:

A is a banded $N \times N$ matrix (equation 63.2),

C is a sparse $N \times L$ matrix,

R is a sparse $L \times N$ matrix,

D is a sparse $L \times L$ matrix.

Usually D is lower triangular and easily inverted, allowing the well terms to be eliminated. Thus the equation for x becomes

$$(A - CD^{-1}R)x = r - CD^{-1}u \quad \text{Eq. 63.23}$$

which is easily solved by preconditioned Orthomin.

ECLIPSE 300

In ECLIPSE 300, there is a single well variable w , corresponding to well bottom-hole pressure. When an implicit treatment of the gravity head for each completion is used, the well constraint equations cannot be solved as in equation 63.22. In the case of wells with multi-layer completions, extra coupling terms are introduced between all the cells on a given well. This is because the required well bottom hole pressure is a function of the solution values in each completion and each completion flow is a function of this well bottom-hole pressure. An additional contribution to the Jacobian is then defined:

$$J_w = \frac{dQ_w}{dX} \quad \text{Eq. 63.24}$$

which we include in the matrix A above. The column sum (or row sum) of J_w is included on the diagonal of the pre-conditioning matrix B . When using explicit head calculations the well equations do, however, reduce to the form of equation 63.23.

See [Ref. 54], and [Ref. 21] for further information.

Nested factorization for LGRs

The linear equations to be solved take the form

$$Ax = b \quad \text{Eq. 63.25}$$

where A is a banded matrix. The banded nature can be expressed as

$$A = D + L1 + U1 + L2 + U2 + L3 + U3 + L4 + U4 \quad \text{Eq. 63.26}$$

where

D is a diagonal matrix,

$L1, L2, L3, L4$ are lower bands, and

$U1, U2, U3, U4$ are upper bands.

The bands $L4$ and $U4$ represent couplings between sub-grids and are usually irregular.

In nested factorization, an approximate preconditioner M is constructed by the following nested sequence of factorizations:

$$M = (B + L4)B^{-1}(B + U4) = B + L4 + U4 + L4 \cdot B^{-1} \cdot U4 \quad \text{Eq. 63.27}$$

$$B = (P + L3)P^{-1}(P + U3) = P + L3 + U3 + L3 \cdot P^{-1} \cdot U3 \quad \text{Eq. 63.28}$$

$$P = (T + L2)T^{-1}(T + U2) = T + L2 + U2 + L2 \cdot T^{-1} \cdot U2 \quad \text{Eq. 63.29}$$

$$T = (G + L1)G^{-1}(G + U1) = G + L1 + U1 + L1 \cdot G^{-1} \cdot U1 \quad \text{Eq. 63.30}$$

It follows that

$$\begin{aligned} M &= G + L1 + U1 + L2 + U2 + L3 + U3 + L4 + U4 \\ &\quad + L1G^{-1}U1 + L2T^{-1}U2 + L3P^{-1}U3 + L4M^{-1}U4 \end{aligned} \quad \text{Eq. 63.31}$$

that results in

$$M = A + G - D + L1G^{-1}U1 + L2T^{-1}U2 + L3P^{-1}U3 + L4M^{-1}U4 \quad \text{Eq. 63.32}$$

The diagonal matrix G is defined as

$$G = D - L1G^{-1}U1 - \text{rowsum}(L2T^{-1}U2 + L3P^{-1}U3 + L4P^{-1}U4) \quad \text{Eq. 63.33}$$

This choice ensures that

$$\text{rowsum}(A) = \text{rowsum}(M) \quad \text{Eq. 63.34}$$

where $\text{rowsum}(A)$ is the diagonal matrix formed by summing the pressure coefficients of A in rows.

This ensures that M^{-1} is an exact inverse on the sub-space spanned by constant pressure within each sub-grid. This technique is described in [Ref. 123].

ECLIPSE 100

In-place LGRs (specified using keyword [LGRLOCK](#)) are solved using the above technique but with the `rowsum` constraint for black oil models.

ECLIPSE 300

For local grid refinements, the factorization can be enhanced by inner iterations on specific sub-grids. This technique is particularly effective when the inner iterations involve relatively small LGRs such as radial refinements. The sub-grid iterations are controlled by the `LSCRITS` keywords when using the WARP linear solver (see keyword [LSCRIT](#)).

The ECLIPSE 300 WARP linear solver

ECLIPSE 300

The WARP linear solver is the default solver for ECLIPSE 300. It is at present based on nested factorization only. Prior to 2002A the default solver was ECLSOL, which can be activated instead of WARP by using [item 66](#) of the `OPTIONS3` keyword. The WARP solver allows more flexibility for solutions of LGR problems than the ECLSOL solver.

Restrictions of the WARP solver

The 2003A WARP linear solver is incompatible with the following keywords:

- [MPFA](#)
- [NINEPOIN](#)
- [NONNC](#)

The 2002A WARP linear solver is also incompatible with

- [IMPSAT](#)

WARP sub-grids

The base entities that the WARP solver manipulates are sub-grids. These represent a portion of the total cells that are ordered to be a contiguous block of matrix.

- Every active cell in the grid is assigned to a sub-grid. In principle this association could be arbitrary; however, there are severe efficiency implications if there are very many sub-grids, or the sub-grids do not mainly contain cells which are only connected to other cells within the sub-grid.
- Sub-grid-to-sub-grid connections are implemented as NNC connections; the total generated is reported in the PRT file. Connections internal to sub-grids are, in general, stored as a mixture of band and NNC connections.
- Sub-grids may or may not be related to LGR grids. It is possible to have separate sub-grids for LGRs and the base grid. However, it is also possible to have sub-grids:
 - representing an amalgamation of portions of the base grid and portions of LGR grids (in this case the LGR planes are ordered first within any sub-grid to enhance the efficiency of this approach);
 - representing an amalgamation of portions of LGRs in a particular level of refinement.
- The form of the sub-grids is best observed by visualizing the output generated from the ‘ SUBG ’ mnemonic in the [RPTRST](#) keyword.
- The preconditioner applied to each sub-grid is nested factorization (using ROWSUM on pressure only).
- The WARP linear solver performs nested factorization on the block matrix, where each block is a sub-grid. As with the sub-grid pre-conditioner, this block nesting preserves ROWSUM (on pressure only).
- When using the parallel option, sub-grids do not span parallel partitions; each sub-grid is completely contained with a processor.

WARP coloring

Each sub-grid is given a color. The WARP solver introduces extra nesting between colors. Sub-grids of the same color can be executed in parallel.

- NNCs between colors are preconditioned by extra nesting levels. If sub-grids only contain cells of a particular refinement level, then this is used by WARP to force a nesting between refinement levels. That is all colors on different levels of refinement are treated independently. Hence if two levels of refinement exist (because of the presence of LGRs), and each level has sub-grids containing two colors, then effectively four colors exists. 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 performs nested factorization on a sub-grid level. Sub-grids of the same color can be computed in parallel. The pre-conditioner performed on a sub-grid is nested factorization. Note that:

- The generation of NNCs between sub-grids will cause a memory overhead.
- Increasing the number of colors will increase the expense of each linear iteration, as extra levels of nesting are required to solve the block sub-grid matrix.

The creation of sub-grids and coloring is controlled by the `PARALLEL` and `WCOL` keywords. The `PARALLEL` keyword controls the division of the original problem into partitions. Figure 63.1 shows a two-color scheme resulting from a 4-processor run. Note that all the LGRs have inherited the color from the base grid. The LGR coloring scheme is controlled by the `WCOL` keyword.

Figure 63.2 shows a possible set of 8 sub-grids that may arise from these colors. In this case portions of the LGR grids have been amalgamated with the base grid, and the WARP solver has no knowledge of refinement levels. This type of amalgamation is generated if the 3rd item of the `WCOL` keyword is set to 2. By amalgamation of LGR and base sub-grids, the load balancing of the parallel run is more easily facilitated. For a parallel run all colors must have equal work on all processors for a perfect load balance. The more colors that are introduced the more difficult this becomes. This amalgamation strategy ensures no extra levels of nesting are introduced by adding refinement levels. However, for cases where the nested factorization is required between refinement levels this strategy may prove to be far from ideal.

Figure 63.2 shows a different sub-grid configuration for the same coloring as in figure 63.1. Here the LGRs are not amalgamated with the base grid; instead, all LGRs of the same refinement level and the same color are amalgamated together. Hence because the base grid has 8 sub-grids, and there is one level of LGR refinement spread fairly evenly over the grid, 16 sub-grids have been generated. Here WARP has knowledge of the refinement and will utilize this by adding nesting between refinement levels. This sub-grid configuration was generated by setting the 3rd item of `WCOL` to 1.

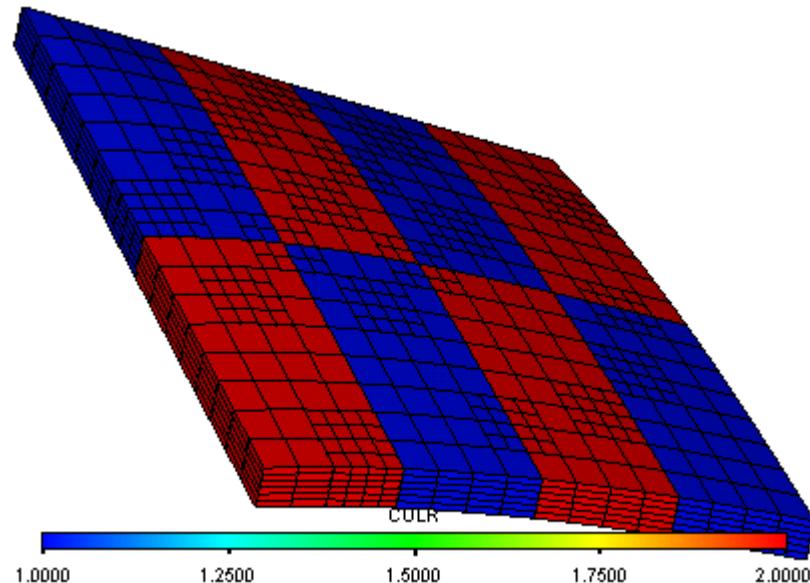


Figure 63.1. Two color (red-blue) split suitable for a 4-processors parallel run

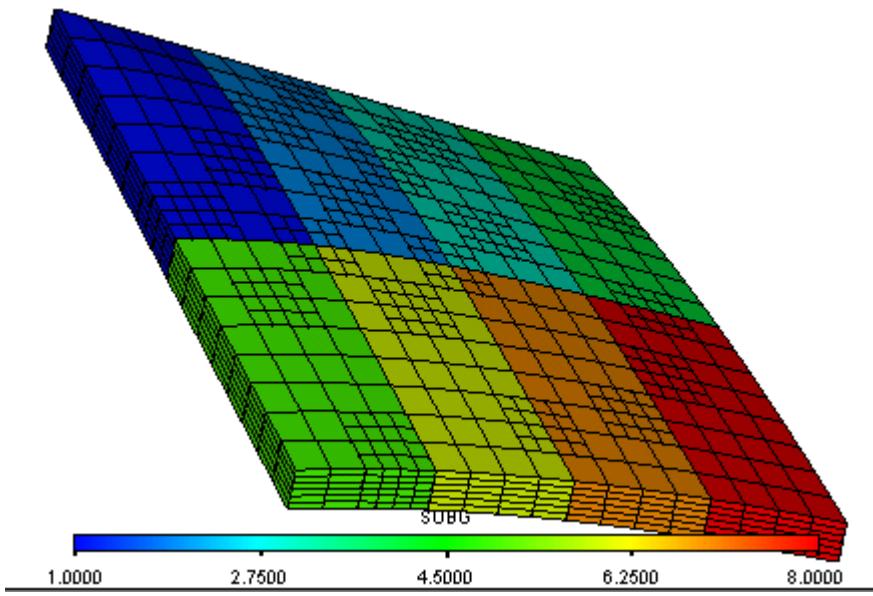


Figure 63.2. The sub-grids from the previous figure with amalgamation of LGRs and base grid

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 2×2 block color matrix thus

$$A = \begin{bmatrix} 0 & 0 \\ L_{\alpha\beta} & 0 \end{bmatrix} + \begin{bmatrix} S_\alpha & 0 \\ 0 & S_\beta \end{bmatrix} + \begin{bmatrix} 0 & U_{\alpha\beta} \\ 0 & 0 \end{bmatrix} \quad \text{Eq. 63.35}$$

The procedure is to perform nested factorization on this block system, that is the pre-conditioner is

$$\tilde{M} = \begin{bmatrix} Q_\alpha & 0 \\ L_{\alpha\beta} & Q_\beta \end{bmatrix} \begin{bmatrix} Q_\alpha & 0 \\ 0 & Q_\beta \end{bmatrix}^{-1} \begin{bmatrix} Q_\alpha & U_{\alpha\beta} \\ 0 & Q_\beta \end{bmatrix} \quad \text{Eq. 63.36}$$

thus

$$\tilde{M} = \begin{bmatrix} Q_\alpha & U_{\alpha\beta} \\ L_{\alpha\beta} & (L_{\alpha\beta} Q_\beta^{-1} + Q_\beta) \end{bmatrix} \quad \text{Eq. 63.37}$$

The matrix Q is chosen to preserve rowsum (of pressure only) so

$$Q_\alpha = G_\alpha \quad \text{Eq. 63.38}$$

and

$$Q_\beta = G_\beta - \text{rowsum}(L_{\alpha\beta} Q_\beta^{-1} U_{\alpha\beta}) \quad \text{Eq. 63.39}$$

where G_α and G_β are the diagonal nested factorization matrices constructed from all the sub-grids associated with colors α and β respectively.

During this procedure Q^{-1} needs to be evaluated. If individual tolerances are imposed on sub-grids (through the `LSCRITS` keyword), then this inversion is approximated by multiple iterations of GMRES, rather than just one nested factorization inversion.

For multiple colors the above procedure is applied recursively. For example if four colors $\alpha, \beta, \gamma, \delta$ are present, the matrix A could be written

$$A = \begin{bmatrix} S_\alpha & U_{\alpha\beta} \\ L_{\alpha\beta} & S_\beta \\ \hline L_{\{\alpha\beta,\gamma\delta\}} & \begin{bmatrix} S_\gamma & U_{\gamma\delta} \\ L_{\gamma\delta} & S_\delta \end{bmatrix} \end{bmatrix} \quad \text{Eq. 63.40}$$

and then nested factorization can be performed on the 2×2 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

$$r_0 = b, \quad \beta = \|r_0\|_2, \quad v_1 = r_0 / \beta \quad \text{Eq. 63.41}$$

Iteration

For $k = 1, 2, \dots$ until convergence, do

Search direction

$$u_k = M^{-1}v_k \quad \text{Eq. 63.42}$$

Matrix multiply:

$$w = Au_k \quad \text{Eq. 63.43}$$

Orthogonalize:

$$\begin{aligned} h_{i,k} &= (w \cdot v_i) \\ w &= w - \sum_{i=1}^k h_{i,k} v_i \end{aligned} \quad \text{Eq. 63.44}$$

Normalize:

$$\begin{aligned} h_{k+1,k} &= \|w\|_2 \\ v_{k+1} &= \frac{w}{h_{k+1,k}} \end{aligned} \quad \text{Eq. 63.45}$$

Solution:

$$x_k = \sum_{i=1}^k \alpha_i u_i \quad \text{Eq. 63.46}$$

where the α_i are chosen to minimize the residual $\| b - Ax_k \|_2$

For complete implementation details see [Ref. 93].

The Constrained Pressure Residual (CPR) Preconditioner

The Constrained Pressure Residual (CPR) preconditioner [Ref. 139] is employed by the **CPR** linear solver and the **JALS** linear solver.

The reservoir equations encountered in reservoir simulation are of mixed character. The pressure equations are often elliptic dominated with long range coupling, whilst the saturation equations can exhibit hyperbolic behavior, with steep local gradients. The CPR preconditioner is designed to tackle the linear equations that arise from the mixed character of the reservoir equations. This section outlines the ECLIPSE implementation, which is used by the CPR linear solver.

The CPR preconditioner consists of a two stage approximation to the inverse of the Jacobian matrix A . The equations can be arranged so that all of the pressure equations appear first, followed by the saturation equations (or molar density equations, for compositional). The Jacobian matrix can then be decomposed into the pressure terms A_{pp} , the saturation terms A_{ss} , and the pressure-saturation coupling terms, A_{ps} and A_{sp} . Similarly, the solution vector, x , and the residual vector, R , are also decomposed into the pressure terms and the saturation terms. This results in the following system of linear equations:

$$\begin{bmatrix} A_{pp} & A_{ps} \\ A_{sp} & A_{ss} \end{bmatrix} \begin{bmatrix} x_p \\ x_s \end{bmatrix} = \begin{bmatrix} R_p \\ R_s \end{bmatrix} \quad \text{Eq. 63.47}$$

The full system contains n_{cell} cells, with n_{eqn} equations per cell. The pressure system consists of just n_{cell} equations. An $(n_{\text{cell}} \cdot n_{\text{eqn}})$ by n_{cell} matrix C can be defined, together with its transpose C^T , to transform between the full system and the pressure system:

$$C = \begin{bmatrix} I_p \\ 0 \end{bmatrix} \quad \text{Eq. 63.48}$$

Where I_p is the n_{cell} by n_{cell} identity matrix, with the same dimensions as the pressure system. To facilitate the first stage of the preconditioner, the full system is restricted to a smaller pressure system:

$$A_p = C^T G A C \quad \text{Eq. 63.49}$$

$$r_p = C^T G R \quad \text{Eq. 63.50}$$

The additional transformation matrix, G , depends on the chosen method of solving the pressure system. In this case, G is constructed to approximately decouple the pressure system from the full system:

$$G = \begin{bmatrix} I_p & -A_{ps} \hat{A}_{ss}^{-1} \\ 0 & I_s \end{bmatrix} \quad \text{Eq. 63.51}$$

The matrix \hat{A}_{ss} is a simple approximation to the saturation matrix. Note that if \hat{A}_{ss} was chosen as the true inverse of A_{ss} , the pressure equations would be fully decoupled, since the matrix A would have a zero block in the position of \hat{A}_{ps} . However, it is not efficient to do this in general as the inverse of A_{ss} is generally dense. For this reason, we replace it with a simple approximation, such as the diagonal matrix $\text{Diag}(A_{ss})$. Therefore:

$$A_p = \begin{bmatrix} I_p & 0 \\ 0 & I_s \end{bmatrix} \begin{bmatrix} I_p & -A_{ps}\hat{A}_{ss}^{-1} \\ 0 & I_s \end{bmatrix} \begin{bmatrix} A_{pp} & A_{ps} \\ A_{sp} & A_{ss} \end{bmatrix} \begin{bmatrix} I_p \\ 0 \end{bmatrix} = A_{pp} - A_{ps}\hat{A}_{ss}^{-1}A_{sp} \quad \text{Eq. 63.52}$$

$$r_p = \begin{bmatrix} I_p & 0 \\ 0 & I_s \end{bmatrix} \begin{bmatrix} I_p & -A_{ps}\hat{A}_{ss}^{-1} \\ 0 & I_s \end{bmatrix} \begin{bmatrix} R_p \\ R_s \end{bmatrix} = R_p - A_{ps}\hat{A}_{ss}^{-1}R_s \quad \text{Eq. 63.53}$$

ECLIPSE employs the CPR preconditioner within the Flexible GMRES (FGMRES) algorithm. The reduced pressure system is also solved using the GMRES algorithm, which results in two levels of GMRES iterations. At the outer level the full solution is obtained by performing a number of outer iterations with the two stage CPR preconditioner. For each outer iteration, a number of inner iterations are performed to obtain the pressure solution. The algorithm consists of the following steps:

1. The residual vector is restricted to the pressure system:

$$r_p = C^T G R \quad \text{Eq. 63.54}$$

2. The reduced pressure system is solved:

$$A_p x_p = r_p \quad \text{Eq. 63.55}$$

To solve the pressure system, ECLIPSE performs inner GMRES iterations with a nested factorization preconditioner, M_p . When the pressure system has been solved to the desired accuracy, the inner GMRES iterations are terminated.

3. The solution to the pressure system is then expanded to the full system, and provides a correction to the original residual:

$$\tilde{r} = r - ACx_p \quad \text{Eq. 63.56}$$

4. A second stage nested factorization preconditioner, \tilde{M} , is applied to the remaining residual to obtain a correction to the solution of the pressure system:

$$\tilde{x} = \tilde{M}^{-1} \tilde{r} \quad \text{Eq. 63.57}$$

5. The first and second stage solutions are combined, to provide the CPR preconditioned vector x to the FGMRES algorithm:

$$x = \tilde{x} + Cx_p \quad \text{Eq. 63.58}$$

Steps 1 to 5 are repeated within each outer FGMRES iteration, until the full solution has converged to within the required tolerance.

Information regarding the outer iterations can be reported with the usual performance summary quantities NLINEARS, MLINEARS and MSUMLINS. Information regarding the inner pressure iterations can be reported with the summary quantities NCPRLINS, NLINEARP, MLINEARP and MSUMLINP. Alternatively, the PERFORMA summary keyword can be used to provide all of this information.

It is more expensive to apply the two stage CPR preconditioner than the single stage nested factorization preconditioner that is used by default. Each linear iteration has a higher computational cost, and therefore takes longer to perform. However, convergence can usually be obtained in a smaller number of iterations of the full system. This makes the CPR preconditioner particularly suitable for simulations when the default linear solver is frequently failing to converge. In these cases, the more robust preconditioning could significantly reduce the number of linear iterations required and reduce the simulation time. The CPR preconditioner is also most suitable for fully implicit simulations, where the pressure system is significantly smaller than the full system.

In ECLIPSE 100, there are some options that cannot be used with the CPR linear solver. These include [IMPES](#), the multiple independent reservoirs option, the Russian Doll option, the friction well option, triple porosity, nested local grid refinements, single phase simulations, item 1 of keyword [WSEGSOLV](#), [BTOBALFA](#), [BTOBALFV](#) and the rivers option.

In ECLIPSE 300, it is currently not possible to use the CPR linear solver for the thermal option (the thermal option uses the [JALS](#) linear solver), the multiple (including triple) porosity option, in a dual porosity (or dual permeability) run containing local grid refinements and the [IMPES](#) option.

In ECLIPSE 300, the CPR linear solver generally performs best when the simulation is fully implicit. For simulations that use the adaptive implicit ([AIM](#)) scheme, the performance may deteriorate if the fraction of fully implicit cells is small.

64

Solvent model

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

This chapter describes a four-component extension of the ECLIPSE blackoil model. Please refer to "[Miscible flood modeling](#)" for more detail on the three-component modeling. The aim of the extension is to enable modeling of reservoir recovery mechanisms in which injected fluids are miscible with the hydrocarbons 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.

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 bypassed 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 is optionally a four-component system consisting of water, oil, reservoir gas and an injected solvent, or a three-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 where 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 viscosities and one for the densities.

Applications

There is a wide variety of secondary and tertiary recovery schemes in which the aim is to enhance the reservoir sweep by using a miscible injection fluid.

Typically the miscible injection fluid is expensive compared with traditional flooding fluids, such as dry gas and water. Miscible fluids are often referred to as ‘solvents’; examples of solvent schemes are listed below:

- High pressure dry gas processes, in which mass transfer effects at the gas-oil contact generate miscible flow conditions between the gas and the oil.
- A solvent such as LPG or propane may be injected as a ‘slug’ to be followed by an extended period of lean gas injection. The slug fluid is miscible with both the gas and oil, and the success of the method depends upon maintenance of both the reservoir pressure and the solvent slug itself.
- Certain non-hydrocarbon gases such as carbon dioxide produce miscible displacement of oil at pressures above a threshold value.
- All-liquid miscible displacements by fluids such as alcohol, normally injected as a slug between the in-place oil and the injected chase water.

In many of the schemes, the cost of the solvent is such that the recovery scheme needs to be ‘engineered’ to allow enough solvent to be injected to increase the total oil recovery while keeping the solvent costs low enough to make the project economic.

In applications where the reservoir pressure falls below the bubble point, and in schemes where a solvent slug is chased by a lean gas flood, there is a requirement to model four components within the reservoir (water, oil, solvent and lean gas). The solvent model in ECLIPSE provides a simple method of modeling these recovery mechanisms.

Model formulation

The solvent model is an extension of the standard three-phase, three-component model (water, oil, and gas), whereby an extra conservation equation is solved for the solvent. Below we detail the modeling of the different flow parts and the mixing rules used which are based on the empirical treatment suggested by M. Todd and W. Longstaff in [Ref. 111].

Relative permeability model: immiscible case

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 three phases water, oil and gas are specified as follows:

$$k_{rw} = k_{rw}(S_w) \quad \text{as a function of water saturation}$$

$$k_{rg} = k_{rg}(S_g) \quad \text{as a function of gas saturation}$$

$$k_{ro} = k_{ro}(S_w, S_g) \quad \text{as a function of both water and gas saturations.}$$

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,

$$k_{rgt} = k_{rg}(S_g + S_s) \quad \text{Eq. 64.1}$$

where S_s is the solvent saturation.

This total relative permeability is obtained from the [SGFN](#) keyword. Then the relative permeability of each gas component is taken as a function of the fraction of each gas component within the gas phase as:

Fraction of solvent:

$$F_{sol} = \frac{S_s}{S_g + S_s} \quad \text{Eq. 64.2}$$

Fraction of reservoir gas:

$$F_{gas} = \frac{S_g}{S_g + S_s} \quad \text{Eq. 64.3}$$

$$\begin{aligned} k_{rs} &= k_{rgt} \cdot k_{rfs}(F_{sol}) \\ k_{rg} &= k_{rgt} \cdot k_{rfg}(F_{gas}) \end{aligned} \quad \text{Eq. 64.4}$$

Typically the functions $k_{rfs} = k_{rs} / k_{rgt}$ and $k_{rfg} = k_{rg} / k_{rgt}$ (equation 64.4) are ‘straight line’ functions such that

$$k_{rfs}(0.0) = 0.0 \quad k_{rfg}(0.0) = 0.0$$

$$k_{rfs}(1.0) = 1.0 \quad k_{rfg}(1.0) = 1.0$$

However, ECLIPSE allows you to specify these gas-solvent functions using the [SSFN](#) keyword.

Relative permeability model: miscible case

In regions where solvent is displacing oil, the hydrocarbon displacement is miscible. However, the two-phase character of the water/hydrocarbon displacement needs to be taken into account. The relative permeabilities are given by:

$$k_{ro} = \frac{S_o}{S_n} k_m(S_n) \quad \text{Eq. 64.5}$$

$$k_{rgt} = \frac{S_s + S_g}{S_n} k_m(S_n) \quad \text{Eq. 64.6}$$

where

$S_n = S_g + S_s + S_o$ is the total hydrocarbon saturation,

k_{rgt} is the total relative permeability of the gas and solvent,

$k_m(S_n)$ is the relative permeability of hydrocarbon to water obtained from the [SOF2](#) keyword.

Then, the gas and solvent component relative permeability are computed from k_{rgt} as above, equations [64.2](#) - [64.4](#). 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:

$$k_{ro} = M_{kro} \left(\frac{S_o}{S_n} \right) k_m(S_n) \quad \text{Eq. 64.7}$$

$$k_{rgt} = M_{krsg} \left(\frac{S_s + S_g}{S_n} \right) k_m(S_n) \quad \text{Eq. 64.8}$$

Note that saturations end-points are taken into account in the previous equations as explained below.

Transition between miscible and immiscible cases

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 to vary from case to case.

The transition is handled by a user specified ‘Miscibility function’. This is a function of the solvent fraction in the gas phase. The function is tabulated 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. The residual oil saturation is

$$S_{or} = S_{orm} M + S_{ori}(1-M) \quad \text{Eq. 64.9}$$

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 entered using the [MISC](#) keyword

Similarly, for the solvent gas

$$S_{sgr} = S_{sgrm} M + S_{sgri}(1-M) \quad \text{Eq. 64.10}$$

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 at the scaled saturations using the new end points. Then the relative permeability is again an interpolation between the two using the miscibility function:

$$k_r = k_{rm} M + k_{ri}(1-M) \quad \text{Eq. 64.11}$$

where

k_{rm} the scaled miscible relative permeability

k_{ri} is the scaled immiscible relative permeability

The formulae above, equations [64.5](#) to [64.8](#), are modified to take into account these new saturation end points, so that:

$$k_{rgt} = \left(\frac{S_g + S_s - S_{sgr}}{S_n - S_{sgr} - S_{or}} \right) k_{rn} (S_n) \quad \text{Eq. 64.12}$$

$$k_{ro} = \left(\frac{S_o - S_{or}}{S_n - S_{sgr} - S_{or}} \right) k_{rn} (S_n) \quad \text{Eq. 64.13}$$

Modifying the straight line miscible functions using the [MSFN](#) keyword will result in

$$k_{rgt} = M_{krsg} \left(\frac{S_g + S_s - S_{sgr}}{S_n - S_{sgr} - S_{or}} \right) k_m (S_n) \quad \text{Eq. 64.14}$$

$$k_{ro} = M_{kro} \left(\frac{S_o - S_{or}}{S_n - S_{sgr} - S_{or}} \right) k_m (S_n) \quad \text{Eq. 64.15}$$

where the M_{kro} and M_{krsg} functions are supplied with the [MSFN](#) keyword.

The saturations supplied in this keyword are normalized and honor the end-point supplied in the **SORWMIS** and **SGCWMIS** keywords. The critical oil to water saturation S_{or} is entered with the **SORWMIS** keyword and the critical oil to gas saturation S_{gc} is supplied with the **SGCWMIS** keyword as a function of the water saturation. See "[Effect of water saturation](#)".

Pressure effect on relative permeability and capillary pressure 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 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. Here we described its usage with the relative permeability and capillary pressure.

For the relative permeabilities and saturation end points, the effect of pressure miscibility is combined with the solvent saturation **MISC** effects, so that the multiplier M coefficient in equations 64.9 to 64.11 is now replaced by:

$$M_T = M \cdot M_p \quad \text{Eq. 64.16}$$

The capillary pressure is interpolated as follows:

$$P_{cog} = M_p P_{cog_m} + (1 - M_p) P_{cog_i} \quad \text{Eq. 64.17}$$

$$\begin{aligned} P_{cog_m} &= P_{cog}(S_g) \\ P_{cog_i} &= P_{cog}(S_g + S_s) \end{aligned} \quad \text{Eq. 64.18}$$

Effect of water saturation

A feature of miscible gas injection processes that may also be modeled is the screening effect of high water saturation which reduces the contact between the miscible gas and the oil in-place. 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 ($S_{or}^* = S_{or} (S_w)$) and is input with the **SORWMIS** keyword. A mobile oil saturation is then calculated by

$$S_o^* = \text{MAX}(S_o - S_{or}, 0.0) \quad \text{Eq. 64.19}$$

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 and input with the **SGCWMIS** keyword. Then the mobile gas saturation is taken as:

$$S_g^* = \text{MAX}(S_g - S_{gc}, 0.0) \quad \text{Eq. 64.20}$$

The mobile oil and gas saturations S_o^* , S_g^* are 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 end-point scaling is used in conjunction with **SGCWMIS**, then S_{sgm} is replaced by the scaled value of **SGCR**.

Viscosity model

The full three hydrocarbon components mixing calculation can be thought of as two separate miscible displacements: gas/solvent and solvent/oil. The effective viscosities of the hydrocarbon components follow the Todd and Longstaff model:

$$\begin{aligned}\mu_{o \text{ eff}} &= \mu_o^{1-\omega} \cdot \mu_{mos}^\omega \\ \mu_{s \text{ eff}} &= \mu_s^{1-\omega} \cdot \mu_m^\omega \\ \mu_{g \text{ eff}} &= \mu_g^{1-\omega} \cdot \mu_{msg}^\omega\end{aligned}\quad \text{Eq. 64.21}$$

where

μ_o, μ_s, μ_g are the component 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_m are defined using the 1/4th-power fluid mixing rule, as follows:

$$\mu_{mos} = \frac{\mu_o \mu_s}{\left(\frac{S'_o}{S'_{os}} \mu_s^{1/4} + \frac{S'_s}{S'_{os}} \mu_o^{1/4} \right)^4} \quad \text{Eq. 64.22}$$

$$\mu_{msg} = \frac{\mu_s \mu_g}{\left(\frac{S'_s}{S'_{sg}} \mu_g^{1/4} + \frac{S'_g}{S'_{sg}} \mu_s^{1/4} \right)^4} \quad \text{Eq. 64.23}$$

$$\mu_m = \frac{\mu_o \mu_s \mu_g}{\left(\frac{S'_o}{S'_n} \mu_s^{1/4} \mu_g^{1/4} + \frac{S'_s}{S'_n} \mu_o^{1/4} \mu_g^{1/4} + \frac{S'_g}{S'_n} \mu_o^{1/4} \mu_s^{1/4} \right)^4} \quad \text{Eq. 64.24}$$

where the mobile saturations are defined by

$$S'_o = S_o - S_{or}$$

$$S'_g = S_g - S_{gc}$$

$$S'_s = S_s - S_{gs}$$

$$S'_n = S'_o + S'_g + S'_s,$$

$$S'_{os} = S'_o + S'_s \text{ and}$$

$$S'_{sg} = S'_s + S'_g$$

In the special case of no mixing, $\omega = 0$, the effective viscosities, equation 64.21 are equal to the pure phase viscosities, while in the case of full mixing $\omega = 1$ the effective viscosities are equal to the mixed viscosities defined by equations 64.22 - 64.24.

These effective viscosities are also used to compute the effective densities as detailed below.

Density model

The treatment of effective oil, gas and solvent densities is based on the fourth power law for the effective viscosities above. 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 effective viscosities are calculated using equations 64.21 to 64.24 above. The value of each effective component viscosity is then used in turn in equations 64.22 to 64.24 to yield an effective saturation fraction to be used in oil, gas and solvent density calculations as:

$$\left(\frac{S_o}{S_n}\right)_{oe} = \frac{\mu_o^{1/4}(\mu_{o\text{ eff}}^{1/4} - \mu_s^{1/4})}{\mu_{o\text{ eff}}^{1/4}(\mu_o^{1/4} - \mu_s^{1/4})} \quad \text{Eq. 64.25}$$

$$\left(\frac{S_o}{S_n}\right)_{ge} = \frac{\mu_s^{1/4}(\mu_g^{1/4} - \mu_g^{1/4})}{\mu_g^{1/4}(\mu_s^{1/4} - \mu_g^{1/4})} \quad \text{Eq. 64.26}$$

$$\left(\frac{S_s}{S_n}\right)_{se} = \frac{\mu_s^{1/4}(S_{gf}\mu_o^{1/4} + S_{of}\mu_g^{1/4}) - \mu_o^{1/4}\mu_g^{1/4}\left(\frac{\mu_s^{1/4}}{\mu_{s\text{ eff}}^{1/4}}\right)}{\mu_s^{1/4}(S_{gf}\mu_o^{1/4} + S_{of}\mu_g^{1/4}) - \mu_o^{1/4}\mu_g^{1/4}} \quad \text{Eq. 64.27}$$

where

$$S_{of} = S'_o / S'_{og}$$

$$S_{gf} = S'_g / S'_{og}$$

$$S'_{og} = S'_o + S'_g$$

The effective oil, gas and solvent densities ($\rho_{o\text{ eff}}, \rho_{g\text{ eff}}, \rho_{s\text{ eff}}$) are now computed from the effective saturation fractions in equations 64.25 to 64.27, and the pure component densities (ρ_o, ρ_g, ρ_s) using the following formulae

$$\rho_{o\text{ eff}} = \rho_o \left(\frac{S_o}{S_n} \right)_{oe} + \rho_s \left[1 - \left(\frac{S_o}{S_n} \right)_{oe} \right] \quad \text{Eq. 64.28}$$

$$\rho_{g\text{ eff}} = \rho_s \left(\frac{S_o}{S_n} \right)_{ge} + \rho_g \left[1 - \left(\frac{S_o}{S_n} \right)_{ge} \right] \quad \text{Eq. 64.29}$$

$$\rho_{s \text{ eff}} = \rho_s \left(\frac{S_s}{S_n} \right)_{se} + \rho_g S_{gf} \left[1 - \left(\frac{S_s}{S_n} \right)_{se} \right] + \rho_o S_{of} \left[1 - \left(\frac{S_s}{S_n} \right)_{se} \right] \quad \text{Eq. 64.30}$$

In the special case of no mixing, $\omega = 0$, the effective saturation fractions defined in equations 64.25 - 64.27 simplify to:

$$\left(\frac{S_o}{S_n} \right)_{oe} = 1, \left(\frac{S_o}{S_n} \right)_{ge} = 0, \left(\frac{S_s}{S_n} \right)_{se} = 1$$

so that the effective densities, equations 64.28 - 64.30 are equal to their pure phase values.

Note that equations 64.25 and 64.26 are singular for unit mobility ratio solvent displacements, $\mu_o = \mu_s$ or $\mu_g = \mu_s$. In that case, the effective densities are computed as follows. First a mixture density is defined as:

$$\rho_m = \rho_o \left(\frac{S_o}{S_n} \right) + \rho_g \left(\frac{S_g}{S_n} \right) + \rho_s \left(\frac{S_s}{S_n} \right) \quad \text{Eq. 64.31}$$

Then

$$\rho_{o \text{ eff}} = (1-\omega)\rho_o + \omega\rho_m \quad \text{Eq. 64.32}$$

$$\rho_{g \text{ eff}} = (1-\omega)\rho_g + \omega\rho_m \quad \text{Eq. 64.33}$$

$$\rho_{s \text{ eff}} = (1-\omega)\rho_s + \omega\rho_m$$

Where ω is the Todd and Longstaff parameter input with the [TLMIXPAR](#) keyword.

Pressure effect on viscosity and density miscibility

As for the relative permeability and capillary pressure, the pressure dependence of the transition between the immiscible and the miscible flow can be modeled for PVT data. This is achieved 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 as follows, with the gas component taken as an example:

$$\frac{1}{B_g} = \frac{1}{B_{gm}} M_p + \frac{1}{B_{gi}} (1 - M_p) \quad \text{Eq. 64.34}$$

$$\frac{1}{(B\mu)_g} = \frac{1}{(B\mu)_{gm}} M_p + \frac{1}{(B\mu)_{gi}} (1 - M_p) \quad \text{Eq. 64.35}$$

where

B_g is the effective 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 effective gas FVF \times viscosity

$(B\mu)_{gi}$ is the gas FVF \times viscosity (immiscible)

$(B\mu)_{gm}$ is the gas FVF \times viscosity (miscible)

M_p is the pressure dependent miscibility function entered using the PMISC keyword.

Note that for the relative permeability the interpolation can be a combination of both MISC and PMISC keywords.

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.

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 four-component model is used. The SOLVENT keyword should only be used with the other three components/phases (Water, Oil, Gas) present.

If the Solvent Model is run in immiscible mode, the mixing is not performed, and the four components take their pure PVT properties. The relative permeabilities are calculated as for a traditional black oil case, with the solvent and gas taking their respective fraction of the total ‘gas’ relative permeability.

PVT data

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)

These PVT data can be used with the mixing parameter ω , input in the PROPS section with the keyword TLMIXPAR as described in the "Viscosity model" and "Density model". If the grid contains cells whose MISCTNUM region is zero, this 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 MISCTNUM in the REGIONS section. The first argument NTMISC of the keyword MISCIBLE sets the maximum number of regions which can be defined.

Relative permeability data

The relative permeability data needs to be specified as follows:

SGFN For k_{rg} in immiscible regions (either **MISNUM** =0 or $S_g \gg 0$)

SWFN For any case containing water.

SOF2 Three-component or miscible case: cases with **MISNUM** =0 in at least one cell, **SOF2** gives k_{rog} .

Four-component case: **SOF2** gives k_{rhc} (k_r of the miscible hydrocarbon phase to water).

SOF3 Three-component cases: Three-phase cases with **MISNUM** =0 in at least one cell.

Four-component cases: immiscible displacement regions

SSFN In immiscible or miscible four-component cases.

In four-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.

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 (four-component)

PMISC Tables of the miscibility function versus oil pressure.

MISC is required for all four-component runs, but **PMISC** is optional. If **PMISC** is not specified, the solvent/oil displacement is assumed to be miscible at all pressures.

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 the fraction of the gas flow as solvent.

It is also possible to initialize the model by enumeration and to include a Solvent saturation. This is achieved using the **SSOL** keyword in the **SOLUTION** section.

Control of numerical diffusion in miscible flow simulation

The relative permeability of the oil and solvent components in the Todd- Longstaff model are specified by equation 64.4. The relative permeability of each component is essentially proportional to the saturation of each component for low saturations. In such cases the influence of numerical diffusion in spreading the miscible components can be unacceptably high and some control of the numerical diffusion errors is required.

A two point upstream algorithm, operating only on the miscible component relative permeabilities, is available for use in the ECLIPSE Miscible Flood option to control the numerical diffusion of oil and solvent. In the present version of ECLIPSE the two-point upsteaming algorithm is only available for three-phase cases.

The two-point upstream projection technique is illustrated in the following figure.

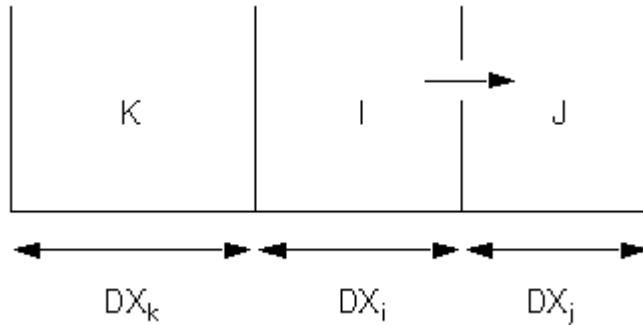


Figure 64.1. The two point upstream projection technique

We need to compute the flow of component C between cells I and J which we take to be aligned along the X-axis. Suppose that cell I is the upstream cell for the flow of component C between I and J. Let cell K be the second upstream cell to cell J and K_{ck} , K_{ci} and K_{cj} denote the component relative permeabilities in cells K, I and J respectively. DX_k , DX_i and DX_j are the X-direction cell sizes.

For single point upstream weighting, the flow rate of component C across the I-J cell interface is $F_{ij,c}$ where

$$F_{ij,c} = T_{ij} \cdot \frac{K_{ci}}{B_{ci} \mu_{ci}} \cdot \delta\Phi_{ij,c} \quad \text{Eq. 64.36}$$

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 upsteaming algorithm operating on component relative permeabilities replaces K_{ci} in the equation 64.36 by a linearly extrapolated relative permeability, K_{ci}^* , obtained from cells I and K,

$$K_{ci}^* = \frac{2DX_i + DX_k}{DX_i + DX_k} K_{ci} - \frac{DX_i}{DX_i + DX_k} K_{ck} \quad \text{Eq. 64.37}$$

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,

$$\text{MIN}(K_{ci}, K_{cj}) < K_{ci}^* < \text{MAX}(K_{ci}, K_{cj}) \quad \text{Eq. 64.38}$$

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

- Cell K lies outside the reservoir mesh (that is the K-I cell boundary is an edge of the reservoir)
- Cell K is inactive
- The K-I cell boundary contains a zero transmissibility multiplier
- Cells I and J are coupled by a non-neighbor connection.

Summary of keywords

RUNSPEC section

Keyword	Description
SOLVENT	Activates the separate solvent component.
MISCIBLE	Initiates the mixing calculation.

PROPS section

Keyword	Description
MISC	Solvent fraction dependent miscibility 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. <ul style="list-style-type: none"> • MISC or PMISC outputs the miscibility function and the pressure miscibility function. • PVDS outputs solvent PVT properties. • SSFN outputs solvent/gas relative permeabilities.
SDENSITY	Solvent density at surface conditions.
SGCWMIS	Miscible critical gas saturation tables versus water saturation.
SORWMIS	Miscible residual oil saturation tables versus water saturation.
SSFN	Gas/solvent relative permeability functions.
TLMIXPAR	Todd-Longstaff mixing parameter.

SOLUTION section

Keyword	Description
RPTSQL	Controls on output from SOLUTION section. <ul style="list-style-type: none"> FIPSOL outputs solvent fluid in place reports. FLOSOL outputs solvent inter-block flow rates. KRN outputs grid block solvent relative permeabilities SSOL outputs initial solvent saturations.
SSOL	Initial solvent saturations.

SUMMARY section

Refer to [Solvent model](#) in the *ECLIPSE Reference Manual*.

SCHEDULE section

Keyword	Description
RPTSCHE	Controls on output from SCHEDULE section. <ul style="list-style-type: none"> FIPSOL outputs solvent fluid in place reports. FLOSOL outputs solvent inter-block flow rates. KRN outputs grid block solvent relative permeabilities. SSOL outputs current solvent saturations.
WSOLVENT	Sets solvent concentrations for gas injection wells.

Example

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'   /
/
COMPDATA
  '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 /
/
/

WEOPEN
  'INJ-G' 'SHUT' /
  'INJ-W' 'OPEN' /
/
TSTEP
  91.25
/
-- SECOND 3 MONTHS -----
-- *GAS*--

TUNING

```

```
2.0 /
/
/
WEOPEN
'INJ-W' 'SHUT' /
'INJ-G' 'OPEN' /
/
TSTEP
91.25
/
--THIRD 3 MONTHS -----
--*WATER*--
TUNING
2.0 /
/
WEOPEN
'INJ-G' 'SHUT' /
'INJ-W' 'OPEN' /
/
TSTEP
91.25
/
-- FOURTH 3 MONTHS -----
--*GAS*--
TUNING
2.0 /
/
WEOPEN
'INJ-W' 'SHUT' /
'INJ-G' 'OPEN' /
/
TSTEP
91.25
/
```

65

Surface tension effects

Miscibility

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The surface tension between the oil and gas phases can be used as a measure of how miscible the two fluids are. The ECLIPSE 300 MISCIBLE model follows that of [Ref. 23], where parameters that are functions of surface tension are used to interpolate properties such as relative permeability and capillary pressure between immiscible and miscible values:

$$K_{ro} = F_k \cdot K_{ro}^{imm} + (1-F_k) \cdot K_{ro}^{mis} \quad \text{Eq. 65.1}$$

$$K_{rg} = F_k \cdot K_{rg}^{imm} + (1-F_k) \cdot K_{rg}^{mis} \quad \text{Eq. 65.2}$$

$$P_{cog} = F_p \cdot P_{cog}^{imm} \quad \text{Eq. 65.3}$$

K_{ro}^{imm} , K_{rg}^{imm} and P_{cog}^{imm} are the immiscible oil and gas relative permeabilities and capillary pressure obtained from the user-defined rock curves (for example [SGFN](#)). The miscible capillary pressure is taken to be zero.

K_{ro}^{mis} and K_{rg}^{mis} are the miscible oil and gas relative permeabilities. By default these are taken to be straight lines, for example $K_{ro}^{mis} = S_o$, although user-defined rock curves can be selected by using the [MISCNUM](#) keyword. The miscible capillary pressure is taken to be zero.

The interpolation parameters F_k and F_p are functions of the surface tension σ between the oil and gas phases. For high surface tension the parameters tend to 1 and the flow becomes immiscible. For low surface tension the parameters tend to 0 and the flow becomes miscible.

In order to add the relative permeability curves sensibly, the critical saturations must also be scaled so that both the immiscible and miscible relative permeabilities have the same endpoints:

$$S_{ocr} = F_k \cdot S_{ocr}^{imm} + (1-F_k) \cdot S_{ocr}^{mis} \quad \text{Eq. 65.4}$$

$$S_{gcr} = F_k \cdot S_{gcr}^{imm} + (1-F_k) \cdot S_{gcr}^{mis} \quad \text{Eq. 65.5}$$

S_{ocr}^{imm} and S_{gcr}^{imm} are the critical immiscible saturations, generally obtained from the user-defined rock curves, though in the case of hysteresis they may be the critical point of a scanning curve (see "Hysteresis" for further information).

The critical saturations for miscible flow S_{ocr}^{mis} and S_{gcr}^{mis} are by default zero (unless the [MISCNUM](#) keyword is used), so these equations simplify to:

$$S_{ocr} = F_k \cdot S_{ocr}^{imm} \quad \text{Eq. 65.6}$$

$$S_{gcr} = F_k \cdot S_{gcr}^{imm} \quad \text{Eq. 65.7}$$

The relative permeabilities defined by equations 65.1 and 65.2 are two phase (oil-gas) relative permeabilities. When there are three phases present, these values are used to calculate three phase relative permeabilities, as described in "[Saturation Functions](#)".

Note the similarity between this model and that provided by the Velocity Dependent Relative Permeability option: see "[Velocity-dependent relative permeabilities](#)". Note also that the length of timestepping is of great importance in simulating surface tension effects.

Relative permeability interpolation parameter

The relative permeability interpolation parameter $F_k(\sigma)$ can either be entered as a table using the [FVST](#) keyword, or by using Coats's formula:

$$F_k = \min \left[1, \left(\frac{\sigma}{\sigma_0} \right)^N \right] \quad \text{Eq. 65.8}$$

where

σ_0 is the reference surface tension, described in "[Reference surface tension](#)",

N is the exponent set by keyword [MISCEXP](#).

Therefore the relative permeabilities defined in equations 65.1 and 65.2 approach the miscible values $K_r \rightarrow K_r^{mis}$ as the surface tension tends to zero, $\sigma \rightarrow 0$, and equal the immiscible values $K_r = K_r^{imm}$ when the surface tension is greater than or equal to the reference surface tension $\sigma \geq \sigma_0$.

For single-phase cells, a value of 1 is used for the interpolation parameter.

Values of F_k can be output to the print file or restart file using the argument FMISC with the [RPTSL](#), [RPTSCHE](#), or [RPTRST](#) keywords. Block values can be output to the summary file using the BFMISC keyword.

Capillary pressure interpolation parameter

The capillary pressure interpolation parameter $F_p(\sigma)$ is given as a linear function of the surface tension:

$$F_p = \min \left[\frac{\sigma_1}{\sigma_0}, \frac{\sigma}{\sigma_0} \right] \quad \text{Eq. 65.9}$$

where σ_1 is the maximum surface tension for capillary pressure calculation, which defaults to $\sigma_1 = \sigma_0$ but can be set larger than this value using the third data item in the [MISCSTR](#) or [MISCSTRR](#) keyword.

Therefore, the capillary pressure defined in equation 65.3 equals the miscible value of zero $P_{cog} = 0$ when the surface tension is zero, $\sigma = 0$, and equals the immiscible value $P_{cog} = P_{cog}^{imm}$ when the surface tension is equal to the reference surface tension $\sigma = \sigma_0$.

For surface tensions greater than the reference surface tension $\sigma \geq \sigma_0$, the capillary pressure can be extrapolated beyond the immiscible values, with the interpolation parameter being allowed to exceed 1:

$$F_p \leq \frac{\sigma_1}{\sigma_0}$$

For single-phase cells a value of 1 will be used for the interpolation parameter.

Values of F_p can be output to the print file or restart file using argument FPC with the [RPTSOL](#), [RPTSCHE](#), or [RPTRST](#) keywords.

Evaluation of surface tensions

Surface tensions can either be entered as a function of pressure using the [STVP](#) keyword, or can be obtained as functions of the oil and gas compositions using the Macleod-Sugden correlation:

$$\sigma = \left[\sum_{i=1}^{N_c} [P]_i (b_L^m x_i - b_V^m y_i) \right]^4 \quad \text{Eq. 65.10}$$

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 gm-M/cc for this evaluation.

the parachors $[P]_i$ should be provided by the [PARACHOR](#) keyword.

For one-phase states the surface tension is zero. The value of σ will go sharply to zero as the phase compositions and phase molar densities become equal, such as occurs near the critical point.

Warnings of un-physically large values of surface tension can be requested using the second data item in the [MISCSTR](#) or [MISCSTRR](#) keyword.

Values of surface tension can be output to the print file or restart file using the argument STEN with the [RPTSOL](#), [RPTSCHE](#), or [RPTRST](#) keywords. Block values can be output to the summary file using the BSTEN keyword.

Reference surface tension

The reference surface tension σ_0 can be defined or calculated in various ways:

- By evaluating the Macleod-Sugden correlation, equation 65.10 at a reference pressure, specified either by the [MISCSTRR](#) or [MISCSTRP](#) keyword.
- By specifying a value directly, using either the MISCSTRR or MISCSTR keyword.

- By taking the maximum value from the [STVP](#) table.
- By evaluating the Macleod-Sugden correlation equation [65.10](#) at the saturation pressure in each grid block.

Use of the MISCTNUM 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. 10] that relative permeabilities may be significantly different from straight lines even at very low surface tensions.

To model such cases the [MISCTNUM](#) 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 MISCTNUM 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 MISCTNUM 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.

66

Surfactant model

x	ECLIPSE 100
x	ECLIPSE 300
	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

ECLIPSE 100

In ECLIPSE 100 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 timestep 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).

ECLIPSE 300

In ECLIPSE 300 surfactant is modeled as a water component as described in "[Water components](#)". A water component must be identified as a surfactant (see the [CWTYPE](#) keyword) and can be input into the reservoir by defining the injection water composition (see the [WINJW](#) keyword).

Surfactant can be adsorbed on to the rock. The ECLIPSE 100 keywords are described in "Treatment of adsorption". For ECLIPSE 300 a solid component can be used to represent adsorbed surfactant, and adsorption and desorption can be modeled using a chemical reaction. See "Chemical reactions" and "Solid phase".

Surfactant effects

The presence of surfactant can affect reservoir performance in three different ways:

- The surfactant modifies the oil-water surface tension (see `SURFST` keyword or `SURFSTES` if `BRINE` option is activated). This affects the capillary pressure directly as described in "Capillary pressure" and the oil and water relative permeabilities as described in "Relative permeability model", via the capillary number.
- The surfactant can modify the water properties. This is described in "Water PVT properties" for ECLIPSE 100, or in "Water component properties" for ECLIPSE 300.
- The adsorbed surfactant can affect the wettability of the rock. This is described in "Modeling the change to wettability" for ECLIPSE 100, or in "Solid wettability affects" for ECLIPSE 300.

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:

$$N_c = \frac{C_N}{C_D} \left\| \frac{T \Delta P_o}{A} \right\| \frac{1}{\sigma_{ow}} = C_N \frac{\| K \cdot \nabla P_o \|}{\sigma_{ow}} \quad \text{Eq. 66.1}$$

where

- | | |
|---------------|--|
| T | is the transmissibility number |
| A | is the flow cross-sectional area |
| K | is the permeability |
| P_o | is the potential |
| σ_{ow} | is the interfacial tension (see the <code>SURFST</code> or <code>SURFSTES</code>) |
| C_D | is the Darcy Constant |
| C_N | is a conversion factor depending on the units used. |

The calculation of the capillary number is described in [Capillary number](#)

Relative permeability model

The Surfactant Relative Permeability Model allows a transition from immiscible relative permeability curves at low capillary number to miscible relative permeability curves at high capillary number. The `SURFCAPD` keyword is used to define an interpolation parameter F_{kr} as a tabulated function of the logarithm (base 10) of the capillary number, $F_{kr} = F_{kr}(\log N_c)$.

The relative permeability used at a value of the miscibility function is calculated in two steps, see figure 66.1. Firstly the saturation end points of the curves are interpolated and both the immiscible and the miscible curves are scaled using F_{kr} to honor these points. The relative permeability values are looked up on both curves as functions of the current grid block saturation, and the final relative permeability is taken as an interpolation between these two values using F_{kr} . This procedure is used to calculate the water relative permeability k_{rw} and the oil-to-water relative permeability k_{row} .

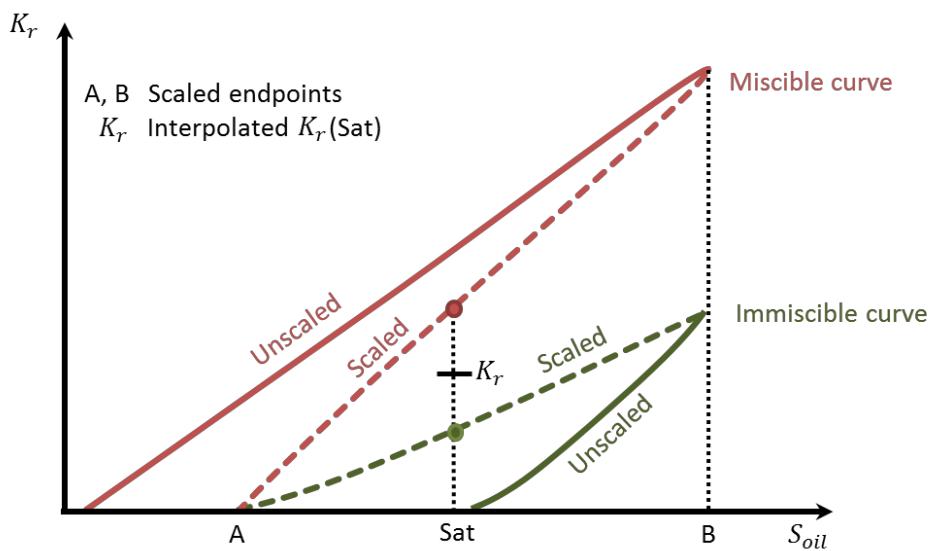


Figure 66.1. Calculation of the relative permeability

In the three-phase case, the oil relative permeability to water k_{row} is then used with either of the two STONE's three-phase relative permeability models or the default model to calculate the oil relative permeability k_{ro} .

ECLIPSE 100

For the default three-phase oil relative permeability model, both the oil relative permeability to water and the oil relative permeability to gas are interpolated in the manner described above before being combined to form the three-phase oil relative permeability.

The well relative permeability model 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 `COMPRT` keyword is used, then that relative permeability table will be used with no modification to account for the surfactant present in the grid block.

The `LOWSALT` keyword can be used in conjunction with the Surfactant Option to model additionally the effects of salinity dependence. See "[Brine tracking](#)" for more information upon the `LOWSALT` option.

ECLIPSE 300

In compositional cases, there is an extra step in calculating the oil relative permeability. The oil-to-water relative permeability k_{row} value is determined as above from the miscible and immiscible values. It is then combined with the gas-to-water relative permeability k_{rgw} to determine a hydrocarbon-to-water relative permeability k_{rhw} . See "[Near critical oil and gas relative permeabilities](#)". The hydrocarbon-to-water relative permeability k_{rhw} is then used in one of the three-phase relative permeability models to calculate k_{ro} .

The Surfactant Model does not currently support the modeling of relative permeability hysteresis in the water and oil phases.

Surfactant model in cells with no surfactant

The capillary number depends on both the velocity and the oil-water surface tension. Therefore, the capillary number, and hence F_{kr} , can vary even when there is no surfactant present. This means that when the surfactant model is used, relative permeability interpolation may occur even if surfactant is not injected.

ECLIPSE 100

This behavior is avoided in ECLIPSE 100. The interpolation of relative permeability is only performed for blocks with a non-zero surfactant concentration. For blocks with zero surfactant concentration the immiscible curves are used. The pre-2010.1 behavior, which is the same as ECLIPSE 300 behavior, of interpolating relative permeability for all blocks, regardless of surfactant concentration, can be restored by setting item 147 of the OPTIONS keyword to a non-zero value.

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:

$$P_{cow} = F_{cp} \cdot P_{cow}(S_w) \quad \text{Eq. 66.2}$$

where

$$F_{cp} = \frac{\sigma_{ow}(C_{surf})}{\sigma_{ow}(0)} \quad \text{is the capillary pressure multiplier}$$

$\sigma_{ow}(C_{surf})$ is the surface tension at the present surfactant concentration,
determined from the table supplied by SURFST keyword,

$\sigma_{ow}(0)$ is the surface tension at zero concentration
determined from the table supplied by SURFST keyword, and

$P_{cow}(S_w)$ is the capillary pressure from the immiscible curves initially scaled to the interpolated end-points calculated in the relative permeability model.

Note: The scaling of the immiscible capillary pressure to honor the interpolated endpoints calculated in the relative permeability model, was not done in versions before 2009.1. Item 119 of the OPTIONS keyword can be used to restore the pre-2009.1 calculations.

Note: If a Leverett J-function is used (JFUNC or JFUNCR), the immiscible capillary pressure is proportional to a user-defined constant surface tension σ_{Jow} . If this surface tension is set equal to the zero concentration surface tension, $\sigma_{Jow} = \sigma_{ow}(0)$ then when the immiscible capillary pressure is multiplied by F_{cp} , the resulting capillary pressure will be equal to that specified by the J-function, but with the surface tension $\sigma_{ow}(C_{surf})$.

If the Brine option is active (and the Polymer option is inactive), the capillary pressure multiplier becomes a function of salt concentration c_s as well:

$$F_{cp} = \frac{\sigma_{ow}(C_{surf}, c_s)}{\sigma_{ow}(0, c_s)}$$

where

$\sigma_{ow}(C_{surf}, c_s)$ is the surface tension at the present surfactant and salt concentrations

$\sigma_{ow}(0, c_s)$ is the surface tension at zero surfactant concentration and present salt concentration.

The surface tension values are determined from the two dimensional table supplied by [SURFSTES](#) and [ESSNODE](#).

Water PVT properties (ECLIPSE 100)

The surfactant modifies the viscosity of the pure or salted water input using the [PVTW](#) or the [PVTWSALT](#) keyword respectively. The surfactant viscosity input as a function of surfactant concentration using the [SURFVISC](#) keyword is used to calculate the water-surfactant solution viscosity as follows:

$$\mu_{ws}(C_{surf}, P) = \mu_w(P) \frac{\mu_s(C_{surf})}{\mu_w(P_{ref})} \quad \text{Eq. 66.3}$$

If the brine option is active (and the polymer option is inactive), the previous equation becomes a function of salt concentration c_s as well:

$$\mu_{ws}(C_{surf}, P, c_s) = \mu_w(P, c_s) \cdot \frac{\mu_s(C_{surf})}{\mu_w(P_{ref}, c_{sref})} \quad \text{Eq. 66.4}$$

where

μ_s is the viscosity from the [SURFVISC](#) keyword

μ_w is the viscosity from the [PVTW](#) or [PVTWSALT](#) keywords

μ_{ws} is the viscosity of the water-surfactant mixture

P_{ref} is the reference pressure in the [PVTW](#) or [PVTWSALT](#) keywords

c_{sref} is the reference salt concentration in the [PVTWSALT](#) keyword.

Note that the [SUMMARY](#) keyword [BEWV_SUR](#) should be used to output the effective water viscosity due to the surfactant concentration.

Treatment of adsorption (ECLIPSE 100)

The adsorption of surfactant is assumed to be instantaneous, and the quantity adsorbed is a function of the surrounding surfactant concentration. The user is required to supply an adsorption isotherm as a function of surfactant concentration (see the [SURFADS](#) keyword). As an alternative to [SURFADS](#), the surfactant adsorption can be specified using keyword [ADSORP](#) with [SURFACT](#) as an argument. [ADSORP](#) specifies coefficients for a generic, analytical adsorption isotherm which allows for dependencies of adsorption on rock permeability and effective brine salinity. When used in conjunction with the Multi-Component Brine Model, an effective salinity for surfactant can be calculated which can then be used in the adsorption

isotherm. See "Brine Tracking" for information on the Multi-Component Brine Model and the calculation of effective salinity for surfactant. See the ADSORP keyword for information on the functional dependence of adsorption on salinity.

The quantity of surfactant adsorbed on to the rock is given by:

$$\text{Mass of adsorbed surfactant} = \text{PORV} \cdot \frac{1-\varphi}{\varphi} \cdot \text{MD} \cdot \text{CA}(C_{\text{surf}}) \quad \text{Eq. 66.5}$$

where

PORV is the pore volume of the cell

φ is the porosity

MD is the mass density of the rock (see [SURFROCK](#) keyword)

CA(C_{surf}) is the adsorption isotherm as a function of local surfactant concentration in solution.

There are two adsorption models that can be selected, using the first argument of SURFROCK. 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 changes in wettability due to surfactant adsorption (ECLIPSE 100)

This option models changes in the rock wettability due to the accumulation of surfactant by adsorption. It is activated by specifying the [SURFACTW](#) keyword, which also activates the Surfactant Model ([SURFACT](#) keyword).

The [SATNUM](#) and [SURFWNUM](#) keywords are used to define the oil-wet and water-wet immiscible saturation regions respectively and hence the associated saturation tables. (The [HWSNUM](#) keyword may be used as an abbreviated alias for [SURFWNUM](#).) The keyword [SURFNUM](#) is used to define the miscible saturation regions. Given two sets of saturation functions, one for immiscible oil-wet and one for immiscible water-wet, the immiscible water and oil table saturation end-points are interpolated firstly according to:

$$\begin{aligned} S_{wco}^{imm} &= F_4 S_{wco}^{ow} + (1-F_4) S_{wco}^{ww} \\ S_{wcr}^{imm} &= F_4 S_{wcr}^{ow} + (1-F_4) S_{wcr}^{ww} \\ S_{wmax}^{imm} &= F_4 S_{wmax}^{ow} + (1-F_4) S_{wmax}^{ww} \\ S_{owcr}^{imm} &= F_4 S_{owcr}^{ow} + (1-F_4) S_{owcr}^{ww} \\ S_{ogcr}^{imm} &= F_4 S_{ogcr}^{ow} + (1-F_4) S_{ogcr}^{ww} \end{aligned} \quad \text{Eq. 66.6}$$

and these are then combined with the miscible table saturation end-points according to:

$$\begin{aligned}
 S_{wco}^i &= F_3 S_{wco}^{mis} + (1-F_3) S_{wco}^{imm} \\
 S_{wcr}^i &= F_3 S_{wcr}^{mis} + (1-F_3) S_{wcr}^{imm} \\
 S_{wmax}^i &= F_3 S_{wmax}^{mis} + (1-F_3) S_{wmax}^{imm} \\
 S_{owcr}^i &= F_3 S_{owcr}^{mis} + (1-F_3) S_{owcr}^{imm} \\
 S_{ogcr}^i &= F_3 S_{ogcr}^{mis} + (1-F_3) S_{ogcr}^{imm}
 \end{aligned} \tag{Eq. 66.7}$$

where

F_3 is tabulated as a function of the capillary number (expressed in terms of its logarithm base ten) and corresponds to the second column of the [SURFCAPD](#) keyword.

F_4 is tabulated as a function of the adsorbed surfactant concentration and corresponds to the second column of the [SURFADDW](#) keyword.

S_{wco} is the saturation table derived connate water saturation.

S_{wcr} is the saturation table derived critical water saturation.

S_{wmax} is the saturation table derived maximum water saturation.

S_{owcr} is the saturation table derived critical oil saturation in water.

S_{ogcr} is the saturation table derived critical oil saturation in gas.

superscripts *ow*, *ww* denote the immiscible oil-wet and water-wet immiscible table derived saturation end-points respectively.

superscripts *imm*, *mis* denote the immiscible and miscible table derived saturation end-points respectively.

superscript *i* denotes the interpolation between the immiscible and miscible table derived saturation end-points.

The immiscible oil-wet and water-wet capillary pressures and relative permeabilities are looked up in the immiscible oil-wet and water-wet saturation tables by applying two-point saturation (horizontal) end-point scaling using the interpolated saturation end-points:

$$\begin{aligned}
 P_{cow}^{ow} &= f(S_w, S_{wco}^i, S_{wmax}^i, P_{cowmax}^{ow}) \\
 P_{cow}^{ww} &= f(S_w, S_{wco}^i, S_{wmax}^i, P_{cowmax}^{ww}) \\
 k_{rw}^{ow} &= f(S_w, S_{wcr}^i, S_{wmax}^i, k_{rwmax}^{ow}) \\
 k_{rw}^{ww} &= f(S_w, S_{wcr}^i, S_{wmax}^i, k_{rwmax}^{ww}) \\
 k_{rog}^{ow} &= f(S_{og}, S_{ogcr}^i, S_{omax}, k_{romax}^{ow}) \\
 k_{rog}^{ww} &= f(S_{og}, S_{ogcr}^i, S_{omax}, k_{romax}^{ww}) \\
 k_{row}^{ow} &= f(S_{ow}, S_{owcr}^i, S_{omax}, k_{romax}^{ow}) \\
 k_{row}^{ww} &= f(S_{ow}, S_{owcr}^i, S_{omax}, k_{romax}^{ww})
 \end{aligned} \tag{Eq. 66.8}$$

The miscible relative permeabilities are looked up in the miscible tables by applying two-point saturation (horizontal) end-point scaling using the interpolated saturation end-points:

$$\begin{aligned}
 k_{rw}^{mis} &= f(S_w, S_{wcr}^i, S_{wmax}^i, k_{rwmax}^{mis}) \\
 k_{rog}^{mis} &= f(S_{og}, S_{ogcr}^i, S_{omax}, k_{romax}^{mis}) \\
 k_{row}^{mis} &= f(S_{ow}, S_{owcr}^i, S_{omax}, k_{romax}^{mis})
 \end{aligned} \tag{Eq. 66.9}$$

where

- f signifies the end-point scaling operation.
- S_w is the grid block water saturation.
- S_{ow} is the oil saturation for oil relative permeability to water.
- S_{og} is the oil saturation for oil relative permeability to gas.
- S_{omax} is the maximum oil saturation, $S_{omax} = (1 - S_{wco}^i - S_{gco}^{ow})$.
- S_{gco} is the saturation table derived connate gas saturation.
- k_{rwmax} is the saturation table derived maximum water relative permeability.
- k_{romax} is the saturation table derived maximum oil relative permeability.
- P_{cowmax} is the saturation table derived maximum oil-water capillary pressure.

For the default three-phase oil relative permeability model, the oil saturations used to look up the oil relative permeabilities are:

$$\begin{aligned}
 S_{ow} &= (1 - S_w - S_g) \\
 S_{og} &= (1 - S_w - S_g)
 \end{aligned} \tag{Eq. 66.10}$$

For the Stone 1 and Stone 2 three-phase oil relative permeability models, the oil saturations used to look up the oil relative permeabilities are:

$$\begin{aligned} S_{ow} &= (1 - S_w) \\ S_{og} &= (1 - S_g - S_{wco}^i) \end{aligned} \quad \text{Eq. 66.11}$$

The immiscible oil-wet and water-wet relative permeabilities and capillary pressure are interpolated according to:

$$\begin{aligned} k_{rw}^{imm} &= F_4 k_{rw}^{ow} + (1-F_4)k_{rw}^{ww} \\ k_{row}^{imm} &= F_4 k_{row}^{ow} + (1-F_4)k_{row}^{ww} \\ k_{rog}^{imm} &= F_4 k_{rog}^{ow} + (1-F_4)k_{rog}^{ww} \\ P_{cow}^{imm} &= F_4 P_{cow}^{ow} + (1-F_4)P_{cow}^{ww} \end{aligned} \quad \text{Eq. 66.12}$$

The immiscible and miscible relative permeabilities are interpolated according to:

$$\begin{aligned} k_{rw}^i &= F_3 k_{rw}^{mis} + (1-F_3)k_{rw}^{imm} \\ k_{row}^i &= F_3 k_{row}^{mis} + (1-F_3)k_{row}^{imm} \\ P_{cow}^i &= P_{cow}^{imm} \end{aligned} \quad \text{Eq. 66.13}$$

Default three-phase oil relative permeability model:

$$k_{rog}^i = F_3 k_{rog}^{mis} + (1-F_3)k_{rog}^{imm} \quad \text{Eq. 66.14}$$

Stone 1 and 2 three-phase oil relative permeability models:

$$k_{rog}^i = k_{rog}^{imm} \quad \text{Eq. 66.15}$$

The water-oil capillary pressure is only interpolated between the immiscible oil-wet and water-wet values. The gas saturation functions are looked up in the immiscible oil-wet table; that is, the gas relative permeability and gas-oil capillary pressure are assumed to exhibit no dependence upon surfactant concentration or surfactant adsorption.

When end-point scaling is specified via the `ENDSCALE` keyword, the interpolations follow a similar pattern to that described above in the absence of end-point scaling but noting that no end-point scaling is applied to the miscible saturation tables. Hence given two sets of immiscible water and oil saturation functions, one for oil-wet and one for water-wet, the scaled saturation end points are interpolated firstly according to:

$$\begin{aligned} S_{WL}^{IMM} &= F_4 S_{WL}^{OW} + (1 - F_4)S_{WL}^{WW} \\ S_{WLPC}^{IMM} &= F_4 S_{WLPC}^{OW} + (1 - F_4)S_{WLPC}^{WW} \\ S_{WCR}^{IMM} &= F_4 S_{WCR}^{OW} + (1 - F_4)S_{WCR}^{WW} \\ S_{WU}^{IMM} &= F_4 S_{WU}^{OW} + (1 - F_4)S_{WU}^{WW} \\ S_{OWCR}^{IMM} &= F_4 S_{OWCR}^{OW} + (1 - F_4)S_{OWCR}^{WW} \\ S_{OGCR}^{IMM} &= F_4 S_{OGCR}^{OW} + (1 - F_4)S_{OGCR}^{WW} \end{aligned} \quad \text{Eq. 66.16}$$

and these are then combined with the miscible table saturation end-points according to:

$$\begin{aligned}
 S_{WL}^I &= F_3 S_{wco}^{mis} + (1-F_3) S_{WL}^{IMM} \\
 S_{WCR}^I &= F_3 S_{wcr}^{mis} + (1-F_3) S_{WCR}^{IMM} \\
 S_{WU}^I &= F_3 S_{wmax}^{mis} + (1-F_3) S_{WU}^{IMM} \\
 S_{OWCR}^I &= F_3 S_{owcr}^{mis} + (1-F_3) S_{OWCR}^{IMM} \\
 S_{OGCR}^I &= F_3 S_{ogcr}^{mis} + (1-F_3) S_{OGCR}^{IMM}
 \end{aligned} \tag{Eq. 66.17}$$

where

S_{WL}	is the scaled minimum water saturation for relative permeability.
S_{WLPC}	is the scaled minimum water saturation for oil-water capillary pressure.
S_{WCR}	is the scaled critical water saturation.
S_{WU}	is the scaled maximum water saturation.
S_{OWCR}	is the scaled critical oil saturation in water.
S_{OGCR}	is the scaled critical oil saturation in gas.

superscripts *IMM*, *MIS* denote the immiscible interpolated and the miscible scaled saturation end-points respectively.

superscript *I* denotes the interpolation between the immiscible and miscible scaled saturation end-points.

Lower and upper case subscripts and superscripts are used here to signify the table derived and the scaled end-points respectively.

The oil-wet immiscible saturation tables are scaled using the set of end-point scaling keywords consisting of: [SWL](#), [SWLPC](#), [SWCR](#), [SWU](#), [SGL](#), [SGLPC](#), [SGCR](#), [SGU](#), [SOWCR](#), [SOGCR](#), [KRW](#), [KRWR](#), [KRG](#), [KRGR](#), [KRO](#), [KRORW](#), [KRORG](#), [PCW](#) and [PCG](#). (For versions prior to 2014.1, this set of end-points was also used for the water-wet immiscible saturation tables. For the 2014.1 version, pre-2014.1 behavior may be restored by setting item 225 of the [OPTIONS](#) keyword to 1.)

The water-wet immiscible saturation tables are scaled using the reduced set of end-point scaling keywords consisting of [HWSWL](#), [HWSLPC](#), [HWSWCR](#), [HWSWU](#), [HWSOWCR](#), [HWSOGCR](#), [HWKRW](#), [HWKRWR](#), [HWKRO](#), [HWKRORW](#), [HWKRORG](#) and [HWPW](#). This set does not include any gas end-points because only the oil-wet immiscible saturation functions are used for gas relative permeability and for oil-gas capillary pressure.

The capillary pressure and relative permeabilities are looked up in the immiscible oil-wet and water-wet saturation tables using the sets of scaled end-points shown below. For capillary pressure, only two-point scaling is employed. For the immiscible relative permeabilities, two-point or three-point end-point scaling is employed according to whether item 1 of the [SCALECRS](#) keyword has been set to NO or YES respectively. (Two-point scaling is selected by default if this keyword is omitted.) For the miscible relative permeabilities, only two-point scaling is applied because no miscible scaled saturation end-points are employed. For the immiscible oil relative permeability to water and for three-point scaling, the critical water saturation end-point is taken from the interpolated value or the respective oil-wet and water-wet values according to whether the default or the Stone 1 or 2 three-phase oil relative permeability models has been selected.

The immiscible oil-wet and water-wet oil-water capillary pressures are calculated according to:

$$\begin{aligned} P_{COW}^{OW} &= f(S_W, S_{WLPC}^I, S_{WU}^I, P_{CW}^{OW}) \\ P_{COW}^{WW} &= f(S_W, S_{WLPC}^I, S_{WU}^I, P_{CW}^{WW}) \end{aligned} \quad \text{Eq. 66.18}$$

For two-point scaling the immiscible oil-wet and water-wet relative permeabilities are calculated according to:

$$\begin{aligned} K_{RW}^{OW} &= f(S_W, S_{WCR}^I, S_{owcr}^{ow}, S_{WU}^I, K_{RW}^{OW}, K_{RWR}^{OW}) \\ K_{RW}^{WW} &= f(S_W, S_{WCR}^I, S_{owcr}^{ww}, S_{WU}^I, K_{RW}^{WW}, K_{RWR}^{WW}) \\ K_{ROG}^{OW} &= f(S_{OG}, S_{OGCR}^I, S_{gcr}^{ow}, S_{OMAX}, K_{RO}^{OW}, K_{RORG}^{OW}) \\ K_{ROG}^{WW} &= f(S_{OG}, S_{OGCR}^I, S_{gcr}^{ow}, S_{OMAX}, K_{RO}^{WW}, K_{RORG}^{WW}) \\ K_{ROW}^{OW} &= f(S_{OW}, S_{OWCR}^I, S_{wcr}^{ow}, S_{OMAX}, K_{RO}^{OW}, K_{RORW}^{OW}) \\ K_{ROW}^{WW} &= f(S_{OW}, S_{OWCR}^I, S_{wcr}^{ww}, S_{OMAX}, K_{RO}^{WW}, K_{RORW}^{WW}) \end{aligned} \quad \text{Eq. 66.19}$$

For three-point scaling the immiscible oil-wet and water-wet relative permeabilities are calculated according to:

$$\begin{aligned} K_{RW}^{OW} &= f(S_W, S_{WCR}^I, S_{OWCR}^I, S_{WU}^I, K_{RW}^{OW}, K_{RWR}^{OW}) \\ K_{RW}^{WW} &= f(S_W, S_{WCR}^I, S_{OWCR}^I, S_{WU}^I, K_{RW}^{WW}, K_{RWR}^{WW}) \\ K_{ROG}^{OW} &= f(S_{OG}, S_{OGCR}^I, S_{GCR}^{OW}, S_{OMAX}, K_{RO}^{OW}, K_{RORG}^{OW}) \\ K_{ROG}^{WW} &= f(S_{OG}, S_{OGCR}^I, S_{GCR}^{OW}, S_{OMAX}, K_{RO}^{WW}, K_{RORG}^{WW}) \end{aligned} \quad \text{Eq. 66.20}$$

Default three-phase oil relative permeability model:

$$\begin{aligned} K_{ROW}^{OW} &= f(S_{OW}, S_{OWCR}^I, S_{WCR}^I, S_{OMAX}, K_{RO}^{OW}, K_{RORW}^{OW}) \\ K_{ROW}^{WW} &= f(S_{OW}, S_{OWCR}^I, S_{WCR}^I, S_{OMAX}, K_{RO}^{WW}, K_{RORW}^{WW}) \end{aligned} \quad \text{Eq. 66.21}$$

Stone 1 and 2 three-phase oil relative permeability models:

$$\begin{aligned} K_{ROW}^{OW} &= f(S_{OW}, S_{OWCR}^I, S_{WCR}^{OW}, S_{OMAX}, K_{RO}^{OW}, K_{RORW}^{OW}) \\ K_{ROW}^{WW} &= f(S_{OW}, S_{OWCR}^I, S_{WCR}^{WW}, S_{OMAX}, K_{RO}^{WW}, K_{RORW}^{WW}) \end{aligned} \quad \text{Eq. 66.22}$$

where

- S_W is the grid block water saturation.
- S_{OW} is the oil saturation for oil relative permeability to water.
- S_{OG} is the oil saturation for oil relative permeability to gas.
- S_{wcr} is the saturation table derived critical water saturation.
- S_{owcr} is the saturation table derived critical oil in water saturation.

S_{OMAX} is the scaled maximum oil saturation, $S_{OMAX} = (1 - S_{WL}^I - S_{GL}^{OW})$.

S_{GL} is the scaled minimum gas saturation.

S_{GCR} is the scaled critical gas saturation.

S_{gcr} is the saturation table derived critical gas saturation.

K_{RW} is the scaled maximum water relative permeability.

K_{RWR} is the scaled water relative permeability at residual oil saturation.

K_{RO} is scaled maximum oil relative permeability.

K_{RORG} is the scaled oil relative permeability at residual gas saturation.

K_{RORW} is the scaled oil relative permeability at residual water saturation.

P_{CW} is the scaled maximum oil-water capillary pressure.

Noting that:

1. For the scaling of both the oil-wet and water-wet immiscible saturation tables, specifically, for the scaling of the relative permeabilities of oil to gas and water, only the oil-wet immiscible minimum and critical scaled gas saturations are used.
2. For two-point scaling, the positioning of the relative permeability at residual saturation of the displacing phase is determined by the two-point scaled table value of the displacing critical phase saturation whereas, for three-point scaling, this is determined by the scaled end-point of the displacing phase critical saturation.
3. For the default three-phase oil relative permeability model, the critical water saturation used in the scaling of the oil-wet and water-wet oil relative permeabilities to water is based upon the interpolated value, whereas for the Stone 1 and 2 three-phase oil relative permeability models, the scaling is based upon the respective immiscible oil-wet and water-wet values.

The miscible relative permeabilities are looked up in the miscible tables by applying two-point saturation (horizontal) scaling using the interpolated saturation end-points together with the table derived values of maximum relative permeabilities:

$$\begin{aligned} K_{RW}^{MIS} &= f(S_W, S_{WCR}^I, S_{WMAX}^I, k_{rwmax}^{mis}) \\ K_{ROG}^{MIS} &= f(S_{OG}, S_{OGCR}^I, S_{OMAX}, k_{romax}^{mis}) \\ K_{ROW}^{MIS} &= f(S_{OW}, S_{OWCR}^I, S_{OMAX}, k_{romax}^{mis}) \end{aligned} \quad \text{Eq. 66.23}$$

The immiscible oil-wet and water-wet relative permeabilities and capillary pressure are interpolated according to:

$$\begin{aligned}
 K_{RW}^{IMM} &= F_4 K_{RW}^{OW} + (1-F_4)K_{RW}^{WW} \\
 K_{ROW}^{IMM} &= F_4 K_{ROW}^{OW} + (1-F_4)K_{ROW}^{WW} \\
 K_{ROG}^{IMM} &= F_4 K_{ROG}^{OW} + (1-F_4)K_{ROG}^{WW} \\
 P_{COW}^{IMM} &= F_4 P_{COW}^{OW} + (1-F_4)P_{COW}^{WW}
 \end{aligned} \tag{Eq. 66.24}$$

The miscible and immiscible relative permeabilities and capillary pressure are interpolated according to:

$$\begin{aligned}
 K_{RW}^I &= F_3 K_{RW}^{MIS} + (1-F_3)K_{RW}^{IMM} \\
 K_{ROW}^I &= F_3 K_{ROW}^{MIS} + (1-F_3)K_{ROW}^{IMM} \\
 P_{COW}^I &= P_{COW}^{IMM}
 \end{aligned} \tag{Eq. 66.25}$$

Default three-phase oil relative permeability model:

$$K_{ROG}^I = F_3 K_{ROG}^{MIS} + (1-F_3)K_{ROG}^{IMM} \tag{Eq. 66.26}$$

Stone 1 and 2 three-phase oil relative permeability models:

$$K_{ROG}^I = K_{ROG}^{IMM} \tag{Eq. 66.27}$$

The gas-oil capillary pressure and gas relative permeability are assumed to be independent of surfactant concentration and surfactant adsorption. Hence the gas-oil capillary pressure is calculated according to:

$$P_{COG} = f(S_G, S_{GLPC}^{OW}, S_{GU}^{OW}, P_{CG}^{OW}) \tag{Eq. 66.28}$$

For two-point scaling the gas relative permeability is calculated according to:

$$K_{RG} = f(S_G, S_{GCR}^{OW}, S_{ogcr}^{ow}, S_{GU}^{OW}, K_{RG}^{OW}, K_{RGR}^{OW}) \tag{Eq. 66.29}$$

For three-point scaling the gas relative permeability is calculated according to:

$$K_{RG} = f(S_G, S_{GCR}^{OW}, S_{OGCR}^{OW}, S_{GU}^{OW}, K_{RG}^{OW}, K_{RGR}^{OW}) \tag{Eq. 66.30}$$

where

S_G is the grid block gas saturation.

S_{GLPC} is the scaled minimum gas saturation for capillary pressure.

S_{GU} is the scaled maximum gas saturation.

S_{ogcr} is the saturation table derived critical oil saturation in gas.

P_{CG} is the scaled maximum oil-gas capillary pressure.

K_{RG} is the scaled maximum gas relative permeability.

K_{RGR} is the scaled gas relative permeability at residual oil saturation.

Hence only the oil-wet table and oil-wet scaled end-points are used for modeling the gas saturation functions. If the current grid block gas saturation exceeds the maximum oil-wet scaled gas saturation, the

values of gas relative permeability and gas-oil capillary pressure are assumed to remain constant at their corresponding maximum values.

The [LOWSALT](#) keyword can be used in conjunction with the Surfactant Option ([SURFACT](#) keyword) or the Surfactant Option with Wettability Dependence ([SURFACTW](#) keyword) to model additionally the effects of salinity dependence. See "[Brine tracking](#)" for more information upon the LOWSALT option.

Saturation function interpolation (ECLIPSE 300)

Relative permeabilities

The interpolation method described in "[Relative permeability model](#)" applies to two relative permeability tables. The calculation here is a bit more complicated, particularly in compositional runs, although essentially it follows the same pattern.

The first step is to determine the saturation endpoints, by interpolating between the miscible and immiscible endpoints. If solid wettability is used, the immiscible endpoints must first be interpolated between the two wettability tables. If endpoint scaling is used, the immiscible curves and endpoints are both scaled, but the miscible curve is not scaled.

The second step is to look up the immiscible and miscible data using the scaled saturation endpoints, and then interpolate between them. If solid wettability is used, the immiscible endpoints must first be interpolated between the two wettability tables:

$$k_{row} = F_{kr}[F_w k_{row1} + (1-F_w)k_{row0}] + (1-F_{kr})k_{rowm}$$

$$k_{rw} = F_{kr}[F_w k_{rw1} + (1-F_w)k_{rw0}] + (1-F_{kr})k_{rwm}$$

Here k_{row0}, k_{rw0} are the immiscible relative permeabilities corresponding to a wettability factor $F_w = 0$ and k_{row1}, k_{rw1} are the immiscible relative permeabilities corresponding to a wettability factor $F_w = 1$. k_{rowm}, k_{rwm} are the miscible relative permeabilities.

In compositional cases, a hydrocarbon in water relative permeability is then calculated to ensure continuous behavior in super-critical regions. See "[Near critical oil and gas relative permeabilities](#)":

$$k_{rhw} = E k_{row} + (1-E)k_{rgw}$$

The final step is to calculate three-phase relative permeabilities using the default method or Stone's method. See "[Three phase oil relative permeability models](#)" and "[Three phase gas relative permeability models](#)":

$$k_{ro} = k_{ro}(k_{rhw}, k_{rog})$$

$$k_{rg} = k_{rg}(k_{rhw}, k_{rgo})$$

Note: In compositional cases, the default extra scaling of the hydrocarbon relative permeabilities is turned off, see keyword [FORMOPTS](#).

Capillary pressures

If the solid wettability model is used, the oil-water capillary pressure is interpolated for the two wettability regions, and the result is then multiplied by the miscibility factor:

$$P_{cow} = F_{cp}[F_w P_{cow1} + (1-F_w)P_{cow0}]$$

Using the ECLIPSE 100 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' argument 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:

Restriction	Keyword	Description
Required	SURFST (or SURFSTES)	Water-oil surface tension in the presence of surfactant (and salts).
Required	SURFVISC	Modified water viscosity.
Required	SURFCAPD	Capillary de-saturation data.
Optional	SURFADS	Adsorption isotherm.
If SURFADS is present.	SURFROCK	Rock properties and adsorption model indicator.

The surfactant properties data can be written to the PRINT file by setting the 'SURFVISC' argument 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 arguments 'SURFBLK', 'SURFADS' and 'FIPSURF' in the [RPTSCHE](#)D keyword. In addition a field/group/well/connection summary can be obtained at each report time by adding the argument 'WELLS' to RPTSCHE. 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](#)" for a more detailed discussion.

Restrictions

If a cell is defined with zero pore volume in the GRID section, then making it active by changing the pore volume in the EDIT section using [PORV](#), is not allowed with the surfactant model.

Using the ECLIPSE 300 surfactant model

Surfactant properties

The [COMPW](#) keyword should be used to set at least two water components - one for water itself and one to represent surfactant. The [CWTYPE](#) keyword should be used to identify which water components behave as surfactants. The water properties are defined using the water component keywords, as described in "[Water PROPS keywords](#)". For example, the water viscosity can be set using the [VREFW](#) keyword.

Adsorption

Adsorbed surfactant can be modeled using the [SOLID](#) model. The [COMPS](#) or [COMPSOL](#) keyword should be used to set the number of non-water components (including solids). The [CVTYPE](#) keyword should be used to identify the solid component. The properties of adsorbed surfactant are defined using the solid keywords, see [62.1](#).

The adsorption and desorption processes can be modeled using the [REACTION](#) model where the properties of the reaction are defined using the reaction keywords, as described in "[Chemical reaction PROPS keywords](#)". The reaction keywords are flexible and you can define the reaction in various ways. One simple method is described in "[Adsorption of dissolved water components](#)". In this method the [STOREAC](#) and [STOPROD](#) keywords are used to define dissolved surfactant as the reactant and adsorbed surfactant as the product of the reaction. The [EQLDREAC](#) keyword (model 4) is used to define a reaction rate that depends on the dissolved-adsorbed equilibrium. The half-life for $(C_{aeq} - C_a)$ is:

$$T_{1/2} = \ln 2 / A_r \quad \text{Eq. 66.31}$$

where

A_r is a constant defined by the [REACRATE](#) keyword,

C_{aeq} is the equilibrium adsorbed solid concentration defined by [EQLDTAB](#),

C_a is the actual adsorbed solid concentration.

This reaction can be one way (forwards for adsorption or backwards for desorption) or two way.

If an ECLIPSE 100 model is converted to ECLIPSE 300, [EQLDTAB](#) data can be derived from the [SURFADS](#) data, which defines equilibrium concentrations C_{aeq} as functions of dissolved concentrations C_d . To convert the data it is necessary to replace the dissolved concentrations C_d with the surfactant mole fractions a_i defined by

$$a_i = \frac{MW_w}{\rho_{ws}} \cdot \frac{C_d}{MW_i}$$

where

MW_w and MW_i are the molecular weights of the water mixture and the surfactant component, and

ρ_{ws} is the surface density of water.

In ECLIPSE 100 the adsorption process is instantaneous, which could be modeled using a large reaction rate constant A_r . However this may make it hard to converge. A reasonable value for A_r can be determined by estimating a half-life and using equation 66.31. For example, the residence time of the surfactant in a cell can be determined from the pore volume and the flow rate $T_{res} \approx V_p / Q$, and the maximum rate of adsorption occurs when there are no adsorbed solids and when the surfactant concentration is at a maximum. If we set $A_r = \ln 2 / T_{res}$ then we would expect about half the surfactant to be adsorbed in the first cell.

Decay

Surfactant decay can be modeled by using the [REACTION](#) model where the properties of the reaction are defined using the reaction keywords, as described in "Chemical reaction PROPS keywords". Surfactant could decay to water, or perhaps to another water component. The [STOREAC](#) and [STOPROD](#) keywords should be used to define surfactant as the reactant and water (or another component) as the product of the reaction. The reaction rate should be made proportional to the amount of surfactant using [REACCORD](#). The [REACRATE](#) keyword should be used to define a constant A_r for the decay rate. This can be determined from the decay half-life $A_r = \ln 2 / T_{1/2}$.

Surfactant effects

The surfactant concentration is calculated in each cell by summing up the weight of the surfactant components and dividing by the volume of water at surface conditions. The surface tension σ_{ow} must be set using the [SURFST](#) keyword, as a function of the surfactant concentration, or [SURFSTES](#) keyword, as a function of the surfactant and salt concentrations. Note that the amount of adsorbed surfactant does not affect the surface tension. The surface tension allows the capillary number N_c to be calculated and determines the capillary pressure multiplier F_{cp} . The [SURFCAPD](#) keyword must be used to determine the effect of surface tension on the relative permeability F_{kr} .

The [SURFNUM](#) keyword must be used to identify the miscible relative permeability tables. Capillary pressures defined by these tables are not used - it is assumed that the miscible capillary pressure is zero.

For cells with low water saturation, a small amount of surfactant will have a high concentration in the water phase, and could therefore have a large effect on the capillary pressure. To prevent numerical errors having an effect, the surfactant concentration in the water phase is set to zero for cells with very small water saturations. Therefore there will be no surfactant effect in cells with water saturations below a threshold. The molar concentrations are unchanged, so the component balance equations are not affected by this threshold. The threshold value can be set using the [SURFOPTS](#) keyword.

Wettability

The effects on wettability of adsorbed surfactant can be modeled by creating two sets of saturation functions, for low and high concentrations of adsorbed solids. The [SATNUM](#) and [SOLWNUM](#) keywords are used to identify the saturation functions corresponding to these two extremes and the [SOLWTAB](#) keyword is used to determine an interpolation parameter dependent on the adsorbed solid concentration.

Injection

The water composition of an injection stream can be defined using the [WELLSTRW](#) keyword, and associated with an injection well using the [WINJW](#) keyword.

Output

Properties of the water, adsorbed surfactant and the adsorption / desorption processes can be output using the SUMMARY vectors and other output keywords described in "[Water component keywords](#)", "[Solid keywords](#)" and "[Chemical reaction keywords](#)". In addition surfactant specific keywords can be used to output the surfactant concentration C_{surf} , surface tension σ_{ow} , capillary number N_c , capillary pressure multiplier F_{cp} , and the miscible / immiscible relative permeability interpolation parameter F_{kr} . See "[ECLIPSE 300 Surfactant SUMMARY output controls](#)" and "[ECLIPSE 300 Surfactant output controls](#)".

Summary of ECLIPSE 100 keywords

RUNSPEC

Keyword	Description
PARTTRAC	Activates and dimensions the Partitioned Tracer option.
SURFACT	Activates the Surfactant Model
SURFACTW	Activates the Surfactant Model and enables modeling of changes of wettability.

GRID

Keyword	Description
RPTGRID	Controls output from the GRID section. Argument KOVERD outputs the K/D values used in the calculation of the capillary number.

PROPS

Keyword	Description
HWSWL , HWSWLP , HWSWCR , HWSWU , HWSOWCR , HWSOGCR , HWKRW , HWKRW , HWKRO , HWKRORW , HWKRORG and HWPCW	Immiscible water-wet saturation end-points.
RPTPROPS	Controls output from the PROPS section. Arguments SURFVISC and so on output surfactant properties.
SOCRS	Scaled critical oil-in-water saturations at miscible conditions.
SURFADDW	Defines weighting between oil-wet and water-wet relative permeabilities as a function of the adsorbed surfactant mass.
SURFADS	Surfactant adsorption isotherm.

Keyword	Description
ADSORP	Analytical adsorption isotherm with salinity dependence.
SURFCAPD	Surfactant capillary de-saturation data.
SURFROCK	Surfactant-rock properties and adsorption model indicator.
SURFST	Water-oil surface tension in the presence of surfactant.
SURFSTES	Water-oil surface tension as a function of surfactant and salt concentrations.
SURFVISC	Modified water viscosity.
SWL, SWLPC, SWCR, SWU, SGL, SGLPC, SGCR, SGU, SOWCR, SOGCR, KRW, KRWR, KRG, KRGR, KRO, KRORW, KRORG, PCW and PCG	Immiscible oil-wet saturation end-points.
TRACER	Defines the tracers (for the Partitioned Tracer option).
TRACERKP	Defines the functions K(P) for the partitioned tracers.

REGIONS

Keyword	Description
HWSNUM	Abbreviated alias for SURFWNUM.
SATNUM	Defines the region number for oil-wet immiscible saturation function table.
SURFNUM	Surfactant miscible region numbers.
SURFWNUM	Defines the region number for water-wet immiscible saturation function tables and additionally the table describing the wettability as a function of surfactant adsorption (SURFADDW).
TRKPFxxx	Specifies the partitioning function for tracer xxx for each grid block ('xxx' = 'SUR' for surfactant).

SOLUTION

Keyword	Description
RPTSOL	Controls output from the SOLUTION section. <ul style="list-style-type: none"> SOCR outputs the critical oil saturations following a surfactant flood. SUFRBLK outputs the surfactant concentration as well as water-oil surface tension (both for salinity dependent and salinity independent surface tension).
SURF	Initial surfactant concentrations.
TVDP	Initial surfactant concentrations with depth
TNUM	Surfactant concentration regions.

SUMMARY

Refer to [Surfactant model](#) in the *ECLIPSE Reference Manual*.

SCHEDULE

Keyword	Description
RPTSCHED	Controls output from the SCHEDULE section. Arguments: <ul style="list-style-type: none"> • FIPSURF: surfactant in place. • SURFADS: adsorbed surfactant concentration. • SURFBULK: surfactant concentrations and the capillary number as well as water-oil surface tension.
RPTRST	Controls output to the restart file. Mnemonic VISC can be requested to output <ul style="list-style-type: none"> • EWV_SUR: effective water viscosity due to surfactant. If a graphics only restart is required (in other words, when mnemonic NORST is requested) then adding mnemonic SURFBULK will output: <ul style="list-style-type: none"> • SURFACT: surfactant concentration. • SURFADS: adsorbed surfactant concentration. • SURFMAX: maximum adsorbed surfactant concentration. • SURFCNM: (log) capillary number (for miscibility transition). • SURFST or (SURFSTES): water-oil surface tension On the other hand, for full restarts (not graphics only) above mentioned arrays will be output by default.
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.

Summary of ECLIPSE 300 keywords

The surfactant component and water properties must be defined by water component keywords. See "[Water component keywords](#)".

A solid component can be used to represent adsorbed surfactant. See "[Solid phase](#)".

Adsorption and desorption could then be modeled using a pair of chemical reactions. See "[Chemical reactions](#)".

PROPS

Restriction	Keyword	Description
	SURFCAPD	Surfactant capillary de-saturation data.
	SURFOPTS	Surfactant model options.
	SURFST	Water-oil surface tension in the presence of surfactant.
	SURFSTES	Salinity dependent water-oil surface tension in the presence of surfactant.
Required if salinity dependent surfactant model is activated.	ESSNODE	Salt concentration nodes for oil-water surface tension.
Required if salinity dependent surfactant model is activated.	SALTEFF	Effective salinity coefficients.
Required to model the effect of absorbed surfactant on the wettability of the rock.	SOLWTAB	Solid wettability table.

Regions

The keywords are:

- [SURFNUM](#) Surfactant miscible region numbers.
- [SOLWNUM](#) Solid wettability region numbers.

SUMMARY

Refer to [Surfactant model](#) in the *ECLIPSE Reference Manual*.

Other Reporting

The following arguments can be used with the [RPTSOL](#), [RPTSCHED](#), [RPTRST](#) keywords to output grid data specific to the Surfactant Model:

Keyword	Description
SURF	Concentration C_{surf} measured as the mass of surfactant per volume of water phase.
SURFST	Oil water surface tension σ_{ow}
SURFCNM	\log_{10} (capillary number)
SURFKR	Relative permeability interpolation factor F_{kr}
SURFCP	Capillary pressure multiplier F_{cp}
ESALTS	Effective salinity of the water phase in presence of surfactant.

ECLIPSE 100 example

Example 1

This is a 2-D example modeling water injection into oil. To aid recovery, a surfactant slug is injected into the reservoir some time after the primary water drive. Water injection is resumed after the surfactant injection period. The performance of the surfactant drive is assumed not to be salt sensitive in this study.

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            0.5  
    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            5.0E-04
  0.1            6.34E-05
  0.2            2.23E-05
  0.5            2.07E-06/
  0.0            5.0E-04
  0.1            6.34E-05
  0.2            2.23E-05
  0.5            2.07E-06/

-- Capillary de-saturation curve
SURFCAPD
-- LOG10(Capillary Number)      Miscibility
--                                         Function
--                                         0.0
  -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
  1                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-'  'KRUNYM'   'KRUNYM-'  'KRNUMZ'   'KRNUMZ-'
'IMBNUM'   'IMBNUMX'   'IMBNUMX-'  'IMBNUMY'   'IMBNUMY-'
'IMBNUMZ'   'IMBNUMZ-'  'ROCKNUM'   'ENDNUM'   'TNUM'
'MISCNUM'   'KRUNMMF'   'IMBNUMMF'   'SURFNUM'   /
```

SOLUTION

```
SOLUTION =====
TBLKFONE
25*0 /

EQUIL
4000 4000 6000 0 0 0 0 0 0 /

RPTSL
'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 =====
RPTSCHE
'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'   /
/
```

```
WELSPCS
'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
```

Example 2

This is a version of the previous example with the salt option activated to study the effect of the interaction between the injected surfactant and the salt.

RUNSPEC

```

RUNSPEC
TITLE
Example Surfactant Flood Case.

DIMENS
 5   5   1  /

NONNC
OIL
WATER
BRINE
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
/
-- brine properties
PVTWSALT
-- Pref      Cref
  2700      0  /
-- SaltConc   Bw      Cw      Vw      Cvw
--                   1/bar    CPOISE    1/psi
--           0       1.030  4.6E-5  0.34 0.0
--           20      1.000  4.6E-5  0.34 0.0
--           30      0.98   4.6E-5  0.34 0.0
--           50      0.95   4.6E-5  0.34 0.0  /
3000  /
-- SaltConc   Bw      Cw      Vw      Cvw
--                   1/bar    CPOISE    1/psi
--           0       1.030  4.6E-5  0.34 0.0
--           20      1.000  4.6E-5  0.34 0.0
--           30      0.98   4.6E-5  0.34 0.0
--           50      0.95   4.6E-5  0.34 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              0.34
  0.1              1.1
  0.2              1.2
  0.8              1.3  /
  0.0              0.34
  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 and salt concentrations
SURFSTES
-- surf      water-oil
-- conc     surface tension
-- LB/STB   (LB/IN)
  0.0      5.0E-04
  5.0E-04
  5.0E-04      /
  0.7      2.23E-05
  2.12E-05
  2.00E-05      /
  3.5      2.07E-06
  1.79E-06
  1.47E-06      /
  4.9      1.04E-06
  8.76E-07
  7.18E-07      /
  7.0      4.95E-07
  4.16E-07
  3.45E-07      /
/
/
-- Salt concentration nodal values for SURFSTES keyword
ESSNODE
  0    10    30  /
/
-- Capillary de-saturation curve
SURFCAPD
-- LOG10(Capillary Number)      Miscibility
--                               Function
--                               0.0
  -9                  0.0
  2                  1.0
  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
  1                  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
'PVNUM'   'SATNUM'   'EQLNUM'   'FIPNUM'   'KRNUMX'
'KRNUMX-' 'KRNAMY'   '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 /

SALTVD
2000 0
4000 0 /

RPTSL
'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 /
/

BTSTSUR
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'  /
/
COMPDATA
'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 /
/
WSALT
'I'  0.6 /
/
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

```

ECLIPSE 300 example

Example 1

This is a 3-D example modeling water injection into oil. To aid recovery, a surfactant slug is injected into the reservoir some time after the primary water drive. The performance of the surfactant drive is assumed not to be salt sensitive in this study.

RUNSPEC

A solid component (to model adsorbed surfactant) and two water components (to model water and dissolved surfactant) are used, together with oil and gas components. A single reversible reaction is used (to model adsorption and desorption). The simulation is run fully implicitly (because reactions are used).

```

RUNSPEC
TITLE
Example E300 Surfactant - blackoil

DIMENS
10 10 3 /
BLACKOIL
OIL
GAS
WATER
SOLID
COMSOL
3 /
COMPW
2 /

REACTION
1 1 1* 1* 1 /
TABDIMS
2 /
METRIC
START
1 'JAN' 2012 /
FULLIMP

```

GRID

```

GRID      =====
TOPS
100*2600 /
DXV
10*50.0 /
DYV
10*50.0 /
DZ
100*0.58 100*0.84 100*0.47 /

PERMX
100*4500 100*3300 100*2400 /

PERMY
100*4500 100*3300 100*2400 /

PERMZ
100*1050 100*1800 100*500 /

PORO
300*0.25 /

```

PROPS

```

PROPS      =====
SWFN
-- Table 1 - Immiscible curves - SATNUM
.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
/
-- Table 2 - Miscible curves - SURFNUM
0.0    0.0    0.0
1.0    1.0    0.0
/
SGFN
0.00  0.000  0
0.02  0.000  0
0.05  0.005  0
0.12  0.025  0
0.20  0.075  0
0.25  0.125  0
0.30  0.190  0
0.40  0.410  0
0.45  0.600  0
0.50  0.720  0
0.60  0.870  0
0.70  0.940  0
0.85  0.980  0
1.00  1.000  0
/
/
SOF3
-- Table 1 - Immiscible curves - SATNUM
.000   .0000   .0000
.205   .0000   .0000
.250   .00006  .00006

```

```

.300 .0009 .0009
.370 .0080 .0080
.440 .0270 .0270
.490 .0520 .0520
.560 .1200 .1200
.635 .2280 .2280
.683 .3340 .3340
.730 .4700 .4700
.780 .6440 .6440
.855 1.000 1.0000
/
-- Table 2 - Miscible curves - SURFNUM
.205 .0000 .0000
.250 1* .00006
.300 1* .0009
.370 1* .0080
.440 1* .0270
.490 1* .0520
.560 1* .1200
.635 1* .2280
.683 1* .3340
.730 1* .4700
.780 1* .6440
.855 1* 1.0000
/
PVTO
 0      1 1.000 1.24 /
 60    220 1.301 1.24
 400   400 1.299 1.24 /
100   300 1.401 1.24
 600   600 1.399 1.24 /
/
PVDG
 1.0    10.0   0.01
 400.0   1.0   0.01
/
ROCK
270.0 .30E-05 /
/
DENSITY
850.0 1000.0 1.0 /

```

Water properties

Here the water component and the surfactant component are given the same density properties, so that the surfactant acts like a tracer. However the surfactant component viscosity is increased so that when the surfactant molar fraction is 3% the water phase viscosity is increased to 5cp.

```

--Water and dissolved surfactant properties
WNAMES
WATER SURF /
CWTYPE
1* SURFS
MW
18.015 18.015 /
PREFW
270 270 /
DREFW
970.9 970.9 /
CREFW
4.6E-5 4.6E-5 /
VREFW
0.34 155.7
0 0 /

```

Solid properties

The adsorbed surfactant density properties are set equal to the dissolved surfactant values, so that the adsorption / desorption processes have no effect on pressure.

```
--Adsorbed surfactant properties
CNAMES
OIL GAS ASURF /
CVTYPE
1* 1* SOLID /
MW
1* 1* 18.015 /
SDREF
1* 1* 970.0 /
```

Adsorption reactions

The [EQLDREAC](#) keyword model 4 is used to provide a reversible reaction between the solid component (component 3 in this example) and the water component (component 5 in this example). The water component must be defined as a reactant ([STOREAC](#)) and the solid component must be defined as a product ([STOPROD](#)) to ensure that the solid component is produced when the reaction rate is positive and the water component is produced when the reaction rate is negative.

```
--Adsorption reactions
STOREAC
--O G A W S
0 0 0 0 1 / Dissolved Surfactant <-> Adsorbed Surfactant
STOPROD
--O G A W S
0 0 1 0 0 / Dissolved Surfactant <-> Adsorbed Surfactant
REACRATE
1 /
EQLDREAC
1 4 5 0 1 BOTH 1 / Rate proportional to EQLDTAB_1(S) - Ads Solid
EQLDTAB
0.0      0.00000
0.001    0.0005
1.000    0.0005 /
```

Surfactant properties

```
-- Water/Oil surface tension vs Surfactant concentration
SURFST
-- Surfactant      Water-Oil Surface Tension
-- concentration
-- Kg/SM3          N/M
0.0              0.05
1.0              1.0e-6
3.0              1.0e-6 /

-- Capillary de-saturation curve
SURFCAPD
-- LOG10(Capillary Number)      Miscibility
--                           Function
-9                0.0
-4.5              0.0
-2                1.0
10               1.0 /
-9                0.0
-4.5              0.0
-2                1.0
10               1.0 /
```

REGIONS

```
REGIONS =====
-- Immiscible table numbers
SATNUM
300*1 /
```

```
-- Miscible table numbers
SURFNUM
300*2 /
```

SOLUTION

```
SOLUTION =====

EQUIL
2600 270 2700 0 0 0 1 /
WMFVD
2600 1.0 0.0
2700 1.0 0.0 /
RSVD
2600 0.0
2700 0.0 /

RPTSQL
'PRES' 'SOIL' 'SWAT' 'SSOLID' 'VWAT'
'SURF' 'SURFST' 'SURFCNM' 'SURFKR' 'SURFCP' /
```

SUMMARY

```
SUMMARY =====

BSURF
1 1 1 /
5 5 1 /
10 10 1 /
/

BSURFST
1 1 1 /
5 5 1 /
10 10 1 /
/

BSURFCNM
1 1 1 /
5 5 1 /
10 10 1 /
/

BSURFKR
1 1 1 /
5 5 1 /
10 10 1 /
/

BSURFCP
1 1 1 /
5 5 1 /
10 10 1 /
/

BKRO
1 1 1 /
5 5 1 /
10 10 1 /
/

BKRW
1 1 1 /
5 5 1 /
10 10 1 /
/

BWPC
1 1 1 /
5 5 1 /
```

```
10 10 1 /
/
```

SCHEDULE

```
SCHEDULE =====
RPTSCHED
'PRES'   'SOIL'     'SWAT'      'SSOLID'    'VWAT'
'SURF'   'SURFST'   'SURFCNM'   'SURFKR'    'SURFCP'   /
RPTRST
'RESTART'          'VWAT'
'SURF'   'SURFST'   'SURFCNM'   'SURFKR'    'SURFCP'   /
WELSPECS
'I'    'G'    1  1  2600  'WAT'   /
'P'    'G'   10 10  2600  'OIL'   /
/
COMPDAT
'I'    1  1  1  3  'OPEN'    0  .0  .157  /
'P'   10 10  1  3  'OPEN'    0  .0  .157  /
/
WCONPROD
'P'  'OPEN'  'RESV' 4* 500 0.0 /
/
WELLSTRW
WATER 1.0  0.0  /
SURF  0.97 0.03 /
/
WINJW
I STREAM WATER /
/
WCONINJE
'I'  'WAT'  'OPEN'  'RESV' 1* 500.0 /
/
TSTEP
3*50 /
WINJW
I STREAM SURF /
/
TSTEP
3*50 /
END
```

Example 2

This is a compositional simulation with multicomponent water with salinity-dependent Surfactant option activated to study the effect of the interaction between the injected surfactant and the effective salinity of the water phase. Furthermore, alkaline represented as one of the water components is injected to study the efficacy of surfactant flooding.

RUNSPEC

A solid component (to model adsorbed surfactant) and two water components (to model water and dissolved surfactant) are used, together with oil and gas components. A single reversible reaction is used (to model adsorption and desorption). The simulation is run fully implicitly (because reactions are used).

```
RUNSPEC
TITLE
E300 effective salinity-sensitive Surfactant model test case - compositional AIM
```

```

DIMENS
 10    10    3    /
OIL
GAS
WATER

COMPW
5 /
COMPS
8 /

METRIC

TABDIMS
2 /

AIM

START
 1 'JAN' 2012  /

```

GRID

```

GRID =====
TOPS
 100*2600 /
DXV
 10*50   /
DYV
 10*50   /
DZ
 100*0.58 100*0.84 100*0.47  /
PERMX
 100*4500 100*3300 100*2400 /
PERMY
 100*4500 100*3300 100*2400 /
PERMZ
 100*1050 100*1800 100*500 /
PORO
 300*0.25 /
ROCKDEN
 300*2650 /

```

PROPS

```

PROPS =====
SWFN
-- Sw      krw     Pc
0.20000  0.00000  6.00000
0.25000  0.00000  5.00000
0.30000  0.00000  4.00000
0.35000  0.00000  3.00000
0.40000  0.04000  2.40000
0.45000  0.06000  1.90000
0.50000  0.08000  1.40000
0.55000  0.11000  1.00000
0.60000  0.15000  0.70000
0.65000  0.20000  0.40000
0.70000  0.25000  0.25000
0.75000  0.32000  0.10000
0.80000  0.40000  0.08000
0.85000  0.50000  0.06000
0.90000  0.62000  0.04000
0.95000  0.80000  0.02000
1.00000  1.00000  0.00000
/

```

```

-- miscible
-- Sw      krw     Pc
0.00000    0.00000    0.00000
1.00000    1.00000    0.00000
/

SGFN
-- Sg      Krg     Pcg
0.000    0       0
0.400    0       0
0.450    0.066   0.80
0.500    0.177   1.56
0.550    0.40    2.24
0.600    0.75    3
/
/

SOF3
-- So      krow    krog
0.0      0       0
0.1      0       0
0.2      0       0
0.25     0       0.01
0.30     0.11   0.02
0.35     0.15   0.03
0.40     0.2    0.04
0.45     0.25   0.08
0.50     0.32   0.14
0.55     0.4    0.225
0.60     0.5    0.33
0.65     0.6    0.434
0.70     0.7    0.575
0.75     0.8    0.72
0.80     0.9    0.9
/
-- miscible
0.0      0.0    0.0
1.0      1.0    0.0
/
-----  

-- WATER, SURFACTANT AND ALKALILNE PROPERTIES
-----  

--Multicomponent water with surfactant
--Water and dissolved surfactant properties
WNAMES
WATER NA CL SURF ALK/  

CWTYPE
1* 1* 1* SURFS ALK/  

MWW
18.015 22.98977 35.453 200 18.015/  

PREFW
270 270 270 270 270/  

DREFW
970.9 970.9 970.9 970.9 970.9/  

CREFW
4.6E-5 4.6E-5 4.6E-5 4.6E-5 4.6E-5/  

VREFW
0.34 0.34 0.34 34 120.7
0      0      0      0      0 /  

EOS
SRK
/

```

```

-- HYDROCARBON AND SOLID COMPONENTS

CNAMES
  C1    HC23    HC46    C8    HC13    HC18    HC26    HC43    /
CVTYPE
  1*    1*    1*    1*    1*    1*    1*    1*    /
MW
  16.04   35.88   67.98  110.14  173.11  248.85  361.77  600.98  /
PCRIT
  46.00   45.59   34.25   30.01   22.30   16.18   11.88   10.27  /
TCRIT
 190.60  338.20  456.95  575.51  666.13  735.04  807.07  914.84  /
VCRIT
 0.09925 0.17071 0.29427 0.49206 0.8008  1.2180  1.8210  2.3880  /
ACF
 0.008   0.1255   0.2325   0.4210   0.7174   0.9849   1.2737   1.6704  /
SDREF
  1*    1*    1*    1*    1*    1*    1*    1*    /
ZI
 0.528   0.1898   0.0703   0.0867   0.0529   0.034   0.0238   0.0145  /
BIC
  0.0
  0.0   0.0
  0.027 0.0   0.0
  0.0   0.0   0.0   0.0
  0.0   0.0   0.0   0.0
  0.0   0.0   0.0   0.0   0.0
  0.0   0.0   0.0   0.0   0.0   0.0
/
PEDERSEN
ROCK
 270 .3E-5 /
DENSITY
 850. 1000. 1. /
RTEMP
 40 /

```

Surfactant properties

```

SURFSTES
-- surf      water-oil
-- conc     surface tension
  0.0    0.05
  0.05
  0.05
  0.05    /
  2.0    4.90E-03
          3.75E-03
          3.50E-03
          4.00E-03    /
 10.0   2.40E-03
          7.00E-04
          2.88E-04
          1.50E-03    /
 14.0   1.82E-03
          4.00E-04
          1.44E-04
          1.00E-03    /
 20.0   8.66E-04
          2.00E-04
          6.87E-05
          5.22E-04    /
ESSNODE
 0    9    20    58.44    /

```

```

SALTEFF
SURF      2  0   0   0   1   0   0   1   0   0   0   0   0   0   /

ALSURFST
0.000 1.0000
0.003 0.9991
0.009 0.99908 /

SURFCAPD
-9    0.0
-4.5  0.0
-2    1.0
10   1.0 /
-9    0.0
-4.5  0.0
-2    1.0
10   1.0 /
-9    0.0
-4.5  0.0
-2    1.0
10   1.0 /

```

REGIONS

```

REGIONS      =====
SATNUM
300*1 / immiscible tables

SURFNUM
300*2 / miscible tables

```

SOLUTION

```

SOLUTION  =====
EQUIL
2600 270 2700 0 0 0 1 /

WMFVD
2600 1.0 0.0 0.0 0.0 0.0
2700 1.0 0.0 0.0 0.0 0.0/

-- 
-- Initialisation Print Output
--
RPTSQL
'PRES' 'SOIL' 'SWAT' 'SURF' 'SURFST' 'SURFCNM' 'SURFKR' 'SURFCP' /
RPTRST
'RESTART' 'SOIL' 'SWAT' 'SURF' 'SURFST' 'SURFCNM' 'SURFKR' 'SURFCP'
'VWAT' 'KRO' 'KRW' 'PCOW' /

```

SUMMARY

```

SUMMARY      =====
BSURF
1  1  1  /
5  5  1  /
10 10 1  /
/

```

```

BSURFST
  1  1  1 /
  5  5  1 /
 10 10 1 /
/

BSURFCNM
  1  1  1 /
  5  5  1 /
 10 10 1 /
/

BSURFKR
  1  1  1 /
  5  5  1 /
 10 10 1 /
/

BSURFCP
  1  1  1 /
  5  5  1 /
 10 10 1 /
/

BKRO
  1  1  1 /
  5  5  1 /
 10 10 1 /
/

BKRW
  1  1  1 /
  5  5  1 /
 10 10 1 /
/

BWPC
  1  1  1 /
  5  5  1 /
 10 10 1 /
/

BALSURF
  1  1  1 /
  5  5  1 /
 10 10 1 /
/

BALSTML
  1  1  1 /
  5  5  1 /
 10 10 1 /
/

```

SCHEDULE

```

SCHEDULE =====
RPTSCHED
'PRES' 'SOIL' 'SWAT' 'SURF' 'SURFST' 'SURFCNM' 'SURFKR' 'SURFCP'
'SOLID' 'VWAT' 'ALSURF' 'ALSTML' /

RPTRST
'RESTART' 'SOIL' 'SWAT' 'SURF' 'SURFST' 'SURFCNM' 'SURFKR' 'SURFCP'
'SOLID' 'VWAT' 'KRO' 'KRW' 'PCOW' 'ALSURF' 'ALSTML' /

WELSPECS
'OP'   'G'    10 10  2600  'OIL'   /
'INJ'  'G'    1  1   2600  'WAT'   /
/

COMPDAT

```

```
'OP      '    10 10 1   3 'OPEN'   0 .0   .157 /
'INJ      '    1   1 1   3 'OPEN'   0 .0   .157 /
/
WCONPROD
'OP' 'OPEN' 'RESV' 4* 500 10 /
/
WELLSTRW
WATER 0.4 0.3 0.3 0.0 0.0/
SURF 0.97 0.0 0.0 0.03 0.0/
ALK 0.98 0.0 0.0 0.0 0.02 /
/
WINJW
INJ STREAM WATER /
/
WCONINJE
'INJ' 'WAT' 'OPEN' 'RESV' 1* 500 500 /
/
TSTEP
2*10 /
/
WINJW
INJ STREAM ALK /
/
TSTEP
1*10 /
/
WINJW
INJ STREAM SURF /
/
TSTEP
3*10 /
/
END
```

67

Temperature option

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The temperature option enables ECLIPSE to model temperature effects, for example the injection of cold water into a hot reservoir.

The major effect of temperature changes in the vicinity of the injection wells is to modify the fluid viscosities. In addition, changes in the reservoir temperature will induce additional stresses within the reservoir which may modify the rock properties. ECLIPSE can output the temperature distribution to the RESTART file; this may be used by a separate program to model the effect of the temperature change on the reservoir rock.

The temperature option is initiated by the keyword [TEMP](#) in the RUNSPEC section.

Particularly for dual porosity calculations, the user should refer to the documentation on the thermal option in ["Dual porosity"](#) for information on how the rock volume is handled in the presence of more than one porosity.

ECLIPSE 100 and 300 differences

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"](#). This leads to a number of important differences:

- In ECLIPSE 100 an energy conservation equation is solved at the end of each converged timestep, and the grid block temperatures are updated. The new temperatures are then used to calculate the oil and water viscosities for the subsequent timestep. 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 a number of specialized processes, including steam injection, hot fluid or gas injection, well bore heaters and combustion. Note that when 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 the reference conditions specified using the [VISCREF](#) keyword. The oil viscosity at the prevailing temperature, pressure and gas/oil ratio is calculated as follows:

$$\mu_o(T, P, R_s) = \mu_T(T) \cdot \frac{\mu_p(P, R_s)}{\mu_p(P_{ref}, R_{s\ ref})} \quad \text{Eq. 67.1}$$

where

μ_T is the viscosity from the [OILVISCT](#) keyword

μ_p is the viscosity from the [PVTO](#) (or [PVDO](#)) keyword

P_{ref} is the reference pressure defined by the [VISCREF](#) keyword

$R_{s\ ref}$ is the reference gas/oil ratio defined by the [VISCREF](#) keyword.

Similarly, the water viscosity at the prevailing temperature, pressure and optionally salt concentration is calculated as follows:

$$\mu_w(T, P) = \mu_T(T) \cdot \frac{\mu_p(P)}{\mu_p(P_{ref})} \quad \text{Eq. 67.2}$$

or, if the brine option is active, as

$$\mu_w(T, P, c_s) = \mu_T(T) \cdot \frac{\mu_p(P, c_s)}{\mu_p(P_{ref}, c_{s\ ref})} \quad \text{Eq. 67.3}$$

where

μ_T is the viscosity from the [WATVISCT](#) keyword

μ_p is the viscosity from the [PVTW](#) or [PVTWSALT](#) keyword

P_{ref} is the reference pressure defined by the [VISCREF](#) keyword

$c_{s\ ref}$ is the reference salt concentration defined by item 2, record 1 of the [PVTWSALT](#) keyword.

Thermal conduction

The thermal conduction of heat in the rock is often a small effect compared with the convection of heat with the injected water. Thermal conduction is optional and is initiated by using either the [THCONR](#) keyword, or for ECLIPSE 300 only, keywords [THCROCK](#), [THCOIL](#), [THCWATER](#), [THCGAS](#) and [THCSOLID](#), in the grid section, to specify the cell rock and fluid thermal conductivity.

ECLIPSE 100

The thermal conductivity and thermal transmissibility can be printed out using the [THCONR](#) and [TRTHER](#) arguments in the [RPTGRID](#) keyword. The thermal transmissibilities calculated by ECLIPSE can be modified using inter-region thermal transmissibility multipliers specified using keywords [MULTREGH](#) and [MULTNUM](#).

ECLIPSE 300

The average cell thermal conductivity can be printed out using the argument [THCAVE](#) in the [RPTGRID](#) keyword. The thermal transmissibility can be printed out using the [HEATTX](#), [HEATTY](#), and [HEATTZ](#) arguments 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 as 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 equation [69.42](#) with default values of the acentric factors and critical temperatures given in table [67.1](#).

The constant F is the ratio of the molar volumes of gas to oil at surface conditions:

$$F = \frac{V_{gas,s}}{V_{oil,s}} = \frac{MW^{gas}}{MW^{oil}} \cdot \frac{\rho_{oil,s}}{\rho_{gas,s}}$$

Values of $\rho_{oil,s}$ and $\rho_{gas,s}$ are taken from the [DENSITY](#) keyword, while default values for the molecular weights are given in the following table.

Property	Symbol	Units	Oil	Gas
Acentric factor	Ac		0.348	0.056
Critical temperature	Tc	K	603	200
Molecular weight	MW	kg/kg-M	120	25

Table 67.1: Default values used in calculating K-values

Setting the initial conditions

The initial temperature is either assumed to be constant throughout the reservoir or can be specified as function of depth. If the initial temperature distribution is specified as a function of depth then the reservoir will not be in thermal equilibrium as there are no heat sources/sinks at the top and bottom of the model. The initial temperature is set using either the [RTEMPA](#) or [RTEMPVD](#) keywords in the [SOLUTION](#) section.

Wells

ECLIPSE 100

In ECLIPSE 100 the temperature of the injected fluid can be specified for each well in the [SCEDULE](#) section, using the [WTEMP](#) keyword. If a well does not have an injection temperature specified, the fluid is taken to be at zero degrees.

ECLIPSE 300

In ECLIPSE 300 the enthalpy of the injected water must be specified. This can be done by specifying the temperature and pressure of the injected water for each well in the [SCEDULE](#) section, using the [WINJTEMP](#) keyword.

Solution output

ECLIPSE 100

In ECLIPSE 100 the temperature can be output using the [TEMP](#) argument in [RPTSCHED](#), and the field and region energy balance reports can be output by using the [FIPTEMP](#) argument.

ECLIPSE 300

In ECLIPSE 300 temperature data can be output using Thermal option keywords, see "[Thermal keywords](#)".

Summary output

ECLIPSE 100

Refer to [Temperature option quantities](#) in the *ECLIPSE Reference Manual*.

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](#)".

Restrictions

If a cell is defined with zero pore volume in the GRID section, then making it active by changing the pore volume in the EDIT section using `PORV`, is not allowed with the temperature option.

Convergence issues

ECLIPSE 100

The effects of unconverged timesteps forced to be accepted by ECLIPSE 100 might be significant although they tend to dissipate rapidly. These anomalies can be corrected if desired by choosing smaller timesteps that ensure the convergence of the solution.

Note: The `MESSOPTS` keyword can be used to control the severity type for messages in the PRT file that report timesteps that are forced to be accepted. Note also that the total number of timesteps forced to be accepted is output in the Error Summary.

Example data set

The example data sets `HEAT100.DATA` and `HEAT300.DATA` which are in the directory `e300\data\therma` can be used with ECLIPSE 100 and ECLIPSE 300 respectively.

68

Tensor permeability

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

Full tensor permeabilities are often required to describe complex reservoirs. Given a detailed heterogeneous reservoir description at the geological scale, upscaling is normally carried out to reduce the model size to one suitable for reservoir simulation. These upscaling procedures will generally create full tensor permeabilities, even if the permeabilities in the fine geological grid are not represented as tensors. Full tensor permeabilities also arise in complex crossbedded systems where the dip directions will not in general coincide with a local coordinate direction. A further case for full tensor permeabilities is in the modeling of fractured systems.

Conventionally, reservoir simulators ignore the full tensor nature of the absolute permeability and assume the permeability can be represented by a diagonal tensor whose principal axes coincide with the local coordinate axes. This allows the construction of the usual 5 point discretization in 2D or 7 point in 3D which can be solved efficiently in the simulator. Introduction of a full tensor permeability extends the stencil of the discretization scheme to 9 points in 2D and either 19 or 27 points in 3D.

ECLIPSE 300 allows a full tensor description of the absolute permeability. The permeability tensor may either be supplied by a pre-processor by upscaling the properties on the geological model, or else it may be entered by keywords [PERMXX](#), [PERMYY](#), [PERMZZ](#) (which describe the diagonal components) and [PERMXY](#), [PERMYZ](#), [PERMZX](#) to describe the off-diagonal components.

Only positive definite symmetric tensors are allowed in the simulator. The principal directions of the tensor must also be supplied. At present, the options are

- the directions for the components of the tensor are aligned with the Cartesian axes;
- aligned with the axes joining the midpoints of each pair of opposite faces in corner point cells; and
- 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

$$\underline{K} = \begin{bmatrix} k_{xx} & k_{xy} & k_{xz} \\ k_{yx} & k_{yy} & k_{yz} \\ k_{zx} & k_{zy} & k_{zz} \end{bmatrix} \quad \text{Eq. 68.1}$$

where by symmetry

$$k_{yx} = k_{xy}, \quad k_{zy} = k_{yz}, \quad k_{zx} = k_{xz}. \quad \text{Eq. 68.2}$$

In ECLIPSE, the coefficients are entered as

$k_{xx} = \text{PERMXX}$, $k_{yy} = \text{PERMYY}$, $k_{zz} = \text{PERMZ}$, $k_{xy} = \text{PERMXY}$, $k_{yz} = \text{PERMYZ}$ and $k_{zx} = \text{PERMZX}$.

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. 1] 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 `MPFNCC` keyword. Further details can be found in [Ref. 43].

The use of the MPFA option is not allowed with the `WARP` solver. Please use the pre-2002A solver (`ECLSOL`) instead by setting `OPTIONS3` item 66 to a non-zero value.

Discretization

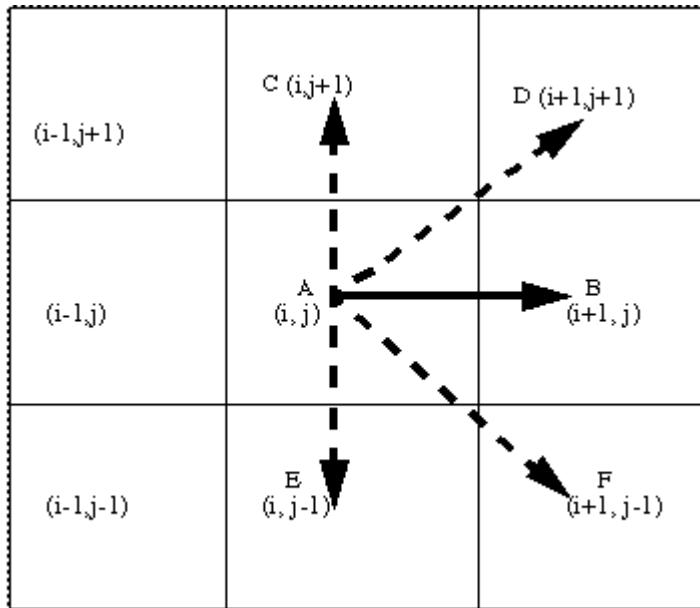


Figure 68.1. Stencil for MPFA flow in 2D between cells (i, j) and $(i + 1, j)$

Referring to "[Formulation of the equations](#)", the standard two-point flow between cells A = (i, j) and B = (i + 1, j) for the corner point grid in figure [68.1](#) can be written for a component c embedded in a phase p ($p = o, w, g$) as

$$F_{pAB}^c = T_{AB} M_{pAB}^c dP_{pAB} \quad \text{Eq. 68.3}$$

where

T_{AB} is the two-point transmissibility between cells A and B

M_{pAB}^c is the generalized mobility of component c in phase p , and is given by equation [23.26](#)

dP_{pAB} is the potential difference of phase p between cells A and B, given by

$$dP_{pAB} = P_A - P_B + P_{cpA} - P_{cpB} - \rho_{pAB} g (D_A - D_B) \quad \text{Eq. 68.4}$$

and where

P_{cp} is the capillary pressure for phase p

ρ_p is the mass density of phase p

g is the acceleration due to gravity

D is the cell center depth.

Alternatively, the MPFA constructs the flow F_{pAB}^c which depends on steering terms from the neighboring cells C, D, E and F in figure [68.1](#) as

$$F_{pAB}^c = M_{pAB}^c \left(T_{AB} dP_{pAB} + \sum_S T_{AS} dP_{AS} \right) \quad \text{Eq. 68.5}$$

where the summation is over *steering* transmissibilities S between cells AC, AD, AE and SF.

The mobility M_{pAB}^c is evaluated in the upstream cell (A or B) according to the direction of flow in equation [68.5](#).

The steering potential differences dP_{AS} are evaluated using pressures and capillary pressures for cells A and S but using the density term ρ_{AB} for the primary flow coefficient obtained from equation [23.36](#), so that

$$dP_{pAS} = P_A - P_S + P_{cpA} - P_{cpS} - \rho_{pAB} G (D_A - D_S) \quad \text{Eq. 68.6}$$

The transmissibilities T_{AB} and T_{AS} are calculated following the procedure described in [Ref. 1] when the permeability 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](#)").

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").

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 $\underline{K} = \text{diag}(\text{PERMX}, \text{PERMY}, \text{PERMZ})$ 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

- the grid system is orthogonal and
- the permeability tensor \underline{K} is diagonal

In general, when the grid is either non-orthogonal or \underline{K} is a full tensor then a 9-point (2D) or 27 point (3D) discretization results. It is possible to control the application of the MPFA scheme based on a cell orthogonality error indicator. Essentially for grids which are nearly orthogonal it is desirable to apply the MPFA discretization only in regions of high non-orthogonality to reduce the CPU cost. In the context of a full tensor permeability, an orthogonality indicator can be defined as follows:

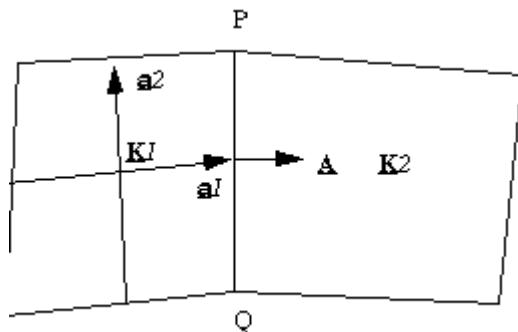


Figure 68.2. K -orthogonality error indicator

Consider an interface PQ between two cells 1 and 2. The permeabilities in each cells are \underline{K}_1 and \underline{K}_2 . The directed area normal on the interface is denoted by 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

$$\underline{a}_i \cdot \underline{a}^j = \delta_{ij}$$

where δ_{ij} is the Kronecker delta.

Now consider a potential gradient in cell 1,

$$\nabla \varphi = \sum_{j=1}^2 \alpha_j \underline{a}^j$$

The flux through PQ from cell 1 is therefore

$$f = \lambda \cdot \underline{A} \cdot \underline{K} \cdot \nabla \varphi = \lambda \sum_{j=1}^2 \alpha_j \underline{A} \cdot \underline{K} \cdot \underline{a}^j$$

where λ accounts for the mobility.

Since this expression can be used for arbitrary values of α_j , it follows that if $\underline{A} \cdot \underline{K}$ is parallel to \underline{a}_1 , such that $\underline{A} \cdot \underline{K} \cdot \underline{a}^2 = 0$, then the flux depends only on the gradient in the covariant direction \underline{a}_1 and implies that a two point discretization can be employed.

The grid is said to be *K-orthogonal* (see [Ref. 43] and [Ref. 1]) when $\underline{A} \cdot \underline{K}$ is parallel to \underline{a}_1 .

A convenient measure of orthogonality error for an interface is therefore

$$\varepsilon = \| \underline{A} \cdot \underline{K} - \frac{[(\underline{A} \cdot \underline{K}) \cdot \underline{a}_1] \underline{a}_1}{\| \underline{a}_1 \cdot \underline{a}_1 \|} \| / \| \underline{A} \cdot \underline{K} \|$$

This is evaluated separately for each of the two cells adjoining the interface 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](#))
- The WARP solver (keyword [WARP](#))

Thermal option

	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](#)")
- combustion (see "[Combustion](#)").

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](#)")

'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](#)".

The thermal option can be used for dual porosity grids, but the combination of the thermal option and dual porosity makes extra assumptions about the distribution of the rock volume between the porosities: see "[Dual porosity](#)".

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](#)".

The next section contains a list of keywords that can only be used with the Thermal live oil and dead oil options, and those ECLIPSE keywords that have special meaning when used with the Thermal option. This is followed by sections describing the thermal properties and features that can be modeled. Example data sets are also provided.

Formulation

The equations used to describe thermal processes are similar to those used in the compositional simulation, "Formulation of the equations", 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

$$X = (P, m_1, \dots, m_N, m_w, e)$$

where

P is the pressure,

m_1, \dots, 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:

$$R_{fl} = \frac{d}{dt} (V_p m_{fl}) + F_{fl} + Q_{fl} = 0 \quad \text{Eq. 69.1}$$

where

V_p is the pore volume,

F_{fl} is the net flow rate into neighboring grid blocks and

Q_{fl} is the net flow rate into wells during the timestep.

The non-linear residual, R_e , is derived from the energy conservation equation in each grid block at each timestep:

$$R_e = \frac{d}{dt} (V_b e) + F_e + C_e + Q_{HL} + Q_e = 0 \quad \text{Eq. 69.2}$$

where

V_b is the bulk volume,

F_e is the convective enthalpy flow rate into neighboring grid blocks,

C_e is the conductive energy flow rate into neighboring grid blocks,

Q_{HL} is the conductive energy flow rate to the surrounding rocks (heat loss), and

Q_e is the net enthalpy flow rate into wells during the timestep.

The non-linear residual, R_v , is derived from the conservation of volume in each grid block at each timestep:

$$R_v = V_p - V_f = 0 \quad \text{Eq. 69.3}$$

where V_f is the fluid volume.

Thermodynamic equilibrium condition

The phase properties used in the residual and Jacobian equations (that is the fluid volume and the flow terms) can be calculated from the component properties once the mole fractions of each component in each phase have been determined.

The number of moles of each component (including water) can be written as

$$m_c = m_t \cdot L \cdot x^c_o + m_t \cdot V \cdot x^c_g + m_t \cdot W \cdot x^c_w \quad \text{Eq. 69.4}$$

where

$m_t \cdot L \cdot x^c_o$, $m_t \cdot V \cdot x^c_g$ and $m_t \cdot W \cdot x^c_w$ are the number of moles in the oil, vapor and water phases.

m_t is the total fluid molar density given by

$$m_t = \sum_c m_c + m_w \quad \text{Eq. 69.5}$$

L , V and W are the oil, vapor and water mole fractions and

x_p^c 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:

$$x_w^c = 0,$$

$$x_w^c = 0, c = 1, \dots, N$$

$$x_w^w = 1,$$

if $W > 0$

It is also assumed the three phases are in thermodynamic equilibrium, determined by user defined K-values:

$$x_o^c = K^c(P, T) \cdot x_o^c \quad \text{Eq. 69.6}$$

$$x_g^w = K^w(P, T) \cdot x_g^w \quad \text{Eq. 69.7}$$

The K-values are functions of pressure P and temperature T only.

Using these equations the mole fractions of each component in each phase can be expressed in terms of the primary variables and the four flash variables, L , V , W , and T :

$$x_o^c = \frac{m_c}{m_t} \cdot \frac{1}{L + K^c \cdot V} \quad c = 1, \dots, N \quad L > 0 \quad \text{Eq. 69.8}$$

$$x_g^c = \frac{m_c}{m_t} \cdot \frac{K^c}{L + K^c \cdot V} \quad c = 1, \dots, N \quad V > 0 \quad \text{Eq. 69.9}$$

$$x_g^w = K^w \quad V > 0 \quad W > 0 \quad \text{Eq. 69.10}$$

Total mole fractions

The total mole fractions, which are used in equations 69.8 and 69.9 are defined as

$$z_c = \frac{m_c}{m_T} \quad z_w = \frac{m_w}{m_T}$$

Equation 69.5 can be written in terms of the mole fractions

$$\sum_c z_c + z_w = 1 \quad \text{Eq. 69.11}$$

The total mole fractions differ from the hydrocarbon mole fractions:

$$\zeta_c = \frac{m_c}{m_T - m_w} = \frac{z_c}{1 - z_w} \quad \text{Eq. 69.12}$$

Substituting this into equation 69.11 gives

$$\sum_c \zeta_c = 1 \quad \text{Eq. 69.13}$$

Thermal flash

The four flash variables can be found from the primary variables by solving the four flash equations in each cell:

$$\sum_c x_o^c = 1 \quad \text{if } L > 0 \quad \text{Eq. 69.14}$$

$$\sum_c x_g^c + x_g^w = 1 \quad \text{if } V > 0 \quad \text{Eq. 69.15}$$

$$L + V + W = 1 \quad \text{Eq. 69.16}$$

and

$$V_b \cdot e = V_p \cdot m_t \cdot (L \cdot e_o + V \cdot e_g + W \cdot e_w) + (V_b - V_p) \cdot e_r \quad \text{Eq. 69.17}$$

where

e_o , e_g and e_w are the user defined molar energies of the oil, gas and water phases, and

e_r is the user defined energy per unit volume of rock.

Fluid volume

The specific volume of each phase can be determined from user defined molar densities for each phase (b_o , b_g and b_w):

$$V_o = \frac{m_t L}{b_o}, \quad V_g = \frac{m_t V}{b_g}, \quad V_w = \frac{m_t W}{b_w}, \quad \text{Eq. 69.18}$$

The volume balance residual equation 69.3 can then be written as:

$$R_v = V_p - V_p \cdot (V_o + V_g + V_w) = 0 \quad \text{Eq. 69.19}$$

The phase saturations S_o , S_g and S_w are defined as

$$S_o = \frac{V_o}{V_o + V_g + V_w}, \quad S_g = \frac{V_g}{V_o + V_g + V_w}, \quad S_w = \frac{V_w}{V_o + V_g + V_w}, \quad \text{Eq. 69.20}$$

When equation 69.19 holds, the phase saturations are equal to the phase specific volumes. Therefore

$$S_o = \frac{m_t L}{b_o} \quad S_g = \frac{m_t V}{b_g} \quad S_w = \frac{m_t W}{b_w} \quad \text{Eq. 69.21}$$

The number of moles of each component equation 69.4 and the energy density, equation 69.17 can now be written in terms of the saturations:

$$m_c = S_o \cdot b_o \cdot x_o^c + S_g \cdot b_g \cdot x_g^c + S_w \cdot b_w \cdot x_w^c \quad \text{Eq. 69.22}$$

$$V_b \cdot e = V_p \cdot (S_o \cdot b_o \cdot e_o + S_g \cdot b_g \cdot e_g + S_w \cdot b_w \cdot e_w) + (V_b - V_p) \cdot e_r \quad \text{Eq. 69.23}$$

These two equations can be used for initializing the simulation - when the primary solution variables need to be calculated from user defined saturations.

Inter-block flows

The net flow rate of a hydrocarbon or water component c from cell i into neighboring cells is obtained by summing the component flow over all phases p and over the neighboring cells n ,

$$F_{ci} = \sum_n \sum_p M_p^c \cdot dP_{pni} \quad \text{Eq. 69.24}$$

The net flow rate of energy from cell i into neighboring cells is obtained in a similar manner:

$$F_{ei} = \sum_n \sum_p M_p^e \cdot dP_{pni} \quad \text{Eq. 69.25}$$

In these formulae

- γ_{ni} is the transmissibility between cells n and i ,
- M_p^c is the generalized mobility of component c in phase p ,
- M_p^e is the generalized mobility of enthalpy in phase p and
- dP_{pni} is the potential difference of phase p between cells n and i .

The generalized mobilities are given by

$$M_p^c = x_p^c \cdot k_{rp}(S_p) \cdot \frac{b_p}{\mu_p} \quad \text{Eq. 69.26}$$

and

$$M_p^e = H_p \cdot k_{rp}(S_p) \cdot \frac{b_p}{\mu_p} \quad \text{Eq. 69.27}$$

where

x_p^c is the mole fraction of component c in phase p ,

H_p is the molar enthalpy in phase p .

k_{rp} is the relative permeability of phase p ,

S_p is the saturation of phase p ,

b_p is the molar density of phase p , and

μ_p is the viscosity of phase p

The fluid mobilities are evaluated in the upstream cell for each phase p separately (cell n if dP_{pni} is positive, cell i if dP_{pni} is negative). The upstream calculation applies separately for each phase (oil, water, gas) so that, for example, oil may flow from cell i to cell n while water flows from cell n to cell i .

The potential difference is made up of a pressure term, a capillary pressure term and a hydrostatic term,

$$dP_{pni} = P_n - P_i + P_{cpn} - P_{cpi} - \rho_{pni} g (D_n - D_i) \quad \text{Eq. 69.28}$$

where

P_{cp} is the capillary pressure for phase p ,

ρ_p is the mass density of phase p ,

g is the acceleration due to gravity, and

D is the cell center depth.

Well flows

The rate of flow of component c into a production well w from cell i is obtained by summing the component flow over all phases p :

$$Q_{cwi} = -\gamma_{wi} \cdot \sum_p (P_{pi} - H_{iw} - P_{bh}) \cdot M_{pi}^c \quad \text{Eq. 69.29}$$

Similarly the rate of flow of energy into a production well w from cell i is

$$Q_{ewi} = -\gamma_{wi} \cdot \sum_p (P_{pi} - H_{iw} - P_{bh}) \cdot M_{pi}^e \quad \text{Eq. 69.30}$$

Here

γ_{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](#)" for further information.

Heat conduction

The heat conduction term for each cell i is given by summing conduction between all neighboring cells n :

$$C_{ei} = -\sum_n \psi_{ni} \cdot A \cdot (T_i - T_n) \quad \text{Eq. 69.31}$$

where

ψ_{ni} is the conduction transmissibility between cells n and i

A is the upstream saturation dependent conductivity multiplier.

Solid phase

Solid components can be used in thermal simulations. A solid component cannot also appear in the liquid or gas phase, but it can be created by chemical reactions, see "[Chemical reactions](#)". If solid components are modeled, then a number of modifications are required to the formulation described above.

The volume balance residual equation 69.3 is modified to include the solid volume V_s :

$$R_v = V_p - V_f - V_s = 0$$

The total fluid molar density m_t is still calculated by equation 69.5, but the sum is taken over the fluid components only.

The energy density equation, 69.17, is modified to include the solid energy e_s :

$$V_b \cdot e = V_p \cdot m_t \cdot (L \cdot e_o + V \cdot e_g + W \cdot e_w) + V_p \cdot e_s + (V_b - V_p) \cdot e_r$$

Equation 69.19, which is derived from the volume balance equation, is replaced by

$$R_v = V_p - V_p \cdot (V_o + V_g + V_w + V_{sol}) = 0$$

where V_{sol} is the specific solid volume.

The equations for the saturations, equation 69.20 are modified to include the solid volume:

$$\hat{S}_o = \frac{V_o}{V_o + V_g + V_w + V_{sol}} = \frac{V_f}{V_f + V_{sol}} \cdot S_o$$

$$\hat{S}_g = \frac{V_g}{V_o + V_g + V_w + V_{sol}} = \frac{V_f}{V_f + V_{sol}} \cdot S_g$$

$$\hat{S}_w = \frac{V_w}{V_o + V_g + V_w + V_{sol}} = \frac{V_f}{V_f + V_{sol}} \cdot S_w$$

$$\hat{S}_{sol} = \frac{V_{sol}}{V_o + V_g + V_w + V_{sol}}$$

The modified saturations satisfy

$$\hat{S}_o + \hat{S}_g + \hat{S}_w + \hat{S}_{sol} = 1$$

Equation 69.20 is still used to define normalized saturations S_o , S_g and S_w , representing the fraction of the fluid volume that is in each fluid phase. The normalized saturations satisfy

$$S_o + S_g + S_w = 1$$

The normalized saturations are used in calculating the relative permeabilities in equations 69.26 and 69.27.

Solution method

For each cell there are $N + 3$ variables (pressure, N hydrocarbon components, one water component, and energy) and $N + 3$ equations ($N + 1$ component conservation equations, including water, equation 69.1, energy conservation, equation 69.2 and volume balance, equation 69.3). The equations can be solved using a fully implicit method ([FULLIMP](#) / default), an implicit pressure, explicit properties method ([IMPES](#)) or a combination of both, ([AIM](#)).

The component conservation equations are written as

$$R_{fl} = \frac{V_p(P_{t+\Delta t}) \cdot m_{fl}(t+\Delta t) - V(P_t) \cdot m_{fl}(t)}{\Delta t} + F_{fl} + Q_{fl} = 0$$

where, in an implicit cell the flow terms and well terms are evaluated at the new time:

$$F_{fl} = F_{fl}(P_{t+\Delta t}, T_{t+\Delta t}, S_{t+\Delta t})$$

$$Q_{fl} = Q_{fl}(P_{t+\Delta t}, T_{t+\Delta t}, S_{t+\Delta t})$$

and in the IMPES method the flow terms and well terms are evaluated using the new pressure but with properties at the previous timestep:

$$F_{fl} = F_{fl}(P_{t+\Delta t}, T_t, S_t)$$

$$Q_{fl} = Q_{fl}(P_{t+\Delta t}, T_t, S_t)$$

The energy conservation equation is dealt with in a similar manner, but the volume balance equation is always evaluated at the new timestep.

The non-linear residual equations

$$R(X) = 0$$

are solved iteratively,

$$X_n = X_{n-1} + \Delta X_{n-1} \quad \text{Eq. 69.32}$$

The solution increment ΔX_{n-1} is found by solving the linear matrix equation

$$0 = R(X_{n-1}) + J \cdot \Delta X_{n-1}$$

where J is the Jacobian matrix:

$$J = \left(\frac{\partial R}{\partial X} \right) \Big|_{X=X_{n-1}}$$

The diagonal Jacobian element corresponding to an IMPES cell has a simple form which can be solved relatively easily:

$$J = \begin{pmatrix} P & m_1 & m_2 & \dots & e \\ \text{Comp1} & x & x & 0 & \dots & 0 \\ \text{Comp2} & x & 0 & x & \dots & 0 \\ \dots & \dots & \dots & \dots & \dots & \dots \\ \text{Energy} & x & 0 & 0 & \dots & x \\ \text{Volume bal} & x & x & x & \dots & x \end{pmatrix}$$

Linear solvers

To compute the solution increment ΔX the linear system involving the Jacobian matrix must be inverted. Whilst the default WARP linear solver may be adequate for relatively simple simulations better performance is usually achieved with the JALS solver, which is used by default for thermal simulations. The JALS linear solver employs the constrained pressure residual (CPR) technique in which an approximate pressure system is solved for each outer iteration. The solution of the pressure system is referred to as the inner iteration. The GMRES algorithm is used for both the outer and inner iteration. The GMRES algorithm is similar to the ORTHOMIN algorithm and requires storing a set of search directions. The JALS solver options can be controlled with the [JALS](#) keyword, which can be inserted in the RUNSPEC section.

Thermal properties

Rocks

Normally in ECLIPSE, a cell is inactive if its porosity is zero. However, in the thermal option, rock filled cells are considered as active cells since they conduct heat. Therefore, a cell is inactive if both its pore volume and rock volume are zero, or, if the [ACTNUM](#) keyword has been used to make it inactive.

Rock heat capacity

The enthalpy per unit volume of rock is given by

$$H_r = C_{r0}(T - T_{\text{ref}}) + \frac{C_{r1}(T - T_{\text{ref}})^2}{2} \quad \text{Eq. 69.33}$$

where

the volumetric heat capacity C_{r0} is defined by the [HEATCR](#) keyword,

the temperature coefficient C_{r1} is defined by the [HEATCRT](#) keyword and

the reference temperature T_{ref} is specified with the [STCOND](#) keyword.

Thermal conductivity

There are two ways to define the thermal conductivity, triggered using either the [THCONR](#) or [THCROCK](#) keyword in the model:

- Using keyword [THCONR](#) to define the thermal conductivity of a rock and liquid filled cell, and optionally [THCONSF](#)
- Using keyword [THCROCK](#), and depending on the phases present in the model, the phase thermal conductivities, [THCOIL](#), [THCWATER](#), [THCGAS](#) and [THCSOLID](#).

For both options, we define the cell average thermal conductivity, [THCAVE](#), which is used to determine the thermal transmissibilities. Depending on the option activated, thermal transmissibilities can be multiplied by a saturation-dependent factor. More details are supplied below.

The keywords from these two separate options must not be mixed together in the dataset.

Option 1: using THCONR

The [THCONR](#) keyword defines the thermal conductivity of a rock and liquid filled cell.

It can be calculated from the rock, oil, gas and water conductivities (K_r , K_o , K_g , K_w) by first determining an average liquid conductivity

$$K_l = (K_o + K_w)/2$$

The conductivity for a liquid filled cell can then be calculated as a porosity weighted average of the liquid and rock conductivities:

$$\text{THCAVE} = \text{THCONR} = \theta \cdot K_l + (1-\theta) \cdot K_r$$

[THCAVE](#) is used to calculate the thermal transmissibilities in the reservoir.

Option 2: using THCROCK

Depending on the phases in the model, the average cell thermal conductivity ([THCAVE](#)) is defined as a porosity weighted average of the phase and rock conductivities using the expression below:

$$\text{THCAVE} = \varphi \frac{(\text{THCOIL} + \text{THCGAS} + \text{THCWATER} + \text{THCSOLID})}{\text{NPHAS}} + (1-\varphi)\text{THCROCK}$$

In this expression, φ is the grid porosity; NPHAS denotes the number of phases in the model; THCOIL, THCGAS, THCWATER and THCSOLID specify the phase thermal conductivities; and THCROCK specifies the rock thermal conductivity.

If some phases are not present in the model, then the relevant phase thermal conductivity will be ignored. THCAVE is used to calculate the thermal transmissibilities in the reservoir.

Thermal transmissibility

Thermal transmissibilities are calculated in a similar manner to the flow transmissibilities, with the thermal conductivity being used instead of the permeabilities. For example, using corner point geometry, the thermal transmissibility between cells i and j is given by $H^{i \rightarrow j}$ where

$$1/H^{i \rightarrow j} = 1/H_i + 1/H_j \quad \text{Eq. 69.34}$$

$$H_i = \text{THCAVE}_i \cdot \frac{A \cdot D_i}{D_i \cdot D_i} \quad \text{Eq. 69.35}$$

$$A \cdot D_i = A_X \cdot D_{iX} + A_Y \cdot D_{iY} + A_Z \cdot D_{iZ} \quad \text{Eq. 69.36}$$

and

$$D_i \cdot D_i = D_{iX}^2 + D_{iY}^2 + D_{iZ}^2 \quad \text{Eq. 69.37}$$

Here

A_X , A_Y and A_Z are the X-, Y- and Z- projections of the mutual interface area of cell i and cell j ,

and D_{iX} , D_{iY} and D_{iZ} are the X-, Y- and Z- components of the distance between the center of cell i and the center of the relevant face of cell i .

See "[Transmissibility Calculations](#)" for further details.

Thermal transmissibilities can also be entered directly, using the keywords `HEATTX`, `HEATTY`, `HEATTZ` (or `HEATTR`, `HEATTTHT`, `HEATTZ` in radial coordinates).

Definition of multiplier for Heat Conduction

Option 1: using THCONR

Thermal transmissibilities can be multiplied by a saturation dependent factor Λ . Following Coats [Ref. 24], it is assumed that the thermal transmissibility is a function of the gas saturation, S_g , only. This method does not distinguish between the thermal transmissibility of a rock filled with oil and the thermal transmissibility of a rock filled with water.

The multiplier Λ is defined to be:

$$\Lambda = (1 - \alpha S_g) \quad \text{Eq. 69.38}$$

where α is specified by the `THCONSF` keyword.

From the definition of THCONR above, a value for α can be determined as:

$$\alpha = \theta \cdot (K_l - K_g) / \text{THCONR}$$

To check these values, we can calculate the conductivities for a liquid filled cell $S_g = 0$ and a gas filled cell $S_g = 1$:

$$\text{THCONR} \cdot A(0) = \theta \cdot K_l + (1-\theta) \cdot K_r$$

$$\text{THCONR} \cdot A(1) = \theta \cdot K_g + (1-\theta) \cdot K_r$$

which are just the porosity weighted averages of the rock and fluid conductivities

Option 2: using THCRock

Thermal transmissibilities are multiplied by a saturation dependent factor A . For this option, A takes the form:

$$A = (\varphi (\sum_p S_p K_p) + (1-\varphi)K_{\text{rock}}) / \text{THCAVE}$$

In the above expression, φ is the porosity; [THCOIL](#) is denoted by K_{oil} , [THCGAS](#) is denoted by K_{gas} , [THCWATER](#) is denoted by K_{water} , [THCSOLID](#) is denoted by K_{solid} and [THCRock](#), specifying the rock thermal conductivity, is denoted by K_{rock} . The phase saturations are represented by S_p .

Dual porosity or dual permeability

A matrix-fracture thermal conductivity can be set using the [THCONMF](#) keyword. The [SIGMATH](#) keyword can be used to set the multiplier used in calculating the matrix-fracture thermal transmissibilities.

Hysteretic rock compaction

During high pressure cyclic steam injection (huff and puff), the rock in the reservoir may be stressed beyond its elastic limits. A simple hysteretic model of rock compaction, the BOBERG option of keyword [ROCKCOMP](#), designed specifically for these circumstances, can be used: see "Rock Compaction".

Relative permeability curves

Keywords [ENPTVT](#), [ENKRVT](#), and [ENPCVT](#) specify the temperature-dependence of relative permeability curves. The form of these keywords is similar to the standard ECLIPSE 100/300 end-point-scaling keywords [ENPTVD](#), [ENKRVD](#), and [ENPCVD](#). In the thermal option, end-point-scaling keywords cannot be used in the same data set as temperature-dependent relative permeability keywords.

Live oil

For a live oil run you must specify temperature dependent properties for each component. The properties required for live components are, K-values to determine equilibrium between the oil and gas phase, and the phase enthalpies, densities and viscosities. These properties are output to the print file in tabular form. The program PVT *i* can be used to output K-values, enthalpy and density data.

Volatility type

In a live oil run, not all the components need be live. The [CVTYPE](#) keyword can be used to define a component as non-volatile, non-condensable or solid.

non-volatile and non-condensable components

A component can be made non-volatile or non-condensable using the CVTYPE keyword. This keyword also resets the component critical temperature, to 104 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 . Four methods are available for specifying oil component K-values:

- Keyword KVCR uses the K-value correlation given in [Ref. 24] and [Ref. 28], where the K-value for each oil component is given by

$$K(P, T) = (A + B / P + C \cdot P) \cdot e^{-D/(T-E)} \quad \text{Eq. 69.39}$$

Some care needs to be taken when using this formula to ensure that K is always positive and increases as T increases. A simplified version that avoids these problems can be used:

$$K(P, T) = (B / P) \cdot e^{-D/T} \quad \text{Eq. 69.40}$$

Values for B and D can be calculated by setting

$$K = P_{sat} / P$$

and finding the best straight line

$$\ln(P_{sat}) = \ln(B) - D / T$$

through a set of points $(1/T, P_{sat})$. Suggested values for some components are given in the following tables.

		B (Psi)	D (oR)
C1	Methane	168900	1904
C2	Ethane	170400	3062
C3	Propane	528400	4571
C6	Hexane	686800	6598

		B (Psi)	D (oR)
C10	Decane	1214000	9966

Table 69.1: K-value coefficients (Field 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

Table 69.2: K-value coefficients (Metric units)

- $\text{THCAVE} = \text{THCONR} = \theta \cdot K_l + (1-\theta) \cdot K_r$ It can be calculated from the rock, oil, gas and water conductivities (K_r, K_o, K_g, K_w) by first determining an average liquid conductivity. Keyword **KVWI** uses the Wilson correlation [Ref. 129], where the K-value for each oil component is given by

$$K(P, T) = e^{5.372697 \cdot (1+A_c) \cdot (1-T_{crit}/T)} \cdot P_{crit} \Big|_P \quad \text{Eq. 69.41}$$

This is the same as equation 69.40 with coefficients

$$B = e^{5.372697 \cdot (1+A_c)} \cdot P_{crit} \quad \text{Eq. 69.42}$$

$$D = 5.372697 \cdot (1+A_c) \cdot T_{crit}$$

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.

Surface K-values

The default K-values for surface calculations are the reservoir K-values. Different values can be used if required:

- If keyword **KVCR** is used to set the reservoir K-values, keyword **KVCRS** can be used to set the surface K-values
- 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, equation 69.41 for surface conditions.

Density

Oil phase density

The molar density b_o of the oil phase is given by:

$$b_o = 1 / V_{oil} \quad \text{Eq. 69.43}$$

where the molar specific volume V_{oil} of the oil phase is calculated using Amagat's law of partial volumes:

$$V_{oil} = \sum_{c=1}^{N_c} x^c V_{oil}^c \quad \text{Eq. 69.44}$$

and x^c is the mole fraction of component c in the oil phase. The component oil phase volume V_{oil}^c is given by:

$$V_{oil}^c = \frac{MW^c}{\rho^c} \quad \text{Eq. 69.45}$$

where

MW^c is the molecular weight of component c given by the [MW](#) keyword, and

the component oil phase density ρ^c is given by:

$$\rho^c = \frac{\rho_{ref}^c}{(1 + C_{T1}(T - T_{ref}))(1 - C_p(P - P_{ref}))} \quad \text{Eq. 69.46}$$

The reference density ρ_{ref}^c is defined by the [DREF](#) keyword; the standard temperature and pressure, T_{ref} and P_{ref} are defined by keywords [TREF](#) and [PREF](#) (or by [STCOND](#)); the thermal expansion coefficient C_{T1} is defined by the [THERMEX1](#) keyword; and the component isothermal compressibility C_p is defined by the [CREF](#) keyword.

Oil phase density using an EoS

An equation of state (EoS) can be used to compute the specific molar volume for the oil phase. The keyword [CODTYPE](#) specifies the components that form the mixture for the EoS calculation.

Let $x_e^c = \begin{cases} x^c & \text{for } c \text{ in EOS} \\ 0 & \text{otherwise} \end{cases}$ as defined by the keyword [CODTYPE](#)

and $x_n^c = \begin{cases} 0 & \text{for } c \text{ in EOS} \\ x^c & \text{otherwise} \end{cases}$ the components not included by the EoS calculation

$$\text{then } x_{esum} = \sum_{c=1}^{N_c} x_e^c \text{ and } x_{emix}^c = \frac{x_e^c}{x_{esum}}.$$

The oil specific molar volume is computed as

$$V_{oil} = \sum_{c=1}^{N_c} x_n^c V_{oil}^c + x_{esum} V_{eos}(P, T, x_{emix}^1, \dots, x_{emix}^{N_c})$$

There are two options when computing the EoS specific molar volume. Using the value [EMIX](#) for a component in [CODTYPE](#), the lowest root (Liquid root) of the cubic equation of state is chosen. For a mixture that cannot exist as an oil under the given pressure and composition, this volume could be unphysical. Another choice is to let the volume used correspond to the stable phase if the composition

$x_{emix}^1, \dots, x_{emix}^{N_c}$ was one-phase, by choosing the root with the lowest Gibbs free energy. If the value [EGFE](#) is selected for a component in [CODTYPE](#), this choice is applied. Typically this could lead to a gas-phase-like specific volume being used together with the overall oil specific volume.

Note: When only a subset of the components are selected for the EoS, usually all heavy components should be selected.

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 [69.45](#) and [69.46](#). The default, if these are not supplied, is to use the reservoir values given by [MW](#), [DREF](#), [TREF](#) and [PREF](#) (or [STCOND](#)).

Gas phase density

The molar density of the gas phase b_g is given by:

$$b_g = 1 / V_{gas} \quad \text{Eq. 69.47}$$

where the molar specific volume V_{gas} of the gas phase is

$$V_{gas} = \sum_{c=1}^{N_c} y^c V_{gas}^c + y^w V_{gas}^w \quad \text{Eq. 69.48}$$

and y^c is the mole fraction of component c in the gas phase. Each hydrocarbon component gas phase molar volume V_{gas}^c is obtained from the gas law,

$$PV_{gas}^c = Z^c RT \quad \text{Eq. 69.49}$$

The component Z-factor Z^c is given by an empirical formula

$$Z^c = Z_0^c - Z_1^c \cdot \frac{P}{RT} \quad \text{Eq. 69.50}$$

where the coefficients Z_0^c and Z_1^c are specified by the user with the [ZFACTOR](#) and [ZFACT1](#) keywords.

The molar specific water vapor volume V_{gas}^w used in equation [69.48](#) is calculated from the steam density.

Determining Z-factor coefficients from the Joule-Thomson coefficient

For an ideal gas the coefficients Z_0^c and Z_1^c are 1 and 0 respectively. The default values are in fact 0.96 and 0, which represents a non-ideal gas, but with constant Z-factors at reservoir conditions. This simple approach can be extended by calculating average reservoir values of Z_0^c in PVT i and exporting the data using the [ZFACTOR](#) keyword. However, in some circumstances it may be necessary to use a temperature dependent Z-factor by specifying values of the coefficient Z_1^c . It can be seen by substituting equation [69.50](#) into [69.49](#) that the coefficient Z_1^c has dimensions volume / mole:

$$V_{gas}^c = Z_o^c \cdot \frac{RT}{P} - Z_1^c \quad \text{Eq. 69.51}$$

Z_1^c can be determined from the Joule-Thomson coefficient μ . For a steady state (constant enthalpy) gas flow the change of temperature as a result of a pressure change is given by [Ref. 2]:

$$\mu = \left(\frac{\partial T}{\partial P} \right)_H = \frac{1}{(MW \cdot C_p)} \cdot \left(T \cdot \left(\frac{\partial V}{\partial T} \right)_P - V \right) \quad \text{Eq. 69.52}$$

where MW and C_p are the fluid molecular weight and specific heat. Substituting equations 69.48 and 69.52 gives

$$\mu = \sum_1^{N_c} y^c \cdot \mu^c + y^w \cdot \mu^w \quad \text{Eq. 69.53}$$

where

$$\mu^c = \frac{Z_1^c}{(MW_{gas} \cdot C_{gas})} \quad \text{Eq. 69.54}$$

Hence the coefficient Z_1^c can be determined using equation 69.54 from the component Joule-Thomson coefficient μ^c and the average molecular weight and heat capacity of the gas phase. The coefficient Z_1^c is also used to calculate the component enthalpy, equations 69.60 and 69.73.

Gas phase density using an EoS

An equation of state (EoS) can be used to calculate the specific molar volume for the gas phase. The keyword `CGDTYPE` specifies the components that form the mixture for the EoS calculation.

Let $y_e^c = \begin{cases} y^c & \text{for } c \text{ in EOS} \\ 0 & \text{otherwise} \end{cases}$ as defined by the keyword `CGDTYPE`

and $y_n^c = \begin{cases} 0 & \text{for } c \text{ in EOS} \\ y^c & \text{otherwise} \end{cases}$ the components not included by the EoS calculation

then $y_{esum} = \sum_{c=1}^{N_c} y_e^c$ and $y_{emix}^c = \frac{y_e^c}{y_{esum}}$.

The gas specific molar volume is computed as

$$V_{gas} = \sum_{c=1}^{N_c} y_n^c V_{gas}^c + y_{esum} V_{eos}(P, T, y_{emix}^1, \dots, y_{emix}^{N_c}) + y^w V_{gas}^w$$

There are two options when computing the EoS specific molar volume. By using the value `EMIX` for a component in `CGDTYPE`, the highest root (Vapor root) of the cubic equation of state is chosen. For a mixture that cannot exist as a gas under the given pressure and composition this volume could be unphysical. Another choice is to let the volume used correspond to the stable phase if the composition

$y_{emix}^1, \dots, y_{emix}^{N_c}$ was one-phase, by choosing the root having the lowest Gibbs free energy. If the value `EGFE` is selected for a component in `CGDTYPE`, this choice is applied. Typically this could lead to an oil-phase-like specific volume being used together with the overall gas specific volume.

Note: When only a subset of the components are selected for the EoS, usually all light components should be selected.

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"). This can be done using keywords `CCTYPE`. In this case equation 69.45 is replaced by:

$$V_{oil}^c = \frac{Z^c RT}{P} \quad \text{Eq. 69.55}$$

Keyword [CCTYPES](#) can be used to set (or reset) the compressibility at surface conditions.

Solid phase density

If a solid phase is modeled, the molar specific volume of solid phase components V_{sol}^c is calculated in a similar manner to the oil volume:

$$V_{sol}^c = \sum_{c=1}^{N_c} s^c V_{sol}^c \quad \text{Eq. 69.56}$$

where s^c is the mole fraction of component c in the solid phase. The component solid phase volume V_{sol}^c is given by:

$$V_{sol}^c = \frac{MW^c}{\rho^c} \quad \text{Eq. 69.57}$$

where MW^c is the molecular weight of component c given by the [MW](#) keyword, and the component solid phase density ρ^c is given by:

$$\rho^c = \frac{\rho_{ref}^c}{(1 + C_{T1}(T - T_{ref}))(1 - C_p(P - P_{ref}))} \quad \text{Eq. 69.58}$$

The reference density ρ_{ref}^c is defined by the [SDREF](#) keyword; the standard temperature and pressure, T_{ref} and P_{ref} are defined by keywords [STREF](#) and [SPREF](#) (or by [STCOND](#)); the thermal expansion coefficient C_{T1} is defined by the [STHERMX1](#) keyword; and the component isothermal compressibility C_p is defined by the [SCREF](#) keyword.

Enthalpy

The enthalpy of the oil phase is calculated using a mole fraction weighted average of the component enthalpies:

$$H_{oil}^c = \sum_{c=1}^{N_c} x^c \cdot MW^c \cdot H_{oil}^c(T) \quad \text{Eq. 69.59}$$

where x^c is the mole fraction of component c in the oil phase and MW^c is the component molecular weight given by the [MW](#) keyword.

The enthalpy of the hydrocarbon components in the gas phase is calculated using a mole fraction weighted average of the component enthalpies:

$$H_{gas}^c = \sum_{c=1}^{N_c} y^c \cdot (MW^c \cdot H_{gas}^c(T) + H_{JT}^c(P)) \quad \text{Eq. 69.60}$$

where y^c is the mole fraction of component c in the gas phase. The gas phase component enthalpies include both a temperature dependent term H_{gas}^c and a pressure dependent (Joule-Thomson) term H_{JT}^c .

The enthalpy of the hydrocarbon components in the solid phase is calculated using a mole fraction weighted average of the component enthalpies:

$$H_{sol} = \sum_{c=1}^{N_c} s^c \cdot MW^c \cdot H_{sol}^c(T) \quad \text{Eq. 69.61}$$

where s^c is the mole fraction of component c in the solid phase.

The component enthalpies in the oil and gas phases are calculated from component specific heats and heats of vaporization. Specific heat values must be given for each hydrocarbon component in at least one fluid phase. If they are only specified in one phase, then the heat of vaporization should be given. The following options are therefore available for the fluid phase enthalpies:

- Set the specific heat of components in the oil and gas phases.
- Set the specific heat of components in the oil phase and the heat of vaporization.
- Set the specific heat of components in the gas phase and the heat of vaporization.

Solid phase enthalpies are only required in simulations where a solid phase is present. The component enthalpies in the solid phase are calculated from component specific heats.

Specific heats

The specific heats C_{oil}^c , C_{gas}^c and C_{sol}^c of a component c in the oil, gas and solid phases are given by

$$C_{oil}^c = C_{oil1}^c + C_{oil2}^c \cdot (T - T_{st}) \quad \text{Eq. 69.62}$$

$$C_{gas}^c = C_{gas1}^c + C_{gas2}^c \cdot (T - T_{st}) \quad \text{Eq. 69.63}$$

$$C_{sol}^c = C_{sol1}^c + C_{sol2}^c \cdot (T - T_{st}) \quad \text{Eq. 69.64}$$

where the oil phase coefficients are defined by keywords [SPECHA](#) and [SPECHB](#), the gas phase coefficients are defined by keywords [SPECHG](#) and [SPECHH](#) and the solid phase coefficients are defined by keywords [SPECHS](#) and [SPECHT](#). T_{st} is the standard temperature defined by [STCOND](#). The specific heats must always be positive.

If oil phase specific heats are defined, the molar enthalpy of component c in the oil phase is:

$$H_{oil}^c = \int_{T_{st}}^T C_{oil}^c \, dT = C_{oil1}^c \cdot (T - T_{st}) + \frac{1}{2} \cdot C_{oil2}^c \cdot (T - T_{st})^2 \quad \text{Eq. 69.65}$$

For temperatures above the critical temperature, $T > T_{crit}^c$, the oil phase molar enthalpy 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:

$$\begin{aligned} H_{gas}^c &= H_{vaps}^c + \int_{T_{st}}^T C_{gas}^c \, dT \\ &= H_{vaps}^c + C_{gas1}^c \cdot (T - T_{st}) + \frac{1}{2} \cdot C_{gas2}^c \cdot (T - T_{st})^2 \end{aligned} \quad \text{Eq. 69.66}$$

where H_{vaps}^c is the heat of vaporization at the standard temperature, defined by [HEATVAPS](#). The oil and gas phase component enthalpies are illustrated in the following figure.

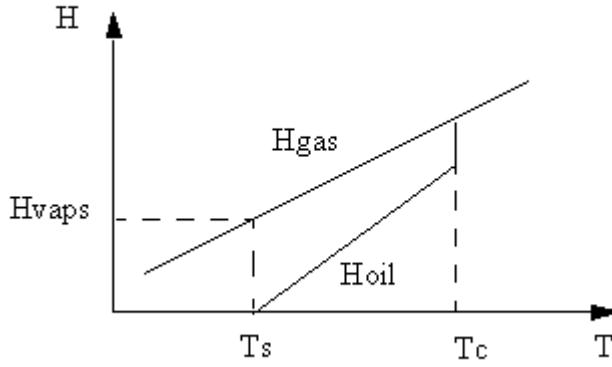


Figure 69.1. Component enthalpy defined by specific heats

If a solid phase is simulated, the molar enthalpy of component c in the solid phase is

$$H_{\text{sol}}^c = \int_{T_{\text{st}}}^T C_{\text{sol}}^c dT = C_{\text{sol}1}^c \cdot (T - T_{\text{st}}) + \frac{1}{2} \cdot C_{\text{sol}2}^c \cdot (T - T_{\text{st}})^2 \quad \text{Eq. 69.67}$$

Heat of vaporization

A temperature dependent heat of vaporization $\Delta H^c(T)$ can be specified instead of either the oil or gas specific heat.

If the specific heat of a component c in the oil phase is not specified, then the oil phase enthalpy is calculated from the gas component enthalpy by:

$$H_{\text{oil}}^c = H_{\text{gas}}^c - \Delta H^c(T) \quad \text{Eq. 69.68}$$

This is illustrated in figure 69.2 where a linear function has been used for the heat of vaporization $\Delta H^c(T)$.

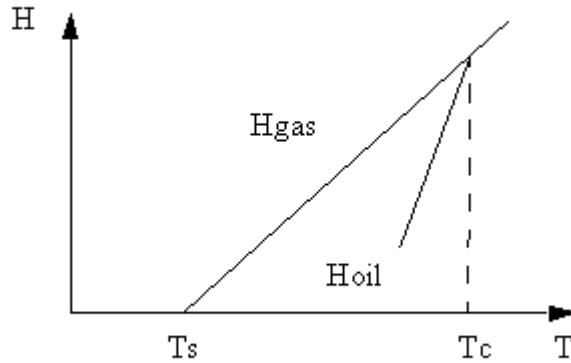


Figure 69.2. Component enthalpy defined by gas specific heat and heat of vaporization

Alternatively, if the specific heat of a component c in the gas phase is not specified, then the gas phase enthalpy is calculated from the oil component enthalpy

$$H_{\text{gas}}^c = H_{\text{oil}}^c + \Delta H^c(T) \quad \text{Eq. 69.69}$$

This is illustrated in figure 69.3 where a linear function has been used for the heat of vaporization $\Delta H^c(T)$.

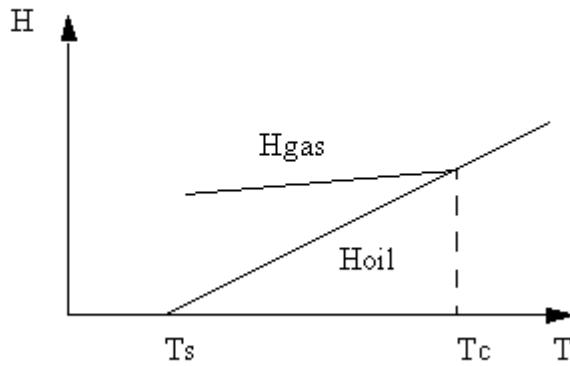


Figure 69.3. Component enthalpy defined by oil specific heat and heat of vaporization

The heat of vaporization is given by Edmister et al. [Ref. 33]:

$$\Delta H^c(T) = A \cdot (1 - T / T_{\text{crit}}^c)^B \quad T < T_{\text{crit}}^c \quad \text{Eq. 69.70}$$

$$\Delta H^c(T) = 0 \quad T \geq T_{\text{crit}}^c \quad \text{Eq. 69.71}$$

where the constant A is defined by the `HEATVAP` keyword, the exponent B is defined by the `HEATVAPE` keyword, and T_{crit}^c is the component critical temperature defined by the `TCRIT` keyword.

Heats of vaporization are usually obtained at the normal boiling point T_{nb} . The constant A , defined by the `HEATVAP` keyword can be found by inverting equation 69.70:

$$A = \frac{\Delta H^c(T_{nb})}{(1 - T_{nb} / T_{\text{crit}})^B} \quad \text{Eq. 69.72}$$

where $\Delta H^c(T_{nb})$ is the heat of vaporization at the normal boiling point. The exponent B , entered with keyword `HEATVAPE`, is defaulted to 0.38, which is the value found in the literature (Edmister et al. [Ref. 33]). However, this can lead to convergence problems, as discussed in the next section.

Pressure dependent gas enthalpy

The gas enthalpy equation 69.60 includes a pressure dependent term H_{JT}^c . This term is only used when the temperature dependent terms in the oil and gas enthalpies are set using specific heats (equations 69.65 and 69.66). It is not used when the heat of vaporization `HEATVAP` is used (equation 69.68 or 69.69). The pressure dependent term is given by

$$H_{JT}^c = -Z_1^c \cdot (P - P_{st}) \quad \text{Eq. 69.73}$$

where Z_1^c is specified with the [ZFACT1](#) keyword and P_{st} is the standard pressure defined by [STCOND](#).

Restrictions on specifying enthalpy data

The component enthalpies must obey the following relationships

$$H_{\text{gas}}^c \geq H_{\text{oil}}^c \text{ for } T < T_{\text{crit}}^c$$

$$\frac{dH_{\text{oil}}^c}{dT} > 0$$

$$\frac{dH_{\text{gas}}^c}{dT} > 0$$

ECLIPSE will issue an error message if any of these relationships does not hold. Errors can occur when specific heats are incorrectly specified, see Figure 69.4; or when using a non-linear heat of vaporization, see figure 69.5. These problems can be avoided by specifying specific heats for both phases, with the oil specific heats equal to the gas specific heats figure 69.6.

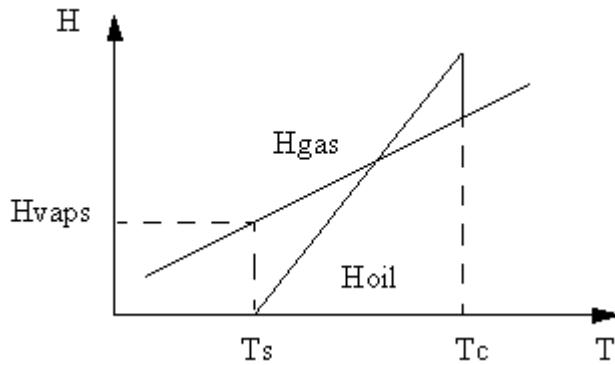


Figure 69.4. Specific heat data leading to $H_{\text{oil}} > H_{\text{gas}}$

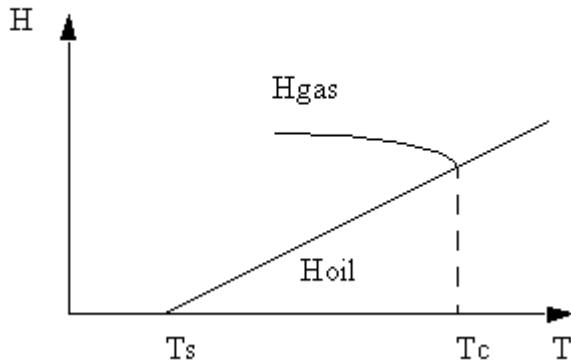


Figure 69.5. Heat of vaporization data leading to decreasing gas enthalpy

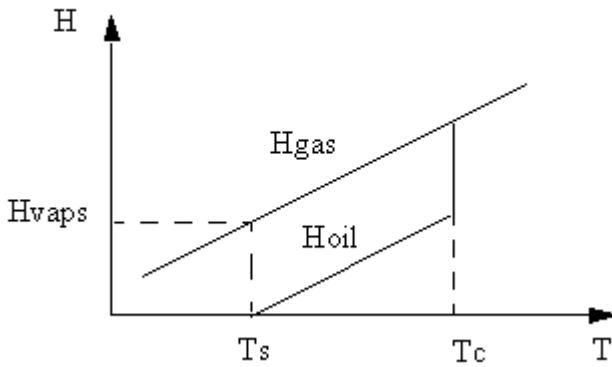


Figure 69.6. Component enthalpy defined by equal specific heats

Sources of enthalpy data

Molar specific heats are often quoted as a third order polynomial

$$C_p = A + B \cdot T + C \cdot T^2 + D \cdot T^3$$

These can be converted to ECLIPSE input values C_{p1} and C_{p2} used in equations 69.62 - 69.64 as follows:

$$C_{p1} = C_p(T_{st})/(MW) = (A + B \cdot T_{st} + C \cdot T_{st}^2 + D \cdot T_{st}^3)/(MW) \quad \text{Eq. 69.74}$$

$$C_{p2} = \frac{dC_p}{dT}|_{T_{st}} / (MW) = (B + 2 \cdot C \cdot T_{st} + 3 \cdot D \cdot T_{st}^2) / (MW) \quad \text{Eq. 69.75}$$

where MW is the component molecular weight.

Note: The ECLIPSE specific heat input data is a function of the standard temperature T_{st} set by keyword [STCOND](#)

Specific heats calculated using equations 69.74 and 69.75 from data in Reid et al. [Ref. 90] are shown in tables 69.3 and 69.4.

The following table contains calculated specific heats in metric units at $T_s = 20^\circ\text{C}$ for selected components. These are calculated from data in Reid et al. [Ref. 90].

Component	C_{p1} kJ/kg/K	C_{p2} kJ/kg/K ²
Methane	2.199	3.505e-3
Ethane	1.725	4.645e-3
Propane	1.650	5.026e-3
n-Butane	1.669	4.570e-3
Isobutane	1.656	4.885e-3
n-Pentane	1.641	4.847e-3

Component	C_{p1} kJ/kg/K	C_{p2} kJ/kg/K ²
Isopentane	1.621	5.008e-3
n-Hexane	1.637	4.826e-3
n-Heptane	1.633	4.809e-3
n-Octane	1.630	4.798e-3
n-Nonane	1.627	4.808e-3
n-Decane	1.625	4.779e-3
n-Unadecane	1.624	4.772e-3
Dodecane	1.623	4.766e-3
n-Tridecane	1.621	4.763e-3
n-Tetradecane	1.620	4.758e-3
n-Pentadecane	1.619	4.754e-3
Hexadecane	1.618	4.752e-3
n-Heptadecane	1.618	4.751e-3
Octadecane	1.618	4.747e-3
n-Nonadecane	1.616	4.743e-3
n-Eicosane	1.615	4.775e-3

Table 69.3: Calculated specific heats in metric units at $T_s = 20$ degrees C

The following table contains calculated specific heats in field units at $T_s = 20$ °C (68 °F) for selected components. These are calculated from data in Reid et al. [Ref. 90]

Component	C_{p1} Btu/lb/°R	C_{p2} Btu/lb/°R2
Methane	5.226e-1	4.659e-4
Ethane	4.080e-1	6.201e-4
Propane	3.895e-1	6.718e-4
n-Butane	3.946e-1	6.099e-4
Isobutane	3.911e-1	6.528e-4
n-Pentane	3.876e-1	6.480e-4
Isopentane	3.827e-1	6.696e-4
n-Hexane	3.866e-1	6.452e-4
n-Heptane	3.857e-1	6.430e-4
n-Octane	3.849e-1	6.415e-4
n-Nonane	3.843e-1	6.430e-4
n-Decane	3.838e-1	6.391e-4
n-Unadecane	3.836e-1	6.381e-4
Dodecane	3.833e-1	6.374e-4
n-Tridecane	3.829e-1	6.370e-4

Component	C _{p1} Btu/lb/°R	C _{p2} Btu/lb/°R2
n-Tetradecane	3.828e-1	6.363e-4
n-Pentadecane	3.825e-1	6.358e-4
Hexadecane	3.823e-1	6.355e-4
n-Heptadecane	3.821e-1	6.353e-4
Octadecane	3.821e-1	6.349e-4
n-Nonadecane	3.818e-1	6.344e-4
n-Eicosane	3.815e-1	6.387e-4

Table 69.4: Calculated specific heats in field units at Ts = 20 degrees C

Viscosity

Oil viscosity temperature dependence

Oil component viscosities can be input in functional form ([OILVISCC](#) or [OILVISCF](#)) or tabular form ([OILVISCT](#)). It is also possible to compute Lorentz-Bray-Clark viscosities for the oil phase by using the keyword [COVTYPE](#). The interpolation method for the tabular form can be specified using the [OILVTIM](#) keyword.

If the [OILVISCC](#) keyword is used the viscosity μ_o^c for oil component c can be specified using one of four standard formulae:

The ASTM correlation:

$$\log_{10}(\mu_o^c + A) = B \cdot T^{-C} \quad \text{Eq. 69.76}$$

The Andrade formula ([Ref. 82] and [Ref. 90]):

$$\log_{10}(\mu_o^c) = A + B / T \quad \text{Eq. 69.77}$$

The Vogel formula ([Ref. 82] and [Ref. 90]):

$$\log_{10}(\mu_o^c) = A + B / (T + C) \quad \text{Eq. 69.78}$$

A logarithmic formula:

$$\log_{10}(\mu_o^c) = A + B \cdot \log_{10}(T) \quad \text{Eq. 69.79}$$

In these equations:

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. 87]:

$$\mu_o^c = (\mu_{o1}^c + A)^{10^{-x}} - A \quad \text{Eq. 69.80}$$

where

$$x = C \cdot \log_{10}(T / T_1),$$

$(1 - A)$ is the limiting viscosity at infinite temperature,

$$B = \log_{10}(\mu_{o1}^c + A) \cdot T_1^{-C} \text{ and}$$

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

$$\mu_o^c = D \cdot T^B \quad \text{Eq. 69.81}$$

that is the same as the logarithmic formula specified by the [OILVISCC](#) keyword when $A = \log_{10}D$.

When all component viscosities have been calculated, the oil phase viscosity is calculated from

$$\log(\mu_o) = \sum_{c=1}^{N_c} f_c(x^c) \cdot \log\left(\mu_o^c\right) \quad \text{Eq. 69.82}$$

where the functions f_c are given by keyword [OILVINDEX](#), or are 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](#).

Lorentz-Bray-Clark oil viscosities

The oil phase viscosities can be computed by the Lorentz-Bray-Clark (LBC) method. Here the components that should be used to form the mixture for the LBC calculation are selected using the keyword [COVTYPE](#). The LBC viscosity is computed based on the fluid density, pressure and temperature. If [LMIX](#) is selected in this keyword, the components with this entry compute their densities and corresponding viscosities using the LBC method. If the [LEMIX](#) entry is present, the densities for these components are computed with an EoS and then applied for the LBC viscosity calculation. The entry [NO](#) means that the regular component viscosity calculation is applied for this component. Care must be taken when this method is applied for heavy oils.

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

$$\mu_g^c = A \cdot T^B \quad \text{Eq. 69.83}$$

where T is the temperature and c is the component number. The gas phase viscosity is calculated to be

$$\mu_g = \sum_{c=1}^{N_c+1} y^c \cdot \mu_g^c \quad \text{Eq. 69.84}$$

Lorentz-Bray-Clark gas viscosities

The gas phase viscosities can be computed by the Lorentz-Bray-Clark (LBC) method. Here the components that should be used to form the mixture for the LBC calculation are selected using the keyword `CGVTYPE`. The LBC viscosity is computed based on the fluid density, pressure and temperature. If `LMIX` is selected with this keyword, the components with this entry compute their densities and corresponding viscosities using the LBC method. If the `LEMIX` entry is present, the densities for these components are computed with an EoS and then applied for the LBC viscosity calculation. The entry `NO` means that the regular component viscosity calculation is applied for this component.

Example

For a two hydrocarbon component study, the PROPS section may look something like this

```

PROPS =====
-- Temperature variation with depth - deg F
TEMPVD
624 105 /
1500 105 /

-- Standard Conditions Temp deg F Press psia
STCOND
60 14.7 /

-- Water Saturation Rel Perm Function
SWFN
.2000 .0000 0.0
.4000 .00035 0.0
.5000 .00101 0.0
.6000 .00213 0.0
.7000 .0038 0.0
/

-- Gas Saturation Rel Perm Function
SGCN
.0000 .0000 0.0
.0200 .0000 0.0
.1000 .07059 0.0
.2000 .15882 0.0
.3000 .24706 0.0
.4000 .33529 0.0
.5000 .42353 0.0
.6000 .51176 0.0
.7000 .6000 0.0
/

-- Two Phase Oil Saturation Rel Perm Functions
SOF3
.2000 .0000 .0000
.3000 .0000 .1666
.4000 .02468 .33333
.5000 .12155 .500000
.6000 .30885 .66667
.7000 .59856 .83333
.8000 1.0000 1.0000
/

-- Stone 3-phase oil rel perm
STONE

-- Relative permeability endpoints vs temperature (deg F)
ENKRVFT
-- temp Krwmax Krgmax Kromax Krwro Krgro Krorg Krorg

```

```

100 1*      1*      1*      0.0038 1*      1*      1*      1*      /
450 1*      1*      1*      0.1      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 vaporization 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
SPECCHA
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
PTW
-- 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.

Specific heat

A constant specific heat is used with DEADOIL. It is defined using the [OILSPECH](#) keyword.

Density

The oil density is defined as

$$\rho_o = \frac{\rho_{ref,o}}{[1 - c_{Po}(P - P_{ref})][1 + c_{T1}(T - T_{ref}) + c_{T2}(T - T_{ref})^2]} \quad \text{Eq. 69.85}$$

where

the oil compressibility c_{Po} and the thermal expansion coefficients c_{T1} and c_{T2}

are defined by the [OILCOMPR](#) keyword,

$\rho_{ref,o}$

is defined by the [DENSITY](#) keyword and

T_{ref} and P_{ref}

are defined by the [STCOND](#) keyword.

Viscosity

Oil viscosity can be specified in tabular form ([OILVISCT](#)) or functional form ([THANALV](#)). The interpolation method for the tabular form can be specified using the [OILVTIM](#) method. The functional form follows Wingard [Ref. 130]:

$$\log_{10}(\mu_o) = A_o + B_o / T_K \quad \text{Eq. 69.86}$$

where

$A_o = -0.8772$, $B_o = 356.681$ and

T_K is the temperature in K.

This is the same as the live oil viscosity given by the Andrade formula, equation 69.77.

Oil viscosity pressure dependence

The oil viscosity can be modified by a factor $\mu(P)/\mu(P_{vref})$ where $\mu(P)$ and P_{vref} are defined by the keywords [PVCO](#) and [VISCREF](#).

Note: Oil viscosity in the [DEADOIL](#) option cannot be lower than 0.1 cP. It will be reset to 0.1 cP

Example

A typical set of dead oil keywords would be:

```
OILCOMPRESSIBILITY      therm. expansion coeff.
 5.00E-06               3.80E-04 0.0 /
OILSPECIFICHEAT          specific heat
 0.5 /                  0.5 /
OILMW                   --effective molecular weight
 600.0 /                600.0 /
DENSITY                 --for thermal, only oil density is needed.
 60.68 2* /              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 gm/gm-M.

K-values

The K-value of water, K_{ww} , is defined as the saturation pressure of water $P_b(T)$ divided by the pressure:

$$K_{ww} = \frac{P_b}{P} \quad \text{Eq. 69.87}$$

By default the saturation pressure and temperature are obtained from the formula of Farouq Ali [Ref. 36]:

$$P_b = \left(\frac{T}{a} \right)^{1/b} \quad \text{Eq. 69.88}$$

$$T_b = aP^b \quad \text{Eq. 69.89}$$

Ejiougu et al. [Ref. 34] give the following values for the constants a and b when the temperature is measured in °C, and in MPa:

$$a = 180.89, b = 0.2350$$

If the [THSTT97](#) keyword is used, the saturation pressure and temperature are calculated from an analytical formulation, as detailed by *The International Association for the Properties of Water and Steam (IAPWS-IF97)*.

Alternatively, the water K-value can be obtained from a correlation of the form

$$K_{ww} = \frac{P_b}{P} = (A + B / P + C \cdot P) \cdot e^{-D/(T-E)}$$

where the coefficients A , B , C , D and E are specified using the [KVCRWAT](#) keyword.

Water density

The water density can be calculated in one of three ways; using an internal analytic function; using an analytic function with user defined coefficients; or using internal steam tables.

Analytic formula with internally defined coefficients

Trangenstein's modification of Kell's 1975 correlations [Ref. 113] is used for the water density (default / [THANALB](#)):

$$\rho_w = \frac{A_0 + A_1 T_C + A_2 T_C^2 + A_3 T_C^3 + A_4 T_C^4 + A_5 T_C^5}{1 + A_6 T_C} e^{c_{Pw}(P_P - A_7)} \quad \text{Eq. 69.90}$$

where

ρ_w is in kg/m³,

the temperature T_C is in °C,

the pressure P_P is in MPa,

$$A_0 = 999.83952 \quad A_1 = 16.955176$$

$$A_2 = -7.987 \times 10^{-3} \quad A_3 = -46.170461 \times 10^{-6}$$

$$A_4 = 105.56302 \times 10^{-9} \quad A_5 = -280.54353 \times 10^{-12}$$

$$A_6 = -16.87985 \times 10^{-3} \quad A_7 = -10.2$$

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, equation, [69.46](#):

$$\rho_w = \frac{\rho_{ref,w}}{\left[1 - c_{Pw}(P - P_{ref})\right]\left[1 + c_{T1}(T - T_{ref}) + c_{T2}(T - T_{ref})^2\right]} \quad \text{Eq. 69.91}$$

where

the reference density is given by $\rho_{ref,w} = \frac{\rho_{s,w}}{B_w(P_{ref})}$ and

$\rho_{s,w}$ is the water density at surface conditions given by the [DENSITY](#) keyword

P_{ref} is the reference pressure given by the [PVTW](#) keyword

$B_w(P_{ref})$ is the formation volume factor given by the PVTW keyword

c_{Pw} is the water compressibility defined by the PVTW keyword or calculated from the WATERTAB data.

T_{ref} is the reference temperature given by the WATDENT keyword

c_{T1} is the first coefficient of thermal expansion, given by the WATDENT keyword

c_{T2} is the second coefficient of thermal expansion, given by the WATDENT keyword

Steam tables

If the [THTABB](#) keyword is used, the water density is calculated from steam tables [Ref. 113].

Industrial formulation

If the [THSTT97](#) keyword is used, the water density is based on an analytical formulation, as detailed by *The International Association for the Properties of Water and Steam* (IAPWS-IF97).

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:

$$\rho_g = \rho_{gb} \frac{T_b}{T} \quad \text{Eq. 69.92}$$

where

T_b is the boiling point temperature and

ρ_{gb} is the density at the boiling point.

The saturated water vapor density is taken from Tortike and Farouq Ali [Ref. 112]:

$$\rho_{gb} = \exp [C_0 + C_1 T_{bK} + C_2 T_{bK}^2 + C_3 T_{bK}^3 + C_4 T_{bK}^4 + C_5 T_{bK}^5] \quad \text{Eq. 69.93}$$

where,

ρ_{gb} is in kg/m³,

T_{bK} is the boiling point temperature in K and

$$C_0 = -93.7072 \quad C_1 = 0.833941 \quad C_2 = -0.003208$$

$$C_3 = 6.57652 \times 10^{-6} \quad C_4 = -6.93747 \times 10^{-9} \quad C_5 = 2.97203 \times 10^{-12}$$

Steam tables

If the [THTABB](#) keyword is used, the steam density is calculated from steam tables [Ref. 82].

Industrial formulation

If the [THSTT97](#) keyword is used, the steam density is based on an analytical formulation, as detailed by *The International Association for the Properties of Water and Steam* (IAPWS-IF97).

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. 82]. Tables from Haywood [Ref. 50] can be selected using the [OPTIONS3](#) keyword.

Analytic formulae, based on Perry's tables, can also be used, using keyword [THANALH](#). If the [THSTT97](#) keyword is used, the water and steam energies and enthalpies are based on an analytical formulation, as detailed by *The International Association for the Properties of Water and Steam* (IAPWS-IF97).

The enthalpy of pure water or steam increases with temperature, as shown below. It also varies with pressure, decreasing as pressure increases for steam (PT values below the saturation curve), but increasing for water (PT values above the saturation curve).

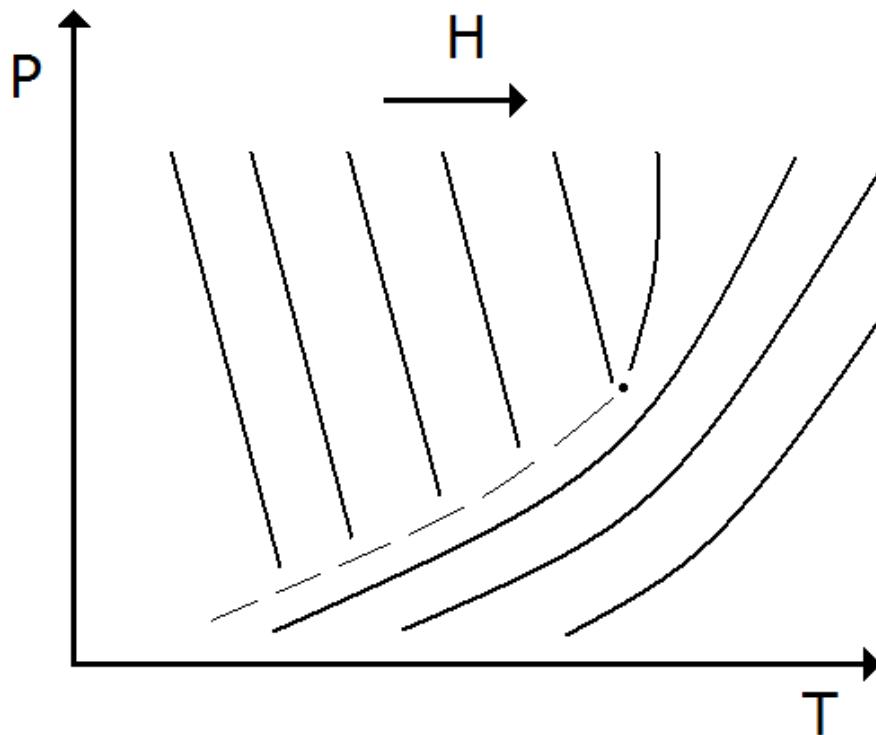


Figure 69.7. Constant enthalpy curves for pure water

For isoenthalpic flows of pure water, the change in temperature is fairly constant over a range of pressures and temperatures, increasing by approximately 1 °C for a 50 bar pressure drop (approximately 1 °F for a 400 psi pressure drop). For isoenthalpic flows of pure steam the temperature change is more variable, decreasing by the order of 1 °C for a pressure drop of 1 bar (approximately 1 °F for a 10 psi drop).

Water viscosity

The water viscosity can be defined in tabular form ([WATVISCT](#)) or in functional form (default). The functional form is taken from Grabowski et al. [Ref. 42]

$$\mu_w = \frac{A_w}{-1 + B_w T_F + C_w T_F^2} \quad \text{Eq. 69.94}$$

where

the temperature T_F is in °F and

the default coefficients are $A_w = 2.1850$ $B_w = 0.04012$ $C_w = 5.1547 \times 10^{-6}$

The coefficients maybe modified using the [THWVC](#) keyword.

Water viscosity pressure dependence

The water viscosity can be multiplied by a factor $\mu_p(P)/\mu_p(P_{vref})$ where μ_p and P_{vref} are defined by the keywords [PVTW](#) and [VISCREF](#).

Steam viscosity

The steam viscosity is taken from Ejiogu and Fiori [Ref. 34]

$$\mu_g = A_g + B_g T_C + C_g P_P^{D_g} \quad \text{Eq. 69.95}$$

where

the temperature T_C is in °C,

the pressure P_P is in MPa

and the default coefficients are:

$$A_g = 4.9402 \times 10^{-3}, \quad B_g = 5.0956 \times 10^{-5}, \quad C_g = 2.9223 \times 10^{-6}, \quad D_g = 2.5077$$

The coefficients maybe modified using the [THSVC](#) keyword.

Combustion

The chemical reaction keywords, "[Chemical Reactions](#)" can be used to model combustion. An example is given in "[Example 1: Combustion](#)".

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](#)". 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 [RPTSQL](#) or with the SUMMARY keywords 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
STOPPROD
      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, and the [ROCKCONL](#) keyword is used to specify the local grid refinement connection data for these cap and base rocks.

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

$$-k \cdot \frac{\partial T}{\partial n}|_{n=0}$$

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. 119]. 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 keywords FHLR and FHLT can be used to obtain information on the field heat loss rate and field heat loss total.

Initialization

Initial values of all the primary variables, P , m_c and e need to be determined at the beginning of the simulation. Four methods are available:

- The solution can be read from a save file or a RESTART file generated by an earlier run.
- The solution can be determined from properties set directly in each grid block.
- The solution can be calculated by the equilibrium facility.
- The pressure can be calculated by equilibrium and the other variables determined from properties set directly in each grid block.

Restarting from a previous run

There are two different ways of restarting from a previous run:

- With a fast restart the `LOAD` keyword can be used to read data that had been previously saved using the `SAVE` keyword. This will only work if the same version of the simulator is used for both runs.
- With a flexible restart the `RESTART` keyword can be used to read data that had been previously output using the `RPTRST` keyword.

Explicit initialization

In an explicit initialization the component molar densities, equation 69.22 and the energy density, equation 69.23 can be determined from the component phase mole fractions and the saturations, equation 69.21:

$$m_c = S_o \cdot b_o \cdot x_o^c + S_g \cdot b_g \cdot x_g^c + S_w \cdot b_w \cdot x_w^c \quad \text{Eq. 69.96}$$

$$e = \frac{V_p}{V_b} \cdot (S_o \cdot b_o \cdot e_o + S_g \cdot b_g \cdot e_g + S_w \cdot b_w \cdot e_w +) \left(1 - \frac{V_p}{V_b} \right) \cdot e_r \quad \text{Eq. 69.97}$$

The phase molar densities b^m_p and energies e_p are functions of the pressure, temperature, component phase mole fractions and the component and rock properties. You must therefore specify

- pressure [PRESSURE](#),
- temperature [TEMP1](#) or [TEMPVD](#),
- two of the three saturations [SOIL](#), [SGAS](#) and [SWAT](#).

Data can also be read from RESTART files using the [GETSOL](#) keyword.

You must also specify the component liquid and vapor mole fractions x_o^c and x_g^c . There are a three ways this can be done:

- directly using the [XMF](#) and [YMF](#) keywords,
- directly, as functions of pressure using the [XMFVP](#) and [YMFVP](#) keywords,
- using the [NEI](#) keyword to specify sample liquid mole fractions.

When specifying the initial component data directly, it is up to you to ensure the fluid composition is in thermodynamic equilibrium, see "[Thermodynamic equilibrium condition](#)". Otherwise, the solution and properties change the first time a flash calculation is performed. This difficulty is avoided if the [NEI](#) keyword is used.

Initialization with NEI

When the [NEI](#) keyword is used, the vapor mole fractions are set so that the composition is in thermodynamic equilibrium, equation 69.7. The simulator tests to see if this composition gives the correct phase saturations - if not it adjusts the composition of the component with the highest gas phase mole fraction, so that the phase saturations are exactly matched.

Equilibration

In the equilibrium calculation a steady state solution of the flow equations is sought. This can be achieved if the pressure gradient in equation 69.28 is zero. For thermal simulations the heat conduction terms, equation 69.31 should also be zero, although this will in fact be determined by the initial temperatures specified by the user. The equilibrium calculation is discussed in detail in the Equilibration section of [Initializing the study](#).

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 equation 69.12.

Mixed initialization

It is possible to engineer a mixed initialization where the pressures are calculated from a reference pressure at a given depth and the other properties are set explicitly. To do this an initial equilibration run is performed using the `EQUIL` keyword. The pressures from this run are output to a restart file:

```
SOLUTION
EQUIL
...      /      set pressure at reference depth
RPTRST
PRESSURE /      output pressure
...
SCHEDULE
END
```

The equilibration run can be stopped at the start of the `SCHEDULE` section. In the main run, explicit initialization is used, but the pressures are read from the equilibration run:

```
SOLUTION
GETSOL
PRESSURE EQUILRUN 0 /      get pressure from previous run
/
SOIL
...           /      set saturations
...
```

Although the equilibration run does not give the correct saturations, the pressures are reasonably accurate, since the oil and water densities are similar. These pressures are combined with the desired saturations in the main run.

Aquifers

The temperature of the aquifer water or gas is specified in the aquifer definition keyword (see the `AQUCT`, `AQUFETP`, `AQUCHWAT`, `AQUCHGAS` or `AQUFLUX` keywords) for analytic aquifers. If the temperature of the analytic aquifer is defaulted and the `TEMPVD` keyword is specified, the aquifer temperature will be interpolated from the `TEMPVD` table using the aquifer depth. Otherwise if the temperature of the analytic aquifer is defaulted and the `TEMPVD` keyword is not specified, the `RTEMP` temperature will be used.

The temperature of the numerical aquifer water is constant. If the `TEMPVD` keyword is used, the aquifer temperature will be the temperature at the aquifer depth. Otherwise the `RTEMP` temperature is used.

Heaters

Heaters can be placed in any grid cell using the `HEATER` keyword, or in an LGR cell using the `HEATERL` keyword. These keywords specify the maximum rate of heating H_{max} and a maximum temperature at which the heater should operate T_{max} . The number of heaters allowed is controlled by the `HEATDIMS` keyword.

Depending on the arguments in the `HEATER` keyword the heater rate is given either by

$$H = \min[(T_{max} - T_c) \times R, H_{max}]$$

where

T_c is the cell temperature and

R the temperature-dependent heat injection rate,

or

$$H = \min\left(H_{max}, V \cdot \frac{\Delta E}{\Delta t}\right) \quad \text{Eq. 69.98}$$

where

V is the cell volume, Δt is the timestep, 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 equation 69.98 will heat the connection cell at a constant rate H_{max} until the cell reaches a temperature T_{max} . However, if there is energy flow into or out of the cell, then equation 69.98 is only approximate. In this case the heater rate may be less than H_{max} for temperatures near T_{max} , or the cell temperature may exceed T_{max} slightly; the discrepancies depend on the size of the heat flow and the size of the timestep.

Output of heater performance data to the print file can be controlled by using item 14 of the [RPTPRINT](#) keyword.

SUMMARY keywords [WHTR](#) and [WHTT](#) contain the energy rate and energy total for individual heaters and require a list of heater names. Vectors [FHTR](#) and [FHTT](#) contain the energy rate and energy total for all heaters.

Well inflow

Keywords [COMPMOBI](#) and [COMPMBIL](#) can be used to set the mobility used in calculating the completion flow for injection wells. Alternatively keywords [COMPKRI](#) and [COMPKRIL](#) can be used to set the relative permeability used in calculating the completion flow for injection wells.

Use of these keywords may improve the speed and stability of thermal runs - although there may be a slight loss in accuracy of the calculated injector BHP. This is not usually a matter of concern if the well is on a rate control. Furthermore, for steam injection, the injection pressure may not depend strongly on the mobility.

Keywords [COMPAGH](#) and [COMPAGHL](#) can be used to set average gravity heads for completions.

See "[Well Inflow Performance](#)" for further information on well inflow calculations.

Water, steam and gas injection

An injection fluid made up of water, steam, gas, oil or a mixture of these, 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, and if the injection fluid contains hydrocarbon oil, the [WINJOIL](#) keyword can be used to specify the oil composition.

The enthalpy of the injected fluid is calculated from data entered using the [WINJTEMP](#) keyword.

If the injection fluid contains just steam or water there are four ways of entering the data:

1. Specify the steam quality Q_i and temperature T_i , pressure derived from the temperature.
2. Specify the steam quality Q_i and pressure P_i , temperature derived from the pressure.
3. Specify the pressure P_i and temperature T_i .
4. Specify the enthalpy H_i and pressure P_i .

The first two methods can be used for injection of two-phase saturated water and steam when the steam quality is between 0 and 1. The enthalpy of each phase is set to the saturated value at the specified temperature or pressure, and the injection fluid enthalpy is then calculated by interpolation:

$$H_i = Q_i \cdot H_s^{sat}(P_i, T_i) + (1-Q_i) \cdot H_w^{sat}(P_i, T_i)$$

where

$$P_i = P_{sat}(T_i) \text{ and } T_i = T_{sat}(P_i)$$

The third method can be used for injection of single phase water or steam. The enthalpy is determined from steam tables

$$H_i = H_w(P_i, T_i) \text{ if } P_i > P_{sat}(T_i),$$

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.

If steam The fourth method can be used in both one-phase and two-phase cases and gas are injected together then the same four ways can be used to specify the enthalpy. In methods 1,2 and 3 the enthalpy of the gas is calculated from the pressure and temperature and added to the steam enthalpy. If method 4 is used, the specified enthalpy is taken to be the enthalpy of the fluid.

If hydrocarbon gas is injected without steam then only methods 3 and 4 should be used to specify the fluid enthalpy.

Temperature at the bottom hole pressure

At the bottom hole pressure P_w , the injection enthalpy is equal to the enthalpy specified by the [WINJTEMP](#) keyword. Therefore the temperature of the well T_w can be found by solving

$$H(P_w, T_w) = H_i(P_i, T_i)$$

The well temperature will only be equal to the temperature specified using the [WINJTEMP](#) keyword, T_i , if the pressure P_i is equal to the bottom hole pressure P_w .

Steam trap

In [DEADOIL](#) simulations, steam production can be prevented by using either the [WELLPROD](#) or [WCONPROD](#) keyword to set the well BHP above the saturated water pressure in the well cells. The difference between the BHP and the saturated water pressure can be controlled either by specifying a pressure offset or a temperature offset.

The same keywords can be used to control the BHP in [LIVEOIL](#) simulations. However, if the reservoir contains volatile hydrocarbons, some steam may be produced at pressures above the water saturation pressure.

Steam production rate

The steam production rate can be controlled by using either the [WELLPROD](#) or [WCONPROD](#) keyword, or modified using the [WELTARG](#) keyword. The steam rate should be specified as a cold water equivalent (C.W.E.) volume/time. The rate is imposed at reservoir conditions, that is the well steam rate is calculated by summing the number of moles of water in the gas phase entering each completion. The simulator converts the molar rate to a C.W.E. rate by multiplying by the surface molar volume of water.

The same value of surface molar volume of water is used for steam rate control and water rate control. In dead oil runs it is calculated from the internal steam tables at the standard conditions ([STCOND](#)). In live oil runs it is calculated from the [DENSITY](#) keyword.

Dual porosity

The use of dual porosity grids in a thermal calculation makes additional assumptions about the distribution of rock volume between the porosities. In single porosity calculations, the rock volume is assumed to be the bulk volume minus the pore volume. This approach cannot be used with dual porosity, as it leads to the rock volume being represented multiple times, one in each porosity. Instead, default assumptions are made about the rock volume distribution, which can be overridden by the user.

The default behavior assumes that the rock is associated with the matrix porosity, as matrix pore volume is contained in pores in the rock, as compared with fracture pore volume, which is associated with gaps in the matrix rock. However, to represent the local thermal effect of rock bounding the fracture, which may cool or heat the fracture fluids before a thermal transient advances far into the matrix rock, an amount of rock equal to the fracture pore volume is associated with the fracture. To prevent negative rock volumes being calculated if the porosities are large, this fracture rock volume will be proportionately reduced, so that the matrix/fracture rock volume ratio equals that of the matrix/fracture pore volume ratio in the case that negative volumes would otherwise be produced.

This approach may not be appropriate in all cases, and the user has the option of overriding this behavior by providing a weighting factor for each cell in each porosity, using the [ROCKSPLV](#) keyword. In this case, the total rock volume, defined as the average bulk volume for all porosities for a particular cell, less the total pore volume over all porosities for that cell, is multiplied by the cell weighting factor specified by [ROCKSPLV](#), to give the rock volume for that porosity. It is not required that the weighting factors sum to unity, and so this keyword can also be used as a multiplier.

The default behavior may also be appropriate for triple porosity, where the non-matrix porosities are larger voids. For general multiple porosity, where the sub-matrix porosities can have a variety of user-defined interpretations, the default behavior may or may not be appropriate, depending on whether the sub-matrix cells are used to model transients in the matrix, or for other purposes.

The use of the [ACTNUM](#) keyword with dual porosity and the thermal option creates additional options for combining the various data inputs, and the user should check carefully that the final result is what is expected, as the ACTNUM and rock proportioning behaviors are applied separately from each other.

Multisegment wells

Multisegment 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 multisegment 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

By default, the convergence criteria are chosen to be reliable across a wide range of thermal models. If necessary, the default convergence criteria can be altered with the [CVCRIT](#) keyword.

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 equation 69.32 is multiplied by a reduction factor λ , $0 < \lambda \leq 1$ chosen so that for every cell:

$$\lambda \cdot |\Delta P| \leq P_{lim}$$

$$\lambda \cdot \left| \Delta X' \cdot \frac{\partial V}{\partial X'} \right| \leq S_{lim},$$

$$\lambda \cdot \left| \Delta e \cdot \frac{\partial T}{\partial e} \right| \leq T_{lim}$$

where X' is any solution variable except the pressure

$$X' = (m_1, \dots, m_N, m_w, e)$$

and V is the total specific volume, equal to the sum of the phase specific volumes defined in equation [69.18](#). The temperature limit is only used in thermal simulations, and only in fully implicit cells. Limits S_{lim} , T_{lim} and P_{lim} are set using items 13, 14 and 15 of the [CVCRIT](#) keyword.

Appleyard chop

After the solution is updated new values of the saturations and temperatures are calculated by performing a flash. If the saturations in an implicit cell change significantly near a critical value - for example 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 the [OPTIONS3](#) keyword. The temperature based chop can be switched off by using [item 27](#) of [OPTIONS3](#). Both chops can be switched off by using [item 25](#) of [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.

Timestepping criteria

By default the timestepping strategy employed in thermal simulations is based upon controlling the number of non-linear iterations that a timestep is expected to take. This is a heuristic timestep selection method in which the overall run time of a simulation can be minimized. In general this is a compromise between taking many cheap short timesteps and taking fewer longer, but computationally more expensive steps. The algorithm is based primarily on selecting a new timestep length depending on the previous timestep length and the number of non-linear iterations required to converge it.

The default timestep selection strategy is generally found to be robust across a wide range of models. However, in some instances, it may be possible to improve upon the default timestep selection by using the [TSCRIT](#) keyword to specify solution change targets and solution change limits.

Rejecting a timestep

A timestep can be rejected even though the non-linear iterations have converged, if the solution change during the timestep is too big. In these cases a smaller timestep will be used. A timestep 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} :

$$\sum_{X'} |X'_{t} - X'_{t-\Delta t}| \cdot \left| \frac{\partial T}{\partial X'} \right| \leq S_{max} \quad \text{Eq. 69.99}$$

$$TPT \leq TPT_{max} \quad \text{Eq. 69.100}$$

$$TTE \leq TTE_{max} \quad \text{Eq. 69.101}$$

where

X' is any solution variable except the pressure $X' = (m_1, \dots, m_N, m_w, e)$,

$\overline{\left| \frac{\partial V}{\partial X'} \right|}$ 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 \(TPT\)](#)". The throughput limit is only applied to IMPES cells.

The time truncation error is a measure of the "acceleration" of the specific volume and is discussed further in "[Time truncation error \(TTE\)](#)".

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} :

$$|SAT_{p,t} - SAT_{p,t-\Delta t}| \leq SAT_{max} \quad \text{Eq. 69.102}$$

$$|T_t - T_{t-\Delta t}| \leq T_{max} \quad \text{Eq. 69.103}$$

$$|e_t - e_{t-\Delta t}| \leq e_{max} = c_p \cdot T_{max} \quad \text{Eq. 69.104}$$

The maximum energy change is defined by the maximum temperature change, using a typical value of specific heat $c_p = 2347 \text{ kJ/m}^3/\text{K} = 35 \text{ Btu/ft}^3/\text{R}$. 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 timestep

New timesteps 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 timestep are estimated by multiplying the rate of change in the previous timestep by the new timestep 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 timestep 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 arguments RATP, RATS or RATT with the [RPTRST](#) keyword. The ratios are extrapolated to the new timestep $t + \Delta t$ and cells are made implicit if these extrapolated ratios exceed values set in [AIMCON](#).

Instability in AIM runs due to thermal conduction

In AIM runs the implicit cells are determined using throughput criteria. However in thermal runs, conduction can cause instability in explicitly solved cells. To prevent this instability:

- Rock filled cells (or near rock filled cells) should be solved fully implicitly. Therefore, in AIM runs item 3 of the [AIMCON](#) keyword should be set to a positive value.
- Problems where conduction is significant should not be solved using AIM.

Restriction	Parameter	Non linear iteration ΔX lim	Non linear iteration ΔX con	Timestep change max	Timestep change tar	Timestep change argument
	Pressure	CVCRIT item 15	CVCRIT item		TSCRIT item 12	PCT
	Specific volume	CVCRIT item 13	CVCRIT item 7	TSCRIT item 13	TSCRIT item 10	SCT
	Time truncation error			TSCRIT item 7	TSCRIT item 6	TTE
IMPES cells	Throughput			TSCRIT item 9	TSCRIT item 8	TPT
Thermal only	Temperature	CVCRIT item 14	CVCRIT item 8	TSCRIT item 18	TSCRIT item 17	TEC
Thermal only	e/cp			TSCRIT item 18	TSCRIT item 17	EnC

Restriction	Parameter	Non linear iteration ΔX lim	Non linear iteration ΔX con	Timestep change max	Timestep change tar	Timestep change argument
Thermal only	Saturation			TSCRIT item 16	TSCRIT item 15	SAC

Table 69.5: Summary of convergence criteria and timestepping criteria

Note: If items 13 and 14 of keyword [CVCRIT](#) are not set, the default values for the non-linear step limits S_{lim} and T_{lim} are calculated from items 10 and 17 of keyword [TSCRIT](#). This means that if the target specific volume is relaxed (item 10 of TSCRIT) - which may be the case if the target saturation is used (item 15 of TSCRIT) - then item 13 of keyword CVCRIT should be used to override the default value for the non-linear step limit S_{lim} .

Tuning thermal models for performance

The additional dependence of the reservoir properties on temperature, as modeled with the Thermal option, can make models significantly more computationally expensive to run. In general, the default tuning is found to be the best compromise across a wide range of models and it is recommended that all Thermal models are run without any additional tuning initially to establish a baseline. This is particularly relevant if the model predates the introduction of the JALS linear solver (default in 2009.1) which was found to significantly improve the performance of many models and render previous tuning redundant and sometimes detrimental. However, obtaining accurate and reliable performance timings for simulations, especially on clusters, is not trivial and it is advised that the trend in elapsed time is also compared with the total number of linear iterations (MSUMLINS) since these two quantities are usually closely correlated. Having established a baseline there are a number of non-default performance options which can be trialed for better performance if necessary.

Item 161 of the OPTIONS3 keyword can be used to activate an alternative linear algebra procedure for simulations with rock-filled cells. In a thermal simulation, for cells with zero pore volume, only the energy conservation equation has to be solved. By employing a slightly different, but mathematically identical, lineup of equations and variables it is possible to eliminate some of the variables from the linear system. With this approach the size of the system passed to the linear solver is reduced by one set of variables and hence can prove to generate a noticeable reduction in simulation time. However, the system passed to the linear solver may also prove to be numerically less favorable so an improvement in performance is not always seen. This option is only worth pursuing if the model contains one or more rock-filled cells.

[TSCRIT](#) item 11 can be used to increase the maximum length of timestep after a well modification, and is active due to the well control modifications or a well opening/closing. Prior to the 2012.1 software release, the default value was set to a low value for thermal simulations (0.5 days) since in some processes the state of the near well zone is very sensitive to the manner in which the well is operating. Short timesteps assist in resolving these features as well as others, including simulation models containing complex geometry and/or heterogeneity (where timesteps less than 0.5 day may be required). However, this constraint can be over cautious especially when using the [JALS](#) solver (the default solver) for simulations where the well modifications are relatively smooth or have less of an impact on the reservoir conditions. The default value for TSCRIT item 11 has now increased to 5 days.

Current limitations

At present a number of options cannot be used with the Thermal option. These include:

- "Diffusion"
- "Transport coefficients"
- "Non-Darcy flow"
- "Optimization"
- "Surface tension effects"

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") 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/stb to $Msfcf/stb$.

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 equation [69.40](#) can be used:

$$K \propto e^{-D/T}$$

with values of D taken from tables [69.1](#) or [69.2](#) 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

$$B_o = \left(V_{oil,r}^{oil} + \frac{R_s}{F} \cdot V_{oil,r}^{gas} \right) / V_{oil,s} \quad \text{Eq. 69.105}$$

where $V_{oil,r}^c$ is the reservoir oil phase volume per mole of component c .

Values of $V_{oil,r}^c$ are determined from user defined densities and compressibilities (see equations [69.45](#) and [69.46](#)), which can be chosen to give a good fit for equation [69.105](#) at the reservoir temperature. As a first step the component compressibility [CREF](#) and thermal expansion coefficient [THERMEX1](#) can be ignored to get approximate values for the reference density [DREF](#):

$$B_o = \frac{\rho_{oil,s}}{\rho_{ref}^{oil}} + R_s \cdot \frac{\rho_{gas,s}}{\rho_{ref}^{gas}} \quad \text{Eq. 69.106}$$

Gas phase density

The gas formation volume factor B_g is given by

$$B_g = \left(F \cdot R_v \cdot V_{gas,r}^{oil} + V_{gas,r}^{gas} \right) / V_{gas,s} \quad \text{Eq. 69.107}$$

where $V_{gas,r}^c$ is the reservoir gas phase volumes per mole of component c :

$$V_{gas,r}^c = \frac{Z^c \cdot R \cdot T}{P} \quad \text{Eq. 69.108}$$

where

R is the gas constant,

P is the pressure,

T the temperature and the compressibility terms Z^c can be set for each component using the [ZFACTOR](#) keyword to match equation [69.107](#) at the reservoir temperature.

The temperature dependence of these terms are fixed. A surface compressibility can be set using the [ZFACTORS](#) keyword to ensure the correct surface gas density:

$$V_{gas,s} = \frac{Z^{gas} \cdot R \cdot T}{P} = \frac{MW_{gas}}{\rho_{gas,s}} \quad \text{Eq. 69.109}$$

Oil phase viscosity

The thermal oil phase viscosity is given by equation 69.82 as a function of temperature and mole fractions. It can be rewritten in terms of R_s to give:

$$\log \mu_{oil} = \frac{1}{F + R_s} \cdot (F \cdot \log \mu_{oil}^{oil} + R_s \cdot \log \mu_{oil}^{gas}) \quad \text{Eq. 69.110}$$

If necessary an extra pressure dependent factor can be included in this formula. Values of the component oil phase viscosities μ_{oil}^c can be found at the reservoir temperature to give a good fit for equation 69.110.

These values can be entered in the [OILVISCT](#) keyword at two temperatures to give a temperature independent oil viscosity.

Gas phase viscosity

The thermal gas phase viscosity is given by equation 69.84 as a function of temperature and mole fractions. It can be rewritten in terms of R_v to give:

$$\mu_{gas} = \frac{1}{F \cdot R_v + 1} \cdot (F \cdot R_v \cdot \mu_{gas}^{oil} + \mu_{gas}^{gas}) \quad \text{Eq. 69.111}$$

Values of the component gas phase viscosities μ_{gas}^c can be found at the reservoir temperature to give a good fit for equation 69.111. These values can be entered in the [GASVISCT](#) keyword at two temperatures to give a temperature independent gas viscosity.

Specific heats

For an isothermal run, the actual values of the specific heats are not important. However, the enthalpy of each component in the gas phase should be larger than the enthalpy of the same component in the oil phase. This can be checked by looking at the property output in the PRT file.

Other data

The [TCRIT](#) and the [PCRIT](#) keywords must be included in the data set. There are no component-wise default values. The users are recommended not to default these to their internal default values.

Water properties

In thermal simulations the temperature dependence of water properties is usually more significant than the pressure dependence. Internal tables are used to define the water density and viscosity. To mimic a blackoil run keywords [WATDENT](#) and [WATVISCT](#) should be used to define the temperature dependence of the water density and viscosity while the pressure dependence is taken from the [PVTW](#) keyword.

Initial conditions

Data from a blackoil run can be used to initialize a thermal run. The keyword [GETSOL](#) can be used to read pressure and saturation data from a blackoil restart file. Keywords [XMF](#), [YMF](#) have to be added.

If the [EQUIL](#) keyword is used item 10 should be set.

Initial compositions can be set by noting that at the saturation pressure

$$z_{oil} = \frac{F}{F + R_s}$$

$$z_{gas} = \frac{R_s}{F + R_s}$$
Eq. 69.112

In thermal runs, water is allowed in the gas phase. This can cause differences in the initialization if a gas cap is present. If the initial composition is not in equilibrium then temperatures and saturations will change the first time the composition is flashed. It is therefore recommended that initial pressures, temperatures and saturations are output to the print file ([RPTSOL](#)) or restart file ([RPTSCHE](#)D) 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 DVD. CASE3T is a thermal version of the black oil data set CASE3B.

Summary of keywords

The ECLIPSE 300 Thermal option can be selected by entering the [THERMAL](#) keyword in the RUNSPEC section of the DATA file. Fluid properties can be specified in three different ways:

LIVEOIL

allows the simulation of multiple hydrocarbon components in both oil and gas phases.

DEADOIL

the hydrocarbon component cannot appear in the gas phase. A simpler set of property keywords is used. More than one hydrocarbon component is not allowed.

BLACKOIL

properties are specified using the Temperature option keywords, see the "[Temperature option](#)".

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)

RUNSPEC

Keyword	Description
BLACKOIL	Request use of Black Oil properties
DEADOIL	Request use of Dead Oil properties
JALS	Switch on the more powerful JALS solver. Recommended for thermal simulations
HEATDIMS	Heater keyword dimensions
LIVEOIL	Request use of Live Oil properties (the default)

Keyword	Description
<code>ROCKDIMS</code>	Base and cap rock dimensions
<code>TCBDIMS</code>	Thermal cut-back table dimensions
<code>THERMAL</code>	Request use of Thermal option

GRID

The following table 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.

Keyword	Description
<code>ACTNUM</code> *	Active grid block identification
<code>HEATCR</code>	Rock volumetric heat capacity
<code>HEATCRT</code>	Rock volumetric heat capacity T coefficient
<code>MINROCKV</code>	Minimum rock volume
<code>MINRV</code>	Minimum rock volume (alias for <code>MINROCKV</code>)
<code>NNC</code> *	Ninth argument is conductivity across the NNC
<code>ROCKCON</code>	Base and cap rock connections to the reservoir
<code>ROCKPROP</code>	Base and cap rock properties
<code>ROCKSPLV</code>	Non-default proportions of the rock in each porosity (for dual porosity).
<code>SIGMATH</code>	Matrix-fracture thermal conductivity multiplier
<code>THCONMF</code>	Matrix-Fracture thermal conductivity
<code>THCGAS</code>	Thermal conductivity for gas phase
<code>THCOIL</code>	Thermal conductivity for oil phase
<code>THCONR</code>	Thermal conductivity of rock and fluid
<code>THCROCK</code>	Thermal conductivity for rock
<code>THCSOLID</code>	Thermal conductivity for solid phase
<code>THCWATER</code>	Thermal conductivity for water phase
<code>THCONSF</code>	Thermal conductivity saturation-dependence

Edit

Keyword	Description
<code>HEATTR</code>	Heat transmissibility overwrite
<code>HEATTHT</code>	Heat transmissibility overwrite
<code>HEATTX</code>	Heat transmissibility overwrite
<code>HEATTY</code>	Heat transmissibility overwrite
<code>HEATTZ</code>	Heat transmissibility overwrite

Keyword	Description
ROCKV	Rock volume

PROPS

Property keywords are given in the following lists. Some keywords can be used with [DEADOIL](#), some with [LIVEOIL](#), and some with both.

The rock fluid property keywords can be used with both [LIVEOIL](#) and [DEADOIL](#).

- [ENKRT](#) Endpoint relative permeabilities versus temperature
- [ENPCVT](#) Endpoint capillary pressures versus temperature
- [ENPTVT](#) Endpoint saturations versus temperature

The following table of hydrocarbon properties 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. These are reservoir keywords that can only be used with [LIVEOIL](#) unless otherwise stated.

Restriction	Keyword	Description
	ACF *	Component acentric factors (used with KVWI).
	CCTYPE * and CCTYPES * (surface keyword)	Component liquid compressibility type.
LIVEOIL and DEADOIL	CGDTYPE	Component gas density type, EoS density.
	CGVTYPE	Component gas viscosity type, LBC viscosity.
	CODTYPE	Component oil density type, EoS density
	COVTYPE	Component oil viscosity type, LBC viscosity.
	CREF	Component compressibilities.
	CVTYPE and CVTYPES (surface keyword).	Component volatility (dead, live, gaseous).
	DREF * and DREFS * (surface keyword)	Component densities
	DREFT * and DREFTS * (surface keyword)	Component densities (alias for DREF)
	GASVISCF	Gas viscosity function of temperature
	GASVISCT	Gas viscosity table as a function of temperature
	GREF *	Reference gravities for live oil components
	HEATVAP	Component heat of vaporization
	HEATVAPE	Heat of vaporization exponent
	HEATVAPS	Component heat of vaporization at standard temperature

Restriction	Keyword	Description
	KVCR and KVCRS (surface keyword)	K-value correlation parameters
	KVGEN	Generate tables of K-values from an EoS
	KVTABTn	Tables of K-values
	KVTEMP	Temperatures for K-value tables
	KWI	Wilson K-value correlation parameters
DEADOIL only	OILCOMPR	Oil compressibility, expansion coefficient
DEADOIL only	OILMW	Oil molecular weight
DEADOIL only	OILSPECH	Oil specific heat
	OILVINDX	Oil viscosity index
	OILVISCC	Oil viscosity-temperature correlations
	OILVISCF	Oil viscosity function
LIVEOIL and DEADOIL	OILVISCT	Oil viscosity table as a function of T.
LIVEOIL and DEADOIL	OILVTIM	Oil viscosity temperature interpolation method.
	PCRIT	Component critical pressure (used with KWI)
	PREF and PREFS (surface keyword).	Component reference pressures
	PREFT * and PREFTS * (surface keyword)	Component reference pressures (alias for PREF)
LIVEOIL and DEADOIL	REACENTH	Reaction molar enthalpy
	SPECCHA	Oil specific heat, 1st coefficient
	SPECCHB	Oil specific heat, 2nd coefficient
	SPECCHG	Gas specific heat, 1st coefficient
	SPECCHH	Gas specific heat, 2nd coefficient
	SPECCHS	Solid specific heat, 1st coefficient
	SPECCHT	Solid specific heat, 2nd coefficient
	TCRIT *	Component critical temperature (used with KWI)
LIVEOIL and DEADOIL	THANALV	Analytic oil viscosities.
	THERMEX1	Liquid thermal expansion coefficient
	TREF * and TREFS * (surface keyword)	Component reference temperatures
	TREFT * and TREFTS *	Component reference temperature (alias for TREF)
LIVEOIL and DEADOIL	VISCREF	Reference conditions for the viscosity/temperature tables.

Restriction	Keyword	Description
	ZFACT1 and ZFACT1S (surface keyword)	Z-factor first coefficient
	ZFACTOR and ZFACTORS (surface keyword)	Z-factor zeroth coefficient

The following water keywords can be used with both [LIVEOIL](#) and [DEADOIL](#).

Keyword	Description
KVCRWAT	Water K-value correlation
THANALB	Analytic water and steam densities
THANALH	Analytic water and steam enthalpies
THSVC	Steam viscosity coefficients
THTABB	Tabular water and steam densities
THWVC	Water viscosity coefficients
WATDENT	Water density temperature dependence
WATVISCT	Water viscosity table as a function of T

REGIONS

The REGIONS keywords are:

- [THERMNUM](#) Thermal region numbers

SOLUTION

The following table 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. The SOLUTION keywords are:

Keyword	Description
RPTRST *	Controls solution output for GRAF: see SOLUTION and SCHEDELE section output
RPTSOL *	Controls PRINT file output: see SOLUTION and SCHEDELE section output

SUMMARY

Refer to the following tables in the *ECLIPSE Reference Manual*.

- [The well and group energy rate options](#)
- [Field, grid block, region quantities \(ECLIPSE 300 Thermal\)](#)
- The segment keywords for multisegment wells are given in [Multisegment wells](#)
- [Analytic aquifer quantities](#)
- [Simulator performance keywords](#)

SCHEDULE

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. The SCHEDULE keywords are:

Keyword	Description
AIMCON *	AIM selection criteria
CVCRIT *	Convergence criteria
HEATER	Position and properties of heater connections
HEATERL	Position and properties of heater connections in LGR
REACCRIT	Reaction interpolation criteria
RPTPRINT *	Item 14 can be used to control heater output
RPTRST *	Controls solution output: see
RPTSCHED *	Controls print file solution output: see
TRANGE	Specify minimum and maximum reservoir temperature
TSCRIT *	Timestepping 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 thermal properties of the tubing/casing Defines the multisegment well structure
WELTARG *	Change well production targets. Change steam production rate
WINJTEMP	Specify steam quality or temperature or enthalpy of injection fluid
WSEGEXSS	Defines external source / sinks for multisegment 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 the arguments that can be used is given below. All arguments can be used for R PTRST, R PTSOL and R PTSCHED output unless otherwise noted,

Keyword	Description
ENERGY	Total energies/bulk volume (energy/(rock + pore volume))
FLOE	Inter-block energy flow

Keyword	Description
GMEG	Gas phase energy mobility
GMO	Oil phase energy mobility
GMEW	Water phase energy mobility
HGAS	Gas enthalpy
HOIL	Oil enthalpy
HSOL	Solid enthalpy
HWAT	Water enthalpy
TCMULT	Saturation dependent thermal conductivity transmissibility multiplier
RATT	Ratio of actual temperature change to temperature target (RPTRST only)
SL	Liquid mole fractions
SPENGAS	Gas internal energy
SPENOIL	Oil internal energy
SPENWAT	Water internal energy
SV	Vapor mole fractions
SW	Water mole fractions
T2	Temperature at the two to three 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 two to three phase transition

All sections

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. The other keywords are:

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.
- 46 A positive value sets the number of Newton iterations within a timestep in which a well can be revived after being closed because it is unable to flow given the current reservoir conditions. The default behavior for the 2001A release when not using the Thermal option is to revive such a well only during the first [NUPCOL](#) Newton iterations. When the Thermal option is used, the default is to allow 21 revivals within a timestep before the well is closed. If this switch is set to a negative value, no well revivals are permitted. If however the switch is set to -999 the simulator reverts to the pre-2001A treatment, whereby the number of revivals permitted within a timestep was fixed at 21.
- 52 In 2001A and later releases, the horizontal thermal transmissibility is based on the full cell cross-sectional area and does not include a net-to-gross factor. If this option is set to 1, the pre-2001A calculation, which includes net-to-gross, is used.
- 92 Alternative timestepping strategy will be implemented. In thermal runs (2006.1 onward) this option is automatically selected and can be turned off by setting a positive value.
- 135 If set to 1, an alternative implementation for predicting the energy density in the [HEATER](#) algorithm is activated
- 158 If set to 1, the Appleyard Chop is applied to components when they appear or disappear. This behavior is the default for simulations using the chemical reactions feature but can be disabled by setting this OPTIONS3 switch to -1.
- 160 If set to 1, the pre-2008.2 thermal non-linear damping is restored.
- 161 If set to 1 an alternative procedure for simulations with rock-filled cells is activated. Mathematically this is a null change but variations due to numerical rounding may occur.
- 190 If set to a non-zero value, the pre-2009.1 setting of well enthalpies in thermal decks is activated. From 2009.1 onwards, the well enthalpies are set whenever an action is triggered.
- 214 If set to a non-zero value, this enables the pre-2009.2 John Appleyard Linear Solver ([JALS](#)).
- 217 If set to a non-zero value, the pre-2010.1 behavior for thermal heavy-oil equilibration will be restored.
- 218 If set to a non-zero value, the pre-2010.1 thermal temperature limits will be restored.

- 219 If set to a non-zero value, this enables the pre-2010.1 thermal rock-filled convergence method.
- 220 If set to 1, this enables the thermal oil viscosity-temperature correlation in the **OILVISCC** keyword, which is based on \log_{10} .
- 235 The transmissibility and diffusivity for dual and multi-porosity connections use the simulation grid-cell bulk-volume and not the sum of the cell pore volume and cell rock volume. For most cases these will be equal, but for thermal and Coal Bed Methane runs, the assignment of rock volume might result in differences. The pre-2010.1 behavior can be restored by setting this item to 1.
- 236 The behavior when failing to converge in a thermal simulation has been improved. This includes not chopping on a minimum timestep until the maximum number of Newton iterations have been solved. The pre-2010.1 behavior can be restored by setting this item to 1.
- 260 If set to 1, the surface gas density will be determined by a surface flash, rather than using the data from the **DENSITY** keyword.

Examples

The following example files come with the ECLIPSE installation:

.DATA File	Description	Auxiliary files
THERM1	Based on problem 1A of [Ref. 7] single well cyclic steam injection radial grid 1 component dead oil	
THERM2	Based on problem 2A of [Ref. 7] steam drive in 1/8th of a nine spot pattern 1 component dead oil	THERM2.GRF
THERM3	Based on problem 3A of [Ref. 7] steam drive in 1/8th of a nine spot pattern 3 component live oil	THERM3.GRF
THERM4	Variant of THERM1 single well cyclic steam injection radial grid 2 component live oil equilibration with a gas cap mixed gas and steam injection	

.DATA File	Description	Auxiliary files
THERM5	Based on case 1 of [Ref. 107] dual well SAGD 2 component live oil MSW	THERM5.GRF and WELL5.TXT
THERM5A	As above, but with perfectly horizontal wells.	
THERM5B	As above, but with inclined wells	
THERM6	Based on case 2 of [Ref. 107]. single well SAGD 2 component live oil MSW	
THERM7	Based on case 3 of [Ref. 107] single well cyclic steam injection 3 component live oil MSW	
THERM8	Based on case 3 of [Ref. 107] single horizontal well, cyclic steam injection 2 component live oil heat loss from surface to reservoir MSW	
THERM9	Based on [Ref. 24] laboratory scale combustion experiment 5 component live oil chemical reactions non-equilibrium initialization (NEI)	THERM9.GRF and OBS9.TXT
THERM10	Foamy oil	THERM10.GRF
THERM10A	As above, but without the foamy oil terms	
THERM10B	As above, but with reactions replacing the flash	
THERM10C	Foamy solids	
THERM11	Based on [Ref. 24] combustion with coke 5 component live oil solid phase chemical reactions	THERM11.GRF

.DATA File	Description	Auxiliary files
THERM12	4 volatile components	
THERM12R	Heaters	
THERM12S	LGRs Corner point geometry Fast and flexible restarts	
THERM13A	Comparison of AIM, FULLIMP and IMPES runs.	THERM13.GRF
THERM13F	Data sets are based on THERM3 (problem 3A of [Ref. 7]), with 2000 cells and 9 wells.	
THERM13		
THERM14D	Variant of THERM1 (problem 3A of [Ref. 7]). Comparison of drift flux and homogeneous flow models in a multisegment well.	THERM14.GRF
THERM14H		
THERM15	Dual porosity and Joule-Thomson effect.	THERM15.GRF
THERM15J		
CASE3E100	Based on [Ref. 124]. 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		

Table 69.6: Thermal example files

70

Timing reports

The argument 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.

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

ECLIPSE 100 timing Reports

```
=====
TIMING REPORTS AT CPU= 558 SECS - ELAPSED= 621 SECS
=====
ZS      1   2   3
          RR      1   2   2
INDX    1   0   0
PR      1   0   0
IR      1   0   0
NNC     1   0   0
XR      1   1   2
JR      1   0   0
YW      12   0   2
TS      11  89  86
          NX      41  88  82
          WC      148   1   1
          WCDC   148   0   0
          CONV   148   2   2
          PAIR   148   0   0
          LXX    107  68  63
          WGIN   107   0   0
          GINV   107   8   8
          WRES   107   0   0
          ITER   107  57  53
          NORM   980   2   2
          SRCH   873  34  32
          WADX   873   0   0
          ORTH   873  14  13
          UPDA   873   5   5
          WSOL   107   0
          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 mnemonic of up to four characters, 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 and saturations for example
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.

ECLIPSE 300 timing reports

```
=====
TIMING REPORTS AT CPU= 52 SECS - ELAPSED= 60 SECS
=====
--XUAIM          1  0.0 %cpu  0.1 %elp
--LSPUT4         3  0.0 %cpu  0.1 %elp
--TS             3 98.7 %cpu 92.7 %elp
----AIM          117 2.1 %cpu 1.8 %elp
----XUAIM         117 0.0 %cpu 0.0 %elp
----LSPUT4        117 0.3 %cpu 0.3 %elp
----NLSTEP        117 93.0 %cpu 83.5 %elp
----DCSP           234 46.2 %cpu 40.3 %elp
----DCED1A        236 28.2 %cpu 24.5 %elp
----XSWAPE        236 0.0 %cpu 0.0 %elp
----DCED1B        236 6.3 %cpu 5.5 %elp
----WSOLVS         234 15.7 %cpu 14.6 %elp
----NLRESD         234 2.1 %cpu 1.8 %elp
----XCNEWD         234 26.5 %cpu 23.5 %elp
----XCMATV         234 7.2 %cpu 6.3 %elp
----ELIM            234 2.1 %cpu 1.9 %elp
----LSSET4          234 0.2 %cpu 0.2 %elp
----LSOLV            234 13.0 %cpu 11.4 %elp
----LWARP            234 13.0 %cpu 11.4 %elp
----LGFAC            234 2.2 %cpu 2.1 %elp
----LGPGMR          234 10.2 %cpu 8.8 %elp
----LGFBS           1877 6.0 %cpu 5.2 %elp
----LGMLT           1877 2.1 %cpu 1.8 %elp
----BACKSU          234 1.8 %cpu 1.6 %elp
----XLIMIT          234 0.2 %cpu 0.2 %elp
----NLUPDT          234 0.1 %cpu 0.2 %elp
----PREDA            585 0.0 %cpu 0.0 %elp
----PSUMR            21  0.0 %cpu 0.0 %elp
```

Again, as with the ECLIPSE 100 timing reports, the top of the report contains a statement which details the total CPU and elapsed time taken so far by the program. The remainder of the report shows how these times were split between various sections of the program. Each program section is identified by a mnemonic of up to six characters, the most important of which are:

AIM Adaptive IMplicit method

AJGRAD Linear solution for adjoint vector

DCSP Main domain calculation
GHMCAL Main routine for gradient calculation
GSOLVS Supervises the gradient calculations at the end of a timestep
LSOLV WARP linear solver
LWARP WARP linear solver
NLCONV Check for convergence of the non-linear equations solved using Newton's method
NLRESD Set up the residual for the Newton system
NLSTEP Solution of the non-linear equations by Newton's method
NLTRAC Solution of the tracer tracking variables
NLUPDT Update the solution variables during a Newton iteration
SAMG Algebraic multi-grid solver used for geomechanics
TS Advances the simulator through time to the next report step
WSOLVS Solve the production and injection well system
XCFLOW Calculate the flows
XCMATV Set up the Jacobian matrix
XLIMIT Limit the solution changes during a Newton iteration
XUAIM Set up the Adaptive IMplicit (AIM) indicator array

71

Total compressibility checks

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

$$V_g = \frac{V \cdot S_g}{B_g} + \frac{V \cdot R_s \cdot S_o}{B_o} \quad \text{Eq. 71.1}$$

$$V_o = \frac{V \cdot S_o}{B_o} + \frac{V \cdot R_v \cdot S_g}{B_g} \quad \text{Eq. 71.2}$$

Equations 71.1 and 71.2 allow the mixture volume V and gas saturation S_g to be computed. If the pressure of the system is increased, the volume occupied by the liquid-vapor system must decrease for physically well-defined fluids. The total compressibility of the system, $C_t = -(1/V)(dV/dP)$, can be shown to be given by the following expression

$$C_t = \frac{S_g}{B_g} \left(-\frac{dB_g}{dP} + \frac{dR_v}{dP} \cdot \frac{B_o - R_s B_g}{1 - R_s R_v} \right) + \frac{S_o}{B_o} \left(-\frac{dB_o}{dP} + \frac{dR_s}{dP} \cdot \frac{B_g - R_v B_o}{1 - R_s R_v} \right) \quad \text{Eq. 71.3}$$

where the pressure derivatives are taken along the saturation curve.

This can also be written as

$$C_t = S_g C_{t,gas} + S_o C_{t,oil} \quad \text{Eq. 71.4}$$

which splits the total compressibility into gas and oil dependent terms.

To derive this expression, we assume without loss of generality that

$$S_o + S_g = 1 \quad \text{Eq. 71.5}$$

and note that

$$dV_o / dp = dV_g / dp = 0 \quad \text{Eq. 71.6}$$

since the volumes of stock tank oil and separator gas are fixed.

Solving for S_o and S_g gives

$$S_o = \frac{R_v V_g - V_o}{V} \cdot \frac{B_o}{R_s R_v - 1} \quad \text{Eq. 71.7}$$

and

$$S_g = \frac{R_s V_o - V_g}{V} \cdot \frac{B_g}{R_s R_v - 1} \quad \text{Eq. 71.8}$$

And since $S_o + S_g = 1$ we have

$$V = \frac{1}{(R_s R_v - 1)} \cdot \left(V_o (R_s B_g - B_o) + V_g (R_v B_o - B_g) \right) \quad \text{Eq. 71.9}$$

Hence

$$\begin{aligned} C_t &= -\frac{1}{V} \cdot \frac{dV}{dp} \\ &= -\frac{(R_s dR_v / dp + R_v dR_s / dp)}{(R_s R_v - 1)} \\ &\quad + \frac{\left\{ V_o \left[R_s \frac{dB_g}{dp} + B_g \frac{dR_s}{dp} - \frac{dB_o}{dp} \right] + V_g \left[R_v \frac{dB_o}{dp} + B_o \frac{dR_v}{dp} - \frac{dB_g}{dp} \right] \right\}}{V_o (R_s B_g - B_o) + V_g (R_v B_o - B_g)} \end{aligned} \quad \text{Eq. 71.10}$$

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 equation 71.3.

- If $R_s = R_v = 0$ then

$$C_t = -\frac{S_g}{B_g} \cdot \frac{dB_g}{dP} - \frac{S_o}{B_o} \cdot \frac{dB_o}{dP} > 0 \quad \text{Eq. 71.11}$$

since the formation volume factor pressure derivatives are < 0 .

- If $R_s > 0$ and $R_v = 0$ then

$$C_t = -\frac{S_g}{B_g} \cdot \frac{dB_g}{dP} - \frac{S_o}{B_o} \cdot \left(\frac{dB_o}{dP} - B_g \frac{dR_s}{dP} \right) \quad \text{Eq. 71.12}$$

so that $C_t < 0$ if $dB_o / dP > B_g dR_s / dP$ and S_g is sufficiently small.

ECLIPSE checks the total compressibility of the hydrocarbon PVT data as it is read from the input data file. For each PVT region in the simulation grid a pressure range is selected from the corresponding oil and

gas PVT data tables to span the complete range of pressure data in the two tables. This pressure range is then subdivided into 30 equally spaced pressure nodes for the evaluation of the total hydrocarbon compressibility using equation 71.3.

The sample pressure range is the maximum of the oil pressure (maximum bubble point pressure in [PVTO](#) or maximum pressure specified in [PVDO](#)) and the gas pressure (maximum dew point pressure specified in [PVTG](#) or maximum gas pressure specified in [PVDG](#)).

ECLIPSE 100

The maximum sample pressure at which total compressibility is checked can be overridden by choosing the second item in the [PMAX](#) keyword. This can be used to check for negative compressibility in extrapolated regions.

At each pressure node two limiting compressibility values are calculated corresponding to $S_g = 0$ and $S_g = 1$ in equation 71.3. If either or both values should be negative, a warning message will be issued indicating the offending pressure value and the gas saturation range for which the compressibility is negative. It should then be possible using equation 71.3 to determine how to change the saturated gas/oil PVT data to ensure the total compressibility remains positive for all pressures.

If the range of sample pressures extends above the maximum bubble point specified in the PVTO table or above the maximum dew point specified in the PVTG table, then ECLIPSE is forced to extrapolate above the highest entered bubble point or dew point. This extrapolation is linear in $1/B_o$ and $1/(B_o\mu_o)$ etc. (see the [PVTO](#) keyword). In this case, it is not unlikely that negative compressibilities could occur as a result of extrapolation.

In some cases, if the pressure is extrapolated severely, then the linearly-extrapolated formation volume factors may become unphysical. It is recommended that the highest bubble point nodes in the PVTO table are constructed so avoid extrapolations above the highest entered R_s 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.

72

Transmissibility calculations

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 $DX \cdot DY \cdot DZ \cdot \varphi \cdot RNTG$

where

φ is the porosity

RNTG the net to gross ratio, and

DX, DY and DZ the cell dimensions.

An alternative is the entry of cell corner points using the [COORD](#) and [ZCORN](#) keywords. The data associated with these keywords is usually prepared using a pre-processor such as PETREL, and entered from an included file. The cell is assumed to have edges defined as straight lines between the corner points, and the cell faces may be bilinear surfaces. Pore volumes may be calculated exactly for such a shape.

It is possible to have more than one set of COORD data, using the [NUMRES](#) keyword in the RUNSPEC section, and [COORDSYS](#) to assign layers to these sets.

Transmissibility calculations

There are at most three types of transmissibility calculation available in ECLIPSE.

The first is the traditional block centered 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 the 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

$$\text{TRANX}_i = \frac{\text{CDARCY} \cdot \text{TMLTX}_i \cdot A \cdot \text{DIPC}}{B} \quad \text{Eq. 72.1}$$

where

TRANX_i Transmissibility between cell i and cell j, its neighbor in the positive X-direction

CDARCY Darcy's constant

$$= 0.00852702 \text{ (E300); } 0.008527 \text{ (ECLIPSE 100) (METRIC)}$$

$$= 0.00112712 \text{ (E300); } 0.001127 \text{ (ECLIPSE 100) (FIELD)}$$

$$= 3.6 \text{ (LAB)}$$

$$= 0.00864 \text{ (PVT-M)}$$

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:

$$A = \frac{\text{DX}_j \cdot \text{DY}_i \cdot \text{DZ}_i \cdot \text{RNTG}_i + \text{DX}_i \cdot \text{DY}_j \cdot \text{DZ}_j \cdot \text{RNTG}_j}{\text{DX}_i + \text{DX}_j} \quad \text{Eq. 72.2}$$

$$B = \frac{\left(\frac{\text{DX}_i}{\text{PERMX}_i} + \frac{\text{DX}_j}{\text{PERMX}_j} \right)}{2} \quad \text{Eq. 72.3}$$

$$\text{DIPC} = \frac{\text{DHS}}{\text{DHS} + \text{DVS}} \quad \text{Eq. 72.4}$$

with

$$\text{DHS} = \left(\frac{\text{DX}_i + \text{DX}_j}{2} \right)^2$$

and

$$\text{DVS} = [\text{DEPTH}_i - \text{DEPTH}_j]^2.$$

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:

$$\text{TRANZ}_i = \frac{\text{CDARCY} \cdot \text{TMLTZ}_i \cdot A}{B} \quad \text{Eq. 72.5}$$

where

TRANZ_i Transmissibility between cell i and cell j, its neighbor in the positive Z-direction (that is below cell i)

TMLTZ_i The Z-transmissibility multiplier for cell i

and

$$A = \frac{\text{DZ}_j \cdot \text{DX}_i \cdot \text{DY}_i + \text{DZ}_i \cdot \text{DX}_j \cdot \text{DY}_j}{\text{DZ}_i + \text{DZ}_j} \quad \text{Eq. 72.6}$$

$$B = \frac{\left(\frac{\text{DZ}_i}{\text{PERMZ}_i} + \frac{\text{DZ}_j}{\text{PERMZ}_j} \right)}{2} \quad \text{Eq. 72.7}$$

Note that there is no dip correction in the Z-transmissibility case.

OLDTRANR transmissibility calculations

ECLIPSE 100

In this case, X- and Y-direction transmissibility values in the Cartesian case are also calculated using cell center separations and cross-sectional areas obtained from DX, DY and DZ, with a dip correction.

The expressions for the Cartesian case are given below:

X- transmissibility

$$\text{TRANX}_i = \frac{\text{CDARCY} \cdot \text{TMLTX}_i \cdot \text{DIPC}}{B} \quad \text{Eq. 72.8}$$

where

TRANX_i Transmissibility between cell i and cell j, its neighbor in the positive X-direction

CDARCY Darcy's constant (in the appropriate units)

TMLTX_i Transmissibility multiplier for cell i

DIPC Dip correction.

B and DIPC are given by the expressions:

$$B = \frac{\left(\frac{\text{DX}_i}{A_i \cdot \text{PERMX}_i} + \frac{\text{DX}_j}{A_j \cdot \text{PERMX}_j} \right)}{2} \quad \text{Eq. 72.9}$$

where

$$A_i = \text{DY}_i \cdot \text{DZ}_i \cdot \text{NTG}_i$$

and

with

$$\text{DIPC} = \frac{\text{DHS}}{\text{DHS} + \text{DVS}} \quad \text{Eq. 72.10}$$

$$DHS = \left(\frac{DX_i + DX_j}{2} \right)^2$$

and

$$DVS = [DEPTH_i - DEPTH_j]^2.$$

RNTG is the net to gross ratio, which appears in the X- and Y-transmissibilities but not in the Z-transmissibility.

Y- transmissibility

The expression for the Y-transmissibility value is entirely analogous to the above, with the appropriate permutations of X,Y and Z.

Z- transmissibility

The Z-transmissibility is given by:

$$TRANZ_i = \frac{CDARCY \cdot TMLTZ_i}{B} \quad \text{Eq. 72.11}$$

where

$TRANZ_i$ Transmissibility between cell i and cell j, its neighbor in the positive Z-direction (that is below cell i)

$TMLTZ_i$ The Z-transmissibility multiplier for cell i

$$B = \frac{\left(\frac{DZ_i}{A_i \cdot PERMZ_i} + \frac{DZ_j}{A_j \cdot PERMZ_j} \right)}{2} \quad \text{Eq. 72.12}$$

with

$$A_i = DX_i \cdot DY_i$$

$$A_j = DX_j \cdot DY_j$$

Note that there is no dip correction in the Z-transmissibility case.

Corner point transmissibility calculations

In this case, the transmissibility values are calculated from the X-,Y- and Z-projections of the mutual interface area of the two cells.

An inner product is then taken with the vector distance from the cell center to the center of the cell face, so that a dip correction is automatically incorporated.

Note: For ECLIPSE 100, this type of calculation is also referred to as NEWTRAN.

X- transmissibility

The X-transmissibility is given by the expression:

$$\text{TRANX}_i = \frac{\text{CDARCY} \cdot \text{TMLTX}_i}{\frac{1}{T_i} + \frac{1}{T_j}} \quad \text{Eq. 72.13}$$

where

$$T_i = \text{PERMX}_i \cdot \text{RNTG}_i \cdot \frac{A \cdot D_i}{D_i \cdot D_i}$$

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 j, these centers being obtained as the appropriate average.

The expression for T_j is analogous.

Y- and Z- transmissibility

The Y- and Z-transmissibility expressions are similar, the net to gross ratio being absent from the Z expression.

The calculation is illustrated by the following figure (where PERMX is represented by Kx):

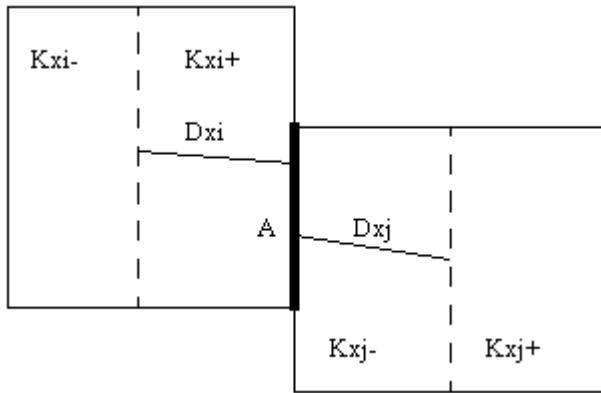


Figure 72.1. Y- and Z- transmissibility expressions

HALFTRAN transmissibility calculations (ECLIPSE 100)

By observing that the transmissibility between cells i and j in the NEWTRAN formulation is composed of a harmonic average between half block transmissibilities T_i and T_j , we have rather more freedom than just using a single permeability value for each cell.

In fact, there are six possible permeability values associated with each cell in the half block transmissibilities. For instance, in cell i, half block x-direction permeabilities Kxil, Kxir contribute to the two x-direction transmissibilities affecting cell i. Similarly in the Y- and Z-directions we can use Kyil, Kyir and Kzil, Kzir.

The use of extra half block permeabilities provides a more accurate upscaling of permeability while retaining the same computational grid.

Within ECLIPSE 100, the [HALFTRAN](#) keyword activates the half block transmissibility calculation.

By writing the half block transmissibilities as

$$\bar{T}_{ir} = Kx_{ir} \cdot RNTG_i \cdot \frac{A \cdot D_i}{D_i \cdot D_i} \quad \text{Eq. 72.14}$$

and

$$\bar{T}_{jl} = Kx_{jl} \cdot RNTG_j \cdot \frac{A \cdot D_j}{D_j \cdot D_j} \quad \text{Eq. 72.15}$$

it is apparent that in terms of the normal NEWTRAN calculation, then

$$\bar{T}_{ir} = \lambda_{ir} T_i$$

and

$$\bar{T}_{jl} = \lambda_{jl} T_j$$

with

$$\lambda_{ir} = Kx_{ir} / Kx_i \text{ and } \lambda_{jl} = Kx_{jl} / Kx_j.$$

When [HALFTRAN](#) is used, ECLIPSE 100 requires the [MULTX](#) and [MULTX-](#) values to be supplied equal to λ_{ir} and λ_{jl} .

At present, the FloGrid program carries out the half-block upscaling and present the modified [MULTX](#) and [MULTX-](#) data to the simulator in order to effect the half block transmissibility calculation. The [PERMX](#) etc. values should be entered as normal. The half block permeabilities are then obtained by multiplying [PERMX](#) by [MULTX](#) or [MULTX-](#) etc. The connection factors are calculated as usual using Peaceman's formula for the block center permeability [PERMX](#) etc.

Radial transmissibility calculations

There is no distinction between the method used to calculate radial transmissibility values whether grid data is supplied in [DR](#) / [DTHETA](#) / [DZ](#) / [INRAD](#) / [TOPS](#) form or [COORD](#) / [ZCORN](#) form.

The expressions used are based upon true radial flow between the pressure equivalent radii, with a dip correction.

R-transmissibility

$$\text{TRANR}_i = \frac{\text{CDARCY} \cdot \text{TMLTR}_i \cdot \text{DIPC}}{\frac{1}{T_i} + \frac{1}{T_j}} \quad \text{Eq. 72.16}$$

with

$$T_i = \frac{\text{PERMR}_i \cdot \text{RNTG}_i \cdot D\theta_i \cdot DZ_i}{D_{1P}}$$

$$D_{1P} = \frac{R_1^2}{(R_2^2 - R_1^2)} \ln(R_1/R_2) + \frac{1}{2}$$

and

$$T_j = \frac{\text{PERMR}_j \cdot \text{RNTG}_j \cdot D\theta_j \cdot DZ_j}{D_{2M}}$$

$$D_{2M} = \frac{R_3^2}{(R_3^2 - R_2^2)} \ln(R_3/R_2) - \frac{1}{2}$$

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

$$DVS = (\text{DEPTH}_i - \text{DEPTH}_j)^2.$$

Azimuthal transmissibility

The azimuthal transmissibility is given by:

$$\text{TRANT}_i = \frac{CDARCY \cdot TMLTT_i \cdot DIPC}{\frac{1}{T_i} + \frac{1}{T_j}} \quad \text{Eq. 72.17}$$

with

$$T_i = \frac{2 \cdot \text{PERMT}_i \cdot \text{RNTG}_i \cdot DZ_i \cdot \ln(R_2/R_1)}{D\theta_i}$$

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

$$DHS = \left(\frac{1}{2} (D\theta_i + D\theta_j) \cdot R_{\text{mean}} \right)^2$$

and

$$DVS = (\text{DEPTH}_i - \text{DEPTH}_j)^2.$$

For ECLIPSE 100, R_{mean} takes a value of one, and for ECLIPSE 300, R_{mean} represents an average of the inner and outer radius.

Vertical transmissibility

The vertical transmissibility is given by:

$$\text{TRANZ}_i = \frac{\text{CDARCY} \cdot \text{TMLTZ}_i}{\frac{1}{T_i} + \frac{1}{T_j}} \quad \text{Eq. 72.18}$$

with

$$T_i = \frac{\text{PERMZ}_i \cdot D\theta_i \cdot (R_2^2 - R_1^2)}{\text{DZ}_i}$$

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 / DTTHETA / 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 OLDTRAN, OLDTRANR or NEWTRAN employs an effective average permeability for the face between two adjacent blocks, which ECLIPSE 100 calculates from the permeabilities and dimensions of these blocks.

Options are available, however, that allow you more control over the average permeability employed in these calculations. You may input the average permeabilities directly (see keyword [LINKPERM](#)), or request alternative averaging formulations (see keyword [PERMAVE](#)).

The options [LINKPERM](#) and [PERMAVE](#) may be useful for comparison with other simulators. It is also possible to override the transmissibilities calculated by ECLIPSE 100 by supplying transmissibilities directly in the EDIT section, for comparison with other simulators.

Use of linking permeabilities in the transmissibility calculation

The [LINKPERM](#) keyword may optionally be used to instruct ECLIPSE 100 to treat the grid block permeabilities (input with [PERMX](#) and related keywords) 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](#)”). 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:

$$\text{TRANX}_i = \frac{\text{CDARCY} \cdot \text{TMLTX}_i \cdot A \cdot \text{DIPC}}{B} \quad \text{Eq. 72.19}$$

with B rewritten as:

$$B = \frac{\text{DX}_i + \text{DX}_j}{2 \cdot \text{PERMX}_{\text{av}}} \quad \text{Eq. 72.20}$$

and the average permeability is given by a harmonic average:

$$\text{PERMX}_{\text{av}} = \frac{(W_i + W_j)}{\left(\frac{W_i}{\text{PERMX}_i} + \frac{W_j}{\text{PERMX}_j} \right)} \quad \text{Eq. 72.21}$$

with weights

$$\begin{aligned} W_i &= \text{DX}_i \\ W_j &= \text{DX}_j \end{aligned} \quad \text{Eq. 72.22}$$

This is just a special case of the power law average given by:

$$\text{PERMX}_{\text{av}} = \left[\frac{W_i \cdot \text{PERMX}_i^{\text{POW}} + W_j \cdot \text{PERMX}_j^{\text{POW}}}{W_i + W_j} \right]^{\frac{1}{\text{POW}}} \quad \text{Eq. 72.23}$$

where $\text{POW} = -1$.

Other forms of average may be obtained by varying the value of POW. A weighted arithmetic average is obtained if $\text{POW} = 1$, and a geometric average results in the limit as $\text{POW} \rightarrow 0$, that is

$$\text{PERMX}_{\text{av}} = \left[\text{PERMX}_i^{W_i} \cdot \text{PERMX}_j^{W_j} \right]^{1/(W_i + W_j)} \quad \text{Eq. 72.24}$$

Secondly, consider the OLDTRANR case. The formulation for OLDTRANR is analogous with OLDTRAN, but now the weights are given by

$$\begin{aligned} W_i &= \frac{\text{DX}_i}{A_i} = \frac{\text{DX}_i}{\text{DY}_i \cdot \text{DZ}_i \cdot \text{RNTG}_i} \\ W_j &= \frac{\text{DX}_j}{A_j} = \frac{\text{DX}_j}{\text{DY}_j \cdot \text{DZ}_j \cdot \text{RNTG}_j} \end{aligned} \quad \text{Eq. 72.25}$$

Finally, consider the NEWTRAN case. The weights corresponding to the default harmonic average of block permeabilities are given by:

$$\frac{1}{W_i} = \text{RNTG}_i \cdot \frac{A \cdot D_i}{D_i \cdot D_i}$$
$$\frac{1}{W_j} = \text{RNTG}_j \cdot \frac{A \cdot D_j}{D_j \cdot D_j}$$

Eq. 72.26

and the alternative permeability averages are readily expressed in the same manner as above.

73

Tracer tracking

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 sulfur 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. Reinjection of tracer gas cannot, however, be modeled in the current tracer option.

The default 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. In ECLIPSE 100, the [Environmental tracers](#) option enables the modeling of contaminants and other substances as they flow within a host water, oil or gas phase. 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 timestep 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 timestep using a fully implicit time discretization. Such a system of difference equations is particularly prone to exhibiting the effects of numerical diffusion errors, particularly in regions of the grid where tracer concentrations vary most rapidly. The smearing of tracer interfaces tends to be most apparent in single phase regions (for example gas injection into a gas cap, water injection into aquifers).

There are two options in ECLIPSE 100 to reduce the effects of numerical diffusion:

Cascade algorithm

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.

In a serial run, the cascade algorithm is always used for **active** tracers whenever no mass transfer is associated with these tracers, whether or not 'DIFF' is enabled in the [RUNSPEC](#) section keyword [TRACERS](#). It is possible to stop the cascade algorithm being used by employing the [NOCASC](#) keyword in the [RUNSPEC](#) section. This forces the linear solver algorithm to be used for solving all single phase tracers.

Note: The cascade algorithm is not available for parallel runs. Thus, switching to parallel in a model containing active tracers automatically forces the linear solver algorithm to be used. Consequently, small changes in results can be expected. To avoid these changes between serial and parallel runs the [NOCASC](#) keyword should be used in the serial run.

Note: Brine is also an active tracer and prior to the 2004A release was also solved automatically with the cascade method. Since the 2004A release, a brine tracer has been solved with the user-selected method. The cascade solver cannot be used for the Multi-component brine option.

Flux limiting scheme

The explicit flux limiting scheme is invoked by setting item 5 to 'DIFF' in keyword [TRACERS](#) in the [RUNSPEC](#) section, and in addition specifying the [TRACTVD](#) keyword in the [PROPS](#) section. The implicit flux limiting scheme is invoked by specifying the [TRACITVD](#) keyword in the [PROPS](#) section. These options only apply to tracers embedded in phases where no mass transfer effects can take place.

The scheme is second order in smoothly varying regions, but uses a limiting function to prevent unphysical oscillations near sharp fronts.

For the explicit flux limiting scheme, tracer concentrations are updated explicitly using a mini 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. 92]).

For the implicit flux limiting scheme, the tracer concentration is computed using the same timestepping control as the standard fully implicit solver.

Lagrangian tracer solver (ECLIPSE 300)

The Lagrangian tracer solver is activated with the [LAGTRAC](#) keyword in the RUNSPEC section and the [LAGTRUPD](#) keyword in the SCHEDULE section.

This solver is used in place of the standard implicit tracer solver for all tracers when activated. It uses FrontSim technology (for example the Pollock method) to trace a set of streamlines based on the flows computed by ECLIPSE. The tracer solver uses these streamlines to find the source of fluid in each cell and updates the tracer concentration in each cell according to the tracer concentration in the source cells.

This technique is not fundamentally conservative and does require that the streamline set accurately represents the flow of fluid in the reservoir. Assuming that this is the case, this method can give a faster, less numerically diffusive tracer solution. To ensure that the streamlines accurately describe the reservoir flow it is important that there are enough streamlines and that the streamlines are updated if the reservoir flow changes due to changes to well controls, movement of fronts and so on. The updates occur according to the data entered in the [LAGTRUPD](#) keyword.

The tracer concentration at the time that the streamline is traced is stored on the streamline; it is this value that is used, along with the time since this value was stored, to determine tracer concentration at the destination cell. This allows a solution with less numerical diffusion. The user can choose to update these concentrations without retracing the streamlines using [LAGTRUPD](#). In any event these values are updated at each report step.

As the Lagrangian Tracer Solver is a streamline-based method, some physical processes are not solved as accurately as others. Gravity segregation can lead to the net flow of a component giving a poor representation of the movement of tracers; as the liquid phase drops and the vapor phase rises the net flow of a component will not necessarily give good results. Another area of difficulty is strongly convergent or divergent flow, such as one might find in a reservoir produced by pressure depletion. While the [LAGTRAC](#) algorithm contains correction terms for compression and divergence, in strong cases these first order corrections will not be sufficient.

Lagrangian tracer solver speed and accuracy

The Lagrangian tracer solver is very quick at calculating tracer distributions once a streamline set has been traced. Tracing a set of streamlines is a slow process. As a result there is a trade-off between speed and accuracy which may be controlled by judicious use of the [LAGTRAC](#) and [LAGTRUPD](#) keywords.

The third argument of the [LAGTRAC](#) keyword should be used to determine the number of streamlines to be traced; more streamlines will slow down both the streamline tracing and the solution of the tracer concentration. The default value has been chosen to give a reliably accurate solution, but this can be reduced in most cases to a value of 0.05.

The [LAGTRUPD](#) keyword may be used to control the frequency with which streamlines are retraced. The default values cause streamlines to be retraced every timestep; this is safe but slow. In the majority of cases the retrace frequency can be reduced dramatically, with a retrace roughly every report step being reasonable, and a value of at least 0.1 for argument 3 also being safe in most cases.

One final point to consider is that a set of streamlines can be used for more than one tracer. This means that multiple tracers which have the same carrier (that is tracers with the same argument 2 in [TRACER](#) and all [LUMPING](#) tracers) are all solved using one streamline set. As the majority of the cost of the Lagrangian tracer option is associated with tracing the streamlines, this means that every tracer after the first for a given carrier has a very marginal cost in memory and CPU time. This contrasts sharply with the standard tracer solver where the cost of calculation is almost proportional to the number of tracers.

Lagrangian tracer solver in parallel

The Lagrangian tracer solver is compatible with the [PARALLEL](#) keyword. This is implemented by gathering the grid and flow information necessary to trace a streamline onto the master processor and then copying this information to all processors. Each processor then traces some of the streamlines required, and the streamlines are finally mirrored across processors. This allows the tracer solver to run without any cross-processor communication being necessary except when tracing streamlines.

Lagrangian tracer summary

The [LAGTRAC](#) keyword is used to activate the Lagrangian Tracer Solver. The [LAGTRUPD](#) keyword is then used to define the solver method and the streamline update strategy. The [RPTSLN](#) keyword can be used to output the streamlines produced by the Lagrangian solver as SLN format files. [LTRACE](#) and [NOLTRACE](#) keywords are also available to allow the user to fine tune the update of the streamlines.

Using the ECLIPSE tracer tracking facility

ECLIPSE 100

The tracer tracking facility in ECLIPSE 100 is activated by means of three integers NOTRAC, NWTRAC, NGTRAC set in keyword [TRACERS](#) in the RUNSPEC section. These three integers represent the maximum number of tracers to be defined in each of the three stock tank phases: water, oil and dry gas. It is recommended that the precise number of tracers to be tracked should be specified in keyword TRACERS in the RUNSPEC section in order to minimize the memory requirements of the model.

ECLIPSE 300

The tracer tracking facility in ECLIPSE 300 does not require the keyword TRACERS in the RUNSPEC section; it is activated by entering the [TRACER](#) keyword in the PROPS section of the input data.

The names of each tracer and their associated stock tank phases are defined using the TRACER keyword in the PROPS section of the input data. In ECLIPSE 100 the maximum number of rows of data is (NOTRAC + NWTRAC + NGTRAC). Each tracer name is, at present, limited to be a name of up to 3 characters in length. The output of TRACER data is controlled by the TRACER argument 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 argument AQUCT or AQUFET in the [RPTSCHE](#) 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 H₂S generating processes.

Note: Injection/production of tracers in wells having significant crossflow may lead to tracer mass balance errors due to current limitations in the tracer model. The mass balance errors can be examined in the tracer-in-place reports in the print file, enabled by argument [FIPTR](#) in keyword [RPTSCHE](#). For non-linear tracers, for example environmental tracers with adsorption enabled, the issue with crossflow in wells may lead to convergence problems. The problem can be resolved by disabling crossflow (see item 10 of the [WELSPECS](#) keyword).

ECLIPSE 100

Reporting of the tracer concentration in each grid cell is activated using the argument TBLK or TRACER in the [RPTSCHE](#) keyword. Output of tracer volumes within each reporting region and the inter-region tracer flows are available using the argument [FIPTR](#) in the [RPTSCHE](#) 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 arguments [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 [RPTSO](#) and [RPTSCHE](#), or output to graphics using [RPTRST](#). For example:

```
RPTSO
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 argument [WELLS](#) in the [RPTSCHE](#) keyword.

ECLIPSE 100

Tracers can be injected in rivers using the keyword [RIVTRACE](#) in the SCHEDULE section. Tracers can be injected in a specified reach of the river and through the upstream end of a river branch. The [RIVTRACE](#) keyword optionally allows the injected tracer concentration to be a function of the total cumulative water flow.

Tracking original fluids

This section describes the operation of the ECLIPSE 300 TRACK option, which enables a division of hydrocarbons in the original reservoir to be tracked through into recovered surface volumes. The option is enabled by entering the [TRACK](#) keyword with appropriate arguments in the RUNSPEC section. To use the enhanced tracking facility, with user defined regions, it is also necessary to define the maximum number of regions to be used in the [REGDIMS](#) keyword in the RUNSPEC section.

Formulation

Suppose that we divide the hydrocarbons in a reservoir into those lying above the GOC (gas-oil contact) and those lying below it. As the reservoir is produced these components will flow through the field to the production wells. Generally, each component will move along a different path. For example, the light components such as methane will fraction into the reservoir vapor, and tend to rise to the top of the reservoir. The heavy components will tend to remain in the oil and stay at the bottom of the reservoir.

At a given producer, each component will have a fraction from below the GOC and a fraction from above it. We can thus define two streams, one of components originating from below the GOC and one from above. If these are run separately through the separator system we can obtain stock tank oil and stock tank gas volumes attributable to each of these streams.

With the enhanced Tracking option it is possible to divide the reservoir in any way the user desires.

Usage

The Tracking option is turned on using the TRACK keyword. This allows the initial fluid to be split at the GOC or at a given depth. The following text discusses splitting at the GOC; splitting at a specified depth is analogous.

In the tracking model, production volume rates originating from hydrocarbons above and below the GOC are reported separately in the PRINT file and when using SUMMARY section keywords.

Summary items are as follows:

For wells

The SUMMARY keyword 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.

Some reported tracer quantities can be prone to rounding errors if the model is made of cells with exceeding small pore volumes. This might happen if modeling a core when using FIELD or METRIC units. In this situation the solution is to convert to LAB units.

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, and so on. In the ‘ REGONLY ’ case it is not possible to report quantities from the regions above and below the GOC, such as FOPRA, FOPRB, FOIPA, and FOIPB, as separate tracers have not been defined in these regions. 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_, and so on. 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:

- Injection gas, which is not from original reservoir components.
- The separator flash is not linear, so the volume of an oil and gas resulting from a mixture is not the same as the sum of the volumes of two sub-mixtures.
- The tracer solution will disperse like any numerical solution, so that the sum of the A and B tracer concentrations will not generally equal exactly one.

Note: The NORM option is provided so that you can request that the volumes be normalized to equal the total rate of the well. If normalization with the [TRACK](#) keyword (argument 3) is activated then FGPRA,FGPRB, FGPTA and FGPTB and similar items for well and group gas rates are calculated as:

$$FGPRA = \frac{FGPR}{FGPRA + FGPRB} FGPRA$$

Alternative production reporting

An alternative method for calculating the production from above and below the GOC is to directly split a well's volumetric flow rate according to the tracer concentrations above and below the contact. So, for example, the well oil production from fluids above the GOC, *WOPR1*, is:

$$WOPR1 = WOPR \cdot \sum_{i=1}^N WXMF_i \cdot T_i \quad \text{Eq. 73.1}$$

where:

WOPR is the well volumetric oil production rate

WXMF_i is the well liquid mole fraction for component *i*

T_i is the above GOC tracer concentration for component *i*

N is the number of components.

Similar equations can be written for the well oil production from fluids below the GOC and for the gas production rates. This way of splitting a well's production ensures that the sum of the volume from above and below the GOC equal the total volume produced by the well in the absence of injection gas and subject to the numerical accuracy of the simulation.

This method is known as the molar volumetric split and the resulting field, group and well rates and totals are calculated in addition to the standard TRACK option production quantities.

The molar volumetric split quantities may be reported using the following SUMMARY section keywords.

Field	Well	Group	Description
F(O,G)P(R,T)1	W(O,G)P(R,T)1	G(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	W(O,G)P(R,T)2	G(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 timestep is then:

$$R_i = M_i^{T+\Delta T} \cdot C_i^{T+\Delta T} - M_i^T C_i^T + \Delta T \sum F^{i \rightarrow n} C_{in}^{T+\Delta T} + \Delta T C_i^{T+\Delta T} Q_{M_i} = 0 \quad \text{Eq. 73.2}$$

where:

M_i is the moles of carrier in cell i

C_i is the tracer concentration in cell i

$F^{i \rightarrow n}$ 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: The tracer conservation equation is linear in C , and thus solves in one iteration of Newton's method.

SUMMARY section keywords

Refer to [Tracer and API tracking quantities](#) in the *ECLIPSE Reference Manual*.

The following table give additional SUMMARY keywords that are compatible with the ECLIPSE 300 Track option if either of the ' GOC ' or ' DEPTH ' options is used. They are not available if the ' REGONLY ' option is used.

Field	Group	Well	Description
FOPRA	GOPRA	WOPRA	Oil Production Rate above GOC
FOPRB	GOPRB	WOPRB	Oil Production Rate below GOC
FOPTA	GOPTA	WOPTA	Oil Production Total above GOC
FOPTB	GOPTB	WOPTB	Oil Production Total below GOC
FOPR1	GOPR1	WOPR1	Oil Production Rate above GOC (molar method)
FOPR2	GOPR2	WOPR2	Oil Production Rate below GOC (molar method)
FOPT1	GOPT1	WOPT1	Oil Production Total above GOC (molar method)

Field	Group	Well	Description
FOPT2	GOPT2	WOPT2	Oil Production Total below GOC (molar method)
FGPRA	GGPRA	WGPR	Gas Production Rate above GOC
FGPRB	GGPRB	WGPRB	Gas Production Rate below GOC
FGPTA	GGPTA	WGPTA	Gas Production Total above GOC
FGPTB	GGPTB	WGPTB	Gas Production Total below GOC
FGPR1	GGPR1	WGPR1	Gas Production Rate above GOC (molar method)
FGPR2	GGPR2	WGPR2	Gas Production Rate below GOC (molar method)
FGPT1	GGPT1	WGPT1	Gas Production Total above GOC (molar method)
FGPT2	GGPT2	WGPT2	Gas Production Total below GOC (molar method)

Example

As an example, we consider the third SPE benchmark problem [Ref. 63], which is a small gas condensate study. The top of the reservoir is at 7315 feet and the bottom of the reservoir is at 7475 feet. We modify the problem slightly to place a gas-oil contact at 7400 feet, and produce the field using the standard gas cycling production strategy.

The TRACK option is selected as follows:

```
TRACK
GOC 1* U C1 C7+2 /
```

This requests initialization with respect to the GOC, no normalization of the output volumes and that tracking use tracers on the C1 and C7+2 components only.

In the SUMMARY section we request output of the field oil production rate, and the contributions from hydrocarbons originally above and below the GOC:

```
FOPR
FOPRA
FOPRB
```

The resulting production profiles are:

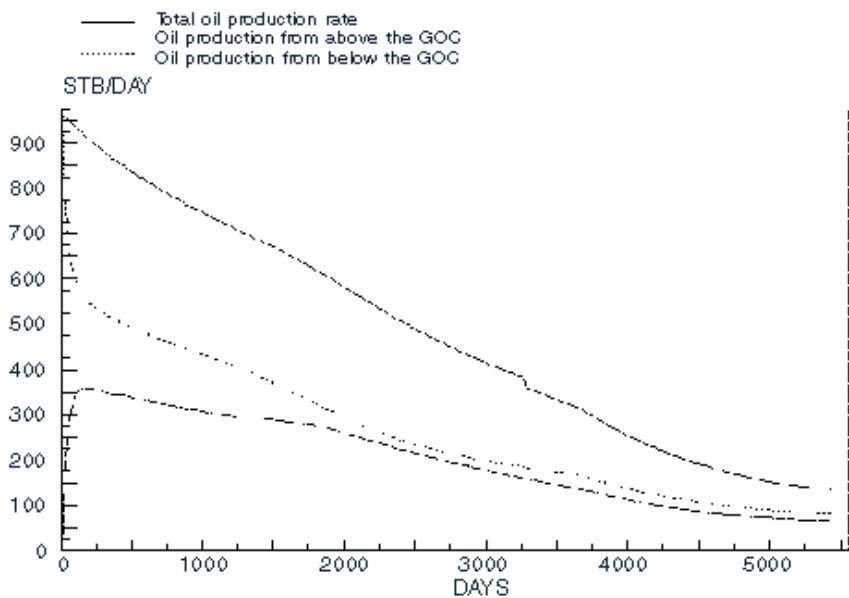


Figure 73.1. Total oil production rate and contributions from above and below GOC

Note: As the production well is completed below the GOC, initially all the production comes from here. This changes quickly as fluid movement driven by gas injection moves hydrocarbon down from cells above the GOC.

Delumping and lumping

This feature, which is available in ECLIPSE 300, is a straightforward implementation of paper SPE 49068 by Leibovici et al. [Ref. 68]. The basic idea is to do a full simulation with as small a number of components as possible and then to use multiple tracers carried on each of these ‘lumped’ components to represent the detailed composition of that component. The rationale and details of the approach are explained in the above paper; here we will concentrate on how to apply this procedure.

The facility is switched on with the `LUMPDIMS` keyword in the `RUNSPEC` section. This gives the number of lumped components (not necessarily as many as the number of components in the ‘lumped’ simulation) and a maximum for the number of detailed components.

The detailed composition of the fluid is described in the `PROPS` section. The `LUMPING` keyword assigns the number of detailed components to each lumped hydrocarbon component and also allows names to be attached to them. **The order in which the detailed components are entered is important as it will be assumed by later keywords.** The initial detailed state of the reservoir is specified with the `DETAILMF` or `DETAILVD` keyword, which determines the proportion of the detailed components making up the lumped components. This can vary with depth. A table is entered for each tracer region (if more than one), defined with keyword `TNUM`.

Various physical properties of the detailed components can also be specified in the `PROPS` section with the `ACFDET`, `MWDETAIL`, `TCRITDET`, `PCRITDET`, `OMEGAADE`, and `OMEGABDE` keywords. These are identical in nature to their ‘normal’ simulation counterparts, though as in a normal simulation, not all the `*CRITDET` keywords are needed in a given simulation.

The detailed composition of an injected fluid is given by using the [WINJEDET](#) keyword in the SCHEDULE section. This must be given for each injection well. For a given well the make-up of the detailed fluid is described in an analogous way to the DETAILED keyword.

Output relies on the standard tracer output with the well, group and field tracer concentration (summary keywords WTPC, GTPC and FTPC) now representing the mole fraction of that detailed component within the overall fluid from that well, with the names of the tracers being those given in the LUMPING keyword. No other summary output is available; in particular the “Traced Mole Fraction” summary keywords; FTMF, GTMF and WTMF will not report useful quantities as these are the product of the tracer concentration with the carrier molar rate.

Diffusion and dispersion

ECLIPSE 100

Prior to 2004A diffusion was only available for environmental tracers. With the 2004A release diffusion was made available for all tracers. Also a form of velocity and concentration dependent dispersion was added, see the keywords [DISPDIMS](#), [TRDIS](#) and [DISPERSE](#).

In 2005A mechanical dispersion (see [Ref. 19]), keyword [HDISP](#), was also introduced. Note that mechanical dispersion is sometimes referred to as hydraulic dispersion, or streamwise diffusion, or streamwise dispersion.

Treating passive tracers as non-linear

Sometimes problems can occur with passive tracers, which show as solution 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.

Tip: 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 timestep is chopped and the whole solution (including the underlying reservoir simulation) is repeated with the new timestep. 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.

Tip: 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.

Tip: 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.

74

Units

There are four unit conventions currently used:

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

- 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 shown below.

Quantity	Metric	Field	Lab	PVT-M
Length, depth, radius	m	ft	cm	m
Time	day	day	hr	day
Area	m ²	ft ²	cm ²	m ²
Density	kg/m ³	lb/ft ³	gm/cc	kg/m ³
Pressure (absolute)	barsa	psia	atma	atma
Pressure (difference)	bars	psi	atm	atm
Temperature (absolute)	K	°R	K	K
Temperature (relative and difference)	°C	°F	°C	°C
Compressibility	1/bars	1/psi	1/atm	1/atm
Viscosity	cP	cP	cP	cP
Permeability	mD	mD	mD	mD
Liquid surface volume	sm ³	stb	scc	sm ³
Gas surface volume	sm ³	Mscf	scc	sm ³
Reservoir volume	rm ³	rb	rcc	rm ³
Volume	m ³	ft ³	cc	m ³

Quantity	Metric	Field	Lab	PVT-M
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.rm ³ /day/bars	cP.rb/day/psi	cP.rcc/hr/atm	cP.rm ³ /day/atm
Mass	kg	lb	gm	kg
Moles	kg-M	lb-M	gm-M	kg-M
Energy	kJ	Btu	J	kJ
Calorific value	kJ/sm ³	Btu/Mscf	J/scc	kJ/sm ³
Heat capacity	kJ/kg/K	Btu/lb/°R	J/gm/K	kJ/kg/K
Volumetric heat capacity	kJ/m ³ /K	Btu/ft ³ /°R	J/cc/K	kJ/m ³ /K
Thermal conduction transmissibility	kJ/day/K	Btu/day/°R	J/hr/K	kJ/day/K
Thermal conductivity	kJ/m/day/K	Btu/ft/day/°R	J/cm/hr/K	kJ/m/day/K
Surface tension	dynes/cm	dynes/cm	dynes/cm	dynes/cm
Surface tension (E100 foam and surfactant models)	N/m	lbf/in	dynes/cm	N/m
Velocity	m/day	ft/day	cm/hr	m/day
Velocity (Multisegment Well Model)	m/s	ft/s	cm/s	m/s
Device strength	bars.day ² /(kg.m ³)	psi.day ² /(lb.ft ³)	atm.hr ² /(gm.cc)	atm.day ² /(kg.m ³)

Table 74.1: A table of the units used for four conventions

Note: The standard definition of 1 mole of an element or compound is in gram-mole and is in terms of the number of atoms or molecules. There are $6.022 \cdot 10^{23}$ atoms in 1 mole of any element. Everything is relative to Carbon-12 so that 1 mole of Carbon-12 weighs 12 grams. In order to work in different unit systems, the number of moles is reported in kilogram-mole (“kg-M”), pound-mole (“lb-M”) or gram-mole (“gm-M”). They are defined for Carbon-12 so that 1 kg-M weighs 12 kg and 1 lb-M weighs 12 lb. This means that there are 1000 times more atoms in 1 kg-M than in 1 gm-M. The relationships with the standard mole are: 1 kg-M = 10^3 mol, 1 lb-M = $0.45359237 \cdot 10^3$ mol, 1 gm-M = 1 mol.

Note: The multiplier M (used for the volume outputs) stands for a million in all units except for the FIELD units, where it means a thousand.

Constants

The following table gives the values of some principal constants in the four unit conventions.

Quantity	Metric	Field	Lab	PVT-M
Gravity constant (E100)	0.0000981 m ² bars/kg	0.00694 ft ² psi/lb	0.000968 cm ² atm/gm	
Gravity constant (E300)	0.0000980665 m ² bars/kg	0.00694444 ft ² psi/lb	0.000967841 cm ² atm/gm	0.0000967841 m ² atm/kg
Darcy constant (E100)	0.008527	0.001127	3.6	
Darcy constant (E300)	0.00852702	0.00112712	3.6	0.00864
Atmospheric pressure (E100)	1.013 barsa	14.7 psia	1.0 atma	
Atmospheric pressure (E300)	1.01325 barsa	14.6959 psia	1.0 atma	1.0 atma
Density of air (E100)	1.22 kg/m ³	0.0763 lb/ft ³	0.00122 gm/cc	
Density of air (E300)	1.2232 kg/m ³	0.076362 lb/ft ³	0.0012232 gm/cc	1.2232 kg/m ³
Density of water (E100)	1000.0 kg/m ³	62.4 lb/ft ³	1.0 gm/cc	
Density of water (E300)	999.014 kg/m ³	62.3664 lb/ft ³	0.999014 gm/cc	999.014 kg/m ³
Gas constant, R	0.083143 m ³ bars/K/kg-M	10.732 ft ³ psi/°R/lb-M	82.055776 cc.atm/K/gm-M	0.08205576 m ³ atm/K/kg-M

Table 74.2: Constants used in the four unit conventions

Standard conditions are taken as one atmosphere and 60 °F.

The Darcy constant is defined by:

- 1 mD.m = 0.00852702 cP.rm³/day/bars
- 1 mD.ft = 0.00112712 cP.rb/day/psi
- 1 mD.cm = 3.6 cP.rcc/hr/atm
- 1 mD.m = 0.00864 cP.rm³/day/atm

Conversion factors

The following table gives some useful conversion factors between the unit systems.

Quantity	Metric	Field	Other units
Length	1 m	=3.28084 ft	
	0.3048 m	=1 ft	
Volume	1 m ³	=35.31466 ft ³	=6.289811 bbl
	0.02831685 m ³	=1 ft ³	=0.1781076 bbl
Mass	1 kg	= 2.204623 lb	=0.001 (metric) ton
	0.45359237 kg	=1 lb	=0.0005 (short) ton
Density	1 kg/m ³	=0.06242797 lb/ft ³	=10 ⁻³ gm/cc
	16.01846 kg/m ³	=1 lb/ft ³	=0.01601846 gm/cc
Pressure	1 bars	=14.50377 psi	=0.986923 atm
	0.06894757 bars	=1 psi	=0.068046 atm
Gas-liquid ratio	1 m ³ /m ³	=5.614583·10 ⁻³ Mcf/bbl	
	178.1076 m ³ /m ³	=1 Mcf/bbl	
Temperatures	1 K	=1/1.8 °R	

Table 74.3: Some useful conversion factors

A note regarding the use of the **ton** as a unit of mass:

- In metric-based systems, a ton (also called a **metric ton** or **tonne**) is 1,000 kg.
- In imperial-based systems, a **short ton** is 2,000 lbs (this is also known as a **net ton** and is the common definition of a **ton** in the U.S.) while a **long ton** is 2,240 lbs
- Prior to 2007.1, when working in FIELD units, ECLIPSE 300 reported both mass rates and mass totals (for example, the WOMR and GWMT summary keywords) in **long tons**. This was changed to **short tons** at 2007.1. The previous behavior can be recovered by setting [OPTIONS3](#) item 112 to a non-zero value. An ‘L’ (for long) or ‘S’ (for short) has been added to the name of the unit in the output from 2007.1 to clarify which type of ton is being used.

API gravity

$$\text{API gravity} = (141.5 / \text{liquid gravity}) - 131.5$$

where the liquid gravity is calculated as the liquid oil density divided by the water density at standard conditions.

Conversion for an ideal gas (Z=1)

For an ‘ideal gas’ Z-factor of unity, we can use the ideal gas equation:

$$PV = nRT \quad \text{Eq. 74.1}$$

where:

P is pressure

V is volume

n is number of moles

R is the gas constant

T is temperature

Using equation 74.1, we can calculate the following:

- When $P = 14.7$ psia and $T = 519.67$ °R,
 - the volume occupied by one lb-mole of gas is:

$$V_{\text{molar}} = 379.39445 \text{ ft}^3$$

- the number of lb-moles in unit volume is:

$$n_{\text{moles}} = 0.00263578 \text{ lb-M/ft}^3$$

- When $P = 1.013$ barsa and $T = 288.15$ K,
 - the volume occupied by one kg-mole of gas is:

$$V_{\text{molar}} = 23.650203 \text{ m}^3$$

- the number of kg-moles in unit volume is:

$$n_{\text{moles}} = 0.04228293 \text{ kg-M/m}^3$$

75

Velocity calculations

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

In a typical reservoir simulation, inter-cell flows are calculated but the fluid velocities are not explicitly calculated. Velocities are only calculated for specific models. These include:

- polymer shear-thinning effects (["Treatment of the non-Newtonian rheology"](#))
- velocity-dependent relative permeabilities and the Forchheimer effect, (["Non-Darcy flow"](#))
- ["Asphaltene deposition"](#)
- tracer dispersion
- miscibility effects due to surfactant (["Surfactant model"](#))
- gas mobility reduction due to foam injection(["Gas mobility reduction"](#)).

Fluid velocities inside the well are also calculated and are discussed in:

- ["Multisegment wells"](#)
- the ["Wellbore friction option"](#).

This chapter describes the way reservoir velocities are calculated.

Velocity concepts

Velocities can be determined from flows, which are calculated across the cell boundaries. This presents two problems. Firstly flows are usually calculated after the fluid properties are calculated (since they themselves depend on the fluid properties), and secondly, flows depend on the pressure and fluid properties in both the upstream and downstream cells. Therefore, unlike other fluid properties, the velocity is a function of the solution in a cell and in the neighboring cells. For these two reasons, velocity terms are often used explicitly, that is they are calculated at the previous timestep, so they can be determined before they are used and the derivative terms can be ignored.

The velocity is a vector, with x , y and z components. The cell boundary velocities represent the components of the velocity in the direction normal to the boundaries. To calculate a single-cell velocity it is convenient to assume the cell boundaries are orthogonal and to use the resultant velocity (with the local cell direction components rather than global Cartesian components):

$$\| v \| = \sqrt{v_x^2 + v_y^2 + v_z^2} \quad \text{Eq. 75.1}$$

where the velocity components are calculated by averaging the flows in and out on opposite faces:

$$v_x = (v_x^+ + v_x^-)/2 \quad \text{Eq. 75.2}$$

This method of averaging fails to deal with the case when a cell may have more than two neighbors in each direction, for example if it is connected to a local grid refinement (LGR) or if there are other non-neighbor connections. In this case, an area-weighted average can be applied more consistently:

$$v_x = (\Sigma A_x v_x) / (\Sigma A_x) \quad \text{Eq. 75.3}$$

where the sum is over all faces (positive and negative) in the same direction. Here the sense of the velocities is in the $+x$ direction, but see below.

If a cell contains a well, then it would be expected that the velocities near the well would be much larger than the background flow velocities. Well flows are not included in these calculations, and the only way at present to capture the well velocity would be to have sufficiently small grid cells around the well, so that the velocity into the cell represents the velocity close to the well. In these cases equation 75.3 can be reinterpreted to represent the average inflow velocity into the cell (the sense of v_x is into the cell rather than $+x$), as, with the original interpretation of equation 75.3, then if the flow is into the well in all directions, the net velocity is zero.

Velocity definitions

Different models use different definitions of velocity. For example, the Darcy velocity u_{Dp} , interstitial velocity u_{Ip} or phase velocity u_p may be used, which are calculated in slightly different ways:

$$u_{Dp} = \frac{Q_p}{A} \quad \text{Eq. 75.4}$$

$$u_{Ip} = \frac{Q_p}{A\varphi} \quad \text{Eq. 75.5}$$

$$u_p = \frac{Q_p}{A\varphi S_p} \quad \text{Eq. 75.6}$$

where

Q_p is the phase volume flow,

A is the cell boundary area,

φ is the porosity,

S_p is the mobile phase saturation.

The Darcy velocity does not take into account the porosity, which limits the flow area. Neither the Darcy velocity nor the interstitial velocity take account of the phase saturations, and so they are similar to superficial velocities used in well flow modeling. If the phase velocity, equation 75.6 is used, but is averaged over the entire fluid volume, by multiplying by the saturation, the resulting average velocity is the interstitial velocity, equation 75.5. All these velocities are cell-scale velocities and are often used to approximate pore-scale phenomena. As such, the choice of which one to use is somewhat arbitrary, and model parameters can be adjusted accordingly, as long as it is clear which velocity is being used.

ECLIPSE 100 velocity calculations

The primary variable calculated by ECLIPSE 100 is the mass flow for each phase through a cell face. The volumetric flow is obtained from this by dividing by the density (equivalently, multiplying by the formation volume factor). The area this flows through is the part of the cell face area occupied by those pores which are occupied by the mobile fraction of this phase. The velocity is obtained from the volumetric flow by dividing by this flow area. This is equivalent to the phase velocity:

$$u_p = \frac{B_p F_p}{A\varphi(S_p - S_c)} \quad \text{Eq. 75.7}$$

where:

u_p is the velocity of phase p ,

B_p is the formation volume factor of phase p ,

F_p is the mass flow of phase p ,

A is the area of the cell face through which it flows

φ is the porosity at the cell face,

S_p is the saturation of phase p , and

S_c is the equivalent critical saturation.

Before the 2009.1 version, the correction for the mobile saturation was not used. The older formulation can be restored by setting item 137 of the OPTIONS to 1.

The polymer shear thinning option still uses the pre-2009.1 calculation to be consistent with its original validation. To output these velocities for visualization, again use OPTIONS 137.

Capillary number

The capillary number is used in both the "Foam model" and the "Surfactant model". The capillary number represents the ratio of shear forces to capillary (surface tension) forces and is often defined as:

$$N_c = \frac{u_p \mu_p}{\sigma_{pq}} \quad \text{Eq. 75.8}$$

where:

u_p is the velocity of phase p ,

μ_p is the viscosity of phase p ,

σ_{pq} is the surface tension between phases p and q .

The velocity across a cell face can be expressed in terms of the volume flow Q_p which can in turn be expressed in terms of the transmissibility T and phase potential P_p :

$$u_p = \frac{Q_p}{A_m} = \frac{C_N}{C_D} \frac{T k_{rp} \Delta P_p}{A_m \mu_p} \quad \text{Eq. 75.9}$$

where:

C_N is a unit conversion constant,

C_D is the Darcy constant,

A_m is the flow area, which could be A , $A\varphi$ or $A\varphi S_p$ depending on the velocity model, and

k_{rp} is the relative permeability of phase p .

When equation 75.9 is substituted into equation 75.8, the capillary number (for a boundary face) is:

$$N_{cn} = \frac{C_N}{C_D} \frac{T k_{rp} \Delta P_p}{A_m \sigma_{pq}} = \left(k_{rp} \frac{A}{A_m} \right) \cdot \left(C_N \frac{K}{\sigma_{pq}} \frac{\partial P_p}{\partial n} \right) \quad \text{Eq. 75.10}$$

where:

A is the cell boundary area,

n is a direction, x , y or z .

The $k_{rp} \frac{A}{A_m}$ term in equation 75.10 is often ignored:

$$N_{cn} = \frac{C_N}{C_D} \frac{T \Delta P_p}{A \sigma_{pq}} = \left(C_N \frac{K}{\sigma_{pq}} \frac{\partial P_p}{\partial n} \right) \quad \text{Eq. 75.11}$$

This is equivalent to using the Darcy velocity ($A_m = A$), and assuming the flow is for a single phase ($k_{rp} = 1$). The cell capillary number is then determined from the boundary capillary numbers:

$$N_c = \frac{C_N}{C_D} \left\| \frac{T \Delta P_p}{A} \right\| \frac{1}{\sigma_{pq}} = C_N \frac{\| K \cdot \nabla P_p \|}{\sigma_{pq}} \quad \text{Eq. 75.12}$$

The value of the unit conversion constant C_N depends on the units of the terms used in equation 75.12 as shown in the following table.

	Metric	Field	Lab	PVT-M
K	mD	mD	mD	mD
ΔP_p	bar/m	psi/ft	atm/cm	atm/m
σ_{pq}	N/m	lbf/in	dyne/cm	N/m
C_N	$10^{-10}(\text{bar/atm}) = 0.9869234 \cdot 10^{-10}$	$12.10^{-15} (\text{m}/\text{ft})^2(\text{bar}/\text{atm}) = 1.274784 \cdot 10^{-13}$	10^{-5}	10^{-10}
C_D	0.00852702 mD.day.bar/m ² /cP	0.001127 mD.day.psi.(ft/bbl)/cP	3.6 mD.hr.atm/cm ² /cP	0.00864 mD.day.atm/m ² /cP

Table 75.1: Capillary number constant and Darcy Constant

For two phases p and q there are two “natural” capillary numbers that could be used:

$$N_{cpq} = \frac{v_p \mu_p}{\sigma_{pq}}$$

or:

$$N_{cqp} = \frac{v_q \mu_q}{\sigma_{pq}}$$

ECLIPSE 100 capillary numbers

ECLIPSE 100 uses the second formulation, equation 75.12 for capillary numbers.

In the surfactant model, the oil phase potential is used together with the oil-water surface tension to determine the capillary number:

$$N_{cow} = C_N \frac{\| K \cdot \nabla P_o \|}{\sigma_{ow}} \quad \text{Eq. 75.13}$$

In the foam model, the oil phase potential is again used together with the water-gas surface tension:

$$N_{cwg} = C_N \frac{\| K \cdot \nabla P_o \|}{\sigma_{wg}} \quad \text{Eq. 75.14}$$

In both these models an RMS value of $\| K \cdot \nabla P_o \|$ is used, after the components have been averaged in a manner analogous to equation 75.1:

$$\| K \cdot \Delta P \| = \sqrt{\left(K_x \cdot \frac{\partial P}{\partial x} \right)^2 + \left(K_y \cdot \frac{\partial P}{\partial u} \right)^2 + \left(K_z \cdot \frac{\partial P}{\partial z} \right)^2} \quad \text{Eq. 75.15}$$

which for a given cell i with neighbor j is calculated in the x direction according to:

$$K_x \cdot \frac{\partial P}{\partial x} = 0.5 \left[\left(\frac{K_x}{D_x} \right)_{i-1, i} \cdot (P_i - P_{i-1}) + \left(\frac{K_x}{D_x} \right)_{i, i+1} \cdot (P_{i+1} - P_i) \right] \quad \text{Eq. 75.16}$$

and similarly for the y and z directions. The K / D value is calculated in an analogous manner to the transmissibility and depends upon how the geometry has been specified.

ECLIPSE 300 capillary numbers

ECLIPSE 300 uses the first formulation in equation 75.12 for capillary numbers.

In the surfactant model, the oil phase potential is used together with the oil-water surface tension:

$$N_{cow} = \frac{C_N}{C_D} \left\| \frac{T \Delta P_o}{A} \right\| \frac{1}{\sigma_{ow}} \quad \text{Eq. 75.17}$$

In the foam model, the gas phase potential is used together with the water-gas surface tension:

$$N_{cwg} = \frac{C_N}{C_D} \left\| \frac{T \Delta P_g}{A} \right\| \frac{1}{\sigma_{wg}} \quad \text{Eq. 75.18}$$

The boundary terms are averaged in a manner analogous to equation 75.2 before the RMS value is calculated.

Matrix-fracture flows

In dual porosity simulations, the matrix-fracture flow contribution can be calculated in a similar manner to equations 75.17 and 75.18 but with the area term calculated by

$$A = V\sqrt{\sigma} \quad \text{Eq. 75.19}$$

where V is the bulk volume of the cell and σ is the sigma factor representing the matrix block size. For fracture cells this contribution to the capillary number is likely to be small and will be ignored.

Asphaltene deposition and entrainment

In the ECLIPSE 300 three-component asphaltene model (see "Deposition - entrainment") the oil phase velocity is calculated using an RMS of the directional velocities:

$$\bar{U}_o = C \parallel \frac{k_{ro} T \Delta P_o}{\mu_o A} \parallel \frac{1}{\varphi(1-S_s)S_o} \quad \text{Eq. 75.20}$$

Here the relative permeability is included in the flow term, and the effective area of flow includes the porosity, oil saturation and solid saturation (representing deposited asphaltene).

The unit conversion constant is given in the following table.

	Metric	Field	Lab	PVT-M
T	cP.m ³ /day/bar	cP.bbl /day/psi	cP.cm ³ /hr/Atm	cP.m ³ /day/Atm
ΔP_o	bar	psi	Atm	Atm
μ_o	cP	cP	cP	cP
A	m ²	ft ²	cm ²	m ²
C	1	5.61458 ft ³ /bbl	1	1
\bar{U}_o	m/day	ft/day	cm/hr	m/day

Table 75.2: Velocity constant

76

Vertical equilibrium

This chapter describes the ECLIPSE 100 vertical equilibrium option.

<input checked="" type="checkbox"/>	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

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 76.1), 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.

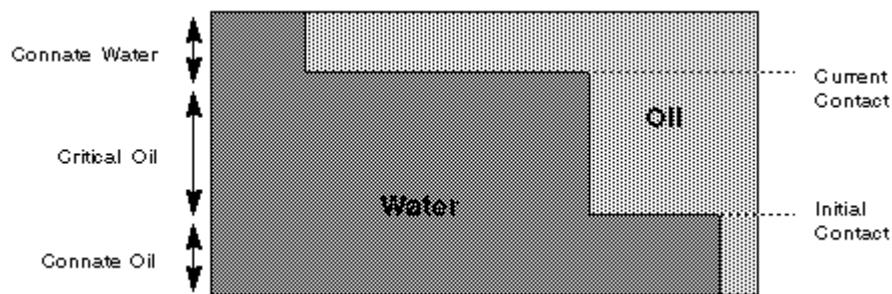


Figure 76.1. A cell with approximate hydrostatic equilibrium

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 arguments **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 cannot be used in combination with the Hysteresis option in the same run.

The endpoint scaling option/initial mobile fluid end point correction should not be used with the vertical equilibrium option.

The cell saturation distribution

The saturation distribution in a cell assumed by the VE model is based on the assumption that the phase hydrostatic potential is independent of depth. In the absence of capillary pressure effects, the saturation distribution becomes a step function. The form of this depends on the phase option used.

The oil-water case

The distribution is derived from the water saturation value, S_w . If Z_c is the current oil water contact depth, and Z_l is the lowest contact depth which has occurred in the run, the distribution is as shown below. This has three depth intervals, as follows:

Region 1

Above the current oil water contact. The water saturation takes the connate value, S_{wco} , and the oil saturation is $1-S_{wco}$.

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 $1-S_{ocr}$.

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.

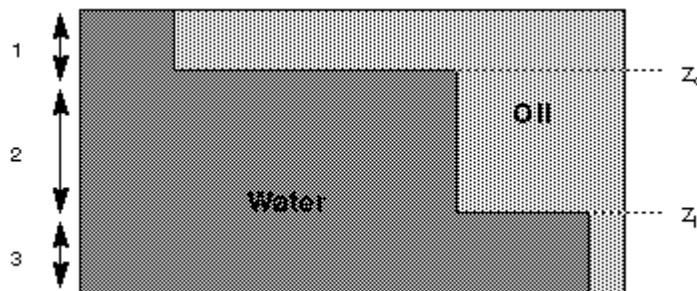


Figure 76.2. Oil-water vertical equilibrium

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.

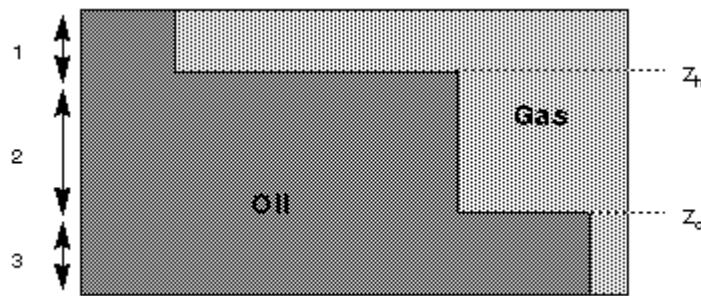


Figure 76.3. Oil-gas vertical equilibrium

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.

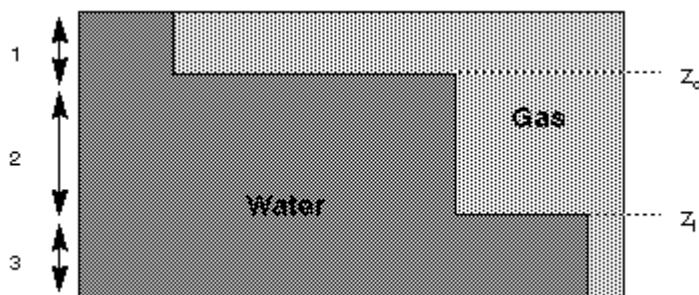


Figure 76.4. Gas-water vertical equilibrium

The three phase case

In this case S_w and S_g are distinct variables. The usual distribution is shown in figure 76.5. There are five depth intervals, as follows:

Region 1

Above the highest gas oil contact. The water saturation has the connate value, S_{wco} , and the oil saturation is the connate with respect to gas, S_{ocog} . The gas saturation is thus $1 - S_{wco} - S_{ocog}$.

Region 2

Below the highest oil gas contact, but above the current. Gas has displaced oil, so the critical oil saturation with respect to gas remains, S_{ogcr} , with $S_w = S_{wco}$ and $S_g = 1 - S_{ogcr} - S_{wco}$.

Region 3

Below the current oil gas contact and above the current oil water contact. The water and gas saturations take their connate values, (S_{wco} , S_{gco}), and the oil saturation is $1 - S_{wco} - S_{gco}$.

Region 4

Below the current oil water contact, but above the lowest oil water contact. Water has displaced oil, so the critical oil saturation with respect to water remains, S_{owcr} , with $S_w = 1 - S_{owcr} - S_{gco}$.

Region 5

Below the lowest oil water contact. The gas saturation is connate, S_{gco} , and the oil saturation is the connate value with respect to water, S_{ocow} . The water saturation is $1 - S_{ocow} - S_{gco}$.

In any of the phase combinations, one or more of the regions may not exist, depending on the conditions in a cell.

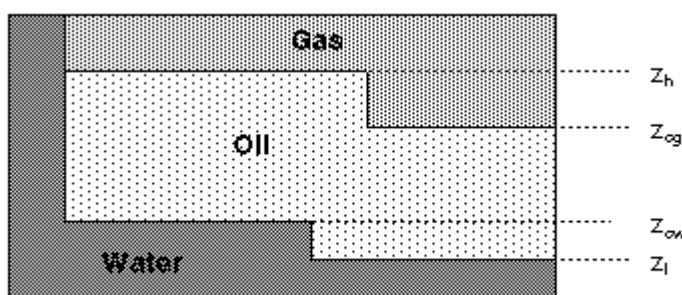


Figure 76.5. Three-phase vertical equilibrium

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):

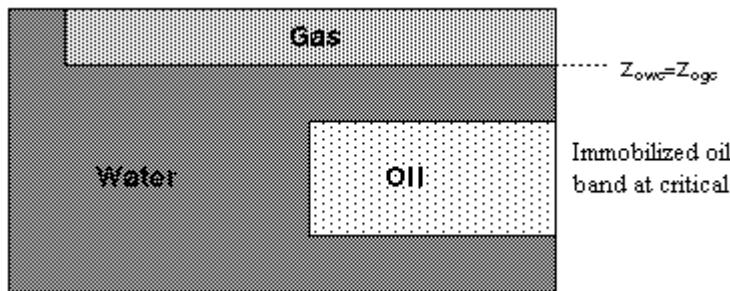


Figure 76.6. Three phase equilibrium with immobilized oil

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:

Saturation	Description
Connate water saturation, S_{wco}	The smallest saturation value in the SWFN table.
Critical water saturation, S_{wcr}	The smallest saturation value for which a non-zero k_{rw} value exists.
Connate gas saturation, S_{gco}	The smallest saturation value in the SGFN table.
Critical gas saturation, S_{gcr}	The smallest saturation value for which a non-zero k_{rg} value exists.
Connate oil saturation in water, S_{ocow}	This is $1 - S_{wmax} - S_{gco}$, where S_{wmax} is the maximum saturation value in the SWFN table.
Connate oil saturation in gas, S_{ocog}	This is $1 - S_{gmax} - S_{wco}$, where S_{gmax} is the maximum saturation value in the SGFN table.
Critical oil saturation in water, S_{owcr}	This is the lowest oil saturation for which a non-zero k_{ro} value exists, with $S_g = S_{gco}$ (that is, from the second column of the SOF3 table in the three phase case or from SOF2 in the two phase case).
Critical oil saturation in gas, S_{ogcr}	This is the lowest oil saturation for which a non-zero k_{ro} value exists, with $S_w = S_{wco}$ (that is from the third column of the SOF3 table in the three phase case or from SOF2 in the two phase case).

VE relative permeabilities and capillary pressures

In the ECLIPSE VE algorithm, the current best solution to the non-linear fluid flow equations is held as a set of cell saturations. The current contact levels are calculated from the saturations using the models for the segregated fluid distributions described in the previous section. (The lowest and highest levels, Z_l and Z_h , are stored as arrays, and will affect 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:

$$P_{cow} = g (\rho_w - \rho_o)(Z_{owc} - Z_{center}) \quad \text{Eq. 76.1}$$

where

ρ_w and ρ_o are the phase densities

g is the acceleration due to gravity constant

Similarly, for the gas phase, the pseudo capillary pressure is given by:

$$P_{cog} = g (\rho_o - \rho_g)(Z_{goc} - Z_{center})$$

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 cutoff 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.

$$K_{r \text{ tot}} = K_{r \text{ VE}} + K_{rr}(S_o) \quad \text{Eq. 76.2}$$

where

$K_{r \text{ tot}}$ is the total relative permeability

$K_{r \text{ VE}}$ is the VE relative permeability calculated from the contact depth

$K_{rr}(S_o)$ is the relative permeability from the RESIDNUM table

S_o is the oil saturation.

The RESIDNUM 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:

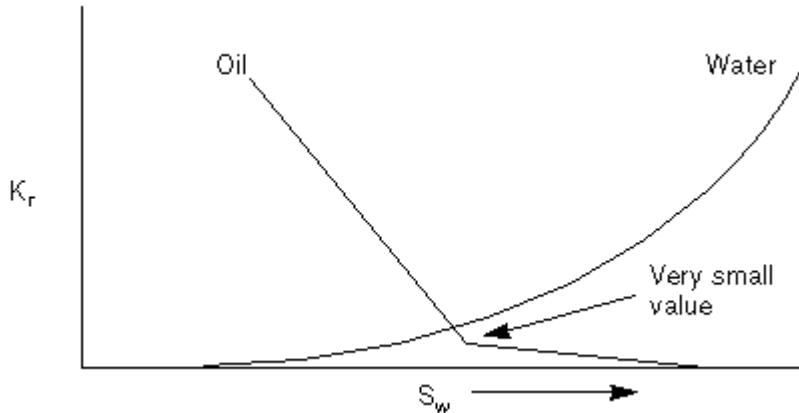


Figure 76.7. Relative permeability behavior for a residual flow model

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.

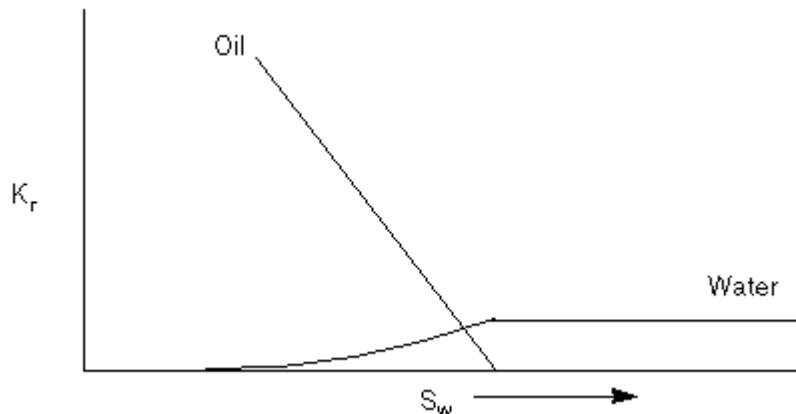


Figure 76.8. V.E Curves (SATNUM)

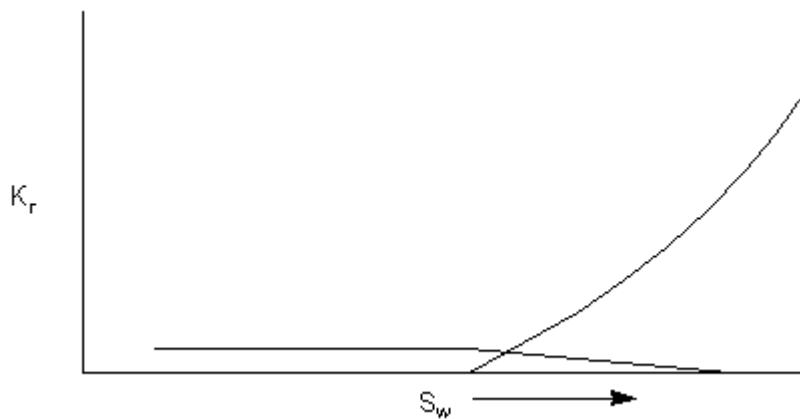


Figure 76.9. Residual flow model

Miscellaneous points

RESTART files produced by VE runs include the lowest and highest contact depths, as required by the phase combination used. It is not possible to restart a VE run from a non-VE run.

Although the saturation distributions assumed by the VE algorithm are based on zero capillary pressures, non-zero values may be entered in the PROPS data section, to be used if the [VEFRAC](#) mixing parameter is not unity; the VE fraction will ignore rock capillary pressures.

The VE model described above assumes a three zone saturation distribution. It is possible to extend this model to allow four zones. In such a case, considering the oil water phase combination, there would be a critical water saturation up to a highest contact depth, and a (possibly different) connate saturation above this. This extension to a 'four cycle' VE model could be provided in future if demand exists.

If twisted sloping faults are used, the intersection of a cell face with a plane of constant depth becomes slightly curved. This curvature is ignored when setting up the cell geometry data for VE. However, the fractional volumes below distinct depths are then re-normalized to ensure that the volume fraction below the highest distinct depth is 1.

If the reservoir geometry is entered using [TOPS](#), [DX](#), [DY](#) etc., rather than with corner point geometry ([COORD](#), [ZCORN](#) etc.), the top and bottom of the cells will be flat. In this case an unrealistic step function change in the relative permeability would be obtained when the fluid contact reaches the top of a cell, which would cause problems in solving the non-linear flow equations. To avoid this, the cell top is 'spread' slightly, over 10% of the cell depth, so that the flow relative permeability rises from zero to its maximum value as the fluid contact traverses this interval.

In a dipping reservoir specified using [TOPS](#), [DX](#) etc. the VE algorithm will prevent flow to the adjacent cell if the water level is below the bottom of the adjacent cell. A better method of specifying a dipping reservoir would be to use corner point geometry ([COORD](#), [ZCORN](#) etc.). However the full face of each cell can be specified for flow by using the [OPTIONS](#) keyword, setting [OPTIONS\(19\)=1](#). A second option can be used by specifying [OPTIONS\(19\)=2](#), in which case the flows are calculated assuming a corner point approximation to the original grid. The pore volume and transmissibilities are preserved, however the VE flow calculations are performed assuming the cells to be part of a continuous layer. This corner point approximation will be output to the [GRID](#) file and hence can be plotted using a pre-processor such as PETREL. This latter option is used in Local Grid Refinement cases if the grid is entered as a block centered grid.

Consider a reservoir specified using corner point geometry. When there is a gap of non-zero thickness between a pair of adjacent active cells in a column which are connected together (such as for a pinchout of non-zero thickness due to use of MINPV), contact depths cannot be resolved in this dead space. In this case, setting OPTIONS(52)=2 instructs ECLIPSE to modify corner-point depths on the bottom face of the topmost cell in the pair to be equal to those for the top face of the bottommost 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 three-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 as 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 \leq 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:

Keyword	Description
DEPTH	
VOLUME	Normalized volume tables, including porosity and NTG values
SOWCR	Oil saturation at critical water (residual oil to water)
SOGCR	Oil saturation at critical gas (residual oil to gas)
SWCR	Critical water saturation
SGCR	Critical gas saturation
SOWMX	Oil saturation at maximum water saturation
SOGMX	Oil saturation at maximum gas saturation
SWCO	Connate water saturation

Keyword	Description
SGCO	Connate gas saturation
KRO1	K_{ro} at SWCO, SGCR
KRO2	K_{ro} at SWCO, SGCO
KRO3	K_{ro} at SWCR, SGCR
KRW1	K_{rw} at SOWCR, SGCO
KRW2	K_{rw} at SOWMX, SGCO
KRG1	K_{rg} at SOGMX, SWCO
KRG2	K_{rg} at SOGCR, SWCO

Table 76.1: Cell volume table

Keyword	Description
DEPTH	Depth
AREA	Normalized surface area tables
KRO2	K_{ro} at SWCO, SGCO
KRW1	K_{rw} at SOWCR, SGCO
KRW2	K_{rw} at SOWMX, SGCO
KRG1	K_{rg} at SOGMX, SWCO
KRG2	K_{rg} at SOGCR, SWCO

Table 76.2: Cell face tables

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.

77

Water components

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

In ECLIPSE, water is traditionally modeled using a single component. Density and viscosity variations caused by salinity changes can be modeled using the Brine Tracking option, see "[Brine tracking](#)". The brine tracking option uses a tracer to track the salt movement in the reservoir.

This single-component model can be applied to many practical cases. However, there are some situations where more than one water component is required:

- as an alternative brine model. When density variations are large, the tracer method may become unstable, since the tracer equations are solved after the flow equations.
- to model brine in multisegment wells, see "[Multisegment wells](#)". At present the brine tracking option cannot be used at with the multisegment well option.
- to model reactions involving water borne components such as bacteria or dissolved solids, refer to "[Chemical reactions](#)".
- to model a surfactant component for the foam model, see "[Foam model](#)".
- to model a surfactant component for surface tension effects, see "[Surfactant model](#)".
- to model a polymer component for water phase viscosity effects, see "[Polymer model](#)".

Water component properties

Density

The molar density of water is given by

$$b_w = \frac{1}{V_w} \quad \text{Eq. 77.1}$$

where V_w is the specific volume given by:

$$V_w = \sum_c w_c \cdot \frac{MW_c}{\rho_{ref_c} \cdot (1 + X_c + 0.5X_c^2)} \quad \text{Eq. 77.2}$$

In equation 77.2 the sum is over the water components where:

w_c is the water component fraction in the water phase

ρ_{ref_c} is the reference density, specified with keyword [DREFW](#)

MW_c is the water component molecular weight, specified by the keyword [MWW](#)

The compressibility terms X_c in equation [77.2](#) are given by

$$X_c = c_{P_c} \cdot (P - P_{ref_c})$$

where

c_{P_c} is the water component compressibility, specified with keyword [CREFW](#)

P_{ref_c} is the reference pressure, specified with keyword [PREFW](#)

Viscosity

The viscosity of water is given by

$$\mu_w = \sum_c w_c \cdot \frac{\mu_{ref_c}}{(1 + Y_c + 0.5 Y_c^2)} \quad \text{Eq. 77.3}$$

where

w_c is the water component fraction in the water phase

μ_{ref_c} is the reference viscosity, specified with keyword [VREFW](#)

The “viscosibility” terms Y_c in equation [77.3](#) are given by

$$Y_c = -c_{V_c} \cdot (P - P_{ref_c})$$

where

c_{V_c} is the “viscosibility”, specified with keyword [VREFW](#)

P_{ref_c} is the reference pressure, specified with keyword [PREFW](#)

Component names

Water components can be named using the [WNAMES](#) keyword. If this keyword is not specified then the components take the default names. If there is only one water component, the default name is WATER. However, if there are more than one water components they are named WAT01, WAT02, and so on. The names are used in reporting and for associating a [TRACER](#) with a component.

Surface properties

For surface calculations, the optional keywords [MWWS](#), [DREFWS](#), [CREFWS](#) and [PREFWS](#) can be used to supply the values used in equations [77.1](#) and [77.2](#). The default, if these are not supplied, is to use the reservoir values given by [MWW](#), [DREFW](#), [CREFW](#) and [PREFW](#) (or [STCOND](#)).

Aquifer properties

The [AQSTRAW](#) keyword must be used to specify the composition of each analytic aquifer defined by [AQUCHWAT](#), [AQUCT](#), [AQUFETP](#) or [AQUFLUX](#). If this is not provided an error message will be emitted to the PRINT file and the water composition of an analytic water aquifer will be made up entirely of the last water component. However if the last water component is a surfactant, polymer or alkaline component (see [CWTYPE](#)), then it will not be used; instead the preceding water component will be used for the default aquifer water composition.

For a numerical aquifer defined by [AQUNUM](#), the aquifer composition is the water composition of the cell representing the aquifer, which is set in the usual way (see [SOLUTION section keywords](#)).

Any water component is allowed to leave the reservoir or enter it, depending on the aquifer composition. If the item 131 of the OPTIONS3 is set to 1, this reverts to the pre-2008.1 behavior where the water entering / leaving the reservoir is made up entirely of the last water component. Additionally, the water properties used in the analytic aquifers are calculated as described above. If the item 119 of OPTIONS3 is set to 1, this reverts to the pre-2008.1 behavior where the PVT tables were used. See "[Aquifer Modeling Facilities](#)".

Converting PVTW data to water component data

In standard simulations, water is modeled as a single component, with properties specified by the [PVTW](#) and [DENSITY](#) keywords. This case can also be modeled using the multiple component water option, with a single water component. Table [77.1](#) gives values for the water component keywords, derived from the PVTW and DENSITY data.

Note: PVTW and DENSITY data are specified for each PVT region, while water component data is specified for each EOS region, so the two models can produce different answers if the regions do not match.

Keyword	Value
MWW	18.015
PREFW	Item 1 of PVTW
DREFW	$DENS_w / BW_{ref}$ where $DENS_w$ is Item 2 of DENSITY BW_{ref} is Item 2 of PVTW
CREFW	Item 3 of PVTW
VREFW	Items 4 and 5 of PVTW

Table 77.1: Water component values

Converting PVTWSALT data to water component data

Water with varying salinity can already be modeled using the Brine Tracking Option, see "[Brine tracking](#)". The Multiple Component Water provides an alternative mechanism, with one component used to model freshwater and another component used to represent brine. In the Brine Tracking Option, water properties are defined using the [PVTWSALT](#) keyword. This can be converted to component data in a similar manner to the [PVTW](#) data, described above, except that the density term must now include the weight of salt

$$DREFW = (DENS_w + SAL_w) / BW_{ref}$$

where SAL_w is the salt concentration.

Effective salinity calculations

Effective salinity calculations are triggered when the multi-component water option is activated ([COMPW](#) in the [RUNSPEC](#) section) and effective salinity parameters are specified using the [SALTEFF](#) keyword in the [PROPS](#) section. In the current release, the keyword [SALTEFF](#) can only be used with the salinity-dependent Surfactant model (see "[Surfactant model](#)") or the Polymer model (see "[Polymer flood model](#)").

Effective salinity calculations can be defined by one of two formulae:

Formula 1: valid for the Surfactant and Polymer models	$E_s = \frac{a_0 + \sum_{i=1}^{N_w} a_i C_i}{b_0 + \sum_{i=1}^{N_w} b_i C_i}$
Formula 2: valid for the Surfactant model only	$E_s = \frac{a_0 + \sum_{i=1}^{N_w} a_i C_i}{b_0 + \frac{1}{C_s} \sum_{i=1}^{N_w} b_i C_i}$

where

C_i is the concentration of the i^{th} water component, and

a_i and b_i , $i = 0, \dots, N_w$ denote the effective salinity parameters specified by the keyword [SALTEFF](#).

The keyword [SALTEFF](#) allows the input of any values of coefficients a_i and b_i . This allows you to define many salinity models (see for example [Ref. 137] *Chapter 3: “Salinity Effect and Ion Exchange”*). The examples below show how these formulae can be used to represent different effective salinity models including those described in the ECLIPSE 100 simulator.

Example 1

The multi-component water option in ECLIPSE 300 can be used as an alternative to the brine model. For example, the user can set three water components: one being fresh water, another representing brine and a final water component representing polymer.

```
COMPW
3 /
WNAMES
WATER SALT POLY /
CWTYPE
1* 1* POLY /
```

The properties of each water component must then be specified with keywords [MWW](#), [PREFW](#), [DREFW](#), [CREFW](#) and [VREFW](#). Information on converting [PVTW](#) and [PVTWSALT](#) data into water component data can be found in sections “[Converting PVTW data to water component data](#)” and “[Converting PVTWSALT data to water component data](#)”.

If the user wishes the effective salinity to represent salt concentration, then the following parameters in [SALTEFF](#) should be specified:

```
SALTEFF
-- Type Eq a0 a1 a2 a3 b0 b1 b2 b3
      POLY 1 0 0 1 0 1 0 0 0 /
```

According to Formula 1, this choice of coefficients in [SALTEFF](#) yields $E_s = C_2$, where C_2 is the salt concentration as desired.

Example 2

Let us now consider the following set of water components: one water component to represent fresh water, one to represent anions in the water phase, another water component to represent divalent cations associated with polymer and finally one water component to represent polymer solution:

```
COMPW
4 /
WNAMES
WATER ANIONS CATIONS POLY /
CWTYPE
1* 1* 1* POLY /
```

If the user wishes the effective salinity to be calculated using formula triggered by keyword [PLYESAL](#) in ECLIPSE 100, then the following parameters should be chosen in [SALTEFF](#)

```
SALTEFF
-- Type Eq a0 a1 a2 a3 a4 b0 b1 b2 b3 b4
      POLY 1 0 0 1 α -1 0 1 0 0 0 0 /
```

where

α is the effective salinity parameter that would be specified in [PLYESAL](#) item 1.

Example 3

Let us now consider a similar set of water components that now contain surfactant: one water component to represent fresh water, one to represent anions in the water phase, another water component to represent divalent cations associated with surfactant and finally one water component to represent Surfactant solution:

```
COMPW
4 /

WNAMES
WATER ANIONS CATIONS SURF /

CWTYPE
1* 1* 1* SURF /
```

If the user wishes the effective salinity to be calculated using the formula triggered by keyword [SURFESAL](#) in ECLIPSE 100, then the following parameters should be chosen in SALTEFF

```
SALTEFF
-- Type Eq a0 a1 a2 a3 a4 b0 b1 b2 b3 b4
SURF 2 0 0 1 0 0 1 0 0 - α 0 /
```

where

α is the effective salinity parameter that would be specified in [SURFESAL](#) item 1.

Units

The extra flexibility in the choice of parameters in SALTEFF means that the units of the resulting effective salinity may vary from one combination of parameters to another:

- In Example 1 above, the coefficients in SALTEFF were chosen in such a way that the resulting effective salinity, according to Formula 1, has the following units: kg/sm³ (METRIC), lb/stb (FIELD), gm/scc (LAB). Considering that the effective salinity in this example equals to the concentration of salt (or the concentration of the second water component), the resulting units are as expected. In general, if all the coefficients b_i , $i = 1, \dots, N_w$ are zero, then the resulting effective salinity will have these units.
- On the other hand in Example 2 above, the resulting effective salinity is, according to Formula 1, dimensionless. This is consistent with the units of effective salinity in ECLIPSE 100 computed according to parameters specified in PLYESAL. In general, units of effective salinity computed using Formula 1 will be dimensionless, with exception of cases described earlier.

Limitations

The multi-component water model can be used in place of the single component water model in most standard simulations (black oil, EoS or K-value). However, it cannot be used at present in certain simulations where water has special properties. Therefore, options that cannot be used with the multi-component water model include:

- The Thermal option, where water is allowed in the gas phase (see "[Thermal option](#)").
- The GASWAT option, where water is modeled using an equation of state (see "[GASWAT option](#)").

- The GASSOL option, where compositional gas is allowed to dissolve in aqueous phase (see "[GASSOL option](#)").
- The CO2SOL option, where CO₂ dissolves in water (see "[CO₂/H₂S solution in the aqueous phase](#)").
- Satellite groups (see [GSATPROD](#)).
- [IMPSAT](#)
- Flux Boundary option (see "[Flux boundary conditions](#)").

In addition, there are currently some limitations when specifying water source terms:

- The water entering a multisegment well using the source / sink term ([WSEGEXSS](#)) is made up entirely of the last water component.

Summary of keywords

RUNSPEC

Keyword	Description
COMPW	Enables the water component model and specifies the number of water components

PROPS

Keyword	Description
CREFW	Compressibility of water components at the reference pressure
CREFWS	Compressibility of water components for surface calculations
CWTYPE	Water component type
DREFW	Density of water components at the reference pressure
DREFWS	Density of water components for surface calculations
MW	Molecular weight of water components
MWWS	Molecular weight of water components for surface calculations
PREFW	Reference pressure for water component density and viscosity calculations
PREFWS	Reference pressure for water component density calculations at surface conditions
VREFW	Viscosity and viscosiblity for water components at the reference pressure
WNAMES	Water component names

SOLUTION

Keyword	Description
AQSTREAW	Sets composition of water analytic aquifer (can also be used in the SCHEDULE section)
WI	Specifies overall initial water composition (can also be used in the PROPS section)
WMF	Specifies initial water composition explicitly

Keyword	Description
WMFVD	Specifies initial water composition with respect to depth (can also be used in the PROPS section)

SUMMARY

The SUMMARY keywords that can be used to obtain water component production and injection data are displayed below.

See the following in the *ECLIPSE Reference Manual*..

- [Analytic aquifer quantities](#)
- ["Format of SUMMARY keywords with a component index"](#)

Field	Group	Well	Connection	Description
FAMF	GAMF	WAMF	CAMF	Water component mole fractions
FWMR	GWMR	WWMR		Water component molar production rate
FWMPT	GWMPT	WWMPT		Water component molar production total
FWMIR	GWMIR	WWMIR		Water component molar injection rate
FWMIT	GWMIT	WWMIT		Water component molar injection total
FCWM				Component in water phase
			CKFR	Component flow rate
			CKFT	Component flow total

Block	Region	Segment	Description
BMLSC			Component moles / reservoir volume
BAMF			Water component mole fractions
	RKFR		Component inter-region flow rate
	RKFT		Component inter-region flow total
BCFRI			Component inter-block flow in the I-direction
BCFRJ			Component inter-block flow in the J-direction
BCFRK			Component inter-block flow in the K-direction
	SKFR		Segment component flow rate

For water component summary keywords, the water component index starts at $N_c + 1$ and ends at $N_c + N_w$ where N_c is the number of components given by [COMPS](#) and N_w is the number of water components given by the [COMPW](#) keyword.

SCHEDULE

Keyword	Description
WELLSTRW	Sets composition of the injection water stream

Keyword	Description
WINJW	Specifies the nature of the injection water

OUTPUT keywords

The arguments AMF and MWAT can be used with the keywords [RPTRST](#), [RPTSCHED](#) and [RPTSOL](#) to output water mole fraction and molar concentrations.

For summary block quantities, such as BMLSC as the water component index starts at $N_c + 1$ and ends at $N_c + N_w$ where N_c is the number of components given by [COMPS](#) and N_w is the number of water components given by the [COMPW](#) keyword.

Examples

The following data sets come with the ECLIPSE installation:

File	Description
MCW0.DATA	Brine model (using tracers, for comparison)
MCW1.DATA	Brine model
MCW2.DATA	MSW Brine model
MCW3.DATA	Brine + solids + reactions
MCW.GRF	GRF to compare runs

Table 77.2: Multi-component water example files

78

Wellbore friction option

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 multilateral wells, in which frictional pressure losses may be significant over the horizontal section of the wellbore 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 wellbore 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

$$\delta P_f = 2f \cdot \frac{L}{D} \cdot \rho \cdot v^2 \quad \text{Eq. 78.1}$$

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 m³/day) and taking into account the unit conversion factors, the pressure drop becomes:

$$\delta P_f = \frac{C_f \cdot f \cdot L \cdot \rho \cdot Q^2}{D^5} \quad \text{Eq. 78.2}$$

where values and units for the equation terms are:

C_f values 4.343E-15 (METRIC),

2.956E-12 (FIELD),

2.469E-13 (LAB)

L units m (METRIC), ft. (FIELD), cm (LAB)

D units m (METRIC), ft. (FIELD), cm (LAB)

ρ units kg/m³ (METRIC), lb/ft.³ (FIELD), gm/cc (LAB)

Q units rm³/day (METRIC), rb/day (FIELD), rcc/hr (LAB)

The Fanning friction factor depends upon the Reynolds number Re . For laminar flow ($Re < 2000$),

$$f = \frac{16}{Re} \quad \text{Eq. 78.3}$$

For $Re > 4000$ we use Haaland's formula ([Ref. 44]),

$$\sqrt{\frac{1}{f}} = -3.6 \log_{10}\left(\frac{6.9}{Re} + \left(\frac{e}{3.7D}\right)^{10/9}\right) \quad \text{Eq. 78.4}$$

where

e is the absolute roughness of the tubing in the same units as D

In the "uncertain region" ($2000 < Re < 4000$) we use a linear interpolation between the values at $Re = 2000$ and $Re = 4000$. The above scheme has the advantage of providing a direct calculation for values of f that are continuous in Re and are in reasonable agreement with the friction factor chart (see [Ref. 125], 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

$$Re = \frac{\rho v D}{\mu} \quad \text{Eq. 78.5}$$

where

μ is the viscosity of the fluid.

Converting v to the volumetric flow rate Q and including the unit conversion factors, this becomes

$$\text{Re} = \frac{C_r \rho Q}{D \mu} \quad \text{Eq. 78.6}$$

where values and units for the equation terms are:

C_r values 0.01474 (METRIC),

0.1231 (FIELD),

0.03537 (LAB)

ρ units kg/m³ (METRIC), lb/ft.³ (FIELD), gm/cc (LAB)

Q units rm³/day (METRIC), rb/day (FIELD), rcc/hr (LAB)

D units m (METRIC), ft. (FIELD), cm (LAB)

μ units cP (METRIC), cP (FIELD), cP (LAB)

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 multisegment wells or the polymer, brine, miscible, solvent or Gi model options in ECLIPSE 100.

Multilateral 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 multilateral 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 78.1, we would enter

```
FRICTION
 1  2  /
```

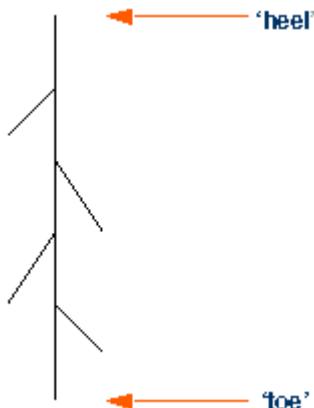


Figure 78.1. A trunk and branch multilateral well

For the “bird's foot” type well shown in figure 78.2, we would enter

```
FRICTION
 1  3  /
```

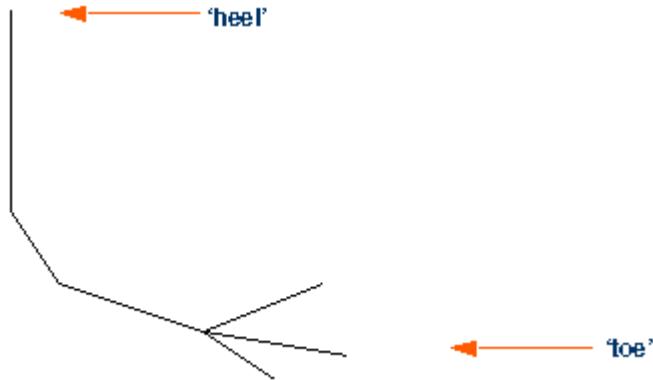


Figure 78.2. A bird's foot multilateral well

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 multilateral 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 (that is 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 entered again 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 multilateral well geometry is defined, consider the well shown in figure 78.3. 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:

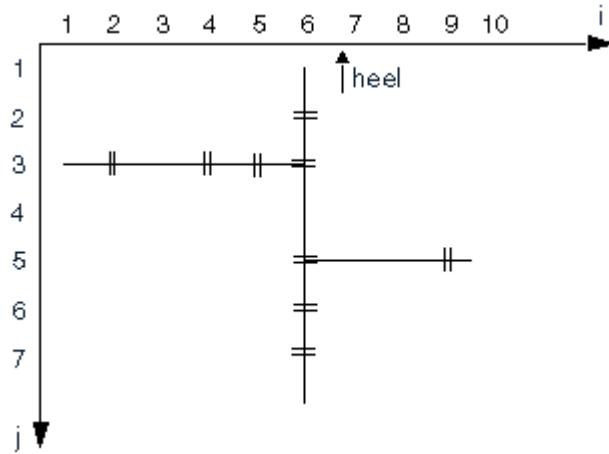


Figure 78.3. Defining multilateral well geometry

```

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 [COMPDLAT](#) 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 multilateral well geometry is flexible enough to allow branches to have sub-branches. As a second example, consider the well shown in the following figure.

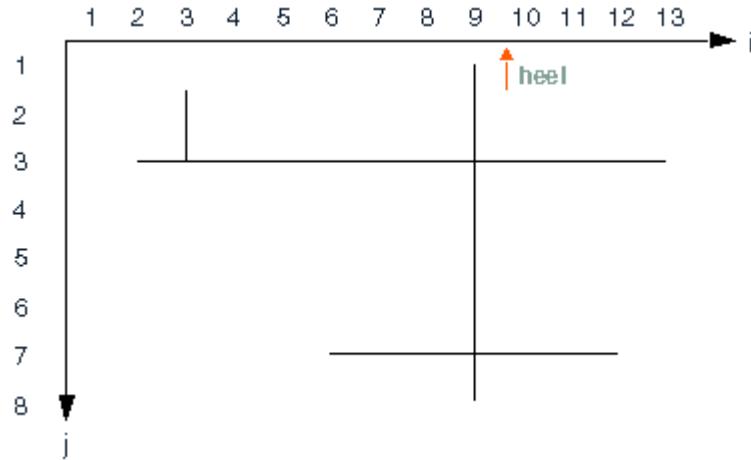


Figure 78.4. A multilateral well with sub-branches

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                           /

```

```

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 multilateral 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 [item 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 timestep, it is very unlikely that the Newton method will need damping. Now the use of item 55 in OPTIONS 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 multilateral 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 78.5 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.

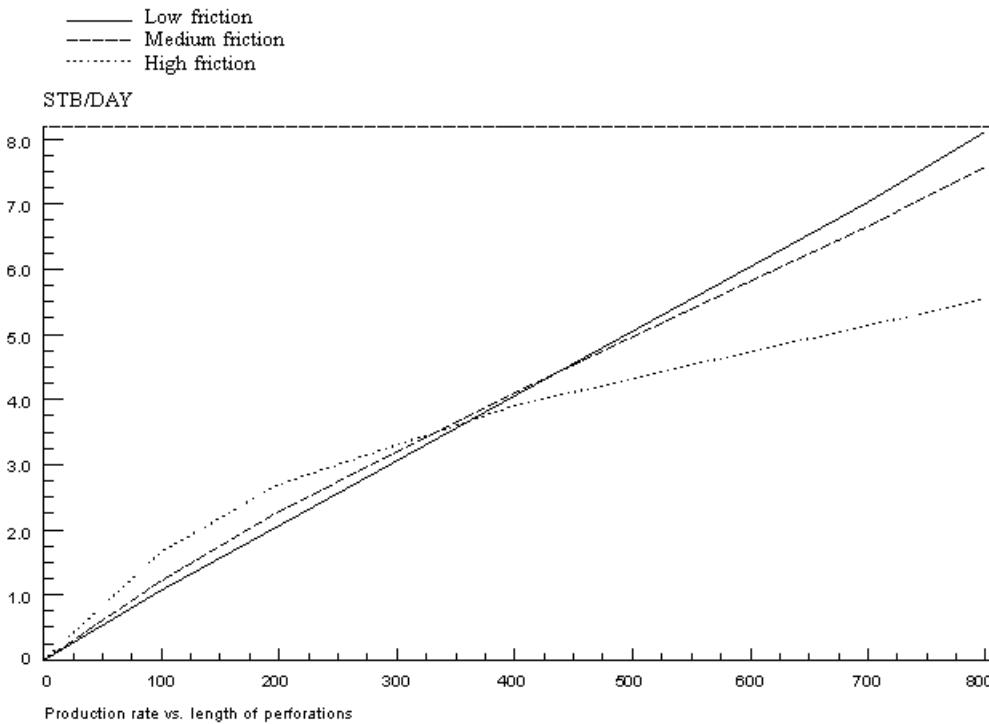


Figure 78.5. Friction effects in a horizontal well

The GRAF program has been adapted to produce graphs of this type. To make one of these graphs, carry out the following steps:

1. In the SUMMARY section of the ECLIPSE DATA file, make sure you have included the flow rates of all the connections in the horizontal well. For example, to plot the oil flow rate from a horizontal well called HORIZ with 6 connections use:

```
SUMMARY
COFR
'HORIZ' 3 5 6 /
'HORIZ' 4 5 6 /
'HORIZ' 5 5 6 /
'HORIZ' 6 5 6 /
'HORIZ' 7 5 6 /
'HORIZ' 8 5 6 /
/
```

The position of each connection (that is the length down the tubing from the bottom hole reference point) is automatically written to the SUMMARY file.

2. When the SUMMARY files produced from the run are loaded into GRAF, you are prompted:

```
The summary files contain Horizontal Well data
Do you wish to process this data?
Y/N (RETURN for N)
```

- a. 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)
- b. 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)
- c. 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, or A to prepare data for all types of plot
- Does data include multilateral wells? Y/N
(RETURN for N)
- d. Reply Y if multilateral wells are present in the model (see "[Multilateral friction well plots in GRAF](#)" for more information), or 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)
- e. 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; or
Q to abandon processing the horizontal well data.

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, ..., for timesteps 0, 1, 2, ... (up to 999). This is essentially the sum of the connection flows from the start of the perforations down to each connection.

3. To generate a horizontal well plot showing the conditions at timestep 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 timestep 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 timestep number. The corresponding length vector ends with _L. For example, to plot the connection pressure (CPR) at the second timestep versus length from the bottom hole reference point:

- b. 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 timestep 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 timestep is stored in the vectors LWOFR0, LWOFR1, ..., up to LWOFR999.

- 4. To generate a horizontal well plot showing the conditions at global timestep 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 timestep 2 versus length would have x-axis = LWCPR_L and y-axis = LWCPR2.

Multilateral friction well plots in GRAF

The horizontal well option in GRAF has been extended to allow you to split a multilateral 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 multilateral 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 WELL_A_3 for the third branch of WELL_A.

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 i11 jj1 kk1
  ....
LOADOPT HWELL www2
```

Use of multilateral 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.

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, for example COFRU, 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 CORFU 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

The keywords are:

- [FRICTION](#) Enables the Wellbore Friction option and sets array dimensions.

SUMMARY

Refer to [Wellbore friction model](#) in the *ECLIPSE Reference Manual*.

SCHEDULE

Keyword	Description
OPTIONS	Switch 55 may be used to damp the Newton iterations if well convergence problems arise.
WFRICTN	Designates a well as a friction well, and defines its tubing properties and the position in the tubing of the well connections with the grid cells.
WFRICTNL	Performs the same task as WFRICTN, but should be used if the well is situated in a local grid.

Example

This dataset describes the 7th SPE Comparative Solution Project [Ref. 76] - 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    /

```

```

FRICITION
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      Viscosity
  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  /

```

```
RPTSQL
'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 =====
RPTSCHE
'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

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:

$$q_{p,j} = T_{wj} M_{p,j} (P_j - P_w - H_{wj}) \quad \text{Eq. 79.1}$$

where

$q_{p,j}$ is the volumetric flow rate of phase p in connection j at stock tank conditions. The flow is taken as positive from the formation into the well, and negative from the well into the formation.

T_{wj} is the connection transmissibility factor, defined below.

$M_{p,j}$ is the phase mobility at the connection.

P_j is the nodal pressure in the grid block containing the connection.

P_w is the bottom hole pressure of the well.

H_{wj} is the well bore pressure head between the connection and the well’s bottom hole datum depth.

$P_w + H_{wj}$ is thus the pressure in the well at the connection j , which we call the “connection pressure”.

The calculation of the terms in this relationship is described in the following sections of this chapter.

Inflow performance relationship in ECLIPSE 300

In ECLIPSE 300 the inflow performance relationship is written in terms of the volumetric production rate of each phase at reservoir conditions:

$$\hat{q}_{p,j} = T_{w,j} \left(\frac{K_{rp,j}}{\mu_{p,j}} \right) (P_j - P_w - H_{wj}) \quad \text{Eq. 79.2}$$

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

$$q_{p,j} = \hat{q}_{p,j} b_{p,j} \quad \text{Eq. 79.3}$$

where $b_{p,j} = \frac{1}{B_{p,j}}$ and $B_{p,j}$ is the phase formation volume factor.

The molar rate is obtained by multiplying by the phase reservoir molar densities, $b_{o,j}^m$. The molar rate of component i is then:

$$q_{i,j} = x_{i,j} b_{o,j}^m q_{o,j} + y_{i,j} b_{g,j}^m q_{g,j} \quad \text{Eq. 79.4}$$

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

$$T_{w,j} = \frac{c\theta K h}{\ln(r_o/r_w) + S} \quad \text{Eq. 79.5}$$

where

- c is a unit conversion factor (0.001127 in field units, 0.008527 in metric units, 3.6 in lab units).
- θ is the angle of the segment connecting with the well, in radians. In a Cartesian grid its value is 6.2832 ($= 2\pi$), as the connection is assumed to be in the center of the grid block. For wells located on an edge (or a corner) of a Cartesian grid, for example in a sector model or symmetry element, use keyword **WPIMULT** after **COMPDAT** to scale the resulting connection factors by 0.5 (or 0.25).
- Kh is the effective permeability times net thickness of the connection. For a vertical well the permeability used here is the geometric mean of the x- and y-direction permeabilities, $K = (K_x K_y)^{1/2}$.
- r_o is the “pressure equivalent radius” of the grid block, defined below.
- r_w is the well bore radius.
- S is the skin factor.

The pressure equivalent radius of the grid block is defined as the distance from the well at which the local pressure is equal to the nodal average pressure of the block. In a Cartesian grid we use Peaceman’s formula, which is applicable to rectangular grid blocks in which the permeability may be anisotropic. The well is assumed to penetrate the full thickness of the block, through its center, perpendicularly to two of its faces.

$$r_o = 0.28 \frac{\left[D_x^2 \left(\frac{K_y}{K_x} \right)^{1/2} + D_y^2 \left(\frac{K_x}{K_y} \right)^{1/2} \right]^{1/2}}{\left(\frac{K_y}{K_x} \right)^{1/4} + \left(\frac{K_x}{K_y} \right)^{1/4}} \quad \text{Eq. 79.6}$$

where

D_x and D_y are the x- and y- dimensions of the grid block

K_x and K_y are the x- and y- direction permeabilities.

Equations 79.5 and 79.6 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 equation 79.6, and in equation 79.5 the expression $Kh = D_x(K_y K_z)^{1/2}$ will be used.

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 innermost grid blocks ($i = 1$), it is assumed to lie on the inner boundary of the block. The connection factor is

$$T_{wj} = \frac{c\theta Kh}{\frac{r_2^2}{r_2^2 - r_w^2} \ln(r_2/r_w) - 0.5 + S} \quad \text{Eq. 79.7}$$

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 outermost blocks ($i = \text{NDIVIX}$), it is assumed to occupy the outer boundary of the block for the purpose of applying boundary conditions. The corresponding connection factor is

$$T_{wj} = \frac{c\theta Kh}{\frac{r_1^2}{r_2^2 - r_1^2} \ln(r_1/r_2) + 0.5 + S} \quad \text{Eq. 79.8}$$

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 ($1 < i < \text{NDIVIX}$), its connection factor is calculated as if it were in a Cartesian grid block with $D_x = r_2 - r_1$ and $D_y = \theta(r_1 + r_2)/2$

Fractured completions

ECLIPSE 300

When using one of the built-in formulae above to calculate the connection factor, the engineer can supply a constant value for the skin factor S , to represent the effects of, for example, formation damage or stimulation, partial penetration and well bore deviation. 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

$$T_{wj} = \frac{8cK_y D_x D_z}{D_y + SD_x / \pi} \quad \text{Eq. 79.9}$$

In the case of FY, the completion connection factor is

$$T_{wj} = \frac{8cK_x D_y D_z}{D_x + SD_y / \pi} \quad \text{Eq. 79.10}$$

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 $2D_x D_z$ or $2D_y D_z$.

Rock crushing modifications

If a transmissibility multiplier is specified with the [ROCKTAB](#) or [ROCKTABH](#) keywords, the well connection transmissibility factor, grid block transmissibilities and NNCs are modified accordingly.

ECLIPSE 300

ECLIPSE 300 also updates the generalized mobility terms for grid cells and well connections. Similar functionality, but based on the corresponding changes in porosity, is also available by specifying the keywords [ROCKTRMX](#), [ROCKTRMY](#) and [ROCKTRMZ](#).

If the ROCKTAB or ROCKTABH table contains directional transmissibility multipliers, as specified in keyword [RKTRMDIR](#), the well connection transmissibility factor is modified according to the formula

$$\tilde{T}_{wj} = T_{wj} (m_x m_y)^{1/2} \quad \text{Eq. 79.11}$$

where

\tilde{T}_{wj} is the pressure-adjusted connection factor,

m_x , m_y are the transmissibility multipliers interpolated from the rock table at the grid block pressure P_j .

The formula assumes a vertically completed well within the grid block j . For other well orientations, permute the directions accordingly.

In Cartesian grids, this transmissibility update is in accordance with modifying the effective Kh and with keeping the Peaceman's formula term $\ln(r_o / r_w)$ unaffected.

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, for example, formation damage or stimulation, partial penetration and well bore deviation. In addition, a flow-dependent contribution can be included to model non-Darcy effects in gas flow near the well,

$$S \rightarrow S + D | q_{fg} | \quad \text{Eq. 79.12}$$

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 equation 79.12, 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

$$D_j = D_w \frac{i}{T_{wj}} \quad \text{Eq. 79.13}$$

where

D_j is the scaled D-factor for the connection j ,

D_w is the measured D-factor for the well, and the

$\sum T_{wi}$ term denotes the sum of the connection transmissibility factors for all the connections in the well i 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

$$D_m = \frac{G}{G + D|q_{fg}|} \quad \text{Eq. 79.14}$$

where G is a geometric factor:

For the Cartesian case:

$$G = \ln\left(\frac{r_o}{r_w}\right) + S \quad \text{Eq. 79.15}$$

For the inner cell radial case:

$$G = \frac{r_2^2}{r_2^2 - r_w^2} \ln\left(\frac{r_2}{r_w}\right) - 0.5 + S \quad \text{Eq. 79.16}$$

For the outer cell radial case:

$$G = \frac{r_1^2}{r_2^2 - r_1^2} \ln\left(\frac{r_1}{r_2}\right) + 0.5 + S \quad \text{Eq. 79.17}$$

For the fractured completion cases:

- Fracture in x-direction:

$$G = \pi \frac{A_y}{A_x} + S \quad \text{Eq. 79.18}$$

- Fracture in y-direction:

$$G = \pi \frac{A_x}{A_y} + S \quad \text{Eq. 79.19}$$

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

$$q_{fg} = T_w D_m M_{fg} \Delta P \quad \text{Eq. 79.20}$$

where ΔP is the drawdown into the connection.

Substituting equation 79.14 for D_m gives the quadratic equation

$$T_w M_{fg} \Delta P = q_{fg} + \left(\frac{D}{G} \right) q_{fg} |q_{fg}| \quad \text{Eq. 79.21}$$

that may readily be solved for $|q_{fg}|$, 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 equation 79.1 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

$$M_{fp,j} = k_{p,j} \lambda_{p,j} \quad \text{Eq. 79.22}$$

where

$k_{p,j}$ is the relative permeability of the phase,

$\lambda_{p,j}$ is defined by

$$\lambda_{p,j} = \frac{1}{B_{p,j} \mu_{p,j}} \quad \text{Eq. 79.23}$$

$\mu_{p,j}$ is the phase viscosity, and

$B_{p,j}$ is the phase formation volume factor.

The quantities in equation 79.22 and 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

$$\begin{aligned} M_{o,j} &= M_{fo,j} + R_{v,j} M_{fg,j} \\ M_{g,j} &= M_{fg,j} + R_{s,j} M_{fo,j} \end{aligned} \quad \text{Eq. 79.24}$$

If a flow-dependent skin factor is used, the free gas mobility is multiplied by the factor D_m defined in equation 79.14.

$$M_{fg,j} \rightarrow D_m M_{fg,j} \quad \text{Eq. 79.25}$$

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. Item 8 of keyword [WELSPECS](#) selects the method to use for each well.

Russell-Goodrich

ECLIPSE 100

The Russell-Goodrich equation [Ref. 138] 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 equation 79.1 if the term $\lambda_{g,j}$ defined in 79.23 is evaluated at the average of the grid block pressure and the well bore pressure at the connection, that is $(P_j + P_w + H_{wj})/2$. The relative permeability is evaluated at the grid block conditions.

Dry gas pseudo-pressure

ECLIPSE 100

In the pseudo-pressure equation, the gas flow rate varies as the difference between the “real gas pseudo-pressure” values at the grid block and well bore pressures. This also can be written in the form of equation 79.1 by setting $\lambda_{g,j}$ equal to its integrated average value between the pressures P_j and $P_w + H_{wj}$. This representation avoids the need to calculate tables of pseudo-pressure vs. pressure. The integration is performed by applying the trapezium rule between each intermediate pair of nodal pressure values in the gas PVT table. This essentially treats λ_g as a linear function of P between adjacent nodal pressure values. The relative permeability is evaluated at the grid block conditions.

At low values of the drawdown the Russell-Goodrich and pseudo-pressure equations will give approximately the same results. When the drawdown is large enough to span a significantly non-linear segment of the graph of λ_g versus P , the pseudo-pressure equation will give more accurate results than the Russell-Goodrich equation.

This option is not available in gas condensate runs. In these cases the generalized pseudo-pressure method may be used instead.

Generalized pseudo-pressure in ECLIPSE 100

ECLIPSE 100

The generalized pseudo-pressure method is intended for use by gas condensate producers. It provides a means of taking account of condensate dropout, as well as compressibility, in the calculation of the mobility integral. It is based on the method described by Whitson and Fevang [Ref. 127].

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,

$$\beta_j = \frac{\int_{(P_w + H_{wj})}^{P_j} (M_g + M_o) dp}{(M_{g,j} + M_{o,j})(P_j - (P_w + H_{wj}))} \quad \text{Eq. 79.26}$$

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:

$$M_o / M_g = M_{o,j} / M_{g,j} \quad \text{Eq. 79.27}$$

This requirement assumes that within the grid block the flows are in steady state. Only the mobile oil is included when calculating this ratio. Immobile dropped-out oil is not included because the oil relative permeability will be zero until the oil saturation exceeds the critical oil saturation.

The integral in equation 79.26 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 79.27 using Newton's method.

The blocking factor is then used to multiply both the oil and gas mobilities in the inflow performance relationship, equation 79.1. 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,

$$\beta_{\text{used}} = f\beta_{\text{new}} + (1-f)\beta_{\text{previous}} \quad \text{Eq. 79.28}$$

where the weighting factor f is specified in the **PICOND** keyword.

Generalized pseudo-pressure in ECLIPSE 300

ECLIPSE 300

The generalized pseudo-pressure method of Whitson and Fevang [Ref. 126] 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 timestepping 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:

$$v_j = \frac{Q_j}{2\pi r h} = \frac{K K_{rj}}{\mu_j} \frac{dP}{dr} \quad \text{Eq. 79.29}$$

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:

$$Q_j = n_j V_{mj} \quad \text{Eq. 79.30}$$

where V_{mj} is phase molar volume.

Combining equations 79.29 and 79.30 and rearranging gives:

$$n_j \frac{dr}{r} = 2\pi K h \frac{K_{rj} b_j}{\mu_j} dp \quad \text{Eq. 79.31}$$

where $b_j = 1 / V_{mj}$, the phase molar density.

For component, i , the component phase molar rate will be:

$$n_{ij} \frac{dr}{r} = 2\pi K h x_{ij} \frac{K_{rj} b_j}{\mu_j} dp \quad \text{Eq. 79.32}$$

where x_{ij} is the mole fraction of the i^{th} component in the j^{th} phase.

Integrating equation 79.32 between the Peaceman pressure equivalent radius r_B and the well radius r_W gives:

$$n_{ij} \int_{r_W}^{r_B} \frac{dr}{r} = 2\pi K h \int_{p_W}^{p_B} x_{ij} \frac{K_{rj} b_j}{\mu_j} dp \quad \text{Eq. 79.33}$$

Summing component phase molar rates in the oil and gas phases for component i :

$$(n_{io} + n_{ig}) \left[\ln \left(\frac{r_B}{r_W} \right) + S \right] = 2\pi K h \int_{p_W}^{p_B} \left[x_i \frac{K_{ro} b_o}{\mu_o} + y_i \frac{K_{rg} b_g}{\mu_g} \right] dp \quad \text{Eq. 79.34}$$

or:

$$n_i = T \int_{p_W}^{p_B} M_i(p) dp \quad \text{Eq. 79.35}$$

where the well connection factor, T , including the skin factor, S , is given by:

$$T = \frac{2\pi K h}{\ln \left(\frac{r_B}{r_W} \right) + S} \quad \text{Eq. 79.36}$$

and the Component Generalized Molar Mobility (CGMM) is given by:

$$M_i = x_i \frac{K_{ro} b_o}{\mu_o} + y_i \frac{K_{rg} b_g}{\mu_g} \quad \text{Eq. 79.37}$$

Normally, we do not consider the pressure dependency in equation 79.35. Instead, we compute the CGMM at the block pressure:

$$\int_{p_W}^{p_B} M_i(p) dp \rightarrow M_i(p_B) [p_B - p_W] \quad \text{Eq. 79.38}$$

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 equation 79.34.

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:

$$z_{pi} = \frac{M_i}{M_T} \quad \text{Eq. 79.39}$$

where:

$$M_T = \sum_i M_i \quad \text{Eq. 79.40}$$

Therefore, equation 79.35 can be written:

$$n_i = T z_{pi} \int_{p_W}^{p_B} M_T(p) dp \quad \text{Eq. 79.41}$$

where:

$$M_T = \frac{K_{ro} b_o}{\mu_o} + \frac{K_{rg} b_g}{\mu_g} \quad \text{Eq. 79.42}$$

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 rewriting 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:

$$n_i = T F_{Bi} M_i(p) [p_B - p_W] \quad \text{Eq. 79.43}$$

where:

$$F_{Bi} = \frac{1}{M_i(p_B)} \frac{1}{(p_B - p_W)} z_{pi} \int_{p_W}^{p_B} M_T(p) dp \quad \text{Eq. 79.44}$$

Because of equation 79.39, we can write:

$$M_i(p_B) = z_{pi} M_T(p_B) \quad \text{Eq. 79.45}$$

therefore, we define the total dimensionless flow blocking factor, F_B , (DFBF)

$$F_B = \frac{1}{M_T(p_B)} \frac{1}{(p_B - p_W)} \int_{p_W}^{p_B} M_T(p) dp \quad \text{Eq. 79.46}$$

and hence the modified inflow equation becomes:

$$n_i = T F_B M_i(p) [p_B - p_W] \quad \text{Eq. 79.47}$$

Note: Item 121 of the OPTIONS3 keyword can be used to impose an upper limit (greater than or equal to unity) on the value of the dimensionless blocking factor in order to prevent convergence issues in the simulation.

The final form of the GPP inflow equation, 79.47, differs only from the normal inflow equation by the additional multiplication factor, F_B , which we calculate from equation 79.46. 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 equation 79.46 appears to be a function of two variables:

$$Q(p_B, p_W) = \int_{p_W}^{p_B} M_T(p) dp \quad \text{Eq. 79.48}$$

but in fact it can be constructed as the difference of two integrals, namely:

$$Q(p_B, p_W) = R(p_B) - R(p_W) = \int_{p_L}^{p_B} M_T(p) dp - \int_{p_L}^{p_W} M_T(p) dp \quad \text{Eq. 79.49}$$

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:

$$\frac{M_o(S_o)}{M_g(S_g)} = \frac{L}{V} \quad \text{Eq. 79.50}$$

where

L and V are the total liquid and vapor moles at the pressure $p_L \leq p \leq p_B$ 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:

$$M_j = \frac{K_{rj} b_j}{\mu_j} \quad \text{Eq. 79.51}$$

Equation 79.50 is a simple non-linear equation in one variable, the oil saturation. The single integral in equation 79.49 is constructed by trapezium rule, following the suggestions made by Whitson and Fevang [Ref. 126]. In practical terms, the integral is constructed at the beginning of each timestep, only if the block conditions have changed significantly.

Note: As described on page 107 of [Ref. 127], and also illustrated in [Ref. 99], an over-coarse grid will lead to an underestimate of well deliverability. It is important that the grid size containing the well should be small enough to model the region 1 steady-state flow using GPP. This will be of the order of 50-100 m for lean gas condensate and 100-200 m for rich gas condensate reservoirs, although sensitivity studies should be carried out to determine the required well grid.

Note: If the rock compaction option has been specified via the `ROCKCOMP` keyword, the values of oil and gas relative permeability used in the above equations will be modified by the transmissibility multipliers specified via the `ROCKTAB` keyword. This can be disabled by setting item 299 of the `OPTIONS3` keyword to 1.

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, equation 79.46 and the phase molar mobility, M_j , equation 79.51 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:

$$\frac{\ln r - \ln r_w}{\ln r_o - \ln r_w} = \frac{m(p) - m(p_w)}{m(p_B) - m(p_w)} \quad \text{Eq. 79.52}$$

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](#)". The 'pipe-equivalent' velocity is calculated from:

$$v_g = \frac{n_g}{2\pi rh\phi(1-S_{wc})b_g} \quad \text{Eq. 79.53}$$

The Capillary Number relative permeabilities may be further modified by a turbulent, Forchheimer effect. A description of the models available for the Forchheimer parameter is presented in "[Forchheimer models](#)". This effect is modeled as a non-Darcy flow factor as described in "[Non-Darcy flow](#)".

Note: If a cell containing a well connection drops below dew point, and velocity dependent effects are strong, rapid variations in PI are occasionally observed. These effects are caused by gas condensate effects being modeled by both the GPP algorithm and the grid block discretization simultaneously. If the connected cell is coarse enough that all of regions 1 and 2 (as described in [\[Ref. 126\]](#) and [\[Ref. 127\]](#)) are contained within the pressure-equivalent radius then these anomalies will be eliminated.

Injecting connections

In injecting connections, an alternative expression for the mobility is required to reflect the upstream saturation conditions in the well bore. In the absence of well bore crossflow, there would be single-phase conditions at all injecting connections. In these circumstances it is standard practice to replace the term $k_{p,j} / \mu_{p,j}$ in equation [79.22](#) and with its sum over all phases, to give in ECLIPSE 100

$$M_{p,j} = \frac{\frac{k_{o,j}}{\mu_{o,j}} + \frac{k_{w,j}}{\mu_{w,j}} + \frac{k_{g,j}}{\mu_{g,j}}}{B_{p,j}} \quad \text{Eq. 79.54}$$

and in ECLIPSE 300

$$M_{p,j} = \frac{k_{o,j}}{\mu_{o,j}} + \frac{k_{w,j}}{\mu_{w,j}} + \frac{k_{g,j}}{\mu_{g,j}} \quad \text{Eq. 79.55}$$

where p here refers to the injected phase. The connection injectivity is thus a function of the total fluid mobility in the grid block. In ECLIPSE 300, the fluid in the wellbore is flashed at the bottom hole pressure to determine the wellbore fluid molar density. For each connection, this wellbore molar density is used with the mobility in equation [79.55](#), the pressure drawdown and the connection factor to form connection molar injection rates. (These are summed to give the total molar injection rate of the well, which is multiplied by a surface volume per mole factor to give the reported injection volume rate.) The ECLIPSE 300 calculation thus relies on quantities at both wellbore conditions (fluid molar density) and grid block conditions (equation [79.55](#)) to determine the injection flow for a given drawdown. ECLIPSE 100, in contrast, only relies on quantities at the grid block conditions (equation [79.54](#)). For gas injectors with very large drawdowns, the difference between the wellbore and grid cell pressures will result in small differences in gas injection flow rates between the simulators for BHP controlled wells.

If gas or water is injected into a grid block initially containing oil, this relationship will cause the well's injectivity to vary until the grid block is completely flooded out with the injected phase. In reality however, most of the pressure drop occurs over a region close to the well, and when this region has flooded out the injectivity will stay approximately constant. If the grid block size is much larger than this region, the calculated injectivity will be incorrect until the whole grid block has flooded out. For cases like this, an alternative approach is available that allows the user to supply a constant value of the injected phase relative permeabilities. In ECLIPSE 100 the keyword [COMPINJK](#) can supply a single relative permeability of the injected phase for non-crossflowing injectors. In ECLIPSE 300 the keywords [COMPKRI](#) and [COMPKRIL](#) supply relative permeabilities for all three phases, which can be applied to injectors regardless of their crossflow status; alternatively [COMPMOBI](#) and [COMPMBIL](#) can supply a fixed value for the injection mobility.

ECLIPSE 100

If crossflow occurs through the well bore, or if the well is a multiphase injector, it is possible for the well to inject a mixture of phases into the formation through one or more connections. The injection mobilities must be adapted to represent the fluid mixture present in the well bore. ECLIPSE 100 determines the proportion of each phase present in the mixture entering the well bore, and ensures that the phases flow in the same proportions through all the injecting connections. The treatment of crossflow is thus based on the assumption that the mixture of phases is uniform throughout the well bore at formation level. To determine the injection mobilities, a set of intermediate quantities is first calculated, which we call the "well volume factors". The well volume factor α_p of a given phase p is defined to be the stock tank volume of the phase that is contained within a unit volume of the well bore at formation level. Its value depends on the well's bottom hole pressure and the mixture of phases entering the well bore. The injection mobilities are set in proportion to the well volume factors. Assuming that the total injectivity of the connection is still proportional to the total fluid mobility within the grid block, the injection mobilities are given by

$$M_{p,j} = \alpha_p \left[\frac{\frac{k_{o,j}}{\mu_{o,j}} + \frac{k_{w,j}}{\mu_{w,j}} + \frac{k_{g,j}}{\mu_{g,j}}}{\alpha_{fo} B_{o,j} + \alpha_w B_{w,j} + \alpha_{fg} B_{g,j}} \right] \quad \text{Eq. 79.56}$$

The expression reduces to the standard injection mobility, equation 79.54, 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 equation 79.14. 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 equation 79.54 or equation 79.56 is replaced by the term $k_{g,j}\lambda_{g,j}B_{g,j}$, where $\lambda_{g,j}$ is evaluated in the form appropriate to each equation, as described above for producing connections.

High mobility injection wells

ECLIPSE 300

It is known that injector wells can cause convergence problems when the mobilities are high in the connecting grid cells. Specially in thermal simulation this might be the case when steam is injected at high pressure resulting in fracturing near the well. The high mobility injection well use the expression

$$\hat{q}_{p,j} = T_{wj} M_{pj} \left(\frac{\sum_i M_{pi}(P_i - H_{wi})}{\sum_i M_{pi}} - P_w \right)$$

for the volume injected by a completion j . The mobilities are the total mobilities as given above. See equation 79.1 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 timesteps.

The wellbore head term

The wellbore head term H_{wj} is the pressure difference in the wellbore 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,

$$\bar{\rho} = \rho_{o,surf}\alpha_o + \rho_{w,surf}\alpha_w + \rho_{g,surf}\alpha_g \quad \text{Eq. 79.57}$$

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 GOR. 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 equation 79.57 would predict a lower density corresponding to a mixture of higher GOR. 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,

$$\rho_{\text{local}} = \frac{\rho_{o,surf}Q_o + \rho_{w,surf}Q_w + \rho_{g,surf}Q_g}{Q_{fo}B_o + Q_wB_w + Q_{fg}B_g} \quad \text{Eq. 79.58}$$

Q here represents the accumulated flow through the upstream connections, and the formation volume factors B are evaluated at the average pressure in the wellbore 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 timestep. 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

$$J = \frac{Q_p}{P_d - P_w} \quad \text{Eq. 79.59}$$

where Q_p is the production rate of the chosen phase, and P_d is the pressure at the drainage radius. Equation 79.59 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:

$$J = \sum_j \left[T_{wj} M_{p,j} \left(\frac{\ln(r_o/r_w) + S}{\ln(r_d/r_w) + S} \right) \right] \quad \text{Eq. 79.60}$$

where \sum_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 equation 79.60.

The simulator's calculation of the productivity index should not be used for a horizontal well, as the assumptions used in equation 79.60 will not be valid. Equation 79.60 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 equation 79.59. 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 79.60 simply reduces to

$$J = \sum_j [T_{wj} M_{p,j}] \quad \text{Eq. 79.61}$$

and the PI should be regarded as the PI of the well within its connecting grid blocks rather than within its drainage region.

Additional forms of the well PI can be output to the SUMMARY file using the SUMMARY section keywords WPI1, WPI4, WPI5 and WPI9. These use equation 79.59 where P_d is replaced with a weighted average pressure within a 1-block, 4-block, 5-block and 9-block region around each connecting grid block. The weighting formula is specified with keywords WPAVE and WWPAVE

ECLIPSE 300

The connection phase mobilities for oil and gas must be modified in the compositional case to take into account phase changes for different components when combined into a single well stream. In this case the mobilities used in equation 79.60 are

$$\tilde{M}_{p,j} = \frac{O_{p,j}}{\sum_k O_{p,k}} M_{p,j} \quad \text{Eq. 79.62}$$

where $O_{p,j}$ is the volume flow rate of phase p per mole of hydrocarbon at connection j . Note that this means that the productivity index for oil and gas may differ between ECLIPSE 100 and ECLIPSE 300. The mobilities used for the water phase are unaffected.

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 keyword WPI.

Note: The PI of the preferred phase of a well is written to the SUMMARY file as a quantity without units in FIELD. 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 (Mscf/day/psi) and liquid (stb/day/psi). The alternative keywords WPIO, WPIG, WPIW and WPIL can be used to output the productivity index for specific phases and are printed with appropriate accompanying units.

80

Well modeling facilities

Well completions

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:

- the transmissibility between the grid block and the well bore
- the mobility of the phase in the grid block at the perforations
- the pressure drawdown between the grid block and the well bore.

These quantities are discussed in more detail in "Well inflow performance".

In certain situations the drawdown in one layer may have the opposite sign to the drawdown in the other completed layers. The simulator will 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 or component mixture entering the well bore. A special case in which crossflow may occur is when the well is plugged off above the formation while the completions remain open to the reservoir. In ECLIPSE 100 the crossflow facility can be turned off if you do not wish this to occur, using a switch in the keyword [WELSPECS](#).

Relative permeabilities in well connections

The relative permeability of a phase in a well connection is a function of the saturations in the grid block containing the connection. By default, the relative permeabilities are calculated using the Saturation Function tables associated with the grid block containing the connection. However, the engineer can supply additional relative permeability tables for calculating the fluid mobilities at the well connections. These tables could take into account the effects of coning and/or partial completion within the grid block, or they could be well **pseudo** tables derived from an earlier fine-grid simulation. The association between the relative permeability tables and the well connections is defined using the keyword [COMPDAT](#).

An alternative means of allowing for coning or partial completion within a grid block is to re-scale the Saturation Function tables to different end-point saturation values. The engineer can optionally supply alternative sets of end-point saturation values for use in particular well connections. These end-point values are used to transform the grid block saturations into the saturations ‘seen by’ the well connections. The alternative end-point saturation values are entered using the keyword [COMPRP](#).

ECLIPSE 100

In simulation runs which use the Vertical Equilibrium option, the keyword [COMPVE](#) should be used instead of [COMPRP](#), to supply depth values for the top and bottom of the perforated interval. The [COMPVE](#) keyword may be used to specify partial penetration data in non-VE runs also; ECLIPSE automatically translates the top and bottom depths of the perforations into [COMPRP](#) type scaled end-points.

Special facilities for gas wells

There are two facilities for improving the accuracy of modeling gas inflow. Firstly, the non-Darcy component of the pressure drawdown resulting from turbulence near the well bore is modeled by a flow-dependent contribution to the skin factor. The turbulence skin factor for a well connection is equal to the product of the free gas flow rate in the connection and the connection’s D-factor. Well D-factors are defined using the keyword [WDFAC](#) or they may be calculated from a correlation using the keyword [WDFACCOR](#).

The second facility is the option to use a special inflow equation to provide a more accurate model of the flow of gas into the well. ECLIPSE 100 offers a choice of three models: the Russell-Goodrich equation, the dry gas pseudo-pressure equation and the generalized pseudo-pressure method. These are all methods of taking into account the pressure-dependence of the gas properties between the grid block pressure and the well bore pressure. The generalized pseudo-pressure equation also takes account of the effects of condensate dropout. They are described in more detail in “[Special inflow equations](#)”. You can select the desired equation by setting a switch in the keyword [WELSPECS](#). The generalized pseudo-pressure equation is also available in ECLIPSE 300, where it may be activated for specific wells using the [WELSPECS](#) keyword, or it may be activated for all production wells using the keyword [PSEUPRES](#).

Scale deposition model

A scale deposition model is available, which accounts for the cumulative effects of scale deposited around the well connections and the resulting degradation of the productivity index.

The rate of scale deposition in each well connection is proportional to the flow rate of water into it and also varies with the amount of impurities dissolved in the water. Formation water and injected sea water contain different impurities and thus the overall rate of scale deposition is dependent on the relative proportion of sea water and formation water.

This scale deposition process is modeled by a table that defines the rate of scale deposition per unit water flow rate as a function of the fraction of sea water present in the water flowing into a connection (see

keyword [SCDPTAB](#)). The sea water fraction is represented by a passive water tracer configured by the user (see keyword [SCDPTRAC](#)).

A second scale damage table defines how the current amount of deposited scale alters the PI of the well connections (see keyword [SCDATAB](#)).

The calculation for the effect of scale damage on a well is performed explicitly. Thus, at the end of each timestep the mass of scale deposited around each connection is incremented according to the flow of water during the timestep and the current sea water concentration. This is translated into a PI reduction factor for each connection, that is used during the following timestep.

You may supply multiple versions of each of the two table types. The [WSCTAB](#) keyword must be supplied to select which tables are to be used by each well for the calculation.

The [WSCCLEAN](#) keyword may be used to reduce the amount of scale deposited around well connections after a scale cleanup operation. (The [WSCCLENL](#) keyword should be used for wells in local grid refinements.)

Well controls and limits

Individual wells can operate at a target value of any of the following quantities:

- Oil flow rate
- Water flow rate
- Gas flow rate
- Liquid flow rate (oil and water)
- Linearly combined rate (user-specified linear combination of oil, water and gas)
- Reservoir fluid volume (or voidage) rate

Note: Voidage rate is the reservoir volume that the produced or injected fluids would occupy at reservoir conditions. The associated reservoir pressure is usually the average hydrocarbon pressure in the field. However, this may be set to be the average pressure in a nominated fluid in place region using argument 13 in keyword [WELSPECS](#). By obtaining voidage rates at a single pressure for all wells we avoid the effects of higher pressures at injectors and lower pressures at producers; setting equal injection and production voidage rates will then yield approximate pressure maintenance.

- Bottom hole pressure
 - Tubing head pressure
 - Wet gas rate (which is the volume that the hydrocarbon mixture would occupy at standard conditions if it were an ideal gas)
- ECLIPSE 300 only*
- Total molar rate

Note: The total molar rate includes the molar rate of water.

- ECLIPSE 300 only*
- Calorific value rate.

One of these quantities is selected as the main control target, while limiting values can be supplied for any of the remaining quantities. The limiting flow rates are treated as upper limits, while the pressure limits are

treated as lower limits in production wells and upper limits in injectors. The well will automatically change its mode of control whenever the existing control mode would violate one of these limits. The target quantity in the old control mode will then become a limit in the new control mode.

For example, consider a production well operating at a target oil flow rate of 1000 stb/day, with a lower limit of 3000 psia on the bottom hole pressure. The well will produce 1000 stb/day of oil until the bottom hole pressure falls below 3000 psia. The well's control mode will then change automatically to maintain a constant bottom hole pressure of 3000 psia, and the oil production rate will decline. If subsequently the oil production rate were ever to exceed 1000 stb/day (resulting for example from stimulation of the well, or the opening of an extra connection), then the control mode will automatically switch back to the original target oil production rate.

In addition to the control quantities listed above, a maximum allowable drawdown can be specified for each production well, using the keyword [WELDRAW](#). This is converted into a maximum liquid or gas production rate at each timestep (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:

- Lower limits for the oil, gas, liquid and reservoir fluid volume 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](#)"), one of these connections will be opened.

- 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, that is 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 crossflow 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 [WELSPCS](#) 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 timestep, and the appropriate action is taken if any are violated. The action is taken at the end of the timestep during which the limit was violated. Thus a well may produce ‘uneconomically’ for one timestep 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 timestep will be repeated after the remedial action has been taken. Thus the action will take effect from the beginning of the timestep 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 timestep, and will be effective for the following timestep. If however the gas-oil ratio reaches above 12 Mscf/stb, the timestep will be repeated after the remedial action has been taken. The tolerance fraction is set with the keyword [WLIMTOL](#). Note that each timestep will be repeated only once. Thus if a well still violates an economic limit at the end of the repeated timestep, that timestep is ‘accepted’. Further remedial action will be applied to the well before starting the next timestep.

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 reopened. 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 restart 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 cutback of flow rate

There is an indirect way of controlling the following production ratios of individual wells subject to gas or water coning:

- Water cut
- Gas-oil ratio
- Gas-liquid ratio
- 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 timestep until the limit is no longer exceeded. The rate cutback is performed at the end of the timestep 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 timestep will be repeated with the well flowing at the reduced rate. The timestep will only be repeated once; if the limit is still exceeded at the end of the repeated timestep, the well's flow rate will be reduced again for the next timestep.

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 cutback limits, there is a corresponding limit that reverses the cutback process. For example, the gas-oil ratio could have a cutback limit of 5.0 Mscf/stb and a cutback 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 cutback process. Cutback 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 cutback pressure limit is a maximum value, and the cutback 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 timestep, rather than the full rates reached when all the wells are operating.

The [WEFAC](#) keyword is best used when wells are brought down individually on a regular basis for short periods of maintenance. But if all the wells in a group are brought down together for regular maintenance, then the efficiency factor should instead be applied to the group. Group efficiency factors are defined with the keyword [GEFAC](#). The treatment of group downtime is entirely analogous to the treatment of well downtime. The group's flow rates reported in the [PRINT](#) and [SUMMARY](#) files reflect its full flow rate. Any flow targets and limits for the group will apply to its full flow rate. But the group rates will be multiplied by the group efficiency factor when summing the group's contribution to its parent group in the hierarchy. Thus the flows of any superior groups will reflect this group's time-averaged flow rate.

Automatic opening of wells and well connections

ECLIPSE 100

An automatic opening facility in ECLIPSE 100 allows wells to be opened in sequence at the earliest opportunity, subject to constraints on the drilling rate and the maximum number of open wells allowed in each group. Wells are placed on automatic opening by setting their OPEN / SHUT status to AUTO, in any of the [WCON](#)... keywords or [WEOPEN](#). 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 overridden 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 midway through a timestep by having their efficiency factors temporarily adjusted to allow for the part of the timestep 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 timesteps 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 [WEOPEN](#). 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](#).

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 [WELSMIN](#). 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](#)".

Group and field control facilities

The multilevel 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 multilevel 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.

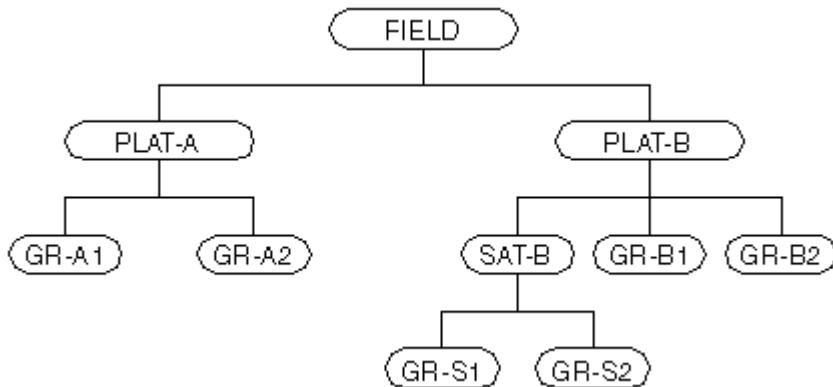


Figure 80.1. A five-level hierarchy

An example of a five-level hierarchy is shown in the figure above. The field occupies the highest level, level 0. PLAT-A and PLAT-B are node-groups at level 1. The groups at level 2 are all well-groups except for SAT-B which is a level 2 node-group. Groups GR-S1 and GR-S2 are level-3 well-groups subordinate to SAT-B. GR-A1 and GR-A2, and their respective wells, are subordinate to PLAT-A. GR-B1 and GR-B2, and their respective wells, are subordinate to PLAT-B. GR-S1 and GR-S2, and their respective wells, are subordinate to both SAT-B and PLAT-B. When the wells are included (on levels 3 and 4), the hierarchy has five levels in total.

Group production control options

Group production rate targets - guide rate control

Selected groups at any level in the hierarchy (including the field) can be made to produce at a target value of any one of the following quantities:

- Oil production rate
- Water production rate
- Gas production rate
- Liquid (oil + water) production rate
- Linearly combined production rate (user-specified linear combination of oil, water and gas)

- Reservoir fluid volume rate
- ECLIPSE 300 only*
- Wet gas rate (which is the volume that the hydrocarbon mixture would occupy at standard conditions if it were an ideal gas)
 - Reservoir volume production balancing fraction (which limits the group's reservoir fluid volume production rate to the specified fraction of its reservoir fluid volume injection rate).
- ECLIPSE 300 only*
- Surface gas volume production balancing fraction (which limits the group's surface gas volume available rate to the specified fraction of its surface gas volume injection rate).
- ECLIPSE 300 only*
- Surface water volume production balancing fraction (which limits the group's surface water volume production rate to the specified fraction of its surface water volume injection rate).
- ECLIPSE 300 only*
- Calorific value rate.

The control target is set using keyword `GCONPROD`. The group's target rate is apportioned between the individual wells in proportion to each well's specified **guide rate**. A guide rate can optionally be defined for any phase; oil, water, gas, liquid or reservoir fluid, using keyword `WGRUPCON`. If the phase differs from the phase under group control, the guide rate is translated into a guide rate for the controlled phase using the well's production ratio for the previous timestep. 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 timestep 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](#)"). 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 [WELSONIN](#) and [WDRILRES](#) described in "[Automatic opening of wells and well connections](#)" 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](#)".

Note: The drilling queue should not be confused with the facility described in "[Automatic opening of wells and well connections](#)". They are both subject to the same constraints on drilling rate, rig availability and maximum number of wells per group. But wells on AUTO are opened as soon as these constraints allow, whereas wells in the drilling queue are opened only when a group or the field cannot meet its rate target.

Group production rate limits

The engineer can impose **upper limits** on any of the quantities listed on "[Group production rate targets - guide rate control](#)", 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.
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.
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](#)". 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.

ECLIPSE 100

Note: When a group which was operating under a rate target for one phase, switches to rate control by another phase in this way, the guide rates for any children of this group are not allowed to be zero.

This is to prevent the following situation. Consider, for example, a group under gas production rate control which also has a limit on the group water production rate to control water breakthrough. When water breakthrough occurs in some of the group's wells and the water rate limit is violated, the group switches to water rate production control. Any wells that do not see any water breakthrough would not have any water production potential, and normally this would mean zero guide rates so these pure gas production wells

would shut - which is obviously not what is required for a group which is primarily intended for gas production.

Instead, a small positive guide rate is imposed on such wells force them to stay open and operate at their potential. **Unfortunately, sometimes this means that the original group gas rate target is exceeded.** Because of this, it is strongly recommended that RATE is not used to control breakthrough in this way. Two possible alternatives are (i) use an alternative procedure such as WELL or CON under [GCONPROD](#) in items 11 onwards, or (ii) use a [GUIDERAT](#) formula that penalizes wells that see breakthrough.

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](#)", 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 80.1 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 timestep. 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 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](#)", 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 watercut. 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 reopened 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 cutback 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 timestep in which they were broken. But if a limit is broken by proportionately more than the **tolerance fraction** specified with keyword [WLIMTOL](#), the timestep 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
- Rejection fraction of a particular phase
- Total voidage replacement fraction of all injected phases.
- Wet gas rate (which is the volume that the injected hydrocarbon mixture would occupy at standard conditions if it were an ideal gas).

ECLIPSE 300 only

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 reinject all its available gas (that is, produced gas minus gas consumed for power), and to inject water so that the water and gas injection together replaces the platform's production voidage. Water is the 'top-up' phase here.

Upper limits can be imposed on the four quantities at any group or the field as a whole. If the injection rate exceeds one of a group's upper limits, the violated limit becomes a target for the group. If the group was already injecting at a target rate, the original target will no longer be met but will act as an upper limit for that quantity. For example, a group can inject water to replace its voidage production, subject to a surface rate limit that represents its injection water handling capacity. Obviously only the 'top-up' phase can have upper limits specified for quantities 2 or 4 since a limit could become a target at some stage in the simulation.

Defining the composition of injected gas

ECLIPSE 300

When gas is injected in compositional simulations, the nature of this gas must be defined. The injected gas may be provided in the following ways:

- The gas may be designated as a named wellstream, whose composition is defined explicitly in keyword [WELLSTRE](#).
- The gas may be provided by the production stream of a named group. Gas used for fuel ([GRUPFUEL](#)) and sales ([GRUPSALE](#)) is by default subtracted from a source group's available gas supply for reinjection, 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 reinjection 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 reinjection 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 [WINJMX](#)). 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 reordered 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 [GCONTROL](#).

Gas accounting tables

ECLIPSE 300

The gas rate accounting table is generated in the print file at each report step, and provides information on how produced gas is used. Each column is described below:

1. Fuel volume rate

The rate at which produced gas is used for fuel. (Controlled by the [GRUPFUEL](#) keyword.)

2. Sales volume rate

The rate at which produced gas is used for sales. (Controlled by the [GRUPSALE](#) keyword.)

3. Reinjection volume rate

The rate at which produced gas is reinjected into the reservoir. (Controlled by the [GCONINJE](#) and [WCONINJE](#) keywords.) This also includes gas which is injected into the reservoir during production balancing of surface gas (controlled by the [GCONPROD](#) keyword).

If item 108 of the [OPTIONS3](#) is non-zero, then this will **not** include any gas injected during the surface gas production balancing group control mode.

4. Excess volume rate

The rate difference between the gas produced and the gas used to meet the fuel, sales and reinjection requirements.

If [OPTIONS3](#) item 108 is non-zero, then gas injected during the surface gas production balancing group control mode will **not** be used in the reporting of the excess (only the reinjected gas during the reinjection group control mode).

5. Sum volume rate

This is the sum of the first four columns and should equal the gas produced by the field for separators that simply accumulate the gas from each stage. (This is the default separator configuration). If gas from an intermediate stage is used as a feed for a subsequent stage and is also a source of reinjection

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".

6. Export volume rate

The sum of the excess gas rate and the sales gas rate, that is the rate produced gas is available for export after the fuel and reinjection requirements have been met.

If OPTIONS3 item 108 is non-zero, then gas injected during the surface gas production balancing group control mode will **not** be used in the reporting of the export (only the reinjected gas during the reinjection group control mode).

7. Advanced gas volume rate

The rate at which advanced gas is used to augment produced gas in a combined reinjection stream. (Controlled by the **GADVANCE** or **WADVANCE** keywords.) The advanced gas is only used for reinjection and not as a source of fuel or sales gas.

This column is not output if the field is operating with a surface gas production balancing limit/target.

8. Make-up gas volume rate

The rate at which a make-up gas stream is used to top-up advanced and re-injected produced gas in order to meet an injection target. (Controlled by the **GINJGAS** and **WINJGAS** keywords.)

This column is not output if the field is operating with a surface gas production balancing limit/target.

9. Reinj + Adv + Make

The sum of volume rates of reinjected produced gas, advanced gas and make-up gas. Note that this may differ from the total field injection gas rate, which could include wellstreams or mixtures not associated with produced gas.

This column is not output if the field is operating with a surface gas production balancing limit/target.

Export tables

Export tables are written to the print file at each report step only when both of the following conditions are met:

- 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 7 of the **SEPCOND** keyword).
- Gas from an intermediate stage of a separator is used as the source of re-injection gas (see item 5 of the **GINJGAS** keyword).

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 reinject 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. In these circumstances, reinjection 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 reinjection work. The difference then between the formation oil and export oil is the oil vapor that is contained in the reinjected 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 reinjection and fuel usage have been accounted for. For this type of separator configuration the sum of the reinjected, 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 keywords (F,G)E(O,G,N)(R,T).

Note: The export rate calculation should be treated with caution when the intermediate separator stage gas is used as part of a mixture to be injected (see the [WINJMIX](#) and [WINJORD](#) keywords) as it may be unable to take the re-injection into account.

Apportioning group injection targets by guide rate

A group or field target injection rate is apportioned between the subordinate wells or groups in a precisely analogous manner to a production target, as described in "[Group production rate targets - guide rate control](#)". 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 timestep. 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.

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 timestep according to a choice of two options. One option sets them equal to the groups' voidage production rate. The other option (available only to the 'top-up' phase) sets them equal to the groups' net voidage, that is the production voidage minus the reservoir volume injection rate in the other phases. An example application is for a platform to reinject its available gas, distributing it to selected groups in proportion to their voidage production rate. The platform could also inject water to replace the overall production voidage, distributing it to all groups in proportion to their net voidage rate. This would give less water to the groups that have gas injection.

Sales gas production control

The associated gas from an oil field, which may at the same time have controls on the oil production rate. The option is selected by using the keyword [GCONSAL](#). 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 reinjecting 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 reinjection control, and its target reinjection fraction is determined dynamically at each timestep to inject the surplus gas. Other limits on gas injection for the group or field (for example 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 timestep to reduce the gas production in the next timestep. 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 reinjection, 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 removes the reinjection fraction limit (by setting the reinjection fraction limit to 1) and continues to recalculate the gas production rate target according to what the injection rate was in the previous iteration. This option therefore maximizes the production rate when the system is constrained by its injection capacity, because if the injection capacity were subsequently to increase then it would allow the production rate to increase accordingly while still meeting the sales target.

There is a flaw in this algorithm if the net sales, import and consumption is zero: stagnation of the injection and production amounts results even if increased injection capacity is available. In this special case, the reinjection fraction limit is set to a larger value (100 by default) so that any increased injection capacity can be used. The value of this increased reinjection fraction limit can be controlled using item 103 of the OPTIONS keyword.

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 timestep. The MAXR option will overwrite the group's gas production rate limit with a value equal to the sales gas target plus the injection rate plus any gas consumption minus any import rate, while the group is operating on gas production rate control. Any user-defined limit on this quantity will be erased.

The RATE and MAXR options require guide rate group control, and are not allowed if prioritization (keyword [GCONPRI](#)) is used to control the group/field's production.

If on the other hand the group or field is not producing enough gas to fulfil its sales requirement, corrective action will be taken at the end of the timestep 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 reinjection fraction target of 1.0, and it will reinject 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 reinject 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 timestep 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 [GCONSAL](#)), 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 reinjection 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 recalculated 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 timestep, and are kept constant for any subsequent iterations. (NUPCOL's value is initialized by keyword [NUPCOL](#) in the RUNSPEC section, and can be reset 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 timestep

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.

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 recalculated 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 timestep. 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 timestep. It is drilled during the timestep 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:

- 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
 - bottom hole pressure versus tubing head pressure
- Producer tables, which describe the vertical flow performance of production wells, entered with keyword [VFPPROD](#). Producer tables may also be used to 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
- tubing head pressure
- water-oil ratio, water cut, or water-gas ratio
- gas-oil ratio, gas-liquid ratio, or oil-gas ratio
- 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 lookup 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 downhole pump, the artificial lift quantity could refer to the pump rating. In ECLIPSE 100 artificial lift can be switched on automatically when a well's production rate falls below a specified limit (see keyword [WLIFT](#)), subject to optional constraints on the total group or field lift capacity (see keyword [GLIFTLIM](#)).

ECLIPSE 100

By default, VFP tables are interpolated linearly in all variables. However, there is an option to apply cubic spline interpolation to the ALQ (Artificial Lift Quantity) variable. This option is primarily intended to be useful for the Gas Lift Optimization facility, in which the gradient of the wells' performance with respect to the lift gas injection rate is important for deciding which wells could gain the most benefit from an increased lift gas supply. The option is enabled with the [VFPTABL](#) keyword.

Note: If the actual simulation conditions fall outside the range given in a VFP table, then linear extrapolation is used. This can produce unrealistic results and should be avoided (by ensuring that each VFP table covers all conditions that will arise during the simulation).

Two or more different wells may use the same VFP table, provided they have similar depths, deviations and internal diameters. Any difference between the bottom hole datum depths of the wells and that of the VFP table will be taken into account using a simple hydrostatic correction to the BHP obtained from the VFP table. In ECLIPSE 100 individual wells may also have a further user-defined adjustment made to their BHPs obtained from the table, specified with keyword [WVFPDP](#). This could be applied, for example, to help match a well's flow rate at a given THP.

When a well is under THP control, its flow rate is determined by the intersection of its inflow performance relationship (IPR) and the VFP curve interpolated at the current value of its water and gas flowing fractions, on a plane of BHP versus flow rate. Often the VFP curve for a producer is U-shaped, and when there are two intersections the one with the higher flow rate is chosen (the other being unstable). If the IPR does not intersect the VFP curve, the well is deemed to have died. Occasionally, wells dying suddenly in this fashion may cause problems in the simulation. The ECLIPSE 100 keyword [WVFPEXP](#) controls some possible remedies, either by using explicit values of the water and gas fractions for the interpolation of the tables or by the use of 'stabilized' VFP curves.

Producer tables are input with the keyword [VFPPROD](#), and injector tables are input with the keyword [VFPINJ](#). A separate interactive program VFP *i* is available to calculate VFP tables, using a choice of multiphase flow correlations to calculate the pressure losses within the tubing string. The program has been specifically tailored for ECLIPSE, in that it can produce any type of VFP table required by the simulator in a form which can be included directly in the input data. It contains extensive graphical facilities for examining and visually editing VFP tables in cross-section and displaying them in 3D.

Well name and well list template matching

The template matching feature of ECLIPSE enables multiple wells to be referenced using a single template containing wildcard characters. The wildcard characters supported by ECLIPSE are the asterisk (*) to match zero or more characters, and the question mark (?) to match a single non-empty character. When these characters are required at the start of a template they should be preceded by a backslash (\) to differentiate between alternative uses of the symbols, for example to indicate a well list (*) and as an alias for the triggering wells with an action block (?). All well name matches are case insensitive.

Consider the specification of the following wells in the SCHEDULE section.

```
WELSPECs
'PR10-OIL'    GR-A1    3    2    1*    OIL /
'PR11-OIL'    GR-A1    3    5    1*    OIL /
'GAS-PR1'     GR-A2    3    8    1*    GAS /
'GAS-PR2'     GR-A2    3    3    1*    GAS /
'IN1-GAS'      GR-A3    4    4    1*    GAS /
'IN2-GAS'      GR-A3    4    6    1*    GAS /
'IN10-WAT'    GR-A4    5    2    1*    WAT /
'IN11-WAT'    GR-A4    6    5    1*    WAT /
'X'           GR-A5    7    2    1*    LIQ /
'Y'           GR-A5    7    5    1*    LIQ /
/
```

New well lists consisting of all the wells containing the sub-string ‘PR’ and another consisting of all the wells containing the sub-string ‘IN’ could then be created in the following manner:

```
WLST
'*PR-WELS'    NEW      '\*PR*' /
'*IN-WELS'    NEW      '\*IN*' /
/
```

Similarly to shut all wells with names composed of seven characters the question mark symbol can be used.

```
WEOPEN
'\???????'    SHUT /
/
```

In this context the preceding backslash (\) is not strictly required; however, if the keyword appears within an action block it is necessary to differentiate between the wildcard character and the alias for the triggering wells. For example, when the following action is triggered all wells with a water cut greater than 0.25 will be shut, whilst all wells with single character names will have their liquid rate target / limit halved.

```
ACTIONs
ACT1 100000 /
WWCT '*' > 0.25 /
/
-- either use the alias ? to select triggering wells
WEOPEN
'?'    SHUT /
/
-- or match template against all the wells in the field with a single
-- character name
WTMULT
'\?'    LRAT    0.5 /
/
ENDACTIO
```

To maintain back-compatibility a single asterisk still represents all the wells in the field. Alternatively, the above action could test only a subset of wells, for example those belonging to a well list

```
ACTIONX
ACT1 100000 /
WWCT '*PR-WELS' > 0.25 /
/
```

or matching a given template

```
ACTIONX
ACT1 100000 /
WWCT '\*GAS*' > 0.25 /
/
```

However, note that it is not possible to use the template matching mechanism to select a subset of the wells triggering the action.

Wildcard characters can be included in well names and well lists in all appropriate SCHEDULE keywords. In addition wildcard characters can be included in well names and well lists for **W*** and **LW*** keywords in the SUMMARY section. For example

```
-- well water production rate for wells containing the
-- substring 'PR' and the wells X & Y
WWPR
'\*PR*' 'X' 'Y' /
-- well mode of control for wells belong to well lists
-- containing the substring 'WELS'
WMCTL
'*WELS*' /
```

Note: For all keywords, well name and well list templates containing wildcard characters must be enclosed within quotes.

The **WTEMPQ** keyword can be used to further explore the well template matching mechanism. The **WTEMPQ** keyword writes a list of all the currently defined wells matching the supplied well name or well list template to the PRT file.

ECLIPSE 100

The pre-2006.1 well name and well list behavior can be restored through [item 93](#) of the OPTIONS keyword.

ECLIPSE 300

The pre-2006.1 well name and well list behavior can be restored through [item 94](#) of the OPTIONS3 keyword.

A

RUNSPEC section: pre-95A fixed record structure

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	Description
Record 2	Title
Record 4	Number of blocks in X,Y,Z directions
Record 6	Phases solved for, that is which saturations, R_s or R_v vary
Record 8	Units
Record 18	Maximum number of wells, groups, wells per group, connections per well.
Record 28	Start date of the simulation
Record 30	Run Simulation (default) or Data Check

An explanatory listing of the contents of the RUNSPEC section will be written to the PRINT file if the RUNSPEC section is followed immediately by the keyword RPTRUNSP, before the GRID keyword.

Detailed description of RUNSPEC records

The contents of each record are listed below. For a more detailed explanation of each item, please refer to the description of the equivalent keyword in the new RUNSPEC format.

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)

- QVEOPT Set to T if the Vertical Equilibrium option is required (keyword V_E)
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
- NTEQLU 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
- QT DISP 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
 - NMFIPIR 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 timestep 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 disk groups (keyword DISKING).
DEFAULT: 0
- IDYNAM GRID data disk 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

B

ECLIPSE Pre-2002A integer controls

This appendix is the pre-2002A listing of the integer controls for the following keywords:

- [RPTPROPS](#)
- [RPTGRID](#)
- [RPTRST](#)
- [RPTSCHED](#)
- [RPTSOL](#)

RPTPROPS Controls on output from PROPS section

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This section contains the pre-2002A listing of the PROPS 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 (/).

In ECLIPSE 100, entering the mnemonic NOTHING clears all the reporting flags.

For additional control over the output in ECLIPSE 100, 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.

Optionally, for back compatibility when using ECLIPSE 100, 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.

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

Input Keyword or Mnemonic	Output	E100 Integer Control	E300
KROR	Output of oil relative permeabilities at residual		x
KRW	Output of maximum water relative permeabilities	9	x
KRWR	Output of water relative permeabilities at residual	9	x
LANGMUIR, LANGSOLV, COALADS	Output of the Langmuir Isotherms	22	
MISC, PMISC	Output of the miscibility function and the pressure miscibility function	17	
OVERBURD	Overburden pressure data		x
PCG	Output of scaled maximum gas capillary pressures	9	x
PCW	Output of scaled maximum water capillary pressures	9	x
PLYVISC etc.	Output of Polymer Flood Model properties	11	
PVDG, PVTG	Output of gas PVT tables	6	
PVDO, PVTO	Output of oil PVT tables	4	x
PVDS	Output of Solvent PVT properties	15	
PVTW	Output of water PVT tables	5	x
PVTWSALT	Output of brine properties	20	x
ROCK,	Output of rock properties	8	
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, three-phase relative permeability 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, VISCVCT	Output of the temperature-dependent properties	24	
SSFN	Output of Solvent/Gas relative permeabilities	16	
STOG, STOW	Output of Surface Tension properties	14	
SURFVISC etc.	Output of surfactant properties	19	

Input Keyword or Mnemonic	Output	E100 Integer Control	E300
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 relative permeability for gas.		x
VDKRO	Velocity-dependent relative permeability 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), for example KRGRZ-.

Examples

Example 1

```
RPTPROPS
  SWFN SGFN PVTO PVTW PVDG /
```

Example 2

Changing the level of output in ECLIPSE 100:

```
RPTPROPS
  SOF3=5 SWFN=5 SGFN=5 /
```

Example 3

The old integer format (ECLIPSE 100):

```
RPTPROPS
  10*1 10*0 1 1 0 1 /
```

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 and DY 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. The data should be terminated by a slash (/).

In ECLIPSE 100, entering the mnemonic NOTHING clears all the reporting flags.

In ECLIPSE 100, to provide 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.

In ECLIPSE 100 there is, optionally, for back compatibility, a list of integers (terminated with a slash) which 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.

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
COALNUM	Output of coal region numbers		x
COALV	Output of grid block coal volumes		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

Input keyword or mnemonic	Output	E100 integer control	E300 availability
DIFFZ or DIFFTZ	Output of Z diffusivities	33	x
DOMAINS	Output of domains (Parallel Option). See also the DOMAINS keyword. The output consists of: <ul style="list-style-type: none"> The solver direction, which ECLIPSE calculates automatically but which may be modified by using the SOLVDIRS keyword. 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
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

Input keyword or mnemonic	Output	E100 integer control	E300 availability
MINPV or MINPVV	Output of cells with pore volume less than the input minimum values	35	x
MPFANUM	Output the MPFANUM data when using the multi-point flux approximation		x
MPFORTH	Output the multi-point flux k-orthogonality error		x
MPFTRAN	Output the multi-point flux forward transmissibilities		x
MPFTREV	Output the multi-point flux reverse transmissibilities		x
MULTNUM	Output of regions for inter-region transmissibility multipliers	53	x
MULTPV	Output of the pore volume multipliers	41	x
MULTREGP	Output region pore volume multipliers	59	x
MULTREGT	Output of multipliers between regions	47	x
MULTX	Output of X transmissibility multipliers	7	x
MULTY	Output of Y transmissibility multipliers	8	x
MULTZ	Output of Z transmissibility multipliers	9	x
NINENUM	Output the NINENUM data when using the nine-point scheme	56	x
NNC	Output of non-neighbor connections	21	x
NTG	Output of grid block net-to-gross ratios	11	x
PERMX	Output of X direction permeabilities	4	x
PERMY	Output of Y direction permeabilities	5	x
PERMZ	Output of Z direction permeabilities	6	x
PERMXY, PERMYZ, 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
PORVF	Output of grid block fluid pore volumes		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

Input keyword or mnemonic	Output	E100 integer control	E300 availability
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	

Table 82.1: RPTGRID output controls

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 centered grids input using the DX, DY, DZ, TOPS keywords can also be output in corner point form.

Control 19

- 0 No output
- 1 Output to the PRINT file.
- 2 COORD keyword written to the DEBUG file in floating point format (F12.2)
- 3 COORD keyword written in exponent format (1PE12.6)

Control 20

- 0 No output
- 1 Output to the PRINT file.
- 2 ZCORN keyword written to the DEBUG file in floating point format (F12.2)
- 3 ZCORN keyword written in exponent format (1PE12.6)

Examples

Example 1

```
RPTGRID  
COORD TRANX TRANY TRANZ ALLNNC /
```

Example 2

ECLIPSE 100

Changing the level of output

```
RPTGRID  
COORD=2 ZCORN=2 /
```

Example 3

ECLIPSE 100

The old integer format:

```
RPTGRID  
14*1 3*0 /
```

RPTRST

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; the use of mnemonics is 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 RPTSLQ 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.

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 nth 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 nth 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 nth month. If set to 6 then a RESTART file is written at every timestep.	1

Input Keyword Or Mnemonic	Output	Integer Control
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
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

Input Keyword Or Mnemonic	Output	Integer Control
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
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 nth 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 nth 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 nth 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 $S_{g,\text{trap}}$ 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

Input Keyword Or Mnemonic	Output	Integer Control
	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 Controls on output from SCHEDULE section

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

In ECLIPSE 100, entering the mnemonic NOTHING clears all the reporting flags.

For additional control over the output in ECLIPSE 100, 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.

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

Input Keyword or Mnemonic	Output	E100 integer control	E300
CPU	<p>Output of breakdown of CPU and elapsed time for current run.</p> <p>If set to 1 then report is sent to Print file.</p> <p>If set to 2 then in addition, a copy of the report is sent to the short form output (terminal)</p> <p>If set to 3 then report is sent to short form output only.</p> <p>See "Timing Reports" for an explanation of the outputs. Internal clocks within TS are not recorded during intermediate timestep when set to 0.</p>	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
FIP	<p>Output of fluid in place and inter-region flow reports.</p> <p>If set to 1: then fluids in place are reported for the whole field.</p> <p>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.</p> <p>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.</p>	8	
FIPFOAM	<p>Output of foam fluid in place reports (for the Foam Model)</p> <p>1 gives a report for the whole field</p> <p>2 in addition gives a report for each fluid in place region</p>	64	
FIPRESV	Output of region reservoir volume reports	76	
FIPSALT	<p>Output of salt fluid in place reports (for Brine option and salt-sensitive Polymer Flood option)</p> <p>1 gives a report for the whole field</p> <p>2 in addition gives a report for each fluid-in-place region</p>	38	
FIPSOL	<p>Output of the Solvent fluid in place reports.</p> <p>1 gives a report for the whole field</p> <p>2 in addition gives a report for each fluid-in-place region</p>	43	

Input Keyword or Mnemonic	Output	E100 integer control	E300
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
FLOE	Output of inter-block energy flow (ECLIPSE 300 Thermal option, using THERMAL keyword)		x
FLOGAS	Output of inter-block gas flows (black oil cases only)	74	x
FLOOIL	Output of inter-block oil flows (black oil cases only)	72	x
FLOSOL	Output of inter-block solvent flows	77	
FLOWAT	Output of inter-block water flows (except for GASWAT and CO2STORE options)	73	x
FMISC	Output of relative permeability miscibility factor (Miscible option)		x
FOAM	Output of grid block foam concentration (for the Foam option)	63	
FOAMADS	Output of grid block foam adsorption (for the Foam option)	68	
FOAMDCY	Output of grid block foam decay (for the Foam option)	69	
FOAMMOB	Output of gas mobility factor due to foam (for the Foam option)	70	
FPC	Output of capillary pressure miscibility factor (Miscible option)		x
FUGG	Output of component fugacities in gas (Activity Driven Diffusion)		x
FUGO	Output of component fugacities in oil (Activity Driven Diffusion)		x
GASCONC	Output of matrix grid block gas concentration (for the Coal Bed Methane option)	54	
GASPOT	Output of gas potentials		x
GDOWAT	Output of gas displaced into oil and water zones (for the VE option) 1 = by regions and field, >1 = by grid blocks	29	

Input Keyword or Mnemonic	Output	E100 integer control	E300
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
GMO	Output oil phase energy mobility (ECLIPSE 300 Thermal)		x
GMEW	Output water phase energy mobility (ECLIPSE 300 Thermal)		x
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	

Input Keyword or Mnemonic	Output	E100 integer control	E300
ODWAT	Output of oil displaced into water zone (for VE option) 1 = by regions and field, >1 = by grid blocks	28	
OILAPI	Output of grid block oil API values (for API Tracking)	31	
OILPOT	Output of oil potentials		x
PART	Output of domain partitions (Parallel option)		x
PB or PUB	Output of grid block bubble point pressures	55	x
RSSAT	1 outputs bubble point pressures only 2 in addition, outputs the saturated dissolved gas-oil ratios		
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	Output of grid block dew point pressures	56	x
RVSAT	1 outputs dew point pressures only 2 in addition, outputs the saturated vaporized oil-gas ratios		
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 ")	71	

Input Keyword or Mnemonic	Output	E100 integer control	E300
RESTART	<p>Output of Restart files</p> <p>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).</p> <p>If set > 1 then Restart files are created at every report time until this switch is reset, and all are kept.</p> <p>If set > 2 then the inter block flows of each phase are written to the Restart files.</p> <p>If set > 3 then the current fluids-in-place and phase potentials in each grid cell are written to the Restart files.</p> <p>If set = 6 then Restart files are created at every timestep.</p> <p>See also the keywords RESTART, 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.</p> <p>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.</p> <p>In ECLIPSE 300 the keyword OUTSOL controls output to the Restart file.</p>	7	
RK	Output of the permeability reduction factor currently being applied to water phase relative permeability (for Polymer Flood option)	37	
ROCKC	<p>Output of the modified pore volume and transmissibility multiplier (for the Rock Compaction model).</p> <p>2: gives the modified DZ assuming compaction in the Z-direction (for hysteretic compaction)</p> <p>3: in addition sums DZ for each column of grid blocks (for hysteretic compaction).</p>	41	
RPORV	Output of reservoir pore volumes		x
RS	Output of grid block solution gas-oil ratios	5	x
RV	<p>Output of grid block vapor oil-gas ratios</p> <p>If set >1: With the discretized matrix option active, the solution values into the matrix are also reported.</p>	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
SGTRAP	Trapped gas saturation and dynamic trapped gas saturation (for WAG Hysteresis model only)	75	

Input Keyword or Mnemonic	Output	E100 integer control	E300
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 timestep 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 the two to three phase transition (ECLIPSE 300 Thermal)		x
T3	Output bubble point (ECLIPSE 300 Thermal)		x
TBLK or TRACER	Output of grid block tracer concentrations (for Tracer Tracking)	33	
TEMP	Output of the grid block temperature (for the Temperature option or ECLIPSE 300 Thermal)	65	x
TET	Output temperatures at last time level (ECLIPSE 300 Thermal)		x
TO	Output temperature at which the oil phase leaves the system (ECLIPSE 300 Thermal)		x
TOTCOMP	Output the total compressibility of the fluid system (excluding the rock)		x
TW	Output temperature at which the water phase leaves the system (ECLIPSE 300 Thermal)		x
TRADS or TRDCY	Output of tracer adsorption and decay (for Environmental Tracers)	47	

Input Keyword or Mnemonic	Output	E100 integer control	E300
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 two to three phase transition (ECLIPSE 300 Thermal)		x
VFPPROD	<p>Output of vertical flow performance tables. (In order to obtain this, RPTSCHED must be placed before VFPPROD etc.)</p> <p>This also turns on network convergence messages in the Network option.</p> <p>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.</p> <p>If set > 2 then only the table comparing the maximum dimensions is written.</p>	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	<p>Output of water displaced into oil and gas zones (for VE option)</p> <p>1 = by regions and field, 1= by grid blocks</p>	30	
WELLS	<p>Output of well reports with production and injection rates and totals.</p> <p>1 gives a report of well flows</p> <p>2 gives a report of well and connection flows</p> <p>3 gives a report of layer totals</p> <p>4 gives a report of layer totals and well flows</p> <p>5 gives a report of layer totals, well and connection flows.</p>	9	
WELSPECS or WELSPECL	Output of connection, well and group keyword data from the SCHEDULE section.	14	

Input Keyword or Mnemonic	Output	E100 integer control	E300
WOC	Output of current water-oil contact (WOC) depths (for VE option) (Output is AAAAA if contact depth is above top of cell; output is BBBBB if contact depth is below bottom of cell)	22	
WOCDIFF	Output of current WOC depth - initial WOC depth (for VE option)	24	
WOCGOC	Output of current WOC depth - current GOC depth (for VE option)	26	
XGAS	Output of gas fractional saturations (Sudation option)		x
XMF	Output of liquid component mole fractions (fraction of each component in liquid)		x
XWAT	Output of water fractional saturations (Sudation option)		x
YMF	Output of gas component mole fractions (fraction of each component in vapor)		x
ZMF	Output of total component mole fractions (fraction of each component in liquid)		x

Note:

- Saturation pressures (PSAT) are not normally used in ECLIPSE 300, and their calculation may involve significant extra CPU time.
- 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

```
RPTSCHE
'OIL'   'SGAS'   'RS'    'RESTART'  'FIP=2'   'WELSPECS'  'WELLS=2'
'SUMMARY=2'  'AQUCT'  'NEWTON' /
```

Example 2

Equivalent to Example 1

```
RPTSCHE
0 1 0 1 1 0 1 2 2 0 2 2 1 1 1 /
```

Example for ECLIPSE 300

```
RPTSCHE
PRES SWAT SOIL SGAS PSAT XMF YMF /
```

RPTSOL**Controls on output from SOLUTION section**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

This section is the pre-2002A 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 (/).

In ECLIPSE 100, Entering the mnemonic NOTHING clears all the reporting flags.

For additional control over the output in ECLIPSE 100, 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.

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

Input Keyword or Mnemonic	Output	E100 Integer value	E300
ENERGY	Internal energy / bulk volume (THERMAL option)		x
EQUIL	Output of equilibration data	9	
FFORG	Forchheimer Non-Darcy flow factor for Gas (VELDEP option)		x
FFORO	Forchheimer Non-Darcy flow factor for Oil (VELDEP option)		x
FIP	Output of fluid in place reports: If set to 1 then initial fluids in place are reported for the whole field, If set to 2 then in addition, initial fluids in place are reported for each fluid in place region defined with the FIPNUM keyword, If set to 3 then in addition, initial fluids in place are reported for all sets of fluid in place regions defined with the FIP keyword.	8	
FIPFOAM	Output of foam fluid in place reports (for the Foam Model) 1 gives a report for the whole field 2 in addition gives a report for each fluid-in-place region	64	
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	

Input Keyword or Mnemonic	Output	E100 Integer value	E300
FIPVE	Output of fluid in place in original oil, water and gas zones in the VE option	21	
FLOCn	Output of inter-block component flows (for component n)		x
FLOE	Output of inter-block energy flow (THERMAL option only)		x
FLOGAS	Output of inter block gas flows (black oil cases only)	74	x
FLOOIL	Output of inter block oil flows (black oil cases only)	72	x
FLOSOL	Output of inter block solvent flows (for the Solvent Model)	77	
FLOWAT	Output of inter block water flows	73	x
FMISC	Output of relative permeability miscibility factor (Miscible option)		x
FOAM	Output of grid block foam concentration (for the Foam option)	63	
FOAMADS	Output of grid block foam adsorption (for the Foam option)	68	
FOAMDCY	Output of grid block foam decay (for the Foam option)	69	
FOAMMOB	Output of gas mobility factor due to foam (for the Foam option)	70	
FPC	Output of capillary pressure miscibility factor (Miscible option)		x
FUGG	Output of component fugacities in gas (Activity Driven Diffusion)		x
FUGO	Output of component fugacities in oil (Activity Driven Diffusion)		x
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

Input Keyword or Mnemonic	Output	E100 Integer value	E300
NCNO	Output of Normalized Capillary Numbers for Oil (VELDEP option)		x
OILAPI	Output of initial grid block API values (for API tracking)	31	
OILPOT	Output of oil potentials		x
PART	Output of domain partitions (Parallel option)		x
PB or PBUB RSSAT	Output of grid block bubble point pressures 1 outputs bubble point pressures only 2 in addition, outputs the saturated dissolved gas-oil ratios	55	x
PBLK or POLYMER	Output of initial grid block polymer concentration values (for Polymer Flood Model)	34	
PCGW	Output of gas-water capillary pressures (GASWAT cases)		x
PCOG	Output of gas-oil capillary pressures		x
PCOW	Output of oil-water capillary pressures		x
PD or PDEW RVSAT	Output of grid block dew point pressures 1 outputs dew point pressures only 2 in addition, outputs the saturated vaporized oil-gas ratios	56	x
PGAS	Output of initial grid block gas phase pressures	19	
PGASD or POTG	Output of initial grid block gas phase pressures corrected to datum depth) (See also the DATUM keyword)	20	
PKRG	Output of processed max. oil relative permeability and the gas relative permeability at residual oil if the end point scaling is activated.	62	
PKRO	Output of processed max. oil relative permeability and the oil relative permeability 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	

Input Keyword or Mnemonic	Output	E100 Integer value	E300
PRES or PRESSURE	Output of initial oil pressures	1	x
PRESMIN	Output of minimum pressures (irreversible Rock Compaction option)		x
PSAT	Output of saturation pressures (Compositional mode)		x
PWAT	Output of initial grid block water phase pressures	17	
PWATD or POTW	Output of initial grid block water phase pressures (corrected to datum depth)	18	
RECOV	Output of recovery mechanisms	71	
RESTART	Output of Restart files: If set > 1 then an initial Restart file is created. If set > 2 then the inter block flows of each phase are written to the Restart files. If set > 3 then the current fluids in place and phase potentials in each grid cell are written to the Restart files. (There must be at least one TIME , TSTEP or DATES keyword if an initial Restart file is required; but the simulation mode can be turned off by the keyword NOSIM in the RUNSPEC section) In ECLIPSE 300 the keyword OUTSOL controls output to the Restart file.	7	
RK	Output of permeability reduction factor currently being applied to the water phase (for Polymer Flood Model)	37	
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 R_v versus depth	11	
SALT	Output of initial grid block salt concentration values (for Brine option and salt-sensitive Polymer Flood Model)	35	x

Input Keyword or Mnemonic	Output	E100 Integer value	E300
SALTVD	Output of tables of initial salt concentration versus depth (entered using SALTVD keyword)	29	x
SGAS	Output of initial grid block gas saturations	4	x
SGCO or SGCR or SGL or SGU	Output of the connate, critical and maximum gas saturations for each grid cell if the end-point scaling or the mobile fluid correction option is activated.	27	
SOCRS	Output of critical oil saturations following a surfactant flood (Surfactant option with end point scaling active)	82	
SOGCR	Output of the critical oil-in-gas saturations for each direction for each grid cell if the end point scaling or the mobile fluid correction is activated.	25	
SOIL	Output of initial grid block oil saturations	2	x
SOWCR	Output of the critical oil-in-water saturations for each direction for each grid cell if the end point scaling or the mobile fluid correction is activated.	24	
SSOL	Output of the initial solvent saturation.	42	
STEN	Output of oil/gas surface tensions (Miscible option)		x
SURFADS	Output of current adsorbed surfactant concentration on the formation rock (for the Surfactant Model)	45	
SURFBLK	Output of the Surfactant concentrations (for the Surfactant Model)	44	
SWAT	Output of initial grid block water saturations	3	x
SWATINIT	Output of scaled maximum water-oil capillary pressures generated from initial water distribution entered with keyword SWATINIT .	30	
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

Input Keyword or Mnemonic	Output	E100 Integer value	E300
VMF	Output of vapor mole fractions (fraction of total hydrocarbon moles in gas)		x
VOIL	Output of grid block oil viscosity	48	x
VWAT	Output of grid block water viscosity	49	x
WATPOT	Output of water potentials		x
WOC	Output of initial water-oil contact depths in the VE option (Output is AAAAA for contact depth above top of cell; output is BBBBB for contact depth below bottom of cell)	22	
XGAS	Output of gas fractional saturations (Sudation option)		x
XMF	Output of liquid component mole fractions (fraction of each component in liquid)		x
XWAT	Output of water fractional saturations (Sudation option)		x
YMF	Output of vapor component mole fractions (fraction of each component in gas)		x
SOLVD	Output of initial solution versus depth tables		x
ZMF	Output of total component mole fractions		x

Examples

Examples 1 and 2 refer to ECLIPSE 100 and example 3 refers to ECLIPSE 300.

Example 1

```
RPTSOR
RESTART=2 SOIL SWAT SGAS /
```

Example 2

```
RPTSOR
0 1 0 1 0 1 0 2 1 0 1 0 /
```

Example 3

```
RPTSOR
PRESSURE SWAT SOIL SGAS PSAT XMF YMF /
```

C

Retired keywords

This appendix provides information on keywords that are no longer supported. They have been retired, are obsolete or are associated with external products that are no longer supported. However, you may have them in your datasets. The keywords are:

- [LSCRITL](#)
- [PSEUDOS](#)
- [WCONINJ](#)

LSCRITL

Linear solution convergence criteria for all local grids

This keyword was retired in 2001A.

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDELE

PSEUDOS**Requests output of data for the PSEUDO package**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

If this keyword is included in the GRID section, ECLIPSE writes out the binary data files which are required for input to the PSEUDO package. The PSEUDO package is now retired and no longer supplied with the ECLIPSE suite. These files are:

`root.INIT` (where root is the file name root for the current run). This file contains a summary of data entered in the GRID, PROPS and REGIONS sections.

`root.X0001, X0002, X0003` and so on. The inter-block flow rates are appended to the restart files. Note that if PSEUDOS is specified, a restart file is generated at **every** timestep, regardless of the setting of the setting of the RESTART mnemonic in the RPTSCHEK keyword or the BASIC mnemonic in the RPTRST keyword.

The file naming convention can vary slightly according to the make of computer.

These files may be formatted (for transfer between computers of different makes, or just for readability), or unformatted (the default). The type of file created is controlled by the keyword FMTOUT in the RUNSPEC section. Formatted files have slightly different names, to distinguish them from unformatted files.

The PSEUDOS keyword has no associated data.

See also keyword INIT and "*File Handling in ECLIPSE*".

WCONINJ

Injection well control data, with no group control

Attention: This keyword is obsolete and has been superseded by the simpler keyword [WCONINJE](#), which is recommended in place of this one. However, if you have not replaced the keyword in your datasets, you can use the information here.

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

This keyword can be used to set individual control targets and limits for injection wells. WCONINJ can also be used to supply individual reinjection and voidage replacement targets for injectors. But reinjection and voidage replacement can be performed with more flexibility by group injection control using keyword [GCONINJE](#), which is recommended wherever possible. Injectors under group control should be specified with the keyword [WCONINJE](#) instead of this one.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated at any item after Item 5. The remaining items will assume their default values. Default values can be specified before the slash by null repeat counts of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name root or well list name

A well name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several wells in one record. Alternatively a well list name, enclosed in quotes and beginning with an asterisk (*), may be used. Well lists are constructed with the keyword [WLIST](#).

2. Injector type

OIL Oil injector

WATER Water injector

GAS Gas injector

3. Open/shut flag for the well

OPEN Well open for injection

STOP Well stopped off above the formation, allowing crossflow through the well bore

SHUT Well completely isolated from the formation

AUTO Well initially SHUT, but are opened automatically as soon as constraints on drilling rate (keyword [WDRILTIM](#)), drilling rig availability (keyword [GRUPRIG](#)), and the maximum number of open wells per group (keyword [GECON](#) item 9) allow. If no such constraints are specified, the well will open immediately. Wells on AUTO are opened in the order in which they are first specified in keyword [WELSPECS](#), unless any belong to a group that already has its maximum number of open wells or has all its available drilling rigs already occupied.

DEFAULT: OPEN

4. Control mode

RATE Controlled by surface flow rate

RESV Controlled by reservoir fluid volume rate

BHP Controlled by BHP

THP Controlled by THP

5. Surface flow rate target or upper limit (the constant part)

For reinjection wells (see items 7 and 8) this quantity is added to the reinjection flow. It can be negative if the reinjection flow is to be reduced by a constant amount. Set this to zero if no constant term is required in the reinjection rate. For wells that do **not** have a reinjection target, a zero value entered here prevents the well from flowing.

UNITS: sm³/day (METRIC), stb/day for oil or water (FIELD), Mscf/day for gas (FIELD), scc/hr (LAB), sm³/day (PVT-M)

DEFAULT: Infinity (1.0E20)

6. Reservoir fluid volume rate target or upper limit (the constant part)

For voidage replacement wells (see items 7 and 8) this quantity is added to the voidage replacement rate. It can be negative if the voidage replacement rate is to be reduced by a constant amount. Set this to zero if no constant term is required in the voidage replacement rate. For wells that do **not** have a voidage replacement target, a zero value entered here prevents the well from flowing.

UNITS: rm³/day (METRIC), rb/day (FIELD), rcc/hr (LAB), rm³/day (PVT-M)

DEFAULT: Infinity (1.0E20)

7. ReInjection or voidage replacement fraction (see Item 8)

DEFAULT: 0.0

8. ReInjection or voidage replacement flag

NONE No reinjection or voidage replacement

GPRD Well reinjects a fraction of its group's production

FPRD Well reinjects a fraction of the field's production

GVDG Well replaces a fraction of its group's voidage

FVD Well replaces a fraction of the field's voidage

DEFAULT: NONE

If GPRD is specified, the target or limiting surface flow rate = Item 5 + Item 7 * group production rate

If FPRD is specified, the target or limiting surface flow rate = Item 5 + Item 7 * field production rate

If GVDG is specified, the target or limiting reservoir volume rate = Item 6 + Item 7 * group voidage rate

If FVD is specified, the target or limiting reservoir volume rate = Item 6 + Item 7 * field voidage rate

Reinjection and voidage replacement wells cannot be under group control and cannot go into the drilling queue.

9. BHP target or upper limit

It is recommended that this quantity should be set to a value not exceeding the highest pressure in the PVT tables, to ensure the tables are not extrapolated in the well. It is best not to default this quantity, as its value is used in calculating the well potential. A warning message will be printed if this is defaulted.

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

DEFAULT: 1.0E5 psia = 6895 barsa = 6804 atma

10. THP target or upper limit

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

DEFAULT: Infinity (1.0E20)

11. Injection well VFP table number (See keyword [VFPINJ](#))

Set this to zero if no THP calculations are required and the limiting THP value has been defaulted

DEFAULT: 0

12. Vaporized oil concentration in the injected gas, or dissolved gas concentration in the injected oil

UNITS: sm³/sm³ (METRIC), stb/Mscf for gas injectors (FIELD), Mscf/stb for oil injectors (FIELD), scc/scc (LAB), sm³/sm³ (PVT-M)

DEFAULT: 0.0

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

See also keyword [WELTARG](#), which can be used to reset selected control quantities individually.

Example

INJ1 Reinjects half its group's gas production less 200 Mscf/day, subject to a BHP upper limit of 5000 psia and a THP upper limit of 1000 psia. The THP is calculated using Injection Well VFP Table number 2.

INJ2 Injects water to replace 0.2 of the field voidage, subject to a BHP upper limit of 4000 psia.

INJ3 Injects water at a constant rate of 500 stb/day, subject to a BHP upper limit of 5000 psia.

```
WCONINJ
INJ1 GAS OPEN RATE -200 1* 0.5 GPRD 5000 1000 2 /
INJ2 WAT OPEN RESV 1* 0.0 0.2 FVDG 4000 /
INJ3 WAT OPEN RATE 500 3* 5000 /
/
```

D

Bibliography

1. Aavatsmark, I., Barkve, T. and Mannseth, T. "Control-Volume Discretization Methods for 3D Quadrilateral Grids in Inhomogeneous, Anisotropic Reservoirs," paper SPE 38000, *Society of Petroleum Engineers Journal* (1998) 3, No. 2, 146-154; also presented at the SPE Reservoir Simulation Symposium, Dallas, Texas, USA (June 1997).
2. Abbott, M.M. and van Ness, H.G. *Thermodynamics with chemical applications*, Schaum's Outline Series, USA, McGraw-Hill (1989).
3. Ansari, A. M., Sylvester N. D., Sarica O., and Shoham J. P. "A Comprehensive Mechanistic Model for Upward Two-Phase Flow in Wellbores," *SPE Production & Facilities* (1994) 9, No. 2, 143-151.
4. Appleyard, J.R. and Cheshire, I. M. "Nested Factorization," paper SPE 12264, presented at the SPE Symposium on Reservoir Simulation, San Francisco, California, USA (November 15-18, 1983).
5. Arri, L.E., Yee, D., Morgan, W.D., and Jeanssone, N.W. "Modeling Coalbed Methane Production with Binary Gas Sorption," paper SPE 24363, presented at the SPE Rocky Mountain Regional Meeting, Casper, Wyoming, USA (May 18-21, 1992).
6. Aziz, K., Govier, G. W., and Fogarasi, M. "Pressure Drop in Wells Producing Oil and Gas," *Journal of Canadian Petroleum Technology* (1972) 11, No. 3, 38-48.
7. Aziz, K., Ramesh, A. B., and Woo, P. T. "Fourth SPE Comparative Solution Project: Comparison of Steam Injection Simulators," *Journal of Petroleum Technology* (1987) 39, No. 12, 1576-1584.
8. Aziz, K. and Settari, A. *Petroleum Reservoir Simulation*, London, UK, Applied Science Publishers (1979) 398.
9. Baker, L. E. "Three-Phase Relative Permeability Correlations," paper SPE 17369, presented at the SPE Enhanced Oil Recovery Symposium, Tulsa, Oklahoma, USA (April 16-21, 1988).
10. Bardon, C. and Longeran, D. G. "Influence of Very Low Interfacial Tensions on Relative Permeabilities," paper SPE 7609, *Society of Petroleum Engineers Journal* (1980) 20, No. 15, 391-401.
11. Barker, J.W. and Fayers F.J. "Transport Coefficients for Compositional Simulation With Coarse Grids in Heterogeneous Media," paper SPE 22591, *SPE Advanced Technology Series* (1994) 2, No. 2, 103-112.
12. Beattie, C.I., Boberg, T.C., and McNab, G.S. "Reservoir Simulation of Cyclic Steam Stimulation in the Cold Lake Oil Sands," paper SPE 18752, *Society of Petroleum Engineers Journal*, (1991) 6, No. 2, 200-206.
13. Beggs, H. D. *Production Optimization Using Nodal Analysis*, OGCI Publications, Oil & Gas Consultants Inc., Tulsa, USA (1991) 81.

14. Bowers, B.E., Brownlee, R.F., and Schrenkel, P.J. "Development of a Downhole Oil/Water Separation and Reinjection System for Offshore Application," *SPE Production and Facilities* (2000) 15, 115-122.
15. Brie A., Pampuri F., Marsala A.F., and Meazza O. "Shear Sonic Interpretation in Gas-Bearing Sands," paper SPE 30595, presented at the SPE Annual Technical Conference & Exhibition, Dallas, USA (October 22-25, 1995).
16. Carlson, F. M. "Simulation of Relative Permeability Hysteresis to the Non-Wetting Phase," paper SPE 10157, presented at the SPE Annual Technical Conference & Exhibition, San Antonio, Texas, USA (October 5-7, 1981).
17. Carter, R. D. and Tracy, G. W. "An Improved Method for Calculating Water Influx," *Transactions of AIME* (1960) 219, 215-417.
18. Chang, Y-B, Coats B. K., and Nolen, J. S. "A Compositional Model for CO₂ Floods Including CO₂ Solubility in Water," paper SPE 35164, *SPE Reservoir Evaluation & Engineering* (1998) 1 No. 2, 155-160; also presented at the Permian Basin Oil and Gas Recovery Conference, Midland, Texas, USA (1996).
19. Charbeneau, Randall J. *Groundwater Hydraulics and Pollutant Transport*, Englewood Cliffs, New Jersey, USA, Prentice Hall (2000).
20. Cheng, L., Reme, A.B., Shan, D., Coombe, D.A., and Rossen, W.R. "Simulating Foam Processes and High and Low Foam Qualities," paper SPE 59287, presented at the SPE/DOE Symposium on Improved Oil Recovery, Tulsa, Oklahoma, USA (April 3-5, 2000).
21. Cheshire, I. M. and Pollard, R. "Recent Advances in Nested Factorization," Proceedings of the Cambridge Conference on Mathematical Techniques for the Oil Industry (1987).
22. Clifford, P.J., and Sorbie, K.S., "The Effect Of Chemical Degradation on Polymer Flooding," paper SPE 13586, presented at the SPE Oilfield and Geothermal Chemistry Symposium, Phoenix, Arizona, USA (March 9-11 1985).
23. Coats, K. H. "An Equation of State Compositional Model," paper SPE 8284, *Society for Petroleum Engineers Journal* (1980) 20, 363-376; also presented at the SPE 54th Annual Technical Conference and Exhibition, Las Vegas, Nevada, USA (September 23-26, 1979).
24. Coats, K. H. "In-Situ Combustion Model," *Society of Petroleum Engineers Journal* (1980) 533-554.
25. Coats, K. H. "Simulation of Gas Condensate Reservoir Performance," paper SPE 10512, presented at the Sixth SPE Symposium on Reservoir Simulation, New Orleans, Louisiana, USA (February 1-3 1982).
26. Coats, K.H. "IMPES Stability: The Stable Step," paper SPE 69225, presented at the SPE Reservoir Simulation Symposium, Houston, Texas, USA (February 11-14 2001).
27. Cook, R. E., Jacoby, R. H., and Ramesh, A. B. "A Beta-type Reservoir Simulator for Approximating Compositional Effects During Gas Injection" paper SPE 4272, *Society of Petroleum Engineers Journal* (1974) 14, No. 5, 471-481.
28. Crookston, H. B., Culham, W. E., and Chen, W. H. "Numerical Simulation Model for Thermal Recovery Processes," *Society of Petroleum Engineers Journal*, Transactions of AIME, (1979) 267, 37-58.
29. Crowe, C. M. and Nishio, M. "Convergence Promotion in the Simulation of Chemical Processes - The General Dominant Eigenvalue Method," *American Institute of Chemical Engineering Journal* (1975) 23, No. 3, 528-533.
30. Dake, L.P. *Fundamentals of Reservoir Engineering*, Amsterdam, The Netherlands, Elsevier Science BV (1978) 256.

31. de Montleau, P., Cominelli, A., Neylon, K., Rowan, D., Pallister, I., Tesaker, O., and Nygard I. “*Production Optimization under Constraints Using Adjoint Gradients*,” Presented at 10th European Conference on the Mathematics of Oil Recovery, Amsterdam, The Netherlands, (September 4-7, 2006).
32. Dennis, J. E. and Schnabel, R. B. *Numerical Methods for Unconstrained Optimisation and Nonlinear Equations*, Classics in Applied Mathematics (unabridged and corrected ed.) Society for Industrial and Applied Mathematics (1996), first published by Prentice-Hall, Englewood Cliffs, New Jersey, USA (1983).
33. Edmister, W. C. and Lee, B. I. *Applied Hydrocarbon Thermodynamics, Volume 1*, (2nd ed.), Houston, Texas, USA, Gulf Publishing Company (1984).
34. Ejiogu, G. C. and Fiori, M. “High-Pressure Saturated-Steam Correlations,” *Journal of Petroleum Technology* (1987) 39, No. 12, 1585-1590.
35. Ewing, R. E., Boyett, B. A., Babu, D. K., and Heinemann, R. F. “Efficient Use of Locally Refined Grids for Multiphase Reservoir Simulation,” paper SPE 18413, *Proceedings of the Tenth SPE Symposium on Reservoir Simulation*, (1989) 55-70; also presented at the SPE Symposium on Reservoir Simulation, Houston, Texas, USA, (February 6-8, 1989).
36. Farouq Ali, S. M. *Oil Recovery by Steam Injection*, Bradford, PA, USA, Producers Publishing Co., (1970).
37. Fenghour, A., Wakeham, W.A., and Vesovic, V. “The Viscosity of Carbon Dioxide,” *Journal of Physical and Chemical Reference Data* (1998) 27, No.1, 31-44.
38. Fetkovich, M. J. “A Simplified Approach to Water Influx Calculations - Finite Aquifer Systems,” *Journal of Petroleum Technology*, (1971) 23, No. 7, 814-828.
39. Garmeh, R., Izadi, M., Salehi, M., Romero, J. L., Thomas, C. P. and Marque, E. J. “Thermally Active Polymer to Improve Sweep Efficiency of Water Floods: Simulation and Pilot Design Approaches”, paper SPE 144234: presented at the SPE Enhanced Oil Recovery Conference, Kuala Lumpur, Malaysia (July 19-21, 2011).
40. Gill, P.E., Murray, W., and Wright, M.H. *Practical Optimization*, London and New York, Academic Press (1981).
41. Gilman, J. R. and Kazemi, H. “Improved Calculation for Viscous and Gravity Displacement in Matrix Blocks in Dual-Porosity Simulators,” paper SPE 16010 (includes a number of associated papers), *Journal of Petroleum Technology* (1988) 40, No. 1, 60-70.
42. Grabowski, J. W. and Rubin, B. “A Preliminary Numerical Simulation Study of In-situ Combustion in a Cold Lake Oil Sands Reservoir,” *Journal of Canadian Petroleum Technology* (1981) 20, No. 2, 79-89.
43. Gunasekera, D., Childs, P., Cox, J., and Herring, J. “A Multi-Point Flux Discretization Scheme for General Polyhedral Grids,” paper SPE 48855, presented at the International Oil & Gas Conference and Exhibition, Beijing, China (November 2-6, 1998).
44. Haaland, S.E. “Simple and explicit formulas for the friction factor in turbulent pipe flow,” *Transactions of the ASME, Journal of Fluids Engineering* (1983) 105, No. 1, 89-90.
45. Hagedorn, A. R. and Brown, K. E. “Experimental Study of Pressure Gradients Occurring During Continuous Two-Phase Flow in Small-Diameter Vertical Conduits,” *Journal of Petroleum Technology* (1965) 17, No. 4, 475-484.
46. Han G., Dusseault, M. “Description of fluid flow around a wellbore with stress-dependent porosity and permeability,” *Journal of Petroleum Science and Engineering* (2003) 40, No. 1, 1-16.

47. Harmathy, T. Z. "Velocity of Large Drops and Bubbles in Media of Infinite and Restricted Extent," *American Institute of Chemical Engineering Journal* (1960) 6, No. 2, 281-288.
48. Hasan, A. R. and Kabir, C. S. "A Simplified Model for Oil/Water Flow in Vertical and Deviated Wellbores," paper SPE 54131, *SPE Production & Facilities* (1999) 14 No. 1, 56-62; also presented at the SPE Annual Technical Conference and Exhibition, New Orleans, Louisiana, USA, (September 27-30, 1998).
49. Hasan, A. R. and Kabir, C. S. "A Study of Multiphase Flow Behavior in Vertical Wells," paper SPE 15138, *SPE Production Engineering* (1988) 3, No. 2, 263-272.
50. Haywood, R. W. *Thermodynamic Tables in SI (Metric) Units* (3rd ed.), Cambridge, UK, Cambridge University Press (1990).
51. Hegre, T. M., Dalen, V., and Henriquez, A. "Generalised Transmissibilities for Distorted Grids in Reservoir Simulation," paper SPE 15622, presented at the SPE Annual Technical Conference and Exhibition, New Orleans, Louisiana, USA (October 5-8, 1986).
52. Henderson, G.D., Danesh, A., Tehrani, D.H., and Al-Kharusi, B. "The Relative Significance of Positive Coupling and Inertial Effects on Gas Condensate Relative Permeabilities at High Velocity," paper SPE 62933, presented at the SPE Annual Technical Conference and Exhibition, Dallas, Texas, USA (October 1-4 2000).
53. Hibbit, Karlsson & Sorensen, Inc. *ABAQUS: Theory Manual*, Hibbit, Karlsson & Sorensen, Inc. (1998).
54. Holmes, J. A. "Enhancements to the Strongly Coupled, Fully Implicit Well Model: Wellbore Crossflow Modeling and Collective Well Control," paper SPE 12259, presented at the SPE Symposium on Reservoir Simulation, San Francisco (November 15-18, 1983).
55. Holmes, J.A., Barkve, T., and Lund, Ø. "Application of a Multisegment Well Model to Simulate Flow in Advanced Wells," paper SPE 50646, presented at the SPE European Petroleum Conference, The Hague, The Netherlands, (October 20-22, 1998).
56. Hustad, O.S. "A Coupled Model for Three-Phase Capillary Pressure and Relative Permeability," paper SPE 74705, *Society of Petroleum Engineers Journal* (2002) 7, No. 1, 59-69.
57. Hustad, O. S. and Browning, D. J. "A Fully Coupled Three-Phase Model for Capillary Pressure and Relative Permeability for Implicit Compositional Reservoir Simulation," paper SPE 125429, presented at the SPE/EAGE Reservoir Characterization and Simulation Conference, Abu Dhabi, UAE (October 19-21, 2009).
58. Hustad, O.S. and Hansen, A.G. "A Consistent Formulation for Three-Phase Relative Permeabilities and Phase Pressures based on Three Sets of Two-Phase Data," *RUTH: A Norwegian Research Program on Improved Oil Recovery, Program Summary, Norwegian Petroleum Directorate* (1996) 183-194; also in Proceedings of the Eighth European Symposium on Improved Oil Recovery, (1995) 1 289.
59. Hustad, O. S. and Holt, T. "Gravity Stable Displacement of Oil by Hydrocarbon Gas After Waterflooding," paper SPE 24116, presented at the SPE/DOE Eighth Symposium on Enhanced Oil Recovery, Tulsa, Oklahoma, USA (April 22-24 1992).
60. Hustad, O.S., Kløv, T., Lerdahl, T. R., Berge, L.I., Stensen, J., Åren, P-E. "Gas Segregation During WAG Injection and the Importance of Parameter Scaling in Three-Phase Models," paper SPE 75138, presented at the SPE /DOE Improved Oil Recovery Symposium, Tulsa, Oklahoma, USA (April 13-17, 2002).

61. Kazemi, H., Merrill JR., L. S., Porterfield, K. L., and Zeman, P. R. "Numerical Simulation of Water-Oil Flow in Naturally Fractured Reservoirs," paper SPE 5719, *Society of Petroleum Engineers Journal* (1976) 16, No. 6, 317-326.
62. Kell, G.S and Whalley, E. "Reanalysis of the density of liquid water in the range 0-150 C and 0-1 kbar," *Journal of Chemical Physics* (1975), 62 No. 9, 3496-3503.
63. Kenyon, D. E. and Behie, A. "Third SPE Comparative Solution Project: Gas Cycling of Retrograde Condensate Reservoirs," paper SPE 12278, *Journal of Petroleum Technology* (1987) 39, No. 8, 981-997.
64. Killough, J. E. "Reservoir Simulation with History-dependent Saturation Functions," paper SPE 5106, *Society of Petroleum Engineers Journal* (1976) 16, No. 1, 37-48.
65. Kovscek, A.R. "Reservoir Simulation of Foam Displacement Processes," Presented at the 7th UNITAR International Conference on Heavy Crudes and Tar Sands, Beijing, China (October 27-31, 1998).
66. Land, C. E. "Calculation of Imbibition Relative Permeability for Two- and Three-Phase Flow from Rock Properties," paper SPE 1942, *Society of Petroleum Engineers Journal* (1968) 8, No. 2, 149-156.
67. Landau, L.D. and Lifshitz, E. M. *Theory of Elasticity: Volume 7 (Course of Theoretical Physics)* (3rd ed.), Oxford, UK, Butterworth-Heinemann (1984).
68. Leibovici, C.F., Barker, J.W., and Waché, D. "A Method for Delumping the Results of a Compositional Reservoir Simulation," paper SPE 49068, presented at the SPE Annual Technical Conference & Exhibition, New Orleans, Louisiana, USA (September 27-30, 1998).
69. Liu, Q., Stone, T., Han, G., Marsden, R., and Shaw, G. "Coupled Stress and Fluid Flow Using a Finite Element Method in a Commercial Reservoir Simulator," paper SPE 88616, presented at the SPE Asia Pacific Oil and Gas Conference, Perth, Australia, (October 18-20, 2004).
70. Lorentz, J., Bray, B. G., and Clark, C. R. J. "Calculating Viscosity of Reservoir Fluids from their Composition," *Journal of Petroleum Technology* (1964) 1171, 231
71. Martin, J. J. "Cubic Equations of State-Which?" *I and EC Fundamentals*, (1973) 18, 81.
72. Mavko, G., Mukerji, T., and Dvorkin, J. *The Rock Physics Handbook: Tools for Seismic Analysis in Porous Media*, Cambridge UK, Cambridge University Press (1998).
73. Mavor, M. J. and Gunter, W. D. "Secondary Porosity and Permeability of Coal vs. Gas Composition and Pressure," paper SPE 90255, *SPE Reservoir Evaluation & Engineering* (2006) 9, No. 2, 114-125.
74. Michelsen, M. L. "The isothermal flash problem. Part I. Stability," *Fluid Phase Equilibria* (1982) 9, 1-19.
75. Mott, R. "Calculating Well Deliverability in Gas Condensate Reservoirs," Presented at the 10th European Symposium on Improved Oil Recovery, Brighton, UK (August 18-20 1999).
76. Nghiem, L. X., Collins D. A., and Sharma, R. "Seventh SPE Comparative Solution Project: Modeling of Horizontal Wells in Reservoir Simulation," paper SPE 21221, presented at the SPE Symposium on Reservoir Simulation, Anaheim, California, USA (February 17-20, 1991).
77. Pal, R. and Rhodes, E. "Viscosity/Concentration Relationships for Emulsions," *Journal of Rheology* (1989) 33, No. 7, 1021-1045.
78. Palmer, I. and Mansoori, J. "How Permeability Depends on Stress and Pore Pressure in Coalbeds: A New Model," paper SPE 52607, *SPE Reservoir Evaluation & Engineering* (1998) 1, No. 6, 539-544.

79. Patillo, P.D., Kristiansen, T.G., Sund, G.V., and Kjelstadli, R.M. "Reservoir Compaction and Seafloor Subsidence at Valhall," paper SPE/ISRM 47274, presented at the Eurock '98 meeting, Trondheim, Norway (July 8-10, 1998).
80. Pedersen, K. S., Fredenslund, Aa., and Thomassen, P. *Properties of Oils and Natural Gases*, Contributions in Petroleum Geology and Engineering, Vol. 5, Houston, Texas, USA, Gulf Publishing Company (1989).
81. Pedrosa, O. A. and Aziz, K. "Use of Hybrid Grid in Reservoir Simulation," paper SPE 13507, *Society of Petroleum Engineers Journal* (1986) 1, No. 6, 611-621.
82. Perry, R. H. and Green, D.W. *Perry's Chemical Engineers' Handbook* (6th ed.), USA, McGraw Hill (1984).
83. Pierre Lebel, J. "Dynamic Fracture Modelling Approach for Cold Lake Cyclic Steam Stimulation," paper SPE 79010, presented at the SPE International Thermal Operations and Heavy Oil Symposium and International Horizontal Well Technology Conference, Calgary, Canada (November 4-7, 2002).
84. Ponting, D. K. "Corner Point Geometry in Reservoir Simulation" Proceedings of the Joint IMA/SPE European Conference on the Mathematics of Oil Recovery, Cambridge University, UK (July 25-27, 1989).
85. Ponting, D. K., Banks, D., Foster, B. A., Naccache, P. F., Nicholas, M. O., Pollard, R. K., Rae, J., and Walsh, S. K. "An Efficient Fully Implicit Simulator," *Society of Petroleum Engineers Journal* (1983) 23, 544-552.
86. Potter II, R. W., Babcock, R. S., and Brown, D. L. "A new method for determining the solubility of salt in aqueous solutions at elevated temperatures," *Journal of Research of the U.S. Geological Survey* (1977) 5, No. 3, 389-395.
87. Prats, M. *Thermal Recovery*, SPE Monograph Series (Vol. 7) Society of Petroleum Engineers (1982).
88. Pritchett, J., Frampton, H., Brinkman, J., Cheung, S., Morgan, J., Chang, K. T., Williams, D. and Goodgame J.: "Field Application of a New In-Depth Waterflood Conformance Improvement Tool", paper SPE 84897, presented at the *SPE International Improved Oil Recovery Conference in Asia Pacific, Kuala Lumpur, Malaysia* (October 20-21 2003).
89. Quandalle, P. and Sabathier, J. C. "Typical Features of a Multipurpose Reservoir Simulator," paper SPE 16007, *SPE Reservoir Engineering* (1989) 4, No. 4, 475-480.
90. Reid, R.C., Prausnitz, J. M. and Polling, B. E. *The Properties of Gases and Liquids*, New York, USA, McGraw-Hill (1987).
91. Rozon, B. J. "A Generalized Finite Volume Discretization Method for Reservoir Simulation," paper SPE 18414, presented at the SPE Symposium on Reservoir Simulation, Houston, Texas, USA (February 6-8 1989).
92. Rubin, B. and Blunt, M. J. "Higher-Order Implicit Flux Limiting Schemes for Black Oil Simulation," paper SPE 21222, presented at the SPE Symposium on Reservoir Simulation, Anaheim, California, USA (February 17-20, 1991).
93. Saad, Y. *Iterative Methods for Sparse Linear Systems*, Boston, USA, PWS Publishing Company (1996).
94. Schlumberger *VISAGE Manual*, Schlumberger 2008.
95. Shaw, G., and Stone, T. "Finite Volume Methods for Coupled Stress/Fluid Flow in a Commercial Reservoir Simulator," paper SPE 93430, presented at the SPE Reservoir Simulation Symposium, The Woodlands, Texas, USA (January 31 - February 2, 2005).

96. Shi, H., Holmes, J.A., Diaz, L. R., Durlofsky, L. J., and Aziz, K. "Drift-Flux Parameters for Three-Phase Steady-State Flow in Wellbores," paper SPE 89836, *Society of Petroleum Engineers Journal* (2005) 10, No. 2, 130-137; also presented at the SPE Annual Technical Conference and Exhibition, Houston, Texas, USA (September 26-29, 2004).
97. Shi, H., Holmes, J.A., Durlofsky, L. J., Aziz, K., Diaz, L. R., Alkaya, B., and Oddie, G. "Drift-Flux Modeling of Two-Phase Flow in Wellbores," paper SPE 84228, *Society of Petroleum Engineers Journal* (2005) 10, No. 1, 24-33; also presented as "Drift-Flux Modeling of Multiphase Flow in Wellbores," at the SPE Annual Technical Conference and Exhibition, Denver, Colorado, USA (October 5-8, 2003).
98. Shiralkar, G. A., and Stephenson, R. E. "A General Formalism for Simulating Physical Dispersion and a New Nine-Point Scheme," paper SPE 16975, *SPE Reservoir Engineering* (1991) 6, No. 1, 115-120.
99. Singh, K. and Whitson, C.H. "Gas Condensate Pseudopressure in Layered Reservoirs," paper SPE 117930, presented at the Abu Dhabi International Petroleum Exhibition and Conference, Abu Dhabi, UAE, (November 3-6, 2008).
100. Sokolnikoff, I.S. and Redheffer, R.M. *Mathematics of Physics and Modern Engineering*, New York, USA, McGraw-Hill (1967).
101. Soreide, I., and Whitson, C.H. "Peng Robinson predictions for hydrocarbons, CO₂, N₂ and H₂S with pure water," *Fluid Phase Equilibria* (1992) 77, 217-240.
102. Spycher, N. and Pruess, K. "CO₂-H₂O mixtures in the geological sequestration of CO₂.II. Partitioning in chloride brines at 12-100 C and up to 600 bar," *Geochimica et Cosmochimica Acta* (2005) 69, No. 13, 3309-3320.
103. Spycher, N. and Pruess, K. "A Phase-Partitioning Model for CO₂-Brine Mixtures at Elevated Temperatures and Pressures: Application to CO₂-Enhanced Geothermal Systems," *Transport Porous Media* (2009) 82, No. 1, 173-196.
104. StatoilHydro ASA "Method and Apparatus for Determining Capillary Pressures in a Three Phase Fluid Reservoir," United States Patent Application No. 12/266307 (May 6, 2010).
105. Stone, H. L. "Estimation of Three-Phase Relative Permeability and Residual Oil Data," *Journal of Canadian Petroleum Technology* (1973) 12, No. 4, 53-61.
106. Stone, H. L. "Probability Model for Estimating Three-Phase Relative Permeability," paper SPE 2116, *Journal of Canadian Petroleum Technology* (1973) 22, No. 2, 214-218.
107. Stone, T.W., Bennett, J., Law, D.H.-S., and Holmes, J.A. "Thermal Simulation with Multisegment Wells," paper SPE 78131, *SPE Reservoir Evaluation & Engineering* (2002) 5, No. 3, 206-218.
108. Stone, T., Bowen, G., Papanastasiou, P., and Fuller, J. "Fully Coupled Geomechanics in a Commercial Reservoir Simulator," paper SPE 65107, presented at the SPE European Petroleum Conference, Paris, France (October 23-24, 2000).
109. Stone, T.W., Xian, C., Fang, Z., Manalac, E., Marsden, R., and Fuller, J. "Coupled Geomechanical Simulation of Stress Dependent Reservoirs," paper SPE 79697, presented at the SPE Reservoir Simulation Symposium, Houston, Texas, USA (February 3-5, 2003).
110. Stüben, K. *SAMG USER's Manual Release 21b1*, St. Augustin, Germany, Fraunhofer Institute (2002).
111. Todd, M. and Longstaff, W. "The Development, Testing and Application of a Numerical Simulator for Predicting Miscible Flood Performance," paper SPE 3484, *Journal of Canadian Petroleum Technology* (1972) 24, No. 7, 874-882.

112. Tortike, W. S. and Farouq Ali, S. M. "Saturated Steam Property Functional Correlations for Fully Implicit Thermal Reservoir Simulation," paper SPE 17094, *Society of Petroleum Engineers Journal* (1989) 4, No. 4, 471-474.
113. Trangenstein, J. A. "Analysis of a Model and Sequential Numerical Method for Thermal Reservoir Simulation," *The Mathematics of Oil Recovery*, King, P.R. (ed.), Oxford, UK, Clarendon Press (1992), 359.
114. Valle, A. "Three Phase Gas-Oil-Water Pipe Flow," Ph.D. Thesis, Imperial College, London, UK (2000)
115. Van Everdingen A.F. and Hurst W. "The Application of the Laplace Transformation to Flow Problems in Reservoirs," paper SPE 949305, *Petroleum Transactions, AIME* (1949) 186, 305-324.
116. van Lingen, P., Sengul, M., Daniel, J. and Cosentino, L. "Single Medium Simulation of Reservoirs with Conductive Faults and Fractures," paper SPE 68165, presented at the SPE Middle East Oil Show, Bahrain (March 17-20, 2001).
117. Vesovic, V., Wakeham, W. A., Olchowy, G.A., Sengers, J.V., Watson, J.T.R. and Millat, J. "The Transport Properties of Carbon Dioxide," *Journal of Physical and Chemical Reference Data* (1990) 19, No.3, 763.
118. Vinsome, P. K. W. "Orthomin, an Iterative Method for Solving Sparse Banded Sets of Simultaneous Linear Equations," paper SPE 5729, presented at the SPE Symposium on Reservoir Simulation, Los Angeles, USA (February 19-20, 1976).
119. Vinsome, P.K.W. and Westerveld, J. "A simple method for predicting cap and base rock heat losses in thermal reservoir simulators," *Journal of Canadian Petroleum Technology* (1980) 19, No. 3, 87-90.
120. Wallis, G. B. *One-dimensional Two-phase Flow*, New York, USA, McGraw-Hill (1969).
121. Wallis, G. B. and Makkenchery, S. "The Hanging Film Phenomenon in Vertical Annular Two-phase Flow," *Journal of Fluids Engineering* (1974) 96, No. 3, 297-298.
122. Wallis, J. R. "Incomplete Gaussian Elimination as a Preconditioning for Generalised Conjugate Gradient Acceleration," paper SPE 12265, presented at the SPE Symposium on Reservoir Simulation, San Francisco, USA (November 15-18, 1983).
123. Wallis, J. R., Kendall, R. P., and Little, T. E. "Constrained Residual Acceleration of Conjugate Residual Methods," paper SPE 13536, presented at the SPE Symposium on Reservoir Simulation, Dallas, Texas, USA (February 10-13, 1985).
124. Weinstein, H.G., Chappellear, J.E., and Nolen, J.S. "Second Comparative Solution Project: A Three-Phase Coning Study," paper SPE 10489, *Journal of Petroleum Technology* (1986) 38, No. 3, 345-353.
125. Welty, J.R., Wicks, C.E., and Wilson, R.E. *Fundamentals of Momentum Heat and Mass Transfer*, New Jersey, USA, John Wiley and Sons (1984).
126. Whitson, C. H. and Fevang, Ø. "Generalised Pseudopressure Well Treatment in Reservoir Simulation," Presented at the IBC Technical Services Conference on Optimisation of Gas Condensate Fields, Aberdeen, UK (June 26-27, 1997).
127. Whitson, C. H. and Fevang, Ø. "Modeling Gas Condensate Well Deliverability," paper SPE 30714, *SPE Reservoir Engineering* (1996) 11, No. 4, 221-230; also presented at the SPE Annual Technical Conference and Exhibition, Dallas, Texas, USA (October 22-25, 1995).
128. Whitson, C.H., Fevang, O., and Saevareid, A. "Gas Condensate Relative Permeability for Well Calculations," paper SPE 56476, presented at the SPE Annual Technical Conference, Houston, Texas, USA (October 3-6 1999).

129. Wilson, G. "A Modified Redlich-Kwong EOS, Application to General Physical Data Calculations," paper 15C, presented at the American Institute of Chemical Engineers National Meeting, Cleveland, Ohio, USA (May 1968).
130. Wingard, J. S. "Multicomponent Multiphase Flow in Porous Media with Temperature Variation," PhD thesis, Stanford University, Palo Alto, California, USA (1988).
131. Yanosik, J. L. and McCracken, T. A. "A Nine-Point, Finite-Difference Reservoir Simulator for Realistic Prediction of Adverse Mobility Ratio Displacements," paper SPE 5734, *Society of Petroleum Engineers Journal* (1979) 19, No. 4, 253-262.
132. Zakirov, I.S., Aanonsen, S.I., Zakirov, E.S., and Palatnik, B.M. "Optimizing Reservoir Performance by Automatic Allocation of Well Rates," Presented at the European Conference on the Mathematics of Oil Recovery, Leoben, Austria (September 3-6 1996).
133. Zaytsev, I.D. and Aseyev, G.G. *Properties of Aqueous Solutions of Electrolytes*, Boca Raton, Florida, USA CRC Press (1993).
134. Zimmerman, R.W. *Compressibility of Sandstones*, New York, USA Elsevier Science (1991).
135. Zuber, N. and Findlay, J. A. "Average Volumetric Concentration in Two-Phase Flow Systems," *Transactions of ASME, Ser. C, Journal of Heat Transfer* (1965) 87, No. 4, 453-468.
136. Clarkson, C.R., Pan, Z., Palmer, I. and Harpalani, S. "Predicting Sorption-Induced Strain and Permeability Increase With Depletion for Coalbed-Methane Reservoirs", SPE 114778-PA, *SPE Journal* (2010) 15, No. 1, 152–159.
137. Sheng J. "Modern Chemical Enhanced Oil Recovery. Theory and Practice." Gulf Professional Publishing. Elsevier (2011).
138. Russell, D.G., Goodrich, J.H., Perry, G.E and Bruskotter, J.F "Methods of Predicting Gas Well Performance", *Transactions of the ASME, Journal of Petroleum Technology* (1966) 99-108.
139. Wallis, J. R., Little, T. E., and Nolen, J. S.: "Constrained Residual Acceleration of Conjugate Residual Methods," paper SPE 13536 presented at the SPE Reservoir Simulation Symposium, Dallas, Texas, USA (February 10-13, 1985).