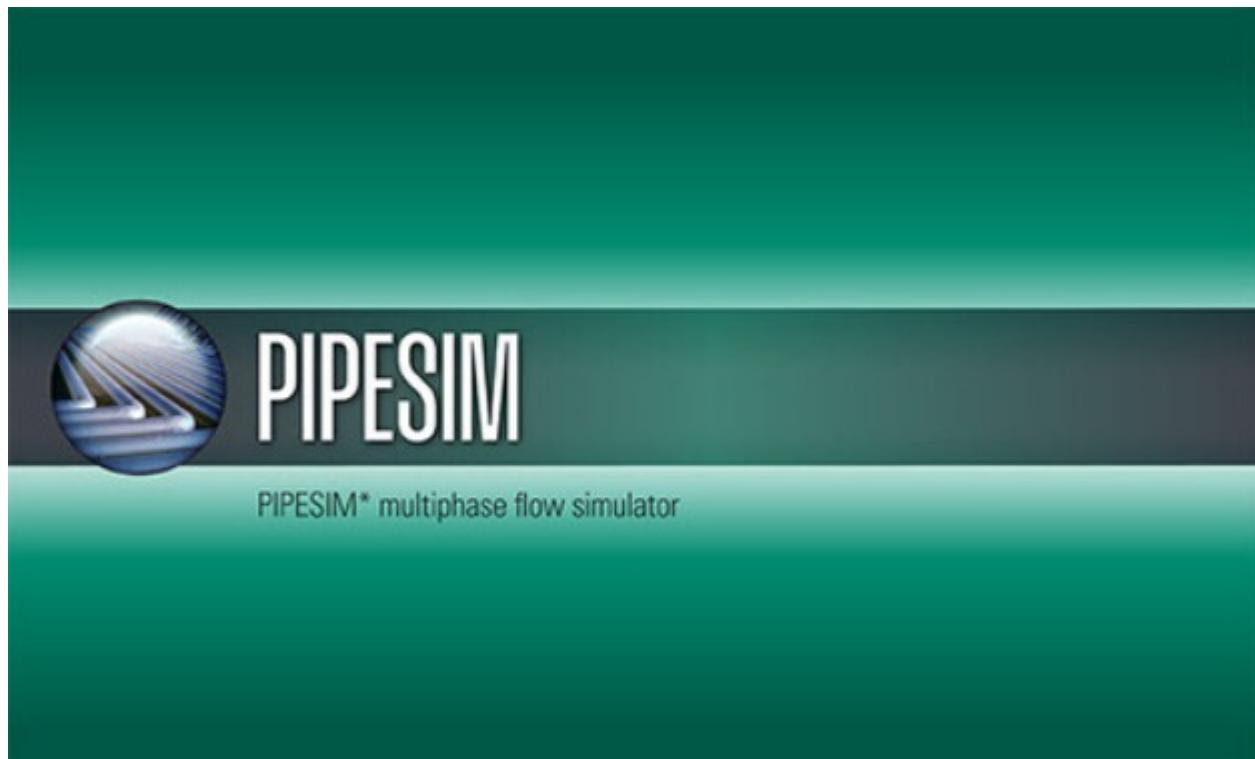


PIPESIM

Version 2012.2.1



User Guide

Schlumberger

No part of this manual may be reproduced, stored in a retrieval system, or translated in any form or by any means, electronic or mechanical, including photocopying and recording, without the prior written permission of Schlumberger Information Solutions, 5599 San Felipe, Suite 1700, Houston, TX 77056-2722, USA.

Use of this product is governed by the License Agreement. Schlumberger makes no warranties, express, implied, or statutory, with respect to the product described herein and disclaims without limitation any warranties of merchantability or fitness for a particular purpose. Schlumberger reserves the right to revise the information in this manual at any time without notice.

The software described herein is configured to operate with at least the minimum specifications set out by Schlumberger. You are advised that such minimum specifications are merely recommendations and not intended to be limiting to configurations that may be used to operate the software. Similarly, you are advised that the software should be operated in a secure environment whether such software is operated across a network, on a single system and/or on a plurality of systems. It is up to you to configure and maintain your networks and/or system(s) in a secure manner. If you have further questions as to recommendations regarding recommended specifications or security, please feel free to contact your local Schlumberger representative.

Copyright 2015 Schlumberger. All rights reserved.

Table of Contents

1	User Guide	1
1.1	Introduction	1
1.1.1	Company profile	1
1.1.2	Getting Started	1
<i>Applications</i>	2	
1.1.3	Reporting	22
1.1.4	Extensibility	22
1.1.5	PIPESIM Hot Keys	22
<i>File Hot Keys</i>	22	
<i>Simulation Hot Keys</i>	23	
<i>Windows Hot Keys</i>	23	
<i>Tools Hot Keys</i>	23	
<i>Editing or General Hot Keys</i>	23	
1.1.6	Main Toolbar	24
1.1.7	Toolbox	25
<i>Single branch Toolbox</i>	25	
<i>Network Toolbox</i>	27	
1.1.8	Wizard Feature	28
<i>Steps</i>	28	
1.1.9	Find	29
<i>Finding an object in a network model</i>	29	
1.1.10	PIPESIM Differences from other Simulators	29
1.1.11	PIPESIM versions	30
<i>PIPESIM Suite (Build26) to PIPESIM Conversion</i>	30	
1.2	Building Models	33
1.2.1	Steps in building a model	33
<i>Basic overview</i>	33	
<i>Creating a new model</i>	33	
<i>Adding objects to a model and connecting them</i>	34	
<i>Adding data to an object</i>	34	
<i>Minimum data</i>	34	
<i>Duplicating an object</i>	34	
<i>Disconnecting objects</i>	34	
<i>Saving the model</i>	35	

1.2.2	Network Models	35
<i>Creating a network model</i>	35	
<i>Network Operations</i>	35	
<i>Source</i>	35	
<i>Sink</i>	36	
<i>Branch</i>	37	
<i>Network Separator</i>	38	
<i>Production Well</i>	38	
<i>Injection Well</i>	39	
<i>Well Curves</i>	40	
<i>Boundary Conditions</i>	41	
<i>Network Constraints</i>	42	
<i>Network Estimates</i>	42	
<i>Optimizing PIPESIM Network Simulation Performance</i>	43	
1.2.3	Single Branch Models	50
<i>Well Performance Analysis Overview</i>	50	
<i>Pipeline tools</i>	91	
<i>Source, Sink and Boundary Conditions</i>	118	
1.3	Flow Correlations	120
1.3.1	Selecting a flow correlation	120
<i>Source</i>	120	
<i>Correlation</i>	121	
<i>Options Control</i>	121	
1.3.2	Defining User Flow Correlations	121
<i>Tuning flow correlations</i>	121	
<i>Data matching</i>	121	
<i>Flow Correlation comparison</i>	122	
<i>Suggested correlations</i>	122	
1.3.3	Flow Regime Number and Flow Pattern	123
1.3.4	Configuration File for user specified flow correlations	125
1.4	Fluid modeling	127
1.4.1	Creating or Editing Fluid Models	127
1.4.2	Modeling systems	128
<i>Water Systems</i>	128	
<i>Gas Condensate Systems</i>	129	
<i>Dry Gas Systems</i>	129	
1.4.3	Managing Multiple Fluid Models	0
<i>Precedence of Fluids when Multiple Fluids Defined</i>	132	
<i>Local Fluid Data</i>	133	
1.4.4	Ensuring consistency among multiple fluid files in a PIPESIM network model	134

1.4.5	Black Oil Fluid Modeling	134
	<i>Introduction</i>	134
	<i>Black Oil Data Sets</i>	135
	<i>Black Oil Options</i>	136
	<i>Calibrate a Fluid to match lab data</i>	136
	<i>Bubble point calibration data</i>	137
	<i>Gas Phase Contaminants</i>	140
	<i>Black Oil Thermal Data</i>	140
1.4.6	Compositional Fluid Modeling	141
	<i>Compositional Template</i>	141
	<i>Specifying the Composition</i>	148
	<i>Compositional using MFL file</i>	153
	<i>Multiflash in the Compositional Fluid mode (native) vs. Multiflash MFL files</i>	127
	<i>PVT Files</i>	157
	<i>Solids Modeling</i>	161
	<i>Compositional Flashing</i>	164
	<i>Stock tank Conditions</i>	166
	<i>Steam</i>	166
1.4.7	Converting Black Oil Models into Compositional Models using Multiflash for PIPESIM*	167
1.5	Options	170
1.5.1	Common Options	170
	<i>Project Data</i>	170
	<i>Units</i>	170
	<i>Heat Transfer Options</i>	171
	<i>Engine Options</i>	172
	<i>Output Options</i>	173
	<i>Erosion and Corrosion Options</i>	176
	<i>Engine Preferences</i>	178
	<i>Parallelized Network Solver</i>	178
1.5.2	Network Options	180
	<i>Network Iterations</i>	180
	<i>Fluid Models</i>	181
1.5.3	View Options	182
	<i>View Properties tab</i>	182
	<i>Objects tab</i>	183
	<i>General Tab</i>	183
	<i>Viewing a large model</i>	184
	<i>Viewing flowline data</i>	184
1.6	Database	184

1.6.1	New ESP/Pump/Compressor Curve	184
	<i>ESP</i>	185
	<i>Generic Pump</i>	185
	<i>Compressor/Expander</i>	185
	<i>New Reciprocating Compressor Curve</i>	185
	<i>User Curves</i>	186
	<i>Electrical Submersible Pumps (ESP) Selection</i>	187
1.6.2	New PCP Curve	190
	<i>Creating a New Curve</i>	190
	<i>Selecting a PCP from the Catalog</i>	191
	<i>Standard PCP Curves</i>	191
	<i>Progressive Cavity Pump (PCP) Selection</i>	192
1.6.3	Gas Lift Valve List	193
1.6.4	Flow Control Valves	194
	<i>How to add an FCV to a completion</i>	194
1.7	Operations	195
1.7.1	Common Operations	195
	<i>Check model</i>	195
	<i>Run Model</i>	195
	<i>Abort Run</i>	195
	<i>Terminate Run</i>	195
1.7.2	Single Branch	196
	<i>System Analysis</i>	196
	<i>Pressure/Temperature profile</i>	196
	<i>Flow correlation comparison</i>	197
	<i>Data matching</i>	198
	<i>Nodal Analysis</i>	200
	<i>Horizontal length</i>	202
	<i>Reservoir tables</i>	202
	<i>Well Performance Curves</i>	203
	<i>Gas Lift rate vs Casing head pressure</i>	204
	<i>Artificial Lift System Performance curves</i>	205
	<i>Wax Deposition</i>	205
	<i>Sensitivity variables</i>	211
	<i>User variable</i>	211
1.7.3	Network Operations	212
	<i>Restart Model</i>	212
	<i>Well Optimization</i>	213
1.8	Artificial Lift analysis	213
1.8.1	How to create artificial lift performance curves	213

1.8.2	General Gas Lift	214
	<i>Gas Lift Operations - Deepest Injection Point</i>	215
	<i>Gas Lift Operations - Bracketing</i>	215
	<i>Gas Lift Operations - Lift Gas Response Curves</i>	216
	<i>Gas Lift Design</i>	217
	<i>Gas Lift Diagnostics</i>	230
	<i>Gas Lift Theoretical background</i>	232
	<i>Gas lift instability</i>	232
	<i>Annular Gas Pressure Design Method</i>	234
1.8.3	Electrical Submersible Pumps (ESP)	235
	<i>General</i>	235
	<i>ESP System Components: Motor</i>	235
	<i>ESP System Components: Pumps</i>	236
	<i>Motor Selection</i>	239
	<i>ESP Database</i>	239
	<i>Selection</i>	239
	<i>Stage-by-stage modeling</i>	240
	<i>Install a Pump</i>	240
	<i>ESP Design</i>	240
	<i>ESP System Components: Cable</i>	241
	<i>Errors</i>	241
	<i>ESP Design</i>	241
	<i>ESP / Compressor / Pump data entry</i>	242
1.8.4	Rod Pump Module	242
	<i>About the Rod Pump Module</i>	242
	<i>Getting Started</i>	243
	<i>RP Operation Menu</i>	244
	<i>Components and Options</i>	250
	<i>Rod Pump Design</i>	251
	<i>Selection of Design Methods</i>	252
	<i>Rod Pump Diagnostics</i>	253
	<i>Operations</i>	253
	<i>Rod Pump Database Configuration</i>	253
	<i>Open Project</i>	253
	<i>Basis Input Data in PIPESIM</i>	253
	<i>RP Application from PIPESIM Drop-down Menu</i>	253
	<i>Optimization Design</i>	254
	<i>Operation Diagnostics</i>	255
	<i>Database</i>	255
1.9	Reports	256
1.9.1	Plotting	256
	<i>Single branch plots</i>	256
	<i>Network plots</i>	256
	<i>PsPlot</i>	256
1.9.2	Report Tool Details	115

1.9.3	Network Report Tool	262
<i>Exporting to Excel</i>	264	
1.10	Expert Mode	264
1.10.1	Expert Mode	264
<i>Calculation engine</i>	265	
<i>Network models</i>	265	
<i>Single branch models</i>	265	
1.10.2	How to work in Expert mode	265
1.10.3	Batch mode	265
<i>How to batch run a number of models, using Expert Mode</i>	266	
1.10.4	Run Multiple Models	266
1.10.5	Expert Data Entry	266
2	Tutorials	268
2.1	Condensate Pipeline Tutorial	269
2.1.1	Flow Assurance Considerations for Subsea Tieback Design	270
<i>Task 1: Developing a Compositional PVT Model</i>	270	
<i>Task 2: Constructing the Model</i>	272	
<i>Task 3: Sizing the Subsea Tieback</i>	274	
2.1.2	Hydrates	275
<i>Task 4: Selecting Tieback Insulation Thickness</i>	277	
<i>Task 5: Determining the Methanol Requirement</i>	277	
2.1.3	Severe Riser Slugging	278
<i>Task 6: Screening for Severe Riser Slugging</i>	280	
2.1.4	Slug Catcher Sizing	280
<i>Task 7: Sizing a Slug Catcher</i>	285	
2.2	Oil Well Performance Analysis Tutorial	43
2.2.1	NODAL Analysis	43
<i>Task 1: Building the Well Model</i>	287	
<i>Task 2: Performing NODAL Analysis</i>	49	
<i>Task 3: Performing a Pressure/Temperature Profile</i>	291	
2.2.2	Fluid Calibration	292
<i>Task 4: Calibrating PVT Data</i>	294	
2.2.3	Pressure/Temperature Matching	297
<i>Task 5: Evaluating Gas Lift Performance</i>	298	
<i>Task 6: Working with Multiple Completions</i>	299	
2.2.4	Flow Control Valve	302
<i>Task 7: Modeling a Flow Control Valve</i>	304	
2.3	Gas Well Performance Analysis Tutorial	304
2.3.1	Compositional Fluid Modeling	305
<i>Task 1: Creating a Compositional Fluid Model for a Gas Well</i>	309	
2.3.2	Gas Well Deliverability	312
2.3.3	Task 2: Calculating Gas Well Deliverability	312
<i>Task 3: Calibrating the Inflow Model Using Multipoint Test Data</i>	315	

2.3.4	Erosion Prediction	316
<i>Task 4: Selecting a Tubing Size</i>	317	
2.3.5	Choke Modeling	319
<i>Task 5: Modeling a Flowline and Choke</i>	320	
<i>Task 6: Predicting Future Production Rates</i>	322	
2.3.6	Liquid Loading	322
<i>Task 7: Determining a Critical Gas Rate to Prevent Well Loading</i>	324	
2.4	Looped Gas Gathering Network Case Study	324
2.4.1	Model a Gathering Network	324
2.4.2	Task 1: Building a Model of a Network	325
2.4.3	Task 2: Performing a Network Simulation	331
2.5	User Equipment DLL Tutorial - User Pump	334
2.5.1	Task 1. Write a DLL to define a piece of equipment for use with PIPESIM	334
<i>Introduction</i>	334	
<i>How to write a DLL</i>	335	
<i>Note on Fortran</i>	336	
<i>Note on C/C++</i>	336	
<i>PIPESIM flash</i>	336	
<i>Exporting routines</i>	337	
<i>Build the DLL</i>	337	
2.5.2	Task 2. Build a single branch model and use the DLL	338
<i>Place the built DLL in the PIPESIM programs directory</i>	338	
<i>Inform PIPESIM of the equipment DLL and its entry points</i>	338	
<i>Build a single branch model with an Engine Keyword Tool to represent the piece of equipment</i>	340	
<i>Note on Network models</i>	341	
2.5.3	Task 3. Run the model and verify that the DLL worked	341
2.6	Field Data Matching Tutorial	344
2.6.1	Measured Data	345
2.6.2	Model Setup	346
2.6.3	Task 1. IPR matching	346
2.6.4	Task 2. Optimizing Flow Correlation & Heat Transfer Coefficient	347
<i>Load Measured Data</i>	347	
<i>Setup Optimizer Options</i>	347	
<i>Select flow correlation parameters and U factor</i>	348	
2.6.5	Task 3. Validate Match	348
2.6.6	Task 4. Determine Choke Settings	349
2.7	Liquid Loading Analysis Tutorial	349
2.7.1	Task 1. Set up the Liquid Loading Model	350
2.7.2	Task 2. Control Liquid Loading Options in PIPESIM	350
2.7.3	Task 3. Analyze Well for Liquid Loading	350
2.7.4	Task 4. Determine Critical Gas Rate to Prevent Well Loading	351
2.8	Ramey Heat Transfer Model Tutorial	351
2.8.1	Task 1. Set up Ramey Heat Transfer Properties	351

2.8.2	Task 2. Set up Detailed Heat Transfer for the Tubing	352
2.8.3	Task 3. Run the Model and View Results	354
2.9	Pipe Inline Heating	354
2.9.1	Tutorial Data	354
2.9.2	Task 1: Build a Representative PIPESIM Model	356
2.9.3	Task 2: Screen for Hydrate Formation	359
2.9.4	Task 3: Prevent Hydrate Formation through Inline Heating	362
2.9.5	Task 4: Sensitize ILH Keywords	365
3	Support and Contact Information	369
3.1	Support	369
3.1.1	SIS web support	369
3.1.2	On-site support	369
3.1.3	SIS Education	369
4	Technical Description	370
4.1	Flow Models	370
4.1.1	Flow Regimes	370
	<i>Flow Regimes Classification for Vertical Two Phase Flow</i>	370
	<i>Flow Regimes Classification for Horizontal Two Phase Flow</i>	371
4.1.2	Horizontal Multiphase Flow Correlations	372
	<i>Baker Jardine (BJA) Correlation</i>	372
	<i>Beggs and Brill Original</i>	372
	<i>Beggs and Brill Revised</i>	373
	<i>Dukler (AGA) and Flanigan</i>	373
	<i>Eaton-Oliemans</i>	373
	<i>Hughmark-Dukler</i>	373
	<i>LEDA</i>	374
	<i>Minami and Brill</i>	374
	<i>Mukherjee and Brill</i>	374
	<i>NOSLIP Correlation</i>	375
	<i>OLGAS 2-phase / OLGAS 2000 3-phase</i>	375
	<i>Oliemans</i>	375
	<i>TUFFP Unified Mechanistic Model (2-phase and 3-phase)</i>	376
	<i>Xiao</i>	376
	<i>Xiao (film modified)</i>	377

4.1.3	Vertical Multiphase Flow Correlations	377
	<i>Ansari</i>	377
	<i>Aziz Govier Fogarasi</i>	377
	<i>Beggs and Brill Original</i>	378
	<i>Beggs and Brill Revised</i>	378
	<i>Duns and Ros</i>	378
	<i>Gomez</i>	379
	<i>Gomez enhanced</i>	379
	<i>Govier and Aziz</i>	379
	<i>Gray</i>	379
	<i>Gray Modified</i>	379
	<i>Gregory</i>	380
	<i>Hagedorn and Brown</i>	381
	<i>Mukherjee and Brill</i>	381
	<i>NOSLIP Correlation</i>	381
	<i>OLGAS 2-phase/ OLGAS</i>	381
	<i>LEDA</i>	382
	<i>Orkiszewski</i>	382
	<i>TUFFP Unified Mechanistic Model (2-phase and 3-phase)</i>	376
4.1.4	Suggested correlations	383
4.1.5	Friction and Holdup factors	385
4.1.6	Single Phase Flow Correlations	385
	<i>Moody (default for liquid or gas)</i>	386
	<i>AGA (for gas)</i>	387
	<i>Cullender and Smith (for gas)</i>	388
	<i>Other friction pressure drops for gas</i>	388
	<i>Hazen-Williams (for liquid water)</i>	389
4.1.7	Swap Angle	390
4.1.8	deWaard (1995) Corrosion Model	390
4.1.9	Cunliffe's Method for Ramp Up Surge	393
4.1.10	Liquid by Sphere	394
	<i>PI-SS (Severe-Slugging Group)</i>	395
4.1.11	Liquid Loading	396
	<i>Critical Unloading Velocity</i>	396
	<i>Critical Gas Rate</i>	397
4.2	Completion (IPR) Models	398

4.2.1	Inflow Performance Relationships for Vertical Completions	398
	<i>Productivity Index (PI)</i>	399
	<i>Vogel's Equation</i>	400
	<i>Fetkovich's Equation</i>	401
	<i>Jones' Equation</i>	401
	<i>Forchheimer Equation</i>	402
	<i>Back Pressure Equation</i>	402
	<i>Pseudo Steady State Equation / Darcy Equation</i>	403
	<i>Transient IPR</i>	408
	<i>Data File</i>	413
	<i>Bubble Point Correction</i>	414
	<i>Vertical Well Skin Factor</i>	415
4.2.2	Inflow Performance Relationships for Horizontal Completions	427
	<i>Theory</i>	427
	<i>Pressure Drop</i>	427
	<i>Inflow Production Profiles</i>	431
	<i>Steady-State Productivity</i>	432
	<i>Pseudo-Steady State Productivity</i>	435
	<i>Solution Gas-Drive IPR</i>	438
	<i>Horizontal Gas Wells</i>	438
	<i>Distributed Productivity Index Method</i>	441
4.2.3	Oil / Water Relative Permeability tables	441
	<i>Keywords</i>	442
4.2.4	Coning	442
4.3	Equipment	443
4.3.1	Chokes, Valves and Fittings	443
	<i>Choke</i>	443
	<i>Choke Subcritical Flow Correlations</i>	446
	<i>Choke Critical Pressure Ratio</i>	449
	<i>Choke Critical Flow Correlations</i>	450
	<i>Flow Control Valves Mechanistic Theory</i>	451
	<i>Fittings</i>	452
4.3.2	Compressors, Pumps, and Expanders	455
	<i>Centrifugal Pumps and Compressors</i>	455
	<i>Reciprocating Compressor Operation</i>	458
	<i>Electrical Submersible Pumps (ESP)</i>	235
	<i>Progressive Cavity Pump (PCP)</i>	465
	<i>Expanders</i>	468
4.3.3	Multiphase Boosting Technology	470
	<i>Guide to Multiphase Booster Efficiencies</i>	484
4.4	Heat Transfer Models	485
4.4.1	Energy Equation for Steady-State Flow	485
4.4.2	Overall Heat Transfer Coefficient	486

4.4.3	Inside Fluid Film Heat Transfer Coefficient	489
<i>Inside Forced Convection</i>	489	
<i>Inside Natural Convection</i>	494	
4.4.4	Conductive Heat Transfer Coefficients	495
4.4.5	Annulus and Outside Convective Heat Transfer Coefficients	497
4.4.6	Heat Transfer Between a Horizontal Flowline and the Ground Surface	499
<i>Fully Buried Ground Heat Transfer Coefficient</i>	499	
<i>Partially Buried Ground Heat Transfer Coefficient</i>	500	
4.4.7	Heat Transfer Between a Vertical Well and the Surrounding Rock	503
<i>Ramey Model</i>	503	
4.5	Fluid Models	504
4.5.1	Black Oil Fluid Modeling	505
<i>Black oil Correlations</i>	506	
<i>Solution Gas-oil Ratio</i>	508	
<i>Oil Formation Volume Factor</i>	512	
<i>Oil Viscosity</i>	515	
<i>Gas Compressibility</i>	522	
<i>Gas Viscosity</i>	525	
<i>Surface Tension</i>	525	
<i>Black Oil Enthalpy</i>	526	
<i>Black Oil Mixing</i>	528	
4.5.2	Compositional Fluid Modeling	141
<i>Cubic Equations of State</i>	536	
<i>Non-cubic Equations of State</i>	539	
<i>Components for Cubic Equations of State</i>	544	
<i>Components for Non-Cubic Equations of State</i>	549	
<i>Viscosity Models for Compositional Fluids</i>	515	
<i>Solid Precipitation</i>	551	
4.5.3	Fluid Property Table Files	554
<i>Internal fluid property tables</i>	555	
4.5.4	Liquid mixture properties	555
<i>Liquid Viscosity and Oil/Water Emulsions</i>	555	
<i>Liquid-gas Surface Tension</i>	560	
4.6	Solvers	561
4.6.1	Tolerance Equations	561
4.7	Typical and Default Data	561
4.7.1	Limits	561
<i>General</i>	561	
<i>Pipeline and facilities</i>	561	
<i>Well Performance</i>	561	
<i>Network</i>	562	
4.7.2	Tubing and Pipeline Tables	562
<i>Tubing/Casing Tables</i>	562	
<i>Pipeline tables</i>	567	

4.7.3	Typical Values	571
<i>Fluid Properties</i>	571	
<i>Roughness</i>	572	
<i>Thermal Conductivities</i>	572	
<i>Permeability</i>	574	
<i>Drainage Radius</i>	574	
<i>Fittings</i>	574	
4.8	Glossary	576
4.8.1	Roman Letters	576
4.8.2	Greek Letters	579
4.8.3	Subscripts	580
4.9	Conversion Factors	581
4.9.1	Length	581
4.9.2	Volume	581
4.9.3	Mass	581
4.9.4	Time	581
4.9.5	Gravity	581
4.9.6	Pressure	582
4.9.7	Energy	582
4.9.8	Power	582
4.9.9	Dynamic viscosity	582
4.9.10	Permeability	582
4.10	References	582
5	Keyword Index	595
5.1	Keyword List	595
5.1.1	A	595
5.1.2	B	595
5.1.3	C	596
5.1.4	D E	596
5.1.5	F	596
5.1.6	G	596
5.1.7	H	597
5.1.8	I	597
5.1.9	J	597
5.1.10	K	597
5.1.11	L	597
5.1.12	M	597
5.1.13	N	598
5.1.14	O	598
5.1.15	P	598
5.1.16	Q R	598
5.1.17	S	598
5.1.18	T	599
5.1.19	U	599

5.1.20	V	599
5.1.21	W	599
5.1.22	XYZ	599
5.2	Input Files and Input Data Conventions	599
5.2.1	General	599
5.2.2	Statements	600
5.2.3	Delimiters	600
<i>Examples</i>	601	
5.2.4	Abbreviations	601
<i>Example</i>	601	
5.2.5	Numeric data	602
<i>Example</i>	602	
5.2.6	Units Description	602
5.2.7	Character Input	602
<i>Example</i>	603	
5.2.8	Comment Statements and Blank Lines	603
<i>Example</i>	603	
5.2.9	Multiple Value Data Sets	603
<i>Examples</i>	604	
5.2.10	Input Files	604
<i>General</i>	604	
<i>The main input ('.PSM' or '.PST') file</i>	605	
<i>Included files and the INCLUDE statement</i>	605	
<i>AUTOEXEC.PSM</i>	606	
<i>modelname.U2P or branchname.U2P</i>	606	
5.3	General Data	606
5.3.1	Changing Parameters within the System Profile	606
<i>Example</i>	607	
<i>Multiple Cases</i>	607	
5.3.2	HEADER - Job Accounting Header (Required)	607
<i>Example</i>	607	
5.3.3	JOB - Job Title (Optional)	607
5.3.4	CASE - Case Title (Optional)	608
5.3.5	UNITS - Input and Output Units (Optional)	608
<i>Example</i>	608	
5.3.6	OPTIONS Calculation Procedure Options (Optional)	609
5.3.7	RATE: Fluid Flow Rate Data	619
5.3.8	ITERN Iteration Data (Optional)	621
5.3.9	INLET System Inlet Data	623
5.3.10	PRINT Output Printing Options (Optional)	624
<i>Per-case output page options</i>	624	
<i>Attributes</i>	627	
<i>Point report subcodes</i>	628	
<i>One-off output pages</i>	630	
5.3.11	PLOT Output Plotting Options (Optional)	631

5.3.12	NOPRINT Output Print Suppression Options (Optional)	634
5.3.13	BEGIN , END - Block delimiters	634
<i>Example</i>	635	
5.3.14	PUSH - Remote Action Editing (optional)	636
5.3.15	PLOTFILEDATA	637
5.3.16	EXECUTE - deferred execution of a statement	637
5.3.17	USERDLL - Equipment	638
5.4	FLOW CORRELATION DATA	638
5.4.1	CORROSION	639
5.4.2	EROSION Erosion Rate and Velocity (Optional)	639
5.4.3	SLUG Slug Calculation Options (Optional)	640
<i>Slug catcher size</i>	641	
5.4.4	VCORR Vertical Flow Correlation Options	641
<i>Summary of Valid Vertical Flow Correlation Combinations</i>	642	
<i>Vertical Flow Correlations - Abbreviations</i>	643	
5.4.5	HCORR Horizontal Flow Correlation Options	645
<i>Summary of Valid Horizontal Flow Correlation Combinations</i>	646	
<i>Horizontal Flow Correlations - Abbreviations</i>	643	
5.4.6	SPHASE Single Phase Flow Options (Optional)	648
5.4.7	USERDLL - Flow Correlations	650
5.5	WELL PERFORMANCE MODELING	650
5.5.1	INTRODUCTION	651
5.5.2	COMPLETION Completion Profile Delimiter	651
<i>Supercode</i>	652	
5.5.3	WELLPI Well Productivity Index (Optional)	653
<i>Subcodes</i>	653	
5.5.4	WPCURVE (Optional)	654
5.5.5	VOGEL Vogel Equation (Optional)	654
<i>Subcodes</i>	654	
5.5.6	FETKOVICH Fetkovich Equation (Optional)	654
<i>Subcodes</i>	654	
5.5.7	JONES Jones Equation (Optional)	655
<i>Subcodes</i>	655	
5.5.8	IFPPSSE : Data for the Pseudo Steady State Equation (Optional)	655
5.5.9	WCOPTION Well Completion Data (Optional)	657
5.5.10	IPRCRV or IFPCRV: Inflow Performance Curve	660
<i>Examples</i>	661	
5.5.11	IFPTAB Inflow Performance Tabulation (Optional)	662
<i>Values</i>	662	
<i>Example</i>	663	
5.5.12	CONETAB Coning Relationship Tabulation (Optional)	663
<i>Example</i>	664	
5.5.13	BACKPRES Back Pressure Equation (BPE) (Optional)	664
<i>Subcodes</i>	664	
5.5.14	HORWELL Horizontal Well Inflow Performance	664

5.5.15	LAYER Reservoir Layer Properties	666
<i>Examples</i>	667	
5.5.16	RESERVOIR	668
5.5.17	PERMCRV: Curves of Relative Permeability versus Saturation (Optional)	668
<i>Example</i>	668	
5.5.18	PERMTAB: Tabulation of Relative Permeability versus Saturation (Optional)	669
<i>Example</i>	669	
5.5.19	HVOGEL (Optional)	670
5.5.20	FORCHHEIMER (Optional)	670
5.5.21	FRACTURE: Data for Hydraulic Fracture	670
5.5.22	TRANSIENT: Data for the Transient Inflow equation (Optional)	671
5.6	SYSTEM DATA	672
5.6.1	CHOKE (Optional)	673
5.6.2	COMPCRV and PUMPCRV: Compressor and Pump performance curves	677
<i>Examples</i>	678	
5.6.3	COMPRESSOR Compressor (Optional)	679
5.6.4	RODPUMP: Rod- or Beam-pump	680
5.6.5	EQUIPMENT Generic Equipment	443
<i>Examples</i>	683	
5.6.6	EXPANDER Expander (Optional)	683
5.6.7	FITTING : Valves and Fittings	684
<i>EXAMPLES</i>	685	
5.6.8	FMPUMP (Optional)	686
5.6.9	FRAMO 2009 (Optional)	686
<i>EXAMPLE</i>	687	
5.6.10	HEATER Heater/Cooler (Optional)	687
5.6.11	GASLIFT: Multiple Injection Ports in Gaslifted Wells	687
<i>Main-code: GASLIFT</i>	688	
5.6.12	INJPORT Gas Lift Injection Valve	691
5.6.13	INJGAS: Injection Gas (Optional) and INJFLUID: Fluid Injection	693
5.6.14	MPBOOSTER (Optional)	695
5.6.15	MPUMP Multiphase Pump (Optional)	696
5.6.16	NODE System Profile Data (Required)	698
5.6.17	PIPE: Pipe or Tubing cross-section dimensions (Required)	699
5.6.18	PUMP Pump (Optional)	701
5.6.19	COMPCRV and PUMPCRV: Compressor and Pump performance curves	677
<i>Examples</i>	678	
5.6.20	REINJECTOR (Optional)	704
5.6.21	RODPUMP: Rod- or Beam-pump	680
5.6.22	SEPARATOR Separator (Optional)	705
5.6.23	WELLHEAD Wellhead Profile Delimiter	706
5.7	HEAT TRANSFER DATA	707
5.7.1	Notes on Heat Transfer Output Printing	707
5.7.2	HEAT Heat Balance Options (Optional)	707
<i>Heat transfer mode</i>	710	

5.7.3	COAT Pipe Coat and Annular Space Medium Data (Optional)	710
<i>Example</i>	711	
5.7.4	TCOAT Pipe Coat Thickness Data (Optional)	711
5.7.5	KCOAT Pipe Coat Thermal Conductivity Data (Optional)	712
<i>Files</i>	713	
<i>Example</i>	713	
5.7.6	FLUID Fluid Thermal Conductivity Data (Optional)	715
5.7.7	CONFIG: Pipe Heat Transfer Configuration Data (Optional)	715
5.7.8	Pipeline burial depth examples	716
5.8	Fluid Models	717
5.8.1	BLACK OIL DATA	717
<i>BLACKOIL: Black Oil Fluid definitions</i>	717	
<i>PROP Fluid Property Data (Optional)</i>	719	
<i>LVIS: Liquid Viscosity Data (Optional)</i>	721	
<i>CPFLUID: Fluid Heat Capacity Data (Optional)</i>	726	
<i>TPRINT Black Oil Table Printing (Optional)</i>	726	
<i>CALIBRATE: Black Oil Property Calibration (Optional)</i>	727	
<i>CONTAMINANTS Gas phase contaminants data (optional)</i>	728	
5.8.2	COMPOSITIONAL DATA	729
<i>AQUEOUS: Aqueous Component Specification</i>	729	
<i>CEMULSION Compositional Liquid Emulsion Data (Optional)</i>	729	
<i>COMPOSITION: Compositional Fluid Specification</i>	732	
<i>LIBRARY: Library Component Specification</i>	736	
<i>MODEL: Model Properties Specification</i>	736	
<i>PETROFRAC: Petroleum Fraction Specification</i>	737	
<i>TPRINT Tabular Data Print Options (Optional)</i>	738	
5.9	PIPESIM OPERATIONS OPTIONS	738
5.9.1	NAPLOT: Nodal Analysis	739
5.9.2	NAPOINT System Analysis Point	743
5.9.3	MULTICASE Introduction and Summary	743
<i>General Rules for use with MULTICASE</i>	744	
5.9.4	Explicit Subcodes	744
5.9.5	General Purpose Subcodes	747
<i>Examples</i>	747	
5.9.6	Combining MULTICASE and CASE/ENDCASE	748
5.9.7	Multiple Case and PS-PLOT	749
5.9.8	Reservoir Simulator Tabular Data Interface	750
5.9.9	ASSIGN Changing Profile Data by Assignment	751
<i>Example</i>	752	
5.9.10	OPTIMIZE	752
<i>Examples</i>	753	

5.9.11	Wax deposition and Time Stepping modeling options	205
	<i>Time subcodes</i>	754
	<i>Termination subcodes</i>	755
	<i>Wax subcodes</i>	756
	<i>BP, DBRS or DBRM method subcodes</i>	757
	<i>Shell subcodes</i>	758
5.10	PIPESIM-Net keywords	759
5.10.1	SETUP	760
	<i>Subcodes</i>	760
5.10.2	BRANCH	762
	<i>Subcodes</i>	762
5.10.3	SOURCE	764
	<i>Subcodes</i>	764
5.10.4	SINK	767
	<i>Subcodes</i>	767
5.10.5	JUNCTION	769
	<i>Subcodes</i>	769
5.10.6	NSEPARATOR	769
	<i>Subcodes</i>	769
5.11	Keyword Index	595
5.11.1	Keyword List	595
	<i>A</i>	595
	<i>B</i>	595
	<i>C</i>	596
	<i>D E</i>	596
	<i>F</i>	596
	<i>G</i>	596
	<i>H</i>	597
	<i>I</i>	597
	<i>J</i>	597
	<i>K</i>	597
	<i>L</i>	597
	<i>M</i>	597
	<i>N</i>	598
	<i>O</i>	598
	<i>P</i>	598
	<i>Q R</i>	598
	<i>S</i>	598
	<i>T</i>	599
	<i>U</i>	599
	<i>V</i>	599
	<i>W</i>	599
	<i>XYZ</i>	599
6	Open Link	776

7 Third party applications	777
Index	778

1

User Guide

1.1 Introduction

PIPESIM was originally developed by a company called Baker Jardine. Baker Jardine was formed in 1985 to provide software and consulting services to the oil and gas industry. In April 2001, Baker Jardine was acquired by Schlumberger.

Schlumberger has invested in the redevelopment of the industry's leading Production Engineering software to ensure that it can solve challenging multiphase flow problems. PIPESIM couples a leading-edge Graphical User Interface (GUI) with a field-proven computation engine.

1.1.1 Company profile

Schlumberger has wide experience in the design and optimization of oil and gas production systems particularly in the transportation of live hydrocarbon fluids, a vital element in the production and processing of hydrocarbons. The development of efficient gathering and transportation systems requires a combination of detailed theoretical knowledge and practical experience of the complex behavior of multiphase hydrocarbon mixtures.

Schlumberger is at the leading edge of software development for the oil and gas industry with the software products such as PIPESIM, OFM, and ECLIPSE. These tools have been successfully applied to numerous systems for the modelling and data management of new and existing oil and gas production and distribution systems for most major oil companies.

PIPESIM is a steady-state, multiphase flow simulator used for the design and diagnostic analysis of oil and gas production systems. PIPESIM software tools models multiphase flow from the reservoir to the separator.

Schlumberger is actively involved in research and development of new multiphase fluid flow technologies, and has managed a diverse range of joint industry projects in this field.

1.1.2 Getting Started

Training courses on the use of PIPESIM can be arranged. Please contact your local Schlumberger Support office or visit www.pipesim.com.

New users are encouraged to read the following sections in the user guide/help system:

- [Case Studies \(p.268\)](#) (either [Oil Well Design \(p.43\)](#) or [Condensate Pipeline \(p.269\)](#)) for details on how to build models.
- Using the [model wizard \(p.28\)](#) to build a model.

Applications

PIPESIM offers a wide ranging capability for modeling entire production systems from the reservoir to the processing facility. Typical applications include:

Well Performance analysis

- Well design
- Well optimization
- Well inflow performance modeling
- Gas Lift design and performance modeling
- ESP design and performance modeling
- PCP performance modeling
- Horizontal well modeling
- Injection well design
- Reservoir VFP table generation
- Detailed sensitivity analysis
- Automated model matching

Pipeline and Facilities

- Point by point generation of pressure and temperature profiles
- Prediction of solids formation (hydrates, wax, asphaltene, scale)
- Slug catcher design (hydrodynamic slugs, pigging, ramp-up)
- Equipment selection (pumps, compressors, multiphase boosters)
- Pipeline design
- Comparing measured data with calculated data

Network analysis module

- Unique network solution algorithm to model wells in large networks
- Rigorous thermal modeling of all network components
- Multiple looped pipeline/flowline capability
- Debottlenecking studies
- Comprehensive pipeline equipment models
- Gathering and distribution networks
- Gas lift optimization

Technical Description of Features

Depending on the type of application, users may select from an extensive set of features to model a wide variety of production systems and fluids. The following tables provide a technical summary of these features. Additional details are available throughout the online help system.

Feature	Description
Black Oil (p.505)	<ul style="list-style-type: none"> Latest industry standard fluid property correlations that cover all types of petroleum fluids from extra heavy oil to light oil and condensate. It can also be used for simplified gas, utility fluids, and so on. Multi-level calibration from simple bubble point matching to advance fluid calibration matching multiple sets of lab data measurements. Wide range of viscosity correlations that includes options for user-specified dead oil and emulsion viscosities. Wide range of emulsion correlations covering tight to light emulsion types. Also, users can specify emulsion tables, and specify or calculate the inversion point. Ability to plot fluid properties at lab or reservoirs conditions. Specify gas contaminants used for compressibility factor adjustment and corrosion calculations. Specify thermal data for all phases of a black oil fluid for accurate thermal modeling and some of the standard methods for fluid enthalpy calculation for accurate energy balance prediction. Comprehensive Fluid Mixing rules.
Compositional (p.141)	<ul style="list-style-type: none"> Choices of Schlumberger developed and third party flash packages, including: <ul style="list-style-type: none"> Eclipse 300 Multiflash DBR GERG 2008 Refprop V8 These packages come with their own standard library of components and binary interaction parameters. Most of these packages allow users to define and calculate properties of pseudo components for accurate modeling of fluid properties in the absence of detailed fluid characterization. Has a wide range of equation of states and transport properties correlations that are based on selected flash packages: <ul style="list-style-type: none"> Flash Packages : 2-3-Peng-Robinson (standard and corrected), SRK (standard and corrected), Association (CPA), BWRS, GERG-2008,

Feature	Description
	<p>NIST default, and so on. Correction for volume shift and Accentric factors as applicable.</p> <ul style="list-style-type: none"> • Binary Interaction Parameters : OilGas1, OilGas2, OilGas3, OilGas4, user specified • Viscosity Models : Pederson, LBC, Aasberg-Petersen, NIST default • Emulsion Methods : Woelflin, Volume ratio, continuous phase, none • Surface Tension : Macleod-Sugden, NIST default • Generate Phase Envelopes, including quality lines and formation curves for hydrates, waxes and asphaltenes. • Has Quick Flash calculations to examine fluid properties at specified P-T Flash/Separation conditions. • Ability to perform phase ratio matching for water, Oil and Gas based on field measurement. This feature is useful for quick updates of fluid composition based on actual measurement of phases at the field separator. • Salinity analysis based on Ion, Salt Components and TDS (Multiflash). <p>Note: Listed features vary depending on selected flash package. Refer to the PIPESIM application for available options against a given flash package.</p>
PVT Files (p.160)	<p>A number of third-party applications are capable of generating PVT files that can be used by PIPESIM. These applications include:</p> <ul style="list-style-type: none"> • PVTSim • HYSYS • UniSim • ScaleChem • DBRSolids • GUTS
Steam (p.166)	<p>PIPESIM allows steam modeling and interprets the properties of fluid based on ASTM97 Steam Tables. Steam models can be used for producers and injectors in both single branch and network modules of PIPESIM.</p>

Table 1.1: Fluid Property Modeling

Feature	Description
Single Phase Flow Correlation (p.385)	<ul style="list-style-type: none"> • Moody • AGA (with tuning option for drag factor) • Panhandle A (with tuning option for flow efficiency)

Feature	Description
	<ul style="list-style-type: none"> • Panhandle B (with tuning option for flow efficiency) • Hazen – Williams (with tuning option for C factor) • Weymouth (with tuning option for flow efficiency) • Cullender – Smith
Vertical Multiphase Flow Correlation (Standard) (p.377)	<p>Standard</p> <ul style="list-style-type: none"> • Beggs & Brill (Original & Revised) • Duns & Ros • Govier, Aziz • Govier, Aziz & Fogarasi • Gray (Original & Modified) • Hagedorn & Brown (Original/Revised - with/without Duns & Ros map) • Mukherjee & Brill • Orkiszewski • No-Slip <p>OLGA</p> <ul style="list-style-type: none"> • Olga-S 2000 Version 6.2.7 (Oct 2010) – 2/3 Phase • Olga-S 2000 Version 5.3.2 (Feb 2009) – 2/3 Phase • Olga-S 2000 Version 5.3 (Feb 2008) – 2/3 Phase • Olga-S 2000 Version 5.0 (Jun 2006) – 2/3 Phase <p>TULSA Unified Mechanistic Model</p> <ul style="list-style-type: none"> • TUFFP Unified 3-Phase v 2011.1 (with default/override emulsion viscosity) • TUFFP Unified 2-Phase v 2011.1 <p>LedaFlow Point Model</p> <ul style="list-style-type: none"> • LedaFlow Point Model v1.0.231.1 (Jun 2011) – 2/3 Phase
Horizontal Multiphase Flow Correlation (Standard) (p.372)	<p>Standard</p> <ul style="list-style-type: none"> • Beggs & Brill (Original & Revised – with/without Taitel Dukler map) • Baker Jardine Revised • Dukler, AGA & Flanagan (with/without Eaton Holdup) • Lockhart & Martinelli (with/without Taitel Dukler map) • Mukherjee & Brill • Oliemans • Xiao • Dukler

Feature	Description
	<ul style="list-style-type: none"> No-Slip <p><u>OLGA</u></p> <ul style="list-style-type: none"> Olga-S 2000 Version 6.2.7 (Oct 2010) – 2/3 Phase Olga-S 2000 Version 5.3.2 (Feb 2009) – 2/3 Phase Olga-S 2000 Version 5.3 (Feb 2008) – 2/3 Phase Olga-S 2000 Version 5.0 (Jun 2006) – 2/3 Phase <p><u>TULSA Unified Mechanistic Model</u></p> <ul style="list-style-type: none"> TUFFP Unified 3-Phase v 2011.1 (with default/override emulsion viscosity) TUFFP Unified 2-Phase v 2011.1 <p><u>LedaFlow Point Model</u></p> <ul style="list-style-type: none"> LedaFlow Point Model v1.0.231.1 (Jun 2011) – 2/3 Phase
Calibration (p.136)	PIPESIM includes a feature that can adjust the holdup-factor, friction-factor, and U-value multiplier automatically to match measured pressures and temperatures. Additionally, the flow correlation comparison operation can quickly sensitize on flow correlations to aid in selecting the most appropriate model.
Flow Regime Maps (p.123)	PIPESIM produces high-resolution flow regime maps at any point in the system that is selected.
Extensibility (p.121)	PIPESIM includes code templates that can assist users in compiling their own 2-phase or 3-phase flow correlation plug-in dll.

Table 1.2: Flow Correlation Options

Feature	Description
Wellbore Heat Transfer (p.100)	<ul style="list-style-type: none"> User Specified Heat Transfer Coefficient Calculated Heat Transfer Coefficient using Ramey's Model (takes into account thermal properties of various layers – rock, cement, completion fluid, casing, tubing, etc)
Flowline & Riser Heat Transfer (p.486)	<ul style="list-style-type: none"> User Specified Heat Transfer Coefficient Calculated Heat Transfer Coefficient taking into account pipe and ground thermal properties and properties of multiple layers of pipe coatings (optional) choosing. PIPESIM rigorously calculates conductive and convective (free and forced) heat transfer for pipes that are fully buried, partially buried and fully exposed to air or water.
Inside Fluid Film Heat Transfer Coefficient (p.489)	<p>Available methods are:</p> <ul style="list-style-type: none"> Kreith Model Kaminsky Model (flow regime dependent) <p>User has the option to consider/ignore Fluid Film Heat Transfer coefficient.</p>

Feature	Description
Energy Balance (p.171)	PIPESIM has comprehensive energy balance calculation taking into account potential energy, kinetic energy, and internal energy to effectively calculate heat loss/gain in the system and heat transfer with the outside environment. Detailed Thermodynamics are considered such as Joule-Thomson heating/cooling and frictional heating.
Flow Assurance Related (p.141)	PIPESIM provides several other calculations to aid accurate flow assurance studies: <ul style="list-style-type: none"> • Calculation and reporting of Hydrate sub-cooling ΔT. • Calculation and reporting of Asphaltene sub-cooling ΔT. • Calculation and reporting of wax sub-cooling ΔT.

Table 1.3: Heat Transfer Calculations

Feature	Description
Wellbore Modeling (p.50)	PIPESIM allows simple and detailed modes for defining a wellbore for single and multilayer producers and injectors. Options include: <ul style="list-style-type: none"> • 2-dimensional deviation surveys • Simple/detailed geothermal data for heat transfer calculations • Tubular, annular or mixed flow (using equivalent hydraulic diameter concept) • Downhole equipment, including chokes, sub-surface safety valve, separators, chemical injectors • Artificial Lift equipment, including gas lift valves, ESP's, PCP's and Rod Pumps • Coil Tubing/velocity string modeling
Skin Calculations (p.68)	Several standard completions options are supported in PIPESIM: <ul style="list-style-type: none"> • Open hole Completion and associated skin calculations due to <ul style="list-style-type: none"> • Damaged zone, Partial penetration • Open hole Gravel Pack Completion and associated skin calculations due to <ul style="list-style-type: none"> • Damaged zone, Partial penetration, Gravel Pack • Perforated Completion and associated skin calculations due to <ul style="list-style-type: none"> • Damaged zone, Compacted Zone, Partial penetration and Perforation (perforation skin calculation use methods like McLeod and Karakas/Tariq; taking into account perforation geometry, density and phase angle) • Gravel Pack and Perforated Completion and associated skin calculations due to

Feature	Description
	<ul style="list-style-type: none"> • Damaged zone, Compacted Zone, Partial penetration, Gravel Pack and Perforation • Frac-Pack Completion and associated skin calculations due to <ul style="list-style-type: none"> • Partial Penetration and Frac-Pack Skin (accounting for fracture half length, fracture width, proppant permeability, frac face damage, fracture choke damage, and so on) <p>Most of the parameters responsible for skin contribution are available for sensitivity and uncertainty analysis.</p> <p>Skin calculations available in PIPESIM across horizontal completions account for effects of the damaged zone, gravel pack, compacted zone and perforations.</p> <p>For both horizontal and vertical completions, the user has the option to override skin calculations by supplying a user defined skin component that may be available from well test data.</p>
Inflow Performance Modeling (p.398)	<p><u>Vertical Completions:</u></p> <ul style="list-style-type: none"> • Well PI - Gas • Well PI – Liquid (with/without taking into account Vogel correction and correction for water phase) • Vogel – (Liquid only) • Fetkovitch – (Liquid only) • Jones – Gas & Liquid • Backpressure – (Gas only) • Forchheimer (Gas only) <p>All of the above equations allow calculation of dependent parameters based on user supplied well test data (if available).</p> <ul style="list-style-type: none"> • Pseudo Steady State – Gas (variation – Pressure squared/Pseudo Pressure and with/without transient calculation) • Pseudo Steady State – Liquid (with/without Vogel correction and with/without transient calculation) • Hydraulic Fracture Model – Gas (with/without transient calculation) • Hydraulic Fracture Model – Liquid (with/without transient calculation and/or Vogel correction) <p><u>Horizontal Completion (Single Point PI Models):</u></p> <ul style="list-style-type: none"> • Steady State Joshi Model for Liquid and Gas based IPR calculation • Pseudo Steady State Babu & Odeh Model for Liquid and Gas based IPR calculations

Feature	Description
	<p>Horizontal Completion (Distributed PI Models): Distributed PI models take into account detailed profile of tubular in the horizontal completion interval:</p> <ul style="list-style-type: none"> • Steady State Joshi Model for Liquid and Gas based IPR calculation • Pseudo Steady State Babu & Odeh Model for Liquid and Gas based IPR calculations • Simple distributed PI Model
Intelligent Completions (p.194)	<p>This feature allows users to control flow from a particular layer to reduce backpressure on other potential contributing layers. This has been implemented by introducing downhole flow control valve linked to the completed layer. Example applications include:</p> <ul style="list-style-type: none"> • Shut-off/control of high watercut layers • Shut-off/control of gassy layers • Control water and gas coning • Regulate flow from various layers • Enhance overall production for well by managing system back-pressure • Control corrosion problems that could be due to fluid contribution from a specific layer • Inject into target zones
Well specific Operations	Refer to PIPESIM Operation features section.

Table 1.4: Well Modeling

Feature	Description
Flowlines (p.95) and Risers (p.112)	<p>PIPESIM allows simple and detailed modes for defining flowlines and risers. Options include:</p> <ul style="list-style-type: none"> • Ability to define pipe undulations to account for uneven ground (flowlines) • Model a Riser and Downcomers • Specify data required from simple to detailed heat transfer calculation (For more information, refer to the heat transfer section.) • Specify measured pressure and temperature (if available). This information is used for model tuning. (For more information, refer to data matching operation under the 'Flow Correlation' section.) • Simplified schematic is available to indicate flowline/riser geometry.
Sources & Sinks (p.118)	<p>Define points of fluid entry into and exiting the system instead of production/injection wells.</p> <p>Network sources may be specified with PQ curves to represent wellhead responses.</p>

Feature	Description
Chokes (p.443) (Surface/ downhole)	<p>PIPESIM allows users to model choke/flow restriction both at surface or downhole/wellbore. Options include:</p> <ul style="list-style-type: none"> User specified or calculated choke bean size. User specified or calculated critical pressure ratio across choke. Various correlations for calculation of pressure losses: <ul style="list-style-type: none"> Subcritical Correlation: Mechanistic, API 14B, Ashford Critical Correlation: Mechanistic, Gilbert, Ros, Achong, Baxendell, Ashford, Poetbeck, Omana, Pilehvari, User Specified Correlation Advanced options are available to tune performance of choke by specifying flow coefficients for liquid and gas phases, discharge coefficient, Cp/Cv, Gas Expansion factor, etc.
Pumps (p.117)	<ul style="list-style-type: none"> Control pump performance by applying limits for DP, Power, and so on, or combination of these Calculate pump parameters – DP, Power, and so on — for single or multiple sets of operating conditions Simple thermodynamic model or user specified curves Most pump performance parameters — head, DP, power, number of stages (if applicable), speed (if applicable), efficiency, and so on are available as sensitivity variables for design or uncertainty analysis Viscosity correction (Turzo method)
Multiphase Booster (p.102)	<ul style="list-style-type: none"> Generic Multiphase booster (treated as pump and compressor in parallel) Twin Screw Multiphase Booster (ability to specify vendor booster performance data) Framo Multiple Boosters: <ul style="list-style-type: none"> Model catalog Tuning factors Models Series vs. Parallel and recirculation behavior Generates detailed performance maps Control booster performance by applying operational limits in various combinations. Calculate booster parameters for single or multiple sets of operating conditions. Most of the booster performance parameters (depending on selected type and model) — head, DP, power, speed, efficiency, head parameter, flowrate parameter, and so on are available as sensitivity variables for design or uncertainty analysis.

Feature	Description
Compressor (p.91) and Expander (p.94)	<ul style="list-style-type: none"> Model both centrifugal and reciprocating compressors/expanders. Control compressor performance by applying limits for DP, Power, and so on, or combination of these. Calculate pump parameters – DP, Power, and so on for single or multiple sets of operating conditions. Ability to model various thermodynamic routes – Adiabatic, Polytropic or Mollier. Reciprocating compressors allows multiple stages with the option to add inter-cooler temperature condition. User can model compressor performance for a series of discharge pressure settings. Users can add Vendor pump performance curves to the PIPESIM database. Most of the compressor performance parameters — head, DP, power, number of stages (if applicable), speed (if applicable), efficiency, and so on are available as sensitivity variables for design or uncertainty analysis.
Generic Equipment (p.91)	<p>Generic equipment allows user the option to model any type of object that is not available in PIPESIM equipment library. Some of the key features are:</p> <ul style="list-style-type: none"> Model equipment with several combination of inputs to offer following types (or combination) of equipment settings: <ul style="list-style-type: none"> Fixed discharge temperature device Heater/cooler Pressure booster or reducer devices Device with fixed duty Several combinations of these. Generic equipment in PIPESIM supports various thermodynamic routes – Isothermal, Isenthalpic, and Isentropic. Control compressor performance by applying limits for DP, Power, and so on, or combination of these.
Separator Inline and Network (Inline for both surface & downhole) (p.116)	<p>PIPESIM has several options for modeling separators, including:</p> <ul style="list-style-type: none"> Inline separators (separated streams get discarded) of various types — Liquid, Gas or Water with user specified separation efficiency. Inline separators can be used both at surface or downhole (downhole separators are also available at the upstream of various artificial lift equipments.) Network separators allows tracking of both product and separated streams. Separation relies on rigorous PIPESIM engine flash calculations performed for all types of fluid definitions based on in-situ P, T conditions. PIPESIM allows separators configured in series/parallel arrangements to model conditions to model multistage separation trains.

Feature	Description
	<ul style="list-style-type: none"> PIPESIM network separators adjust for pressure continuity across separators to allow boundary condition matching for each outlet stream.
Heat Exchangers (p.98)	<p>Generic equipment allows user the option to model any type of object that is not available in PIPESIM equipment library. Some of the key features are:</p> <ul style="list-style-type: none"> Model heat exchangers with several combination of inputs to offer following types (or combination) of equipment settings: <ul style="list-style-type: none"> Fixed discharge temperature, temperature differential or duty. Fixed discharge pressure or pressure drop. All the above parameters are available as sensitivity variables. Wide range of reports are available at inlet and outlet conditions.
Fluid Injection (Surface/downhole) (p.89)	<p>PIPESIM allows modeling of fluid injectors both at the surface and in the wellbore. Fluid injection can be used to inject chemicals or other fluids to handle flow assurance issues. Key features include:</p> <ul style="list-style-type: none"> Ability to inject any type of petroleum fluid (Black Oil, compositional, MFL file (generated from Multiflash Standalone.) Injected fluid is seamlessly mixed with the main fluid to predict mixture properties.
Adders/Multipliers	<p>Adders and/or multipliers are used as a tool to simulate performance of a flowing production/injection system to account for many scenarios. Some examples are:</p> <ul style="list-style-type: none"> Designing a pipeline system accounting for expected wells in future. Simulating turndown scenarios in surface networks. Designing parallel pipelines. Many other similar scenarios.
Report Tool (p.115)	<p>Report tools are widely used to generate detailed report at any point in the flowing system.</p> <ul style="list-style-type: none"> Available reports include: <ul style="list-style-type: none"> Flow map (to generate high resolution flow regime map) Phase split (compositional fluids) Fluid properties (stock tank condition) Fluid properties (flowing condition) Cumulative values Multiphase flow parameters Slugging Values Sphere generated liquid volumes Heat Transfer input values

Feature	Description
	<ul style="list-style-type: none"> • Heat Transfer output values • Compositional detail (in-situ condition) • Phase envelope • These reports have several uses. Some are: <ul style="list-style-type: none"> • As FEED for designing any equipment to be installed at any point in the system. • Detailed performance analysis across any surface equipment. • Analyzing performance of production system on phase envelope to understand possible flow assurance issues.
Engine Keyword Tool (p.94)	<p>This is an advanced tool that allows expert PIPESIM users to perform advanced modeling tasks beyond the range of functionalities exposed in the PIPESIM user interface. There are many tasks that can be performed using engine keyword tool. Some of the high level applications are:</p> <ul style="list-style-type: none"> • Perform advanced settings such as changing flow correlations, heat transfer methods and other calculation options in the middle of a flowing system. • Use keyword tools to build special equipment or combination of equipment not supported by the PIPESIM User Interface. • Add special equipment such as inline heaters, coolers, pressure boosters, and so on. • Configure special reports at any point in the system.

Table 1.5: Surface Equipment

Feature	Description
Gas Lift Systems (p.214)	<p>PIPESIM uses advanced methods to perform design and diagnostics for a gas lifted wells. There are many other associated operations also available in PIPESIM to analyze performance of a gas lifted well. Some of the key features include:</p> <p>Gas Lift Valve Database</p> <ul style="list-style-type: none"> • Extensive database of gas lift valves of several types/series and sizes from various manufacturers. <ul style="list-style-type: none"> • Bompet • Daniel • Hughes • Macco • Schlumberger (Camco) • Schlumberger (Merla) • Weatherford

Feature	Description
	<p>Users can easily add new valves to the database.</p> <p>Gas Lift Design</p> <ul style="list-style-type: none"> • Several design methods available (IPO-Surface Close, IPO-pt-Min-Max & PPO) • Valve sizing and mandrel spacing calculations for optimum design • Valve sizing for existing mandrel spacings • Design can be based on pressure boundary conditions or at a fixed target production rate • Design takes into account detailed hydraulic calculations inside the tubing as well as frictional pressure loss for injected gas through the annulus. • User configurable design bias/safety factors to control design (conservative vs. worst case scenario) • User control over choice of valve manufacturer, size, series • Redesign option available with change in spacing, change in one or more valves, change in design temperature profile, and so on • Design results and plots include: <ul style="list-style-type: none"> • Recommended valve information – model, spacing, size, and so on • Recommended test rack opening pressure for all valves • Valve throughput calculations • Pressure and temperature profiles in the tubing and annulus section as well as values across each valve. • A well formatted report including input data (design control, design parameters, design bias, fluid data, and so on), design calculation and spacing plot. <p>Gas Lift Diagnostics</p> <ul style="list-style-type: none"> • Gas lift diagnostics take into account injection gas conditions and operating boundary conditions to provide operational status of each valve (open/closed/throttling.) • Graphical diagnostic results provide production and injection profiles and the status of valves. • Tabulated performance data sheet reporting operating status of each valve. <p>Other Analysis</p> <ul style="list-style-type: none"> • Deepest Injection Point Operation : Calculated deepest injection point and associated production rate for given fluid and operating conditions • Gas Lift Bracketing Operation : Calculated deepest injection point and associated production rate for minimum and maximum sets of conditions (typically current and future performance.)

Feature	Description
	<ul style="list-style-type: none"> Lift Gas Response : Analyzes performance of a well under gas lift for various conditions such as changing fluid data, injection and operating conditions and/or varying injection depths. Gas Lifted Wells in PIPESIM are also exposed to all PIPESIM standard single branch and network operations and gas lift optimization.
Electrical Submersible Pumps (ESP's) (p.235)	<p>PIPESIM uses advanced methodology to perform ESP Design and associated operations. Key features include:</p> <p>ESP Database</p> <ul style="list-style-type: none"> Extensive database of Electrical Submersible Pump performance curves covering a wide range of production rates. Pumps curves are available from the following manufacturers: <ul style="list-style-type: none"> Centrilift ODI Ramco-Alnas Schlumberger-REDA Trico Wood Group Users can easily add performance curves from any manufacturer and use it for design and simulation purposes. Database also includes motors of various series and power ratings and cables of various sizes (AWG) and current ratings.

ESP Selection & Design Calculations

- PIPESIM recommends series of pumps in the order of decreasing efficiency based on target production rates and size constraints.
- ESP Design calculates the required number of stages (taking into account losses between stages), as well as performance data for pumps at design condition including head, differential pressure, intake gas volume fraction, power required, and so on.
- Effects of downhole separation, head factor tuning and viscosity corrections are considered in the design and staging calculations.
- Design results and plots Includes:
 - Standard performance curve with operating condition annotated.
 - Multispeed pump performance curves at operating conditions indicating gas volume fraction at pump intake.
 - A formatted report that includes input design data, pump performance data, motor and cable performance data.

ESP Well Simulation

Feature	Description
	<ul style="list-style-type: none"> • All PIPESIM standard operations can be used to model ESP wells (such as PT profile, nodal analysis, system analysis, network simulation.) • ESP parameters (speed, number of stages and power) are available as sensitivity variables.
Rod Pump Module (p.242)	<p>PIPESIM uses a third party Rod Pump module for the design and diagnostics. For standard simulation operations (such as, PT profile, nodal analysis, system analysis, network simulation) a rod pump may be modeled.</p> <p><u>Rod Pump Equipment Database (3rd Party)</u></p> <ul style="list-style-type: none"> • Extensive database of pumping units (including various geometric configurations and ratings) from leading vendors, including: <ul style="list-style-type: none"> • International Vendors - American Conventional, Ampscot, Baker Torqmaster, LUFKIN – Conventional, LUFKIN – Mark II, Ramco-Alnas • Chinese vendors - ER JI, DA AN, BAO JI, LAN TONG, SI JI, TONG HUA, LAN SHI, FU SHUN, SAN JI, XU ZHOU, XIN JIANG • Motor vendor: <ul style="list-style-type: none"> • International Vendors – Sargent, General Electric, Reliance, Robbins & Meyers, Westinghouse, Baldor, • Chinese vendors - ER JI, DA AN, BAO JI, LAN TONG, SI JI, TONG HUA, LAN SHI, FU SHUN, SAN JI, XU ZHOU, XIN JIANG • Extensive database of Steel and Fiberglass Rods of various grades from API, Axelson, Continental Emmsco, COROD, Metalmecanica UHS, Norris, Trico, Upco, Weatherford XD, etc. <p><u>Rod Pump Design Module (3rd Party)</u></p> <ul style="list-style-type: none"> • Derives basic reservoir, wellbore and fluid data from base PIPESIM model. • Clockwise & counterclockwise crank rotations. • Rod String selection may be calculated or user specified. Supports tapered rod design and sinker bars. • Design basis may be defined by pump intake pressure, fluid level, target production rate or fixed pumping parameter. • Support for downhole gas separation. • Design results include: <ul style="list-style-type: none"> • PLOTS: Pressure profile of wellbore, rod loading analysis/stress plot, pump efficiency plot. • REPORT: Formatted report showing design parameters, hydraulic condition across pump, motor sizing, polished rod loading, rod string stress analysis, pump efficiency analysis, and so on. <p><u>Rod Pump Diagnostics (3rd Party)</u></p> <ul style="list-style-type: none"> • PIPESIM supports various dynocard file formats.

Feature	Description
	<ul style="list-style-type: none"> Result include calculated dynocards (surface, rod sections and pump), torque curve, load balance analysis, rod string stress loading analysis, pump conditions, and so on. <p>Rod Pump Simulation</p> <ul style="list-style-type: none"> PIPESIM allows simulation of rod pumps as downhole equipment with required data, including support for downhole separator (optional) and the ability to recombine annulus gas at the wellhead.) Rod pumping wells can be simulated for all types of standard PIPESIM operations including network simulation (wells must be flow rate specified.)
Progressive Cavity Pump (PCP) (p.465)	<p>PIPESIM uses industry standard methods to simulate progressive cavity pumps. Key features include:</p> <p>PCP Pump Database</p> <ul style="list-style-type: none"> PCP's performance curves for various sizes and nominal rates are available from the following manufacturers: <ul style="list-style-type: none"> PCM Weatherford Users can easily add performance curves from any manufacturer. <p>PCP Well Simulation</p> <ul style="list-style-type: none"> PIPESIM supports PCP's with both top and bottom drive configurations. Options to adjust head factor, viscosity corrections and gas separation.

Table 1.6: Artificial Lift Design and Simulation

Feature	Description
Single Branch Operations (p.50)	<p>Single-branch operations pertain to analysis performed on a model configuration that has typically a single source fluid and single delivery point. These sources can be completed reservoir layers (in a well) or a generic source representing a surface inlet stream. There are special scenarios where multiple sources are allowed but the flow path must have no branching. These include:</p> <ul style="list-style-type: none"> Multilayer wells Fluid/chemical injection at any point in the branch Lift Gas injection (tubing or riser base) <p>Key single-branch operations include:</p> <ul style="list-style-type: none"> Pressure Temperature Profile: Reports many detailed variables (such as, flow, pressure distribution, fluid properties, thermal properties, multiphase flow characteristics and flow assurance parameters) over the length of flow path. Solves for the unknown boundary condition – pressure or flowrate. Equipment operating conditions if both pressures and flowrate are provided.

Feature	Description
	<ul style="list-style-type: none"> • Sensitivity analysis for model objects, fluid properties, boundary conditions. • Profile plots with hundreds of potential result variables available. • Used for well performance analysis, well design, pipeline design, flow assurance, and many other analysis. • Nodal Analysis: Standard well performance analysis operation that can also be applied to a simple pipeline system. Applications include well & completion design, artificial lift selection and design, equipment sizing, system debottlenecking, flow assurance analysis and many other applications. <ul style="list-style-type: none"> • Inflow & outflow sensitivities • Nodal Plot – with several plot control options • System Analysis: One of the most versatile analysis tools in PIPESIM allows users to analyze performance of production and/or injection systems (well, pipeline, etc). It has advanced sensitivity options that enables varying multiple parameters through either permutations or on a case-by-case basis • Flow Correlation Comparison: Aids in selection of suitable flow correlation by comparing several correlation against measured data. • Data Matching: Advanced tool that uses optimization techniques to select and match best suited flow correlations and adjust friction, holdup and heat transfer factors to match field pressure and temperature data. • Optimum Horizontal Well Length: Specific operation to optimize horizontal well design. • Reservoir Tables: Generates vertical lift performance curves for both production and injection wells allowing sensitivities for flowrates, fluid properties, outlet pressure and artificial lift quantities. Results are generated in specific formats for several reservoir simulators: <ul style="list-style-type: none"> • Eclipse • Pores • VIP • Comp4 • MoRes • Well Performance Curves: Specific well performance analysis operation to generate PQ curve for the well for multiple sensitivities. • Artificial Lift Performance Curves: Specific well performance analysis operation to generate PQ curves for artificially lifted wells for multiple sensitivities. • Gas Lift Rate vs Casing Head Pressure: Specific operation for gas lifted wells to determine the required casing head pressure to achieve a target production rate.

Feature	Description
	<ul style="list-style-type: none"> Wax Deposition: Predicts thickness and volume of wax deposition over time. This operation requires specific wax properties files based on selected deposition models.
Network Simulation	<p>PIPESIM has a very advanced network solver that can solve any type of network including large and complex networks having thousands of well and branches and may include multiple loops and crossovers.</p> <p>Key features</p> <ul style="list-style-type: none"> Solves for unspecified boundary conditions (pressure and/or flowrate). Handles multiple rate constraints.
Well Optimizer	<p>PIPESIM offers very powerful network optimization capabilities to maximize production from a network of wells (self flowing, wells with choke, wells on gas lift/ESP) under various types of constraints. This optimizer has been successfully used to optimize large network containing thousands of wells.</p> <p>Key features of the Well Optimizer</p> <ul style="list-style-type: none"> Optimization for maximum oil or Liquid production using: <ul style="list-style-type: none"> Optimum lift gas distribution (for gas lifted wells) Optimum ESP power/speed settings (for ESP wells) Optimum choke setting (for wells under choke control) Selected wells ON/OFF (to reduce system back pressure) Any combinations of the above Several state-of-the-art algorithms are available: <ul style="list-style-type: none"> Newton-Raphson Genetic algorithm SDR Lexico SDR MINLP Comprehensive constraint handling capabilities: <ul style="list-style-type: none"> General: Well stability, Min/Max flowrates (liquid, water), well drawdown limits, control for gas coning, bubble point pressure margin Flow assurance: erosion control, CO2/H2S limits Operational constraints: Available lift gas, available ESP power, facility capacity limits, and so on Gas Lifted Wells: Lift gas limits (min/max), casing head pressure, dual string wells ESP Wells: ESP Power limit (min/max) Wells on Choke: Min/max choke setting Any combinations of the above

Feature	Description
	<ul style="list-style-type: none"> Other associated features include network validation, group constraints application (high watercut wells, sour gas producers, and so on) Several types of results and plots are available including and overall solution plot, comparison tables (to compare results from multiple scenarios), results overlay on a network diagram (bubble map, and so on.)

Table 1.7: PIPESIM General Simulation/Optimization Modules

Feature	Description
Liquid Loading (p.396)	<ul style="list-style-type: none"> Liquid loading analysis is available primarily to determine the minimum stable flow rate for vertical gas wells. User can tune predicted calculation by applying a correction factor. Analyze a well or network for locations susceptible to liquid loading.
Hydrates (p.552)	<ul style="list-style-type: none"> Uses the Multiflash compositional package to generate hydrate formation curves on the phase envelope. Users can create a production profile superimposed on a phase envelope to predict occurrence and location of Hydrate formation. Report hydrate formation temperatures and/or hydrate sub-cooling delta-temperature to determine occurrence and location of hydrate formation in a single well or large networks. Analyze effects of hydrate inhibitor and determine the required treatment quantity to prevent hydrate formation.
Asphaltenes (p.551)	<ul style="list-style-type: none"> Uses the Multiflash compositional package to generate Asphaltene formation curves on the phase envelope. Users can create a production profile superimposed on a phase envelope to predict occurrence and location of Asphaltene formation. Report Asphaltene formation temperatures and/or Asphaltene sub-cooling delta-temperature to determine occurrence and location of Asphaltene in a single well or large network.
Waxes (p.554)	<p>There are several modules in PIPESIM for wax precipitation and deposition analysis:</p> <ul style="list-style-type: none"> Multiflash thermodynamic prediction module: <ul style="list-style-type: none"> Generates a wax formation curve on the phase envelope. Superimpose a production profile on top of the phase envelope to predict occurrence and location of wax. Report critical wax formation temperatures and/or sub-cooling delta-temperature to determine occurrence and location of wax in a single well or large network. DBR Wax deposition module:

Feature	Description
	<ul style="list-style-type: none"> DBR-Solids package provides detailed characterization of fluid for wax properties. PIPESIM wax deposition module uses wax properties file generated by DBR-Solids to perform detailed thermodynamic calculation to predict occurrence and quantity of wax deposition. Reported parameters include wax volume in pipeline (at different time steps), wax deposition rate in pipeline, wax thickness along pipe profile (at different time steps), wax thermal properties (thickness, conductivity, heat transfer coefficient, and so on.)
Emulsion Modeling (p.146)	<ul style="list-style-type: none"> Available Emulsion Models include: <ul style="list-style-type: none"> Use Continuous Phase Viscosity Volume-weighted mixture viscosity Woelflin loose, medium and tight models Brinkman Vand (Vand coefficients, Barnea & Mizrahi coefficients or user specified coefficients) Richardson (with tuning for K factor) Leviton & Leiton User specified Emulsion Table Inversion watercut may be user specified or calculated using the Brauner-Ullman equation
Corrosion Modeling (p.176)	<ul style="list-style-type: none"> DeWaard Corrosion Model for predicting CO2 corrosion. Tuning options for efficiency User may override pH calculation or derive from ScaleChem PVT files Effect of corrosion inhibitors (MEG, DEG) are accounted for Available results include corrosion rate, pH, glycol inhibition factor, and many other variables
Erosion Modeling (p.161)	<ul style="list-style-type: none"> Erosion models available include: <ul style="list-style-type: none"> API 14E Salama Model (for sand-laden fluids) Tuning options for efficiency Available results include erosional velocity limit, erosional velocity ratio, erosion rate and many other variables.
Slug and Pigging Analysis	<ul style="list-style-type: none"> Slug length correlations Slug growth calculations Probabilistic slug length distribution

Feature	Description
	<ul style="list-style-type: none"> • Severe riser slugging indicator • Pig generated slug volume calculation
Scaling Analysis (p.163)	<ul style="list-style-type: none"> • Ability to use ScaleChem generated PVT file for predicting occurrence, type location and severity of scale formation

Table 1.8: PIPESIM Flow Assurance Capabilities

1.1.3 Reporting

- Flexible plotting functionality with hundreds of potential variables to select from
- Configuration of multiple y-axis
- Detailed spot reports including phase splits and flow regime maps
- Interactive network simulation result viewer
- Summary and Detailed output report
- Specially formatted reports for nodal analysis and gas lift design

1.1.4 Extensibility

PIPESIM includes a rich and fully documented API called Openlink that enables integration with various other software products, including:

Avocet IAM	Schlumberger product used to integrate PIPESIM to reservoir, process and economics models.
Avocet	Schlumberger production operations platform used to integrate PIPESIM to high-frequency production data.
HYSYS	Process simulation software developed by Aspen Technology. PIPESIM single-branch and network models can enhance the models built in the process simulation software HYSYS. For more information, refer to the Aspentech website.
UniSim	Process simulation software developed by Honeywell Process Systems. PIPESIM single-branch and network models can enhance the models built in the process simulation software UniSim Design. For more information, refer to Honeywell website.

1.1.5 PIPESIM Hot Keys

Hot keys are short cuts for menu options.

File Hot Keys

The following specific hot keys are available:

Create New Single Branch Model	CTRL+W
Create New Network model	CTRL+N

Open model	CTRL+O
Open engine file for text edit	CTRL+T
Save model	CTRL+S
Close PIPESIM	ALT+F4
Export to Engine file	CTRL+E
Purge Engine Files	CTRL+Y
Open Single Branch Wizard	CTRL+ALT+W

Simulation Hot Keys

The following specific hot keys are available:

Run model	CTRL+G
Restart Model (applicable only for PIPESIM Net models)	CTRL+R

Windows Hot Keys

The following specific hot keys are available:

New Model Window for Selection	CTRL+ALT+N
Close Active Window	CTRL+F4
Go to Next Window	CTRL+F6 or CTRL+TAB
Go to Previous Window	CTRL+SHIFT+F6 or CTRL+SHIFT+ TAB

Tools Hot Keys

The following specific hot keys are available:

Print	CTRL+P
Access Help	F1

Editing or General Hot Keys

The following hot keys are available:

Access Pull-down menus	ALT or F10
Cut	CTRL+X
Copy	CTRL+C
Paste	CTRL+V
Delete	Del
Select All	CTRL+A
Find	CTRL+F

Sticky key mode	SHIFT
Zoom in	SHIFT+Z
Zoom out	SHIFT+X
Zoom Full View	SHIFT+F
Restore View	SHIFT+R

1.1.6 Main Toolbar

The main toolbar (like all toolbars in PIPESIM) is a docking toolbar. That is, it can be re-positioned on the screen.



From left to right, the icons are:

1. [New model \(p.28\)](#)
2. [Single Branch Wizard \(p.28\)](#)
3. Open an existing model,
4. Save active model (p.35)
5. Save as (p.35)
6. Save all open models (p.35)
7. Find (p.184)
8. Boundary Conditions (p.118)
9. Cut
10. Copy
11. Paste
12. [Run Model \(p.195\)](#)
13. [Restart \(p.212\)](#)
14. [Abort Run \(p.195\)](#)
15. View summary file
16. View Output file
17. View System plot
18. View profile plot
19. View flow regime map
20. [Report Tool \(p.262\)](#)
21. Export engine files (for FPT)
22. Help

See also: [Well Performance \(p.50\)](#), [Pipeline tools \(p.91\)](#) and [Network \(p.35\)](#)

1.1.7 Toolbox

All tools can be accessed using the **Tools** menu.

See also [Hot Keys \(p.22\)](#), [Main toolbar \(p.24\)](#), [Wells \(p.50\)](#), [Pipeline tools \(p.91\)](#), [Network \(p.35\)](#) components, [Case studies \(p.43\)](#), and [How to add objects \(p.34\)](#)

Single branch Toolbox

Button	Function
	Returns the mouse pointer to its original function. If you place an object, such as a node, in the work area, further clicks will continue to place objects of that type until the select arrow button is pressed.
	Adds a text box to the model. Any number of text boxes can be added to the model. The size and color of the text and the background can be changed.
	Allows two connection objects to be connected together where no equipment is located between them. In the network module, boundary nodes are used to identify the "ends".
	Final node in a single branch model where the branch connects to the network..
	The generic source object is a means by which you can specify explicit upstream boundary conditions of pressure and temperature in a given model.
	The vertical completion component models flow from the reservoir to the bottom hole using an Inflow Performance Relationship (IPR). A multilayer reservoir model (p.51) can be defined by several layers (completions) which, can, if required, be separated by a section of tubing.
	A horizontal completion with multiple sources along the horizontal wellbore. This takes into account reservoir drawdown and wellbore pressure drop.
	The basic pump model uses centrifugal pump equations to determine the relationship between inlet pressure and temperature, outlet pressure and temperature, flowrate, shaft power, hydraulic power and efficiency.
	A device that boosts the pressure of an oil-gas mixture.

Button	Function
 Separator (p.116)	Placing a separator in the model removes up to 100% of the gas, water or liquid (oil plus water) phase.
 Compressor (p.91)	Either centrifugal or reciprocating compressors can be modeled.
 Expander (p.94)	The basic expander model uses centrifugal expander equations to determine the relationship between inlet pressure and temperature, outlet pressure and temperature, flowrate, shaft power, and efficiency.
 Heat Exchanger (p.98)	A heat exchanger in the model allows a fluid temperature change to be modelled.
 Choke (p.443)	A choke is a device that restricts the flow rate.
 Injection Point (p.101)	Placing an injection point in the model allows a side stream to be injected without creating a new pressure boundary condition.
 Equipment (p.443)	Simulates a generic unit operation in which the pressure and/or temperature of the stream are modified.
 Adder/Multiplier (p.112)	A rate change device in the model that can increase or decrease the fluid flowrate at that point in the system
 Report (p.115)	Placing a report tool in the model gives additional reporting of the conditions at that point in the model.
 Engine Keyword Tool (p.94)	Placing an Engine Keyword Tool (EKT) in the model allows access to the PIPESIM Input Language
 Nodal analysis Point (p.88)	Defines where the system is to be broken in two for the Nodal Analysis operation
 Connector	Allow two objects to be connected by a "zero-length" flowline. This is normally used to connect two items of equipment together where there is no significant pressure or temperature change between them.

Button	Function
	Placing a flowline in the model allows the modeling of horizontal or near-horizontal flow (up or downhill).
	Placing the tubing object in the model allows the modeling of vertical or near-vertical flow (production or injection) in a well bore.
	Placing a riser in the model allows the modelling of vertically or near-vertical flow (up, down or inclined).

Network Toolbox

Button	Function
	Allows the user to select, drag and drop any object in the working window
	Allows a text box to be added to the model.
	A junction is a location in the model where two, or more, branches meet. The fluid from the incoming branches are then mixed at the junction. The junction itself has no associated pressure drop.
	A branch is an object that connects two junctions or a well sources/sink to a junction. A branch may contain many equipment objects.
	A point in the network where a stream removed from a separator can be directed to an injection well or sink.
	A source is a point where the fluid enters the network. A network model can have any number of sources.
	A sink is a point in the network where the fluid leaves the system. Normally used to represent a surface outflow point as opposed to an injection well.
	A production well is a well where the fluid enters the network.

Button	Function
 Injection well (p.39)	A injection well is a well where the fluid exits the network.
 Folder	Allows parts of network to be "collapsed" in to a sub-network of the main model. This could be used to divide a large model into a number of smaller sections. Place a folder on the model window and double-click to enter. A sub-network can then be built in the folder. Double-click on any "white" background in a folder to take you up a level. Links can be made into the folder by connecting a node to the folder via a branch. The "dangling" end of the branch within the folder must then be connected.

1.1.8 Wizard Feature

The wizard can be used to create the following new models:

- Production well
- Injection well
- Surface and Facilities Model

Steps

To open the wizard, click the **Single Branch Wizard** button on the [Main Toolbar \(p.24\)](#). The wizard involves the following steps:

1. Supply general project information.
2. Select the model type required.
3. Select the operation type and, if it's Nodal Analysis, the Nodal Analysis point.
4. (Optional) specify the model file name and directory.
5. Select the [units \(p.170\)](#) to use.
6. Select the fluid model type, one of the following:
 - Black Oil
 - Compositional
 - PVT File
7. Select the following flow correlations (also friction and holdup factors, and swap angle):
 - Vertical flow
 - Horizontal flow
 - Single phase
8. Identify the model source, one of the following:
 - vertical completion
 - generic source

- horizontal completion
9. Identify Pipes and Equipment objects to include in the model, by using one of the following methods:
- Double-click on the object type to add it to the model.
 - Highlight the object then use the **Add Pipe>>** or **Add Equip>>** button.
-

Note: Objects MUST be added in the correct order (from source to sink), for example, tubing, flowline, riser, choke, compressor, and so on.

- 10.(For an injection model only) Identify the model sink.
 - 11.The required objects are connected in the model window. If the model window appears to be blank, scroll down to locate the model.
 - 12.Click **Finish** to close the wizard.
-

Note: Do not forget to save the model.

1.1.9 Find

Use the **Find** tool to find quickly any object (well, source, flowline, and so on) in a PIPESIM model.

Finding an object in a network model

To find an object, do the following:

1. Launch the Find tool from the [main toolbar](#) (p.24) or by using **Edit > Find**.
2. The **Find** dialog lists object types (branch, well, source, sink, junction).
3. Do one of the following:
 - enter the object name in the text box and click **Find**. The appropriate part of the tree structure opens to show it selected.
 - select the object type in the tree and then select the actual object, by name.
4. The chosen object is highlighted in the model and the screen display updated.
5. The **Edit** button can be used to modify the data.

1.1.10 PIPESIM Differences from other Simulators

It is important to understand how PIPESIM works in order to assess its performance in comparison with other network simulators, which may or may not appear to be faster. PIPESIM differs from other simulators in the following ways:

- PIPESIM is a multiphase flow simulator. Other simulators with apparent faster performance may be single-phase simulators, which cannot capture important multiphase effects.
- PIPESIM can model general networks including loops and crossovers. Other simulators may be limited to solving gathering networks only (multiple sources, 1 sink).

- PIPESIM does not require (good) initial estimates at each source and sink, which may be a requirement for other simulators.
- PIPESIM does not require (good) internal node estimates, which may be a requirement for other simulators.
- The tolerance in PIPESIM may be defined differently from other simulators.
- PIPESIM performs a rigorous heat balance, which may not be the case for other simulators.
- Other simulators may have to define the fluid composition for each branch in the model at the start of the simulation, before the flow rates are known! This is not a PIPESIM requirement.
- PIPESIM rigorously checks for network inconsistencies, for example elevation mismatches, prior to the simulation, which is a step other simulators may skip.
- Other simulators may need to have non-return valves placed in lines to indicate the direction of flow. This is not a PIPESIM requirement.
- PIPESIM has a strong and rigorous fluid Compositional PVT characterization supported by the Multiflash package, which is also embedded in OLGA, allowing better alignment and transition from steady-state to transient workflows.
- PIPESIM includes more PVT correlations for heavy oil characterization.
- PIPESIM includes a comprehensive list of flow correlations; single-phase, multiphase, empirical and state-of-the-art mechanistic flow correlations such as the OLGA-S correlations.
- PIPESIM has more engineering tools for flow assurance analysis (hydrates, asphaltenes, wax).
- PIPESIM data matching is more rigorous as the (U value and pressure hydraulics) are simultaneously tuned to give a more accurate thermo-hydraulic representation of the system being modeled.

1.1.11 PIPESIM versions

Only one version of PIPESIM can be installed on a machine at a time. To install a new version, first uninstall any existing version. PIPESIM models are forward compatible. For example, a model created using PIPESIM 2009.1 can be read and used with PIPESIM 2011.1. However, models are not back-compatible. Once it has been saved with a version of PIPESIM it can not then be read by older versions. Versions of PIPESIM prior to 2001 may need to be [converted \(p.30\)](#) before they can be used.

PIPESIM Suite (Build26) to PIPESIM Conversion

The converter has been tested with PIPESIM Suite Build 26, release in 1999. Previous versions may not convert correctly.

The following Build 26 files can be converted:

- PIPESIM: *.PSW
- PIPESIM-Net: *.NET and *.PSW

The following do not need conversion. They can be loaded directly in PIPESIM:

- Compositional: *.PVT. These do not need to be converted. Load the PVT file directly into PIPESIM using the **Setup » Compositional.... » Import** menu option.

- Plotting: *.PLT and *.PLC. PLOT (PLT, PLC) files do not need to be converted. Load the plot file directly into the **PsPlot** tool.

What is not converted

PIPESIM models cannot be converted to PIPESIM Suite models.

The following options from Build 26 input files are not converted:

- The Black box object does not exist in PIPESIM. It is converted into a heat exchanger.
- Gas Lift Design
- Horizontal completion data
- PIPESIM operations
- Upstream inheritance is not supported. Therefore any models that use this feature will have data (normally flowline ID) missing.

Pre-conversion preparation

Before conversion, do the following:

- Back up your data.
- Load the PIPESIM Suite model into Build 26 and run the model with Build 26 to verify that the results are correct.

Note: The converter does not prompt for file save; any existing previous conversion is overwritten.

Converting a model

Do the following:

1. Select **File** » **Import Build26**.
2. Open the required Build 26 model (*.psw or *.net). The file is converted and written to the same directory as the build 26 input, along with a log file.
3. For a large PIPESIM-Net model, if an Out of Memory error appears, use the conversion utility instead.

Using the conversion utility

Use this external method of file conversion if there's an Out of Memory error (due to a large file), or if a number of models are to be converted and used in PIPESIM at a later date. Do the following:

1. From the Windows Start menu, select **Programs** » **Schlumberger** » **PIPESIM** » **Utilities** » **B26 to P2K Converter**.
2. This opens the **Network File Converter** dialog. In its **Options** menu, choose whether to overwrite existing files. This must be off (don't overwrite) if the file is being converted because of a an out of memory error.
3. From the converter's **File** menu, choose the Build 26 file (*.psw or *.net) to convert

4. If an Out of Memory message appears, click **Cancel** and rerun the conversion. With the option not to overwrite, the converter starts from where it last stopped. Otherwise it restarts completely..
5. When the model has been successfully converted, all the *.bps files in the directory can be deleted.

Post-conversion QC

After conversion:

- The log file *.lgg for PIPESWIM, or *.p2k for PIPESIM-Network, is in the same directory as the PIPESIM model.
- The new model is saved to the same directory as the Build 26 model. The file extension for a PIPESIM single branch model is .bps., and .bpn for a PIPESIM Network model. All input data is now saved in one file.
- The PIPESIM file might be large. All flowlines are converted as if they were defined in PIPESIM as detailed profiles, that is the node data is imported. This is done so that no important data is lost. This can be reversed in PIPESIM by using **Setup » Flowline Properties » Simple profile**. All detailed profiles are then ignored.
- Check the default units. The converter saves the model in Engineering units.
- Run the new model in PIPESIM and verify that the results are the same as those produced by the PIPESIM Suite. There may be small differences due to changes in the calculation engine.
- If you have any problems with the file conversion, please send a copy of the file(s) to [PIPESIM support](#).
- Check the default units. The converter will save the model in Engineering units.
- Run the new model in PIPESIM
- Verify that the results are the same as those produced by the PIPESIM Suite. There may be small differences due to changes in the calculation engine. If you are unsure about any results, please contact [PIPESIM support](#).
- If you have any problems with thefile conversion, please send a copy of the file(s) to [PIPESIM support](#)

Troubleshooting

If the results from PIPESIM Suite and PIPESIM are not the same, please try the following before contacting the PIPESIM support.

1. Units settings -Save (a copy) of the PIPESIM Suite model with the following settings:
 - a. Set the units to standard Eng or SI, that is not customized. Use **Setup » units » Eng**.
 - b. Set the option to write file with default units, using **Preferences » Options » General » Write file with default units**.
 - c. All units can be restored later in PIPESIM.
 - d. Repeat the conversion.
2. Language -Save (a copy) the PIPESIM Suite model with the following settings

- a. Set the language to English by using **Setup** » **Preferences** » **Language** » **English** or **Setup** » **Preferencias** » **Idioma** » **English**.
- b. Repeat the conversion.
3. Contact [PIPESIM support](#) with a copy of your input file(s). Do not forget that for PIPESIM-Net models both the *.net and *.psw files are needed.

1.2 Building Models

1.2.1 Steps in building a model

This topic outlines the basic steps involved in building models.

Basic overview

The steps in building a PIPESIM model are slightly different for each module, but all involve the following basic steps:

1. Select units.
2. Set fluid data and (optionally) calibrate it.
3. Define components of the model.
4. Add well components (completion, tubing).
5. Add pipeline components.
6. Add field equipment.
7. Set heat transfer options.
8. Select flow correlations.
9. Save the model.
10. Perform an operation.
11. Analyze the graphical and tabular results.
12. Use the schematic.

Creating a new model

To create a new model, select **File** » **New** and then select from the following:

- pipeline and facilities model
- well performance model (production and injection wells)
- network model (more than one well or source)

Alternatively, use the [wizard \(p.28\)](#).

A model is made up of "node" and "connection" type objects.

Before an [operation \(p.195\)](#) can be performed on the model it must be saved. Single branch models have the file extension .bps and network models, .bpn.

Adding objects to a model and connecting them

The objects in the [toolbox \(p.25\)](#) are defined as either node or link objects, as follows:

- node objects — for example, source, sink, junction/node, completion, equipment, Nodal analysis point, report tool, and so on. Add node objects to the model first. Select a node object in the toolbar then click to place it in the model window. To add a number of the same type of node object, press the SHIFT key to turn the selector into a "sticky" mode where the same object is added to the model window whenever the mouse button is pressed.
- link (connection) objects — for example, tubing, flowline, riser, connection, and so on. Link type objects are used to connect two node type objects. The node objects must have already been added. Select the link object from the toolbox. Hover the mouse over a node object, then press and hold down the (left) mouse button and "drag" the resulting link to another node object.

Adding data to an object

After adding the object to the model, double-click it to open its data entry screen. All the data in an object is self-contained and does not affect, or rely, on data from any other object.

Minimum data

Missing data is reported at the following levels:

- Dialog
- Object
- Folder

All dialogs that have data entry fields display a red box around any mandatory data. This shows the minimum data required. The red box disappears when the data is entered.

In the model objects and folders that have data missing are displayed with a red box around them. Missing data includes data in dialog fields with a red box, the fluid model, and boundary conditions. Again, once the necessary data has been entered, the red box disappears.

Data range checks are made on some data entry fields.

Duplicating an object

The **Edit » Copy** and **Edit » Paste** (or Ctrl+c, Ctrl+v) commands can be used to copy an object and all its associated data. The only data that is changed in the copy is the identifier of the object.

This option works in both single branch and network mode. It also works on multiple selections, so you can duplicate more than one object at a time.

Disconnecting objects

To disconnect objects, do the following:

1. Select the link (connection) object. End markers (small squares) appear at each end of the object.
2. Select the end marker (small square). The pointer changes into an arrow.
3. Press the left mouse button and hold it down while dragging the link to its new node object.

Saving the model

The PIPESIM models are stored in binary data files with the following extensions:

- .bps - single branch model
- .bpn - network model. A network model is stored in a single input file. It is not necessary to store each network model in a separate directory. However, it is important to note that each individual branch has its own output files. Thus, using separate directories ensures that the output files are not overwritten. Using separate directories may be useful in the following cases:
 - If results from models having the same well/branch names are to be compared.
 - If models having the same well/branch names are to be run simultaneously.

1.2.2 Network Models

Network models contain multiple single branch models, linked at junctions. They can be used to model parts of, or complete, production or injection systems, from the reservoir to the final delivery point(s).

Creating a network model

To build a network model, perform the following basic steps:

1. Select a [units \(p.170\)](#) set.
2. Develop the network model. The basic building blocks — sources, sinks, branches and junctions — can be found in the [Network Toolbox \(p.27\)](#). Branches can be built up of equipment items, using the [Single branch Toolbox \(p.25\)](#). Prebuilt single branch models of wells and flowlines can also be used.
3. [Set the fluid properties \(p.127\)](#).
4. (Optional) [Calibrate the fluid \(p.505\)](#).
5. [Set the boundary conditions \(p.41\)](#).
6. [Save the model \(p.35\)](#).

Network Operations

Operations for network models differ from those available for single branch models. They are:

- [Run Model \(p.195\)](#)
- [Restart Model \(p.212\)](#)
- Gas Lift Optimisation
- Linking to reservoir models

Source

A source is a point where fluid enters the network. A network model can have any number of sources. A [production well \(p.38\)](#) can be used instead of a source.

Right-click on a source to display the following menu:

Data	Access the Source Properties (p.36) and Fluid Model (p.181) tabs
Fluid Model	Access the Fluid Model tab
Active	If active is selected, the source is included in the model at run time, otherwise it is left out. This flag can be used to determine the effects of switching off parts of the network without having to delete any data.
Plot Results	Plots the results for the branch containing this source.
Cut / Copy / Paste / Delete	Allows items in the network diagram to be cut, copied, pasted and deleted.
Move to Top / Move to Back	For use when several icons are overlaid, this moves the selected icon to the top or bottom.

Source Properties tab

Double-click a source to open the **Source Properties** tab, where you can set the following parameters:

Temperature	Fluid Temperature at the inlet
Pressure/Flowrate Boundary Condition	Specify the boundary condition in terms of a fixed pressure and/or flowrate. The flowrate type can be selected. The flowrate can be entered in either stock tank or flowing conditions by using the "@" button.
PQ Curve	Specify the boundary conditions in terms of a Pressure versus Flowrate (PQ) curve (there must be at least 2 data points and a maximum of 30)
Type	Flow type for the PQ Curve flowrate data.

Sink

A sink is a point in the network where the fluid leaves the system. It is normally used to represent a surface outflow point as opposed to an injection well. A network model can have any number of [sinks \(p.118\)](#). An [injection well \(p.39\)](#) can be used instead of a sink.

Right-click on a sink to display the following menu:

Data	Access the Sink Properties (p.37) tab
Active	If active is selected, the sink is included in the model at run time, otherwise it is left out. This flag can be used to determine the effects of switching off parts of the network without having to delete any data.
Plot Results	Plots the results for the branch containing this sink.
Cut / Copy / Paste / Delete	Allows items in the network diagram to be cut, copied, pasted and deleted.

Move to Top / Move to Back	For use when several icons are overlaid, this moves the selected icon to the top or bottom.
-----------------------------------	---

Sink Properties tab

Double-click a sink to open the **Sink Properties** tab, where you can set the following parameters:

Pressure Outlet pressure

Flow rate Flowrate in Liquid/Gas/Mass units at stick tank conditions. The flowrate can be entered in flowing conditions by the "@" button.

Branch

A branch is an object that connects two junctions or a source/sink to a junction. The branch can contain flowlines and equipment. [Production wells \(p.38\)](#) and [injection wells \(p.39\)](#) are special types of branches that combine a branch with a [source \(p.35\)](#) or [sink \(p.36\)](#).

Right-click on a branch to display the following menu:

Data	Gives access to the Branch Properties (p.37) , Limits (p.42) and Flow Correlations (p.120) tabs.
Flow Correlations	Gives access to the Flow Correlations tab.
Import Single Branch Model	Allows a model already defined in a single branch mode to be imported. The file name of the model to import will be requested.
Active	If active is selected, the branch is included in the model at run time, otherwise it is left out. This flag can be used to determine the effects of switching off parts of the network without having to delete any data.
Switch Flowline Geometry	Changes the assumed direction of flow for this branch.
Plot Results	Plots the results for this branch.
Cut / Copy / Paste / Delete	Allows items in the network diagram to be cut, copied, pasted and deleted.
Move to Top / Move to Back	For use when several icons are overlaid, this moves the selected icon to the top or bottom.

Branch Properties tab

Double-click a branch to open the **Branch Properties** tab, where you can set the following parameters:

Block Determines whether the branch is allowed to flow in both directions. Block reverse acts as a non-return valve.

Estimates Estimates of the Pressure and/or flowrate for the branch (optional).

Network Separator

A point in the network where a stream removed from a separator can be directed to a sink or injection well. Sometimes this is also known as a re-injector.

The network separator (reinjection node) acts in a similar manner to a junction. It links to the following branches:

Incoming / feed stream branch

The branch upstream of the network separator, that is before separation takes place.

Outgoing main stream branch

The branch where the main fluid, that is the remaining fluid after separation, goes.

Outgoing separated stream branch

The branch where the separated fluid goes.

Note: This branch MUST not be connected directly to a sink. Place a node before the sink and add a dummy branch. In addition if there is a single branch connecting the separated stream branch to a sink, that branch CANNOT be constrained

The following must be defined:

- The incoming feed stream and the separated stream branches.
- Type of separator (liquid, gas, or water) and its efficiency.

Note: There will be a pressure discontinuity between the separator and the separated branch inlet. This represents the pump, compressor, or choke required to adjust the stream's pressure to that necessary to balance the remainder of the network.

See also: [How to add objects \(p.34\)](#), [Separator Details \(p.116\)](#)

Production Well

A production well is a well where the fluid enters the network. A network model can have any number of production wells. A well can also be modelled using a [well head performance curve \(p.40\)](#). This aids in the solution time of the network.

A production well is a combination of a completion, tubing, and surface and downhole equipment.

Right-click on a production well to display the following menu:

Data	Gives access to the Production Well Properties (p.39) , Limits (p.42) , Fluid Model (p.181) , Flow Correlations (p.120) and Estimates (p.42) tabs.
Fluid Model	Gives access to the Fluid Model tab.
Flow Correlations	Gives access to the Flow Correlations tab
Import Single Branch Model	Allows a model already defined in a single branch mode to be imported. The file name of the model to import will be requested.

Active	If active is selected, the well is included in the model at run time, otherwise it is left out. This flag can be used to determine the effects of switching off parts of the network without having to delete any data.
Plot Results	Plots the results for this well.
Cut / Copy / Paste / Delete	Allows items in the network diagram to be cut, copied, pasted and deleted.
Move to Top / Move to Back	For use when several icons are overlaid, this moves the selected icon to the top or bottom.

Production Well Properties tab

Right-click on a production well and select **Data** to open the **Properties** tab, where you can set the following pressure and flow boundary condition parameters:

Block	Determines whether the well is allowed to flow in both directions. Block reverse acts as a non-return valve and will stop the well from becoming an injection well. If the solution algorithm determined that a production well, with block reverse set, was an injector then it will be shut-in.
Pressure	Inlet pressure, normally the static reservoir pressure.
Flow rate	Flowrate in Liquid/Gas/Mass units at stick tank conditions. The flowrate can be entered in flowing conditions by the "@" button.
Well Curves	Models the well using a well head performance curve (p.40) .

Injection Well

An injection well is a well where the fluid leaves the network. A network model can have any number of injection wells.

An injection well is a combination of a completion, tubing, and surface and downhole equipment.

Right-click on an injection well to display the following menu:

Data	Gives access to the Injection Well Properties (p.40) , Limits (p.42) , Flow Correlations (p.120) Estimates (p.42) tabs.
Flow Correlations	Gives access to the Flow Correlations tab.
Import Single Branch Model	Allows a model already defined in a single branch mode to be imported. The file name of the model to import will be requested.
Active	If active is selected, the source is included in the model at run time, otherwise it is left out. This flag can be used to determine the effects of switching off parts of the network without having to delete any data.
Plot Results	Plot the results for this well.

Cut / Copy / Paste / Delete	Allows items in the network diagram to be cut, copied, pasted and deleted.
Move to Top / Move to Back	For use when several icons are overlaid, this moves the selected icon to the top or bottom.

Injection Well Properties tab

Right-click on an injection well and select **Data** to open the **Properties** tab, where you can set the following pressure and flow boundary condition parameters:

- Block** Determines whether the well is allowed to flow in both directions. Block reverse acts as a non-return valve and will stop the well from becoming a producer. If the solution algorithm determined that an injection, with block reverse set, was a producer then it will be shut-in
- Pressure** Outlet pressure, normally the static reservoir pressure.
- Flow rate** Flowrate in Liquid/Gas/Mass units at stick tank conditions. The flowrate can be entered in flowing conditions by the "@" button.

Well Curves

A production well in a network model can be modeled using a well performance curve of flowrate versus [outlet] well head pressure - [Well performance curves \(p.203\)](#). These well performance curve(s) are stored in ASCII files and can be created using a PIPESIM [operation \(p.195\)](#) or any another suitable Nodal analysis package.

This option has been introduced for the following reasons:

- To reduce the time required to solve large networks. A significant amount of time is spent computing well bore pressure losses.
- To ensure that the well operates in its stable region.
- To interface with 3rd party applications so that they can create these well curves, that is Shell's WePs module.
- To ease debugging when wells do not flow and the network solution does not converge . In this case the individual well curves can be examined to insure that the well will flow at the required pressure.

To simulate a well, right-click on it and select **Data**. On the **Properties** tab, select the **Well Curves** check box. Each production well can be simulated by one of the following options:

Create (during network run) only when necessary

The time stamp of the last created performance curve is checked. The curve is regenerated automatically, if necessary. Once the file has been created, it can be viewed using the **Plot** button.

Create on every network run

The performance curve is always regenerated. This is time consuming.

Online

Model the well online (the default). The well's details must be included in the PIPESIM Network model.

Offline from a well performance curve file

The curve is generated by the simulator automatically, as it is needed. The well's details must be included in the PIPESIM Network model.

With the curve(s) being pre-generated, the well's details need not be included in the PIPESIM Network model. The data file may contain a number of well head curves over a range of values, for example, reservoir pressure, watercut, and so on, so that it need only be recreated if the operating conditions are outside the specified ranges. Before the network can be solved, the exact value to use during the simulation must be defined.

Data Entry

If use Well Curves is selected.

Use File

Use a specific performance curve file for this well. A single file can be used by a number of wells. The file can contain data over a range of values or a single point. Once you specify the file, the options are as follows:

Sensitivities

Interrogates the performance curve file to determine what range of values it was created over. A specific value for each variable must then be defined. Extrapolation is not allowed.

Plot Interpolated Curve

Plots the resulting interpolated curve that will be used during the simulation.

Plot

Plots all curves in the file.

Boundary Conditions

The network engine solves the mass, momentum and energy conservation equations for the fluid pressures, flow rates and temperatures, in a network. Users must specify acceptable boundary conditions. To view and/or modify boundary conditions, select **Setup > Boundary Conditions....**. This also displays the number of boundary conditions that are required and the number that have been set.

Hydraulic boundary conditions (pressure and flow)

The requirements are as follows:

- The number of boundary conditions required for a model is known as the model's Degrees of Freedom. This is equal to the total number of boundary nodes. That is, number of wells (production and injection) + number of sources + number of sinks.
- A boundary condition specifies the pressure (P) or flowrate (Q). For sources, a pressure-flowrate curve (C) can be specified.
- At least one pressure boundary condition must be specified.

- Normally there should be one boundary condition (P, Q or C) for each boundary node. However, you can specify two boundary conditions, pressure and flow rate (PQ) at some sources or production wells, and specify no boundary conditions at some sinks, as long as the total number of boundary conditions equals the Degrees of Freedom.

Note: Use this flexibility with care as it may produce networks that cannot be solved.

Temperature boundary conditions

Users must specify the fluid temperature at all sources and the reservoir temperature for all production wells. The fluid temperatures at all sinks and injection wells are always calculated. Energy will be transferred from the fluid to the environment by heat loss, so ambient temperatures and heat loss properties must be specified for each branch in a network.

Input fluids

The [Fluid model \(p.181\)](#) must be specified for each production well and each source.

Network Constraints

To set network constraints, select **Setup** » **Flowrate Limits....**

You have two options:

- Set flowrate limits for each branch in a network model. The flowrate limits apply both to forward and reverse flows. So a flow limit of 20 STB/d will limit the flow to 20 STB/d independently of whether the flow is backward or forward.
- Set flowrate limits for an individual network branch (network connector - flowline). Right-click on the branch and select data, then click on the **Limits** tab.

The following flowrate limits can be set:

- [Mass \(p.619\)](#)
- [Liquid \(p.619\)](#)
- Water
- Oil
- [Gas \(p.619\)](#)

Network Estimates

Each time the network is solved, an initial estimate of the unknown boundary condition (inlet pressure, outlet pressure or flowrate) in each branch is required.

Internal default estimates are provided as follows:

- Production/injection well static pressure = 5,000 psia
- Source/sink/node pressure = 1,000 psia
- Flowrate = 10 lb/s

However, it is possible to speed up network convergence by providing good estimates of the unknown boundary condition in each branch. User estimates can be supplied locally and globally,

and four hierarchy options are available to specify which estimates should be used. To do this, select **Setup** » **Estimates**. The following parameters are available:

Do not use any estimates

The internal default values, shown above, are used. This is sufficient in most cases.
(Default)

Use local if present else use global

If local estimates have been set, use these. Otherwise, use the global values (if set), otherwise use the internal defaults.

Use local everywhere

If local estimates have been set, use these. Otherwise, use the internal defaults.

Use global everywhere

If global estimates have been set, use these. Otherwise, use the internal defaults.

Set the local estimates by using the **Pressure** and **Flowrate** fields.

To aid convergence and speed of solution, the results from a previous converged solution can be used as initial estimates. To do this, use the [restart \(p.212\)](#) feature. This overrides all settings here, including the internal defaults.

Optimizing PIPESIM Network Simulation Performance

In general, Performance is a trade-off between speed and accuracy. When dealing with models that are taking too long to run, there are several approaches that can be taken to improve the network speed. Improving the speed may compromise the accuracy and you may need to reverse some of the changes outlined in the approaches below to restore the appropriate level of accuracy, once you have fine-tuned the model.

Approaches for Improving Network speed

- Approach 1: Change the PIPESIM execution and reporting settings
 - Approach 2: Make high and low-level changes to the PIPESIM model
-

Note: After using the approaches above to improve the network speed and fine-tune the model, it is important that you carefully reverse some/all of the changes, in order to regain accuracy.

Approach 1: Change the PIPESIM execution and reporting settings

The options outlined below do not modify the model but attempt to reduce the engine workload to improve speed. You may need to do some trial-and-error to determine which one, or combination of options below, is best for speeding up your model.

Option	Details
Increase the no. of allocated processors	PIPESIM 2012 (and newer) introduced a parallelized network solver where you can run network simulations with multiple processors to increase the speed. How do I do this?

Option	Details
	Go to Setup » Engine Preferences . Increase the Number of processes for Network engine. For more information, see Parallelized Network Solver (p.178)
Add -v1 switch	<p>Adding the -v1 switch to the engine will minimize the simulation output written to the PIPESIM engine console window.</p> <p>Note: This is already the default setting for PIPESIM versions starting from 1.30.</p> <p><u>How do I do this?</u></p> <p>Go to Setup > Engine Preferences. Click the Advanced tab and type "-v1" in the fields for the Single Branch and/or Network Engine Command Line Parameters.</p>
Use a Restart file	<p>Using a restart file initializes the simulation by using the results from the previous simulation as estimates for the unknown variables. This is most effective when you are running many similar scenarios with only small variations. If minor changes (such as flow rates, pipe dimensions, and etc.) have been made to a network, use the Restart function. However, if structural changes (such as new pipes, wells deleted, inactive branches reactivated and etc.) have occurred, run the model from scratch, or use the -r Restart option.</p> <p><u>How do I do this?</u></p> <p>Refer to the topic: Restart Model (p.212) for the steps to do this and the limitations. Before using the restart function, make a backup of the restart file (*.p00.rst) in the model folder. If the model fails to solve, the saved restart file can be used to make another attempt.</p>
Minimize engine console window	<p>Minimizing the PIPESIM-Net console window after the engine starts will allow the computer to optimize updates to the window.</p> <p><u>How do I do this?</u></p> <p>Once the simulation starts running, click Minimize on the top right of the engine console window.</p>
Run the model locally	<p><u>How do I do this?</u></p> <p>Save the model to the local PC rather than to a Network drive. This will eliminate any potential network delays. Also use a local PIPESIM license file, rather than a network license, if possible.</p>

Approach 2: Make high and low-level changes to the PIPESIM model

The following options will increase the simulation speed, but may sacrifice accuracy in doing so. Use these options to fine-tune the model, but reverse them to get more accuracy, once this is done. You may need to do some trial-and-error to determine which one, or combination of options below, is best for speeding up your model. The high level changes are easy to reverse, the low level changes might require a bit more work to reverse.

Option	Level of Change	Details
Increase the Tolerance	High and Low	<p>PIPESIM solves the network using an iterative approach. It stops the calculation when the iterative error is less than a given tolerance. Thus, the specified tolerance has a direct impact on the number of iterations and the time taken to achieve an acceptable result. The default tolerance is 1%. Increasing the tolerance will increase the speed but will compromise the accuracy.</p> <p>Note: The results with tolerance greater than 2% are not recommended.</p> <p><u>How do I do this?</u></p> <p>For a network model, go to Setup > Iterations and enter a higher tolerance value.</p>
Specify flow rate as the inlet boundary conditions	Low	<p>Specifying flow rates as the boundary conditions at inlet nodes usually result in faster performance.</p> <p><u>How do I do this?</u></p> <p>For a network model, go to Setup > Boundary Conditions. Delete the pressure boundary conditions and enter flow rate boundary conditions instead, but ensure that at least 1 pressure is specified to satisfy the criteria required for the network to solve.</p>
Change the calculation method for the Moody friction factor to Approximate	High and Low	<p>The Moody friction factor is calculated as part of the multiphase pressure drop calculations (vertical and horizontal) when the single phase flow correlation option is set to Moody or Cullender-Smith. For more information, see Single Phase Flow Correlations. (p.385) There are three (3) options for the Moody friction factor calculation. In increasing order of accuracy, they are: Approximate/Moody (refer to the Moody paper (p.582)), Explicit/Sonnad (refer to the Sonnad and Goudar paper (p.582)), and Implicit/Iterative (Colebrook-White equation or Moody chart). The default option is Explicit. Changing the calculation method to Approximate will increase the speed but decrease the accuracy.</p> <p><u>How do I do this?</u></p> <p>Go to Setup > Engine options and enter the following lines of PIPESIM keywords (For a network model, enter the keywords in the field; BOTTOM of the engine network file).</p> <p>OVERRIDE</p> <p>SPHASE MOODYCALC = APPROXIMATE</p>
Decrease the number of segments per pipe length	High and Low	<p>PIPESIM divides pipes into shorter segment lengths to do the pressure drop calculations. The greater the pipe segmentation, the better the accuracy, but the slower the performance. The default number of segments per pipe length in PIPESIM is 4. Decreasing this</p>

Option	Level of Change	Details
		<p>number to 3, for example, will speed up the simulation. Decreasing it to 2 will further speed up the simulation, but the answers may become more unstable. Furthermore, if when using the user-specified number of segments, PIPESIM encounters discontinuities, it will override the specification and this will ultimately slow down the simulation.</p> <p><u>How do I do this?</u></p> <p>Go to Setup > Engine options.</p> <p>For the single branch model, enter a value less than the default value of 4, in the Segments per pipe length field.</p> <p>For the network model, click the Option Control tab and follow the previous step.</p>
Deactivate the option to include extra one foot segments	High	<p>PIPESIM calculates fluid properties at the average pressure and temperature for each segment. The average values for these properties may not be representative for the beginning and end of the segment (i.e. the nodes), particularly if the segment is long and there are significant changes in pressure and temperature across it. PIPESIM resolves this by adding short 1 foot segments at both ends of each segment, by default. This will ensure accurate values at the start and end of each node are reported, but it also slows down the engine. If you are not interested in the exact values at the beginning and end of each node, or are performing some fine tuning, you may deactivate this option to speed up the simulation.</p> <p><u>How do I do this?</u></p> <p>For the network model, go to Setup > Engine Options. Click the Option Control tab and deselect the box beside Additional short segments either side of each node.</p> <p>Alternatively, you may enter the keywords below in the field; BOTTOM of the engine network file.</p> <p>OVERRIDE OPTION EOFS =OFF</p>
Model wells in offline mode using well performance curves	High and Low	<p>PIPESIM supports the ability to model production wells in a network, in offline mode. This basically means the wells are modeled using well performance curves for example, curves of flow rate versus wellhead pressure generated as ASCII files by running multiple sensitivities in PIPESIM or any suitable Nodal Analysis package. Choosing to model wells in offline mode in network models significantly speeds up the simulation, especially if the networks are large, because the time required to compute wellbore pressure losses is eliminated. Additionally, a smooth curve ensures the well operates in the stable region.</p>

Option	Level of Change	Details
		<p><u>How do I do this?</u></p> <ol style="list-style-type: none"> For the network model, go to Setup > Boundary Conditions. Check the PQ Curve box for each well that you want to use a well performance curve for. Right-click each well in the Network diagram and select Data. You will observe that the Well Curves box is checked. This is the result of Step 1. Choose the most appropriate of the 3 options for generating the well curves. Refer to the topic, Well Curves (p.40) for more details on each option. <hr/> <p>Note: If the 3rd option to use files for the well performance curves is chosen, the files can be generated by double-clicking on the well to go to the single-branch mode and going to Operations > Well performance curves to run the operation.</p> <hr/> <p>Refer to the topic: Well performance curves (p.203) for more details. Alternatively, if you are more comfortable with PIPESIM keywords, you may enter the keywords below under Setup > Engine Options at the bottom of the engine network file.</p> <p> OVERRIDE SETUP WOFLMODE= xxxx</p> <p>Where xxxx is the keyword for your preferred well curve option. There are 3 of them: OFF, CREATE? and CREATE. Refer to the main code SETUP (p.760) and the subcode WOFLMODE for details on each option.</p>
Changing the Flashing Settings	High	<p>In the Compositional fluid mode, flashing the fluid is computationally expensive. An option for speeding up the network is selecting a faster, but less accurate, flashing option. PIPESIM has 3 flashing options. In order of increasing accuracy but decreasing network speed, they are:</p> <ul style="list-style-type: none"> <u>Always Interpolate (fastest)</u>: This option uses interpolation between physical properties determined by in a predefined grid of temperature and pressure points. <u>Rigorous Flash when close to the Phase Envelope, interpolation elsewhere</u>: This is a compromise between speed and accuracy, which assumes that properties will change more rapidly when close to a phase boundary. Interpolation is performed whenever the grid points comprising a rectangle all show the presence of the same phases. For example, if all 4 points in the rectangle have some oil, some gas, and no water, then we assume the rectangle

Option	Level of Change	Details
		<p>lies entirely within the 2-phase region of the hydrocarbon phase envelope, so interpolation is appropriate. If however one, two or three of the points have no oil, then clearly the hydrocarbon dew point line crosses the rectangle, so a rigorous flash is required.</p> <ul style="list-style-type: none"> • <u>Always Rigorous Flash (slowest)</u>: Interpolation never occurs. Properties are obtained by flashing at the required pressure and temperature. This is the slowest but the most accurate method. <p>Refer to the PPMETHOD subcode in the Options Calculations procedure (p.609) for more details.</p> <p><u>How do I do this?</u></p> <p>Double-click on each well can change the flash settings under Setup > Flashing.</p>
Switch to a Black Oil fluid model	Low	<p>Generally, black oil fluid models run faster than compositional fluid models. However, Compositional fluid models are more accurate particularly when dealing with gas condensates and volatile oils. If your model does not undergo a lot of compositional or phase changes and/or the difference in results between running the simulation in black oil vs. compositional mode is minimal, then it would be reasonable to run the model in black oil mode to speed up the simulation.</p> <p><u>How do I do this?</u></p> <p>If you have a compositional fluid model, change it to Black Oil by selecting Setup > Black Oil. Enter the required values.</p>
Changing Flow Correlations	Low	<p>Changing flow correlations is another way of speeding up simulations, but this option should be used with great caution. Flow correlations should be chosen based on their ability to reproduce/match the flowing pressures holdups, etc. observed in the field. However, if different correlations yield similar (accurate) results but varying simulation speeds, then it would be reasonable to choose the flow correlation that yields the fastest simulation speed. The native bja package is the fastest. 3rd party flow correlations, specifically the 3-phase mechanistic flow correlations, will typically be the slowest, but most accurate.</p> <p><u>How do I do this?</u></p> <p>Change the flow correlations under Setup > Flow Correlations.</p>
Avoid loops in the network topology	Low	<p>Loops in the network require PIPESIM to do extra checks to ensure overall consistency (for example, elevation difference). Avoid loops where possible to speed up the performance.</p>
Provide reasonable initial boundary	High	<p>By default, the PIPESIM engines use internally set default estimates for the boundary conditions. Under normal circumstances, the default user estimates are satisfactory. However, if the model has</p>

Option	Level of Change	Details
condition estimates		<p>particularly extreme flow rates and or pressures, these should be entered in the User estimates dialog box. Changing these to better match the conditions of the system improves the convergence and consequently, the speed.</p> <p><u>How do I do this?</u></p> <p>Go to Setup > Estimates. Select the behavior and enter the estimates. Refer to the topic Network Estimates (p.42) for details.</p>
Follow these general tips	High and Low	<ul style="list-style-type: none"> • Try to split the model into smaller networks, which can be solved independently, before linking them all together. (This helps in troubleshooting the model.) • When first building the model, leave out equipment such as compressors and separators, then incorporate them one at a time. (Again, this helps troubleshooting.) • When using a compressor or pump, define it initially with a Delta P rather than with a power or user curve. It can be changed as required later. Also, avoid defining a compressor with discharge pressure, as this can have the effect of over-constraining a system. • Try to avoid unnecessary nodes in a network, as this increases the computing time required to solve it. • Avoid dangling or redundant branches. • If the sinks are flow rate specified, and are consistently being reported at atmospheric pressure upon simulation (see messages in engine window), try changing the boundary condition to an outlet pressure to see what flow rate can be achieved. • When first attempting to solve a large network, increase the convergence tolerance to 5% and check the validity of the results. The tolerance can later be reduced and the model restarted. • If a branch appears to be behaving strangely, or is ill-conditioned, split it into smaller segments. This aids troubleshooting and improves continuity along the branch. • If the program crashes part way through an iteration with "file open" or "macopen" errors, this is due to the processor running out of memory. Simply restart the model; the program will start from where it left off. Use the PIPESIM toolbar Restart button in this case. • Try to avoid having long flowlines and risers in the same branch.

Reversing the changes made to PIPESIM models to optimize their simulation performance

After using the approaches above to optimize the network performance and fine-tune the PIPESIM model, it is important that you carefully reverse some/all of the above changes in order to regain

accuracy. A subset of some (not all) of the changes that may need to be reversed are outlined below:

- **Tolerance:** Restore the default tolerance of 1%. Generally, increasing the tolerance above the default value of 1% will increase network speed but decrease accuracy. Decreasing the tolerance to 0.1% or lower will significantly increase the simulation time.
- **Moody friction factor:** Change the Moody friction factor calculation method back to the default, EXPLICIT or the most accurate method, IMPLICIT. Do this by replacing the keyword APPROXIMATE, which was recommended in the previous section to speed up the performance, with EXPLICIT or IMPLICIT (Refer to the previous section for Help with entering the keywords correctly).
- **Boundary conditions:** Enter appropriate boundary conditions that are fit for purpose.
- **Extra one foot segments:** Reactivate the option to add extra one foot segments under Engine options or by deleting the keywords recommended in the previous section (Refer to the previous section for details).
- **Wells offline:** Switch wells from offline mode, where they are modeled using well performance curves, back on, so the detailed wellbore pressure loss calculations can be done. Do this by unchecking the option to use PQ curves in the Boundary conditions (Refer to the previous section for details).
- **Flashing settings:** If working with a Compositional fluid, select a more accurate flashing option; *Rigorous Flash when close to the Phase Envelope, interpolation elsewhere* or *Always Rigorous Flash*. Refer to the previous section for details.
- **Flow correlations:** Select the flow correlations that most closely reproduce the rates, pressures, holdups, etc. recorded in the field.
- **Loops:** Enter accurate and representative topology for the loops in the network.

1.2.3 Single Branch Models

Well Performance Analysis Overview

Well Performance analysis involves modeling a production or injection well, including any artificial lift.

Well Performance Specific Operations

The following operations are available:

- [Nodal Analysis \(p.200\)](#)
- [Reservoir tables \(p.202\)](#)
- [Optimum horizontal well length \(p.52\)](#)
- [Artificial Lift Performance \(p.213\)](#)
- [Pressure Temperature Profile \(p.196\)](#)
- [Gas lift rate vs casing head pressure \(p.204\)](#)
- [Flow correlation comparison \(p.197\)](#)
- [Systems Analysis \(p.196\)](#)

Well Performance

Perform the following basic steps to build a well model (single or multiple completion):

1. Select the [units \(p.170\)](#) set to use.
2. Specify the well's completion type, one of the following:
 - [Vertical \(p.57\)](#)
 - [Multiple \(p.51\)](#)
 - [Horizontal \(p.51\)](#)
3. Add the necessary components to the model (tubing, choke, and so on) and define the necessary data.
4. Define the fluid specification ([black oil \(p.505\)](#) or [compositional \(p.141\)](#)).
5. (Optional) [Calibrate the fluid \(p.505\)](#).
6. Define suitable vertical correlations ([p.641](#)) and [Horizontal correlations \(p.645\)](#).
7. [Save the model \(p.35\)](#)

Multilayer Reservoir IPR

A Multilayer reservoir model can be defined by several layers (completions). These can, if required, be separated by sections of tubing.

In a multilayer model, each layer can have the following:

- A different IPR method.
- Different fluid model properties. That is, each layer can have a different watercut, GOR, or a different composition, and so on.
- Different fluid types for each layer. Oil and gas layers can be mixed.

Note: As of release 2008.1, multi-layer injection well models can be simulated in PIPESIM.

[More details \(p.74\)](#)

Horizontal Well Completion

This refers to a horizontal completion with multiple sources along the horizontal wellbore, taking into account reservoir drawdown and wellbore pressure drop. The horizontal well performance models included in PIPESIM allow the user to accurately predict hydraulic wellbore performance in a horizontal completion. As such, they are an integral part of the reservoir-to-surface analysis.

A horizontal well model can be used in all Operations modes. However, you may be especially interested in investigating the productivity of a horizontal completion using the [Optimum Horizontal Completion Length \(p.52\)](#) option.

Reservoir inflow and wellbore pressure drop equations are solved to calculate the changing production rate along the well length. [More details \(p.427\)](#)

Distributed PI mode

In **Distributed PI mode** the inflow performance is expressed as a Productivity Index (PI) per unit length which can be assigned explicitly (Distributive PI) or calculated using the following:

- Steady-State (oil reservoirs)
- Steady-State (gas reservoirs)
- Pseudo-Steady-State (oil reservoirs)
- Pseudo-Steady-State (gas reservoirs) productivity equations.

In this mode the pressure drop along the horizontal completion is computed.

These equations take account of the effect of the vertical/horizontal permeability ratio, completion skin, and reservoir thickness.

Single point PI mode

In **Single Point PI mode** inflow is assumed to be at the heel of the well only (no pressure drop along the horizontal completion is computed). The following IPRs are available:

- Pseudo Steady-State (oil reservoirs)
- Pseudo Steady-State (gas reservoirs)
- Steady-State (oil reservoirs)
- Steady-State (gas reservoirs)

Horizontal completion length

Using the **Optimum Horizontal Completion analysis** option, PIPESIM accurately predicts the hydraulic wellbore performance in the completion. This is an integral part of PIPESIM's reservoir-to-surface analysis.

The technique subdivides the horizontal completion into vertical cross-sections and treats flow independently from other cross-sections. This multiple source concept leads to a pressure gradient from the blind-end (Toe) to the producing-end (Heel) which, if neglected, results in over-predicting deliverability. The reduced drawdown at the Toe results in the production leveling-off as a function of well length and it can be shown that drilling beyond an optimum length would yield no significant additional production. Several IPRs are available. These are solved with the wellbore pressure drop equations to yield the changing production rate along the well length.

Note: To use the **Optimum Horizontal Completion** option, you must include a horizontal well completion in the system model.

How to determine the optimal horizontal completion length

Follow these basic steps to determine the optimal horizontal completion length:

1. [Build the well performance model \(p.51\)](#), include the horizontal completion.
2. Select the **Horizontal completion length** operation.
3. Enter the required data.
4. [Run \(p.195\)](#) the operation.

5. Save the model (p.35).

Horizontal Completion data entry

To access this area, double-click a horizontal completion.

Properties Tab

Use this tab to set up the Reservoir data and IPR model.

Reservoir data

The reservoir boundary conditions must be specified:

Static Pressure:

The static reservoir pressure

Temperature

The reservoir temperature

IPR Model

Select Distributed or Single Point model and select the model type, one of the following:

Distributed PI (Finite Conductivity)

The complete length of the horizontal section is modeled. See [Well bore... \(p.95\)](#)

Single Point PI (Infinite Conductivity)

The input data is used to compute an equivalent straight line PI value and this is then used as single point inflow.

Model type is the type of IPR model to use for calculation. Each option requires a different set of parameters, as described below.

SS Oil (Joshi) and SS Gas (Joshi)

Reservoir Size

Rextn

External boundary radius of the drainage area

Thickness

Reservoir thickness

Well Location

Eccen

Well bore eccentricity. This is the offset of the well from the center of the pay zone.

Reservoir Properties

Kx

Permeability in the x-direction, that is Kh

Ky

Permeability in the y-direction, that is parallel to the well

Kz

Permeability in the z-direction, that is Kv

Well Properties**Length**

Length of the horizontal well (completion). Assuming that the horizontal completion is exactly horizontal. A profile for the horizontal section can be entered using the Well bore... button. The length must not be greater than the reservoir diameter (that is twice the reservoir extension radius Rextn).

Rw

Sandface radius, that is. pipe+annulus+cement

Skin

Mechanical skin factor. Can be entered or computed. To compute the skin, select the **Calculate Skin** check box. This activates the [Options \(p.56\)](#) button. If the value is computed, sensitivity cannot be performed directly on the skin value.

Single Point PI only

The following data are required for single point PI only:

Fluid properties

At reservoir conditions (the first two parameters are alternatives):

OFVF

Oil Formation Volume Factor (oil well)

Gas Z

gas compressibility factor (gas well)

Viscosity

fluid viscosity

Calculate PI

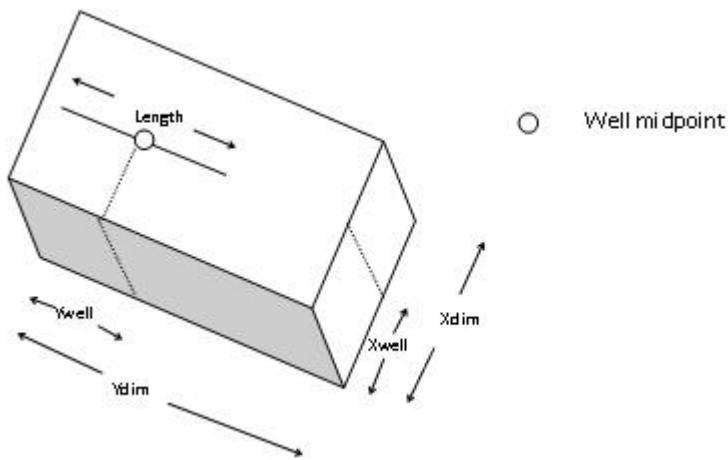
Compute the single point PI values from the data supplied.

Distributed PI only

[Well bore... \(p.95\)](#)

PSS Oil (Babu and Odeh) and PSS gas (Babu and Odeh)

Reservoir Size



Note: this is referenced only in PSS gas.

X dim

drainage width perpendicular to the well

Y dim

drainage width parallel to the well

Thickness

reservoir thickness

Well Location (based on the well midpoint)

Note: this is referenced only in PSS gas.

X well

x coordinates of the horizontal well trajectory.

Y well

y coordinates of the horizontal well trajectory.

Z well

z coordinates of the horizontal well trajectory.

Reservoir Properties

Kx

Permeability in the x-direction, that is Kh

Ky

Permeability in the y-direction, that is parallel to the well

Kz

Permeability in the z-direction, that is Kv

Well Properties**Length**

length of the horizontal well (completion). (This assumes that the horizontal completion is exactly horizontal.) A profile for the horizontal section can be entered using the

[Well bore... \(p.95\)](#)

button.

Rw

Sandface radius. That is, pipe+annulus+cement

Skin

Mechanical skin factor. Can be entered or computed. To compute the skin, select the **Calculate Skin** check box . This activates the [Options \(p.56\)](#). If the value is computed, sensitivity cannot be performed directly on the skin value.

Single Point PI only

The following data are required when you specify **Single Point PI** as the IPR model:

Fluid properties

enter these for reservoir conditions

OFVF

Oil Formation Volume Factor (oil well)

Gas Z

gas compressibility factor (gas well)

Viscosity

fluid viscosity

Calculate PI

Click this button to compute the single point PI values from the data supplied.

Distributed PI only

The following data are required when you specify **Distributed PI** as the IPR model and select **Distributed PI** as the model type:

Distributed PI

Enter a straight line PI value for liquid or gas.

Well bore... (p.95)

Click the button to set the properties.

Horizontal Completion Well Properties Options

This dialog opens when you select the **Calculate skin** check box on the **Properties** tab and click the **Options** button.

Damaged Zone

Diameter

Diameter of the damaged zone around the well bore. The default is the well bore radius, that is the damaged zone does not exist

Permeability

Permeability of the damaged zone around the well bore. The default is the formation permeability.

Gravel Pack

Permeability

Permeability of the gravel pack. The default is estimated according to the sieve size.

Tunnel

Length of the tunnel. This is usually the sum of the thickness of cement, casing and annulus. The default =0

Compacted zone

Diameter

Diameter of the compacted zone (or crushed zone) around the perforation. The default is the diameter of the perforation (that is, the compacted zone does not exist).

Permeability

Permeability of the compacted zone (or crushed zone) around the perforation. The default is the permeability of the damaged zone.

Perforation

Diameter

Diameter of the perforation into the formation. The default = 0.5 inches, 12.7 mm

Length

Length of the perforation into the formation. The default = infinity (which results in a zero skin due to perforation)

Shot Density

Shot density. The default = 4 shots/ft ,13.12 shots/m

Calculate Skin

Calculate

Click the **Calculate** button to compute the skin from the data supplied. This overwrites any value already set.

Vertical Well Completion

The vertical completion component models flow from the reservoir to the bottom hole using an inflow performance relationship. [More details \(p.398\)](#).

The following data must be entered:

Reservoir Data

- Static Pressure
 - Temperature
-

Note: The pressure may be a calculated variable, provided that the other two boundary conditions, flowrate and outlet pressure, are entered.

IPR Model

Inflow Performance Relationship (p.398)

Relationship between completion drawdown and flowrate.

Flow Control Valve (FCV)

The default is a completion without FCV. To use an FCV, select the check box and click **FCV Properties** to configure it.

Local fluid properties

Local Fluid properties that pertain to this completion.

Note: Vertical completions can be "stacked" to form a multiple completion well and (optional) tubing can be placed between the completions. Thus each completion can have its own fluid properties.

PI Data

Data required

The following parameters are required for a [PI \(p.399\)](#) IPR.

PI coefficient

A function of bottom hole flowing pressure (Pwf), Static reservoir pressure (Pws) and Flowrate (Q).

Vogel Below bubble point correction

Use this in cases of undersaturated reservoirs where wellbore pressure may be above or below the bubble point. **Note:** The "Well PI with Vogel correction below the bubble point" completion model is only intended for use when the reservoir pressure is above the bubble point. If the static reservoir pressure is below the bubble point, then you should use a different completion model, for example [Vogel \(p.400\)](#) or [Fetkovich \(p.401\)](#) which are intended for saturated fluids.

Calculate/Graph button

The required parameters can be computed from [multi-rate test \(p.64\)](#) data.

Vogel's Equation Data

The [Vogel equation \(p.400\)](#) was developed to model saturated oil wells.

Data required

If you select Vogel's Equation as the IPR Model on the **Vertical Completion Properties** tab, the following parameters are displayed:

Abs. Open Flow Potential

The maximum liquid flowrate that the well could deliver if the bottom hole pressure was 0.

In the **Absolute Open Flow Potential** section, enter the following data that will be used to calculate the AOFP:

PI Coefficient

The value is usually around 0.8 (the default).

Q

The actual flowrate of the well from a well test.

Pwf

Flowing bottom hole pressure

Pws

Static Reservoir pressure

Calculate AOFP

use the above data to compute the AOFP. This overwrites any values already entered for the AOFP.

Fetkovich's Equation Data

[Fetkovich's equation \(p.401\)](#) is a development of the Vogel equation to take account of high velocity effects.

Data required

If you select Fetkovich's Equation as the IPR Model on the **Vertical Completion Properties** tab, the following parameters are displayed:

Open Flow Potential

The well's maximum flowrate.

n exponent:

Calculate/Graph button

The required parameters can be computed from [multi-rate test \(p.64\)](#) data.

Jones' Equation Data

Data required

If you select [Jones' equation \(p.401\)](#) as the IPR Model on the **Vertical Completion Properties** tab, the following parameters are displayed:

Fluid type; Liquid or Gas:

Changes the units of the A and B coefficients

A (turb)

turbulent coefficient, must be => 0

B (lam)

laminar coefficient, must be => 0

Calculate/Graph... button

The required parameters can be computed from [multi-rate test \(p.64\)](#) data.

Jones 4-point test

To run a Jones 4-point test, do the following:

1. Select **Jones IPR**
2. Click **Calculate » Graph....**
3. Select [multipoint \(p.64\)](#)
4. Enter the static reservoir pressure.
5. Enter the reservoir temperature.
6. Enter up to 4 test rates and associated test pressures.
7. Press the **Plot IPR** or **Plot fit** button and the coefficients A and B will be computed.
8. Select OK

Backpressure Data

Rawlins and Schellhardt developed the Backpressure equation in 1935.

Data required

If you select [Backpressure equation \(p.402\)](#) as the IPR Model on the **Vertical Completion Properties** tab, the following parameters are displayed:

C

Constant (intercept at log flow rate = 1.0)

n

Inverse Slope. For pure laminar flow n=1 and 0.5 for completely turbulent flow. n is limited to 0.5<n<1

Calculate/Graph button

Click this to compute the required parameters C and n from [multi-rate test \(p.64\)](#) data. At least three data sets must be supplied.

4-point test

To run a Back Pressure 4-point test, do the following:

1. On the **Vertical Completion Properties** tab, select **Backpressure IPR**.
2. Select **Calculate » Graph..**
3. Select [multipoint \(p.64\)](#).
4. Enter the static reservoir pressure.

5. Enter the reservoir temperature.
6. Enter up to 4 test rates and associated test pressures.
7. Click **Plot IPR** or **Plot fit**. The coefficients C and n will be computed.
8. Click **OK**.

Pseudo Steady State Equation / Darcy Data

The [Pseudo steady-state equation \(p.403\)](#) assumes that the fluid is single phase, that laminar flow exists, and that the fluid is (essentially) incompressible. The equation is based on single phase flow, therefore you are prompted to select the basis for the equation; either Liquid or Gas.

A Vogel correction is available for liquid flow below the bubble point. For gas systems, you can choose to use gas pseudo pressure or a pressure squared approximate (only reliable for low pressure systems).

Data required

If you select Pseudo Steady-State as the IPR Model on the **Vertical Completion Properties** tab, the following parameters are displayed:

Basis of IPR calculation

Specify whether the calculations are to be performed on a Liquid or Gas basis. Your selection changes the following parameter choices.

Use Vogel below bubble point

Apply the equation for calculating the productivity above the bubble point and the Vogel relationship to calculate the IPR curve below the bubble point.

Note: This check box is available only if you selected Liquid as the IPR Basis.

Use pseudo pressure method

Note: This check box is available only if you selected Gas as the IPR Basis.

Reservoir Thickness

Average formation thickness.

Wellbore Diameter

Diameter of the wellbore (drilled hole) outside of the casing and cement. The default is 6 inches.

Permeability

This is the average formation permeability. [Typical values \(p.574\)](#).

Reservoir Size/Shape

Take into account the effect of the shape of the reservoir (and the position of the well relative to boundaries), by way of a shape factor. The default is a circular reservoir shape (shape factor 31.62).

Drainage Radius

Radius of external boundary of drainage area. Default 2,000 feet. [Typical values \(p.574\)](#).

Shape factor

Identify the physical location of a well in relationship to the reservoir boundaries. Default 31.62 circular reservoir, see figure below.

Reservoir Area

Area of the reservoir. [Typical values \(p.574\)](#).

Shape Factors for Pseudo Steady State Inflow

Geometry	C_A	S_A	Geometry	C_A	S_A
	31.62	0.000		10.84	0.535
	30.88	0.012		4.514	0.973
	31.60	0.000		2.077	1.362
	27.6	0.068		2.690	1.232
	27.1	0.077		0.232	2.458
	21.9	0.184		0.115	2.806
	21.84	0.185		3.335	1.125
	5.379	0.886		3.157	1.152
	2.361	1.298		0.581	1.998
	12.98	0.445		0.111	2.827
	4.513	0.973		0.098	2.888

Figure 1.1. Shape Factors

See [Earlougher \(1977\) \(p.586\)](#) for reference.

Skin

Skin value has two components, a Mechanical (constant) term and a rate dependent term. (For example, if the rate is 20 mmscf/d, the constant skin is 3, and the rate dependent skin is 0.1/

mmscf/d, the total skin would be 5). Both mechanical and rate dependent skin terms can be entered or calculated (from completion description).

Mechanical Skin

Enter Skin

Dimensionless constant skin factor

Calculate skin

Model the completion in detail and thus compute the skin factor using [completion options \(p.68\)](#). If the skin is computed, sensitivity cannot be performed directly on the skin value, but sensitivities can be performed on any completion description parameter (for example shots per foot, perforation depth, and so on).

Rate Dependent Skin

Enter Skin

Dimensionless Rate dependent skin factor

Calculate

Calculate the rate dependent skin terms for all the components specified in the [completion options \(p.68\)](#) description.

Forchheimer Equation Data

Data required

If you select [Forchheimer's equation \(p.402\)](#) as the IPR Model on the **Vertical Completion Properties** tab, the following parameters are displayed:

F (turb)

Turbulent coefficient, this must be => 0

A (lam)

Laminar coefficient, this must be => 0

Calculate/Graph button

The required parameters can be computed from [multi-rate test \(p.64\)](#) data.

Multi-rate

In addition to the standard IPR equations, test data can be utilized so that the inflow can be matched to actual measured data. A minimum of three data points is required. Two types of multi-rate test are available:

Multipoint (default)

A 'flow-after-flow' test sequence. Static pressure is taken as a constant throughout the test period. The flowrate (Q) and corresponding flowing bottom hole pressure (Pwf) are required. These are entered into the spreadsheet.

Isochronal

This type of test is normally performed in reservoirs with low permeability where the time taken to reach stabilized flow conditions is unacceptably long (such as low permeability

sands). Isochronal testing is performed by periods of flowing followed by shutting-in of a well (normally with increasing rate). The wellbore flowing pressure is recorded during each flow period at a specific time (for example if the time is 4 hours, then the test is referred to as a 4-hour isochronal test). Due to the long stabilization time normally associated with the isochronal test, reservoir conditions need not return to the original static pressure. Hence a different static reservoir pressure is recorded. The flowrate (Q), flowing bottom hole pressure (P_{wf}) and static reservoir pressure (P_{ws}) are required. These are entered into the spreadsheet.

Once the test data has been entered it can be plotted using either;

Plot IPR

plots the actual IPR to be used, with the test data superimposed.

Plot Fit

plots the log log fit, with the test data superimposed.

The IPR constants (for example PI , A and B , C and n , and so on) will be computed and displayed in the **Fitted constants** section.

Select the **Chart > Print** option to print or export the plot.

Hydraulic Fracture Reservoir Properties

This topic describes the reservoir data to enter when you use the hydraulic fracture equation for vertical completions.

Note: The Fractured Well IPR type uses a digitized, constant rate, finite-conductivity, closed square, fractured well type-curve to calculate the effect of a vertically drilled well that has been hydraulically fractured. This is the same method used in the Schlumberger FracCADE software.

- The well is assumed to be in the center of a square reservoir with an aspect ratio of 1:1.
- The type curves used in the calculation are taken from *Reservoir Stimulation* 2nd Edition by Economides and Nolte, Chapter 8 by Hai-Zui Meng and SPE paper 16435 and are best suited for tight gas wells. Type curves are generated using single-phase, two-dimensional finite difference simulators for ranges of system properties (permeability, porosity, fluid viscosity, total system compressibility) and the characteristic length of the system, fracture half-length. These are then used to compute Dimensionless time (valid range: 10e-5 - 10e3), Dimensionless wellbore pressure, and Dimensionless fracture conductivity (valid range 0.1 - 500).

Property	Description
Reservoir pressure	Static reservoir pressure
Reservoir temperature	Reservoir temperature
IPR basis	Basis for IPR calculation (liquid or gas)
Use Vogel below bubble point	(Available for liquid-based IPR only) Uses a type-curve equation for calculating the productivity above the bubble point, and the Vogel

Property	Description
	relationship to calculate the IPR curve below the bubble point. If the watercut exceeds 60%, using Vogel's equation is not recommended.
Reservoir thickness	Average formation thickness.
Reservoir permeability	Average formation permeability. For a gas well, this is gas permeability. For an oil well, this is total liquid permeability.
Reservoir radius	Radius of external boundary of drainage area. The default value is 2,000 feet.
Borehole diameter	Diameter of the wellbore (drilled hole) outside of the casing and cement. The default value is 6 inches.
Fracture half length	The length of the fracture extending out in one direction from the wellbore, which is half of the total fracture length.
Fracture permeability	The effective permeability to the primary fluid of the fracture proppant under reservoir conditions.
Fracture width	Average width of the fractures in a hydraulically fractured reservoir.
Use transient model	Select the Transient model check box to model a well when the well has not reached pseudo-steady-state conditions. When selected, the following properties appear: Time Time well has been producing Porosity Pore volume/bulk volume Compressibility Total compressibility of the reservoir

Transient IPR Data

Data required

If you select [Transient \(p.408\)](#) as the IPR Model on the **Vertical Completion Properties** tab, the following parameters are displayed:

Basis of IPR calculation

Specify whether the calculation are to be performed on a Liquid or Gas basis. Your selection changes the following parameter choices.

Use Vogel below bubble point

Apply a type-curve equation for calculating the productivity above the bubble point and the Vogel relationship to calculate the IPR curve below the bubble point. The Vogel correction is not recommended if the water cut exceeds 60%.

Note: This check box is available only if you selected Liquid as the IPR Basis.

Use pseudo pressure method

Using $(\text{Pres}^2 - \text{Pbhp}^2)$ for the delta P term in the pseudo steady state equation for Gas is correct only for low pressures ($< \sim 2,000$ psia).

A more accurate way is to use $(M\text{Pres} - M\text{Pbhp})$ instead where MP is the pseudo pressure (which itself is a function of pressure, Z factor and viscosity). This is more valid over the entire range of pressures.

Note: This check box is available only if you selected Gas as the IPR Basis.

Permeability and other Parameters

To plot the IPR, click **Plot IPR**. Enter the following parameters:

Reservoir Permeability

Average formation permeability. The default is infinity. [Typical values \(p.571\)](#).

Reservoir Thickness

Average formation thickness. The default is infinity.

Wellbore Diameter

The default is 6 inches.

Drainage Radius

Radius of external boundary of drainage area. The default is 2,000 feet. [Typical values \(p.574\)](#)

Time

Duration of the drawdown period in hours.

Porosity

Average porosity of the reservoir rock.

Total Compressibility

Saturation weighted compressibility of the entire reservoir-liquid system
 $(ct = coSo + cwSw + cf)$

Skin parameters

Skin value has two components, a Mechanical (constant) term and a rate dependent term (for example if the rate is 20 mmSCF/d and the constant skin is 3, and the rate dependent skin is 0.1/mmSCF/d, then the total skin would be 5). Both mechanical and rate dependent skin terms can be entered or calculated (from completion description).

Mechanical Skin.

Choose from the following:

Enter Skin

Dimensionless constant skin factor.

Calculate

Model the completion in detail and thus compute the skin factor using [completion options \(p.68\)](#). If the skin is computed then sensitivity cannot be performed directly on the skin value, but sensitivities can be performed on any completion description parameter (for example shots per foot, perforation depth and so on).

Rate Dependent Skin.

Choose from the following:

Enter Skin

Dimensionless Rate dependent skin factor.

Calculate

Calculate the rate dependent skin terms for all the components specified in the [completion options \(p.68\)](#) description.

Completion Options

Both the [Pseudo Steady State \(p.61\)](#) and the [Transient \(p.408\)](#) IPRs use a skin factor to determine the flow into the well. PIPESIM allows you to enter a skin factor, or to calculate it using **Completion Options**. To use **Completion Options**, first select one or both of the **Calculate** radio buttons in the **Mechanical Skin** and **Rate Dependent Skin** boxes.

Skin factor

The skin factor is calculated by summing contributions from five different components:

Damaged Zone Skin

models the effect of reduced (or improved) permeability in a zone around the well. The diameter and permeability of the damaged zone must be supplied, otherwise they will default to the well bore diameter and the formation permeability, and the damaged zone skin will be zero.

Partial Penetration / Deviation Skin

models geometric effects, such as partial penetration of the reservoir layer and deviation of the well from vertical. If the open interval is equal to the reservoir thickness and the well is vertical (0 degrees deviation) then the partial penetration / deviation skin will be zero.

Gravel Pack Skin

models the effect of a gravel pack in the well bore, between a screen and the reservoir. Also, the pressure drop contribution from the gravel pack is calculated and reported in the output and system plot files.

Perforation Skin

models the effects of perforations. A compacted zone around the perforations is included.

Frac Pack Skin

models a well that has been fractured.

The Skin factor can be either negative (enhanced inflow) or positive (reduced inflow).

Completion models

In **Completion Options** you can select one of five completion models. Skin components appropriate to each completion model are calculated, as detailed in [Vertical Well mechanical skin factor calculation \(p.415\)](#). You can choose to turn off any component of the skin.

Completion model		Damaged Zone Skin	Partial Penetration / Deviation Skin	Gravel Pack Skin	Perforation Skin	Frac Pack Skin
Open Hole	Assumes the well is not lined or cemented	Yes	Yes			
Open Hole Gravel Pack	Assumes the well bore is Open Hole, with the addition of a gravel pack	Yes	Yes	Yes		
Perforated	Assumes the well is lined or cemented, and perforated	Yes	Yes		Yes	
Gravel Packed and Perforated	Assumes the well is lined or cemented, and perforated, with the addition of a gravel pack	Yes	Yes	Yes	Yes	
Frac Pack	Assumes the well is lined or cemented, perforated and has been fractured.		Yes	Yes	Yes	Yes

The data entered in the **Completion Options** dialog is passed to the engines to calculate the IPR during simulation. An approximate value of the skin used during the simulation can be determined by clicking on the **Calculate Skin** button. This value may not be exact, since some of the skin calculations may depend on fluid properties which are only calculated during the simulation.

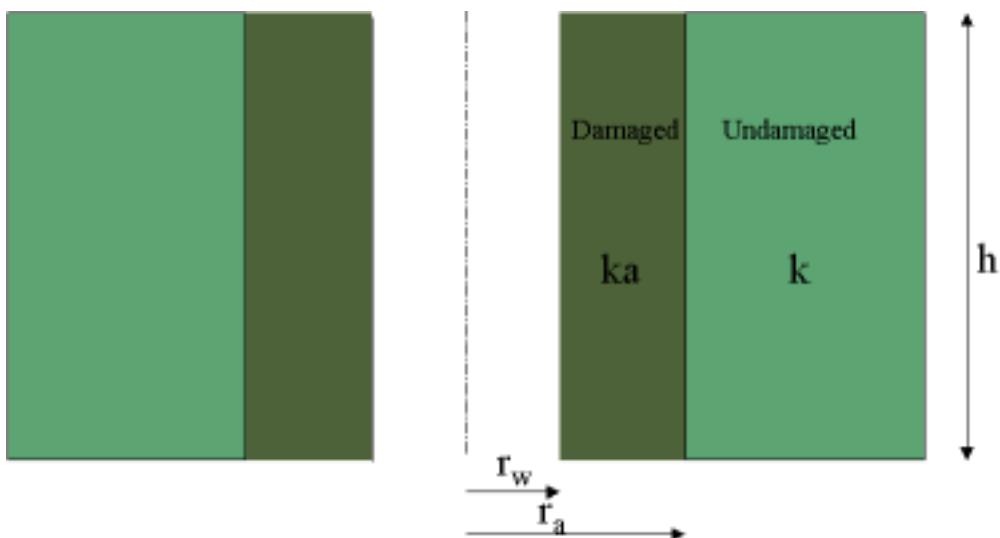


Figure 1.2. Open Hole Completion

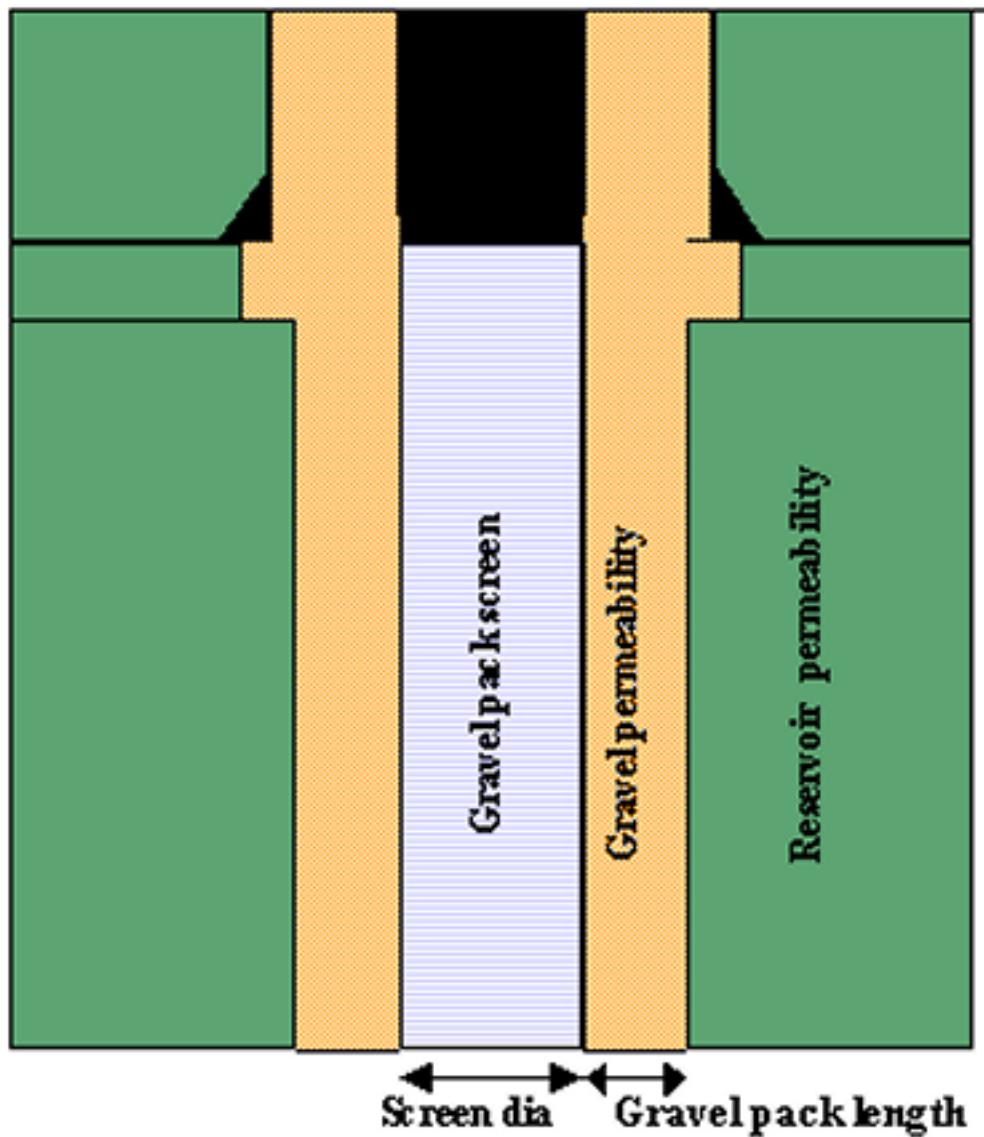
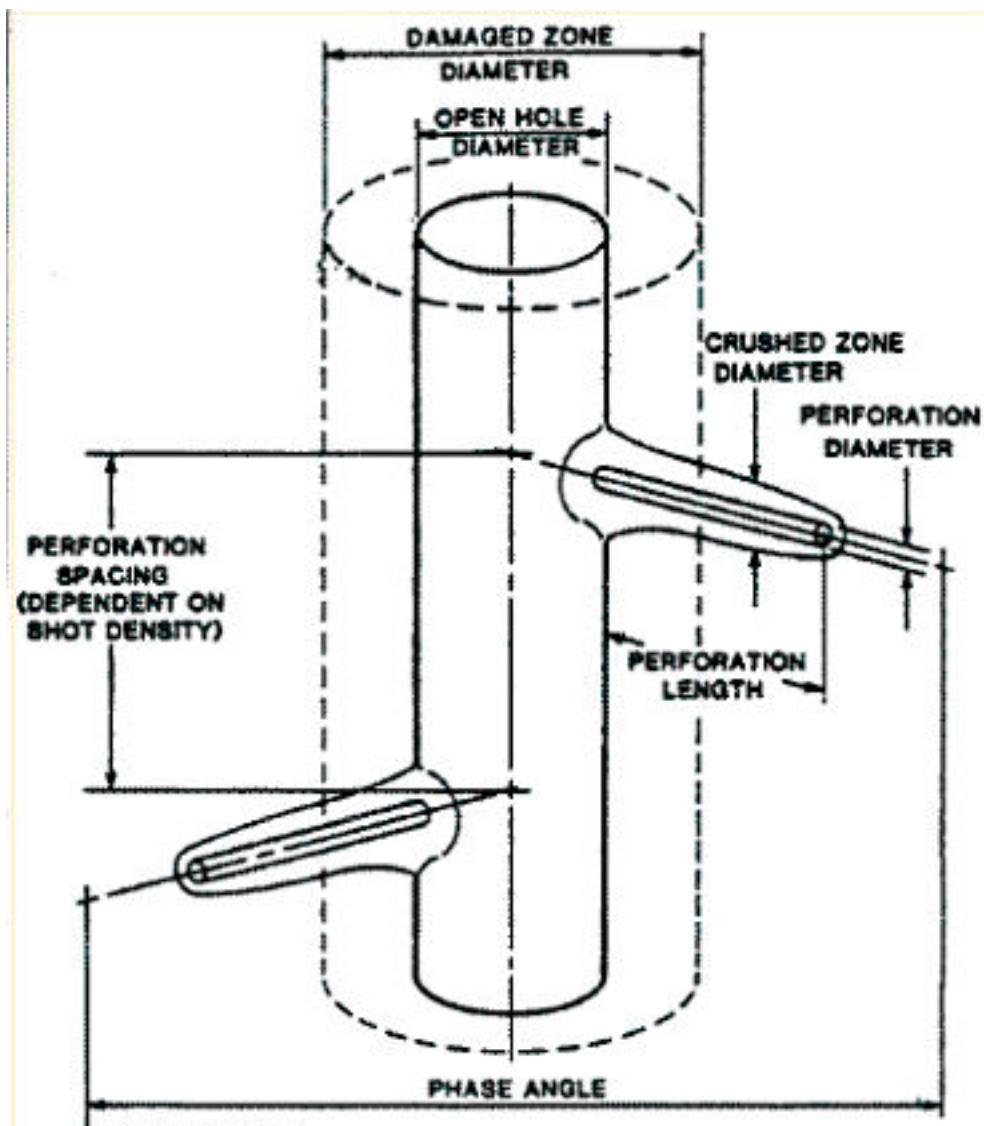


Figure 1.3. Open Hole Gravel Pack Completion



Ref: SPE18247

Figure 1.4. Perforated Completion

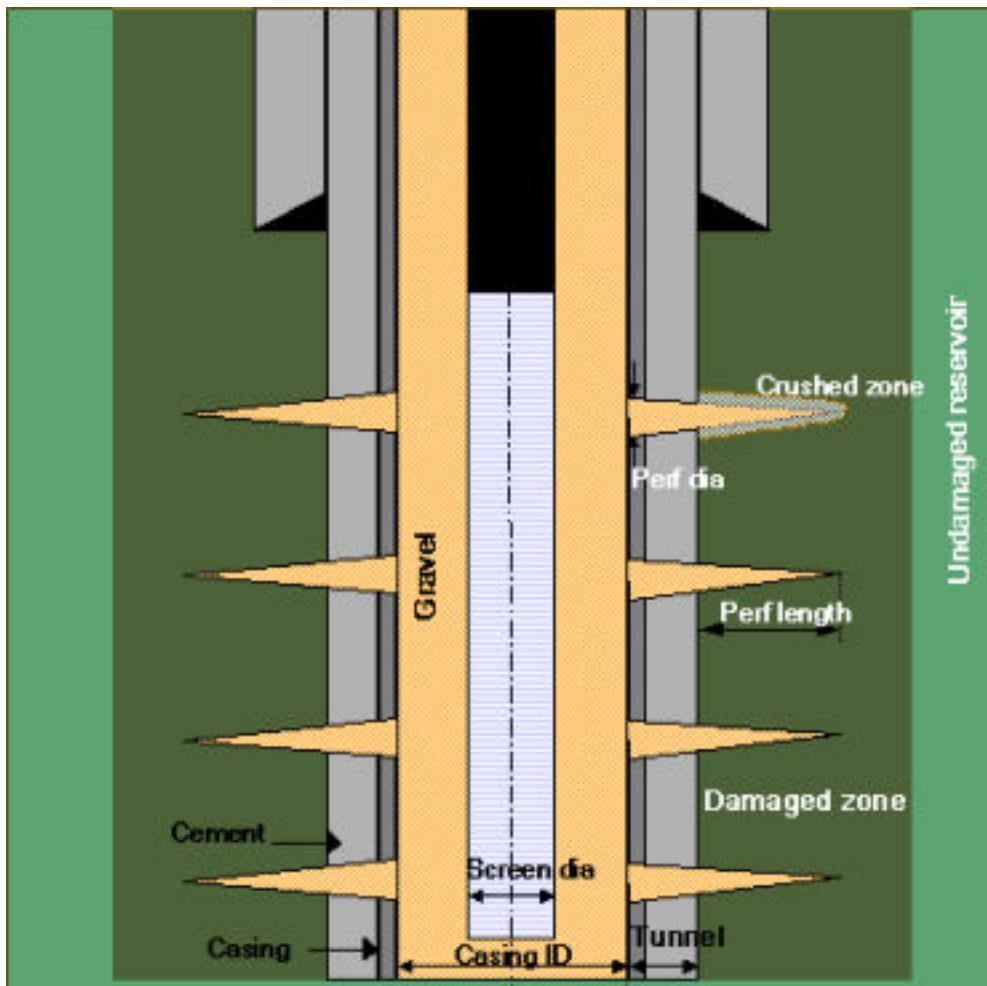


Figure 1.5. Gravel Packed and Perforated Completion

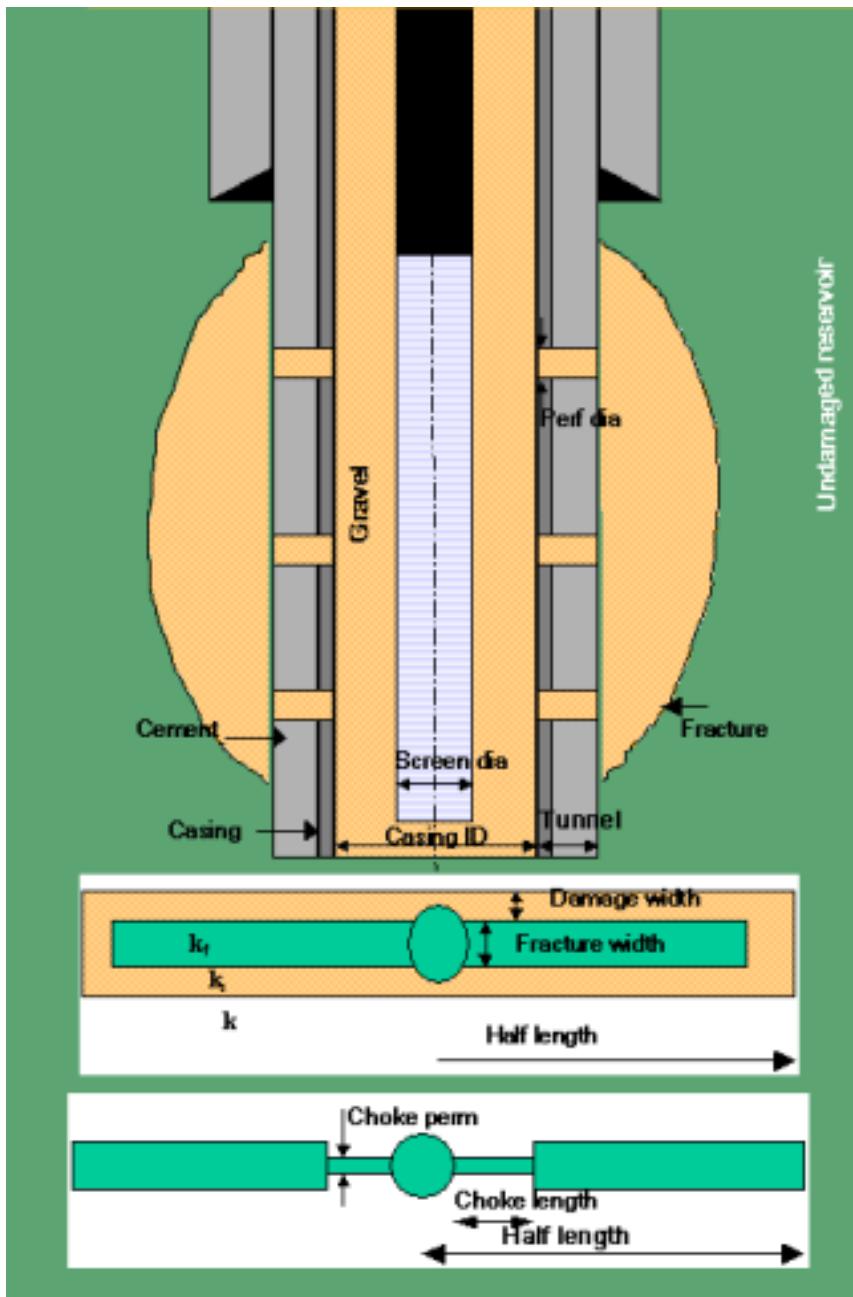


Figure 1.6. Frac Pack Completion

Multilayer Reservoir IPR

A multilayer reservoir model is easy to construct. If the fluids from multiple layers are NOT commingled, the individual layers can be treated as two separate wells and modeled in the Network model connected at the wellhead. Otherwise the **Well Performance** module can be used to model the commingled flow.

Any number of reservoir intervals (layers) can be added to the model. The layers can be either stacked on top of each other (that is with common flowing bottom hole pressure) or separated by a length of conduit (such as tubing or casing) if required.

Note: As of release 2008.1, multi-layer injection well models can be simulated in PIPESIM.

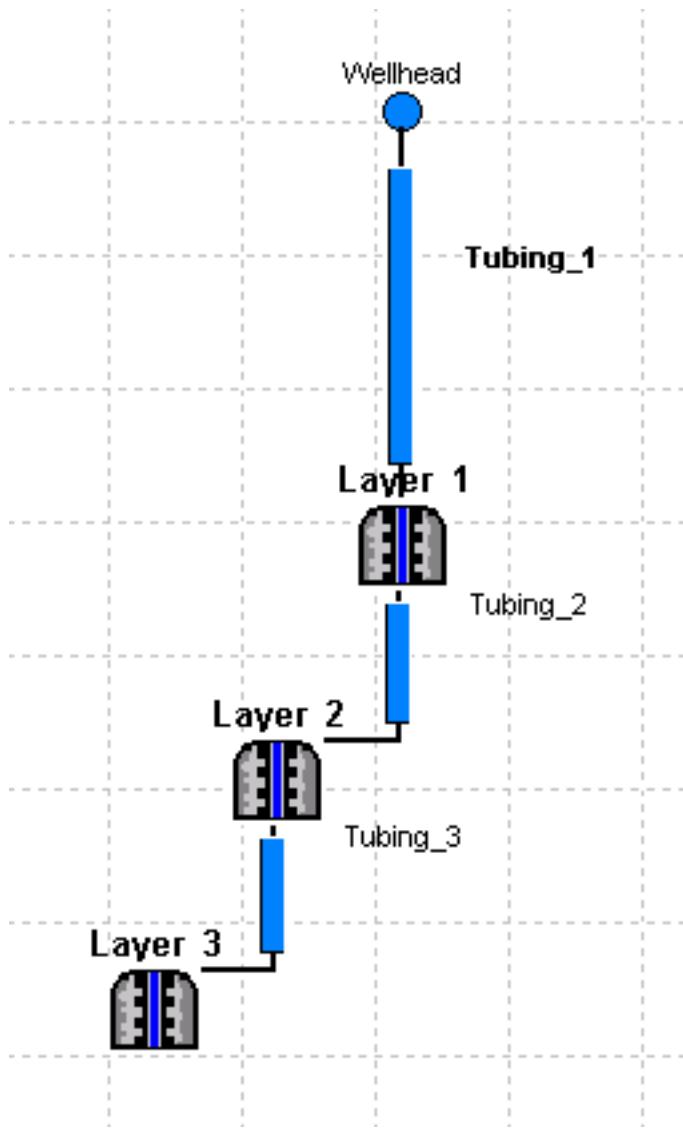
Layers

A completion icon represents each layer:



Therefore, if required, each layer can have a different Inflow Performance Relationship (IPR) and also a different fluid model.

Tubing strings or connectors may be used to separate layers. This ensures the layers are positioned at different elevations. If a tubing string is used, elevational and frictional pressure drops are calculated between the layers, and each layer has a different production pressure (PWF).



Care should be taken when defining the dimensions of the tubing between the layers. Each tubing object will have to be defined independently. It is only important to ensure that the relative distance from top to bottom of each icon (that is between each completion) is correct. Therefore, for example, each tubing icon can be assumed to start from a depth reference of zero. Alternatively the TVDs and MDs at the tubing/layer junctions can be made to match, so each deeper tubing object would be made to start at the deepest point of the adjoining shallower tubing.

Make sure also that the correct ambient temperature(s) for each tubing is specified.

By default PIPESIM allows back flow (that is reverse flow) into (or, in an injection well, out of) any layer. To block back flow in any layer, the appropriate keyword can be set ([LAYER INJECT = NO \(p.666\)](#)) by placing an [EKT \(p.94\)](#) upstream of the completion in question.

Tubing Tool

Placing a tubing object in the model allows modeling of vertical or deviated flow (production or injection) in a well bore. [Heat transfer \(p.100\)](#) can be modeled by either entering or computing an

overall heat transfer coefficient (U value) or using the [Ramey Model \(p.81\)](#). The tubing object is a [connection \(p.34\)](#) object, so must be placed between two [node \(p.34\)](#) objects.

Normally only one tubing object is required for a well model. The exception is when multiple completions are modeled.

The tubing can be defined as either of the following types of model:

- [simple \(p.77\)](#) model — simplified interface
- [detailed \(p.79\)](#) model

Data entered under the simple model can be converted to a detailed model using the **Convert to detailed model** button at any time.

Use the detailed profile model if any of the following situations apply:

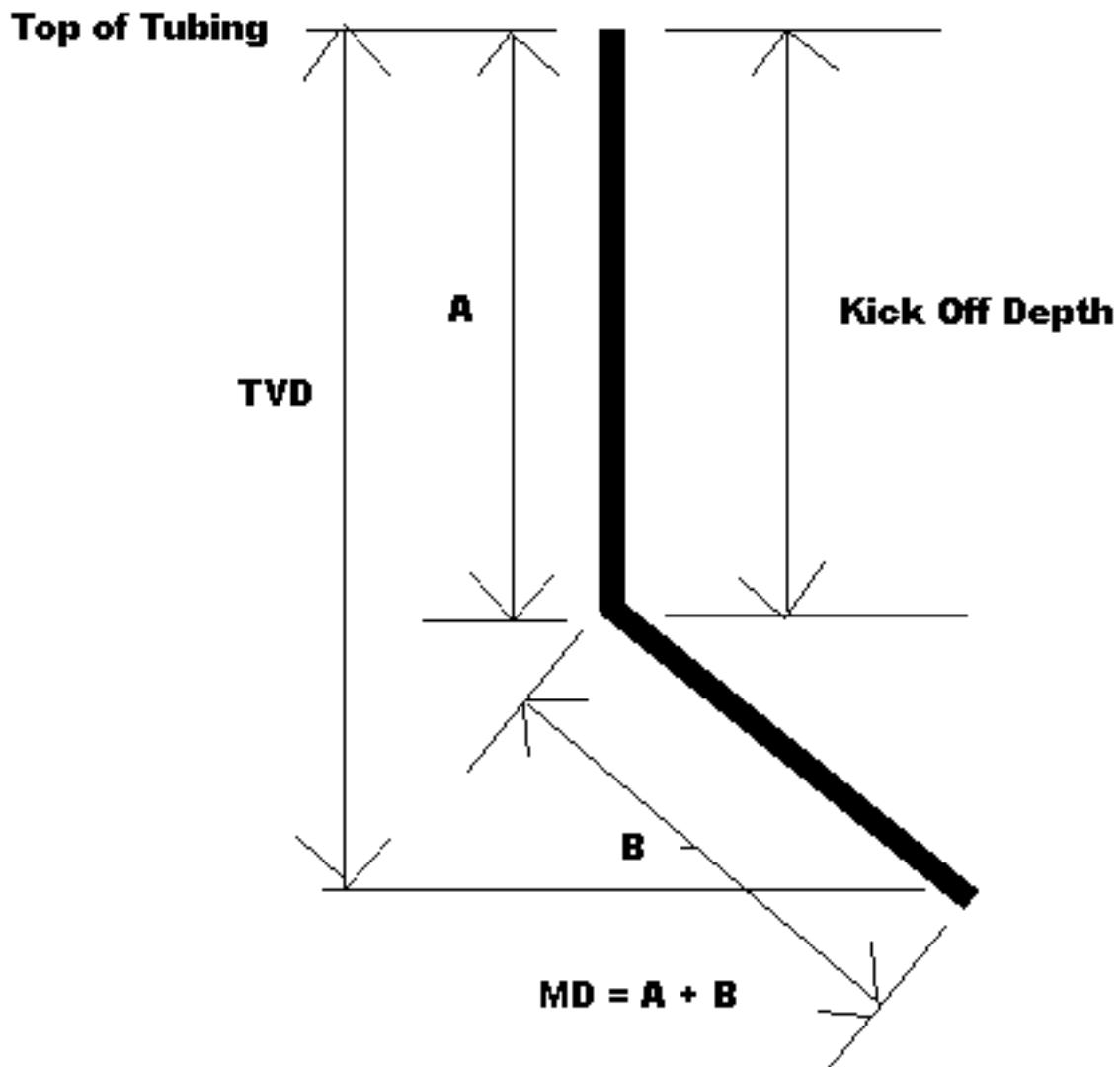
- The well is highly deviated
- It has more than four changes in tubing size
- A geothermal gradient is known
- The overall heat transfer coefficient is not the same as the default (0.2 Btu/hr/ft²), or needs to vary through the depth of the tubing
- More than one piece of downhole equipment needs to be modeled. That is, there are multiple gas lift injection points.

Note: If different data is entered in both the simple and detailed models, PIPESIM uses the data in the **Preferred Tubing model** that was selected when the **Tubing Tool** dialog was closed.

See also: [Coiled Tubing/Velocity String \(p.87\)](#)

Simple Profile Model

The [tubing \(p.76\)](#) object, [once added to the model \(p.34\)](#), requires the following information:



Perforations

This data determines the extent of the tubing modeled. That is, the MD/TVD is taken as the fluid inflow point. Enter the following parameters:

MD (Measured depth)

total length of the tubing to the perforations. This will always be equal to, or greater than, the TVD

TVD

true vertical depth

Reservoir Temperature

fluid temperature

Tubing sections

Note: Up to four tubing items can be defined. If more are required, use the [detailed model \(p.79\)](#) profile.

Enter the following parameters for each section:

From MD

start MD for this tubing's section. This is computed automatically from the initial MD or from the last set tubing depth, that is the "to MD column".

To MD

end MD for this tubing's section. This MUST always be entered.

ID

inner diameter of the tubing section.

Convert to Detailed model

The data entered into the simple model can be exported to the [detailed \(p.79\)](#) model using the **Convert to detailed model** button.

Note: Any data already entered under the detailed model is overwritten.

Summary table

[Summary table \(p.82\)](#)

Detailed profile model

To configure a [tubing \(p.76\)](#) object, once added to the model (p.34), double-click the tubing, select **Detailed Model**, and use the following tabs.

Deviation Survey Tab

Set two of the MD, TVD and angle for at least two data points. Compute the third using the **Calculate** button. (Default = compute angle). The wellbore profile can be copied from any other spreadsheet application. If the well profile is straight, the deviation survey can be omitted.

Geothermal Survey Tab

Enter ambient temperatures at various depths (the depths can be either based on MD or TVD). If the **Input U value** radio button is selected, also enter the overall heat transfer coefficient values (U-values) at the various depths. The minimum data you must enter is at the well head and bottom hole. If you want the U-values to be calculated, select the **Calculate U Value** radio button; the **U Value** column is greyed-out so that you cannot enter any values. Click **Heat Transfer Properties** to enter the [Tubing Heat Transfer Properties \(p.81\)](#) for the heat transfer calculation.

Tubing Configurations Tab

Up to 200 changes of tubing configuration can be modeled. The data entered here determine the extent of the tubing modeled. That is, the bottom of the last tubing section is taken as the fluid inflow point.

The parameters are as follows:

Bottom MD

Measured Depth of the bottom of the tubing.

ID

tubing inside diameter, See also [tubing tables \(p.562\)](#)

Wall Thickness

tubing thickness, default 0.5 inches 12.7 mm. See also [tubing tables \(p.562\)](#)

Roughness

absolute pipe roughness, default 0.001 inches, 0.0254 mm. See also [Roughness tables \(p.572\)](#)

Casing ID

casing ID is only required if the flow type is set to "Annular" or "Tubing+Annulus" flow or the [Alhanati \(p.232\)](#) instability check is to be made. See also [Casing tables \(p.562\)](#).

Flow Type

flow path up the tubing. Select one of the following options:

- Tubing (default)
- Annulus or
- Tubing+Annulus.

The label is optional, but allows key points in the profile to be highlighted in the output report.

Downhole Equipment Tab

Select the type, position (in terms of its MD), properties and optional label of any downhole equipment. Click **Properties** to configure each item of equipment, as follows:

- [Gas Lift Injection Properties \(p.84\)](#) button
- [ESP Properties \(p.240\)](#) button
- [PCP Properties \(p.192\)](#) button
- [Rod Pump Properties \(p.242\)](#)
- [Choke Properties \(p.443\)](#), button
- [SSSV \(p.443\)](#). Enter the bean ID using the **Properties** button.
- [Separator Properties \(p.116\)](#) button
- [Keyword Tool Properties \(p.94\)](#) button
- [Injector Properties \(p.101\)](#) button

Gas lift valves

Click [G/L Valve system \(p.84\)](#) to enter Gas lift valves.

Note: Using Gas lift injection points is different from adding Gas Lift valves. The valve option takes into account the physics of the gas lift system, whereas the injection point assumes that the amount of gas supplied can always be injected. See [Gas Lift system \(p.214\)](#) for more details.

Wellbore Heat Transfer

Various input data can be entered in the **Tubing Heat Transfer Properties** dialog to compute the [overall heat transfer coefficient \(p.486\)](#).

To open this dialog, open a detailed tubing model, select the **Geothermal Survey** tab and click **Heat Transfer Properties**.

Ground Properties

The parameters are as follows:

Thermal Conductivity

The ground/rock thermal conductivity (default = 1.5 BTU/hr/ft/F).

Specific Heat Capacity

The ground/rock specific heat capacity (default = 0.2 BTU/lb/F).

Density

The ground/rock density (default = 140 lb/ft³)

Wellbore Properties

The parameters are as follows:

Production/Injection Time

The time the well has been producing/injecting (default = 168 hr).

Tubing Thermal Conductivity

The thermal conductivity of the tubing material (default = 35 BTU/hr/ft/F).

Completion Fluid Thermal Conductivity

The thermal conductivity of the completion fluid in the annular space between the tubing and the casing (default = 3.2 BTU/hr/ft/F).

Casing Thermal Conductivity

The thermal conductivity of the casing material (default = 35 BTU/hr/ft/F).

Casing Thickness

The thickness of casing.

Cement Thermal Conductivity

The thermal conductivity of the cement between the casing and the rock/ground (default = 2 BTU/hr/ft/F).

Cement Thickness

The thickness of cement.

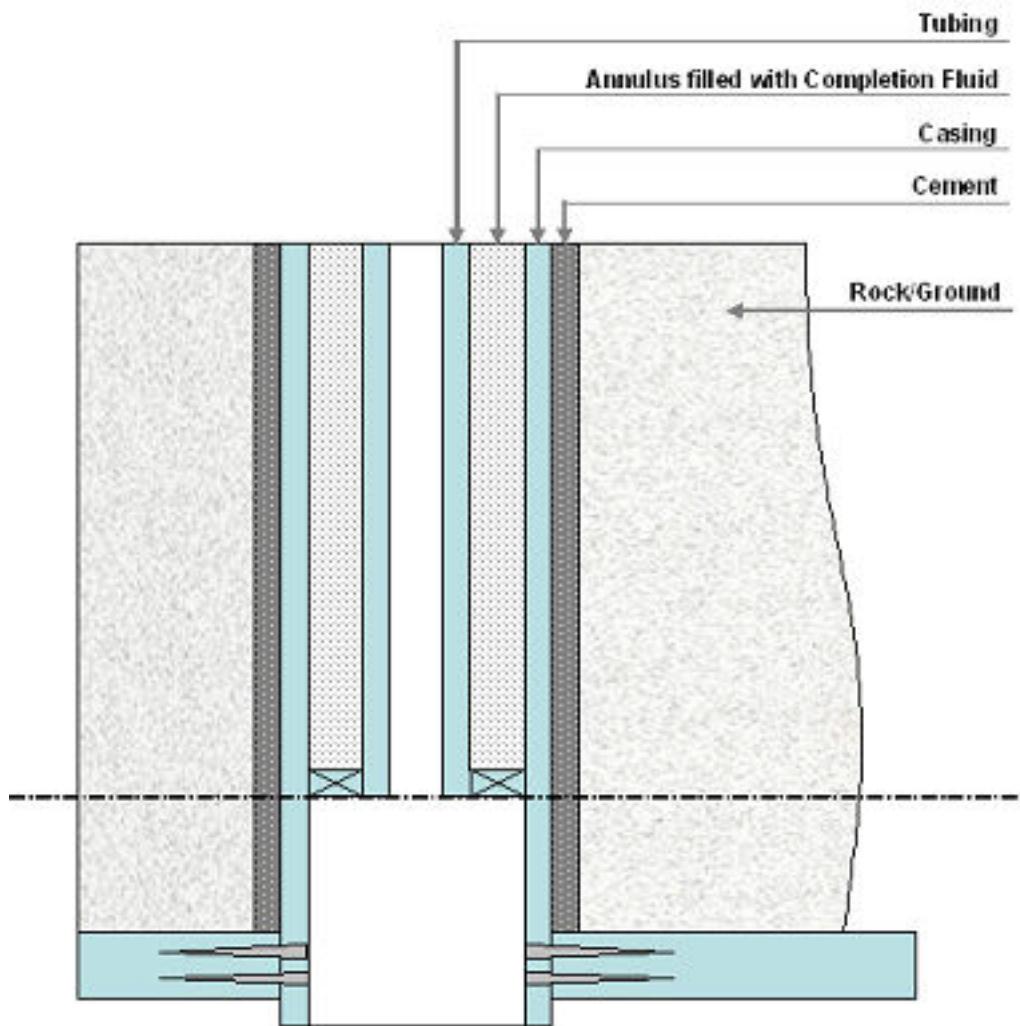


Figure 1.7. Typical Wellbore

Summary tubing table

To view this table, double-click a tubing object and click **Summary Table** in the dialog. The dialog has the following three sections:

Configuration summary

There is just one parameter, as follows:

Spreadsheet

shows a summary of the nodes that will be used to model the well and any equipment or ID changes in the tubing. The data cannot be changed here.

Options

The parameters are as follows:

Distance between nodes

to speed up the calculations, increase the distance. To make the calculation more accurate, decrease it. (Default =1,000 ft).

Refresh

update the spreadsheet after the node distance has been changed.

Schematic

This shows a schematic of the well with any equipment placement noted.

Gas Lift Properties

To open this dialog, double-click a tubing object. Select one of the following tubing models:

- [simple tubing model \(p.77\)](#): Set the Artificial Lift method to Gas Lift, then click **Properties** under **Artificial Lift**.
- [detailed tubing model \(p.79\)](#) On the **Downhole Equipment** tab, add Gas Lift Injection.

Properties of the lift gas

The flow rate of the injected lift gas can be specified directly, or calculated to give specified values of the stock tank GLR of the produced fluid. Select from the following options:

Injection Gas Rate:

The actual (fixed) amount of lift gas to be injected. PIPESIM will always inject this amount at the specified depth regardless of whether or not the gas injection systems (that is the casing head pressure or gas lift valve) can deliver this amount.

Set GLR to ...:

The injection rate will be calculated so that the stock tank gas lift ratio of the tubing fluid downstream of the injection point is equal to this value. **Note**, if this value is less than the gas lift ratio of the tubing fluid upstream of the injection, then gas will actually be removed.

Increase GLR by ...

The injection rate will be calculated so that the stock tank gas lift ratio of the tubing fluid is increased by an amount equal to this value.

Sensitivity of the gas lift quantity

To perform sensitivity of the gas lift quantity, use the [artificial lift operation \(p.213\)](#).

Surface temperature

the temperature of the lift gas at the surface. This is used to compute the lift gas temperature at the injection point. If the temperature is omitted, the lift gas temperature is assumed to be the same as the production fluid temperature.

Injection gas

The following injection gas properties depend on the fluid model:

Blackoil fluid

Specify the gas specific gravity. If this is omitted, the specific gravity of the gas from the production fluid is used (default 0.64).

Compositional fluid:

Specify the composition of the injection gas fluid.

Optional data

This data does not affect the amount of gas that can be passed through the gas lift point. The data allows further calculations to be performed. For example, if the Valve Port Diameter is provided, the pressure drop across the valve and the Joule-Thomson temperature change are calculated. This data is also needed if the [Alhanati instability \(p.232\)](#) check is required — see [OPTIONS \(p.583\)](#).

Valve Port Diameter

The operating gas lift valve port (orifice) diameter.

Surface injection pressure

The gas lift injection surface pressure (upstream of the surface injection choke).

Set Gas Lift Injection point/points

The Gas Lift Injection points do not take into account the gas lift system. It is assumed that the quantity of lift gas requested is fully injected into the production string at the specified depth(s) and that this takes no account of the available injection pressure or valve details. You can specify different injection depths and injection rates at each depth. To allow the program to calculate the injection depth, based on the available injection pressure, use the [Gas Lift valves \(p.84\)](#) method.

PIPESIM offers the following options for setting the gas lift injection points.

If the depth(s) of injection and injection rate(s) are known, do the following:

1. Create a tubing object and double-click it to edit it.
2. In simple profile enter the gas lift depth and the gas lift properties.
3. In detailed profile set the Gas Lift injection points in the **Downhole equipment** tab and enter the gas lift properties .

Gas Lift Valves

The actual gas lift valve depths are specified. For normal PIPESIM operations, gas is injected at the specified rate at the deepest possible valve depth (taking into account the available injection pressure). For the **Gas Lift Diagnostics** operation only, the actual gas throughput for each valve is calculated based on the injection pressure, production pressure, valve details and valve status. For this operation, details of the gas lift system are required (valve size, P_{tro} and so on). The valve throttling response is modeled (based on the bellows load rate of the valve).

Allows gas lift valves to be selected from a database and placed in the tubing string.

Installed Design**Spreadsheet**

This is a list of installed valves:

Valve MD

the depth of the valve

Valve Choke size

(optional)

All other data is populated from the database when a valve is added by using the **Add** button.

Note: To expand the spreadsheet and to see available buttons, select **Edit Valve Details (only used for Gas Lift Diagnostics)** check box.

Add (p.86)... (Valve Lookup)

add a valve from the database to the installed valve table. The valve will be added to the bottom of the current list.

Edit Selected Valve

replace selected valve with the selected entry from the database. You can also double click the cell under model or Port size that corresponds to the required depth.

Remove Selected Valve

remove the selected valve

Remove All Valves

remove all valves

Note: Take care with this option.

Gas Lift System Properties**Available Injection Pressure**

the available injection operating pressure

Injection Gas Rate

the specified injection gas rate for normal PIPESIM operations. (For the Diagnostics operation only, this is the maximum amount of gas that can be injected through all valves).

Surface Injection Temperature

the temperature of the lift gas at the surface. This will then compute the lift gas temperature at the injection point. If the temperature is omitted then the lift gas temperature will be assumed to be the same as the production fluid temperature.

Minimum Valve Injection DP

a minimum differential between injection and production pressures to allow injection at a valve depth.

Max Allowable Injection TVD

maximum possible injection depth (TVD). Normally the packer depth.

Gas specific gravity

the specific gravity of the lift gas. If omitted then the specific gravity of the gas from the production strung will be used (default 0.64)

Valve database[Valve Database \(p.193\)](#)

The database contains a list of the current valves available. To add your own valves to the database, use **Data » New/Edit Gas Lift Valve...**

Spreadsheet

list all the valves in the database that match the criteria defined by the filter. A valve can be selected by selecting the appropriate row.

Add valve

add the selected valve to the installed design table.

Filter

Use this to apply a filter to the listed valves.

Manufacturer

valve manufacturer filter

Type

Valve type filter

- IPO
- PPO-N: Production Pressure Operated Valve that uses a nitrogen charged dome as the loading element to cause the valve to close. Most gas lift equipment manufacturers use a valve setting temperature base of 60 degrees for nitrogen charged gas lift valves. The valve is submerged in a 60 degrees F water bath to ensure a constant nitrogen temperature in the dome of each valve during the test rack setting procedure.
- PPO-S: Production Pressure Operated Valve with a spring to preload the bellows and hold the valve stem on the port (for example, a spring is used as the loading element to cause the valve to close). They are also called as PPO unbalanced spring valve. The advantage of this type of PPO valve is that there is no temperature effects to consider when setting the valves opening pressure.
- Orifice
- Dummy

Size filter

Valve outer diameter

Series filter

series type

Refresh

update the spreadsheet listing by applying the filters

Remedial Coiled Tubing / Velocity String

This option allows the modeling of a remedial coiled tubing string inserted inside the tubing. You enter the coiled tubing Outside Diameter and its injection gas rate (normally nitrogen). The PIPESIM model then simulates an annular flow path between the coiled tubing string and the tubing specified in the **Tubing Configurations** window.

Gas is injected into the produced fluid (at the depth of the coiled string) at the user specified gas injection rate. You can also optionally enter the coiled tubing surface injection pressure (and coiled tubing ID). PIPESIM will then calculate whether the injection pressure is sufficient to inject the specified gas rate or not. If the pressure is not sufficient, then a warning message is written to the output and summary files, however, the specified gas rate will still be injected. Therefore, PIPESIM will not calculate the coiled tubing injection gas rate; it is user specified.

The remedial coiled tubing description in PIPESIM has no association (or functionality) with Gas Lift Design routines. Therefore PIPESIM does not perform coiled tubing gas lift design. The gas lift mandrels and valves will be assumed to be installed in the tubing specified in **Tubing Configurations** window.

To model coiled tubing, do the following:

1. Double-click the tubing object. Select **Detailed Model**.
2. Select the **Remedial Coiled Tubing** option on the tubing's **Downhole Equipment** tab.
3. Properties
 - Bottom MD:
 - OD
 - Injection
4. Injection Gas Rate
5. GLR
 - Surface Injection Gas Temperature
 - Gas specific Gravity = 0.967 for N₂
6. Coiled Tubing Pressure Drop Calculation (optional)
 - Coiled Tubing ID
 - Surface Injection Gas Pressure
7. To model a velocity string, set the gas injection rate to 0.

Tubing and Pipeline Tables

Equipment/Tubing sizing OR physical property sensitivity (1 parameter)

To size tubing or a piece of equipment, perform the following basic steps:

1. Build the required model, [pipeline tools \(p.91\)](#) or [well \(p.51\)](#).
2. Include one of the following:
 - tubing/flowline/equipment to be sized
 - the physical property to vary

3. Select the [Pressure/Temperature profile \(p.196\)](#) operation.
4. Select the sensitivity parameter.
5. Enter the data for the sensitivity parameter.
6. [Run \(p.195\)](#) the operation.
7. [Save the model \(p.35\)](#).

Equipment/Tubing sizing OR physical property sensitivity (multiple parameters)

To size tubing or a piece of equipment with multiple sensitivity parameters, perform the following basic steps:

1. [Build the required model, \(p.51\) pipeline tools \(p.91\) or well \(p.51\)](#).
2. Include one of the following:
 - tubing/flowline/equipment to be sized
 - the physical property to vary
3. Select the [System Analysis \(p.196\)](#) operation.
4. Select the multiple sensitivity option.
5. Select the x-axis parameter.
6. Enter the data for the x-axis parameters.
7. Select the multiple sensitivity parameters.
8. Enter the data for the sensitivity parameters.
9. Decide whether the sensitivity parameters are permuted or change in step.
10. [Run \(p.195\)](#) the operation.
11. [Save the model \(p.35\)](#).

Nodal Analysis Point

This defines where the system is to be broken into two for the [Nodal Analysis \(p.200\)](#) operation. Nodal analysis breaks the system into two parts around a particular (solution) point and then computes the inflow to and outflow from that point. The solution node is defined as the location where the pressure differential upstream (inflow) and downstream (outflow) of the node is zero. This is represented graphically by the intersection point of the inflow and outflow performance curves; this is the operating point. The Nodal Analysis point must be connected between two objects.

Nodal Analysis is normally performed at the following locations:

Bottom hole

Place the Nodal Analysis point between the completion and the tubing.

Well head - upstream of any wellhead equipment

Place the Nodal Analysis point between the tubing and the equipment.

Well head - downstream of any wellhead equipment.

Place the Nodal Analysis point between the equipment and the following object (flowline, riser, and so on).

Riser base

Place the Nodal Analysis point between the flowline and the riser.

See also: [Connecting objects \(p.34\)](#)

Injections Wells

Besides standard production wells, you can model the following injections wells in PIPESIM:

To model an injection well, do the following:

1. Define the well (that is, its flowline, tubing, completion, fluid model, and so on), as for a production well.
2. Add a [source \(p.118\)](#) to the start of the system.
3. Set the well injectivity using the **Well PI** option in the **Well completion** dialog.
4. Define the operation. Remember that inlet conditions are now at the surface, and outlet conditions at the completion or reservoir.

In the [network module \(p.39\)](#), injection wells are modeled explicitly using the Injection Well icon.

Injection Performance

Perform the following basic steps to build an injection well model:

1. Select the [units \(p.170\)](#) set of your preference.
2. Add a source object at the inlet.
3. [Add \(p.34\)](#) the necessary [components \(p.50\)](#) to the model (tubing, choke, and so on) and defined the necessary data.
4. Determine the completion of the well, one of the following:
 - [Vertical \(p.57\)](#)
 - [Horizontal \(p.51\)](#)
5. Define the fluid specification ([black oil \(p.505\)](#), [compositional \(p.141\)](#) or [steam \(p.166\)](#)). Water injection systems can be modeled by black oil or compositional.
6. (Optional) [Calibrate the fluid \(p.505\)](#).
7. Define the flow correlation to use.
8. [Save the model \(p.35\)](#).

Once the basic model has been developed, a number of operations can be performed, or the well model can be utilized in additional PIPESIM modules.

Operations

- [Correlation matching \(p.121\)](#)
- [Pressure/Temperature profile \(p.196\)](#)
- [Physical property sensitivity - 1 parameter \(p.87\)](#)
- [Physical property sensitivity - Multiple parameters \(p.88\)](#)
- [Equipment/flowline sizing - 1 parameter \(p.87\)](#)

- Equipment/flowline sizing - Multiple parameters (p.88)
- Nodal Analysis (p.200)

Multiple Completion Well

Model multiple completions by setting the following properties to different values for each layer:

- Static pressure
- Temperature
- Depth
- IPR specification
- Fluid model

The IPR for each individual layer can be specified by using any of the [standard completion \(p.398\)](#) options. Similarly the fluid description for each layer can be specified using the standard [black oil \(p.505\)](#) or [compositional \(p.141\)](#) fluid description.

A multiple completion model is defined by adding the required number of completions to the model and connecting with the [tubing \(p.76\)](#) object, (the length of each tubing string should be set to reflect the depth of the completions). If the layers are stacked, then they can be connected using the connector object rather than by the tubing object as the connector object has zero length.

When performing [Nodal Analysis \(p.200\)](#) on multiple completions the composite IPR curve will be created (and shown) for the inflow performance. The individual layer IPR curves can however, be displayed using the plot utility.

Note: As of release 2008.1, multi-layer injection well models can be simulated in PIPESIM.

Liquid Loading

Implementation

The functional form of Turner's Equation is implemented in PIPESIM and you have the ability to specify a correction factor (E). The default correction factor is 1.2, corresponding to Turner's original model. You can specify E factors suggested by any of the authors listed in the [table \(p.396\)](#), or input your own E factor between 0.1-10. This is done using the ELIQLOADING subcode of the [OPTIONS \(p.609\)](#) maincode.

You can control what velocity ratio

$$\frac{V_t}{V_m} \text{ - or } \frac{V_t}{V_g} \text{ - is reported using the LLVELOCITY subcode of the } \text{OPTIONS (p.609)} \text{ maincode.}$$

Inclination Angle

There is a critical pipe angle where the droplets fall back onto a liquid film on the lower section of the pipe instead of falling back to the bottom of the well. Thus, the stable rate would depend on liquid film characteristics rather than a force balance on a droplet. This may indicate that the liquid loading equations are valid for vertical wells and should not be used beyond a certain angle. You

can select a maximum pipe angle to restrict ν_t calculation for vertical or near vertical section of well using the LLANGLE subcode of the [OPTIONS \(p.609\)](#) maincode.

Pipeline tools

Pipeline and facilities specific operations

The operations specific to pipeline and facilities are the following:

- [Pressure Temperature Profile \(p.196\)](#)
- [Flow correlation comparison \(p.197\)](#)
- [Systems Analysis \(p.196\)](#)

Once the basic model has been developed, the following operations can be performed, or the model can be utilized in additional PIPESIM modules.

- [Correlation matching \(p.121\)](#)
- [Pressure/Temperature profile \(p.196\)](#)
- [Physical property sensitivity - 1 parameter \(p.87\)](#)
- [Physical property sensitivity - Multiple parameters \(p.88\)](#)
- [Equipment/flowline sizing - 1 parameter \(p.87\)](#)
- [Equipment/flowline sizing - Multiple parameters \(p.88\)](#)
- [Multiphase booster design \(p.102\)](#)

See also [Hot Keys \(p.22\)](#), [Main toolbar \(p.24\)](#), [Well Performance \(p.50\)](#), [Network \(p.35\)](#), [components, case studies \(p.269\)](#) and [How to add objects \(p.34\)](#).

Pipeline and facilities model

Perform the following basic steps to build a pipeline and facilities model:

1. Select the [units \(p.170\)](#) set of your preference.
2. Add a source to the model.
3. [Add \(p.34\)](#) the necessary components to the model (flowline, equipment, and so on.) and defined the necessary data.
4. Set [heat transfer \(p.100\)](#) options for each flowline.
5. Define the fluid specification ([black oil \(p.505\)](#) or [compositional \(p.141\)](#)).
6. (Optional) [Calibrate the fluid \(p.505\)](#).
7. Define suitable Horizontal correlations (p.170) and [vertical correlations \(p.170\)](#).
8. [Save the model \(p.35\)](#).

Compressor Details

Either centrifugal or reciprocating compressors can be modeled. The basic compressor model uses centrifugal compressor equations to determine the relationship between inlet pressure and temperature, outlet pressure and temperature, flowrate, shaft power, and efficiency. It is also possible to use user-defined [compressor curves \(p.186\)](#) to describe the relationship between

differential pressure, flowrate, and efficiency for a range of compressor speeds. Reciprocating compressors only work off user entered [performance curves \(p.185\)](#). If compressor curves are used the compressor speed and number of stages become additional factors.

Double-click the compressor to open its **Properties** tab. Enter at least one of the following parameters:

Discharge Pressure

Pressure at the compressor outlet (default 20,000 psia). This is not allowed in [Network \(p.35\)](#) models.

Pressure Differential

Pressure increase (positive) or decrease (negative) across the compressor. The default is 10,000 psia.

Pressure Ratio

Discharge pressure / suction pressure ratio. (P_{out}/P_{in}) The default is 1,000.

Power (shaft power) (default unlimited).

Power of the compressor.

Route

Select one of the following options: (see also [Reciprocating Compressor \(p.93\)](#))

- **Adiabatic** — compressor follows an adiabatic (no heat transfer) compression process (the default). This is available for blackoil modeling and compositional modeling.
- **Polytropic** — compressor follows a polytropic compression process. This is available for black oil modeling and compositional modeling. This is the default for the black oil model.
- **Mollier** — compressor follows an isoentropic (no change in entropy) compression process. This is available for compositional models only, it is their default.

Efficiency (default 100%).

Efficiency of the compressor.

Honor Stonewall limit

If this is set, the compressor will honor the stonewall.

The remaining quantities are calculated using centrifugal compressor equations. If more than one value is supplied, the parameter which leads to the smallest compressor differential pressure is used, and all other supplied parameters are discarded.

See also: [How to add objects \(p.34\)](#)

User Curves

Select the **User Curves** check box to specify that a compressor curve is to be used. This displays the parameters listed below:

Select

Select the compressor curve to use from the [database \(p.187\)](#).

Compressor type:

Specify Centrifugal or Reciprocating.

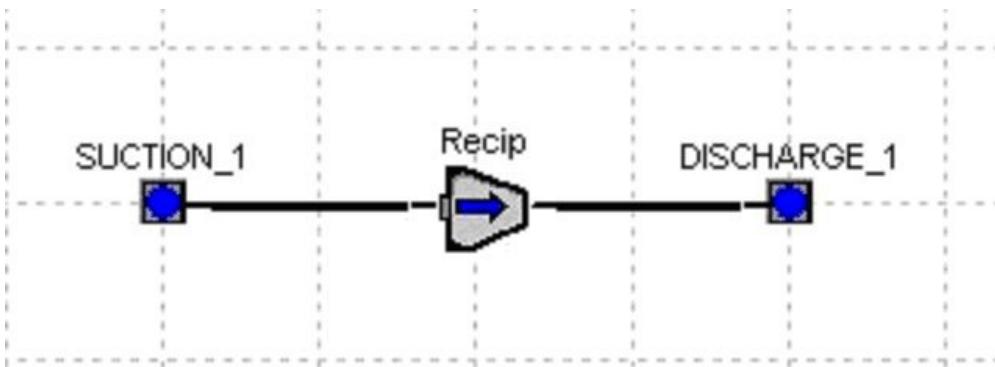
Speed

If a compressor curve is supplied, the speed can also be specified. This is used to adjust the supplied curve against its specified speed. The adjustment is made using the so called affinity or fan laws, which state that "capacity is directly proportional to speed, head is proportional to square of speed, and power is proportional to cube of speed". See [Data Entry \(p.184\)](#) for entering curve data. If a reciprocating compressor is selected, a variable speed device may be used and this requires the minimum and maximum operating speed.

Reciprocating Compressor Details***Adding a new Reciprocating Compressor to the PIPESIM database***

To add a new reciprocating compressor to the database:

1. Select **Data** » **New Reciprocating Compressor** (only available in the single-branch menu).
2. Set the **Abs. Min. Suction Pressure** to zero and the **Abs. Maximum Capacity** to a value equal to or less than the maximum flowrate on the suction pressure curves.



3. Add at least two Suction Pressure versus FlowRate [performance curves \(p.185\)](#) for a range of discharge pressures covering all likely eventualities. These two curves can be identical if required

Adding a Reciprocating Compressor to a PIPESIM-Net Model

The compressor must exist in its own PIPESIM-Net branch. That is, no other piping or equipment can be present.

To add a compressor to a model, do the following:

1. Double-click on the compressor icon and select **User Curves**.
2. Choose the required reciprocating compressor from the PIPESIM database and enter the operating speed.
3. Leave all other compressor parameters (**Discharge Pressure**, **Pressure Differential**, and so on) blank.

Note: If the model has two or more reciprocating compressors in series (for example, field compressors followed by a plant compressor), any downstream compressor must have a greater capacity than the upstream compressor. (Even if only fractionally greater, for example 10.00 mmscf/d followed by 10.01 mmscf/d.)

Engine Keyword Tool

Placing an Engine Keyword Tool (EKT) in the model allows access to the [PIPESIM Input Language \(p.595\)](#). This is a powerful input language that gives direct access to the calculation engine via the GUI, reducing the need for you to embark on the more complex Expert mode, or Keyword mode. Expert mode is where keywords are entered directly into a text file and the calculation engine processes the contents of the file to produce results.

The EKT can be used in the following situations:

- to access advanced features
- to access new features that do not have an entry mechanism in the GUI yet.

See also adding [keywords at the start of the model \(p.595\)](#).

Expander Details

The basic expander model uses centrifugal expander equations to determine the relationship between inlet pressure and temperature, outlet pressure and temperature, flowrate, shaft power, and efficiency. Built in, or user-developed expander curves can be used to describe the relationship between differential pressure, flowrate, and efficiency for a range of expander speeds. If expander curves are used, therefore, the expander speed and number of stages become additional factors.

Enter at least one of the following parameters:

Discharge Pressure

Pressure at the expander outlet (not allowed in [Network \(p.35\)](#) models). (The default is 20 psia.)

Pressure Differential

Pressure increase (positive) or decrease (negative) across the expander. (The default is 10,000 psia.)

Pressure Ratio

Inlet pressure / discharge pressure ratio.(Pin/Pout). (The default is 1,000.)

Power

Shaft power, the power of the expander. (The default is unlimited.)

Route

[Adiabatic \(p.468\)](#)

[Polytropic \(p.468\)](#)

default for black oil model

Mollier (p.468)

default for compositional models

Efficiency

Efficiency of the expander (default =100%).

The remaining quantities are calculated using centrifugal expander equations. If more than one value is supplied, the parameter which leads to the smallest expander differential pressure is used, and all other supplied parameters are discarded.

See also [keywords \(p.683\)](#)

User Curves

Select the **User Curves** check box to specify that an expander curve is to be used. This displays the parameters listed below:

Select

Select the expander curve to use from the [database \(p.187\)](#).

% Speed

If an expander curve is supplied, the % speed may also be specified. This is used to adjust the supplied curve against its specified speed. The adjustment is done using the so called affinity or fan laws, which state that "capacity is directly proportional to speed, head is proportional to square of speed, and power is proportional to cube of speed". See [Data Entry \(p.184\)](#) for entering curve data.

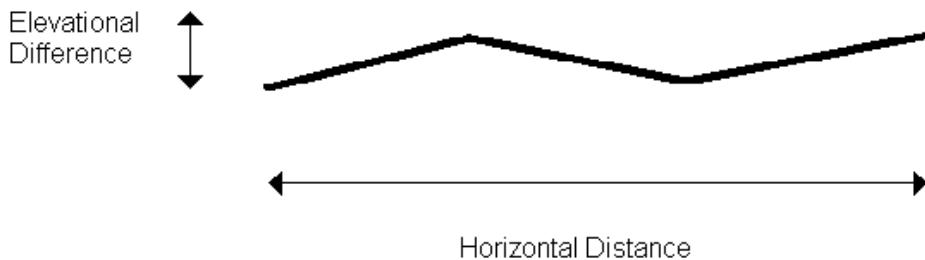
Flowline Details

Placing a flowline in the model allows modeling of horizontal or near-horizontal flow (up or downhill). [Heat transfer \(p.100\)](#) can be modeled by entering or computing an overall heat transfer coefficient (U value).

Each flowline has fixed characteristics in terms of inner diameter, outer diameter, roughness, and so on. If any of these dimensions changes along the pipeline length, add an additional flowline object to the model.

Double-click the flowline to view its **Properties** tab. The flowline's profile (distance vs. elevation) can be defined by either a [simple \(p.96\)](#) or [detailed \(p.97\)](#) model. Specify which to use, using the **Preferred Pipe Description** list. The simple model is often used when an initial model is being developed and the exact profile of the flowline is unknown. When additional data becomes available, you can switch to the detailed model.

Simple model



For a simple model, provide the following information:

Rate of Undulations

An artificial factor that can be used to automatically introduce some undulations into the flowline to reflect terrain effects. The value entered is the total change in elevation for every 1,000 units (feet, m, and so on). Thus, a rate of undulations of 10 on a flowline 1,000 m long would have a peak of 5 m at 500 m along the flowline. This is different to the manner in which this was handled in the PIPESIM Suite. To model a totally flat flowline, set the rate to 0. Default = 10.

Horizontal Distance

the horizontal distance covered by the complete flowline, NOT the length of the pipeline!

Elevation Difference

the change in elevation between the start (source end for a single branch model) and the end of the flowline object. Enter a negative value for a downhill flowline, a positive value for an uphill one. Thus the elevation change is relative to the object, not from some datum.

Inner Diameter (ID)

the flowline inner diameter for the complete flowline. If this value changes significantly part way along the flowline, add a second flowline object.

Wall Thickness

the wall thickness, excluding any coatings for the complete flowline. If this value changes significantly part way along the flowline, add a second flowline object. Default = 0.5 inches, 12.7 mm.

Roughness

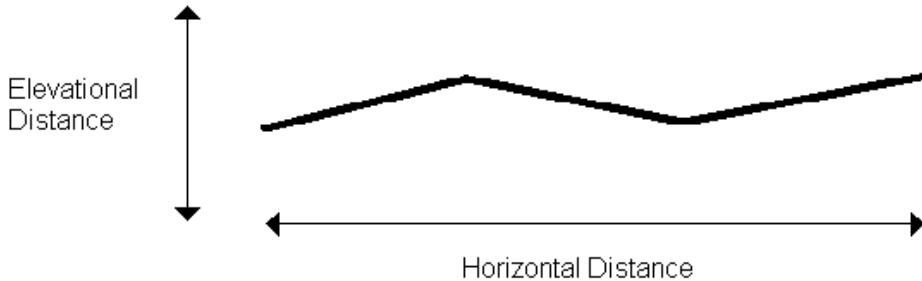
the absolute pipe roughness for the complete flowline. [Typical values \(p.572\)](#). If this value changes significantly part way along the flowline, add a second flowline object. Default = 0.001 inches, 0.0254 mm

Ambient Temperature

the surrounding ambient temperature for the complete flowline. If this value changes significantly part way along the flowline, add a second flowline object.

Once the flowline details have been entered its profile can be plotted using the **Schematic** button.

Detailed model



For a detailed model, provide the following information:

Distance / Elevation

node points detailing the flowline profile. This can be cut and pasted from another spreadsheet application. PIPESIM uses the difference in elevation between each point. Therefore the datum is not important.

Ambient Temperature profile

this is optional except at the first and last nodes, which **must** have a temperature defined.

U Value

this is optional except at the first and last nodes, which **must** have a U value specified. If the option to calculate the U value was selected, this column will show "calc".

Inner diameter (ID)

flowline inner diameter for the complete flowline. If this value changes significantly part way along the flowline then a second flowline object should be added.

Wall Thickness

wall thickness, excluding any coatings for the complete flowline. If this value changes significantly part way along the flowline then a second flowline object should be added. Default = 0.001 inches, 0.0254 mm

Roughness

absolute pipe roughness for the complete flowline. [Typical values \(p.572\)](#). If this value changes significantly part way along the flowline then a second flowline object should be added. Default = 0.001 inches, 0.0254 mm.

Label

text field that can be used to identify key points in the flowline. These will then appear in the main output file.

Once the flowline details have been entered its profile can be plotted using the **Schematic** button.

Flowline Properties

Use **Setup > Flowline Properties..** to display, edit and sort flowline details in a model. The following parameters can be edited using this view:

Distance

horizontal distance of the flowline. This cannot be changed if the detailed profile model has been used to define the flowline.

Elevation Difference (+ve for uphill, -ve for down hill).

this cannot be changed if the detailed profile model has been used to define the flowline.

ID

Inside Diameter

Wall Thickness

pipe wall thickness

Roughness

pipe roughness

Ambient Temperature

surrounding ambient temperature. This cannot be changed if the detailed profile model has been used to define the flowline

Use Detailed Profile

If this is set, the flowline has been defined in detail and the Distance, elevation difference and ambient temp cannot be changed here.

Other data associated with a flowline can be edited by selecting the required flowline and using the **Edit...** button. This displays the normal data entry screens for a flowline.

Heat Exchanger Details

[Adding \(p.34\)](#) a heat exchanger in the model allows a fluid temperature and/or pressure change to be modeled. Double-click the Heat Exchanger to view the options.

Temperature Change

To model a temperature change, do the following:

1. Specify the temperature effects of the heat exchanger on the flowing fluid.

Note: The heating or cooling media itself is not modeled.

2. Select the parameter to use from the following and enter a value for it:

Discharge Temperature

a fixed flowing fluid outlet temperature

Temperature differential

temperature increase (positive) or decrease (negative) across the heat exchanger

Duty

enthalpy change across the exchanger (positive values mean that heat is added to the fluid)

Pressure Change

To model a pressure change, do the following:

1. Specify the pressure effects of the heat exchanger on the flowing fluid.
2. Select the parameter to use from the following and enter a value for it:

Discharge Pressure

a fixed flowing fluid discharge (outlet pressure)

Pressure Drop

pressure decrease (negative) across the heat exchanger

See also: [How to add objects \(p.34\)](#)

Equipment

Adding an equipment item to the model allows simulation of a generic unit operation, in which the pressure and/or temperature of the stream are modified.

To configure the equipment, double-click it to view its **Properties** tab.

Route

Specify the thermodynamic mode of operation of the equipment. This is used to calculate the fluid outlet temperature. The options are:

Isenthalpic

Constant enthalpy (the default)

Isentropic

Constant entropy

Isothermal

Constant temperature

For instance, to simulate chokes and the Joule-Thomson cooling across pressure reduction valves, the most appropriate choice is isenthalpic.

Temperature Change

Use this to specify the temperature effects of the equipment on the flowing fluid.

Note: The heating or cooling media itself is not modeled.

Select one of the following and enter a data value:

Discharge Temperature

a fixed flowing fluid outlet temperature

Temperature differential

temperature increase (positive) or decrease (negative) across the equipment

Duty

enthalpy change across the equipment (positive values mean that heat is added to the fluid)

Pressure Change

Use this to specify the pressure effects of the equipment on the flowing fluid.

Select one of the following and enter a data value:

Discharge Pressure

a fixed flowing fluid discharge (outlet pressure)

Pressure Drop

pressure decrease (negative) across the equipment

See also: [How to add objects \(p.34\)](#)

Heat Transfer

Heat transfer between the fluid and surroundings occurs, depending upon the temperature gradient. To configure this, right-click on a flowline and select **Data**, then select the **Heat Transfer** tab.

There are two options for modeling the heat transfer:

- [Input U value \(p.100\)](#) — you enter an overall heat transfer coefficient (U value) based upon the pipe outside diameter.
- [Calculate U value \(p.100\)](#) — the heat transfer coefficient is calculated from entered pipe and conductivity data.

Input U value

You can select the most appropriate U value, or use a default value for an insulated, coated or bare flowline.

Note: These values only apply when the [simple view \(p.96\)](#) is used. When the [detailed view \(p.97\)](#) is selected, the U values are entered directly into the detailed view spreadsheet. The U value is based on the pipe outside diameter and reported in the output file. To allow the value to be reported at a different reference point, use the [HEAT HTCRD \(p.707\)](#) keyword through the [EKT \(p.94\)](#).

In addition the Inside Film Coefficient (IFC) can either be included in the overall U value, or calculated separately and added to the U value.

Calculate U value

Detailed heat transfer input and output information can be obtained by selecting **Setup** » **Define Output** and then selecting the heat transfer options in the [Define output \(p.173\)](#) dialog.

Enter the following information to compute the overall Heat Transfer coefficient:

Pipe coatings

Up to 26 pipe coatings can be added in the table. For each pipe section, specify the following:

- K (Thermal conductivity) of the material. [Typical values \(p.572\)](#)
- Thickness of the pipe coating.
- Description - for reference purposes only.

Pipe overall outside Diameter

This is automatically computed from data supplied.

Pipe conductivity

The default is 35 Bth/hr/ft/F [Typical values \(p.572\)](#)

Ambient fluid

Specify either air or water depending on whether the system is onshore or offshore.

Ambient median

For a pipe that is exposed to different media along its length, use two (or more) pipe sections. Select Air or Water.

Velocity

Average velocity of the surrounding median. (Default = 0.1 ft/s)

Pipe burial data

Burial

The burial depth of the flowline from the pipe's center line. Enter a positive value for buried pipe and negative for a pipe elevated above the ground. Enter 0 for a half-buried pipe. The default value is -2000 mm. [More examples \(p.716\)](#)

Ground conductivity

This is for flowlines only.

Status

This changes, based on the entries above.

Injection Point

Placing an injection point in the model allows a side stream to be modeled.

Double-click the injection point and specify the following :

Fluid

fluid to be injected. Select the radio button for one of the following:

- [Black Oil \(p.505\)](#)
- [Compositional \(p.141\)](#)
- [PVT File](#)
- [MFL File \(p.153\)](#)

Temperature

temperature of the incoming fluid at the injection point

Rate

incoming liquid/mass/gas rate (this can be entered at stock tank or flowing conditions using the "@" button).

In a network model a stream removed at the separator can be reinjected using the [re-injection tool](#) (p.704).

Important Notes

Note the following:

- The injected fluid type must be consistent with the main fluid type set in the model.
- The injection fluid toolbox can be used anywhere within the model; at the surface, or in the tubing as a downhole object.

Multiphase Boosters

Multiphase boosters can be used where the fluid contains both liquid and gas phases. A number of models for booster performance are available, based on the different types of booster available on the market:

- [Generic Multiphase Booster](#) (p.102)
- Twin Screw Multiphase Booster
 - [Nuovo Pignone](#) (p.104)
 - [User defined](#) (p.105)
 - [Generic](#) (p.103)
- Helico-Axial Multiphase Booster
 - [Framo 2009](#) (p.105)
 - [Framo 1999](#) (p.108)

Generic Multiphase Booster

The **generic multiphase booster** module is based on splitting the multiphase fluid in its constituent liquid and gaseous phases at booster suction. Conventional pump theory is used to calculate the shaft power required to raise the pressure of the liquid phase and conventional compressor theory is used to calculate the shaft power required to raise the pressure of the gaseous phase. The total shaft power required by the multiphase booster is the sum of the two computed shaft powers. The use of the generic multiphase booster module is envisaged for preliminary checks only and can be used to benchmark the multiphase booster power requirement.

As a rule, using the generic booster module with realistic values for pump efficiency and compressor efficiency, a 'best case' estimate for multiphase boosting power is obtained. That is, using a specifically designed multiphase booster, its power requirement will be higher than that calculated using the generic booster module (up to 100% higher is possible). Realistic values for pump and compressor efficiency are considered to be 75% for initial screening purposes.

Data Entry

Double-click the Multiphase booster to open its **Properties** tab. Enter one or more of the following parameters:

Discharge Pressure

discharge pressure from multiphase pump (default 20,000 psia)

Pressure Differential

pressure differential across multiphase pump (default 10,000 psia)

Pressure Ratio

the multiphase pump pressure ratio is the absolute discharge pressure divided by the absolute suction pressure (pressure ratio must be between 0 and 5)

Power

power available for pump (default unlimited)

Pump Efficiency

Efficiency of the pump. The default is 100%.

Compressor Efficiency

Efficiency of the compressor. The default is 100%.

When differential pressure, pressure ratio, or discharge pressure is specified, the program calculates the required power for multiphase boosting. Alternatively, when power is specified the program calculates the differential pressure (or pressure ratio or discharge pressure) that can be developed by the booster. When more than one variable is specified, the program determines the "most limiting specification" during the course of the simulation, and recalculates the other parameters.

As the generic booster is modeled as a parallel pump/compressor, you can define pump and compressor efficiency. Pump efficiency and compressor efficiency are percentage values; the defaults are 100%. [Typical efficiency values \(p.484\)](#)

The output provided using the **generic multiphase booster** option includes the calculated total power requirement, as well as its constituent shares of compression power and pumping power. See also [MPUMP keyword \(p.696\)](#).

Twin Screw Boosters

After adding a Multiphase booster, double-click it to open the Multiphase Booster dialog. In the **Properties** tab, select a multiphase booster type from the list .

The following twin screw multiphase boosters are available:

Generic Twin Screw Multiphase Booster

In essence, this option provides the same functionality and [input \(p.103\)](#) as the generic type multiphase booster, but rather than assuming a parallel pump/compressor process, actual twin screw multiphase booster performance data is used to predict pump performance.

Based on actual pump operating conditions, the program selects a suitable pump from the generic database available in PIPESIM and calculates performance accordingly. Note that, using the **generic twin screw multiphase booster** option, you do NOT specify a pump model, but leave it

to the program to select a suitable pump for the operating conditions. If different operating conditions need to be investigated, the program may select a different pump for each of these; if you want to investigate the performance of one specific model under all these operating conditions, use one of the modeling options below.

Based on the calculations performed under this option, the main parameters characterizing the multiphase booster operating point are provided in the output reports or plots. These include booster speed (as a % of nominal speed), total volumetric flow rate at booster suction, gas volume fraction (GVF) at booster suction, booster differential pressure, and booster power requirement. In addition, the program suggests which pump to select from the generic pump database for a more detailed investigation.

Twin Screw Multiphase Booster

This option is based on nominal capacity data only, rather than offering a detailed model booster from vendor catalogues or similar. This approach provides sufficient detail for you to make a first assessment, does not require you to fully know all of the booster models available for use. It lets you progress your studies without requiring detailed vendor information. If you wish to simulate a specific vendor model, use the **user defined** curve.

The generic database contains performance data for seven different twin screw multiphase booster models. Rather than asking the program to select a suitable pump for investigated operating conditions, you can preselect one of these seven boosters and assess its performance under different operating conditions. This lets you, for example, study the effect of the same booster under different operating conditions (for example, later in field life with lower reservoir pressure), or study the effect of changes to pump operations (for example, the effect of changes in pump speed). Unlike the [Generic Twin Screw Multiphase Booster \(p.103\)](#) option, this option therefore allows you to explore the operating envelope of a specific pump model.

The seven different twin screw multiphase boosters available from the generic database in PIPESIM are characterized by their theoretical flow rate at 100% speed, typically referred to as 'booster capacity' or 'nominal capacity'. It is the flow rate of the selected booster when operated at 100% speed and zero differential pressure (that is no internal leakage). The seven nominal capacities available are:

- 100 m³/h
- 250 m³/h
- 500 m³/h
- 700 m³/h
- 1000 m³/h
- 1400 m³/h
- 2000 m³/h

Select a booster from the **Nominal Booster** list.

In the **% speed** box, you can also specify booster speed as a percentage of nominal speed; the speed range a specific booster model can operate at is typically from 50% to 110% of nominal booster speed.

Based on the calculations performed under this option, the main parameters characterizing the multiphase booster operating point are provided in the output reports or plots. These include booster speed (in % of nominal speed), total volumetric flow rate at booster suction, GVF at booster suction, booster differential pressure and booster power requirement. PIPESIM also gives the option of downloading the simulation output onto a multiphase pump process data sheet. This allows multiphase pump operating conditions to be communicated and further information to be obtained directly from the vendors.

Note the following:

- Performance curves in the PIPESIM generic database are provided for different values of booster speed, suction pressure and GVF at suction. Where actual conditions do not exactly match the conditions for which the performance curve is supplied, the program interpolates as appropriate.
- Performance curves in the PIPESIM generic database are valid for a liquid viscosity of 6 cSt. For viscosities different from this, the program incorporates a module to make the required corrections.

Vendor Twin Screw Multiphase Booster

Offering the same functionality and [input \(p.103\)](#) as the [Twin Screw Multiphase Booster \(p.104\)](#) option, you can input this data as a **user defined booster performance**. Thus this entered data will allow you to model a booster specifically of interest and determine the effects of changing operating parameters on system and booster performance.

This option may be of specific interest when you need to assess the suitability of a multiphase pump of which performance data is well known. This could be when you have received a technical proposal from a vendor, or when a booster has already entered field operation, but you want to explore the effects of changing operating conditions or booster operations for that specific pump and system where the booster is placed in.

Obtain the booster data file from the vendor. Enter the location of the file in the **Full Path of Vendor File** box.

In the **% speed** box, you can also specify booster speed as a percentage of nominal speed; the speed range a specific booster model can operate at is typically from 50% to 110% of nominal booster speed.

Note: Schlumberger does not supply any vendor data file, but please contact us for more details. See also [Vendor Performance Curve Format. \(p.110\)](#)

Framo 2009 Helico-Axial Multiphase Booster

See also [Multiphase Boosting Technology \(p.470\)](#), [FRAMO 2009 \(Optional\) \(p.686\)](#).

The Framo 2009 pump model supersedes the earlier 1999 model ([p.108](#)). You can select the **Framo 2009 Multiphase Booster** from the list in the **Multiphase Booster** dialog box.

Pump performance is defined by pump data files provided by [Framo \(p.586\)](#). The pumps are classified based on their nominal capacity and maximum pressure differential achievable. The nominal capacity refers to the total volumetric flowrate (gas & liquid) at suction conditions and is thus dependent on suction pressure and fluid properties. The quickest way to determine the pump requirements is to first model the system using a [generic multiphase pump \(p.102\)](#) based on

expected operating conditions and inspect the summary or output file to determine the total volumetric flowrates at the suction.

The following pumps are available as part of the standard Framo catalog:

Pump	Impeller Diameter		Nominal Capacity			Maximum DP	
	mm	in	m^3/h	m^3/day	BBL/d	bar	psi
FRAMO Helico-Axial 310-250/180	310	12.2	250	6000	37739	180	2611
FRAMO Helico-Axial 310-400/180	310	12.2	400	9600	60382	180	2611
FRAMO Helico-Axial 310-500/45	310	12.2	500	12000	75478	45	653
FRAMO Helico-Axial 310-500/180	310	12.2	500	12000	75478	180	2611
FRAMO Helico-Axial 310-600/120	310	12.2	600	14400	90573	120	1740
FRAMO Helico-Axial 310-700/45	310	12.2	700	16800	105669	45	653
FRAMO Helico-Axial 310-800/120	310	12.2	800	19200	120764	120	1740
FRAMO Helico-Axial 310-900/45	310	12.2	900	21600	135860	45	653
FRAMO Helico-Axial 310-1100/45	310	12.2	1100	26400	166051	45	653
FRAMO Helico-Axial 310-1100/120	310	12.2	1100	26400	166051	120	1740
FRAMO Helico-Axial 360-1200/38	360	14.2	1200	28800	181146	38	551
FRAMO Helico-Axial 360-1500/38	360	14.2	1500	36000	226433	38	551
FRAMO Helico-Axial 360-1800/38	360	14.2	1800	43200	271719	38	551

Note: The numbers in the pump name refer to impeller diameter (mm); nominal capacity (m³/hr); Max. dP (bar).

New files can be added if they become available, as follows:

1. Add any new file to the Framo pump directory (for a default installation this is C:\Program Files \Schlumberger\PIPESIM\Data\framo09).
2. Update the index of pump files to reference the new file; see the file pipesim_framo.xml in the above directory for more details.

Modeling the Pump

All types of multiphase boosters allow standard operating constraints to be defined. The user may enter any combination of the following parameters or none at all. Note that the default values for all parameters are unrealistically high; this ensures that each parameter will not be the limit on pump performance, unless the user supplies a lower, more realistic value. If none of the parameters is specified, the limit of pump performance will usually be the speed, and which will in this case be 100%; or otherwise the DP limit for the specific pump model.

- Discharge pressure: Pressure at the pump outlet. Not allowed in Network models and generally not advisable to specify. *The default is 20,000 psia.*
- Pressure differential: Pressure increase across the pump. *The default is 10,000 psia.*

- Pressure ratio (Pout/Pin): Discharge pressure / suction pressure ratio. *The default is 1,000.*
- Power (shaft power): Power available for the pump. *The default is unlimited*, though it may be constrained by the selected pump model.

For example, when differential pressure is specified, the program will calculate the required power for multiphase boosting, or alternatively when power is specified the program will calculate the differential pressure that can be developed by the pump. When both values are specified, the program will determine the "most limiting specification" during the course of the simulation, and recalculate the other parameter. When none of the two operating parameters are specified, the program will calculate the differential pressure that can be developed by the selected pump for the prevailing suction conditions.

There are several operating parameters specific to the Framo 2009 Multiphase booster which may be optionally supplied. These include:

- Tuning factor: This is a linear multiplier on calculated pump differential pressure used to match field operating conditions. Allowable range is from 0.7 to 1.5.
- Number of pumps in parallel: Used to simulate multiple identical pumps operating in parallel (equal flow split between pumps). Allowable range is from 1 to 7.
- Speed Limit: Allows the user to specify a maximum speed limit for the pump. Allowable range is from 20% to 100%.
- Flow in recirculation: Allows the user to specify the quantity of flow in recirculation.

It should be noted that if the required flowrate through the pump is less than the minimum required the amount of fluid recirculated through this pump to achieve the minimum rate is automatically added.

Output

Based on the calculations performed, parameters characterizing the multiphase pump operating point are provided in the summary and output reports and available as system plot variables.

These include:

- Pump speed
- Total volumetric flow rate at pump suction
- GVF at pump suction
- Pump differential pressure
- Power requirement
- Fraction of the flow in recirculation (if applicable)
- Minimum flowrate (with zero recirculation)

Additionally, the limiting constraint will be reported in the output report.

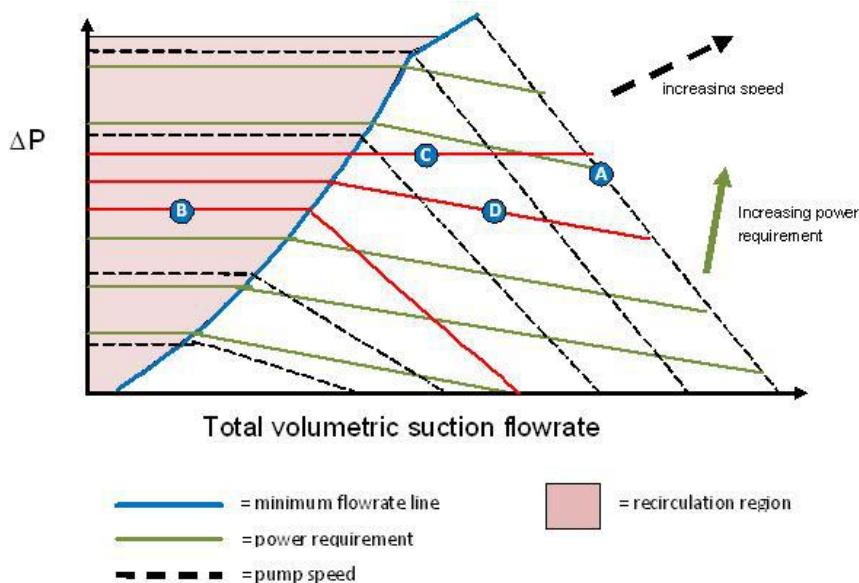
If **Simulated Operating Conditions Performance Map** is selected, a performance map file is created for each case. These files can be viewed using **PsPlot**. The files are named "pump_n.pfm", where "pump" is the pump model name, "n" is the case number, and ".pfm" is the file extension. The file created by the first case can be plotted by right-clicking on the **Multiphase Booster** icon in the model view, and selecting the option "Booster map". If you want to plot files from subsequent cases, you should select the desired file from **PsPlot's** file > open menu.

Note: The pump name should be unique, otherwise plot files will over write each other. Alternatively the user can rename the pump for each scenario to see the booster map for each case/scenario.

A performance map is valid for a given set of operating conditions, including:

- Gas volume fraction
- Suction pressure
- Liquid density
- Liquid viscosity

A typical performance map is shown below.



The operating point is displayed on the map. Several scenarios are illustrated.

- A. Pump is operating unconstrained. Pump speed is 100%.
- B. Pump is limited by a speed constraint. Since the operating point falls to the left of the minimum flowrate line, some fluid is being recirculated.
- C. Pump is limited by a constraint on pressure differential. Pump speed is reduced.
- D. Pump is limited by power available. Pump speed is reduced.

Framo 1999 Helico-Axial Multiphase Booster

The Framo 1999 model has now been superseded by the [2009 model \(p.105\)](#).

You can select the **Framo 1999 Multiphase Booster** from the list in the **Multiphase Booster** dialog box.

When choosing the helico axial multiphase booster, you can enter the pump driver type, pump parameters and pump operating conditions, as detailed below.

You can select one of three drive types from the dropdown-menu in the dialog box

Hydraulic Drive

characterizes Framo hydraulically driven multiphase pump, usable subsea and topsides/onshore

Electrical Air cooled drive

characterizes Framo E-motor driven multiphase pump, usable topsides/onshore only

Electrical Oil cooled drive

characterizes Framo E-motor driven multiphase pump, usable subsea and topsides/onshore

To define the size of the pump and number of stages, enter the following pump parameters:

Flowrate Parameter (FQ)

The pump flow parameter determines the size of the multiphase pump relative to a reference pump; FQ=1 denotes a pump equal in size to the reference pump. The range of values for FQ is 0.1 to infinity, values FQ>1 typically indicate that more than one pump is in parallel operation. The default value for FQ is 1.0.

Head Parameter (FZ)

The pump head parameter determines the number of stages inside the multiphase pump relative to a reference pump; FZ=1 denotes a pump with a number of stages equal to the maximum number of stages in the reference pump. The range of values for FZ is 0 to 1.0. The default value for FZ is 1.0.

Based on the values for FQ and FZ, the program will calculate the required booster speed for the prevailing operating conditions. Booster speed is shown as pump speed parameter FN, being the speed expressed as a fraction of the pump maximum continuous speed. FN is recommended to have values within the range of 0.50 - 0.85.

A value for FN significantly larger than 0.85 means that the pump is running at an unnecessary high speed because the actual multiphase flow rate is higher than the flow rate recommended for the selected pump. It will not be harmful to run with FN>0.85, but there will be no extra margin to further increase speed should operating conditions require it. Therefore, for initial screening purposes, a maximum value of FN = 0.85 is rather prudent.

A value for FN smaller than 0.50 means that the pump is running at an unnecessary low speed because the actual multiphase flow rate is lower than the flow rate recommended for the selected pump.

The differential pressure developed by the booster is proportional to pump head, which is dependent on the number of stages and pump speed. With pump speed targeted to be between 50 and 85% of the booster maximum continuous speed, changes to pump head are mainly implemented by changing the number of stages within a pump. The number of stages is reflected in pump head parameter FZ; the larger the value of FZ, the higher the number of stages.

The differential pressure developed by a single multiphase pump is set to be no greater than 70 bar, the maximum available differential pressure for the multiphase booster.

Whereas the selection of pump driver type is straightforward, the selection of a suitable pump flow rate parameter FQ and pump head parameter FZ is somewhat more complicated. For this reason, you have the option to omit suggested values for FQ and FZ in the first instance, upon which the

program (once a simulation has been run) will recommend values for FQ and FZ. If FQ and FZ are entered by the user, the program will perform the simulation on the basis of the pump model as defined by those parameters.

For pump operating conditions, you can enter any of the following values, or a combination of these:

Pressure Differential

pressure differential across multiphase pump

Power

power available for pump

When differential pressure is specified, the program will calculate the required power for multiphase boosting, or alternatively when power is specified the program will calculate the differential pressure that can be developed by the booster. When both values are specified, the program will determine the "most limiting specification" during the course of the simulation, and recalculate the other parameter. When none of the two operating parameters are specified, the program will calculate the differential pressure that can be developed by the selected booster for the prevailing suction conditions and associated shaft power requirement; the thus calculated operating point will be on the 'best efficiency line' as shown in the user guide.

Based on the calculations performed under the option 'helico-axial multiphase booster', main parameters characterizing the multiphase booster operating point are provided to the output reports or plots, and include booster speed parameter, total volumetric flow rate at booster suction, GVF at booster suction, booster differential pressure and booster power requirement. In addition, PIPESIM has the option to download the simulation output onto a multiphase pump process data sheet. This will therefore allow multiphase pump operating conditions to be communicated and further information to be obtained by the user directly from the vendors. See also [FMPUMP \(p.686\)](#) keyword.

Vendor Performance Curve Format for Twin Screw Multiphase Pumps

PIPESIM allows you to model the behavior of twin screw multiphase pumps by defining a set of performance curves. These curves specify volumetric rate/dP performance and associated power requirement to account for variations in pump speed, suction pressure and GVF at suction conditions. The following sensitivities are suggested:

Gas Volume Fraction

0%, 20%, 50%, 70%, 85%, 95%

Speed

50%, 70%, 80%, 90%, 100%

Suction pressure

1 bara, 10 bara, 50 bara

The first line of the file must be:

Booster Performance Data

The second line of the file should be the vendor name.

The file then contains sets of curve data. The first line of the curve data should contain the pump model, nominal flow rate in m³/h, the GVF, the speed in % and the suction pressure all in the correct order. This is followed by data lines giving the points on the curves and should contain the pump differential pressure (bar), volumetric flowrate (m³/h), and the power (kW).

```
Model      NomQv (m3/hr)  GVF Speed (%max)  Psuction (bar)
dP (bar)   Qvol (m3/hr)   Power
"          "           "
"          "           "
"          "           "
```

At the end of the file a capital END must be present.

Typical Booster Performance Data File

```
Booster Performance Data
NUOVO_PIGONE
PSP250 2000 0 100 1
0 2001 150
10 1907 706
20 1842 1262
30 1787 1817
40 1735 2373
50 1684 2929
60 1633 3485
70 1582 4041
PSP250 2000 20 100 1
0 2001 134
10 1997 702
20 1991 1259
30 1979 1815
40 1960 2371
50 1930 2927
60 1911 3483
70 1892 4039
...
END
```

Notes:

1. The data should always be separated by at least once character space.
2. Comment lines (beginning with a "!") and blank lines can be used to make the table easier to read. They will be ignored by the program.
3. The Model name should contain at least one non numeric character to allow the program to determine where each set of curve data begins and ends.
4. The maximum number of different nominal flow rates (NomQv) is 10.
5. The maximum number of different GVF values is 10.
6. The maximum number of different speed values is 10.
7. The maximum number of different suction pressure values is 10.
8. There must be one curve for each combination of nominal flow rate, GVF, speed and suction pressure.

9. There must be at least two points on each curve.
 10. The total number of curve points must not exceed 10000.
 11. There should be no stationary points on the curves, that is, for each curve the dP values should increase, the Qvol values should decrease and the Power values should increase.
-

Flowrate Operator Details

Only use the Flowrate Operator in a single branch model, not in the network module.

A flowrate operator is a rate change device that can increase or decrease the fluid flowrate at that point in the system. All fluid properties remain the same as those upstream of the object. The flowrate units (for the adder) are the same as those specified for the source flowrate (that is liquid or gas), as set by an operation. For example, if the operation sets the flowrate as 1,000 STB/d then the adder will add x STB/d.

Double-click the Flowrate Operator to set the following properties:

Adder

values > 0 apply a rate increase, < 0 apply a rate decrease

Multiplier

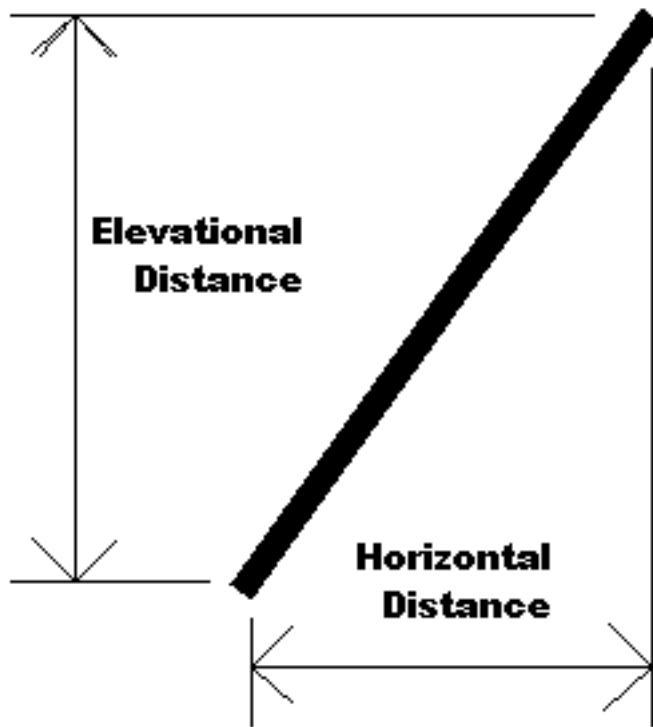
values > 1 apply a rate increase, < 1 apply a rate decrease.

Note: This feature cannot be used when the feed flowrate is computed.

Riser Details

Placing a riser in the model allows the modelling of vertically or near-vertical flow (up, down or inclined). [Heat transfer \(p.100\)](#) is modeled by either entering or computing an overall heat transfer coefficient (U value).

Note: The direction of the riser on the screen has no effect on the flow direction. If the riser is defined as going downhill, the elevation difference MUST be set to a negative (-ve) value. To model a riser going uphill, the elevation difference must be set to a positive (+ve) value. Take care to ensure that the correct sign is used, as the definition of up or downhill depends on your starting point!



A riser can be defined by either a [simple \(p.113\)](#) or [detailed \(p.114\)](#) profile.

Simple profile

Provide the following information:

Horizontal distance

the horizontal distance covered by the complete riser, NOT the length of the riser!

Elevation difference

the change in elevation between the start (source end for a single branch model) and the end of the riser. Enter a negative value for a downhill riser and positive for an uphill one.

Inner diameter (ID)

the riser inner diameter for the complete riser. If this value changes significantly part way along the riser then a second riser object should be added.

Wall Thickness

the wall thickness, excluding any coatings for the complete riser. If this value changes significantly part way along the riser then a second riser object should be added.

Roughness

the absolute pipe roughness for the complete riser. [Typical values \(p.572\)](#). If this value changes significantly part way along the riser then a second riser object should be added.

Ambient temperature

the surrounding ambient temperature for the complete riser. If this value changes significantly part way along the riser then a second riser object should be added.

Once the riser details have been entered, its profile can be plotted using the **Schematic** button.

Detailed profile

Provide the following information:

Distance / Elevation

Node points detailing the riser profile. This can be cut and pasted from another spreadsheet application.

Ambient temperature profile (optional), at the point specified.

First and last nodes MUST have a temperature defined.

Heat transfer profile (optional).

First and last nodes MUST have a U value specified defined. If the option to calculate the U value has been invoked then this column will show "calc".

Inner diameter (ID)

the riser inner diameter for the complete riser. If this value changes significantly part way along the riser then a second riser object should be added.

Wall Thickness

the wall thickness, excluding any coatings for the complete riser. If this value changes significantly part way along the riser then a second riser object should be added.

Roughness

the absolute pipe roughness for the complete riser. [Typical values \(p.572\)](#). If this value changes significantly part way along the riser then a second riser object should be added.

Label

A text field that can be used to identify key points in the riser. These will then appear in the main output file.

Once the riser details have been entered, its profile can be plotted using the **Schematic** button.

Gas lift in the riser

PIPESIM can be used to model the common situation of gas lifting at the riser base. This is becoming a common technique in the industry.

As the riser object does not have a location to enter the gas lift location or the gas lift quantity, you may insert a tubing to "represent" the riser. By doing this the gas injection location and quantity can be specified.

In the tubing dialog, set the parameters as follows:

1. wellhead TVD to 0 - represents the top of the riser
2. riser height in the (TVD) perforations field
3. Artificial lift to single injection point
4. gas lift injection point

5. gas injection quantity (under the single G'L valve button) or a sensitivity variable in one of the operations
 6. Riser diameter (using tubing configuration)
 7. Ambient temperature at the top and base of the riser (using the **Ambient Temperature** field on the **Tubing** dialog, **Properties** tab).
 8. For clarity the objects identified can be changed to, say, "Riser 1" from "Tubing 1".
-

Note: If the artificial lift operation is to be used then in addition to the above the "normal" tubing, must be replaced with a riser object (this is due to the artificial lift dialog assuming that gas lift will be allocated to the first tubing after the completion). Other operations allow the required tubing object to be set and do not need this change.

Report Tool Details

Placing a report tool in a single branch model gives additional reporting of the conditions at that point in the model. To do this, click the Report Tool icon on the main toolbar, then click on the model. Double-click the Report Tool object to open its **Properties** tab. There is a check box for each report item available, as listed below. Select the box for each item you want to report on.

Flow Map

This applies to multiphase flow regions only. If this box is ticked, a high resolution map is produced, showing the multiphase flow regimes plotted against superficial liquid and gas velocities. Note that the map is scaled so that the operating point is always inside it. To display a map, select **Reports** » **Flow Regime Map** and select it in the list of available maps. The list shows all available maps for the case that has been run.

Phase Split

This applies to compositional cases only. If this box is ticked, a table is printed in the output file, showing the constituents of each phase.

Stock Tank Fluid Properties

If this box is ticked, a table is printed in the output file, showing various fluid properties such as gas and liquid volumetric flowrate at stock tank conditions.

Flowing Fluid Properties

If this box is ticked, a table is printed in the output file, showing various fluid properties such as gas and liquid volumetric flowrate at flowing conditions.

Cumulative Values

If this box is ticked, a table is printed in the output file, showing properties, such as liquid holdup, where the cumulative value may be of interest

Multiphase Flow Values

If this box is ticked, a table is printed in the output file, showing various multiphase fluid flow values such as superficial gas and liquid velocities.

Slugging Values

If this box is ticked, a table is printed in the output file, showing various slug values such as slug lengths and frequencies.

Sphere Generated Liquid Volume

If this box is ticked, a table is printed in the output file, showing various sphere generated liquid values.

Heat Transfer Input Values

If this box is ticked, a table is printed in the output file, showing heat transfer input data.

Typically, the following are printed: Pipe and Coating Thicknesses and Thermal Conductivities, Soil Thermal Conductivity, Burial Depth, Ambient Fluid Velocity, and Ambient Temperature.

Heat Transfer Output Values

If this box is ticked, a table is printed in the output file showing heat transfer output data.

Typically, the following information is printed: Distance, Fluid Temperature, Fluid Enthalpy, and Heat Transfer Coefficients.

Composition Details

This applies to compositional cases only. If this box is ticked, a table is printed in the output file showing some compositional data such as water specific gravity.

Phase Envelope

This applies to compositional cases only. If this box is ticked, the fluid's phase envelope is automatically computed and stored in the branch's plot file. In network cases, the file that contains the phase envelope will be noted in the output file.

Separator Details

Placing a separator in the model removes up to 100% of the gas, water or liquid (oil plus water) phase. A flash is performed to determine the quantity of each phase at the separator inlet. Double-click the separator and set the following properties:

Type

Liquid, Gas or Water separator.

Efficiency

The efficiency or efficiency fraction refers to the amount of that material removed.

For example, a 90% efficient water separator removes 90% of the water. From that point onward only flow of the remaining fluids will be modeled.

Note: To remove both gas and water place two separators in the model.

See also: [How to add objects \(p.34\)](#)

Track all streams from separator

A stream can only be removed from the system by the separator. The removed fluid is normally lost to the model, but it can be reinjected into the network by using the [reinjection tool \(p.704\)](#). All the fluid removed at the separator is reinjected. (To only reinject a percentage of the removed fluid then the [multiplier/added \(p.112\)](#) object should be used in the separated branch.)

The following are required;

- The branch that contains the separator of the fluid stream to reinject

- The reinjection temperature. If this is omitted then the separator temperature is used
- An estimate of the fluids flowrate (required)

Pump Details

The basic pump model uses centrifugal pump equations to determine the relationship between inlet pressure and temperature, outlet pressure and temperature, flowrate, shaft power, hydraulic power and efficiency. Built in or user-developed pump curves can be used to describe the relationship between differential pressure, flowrate, and efficiency for a range of pump speeds. If pump curves are used, therefore, the pump speed and number of stages become additional factors.

Enter at least one of the following parameters:

Discharge pressure

Pressure at the pump outlet. Not allowed in [Network \(p.35\)](#) models. The default is 20,000 psia.

Pressure differential

Pressure increase (positive) or decrease (negative) across the pump. The default is 10,000 psi.

Pressure ratio (Pout/Pin)

Discharge pressure / suction pressure ratio. The default is 1,000.

Power (shaft power)

Power of the pump. The default is unlimited.

Efficiency

Efficiency of the pump. The default is 100%.

User Pumps

Viscosity Correction

Available for user-supplied pumps. Pump curve correction for viscosity effect using the [TURZO method \(p.514\)](#). Because viscosity degrades head, flow rate, and efficiency, usually more power is needed to lift the same amount of fluid. To apply viscosity correction using another method, use the [engine keywords \(p.701\)](#).

User Curves

Select **User Curves** to use a performance curve for the pump. See [data entry \(p.184\)](#) for details of how to enter pump curve data.

Speed

If a pump curve is supplied, the speed can also be specified. This is used to adjust the supplied curve against its specified speed. The adjustment is made using the so called affinity or fan laws, which state that "capacity is directly proportional to speed, head is proportional to square of speed, and power is proportional to cube of speed".

Stages

Number of stages of the pump.

The remaining quantities will then be calculated using centrifugal pump equations. If more than one value is supplied, then the parameter which leads to the smallest pump differential pressure will be used, and all other supplied parameters will be discarded.

The main pump equations used are as follows: Hydraulic Power = Flowrate x Differential Pressure
Hydraulic Power = Shaft Power x Efficiency

See also [keywords \(p.701\)](#).

Source

The generic source object is a means by which you can specify explicit upstream boundary conditions of pressure and temperature in a given model. By its nature, the source component is generic and must be placed upstream of the first component in any model. Examples of where a source component can be used to emulate input boundary flow conditions include:

- Wellhead conditions in a subsea production flowline system.
- Export flow conditions from an offshore platform.
- Production well test conditions at a down hole gauge.

Note: The generic source object can be used in place of a well performance model or horizontal well performance model in cases where reservoir inflow is a fixed value relationship. [More details \(p.118\)](#).

Source, Sink and Boundary Conditions

In order to solve the system, boundary conditions must be provided at the source or sink. This means setting values for the following:

- Pressure
- Flowrate
- Temperature - always required at a source and always computed at the sink.

In a single branch model, the boundary conditions are: Inlet pressure, Outlet pressure, Flowrate and Inlet temperature. Of these the inlet temperature and two of the others must be set. Set these either by using the source dialog or from the operation that is being performed.

The **Source** dialog requires the following:

Properties

Pressure

Required, but can be overwritten by the [operation \(p.195\)](#).

Temperature

Required, but can be overwritten by the operation.

Fluid model

This is used to set local fluid data that overrides the global setting, that is:

- Water cut
- GOR

Example

In the single branch mode there is no sink object. The outlet of the last object (furthest from a source object) is defined as the sink point. For example, a model with a vertical completion tubing, choke and flowline, will have the following;

Inlet pressure

Static reservoir pressure

Outlet pressure

Pressure at the end of the flowline

Flowrate

Flowrate (liquid, gas or mass) at the model's source.

Note: The flowrate may be different at the model's sink due to a rate change device added to the model (for example, an [adder \(p.112\)](#), [multiplier \(p.112\)](#), [side stream injection \(p.101\)](#), and so on).

Any of the above values can be computed. For example:

- Set: static reservoir pressure and outlet pressure. PIPESIM computes the maximum flowrate that can be obtained.
- Set: static reservoir pressure and flowrate. PIPESIM computes the outlet pressure. (This is the fastest option)
- Set: static flowrate and outlet pressure. PIPESIM computes the static reservoir pressure by using the IPR after the bottom hole flowing pressure has been determined.

Inlet Pressure

This is the pressure that exists at the inlet to the first object in the model. This table shows examples:

In a model with:	The inlet pressure is:
Vertical Completion - Tubing - Choke	the static reservoir pressure
Source - Flowline	the pressure inlet to the flowline
Source - Tubing - Vertical completion	the well head pressure (this is an injection well)
Horizontal Completion - Tubing	the "toe" pressure

Outlet Pressure

This is the pressure that exists at the end of the last object in the model. This table shows examples:

In a model with:	The outlet pressure is:
Vertical Completion - Tubing - Choke - Flowline - Riser	at the top of the riser
Vertical Completion - Tubing - Choke	downstream of the choke

In a model with:	The outlet pressure is:
Source - Flowline	at the end of the flowline
Source - Tubing - Vertical Completion	the static reservoir pressure (this is an injection well)

Flowrate

This is the flowrate at the inlet to the first object in the model. The inlet conditions (pressure and temperature) that the flowrate pertains to can be entered. The flowrate at the model outlet can differ from that at the inlet if any of the following are used in the model:

- Separator
- Flowrate Operator
- Injection point

1.3 Flow Correlations

Select flow correlations using **Setup » Flow Correlations...** Make the following selections for vertical and horizontal correlation from either those supplied as standard or from a [User Defined Correlations \(p.650\)](#).

For more information about flow correlations, refer to [Horizontal Multiphase Flow Correlation \(p.372\)](#) and [Vertical Horizontal Multiphase Flow Correlations \(p.377\)](#).

1.3.1 Selecting a flow correlation

Source

This defines the source of the correlation for either the vertical or horizontal correlation. Options are:

bja

The code for the correlation has been developed by Baker Jardine (now Schlumberger) and has been extensively tested. (default)

tulsa

The code for the correlation was developed by Tulsa University Fluid Flow Projects (TUFFP). This is the original library of empirical correlations developed in 1989. The code is usually of academic quality and may give errors. This library is included mainly for benchmarking purposes. It is not recommended for general use.

OLGAS

The OLGAS steady-state model developed by SINTEF/SPT. (Only available if the OLGAS 2-phase or 3-phase option has been purchased.)

Tulsa 3-Phase Flow Correlation

Latest unified mechanistic model developed by the Tulsa University Fluid Flow Projects (TUFFP). It is available for general use.

LEDA

The Leda steady-state model developed by SINTEF/Kongsberg.

Neotec

The Neotec flow correlations were developed by a company called Neotec based in Calgary. Neotec was formed in 1972 by Gary Gregory and Khalid Aziz, professors at the University of Calgary who specialized in Multiphase Flow research. Neotec developed several software applications used in the oil and gas industry, including WELLFLO, PIPEFLO and FORGAS. In 2010, Neotec was acquired by SPT Group and became part of Schlumberger in 2012 when Schlumberger acquired SPT Group.

See also [User Defined Correlations \(p.650\)](#)

Correlation

The available correlations depend on the "source" selected.

The [Flow Correlation Comparison \(p.197\)](#) operation allows users to compare various flow correlations with an option of using measured data. The [Data Matching \(p.198\)](#) operation has been specifically developed to assist with this task of determining the most suitable flow correlation from well test data and calculating the friction and holdup multipliers to achieve a best match.

Options Control

This allows to control globally in conjunction with the flow correlation selected at each branch level, which flow correlation is used in the network.

When you select:

- Use local branch options: PIPESIM will use whatever setting is at the branch level (locally defined flow correlation or network global flow correlation - see also the flow correlation window accessible by right clicking on any branch or in the single branch viewer window)
- Use network options: PIPESIM will use the network globally defined flow correlation for all branches. This is achieved with an override statement in the network engine file.
- Apply network options to all branches: PIPESIM will override in all branches the flow correlation settings by the network global setting. **WARNING:** You will lose ANY local flow correlation defined in any branch.

1.3.2 Defining User Flow Correlations

PIPESIM lets you define your own multiphase flow correlations and implement them directly. See [User Defined correlations \(p.650\)](#) for further details on how to create your own flow correlation DLL.

Tuning flow correlations

Flow correlations can be adjusted (tuned) by using friction and hold up factors.

Data matching

If measured pressure data is available, the [Data matching \(p.198\)](#) operation can be used to calculate friction and holdup factors automatically.

Flow Correlation comparison

The [Flow correlation comparison \(p.197\)](#) operation can be used to compare different flow correlations with measured data. Unlike the [Data matching \(p.198\)](#) operation it does not try to tune any parameters, so will be quicker to run.

Suggested correlations

If no production data are available, Schlumberger have found the following to give satisfactory results based on previous studies using field data:

Single phase system

[Moody \(p.386\)](#)

Vertical oil well

[Hagedorn and Brown \(p.381\)](#)

Highly deviated oil well

[Hagedorn and Brown \(p.381\)](#) or [Duns and Ros \(p.378\)](#) or [OLGA-S \(p.381\)](#)

Gas/condensate well

[Hagedorn and Brown \(p.381\)](#)

Oil pipelines

[Oliemans \(p.375\)](#)

Gas/condensate pipelines

[BJA Correlation \(p.372\)](#)

Correlation	Vertical and Predominantly Vertical Oil Wells (p.377)	Highly Deviated Oil Wells (p.377)	Vertical Gas/Condensate Wells (p.377)	Oil Pipelines (p.372)	Gas/Condensate Pipelines (p.372)
Duns and Ros	yes	yes	yes	yes	yes
Orkiszewski	yes	no	yes	no	no
Hagedorn and Brown	yes	no	yes	no	no
Beggs and Brill Revised	yes	yes	yes	yes	yes
Beggs and Brill Original	yes	yes	yes	yes	yes
Mukherjee and Brill	yes	yes	yes	yes	yes
Govier, Aziz and Forgasi	yes	yes	yes	yes	yes
NoSlip	yes	yes	yes	yes	yes
OLGAS	yes	yes	yes	yes	yes

Ansari	yes	no	yes	no	no
BJA for Condensates	no	no	yes	no	yes
AGA and Flanigan	no	no	no	no	yes
Oliemans	no	no	no	yes	yes
Gray	no	no	yes	no	no
Gray Modified	no	no	yes	no	no
Xiao	no	no	no	yes	yes
LEDA	yes	yes	yes	yes	yes
TUFFP	yes	yes	yes	yes	yes

1.3.3 Flow Regime Number and Flow Pattern

The following flow regimes (patterns) are available for plotting:

- gas-liquid
- oil-water

Gas-Liquid

The following table shows the current gas-liquid regimes:

Flow Regime	Flow Map	Number
'Undefined'	'? '	-1
'Smooth'	'Sm'	00
'Stratified'	'St'	01
'Annular'	'A '	02
'Slug'	'Sl'	03
'Bubble'	'B '	04
'Segregated'	'Sg'	05
'Transition'	'T '	06
'Intermittent'	'I '	07
'Distributed'	'D '	08
'Strat. Smooth'	'Ss'	09
'Strat. Wavy'	'Sw'	10
'Strat. Dispersed'	'Sd'	11
'Annular Disp.'	'Ad'	12
'Intermit./Slug '	'Is'	13

Flow Regime	Flow Map	Number
'Churn'	'C'	14
'Dispersed Bubble'	'Db'	15
'Single Phase'	'Sp'	16
'Mist'	'M'	17
'Liquid'	'L'	18
'Gas'	'G'	19
'Dense Phase'	'De'	20
'Annular Mist'	'Am'	21
'Two Phase '	'2'	22
'Wave '	'W'	23
'Dispersed '	'Dp'	24
'Plug '	'P'	25
'Tr. Bubble/Slug'	'Tb'	26
'Tr. Froth/Slug'	'Tf'	27
'Heading'	'H'	28
'Oil '	'O'	29
'Oil/Water '	'Ow'	30
'Water/Oil '	'Wo'	31
'Water '	'Wa'	32
'Froth '	'F'	33
'Strat. 3-phase '	'S3'	34
'Bubbly'	'B'	35

Oil-Water

The following table shows the current oil-water regimes:

Flow Regime	Flow Map	Number
'Undefined'	'?'	-1
'Stratified'	'St'	00
'Strat. liq. film in slug flow'	'Sf'	01
'Oil/Water'	'Ow'	02
'Water/Oil'	'Wo'	03
'Water'	'Wa'	04
'Oil'	'O'	05
'Emulsion'	'Em'	06

1.3.4 Configuration File for user specified flow correlations

The **userdll.dat** is a text file that contains PIPESIM installation configuration information. For a standard PIPESIM installation, the file can be edited by selecting **Setup » Preferences » Choose Paths** and selecting “edit” next to the userdll path. If you want to select flow correlation(s) from the **Setup » Flow Correlations** menu, you must edit the dll to define the flow correlation(s). Each dll requires a section starting with:

```
[UserDLL_32_X]
```

where 'X' can be 1, 2, 3, ... 20. This number must be unique within this userdll.dat file. Each section contains lines that start with the strings 'filename=' , "title=' , psname=' , 'epname=' , 'linktype=' , 'eptype=' , 'mpflow title =' , 'options=' , and so on.

The general syntax is as follows:

'01'	'FILENAME',	req,	pres,	str,	0,	0,	0,	pcnone,	&
'02'	'EPNAME',	opt,	pres,	str,	0,	0,	0,	pcnone,	&
'03'	'PSNAME',	opt,	pres,	str,	0,	0,	0,	pcnone,	&
'04'	'EPTYPE',	opt,	chr1,	str,	BUS01,	NUS01,	4,	pcnone,	&
'05'	'LINKTYPE',	opt,	int1,	str,	1,	2000,	2,	pcnone,	&
'06'	'SDESCRIPTION',	opt,	pres,	str,	0,	0,	0,	pcnone,	&
'07'	'LDESCRIPTION',	opt,	pres,	str,	0,	0,	0,	pcnone,	&
'07'	'TITLE',	opt,	pres,	str,	0,	0,	0,	pcnone,	&
'08'	'OPTIONS',	opt,	pres,	str,	0,	0,	0,	pcnone,	/

The EPTYPE can be

'MPFLOW',	'1',	&
'HMPFLOW',	'2',	&
'VMPFLOW',	'3',	&
'SLUG',	'0',	&
'OTHER',	'0',	&
'ETC',	'0',	&
'EQUIPMENT',	'4'	

filename =

Filename when no path is provided. The assumption is that the file exists in same directory as the PIPESIM executables. Filenames should confirm to the accepted syntax of the environment being used. Do not use special characters or embedded spaces.

Title or LDESCRIPTION =

The DLL that you can access from the **Setup » Flow Correlations** menu.

psname =

The SLB internal identifier for this selection. To specify a user equipment DLL routine, enter a unique string within the user dll for flow correlation and any engine options.

EPNAME =

The actual name of the subroutine or entry point exported from the DLL. It is case-sensitive. EPNAME can be up to 40 characters long, and must obey the ANSI FORTRAN-77 standard. For example, it must be composed of uppercase alphabetical and decimal digits and it must start with a letter of the alphabet. You can code the routines in any language, but you must compile the routines to present a Microsoft FORTRAN-compatible calling linkage.

In the userdll.dat file, you can enter the same EPNAME many times on different lines with various options and titles.

EPTYPE =

An integer that describes the job that the routine performs. It also controls where and why the PIPESIM engine calls it. Valid TYPES are:

- 1 — Multiphase Flow correlation, valid for both vertical and horizontal flow
- 2 — Multiphase Flow correlation, valid for vertical flow
- 3 — Multiphase Flow correlation, valid for horizontal flow

LINKTYPE =

An integer that specifies the LINKAGE of the routine. The LINKAGE comprises the exact number, type, meaning, and order of the arguments or formal parameters declared by the routine, the direction of the data for each argument (IN to or OUT of the routine), the Units of the data (if any), and its valid range. The following LINKAGE values are valid:

- 21 (Compaq) and 1021 (Intel) — Three phase model
- 22 (Compaq) and 1022 (Intel) — Two phase model

SDESCRIPTION =

A character string that describes the correlation or job performed by the routine.

OPTIONS =

Keyword(s) for the PIPESIM engine or for the particular correlation. This text must be passed to the engine as-is, enclosed in quotes, as the value of the OPTIONS= keyword.

The engine recognizes the following options:

+SLUG	SRTCA only. Selects artificial slug model
+DP	SRTCA only. Adds slug pressure drop; requires SLUG
+3PHASE	SRTCA only. Selects 3-phase model
+WOD	SRTCA only. Adds water-oil dispersion; requires 3PHASE
+DISP	Same as WOD
+G	Specifies the correlation that handles single-phase gas
+L	Specifies the correlation that handles single-phase liquid

+OW	Specifies the correlation that handles oil & water with no gas
+WO	Same as +OW
+EMULn	<p>Specifies whether the emulsion model used is from PIPESIM or from a user-provided dll that is based on the value of n.</p> <ul style="list-style-type: none"> n=0 corresponds to the default model supplied in the user DLL. n=1 corresponds to the model that you selected in the PIPESIM GUI. (The supplied liquid emulsion viscosity in the linkage 21 specification “overrides” any liquid viscosity used internally in the user DLL.) n=k (where k>1) corresponds to any further models supplied in the user DLL.
+EXTRAINx.x! yy!z	The extra input as “!” separated values. All extra input should be converted already to eng units and parsed in an array of 200 real and passed to the user dll as is. Any enumeration should be converted to a number. The extra input are written in the same order as provided by the get_extra_inputs function.
olga_key	LIBOLGAS only. This keyword activates olga-specific call to Set_Vendor_Key, which is required for olga release 4.13, nov 2004.
N.B.	All unrecognized strings will be silently ignored.

As an example:

```
[UserDll_32_9]
title = User-defined 3P
ident = 3P_Demo_UFC
filename = UFC3P_Demo.dll
Copyright = $3-Phase_demo_UFC 32-bit DLL
ep1 = 3P_Demo_UFC, UFC_3P, 1021, 1, Joe Bloggs Demo BBR Correlation,+G +L +
OW +EXTRAIN1!200!30000!50 +EMUL0
```

1.4 Fluid modeling

In a network model different fluid descriptions cannot be used. That is, the model must be either black oil, compositional or steam. However, each source can have its own fluid properties or use shared (common) data. Changes to the common (shared) data will results in all sources using that data being changed.

1.4.1 Creating or Editing Fluid Models

Fluid modeling is a fundamental aspect of multiphase flow simulation. Before running any simulations, you need to create one or more fluid models. Fluid models are used to describe phase behavior and provide physical and transport properties of the fluid required for any simulation run.

PIPESIM* supports several types of fluids. These fluid types are currently available:

Fluid Type	Description
Black Oil	Black oil fluids are modeled as three phases: oil, gas, and water. The amount of each phase is defined at stock tank conditions by specifying gas and water phase ratios, typically the gas/oil ratio (GOR) and the watercut. Properties at pressures and temperatures other than stock tank are determined by correlations. Water is assumed to remain in the liquid phase. The key property for determining the phase behavior of hydrocarbons is the solution gas/oil ratio, which is used to calculate the amount of the gas dissolved in the oil at a given pressure and temperature.
Compositional fluid	Compositional fluid refers to a fluid made up of a number of components. These can be real molecules, such as methane, ethane, or water, known as library components, or user-defined pseudo-components that represent the properties of several molecules known as petroleum fractions. The Flash packages available in PIPESIM include ECLIPSE 300, GERG, and Multiflash. When the Multiflash package is chosen in the Compositional fluid mode, the fluid definition is done using the PIPESIM interface (Multiflash "native"). However, when the MFL File mode is chosen, the fluid definition is done using files generated by launching the Multiflash interface (Multiflash MFL file). Refer to the section Multiflash in the Compositional Fluid mode (native) vs. Multiflash MFL files (p.127) for more details on these two options. A compositional fluid can be defined within PIPESIM and written to a PVT file.
PVT File	PVT files are generated from a third-party PVT simulator such as AspenTech's HYSYS®, Calsep's PVTsim, KBC's Multiflash™, GUTS, DBR Solids, and OLI's ScaleChem. The PVT simulator writes a data file that is stored externally to PIPESIM in an ASCII file. When properties are required at a specific pressure and temperature (PT), the data file will be interrogated, and interpolation (or extrapolation) used to find the properties at the required PT point. You may define only one PVT fluid per model.
MFL File	MFL files are generated from KBC's Multiflash software, a 3 rd -party PVT flash package available as a separate licensed module in PIPESIM. Multiflash enables full phase behavior modeling of multiphase fluids and solids using standard models with petroleum fluid characterization. You may define a new MFL fluid or edit existing MFL fluid files by launching the Multiflash interface from PIPESIM or simply use existing MFL files by pointing to their locations. Multiple MFL files can be defined in one PIPESIM model and mapped to different sources and wells in the Fluid Manager, however care must be taken to ensure that the models and components are consistent across all MFL files.

1.4.2 Modeling systems

You can model the following systems in PIPESIM:

Water Systems

Water systems can be modeled by specifying the fluid model as either:

Black Oil (p.136)

Set the water cut to 100% and specify the water S.G. All other properties will be ignored

Compositional (p.142)

Select the water component to water and set its molar rate to 1.

Gas Condensate Systems

Gas Condensate systems can be modeled by specifying the fluid model as either:

Black Oil (p.136)

Set the (oil/gas ration) OGR or (liquid/gas ratio) LGR, Gas S.G, watercut, water S.G. and API

Compositional (p.142)

Select the required components and set their molar rates.

See also: [Gas Condensate IPR \(p.398\)](#)

Dry Gas Systems

Dry Gas systems can be modeled by specifying the fluid model as either:

Black Oil (p.136)

Set the (liquid/gas ratio) LGR = 0, Gas S.G, watercut (to model a 2-phase system), water S.G.

Compositional (p.142)

Select the required components and set their molar rates.

See also: [Gas IPR \(p.398\)](#).

Fluids can be defined as follows:

- Black oil data — for black oil fluids. Please note that black oil fluids cannot be active in the same model as any type of compositional fluid.
- Compositional data — for compositional fluids. Please note that compositional fluids cannot be active in the same model as a black oil fluid.
- Steam
- PVT file — This can contain table information, for table fluids, or compositional information, for compositional fluids, or both.
- MFL file — This is a Multiflash file for Multiflash compositional fluids.

Compositional Fluid Types

Type	Description	Notes
"Vanilla" or plain composition	Specified by the user in a PIPESIM GUI dialog.	Contains default compositional properties.
MFL file composition	Created with the Multiflash GUI and stored in an .MFL file.	

Type	Description	Notes
PVT file composition	Specified by the user in a PIPESIM GUI dialog and saved to a file.	The PIPESIM GUI writes a PVT file to convey the plain compositions to the engine. Note: PVT file and PVT File table are not differentiated by the user interface. Make sure that you do not mix the two types of PVT files.
PVT file table	Specified by the user in a third-party program and saved by the user to a file.	Note: PVT file and PVT File table are not differentiated by the user interface. Make sure that you do not mix the two types of PVT files.

Important: The PIPESIM GUI enforces a single active fluid mode of the four possible types:

- Blackoil
- “Vanilla” or plain composition
- MFL file
- PVT file

You cannot mix composition with PVT, composition with blackoil, PVT with MFL, and so forth.

If possible, the PIPESIM GUI preserves the state (local or global) and the settings. For MFL and PVT files, the action of toggling between the two modes resets the file selected. Otherwise, the file selected is kept.

Where Fluids Are Defined

	Black Oil data	Compositional data	PVT file	MFL file
Menu	Setup » Black Oil...	Setup » Compositional Template... Setup » Compositional... (the network default)	Setup » PVT File...	Setup » MFL File...
Black Oil Fluid	Yes			
Compositional Fluid		Yes	Yes — if the file contains compositional data	Yes
Table Fluid			Yes — if the file contains property tables	

Table fluids are not suitable for mixing.

PVT files that contain compositional information can also be imported into PIPESIM and used as Compositional data.

1.4.3 Managing Multiple Fluid Models

Fluids can be defined as follows:

- Black oil data — for black oil fluids. Please note that black oil fluids cannot be active in the same model as any type of compositional fluid.
- Compositional data — for compositional fluids. Please note that compositional fluids cannot be active in the same model as a black oil fluid.
- Steam
- PVT file — This can contain table information, for table fluids, or compositional information, for compositional fluids, or both.
- MFL file — This is a Multiflash file for Multiflash compositional fluids.

Compositional Fluid Types

Type	Description	Notes
"Vanilla" or plain composition	Specified by the user in a PIPESIM GUI dialog.	Contains default compositional properties.
MFL file composition	Created with the Multiflash GUI and stored in an .MFL file.	
PVT file composition	Specified by the user in a PIPESIM GUI dialog and saved to a file.	The PIPESIM GUI writes a PVT file to convey the plain compositions to the engine. Note: PVT file and PVT File table are not differentiated by the user interface. Make sure that you do not mix the two types of PVT files.
PVT file table	Specified by the user in a third-party program and saved by the user to a file.	 Note: PVT file and PVT File table are not differentiated by the user interface. Make sure that you do not mix the two types of PVT files.

Important: The PIPESIM GUI enforces a single active fluid mode of the four possible types:

- Blackoil
- "Vanilla" or plain composition

- MFL file
- PVT file

You cannot mix composition with PVT, composition with blackoil, PVT with MFL, and so forth.

If possible, the PIPESIM GUI preserves the state (local or global) and the settings. For MFL and PVT files, the action of toggling between the two modes resets the file selected. Otherwise, the file selected is kept.

Where Fluids Are Defined

	Black Oil data	Compositional data	PVT file	MFL file
Menu	Setup » Black Oil...	Setup » Compositional Template... Setup » Compositional... (the network default)	Setup » PVT File...	Setup » MFL File...
Black Oil Fluid	Yes			
Compositional Fluid		Yes	Yes — if the file contains compositional data	Yes
Table Fluid			Yes — if the file contains property tables	

Table fluids are not suitable for mixing.

PVT files that contain compositional information can also be imported into PIPESIM and used as Compositional data.

Precedence of Fluids when Multiple Fluids Defined

In a network, fluids can be defined at the

- network level ([Setup » Black Oil](#))
- well or source level ([Setup » Fluid Models](#) or right-click the well or source and select Fluid Models)
- equipment level

The following table shows which value PIPESIM uses when fluids are defined at the different levels:

Network Level	Well or Source Level	Equipment Level	Fluid Definition Used
Defined	Defined	Defined	Equipment Level
Defined	Defined	Not defined	Well or Source Level
Defined	Not defined	Defined	Equipment Level

Network Level	Well or Source Level	Equipment Level	Fluid Definition Used
Defined	Not defined	Not defined	Network Level

Note:

- Injection points and gas lift injection points always use locally defined fluids.
- Multi-layered wells always use the default setting, unless a local fluid model is defined at the completion level. (The fluid model cannot be defined at the network level.)

Local Fluid Data

There are two types of branches in PIPESIM: well branches and normal branches. Well branches in a network use the global fluid by default (except for multi-layer). A normal branch uses the default unless a local fluid is defined. The definition of a local fluid for a normal branch means that the branch uses the local fluid.

Fluid data must be defined for each well, source or injection point in a simulation model. The fluid type (black oil, compositional, or table fluid) must be the same as the [default fluid \(p. 0 \)](#) type for the simulation model. A local fluid can be set to be the same as the default fluid, or can be locally defined.

To edit the local fluid:

1. Right-click on the source/completion icon.
2. In the menu displayed, select the **Fluid Model** option.
3. Set **Use locally defined fluid model** and click **Edit** to set the fluid properties.

Alternatively, in a network model, select **Setup » Fluid Models....** To edit a fluid, double-click its row.

Local overrides

For black oil or compositional fluids, once a local fluid has been defined, the surface phase ratios can be changed without editing the fluid properties

To set the phase ratios:

1. Right-click on the source / completion icon.
2. In the menu displayed, select the **Fluid Model** option and select **Override fluid model parameters**.
3. Enter the data.

Alternatively, in network models, select **Setup » Fluid Models....**

This allows access to the watercut and GOR/GLR/OGR/LGR entry fields. Data entered here overrides the local or default fluid data. This is especially useful for multi-zone wells or networked wells that have the same [default] fluid properties but different watercut, and so on. For compositional fluids, the composition will be modified to give the correct phase ratios, in a similar manner to GLR matching.

1.4.4 Ensuring consistency among multiple fluid files in a PIPESIM network model

To ensure reasonable simulation results for network models using multiple fluid files (for example, MFL files), it is important to maintain consistency in the fluid characterization in the various fluid files used.

Here are a few guidelines to follow when using multiple fluid files in a single PIPESIM network model:

- All fluid files should have the same template of components

Note: Some components may be set to have zero number of moles in the different fluids, but the component set should be the same across all fluids.

- All fluid files should be characterized to the same number of pseudo-components and use the same correlations and methods to estimate the properties of the pseudo-components (for example, critical properties, acentric factors, omegas, etc.).
- All fluid files should be defined with the same Binary Interaction Parameter (BIP) set.

Note:

- When only one MFL file is mapped in the PIPESIM workspace, all of the information in the MFL file will be honored by PIPESIM in the simulation. This includes the Equation of State, Models for Viscosity, Thermal Conductivity, Surface Tension, BIP sets, etc. including any tuning done to the fluid.
- PIPESIM can currently use tuned data in only one MFL fluid file in the workspace. If you tune the models (EOS, Viscosity, etc.) to match experimental data, e.g. viscosity, density, etc., in the Multiflash interface, it is strongly recommended that you use only one MFL file (the one with the tuned data) in the workspace. If you use multiple MFL files with tuned data in the PIPESIM workspace, the tuned data in only one of the MFL files will be used in the PIPESIM simulation run.

1.4.5 Black Oil Fluid Modeling

Introduction

Fluid properties can be predicted by blackoil correlations, which have been developed by correlating gas/oil ratios for live crudes with various properties, such as oil and gas gravities. The selected correlation is used to predict the quantity of gas dissolved in the oil at a particular pressure and temperature.

The black oil correlations have been developed specifically for crude oil/gas/water systems and are therefore most useful in predicting the phase behavior of crude oil well streams. When used in conjunction with the calibration options, the black oil correlations can produce accurate phase behavior data from a minimum of input data. They are particularly convenient in gas lift studies where the effects of varying GLR and water cut are under investigation. However, if the accurate

phase behavior prediction of light hydrocarbon systems is important, it is recommended that the more rigorous compositional model is employed.

The Black Oil fluid description can be used for the following fluid types:

- Water
- Dry Gas
- Condensate (not recommended)
- Volatile Oil (not recommended)

Black Oil Data Sets

A name and optional description/comment must be entered for each black oil data set. A data set consists on all the black oil properties including viscosity, coning data, calibration data, and so on.

PIPESIM also allows black oil data sets to be stored [externally \(p.171\)](#) [to the model in a database], if required, so that it can be shared between sources in a single model, different models and different users or groups, etc.

Note: The external database base is used only as a data storage system to allow data to be transferred. It should be regarded as allows the black oil data set to be imported and exported. The actual (black oil) data for a source is stored locally with the source. For example the database could be located on a LAN and the model on a PC. When the PC is disconnected from the LAN the model will still run as the black oil data for the sources is stored within the model. Thus, if the database set is changed the local source data is not affected.

Exporting a data set

To export a data set to the database:

1. Select **Setup > Black Oil**.
2. In the **Black Oil properties** dialog, set the required data, include and viscosity or calibration data.
3. Provide the data set with a name and description.
4. Click the **Export** button. A confirmation dialog appears.

The data set can now be shared.

Importing a data set

To import a data set, for a source, from the database for a Network model:

1. Select the source (or sources). To make multiple selections, hold down the Shift key while selecting the objects.
2. Right-click and select the **Fluid Model** option from the menu displayed.
3. Deselect the **Use Default Black Oil set** option.
4. Click **Import**.

5. Select the name of the fluid to import, take a copy of, the data from the database to the local source. It **does not** point to the data set in the database. That is, if the data in the database is changed, the black oil data for the selected source is **not** changed.

Black Oil Options

To access these options, select **Setup » Black Oil**.

[Keyword Reference. \(p.728\)](#)

Stock Tank Properties

On the **Black Oil Properties** tab, enter the data at [stock tank conditions \(p.166\)](#) for the following parameters:

Watercut

The % of the liquid phase that is water (default 0%)

GOR/GLR

Gas oil/liquid ratio for oil systems (default not set).

OGR/LGR

Oil/liquid ratio for gas systems (default not set).

Gas S.G.

Gas specific gravity (default 0.64)

Water S.G

Water specific gravity (default 1.02)

API/DOD

API/Dead oil density (default 30 API and 54.68 lb/ft³)

See also [Database Sets \(p.135\)](#), [Calibration Data at Bubble Point \(p.137\)](#), [Solution Gas Correlations \(p.508\)](#), [Coning \(p.513\)](#), [Viscosity data \(p.515\)](#), [Advanced Calibration Data \(p.138\)](#), [Contaminates \(p.140\)](#) and [Thermal Data \(p.140\)](#).

Calibrate a Fluid to match lab data

The method to use to model (and calibrate) the fluid will depend upon the fluid type (oil or gas) and the data available. Most oil systems are modeled using the black oil correlations whereas gas-condensate and gas systems are normally modeled using the more rigorous [compositional \(p.141\)](#) model.

Black Oil Calibration

To calibrate the black oil defined fluids, perform the following basic steps:

1. Select a [units \(p.170\)](#) set.
2. Enter the basic fluid data.
3. Enter the Bubble Point data.
4. Either OFVF, density or Compressibility above the bubble point.

5. Either OFVF or density and Live oil viscosity, Gas viscosity and Gas Z factor at or below the bubble point.
6. (Optional) Enter the Advanced calibration data.
7. Run the operation.
8. [Save the model \(p.35\)](#)

In a network model the calibration data is "mixed" at junctions to provide average calibration data for the resulting stream.

Calibration on Black Oil data is limited to 1 data point per property. After entering the basic Black oil data (API, GOR, viscosity, and so on) by using **Setup » Black oil**, enter the following:

- Bubble point data (required for calibration).
- One of the following:
 - Either OFVF, density or Compressibility above the bubble point.
 - Either OFVF, density and Live oil viscosity, Gas viscosity and Gas Z factor at or below the bubble point.

A [calibration factor \(p.727\)](#) is then computed for each property. The effect of the measured data can be further examined by plotting the PVT data.

Correlations

See the [Technical Description \(p.506\)](#).

Bubble point calibration data

To access these options, select **Setup » Black oil**.

Note: If you want the following options to appear on the **Black Oil Properties** tab in the dialog, set the **Calibration** option, under the **Advanced Calibration data** tab to **No Calibration**.

The oil saturated gas content at a known temperature and pressure (for example at reservoir conditions) should be entered to allow calibration of the black oil model. Such calibration will significantly improve the accuracy of the predicted gas/liquid ratios. If the calibration data is omitted the program will calibrate the correlation on the basis of oil and gas gravity alone and there will be a consequent loss in accuracy.

Sat. Gas

Quantity of gas which will dissolve in the oil, and saturate it, at a given pressure and temperature

Pressure

Pressure at which the above applies

Temperature

Temperature at which the above applies

Advanced Calibration Data

Select **Setup** » **Black oil** and select the **Advanced Calibration Data** tab. The following options are available:

No Calibration

No (advanced) calibration data is available for the fluid. The bubble point information can however be entered.

Single Point Calibration

In many cases, actual measured values for some properties show a slight variance when compared with the value calculated by the blackoil model. In this situation it is useful to "calibrate" the property using the measured point. PIPESIM can use the known data for the property to calculate a "calibration constant" K_c :

$K_c = \text{Measured Property } @(\text{P},\text{T}) / \text{Calculated Property } @(\text{P},\text{T})$

This calibration constant is then used to modify all subsequent calculations of the property in question, that is: Calibrated value = K_c (Predicted value)

If you select **Single Point Calibration** the following properties can be calibrated:

Above Bubble Point

Select the property in the list and enter data for it. The options are:

- [Oil Formation Volume Factor \(OFVF\) \(p.505\)](#) . This is the ratio of the liquid volume at stock-tank conditions to that at reservoir conditions.
- Density
- Compress. — the [Gas Z \(compressibility\) \(p.522\)](#)

At Bubble point

Enter the oil saturated gas content at a known temperature and pressure (for example at reservoir conditions) to allow calibration of the black oil model. The required [correlation \(p.505\)](#) can also be selected.

Sat. Gas

Quantity of gas which will dissolve in the oil, and saturate it, at a given pressure and temperature.

At or Below the Bubble point

Select the property in the list and enter data for it. The required [correlation \(p.505\)](#) can also be selected. The options are:

- [Oil Formation Volume Factor \(OFVF\) \(p.505\)](#) — The ratio of the liquid volume at stock-tank conditions to that at reservoir conditions.
- [Live oil viscosity \(p.518\)](#)
- [Gas viscosity \(p.525\)](#)
- [Gas Z \(compressibility\) \(p.522\)](#)

Plot PVT Data

These two buttons produce plots of properties as a function of pressure versus temperature:

@ Laboratory conditions - GOR = GSAT

sets the GOR to that defined by the **Saturation Gas** value under the Bubble Point data section on the main **Black Oil** dialog.

@ Reservoir conditions

uses the GOR as defined by the GOR value on the main **Black Oil** dialog.

Generate tables

If you select this check box, the output file (*.out) will contain tables of the properties as a function of pressure and temperature.

Multi Point Calibration

The black oil correlations will be tuned so that the correlation goes through ALL the data points.
Note: This is not a best fit method - all points will be fitted exactly. Any outlying data MUST be smoothed before entering into PIPESIM.

Temperature

The temperature at which the properties apply. All other data is assumed to be at this temperature. The blackoil correlations will then be used to calculate fluid properties at different temperatures.

Below the Bubble point

Enter the property at various pressures. The required correlation (p.505) can also be selected.

1. [Oil Formation Volume Factor \(OFVF\) \(p.137\)](#). The ratio of the liquid volume at stock-tank conditions to that at reservoir conditions.
2. [Solution Gas \(p.137\)](#) Quantity of gas which will dissolve in the oil, and saturate it, at a given pressure
3. Density
4. [Live oil viscosity \(p.518\)](#)
5. [Gas Z \(compressibility\) \(p.522\)](#)
6. [Gas viscosity \(p.525\)](#)

At Bubble Point

1. [Oil Formation Volume Factor \(OFVF\) \(p.137\)](#). The ratio of the liquid volume at stock-tank conditions to that at reservoir conditions.
2. [Solution Gas \(p.137\)](#) Quantity of gas which will dissolve in the oil, and saturate it, at a given pressure
3. Density
4. [Live oil viscosity \(p.518\)](#)
5. [Gas Z \(compressibility\) \(p.522\)](#)
6. [Gas viscosity \(p.525\)](#)

Above Bubble Point

Select Density or [Oil formation volume factor \(OFVF\) \(p.137\)](#) (the ratio of the liquid volume at stock-tank conditions to that at reservoir conditions), then enter the following properties at various pressures:

- [Oil formation volume factor \(OFVF\) \(p.505\)](#) . This is the ratio of the liquid volume at stock-tank conditions to that at reservoir conditions.
- Density
- [Live oil viscosity \(p.518\)](#)
- [Gas Z \(compressibility\) \(p.522\)](#)
- [Gas viscosity \(p.525\)](#)

In the [Correlation \(p.137\)](#) list, select the correlation to use for each property.

Plot PVT Data

This produces a plot of properties as a function of pressure versus temperature.

Plot Calibration Data Points Only

Displays a plot of the calibration data.

@ Laboratory Conditions- GOR = GSAT.

This sets the GOR to that as defined by the **Saturation Gas** value under the **Bubble Point data** section on the main **Black Oil** dialog.

@ Reservoir Conditions

Uses the GOR as defined under the GOR value on the main **Black Oil** dialog.

Generate tables

The output file (*.out) will contain tables of the properties as a function of pressure and temperature.

[More details \(p.727\).](#)

Gas Phase Contaminants

This allows gas phase contaminants (CO₂, H₂S, N₂, H₂ and CO) to be tracked through the system. Enter the mole fraction of each contaminant (<1) of the gas phase at stock-tank conditions. The default for each contaminant is 0.

Contaminants are used to improve the accuracy of the compressibility (z) factor and may be used for corrosion calculations and optimization constraints. For more information, see [deWaard Corrison Model \(p.390\)](#).

Black Oil Thermal Data

To set up Black Oil thermal data, do the following:

1. Select **Setup > Black Oil....**
2. On the **Thermal Data** tab, enter the following thermal properties for the three phases (gas, oil and water) of the black oil fluid:

- Specific Heat Capacity
 - Thermal Conductivity
3. Select the [Black Oil Enthalpy Calculation method \(p.526\)](#) to use. Currently, there are two options:
- 1983 Method — original method implemented in PIPESIM.
 - 2009 Method — for this option, you are asked to supply a **Specific Latent Heat of Vaporisation**.

1.4.6 Compositional Fluid Modeling

PIPESIM allows full compositional modeling of the fluid by accessing a PVT modeling package. This uses a compositional interface as an alternative to the [blackoil model \(p.505\)](#).

Compositional fluid modeling is generally regarded as more accurate, but also more expensive in terms of runtime than black oil fluid modeling. It is justified for problems involving volatile fluids which need rigorous heat transfer calculations and more accurate phase fractions.

Compositional Template

Multiple fluids can be used in both network and single branch simulations. For example, in a network model each source or well can have a separate fluid. In a single branch model, an injection fluid can be different from the fluid in the branch. At the same time, PIPESIM allows users to select a variety of fluid packages, components, thermodynamic and transport models, to describe compositional fluids. However, it is important that there is some consistency between all the fluids used in a model, to ensure they can be mixed. The Compositional Template imposes a consistent global set of properties on all fluids in the model.

Creating a compositional fluid is therefore a two stage process, as follows:

1. Define (or update) the compositional template by selecting **Setup » Compositional Template**.
2. [Specify the composition \(p.148\)](#).

Creating a compositional template

To build a compositional model, do the following:

1. Select the [compositional package \(p.142\)](#).
2. Select [library components \(p.142\)](#).
3. (Optional) Define any [petroleum fractions \(p.143\)](#).
4. Select the [compositional property models \(p.145\)](#).

Alternatively, [import \(p.142\)](#) a template from an existing PVT file. The template ensures the following:

- the same package is used for all fluids
- the total number of components used by all fluids is less than the limit for the package
- the thermodynamic and transport models are consistent for all fluids

Updating the compositional template

When adding a new fluid to the model, you must first check that it is compatible with the template. If the fluid contains components that are not already in the template, they must be added to it, as follows:

1. Select [library components \(p.142\)](#).
2. Define any [petroleum fractions \(p.143\)](#).

Any new component added to the template will be added to all the existing fluids, but with zero moles.

Importing a compositional template

To import a template from an existing PVT file, click the **Import** button. There are three options when importing a template:

- Imports only Component List, Petroleum Fractions and Property Models. This option removes any components already in the template and changes all template [compositional property models \(p.145\)](#) (EoS, Viscosity etc.)
- Merge with existing template Component List and Petroleum Fraction. This option adds any new components in the file to the template. It does not remove any existing components from the template, and it does not affect the template [compositional property models \(p.145\)](#).
- Imports only Property Models information. This option only changes the template [compositional property models \(p.145\)](#) (Equations of State, viscosity, and so on.)

Note: If you only import the property information and the property is not selected in the dialog box, the property for the item that is selected is altered.

Phase Behavior Package

PIPESIM currently has access to the following compositional packages:

- E300 (from Schlumberger)
- DBR (from Schlumberger)
- REFPROP v8.0 (from The National Institute of Standards and Technology, United States)
- GERG-2008 (from Ruhr-Universitat Bochum)
- Multiflash (from InfoChem)

The E300 flash package is the same PVT package that is used by Schlumberger Reservoir Simulation products such as ECLIPSE Compositional, PVTi, VFPI and so on.

See also: [Viscosity \(p.515\)](#), [Binary Interaction Parameters \(p.147\)](#), [Emulsions \(p.146\)](#), [Equation of State \(p.536\)](#), [Technical Description \(p.141\)](#)

Selecting Components

Three types of component are available: aqueous, library components and [petroleum fractions \(p.143\)](#). However, note that the maximum number of components allowed is as follows:

- **50** for Multiflash

- **50** for E300 flash
- **50** for DBR flash
- **20** for REFPROP and GERG-2008 flashes. This excludes water, which can be added to make 21 in total

To change components, select **Setup » Compositional Template** and then select the **Component Selection** tab.

To select the library components, click on each required component in the list, then on **Add>>**. This adds the selected components into one of the **Selected Components** lists on the right hand side of the window. All but the [aqueous \(p.548\)](#) components will appear in the **Hydrocarbons** list. (Aqueous components are displayed in blue.)

To define a [Petroleum fraction \(p.143\)](#), specify its name and relevant characterization parameters. Once the petroleum fraction data has been entered, it MUST be added to the component list. To do this, click **Add to Composition>>**.

To remove an unwanted component from the spreadsheet lists, do the following:

1. Click on the row number to the left of the component name in the spreadsheet. This highlights the whole row.
2. Click **<<Delete**.

Petroleum Fractions

Oil systems contain in reality many thousands of pure components, consisting of a spectrum of molecules with different carbon numbers and exponentially increasing numbers of different isomers of each. It would be impossible to model the behavior of such systems by explicitly defining the amount of each of these molecules, both because of the excessive computing power needed and the fact that laboratory reports could not possibly supply all this information. Luckily, since the alkane hydrocarbons are non-polar and therefore mutually relatively ideal, lumping them together in the form of a number of 'pseudo-components' results in fairly accurate phase behavior and physical property predictions.

Petroleum fractions are normally defined by splitting off sections of a laboratory distillation of the C7+ mixture. Curves of boiling point, density and molecular weight are produced from which the properties of the individual pseudo-components may be derived. The default characterization parameters are boiling point (BP), the molecular weight (MW) and the specific gravity (SG). The maximum value allowed for the specific gravity in PIPESIM is 1.3.

There is an alternate specification for users, with additional information in the form of critical temperature (TC), critical pressure (PC) and an acentric factor.

Petroleum Fraction data entry

Select **Setup » Compositional** and, on the **Petroleum Fractions** tab, enter the following information:

Note: You can enter a maximum of **50** Petroleum fractions. Some properties may not be applicable to a PVT package.

Name

A name to describe the petroleum fraction, for example C7+.

Property data**For Multiflash**

The minimum data that must be entered are either

- M.W. and S.G.
- M.W. and B.P.
- B.P. and S.G.
- T.C., P.C. and Acentric Factor

For E300 and DBR

The minimum data that must be entered is

- M.W.

For REPROP and GERG-2008

Petroleum fractions cannot be entered

Input and Calculated Data

The input and calculation availability of all property data is summarized in the following table:

- I = Input only
- C = Calculated (These read-only properties are disabled for input and highlighted in grey. The calculated values display if button “Calculate properties” is performed.)
- N = Non-applicable (highlighted in grey)
- I, C = Calculated if not input

	Multiflash	DBR	Eclipse 300
Name	I	I	I
B.P.	I, C	C	C
M.W.	I, C	I	I
S.G.	I, C	I, C	C
T.C.	I, C	I, C	I, C
P.C.	I, C	I, C	I, C
Liq. Visc.	N	C	I, C
V.C.	N	C	I, C
Ac. factor	I	I, C	I, C
Volume Shift	N	C	I, C
Ref. Dens.	N	C	C

Ref. Temp.	N	C	C
Watson K	N	C	C
Omega A	N	I, C	I, C
Omega B	N	I, C	I, C

Add to composition >>

When the petroleum fraction is added it is initially displayed in red. This shows that not enough data has been entered. When sufficient data has been added, the row reverts to black. After defining the petroleum fraction, you MUST add it to the main component list by selecting the row and clicking **Add to Compositions>>**. Petroleum fractions that have been added to the main component slate are shown in green. Next set its composition by using the **Component Selection** tab.

Delete

Click this to delete the selected row(s).

Calculate properties

Once the minimum data has been entered then all properties not entered and marked as "C" in the table above can be computed (if required) using the **Calculate properties** button. However, before this can be done the petroleum fractions MUST be added to the main component list and its composition set. That is, the petroleum fraction must be displayed in green and have a molar rate defined.

Notes:

- This option is only available for the Multiflash, DBR, and Eclipse 300 PVT packages.
- When you import a file and calculate petroleum fraction properties, the properties are updated and the petroleum fractions are changed. If you import the original file again, a new set of the original petroleum fractions are imported. The original petroleum fractions are different than the calculated ones.

Recalculate properties

Before recalculating some properties, you MUST remove the values to be computed from the spreadsheet. This can be easily done by selecting a group of cells and using Ctrl-X.

Compositional Property Models

Compositional thermodynamic and transport properties are determined by using the following models and parameters:

- [Equation of State models \(p.536\)](#) and [Binary interaction parameter \(p.147\)](#) used to calculate the following:
 - fugacity, to determine the phases present and the composition of the phases
 - density
 - enthalpy

- entropy
- [Viscosity models \(p.515\)](#)
- Surface Tension models
- Thermal Conductivity models

Model scope

Use of the same set of models and parameters for all fluids in a network or single branch simulation is recommended, as this allows the fluids to be mixed correctly. For back-compatibility, you can define a different model for each fluid. This facility is also useful if a network is made up of several disjoint sub-networks, where the fluids may be characterized differently. If different property models are used for different fluids, results may be unpredictable when these fluids mix. PIPESIM chooses the property models corresponding to the fluid with the most mass to model the mixture fluid.

To set the scope of the compositional property models, select **Setup** » **Compositional Template** and use the **Property Models** tab.

If the scope is set to global, the template property models are used for all fluids. If the scope is set to local, individual property models can be set for each fluid in the **Compositional Template Properties** dialog.

Emulsion Viscosities

Three mixing rules have been implemented for the liquid viscosity. These are only valid for the Multiflash PVT package.

Select one of the following options :

- [Set to oil viscosity \(Inversion method\) \(p.146\)](#)
- [Volume ratio \(p.146\)](#)
- [Woelflin \(p.146\)](#)
- [None \(p.147\)](#)

Set to oil viscosity (Inversion method)

1. Use Oil viscosity at water cut less than or equal to CUTOFF
2. Use Water viscosity at water cut greater than or equal to CUTOFF

Volume ratio of oil and water viscosities

$$\mu_L = \mu_o \left(1 - \frac{wcum}{100}\right) + \mu_w \times \frac{wcum}{100} \quad \text{Eq. 1.1}$$

Woelflin

1. Use Woelflin correlation at watercut less than or equal to CUTOFF. The correlation is defined as follows:

$$\mu_L = \mu_o \left(1.0 + 0.00123 \times wcut^{2.2}\right) \quad \text{Eq. 1.2}$$

2. Use Water viscosity at watercut greater than CUTOFF.

None

This option calculates the liquid viscosity from the oil and water viscosities, using the Kendal & Monroe equation.

How to set the cutoff

You can define the CUTOFF, but it is typically set to 60% (default).

Cutoff can be set for

- Black Oil — Use **Setup » Black Oil**. On the **Viscosity Data** tab, enter the **Watercut Cutoff Method** in the **User Specified** field.
- Compositional — Use **Setup » Compositional**. For the name, enter WCut. For the comment, enter *EMULSION and the percent. Enter the word EMULSION in all uppercase. Do not enter the percent sign. For example, “*EMULSION 60” means a 60% water cut.

See also: [Package \(p.142\)](#), [Viscosity \(p.515\)](#), [Binary Interaction Parameters \(p.147\)](#), [Equation of State \(p.536\)](#)

Binary Interaction Parameter(BIP) Set

Select the Binary Interaction Parameters to use: Select or choose

- OilGas1 (only available for Multiflash)
- OilGas2 (only available for Multiflash)
- OilGas3 (only available for Multiflash)
- OilGas4 (only available for Multiflash)
- E300 Flash default (only available for E300)
- DBR Flash default (only available for DBR)
- REFPROP default (only available for REFPROP)
- GERG-2008 default (only available for GERG-2008)
- [User BIPs \(p.148\)](#).

Note: User BIPs are not available under REFPROP or GERG-2008.

Binary interaction parameters (BIPs) are adjustable factors, which are used to alter the predictions from a model until the predictions match experimental data as closely as possible. BIPs are usually generated by fitting experimental VLE or LLE data to the model in question. BIPs apply between pairs of components, although the fitting procedure may be based on both binary and multi-component phase equilibrium information.

BIP Set	Multiflash	E300	DBR	REFPROP	GERG-2008
OilGas1	x				
OilGas2	x				
OilGas3	x				
OilGas4	x				

E300 Flash Default		x			
DBR Flash Default			x		
NIST REFPROP default				x	
GERG-2008 default					x

For Multiflash

The interaction parameters in OILGAS1 were from the correlation recommended by Nishumi et al. They correlated parameters between all the light gases and hydrocarbons up to Decane. However, when these were used in the prediction of hydrate dissociation temperatures for systems containing a significant amount of condensate or crude, it became apparent that neither the bubble or dew points nor the hydrate dissociation temperatures were being predicted accurately.

After further investigation new correlations contained in OILGAS2 based on BIPs recommended by Whitson for Methane with heavy hydrocarbons were introduced. The new correlations provide heavy hydrocarbon interaction parameters for Ethane to Pentane, all of which were previously set to zero. For systems containing Methane and alkanes up to C10 there is no major difference between OILGAS1 and OILGAS2 BIP sets, but after C10 OILGAS1 parameters decrease rapidly in value, reaching negative values after C16. In contrast, the values of OILGAS2 parameters continue to increase gently with increasing carbon number.

OILGAS3 and OILGAS4 were later introduced and the only difference is in the parameters for MEG/alkanes for the RKSA-info (hydrate) model. The newer parameters predict a lower (correct) solubility for MEG in heavy hydrocarbons but are less accurate for the solubility of hydrocarbons in MEG. We would generally recommend the use of the latest parameters, OILGAS4.

How to...: **Setup** » **Compositional** » **Options**

See also: [Package \(p.142\)](#), [Viscosity \(p.515\)](#), [user-defined BIPs \(p.148\)](#), [Emulsions \(p.146\)](#), [Equations of State \(p.536\)](#)

User Defined BIPs

If you have interaction parameters available and wish to supplement or overwrite those used, you can do this by selecting the **User BIPs** option.

1. After selecting the components, select **User BIPs** as the **BIP Set**
2. and then **Load BIPs**.

This will load the default [BIP \(p.147\)](#) set for all the components that have been defined in the main component list. These values can then be modified accordingly. The modified values will then be used providing that the **BIP Set** is set to **User BIPs**.

Note: You must make sure that the BIPs you supply conform to the model definition used. Not available under REFPROP, or GERM-2008.

Specifying the Composition

Once the component list has been selected and added in the [Compositional Template \(p.141\)](#), you can enter the desired composition for each component.

The **Aqueous** list contains the aqueous components. Typically their input units are per volume of stock-tank gas (for gas systems) or per volume of stock-tank liquid (for oil systems). In order to use the [watercut matching \(p.152\)](#) option, the water MUST be expressed on a mole basis.

The **Hydrocarbons** list contains the oil and gas components, whose composition is normally specified in mole percent (water-free). As the mole rates are added the total number of moles is computed and displayed automatically.

Normalize

After entering the composition, you can normalize the total number of mole (to 1.0) by clicking **Normalize**. If the aqueous components have been entered on a volume or mass basis, they are first converted to a Mole basis and then normalized. Once a component set has been normalized, it cannot be reversed.

Importing the Composition

A composition can be imported from the following file types: PVT, PTT, and PVO. The data in the imported file overwrites the existing fluid. Any component in the imported file that is not in the [Compositional Template \(p.141\)](#) is added to it. The thermodynamic or transport models in the imported file are not imported.

Note: When importing a PVO file, select the units in which the file was written. The default is metric.

Phase Envelope

To determine a fluid phase envelope, do the following:

1. Select **Setup** » **Compositional**.
2. In the **Compositional** tab, enter the components, including any petroleum fractions and their composition.
3. In the **Compositional** tab, click **Phase Envelope**.
4. The phase envelope is displayed (automatically) if appropriate, showing the following:
 - Dew line
 - Bubble line
 - Critical point
 - [Hydrate lines \(p.552\)](#) — Multiflash only
 - [Ice line \(p.553\)](#) — Multiflash only
 - Water or aqueous line — Multiflash only

For the hydrocarbon lines, which include the dew and bubble lines, PIPESIM uses a two-phase stream to perform the phase envelope. In other words, if a given compositional fluid contains aqueous components (water, methanol, and so on), PIPESIM removes all those aqueous components from the original fluid and only uses the remaining subset fluid to perform and display the hydrocarbon lines. However, when Multiflash MFL files are used to perform the phase envelope, the aqueous components are not removed from the stream. Hence, for the same compositional fluid the hydrocarbon lines may be different depending whether the MFL file is used or not.

In addition to the above phase plot lines, when using the MFL files, the phase envelope can also display (if appropriate).

- [Wax line \(p.553\)](#)
- [Asphaltene locus \(p.551\)](#)

For phase envelopes generated using the Multiflash PVT package, the maximum pressure plotted is initially set to 300 bars for the hydrocarbon lines and 400 bars for the water and solids (hydrates, ice, and wax) lines. These values will, if necessary, automatically increase up to 1000 bars for the hydrocarbon lines and 1200 bars for the water and solids lines.

The [quality lines \(p.150\)](#) can also be added.

The phase envelope can also be superimposed over the flowline temperature or pressure profile. [See how \(p.261\)](#).

Comment

You can add a comment line to the fluid definition.

For a phase envelope created with Multiflash, the extent of the pressure (Y) axis can be set by using the following keyword placed in the comment line:

```
.....  
**penv_pmult=n  
.....
```

where n is a factor. The pressure axis is then plotted to a value of n times the phase envelope cricondenbar value.

Search

This allows the component database to be searched and sorted. To locate a specific component, enter either its name or formula.

Show Quality lines on a Phase Envelope

To show quality lines on a phase envelope, do the following:

1. Define your composition using **Setup** » **Compositional** and the **Compositional** tab. Include any Petroleum Fractions.
2. Select the **Phase Envelope** tab.
3. Enter the quality lines that you want to view.
4. Click **Phase Envelope**.

Quality Lines

To add quality lines to phase envelope plots:

1. Select **Setup** » **Compositional** » **Phase Envelope**.
2. Go to the **Phase Envelope** tab and specify the quality line percentages that you want to see on the phase envelope.
3. Press the **Phase Envelope** button.

Note:

1. Selecting the **Phase Envelope** button from the **Component Selection** tab will NOT produce quality lines.
2. The behavior of this option is different for the Multiflash package.
 - For Multiflash, two lines are plotted for each percentage value entered, one corresponding to the gas quality and the other to the liquid quality. For example, in a Multiflash case, if the 10% line is selected, a 10% gas quality line and a 10% liquid (90% gas) quality line are plotted together. For Multiflash, the first Quality Line Percentage should always be 0.00%, which will plot both the bubble line (0% gas) and the dew line (0% liquid / 100% gas).
 - For all the other flash packages, bubble and dew lines are always plotted, but only the gas quality lines are plotted.

Therefore, to plot quality lines at 10% intervals, for Multiflash select (0,10,20,30,40,50), for all other packages select (10,20,30,40,50,60,70,80,90).

Flash/Separation

Flash/separation allows a single point flash of the current composition to be performed, or a multi stage separation.

Do the following:

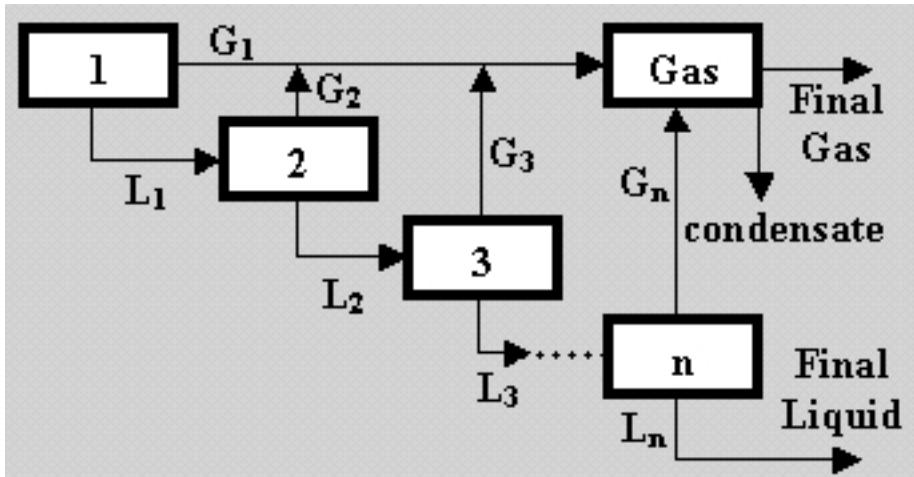
1. Select **Setup** » **Compositional** » **Flash Separation**.
2. Select the **Flash type** and enter the (PT) conditions, press the **Perform Flash** button. The results are displayed in a spreadsheet that can be copied (CTRL-C) and pasted (CTRL-V) to other applications.

Multi-Stage Separation

Multi stage separation allows the input of pressure and temperature conditions at multi stage liquid-vapor separation levels with a final gas separation stage used to flush out any remaining liquid in the gas stream. Each separator operates at 100% efficiency, that is the two exit streams from each stage are pure liquid and vapor.

1. Input the conditions (pressure and temperature) for each of the stages as well as for the final gas separation stage. By clicking on **Perform Flash**, the component mole fraction (in percentages) and fluid properties of the specified liquid stage are displayed. To get the properties of the fluid at the final exit streams (liquid and gas), enter a liquid stage of 0 before clicking on Perform Flash or enter a liquid stage of 1 to n, click on Perform flash then click on Final stage. After a flash has been performed, you can also get a specific stage (gas, liquid or final) by clicking on the appropriate button as described below.
 - By entering the liquid stage number (1, 2,n) and clicking on **Liquid stage**, the mole fraction and fluid properties for the liquid stage exit stream are displayed. Note that a liquid stage of 0 is not valid at this point and is allowed only when performing the flash to indicate the final stage.
 - By clicking on **Gas stage**, the mole fraction and fluid properties for the gas stage exit stream are displayed.

- Clicking on **Final stage** displays the mole fraction and fluid properties for final gas and final liquid exit streams.



GLR Matching

Tune the composition to match a specified water cut and gas/oil ratio.

- Select **Setup** » **Compositional** » **GLR**.
- Enter all of the following:
 - Water cut (percentage of the liquid phase that is water) of the fluid. Water MUST be included in the component list and expressed in a mole basis if this is set to anything other than 0%.
 - GLR/GOR/LGR/OGR of the fluid.
 - Pressure and Temperature at which the above conditions apply. Default is stock-tank conditions.
- Press **Perform Calculation** to compute the molar flow rates of the composition to match those conditions if this is possible. If the conditions cannot be met then an error message is displayed. This is displayed in the calculated column.
- To update the main composition, press the **Update Composition list** button after the new composition has been computed.

Salinity (Salt) Analysis

One of the components in the database is a "salt component". This is used to represent a salt pseudo-component, based on sodium chloride equivalence, for use in calculating freezing point depression or hydrate inhibition. Define the salinity content of the water. To carry out such calculations you need to provide the amount of salt in the stream. Often the data for the salt, brine or formation/production water ion analysis will not be available. The Salt calculator can assist.

The water content of the stream MUST be defined. The Salt component can either be added from the component list or will be automatically added when the salt quantity has been computed.

To define the salt, select **Setup** » **Compositional** » **Salinity Analysis**. The salt can be defined in the following ways:

- Ion Analysis

- a. Density (SG) of solution
- Salt Analysis table
- Total amount of dissolved solid (TDS)
 - a. Density (SG) of solution
 - b. Total dissolved solid

After entering the required data, click the **Calculate** button. This computes the salt quantity and adds that amount to the component list.

Salt quantity is always in mole units.

Compositional using MFL file

Some considerations exist for mixing MFL files. For instance, multiple MFL Files may be used in a single model. However, care must be taken to ensure that the methods and components are consistent across all MFL files. The standard PIPESIM compositional user interface enforces this consistent use through the Compositional Template definition. When using MFL files, you must ensure that all fluid models are consistent.

Two examples of MFL usage include the following:

- A reservoir composition may include many components and be mixed with an MFL file used for gas lift.
- The gas lift MFL file must include all of the same components as the MFL file for the reservoir fluid, even if many of them (e.g., all except methane) are defined as zero.

To use an MFL file, select **Setup** » **MFL file**. This opens a dialog with the following fields and buttons:

Select MFL File

Should either contain the full path and name of a previously created MFL file or be left blank if an MFL file is intended to be created by selecting **Create New**.

Browse

Locate and select a previously created MFL file.

Create New

Allows you to define this fluid by using InfoChem's MultiFlash GUI. The MultiFlash GUI will be launched. From this location, the component list, component quantities, equation of state, transport property, wax, asphaltene, hydrate, or combined solid models can be selected.

Edit

Enables you to edit an MFL file that has been selected and is displayed in the field above. The MultiFlash GUI is launched and displays the component and model selection in the MFL file to be edited. These elements can be modified as required.

Phase Envelope

Plots the phase envelope of the fluid selected in the File field. After an MFL file has been created, edited, or selected after browsing to it, the phase envelope can be plotted. The hydrocarbon phase envelope and any other phase selected in the MFL file, will be plotted.

Availability of Multiflash models in PIPESIM using the MFL file fluid option

Multiflash is a 3rd party flash package that enables full phase thermodynamic modeling of multiphase fluids and solids using standard and state-of-the-art models. Multiflash incorporates an extensive suite of equations of state for advanced flashes and viscosity, interfacial tension and thermal conductivity models for the prediction of transport properties. Multiflash enables flashes that can result in up to 7 separate phases simultaneously including gas, liquid, water, ice, hydrates, wax and asphaltene.

Multiple MFL files can be defined in one PIPESIM model and mapped to different sources and wells in the Fluid Manager, however care must be taken to ensure that the models and components are consistent across all MFL files. Refer to the section [Ensuring consistency among multiple fluid files in a PIPESIM network model \(p.134\)](#) for more details.

Note:

- When only one MFL file is mapped in the PIPESIM model, all of the information in the MFL file will be honored by PIPESIM in the simulation. This includes the Equation of State, Models for Viscosity, Thermal Conductivity, Surface Tension, BIP sets, etc. including any tuning done to the fluid.
 - PIPESIM can currently use tuned data in only one MFL fluid file in the model. If you tune the PVT models (EOS, Viscosity, etc.) to match experimental data, e.g. viscosity, density, etc. in the Multiflash interface, it is strongly recommended that you use only one MFL file (the one with the tuned data) in the PIPESIM model. If you use multiple MFL files with tuned data in the PIPESIM model, the tuned data in only one of the MFL files will be used in the PIPESIM simulation run.
-

Viewing Wax or Asphaltene Curves on Phase Envelopes

Waxes are complex mixtures of solid hydrocarbons that freeze (solidify) out of crude oils if the temperature is low enough - below the critical wax deposition temperature. They are formed from normal paraffins (n-paraffins) and isoparaffins and naphthenes, if present.

Asphaltenes are defined as the fraction of crude oil that is insoluble in n-alkanes (for example, n-heptane or n-pentane) but soluble in aromatic solvents such as benzene and toluene. They are extremely complex mixtures whose molecular structure is difficult to determine because the molecules tend to stick together in solution. They do not have a specific chemical formula but are generally made up of large rings of aromatic molecules consisting of carbon, hydrogen, sulfur, oxygen, and nitrogen.

The Wax precipitation line and Asphaltene precipitation envelope can only be visualized in PIPESIM using Multiflash MFL fluid files. The requirements for displaying these precipitation lines on the PIPESIM phase envelope are outlined below.

Requirements for Display of Wax Precipitation Line on Phase Envelope

- The Coutinho model for the precipitation of the Wax phase must be used in conjunction with the RKSA equation of state for the phase equilibria of the other phases (This is done by choosing **Waxes** or **Combined Solids** in the Model set when defining the fluid in Multiflash).

- Composition of Live Oil or Stock Tank Oil from a gas chromatography analysis, entered under **Select > PVT Lab. Input** (if measured n-paraffin distribution is not available) or **Select > PVT Input with n-paraffin** (if measured n-paraffin distribution is available).
- The wax content must be provided in Multiflash when defining the fluid using any one of the following options:
 - Enter a lab-measured n-paraffin distribution under **Select > PVT Input with n-paraffin** (most accurate and recommended)
 - Enter a Total Wax content under **Select > PVT Lab. Input**. In this case, the n-paraffin distribution will be estimated by Multiflash based on the provided total wax content using the Coutinho & Daridon method
 - Check the box **Estimate Wax Content** under **Select > PVT Lab. Input**. Multiflash will estimate both the total wax content and the n-paraffin distribution. The wax content will be estimated empirically and the n-paraffin distribution will be estimated using the Coutinho & Daridon method (least accurate and not recommended)
- Optional Tuning Data for improving wax prediction accuracy
 - Measured Bubble Points (under **Tools > Matching > Bubble Point / GOR** in Multiflash)
 - Measured Wax Appearance Temperatures at corresponding pressures (under **Tools > Matching > Wax Phase** in Multiflash)
 - Measured amounts of precipitated wax at corresponding pressures and temperatures (under **Tools > Matching > Wax Phase** in Multiflash)

Requirements for Display of Asphaltene Precipitation Envelope on Phase Envelope

- The version of the RKSA equation of state that includes association terms for Asphaltene-Asphaltene and Asphaltene-Resin interactions must be defined (This is done by choosing **Asphaltenes** or **Combined Solids** in the Model set when defining the fluid in Multiflash).
- Composition of Live Oil or Stock Tank Oil from a gas chromatography analysis, entered under **Select > PVT Lab. Input** (for asphaltene precipitation only) or **Select > PVT Input with n-paraffin** (for both asphaltene and wax precipitation, if measured n-paraffin distribution is available for the wax).
- The amount of asphaltene in the oil and the ratio of resins to asphaltene, using any one of the following options in Multiflash:
 - Lab-measured, complete SARA analysis for example, Amount of Saturates, Aromatics, Resins and Asphaltenes (Most accurate and recommended).
 - Amount of Resins and Asphaltenes, as measured in the lab
 - Check the box **Estimate RA** (Resin-Asphaltene ratio) under **Select > PVT Lab. Input** or **Select > PVT Input with n-paraffin**. Checking this box will cause Multiflash to estimate the resin-asphaltene ratio using proprietary methods (least accurate and not recommended)
- Optional Tuning Data for improving asphaltene prediction accuracy
 - Measured Bubble Point(s) (under **Tools > Matching > Bubble Point / GOR** in Multiflash)

- All of the following (if available) under **Tools** » **Matching** » **Asphaltene** in Multiflash:
 - Measured Asphaltene Onset Pressures for Live Oil, ideally at two different temperatures (most accurate, recommended)
 - Measured Amount of n-Heptane required for the Onset of Asphaltene precipitation for the Dead Oil (most accurate, recommended in addition to (Measured Asphaltene Onset Pressures for Live Oil) if available)
 - Reservoir pressure and temperature (least accurate, should be provided if (i) and (ii) are not available)

Using MFL files with Wax and Asphaltene Phases

Once the MFL fluid file has been created to meet the Requirements for Display of Wax Precipitation Line on Phase Envelope or Requirements for Display of Asphaltene Precipitation Envelope on Phase Envelope, you may incorporate it in your PIPESIM model by browsing to it under **Setup** » **MFL file**. You can also report and plot the following wax and asphaltene system and profile variables by adding them to the report template using keywords:

- Wax formation temperature (profile): This is the wax precipitation temperature along the profile.
- Wax sub-cooling delta temperature (profile): This is the wax precipitation temperature minus the fluid temperature along the profile. A negative wax sub-cooling delta temperature indicates that the fluid is warmer than the wax formation temperature and there is no risk of forming wax. Conversely, a positive value indicates there is tendency for wax to form at that location.
- Maximum wax subcooling temperature difference (system): This is the maximum value of the wax sub-cooling delta temperature and pinpoints the location in the entire system that is at the greatest risk of forming wax, if it is a positive value.
- Asphaltene formation temperature (profile): This is the asphaltene precipitation temperature along the profile.

Multiflash in the Compositional Fluid mode (native) vs. Multiflash MFL files

PIPESIM enables two options for using the Multiflash package:

Package Type	Description
Multiflash in the Compositional Fluid mode ("native")	This option is enabled when you create a Compositional fluid and choose Multiflash as the PVT package. With this option, the entire fluid definition is done at a global level using the PIPESIM interface. The models selected (equation of state, viscosity, BIP set, etc.) are applied to all individual fluids defined in the model. The models available with this option are a subset of the full extent of the models available with the Multiflash MFL files option, which gives you access to the standalone Multiflash program directly.
Multiflash MFL files	This option is enabled when you reference an MFL file by selecting Setup » MFL file . The fluid definition is done using files generated by launching the Multiflash interface from within PIPESIM. This option gives you access to the full extent of the models available in Multiflash and is the required option for wax and asphaltene thermodynamics.

PVT Files

PVT files are one of the fluid modeling options available for running a PIPESIM simulation. The structure and content of a PVT file varies, depending on the software it was generated from. At the minimum, a PVT file contains a grid of thermodynamic and transport properties calculated at specific pressures and temperatures, by a PVT flash package. These properties must be calculated over a range of pressures and temperatures that cover the expected simulation conditions of the PIPESIM model. Some PVT files also contain the thermodynamic models used e.g. the equation of state and transport models, the component list as well as the moles of each component. The structure of a PVT file governs the way it can be used in PIPESIM.

A list of the typical properties available in a PVT file are presented below. It is not exhaustive. Depending on the source of the PVT file; a subset of the following properties will be available in it. For certain PVT flash packages, additional information such as the wax and hydrate precipitation lines can be exported as part of the PVT file, or as separate files.

PVT File Properties

Pressure	Temperature	Liquid volume fraction
Watercut volume fraction	Water mass fraction	Water thermal conductivity
Liquid density	Liquid viscosity	Liquid heat capacity
Liquid surface tension	Gas heat capacity	Gas density
Gas compressibility	Gas molecular weight	Gas viscosity
Gas enthalpy	Oil entropy	Gas thermal conductivity
Oil viscosity	Oil density	Oil mass fraction
Oil compressibility factor	Oil molecular weight	Oil heat capacity
Oil thermal conductivity	Oil enthalpy	Oil entropy
Oil-Gas surface tension	Total enthalpy	Total entropy
Wax density	Wax thermal conductivity	Derivative of gas density with respect to pressure
Derivative of gas density with respect to temperature	Derivative of oil density with respect to pressure	Derivative of oil density with respect to temperature

Structure of PVT Files

A PVT file consists of some or all of the following sections, depending on the software from which it was exported.

Equation of State	Viscosity model	Viscosity model coefficients
Component list	Moles of each component	Molecular weight of each component
Density of each component	Omega A for each component	Omega B for each component

Critical temperatures for each component	Critical pressures for each component	Critical volumes for each component
Critical Z-factors for each component	Volume shift for each component	Acentric factors for each component
Parachors for each component	Critical volumes for viscosity calculations	Critical Z-factors for viscosity calculations
Binary interaction coefficients or BIP set		

Table 1.9: Model Information

Grid Information

A pressure-temperature grid of a subset of the thermodynamic and transport properties listed under PVT File Properties. The minimum grid size that PIPESIM can handle is 1 x 1.

Importing a PIPESIM-generated PVT File

A PVT file exported from PIPESIM can be re-imported into it.

1. Import the components in the PVT file first, under **Setup** » **Compositional Template** » **Import**, and browse to select the file.
2. Import the component moles in the PVT file, under **Setup** » **Compositional**, and browse to select the same file.

Viewing the Contents of a PVT File

PVT files with a .pvt extension, are stored in the **PsPlot** format. **PsPlot** is the standard plotting utility used by PIPESIM. **PsPlot.exe** can be launched to browse to the PVT file and view its contents in graphical format. All the physical properties generated can be viewed by the **PsPlot** task.

Exporting PVT Files from PIPESIM

A compositional fluid defined using the Compositional template in PIPESIM can be exported as a [PVT file \(p.157\)](#).

1. Create the Compositional fluid by defining the models and component list under **Setup** » **Compositional template**.
2. Enter the moles of each component under **Setup** » **Compositional**.
3. In the Compositional properties window, click **Export Property Table**, and then enter the pressure and temperature values at which the PVT properties should be calculated, and exported to the file.
The minimum grid size is 1 x 1.
 - Click **Reset to default** to reset the Temperature-Pressure grid to its original values.
 - Click **Clear All** to delete all the values.
4. Click **Export Property Table** and select the location where you want the PVT file to be saved.

You can share the exported PVT file with other PIPESIM users. You can also use the exported PVT file in different models.

Important: Take care when creating (and using) PVT files to ensure that the selected Pressure and Temperature points span the range of operating conditions and are correctly spaced around phase boundaries. Running a check using the [rigorous](#) (p.164) option is recommended.

It is also important to understand the way PIPESIM uses PVT files. For more information, see [PIPESIM Handling of PVT files](#) (p.159).

PIPESIM Handling of PVT Files

PVT files (p.157) can be generated by PIPESIM*, as well as other software such as ECLIPSE PVTi, Multiflash, DBRSolids, PVTSim, etc. However, the structure and content of the PVT file is different, depending on the software it was generated from. For example, PVT files generated by PIPESIM and PVTi contain model information; such as equation of state, fluid composition, as well as the PVT properties at the specified grid of pressures and temperatures. For certain other software, only the PVT properties at the specified grid of pressures and temperatures are written to the file i.e. the model information is not exported.

Note: Multiflash versions embedded in PIPESIM Classic versions, older than 2012.2, export both the model and PVT property grid information. However, it is important to note that starting from PIPESIM 2012.2 and newer versions of PIPESIM Classic, PVT files generated from Multiflash specifically, do not contain any model information. They only contain the PVT properties at the specified grid of pressures and temperatures.

It is important to review each PVT file with a text editor, to understand what kind of information it contains. This is because the information contained in the PVT file determines how it can be used in PIPESIM, as outlined below.

PVT files can be used in PIPESIM in two ways; Imported and Linked. The table below outlines how specific PVT files, from different sources, can be used in PIPESIM.

1. **Imported:** Certain PVT files can be imported as Compositional fluids in PIPESIM. These are PVT files that contain model information such as composition, equation of state, viscosity model, etc. This approach creates a new Compositional fluid using the information contained in the file. To import PVT files, perform the following steps:
 - a. Import the components in the PVT file first, under **Setup** » **Compositional Template** » **Import**.
 - b. Import the component moles in the PVT file, under **Setup** » **Compositional**.
- When PIPESIM does a simulation with a PVT file imported as a compositional fluid, it actually performs a flash of the compositional fluid, using the assigned PVT package at the pressures and temperatures encountered along the flow path.
2. **Linked:** A PVT file can also be used in PIPESIM via a loose link to the file. This link is established under **Setup** » **PVT file**. The PVT file must be available, in this location for the simulation to run. In this scenario, the PVT file does not need to contain any model or composition information, it just needs to have the thermodynamic and transport PVT properties

at a specified grid of pressures and temperatures. The PVT properties also need to cover the expected pressures and temperatures that will be encountered during simulation.

When you select a PVT file under **Setup > PVT file**, you have the option to **View** and **Verify against Composition Template**. These 2 options are only available for supported PVT files that have compositional information, which are PIPESIM-exported PVT files. You will get an error if you select either of these options with an unsupported PVT file, or one that has no composition information. You can alternatively [view the contents of a PVT file \(p.158\)](#) using the PIPESIM PsPlot utility.

When PIPESIM runs a simulation with a linked PVT file, as long as the PVT grid is at least 4 x 4, and there is no kind of fluid mixing or separation that changes the fluid attributes (e.g. a network model, a model with a separator, gas lift, or with GOR or Water cut tuning) it interpolates the PVT file to determine the PVT properties at the pressures and temperatures encountered during the simulation. However, if there is any kind of fluid mixing or separation, then PIPESIM must use the composition information in the PVT file to perform flashes and determine the correct fluid attributes. It cannot interpolate PVT properties for these special scenarios. To reiterate, whenever fluids are mixed or separated in a PIPESIM model using PVT files, the composition information must be present in the PVT files for the simulation to run successfully.

The table below outlines how specific PVT files, from different sources, can be used in PIPESIM.

PVT File Source	File Extension	Can be imported?	Can be linked?
PIPESIM	.pvt	Yes	Yes
PVTi	.pvo	Yes	No
DBR Solids	.pvt	No	Yes
PVTSim	.pvt	No	Yes
Multiflash (in PIPESIM Classic versions older than 2012.2)	.pvt	Yes	Yes
Multiflash (in PIPESIM Classic version 2012.2, and newer)	.pvt	No	Yes
ScaleChem	.pvt	No	Yes
GUTS	.pvt	No	Yes
AsphWax	.pvt	No	Yes

Refer to the topic: [Managing Multiple Fluid Models \(p. 0 \)](#) to learn more about the other fluid modeling options available in PIPESIM.

Fluid Property Table (External PVT Tables)

PIPESIM provides various levels of support for several PVT packages; Schlumberger and 3rd party. One way in which PVT data from other sources can be used in PIPESIM, is via [PVT files \(p.157\)](#). A PVT file is simply a table of PVT properties generated over a grid of specified pressures and temperatures. PVT files can be generated by PIPESIM, as well as other software such as ECLIPSE PVTi, Multiflash, DBRSolids, PVTSim, GUTS, etc., and can be used for simulation.

A PVT data file is written by the PVT simulator and stored externally to PIPESIM in an ASCII file. When properties are required at a specific pressure and temperature (PT) the data file will be interrogated, and interpolation (or extrapolation) used to find the properties at the required PT point. It is important to note that the format of the PVT file is different, depending on the software it was generated from. The information contained in the PVT file determines how the file can be used in PIPESIM. For more details on the differences in PVT files generated from different sources, and the differences in the ways they can be used, see [PIPESIM Handling of PVT Files \(p.159\)](#).

External PVT tables can also be created from within PIPESIM. For more information, see [Exporting PVT Files from PIPESIM \(p.158\)](#).

Solids Modeling

Solids are modeled in the following ways:

- Sand is modeled for its effects on erosion. (Refer to [Erosion and Corrosion Options \(p.176\)](#).)
Sand inventory is traced in the model. (Refer to [Sand Modeling \(p.161\)](#).)
- Wax is modeled for its deposition behavior. (Refer to [Wax Deposition \(p.205\)](#).)
- Wax, Asphaltene, and Scale are modeled for their appearance in the fluid. (Refer to [Solids Appearance \(p.162\)](#).)

See also [Scale Predictions \(p.163\)](#).

Sand Modeling

In PIPESIM, the purpose of sand modelling is to predict its erosion effect. Sand has **no effect** on pressure drop, heat transfer calculations, or on overall fluid mass balance across the network. Sand is assumed to be transported at the fluid mean velocity. There is no separate modeling of sand velocity, sand deposition, or accumulation in low velocity pipe bends or sumps. Fluid density, viscosity, and other transport properties, are not affected by sand rate.

In a **single-branch model**, specify the sand rate using **Setup » Erosion & Corrosion Properties**. For more information, refer to [Erosion and Corrosion Options \(p.176\)](#). The sand rate applies to the whole branch. (At present, there is no support for different sand rates from individual reservoirs or completions).

In a **network model**, specify the sand rate in each source branch by double-click the branch to bring up its individual window. Then select **Setup » Erosion & Corrison Properties** and enter the desired sand rate for the branch. The network solution keeps track of sand inventory in each branch and mixes it into downstream branches at network junctions. Thus, the erosion effects of the sand in link and sink branches are accounted for correctly. (Sand rate can in fact be entered for **any** branch in the network. PIPESIM makes no attempt to restrict you to source branches.)

However, any sand rates that you enter for link and sink branches are ignored by the network solver. PIPESIM uses the calculated mixed values.)

Sand is assumed to be present in the liquid phase of the fluid for the purpose of separator modeling.

Wax Deposition Modeling

PIPESIM can model the **deposition** of solid wax on the inside surface of the pipe system.

Note: This is distinct and separate from modeling the solids appearance in the fluid. See [Solids Appearance \(p.162\)](#).

Wax deposition reduces the pipe internal diameter, and therefore affects the fluid velocity and system pressure drop. The wax deposit causes a change in the heat transfer characteristics of the pipe system, which affects the fluid temperature. PIPESIM models both effects.

Wax deposition is strictly a transient effect. PIPESIM is a steady-state simulator and cannot perform a rigorous wax deposition simulation. Nevertheless, wax deposition typically occurs over a timescale of weeks or months, so a series of steady-state timesteps is a good approximation to real system behavior. This is what PIPESIM does. See [Operations » Wax Deposition](#). ([Wax Deposition \(p.205\)](#)) This operation allows a series of timesteps to be performed for modeling the buildup of wax in the system and its effect on pressure drop, deliverability, and temperature.

Wax deposition can also be modeled as an effect caused by other system sensitivity variables. If sufficient wax deposition data is available (see [Setup » Wax Properties](#)), a rate of wax deposition is also available at any point in the system. This information can be viewed in the system and profile plots along with any other result variables as a function of selected sensitivity data. For example, you could use the operation [Pressure / Temperature profile \(p.196\)](#) to sensitize on system inlet temperature, and to view the effect of this on the profile of wax deposition rate.

Deposition of wax has no effect on fluid composition, fluid transport properties, or on overall mass balance in the system.

See also [Solids Modeling. \(p.161\)](#)

Solids Appearance

The phase behavior of fluids can result in the formation of one or more solid phases. PIPESIM does not simulate the flow and transport of solids in any rigorous manner, which is comparable to the support of 2- and 3-phase fluids with flow correlations. However, the formation of solids — such as Wax, Asphaltene, and Scale in the bulk fluid — can be modeled and predicted using the PVT tables that are written by third-party programs, such as the Schlumberger product DBR Solids and OLI's ScaleChem. This modeling is confined to the **appearance** of the solid phase in the bulk fluid and **does not model the deposition** of the solid on the pipe or tubing walls.

Third-party programs write a PVT table file in a PIPESIM enhanced PVT file format. That file contains fluid phase behavior and transport properties on a grid of pressures and temperatures. Additional properties in the file describe the solids phases that the third-party programs model. The exact nature of the solids phase(s) and the properties associated with them depend exclusively on third-party programs. Because PIPESIM deals with solids phases in a generalized manner, updates to the third-party programs that write the enhanced PVT files should be compatible with existing PIPESIM functionality.

Once the PVT files are created, PIPESIM accesses those files when specific pressure-temperature data is needed. For more information, refer to [Fluid Property Table \(External PVT Tables\) \(p.160\)](#).

Note: Because PVT files contain fluid phase behavior and transport properties, solids appearance modeling cannot be combined with other methods of fluid PVT modeling, such as Black oil or Compositional fluids.

To view the additional solids data in the PVT tables, use the profile plot. For example, if you model Wax and Scale results, wax and scale appearance temperatures are available in the fluid-temperature profile plot by setting the x-axis to temperature and the y-axis to pressure. If you want to view solids appearance in the fluid, plot the solid phase's Mass Fraction data.

See also [Scale Prediction \(p.163\)](#) and [Solids Modeling. \(p.161\)](#)

Scale Prediction

Prediction of scale formation may be performed by using a [PVT file \(p.160\)](#) generated by OLI's [ScaleChem](#) (<http://www.olisystems.com/oliscale.htm>) program (purchased separately). No extra license for PIPESIM is required.

To use the PVT file, select **Setup » PVT file** and browse to the PVT file generated by ScaleChem.

The PVT file contains tables of all fluid phases and transport properties required for thermo-hydraulic calculations. Additionally, this file contains detailed water chemistry information that enables PIPESIM to determine the occurrence, type, location, and severity of scale formation. Various profile properties related to scale formation are reported, including:

- Total scale concentration
- Individual scale species concentrations
- Mass ratio of individual scale species
- Pre-Scale Index
- Post-Scale Index

Scaling tendencies are reported by ScaleChem in two forms: the pre-scale index and the post-scale index. The pre-scale index is based on a condition where everything remains in solution (no solid phases are allowed to form). The pre-scale index values can be greater than 1.0 (supersaturated). The post-scale index (more commonly used) represents equilibrium conditions. If the water is saturated (the scale species precipitates), then the post-scale index is 1. The post-scale index is also useful for understanding the potential risk of scaling based on the degree of undersaturation, that is, how close the post-scale index is to 1.

These properties may be viewed in profile plots and tables.

To view phase appearance lines on the same plot as the production profile, select temperature as the x-axis and pressure as the y-axis. Each scale species will be described by either a phase "appearance" line, a phase "disappearance" line, or both. Moving from the left side of the plot to the right (from low to high temperature at a constant pressure):

- If a phase appearance line is encountered first, then solids will exist only at temperatures above this line until (and if) a phase disappearance line is encountered. This behavior is typical for CaCO₃ and CaSO₄.
- If a phase disappearance line is encountered first, then solids exist only at temperatures below this line. This behavior is typical for BaSO₄.

ScaleChem PVT files may be used for [corrosion prediction \(p.390\)](#). OLI's Corrosion Analyzer may be used for more detailed corrosion analysis.

Validating contents of PVT file

To view information contained in the PVT file, double-click the file to open it in PSPlot. The variables in the PVT file can be viewed graphically as a function of pressure and temperature. The composition of the fluid used to generate the PVT file is recorded in the comments section in the header of the PVT file. This file can be viewed in a text editor. To view the contents of the PVT file in a tabular format, under **Setup » Define Output**, select "Fluid Property Tables". When you run a simulation, all fluid properties are written to an output file.

See also

- Example files: Case Studies/Well Design and Performance/ScaleChem
- [Fluid Properties Tables Files \(External PVT Tables\) \(p.160\)](#)
- [Solids Appearance \(p.162\)](#)

Compositional Flashing

[More details. \(p.141\)](#)

Compositional Flashing determines the way in which physical properties are computed. The balance is between speed and accuracy.

Note: This is only valid for compositional models.

Select **Setup » Flashing** to open the **Compositional Flashing** tab.

Temperature Energy Balance / Physical Properties

The following methods are available for calculating compositional fluid physical properties:

Always Interpolate (fast)

This option uses interpolation between physical properties determined by in a predefined grid of temperature and pressure points. This grid can be modified from the compositional menu. (default)

Rigorous Flash when close to the Phase Envelope, interpolate elsewhere

This is a compromise between speed and accuracy, which assumes that properties will change more rapidly when close to a phase boundary. Interpolation is performed whenever the grid points comprising a rectangle all show the presence of the same phases. For example if all 4 points in the rectangle have some oil, some gas, and no water, then we assume the rectangle lies entirely within the 2-phase region of the hydrocarbon phase envelope, so interpolation is appropriate. If however one, two or three of the points have no oil, then clearly the hydrocarbon dew point line crosses the rectangle, so a rigorous flash is required.

Always Rigorous Flash (slow)

Interpolation never occurs: properties are obtained by flashing at the required pressure and temperature. This is the slowest but most accurate method.

These apply in the following sections:

Temperature Energy Balance (TH)

These values are used to maintain the temperature/enthalpy/entropy balance of the fluid.

Physical properties (PP)

These are the values required to perform the multiphase fluid flow and heat transfer calculations. They include phase volume fractions, densities, viscosities, heat capacities, and surface tension.

Note the following:

- In most simulations, for every PP flash that is performed, there are about 5 to 10 TH flashes. Thus, these (TH flashes) clearly have the greatest effect on speed and run time. The inaccuracies of TH interpolated flashes are usually minimal.
- The speed impact of each choice obviously depends on the composition, and the phase behavior in the PT region of interest. As a rough guide, taking the base case as interpolation, swapping just the PP flashes to "rigorous" will multiply your run time by about 4. With TH flashes also "rigorous", run time will probably increase at least 20 fold. Use of the 'compromise' choices will be faster.
- For those requiring more accuracy, we have found the "most useful" settings (that is the greatest increase in accuracy for the smallest effect on performance) to be the following:
 - PP= Rigorous Flash when close to the Phase Envelope, interpolation elsewhere
 - TH= Always Interpolate.

In-line flashing (default)

These are PVT tables that built (in memory) when the simulation is underway. There are three options to this method, as follows:

Interpolation

In order to maximize the speed of the simulation not all requested PT points are flashed, A pressure/temperature grid is defined and only these points are created. For points not lying exactly on a grid point, four-point interpolation is used. The default grid points can be changed using the **compositional option**. The fastest but least accurate method.

Interpolate when close to phase boundary

The above table is used except where 1 or more of the four-points used for the interpolation is in a different phase. In this case a full flash is performed and the data point added to the table. This will improve accuracy, but at the cost of speed

Rigorous

a full flash is always performed. Very accurate, but slow!

[More details \(p.164\)](#)

In a Network model each source can have a different composition assigned to it, the streams are mixed at junctions. These mixer streams are then reported for all branches in the output file. Additional information on the fluid properties can be reported by adding the [report tool \(p.115\)](#) object to a model and selecting the compositional reporting options.

The compositional interface takes advantage of concepts and terminology specific to petroleum engineers, providing an easy to use front-end to a complex multiphase flash packages. It allows the following:

- compositions to be specified (including petroleum fractions),
- PVT tables to be generated
- phase envelopes (with water and hydrate prediction) to be generated
- compositions to be exported and imported
- PVT Data matching

Stock tank Conditions

Stock tank conditions are defined as:

Pressure	Temperature
14.7 psia	60 F
1 bara	15.5 C

Flowing Conditions

Flowing conditions are defined at the actual in-situ flowing pressure (P) and temperature (T).

Examples

Flowing gas flowrate

the gas flowrate at stock-tank conditions (mmscf/d), this rate does not include the dissolved gas and therefore its value increases along the flow system due to pressure losses. If the outlet pressure is set to 14.7 psia, then the profile will eventually converge to the stock-tank gas rate (3).

Flowing gas volume flowrate

the gas volume flowrate at in-situ flowing pressure (P) and temperature (T) (mmcf/d). This rate has lower values than those of (1).

Stock Tank gas flowrate

the gas flowrate at stock-tank conditions (mmscf/d), this is the total gas (free + dissolved) which remains constant throughout the flow system.

Steam

For steam systems (production and injection) PIPESIM uses "ASTEM97 - IAPWS IF97 Properties of Water and Steam for Industrial Use," Copyright Edward D. Throm (C) 2005.

When modeling steam systems the pressure and quality or temperature are required. If the quality is not provided, superheated (quality =100%) or sub-cooled (quality=0%) then the temperature is required.

Steam systems can be modeled in both single branch and network models using **engine keywords**. These can be specified from the Pipesim GUI as described below. However:

Note: Because the GUI does not understand steam as a fluid model choice, it will require you to specify a valid fluid model, either as Black Oil, or Compositional. The steam keywords will override this, so the choice is not really relevant when the model is working.

Single branch steam

1. To model steam in a single branch PIPESIM model, go to **Setup » Engine Options** in the single branch view of the steam source branch, and specify the following keywords, for example:.

```
STEAM  
INLET QUALITY = 0.5
```

The inlet steam quality needs to be specified, if not, the engine will assume it to be either 0.0 or 1.0 depending on the pressure and temperature at the inlet.

2. Make sure you have a black oil fluid specified, with a GLR of zero and a watercut of 100% correctly.
3. Mass flow rates must be used with steam. Any operation that specifies a flowrate, or sets a flowrate limit, must do so with a mass rate, not a gas or liquid rate.

When steam quality is provided, it will be used with the Inlet pressure to calculate the resulting steam temperature and enthalpy; Any inlet temperature you specify will be ignored.

If quality is not provided, enthalpy will be used instead. If Enthalpy is not provided, the system will be flashed at the specified inlet pressure and temperature, and as a result will be 100% liquid or 100% vapour at the system inlet.

Network model steam

1. To model steam sources in a network model, go to **Setup » Engine Options** in the network view, and enter the following data for example in the lower section of the window:

```
SETUP COMP = STEAM  
SOURCE NAME = SS1 QUALITY = 0.8
```

2. Enter the quality for all the steam sources. If the quality is not entered, it will be determined from the temperature and pressure given for that source. If it is entered the source will be considered saturated at that pressure and the temperature will be adjusted accordingly.

Note: Steam is considered as a third thermodynamic model (after blackoil and compositional). At present only one thermodynamic model is allowed per network, so steam systems have to be modeled as a separate network from the hydrocarbon production or injection networks.

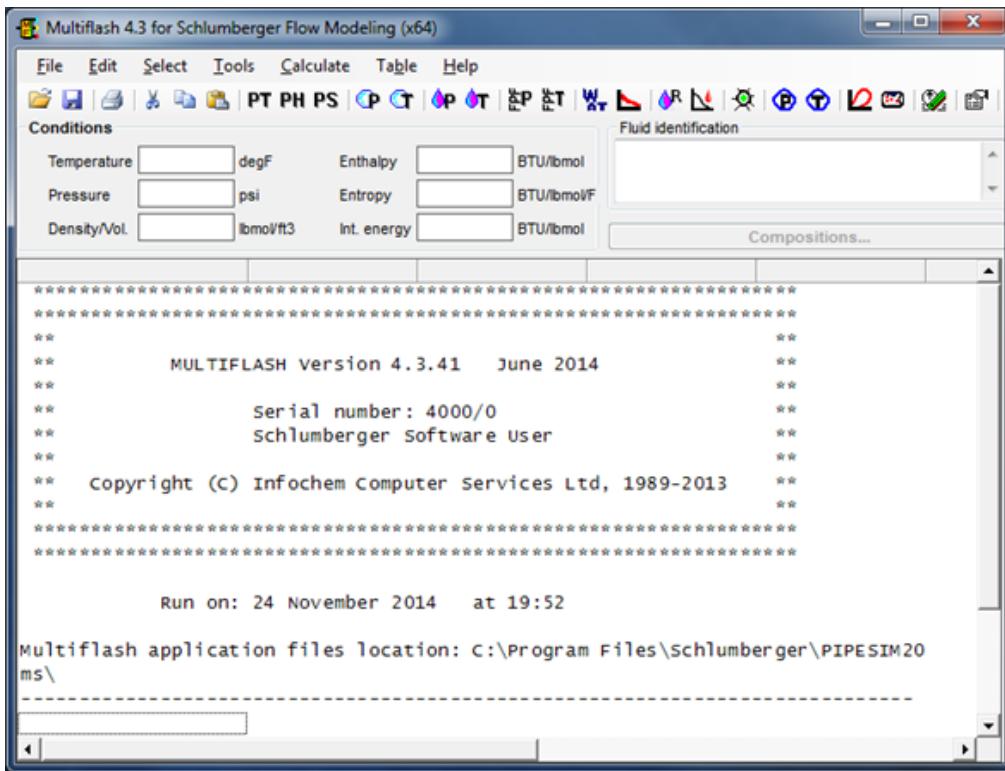
1.4.7 Converting Black Oil Models into Compositional Models using Multiflash for PIPESIM*

The black oil analysis within Multiflash™ allows you to generate a normal compositional analysis from a very limited input specification (known as Black Oil input).

Knowledge of the type of reservoir fluid to model is one of the main requirements to run any PIPESIM* Production System Analysis Software simulation. Black Oil and Compositional models are available in PIPESIM throughout the GUI or by using PVT (Pressure - Volume - Temperature) or MFL (Multiflash) files.

The black oil model is used for simulating dry gas, water, and non-volatile oils while the compositional model is best suited for light oils, condensates, and natural gases. Any reservoir fluid could be described using a compositional model that allows more detailed analysis of the fluid behavior. However, you rarely have compositional data so are forced to use a black oil model without being able to analyze further problems like flow assurance issues. However, Multiflash provides a useful feature to create an equivalent Compositional fluid from basic Black oil data. The steps to do this are outlined below.

1. Open the PIPESIM application, choose to create a New Single Branch model, and then click **Setup » MFL file**.
2. Click **Create New** to launch the Multiflash interface.



3. Click **File » Save Problem Set Up...** to create your Multiflash model. Save periodically during the rest of this procedure.
4. Click **Select » Units**, choose the units of your preference (for example, **All British**) and then click **OK**.
5. Define the model set.
 - a. Click **Select » Model set...**, on the **Cubic EoS Models** tab, select the desired model, and then click **Define Model**.

Note: Refer to the Multiflash Help for a detailed explanation on model selection, or leave the default selections.

- A message displays that shows your selection.
- b. Click **OK** and close the Select Model Set window.
 6. Enter values for black oil analysis.
 - a. Click **Select » PVT Lab Input...** and click on the **Black Oil Analysis** tab, enter the **Minimum input for Black Oil Analysis**.
 - b. Enter the available black oil data in the section **Minimum input for Black Oil Analysis**.

It is strongly recommended that you enter the Watson K-factor value if it is available, even though it is listed as optional. This is because it is used to determine the molecular weight of the stock tank liquid for the created, equivalent compositional fluid.

Note: You can also enter values for SARA Analysis, Total Wax Content, and Water Cut (as % of total liquid).

7. Configure the pseudocomponent settings for the fluid characterization.
 - a. Select the carbon number you would like the pseudocomponents to start from, when the characterization is done. Do this by selecting from the **Start pseudocomponents at** dropdown menu.
 - b. Select the number of pseudocomponents you want in the created compositional fluid. Do this by selecting from the **Number of pseudocomponents required** dropdown menu.
8. Click **Do Characterisation**.

A message displays indication that black oil characterization was successfully completed.

This process generates an equivalent compositional fluid made up of pure components and petroleum fractions in the Multiflash Results window. The pseudocomponents generated will depend on the settings you configured in steps 7a-b. The more data you enter in the Black Oil Analysis tab, the closer the match between the created compositional fluid, and the original black oil fluid.
9. Click **OK** and close the PVT Lab Fluid Analysis window.
10. View the phase envelope for the fluid by clicking the **Phase envelope** or **Phase envelope for solids** icons on the toolbar. They can also be accessed via **Calculate » Phase envelope**. Exit the phase envelope.
11. Save the MFL file again and close the Multiflash interface. This takes you back to the PIPESIM interface and displays a link to the MFL file you just created. You may now run the PIPESIM simulation with this file, as long as it is present in the location specified under **Setup » MFL file**.

1.5 Options

1.5.1 Common Options

Project Data

Use **Setup » Project Data** to enter any of the following optional information on the PIPESIM project:

- Project
- User
- Job
- Manager
- Company
- Work order
- Client
- Remarks

The project title will be displayed on all reports.

Units

These are the default units of measurement for the main engineering quantities which appear as a first choice for input data. The units can be in one of the predefined data sets "Engineering" or "SI". Alternatively, you can specify a customized set of units by selecting the **Custom** option.

The current selected data set can also be set so that it is the default when a new model is created.
[See how... \(p.170\)](#)

Some custom data sets are provided. These include Canadian, Mexican and SI unit sets. These can be imported from the "PIPESIM/data" directory.

Selecting a units set

To select the unit set, do the following:

1. Select **Setup » Units....** On the **Set Units** tab, do one of the following:
 - Select **Eng** to use engineering units.
 - Select **SI**.
 - Select **Custom** and define the unit set for each property.
2. If required, click **Set as Default**. This makes this unit set the default used when any new model is opened. Existing models will retain the unit that they were saved with.

Custom units

A custom data set can also be saved to a data file (*.unf) for use by other users or models. A custom set is, by default, based on the Eng. data set. However, it can be based on the SI set if you

first import the SI set (SI.unf file). Set the required units and provide a data set name, then click **Export** to be prompted for a file name and location.

The **More Units>>** button allows more control over the units selected.

Choose Paths

Note: Only change these entries if advised to by Schlumberger.

To change the path of any tool used, do the following:

1. Select **Setup > Preferences > Choose Paths....** Alternatively **Expert > Display > Preferences > Choose Paths...**
2. Change the file and full path of any the following:

Editor

can be changed to use your preferred text editor

Single branch calculation engine

can be changed if a new engine is provided by Schlumberger

Network calculation engine

can be changed if a new engine is provided by Schlumberger.

Plot

plot utility to use

Database

database location. Note: If this is changed, close and restart PIPESIM so the changes take effect.

USERDLL

Location of the USERDLL.dat file. This is used to edit/specify to PIPESIM the characteristics of USER defined correlations. **Changing this may remove some correlations from PIPESIM. Only for expert Users.**

Note: If this is changed, close and restart PIPESIM so the changes take effect.

Heat Transfer Options

To set up the Heat Transfer options, select **Setup > Heat Transfer Options.**

Pipe Burial Method

You can select the model to be used for pipeline heat transfer calculations. The calculations take into consideration the burial configuration of the pipe (fully buried, partially buried, or fully exposed) and give different U-value results based on the model selected. The options available, in decreasing order of accuracy, are the following:

- [2009 Method \(p.499\)](#)
- [2000 Method \(p.501\)](#)

- 1983 Method (p.500)

The three options give identical results for a fully exposed pipeline. However, for a fully buried pipe or partially buried pipe they give different results. The 2009 Method is recommended for use and is the current default. See [Further details \(p.486\)](#).

Also see [heat transfer](#). (p.100)

Inside Film Coefficient Method

You can specify the inside film coefficient calculation model to be used during heat loss calculations. You can choose between the [Kaminsky Method \(p.492\)](#) and the [Kreith Method \(p.489\)](#)

U Value Multiplier

You can specify a multiplier for user enter U-values in heat loss calculations. This is particularly useful when performing a temperature match in the Data Matching Operation.

Hydrate Sub-Cooling Calculation

This switch turns on the calculation and reporting of the hydrate formation temperature. The maximum amount of sub-cooling (Hydrate formation temperature - Fluid Temperature) is reported for each branch of a network. This can help to determine where the hydrate problem starts in the flowline.

You can plot the following:

- Fluid Temperature versus Distance
- Hydrate Formation Temperature versus Distance
- Degrees of Subcooling versus Distance

You can display all of the plots on the same plot.

The maximum amount of sub-cooling in a branch is calculated and reported to the system plot file.

Note: This calculation may increase runtime significantly.

Engine Options

To set these options, select **Setup > Engine Options**. The parameters are as follows:

Segments per pipe length and Maximum Segment Length

These data fields allow you to set the initial maximum segment length either as a length quantity, or as a division factor of each section. If both are specified, the smaller will be taken. The segment may be further subdivided by the program, as required, so as to obtain a converged solution.

Iteration Estimate

This data field allows you to provide an estimated inlet pressure or an estimated mass flow rate. For further information see [ITERN Iteration Data \(Optional\) \(p.621\)](#).

Additional engine keywords

This data field allows you to provide additional engine keywords for the PIPESIM single branch or network engine.

PIPESIM Network Option Control

Select **Setup** » **Define Output** and select the **Output Control** tab. The parameters are as follows:

Use local branch options

If this option is selected engine will use corresponding values specified for each local branch.

Use network options (in addition to local branch options)

If this option is selected it allows you to add output options defined for network to options defined for each local branch

Apply network options to all branches

This overrides specific options defined on branch level with values defined for network.

Output Options

[More details. \(p.624\)](#)

Define Output

To change the options, select **Setup** » **Define Output**.

Options that apply to the data reported in the main output file for a single branch (*.out) are as follows:

Primary Output Page(s)

Primary Output Page(s) typically contain the following information for each node :
Distance, Elevation, Horizontal Angle, Vertical Deviation, Pressure, Temperature, Mean Velocity, elevational and Frictional Pressure Drop, Liquid Flowrate, Free Gas Flowrate, Liquid and Gas Densities, Slug Number and Flow Pattern.

Auxiliary Output Page(s)

Auxiliary Output Page(s) typically contain the following information for each node :
Distance, Elevation, Pipe ID, Superficial Liquid and Gas Velocities, Liquid and Gas Mass Flowrates, Liquid and Gas Viscosities, Reynolds Number, Liquid Volume Fraction, Liquid Holdup Fraction, Enthalpy, Iteration Number, Flash/Table Interpolation Diagnostics.

Well Inflow Performance Data

The well Inflow performance data output page contains the well inflow performance relationship input and derived values used to model the relationship between well drawdown and flowrate. This page only applies to models containing a completion.

System Profile

Primary Output Page(s) typically contain the following information for each node :
Distance, Elevation, Section Length, Total Length, Ambient Temperature, Input U, and Fluid Data File.

Echo Input Fluid Properties

Fluid Property Pages contain the fluid property input data and some calculated properties at stock tank conditions.

Echo Engine Batch Language

Prints out the input keywords at the top of the output file.

Segment Data in Primary Output

A "section" of the model is defined as the thing between two nodes. Sections are further divided into "segments" for calculation purposes. Segment data can be included in the Primary Output Page by checking this box.

Heat Transfer Input Data

The Heat Transfer Input Data Page(s) typically contain the following information for each node: Pipe and Coating Thicknesses and Thermal Conductivities, Soil Thermal Conductivity, Burial Depth, Ambient Fluid Velocity, and Ambient Temperature.

Heat Transfer Output Data

The Heat Transfer Output Data Page(s) typically contain the following information for each node: Distance, Fluid Temperature, Fluid Enthalpy, Temperature Gradients, and Heat Transfer Coefficients.

Slug Output Pages

Slug Output Page(s) typically contain the following information for each node : Distance, Elevation, Pipe ID, Slug Lengths and Frequencies, Slug Number and Flow Pattern.

Iteration Progress Log

This prints the log of iterations for iterative models only, that is those where the calculated variable is inlet pressure or flowrate.

Wax Page(s)

Create specific pages for the wax operation.

Fluid property tables

Create an output file (*.out) that contains tables of the properties as a function of pressure and temperature for compositional fluids ONLY. [Blackoil fluids \(p.137\)](#) are controlled under **Setup » Blackoil**. See also [TPRINT Tabular Data Print Options \(p.738\)](#).

No Cases to Print (Single branch mode only)

Determines the number of cases to print full results for in a single branch sensitivity case. By default full results are only printed for the 1st case.

Plot File Format

Determines the format of the created plot file.

PS-PLOT

PIPESIM's graphical post-processor (Default)

Lotus compatible

Lotus formatted file

Excel compatible

Excel formatted file

Excel compatible (packed).

Excel formatted file that is compacted.

Neutral

standard ASCII text file.

Report Tool Output Goes To

When a report tool is placed in the model, its results are written to the output file (*.out).

The section of the output file that the report goes to can be selected, the default is

Dedicated.**Export data to EXCEL**

From the **calculation engine**, the basic [plot] data can be written to an Microsoft Excel formatted file by specifying one of the Excel options for the Plot File Format under the **Setup » Define Output**. This creates an Excel data file with the extension .csv. See [Setup/Define Output \(p.173\)](#).

From the GUI:

1. From the network module (after a simulation) the field results can be exported to Excel using **Reports » Report Tool**.
2. In the Report Tool, select the required field data and click the **Excel** button. If Excel is located on the PC, this step opens an [interface \(p.264\)](#) to Excel.

From **PsPlot**, use the [Excel button \(p.264\)](#) on the **Data** tab.

Produce a custom report

The standard reports (full output and summary files) have been designed to provide commonly reported results after a simulation. However there will be situations where you only require a subset of the information or some data that is not available in the standard report. In this case the custom report feature can be utilized.

Custom reports are invoked by using the PIPESIM Input Language (PIL) using the [additional EKT \(p.595\)](#).

Note: The [EKT \(p.94\)](#) (spanner tool) can not be used for this operation.

The sub-command [CUSTOM \(p.624\)](#) of the [PRINT \(p.624\)](#) keyword is required to create custom reports. The keyword PRINT CUSTOM requires a list of variables to print to the custom report, these can be determined using the [PLOT SYNTAX \(p.631\)](#) keyword. The codes returned from this list can then be used as argument on the PRINT CUSTOM command.

Example 1

To obtain a list of available custom report variable, use the additional EKT option, add PLOT SYNTAX and run the model. The output file contains a list of variables and codes for "Profile plot file, SPOT & CUSTOM reports".

Example 2

To produce a custom report with "Horizontal distance", "Total distance" and "Elevation". In the output file locate the codes for the required properties, in this case A, Band C. Using the [additional EKT \(p.595\)](#) option, add PRINT CUSTOM = (A,B,C) (the PLOT SYNTAX can be left if required) and run the model. The output file contains a custom report section with the variables required.

View tabular results

Tabular (text) results files are created, from the calculation engine, in standard ASCII format (*.out and *.sum) and can be viewed by any text editor.

You can change this to use a preferred text editor using the [Choose Path option \(p.171\)](#).

Erosion and Corrosion Options

The erosion and corrosion options are used to calculate the following:

- erosion and corrosion rates
- erosional velocity limit

These are reported per-node in the **Auxiliary Output** page of the main output file, and in the profile plot file. The maxima for the case are reported in the system plot file. The erosional velocity is the maximum fluid velocity as recommended by the chosen correlation. The erosional velocity appears in the **Auxiliary Output** page as a ratio with the fluid mean velocity, so values of 1 or greater indicate the need for attention.

To set the options, select **Setup » Erosion & Corrosion Properties**.

Erosion

Erosion can occur in solids-free fluids, but is usually caused by entrained solids (sand). The rate of sand production is the main determinant of erosion rate. Sand production rate is specified per-branch in a network model. Sand produced in source or well branches will be tracked across manifolds and mixed along with fluids, so an accurate erosion rate will be calculated without the need for you to specify the sand rate in link branches. Sand is assumed to follow liquid production when a separator is modeled.

The following erosion models are available:

- API 14 E
- Salama

API 14 E

The API 14 E model comes from the American Petroleum Institute, Recommended Practice, number 14 E. This is a solids-free model which will calculate an erosion velocity only (no erosion rate). The erosion velocity V_e is calculated with the formula

$$V_e = \frac{K}{\rho^{0.5}} \quad \text{Eq. 1.3}$$

where ρ is the fluid mean density and K is an empirical constant. K has dimensions of $\left(\frac{\text{mass}}{(\text{length} \times \text{time}^2)}\right)^{0.5}$. Its default value in engineering units is 100, this corresponds to 122 in SI units.

Salama

The Salama model was published in Journal of Energy Resources Technology, Vol 122, June 2000, "An Alternative to API 14 E Erosional Velocity Limits for Sand Laden Fluids", by Mamdouh M. Salama. This models erosion rate and erosional velocity.

The parameters required for the model are as follows:

Acceptable Erosion rate

rate of erosion that is deemed 'acceptable'. This is used to calculate the erosional velocity. Erosional velocity appears as a ratio with fluid mean velocity in the Auxiliary output page. The default is 0.1 mm/year.

Sand production ratio

a measure of sand production rate, as a ratio with liquid rate. Units are Parts Per Million, by volume, against **stock-tank liquid** rate. If sand production ratio is zero, erosion rate will not be calculated.

Sand Grain Size

mean size of the sand grains. The default 0.25 mm.

Geometry Constant

this is the geometry dependent constant **Sm**. The default is 5.5

Efficiency

a multiplier used to match field data. The default is 1.

The equations in Salama's paper use a sand rate in Kg/day. This is obtained from the supplied volume ratio using Salama's 'typical value' for sand density, 2650 kg/m³.

See also [EROSION \(p.639\)](#).

Corrosion

The [de Waard \(1995\) corrosion model \(p.582\)](#) calculates a corrosion rate caused by the presence of CO₂ dissolved in water. Concentrations of CO₂ and water are obtained from the fluid property definitions, (black oil, compositional, ScaleChem generated PVT files). If CO₂ is absent from the fluid and no liquid water phase is present, the corrosion rate is zero.

The parameters required for the model are:

Efficiency

multiplier C_c to correct for inhibitor efficiency, or to match field data.

Actual pH

if supplied, this value is used instead of the calculated one.

See also [CORROSION \(p.639\)](#) and [deWaard \(1995\) Corrosion Model \(p.390\)](#).

Engine Preferences

To change the engine preferences, select **Setup** » **Preferences** » **Engine**. The dialog this opens has two tabs, **Engine Preferences** and **Advanced**.

Engine Preferences tab

The following Preference settings affect the running of the calculation engine and apply to PIPESIM, every time that it is started.

Run Plot tool with single branch engine	Allows the plot utility to be run and display results graphically as the calculation engine is running. The default is on.
Run engine minimized	Runs the engine in a minimized state. The default is on. If you want to see the progress of the simulation, deselect this option or maximize the PIPESIM engine window.
Add unit descriptors to the engine files	When the data is written to the engine file, the units of each property are added. The default is off.
Run engine via batch file	Running the engine via a batch file allows the window to be kept open if the engine fails, so any error messages can be viewed. This also allows license errors to be identified. The default is on.
Number of processes for Network engine	PIPESIM 2012 (and newer) introduced a parallelized network solver where you can run network simulations with multiple processors to increase the speed. The selection for this will be limited to the number of available processors on your hardware as reported by Windows. The larger the selected value the faster the network simulation. Set this to a smaller value if you would like to limit the number of processes used for simulation due to other running applications that may need processing resources. The parallelized network solver requires the installation of a compatible version of Intel MPI (the version in the PIPESIM install kit). Most issues with incompatible Intel MPI versions can be easily resolved by uninstalling the incompatible version and installing the compatible one. For more information, see Parallelized Network Solver. (p.178)

Advanced tab

To change these, select the **Advanced** tab.

Additional Single Branch Engine Command Line Parameters

The following are command line arguments for the single branch engine:

-vx Controls screen output levels. The default = -v1

-b Do not issue dialog box when simulation has finished

Additional Network Engine Command Line Parameters

The following are command line arguments for the Network engine:

- v**x Controls screen output levels. The default = -v1
- b** Do not issue dialog box when simulation has finished.
- c** Creates well performance curves.
- r** Restart but do not keep branches skipped in the restart file permanently skipped. This option has to be used ONLY with the run button in the toolbar



Note: Use of the restart button



, overrides this option and keeps all branches skipped in the restart file permanently skipped. See [Restart Model \(p.212\)](#).

Parallelized Network Solver

PIPESIM 2012 (and newer) introduced a parallelized network solver where you can run network simulations with multiple processors to increase the speed. This is configured under [Setup » Preferences » Engine » Number of processes for Network engine](#).

Resolving Intel MPI incompatibility issues

While multiple Intel MPI versions can be installed on a machine, if any of these versions is incompatible with PIPESIM, such as the Intel MPI version installed with Avocet IAM 2011 (and older), and was installed last (for example, after a compatible Intel MPI version was already installed), PIPESIM network simulations will be unable to run with multiple processors.

To resolve Intel MPI incompatibility, perform the following steps:

1. Uninstall all Intel MPI versions including the incompatible Intel MPI version (4.01.007 and older) and any previously-installed compatible versions. Re-install the compatible version in the PIPESIM install kit.
 2. Uninstall Avocet IAM 2011 (which will also uninstall the incompatible Intel MPI version) and all other Intel MPI versions and upgrade to Avocet IAM 2013 (or newer) which includes a compatible Intel MPI version.
-

Note:

If you do not want to change the Intel MPI version, you still have the option of running the network simulation with only 1 processor by manually selecting this under **Setup » Preference » Engine » Number of processes for Network engine**.

If you have only one (1) Intel MPI version installed and it is the incompatible version, the PIPESIM network simulation will run, but will default to using only one (1) processor.

1.5.2 Network Options

Network Iterations

To configure Network iterations, select **Setup » Iterations**. The parameters are as follows:

Tolerance

The tolerance to converge the network model to. (Default 1%). A network has converged when the pressure balance and mass balance at each node is within the specified tolerance. The calculated pressure at each branch entering and leaving a node is averaged.

Max Iterations

The maximum number of iterations in which to try and determine a solution. The simulation will stop after this number of iterations unless the tolerance has been met. (Default 100)

Background information

The network module uses a GNET algorithm to solve all networks. Reaching a solution involves continually estimating and refining a matrix of results for each branch while simultaneously taking into consideration the many sources of discontinuity within the network. These sources of discontinuity include, dead wells, two phase vertical flow, critical flow, phase changes and flow regime boundaries.

Note: If there is no convergence after completing the specified number of iterations and the last iteration performed is not the best solution, the solver performs another iteration to calculate the best solution from all the iterations. The best iteration is based on where the maximum pressure and flow rate residual errors are the lowest. PIPESIM publishes the data from the best iteration to the network report. When there is no convergence, the report tool displays a warning message that the model did not converge and it lists the tolerance that was achieved.

By default the tolerance for PIPESIM simulations is 1%. Mathematically this means that the simulation will terminate when all of the root mean square errors for pressure and flowrate at nodes are less than 1%. If you decrease this value then you are forcing the solver to do more calculations and produce more accurate results. It should be noted, that for long chains of branches this error may become cumulative, resulting in a much larger overall error across the network

By default the solver will complete a maximum of 100 iterations per simulation. If after 100 iterations no solution meeting the required tolerance has been found then solver will stop and display existing results. If you find that a particular system is not converging then, generally, it is best to relax the tolerance rather than increase the maximum number of iterations.

The solver makes use of convergence techniques that are tuned to suit each individual problem. These routines are based on well accepted mathematical theorems but modified to allow for the discontinuities that might be generated by different flow correlations, as well as those originating from the problem specification. However, as a result of the inherent difficulties associated with sources of discontinuity, you may find that some network simulations converge slowly, or in some rare cases, not at all. In these circumstances, the table shown below is a guide to obtaining satisfactory convergence:

Problem	Cause	Solution
Oscillates about the solution but does not converge	Machine accuracy or other limitation	Relax tolerance if intermediate results is sufficiently accurate
Slow reduction in error followed by a large increase followed by slow reduction, and so on	Discontinuity in chosen flow correlation, not already trapped with the solver.	Use the results from a NOSLIP run as estimates for the simulation using the original correlation
Alternating forward and reverse flow predicted	Too many non-return valves in network	Remove all non-return valves, except in lines where reverse flow is guaranteed.

Fluid Models

This dialog allows fluid models in a network model to be displayed, edited and sorted in a spreadsheet type display. This allows you to view what fluid properties are used by all network wells or sources without having to drill down to the individual fluid model screens.

Setup Fluid Models

The screen allows display and editing of black oil fluid models. If the network is using compositional fluid models, the screen only allows viewing of fluid models being used and selection of either Global or Local fluid models for sources.

The following can be edited:

Default / Local

Select whether to use the default (global) fluid model or the local fluid model defined for the well / source.

Override Wcut/GOR

Select the check box to allow override of the fluid model water cut and gas fraction (GOR/GLR/OGR/LGR).

Local fluid models

You can edit the following only for local fluid models:

Water Cut

fluid model water cut value.

GOR

fluid model gas fraction value, can be GOR, GLR, OGR or LGR.

Gas SG

stock tank gas specific gravity.

Water SG

stock tank water specific gravity.

Oil Gravity

stock (dead) oil density or API.

Dead Oil Visc

the dead oil viscosity correlation to use. Alternatively, specify **User's Data**.

Visc 1 & Visc 2

Two dead oil viscosity values. These values are read only if a dead oil viscosity correlation is selected. If **User's Data** is selected, these two values are user specified.

Temp 1 & Temp 2

Two temperature values at which to define the dead oil viscosity values.

Note: If you attempt to edit a Global fluid model parameter from this screen, the **Global fluid model** dialog boxes open. This is because changing the Global fluid model parameters affects all other sources that are using the Global fluid.

1.5.3 View Options

To change these options, select **View » Options**. This opens the **View Options** dialog, which has the following tabs.

View Properties tab

This controls background color and grid lines.

Background Color

Double-click the color box to change the color. The default is white.

Grid Lines

The parameters are the following:

Draw Grid

turn the back ground grid on and off

Horizontal spacing

distance between the horizontal grid lines (in pixels) if the background grid is displayed.
Default =50.

Vertical spacing

distance between the vertical grid lines (in pixels) if the background grid is displayed.
Default =50.

Color

Color of the grid line, if displayed. Double click the color box to change. Default light gray.

Objects tab

Identifiers

The options for object identifiers are as follows:

Leave Display unchanged

default

Show all identifiers

shows all object identifiers on the schematic

Hide all identifiers

Remove all object identifiers from the schematic

Tool Tip Options

You can switch off Tool tips. This is useful on a large model as having these enabled slows model navigation.

Activate on mouse move

default

Activate on mouse click

Switch off

Red box

Draw red box around incomplete objects

The red box warns the user that a parameter is incomplete. The default = on.

General Tab

This tab appears on most dialogs which show object properties.

Activate/Deactivate

turn the object off (that is remove it from the simulation)

Identifier

The parameters are as follows:

Text box

the object's identifier. This cannot be more than 16 characters, must be unique and must not contain any of the following characters: .\|{}<>|=?" or spaces.

Bounding rectangle

this places a rectangle around the name

Show identifier

this displays the object's identifier

Back Color

color of the bounding rectangle

Position

Screen coordinates of the object. The limits are as follows:

Top left

X= -2,101 Y= 2,089

Top right

X= 2,064 Y=2,084

Bottom left

X= -2,101 Y= -2,089

Bottom right:

X=2,064 Y=-2,0898

Viewing a large model

When the model is too large to fit on the screen, zooming out can change the view. Zoom options are located on the Format tab in the Zoom group.

To zoom into the smallest area that will show all your completions, click **Zoom to Fit**.

To use the zoom out feature, select **View » Zoom Out** or press Shift+X. This reduces the size of the objects accordingly.

- To Zoom out to the maximum extent, use Shift+F or **View » Zoom Full View**.
- To Zoom in, use Shift+Z or **View » Zoom In**.
- To restore the original (default) view setting, use Shift+R or **View » Restore View**.

Viewing flowline data

To view the flowline descriptions for all flowlines, select **Setup » Flowline Properties**. The data displayed here can be changed. If the flowline has been defined using the detailed profile, the data is grayed out.

1.6 Database

1.6.1 New ESP/Pump/Compressor Curve

The user curves are used to determine the performance of the device. The data is stored in an external Access database.

To define a new curve, do the following:

1. Set the device type: **ESP**, **Generic Pump** or **Compressor**. The new device can be based on data from an existing one.
2. Enter the new manufacturer's name.

3. Enter the new model reference.

The defined curve can then be used in the [ESP \(p.235\)](#), [pump \(p.117\)](#), [compressor \(p.91\)](#), or [expander \(p.94\)](#) objects to define the performance. At this stage the actual operating speed, number of stages, and so on, can also be set.

ESP

For an ESP enter the following:

1. The ESP diameter
2. Minimum allowable flowrate
3. Maximum allowable flowrate
4. Base speed that the curves are based upon, typically 60 Hz
5. Base stages that the curves are based upon, typically 1
6. Allowed stages. The stages that are recommended.
7. Press **Next** to enter the curve data in terms of Flow rate, Head and Efficiency.

Generic Pump

For a Generic Pump enter the following:

1. Minimum allowable flowrate
2. Maximum allowable flowrate
3. Base speed that the curves are based upon, typically 60 Hz
4. Base stages that the curves are based upon, typically 1
5. Allowed stages. The stages that are recommended.
6. Press **Next** to enter the curve data in terms of Flow rate, Head and Efficiency.

Compressor/Expander

For a Compressor/Expander enter the following:

1. Minimum allowable flowrate
2. Maximum allowable flowrate
3. % design speed that the curves are based upon, typically 100
4. The curves should be entered for a single stage device. Press **Next** to enter the curve data in terms of rate, Head and Efficiency. Note the units.
5. Once the curve data has been defined, the device's performance curve can be displayed (and printed).

New Reciprocating Compressor Curve

The user curves are used to determine the performance of the device. The date is stored in an external Access database. To define a new curve, do the following:

1. Enter the new manufacturer's name. The new device can be based on data from an existing one.
2. Enter the new model reference.
3. Enter the following:
 - Base Speed
 - Absolute Minimum Suction Pressure
 - Absolute Maximum Suction Pressure
 - Absolute Maximum Capacity
 - Stages
 - Interstage Temperature. The curves should be entered for a single stage device.
4. Press **Next** to enter the curve data in terms of Flowrate, Suction Pressure and Efficiency or Power for various Discharge Pressures, at least two are required. The Discharge Pressure you must supply is the one at which the data for the associated curve is valid for. If the curve is valid for a range of Discharge Pressures, then supply the compressor's maximum discharge pressure. Note the units.

Once the curve data has been defined, the device's performance curve can be displayed (and printed).

The defined curves can then be used within the [compressor \(p.91\)](#) object. Set the **User curves** option and the compressor type to **Reciprocating** to define the performance. At this stage, the actual operating speed and or minimum and maximum operating speeds can also be set.

User Curves

User curves allow the user to enter basic data to represent the following elements:

- Multiphase Booster

ESP and Pumps

The ESP or pump is defined as curve that represents its performance over a range of operating conditions. The following data are required:

- Manufacturer
- Model
- Diameter (ESP only) - used only when a search is performed to locate items that fall within certain ranges
- Min flowrate - minimum recommended flowrate. The performance curve can be constructed outside of this range. Warning messages show where the operating point is outside this limit.
- Max flow rate - maximum recommended flowrate. The performance curve can be constructed outside of this range. Warning messages show where the operating point is outside this limit.
- Base speed at which the performance curve is defined. This can be changed for the simulation.
- Base stages at which the performance curve is defined. This can be changed for the simulation.
- Allowed stages: The number of stages that can be used for this particular item.

- Performance curve
 - Flow Rate - flow rate in flowing conditions (not stock tank)
 - Head -
 - Power - the power requirements
- The defined curve can also be viewed and printed.

The [ESP \(p.240\)](#) or curve can then be used in the simulation.

Compressors

The compressor is defined as a curve that represents its performance over a range of operating conditions. The following data are required:

Manufacturer

a new curve can be based on an existing model.

Model

Min flowrate

minimum recommended flowrate. The performance curve can be constructed outside of this range. Warning messages will be provided where the operating point is outside this limit.

Max flow rate

maximum recommended flowrate. The performance curve can be constructed outside of this range. Warning messages will be provided where the operating point is outside this limit.

Base Speed

speed at which the performance curve is defined. This can be changed for the simulation.

Performance curve

A set (minimum of 3 points) of data comprising the following:

Flow Rate

flow rate in flowing (actual) conditions (not stock tank)

Head

compressor head.

Efficiency

adiabatic efficiency of the compressor.

The defined curve can also be viewed and printed.

The [Compressor \(p.91\)](#) curve can then be used in the simulation.

See also [keywords \(p.677\)](#)

Electrical Submersible Pumps (ESP) Selection

To simulate an ESP, PIPESIM maintains a database of manufacturers and models from which you can select. For each model the diameter, minimum and maximum flowrate and base speed are

provided. A plot of the ESP's performance is also available. If the required ESP is not in the database, you can easily enter the basic data required for it into the database using **Data » New ESP/Pump/Compressor**. See [Data/NewESP-Pump-Compressor Curve \(p.184\)](#).

Overview of the Steps

1. To open the **Tubing** dialog box, double-click the tubing.
2. In the **Artificial Lift** section, select **ESP** for the lift type.
3. Click **Properties**.
4. Select the appropriate manufacturer and model.
5. To select a pump or add your data to the **ESP Selection**, click **Advanced Select** and enter your information.
6. Update the **Design Data** section and the **Calculation Options** section as necessary.
7. To change data on the **Performance Table**, click the Performance Table tab and enter the appropriate information.
8. Click **OK** to save your changes.
9. To view the curves, click **Standard Curves** tab or **Variable Speed Curves** tab.

You can export the data or print the curve.

Selection

When modeling an ESP, it is important that the correct size (expected design flowrate and physical size) ESP is used. A search facility is available, based on these two parameters, to select the appropriate ESP from the database. The search can, if required, be restricted to a particular manufacturer. Pumps that meet the design criteria will be listed.

Stage-by-stage modeling

Stage-by-stage modeling is selected by selecting the checkbox next to the calculate button. Alternatively by inserting Engine Keywords ([PUMP STAGECALCS \(p.701\)](#)) into the model, using the [EKT \(p.94\)](#).

Install a Pump

Once the ESP [manufacturer and model \(p.235\)](#) has been selected from the [database of common ESP's \(p.239\)](#) some parameters can be altered. The performance curves for each model are (normally) based on a Speed of 60Hz and 1 stage.

Design data

Speed

The actual operating speed of the ESP

Stages

The actual number of stages of the ESP

Head factor

Allows the efficiency to be factored (default = 1)

Calculation Options

Viscosity Correction

Allow a viscosity correction factor to be applied to take account of changes to the fluid viscosity by the pressure and temperature.

Gas Separator present

Allow a gas separator to be added (automatically) with an efficiency: Separator efficiency - efficiency of an installed gas separator (default = 100% if installed)

Performance table

The data used to predict the performance of the ESP

Standard Curves

The standard performance curves for the ESP - can be printed/exported

Variable Speed Curves

Variable speed curves at 30 - 90 Hz.- can be printed/exported

ESP Design

The ESP option is selected from the Artificial Lift. To design an ESP the following stages are required:

[Select a Pump \(p.236\)](#)

[Select a Motor \(p.235\)](#)

[Select a Cable \(p.241\)](#)

The ESP should then be installed, added into the tubing, at the required depth. This can either be performed manually or by using the **Install** button. Installing automatically removes any existing ESPs in the tubing. However, any gas lift values or injections points are not removed.

See also: [ESP \[Reda\] web site](#)

ESP System Components: Cable

Cable Selection can be determined after a Pump and motor have been selected.

1. Motor/Cable Selection tab
2. Cable Selection
3. Select Cable

Cable Length

The length of the cable, can be modified

NP Current @ Design Frequency

The [Name Plate] Current at the design frequency. Cannot be changed.

Computed values

Selected Cable

Cable length

Voltage drop

Downhole voltage

Surface voltage

Total System KVA

Design Report

Display a report that details all the selected components of the ESP system.

Errors

Occasionally a pump may not be able to be determined and a Convergence error will be reported. There could be a number of reasons for such an error and the user is advised to view the output report.

Common problems:

1. The system cannot reach the outlet pressure specified. Try increasing the outlet pressure.

1.6.2 New PCP Curve

User defined curves are used to determine the performance of the device. Data is stored in an external Access database. (**Setup Preferences Choose paths**)

To define a new PCP curve, use one of the following methods:

- [Create a new curve \(p.190\)](#) by entering the information in the **PCP Selection** window.
- [Select a PCP from the catalog \(p.191\)](#) and edit the data.

Creating a New Curve

To create a PCP curve, do the following:

1. Select **Data » New/Edit PCP**.
2. Enter the new Manufacturer name.
3. Enter the new Model name.
4. Enter the Diameter.
5. Enter the Nominal Rate.
6. Enter the Base Speed.
7. Click **Next**.
8. On the Performance Table dialog box, enter the Head, Flowrate, Power, and Torque data for the curve.
9. To view the standard curve, click **Next**.
10. If the variable speed curves were calculated, click **Next** to view them.

Once the curve data has been defined, you can display, export, or print the device's performance curve at base speed or at variable speeds (when available). You can use the data within a PCP object.

Selecting a PCP from the Catalog

To select a PCP from the catalog and edit the information, do the following:

1. Select **Data » New/Edit PCP**.
2. Click **Select PCP from Catalog**.
3. When asked whether to continue because switching to the catalog will reset all unsaved data, click **Yes**.
4. To edit the information on the page, click **Copy to User Defined PCP**.
5. Enter your changes.
6. Click **Next**.
7. Edit the information in the performance table, as necessary.
8. To view the curve, click **Next**.

Once the curve data has been defined, you can display, export, or print the device's performance curve at base speed or at variable speeds (when available). You can use the data within a PCP object.

Standard PCP Curves

When entering the data for standard curves, enter the basic data.

PCP

The PCP requires a curve that represents its performance over a range of operating conditions. The following information is required:

- Manufacturer
- Model
- Diameter
- Nominal Rate — maximum flowrate through a PCP at a specified speed and zero pressure differential
- Base speed — testing catalog speed from the manufacturer. Users can input any different working speed (normally in the range of 50–500 RPM), and PIPESIM calibrates the performance curves correspondingly.
- Rod Diameter — used for top drive PCPs
- Performance curve
 - Flow Rate — flow rate in flowing conditions (not stock tank)
 - Head — increased pressure head in feet or meter
 - Power — the power requirements
 - Torque — tendency of a force to rotate the PCP (moment of force)
- The defined curve can also be viewed, exported, and printed.

Progressive Cavity Pump (PCP) Selection

To simulate a PCP, PIPESIM maintains a database of manufacturers and models from which you can select. If the required PCP is not in the database, you can enter the required data into the database using **Data » New / Edit PCP**. [See how... \(p.190\)](#)

Overview of the Steps

1. To open the Tubing dialog box, double-click the tubing.
2. In the Artificial Lift section, select PCP for the lift type.
3. Click **Properties**.
4. Select the appropriate manufacturer.
5. Update the Design Data fields as necessary.
6. To edit the data on the PCP Selection and Performance Table tab, click **Select PCP from Catalog**.
7. To save the model that you defined, change the name of the manufacturer and the model and click **Save**.
8. On the Performance Table tab, update the data as necessary.
9. To see the standard curves, click the Standard Curves tab. You can export or print the data from this tab.
10. To see the variable speed curves, click the Variable Speed Curves tab. You can export or print the data from this tab.
11. Click **OK**.

Selecting a Pump

Once the PCP manufacturer and model have been selected from the database of common PCPs, you can alter some of the parameters. Typically, the performance curves for each model are based on a Speed of 100 RPM.

PCP Design Data

The following parameters must be specified.

Speed

The speed at which the PCP is expected to run.

Top Drive

Specifies whether the drive is a top-drive or a bottom-drive. This is used for torque calculations.

Rod Diameter

Specifies the rod diameter (top drive only.)

Head Factor

Allows the pump head to be adjusted to better match field performance data or account for wear. The default is 1.

Calculation Option

Viscosity Correction

Allows a viscosity correction factor to be applied to account for reduced slippage.

Gas Separator present

Allows a gas separator to be modeled.

Separator Efficiency (%)

Specifies efficiency of the gas separator. The default is 100%.

Errors

If the PCP is modeled outside its intended operating range, warning or error messages will be reported. For more details, please view the output report. ([Reports » Output file](#))

1.6.3 Gas Lift Valve List

The database contains a list of available valves . You can add valves to this.

To add a valve to the database, do the following:

1. Select **Data » New/Edit Gas Lift Valve....**
2. The Gas Lift Valve database opens, for checking or editing. Click **Add** and enter values for the following parameters :
 - Manufacturer – the valve maker
 - Series – the valve series or model name (for example BK-1 valve)
 - Port Size – the diameter of the valve port
 - Type – the valve type - IPO, PPO-N, PPO-S, Orifice or Dummy valve
 - Outer Diameter – the valve outside diameter - valves are either 1" or 1 1/2"
 - Port Area – the cross sectional area of the port
 - Bellows Area – the cross sectional area of the bellows
 - Discharge Coefficient – a discharge coefficient for the Thornhill Craver equation, used to calculate valve gas throughput (given injection and production pressure)
 - DP [Delta P] to fully Open – this is only needed for Diagnostics operation if the throttling behavior of the valve is to be modeled. It is the difference between the production pressure when the valve is fully open to fully closed (for a fixed injection pressure).
 - Inactive – if this is **Yes**, the valve is not available for a new design
3. Save the new valve.

Note: To edit an entry, select it and click **Edit**.

1.6.4 Flow Control Valves

Down hole Flow Control Valves (FCVs) allow 'intelligent' or 'smart' wells to be modeled. The methodology implemented provides a simple way of modelling single branch (that is non multilateral) intelligent wells where the FCVs are located close to the reservoir.

FCVs can restrict the completion flowrate through the system, but they are only available for vertical completions. By default, a completion does not have an FCV associated with it.

The purpose of an FCV is to provide a restriction to fluid flow, thereby reducing the productivity or injectivity of a given completion. They are useful in a model containing multiple completions (although there is nothing stopping you from adding one to a single-completion model) to distribute flow from or to each completion in the desired manner.

An FCV is very similar to a choke. Like a choke, it can be modeled as a fixed-size orifice, in which form it presents a restriction to flow that results in a pressure drop that increases as flowrate increases. Unlike a choke however, a maximum flowrate can also be specified: this is applied to the completion, and if necessary the choke bean diameter is reduced to honour the limit. The choke diameter and flowrate limit can be applied separately or together. If both are supplied, they are treated as maximum limits.

How to add an FCV to a completion

To add an FCV to a vertical completion, do the following:

1. Select the **Flow Control Valve** check box in the **Completion** dialog.
2. Click the **FCV Properties** button.
3. The **Flow Control Valve** dialog opens.
4. Select **Generic valve** or **Specific valve** and enter values as described below:

Generic valve

Specify the **Equivalent Choke Area**, **Gas** and **Liquid Flow Coefficients**, and choice of **Gas Choke Equation** method. The choke area can be omitted if a Maximum Flowrate is specified (see below). If it is present, the FCV is modeled with that choke area, but if the resulting flowrate exceeds the limit, the area is reduced to honor the limit.

Specific Valve

Choose from the list of available valves. To add a new valve to the database, use [Data > New/Edit Flow Control Valve Data](#). Select the required **Manufacturer**, **Valve Type**, and **Part Number**. Many of the Specific Valves are multi-position devices: they allow the effective choke area to be selected from a range of pre-installed fixed chokes. If a flowrate limit is supplied, the simulation will select the choke position required to honor the limit. Since the choke area cannot be calculated to match the limit exactly, this will usually result in the flowrate being lower than the limit. The valve position must be specified. The FCV will be modeled with the corresponding choke area, but if the resulting flowrate exceeds the limit, a lower position number will be used. Valve positions are numbered in order of increasing choke size, starting with position zero, which usually specifies a diameter of zero to allow the valve to be shut. An FCV may have up to 30 positions.

1.7 Operations

1.7.1 Common Operations

Check model

Allows all the data inputs and connectivity to be checked and verified before any simulation is performed. No lone nodes are allowed in a single branch model. Any errors or omissions will be reported.

To check a model, select **Operations** » **Check Model**.

Run Model

The **Run Model** option validates the system, creates the necessary engine input files, and then runs the model.

To use **Run Model**, do one of the following:

- Click the **Run Model** button in any of the operating mode dialog boxes that are accessed using the **Operations** Menu.
- Select **Operations** » **Run Model** .
- Click the **Run model** button



on the toolbar.

Abort Run

This allows a simulation to be stopped immediately. This kills the engine process. However, if the engine is being [run via a batch file \(p.178\)](#) this will only kill the batch process, not the engine. In this circumstance, use [Terminate Run \(p.195\)](#).

To stop a simulation, select **Operations** » **Abort Run**.

Terminate Run

This operation terminates a simulation in a tidy manner. A message is sent to the engine, which terminates at the next convenient point.

Terminating the run using this method allows the engine to delete temporary files it has created. Simply closing the simulation window or [aborting the run \(p.195\)](#) does not clean up these files, which may eventually clutter up the hard disk.

To stop a simulation cleanly, select **Operations** » **Terminate Run**.

1.7.2 Single Branch

System Analysis

Use the **System Analysis** operation to determine the performance of a given system for varying operating conditions on a case-by-case basis. An example is Pressure or Temperature Profiles where performance is evaluated on a point-by-point basis.

How to perform a System Analysis

To perform a system analysis, do the following:

1. [Create and save your model. \(p.33\)](#)
2. Select **Operations » System Analysis**.
3. Set the **Calculated Variable** to [Inlet Pressure \(p.119\)](#), [Outlet Pressure \(p.119\)](#), [Flowrate \(p.120\)](#) or [User Variable \(p.211\)](#). If an unselected variable has an adjacent data-entry box, enter its value(s)
4. Choose the X-axis variable: this is a two stage process. First, choose the desired model **Object**, by clicking on the down-arrowhead in the topmost combo-box in the **X-Axis Values** column (just underneath the **Range...** button), and selecting one. Repeat this for the next lower combo box to choose a **Sensitivity variable name**. Check the units revealed in the next box, and adjust if desired.
5. Add the desired values of the X-axis into the spreadsheet cells for the X-axis. To enter a range of values with a starting value, end value and increment, use the “Range...” button.
6. (Optional) Select [sensitivity variables \(p.211\)](#) by repeating steps 4 and 5 for the columns marked **Sens Var 1**, **Sens Var 2**, etc.
7. If one or more sensitivity variables are defined, choose how they should be combined with the X-axis, by selecting a radio button in the **Sensitivity Variables** box at top right of the dialog. There are three choices:
 - **Permuted against each other** runs a case for every combination of X-axis and all sensitivity variables. This produces a plot with most lines (and takes the longest time to run).
 - **Change in step with Sens Var 1** runs a case for every combination of X-axis and **Sens var 1**, with the remaining sensitivity variables following **Sens var 1** in step. This produces a plot with the number of lines equal to the number of **Sens var 1** values.
 - **Change in step with X-axis** runs a case for every X-axis variable value, with all sensitivity variables following the X-axis in step. This produces a plot with just one line (and takes the least time to run).
8. Select the **Active** check box for each sensitivity variable you want to use in this simulation.
9. Click **Run Model**.

Pressure/Temperature profile

Pressure or temperature profiles can be generated as a function of distance along the system. Both temperature and pressure profiles are generated on a node-by-node basis for the system.

Perform the following basic steps to determine the pressure or temperature profile along the system:

1. Build the required model, [pipeline tools \(p.91\)](#) or [well \(p.51\)](#).
2. Select **Operations » Pressure/Temperature profile**.
3. Determine the boundary condition to compute.
4. (Optional) Select and enter any sensitivity parameters.
5. [Run \(p.195\)](#) the operation.
6. [Save the model \(p.35\)](#).

The operation can be run in any of the following modes:

- Calculate [inlet pressure \(p.119\)](#) given outlet pressure and flow rate
- Calculate [outlet pressure \(p.119\)](#) given inlet pressure and flow rate
- Calculate [flowrate \(p.120\)](#) given inlet pressure and outlet pressure
- Calculate the value of a [User Variable \(p.211\)](#) given inlet pressure, outlet pressure and flow rate

How to perform a pressure or temperature profile

To perform a pressure or temperature profile, do the following:

1. [Create and save your model. \(p.33\)](#)
2. Select **Operations » Pressure/Temperature profile**.
3. Select the property to calculate: [inlet pressure \(p.119\)](#), [outlet pressure \(p.119\)](#), [flow rate \(p.120\)](#), or [User Variable \(p.211\)](#).
4. (Optional) Select [sensitivity variable \(p.211\)](#) and supply values for it.
5. Select the desired initial default profile plot. You may have either a pressure or temperature profile, plotted against either total distance or elevation. This selection affects only the initial view of the profile plot; once the operation is complete the plot may be changed to display any desired profile.

Note: If unified multiphase models are to be compared for all inclination angles, select "horizontal" and change the swap angle to 90 degrees from horizontal. ([Setup Flow Correlations](#))

6. Click **Run Model**.

Flow correlation comparison

Use this operation to match test data against each multiphase flow correlation for a particular system, to determine the most suitable correlation for each system model.

How to perform a flow correlation match

To perform a flow correlation match, do the following:

1. [Create and save your model. \(p.33\)](#)

2. Select **Operations** » **Flow Correlation Comparison**.
3. Select the property to compute, [inlet pressure \(p.119\)](#), [outlet pressure \(p.119\)](#) or [flowrate \(p.120\)](#). If the inlet pressure is the same as the pressure entered in a source object ([completion \(p.398\)](#) or [source \(p.118\)](#)) it can be omitted.
4. Enter the measured data: pressure and/or temperature.
5. Select the type of correlation to examine; **Horizontal** or **vertical**.
6. Select the multiphase flow correlations to examine.
7. Click **Run Model**.

See also [Other Variable \(p.211\)](#).

Adjusting parameters to match measured data

A better match can be achieved by [automatically adjusting parameters \(p.198\)](#) (such as flow correlation factors).

Data matching

This option allows you to select parameters that will be automatically adjusted to match measured data for a particular system.

See also [User Variable \(p.211\)](#) which can be used to select a single parameter that will be automatically adjusted to match the boundary conditions.

How to perform a data match

To perform a data match, do the following:

1. Create and save your model. [See how... \(p.33\)](#)
2. Add and save measured pressure

$$\{P_M\} \quad \text{Eq. 1.4}$$

and temperature

$$\{T_M\} \quad \text{Eq. 1.5}$$

data for your model. For tubing, data can be added using the **Data** menu. For flowlines and risers, measured data can be added to the **properties** tab of the detailed view.

3. From the toolbar, select **Operations** » **Data matching**.
4. Select up to five of the following parameters:
 - U-value multiplier: multiplies all user defined u-values. It can only be selected if measured temperature data is supplied.
 - Vertical flow correlation friction factor. This can only be selected if measured pressure data is supplied.
 - Vertical flow correlation holdup factor. This can only be selected if measured pressure data is supplied.
 - Horizontal flow correlation friction factor. This can only be selected if measured pressure data is supplied.

- Horizontal flow correlation holdup factor. This can only be selected if measured pressure data is supplied.
5. Set limits for each parameter selected. Do not use extreme values, as these may cause convergence problems. Suggested limits are as follows:
 - $0.01 < U\text{-value multiplier} < 100$
 - $0.1 < \text{Friction factor} < 10$
 - $0.1 < \text{Holdup factor} < 2$
 6. (Optional) Select **Flow Correlations**. If flow correlations are not specified here, Data matching is performed with the flow correlations specified using **Setup » Flow Correlations**.
 7. Select the property to compute, [inlet pressure \(p.119\)](#), [outlet pressure \(p.119\)](#), or [flowrate \(p.120\)](#). If the inlet pressure is the same as the pressure entered in a source object ([completion \(p.398\)](#), [source \(p.118\)](#)), it can be omitted.
 8. Select weight factors. If both pressure and temperature data have been specified then use the weight factors w_P and w_T to set the relative importance of the pressure and temperature error terms. PIPESIM will minimize the total error term $RMS = w_P \cdot RMS_P + w_T \cdot RMS_T$, where the pressure error term is given by:

$$RMS_P = \sqrt{\frac{1}{n_P} \cdot \sum (P - P_M)^2} \quad \text{Eq. 1.6}$$

and the temperature error term is given by:

$$RMS_T = \sqrt{\frac{1}{n_T} \cdot \sum (T - T_M)^2} \quad \text{Eq. 1.7}$$

9. [Run \(p.195\)](#) the model. The optimizer performs a number of PIPESIM runs, until it has minimized the RMS value. The accuracy of the optimization and the number of iterations allowed can be controlled by using the [OPTIMIZE \(p.752\)](#) keyword, which can be entered using the [Engine Options \(p.172\)](#) dialog.

For each set of flow correlations selected, two sets of results will be tabulated, an Initial run using default parameters, and an Optimized run, using fitted parameters. The pressure, temperature and total RMS values will be displayed.

10. Select the run that gives the best fit, by clicking on the run number in the left hand column. Then click on the **Save Selected Results**. This updates the parameters in your model.

Note: The run with the smallest RMS gives the best fit, but it is worth considering how far the multipliers have been changed from their default value (1). A flow correlation where the multipliers are close to 1 may be a better model than one where extreme values are needed to give a good fit.

See also the [OPTIMIZE \(p.752\)](#) keyword.

Nodal Analysis

Nodal (or system) analysis in PIPESIM is defined as solving the total-producing-system by placing nodes at the reservoir sand-face, the well tubing, the flowline and the separator.

A node is classified as functional when a pressure differential exists across it. In nodal analysis, the producing system is divided into two halves at the solution node. The solution node is defined as the location where the pressure differential upstream (inflow) and downstream (outflow) of the node is zero. This is represented graphically as the intersection points of the inflow and outflow performance curves. Solution nodes can be judiciously selected to show the effect of certain variables such as inflow performance, perforation density, tubing IDs, flowline IDs and separator pressures.

The solution node can be placed between any two objects, that is bottom hole (between completion and tubing), wellhead (between tubing and choke), riser base (between flowline and riser), and so on. Use the [Nodal Analysis point \(p.88\)](#) for this.

How to perform a Nodal Analysis

To perform a Nodal Analysis, do the following:

1. [Create and save your model \(p.33\)](#)
2. [Build the well performance model \(p.51\)](#)
3. Determine the Nodal Analysis point and insert the [Nodal Analysis point \(p.88\)](#) object into the model (this is a node type object)
4. Select **Operations » Nodal Analysis**.
5. Determine the inflow and outflow parameters.
6. Once in the **Nodal Analysis Data** screen, set the [outlet pressure \(p.119\)](#) and the [maximum flowrate \(p.120\)](#) that the well will attain (this is to limit the outflow curves).
7. Set any limits (optional) using the [Limits \(p.200\)](#) button. Limit the extent of the resulting plot.
8. Select the Inflow and outflow [sensitivity object and variables \(p.211\)](#) and enter the data.
9. [Run \(p.195\)](#) the model.
10. [Save the model \(p.35\)](#).

Note: If a gas system is being modeled then the [liquid loading \(p.90\)](#) point, for each outflow curve, will automatically be displayed.

Nodal Analysis Limits

Sets limits on the resulting Nodal Analysis plot.

- The maximum flowrate to be used (optional). This always applies to the outflow curves, and optionally to the inflow curves, see below. If left blank, the outflow curves will extend to the maximum AOFP of the inflow curves or to the Maximum Pressure limit (see below).
- Number of points on each inflow curve. Default = 20, maximum 200
- Number of points on each outflow curve. Default = 20, maximum 200.

- Rate Limit option for the inflow curves: these can either obey the maximum flowrate, or be allowed to extend to the AOFP (the rate where the curve meets the X-axis).
- Pressure limit option for the outflow curves: these can either be limited to the Maximum Pressure (see below), or be allowed to extend to the flowrate limit.
- Maximum pressure for outflow curves. If supplied, the outflow curves will extend to this pressure, or the maximum rate, whichever gives the smallest curve. If left blank, a default value will be assumed, this will be calculated at run-time to be the highest pressure on any of the inflow curves, plus 50%. Either way, the pressure limit option (see above) can be used to make the limit be ignored.

Operating Points

The intersection of one inflow curve and one outflow curve is known as an Operating Point.

As of PIPESIM release 2010.1, the nodal analysis operation generates the operating points and displays them on the plot.

While it is possible to infer the system flowrate geometrically from the line intersections alone, it is more accurate and far safer to calculate the flowrate by simulating the system end-to-end, which PIPESIM is well designed to do. The resulting pressure and flow rate is displayed on the Nodal Analysis graph as an Operating Point (usually a small circle marker). This explicit calculation ensures the inflow and outflow fluid properties and temperature are identical, thus eliminating the possibility of a mismatch and consequent error in answer interpretation.

Operating points are generated for each permutation from the lists of inflow and outflow sensitivity variables. However, it is possible to set up the sensitivities so that some combinations are invalid, and these do not result in operating points being generated and displayed. For example, if you set both inflow and outflow sensitivity to the fluid watercut, most of the permutations will be invalid, because the fluid at the intersection cannot have 2 different values for watercut. With Operating point generation enabled, the valid intersections are clearly distinguishable from the invalid ones: operating points will only be generated for "valid" combinations.

Sometimes it will happen that the displayed operating point does not coincide with the geometric intersection. The cause of this will always be that the inflow or outflow fluid properties or temperature do not match that of the operating point. The fact that the mismatch is evident should be regarded as a feature, not a bug, and should alert the user to a problem or condition that requires particular caution and attention.

With operating point generation enabled, the profile plot file will contain valid profile plots for each operating point: these can be viewed by selecting **Reports » Profile Plot**. If you do not want operating points to be generated, use the OPPOINTS= subcode of **NAPLOT**. (p.739)

Display Liquid Loading Point

The [liquid loading point \(p.396\)](#), for a gas system, is automatically displayed when a [Nodal Analysis \(p.200\)](#) plot is requested. The liquid loading point will be displayed for all outflow curves.

The liquid loading point of a given tubing string is the point at which the reservoir energy is not capable of overcoming the frictional and hydrostatic losses of the given tubing string as a function of wellhead pressure. Liquid loading in a tubing string is identified when the corresponding Tubing Performance Curve (TPC) slope approaches zero.

However, in low rate shallow gas wells with high LGRs, it becomes difficult to identify the "minimum rate to lift liquids" in a given TPC plot. The Turner et al correlation can be used to identify the minimum gas producing rate that is required to keep a well unloaded. This minima will, of course, be approaching the "unstable flow point" of a given tubing configuration.

Horizontal length

Optimum Horizontal Well Length analysis accurately predicts the hydraulic wellbore performance in the completion. It is an integral part of PIPESIM's reservoir-to-surface analysis.

The technique subdivides the horizontal completion into vertical cross-sections and treats flow independently from other cross-sections. This multiple source concept leads to a pressure gradient from the blind-end (toe) to the producing-end (heel), which, if neglected, results in over-predicting deliverability. The reduced drawdown at the toe results in the production leveling-off as a function of well length. It can be shown that drilling beyond an optimum length would yield no significant additional production.

Several Inflow Performance Relationships are available. These are solved with the wellbore pressure drop equations to yield the changing production rate along the well length.

How to perform a Horizontal length optimization

To perform a Horizontal length optimization, do the following:

1. [Create and save your model \(p.33\)](#). This must include a horizontal well completion.
2. Select **Operations » Optimum Horizontal Well Length**.
3. Set the outlet pressure and the completion lengths to examine.
4. Click **Run Model**.

Reservoir tables

The reservoir simulator interface allows you to create tabular performance data, to a file, for input into a reservoir simulation model.

It is often necessary, for the purpose of reservoir simulation, to generate VFP curves for input to a reservoir simulator program. The VFP curves supply the simulator with the necessary data to define bottom hole pressure and tubing head pressures as a function of various parameters such as flow rate, GOR or GLR (where applicable), watercut, surface pressure and the injection gas rate. Further choices of input parameters are available through the Expert mode. The effects of variations of up to five parameters can be investigated and reported. Tabular data is then created in a format specific to the reservoir simulator selected.

The reservoir simulator interface allows you to write tabular well performance data (in the form of bottom hole pressures) to a file for input into a reservoir simulation model. Currently, the following reservoir simulators are supported:

- ECLIPSE
- PORES
- VIP
- COMP4
- MoRes (Shell's In-house simulator)

All combinations of the variables input by you will be used to generate the tables.

You may wish to model networks in their reservoir simulator, by generating VFP curves items of well tubing, flowline or riser. This may not result in an accurate model of the surface network as temperatures at network connections will not be modeled correctly.

It is not recommended to generate system plots after running this operation. For system plots, system analysis operation should be performed instead of reservoir tables.

How to create reservoir look-up tables

To create reservoir look-up tables, do the following:

1. [Create and save your model \(p.33\)](#). This should include a completion and tubing.
2. [Build the well performance model \(p.51\)](#).
3. Select **Operations » Reservoir tables**.
4. Enter the data for the sensitivity variables.
5. Select the reservoir simulator.
6. Select well type if the model does not have any completion. Otherwise, injection well is selected by PIPESIM if the model has a generic source or production well is selected if the model has no generic source. The well type cannot be changed by the user when the model has a completion..
7. For ECLIPSE simulator, you can set the following additional options
 - a. For production wells, you can check the “additional temperature table” box to generate a temperature VFP table in addition to the pressure VFP table.
 - b. If you would like to specify an elevation to be written in the VFP table, enter the user-specified bottom hole datum depth in “User BH Datum Depth” input area. The input value cannot be negative. (The default value in output VFP table is the total elevation change from inlet to outlet if this area is left empty)
8. Enter the required data.
9. Click **Run Model**.
10. [Save the model \(p.35\)](#).

The resulting ASCII file can be used directly by the reservoir simulator. The file contains the data in the required format with the following file names:

- For ECLIPSE simulator, the files are named <filename>.VFP<PROD|INJ>.<BHP|TEMP>.<tabnum>.txt, where <filename> is the base model name, <PROD|INJ> indicates Production or Injection well, <BHP|TEMP> indicates BHP or Temperature VFP table, and <tabnum> is the table number to be included in ECLIPSE file.
- For other simulators, the files are named <filename>.t<tabnum>, where <filename> is the base model name and <tabnum> is a two-digit table number.

Well Performance Curves

This operation creates a well [head] performance curve file. This file can be utilized in the network model to represent the well's performance using the [wells offline \(p.40\)](#) option. This enables the network to be solved faster.

How to create a well performance curve

To create a well performance curve, do the following:

1. [Create and save your model \(p.33\)](#).
2. Select **Operations » Well Performance Curves**.
3. (Optional) Select up to five [sensitivity variables \(p.211\)](#). The actual value to use for each sensitivity variable has to be set before the network simulation.
4. Click **Run Model**.

An ASCII file of <model name>.pwi is created in the model directory. This file can be utilized by the [well curves \(p.40\)](#) feature in the **Network** module.

Gas Lift rate vs Casing head pressure

This operation calculates the gas lift injection rate and resulting production flowrate as a function of casing head pressure (that is, injection pressure).

The Gas lift injection rate and production rates are calculated over a user-specified range of casing head pressures. The operation assumes the operating valve is an orifice and the injection depth is defined as the gas lift injection point in the tubing description.

The result of this operation identifies the relationship between gas lift injection rate and casing head pressure. This may be particularly useful for wells where the method of gas lift rate control may be by adjustment of casing head pressure.

The resulting performance curve may also be used by ProdMan to provide casing head pressure boundary conditions for gas lift allocation and optimization.

How to perform a Casing Head Pressure analysis

To perform a Casing Head Pressure analysis, do the following:

1. [Create and save your model. \(p.33\)](#)
2. Select **Operations » Gas Lift Rate vs Casing Head Pressure**.
3. Enter the following data in the dialog:

- Tubing

System Outlet Pressure

pressure downstream of the last object in the model.

- Gas Lift

Max Available Gas Rate

maximum quantity of lift gas that you want to inject.

Gas Temperature

lift gas temperature at the casing head

Orifice Diameter

orifice diameter of the gas lift valve.

Cv

Cv of the gas lift valve

- Casing Head pressure

Min. Casing Head Pressure

minimum casing head pressure to use to determine the well's flowrate

Max Casing Head Pressure

maximum casing head pressure to use to determine the well's flowrate

Step Size

The flowrate is computed from the minimum casing head pressure to the maximum casing head pressure, in increments of this value.

4. Click **Run Model**.

Effect of gas lift rate on a well

To analyze the effects of gas lift rate on the casing head pressure for a well, work through the following basic steps:

1. [Build the well performance model \(p.50\)](#).
2. Ensure that the gas lift depth and gas lift quantity has been set.
3. Select **Operations » Gas Lift Rate vs Casing Head Pressure**.
4. Enter the required parameters.
5. [Run \(p.195\)](#) the operation.
6. [Save the model \(p.35\)](#).

Artificial Lift System Performance curves

The **Artificial Lift Performance** operation analyzes the effects of artificially lifting the well (by gas lift, ESP lift, or PCP lift). PIPESIM generates lift performance curves of gas lift injection rate, ESP speed versus gross liquid flowrate, or PCP speed versus gross liquid flowrate from the standard system model data. The performance curves can also be created with sensitivity analysis on various parameters, such as wellhead pressure, watercut, tubing ID, and flowline ID. [More details \(p.213\)](#).

Wax Deposition

To access this area, select **Setup » Wax Properties**.

The following proprietary methods are available:

- [Schlumberger DBR singlephase Method \(p.210\)](#) - available to anyone using an additional license
- [Schlumberger DBR multiphase Method \(p.210\)](#) - available to anyone using an additional license
- [Shell Method \(p.206\)](#) - only available to Shell companies
- [BP Method \(p.208\)](#) - only available to BP companies

Wax Deposition Limits

These set limits for the wax deposition calculations. To access this area, select **Operations** ➤ **Wax Deposition** and click **Limits**.

General

The following options are available:

Start/Restart time

The starting/restarting time

Reporting interval

The interval between reporting steps. This can be set independently of the timestep size to allow a number of timesteps to occur with no reported output, if desired. The timestep size will be adjusted to ensure that one ends at each report interval, in order to allow the report to be written.

Termination Mode

The simulation will finish when the first stopping criterion is met. The stopping criteria may be any of the following:

End time

The finish time for the simulation, if no other stopping criterion is met

Maximum Pig DP / Maximum Wax Volume

The maximum delta pressure available to push a wax removal scraper pig through the line. The simulation will terminate early when sufficient wax has deposited to cause the specified DP to occur.

Maximum Wax Thickness

An upper limit in the thickness of the wax deposit anywhere in the system

Minimum Production

A lower limit for system stock-tank liquid/gas/mass rate

Maximum System DP

An upper limit on the Delta Pressure between system inlet and outlet

Timestep Calculation criteria

There is just one parameter currently:

Step size

The time step size

Shell Wax Method

Wax Properties Setup

To access this area, select **Setup** ➤ **Wax Properties** and select **Shell**. The dialog allows you to change data for prediction of wax in the model.

Wax Properties - required

Density

Wax density (Must be supplied if the max. wax plug DP/max. volume stopping option is used. Reasonable value: 55 lb/ft³)

Thermal Conductivity

Wax thermal conductivity. (Reasonable value: 0.15 Btu/hr/ft²/F)

Yield Strength

The yield strength of the deposited wax. (Reasonable value: 0.3 psi).

CWDT (critical wax deposition temperature)

A table of pressures and deposition temperatures may be supplied.

Rate model #

Deposition rate model number. Currently there is only one rate model, number 1.

Modeling Parameters

A table of modelling parameters temperatures, coefficients A and B may be supplied.

Wax Deposition Limits

Sets limits for the wax deposition calculations.

General

The following options are available:

Start/Restart time

The starting/restarting time.

Reporting interval

The interval between reporting steps. This can be set independently of the timestep size to allow a number of timesteps to occur with no reported output, if desired. The timestep size will be adjusted to ensure that one ends at each report interval, in order to allow the report to be written.

Termination Mode

The simulation will finish when the first stopping criteria is met. The stopping criteria may be any of the following:

End time

The finish time for the simulation, if no other stopping criteria is met.

Maximum Pig DP / Maximum Wax Volume

The maximum delta pressure available to push a wax removal scraper pig through the line.

The simulation will terminate early when sufficient wax has deposited to cause the specified DP to occur.

Maximum Wax Thickness

An upper limit in the thickness of the wax deposit anywhere in the system

Minimum Production

A lower limit for system stock-tank liquid/gas/mass rate.

Maximum System DP

An upper limit on the Delta Pressure between system inlet and outlet.

Timestep Calculation criteria**Minimum step**

The minimum time step size.

Relaxation parameter

The relaxation factor for automated timestep adjustment. Must be a real number between 0 and 1 — higher values favour the new value, lower the old.

Step size

The time step size.

DP Factor

Fraction of the pressure drop change allowed with the new timestep.

Minimum Dx

The minimum allowable increase in wax ID.

Set Dx

The maximum increase in wax ID.

HTC limit

Controls the application of the Heat Transfer Coefficient limit on the timestep size.

BP Wax Method***Wax Properties Setup***

To access this area, select **Setup** » **Wax Properties** and select **BP**. This dialog allows you to change or input data for prediction of wax in the model.

Wax Properties - required**Conductivity multiplier****Yield Strength**

Wax thermal conductivity. (Reasonable value: 0.15 Btu/hr/ft²/F)

Properties Filename

File that contains the wax properties data - *.thm file.

Diffusion Coefficient Method

Diffusion coefficient method. Can be:

- Wilke-Chang
- Hayduk-Minhas
- User-supplied (with a diffusion coefficient multiplier).

Diffusion Coefficient Multiplier

Molecular Diffusion coefficient multiplier (number between 0.1 to 1).

Oil Fraction in Wax

Oil fraction in the wax (number between 0 and 1).

Roughness Multiplier

Roughness multiplier (number between 0 and 1).

Shear Multiplier

Shear reduction multiplier (number between 0 and 1) to simulate wax stripping.

Wax Deposition Limits

Sets limits for the wax deposition calculations.

General

The following options are available:

Start/Restart time

The starting/restarting time.

Reporting interval

The interval between reporting steps. This can be set independently of the timestep size to allow a number of timesteps to occur with no reported output, if desired. The timestep size will be adjusted to ensure that one ends at each report interval, in order to allow the report to be written.

Termination Mode

The simulation will finish when the first stopping criteria is met. The stopping criteria may be any of the following:

End time

The finish time for the simulation, if no other stopping criteria is met.

Maximum Pig DP / Maximum Wax Volume

The maximum delta pressure available to push a wax removal scraper pig through the line.

The simulation will terminate early when sufficient wax has deposited to cause the specified DP to occur.

Maximum Wax Thickness

An upper limit in the thickness of the wax deposit anywhere in the system

Minimum Production

A lower limit for system stock-tank liquid/gas/mass rate.

Maximum System DP

An upper limit on the Delta Pressure between system inlet and outlet.

Timestep Calculation criteria

Step size

The time step size.

Schlumberger DBR Wax Methods

Two methods are available; a single phase and a multi-phase method.

Schlumberger DBR wax deposition model

Wax Properties Setup

To access this area, select **Setup** » **Wax Properties** and select **Schlumberger**. The dialog allows you to change or input data for prediction of wax in the model.

- File that contains the wax properties data,- *.DBRWax file. This file is generated using a third party package DRR Solids version 4.1 and above.
- Properties are read from the data file and can be viewed using a text editor. Properties can be over written by entering values in the dialog. The use of multipliers can be switched on or off by selecting the tick box.

Wax Properties Sensitivity

It is possible to sensitize on some of the wax property variables when using the DBR wax deposition models. For both single-phase and multiphase models, the following properties are available for sensitivity:

Sensitivity variable	Acceptable range (min, max)
Shear reduction multiplier	[−10 : 10]
Molecular diffusion multiplier	[−10 : 10]
Oil fraction in wax	[0 : 0.99]
Wax surface roughness	[1E-8 : 0.99999] cm or [3.933701E-9 : 0.3937] inch
Wax thermal conductivity	[0 : 10] W/m/K or [0 : 5.78035] BTU/hr/ft/F

For all the sensitivity variables given above, the default value used in the code is the one specified in the .DBRWAX property file.

For the multiphase DBR wax deposition model, it is possible to sensitize on extra properties which are:

Sensitivity variable	Acceptable range (min, max)
Shear stress coefficient	[-5 : 5]
Shear factor coefficient for porosity	[-5 : 5]
Temperature factor coefficient for oil fraction	[-5 : 5]
Liquid holdup coefficient	[0 : 5]

Wax Deposition Limits

This dialog sets limits for the wax deposition calculations

[Limits \(p.206\)](#)

Sensitivity variables

One of the application's strongest benefits is its ability to perform sensitivity studies on a single or a number of variables at a time.

Important: Some variables are available many times in the sensitivity list (especially when using System Analysis) or in the single branch task window in general. If you set sensitivity for duplicate variables, the system does not detect the duplicates and will not work. For example, sensitizing on the system inlet pressure and on the bottom-most completion pressure will give inconsistent results. Similarly, when using sensitivity and calculated variable in an operation, the system will not detect duplicates. In a case such as this one, you still need to specify the outlet pressure under calculated variable even if you are sensitizing on it.

To perform sensitivity:

1. Select the appropriate operation, (P/T profile, system analysis, and so on.)
2. From the **component** row, use the drop down list to select the object that the sensitivity is to be performed on, that is System data, completion, flowline, and so on. The list of objects displayed here will reflect the names of each individual object in the model.
3. From the **variable** row, select the variable from the chosen object, that is rate, well PI, flowline ID, and so on. The list of variables displayed here will reflect the object chosen in step 2.
4. From the **units** row, select the units for the variable.
5. Enter the date for the variables from row 1. If a set of data is equally spaced, that is 5, 10, 15, 20, 25, 30, 35, then the **Range** button can be used. In this example enter, the start and end value and the step size for example 0, 35, step 5 and press **Apply**.
6. To disable the data in a column, that is perform the operation but ignore a particular sensitivity variable, then deactivate the **Active** check box for the required column.
7. Repeat the above process for all the required sensitivity variables.

User variable

In an operation, if you wish to specify all three of [inlet pressure \(p.119\)](#), [outlet pressure \(p.119\)](#) and [flowrate \(p.120\)](#), you must also tell the engine how to achieve the specified outlet pressure. This is done by defining the **User Variable**. This is similar to a [sensitivity variable \(p.211\)](#), but instead of requiring you to provide a series of values for it, its value is calculated as part of the simulation.

Note: When the engine performs calculations on a user-defined variable, it assumes that the inlet pressure and flowrate are fixed; therefore, the user-defined variable cannot be the inlet pressure nor the flowrate.

You must choose an **object** and a **variable**, whose value will have an effect on the system outlet pressure. For example, in a production well model, a choke positioned at the wellhead could be chosen as the object, and the Bean Diameter as the variable. Any object and any variable can be chosen, as long as you consider it will have an effect on the system outlet pressure.

You must specify the allowable **maximum** and **minimum** values for the variable, and the **proportionality relationship** (that is, whether an increase in the variable's value causes an increase or a decrease in outlet pressure). If **Direct** is selected, the outlet pressure is assumed to be directly proportional to the variable, and will increase with it; for example, the choke bean diameter would behave like this. If **Inverse** is selected, the outlet pressure is assumed to be inversely proportional to the variable, and will decrease when the variable is increased. An example of this might be the watercut of a black oil fluid in a production well: as watercut increases, the well's static DP increases and hence its outlet pressure decreases.

Sometimes, depending on the choice of object and variable, the proportionality relationship can be difficult to predict. For example if **Tubing Inside Diameter** is used in an oil production well, you would expect outlet pressure to increase as diameter is increased, when starting from a small diameter value. However, once the diameter exceeds a certain critical value, the well will probably start to suffer from excessive liquid holdup, causing the outlet pressure to decrease. In this situation the simulation may have two solutions, one with a small ID, another with a much larger ID. The choice of proportionality relationship allows you to pick the solution you want. However, also in this situation, the simulation may have no solution; this will happen if the specified outlet pressure is too big.

1.7.3 Network Operations

Restart Model

This is only available in the Network module. See also [Engine preferences \(p.178\)](#), and [Optimizing PIPESIM Network Simulation Performance. \(p.43\)](#)

Note: The model can be restarted in the following ways:

- Use the Restart button



- Use the Run button



and set the network engine advanced Command Line Parameters (see [Engine preferences \(p.178\)](#)) as follows:

- **-r** : Restart but do not keep branches skipped in the restart file permanently skipped.
- **-R** : Restart but permanently skip all branches skipped in the restart file (this is the default PIPESIM behavior)

When a Network simulation finishes, the final solution results are stored in a restart file. If the model is restarted



(as opposed to rerun), these final results are used as the starting point for the run, instead of the initial [estimates \(p.42\)](#).

The default restart function in PIPESIM restarts the model by keeping all skipped branches permanently skipped. So if the user has deactivated a branch, run the model, and reactivated the branch again before using the Restart button



, the deactivated branch remains deactivated.

Running a network model by using Restart , as opposed to Run, should also speed up the solution. However, the follow limitations apply:

- If the model has changed significantly (for example, a well was added, or a branch or node was de- or re-activated), the use of Restart may actually slow down the simulation. In addition, the user should pay attention to the restart option used (-r or -R).
- If minor changes have been made to a network (for example, a flowrate or pipe dimension change), Restart should provide a faster convergence than a normal run.
- If the program crashes part way through an iteration, or the model does not solve in the allowed number of iterations, or the run is terminated prematurely by user intervention or other system error, the Restart function can be useful. The model can simply be restarted and the program will start from where it left off. Use the Restart button on the PIPESIM toolbar in this case.

Well Optimization

This section provides information on the gas lift optimization functionality of PIPESIM.

This functionality allows an oil field to be optimized in terms of day-to-day oil production for gas lifted wells. In addition well, manifold and field constraints can be taken into account.

Please select [Help » Well Opt. Help](#) for further information.

1.8 Artificial Lift analysis

Perform the following basic steps to analyze the effects of artificial lift on a well:

1. [Build the well performance model \(p.51\)](#).
2. Ensure that the gas lift or ESP lift depth has been set.
3. Select **Operations » Artificial Lift Performance**.
4. Select the sensitivity parameters.
5. [Run \(p.195\)](#) the operation.
6. [Save the model \(p.35\)](#).

1.8.1 How to create artificial lift performance curves

To create an artificial lift performance curve, do the following:

1. Create and save your model ([p.33](#)): include a tubing object and one or more artificial lift points ([Injection points \(p.84\)](#), [gas lift valves \(p.84\)](#) or [ESP \(p.235\)](#)).
2. Select **Operations » Artificial Lift Performance**.
3. Set the [outlet pressure \(p.119\)](#).
4. (Optional) Select [sensitivity variables \(p.211\)](#).
5. Select the Artificial lift method, **Gas** or **ESP**, and set the appropriate data.
6. [Run \(p.195\)](#) the model.

1.8.2 General Gas Lift

This is the process of raising or lifting fluid from a well by injecting gas down the well through the tubing, casing, annulus, or riser. Injected gas aerates the fluid to make it exert less pressure than the formation does; the resulting higher formation pressure forces the fluid out of the wellbore. Gas may be injected continuously or intermittently, depending on the producing characteristics of the well and the arrangement of the gas lift equipment.

Note: PIPESIM models the continuous injection process only. For details on Artificial Lift, see the web site.

PIPESIM has two methods for modeling gas lift systems:

Gas Lift injection points

this uses fixed injection depths and injection rates. It is assumed that the quantity of lift gas requested is fully injected into the production string at the specified depth(s) and takes no account of the available injection gas pressure.

Gas Lift Valve System

the actual gas lift valve depths are specified. For normal PIPESIM operations, gas is injected at the specified rate at the deepest possible valve depth (taking into account the available injection pressure). For the Gas Lift Diagnostics operation only, the actual gas throughput for each valve is calculated, based on the injection pressure, production pressure, valve details, and valve status. For this operation, details of the gas lift system are required (valve size, P_{tro} , and so on). The valve throttling response (based on the bellows load rate of the valve) is modeled.

PIPESIM allows the following modes of operation:

- [Deepest Injection Point \(p.215\)](#)
- [Lift Gas Response \(p.216\)](#)
- [Bracketing \(p.215\)](#)
- [Gas Lift Design \(p.217\)](#)
- [Gas Lift Diagnostics \(p.230\)](#)

Gas Lift Operations - Deepest Injection Point

This operation computes the Deepest Injection Point [DIP] and the expected production rate, based on the data entered. To run it, select **Artificial Lift » Gas Lift » Deepest Injection Point**.

Set the **Outlet Pressure** to the pressure downstream of the last object in the model.

The input parameters are as follows:

Injection Gas Rate

injection gas rate

Injection Gas Surface Pressure

gas injection pressure at the surface

Injection Gas Surface Temperature

gas injection temperature at the surface

Injection Gas Specific Gravity

specific gravity of gas

Minimum Valve Injection DP

pressure drop across the valve

Max Allowable Injection TVD

a limit on the depth at which injection can occur

Annular Gas Pressure Design Gradient Method (p.234)

Choose from the following options:

- **Use static gradient** — the default
- **Use Rigorous Friction and Elevation DP**

Gas Injection Depth

Choose from the following options:

- **Optimum Depth of injection** — gas injection can occur at any depth in the wellbore
- **Injection at Valve Depths only** — gas injection is restricted to the specified valve/mandrel depths only, as defined in the model tubing description.

Use the buttons as follows:

- **Calculate** computes the liquid flow rate, pressure profile and the Deepest Injection Point.
- **Copy** copies the pressure profile to the clipboard.
- **Print** prints the pressure profile.
- **Summary File** displays the PIPESIM standard summary file.
- **Output File** displays the PIPESIM standard output file.

Gas Lift Operations - Bracketing

This operation computes the Deepest Injection point [DIP] and the expected production rate, based on the data entered for two set of conditions. (Normally the set of production rate conditions, now and at some time in the future).

These conditions are used to compute the flowing pressure profiles for the two cases, and therefore the deepest possible injection depths for the two sets of conditions.

The flowing pressure profiles are computed from the given outlet pressure and flow rates (and GLR and Watercut). Unlike the other Gas Lift operations, here the model reservoir pressure and IPR are not used to determine the flowing pressures or flow rates.

To run the operation, select **Artificial Lift** » **Gas Lift** » **Bracketing**.

Set the **Outlet Pressure** to the pressure downstream of the last object in the model.

The input parameters are as follows:

Injection Gas Surface Pressure

gas injection pressure at the surface

Injection Gas Surface Temperature

gas injection temperature at the surface

Injection Gas Specific Gravity

specific gravity of gas

Minimum Valve Injection DP

pressure drop across the valve

Max Allowable Injection TVD

a limit on the depth at which injection can occur

Annular Gas Pressure Design Gradient Method (p.234)

There are two options: **Use Static Gradient** (the default) and **Use Rigorous Friction and Elevation DP**

The **Bracketing Input** section shows the minimum and maximum data for:

- Q: Liquid Flow rate
- GLR or Gas Injection rate [Qgi]
- WCUT: Water cut

Use the buttons as follows:

- **Calculate** computes pressure profiles and the Deepest Injection Points for the two cases.
- **Copy** copies the pressure profile to the clipboard.
- **Print** prints the pressure profile.
- **Summary File** displays the standard summary file.
- **Output File** displays the standard output file.

Gas Lift Operations - Lift Gas Response Curves

This operation computes the response of the gas lift system to changes in any variable. You can select a range of gas lift rates and a set of sensitivity values. The resulting gas lift response curves show how the production rate and possible gas injection depth vary as a function of injection rate and the sensitivity parameter.

Use this operation to investigate how the well responds to various parameters before proceeding with a gas lift design.

To run the operation, select **Artificial Lift** » **Gas Lift** » **Lift Gas Response**.

The parameters are as follows:

Outlet Pressure

pressure downstream of the last object in the model.

Minimum Valve Injection DP

pressure drop across the valve.

Injection Gas Surface Temperature

gas injection temperature at the surface.

Max Allowable Injection TVD

a limit on the depth at which injection can occur

Inj Gas SG

injection gas specific gravity

Annular Gas Pressure Design Gradient Method (p.234)

There are two options: **Use static gradient** (default) and **Use Rigorous Friction & Elevation DP**.

Sensitivity Data

Perform sensitivity on any parameter in the model

Gas Injection Depth

Choose from the following options:

- **Optimum Depth of injection** — gas injection can occur at any depth in the wellbore
- **Injection at Valve Depths only** — gas injection is restricted to the specified valve/mandrel depths only, as defined in the model tubing description

Injection Gas rate

a range of injection gas rates

Use the buttons as follows:

- **Run Model** computes the liquid production rate, and the (deepest) injection depths for the specified range of gas injection rates and sensitivity parameters.
- **System Plot** displays the system plot.
- **Summary File** displays the PIPESIM standard summary file.
- **Output File** displays the PIPESIM standard output file.

Gas Lift Design

The Gas Lift Design (GLD) feature allows detailed design of the location and type of valves to insert into the tubing to maximize the potential of the well. A gas lift design can be conducted for the following:

- An existing mandrel spacing (Current Spacing)
- To create a new mandrel spacing (New Spacing)

Data Entry

To run the operation, select **Artificial Lift » Gas Lift » Gas Lift Design**. This opens a dialog with the following tabs:

- [Design Control \(p.218\)](#)
- [Design Parameters \(p.223\)](#)
- [Safety Factors \(Design Bias\) \(p.224\)](#)

[Project Data Gas Lift Design - Summary \(p.228\)](#) (this is only displayed after a successful design has been created).

Gas Lift Design

To perform a Gas Lift Design, do the following:

1. Create and save your model. [See how... \(p.33\)](#)
2. Select **Artificial Lift » Gas Lift » Gas Lift Design**.
3. Set the [required data \(p.217\)](#).
4. Select **Perform Design**.... A redesign can be performed after the design has been created.

Design Control

To set the following parameters, select **Artificial Lift » Gas Lift » Gas Lift Design** and select the **Design Control** tab.

Design Spacing

Select from the following options:

New Spacing

calculate a new mandrel spacing for the well as part of the gas lift design and overwrite (without prompting) any existing valve settings.

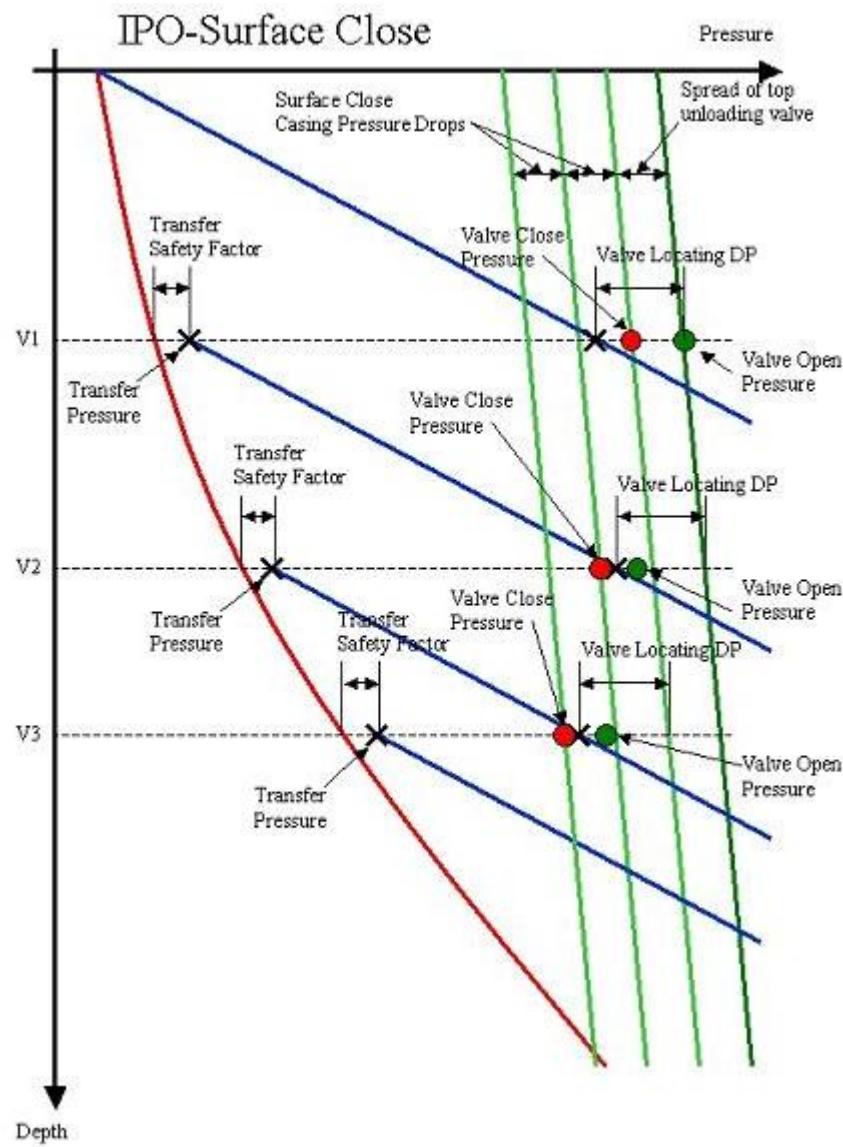
Current Spacing:

use the existing mandrel spacing as defined in the PIPESIM model and calculate appropriate valve parameters.

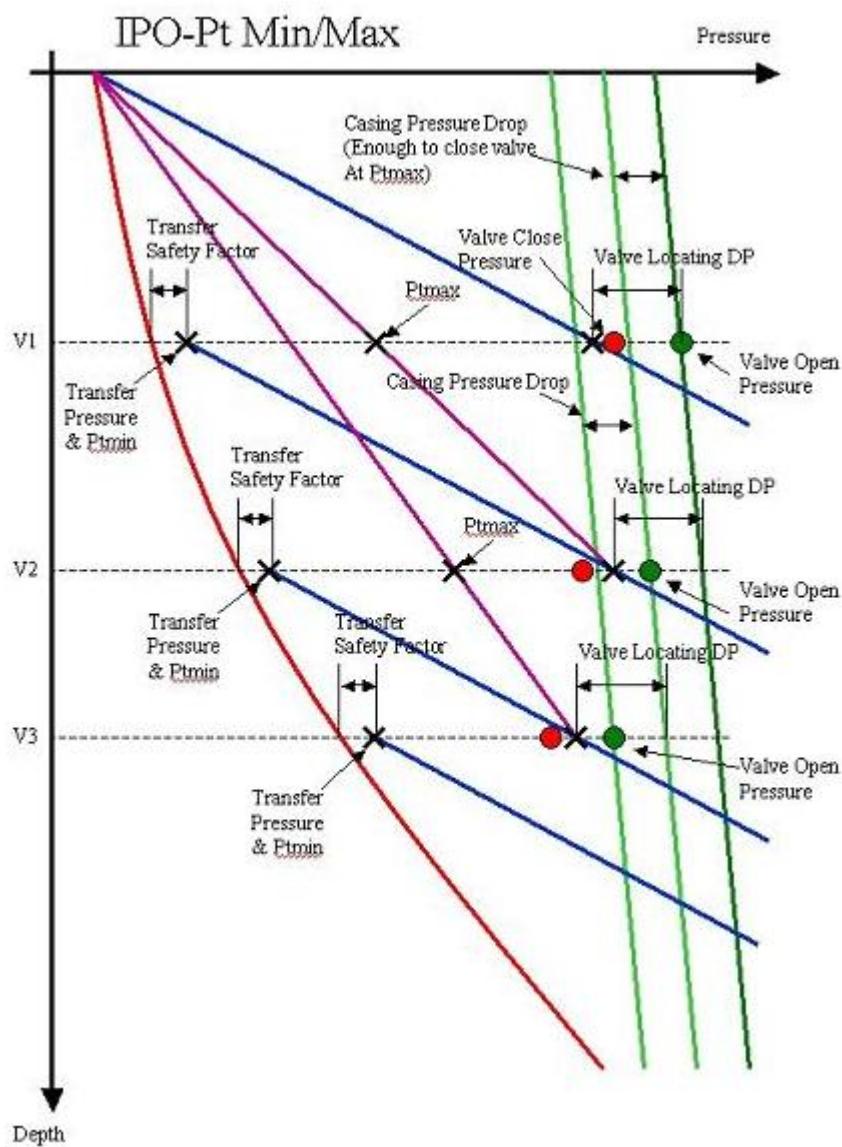
Design Method

The three following design methods are available (two for IPO design and one for PPO design):

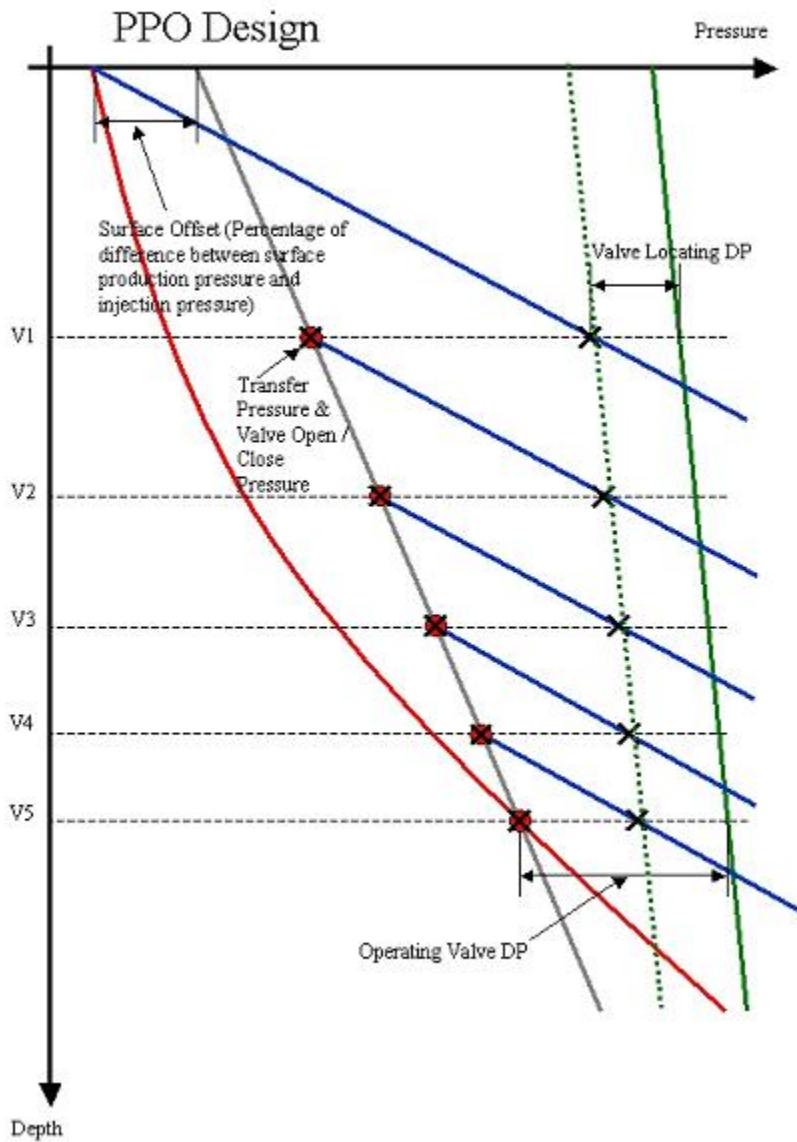
- Uses a user specified fixed surface close (injection) pressure drop between valves.



- Uses a calculated surface close (injection) pressure drop between valves (with user specified minimum).



- Uses a transfer gradient based on user entered Surface Offset and Operating Valve DP values. Normally used for design with production pressure operated (PPO) valves.



Surface offset

Either a percentage of the difference between the operating production (wellhead) pressure and the surface injection pressure, or a DP value (added to the wellhead pressure). The transfer gradient used for design is then a straight line drawn between the surface offset and the operating valve DP (at the operating valve location).

Operating Valve DP

The difference between the injection pressure and the production pressure at the operating valve location.

Top Valve Location

This is used as the basis for spacing the top unloading valve. The options are as follows:

Assume Liquid to Surface

Assumes that liquid is present in the wellbore to the surface.

Assume Liquid NOT to Surface

Assumes that a liquid level exists, based on the static bottom hole pressure and static gradient.

Well Absorbs Fluid

If this check box is selected, this assumes that the reservoir can readily absorb fluid from the wellbore, thus maintaining a constant tubing liquid level during unloading. If it is not selected, it is assumed that the reservoir cannot absorb any fluid which is unloaded from the annulus to the tubing.

Valve Selection Filter

You can select the valve manufacturer, type of valve to assume, valve size (nominal OD), valve series, and the minimum port diameter for the design. The filter selects the appropriate valve from the gas lift valve database. The appropriate valve parameters (port/bellows dimensions and so on.) will be used for the design. You can enter new valves into the PIPESIM database if required. Note that the type of valve selected (IPO or PPO) determines the default safety factors and calculation of open / close points.

Unloading Temperature

You can select whether to use the expected final production temperature, the ambient (geothermal) temperature, the injection gas temperature or the calculated unloading production temperature for the valve test rack pressure determination. You can select a different option for the top valve and for other valves. You can make a first pass for design based on one of these temperature profiles and then edit the temperature values for any of the valves on the design summary screen and recalculate the test rack pressures (using the [Re-Design \(p.228\)](#) option).

Max. Allowable Inj. TVD

This depth (true vertical) is taken as the maximum depth for valve placement in the spacing calculations. Normally this should be about 100 ft above packer depth.

Production Pressure Curve

Select a flowing production pressure curve to use for the design, one of the following:

Production Pressure Model

uses the objective flowing pressure gradient calculated with the PIPESIM model, assuming the target injection gas rate injected at the operating valve location (default).

Equilibrium curve

calculates and uses an equilibrium curve based on the target gas injection rate in the model. The equilibrium curve is a curve of the expected flowing pressure at any depth, assuming gas lift is being injected at that depth at the target gas injection rate.

[Perform Design \(p.227\)](#)

[Gas Lift Design - Summary \(p.228\)](#)

Note: The summary is only displayed after a successful design has been created.

Design Parameters

To set these parameters, select **Artificial Lift » Gas Lift » Gas Lift Design** and select the **Design Parameters** tab. The parameters are as follows:

Kickoff Pressure

Maximum possible injection pressure to kick off the well. Used for top valve placement only.

Operating Injection Pressure

The maximum available injection pressure to be used for the design calculation.

Unloading Production Pressure

Minimum possible production (wellhead) pressure during unloading. Used for placement of top valve only.

Operating Production

The expected production (wellhead) pressure. Used to calculate the production pressure curve.

Target Inj. Gas Flowrate

The target injection gas flowrate for design. Used to calculate the production pressure curve and operating valve sizing. Note that the effect of gas injection flowrate on well performance can be investigated with the PIPESIM model using the Lift Gas Response Curves operation. This should give you an understanding of the gas lift performance of the well and enable determination of the optimum target gas injection flowrate.

Inj. Gas Surface Temperature

The surface temperature of the injection gas. Used to determine the injection gas temperature profile.

Inj Gas Specific Gravity

Injection gas specific gravity. Used to determine injection gas pressure profile.

Min Unloading Liq rate:

The design includes calculation of the unloading liquid and gas injection rates for the unloading valves. You can specify the minimum unloading liquid rates for unloading.

Unloading Gradient

Gradient of the well fluid for unloading and spacing calculation.

Minimum Valve Spacing

Minimum distance for valve spacing (TVD). Valves will not be placed deeper if they are spaced closer than this value.

Minimum Valve Inj DP

Minimum difference between injection and production pressure required for valve placement. Valves will not be placed deeper if the pressure difference is less than this value.

Solution Point Rate/Fixed rate

Use either a specified liquid production flowrate or a specified reservoir pressure in the calculation of the Production Pressure Curve (the remaining property will be computed).

Reservoir Pressure

Static reservoir pressure used in the model. (The liquid production rate is calculated).

Liquid Production Rate

The desired liquid production rate. (The reservoir pressure and inflow is ignored).

Bracketing

Select **Enable bracketing Options** to view the following options, which control the bracketing interval for placement of bracketing valves:

Note: The type of bracketing valves depends on the type of operating valve.

- **Max TVD** — the maximum depth for bracketing valves
- **Spacing** — distance for bracketing valve spacing (TVD)

Annular Gas Pressure Design Gradient Method (p.234)

Select one of the following:

- **Use static gradient** — the default
- **Use Rigorous Friction and Elevation DP**

Safety Factors (Design Bias)

To set the following parameters, select **Artificial Lift** » **Gas Lift** » **Gas Lift Design** then select the **Safety Factors (Design Bias)** tab:

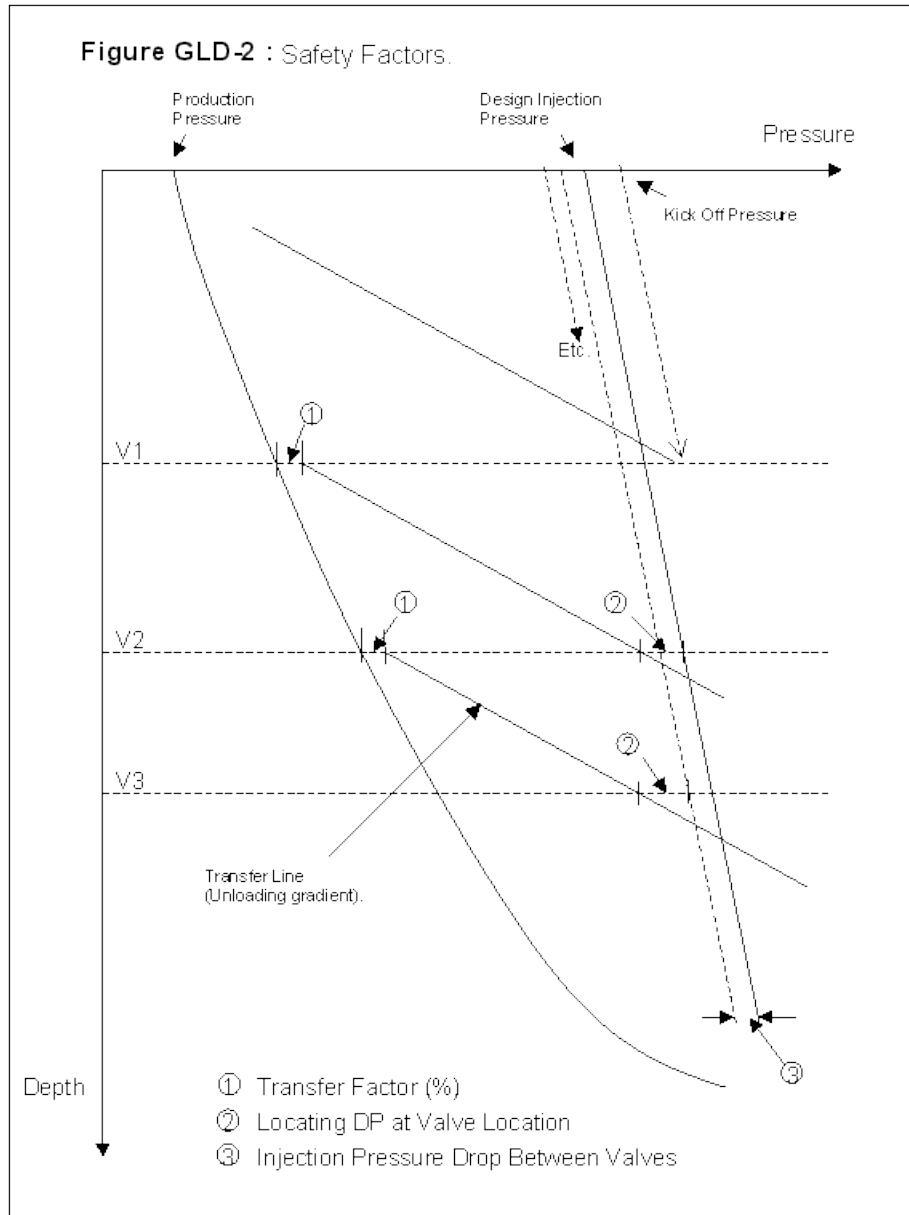
Surface Close Pressure Drop Between Valves

[IPO valves only] Safety factor normally used for Injection Pressure Operated (IPO) valves only. Surface close (injection) pressure drop for each successive valve down hole to ensure valve above closes. For IPO Surface Close method a fixed value is used for all valves. For Pt Min/Max method it is the minimum value since the design process will calculate this value. Default value for injection pressure operated valves is 15 psi.

Note: PPO Design does not use this safety factor.

Locating DP at Valve Location

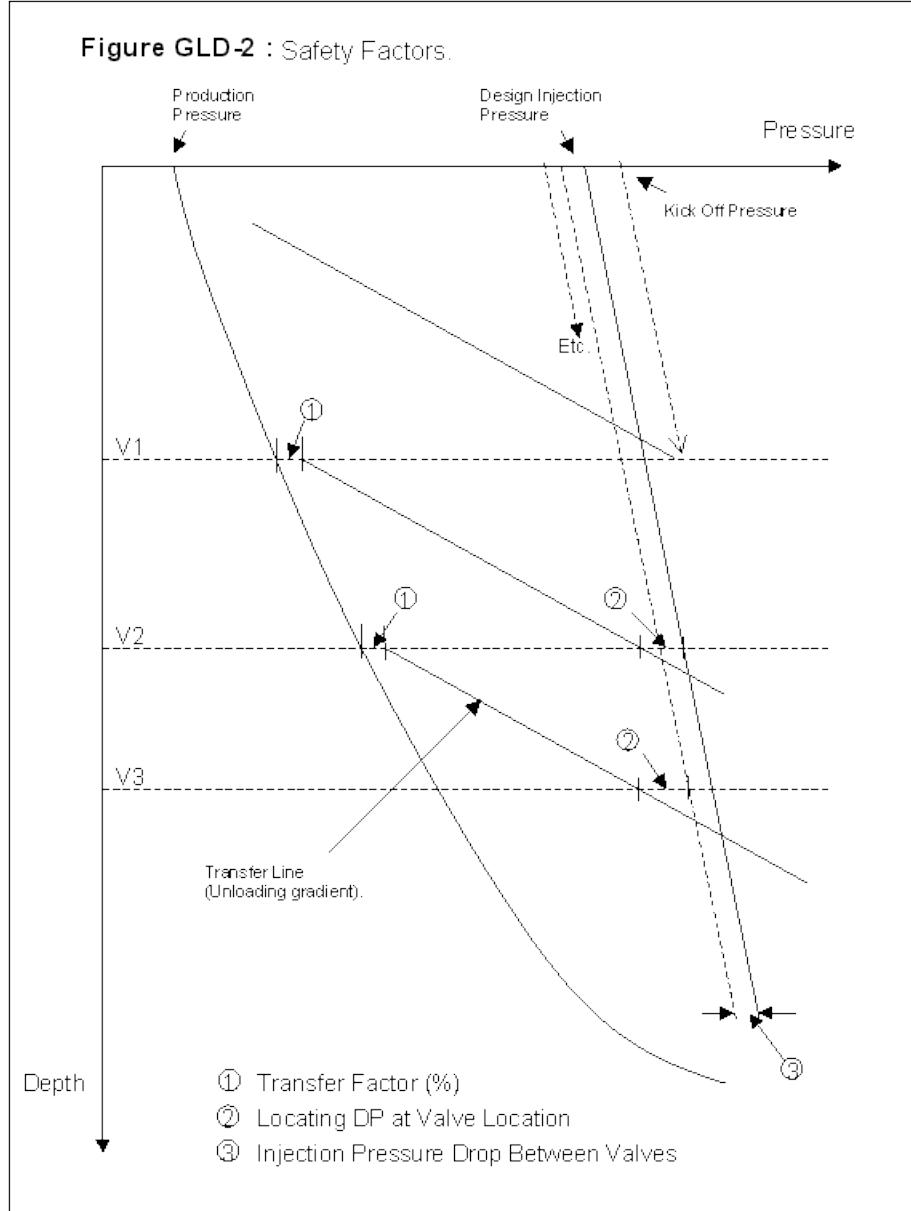
Safety factor normally used for both injection pressure operated and production pressure operated valves. This is an offset from the injection pressure line to the start of the transfer (unloading) line at each valve location. Default value for injection pressure operated valves is 50 psi. and default for production pressure operated valves is 100 psi.



Transfer Factor

[IPO valves only] Safety factor normally used for injection pressure operated valves only. This is an offset from the production pressure line to the start of the transfer line (transfer point) at each valve location. This can be entered as a percentage of the difference between the production pressure and injection pressure at each valve location or as a percentage of the production pressure. Default value is zero.

Note: PPO Design does not use this safety factor (the transfer gradient is itself a transfer safety factor).



Place Orifice at operating valve location

Select this check box to use an orifice in the operating valve location. You can then edit the default **Discharge Coefficient for Orifice** (used for gas throughput calculations).

Subtract delta P from Ptro at Operating Valve Location

If the operating valve is an IPO or PPO valve, you can elect to subtract a delta P from the calculated Ptro value for this valve (flag the valve). This causes an additional injection pressure drop at this valve during unloading. Default value is zero.

Place IPO valve at Operating valve location

[PPO Design only] For a PPO design, you can opt to place an IPO valve in the operating valve location.

Project Data

Data that relates to the current project. This will be displayed on the Gas Lift Design report.

Perform Design

After selecting **Artificial Lift** » **Gas Lift** » **Gas Lift Design**, enter data on the [Design Control \(p.218\)](#), [Design Parameters \(p.223\)](#) and [Safety Factors \(p.224\)](#) tabs. The data are used to design the gas lift system; see [minimum data set \(p.227\)](#).

When all data is present, click the **Perform Design** button. The following calculations are performed using the design data and selected design method:

- The objective production pressure curve is generated.
- The mandrel depths are calculated for New Spacing. For Current Spacing, the specified mandrel depths (in the tubing description) are used.
- The valve port sizes are calculated. The valve port size is selected to pass the unloading gas rate for the unloading valves and the target gas rate for the operating valve. If no unloading calculations are possible (no inflow means that the well does not flow), the minimum port size for the series is selected by default.
- The unloading injection gas requirement and unloading liquid rates are calculated (by nodal simulation) for each valve station. Also, the maximum possible gas throughput for each valve is calculated.
- The valve open pressures, close pressures and test rack open pressures (P_{tro}) are calculated.

The depth of the operating valve is determined when any of the following conditions is met:

- the position of the next unloading valve would be closer than the minimum valve spacing criterion
- the injection pressure minus the production pressure is less than the Minimum Valve Injection DP
- the Maximum Allowable Injection TVD is reached.

If the **Bracketing** option was selected, bracketing valves are placed below the operating valve down to the specified max TVD at the specified bracketing spacing. If the operating valve is either an orifice, a flagged valve, or an IPO valve in a PPO design, the bracketing mandrels are assumed to contain dummy valves.

When the design calculations are complete, a [summary sheet \(p.228\)](#) is displayed. On this some valve data can be changed and a [re-design \(p.228\)](#) performed if required.

The standard Gas Lift design graph (Pressure versus Depth plot) and report can also be displayed and printed.

Once the design has been verified, it can be transferred to the well model using the **Install Design** button.

Minimum Data Set

The minimum data set required to perform a gas lift design is the following:

- Target Injection Gas Flowrate
- Injection Gas Surface Temperature

- Injection Gas Specific Gravity
- Operating Prod. Pressure (Flowing tubing head pressure).
- Available Injection Pressure.

In addition, a well model is required. This must contain at least a completion (reservoir), tubing icons, and a valid fluid model.

Perform Re-Design

You can edit certain parameters in the Gas Lift Design [summary sheet \(p.228\)](#) and perform a redesign of the gas lift system.

You can edit the following fields:

- Valve Depth (if the **Edit Depth** check box is selected).
- Valve Model and Port size (if the **Edit Valve** check box is selected).
- Valve Temperature (if the **Edit Temp** check box is selected).

To change the valve model and/or port size:

1. Double-click the valve to change out.
2. The [gas lift valve \(p.84\)](#) database opens. You can filter for certain types of valves using the **Filter** parameters on the right hand side.
3. Double click the required valve. It is placed in the summary field.

After changing any parameter, click on the **Perform Re-Design** button to recalculate the design parameters to reflect the changes.

Note: Only the parameter which has its check box selected remains unaltered. Any other parameters may be changed by the design process. It is therefore recommended that you change only one parameter at a time and then perform a redesign.

In this way you can modify the initial gas lift design, for example in the following ways:

1. Reset the valve temperatures for each valve and recalculate test rack pressures.
2. Manually reset the design mandrel depths.
3. Replace the operating valve with an orifice valve or other valve.
4. Replace unnecessary valves with dummy valves.
5. Manually re-size upper valves (port sizes) to user preferred port sizes.

The standard Gas Lift Design graph (Pressure - Depth plot) can also be displayed and printed.

Once the design has been verified, transfer it to the well model using the **Install Design** button.

See also: [Perform Design \(p.227\)](#)

Gas Lift Design - Summary

This summary screen is displayed after the **Perform Design** operation is successfully executed. Use it allows to review and/or change the design, or perform a [Re-Design \(p.228\)](#) or a diagnostic.

Proposed design

Valves and design parameters

This table lists the proposed valves and their design parameters, as follows:

- Valve depth (MD or TVD).
- Valve series
- Port Size - the valve port size is selected to pass the unloading gas rate for the unloading valves and the target gas rate for the operating valve. If no unloading calculations are possible (no inflow - well does not flow), then the minimum port size for the series is selected by default.
- Ptro - test rack open pressure.
- Open Pressure at Surface (for IPO design only).
- Close Pressure at Surface (for IPO design only).
- Pdome - Valve dome pressure at depth (for PPO design only).
- Open Pressure at Depth (for PPO design only).
- Gas Rate (Unloading) - injection gas rate required to unload the well to the transfer point (Not calculated if no inflow from the reservoir).
- Unloading Liquid Rate - calculated liquid unloading rate at the transfer point. (Not calculated if no inflow from the reservoir).
- Max Valve Throughput - maximum rate that can be passed through the valve given the injection pressure, production pressure and port size.
- Valve Temperature - Valve temperature used for Petro calibration.
- Injection Pressure Drop - surface close injection pressure drop between valves.
- Cd - discharge coefficient (from the valve database).
- Valve Choke (optional) - can be used if chokes are placed in valve ports.

Buttons

The following buttons are available:

Add (Valve Lookup)

To add a valve from the database to the design valve table, click [Add.\(Valve Lookup\) .. \(p.86\)](#). The valve will be added to the bottom of the current list.

To add [change out] a valve at a particular depth:

1. Select the **Edit Valve** check box.
2. Double click the cell under **Valve Series** or **Port Size** that corresponds to the required depth.

Edit Selected Valve

Use this to edit the selected valve. (This is only possible if the **Edit Valve** check box is selected.)

Remove Selected Valve

Use this to remove the selected valve. (This is only possible if the **Edit Valve** check box is selected.)

Perform Re-Design

Use this to change some parameters and perform a Re-Design of the system.

Re-Design is only possible if one of the following parameters is edited in the summary screen:

- **Valve Depth** (the **Edit Depth** check box must be selected)
- **Valve Series or Port size** (the **Edit Valve** check box must be selected)
- **Valve Temperature** (the **Edit Temp** check box must be selected)

Graph

Click **Graph...** to display the traditional Gas Lift Design graph (Pressure versus Depth plot). This can also be printed

Install Design

Click **Install Design** to install the proposed design in the current tubing description. This overwrites any gas lift valves already in place for this well.

Target Operating Injection Pressure

This is computed from the design. It can be used, or modified, for the diagnostics operation.

Run Diagnostics

Performs [gas lift diagnostics \(p.230\)](#) on the current gas lift valve system with the operating injection pressure specified.

Gas Lift Diagnostics

This operation performs a simulation of the gas lift system. It calculates (simulates) the status and actual throughput for each valve, based on the simulated injection and production pressures. To run it, select **Artificial Lift** » **Gas Lift** » **Gas Lift Diagnostics**.

The valve status can be closed, throttling or fully open. The valve performance characteristics take into account the throttling response of the valves. The throttling response is based on the bellows load rate of the particular valves.

The diagnostic simulation is iterative. The throughput of each valve depends on the simulated injection and production pressures (in addition to valve characteristics). However, the simulated production pressure itself depends on the throughput of each valve. Therefore the system is solved iteratively.

The assumed conditions for the diagnostic simulation are as follows:

- The operating injection pressure, taken from the design result.
- The operating production pressure, taken from the design input.
- The operating reservoir pressure, taken from the design input.
- The valve positions and parameters, taken from the design summary.

All other model parameters are as defined in the model.

The diagnostic results are displayed graphically on a pressure-depth plot. The valve status is shown on the left hand well schematic as red if closed or green if open (or throttling). To view the valve details, place the cursor over a valve on the well schematic.

Gas Lift Diagnostics calculates the injection rate and the maximum injection rate for the valves by using the Thornhill-Craver equation. Total gas lift rates are shown in the **Rates** section below the well schematic. This shows the following:

- **Measured** — the target gas injection rate
- **Calc** — the value calculated when Diagnostics is run
- **%Max** — the ratio of the calculated flow rate to the maximum flow rate, expressed as a percentage

To view the detailed results in tabular form, click **Data Sheet...**.

Gas Lift Diagnostics

This operation determines which valve or valves a gas lift system is lifting from. It creates a well performance curve. To use this feature a [Gas Lift \(p.217\)](#) system must have been installed.

How to create a well performance curve

To create a well performance curve, do the following:

1. Create and save your model. [See how... \(p.33\)](#)
2. Select **Artificial Lift** » **Gas Lift** » **Gas Lift diagnostics**.
3. Enter the [required data \(p.223\)](#).
4. [Run \(p.217\)](#) the diagnostics.

Gas Lift Diagnostics Data

The following parameters are required by the **Diagnostics** operation:

Outlet Pressure

system outlet pressure (usually wellhead pressure).

Operating Injection Pressure

actual injection pressure.

Target Gas Lift Rate

estimated (measured) gas lift injection rate. This value will be calculated by the diagnostics routine (based on the injection pressure, production pressure and valve status).

Surface Injection Temperature

surface temperature of the injection gas. Used to determine the injection gas temperature profile.

Gas Specific Gravity

injection gas specific gravity. Used to determine injection gas pressure profile.

To set these parameters, select **Artificial Lift** » **Gas Lift** » **Gas Lift Diagnostics**.

Gas Lift Theoretical background

Gas lift is a method of lifting fluid by using relatively high pressure (250 psi minimum) gas as the lifting method through a mechanical process.

Gas lift is particularly applicable for lifting wells where high-pressure gas is available. Gas compressors may have been installed for gas injection or high-pressure gas wells may be nearby. Since the cost of compression far exceeds the cost of downhole gas lift equipment, always consider gas lift when an adequate volume of high-pressure gas is available for wells requiring artificial lift.

Gas lift is accomplished by one of the following methods:

Continuous flow

In continuous flow a continuous volume of high pressure gas is introduced into the eductor tube to aerate or lighten the fluid column until reduction of the bottom-hole pressure allows a sufficient differential across the sand face, causing the well to produce the desired rate of flow. This requires the following:

- a flow valve that permits the deepest possible one point injection of available gas lift pressure
- a valve that acts as a changing or variable orifice to regulate gas injected at the surface depending upon tubing pressure.

This method is used in wells with a high productivity index and a reasonably high bottom-hole pressure relative to well depth. In this type of well, fluid production can range from 200-20,000 B/d through normal size tubing strings. On casing flow it is possible to lift in excess of 80,000 B/d. The internal diameter (ID) of the pipe governs the amount of flow, provided the well productivity index, bottom-hole pressure, gas volume and pressure, and the mechanical conditions are ideal.

Smaller volumes can be efficient using continuous flow where small 'macaroni' tubing is used. As low as 25 B/d may be produced efficiently through 1-in. tubing by continuous flow.

Intermittent flow

Intermittent flow involves expansion of a high pressure gas ascending to a low-pressure outlet. A valve with a large port permits complete volume and pressure expansion control of gas entering into the tubing. This either regulates lift of the accumulated fluid head above the valve with a maximum velocity to minimize slippage, or controls liquid fall back, fully ejecting it to the tank with minimum gas.

Intermittent lift is generally used in conjunction with a surface time cycle controller (intermitter). It is used on wells with relatively low fluid volumes, or wells that present high or low PI with low bottom-hole pressure. In intermittent lift, gas is injected at regular intervals by the intermitter with regulated cycling to coincide with fluid fill-in rate from the producing formation into the well bore. The multipoint injection gas through more than one lift valve may also accomplish intermittent lift.

Gas lift instability

Unstable operational conditions may occur in a continuous gas lift well because the characteristics of the system mean that small perturbations can degenerate into huge oscillations in the flow parameters. Therefore, a clearly defined mechanism is required to show the relative importance of the different factors involved, to help to assure stable flow conditions at the design phase, or to decide what to do to stabilize an unstable gas lift well.

Unified instability criteria were developed by Alhanati et al. (1993) for continuous gas lift wells to overcome drawbacks in previous developments. The unified criteria can be used for all possible flow regimes for the gas-lift valve and surface gas injection choke. The unified criteria were developed using a number of simplifying assumptions, so are not highly accurate or applicable to every type of instability experienced in a gas lift installation. However, they cover a number of common cases encountered in the industry and certainly indicate what can be done to improve operating instability.

For the Alhanati factors to be computed, the well model must have the following:

- Well IPR ([p.398](#)) modeled by the PI ([p.399](#)) method
- Casing inside diameter ([p.76](#)) is set. The simple ([p.77](#)) or detailed ([p.79](#)) tubing model can be used.
- Valve Port Diameter ([p.83](#)) set. The operating Gas Lift Valve Port (Orifice) Diameter.
- Surface injection pressure ([p.83](#)) set. This is the gas lift injection surface pressure, upstream of the surface injection choke (which must be set higher than the internally calculated casing head pressure).
- To ensure accurate results from the Alhanati instability check, the model should be developed only to the well head.

Note: The factors will not be computed if data is missing or incorrect.

From this data the well model automatically calculates the steady state casing and tubing pressures. The Gas Lift performance curve should then be developed as normal; the Alhanati factors C1 and C2 are automatically generated. For stable gas lift operations, both these factors must be greater than zero.

To view the Alhanati factors (C1 and C2) graphically (using the system plot), select:

- X-axis: Total Injection Gas
- Left Y-axis: Alhanati Criterion 1.
- Right Y-axis ([p.257](#)): Alhanati Criterion 2. For stable gas lift operations, both these factors must be greater than zero.

If the graphical display is empty, this implies that one or more of the above data items is missing/incorrect.

Assumptions of the Alhanati model

The model assumes:

- Constant pressure at the gas injection manifold, which is upstream of the surface injection choke
- Adiabatic flow through the choke.

In the unified criteria, two sets of criteria were defined, namely C1 and C2, and both must be greater than zero for stable gas lift operation.

$$C_2 = \left(F_1 \frac{\gamma_v}{\mu_v} \right) + \frac{\gamma_v}{F_c} \quad \text{Eq. 1.8}$$

where

$$F_3 = \frac{(q_{fo} + q_{Go}) \cdot A_t}{(p_f - p_G) \cdot g} \cdot \frac{P_{co}}{q_{fo}} \quad \text{Eq. 1.9}$$

$$\mu_{ch} = \frac{(ZT)_c}{(ZT)_m} \quad \text{Eq. 1.10}$$

Nomenclature

A_t	Cross sectional area of tubing	in^2
g	Acceleration due to gravity	ft^2/s
P_{co}	Steady state casing pressure	$psia$
P_g		
P_f		
q_{fo}	Steady state reservoir fluids flow rate	$stbd$
q_{Go}	Steady state injected gas flow rate	$mmscf$
γ	Gas expansion factor	
T	Temperature	$^{\circ}F$
Z	gas compressibility factor	

subscripts

v	gas lift valve
ch	gas injection choke
t	tubing
c	casing
m	manifold

Annular Gas Pressure Design Method

PIPESIM can calculate the pressure drop caused by gas friction through annular space. Annular space is defined as the space between the outer diameter of the tubing ($ID+2*$ wall thickness) and the Casing ID. You must specify the annular dimension in the [Tubing dialog \(p.76\)](#).

Note: The Casing ID must be bigger than the outside of the tubing diameter and a Casing ID must be given for all depths shallower than the Maximum Allowable Injection TVD.

This calculation is available in the following Gas Lift Operations:

- [Deepest Injection Point \(p.215\)](#)
 - [Bracketing \(p.215\)](#)
 - [Lift Gas Response Curves \(p.216\)](#)
 - [Gas Lift Design \(p.223\)](#)
-

Note: The default is for PIPESIM to use the static annulus gas gradient rather than calculate the pressure drop. The **Annulus Gas Pressure Design** option you select applies to all four operations. For instance, if you select the **Use Rigorous Friction and Elevation DP** in the **Deepest Injection** dialog, this is adopted for the other three operations.

1.8.3 Electrical Submersible Pumps (ESP)

General

The electric submersible pump (ESP) is perhaps the most versatile of the artificial lift methods. The ESP comprises a down hole pump, electric power cable, motor and surface controls. In a typical application, the down hole pump is suspended on a tubing string hung on the wellhead and is submerged in the well fluid. The pump is close-coupled to a submersible electric motor that receives power through the power cable and surface controls.

ESPs are used to produce a variety of fluids and the gas, chemicals and contaminants commonly found in these fluids. Aggressive fluids can be produced with special materials and coatings. Sand and similar abrasive contaminants can be produced with acceptable pump life by using modified pumps and operation procedures.

ESPs usually do not require storage enclosures, foundation pads, or guard fences. An ESP can be operated in a deviated or directionally drilled well, although the recommended operating position is in a vertical section of the well.

The ESP has the broadest producing range of any artificial lift method ranging from 100 b/d of total fluid up to 90,000 b/d.

ESPs are currently operated in wells with bottom hole temperatures up to 350 degree Fahrenheit. Operation at elevated ambient temperatures require special components in the motor and power cables of sustained operation at high temperatures, and have efficiently lifted fluids in wells deeper than 12,000 ft. System efficiency ranges from 18 to 68%, depending on fluid volume, net lift and pump type.

ESP System Components: Motor

The ESP system's prime mover is the submersible motor. The motor is a two-pole, three-phase, squirrel-cage induction type. Motors run at a nominal speed of 3,500 rev/min in 60-Hz operation. Motors are filled with a highly refined mineral oil that provides dielectrical strength, bearing

lubrication and thermal conductivity. The design and operation voltage of these motors can be as low as 230 volt or as high as 4,000 volt. Amperage requirement may be from 17 to 110 amps. The required horsepower is achieved by simply increasing the length of the motor section. The motor is made up of rotors, usually about 12 to 18 inches (300-460 mm) in length that are mounted on a shaft and located in the electrical field (stator) mounted within the steel housing. The larger single motor assemblies will approach 33 feet (10 m) in length and will be rated up to 400 horsepower, while tandem motors will approach 90 feet (27.5 m) in length and will have a rating up to 750 horsepower. The rotor is also composed of a group of electromagnets in a cylinder with the poles facing the stator poles. The speed at which the stator field rotates is the synchronous speed, and can be computed from the equation:

$$v = \frac{120 f}{M} \quad \text{Eq. 1.11}$$

Where: v is speed in rev/min, f is frequency in cycles/sec and M is number of magnetic poles.

The number of poles the stator contains is determined by the manufacturer. Therefore to change the speed of the stator magnetic field, the frequency will have to change.

Heat generated by the motor is transferred to the well fluid as it flows past the motor housing. Because the motor relies on the flow of well fluid for cooling, a standard ESP should never be set at or below the well perforations or producing interval, unless the motor is shrouded.

Motors are manufactured in four different diameters (series) 3.75, 4.56, 5.40 and 7.8 in. Thus motors can be used in casing as small as 4.5 in. 60-Hz horsepower capabilities range from a low of 7.5 hp in 3.75-in series to a high of 1,000 hp in the 7.38-in series.

Motor construction may be single section or several "tandems" bolted together to reach a specific horsepower. Motors are selected on the basis of the maximum diameter that can be run easily in a given casing size.

ESP System Components: Pumps

The ESP is a multistage centrifugal pump. Each stage of a submersible pump consists of a rotating impeller and a stationary diffuser. The pressure-energy change is accomplished as the liquid being pumped surrounds the impeller. As the impeller rotates it imparts two rotating motion components to the liquid: one is in a radial direction outward from the center of the impeller (centrifugal force), the other motion moves in a direction tangential to the outside diameter of the impeller. The resultant of these two components is the actual direction of flow. The type of stage used determines the rate of fluid production. The number of stages determines the total design head generated and the motor horsepower required. The design falls into one of two general categories: the smaller flow pumps are generally of radial flow design. As the pumps reach design flows of approximately 1,900 B/D, the design change to a mixed flow.

The impellers are of a fully enclosed curved vane design, whose maximum efficiency is a function of the impeller design and type and whose operating efficiency is a function of the percent of design capacity at which the pump is operated. The mathematical relationship between head, capacity, efficiency and brake horse power is expressed as:

$$\text{Power} = \frac{q_v H \gamma}{\eta} \quad \text{Eq. 1.12}$$

Where: q_v is the volume flow rate, H is the head, γ is the fluid specific gravity and η is the pump efficiency

The discharge rate of a submersible centrifugal pump depends on the rotational speed (rpm), size of the impeller, impeller design, number of stages, the dynamic head against which the pump is operating and the physical properties of the fluid being pumped. The total dynamic head of the pump is the product of the number of stages and the head generated by each stage.

"Bolt-on" design makes it possible to vary the capacity and total head of a pump by using more than one pump section. However, large-capacity pumps typically have integrated head and bases.

Pump Selection

Select **Artificial Lift » ESP » ESP Design** and use the **Pump Selection** tab . The tab has two sections, **Pump Design Data** and **Pump Parameters**. Select a pump based on certain design criteria.

Pump Design data

Design Production rate

Desired flowrate through the pump in stock-tank units. The actual flowing quantity will be computed.

Design Outlet Pressure

the required outlet pressure of the PIPESIM model when the pump is installed. It is recommended to only model the well, and no associated flowline or riser, while designing the ESP system. In this case the outlet pressure would then be the wellhead pressure

Static Reservoir Pressure

If you entered a value previously, that value is preserved. However, if this field is empty, the value is taken from the PIPESIM model.

Water cut

If you entered a value previously, that value is preserved. However, if this field is empty, the value is taken from the PIPESIM model.

GOR (or GLR)

If you entered a value previously, that value is preserved. However, if this field is empty, the value is taken from the PIPESIM model.

Pump Depth

The depth at which the pump is to be installed. This is taken from the PIPESIM model if a pump is already installed or can be entered.

Casing ID

The casing size that the pump has to fit into. Usually 3.38 to 11.25 in.

Design Frequency

The frequency/speed that the pump is expected to run at.

Gas Separator Efficiency

The efficiency of the gas separator if installed.

Head factor

Allows the pump efficiency to be factored (default = 1).

Viscosity Correction

All pump performance curves are based on water systems, this option will correct for oil viscosity.

Select Pump

Will use the available data to select suitable pumps from the database. The pump intake conditions will first be computed. The resulting pump list can be sorted by efficiency or Maximum flowrate by selecting the column header. The Manufacturers to select from can be filtered. [Errors in the simulation \(p.241\)](#).

Pump parameters**Calculate**

Calculate pump performance at the conditions specified. [Errors in the simulation \(p.241\)](#)

Stage-by-stage

Perform the stage calculations on a stage-by-stage basis. Default = stage-by-stage.

Selected Pump

The pump selected, by the user, from the design data

No. of Stages required

The computed number of stages for this pump under these conditions.

Pump efficiency @ Design rate

The efficiency of the pump at the design production rate

Pump power required

The power required for this pump to deliver the required flowrate.

Pump intake pressure

The computed pump intake pressure.

Pump discharge pressure

The computed pump discharge pressure.

Head required

The computed pump head required

Liquid density

The computed liquid density at the pump intake

Free gas fraction at inlet conditions

The computed gas fraction.

Pump performance plot

plot performance curves at different speeds

Pump curves

plot standard performance curves

Install pump

Install the pump into the tubing of model. This will replace any existing ESP but not gas lift valves.

See also: [Select a Motor \(p.235\)](#), [Select a Cable \(p.241\)](#)

Motor Selection

This can only be performed after a pump has been selected.

Select **Artificial Lift** » **ESP** » **ESP Design** and use the **Motor/Cable Selection** tab.

Name of the selected Pump

Selected from the **Pump** tab.

No. of stages

Computed.

Pump efficiency

Computed

Pump power required

Computed.

Select Motor

The resulting motor list can be sorted by power, voltage, current, etc. by selecting the column header. [Errors \(p.235\)](#).

Various parameters associated with the motor will be computed and displayed at both 60Hz and the initially entered Design Frequency.

NP [Name Plate] Power

NP [Name Plate] Voltage

NP [Name Plate] Current

See also: [Select a Pump \(p.236\)](#), [Select a Cable \(p.241\)](#)

ESP Database

To simulate an ESP, PIPESIM maintains a database of manufacturers and models from which the user can select. For each model the diameter, minimum and maximum flowrate and base speed are provided. A plot of the ESP's performance is also available. If the required ESP is not in the database, you can easily enter the basic data required for it into the database using **Data** » **New ESP/Pump/Compressor**. See [Data/NewESP-Pump-Compressor \(p.242\)](#).

Selection

When modeling an ESP, it is important that the correct size (expected design flowrate and physical size) ESP is used. A search facility is available, based on these two parameters, to select the appropriate ESP from the database. The search can, if required, be restricted to a particular manufacturer. Pumps that meet the design criteria will be listed.

Stage-by-stage modeling

Stage-by-stage modeling is selected by selecting the checkbox next to the calculate button. Alternatively by inserting Engine Keywords ([PUMP STAGECALCS](#)) (p.701) into the model, using the [EKT](#) (p.94).

Install a Pump

Once the ESP manufacturer and model (p.235) has been selected from the [database of common ESP's](#) (p.239) some parameters can be altered. The performance curves for each model are (normally) based on a Speed of 60Hz and 1 stage.

Design data

Speed

The actual operating speed of the ESP

Stages

The actual number of stages of the ESP

Head factor

Allows the efficiency to be factored (default = 1)

Calculation Options

Viscosity Correction

Allow a viscosity correction factor to be applied to take account of changes to the fluid viscosity by the pressure and temperature.

Gas Separator present

Allow a gas separator to be added (automatically) with an efficiency: Separator efficiency - efficiency of an installed gas separator (default = 100% if installed)

Performance table

The data used to predict the performance of the ESP

Standard Curves

The standard performance curves for the ESP - can be printed/exported

Variable Speed Curves

Variable speed curves at 30 - 90 Hz.- can be printed/exported

ESP Design

The ESP option is selected from the Artificial Lift menu. To design an ESP the following stages are required:

[Select a Pump](#) (p.236)

[Select a Motor](#) (p.235)

[Select a Cable](#) (p.241)

The ESP should then be installed, added into the tubing, at the required depth. This can either be performed manually or by using the **Install** button. Installing automatically removes any existing ESPs in the tubing. However, any gas lift values or injections points are not removed.

See also: [ESP \[Reda\] web site](#)

ESP System Components: Cable

Cable Selection can be determined after a Pump and motor have been selected.

1. Motor/Cable Selection tab
2. Cable Selection
3. Select Cable

Cable Length

The length of the cable, can be modified

NP Current @ Design Frequency

The [Name Plate] Current at the design frequency. Cannot be changed.

Computed values

Selected Cable

Cable length

Voltage drop

Downhole voltage

Surface voltage

Total System KVA

Design Report

Display a report that details all the selected components of the ESP system.

Errors

Occasionally a pump may not be able to be determined and a Convergence error will be reported. There could be a number of reasons for such an error and the user is advised to view the output report.

Common problems:

1. The system cannot reach the outlet pressure specified. Try increasing the outlet pressure.

ESP Design

To select an ESP, do the following:

1. Create and save your model. (A well model need not be created in order to select an ESP, but if it is, the data entered will be used.) [See how... \(p.33\)](#)
2. From the toolbar, select **Artificial Lift** » **ESP** » **ESP design**

3. Set the required data (p.240)

ESP / Compressor / Pump data entry

The supplied ESP database may not always contain the most up to date ESP data or details of the exact ESP you are using. To ensure that the latest data can be utilized, PIPESIM allows you to enter your own ESP performance data. [See how... \(p.184\)](#)

As compressors and pumps are typically custom made, there is no database for these.

1.8.4 Rod Pump Module

[Design \(p.244\)](#)

[Diagnostics \(p.246\)](#)

[Options \(p.250\)](#)

[New / Edit RP Data \(p.248\)](#)

[New / Edit Pumping Unit \(p.249\)](#)

About the Rod Pump Module

Use this module to manage rod pump lifted production wells. The rod pump module offers options in equipment selection and design control specification so you can conduct various optimal design scenario studies. You also can perform a design study based on your experience and assumptions. For diagnostics, the combination of neural network technology and comprehensive analysis provides you with more detailed and rigorous diagnostics analysis.

The module comprises the following main parts:

- Equipment Database
- Rod Pump Design
- Rod Pump Diagnostics with Dynamometer Card

Limitations

The following items are current rod pump limitations:

- The well model should be PI or Vogel.
- The well must not be a gas well and it should have a Liquid IPR.
- The model must have a single vertical completion; it cannot be a multilayer model.
- The fluid model must be BlackOil.
- The bubble point pressure must be specified.
- The well must have at least one tubing section defined.

Equipment Database

The equipment database in the RP module includes Pumping Unit data, Rod data, and Pump data. You can browse the data in the database. You can add, modify, or delete any equipment data that you entered.

Rod Pump Design and Optimization

The Rod Pump Design module uses the following design methods:

- Specify target flowrate
- Specify pump intake pressure
- Specify dynamic fluid level.

The Rod Pump Design module provides the following design options:

- Specify or select a motor
- Specify or select a stroke
- Specify or select stroke frequency
- Specify or select plunger size
- Specify or select a rod string
- Specify or select a heavy rod

Rod Pump Diagnostics with Dynamometer Card

This module provides the following types of diagnostic analysis based on the dynamometer card data:

- Plunger Operation Condition
- Stress Analysis of Rod String Torque Plots and Pumping Unit
- Balance Condition
- Multiple Dyno. curves at various sections of rod string

The module offers the following diagnostic options:

- Specify or calculate damping factor
- Use the built-in dyno. card formats. (There are four types supported.)

Getting Started

The rod pump design/diagnostics model is an independently developed module that plugs into PIPESIM. Well and fluid data defined in PIPESIM is transferred to the rod pump module and vice-versa.

Reservoir Inflow Performance Relationship (IPR)

PIPESIM transfers the specified IPR to the Rod Pump module.

Well data

PIPESIM transfers well basic data — such as reservoir static pressure, reservoir temperature and tubing configurations — to the Rod Pump module.

Specific data for rod pump

The data required for Rod Pump Design — such as production flowrate and dynamic fluid level — must be specified in the Rod Pump module.

Unit System

PIPESIM transfers any specified units to the Rod Pump module.

Project Save and Import

PIPESIM saves all parameters and data information specified in Rod Pump in the .bps project file.

RP Operation Menu

To access the Rod Pump module, select **Artificial Lift** ➤ **Rod Pump**. The following options are available:

- Design
- Diagnostics
- Options
- Browse/Edit RP Data
- Browse Pumping Unit

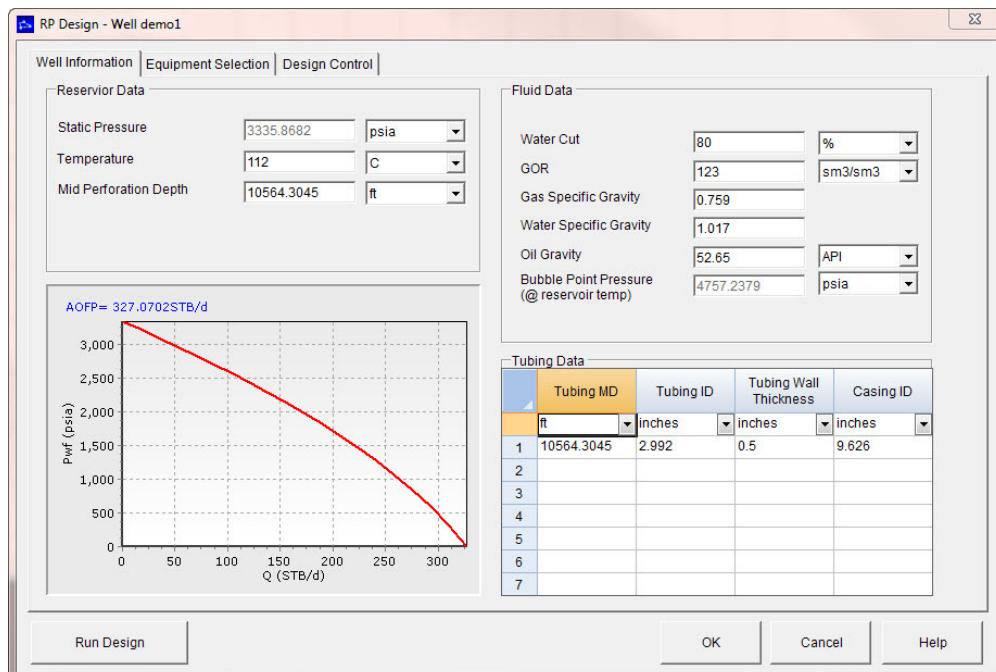
Design...

The **RP Design** dialog box has the following tabs available:

- Well Information
- Equipment Selection
- Design Control

Well Information

The following image shows the properties on the **Well Information** tab.

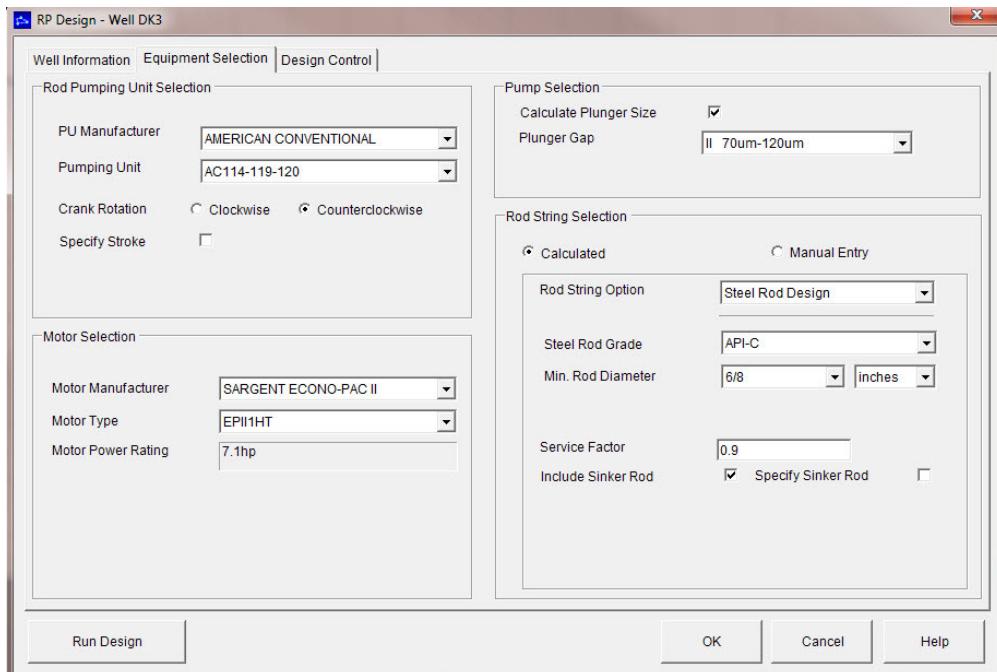


Equipment Selection

Here are a few key points when specifying equipment:

- You should specify the crank rotation direction.
- The stroke depends on pumping unit.
- You can select a different motor for Pumping Unit.
- The speed depends on motor.

The following image shows the properties on the **Equipment Selection** tab.



Simulated Design Results (that is Pressure Profiles)

The following image graphically shows the simulated results.



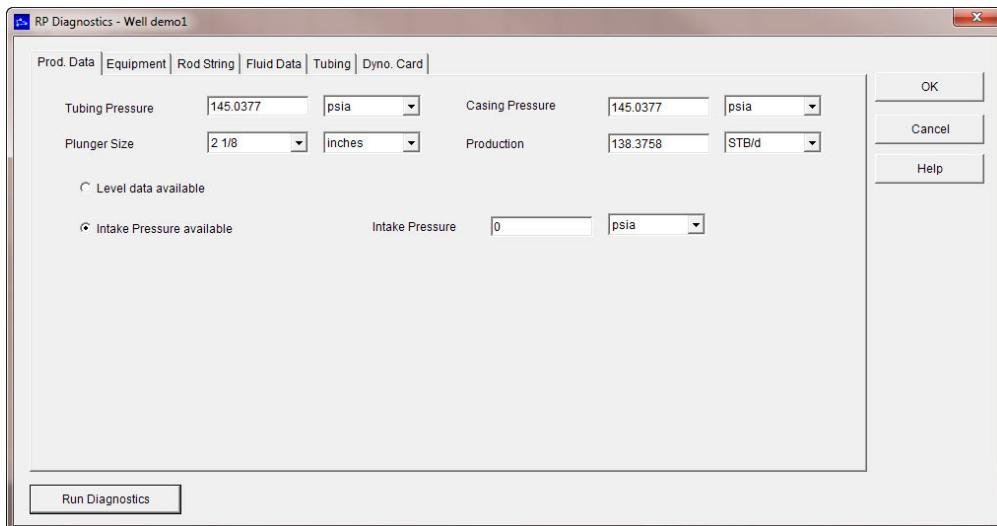
Diagnostics...

To input data, the **RP Diagnostics** dialog box has the following tabs available:

- Prod. Data
- Equipment
- Rod String
- Fluid Data
- Tubing
- Dyno. Card

Prod. Data

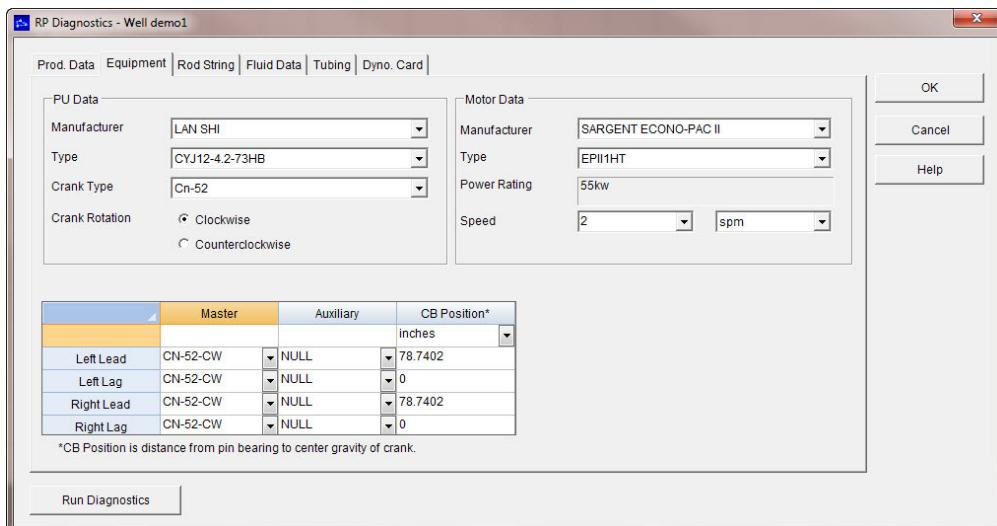
The following image shows the properties on the **Prod. Data** tab.



Equipment

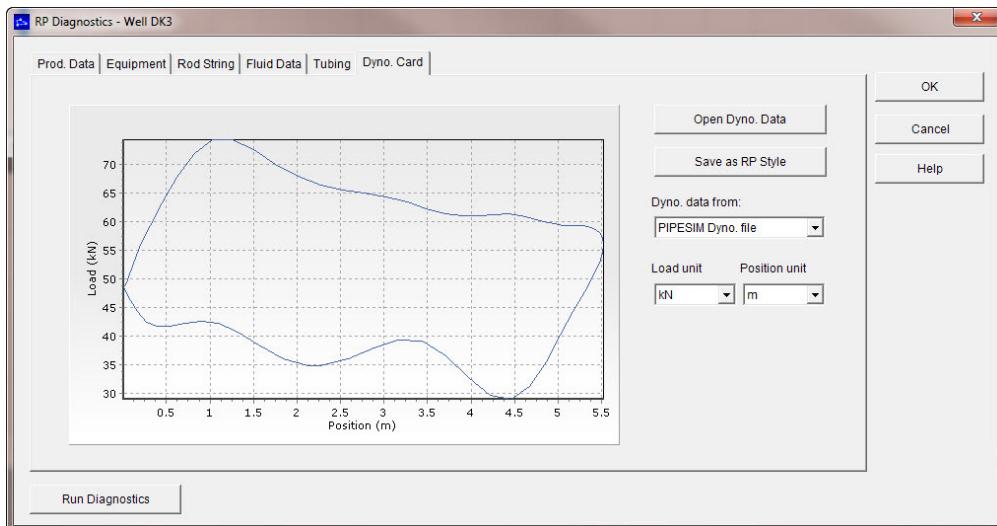
- You should specify crank type and the rotation direction.
- You can specify master weights and auxiliary weights and counterweight positions respectively.

The following image shows the properties on the **Equipment** tab.



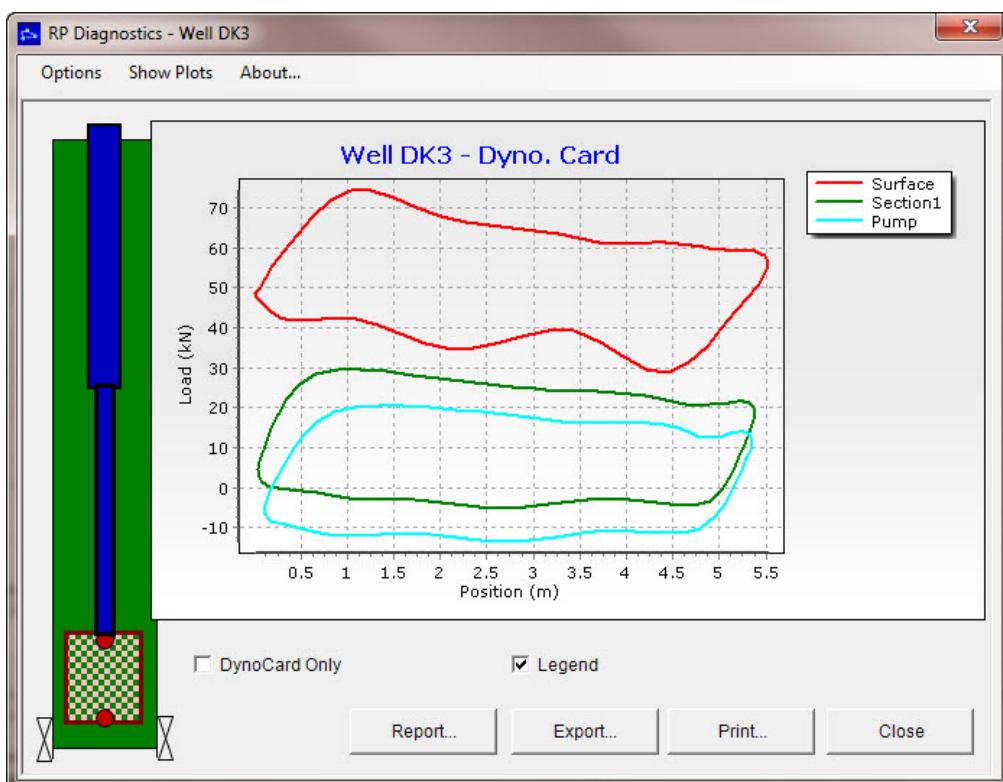
Reading / Loading Dynamometer Card

The following image shows the properties on the **Dyno.Card** tab.



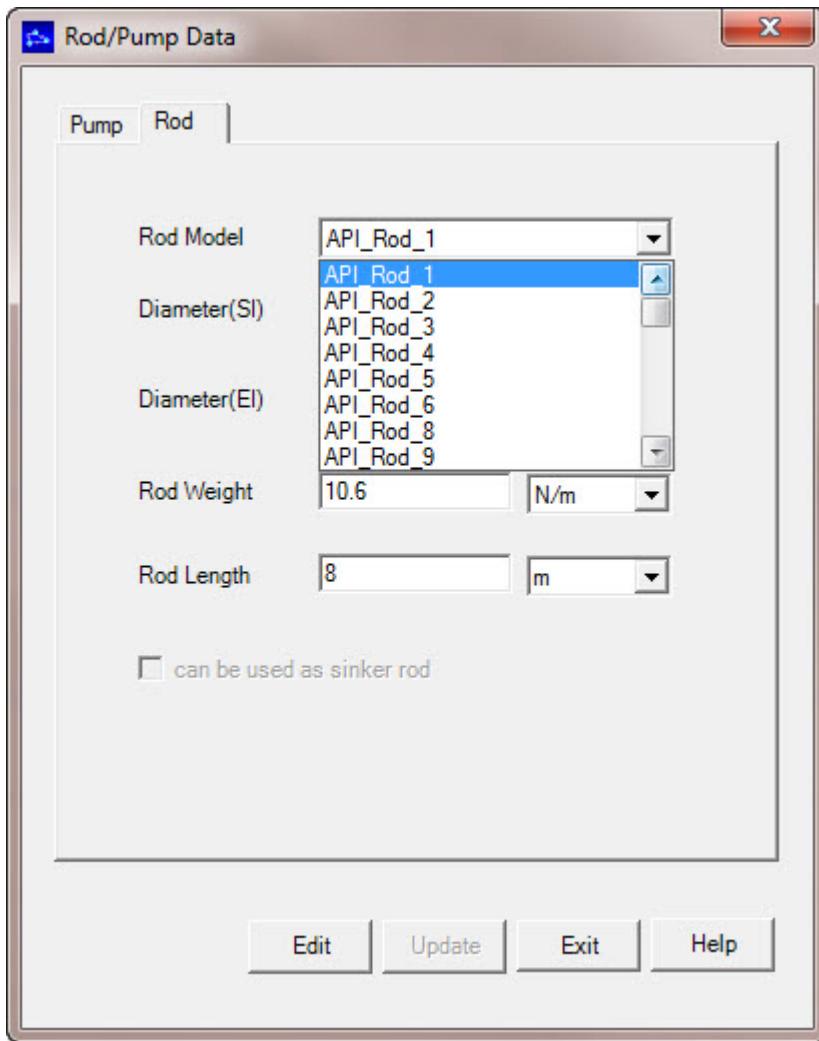
Simulated Diagnostics Results

The following image graphically shows the simulated diagnostic results.



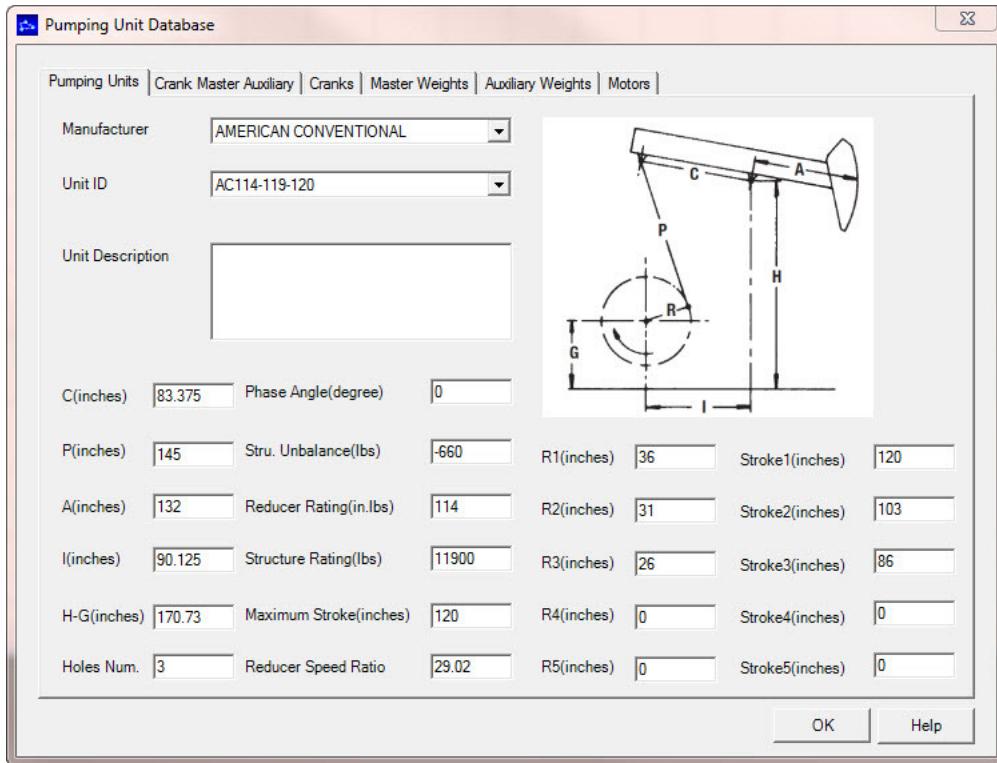
Browse/Edit RP Data...

The following image shows the **Browse/Edit RP Data** dialog box.



Browse Pumping Unit...

The following image shows the **Browse Pumping Unit** dialog box.



Components and Options

This section covers the Rod Pump Components and Data Information.

General Data

Reservoir Inflow Performance Relationship (IPR)

You can use the Rod Pump module for vertical well completion only. Before running RP, you must provide well completion information so the required IPR curve can be calculated. The productivity analysis is the basis of the diagnostic and design study. To get more accurate results,

- For new completion wells, ensure that a proper PI is specified.
- For existing wells, ensure that the IPR matches current production conditions.

Note: Only Productivity Index (PI) and Vogel IPR's are supported.

Basic parameters

The basic Rod Pump parameters refer to the parameters specified in PIPESIM — such as well completion data, tubing data and fluid property (PVT) data. Once you specify these parameters in PIPESIM, the values are transferred to the Rod Pump module automatically. With the exception of well completion data, you can modify the data during the Rod Pump design and Rod Pump diagnostics study. When you exit the Rod Pump GUI, you have the option to save the changes that you made to the PIPESIM well model.

Rod Pump Parameters

Rod Pump parameters refer to the data required by the Rod Pump module. The parameters can be categorized into the following groups:

- Rod Pump equipment data — static characteristic data of the motor, rod string and plunger. You can add your own/local data to this database.
- Production data — the equipment configuration data and the related test data. Before executing Rod Pump Design or Rod Pump Diagnostics study, ensure that you specified the data based on the existing well operating condition.

Note: If any of the required data is missing, the Rod Pump model will not run.

Dynamometer Card (Dyno.Card) Data

The dyno.card data measured at the surface is required data for the Rod Pump Diagnostics study. The surface dyno.card is the relationship curve of rod loading vs. rod displacement. You can get this information from the dynamometer; however, the data file typically contains dyno.card data of more than 10 wells. You must find the dyno.card for the current well and save it in the Rod Pump format so the Rod Pump module can read it.

There are four commonly used dyno.card formats that can be read in the Rod Pump module. If the dyno.card format cannot be read by Rod Pump, convert the format to one of the Rod Pump readable formats.

Units

The unit system in the Rod Pump module is consistent with the units in PIPESIM. For example, if you change the unit of each variable in PIPESIM, and then set it as default, the changes are transferred to Rod Pump automatically.

Rod Pump Design

Overview

There are two types of sucker rod pump systems:

- Beam Pump system
- PC Pump system

The Beam Pump is the most popular artificial lift method because of its simple structure, flexibility and longevity. It has three main parts: pumping unit, sucker rod, and pump. The motor transfers high speed rotation movement to the reducing gear box. After a two-level speed down using three shafts, the rotation movement is transferred to the up and down swing of the beam using a crank connection rod. This movement drives the pump plunger to perform a reciprocal movement using the rod to lift fluid (mainly oil) from the bottom hole to the surface.

The pump unit is the main surface equipment of the rod pump system. The beam-balanced pumping unit transforms the circle movement of the crank to the up and down swing of the horse head using the connection rod between the beam and crank. Structurally, the process can be divided into two types: conventional and forward-established. In the oil fields, the conventional type is the most popular pumping unit. Its main characteristic is a support between the horse head and

crank connection rod. The time for the up and down stroke is the same. Rod Pump is used mainly for conventional pumping system.

The pump is the main part of the rod pump system. It consists of a pump barrel (outer barrel and sleeve), a plunger and a valve (traveling valve and standing valve). Sometimes, the traveling valve is called a discharge valve and the standing valve is called an intake valve. Based on the different structure, the pump can be divided into two groups, the tubing pump and the rod pump.

A polished rod links the steel cable on the horse head with the down hole sucker rod. The polished rod also helps seal the wellhead together with the sealing box. Polished rods are available in two different types: a common type and one that has a larger ID on one side. On the common type rod, you can change the position of the two sides when one side becomes worn; however, if one side has a larger ID, you cannot switch the sides. The larger ID provides better connectivity.

Introduction of Rod Pump Optimization and Design

The sucker rod pump system is a very complex system that has several components — such as a motor with a reducing gear box that drives the pumping unit system, a sucker rod, a down hole pump and tubing, and so on. The optimization and design in Rod Pump includes the optimization of the swabbing parameters — such as pump diameter, stroke, and frequency of stroke — and the selection of the swabbing equipment — such as the structure of the rod. Diversity in the optimization target with various influence factors and the continuity of the changing of design parameters results in a complex design process.

Selection of Swabbing Parameters

You can specify the combination of the rod and swabbing parameters, or you can let the designer select the parameters automatically. If all the parameters are specified by the designer, Rod Pump calculates the production condition based on that data. That is, it uses PIPESIM-RP as a tool to predict the designed scenarios based on experience or other factors.

Note: Sometimes specifying the strike and frequency of stroke is done because these two parameters of the actual rod pump system are not standard. Another reason is because this design method also fits the requirement of sand control.

Selection of Design Methods

There are three design methods in Rod Pump:

- Specify the flowrate — used to meet the requirement of the production allocation of each well
- Specify the intake pressure
- Specify the dynamic fluid level for the well with high GOR — used to improve the pump volumetric efficiency which affects the whole system efficiency

These methods are based on the principle of equivalent intensity, which re-designs the rod, pump, and pump depth without taking into account of the current rod pump system. The rod material can be redesigned as either a common rod or a weighted rod. The optimized solution ensures the stress percentages on the top side of each rod section are equal.

Rod Pump Diagnostics

Introduction

The diagnostic technology of the rod pump system that was proposed by S.G.Gibbs and A.B.Neily is based on the principle of treating the rod as a downhole dynamic signals conduct line and the pump as a transmission device. The working condition of the pump (loading change on the plunger) is transmitted to surface via rod in forms of a stress wave that is received by the dynamometer. By building the wave equation with a damping coefficient, the dyno.card of each cross section of the rod and pump can be deduced. The whole well working condition can be analyzed and judged.

The traditional pump dyno.card recognition method is to use a numerical value to represent dyno.card according to a particular specification. Experience shows that the simple specification cannot classify the dyno.card properly. In the RP module, an NN (Neural Network) method is introduced. It contains a three level neural network technology for the dyno.card diagnostic.

Because of the complexity of the rod pump system, the use of the dyno.card to recognize the pump working condition is not enough. You also need a good understanding of loading the rod and the balance of the pump unit. Comprehensive analysis is added in the Rod Pump diagnostic module by adding the neural network technology. This method produces a more rigorous and comprehensive judgment of the working condition of the well.

Operations

This information is related to Rod Pump general operations, optimized design, working condition, and the diagnostic equipment database.

Rod Pump Database Configuration

The data in Rod Pump database contain the equipment parameters that refer to the static performance data of the motor, rod string, and pump. You may add your equipment data to the Rod Pump database. When you finish adding the new data, we recommend that you do not change the parameters in the database.

Open Project

Before running the Rod Pump module, you must build a new project or open an existing project in PIPESIM.

Basis Input Data in PIPESIM

For Rod Pump module, the following data should be specified in PIPESIM:

- Well completion data (well productivity)
- Tubing data (only for vertical well completion)
- PVT data (only for black oil including P_b)

RP Application from PIPESIM Drop-down Menu

The **Artificial Lift » Rod Pump** menu has the following operations:

- Design
- Diagnostics
- New/Edit RP data
- New/Edit Pumping Unit

Optimization Design

For an old production well, if you do not need to change the pumping unit, choose an existing pumping unit and production manufacturer. For a new production well or for an old well that requires a change in pumping unit, select a model type, and then simulate it to check whether this pumping unit is suitable for the studied well. The model serial number and manufacturer for new pumping units may need to be added to the equipment database.

Selection of Pumping Unit

Selection of Motor

You can specify the motor type or you can let the model select one that matches the selected pumping unit.

Selection of Pump

You can specify a particular pump diameter, or you can let the model select the most suitable pump diameter. If you specify a new pump diameter, you must add it to the equipment database.

Selection of Stroke

You can specify stroke, or you can let the model select. The optimal stroke is selected from the various stroke combinations available in equipment database.

Selection of Pumping Speed

You can specify the pumping speed or you can let the model simulate one. If the optimal simulated speed from the model is not selected from various speed combinations in the equipment database, you must specify the speed.

Selection of Rod String

You can specify a rod combination, or you can let the model select one. A rod combination includes rod length, rod diameter, rod elasticity / tensile strength Grade, and the principles of the rod strength for automatic rod design. If you specify a new rod diameter, enter it into the equipment database.

Specify Pump Depth

No matter what type of design method you select, you must specify a suitable pump depth. You can determine the depth from the well production performance curve and pressure profile or from the corresponding profiles.

Select Design Criteria

The following design principles are available:

- Specified production

- Specified pump inlet pressure
- Specified dynamic liquid level

Once the design principle is chosen, input the specified parameters to continue the design.

Operation Diagnostics

Input Liquid Level or Pump Inlet Pressure

Pump inlet pressure is a compulsory input parameter for diagnostics study. You can input the parameter directly, or you can input the dynamic liquid level and let the model calculate the pump inlet pressure.

Input Rod String Information

The operation diagnostic is only applicable to old production wells. The rod string information is required for the rod combination when the well was in production.

Select Well Dynamometer Card

The well dynamometer card is a required parameter in diagnostic study. You must specify a set of actual tested dynamometer card data.

Specify Damping Coefficients to obtain Pattern

When applying wave equation to solve a pump performance curve, the magnitude of the damping coefficients of the rod influences the calculation results. In the module, it offers the following choices:

- module to automatically calculate damping coefficients
- user specified / input damping coefficients

Database

View Equipment Parameter

To view the equipment database, use the **Artificial Lift » Rod Pump » Browse Pumping Unit** menu. The database contains equipment values that can be plotted.

Delete Equipment

Deletes the selected equipment model from the application.

Add New Equipment

Adds a new model type for the existing equipment. You must specify a new diameter for rod plunger and rod pump. When adding a new pump, ensure that the pump stroke combination and the crank radius combination are equal.

1.9 Reports

1.9.1 Plotting

After a simulation has been completed, the results can be viewed graphically using the plot utility, **PsPlot**.

Single branch plots

Results are (by default) plotted dynamically as the simulation runs. This feature can be enabled or disabled using **Setup** » **Preferences** » **Engine** and selecting **Run Plot tool with Single Branch engine**.

In addition the following plot types are available from the main toolbar:

- System plot — displays data as a result of changing some model parameter.
- Profile plot — displays data in terms of distance from the model's inlet.

Network plots

PsPlot can also be accessed using the **Report** menu or directly from the main toolbar. Select the branches that you want to plot, then select the Report option. The results from the selected branches will be concatenated.

PsPlot

The following list shows what you can do with PsPlot:

- [Plot a 2nd Y-Axis \(p.257\)](#)
- [Print or Copy the plot \(p.257\)](#)
- [Close/minimize the plot utility \(p.257\)](#)
- [Export the data to Microsoft Excel \(p.257\)](#)
- [Remove/Add the background grid \(p.257\)](#)
- [Copy the graph to another application \(p.258\)](#)
- [Change the axis or title description \(p.258\)](#)
- [Change the legend description \(p.258\)](#)
- [Change the values on the axis \(p.258\)](#)
- [List/Disable lines \(p.258\)](#)
- [Scale the axes \(p.259\)](#)
- [Change the color display mode \(p.259\)](#)
- [Change the appearance of a line \(p.259\)](#)
- [Sort data \(p.259\)](#)
- [Zoom in & out \(p.260\)](#)
- [Specify Landscape or Portrait plots \(p.260\)](#)

- Open a new or additional plot file (p.260)
- Remove or change the legend location (p.260)
- Superimpose the phase envelop on to a temperature / pressure profile (p.261)
- Translate plot labels and titles (p.261)

Plot a 2nd Y-Axis

Do the following:

1. Select the **Series** menu option.
2. Select the property to plot on the **Right Y Axis** from the list.

Note: The X Axis remains the same for both Y Axes.

Print or copy the plot

To print, do the following:

1. Select **File » Print**.
2. To preview the plot before printing, or to change any print options, select **File » Print Preview**.

To copy, select **Edit » Copy to Clipboard**.

The plot will be available using the clipboard in other applications that support copy-and-paste.

Close/minimize the plot utility

To Close the PLOT utility, do one of the following:

- Select the close button (top right of the dialog) or
- Select **File » Exit**.

To Minimize the PLOT utility select the minimize button (top right of the dialog).

Exporting graphical data to Excel

Select **File » Export Data to Excel**. An Excel data file name is requested and Excel loads (providing it is installed on the PC) and is populated with the data.

Remove / Add the background grid

Do the following:

1. Select **Edit » Advanced Plot Setup**.
2. In the resulting dialog, select the **Chart, Axis, Ticks** tab. Select the required Axis and then the **Grid Border** button.
3. Change the **Visible** check box to display or hide the grid lines (for that axis).

Changes are dynamic that is the effect of a change can be seen on the plot as they are made without the dialog having to be closed.

Note: These settings are NOT saved between sessions.

Copy the graph to another application

Do the following:

1. Load the required graph and select **Edit** » **Copy to Clipboard**.
2. The plot will now be available using the clipboard in other applications that support copy-and-paste.

Change the axis or title description

Do the following:

1. Select **Edit** » **Advanced Plot Setup**.
2. In the resulting dialog, select the **Chart, Axis** tab and **Title**. Select the Axis to change from the **Axis** radio button and enter the new axis title, angle, style, and so on.

Changes are dynamic that is the effect of a change can be seen on the plot as they are made without the dialog having to be closed.

Note: These settings are NOT saved between sessions.

Change the legend description

Do the following:

1. Select **Edit** » **Advanced Plot Setup**.
2. In the resulting dialog, select the **Chart, Series** tab.
3. Highlight the legend title to change and select title. Enter the new title.

Changes are dynamic. That is, the effect of a change can be seen on the plot as changes are made without the dialog having to be closed.

Note: These settings are NOT saved between sessions.

Change the values on the axis

Select the **Series** menu. Select the property to plot on the Right X Axis from the list. Select the property to plot on the Left Y Axis from the list.

List/Disable lines

Do the following:

1. Select **Edit** » **Advanced Plot Setup**, then choose **Chart** and its **Series** tab.
2. Select the check box for each line you want to plot.

Note: These settings are NOT saved between sessions.

Scale the axes

Do the following:

1. Select **Edit** » **Advanced Plot Setup**.
 2. In the resulting dialog, select the **Chart Axis** tab and scales.
 3. Select the **Axis** to change from the radio buttons and disable **Automatic**.
 4. Set the maximum and minimum values using the **Change** button or set to automatic.
-

Note: These settings are NOT saved between sessions.

Change the color display mode

On the **Display** menu, select one of the following available options:

- **Color by Series Data** - Graph lines are colored according to which Data Series they belong (the default).
 - **Color by Axes Data** - Graph lines are colored according to which axes they belong to (with Data Series differentiated by marker symbol and line style).
 - **Color by File** - Graph lines are colored according to which file they belong (with Data Series differentiated by marker symbol and line style).
-

Note: These settings are NOT saved between sessions.

Change the appearance of a line

Do the following:

1. Select **Edit** » **Advanced Plot Setup**.
2. Select the **Series** tab and then the required Data Series from the drop down list
3. Select the **Format** tab and then the **Color** button to change the line color or the **Border** button to change the line pattern.
4. Select the **Point** tab and then the **Style Combo** to change the style of the marker at each data point.

Changes are dynamic. That is, the effect of a change can be seen on the plot as changes are made without the dialog having to be closed.

Note: Changes to the first five colors and marker styles ARE saved in the registry between sessions.

Sort data

Do the following:

1. Select **Edit** » **Sort Values**.
 2. In the resulting dialog, select the required sorting option.
-

Note: These settings are NOT saved between sessions.

Zoom in or out

Do the following:

Zoom in

On the graph area, place the pointer at the top left position of the area to zoom in to. Select and hold down the left mouse button and drag the resulting box to the required size.

Default size

On the graph, place the pointer at the bottom right position of the area to zoom out to. Select and hold down the left mouse button and drag the resulting box. The plot will revert to its default size.

Zoom out

There is no automatic zoom out feature. [Scale \(p.259\)](#) the x- and y-axis limit to achieve this.

Specify Landscape or Portrait plots

To do this, on the **Print preview** dialog, select the required paper orientation.

Open a new or additional plot file

When a new plot file is to be opened, the existing plot file can be handled in the following ways:

- Retained — to retain the existing plot and add the data to it, select **File** » **Append**.
- Removed — to remove the existing file, select **File** » **Open**.

Opened files can be temporarily made **non-active** by using **Files** » **Files List**.

Remove or change the legend location

Do the following:

1. Select **Edit** » **Advanced Plot Setup**.
2. In the resulting dialog, select the **Legend** tab.
3. Select **Visible** to show or hide the legend
4. Select the required position of the legend.

Changes are dynamic that is the effect of a change can be seen on the plot as they are made without the dialog having to be closed.

Note: These settings are NOT saved between sessions.

Superimpose the phase envelope on to a temperature / pressure profile

To undertake this task, the model must be [compositional \(p.141\)](#). Do the following:

1. Set the point in the system where you want to create the phase envelope. To do this, add a [report tool \(p.115\)](#) to the model and select **Phase Envelope**. The phase envelope will be created at this point in the system
2. Run the model. This will normally be a [Pressure/Temperature operation \(p.196\)](#).
3. Select and display the profile plot, using **Reports » Profile Plot**.
4. Using the **Series** menu, select **Temperature** as the X-Axis and **Pressure** as the Y-axis.

The phase envelope will be superimposed on the plot.

Translate plot labels and titles

Load Dictionary:

- Use the supplied label translation dictionary template (PsPlotDict.txt) to create a translation dictionary, with translation of each label on the same line separated by a comma ','.
- (A usable example is provided for Russian in the file PsPlotDictRus.txt)
- Select **File » Dictionary** to browse for the translation dictionary file and open it.
- The **Dictionary** option will remain selected until you cleared it.

Clear Dictionary:

- To clear the dictionary, select **File » Dictionary** and select **Cancel**

A dictionary cannot be loaded unless it is based on the template and in the correct format.

1.9.2 Report Tool Details

Placing a report tool in a single branch model gives additional reporting of the conditions at that point in the model. To do this, click the Report Tool icon on the main toolbar, then click on the model. Double-click the Report Tool object to open its **Properties** tab. There is a check box for each report item available, as listed below. Select the box for each item you want to report on.

Flow Map

This applies to multiphase flow regions only. If this box is ticked, a high resolution map is produced, showing the multiphase flow regimes plotted against superficial liquid and gas velocities. Note that the map is scaled so that the operating point is always inside it. To display a map, select **Reports » Flow Regime Map** and select it in the list of available maps. The list shows all available maps for the case that has been run.

Phase Split

This applies to compositional cases only. If this box is ticked, a table is printed in the output file, showing the constituents of each phase.

Stock Tank Fluid Properties

If this box is ticked, a table is printed in the output file, showing various fluid properties such as gas and liquid volumetric flowrate at stock tank conditions.

Flowing Fluid Properties

If this box is ticked, a table is printed in the output file, showing various fluid properties such as gas and liquid volumetric flowrate at flowing conditions.

Cumulative Values

If this box is ticked, a table is printed in the output file, showing properties, such as liquid holdup, where the cumulative value may be of interest

Multiphase Flow Values

If this box is ticked, a table is printed in the output file, showing various multiphase fluid flow values such as superficial gas and liquid velocities.

Slugging Values

If this box is ticked, a table is printed in the output file, showing various slug values such as slug lengths and frequencies.

Sphere Generated Liquid Volume

If this box is ticked, a table is printed in the output file, showing various sphere generated liquid values.

Heat Transfer Input Values

If this box is ticked, a table is printed in the output file, showing heat transfer input data. Typically, the following are printed: Pipe and Coating Thicknesses and Thermal Conductivities, Soil Thermal Conductivity, Burial Depth, Ambient Fluid Velocity, and Ambient Temperature.

Heat Transfer Output Values

If this box is ticked, a table is printed in the output file showing heat transfer output data. Typically, the following information is printed: Distance, Fluid Temperature, Fluid Enthalpy, and Heat Transfer Coefficients.

Composition Details

This applies to compositional cases only. If this box is ticked, a table is printed in the output file showing some compositional data such as water specific gravity.

Phase Envelope

This applies to compositional cases only. If this box is ticked, the fluid's phase envelope is automatically computed and stored in the branch's plot file. In network cases, the file that contains the phase envelope will be noted in the output file.

1.9.3 Network Report Tool

To use the report tool, select **Reports** » **Report Tool** (or click the report tool icon directly at the top of the menu bar) after a successful network simulation completes. Its primary function is to provide a report of the computed data in a network simulation. However, it can be also used for single branch analysis. The report includes a node report (single point properties) and a branch report (distributed properties), and they can switch by selecting "Node" or "Branch" in group "Type".

Note: If there is no convergence after completing the specified number of iterations and the last iteration performed is not the best solution, the solver performs another iteration to calculate the best solution from all the iterations. The best iteration is based on where the maximum pressure

and flow rate residual errors are the lowest. PIPESIM publishes the data from the best iteration to the network report. When there is no convergence, the report tool displays a warning message that the model did not converge and it lists the tolerance that was achieved.

All properties for node report are listed below:

Name, Type, Upstream, Downstream, Temperature, Pressure, ST Gas Flowrate (Stock-tank Gas Flowrate), ST Liq Flowrate (Stock-tank Liquid Flowrate), ST GLR (Stock-tank Gas Liquid Ratio), ST Water Cut (Stock-tank Water Cut), Mass Flowrate, Flowing Gas Flowrate (Gas Flowrate at flowing condition), Flowing Liq Flowrate (Liquid Flowrate at flowing condition), Flowing Water Cut (water cut at flowing condition), Mean Vel (Mean Velocity), Liq Vel (Liquid Velocity), Gas Vel (Gas Velocity), Erosion Vel (Erosion Velocity), EVR (Erosion Velocity Ratio).

Note: The meanings of columns “Name”, “Type”, “Upstream”, and “Downstream” are described as below:

Name	Type	Upstream	Downstream
Object label	SOURCE: source or production well	None	None
	SINK: sink or injection well	None	None
	JUNCTION: junction node	None	None
Branch label	INLET: branch inlet	None	Type of downstream equipment
	OUTLET: branch outlet	Type of upstream equipment	None
	INTERNAL: node between any two neighboring equipments in a branch	Type of upstream equipment	Type of downstream equipment

Note: Branch internal nodes in branch “bname” are named as “bname-Node[i]” with index i ascending from upstream to downstream.

All properties for branch report are listed below:

Name, Type, Power (Equipment power (if applicable)), DT (Temperature drop), DP (Pressure drop), Max Press (Maximum Pressure), Max Mean Vel (Maximum Mean Velocity), Max Liq Vel (Maximum Liquid Velocity), Max Gas Vel (Maximum Gas Velocity), Max Eros Vel (Maximum Erosion Velocity), Max EVR (Maximum Erosion Velocity Ratio).

Note: Equipment in branch “bname” is named as “bname-Equip[i]” with index i ascending from upstream to downstream.

The following custom report operations can be applied to the displayed data:

Operation	Description
Clear	Remove all the data in the report. The underlying data file (*.pnsx) is not removed and can be restored using the Restore button.
Delete	Delete highlighted rows in the report. The underlying data file (*.pnsx) is not removed and can be restored using the Restore button.
Restore	Restore all rows to the report which are removed previously.

Print	Print the current custom report.
Plot	Plot the report data (if applicable).
Sort	Sort report rows with respect to user specified properties.
Filters	Filter the report to display row data only with specified types or ranges.
Config	Configure report to determine what columns are displayed.
Excel	Export to an Excel spreadsheet.
Close	Close the report
Browse	Load report files. To load results from a previous simulation, browse to its results file (*.pnsx). The name of the file is displayed in the adjacent dropdown box.

Note: Users can also single click a node or branch object in a network model to add the corresponding results in the report spreadsheet.

Exporting to Excel

In addition to the copy and paste features of spreadsheets in PIPESIM, the Network Report Tool can also export data to Microsoft Excel. When you click the **Excel** button above, an **Export to Excel** dialog asks for the following information:

Excel File

All the data viewed in the Report tool will be exported to the specified Microsoft Excel file.
The Excel file must exist already.

Worksheet

The worksheet, within the Excel file, to export the data to.

Start Row

The row in the Excel spreadsheet where the first data item will be placed. The default is row 1.

Start Column

The column in the Excel spreadsheet where the first data item will be placed. The default is column A.

1.10 Expert Mode

The PIPESIM calculation engines are [keyword \(p.595\)](#) driven, with the keywords normally written by the GUI. However, you can gain access to the keywords using the following methods:

- [Engine keyword tool \(p.94\)](#)
- Expert Mode

1.10.1 Expert Mode

Expert Mode allows the following;

Calculation engine

The calculation engine to be driven directly using [keyword \(p.595\)](#) input files. This could be when, for example, new features are available in the calculation engine but not yet in the Graphical Interface. The keyword input file can either be created, from scratch, using a text editor or from a model already developed in the GUI. [See how \(p.266\)](#) (Only recommended for single branch models).

Network models

A number of Network models can be run in batch mode. This could be to run a set of case studies over night for example.

Single branch models

A number of Single branch models can be run in batch mode, but note the following:

- Models must have been run once already using the GUI
- The operational data used for the last run in the GUI will be used.
- Models in the same directory may have file sharing problems and therefore not run
- You may want to turn off the [automatic dynamic plotting \(p.178\)](#) of results if multiple single branch models are to be run.

1.10.2 How to work in Expert mode

Do the following:

1. Create your basic model(s).
2. Use **File** » **Export Files** to export the models to files. This creates:
 - For a Network model: a *.tnt and associated *.pst files
 - For a single branch model: a *.psm file
3. Select **Expert** » **Display** and select the above *.tnt or *.psm files by using the **Browse** button. Select the **Select** check box for each model, or click **Select All**.
4. Edit the relevant text file(s) by clicking **Edit**.
5. Run the model by clicking **Run**.
6. To view results, click **Output File**, **Summary**, **System Plot**, or **Profile Plot**.

Note: In Expert Mode, System Plot and Profile Plot results are only available for single branch models.

1.10.3 Batch mode

This Expert mode method can also be used to batch run models developed in the GUI.

How to batch run a number of models, using Expert Mode

To batch run a number of models, do the following:

1. Create your basic model using the GUI.
2. Use **File » Export Files** to export the models to files. This creates:
 - For a Network model: a *.tnt and associated *.pst files
 - For a single branch model: a *.psm file
3. Select **Expert... » Display** and select the above *.tnt or *.psm files by using the **Browse** button. Select the **Select** check box for each model, or click **Select All**.
4. Click **Run**.
5. To view results, click **Output File, Summary, System Plot, or Profile Plot**.

1.10.4 Run Multiple Models

To run multiple models, use the **Expert Mode** feature.

Do the following:

1. Use **File » Export Files** to export the models to files. This creates:
 - For a Network model: a *.tnt and associated *.pst files
 - For a single branch model: a *.psm file
2. Select **Expert » Display** to enter Expert mode.
3. Select the above file (*.tnt or *.psm) using the **Browse** button and check the **Select** box, or click **Select All**
4. Select **Edit** to modify the engine keyword file, if necessary.
5. You may want to turn off the [automatic dynamic plotting \(p.178\)](#) of results if multiple single branch models are to be run.
6. Click **Run** to run all the selected models.
7. To view results, click **Output File, Summary, System Plot or Profile Plot**.

[Batch Mode \(p.265\)](#).

1.10.5 Expert Data Entry

Select

check box to activate the chosen model. When the run option is selected all models that have been selected will be run.

Engine Files

A list of the engine files available for running, *.psm for single branch and *.tnt for network models. Engine input files are created either manually or using the **File » Export Files** command.

Browse

locate the required engine files.

Run

Run all the selected models, providing the **Selection only** option is on.

Note: These are run in silent mode (minimized and no completion dialog displayed).

Edit

edit the selected engine file using a text editor and modify the engine keywords. See also the [keyword list \(p.595\)](#)

Output File

view the selected models output file after a simulation.

Summary

view the selected models summary file after a simulation.

System Plot

view the selected models system plot after a simulation.

Profile Plot

view the selected models profile plot after a simulation.

Options**Preferences**

- Select the path of the engine: [Choose Paths \(p.171\)](#)
- Set preferences: [Engine Preferences \(p.178\)](#). You may want to turn off the [automatic dynamic plotting \(p.178\)](#) of results if multiple single branch models are to be run.
- [Advanced \(p.178\)](#)

Select All

Select all models in the list

Deselect All

deselect all models in the list

Selection Only

Only run selected models. Default = on.

2

Tutorials

The following tutorials are described in detail:

- Condensate Pipeline (p.269)
- Oil Well Performance Analysis (p.43)
- Gas Well Performance Analysis (p.304)
- Looped Gas Gathering Network (p.324)
- User Pump (p.334)
- Field Data Matching (p.344)
- Liquid Loading Analysis (p.349)
- Ramey Heat Transfer Model (p.351)
- Pipe Inline Heating (p.354)

Summary of features and tutorials:

Tutorials	Module
Gas Network	Network Analysis (p.35)
Loop (Folders)	Network Analysis (p.35)
Loop Network (p.324)	Network Analysis (p.35)
Small Network	Network Analysis (p.35)
Water Inj. Net	Network Analysis (p.35)
Water Reinjection	Network Analysis (p.35)
Gas Lift (p.217) Performance	Open Link (p.776)
Production Network	Open Link (p.776)
Template	Open Link (p.776)
Condensate Pipeline (p.269)	Pipeline Tools (p.91)
Riser (p.112) and Flowline (p.95)	Pipeline Tools (p.91)

Surface facilities	Pipeline Tools (p.91)
Wax Deposition (p.205)	Pipeline Tools (p.91)
Pump (p.334)	User Equipment (p.638)
UserPump (p.334)	User Equipment (p.638)
Bottom Hole Nodal Analysis (p.200)	Well Design and Performance (p.50)
Compositional (p.141) Nodal Analysis (p.200)	Well Design and Performance (p.50)
ESP (p.235) Design	Well Design and Performance (p.50)
ESP (p.235) Lift	Well Design and Performance (p.50)
Flow Correlation Match (p.197)	Well Design and Performance (p.50)
FracPack (p.68)	Well Design and Performance (p.50)
Gas Lift Design (p.217)	Well Design and Performance (p.50)
Gas Lift (p.217) Performance	Well Design and Performance (p.50)
Horizontal Well Performance (p.664)	Well Design and Performance (p.50)
Injection Well	Well Design and Performance (p.50)
Multilayer Well (p.74)	Well Design and Performance (p.50)
Multiphase Booster (p.102)	Well Design and Performance (p.50)
P and T Profiles (p.196)	Well Design and Performance (p.50)
PCP Lift (p.465)	Well Design and Performance (p.50)
Qgi vs CHP Curve (p.204)	Well Design and Performance (p.50)
Reservoir VFP Tables (p.202)	Well Design and Performance (p.50)
System Analysis (p.196)	Well Design and Performance (p.50)
Wellhead Nodal Analysis (p.200)	Well Design and Performance (p.50)

2.1 Condensate Pipeline Tutorial

The offshore frontier poses some of the greatest technical challenges facing the oil and gas industry, particularly as we venture into ever deeper waters and more remote locations.

Development costs can be substantial and many new production systems must be designed to accommodate subsea multiphase flow across long distances to be economically viable.

Managing costs over extended distances introduces a number of complex risks and reliability becomes a key concern due to high intervention costs and potential for downtime. Characterizing and managing these risks requires detailed multidisciplinary engineering analysis and has led to the emergence of a new field called flow assurance.

Design of subsea tiebacks requires multiphase flow simulation to assure that fluids will be safely and economically transported from the bottom of the wells all the way to the downstream processing plant.

Four flow assurance issues are discussed in this task, including hydrates, heat loss, erosion, and liquid sluging.

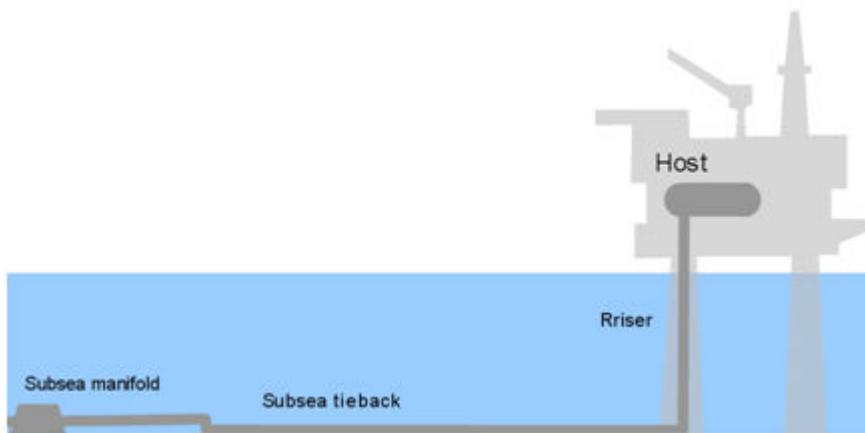
This tutorial involves the following tasks:

- Develop a Compositional PVT Model (p.270)
- Construct the Model (p.272)
- Size the Subsea Tieback (p.274)
- Select Tieback Insulation Thickness (p.277)
- Determine the Methanol Requirement (p.277)
- Screen for Severe Riser Slugging (p.280)
- Size a Slug Catcher (p.285)

2.1.1 Flow Assurance Considerations for Subsea Tieback Design

In this tutorial, a client plans to produce four condensate wells into a subsea manifold through a subsea tieback and up a riser to a platform. The oil and gas will be separated, with the oil pumped to shore and the gas compressed to shore.

Figure 2.1. Subsea Tieback



Task 1: Developing a Compositional PVT Model

In this section, you develop a compositional PVT model based on the data in the tables that follow.

Component	Moles
Carbon Dioxide	3
Methane	72
Ethane	6
Propane	3
Isobutane	1
Butane	1
Isopentane	1

Component	Moles
Pentane	0.5
Hexane	0.5

Table 2.1: Pure Hydrocarbon Components

Name	Boiling Point (degF)	Molecular Weight	Specific Gravity	Moles
C7+	214	115	0.683	12

Table 2.2: Petroleum Fractions

Component	Volume ratio (%bbl/bbl)
Water	10

Table 2.3: Aqueous Component

To develop a Compositional PVT model:

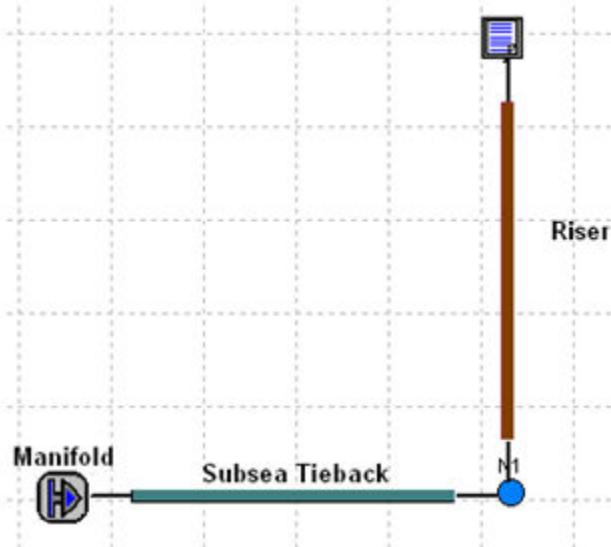
1. Open the **Setup » Compositional Template** menu.
 2. Choose PIPESIM as *PVT Framework*.
 3. Choose Multiflash as *PVT Package*.
 4. To enter the pure components noted in the preceding tables, select the pure hydrocarbon components from the component database.
-
- Note:** Make multiple selections by holding down the **Ctrl** key.
-
5. After selecting all pure hydrocarbon components, click **Add >>**.
 6. Select the Petroleum Fractions tab and characterize the petroleum fraction C7+ by entering these parameters:
 - Petroleum fraction name
 - BP
 - MW
 - SG in Row 1
 7. Highlight the row by clicking Row 1 and click **Add to composition >>**.
 8. Return to the **Component Selection** tab to see that petroleum fraction displays in the component list table on the right.
 9. Click the **Property Model** tab and check the radio button **Use Template Models for all fluids**.
 10. Select **SRK Equation of State** and **Pedersen viscosity model**. Leave all other options as default.
 11. Select **Setup » Compositional Local Default** and add mole fractions for all library and pseudo components, as shown in the above tables.
 12. To generate the hydrocarbon phase envelope, click **Phase Envelope**.

Task 2: Constructing the Model

In this section, you construct the subsea tieback model.

To construct the model:

1. Using the Single Branch toolbar, insert the objects shown in the figure below.



2. Specify each object based on the data provided in the tables that follow.

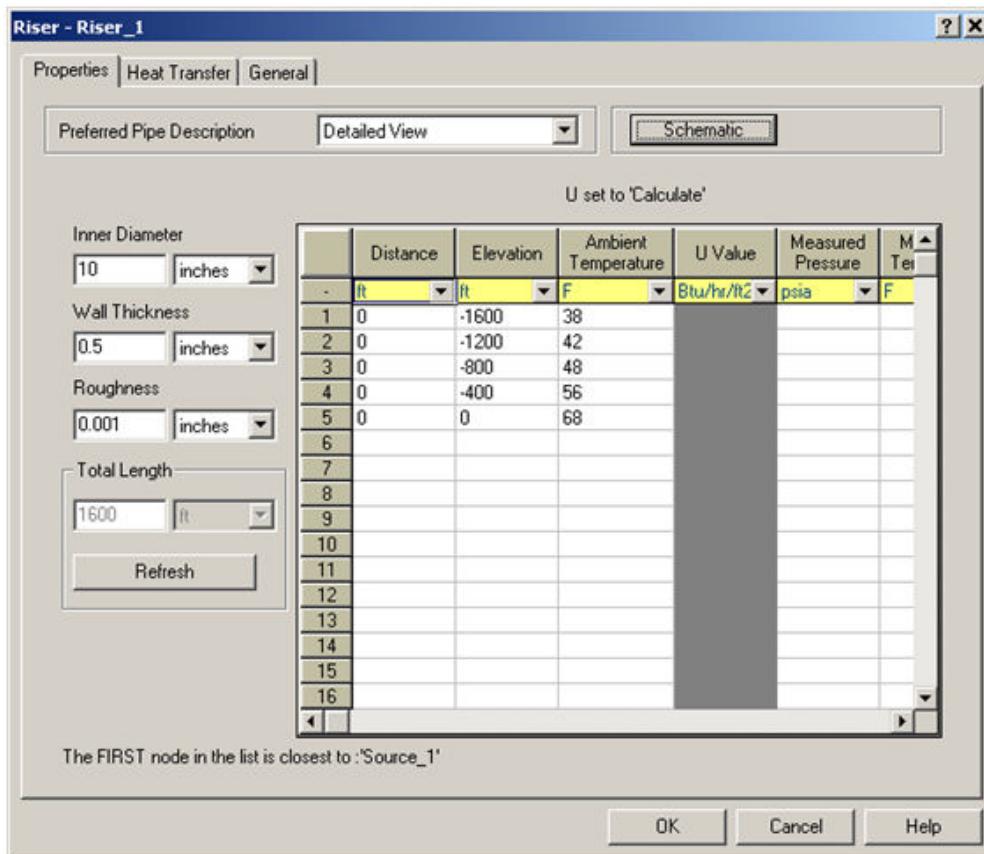
Note: To enter the detailed heat transfer data in the flowline and riser, select the Heat Transfer tab and click Calculate U value. Ensure that your Riser Elevation survey matches that shown below.

Manifold Data	
Temperature	176 degF
Pressure	1,500 psia

Subsea Tieback Data	
Rate of undulations	0'/1000 feet (not hilly)
Horizontal Distance	6 miles
Elevational difference	0 feet (horizontal)
Available IDs	9,10,11 inches
Heat Transfer	
Ambient temperature	38 degF
Pipe thermal conductivity	35 Btu/hr/ft/degF
Insulation thermal conductivity	0.15 Btu/hr/ft/degF
Insulation thicknesses available	0.50 in + 0.25 in increments

Subsea Tieback Data	
Ambient fluid	water
Ambient fluid velocity	1.5 ft/sec
Burial depth	Leave it blank (Elevated above ground)
Ground conductivity	1.5 Btu/hr/ft/degF

Riser (use detailed profile)	
Horizontal Distance	0 feet (vertical pipe)
Elevational difference	1,600 feet
Available IDs	9,10,11 inches
Heat Transfer	
Ambient temperature @ riser base	38 degF
Ambient temperature @ 1,200 feet	42 degF
Ambient temperature @ 800 feet	48 degF
Ambient temperature @ 400 feet	56 degF
Ambient temperature @ topsides	68 degF
Pipe thermal conductivity	35 Btu/hr/ft/degF
Insulation thermal conductivity	0.15 Btu/hr/ft/degF
Insulation thickness	0.50 in (plus additional 0.25 in increments if required)
Ambient fluid	water
Ambient fluid velocity	1.5 ft/sec



Task 3: Sizing the Subsea Tieback

You will now determine the required ID for the subsea tieback, such that the separator pressure for the maximum expected rate is no less than 400 psia.

The expected production rate is 14,000 STBD. The system will be designed to accommodate between 8,000 STBD (turndown case) and 16,000 STBD, should the wells produce more than expected.

The riser must be the same ID as the tieback, and you must not exceed the erosional velocity.

To size the subsea tieback:

- From the **Setup > Flow Correlations** menu, make the following selections:
 - Vertical Flow Correlation = Hagedorn Brown (Duns & Ros map)
 - Horizontal Flow Correlation = Beggs-Brill Revised
- Perform a System analysis with the minimum, maximum, and expected flow rates as the X-axis variable and the available IDs for the flowline and riser as Change in Step (with Sensitivity variable 1) sensitivity variables.
- Determine the minimum flowline ID that satisfies the separator pressure requirement (400 psia) for the maximum flow rate.
- Change the Y-axis to display Erosional Velocity Ratio Maximum.

5. Verify that the selected flowline ID does not exceed an erosional velocity ratio of 1.0 for the expected flow rate.

Property	Value
Pipeline and Riser ID	10 inch
Max. erosional velocity ratio for selected ID	0.828
Min. Separator pressure for selected ID	946 osua
Max. separator pressure for selected ID	1,265 psia

Table 2.4: Results

2.1.2 Hydrates

Gas hydrates are crystalline compounds with a snow-like consistency that occur when small gas molecules come into contact with water at below a certain temperature. Hydrate formation temperature increases with increasing pressure, therefore, hydrates risk increases at higher pressures and lower temperatures. When hydrates form inside the pipeline, the flow can be blocked by hydrate plugs.

Hydrate forming molecules most commonly include methane, ethane, propane, carbon dioxide, and hydrogen sulfide.

Three hydrate crystal structures have been identified - Structures I, II, and H. The properties of Structures I and II hydrates are well defined. Structure H hydrates are relatively new, and their properties are less well defined.

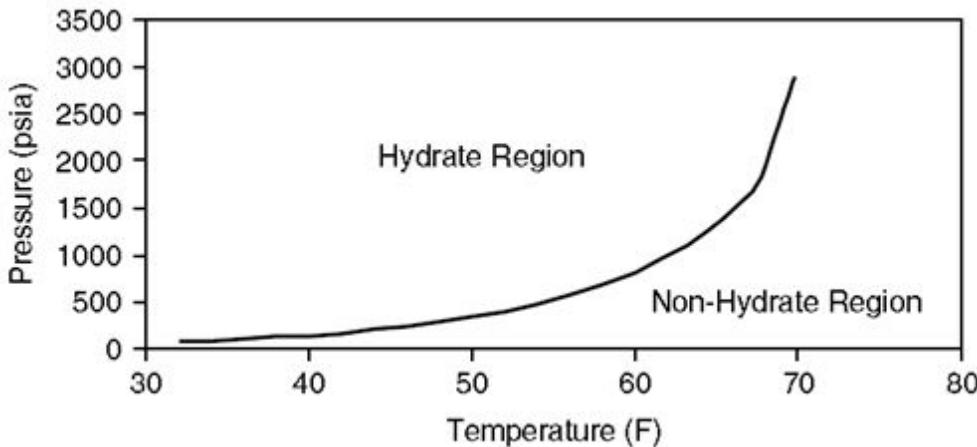
Hydrates can very easily form downstream of a choke where fluid temperature can drop into the hydrate formation region due to Joule-Thompson cooling effects.

The Hydrate curve (below), shows a typical gas hydrate curve which is very useful for subsea pipeline design and operations. On the left side of the curve is the hydrate formation region. When pressure and temperature are in this region, water and gas will start to form hydrate.

Many factors impact the hydrate curve, including fluid composition, water salinity and presence of hydrate inhibitors.

Note: Generating Hydrate curves requires the PIPESIM Multiflash Hydrate Package.

Figure 2.2. Hydrate curve



Hydrate Mitigation Strategies in PIPESIM

Two common strategies available in PIPESIM to mitigate hydrates formation are thermal insulation and chemical inhibitors. Thermal insulation carries a higher upfront capital cost whereas chemical inhibition carries a higher operational cost.

Thermal insulation

Heat transfer between the fluid and surroundings occurs, depending upon the temperature gradient. There are two options for modeling the heat transfer in PIPESIM – **Input U value** and **Calculate U value**.

Input U value is an overall heat transfer coefficient (U value) based upon the pipe outside diameter is entered.

Calculate U value includes the following information, which can be entered to compute the overall Heat Transfer coefficient.

- Pipe coatings
 - Thickness of the pipe coat.
 - K (Thermal conductivity) of the material
- Pipe conductivity
- Ambient fluid (Air or Water)
- Ambient Fluid Velocity
- Pipe burial Depth
- Ground conductivity (for flowlines only)

Chemical Inhibitors

Thermodynamic inhibitors can be used to shift the hydrate curve towards the left, thereby lowering the hydrate formation temperature. Examples of inhibitors include methanol and ethylene glycol.

Kinetic and anti-agglomerate inhibitors comprise a category known as Low Dosage Hydrate Inhibitors (LDHIs). These inhibitors do not lower the hydrate formation temperature; instead, they help prevent the nucleation and agglomeration of hydrates to avoid blockage formation. The effects of these types of inhibitors cannot be modeled with PIPESIM.

Task 4: Selecting Tieback Insulation Thickness

Using the tieback/riser ID selected above, determine the thickness of the insulation required for both the flowline and the riser, such that the temperature of the fluid does not cross the hydrate curve for all possible flow rates.

To select tieback insulation thickness:

1. Double-click on the **Report** tool and ensure that **Phase Envelope** is selected.
2. Select **Operations » Pressure/Temperature profile**.
3. Specify **Separator (outlet) pressure** as the calculated variable and the three design flow rates as the sensitivity variables.
4. Use the **Series** menu on the resulting plot to change the X-axis to **Temperature** and the Y-axis to **Pressure** to display the phase envelope.
5. Observe the production path on the phase envelope and its proximity to the hydrate curve.
6. If required, perform successive runs while increasing the insulation thickness of both the flowline and riser by 0.25 inch increments until sufficient.

Property	Value
Req. Insulation thickness	0.75 in

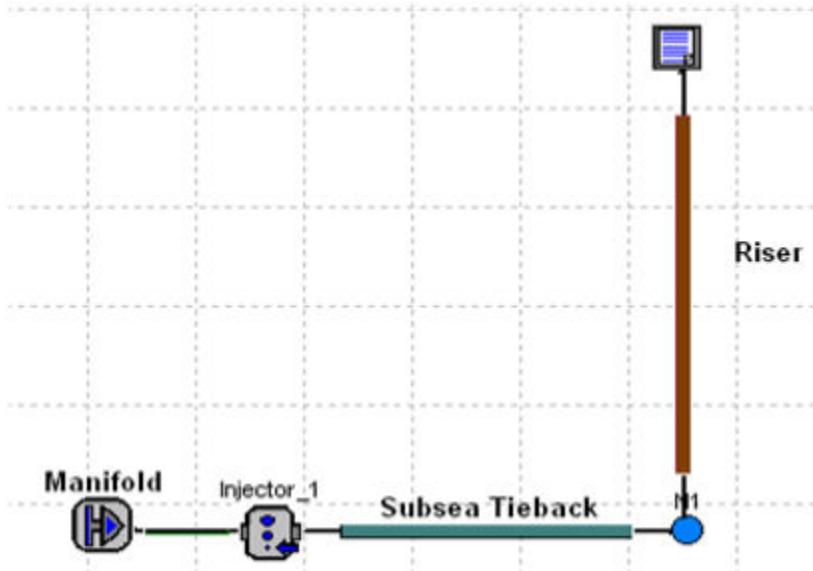
Table 2.5: Results

Task 5: Determining the Methanol Requirement

Assume the flowline and riser have been insulated but they are under-insulated with only 0.25 inch of insulation. In this section, you determine the required injection volume of methanol to ensure that hydrates do not form.

To determine the methanol requirement:

1. Insert an injector just downstream of the source, as shown in the figure below.



2. Specify Methanol as Injector Fluid.
3. (Use injection temp. = 68 degF). To do this:
 - a. Select **Setup** » **Compositional Template**.
 - b. Add Methanol to the listed of added components.
 - c. Double-click the **Injector** and choose **Edit Composition**.
 - d. Specify a composition of **100% Methanol**.
 - e. Specify **Injection Temperature** and any **injection rate**.
4. Select **Setup** » **Heat Transfer Options**, and verify that **Enable Hydrate Sub-Cooling Calculation** is selected.
5. Select **Operations** » **System Analysis**.
 - a. Specify a liquid rate of 8,000 BPD and select **calculated variable** as the outlet pressure.
 - b. For the X-axis variable, select the **Injector** as the object and **Rate** as the Variable.
 - c. Select **Range** and enter a range of 200 to 600 BPD in increments of 50 BPD.
 - d. Uncheck the active status on all sensitivity variables with defined values.
 - e. Run the model.
6. On the resulting plot, change the Y-axis to display Maximum Hydrate Subcooling Temperature.
7. From the plot, determine the required Methanol injection rate, such that the flowing temperature is always above the stable hydrate temperature.

Note: A Positive Hydrate Sub-cooling in the output file indicates the fluid temperature is below the hydrate stability temperature.

Property	Value
Req. Methanol Injection Volume (bbl/d)	~308

Table 2.6: Results

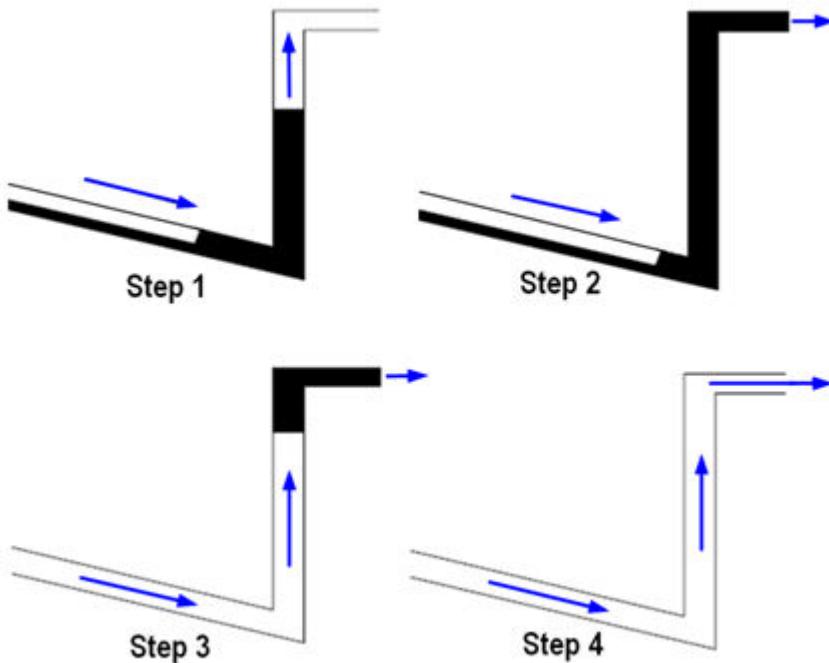
2.1.3 Severe Riser Slugging

Severe slugging in risers can occur in a multiphase transport system consisting of a long flowline followed by a riser. Severe slugging is a transient phenomenon that can be split into four steps, as shown in the next figure, **The four slugging steps**.

Step 1:	Slug formation corresponds to an increase of the pressure in bottom of the riser. The liquid level does not reach the top of the riser. During this period, the liquid is no longer supported by the gas and begins to fall, resulting in blockage to the riser entrance and pipeline pressure buildup, until the liquid level in the riser reaches to the top.
Step 2:	In slug production, the liquid level reaches the riser outlet, and the liquid slug begins to be produced until gas reaches the riser base.

Step 3:	In bubble penetration, gas is again supplied to the riser, so the hydrostatic pressure decreases. As a result, the gas flow rate increases.
Step 4:	This corresponds to gas blowdown. When the gas produced at the riser bottom reaches the top, the pressure is minimal and the liquid is no longer gas-lifted. The liquid level falls and a new cycle begins.

Figure 2.3. The four slugging steps



PIPESIM does not rigorously model severe slugging associated with risers, as this is a transient phenomena, but it does report a dimensionless indicator of the likelihood of this occurring (PI-SS number in PIPESIM output file).

Severe slugging is most prevalent in cases in which a long flowline precedes a riser, especially for cases in which the flowline inclination angle is negative going into the riser.

In cases of severe slugging, the slug catcher must be able to receive a volume of liquid at least equal to the volume of the riser. However, severe slugging can be mitigated by topsides choking or riser base gas lift including self-lifting mechanisms.

PI-SS Indicator (Severe-Slugging Group)

$$\text{PI-SS} = \frac{ZRT}{g L_F \bar{\alpha}_{GF}} \frac{W_G}{W_L}$$

where,

Z = Gas compressibility factor

R = Gas universal constant

T = Temperature (K)

M = Molecular weight of gas

W_G = Gas mass flow rate (kg/s)

W_L = Liquid mass flow rate (kg/s)

g = Acceleration due to gravity (m/s²)

LF = Flowline length (m)

$\overline{\alpha}_{GF}$

= Average flowline gas holdup

Severe slugging is expected when the Pots' number is equal to, or less than, unity. Pots' model can be used to determine the onset of severe slugging, but the model cannot predict how long the severe slugs will be and how fast severe slugs will be produced into the separator.

The PI-SS indicator is available as part of the PRIMARY output in PIPESIM.

Task 6: Screening for Severe Riser Slugging

To screen for severe riser slugging:

1. Deactivate the methanol injector and reset the insulation thickness to that determined to prevent hydrate formation.
2. Under **Setup > Define Output**, select three cases to print. This reports the full output of each sensitivity value with the Report tool selections appended to the bottom of each sensitivity output.
3. Perform a System analysis with an inlet pressure of 1,500, outlet pressure calculated and liquid rates of 8,000; 14,000 and 16,000 BPD.
4. To check for severe slugging:
 - a. Configure the Y-axis of the System analysis plot to display the PI-SS number. This represents the maximum value of the PI-SS number along the flowline.
 - b. View the Output report by selecting **Reports > Output File**, to determine the prevalent flow regime at the riser base for the different rates.

Severe Slugging	8,000 stb/d	14,000 stb/d	16,000 stb/day
PI-SS number at riser base	0.938	1.180	1.263
Flow pattern at riser base	Intermittent	Intermittent	Intermittent

Table 2.7: Results

2.1.4 Slug Catcher Sizing

PIPESIM is frequently used to estimate the capacity requirements for slug catchers. More detailed analysis is typically performed with transient simulators such as OLGA. For offshore platforms, you

must balance the high cost of added weight to the platform with the potential of a large slug overwhelming the liquids handling capacity and shutting down the entire system.

There are three typical scenarios to consider in the sizing of slug catchers for this type of system:

- Hydrodynamic slugging
- Pigging
- Ramp-up

Hydrodynamic Slugging

Most multiphase production systems will experience hydrodynamic slugging. Designing systems simply to avoid hydrodynamic slugging, such as larger pipe ID, is not a common practice. Because hydrodynamic slugs grow as they progress through the pipe, long pipelines can produce very large hydrodynamic slugs.

PIPESIM calculates the mean slug length as a function of distance traveled by using the SSB or Norris Correlations. A continuous intermittent flow regime is required for this to occur. A probabilistic model (again, based on Prudhoe Bay field data) is applied to calculate the largest slug out of 10, 100 and 1,000 occurrences. The 1/1000 slug length is often used to determine slug catcher volume requirement.

The slug output from PIPESIM yields the length and frequency for the selected slug size correlation:

- Mean slug length (distribution is assumed skewed log normal)
- 1 in 1,000 slug length and frequency
- 1 in 100 slug length and frequency
- 1 in 10 slug length and frequency

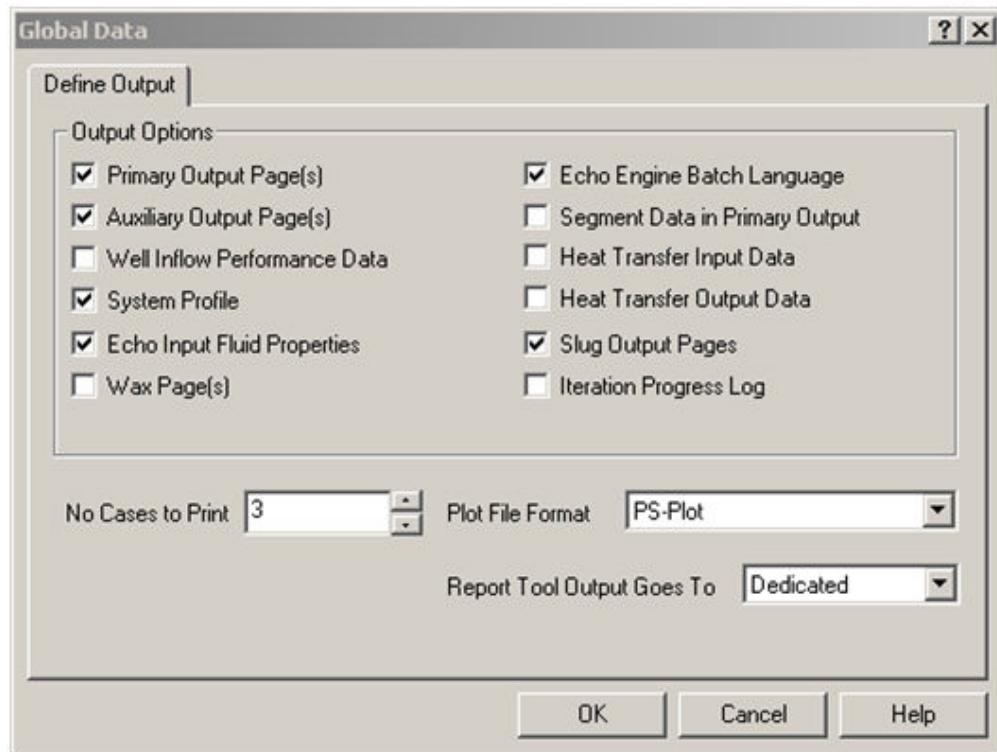
The preceding probabilities represent various levels of confidence regarding the maximum slug size. For example, a 1 in thousand slug length of 50 meters indicates there is only 0.1% probability of the maximum slug length exceeding 50 meters.

Symbols that can be included in the slug output have the following meanings:

0.0	Flow is not in a slugging regime (as calculated by the relevant flow map correlation at spot report) and, thus, no hydrodynamic slugs are required.
N/A	The slug length calculated using the chosen slugging correlation is negative and, therefore, slug size is indeterminate at this point in the flowline.

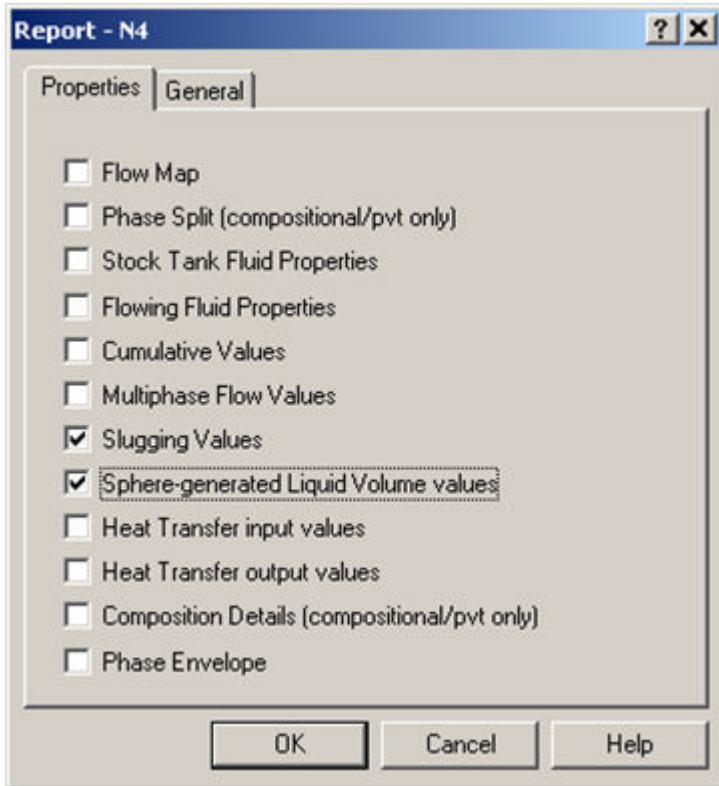
It should be noted that the slug size data output is only printed if SLUG is specified in the **Windows** menu option **Define Output**.

Figure 2.4. Define Output menu options



Alternatively, you can insert the Report tool and check Slugging values and Sphere-generated Liquid Volume values, as shown in the next figure.

Figure 2.5. Selecting report properties

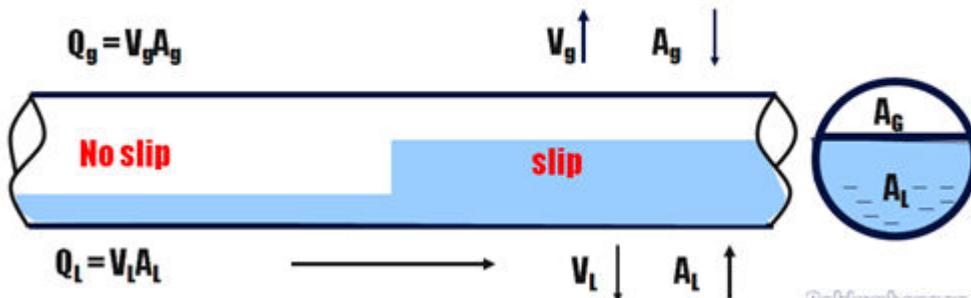


Pigging

In multiphase flow in horizontal and upwards inclined pipe, the gas travels faster than the liquid due to lower density and lower viscosity. This is called slippage. Multiphase flow correlations predict the 'slip-ratio' which depends on many factors, such as fluid properties, pipe diameter and flow regime.

In steady-state flow, the gas travels faster, so it will slip past the liquid and occupy less pipe volume. This gives rise to a higher liquid volume fraction than if the gas traveled at the same velocity, resulting in 'liquid holdup,' as illustrated in the Liquid Holdup figure, below.

Figure 2.6. Liquid Holdup

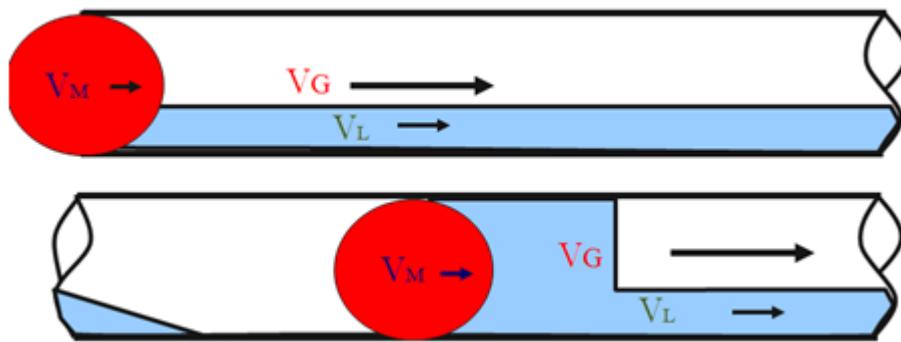


During a pigging operation, a solid object the diameter of the pipeline is sent through the line to push out liquids and debris. As a pipeline is pigged (see next figure), a volume of liquid builds up ahead of the pig and is expelled into the slug catcher as the pig approaches the exit.

PIPESIM considers that the pig travels at the mean fluid velocity and, thus, the volume of liquid that collects ahead of the pig is a function the degree of slip between the gas and liquid phases (such as magnitude of liquid holdup). PIPESIM reports this volume as the sphere generated liquid volume (SGLV). The slip ratio (SR) is also reported, which is the average speed of the fluid divided by the speed of the liquid.

The volume of liquid expelled at the receiving terminal as a result of pigging can be estimated using steady-state analysis as a first order approximation.

Figure 2.7. Pigging operation



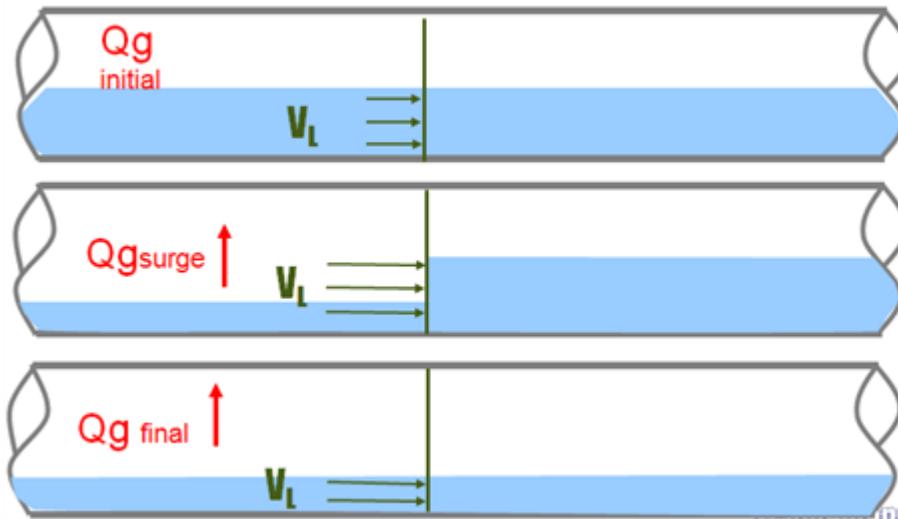
Ramp-up

When the flow rate into a pipeline increases, the overall liquid holdup typically decreases because the gas can more efficiently sweep out the liquid phase. When a sudden rate increase (ramp-up) occurs, the liquid volume in the pipeline is accelerated resulting in a surge.

A ramp-up operation is illustrated in the next figure. The size of the surge is influenced by the sensitivity of liquid holdup with respect to the overall flow rate. A simple material balance approach can be applied to estimate the volume of the associated surge.

For more details, see [Cunliffe's method \(p.393\)](#).

Figure 2.8. Ramp-up operation



Evaluating Each Scenario

For a more detailed analysis of slug catcher sizing, you should also consider the drainage rates of the primary separator and slug catcher. Hydrodynamic slugs and pig-generated slugs typically occur over a short duration (minutes), while the surge created by a ramp-up operation can be a long duration (hours/days).

Task 7: Sizing a Slug Catcher

In this section, you screen for severe slugging and determine the required size of the slug catcher based on the largest of the following criteria, multiplied by a safety factor of 1.2.

Consider these criteria:

- Hydrodynamic slugging, which is the requirement to handle the largest slugs envisaged, chosen to be statistically the 1/1000 population slug size. This is determined by using the SSB or Norris Correlations.
- The requirement to handle liquid swept in front of a pig.
- Transient effects, such as the requirement to handle the liquid slug generated when the production flow is ramped up from 8,000 to 16,000 STB/D, such as Ramp-up surge.

Note: For the purposes of sizing a slug-catcher, it is assumed that severe riser slugging can be mitigated with topsides choking or riser-based gas lift.

To size the slug catcher:

1. In the Report tool, verify that slugging values and sphere generated liquid volume are selected.
2. Re-run the System analysis configured in the previous .
3. For each sensitivity value, scroll down and read the reported 1/1000 slug volume and the Total Sphere Generated Liquid Volume So Far.
4. For the ramp-up case, calculate the difference in total liquid holdup, as this will be the surge volume. You must convert from ft³ > bbl. The conversion factor is 5.615 ft³/bbl.

Note: The surge associated with ramp-up occurs over a much longer time period than the other cases. The ramp-up volume does not consider the drainage rate of the separator or the duration of the ramp-up. See [Cunliffe's method \(p.393\)](#) for information on how to calculate the ramp-up duration.

5. Inspect the output file and observe the flow regimes along the profile for each case.
6. Based on the results in the table below, select a slug catcher size that will be able to handle the largest slug volume for all conditions.

Slug Catcher Sizing	8,000 stb/d	14,000 stb/d	16,000 stb/d
1/1000 slug volume (bbl)	164	176	207
Sphere generated liquid volume (bbl)	467	437	429
Ramp-up volume (bbl)	$982 - 812 = 170$		
Design volume for slug catcher (bbl) (use 20% safety factor)	$467 * 1.2 = 560$		

Table 2.8: Results

2.2 Oil Well Performance Analysis Tutorial

This module examines a producing oil well located in the North Sea. You analyze the performance of this well using NODAL analysis, calibrate black oil fluid (low GOR) using laboratory data, and match flow correlations with pressure survey data.

You will also analyze the behavior of the well with increased water cut and find an opportunity to inject gas at a later stage when the well is unable to flow naturally.

This tutorial the following tasks:

- [Build the Well Model \(p.287\)](#)
- [Perform NODAL Analysis \(p.49\)](#)
- [Perform a Pressure/Temperature Profile \(p.291\)](#)
- [Calibrate PVT Data \(p.294\)](#)
- [Evaluate Gas Lift Performance \(p.298\)](#)
- [Work with Multiple Completions \(p.299\)](#)
- [Modeling a Flow Control Valve \(p.304\)](#)

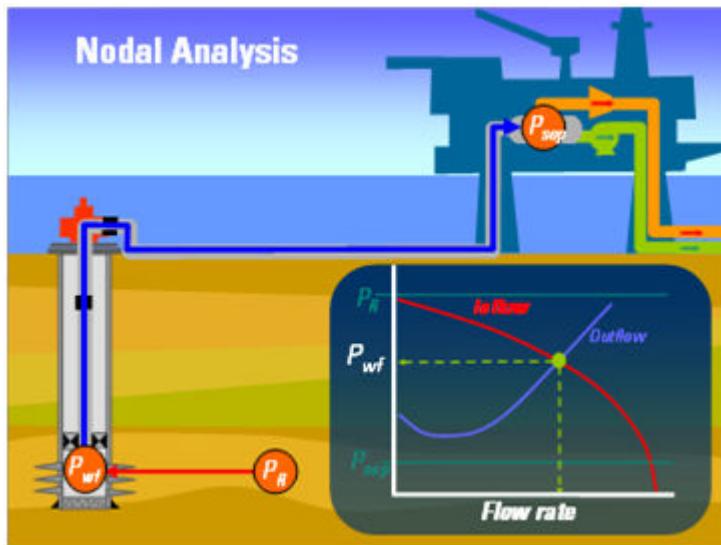
2.2.1 NODAL Analysis

NODAL analysis is used to evaluate the performance of an oil well. It involves specifying a nodal point, usually at the bottomhole or wellhead, and dividing the producing system into two parts - the inflow and the outflow. This is represented graphically in the next figure, Intersection points of the inflow and outflow performance curves .

The solution node is defined as the location where the pressure differential upstream (inflow) and downstream (outflow) of the node is zero.

Solution nodes can be judiciously selected to isolate the effect of certain variables. For example, if the node is taken at the bottomhole, factors that affect the inflow performance, such as skin factor, can be analyzed independently of variables that affect the outflow, such as tubing diameter or separator pressure.

Figure 2.9. Intersection points of the inflow and outflow performance curves



Getting Started

Before beginning an oil well performance analysis:

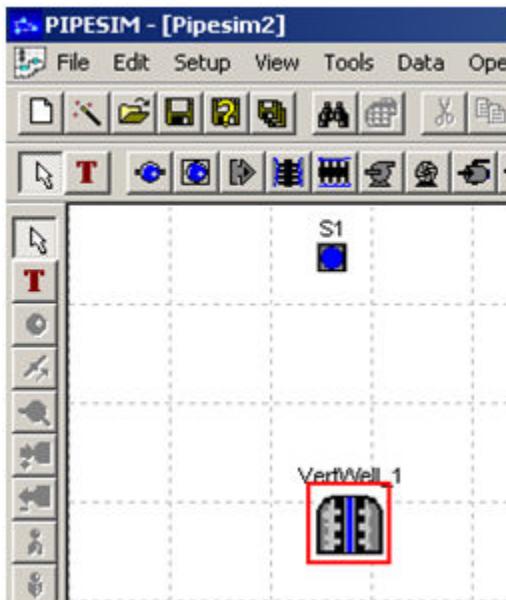
1. Select **File » New » Well Performance Analysis**.
2. From **Setup » Units**, set the engineering units.

Task 1: Building the Well Model

Model building refers to setting up all objects, from the source to the sink, and defining the properties of these objects. You can select PIPESIM single branch objects using either the Tool menu or the toolbar at the top of PIPESIM window.

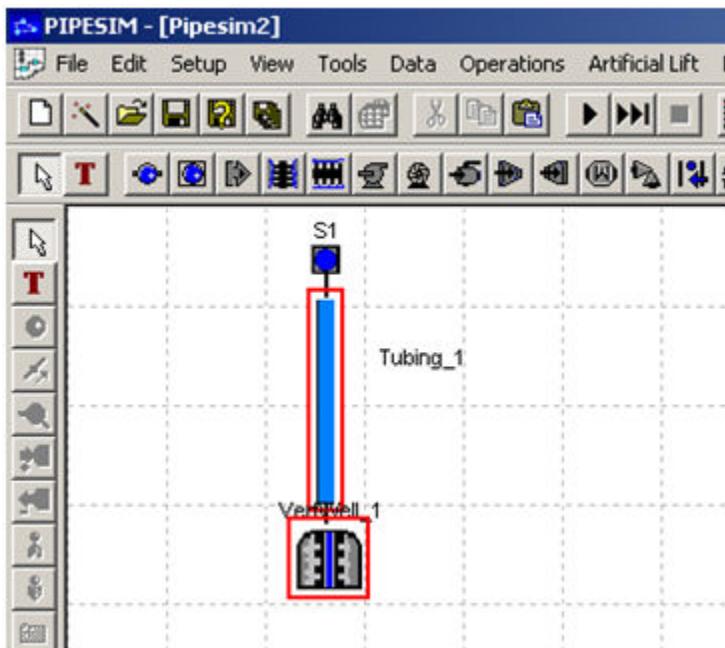
To build the well model:

1. Select a **Vertical Completion** object from the single branch toolbar, and place it in the **Single Branch** flow diagram.
2. Select a **Boundary Node** and place it in the flow diagram.



3. Select a **Tubing** object and connect **VertWell_1** to the **End Node S1** by clicking and dragging from **VertWell_1** completion to the **End Node S1**.

Note: The red outlines on VertWell_1 and Tubing_1 indicate that essential input data are missing.



4. Double-click on the completion and enter the properties listed in the table.

Completion model	Well PI
Use Vogel?	Yes
Reservoir Pressure	3,600 psia
Reservoir Temperature	200 degF
Liq. Productivity Index	8 stb/d/psi

Table 2.9: Reservoir and Inflow Data

- Double-click on the tubing object and enter the tubing properties based on data listed in the table.

Measured Depth (ft)	True Vertical Depth (ft)
0	0
1,000	1,000
2,500	2,450
5,000	4,850
7,500	7,200
9,000	8,550

Table 2.10: Deviation Data

Measured Depth (ft)	Ambient Temp. (degF)
0	50
9,000	200

Table 2.11: Geothermal Gradient

Bottom MD (ft)	Internal Diameter (inches)
8,600	3.958
9,000	6.184

Table 2.12: Tubing Data

- Specify an Overall Heat Transfer Coefficient = 5 btu/hr/ft²/F (override the default value).

Note: You can use the overall heat transfer coefficient to calculate total heat transfer through the pipe wall. The overall heat transfer coefficient depends on the fluids and their properties on both sides of the wall, as well as the properties of the wall and the transmission surface.

- Click the **Summary table** button to observe the configuration summary.
- Set the Distance between nodes to 100 ft.
- Select **Setup ➤ Black Oil**.
- Enter the fluid properties, as shown in the table. Assume default PVT correlations and no calibration data.

Water Cut	10 %
GOR	500 scf/stb
Gas SG	0.8
Water SG	1.05
Oil API	36 °API

Table 2.13: Black Oil PVT Data

Note: The fluid physical properties are calculated over the range of pressures and temperatures encountered by the fluid. These physical properties are subsequently used by multiphase flow correlations to determine the phases present, the flow regime, and the pressure losses in single and multiphase flow regions. The heat transfer calculations use the fluid thermal properties.

- From the **Setup » Flow Correlation**, ensure that the Hagedorn-Brown correlation is selected for vertical flow and the Beggs-Brill Revised correlation is selected for horizontal flow.

Note: Select the correlation that is best suited for the fluid and operating conditions of interest. There is no universal rule for selecting a multiphase flow correlation that is good for all operating scenarios. Refer to the [Flow Correlation \(p.370\)](#) topic for information on the applicability of flow correlations

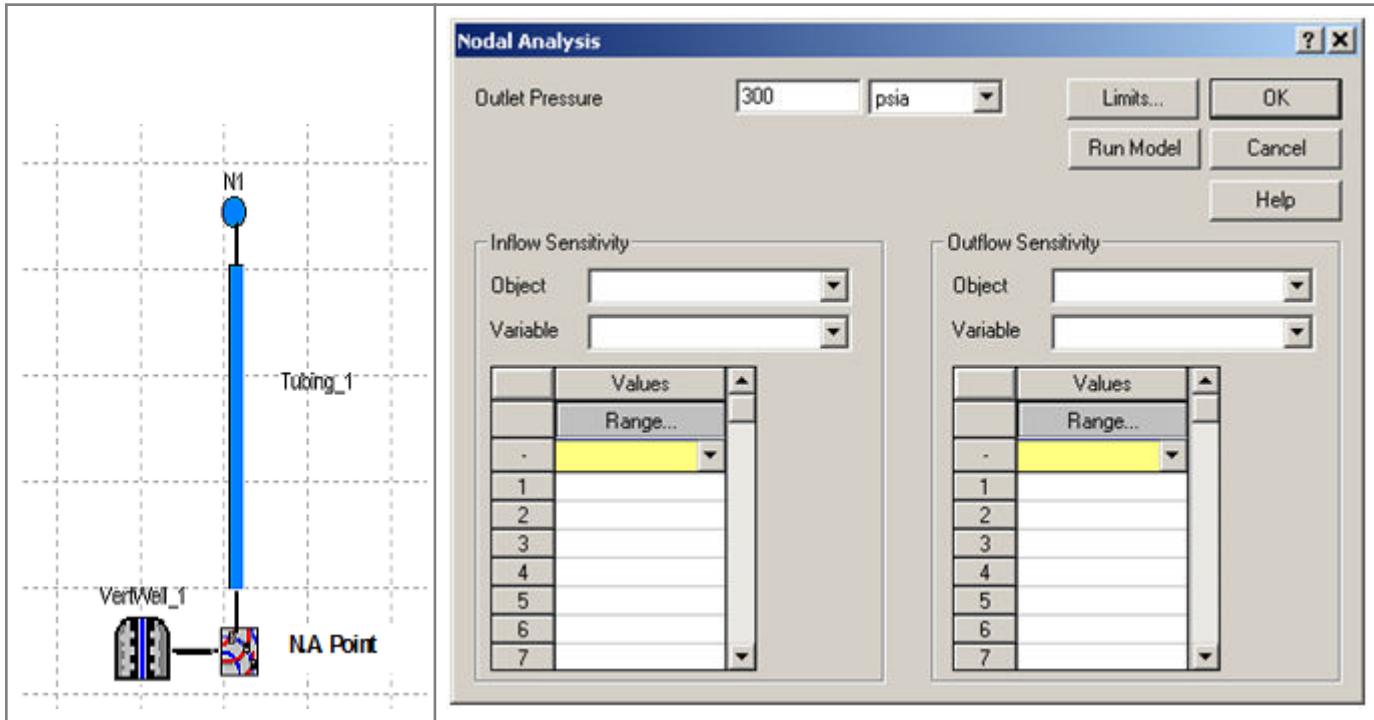
- Save the model as CaseStudy1_Oil_Well.bps.

Task 2: Performing NODAL Analysis

In this section, you perform a NODAL analysis operation for a given outlet (wellhead) pressure to determine the operating point (intersection) and the absolute open flow potential (AOFP) of the well. To do this, add a NODAL analysis point at the bottomhole to divide the system into two parts. Part A extends from reservoir to the bottomhole, while Part B runs from the bottomhole to the wellhead.

To perform a NODAL analysis:

- Select a NODAL analysis point from the toolbar and drop it near the completion.
- Click on the tubing and drag its bottom tip over to the NODAL analysis point.
- Insert a connector to link the completion with the NODAL analysis point.



4. Select **Operations » NODAL Analysis**.
5. Enter an Outlet Pressure (Boundary Condition) of 300 psia.
6. Leave Inflow Sensitivity and Outflow Sensitivity empty.

Note: PIPESIM has implemented several modifications in Nodal Analysis calculation. The most significant is displaying the intersection point on the nodal plot. As a result, you do not depend on reading from the plot and the solution points are calculated with the values presented in Data tab. There is no need to specify/change number of points for inflow and outflow curve unless you wish to use those data for further processing. The PIPESIM engine automatically determines the number of points and their spacing for both inflow and outflow curves.

7. Run the model.
8. Inspect the plot and select the Data tab to get these answers.

(Outlet) Wellhead Pressure	300 psia
Operating Point Flow rate	8,514 stb/d
Operating Point BHP	2,536 psia
AOFP	21,290 stb/d

Table 2.14: Results

Task 3: Performing a Pressure/Temperature Profile

The Pressure/Temperature profile calculates pressure and temperature on a node-by-node basis for the system. The results are plotted for pressure or temperature as a function of distance/elevation along the flow path.

To estimate bottomhole flowing conditions:

1. Run **Operations » Pressure / Temperature Profile**.
2. Enter the Outlet (Tubing head) pressure of 300 psia.
3. Specify the liquid rate as the calculated variable.
4. Leave Sensitivity Data empty.

Note: Inlet and outlet pressure always reference the boundaries of the system. In this particular case, inlet pressure is the reservoir pressure, while the outlet pressure corresponds to wellhead pressure. The inlet pressure is specified at the completion or source level, whereas the outlet pressure is always specified manually within the operation.

5. Run the model.

Note: PIPESIM generates a Profile plot for every valid combination of inflow-outflow cases. Because of this, there is no need to run a separate Pressure Temperature Profile operation.

6. Inspect the plot and summary output report to determine answers.

Wellhead Pressure	300 psia
Production Rate	8,514 stb/d
Flowing BHP	2,536 psia
Flowing WHT	133 degF
Depth at which gas appears	7,600 ft

Table 2.15: Results

2.2.2 Fluid Calibration

Fluid properties (also known as PVT properties) are predicted by correlations developed by fitting experimental fluid data with mathematical models. Various correlations have been developed over the years based on experimental data sets covering a range of fluid properties.

The PIPESIM help system describes the range of fluid properties used to develop each correlation, which helps you select the most appropriate correlation for the fluid at hand. The default correlations in PIPESIM are based on the overall accuracy of the correlations as applied to a broad range of fluids.

To increase the accuracy of fluid property calculations, PIPESIM provides functionality to match PVT fluid properties with laboratory data. Calibration of these properties can greatly increase the accuracy of the correlations over the range of pressures and temperatures for the system being modeled.

For example, calibration of the bubblepoint pressure can result in the initial appearance of gas at a depth of perhaps a thousand feet higher or lower than an uncalibrated model. This results in a significantly different mixture fluid density and, thus, a much different elevational pressure gradient.

Likewise, calibration of the fluid viscosity can drastically improve the calculation of the frictional pressure gradient, especially in heavy oils and emulsions.

If the calibration data is omitted, PIPESIM calibrates on the basis of oil and gas gravity alone, resulting in a loss of accuracy.

After the calibration is performed, a calibration factor calculated as ratio of measured value to the value calculated by selected correlation.

There are two calibration options available in PIPESIM:

- Single Point calibration
- Multi-Point calibration

Single Point Calibration

In many cases, actual measured values for some properties show a slight variance from calculated values. When this occurs, it is useful to calibrate the property using the measured point. PIPESIM can use the known data for the property to calculate a calibration constant K_c :

$$K_c = \text{Measured Property } @ (P, T) / \text{Calculated Property } @ (P, T)$$

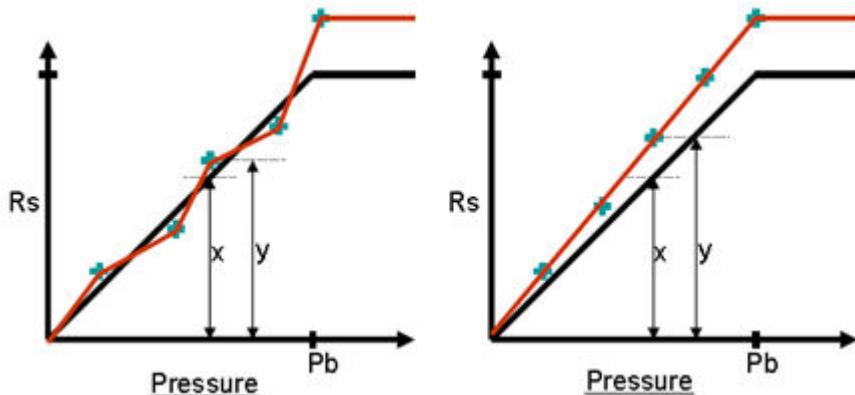
This calibration constant is used to modify all subsequent calculations of the property in question, that is:

$$\text{Calibrated value} = K_c \text{ (Predicted value)}$$

Multi-Point Calibration

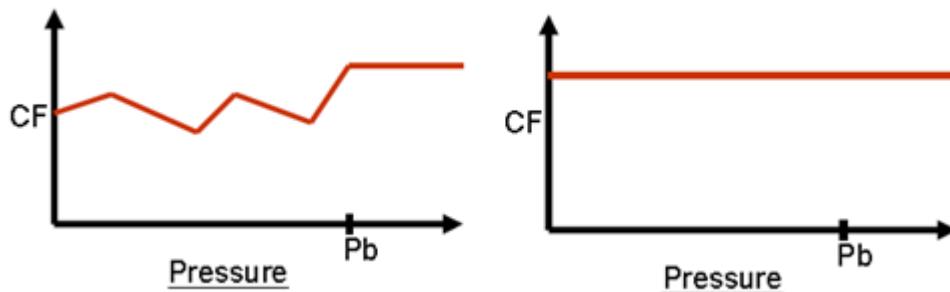
In multi-point calibration, black oil correlations are tuned so that the correlation honors all data points as shown in the next figure.

Figure 2.10. Correlation running through all data points



A calibration factor is calculated for every measurement point, and a plot is generated for the Pressure vs. Calibration factor, as shown in the next figure.

Figure 2.11. Pressure vs. Calibration factor



Note: This is not a best fit method, as all points are fitted exactly. Any outlying data should be smoothed before entering it into PIPESIM.

Task 4: Calibrating PVT Data

To calibrate PVT data:

1. From **Setup** » **Black Oil**, select the Viscosity Data tab.
2. Enter the following calibration data:
3. Under Dead Oil Viscosity, select User's 2 Data points as the correlation.
4. Enter the following measurements:

Property	Temperature (degF)	Value
Viscosity	200	1.5 cp
	60	10 cp

Table 2.16: Dead Oil Viscosity Measurements

5. For Live Oil Viscosity, ensure that the **Chew and Connally** correlation is selected.
6. For the Emulsion Viscosity Method, select the **Brinkman 1952 correlation**.
7. For the Undersaturated Oil Viscosity, select the **Bergman-Sutton correlation**.
8. Select the Advanced Calibration Data tab and click Single-Point Calibration.
9. Enter the measured data to calibrate the PVT model.

Range	Property	Value	Pressure (psia)	Temp (degF)
P > Pb	OFVF	1.18	3,000	200
P = Pb	Sat. Gas	500 scf/stb	2,100	200
P <= Pb	OFVF	1.22	2,100	200
	Live Oil Viscosity	1.1 cp	2,100	200
	Gas viscosity	0.029 cp	2,100	200
	Gas Z factor	0.8	2,100	200

Table 2.17: PVT Calibration Data

10. Select the following PVT correlations:

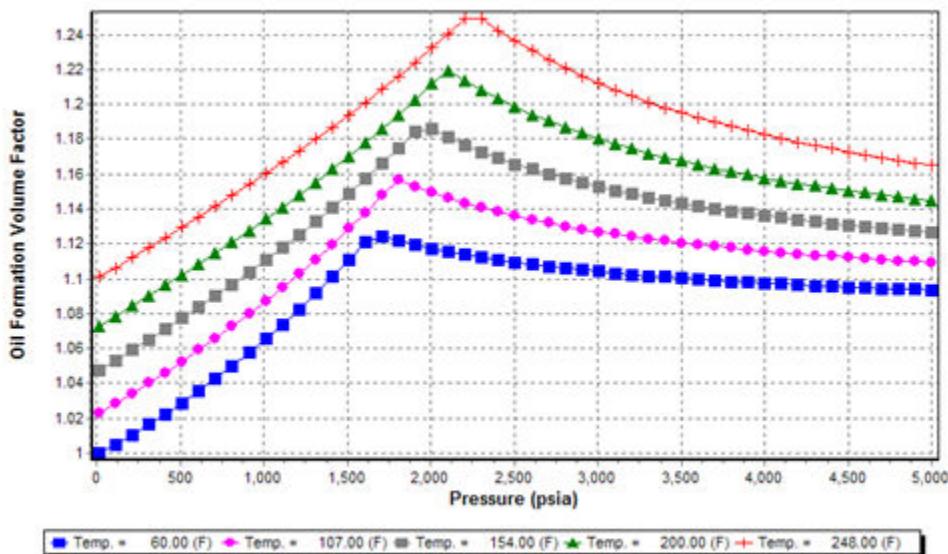
Property	Correlation
Saturated gas	Lasater
OFVF at / below bubblepoint	Standing
Live oil viscosity	Chew and Connally
Gas Z	Standing

Property	Correlation
Saturated gas	Lasater
OFVF at/below bubblepoint	Standing
Live oil viscosity	Chew and Connally
Gas Z	Standing

11. From the **Advanced Calibration Data** tab, select **Plot PVT Data** (Laboratory Conditions GOR = GSAT) to generate a plot of the PVT properties for various pressures and temperatures.

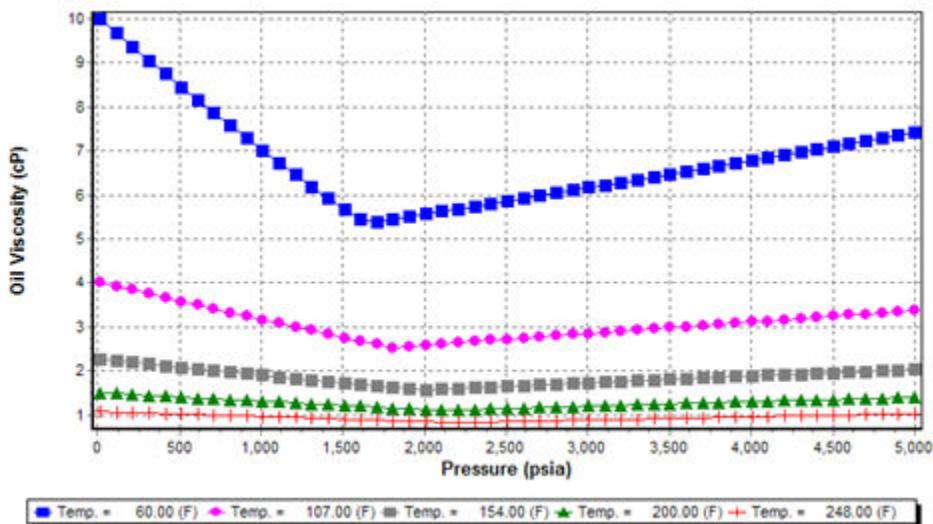
12. Select **Series** and change the y-axis to **Oil Formation Volume Factor**.

13. Verify that the predicted values match the calibration points.



14. Repeat steps 12 and 13 for Oil viscosity and Gas viscosity to ensure the predicted values are correct.

Note: Dead Oil conditions are at 14.7 psia.



Notice that the predicted oil viscosity value at a temperature of 60 degF and 14.7 psia is 10.0 P, consistent with the laboratory dead oil data.

15. Now that the fluid model is calibrated, re-run the **Pressure-Temperature Profile**.
16. Determine the flowing bottomhole pressure, flowing wellhead temperature, and production rate for the given wellhead pressure.
17. Compare your answers to the uncalibrated model results from [2.15 \(p.292\)](#).
18. Inspect the plot and summary output to determine answers.

Wellhead Pressure	Calibrated	Uncalibrated
Production Rate	7,810 stb/d	8,514 stb/d
Flowing BHP	2,624 psia	2,536 psia
Flowing WHT	129 degF	133 degF
Depth where gas appears	7,300 ft	7,600 ft

Table 2.18: Results

Note:

The quantity defined by PIPESIM as 'stock tank' GOR is actually the produced GOR, a dynamic property. The solution gas GOR calibration, an intrinsic property, is specific to the reservoir oil at reservoir conditions and is obtained through laboratory experiments.

The solution gas liberated at standard conditions is called the associated gas. Produced gas can also include a contribution from the gas cap, otherwise known as free gas. In other words:

$$\text{Produced gas} = \text{associated (solution) gas} + \text{free gas}$$

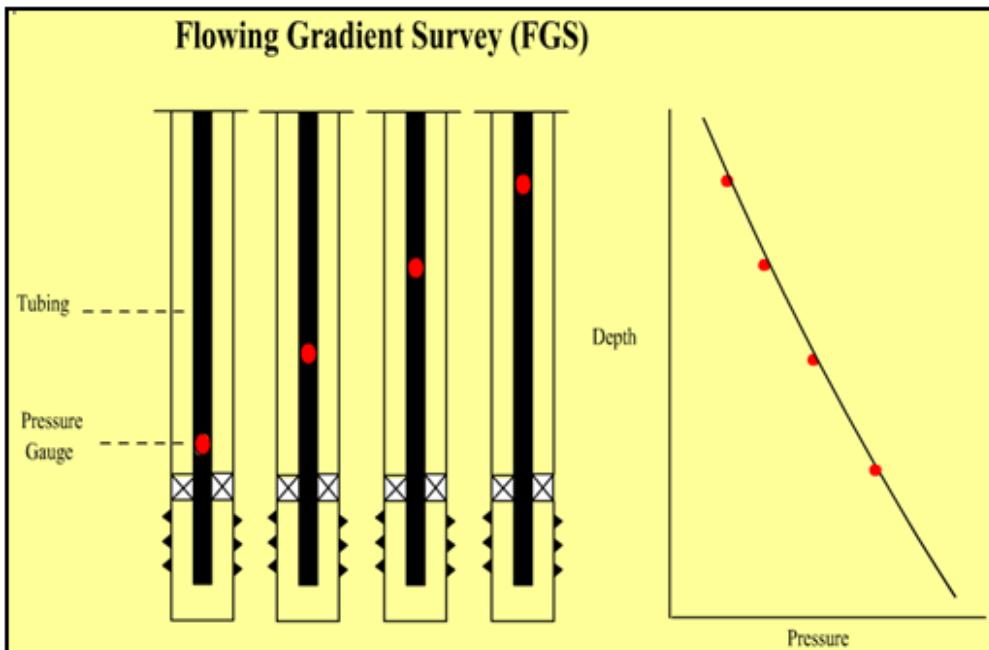
If free gas is produced, the produced GOR will be higher than the solution GOR and, therefore, the calculated bubblepoint based on the specified produced GOR will be higher than that defined by the solution GOR calibration point.

2.2.3 Pressure/Temperature Matching

The pressure distribution of the fluid as it flows through the tubing is very important in production engineering tasks such as selecting tubing sizes, forecasting well productivity, and designing artificial lift installations.

Pressure distribution along particular tubing can be obtained from actual measurements taken with pressure gauges using wireline/slickline at different depths in the well while it is flowing at a constant rate. The result of this measurement is a plot of fluid pressure along tubing versus vertical depth, called a Flowing Gradient survey (FGS) and shown in the next figure.

Figure 2.12. Flowing Gradient Survey



When an FGS is available, it is always best to compare different multiphase flow correlations with the FGS, to determine the one that best matches the FGS.

Additionally, the correlation can be tuned to more accurately match the data. Optimization routines in PIPESIM allow the PIPESIM Single Branch engine to calculate optimal values of parameters to match measured pressure and/or temperature data.

The match is performed by tuning parameters, such as friction and hold-up factor multiplier for pressure matching, and a U-factor multiplier for temperature matching. After the model is tuned, you should validate it against test data measured at different conditions.

Note: WARNING: Avoid using large tuning factors. The recommended tuning range of friction and holdup factor multipliers are +/- 15% (such as 0.85 - 1.15). If it needs > +/- 15% to match the actual measured data, you should review the data again. Large adjustments in friction and holdup factors could also be due to poor fluid characterizations.

Task 5: Evaluating Gas Lift Performance

The basic principle behind gas lift injection in oil wells is to lower the density of the produced fluid in the tubing. This results in a reduction of the elevational component of the pressure gradient above the point of injection and a lower bottomhole pressure. Lowering the bottomhole pressure increases reservoir drawdown and, thus, production rate.

In this section, you examine how this well responds to gas lift by introducing a Gas Lift Injection point at 8,000 feet MD in the tubing equipment.

You have two tasks to accomplish:

- Determine how the well responds to gas lift when the water cut is 10% and 60%.
- Determine the liquid production rates as a function of the gas lift rate and water cut.

Wellhead Pressure (psia)	300
Injection Gas SG	0.6
Injection Gas Surface Temp (degF)	100

Table 2.19: Gas Lift Data

To evaluate gas lift performance:

1. Double-click on Tubing and select the **Downhole Equipment** tab.
2. Under Equipment, select **Gas Lift Injection** and specify a depth of 8000 ft. MD.
3. Click **Properties**.
4. Enter a default gas lift rate of 1 mmscf/d.
5. Go to **Operations » Artificial Lift Performance** and enter the Outlet Pressure as 300 psia.
6. For Sensitivity Data, enter water cut values of 10% and 60%.
7. For the Gas Lift Injection Rate:
 - a. Select **Range**.
 - b. Enter a start value of 1.0.
 - c. Enter an end value of 10.0.
 - d. Enter increments of 0.5.
8. Run the model to generate a plot of calculated liquid rate vs. gas lift rate for different water cuts.
9. Inspect the plot and summary output to determine answers.

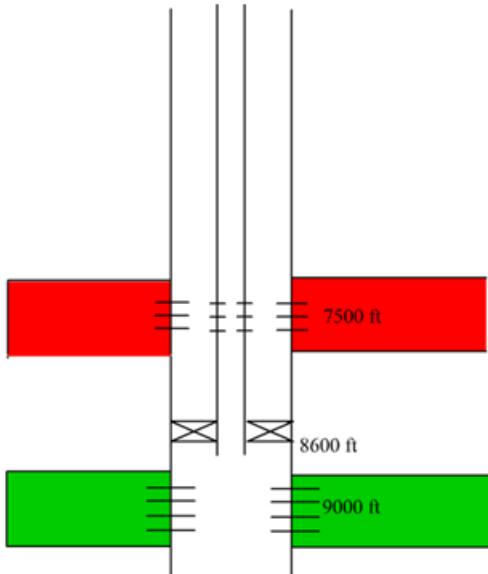
Gas Lift Rate (mmscf/d)	Liq. Prod. Rate (stb/d) @ 10% Wcut	Liq. Prod. Rate (stb/d) @ 60% Wcut
1	8,978	5,810
2	9,606	6,454
4	10,260	7,921
6	10,545	8,381

Gas Lift Rate (mmscf/d)	Liq. Prod. Rate (stb/d) @ 10% Wcut	Liq. Prod. Rate (stb/d) @ 60% Wcut
10	10,674	8,810

Table 2.20: Results

Task 6: Working with Multiple Completions

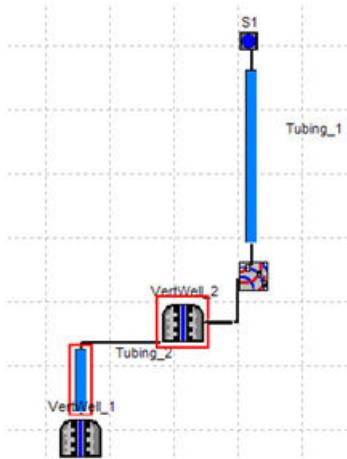
Log analysis shows that a shallow gas zone exists at a TVD of 7,500 feet. As an alternative to gas lift injection, you can investigate the benefits of perforating this zone and self lifting the well.

Figure 2.13. Shallow zone at 7,500 feet

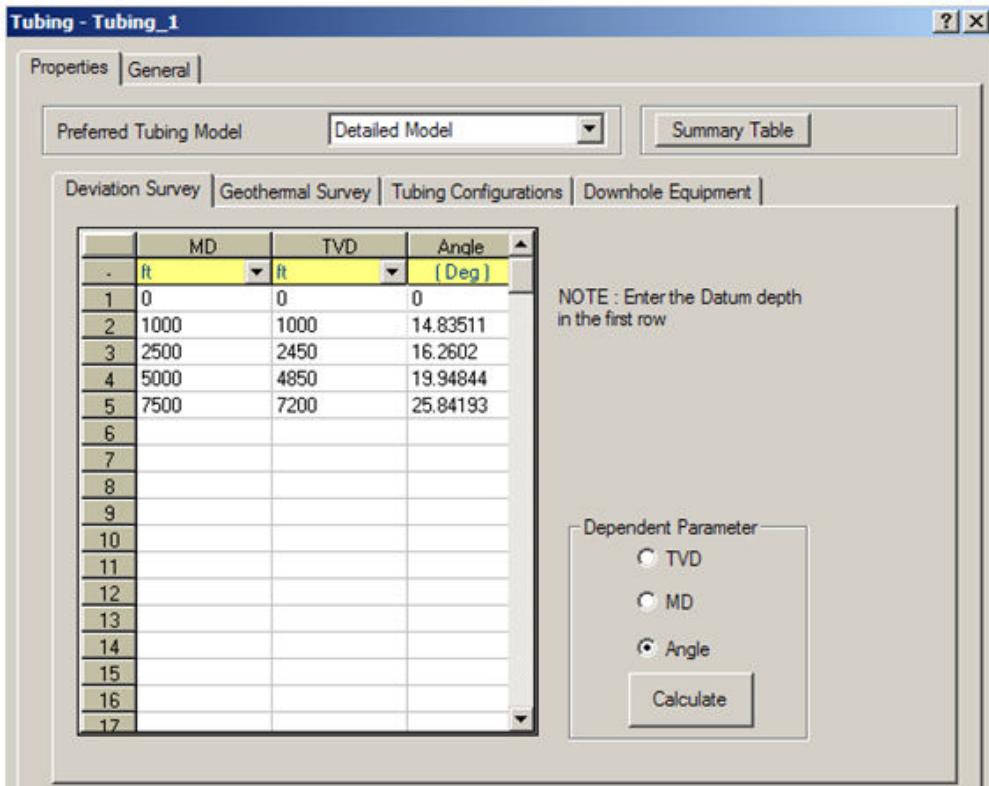
Defining a Second Completion

To define a second completion:

1. Insert a second vertical completion below the NODAL analysis point.
2. Connect to the original completion using a separate tubing model, as shown below.

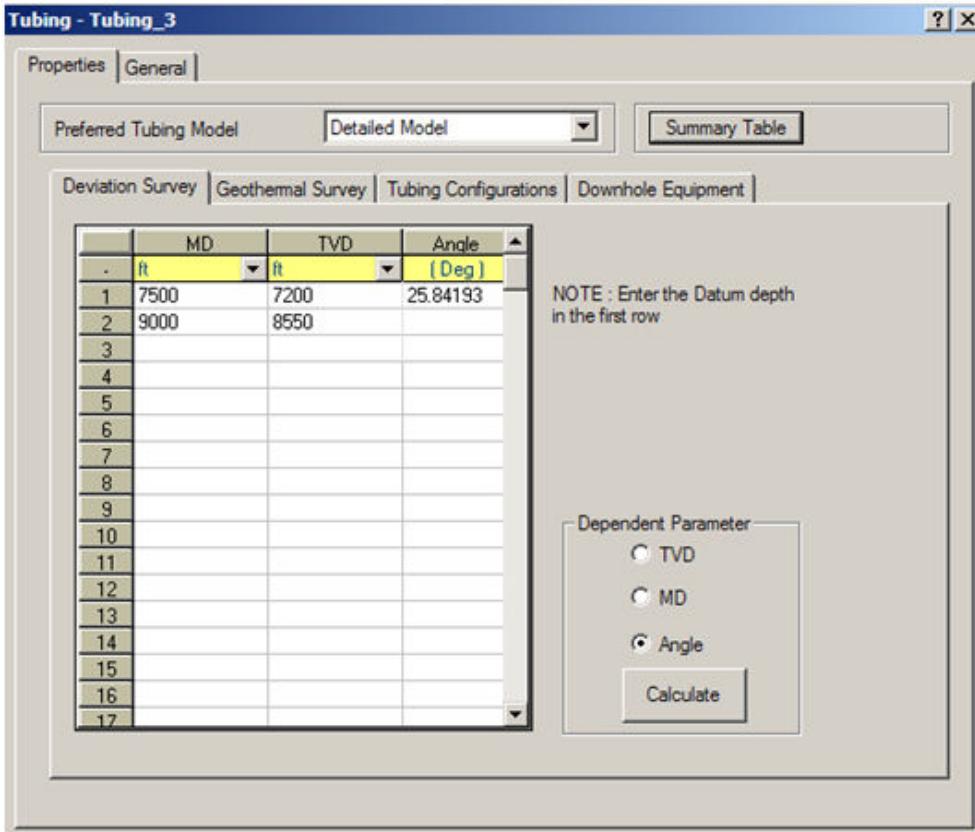


3. Modify the upper tubing string to extend only to the top of the upper perforations.
 - a. Modify the Deviation survey such that it will extend to only 7,200 feet TVD.



- b. Modify the Geothermal survey such that the ambient temperature at an MD of 7,500 feet is 180 degF.
- c. In the **Tubing Configurations** tab, specify a bottom MD of 7,500 feet and a tubing ID of 3.958 inches.
- d. In the **Downhole Equipment** tab, remove the gas lift injection.
- e. Click **OK** to close the menu.

4. Double-click on the lower tubing string to define its properties,
 - a. In the **Deviation Survey** tab, define the lower tubing string profile, as shown.



- b. In the **Geothermal Survey** tab, specify temperatures of 180 degF at 7,500 feet and 200 degF at 9,000 feet.
 - c. Specify the U value as 5 Btu/hr/ft²/F.
 - d. In the **Tubing Configuration** tab, specify a tubing ID of 3.958 inches to a depth of 8,600 feet MD and 6.184 inches to a depth of 9,000 feet.
 - e. Click **OK** to close the menu.
5. With no test data at hand, model the reservoir performance of the upper zone using the pseudo-steady state Darcy equation. Specify the upper completion using the following data:

Model	Pseudo-steady state
Basis of IPR Calculation	Gas
Use Pseudo-pressure?	yes
Reservoir pressure	3,000 psia
Reservoir Temperature	180 degF
Thickness	5 feet
Wellbore Diameter	8.5 inches

Model	Pseudo-steady state
Permeability	20 md
Mechanical Skin	0
Rate Dependant Skin	0

Table 2.21: Reservoir Properties — Upper Gas Zone

6. Select the **Fluid model** tab within the completion dialog and enter the following:
 - a. Use a locally-defined fluid model with an OGR of 0 STB/mmscf and a WGR of 0 (all gas).
 - b. Specify a gas gravity of 0.67.
 - c. Leave all other properties and correlations at their default settings.

Note: The fluid data used for a well/source is defined by a default, local data set or an override value [for water cut and/or GOR/GLR/OGR/LGR]. If there are multiple fluids present in the system with different intrinsic properties, define the main fluid as the default and all others as local fluids.

7. To analyze the effect of perforating the upper zone (compared with gas lift injection), run a Pressure/Temperature Profile for the 60% water cut case.
 - a. From **Setup** » **Black Oil**, set the water cut to 60%.

Note: This water cut affects only the lower zone because the lower zone uses the default fluid model, while the upper zone is defined with a local fluid model.

 - b. Select **Operations** » **Pressure/Temperature Profile**.
 - c. Specify the Outlet Pressure as 300 psia.
 - d. Specify the Liquid Rate as the Calculated Variable.
 - e. Run the model.
 - f. Inspect the output file to determine the results.

Wellhead Pressure	300 psia
Liquid Rate (stb/d)	8,251
Gas Rate (upper zone) (mmscf/d)	6.3844

Table 2.22: Results

2.2.4 Flow Control Valve

A downhole flow control valve (FCV) allows you to model so-called 'intelligent' or 'smart' wells. The methodology implemented provides a simple way of modeling single branch (non-multilateral) intelligent wells in which FCVs are located close to the reservoir.

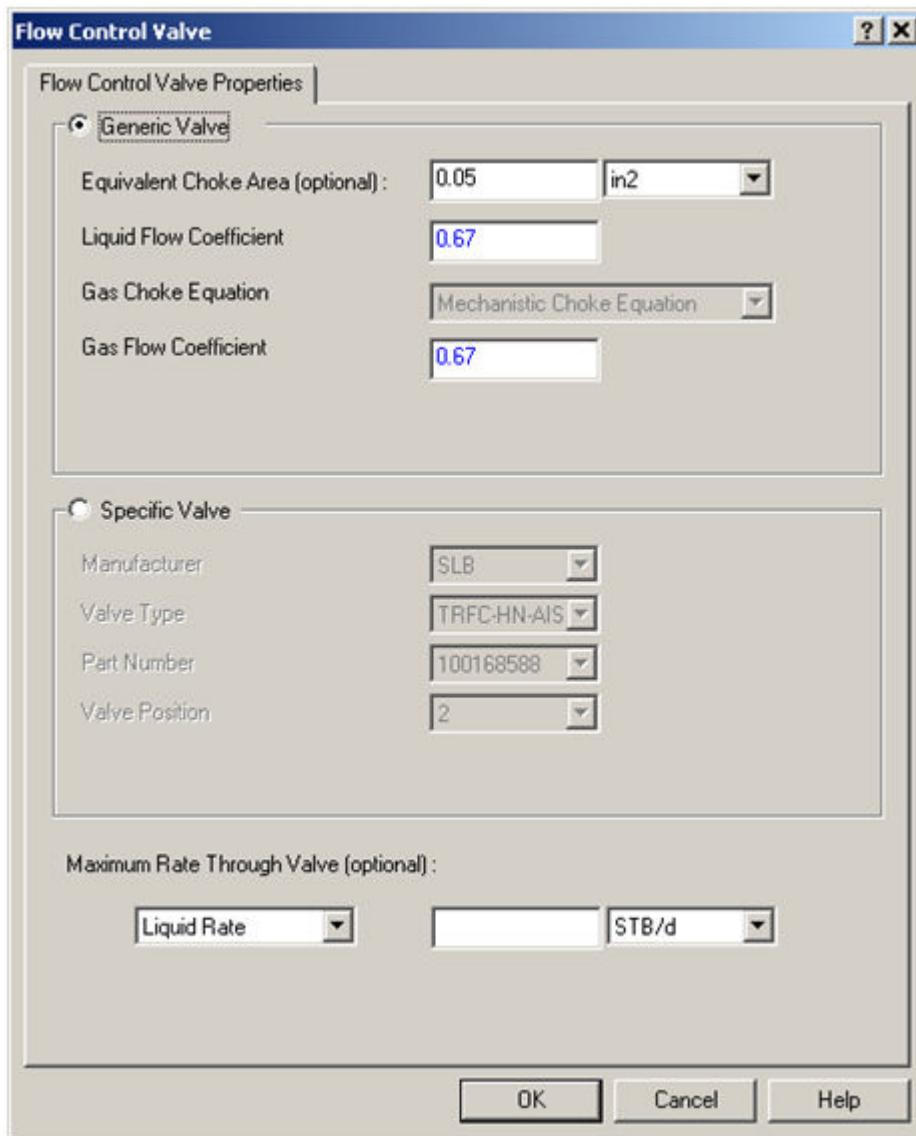
An FCV can restrict the completion flow rate through the system; however, they are available only for vertical completions. The purpose of an FCV is to provide a restriction to fluid flow, thereby reducing the productivity (or injectivity) of a given completion. They are useful in a model containing multiple completions.

An FCV is very similar to a choke. Like a choke, it can be modeled as a fixed-size orifice, in which form it presents a restriction to flow resulting in a pressure drop that increases as flow rate increases.

Unlike a choke however, a maximum flow rate can also be specified. This is applied to the completion and, if necessary, the choke bean diameter is reduced to honor the limit. The choke diameter and flow rate limit can be applied separately or together. If they are both supplied, they are treated as maximum limits.

As shown in next figure, the Flow Control Valve dialog uses radio buttons to present a choice between a Generic valve and a Specific valve.

Figure 2.14. Flow Control Valve properties



A generic valve is specified with its Equivalent Choke Area, Gas and Liquid Flow Coefficients, and choice of Gas Choke Equation method. The choke area can be omitted if a Maximum Rate

Through Valve is specified. If it is present, the FCV is modeled with that choke area but, if the resulting flow rate exceeds the limit, the area is reduced to honor the limit.

You must choose a specific valve from the list of available valves provided in the PIPESIM database. Many of the specific valves are multi-position devices, as they allow you to select the effective choke area from a range of pre-installed fixed chokes.

If a flow rate limit is supplied, the simulation selects the choke position required to honor the limit. Because the choke area cannot be calculated to match the limit exactly, this usually results in the flow rate being lower than the limit.

The valve position can be specified or omitted. If specified, the FCV is modeled with the corresponding choke area, but if the resulting flow rate exceeds the limit, a lower position number is used.

Valve positions are numbered in order of increasing choke size, starting with position zero. This position usually specifies a diameter of zero to allow the valve to be shut. An FCV can have as many as 30 positions.

Task 7: Modeling a Flow Control Valve

A formation integrity test indicates you should not flow more than 2 mmSCFD of gas from the upper formation. To make sure, install the FCV in the upper completion.

To model a flow control valve:

1. Double-click on the upper completion and check Flow Control Valve.
2. In the **FCV Properties** window, set the Maximum Rate through Valve to 2 mmSCFD.
3. Leave Equivalent Choke Area empty.
4. Select **Operations » Pressure Temperature Profile**. Ensure that the Liquid Rate is the calculated variable and the outlet pressure is set to 300 psia.
5. Run the model and view the output file for Bean Size. Required Bean Size: 0.0417 in²
6. (Optional) Select any Specific Valve to sensitize on FCV and generate a plot liquid flow rate vs. FCV position.

Note: TIP: Select **SLB : TRFC-HN-AIS** value and use System Analysis and mass flow rate.

2.3

Gas Well Performance Analysis Tutorial

A gas well has been drilled for which Drill Stem Test (DST) and compositional fluid data are available. In this section, you will model the performance of this well.

In this tutorial, you will perform the following tasks:

- Create a Compositional Fluid Model for a Gas Well (p.309)
- Calculate Gas Well Deliverability (p.312)
- Calibrate the Inflow Model Using Multipoint Test Data (p.315)
- Select a Tubing Size (p.317)

- Model a Flowline and Choke (p.320)
- Predict Future Production Rates (p.322)
- Determine a Critical Gas Rate to Prevent Well Loading (p.324)

2.3.1 Compositional Fluid Modeling

PIPESIM offers fully compositional fluid modeling as an alternative to the Black Oil model.

Compositional fluid modeling is generally regarded as more accurate, especially for wet gas, condensate and volatile oil systems. However, detailed compositional data is less frequently available to the production engineer.

PIPESIM currently has access to two compositional PVT Frameworks that provide several PVT flash packages.

Original PIPESIM PVT Framework:

- Multiflash, a third-party compositional package (InfoChem).

New PVT Toolbox Framework (available after PIPESIM 2010.1):

- Eclipse 300 Flash, a new interface to ECLIPSE two-phase flash, allowing additional Equation of States. This is the same Equation of State package used by other GeoQuest products, such as ECLIPSE Compositional, PVTi, VFPI, and others.
- DBR Flash, two-phase flash developed by the Schlumberger DBR Technology Center. It has a more extensive component library than ECLIPSE Flash.
- NIST Refprop Flash, two-phase flash using HelmHoltz Equation of State.
- GERG 2008, two-phase flash using HelmHoltz Equation of State.

Equations of State (EoS)

Equations of State describe the pressure, volume and temperature (PVT) behavior of pure components and mixtures. Most thermodynamic and transport properties are derived from the Equation of State. They are a function of pressure and temperature.

One of the simplest Equations of State for this purpose is the ideal gas law, $PV=nRT$, which is roughly accurate for gases at low pressures and high temperatures.

Note: The Black Oil model uses this equation along with a compressibility factor (z) to account for non-ideal behavior.

However, this equation becomes increasingly inaccurate at higher pressures and temperatures, and it fails to predict condensation from a gas to a liquid. As a result, much more accurate Equations of State have been developed for gases and liquids.

The Equations of State available in PIPESIM include:

Multiflash	<ul style="list-style-type: none">• Standard Peng-Robinson• Advanced Peng-Robinson• Standard Soave-Redlich-Kwong (SRK)
-------------------	--

	<ul style="list-style-type: none"> Advanced Soave-Redlich-Kwong (SRK) Benedict-Webb-Rubin-Starling (BWRS) Association (CPA).
DBR Flash	<ul style="list-style-type: none"> Peng-Robinson (with/without Volume Shift) Soave-Redlich-Kwong (with/without Volume Shift Correction).
ECLIPSE 300 Flash	<ul style="list-style-type: none"> Peng-Robinson (with/without Volume Shift + Accentric Factor Correction) Soave-Redlich-Kwong (with/without Volume Shift Correction).
NIST Refprop Flash	<ul style="list-style-type: none"> HelmHoltz Equation of State
GERG 2008	<ul style="list-style-type: none"> HelmHoltz Equation of State

Viscosity

Compositional fluid models also use Viscosity models based on corresponding state theory.

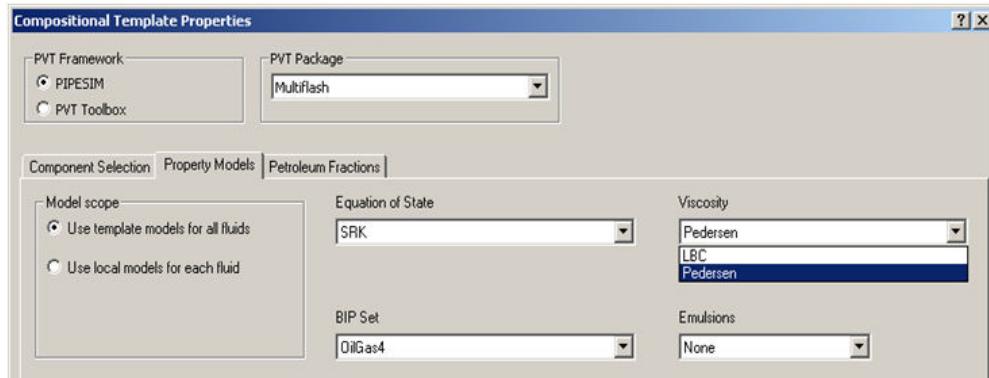
Available Viscosity models include:

- Pederson (default)
- Lohrenz-Bray-Clark (LBC)
- Aasberg-Petersen

Comparative testing has shown the Pedersen method to be the most widely applicable and accurate for oil and gas viscosity predictions. Multiflash uses the Pedersen method as the default viscosity model, though an option is available to choose the LBC model for backward compatibility.

The choice you make of the Equation of State has a large effect on the viscosities predicted by these methods. The LBC method is more sensitive to the Equation of State effects than the Pedersen method.

Figure 2.15. Selecting the default viscosity option

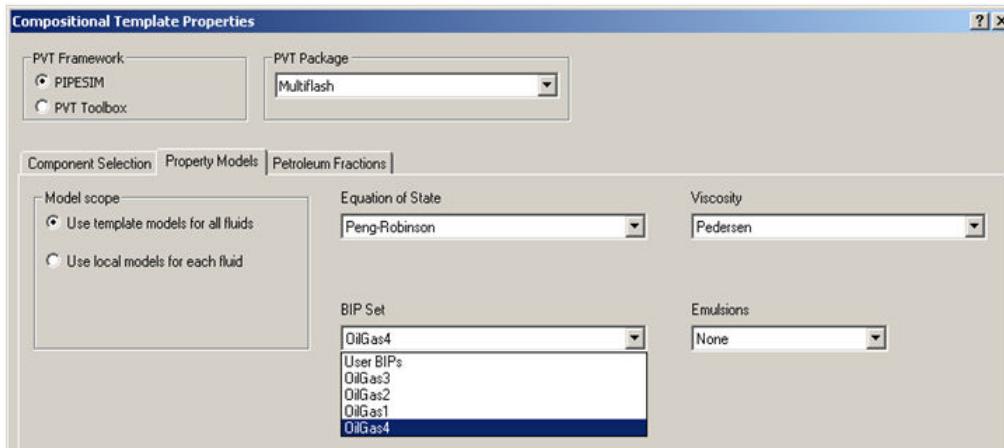


Binary Interaction Parameter (BIP) Set

Binary interaction parameters (BIPs) are adjustable factors used to alter the predictions from a model until the predictions match experimental data as closely as possible.

BIPs are usually generated by fitting experimental VLE or LLE data to the model in question. BIPs apply between pairs of components, although the fitting procedure can be based on both binary and multi-component phase equilibrium information.

Figure 2.16. Selecting a BIP in the Compositional Properties window



Emulsion Viscosities

An emulsion is a mixture of two immiscible liquid phases. One phase (the dispersed phase) is carried as droplets in the other (the continuous phase). In oil/water systems at low water cuts, oil is usually the continuous phase.

As water cut is increased, there comes a point at which phase inversion occurs, and water becomes the continuous phase. This is the Critical Water cut of Phase Inversion, otherwise called the cutoff, which occurs typically between 55% and 70% water cut. The viscosity of the mixture is usually highest at, and just below, the cutoff.

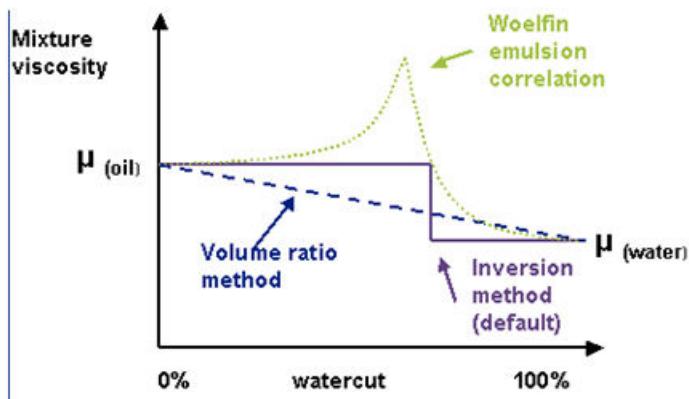
Emulsion viscosities can be many times higher than the viscosity of either phase alone.

Three mixing rules have been implemented that are identical to the options currently available in the Black Oil section.

You can choose any of these options:

- Set to oil viscosity
- Volume ratio of oil and water viscosities
- Woelflin, which uses Woelflin correlation at water cut less than, or equal to, CUTOFF, and water viscosity at water cut greater than CUTOFF.

Figure 2.17. Mixing Options



Flashing Options

Flash calculations are an integral part of all reservoir and process engineering calculations. They are required whenever it is desirable to know the amounts (in moles) of hydrocarbon liquid and gas coexisting in a reservoir or a vessel at a given pressure and temperature.

These calculations are also performed to determine the composition of the existing hydrocarbon phases.

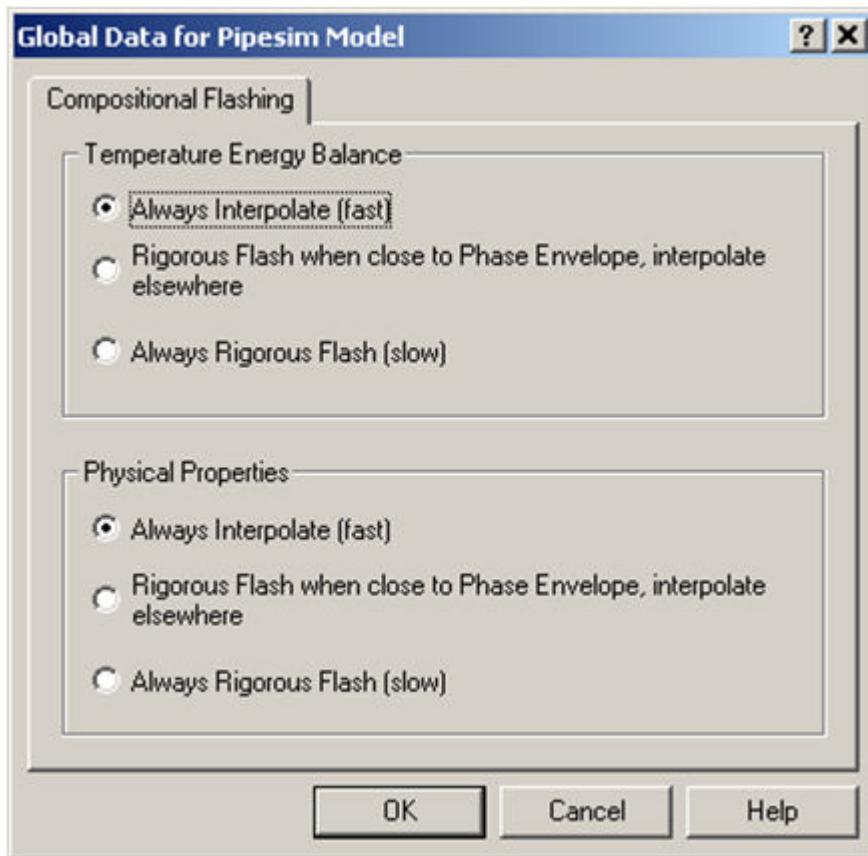
Given the overall composition of a hydrocarbon system at a specified pressure and temperature, flash calculations can determine four factors:

- Moles of the gas phase
- Moles of the liquid phase
- Composition of the liquid phase
- Composition of the gas phase

The compositional module uses inline flashing (PVT tables built in memory) as the default mode of compositional simulation. For inline flashing, PIPESIM has three options – Interpolation, Interpolation when close to phase boundary, and Rigorous.

Interpolation	<p>To maximize the speed of the simulation, not all requested P/T points are flashed. A pressure/temperature grid is defined and only these points are created.</p> <p>For points not lying exactly on a grid point, four-point interpolation is used. The default grid points can be changed via the compositional option.</p> <p>This is the fastest, but least accurate, method.</p>
Interpolation when close to a Phase Boundary	<p>In a case where one or more of the four points used for the interpolation is in a different phase, a full flash is performed and the data point added to the table.</p> <p>This will improve accuracy, but at the cost of speed.</p>
Rigorous	A full flash is always performed. Very accurate, but slow!

Figure 2.18. Flashing options



Task 1: Creating a Compositional Fluid Model for a Gas Well

To create a compositional fluid model:

1. Start with a new PIPESIM case – Well Performance Analysis.
2. Open the Compositional Fluid Template menu by selecting **Setup > Compositional Template**.
3. Choose PVT Framework as PIPESIM and select Multiflash as PVT Package.

Note: Schlumberger employees select PVT Toolbox Framework, E300 Flash Package. Your results will be slightly different.

4. Click the **Component Selection** tab.
5. Add following library components by selecting the desired components from the list and click **Add >>**.

Methane	Butane
Ethane	Isopentane
Propane	Pentane

Isobutane	Hexane
-----------	--------

Table 2.23: Components

6. Add the C7+ pseudo-component:
 - a. Select the **Petroleum Fractions** tab.
 - b. Enter the pseudo-component name and data.
 - c. Highlight the row number for the pseudo-component and click **Add to Composition**.

C7+ BP	214 degF
C7+ MW	115
C7+ SG	0.683

Table 2.24: Pseudo-Component Stock Tank Properties

7. Leave Property Models as default.
8. Open the **Compositional (Local Default)** menu by selecting **Setup ➤ Compositional (local default)**.
9. Under the **Component Selection** tab, you will notice all the components pre-defined in step-4 above. Add the mole fraction to these components.

Methane	78
Ethane	8
Propane	3.5
Isobutane	1.2
Butane	1.5
Isopentane	0.8
Pentane	0.5
Hexane	0.5
C7+	6

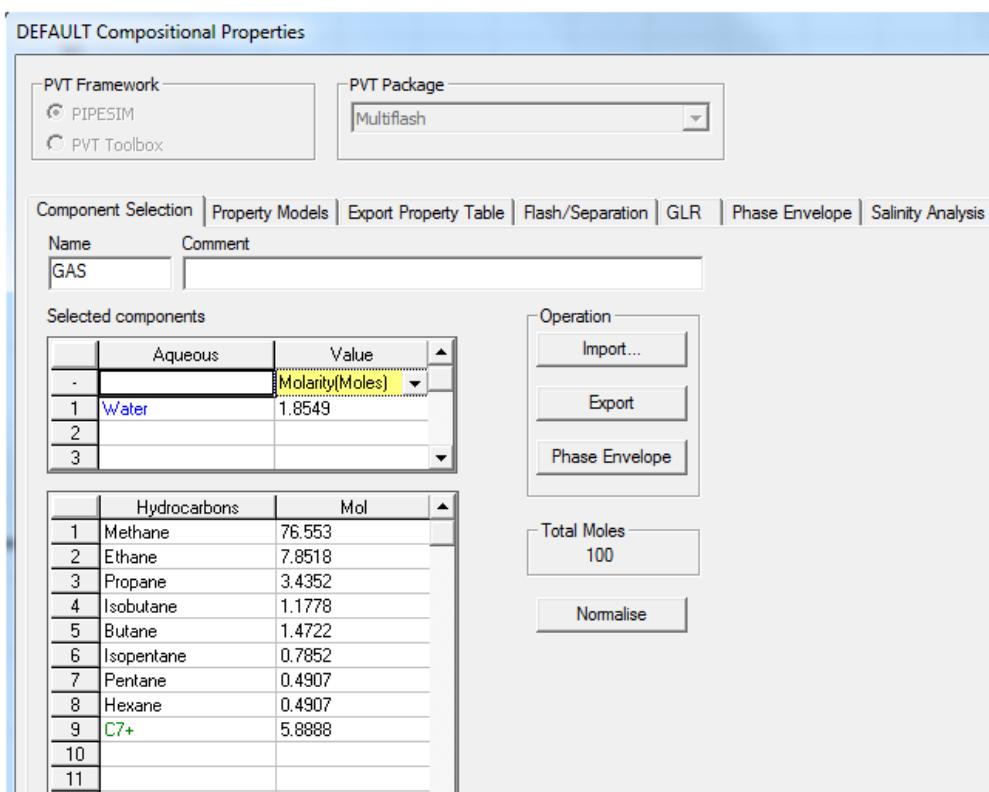
Table 2.25: Componistion (%)

10. To determine the water content at saturation at reservoir conditions:
 - a. Go back to the **Compositional Template** UI and add Water as additional component.
 - b. Now come back to **Compositional (Local default)** UI and add an arbitrary amount of water, such as 20 moles, to the composition.
 - c. Select the **Flash/Separation** tab.
 - d. Click the **PT** button and enter the reservoir pressure and temperature, 4,600 psia and 280 degF, respectively.
 - e. Perform a flash and read the water content for the vapor fraction from the screen.

Note: The hydrocarbon vapor components will be normalized to include the mole fraction of water.

- f. Copy and paste (**Ctrl + C** and **Ctrl + V**) the water and the normalized hydrocarbon composition back into the compositional editor main screen.

Note: Water can be carried along with the gas in the vapor phase or entrained in the gas in droplet form. There exists at any temperature and pressure a maximum amount of water vapor that a gas is able to hold. A gas is completely saturated when it contains the maximum amount of water vapor for the given pressure and temperature conditions. Keeping the volume and pressure constant on water vapor-saturated gas, water will condense out at lower temperatures because the capacity of the gas to hold water is less. The same is true if the volume and temperature are kept constant, but the pressure is allowed to increase.



11. Click **Phase Envelope** to generate a phase envelope using the water-saturated composition.
12. From the main **Component Selection** tab, click **Export**, name the composition sat_gas and click **Save**.
13. Select **Setup** » **Flow Correlations** and choose **Gray Modified** for the vertical flow correlation.
14. Select **File** » **Save As** and save the model as **GasWell.bps**.

2.3.2 Gas Well Deliverability

Based on the analysis for flow data obtained from a large number of gas wells, Rawlins and Schellhardt (1936) presented a relationship between the gas flow rate and pressure drawdown that can be expressed as:

$$Q_{SC} = C(p_R^2 - p_{wf}^2)^n$$

Where

Q_{SC} = gas rate (mmscf/d)

p_R = average reservoir pressure (PSIA)

p_{wf} = flowing bottomhole pressure

C = flow coefficient (mmscf/day/psi²)

n = non-Darcy exponent

The exponent n is intended to account for the additional pressure drop caused by the high-velocity gas flow, such as turbulence. Depending on the flowing conditions, the exponent n can vary from 1.0 for completely laminar flow to 0.5 for fully turbulent flow.

The performance coefficient C in above equation is included to account for:

- Reservoir rock properties
- Fluid properties
- Reservoir flow geometry

This equation is commonly called the deliverability or back-pressure equation. If you can determine the coefficients of the equation - n and C - you can calculate the gas flow rate Q_{SC} at any bottomhole flow pressure p_{wf} and construct the IPR curve.

There are essentially three types of deliverability tests:

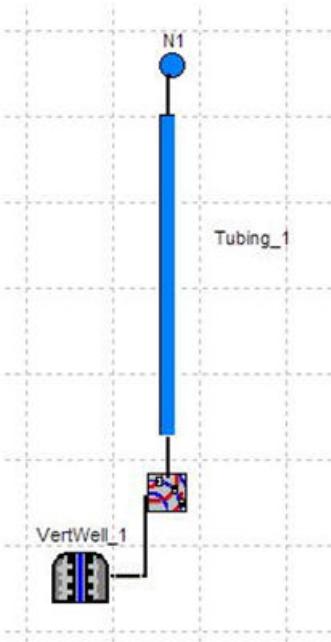
- Conventional deliverability (back-pressure) test
- Isochronal test
- Modified isochronal test

Essentially, these tests consist of flowing wells at multiple rates and measuring the bottomhole flowing pressure as a function of time. When the recorded data are properly analyzed, it is possible to determine the flow potential and establish the inflow performance relationships of the gas well.

2.3.3 Task 2: Calculating Gas Well Deliverability

In this section, you construct the simple physical well model shown below and perform a simulation to calculate deliverability.

1. Using the **Single Branch** toolbar, insert a vertical completion, tubing, and NODAL analysis point, as shown in the figure.



2. Edit the reservoir and tubing data according to the data in the table.

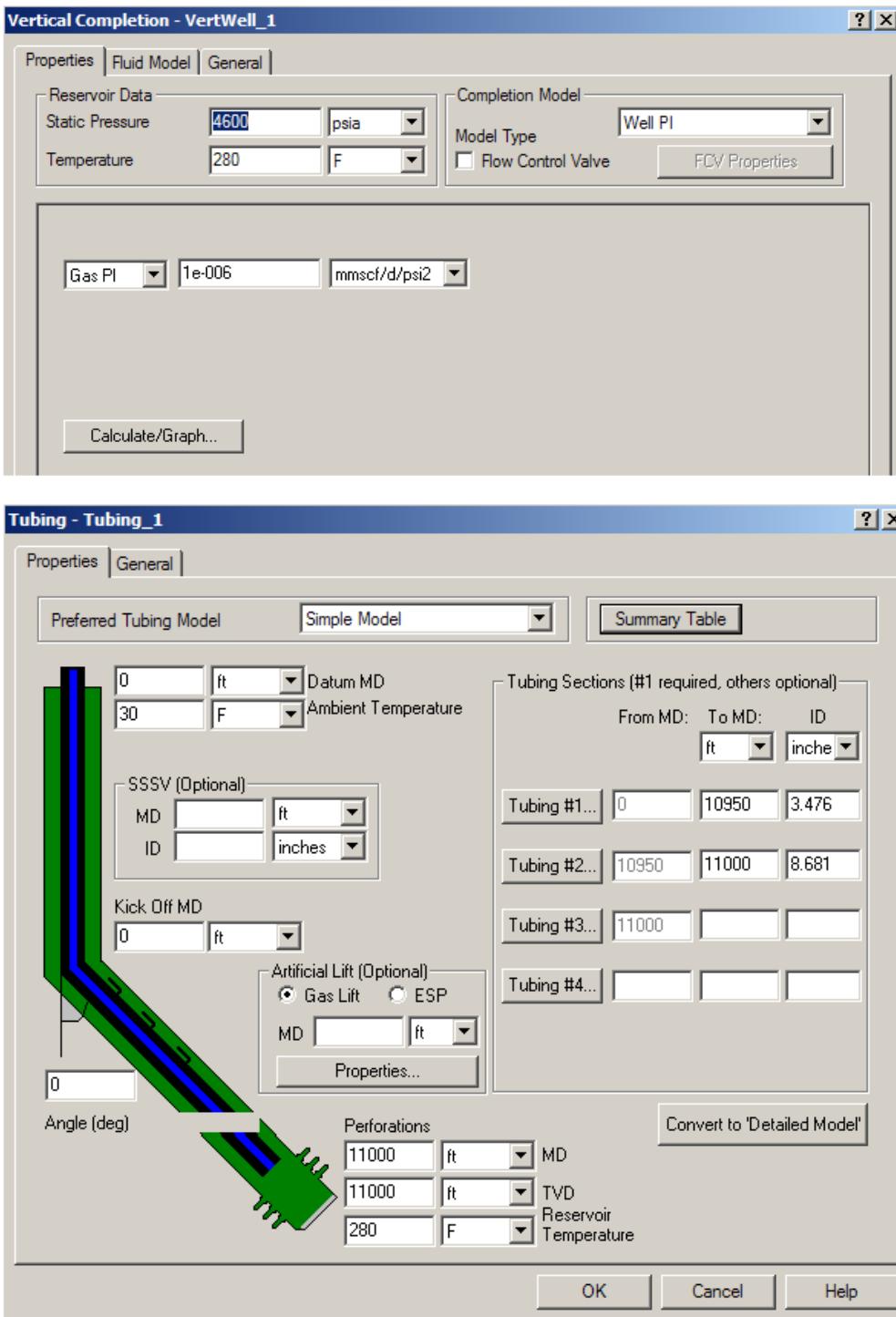
Static Pres	4,600 psia
Reservoir Temp.	280 degF
Gas PI	1×10^{-6} mmscf/d/psi ²

Table 2.26: Reservoir Data

Mid perf TVD	11,000 feet
Mid perf MD	11,000 feet
Ambient temp	30 degF
EOT MD	10,950 feet
Tubing ID	3.476 inches
Casing ID	8.681 inches

Table 2.27: Tubing Data

The vertical completion properties for Well_1 are shown in the figure below, followed by an example of tubing properties for a simple model.



3. Select Operations » Pressure/Temperature Profile Operation.
 - a. Select the **Gas Rate** as the calculated variable.
 - b. Specify an **Outlet Pressure** of 800 psia and click **Run**. The flow rate displays below the plot. You can read the bottomhole flowing pressure on the plot.
4. On the **Plot** menu, select **Series**.

5. Change the Y-axis to **Temperature**. You can read the bottomhole and wellhead temperatures on the plot.

Pres = 4,600 psia, Tres = 280 degF	
% H ₂ O @ saturation	1.8549
P_o = 800 psia	
Q _G	18.21 mmscf/d
P _{wf}	1,718 psia
BHT	236 degF
WHT	172 degF

Table 2.28: Results

Task 3: Calibrating the Inflow Model Using Multipoint Test Data

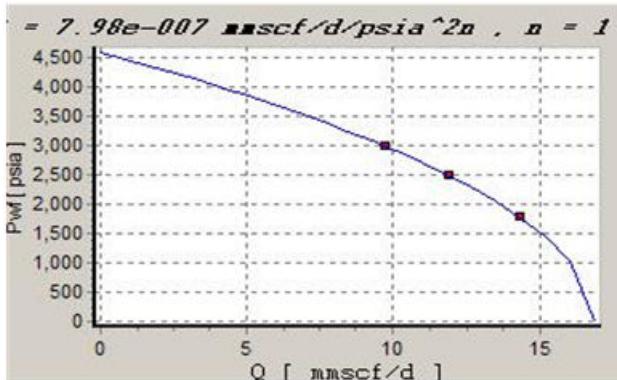
In this section, you use the back-pressure equation for inflow performance relationship for a gas well producing at a pseudo-steady state. Using a multipoint well test, the C and n parameters are calculated.

1. Double-click **Completion**.
2. Choose the **Back Pressure Equation** from the drop-down menu.
3. Click **Calculate/Graph** and enter the test data listed in the table.

QGas (mmscf/d)	Pwf (psia)
9.7	3,000
11.9	2,500
14.3	1,800

Table 2.29: Multiple Test Data

4. Click **Plot IPR**.



Note: To position data points, right-click and drag on a plot. To zoom in, click and drag a window across the data points towards the lower right. To zoom out, click and drag a window towards the upper-left.

5. Rerun the **Pressure/Temperature Profile** operation to determine the following:
 - Gas flow rate
 - Bottomhole flowing pressure
 - Bottomhole flowing temperature
 - Wellhead temperature
6. Inspect the profile plot and summary file to determine the results.

Back Pressure Equation	
Parameter C	7.9793682e-007
Parameter n	1
$P_o = 800 \text{ psia}$	
Q_G	14.97 mmscf/d
P_{wf}	1,548 psia
T_{bh} (degF)	232 degF
T_{wh} (degF)	168 degF

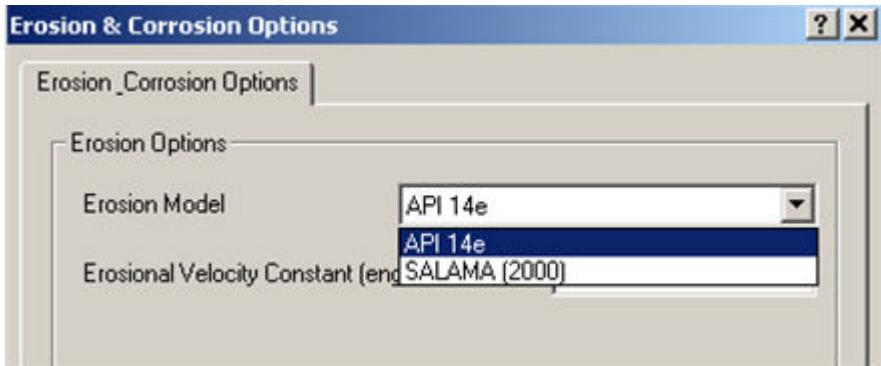
Table 2.30: Results

2.3.4 Erosion Prediction

Erosion has been long recognized as a potential source of problems in oil and gas production systems. Erosion can occur in solids-free fluids but, usually, it is caused by entrained solids (sand).

Two erosion models are available in PIPESIM – API 14 E and Salama.

Figure 2.19. Selecting erosion options



API 14 E

The API 14 E model comes from the American Petroleum Institute, Recommended Practice, number 14 E. This is a solids-free model which calculates only an erosion velocity (no erosion rate). The erosion velocity V_e is calculated with the formula:

$$V_e = \frac{C}{\sqrt{p_m}}$$

Where p_m is the fluid mean density and C is an empirical constant. C has dimensions of (mass/(length*time²))^{0.5}. Its default value in engineering units is 100, which corresponds to 122 in SI units.

The current practice for eliminating erosional problems in piping systems is to limit the flow velocity to that calculated by this correlation.

Salama

The Salama model was published in Journal of Energy Resources Technology, Vol 122, June 2000, "An Alternative to API 14 E Erosional Velocity Limits for Sand Laden Fluids," by Mamdouh M. Salama.

This model calculates erosion rate and erosional velocity. The parameters required for the model are Acceptable Erosion rate, Sand production ratio, Sand Grain Size, Geometry Constant and Efficiency.

The equations in Salama's paper use a sand rate in Kg/day. This is obtained from the supplied volume ratio using Salama's 'typical value' for sand density - 2650 kg/m³.

Task 4: Selecting a Tubing Size

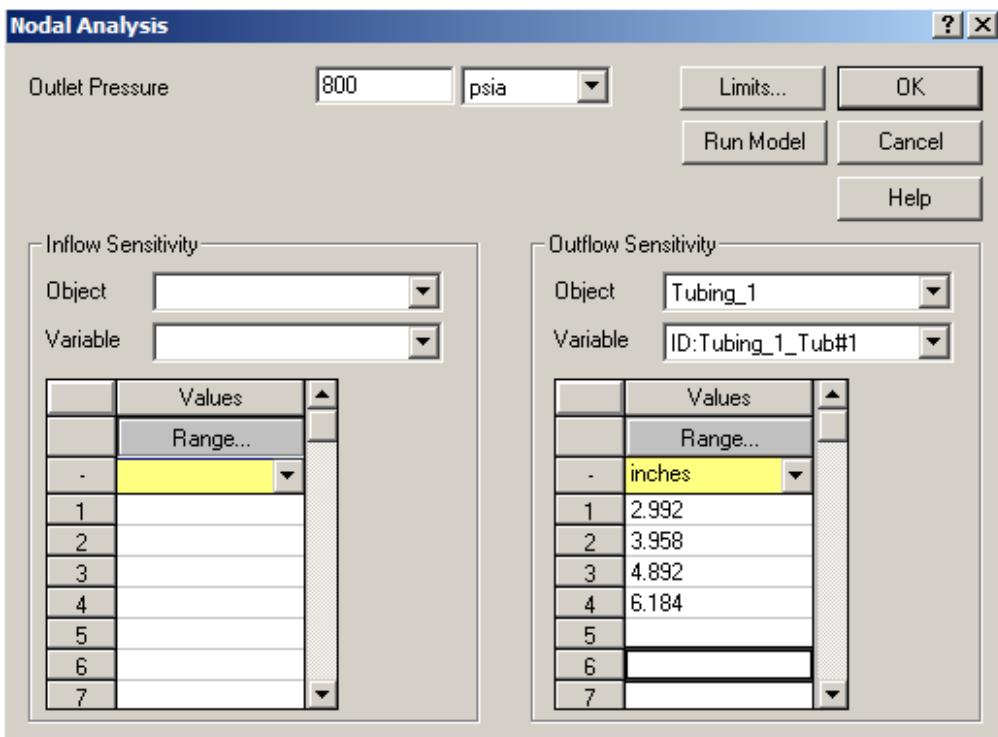
In this section, you perform a NODAL analysis to select an optimum tubing size. The available tubing sizes have IDs of 2.992 inches, 3.958 inches, 4.892 inches, and 6.184 inches.

Your final decision will be based on these criteria:

- Flow rate (High)
- Erosional velocity ratio (<1).
- Cost (Generally increases with size)

To select a tubing size:

1. Ensure that the model includes a NODAL analysis object located between the tubing and the completion.
2. Select **Operations > NODAL analysis**.
 - a. Enter 800 psia as the Outlet Pressure.
 - b. Enter the tubing IDs as the Outflow Sensitivity.
 - c. Run the model and observe the outflow curves.



3. Another way to analyze the effect of the tubing ID, is to perform a Pressure/Temperature profile.
 - a. Select **Operations > Pressure/Temperature Profile**.
 - b. Enter the tubing size as the sensitivity.
 - c. Specify the flow rate as the calculated variable and run the model.
 - d. From the profile plot, change the X-axis to Erosional Velocity Ratio (EVR = actual velocity / API 14e limit) by selecting the Series option from the toolbar. This lets you determine the maximum erosional velocity ratio.

Selected Tubing	3.958 inch
P_o = 800 psia	
Q _G	15.39 mmscf/d
P _{wf}	1,369 psia
BHT	228 degF

WHT	167 degF
Max Erosional velocity ratio	0.7693

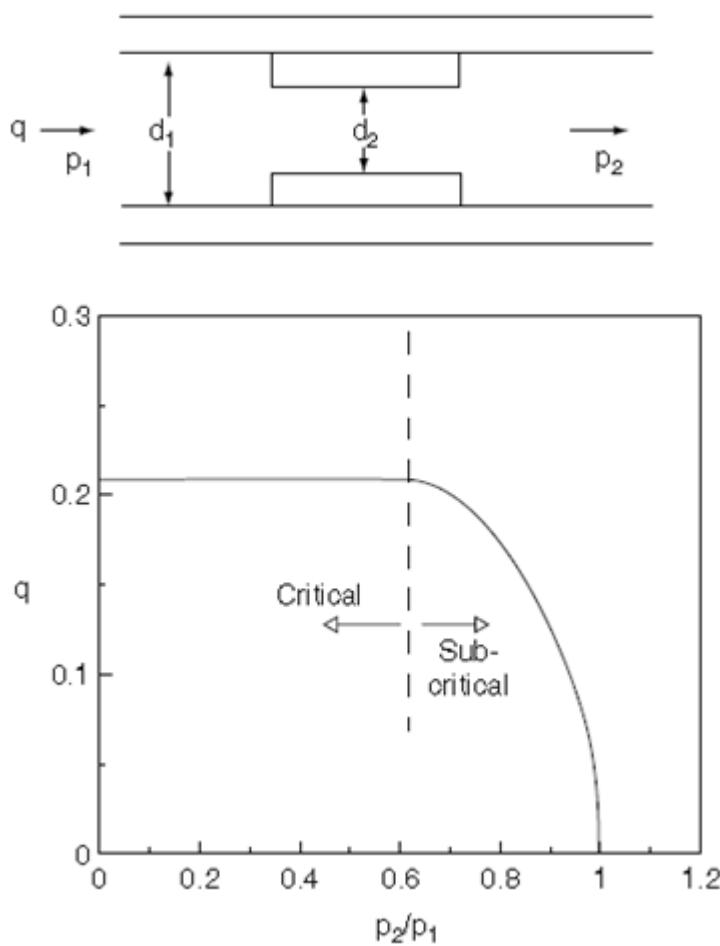
Table 2.31: Results

2.3.5 Choke Modeling

Wellhead chokes are used to limit production rates to meet surface constraints, protect surface equipment from slugging, avoid sand problems due to high drawdown, and control flow rate to avoid water or gas coning. Placing a choke at the wellhead increases the wellhead pressure and, thus, the flowing bottomhole pressure which reduces production rate.

Pressure drop across wellhead chokes is usually very significant, and various choke flow models are available for critical (sonic) and sub-critical flow as shown in the next figure.

Figure 2.20. Gas fraction in the fluid and flow regimes



Sound waves and pressure waves are both mechanical waves. When the fluid flow velocity in a choke reaches the traveling velocity of sound in the fluid under the in situ condition, the flow is called sonic flow. Under sonic flow conditions, the pressure wave downstream of the choke cannot go upstream through the choke because the medium (fluid) is traveling in the opposite direction at

the same velocity. As a result, a pressure discontinuity exists at the choke, which means that the downstream pressure does not affect the upstream pressure.

Because of the pressure discontinuity at the choke, any change in the downstream pressure cannot be detected from the upstream pressure gauge. Any change in the upstream pressure cannot be detected from the downstream pressure gauge either. This sonic flow provides a unique choke feature that stabilizes the well production rate and separation operation conditions.

Whether a sonic flow exists at a choke depends on a downstream-to-upstream pressure ratio. If this pressure ratio is less than a critical pressure ratio, sonic (critical) flow exists. If this pressure ratio is greater than, or equal to, the critical pressure ratio, sub-sonic (sub-critical) flow exists.

The critical pressure ratio is about 0.55 for natural gas, and a similar constant is used for oil flow.

In some wells, chokes are installed in the lower section of tubing strings. This choke arrangement reduces wellhead pressure and enhances oil production rate as a result of gas expansion in the tubing string. For gas wells, a downhole choke can reduce the risk of gas hydrates. A major disadvantage of using downhole chokes is that replacing a choke is costly.

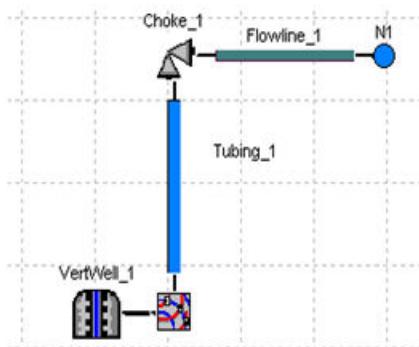
Task 5: Modeling a Flowline and Choke

In this section, you add a horizontal flow line and a choke to the model. You use the gas rate calculated in the previous to determine the choke bean size that results in a manifold (end of flowline) pressure of 710 psia.

To model a flowline and choke:

1. Ensure the tubing ID is set to 3.958 inches.
2. Insert a choke at the wellhead and reconnect the tubing to the choke.
3. Select the mechanistic model for both critical and sub-critical flow.

Note: TIP: You can enter any choke size you wish, but it will be overridden by the sensitivity variable.



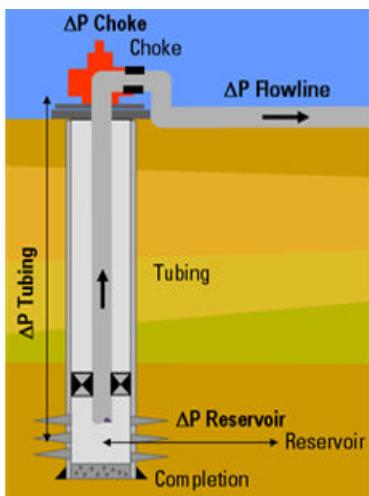
4. Insert a flowline downstream of the choke and connect it to a node representing the manifold.
5. Specify the flowline using the data in the table.

Flow-line length	300 feet
Flow-line ID	6 inches

Pipe Roughness	0.001 inches
Wall thickness	0.5 inches
Ambient Temp	60 degF

6. Select Operations » Pressure Temperature Profile.

- a. Select User Variable as calculated and input a choke size. A good estimate is a size between 1 inch and 3 inches.
 - b. Set the Outlet Pressure to 710 psia.
 - c. Specify the gas flow rate calculated in the previous .
 - d. Run the model and see the PsPlot for the choke size.
7. Enter the resulting choke size into the choke model.
8. Re-run the Pressure/Temperature profile with outlet pressure as the calculated variable to verify that the calculated wellhead pressure is 800 psia.
9. Inspect the output file to determine individual pressure drops for the reservoir, tubing, choke and flow line.



P_o = 710 psia	
Choke size	1.515 inch
Pressure losses across system	
Δ P Reservoir	3232 psi
Δ P Tubing	568 psi
Δ P Choke	88 psi
Δ P Flow-line	2 psi

Table 2.32: Results

Task 6: Predicting Future Production Rates

In this section, you use System analysis to calculate the gas rate as a function of reservoir pressure.

To predict future production rates:

1. Right-click and choose **Active** to deactivate the choke and flowline. These objects should be highlighted in red to indicate they are inactive.
2. Select **Operations » System Analysis**.
3. Choose Gas Rate as the calculated variable.
4. Set the wellhead pressure to 800 psia.
5. Use Reservoir (Static) Pressure as the X-axis variable and set these values:
 - 4,600 psia
 - 4,300 psia
 - 3,800 psia
 - 3,400 psia
6. Run the model and view the resultant plot.

Reservoir Pressure (psia)	Gas Rate (mmscfd)
4600	8.059
4200	10.281
3800	13.374
3400	15.388

Table 2.33: Results

2.3.6 Liquid Loading

Gas wells usually produce natural gas-carrying liquid water and/or condensate in the form of mist. As the gas flow velocity in the well drops because of reservoir pressure depletion, the carrying capacity of the gas decreases. When the gas velocity drops to a critical level, liquids begin to accumulate in the well (liquid loading).

This increases the bottomhole pressure, which reduces the gas production rate. A low gas production rate will cause gas velocity to drop further and, eventually, the well will cease producing.

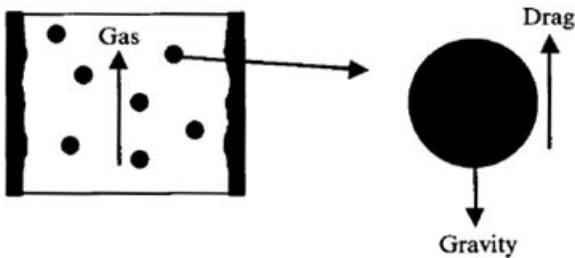
Turner Droplet Model

In predominantly gas wells operating in the annular-mist flow regime, liquids flow as individual particles (droplets) in the gas core and as a liquid film along the tubing wall.

By analyzing a large database of producing gas wells, Turner found that a force balance performed on a droplet could predict whether the liquids would flow upwards (drag forces) or downwards

(gravitational forces). If the gas velocity is above a critical velocity, the drag force lifts the droplet, otherwise the droplet falls and liquid loading occurs. This is illustrated in next figure.

Figure 2.21. Turner Droplet model



When the drag is equal to weight, the gas velocity is at critical. Theoretically, at the critical velocity, the droplet would be suspended in the gas stream, moving neither upward nor downward. Below the critical velocity, the droplet falls and liquids accumulate in the wellbore.

In practice, the critical gas velocity is generally defined as the minimum gas velocity in the tubing string required to move droplets upward.

The general form of Turner's equation is given by:

$$V_t = \frac{1.593\sigma^{1/4}(\rho_l - \rho_g)^{1/4}}{\rho_g^{1/2}} \text{ ft/sec}$$

where

ρ_g = gas phase density (lbm/ft³)

ρ_l = liquid phase density (lbm/ft³)

σ = interfacial tension (dynes/cm)

v_t = terminal velocity of liquid droplet (ft/sec)

Liquid loading calculations are performed in every operation and are available for review in output files and plot reports. Review the output file to determine if the well is under liquid loading.

A value of Liquid Loading Velocity Ratio in excess of 1 indicates loading.

The NODAL analysis plot will report the Liquid Loading Gas Rate when the X-axis is configured to display gas rate. For every point on the outflow curve, the value of Liquid Loading Velocity Ratio is calculated and the critical gas rate is calculated at a point where liquid loading velocity ratio is equal to 1.

Note: The reported value comes from interpolation of the outflow curve between two points, one with a velocity ratio below 1 and another with a velocity ratio above 1. Therefore, the accuracy of the results depends on the number of points on the outflow curve.

Task 7: Determining a Critical Gas Rate to Prevent Well Loading

To determine the Critical Gas Rate:

1. Select **Operations** » **NODAL analysis**.
2. Select **Limits** and change these settings:
 - Number of points on each inflow curve = 100
 - Number of points on each outflow curve = 200
 - Inflow curves to extend to the AOFP
 - Outflow curves limited to the pressure range of the inflow curves
3. Set the outlet pressure to 800 psia and run the model.
4. Plot the Pressure at NA point vs. Stock Tank Gas Rate. Note the stock tank gas rate on the Data tab. The reported critical gas rate is 2.069 mmscfd

Note: The reported critical gas rate refers to the outflow curve, which you can validate by performing a Pressure/Temperature Profile operation at the same conditions (flow rate and outlet pressure).

5. Perform a Pressure/Temperature Profile operation to calculate inlet pressure at the given critical gas rate corresponding to outflow outlet pressure of 800 psia.
6. View the output file to see if the Maximum Liquid Loading Velocity Ratio is close to 1, which is consistent with the results of the NODAL analysis.

2.4 Looped Gas Gathering Network Case Study

In this case study, you must model the network as a complete system to account for the interaction of wells producing in a common gathering system. The wellhead pressure and, by extension, the deliverability of any particular well is influenced by the backpressure imposed by the production system.

Modeling the network as a whole allows the engineer to determine the effects of such actions as adding new wells, adding compression, looping flowlines and changing the separator pressure.

In this case study, you learn how to build a gathering network and perform a network simulation to evaluate the deliverability of the complete system. This case study involves the following tasks:

- [Build a Model of a Network \(p.331\)](#)
- [Perform a Network Simulation \(p.331\)](#)

2.4.1 Model a Gathering Network

Network models are constructed using the network module and solved using its calculation engine. The basic stages involved in developing a network model are:

1. Build a model of the field, including all wells and flowlines.
2. Specify the boundary conditions.
3. Run the model.

Boundary Conditions

To solve the network model, you must enter the correct number of boundary conditions. Boundary nodes are those that have only one connecting branch, such as a production well, injection well, source or sink.

The number of boundary conditions required for a model is determined by the model's Degrees of Freedom, determined as follows:

Degrees of Freedom = number of wells (production and injection) + number of sources + number of sinks

For example, a 3 production well system producing fluid to a single delivery point has 4 degrees of freedom (3+1), regardless of the network configuration between the well and the sink.

Each boundary can be specified in terms of Pressure OR Flow rate OR Pressure/Flow rate (PQ) curve.

Additionally, the following conditions must be satisfied:

- The number of pressure, flow rate or PQ specifications must equal the degrees of freedom of the model.
- At least 1 pressure must be specified.
- At each source (production well & source) the fluid temperature must be set.

Solution Criteria

A network has converged when the pressure balance and mass balance at each node are within the specified tolerance. The calculated pressure at each branch entering and leaving a node is averaged, and the tolerance of each pressure is calculated from the equation:

$$Ptol = \frac{P - Pave.}{Pave. \times 100 \%}$$

If all Ptol values are within the specified network tolerance, that node has passed the pressure convergence test. This is repeated for each node.

The total mass flow rate into and the total mass flow rate out of a node are averaged. The tolerance is calculated from the equation:

$$Ptol = \frac{(Total\ mass\ flow\ rate\ in) - (Total\ mass\ flow\ rate\ average)}{Total\ mass\ flowrate\ average \times 100 \%}$$

If the Ftol value is within the specified network tolerance, that node has passed the mass convergence test. This is repeated for each node.

The network has converged when all of the foregoing conditions are satisfied.

2.4.2 Task 1: Building a Model of a Network

In this case study, your goal is to establish the deliverability of a production network. The network connects three producing gas wells in a looped gathering system and delivers commingled product to a single delivery point.

Getting Started

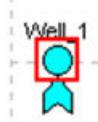
1. Open PIPESIM and go to **File** » **New** » **Network** to create a new network model.
2. Go to **File** » **Save As** to save the model in your training directory, such as c:\training\pn01.bpn.

Building the Model

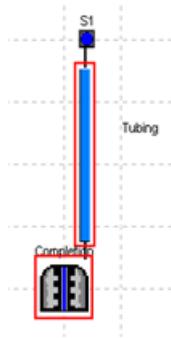
Using the engineering data available at the end of this case study, build a model of a network.

To build the model:

1. Click **Production Well** to place Well_1 in the work area, as shown.



2. Double-click on Well_1 to reveal the components, as shown below.



3. Double-click on the vertical completion to enter the inflow performance data.
4. Enter a gas PI of 0.0004 mmscf/d/psi² and a reservoir temperature of 130 degF.

Note: You will enter the reservoir pressure later when the network boundary conditions are specified. For now, enter any value you wish for reservoir pressure.

5. Double-click on the tubing and select Simple Model as the preferred tubing model.
6. Define vertical tubing with a wellhead datum MD of 0 feet and mid perforations TVD and MD of 4,500 feet.
7. The ambient temperatures are 130 degF at mid-perforations and 60 degF at the wellhead. The tubing has an I.D. of 2.4 inches.

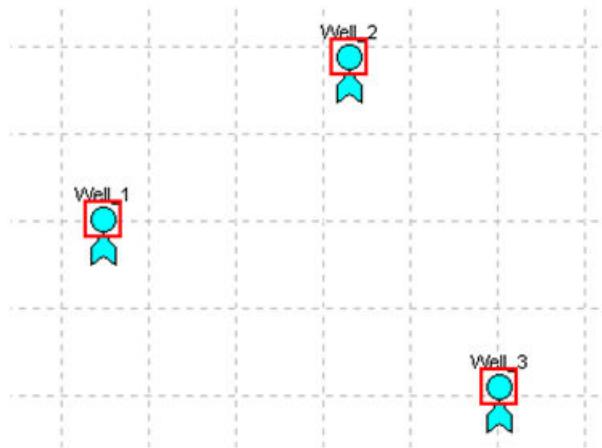
Note: Essential data fields are shown in a red outline; if the fields are not outlined, data entry is optional.

8. Close the view of Well_1 by clicking at the upper-right corner of the window, or by selecting **File** » **Close** to return to the network view.
9. Copy the data to Well_2 and Well_3.
 - a. Select Well_1.

- b. Using the commands **Edit » Copy** and **Edit » Paste** (or **Ctrl + C** and **Ctrl + V**), create two copies of the well.

Note: By default, the names of the copied wells will be Well_2 and Well_3 and contain the same input data as Well_1.

10. Position the new wells, as shown.



11. Modify the data of Well_3.

- Double-click on Well_3 and modify the completion and tubing data.
- For the vertical completion, enter a gas PI of 0.0005 mmSCF/d/psi² and a reservoir temperature of 140 F.
- Define vertical tubing with a wellhead TVD of 0 and mid-perforations TVD and MD of 4,900 feet.
- The ambient temperatures are 140 degF at the mid-perforation depth and 60 degF at the surface. The tubing has an I.D. of 2.4 inches.
- Close the view of Well_3 to return to the network view.

12. Specify the composition of each production well. This step defines the compositions at the production wells. Well_1 and Well_2 are producing from the same zone and, thus, are assumed to have the same composition. Well_3 has a composition that is different than that shown in the data section at the end of the case study. The most efficient way to define the compositions is to set the more prevalent composition (that for Wells_1 and Well_2) as the global composition, then specify the composition of Well_3 as a local composition.

Note: The composition data of all wells is provided at the end of this in Summary data.

- Save the current network model.
- Define the global template of all components used in the network model.
 - Select **Setup » Compositional Template** menu.
 - Add all library components (Hydrocarbon as well as aqueous components).

- c. Under the Petroleum Fraction tab, specify the name and properties of the petroleum fraction and add it to the list of template components.
- d. Select **Setup > Compositional (Network Default)** and enter the mole fraction for all components to define global composition (Well_1 and Well_2).

Note: By default, the network global composition applies to all sources/wells in the network model. This can be checked by viewing the network fluid summary under **Setup > Fluid Models**. To define a different composition for any particular source/well, you must set it locally.

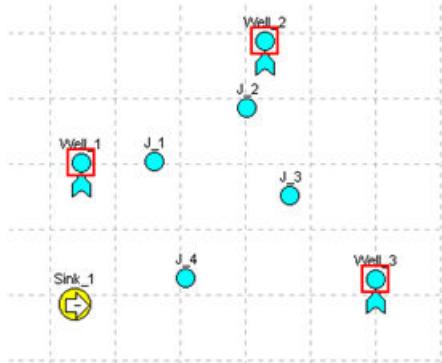
- e. Define the local composition for Well_3:
 1. Right-click on Well_3.
 2. Choose Fluid Model.
- f. Select **Use locally defined fluid model** and click **Edit**.
- g. Choose **Local Compositional** and click **Edit Composition**.
- h. Enter the composition of Well_3.

13. Connect the network together.

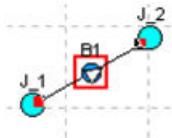
- a. Insert a sink and some junction nodes.

Note: Holding down the Shift key while placing junction nodes allows for multiple insertions. Be sure to release the Shift key before the final insertion.

The network should now look like this:



- b. Use the **Branch** button to connect J_1 to J_2. To do this:
 1. Click the **Branch** object.
 2. Hold down the left mouse button over J_1 and drag the cursor to J_2.
 3. Release the mouse button. A connected branch is shown in the figure.



14. Double-click on the arrow in the center of B1 to enter data for that branch.

- Double-click on the flowline to enter the following data:

Rate of Undulations	10/1000
Horizontal distance	30,000 feet
Elevation difference	0 feet
Inner Diameter	6 inches
Wall Thickness	0.5 inches
Roughness	0.001 inches
Ambient Temperature	60 degF

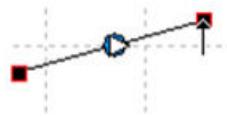
- Close the **B1** window to return to the network view.

15. As the looped gathering lines are all identical, the data for branch B1 can be used to define the other looped gathering lines.

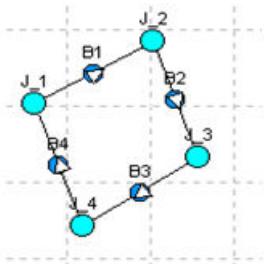
- Select **B1**. Click the arrow in the middle of the branch and copy/paste B1 to create B2, B3, and B4.

- To connect a pasted branch:

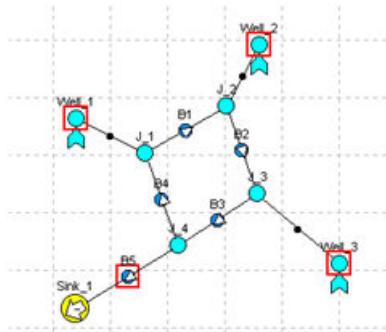
- Click the arrow in the middle of the new branch. You will see that highlight boxes display at either end of the branch.
- Move the cursor over the right-hand, highlight box. The cursor changes to an up arrow. Use this end of the branch to drag and drop onto a junction node.



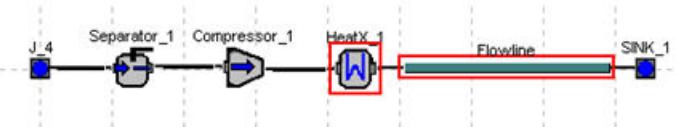
- Position the new branches.



- Connect the wells to the adjacent junction node and connect J_4 to the sink.



16. Double-click on branch **B5** and insert the following objects in the left-to-right order shown in the figure:



- Liquid separator with an efficiency of 100%
- Compressor with a pressure differential of +400 psi and an efficiency of 70%
- After-cooler (heat exchanger) with an outlet temperature of 120 degF and ΔP of 15 psi
- Flowline with the following properties:

Rate of undulations	10/1000
Horizontal distance	10,000 feet
Elevation difference	0 feet
Inner Diameter	8 inches
Wall Thickness	0.5 inches
Roughness	0.001 inches
Ambient Temperature	60 degF

- a. Click **Connector** to join the equipment together.
- b. Close the **Single Branch** window.

17. Select **Setup** ➤ **Flow Correlations** menu and choose **Beggs-Brill Revised** as the global vertical and horizontal multiphase flow correlations.

18. In the Options Control tab of the Flow Correlations menu:

- a. Select **Use network options**.
- b. Click **Apply network options to all branches**.

19. Select **Setup** ➤ **Erosion and Corrosion Options** and choose the **deWaard Corrosion** model.

Note: This model calculates a corrosion rate caused by the presence of CO₂ dissolved in water. Concentrations of CO₂ and water are obtained from the fluid property definitions, (black

oil or compositional). The corrosion rate will be zero if CO₂ or the liquid water phase is absent from the fluid.

20. In the **Options Control** tab of the **Erosion and Corrosion Options** menu:

- a. Select **Use network options**.
- b. Click **Apply network options to all branches**.

21. Save the model as gas_network.

2.4.3 Task 2: Performing a Network Simulation

To perform a network simulation:

1. Select **Setup » Boundary Conditions** and specify these boundary conditions:

Node	Pressure
Well_1	2,900 psia
Well_2	2,900 psia
Well_3	3,100 psia
Sink_1	800 psia

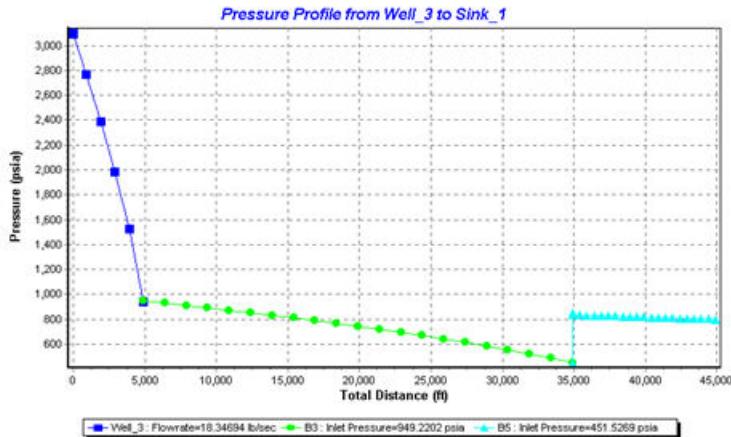
All flow rates are calculated by the network solver.

Note: Any pressure specification defined in the single branch model must be re-specified in the network model. However, the boundary pressures specified in the Network view will update the pressures defined in the single branch model for use in single branch operations.

2. Open the **Setup » Iterations** menu to set the network tolerance to 1%.
3. Save the model.
4. Click **Run**.
5. When the network has solved, you should see the message: Gas_networkbpn01 – Finished OK. When this message displays, click **OK**.
6. Click **Report Tool**. What is the gas production rate at the sink? 42.51 mmscf/d.

Note: More comprehensive reporting is available by clicking **Summary File**.

7. Hold down the **Shift** key and select the flow route from Well_3, branch B3 and branch B5.
8. Click **Profile Plot**. You should obtain the pressure profile for these three branches, and the effect of the compressor at J_4 on the system pressure should look similar to the figure below.



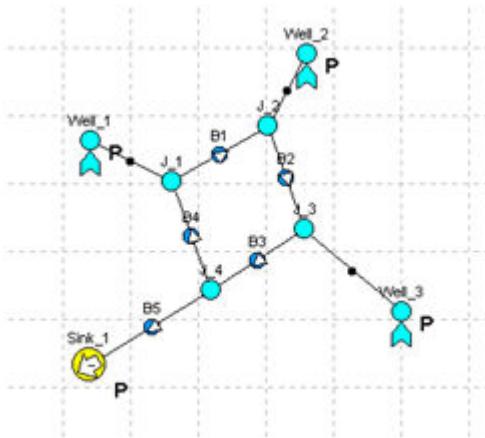
9. Select **Series** and change the Y-axis to Corrosion Rate to observe the calculated corrosion rate. Maximum Corrosion Rate in network = 44.902 mm/year
10. Determine the field production rate in the event of a compressor shutdown. Assuming a bypass line exists around the compressor, deactivate the compressor object and rerun. Gas production rate at the Sink: 38.47 mmscfd

Note: Edit the legend and title on PsPlot to improve the graphical presentation.

Looped Gathering Network Data

The tables that follow contain the data for task 3.

Figure 2.22. Network layout



	Well_1 and Well_2	Well_3
Gas PI	0.0004 mmscf/d/psi ²	0.0005 mmscf/d/psi ²
Wellhead TVD	0	0
Mid Perforations TVD	4,500 feet	4,900 feet

	Well_1 and Well_2	Well_3
Mid Perforations MD	4,500 feet	4,900 feet
Tubing I.D.	2.4 inch	2.4 inch
Wellhead Ambient Temperature	60 degF	60 degF
Mid Perforations Ambient Temperature	130 degF	140 degF

Table 2.34: Completion and Tubing Data

Component	Moles
Carbon Dioxide	3
Methane	72
Ethane	6
Propane	3
Isobutane	1
Butane	1
Isopentane	1
Pentane	0.5
Hexane	0.5

Table 2.35: Pure Hydrocarbon Components (Well_1 and Well_2)

Name	Boiling Point (degF)	Moecular Weight	Specific Gravity	Moles
C7+	214	115	0.683	12

Table 2.36: Petroleum Fraction (Well_1 and Well_2)

Component	Volume ratio (%bbl/bbl)
Water	10

Table 2.37: Aqueous Component (Well_1 and Well_2)

Component	Moles
Carbon Dioxide	2
Methane	71
Ethane	7
Propane	4
Isobutane	1.5
Butane	1.5
Isopentane	1.5
Pentane	0.5

Component	Moles
Hexane	0.5

Table 2.38: Pure Hydrocarbon Components (Well_3)

Name	Boiling Point (degF)	Molecular Weight	Specific Gravity	Moles
C7+	214	115	0.683	10.5

Table 2.39: Petroleum Fraction (Well_3)

Component	Volume ratio (%bbl/bbl)
Water	5

Table 2.40: Aqueous Component (Well_3)

Rate of undulations	10/1000
Horizontal distance	30,000 feet
Elevation difference	0 feet
Inner diameter	6 inch
Wall thickness	0.5 inch
Roughness	0.001 inch
Ambient temperature	60 degF
Overall heat transfer coefficient	0.2 Btu/hr/ft ² /degF

Table 2.41: Data for Looped Gathering Lines (B1, B2, B3, and B4)

2.5 User Equipment DLL Tutorial - User Pump

A 32-bit DLL, compatible with PIPESIM, is to be produced to model a piece of user equipment, in this case a pump. This tutorial involves the following tasks:

- Write a DLL to define a piece of equipment for use with PIPESIM (p.334).
- Build a single branch model and use the DLL (p.338).
- Run the model and verify that the DLL worked (p.341)

2.5.1 Task 1. Write a DLL to define a piece of equipment for use with PIPESIM

Introduction

The DLL can be written in any programming language. Examples are provided in Fortran and C++ together with the necessary library and header files. Also provided are READ ME's for Fortran and C++ code. For a standard installation of PIPESIM, the following files are provided.

C:\Program Files\Schlumberger\PIPESIM\Developer Tools \User Equipment\Fortran_Code\UserPump.vfproj	Visual Fortran Project
C:\Program Files\Schlumberger\PIPESIM\Developer Tools \User Equipment\Fortran_Code\UserPump.f90	Fortran code
C:\Program Files\Schlumberger\PIPESIM\Developer Tools \User Equipment\C++_Code\UserPump.vcproj	Visual C++ Fortran Project
C:\Program Files\Schlumberger\PIPESIM\Developer Tools \User Equipment\C++_Code\UserPump.cpp	C++ code
C:\Program Files\Schlumberger\PIPESIM\Developer Tools \User Equipment\libs\fluidman.i	PIPESIM include file for Fortran
C:\Program Files\Schlumberger\PIPESIM\Developer Tools \User Equipment\libs\fluidman.h	PIPESIM header file for C++
C:\Program Files\Schlumberger\PIPESIM\Developer Tools \User Equipment\libs\fluidman.lib	PIPESIM fluid manager library
C:\Program Files\Schlumberger\PIPESIM\Developer Tools \User Equipment\Fortran_Code\READ.ME	READ ME for Fortran code
C:\Program Files\Schlumberger\PIPESIM\Developer Tools \User Equipment\C++_Code\READ.ME	READ ME for C++ code
C:\Program Files\Schlumberger\PIPESIM\Case Studies\User Equipment\UserPump.BPS	Example case using user supplied DLL
C:\Program Files\Schlumberger\PIPESIM\Case Studies\User Equipment\Pump.bps	Example case for comparison with UserPump.bps

How to write a DLL

The PIPESIM simulators are written in Fortran and can call an external routine using the following interface:

```
interface
  subroutine equipment(op1,op2,istream,ostream,omsg,rcode)
    character(*) , intent(in) :: op1
    character(*) , intent(in) :: op2
    type(fman_stream_type), intent(in) :: istream
    type(fman_stream_type), intent(out) :: ostream
    character(*), intent(out) :: omsg
    integer , intent(out) :: rcode
  end subroutine equipment
end interface
```

The first two arguments are options provided by the user, through the PIPESIM GUI, and passed via the PIPESIM simulators to the user DLL. The first option is intended to be a global option, perhaps specified by the author of the DLL, while the second option is a local option, specified by the user of the DLL. The difference between these two arguments will become more apparent in the section on [Engine keyword tool \(p.340\)](#). Arguments three and four define, respectively, the fluid input to, and output from, the equipment. PIPESIM fluid properties are stored in a type called 'fman_stream_type'. The definition of this type is provided by an include file (fluidman.i for Fortran)

or header file (fluidman.h for C++). Argument five is an error message. This will be displayed by PIPESIM if argument six is non-zero.

Any name can be used for the routine.

Note: The routine is called pump in the example code UserPump.f90 and UserPump.cpp.

Note on Fortran

If the dll needs to read from or write to a file, then care must be taken since Fortran I/O unit numbers will be shared with PIPESIM. The following guidelines are recommended:

- Use unit numbers greater than 100.
- Use the INQUIRE statement to make sure the unit is not already opened.
- Close all units on exit from the DLL.

Note on C/C++

If the DLL is written in C or C++, then extra arguments are required after each character argument, because the Fortran compiler passes the address of the character string and its length. Therefore the C++ declaration is:

```
void equipment(char * op1, long lenop1,
               char * op2, long lenop2,
               fman_stream_type & istream,
               fman_stream_type & ostream,
               char * omsg, long lenomsg,
               long & rcode)
```

The input character arrays will not be null terminated.

PIPESIM flash

The fman_stream_type contains fluid data that is input to the flash, (for example, pressure, temperature, compositional handle, compositional type), and data output from the flash (for example phase splits, densities, viscosities). Two flash routines are provided with the following interfaces:

```
interface
  subroutine fman_stream_flow(stream, rcode)
    use fluidman_types
    type(fman_stream_type) :: stream
    integer, intent(out) :: rcode
  end subroutine fman_stream_flow
  subroutine fman_stream_st(stream, rcode)
    use fluidman_types
    type(fman_stream_type) :: stream
    integer, intent(out) :: rcode
  end subroutine fman_stream_st
end interface
```

Note: `c_fman_stream_flow` is merely a wrapper to `fman_stream_flow` and is intended to be used by C/C++ applications.

These two routines can be used to perform three types of flash:

- a PT-flash at user defined pressure (P) and temperature (T). To perform a PT flash at a given pressure (P) and temperature (T) on an `fman_stream_type` called `stream`, set the stream values:
 - `stream%ipres=P`
 - `stream%itemp=T`
 - `stream%ienth = r4undef` (i.e. undefined)and call the PIPESIM routine `fman_stream_flow`.
- a PH-flash at user defined pressure and enthalpy (H). To perform a PH flash at a given pressure (P) and enthalpy (H) on an `fman_stream_type` called `stream`, set the stream values:
 - `stream%ipres=P`
 - `stream%itemp=r4undef` (i.e. undefined)
 - `stream%ienth =H`
- a stock tank flash at atmospheric pressure and 60°F. To perform a stock tank flash call the routine `fman_stream_st`.

Note: A PH flash is used in the example code `UserPump.f90` and `UserPump.cpp`.

Results are stored in the stream. For example, `stream%ptdl` contains the liquid density calculated by either a PT flash or a PH flash; `stream%stdl` contains the stock tank liquid density calculated by the stock tank flash (See `fluidman.i` and `fluidman.h` for more details). Examples of calling `fman_stream_flow` and `c_fman_stream_flow` are given in `UserPump.f90` and `UserPump.cpp` respectively. An example of calling `fman_stream_st` is not provided since calling this routine should not be necessary; the stock tank conditions of the outlet stream will be the same as those of the input conditions (unless the equipment changes the fluid).

Exporting routines

The DLL export (and preprocessor) commands for these compilers are given in the example files `UserPump.f90` and `UserPump.cpp`.

Build the DLL

PIPESIM is compiled using Intel Visual Fortran. We have tested the example projects by compiling and building with Intel Visual Fortran 9.1 and Microsoft Visual C++ 2005. The built DLL will be put in the following directory for a standard installation of PIPESIM:

C:\Program Files\Schlumberger\PIPESIM\Developer Tools\User Equipment\Programs.

2.5.2 Task 2. Build a single branch model and use the DLL

There are three steps to using the equipment DLL in PIPESIM:

- Place the built DLL in the PIPESIM programs directory (p.338).
- Inform PIPESIM of the equipment DLL and its entry points (p.338).
- Build a single branch model with an Engine Keyword Tool to represent the piece of equipment (p.340).

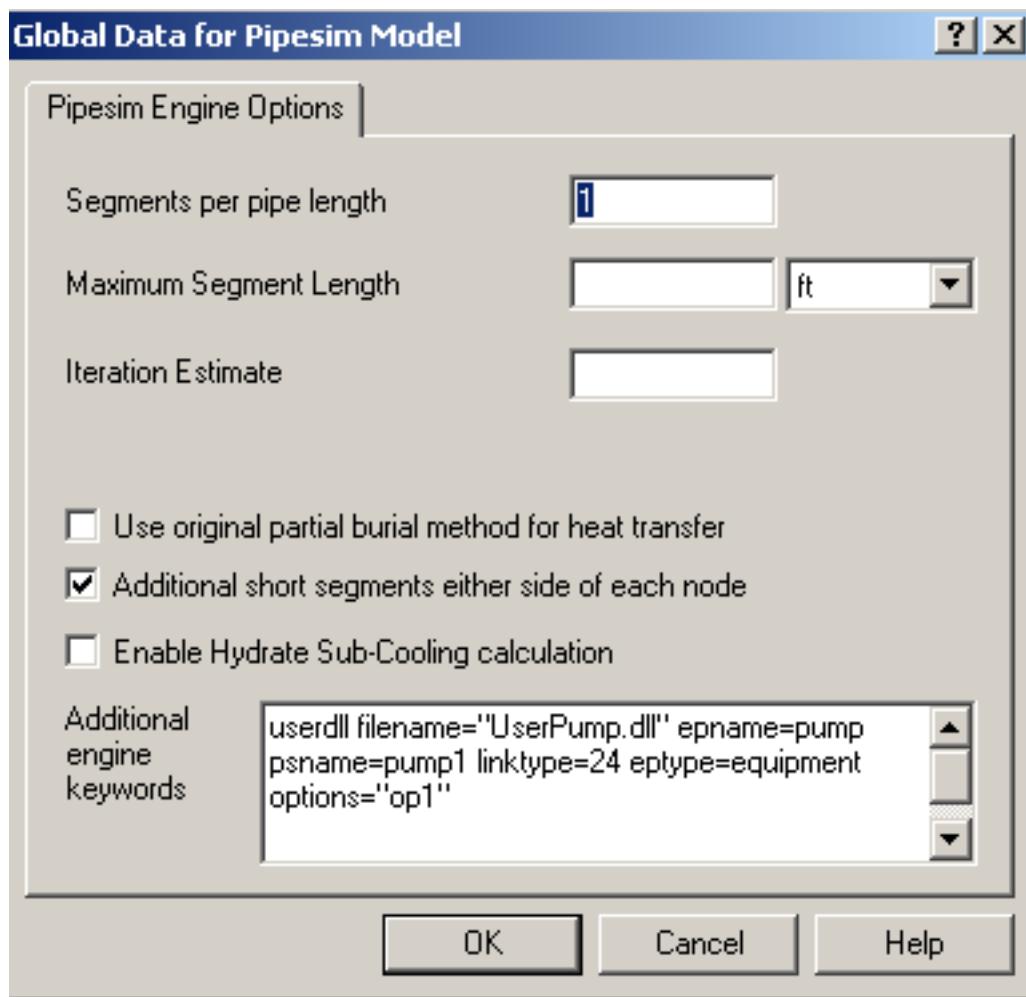
Place the built DLL in the PIPESIM programs directory

You **must** place the DLL in the PIPESIM programs directory. For a standard installation of PIPESIM this is:

C:\Program Files\Schlumberger\PIPESIM\Programs

Inform PIPESIM of the equipment DLL and its entry points

In the UserPump.bps example case, the DLL information has been put in **Setup » Engine Options**:

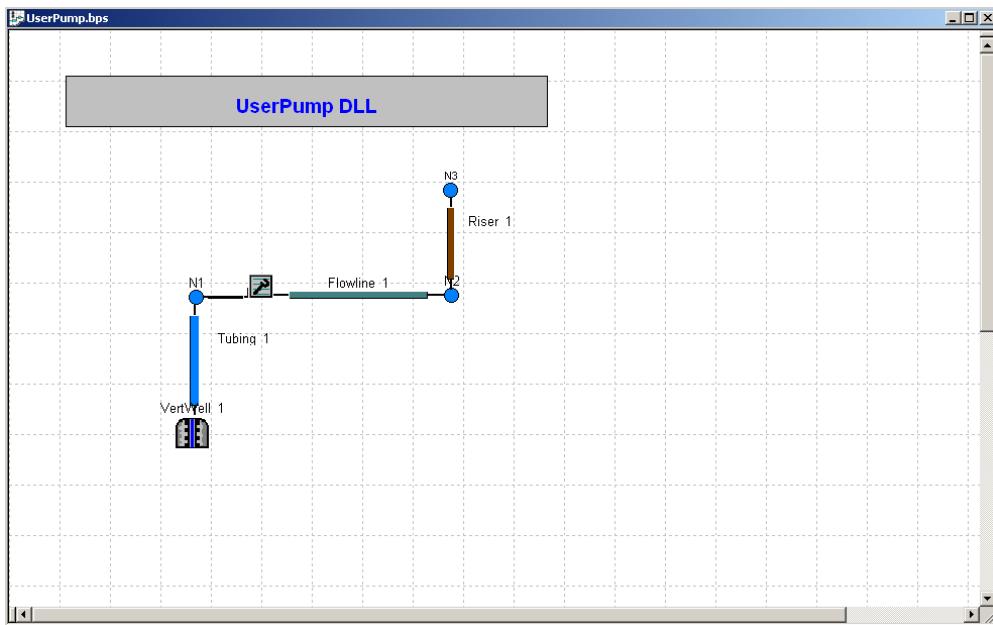


You must specify the maincode **userdll** and values for all of its subcodes. The subcodes have the following meanings:

- **filename** specifies the name of the DLL.
- **epname** (entry point name) specifies the entry point of the DLL, the actual name of the routine as exported from the DLL.
- **psname** (PIPESIM name) specifies the internal PIPESIM name of the routine. The psname's must be unique. Check that other DLLs specified in the userdll.dat file (located in C:\Program Files\Schlumberger\PIPESIM\data for a standard installation of PIPESIM - look for ep_ident) do not use the same psname's.
- **linktype** specifies the DLL linkage type. This must be 24.
- **eptype** (entry point type) specifies the type entry point for the DLL. Note that it must be **equipment** to distinguish it from flow correlations.
- **options** specifies the string that will be sent as the first argument to the routine. (This is a global option, perhaps specified by the author of the DLL).

Build a single branch model with an Engine Keyword Tool to represent the piece of equipment

The only way of using this equipment piece is by inserting an “Engine keyword tool” into the branch, as shown in the UserPump.bps example case:

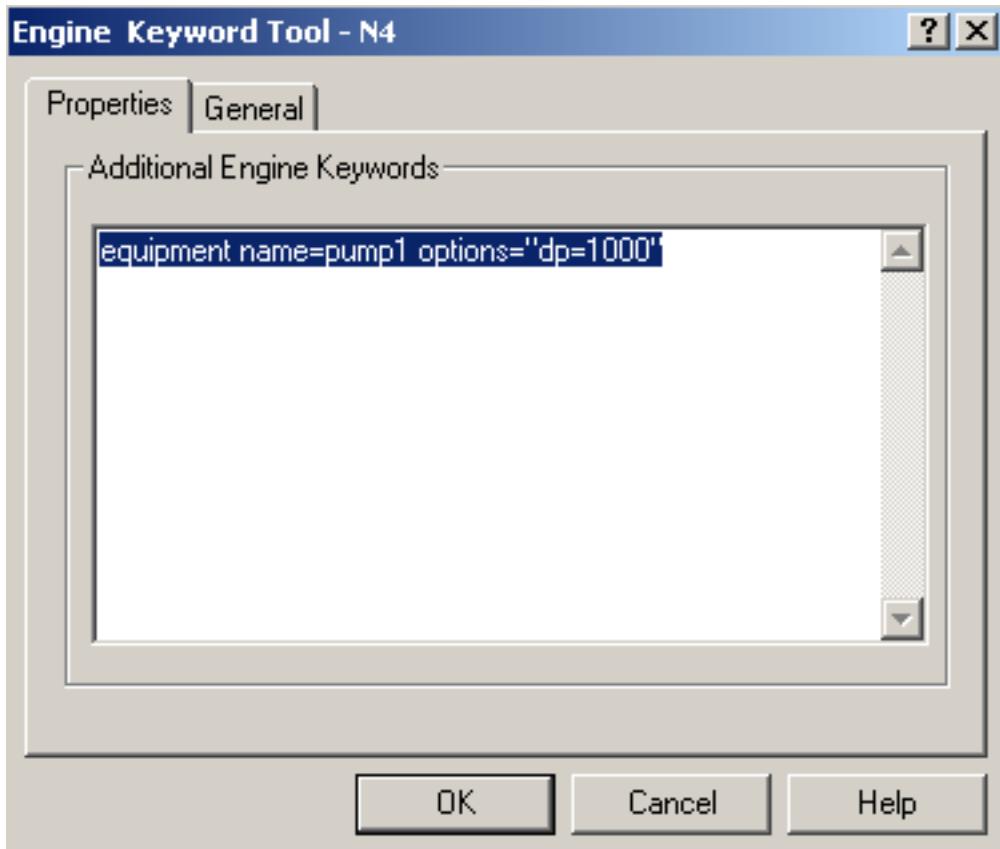


Use this tool to add the following data:

equipment name=value options="op2"

Where:

- Value must be one of the psname values given with the userdll maincode (see [Inform PIPESIM of the equipment DLL and its entry points \(p.338\)](#))
- The string “op2” is passed as the 2nd argument to the particular DLL routine linked to value. In the UserPump.bps example provided, “dp=1000” (represents a pressure differential across of 1000 psia across the equipment piece) is passed to routine pump in DLL UserPump.dll:



The author of the DLL should make sure the users know what options (if any) are allowed.

Note on Network models

The user DLL can also be used with network models. The user DLL information can again be added using **Setup** » **Engine Options**. It can be added either at the bottom or the top of the tnt file.

2.5.3 Task 3. Run the model and verify that the DLL worked

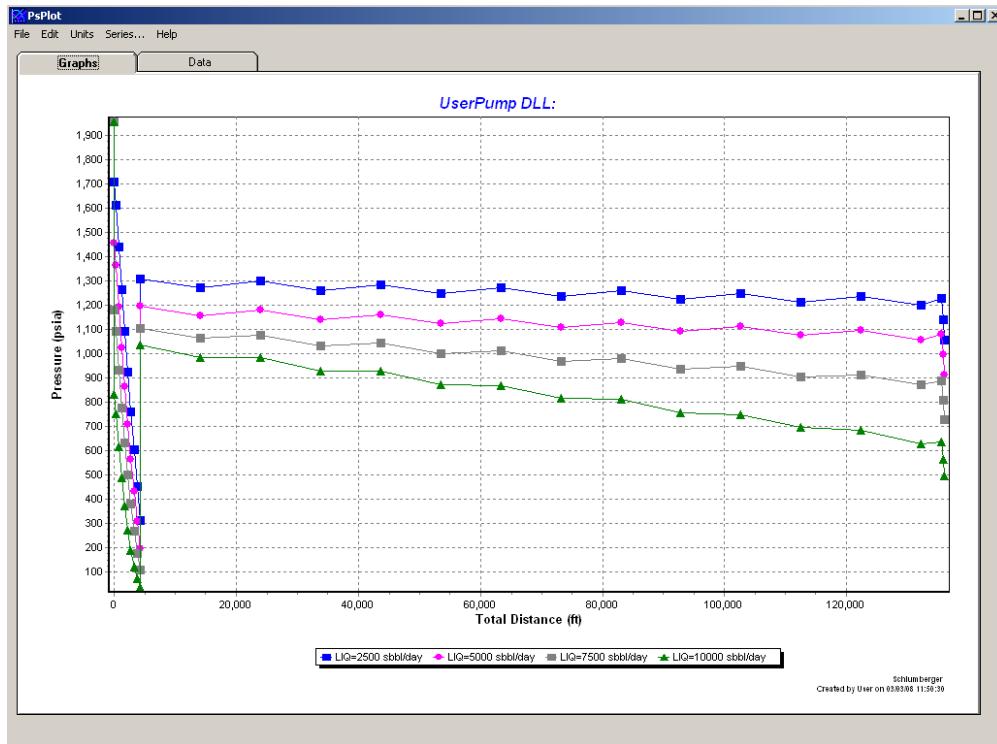
Perform the following steps to verify that the DLL worked:

1. Run the UserPump.bps model by clicking

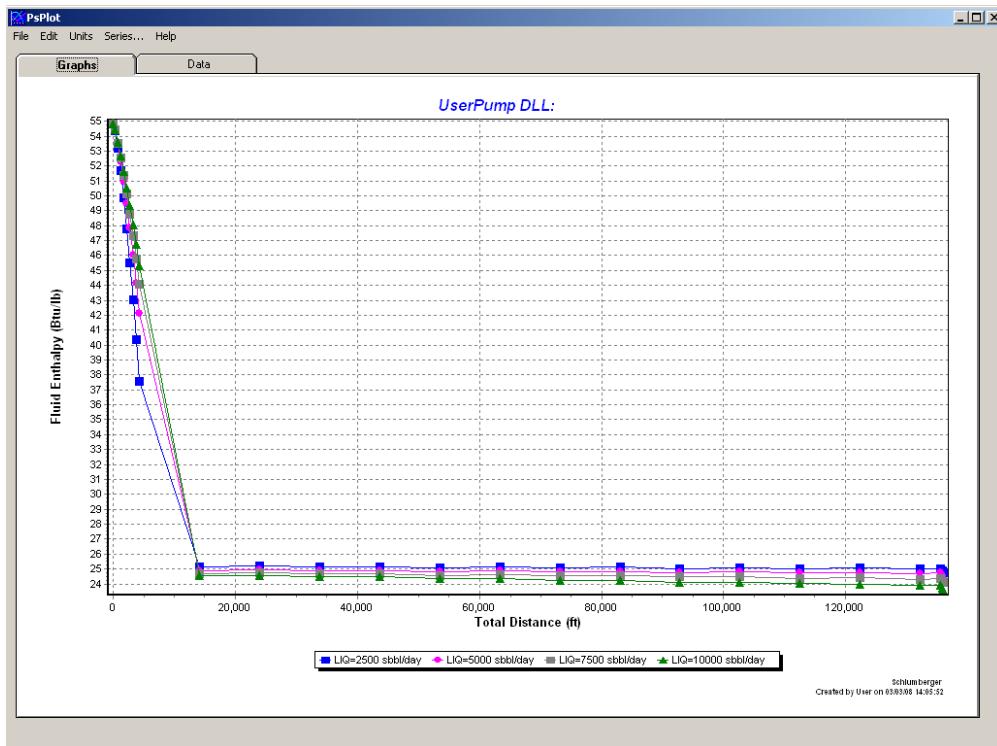


2. View the profile plots.

The following profile plots should be obtained for pressure vs total distance:

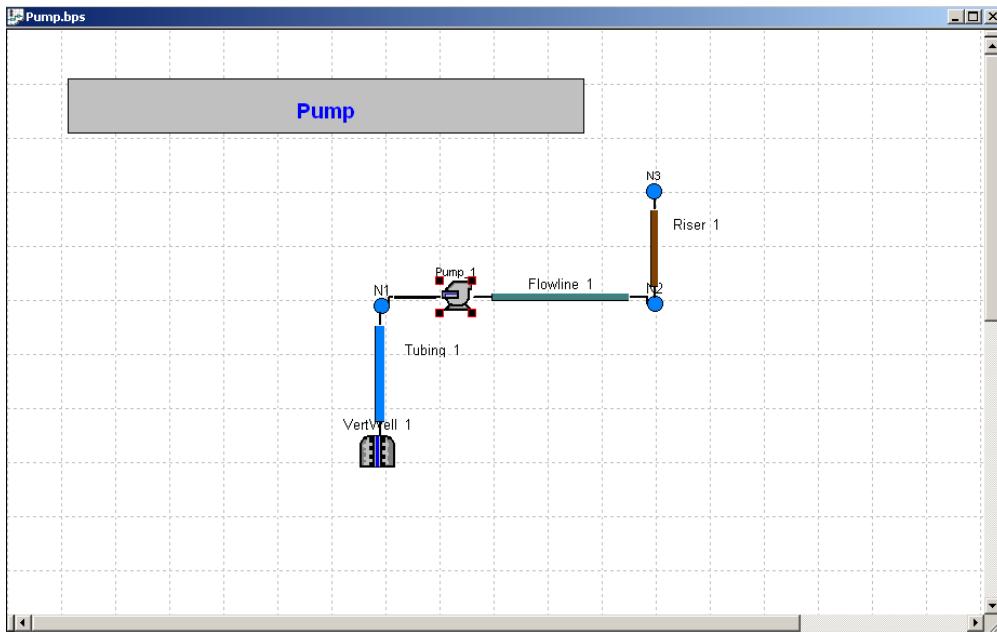


and fluid enthalpy vs total distance:

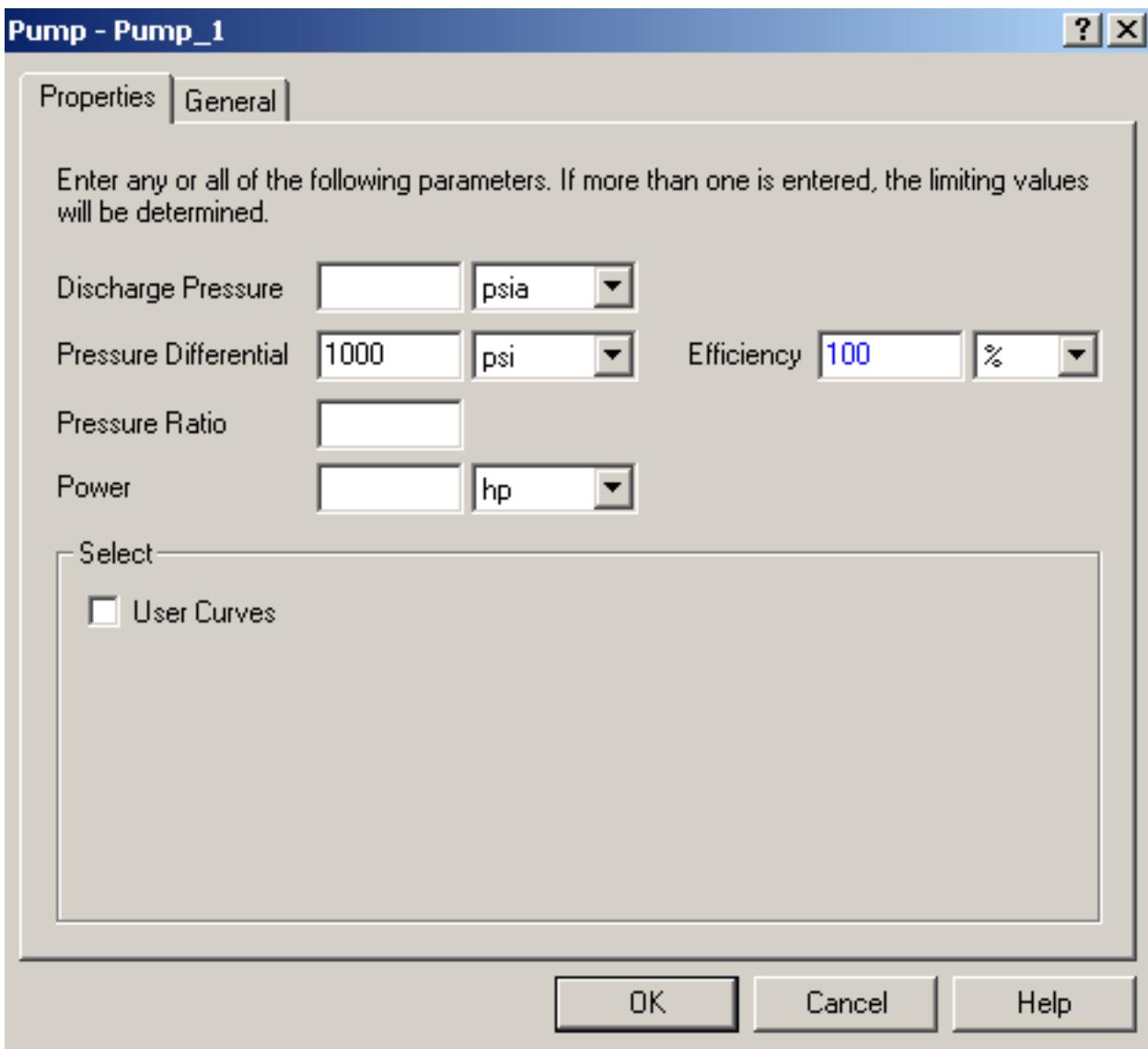


It is clear from the plots that the DLL is working as intended, since a pressure differential of 1000 psia at a constant enthalpy of about 25 BTU/lb is applied across the equipment.

An example case entitled Pump.bps is provided, for comparison with UserPump.bps. This case is identical to UserPump.bps except that the Engine Keyword Tool representing the user defined pump (DLL) is replaced by a standard PIPESIM pump:



Similarly, a pressure differential of 1000 psia is applied across the 100% efficient (i.e. at constant enthalpy) pump:



Perform the following steps:

1. Run the Pump.bps model by clicking on the



2. View the profile plots.

You should get plots of pressure vs total distance and fluid enthalpy vs total distance similar to those obtained in the UserPump.bps example case.

2.6 Field Data Matching Tutorial

Field data matching is important in building an accurate model. PIPESIM lets you perform fluid properties matching, IPR matching, flow correlation matching, temperature matching, equipment settings, and so on. The aim of such matching is to eliminate uncertainties based on measured

data or operating conditions and create a model that follows field measurement. This tutorial involves the following tasks:

- [IPR matching \(p.346\)](#).
- [Optimize Flow Correlation and U Value \(p.347\)](#).
- [Validate Match \(p.348\)](#)
- [Determine Choke Settings \(p.349\)](#)

For this demo, we use the Data Matching Base model Liquid Loading which for a standard installation of PIPESIM is located in C:\Program Files\Schlumberger\PIPESIM\Case Studies \Tutorials\ directory.

2.6.1 Measured Data

Flowing gradient Survey @ Liquid Rate 15000:

S. No.	Depth (MD)	Pressure (Psia)	Temperature (F)
1	0	785	145
2	3000	1355	
3	5000	1838	
4	7500	2550	
5	10000	3186	
6	11600 (mid Perf.)	3600	

Static Pressure measured at mid-perforation is 4224 psia. Temperature = 210 F.

Flowline:

S. No.	Depth (MD)	Pressure (Psia)	Temperature (F)
1	0	785	
2	1250	775	
3	2500	760	

Riser:

S. No.	Depth (MD)	Pressure (Psia)	Temperature (F)
1	0	760	
2	80	750	143

Multi-Rate Test Data:

Test Point	Flow Rate (STB/Day)	Tubing Head Pressure (psia)	Water Cut (%)	GOR (scf/STB)
1	15000	750	0	892

Test Point	Flow Rate (STB/Day)	Tubing Head Pressure (psia)	Water Cut (%)	GOR (scf/STB)
2	12000	890	0	892
3	9000	1004	0	892
4	6000	1089	0	892

2.6.2 Model Setup

Do the following:

1. Open the Data Matching Base model.
2. Modify the flowline element by setting **Rate of Undulations** to 0.
3. Double click a **Tubing** object to open the **Tubing** dialog. Click **Convert to Detailed Model** and then select **Detailed Model** from the **Preferred Tubing Model** list and click **OK**. This changes the tubing profile to **Detailed** mode.

2.6.3 Task 1. IPR matching

IPR matching can be performed by performing simulation across completion using measured pressure data at mid-perforation. The objective is to look for the uncertain variable and tune it to get the best IPR match. The IPR model in this case is **Well PI** and, once the Static Reservoir Pressure is available, the only uncertain parameter is Productivity Index. You will use the [Pseudo General Iterative Method \(p.211\)](#) to get the correct value of PI.

Do the following:

1. Update Static Reservoir Pressure and Temperature in the model.
2. Deactivate physical objects downstream of the completion (in this case the tubing object) by selecting the equipment item(s), right-clicking and pressing **Active**. The tick next to the **Active** becomes unchecked and the deactivated equipment item changes from blue to red.
3. Select **Operations » Pressure/Temperature Profile** and enter the known values for **Inlet Pressure**, **Outlet Pressure** (set as measured pressure at mid-perforation at 3600 psia) and liquid rate (measured test rate at 15000 STB/day).
4. Set [User Variable \(p.211\)](#) as the calculated parameter and define **Productivity Index** to be the user variable.
5. Set a valid minimum and maximum range for **Productivity Index** (for this case we use the range from 1 STB/day/psi to 30 STB/day/psi). Set **Proportionality Relationship** to Direct.
6. Run the operation. (Be sure to remove the data set used for sensitivity analysis on Water Cut (these data were used in the standard P&T; model) and view the plot of pressure vs elevation. Note that the calculated liquid PI (expected value of PI in this case should be 24.05147 STB/Day/Psi).
7. Double-click the completion object and update the value of Productivity Index (PI).

2.6.4 Task 2. Optimizing Flow Correlation & Heat Transfer Coefficient

Do we need to optimize? Before proceeding with optimization, we would like to see how the simulated result compares to measured data. Re-activate all the components of the model and perform simulation (Pressure / Temperature profile) to calculate outlet pressure for a liquid rate of 15000 STB/day and make a note of Outlet pressure and temperature. In this case we obtain:

- Outlet pressure of 728 psia (this gives 22 psia lower than measured value of 750 psia, i.e. ~3% error)
- Outlet temperature of 154 F (again 11 F higher than the measured temperature of 145 F, i.e. 8% error).

To correct these errors, we need to tune the model using the optimization routine (read on). PIPESIM optimization routines allow the single branch engine to calculate optimal values of parameters to match measured pressure and/or temperature data.

You can perform any of the following:

- Pressure Match
- Temperature Match
- Simultaneous Match for Pressure and Temperature

The matching is performed by tuning parameters like friction and holdup factor for Pressure matching and U-factor for Temperature matching. See [optimize \(p.752\)](#) for further details.

Load Measured Data

Measured data must be set up to use the optimizer. Measured data for Flowline and Riser can be directly entered into their respective interfaces. Measured data can only be entered using a detailed profile for flowline and riser:

- Open Flowline/Riser interface and set **Preferred Pipe Description** to **Detailed View**.
- Enter **Measured Pressure** and **Measured Temperature** and click **OK**.

Enter measured data for tubing into the database, as follows:

1. Select **Data** » **Load/Add Measured Data**.
2. Select **New** and enter the **Well Name, Survey Date** and the measured data (Pressures and Temperatures at various measured depths).
3. Click **Save Changes**.

Setup Optimizer Options

Do the following:

1. Select **Operations** » **Data Matching** and click the **Parameters** tab. Ensure that the **Calibrate** check buttons are ticked for U-value multiplier, Vertical flow friction factor, and Vertical flow holdup factor. Enter the following Min and Max values:

	Min	Max
U-value multiplier	0.5	10

	Min	Max
Vertical flow friction factor	0.7	1.3
Vertical flow holdup factor	0.7	1.3
Horizontal flow friction factor	0.7	1.3
Horizontal flow holdup factor	0.7	1.3

2. On the **Flow Correlations** tab, select suitable horizontal and vertical flow correlations. For this case we will use following correlations:

Vertical Flow Correlations	Horizontal Flow Correlations
Ansari	Beggs & Brill Revised
Duns & Ros	Mukherjee & Brill
Hagerdorn & Brown	

3. On the **Run** tab, select **Outlet Pressure** as calculated variable (under **Operation Run Data**) and enter the given Flowrate (15000 STB/Day). Click **Run model**. (Set the RMS weight factors for the pressure and temperature to 1 to give equal weighting to P & T match).

Select flow correlation parameters and U factor

Do the following:

1. Review the table of results. It shows two cases for each combination of Horizontal and Vertical flow correlations, an initial and the optimized value. The u-value multiplier (U), vertical friction factor (VF), vertical holdup factor (VH), horizontal friction factor (VF) and horizontal holdup factor (VH) are all listed before (Initial) and after the optimization (Optimized) along with the root-mean-square errors in the pressure (P) and temperature (T) match and the total RMS error (Total).
2. We should be selecting the combination that gives the lowest total RMS error. In this example, we notice that the optimized combination of Hagedorn & Brown (Vertical Correlation) and Beggs & Brill Revised (Horizontal Correlation) is the best option, with the lowest RMS error.
3. Select Optimized HBR + BBR and click **Save Selected Results**.
4. Go back to the model and select **Setup** ➤ **Flow Correlations**. You will see that the optimized flow correlation results have been applied to the model. The friction factor and holdup factor have also been updated in the model.
5. Select **Setup** ➤ **Heat Transfer Options**. You will see that the optimized U-value multiplier has also been applied to the model. Now all u-values listed in the models (Tubing, Flowline and Riser) will be multiplied by the U-value multiplier.

The model is now ready for use.

2.6.5 Task 3. Validate Match

Once the model is tuned, validate it against test data measured at different conditions. We will use a Multi-rate test data for validation.

Do the following:

1. Select **Operations » Pressure/Temperature Profile** to calculate **Outlet Pressure** for known liquid rates. (Use **Liquid Rate** as the Sensitivity Variable).
2. Run the operation and review the result. Compare the simulated result with the measured data and see the difference:

Case No.	Flow rate (STB/Day)	THP - (psia) Measured	THP - (psia) Simulated	Difference (%)
1	6000	1089	1109	1.84
2	9000	1004	1021	1.69
3	12000	890	903	1.46
4	15000	750	751	0.13

This confirms a valid match.

2.6.6 Task 4. Determine Choke Settings

The well above will be connected to a common surface facility operating at 400 psia. To avoid a back-pressure effect on other wells connected to the system, the well must be choked back to give final delivery pressure of 400 psia, delivering a flowrate of 15000 STB/d. Do the following:

1. Modify the model to add a choke object at the wellhead. Set the bean diameter to any value (say 0.5 inches).
2. Perform a **Pressure/Temperature Profile** operation with known values for **Inlet Pressure**, **Outlet Pressure** (400 psia) and **liquid rate** (15000 STB/day).
3. Set **User Variable** as the calculated parameter and define **Choke Bean Size** as the user variable.
4. Set a valid minimum and maximum range for choke size (for this case we use the range from 0 in to 2 in). **Proportionality Relationship** should be set to **Direct**.
5. Run the model and view the results. Viewing the pressure versus elevation plot indicates the choke bean size as 1.6448 inches.

Set the new choke size and perform a **Pressure/Temperature Profile** operation with known values for Inlet Pressure, Outlet Pressure (400 psia) and request the liquid rate to be calculated. You will get the expected value of 15000 STB/d for liquid rate. The well is now ready to produce to the common facility without any back-pressure effect on other wells.

2.7 Liquid Loading Analysis Tutorial

Field data matching is an important exercise to build an accurate model. PIPESIM allows user to perform fluid properties matching, IPR matching, flow correlation matching, temperature matching, equipment settings, and so on. The objective of such matching is to eliminate uncertainties based on measured data, operating conditions and create a model that follows field measurement. This tutorial involves the following tasks:

- Set up Liquid Loading Model (p.350).
- Control Liquid Loading Options in PIPESIM (p.350).

- Analyze Well for Liquid Loading (p.350)
- Determine Critical Gas Rate to Prevent Well Loading (p.351)

2.7.1 Task 1. Set up the Liquid Loading Model

Liquid Loading is performed in PIPESIM by default. The default model is Turner's Model, but the user can select different models through engine keywords. See [Liquid Loading \(p.396\)](#) for further details. In this exercise we use the Coleman (1991) model.

Do the following:

1. Open the Liquid Loading Base model which for a standard installation of PIPESIM is located in C:\Program Files\Schlumberger\PIPESIM\Case Studies\Tutorials\ directory.
2. Set up the Coleman Liquid Loading Model and other parameters:
 - Pick the value of correction (Efficiency) factor for Coleman Liquid Loading Model from the [table \(p.396\)](#) (this value is 1.0).
 - Select **Setup » Engine Options** and, in the space provided for **Additional engine keywords**, enter the following:

```
OPTION ELILOADING = 1.0
```

This picks Coleman's model.

2.7.2 Task 2. Control Liquid Loading Options in PIPESIM

You have the option of controlling Liquid Loading Calculations for every liquid loading model selected. This is done using engine keywords. The parameters to control are:

- Implementation and reporting of [Liquid loading Velocity Ratio \(p.90\)](#).
- [Inclination angle \(p.90\)](#) to prevent liquid loading calculations.

The user should add the following engine keywords:

```
OPTION LLVELOCITY = GAS  
OPTION LLANGLE = 30
```

The first option reports Liquid Loading Velocity Ratio as Terminal Settling Velocity/Gas Velocity.

The second option prevents calculation of Liquid Loading for a Pipe angle up to 30 degrees from horizontal.

2.7.3 Task 3. Analyze Well for Liquid Loading

Liquid loading calculations are performed in every operation and are available for review through output and plot reports. Do the following:

1. Select **Pressure » Temperature Profile** and calculate the Gas Rate for an outlet pressure of 450 psia.
2. Review the output file and plots to see whether the well is under liquid loading. A **Liquid Loading Velocity Ratio** in excess of 1 indicates loading.

The result indicates the value for Liquid Loading Velocity Ratio much less than 1 at every node in tubing; this indicates the well is free from Liquid Loading. (The auxiliary report should show no calculation performed for the flowline section due to option control set for inclination angle.)

2.7.4 Task 4. Determine Critical Gas Rate to Prevent Well Loading

A Nodal Analysis plot reports Liquid Loading Gas Rate by default. For every point on the outflow curve, the value of **Liquid Loading Velocity Ratio** is calculated and the [critical gas rate \(p.396\)](#) is calculated at a point where **Liquid Loading Velocity Ratio** is 1. The reported value comes from interpolation of the outflow curve between two points, one with a velocity ratio below 1 and another with a velocity ratio above 1. **The accuracy of the result therefore depends on the number of points on the outflow curve.**

Do the following:

1. Add a Nodal Analysis Point between the tubing and completion.
2. Select **Operations » Nodal Analysis** and add the following settings:
 - Number of points on each inflow curve = 100
 - Number of points on each outflow curve = 200
 - Allow the inflow curves to extend to the AOFP
 - Limit the outflow curves to lie within the pressure range of the inflow curves
3. View the NA plot. Plot the Pressure at NA point versus Stock Tank Gas Rate. Make a note of the stock tank gas rate under the **Data** tab. The reported critical gas rate is 2.2584 mmscf/d
4. The reported critical gas rate refers to the outflow curve. This can be validated by performing **Operations » Pressure/Temperature Profile** at the same conditions (flowrate and outlet pressure). Perform the operation to calculate inlet pressure at the given critical gas rate (2.2584 mmscf/d) corresponding to outflow outlet pressure of 450 psia. Viewing the output file should reveal that the **Maximum Liquid Loading Velocity Ratio** is close to 1. This is consistent with the Nodal Analysis result.

2.8 Ramey Heat Transfer Model Tutorial

Heat transfer between a well and its surroundings varies with time - the well exchanges energy with the formation, heating it up (or cooling it down), until the formation is at the same temperature as the well. This tutorial helps user in implementing detailed steady state heat transfer in the wellbore using the [Ramey model \(p.81\)](#). This tutorial involves the following tasks:

- Set up Ramey Heat Transfer Properties ([p.351](#)).
- Set up Detailed Heat Transfer for the Tubing ([p.352](#)).
- Run Model and View Results ([p.354](#))

2.8.1 Task 1. Set up Ramey Heat Transfer Properties

Modeling wellbore heat transfer using the Ramey model requires setting up ground thermal properties and other parameters like time and heat transfer reference diameter. This can be done using engine keywords as follows:

1. Open the Wellbore Heat Transfer Base model which for a standard installation of PIPESIM is located in C:\Program Files\Schlumberger\PIPESIM\Case Studies\Tutorials\ directory.
2. Set up Ramey Heat Transfer Properties. Perform the following steps:
 - Double click the tubing to open the **Tubing editor**.
 - On the **Geothermal Survey** tab, select the **Calculate U Value** radio button, click **Heat Transfer Properties** and enter the following ground data:

Thermal Conductivity (BTU/hr/ft/F)	Specific Heat Capacity (BTU/lb/F)	Density lb/ft ³	Production/Injection Time (hrs)
2.2	0.24	126	500

- Select **Setup » Engine Options** and, in the **Additional engine keywords** box , enter the following:

```
HEAT HTCRD = 11.741
```

The second line refers to the time the well has been operating. Typically, with a value in excess of 168 hours (one week), the surrounding rock should reach steady state heat transfer.

2.8.2 Task 2. Set up Detailed Heat Transfer for the Tubing

The next step is to define detailed heat transfer properties for the tubing, including pipe and coatings information. Looking at a typical wellbore (Figure 1), it is apparent that the upper section (from the bottom of the tubing to the surface) has got several layers of materials separating the flowing wellbore fluid from the surrounding rock/ground. These include:

- Tubing material
- Annulus filled with completion fluid/gas
- Casing material

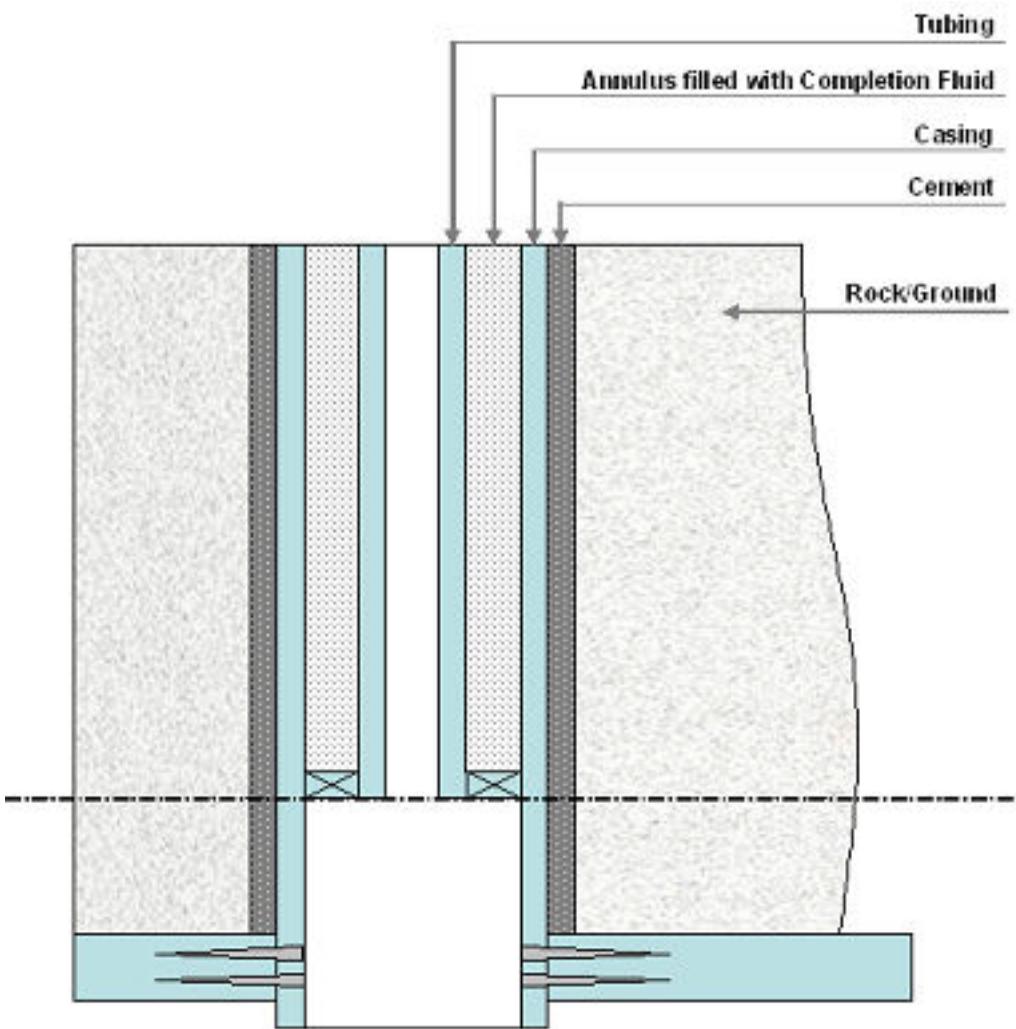


Figure 2.23. Typical Wellbore

Similarly, the lower section (from tubing shoe down to mid-perforation) has the following layers of material separating flowing fluid with the surrounding rock:

- Casing material
- Cement behind casing

These layers act as thermal coatings between the flowing fluid and surrounding rock. Enter the following information in the **Wellbore properties** section of the **Tubing Heat Transfer Properties** dialog:

Tubing Thermal Conductivity (BTU/hr/ft/F)	Completion Fluid Thermal Conductivity (BTU/hr/ft/F)	Casing Thermal Conductivity (BTU/hr/ft/F)	Casing Thickness (ft)	Cement Thermal Conductivity (BTU/hr/ft/F)	Cement Thickness (ft)
35	3.25	35	0.78	2	0.75

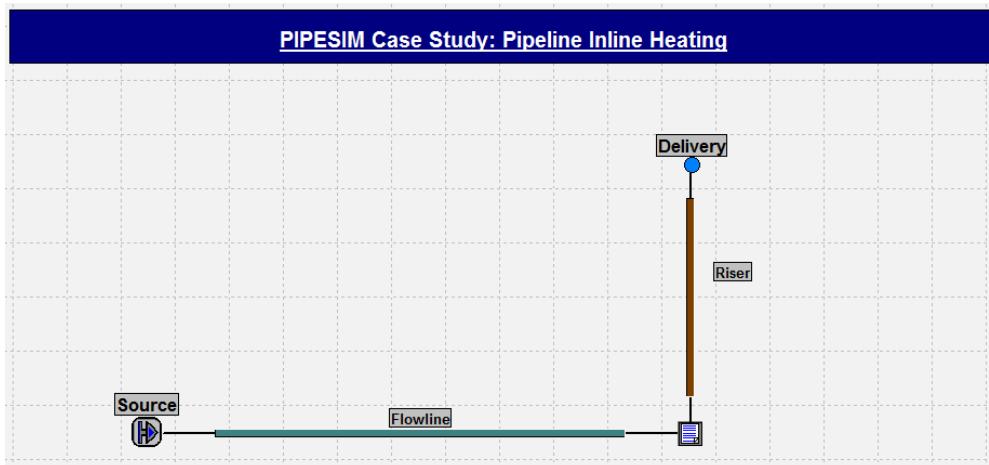
2.8.3 Task 3. Run the Model and View Results

Do the following:

1. Select **Setup** » **Define Output** and select **Heat Transfer Input Data** and **Heat Transfer Output Data**. Click **OK**.
2. Run the model and view the heat transfer input and output data in the output file.

2.9 Pipe Inline Heating

This tutorial will show you how to use the PIPESIM* electrical inline heating feature to mitigate hydrate formation for a 14-mile condensate transportation pipeline. The incoming fluid is at 4200 psia and 102 F and the pressure at the delivery point must be maintained at 1200 psia. Considering the cold climate and presence of water and light hydrocarbon in the fluid stream, hydrate formation is expected to occur in flowline and riser.



In the tutorial you will design and implement pipeline heating to prevent hydrate formation and determine the heating power required in order to maintain pipeline temperature above the hydrate formation temperature.

In this tutorial, you will perform the following tasks:

- [Build a Representative PIPESIM Model \(p.356\)](#)
- [Screen for Hydrate Formation \(p.359\)](#)
- [Prevent Hydrate Formation through Inline Heating \(p.362\)](#)
- [Analysis using ILH Keywords \(p.365\)](#)

2.9.1 Tutorial Data

Use the following data for the tutorial.

Source and Delivery Conditions	
Pressure at the Source	4200 psia
Temperature at the Source	102 deg.F

Source and Delivery Conditions	
Minimum pressure at the delivery	1200 psia

Flowline and Riser Data	
Flowline length	14 miles
Riser height	1000 ft
Inside diameter (flowline and riser)	4.026 inch
Wall thickness (flowline and riser)	0.237 inch
Pipe roughness	0.0018 inch

Heat Transfer Data (Flowline & Riser)	
Pipe Conductivity	28 Btu/hr/ft/F
Ambient fluid (air) velocity	0.6 ft/s
Ambient temperature	38 deg.F
Coating layer conductivity	0.6 Btu/hr/ft/F
Coating thickness	0.76 inch

Fluid Composition Data (mole %)	
Methane	42
Ethane	12
Propane	8
Butane	5
Isobutane	1
Pentane	2
Isopentane	1
Hexane	1
Water	12
C7+ (Petroleum fraction)	16

Petroleum Fraction Properties	
C7+ (Boiling point)	298 F
C7+ (Molecular weight)	115

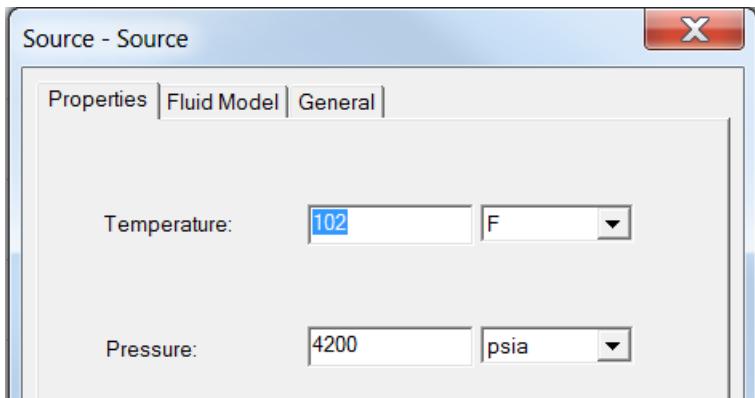
Note: Use Multiflash (PVT package) with Peng-Robinson (EoS), LBC (viscosity model) and OilGas4 (BIP set). Keep the Emulsion option as none.

2.9.2 Task 1: Build a Representative PIPESIM Model

In this task, you will use the model schematic and data provided to create a PIPESIM model.

Do the following:

1. Open a new PIPESIM single branch window. Add **Source**, **Junction**, **Report tool**, and connect them using flowline and riser to create a model as shown in the previous figure.
2. Specify the **Source** properties as shown in the following figure.



3. Specify the **Flowline** properties including the heat transfer data as shown in the following figures.

Flowline - Flowline

Properties	Heat Transfer	General
Preferred Pipe Description Simple View		
Rate of Undulations	0	/ 1000
Horizontal Distance	14	miles
Elevation Difference	0	ft
Inner Diameter	4.026	inches
Wall Thickness	0.237	inches
Roughness	0.0018	inches
Ambient Temperature	38	F

Flowline - Flowline

Properties	Heat Transfer	General																																	
Mode																																			
<input type="radio"/> Input U value	<input checked="" type="radio"/> Calculate U value																																		
Pipe Coating (starting from pipe surface)																																			
<table border="1"> <thead> <tr> <th>K</th> <th>Thickness</th> <th>Description</th> </tr> </thead> <tbody> <tr> <td>Btu/hr/ft/F</td> <td>inches</td> <td>(Optional)</td> </tr> <tr> <td>0.6</td> <td>0.76</td> <td>Coating</td> </tr> <tr> <td>2</td> <td></td> <td></td> </tr> <tr> <td>3</td> <td></td> <td></td> </tr> <tr> <td>4</td> <td></td> <td></td> </tr> <tr> <td>5</td> <td></td> <td></td> </tr> <tr> <td>6</td> <td></td> <td></td> </tr> <tr> <td>7</td> <td></td> <td></td> </tr> <tr> <td>8</td> <td></td> <td></td> </tr> <tr> <td>9</td> <td></td> <td></td> </tr> </tbody> </table>			K	Thickness	Description	Btu/hr/ft/F	inches	(Optional)	0.6	0.76	Coating	2			3			4			5			6			7			8			9		
K	Thickness	Description																																	
Btu/hr/ft/F	inches	(Optional)																																	
0.6	0.76	Coating																																	
2																																			
3																																			
4																																			
5																																			
6																																			
7																																			
8																																			
9																																			
Pipe Conductivity 28 Btu/hr/ft/F																																			
Ambient Fluid <input checked="" type="radio"/> Air <input type="radio"/> Water																																			
Velocity 0.6 ft/s																																			
Pipe Burial Data Burial Depth inches Ground 1.5 Btu/hr/ft/F Conductivity Elevated above ground																																			
Pipe Overall Outside Diameter 6.02 inches																																			

4. Specify **Riser** properties including heat transfer data as shown in the following figures.

Riser - Riser

Properties Heat Transfer General

Preferred Pipe Description Simple View

Horizontal Distance: 0 ft

Elevation Difference: 1000 ft

Inner Diameter: 4.026 inches

Wall Thickness: 0.237 inches

Roughness: 0.0018 inches

Ambient Temperature: 38 F

Riser - Riser

Properties Heat Transfer General

Mode: Input U value Calculate U value

Pipe Coating (starting from pipe surface)

	K Btu/hr/ft/ $^{\circ}$ F	Thickness inches	Description
-	0.6	0.76	(Optional)
1			Coating
2			
3			
4			
5			
6			
7			
8			
9			

Pipe Conductivity: 28 Btu/hr/ft/ $^{\circ}$ F

Ambient Fluid: Air Water

Velocity: 0.6 ft/s

Pipe Overall Outside Diameter: 6.02 inches

5. Go to **Setup** » **Flow Correlations** and select **Duns & Ros** (Vertical Multiphase Flow Correlation) and **Beggs & Brill Revised** (Horizontal Multiphase Flow correlation). Keep the single phase correlation as **Moody**.
6. Go to **Setup** » **Heat Transfer Options** tab and ensure **Hydrate Sub-cooling** is selected.

7. Create a custom report template to include additional variables related to inline heating and hydrate screening as shown in the following figures:
 - Go to **Setup** » **Compositional Template**, and select **Multiflash as PVT package**.
 - On the **Component Selection** tab, select and add all the database components.
 - On the **Petroleum Fractions** tab, define C7+ and click **Add to composition**.
 - On the **Property Models** tab, select to **Use template models** for all the fluid and specify **Peng-Robinson (EoS)**, **LBC (Viscosity model)**, **OilGas4 (BIP Set)**. Keep the Emulsion option as none.
 - Click **OK** to close the **Template** window.
 - Go to **Setup** » **Compositional** (local default) and specify mole(%) for all components.
 - Click **OK** to exit.
8. Save the model as `Pipe_Inline_Heating.bps`.

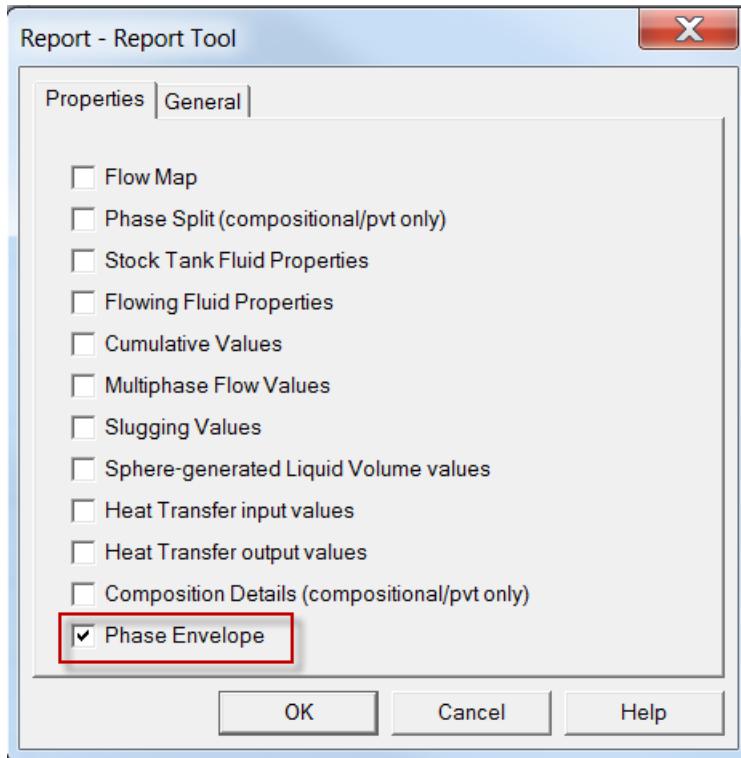
2.9.3 Task 2: Screen for Hydrate Formation

In this task you will screen for hydrate formation in the flowline and riser, and identify the location where hydrate is likely to form.

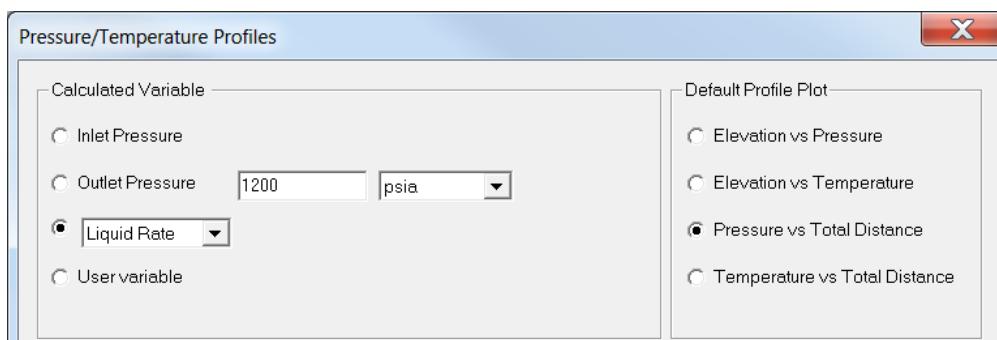
Do the following:

1. Open the **Report Tool** and ensure that **Phase Envelope** is selected.

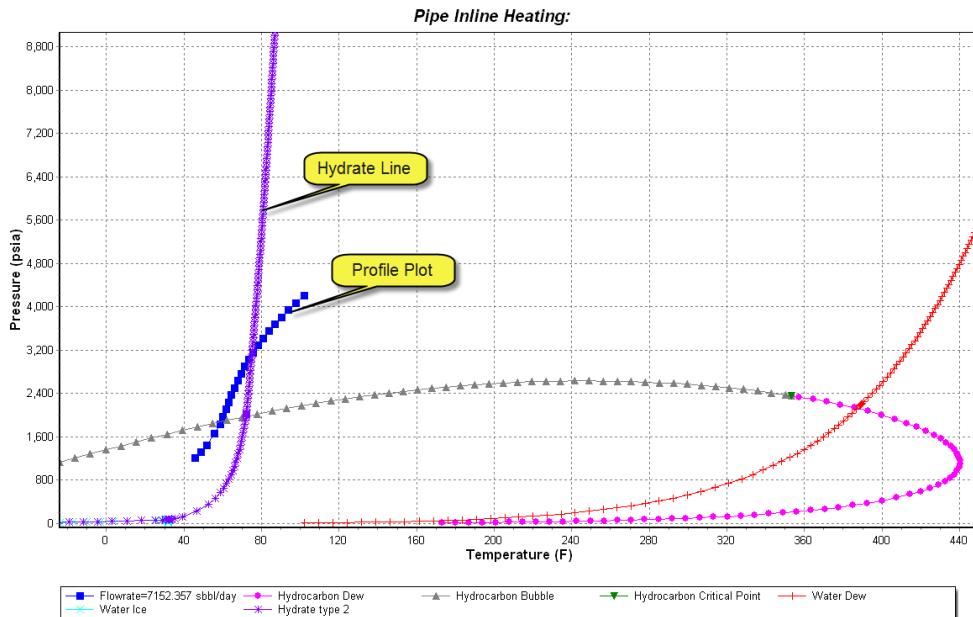
Select **Source**, configure the **Pressure Temperature Profile**, and select **Liquid Rate** as the **Calculated Variable** at a given outlet pressure of 1200 psia. Set the default plot to **Pressure vs. total distance**.



2. Go to **Operations** » **Pressure Temperature Profile**, and select **Liquid Rate** as the **Calculated Variable** at a given outlet pressure of 1200 psia. Set the default plot to **Pressure vs Total Distance**.

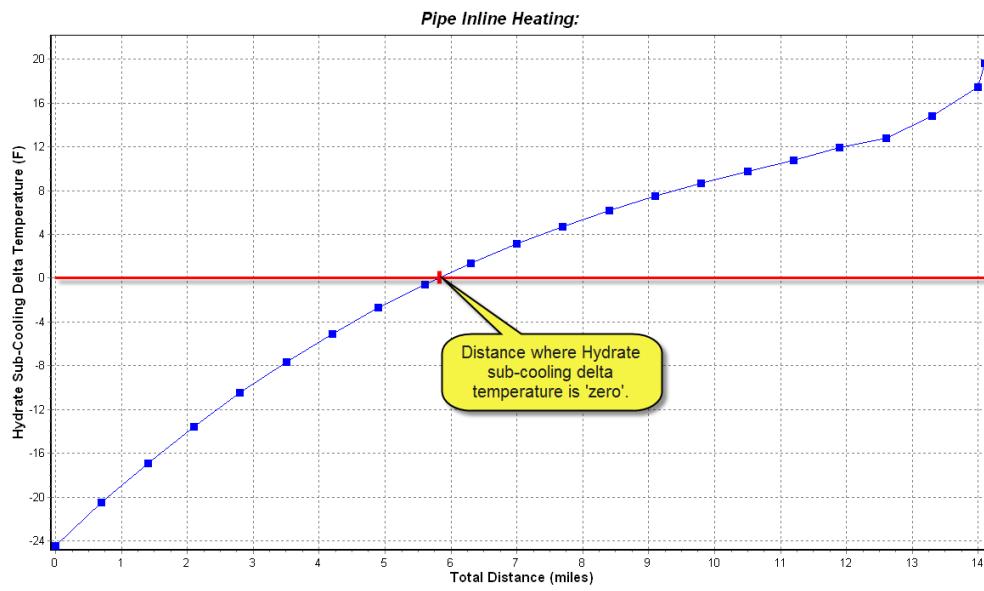


3. Run the model and observe the following result:
Stock Tank Liquid Flowrate = 7152.36 STB/day
4. Modify the Profile plot to generate a profile plot superimposed over phase envelope by modifying the plot to Temperature (X-axis) vs Pressure (Y-axis). You can see that there is hydrate formation in the system identified as **Hydrate 2** in the following figure.



Note: Part or whole of the profile plot falling to the left of hydrate line indicates hydrate formation in the system.

- Identify the location of hydrate formation by generating a plot of Total distance (X-axis) vs Hydrate subcooling delta Temperature (Y-axis). You can see in the following figure that the hydrate formation starts 6 miles from the source in the flowline.



Note: A positive hydrate sub-cooling delta temperature indicates that the fluid temperature is below the hydrate formation temperature.

6. Identify the minimum temperature that must be maintained in the system to prevent hydrate formation by doing the following:
 - Generate a plot of Total distance (X-axis) vs Temperature (Y-axis).
 - Add existing Hydrate Structure (Type 2 in this case) to the left (Y-axis).
 - Get the intersection point temperature and add some safety margin.

2.9.4 Task 3: Prevent Hydrate Formation through Inline Heating

In this task you will apply inline heating to maintain a higher system temperature (above hydrate formation temperature) to prevent hydrate formation.

Inline heating can be applied using engine keywords under **Setup » Engine Options**. The inline heating goes under **PIPE** main code, and there are two sub codes described in the following table:

Sub Code	Description
ILH=or ILHPOWER=or ILHMAXPOWER	This sub code allows a fixed heating power per unit length of pipeline (BTU/hr/ft or Kw/m). This will result in higher fluid temperature.
In-Line Heater	This sub code allows a fixed power or duty to be specified, that is used to transfer heat to the fluid flowing in the pipe. The value is interpreted as power per unit length (BTU/hr/ft or Kw/m).
ILHMINTEMP	This sub code allows maintaining a fixed minimum temperature across the pipeline by assigning required variable heating power per unit length of pipe (BTU/hr/ft or Kw/m).
ILHMINTEMP and ILH (or ILHPOWER, or ILHMAXPOWER) are used together	When both these sub codes are used, the supplied power is treated as the maximum limit. Specified minimum temperature is maintained along the pipeline as long as required power does not exceed available maximum power. When required temperature cannot be maintained without exceeding the power limit, the available power will be used as fixed power and resultant temperature will be calculated.

In the previous task you determined that the required minimum temperature is 80 deg. F. You will use the ILHMINTEMP sub code for this task.

Do the following:

1. Go to **Setup » Engine Options** enter the following single branch keywords:

PIPE ILHMINTEMP = 80

2. Configure the model to generate additional plots and results for inline heating related parameters (they are not reported by default) by using following additional keywords.

PLOT PROFILE =+Q27R27S27 (adds additional plot variables)

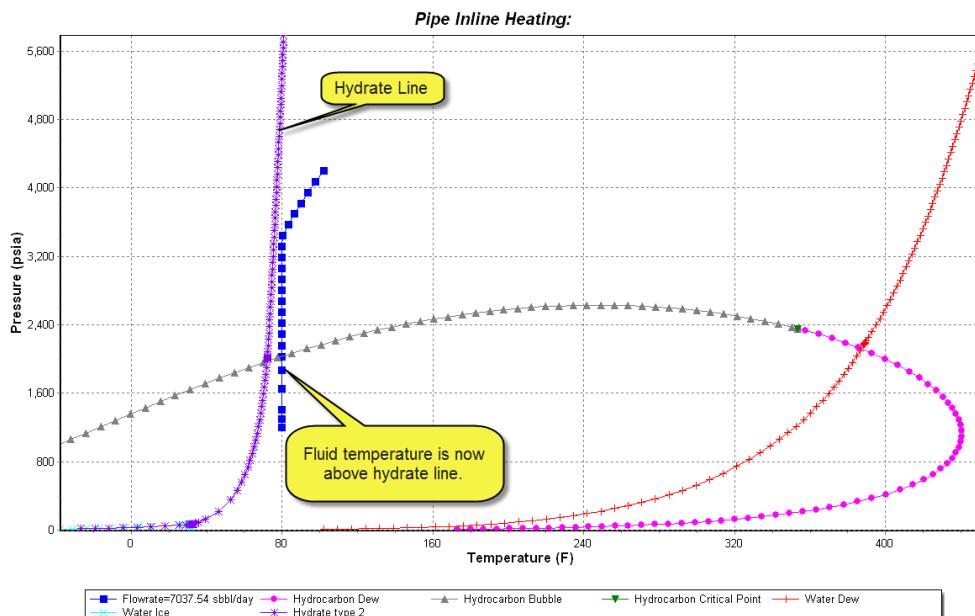
PRINT CUSTOM =(B,E,Q27,R27,S27) (generates custom page in the output)

Variable Code	Variable Description
B	Total distance

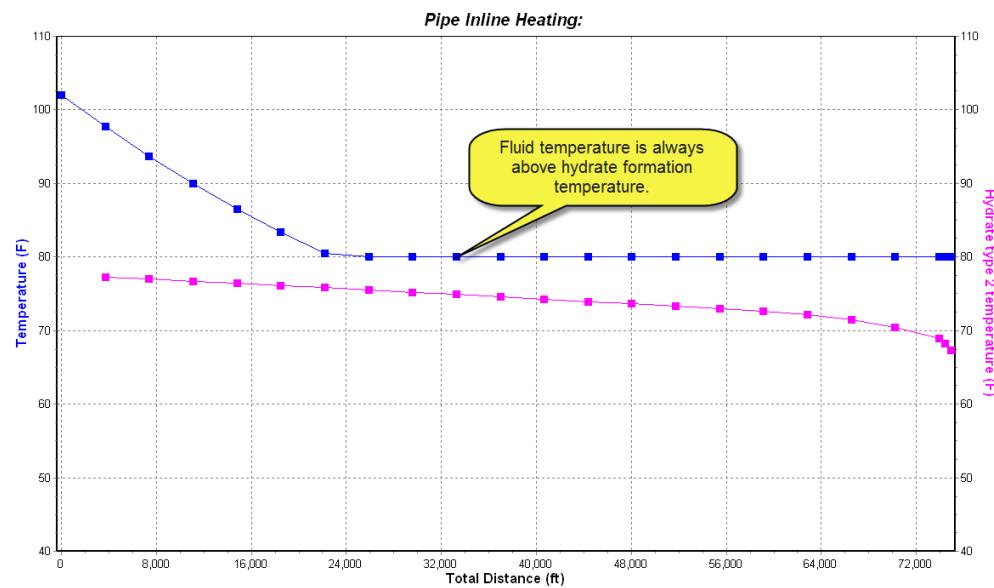
Variable Code	Variable Description
E	Temperature
Q27	Inline heater power used
R27	Inline heater power available
S27	Inline heater minimum temperature

3. Re-run the **Pressure Temperature Profile** and generate a **Profile plot** superimposed over **Phase envelope**.

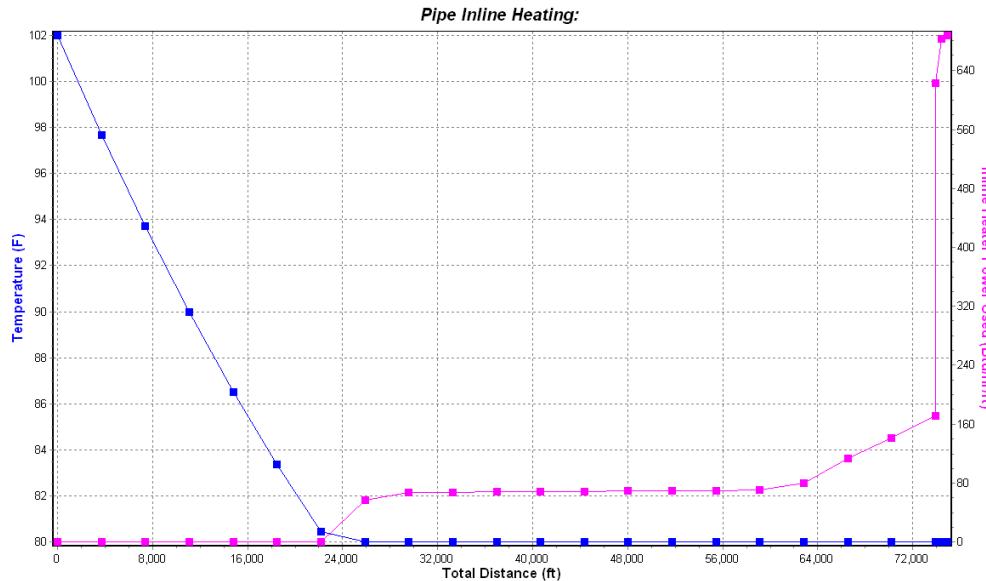
In the following figure you can see that the hydrate issue has been resolved.



4. Re-generate a plot of fluid temperature and hydrate temperature vs total distance to confirm the result.



5. Estimate maximum inline heating power used to prevent hydrate formation.



Horizontal Distance (ft)	Total Distance (ft)	Elevation (ft)	Pressure (psia)	Temp. (F)	Inline Heater Power Used (Btu/hr/ft)	Inline Heater Available (Btu/hr/ft)	Inline Heater Min Temp. (F)
Flowline							
1 0.	0.	0.	4200.000	102.0000	0.	n/a	80.00000
2 3696.000	3696.000	0.	4073.421	97.67924	0.	n/a	80.00000
3 7392.000	7392.000	0.	3946.962	93.70575	0.	n/a	80.00000
4 11088.00	11088.00	0.	3820.564	89.98012	0.	n/a	80.00000
5 14784.00	14784.00	0.	3694.207	86.51810	0.	n/a	80.00000
6 18480.00	18480.00	0.	3567.874	83.36144	0.	n/a	80.00000
7 22176.00	22176.00	0.	3441.541	80.45299	0.	n/a	80.00000
8 25872.00	25872.00	0.	3315.022	80.00000	56.23746	n/a	80.00000
9 29568.00	29568.00	0.	3188.142	80.00000	66.66880	n/a	80.00000
10 33264.00	33264.00	0.	3060.868	80.00000	66.79959	n/a	80.00000
11 36960.00	36960.00	0.	2933.166	80.00000	67.83301	n/a	80.00000
12 40656.00	40656.00	0.	2805.014	80.00000	67.79934	n/a	80.00000
13 44352.00	44352.00	0.	2676.405	80.00000	67.92745	n/a	80.00000
14 48048.00	48048.00	0.	2547.304	80.00000	69.06995	n/a	80.00000
15 51744.00	51744.00	0.	2417.682	80.00000	69.03641	n/a	80.00000
16 55440.00	55440.00	0.	2287.530	80.00000	69.74672	n/a	80.00000
17 59136.00	59136.00	0.	2156.789	80.00000	70.41646	n/a	80.00000
18 62832.00	62832.00	0.	2025.388	80.00000	79.65849	n/a	80.00000
19 66528.00	66528.00	0.	1863.714	80.00000	112.8574	n/a	80.00000
20 70224.00	70224.00	0.	1651.655	80.00050	140.7690	n/a	80.00000
21 73920.00	73920.00	0.	1409.272	80.00021	171.5430	n/a	80.00000
Riser							
Riser Base							
22 73920.00	73920.00	0.	1409.272	80.00021	623.4984	n/a	80.00000
23 73920.00	74420.00	500.0000	1302.420	80.00000	682.9677	n/a	80.00000
24 73920.00	74920.00	1000.000	1200.226	80.00000	687.8430	n/a	80.00000
Topsides							

2.9.5 Task 4: Sensitize ILH Keywords

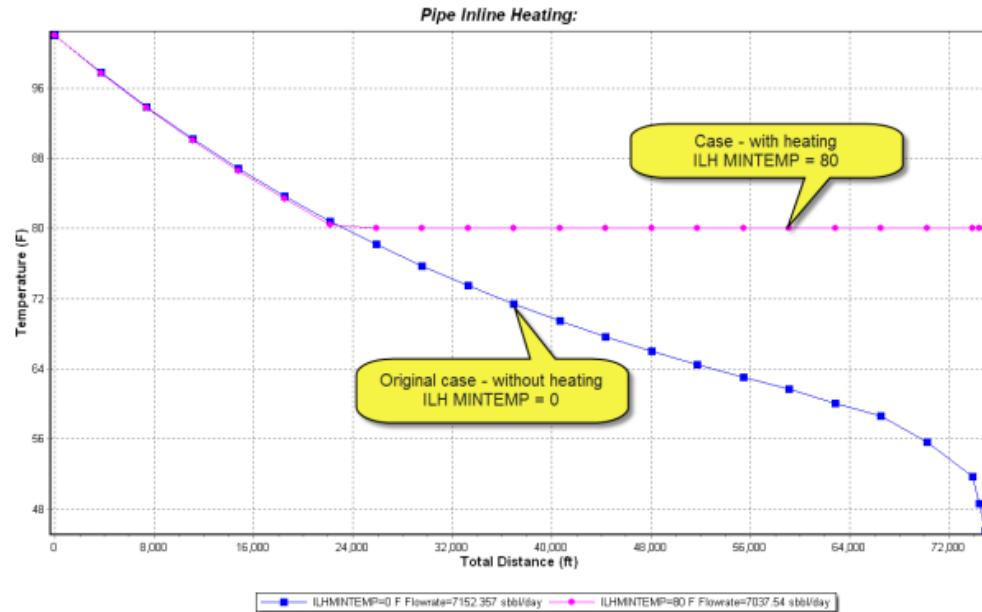
This task will demonstrate the use of inline heating keywords for ILH parameter sensitizing.

Sensitizing on Minimum Temperature

Re-run the P-T Profile using following ILH keywords to perform sensitivity on minimum temperature:

Multicase ?beta = (0,80)

PIPE ILHMINTEMP = ?beta

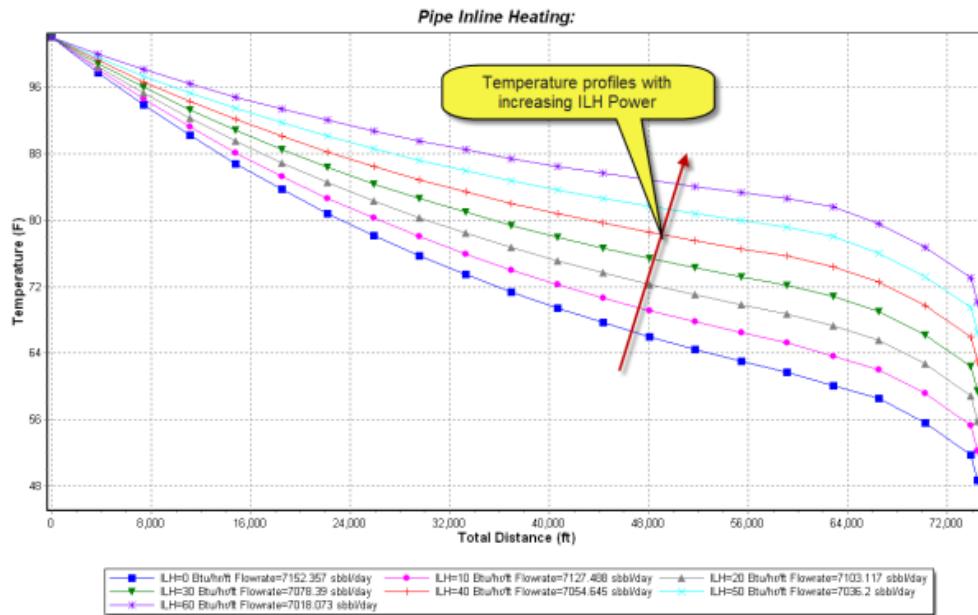


Analysis using ILH Keywords

Re-run the P-T Profile using following ILH keywords to perform sensitivity on heating power:

Multicase ?beta = (0,10,20,30,40,50)

PIPE ILH = ?beta



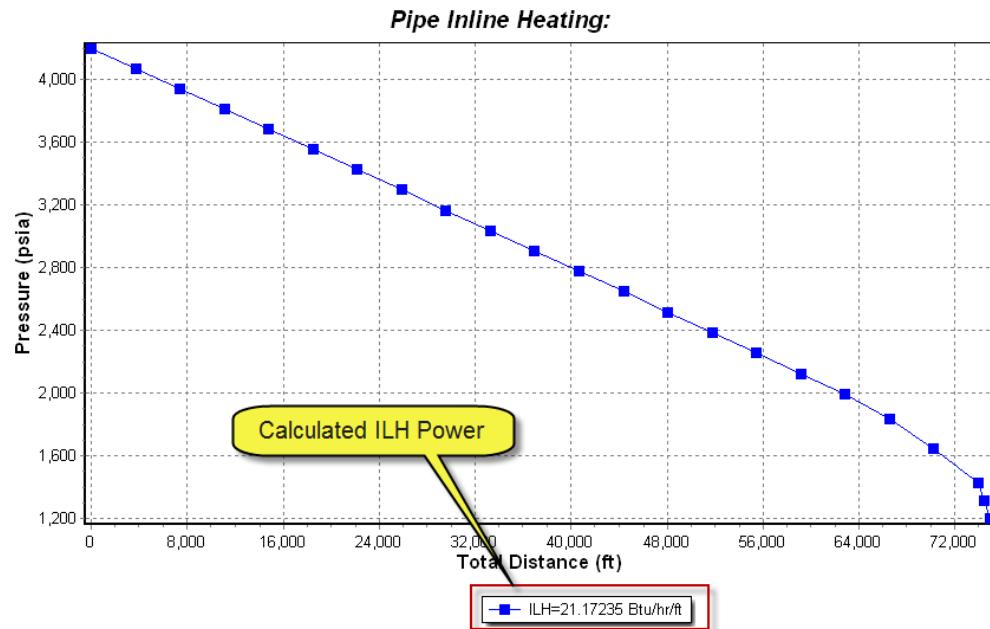
ILH Power as Calculated Variable

Variation in ILH power results changes in temperature and pressure profile of the system and thus impacts the calculated flowrate. Using pseudo iterative method, you can estimate ILH power for a given flowrate and boundary condition doing the following:

1. Modify the **P-T profile** operation to calculate ILH power required to achieve a target liquid flowrate of 7000 STB/day for given inlet and outlet condition. To do this, use following keywords:

ILH Power as Calculated Variable	
PLOT casedata jobdata xycase = dc	Sets plot parameter
RATE liq = 7000 stb/day	Sets liquid rate
ITERN type = PGEN- pout = 1200 psia xmin=0 xmax=50	Sets P-out and ILH power range
push mainc=pipe label='flowline' text='ilh = ?xiter'	Sets ILH as calculated variable

2. Rerun the **P-T Profile** and you will see that the ILP power is calculated as 21.17 Btu/hr/ft.



3

Support and Contact Information

3.1 Support

Schlumberger Information Solutions (SIS) provides a variety of options for receiving support:

3.1.1 SIS web support

If you have a problem that cannot be resolved using PIPESIM Help, you can send a support request via the Schlumberger Information Solutions Support [Web page](#). The Support Portal provides a single, online location for all your support needs. Within the Support Portal you can quickly search a vast knowledge base for the answers you need, participate with your peers in discussion forums, and receive the latest news about SIS products and services.

All support requests are entered into the SIS Customer Care Center incident tracking system, where they are resolved by local support staff. For those times when you need to speak with a support specialist, contact numbers are provided for your local support center.

3.1.2 On-site support

Schlumberger Information Solutions (SIS) supplements the standard maintenance agreement by offering extended on-site support worldwide. This enhanced level of support includes on-site assistance, installation, troubleshooting and maintenance services of licensed SIS software. Contact the help desk or refer to your licensing agreement for more information.

3.1.3 SIS Education

A large number of training courses are offered at various SIS locations, or on-site for groups or individuals upon request. These training courses can help you increase the value you are receiving from your SIS software products. The courses can also act as a form of preventive maintenance as you learn to achieve smoother and more trouble-free performance with the products. See the [SIS Training Web site](#) for more information.

4

Technical Description

This section of the User Guide provides additional details and references on the theory and methods implemented in PIPESIM.

- [Flow Models \(p.370\)](#)
- [Completion \(IPR\) Models \(p.398\)](#)
- [Equipment \(p.443\)](#)
- [Heat Transfer Models \(p.486\)](#)
- [Fluids Models \(p.504\)](#)

4.1 Flow Models

4.1.1 Flow Regimes

Flow Regimes Classification for Vertical Two Phase Flow

The general problem of predicting the pressure drop for the simultaneous flow of gas and liquid is complex.

The problem consists of being able to predict the variation of pressure with distance along the length of the flow path for known conditions of flow. Multiphase vertical flow can be categorized into four different flow patterns or flow regimes, consisting of bubble flow, slug flow, slug-mist transition (churn) flow and mist flow.

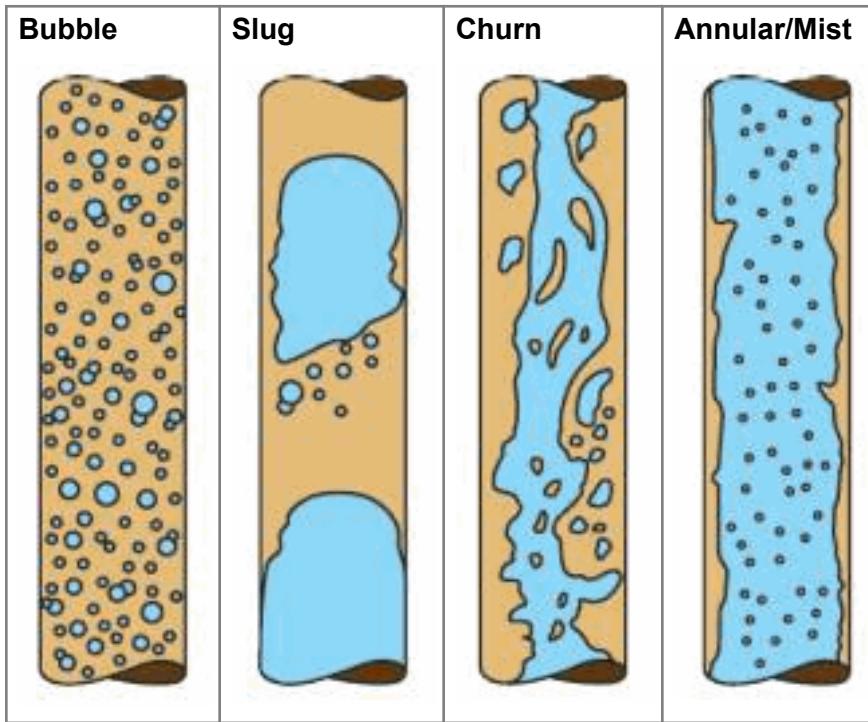
A typical example of bubble flow is the liberation of solution gas from an undersaturated oil at and above the point in the flow path where its bubble point pressure is reached.

In slug flow, both the gas and liquid phases significantly contribute to the pressure gradient. the gas phase exists as large bubbles almost filling the pipe and separated by slugs of liquid. In transition flow, the liquid slugs between the gas bubbles essentially disappear, and at some point the liquid phases becomes discontinuous and the phase becomes continuous.

The pressure losses in transition (churn) flow are partly a result of the liquid phase, but are more the result of the gas phase. Mist flow is characterized by a continuous gas phase with liquid

occurring as entrained droplets in the gas stream and as a liquid film wetting the pipe walls. A typical example of mist flow is the flow of gas and condensate in a gas condensate well.

PIPESIM Vertical Flow correlations

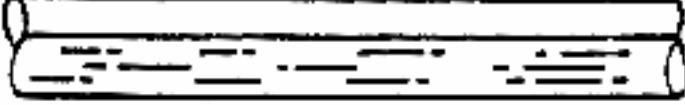
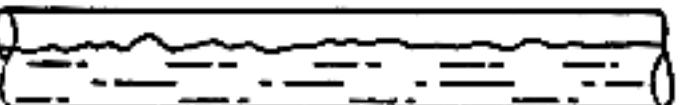


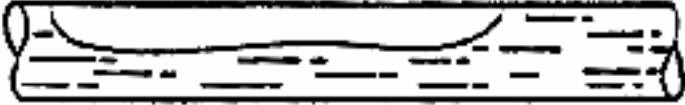
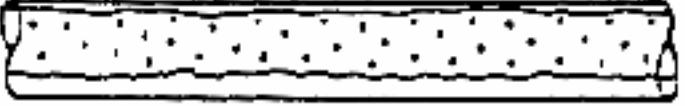
Flow Regimes Classification for Horizontal Two Phase Flow

Prediction of liquid holdup is less critical for pressure loss calculations in horizontal flow than for inclined or vertical flow, although several correlations will require a holdup value for calculating the density terms used in the friction and acceleration pressure drop components. The acceleration pressure drop is usually minor and is often ignored in design calculations; however, PIPESIM includes them.

As in the vertical flow, the two-phase horizontal flow can be divided into the following flow regimes: Stratified Flow (smooth, wavy), Intermittent Flow (plug and slug) and Distributed Flow (bubble and mist).

PIPESIM Horizontal Flow correlations

Stratified Flow	Smooth	
	Wavy	

Intermittent Flow	Slug	
	Elongated bubble/Plug	
Distributed	Annular/Mist and Bubble	
	Dispersed Bubble	

See also: [Flow regimes \(p.370\)](#),

4.1.2 Horizontal Multiphase Flow Correlations

The following horizontal multiphase flow correlations are available:

Baker Jardine (BJA) Correlation

[Baker Jardine \(p.583\)](#) (now Schlumberger) has developed a correlation for two phase flow in gas-condensate pipelines. This model represents no major advance in theory, but rather a consolidation of various existing mechanistic models, combined with a modest amount of theoretical development and field data testing. The model uses the Taitel Dukler flow regime map and a modified set of the Taitel Dukler momentum balance to predict liquid holdup. The pressure loss calculation procedure is similar in approach to that proposed by Oliemans, but accounts for the increased interfacial shear resulting from the liquid surface roughness. The BJA correlation is used for pressure loss and holdup with flow regime determined by the Taitel Dukler correlation. The BJA correlation has been developed specifically for applications involving low liquid/gas ratios, for example gas/condensate pipelines with a no-slip liquid volume fraction of lower than 0.1.

Beggs and Brill Original

ORIGINAL: The original [Beggs and Brill \(p.583\)](#) correlation is used for pressure loss and liquid holdup. Flow regime is determined by either the Beggs and Brill or Taitel Dukler correlation. The Beggs and Brill correlation was developed following a study of two-phase flow in horizontal and inclined pipes. The correlation is based upon a flow regime map which is first determined as if the flow was horizontal. A horizontal holdup is then calculated by correlations, and this holdup is corrected for the angle of inclination. The test system included two 90 ft long acrylic pipes, winched to a variable elevation in the middle, so as to model incline flow both upwards and downwards at angles of up to 90°.

Beggs and Brill Revised

REVISED: As above except that the revised version of the Beggs and Brill correlation is used, with rough pipe friction factors, holdup limits and corrective constants as proposed by [Palmer \(p.591\)](#) and [Payne \(p.591\)](#). The following enhancements to the original method are used; (1) an extra flow regime of froth flow is considered which assumes a no-slip holdup, (2) the friction factor is changed from the standard smooth pipe model, to utilize a single phase friction factor based on the average fluid velocity.

Dukler (AGA) and Flanigan

1. The AGA and Flanigan correlation was developed for horizontal and inclined two phase flow of gas-condensate gathering systems. The Taitel Dukler flow regime map is used which considers five flow regimes, stratified smooth, stratified wavy, intermittent, annular dispersed liquid, and dispersed bubble. The [Dukler \(p.585\)](#) equation is used to calculate the frictional pressure loss and holdup, and the [Flanigan \(p.586\)](#) equation is used to calculate the elevational pressure differential.
2. As above but with liquid holdup calculated according to the [Eaton \(p.586\)](#) correlation. The Eaton liquid holdup correlation is based on a study performed on 2 in. and 4 in. steel pipe using water and natural gas as test fluids. Test pressures ranged from 305 to 865 psia and liquid holdup measurements ranged from .006 - 0.732.

Eaton-Oliemans

The Eaton, Oliemans combination of methods uses the correlation developed by Eaton et al (1967) to predict liquid holdup and the Oliemans Pressure Drop Calculations correlation (1976) to predict frictional pressure losses. This set of correlations has been found to be reliable for gas-condensate systems in which the liquid loading varies from very small amounts to levels high above that which is normally found in gas gathering systems. Additionally, while the Eaton method tends to over-predict liquid holdup, the results for crude oil systems are generally reasonable. Note that since the Eaton et al correlation does not incorporate elevation change in its computation of liquid holdup, hydrostatic pressure losses can be significantly underestimated in cases of low flow rates over hilly terrain.

The Oliemans correlation was developed following the study of large diameter condensate pipelines. The flow regime is predicted using the Taitel Dukler flow regime map, and a simple model, which obeyed the correct single phase flow limits was introduced to predict the pressure drop. The model was based on a limited amount of data from a 30-in, 100-km pipeline operating at pressures of 100 barg or higher. The Oliemans pressure loss correlation can be used with the Eaton, BJA1, BJA2, BRIMIN1 or BRIMIN2 holdup correlations.

Hughmark-Dukler

The [Hughmark \(1962\) \(p.587\)](#) / [Dukler et al \(1964\) \(p.585\)](#) method is the procedure that was recommended by the AGA /API (1970). This approach uses the Dukler model for pressure loss calculations and the Hughmark model for liquid holdup calculations.

The use of the Hughmark (1962) liquid volume fraction correlation for pipelines is somewhat anomalous since it was originally based solely on data for flow in vertical pipes. Hughmark did however, compare its predictions with some limited data from horizontal pipes and found the agreement to be reasonable. Since then, a number of studies (Dukler et al, 1964; Mandhane et al,

1975; Gregory, 1975; Gregory and Fogarasi, 1985) have confirmed it to be one of the better correlations for pipeline applications.

It can generally be expected to give reasonable pressure drop and liquid holdup results for gas-crude oil pipelines. This procedure is not recommended however for gas-condensate systems, where the Hughmark correlation generally predicts excessive liquid holdups; errors of up to 600% have been observed.

LEDA

The Leda Point Model (PM) (<http://www.kongsberg.com/ledaflow>) is a mechanistic model applicable for all inclination angles, pipe diameters and fluid properties. The 2-phase model considers gas-liquid flow whereas the 3-phase model considers gas-oil-water flow.

The 3-phase Leda PM considers 9 fields in the mass (continuity) equations (oil, gas, water, oil in gas and water, gas in oil and water, water in oil and gas). Separate momentum equations are solved for oil, gas and water.

The 2-phase Leda PM considers 4 fields in the mass (continuity) equations (liquid, gas, liquid in gas and gas in liquid). Separate momentum equations are solved for gas and liquid phases. The flow regimes predicted by LedaPM are stratified smooth flow, stratified wavy flow, slug flow, annular and bubbly flow. The Leda 2-phase model uses the liquid viscosity associated with the fluid model defined in PIPESIM. The Leda 3-phase model assumes that the liquid viscosity is equal to that of the continuous phase; liquid viscosity options defined with the PIPESIM fluid model are ignored. The continuous phase is determined by the [Brauner-Ullman \(p.584\)](#) inversion criteria.

The Leda Point Model is the steady-state version of the transient model developed by SINTEF in collaboration with Total and ConocoPhillips and commercialized by Kongsberg. The model has been calibrated against data collected at the SINTEF Multiphase Flow Laboratory near Trondheim Norway. Over 10,000 experimental data points have been collected for single-phase, two-phase (oil-water, water-gas) and three-phase (oil-water-gas) flow. Pipe diameters ranging from 4-12" were used at pressures up to 90 barg. The models have been validated with field data supplied by ConocoPhillips and Total.

Minami and Brill

The Minami and Brill correlation calculates liquid holdup though does not predict flow regime or pressure gradient. The experimental holdup data was obtained by passing spheres through a 1,333 ft long 3" steel horizontal pipe and measuring the liquid volumes removed. Holdup measurements ranged from .001 to .44. Fluids used in the experiment included air, kerosene and water with the liquid viscosities ranging from .6 cp to 2 cp.

Two correlations were proposed. The first (BRIMIN1) is valid for all ranges of liquid holdup; the second (BRIMIN2) is strictly for wet gas pipelines (holdup < .35).

The [Minami and Brill \(p.590\)](#) holdup correlations can be used with any correlation except Mukherjee and Brill and No Slip. To activate the Minami and Brill correlation, enter the appropriate engine keyword under **Setup » Engine Options** (for example, hcorr holdup = brimin1)

Mukherjee and Brill

The [Mukherjee and Brill \(p.590\)](#) correlation is used for Pressure loss, Holdup and Flow Map. Note: selection of alternative flow maps and/or holdups will cause unpredictable results. The Mukherjee

and Brill correlation was developed following a study of pressure drop behavior in two-phase inclined flow. The test facility consisted of a U-Shaped pipe that was inclinable +/-90°. Each leg of the U section was 56 ft with 22 ft entrance lengths and a 32 ft test sections on both sides. Fluids were air, kerosene and lube oil with liquid viscosities ranging from .9 to 75 cp. Approximately 1000 pressure drop measurements and 1500 liquid holdup measurements were obtained from a broad range of oil and gas flows.

For bubble and slug flow, a no-slip friction factor calculated from the Moody diagram was found adequate for friction head loss calculations. In downhill stratified flow, the friction pressure gradient is calculated based on a momentum balance equation for either phase assuming a smooth gas-liquid interface. For annular-mist flow, a friction factor correlation was presented that is a function of holdup ratio and no-slip Moody friction factor. Results agreed well with the experimental data and correlations were further verified with Prudhoe Bay and North Sea data.

NOSLIP Correlation

The NOSLIP correlation assumes homogeneous flow with no slip between the phases. Fluid properties are taken as the average of the gas and liquid phases and friction factors are calculated using the single phase MOODY correlation.

OLGAS 2-phase / OLGAS 2000 3-phase

The OLGAS mechanistic models are applicable for all inclination angles, pipe diameters and fluid properties. The 2-phase [Bendiksen \(p.583\)](#) model considers gas-liquid flow, whereas the 3-phase model considers gas-oil-water flow.

This model employs separate continuity equations for gas, liquid bulk and liquid droplets, which are coupled through interphase mass transfer. Two momentum equations are solved: one applied to the combined balance for the gas and liquid droplets, if present, and a separate momentum equation for the liquid film. OLGAS considers four flow regimes: stratified, annular, slug and dispersed bubble flow; and uses a unique minimum slip criteria to predict flow regime transitions.

The OLGA 2-Phase model uses the liquid viscosity model defined within the PIPESIM fluid property definition. The 3-Phase model uses the Pal and Rhodes emulsion correlation to calculate liquid viscosity based on the oil and water viscosities defined with the PIPESIM fluid model definition; liquid viscosity options defined with the PIPESIM fluid model are ignored.

OLGAS is based in large part on data from the SINTEF multiphase flow laboratory near Trondheim, Norway. The test facilities were designed to operate at conditions that approximated field conditions. The test loop is 800 m long and 8 inches in diameter. Operating pressures between 20 and 90 barg were studied. Gas superficial velocities of up to 13 m/s, and liquid superficial velocities of up to 4 m/s were obtained. In order to simulate the range of viscosities and surface tensions experienced in field applications, different hydrocarbon liquids were used (naphtha, diesel, and lube oil). Nitrogen was used as the gas. Pipeline inclination angles between 1° were studied in addition to flow up or down a hill section ahead of a 50m high vertical riser. Over 10,000 experiments were run on this test loop during an eight year period. The facility was run in both steady state and transient modes.

Oliemans

The Oliemans correlation was developed following the study of large diameter condensate pipelines. The flow regime is predicted using the Taitel Dukler flow regime map, and a simple

model, which obeyed the correct single phase flow limits was introduced to predict the pressure drop. The model was based on a limited amount of data from a 30-in, 100-km pipeline operating at pressures of 100 barg or higher. The [Oliemans](#) ([p.590](#)) pressure loss correlation can be used with the Eaton, BJA1, BJA2, BRIMIN1 or BRIMIN2 holdup correlations.

TUFFP Unified Mechanistic Model (2-phase and 3-phase)

The TUFFP Unified Mechanistic Model is the collective result of many research projects performed by the Tulsa University Fluid Flow Projects (TUFFP) research consortium. The model determines flow pattern transitions, pressure gradient, liquid holdup and slug characteristics. A 2-phase version is available for gas-liquid flow [[Zhang et.al, development](#) ([p.594](#)) and [validation](#) ([p.594](#))] and a 3-phase version is available for gas-oil-water pipe flow [[Zhang and Sarica](#) ([p.594](#))]. The model is valid for all inclination angles, pipe diameters and fluid properties.

The principle concept underlying the model is the premise that slug flow shares transition boundaries with all the other flow patterns. The flow pattern transition from slug flow to stratified and/or annular flow is predicted by solving the momentum equations for slug flow. The entire film zone is treated as the control volume and the momentum exchange between the slug body and the film zone is introduced into the combined momentum equation. This approach differs from traditional methods of using separate models for each transition. The advantage of a single hydrodynamic model is that the flow pattern transitions, slug characteristics, liquid holdup and pressure gradient are implicitly related.

The 3-phase model contains separate momentum balances for the gas, oil and water phases. The model determines whether the oil and water phases are separated or fully mixed. If the phases are separated, individual phase viscosities are used. If the phases are fully mixed, the liquid viscosity can be determined either by the method within the TUFFP model (emul default option) or overridden (emul override option) by the liquid viscosity method defined with the PIPESIM fluid model, which is useful when rheology data are available. In the latter case, for black oil fluid models, selecting the Brinkman emulsion viscosity method with the Brauner-Ullman watercut cutoff method will replicate the method used within the TUFFP model. For the 2-phase (gas-liquid) model, the liquid viscosity from PIPESIM is always used, so the emulsion options defined in the PIPESIM fluid definition always apply.

The closure relationships included in the model are based on focused experimental research programs at University of Tulsa and elsewhere. As new and improved closure relationships become available, the TUFFP Unified Model is updated and validated.

Note: The TUFFP Unified 2-Phase Model v 2007.1 is no longer supported in PIPESIM. Upon import, TUFFPU2P is used instead.

Xiao

The Xiao comprehensive mechanistic model was developed as part of the TUFFP research program. It was developed for gas-liquid two-phase flow in horizontal and near horizontal pipelines. The model first predicts the existing flow pattern, and then calculates flow characteristics, primarily liquid holdup and pressure drop, for the stratified, intermittent, annular, or dispersed bubble flow patterns. The model was tested against a pipeline data bank. The data bank included large diameter field data culled from the AGA multiphase pipeline data bank, and laboratory data published in literature. Data included both black oil and compositional fluid systems. A new

correlation was proposed which predicts the internal friction factor under stratified flow. The former has the advantage of making the film friction sensitive to both the gas and liquid velocities making the model as a whole more interconnected and a better holdup predictor. The Xiao et al. model is valid for all fluid types and pipe inclinations between -15degC and +15degC relative to horizontal.

Xiao (film modified)

The Xiao mechanistic model was developed as part of the TUFFP research consortia at the University of Tulsa ([p.594](#)). The standard implementation was modified in the stratified flow pattern to produce a second method called Xiao film modified. Unlike the Baker-Jardine implementation (which is based on the TUFFP version of the code, yet also contains a film modification), the Neotec version was coded independently and therefore the numerical methods and film modification approach are slightly different.

The modification consists of using the Ouyang and [Aziz \(p.583\)](#) two-phase correlation for film wall friction instead of the more traditional single phase correlation. The former has the advantage of making the film friction sensitive to both the gas and liquid velocities making the model as a whole more interconnected and a better holdup predictor. The Xiao et al. model is valid for all fluid types and pipe inclinations between -15degC and +15degC relative to horizontal.

4.1.3 Vertical Multiphase Flow Correlations

Setup » Flow Correlations

See also: [Flow regimes \(p.370\)](#), [Suggested flow correlation \(p.383\)](#),

The flow correlations available are affected by the Moody friction Factor calculation method option. By default, PIPESIM uses the explicit Moody friction factor calculation method ("Explicit Reformulation of the Colebrook-White Equation for turbulent Flow friction Factor calculation" by J. Sonnad and C. Goudar, *Ind. Eng. Chem. Res*, 2007, 46, pp. 2593-2600).

The following vertical multiphase flow correlations are available:

Ansari

The Ansari mechanistic model was developed as part of the Tulsa University Fluid Flow Projects (TUFFP) research program. A comprehensive model was formulated to predict flow patterns and the flow characteristics of the predicted flow patterns for upward two-phase flow. The comprehensive mechanistic model is composed of a model for flow pattern prediction and a set of independent models for predicting holdup and pressure drop in bubble, slug, and annular flows. The model was evaluated by using the TUFFP well databank that is composed of 1775 well cases, with 371 of them from Prudhoe Bay data.

Aziz Govier Fogarasi

The Aziz, Govier, and Fogarasi model was developed especially for wellbore pressure drop calculations for upward flow in production wells. The flow regime (for example, annular-mist, slug, etc.) is determined using the correlation of Govier and Aziz (1972). The flow pattern is predicted first, and then a corresponding correlation is used to calculate liquid holdup and frictional pressure loss. The Duns and Ros method is used for holdup and pressure calculations in the annular mist flow regime as recommended in the published work.

The correlation of Aziz, Govier, and Forgasi is used for pressure loss, holdup, and flow regime. The Govier, Aziz and Fogarasi correlation was developed following a study of pressure drop in wells producing gas and condensate. Actual field pressure drop versus flowrate data from 102 wells with gas-liquid ratios ranging from 3,900 to 1,170,000 scf/bbl were analyzed in detail. The phase conditions in the well bore were determined by standard flash calculations. Pressure-gradient data for flow under single-phase conditions were compared with conventional predictions, and found generally to confirm them. For the test in which two-phase conditions were predicted throughout the well bore, the field data were compared with several wholly empirical prediction methods, with a previously proposed method, and with a new prediction method partly based on the mechanics of flow. The new prediction method incorporates an empirical estimate of the distribution of the liquid phase between that flowing as a film on the wall and that entrained in the gas core. It employs separate momentum equations for the gas-liquid mixture in the core and for the total contents of the pipe.

Note: This method tends to overpredict the minimum stable flow rate (minimum rate to lift liquids) and thus can overpredict pressure losses, especially for gas-water wells.

Beggs and Brill Original

ORIGINAL: The Original [Beggs and Brill \(p.583\)](#) correlation is used for pressure loss and holdup. Flow regime is determined by either the Beggs and Brill or Taitel Dukler correlation. The Beggs and Brill correlation was developed following a study of two-phase flow in horizontal and inclined pipes. The correlation is based upon a flow regime map which is first determined as if the flow was horizontal. A horizontal holdup is then calculated by correlations, and this holdup is corrected for the angle of inclination. The test system included two 90 ft long acrylic pipes, winched to a variable elevation in the middle, so as to model incline flow both upwards and downwards at angles of up to 90°.

Beggs and Brill Revised

REVISED: As above except that the revised version of the Beggs and Brill correlation is used, with rough pipe friction factors, holdup limiters and corrective constants as proposed by [Palmer \(p.591\)](#) and [Payne \(p.591\)](#). The following enhancements to the original method are used; (1) an extra flow regime of froth flow is considered which assumes a no-slip holdup, (2) the friction factor is changed from the standard smooth pipe model, to utilize a single phase friction factor based on the average fluid velocity.

Duns and Ros

The Duns and Ros correlation is used for pressure loss and holdup with flow regime determination by either the [Duns and Ros \(p.586\)](#) or the [Taitel \(p.593\)](#) [Dukler \(p.585\)](#) correlations. The Duns and Ros correlation was developed for vertical flow of gas and liquid mixtures in wells. Equations were developed for each of three flow regions, (I) bubble, plug and part of froth flow regimes, (II) remainder of froth flow and slug flow regimes, (III) mist flow regime. These regions have low, intermediate and high gas throughputs respectively. Each flow region has a different holdup correlation. The equations were based on extensive experimental work using oil and air mixtures.

Gomez

The Gomez mechanistic flow model was developed at The University of Tulsa (p.587), and the code written by Neotec based on the published work.

The Gomez et al. model is valid for all fluid types and inclinations between 0 and 90degrees. (Recommended for 30-90 degrees).

Gomez enhanced

The standard Gomez et al. implementation was modified by Neotec to produce the Gomez et al. enhanced method.

The modification consists of using the [Oliemans \(p.590\)](#) liquid entrainment correlation for vertical annular mist flow instead of the standard Wallis correlation. Even though the Oliemans correlation was developed using low pressure, mainly water-air, small diameter data, it does a good job of smoothing the response surface around the slug to annular mist transition region where the Gomez et al. correlation shows unusual behavior. This improves the statistical performance of this method in simulations of gas-lift wells as shown by [Adames \(p.582\)](#). The result is an improved method that works well for all types of wells.

Govier and Aziz

The correlation of [Aziz, Govier, and Forgasi \(p.583\)](#) is used for pressure loss, holdup, and flow regime. The Govier, Aziz and Fogarasi correlation was developed following a study of pressure drop in wells producing gas and condensate. Actual field pressure drop versus flowrate data from 102 wells with gas-liquid ratios ranging from 3,900 to 1,170,000 scf/bbl were analyzed in detail. The phase conditions in the well bore were determined by standard flash calculations. Pressure-gradient data for flow under single-phase conditions were compared with conventional predictions, and found generally to confirm them. For the test in which two-phase conditions were predicted throughout the well bore, the field data were compared with several wholly empirical prediction methods, with a previously proposed method, and with a new prediction method partly based on the mechanics of flow. The new prediction method incorporates an empirical estimate of the distribution of the liquid phase between that flowing as a film on the wall and that entrained in the gas core. It employs separate momentum equations for the gas-liquid mixture in the core and for the total contents of the pipe.

Gray

The Gray Vertical Flow correlation is used for pressure loss and holdup. This correlation was developed by H E Gray of Shell Oil Company for vertical flow in gas and condensate systems which are predominantly gas phase. Flow is treated as single phase, and dropped out water or condensate is assumed to adhere to the pipe wall. It is considered applicable for vertical flow cases where the velocity is below 50 ft/s, the tube size is below 3.5 in, the condensate ratio is below 50 bbl/mmscf, and the water ratio is below 5 bbl/mmscf.

Gray Modified

As above, but with the following modifications: (1) Actual Reynolds number used (Gray Original assumed Reynolds number to always be 1 million), and (2) Pseudo-roughness is constrained to be less than the pipe radius.

Gregory

The Gregory et al model (1989) is a modification of the Aziz, Govier and Fogarasi (1972) method (described in [Aziz, Govier and Fogarasi \(p.582\)](#)). The Gregory et al model uses the Govier and Aziz flow pattern map (1972) except for the transition from annular-mist flow to froth flow. The transition between annular-mist (stable flow) and froth flow (unstable flow) is computed using the technique proposed by Turner et al (1969). Turner et al postulated that the minimum gas velocity required to lift liquids would correspond to the terminal velocity of the largest stable liquid droplet that would form. The Gregory model uses the procedure recommended by Coleman which does not include the 20% increase in velocity added by Turner. If the gas velocity (superficial gas velocity divided by the gas volume fraction in the input stream) is larger than the velocity expressed in the equation below, the flow pattern will be annular-mist, otherwise froth flow will be assumed.

$$V = 1.2979 \left[\frac{\sigma (\rho_L - \rho_G)}{C_D \rho_G^2} \right]^{1/4}$$

where

σ

gas-liquid surface tension (dynes/cm)

C_D

droplet drag coefficient

ρ_L

liquid density (lbm/ft³)

ρ_G

gas density (lbm/ft³)

V

velocity at the boundary between froth and annular-mist (ft/s)

C_L

liquid input volume fraction

The rest of the calculations are the same as for the Aziz, Govier and Fogarasi method (described in [Aziz, Govier and Fogarasi](#)), with the exception that the parameter V_{Gfm} (required for froth flow calculations) is computed as shown in the following equation:

$$V_{Gfm} = V (1 - C_L)$$

A default value of 0.44 (which corresponds to a spherical droplet shape) is provided for the droplet drag coefficient. Additionally, the Gray Revised method for pressure drop is used instead of the Duns and Ros method in the annular-mist regime.

Hagedorn and Brown

The correlation of [Hagedorn and Brown \(p.587\)](#) is used for pressure loss and holdup. While the Hagedorn and Brown correlation does not predict flow pattern, the flow pattern as predicted by Orkiszewski is reported. The Duns and Ros flow pattern prediction can also be reported. Neither of these flow pattern prediction methods affects any of the calculations. The Hagedorn and Brown correlation was developed following an experimental study of pressure gradients occurring during continuous two-phase flow in small diameter vertical conduits. A 1,500 ft experimental well was used to study flow through 1 in., 1.25 in., and 1.5 in. nominal size tubing. Air was the gas phase and four different liquids were used: water and crude oils with viscosities of about 10, 30 and 110 cp. Liquid holdup was not directly measured, rather a pseudo liquid-holdup value was determined that matched measured pressure gradients.

Further work by Brill and Hagedorn have led to two modifications: (1) If the Griffith and Wallis criteria predicted the occurrence of bubble flow, the Griffith bubble-flow method should be used to predict pressure gradient, and (2) If the predicted liquid holdup is less than the no-slip liquid holdup, then the no-slip liquid holdup is used.

All of the correlations involve only dimensionless groups, which is a condition usually sought for in similarity analysis but not always achieved.

Mukherjee and Brill

The [Mukherjee and Brill \(p.590\)](#) correlation is used for Pressure loss, Holdup and flow map. Note: selection of alternative flow maps and/or holdups will cause unpredictable results. The Mukherjee and Brill correlation was developed following a study of pressure drop behavior in two-phase inclined flow. For bubble and slug flow, a no-slip friction factor calculated from the Moody diagram was found adequate for friction head loss calculations. In downhill stratified flow, the friction pressure gradient is calculated based on a momentum balance equation for either phase assuming a smooth gas-liquid interface. For annular-mist flow, a friction factor correlation was presented that is a function of holdup ratio and no-slip Moody friction factor. Results agreed well with the experimental data and correlations were further verified with Prudhoe Bay and North Sea data.

NOSLIP Correlation

The NOSLIP correlation assumes homogeneous flow with no slip between the phases. Fluid properties are taken as the average of the gas and liquid phases and friction factors are calculated using the single phase MOODY correlation. Note: selection of alternative flow maps and/or holdups will cause unpredictable results.

OLGAS 2-phase/ OLGAS

The OLGAS mechanistic models are applicable for all inclination angles, pipe diameters and fluid properties. The 2-phase [Bendiksen \(p.583\)](#) model considers gas-liquid flow, whereas the 3-phase model considers gas-oil-water flow.

This model employs separate continuity equations for gas, liquid bulk and liquid droplets, which are coupled through interphase mass transfer. Two momentum equations are solved: one applied to the combined balance for the gas and liquid droplets, if present, and a separate momentum equation for the liquid film. OLGAS considers four flow regimes: stratified, annular, slug and dispersed bubble flow; and uses a unique minimum slip criteria to predict flow regime transitions.

The OLGA 2-Phase model uses the liquid viscosity model defined within the PIPESIM fluid property definition. The 3-Phase model uses the Pal and Rhodes emulsion correlation to calculate liquid viscosity based on the oil and water viscosities defined with the PIPESIM fluid model definition; liquid viscosity options defined with the PIPESIM fluid model are ignored.

OLGAS is based in large part on data from the SINTEF multiphase flow laboratory near Trondheim, Norway. The test facilities were designed to operate at conditions that approximated field conditions. The test loop is 800 m long and 8 inches in diameter. Operating pressures between 20 and 90 barg were studied. Gas superficial velocities of up to 13 m/s, and liquid superficial velocities of up to 4 m/s were obtained. In order to simulate the range of viscosities and surface tensions experienced in field applications, different hydrocarbon liquids were used (naptha, diesel, and lube oil). Nitrogen was used as the gas. Pipeline inclination angles between 1° were studied in addition to flow up or down a hill section ahead of a 50m high vertical riser. Over 10,000 experiments were run on this test loop during an eight year period. The facility was run in both steady state and transient modes.

LEDA

The ([Leda Point Model \(PM\)](#)) is a mechanistic model applicable for all inclination angles, pipe diameters and fluid properties. The 2-phase model considers gas-liquid flow whereas the 3-phase model considers gas-oil-water flow.

The 3-phase Leda PM considers 9 fields in the mass (continuity) equations (oil, gas, water, oil in gas and water, gas in oil and water, water in oil and gas). Separate momentum equations are solved for oil, gas and water.

The 2-phase Leda PM considers 4 fields in the mass (continuity) equations (liquid, gas, liquid in gas and gas in liquid). Separate momentum equations are solved for gas and liquid phases. The flow regimes predicted by LedaPM are stratified smooth flow, stratified wavy flow, slug flow, annular and bubbly flow. The Leda 2-phase model uses the liquid viscosity associated with the fluid model defined in PIPESIM. The Leda 3-phase model assumes that the liquid viscosity is equal to that of the continuous phase; liquid viscosity options defined with the PIPESIM fluid model are ignored. The continuous phase is determined by the [Brauner-Ullman \(p.584\)](#) inversion criteria.

The Leda Point Model is the steady-state version of the transient model developed by SINTEF in collaboration with Total and ConocoPhillips and commercialized by Kongsberg. The model has been calibrated against data collected at the SINTEF Multiphase Flow Laboratory near Trondheim Norway. Over 10,000 experimental data points have been collected for single-phase, two-phase (oil-water, water-gas) and three-phase (oil-water-gas) flow. Pipe diameters ranging from 4-12" were used at pressures up to 90 barg. The models have been validated with field data supplied by ConocoPhillips and Total.

Orkiszewski

The [Orkiszewski \(p.591\)](#) correlation is used for pressure loss, holdup, and flow regime. The Orkiszewski correlation was developed for the prediction of two phase pressure drops in vertical pipe. Four flow regimes were considered, bubble, slug, annular-slug transition, and annular mist. The method can accurately predict, to within 10%, the two phase pressure drops in naturally flowing and gas lifted production wells over a wide range of well conditions. The precision of the method was verified when its predicted values were compared against 148 measured pressure

drops. Unlike most other methods, liquid holdup is derived from observed physical phenomena, and is adjusted for angle of deviation.

TUFFP Unified Mechanistic Model (2-phase and 3-phase)

The TUFFP Unified Mechanistic Model is the collective result of many research projects performed by the Tulsa University Fluid Flow Projects (TUFFP) research consortium. The model determines flow pattern transitions, pressure gradient, liquid holdup and slug characteristics. A 2-phase version is available for gas-liquid flow [Zhang et.al, development (p.594) and validation (p.594)] and a 3-phase version is available for gas-oil-water pipe flow [Zhang and Sarica (p.594)]. The model is valid for all inclination angles, pipe diameters and fluid properties.

The principle concept underlying the model is the premise that slug flow shares transition boundaries with all the other flow patterns. The flow pattern transition from slug flow to stratified and/or annular flow is predicted by solving the momentum equations for slug flow. The entire film zone is treated as the control volume and the momentum exchange between the slug body and the film zone is introduced into the combined momentum equation. This approach differs from traditional methods of using separate models for each transition. The advantage of a single hydrodynamic model is that the flow pattern transitions, slug characteristics, liquid holdup and pressure gradient are implicitly related.

The 3-phase model contains separate momentum balances for the gas, oil and water phases. The model determines whether the oil and water phases are separated or fully mixed. If the phases are separated, individual phase viscosities are used. If the phases are fully mixed, the liquid viscosity can be determined either by the method within the TUFFP model (emul default option) or overridden (emul override option) by the liquid viscosity method defined with the PIPESIM fluid model, which is useful when rheology data are available. In the latter case, for black oil fluid models, selecting the Brinkman emulsion viscosity method with the Brauner-Ullman watercut cutoff method will replicate the method used within the TUFFP model. For the 2-phase (gas-liquid) model, the liquid viscosity from PIPESIM is always used, so the emulsion options defined in the PIPESIM fluid definition always apply.

The closure relationships included in the model are based on focused experimental research programs at University of Tulsa and elsewhere. As new and improved closure relationships become available, the TUFFP Unified Model is updated and validated.

Note: The TUFFP Unified 2-Phase Model v 2007.1 is no longer supported in PIPESIM. Upon import, TUFFP version 2011.1 is used instead.

4.1.4 Suggested correlations

Use the **Flow correlations** tab to set flow correlation options at the global level or at local levels. If you set flow correlation options at the local level, the source, correlation, friction factor, and holdup factor appear as individual columns for both vertical and horizontal components.

If no production data are available, Schlumberger have found the following to give satisfactory results based on previous studies using field data:

Single phase system

[Moody \(p.386\)](#)

Vertical oil well[Hagedorn and Brown \(p.381\)](#)**Highly deviated oil well**[Hagedorn and Brown \(p.381\) or Duns and Ros \(p.378\) or OLGA-S \(p.381\)](#)**Gas/condensate well**[Hagedorn and Brown \(p.381\)](#)**Oil pipelines**[Oliemans \(p.375\)](#)**Gas/condensate pipelines**[BJA Correlation \(p.372\)](#)

Correlation	Vertical and Predominantly Vertical Oil Wells (p.377)	Highly Deviated Oil Wells (p.377)	Vertical Gas/Condensate Wells (p.377)	Oil Pipelines (p.372)	Gas/Condensate Pipelines (p.372)
Duns and Ros	yes	yes	yes	no	no
Orkiszewski	yes	no	yes	no	no
Hagedorn and Brown	yes	no	yes	no	no
Beggs and Brill Revised	yes	yes	yes	yes	yes
Beggs and Brill Original	yes	yes	yes	yes	yes
Mukherjee and Brill	yes	yes	yes	yes	yes
Govier, Aziz and Forgasi	yes	yes	yes	no	no
NoSlip	yes	yes	yes	yes	yes
OLGAS	yes	yes	yes	yes	yes
Ansari	yes	no	yes	no	no
BJA for Condensates	no	no	yes	no	yes
AGA and Flanigan	no	no	no	no	yes
Oliemans	no	no	no	yes	yes
Gray	no	no	yes	no	no
Gray Modified	no	no	yes	no	no
Xiao	no	no	no	yes	yes

LEDA	yes	yes	yes	yes	yes
TUFFP	yes	yes	yes	yes	yes

4.1.5 Friction and Holdup factors

These two factors can be used to adjust the friction and holdup prediction of a particular flow correlation. By default these factors are 1.

A linear relationship is used for the friction pressure drop. Setting the friction factor to 0.5, for example, will mean that the friction element of pressure drop computed by the correlation will be halved.

A non-linear relationship is used to calculate the liquid holdup H_L from the value predicted by the correlation H_{Lc} :

$$H_L = f_H \cdot H_{Lc} + (1 - f_H) \cdot H_{Lc}^2 \quad \text{Eq. 4.1}$$

This ensures that the liquid holdup is sensible $0 \leq H_L \leq 1$ when $0 \leq f_H \leq 2$.

These factors are often used as calibration factors when a good match to field data cannot be obtained by any other method. Changing these factors will affect the results and should be undertaken with care.

4.1.6 Single Phase Flow Correlations

See also: [SPHASE Single Phase Flow Options \(p.648\)](#)

The steady-state pressure gradient in single phase sections is given by the equation:

$$\frac{dp}{dL} = \left(\frac{dp}{dL} \right)_{elev.} + \left(\frac{dp}{dL} \right)_{fric.} + \left(\frac{dp}{dL} \right)_{acc.} \quad \text{Eq. 4.2}$$

where elevation, friction and acceleration components of the pressure drop are:

$$\left(\frac{dp}{dL} \right)_{elev.} = - \rho g \sin \theta \quad \text{Eq. 4.3}$$

$$\left(\frac{dp}{dL} \right)_{fric.} = - \frac{f \rho v^2}{2D} \quad \text{Eq. 4.4}$$

$$\left(\frac{dp}{dL} \right)_{acc.} = - \rho v \frac{dv}{dL} \quad \text{Eq. 4.5}$$

where

f	is the friction factor	dimensionless
ρ	is the fluid density	lb/ft^3
v	is the fluid velocity	ft/s

g	is the gravitational acceleration	ft/s^2
θ	is the angle of the pipe to the horizontal	<i>degrees</i>
D	is the pipe diameter	ft
L	is the length of the pipe	ft

There are a number of different ways of calculating the friction factor, which usually depends on the Reynolds number:

$$Re = \frac{\rho v D}{\mu} \quad \text{Eq. 4.6}$$

where:

μ	is the fluid viscosity	$Ib/ ft \cdot s$
-------	------------------------	------------------

Moody (default for liquid or gas)

See [Sonnad and Goudar paper \(p.593\)](#) and [Moody paper \(p.590\)](#) for more technical details.

For laminar flow ($Re < 2000$)	$f_{Lam} = \frac{64}{Re}$
For turbulent flow ($Re > 4000$)	$\frac{1}{f_{Turb}}^{1/2} = a \left[\ln \left(\frac{c}{q} + \delta \right) \right]$
For transition flow ($2000 \leq Re \leq 4000$)	$f = \frac{(Re - Re_{min})(f_{Turb} - f_{Lam})}{(Re_{max} - Re_{min})} + f_{Lam}$

where:

f_{Turb}	is the Moody friction factor
Re	is the Reynolds Number
a	is $\frac{2}{\ln(10)}$
ϵ	is the pipe roughness
D	is the pipe diameter
b	is $\frac{\epsilon/D}{3.7}$
c	is $\left(\frac{\ln(10)}{5.02} \right) Re$
s	is $bc + \ln(c)$

q	is $s^{\lceil s/(s+1) \rceil}$
z	is $\ln\left(\frac{q}{g}\right)$
g	is $bc + \ln\left(\frac{c}{q}\right)$
δ	is $\left(\frac{g}{g+1}\right)z$

The friction factor is interpolated in the transition region ($2000 < \text{Re} < 4000$). The limits for the transition zone and the interpolation method can be reset by the user.

The various friction factor calculation methods available are:

Friction Factor Calculation method	Approximation used	Equation
EXPLICIT or SONNAD	Sonnad 2007 linear approximation (default)	$\frac{1}{f_{Turb}^{1/2}} = a \left[\ln\left(\frac{c}{q} + \delta\right) \right]$
APPROXIMATE or MOODY	Moody 1947 approximation	$f_{Turb} = 0.0055 \left[1 + \left(20000 \frac{\epsilon}{D} + \frac{10^6}{\text{Re}} \right)^{1/3} \right]$
IMPLICIT or ITERATIVE	Colebrook-White equation (Moody chart)	$\frac{1}{f_{Turb}^{1/2}} = 1.74 - 2 \log_{10} \left(\frac{2\epsilon}{D} + \frac{18.7}{\text{Re} f_{Turb}^{1/2}} \right)$

AGA (for gas)

The AGA friction factor is the same as the Moody friction factor at high and low Reynolds numbers, but differs in between:

For laminar flow ($\text{Re} < 1000$)	$f = \frac{64}{\text{Re}}$
For transition flow $(1000 < \text{Re} < 4 \frac{c_2}{c_1} \left(\frac{3.7D}{\epsilon} \right)^{1/c_1} \log_{10} \left(\frac{3.7D}{\epsilon} \right))$	$\frac{1}{f^{1/2}} = 2c_1 \log_{10} \left(\frac{\text{Re}}{2} \frac{c_1}{c_2} f^{1/2} \right)$
For turbulent flow $(\text{Re} > 4 \frac{c_2}{c_1} \left(\frac{3.7D}{\epsilon} \right)^{1/c_1} \log_{10} \left(\frac{3.7D}{\epsilon} \right))$	$\frac{1}{f^{1/2}} = 2 \log_{10} \left(\frac{3.7D}{\epsilon} \right) = 1.74 - 2 \log_{10} \left(\frac{2\epsilon}{D} \right)$

where:

$c_1 = 0.98$	is the drag factor
$c_2 = 10^{0.15}$	is a constant

Cullender and Smith (for gas)

The total pressure drop can be calculated from

$$\frac{dp}{dL} = \frac{p_{down} - p_{up}}{L}$$

where:

$$p_{up}^2 = \frac{p_{down}^2 - a^2}{b}$$

where:

$$a^2 = \frac{25 f q_{vG}^2 \bar{T}^2 Z_G^2 (b - 1)}{0.0375(12D)^5}$$

$$b = \exp\left(\frac{0.0375 \gamma_G L}{\bar{T} Z_G}\right)$$

f	is the Moody friction factor	dimensionless
L	is the pipe length	ft
p_{down}	is the downstream pressure	psi
p_{up}	is the upstream pressure	psi
q_{vG}	is the stock tank gas volume flow rate	scf / day
\bar{T}	is the average temperature	° R
Z_G	is the gas compressibility factor	dimensionless
γ_G	is the gas specific gravity	dimensionless

Other friction pressure drops for gas

The friction pressure drop can be calculated from

$$\left(\frac{dp}{dL}\right)_{fric.} = \frac{p_{down} - p_{up}}{L}$$

where:

$$p_{up}^2 - p_{down}^2 = \frac{\overline{T}Z_G L}{5280} \left(\frac{P_s}{T_s} \right)^2 \left(\frac{1}{a_1} \cdot \frac{q_{vG}}{\eta} \cdot \frac{\gamma_G^{a_4}}{(12D)^{a_5}} \right)^{1/a_3}$$

where:

L	is the pipe length	ft
p_{down}	is the downstream pressure	psi
p_{up}	is the upstream pressure	psi
P_s	is the stock tank pressure	psi
q_{vG}	is the stock tank gas volume flow rate	scf / day
\overline{T}	is the average temperature	° R
T_s	is the stock tank temperature	° R
Z_G	is the gas compressibility factor	dimensionless
γ_G	is the gas specific gravity	dimensionless
η	is a flow efficiency factor	dimensionless

and the constants are given by

	a_1	a_3	a_4	a_5
Panhandle A	435.87	0.5394	0.4604	2.618
Panhandle B	737.00	0.5100	0.4900	2.530
Weymouth	433.50	0.5000	0.5000	2.667

Hazen-Williams (for liquid water)

The friction pressure drop can be calculated from:

$$\left(\frac{dp}{dL} \right)_{fric.} = \frac{0.015 \rho_m}{144(12D)^{4.87}} \cdot \left(\frac{q_{vL}}{c} \right)^{1.85} \quad \text{Eq. 4.7}$$

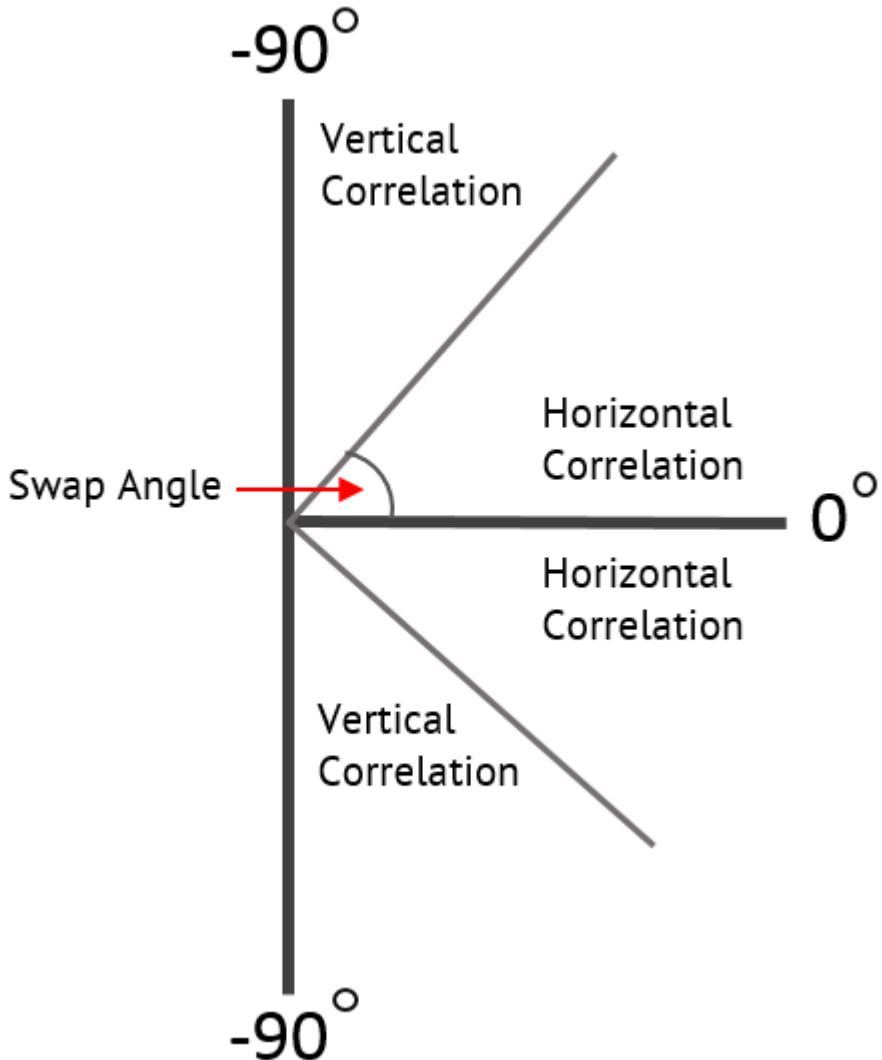
where:

c	is the pipe condition factor	
q_{vL}	is the liquid volume flow rate	bb1 / day
ρ_m	is the mixture density	lb / ft ³

4.1.7 Swap Angle

The multiphase flow correlations used to predict the pressure loss and holdup are split into two categories: vertical and horizontal. Each category lists the correlations that are appropriate for that type of flow.

By default the selected vertical correlation is used in the situation where the tubing/pipe is within 45 degrees of the vertical, up (+90 degrees) or down (-90 degrees). Outside this range the selected horizontal correlation is used. This angle can be changed.



4.1.8 deWaard (1995) Corrosion Model

The [de Waard model \(p.585\)](#) predicts the corrosion rate of carbon steel in the presence of water and CO₂. The model was developed primarily for use in predicting corrosion rates in pipelines where CO₂ is present in a vapor phase. The model has not been validated at high pressures where CO₂ is entirely in the liquid phase. Corrosion rate is calculated as a function of:

- Temperature

- Pressure
- Mol% CO₂
- Wt% Glycol (Multiflash and ScaleChem only)
- Liquid velocity
- Pipe Diameter
- pH

The model accounts for the flow-independent kinetics of the corrosion reaction as well as the flow-dependent mass transfer of dissolved CO₂ using a resistance model. Additionally, effects of protective scale at high temperatures are considered in addition to glycol inhibition.

Note: The equations that follow are based on the [de Waard 1995 model \(p.585\)](#). This model is a revision to the [de Waard 1991 model \(p.585\)](#). Some of the equations below appear only in the original paper].

General Equation

$$V_{cor} = \frac{CcFsFg}{\left[\frac{1}{V_r} + \frac{1}{V_m} \right]} \quad \text{Eq. 4.8}$$

CO₂ Partial Pressure/Fugacity

$$pCO_2 = \frac{(\text{mole \% } CO_2 * P_{total})}{100} \quad \text{Eq. 4.9}$$

$$\log (f_{CO_2}) = \log (pCO_2) + (.0031 - \frac{1.4}{t + 273})P$$

Reaction Rate term (Vr)

$$\log (V_r) = 4.93 - \frac{1119}{T} + 0.58\log (fCO_2) - .34(pH_{act} - pH_{CO_2}) \quad \text{Eq. 4.10}$$

pH

By default, the correlation assumes that the actual pH of the water is affected strictly by the presence of CO₂. However, the user may specify the actual pH of a water sample that accounts for the additional presence of electrolytes and dissolved FeCO₃ liberated from the pipe wall. Since pH is dependent on pressure and temperature, care must be taken when specifying this value. If a ScaleChem generated PVT file is used, the actual pH is taken from the ScaleChem fluid description.

$$pH_{CO_2} = 3.82 + .00384t - 0.5\log (fCO_2) \quad \text{Eq. 4.11}$$

$pHact$ = assumed to equal $pHco2$ unless user specified or ScaleChem PVT file is used

Mass Transfer rate term (Vm)

$$Vm = 2.45 \frac{U_L^{0.8}}{d^{0.2}} fCO_2 \quad \text{Eq. 4.12}$$

Effect of Temperature (protective scale)

$$Ts = \frac{2400}{6.7 + 0.44\log(fCO_2)} \quad \text{Eq. 4.13}$$

(if $T > Ts$)

$$\log(F_s) = 2400 \left[\frac{1}{T} - \frac{1}{Ts} \right] \quad \text{Eq. 4.14}$$

Else,

$$F_s = 1 \quad \text{Eq. 4.15}$$

Glycol Reduction Effect

$$\log F_g = 1.6[\log(W\%) - 2] \quad \text{Eq. 4.16}$$

Where W% is the weight percent of water in a water-glycol mixture (100% water results in a factor of 1.0). The Glycol component is only available when using Multiflash (MEG or DEG) or with ScaleChem (MEG).

Variable	Units	Description	Default	Acceptable Input Range	Variable Source
$Vcor$	(mm/yr)	corrosion rate			calc
T	°K	temperature			pipesim
t	°C	temperature			pipesim
pCO^2	atm	partial pressure of CO ₂			calc
fCO^2	atm,	fugacity of CO ₂			calc
$mol\%CO_2$	—	mol % CO ₂ (comp, BO, ScaleChem PVT file)			pipesim
P_{total}	atm	pressure			pipesim
$pHact$	—	actual pH of the system	$pHco_2$	1.0–10.0	user spec
$pHCO_2$	—	pH of dissolved CO ₂ in pure water			calc
U_L	m/s	liquid velocity			pipesim

Variable	Units	Description	Default	Acceptable Input Range	Variable Source
d	m/s	pipe diameter			pipesim
$W\%$	fraction	Weight percent water in a water-glycol mixture	100		pipesim
T_s	°K	Vcor inversion temperature			calc
F_s	—	scaling factor			calc
C_c	—	multiplier to correct for inhibitor efficiency or match to field data	1	0.1–10.0	user spec

4.1.9 Cunliffe's Method for Ramp Up Surge

Cunliffe's Method is used to predict the liquid surge rate due to an overall gas rate change for condensate pipelines. This method is particularly useful for estimating liquid handling capacity for ramp-up (increasing gas rate) cases. As the gas rate increases, the total liquid holdup in the line will drop owing to less slippage between the gas and liquid phases. The liquid residing in the line is therefore accelerated to the equilibrium velocity at the final gas rate and thus expelled at a rate higher than the final equilibrium liquid rate for the duration of the transition period. The transition period is assumed to be equal to the residence time at the final gas rate, that is, the time it takes the liquid to travel from one end of the line to the other.

The average liquid rate during the transition period can be determined as follows:

$$q_{L_T} = q_{L_i} + \frac{(H_{L_{Ti}} - H_{L_{Tf}})}{t_r}$$

$$q_{L_t} = q_{G_i} (LGR_{out})$$

$$t_r = \frac{H_{L_{Tf}}}{q_{L_i}}$$

where:

q_{L_T}	liquid rate during the transition period
q_{L_i}	initial liquid rate
$q_{G_i} = H_{L_T}$	total liquid holdup volume in line - initial gas rate
$H_{L_{Tf}}$	total liquid holdup volume in line - final gas rate

LGR_{out}	liquid/gas ratio at outlet pressure (assumed constant)
t_r	liquid residence time (at final flowrate)

Note: The total liquid holdup volume in the line is provided in the summary output report. Cunliffe tested this method with field measurements for a 67 mi. 20 in. pipeline with an average operating pressure of 1300 psig and an LGR of 65 bbl/MMscf. He found that the change in condensate flow rate can be predicted to within 15% using this method.

Reference: Cunliffe, R.: "Prediction of Condensate Flow Rates in Large Diameter High Pressure Wet Gas Pipelines", APEA Journal (1978), 171-177.

4.1.10 Liquid by Sphere

The liquid holdup throughout the pipe will be divided into two notional fractions, that is . the 'moving' and the 'static'. Since the liquid normally flows slower than the gas, the division will normally result in a positive value for both of these volumes. (If the pipe goes downhill the liquid often flows faster, so the 'static' will be negative in these sections, but this does not affect the equation.) If the fluid's phase split is assumed to be constant throughout the pipe, the size of the slug that issues when spheroid can be calculated using the following formula:

$$SGLV = \left(\frac{SLV}{TPV - MLV} \right) \times MLV + SLV \quad \text{Eq. 4.17}$$

where:

$SGLV$ is Sphere Generated Liquid Volume

SLV is Static Liquid Volume in pipe

MLV is Moving Liquid Volume in pipe

TPV is Total Pipeline Volume

Note: $SLV + MLV$ = Total pipeline holdup, which PIPESIM calculates and writes to the summary output.

The explanation for this formula is as follows. The slug of liquid starts to issue from the pipe when the pipe is full of liquid from its exit, back along to the position of the sphere. The liquid in the slug comprises 2 notional fractions: firstly, the entire SLV in the pipe, and secondly, that portion of the MLV that lies between the sphere and the outlet. Now: the volume available for the SLV to occupy in the pipe is $TPV - MLV$. Dividing this into SLV gives us the position of the sphere in the pipe as a value between 0 and 1, where 0 is the outlet. Multiplying the MLV by this gives us the portion of the MLV that is entrained in the slug, so adding this to the SLV gives the total slug volume.

The liquid holdup is calculated from the integration of the predicted holdup from the selected Multiphase Flow Correlation (MFC) along the entire pipeline length. The pipeline is simulated in segments, each of which has a length and cross sectional area, which multiplied together yield its volume. The MFC calculates a value for holdup in the range 0 to 1, so this multiplied by the

segment volume gives holdup for the segment. The holdups for all the pipe segments are added together to yield the pipeline total holdup as reported in the summary file.

When a sphere is introduced into the line, it will gather in front of itself a liquid slug made from "all the liquid that is flowing slower than the mean fluid flowrate in the pipeline at any given point". Thus the crucial value that determines Sphere Generated Liquid Volume (SGLV) is the Slip Ratio (SR), which is the average speed of the fluid divided by the speed of the liquid. If the liquid and gas move at the same speed, the slip ratio will be 1, that is there is 'no slip' between the phases. In this situation the sphere will not collect any liquid, so the SGLV will be zero. Normally the liquid flows slower than the gas, that is the slip ratio is greater than 1, so "some" of the liquid in the pipeline will collect in front of the sphere to form the SGLV. The only way that "all" of the liquid in the pipeline will collect to form the SGLV, is if the liquid velocity is zero, i.e.. the slip ratio is infinite. This cannot happen in a steady-state reality, so the SGLV is always smaller than the total liquid holdup.

One complicating factor is that the slug of liquid swept up by the sphere will begin to emerge from the end of the pipe some considerable time before the sphere itself emerges. This slug will be composed of the liquid that the sphere collected on its way, plus the normal liquid production of the system. This total volume is the figure required to size the slug catcher, which is why we report it as "Volume by sphere".

To determine the sizes of terrain slugs or slugs from start up it is necessary to use a dynamic multiphase flow simulator such as LEDA or OLGA. [More details. \(p.640\)](#)

PI-SS (Severe-Slugging Group)

PI-SS (severe-slugging group) is the ratio between the pressure buildup rates of gas phase and that of liquid phase in a flowline, when followed by a riser:

$$\Pi_{ss} = \frac{ZRT}{gL <\alpha_{GF}>} \frac{W_G}{W_L} \quad \text{Eq. 4.18}$$

where

Z	Gas compressibility factor	
$R = 8314 \text{ J/K} \cdot \text{kmol}$	Gas constant	
T	Temperature	K
M_G	Molecular weight of gas	(kg/kmol)
W_G	Gas mass flow rate	kg/s
W_L	Liquid mass flow rate	kg/s
$g = 9.81 m/s^2$	Acceleration due to gravity	
L	Flowline length	m
$<\alpha_{GL}>$	Average flowline gas holdup	

This expression is with assumptions of no mass transfer between the phases $\rho_L \gg \rho_G$, and the liquid fall back in the riser is neglected.

This PI-SS expression is based upon a correlation developed at Koninklijke Shell-Laboratorium (see [Pots and Bromilow \(p.591\) 1985](#)) to quantify the likelihood of severe riser slugging, that is . when $\Pi_{ss} < 1.0$.

For severe slugging to occur, at least two conditions must be in evidence:

1. the flowline gas flow must be completely inhibited during slug buildup (that is due to a partly declining flowline or the presence of flowline undulations).
2. the rate of hydrostatic pressure buildup in the riser due to the growth of the slug must exceed the rate of gas pressure buildup in the flowline.

Under such conditions, the riser becomes filled with liquid before the gas pressure can drive the liquid slug out of the line.

In PIPESIM if the value of Π_{ss} is less than one at the riser base and the flow regime (as predicted by the Taitel-Dukler correlation) is stratified (or wavy stratified), then severe riser slugging is possible. Conversely, Π_{ss} values significantly greater than one indicate that severe riser slugging is not likely.

The PI-SS number (Π_{ss}) can also be used to estimate slug size. As a rule of thumb the slug length will be approximately equal to the riser height divided by Π_{ss} :

$$\text{Slug Length} = \text{Riser Length} / \Pi_{ss}$$

Hence PI-SS values (Π_{ss}) less than unity imply slug lengths greater than the riser height. PI-SS is calculated at each node in the flowline (while PISS=ON) using averaged holdup data

4.1.11 Liquid Loading

Critical Unloading Velocity

The critical unloading velocity is defined as the minimum gas velocity required to lift liquid droplets out of a gas well. Lower flowing gas velocities will result in liquid loading in the well. The critical unloading velocity is predicted by Turner's Equation.

$$V_t = \frac{N \left[\sigma (\rho_L - \rho_G) \right]^{0.25}}{(C_D^{25} \rho_G^{0.5})} \quad \text{Eq. 4.19}$$

where

ρ_g	is the gas phase density	lb/ft^3
ρ_L	is the liquid phase density	lb/ft^3

σ	is the interfacial tension	dynes / cm
v_t	is the terminal velocity of liquid droplet	ft / s
θ	is pipe angle from vertical	°
C_D	is the drag coefficient	dimensionless
N	is a constant	dimensionless

The values of N and C_D are given in the following table for Turner's model and various others:

Model	N	C_D
Turner (1969)	1.56	0.44
Coleman (1991)	1.3	0.44
Nossier II (2000)	1.482	0.2
Li (2002)	0.724	1.0

Combining N and C_D , and discounting Turner's "built-in" 20% "correction factor" gives a constant of 1.593. The correction factor is split out into the E term below.

Turner's Equation (General)

Turner's Equation (General Form):

$$v_t = \frac{1.593E[\sigma(\rho_L - \rho_G)]^{0.25}}{\rho_G^{0.5}} \quad \text{Eq. 4.20}$$

Where E is the correction (efficiency) factor. The values of E for Turner's model and various others are given in the following table:

Model	E
Turner (1969)	1.2
Coleman (1991)	1.0
Nossier II (2000)	1.391
Li (2002)	0.454

Critical Gas Rate

The critical gas rate is the minimum gas rate required to prevent liquid loading.

4.2 Completion (IPR) Models

4.2.1 Inflow Performance Relationships for Vertical Completions

Inflow performance relationships (IPRs) have been developed to model the flow of fluids from the reservoir, through the formation, and into the well. They are expressed in terms of the well static (or reservoir) pressure P_{ws} , the well flowing (or bottom hole) pressure P_{wf} , and flowrate Q . Typically, volume flow rates are proportional to the pressure drawdown:

$$Q_V \propto (P_{ws} - P_{wf}) \quad \text{Eq. 4.21}$$

For liquid IPRs the stock tank liquid rate is roughly proportional to the volume flow rate at well conditions, and this form of the equation is used:

$$Q_L \propto (P_{ws} - P_{wf}) \quad \text{Eq. 4.22}$$

For gas IPRs the stock tank flow rates are roughly proportional to the volume flow rate at reservoir conditions times the average reservoir pressure:

$$Q_G \propto Q_V \cdot \frac{(P_{ws} + P_{wf})}{2} \propto (P_{ws}^2 - P_{wf}^2) \quad \text{Eq. 4.23}$$

See also [Vertical Completion Options](#) When the selected IPR model is Darcy and one of the Skin options is set to calculate, the following vertical completion options are available: Multilayer Completions

PIPESIM offers a comprehensive list of IPR options, for both oil and gas reservoirs, as follows:

IPR	Oil reservoirs	Gas and Gas Condensate Reservoirs	Multi-rate test In addition to the standard IPR equations, test data can be utilized so that the inflow can be matched to actual measured data. A minimum of three data points is required. Two types of multi-rate test are available:
Backpressure Equation (p.402)		Yes	Yes
Fetkovich (p.401)	Yes		Yes
Hydraulically fractured (p.65)	Yes	Yes	
IPR Table (p.413)	Yes		
Jones / Forchheimer (p.401)	Yes	Yes	Yes
Pseudo Steady State Equation / Darcy (p.403)	Yes	Yes	

Transient (p.408)	Yes	Yes	
Vogel (p.400)	Yes		
Well PI (Productivity Index) (p.399)	Yes	Yes	Yes

The Well PI (p.399), Pseudo Steady State (p.403) and Transient (p.408) liquid IPRs can be combined with a Vogel (p.400) IPR to model flow at pressures below the bubble point; see bubble point correction (p.414) .

Productivity Index (PI)

PI is one of a number of methods that can be used to specify the Inflow Performance Relationship (p.398) (IPR) for a completion. It can be regarded as a simplified version of the pseudo-steady state (p.403) or transient (p.408) IPRs.

Liquid PI

The (straight line) productivity index relationship for liquid reservoirs is perhaps the simplest and most widely used IPR equation. It states that rate is directly proportional to pressure drawdown between the bottom hole and the reservoir.

$$Q_L = J_L \cdot (p_{ws} - p_{wf}) \quad \text{Eq. 4.24}$$

where:

Q_L is the stock-tank oil rate

p_{ws} is the well static (or reservoir) pressure

p_{wf} is the well flowing (or bottom hole) pressure

J_L is the liquid productivity index.

Below bubble point correction

The liquid PI equation can be combined with a Vogel equation (p.400) to model inflows when the bottom hole pressure is below the bubble point, see, Bubble point correction. (p.414)

Gas PI

For gas reservoirs a non-linear relationship is used:

$$Q_G = J_G \cdot \left(p_{ws}^2 - p_{wf}^2 \right) \quad \text{Eq. 4.25}$$

where:

Q_G is the stock-tank gas rate

p_{ws} is the well static (or reservoir) pressure

p_{wf} is the well flowing (or bottom hole) pressure

J_G is the gas productivity index

Vogel's Equation

Vogel's (1968) (p.593) equation is one of a number of methods that can be used to specify the Inflow Performance Relationship (p.398) (IPR) for a completion. It was developed to model saturated oil wells. Vogel's equation is a best-fit approximation of numerous simulated well performance calculations. Vogel's work considers only the effect of rock and fluid properties on saturated systems. The Vogel relation does not account for high-velocity-flow effects that may exist in high-rate wells, see the Fetkovich equation (p.401).

Vogel's equation is:

$$Q = Q_{\max} \left(1 - \left(1 - C \right) \left(\frac{p_{wf}}{p_{ws}} \right) - C \left(\frac{p_{wf}}{p_{ws}} \right)^2 \right) \quad \text{Eq. 4.26}$$

Where

Q is the liquid flow rate (STB/D or m³/d)

Q_{\max} is the absolute open hole flow potential, that is the liquid flow rate when the bottom hole pressure is zero

p_{wf} is the well flowing (or bottom hole) pressure (psia or bara)

p_{ws} is the well static (or reservoir) pressure (psia or bara)

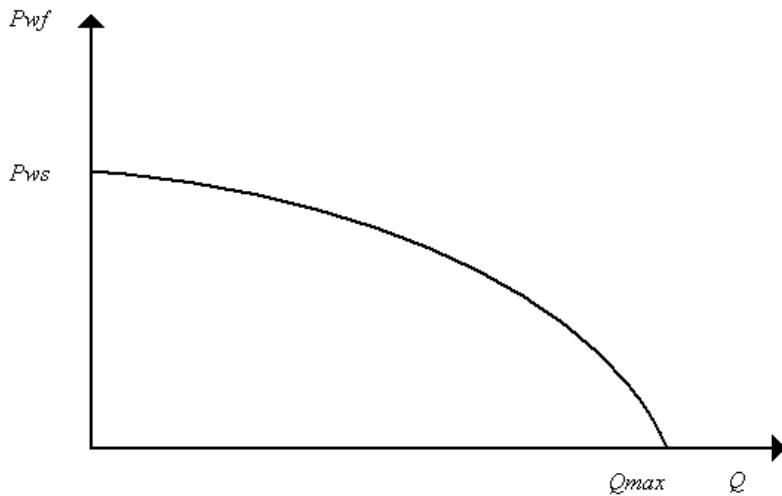
C is the Vogel coefficient.

The Vogel equation has the following properties:

$$Q = Q_{\max} \quad \text{at } p_{wf} = 0$$

$$Q = 0 \quad \text{at } p_{wf} = p_{ws}$$

$$\text{Productivity index } \frac{\partial Q}{\partial p_{wf}} = - \frac{Q_{\max} \cdot (1 + C)}{p_{ws}} \quad \text{at } p_{wf} = p_{ws}$$



Fetkovich's Equation

Fetkovich's equation is one of a number of methods that can be used to specify the [Inflow Performance Relationship \(p.398\)](#) (IPR) for a completion. The Fetkovich equation is a development of the [Vogel equation \(p.400\)](#) to take account of high velocity effects.

$$Q = Q_{\max} \left(1 - \left(\frac{P_{wf}}{P_{ws}} \right)^2 \right)^n \quad \text{Eq. 4.27}$$

Where

Q is the liquid flow rate (STB/D or m³/d)

Q_{\max} is the absolute open hole flow potential, that is the liquid flow rate when the bottom hole pressure is zero

P_{wf} is the well flowing (or bottom hole) pressure (psia or bara)

P_{ws} is the well static (or reservoir) pressure (psia or bara)

n is the Fetkovich exponent.

Jones' Equation

Jones' equation ([p.588](#)) is one of a number of methods that can be used to specify the [Inflow Performance Relationship \(p.398\)](#) (IPR) for a completion. It is similar to the [PI \(p.399\)](#) method but contains an extra term to model turbulence.

Jones equation for gas inflow

The Jones equation for gas reservoirs is :

$$P_{ws}^2 - P_{wf}^2 = A Q_G^2 + B Q_G \quad \text{Eq. 4.28}$$

where

Q_G is the stock-tank gas rate

p_{ws} is the well static (or reservoir) pressure

p_{wf} is the well flowing (or bottom hole) pressure

$A \geq 0$ is the turbulence coefficient

$B \geq 0$ is the laminar coefficient

In the case when $A = 0$ the Jones equation is the same as the [gas PI \(p.399\)](#) equation with productivity index $J_G = 1 / B$. Values of $B > 0.05$ (psi²/MMscf/d) indicate low permeability or the presence of skin damage .

Jones equation for liquid inflow

Jones proposed the equation for gas flow, but it can also be used to model oil wells. However the [Fetkovich equation \(p.401\)](#) can also be used for saturated oil wells and is the recommended method for IPRs in reservoirs producing below the bubble point.

The Jones equation for liquid reservoirs is :

$$P_{ws} - P_{wf} = A Q_L^2 + B Q_L \quad \text{Eq. 4.29}$$

where

Q_L is the stock-tank oil rate

In the case when $A = 0$ the Jones equation is the same as the [liquid PI \(p.399\)](#) equation with productivity index $J_L = 1 / B$

Forchheimer Equation

[Forchheimer, 1901 \(p.586\)](#) gave an equation for non-Darcy flow in the reservoir, which is essentially the same as the [Jones equation \(p.402\)](#) for liquid inflow.

Back Pressure Equation

The Back Pressure Equation is one of a number of methods that can be used to specify the [Inflow Performance Relationship \(p.398\)](#) (IPR) for a completion.

The Back Pressure Equation was developed by Rawlins and Schellhardt (1935) (p.592) after testing 582 wells. The equation is typically applied to gas wells although its application to oil wells has also been proven. If correlations already exist for oil wells, use the Back Pressure Equation on gas wells only. The equation has the following form:

$$Q_G = C \cdot \left(P_{ws}^2 - P_{wf}^2 \right)^n \quad \text{Eq. 4.30}$$

where

Q_G is the gas flow rate (MMscf/d) (m^3/d),

P_{ws} is the well static (or reservoir) pressure (psia) (bara)

P_{wf} is the well flowing (or bottom hole) pressure (psia) (bara)

C is the back pressure constant (MMscf/d/(psia²)ⁿ) ($\text{m}^3/\text{d}/(\text{bar}^2)^n$)

n is the dimensionless back pressure exponent

The back pressure exponent, n , which ranges between 0.5 and 1.0, accounts for high velocity flow (turbulence). When $n = 1$ the back pressure equation is the same as the [gas PI \(p.399\)](#) equation.

The back pressure constant, C , represents reservoir rock and fluid properties, flow geometry and transient effects.

The parameters C and n must be obtained by [multi-rate testing](#) in addition to the standard IPR equations, test data can be utilized so that the inflow can be matched to actual measured data. A minimum of three data points is required. Two types of multi-rate test are available: of the well.

Since

$$\log Q_G = \log C + n \cdot \log \left(P_{ws}^2 - P_{wf}^2 \right) \quad \text{Eq. 4.31}$$

A plot of flow rate Q_G versus $P_{ws}^2 - P_{wf}^2$ on a log-log scale will give a line with slope n and intercept C . To avoid unit conversion problems when obtaining the parameters, check that the slope has a value between 0.5 and 1.0. If n is less than 0.5, this implies that the reservoir stabilization conditions are slow, or that liquid has accumulated in the wellbore (in gas condensate wells). The value of n can be greater than 1.0 if liquid is removed from the well during testing, or by removing drilling or stimulation fluids. Also, changes in well capacity during isochronal testing will cause a wider scatter of data points. This might be the result of liquid accumulation or cleaning of the wells.

Pseudo Steady State Equation / Darcy Equation

The pseudo steady state IPR (p.398) equation (PSS), is derived from the equation for single phase Darcy flow into a well. A number of versions of the equation can be used (some require [keywords \(p.655\)](#)):

- for liquid flow the PSS equation is written in terms of the [stock tank liquid \(p.406\)](#) flow rate
 - this can be optionally combined with a Vogel formula for pressures [below the bubble point \(p.406\)](#).
 - the liquid flow can be modelled using a two phase version of the radial flow equations for [oil and water \(p.406\)](#)
- for gas flow the PSS is written in terms of the [stock tank gas \(p.407\)](#) flow rate
 - a version using the [gas pseudo pressure \(p.408\)](#) (more accurate for high pressure systems).
- the PSS expressed in terms of [reservoir flow \(p.404\)](#) rates can be used for either liquid or gas flow.
 - the liquid flow can be modelled using a two phase version of the reservoir flow equations for [oil and water \(p.406\)](#)

Reservoir flow

The pseudo steady state equation, like the [transient IPR \(p.408\)](#), is calculated by solving the radial, single phase, Darcy flow into a well. It applies for relatively long times, after the well has passed through the transient stage. The solution is given by [Dake 1978 \(p.585\)](#):

$$Q_{R\Phi} = M_\Phi \cdot T \cdot (P_{ws} - P_{wf}) \quad \text{Eq. 4.32}$$

where the PSS transmissibility term is defined by:

$$T = \frac{2\pi kh}{C_1 \left[\ln \left(\frac{r_e}{r_w} \right) - 0.75 + S \right]} \quad \text{Eq. 4.33}$$

Here:

$Q_{R\Phi}$	is the volume flow rate at reservoir conditions of phase Φ	RB/d or MCF/d
M_Φ	is the mobility of phase Φ	$1/cp$
p_{ws}	is the average reservoir pressure	$psia$
p_{wf}	is the bottom hole pressure	$psia$
k	is the formation permeability	mD
h	is the formation thickness	ft
r_w	is the wellbore radius	ft
r_e	is the drainage radius	ft

S

is the skin

 C_1 is a conversion factor depending
on the flow units

$$C_1 = \frac{14.7 \cdot 0.3048^2 \cdot 5.61458 \cdot 10^{-3}}{86400 \cdot 10^{-10}} = 2\pi \cdot 141.2$$

If the flow is in RB/d

$$C_1 = \frac{14.7 \cdot 0.3048^2}{86400 \cdot 10^{-10}}$$

If the flow is in
 MCF/d

Note: The constant 0.75 comes from using the average reservoir pressure $p_{ws} = \bar{p}$. A similar formula can be derived using the pressure at the drainage radius $p_{ws} = p(r_e)$, but the value 0.75 is replaced by 0.5.

The phase mobility is defined in terms of the phase relative permeability and viscosity:

$$M_\Phi = \frac{k_{r\Phi}}{\mu_\Phi} \quad \text{Eq. 4.34}$$

$k_{r\Phi}$ is the relative permeability for phase Φ

μ_Φ is the viscosity of phase Φ at reservoir conditions cp

For single phase flow the relative permeability is $k_{r\Phi} = 1$, and the inflow equation simplifies to :

$$Q_{R\Phi} = \frac{1}{\mu_\Phi} \cdot T \cdot (P_{ws} - P_{wf}) \quad \text{Eq. 4.35}$$

This version of the PSS IPR can be used for liquid or gas inflow.

For multiphase inflow, the total inflow can be written as the sum of the phase inflows:

$$Q_R = Q_{RO} + Q_{RW} + Q_{RG} \quad \text{Eq. 4.36}$$

This can be rearranged to give:

$$Q_R = M \cdot T \cdot (P_{ws} - P_{wf}) \quad \text{Eq. 4.37}$$

Where the total mobility is defined by

$$M = M_O + M_W + M_G \quad \text{Eq. 4.38}$$

Oil and water flow

A two phase version of the multiphase inflow equation can be used to model liquid inflow.

$$Q_{RL} = \left(\frac{k_O}{\mu_O} + \frac{k_W}{\mu_W} \right) \cdot T \cdot (P_{ws} - P_{wf}) \quad \text{Eq. 4.39}$$

The relative permeabilities can be determined from [permeability tables \(p.441\)](#).

Injection

The reservoir injection flow equation is similar to the PSS production IPR:

$$Q_R = M_I \cdot T \cdot (P_{wf} - P_{ws}) \quad \text{Eq. 4.40}$$

Here however, the mobility term represents the mobility of the fluid in the reservoir, rather than that of the injection fluid, and must be specified. In production, the fluid being produced is the same as that moving through the reservoir. In injection systems the two fluids may be different. For example, we would expect different flow rates if gas is injected into a liquid filled reservoir or a gas filled reservoir. If the injection fluid does differ from the reservoir fluid, then the injection mobility will change with time, as the reservoir fluid changes.

Stock tank liquid flow

The pseudo steady state equation can be expressed in stock tank flow rates. For liquid flow, the stock tank flow rate $Q_L = Q_R / B_L$ is given by

$$Q_L = \frac{2\pi kh(p_{ws} - p_{wf})}{C_1 \mu_L B_L \left[\ln\left(\frac{r_e}{r_w}\right) - 0.75 + S \right]} \quad \text{Eq. 4.41}$$

Q_L is the liquid flowrate STB/d

B_L is the liquid volume formation factor RB/STB

μ_L is the liquid viscosity at reservoir conditions cp

Below bubble point correction

The pseudo steady state equation can be combined with a [Vogel equation \(p.400\)](#) to model inflows when the bottom hole pressure is below the bubble point, see, [Bubble point correction. \(p.414\)](#)

Oil and water flow

The [two phase version \(p.406\)](#) of the reservoir liquid flow equation can also be written in terms of stock tank liquid flow rate:

$$Q_L = Q_R / B_L = Q_{RO} / B_L + Q_{RW} / B_L \quad \text{Eq. 4.42}$$

Stock tank gas flow

This pseudo steady state equation can be expressed in stock tank flow rates. For gas flow, the formation volume factor can be expressed in terms of pressure and temperature

$B_G = \frac{V}{V_s} = \frac{ZRT}{P} \cdot \frac{P_s}{Z_s RT_s}$. The reservoir pressure is taken to be the average pressure in the

reservoir: $P = \frac{P_{ws} + P_{wf}}{2}$, which gives a stock tank flow rate $Q_G = Q_R / B_G$:

$$Q_G = \frac{2\pi kh(p_{ws}^2 - p_{wf}^2)}{C_2 \mu_G TZ \left[\ln\left(\frac{r_e}{r_w}\right) - 0.75 + S + DQ_G \right]} \quad \text{Eq. 4.43}$$

- The quadratic term in pressure arises from a combination of the pressure difference and the reservoir average pressure term

$$(p_{ws}^2 - p_{wf}^2) = (p_{ws} - p_{wf}) \cdot (p_{ws} + p_{wf}).$$
- The constant term arises from a combination of the conversion factor and stock tank properties

$$C_2 = \frac{2C_1 Z_s P_s}{T_s}.$$

- The skin has been modified to include a flow rate dependent term.

Q_G	is the gas flowrate	<i>MSCF/d</i>
B_G	is the gas volume formation factor	<i>CF/SCF</i>
μ_G	is the gas viscosity at reservoir conditions	<i>cp</i>
S	is the constant skin	
DQ	is the near wellbore turbulence factor or rate dependent skin	
T	is the reservoir temperature	$^{\circ}R$
Z	is the reservoir compressibility factor	
P_s	is the stock tank pressure	14.7 <i>psi</i>
T_s	is the stock tank temperature	519.67 $^{\circ}R$

Z_s

is the stock tank compressibility factor

1

$$C_2 = \frac{2 \cdot 14.7^2 \cdot 0.3048^2}{86400 \cdot 10^{-10} \cdot 519.67} = 2\pi \cdot 1422$$

is a constant, arising from conversion factors
and stock tank properties

Gas pseudo pressure

Dake 1978 (p.585) gives another version of the Pseudo steady state IPR for gas inflow, that is more accurate for large drawdowns, based on work by Al-Hussainy et al (p.583):

$$Q_G = \frac{2\pi kh[m(p_{ws}) - m(p_{wf})]}{C_2 T \left[\ln\left(\frac{r_e}{r_w}\right) - 0.75 + S + DQ_G \right]} \quad \text{Eq. 4.44}$$

Here the gas pseudo pressure is given by:

$$m(p) = 2 \int \frac{p}{\mu_G Z} d p \quad \text{Eq. 4.45}$$

Transient IPR

The transient IPR (p.398) equation, is derived from the equation for single phase Darcy flow into a well. A number of versions of the equation can be used (some require keywords (p.671)):

- for liquid flow the transient IPR is written in terms of the stock tank liquid (p.411) flow rate
 - this can be optionally combined with a Vogel formula for pressures below the bubble point (p.411).
 - the liquid flow can be modelled using a two phase version of the radial flow equations for oil and water (p.411)
- for gas flow the transient IPR is written in terms of the stock tank gas (p.412) flow rate
 - a version using the gas pseudo pressure (p.413) (more accurate for high pressure systems).
- the transient IPR expressed in terms of reservoir flow (p.408) rates can be used for either liquid or gas flow.
 - the liquid flow can be modelled using a two phase version of the reservoir flow equations for oil and water (p.410)

Reservoir flow

The transient IPR, like the pseudo steady state IPR (p.403), is calculated by solving the radial, single phase, Darcy flow into the well. It applies for relatively small times, before the well has reached the pseudo steady state. A similarity solution is given by Dake 1978 (p.585):

$$Q_{R\Phi} = M_\Phi \cdot T \cdot (P_{ws} - P_{wf}) \quad \text{Eq. 4.46}$$

where the transient IPR transmissibility is defined by:

$$T = \frac{2\pi kh}{C_1 \left[\frac{1}{2} \ln \left(\frac{4kt}{C_0 \gamma \theta \mu C r_w^2} \right) + S \right]} \quad \text{Eq. 4.47}$$

$Q_{R\Phi}$	is the volume flow rate at reservoir conditions of phase Φ	RB/d or MCF/d
M_Φ	is the mobility of phase Φ	$1/cp$
p_{ws}	is the average reservoir pressure	$psia$
p_{wf}	is the bottom hole pressure	$psia$
t	time	<i>hours</i>
k	is the formation permeability	mD
h	is the formation thickness	ft
r_w	is the wellbore radius	ft
S	is the skin	
θ	is the reservoir porosity	
C	is the total compressibility of the reservoir and the reservoir fluids	$1/psi$
γ	is a constant equal to the exponential of Euler's constant	$\gamma = e^{0.5772} = 1.781$
C_0	is a conversion factor	$C_0 = \frac{14.7 \cdot 0.3048^2 \cdot 10^{-3}}{10^{-10} \cdot 3600}$
C_1	is a conversion factor depending on the flow units	

$$C_1 = \frac{14.7 \cdot 0.3048^2 \cdot 5.61458 \cdot 10^{-3}}{86400 \cdot 10^{-10}} = 2\pi \cdot 141.2 \text{ RB/d}$$

If the flow is in
RB/d

$$C_1 = \frac{14.7 \cdot 0.3048^2}{86400 \cdot 10^{-10}}$$

If the flow is in
MCF/d

The transient IPR equation can be written in similar terms to the [pseudo steady state IPR \(p.403\)](#) by defining a radius

$$r^2 = \frac{4kt}{C_0 \gamma \theta \mu C} \quad \text{Eq. 4.48}$$

$$T = \frac{2\pi kh}{C_1 \left[\ln \left(\frac{r}{r_w} \right) + S \right]} \quad \text{Eq. 4.49}$$

The phase mobility is defined in terms of the phase relative permeability and viscosity:

$$M_\Phi = \frac{k_{r\Phi}}{\mu_\Phi} \quad \text{Eq. 4.50}$$

$k_{r\Phi}$ is the relative permeability for phase Φ

μ_Φ is the viscosity of phase Φ at reservoir conditions cp

For single phase flow the relative permeability is $k_{r\Phi} = 1$, and the inflow equation simplifies to :

$$Q_{R\Phi} = \frac{1}{\mu_\Phi} \cdot T \cdot (P_{ws} - P_{wf}) \quad \text{Eq. 4.51}$$

This version of the transient IPR can be used for liquid or gas inflow.

For multiphase inflow, the total inflow can be written as the sum of the phase inflows:

$$Q_R = Q_{RO} + Q_{RW} + Q_{RG} \quad \text{Eq. 4.52}$$

This can be rearranged to give:

$$Q_R = M \cdot T \cdot (P_{ws} - P_{wf}) \quad \text{Eq. 4.53}$$

Where the total mobility is defined by

$$M = M_O + M_W + M_G \quad \text{Eq. 4.54}$$

Oil and water flow

A two phase version of the multiphase inflow equation can be used to model liquid inflow.

$$Q_{RL} = \left(\frac{k_{rO}}{\mu_O} + \frac{k_{rW}}{\mu_W} \right) \cdot T \cdot (P_{ws} - P_{wf}) \quad \text{Eq. 4.55}$$

The relative permeabilities can be determined from [permeability tables \(p.441\)](#).

Stock tank liquid flow

This transient IPR equation can be expressed in stock tank flow rates. For liquid flow, the stock tank flow rate $Q_L = Q_R / B_L$ is given by

$$Q_L = \frac{2\pi kh(p_{ws} - p_{wf})}{C_1 \mu_L B_L \left[\frac{1}{2} \ln \left(\frac{4kt}{C_0 \gamma \theta \mu_L Cr_w^2} \right) + S \right]} \quad \text{Eq. 4.56}$$

Q_L is the liquid flowrate STB/d

B_L is the liquid volume formation factor RB/STB

μ_L is the liquid viscosity cp

The equation can be written using base 10 logarithms, since

$$\ln \frac{4x}{C_0 \gamma} = \ln 10 \cdot \log \frac{4x}{C_0 \gamma} = 2.302 \cdot (\log x + \log \frac{4}{C_0 \gamma}) = 2.302 \cdot (\log x - 3.23):$$

$$Q_L = \frac{2\pi kh(p_{ws} - p_{wf})}{1.151 \cdot C_1 \mu_L B_L \left[\log \left(\frac{kt}{\theta \mu_L Cr_w^2} \right) - 3.23 + \frac{S}{1.151} \right]} \quad \text{Eq. 4.57}$$

Below bubble point correction

The transient IPR equation can be combined with a [Vogel equation \(p.400\)](#) to model inflows when the bottom hole pressure is below the bubble point, see, [Bubble point correction. \(p.414\)](#)

Oil and water flow

The [two phase version \(p.410\)](#) of the reservoir liquid flow equation can also be written in terms of stock tank liquid flow rate:

$$Q_L = Q_R / B_L = Q_{RO} / B_L + Q_{RW} / B_L \quad \text{Eq. 4.58}$$

Stock tank gas flow

This transient IPR equation can be expressed in stock tank if $B_G = \frac{V}{V_s} = \frac{ZRT}{P} \cdot \frac{P_s}{Z_s RT_s}$ low

rates. For gas flow, the formation volume factor can be expressed in terms of pressure and temperature: . The reservoir pressure is taken to be the average pressure in the reservoir:

$P = \frac{p_{ws} + p_{wf}}{2}$, which gives a stock tank flow rate $Q_G = Q_R / B_G$:

$$Q_G = \frac{2\pi kh(p_{ws}^2 - p_{wf}^2)}{C_2 \mu_G TZ \left[\frac{1}{2} \ln \left(\frac{4kt}{C_0 \gamma \theta \mu_G Cr_w^2} \right) + S + DQ_G \right]} \quad \text{Eq. 4.59}$$

- The quadratic term in pressure arises from a combination of the pressure difference and the average pressure term $(p_{ws}^2 - p_{wf}^2) = (p_{ws} - p_{wf}) \cdot (p_{ws} + p_{wf})$.
- The constant term arises from a combination of the conversion factor and stock tank properties

$$C_2 = \frac{2C_1 Z_s P_s}{T_s}$$

- The skin has been modified to include a flow rate dependent term.

Q_G is the gas flowrate $MSCF/d$

B_G is the gas volume formation factor CF/SCF

μ_G is the gas viscosity cp

S is the constant skin

DQ is the near wellbore turbulence factor or rate dependent skin

T is the reservoir temperature ${}^{\circ}R$

Z is the reservoir compressibility factor

P_s is the stock tank pressure 14.7 psi

T_s is the stock tank temperature $519.67 {}^{\circ}R$

Z_s is the stock tank compressibility factor 1

C_2 is a constant, arising from conversion factors and stock tank properties

For $MSCF/d$:

$$C_2 = \frac{2 \cdot 14.7^2 \cdot 0.3048^2}{86400 \cdot 10^{-10} \cdot 519.67} = 2\pi \cdot 1422$$

The equation can also be written using base 10 logarithms:

$$Q_G = \frac{2\pi kh(p_{ws}^2 - p_{wf}^2)}{1.151 \cdot C_2 \mu_G TZ [\log(\frac{kt}{\theta \mu_G Cr_w^2}) - 3.23 + \frac{S + DQ_G}{1.151}]} \quad \text{Eq. 4.60}$$

Gas pseudo pressure

Dake 1978 (p.585) gives another version of the Pseudo steady state IPR for gas inflow, that is more accurate for large drawdowns, based on work by Al-Hussainy et al (p.583):

$$Q_G = \frac{2\pi kh[m(p_{ws}) - m(p_{wf})]}{C_2 T [\frac{1}{2} \ln(\frac{4kt}{C_0 \gamma \theta \mu_G Cr_w^2}) + S + DQ_G]} \quad \text{Eq. 4.61}$$

Here the gas pseudo pressure is given by:

$$m(p) = 2 \int \frac{p}{\mu_G Z} d p \quad \text{Eq. 4.62}$$

Time to pseudo steady state solution

According to Dake 1978 (p.585), the solution to the well inflow equations changes from transient to pseudo steady state (p.403) when the dimensionless time t_{DA} is given by

$$t_{DA} = \frac{kt}{C_0 \theta \mu CA} > 0.1 \quad \text{Eq. 4.63}$$

Writing $A = \pi r_e^2$, where r_e is the drainage radius of the reservoir, the time when the pseudo steady state (p.403) solution becomes applicable is

$$t_{pss} = (0.1 \cdot \pi \cdot C_0) \cdot \frac{\theta \mu C r_e^2}{k} \quad \text{Eq. 4.64}$$

A warning will be issued if the time t exceeds this value.

Data File

Enter an IPR in table form (Flowrate versus Pressure) rather than using an IPR (p.398) equation. This feature is currently only available by using an EKT (p.94) or in expert mode (p.264).

Place the EKT (Spanner icon) between the completion and the tubing and enter the [IFPTAB](#) ([p.662](#)) data.

Example:

```
! n liq pwf gor wcut
ifptab 0 0      3000 986  0
ifptab 0 1000   2990 986  2.0
ifptab 0 2699   2920 1096 2.2
ifptab 0 6329   2800 2540 2.8
ifptab 0 7288   2600 2980 3.9
ifptab 0 8082   2400 3370 5.6
ifptab 0 8805   2003 3770 8.0
ifptab execute
```

Note: The GOR and water cut values are optional.

All IPR data associated with the completion will be ignored.

Bubble Point Correction

The [Productivity Index](#) ([p.399](#)), [Pseudo steady state](#) ([p.403](#)) and [Transient](#) ([p.408](#)) IPRs for liquid inflow can be modified to use a form of [Vogel's equation](#) ([p.400](#)) below the bubble point ($p_{wf} < p_{bp}$). This allows the effects of gas break-out to be modelled.

$$Q - Q_{bp} = Q_{\max} \left(1 - (1 - C) \left(\frac{p_{wf}}{p_{bp}} \right) - C \left(\frac{p_{wf}}{p_{bp}} \right)^2 \right) \quad \text{Eq. 4.65}$$

Where

Q is the liquid flow rate (STB/D or m³/d)

Q_{bp} is the flow at the bubble point flow

$Q_{\max} = \frac{Q_{bp}}{1 + C} \cdot \frac{P_{bp}}{P_{ws} - P_{bp}}$ is the absolute open hole flow potential, that is the liquid flow rate when the bottom hole pressure is zero

p_{bp} is the bubble point pressure (psia or bara)

p_{wf} is the well flowing (or bottom hole) pressure (psia or bara)

p_{ws} is the well static (or reservoir) pressure (psia or bara)

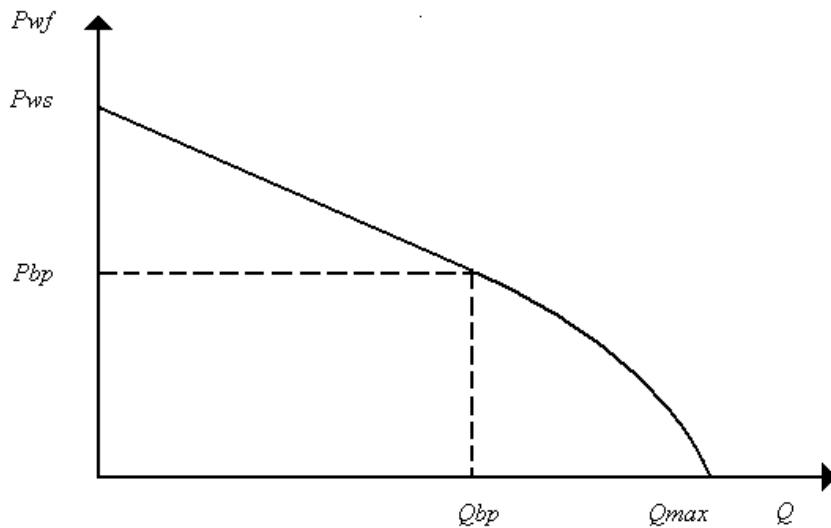
C is the Vogel coefficient.

The Vogel equation has been shifted to match a linear IPR above the bubble point:

$$Q = Q_{bp} \quad \text{at } p_{wf} = p_{bp}$$

$$\text{Productivity index } \frac{\partial Q}{\partial p_{wf}} = - \frac{Q_{\max} \cdot (1 + C)}{p_{bp}} = - \frac{Q_{bp}}{p_{ws} - p_{bp}} \quad \text{at } p_{wf} = p_{bp}$$

This correction only works if the bubble point pressure is less than the static (reservoir) pressure,
 $p_{bp} < p_{ws}$.



Vertical Well Skin Factor

The skin factor S is used in the [pseudo steady state \(p.403\)](#) and [transient \(p.408\)](#) IPRs to represent friction caused by damage to the formation close to the well (mechanical skin) and the effects of high flow (dynamic skin).

$$S = S_M + D \cdot Q \quad \text{Eq. 4.66}$$

Mechanical skin factor

The [pseudo steady state \(p.403\)](#) and [transient \(p.408\)](#) IPRs are derived from Darcy's equation for a homogenous reservoir with a vertical completion. The mechanical skin can be used to represent friction terms arising from any departure from this idealized model. The mechanical skin has a number of separate components:

$$S_M = S_{pp} + S_\theta + S_d + S_g + S_p + S_f \quad \text{Eq. 4.67}$$

Different components are used in different completion types:

		Open hole	Open hole gravel pack	Perforated	Gravel Packed & Perforated	Frac Pack
--	--	-----------	-----------------------	------------	----------------------------	-----------

S_{pp}	partial penetration (p.418)	Yes	Yes	Yes	Yes	Yes
S_θ	deviation (p.418)	Yes	Yes	Yes	Yes	Yes
S_d	damaged zone (p.419)	Yes	Yes	Yes	Yes	
S_g	gravel pack (p.419)		Yes		Yes	Included in S_f
S_p	perforated well (p.420)			Yes	Yes	Included in S_f
S_f	frac pack (p.424)					Yes

Dynamic skin factor

The [pseudo steady state \(p.403\)](#) and [transient \(p.408\)](#) IPRs are derived from Darcy's equation for a homogenous reservoir with a vertical completion. The dynamic skin can be used to represent friction terms arising from turbulence in the flow entering the well. The dynamic skin has a number of separate components:

$$D = D_d + D_r + D_g + D_c + D_f \quad \text{Eq. 4.68}$$

Different components are used in different completion types:

		Open hole	Open hole gravel pack	Perforated	Gravel Packed & Perforated	Frac Pack
D_d	damaged zone	Yes	Yes	Yes	Yes	
D_r	reservoir	Yes	Yes	Yes	Yes	
D_s	gravel pack screen		Yes			
D_g	gravel pack				Yes	
D_c	crushed zone			Yes	Yes	
D_f	frac pack					Yes

Formulas for these skin components can be found in [Golan and Whitson \(1986\) \(p.587\)](#). The damaged zone, reservoir and gravel pack screen dynamic skins all use the same formula D_G for gas flow and the same formula for liquid flow D_L :

$$D_d = \begin{cases} D_G(r_w, r_d, \beta_d) \\ D_L(r_w, r_d, \beta_d) \end{cases} \quad \text{Eq. 4.69}$$

$$D_r = \begin{cases} D_G(r_d, r_r, \beta_r) \\ D_L(r_d, r_r, \beta_r) \end{cases} \quad \text{Eq. 4.70}$$

$$D_s = \begin{cases} D_G(r_s, r_w, \beta_s) \\ D_L(r_s, r_w, \beta_s) \end{cases} \quad \text{Eq. 4.71}$$

The general gas flow dynamic skin term is given by:

$$D_G(r_{in}, r_{out}, \beta) = 2.222 \cdot 10^{-18} \cdot \beta \cdot \frac{k \cdot h \cdot \gamma_G}{h_c^2 \cdot \mu_G} \cdot \left(\frac{1}{r_{in}} - \frac{1}{r_{out}} \right) \quad \text{Eq. 4.72}$$

The general liquid flow dynamic skin term is given by:

$$D_L(r_{in}, r_{out}, \beta) = 1.635 \cdot 10^{-16} \cdot \beta \cdot \frac{k \cdot h \cdot \rho_L \cdot B_L}{h_c^2 \cdot \mu_L} \cdot \left(\frac{1}{r_{in}} - \frac{1}{r_{out}} \right) \quad \text{Eq. 4.73}$$

The gravel pack dynamic skin for gas flow is given by:

$$D_g = 2.45 \cdot 10^{-13} \cdot \beta_g \cdot \frac{k \cdot h \cdot \gamma_G \cdot L_{pack}}{(2r_p)^4 \cdot n_p^2 \cdot \mu_G} \quad \text{Eq. 4.74}$$

The gravel pack dynamic skin for liquid flow is given by:

$$D_g = 1.8 \cdot 10^{-11} \cdot \beta_g \cdot \frac{k \cdot h \cdot \rho_L \cdot B_L \cdot L_{pack}}{(2r_p)^4 \cdot n_p^2 \cdot \mu_L} \quad \text{Eq. 4.75}$$

The crushed zone dynamic skin for gas flow is given by:

$$D_c = 3.84 \cdot 10^{-15} \cdot \beta_c \cdot \frac{k \cdot h \cdot \gamma_G}{L_p^2 \cdot n_p^2 \cdot r_p \cdot \mu_G} \quad \text{Eq. 4.76}$$

The crushed zone dynamic skin for liquid flow is given by:

$$D_c = 0 \quad \text{Eq. 4.77}$$

The frac pack dynamic skin is given as the sum of the tunnel and annulus terms:

$$D_f = D_t + D_a \quad \text{Eq. 4.78}$$

The frac pack dynamic skin annulus term uses the same general formula as used for the damaged zone, reservoir and gravel pack screens:

$$D_a = \begin{cases} D_G(r_s, r_c, \beta_s) \\ D_L(r_s, r_c, \beta_s) \end{cases} \quad \text{Eq. 4.79}$$

The frac pack dynamic skin tunnel term for gas flow:

$$D_t = 2.222 \cdot 10^{-18} \cdot \beta_s \cdot 4 \cdot \frac{k \cdot h \cdot \gamma_G \cdot L_{tun}}{\left(\frac{r_p}{12}\right)^4 \cdot den_{shot}^2 \cdot \mu_G} \quad \text{Eq. 4.80}$$

The frac pack dynamic skin tunnel term for liquid flow:

$$D_t = 1.635 \cdot 10^{-16} \cdot \beta_s \cdot 4 \cdot \frac{k \cdot h \cdot \rho_L \cdot B_L \cdot L_{tun}}{\left(\frac{r_p}{12}\right)^4 \cdot den_{shot}^2 \cdot \mu_L} \quad \text{Eq. 4.81}$$

Partial Penetration Skin

[Brons and Marting \(p.584\)](#) (1961) (quoted in [Golan and Whitson \(p.587\)](#)) expressed the effect of partial penetration and limited entry as a skin factor :

$$S_{pp} = \left(\frac{h}{L} - 1 \right) \left[\ln \left(\frac{h}{r_w} \sqrt{\frac{k_r}{k_z}} \right) - Y \right] \quad \text{Eq. 4.82}$$

with

$$Y = 2.948 - 7.363X + 11.45X^2 - 4.675X^3 \quad \text{and} \quad X = \frac{L}{h}$$

h Reservoir Thickness

r_w Wellbore Radius

k_r Reservoir Permeability

k_z Completion Vertical Permeability

L Completion Open Interval or Perforated Interval

The skin factor is dimensionless. The equation for the skin factor involves ratios of permeability and ratios of length. It is assumed the same units (e.g. md) are used for all permeability terms, and the same units are used for all lengths (e.g. feet).

Deviation Skin

[Cinco et al. \(p.584\)](#) (1975) approximate the pseudo-skin factor caused by the slant of a well as :

$$S_\theta = - \left(\frac{\theta'}{41} \right)^{2.06} - \left(\frac{\theta'}{57} \right)^{1.865} \log_{10} \left(\frac{h}{100r_w} \sqrt{\frac{k_r}{k_z}} \right) \quad \text{Eq. 4.83}$$

θ' is measured in degrees:

$$\tan(\theta') = \sqrt{\frac{k_z}{k_r}} \cdot \tan(\theta) \quad \text{Eq. 4.84}$$

h Reservoir Thickness

r_w Wellbore Radius

k_r Reservoir Permeability

k_z Completion Vertical Permeability

$0 < \theta < 75^\circ$ deviation from vertical in degrees

The skin factor is dimensionless. The equation for the skin factor involves ratios of permeability and ratios of length. It is assumed the same units (e.g. md) are used for all permeability terms, and the same units are used for all lengths (e.g. feet).

Damaged Zone Skin

The effect of formation damage on productivity was treated analytically by [Muskat \(p.590\)](#) (1937). [Hawkins \(p.587\)](#) (1956) translated the Muskat Model of a near wellbore altered permeability into the following expression for the skin factor :

$$S_d = \left(\frac{k_r}{k_a} - 1 \right) \ln \left(\frac{d_a}{d_w} \right) \quad \text{Eq. 4.85}$$

d_a Damaged Zone Diameter

d_w Wellbore Diameter

k_r Reservoir Permeability

k_a Damaged Zone Permeability

The skin factor is dimensionless. The equation for the skin factor involves ratios of permeability and ratios of length. It is assumed the same units (e.g. md) are used for all permeability terms, and the same units are used for all lengths (e.g. feet).

Gravel Pack Skin

Two different formula are used in PIPESIM. The skin factor is dimensionless. The equations for the skin factor involves ratios of permeability and ratios of length. It is assumed the same units (e.g. md) are used for all permeability terms, and the same units are used for all lengths (e.g. feet).

Open hole gravel pack skin

Assuming a radial flow through formation and gravel, the contribution to the skin is expressed as:

$$S_g = \frac{k_r}{k_g} \cdot \ln\left(\frac{r_{ev}}{r_s}\right) \quad \text{Eq. 4.86}$$

r_{ev} Reservoir Drainage Radius

r_s Screen Radius

k_r Reservoir Permeability

k_g Gravel Permeability

Compare this with the [Annulus skin \(p.426\)](#) for a Frac Pack.

Gravel pack skin

Applying Darcy's law for linear flow in packed perforations for the steady state skin term due to gravel pack gives :

$$S_g = 2 \cdot \frac{k_r}{k_g} \cdot \frac{h \cdot I_t^{'}}{n \cdot r_p^2} \quad \text{Eq. 4.87}$$

where

$$I_t^{'} = I_t + r_{ic} - r_s$$

h Reservoir Thickness

I_t Tunnel Length

r_{ic} Casing Internal Radius

r_p Perforation Radius

Compare this with the [Gravel skin \(p.426\)](#) for a Frac Pack.

Perforated Well Skin

McLeod Model

McLeod ([p.420](#)) (1983) used a model of a "horizontal microwell" with formation damage around it as an analogy to a perforation surrounded by a crushed zone. He quantified the effect of the crushed zone as the following skin factor :

Compacted or crushed zone

$$S_p = \frac{1}{n_p \cdot L} \cdot \frac{h}{I_p} \cdot \left(\frac{k_r}{k_c} - \frac{k_r}{k_a} \right) \cdot \ln \left(\frac{d_c}{d_p} \right) \quad \text{Eq. 4.88}$$

where:

h Reservoir Thickness

L Completion Open Interval or Perforated Interval

n_p Perforation Density

I_p Depth of Penetration (or perforation length)

d_c Compacted Zone Diameter

d_p Perforation Diameter

k_r Reservoir Permeability

k_c Compacted Zone Permeability

k_a Damaged Zone Permeability

Karakas Model

Karakas and Tariq (p.588) (1991) have developed a semi analytical solution for the calculation of the perforation skin effect. Depending on the size of the damaged zone and the length of the perforation , the well radius and the perforation length, or their modified value are used in the model .

The thickness of the damaged zone is :

$$I_a = \frac{1}{2}(d_a - d_w) \quad \text{Eq. 4.89}$$

For perforation sitting inside the damaged zone :

$$\begin{cases} I'_w = I_w \\ I'_w = I_p \end{cases} \text{if } I_p \leq I_a \quad \text{Eq. 4.90}$$

For perforations extending beyond the damaged zone

$$\left. \begin{aligned} r'_w &= r_w + \left(1 - \frac{k_a}{k_r}\right) I_a \\ I'_p &= I_p - \left(1 - \frac{k_a}{k_r}\right) I_a \end{aligned} \right\} \text{if } I_p > I_a \quad \text{Eq. 4.91}$$

r_w' Wellbore Radius ($d_w/2$)

r'_w Wellbore Radius or modified Wellbore Radius

I'_p Perforation Length or modified Perforation Length

The perforation skin effect is divided into the following components :

Horizontal Component of the skin

$$S_h = \ln \left(\frac{r'_w}{r'_{wc}} \right) \quad \text{Eq. 4.92}$$

where

$$r'_{wc} = \alpha(\varphi) [r'_w + I'_p] \quad \text{Eq. 4.93}$$

φ Phase Angle

$\alpha(\varphi)$ Function of phase angle φ (see table 4.1 (p.423))

Well bore skin

$$S_w = c_1 \exp \left[c_2 \frac{r'_w}{(I'_p + r'_w)} \right] \quad \text{Eq. 4.94}$$

with

$c_1 = c_1(\varphi)$ and $c_2 = c_2(\varphi)$ Functions of the phase angle φ (see table 4.1 (p.423))

Vertical skin

$$S_v = 10^A H_n^{B-1} R_n^B \quad \text{Eq. 4.95}$$

with

$$H_n = \frac{1}{n_p I'_p} \sqrt{\frac{k_r}{k_z}}$$

$$R_n = \frac{n_p d_p}{4} \left(1 + \sqrt{\frac{k_z}{k_r}} \right)$$

$$A = a_1(\varphi) \times \log_{10} R_n + a_2(\varphi)$$

$$B = b_1(\varphi)R_n + b_2(\varphi)$$

$a_1(\varphi)$, $a_2(\varphi)$, $b_1(\varphi)$ and $b_2(\varphi)$ functions of phase angle φ (see table 4.1 (p.423))

The equation 4.95 (p.422) is valid for $H_n \leq 10$ and $R_n \geq 0.01$

Crushed zone effect

$$S_{ck} = \frac{1}{n_p I_p} \left(\frac{k_r}{k_c} - 1 \right) \ln \left(\frac{d_c}{d_p} \right) \quad \text{Eq. 4.96}$$

The combined skin effect caused by perforations added to the crushed zone effects is given by :

$$S'_t = S_h + S_w + S_v + S_{ck} \quad \text{Eq. 4.97}$$

If the perforations go beyond the damaged zone, the total perforation skin is the sum of these four contributions :

$$S_t = S'_t \quad \text{for } I_p > I_a \quad \text{Eq. 4.98}$$

Damaged zone effect

For perforations within damaged zone, the skin caused by the combined effects of perforations and damage is :

$$S_t = \left(\frac{k_r}{k_a} - 1 \right) \ln \left(\frac{d_a}{d_w} \right) + \frac{k_r}{k_a} (S'_t + S_x) \quad \text{for } I_p \leq I_a \quad \text{Eq. 4.99}$$

$$S_x = S_x(r) \quad \text{and} \quad r = \frac{d_a}{d_w + 2I_p}$$

r Ratio of the damaged zone diameter over the penetration zone diameter

$S_x(r)$ function of r (see table 4.2 (p.424))

φ (degree)	45	60	90	120	180	360 (0)
α	0.860	0.813	0.726	0.648	0.500	0.250
a_1	-1.788	-1.898	-1.905	-2.018	-2.025	-2.091
a_2	0.2398	0.1023	0.1038	0.0634	0.0943	0.0453
b_1	1.1915	1.3654	1.5674	1.6136	3.0373	5.1313
b_2	1.6392	1.6490	1.6935	1.7770	1.8115	1.8672

c_1	4.6×10^{-5}	3.0×10^{-4}	1.9×10^{-3}	6.6×10^{-3}	2.6×10^{-2}	1.6×10^{-1}
c_2	8.791	7.509	6.155	5.320	4.532	2.675

Table 4.1: Karakas and Tariq Skin Correlation Coefficients

$r = d_a / (d_w + 2l_p)$	S_x
18.0	0.000
10.0	-0.001
2.0	-0.002
1.5	-0.024
1.2	-0.085

Table 4.2: Skin caused by boundary effect, 180 degree phasing

Frac Pack Skin

The Frac Pack Skin is calculated only in association with a case hole gravel pack. If the gravel pack is not defined the Frac Pack Skin is 0.

[Pucknell and Mason \(p.592\)](#) (1992) give a review of the contributions to the skin in a cased hole gravel pack completion.

$$S_f = S_{hf} + S_{ff} + S_{cf} + S_{an} + S_{tg}$$

S_{hf}	Hydraulic fracture (p.424)
S_{ff}	Fracture face skin (p.425)
S_{cf}	Choke fracture skin (p.426)
S_{an}	Annulus skin (p.426)
S_{tg}	Tunnel gravel skin (p.426)

The skin factor is dimensionless. The equations for the skin factor involves ratios of permeability and ratios of length. It is assumed the same units (e.g. md) are used for all permeability terms, and the same units are used for all lengths (e.g. feet).

Hydraulic fracture

The following model is also applied in IPR Model “Hydraulic Fracture”.

Hydraulic fracture is characterized by its length, capacity or conductivity and related equivalent skin effect. [Prats \(p.592\)](#) (1961) introduced the concept of effective wellbore radius, providing pressure profiles in a fractured reservoir as functions of the fracture half-length and a relative capacity. He provided a graph relating the effective well radius and the capacity. [Cinco-Ley et al. \(p.584\)](#) (1978, 1981a) (see also [Economides et al. \(p.586\)](#) 1994) introduced the fracture conductivity instead,

which is proportional to the inverse of the capacity, and provided an alternative graph relating the fracture conductivity to the skin. The following correlations are derived from Cinco-Ley and Samaniego graph.

Dimensionless fracture conductivity:

$$F_{cd} = \frac{k_p \cdot w_f}{k_r \cdot x_f} \quad \text{Eq. 4.100}$$

Pseudo-skin factor for a well with a finite-conductivity vertical fracture

$$S'_{hf} = \begin{cases} -0.7205 * \ln(F_{cd}) + 1.6368 & \text{if } F_{cd} < 1 \\ 3.0386 - 2.349 \exp(-0.511 F_{cd}^{-0.909}) & \text{if } 1 \leq F_{cd} < 1000 \\ 0.692 & \text{if } F_{cd} \geq 1000 \end{cases} \quad \text{Eq. 4.101}$$

and the hydraulic skin is given by:

$$S_{hf} = S'_{hf} - \ln\left(\frac{x_f}{r_w}\right) \quad \text{Eq. 4.102}$$

x_f Fracture Half Length

w_f Fracture Width

k_r Reservoir Permeability

k_p Proppant Permeability

r_w Wellbore Radius

Damaged hydraulic fracture performance can deviate substantially from undamaged fracture. Two types of damage are considered: [fracture face \(p.425\)](#) and [choke fracture \(p.426\)](#).

Fracture Face Skin

Cinco-Ley and Samaniego ([p.585](#)) (1981b) quantified the damage that may develop on the fracture face, by a skin effect of the form

$$S_{ff} = \frac{\pi}{2} \cdot \frac{w_{af}}{x_f} \cdot \left(\frac{k_r}{k_{af}} - 1 \right) \quad \text{Eq. 4.103}$$

w_{af} Depth of Damage (normal to the fracture face) is very thin (0.2 ft or less)

k_{af} Frac Face Damage Permeability

Choke Fracture Skin

Damage at the connection between the fracture and a well is referred to as a choke. Different to the fracture face skin, the damaged zone is within the fracture. [Romero et al. \(p.592\) \(2002\)](#) express the extra pressure drop in the fracture in term of a skin effect:

$$S_{cf} = \pi \cdot \frac{x_{cf}}{x_f} \cdot \left(\frac{k_r}{k_{cf}} - \frac{k_r}{k_p} \right) \quad \text{Eq. 4.104}$$

x_{cf} Choke Length

k_{cf} Frac Choke Permeability

Annulus Skin

Between the casing internal radius and the screen outer radius, assuming a radial flow through the gravel the contribution to the skin is expressed as:

$$S_{an} = \frac{k_r}{k_g} \cdot \ln \left(\frac{d_{ic}}{d_s} \right) \quad \text{Eq. 4.105}$$

d_{ic} Casing Internal Diameter

d_s Screen Diameter

k_g Gravel Permeability

Gravel Skin in tunnel

If a perforation is not defined, a default perforation diameter and a default perforation density are set (equal to 0.5 inches and 4 shots/ft respectively) for the calculation of the Frac Pack Skin. If the perforation tunnels through the casing and cement, where the most significant pressure drops usually occur, the skin component is:

$$S_{tg} = 2 \cdot \frac{k_r}{k_g} \cdot \frac{l_t}{n_p \cdot r_p^2} \quad \text{Eq. 4.106}$$

l_t Tunnel length

n_p Perforation Density

r_p Perforation radius

4.2.2 Inflow Performance Relationships for Horizontal Completions

Theory

The main purpose of drilling horizontal wells is to enhance production. There are also many circumstances that lead to drilling horizontal wells (Cooper, 1988):

Thin reservoirs

The increased area of contact of the horizontal well with the reservoir is reflected by the productivity index (PI). Typically, the PI for a horizontal well may be increased by a factor of 4 when compared to a vertical well penetrating the same reservoir.

Heterogeneous reservoirs

When irregular reservoirs exist, the horizontal well can effectively intersect isolated productive zones which might otherwise be missed. A horizontal well can also intersect vertical natural fractures in a reservoir.

Reduce water/gas coning

A horizontal well provides minimum pressure drawdown, which delays the onset of water/gas breakthrough. Even though the production per unit well length is small, the long well length provides high production rates.

Vertical permeability

If the ratio of vertical permeability to horizontal permeability is a high, a horizontal well may produce more economically than a vertical well.

Pressure Drop

Effect of Pressure Drop on Productivity

In many reservoir engineering calculations, the horizontal wellbore is treated as an infinite conductivity fracture, that is the pressure drop along the well length is negligible. However, in practice, there must be a pressure drop from the toe (tip-end) of the horizontal wellbore to the heel (producing-end) so as to maintain fluid flow within the wellbore (see Figure 1).

Dikken (1989), Folefac (1991) and Joshi (1991) have addressed the effect of wellbore pressure gradient on horizontal well production performance.

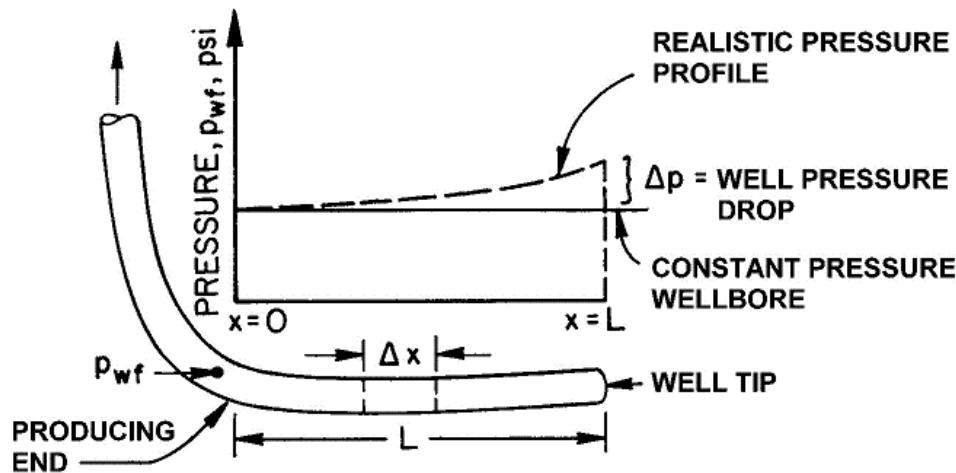


Figure 4.1. Along-hole pressure gradient of a horizontal well (Joshi, 1991)

Dikken (p.585) (1990) and Folefac (p.586) (1991) contend that the assumption of constant pressure wellbore is reasonable for single phase laminar flow but is no longer valid when turbulent or multiphase flow occurs. Folefac (1991) showed that a typical well with the following properties: $\rho_o = 800 \text{ kg/m}^3$; $\mu = 1.0 \text{ cp}$; $d = 0.1968 \text{ m}$; and $Q = 5000 \text{ RB/d}$ gives a N_{RE} of 4000 which is well above the turbulence transition limit of 2000. In most practical situations, Dikken (1990) asserts that horizontal wells will exhibit non-laminar flow. In addition, the pressure drop will be even greater when multiphase flow exists.

Joshi (p.588) (1991), thus, asked the question: What is the magnitude of the wellbore pressure drop as compared to pressure drop from the reservoir to the wellbore? If the wellbore pressure drop is significant as compared to the reservoir drawdown, then the reservoir drawdown, and consequently, the production rate along the well length will change. Thus, there is a strong interaction between the wellbore and the reservoir. The reservoir flow and wellbore equations must be solved simultaneously as shown in Figure 2.

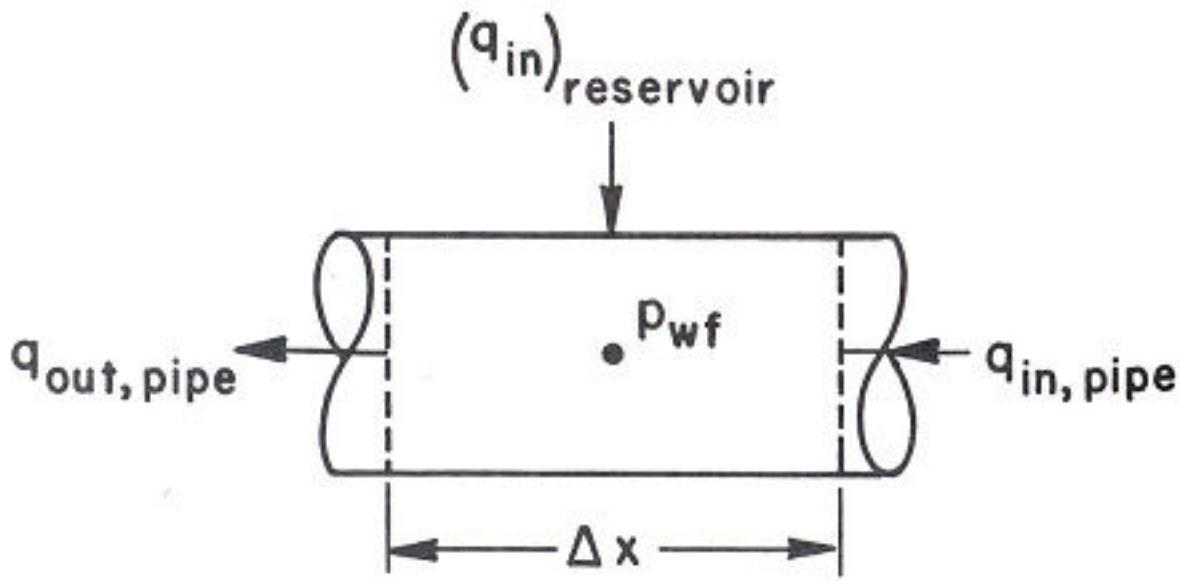


Figure 4.2. Schematic of reservoir and flow relationship (Joshi, 1991)

The coupled equations were solved by Dikken (1990) analytically by simplified boundary conditions, notably, no inflow from the toe-end. Folefac (1991) used a Black Oil type model that involved a finite volume technique.

Folefac (1991) concluded that the well length, wellbore diameter and perforated interval had the most profound effect on the level of pressure drop in the wellbore. Folefac (1991) pointed out that the wellbore pressure profile is non-linear with respect to the well length. This is because the mixture momentum equation has a non-linear term in velocity, the friction force. This in turn will result in an uneven drawdown in the reservoir that is otherwise considered homogenous.

Furthermore, Folefac (1991) showed that as the wellbore radius increased from 64.5 mm (2.5") to 114.3 mm (4.5"), the rate at which pressure dropped along the wellbore became nearly constant. This is mainly due to the turbulent flow being converted to laminar flow by drilling a larger size hole.

Joshi (1991) mentions other situations where wellbore pressure drop is considerable:

- High flowrates of light oil (10,000 to 30,000 RB/d)
- High viscous crudes (heavy oils and tar sands)
- Long well lengths

The wellbore pressure drop effects well deliverability and in turn influences well completion and well profile design. The need to accurately calculate well flowrates and wellbore pressures is therefore, essential.

Joshi (1991) lists a few remedies to minimize high wellbore pressure drops:

- Drilling a larger diameter hole would dramatically reduce the pressure drop. The reason being that for single phase flow, $D P \propto 1/d^5$. For example, Joshi (1991), states "for a given production rate, by increasing the well diameter twofold, the pressure drop can be reduced at least thirty-two fold".

- Varying the shot density of a cemented hole or the slot size of a slotted liner would control production rates and minimize pressure drop along the wellbore
- Gravel packs are used in high permeability reservoirs. If the well is completed with a slotted liner, the slots should be placed as far apart as possible. Joshi (1991) states that "this will let the gravel pack act as a choke and facilitate maintaining minimum pressure drop across the well length".

Therefore, by selecting the appropriate well geometry, hole size and length, wellbore pressure drops can be minimized.

Single Phase Pressure Drop

Assuming that the horizontal wellbore can be treated as a horizontal pipe, the single phase flow pressure drop calculation for oil flow can be written as follows:

$$\Delta p = (1.14644 \times 10^{-5}) f \rho q^2 \frac{L}{D^5} \quad \text{Eq. 4.107}$$

where,

Δp is pressure drop, psia

f is Moody's friction factor, dimensionless

ρ fluid density, gm/cc

q is flowrate at reservoir conditions, RB/d

L is horizontal length, ft

D is internal diameter of pipe, inches

For gas flow, however, the pressure drop calculations are more complex. This is due to friction which could change the temperature of the gas as it travels through the wellbore. Moreover, density and viscosity are strong functions of gas pressure and temperature. This would result in a changing pressure drop per foot length of a well along the entire well length. The Weymouth equation for dry gas is the simplest equation to estimate pressure drop in a horizontal pipe

$$q_g = 15320 \sqrt{\frac{(p_1^2 - p_2^2) D^{16}}{\gamma_g T Z L}} \quad \text{Eq. 4.108}$$

where

q_g is gas flowrate, scfd

p_1 is pipe inlet pressure, psia

p_2 is pipe outlet pressure, psia

L is pipe length, miles

T is average temperature, °R

Z is average gas compressibility factor

D is pipe diameter, in

γ_g gas specific gravity

Also, several multiphase correlations (Brill, 1988) are applicable for a single phase flow of either oil or gas.

Multiphase Pressure Drop

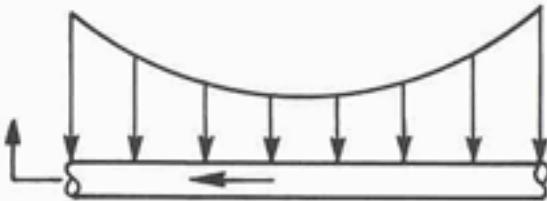
There is very little discussion on multiphase pressure drop in horizontal wells. Folefac (1991) studied the effect of two phase flow (hydrocarbon liquid and water are treated as one phase with identical velocity but averaged properties). The pressure drop along the horizontal wellbore was similar to that for single phase flow. However, the pressure drop was higher than for single phase flow for the same volume of fluid intake.

For a horizontal pipe, numerous multiphase flow correlations have been discussed by [Brill \(p.584\)](#) (1988). Slip velocities between phases make these equations more complex than single phase flow equations. In general, Joshi (1991) states that, "different multiphase correlations may give different values of the pressure drop". The various correlations should be compared with actual pressure drop data. However, measuring the pressure at both ends of a horizontal well and calibrating the data is very difficult. There is a definite need for further study on multiphase flow in horizontal wells.

Inflow Production Profiles

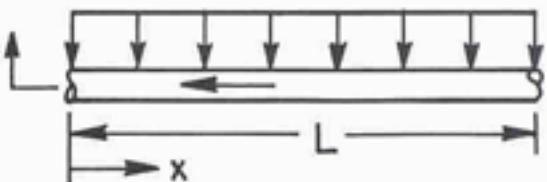
Horizontal wellbore pressure drops also depend upon the type of fluid inflow profiles. Figure 3 shows some horizontal well fluid inflow profiles. On the basis of well boundary condition and reservoir heterogeneity, several profiles are possible. Joshi (1991) examined the effect of different fluid entry profiles on the wellbore pressure drop. Depending on the type of profile, Joshi concluded that the total pressure drop varied from 6 psi to 14.5 psi but it was not large enough to effect the wellhead pressure.

a. UNIFORM WELBORE PRESSURE
(INFINITE - CONDUCTIVITY)

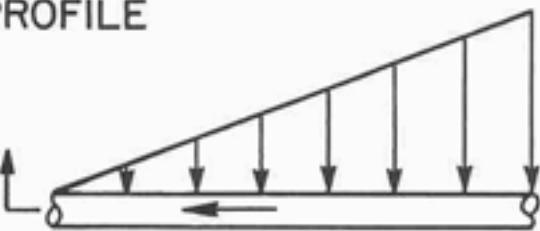


b. UNIFORM FLUX ENTRY

$$q(x) = \frac{q_{\text{total}}}{L}$$



c. TRIANGULAR PROFILE



d. TRIANGULAR PROFILE

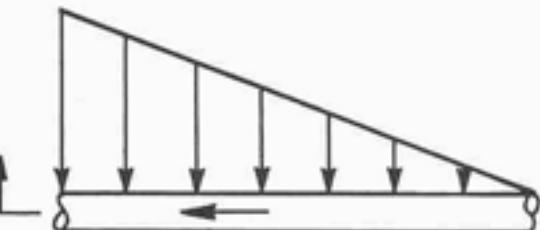


Figure 4.3. Horizontal Well Inflow Profiles (Joshi, 1991)

Steady-State Productivity

The simplest form of horizontal well productivity calculations are the steady-state analytical solutions which assume that the pressure at any point in the reservoir is constant over time. According to Joshi (1991), even though very few reservoirs operate under steady-state conditions, steady state solutions are widely used because:

Analytical derivation is easy.

The concepts of expanding drainage boundary over time, effective wellbore radius and shape factors allows the conversion to either transient or pseudo-steady state results to be quite straightforward.

Steady-state mathematical results can be verified experimentally.

Giger (1984), Economides (1989), Mukherjee (1988) and numerous others have developed solutions to predict steady-state productivity. Most are similar in form to the equation given by Joshi (1988) who simplified the 3-D Laplace equation ($\Delta p=0$) by coupling two 2-D problems. This was based on the assumption that a horizontal well drains an ellipsoidal volume around the wellbore of length L as shown below.

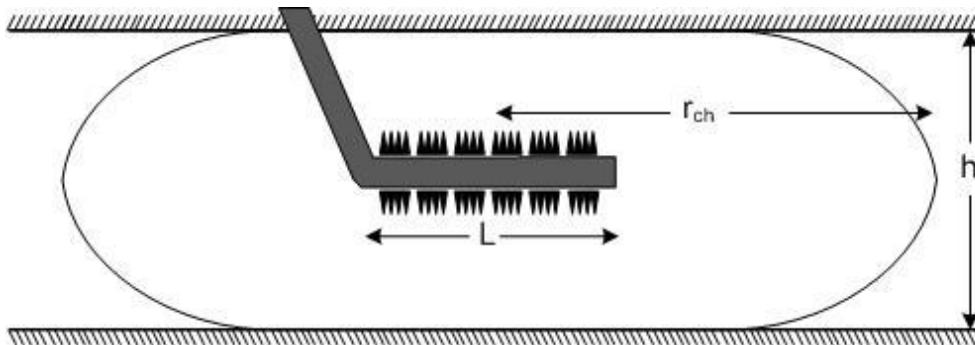


Figure 4.4. Horizontal Well Drainage Pattern

For isotropic reservoirs $k_h = k_v$

$$q_h = \frac{k_h h \Delta p}{141.2 \mu_o B_o \left(\ln \left[\frac{a + \sqrt{a^2 - (L/2)^2}}{L/2} \right] + \left(\frac{h}{L} \right) \ln \left[\frac{h}{2r_w} \right] \right)} \quad \text{Eq. 4.109}$$

and

$$a = \left(\frac{L}{2} \right) \left[0.5 + \sqrt{0.25 + \left(\frac{2r_{eh}}{L} \right)^4} \right]^{0.5} \quad \text{Eq. 4.110}$$

where

q_h	is the flowrate	STB / d
a	is half the major axis of the drainage ellipse	ft
Δp	is the pressure drop	psi
L	is the horizontal well length	ft
h	is reservoir height	ft
r_w	is the wellbore radius	ft

r_{eh}	is the effective drainage radius of horizontal well	ft
μ_o	is the oil viscosity	cp
B_o	is the oil volume formation factor	RB / STB
k_h	is the horizontal permeability	mD

If the length of the horizontal well is significantly longer than the reservoir height, that is $L \gg h$, then the second term in the denominator of the [4.109 \(p.433\)](#) equation is negligible and the solution simplifies to

$$q_h = \frac{k_h h \Delta p}{141.2 \mu_o B_o \ln \left[\frac{4r_{eh}}{L} \right]} \quad \text{Eq. 4.111}$$

[Muskat \(p.590\)](#) (1937) suggested a simple transformation to account for permeability anisotropy.

An effective permeability, k_{eff} , is defined as

$$k_{eff} = \sqrt{k_v k_h} \quad \text{Eq. 4.112}$$

To account for vertical anisotropy, the reservoir thickness can be modified as follows

$$h = h \sqrt{\frac{k_h}{k_v}} \quad \text{Eq. 4.113}$$

In addition, the influence of well eccentricity (distance from the center of the reservoir in the vertical plane) was also implemented. Thus, equation [4.109 \(p.433\)](#) was transformed as follows

$$q_h = \frac{k_h h \Delta p}{141.2 \mu_o B_o \left(\ln \left[\frac{a + \sqrt{a^2 - (L/2)^2}}{L/2} \right] + \beta^2 \frac{h}{L} \ln \left(\frac{h}{2r_w} \right) \right)} \quad \text{Eq. 4.114}$$

where

$$\beta = \sqrt{\frac{k_h}{k_v}} \quad \text{Eq. 4.115}$$

Productivity comparisons of a horizontal well to that of a vertical well can easily be made by using the [4.114 \(p.434\)](#) equation. In converting the productivity of a horizontal well into that of an equivalent vertical well, an effective wellbore radius can be calculated, $r_{w,eff}$

$$r_{w,eff} = r_w \exp(-S) \quad \text{Eq. 4.116}$$

The effective wellbore radius is defined as the theoretical well radius which will match the production rate. Joshi (1991) assumed equal drainage volumes, $r_{eh} = r_{ev}$, and equal productivity indices, $J_h = J_v$ to give the following for an anisotropic reservoir

$$r_{w,eff} = \frac{r\left(\frac{L}{2}\right)}{a\left[1 + \sqrt{1 - \left(\frac{L}{2a}\right)^2}\right] + \left[\frac{\beta h}{r_w}\right]^{\frac{\beta h}{L}}} \quad \text{Eq. 4.117}$$

In this way, controlling parameters like well length, permeability and formation thickness can be used to screen potential candidates for further simulation studies.

[Renard \(p.592\)](#) (1990) studied the effect of formation damage around the wellbore and modified the steady-state equation to include skin. Renard (1990) concluded that due to the lower productivity index per unit length in horizontal wells, the effect of skin damage is not as pronounced as it is in vertical wells. [Celier et al. \(p.584\)](#) (1989) came to the same conclusion with respect to the effect of non-Darcy flow.

Pseudo-Steady State Productivity

It is often desirable to calculate productivity from a reservoir with unique boundary conditions, such as a gas cap or bottom water drive, finite drainage area, well location, and so on. In these instances pseudo-steady state equations are employed. Pseudo-steady state or depletion state begins when the pressure disturbance created by the well is felt at the boundary of the well drainage area

Pseudo-Steady State Productivity

[Dake \(p.585\)](#) (1978) and [Golan \(p.587\)](#) (1986) describe the pseudo-steady state flow of an ideal fluid (liquid) in a closed circular drainage area. Rearranging the equation gives the familiar vertical well productivity

$$q_v = \frac{kh\Delta p}{141.2 \mu_o B_o [\ln(\sqrt{\frac{2.2458A}{C_A r_w^2}}) + S + S_m + Dq_v]} \quad \text{Eq. 4.118}$$

where

q_v	is the flowrate	STB/d
Δp	is the pressure drop between the reservoir and the wellbore	psi
S_m	is the mechanical skin factor due to drilling and completion related well damage	
S	is the skin due to perforations, partial penetration and stimulation	
C_A	is the shape factor	
Dq_v	is the near wellbore turbulence factor or rate dependent skin	

μ_o	is the oil viscosity	cp
B_o	is the oil volume formation factor	RB / STB
k	is the formation permeability	mD
h	is the formation thickness	ft
A	is the drainage area	ft^2
r_w	is the wellbore radius	ft
r_{eh}	is the drainage radius	ft

The above equation can be reduced to the following single-phase pseudo-steady state equation for oil flow (assuming $S = 0$, $S_m = 0$ and $Dq_v = 0$),

$$q_v = \frac{kh\Delta p}{141.2\mu_o B_o \left[\ln\left(\frac{r_{eh}}{r_w}\right) - 0.75 \right]} \quad \text{Eq. 4.119}$$

The equation is for a vertical well which is located in the center of a circular drainage area. [Fetkovich \(p.586\)](#) (1985) wrote the shape factor in terms of an equivalent skin. This skin was expressed by choosing a reference shape factor of a well at the center of circular drainage area

$$S_{CA} = \ln \sqrt{\frac{C_{Aref}}{C_A}} \quad \text{Eq. 4.120}$$

The horizontal well shape factor depends on the following:

- drainage area shape,
- well penetration.
- dimensionless well length, $L_D = \left(\frac{L}{2h} \right) \sqrt{\frac{k_v}{k_h}}$

L	is the length of the horizontal well	ft
h	is the formation thickness	ft
k_v	is the vertical permeability	mD
k_h	is the horizontal permeability	mD

[Joshi \(p.588\)](#) (1991) explains that the well performance approaches a fully penetrating infinite-conductivity fracture when the horizontal well length is sufficiently long, i. e. $L_D > 10$.

[Babu \(p.583\)](#) (1989), [Goode \(p.587\)](#) (1989) and [Mutalik \(p.590\)](#) (1988) have developed methods to calculate pseudo-steady state productivity for single phase flow in horizontal wells. Shape factors were used to arbitrarily locate the well within a rectangular bounded drainage area and the reservoir was bounded in all directions. Mutalik's model assumed the horizontal well as an infinite conductivity well (i.e. the wellbore pressure drop is negligible). Babu's model assumed uniform-flux boundary condition. Goode's model used an approximate infinite conductivity solution where the constant wellbore pressure is estimated by averaging the pressure values of the uniform-flux solution along the well length. Goode (1989) also considered the effects of completion type on productivity. Their model allowed for cased completion, selectively perforated completion, external casing packers to selectively isolate the wellbore and slotted liner completion with selectively isolating zones.

Babu (1989) developed a physical model consisting of a well drilled in a box-shaped drainage volume, parallel to the y direction (see figure 4.5 (p.437)).

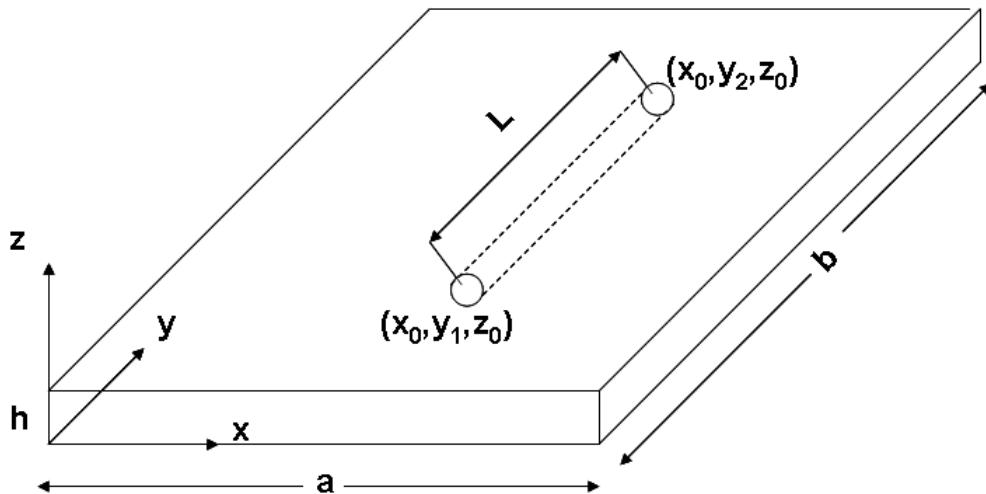


Figure 4.5. Babu and Odeh physical model

The derived pseudo-steady productivity equation is

$$q_h = \frac{7.08 \times 10^{-3} b \sqrt{k_x k_z} \Delta p}{\mu_o B_o \left[\ln \left(\frac{\sqrt{A_l}}{r_w} \right) + \ln C_H - 0.75 + S_R \right]} \quad \text{Eq. 4.121}$$

where

b	is extension of the drainage volume in the direction along the well axis Oy	ft
S_R	is the skin factor due to partial penetration	
C_H	is the geometric shape factor defined by Babu (1989)	

k_x	is the permeability along the x-axis	mD
k_z	is the permeability along the z-axis	mD
A_1	is the drainage area in the vertical plane	ft^2
r_w	is wellbore radius	ft

The validation rules for Babu and Odeh IPR model are:

- 1. Heel location (X position) must be from 0.0 to ReservoirXDim. In the original publication (see Babu and Odeh 1989), the requirement in the x-direction for the second case considered is that heel location (X position) must be from ReservoirXDim * 0.25 to ReservoirXDim * 0.75. While this requirement is not enforced in this product, user should take caution when operating outside of the requirement.
- 2. Heel location (Y position) + well length must be less than or equal to the reservoir Y dimension. Also, heel location (Y position) must be greater than or equal to zero.
- 3. Heel location (Z position) + well radius must be less than or equal to the reservoir thickness. Also, heel location (Z position) must be greater than or equal to well radius.

The equation 4.121 (p.437) is derived from a very complex general solution. It requires the calculation of C_H and S_R . The geometric shape factor accounts effect of permeability anisotropy, well location and relative dimensions of the drainage volume. The skin accounts for the restricted entry associated with the well length. Babu (1989) reported an error of less than 3% when compared to the more rigorous solution.

Solution Gas-Drive IPR

Cheng (1990), Joshi (1991) and Bendakhlia (1989) have studied the inflow performance relationship (IPR) for solution gas-drive reservoirs. Bendakhlia followed the same approach used by Vogel for vertical wells and developed the following equation:

$$\frac{q_0}{q_{0,\max}} = \left[1 - V \left(\frac{P_{wf}}{P_R} \right)^n - 1 - V \left(\frac{P_{wf}}{P_R} \right)^2 \right]^n \quad \text{Eq. 4.122}$$

The equation can be used under the assumptions of Vogel's original IPR correlation. The parameter V and n were correlated as a function of recovery factor.

Horizontal Gas Wells

The preceding sections have dealt with oil flow. However, horizontal wells are also appropriate for gas reservoirs. For example, in high-permeability gas reservoirs wellbore turbulence limits the deliverability of a vertical well. The most effective way, according to Joshi (p.588) (1991), to reduce gas velocity around the wellbore is to reduce the amount of gas production per unit well length which can be accomplished by horizontal wells. Joshi (1991) describes two methods for the relationship between pressure and flowrate.

The gas flowrate is proportional to the pressure square terms.

Al-Hussainy et al. (p.583) (1966) defined a pseudo-pressure $m(p)$. The gas flowrate is directly proportional to the pseudo-pressures which is defined as

$$m(p) = \int_{p_0}^p \frac{2pd\bar{p}}{\mu(p)z(p)} \quad \text{Eq. 4.123}$$

A comparison of the two methods was done by Joshi (1991). Below reservoir pressures of 2500 psia, either method can be employed. However, above 2500 psia, the pseudo-pressure should be used.

Steady state gas flow equation

The steady-state equation for gas flow is

$$q_h = \frac{7.027 \times 10^{-4} k_h h (p_e^2 - p_{wf}^2)}{\mu Z T \ln \left(\frac{r_{eh}}{r'_w} \right)} \quad \text{Eq. 4.124}$$

where

q_h	is the gas flowrate	mmscf / d
p_e	is the pressure at external radius	psia
p_{wf}	is the wellbore flowing pressure	psia
k_h	is the horizontal permeability	mD
h	is the reservoir height	ft
r_{eh}	is the drainage radius	ft
r'_w	is the effective wellbore radius	ft
μ	is the average viscosity	cp
Z	is the average compressibility factor	
T	is the reservoir temperature	${}^\circ\text{R}$

Pseudo steady state gas flow equation

The pseudo-steady state gas flow equation can be written as follows (Joshi, 1991)

$$q_h = \frac{7.027 \times 10^{-4} kh \left(p_r^2 - p_{wf}^2 \right)}{\mu Z T \left[\ln \left[\frac{r_{eh}}{r_w} \right] - 0.75 + S + S_m + S_{ca} - C + D q_h \right]} \quad \text{Eq. 4.125}$$

where

$$D = \frac{2.222 \times 10^{-15} (\gamma_G k_a h \beta)}{\mu_{pwf} r_w^2 h_p} \quad \text{Eq. 4.126}$$

In equation 4.126 (p.440), the high velocity flow coefficient is given by:

$$\beta = 2.73 \times 10^{10} k_a^{-1.1045} \quad \text{Eq. 4.127}$$

where

q_h	is the gas flowrate	mmscf / d
p_r	is the average reservoir pressure	psia
p_{wf}	is the wellbore flowing pressure	psia
S	is the negative skin factor due to horizontal well (or well stimulation)	
S_m	is the mechanical skin damage	
S_{ca}	is the shape related skin factor	
C	is the shape factor conversion constant	
k	is the permeability	mD
h	is the reservoir height	ft
r_{eh}	is the drainage radius	ft
r_w	is the wellbore radius	ft
μ	is the average viscosity	cp
Z	is the average compressibility factor	
T	is the reservoir temperature	°R
μ_{pwf}	is the gas viscosity at well flowing conditions	cp
β	is the high velocity flow coefficient	1 / ft
γ_G	is the gas specific gravity	dimensionless

h_p	is the perforated interval	ft
k_a	is the permeability in the near wellbore region	mD

The equation 4.125 (p.440) is based upon circular drainage area as a reference area. In this equation, Dq_h is the turbulence term, also called turbulence skin, or rate dependent skin factor . (see [Joshi \(p.588\)](#) (1991), [Brown \(p.584\)](#) (1984) and [Golan and Whitson \(p.587\)](#) (1986)). This term accounts for the extra pressure drop in the near wellbore region due to high gas velocity. This term was neglected when dealing with oil flow. In addition, the term makes the solution of [4.125 \(p.440\)](#) iterative.

The equation 4.127 (p.440) is given in [Golan and Whitson \(p.587\)](#) (1986)

Conclusions

The following can be concluded from this review:

The assumption of constant pressure drop in the wellbore is no longer valid, especially for long well lengths and when turbulent/multiphase flow occurs.

More realistic production geometries are being used in the existing models to calculate horizontal well productivity.

Existing models need to be verified and validated with actual field data. The absence of case studies in the literature is indicative of the 'tight-hole' status of most horizontal well projects.

Distributed Productivity Index Method

for liquid reservoirs

$$Q = J(P_{ws} - P_{wf})L \quad \text{Eq. 4.128}$$

for gas reservoirs

$$Q = J(P_{ws}^2 - P_{wf}^2)L \quad \text{Eq. 4.129}$$

where J is the distributed productivity index.

4.2.3 Oil / Water Relative Permeability tables

A table of oil and water relative permeabilities as functions of water saturation ($k_{ro}(S_{wat})$, $k_{rw}(S_{wat})$) can be defined in conjunction with the [Pseudo Steady-State \(p.403\)](#) or [Transient \(p.408\)](#) liquid IPRs for vertical completions or the [Steady State \(p.432\)](#) or [Pseudo-Steady State \(p.435\)](#) liquid IPRs for horizontal completions.

If the reservoir water saturation is known, the water cut of the fluid flowing into the well can be calculated:

$$WCUT = 100 \cdot \frac{Q_w}{Q_o + Q_w} = 100 \cdot \frac{k_{rw}(S_{wat}) / \mu_w}{k_{ro}(S_{wat}) / \mu_o + k_{rw}(S_{wat}) / \mu_w} \quad \text{Eq. 4.130}$$

Alternatively, if the water cut is known, the water saturation can be found by solving the same equation for the water saturation, S_{wat} .

The oil and water inflows can then calculated separately using the relevant liquid IPR equation and summed to give the liquid flow rate.

Keywords

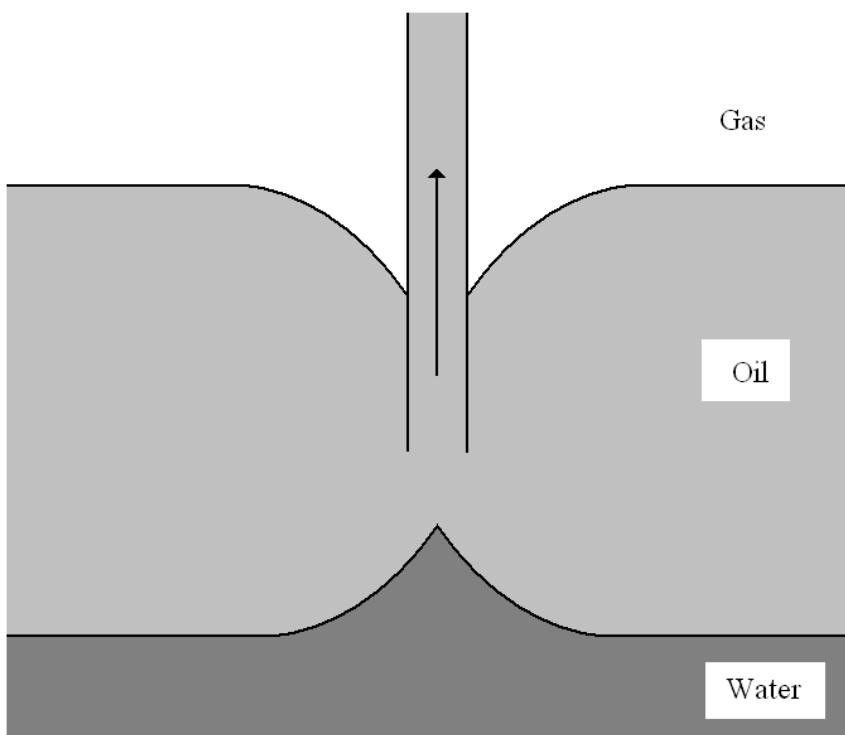
Data is entered, in keyword mode, using the [additional engine keyword feature \(p.595\)](#). Use the [PERMTAB \(p.669\)](#) keyword to define the table. Use the LAYER keyword to define the reservoir water saturation, if required.

See also [other IPR methods \(p.398\)](#)

4.2.4 Coning

In order to simulate gas and/or water breakthrough from the reservoir, flowrate-dependent values of GOR and watercut may be entered. In a homogeneous reservoir, analysis of the radial flow behavior of reservoir fluids moving towards a producing well shows that the rate dependent phenomenon of coning may be important. The effect of increasing fluid velocity and energy loss in the vicinity of a well leads to the local distortion of a gas-oil contact or a water-oil contact. The gas and water in the vicinity of the producing wellbore can therefore flow towards the perforation. The relative permeability to oil in the pore spaces around the wellbore decreases as gas and water saturation increase. The local saturations can be significantly different from the bulk average saturations (at distances such as a few hundred meters from the wellbore). The prediction of coning is important since it leads to decisions regarding:

- Preferred initial completions
- Estimation of cone arrival time at a producing well
- Prediction of fluid production rates after cone arrival
- Design of preferred well spacing



4.3 Equipment

4.3.1 Chokes, Valves and Fittings

Choke

A choke is a mechanical device that limits the flow rate through the pipe. The fluid velocity increases through the constriction and for compressible fluids can reach the sonic velocity. As the pressure difference across the choke increases the flow velocity increases too. At the point the velocity becomes sonic, the flow is said to be critical, and is independent of the down stream pressure. See [Brill and Mukerjee \(p.590\) \(1999\)](#) for a detailed description of flow through chokes and restrictions.

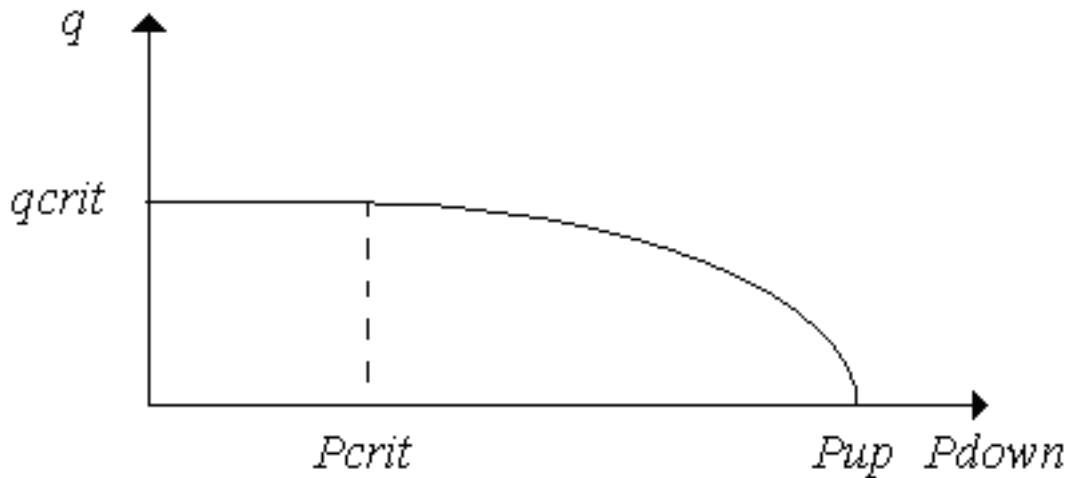


Figure 4.6. Typical flow-pressure relationship for a choke

The choke is modeled by splitting the flow into two regimes:

flow is subcritical	$P_{crit} < P_{down} < P_{up}$	$q < q_{crit}$
flow is critical	$0 < P_{down} < P_{up}$	$q = q_{crit}$

where

P_{up}	is the upstream pressure	psi or lbf/in^2	N/m^2
P_{down}	is the downstream pressure	psi or lbf/in^2	N/m^2
P_{crit}	is the critical (downstream) pressure	psi or lbf/in^2	N/m^2
q	is the flow rate	lb/s	kg/s
q_{crit}	is the critical flow rate	lb/s	kg/s

The choke performance is determined by the following:

1. The choke geometry and fluid properties
2. The subcritical flow correlation
3. The critical pressure ratio
4. The critical flow correlation

Choke geometry

The main choke parameters are:

d_{up}	upstream diameter	in
d_{bean}	constriction (bean) diameter	in
c_v	flow coefficient (used in the Ashford & Pierce (p.446) correlation)	
c_{vL}	liquid flow coefficient (used in the Mechanistic (p.448) correlation)	
c_{vG}	gas flow coefficient (used in the Mechanistic (p.448) correlation)	
c_d	discharge coefficient (used for calculating the flow coefficients)	

The flow coefficients can either be specified or calculated from the discharge coefficient:

$$c_v = \frac{c_d}{\sqrt{1 - \delta^4}} \quad \text{Eq. 4.131}$$

where:

$\delta = \frac{d_{bean}}{d_{up}}$	is the diameter ratio	dimensionless
------------------------------------	-----------------------	---------------

Subcritical flow correlations

There are essentially two subcritical flow models used in PIPESIM, the [Mechanistic \(p.448\)](#) correlation and [Ashford and Pierce \(p.446\)](#) (1975) correlation . A third correlation [API-14B \(p.449\)](#) is a modification of the Mechanistic correlation

Critical pressure ratio

For single phase liquids, the sonic velocity is high and flow is always subcritical. For single phase gas flow and multiphase flow, the critical pressure is given by

$$P_{crit} = C_{PR} \cdot P_{up} \quad \text{Eq. 4.132}$$

The value of the critical pressure ratio C_{PR} can be set by the user or [calculated \(p.449\)](#).

Critical flow correlations

A critical flow correlation can be used to set the critical flow rate q_{crit} . There is a danger that this will not match the subcritical flow at the critical pressure ratio. PIPESIM therefore adjusts the subcritical flow correlation to ensure the flow is correct at the critical pressure. To do this it first calculates the subcritical flow at the critical pressure:

$$q_{lim} = q_{sub}(P_{up}, P_{down}) \text{ evaluated at } P_{down} = P_{crit}$$

The choke downstream pressure is then calculated from the subcritical correlation using the upstream pressure and a scaled flow rate (q_{lim}/q_{crit}) q . This matching can be turned off, in which case the critical flow correlation is ignored when calculating the pressure drop, although it is used for reporting purposes.

One of twelve correlations of four distinct types can be selected for the critical flow:

1. Mechanistic (p.450), API-14B (p.450)
2. Ashford Pierce (p.450), A-P Tulsa, Poettmann-Beck (p.450)
3. Omana (p.450)
4. Achong (p.451), Baxendale (p.451), Gilbert (p.451), Pilehvari (p.451), Ros (p.451), User defined (p.451)

Engine keywords

See [Choke keyword](#) (p.673)

Choke Subcritical Flow Correlations

Two subcritical flow correlations, [Ashford-Pierce \(p.446\)](#) and [Mechanistic \(p.448\)](#) are available. A third, [API 14B \(p.449\)](#) is a version of the mechanistic correlation.

Ashford-Pierce

Ashford-Pierce (1975) (p.583) give the following equation for oil flow rate through a choke:

$$q_o = \frac{c_1 c_v (64 d_{bean})^2}{\sqrt{c_2}} \cdot \sqrt{P_{up}} \cdot \frac{\sqrt{1 - \varepsilon + R_L (1 - \varepsilon^k)/k}}{1 + R_L \varepsilon^{-1/\gamma}} \cdot \frac{1}{\sqrt{B_o + F_{wo}}} \cdot \frac{\sqrt{\gamma_o + c_3 \gamma_G R_s + F_{wo} \gamma_w}}{\gamma_o + c_3 \gamma_G R + F_{wo} \gamma_w} \quad 4.133$$

where

B_o	is the oil formation factor volume factor	bbi / STB
$c_1 = 3.51$	is a constant	
$c_2 = 198.6$	is a constant	
$c_3 = 0.000217$	is a constant	
c_v	is the flow coefficient	
d_{bean}	is the bean diameter	1 / 64 in
$k = \frac{\gamma - 1}{\gamma}$		dimensionless
F_{wo}	is the upstream water to oil ratio	

P_{up}	is the upstream pressure	
P_{down}	is the downstream pressure	
q_o	is the oil flow rate at standard conditions	bbl/d
R_s	is the upstream gas oil ratio	scf / STB
R	is the gas oil ratio at standard conditions	scf / STB
$R_L = \frac{T_{up}Z_{up}(R - R_s)}{198.6P_{up}}$	is the upstream gas liquid ratio	dimensionless
$\varepsilon = \frac{P_{down}}{P_{up}}$	is the pressure ratio	dimensionless
$\gamma = \frac{C_p}{C_V}$	is the ratio of specific heats	dimensionless
γ_o	is the upstream oil specific gravity	dimensionless
γ_G	is the upstream gas specific gravity	dimensionless
γ_w	is the upstream water specific gravity	dimensionless

The Ashford and Pierce formula is based on the following assumptions:

- polytropic expansion of gas-liquid mixture
- equal gas and liquid velocities at the throat
- incompressible liquid phase
- liquid dispersed in a continuous gas phase
- negligible friction losses

Recommended values for the flow coefficient c_v are:

diameter in 1/64 in	d_{bean}	
8	0.125	1.2
12	0.1875	1.2
20	0.3125	0.976
24	0.375	0.96
32	0.5	0.95

Mechanistic Correlation

The pressure drop across the choke is given by the weighted average of the liquid and gas phase pressure drops:

$$\Delta P = \lambda_L \cdot \Delta p_L + \lambda_G \cdot \Delta p_G \quad \text{Eq. 4.134}$$

The liquid phase pressure drop is given by Bernouilli's equation:

$$\Delta p_L = \frac{\rho_n}{2 \cdot c} \left(\frac{v^2}{c_{vL} \cdot Z_L} \right)^2 \quad \text{Eq. 4.135}$$

The gas phase pressure drop is given by Bernouilli's equation:

$$\Delta p_G = \frac{\rho_n}{2 \cdot c} \left(\frac{v^2}{c_{vG} \cdot Z_G} \right)^2 \quad \text{Eq. 4.136}$$

$v = \frac{q}{A_{beam} \cdot \rho_n}$	is the mixture velocity through the choke	ft/s	m/s
q	is the mass flow rate	lbf/s	kg/s
$A_{beam} = \frac{\pi \cdot d_{beam}^2}{4}$	is the choke area at the constriction	ft^2	m^2
$\rho_n = \lambda_L \cdot \rho_L + \lambda_G \cdot \rho_G$	is the no-slip density	lbf/ft^3	kg/m^3
λ_L and λ_G	are the liquid and gas phase flowing fractions		
ρ_L and ρ_G	are the liquid and gas phase densities	lbf/ft^3	kg/m^3
$Z_L = 1$ and $Z_G = 1 - \frac{0.41 + 0.35 \delta^4}{\gamma} \cdot \frac{\Delta P}{P_{up}}$	are the liquid and gas compressibilities		
c	is a conversion factor for engineering units	$c = 144 \cdot g$ $\text{lbf/(ft \cdot s}^2\text{)} / \text{psi}$	$c = 1$

Total pressure drop for the two-phase system is therefore:

$$\Delta P = \frac{\rho_n \cdot v^2}{2 \cdot c} \cdot \left[\frac{\lambda_L}{(c_{vL} \cdot Z_L)^2} + \frac{\lambda_G}{(c_{vG} \cdot Z_G)^2} \right] \quad \text{Eq. 4.137}$$

API 14-B Formulation

The API 14-B formulation is similar to the mechanistic formulation, with the addition of the following assumptions:

1. Liquid flow through the choke is incompressible. The discharge coefficient is constant with a value of

$$c_{vL} = 0.85.$$

2. Subcritical gas flow through the choke is adiabatic and compressible. The discharge coefficient is constant with a value of

$$c_{vG} = 0.9.$$

Choke Critical Pressure Ratio

The critical pressure ratio C_{PR} is used to determine the downstream pressure when critical flow occurs in the choke. You can set a value of C_{PR} or it can be calculated, either from the single-phase gas formula (used with the Mechanistic subcritical flow correlation) or using the Ashford and Pierce formula (used with the Ashford and Pierce subcritical flow correlation).

Single phase gas critical pressure ratio

For a single phase gas flow, the critical pressure is given as a function of the specific heat ratio:

$$C_{PR} = \left[\frac{2}{\gamma + 1} \right]^{\frac{\gamma}{\gamma - 1}} \quad \text{Eq. 4.138}$$

The value of $\gamma = C_p / C_v$ is calculated by the program, but can be overridden by the user. For diatomic gases (for example air) $\gamma \approx 1.4$ and $C_{PR} = 0.53$

Ashford and Pierce critical pressure ratio

[Ashford-Pierce \(1975\) \(p.583\)](#) give the critical flow condition $\frac{\partial q_o}{\partial \varepsilon} = 0$ at $\varepsilon = C_{PR}$. [4.133 \(p.446\)](#) for q_o and simplifying gives:

$$\frac{\partial}{\partial \varepsilon} \left[\frac{\sqrt{1 - \varepsilon + R_L (1 - \varepsilon^k) / k}}{1 + R_L \varepsilon^{-1/\gamma}} \right] = 0 \quad \text{Eq. 4.139}$$

This can be manipulated to give an equation for $\varepsilon = C_{PR}$:

$$(1 + R_L \varepsilon^{-1/\gamma})^2 = \frac{2R_L}{\gamma} \cdot \varepsilon^{-\frac{\gamma}{\gamma+1}} \cdot (1 - \varepsilon + R_L \cdot (1 - \varepsilon^k) / k) \quad \text{Eq. 4.140}$$

Choke Critical Flow Correlations

The following choke correlations are available:

Ashford and Pierce / Sachdeva / Poetmann-Beck

The [Ashford-Pierce \(1975\) \(p.583\)](#) critical flow can be obtained by evaluating [4.133 \(p.446\)](#) at $\varepsilon = C_{PR}$, determined from [4.140 \(p.449\)](#). The stock tank critical oil flow rate takes the form:

$$q_o = \frac{c_1 c_v (64 d_{bean})^2}{\sqrt{c_2}} \cdot \sqrt{P_{up}} \cdot \frac{\sqrt{1 - \varepsilon + R_L (1 - \varepsilon^k) / k}}{1 + R_L \varepsilon^{-1/\gamma}} \cdot \frac{1}{\sqrt{B_o + F_{wo}}} \cdot \frac{\sqrt{\gamma_o + c_3 \gamma_g R_s + F_{wo} \gamma_w}}{\gamma_o + c_3 \gamma_g R + F_{wo} \gamma_w} \quad \text{Eq. 4.141}$$

The [Sachdeva \(p.582\)](#) critical flow correlation takes a similar form:

$$q_o = 0.858 c_v (64 d_{bean})^2 \cdot \sqrt{P_{up}} \cdot \frac{\sqrt{R_L + 0.76}}{R_L + 0.56} \cdot \frac{1}{\sqrt{B_o + F_{wo}}} \cdot \frac{1}{\sqrt{(62.4(\gamma_o + c_3 \gamma_g R + F_{wo} \gamma_w))^2 + (62.4(\gamma_o + c_3 \gamma_g R_s + F_{wo} \gamma_w))^{-1}}} \quad \text{Eq. 4.142}$$

The [Poetmann-Beck \(p.591\)](#) critical flow correlation takes a similar form:

$$q_o = 88992 \cdot \sqrt{9273.6} \cdot 0.4513 \cdot A_{bean} \cdot \sqrt{P_{up}} \cdot \frac{\sqrt{R_L + 0.766}}{R_L + 0.5663} \cdot \frac{1}{5.61 \rho_L + 0.0765 \gamma_G R} \cdot \frac{\rho_L + R_L \rho_G}{\sqrt{\frac{3}{2} \rho_L + R_L \rho_G}} \quad \text{Eq. 4.143}$$

Mechanistic / API14B

The critical mass flow rate can be found by inverting the [4.137 \(p.448\)](#) and evaluating it at the critical value of the pressure drop:

$$q = A_{bean} \sqrt{\frac{2g \cdot \rho_n \cdot \Delta P}{c_1 \cdot \left[\frac{\lambda_L}{(c_{vL} \cdot Z_L)^2} + \frac{\lambda_G}{(c_{vG} \cdot Z_G)^2} \right]}} \quad \text{Eq. 4.144}$$

$$\Delta P = (1 - C_{PR}) P_{up} \quad \text{Eq. 4.145}$$

The API14B critical flow uses the mechanistic critical flow formula, with $c_{vL} = 0.85$ and $c_{vG} = 0.9$

Omana Correlation

The [Omana \(p.591\)](#) correlation gives a formula for the stock tank critical liquid flow rate:

$$q_L = 1.953 \times 10^{-3} \cdot \sigma_L^{-1.245} \cdot \rho_L^{1.545} \cdot (1 + R_L)^{-0.657} \cdot d_{bean}^{1.8} \cdot \rho_G^{-3.49} \cdot P_{up}^{3.19} \quad \text{Eq. 4.146}$$

where:

σ	surface tension at upstream conditions (dynes/cm)
----------	---

Gilbert, Ros, Baxendall, Achong, Pilehvari and User defined correlations

The equations proposed by Gilbert, Ros, Baxendall, Archong and Pilehvari ([Ghassan \(p.586\)](#)) for stock tank critical liquid flow are all of the form:

$$q_L = \left[\frac{P_{up} \cdot (64d_{bean})^c}{a \cdot GLR^b} \right]^{1/e} \quad \text{Eq. 4.147}$$

where

GLR - producing gas liquid ratio (scf/STB)

a, b, c - empirical coefficient given below

Correlation	a	b	c	e
Achong	3.82	0.650	1.88	1
Baxendall	9.56	0.546	1.93	1
Gilbert	10	0.546	1.89	1
Pilehvari	46.67	0.313	2.11	1
Ros	17.4	0.5	2.00	1

Users can also define their own parameters for this formula by using engine [keywords \(p.673\)](#). For example:

```
CHOKE CCORR=USER a=0.1 b=0.546 c=1.89 e=1.0 ADJUSTSC dbean = 3
```

Keywords can be entered in the GUI by replacing the choke with an [Engine Keyword Tool \(p.94\)](#).

Flow Control Valves Mechanistic Theory

PIPESIM's mechanistic choke equation is based on the theory for subcritical flow (see Brill and Mukherjee, 1969).

Subcritical flow

The mass flow rate is given in terms of the pressure drop as follows:

$$Q_{sc} = 12A_{bean} \sqrt{\frac{2g\rho_{ns}\Delta P}{\left[\frac{f_L}{(Z_L c_L)^2} + \frac{f_G}{(Z_G c_G)^2} \right]}} \quad \text{Eq. 4.148}$$

where:

f_L and f_G	are the liquid and gas phase fraction
-----------------	---------------------------------------

c_L and c_G	are the liquid and gas flow coefficient
A_{bean}	is the choke cross-section

ΔP is the pressure drop, which is given by:

$$\Delta P = f_L \Delta P_L + f_G \Delta P_G \quad \text{Eq. 4.149}$$

and

$$\Delta P_L = \left(\frac{1}{2g\rho_{ns}} \right) \left[\frac{q}{(12Z_L c_L A_{bean})} \right]^2 \quad \text{Eq. 4.150}$$

$$\Delta P_G = \left(\frac{1}{2g\rho_{ns}} \right) \left[\frac{q}{(12Z_L c_L A_{bean})} \right]^2 \quad \text{Eq. 4.151}$$

where

$Z_L = 1$	is the liquid compressibility factor
$Z_G = Z_G(k, DP, P_{up})$	is the gas compressibility factor
$\rho_{ns} = f_L \rho_L + f_G \rho_G$	is the no slip density

Critical flow

The critical mass flow is given by the subcritical correlation evaluated at the critical pressure drop:

$$\Delta P_{crit} = P_{up} (1 - C_{PR}) \quad \text{Eq. 4.152}$$

where C_{PR} is the critical pressure ratio.

Fittings

The pressure drop across a fitting is given by the [Crane Technical Paper 410 \(p.585\)](#):

$$\Delta P = K \cdot \frac{\rho \cdot v^2}{2 \cdot c} \quad \text{Eq. 4.153}$$

K	is a dimensionless friction factor or resistance		
v	is the fluid velocity	ft/s	m/s
ρ	is the fluid density	lb/ft^3	kg/m^3
c	is a conversion factor for engineering units	$c = 144 \cdot g \cdot lb/(ft \cdot s^2) / psi$	$c = 1$

The velocity of the fluid in the fitting depends on the internal diameter of the fitting where the velocity is measured. If the fitting has two internal diameters, d_1 and d_2 , the velocities are related by:

$$\frac{\rho_1 \cdot \pi \cdot d_1^2 \cdot v_1}{4} = \frac{\rho_2 \cdot \pi \cdot d_2^2 \cdot v_2}{4} = q \quad \text{Eq. 4.154}$$

For incompressible fluids the density can be taken as constant and the velocities are inversely proportional to the square of the diameters. Therefore the pressure drop can be written in terms of either velocity:

$$\Delta P = K_1 \cdot \frac{\rho \cdot v_1^2}{2 \cdot c} = K_2 \cdot \frac{\rho \cdot v_2^2}{2 \cdot c} \quad \text{Eq. 4.155}$$

The fitting resistances are related by:

$$K_2 = \frac{1}{\delta^4} \cdot K_1 \quad \text{Eq. 4.156}$$

and $\delta = d_1 / d_2$ is the ratio of the internal diameters.

Comparison with the choke model

The fitting can be modeled as a [choke](#) ([p.443](#)) using a mechanistic [sub-critical](#) ([p.448](#)) liquid flow correlation. The choke diameter is taken as the minimum diameter of the fitting d_1 and the flow coefficient is calculated from the fitting resistance at d_1 :

$$c_v = \frac{1}{\sqrt{K_1}} \quad \text{Eq. 4.157}$$

Resistance calculation

The fitting resistance K can be specified by the user. Since it is a function of the internal diameter, d , this value must also be specified to allow the velocity to be calculated correctly.

The fitting resistance can also be calculated by PIPESIM using formulae from the [Crane Technical Paper 410](#) ([p.585](#)). The resistance is a function of the fitting type, the pipe nominal diameter, d_N the internal diameter, d_2 and the diameter of any constriction, d_1 . These [Crane Technical Paper 410](#) ([p.585](#)) formula can be written as:

$$K_1 = a_1 \cdot f_T(d_N) + a_2 \cdot 0.5 \cdot (1 - \delta^2) + a_3 \cdot (1 - \delta^2)^2 \quad \text{Eq. 4.158}$$

The first term $a_1 \cdot f_T$ represents friction due to the shape of the pipe fitting, the second term $a_2 \cdot 0.5 \cdot (1 - \delta^2)$ is the resistance due to sudden contraction through any constriction in the fitting

and the third term $a_3 \cdot (1 - \delta^2)^2$ is the resistance due to sudden expansion after a constriction. The constants a_1 , a_2 and a_3 depend on the fitting type and are given by:

Fitting	a_1	a_2	a_3
Check Swing Valve Conventional	100	0	0
Check Swing Valve Clearway	50	0	0
Standard 90 degree Elbow	30	0	0
Standard 45 degree Elbow	16	0	0
Standard 90 degree Short Radius Elbow	14	0	0
Standard 90 degree Long Radius Elbow	12	0	0
Tee - Flow through run	20	0	0
Tee - Flow through branch	60	0	0
Check Lift Globe Valve	600	δ	δ
Globe Valve Conventional	340	δ	δ
Angle Valve Conventional	150	δ	δ
Globe Valve Y-Pattern	55	δ	δ
Check Lift Angle Valve	55	δ	δ
Gate Valve $\theta < 45^\circ$	8	$1.6 \cdot \sin \frac{\theta}{2}$	$2.6 \cdot \sin \frac{\theta}{2}$
Gate Valve $45^\circ < \theta < 180^\circ$	8	$\sqrt{\sin \frac{\theta}{2}}$	1
Ball Valve $\theta < 45^\circ$	3	$1.6 \cdot \sin \frac{\theta}{2}$	$2.6 \cdot \sin \frac{\theta}{2}$
Ball Valve $45^\circ < \theta < 180^\circ$	3	$\sqrt{\sin \frac{\theta}{2}}$	1

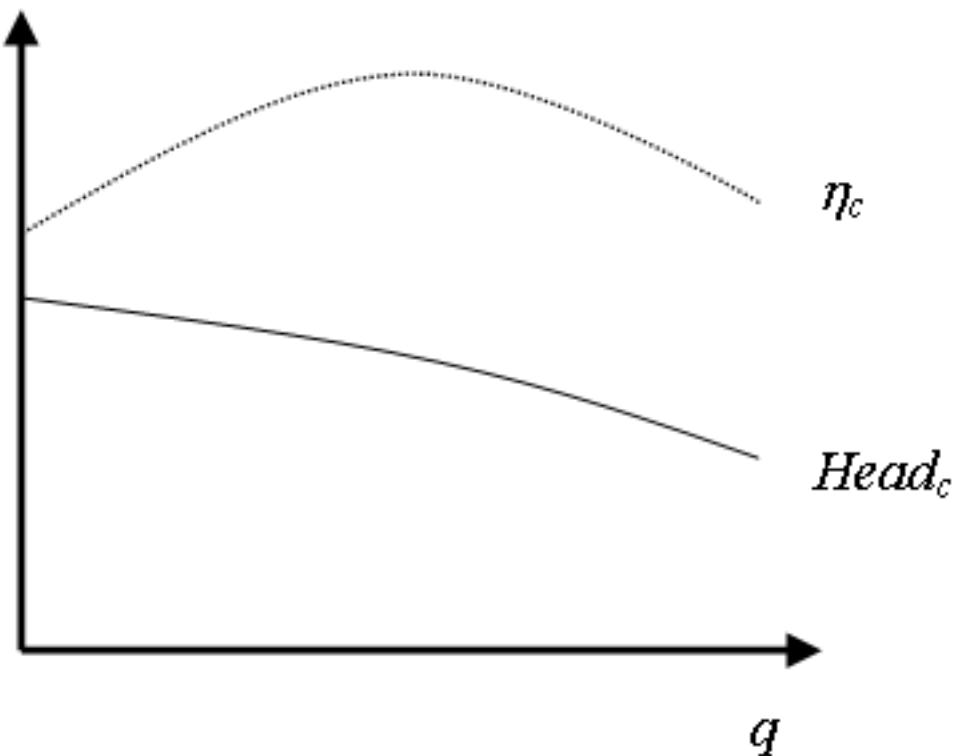
The friction factor f_T , depends on the nominal size of the pipe:

Nominal size d_N (inch)	f_T
1/2	.027
3/4	.025
1	.023
1 1/4	.022
1 1/2	.021

2	.019
2 1/2, 3	.018
4	.017
5	.016
6	.015
8 - 10	.014
12 - 16	.013
18 - 24	.012

4.3.2 Compressors, Pumps, and Expanders

Centrifugal Pumps and Compressors



Centrifugal pumps and compressors are described by curves of head and efficiency as functions of the flow rate for a given speed:

$$Head(q, N_c) = Head_c(q) \quad \text{Eq. 4.159}$$

$$\eta(q, N_c) = \eta_c(q) \quad \text{Eq. 4.160}$$

where:

$Head$	is the head	$ft \cdot lbf / lb$	Nm / kg
--------	-------------	---------------------	-----------

η	is the efficiency, expressed as a fraction, $0 < \eta \leq 1$		
q	is the flow rate	lb/s	kg/s
N_c	is the compressor speed for the curve		

The fan laws can be used to determine the head and efficiency for speeds N that differ from the curve speed :

$$Head(q, N) = \left(\frac{N}{N_c}\right)^2 Head_c\left(\frac{q}{N/N_c}\right) \quad \text{Eq. 4.161}$$

$$\eta(q, N) = \eta_c\left(\frac{q}{N/N_c}\right) \quad \text{Eq. 4.162}$$

The change in pressure of the fluid and the power needed to run the pump or compressor can be determined from the head and efficiency:

$$\Delta P = P_{out} - P_{in} = c_1 \cdot Head \cdot \langle\rho\rangle \quad \text{Eq. 4.163}$$

$$Power = \frac{c_2 \cdot q \cdot Head}{\eta} \quad \text{Eq. 4.164}$$

where

$\langle\rho\rangle = \frac{[\rho(P_{in}, T_{in})] + [\rho(P_{out}, T_{out})]}{2}$	is the average density	lb/ft^3	kg/m^3
P_{in}	is the suction pressure	psi or lbf/in^2	N/m^2
P_{out}	is the discharge pressure	psi or lbf/in^2	N/m^2
T_{in}	is the suction temperature	$^{\circ}R$	K
T_{out}	is the discharge temperature	$^{\circ}R$	K
$Power$	is the power required by the pump or compressor	hp	W
c_1	is a conversion factor for engineering units	$\frac{1}{144} \left(\frac{in}{ft}\right)^2$	
c_2	is a conversion factor for engineering units	$\frac{1}{550} \frac{hp}{ft \cdot lbf/s}$	

The outlet temperature depends on how much of the pump energy is transferred to the fluid. Three different models can be used:

Adiabatic Route:	$\Delta T = T_{out} - T_{in} = \frac{T_{in}}{\eta} \cdot \left(\left(\frac{P_{out}}{P_{in}} \right)^{\frac{\langle \gamma \rangle - 1}{\langle \gamma \rangle}} - 1 \right)$	Eq. 4.165
Polytropic Route:	$\Delta T = T_{out} - T_{in} = T_{in} \cdot \left(\left(\frac{P_{out}}{P_{in}} \right)^{\frac{\langle n \rangle - 1}{\langle n \rangle}} - 1 \right)$	Eq. 4.166
Mollier Route (Isentropic):	$S(P_{out}, T_{out}) = S(P_{in}, T_{in})$	Eq. 4.167

where

$\langle \gamma \rangle = \frac{[\gamma(P_{in}, T_{in})] + [\gamma(P_{out}, T_{out})]}{2}$	is the average value of γ		
$\gamma = \frac{C_p}{C_V}$	is the ratio of specific heats		
$\langle n \rangle = \frac{[n(P_{in}, T_{in})] + [n(P_{out}, T_{out})]}{2}$	is the average value of n		
$\frac{n}{n-1} = \eta \cdot \frac{\gamma}{\gamma-1}$	is the polytropic coefficient		
S	is the specific entropy	$\frac{BTU}{lb \cdot ^\circ F}$	$\frac{J}{kg \cdot K}$

Note that:

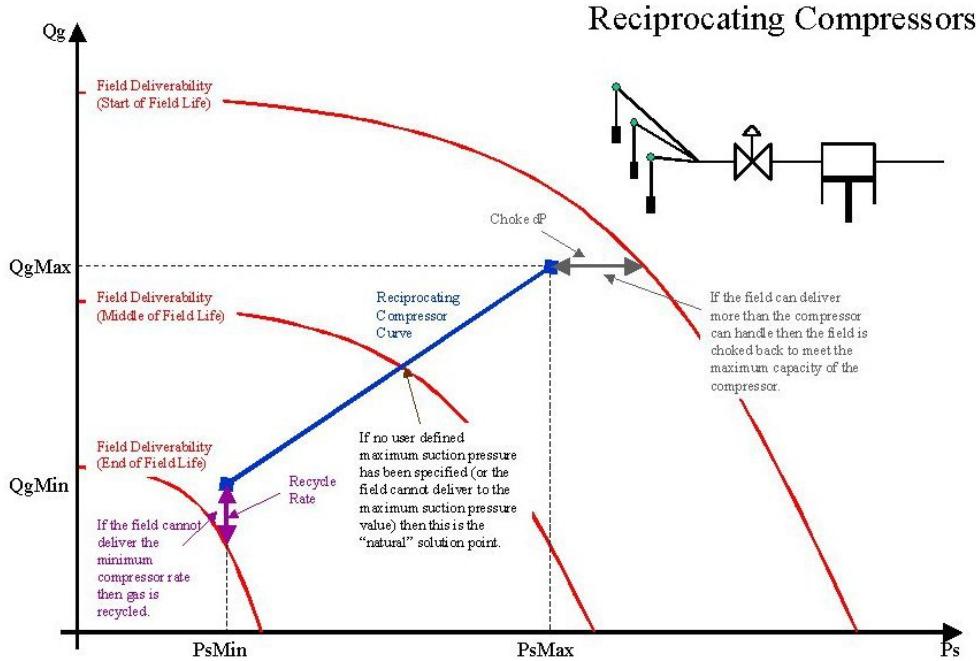
- Only a fraction η of the power is converted to head. When using the adiabatic route, the energy that is not converted to head is assumed to be converted to fluid heat. The usual adiabatic temperature increase is multiplied by a factor $1/\eta \geq 1$.
- The polytropic route $PV^n = constant$ can be used to model constant pressure ($n = 0$) , constant temperature ($n = 1$) , constant enthalpy ($n = \gamma$) and constant volume ($n = \infty$) changes as well as intermediate routes. PIPESIM uses a value of n that is a function of the efficiency (η) and the specific heat ratio (γ). This value can only be used when $\eta > (\gamma - 1)/\gamma$.
- In the special case when the efficiency $\eta = 1$, the polytropic coefficient equals the specific heat ratio $n = \gamma$ and the polytropic and adiabatic formulas are the same.
- The Mollier Route can only be used in compositional models, the PIPESIM blackoil model does not calculate entropy.

Engine keywords

See [compressor keywords \(p.679\)](#).

Reciprocating Compressor Operation

This graph shows how the reciprocating compressor will operate for various field deliverabilities:



Note the following:

1. If the field deliverability falls below the minimum compressor flowrate, recycle mode will be invoked. Due to the low pressure operation in this region, it may be necessary to add a reverse block to the branch containing the compressor.
2. If the field deliverability falls below the minimum suction pressure of the compressor, no solution is possible.

Note: This occurrence may be identified using FPT with an event conditional on the minimum rate through the compressor. To limit the flowrate through a compressor, place a choke in the branch upstream and specify a maximum gas constraint in FPT.

3. Always run the network model in **Wells Offline** mode, with no reverse blocks set.

Electrical Submersible Pumps (ESP)

General

The electric submersible pump (ESP) is perhaps the most versatile of the artificial lift methods. The ESP comprises a down hole pump, electric power cable, motor and surface controls. In a typical application, the down hole pump is suspended on a tubing string hung on the wellhead and is submerged in the well fluid. The pump is close-coupled to a submersible electric motor that receives power through the power cable and surface controls.

ESPs are used to produce a variety of fluids and the gas, chemicals and contaminants commonly found in these fluids. Aggressive fluids can be produced with special materials and coatings. Sand

and similar abrasive contaminants can be produced with acceptable pump life by using modified pumps and operation procedures.

ESPs usually do not require storage enclosures, foundation pads, or guard fences. An ESP can be operated in a deviated or directionally drilled well, although the recommended operating position is in a vertical section of the well.

The ESP has the broadest producing range of any artificial lift method ranging from 100 b/d of total fluid up to 90,000 b/d.

ESPs are currently operated in wells with bottom hole temperatures up to 350 degree Fahrenheit. Operation at elevated ambient temperatures require special components in the motor and power cables of sustained operation at high temperatures, and have efficiently lifted fluids in wells deeper than 12,000 ft. System efficiency ranges from 18 to 68%, depending on fluid volume, net lift and pump type.

ESP System Components: Motor

The ESP system's prime mover is the submersible motor. The motor is a two-pole, three-phase, squirrel-cage induction type. Motors run at a nominal speed of 3,500 rev/min in 60-Hz operation. Motors are filled with a highly refined mineral oil that provides dielectrical strength, bearing lubrication and thermal conductivity. The design and operation voltage of these motors can be as low as 230 volt or as high as 4,000 volt. Amperage requirement may be from 17 to 110 amps. The required horsepower is achieved by simply increasing the length of the motor section. The motor is made up of rotors, usually about 12 to 18 inches (300-460 mm) in length that are mounted on a shaft and located in the electrical field (stator) mounted within the steel housing. The larger single motor assemblies will approach 33 feet (10 m) in length and will be rated up to 400 horsepower, while tandem motors will approach 90 feet (27.5 m) in length and will have a rating up to 750 horsepower. The rotor is also composed of a group of electromagnets in a cylinder with the poles facing the stator poles. The speed at which the stator field rotates is the synchronous speed, and can be computed from the equation:

$$v = \frac{120 f}{M} \quad \text{Eq. 4.168}$$

Where: v is speed in rev/min, f is frequency in cycles/sec and M is number of magnetic poles.

The number of poles the stator contains is determined by the manufacturer. Therefore to change the speed of the stator magnetic field, the frequency will have to change.

Heat generated by the motor is transferred to the well fluid as it flows past the motor housing. Because the motor relies on the flow of well fluid for cooling, a standard ESP should never be set at or below the well perforations or producing interval, unless the motor is shrouded.

Motors are manufactured in four different diameters (series) 3.75, 4.56, 5.40 and 7.8 in. Thus motors can be used in casing as small as 4.5 in. 60-Hz horsepower capabilities range from a low of 7.5 hp in 3.75-in series to a high of 1,000 hp in the 7.38-in series.

Motor construction may be single section or several "tandems" bolted together to reach a specific horsepower. Motors are selected on the basis of the maximum diameter that can be run easily in a given casing size.

ESP System Components: Pumps

The ESP is a multistage centrifugal pump. Each stage of a submersible pump consists of a rotating impeller and a stationary diffuser. The pressure-energy change is accomplished as the liquid being pumped surrounds the impeller. As the impeller rotates it imparts two rotating motion components to the liquid: one is in a radial direction outward from the center of the impeller (centrifugal force), the other motion moves in a direction tangential to the outside diameter of the impeller. The resultant of these two components is the actual direction of flow. The type of stage used determines the rate of fluid production. The number of stages determines the total design head generated and the motor horsepower required. The design falls into one of two general categories: the smaller flow pumps are generally of radial flow design. As the pumps reach design flows of approximately 1,900 B/D, the design change to a mixed flow.

The impellers are of a fully enclosed curved vane design, whose maximum efficiency is a function of the impeller design and type and whose operating efficiency is a function of the percent of design capacity at which the pump is operated. The mathematical relationship between head, capacity, efficiency and brake horse power is expressed as:

$$\text{Power} = \frac{q_v H \gamma}{\eta} \quad \text{Eq. 4.169}$$

Where: q_v is the volume flow rate, H is the head, γ is the fluid specific gravity and η is the pump efficiency

The discharge rate of a submersible centrifugal pump depends on the rotational speed (rpm), size of the impeller, impeller design, number of stages, the dynamic head against which the pump is operating and the physical properties of the fluid being pumped. The total dynamic head of the pump is the product of the number of stages and the head generated by each stage.

"Bolt-on" design makes it possible to vary the capacity and total head of a pump by using more than one pump section. However, large-capacity pumps typically have integrated head and bases.

Pump Selection

Select **Artificial Lift » ESP » ESP Design** and use the **Pump Selection** tab . The tab has two sections, **Pump Design Data** and **Pump Parameters**. Select a pump based on certain design criteria.

Pump Design data

Design Production rate

Desired flowrate through the pump in stock-tank units. The actual flowing quantity will be computed.

Design Outlet Pressure

the required outlet pressure of the PIPESIM model when the pump is installed. It is recommended to only model the well, and no associated flowline or riser, while designing the ESP system. In this case the outlet pressure would then be the wellhead pressure

Static Reservoir Pressure

If you entered a value previously, that value is preserved. However, if this field is empty, the value is taken from the PIPESIM model.

Water cut

If you entered a value previously, that value is preserved. However, if this field is empty, the value is taken from the PIPESIM model.

GOR (or GLR)

If you entered a value previously, that value is preserved. However, if this field is empty, the value is taken from the PIPESIM model.

Pump Depth

The depth at which the pump is to be installed. This is taken from the PIPESIM model if a pump is already installed or can be entered.

Casing ID

The casing size that the pump has to fit into. Usually 3.38 to 11.25 in.

Design Frequency

The frequency/speed that the pump is expected to run at.

Gas Separator Efficiency

The efficiency of the gas separator if installed.

Head factor

Allows the pump efficiency to be factored (default = 1).

Viscosity Correction

All pump performance curves are based on water systems, this option will correct for oil viscosity.

Select Pump

Will use the available data to select suitable pumps from the database. The pump intake conditions will first be computed. The resulting pump list can be sorted by efficiency or Maximum flowrate by selecting the column header. The Manufacturers to select from can be filtered. [Errors in the simulation \(p.241\)](#).

Pump parameters**Calculate**

Calculate pump performance at the conditions specified. [Errors in the simulation \(p.241\)](#)

Stage-by-stage

Perform the stage calculations on a stage-by-stage basis. Default = stage-by-stage.

Selected Pump

The pump selected, by the user, from the design data

No. of Stages required

The computed number of stages for this pump under these conditions.

Pump efficiency @ Design rate

The efficiency of the pump at the design production rate

Pump power required

The power required for this pump to deliver the required flowrate.

Pump intake pressure

The computed pump intake pressure.

Pump discharge pressure

The computed pump discharge pressure.

Head required

The computed pump head required

Liquid density

The computed liquid density at the pump intake

Free gas fraction at inlet conditions

The computed gas fraction.

Pump performance plot

plot performance curves at different speeds

Pump curves

plot standard performance curves

Install pump

Install the pump into the tubing of model. This will replace any existing ESP but not gas lift valves.

See also: [Select a Motor \(p.235\)](#), [Select a Cable \(p.241\)](#)

Motor Selection

This can only be performed after a pump has been selected.

Select **Artificial Lift** » **ESP** » **ESP Design** and use the **Motor/Cable Selection** tab.

Name of the selected Pump

Selected from the **Pump** tab.

No. of stages

Computed.

Pump efficiency

Computed

Pump power required

Computed.

Select Motor

The resulting motor list can be sorted by power, voltage, current, etc. by selecting the column header. [Errors \(p.235\)](#).

Various parameters associated with the motor will be computed and displayed at both 60Hz and the initially entered Design Frequency.

NP [Name Plate] Power

NP [Name Plate] Voltage

NP [Name Plate] Current

See also: [Select a Pump \(p.236\)](#), [Select a Cable \(p.241\)](#)

ESP Database

To simulate an ESP, PIPESIM maintains a database of manufacturers and models from which the user can select. For each model the diameter, minimum and maximum flowrate and base speed are provided. A plot of the ESP's performance is also available. If the required ESP is not in the database, you can easily enter the basic data required for it into the database using **Data » New ESP/Pump/Compressor**. See [Data/NewESP-Pump-Compressor \(p.242\)](#).

Selection

When modeling an ESP, it is important that the correct size (expected design flowrate and physical size) ESP is used. A search facility is available, based on these two parameters, to select the appropriate ESP from the database. The search can, if required, be restricted to a particular manufacturer. Pumps that meet the design criteria will be listed.

Stage-by-stage modeling

Stage-by-stage modeling is selected by selecting the checkbox next to the calculate button. Alternatively by inserting Engine Keywords (**PUMP STAGECALCS**) (p.701) into the model, using the [EKT \(p.94\)](#).

Install a Pump

Once the ESP [manufacturer and model \(p.235\)](#) has been selected from the [database of common ESP's \(p.239\)](#) some parameters can be altered. The performance curves for each model are (normally) based on a Speed of 60Hz and 1 stage.

Design data

Speed

The actual operating speed of the ESP

Stages

The actual number of stages of the ESP

Head factor

Allows the efficiency to be factored (default = 1)

Calculation Options

Viscosity Correction

Allow a viscosity correction factor to be applied to take account of changes to the fluid viscosity by the pressure and temperature.

Gas Separator present

Allow a gas separator to be added (automatically) with an efficiency: Separator efficiency - efficiency of an installed gas separator (default = 100% if installed)

Performance table

The data used to predict the performance of the ESP

Standard Curves

The standard performance curves for the ESP - can be printed/exported

Variable Speed Curves

Variable speed curves at 30 - 90 Hz.- can be printed/exported

ESP Design

The ESP option is selected from the Artificial Lift menu. To design an ESP the following stages are required:

[Select a Pump \(p.236\)](#)

[Select a Motor \(p.235\)](#)

[Select a Cable \(p.241\)](#)

The ESP should then be installed, added into the tubing, at the required depth. This can either be performed manually or by using the **Install** button. Installing automatically removes any existing ESPs in the tubing. However, any gas lift values or injections points are not removed.

See also: [ESP \[Reda\] web site](#)

ESP System Components: Cable

Cable Selection can be determined after a Pump and motor have been selected.

1. Motor/Cable Selection tab
2. Cable Selection
3. Select Cable

Cable Length

The length of the cable, can be modified

NP Current @ Design Frequency

The [Name Plate] Current at the design frequency. Cannot be changed.

Computed values

Selected Cable

Cable length

Voltage drop

Downhole voltage

Surface voltage

Total System KVA

Design Report

Display a report that details all the selected components of the ESP system.

Errors

Occasionally a pump may not be able to be determined and a Convergence error will be reported. There could be a number of reasons for such an error and the user is advised to view the output report.

Common problems:

1. The system cannot reach the outlet pressure specified. Try increasing the outlet pressure.

Progressive Cavity Pump (PCP)

General

Progressive Cavity Pumps (PCP's) are a special type of rotary positive displacement pump sometimes referred to as "single-screw pumps". Unlike ESP's, PCP performance is based on the volume of fluid displaced and not on the pressure increase dynamically generated through the pump. PCP's are an increasingly common form of artificial lift for low- to moderate-rate wells, especially onshore and for heavy (solid laden) fluids.

Invented in the late 1920's by Rene Moineau, PCP's were not used in the oilfield until the late 1970's. Their use is becoming increasingly common for low- to moderate- rate onshore wells, particularly for heavy-oil and sand-laden fluids. (See [SPE Production Engineering Handbook \(p. 0 \)](#).)

PCP systems have several advantages over other lift methods:

- Overall high energy efficiency (typically 55-75%)
- Ability to handle solids
- Ability to tolerate free gas
- No valves or reciprocating parts
- Good resistance to abrasion
- Low internal shear rates (limits fluid emulsification through agitation)
- Relatively lower power costs (prime mover capacity fully utilized)
- Relatively simple installation and operation (low maintenance)
- Low profile surface equipment and noise levels

Limitations of PCP systems include:

- Maximum production rates of approximately 5000 BPD
- The maximum installation depth is about 4,500 ft.
- Maximum operating temperature of approximately 300 ° F.
- Corrosive fluids may damage elastomer and result in higher slippage
- Pump stator may sustain permanent damage if run dry even for short periods
- Rod sting and tubing wear can be problematic for directional and horizontal wells (though downhole drives can be used to avoid this)

Principle of Operation

PCP's are most commonly driven by surface mounted electrical motors (Figure 4.7 (p.466)), although downhole electric and hydraulic drive systems are available.

A PCP is comprised of two helical gears, a steel rotating gear called the rotor ("internal gear") and a stationary gear called the stator ("external gear"), which is commonly made of elastomer but may be steel as well. The rotor is positioned inside the stator and rotates along the longitudinal axis (Figure 4.7 (p.466)):

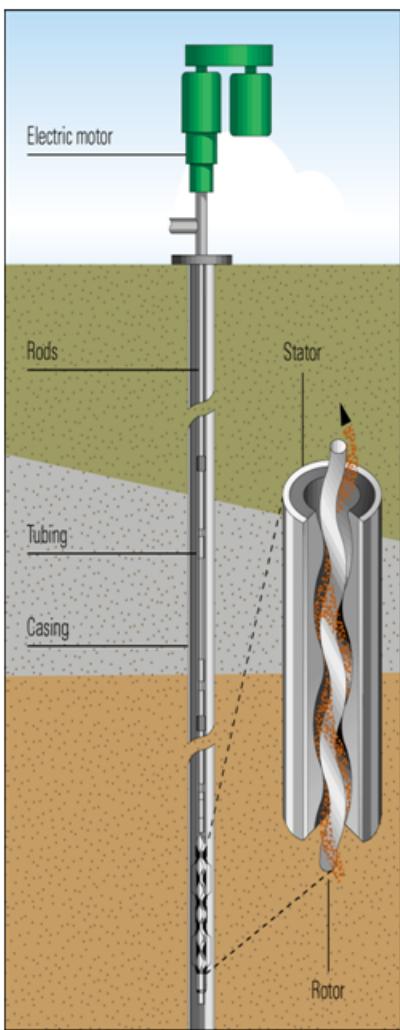


Figure 4.7. PCP Pump Illustration

The volume between the stator and rotor forms a sealed cavity that traps the fluid and as the rotor turns this cavity "progresses" the fluid from the inlet to the outlet of the pump. The volume of the cavity and the rotational speed (N) determine the flow rate achieved by the pump.

The volume of the cavity may be calculated based on geometric parameters. The volume of the cavity is defined by the diameter of the rotor (D_r) times the stator pitch length (L_s) times the eccentricity (e). The eccentricity is defined as the distance between the centerlines of the major and minor diameters of the rotor.

Therefore, the flow rate through the pump can be expressed as:

$$Q = 4eD_rL_sN$$

In field units, P_s , e and D are in feet and N in *revolutions per-minute* to give a rate in ft³/min.

Multiply by 256.46 to convert to BPD. (See [Bellarby \(p. 0\)](#).) The geometric parameters required for this calculation vary considerably among vendors and are generally not published.

Hydraulic power can then be calculated by:

$$Hhp = 1.7 \times 10^{-5} \Delta PQ$$

Where ΔP is the pressure differential across the pump (psi) and Q is rate (BPD).

In practice, the clearance between the rotor and stator are not perfect due mainly to deformation of the elastomeric stator as a function of pressure, temperature, and wear. This causes some of the fluid to slip back into preceding cavities. Slip increases with increasing pressure and number of stages. Higher viscosity fluids exhibit less slip.

For simulation purposes, PCP performance curves are generally used. While the format of performance curves varies by vendor, PIPESIM has adopted the format suggested by [ISO 15136-1 \(2009\) \(p. 0\)](#). PIPESIM provides performance curves from several vendors based on reference conditions (generally water at standard conditions). While catalog performance curves for rotodynamic-type pumps (such as ESP's) are generally consistent with field performance, PCP performance curves vary considerably based on the operating conditions (pressure and temperature) as well as the fluid properties. Therefore, the catalog curves available from within PIPESIM should only be used for preliminary analysis. It is common for PCP's to undergo "bench tests" to generate performance curves for specific pumps at intended operating conditions. It is therefore recommended that these curves be used for more detailed simulation studies.

Viscosity Effects

PIPESIM has the option to apply a viscosity correction to reduce slippage effects for higher viscosity fluids. The method of [Karassik et al. \(p. 0\)](#) is used.

$$\frac{q_{v2}}{q_{v1}} = \sqrt{\frac{v_1}{v_2}} \quad (1)$$

Eq. 4.170

Where v = the kinematic viscosity, SSU (Saybold Seconds Universal)

q = the slippage, (BPD)

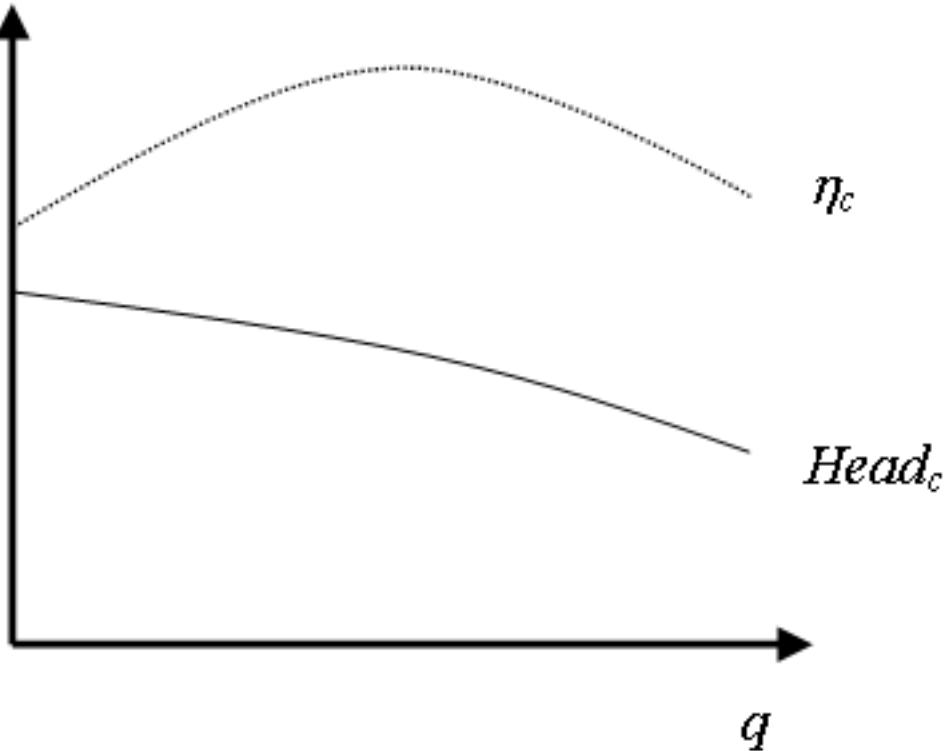
The range of kinematic viscosity is 100 to 10,000 SSU for this viscosity correction. If the reference fluid is water with kinematic viscosity of about 32 SSU, the equation reduces to:

$$q_{s(v2)} = \sqrt{\frac{32}{v_2}} q_{s(curve)} \quad (2)$$

Eq. 4.171

Note: SSU is a viscosity unit that is equal to the measure of the time that 60 cm³ of oil takes to flow through a calibrated tube at a controlled temperature. This should not be confused with the dynamic (absolute) viscosity, unit of cp or Pa·s.

Expanders



Expanders are modeled in a similar way to [centrifugal compressors \(p.455\)](#), except they work in reverse. Fluid flows through the expander and power is extracted. As with compressors, expanders can be described by curves of head and efficiency as functions of the flow rate for a given speed:

$$Head(q, N_c) = Head_c(q) \quad \text{Eq. 4.172}$$

$$\eta(q, N_c) = \eta_c(q) \quad \text{Eq. 4.173}$$

where:

$Head$	is the head	$ft \cdot lbf / lb$	Nm / kg
η	is the efficiency, expressed as a fraction, $0 < \eta \leq 1$		
q	is the flow rate	lb / s	kg / s
N_c	is the expander speed for the curve		

The fan laws can be used to determine the head and efficiency for speeds N that differ from the curve speed :

$$Head(q, N) = \left(\frac{N}{N_c}\right)^2 Head_c\left(\frac{q}{N/N_c}\right) \quad \text{Eq. 4.174}$$

$$\eta(q, N) = \eta_c\left(\frac{q}{N/N_c}\right) \quad \text{Eq. 4.175}$$

The change in pressure of the fluid and the power needed to run the pump or compressor can be determined from the head and efficiency:

$$\Delta P = P_{in} - P_{out} = c_1 \cdot Head \cdot \langle\rho\rangle \quad \text{Eq. 4.176}$$

$$Power = c_2 \cdot \eta \cdot q \cdot Head \quad \text{Eq. 4.177}$$

where

$\langle\rho\rangle = \frac{[\rho(P_{in}, T_{in})] + [\rho(P_{out}, T_{out})]}{2}$	is the average density	lb/ft^3	kg/m^3
P_{in}	is the suction pressure	psi or Ibf/in^2	N/m^2
P_{out}	is the discharge pressure	psi or Ibf/in^2	N/m^2
T_{in}	is the suction temperature	oR	K
T_{out}	is the discharge temperature	oR	K
$Power$	is the power extracted by the expander	hp	W
c_1	is a conversion factor for engineering units	$\frac{1}{144} \left(\frac{in}{ft}\right)^2$	
c_2	is a conversion factor for engineering units	$\frac{1}{550} \frac{hp}{ft \cdot Ibf/s}$	

The outlet temperature depends on how much of the energy is removed from the fluid. Three different models can be used:

Adiabatic Route:	$\Delta T = T_{out} - T_{in} = \eta \cdot T_{in} \cdot \left(\left(\frac{P_{out}}{P_{in}} \right)^{\frac{\langle\gamma\rangle-1}{\langle\gamma\rangle}} - 1 \right)$	Eq. 4.178
-------------------------	--	-----------

Polytropic Route:	$\Delta T = T_{out} - T_{in} = T_{in} \cdot \left(\left(\frac{P_{out}}{P_{in}} \right)^{\frac{\langle n \rangle - 1}{\langle n \rangle}} - 1 \right)$	Eq. 4.179
--------------------------	---	-----------

Mollier Route (Isoentropic):	$S(P_{out}, T_{out}) = S(P_{in}, T_{in})$	Eq. 4.180
-------------------------------------	---	-----------

where

$\langle \gamma \rangle = \frac{[\gamma(P_{in}, T_{in})] + [\gamma(P_{out}, T_{out})]}{2}$	is the average value of γ		
$\gamma = \frac{C_p}{C_v}$	is the ratio of specific heats		
$\langle n \rangle = \frac{[n(P_{in}, T_{in})] + [n(P_{out}, T_{out})]}{2}$	is the average value of n		
$\frac{n}{n-1} = \frac{1}{\eta} \cdot \frac{\gamma}{\gamma-1}$	is the polytropic coefficient		
S	is the specific entropy	$\frac{BTU}{lb \cdot ^\circ F}$	$\frac{J}{kg \cdot K}$

Notes:

- Only a fraction η of the head is converted to power. When using the adiabatic route, the energy that is not converted to power is assumed to be converted to fluid heat. The usual adiabatic temperature decrease is multiplied by a factor $\eta \leq 1$.
- The polytropic route $PV^n = \text{constant}$ can be used to model constant pressure ($n = 0$), constant temperature ($n = 1$), constant enthalpy ($n = \gamma$) and constant volume ($n = \infty$) changes as well as intermediate routes. PIPESIM uses a value of n that is a function of the efficiency (η) and the specific heat ratio (γ). This value can only be used when $\eta < \gamma/(\gamma - 1)$.
- In the special case when the efficiency $\eta = 1$, the polytropic coefficient equals the specific heat ratio $n = \gamma$ and the polytropic and adiabatic formulas are the same.
- The Mollier Route can only be used in compositional models; the PIPESIM blackoil model does not calculate entropy.

Engine keywords

See [expander keywords \(p.683\)](#)

4.3.3 Multiphase Boosting Technology

Multiphase boosting technology (also referred to as multiphase pumping technology) for the oil and gas industry has been in development since the early 1980s, and is now rapidly gaining acceptance as a tool to optimize multiphase production systems (Oxley, Ward and Derk 1999). Multiphase boosting has been recognized as a vital technology, preferable to the standard approach of separation, gas compression, liquid pumping and the use of dual flow lines back to the host facility. It is particularly beneficial for the development of satellite fields. Multiphase boosting enables the full (non-separated) well stream to be boosted in a single machine, thus greatly

simplifying the production system, resulting in significant cost savings that in many scenarios, have made the development of marginal fields, economic.

Since 1990, thousands of multiphase boosters have been installed worldwide, with the vast majority of the installations based onshore or offshore topsides. Over the years, the development of multiphase boosting has led to two categories of commercial boosters:

Positive Displacement

The most common types are the Twin screw type & Progressive cavity type multiphase boosters.

Rotodynamic

The most common type is the Helico-axial type multiphase booster.

The figure below depicts the difference between multiphase boosting technology and the more traditional technology of separation, pumping and compression.

Traditional Multiphase Production approach	
<ul style="list-style-type: none"> • The incoming fluid is separated into its constituent liquid and gas phases. • The separated liquids are pumped up to the required pressure and exported via the liquid export line. • Separated gas is compressed up to the required pressure and exported via the gas export line. 	
Alternative Multiphase Production approach	
<ul style="list-style-type: none"> • The incoming fluid is separated into its constituent liquid and gas phases. • The separated liquids are pumped up to the required pressure and separated gas is compressed up to the required pressure. • The two phases are recombined and exported using a multiphase export line. 	
Multiphase Boosting	
<ul style="list-style-type: none"> • The incoming fluid is directly boosted up to the required pressure without separation of the gas and liquid phases. • It is exported using a multiphase export line. 	

Multiphase boosters are pumps/compressors that can accommodate fluids ranging from 100% liquid to 100% gas, and anywhere in between. Although commonly referred to as multiphase pumps, the terminology used in this document is 'multiphase booster' to recognize the fact that

100% gas can also be handled by this equipment (albeit with some restrictions, as outlined in later sections of this topic).

Multiphase boosters are used primarily for the following reasons:

Production Enhancement

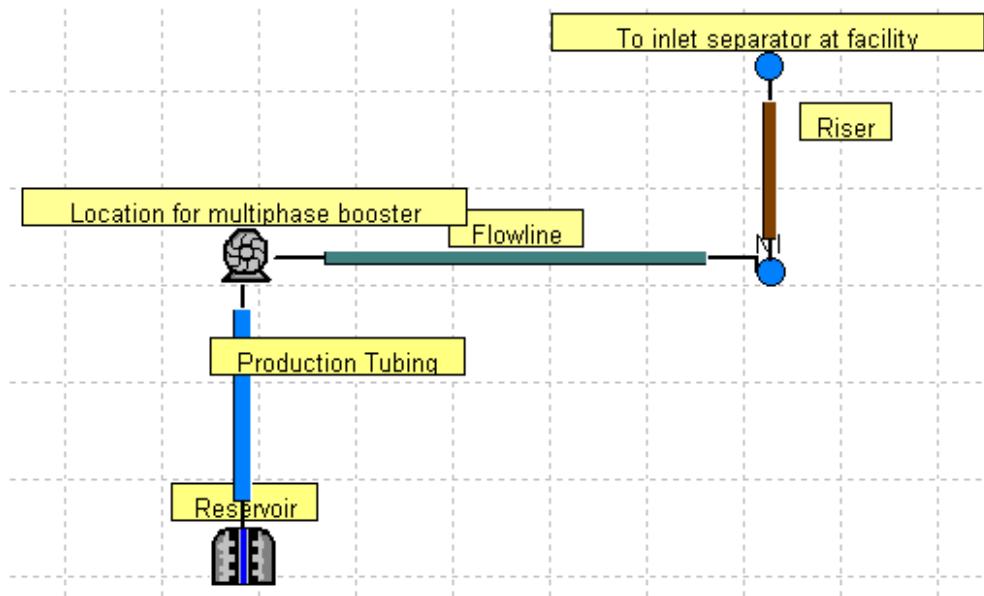
Multiphase boosting helps to accelerate and/or increase hydrocarbon production by lowering backpressure on wells.

Pressure Boosting

Multiphase boosting increases fluid pressure thus enabling the transportation of multiphase fluids over long distances. It also helps to move fluids from low pressure systems to higher pressure systems.

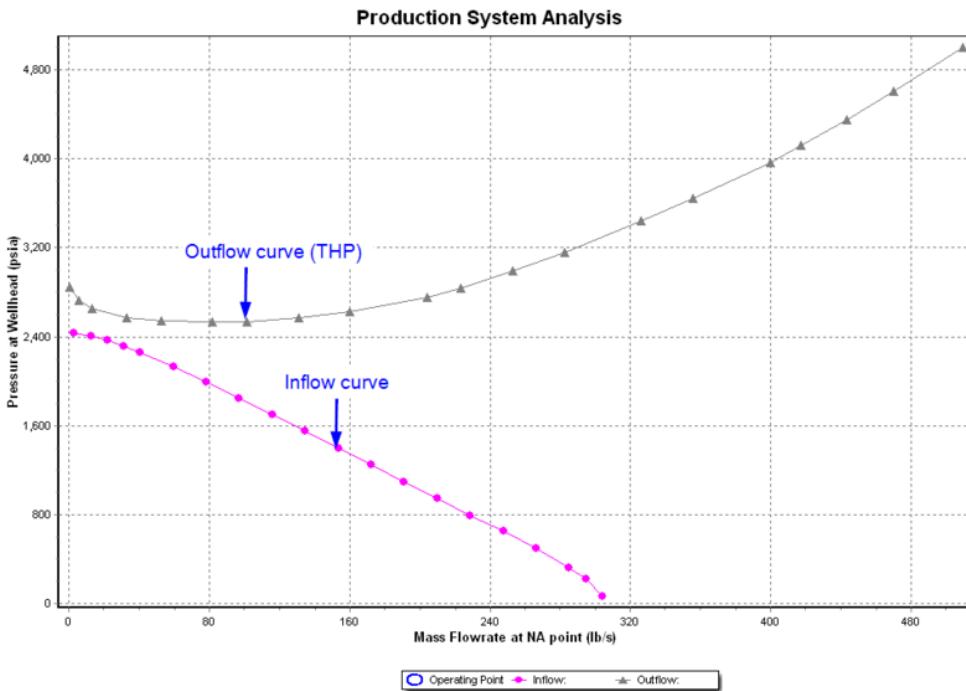
In many cases, Multiphase boosters will deliver the combined benefit of production enhancement and pressure boosting. For example, lowering the backpressure on a well by using a multiphase booster may increase the rate and simultaneously supply the fluid at a higher pressure at the flowline inlet.

To demonstrate the principle of multiphase boosting, take the example of a well which is connected using a flowline and riser to the inlet separator on the host facility, as in the following diagram.



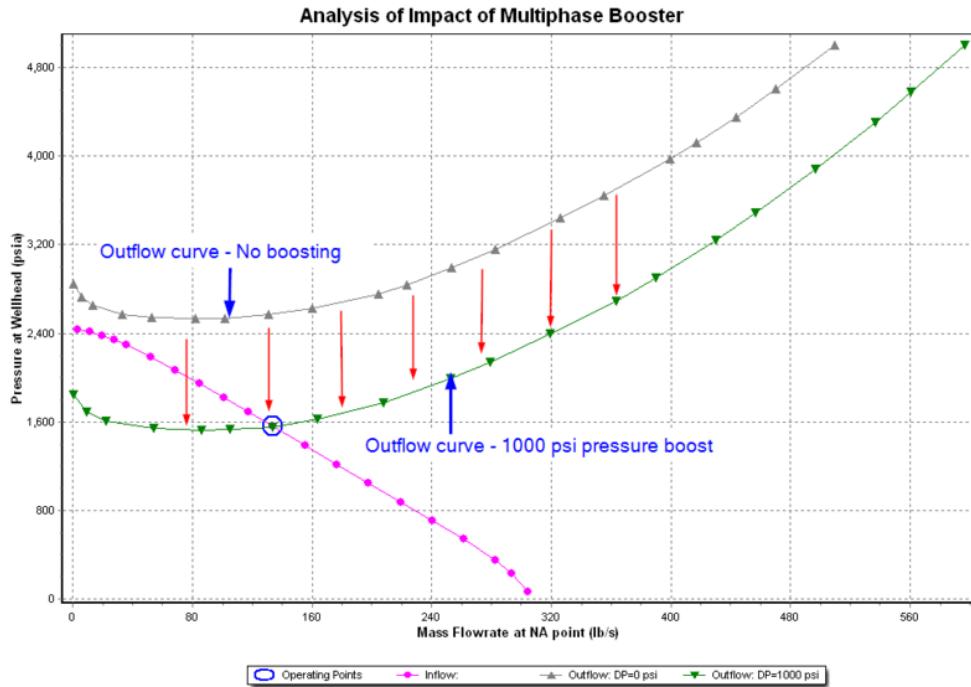
If the Wellhead is selected as the node, the inflow would represent the P-Q (pressure-flowrate) relationship from the reservoir up to the wellhead and the outflow would represent the P-Q relationship downstream of the wellhead, including the multiphase booster, flowline and riser. Both Inflow and Outflow are represented in the Systems plot. The point of intersection of the two curves is the system operating point, for example, the flowing wellhead pressure and production rate the system would operate at. In this example, there is no intersection point, which indicates that the system would not produce at these conditions.

Production system analysis: THP curve and outflow curve



For this example, you can see from the Production System Analysis graphic that the system is incapable of producing naturally. From the THP curve, it is clear that if the back pressure on the well could be lowered, production could be restored. Assuming that you could install a booster directly downstream of the wellhead, that would provide a pressure 'boost' of 1000 psi to the well fluids, the outflow curve could be lowered as shown in the figure below. The system would now produce 134 lb/s (32,412 stb/d of liquid) at a flowing wellhead pressure of 1554 psia. In this example, multiphase boosting has transformed a dead well to one that produces over 30,000 stb/d of fluid.

Production system analysis: the effect of multiphase boosting visualized



Through the type of analysis outlined above, the effect of multiphase boosting on a production system can be easily evaluated, and the requirements of the multiphase booster such as power requirement, speed, etc. can be determined.

Positive Displacement Multiphase Pumps

Positive displacement type pumps work by transferring a definite amount of fluid through a pumping chamber operating at a particular speed. As the fluid is passed from the suction side to the discharge end, differential pressure is added hydrostatically rather than dynamically, which results in these pumps being less sensitive to fluid density than rotodynamic type pumps. This feature makes positive displacement type pumps more attractive for surface installations than rotodynamic type pumps. This is primarily because fluids at surface conditions are at lower pressures and temperatures and tend to have higher gas fractions and a greater tendency for density change than fluids at subsea conditions (Butler 1999).

There are four (4) types of Positive displacement pumps: Twin Screw, Progressive Cavity (Single Screw), Piston & Diaphragm, but commercial development has focused mainly on the Twin Screw and Progressive Cavity types.

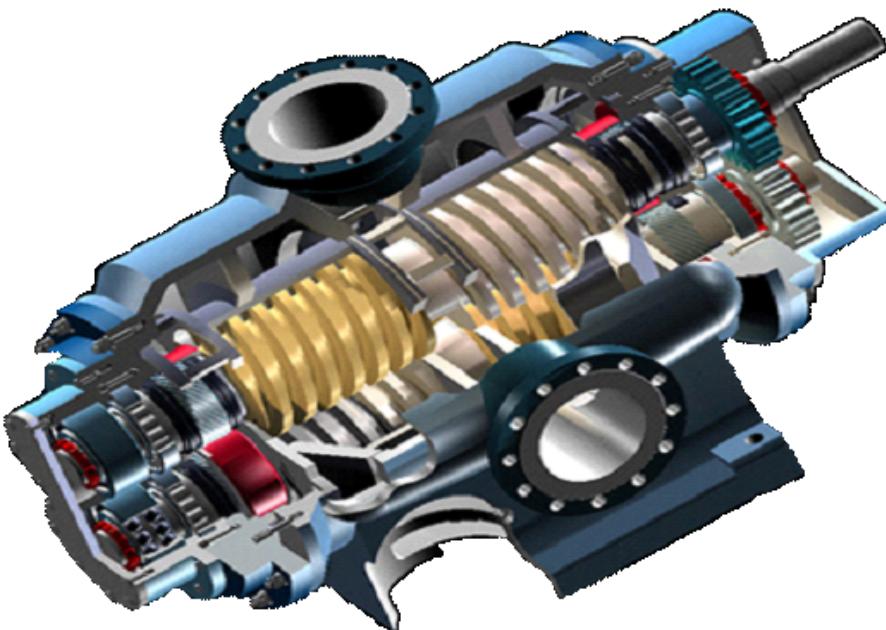
The majority of positive displacement type multiphase boosters on the market are of the Twin screw type, and they will be the primary focus of this topic.

Twin Screw Type

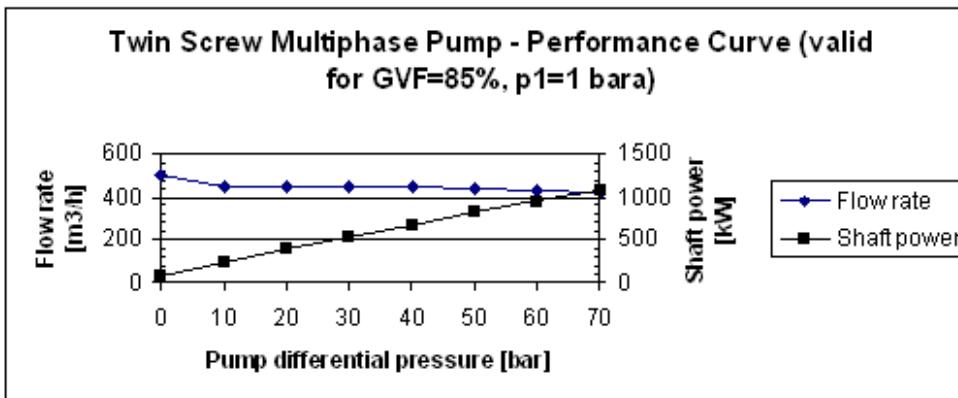
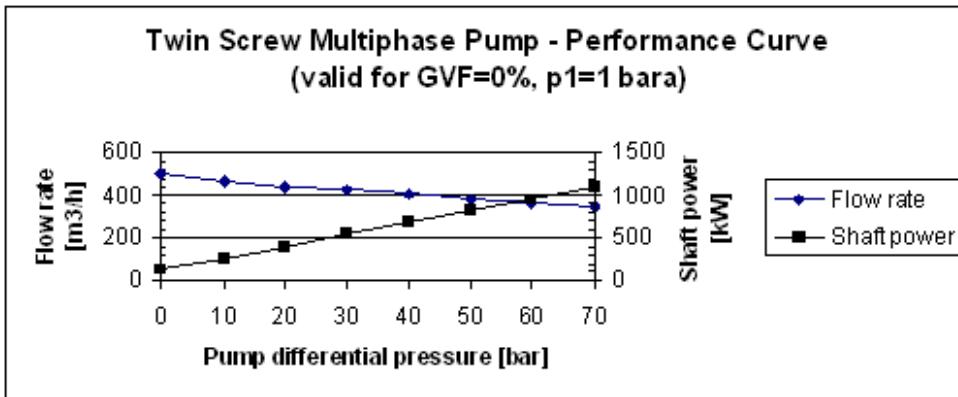
The twin screw type booster, also referred to as two-spindle screw pump, works on the basis of fluid carried between the screw threads of two intermeshing feed screws and displaced axially as the screws rotate and mesh. The fluid is split into two inlets on opposite sides of the pumps. This equalizes stresses associated with slugging and better enables this type of pump to handle fluctuating inlet conditions. The fluid passes through a chamber created by the twin interlocking

screws and moves along the length of the screws to the outlet at the top of the pump. The volumetric rate pumped depends on the screw pitch, diameter and rotational speed.

The following figure shows an example of a twin screw type pump.



It should be noted that, unlike screw type compressors, the volume of the chambers is not reduced from pump suction to pump discharge, for example, there is no in-built compression in the twin screw type multiphase boosters. Pressure buildup in the twin screw type multiphase booster is entirely based on the fact that a definite amount of fluid is delivered into the outlet system with every revolution of the feed screws, and the pressure developed at pump discharge is solely the result of resistance to flow in the outlet system. Additionally, as the fluid makes its way from suction to discharge, gas is compressed and liquid slips back, resulting in a reduction in the volumetric efficiency of the pump. This is due to the development of a pressure gradient across the moving chambers from pump discharge to suction, which causes an internal leakage in the pumping elements. This internal leakage/slip causes the pump net flow to be less than its theoretical capacity, as demonstrated in the pump performance curves shown below.



As can be seen from the typical pump performance curves above, pump flow rate is dependent on pump differential pressure: the higher the pump differential pressure, the higher the internal leakage, and thus the lower the pump flow rate.

The theoretical capacity of the pump, i.e. the flow rate if no internal leakage is present; is the flow rate at zero pump differential pressure. For the pump represented in the pump performance curves above, its theoretical capacity is 500 m³/h. The difference between the theoretical flow rate and the actual flow rate, is the internal leakage, also called 'pump slip'. As an example, for the pump represented in the GVF=0% pump performance curve, the actual flow rate for a pump differential pressure of 40 bar, would be 400 m³/h, and the pump slip would be 100 m³/h (for example, 500 - 400). Given the relative insensitivity of flow rate to differential pressure, especially at higher GVF's, the twin screw multiphase booster is sometimes referred to as a 'constant flow rate' pump. The twin screw pump is good for handling GVF's up to 98% at suction conditions and is the preferred technology for high viscosity fluids.

It can be seen from the pump performance curves that pump flow rate is dependent on GVF, but GVF has minimal impact on pump shaft power.

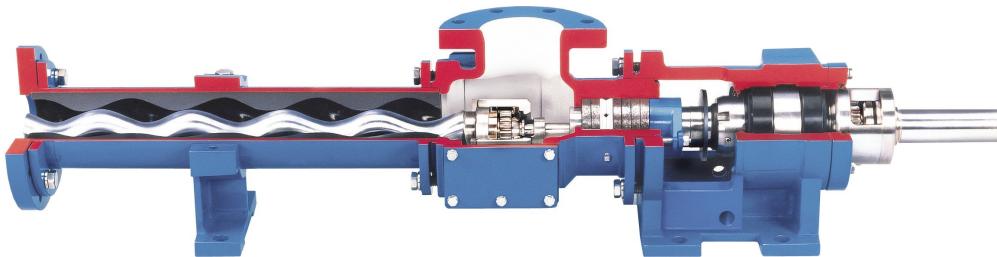
The pump performance curves may suggest that there is an unlimited variety of twin screw multiphase pumps available for an unlimited number of DP-Q (differential pressure - flow rate) combinations; however, in practice, there are several physical limitations that restrict pump options, as below:

- Pump differential pressure is typically limited to 70 bar to avoid excessive deflection of feed screws and possible contact between rotating screws and stator housing

- Pump flow rate (i.e. total volumetric flow rate at pump suction) is presently limited to approximately 2000 m³/h per pump
- Gas volume fraction at pump suction is typically limited to 95% maximum (for GVF > 95%, some form of liquid recirculation is typically required to maintain GVF at suction at 95% maximum)
- Pump inlet pressure and outlet pressures are restricted by casing design pressure and seal design pressure

Progressive Cavity Type

The progressive cavity type pump (also known as single-rotor screw pump) operates on the basis of an externally threaded screw, also called rotor, turning inside an internally threaded stator. It is the same artificial technology used in wells for production enhancement that was adapted for surface multiphase pumping.



As with the screw type pump, as the rotor rotates within the stator, chambers are formed and filled with fluid that progress from the suction side of the pump to the discharge side of the pump. The continuous seal line between the rotor and the stator helix keeps the fluid moving steadily at a fixed flow rate proportional to the pump rotational speed. Application of the progressive cavity type pump for multiphase boosting has been less widespread than the twin screw type multiphase booster, and flow rates and differential pressures are typically lower than those achievable with the twin screw type (< 30,000 bbl/d total volume).

An example of a progressive cavity type pump for multiphase applications is Moyno's R&M Tri-Phaze® System, which is considered one of the largest; capable of transferring multiphase flows up to 29,000 bbl/day (192 m³/h) at differential pressures up to 300 psi (20.7 bar). Progressive cavity pumps can tolerate high solids content and can be adapted to deliver higher flow rates and differential pressures by installing them in series or parallel arrangements, which increases the complexity (Mirza 1999).

Given the wider operating range and greater popularity of twin screw pumps in the oil and gas industry, PIPESIM has chosen to focus its modeling capabilities in the positive displacement category of multiphase boosters, on twin screw pumps.

Rotodynamic Multiphase Pumps

Rotodynamic type pumps work by adding kinetic energy to the fluid, which is then converted to pressure, thus boosting the fluid. The actual increase in pressure is directly proportional to the density of the pumped fluid, for example, the higher the fluid density, the higher the pressure increase. Because of this, dynamic type pumps are more sensitive to fluid density than positive displacement type pumps, and tend to be used in applications with lower maximum gas volume fractions; for example, in subsea applications.

The commercial development of dynamic type multiphase boosters has been focused on the helico-axial type, based on helico-axial hydraulics developed and licensed by Institute François du Petrole (IFP). For very high gas volume fractions (GVF > 95%), the contra-rotating axial (CRA) machine was specially developed; originally by Framo Engineering AS and Shell.

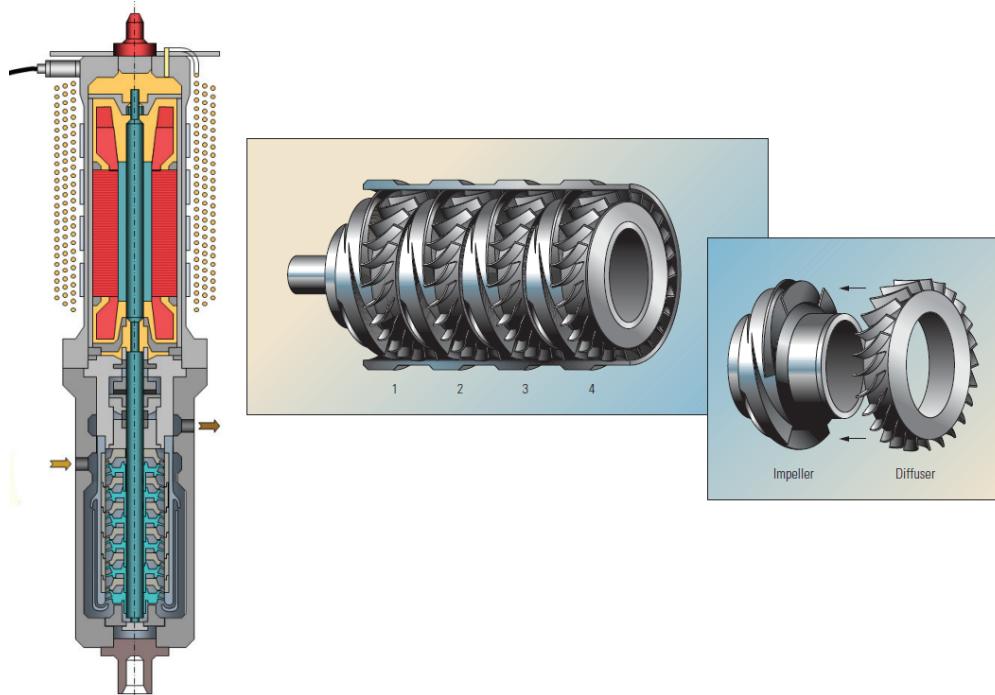
The design of the helico-axial type pump has also concentrated on its driver mechanism. For subsea use, there are electric motor driven units as well as hydraulic turbine driven units. For onshore or offshore topsides applications, other driver types can also be used.

Helico-Axial Type

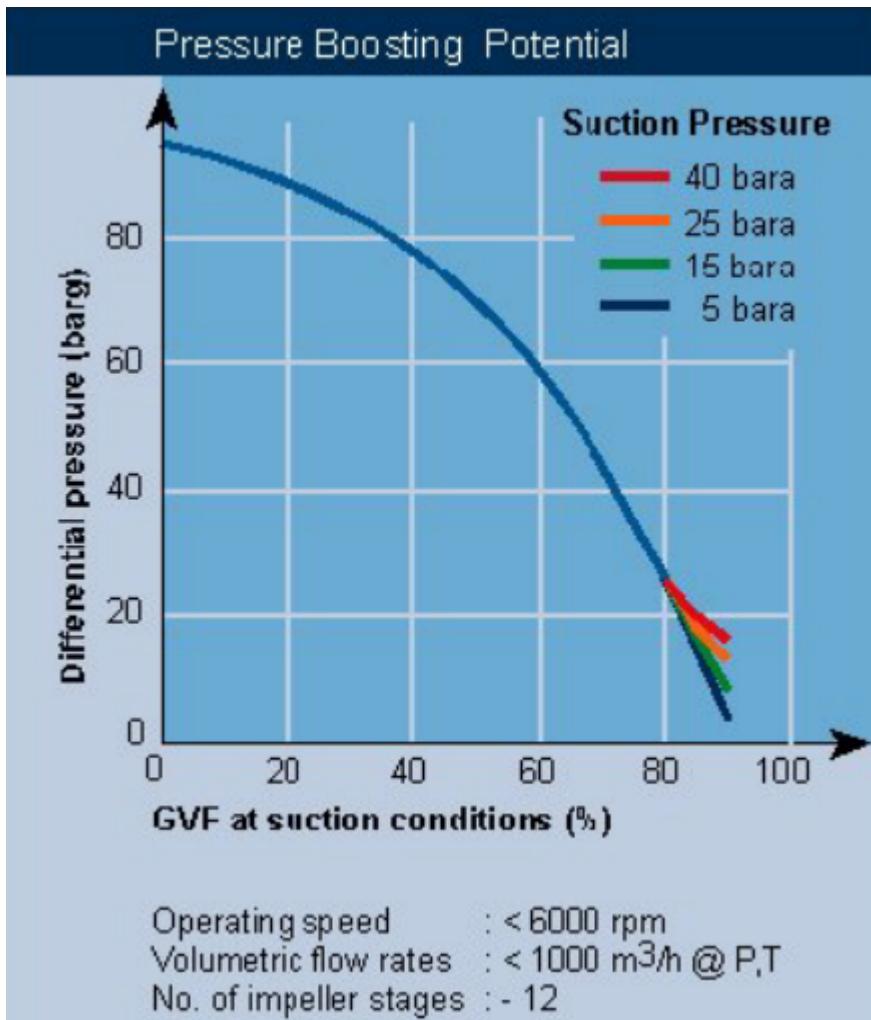
The helico-axial type multiphase booster features a number of individual booster stages, each consisting of an impeller mounted on a single rotating shaft, followed by a fixed diffuser. In essence, the impeller imparts kinetic energy to the fluid, which is converted to pressure in the diffuser. The diffuser homogenizes the fluid and redirects it to the next impeller stage. This interstage mixing prevents separation of the gas-oil mixture, enabling stable pressure-flow characteristics and increased overall efficiency. The impeller blades have a typical helical shape, and the profile of the open type impeller and diffuser blade arrangement are specifically designed to prevent the separation of the multiphase mixture inside the pump (de Marolles and de Salis, 1999).

Helico-axial pumps are able to pump large fluid volumes compared to positive displacement pumps, which is the reason they are installed in the majority of offshore and subsea applications. They can also handle limited amounts of sand but are more prone to stresses associated with slugging. They are good for handling GVF_s up to 95%.

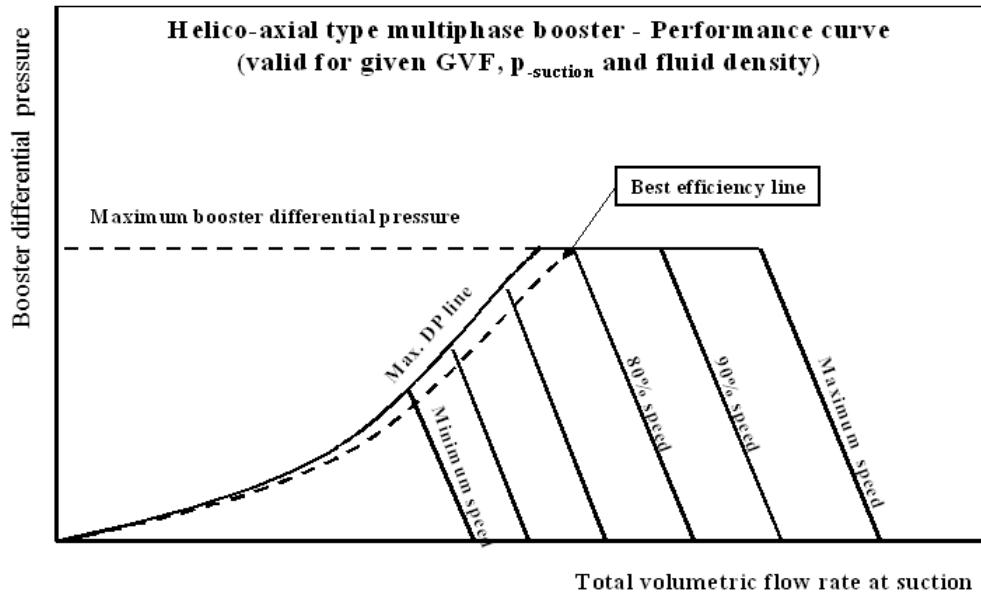
The following figure shows a vertically-configured helico-axial pump and a close-up of four individual stages.



The boosting capabilities of the helico-axial type pump are a function of GVF at suction, suction pressure, speed, number of impeller stages and impeller size.



As can be seen from the above figure, the pressure boosting capability drastically reduces with increasing GVF. Also, for lower speeds or a reduced number of stages, the pressure boosting capability will be less than the maximum shown in the figure. For a given pump with a given number of stages, speed and impeller diameter, pump performance curves can be provided as shown in the figure. These curves are valid for a given GVF at suction, suction pressure and fluid density only. New performance curves will have to be generated for conditions differing from those represented in a specific set of performance curves.

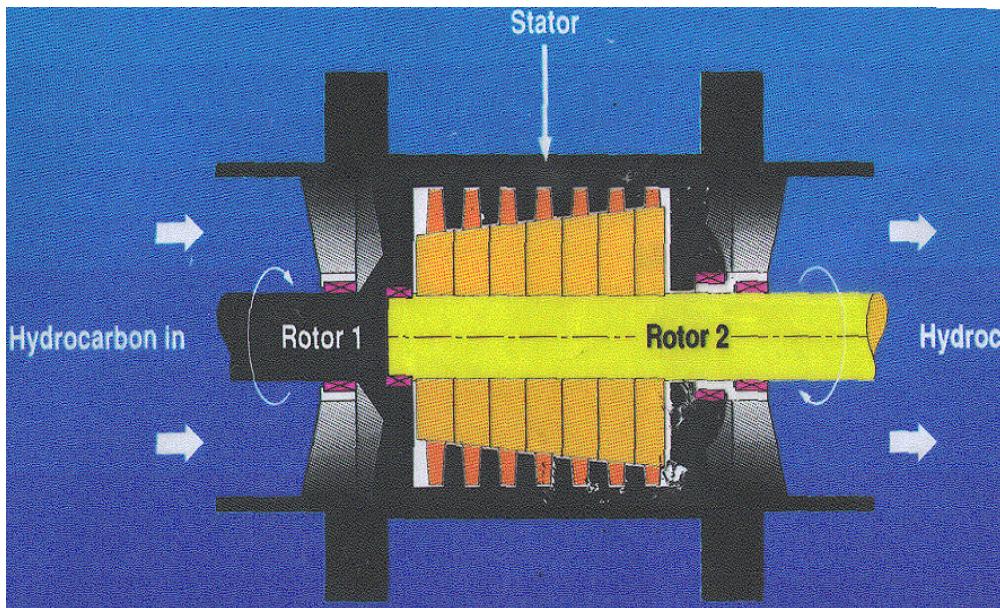


Practical operating limits of the helico-axial type multiphase booster are (Siep-RTS 1998):

- Pump differential pressure typically limited to 70 bar
- Pump flow rate (for example, total volumetric flow rate at pump suction) presently limited to approximately 1500 m³/h per pump
- Gas volume fraction at pump suction typically limited to 95% maximum
- Pump inlet pressure, 3.4 bara minimum
- Pump outlet pressure restricted by casing design pressure and seal design pressure

Contra-Rotating Axial Type

The CRA operates on the basis of axial compressor theory, but rather than having one rotor and a set of stator vanes, the CRA employs two contra-rotating rotors. The inner rotor consists of several stages mounted on the outside of an inner cylinder. The outer rotor consists of several stages on the inside of a concentric, larger diameter cylinder.



The exact mechanism underlying pressure buildup inside the CRA compressor is not yet fully understood, nor are there sufficiently mature design rules available for the scale-up of CRA performance to larger flow rates.

CRA can handle flow rates of the same order of magnitude as the helico-axial type multiphase booster, however they can achieve significantly lower differential pressures (maximum 20 bar) and efficiencies (approximately 25%) than conventional boosting systems.

Given the wider operating range and greater popularity of helico-axial multiphase boosters in the oil and gas industry, PIPESIM currently focuses its modeling capabilities in the rotodynamic category of multiphase boosters, on the helico-axial type.

Alternative Multiphase Production approach

The alternative multiphase production approach described in the figure can also be modeled in PIPESIM. This is done using the generic booster option, which splits the fluid into liquid and gas; and pumps the liquid and compresses the gas. Efficiency values for the compressor have been obtained from field data and are available in the help system.

Wet Gas Compressors

Wet Gas Compressors are a special category of multiphase booster that are used for well streams with high gas volume fractions and small amounts of liquid. Multiphase pumps operating under these conditions are termed "wet gas compressors."

These well streams are often found in marginally economic fields where optimizing production and minimizing cost are critical targets.

The same guidelines that are used to design a multiphase pump, apply for wet gas compression service. But special attention must be paid to the design of the wet gas compression service, to ensure it can handle thermal expansion, quick temperature changes, as well as high equipment temperature due to compression heat generated.

Wet gas compressors can handle GVF's greater than 98% and small volumes of low viscosity fluids. The excessive heat generated by the compression of mostly gas in the well stream often necessitates the installation of a product cooler. PIPESIM currently cannot model Wet gas compressors, but support will provided for this option in the near future.



References

Subsea Development from Pore to Process, Oilfield Review, Volume 17, Issue 1, Publication Date: 3/1/2005, Amin Amin, Mark Riding, Randy Shepler, Eric Smedstad Schlumberger and John Ratulowski Shell.

Guide to Multiphase Booster Efficiencies

Tables 1 and 2 gives guidelines on the (pump and compressor) efficiencies to enter in the generic multiphase booster module, when the generic model is needed to simulate a [Helico-Axial Booster \(p.484\)](#) or [Twin Screw Booster \(p.484\)](#).

Helico-Axial

The following table gives guidelines on the efficiencies to enter in the generic multiphase booster module to simulate a Helico-Axial multiphase booster.

FLUID GVF (%)	APPROXIMATE PUMP EFFICIENCY (%)	APPROXIMATE COMPRESSOR EFFICIENCY (%) (see 2) (p.485)
0 (see 1) (p.485)	10	10-100
10	50	20 -100
20	40	60-100
30	40	80-100
40	30-40	80-100
50	40(50) (see 3) (p.485)	40 (20) (see 3) (p.485)
60	40(50) (see 3) (p.485)	30(20) (see 3) (p.485)
70	30	60
80	30	50
90	20	70
100	10	100

Table 4.3: Helico-Axial Multiphase Booster

Twin screw

The following table gives guidelines on the efficiencies to enter in the generic multiphase booster module to simulate a Twin Screw multiphase booster.

FLUID GVF (%)	APPROXIMATE PUMP EFFICIENCY (%)	APPROXIMATE COMPRESSOR EFFICIENCY (%) (see 2) (p.485)
0	5	20 -100
10	30	20 -100
20	30	70 -100
30	30	80 -100
40	30	90
50	40(50) (see 3) (p.485)	40(20) (see 3) (p.485)
60	40	50
70	30	70

80	20	60
90	10	30
100 (see 4) (p.485)	10	100

Table 4.4: Twin Screw Multiphase Booster

See also the [Twin screw curve format \(p.110\)](#) description.

Notes:

1. Helico-Axial multiphase booster not recommended for pure liquid operations.
2. When using fluids with high liquid content the compressor efficiency has little effect as long as the compressor efficiency is within the range indicated.
3. Two sets of pump and compressor efficiencies are valid for fluids with these gas volume fractions.
4. Twin screw multiphase booster not recommended for pure gas operations

4.4 Heat Transfer Models

4.4.1 Energy Equation for Steady-State Flow

PIPESIM uses the **first law of thermodynamics** to perform a rigorous heat transfer balance on each pipe segment. The first law of thermodynamics is the mathematical formulation of the principle of conservation of energy applied to a process occurring in a closed system (a system of constant mass m). It equates the total energy change of the system to the sum of the heat added to the system and the work done by the system. For steady-state flow, it connects the change in properties between the streams flowing into and out of an arbitrary control volume (pipe segment) with the heat and work quantities across the boundaries of the control volume (pipe segment). For a multiphase fluid in steady-state flow, the energy equation is given by:

$$\Delta \left[\left(H + \frac{1}{2} v_m^2 + gz \right) dm \right] = \sum \delta Q - \sum \delta W_s \quad \text{Eq. 4.181}$$

where the specific enthalpy:

$$H = U + PV \quad \text{Eq. 4.182}$$

is a state property of the system since the internal energy U the pressure P and the volume V are state properties of the system.

It is clear from the left-hand side of equation 4.181 (p.485), that the change in total energy is the sum of the change in enthalpy energy,

$$\Delta[Hdm] = \Delta[(U + PV)dm] \quad \text{Eq. 4.183}$$

the change in gravitational potential energy:

$$\Delta E_P = \Delta[(gz)dm] \quad \text{Eq. 4.184}$$

and the change in total kinetic energy (based on the mixture velocity v_m)

$$\Delta E_K = \Delta \left[\left(\frac{1}{2} v_m^2 \right) dm \right] \approx 0 \quad \text{Eq. 4.185}$$

which is assumed to be negligible.

On the right-hand side of equation 4.181 (p.485), $\sum \delta Q$ includes all the heat transferred to the control volume (pipe segment) and δW_s represents the shaft work, that is work transmitted across the boundaries of the control volume (pipe segment) by a rotating or reciprocating shaft.

4.4.2 Overall Heat Transfer Coefficient

Steady state heat transfer between the fluid inside a pipe (flowline, riser or tubing) and its surroundings occurs due to the difference between the bulk fluid temperature T_b and the ambient temperature T_a . In the case of a flowline or riser, the ambient temperature is the temperature of the ambient fluid (air or water) moving above the mud line. In the case of a tubing, the ambient temperature is the ground temperature at a distance far from the well, given by the geothermal gradient at the tubing depth. The rate at which heat is transferred depends on various thermal resistances such as:

- Inside fluid film (which is used to model heat transfer between a moving fluid and the pipe wall)
- Wax layers on the inside of the pipe wall
- Pipe wall and surrounding layers (for example coatings, fluid-filled annuli)
- Ground and surrounding medium (air or sea)

The heat transfer Q per unit length of pipe can be expressed as:

$$Q = UA(T_b - T_a) \quad \text{Eq. 4.186}$$

where $A = \pi D_o$ is a reference area based on the pipe outside diameter and U is the overall heat transfer coefficient. The overall heat transfer coefficient can be calculated from the heat transfer coefficients for each resistance, which in turn can be found from theoretical heat transfer models. The method of calculation depends on whether the resistances are in series, parallel, or both.

Resistances in series

For resistances in series, (for example pipe coatings, see Fig 4.8 (p.487)) the temperature difference can be written as the sum of the temperature differences across each resistance:

$$T_a - T_b = \sum_i \Delta T_i \quad \text{Eq. 4.187}$$

Therefore

$$\frac{1}{U} = - \frac{A}{Q} \sum_i \Delta T_i = \sum_i \frac{1}{h_i} \quad \text{Eq. 4.188}$$

Here h_i is the heat transfer coefficient for resistance i given by:

$$\frac{1}{h_i} = - \frac{A}{Q} \Delta T_i \quad \text{Eq. 4.189}$$

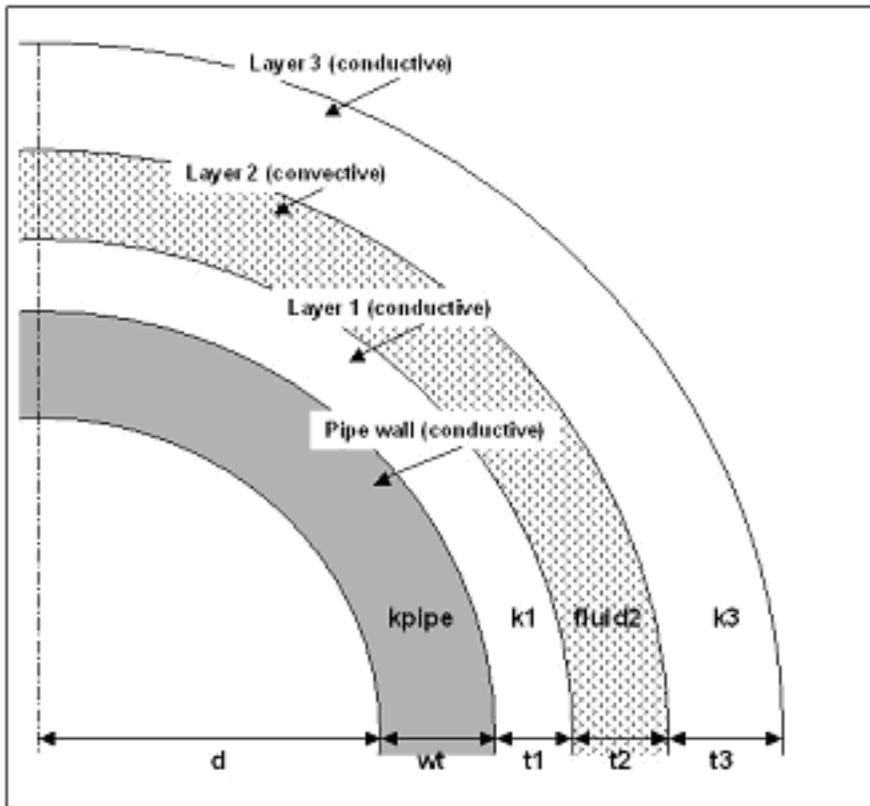


Figure 4.8. Pipeline and Layers

Resistances in parallel

For resistances in parallel, (for example partially buried pipes, see Fig 4.9 (p.488)) the overall heat transfer can be written as the sum of the heat transfer through each resistance:

$$Q = \sum_i Q_i = \sum_i U_i A (T_b - T_a) \quad \text{Eq. 4.190}$$

Therefore the overall heat transfer coefficient can be found by summing the heat transfer coefficients for each resistance in parallel:

$$U = \sum_i U_i \quad \text{Eq. 4.191}$$

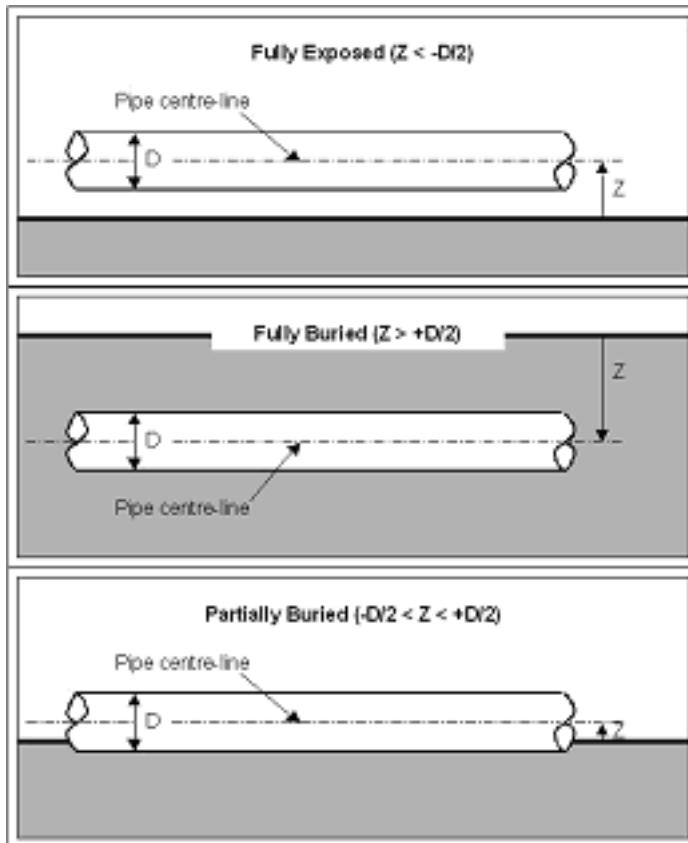


Figure 4.9. Burial configurations

Heat transfer models

Heat transfer models are required for:

- radial heat transfer between a moving fluid and the pipe wall, see [Inside film coefficient \(p.489\)](#).
- radial heat transfer through a [conductive layer \(p.495\)](#), such as internal wax layers, the pipe wall and insulation.
- radial heat transfer through a [convective layer \(p.497\)](#), such as a fluid-filled annulus.
- heat transfer through the ground
 - between the pipe and the surface for buried and partially buried [horizontal flowlines \(p.499\)](#)
 - radially, between the pipe and the far field geothermal temperature gradient for [vertical wells \(p.503\)](#)
- heat transfer through the ambient fluid
 - between the ground and the ambient fluid for buried and partially buried horizontal flowlines
 - between the pipe and the ambient fluid for partially buried and fully exposed horizontal flowlines
 - between the pipe and the ambient fluid for fully exposed vertical risers

Reference diameter for Overall Heat Transfer Coefficient

By default in PIPESIM, all Heat Transfer Coefficients (including local Inside, Outside, and Wall Coefficients, as well as the Overall Heat Transfer Coefficient) are referenced to the pipe outside diameter (and outside area), in keeping with one industry standard for heat exchanger design. However, the HEAT HCRD keyword can be used from PIPESIM's EKT to override that default basis with a different user-specified diameter (effectively imposing a different reference area for all reported Heat Transfer Coefficients). This can be useful for direct comparison with Heat Transfer Coefficients which are reported by some other flow assurance software on an inside diameter and area basis (for example, OLGA, PIPEFLO).

4.4.3 Inside Fluid Film Heat Transfer Coefficient

This inside film heat transfer coefficient accounts for resistance to heat flow between the bulk of the fluid and the inside of the pipe wall. For the most common PIPESIM cases where forced convection dominates internal heat transfer, a Nusselt number correlation is selected depending on whether that forced flow is laminar, turbulent or in the transition region.

For the less common PIPESIM case where inside natural convection may dominate internal heat transfer, a Natural Convection Nusselt number is also computed. In such cases, a check of these two competing Forced Convection and Natural Convection mechanisms is performed. PIPESIM then sets its Inside Fluid Film Heat Transfer Coefficient to the higher of these values.

Inside Forced Convection

A number of inside film coefficient (IFC) correlations were added over time to the legacy PIPESIM engine. Several of those legacy correlations are no longer commonly used for new models. However, these may still be used for compatibility with historic work, through the HEAT keyword.

The following document PIPESIM's two primary methods for computation of the Inside Forced Convective Heat Transfer Coefficient:

- [Kreith Method \(p.489\)](#) (default, with mixed multiphase properties)
- [Kaminsky Method \(p.492\)](#) (noted for special consideration of Inside Fluid Film Heat Transfer Coefficients in cases with a stratified multiphase flow regime)

Kreith

This is the default method in PIPESIM.

Kreith (averaged) mixture properties

For cases with multiphase flow, the Kreith method uses mixture properties for a pseudo single-phase, based on the local (slip-flow) liquid holdup. These properties are calculated as follows.

First, liquid and gas Reynolds numbers are calculated based on the superficial velocities v_{SL} and v_{SG} :

$$\text{Re}_{SL} = \frac{\rho_L v_{SL} D}{\mu_L} \quad \text{Eq. 4.192}$$

$$\text{Re}_{SG} = \frac{\rho_G v_{SG} D}{\mu_G} \quad \text{Eq. 4.193}$$

where ρ is the density, μ the viscosity, D the pipe diameter and the subscripts L and G refer to the liquid and gas phase properties.

A total Reynolds number is then obtained:

$$\text{Re}_{TOTAL} = \text{Re}_{SL} + \text{Re}_{SG} \quad \text{Eq. 4.194}$$

A Prandtl number is then calculated using fluid mixture properties:

$$\text{Pr}_m = \frac{\mu_m c_{p_m}}{k_m} \quad \text{Eq. 4.195}$$

where c_p is the specific heat capacity, k the thermal conductivity, and M the viscosity, and the subscript m refers to the mixture.

The mixture thermal conductivity is given by:

$$k_m = \frac{1}{\frac{H_L}{k_L} + \frac{(1 - H_L)}{k_g}} \quad \text{Eq. 4.196}$$

and the mixture heat capacity:

$$C_{p_m} = H_L C_{p_L} + (1 - H_L) C_{p_G} \quad \text{Eq. 4.197}$$

where H is the holdup.

Kreith Single-Phase Nusselt Number relations

In both single-phase and multiphase cases, PIPESIM's Kreith inside fluid film coefficient is based on the following methods for prediction of the inside film Nusselt Number.

For turbulent flow ($\text{Re}_{TOTAL} \geq 6000$):

Kreith recommends the McAdams enhancement to the respected Dittus-Boelter equation for turbulent inside Nusselt Number (Nu). McAdams fixes the Pr exponent at 0.33 - in agreement with the other respected Sieder-Tate equation for turbulent inside Nu. McAdams also applies a 'short-pipe' entrance effect (D/L) multiplier to Dittus-Boelter's turbulent inside Nu method, similar to the entrance effects found in all laminar forced-flow Nu methods.

$$Nu_{1P_{Turb}} = 0.023 \text{Re}_{TOTAL}^{0.8} \text{Pr}^{0.33} \left(1 + \left(\frac{D}{L} \right)^{0.7} \right) \quad \text{Eq. 4.198}$$

For laminar flow ($\text{Re}_{TOTAL} \leq 2000$):

$$Nu_{1P,Lam} = \begin{cases} Nu_{1P}^{SD} \equiv -0.25\text{Re}_{TOTAL} \Pr\left(\frac{D}{L}\right) \ln\left(1 - \frac{2.645}{\Pr^{0.167} \sqrt{\text{Re}_{TOTAL}} \Pr\left(\frac{D}{L}\right)}\right); & \frac{L}{D} \leq 10 \\ Nu_{1P}^{MD} \equiv -\left[1 - \frac{\left(\frac{D}{L} - 10\right)}{30 - 10}\right] Nu_{1P}^{SD} + \left[\frac{\left(\frac{D}{L} - 10\right)}{30 - 10}\right] Nu_{1P}^{LD}; & 10 < \frac{L}{D} \leq 30 \\ Nu_{1P}^{LD} \equiv 1.86 \left[\text{Re}_{TOTAL} \ Pr\left(\frac{D}{L}\right)\right]^{\frac{1}{3}}; & \frac{L}{D} > 30 \end{cases}$$

where the superscripts SD, MD and LD stand for short duct, medium duct and long duct, respectively.

For transition flow ($2000 \leq \text{Re}_{TOTAL} \leq 6000$):

$$Nu_{1P} = Nu_{1P,Lam} \left[\frac{\text{Re}_{TOTAL}}{2000} \right] \left(\frac{\ln(Nu_{1P,Turb}) - \ln(Nu_{1P,Lam})}{\ln(\text{Re}_{max}) - \ln(\text{Re}_{min})} \right) \quad \text{Eq. 4.200}$$

Note: As the Reynolds number decreases, the laminar flow Nusselt number is approaches 4. So if the Reynolds number is less than 2000, then PIPESIM limits the Reynolds number to a minimum of 4.

Reference: [Kreith \(p.589\)](#)

Kreith Multiphase Inside Fluid Film Coefficient

If the flow is multiphase then the void fraction is given by

$$\phi = \frac{Av_G}{Av_G + Av_L} \quad \text{Eq. 4.201}$$

where the cross-sectional area of the pipe:

$$A = \frac{\pi D^2}{4} \quad \text{Eq. 4.202}$$

The gas-weighted two phase fluid thermal conductivity is defined as:

$$k_{2P} = \phi k_G + (1 - \phi) k_L \quad \text{Eq. 4.203}$$

The two phase inside film coefficient for the correlations below (unless otherwise stated) is defined as:

$$h_{i_{2P}} = \frac{Nu_{2P} k_{2P}}{D} \quad \text{Eq. 4.204}$$

Note: Nu2P in this equation is computed by applying the Kreith (averaged) mixture properties to the Kreith Single-Phase Nusselt Number relations.

Kaminsky

The Kaminsky method may give enhanced prediction for cases where multiphase stratified-flow heat transfer effects will strongly affect the Overall Heat Transfer Coefficient (as one of its largest series resistances).

Kaminsky (regime-dependent) Film Phase and Dimensionless Parameters

If the flow regime is mist, single gas phase or froth then the Kaminsky method's fluid film at the inside wall is considered to be a single-phase gas and the base superficial Reynolds number is:

$$\text{Re}_S = \text{Re}_{SG} = \frac{\rho_G v_{SG} D}{\mu_G} \quad \text{Eq. 4.205}$$

and the Prandtl number is:

$$\text{Pr} = \text{Pr}_G = \frac{\mu_G c_{pG}}{k_G} \quad \text{Eq. 4.206}$$

where ρ is the density, μ the viscosity, v the velocity, c_p the specific heat capacity, k the thermal conductivity, D the pipe diameter and the subscript *SG* refers to the superficial gas phase properties.

For all other flow regimes, the Kaminsky method's fluid film at the inside wall is considered to be a single-phase liquid and the base superficial Reynolds number is:

$$\text{Re}_S = \text{Re}_{SL} = \frac{\rho_L v_{SL} D}{\mu_L} \quad \text{Eq. 4.207}$$

and the Prandtl number is:

$$\text{Pr} = \text{Pr}_L = \frac{\mu_L c_{pL}}{k_L} \quad \text{Eq. 4.208}$$

where the subscript *SL* refers to the superficial liquid phase properties.

The following minimum and maximum superficial Reynolds numbers are defined as boundaries of the laminar-turbulent transition region:

$$\text{Re}_{S_{\min}} = 2000 \quad \text{Eq. 4.209}$$

$$\text{Re}_{S_{\max}} = 6000 \quad \text{Eq. 4.210}$$

Kaminsky base Single-Phase Film Nusselt Number

The following minimum and maximum superficial Reynolds numbers are defined as boundaries of the laminar-turbulent transition region: $\text{Re}_{S_{\min}} = 2000$

$$\text{Re}_{S_{\max}} = 6000$$

The Kaminsky method bases its single-phase Nusselt Number on the Sieder and Tate equations for turbulent and laminar flows of the film phase, as follows.

For turbulent Kaminsky Film Phase ($\text{Re}_S > \text{Re}_{S_{\max}}$):

$$Nu_{1PTurb} = 0.023 \text{Re}_S^{4/5} \text{Pr}^{1/3} \left(\frac{\mu}{\mu_W} \right)^{0.14} \quad \text{Eq. 4.211}$$

For laminar Kaminsky Film Phase ($\text{Re} \leq \text{Re}_{S_{\max}}$) :

$$Nu_{1PLam} = [1.86 \text{Re}_S \text{Pr} \left(\frac{D}{L} \right)^{\frac{1}{3}}] \left(\frac{\mu}{\mu_W} \right)^{0.14} \quad \text{Eq. 4.212}$$

where L is the length of the pipe and the subscript W refers to wall properties. The viscosity μ is either the liquid or gas viscosity depending on the flow regime (as described above).

Throughout the transition region ($2000 \leq \text{Re} \leq 6000$), PIPESIM prorates the Kaminsky Nu1P by applying

$$Nu_{1PTurb} \quad \text{Eq. 4.213}$$

from Kaminsky Film Equation 1081.16 and

$$Nu_{1PLam} \quad \text{Eq. 4.214}$$

from Kaminsky Film Equation 1081.17 to the same

$$Nu_{1P} \quad \text{Eq. 4.215}$$

Interpolation Method listed above as Kreith Equation 1081.9.

Reference: [Sieder and Tate \(p.593\)](#)

Kaminsky Single-Phase Inside Fluid Film Coefficient

If the flow regime is mist, single gas phase or froth, this PIPESIM method regards the entire inside bulk fluid as a single-phase gas and the Inside Film Coefficient is:

$$h_{i_{1P}} = \frac{k_G N u_{1P}}{D} \quad \text{Eq. 4.216}$$

Similarly, for the case of a single-phase bulk liquid flow the inside film coefficient reduces to:

$$h_{i_{1P}} = \frac{k_L N u_{1P}}{D} \quad \text{Eq. 4.217}$$

Kaminsky Multiphase Inside Fluid Film Coefficient

For all other (multiphase) flow regimes, with a turbulent liquid film ($\text{Re} \geq 2300$):

$$h_{i_{2P}} = h_{i_{1P,SL}} \sqrt{S \frac{\Delta P_{2Pf}}{\Delta P_{1Pf}}} \quad \text{Eq. 4.218}$$

Its required input of the single-phase turbulent heat transfer film coefficient h_1 , must first be computed by the Sieder-Tate correlation. This Multiphase Pressure-Drop/Heat-Transfer Analogy Method also implicitly requires that both the Superficial Liquid Pressure Drop and the Multiphase Pressure Drop be pre-calculated as additional inputs, before its h_1 can be computed.

For horizontal (for example, if the pipe angle $|\beta| < \beta_{swap}$) stratified flow, the wetting of the pipe wall is calculated from

$$S = \pi D \theta \quad \text{Eq. 4.219}$$

where θ is the wetted wall fraction given by [Grolman's correlation \(p.587\)](#):

$$\theta = \theta_0 \left(\frac{\sigma_w}{\sigma} \right)^{0.15} + \frac{\rho_g}{\rho_l + \rho_g} \frac{1}{\cos(\beta)} \left\{ \frac{\rho_l u_{ls} D}{\sigma} \right\}^{0.25} \left\{ \frac{u_{gs}^2}{(1 - H_l)^2 g D} \right\}^{0.8} \quad \text{Eq. 4.220}$$

in which the minimum wetted wall fraction θ_0 is approximated by:

$$\theta_0 \approx 0.624 H_l^{0.374} \quad \text{Eq. 4.221}$$

For all other types of flow, heat transfer it is reasonable to assume that heat transfer is circumferentially uniform (i.e. $S = 1$).

For laminar flow ($\text{Re} < 2300$):

$$h_{i_{2P}} = \frac{(2 - H_l) h_{i_{1PSL}}}{H_l^{\frac{2}{3}}} \quad \text{Eq. 4.222}$$

The single-phase laminar heat transfer is estimated by the Sieder-Tate correlation.

Reference: [Kaminsky \(p.588\)](#)

Inside Natural Convection

As described for Annulus and Outside Convective Heat Transfer Coefficients, Natural Convection Heat Transfer inside a pipe is also a function of the Grashof Number (Eq. 1081.3).

Whenever fluid temperature may be significantly influenced by Natural Convection heat transfer, PIPESIM checks for the possible influence of Natural Convection inside the pipe.

Inside Natural Convection Nusselt Number

PIPESIM computes an Inside Natural Convection Nusselt Number as follows:

$$Nu_{NC} = \max[3.66, 0.184(\text{GrPr})^{\frac{1}{3}}]$$

This equation captures the Natural Convective Effect whenever significant, in a simple, compute-inexpensive, and numerically stable way. Its 'floor' is set to Nu = 3.66 - the Conduction-Only Limiting Minimum Nu for a Circular Pipe.

PIPESIM computes this NuNC for every Laminar case, to check if this Natural Convection Nu would dominate.

Further, PIPESIM also computes this NuNC for Transition and Turbulent (Forced Convection) cases if: $Re < 10000$ (transition Re)

Maximum Nu Method for Competing Natural and Forced Convection

Whenever it is computed (for Laminar cases, or for Transition and Turbulent cases within the Gr/Re or Re range described above), this Inside Natural Convection Nu is compared with PIPESIM's Inside Forced Convection Nu, and the larger Nu value is used to compute PIPESIM's Inside Fluid Film Coefficient, as: $Nu = \max(Nu_{NC}, Nu_{FC})$

where, NC denotes this Natural Convection Nu, and FC denotes the Forced Convection Nu.

This numerical method approximates a physical reality. One or the other of Natural or Forced Convection will suppress the other as buoyant density differences, flowrates, and turbulence are increased or decreased in different cases. This method models that competition, while maintaining a continuous trend of Inside Fluid Film Nusselt Number for parametric studies with differing flowrates, insulation levels, ambient temperatures, etc.

Reference: [VDI, Incropera, Scandpower \(p.588\)](#)

4.4.4 Conductive Heat Transfer Coefficients

[Brill and Mukherjee \(p.584\)](#) give a formula for radial heat transfer Q per unit length of pipe through a conductive layer:

$$\Delta T = T_o - T_i = -\frac{Q \ln(r_o/r_i)}{2\pi k} \quad \text{Eq. 4.223}$$

where

k is the conductivity of the layer.

r_i is the inner radius of the layer

r_o is the outer radius of the layer

T_i is the temperatures at the inside edge of the layer

T_o is the temperatures at outside edge of the layer

This equation can be used to calculate the heat transfer coefficient for a conductive layer:

$$\frac{1}{h} = -\frac{A}{Q}\Delta T = \frac{A}{2\pi} \cdot \frac{\ln(r_o/r_i)}{k} \quad \text{Eq. 4.224}$$

where

$\frac{A}{2\pi} = \frac{D_o}{2}$ is the radius of the reference area (normally the pipe outside radius)

Wax Heat Transfer Coefficient

[4.224 \(p.496\)](#) can be used for heat transfer through a wax layer on the inside wall of the pipe, where

$k = k_{wax}$ is the conductivity of the wax layer.

$r_i = \frac{D_i}{2} - r_{wax}$ is the inner radius of the wax layer (equal to the pipe inner radius minus the wax thickness)

$r_o = \frac{D_i}{2}$ is the outer radius of the wax layer (equal to the pipe inner radius)

Pipe wall heat transfer coefficient

[4.224 \(p.496\)](#) can be used for heat transfer through the pipe wall, where

$k = k_{pipe}$ is the conductivity of the pipe wall.

$r_i = \frac{D_i}{2}$ is the inner radius of the pipe

$r_o = \frac{D_o}{2}$ is the outer radius of the pipe

Conductive layer heat transfer coefficient

[4.224 \(p.496\)](#) can be used for heat transfer through conductive layers, such as foam insulation or cement, where

$k = k_n$ is the conductivity of the n th layer

$r_i = \frac{D_{ni}}{2}$ is the inner radius of the n th layer

$r_o = \frac{D_{no}}{2}$ is the outer radius of the n th layer

4.4.5 Annulus and Outside Convective Heat Transfer Coefficients

Convective heat transfer can occur in a number of places in a well and surface network, across a fluid filled annulus; between a pipe or surface and the air or sea. Free (or natural) convection occurs when the bulk fluid is at rest and convection is driven by buoyancy effects alone. Forced convection occurs when the fluid is moving, which will increase the rate at which heat is transferred.

The heat transfer coefficient for free convection at a wall is given in terms of the Nusselt number (Nu), the fluid conductivity (k) and a length scale (L):

$$h = \frac{k \cdot Nu}{L} \quad \text{Eq. 4.225}$$

The Nusselt number can be found experimentally, depending on the geometry of the convective surfaces. It also depends on the fluid properties, which are encapsulated in two dimensionless numbers, the Prandtl number (Pr) representing the ratio of velocity and temperature gradients:

$$Pr = \frac{c_p M}{k} \quad \text{Eq. 4.226}$$

and the Grashof number representing the ratio of buoyancy to viscous forces:

$$Gr = \frac{L^3 P^2 B g \Delta T}{\mu^2} \quad \text{Eq. 4.227}$$

β	Fluid thermal expansion coefficient	K^{-1}
μ	Fluid dynamic viscosity	$kg \cdot m^{-1} \cdot s^{-1}$
ρ	Fluid density	$kg \cdot m^{-3}$
c_p	Fluid heat capacity	$W \cdot kg^{-1}$
k	Fluid thermal conductivity	$W \cdot m^{-1} \cdot s^{-1}$

Fluid properties are calculated at a film temperature T_{film} half way between the wall temperature and the bulk fluid temperature:

$$T_{f\bar{f}lm} = (T_{wall} + T_f)/2 \quad \text{Eq. 4.228}$$

The wall temperature and bulk fluid temperature are used to calculate the temperature difference in the formula for the Grashof number:

$$\Delta T = |T_f - T_{wall}| \quad \text{Eq. 4.229}$$

Because the fluid properties and Grashof numbers are functions of the wall temperatures, the heat transfer coefficient is also a function of the wall temperatures. The heat loss calculation therefore needs to be solved iteratively.

Convection in a fluid filled vertical annulus

PIPESIM can model the heat transfer in a fluid filled annulus by free convection. The heat transfer coefficient can be determined from the heat transfer coefficients at the inner and outer walls:

$$\frac{1}{h_{annulus}} = \frac{1}{h_{inner}} + \frac{1}{h_{outer}} \quad \text{Eq. 4.230}$$

For vertical pipes ($\text{angle} \geq 45^\circ$), the Nusselt number is given by [Eckert and Jackson \(1950\) \(p.586\)](#) (quoted in [Kreith and Bohn\(1997\) \(p.589\)](#)) in terms of the Rayleigh number (Ra):

$$Nu = 0.555 Ra^{0.25} \quad \text{for } Ra \leq 10^9 \quad \text{Eq. 4.231}$$

$$Nu = 0.021 Ra^{0.4} \quad \text{for } Ra > 10^9 \quad \text{Eq. 4.232}$$

where

$$Ra = \Pr \cdot Gr \quad \text{Eq. 4.233}$$

The bulk fluid temperature is assumed to be the average of the annulus wall temperatures:

$$T_f = (T_{inner} + T_{outer})/2 \quad \text{Eq. 4.234}$$

Convective Heat transfer through fluid-filled annuli can be modeled by the use of the [EKT \(p.94\)](#). Refer to the Expert Mode Keyword Reference section on [fluid coats \(p.712\)](#).

Convection in a fluid-filled horizontal annulus

PIPESIM uses the same equations to calculate the heat transfer for a horizontal fluid annulus as for a vertical annulus. except that the Nusslet number is given by:

$$Nu = 0.53 Ra^{0.25} \quad \text{Eq. 4.235}$$

Fully exposed pipe

For a flowline or riser exposed to the sea or the air, the “ambient” heat transfer coefficient can be calculated by summing the free and forced convection heat transfer coefficients:

$$h_a = h_{forced} + h_{free} \quad \text{Eq. 4.236}$$

For forced convection, the heat transfer coefficient depends on the Reynolds number of the flow

$$Nu_{forced} = (0.4Re^{0.5} + 0.06Re^{0.67})\Pr^{0.4} \quad \text{Eq. 4.237}$$

4.4.6 Heat Transfer Between a Horizontal Flowline and the Ground Surface

Fully Buried Ground Heat Transfer Coefficient

The fully buried heat transfer coefficient for a flowline is evaluated by determining a conduction shape factor to account for the geometrical and thermal effects of the burial configuration. Once the shape factor is known, the ground heat transfer coefficient is calculated from:

$$h_g = \frac{k_g S}{R} \quad \text{Eq. 4.238}$$

where R is a chosen reference length. By default, in PIPESIM, this is the outer radius of the pipe. The shape factor used differs depending on the [partial burial option \(p.171\)](#) that is selected.

A pseudo film coefficient is then added in series in order to model the ambient fluid moving above ground level:

$$\frac{1}{h_{ext}} = \frac{1}{h_g} + \frac{1}{h_a} \quad \text{Eq. 4.239}$$

2009 Method

The conduction shape factor is obtained from a solution to the steady-state heat conduction equation (the Laplace equation) with convective boundary conditions on the pipe inside wall and ground surfaces:

$$S = \frac{B_p a_{bur}}{\left[\left(\cosh \alpha_0 - B_p a_{bur} \alpha_0 + \frac{B_p}{B_g} \right)^2 - \left(1 + \frac{B_p}{B_g} \right)^2 \right]^{\frac{1}{2}}} \quad \text{Eq. 4.240}$$

where

$$\alpha_0 = -\cosh^{-1} \left(-\frac{Z}{R} \right) \quad \text{Eq. 4.241}$$

is a auxiliary geometrical quantity and

$$a_{bur} = -\sinh \alpha_0 = \sqrt{\left(\frac{Z}{R}\right)^2 - 1} \quad \text{Eq. 4.242}$$

is a scale factor for bicylindrical coordinates and

$$B_p = \frac{U_{ipc} R}{k_g} \quad \text{Eq. 4.243}$$

is the Biot number of the pipe and

$$B_g = \frac{h_a R}{k_g} \quad \text{Eq. 4.244}$$

is the Biot number of the ground. U_{ipc} is the combined heat transfer coefficient of the inside film, pipe, coatings (and wax)

$$\frac{1}{U_{ipc}} = \frac{1}{h_i} + \frac{1}{h_{wax}} + \frac{1}{h_{pipe\&layers}} \quad \text{Eq. 4.245}$$

Equation 4.240 (p.499) is not valid when the pipe&layers surface is just touching the ground surface ($Z/R=1$). In such a case, the shape factor is calculated from the following asymptotic expression

$$S \sim \frac{B_p}{\left[\left(1 + \frac{B_p}{B_g} \right) \left(1 + 2B_p \right) \right]^{\frac{1}{2}}} \quad \text{Eq. 4.246}$$

We obtain the ground heat transfer coefficient from:

$$h_g = \frac{k_g S}{R} \quad \text{Eq. 4.247}$$

Note: This is the default method in PIPESIM. The shape factor above is accurate to within 2.5% of the numerical simulation studies given by Schneider (p.592). For information about how the results of the 2009 method compare to Schneider's, see Ouworrie (p.591).

1983 & 2000 Methods

The conduction shape factor is obtained from a solution to the steady-state heat conduction equation (the Laplace equation) with isothermal boundary conditions on the pipe inside wall and ground surfaces:

$$S = \frac{2\pi}{\cosh^{-1}\left(\frac{Z}{R_{pipe\&layers}}\right)} \quad \text{Eq. 4.248}$$

Reference: Kreith (p.589)

Partially Buried Ground Heat Transfer Coefficient

To calculate the overall heat transfer coefficient for a partially buried pipeline, buried and exposed heat transfer coefficients must be calculated and combined in parallel. The method of combination and the ground conduction shape factors used differ depending on the partial burial option (p.100) that is selected.

2009 Method

1. A fully exposed pseudo pipe of the same diameter is created and an overall heat transfer coefficient (U_{exp}) is calculated using the methods described in the sections above.

2. A partially buried conduction shape factor is calculated using the methods described in the sections above. The shape factor is computed from

$$S = \begin{cases} \frac{2B_p a_{part} \tan^{-1} \left(\sqrt{\frac{1 - A_{part}}{1 + A_{part}}} \right)}{\pi \left(1 + \frac{B_p}{B_g} \right) \sqrt{1 - A_{part}^2}}; & -1 < A_{part} < 1 \\ \frac{B_p a_{part}}{\pi \left(1 + \frac{B_p}{B_g} \right)}; & A_{part} = 1 \\ \frac{2B_p a_{part} \tanh^{-1} \left(\sqrt{\frac{A_{part} - 1}{A_{part} + 1}} \right)}{\pi \left(1 + \frac{B_p}{B_g} \right) \sqrt{A_{part}^2 - 1}}; & A_{part} > 1 \end{cases} \quad \text{Eq. 4.249}$$

where

$$A_{part} = \left(1 + \frac{B_p}{B_g} \right)^{-1} \left(\cos \beta_0 + B_p a_{part} \left(\pi + \beta_0 \right) - \frac{B_p}{B_g} \right) \quad \text{Eq. 4.250}$$

is an auxiliary geometrical quantity and

$$a_{part} = -\sin \beta_0 = \sqrt{1 - \left(\frac{Z}{R} \right)^2} \quad \text{Eq. 4.251}$$

is a scale factor for bicylindrical coordinates.

3. The fully buried and fully exposed heat transfer coefficients are then combined in parallel (according to the fraction of pipe exposed and the fraction of pipe buried) using equation 4.252 (p.502) to give the overall heat transfer coefficient:

Note: This is the default method in PIPESIM. For more information, see [Ovuworie \(p.591\)](#).

2000 Method

1. A fully exposed pseudo pipe of the same diameter is created and an overall heat transfer coefficient (U_{bur}) is calculated using the methods described in the sections above.
2. A fully buried pseudo pipe ($Z=+R$) of the same diameter is created and an overall heat transfer coefficient (U_{exp}) is calculated using the methods described in the sections above.

3. The fully buried and fully exposed heat transfer coefficients are then combined in parallel (according to the fraction of pipe exposed and buried) to give the overall heat transfer coefficient:

$$U = \left(1 + \frac{\beta_0}{\pi}\right) U_{\text{exp}} - \frac{\beta_0}{\pi} U_{\text{bur}} \quad \text{Eq. 4.252}$$

where the negative of half of the angle of the exposed arc:

$$\beta_0 = -\cos^{-1}\left(-\frac{Z}{R}\right) \quad \text{Eq. 4.253}$$

1983 Method

1. A fully exposed pseudo pipe with diameter corresponding to the exposed surface area is created and an overall heat transfer coefficient (U_{exp}) is calculated using the methods described in the sections above.
2. A fully buried pseudo pipe with diameter corresponding to the buried surface area is created and an overall heat transfer coefficient (U_{bur}) is calculated using the methods described in the sections above.
3. The fully buried and fully exposed heat transfer coefficients are then combined in parallel (according to the surface areas of pipe exposed and buried) to give the overall heat transfer coefficient:

$$U = \frac{A_{\text{exp}}}{A} U_{\text{exp}} + \frac{A_{\text{bur}}}{A} U_{\text{bur}} \quad \text{Eq. 4.254}$$

where the total surface area of the buried pipe:

$$A = 2\pi R \quad \text{Eq. 4.255}$$

The surface area of the exposed portion of the pipe is:

$$A_{\text{exp}} = \pi R \left(1 - \frac{\theta_{\text{bur}}}{2\pi}\right) \quad \text{Eq. 4.256}$$

where the angle of the buried arc:

$$\theta_{\text{bur}} = \sin^{-1}\left(\frac{Z}{R}\right) \quad \text{Eq. 4.257}$$

The surface area of the buried portion of the pipe is:

$$A_{\text{bur}} = A - A_{\text{exp}} \quad \text{Eq. 4.258}$$

4.4.7 Heat Transfer Between a Vertical Well and the Surrounding Rock

Ramey Model

Heat transfer between a well and its surroundings varies with time: the well exchanges energy with the formation, heating it up (or cooling it down), until the formation is at the same temperature as the well.

The [Ramey \(1962\) \(p.592\)](#) model is an analytical method for determining the ground heat transfer coefficient, h_g , given the length of time t a well has been operating. The model assumes that heat transfer in the wellbore is steady-state, whilst heat transfer to the formation is by transient radial conduction. In his paper, Ramey quotes various solutions for different boundary conditions. He observed that the solutions eventually converge after about a week. He concluded that a line source with constant heat flux gives a good asymptotic solution for long times (times greater than one week).

The wellbore (ground) heat transfer coefficient is given by:

$$h_g = \frac{2k_g}{Df(t)} \quad \text{Eq. 4.259}$$

where the time function:

$$f(t) = \frac{1}{2} E_1\left(\frac{D^2}{4\alpha T}\right) \exp\left(\frac{D^2}{4\alpha T}\right) \quad \text{Eq. 4.260}$$

The exponential integral is given by:

$$E_1\left(\frac{D^2}{4\alpha t}\right) = \int_0^{\frac{D^2}{4\alpha t}} \frac{1 - \exp(-r)}{r} dr - \ln\left(\frac{D^2}{4\alpha t}\right) - \gamma \quad \text{Eq. 4.261}$$

For large values of time t , Ramey uses a series expansion for the exponential integral, which to leading order gives:

$$f(t) \approx -\ln\left(\frac{D_{co}}{4\sqrt{\alpha t}}\right) - \frac{\gamma}{2} \quad \text{Eq. 4.262}$$

k_g	ground thermal resistance	$\text{W m}^{-1} \text{K}^{-1}$
D	outside diameter of pipe	m
D_{co}	outside diameter of pipe and thermal coatings	m
$\alpha = \frac{k_g}{\rho_g c_g}$	ground thermal diffusivity	$\text{m}^2 \text{s}^{-1}$
c_g	ground specific heat capacity	$\text{J kg}^{-1} \text{K}^{-1}$
ρ_g	ground density	kg m^{-3}

r	radial distance from the centre of the well	m
$\gamma \approx 0.577$	Euler-Mascheroni gamma constant	dimensionless

In the case of a tubing we see that:

$$\frac{1}{h_{ext}} = \frac{1}{h_g} \quad \text{Eq. 4.263}$$

and the ambient temperature used in equation 4.259 (p.503) is given by the geothermal temperature at some radial distance far from the centre of the well.

Note: To compute a geothermal gradient and hence a geothermal temperature at a particular well depth,, PIPESIM requires knowledge of at least two ambient temperatures at two corresponding measured depths (MD) or true vertical depth (TVD) — usually these are the ambient temperatures at top and bottom of the tubing.

Reference: "[Wellbore Heat Transmission](#)", H.J. Ramey (p.592)

4.5 Fluid Models

A number of fluid and solid phases may be present in oil and gas pipeline. These include:

- **Fluids**
 - Vapour hydrocarbon and water (gas)
 - Liquid hydrocarbon (oil)
 - Liquid water
 - Other liquids (e.g. liquid CO₂)
- **Solids**
 - Hydrate I
 - Hydrate II
 - Wax
 - Asphaltene
 - Ice
 - Scale

PIPESIM simulates flow of only three fluid phases, oil, gas and water. In fact some flow models only consider two phases, liquid and gas. Liquid properties are determined by combining the oil and water properties.

PIPESIM can be used to model wax precipitation and deposition. Other solid phases cannot be modelled, although the appearance of hydrates, asphaltene and ice can be predicted. PIPESIM can model scale prediction.

Fluid models are used to determine the phase state (e.g. single phase oil, single phase gas, two phase oil and gas etc) and the phase thermodynamic and transport properties needed for

simulation (e.g. density, enthalpy and viscosity). PIPESIM allows three different types of fluid description:

- **Black oil (p.505)** Three phases are allowed, oil, gas and water. The hydrocarbon fluid is made up of oil and gas. Simple correlations are used to determine how much gas can dissolve in oil and the phase properties.
- **Compositional (p.141)** The number of phases allowed depends on the flash package. Fluid is made up of components, such as methane, ethane, water etc. Phase state is determined by minimizing Gibbs energy of the system (the flash). This can be a complicated calculation and is therefore significantly slower than black oil. PIPESIM can use a number of different flash packages.
- **Fluid Property Table Files (p.554)** Two phase (liquid and gas) properties can be output from compositional packages in a tabular form that PIPESIM can read.

4.5.1 Black Oil Fluid Modeling

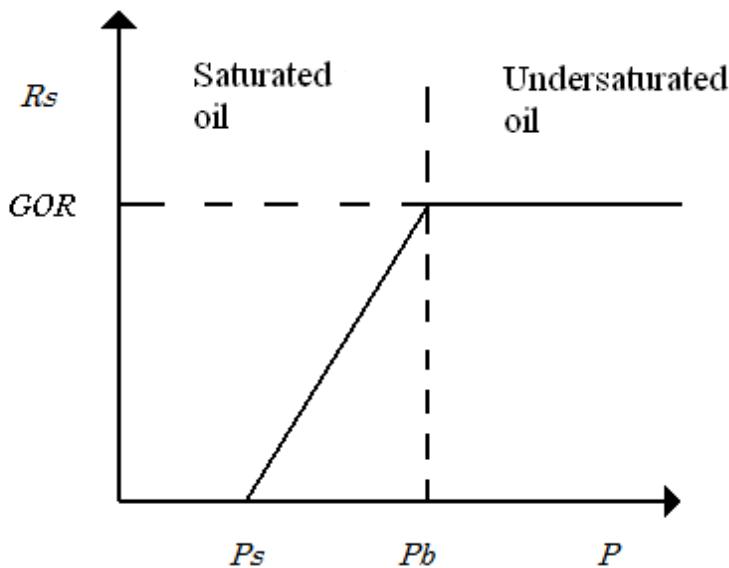
Black oil fluids are modelled as three phases, oil, gas and water. The amount of each phase is defined at stock tank conditions, by specifying two ratios, typically the gas oil ratio (GOR) and the water cut (WCUT). Properties at pressures and temperatures other than stock tank are determined by [correlations \(p.506\)](#). Water is assumed to remain in the water phase. The key property for determining the phase behaviour of the hydrocarbons is the [solution gas—oil ratio \(p.508\)](#),

$R_s(P, T)$, which is used to calculate the amount of the gas dissolved in the oil at a given pressure and temperature:

Stock tank volume of gas dissolved in oil: $R_s \cdot V_O$

Stock tank volume of free gas: $V_G = (GOR - R_s) \cdot V_O$

At stock tank conditions $R_s = 0$. The [bubble point pressure \(p.512\)](#) $P_b(T)$ can be found by calculating the pressure at which all the gas is dissolved in the oil $R_s(P_b, T) = GOR$



For pressures below the bubble point the oil is **saturated** (no more gas can dissolve in it at that pressure and temperature). For pressures above the bubble point, there is no vapour phase and the oil is **undersaturated**, since more gas could be dissolved in it if it were available. Above stock tank pressure $P > P_s$ the oil contains dissolved gas, and is known as **live oil**. Oil at stock tank pressure (or oil with $GOR=0$) is known as **dead oil**. Different correlations apply for dead oil, saturated live oil, and unsaturated live oil properties.

[Correlations \(p.506\)](#) are needed for the fluid properties needed for simulation:

- the [oil formation volume factor \(p.512\)](#) (which is used to determine oil density),
- the [gas compressibility \(p.522\)](#) (to determine the gas density)
- the water density
- the [oil viscosity \(p.515\)](#)
- the [gas viscosity \(p.525\)](#)
- the water viscosity
- the [fluid enthalpy \(p.526\)](#)
- the [oil-gas surface tension \(p.525\)](#)
- the [water-gas surface tension \(p.526\)](#)

[Liquid properties \(p.555\)](#) are calculated by combining the oil and water properties.

Black oil Correlations

The following black oil correlations are available:

Solution gas (p.508) and bubble point pressure (p.512)	Lasater (p.511) , Standing (p.511) , Vasquez and Beggs (p.511) , Kartoatmodjo and Schmidt (p.510) ,
--	---

	Glasø (p.510), De Ghetto et al (p.508) or Petrosky and Farshad (p.511).
Oil formation volume factor of saturated systems	Standing (p.513), Vasquez and Beggs (p.513), Kartoatmodjo and Schmidt (p.514)
Oil formation volume factor of undersaturated systems	Vasquez and Beggs (p.514)
Dead oil viscosity (p.516)	Beggs and Robinson, Glasø, Kartoatmodjo, De Ghetto, Hossain, Petrosky, Elsharkawy or Users data.
Live oil viscosity of saturated systems (p.518)	Chew and Connally, Kartoatmodjo, Khan, De Ghetto, Hossain, Petrosky, Elsharkawy, or Beggs and Robinson.
Live oil viscosity of undersaturated systems (p.520)	Vasquez and Beggs, Kouzel, Kartoatmodjo, Khan, De Ghetto, Hossain, Petrosky, Elsharkawy, Bergman or None.
Viscosity of oil/water mixtures (p.555)	Inversion, Volume Ratio, or Woelflin.
Gas viscosity (p.525)	Lee et al.
Gas compressibility (p.522)	Standing, Hall and Yarborough, or Robinson et al.
Oil-gas surface tension (p.525)	
Water-gas surface tension (p.526)	

Correlation data

The data points spanned the following ranges :

		Lasater (p.589)	Standing (p.593)	Vasquez and Beggs (p.514)
Data		Correlation was developed in 1958 from 158 experimental data points	Correlation was based on 105 experimentally determined bubble point pressure of California oil systems.	Correlations use data from more than 600 oil systems. Approximately 6,000 measured data points were collected.
P_b	bubble point pressure (psia)	48 to 5,780	130 to 7,000	50 to 5,250
T	temperature (°F)	82 to 272	100 to 258	70 to 295
API	API gravity (°API)	17.9 to 51.1	16.5 to 63.8	16 to 58
γ_G	gas specific gravity	0.574 to 1.223	0.59 to 0.95	0.56 to 1.18
R_{sb}	solution gas at bubble point pressure (scf/ STB)	3 to 2,905	20 to 1,425	20 to 2,070

		Beggs and Robinson	Chew and Connally
Data		Data from 600 oil systems were used to develop correlations for dead and live oil viscosity. 460 dead oil observations and 2,073 live oil observations were used.	Data from 457 oil systems was used to develop correlation for live oil viscosity
P_b	bubble point pressure (psia)	50 to 5,250	132 to 5,645
T	temperature (°F)	70 to 295	72 to 292
API	API gravity (°API)	16 to 58	
γ_G	gas specific gravity		
R_{sb}	solution gas at bubble point pressure (scf/ STB)	20 to 2,070	51 to 3,544

Glasø ([p.587](#)) developed PVT correlations from analysis of crude oil from the following North Sea Fields:- Ekofisk Stratfjord Forties Valhall COD 30/7-2A.

Solution Gas-oil Ratio

The solution gas-oil ratio, R_s (scf/STB), can be determined using one of a number of correlations:

- [De Ghetto et al \(p.508\)](#)
- [Glasø \(p.510\)](#)
- [Kartoatmodjo and Schmidt \(p.510\)](#)
- [Lasater \(p.511\)](#)
- [Petrosky and Farshad \(p.511\)](#)
- [Standing \(p.511\)](#)
- [Vasquez and Beggs \(p.511\)](#)

The correlations depend on:

P	pressure (psia)
T	temperature (°F)
API	API gravity (°API)
γ_G	gas specific gravity

De Ghetto et al.

De Ghetto et al. give different correlations for the solution gas-oil ratio and the bubble point pressure. In PIPESIM it is important to use related formula for these two properties to ensure

consistency. The PIPESIM implementation of the solution gas-oil ratio is therefore derived from the De Ghetto *et al* equations for the bubble point pressure.

Extra heavy oil, API < 10

For extra heavy oil the De Ghetto formula is a modified version of the [Standing \(p.511\)](#) formula:

$$R_s(P, T) = C \cdot \gamma_G \cdot \left[\frac{P}{10.7025 \cdot A(T)} \right]^{1.1128} \quad \text{Eq. 4.264}$$

Here A is a function of the fluid temperature and the oil API density:

$$\log_{10} A = 0.002 \cdot T - 0.0142 \cdot API \quad \text{Eq. 4.265}$$

C is a [calibration \(p.512\)](#) constant.

Heavy oil, 10 < API < 22.3

For heavy oil the De Ghetto formula is a modified version of the [Standing \(p.511\)](#) formula:

$$R_s(P, T) = C \cdot \gamma_G \cdot \left[\frac{P}{15.7286 \cdot A(T)} \right]^{1/0.7885} \quad \text{Eq. 4.266}$$

Here A is a function of the fluid temperature and the oil API density:

$$\log_{10} A = 0.002 \cdot T - 0.0142 \cdot API \quad \text{Eq. 4.267}$$

C is a [calibration \(p.512\)](#) constant.

Medium oil, 22.3 < API < 31.1

For medium oil the De Ghetto formula is a modified version of the [Kartoatmojdo and Schmidt \(p.510\)](#) formula:

$$R_s = C \cdot C_1 \cdot \left[\gamma_G \cdot (1 + g_{corr}) \right]^{C_2} \cdot A(T) \cdot P^{C_4} \quad \text{Eq. 4.268}$$

Here A is a function of the fluid temperature and the oil API density:

$$\log_{10} A = C_3 \cdot \frac{API}{T + 460} \quad \text{Eq. 4.269}$$

If the separator pressure and temperatures are known then a non-zero gas specific gravity correction factor is used:

$$g_{corr} = 0.1595 \cdot API^{0.4078} \cdot T_{sep}^{-0.2466} \cdot \log_{10} \left(\frac{P_{sep}}{114.7} \right) \quad \text{Eq. 4.270}$$

C is a [calibration \(p.512\)](#) constant.

The constants C_1 , C_2 , C_3 and C_4 :

C_1	C_2	C_3	C_4
-------	-------	-------	-------

0.10084	0.2556	7.4576	0.9868
---------	--------	--------	--------

Light oil, $31.1 < API$

For light oil the De Ghetto formula is a modified version of the Standing (p.511) formula:

$$R_s(P, T) = C \cdot \gamma_G \cdot \left[\frac{P}{31.7648 \cdot A(T)} \right]^{1/0.7885} \quad \text{Eq. 4.271}$$

Here A is a function of the fluid temperature and the oil API density:

$$\log_{10} A = 0.0009 \cdot T - 0.0148 \cdot API \quad \text{Eq. 4.272}$$

C is a [calibration \(p.512\)](#) constant.

Glasø

The Glasø formula for the solution gas-oil ratio is:

$$R_s = \gamma_G \cdot [f(P)]^{1.22549} \cdot API^{1.212009} \cdot T^{-0.210784} \quad \text{Eq. 4.273}$$

Here:

$$\log_{10} f(P) = 2.887 \cdot \left[1 - \sqrt{1 - 0.397 \cdot (\log_{10} P - C)} \right] \quad \text{Eq. 4.274}$$

C is a [calibration \(p.512\)](#) constant.

Kartoatmodjo and Schmidt

The Kartootmodjo and Schmidt formula for the solution gas-oil ratio:

$$R_s = C \cdot C_1 \cdot [\gamma_G \cdot (1 + g_{corr})]^{C_2} \cdot A(T) \cdot P^{C_4} \quad \text{Eq. 4.275}$$

Here A is a function of the fluid temperature and the oil API density:

$$\log_{10} A = C_3 \cdot \frac{API}{T + 460} \quad \text{Eq. 4.276}$$

If the separator pressure and temperatures are known then a non-zero gas specific gravity correction factor is used:

$$g_{corr} = 0.1595 \cdot API^{0.4078} \cdot T_{sep}^{-0.2466} \cdot \log_{10} \left(\frac{P_{sep}}{114.7} \right) \quad \text{Eq. 4.277}$$

C is a [calibration \(p.512\)](#) constant.

The constants C_1 , C_2 , C_3 and C_4 depend on the oil API density:

	C_1	C_2	C_3	C_4
$API < 30$	0.05958	0.7972	13.1405	1.0014
$API > 30$	0.0315	0.7587	11.2895	1.0937

Lasater

The Lasater formula for the solution gas-oil ratio:

$$R_s(P, T) = C \cdot 132755 \cdot \frac{Y_G}{(1 - Y_G)} \cdot \frac{\gamma_O}{MW_O} \quad \text{Eq. 4.278}$$

$$Y_G = 0.08729793 + 0.37912718 \cdot \ln \left(\frac{P \cdot \gamma_G}{T + 460} + 0.769066 \right) \quad \text{Eq. 4.279}$$

The oil molecular weight is given by

$$MW_O = 677.3893 - 13.2161 \cdot API + 0.024775 \cdot API^2 + 0.00067851 \cdot API^3 \quad \text{Eq. 4.280}$$

The oil specific gravity is given by

$$\gamma_O = \frac{141.5}{API + 131.5} \quad \text{Eq. 4.281}$$

C is a [calibration \(p.512\)](#) constant.

Petrosky and Farshad

The Petrosky and Farshad formula for the solution gas-oil ratio is

$$R_s(P, T) = C \cdot \left[\left(\frac{P}{112.727} + 12.34 \right) \cdot \frac{\gamma_G^{0.8439}}{A(T)} \right]^{\frac{1}{0.5774}} \quad \text{Eq. 4.282}$$

Here A is a function of the fluid temperature and the oil API density:

$$A(T) = 4.561 \cdot 10^{-5} \cdot T^{1.3911} - 7.916 \cdot 10^{-4} \cdot API^{1.541} \quad \text{Eq. 4.283}$$

C is a [calibration \(p.512\)](#) constant.

Standing

The Standing formula for the solution gas-oil ratio used in PIPESIM is:

$$R_s(P, T) = C \cdot \gamma_G \cdot \left[\frac{P}{A(T) \cdot 18} \right]^{1/0.83} \quad \text{Eq. 4.284}$$

Here A is a function of the fluid temperature and the oil API density:

$$\log_{10} A = 0.00091 \cdot T - 0.0125 \cdot API \quad \text{Eq. 4.285}$$

C is a [calibration \(p.512\)](#) constant.

Vasquez and Beggs

The Vasquez and Beggs formula for the solution gas-oil ratio used in PIPESIM is:

$$R_s(P, T) = \frac{C}{C_1} \cdot \gamma_g \cdot (P - 14.7)^{C_2} \cdot A(T) \quad \text{Eq. 4.286}$$

Here A is a function of the fluid temperature and the oil API density:

$$\log_{10} A = \frac{C_3 \cdot API}{T + 460} \quad \text{Eq. 4.287}$$

C is a [calibration \(p.512\)](#) constant.

The constants C_1 , C_2 and C_3 depend on the oil API density:

	C_1	C_2	C_3
$API < 30$	11.172	1.0937	11.172
$API > 30$	10.393	1.187	10.393

Calibration

If a calibration data point is provided, $R_s^{scal} = R_s(P_{cal}, T_{cal})$, then the calibration term C is calculated to ensure the calibration point is a solution of the relevant solution gas-oil ratio equation. For example, for the [Vasquez and Beggs \(p.511\)](#) equation, the calibration term will be given by

$$R_s^{scal} = \frac{C}{C_1} \cdot \gamma_g \cdot (P_{cal} - 14.7)^{C_2} \cdot A(T_{cal}) \quad \text{Eq. 4.288}$$

Hence the [Vasquez and Beggs \(p.511\)](#) equation for the solution gas oil ratio can be re-written as:

$$R_s(P, T) = R_s^{scal} \cdot \left(\frac{P - 14.7}{P_{cal} - 14.7} \right)^{C_2} \cdot \frac{A(T)}{A(T_{cal})} \quad \text{Eq. 4.289}$$

It is assumed that the calibration point is a [bubble point \(p.512\)](#), although this will in fact only be the case if the calibration solution gas-oil ratio R_s^{scal} is equal to the fluid GOR.

If no calibration data is provided, PIPESIM uses $C = 1$.

Bubble point pressure

The bubble point pressure $P_b(T)$ is the pressure at which all the free gas is dissolved, i.e. when the solution gas-oil ratio is equal to the fluid GOR:

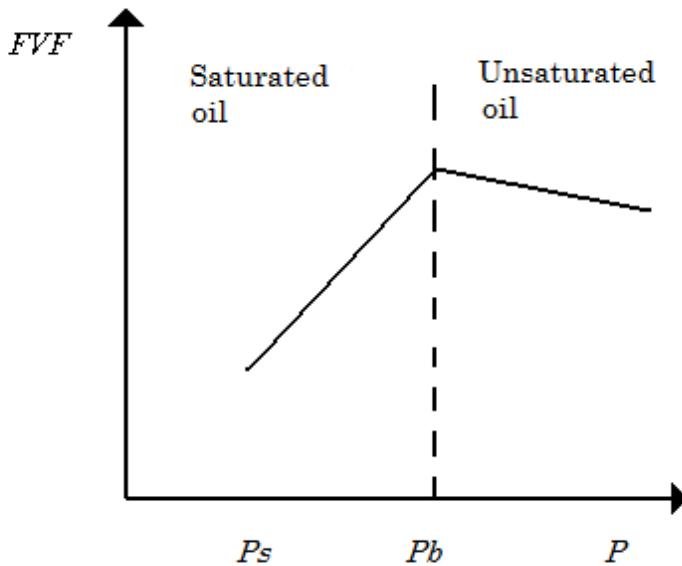
$$R_s(P_b, T) = R_{sb} \quad \text{Eq. 4.290}$$

The bubble point can therefore be determined by solving the relevant [solution gas-oil ratio \(p.508\)](#) equation.

Oil Formation Volume Factor

The oil formation volume factor (FVF) is the ratio of the oil volume (at a given pressure and temperature) to the stock tank oil volume. As pressure increases, two competing processes take place: gas is dissolved in oil which increases the volume, and the oil is compressed, which decreases the volume. Below the bubble point, the effect of gas dissolving in oil dominates and the saturated oil FVF increases with pressure. However at the bubble point pressure, all the available

gas has dissolved in the oil. Therefore above the bubble point pressure the only effect is compressibility and the undersaturated oil FVF increases with pressure.



Separate correlations are available for the [saturated oil FVF \(p.513\)](#) and [undersaturated oil FVF \(p.514\)](#).

Oil Formation Volume Factor for Saturated Systems

For saturated systems $P < P_b$ the oil formation volume factor B_{ob} (bbl/STB) depends on the solution gas-oil ratio R_s and the temperature T .

Standing

The saturated oil formation volume factor is given by:

$$B_{ob} = 0.972 + 0.000147F^{1.175} \quad \text{Eq. 4.291}$$

where the correlating factor is calculated using :

$$F = R_s \left(\frac{\gamma_g}{\gamma_o} \right)^{0.5} + 1.25T \quad \text{Eq. 4.292}$$

[Data used to develop correlation \(p.507\)](#)

Vasquez and Beggs

The saturated oil formation volume factor is given by:

$$B_{ob} = 1 + C_1 R_s + C_2 (T - 60) \left(\frac{API}{\gamma_G} \right) + C_3 R_s (T - 60) \left(\frac{API}{\gamma_G} \right)^2 \quad \text{Eq. 4.293}$$

	C_1	C_2	C_3
$API < 30$	$4.677 \cdot 10^{-4}$	$1.751 \cdot 10^{-5}$	$-1.81 \cdot 10^{-8}$
$API > 30$	$4.67 \cdot 10^{-4}$	1.100×10^{-5}	1.337×10^{-9}

[Data used to develop correlation \(p.507\)](#)

Kartoatmodjo and Schmidt

The saturated oil formation volume factor is given by:

$$B_{ob} = 0.98496 + 0.0001F^{1.50} \quad \text{Eq. 4.294}$$

Where the correlating factor

$$F = R_S \frac{\gamma_g^{0.755}}{\gamma_o^{0.25}} + 0.45T^{-1.50} \quad \text{Eq. 4.295}$$

Oil Formation Volume Factor for Undersaturated Systems

The oil formation volume factor B_o (bbl/STB) for pressures above the bubble point is given by a simple compressibility law:

$$B_o = B_{ob}(R_{sb}) \cdot \exp[\lambda Z_o(p_b - p)] \quad \text{Eq. 4.296}$$

where Z_o is the oil compressibility and λ is a calibration factor (used in mixing different fluids).

Vasquez and Beggs

The Vasquez and Beggs correlation for the oil compressibility is

$$Z_o = 10^{-5} \cdot \frac{5 \cdot R_{sb} + 17.2 \cdot T - 1180 \cdot \gamma_G + 12.61 \cdot API - 1433}{P} \quad \text{Eq. 4.297}$$

[Data used to develop correlation \(p.507\)](#)

TURZO Method

The performance of a rotodynamic (centrifugal or vertical) pump on a viscous liquid differs from the performance on water, which is the basis for most published curves. Typically, head and rate of flow decrease as viscosity increases, while power and the net positive suction head required (NPSHR) increases. Starting torque could be affected.

The following formula is the TURZO equation for viscosity correction:

$$\text{Power} = \frac{Q \cdot H}{E} \quad \text{Eq. 4.298}$$

where

P is power.

Q is the rate.

H is the head.

E is the efficiency.

$$Q_v = Q \cdot f_Q \quad \text{Eq. 4.299}$$

$$H_v = H \cdot f_H \quad \text{Eq. 4.300}$$

$$E_v = E \cdot f_E \quad \text{Eq. 4.301}$$

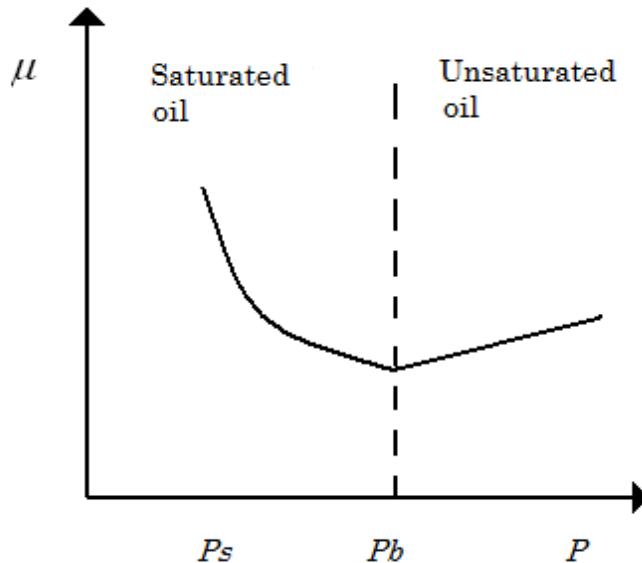
where the value of each equation is less than 1.

The viscosity correction is calculated as follows:

$$\frac{P_v}{P} = \frac{f_Q \cdot f_H}{f_E} \quad \text{Eq. 4.302}$$

Oil Viscosity

As pressure increases, two competing processes take place: gas is dissolved in oil which lightens the oil, reducing its viscosity, and the oil is compressed, which increases the viscosity. Below the bubble point, the effect of gas dissolving in oil dominates and the saturated viscosity decreases with pressure. However at the bubble point pressure, all the available gas has dissolved in the oil. Therefore above the bubble point pressure the only effect is compressibility and the undersaturated viscosity decreases with pressure.



Three sets of correlations are used to determine the oil viscosity:

1. At stock tank pressure the oil viscosity is given by [dead oil viscosity correlations \(p.516\)](#) as a function of the flowing fluid temperature $\mu_o(P_s, T) = \mu_{od}(T)$.
2. At pressures below the bubble point the oil viscosity is given by [live oil viscosity correlations \(p.518\)](#) as a function of the dead oil viscosity and the solution gas-oil ratio $\mu_o(P, T) = \mu_{ob}(\mu_{od}, R_s)$.

3. At pressures above the bubble point the oil viscosity is given by [undersaturated oil viscosity correlations \(p.520\)](#), as a function of the bubble point viscosity and the pressure

$$\mu_o(P, T) = \mu_{ou}(\mu_{ob}, P).$$

Dead Oil Viscosity

The correlations available for calculating dead oil viscosity are:

Beggs and Robinson

Dead oil viscosity is calculated as follows :

$$\mu_{od} = 10^x - 1 \quad \text{Eq. 4.303}$$

where $x = yT^{-1.163}$

and $y = 10^z$ and $z = 3.0324 - 0.02023 \cdot g_{API}$

[Data used to develop correlation \(p.507\)](#)

Glasø

Dead oil viscosity is calculated as follows :

$$\mu_{od} = c[\log_{10}(g_{API})]^d \quad \text{Eq. 4.304}$$

where

$$c = 3.141 \cdot 10^{10} \cdot T^{-3.444} \text{ and } d = 10.313 \cdot \log_{10}(T) - 36.447$$

Kartoatmodjo and Schmidt

Dead oil viscosity is calculated as follows :

$$\mu_{od} = c[\log_{10}(g_{API})]^d \quad \text{Eq. 4.305}$$

where

$$c = 16 \cdot 10^8 \cdot T^{-2.8177} \text{ and } d = 5.7526 \cdot \log_{10}(T) - 26.9718$$

De Ghetto et al

De Ghetto et al. use a combination of four correlations to compute the dead oil viscosity depending on the value of the API.

For API < 10 (extra heavy oils) the following correlation is used:

$$\mu_{od} = 10^x - 1 \quad \text{Eq. 4.306}$$

where $x = 10^y$ and $y = 1.90296 - 0.012619 \cdot g_{API} - 0.61748 \cdot \log_{10}(T)$

For 10 < API < 22.3 (heavy oils) the following correlation is used:

$$\mu_{od} = 10^x - 1 \quad \text{Eq. 4.307}$$

where $x = 10^y$ and $y = 2.06492 - 0.0179 \cdot g_{API} - 0.70226 \cdot \log_{10}(T)$

For $22.3 < \text{API} < 31.1$ (medium oils) the following correlation is used:

$$\mu_{od} = c[\log_{10}(g_{API})]^d \quad \text{Eq. 4.308}$$

where $c = 220.15 \cdot 10^9 \cdot T^{-3.556}$ and $d = 12.5428 \cdot \log_{10}(T) - 45.7874$

For $\text{API} > 31.1$ (light oils) the following correlation is used

$$\mu_{od} = 10^x - 1 \quad \text{Eq. 4.309}$$

where $x = 10^y$ and $y = 1.67083 - 0.017628 \cdot g_{API} - 0.61304 \cdot \log_{10}(T)$

Petrosky and Farshad

Dead oil viscosity is calculated as follows:

$$\mu_{od} = c[\log_{10}(g_{API})]^d \quad \text{Eq. 4.310}$$

where $c = 2.3511 \cdot 10^7 \cdot T^{-2.10255}$ and $d = 4.59388 \cdot \log_{10}(T) - 22.82792$

Hossain et al

Hossain et al. correlation for dead oil viscosity is only valid for heavy oils ($10 < \text{API} < 22.3$) and it is given as follows:

$$\mu_{od} = 10^A \cdot T^B \quad \text{Eq. 4.311}$$

where $A = -0.71523 \cdot g_{API} + 22.13766$ and $B = 0.269024 \cdot g_{API} - 8.268047$

Elsharkawy and Alikhan

Elsharkawy and Alikhan dead oil viscosity is only valid in the API range 20-48 and is calculated as follows:

$$\mu_{od} = 10^x - 1 \quad \text{Eq. 4.312}$$

where $x = 10^y$ and $y = 2.16924 - 0.02525 \cdot g_{API} - 0.68875 \cdot \log_{10}(T)$

User's data

If **user's data** is selected for the dead oil viscosity method, then a curve is fitted through the two supplied data points (μ_1, T_1) and (μ_2, T_2) of the following form:

$$\log(\mu_{od}) = \log(B) - C \log(T) \quad \text{Eq. 4.313}$$

where

$$C = \frac{\log\left(\frac{\mu_1}{\mu_2}\right)}{\log\left(\frac{T_2}{T_1}\right)}$$
Eq. 4.314

and

$$B = \mu_1 T_1^C = \mu_2 T_2^C$$
Eq. 4.315

Live Oil Viscosity Correlations

Many of the correlations available for calculating live oil viscosity are of the form

$$\mu_{ob} = A \cdot \mu_{od}^B$$
Eq. 4.316

where A and B are functions of the Solution gas-oil ratio R_s :

Correlation		A	B
Chew and Connally	Data used to develop correlation (p.507)	$0.2 + \left(\frac{0.8}{10^{0.000852R_s}} \right)$	$0.482 + \left(\frac{0.518}{10^{0.000777R_s}} \right)$
Beggs and Robinson	Data used to develop correlation (p.507)	$10.715 \cdot (R_s + 100)^{-0.515}$	$5.44 \cdot (R_s + 150)^{-0.338}$
Elsharkawy and Alikhan		$1241.932 \cdot (R_s + 641.026)^{-1.12410}$	$1768.841 \cdot (R_s + 1180.335)^{-1.06622}$
Hossain et al		$1 - 1.7188311 \cdot 10^{-3} \cdot R_s +$ $+ 1.58031 \cdot 10^{-6} \cdot R_s^2$	$1 - 2.052461 \cdot 10^{-3} \cdot R_s +$ $+ 3.47559 \cdot 10^{-6} \cdot R_s^2$
Petrosky and Farshad		$0.1651 + 0.6165 \cdot 0.9886 \cdot 10^{-4R_s}$	$0.5131 + 0.5109 \cdot 0.9973 \cdot 10^{-3R_s}$

Other authors use more complicated formulas:

Kartoatmodjo and Schmidt

Live oil viscosity is calculated as follows:

$$\mu_{ob} = -0.06821 + 0.9824F + 0.0004034F^2$$
Eq. 4.317

where

$$F = A \cdot \mu_{od}^{0.43+0.5165 \cdot B} \quad \text{Eq. 4.318}$$

and

$$A = 0.2001 + 0.8428 \left[10^{-0.000845 R_s} \right] \quad \text{Eq. 4.319}$$

and

$$B = 10^{-0.00081 R_s} \quad \text{Eq. 4.320}$$

Khan

Live oil viscosity calculated by Khan is a function of the gas and oil specific gravities (γ_G , γ_O), the solution gas-oil ratio (R_s), the bubble pressure (P_b), and the flowing pressure (P). It is given as follows:

$$\mu_{ob} = A \cdot e^y \quad \text{Eq. 4.321}$$

where

$$y = \ln(0.09) + 0.5 \cdot \ln(\gamma_G) - \frac{1}{3} \cdot \ln(R_s) - 4.5 \cdot \ln\left(\frac{T}{460}\right) - 3 \cdot \ln(1 - \gamma_O) \quad \text{Eq. 4.322}$$

and

$$A = \left(\frac{P}{P_b} \right)^{-0.14} \cdot e^{-2.5 \times 10^{-4}(P - P_b)} \quad \text{Eq. 4.323}$$

De Ghetto et al

De Ghetto et al. expression of the live oil viscosity is a combination of 4 correlations depending on the value of oil API.

		F	A	B
For API < 10 (extra heavy oils)	$\mu_{ob} = 2.3945 + 0.8927F + 0.01567F^2$	$A \cdot \mu_{od}^{0.5798+0.3432B}$	$-0.0335 + 1.0875 \left[10^{-0.000845 \cdot R_s} \right]$	$10^{-0.00081 \cdot R_s}$
For $10 < \text{API} < 22.3$ (heavy oils)	$\mu_{ob} = -0.6311 + 1.078F - 0.003653F^2$	$A \cdot \mu_{od}^{0.4731+0.5664B}$	$0.2478 + 0.6114 \left[10^{-0.000845 \cdot R_s} \right]$	$10^{-0.00081 \cdot R_s}$
For $22.3 < \text{API} < 31.1$	$\mu_{ob} = 0.0132 + 0.9821F - 0.005215F^2$	$A \cdot \mu_{od}^{0.3855+0.5664B}$	$0.2038 + 0.8591 \left[10^{-0.000845 \cdot R_s} \right]$	$10^{-0.00081 \cdot R_s}$

(medium oils)				
For API > 31.1 (light oils)	$\mu_{ob} = A \cdot \mu_{od}^B$		$25.1921 \cdot (R_s + 100)^{-0.6487}$	$2.7516 \cdot (R_s + 150)^{-0.6487}$

Undersaturated Oil Viscosity

The correlations available for calculating undersaturated oil viscosity are:

Vasquez and Beggs

Undersaturated oil viscosity is calculated as follows:

$$\mu_{ou} = \mu_{ob} \left(\frac{p}{p_b} \right)^A \quad \text{Eq. 4.324}$$

where $A = 2.6 p^{1.187} \exp(-8.98 \times 10^{-5} p - 11.513)$

Data used to develop correlation (p.507)

Kouzel

Undersaturated oil viscosity is derived from the equation:

$$\mu_{ou} = \mu_{ob} \cdot \frac{10^{F(p)}}{10^{F(p_b)}} \quad \text{Eq. 4.325}$$

$$F(p) = \frac{p - 14.7}{1000(A + B\mu_{od}^{0.278})} \quad \text{Eq. 4.326}$$

Where A and B are parameters entered by the user. Suggested values for A and B are 0.0239 and 0.01638 respectively.

Kartoatmodjo and Schmidt

Undersaturated oil viscosity is calculated as follows:

$$\mu_{ou} = 1.00081 \mu_{ob} + 0.001127 A(p - p_b) \quad \text{Eq. 4.327}$$

$$A = -0.006517(\mu_{ob}^{1.8148}) + 0.038(\mu_{ob}^{1.59})$$

Khan

Undersaturated oil viscosity is calculated as follows:

$$\mu_{ou} = \mu_{ob} \exp[9.65 e^{-5}(p - p_b)] \quad \text{Eq. 4.328}$$

De Ghetto et al

De Ghetto et al. expression of the undersaturated oil viscosity is a combination of 3 correlations depending on the value of oil API.

For API < 10 (extra heavy oils) the following correlation is used:

$$\mu_{ou} = \mu_{ob} - \left(1 - \frac{p}{p_b}\right) \left(\frac{A}{B}\right) \quad \text{Eq. 4.329}$$

where $A = 10^{-2.19} \left(\mu_{od}^{1.055}\right) \left(p_b^{0.3132}\right)$ and $B = 10^{(0.0099g_{API})}$

For 10 < API < 22.3 (heavy oils) the following correlation is used:

$$\mu_{ou} = 0.9886\mu_{ob} + 0.002763A(p - p_b) \quad \text{Eq. 4.330}$$

where $A = -0.01153 \left(\mu_{ob}^{1.7933}\right) + 0.03610 \left(\mu_{ob}^{1.5939}\right)$

For API > 22.3 (medium and light oils) the following correlation is used:

$$\mu_{ou} = \mu_{ob} - \left(1 - \frac{p}{p_b}\right) \frac{A}{B} \quad \text{Eq. 4.331}$$

where $A = 10^{-2.19} \left(\mu_{od}^{1.055}\right) \left(p_b^{0.3132}\right)$ and $B = 10^{(-0.00288g_{API})}$

Hossain et al

Undersaturated oil viscosity is calculated as follows:

$$\mu_{ou} = \mu_{ob} + 0.004481(p - p_b)(A - B) \quad \text{Eq. 4.332}$$

where $A = 0.555955 \left(\mu_{ob}^{1.068099}\right)$ and $B = 0.527737 \left(\mu_{ob}^{1.063547}\right)$

Petrosky and Farshad

Undersaturated oil viscosity is calculated as follows:

$$\mu_{ou} = \mu_{ob} + 1.3449E^{-3}(p - p_b)(10^A) \quad \text{Eq. 4.333}$$

where $A = -1.0146 + 1.3322X - 0.4876X^2 - 1.15036X^3$ and $X = \log_{10}(\mu_{ob})$

Elsharkawy and Alikhan

Undersaturated oil viscosity is calculated as follows:

$$\mu_{ou} = \mu_{ob} + A[10^{-2.0771}](p - p_b) \quad \text{Eq. 4.334}$$

where $A = \left[\mu_{od}^{1.19279}\right] \cdot \left[\mu_{ob}^{-0.40712}\right] \cdot \left[p_b^{-0.7941}\right]$

Bergman and Sutton

Undersaturated oil viscosity is calculated as follows:

$$\mu_{ou} = \mu_{ob} \exp \left[A(p - p_b)^B \right] \quad \text{Eq. 4.335}$$

where $A = 2.278877 \cdot 10^{-4} - 1.48211 \cdot 10^{-5} \cdot X + 6.5698 \cdot 10^{-7} \cdot X^2$,

$$B = 0.873204 + 2.24623 \cdot 10^{-2} \cdot X \text{ and } X = \log_{10}(\mu_{ob})$$

Disabling the calculation of undersaturated oil viscosity

If you select **None** as the undersaturated oil viscosity method, then the undersaturated oil viscosity is assumed to be the same as the saturated live oil viscosity at the same temperature and pressure.

Gas Compressibility

The real gas law is given by $pV = ZRT$ where

p	pressure
V	volume
R	universal gas constant
T	absolute temperature
Z	gas compressibility factor

Numerous equations of state have been proposed to predict this Z-factor. Standing and Katz presented a generalized Z-factor chart for predicting the volumetric behavior of natural gases. To employ this chart, we require knowledge of the critical properties of the gas (namely, critical pressure and critical temperature) as a function of the specific gravity. These are given in the black oil model by Standing (1977) for natural gas and gas-condensate systems:

Gas Systems

$$T_c = 168 + 325 \gamma_G - 12.5 \gamma_G^2 \quad \text{Eq. 4.336}$$

Gas-Condensate Systems

$$T_c = 187 + 330 \gamma_G - 71.5 \gamma_G^2 \quad \text{Eq. 4.337}$$

$$p_c = 706 - 51.7 \gamma_G - 11.1 \gamma_G^2 \quad \text{Eq. 4.338}$$

where

T_c	critical temperature
-------	----------------------

p_c	critical pressure
γ_G	specific gravity of the gas mixture

This allows us to calculate the reduced temperature and reduced pressure, defined respectively as:

$$T_c = \frac{T}{T_c} \quad \text{Eq. 4.339}$$

$$p_c = \frac{p}{p_c} \quad \text{Eq. 4.340}$$

Various correlations have been proposed for curve fitting this reduced pressure-reduced temperature Z-factor chart and are available in PIPESIM:

[Hall-Yarborough Z-Factor Correlation \(p.523\)](#)

[Standing Z-Factor Correlation \(p.523\)](#)

[Robinson et al. Z-Factor Correlation \(p.524\)](#)

Hall-Yarborough Z-Factor Correlation

Gas compressibility (Z-factor) is calculated as follows:

$$Z = \left(\frac{0.06125 p_R T_R}{\rho_R} \right) e^{x \left[-1.2 (1 - T_R)^2 \right]} \quad \text{Eq. 4.341}$$

where the reduced density is a root of the following equation:

$$F(\rho_R) = 0.06125 p_R T_R \left[-1.2 (1 - T_R)^2 \right] + \frac{\left(\rho_R^2 + \rho_R^3 + \rho_R^4 - \rho_R^5 \right)}{\left((1 - \rho_R)^3 - \left(14.67 T_R^2 - 9.76 T_R^2 4.85 T_R^3 \right) \rho_R \right)^2} + \left(90.7 T_R - 242.2 T_R^2 + 42.4 T_R^3 \right) \rho_R^{2.18+2.82 T_R} = 0 \quad \text{Eq. 4.342}$$

where

where ρ_R	reduced density
p_R	reduced pressure
T_R	reciprocal of the reduced temperature

The method is not recommended for use within a pressure range $p_R = [0, 1]$.

Standing Z-Factor Correlation

Gas compressibility (Z-factor) is calculated as follows:

$$Z = \frac{A + (1 - A)}{e^x B + F P_r^G} \quad \text{Eq. 4.343}$$

where the coefficients A to G are given by:

$A = 1.39(T_r - 0.92)^{0.5} - 0.36 T_R - 0.101$
$B = (0.62 - 0.23 T_r) p_R + \left[\frac{0.666}{(T_r - 0.86)} - 0.037 \right] p_R^2 + \frac{0.32 p_R^6}{10^9 (T_R^{-1})}$
$C = (0.132 - 0.32 \log(T_R))$
$D = 10^{(0.3016 - 0.49 T_R + 0.1824 T_R^2)}$

The method is not valid for $T_R < 0.92$.

Robinson et al. Z-Factor Correlation

Gas compressibility (Z-factor) is calculated as follows:

$$Z = 1 + \left[A_1 + \left(\frac{A_2}{T_R} \right) + \left(\frac{A_3}{T_R^3} \right) \right] \rho_R + \left[A_4 + \left(\frac{A_5}{T_D} \right) \right] \rho_R^2 + \left[\frac{A_5 A_6}{\rho_R} \right] D_R^5 + \left[\frac{A_7}{T_r^3} \right] \rho_R^2 \left(1 + A_8 \rho_R^2 \right) e^{x(-A_8 \rho_R^2)} \quad \text{Eq. 4.344}$$

where

$D = \frac{0.27 p_R}{T_R}$
$A_1 = 0.310506237$
$A_2 = -1.4067099$
$A_3 = -0.57832729$
$A_4 = 0.53530771$
$A_5 = -0.61232032$

$A_6 = -0.10488813$
$A_7 = 0.68157001$
$A_8 = 0.68446549$

The method is valid within a temperature and pressure range of $T_r = [1.05, 3.0]$ and $P_r = [0.2, 3.0]$.

Gas Viscosity

Gas viscosity is calculated using the [Lee et al \(p.589\)](#) correlation as follows:

$$\mu_g = K \cdot \exp \left[X \cdot \rho_g^Y \right] \quad \text{Eq. 4.345}$$

where

$$K = \frac{(7.77 + 0.183 \cdot \gamma_G) \cdot (T + 460)^{1.5}}{(122.4 + 373.6 \cdot \gamma_G + T + 460)} \cdot 10^{-4}$$

$$X = 2.57 + \frac{1914.5}{T} + 0.275 \gamma_g$$

$$Y = 1.11 + 0.04X$$

μ_g is the gas viscosity (cp)

γ_g is the gas specific gravity

ρ_g is the gas density (g/cc)

T is the temperature ($^{\circ}\text{F}$)

Surface Tension

Oil-gas Surface Tension

The oil-gas surface tension is given by [Baker and Swerdloff \(p.583\)](#):

$$\sigma_O = [37.7 - 0.05 \cdot (T - 100) - 0.26 \cdot API] \cdot \left[1 - 7.1 \cdot 10^{-4} \cdot P + 2.1 \cdot 10^{-7} \cdot P^2 + 2.37 \cdot 10^{-11} \cdot P^3 \right] \quad \text{Eq. 4.346}$$

σ_O is the surface tension between the oil and the gas (dynes/cm)

P is the pressure (psia)

T is the temperature ($^{\circ}\text{F}$)

API is the oil API gravity

Water-gas Surface Tension

The water-gas surface tension is given by Katz:

$$\sigma_W = 70 - 0.1 \cdot (T - 74) - 0.002 \cdot P \quad \text{Eq. 4.347}$$

σ_W is the surface tension between the water and the gas (dynes/cm)

P is the pressure (psia)

T is the temperature ($^{\circ}\text{F}$)

Black Oil Enthalpy

Black Oil Fluid Enthalpy Model

The black oil fluid model makes some approximations in the entropy balance based upon the thermodynamic behavior of typical hydrocarbon fluids. The black oil model is suitable for light, medium and heavy oil based fluids, particularly if significant quantities of water are present. The black oil model is fast, simple to use and easy to calibrate. It is also suitable for gas and gas/condensate screening studies.

There are currently two black oil enthalpy calculation methods available in PIPESIM.

2009 Method

The enthalpy of the gas phase is given by:

$$H_g = c_{p_g} T - \eta_g c_{p_g} P + \Delta H_{vap} \quad \text{Eq. 4.348}$$

The enthalpy of the oil phase is given by:

$$H_o = c_{p_o} T - \eta_o c_{p_o} P \quad \text{Eq. 4.349}$$

The enthalpy of the water phase is given by:

$$H_w = c_{p_w} T - \eta_w c_{p_w} P \quad \text{Eq. 4.350}$$

where the gas, oil and water Joule Thomson coefficients are approximated by (Ref: [Alves, Alhanati and Shoham \(p.583\)](#):

$$\eta_g = \frac{1}{\rho_g c_{p_g}} \left[\frac{T}{Z} \frac{\partial Z}{\partial T} \right] \Big| 5.40395 \quad \text{Eq. 4.351}$$

$$\eta_o = - \frac{1}{\rho_o c_{p_o}} \Big| 5.40395 \quad \text{Eq. 4.352}$$

$$\eta_w = - \frac{1}{\rho_w c_{p_w}} \Big| 5.40395 \quad \text{Eq. 4.353}$$

The total enthalpy of the fluid is given by:

$$H = H_g w_g + H_o w_o + H_w w_w \quad \text{Eq. 4.354}$$

where:

H	is the specific enthalpy	BTU / lb
T	is the flowing temperature	oF
P	is the flowing pressure	$psia$
c_p	is the specific heat capacity at constant pressure	$BTU / lb ^oF$
η	is the Joule Thomson coefficient	$^oF / psia$
ρ	is the flowing density	lb / ft^3
Z	is the gas compressibility factor	dimensionless
w	is the flowing mass fraction	dimensionless
ΔH_{vap}	is the latent heat of vaporization	BTU / lb

$$1 BTU / ft^3 = 5.40395 psia$$

1983 Method

The enthalpy of the gas phase is given by:

$$H_g = c_{p_g} T + P \left[(1.619 \times 10^{-10} P + 1.412 \times 10^{-6}) P - 0.02734 \right] \quad \text{Eq. 4.355}$$

The enthalpy of the oil phase is given by:

$$H_o = c_{p_o} T + 3.36449 \times 10^{-3} P \quad \text{Eq. 4.356}$$

The enthalpy of the water phase is given by:

$$H_w = c_{p_w} T + \left(\frac{2.9641 \times 10^{-3}}{\gamma_w} \right) P \quad \text{Eq. 4.357}$$

The total enthalpy of the fluid is given by:

$$H = H_g m_g + H_o m_o + H_w m_w \quad \text{Eq. 4.358}$$

where:

m	is the stock tank mass fraction	dimensionless
γ	is the stock tank specific gravity	dimensionless

Black Oil Mixing

Introduction

Mixing occurs in network models, when two or more streams meet at a junction and in single branch models where injected fluid, or fluids from multiple completions mix with fluid already in the branch. The fluid properties of the mixed stream need to be determined.

Stock Tank Oil Properties

Phase ratios (Gas Oil Ratio / Water cut)

The phase ratios for a mixed stream are calculated by adding the individual phase rates of each stream and then calculating the ratio of the phases. The calculations are at stock tank conditions.

$$GOR_{mix} = \frac{Q_{vg,mix}}{Q_{vo,mix}} \quad \text{Eq. 4.359}$$

$$WCUT_{mix} = \frac{Q_{vw,mix}}{Q_{vw,mix} + Q_{vo,mix}} \quad \text{Eq. 4.360}$$

Here:

GOR_{mix}	is the gas oil ratio of the mixture
$WCUT_{mix}$	is the water cut of the mixture
$Q_{vo,mix} = \sum_{i=1}^n Q_{vo,i}$	is the stock tank oil volume rate of the combined stream
$Q_{vw,mix} = \sum_{i=1}^n Q_{vw,i}$	is the stock tank water volume rate of the mixture
$Q_{vg,mix} = \sum_{i=1}^n Q_{vg,i}$	is the stock tank gas volume rate of the mixture

$Q_{vo,i} = Q_{vL,i} \times (1 - WCUT_i)$	is the stock tank oil volume rate of stream i
$Q_{vw,i} = Q_{vL,i} \times WCUT_i$	is the water volume rate of stream i
$Q_{vg,i} = Q_{vL,i} \times GOR_i$	is the stock tank gas volume rate of stream i
$Q_{vL,i} = Q_{vo,i} + Q_{vw,i}$	is the stock tank liquid volume rate of stream i
$WCUT_i$	is the water cut of stream i
GOR_i	is the gas oil ratio of stream i
n	is the number of streams in the mixture

Phase densities

The phase densities (and specific gravities) are determined as a volumetric average of the input stream densities:

$$DOD_{mix} = \frac{\sum DOD_i \times Q_{vo,i}}{Q_{vo,mix}} \quad \text{Eq. 4.361}$$

$$GSG_{mix} = \frac{\sum GSG_i \times Q_{vg,i}}{Q_{vg,mix}} \quad \text{Eq. 4.362}$$

$$WSG_{mix} = \frac{\sum WSG_i \times Q_{vw,i}}{Q_{vw,mix}} \quad \text{Eq. 4.363}$$

Here:

DOD_{mix}	is the dead oil density of the mixture
GSG_{mix}	is the gas specific gravity of the mixture
WSG_{mix}	is the water specific gravity of the mixture
DOD_i	is the dead oil density of stream i
GSG_i	is the gas specific gravity of stream i
WSG_i	is the water specific gravity of stream i

Contaminants

The mole fractions of contaminants for the mixed stream is determined using a gas phase volumetric average of the individual stream mole fractions:

$$Z_{j,mix} = \frac{\sum Z_{j,i} \times Q_{vg,i}}{Q_{vg,mix}} \quad \text{Eq. 4.364}$$

Here:

$Z_{j,mix}$	is the mole fraction of contaminant j in the mixture
$Z_{j,i}$	is the mole fraction of contaminant j in stream i

Thermal Data (Heat Capacity and thermal conductivity)

The phase thermal properties of mixed streams are calculated using mass averages of the phase properties of the input streams:

$$CP_{\varphi,mix} = \frac{\sum CP_{\varphi,i} \times Q_{\varphi,i}}{Q_{\varphi,mix}} \quad \text{Eq. 4.365}$$

$$K_{\varphi,mix} = \frac{\sum K_{\varphi,i} \times Q_{\varphi,i}}{Q_{\varphi,mix}} \quad \text{Eq. 4.366}$$

$$\Delta H_{vap,mix} = \frac{\sum \Delta H_{vap,i} \times Q_{g,i}}{Q_{g,mix}} \quad \text{Eq. 4.367}$$

Here:

φ	is the phase, oil $\varphi = O$, vapor (gas) $\varphi = G$, or water $\varphi = W$
$CP_{\varphi,mix}$	is the heat capacity of phase φ in the mixture
$K_{\varphi,mix}$	is the thermal conductivity of phase φ in the mixture
$\Delta H_{vap,mix}$	is the latent heat of vaporization of the gaseous phase g in the mixture
$CP_{\varphi,i}$	is the heat capacity of phase φ in stream i
$K_{\varphi,i}$	is the thermal conductivity of phase φ in stream i
$\Delta H_{vap,i}$	is the latent heat of vaporization of the gaseous phase g in stream i
$Q_{\varphi,mix} = \sum_{i=1}^n Q_{\varphi,i}$	is the mass flow rate of phase φ of the mixture
$Q_{\varphi,i}$	is the mass flow rate of phase φ of stream i

Correlations

Unlike other properties, the choice of correlations used for the combined fluid can not be decided by averaging. Instead, the selected correlation for the mixed stream is chosen as the one which

has the highest flow rate associated with it for the relevant phase. For example the correlation for mixture Oil Viscosity is set to be the correlation that has maximum stock tank rate associated with it.

While deciding the correlation for the mixed stream, we have to consider following rules:

- All properties are independent of each other. For example the choice for mixture dead Oil viscosity correlation has nothing to do with mixture live Oil viscosity
- Resultant mixture correlation is decided based on associated phase rate (for example, oil if we are deciding Oil property) ; not based on number of streams using that correlation
- Stock tank flow rates are used at the point of mixing.

Example 1

For an example assume we are mixing 7 flow streams which have different sets of correlations as tabulated below:

Stream	Flow rate (STB/day)	Dead Oil Viscosity	Live Oil Viscosity	Under-saturated Oil Viscosity
1	5000	Hossain	Kartoatmodjo	Vasquez and Beggs
2	2000	Glasso	Khan	Kouzel
3	4000	Petrosky-Farshad	Chew and Connally	Kouzel
4	3000	Beggs and Robinson	Khan	Kouzel
5	6000	Beggs and Robinson	Kartoatmodjo	Bergman and Sutton
6	8000	Glasso	Hossain	Bergman and Sutton
7	2000	Beggs and Robinson	Elsharkawy	Kouzel

The total oil flow for each correlation, and the correlations selected for the combined fluid are tabulated below:

Stream	Flow rate (STB/day)	Dead Oil Viscosity
4,5,7	11000	Beggs and Robinson
2,6	10000	Glasso
1	5000	Hossain
3	4000	Petrosky-Farshad
combined	30000	Beggs and Robinson

Stream	Flow rate (STB/day)	Live Oil Viscosity
1,5	11000	Kartoatmodjo
6	8000	Hossain
2,4	5000	Khan
3	4000	Chew and Connally
7	2000	Elsharkawy

combined	30000	Kartoatmodjo
Stream	Flow rate (STB/day)	Under-saturated Oil Viscosity
5,6	14000	Bergman and Sutton
2,3,4,7	11000	Kouzel
1	5000	Vasquez and Beggs
combined	30000	Bergman and Sutton

Example 2

Mixing of fluids that use different correlations may produce unexpected results. In the above example, a 51%-49% mixture of streams 1 and 2 will use the same correlations as stream 1, but a 49%-51% mixture will use the same correlations as stream 2. So, even though these two mixtures are similar, their properties may be modelled completely differently. Therefore it is important to select compatible correlations when modelling networks.

Calibration data

A number of correlations are calibrated using user supplied data. This section describes how streams with different calibration data are mixed.

Dead oil viscosity

Dead oil viscosity ([p.516](#)) can be specified using no calibration data; 2-point $\{(T_1, \mu_1), (T_2, \mu_2)\}$ calibration data; or as a **User Supplied Table** with multipoint calibration data $\{(T_1, \mu_1), \dots, (T_n, \mu_n)\}$. If none of the streams in a mixture use calibration data, then the mixing is done by simply determining the mixture correlation, as outlined in [Correlations \(p.530\)](#). If however, at least one of the input streams uses dead oil viscosity with calibration data then a **User Supplied Table** is used for the mixture deadoil viscosity. The table entries are calculated in three steps:

1. The number and value of the temperature points $\{T_x\}$ in the table are determined:
 - If the inlet streams use multipoint calibration, then the mixed stream will use multipoint calibration. PIPESIM will try to include all the input temperatures in the mixture table, up to a maximum of 40 points.
 - If the inlet streams only use 2-point calibration data, then the mixed stream will use only two points. The Temperature will be set to the minimum and maximum temperatures of the input stream calibration temperatures.
2. The viscosity of each inlet stream $\mu_i(T_x)$ is calculated at each temperature in the mixed stream table.
3. The mixture viscosity is calculated at each point in the stream using the Kendall and Monroe cubic mixing rule:

$$\mu_{comb}(T_x) = \left(\sum_{i=1}^n \frac{Q_{vo,i}}{Q_{vo,mix}} \cdot (\mu_i(T_x))^{1/3} \right)^3 \quad \text{Eq. 4.368}$$

Example 3

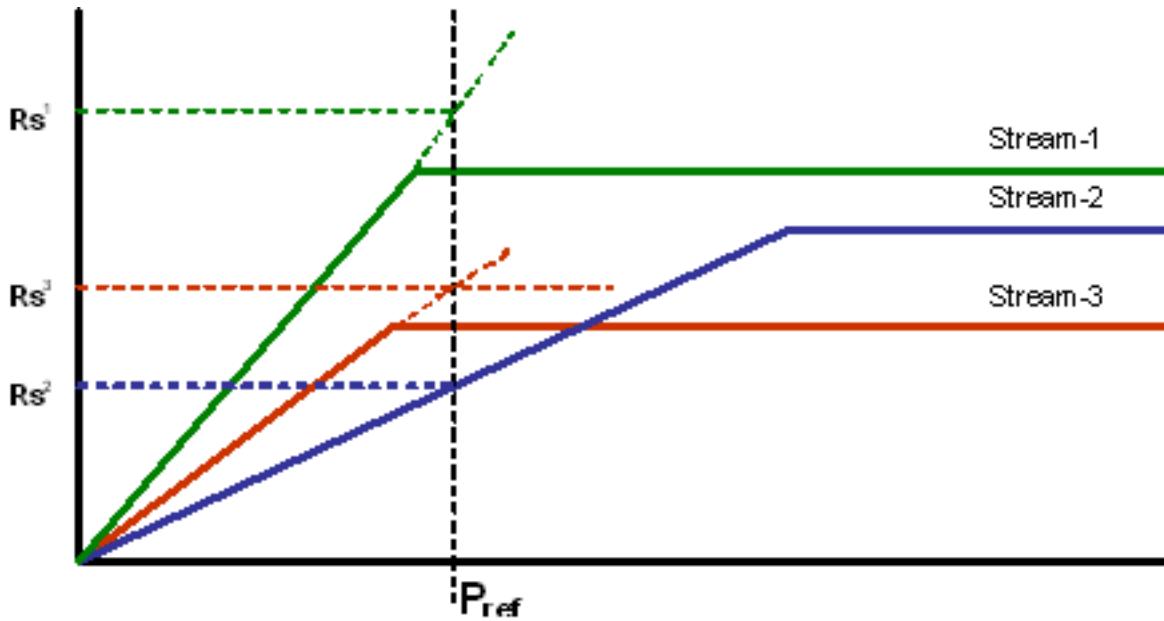
Streams 1–3, defined in the table below, mix at a junction. All streams use dead oil correlations without data, so the mixed stream, stream 4, uses the dead oil correlation with the biggest flow, in this case Beggs and Robinson. Stream 4 then mixes with stream 5 at another junction. Stream 5 uses a correlation with calibration data. Even though stream 5 has less flow than stream 4, the mixture, stream 6, will use a User Supplied Table to define its dead oil correlation.

Stream	Flow rate (STB/day)	Dead Oil Viscosity
1	2000	Beggs and Robinson
2	2500	Chew and Connally
3	1000	Beggs and Robinson
4= 1+2+3	5500	Beggs and Robinson
5	2500	Any correlation with 2-point data, or a User Supplied Table
6 = 4+5	8000	User Supplied Table

Solution Gas Rs

If one or more of the input streams has a single point calibration data, $R_{si}(P_{ref,i}, T_{ref,i})$, then the mixed stream solution gas will also be calibrated using a single point:

1. Determine the [correlation](#) (p.530) for the mixed stream.
2. Determine reference pressure and temperature values $(P_{ref,mix}, T_{ref,mix})$ for calibrating the mixed stream viscosity. These are calculated as the mass flow rate average of the input stream reference pressures and temperatures — for those input streams with calibration data.
3. Determine the solution gas of each stream at the reference pressure and temperature for the mixed stream $R_{si}(P_{ref,mix}, T_{ref,mix})$. For those streams that are undersaturated at $(P_{ref,mix}, T_{ref,mix})$ the “potential” solution gas is determined by extrapolation.



- The solution gas for the mixed stream is determined as a volume average of all the input stream solution gas (or potential solution gas) values.

Live oil viscosity

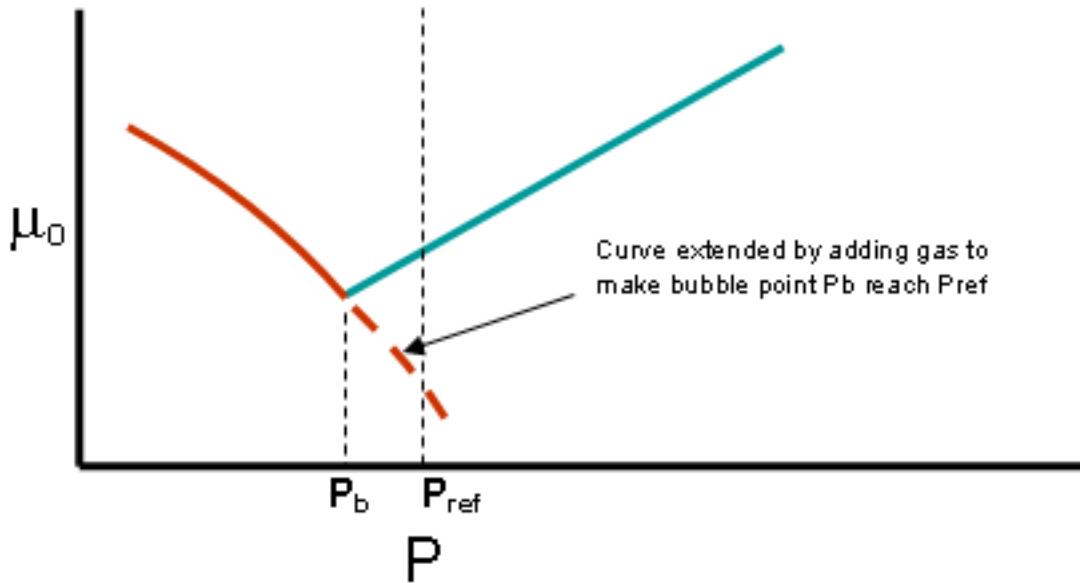
A number of steps are needed to determine the live oil viscosity of a mixture:

- Determine the [correlation \(p.530\)](#) for the mixed stream.
- Determine reference pressure and temperature values ($P_{ref,mix}$, $T_{ref,mix}$) for calibrating the mixed stream viscosity. These are calculated as the mass flow rate average of the input stream reference pressures and temperatures, for those input streams with calibration data. If the mixed stream is not saturated at the calculated reference pressure and temperature, the reference pressure is reduced to the saturation pressure

$$T_{ref,mix} = \sum_{i=1}^n \frac{Q_{o,i}}{Q_{o,mix}} T_{ref,i}$$

$$P_{ref,mix} = MIN \left[\sum_{i=1}^n \frac{Q_{o,i}}{Q_{o,mix}} P_{ref,i}, P_{sat,mix}(T_{ref,mix}) \right]$$

- Determine the input stream viscosities at the mixture reference pressure and temperature $\mu_{o,i}(P_{ref,mix}, T_{ref,mix})$. If an input stream is undersaturated at $(P_{ref,mix}, T_{ref,mix})$ then its viscosity is calculated by first adding more gas so that the stream is saturated. The added gas has a specific gravity equal to that of the mixture.



4. The live oil viscosity of the mixture is calculated at $(P_{ref,mix}, T_{ref,mix})$ using the [4.368 \(p.533\)](#) equation.
5. The live oil correlation can then be calibrated using $\mu_{o,mix}(P_{ref,mix}, T_{ref,mix})$

4.5.2 Compositional Fluid Modeling

In compositional fluid models the user can specify a number of components that make up the fluid. These can be real molecules, such as methane, ethane or water, known as library components or pseudo components that represent the properties of several molecules, known as petroleum fractions. The phase behaviour and thermodynamic properties are determined by an equation of state (EOS). This equation of state is either a [cubic equation \(p.536\)](#) (this is a modified form of the Van der Waals equation) or a [non-cubic equation \(p.539\)](#). The number of phases that can be modelled depends on the flash package:

- **Two phase flash.** Water is removed from the fluid and the remaining hydrocarbons are flashed to determine the amount of oil and vapour. This method is used for most compositional flash packages (and for black oil models). This means that water only appears in the water phase and does not appear in the vapour phase.
- **Three phase flash.** If the Multiflash compositional package is used, then a three phase flash is performed. This means that there is a possibility that water will appear in the vapour phase, and some components (e.g. water, methanol) will appear in the aqueous phase.

The three phase flash gives a more accurate model of water behaviour than the two phase flash. However, there can be problems when the flash produces two non-aqueous liquids — one of these may be mis-identified as water.

- **Multiphase flash.** The Multiflash compositional standalone package can be used to model vapour and three liquid phases as well as solid phases. Within PIPESIM flow simulations, Multiflash is only ever used to model two liquid phases. However, it can be used within

PIPESIM to plot phase envelopes and to predict whether solid phases (asphaltene, hydrates, wax and ice) would be present.

Cubic Equations of State

Equations of state (EoS) describe the pressure, volume and temperature (PVT) behavior of pure components and mixtures. The phase state and most thermodynamic properties (e.g. density, enthalpy, entropy) are derived from the equation of state. Separate models are used for transport properties, such as [Viscosity \(p.515\)](#), thermal conductivity and surface tension. PIPESIM can use both cubic and [non-cubic \(p.539\)](#) Equations of State.

Volume shift (three-parameter) and acentric factor corrections, if available, are recommended for the cubic equations of state.

The cubic equation of state can be written as:

$$P = \frac{nRT}{V - b} + \frac{a}{(V + m_1 \cdot b) \cdot (V + m_2 \cdot b)} \quad \text{Eq. 4.369}$$

Where:

P	is the pressure of the fluid
V	is the total volume of the container containing the fluid
a	is a measure of the attraction between particles
b	is the volume excluded from V by a particle
n	is the number of moles
T	is the temperature
R	is the gas constant
m_1, m_2	constants: for the Peng-Robinson EOS $(m_1, m_2) = (1 + \sqrt{2}, 1 - \sqrt{2})$ for the Soave—Redlich—Kwong EOS $(m_1, m_2) = (1, 0)$

The EoS is a cubic equation for the volume, as a function of the pressure, temperature and EoS parameters. It is often written in terms of the compressibility:

$$Z = \frac{PV}{nRT} \quad \text{Eq. 4.370}$$

In the special case $(m_1, m_2) = (0, 0)$ the cubic EoS reduces to van der Waals equation, and in the special case when $a = b = 0$ the cubic EoS reduces to the ideal gas equation.

The parameters a and b are in fact functions of the pressure, temperature, composition, component properties and the mixing rules. If there is more than one phase present, the composition of each phase differs and hence each phase has different equation of state parameters. Assuming a quadratic mixing rule for a and a linear mixing rule for b the parameters for phase φ are given by

$$a_{\varphi} = \frac{n^2 \cdot R^2 \cdot T^2}{P} \cdot \sum \sum (A_i \cdot A_j)^{1/2} \cdot (1 - \delta_{ij}) \cdot x_{\varphi i} \cdot x_{\varphi j} \quad \text{Eq. 4.371}$$

$$b_{\varphi} = \frac{n \cdot R \cdot T}{P} \cdot \sum B_i \cdot x_{\varphi i} \quad \text{Eq. 4.372}$$

Where:

$x_{\varphi i}$	is the mole fraction of component i in phase φ
A_i	is a function of the temperature T , the component critical pressure P_{ci} , critical temperature T_{ci} and acentric factor ω_i
B_i	is a function of the component critical pressure P_{ci} and critical temperature T_{ci}
δ_{ij}	is the Binary Interaction Parameter (p.147) between component i and component j

Thermodynamic properties

Thermodynamic properties can be calculated from the equation of state. The method may vary between flash packages. The following equations are used in E300 flash.

The fugacity coefficient $\Phi_{\varphi i}$ for each component in each phase is used to determine the phase state and phase split. It is given by:

$$\ln \Phi_{\varphi i} = \frac{-1}{RT} \int_V^\infty \left[\frac{RT}{V} - \left(\frac{\partial P}{\partial n_i} \right)_{T, V, n_j} \right] dV - \ln \frac{PV}{RT} \quad \text{Eq. 4.373}$$

The phase density can be found from the phase volume. For a two parameter Equation of State, this is found by solving the cubic equation. However, this can give poor prediction of the liquid density. For a three parameter Equation of State, the phase volume is modified by subtracting a volume shift term:

$$V_{\varphi} = V_{\varphi}^{eos} - \sum x_{\varphi i} \cdot V_{si} \quad \text{Eq. 4.374}$$

The phase enthalpy H_{φ} is calculated from the ideal gas enthalpy: H_{φ}^o :

$$H_{\varphi} = H_{\varphi}^o - \int_V^\infty \left[T \left(\frac{\partial P}{\partial T} \right)_V - P \right] dV - RT + PV \quad \text{Eq. 4.375}$$

The phase entropy S_{φ} is calculated from the ideal gas entropy: S_{φ}^o :

$$S_{\varphi} = S_{\varphi}^o - \int_V^\infty \left[\left(\frac{\partial P}{\partial T} \right)_V - \frac{R}{V} \right] dV + R \cdot \ln \frac{PV}{RT} \quad \text{Eq. 4.376}$$

The ideal gas enthalpy and entropy are determined from the ideal gas specific heat $C_{p\varphi}^o$:

$$H_{\varphi}^o = \int_{T_{ref}}^T C_{p\varphi}^o dT \quad \text{Eq. 4.377}$$

$$S_{\varphi}^o = \int_{T_{ref}}^T \frac{C_{p\varphi}^o}{T} dT - R \cdot \ln \frac{P}{P_{ref}} - R \cdot \sum (x_{\varphi i} \cdot \ln x_{\varphi i}) \quad \text{Eq. 4.378}$$

The ideal gas specific heat is calculated by summing the component specific heats

$$C_{p\varphi}^o = \sum C_{pi}^o \cdot x_{\varphi i} \quad \text{Eq. 4.379}$$

For library components, the component specific heat C_{pi}^o is a known function of temperature. For user defined petroleum fractions, the component specific heat is calculated as a function of temperature and the component molecular weight MW_i , boiling point temperature T_{Bi} , specific gravity γ_i and acentric factor ω_i .

E300 Flash

The ECLIPSE version of the Peng Robinson EoS has an option correction for the A_i term for large acentric factors (ECLIPSE PRCORR keyword).

E300 flash name	Peneloux Volume Shift Correction	Peng Robinson 1978 Acentric Factor Correction
Peng-Robinson		
Peng-Robinson		Yes
Peng-Robinson	Yes	
Peng-Robinson	Yes	Yes
SRK	Yes	
SRK		

Multiflash

The Multiflash Implementation in PIPESIM has the cubic Equations of State, Peng-Robinson and RKS, along with the Cubic Plus Association (CPA) model, which is an extension of the RKS (advanced) cubic EoS to handle polar and hydrogen bonding components. The Multiflash implementation also includes [non-cubic EoS \(p.539\)](#).

EoS names differ from those in the Multiflash GUI.

PIPESIM GUI name	Multiflash GUI name	Peneloux Volume Shift Correction	Peng Robinson 1978 Acentric Factor Correction
Peng-Robinson	PR (Advanced)	Yes	
NOT CURRENTLY AVAILABLE	PR78 (Advanced)		Yes

SRK	RKS (Advanced)	Yes	
Association (CPA)	Association (CPA-Infochem)	Yes	

Versions of the Peng-Robinson and SRK equations of state without the volume shift correction are available, but are not recommended. Liquid densities predicted by these equations of state can be poor. In particular the liquid water density is out by about 15%. This causes problems in PIPESIM, since it can predict water being lighter than oil. This particular problem does not arise in two phase flashes, where the water properties are not determined by the equation of state. It does occur in 3 phase flashes, such as Multiflash.

PIPESIM GUI name	Multiflash GUI name	Volume Shift Correction	Peng Robinson 1978 Acentric Factor Correction
Standard Peng-Robinson	PR		
NOT CURRENTLY AVAILABLE	PR78		Yes
Standard SRK	RKS		

DBR flash

The DBR flash has both the Peng Robinson and the Soave—Redlich—Kwong equations of state, with both two and three parameter (volume shift) options:

DBR flash	Volume Shift Correction
Peng-Robinson	
Peng-Robinson	Yes
SRK	
SRK	Yes

Non-cubic Equations of State

Equations of state (EoS) describe the pressure, volume and temperature (PVT) behavior of pure components and mixtures. The phase state and most thermodynamic properties (e.g. density, enthalpy, entropy) are derived from the equation of state. Separate models are used for transport properties, such as viscosity, conductivity and surface tension.

PIPESIM can use both [cubic \(p.536\)](#) and non-cubic and Equations of State.

Multiflash

BWRS

The BWRS is an 11-parameter non-cubic equation. The BWRS equation gives much more accurate volumetric and thermal property predictions for light gases and hydrocarbons. It should give reasonable vapor-liquid phase equilibrium predictions, but owing to its complexity, it requires more computing time than the cubic EOS (e.g. SRK or Peng-Robinson). The EoS is similar to a virial expansion in density:

$$P = \frac{RT}{V} \cdot \left[n + \frac{B}{V} + \frac{C}{V^2} + \frac{D}{V^5} + \frac{C'}{V^5} \cdot \left(1 + \frac{\gamma^2}{V^2} \right) \cdot \exp \left(\frac{-\gamma^2}{V^2} \right) \right] \quad \text{Eq. 4.380}$$

The BWRS EoS can be used with most of the components that can be used with the cubic EoS. ([p.544](#))

Note: It does not work with the Hydrogen component or with aqueous components.

CSMA

CSMA is the Multiflash multi-reference fluid corresponding states model. The CSMA model is based on a collection of very accurate equations of state for a number of common substances. The density, thermal properties and VLE of each substance are generally reproduced to within the accuracy of experimental measurements. The properties of mixtures can be estimated from a model that reduces to the (accurate) pure component values as the mixture composition approaches each pure component limit.

An important application is mixtures containing CO₂, H₂S and light hydrocarbons. It can only be used with a limited selection of components ([p.549](#)).

Note: It can only be used via an MFL file.

CPA

The CPA (cubic-plus-association) model extends the capabilities of industry-standard cubic equations of state to polar and hydrogen-bonding components. It is applicable to a wide variety of systems of importance to the upstream oil industry such as hydrocarbons, gases, water and hydrate inhibitors (alcohols and glycols).

The Multiflash CPA model is based on the Infochem RKSA (advanced Redlich-Kwong-Soave) equation of state. It has the advantage for non-polar substances, because it reduces the RKSA eos so that all the characterization methods and parameters for standard oil and gas mixtures can be used. Extra terms in the equation describe polar and associating compounds. The main application in Multiflash is representing the fluid phases when modeling hydrates and hydrate inhibition. CPA shows improvements over standard cubic eos for other systems such as acid gases and water.

Note: Salt components are not supported.

The CPA model is the subject of an active research program that is extending its applicability to many other systems of industrial importance.

Reference Fluid Thermodynamic and Transport Properties — REFPROP

REFPROP is an acronym for REReference fluid PROProperties. The flash package provides thermodynamic and transport properties of industrially important fluids and their mixtures with an emphasis on refrigerants and hydrocarbons, especially natural gas systems. It is developed by the National Institute of Standards and Technology (NIST).

REFPROP is based on highly accurate single component and mixture models based on the Helmholtz energy.

NIST recommendation for pure fluids / mixture

For single components, REFPROP has a recommended set of equations of state explicit in Helmholtz energy. That is, for each single component, a specific equation of state explicit in Helmholtz energy is chosen. e.g. for carbon dioxide this is [Span and Wagner \(1996\) \(p.593\)](#). Mixture calculations employ a model that applies mixing rules to the Helmholtz energy of the mixture components; it uses a departure function to account for the departure from ideal mixing.

GERG

Introduction

GERG is an acronym for Groupe Européen de Recherches Gazières, which is supported by the European natural-gas companies. The European natural-gas companies include E.ON Ruhrgas (Germany), Enagás (Spain), Gasunie (The Netherlands), Gaz de France (France), Snam Rete Gas (Italy) and Statoil (Norway).

The flash package, developed at Ruhr-Universität Bochum, provides thermodynamic and transport properties of industrially important gases and other mixtures with an emphasis on hydrocarbons and further components. Lehrstuhl fuer Thermodynamik (Department of Thermodynamics) of Ruhr-Universität Bochum, Germany have developed a wide-range equation of state (EOS) for natural gases and other mixtures that meets the requirements of standard and advanced natural gas applications.

The first published equation of state by Ruhr-Universität Bochum covers mixtures consisting of up to 18 components as listed below:

Methane	n-Butane	Hydrogen
Nitrogen	Isobutane	Carbon Monoxide
Carbon Dioxide	n-Pentane	Water
Ethane	Isopentane	Oxygen
Propane	n-Hexane	Argon
	n-Octane	Helium

Annotations:

Yellow – natural gas main components

Red – further hydrocarbons

Blue – further components

In 2004, the new equation of state was evaluated by the GERG group and then adopted under the name GERG-2004 equation of state (or GERG-2004 for short) as an international reference equation of state for natural gases and similar mixtures (GERG standard).

GERG-2008

In 2008, Ruhr-Universität Bochum further extended GERG-2004 by including three additional components *n-nonane*, *n-decane* and *hydrogen sulfide*, making its component list up to 21

components. This expanded equation of state was called GERG-2004 XT08, where "XT08" meant "eXTension 2008". In 2010, upon the request of the ISO Working Group (ISO TC193 SC1 WG13), Ruhr-Universitat Bochum simplified the name of GERG-2004 XT08 to GERG-2008, the current version of GERG.

The GERG-2008 equation of state is under consideration to be adopted as an ISO Standard (ISO 20765-2 and ISO 20765-3) for natural gases. The ISO group ISO TC 193/SC 1/WG 13 is working on this matter.

Description

Structure

The GERG equation of state is based on a multi-fluid approximation, which is explicit in the reduced Helmholtz energy $\alpha = a/(RT)$ [a = Alpha in the figures] dependent on the density ρ , the temperature T and the composition x (mole fractions) of the mixture. The structure of the equations of state is shown in the following figure:

$$\alpha(\delta, \tau, \bar{x}) = \alpha^0(\rho, T, \bar{x}) + \sum_{i=1}^N x_i \alpha_{0i}^r(\delta, \tau) + \Delta\alpha^r(\delta, \tau, \bar{x})$$

Ideal gas part Contribution of the pure substances Departure function

Reduced density and temperature of the mixture
 $\delta = \rho / \rho_r(\bar{x})$ $\tau = T_r(\bar{x}) / T$
 Reducing functions, only dependent on the composition of the mixture

Figure 4.10. The basic structure of the equations of state GERG-2004 ($N = 18$) and GERG-2008 ($N = 21$) for natural gases and other mixtures.

Three elements are necessary to set up a multi-fluid approximation:

- Pure substance equations of state for all components
- Reducing functions for density and temperature
- Departure function

The reducing functions as well as the departure function were developed to describe the behaviour of the mixture and contain substance and mixture specific parameters. From the reducing functions, the reducing values ρ_r and T_r for the density and the temperature of the mixture are calculated. They only depend on the mixture composition and turn into the critical properties p_c and T_c , respectively, for the pure components. The departure function depends on the reduced density δ , the inversely reduced temperature τ ($\tau = \text{Tau}$ in the figures), and the composition x of the mixture. It contains the sum of binary specific and generalized departure functions, which can

be developed for single binary mixtures (binary specific) or for a group of binary mixtures (generalized). The following equation illustrates this summation:

$$\Delta\alpha^r(\delta, \tau, \bar{x}) = \sum_{j=i+1}^N \sum_{i=1}^{N-1} \Delta\alpha_{ij}^r(\delta, \tau, \bar{x})$$

Figure 4.11. The departure function for the mixture in a multi-fluid approximation as a double summation over all binary specific and generalized departure functions developed for the binary subsystems; GERG-2004: N = 18; GERG-2008: N = 21.

The mathematical structure of the part of the binary specific and generalized departure functions that depends on δ and τ is similar to the structure of pure substance equations of state and is determined by our method for optimizing the structure of equations of state. Furthermore, the departure functions contain a factor that only depends on the composition of the mixture. For further details, see the references given at the end of this description.

In order to obtain a reference equation of state that yields accurate results for various types of natural gases and other multi-component mixtures over wide ranges of composition, the reducing and departure functions were developed using only data for binary mixtures. The 18 pure components covered by GERG-2004 form 153 different binary mixtures, and the 21 pure components covered by GERG-2008 result in 210 possible binary mixture combinations. Departure functions $\Delta\alpha_{ij}^r(\delta, \tau, x)$ were developed only for such binary mixtures for which accurate experimental data existed. For binary mixtures with limited or poor data, no departure functions were developed, and only the parameters of the reducing functions $p_r(x)$ and $T_r(x)$ were fitted; in case of very poor data, simplified reducing functions without any fitting were used.

The multi-fluid approximation used enables a simple inclusion of additional components in future developments. This means that, for example, fitted parameters of the existing equation of state do not have to be refitted when incorporating new components. This also holds for the departure function with its optimized structure, which remains unchanged when expanding the model.

Range of Validity and Accuracy

The entire range of validity of GERG-2008 covers the following temperatures and pressures:

- Normal range: $90 \text{ K} \leq T \leq 450 \text{ K}$, $p \leq 35 \text{ MPa}$
- Extended range: $60 \text{ K} \leq T \leq 700 \text{ K}$, $p \leq 70 \text{ MPa}$

Moreover, the equation can be reasonably extrapolated beyond the extended range, and each component can basically cover the entire composition range, i.e. (0-100)%.

GERG-2008 represents most of the experimental data, including the most accurate measurements available, to within their uncertainties. The uncertainty values given in the following correspond to the uncertainties of the most accurate experimental data.

In the gas region, the uncertainties in density and speed of sound are 0.1%, in enthalpy differences (0.2-0.5)% and in heat capacities (1-2)%. In the liquid region, the uncertainty in density is (0.1-0.5)%, in enthalpy differences (0.5-1)% and in heat capacities (1-2)%. In the two-phase region, vapour pressures are calculated with a total uncertainty of (1-3)%, which corresponds to the uncertainties of the experimental VLE data. For mixtures with limited or poor data, the uncertainty values stated above can be somewhat higher.

These accuracy statements are based on the fact that GERG-2008 represents the corresponding experimental data to within their experimental uncertainties (with very few exceptions).

References

The comprehensive descriptions of GERG (with the entire numerical information, experimental data used, quality, range of validity, etc.) are retrievable from the following reference:

Kunz, O., Klimeck, R., Wagner, W., Jaeschke, M. The GERG-2004 wide-range equation of state for natural gases and other mixtures. GERG TM15 2007. Fortschr.-Ber. VDI, Reihe 6, Nr. 557, VDI Verlag, Düsseldorf, 2007; also available as GERG Technical Monograph 15 (2007).

Kunz, O., Wagner, W. The GERG-2008 wide-range equation of state for natural gases and other mixtures: An expansion of GERG-2004. To be submitted to J. Chem. Eng. Data (2011).

Note: This GERG Monograph is available for downloaded from the website of GERG - <http://www.gerg.eu/publications/tm.htm>

Components for Cubic Equations of State

For the [cubic equations of state \(p.536\)](#), non-aqueous components can either be selected from a [library \(p.544\)](#), or user-defined components ([petroleum fractions \(p.549\)](#)) can be selected by defining their properties.

For two phase flashes, water can be selected, but it is treated differently from the other components. It is removed from the flash calculations, and the water phase properties are calculated separately. For three phase flashes (Multiflash), [aqueous components \(p.548\)](#) can be selected from the component library. User-defined aqueous components are not allowed.

Non-aqueous library components

Different library components are available for each package. When converting from one package to another, if a library component is not available in the new package, it will be converted into a [petroleum fraction \(p.549\)](#). Default molecular weight and boiling point temperature are used to define the petroleum fraction — data for pure components are taken from [Poling et al \(p.591\)](#).

The non-aqueous library components can be divided into three types

1. [Pure hydrocarbon components \(p.544\)](#)
2. [Non-hydrocarbon components \(p.546\)](#)
3. [Pseudo-components \(p.547\)](#)

Pure hydrocarbon components

The following pure library components can be selected:

Formula	Multiflash	E300 flash	DBR 2-phase flash	MW (g/gmol)	Tbp (K)
CH4	Methane	C1	C1	16.043	111.66
C2H4	Ethylene		Ethylene	28.054	169.42
C2H6	Ethane	C2	C2	30.070	184.55
C3H4			Propadiene	40.065	238.77

C3H6			Propylene	42.081	225.46
C3H6			Cyclo-C3	42.081	240.34
C3H8	Propane	C3	C3	44.097	231.02
C4H6			1,3-Butadiene	54.092	268.62
C4H6			1,2-Butadiene	54.092	269.00
C4H8			iso-Butene	56.108	266.24
C4H8			1-Butene	56.108	266.92
C4H8			tr2-Butene	56.108	274.03
C4H8			cis2-Butene	56.108	276.87
C4H10	Isobutane	IC4	i-C4	58.123	261.34
C4H10	N-Butane	NC4	n-C4	58.123	272.66
C5H10			1-Pentene	70.134	322.38
C5H10	Cyclopentane		Cyclo-C5	70.134	303.11
C5H12	2,2-Dimethylpropane			72.150	282.65
C5H12	Isopentane		i-C5	72.150	300.99
C5H12	N-Pentane		n-C5	72.150	309.22
C6H6	Benzene	BEN	Benzene	78.114	353.24
C6H12			1-Hexene	84.161	336.63
C6H12	Methylcyclopentane		Mcyclo-C5	84.161	344.98
C6H12	Cyclohexane		Cyclo-C6	84.161	353.93
C6H14	N-Hexane		n-C6	86.177	341.88
C7H8	Toluene	TOL	Toluene	92.141	383.79
C7H14			1-Heptene	98.188	366.79
C7H14	Methylcyclohexane		Mcyclo-C6	98.188	374.09
C7H16	3-Methyl Hexane			100.204	365.00
C7H16	N-Heptane		n-C7	100.204	371.57
C7H16	3-Methyl Hexane			100.204	365.00
C8H10	Ethylbenzene		C2-Benzene	106.167	409.36
C8H10	P-Xylene		p-Xylene	106.167	411.53
C8H10	M-Xylene		m-Xylene	106.167	412.34
C8H10	O-Xylene		o-Xylene	106.167	417.59
C8H16			1-Octene	112.215	394.44
C8H16	Ethylcyclohexane			112.215	404.00
C8H18	N-Octane		n-C8	114.231	398.82
C9H12			124-MBenzene	120.194	442.49

C9H12	Cumene			120.194	425.52
C9H20	N-Nonane		n-C9	128.258	423.97
C10H14	1,2-Diethylbenzene			134.22	456
C10H22	N-Decane		n-C10	142.285	447.30
C11H24	N-Undecane			156.312	469.08
C12H26	N-Dodecane			170.338	489.48
C13H28	N-Tridecane			184.365	508.63
C14H10			Phenanthrene	178.233	611.55
C14H30	N-Tetradecane			198.392	526.76
C15H32	N-Pentadecane			212.419	543.83
C16H34	N-Hexadecane			226.446	559.98
C17H36	N-Heptadecane			240.473	574.56
C18H38	N-Octadecane			254.500	588.30
C19H40	N-Nonadecane			268.527	602.34
C20H42	N-Eicosane			282.554	616.84
C21H44	N-Heneicosane				
C22H46	N-Docosane				
C23H48	N-Tricosane				
C24H50	N-Tetracosane				
C25H52	N-Pentacosane				
C26H54	N-Hexacosane				
C28H58	N-Octacosane				
C29H60	N-Nonacosane				
C30H62	N-Triacontane				
C32H66	N-Dotriacontan				
C36H74	N-Hexatriacontane				

Non-hydrocarbons

The following pure library components can be selected:

Formula	Multiflash	E300 flash	DBR 2-phase flash	MW (g/gmol)	Tbp (K)
H2	Hydrogen	H2	H2	2.016	20.38
He	Helium		Helium	4.003	4.30
N2	Nitrogen	N2	N2	28.014	77.35
O2	Oxygen		O2	31.999	90.17
Ar	Argon		Argon	39.948	87.27

Kr			Krypton	83.800	119.74
Xe			Xenon	131.290	165.01
NH3	Ammonia		NH3	17.031	239.81
H2S	Hydrogen Sulphide	H2S	H2S	34.082	212.84
CO	Carbon Monoxide	CO	CO	28.010	81.66
CO2	Carbon Dioxide	CO2	CO2	44.010	
SF6			SF6	146.06	209.00

Pseudo-hydrocarbon components

Some packages contain pseudo components, essentially pre-defined [petroleum fractions](#) ([p.549](#)), that can be used to represent the heavy end of the oil.

Formula	Multiflash	E300 flash	DBR 2-phase flash	MW (g/gmol)	Tbp (K)
		C4		58.124	268.9
		C5		72.151	305.9
		C6	C6	84.00	341.9
		C7	C7	96.00	371.6
C8H10			m&p-Xylene	106.167	411.9
		C8	C8	107.00	398.8
		C9	C9	121.00	424.0
		C10	C10	134.00	447.0
		C11	C11	147.00	469.0
		C12	C12	161.00	489.0
		C13	C13	175.00	508.0
		C14	C14	190.00	527.0
		C15	C15	206.00	544.0
		C16	C16	222.0	560.0
		C17	C17	237.00	575.0
		C18	C18	251.00	589.0
		C19	C19	263.00	603.0
		C20	C20		
		C21	C21		
		C22	C22		
		C23	C23		
		C24	C24		
		C25	C25		

		C26	C26		
		C27	C27		
		C28	C28		
		C29	C29		
		C30	C30		
		C31	C31		
		C32	C32		
		C33	C33		
		C34	C34		
		C35			
		C36			
		C37			
		C38			
		C39			
		C40			
		C41			
		C42			
		C43			
		C44			
		C45			
		Bitumen	544.00	773.2	

Aqueous library components

Aqueous components are defined as those that will distribute mainly in the second liquid phase. These are water and the hydrate suppressants methanol, ethylene glycol, diethylene glycol and triethylene glycol. The calculation in of the amount of aqueous components to add corresponds to an agreed definition. The water phase is added as a proportion of the DRY gas at stock tank conditions (15 degrees C, 1bar). The discrepancy between hand calculations and software is because the software makes a correction for the amount of aqueous components that will partition to the gas phase. That is, it aims to add the amount of aqueous components requested as a separate aqueous phase. There will also be some loss to the hydrocarbon liquid phase but this will not be significant unless the aqueous phase contains a lot of methanol. The amount lost to the vapor phase will be significant if there is a large amount of gas present relative to other phases.

Formula	Multiflash
H ₂ O	Water
CH ₃ -OH	Methanol
C ₂ H ₅ -OH	Ethanol

C2H6O2	Ethylene Glycol (MEG)
C4H10O3	Diethylene Glycol (DEG)
C6H14O4	Triethylene Glycol (TEG)
NaCl	Salt Component

Petroleum fractions

User-defined components can be created by defining key properties: molecular weight MW_i , critical pressure P_{ci} , critical temperature T_{ci} , boiling point temperature T_{Bi} , specific gravity γ_i and acentric factor ω_i . It is not always necessary to supply values for all these properties — the flash packages can use correlations to determine unspecified properties from those that are specified. Minimum data required is shown in the table below:

	Multiflash	E300 flash / DBR flash
MW_i, γ_i	yes	
MW_i, T_{Bi}	yes	
T_{Bi}, γ_i	yes	
MW_i		yes
P_{ci}, T_{ci}, ω_i	yes	

Note: The Multiflash routines that calculate the petroleum fraction properties assume the molecular weight of petroleum fractions exceed 72.

Components for Non-Cubic Equations of State

Only library components can be used for the CSMA, NIST recommendation for pure fluid/mixture and GERG-2008 [non-cubic equations of states \(p.539\)](#).

Formula	Multiflash (CSMA)	REFPROP (NIST recommendation for pure fluid/mixture)	GERG-2008 (GERG-2008)
CH4	Methane	methane	Methane
C2H6	Ethane	ethane	Ethane
C3H8	Propane	propane	Propane
i-C4H10	Isobutane	isobutane	Isobutane
n-C4H10	Butane	butane	n-Butane
i-C5H12	Isopentane	isopentane	Isopentane
n-C5H12	Pentane	pentane	n-Pentane

n-C6H14	Hexane	hexane	n-Hexane
n-C7H16	Heptane	heptane	n-Heptane
n-C8H18	Octane	octane	n-Octane
n-C9H20		nonane	n-Nonane
n-C10H22		decane	n-Decane
C2H2	Ethylene		
C6H12	Cyclohexane		
C7H8	Toluene		
H2	Hydrogen	hydrogen (normal)	Hydrogen
He	Helium	helium	Helium
N2	Nitrogen	nitrogen	Nitrogen
O2	Oxygen	oxygen	Oxygen
Ar	Argon	argon	Argon
H2S	Hydrogen Sulphide	hydrogen sulfide	Hydrogen sulphide
CO	Carbon Monoxide	carbon monoxide	Carbon monoxide
CO2	Carbon Dioxide	carbon dioxide	Carbon dioxide
NH3	Ammonia		

Multiflash is a 3-phase flash and allows aqueous components. REFPROP and GERG-2008 allow water, but are two phase flashes. The water component is therefore removed before the flash calculation, and water properties are calculated outside the flash.

Formula	Multiflash (CSMA)	NIST-REFPROP	GERG-2008
H2O	Water	water	Water
C2H5-OH	Ethanol		

Viscosity Models for Compositional Fluids

Select **Setup** » **Compositional**, then on the **Property Models** tab, select one of the following models for determining viscosity:

- Pedersen (the default)
- LBC (Lohrenz-Bray-Clark)
- Aasberg-Petersen (only available for E300 and DBR)
- NIST recommendation for pure fluids / mixture (only available for REFPROP and GERG-2008)

Preliminary testing has shown the Pedersen method to be the most widely applicable and accurate for oil and gas viscosity predictions. Multiflash uses the Pedersen method as the default viscosity model, though an option is available to choose the LBC model for backward compatibility. The Pedersen method has been adopted as the default in response to the deficiencies of the LBC method. The Pedersen method is based on the corresponding state theory, as is the LBC method.

The results for different components are as follows:

Lower Alkanes

Predicted liquid viscosities using LBC and Pedersen methods have been compared to experimental data for Methane and Octane as a function of both temperature and pressure and for Pentane as a function of temperature. For both Methane and Pentane the Pedersen method predictions show close agreement with experimental data. For Octane, the Pedersen and LBC methods give comparable results. For the aromatic compound, Ethyl Benzene, the Pedersen method is not as good as LBC.

Higher Alkanes

The results for higher alkanes Eicosane and Triacontane are mixed: the Pedersen method is adequate for Eicosane whereas LBC is slightly better than Pedersen for Triacontane. For Triacontane both the LBC and Pedersen methods are inadequate. However, in the majority of cases the higher hydrocarbons should be treated as petroleum fractions rather than as single named components.

Petroleum Fractions

The LBC method describes viscosity as a function of the fluid critical parameters, acentric factor and density. The LBC model is therefore very sensitive to both density and the characterization of the petroleum fractions.

Water

The Pedersen method suffers the same drawback as LBC in that it is unable to predict the temperature dependence of water, a polar molecule. To overcome this problem, the Pedersen method has been modified especially for water so that it now accurately models the viscosity of water in the liquid phase. This was achieved by the introduction of a temperature-dependent correction factor. However the prediction of the viscosity of the gas phase is also affected, though in only a minor way.

Methanol

Neither the LBC nor Pedersen method can deal with polar components, with the Pedersen method slightly worse than LBC. This is not surprising, as both methods were developed for non-polar components and mixtures. The Pedersen method works best with light alkanes and petroleum mixtures in the liquid phase. It performs as well or better than the LBC method in nearly all situations.

The choice of the equation of state has a large effect on the viscosities predicted by both methods. The LBC method is more sensitive to the equation of state effects than is the Pedersen method.

See also: [Package \(p.142\)](#), [Binary Interaction Parameters \(p.147\)](#), [Emulsions \(p.555\)](#), [Equation of State \(p.536\)](#)

Solid Precipitation

Asphaltene Prediction

ONLY AVAILABLE WITH MFL FILES

Asphaltene formation line is displayed automatically on the phase envelope to enable the determination of the conditions (temperature and pressure) at which asphaltene appears

Hydrates

Only available with Multiflash.

Requires additional licensing option.

Hydrate lines are displayed automatically on the [phase envelope \(p.149\)](#) if water is in the component list and hydrates will form. The amount of water may influence the results of the calculations, particularly when inhibitors or water-soluble gases are present.

Background

Natural gas hydrates are solid ice-like compounds of water and light components of natural gas. The phase behavior of the systems involving hydrates can be very complex because up to six phases must normally be considered. The behavior is particularly complex if there is significant mutual solubility between phases.

The Multiflash hydrate model uses a modification of the SRK equation of state for the fluid phases plus the van der Waals and Platteeuw model for the hydrate phases. The model can explicitly represent all the effects of the presence of inhibitors.

Hydrate Inhibitors

Hydrate inhibitors decrease the hydrate formation temperature or increase the hydrate formation pressure in a given gas mixture. The model includes parameters for the commonly used inhibitors such as Methanol, and the glycols MEG, DEG and TEG.

A new mixing rule has been developed for the SRK equation of state to model the inhibitors' effects on the fluid phases. The treatment of hydrate inhibition has the following features. The model can represent explicitly all the effects of inhibitors, including the depression of hydrate formation temperature, the depression of the freezing point of water, the reduction in the vapor pressure of water (i.e. the dehydrating effect) and the partitioning of water and inhibitor into the oil, gas and aqueous phases. The model has been developed using all available data for mixtures of water with Methanol, MEG, DEG and TEG. This involves simultaneously representing hydrate dissociation temperatures, depression of freezing point data and vapor-liquid equilibrium data. The solubilities of hydrocarbons and light gases in water/inhibitor mixtures have also been represented. There is no fundamental difference between calculations with and without inhibitors. To investigate the effect of an inhibitor it must be added to the list of components in the mixture and the amount must be specified just as for any other component. It is not possible to specify the amount of inhibitor in a particular phase, only the total amount in the mixture. This is because the inhibitor will be split among the different phases present at equilibrium with the amount in a particular phase depending on the ambient conditions and the amounts of other components present in that phase. This is exactly what happens in reality. The amount of inhibitor typically needed would be approximately 35% by mass of inhibitor relative to water.

Model features

The main features of the model are:

- The description of the hydrate phase behavior uses a thermodynamically consistent set of models for all phases.
- The vapor pressures of pure water are reproduced. The following natural gas hydrate formers are included: METHANE, ETHANE, PROPANE, ISOBUTANE, BUTANE, NITROGEN, CO₂ AND H₂S.

- The thermal properties (enthalpies and entropies) of the hydrates are included, permitting flashes involving these phases.
- The properties of the hydrates have been fixed by investigating data for natural gas components in both simple and mixed hydrates to obtain reliable predictions of both structure I and structure II hydrates.
- The properties of the empty hydrate lattices have been investigated and the most reliable recent values have been adopted.
- Proper allowance has been made for the solubilities of the gases in water so that the model parameters are not distorted by this effect. This is particularly important for Carbon Dioxide and Hydrogen Sulphide, which are relatively soluble in water.
- Correct thermodynamic calculations of the most stable hydrate structure have been made. The model has been tested on a wide selection of open literature and proprietary experimental data. In most cases the hydrate dissociation temperature is predicted to within 1 degree Kelvin.

To ensure that reliable results are obtained, it is particularly important to use the correct set of models and phase descriptors. The hydrates model set contains a complete set of model and phase specifications.

Hydrate Model Details

The hydrate model set has the following definitions:

FLUID PHASE MODEL

The recommended fluid phase model is the advanced SRK equation of state with the a parameter fitted to the pure component vapor pressure, the Peneloux density correction and the INFOCHEM mixing rule. The required binary interaction parameters (BIP) for hydrocarbons, light gases, water and inhibitors are available from the OILGAS2 BIP data set.

HYDRATE MODELS

The hydrate model consists of lattice parameters for the empty hydrate and parameters interaction of gas molecules with water in the hydrate. There are different parameter values for each hydrate structure, HYDRATE I and II. In addition, the hydrate must be associated with a liquid phase model that is used to obtain the properties of water. It is important that this is the same model that is used for water as a fluid phase.

PHASES

In most cases, six phase descriptors are required: gas, hydrocarbon liquid, aqueous liquid, hydrate I, hydrate II and ice. At high pressures and/or low temperatures the 'gas' phase may become liquid-like and a second non-aqueous liquid PD is needed. This is also the case if there is a significant amount of CO₂ or H₂S present. In most practical cases, the gas contains propane and has a hydrate II stable hydrate structure. Key components are defined to distinguish between the hydrocarbon and aqueous liquid phases. In principle, hydrate calculations and phase envelope plotting with Multiflash are no different from flash calculations and envelope plotting for the fluid phases alone. Multiflash treats fluid and solid phases the same. It can carry out a full range of flashes for streams with hydrates.

Ice Prediction

Only available with Multiflash Flash.

Ice is treated as a pure solid phase. The ice formation line is displayed automatically on the phase envelope if water is in the component list and ice will form.

Wax Prediction

ONLY AVAILABLE WITH MFL FILES

Waxes are complex mixtures of solid hydrocarbons that freeze (solidify) out of crude oils if the temperature is low enough - below the critical wax deposition temperature. They are mainly formed from normal paraffins or if isoparaffins and naphthenes are present. The wax formation line is displayed automatically on the phase envelope to enable the determination of the conditions (temperature and pressure) at which wax could deposit.

[Wax deposition \(p.205\)](#) can also be modelled.

4.5.3 Fluid Property Table Files

Fluid properties can be pre-calculated for a range of pressure and temperature values and stored in a table. PIPESIM reads the table and can interpolate it to get properties for any pressure and temperature. Tables in a format that can be read by PIPESIM can be generated by a range of Compositional Fluid Packages, including PIPESIM itself. Table files representing different fluids cannot be mixed. They are therefore useful in single branch models, but less so in network models, unless the entire network contains only a single fluid.

The tables contain liquid and gas properties:

Property		
Pressure	<i>psia</i>	<i>kPa</i>
Temperature	<i>F</i>	<i>K</i>
Liquid Volume Fraction	%	%
Water cut	%	%
Liquid Density	<i>lb / ft³</i>	<i>kg / m³</i>
Gas Density	<i>lb / ft³</i>	<i>kg / m³</i>
Gas Compressibility		
Gas Molecular Weight	<i>lb / lbmol</i>	<i>kg / kgmol</i>
Liquid Viscosity	<i>cP</i>	<i>cP</i>
Gas Viscosity	<i>cP</i>	<i>cP</i>
Total Enthalpy	<i>BTU / lbmol</i>	<i>kJ / kgmol</i>
Total Entropy	<i>BTU / (lbmol · F)</i>	<i>kJ / (kgmol · K)</i>
Liquid Heat Capacity	<i>BTU / (lbmol · F)</i>	<i>kJ / (kgmol · K)</i>
Gas Heat Capacity	<i>BTU / (lbmol · F)</i>	<i>kJ / (kgmol · K)</i>
Liquid Surface Tension	<i>dyne / cm</i>	<i>dyne / cm</i>

Gas Thermal Conductivity	$BTU/(hr \cdot ft \cdot F)$	$W/(m \cdot K)$
Oil Thermal Conductivity	$BTU/(hr \cdot ft \cdot F)$	$W/(m \cdot K)$
Water Thermal Conductivity	$BTU/(hr \cdot ft \cdot F)$	$W/(m \cdot K)$

The number of phases and the phase volume fractions, can be determined from the liquid fraction and water cut. However, since only liquid and gas properties are available, these tables are only suitable for use with two-phase flow models.

The table files also contain the total molecular weight of the fluid, which is independent of the pressure and temperature. This allows PIPESIM to calculate the liquid molecular weight from the other table properties. The total molecular weight, liquid molecular weight and gas molecular weight are then used to convert the molar quantities (enthalpy, entropy and heat capacities) to mass based quantities.

Table files may also contain compositional data. In this case the table data may be ignored and PIPESIM may use normal compositional flashing.

Internal fluid property tables

PIPESIM can also use property tables to store fluid data in compositional runs. Properties are interpolated from the table properties and as the simulation progresses, tables values are filled in when necessary. This can speed up compositional simulations, although it requires more memory to store the data. These tables can be used in network simulations with multiple fluids — a new mixture can be created by mixing the fluid components and a new internal table created for the new fluid.

4.5.4 Liquid mixture properties

Two phase flow correlations model flow of a liquid and a gas phase. When there are two liquid phases, such as oil and water, the two liquid phases need to be combined and modelled as a single liquid phase. The oil and water will flow at the same velocity. The liquid density can be simply averaged. However more complicated models are used for [liquid viscosity \(p.555\)](#) and [liquid-gas surface tension \(p.560\)](#).

Liquid Viscosity and Oil/Water Emulsions

An **emulsion** is a mixture of two immiscible liquid phases. One phase (the dispersed phase) is carried as droplets in the other (the continuous phase). In Oil/Water systems at low watercuts, oil is usually the continuous phase. As watercut is increased there comes a point where **phase inversion** occurs, and water becomes the continuous phase. This is the **Critical Watercut of Phase Inversion**, otherwise called the **cutoff**. It occurs typically between 55% and 70% watercut. The viscosity of the mixture is usually highest at and just below the cutoff. Emulsion viscosities can be many times higher than the viscosity of either phase alone.

Correlations and methods

The methods available for calculating the oil/water mixture viscosity are:

- Inversion
- Volume Ratio

- User-supplied Table

In addition a number of emulsion correlations are available:

- Woelflin
- Brinkman
- Vand
- Richardson
- Leviton and Leighton

The cutoff can be entered as a watercut, or calculated using the Brauner and Ullman correlation. The cutoff is used by all the emulsion correlations and methods, except for the Volume ratio method.

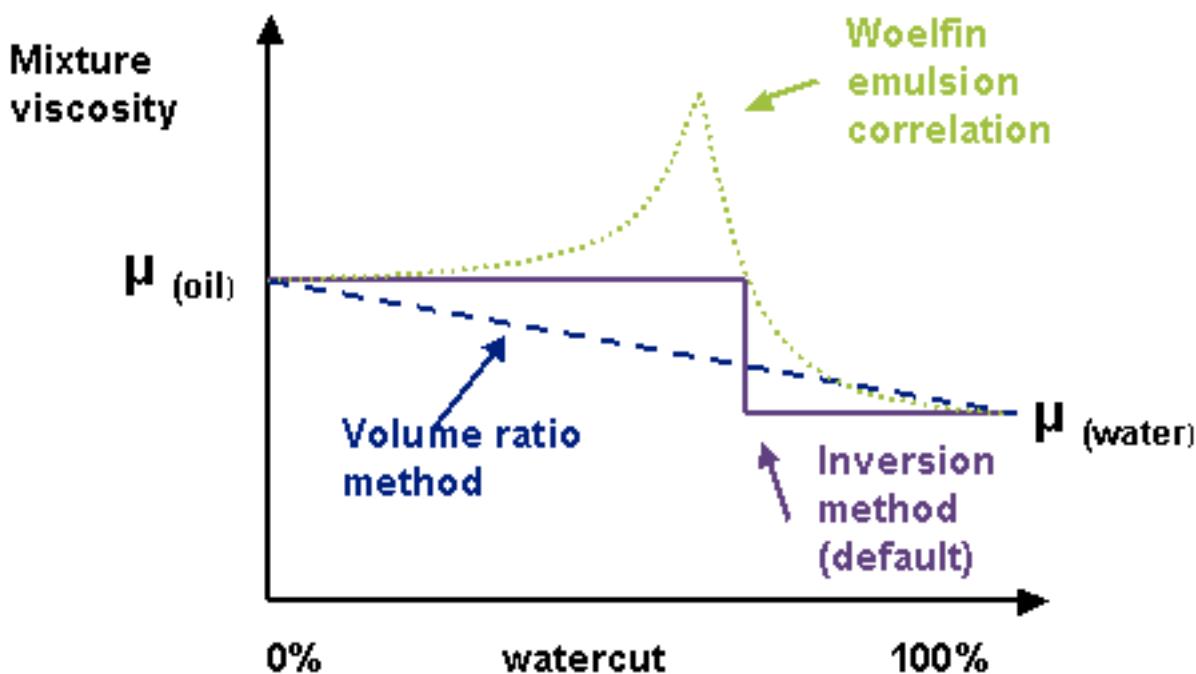


Figure 4.12. Viscosity of oil/water mixtures

Inversion

The inversion method sets the liquid viscosity to the viscosity of the continuous phase. This means that, at a watercut below or equal to the cutoff, water droplets are carried by a continuous oil phase, and the mixture assumes the viscosity of the oil. At a watercut above the cut-off value, oil droplets are carried by a continuous water phase, and the mixture assumes the viscosity of the water.

Volume ratio

The Volume ratio method calculates mixture viscosity as follows:

$$\mu_m = \mu_o \varphi_o + \mu_w \varphi_w \quad \text{Eq. 4.381}$$

where

μ_o	is oil viscosity
φ_o	is the volume fraction of oil
μ_w	is the water viscosity
φ_w	is the volume fraction of water

User-Supplied Table

This method uses a user-supplied table of viscosities or viscosity multipliers against flowing (in situ) watercut. The table is entered in the dialog revealed by pressing the **Setup emulsion table** button. The first watercut value in the table must be zero. The viscosity value at zero watercut is used to divide into all the others to yield multipliers. Therefore, the viscosity table can be populated with absolute viscosity values, or with multipliers.

The table is applied to watercuts from zero up to the supplied watercut cutoff value; above this, the liquid viscosity is set to the water viscosity using a transition region.

Woelflin

The Woelflin correlations assume that the continuous phase changes from oil to water at a given watercut cutoff point. This means that, at a watercut below or equal to the cutoff value, a water-in-oil emulsion forms, and the emulsion viscosity is given by the Woelflin correlation. At a watercut above the cutoff value, oil droplets are carried by a continuous water phase, and the mixture assumes the viscosity of the water, using a transition region.

In his 1942 paper, Woelflin described 3 types of water-in-oil emulsions, which he labeled **loose**, **medium** and **tight**. The paper provides tables of viscosity multiplication factors as a function of watercut for the 3 types, and a chart showing curves to fit the data. The PIPESIM implementation is a digitization of the curves.

The viscosity of all 3 emulsion types increases with watercut up to the specified cutoff value, above which it falls and assumes the value of the water viscosity. It should be noted that all 3 emulsion types can yield **emulsion viscosities many times greater than the oil viscosity**. In the case of the tight emulsion, multiplier values in the region of 100 are readily obtained. In his experiments on tight emulsions, Woelflin reported that the viscosity of a 60% watercut emulsion could not be determined, because the mixture was too viscous to flow through the viscometer.

Versions of PIPESIM prior to the 2007.1 release implemented only the loose emulsion correlation, using a curve-fit as follows:

$$\mu_m = \mu_o \left(1 + 0.00123 V_w^{2.2} \right) \quad \text{Eq. 4.382}$$

where μ_o is oil viscosity, and φ_w is volume fraction of water. This option is retained for backwards compatibility and is called **Pipesim Original Woelflin**. It gives very similar answers to the new loose emulsion option up to a watercut of 60%, but diverges above this.

Brinkman

The Brinkman correlation calculates elevated emulsion viscosities on either side of the cutoff, using the formula

$$\mu_L = \mu_c(1 - \varphi_d)^{-2.5} \quad \text{Eq. 4.383}$$

where μ_c is the viscosity of the continuous phase and φ_d is the volume fraction of the discontinuous phase.

Vand

The Vand correlations calculate elevated emulsion viscosities on either side of the cutoff, using the formula

$$\mu_L = \mu_c e^{\left[\frac{(k_1 \varphi_d)}{(1-k_2 \varphi_d)} \right]} \quad \text{Eq. 4.384}$$

where μ_c is the viscosity of the continuous phase,

φ_d is the volume fraction of the discontinuous phase,

and the coefficients k_1 and k_2 are selected as follows. The Vand coefficients are 2.5 and 0.609; Barnea and Mizrahi are 2.66 and 1.0; the user-supplied coefficients are entered in the accompanying data entry boxes.

Richardson

The Richardson correlation calculates elevated emulsion viscosities on either side of the cutoff, using the formula

$$\mu_L = \mu_c(k \varphi_d)^e \quad \text{Eq. 4.385}$$

where μ_c is the viscosity of the continuous phase,

φ_d is the volume fraction of the discontinuous phase,

and k is a user-supplied constant. Separate values of k can be provided for oil-in-water and water-in-oil conditions, the default values are 3.8 and 6.6.

Leviton and Leighton

The Leviton and Leighton correlation calculates elevated emulsion viscosities either side of the cutoff, using the formula

$$\mu_L = \mu_c \left[\frac{2.5(\mu_d + 0.4\mu_c)}{(\mu_d + \mu_c)(\varphi_d^{1.66} + \varphi_d^{3.66})} \right]^e \quad \text{Eq. 4.386}$$

where μ_d and μ_c are the viscosities of the discontinuous and continuous phases, and φ_d is the volume fraction of the discontinuous phase.

Brauner and Ullman

The Brauner and Ullman correlation can be used to calculate the watercut cutoff value. It uses the formula

$$c = 1 - \left(\frac{\rho t^{0.6} \mu_t^{0.4}}{(1 + \rho t^{0.6} \mu_t^{0.4})} \right) \quad \text{Eq. 4.387}$$

where

c	is the cutoff/100
$\mu_t = \frac{\mu_o}{\mu_w}$	
μ_o	is the oil viscosity (in cP)
μ_w	is the water viscosity (in cP)
$\rho_t = \frac{\rho_o}{\rho_w}$	
ρ_o	is the oil density (in lb/ft ³)
ρ_w	is the water density (in lb/ft ³)

See also: [Inversion \(p.556\)](#) and [Volume Ratio \(p.556\)](#).

Limits and Safety factors

Emulsion correlations and methods have the potential to cause difficulty for the calculation engine. Extremely large viscosities can be predicted, this can cause convergence failure. The discontinuous behavior around the inversion point (cutoff) can also cause problems, particularly in a network model. As a result, a number of limits and safety factors have been introduced, as described below. All of these are **properties of the model**: only one value is held, and it is applied to all fluids in the model.

Maximum cutoff

A typical value for the cutoff is between 55% to 70%, and the default is 60%. The Woelflin correlations are particularly sensitive to high cutoff values, so there is a maximum limit of 70%, which will be applied silently. The limit may be extended using the keyword [MAXCUTOFF= \(p.609\)](#).

Transition region above the cutoff

The Woelflin and user-supplied table methods exhibit pronounced discontinuity about the cutoff. For these methods therefore, the discontinuity about the cutoff is smoothed with a transition region, extending from the cutoff value in the direction of increasing watercut. By default this is 5% wide. Predicted viscosities in this region are interpolated between the value predicted at the cutoff (the maximum emulsion viscosity point, with oil the continuous phase) and the value at cutoff plus transition region width (the viscosity predicted from a continuous water phase). The size of the transition region can be controlled with the keyword [SMOOTHCUTOFF \(p.609\)](#) =.

Maximum Emulsion Table Multiplier

The user-supplied and Woelflin correlations are subject to a maximum multiplier limit, default value 100. This can be controlled with the keyword [MAXEMULSION \(p.609\)](#) =.

Maximum liquid viscosity

As viscosity increases, the resistance to fluid flow also increases. There comes a point where viscosity is so high that the term 'fluid' ceases to be appropriate, and the prediction of pressure drops due to fluid flow can be regarded as non physical. The maximum liquid viscosity is by default 1.0E+7 (ten million) cP, this can be controlled by the keyword [MAXLIQVISC \(p.609\)](#) =.

Liquid-gas Surface Tension

The surface tension between the liquid and gas phase is used in two phase flow correlations, for example to calculate bubble velocity. If there are two liquid phases present, the surface tension will depend on the oil-gas surface tension, the water-gas surface tension, and the water cut.

In **black oil models**, when there is sufficient oil, it is assumed that the liquid is segregated with the water below the oil. The gas is therefore only in contact with the oil, and the surface tension is given by:

$$\sigma_L = \sigma_O \quad WCUT < 60$$

If water dominates the liquid then the surface tension is taken as an average of the oil-gas and water-gas surface tensions:

$$\sigma_L = \sigma_O \cdot \left(1 - \frac{WCUT}{100}\right) + \sigma_O \cdot \frac{WCUT}{100} \quad WCUT > 60$$

σ_L is the surface tension between the oil and the gas (dynes/cm)

σ_O is the surface tension between the oil and the gas (dynes/cm)

σ_W is the surface tension between the water and the gas (dynes/cm)

$WCUT$ is the water cut (%)

4.6 Solvers

4.6.1 Tolerance Equations

The tolerance of each pressure is calculated from the equation:

$$tol = abs\left(\frac{(p - p_{av})}{(p_{av} \times 100 \%)}\right) \quad \text{Eq. 4.388}$$

If all tolerance values are within the specified network tolerance then that node has passed the pressure convergence test. This is repeated for each node. The total mass flowrate into and the total mass flowrate out of a node are averaged.

The tolerance is calculated from the equation:

$$tol = abs\left(\frac{(Totalmq_{in} - Totalmq_{av})}{(Totalmq_{av} \times 100 \%)}\right) \quad \text{Eq. 4.389}$$

4.7 Typical and Default Data

4.7.1 Limits

The following limitations apply to the PIPESIM modules.

General

- Maximum number of components in a stream: 50

Pipeline and facilities

- Maximum number of sources: 1
- Maximum number of sinks: 1
- Maximum number of pipe coatings: 4
- Maximum number of nodes for a pipeline or riser: 101

Well Performance

- Maximum number of completions: 10
- Maximum number of sinks: 1
- Maximum number tubing coatings (using Expert mode) : 10
- Maximum number of nodes for a tubing: 100
- Maximum number of geothermal survey points: 100
- Maximum number of tubing strings: Detailed model = 20, Simple model = 4

Network

- Maximum number of wells / branches: Unlimited.
- Maximum number of nodes: Unlimited.
- Maximum number of PVT files: 500
- Maximum number of compositions: 1,000
- Maximum number of Black Oil compositions: 1,024
- Maximum number of PQ data points: 30

Note: Although the maximum number of wells, and so on, is unlimited practical limits may apply depending upon the configuration of the PC used. The limiting factors [for large models] will be memory and processor speed. Please see the License and Installation Guide for your version of PIPESIM for recommendations on memory.

4.7.2 Tubing and Pipeline Tables

Tubing/Casing Tables

Nominal Bore	Weight lb/ft	OD in	ID in	WT in		
1 1/4in	3.02	1.660	1.278	0.191		
1 1/4in	2.3	1.660	1.379	0.140	H-40,J-55,C-75,L-80,N-80,C-90	Tubing
1 1/4in	2.33	1.660	1.379	0.140		
1 1/4in	2.4	1.660	1.379	0.140	H-40,J-55,C-75,L-80,N-80,C-90	Tubing
1 1/4in	2.1	1.660	1.410	0.125		
2 3/8in	7.7	2.375	1.703	0.336		
2 3/8in	6.2	2.375	1.853	0.261		
2 3/8in	5.8	2.375	1.867	0.254	C-75,L-80,N-80,C-90,P-105	Tubing
2 3/8in	5.95	2.375	1.867	0.254	C-75,L-80,N-80,C-90,P-105	Tubing
2 3/8in	5.3	2.375	1.939	0.218		
2 3/8in	4.6	2.375	1.995	0.190	H-40,J-55,C-75,L-80,N-80,C-90,P-105	Tubing
2 3/8in	4.7	2.375	1.995	0.190	H-40,J-55,C-75,L-80,N-80,C-90,P-105	Tubing
2 3/8in	4	2.375	2.041	0.167	H-40,J-55,C-75,L-80,N-80,C-90	Tubing
2 7/8in	11	2.875	2.065	0.405		
2 7/8in	10.7	2.875	2.091	0.392		
2 7/8in	9.5	2.875	2.195	0.340		
2 7/8in	8.6	2.875	2.259	0.308	C-75,L-80,N-80,C-90,P-105	Tubing
2 7/8in	8.7	2.875	2.259	0.308	C-75,L-80,N-80,C-90,P-106	Tubing

2 7/8in	7.8	2.875	2.323	0.276	C-75,L-80,N-80,C-90,P-105	Tubing
2 7/8in	7.9	2.875	2.323	0.276	C-75,L-80,N-80,C-90,P-106	Tubing
2 7/8in	6.4	2.875	2.441	0.217	H-40,J-55,C-75,L-80,N-80,C-90,P-105	Tubing
2 7/8in	6.5	2.875	2.441	0.217	H-40,J-55,C-75,L-80,N-80,C-90,P-105	Tubing
3 1/2in	16.7	3.500	2.480	0.510		
3 1/2in	15.8	3.500	2.548	0.476		
3 1/2in	12.7	3.500	2.750	0.375	C-75,L-80,N-80,C-90,P-105	Tubing
3 1/2in	12.95	3.500	2.750	0.375	C-75,L-80,N-80,C-90,P-106	Tubing
3 1/2in	12.8	3.500	2.764	0.368		
3 1/2in	9.2	3.500	2.992	0.254	H-40,J-55,C-75,L-80,N-80,C-90,P-105	Tubing
3 1/2in	9.3	3.500	2.992	0.254	H-40,J-55,C-75,L-80,N-80,C-90,P-105	Tubing
3 1/2in	10.2	3.500	2.992	0.254	H-40,J-55,C-75,L-80,N-80,C-90	Tubing
3 1/2in	7.7	3.500	3.068	0.216	H-40,J-55,C-75,L-80,N-80,C-90	Tubing
4 in	13.4	4.000	3.340	0.330		
4 in	11.6	4.000	3.428	0.286		
4 in	11	4.000	3.476	0.262	H-40,J-55,C-75,L-80,N-80,C-90	Tubing
4 in	9.5	4.000	3.548	0.226	H-40,J-55,C-75,L-80,N-80,C-90	Tubing
4 1/2 in	19.2	4.500	3.640	0.430		
4 1/2 in	19.1	4.500	3.626	0.437	Q-125*,V-150*	Casing
4 1/2 in	16.6	4.500	3.750	0.375	Q-125*,V-150*	Casing
4 1/2 in	15.1	4.500	3.826	0.337	HC-95*,P-110,Q-125,V-150*	Casing
4 1/2 in	15.5	4.500	3.826	0.337		
4 1/2 in	13.5	4.500	3.920	0.290	J-55,K-55,C-75,L-80,N-80,C-90,HC-95*,P110	
4 1/2 in	12.6	4.500	3.958	0.271	H-40,J-55,C-75,L-80,N-80,C-90	Tubing
4 1/2 in	12.75	4.500	3.958	0.271	H-40,J-55,C-75,L-80,N-80,C-90	Tubing
4 1/2 in	11.6	4.500	4.000	0.250	J-55,K-55,C-75,L-80,N-80,C-90,HC-95*,P110	Casing
4 1/2 in	10.5	4.500	4.052	0.224	J-55,K-55	Casing
4 1/2 in	9.5	4.500	4.090	0.205	H-40,J-55,K-55	Casing
5 in	24.2	5.000	4.000	0.500	C-75,L-80,N-80,C-90,HC-95*,P110,Q125	Casing
5 in	23.2	5.000	4.044	0.478	C-75,L-80,N-80,C-90,HC-95*,P110,Q125,V150*	Casing
5 in	21.4	5.000	4.126	0.437	C-75,L-80,N-80,C-90,HC-95*,P110	Casing
5 in	20.8	5.000	4.156	0.422		
5 in	20.3	5.000	4.184	0.408		
5 in	18	5.000	4.276	0.362	C-75,L-80,N-80,C-90,HC-95*,P110,Q125,V150*	Casing

5 in	15	5.000	4.408	0.296	J-55,K-55,C-75,L-80,N-80,C-90,C-95,HC-95*,P110,Q125,V150*	Casing
5 in	13	5.000	4.494	0.253	J-55,K-55	Casing
5 in	11.5	5.000	4.560	0.220	J-55,K-55	Casing
5 1/2in	35	5.500	4.200	0.650	C-90	Casing
5 1/2in	26.8	5.500	4.500	0.500	Q-125*,V-150*	Casing
5 1/2in	26	5.500	4.548	0.476	C-90	Casing
5 1/2in	23	5.500	4.670	0.415	C-75,L-80,N-80,C-90,C-95,HC-95*,P110,Q125,V150*	Casing
5 1/2in	20	5.500	4.778	0.361	C-75,L-80,N-80,C-90,C-95,HC-95*,P110,Q125*,V150*	Casing
5 1/2in	17	5.500	4.892	0.304	J-55,K-55,C-75,L-80,N-80,C-90,C-95,HC-95*,P110,Q125*	Casing
5 1/2in	15.5	5.500	4.950	0.275	J-55,K-55	Casing
5 1/2in	14	5.500	5.012	0.244	H-40,J-55,K-55	Casing
5 1/2in	13	5.500	5.044	0.228		
6 in	26	6.000	5.132	0.434		
6 in	23	6.000	5.240	0.380		
6 in	20	6.000	5.352	0.324		
6 in	18	6.000	5.424	0.288		
6 in	15	6.000	5.675	0.163		
6 5/8 in	32	6.625	5.524	0.550	C-75,L-80,N-80,C-90,C-95,P110,Q125*,V150*	Casing
6 5/8 in	28	6.625	5.791	0.417	C-75,L-80,N-80,C-90,C-95,P110,Q125*,V150*	Casing
6 5/8 in	24	6.625	5.921	0.352	J-55,K-55,C-75,L-80,N-80,C-90,C-95,P110,Q125*,V150*	Casing
6 5/8 in	20	6.625	6.049	0.288	H-40,J-55,K-55	Casing
6 5/8 in	17	6.625	6.135	0.245		
7 in	42.7	7.000	5.750	0.625	Q125*,V150*	Casing
7 in	38	7.000	5.920	0.540	C-75,L-80,N-80,C-90,C-95,HC-95*,P110,Q125,V150*	Casing
7 in	35	7.000	6.004	0.498	C-75,L-80,N-80,C-90,C-95,HC-95*,P110,Q125,V150*	Casing
7 in	32	7.000	6.094	0.453	C-75,L-80,N-80,C-90,C-95,HC-95*,P110,Q125*,V150*	Casing
7 in	29	7.000	6.184	0.408	C-75,L-80,N-80,C-90,C-95,HC-95*,P110,Q125*,V150*	Casing
7 in	26	7.000	6.276	0.362	J-55,K-55,C-75,L-80,N-80,C-90,C-95,HC-95*,P110	Casing
7 in	23	7.000	6.366	0.317	J-55,K-55,C-75,L-80,N-80,C-90,C-95,HC-95*	Casing
7 in	20	7.000	6.456	0.272	H-40,J-55,K-55	Casing
7 in	17	7.000	6.538	0.231	H-40	Casing
7 5/8 in	47.1	7.625	6.375	0.625	C-75,L-80,N-80,C-90,C-95,P110,Q125	Casing
7 5/8 in	45.3	7.625	6.435	0.595	C-75,L-80,N-80,C-90,C-95,HC-95*,P110,Q125,V150*	Casing
7 5/8 in	39	7.625	6.625	0.500	C-75,L-80,N-80,C-90,C-95,HC-95*,P110,Q125,V150*	Casing

7 5/8 in	33.7	7.625	6.765	0.430	C-75,L-80,N-80,C-90,C-95,HC-95*,P110,Q125*,V150*	Casing
7 5/8 in	29.7	7.625	6.875	0.375	C-75,L-80,N-80,C-90,C-95,HC-95*,P110,Q125*,V150*	Casing
7 5/8 in	26.4	7.625	6.969	0.328	J-55,K-55,C-75,L-80,N-80,C-90,C-95,HC-95*	Casing
7 5/8 in	24	7.625	7.025	0.300	H40	Casing
7 5/8 in	20	7.625	7.125	0.250		
8 5/8 in	49	8.625	7.511	0.557	C-75,L-80,N-80,C-90,C-95,HC-95*,P110,Q125,V150*	Casing
8 5/8 in	44	8.625	7.625	0.500	C-75,L-80,N-80,C-90,C-95,HC-95*,P110,Q125*,V150*	Casing
8 5/8 in	40	8.625	7.725	0.450	C-75,L-80,N-80,C-90,C-95,HC-95*,P110,Q125*	Casing
8 5/8 in	36	8.625	7.825	0.400	J-55,K-55,C-75,L-80,N-80,C-90,C-95,HC-95*	Casing
8 5/8 in	32	8.625	7.921	0.352	H-40,J-55,K-55	Casing
8 5/8 in	28	8.625	8.017	0.304	H-40	Casing
8 5/8 in	24	8.625	8.097	0.264	J-55,K-55	Casing
9 5/8 in	71.8	9.625	8.125	0.750		
9 5/8 in	70.3	9.625	8.157	0.734	V150*	Casing
9 5/8 in	61.1	9.625	8.375	0.625	HC-95*,Q125*,V150*	Casing
9 5/8 in	58.4	9.625	8.435	0.595	HC-95*,Q125*,V150*	Casing
9 5/8 in	53.5	9.625	8.535	0.545	C-75,L-80,N-80,C-90,C-95,HC-95*,P110,Q125,V150*	Casing
9 5/8 in	47	9.625	8.681	0.472	C-75,L-80,N-80,C-90,C-95,HC-95*,P110,Q125	Casing
9 5/8 in	43.5	9.625	8.755	0.435	C-75,L-80,N-80,C-90,C-95,HC-95*,P110	Casing
9 5/8 in	40	9.625	8.835	0.395	J-55,K-55,C-75,L-80,N-80,C-90,C-95,HC-95*	Casing
9 5/8 in	36	9.625	8.921	0.352	H-40,J-55,K-55	Casing
9 5/8 in	32.3	9.625	9.001	0.312	H-40	Casing
9 5/8 in	29.3	9.625	9.063	0.281		
10 3/4 in	79.2	10.75 0	9.282	0.734	Q125*,V150*	Casing
10 3/4 in	73.2	10.75 0	9.406	0.672	Q125*,V150*	Casing
10 3/4 in	71.1	10.75 0	9.450	0.650	HC-95*,Q125*,V150*	Casing
10 3/4 in	65.7	10.75 0	9.560	0.595	HC-95*,P-110,Q125,V150*	Casing
10 3/4 in	60.7	10.75 0	9.660	0.545	HC-95*,P-110,Q125,V150	Casing
10 3/4 in	55.5	10.75 0	9.760	0.495	L-80,N-80,C-90,C-95,HC-95*,P-110,Q125*	Casing
10 3/4 in	51	10.75 0	9.850	0.450	J-55,K-55,C-75,L-80,N-80,C-90,C-95,HC-95*,P-110	Casing

10 3/4 in	45.5	10.75 0	9.950	0.400	H-40,J-55,K-54	Casing
10 3/4 in	40.5	10.75 0	10.05	0.350	H-40,J-55,K-55	Casing
10 3/4 in	32.75	10.75 0	10.19 2	0.279	H-40	Casing
11 3/4 in	66.7	3.915	11.75 0	10.65 6	Q-125*,V-150*	Casing
11 3/4 in	60	3.522	11.75 0	10.77 2	J-55,K-55,C-75,L-80,N-80,C-90,C-95,HC-95*,P-110,Q-125	Casing
11 3/4 in	54	11.75 0	10.88 0	0.435	J-55,K-55	Casing
11 3/4 in	47	11.75 0	11.00 0	0.375	J-55,K-55	Casing
11 3/4 in	42	11.75 0	11.08 4	0.333	H-40	Casing
11 3/4 in	38	11.75 0	11.15 0	0.300		
13 3/8 in	100.3	13.37 5	11.90 7	0.734	V-150*	Casing
13 3/8 in	98	13.37 5	11.93 7	0.719		
13 3/8 in	92.5	13.37 5	12.03 1	0.672	Q-125*	Casing
13 3/8 in	86	13.37 5	12.12 5	0.625	HC-95*	Casing
13 3/8 in	85	13.37 5	12.15 9	0.608		
13 3/8 in	77	13.37 5	12.27 5	0.550		
13 3/8 in	71	13.37 5	12.28 1	0.547	Q-125*	Casing
13 3/8 in	72	13.37 5	12.34 7	0.514	C-75,L-80,N-80,C-90,C-95,HC-95*,P-110,Q-125	Casing
13 3/8 in	68	13.37 5	12.41 5	0.480	J-55,K-55,C-75,L-80,N-80,C-90,C-95,P-110	Casing
13 3/8 in	61	13.37 5	12.51 5	0.430	J-55,K-55	Casing
13 3/8 in	54.5	13.37 5	12.61 5	0.380	J-55,K-55	Casing

13 3/8 in	48	13.37 5	12.71 5	0.330	H-40	Casing
16 in	109	16.00 0	14.68 8	0.656		
16 in	84	16.00 0	15.01 0	0.495	J-55,K-55	Casing
16 in	75	16.00 0	15.21 4	0.393	J-55,K-55	Casing
16 in	65	16.00 0	15.25 0	0.375	H-40	Casing
16 in	55	16.00 0	15.37 6	0.312		
18 5/8 in	87.5	18.62 5	17.75 5	0.435	H-40,J-55,K-55	Casing
20 in	133	20.00 0	18.73 0	0.635	J-55,K-55	Casing
20 in	106.5	20.00 0	19.00 0	0.500	J-55,K-55	Casing
20 in	94	20.00 0	19.12 4	0.438	H-40,J-55,K-55	Casing

Pipeline tables

Nominal Bore	Schedule	OD in	ID in	Wall Thickness in
1/8in	Sch 80	0.406	0.217	0.094
1/8in	Sch 40	0.406	0.268	0.069
1/4in	Sch 80	0.539	0.303	0.118
1/4in	Sch 40	0.539	0.362	0.089
3/8in	Sch 80	0.673	0.421	0.126
3/8in	Sch 40	0.673	0.492	0.091
1/2in	XXS	0.839	0.252	0.293
1/2in	Sch 160	0.839	0.461	0.189
1/2in	Sch 80	0.839	0.543	0.148
1/2in	Sch 40	0.839	0.622	0.108
3/4in	XXS	1.051	0.437	0.307
3/4in	Sch 160	1.051	0.614	0.219
3/4in	Sch 80	1.051	0.744	0.154
3/4in	Sch 40	1.051	0.827	0.112

1in	XXS	1.315	0.599	0.358
1in	Sch 160	1.315	0.815	0.250
1in	Sch 80	1.315	0.957	0.179
1in	Sch 40	1.315	1.049	0.133
1 1/4in	XXS	1.661	0.898	0.382
1 1/4in	Sch 160	1.661	1.161	0.250
1 1/4in	Sch 80	1.661	1.280	0.191
1 1/4in	Sch 40	1.661	1.382	0.140
1 1/2in	XXS	1.902	1.102	0.400
1 1/2in	Sch 160	1.902	1.339	0.281
1 1/2in	Sch 80	1.902	1.500	0.201
1 1/2in	Sch 40	1.902	1.610	0.146
2in	XXS	2.375	1.503	0.436
2in	Sch 160	2.375	1.687	0.344
2in	Sch 80	2.375	1.939	0.218
2in	Sch 40	2.375	2.067	0.154
2 1/2in	XXS	2.874	1.772	0.551
2 1/2in	Sch 160	2.874	2.469	0.374
2 1/2in	Sch 80	2.874	2.323	0.276
2 1/2in	Sch 40	2.874	2.126	0.203
3in	XXS	3.500	2.300	0.600
3in	Sch 160	3.500	2.624	0.438
3in	Sch 80	3.500	2.900	0.300
3in	Sch 40	3.500	3.068	0.216
4in	XXS	4.500	2.728	0.886
4in	Sch 160	4.500	3.438	0.531
4in	Sch 120	4.500	3.622	0.439
4in	Sch 80	4.500	3.826	0.337
4in	Sch 40	4.500	4.026	0.237
5in	XXS	5.563	4.063	0.750
5in	Sch 160	5.563	4.311	0.626
5in	Sch 120	5.563	4.563	0.500
5in	Sch 80	5.563	4.815	0.374
5in	Sch 40	5.563	5.047	0.258
6in	XXS	6.625	4.897	0.864

6in	Sch 160	6.625	5.187	0.719
6in	Sch 120	6.625	5.504	0.561
6in	Sch 80	6.625	5.761	0.432
6in	Sch 40	6.625	6.211	0.280
8in	Sch 160	8.626	6.815	0.906
8in	XXS	8.626	6.878	0.874
8in	Sch 140	8.626	7.004	0.811
8in	Sch 120	8.626	7.189	0.719
8in	Sch 100	8.626	7.437	0.594
8in	Sch 80	8.626	7.626	0.500
8in	Sch 60	8.626	7.815	0.406
8in	Sch 40	8.626	7.980	0.323
8in	Sch 30	8.626	8.071	0.278
8in	Sch 20	8.626	8.126	0.250
10in	Sch 160	10.748	8.496	1.126
10in	Sch 140	10.748	8.748	1.000
10in	Sch 120	10.748	9.059	0.844
10in	Sch 100	10.748	9.311	0.719
10in	Sch 80	10.748	9.559	0.594
10in	Sch 60	10.748	9.748	0.500
10in	Sch 40	10.748	10.020	0.364
10in	Sch 30	10.748	10.134	0.307
10in	Sch 20	10.748	10.248	0.250
12in	Sch 160	12.752	10.126	1.313
12in	Sch 140	12.752	10.500	1.126
12in	Sch 120	12.752	10.752	1.000
12in	Sch 100	12.752	11.063	0.844
12in	Sch 80	12.752	11.378	0.687
12in	Sch 60	12.752	11.630	0.561
12in	Sch 40	12.752	11.941	0.406
12in	Sch 30	12.752	12.091	0.331
12in	Sch 20	12.752	12.252	0.250
14in	Sch 160	14.000	11.189	1.406
14in	Sch 140	14.000	11.500	1.250
14in	Sch 120	14.000	11.811	1.094

14in	Sch 100	14.000	12.126	0.937
14in	Sch 80	14.000	12.500	0.750
14in	Sch 60	14.000	12.811	0.594
14in	Sch 40	14.000	13.122	0.439
14in	Sch 30	14.000	13.252	0.374
14in	Sch 20	14.000	13.378	0.311
14in	Sch 10	14.000	13.500	0.250
16in	Sch 160	16.000	12.811	1.594
16in	Sch 140	16.000	13.126	1.437
16in	Sch 120	16.000	13.563	1.219
16in	Sch 100	16.000	13.937	1.031
16in	Sch 80	16.000	14.311	0.844
16in	Sch 60	16.000	14.689	0.656
16in	Sch 40	16.000	15.000	0.500
16in	Sch 30	16.000	15.252	0.374
16in	Sch 20	16.000	15.378	0.311
16in	Sch 10	16.000	15.500	0.250
18in	Sch 160	18.000	14.437	1.781
18in	Sch 140	18.000	14.874	1.563
18in	Sch 120	18.000	15.252	1.374
18in	Sch 100	18.000	15.689	1.156
18in	Sch 80	18.000	16.126	0.937
18in	Sch 60	18.000	16.500	0.750
18in	Sch 40	18.000	16.878	0.561
18in	Sch 30	18.000	17.122	0.439
18in	Sch 20	18.000	17.378	0.311
18in	Sch 10	18.000	17.500	0.250
20in	Sch 160	20.000	16.063	1.969
20in	Sch 140	20.000	16.500	1.750
20in	Sch 120	20.000	17.000	1.500
20in	Sch 100	20.000	16.650	1.675
20in	Sch 80	20.000	17.937	1.031
20in	Sch 60	20.000	18.378	0.811
20in	Sch 40	20.000	18.811	0.594
20in	Sch 30	20.000	19.000	0.500

20in	Sch 20	20.000	19.252	0.374
20in	Sch 10	20.000	19.500	0.250
24in	Sch 160	24.000	19.311	2.344
24in	Sch 140	24.000	19.874	2.063
24in	Sch 120	24.000	20.378	1.811
24in	Sch 100	24.000	20.937	1.531
24in	Sch 80	24.000	21.563	1.219
24in	Sch 60	24.000	22.063	0.969
24in	Sch 40	24.000	22.622	0.689
24in	Sch 30	24.000	22.878	0.561
24in	Sch 20	24.000	23.252	0.374
24in	Sch 10	24.000	23.500	0.250
30in	Sch 30	30.000	28.748	0.626
30in	Sch 20	30.000	29.000	0.500
30in	Sch 10	30.000	29.378	0.311

4.7.3 Typical Values

Fluid Properties

The table below gives typical values for properties, in Engineering units and data for various oil locations worldwide.

	Default	Min	Max	North Sea	North America	South America	Middle East	Far East	Australia
Black Oil Properties									
Water cut	0	0	100						
GOR	required				340 -1,100	600-20,000			
Gas s.g.	0.64				0.64 - 0.81	0.65 - 0.8			
Water s.g.	1.02				1.01 - 1.03	1.01 - 1.05			
API	30			38	13 - 56	7-45	32-44	34	
Solution Gas Correlation	Lasater			Glaso	Lasater				
Viscosity Data									
Dead Oil viscosity	Beggs and Robinson				Beggs and Robinson				

Viscosity @ 200F	2.247			0.4 - 11	5 - 10,000			
Viscosity @ 60F	117.8			3.8 - 82,000	120 - 10,000			
Heat capacities								
Oil (p.726)	0.45							
Gas (p.726)	0.55							
Water (p.726)	1.0							

Default values can be changed - click the relevant link.

Roughness

Material	ft.	in
Drawn tubing (brass, lead, glass, and the like)	0.000005	0.00006
Commercial steel or wrought iron	0.00015	0.0018
Asphalted cast iron	0.0004	0.0048
Galvanized iron	0.0005	0.006
Cast iron	0.00085	0.010
Wood stave	0.0006-0.003	0.0072-0.036
Concrete	0.001-0.01	0.012-0.12
Riveted steel	0.003-0.03	0.036-0.36

Thermal Conductivities

Material	Density (kg/m ³)	Thermal Conductivity Btu/hr/ft/F	Thermal Conductivity (W/m/K)
Anhydrite		0.75	
Carbon Steel	7900	28.9	50
Concrete Weight Coat	2000 - 3000	0.81 - 1.15	1.4 - 2.0
Corrosion Coat (Bitumen)	-	0.19	0.33
Corrosion Coat (Epoxy)	-	0.17	0.30
Corrosion Coat (Polyurathane)	-	0.12	0.20
Dolomite		1.0	
Ground (Earth)		0.37 - 1.5	
Gypsum		0.75	

Halite		2.8	
Ice	900	1.27	2.2
Lignite		2.0	
Limestone		0.54	
Line pipe		27	46.7
Mild Steel tubing		26	45
Mud	1500	0.75 - 1.5	1.3 - 2.6
Neoprene Rubber	-	0.17	0.3
Plastic coated pipe		20	34.6
Plastic coated tubing		20	34.6
Polyurathane Foam (dry)	30 - 100	0.011 - 0.023	0.02 - 0.04
Polyurathane Foam (wet)	-	0.023 - 0.034	0.4 - 0.6
PVC Foam (dry)	100 - 340	0.023 - 0.025	0.040 - 0.044
Sandstone		1.06	
Shale		0.7	
Stainless Steel	-	8.67	15
Stainless steel (13%)		18	31.14
Stainless steel (15%)		15	26
Syntactic Foam (dry)	500	0.052	0.09
Syntactic foam (wet)	-	0.17	0.3
Volcanics		1.6	
Wet Sand	1600	1.04 - 1.44	1.8 - 2.5

Thermal Conductivities in W/m/K (Liquids and Gases)

Fluid	Default	Temperature	Temperature	Temperature	Temperature
		5°C	20°C	100°C	200°C
Air		0.024	0.026	0.030	0.037
Crude Oil (p.715) (30 API)	0.138	0.14	0.14	0.12	0.10
Glycol (DEG)		0.26	0.25	0.20	0.14
Natural Gas (p.715) (P=1 bara)	0.035	0.030	0.032	0.045	0.062
Natural Gas (p.715) (P=100 bara)		0.045	0.045	0.052	0.070
Natural Gas (p.715) (P=200 bara)		0.071	0.067	0.064	0.074

Natural Gas (p.715) (P=300 bara)		0.090	0.085	0.076	0.083
Water (p.715)	0.605	0.56	0.59	0.68	0.68

Default values can be changed - click the relevant link.

Permeability

For a gas well, this is gas permeability. For an oil well, this is total liquid permeability.

Typical values are:

- < 1 md : Very low
- 1 - 10 md: Low
- 10 - 50 md: Mediocre
- 50 - 200 md: Average
- 200 - 500 md: Good
- > 500 md: Excellent

Drainage Radius

Common drainage radii are:

- 40 acres 745 ft (227 m)
- 80 acres 1053 ft (321 m)
- 160 acres 1490 ft (454 m)
- 640 acres 2980 ft (908 m)

Fittings

Model fittings

Fittings (elbows, valves and tees) are modeled by the standard practice of utilizing equivalent length. From the fittings table determine the extra length of pipe that needs to be added to the model to exert the same pressure drop as the required fitting.

Valves - Equivalent lengths of 100% open valves in feet

Model as a pipe with the required ID and set the equivalent length as indicated in the table below. For example to model a 3/4 inch angle valve add a pipe section of ID 3/4 and a length of 12 feet.

Pipe Size (Inches)	Gate Valve (feet)	Globe Valve (feet)	Angle Valve (feet)
1/2	.35	17	8
3/4	.50	22	12
1	.6	27	14
1 1/4	.8	38	18

1 1/2	1.0	44	22
2	1.2	53	28
2 1/2	1.4	68	33
3	1.7	80	42
4	2.3	120	53
5	2.8	140	70
6	3.5	170	84
8	4.5	220	120
10	5.7	280	140
12	9	400	190
14	10	450	210
16	11	500	240
18	12	550	280
20	14	650	300
22	15	688	335
24	16	750	370

Elbows: Equivalent length of elbows in feet

Model as a pipe with the required ID and set the equivalent length as indicated in the table below. For example to model a 3/4 standard elbow add a pipe section of ID 3/4 and a length of 2.2 feet.

Pipe Size:	St'd elbow	Med. sweep elbow	Long sweep elbow
Inches	feet	feet	feet
1/2	1.5	1.3	1
3/4	2.2	1.8	1.3
1	2.7	2.3	1.7
1 1/4	3.6	3	2.3
1 1/2	4.5	3.6	2.8
2	5.2	4.6	3.5
2 1/2	6.5	5.5	4.3
3	8	7	5.2
4	11	9	7
5	14	12	9
6	16	14	11
8	21	18	14
10	26	22	17

Tees: Equivalent length of Tees in feet

Model as a pipe with the required ID and set the equivalent length as indicated in the table below. For example to model a 3/4 inch Tee (straight through) add a pipe section of ID 3/4 and a length of 1.3 feet

Pipe Size:	Tee (straight through)	Tee (rt. angle flow)
Inches	feet	feet
1/2	1	3.2
3/4	1.3	4.5
1	1.7	5.7
1 1/4	2.3	7.5
1 1/2	2.8	9
2	3.5	12
2 1/2	4.3	14
3	5.2	16
4	7	22
5	9	27
6	11	33
8	14	43
10	17	53

4.8 Glossary

The PIPESIM help uses the following symbols:

4.8.1 Roman Letters

a	is the major axis of the drainage ellipse	ft	m
$A = \frac{\pi D^2}{4}$	is the pipe cross- sectional area	ft^2	m^2
$B = \frac{hR}{k}$	is the Biot number	dimensionless	dimensionless
B_o	is the oil formation volume factor	$bbI / STBO$	
c, C	is the specific heat capacity	$BTU / lb \cdot {}^{\circ}F$	$J / kg \cdot K$

d, D	is the pipe diameter	ft	m
E	is the specific total energy	BTU / lb	J / kg
f	is the friction factor	dimensionless	dimensionless
	is the frequency	Hz	Hz
F, R	is the gas/oil ratio	dimensionless	dimensionless
g	is the acceleration due to gravity	= 32.17 ft / s ²	= 9.81 m / s ²
$Gr = \frac{L^3 \rho^2 \beta g \Delta T}{\mu^2}$	is the Grashof number	dimensionless	dimensionless
h	is the local heat transfer coefficient	BTU / h · ft ² · °F	W / m ² · K
$H = U + PV$	is the specific enthalpy	BTU / lb	J / kg
H_L	is the liquid holdup	dimensionless	dimensionless
$Head$	is the head	ft · lb _f / lb	N · m / kg
J	is the productivity index		
k	is the thermal conductivity	BTU / h · ft · °F	W / m · K
	is the absolute permeability	ft ²	m ²
L	is the pipe length	ft	m
	is the horizontal well length		
m	is the mass	lb	kg
M	is the molecular weight	lb / mol	kg / kmol
	is the number of magnetic poles in an ESP	dimensionless	dimensionless

n	is the polytropic coefficient	dimensionless	dimensionless
		mol	mol
	is the number of moles		
N	is the compressor speed		
$Nu = \frac{hL}{k}$	is the Nusselt number	dimensionless	dimensionless
p, P	is the pressure	psi or lbf/in^2	N/m^2
$Power$	is the power	hp	W
$Pr = \frac{\mu c_p}{k}$	is the Prandtl number	dimensionless	dimensionless
q	is the mass flow rate	lb/s	kg/s
Q	is the heat transfer rate	BTU/h	W
	is the average liquid rate		
r	is the radial distance from the centre of well	ft	m
r_w	is the wellbore radius	ft	m
r_{eh}	is the drainage radius of a horizontal well	ft	m
R	is the pipe radius	ft	m
	is the gas constant	$= 1545.35 lb_f \cdot ft/lb - mol \cdot {}^oR = 8.314 J/K \cdot mol$	
$Ra = Pr \cdot Gr$	is the Rayleigh number	dimensionless	dimensionless
Re	is the Reynolds number	dimensionless	dimensionless
S	is the specific entropy	$\frac{BTU}{lb \cdot {}^oF}$	$\frac{J}{kg \cdot K}$

$S = \frac{Q}{2\pi k \Delta T}$	is the pipe burial shape factor is the skin	dimensionless	dimensionless
t	is the well operating time	h	s
T	is the temperature	${}^{\circ}F, {}^{\circ}R$	K
U	is the specific internal energy is the overall heat transfer coefficient	BTU/lb $BTU/h \cdot ft^2 \cdot {}^{\circ}F$	J/kg $W/m^2 \cdot K$
v	is the fluid velocity is the ESP speed	ft/s rev/min	m/s rev/s
V	is the volume	ft^3	J/kg
wt	is the pipe wall thickness	ft	m
W_s	is the shaft work	BTU	J
z	is the vertical displacement above a gravitational datum level	ft	m
Z	is the compressibility is the pipe burial depth	ft	m

4.8.2 Greek Letters

θ	is the angle of the pipe to the horizontal		
$\theta_{bur} = \sin^{-1}\left(\frac{Z}{R}\right)$	is the angle of the buried arc of a partially buried pipe		
$\alpha = \frac{k}{\rho c_p}$	is the thermal diffusivity	ft^2/s	m^2/s
$\beta = -\frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_H$	is the volumetric thermal expansion coefficient	$1/{}^{\circ}F, 1/{}^{\circ}R$	$1/K$

$\gamma = \frac{C_p}{C_V}$	is the ratio of specific heats is the specific gravity (relative density)	dimensionless	dimensionless
$\delta = \frac{d_{beam}}{d_{up}}$	is the choke diameter ratio	dimensionless	dimensionless
ϵ	is the pipe roughness	ft	m
η	is the efficiency, expressed as a fraction, $0 < \eta \leq 1$		
$\phi = \frac{Av_G}{Av_G + Av_L}$	is the void fraction	dimensionless	dimensionless
μ	is the fluid dynamic viscosity	$cp = 10^{-3} Pa \cdot s \quad Pa \cdot s = kg / m \cdot s$	
ρ	is the density	Ib / ft^3	kg / m^3
$\rho_{ns} = \lambda_L \rho_L + \lambda_G \rho_G$	is the no-slip density	Ib / ft^3	kg / m^3
λ	is the flowing fraction	dimensionless	dimensionless
σ	is the interfacial (surface) tension	dynes / cm	N / m

4.8.3 Subscripts

b bubble point

bulk

c critical

G gas phase

L liquid phase

m mixture

o oil

r reduced

R reservoir

- s gas/oil ratio, solution
- slip or slippage
- v vaporization or vapor phase
- volume or volumetric
- w water

4.9 Conversion Factors

Common conversion factors used in PIPESIM are tabulated below.

4.9.1 Length

$$1 \text{ ft} = 0.3048 \text{ m} \quad 1 \text{ m} = 3.28084 \text{ ft}$$

$$1 \text{ ft} = 12 \text{ in}$$

4.9.2 Volume

$$1 \text{ ft}^3 = 0.02832 \text{ m}^3 \quad 1 \text{ m}^3 = 35.31467 \text{ ft}^3$$

$$1 \text{ barrel} = 5.61458 \text{ ft}^3 \quad 1 \text{ ft}^3 = 0.17811 \text{ barrel}$$

$$1 \text{ barrel} = 0.15899 \text{ m}^3 \quad 1 \text{ m}^3 = 6.28981 \text{ barrel}$$

4.9.3 Mass

$$1 \text{ lb} = 0.4536 \text{ kg} \quad 1 \text{ kg} = 2.2046 \text{ lb}$$

4.9.4 Time

$$1 \text{ hour} = 3600 \text{ s}$$

$$1 \text{ day} = 86400 \text{ s}$$

4.9.5 Gravity

$$g = 32.18 \text{ ft} \cdot \text{s}^{-2} \quad g = 9.81 \text{ m} \cdot \text{s}^{-2}$$

$$g = 1/144 \text{ psi} \cdot \text{ft}^2 \cdot \text{lb}^{-1}$$

4.9.6 Pressure

The engineering units of pressure psi, needs to be treated with care. One psi is one pound-force per square inch, or 144 pound force per square foot. A pound force is the force exerted by one pound weight, which is one pound times the acceleration due to gravity $g = 32.18 \text{ ft} \cdot \text{s}^{-2}$.

$$1 \text{ psi} = 144 \frac{\text{lbf}}{\text{ft}^2} = 144 \cdot g \text{ lb} \cdot \frac{\text{ft}}{\text{s}^2} \cdot \frac{1}{\text{ft}^2}$$

$$1 \text{ bar} = 10^5 \text{ Pa}$$

$$1 \text{ bar} = 14.504 \text{ psi}$$

$$1 \text{ Atm} = 1.01325 \text{ bar}$$

$$1 \text{ Atm} = 14.70 \text{ psi}$$

4.9.7 Energy

$$1 \text{ BTU} = 1.055056 \text{ kJ} \quad 1 \text{ kJ} = 0.947817 \text{ BTU}$$

$$1 \text{ kJ} = 10^{-3} \text{ Pa} \cdot \text{m}^3$$

4.9.8 Power

$$1 \text{ hp} = 550 \cdot g \text{ ft}^2 \cdot \text{lb} \cdot \text{s}^{-3} \quad 1 \text{ hp} = 0.7457 \text{ kW}$$

4.9.9 Dynamic viscosity

$$1 \text{ cP} = 10^{-3} \text{ Pa} \cdot \text{s}$$

4.9.10 Permeability

$$1 \text{ mD} = 10^{-10} \frac{\text{Pa}}{\text{Atm}} \cdot \text{m}^2$$

4.10 References

Adames, P.E.

"Analysis and comparison of three comprehensive mechanistic hydrodynamic models for vertical upward and deviated gas-liquid flow through pipes". MSc thesis, University of Calgary, 2009.

Aggour

"Hydrodynamics and Heat Transfer in Two-Phase Two-Component Flow", PhD thesis, University of Manitoba, Winnipeg, Canada (1978).

- Al-Hussainy, R., Ramey Jr., H. J. and Crawford, P. B. "The Flow of Real Gases Through Porous Media", JPT (1966) 624-636.
- Alhanati, F. J. S., Schmidt, Z., Doty, D. R. and Lagerlef, D. D. "Continuous Gas-Lift Instability: Diagnosis, Criteria, and Solutions", SPE 26554 (1993)
- Alves,I.N., Alhanati, F. J. S. and Shoham,O. "A Unified Model for Predicting Flowing Temperature Distribution in Wellbores and Pipelines", SPE 20632 (1992)
- Ashford, F.E. and Pierce, P.E. "Determining Multiphase Pressure Drops and Flow capacities in Down-Hole Safety Valves", Journal of Petroleum Technology, Paper No. SPE-5161, (September, 1975).
- Aziz, K., Govier, G. W. and Forgasi, M. "Pressure Drop in Wells Producing Oil and Gas," J. Cdn. Pet. Tech. (July-Sept. 1972) 38-48.
- Babu, D. K. and Odeh, A. S. "Productivity of a Horizontal Well", SPE Reservoir Engineering (November 1989) 417-421.
- Baker, A., Nielsen, K., and Gabb, A. "Pressure Loss, Liquid-Holdup Calculations Developed," Technology, Oil & Gas Journal (Mar. 14, 1988).
- Baker, O. and Swerdloff, W. "Calculation of Surface Tension - 3";, Oil & Gas Journal (Dec. 5, 1955) 141.
- Beal, C. "The Viscosity of Air, Water, Natural Gas, Crude Oil and its Associated Gases at Oil Temperatures and Pressures", Trans. AIME (1946) 94.
- Beggs, H. D. "Gas Processing Operations", OCGI Publications, (1984), ISBN 0-930972-06-6
- Beggs, H. D., and Brill, J. P. "A Study of Two Phase Flow in Inclined Pipes", J. Pet. Tech. (May 1973) 607-617.
- Beggs, H. D. and Robinson, J. R. "Estimating the Viscosity of Crude Oil Systems", J. Pet. Tech. (Sept. 1975) 1140-1.
- Bellarby, Jonathan "Well Completion Design", Chapter 6, Elsevier, 2009.
- Bendakhlia, H. and Aziz, K. "Inflow Performance Relationships for Solution-Gas Drive Horizontal Wells", SPE paper 19823 presented at the Annual Technical Conference and Exhibition, San Antonio, (October 1989).
- Bendiksen et al. "The Dynamic Two-Fluid Model OLGA: Theory and Application," SPE Production Engineering, 171-180, May 1991.
- Bergman, D. F. and Sutton, R. P. "Undersaturated Oil Viscosity Correlation for Adverse Conditions", SPE 103144 (2006).

- Bhatti and Shah "Turbulent and Transition Flow Convective Heat Transfer in Ducts", Handbook of Single-Phase Convective Heat Transfer, ed. Kakac, Shah, Aung Wiley Interscience, NY (1987)
- Brauner, N. and Ullmann, A. "Modeling of Phase Inversion Phenomenon in Two-Phase Pipe Flows," Int. J. Multiphase Flow, 28, 1177, 2002.
- Brill, J. P. et al. "Analysis of Two-Phase Tests in Large Diameter Flow Lines in Prudhoe Bay Field", SPEJ (June 1981).
- Brill, J. P. and Beggs, D. H. "Two-Phase Flow in Pipes", 6th Edition, University of Tulsa, Tulsa, Oklahoma, (December 1988).
- Brill, J.P. and Mukherjee, H. "Multiphase Flow in Wells", SPE Monograph 17, (1999)
- Brons, F. and Marting,V. E. "The Effect of Restricted Fluid Entry on Well Productivity", Trans., AIME (1961) 222, 1972.
- Brown, T.S., Niesen, V.G., And Erickson, D.P. "Measurement and Prediction of Kinetics of Paraffin Deposition", SPE 26548, 68th Annual Technical Conference and Exhibition of SPE Houston, TX, 3-6 October, 1993.
- Brown, K.E. "The Technology of Artificial Methods", Penwell Publishing Company, Tulsa, Oklahoma, 1984.
- Butler, B. "Success grows in pumping high-gas-fraction multiphase fluids", Petroleum Engineer International, July 1999.
- Celier, G. C. M. R., Jouault, P. and de Montigny, O. A. M. C. "Zuidwal: A Gas Field Development With Horizontal Wells", SPE paper 19826 presented at the Annual Technical Conference and Exhibition in San Antonio, October 1989.
- Cheng, A.M. "Inflow Performance Relationships for Solution-Gas-Drive Slanted/Horizontal Wells", SPE paper 20720 presented at the Annual Technical Conference and Exhibition, New Orleans, September 1990.
- Chew, J. and Conally, C. A. Jr. "A Viscosity Correlation for Gas Saturated Crude Oils", Trans., AIME (1974) 23.
- Chu and Jones "Convective Heat Transfer Coefficient Studies in Upward and Downward, Vertical, Two-Phase, Non-Boiling Flows", AIChE Symp. Ser., vol. 76, 79-90 (1980).
- Cinco, H., Miller, F.G.,and Ramey H.J. "Unsteady-State Pressure Distribution created By a Directionally Drilled Well," JPT 5131, (1975)
- Cinco, H., Samaniego, V., and Dominguez A. "Transient Pressure Behavior for a Well With a Finite-Conductivity Vertical Fracture", SPE 6014, August 1978.

- Cinco-Ley, H., and Samaniego, F. "Transient Pressure Analysis for Fractured Wells", JPT, 1749 - 1766, September 1981a.
- Cinco-Ley, H., and Samaniego, F. "Transient Pressure Analysis: Finite Conductivity Fracture Case versus Damage Fracture Case", SPE Paper 10179, 1981b.
- Colburn "A Method of Correlating Forced Convection Heat Transfer Data and a Comparison With Fluid Friction", Trans. AIChE, 19, 174 (1933) [reprinted in Int. J. Heat Mass Transfer, 7, 1359 (1964)..
- Cooper, R.E., and Troncoso, J.C. "An Overview of Horizontal Well Completion Technology", SPE paper 17582 presented at the International Meeting on Petroleum Engineering, Tianjin, China, (November 1988).
- Crane Technical Paper 410 "Flow of fluids through valves, fittings and pipe", 1988, Crane Co.
- Dake, L.P. "Fundamentals of Reservoir Engineering", Elsevier Scientific Publishing Co., New York, (1978).
- De Ghetto, G. et al. "Reliability Analysis on PVT Correlations", SPE 28904 (1994).
- De Ghetto, G. et al. "Pressure-Volume-Temperature Correlations for Heavy and Extra Heavy Oils", SPE 30316 (1995).
- de Marolles, C. abd de Salis, J. "Innovations in multiphase hydrocarbon operations", www.pump-zone.com, (1999).
- C.de Waard, U. Lotz "Influence of liquid flow velocity on CO₂ corrosion: a semi-empirical model." CORROSION/95, Paper No. 128, NACE, 1995.
- C.de Waard, U. Lotz, Milliams, D.E "Predictive model for CO₂ Corrosion Engineering in Natural Gas Pipelines", Corrosion, Vol. 47, No. 12, Dec. 1991 (976-985).
- Dikken, B.J. "Pressure Drop in Horizontal Wells and its Effect on Production Performance", JPT (November 1990) 1426-1433.
- Dipprey and Sabersky "Heat and Momentum Transfer in Smooth and Rough Tubes at Various Prandtl Numbers", Int. J. Heat Mass Transfer, 6, 329 (1963)
- Dittus and Bolter "Heat Transfer in Automobile Radiators of the Tubular Type", University of California Pub. Eng., 2, 443 (1930).
- Drexel and McAdams "Heat Transfer Coefficients for Air Flowing in Round Tubes and Around Finned Cylinders", NACA ARR No. 4F28, [also Wartime Report W-108 (1945)].
- Dukler, E. A., et al. "Gas-Liquid Flow in Pipelines, I. Research Results", AGA-API Project NX-28 (May 1969).

- Duns, H., and Ros, N. C. J. "Vertical Flow of Gas and Liquid Mixtures in Wells", 6th. World Pet. Congress (1963) 452.
- Earlougher, R. C. Jr, "Advances in Well Test Analysis", SPE Monograph 5, (1977)
- Eaton, B. A. "Prediction of Flow Patterns, Liquid Holdup and Pressure Losses Occurring During Continuous Two-Phase Flow in Horizontal Pipelines", Trans., AIME (1967) 815.
- Economides, M.J., Hill, D.A., and Economides, C.E. "Petroleum Production Systems", PTR Prentice Hall, 1994.
- Economides, M.J., McLennan, J.D., Brown, E., and Roegiers, J.C. "Performance and Stimulation of Horizontal Wells", World Oil, (July 1989) 69-76.
- Eckert, E.R.G. and Jackson, T.W. "Analysis of Turbulent Free Convection Boundary Layer on a Flat Plate", NACA Report 1015 (July 1950)
- Elsharkawy, A.M. and Alikhan, A.A. "Models for predicting the viscosity of Middle East crude oils", Fuel 78 (1999), 891-903.
- Elamvaluthi and Srinivas "Two-Phase Heat Transfer in Two Component Vertical Flows", Int. J. Multiphase Flow, vol. 10, no. 2, p. 237-242 (1984).
- Fetkovich, M.J. "The Isochronal Testing of Oil Wells", SPE 4259, (1973).
- Fetkovich, M.J. and Vienot, M.E. "Shape Factors, CA, Expressed as a Skin, sCA", JPT (February 1985) 321-322.
- Flanigan, O. "Effect of Uphill Flow on Pressure Drop in Design of Two-Phase Gathering Systems", Oil and Gas J. (March 10, 1958) 56, 132.
- Folefac, A. N., Archer, J. S. and Issa, R. I. "Effect of Pressure Drop Along Horizontal Wellbores on Well Performance", SPE paper 23094 presented at the Offshore Europe Conference held in Aberdeen (September 1991).
- Forchheimer, Ph. "Wasserbevegung durch Boden", Z. Ver. Deutch. Ing., Vol. 45 (1901) p. 1781 - 1788
- Framo Webpage <http://www.framoeng.no/pump.php>
- Friend and Metzner "Turbulent Heat Transfer inside Tubes and the Analogy among Heat, Mass, and Momentum Transfer", AIChE J., 4, 393 (1958).
- Ghassan, H. A., and Maha, R. A. "Correlations developed to predict two-phase flow through wellhead chokes", The Journal of Canadian Petroleum technology, Volume 30, No 6, 1991

- Giger, F. M., Reiss, L. H., and Jourdan, A. P. "The Reservoir Engineering Aspects of Horizontal Drilling", SPE paper 13024 presented at the Annual Technical Conference and Exhibition in Houston, September 1984.
- Glasø, O. "Generalized Pressure Volume Temperature Correlation", J. Pet. Tech. (May 1980) 785.
- Golan, M. and Whitson, C.H. "Well Performance", International Human Resources Corporation, Boston, MA (1986).
- Gomez, L.E., Shoham, O., Schmidt, Z., Chokshi, R.N., and Northug, T. "Unified mechanistic model for steady-state two-phase flow: Horizontal to vertical upward flow". SPE Journal, 5(3):339-350, 2000.
- Goode, P. A. and Wilkinson, D. J. "Inflow Performance of Partially Open Horizontal Wells", SPE paper 19341 presented at the SPE Eastern Regional Meeting, Morgantown, WV, October, (1989).
- Gowen and Smith "Turbulent Heat Transfer from Smooth and Rough Surfaces", Int. J. Heat Mass Transfer, 11, 1657 (1968)
- Grolman, E. "Gas-Liquid Flow with Low Liquid Loading in Slightly Inclined Pipes", Ph.D. Dissertation, U. of Amsterdam, Amsterdam, The Netherlands (1994).
- Grolman, Commandeur, de Basst and Fortuin "Wavy-to-Slug Flow Transition in Slightly Inclined Gas-Liquid Pipe Flow", AIChE Journal, vol 42. No. 4, p901 (1996)
- Gurley, D. G., Copeland, C. T. and Hendrick, J. L. "Design Plan and Execution of Gravel-Pack Completion," J. Pet. Tech. (Oct. 1977).
- Hagedorn, A. R. and Brown, K. E.: "Experimental Study of Pressure Gradients Occurring During Continuous Two-Phase Flow in Small-Diameter Vertical Conduits", J. Pet. Tech. (April 1965) 475-484.
- Hausen "Neue Gleichungen fur die Wärmeübertragung bei freier oder erzwungener Stromung", Allg. Wärmetchn., 9, 75 (1959).
- Hawkins, M. F., Jr. "A Note on the Skin Effect", Trans. AIME, 207: 356-357, (1956).
- Hayduk, W. and Minhas, B.S. "Correlations for Predictions of Molecular Diffusivities in Liquids", The Canadian Journal of Chemical Engineering, 60, 295 (1982).
- Hernandez, O.C., Hensley, H., Sarica, C., Brill, J.P., Volk, M., Delle-Case, E. "Improvements in Single-Phase Paraffin Deposition Modeling", SPE Production and Facilities, 237, (2004).
- Hossain, M.S. et al. "Assessment and development of Heavy-Oil Viscosity Correlations", SPE/PS-CIM/CHOA 97907, PS2005-407.
- Hughmark, G. A. "Holdup in Gas Liquid Flow", Chem. Eng. Prog., v.58, p.62.

- Incropera, F. P. and DeWitt, D.P. "Fundamentals of Heat and Mass Transfer", Fourth Edition, Wiley, New York. (1996).
- ISO Standard 15136–1 "Petroleum and Natural Gas Industries – Progressing Cavity Pump Systems for Artificial Lift", Second Edition, 2009-11-15.
- Jones, L.G., Blount, E.M. and Glaze, O.H. "Use of Short Term Multiple Rate Flow Tests To Predict Performance of Wells Having Turbulence", SPE 6133 (1976).
- Jones, L. G. and Slusser, M. L. "The Estimation of Productivity Loss Caused by Perforation - Including Partial Completion and Formation Damage", SPE paper 4798 (1974).
- Joshi, S. D. "Horizontal Well Technology", Penwell Publishing Company, Tulsa, Oklahoma (1991).
- Joshi, S. D.: "A Review of Horizontal and Drainhole Technology", SPE paper 16868 presented at the Rocky Mountain Regional Meeting in Casper, WY (May 1988).
- Kaminsky, R. D. "Estimation of Two-Phase Flow Heat Transfer in Pipes", J Energy Res Tech, vol 121, p 75 (1999).
- Karakas, M. and Tariq, S,M. "Semianalytical Productivity Models for Perforated Completions", SPE 18247 (1991).
- Karassik, I.J., Messina, J.P., Cooper, P., Heald, C. C. "Pump Handbook," 3rd edition, McGraw-Hill Inc., 1991
- Karassik, I. J. "Pump Handbook," 3rd Edition. McGraw-Hill, New York, 2001.
- Karassik, J. et al. "Pump Handbook", 2nd edition, McGraw-Hill Inc., 1986
- Kartoatmodjo, R.S.T. and Schmidt, Z. "New Correlations For Crude Oil Physical Properties", SPE 23556 (1991).
- Katz, D. L. et al. "Handbook of Natural Gas Engineering", McGraw Hill Book Co., Inc., New York (1959).
- Kawase and Ulbrecht "Turbulent Heat and Mass Transfer in Dilute Polymer Solutions", Chem Eng Sci, 37, 1039 (1982)
- Kawase and De "Turbulent Heat and Mass Transfer in Dilute Polymer Solutions Flowing Through Rough Tubes", Int. J. Heat Mass Transfer, 27, 140 (1984)
- Kern, D.Q. and Seaton, R.E. "A Theoretical Analysis of Thermal Surface Fouling", British Chemical Engineering, Brit. Chem. Eng. 4, 258 (1959).

Khoze, Dunayev, Sparin	"Heat and Mass Transfer in Rising Two-Phase Flows in Rectangular Channels", Heat Transfer-Sov. Res., vol. 8, no. 3, p 87-90, (1976)
King	"Heat Transfer and Pressure Drop for an Air-Water Mixture Flowing in a 0.737 Inch I.D. Horizontal Tube", MS Thesis, University of California, Berkeley, CA, (1952).
Knott et al.	"An experimental Study of Heat Transfer to Nitrogen-Oil Mixtures", Ind. Eng. Chem., vol. 51, no. 11, p. 1369-1372 (1959).
Kreith, F. and Bohn, M.	"Principles of Heat Transfer", 5th Edition, PWS Publishing Company (1997).
Kudirka, Grosh, McFadden	"Heat Transfer in Two-Phase Flow of Gas-Liquid Mixtures", I&EC Fundamentals, vol 4, no 3, p 339-344 (1965)
Kunz, Klimeck	The GERG-2004 wide-range equation of state for natural gases and other mixtures. GERG TM15 2007. Fortschr.-Ber. VDI, Reihe 6, Nr. 557, VDI Verlag, Düsseldorf, 2007; also available as GERG Technical Monograph 15 (2007).
Kunz, Wagner	The GERG-2008 wide-range equation of state for natural gases and other mixtures: An expansion of GERG-2004. To be submitted to J. Chem. Eng. Data (2011).
Lasater, J. A.:	"Bubble Point Pressure Correlation", Trans., AIME (1958) 379.
Lee, A. L., Gonzales. M.H. and Eakin, B.E.	"The Viscosity of Natural Gases", Trans., AIME (1966), 237, 997-1000.
Lockhart, R. W. and Martinelli, R. C.	"Proposed Correlation of Data for Isothermal Two-phase, Two-Component Flow in Pipes", Chem. Eng. Prog. (January 1949) 45, 39.
Martin and Sims	"Forced Convection Heat Transfer to Water with Air Injection in a Rectangular Duct", Int. J. Heat Mass Transfer, vol 14, p 1115-1134, (1971).
Matzain, A.	"Multiphase Flow Paraffin Deposition Modeling", Ph.D. Dissertation, U. Tulsa, Tulsa, OK (2001).
Matzain, A., Apte, M.S., Zhang, H.-Q., Volk, M., Redus, C.L., Brill, J.P., Creek, J.L.	"Multiphase Flow Wax Deposition Modeling", Proceedings of ETCE 2001, ETCE2001-17114, 927 (2001).
Matzain, A., Apte, M.S., Zhang, H.-Q., Volk, M., Brill, J.P., Creek, J.L.	"Investigation of Paraffin Deposition During Multiphase Flow in Pipelines and Wellbores-Part 1: Experiments", Transactions of the ASME, 124, 180 (2002).

- McAdams, W.H. "Heat Transmission", McGraw Hill, New York (1954).
- McCain, William D. Jr. "Petroleum Fluids", edition 2.
- McLeod, H. O. "The Effect of Perforating Conditions on Well Performance", JPT (Jan. 1983).
- Manabe, R. "A comprehensive heat transfer model for two phase flow with high pressure flow pattern validation", University of Tulsa, (2001).
- Manhane, J. M., Gregory, G. A. and Aziz, K. "A Flow Pattern Map for Gas-Liquid Flow Pattern in Horizontal Pipes", Int. J. of Multiphase Flow.
- Martinelli "Heat Transfer to Molten Metals", Trans. ASME, 69, 947 (1947).
- Minami, K. and Brill, J. P. "Liquid Holdup in Wet Gas Pipelines", SPE J. Prod. Eng. (May 1987).
- Mirza, K. Z. "Progressing cavity multiphase pumping systems: expanding the possibilities", presented at BHR Conference Multiphase, 1999
- Moody, L. "An approximate Formula for Pipe Friction Factors", Transactions ASME, vol 69, pp. 1005 (1947)
- Mukherjee, H. and Brill, J. P. "Liquid Holdup Correlations for Inclined Two-Phase Flow", JPT (May 1983) 1003-1008.
- Mukherjee, H. and Economides, M. J. "A Parametric Comparison of Horizontal and Vertical Well Performance", SPE paper 18303 presented at the Annual Technical Conference and Exhibition in Houston, October 1988.
- Muskat, M. "The Flow of Homogeneous Fluids Through Porous Media", I.H.R.D.C., Boston (1937).
- Mutalik, P. N., Godbole, S. P. and Joshi, S. D. "Effect of Drainage Area Shapes on Horizontal Well Productivity", SPE paper 18301 presented at the Annual Technical Conference and Exhibition, Houston (October 1988).
- Norris, L. "Correlation of Prudhoe Bay Liquid Slug Lengths and Holdups Including 1981 Large Diameter Flowlines Tests", Internal Report Exxon (October 1982).
- Nunner "Wärmeübergang und Druckabfall in Rauhen Rohren", VDI-Forschungsheft 445, ser B, 22, 5 (1956) or 786-Atomic Energy Research Establishment, Harwell UK
- Oliemans, R. V. A. "Two-Phase Flow in Gas-Transmission Pipeline", ASME paper 76-Pet-25, presented at Pet. Div. ASME meeting Mexico City (Sept. 1976).

- Oliver and Wright "Pressure Drop and Heat Transfer in Gas-Liquid Slug Flow in Horizontal Tubes", Bri. Chem. Eng., vol. 9, no. 9, p 590-596 (1964)
- Omana, R. et al. "Multiphase Flow Through Chokes", SPE 2682, 1969
- Orkiszewski, J. "Predicting Two-Phase Pressure Drops in Vertical Pipes", J. Pet. Tech. (June 1967) 829-838.
- Ovworrie, Chukwuemeka "Steady-State Heat Transfer Models For Fully And Partially Buried Pipelines," SPE 131137, International Oil and Gas Conference and Exhibition in Beijing, China, 8-10 June 2010.
- Oxley, K. C., Ward, J.M. and Derks, W.G. "How multiphase pumping can make you money", presented at Facilities 2000 Conference, New Orleans 1999
- Palmer, C. M. "Evaluation of Inclined Pipe Two-Phase Liquid Holdup Correlations Using Experimental Data", M.S. Thesis, The University of Tulsa (1975).
- Pan, S., Zhu, J., Zhang, D., Razouki, A., Talbot, M, and Wierzchowski, S. "Case Studies on Simulation of Wax Deposition in Pipelines", IPTC-13420-PP, 7-9 December 2009
- Park, H. Y., Falcone, G. and Teodoriu, C. "Decision Matrix for Liquid Loading in Gas Wells for Cost/Benefit Analyses of Lifting Options," Journal of Natural Gas Science and Engineering 1 (2009) 72-83.
- Payne, G. A.: "Experimental Evaluation of Two-Phase Pressure Loss Correlations for Inclined Pipe", M.S. Thesis, The University of Tulsa (1975).
- Poettman, F. H. and Beck, R. L. "New Charts Developed to Predict Gas-Liquid Flow Through Chokes", World Oil, March 1963, 95-101
- Prandtl, L. "Furrrer durch die Stomungslehre", Vieweg, Braunschweig, p 359 (1944)
- Petrosky, G.E. Jr. and Farshad, F .F. "Pressure-Volume-Temperature Correlations for Gulf of Mexico Crude Oils", SPE 51395 (1998)
- Petukhov and Gnielinski Int. Chem. Eng., 16-2, p 359, (1976)
- Petukhov and Kirillov "Heat Transfer and Friction in Turbulent Pipe Flow with Variable Physical Properties", Adv. Heat Tranfer, vol. 6, p 505-564 (1970)
- Poling, B.E., Prausnitz, J.M. and O'Connell, J.P. "The Properties of Gases and Liquids", Fifth Edition, McGraw-Hill
- Pots, B. F. M., Bromilow, I. G. and Konijn, M. J. W. "Severe Slug Flow on Offshore Flowline/Riser Systems", SPE paper 13723, (March 1985).

- Prats, M. "Effect of Vertical Fractures on Reservoir Behavior-Incompressible Fluid Case", SPEJ, pp. 105-118, June 1961.
- Pucknell, J.K. and Mason J.N.E. "Predicting the Pressure Drop in a Cased-Hole Gravel Pack Completion", SPE 24984, 1992.
- Ramey, H.J. "Wellbore Heat Transmission", JPT 435 Trans AIME, No. 225 (April 1962)
- Ravipudi and Godbold "The Effect of Mass Transfer on Heat Transfer Rates for Two-Phase Flow in a Vertical Pipe", Proc. 6th Int. Heat Transfer Conf.- Toronto, vol. 1, p 505-510, (1978)
- Rawlins, E.L. and Schellhardt, M.A, "Backpressure Data on Natural Gas Wells and Their Application to Production Practices", Monograph Series, USBM 7, (1935).
- Renard, G. I. and Dupuy, J. M. "Influence of Formation Damage on the Flow Efficiency of Horizontal Wells", SPE paper 19414 presented at the Formation Damage Control Symposium, Lafayette (February 1990).
- Rezkallah and Sims "An Examination of Correlations of Mean Heat Transfer Coefficients in Two-Phase and Two-Component Flow in Vertical Tubes", AIChE Symp. Ser., vol. 83, pp. 109-114, 1987.
- Romero, D.J., Valko, P.P., and Economides, M.J. "The Optimization Of The Productivity Index And The Fracture Geometry Of A Stimulated Well With Fracture Face And Choke Skins", SPE 73758, 2002.
- Sandall et al. "A New Theoretical Formula for Turbulent Heat and Mass Transfer with Gases or Liquids in Tube Flow", Can. J. Chem. Eng., 58, 443 (1980)
- Scandpower PT Tech Note TN-11 (internal) "TN-11: Heat Transfer Calculations", Scandpower PT, Oslo (February 2003).
- Shell report SIEP 98-5463 "Satellite multiphase boosting - Multiphase boosting study", Siep-RTS, ABB Lummus Global
- Scott, S. L., Shoham, O., and Brill, J. P. "Prediction of Slug Length in Horizontal Large-Diameter Pipes", SPE paper 15103 (April 1986).
- Schlunder, E.U. (ed) VDI Wärmeatlas, VDI-Verlag, Düsseldorf (1984).
- Schneider, G. E. "An Investigation Into Heat Loss Characteristics of Buried Pipes", Journal of Heat Transfer Vol. 107 (pp696-699) (Aug 1985).
- Shah "Generalized Prediction of Heat Transfer during Two Component Gas-Liquid Flow in Tubes and Other Channels", AIChE Symp. Ser., vol. 77, no. 208, p. 140-151, (1981).

- Sieder and Tate "Heat Transfer and Pressure Drop of Liquids in Tubes", Ind. Eng. Chem., 28, 1429 - 1453 (April 1936).
- Singh, P. "Gel Deposition on Cold Surfaces", U. of Michigan, Ann Arbor, Michigan (2000).
- SPE Production Engineering Handbook Volume IV, Chapter 15, Society of Petroleum Engineers, 2007.
- Sonnad, J. and Goudar, C. "Explicit Reformulation of the Colebrook-White Equation for turbulent Flow friction Factor calculation", Ind. Eng. Chem. Res, 46, pp. 2593-2600, (2007)
- Span, R. and Wagner, W. "A New Equation of State for Carbon Dioxide Covering the Fluid Region from the Triple-Point Temperature to 1100 K at Pressures up to 800 MPa," J. Phys. Chem. Ref. Data, 25(6):1509-1596, 1996.
- Standing, M. B. "Volumetric and Phase Behavior of Oil Field Hydrocarbon Systems", Society of Petroleum Engineers, (1977) 121.
- Standing, M. B. "A General Pressure Volume-Temperature Correlation for Mixtures Of California Oils and Greases," Drill. and Prod. Prac., API (1947) 275.
- Standing, M. B. and Katz, D. L. "Volumetric and Density of Natural Gases", Trans., AIME (1942) 140.
- Taitel, Y. and Dukler, A. E. "A Model for Predicting Flow Regime Transitions in Horizontal Gas-Liquid Flow", AICHE J. (vol. 22, no. 1) (Jan. 1976) 47-55.
- Vasquez, M., and Beggs, H. D. "Correlations for Fluid Physical Property Prediction", SPE paper 6719, presented at the 52nd Annual Technical Conference and Exhibition of the Society of Petroleum Engineers, Denver, Colorado (1977).
- Venkatesan, R. "The Deposition and Rheology of Organic Gels", U. of Michigan, Ann Arbor, Michigan (2004).
- Vijay, Aggour, and Sims "A Correlation of Mean Heat Transfer Coefficients for Two-Phase Two-Component Flow in a Vertical Tube", Proc. 7th Int. Heat Transfer Conf., vol. 5, p. 367-372 (1982).
- von Karman, "The Analogy Between Fluid Friction and Heat Transfer", Trans. ASME, 61, 705 (1939).
- Vogel, J. V. "Inflow performance relationships for solution gas drive wells", Journal Pet. Tech., SPE 1476, (1968)
- Webb "A Critical Evaluation of Analytical Solutions and Reynolds Analogy Equations for Turbulent Heat and Mass Transfer in Short Tubes", Wärme-und Stoffübertrag, 4, 197 (1971).

- Woelflin, W. "The Viscosity of Crude-Oil Emulsions", Drill. and Prod. Prac., API (1942) 148.
- Xiao, J.J., Shoham, O., and Brill J.P. "A comprehensive mechanistic model for two-phase flow". In Proceedings of the 65th SPE Annual Technical Conference and Exhibition, pages 167-180, 1990.
- Zhang, H.Q., Wang, Q., Sarica, C., Brill, J.P. "Unified Model for Gas-Liquid Pipe Flow Via Slug Dynamics - Part 1: Model Development", ASME Journal of Energy Resources Technology, Vol.125 (December 2003) 274.
- Zhang, H.-Q., Wang, Q., Sarica, C. and Brill, J.P. "Unified Model for Gas-Liquid Pipe Flow Via Slug Dynamics - Part 2: Verification", ASME Journal of Energy Resources Technology, Vol.125 (December 2003) 266.
- Zhang, H.-Q. and Sarica C. "Unified Modeling of Gas/Oil/Water Pipe Flow – Basic Approaches and Preliminary Validation," SPE Project Facilities & Construction 1(2), pp. 1- 7, 2006.

5

Keyword Index

[Input Files and Input Data Conventions \(p.599\)](#)

[General Data \(p.606\)](#)

[Compositional Data \(p.729\)](#)

[Blackoil Data \(p.717\)](#)

[Heat Transfer Data \(p.707\)](#)

[Flow Correlation Data \(p.638\)](#)

[System and Equipment Data \(p.672\)](#)

[Well Performance Modeling \(p.650\)](#)

[PIPESIM Operations \(p.738\)](#)

[PIPESIM-Net Keywords \(p.759\)](#)

5.1 Keyword List

[A \(p.595\)](#) [B \(p.595\)](#) [C \(p.596\)](#) [D \(p.596\)](#) [E \(p.596\)](#) [F \(p.596\)](#) [G \(p.596\)](#) [H \(p.597\)](#) [I \(p.597\)](#) [J \(p.597\)](#) [K \(p.597\)](#) [L \(p.597\)](#) [M \(p.597\)](#) [N \(p.598\)](#) [O \(p.598\)](#) [P \(p.598\)](#) [Q \(p.598\)](#) [R \(p.598\)](#) [S \(p.598\)](#) [T \(p.599\)](#) [U \(p.599\)](#) [V \(p.599\)](#) [W \(p.599\)](#) [X \(p.599\)](#) [Y \(p.599\)](#) [Z \(p.599\)](#)

5.1.1 A

- [ASSIGN \(p.751\)](#)

5.1.2 B

- [BACKPRES \(p.664\)](#)
- [BEGIN \(p.634\)](#)
- [BLACKOIL \(p.717\)](#)
- [BRANCH \(p.762\)](#)

5.1.3 C

- CASE (p.608)
- CALIBRATE (p.727)
- CHOKE (p.673)
- COAT (p.710)
- COMP (p.732)
- COMPCRV (p.677)
- COMPLETION (p.651)
- COMPRESSOR (p.679)
- CONETAB (p.663)
- CONFIG (p.715)
- CONTAMINANTS (p.728)
- CORROSION (p.639)
- CPFLUID (p.726)

5.1.4 D E

- END (p.634)
- ENDCASE (p.634)
- ENDJOB (p.634)
- EROSION (p.639)
- EQUIPMENT (p.443)
- ESP (p.701)
- EXPANDER (p.683)

5.1.5 F

- FETKOVICH (p.654)
- FITTING (p.684)
- FLOWLINE (p.651)
- FMPUMP (p.686)
- FORCHHEIMER (p.670)
- FRACTURE (p.670)
- FRAMO2009 (p.686)

5.1.6 G

- GASLIFT (p.687)

5.1.7 H

- HCORR (p.645)
- HEATER (p.687)
- HEADER (p.607)
- HEAT (p.707)
- HORWELL (p.664)
- HVOGEL (p.670)

5.1.8 I

- IFPPSSE (p.655)
- IFPTAB (p.662)
- IFPCRV (p.660)
- INLET (p.623)
- INJGAS (p.693)
- INJFLUID (p.693)
- INJPORT (p.691)
- IPRCRV (p.660)
- ITERN (p.621)

5.1.9 J

- JOB (p.607)
- JONES (p.655)
- JUNCTION (p.769)

5.1.10 K

- KCOAT (p.712)

5.1.11 L

- LAYER (p.666)
- LVIS (p.721)

5.1.12 M

- MPBOOSTER (p.695)
- MPUMP (p.696)
- MULTICASE (p.743)

5.1.13 N

- NAPLOT (p.739)
- NAPOINT (p.743)
- NODE (p.698)
- NOPRINT (p.634)
- NSEPARATOR (p.769)

5.1.14 O

- OPTIONS (p.609)
- OPTIMIZE (p.752)

5.1.15 P

- PCP (p.701)
- PERMCRV (p.668)
- PERMTAB (p.669)
- PETROFRAC (p.737)
- PIPE (p.699)
- PLOT (p.631)
- PRINT (p.624)
- PROP (p.719)
- PUMP (p.701)
- PUMPCRV (p.677)
- PUSH (p.636)

5.1.16 Q R

- RATE - Compositional (p.738)
- RATE - Blackoil (p.619)
- REINJECTOR (p.704)
- RISER (p.651)

5.1.17 S

- SEPARATOR (p.705)
- SETUP (p.760)
- SINK (p.767)
- SLUG (p.640)

- [SOURCE](#) (p.764)
- [SPHASE](#) (p.648)

5.1.18 T

- [TABLE](#) (p.750)
- [TCOAT](#) (p.711)
- [TIME](#) (p.205)
- [TPRINT - Compositional](#) (p.738)
- [TPRINT - Blackoil](#) (p.726)
- [TRANSIENT](#) (p.671)
- [TUBING](#) (p.651)

5.1.19 U

- [UNITS](#) (p.608)
- [USERDLL - Flow Correlations](#) (p.650)
- [USERDLL - Equipment](#) (p.638)

5.1.20 V

- [VCORR](#) (p.641)
- [VOGEL](#) (p.654)

5.1.21 W

- [Wax Deposition](#) (p.205)
- [WELLHEAD](#) (p.706)
- [WELLPI](#) (p.653)
- [WCOPTION](#) (p.657)
- [WPCURVE](#) (p.654)

5.1.22 XYZ

5.2 Input Files and Input Data Conventions

5.2.1 General

The PIPESIM engine input processor accepts data under a system of main-code and sub-code keywords. The entire input data file is checked for syntax errors before program execution begins; if any input errors are detected, diagnostics are written to the terminal (for interactive jobs) and to the Input Data Echo in the main job output, and program execution is halted.

5.2.2 Statements

Data is entered in statements. Each statement must begin with a main-code keyword (unless it is a comment card or a blank line), and is followed by sub-code keywords and equated values appropriate to the maincode. Statements are usually entered one per line in the file; but if desired, a statement can be split across multiple lines, or multiple statements can be provided on one line. In all cases there is a limit of 256 characters per statement, and per line, including spaces.

Statements are composed of printable text characters. Valid characters are those in the 7- or 8-bit ASCII set between decimal values 32 (space, []) and 126 (tilde, [~]). Some computers or installations will generate characters outside this range, due to differences in national language alphabets and punctuation. Usually this will not cause any problems but this cannot be guaranteed.

(Files containing ASCII codes greater than 128 or less than 32 are often created using a word-processing program, because these characters are used as formatting instructions to produce a correctly formatted printed page. For PIPESIM however they are not required and will sometimes cause the program to generate large numbers of syntax errors. Use of such programs for preparing input files is best avoided, you should use a text editor instead; alternatively acceptable results may be obtained by requesting the word-processor to produce ASCII text files as output.)

Upper and lower case can be freely mixed, except where doing so would cause the computer system to assign a different meaning to the data. All Maincodes and Subcodes described in this document are case-insensitive, but for example some computer systems are case-sensitive in filenames, so where these are specified care must be taken to provide the correct case.

5.2.3 Delimiters

There are a number of characters reserved for use as delimiters. In general these can only be used for the purpose described, however if a reserved delimiter character is required for use in a character string, the string can be quoted with apostrophes or double quotes. The delimiters (with ASCII decimal codes) are:

- [] One or more blanks or spaces can be used to delimit main-code and sub-code entries, and to improve readability in conjunction with other delimiter characters. (ASCII 32)
- [,] A comma (with or without one or more blanks) can also be used to delimit main-code and subcode entries, but its main use is to delimit data items when a multiple set of values is provided with parentheses (see below). (ASCII 46)
- [=] An equal's sign is used to separate each subcode from its associated numeric or character data. Additional spaces may be inserted either side of the equals to improve readability. Some subcodes do not require values; if no value is provided the equals must be omitted. (ASCII 61)
- ['] Apostrophes (also known as closing single quotes) can be used in matching pairs to delimit character data which itself contains delimiter characters, e.g. embedded blanks. (Do not confuse apostrophe with the opening single quote['], and do not attempt to match opening and closing single quotes with one another.) (ASCII 39)
- ["] Double quotes can be used in matching pairs as an alternative to apostrophes in delimiting character data. This is useful when the character string contains one or more single quotes. (Note that double quote is itself a single character or keystroke, and is not equivalent to two single quotes.) (ASCII 34)

- [\$] or [!] Either a dollar sign or an exclamation point is used to delimit end-of-line comments from input data. All characters on the line between the comment delimiter and end-of-line will be ignored. (ASCII 36 and 33)
- [() ()] Parentheses are used to enclose Multiple Value Data Sets when 2 or more values are provided for a subcode. Supplied data values should be separated with commas. (Additional separators are valid in multiple data sets, see below.) (ASCII 40 and 41)
- [&] The ampersand is used to continue a statement across two or more lines. It is placed as the last character on a line to specify that the statement continues on the next line. There is no limit to the number of continuation lines, but the complete statement cannot span more than 256 characters, including spaces. The ampersand should appear between subcodes. Continued lines can be separated with blank lines, but not with comment lines. (ASCII 38)
- [;] The semicolon is used to separate multiple statements provided on a single line. (ASCII 59)

Examples

```
RATE LIQUID = 6000 GLR = 400 WCUT = 20
RATE, LIQ=6000, GLR=400, WCUT=20
MULTICASE LIQ = ( 200, 250, 300 ) GLR = ( 95, 105 )
HEADER PROJECT=TEST, USER=J. BLOGGS
HEADER PROJECT=TEST, USER='JOE BLOGGS'
$ This line is a comment and will be ignored
! This line is also a comment
MULTICASE LIQ=(200.250,300) GLR=(95,105) ! This is an end-of-line comment
RATE, LIQ=6000, GLR=400, WCUT=20 $ This is also an end-of-line comment
MULTICASE LIQ=(200.250,300) & ! This statement is continued on the next line
GLR=(95,105)
! the next line contains two statements
RATE, LIQ=6000, GLR=400, WCUT=20 ; MULTICASE
LIQ=(200,250,300) GLR=(95,105)
```

5.2.4 Abbreviations

Main-code and sub-code keywords can be abbreviated down to the minimum number of letters required to make them unique in their context. For maincodes the context is all other main-codes; thus for example the GASLIFT maincode can be abbreviated to G because no other maincode starts with G, but COMP is an illegal abbreviated maincode because it matches COMPRESSOR, COMPLETION and COMPOSITION. The context for subcodes is restricted to the set of legal subcodes for the maincode concerned. If the keyword is abbreviated too much, the input processor will generate a syntax error and processing will terminate.

Example

For example the following 2 lines are equivalent:

```
OPTIONS SEGMENTS=10
OPT S=10
```

5.2.5 Numeric data

Except where noted, all numeric data must be equated to a preceding subcode with an equals sign [=]. Decimal points are optional, if provided only one is allowed and it must be the period or full stop character, [.] (ASCII 46). Large and small values may carry an exponent, for example, 1200000 may be written as 1.2e6 (or 0.12E7, .12e7, 1.2+6, 1.2e+6, 1.2D6, and so on). For example 0.000034 may be written as 3.4e-5 (or 0.34e-6, and so on). Do not embed spaces or commas in numeric data; they will be interpreted as delimiters signaling the end of the data item, and the remaining digits will then cause a syntax error.

Some main-codes allow data to be provided without keywords, in a strict positional order. Usually this is to allow easy entry of tabular data. Examples are [PERMTAB \(p.669\)](#), [CONE TAB \(p.663\)](#), [IFPTAB \(p.662\)](#).

Example

In the following example, all lines are equivalent:

```
RATE MASS=224.0
RATE MASS=.224e+3
RATE MASS=224
```

5.2.6 Units Description

A Units Description String can accompany numeric data. Such strings are the best way to provide data in units different to the defaults established with the UNITS statement. The string must contain no embedded blanks or other recognized delimiters. It can appear after the data to which it refers, or between the keyword and the equals. For example:

```
RATE LIQ = 3000 bbl/day WCUT = 55 % GOR = 300 scf/bbl
RATE LIQ bbl/day = 3000 wcut % = 55 GOR scf/bbl =
300
```

Multiple value data sets equated to non-symbolic subcodes can also accept unit's description strings placed outside the parentheses, for example:

```
PUMPCRV head = (300, 250, 200, 150, 100, 50) kj/kg
```

Symbolic subcodes (eg ?ALPHA, ?BETA on Multicase) will not allow units description strings. Instead, the description can be placed on the line where the symbol is used, for example:

```
MULTICASE ?ALPHA=(20,30,40)
PUMP DP = ?ALPHA kg/cm2
```

5.2.7 Character Input

Character data should be enclosed in a single or double quotes if it contains reserved delimiter characters (for example embedded blanks). It must be equated to a sub-code with an [=] sign.

Example

For example:

```
HCORR PLOSS = BJA HOLOUP = BJA MAP = TD
NODE dist=0 elev = 0 t=100 u=0.8 label = "Station Z' to
J2"
```

5.2.8 Comment Statements and Blank Lines

Any information to the right of a comment delimiter [\$ or !] is ignored. Lines beginning with a comment delimiter sign will be ignored completely and can be entered at any location in the input data file.

Blank lines are also ignored and so may be used to improve layout and readability.

Example

```
$ ----- THIS LINE WILL BE IGNORED -----
RATE, MASS=224 $ THIS IS AN IN-LINE COMMENT
```

5.2.9 Multiple Value Data Sets

Some subcodes accept more than one value, and for these an explicit syntax is available using parentheses. At its simplest the Multiple Value Data Set is a 2 or more values, separated by commas, and enclosed by parentheses. For example:

```
MULTICASE LIQ=(200,250,300) GLR=(95,105)
CHOKE dbean = 0.5 ccorr = (pratio, sonicup, flowrate)
DOVCORR uovcorr=table temps=(80,100,120,140)
viscscs=(6.5,5,2,1.5)
PRINT CUSTOM=(g3,h3,i3,k1,l1,m4,g4,f4,h11,i11,j11,m11)
PLOT CASE=(+,f7,g7,h7)
```

Multiple value sets can become very long, and care should be taken to avoid the maximum statement limit of 256 characters.

A range of data values can be specified for numeric data, this is a convenient alternative to entering long strings of explicit values. The syntax is (start : finish ; iop increment), which specifies a Starting value, a Finishing value, an Increment Operator, and an Increment Value. The special characters used are:

colon [:] separates the starting value from the finishing value

semicolon [:] separates the finishing value from the increment operator

plus [+] the addition operator

minus [-] the subtraction operator

asterisk [*] the multiplication operator

hash/sharp/number sign the enumeration operator.
[#]

Note: On computers outside the USA, the hash character will sometimes display as the national currency symbol. The required ASCII code is 35 decimal.

Examples

Some examples:

Example 1

To specify values from 10 to 100 by repeated addition of 5:

```
(10:100;+5) .
```

Example 2

To specify values from 10 to 100 in 50 equal-sized steps:

```
(10:100;#50)
```

Example 3

To specify values from 10 to 100 by repeated multiplication by 1.5:

```
(10:100;*1.5)
```

Example 4

To specify values from 100 to 10 by repeated subtraction of 7:

```
(100:10;-7)
```

Example 5

The range syntax can appear many times and be combined with other values, E.g.:

```
MULTICASE GLR=( 0, 1:1000;*1.5, 1200, 1600, 2000:10000;+1000 )
```

5.2.10 Input Files

General

The input data may appear in more than one file. At least one file is required, this may contain explicit references to further files if desired. In addition certain other files, if they exist, will be automatically read and processed in addition to your main input file.

The main input ('.PSM' or '.PST') file

Conventionally, the main input file has a filename with an extension of '.PSM'. It is specified on the engine command line or in answer to the engine prompt 'INPUT FILE NAME:'. In fact however any extension can be chosen, and will be used as specified. Beware however that word-processing programs can generate files with embedded formatting characters (that the engine will not recognize) if certain extensions are used, for example .TXT, .DOC. You are advised to use .PSM as the extension for all PIPESIM keyword main input files.

Note: The PIPESIM Graphical User Interface (GUI) program generates temporary engine keyword files with the extension .PST. All such files are assumed by it to be volatile, so if you choose to create files with a .PST extension, they are likely to be overwritten with no warning. Use .PSM instead.

Included files and the INCLUDE statement

Input data can be explicitly split among 2 or more files by use of the INCLUDE statement. INCLUDE has no subcodes, and the only value allowed is the name of the file to be included. The contents of the included file will be processed as though it appeared in the input file instead of the INCLUDE statement. For example if the file 'oil23.inc' contains the following:

```
UNITS in=eng
BLACKOIL
PROP gassg=.68 watersg=1.05 api=37.6
PROP psat=600 psia tsat=120 F GSAT=370
RATE wcum=20 GOR=320
```

This file can be referenced from the main input file by use of an INCLUDE at the appropriate point, for example:

```
INCLUDE oil23.inc
```

The Included file is assumed to reside in the same directory as the main input file; if this is not the case a path can be provided, such as:

```
include ..\..\proj-45\common\oil23.inc
```

Filenames containing spaces and other delimiter characters must be quoted, for example:

```
INCLUDE "k:\my special projects\pipesim files\my common
files\proj-45\common\oil type 23c.inc"
```

Included files can themselves contain INCLUDE statements, and such nested includes can go to a maximum of 10 levels. Take care to ensure such an include nest does not attempt to include a file that has already been included.

All Included files should specify a UNITS statement before any numeric data is supplied. Failure to do this is not an error, but the interpretation of the contents of the included file will then depend on the units in force in the main input file at the point of the INCLUDE statement. This is an unsafe situation, and can lead to unforeseen errors, which do not necessarily manifest themselves immediately. Any UNITS statement in an included file will only affect data in that file, and will not be

remembered when processing returns to the main input file. Thus an included file cannot be used to alter units settings in the main input file.

AUTOEXEC.PSM

This is a special include file that, if it exists in the same directory as the main input file, is automatically included for processing as though an INCLUDE statement referenced it. The file is named after the MS-DOS control file AUTOEXEC.BAT because of the obvious parallels between them. Note however that, while AUTOEXEC.BAT must reside in the root directory of a DOS boot disk to do its job, AUTOEXEC.PSM must instead reside in the same directory as the engine main input file. It is therefore possible to have many different autoexec.psm files in different directories of a computer's file system.

modelname.U2P or branchname.U2P

The .U2P file is a file whose contents are defined identically to AUTOEXEC.PSM, but whose applicability is limited to just one main input file name. This is specified by matching the rootnames of the files. For example if the main input file is called **fred.psm**, the matching .u2p file is **fred.u2p**.

5.3 General Data

[HEADER \(p.607\)](#) Job Accounting Header

[JOB \(p.607\)](#) Job Title

[CASE \(p.608\)](#) Case Title

[UNITS \(p.608\)](#) Input and Output Units

[OPTIONS \(p.609\)](#) Calculation Procedure Options

[RATE \(p.619\)](#) Fluid Flow Rate Data

[ITERN \(p.621\)](#) Iteration Data (Optional)

[INLET \(p.623\)](#) System Inlet Data

[PRINT \(p.624\)](#) Output Printing Options

[NOPRINT \(p.634\)](#) Output Print Suppression Options

[PLOT \(p.631\)](#) Output Plotting Options

[BEGIN, END \(p.634\)](#) Block delimiters

[PUSH \(p.636\)](#) Remote action editing

[PLOT FILE DATA \(p.637\)](#)

[EXECUTE \(p.637\)](#)

[USERDLL \(p.638\)](#) Equipment

5.3.1 Changing Parameters within the System Profile

One of the features of PIPESIM which gives a great deal of flexibility is the ability to change any parameter, for example, flow rate or fluid property, at any point (node) in the system. In fact, almost any main-code can be inserted at any point in the System Profile, that is the cards between the first

NODE card and the ENDCASE card. There are one or two exceptions where the main-code must appear before the first NODE card (for example ITERN main-code) and these are documented in the relevant sections. This feature allows changes of pipe diameter, fluid inflow and outflow, and so on, to be easily modeled.

The main-code to change parameters within the System Profile should be inserted after the NODE card at which it is to take effect. There is no limit to the number of parameters which can be changed at a particular node.

Example

Example:

A pressure control valve is located at a position 2000' down a flowline and sets the downstream pressure to 800 psia. There is a change of pipe diameter (to 6"), and another flowline from a similar well joins thus doubling the flow rate.

Multiple Cases

If multiple cases are to be considered, where the same feature within the profile is to be repeated but with a different value assigned to it, then the user has a choice. Either the whole profile may be repeated or an [ASSIGN \(p.751\)](#) card may be used to avoid repetition of the profile.

Note: The [MULTICASE \(p.743\)](#) card provides a convenient alternative to the use of repeated ENDCASE cards.

5.3.2 HEADER - Job Accounting Header (Required)

Main-code: HEADER

The HEADER card must be the first card in a job and must contain both a PROJECT and USER sub-code.

PROJECT= Project name (12 characters maximum) which should be entered in quotes if the string contains delimiters (such as blanks or commas).

USER= User name (12 characters maximum) which should be entered in quotes if the string contains delimiters (such as blanks or commas).

PASSWORD= Password (12 characters maximum) which should be entered in quotes if the string contains delimiters (such as blanks or commas).

Example

```
HEADER PROJ=TEST USER='JOE BLOGGS'
```

5.3.3 JOB - Job Title (Optional)

Main-code: JOB

JOB Job title (70 characters maximum). Quotes are not required even if the title string includes delimiters.

5.3.4 CASE - Case Title (Optional)

Main-code: CASE

CASE Case title (70 characters maximum). Quotes are not required even if the title string includes delimiters.

5.3.5 UNITS - Input and Output Units (Optional)

Main-code: UNITS

INPUT= Specifies the units in the input file

SI Input data in SI units (default).

ENG Input data in engineering units

OUTPUT= Specifies the units in the output file

SI Output data in SI units (default).

ENG Output data in engineering units

ALL= Specifies the units in both the input and output files.

SI Input data in SI units (default).

ENG Input data in engineering units

The UNITS statement should appear before the input data to which it relates and is therefore usually placed at the top of the input file after the HEADER statement.

If input data is provided in additional files, viz. [AUTOEXEC.PSM, \(p.599\)](#) branchname .U2P, or files specified on INCLUDE statements, each file should commence with its own UNITS statement to ensure the units in the file are not dependent on the position in the main input file where it is processed. A UNITS statement in an additional file does not affect the units already established for the main input file.

The UNITS statement may appear many times in the input file, to ensure the subsequent data has the desired units system. Please note however it is preferable for each data item to be qualified with its own units description string.

Example

```
.....  
UNITS INPUT=SI OUTPUT=ENG  
.....
```

5.3.6 OPTIONS Calculation Procedure Options (Optional)

Main-code: OPTIONS

SEGMENTS=	The number of <i>segments</i> in each pipe or tubing <i>section</i> . The pipeline is divided into <i>sections</i> by entering distance and elevation data (or TVD/MD) on NODE statements. Each <i>section</i> is then sub-divided by the program into a number of <i>segments</i> for calculation purposes. The default number of segments is 4; allowable range is 1 to 500. The intermediate segment data will only be printed if PRINT (p.624) sub-code SEGMENT is selected. The MAXSEGLEN= subcode (below) can also be used to subdivide sections. The EOFS= setting may add 2 additional short segments to the section, see below.
MAXSEGLEN=	The maximum segment length to be used by the program (ft or m). The number of segments in each section is computed by dividing the section length by the specified MAXSEGLEN= length. The final number of segments used is the maximum of this calculation, rounded up, and the number specified by the SEGMENTS= subcode. above. The default for MAXSEGLEN= is infinite. The EOFS= setting may add 2 additional short segments to the section, see below.
MINSECTLEN or DUPENODELEN	The minimum length of any section of pipe that the PIPESIM engine will compute properties such as pressure, temperature, etc. for. Any node which is closer than this length from the previous node will be ignored and removed from the system profile.
EOFS=	“Extra One-Foot Segments”. Pipe sections between NODE statements are divided into segments for calculation purposes, under the control of the SEGMENTS= and MAXSEGLEN= subcodes above. In addition, extra short segments are added to the start and end of the section to ensure the reported fluid properties and flowrates are calculated at an almost identical temperature and pressure as that reported at the node. In fact, the fluid properties are calculated at segment average pressure and temperature. With EOFS enabled, the discrepancies caused by this mismatch are minimized; however, it does have some effect of the runtime. Can be set to ON or OFF. Default is ON.
SEC=	Controls the calculation of pressure losses due to Sudden Expansions and Contractions (SEC) of pipe diameter. Can be set to ON or OFF. The default is ON.
SECLIM=	The lower limit for printing of SEC pressure losses. Any SEC DP less than this value will not be reported. DPs are reported with a one-line message in the primary output page. Default is 10 psi.
GRIDPRES=(...)	Values of pressure to be used in the P/T grid for compositional interpolated flash calculations. Psia or bara. Exclusive with NUMGRIDPRES=.

GRIDTEMP=(...)	Values of temperature to be used in the P/T grid for compositional interpolated flash calculations. F or C. Exclusive with NUMGRIDTEMP=.
NUMGRIDPRES=	The number of grid pressure points desired in the P/T grid for compositional interpolated flash calculations. A number between 60 and 100, default 60. If this subcode is specified, the actual values of grid pressure will be generated internally using an arithmetic increment algorithm, between atmospheric pressure and approximately 40,000 psi. Exclusive with GRIDPRES=.
NUMGRIDTEMP=	The number of grid temperature points desired in the P/T grid for compositional interpolated flash calculations. A number between 60 and 100, default 60. If this subcode is specified, the actual values of grid temperature will be generated internally using a linear increment algorithm, between -60 and +300 F. Exclusive with GRIDTEMP=.
2010GRID	Reverts from the current 60x60 compositional table pressure-temperature grid size to the 21x21 grid size present in PIPESIM 2010.1 and earlier releases. Can be specified either as 2010GRID or 2010GRID=ON
HTCRD=	"Heat Transfer Coefficient Reference Diameter". All HTCs printed on the Heat Transfer Output pages will normally use a reference diameter equal to the local pipe outside diameter. However, if a value is provided for HTCRD=, the supplied reference diameter will be used instead. This is useful when sensitizing on pipe diameter, or when different pipe diameters are present in the system, because it allows HTCs to be compared without the need to convert for different diameters. Units are In. or mm.
GTGRADIENT=	Controls geothermal gradient assumptions in Pipe objects. When a temperature profile is provided for a well tubing with TEMP= on the various NODE statements, the values are assumed to specify measured points on a geothermal gradient, so temperatures are interpolated between them based on the true vertical Depth. However, when temperatures are provided for flowlines and risers, they are assumed to be specifications that exhibit a step-change at the nodes. This subcode can be set to ON, OFF or AUTO. ON makes all pipe objects, notably flowlines and risers, interpolate the temperatures according to the elevation or depth (NB, not distance). OFF prevents all pipe objects, notably well tubing and horizontal completions, from interpolating. AUTO is the default setting, which allows well tubing pipes to interpolate, but prevents flowlines and risers.
PPMETHOD=	Compositional Flashing method Use the Advanced tab to configure additional calculation options and specify keyword input. for determination of fluid transport properties. Can be set to 1, 2, or 3. The default is 1. The meaning of these is:

- 1: Interpolate (fastest). This option uses interpolation between physical properties from flash results in a predefined grid of temperature and pressure points. This grid can be modified using the GRIDPRES= and GRIDTEMP= subcodes above.
- 2: Hybrid when close to the Phase Envelope, interpolation elsewhere. This is a compromise between speed and accuracy, which assumes that properties will change more rapidly when close to a phase boundary. Interpolation is performed whenever the grid points comprising a rectangle all show the presence of the same phases. For example, if all 4 points in the rectangle have some oil, some gas, and no water, then we assume the rectangle lies entirely within the 2-phase region of the hydrocarbon phase envelope, so interpolation is appropriate. If however one, two or three of the points have no oil, then clearly the hydrocarbon dew point line crosses the rectangle, so a rigorous flash is required.
- 3: Rigorous (slowest). Interpolation never occurs: properties are obtained by flashing at the required pressure and temperature. This is the most accurate method, but it is also the slowest.

THMETHOD=	Compositional Flashing method Use the Advanced tab to configure additional calculation options and specify keyword input. for Temperature/enthalpy balance calculation. In most simulations, for every PP flash that is performed, there are about 5 to 10 TH flashes, so these have the greatest effect on speed and run-time. The inaccuracies of TH interpolated flashes are usually minimal. Can be set to 1, 2, or 3, as for PPMETHOD=.
IFP=	This is a way to switch off all existing completion options in the model. Should only be used by another program controlling the engine as a sub-task. Can be set to ON or OFF. The default is ON.
ACTIVELAYER=	In a multi-layer well, specifies that only one of the reservoir layers is to be active. Must be set to a number between 1 and the number of reservoir layers in the well. Layers are numbered starting with 1 for the deepest.
COMPLETION= or COMPHBAL=	Controls the way temperature and enthalpy changes are handled when modelling the pressure drop calculations across a completion. may be set to: ADIABATIC or ISENTHALPIC: The pressure change will be at constant enthalpy, so the fluid will undergo a temperature change according to its Joule-Thomson coefficient. For Liquids this will result in a temperature increase; for gases, temperature will usually decrease, but in high-pressure reservoirs it may increase. ISOTHERMAL: The pressure change will be at constant temperature. The fluid temperature will not change, so a consequent enthalpy change will occur,

MAXEMULSION=	Maximum value for the multiplier as interpolated or extrapolated from the emulsion viscosity table for user-supplied and Woleflin emulsion correlations (p.721) . Default 100. The limit will be applied silently. This is a global (model-wide) option.
MAXCUTOFF=	Maximum value for the BOUNDARY= and CUTOFF= (p.721) keywords on the LVIS statement, default 70. Limit will be applied silently. This is a global (model-wide) option.
MAXLIQVISC=	Maximum liquid viscosity the engine will allow. If any correlation predicts values greater than this they will be limited to it. Default 1e7 cP. Limit will be applied with warnings. This is a global (model-wide) option.
SMOOTHCUTOFF=	Size of the transition region that is used to interpolate the viscosity multiplier when watercut is above the cutoff value. Default 5%. This is a global (model-wide) option.
EMUL3PHASE=	<p>Method for assigning priority between Emulsion options and a 3-phase flow correlation. This is a PIPE component level option. May be set to one of the following values:</p> <p>EMULSION: Emulsion option will take priority. Any fluid with an emulsion viscosity will override a 3-phase correlation. Oil and water phase Viscosities and densities will be set to the emulsion liquid phase values before calling the correlation.</p> <p>3PHASE: 3 phase correlation will take priority. Separate oil and water phase densities and viscosities will be passed to it, and its answers will be used as-is. The liquid phase emulsion viscosity will be ignored. This is the default setting, chosen for backward compatibility with previous versions of the engine.</p> <p>HYBRID: The 3 phase correlation will be called as for the 3PHASE option, and its prediction of the mixing status of the liquid phase will be examined. If it predicts separate, unmixed oil and water phases, the answers will be used as-is. If however it predicts mixed oil and water, and the fluid has an emulsion viscosity, then the answers will be discarded, and a further call made in the EMULSION mode as described above.</p>
HYDRATECALC=	Controls calculation of Hydrate Formation Temperature (HFT) in compositional models. HFT optionally appears in the profile plot file, but

Note: The test for "the fluid has an emulsion viscosity" is that the mixed liquid viscosity has to be at least 1% greater than the maximum of the oil and water phase viscosities. The test will therefore give a positive result for any emulsion option, that is, it is not restricted to Woleflin and user-supplied-table options

	calculating it results in a large increase in CPU time required, hence it is not enabled by default. Can be set to OFF or ON.
WAXCALC=	Controls calculation of Wax Formation Temperature (WFT or Cloud Point) in compositional models. WFT optionally appears in the profile plot file, but calculating it results in a large increase in CPU time required, hence it is not enabled by default. Can be set to OFF or ON.
ASPHALT CALC=	Controls calculation of Asphaltene Formation Temperature (AFT) in compositional models. AFT optionally appears in the profile plot file, but calculating it results in a large increase in CPU time required, hence it is not enabled by default. Can be set to OFF or ON.
ALHANATI=	Controls calculation of Alhanati Gas Lift Instability (GLI) criteria. GLI criteria can be calculated for wells that have gas lift, but the calculation requires a number of additional data items that are not needed for any other purpose. Can be set to ON or OFF, default OFF. ON: GLI criteria calculation is requested. If the additional data items are available, the calculated criteria values will be written to the system plot file; otherwise, diagnostic message(s) will be issued to enumerate the missing data. To remove the messages, you can either supply the missing data, or switch the calculation OFF. OFF: GLI criteria will not be calculated, and diagnostic messages will not appear.
GLMAXMASS=	Specifies a maximum gas lift rate limit, in mass ratio terms. Unlimited gas lift in a network branch can lead to the existence of multiple network solutions, so the network may converge to an unwanted solution, where a well produces nothing but lift gas. This subcode specifies the maximum mass rate of gas that can be injected, as a ratio with the current production mass flow rate. Its purpose is to prevent the well from converging at the unwanted solution. It is only applied in a network model. The default value is 0.2, thus the gas lift mass rate will be limited to 20% of the current production mass flowrate in a network model.
GLMAXGLR=	Specifies a maximum gas lift rate limit as a GLR. Unlimited gas lift in a network branch can lead to the existence of multiple network solutions, so the network may converge to an unwanted solution, where a well produces nothing but lift gas. This subcode specifies the maximum rate of gas that can be injected, as a volume ratio with the current production stock-tank liquid flow rate. Its purpose is to prevent the well from converging at the unwanted solution. The default value is infinite. A sensible value for this limit is in the range 1000 to 2000 scf/sbbl. Units are scf/sbbl or sm ³ /sm ³ .
FMMINTEMP=	Specifies the minimum temperature used in fluid property flash calculations. Units are F or C. The default value is 0K (absolute zero). However some flash packages will refuse to produce results at

temperatures higher than this. For example, NIST Refprop requires temperature to be above -100F. This keyword can be used to limit the temperature range for the iterative PH flash algorithm when calculating temperature from enthalpy. If the temperature is out of the flash package range, you get an error message stating this. Only the minimum temperature the package will allow is reported. Use this keyword to set the equivalent minimum temperature. A similar keyword FMMAXTEMP= is used for the maximum temperature.

SYSTEMTYPE=	Specifies if the branch or model represents a production well or an injection well. Usually there is no ambiguity between these and it is unnecessary for this to be explicitly stated by the user. However sometimes the model is open to interpretation either way. If it makes a difference, the user can supply an override value here. One example where this can be important is the production of VFP tables for a flowline branch. If there is no elevation difference between the branch start and end, then the system type is moot. However, VFP tables must be written with VFPPROD for a production branch and VFPINJ for an injection branch. Also, a VFPPROD allows sensitivity on GLR, Watercut and Artificial lift, and these must be provided in the table. Can be set to PRODUCER or INJECTOR. If omitted, PIPESIM will determine the system type, based on overall branch elevation change and the location(s) and numbers of completions it contains.
LAYERINJECT=	Controls the ability for reservoir layers to operate in injection mode. By default layers are able to accept fluid injection if the tubing pressure exceeds the layer pressure. Setting this option to NO makes all layers to refuse to allow injection; if the tubing pressure exceeds the layer pressure, the layer's flowrate will be zero. Can be set to YES or NO, default is YES.
ELIQLOADING=, LLE=	Specifies the correction factor to be applied to Turner's general equation in liquid loading (p.396) calculations. Minimum is 0.1, maximum is 10.0 and the default is 1.2.
LLVELOCITY=	Controls which Gas Velocity is used in liquid loading (p.396) calculations to get the Critical Gas Rate (CGR). Choices are: VSG, VM, VG, and EQN, whose meanings are: VSG: The Superficial Gas Velocity is used. This yields a result that is closest to that obtained by a hand calculation (from which it differs because the fluid phase behaviour is predicted by the selected fluid PVT package). However it is insensitive to Liquid Volume Fraction (LVF), and under some conditions can predict a CGR that reduces when LVF increases. This is the default. VM: The fluid Mean Velocity is used (i.e. the average of the gas and liquid phase velocities, the velocity at no-slip conditions). This yields a conservative result, i.e. a CGR somewhat higher than that obtained with

VSG. Its main advantage is that the CGR it calculates should increase with VFL.

VG: The Actual Gas Velocity is used. VG is calculated by the selected multiphase flow correlation, thus in principle it ought to be the most accurate choice. The CGR it calculates is generally the largest, or most conservative. However it will always be considerably larger than a hand calculated result, and will be strongly affected by the choice of multiphase flow correlation.

EQN: The gas velocity is calculated from the Stock-tank gas flowrate using the equation: $Vg = Q_{gas} * (T+460) * Z / (3.067 * P * A)$. This allows the resulting CGR to be verified by hand calculation. However, it takes no account of the fluid PVT behaviour.

LLANGLEMIN= or
ALIQLOADING=

Maximum pipe angle for [liquid loading \(p.90\)](#) calculations. The Turner equation assumes vertical or near vertical uphill flow. As deviation increases, so the equation becomes less applicable; so it makes sense to restrict it to pipe sections where the local deviation angle is a reasonable approximation to the vertical. By default the limit is 45 degrees. It can be set to any vertical deviation angle between 0.1 and 90 degrees. When the Pipe is deviated greater than this, the calculation is not performed.

LLFRNLIQMAX=

Maximum Liquid Volume Fraction (LVF) for [liquid loading \(p.90\)](#) calculations. The Turner equation assumes a continuous gas phase with small dispersed liquid droplets entrained in it. As LVF increases, so the equation becomes less applicable, and it makes sense to restrict it to pipe segments where the LVF is consistent with the description "liquid droplets in a continuous gas phase". By default the limit is 0.1. It can be set to any value between 0 and 1. When the LVF is greater than this, the calculation is not performed.

RAMEYTIME=

Specifies the length of time a well has been operating when [HEAT \(p.707\)](#) subcode RAMEYMETHOD is invoked for a piece of tubing. Minimum is 0 hour and default is 168 hours. The minimum recommended value for RAMEYMETHOD=LARGETIME is 168 hours.

UFACTOR=

Specifies a multiplier for all supplied (not calculated) U-values in heat loss calculations. U-values are entered on the numerous NODE statements that specify the geometry of the pipe and tubing of the branch. This multiplier is applied to all of these before they are used in calculation of heat transfer and temperature change of the fluid. This is useful if you want to sensitize on the overall effective U-value for the branch. Note it is NOT used if the U-value is calculated. Default is 1, allowed range is 0 to 1e10.

SEPMASSCALC=

Method for calculating the flowrate of fluid separated when a compositional fluid passes through a separator. Can be set to TABLE or FLASH, default TABLE.

MPBOOSTROUTE=	Specifies an override thermodynamic route to adjust the fluid temperature and enthalpy at the discharge of any multiphase booster. Can be set to ISENTHALPIC, ISOTHERMAL, or NONE.
FCVSHUTMODE=	Shut-in behaviour for Flow Control Valves (FCVs). An FCV can be specified with a table of fixed bean areas, and a flowrate limit. PIPESIM enforces the flowrate limit by selecting the largest area that results in a flowrate at or below the limit. However, it may happen that the smallest bean area in the table is too large to enforce the limit. In this case the value specified on this subcode is used to select a mode of behaviour. Can be set to OPEN, SHUT, or EXACT, whose meanings are: OPEN: The smallest non-zero bean area is used, so the flowrate will exceed the specified limit. SHUT: The valve will be set to the closed position, so the flowrate will be zero. EXACT: The flowrate will be set to the limit, and the required bean area will be calculated and reported.
RETAINHEEL=	Can be set to YES or NO. If YES, selects the Multiple Completion algorithm for the well's iterative solution, regardless of the number of completions the well may contain. The default is NO.
IFC=	An override on the state of the IFC= subcode on the HEAT statement. Can be set to INPUT or CALC; if set to CALC, this overrides any subsequent HEAT statement that may set it to INPUT.
SLUGREGIME=	Specifies the Flow Regimes that allow slug length calculations.
NOSLUGREGIME=	Specifies the Flow Regimes that do not allow slug length calculations.
MINSEGLEN=	The minimum segment length to be used when pipe sections are subdivided. (ft or m).
OPPOINTS=	Controls the explicit generation of Operating Points in the Nodal Analysis operation. Can be set to YES, to generate them, or NO, to omit them. This subcode is also available on the NAPLOT (p.739) statement; it is duplicated here so that it can be used without the additional effects that occur when NAPLOT is used.
DOWNHILLPREC=	Downhill Pressure Recovery method. When a two-phase fluid flows in a pipe that is angled downhill, the liquid phase usually flows faster than the gas. Sometimes the liquid flows downhill at its <i>terminal</i> , or <i>critical</i> , velocity, whereby its speed is limited by friction against the pipe walls, and there is no net pressure gain due to the elevation change. This is often called <i>slack</i> flow conditions. The multiphase flow correlations will not usually model slack flow, so this keyword allows a choice of methods for

simulating how pressure recovery is modelled in downhill pipe sections, and will affect the value for Elevation Pressure gradient:

CORR or NORMAL: Downhill pressure recovery is modelled by the selected multiphase flow correlation. The exact calculations performed will depend on the selected correlation. Typically, they use some mixture fluid density based on the calculated liquid holdup. This is the default method.

GAS: Downhill pressure recovery is calculated using the density of the gas phase alone. This option assumes the liquid will segregate into a stream at the bottom of the pipe and flow at its terminal velocity, similar to open-channel flow. N.B. This option is applied only when the in-situ liquid volume fraction is less than 0.8; at higher values, the CORR method is used.

NONE: Downhill Pressure Recovery is disabled. The elevation pressure gradient is set to zero in downhill pipe sections.

SSMETHOD=

Segment Solution Method. The pipe and tubing objects (for example, flowlines, risers, tubing strings, and distributed completions) are divided into computational elements called *segments*. Each segment is simulated one after another in a so-called *marching algorithm*. This subcode allows a choice of segment length selection and solution method. (Note: To activate this subcode requires a specific debug flag. To get the debug flag, please contact Schlumberger.) Currently, the following choices are available:

1: This is the Original method, which uses predefined, fixed segment lengths. The segment length is defined using the SEGMENTS= and MAXSEGLEN= keywords. (See above.) All pressure drop, heat transfer, and fluid inflow calculations are based on this segment length. If the segment convergence algorithm fails, then the entire section (NB, not just the current segment) is subdivided into double the previous number of segments and the calculation is restarted from the start of the section.

2: This is the Gradient method, which uses values of *gradients* to select a suitable segment length at the position in the system being simulated. The gradients considered are Pressure, Temperature, Enthalpy, and Reservoir Fluid Inflow. These are expressed as a change in the quantity per unit length of pipe. For example, pressure gradient is expressed as psi/ft or bar/m, temperature gradient is expressed as F/ft or C/m, and so on. Each gradient is evaluated at the segment boundaries. For each gradient, a user-specifyable ***tolerance*** exists which, when divided by the gradient, yields a segment length. The minimum of these segment lengths is used as the length of the next segment in the simulation. (For reasons of backwards-compatibility, the values of SEGMENTS= and MAXSEGLEN= keywords are also honoured in this method.) If the segment convergence algorithm fails, the segment length is halved and

the calculation is restarted. This is known as a "chop". NB, unlike the original method, the chop applies only to the current segment and not to the entire section.

DPRTOL=	Delta Pressure Relative Tolerance is a tolerance used by the Gradient method. This is a unitless ratio of pressure expressed as (PIN-POUT)/PIN, where PIN is the segment inlet pressure, and POUT is the segment outlet pressure. The segment length for the next segment is calculated as DPRTOL*PIN/PGRAD, where PGRAD is the pressure gradient (DP per length) from the previous segment. The default is 0.04. Smaller values will result in smaller, and therefore more, segments.
DPATOL=	Delta Pressure Absolute Tolerance is a tolerance used by the Gradient method. This is a value of Delta Pressure (DP) in units of psi or Bar. The segment length for the next segment is calculated as DPATOL/PGRAD, where PGRAD is the pressure gradient (DP per length) from the previous segment. The default is 1 psi. Smaller values will result in smaller, and therefore more, segments.
DTATOL=	Delta Temperature Absolute Tolerance is a tolerance used by the Gradient method. This is a value of Delta Temperature (DT) in units of Farenheit or Celcius. The segment length for the next segment is calculated as DTATOL/TGRAD, where TGRAD is the temperature gradient (DT per length) from the previous segment. This ensures that the temperature change across any segment is never more than DTATOL. The default is 5 F. Smaller values will result in smaller, and therefore more, segments.
DHATOL=	Delta Enthalpy Absolute Tolerance is a tolerance used by the Gradient method. This is a value of Delta Enthalpy in units of BTU/lb or KJ/Kg. The segment length for the next segment is calculated as DHATOL/HGRAD, where HGRAD is the enthalpy gradient (DH per length) from the previous segment. This ensures that the enthalpy change across any segment is never more than DHATOL. The default is 10 btu/lb. Smaller values will result in smaller, and therefore more, segments.
DPGTOL=	Delta Pressure Gradient Tolerance is a tolerance used by the Gradient method. This is a unitless ratio of pressure gradients that is used to validate the results of the current segment's DP calculation. The pressure gradient in the current segment is compared to the previous segment. If the difference is greater than this tolerance, the segment is "chopped", or divided into two smaller segments. This allows the algorithm to identify and recover from a pressure gradient discontinuity, such as what is often encountered in multiphase flow correlations. The default value is 0.05. Smaller values will result in smaller, and therefore more, segments.
DTGTOL=	Delta Temperature Gradient Tolerance is a tolerance used by the Gradient method. This is a unitless ratio of temperature gradients that is used to validate the results of the current segment's DT calculation. The

temperature gradient in the current segment is compared to the previous segment. If the difference is greater than this tolerance, the segment is "chopped", or divided into two smaller segments. This allows the algorithm to identify and recover from a temperature gradient discontinuity, such as what is caused step-changes in ambient temperature. The default value is 0.25. Smaller values will result in smaller, and therefore more, segments.

- DQGTOL= Delta Flowrate Gradient Tolerance is a tolerance used by the Gradient method. This is a unitless ratio of flowrates and/or flowrate gradients. It is relevant only in distributed completions and horizontal wells. It is used in the following ways:
- The segment length for the next segment is calculated as DQGTOL/ QGRAD. QGRAD is the inflow rate gradient that is calculated from QI/QP/Segl, where QI is segment inflow rate (the rate entering the segment from the reservoir), QP is segment production rate (the rate entering at the segment inlet), and Segl is segment length. These values are all from the previous segment. This ensures the flowrate change across any segment, when expressed as a ratio, is never more than DQATOL.
 - Used to validate the results of the current segment's Delta Flowrate (DQ) calculation. The inflow gradient in the current segment is compared to the previous segment. If the difference is greater than this tolerance, then the segment is "chopped", or divided into two smaller segments. This allows the algorithm to identify and recover from an inflow gradient discontinuity, such as what is caused by step-changes in reservoir properties (for example, pressure, and step-changes in reservoir fluid properties.)

The default value is 0.2 .Smaller values will result in smaller, and therefore more, segments.

- MEMCHOPFACTOR= Memory Chop factor is a factor used by the Gradient method. When a previous segment length was set as a result of a "chop" (see above), it is desirable to restrict the speed at which subsequent segments are allowed to grow. The maximum segment length for the current segment is limited to the length of the previous segment multiplied by this factor. The default value is 2.

5.3.7 RATE: Fluid Flow Rate Data

Main-code: RATE

RATE allows flow rate to be defined for all fluid types.

For both Blackoil and Compositional fluids, a flow rate may be defined in volumetric terms using the GAS= or LIQ= subcodes, or in mass terms using MASS=. A mass rate refers to the total stream without regard for which phases may exist. A volumetric rate refers only to the phase it specifies, and is always measured at stock-tank conditions. The other phase may or may not be

present at stock-tank conditions, depending on the fluid definition, but it is never included in the specified flowrate.

Stock tank conditions are 1.013 bara and 15.6 °C, or 14.696 psia and 60 °F.

LIQ=	Gross liquid flow rate at stock tank conditions (sm^3/d or STB/D). The liquid phase includes both hydrocarbon and aqueous phases (oil and water), but not gas.
GAS=	Gas flow rate at stock tank conditions (MMsm $^3/\text{d}$ or MMscf/d).
MASS=	The total mass flow rate (kg/s or lb/s). Note this defines the mass flow rate of the total stream, in contrast to LIQ= and GAS=, which defines a flow rate for one phase only.
MULTIPLIER=	Factor to mix or split a previously defined flow rate by a fixed ratio. Supplied value must be greater than zero. Valid only within the system profile, that is after the PROFILE or the first NODE statement.
ADDLIQ=	Quantity to be added to a previously defined stock-tank liquid flow rate. Supplied value may be greater or less than zero. (sm^3/d or STB/D). Valid only within the system profile, that is after the PROFILE or the first NODE statement. See also the INJFLUID. (p.693) statement.
ADDGAS=	Quantity to be added to a previously defined stock-tank gas flow rate. Supplied value may be greater or less than zero. (MMsm $^3/\text{d}$ or MMscf/d). Valid only within the system profile, that is after the PROFILE or the first NODE statement.. See also the INJFLUID and INJGAS. (p.693) statements.
ADDMASS=	Quantity to be added to a previously defined total mass flow rate. Supplied value may be greater or less than zero. (kg/s or lb/s). Valid only within the system profile, that is after the PROFILE or the first NODE statement.. See also the INJFLUID. (p.693) statement.
ADDER=	Quantity to be added to a previously defined flow rate (may be greater or less than zero). Note The units of ADDER= are inferred from the type of flowrate as originally defined, viz. Gas, Liquid or Mass, and the system of unit conversions currently in force, that is Engineering or SI.). Valid only within the system profile, that is after the PROFILE or first NODE statement.
WCUT=	Obsolete: see the BLACKOIL. (p.717) statement.
GWR=	Obsolete: see the BLACKOIL. (p.717) statement.
WGR=	Obsolete: see the BLACKOIL. (p.717) statement.
GLR=	Obsolete: see the BLACKOIL. (p.717) statement.
GOR=	Obsolete: see the BLACKOIL. (p.717) statement.
LGR=	Obsolete: see the BLACKOIL. (p.717) statement.

OGR= Obsolete: see the [BLACKOIL. \(p.717\)](#) statement.

A Blackoil fluid must define its stock-tank volume phase split on the BLACKOIL statement, using the subcodes: GLR=, GOR=, OGR=, or LGR=, and WCUT=, WGR= or GWR=. For historical reasons these subcodes are also available on the RATE statement. However, since RATE applies to all types of fluid (Compositional and Steam in addition to Blackoil), it is natural to assume that GLR= and so on behave similarly. **Alas they do not**, they apply to black oil fluids only. You are strongly encouraged to refrain from using these subcodes on RATE; use them on the BLACKOIL or [COMPOSITION. \(p.732\)](#) statement instead.

5.3.8 ITERN Iteration Data (Optional)

Allows the System Outlet Pressure to be specified.

Because PIPESIM performs a heat balance simultaneously with the pressure loss calculations, it is necessary for the calculation procedure to begin at the pipeline source and proceed in the direction of flow. A problem with a fixed delivery pressure therefore requires an iterative solution. The program will iterate on the user's specified Control variable, which can be System Inlet Pressure, Flow Rate, or a user-defined variable, as specified with the TYPE subcode.

The ITERN main-code should appear in the initial part of the input file, i.e. before the PROFILE or NODE statements.

Main-code: ITERN

POUT= The required System Outlet Pressure (psia or Bara).

TYPE= Specifies the identity of the Control variable ('guess') to be changed ('guessed') in order to match the specified outlet pressure. May be one of:

PRESSURE or IPRESSURE: The system inlet pressure.

GFLOW: The system gas flow rate.

LFLOW: The system liquid flow rate.

MFLOW: The system mass flow rate.

PGEN+: A User-specified variable, as defined with the special symbol ?XITERN. See note 1 below. Outlet pressure is expected to increase as ?XITERN increases.

PGEN-: A User-specified variable, as defined with the special symbol ?XITERN. See note 1 below. Outlet pressure is expected to decrease as ?XITERN increases.

XEST= Initial Estimate of the Control Variable ('guess') to be changed ('guessed') in order to match the specified outlet pressure. The units for this is dependent on the TYPE. For example: = Estimated inlet pressure (bara or psia) if TYPE=PRES. = Estimated mass flow rate (kg/s or lb/s) if TYPE=MFLOW. = Estimated gas flow

	rate (kg/s or lb/s) if TYPE=GFLOW. = Estimated liquid flow rate (kg/s or lb/s) if TYPE=LFLOW
PTOL=	Allows control over the Outlet Pressure Tolerance. The program will iterate until the difference between the calculated outlet pressure and the pressure specified in the POUT sub-code is less than outlet pressure tolerance. The user can specify the required tolerance, as a percentage, by use of the PTOL sub-code. If PTOL is not specified, the program will use a value of 1% or 1 psi, whichever is the smaller.
XTOL=	Allows control over the Control Variable ('guess') Tolerance. This is important if the system being simulated is prone to becoming 'ill-conditioned' (i.e., when a small change in guess results in a disproportionately large change in outlet pressure). Under such conditions it may take the iterative procedure many more iterations than usual to calculate a solution to within the outlet pressure tolerance (if it can manage it at all). Also, the user may not be interested in such accuracy, because for example, s/he may only be able to control the guess to within fairly coarse limits. XTOL exists to stop the program performing numerous unnecessary iterations, by terminating the iterative procedure when two successive guesses fall within the specified tolerance. The user can control the value of this tolerance, as a percentage, with the XTOL sub-code. The default value for XTOL is 1.0E-4 %.
XMIN=	Specifies a lower bound for the Control variable. If the iterative procedure attempts to guess below this limit, the guess will be reset to the limit. If this shows the required answer lies below the limit, the iterative procedure will terminate with a suitable diagnostic, and case output will be written.
XMAX=	Specifies an upper bound for the Control variable. If the iterative procedure attempts to guess above this limit, the guess will be reset to the limit. If this shows the required answer lies above the limit, the iterative procedure will terminate with a suitable diagnostic, and case output will be written.
LIMIT=	Specifies the maximum allowed number of iterations. The default value is 40 and the maximum is 100. If a solution has not been obtained within this number of iterations, the iterative procedure will be terminated, and results printed for this case with the current (i.e. last guessed) value of input pressure or flow rate.
SCREEN	Gives node by node output on the user's terminal for each iteration. If this sub-code is omitted, the only output that appears on the terminal during the iteration procedure is one line for each iteration, summarizing the iteration progress so far. Note this sub-code has no effect if PIPESIM is running in batch mode.
OPWI	Enables OPWI ("Output Printing While Iterating") mode. Node-by-node output for the system profile is written to the output files during every iteration. Normally, this output would be suppressed until the iterative procedure has converged. This is useful for debugging the iterative procedure.

LASTANSWER Using this sub-code, the guess from the previous case is used as an estimate for the next case. Thus the value which you have set for XEST will only be used in the first case.

CFCMODE=	Controls the iteration routine's interaction with a system containing a Choke in critical flow. Can be set to ON or OFF, default is ON. In the ON state, a choke in critical flow will terminate the iterative procedure early. Thus is usually beneficial, since the converged solution is likely to require the choke to be in critical flow. However when TYPE=PGEN, and the Control variable is applied downstream of the choke, an early iterative termination will prevent correct convergence, so the OFF state is preferable.
CFC SOLN=	Controls the post-convergence behavior when a case has converged with a choke in critical flow. Can be set to ON or OFF, default ON. In the ON state, a further round of iteration is performed, to converge on the pressure downstream of the choke that allows the specified system outlet pressure to be achieved. In the OFF state, this further round is omitted, thus the system outlet pressure will be higher than the one specified.
TITLE=	Specifies the title to be used for the Control variable in the system and profile plot files. TITLE keyword is working only when TYPE = PGEN+ or PGEN-.

Note: When TYPE=PGEN+ or PGEN- is used, the iterations will guess the value of a user-defined variable in order to achieve the specified outlet pressure. The variable is called ?XITERN, and the user must arrange that this name appears at a suitable point in the input file as the value of a subcode that will have an effect on the system outlet pressure. This is how PIPESIM implements its [user variable \(p.211\)](#) feature. PGEN is an acronym for "Pseudo-GENeral iterative mode" (it is not truly general since it converges only on outlet pressure).

5.3.9 INLET System Inlet Data

Main-code: INLET

The INLET statement is optional. It is useful if the system contains no reservoir completions, and is typically used to define a Generic Source at the start of a surface pipeline model. If supplied, must appear before the PROFILE or first NODE statement.

TEMPERATURE= The temperature of the fluid entering the system at the System inlet. (°C or °F). If omitted, the inlet fluid is assumed to enter the system at the ambient temperature as defined on the first NODE statement.

PRESSURE= The System inlet pressure (bara or psia). Not required if the inlet pressure is to be determined using the iteration option (see the ITERN main-code), or if the reservoir pressure is supplied with a well inflow performance option .

ENTHALPY= or H= As an alternative to temperature, the inlet fluid enthalpy can be supplied; PIPESIM will then calculate its temperature. (btu/lb or Kj/kg)

QUALITY= As an alternative to Temperature or Enthalpy, and only if the fluid is specified as Steam, the inlet steam quality can be supplied. Quality is the steam mass fraction vapour: must be in the range 0 to 1.

5.3.10 PRINT Output Printing Options (Optional)

Main-code: PRINT

PIPESIM offers a wide choice as to the amount of printed output. The PRINT maincode controls most of the available options, which can be divided into 4 behavioral categories:

Per-case Output pages

Subcodes such as PRIMARY and AUXILIARY control the production of complete pages which are repeated for as many cases as desired (as specified on CASES=). Pages are typically 132 characters wide and usually have (at least) one line for every node in the system, or otherwise have about 60 lines of relevant data. These page selections should be made at the start of the job or between cases.

Attributes

Subcodes such as SEC and TITLES affect the way data appears on output pages, e.g. by adding something to an existing page or changing the representation of the data on the page. Page attributes are best selected at the start of the job and not changed thereafter.

Point reports

Subcodes such as SPOT and PHASE SPLIT control the production of localized reports dedicated to an aspect of the system at a chosen position, or to a piece of equipment within it. Reports are requested by supplying the PRINT statement, with the required subcode, at the position within the system profile where the information is required. They are written to one of the selected output pages (as specified on REPORTS=) and appear for as many cases, and as many positions, as desired.

One-off Output pages

Subcodes such as SYNTAX, ECHO and NARESULT control the production of single page reports that appear only once per job. These page selections should be made at the start of the job.

Except where noted, the options can appear without a value, in which state a value of ON will be assumed. Values of OFF or ON can be provided if desired

Per-case output page options

The per-case output page options are as follows:

		Default
PRIMARY	The primary output page consists of a line for each node, containing node distance and elevation, pipe angle, fluid pressure, temperature and mean velocity, friction and elevation pressure drop, phase flowrates, phase densities, and Flow Regime pattern.	ON

AUXILIARY	The Auxiliary output page consists of a line for each node, containing node distance and elevation, , phase superficial velocities, mass flow rates and viscosities, overall Reynolds number, Liquid volume fraction, Liquid Holdup fraction, Flowing Liquid Watercut, total enthalpy, Erosion velocity, Erosion rate, Corrosion rate, Hydrate sub-cooling Delta Temperature, Liquid loading Velocity ratio, and segment iteration counters. For compositional fluids 3 additional columns hold table interpolation diagnostics.	ON
HORWELL	The Horizontal Well output page consists of a line for each node, containing: node distance and elevation; Reservoir and Wellbore Pressure; Reservoir, Inflow, and Wellbore temperature; Inflow Joule-Thomson Coefficient; Distributed Productivity Index; Wellbore flowrate; Specific inflow (i.e., flowrate between wellbore and reservoir, per unit length); Friction Gradient; Reservoir and Wellbore GLR and Watercut; and Liquid Viscosities for Reservoir, Inflow and Wellbore. The Horwell output is restricted to the portion of the system that is defined to be a distributed completion (see the COMPLETION (p.651) statement).	OFF
FLUID	The Input Fluid data page shows the definition of the Blackoil or Compositional fluids used as input to the case (fluid definitions resulting from mixing or separation can be obtained with the SPOT=FLUIDSPEC subcode, see below). A compositional fluid is specified mostly by its component list and their respective molar flowrates, along with other data controlling the attributes of the selected PVT package and table interpolation control data. A blackoil fluid is specified by a number of correlation choices and tunable values. This page also specifies fluid input flowrates.	ON
PROFILE	The profile and Flow Correlations output page consists of a line for each node, containing node distance and elevation, pipe section length, cumulative length, ambient temperature, input U-value, node TVD and MD, and fluid definition detail. In addition the selected Horizontal, Vertical, and Single Phase flow correlation choices will be echoed, along with pertinent options currently in force.	ON
ITERATION	The case-level iteration progress log page. This page will only appear if the case is iterative, i.e. the Outlet Pressure has been specified. Data is one line per iteration plus information on how each iteration's guess is computed. Errors encountered during iteration will also appear on this page.	ON
INFLOW	Details of the selected Inflow Performance Relationship (IPR) appear on this page. Data includes relevant input values and all derived or computed values and answers. If the model contains multiple completions, each will have its own section on this page.	OFF
HINPUT	The heat Transfer Input data page has a line for each node showing the input data for detailed heat transfer calculations across multiple	OFF

	layers of pipe and coatings. Values are: node distance, wax pipe and coatings thicknesses, wax pipe and coatings conductivities, burial depth, ambient fluid velocity, and ambient temperature.	
HOUTPUT	The Heat Transfer Output page has a line for each node showing the results of heat transfer calculations. Values are: Node distance, fluid temperature and enthalpy, Overall Heat Transfer Coefficient (HTC), Fluid film HTC, wax pipe and coatings HTCs, soil/ambient HTC, and text description of burial configuration. (All HTCs are referenced to the Pipe (Note: not coatings) outside diameter, this can be changed with the HTCRD= subcode of HEAT (p.707)).	OFF
SLUG	The Slug output page has a line for every node showing the results of slugging calculations. Values are: node distance and elevation, mean slug length and frequency, 1 in a thousand slug length and frequency, 1 in a hundred slug length and frequency, 1 in ten slug length and frequency, PI-SS, and flow regime pattern.	OFF
GLINPUT	The gas Lift Input data page has a line for every gas lift valve in the system, showing the input data supplied for it.	OFF
Note: The lines in this page appear in order of depth from the wellhead, i.e. shallowest at the top of the page, deepest at the bottom; this is the opposite to the direction of fluid flow in a gas lifted well, so this page will usually be in reverse order when compared with all other pages.		
	Values are: Valve TVD and MD, valve port diameter, Cv, test rack pressure, Ap/Ab, Throttling factor, valve type, and valve operation mode. (To request this page be produced in the same direction as the rest of the output pages specify GLINPUT=*FWD). N.B. The values in this page are only useful when MODE=SIMULATE has been specified on a GASLIFT (p.687) statement.	
GLOUTPUT	The Gas Lift Output page has a line for every gas lift valve in the system, showing all calculated values for the valve. The lines are ordered shallowest first as for the GLINPUT page (see above). Values are: valve MD, test rack dome pressure, valve operating temperature at depth, dome pressure at depth, casing and tubing pressure at depth, valve opening and closing pressures, DP across valve, Orifice gas flowrate, throttled gas flowrate, actual gas flowrate, and text description of valve operating status. (To request this page be produced in the same direction as the rest of the output pages specify GLOUTPUT=*FWD). N.B. The values in this page are only useful when MODE=SIMULATE has been specified on a GASLIFT (p.687) statement.	OFF
3PHASE	For Shell clients only, this page has a line for every node showing three phase flow values as calculated by the SRTCA 3-phase flow	OFF

	correlation. The 3-phase SRTCA (p.645) correlation must be the selected multiphase flow correlation for this page to appear.	
ARTSLUG	For Shell clients only, this page has a line for every node showing the results of Artificial Slug calculations. The Artificial Slug SRTCA (p.645) correlation must be the selected multiphase flow correlation for this page to appear.	OFF
WAX	this page has a line for every node showing Wax Deposition input data, calculations and results.	OFF
CUSTOM=(x,x,x)	The Custom output page allows you to create your own page of output organized one node per line. Values appear in columns on the page and are chosen from the set of Profile Plot variables. Identifiers are provided as a multiple value set (p.599) . An up-to-date list of the available identifiers can be obtained using the SYNTAX subcode of PLOT (p.632) . Each column will be 10 characters wide plus one space, so 11 columns will conveniently fit on a standard width page. If desired you can specify up to 40 identifiers, but be aware this will give an output page that is 440 characters wide.	OFF
INDATA	This is a combination of PROFILE and FLUID.	ON
EXTRA=	The Extra output page allows installation-specific data to be printed. This subcode requires an equated value. If the value TGRAD is provided, the result is a page containing a line for each node with temperature Gradient information from heat transfer calculations. Other values are installation-specific..	OFF

Attributes

The Attributes are:

		Default
TITLES	Print case titles on job summary output	ON
SEGMENTS	Print segment data. The pipe or tubing section between each node is sub-divided for computation purposes into a number of segments (as controlled by OPTIONS SEG= (p.609) and MAXSEGLEN= (p.609) , and the accuracy needs of the calculation at each point). With this subcode selected, each segment will have its own line of output in all the per-case output pages; without it, the output will be restricted to each node.	OFF
SEC	Print details of pressure drops caused by Sudden Expansion and Contraction. When Pipe ID changes, the junctions between the non-matching diameters are assumed to be straight-edged, and to cause pressure reduction due to turbulence effects. With this subcode enabled a one-line message will be written to the primary output where SEC	ON

losses are greater than a specified threshold value (as defined with OPTIONS SECLIM=).

CASES=	Specifies the number of cases to print. This subcode requires a numeric value. The selected per-case output pages will appear for as many cases as are specified. In Nodal Analysis jobs, the value applies to both the inflow and the outflow cases, thus the actual number of cases printed will be double.	1
REPORTS=	Specifies the name of the per-case output page to receive point report output. Any of the per-case page names can be provided to direct the point reports to the specified page. In addition the value DEDICATED specifies that an additional page be created to hold them instead.	PRIMARY

Point report subcodes

The point report subcodes must appear on a PRINT statement, positioned within the system profile, at the position where the values are required. If multiple reports are required at any point a separate PRINT statement must be used for each report. Each report is written to the output page chosen with the REPORTS= subcode above. Reports vary in length between 6 and 200 lines. They are:

SPOT=STPROPS	Fluid Stock-tank properties: phase flowrates and physical properties are reported at stock-tank conditions, viz. 14.696 psia, 60 F.
SPOT=FLPROPS	Fluid flowing properties: phase flowrates and physical properties are reported at the current pressure and temperature
SPOT=MPFLOW	Multiphase Flow values: Fluid properties, pipe dimensions, and calculated values with particular relevance to multi-phase flow calculations. See note 1.
SPOT=SLUG	Slug flow values: fluid properties and calculated values with relevance to slug size calculations. See note 1.
SPOT=SGLV	Sphere-Generated Liquid Volume values: input data and results from SGLV calculations. See note 1.
SPOT=HTINPUT	Heat Transfer Input values: fluid properties, pipe and coatings thicknesses and conductivities etc. as used in heat transfer calculations. See note 1.
SPOT=HTOUTPUT	Heat transfer output and calculated values: heat transfer coefficients, coating layer temperatures, film coefficients and dimensionless groups. See note 1.
SPOT=FLUIDSPEC	Fluid specification values. The complete set of values that define the fluid. A compositional fluid is specified mostly by its component list and their respective molar flowrates, along with other data controlling the attributes of the selected PVT package and table interpolation control data. A

	blackoil fluid is specified by a number of correlation choices and tunable values.
SPOT=ACVALUES	'Acculated' values. These are values that accumulate over the length of the system, for example total liquid holdup, total friction DP, total pipeline volume, etc.
SPOT=SHELL	Shell clients only, a report specific to the SRTCA slugging and 3-phase flow correlation. See note 1.
SPOT=COMPLETION	Distributed or multipoint completion values: reservoir inflow, drawdown, Distributed P.I., Skin values, relevant pipe dimensions, fluid phase flowrates and physical properties. See note 1.
USPOT=(x,x,x)	Custom spot report: allows you to create a report of values of your choice. Values are chosen from the set of Profile Plot variables. Identifiers are provided as a multiple value set (p.599) . An up-to-date list of the available identifiers can be obtained using the SYNTAX subcode of PLOT. (p.631)
MAP	Print the flow regime map at the current position. A flow regime map is specific to each choice of multiphase flow correlation, and is affected by fluid properties, pipe dimensions, and (critically) pipe angle. Since the map must be requested at a node position, please note that the pipe angle (and perhaps other dimensions) may change across the selected node. The dimensions and angle used to generate the map are those of the Upstream pipe section. The fluid properties used are those at the current pressure and temperature. If the map is requested at the start of the profile, pipe dimensions and angle are taken from the first pipe section.
PHASESPLIT	Print a Phase Split report, for compositional fluids only. This lists the molar flowrates of all components in the feed stream and in the phases that exist at that pressure and temperature. Additional phase properties such as density, viscosity etc. are also printed.
PRESSURE=	Pressure value for use with PHASESPLIT; if provided, will be used instead of the system current pressure. (psia or Bara)
TEMPERATURE=	Temperature value for use with PHASESPLIT; if provided, will be used instead of the system current temperature. (F or C)
PHASENV=	Produce a plot file containing the Phase Envelope (and other lines) for the current compositional fluid. Note this option produces no printed output; instead, a plot file will be created, named with an 8-character code known as the handle, and with an extension of .ENV. This file can be processed by the plotting post-processor PS PLOT to display the phase envelope. N.B., you do not need to know the name of the file to plot the phase envelope. From the GUI, select Profile Plot, then select Series, and choose axes of pressure and temperature. The phase envelope file(s) will be automatically processed along with the model's profile plot data, so you

should see the phase envelope and other available line(s) along with the pipeline system's pressure-temperature traverse.

By default the phase envelope file will contain a number of lines, depending on the phase behaviour of the fluid, the capabilities of the selected PVT package, and the PVT feature licenses you have available. The lines can be selected by supplying a list of line types as a [multiple value set. \(p.603\)](#) Available line types are:

HYDROCARBON: the hydrocarbon phase envelope, consisting of a bubble point line and a dew point line

CRITICALPOINT: the Hydrocarbon critical point

WATERDEW: the water dew point line

HYDRATE1: Hydrate type 1 line

HYDRATE2: Hydrate type 2 line

WATERICE: the water ice line

WAX: the wax appearance, or cloud point, line

ASPHALTENE: the asphaltene appearance line

If you do not supply a list of line types, the file will contain as many lines as the PVT package is capable of generating for the fluid, and for which you have a valid license.

QUALITY=	Values of Quality for use with PHASENV. If present, must be equated to a multiple value set (p.603) of quality values, each in the range 0 to 1. The resulting plot file will contain a hydrocarbon quality line for each value.
----------	--

Note: The values in this report are calculated during the simulation of a piece of pipe, and therefore refer to the pipe segment immediately upstream of the statement's position.

One-off output pages

The One-off output pages are:

		Default
NARESULT	Nodal Analysis result page. Lists the pressure, temperature and flowrate at the Nodal Analysis point for all cases in each inflow and outflow curve. Will only appear in Nodal Analysis jobs.	ON
SUMFILE	Controls the generation of the summary output file. This file is named from the model file's root name with an extension of .SUM, and contains a line for each labelled node for every case that was run in the job. Data values are: Stock-tank waternut, Stock-tank liquid flow	ON

	rate, flowing free gas flow rate, pressure, temperature, friction elevation and total pressure losses, mixture velocity, liquid holdup fraction, liquid holdup volume, and flow regime pattern.	
SUMMARY	Requests that the summary file be copied to the end of the main output file at the end of the job. Requires that SUMFILE be set to ON.	ON
WAXRESULT	For Shell clients only, this page has one line per case or per reporting interval showing the history of wax deposition in the system.	OFF
ECHO	Writes a line-numbered copy of the input data file at the start of the job. All included files will be expanded in-line, and any syntax errors generated will appear after the line that caused them.	ON
SYNTAX	Print formatted table of valid keyword input data. This extends over about 20 pages and lists the available maincodes, subcodes, value types they can be equated to, conversion factors, maximum and minimum limits for numeric data, and allowable character values. This report is generated from the engine code used to read and validate the input data file. It is useful when up-to-date documentation is not to hand or appears to be incorrect.	OFF
NEWINPUTDATA	Writes a copy of the input data file, but with the numeric data converted to the units system specified for output (with UNITS OUT=).	OFF

5.3.11 PLOT Output Plotting Options (Optional)

Main-code: PLOT (or NOPLOT to switch off)

The PLOT statement requests the production of plot files, and controls various aspects of program behavior with relevance to plotting. Plot files are post-processed with the BJA plotting program PSPLLOT, or your chosen plotting program (for example Microsoft EXCEL). When driven using the PIPESIM GUI, plots will normally appear concurrently with running the engine.

CASE or PROFILE	Requests the production of the Profile Plot File, which contains data that is organized to be plotted against position in the pipeline system. For example a pressure profile for a flowline shows pressure on the Y-axis against total distance on the X-axis; A temperature profile shows temperature against total distance. For a well the axes may be reversed, and/or the Y-axis might be elevation or depth. The profile plot file will contain many quantities that can sensibly be plotted against distance, elevation, or total length. (They may also be plotted against one another, with varying degrees of usefulness.) By default, each NODE (p.698) in the system will produce a point on the plot, and each case (p.743) will produce a separate line on the plot. The SEGMENT subcode (see below) will increase the number of plot points on each line.
CASE= or PROFILE=	As above, and if a value is provided, it specifies the data that is to appear in the plot file. Identifiers are provided as a multiple value set (p.603) . An up-to-date list of the available identifiers can be obtained using the

	SYNTAX subcode (see below). If no value is supplied a default set of plot file data will be written. If your supplied value starts with a plus sign [+], your identifiers will be added to the default set instead of replacing it.
JOB or SYSTEM	Requests the production of the System Plot File, which contains data that is organized to be plotted against sensitivity variables. For example a job that sensitizes on flowline ID could produce a plot of ID on the X-axis against flowrate on the Y-axis (or ID against holdup, or flowrate against holdup, etc...). The plot file will contain many values, and you can plot anything against anything, with varying degrees of usefulness. Each case will produce one point on the plot, and separate lines on the plot are produced by combinations of sensitivity variables as specified by MULTICASE or NAPLOT statements.
JOB= or SYSTEM=	As above, and if a value is provided, it specifies the data that is to appear in the plot file. Identifiers are provided as a multiple value set. An up-to-date list of the available identifiers can be obtained using the SYNTAX subcode (see below). If no value is supplied a default set of plot file data will be written. If your supplied value starts with a plus sign [+], your identifiers will be added to the default set instead of replacing it.
SEGMENT	Requests that profile plot data be written for every segment. By default a point is written only for each node, but the pipe between each node is usually sub-divided into a number of segments for calculation purposes. With this subcode you can plot the intermediate segment data as well.
HERE=	Requests that specified profile plot variables be added to the existing system plot file at the current profile position. This can be used for example to obtain fluid properties at any point, so they can be plotted against sensitivity variables. This subcode is only valid within the system profile. Identifiers are provided as a multiple value set (p.603) , chosen from the profile plot variables list (see SYNTAX below).
SYNTAX	Will create a list (in the standard output file) in two columns of available plot file variables and their identifiers (for example A, B, Y2, etc.) These identifiers can be provided to the JOB=, CASE= or HERE= subcodes (see above), and to SPOT= and CUSTOM= subcodes of PRINT (p.624) .
EQUIPJOB=	Controls the addition of equipment plot variables to the system plot file. Each item of equipment (for example, pumps, chokes, heaters, and so on) placed in the profile will, by default, result in additional plot variables being added to the system plot file. For each equipment item, between 6 and 20 additional variables will be added, the exact number and selection being specific to the equipment concerned. Can be set to ON or OFF, default ON.
PVTDATA=	Presence of this subcode triggers production of a fluid calibration plot file, similar to that produced when one of the PLOT buttons in the Black oil dialog advanced calibration tab is pressed. If a value is provided it must be a multiple value set of identifiers specifying the fluid properties to be

written to the plot file. The file created is named from the model or branch root file name with an extension of .PEX.

FORMAT= The overall textual layout of the plot files. Can be set to:

BJA: Write the plot files in BJA (i.e., original PIPESIM) format. This is the default option. BJA format plot files are composed of printable ASCII characters, arranged in lines of less than 200 columns. Header information is present at the start of the file. This is the option to use if you intend to read the file with PSPLT.

LOTUS: Write the plot files in LOTUS '.PRN' file format. Files will be named with the extension .PRN. The Lotus 123 spreadsheet program will recognize .PRN files and will often read them without further user intervention.

NEUTRAL: Write the plot files in NEUTRAL format. NEUTRAL format consists entirely of lines of numeric data arranged in columns. No header information is written.

CSV: Write plot files in Comma Separated Value format. Files will be named with the extension .CSV. The EXCEL spreadsheet program will recognize .CSV files and will usually read them without further user intervention.

PACKEDCSV: As for CSV above, but the data is written in a compressed form that occupies less space, thus using less disk space; however, it takes more run-time to produce.

GOAL: Write files in GOAL-compatible format. This is a combination of revision B (see below) and BJA.

XYJOB= For the System plot file, specifies the identifiers to be used as the X and Y axes when the plotfile is first opened by PSPLT. Identifiers are provided as a multiple value set.

XYCASE= For the Profile plot file, specifies the identifiers to be used as the X and Y axes when the plotfile is first opened by PSPLT. Identifiers are provided as a multiple value set.

VERSION= or
REVISION= Specifies the **revision standard** that the plot file is to be written to conform to. May be set to B or C, whose meanings are:

B: Revision B plot files conform to an older standard that contains some fixed-format data and hence is not forward compatible. Some older programs that read plot files, notably GOAL, can only process revision B plot files.

C: Revision C plot files contain additional information, and are written using textual Tags at the start of every line. This allows a measure of forward compatibility, thus additional features may be present in the file,

and these can be silently ignored by an older reading program, without causing an error.

COMPOSITIONS=	Controls the addition of composition records to the system plot file. Composition records specify the fluid definition at the system outlet, and are important in .PWH files created for use in PIPESIM.net's Wells Offline mode. With composition records present, a PWH file can be used to replace a well definition in a network run, resulting in considerable speedup of the network solution. Can be set to YES (the default) or NO.
CASEFILENAME= or PROFILEFILENAME=	Specifies the name of the profile plot file. By default this will be created at run-time from the root name of the branch or model input file name, with an extension of .PLC.
JOBFILENAME= or SYSTEMFILENAME=	Specifies the name of the system plot file. By default this will be created at run-time from the root name of the branch or model input file name, with an extension of .PLT.
PVTFILENAME=	Specifies the name of the Fluid calibration plot file produced with the PVTDATA= subcode. By default this will be created at run-time from the root name of the branch or model input file name, with an extension of .PEX.
ALHANATI=	Controls the calculation of Alhanati gas lift Instability criteria. The Alhanati criteria are required by GOAL, so production of GOAL-format files will enable this option. If it is enabled, but some of the data it requires is missing, warning messages will be produced: these will list the nature of the required missing data. This subcode allows the calculation to be controlled explicitly, thus the messages can be suppressed if the calculation is not required. Can be set to YES or NO, default being dependent on model input data.

A PLOT statement should appear before the first NODE card in a case, to specify the required SYSTEM and PROFILE plot options. Additional PLOT HERE statements can appear anywhere in the profile.

5.3.12 NOPRINT Output Print Suppression Options (Optional)

Main-code: NOPRINT

The NOPRINT card has the opposite effect to [PRINT \(p.624\)](#) and suppresses printing of the specified data. The same sub-codes as specified under PRINT are valid (with the exception of MAP). This card is often used to suppress output in the second and subsequent cases of a job.

5.3.13 BEGIN , END - Block delimiters

Main-codes: BEGIN, END

The BEGIN and END statements delimit a block of one or more further statements that collectively define an entity, and give it a name which can be referred to later. There are two types of entity

that can be defined, a CURVE, or a FLUID. The input file can contain as many BEGIN..END blocks as are needed to define as many fluids or curves as desired.

- FLUID** Specifies that the block defines a fluid. A Blackoil or Compositional fluid can be specified with as many delimited statements as are necessary, and the resulting fluid can be referred to on subsequent main-codes (such as . LAYER, INJGAS, INJFLUID, GASLIFT, BLACKOIL, COMPOSITION) to specify the injected or reservoir layer fluids
- CURVE** Specifies that the block defines a curve. Curve definitions are used in 2 situations:
Inflow performance : a reservoir or layer can be characterized by a curve of Bottom hole pressure against flowrate. Also, variation of GLR and Watercut can be specified as a coning relationship. Pumps and compressors: these devices can be specified with curves of flowrate against head, power and efficiency
- NAME** The name of the entity being defined.
- INHERIT** Optional, for FLUID blocks only. Controls inheritance of black oil fluid properties from the 'current' fluid. By default, each new fluid starts off with nothing defined. However the fluid already defined and currently in use can be inherited as the basis for a new fluid if desired. This is useful in legacy .PSM files which define only one black oil fluid and do not give it a name, and when additional fluids are being defined in [additional input files](#). ([p.604](#))

Example

The subcodes can appear on either maincode. Blocks cannot be nested, but it is possible to refer to an earlier block when defining a subsequent block.

For example:

```

begin fluid name=oil1
    BLACKOIL
    PROP API = 33 GASSG=0.65 PSAT=4000 TSAT=250
    GSAT=320
    LVIS T1=250 VIS1=0.6 T2=60 VIS2=20
    RATE GOR=320 WCUT=30
end fluid

begin
    BLACKOIL
    PROP API = 45 GASSG=0.6 PSAT=3770 TSAT=240
    GSAT=350
    LVIS T1=250 VIS1=0.63 T2=60 VIS2=22
    RATE GOR=300 WCUT=10
end fluid name=oil2

begin fluid name=oil3
    BLACKOIL USE = oil1
    RATE GOR=600 WCUT=12
end

BLACKOIL use = oil2

```

5.3.14 PUSH - Remote Action Editing (optional)

The PUSH statement is provided primarily to allow other computer programs to exert control over a PIPESIM engine run, without the need to modify an existing input file. For a human, almost everything that is possible with the PUSH statement can be accomplished far more easily by editing the main input ('.psm') file with a text editor. However, designing a computer program to reliably interpret and correctly modify a .PSM file without human help is surprisingly difficult. PUSH is best viewed a replacement for a text editor and a human. Nevertheless, humans can sometimes find PUSH statements useful as an alternative way to organize input data. (Beware however that a .psm file containing PUSH statements may not behave as expected if it is itself the subject of control by another program using PUSH.)

The PUSH statements are generally supplied in an additional [input file \(p.604\)](#) but this is not a requirement.

PUSH allows an editing action (the action) to be performed on a subsequent statement (the target). The target is specified by its maincode and label. The action can be: the addition of extra text on the end of the target statement; addition of an extra statement before or after the target; or the removal of the target statement.

Main-code: PUSH

MAINCODE=	Required: Specifies the target maincode.
LABEL= or OBJECT=	Specifies the label of the target statement. Serves to distinguish the required target statement when multiple statements having the same maincode are present. To specify that the target has no label (and thus prevent an earlier statement that does have a label from being the target), supply LABEL=*NONE.
TEXT=	Text to be appended to the target statement. The text should be enclosed in quotes since it will usually contain spaces, and equated pairs of keywords and values. The supplied text must conform to the syntax necessary for the target maincode, otherwise a syntax error will occur and processing will terminate.
ETEXT=	Exclusive text to be appended to the target statement. When 2 or more PUSH operations append text to the same target, the appended text will normally grow as each push is actioned; however if ETEXT= is specified the current text will replace any existing text resulting from earlier push(es).
LINE= OR LINEAFTER=	Text to be added as a separate line after the target statement.
LINEBEFORE=	Text to be added as a separate line before the target statement.
REMOVE	Results in the target statement being removed from the input. (This is actually achieved by transforming it into a comment by prepending the comment character '!'.)
ERROR=	Sets the severity of the action when errors occur. The most common error is that the position or target was not found, so the action did not occur. May be set to one of the following:

FATAL: Errors will be fatal, i.e. processing will terminate. A diagnostic message will be issued to the screen and the output file. This is the default behavior.

WARNING: Errors will result in a diagnostic message on the output file and a message box on the screen, but processing will continue.

NOTE: Errors will result in a diagnostic message on the output file, processing will continue.

SILENT: Errors will be silently ignored.

GLOBAL Specifies that this push statement is to be applied to all matching statements. If GLOBAL is not specified, the first statement that matches the specified maincode and label will be the only target.

Notes:

- Multiple PUSH statements may be present in the input file or additional files.
 - If many PUSHes specify the same target, the order in which the actions occur is the order in which they appear in the file. However the result may turn out to be reversed from that expected by the user. For example, if 2 pushes each add a line after the same target (the LINE= subcode), the second push will insert its new line immediately after the target thus displacing the one added by the first push. For the TEXT= and LINEBEFORE= subcodes this does not cause a problem, because the definition of the action corresponds to what the user expects. If one push specifies REMOVE=, then all subsequent pushes will not find the target, so position this push last.
 - The text added with TEXT=, LINE= etc can be any text valid for the specified position in the file. Multiple statements can be provided by separating them with a semicolon (';'). Remember to enclose the text in quotes ("") or apostrophes (''). If the text you are adding itself contains quotes , enclose it in apostrophes, and vice-versa.
 - The subcodes TEXT=, ETEXT=, LINE=, LINEBEFORE= and REMOVE are mutually exclusive.
 - Any statement that has a label starting with an exclamation point (!) will be excluded from being selected as a PUSH target. This is useful to prevent a line that was previously inserted with one push from being modified or removed by a subsequent push.
-

5.3.15 PLOTFILEDATA

Main-code: PLOTFILEDATA

5.3.16 EXECUTE - deferred execution of a statement

Main-code: EXECUTE

The EXECUTE statement allows some other statement to be positioned within the system profile, to be executed during system simulation. Normally, any statement in the profile is processed by the input processor, and is used to build the system model. The system model consists of a set of

global values, (for example fluid properties, options, inlet pressure, and so on.) and a set of connected equipment items (for example pipes, pumps, chokes, flowlines, and so on). When the system model is simulated, the global values cannot normally be changed, but use of the EXECUTE statement makes some values available for control.

EXECUTE has no subcodes: instead, any text supplied on it will be stored, and interpreted as a statement by the input processor when the system is simulated.

The EXECUTE statement should appear within the profile, that is after the PROFILE statement.

EXECUTE text comprising an otherwise complete and valid statement

5.3.17 USERDLL - Equipment

The API for the inclusion of user-defined 32-bit equipment DLL's is provided by Schlumberger.

See [User Equipment DLL Case Study - User Pump \(p.334\)](#)

Main-code: USERDLL

FILENAME=	The name of the DLL.
EPNAME=	The entry point of the DLL - the actual name of the routine as exported from the DLL
PSNAME=	The internal PIPESIM name of the routine. The psname's must be unique - the user should check that other DLLs specified in the userdll.dat file (located in C:\Program Files\Schlumberger\PIPESIM\data for a standard installation of PIPESIM - look for ep_ident) do not use the same psname's.
LINKTYPE= 24	The DLL linkage type. Note that it must be 24
EPTYPE= EQUIPMENT	The type entry point for the DLL. Note that it must be equipment to distinguish it from flow correlations.
TITLE=	The title text describing the DLL.
OPTIONS=	The string that will be sent as the first argument to the routine. (This is a global option, perhaps specified by the author of the DLL).
SDESCRIPTION=	
LDESCRIPTION=	

5.4 FLOW CORRELATION DATA

[VCORR \(p.641\)](#) Vertical Flow Correlation Options

[HCORR \(p.645\)](#) Horizontal Flow Correlation Options

[Single Phase Flow Options \(p.648\)](#)

[User Defined DLL \(p.650\)](#)

5.4.1 CORROSION

Maincode: CORROSION

Subcodes:

This maincode allows corrosion rate to be calculated.

DEWAARD Uses the de Waard model.

NONE disables corrosion calculations

PHACT= Optional: Specifies the actual pH of the fluid system. If not supplied the value will be calculated internally.

CC= or
EFFICIENCY= The multiplier Cc to correct for inhibitor efficiency or to match field data

5.4.2 EROSION Erosion Rate and Velocity (Optional)

Maincode: EROSION

This maincode allows erosion rate and erosional velocity to be calculated.

METHOD= Specifies the correlation method to be used. Available methods are:

API14E: The API 14 E method. This calculates erosional velocity assuming solids-free production. Erosion rate is not calculated. The only other subcode this method recognizes is K=, all others are ignored

SALAMA The SALAMA 2000 method.

K= or KEROS= The desired constant in the API 14 E equation. Default value is 100 in engineering units. A value of 100 specified when SI units are being used will be in SI units: this translates to approximately 82 in engineering units. The value may be qualified with the units descriptor 'ENG' or 'SI' to specify which units system to use when interpreting it.

H= or EROSRATE= The acceptable erosion rate. Used to calculate erosional velocity. Units are in/1e3/year or mm/year, default 0.1 mm/year.

SANDRATIO= The rate of sand production, specified as a ratio with liquid rate. Units are Parts Per Million , **by volume**, against **stock-tank liquid** rate. (The equations in Salama's paper use a sand rate in Kg/day. This is obtained from the supplied volume ratio using Salama's 'typical value' for sand density, 2650 kg/m³.) If sand production ratio is zero, erosion rate will not be calculated

W= or SANDRATE= The absolute rate of sand production, kg/day or lb/day. Use of this subcode is not recommended unless the model also fixes the system flowrate. Sand

production rate is better specified as a ratio with liquid rate, using the SANDRATIO= subcode (see above).

SM= or S=	This is the Geometry constant Sm in the Salama method, default 5.5.
CE= or EFFICIENCY=	Multiplier to match field data, default 1.
D= or SANDSIZE=	The mean size of the sand grains. Units are in/1e ³ or mm. Default 0.25 mm
SANDDENSITY=	Density of the sand grains. Units are lb/ft ³ or kg/m ³ . Default 2650 kg/m ³ .
SANDSG=	Specific gravity of the sand grains relative to water. Default 2.650

5.4.3 SLUG Slug Calculation Options (Optional)

Main-code: SLUG

The SLUG main-code allows the selection of slug behavior correlations. At present three slug correlations are available: the severe-sludging group PI-SS proposed by [Pots \(p.591\)](#), and the slug sizing correlations of [Norris \(p.590\)](#) and of [Scott, Shohan and Brill \(p.592\)](#).

PISS= ON	Start calculation of PI-SS
OFF	End calculation of PI-SS
SIZE= SSB	Switch on Scott, Shohan and Brill slug size correlation.
NORRIS	Switch on Norris slug size correlation.
OFF	Switch off slug size correlation.
BP= ON	Use BP Slug method. To see the results of this the following should also be used: print custom = (b,o,a24, b24, c24, d24, e24,f24,g24,h24,i24)

Note: The SIZE and PISS sub-codes are not related, and can be set independently of one another.

The PI-SS routine is based upon a correlation developed at Koninklijke Shell Laboratory. PI-SS is a dimensionless number that is a means of quantifying the likelihood of severe riser-slugging. Normally one would turn the PI-SS calculation on after the first node of the flowline and switch it off at the downstream riser base. If the value of PI-SS is less than one at the riser base and the flow regime (as predicted by the Taitel-Dukler correlation) is stratified, then severe riser slugging is possible. Conversely, PI-SS values significantly greater than one indicate that severe riser slugging is not likely. The PI-SS number can also be used to estimate slug size. As a rule of thumb the slug length will be approximately equal to the riser height divided by PI-SS, that is PI-SS values less than unity imply slug lengths greater than the riser height. PI-SS is calculated at each node in the flowline (while PISS=ON) using averaged holdup data, etc., but it is only the value recorded at the downstream riser base which is of any real significance. PI-SS is printed as part of the PRIMARY output (see the [PRINT \(p.624\)](#) main code).

The SIZE sub-code enables the user to specify a slug sizing correlation. At present two correlations are available, NORRIS and SSB. The NORRIS correlation was developed from Prudhoe Bay operational data and gives slug size as a function of pipe diameter. The SSB correlation was developed by Scott, Shohan and Brill and published in SPE paper 15103 in April 1986. The correlation takes account of slug growth. Normally one would switch the SIZE option on at the start of the profile and slug sizes will be automatically estimated whenever the flow regime (as predicted by the chosen correlation) is one that will support slugs. It should be noted that the slug size data output is only printed if SLUG is specified on the PRINT main code.

Slug catcher size

The following comments may help to determine the size of a slug catcher.

The slug output pages should be switched on from the Define Output dialog.

The size of a slug catcher is determined by one of the following parameters.

1. The amount of liquid generated by pigging the lines.
2. The amount of liquid generated by changing the flowrate in the flowline. At low flowrates there will be a large holdup of liquid in the pipeline and at high flowrates there will be a small holdup of liquid in the line. As the flowrate is increased you get a surge of liquid from the pipeline. The flowrate increase can be calculated using [Cunliffe's method \(p.393\)](#).
3. Dealing with slugs created by severe riser slugging. The likelihood of severe riser slugging is determined by the PI-SS correlation. Slugging will occur if there is a segregated flow regime and a PI-SS number less than one. The size of the slug is determined by using the following formula. $\text{Slug Size} = \text{Riser Volume}/\text{PI-SS number}$.
4. Dealing with hydrodynamic Slugging. This is determined by use of the SSB or Norris Correlations. You need a slugging flow regime for this to occur such as intermittent. The slug size and frequency is taken from the slug length and frequency table in the output. It is normal that the slug catcher is sized for the 1 in 1000 slug. These two correlations can predict huge slug sizes with volumes greater than the holdup in the pipeline. Therefore one must be careful to check the holdup as the slug cannot be bigger than the total amount of liquid in the pipeline.
5. Dealing with terrain slugging. PIPESIM cannot accurately predict slugging. If the holdup increases as the pipeline goes over successive humps - this may indicate a propensity for terrain slugging.

5.4.4 VCORR Vertical Flow Correlation Options

See also: [SPHASE Single Phase Flow Options \(p.648\)](#)

Main-code: VCORR

PLOSS= Pressure loss correlation (refer to the [Summary of Valid Vertical Flow Correlation Combinations \(p.642\)](#)).

HOLDUP= Holdup correlation (refer to the [Summary of Valid Vertical Flow Correlation Combinations \(p.642\)](#)).

MAP= Flow regime map (refer to the [Summary of Valid Vertical Flow Correlation Combinations \(p.642\)](#)).

ANGLE= Angle above which vertical flow correlations are used (default = 45 °)

TYPE= This sub-code allows the commonly recommended combinations of flow regime maps, holdup, and pressure loss correlations to be specified with one sub-code instead of separate MAP=, HOLDUP=, and PLOSS= sub-codes. Please refer to the table next page.

FFACTOR= Correlating or matching factor to be applied (as a multiplier) to the calculated friction pressure gradient (default = 1.0). This subcode can be used to adjust ('tune') the friction pressure drop values calculated by the correlation to match measured data.

HFACTOR= Correlating or matching factor to be applied (as a multiplier) to the calculated liquid holdup fraction (default = 1.0). This subcode can be used to adjust ('tune') the liquid holdup (and hence elevation pressure drop) values calculated by the correlation to match measured data.

SOURCE=

OVERRIDE=

ACCELL=

SWITCHES=

ENTRAINMENT=

OPTIONS=

Summary of Valid Vertical Flow Correlation Combinations

The following table summarizes the valid combinations of pressure loss, holdup and flow pattern map available for vertical flow. Entering non-valid combinations will result in an input data error. For details on the vertical flow correlation abbreviations, refer to [Vertical Flow Correlations - Abbreviations \(p.643\)](#).

PLOSS	HOLDUP	MAP	TYPE
DR	DR	DR/TD	DR
BBO	BBO	BB/TD	BBO
BBR	BBR	BB/TD	BBR
ORK	ORK	ORK	ORKISZEWSKI
GA	GA	GA	
HB	HB	BB/DR/BJA	HBR
HBO	HBO	BB/DR/BJA	

BJA	BJA1/BJA2	TD	BJA
MB	MB	MB	
Any	BRIMIN1	Any	
Any	BRIMIN2	Any	
NOSLIP	NOSLIP	NOSLIP	NOSLIP
GRAY	GRAY	TD	GRAY

Vertical Flow Correlations - Abbreviations

The abbreviations for vertical flow correlations is different for each source. This topic covers the BJA and TULSA sources. For OpenLink users and for flow correlations like OLGAS,LEDA, TUFFP defined in the userdll.dat file, the source is the identifier (IDENT) for the flow correlation, while the abbreviation is the entry point identifier (ep_ident) for the selection that the user wants to use.

BJA

The abbreviations for BJA are as follows:

ANSARI

Ansari Vertical Flow Correlation

BBO

Beggs & Brill Original

BBR.

Beggs & Brill Revised

BJA

BJA correlation

BJA1

Original BJA holdup correlation

BJA2

Revised BJA holdup correlation

BRIMIN 1 or 2

Brill & Minami Holdup Correlation

DR

Duns and Ros

GA

Govier and Aziz and Forgassi

GRAY

Gray Vertical Flow Correlation

GRAYM

Gray (modified)

GRAYO

Gray (original)

HB

Hagedorn and Brown (Revised)

HBO

Hagedorn & Brown (Original)

HBR

Hagedorn & Brown

HBRDR

Hagedorn & Brown, Duns & Ros map

LEDA

LEDA steady-state correlation

MB

Mukherjee and Brill

NOSLIP

No Slip Assumption

OLGA

OLGA-S steady-state correlation

ORK

Orkiszewski

TD

Taitel Dukler

TU2P

TUFFP Unified 2-phase v2007.1

TULSA

The abbreviations for Tulsa are as follows:

TBB

Beggs & Brill

TDR

Duns & Ros

TGA

Govier, Aziz

THB

Hagedorn & Brown (Original)

THBR

Hagedorn & Brown (Revised)

TMB

Mukherjee & Brill

TORK

Orkiszewski

5.4.5 HCORR Horizontal Flow Correlation Options

See also: [SPHASE Single Phase Flow Options \(p.648\)](#)

Main-code: HCORR

PLOSS=	Pressure loss correlation (refer to the Summary of Valid Horizontal Flow Correlation Combinations (p.646)).
HOLDUP=	Holdup correlation (refer to the Summary of Valid Horizontal Flow Correlation Combinations (p.646)).
MAP=	Flow regime map (refer to the Summary of Valid Horizontal Flow Correlation Combinations (p.646)).
TYPE=	This sub-code allows the commonly recommended combinations of flow regime maps, holdup, and pressure loss correlations to be specified with one sub-code instead of the separate MAP=, HOLDUP=, and PLOSS= sub-codes. Please refer to the table next page.
FFACTOR=	Correlating or matching factor to be applied (as a multiplier) to the calculated friction pressure gradient (default = 1.0). This subcode can be used to adjust ('tune') the friction pressure drop values calculated by the correlation to match measured data.
HFACTOR=	Correlating or matching factor to be applied (as a multiplier) to the calculated liquid holdup fraction (default = 1.0). This subcode can be used to adjust ('tune') the Liquid holdup (and hence elevation pressure drop) values calculated by the correlation to match measured data.
SOURCE=	
ANGLE=	
OVERRIDE=	
ACCELL=	
SWITCHES=	
ENTRAINMENT=	

Summary of Valid Horizontal Flow Correlation Combinations

The following summarizes the valid combinations of pressure loss, holdup and flow pattern map available for horizontal or inclined flow. Entering non-valid combinations will result in an input data error. For details on the horizontal flow correlation abbreviations, refer to [Horizontal Flow Correlations - Abbreviations \(p.643\)](#).

PLOSS	HOLDUP	MAP	TYPE
DR	DR/BJA	DR/TD	DR
DKAGAF	DKAGA	TD	
DKAGAF	EATON	TD	DKAGAF
BBO	BBO/BJA1/BJA2	BB/TD	BBO
BBR	BBR/BJA1/BJA2	BB/TD	BBR
BJA	BJA1/BJA2/EATON	TD	BJA
BJA1	BJA1/BJA2/EATON	TD	
MB	MB	MB	
HB	HB	BB/DR/BJA	HBR
HBO	HBO	BB/DR/TD	
OLI	BJA1/BJA2/EATON	TD	OLIEMANS
MB	MB	MB	
Any	BRIMIN1	Any	
Any	BRIMIN2	Any	
NOSLIP	NOSLIP	NOSLIP	NOSLIP

Horizontal Flow Correlations - Abbreviations

The abbreviations for vertical flow correlations is different for each source. This topic covers the [BJA](#) and [TULSA](#) sources. For OpenLink users and for flow correlations like OLGAS,LEDA, TUFFP defined in the userdll.dat file, the source is the identifier (IDENT) for the flow correlation, while the abbreviation is the entry point identifier (ep_ident) for the selection that the user wants to use.

BJA

The abbreviations for BJA are as follows:

BBR

Beggs and Brill (Revised)

BBO

Beggs and Brill (Original)

BBOTD

Beggs & Brill, Taitel Dukler map

BJA

Baker Jardine Revised

BJA1

BJA correlation

BJA2

Revised BJA holdup correlation

BRIMIN 1 or 2

Brill and Minami Holdup Correlation

DKAGA

Dukler (AGA)

DKAGAD

Dukler, AGA & Flanagan

DKAGAF

Dukler, AGA & Flanagan (Eaton Holdup)

DR

Duns and Ros

HB

Hagedorn and Brown Revised

HBO

Hagedorn and Brown Original

LEDA

LEDA steady-state correlation

LOCKMAR

Lockhart & Martinelli

LOCKMARTD

Lockhart & Martinelli

MB

Mukherjee and Brill

NOSLIP

No Slip Assumption

OLIEMANS

Oliemans

OLGA

OLGA-S Steady-State Correlation

OLI

Oliemans Correlation

TD

Taitel Dukler

TU2P

TUFFP Unified 2-phase v2007.1

XIAO

Xiao horizontal mechanistic model

TULSA

The abbreviations for TULSA are as follows:

TBB

Beggs & Brill

TDUK

Dukler

TMB

Mukherjee & Brill

5.4.6 SPHASE Single Phase Flow Options (Optional)

See also: [Single Phase Flow Correlations \(p.385\)](#), [Horizontal Flow Correlation Options \(p.372\)](#), [Vertical Flow Correlation Options \(p.377\)](#)

PIPESIM will automatically select either the specified two-phase or single-phase correlation depending on the phase behavior at the particular section in the pipeline. The single phase correlation is set by default to the MOODY correlation. **If no single-phase correlation is specified but single-phase flow is encountered in the pipeline, the program automatically switches to the MOODY correlation.**

In addition, when the specified phase correlation is the Moody correlation or the Cullender-Smith correlation, PIPESIM will calculate the Moody friction factor using either an iterative implicit method (Colebrook-White equation (Moody chart)), an explicit method (see the [Sonnad and Goudar paper \(p.593\)](#)) or a fast explicit or approximate method (see the [Moody paper \(p.590\)](#)). The default calculation method for the friction factor is the explicit method. The Moody friction factor calculation method will also have an impact on the horizontal and vertical flow correlations as the friction factor used to compute the pressure gradient in the flow correlations will be evaluated based on the method specified by the Moody friction factor calculation method.

Main-code: SPHASE

CORRELATION= Single-phase flow correlation.

AGA Use the AGA dry gas equation for single phase flow.

	MOODY	At Reynolds numbers greater than 2000, use the method specified by the MOODYCALC option and at Reynolds numbers less than 2000, assume laminar flow ($f=64/Re$) (default).
	PANA	
	PANB	
	WEYMOUTH	
	HAZENWILL	
	CULLSMITH	Uses the Cullender and Smith Correlation for Gas with a Moody friction factor calculated using the method specified by the MOODYCALC option.
DRAGFACTOR=		The AGA drag factor (default = 0.98).
LFMIN=		The liquid volume fraction below which single phase gas flow is assumed to exist (default = 0.00001).
LFMAX=		The liquid volume fraction above which single phase liquid flow is assumed to exist (default = 0.99).
TRMIN=		
TRMAX=		
TRMETHOD=		
	INTERPOLATE	
	CUTOFF	
	MAXIMUM	
	CUTOFF	
COMPARE=		
	ON	
	OFF	
C=		Hazen-Williams C parameter
LFPROP=		
MOODYCALC	EXPLICIT or SONNAD	Sonnad 2007 linear approximation (default)
	APPROXIMATE or MOODY	Moody 1947 approximation

IMPLICIT or ITERATIVE	Colebrook-White equation (Moody chart)
--------------------------	--

5.4.7 USERDLL - Flow Correlations

The API for user-defined multiphase flow correlation plug-in is provided by Schlumberger.

For a standard installation of PIPESIM, example Fortran source code is provided in the following directory (assuming the default installation location):

C:\Program Files\Schlumberger\PIPESIM\Developer Tools\User Flow Correlations
\Fortran_code

. Two files are included, “UFC2P_Demo.f90” for 2-phase correlations and “UFC3P_Demo.f90” for 3-phase correlations. These files are self-documenting templates that will compile as is (using Beggs-Brill as an example) and can be modified to interface with your own correlation and compiled into a dll that is called directly by the PIPESIM engine. Configuration of the flow correlations and related options is contained within the USERDLL.dat file which may be edited by selecting **Setup** » **Preferences** » **Choose Paths**.

5.5 WELL PERFORMANCE MODELING

[INTRODUCTION \(p.651\)](#)

[WELLPI \(p.653\)](#) Well Productivity Index

[VOGEL \(p.654\)](#) Data for the Vogel Equation

[FETKOVICH \(p.654\)](#) Data for the Fetkovich Equation

[JONES \(p.655\)](#) Data for the Jones Equation

[IFPPSSE \(p.655\)](#) Data for the Pseudo-steady state inflow equation

[WCOPTION \(p.657\)](#) Well Completion Data

[IPRCRV or IFPCRV \(p.660\)](#) Well performance and/or coning relationship tabulation

[IFPTAB \(p.662\)](#) Inflow Performance Tabulation (obsolete)

[CONETAB \(p.663\)](#) Coning relationship Tabulation (obsolete)

[BACKPRES \(p.664\)](#) Backpressure Equation (BPE)

[NAPOINT \(p.743\)](#) System Analysis Point

[NAPLOT \(p.739\)](#) System Analysis

[HORWELL \(p.664\)](#) Horizontal Well Inflow Performance

[LAYER \(p.666\)](#) Reservoir Layer properties

[PERMTAB \(p.669\)](#) Permeability Saturation Relationship Tabulation

[HVOGEL \(p.670\)](#)

[FORCHHEIMER \(p.670\)](#) Data for the Forchheimer Equation

FRACTURE Data for the Hydraulic Fracture IPR

[TRANSIENT](#) (p.671) Data for the Transient inflow equation.

5.5.1 INTRODUCTION

Several options for well performance modeling have been introduced. A number of basic options are presently available and are summarized below with application limits:

1. [Well Productivity Index](#) (p.653). Oil and gas reservoirs. Black oil and compositional.
2. [Vogel's Equation](#) (p.654). Oil reservoirs. Black oil only.
3. [Fetkovich's Equation](#) (p.654). Oil reservoirs. Black oil only.
4. [Jones' Equation](#) (p.655) . Oil and gas reservoirs. Black oil and compositional.
5. [Pseudo Steady State Equation](#) (p.655) . Oil and gas reservoirs. Black oil and single phase compositional.
6. [Well Completion Options](#) (p.657) (such as perforation and gravel steady state pack models) are available in association with the pseudo equation..
7. [Inflow Performance Tabulation](#) (p.662). Oil and gas reservoirs. Black oil and compositional.

Options 1 to 7 are mutually exclusive (except the Well Completion options, which must be used in combination with the Pseudo-Steady-State Equation). If more than one option is entered, the last one entered will be invoked. Normally inflow performance data would be entered after the INLET statement, and must appear before the first NODE card in a case. However, if injection wells are modeled, the system profile should describe the well geometry in the direction of flow, that is ending at the bottom hole. The appropriate inflow performance data should appear after the bottom hole and before the ENDCASE.

Printing Inflow Performance Data

A comprehensive printout of the well inflow performance data can be obtained by invoking the PRINT INFLOW option (Ref. Section 1.6).

Definition of Reservoir Type

For black oil cases, PIPESIM will interpret the reservoir type (oil or gas) from the way in which the flow rate is defined under the RATE or ITERN statement as follows:

- If the rate is defined on the basis of liquid flow plus a gas/liquid ratio (that is LIQ plus GLR sub-codes) then an "oil" reservoir is assumed.
- If the rate is defined on the basis of gas flow plus a liquid/gas ratio (that is . GAS plus LGR sub-codes) then a "gas" reservoir is assumed.

5.5.2 COMPLETION Completion Profile Delimiter

Main-code: COMPLETION

The profile delimiters (supercodes) are used by PIPESIM as required flags if the model contains horizontal wells or if you wish to perform system analysis anywhere in the system profile. The presence of the COMPLETION delimiter informs PIPESIM that subsequent wellbore sections form a "completion," or "productive interval." The program will therefore model the flow of reservoir fluid into the wellbore.

INLINE= If this sub code is present, the entire profile is modeled as a single unit. If it is absent, the completion is modeled separately from the rest of the system profile.

EFFLENG= The effective length of the horizontal completion (m or ft). This allows you to specify a completion length which is less than the actual length supplied with subsequent NODE maincodes. Thus, sensitivities on length can be performed using the NAPLOT maincode.

IPRPOINT=

DPRETIO=

TOL=

LABEL=

TYPE=

For datum reset feature, please refer to [Node \(p.698\)](#) .

Supercode

The supercodes are:

TUBING

Tubing Profile Delimiter

FLOWLINE

Flowline Profile Delimiter

RISER

Riser Profile Delimiter

Main-code: TUBING, FLOWLINE, RISER

The profile delimiters (supercodes) are used by PIPESIM as required flags if the model contains horizontal wells or if you wish to perform system analysis anywhere in the system profile. The portions of profile so delimited are sometimes described as objects.

When any of these are encountered after the COMPLETION delimiter, the inflow modeling is switched off, and the resulting flowrate is used for the remainder of the system profile.

Other modes of program behavior depend on the current delimiter, and the junctions of different delimiters. For example, Heat Transfer data implying that a pipe is buried, will not be applied to a riser; the junction of a flowline and an upward-going riser is identified as a riser-base and triggers checks on slugging parameters; the junction of tubing and flowline triggers actions relevant to the wellhead.

LABEL= or
NAME= The name of the profile object. This is used to print on the output file, and for object identification with the [PUSH \(p.636\)](#) statement.

RESETDATUM= Can be set to YES (the default) or NO. The NODE statements on either side of a supercode are, by default, assumed to be coincident. This allows the last

node of (for example) a previous flowline to specify the same position as the first node of the next flowline, with no intermediate length of pipe joining them, regardless of the values of distance and elevation these 2 nodes may specify. This behavior can be reversed with RESETDATUM=NO, which will model a pipe section between the 2 nodes in the same way as between any other 2 nodes in the same object.

INHERIT=	Can be set to YES (the default) or NO. Controls the application of Upstream Inheritance. Pipe object dimensions (for example Pipe ID, wall thickness, coatings thickness and conductivity, burial configuration, and so on.) are by default inherited from upstream objects. This allows each subsequent object to be specified with a minimum of input data, as the only required values are those that change between objects. However, mistakes in the specification of data can easily occur with this mode of behavior, particularly when complex pipe coatings and burial configurations are being specified, as unwanted data from previous objects can be mistakenly inherited by the current object. Specification of INHERIT=NO will ensure that each new object inherits nothing from its upstream neighbor.
----------	---

5.5.3 WELLPI Well Productivity Index (Optional)

Main-code: WELLPI

The WELLPI statement allows the [Productivity Index \(p.399\)](#) to be specified for a point-type completion or a distributed completion. Exactly one of the subcodes LPI=, GPI=, MIPI=, MCPI=, LDPI=, GDPI=, MIDPI=, or MCDPI= should be provided.

Subcodes

PWSTATIC= Static bottom hole pressure (bara or psia). This is the bottom hole pressure at zero flow rate.

LPI= Liquid Productivity Index (bbl/day/psi or sm³/day/bar)

GPI= Gas Productivity Index (MMscf/day/psi/psi or MMsm³/day/bar/bar) Note: Under normal circumstances, a gas well PI is much smaller than the typical liquid well PI. Typical gas well PIs are between 1E-3 and 1E-6 mmscf/day/psi². Oil well PIs usually vary between 1 and 40 STB/D/psi.

MIPI= Mass Incompressible Productivity Index (lb/sec/psi or kg/s/bar)

MCPI= Mass Compressible Productivity Index, (lb/sec/psi/psi or kg/s/bar/bar)

LDPI= Liquid Distributed Productivity Index (bbl/day/psi/ft or sm³/day/bar/m)

GDPI= Gas Distributed Productivity Index (MMscf/day/psi/psi/ft or MMsm³/day/bar/bar/m)

MIDPI= Mass Incompressible Distributed Productivity Index (lb/sec/psi/ft or kg/s/bar/m)

- MCDPI= Mass Compressible Distributed Productivity Index, (lb/sec/psi/psi/ft or kg/s/bar/bar/m)
- BPCORR= Allows a correction to the straight-line PI to allow for gas breakout when the fluid goes below its bubble point pressure: can be set to ON or OFF (default OFF). If enabled, the portion of the IPR below the bubble point is modelled with a Vogel relationship.
- PICOEF= Specifies the PI coefficient for the Vogel equation used if BPCORR= is enabled. (default 0.8)

5.5.4 WPCURVE (Optional)

Main-code: WPCURVE

This statement is obsolete, please do not use it.

5.5.5 VOGEL Vogel Equation (Optional)

Main-code: VOGEL

This keyword is used to specify data for [Vogel's Equation \(p.400\)](#).

Subcodes

- PWSTATIC= Static bottom hole pressure (bara or psia). This is the bottom hole pressure at zero flow rate.
- AOFP= Absolute Open Flow Potential of the well (sm^3/d or STB/D). This is a hypothetical liquid flow rate when bottom hole pressure is set to 0.0 psia
- PICOEF= The PI-coefficient used in Vogel's equation to adjust the degree of curvature of the inflow performance curve. Curvature increases with increasing PICOEF. A straight line is produced when PICOEF=0. (Default = 0.8).

5.5.6 FETKOVICH Fetkovich Equation (Optional)

Main-code: FETKOVICH

This keyword is used to specify data for [Fetkovich's Equation \(p.401\)](#).

Subcodes

- PWSTATIC= Static bottom hole pressure (bara or psia). This is the bottom hole pressure at zero flow rate.
- AOFP= Absolute Open Flow Potential of the well (sm^3/d or STB/D). This is a hypothetical liquid flow rate when bottom hole pressure is set at 0.0 psia.
- EXP= Exponent used in the Fetkovich equation to adjust the degree of curvature of the inflow performance curve. Unlike the Vogel equation it is not possible to produce a

linear well inflow characteristic as a special case of the Fetkovich equation. The default is 1.0.

5.5.7 JONES Jones Equation (Optional)

Main-code: JONES or FORCHHEIMER

This keyword is used to specify data for the [Jones Equation \(p.401\)](#).

Subcodes

PWSTATIC=	Static bottom hole pressure (bara or psia). This is the bottom hole pressure at a flow rate of zero.
A=	Turbulent flow coefficient. (Use TYPE= to define fluid type.)
B=	Laminar flow coefficient. (Use TYPE= to define fluid type.)
TYPE=	Used to define the type of fluid. LIQ Liquid GAS Gas
LA=	Liquid turbulent flow coefficient. (psi /MMscf $^2/d^2$ or bar/m $^6/d^2$)
LB=	Liquid laminar flow coefficient. (psi/MMscf/d or bar/m $^3/d$)
GA=	Gas turbulent flow coefficient. (psi $^2/MMscf^2/d^2$ or bar $^2/m^6/d^2$)
GB=	Gas laminar flow coefficient. (psi $^2/MMscf/d$ or bar $^2/m^3/d$)

5.5.8 IFPPSSE : Data for the Pseudo Steady State Equation (Optional)

Main-code: IFPPSSE

The Pseudo Steady-state equation employs a radial reservoir model. The equation takes into account both the effects of laminar and turbulent flows on pressure drawdown.

PWSTATIC=	Static bottom hole pressure (bara or psia). This is the bottom hole pressure at zero flow rate.
PERM=	Average formation permeability (md).
THICKNESS=	Average formation thickness (metres or feet).
RADE=	Radius of external boundary of drainage area (metres or feet). Default = 609.6 m or 2000 ft.
SKIN=	Dimensionless skin factor (mechanical). Default = 0.
DIAMWELL=	Wellbore diameter.

IPRTYPE=	Inflow performance relationship model
PSS	Pseudo steady-state model
JONES	Jones mode;
BASIS= or FLOWTYPE=	
LIQUID	
GAS	
2PHASE	
GASMETHOD=	
PSEUDO	Use pseudo pressure
SQUARED	Use pressure squared
GA=	
GB=	
SOURCE=	
ST	Use the stock tank flow formulation of the pseudo steady equation (default)
RES	Use the reservoir flow formulation of the pseudo steady equation
BPCORRECTION=	Controls whether to apply the Vogel correction below the bubble point
YES	
NO	
PICOEFF=	Vogel coefficient for Vogel correction. Default = 0.8
DSKINLIQUID=	Dynamic skin for liquid phase. Default = 0.
DSKINGAS=	Dynamic skin for gas phase. Default = 0.
RESAREA=	Reservoir area. (If entered, reservoir radius is calculated)
SHAPEFACTOR=	Reservoir shape factor. (If entered, reservoir radius is calculated) Default = 31.62
DRAINAGESKIN=	Reservoir drainage skin.

RESMOBILITY=	Reservoir mobility. Should only be used for injection systems (1/cp).
--------------	---

Note: If the skin is entered with the IFPPSSE main-code, any skin associated with the Well Completion options (see Section 7.6) will be overwritten by this value.

5.5.9 WCOPTION Well Completion Data (Optional)

Main-code: WCOPTION

Well Completion Options (WCOPTION) allow the mechanical and dynamic skin factors to be calculated from the details of the well completion configuration. Note: The well completion options are only valid when used in conjunction with the Pseudo-Steady-State or Transient equations (as defined under the IFPPSSE and TRANSIENT main-codes).

The WCOPTION maincode is an alternative to the FRACTURE maincode, and is exclusive with it.

TYPE=	Specifies the type of well completion
OPENHOLE	Openhole completion option. The program calculates the skin factors assuming the well is not cemented. The required data are DDAMAGE, PDAMAGE, INTERVAL, PVERT and DEVIATION. All other sub-codes will be ignored.
OPENGRAVEL	Openhole gravel pack completion option. The required data are DDAMAGE, PDAMAGE, INTERVAL, PVERT, DEVIATION, PGRAV and DSCREEN. All other sub-codes will be ignored.
PERFORATED	Perforated completion option. The program calculates the skin factors using the McLeod (p.420) or Karakas/Tariq model. The required sub-codes are; DDAMAGE, PDAMAGE, INTERVAL, PVERT, DEVIATION, PERFSNMTHD, SHOTS, LPERF, DPERF, PHASEANGLE, DCOMP, PCOMP. All other sub-codes will be ignored.
GRAVELPACKED	Gravel-packed and perforated completion Option. The required sub-codes are; DDAMAGE, PDAMAGE, INTERVAL, PVERT, DEVIATION, PERFSNMTHD, SHOTS, LPERF, DPERF, PHASEANGLE, DCOMP, PCOMP, PGRAV, DSCREEN, CASINGID and LTUNNEL. All other sub-codes will be ignored.
FRACPACK	Fracpack completion option. (i.e. a gravel packed hydraulic fracture model). The required sub-codes are; INTERVAL, PVERT, DEVIATION, SHOTS, DPERF, PGRAV, DSCREEN, CASINGID, LTUNNEL,

	FPHLFRAC, FPWFRAC, FPPPERM, FPDEPTH, FPDPERM, FPCLEN, FPCPERM. All other sub-codes will be ignored
INTERVAL=	Completion interval length (metres or feet). Default = formation thickness, as defined previously under the IFPPSSE (p.655) main-code.
PDAMAGEDZONE=	Permeability of the damaged zone around well bore (md). The formation data should be given under the IFPPSSE (p.655) main-code. Default = formation permeability.
DDAMAGEDZONE=	Diameter of the damaged zone around well bore (mm or inches). Default = well bore radius (i.e., damaged zone does not exist).
LPERFORATION=	Length of perforation into the formation (mm or inches). Default = infinity (which will result in a zero skin due to perforation).
SHOTS=	Shot density (shots/m or shots/ft). Default = 13.12 shots/m or 4 shots/ft.
PCOMPACTEDZO=	Permeability of the compacted zone (or crushed zone) around the perforation. Default = permeability of the damaged If neither the damaged zone nor compacted zone permeability is defined, the default value for compacted zone permeability will be the formation permeability specified under IFPPSSE. zone.
PVERTICAL=	Permeability in the vertical direction (md). Default = formation permeability, as defined previously under the IFPPSSE (p.655) main-code.
DPERFORATION=	Diameter of perforation (mm or inches). Default = 12.7 mm or 0.5 inches.
DCOMPACTEDZO=	Diameter of the compacted zone (or crushed zone) around the perforation. Default = diameter of the perforation (i.e., compacted zone does not exist).
PGRAVEL=	Permeability of gravel pack (md). Default = estimated according to sieve size input.
SIEVESIZE=	
LTUNNEL=	Length of tunnel; which is usually the sum of the thicknesses of cement, casing and annulus (mm or inches). Default = 0.0.

DSCREEN=	Gravel pack screen ID (mm or inches).
PHASEANGLE=	Perforation phase angle (degrees).
DEVIATION=	Well deviation (degrees).
CASINGID=	Casing ID (mm or inches).
FPDDEPTH=	Fracture damage depth (mm or inches). For fracture face skin term.
FPDPERM=	Fracture face damage permeability (md). For fracture face skin term.
FPPPERM =	Fracture proppant permeability (md). Frac pack proppant permeability.
FPCLEN =	Frac pack choke length (mm or inches). Choke length for choke fracture skin term.
FPCPERM=	Frac pack choke permeability (md). Choke permeability for choke fracture skin term.
FPWFRAC=	Fracture width (mm or inches). Frac pack fracture width.
FPHLFRAC=	Fracture half length (ft or m). Frac pack fracture half length.
PERFSKNMETHOD=	Method for calculation of perforation skin.
0	McLeod (p.420) model
1	Karakas / Tariq model
DZSKINCALC=	
GPSKINCALC=	
PFSKINCALC=	
PPDSKINCALC=	
FPSKINCALC=	
FACESKINCALC=	
CHOKESKINCALC=	

Note: The well completion options are only valid when used in conjunction with either the Pseudo-Steady-State equation or the Transient equation (as defined under the [IFPPSSE \(p.655\)](#) and

[TRANSIENT](#) ([p.671](#)) maincodes). All data entered under the WCOPTION main-code will be ignored if the IFPPSSE or TRANSIENT main-codes are not also specified.

5.5.10 IPRCRV or IFPCRV: Inflow Performance Curve

The IPRCRV statement is a generalized replacement for IFPTAB, GIFPTAB, and CONETAB. It allows a tabular Inflow Performance Relationship (IPR) to be specified using PIPESIM's [multiple value syntax](#) ([p.603](#)). A Coning relationship can also be supplied, either stand-alone, or in addition to a tabular IPR. The available range of formats in which the data can be supplied is a superset of those available with IFPTAB, GIFPTAB and CONETAB.

Maincode: IPRCRV or IFPCRV

NAME=	Required. Defines the name of the curve. This name is then used on (a) subsequent LAYER statement(s).
GAS=(...)	Specifies values of Gas flowrate (mmscf/d or mmsm ³ /d). Exclusive with LIQ= and MASS=.
LIQ=(...)	Specifies values of liquid flowrate (sbb/d or sm ³ /d). Exclusive with GAS= and MASS=.
MASS=(...)	Specifies values of mass flowrate (lb/sec or kg/sec). Exclusive with GAS= and LIQ=.
GLR=(...)	Specifies values of Gas Liquid Ratio (scf/sbb or sm ³ /sm ³). Exclusive with GOR=, OGR=, LGR=.
GOR=(...)	Specifies values of Gas Oil Ratio (scf/sbb or sm ³ /sm ³). Exclusive with GLR=, OGR=, LGR=.
OGR=(...)	Specifies values of Oil Gas Ratio (sbb/mmscf or sm ³ /mmsm ³). Exclusive with GOR=, GLR=, LGR=.
LGR=(...)	Specifies values of Liquid Gas Ratio (sbb/mmscf or sm ³ /mmsm ³). Exclusive with GOR=, OGR=, OGR=.
GWR=(...)	Specifies values of Gas Water Ratio (scf/sbb or sm ³ /sm ³). Exclusive with WGR=, WCUT=.
WGR=(...)	Specifies values of Water Gas Ratio (sbb/mmscf or sm ³ /mmsm ³). Exclusive with GWR=, WCUT=.
WCUT=(...)	Specifies values of Watercut (% vol/vol). Exclusive with WGR=, GWR=.
PWSTATIC=	Specifies the reservoir static (zero flowrate) pressure (psia or Bara).

PWF=(...)	Specifies values of Flowing Bottom Hole Pressure (psia or bara). Exclusive with DP=. Must be accompanied by PWSTATIC= unless the first flowrate value point is zero.
DP=(...)	Specifies values of drawdown (Delta pressure, the difference between PWS and PWF). (psi or bar). Exclusive with PWF=, and must be accompanied by PWSTATIC=.
GASSG=(...)	Specifies values of Gas Specific Gravity in a coning table. Exclusive with CONEDGASSG=.
DEGREE=	The degree of polynomial to fit to the data, default 1.
CONEDGASSG=	Specifies the Specific gravity of the gas in the gas cap. Used to calculate the produced gas SG. Exclusive with GASSG=.

Exactly one of the flowrate subcodes GAS=, LIQ= or MASS= must be specified. Then:

- To specify an IPR table, supply also one of PWF= or DP=.
- To specify a gas or liquid coning table, supply also one of GLR=, GOR=, LGR=, or OGR=. The gas specific gravity may be provided with GASSG= values, or a single value of CONEDGASSG=.
- To specify a water coning table, supply one of WGR=, GWR=, or WCUT=.

Gas and Water coning can be supplied with IPR data in the same table.

Care must be taken when combining the coning subcodes, since some combinations can cause unphysical situations, and others can leave the system undefined. For example, if OGR= and WCUT= are provided, the water flowrate is undefined when OGR is zero, so WGR= should be used instead of WCUT= or LGR= instead of OGR=. If GLR= and GWR= are provided, the GWRs must always be less than the GLRs, so WCUT= should be used instead of GWR=, or OGR= instead of LGR=

Examples

1. This defines a coning table for a liquid production well. The statement has been provided on 2 lines using the '&' character as the continuation marker at the end of the first line. The entire statement (i.e. the total characters in all the continued lines) may be no more than 255 characters in length.

```
IFPCRV name=ccl LIQ=(0,1000,2000,3000,4000) GLR=(300,300,550,600,620) &
WCUT=(10,10,18,22,30) GASSG=(.71 ,.71 ,.68 ,.67 ,.669)
```

2. This defines the same coning table as above, but shows how multiple statements can be used. Each statement may be no more than 255 characters in length, but the use of multiple statements allows more data points to be entered if necessary. Note that the curve name must appear on every statement.

```
IFPCRV name=ccl LIQ= (0 ,1000,2000,3000,4000)
IFPCRV name=ccl GLR= (300 ,300 ,550 ,600 ,620 )
```

```
IFPCRV name=cc1 WCUT= (10 ,10 ,18 ,22 ,30 )
IFPCRV name=cc1 GASSG=(.71 ,.71 ,.68 ,.67 ,.669)
```

5.5.11 IFPTAB Inflow Performance Tabulation (Optional)

Main-code: IFPTAB

Note: The IFPTAB statement is obsolete, its functionality has been replaced by [IPRCRV \(p.660\)](#).

Specifies a liquid inflow performance relationship in the form of a table of bottom hole pressure versus flow rate. Each IFPTAB statement holds a single data point. To complete the tabulation, at least four statements are required.

Values

should be provided without keywords in a strict positional order, as follows:

- value 1 Data point ordinal. If the IFPTAB statements are provided in order of increasing flowrate this can be set to 0 for all statements (recommended) ; otherwise, it must be a value between 1 and 30 to specify the ordinal.
- value 2 Liquid flow rate value at stock tank conditions. (sbbl/day or sm³/day)
- value 3 Bottom hole pressure (bara or psia).
- value 4 Gas Oil Ratio (optional, scf/sbbl or sm³/sm³)
- value 5 Watercut (optional, %)

EXECUTE see note 4.

Notes:

1. The first 3 values are mandatory. If values 4 and 5 are present they define a table of coning performance for the well or completion
2. One of the data points must be at zero flow rate such that the corresponding pressure is the static bottom hole (and reservoir) pressure.
3. The ordinal (value 1) is present for historical reasons to ensure backwards compatibility with earlier versions of the PIPESIM engine. As long as the statements are provided in order of increasing flowrate, and no other statements apart from IFPTAB appear in the middle of the table, the value can be left at zero.
4. If the IFPTAB table is provided inside the system profile, the last statement must contain no values, but instead must contain the EXECUTE sub-code. This syntax ensures that only one completion is actually executed regardless of the number of IFPTAB statements actually present. If IFPTAB is provided outside the system profile the EXECUTE subcode is unnecessary.

-
5. If your model contains multiple completions, 2 or more IFPTAB tables can be used to enter data relevant to each completion.
-

Example

```
!      n liq    pwf gor wcut
ifptab 0 0     3000 986   0
ifptab 0 1000  2990 986   2.0
ifptab 0 2699  2920 1096  2.2
ifptab 0 6329  2800 2540  2.8
ifptab 0 7288  2600 2980  3.9
ifptab 0 8082  2400 3370  5.6
ifptab 0 8805  2003 3770  8.0
ifptab execute
```

The ! line is a comment line.

5.5.12 CONETAB Coning Relationship Tabulation (Optional)

Main-code: CONETAB

The CONETAB statement is obsolete, its functionality has been replaced by [IPRCRV \(p.660\)](#).

Specifies a coning relationship for a well or completion, in the form of a table of stock-tank liquid flowrate versus produced GOR and watercut. Each CONETAB statement holds a single data point. To complete the tabulation, at least four statements are required. CONETAB does not define an inflow performance relationship, it is intended to be used in addition to an existing maincode that specifies the desired IPR. Values should be provided without keywords in a strict positional order, as follows:

LIQUID= Liquid flow rate value at stock tank conditions. (sbbl/day or sm³/day)

GOR= Gas Oil Ratio (scf/sbbl or sm³/sm³)

WCUT= Watercut (%)

Notes:

1. The CONETAB table should be provided in the system profile immediately before the maincode specifying the required IPR.
 2. No other maincode should appear in the body of the table.
 3. Values should be provided in increasing order of liquid flow rate.
 4. If your model contains multiple completions, 2 or more CONETAB tables can be used to enter data for each completion.
 5. For [BLACKOIL \(p.717\)](#) fluids, the Specific Gravity of the coned and associated gas should be provided with the [PROP \(p.719\)](#) statement. See example below.
-

Example

For example:

```
! liq gor wcut conetab 0 986 0 conetab 1000 986 2.0 conetab 2699 1096 2.2
conetab 6329 2540 2.8 conetab 7288 2980 3.9 conetab 8082 3370 5.6 conetab
8805 3770 8.0
```

5.5.13 BACKPRES Back Pressure Equation (BPE) (Optional)

Main-code: BACKPRES

This keyword is used to specify data for the [Back Pressure Equation \(p.402\)](#).

Subcodes

PWSTATIC The static reservoir pressure (psia or bar)

N The back pressure exponent (dimensionless).

C The back pressure constant (dimensions of (mmscf/d)/ (psi²)ⁿ or equivalent SI).

5.5.14 HORWELL Horizontal Well Inflow Performance

The following Inflow Performance options are available in addition to the Distributed Productivity Index Inflow option (see [WELLPI \(p.653\)](#))

- Pseudo-steady state equation of Babu and Odeh for oil wells
- Joshi's steady-state equation
- Backpressure equation

The backpressure equation is accessed via the BACKPRES maincode [BACKPRES \(p.664\)](#), which applies to horizontal completions as well as vertical. As with the WELLPI maincode, [WELLPI \(p.653\)](#), the backpressure C and N parameters are assumed to apply per unit length of wellbore. The steady-state and pseudo steady-state options are both accessed via the TYPE= subcode of the HORWELL maincode. The well completion option, [WCOPTION \(p.657\)](#), can be used in conjunction with the HORWELL maincode.

Main-code: HORWELL

TYPE= PSSOIL	The PSSOIL subcode calculates the horizontal well distributive productivity index based on Babu and Odeh's (p.583) SPE paper 18298. It is recommended the user read this reference before applying the equation. The equation is based upon the pseudo-steady state IPR well model applied to a rectangular drainage area.
ADIM	Drainage width perpendicular to the well (ft or m).
BDIM	Drainage width parallel to the well (ft or m).
THICK	Reservoir thickness (ft or m).

KX	Permeability in the x-direction (that is K_h) (mD)
KY	Permeability in the y-direction (parallel to well) (mD).
KZ	Permeability in the z-direction (that is K_v) (mD).
XZERO	X-ordinate of horizontal well trajectory (ft or m).
YONE	Starting y-ordinate of horizontal well trajectory (ft or m).
YTWO	Ending y-ordinate of horizontal well trajectory (ft or m).
ZZERO	Z-ordinate of horizontal well trajectory (ft or m).
RWELL	Sandface radius (such as pipe + annulus + cement) (in or mm).
SKIN	Mechanical skin factor (dimensionless).
PSSGAS	The pseudo-steady state gas flow equation is based upon a circular drainage area and is described in Joshi's (p.588) "Horizontal Well Technology". It is recommended that the user read this reference and Inflow Performance Relationships for Horizontal Completions (p.427) . This equation contains two skin terms; the skin due to drilling/perforations and the rate-dependent skin due to turbulent gas flow around the wellbore.
THICK	Reservoir thickness (ft or m).
KX	Permeability in the x-direction (that is K_h) (mD).
KY	Permeability in the y-direction (parallel to well) (mD).
KZ	Permeability in the z-direction (that is K_v) (mD).
RWELL	Sandface radius (i.e. pipe+annulus+cement) (in or mm).
REXT	External boundary radius of drainage area (ft. or m). Default = infinity
SKIN	Mechanical skin factor (dimensionless).
SSOIL	The simplest form of horizontal well productivity calculations are the steady-state analytical solutions which assumes that the pressure at any point in the reservoir does not change with time. The steady-state distributive productivity index is based upon Joshi's (p.588) SPE 16868 "Review of Horizontal and Drainhole Technology". The equation is based on the assumption that the horizontal well drains an ellipsoidal volume around the wellbore of length L. See Inflow Performance Relationships for Horizontal Completions (p.427) , for more details.
ECCENT	Wellbore eccentricity (i.e. offset of the well from the centre of the pay zone) (in or mm).

THICK	Reservoir thickness (ft or m).
KX	Permeability in the x-direction (that is K_h) (mD).
KY	Permeability in the y-direction (parallel to well) (mD).
KZ	Permeability in the z-direction (i.e. K_v) (mD).
RWELL	Sandface radius (that is pipe+annulus+cement) (in or mm).
REXT	External boundary radius of drainage area (ft or m).
SKIN	Mechanical skin factor (dimensionless).
SSGAS	The steady-state gas distributive productivity equation is described in Joshi's (p.588) "Horizontal Well Technology", Chapter 9, and Inflow Performance Relationships for Horizontal Completions (p.427)
THICK	Reservoir thickness (ft or m).
KX	Permeability in the x-direction (that is K_h) (mD).
KY	Permeability in the y-direction (parallel to well) (mD).
KZ	Permeability in the z-direction (that is K_v) (mD).
RWELL	Sandface radius (that is pipe+annulus+cement) (in or mm).
REXT	External boundary radius of drainage area (ft or m).
SKIN	Mechanical skin factor (dimensionless).

5.5.15 LAYER Reservoir Layer Properties

Main-code: LAYER

LAYER specifies the presence of a distinct reservoir Layer or Zone. It allows the properties of a fluid to be defined, along with its pressure and temperature. Some additional layer-specific properties can also be set. LAYER is intended to be used in a model that contains multiple completions, which may be point-type, or distributed/horizontal. It should only appear within the system profile, and be followed by a statement that selects a choice of IPR relationship, for example, WELLPI, FETKOVICH, IFPPSSE, and so on.

All subcodes are optional.

PWSTATIC= or PRESSURE=	Reservoir layer (Static bottom hole) pressure (bara or psia). This can also be defined on the selected IPR maincode, in which case it may be omitted from LAYER. If supplied on both, the one on the IPR will be used.
TEMPERATURE=	Reservoir layer temperature (F or C). This should be the same as the ambient temperature on the node, and is therefore unnecessary.

USE= or FLUIDNAME=	Name of a Black Oil or Compositional fluid, as previously defined with BEGIN FLUID (p.634) . Exclusive with PVTFILE=.
PVTFILE=	Name of an existing .PVT file defining a compositional fluid. Exclusive with USE=.
INJECT=	Controls if the a layer will accept fluid injection: can be set to YES or NO, default YES.
IPRCURVENAME=	Specifies the name of an Inflow Performance and/or Coning curve for the Completion, as defined in (a) previous IPRCRV (p.660) statement(s).
LDORATE= or MDORATE= or GDORATE=	Specifies a fixed, overriding value for the Specific Inflow Rate in a horizontal completion. Normally the inflow rate of a horwell is calculated from the supplied Inflow Performance data, and reservoir and wellbore pressure difference. If one of these subcodes is supplied however, all other data is ignored, and the reservoir inflow rate is unconditionally set to this value. LDORATE= supplies a Liquid rate (bbl/day/ft or m3/day/m); GDORATE= supplies a gas rate (mmscf/day/ft or mmsm3/day/m), MDORATE= supplies a mass rate (lb/sec/ft or Kg/sec/m).
EXECUTE=	For use with IPRCURVENAME=. If the supplied curve specifies a complete Inflow Performance relationship, this subcode can be used to render it executable.
PERMCURVENAM= or PERMCRVNAME=	Specifies the name of an Oil/Water Relative Permeability table or curve for the completion, as defined in (a) previous PERMCRV (p.668) statement(s).
SATURATION=	For use with PERMCURVENAME=. Allows the reservoir saturation value to be specified. This results in the calculation of the watercut of the produced fluid. If absent, the specified fluid watercut is used, and reservoir saturation calculated from it.
HBALANCE= or ROUTE=	Specifies the thermodynamic route for calculation of temperature/enthalpy change consequent upon the DP across the completion. May be set to ISENTHALPIC, for constant enthalpy (the default), or ISOTHERMAL, for constant temperature.

Examples

Example 1

This is a simple layer and point completion such as might appear as part of a larger multi-completion model:

```
LAYER temp = 220 use = 'fluid A' label = 'Layer one'
WELLPI pwstatic = 3650 LPI=9.5
```

Example 2

This completion includes a coning relationship defined with [IPRCRV \(p.660\)](#), which must appear first. LAYER follows because it references the curve name defined in the IPRCRV. Finally the selected IPR, JONES, comes last:

```
IFPCRV name=cc1 LIQ= (0 ,1000,2000,3000,4000)
IFPCRV name=cc1 GLR= (300 ,300 ,550 ,600 ,620 )
IFPCRV name=cc1 WCUT= (10 ,10 ,18 ,22 ,30 )
IFPCRV name=cc1 GASSG=(.71 ,.71 ,.68 ,.67 ,.669)
LAYER temp=240 F pres=4503 psia use='fluid' B inject=NO IPRCURV=cc1
label='Layer two'
JONES LA=1e-4 LB=3e-2
```

5.5.16 RESERVOIR

Main-code: RESERVOIR

This maincode is obsolete, please do not use it.

5.5.17 PERMCRV: Curves of Relative Permeability versus Saturation (Optional)

Main-code: PERMCRV

The PERMCRV statement is an alternative to [PERMTAB \(p.669\)](#). Both these statements allow a table/curve of oil/water relative permeability to be entered as a function of water saturation. They differ in the format of the statements: PERMTAB allows a tabular data entry format using multiple statements, whereas PERMCRV offers subcodes that accept PIPESIM's [multiple value syntax \(p.603\)](#). PERMCRV requires the curve to be given a name that can be referenced in one or more subsequent completions.

NAME= Required. Defines the name of the curve. This name is then used on (a) subsequent LAYER statement(s).

SAT= Water Saturation in the reservoir, as a ratio 0 to 1.

OIL= Oil relative permeability, as a ratio 0 to 1.

WAT= Water relative permeability, as a ratio 0 to 1.

The complete definition of a curve requires all subcodes to be specified, but they may be spread over 2 or 3 statements that reference the same curve name. NAME= must appear on all statements.

Example

This example shows a table containing the same data as the example given for [PERMTAB \(p.669\)](#):

```
PERMCRV name=pc1 SAT = (0.0, 0.1, 0.2, 0.3 , 0.4 , 0.5 , 0.6, 0.7 , 0.8 ,
0.9 , 1 )
```

```

: PERMCRV name=pc1 OIL = (0.9, 0.9, 0.9, 0.6 , 0.43, 0.35 , 0.2, 0.13, 0.07,
0 , 0 )
PERMCRV name=pc1 WAT = (0.0, 0.0, 0.0, 0.05, 0.1 , 0.125, 0.2, 0.33, 0.44,
0.44, 0.44)
layer    PERMCURV=pc1    temp = 185 F use = oil44 inject=no label =  'strat
bk'
ifppsse pwstatic = 4269 psia perm = 200 md thickness = 50 ft &
rade = 2000 ft skin = 2 diamwell = 5 in

```

5.5.18 PERMTAB: Tabulation of Relative Permeability versus Saturation (Optional)

Main-code: PERMTAB

The PERMTAB statement is an alternative to [PERMCRV](#). (p.668) Both statements allow entry of a table of oil and water relative permeability against water saturation, in a different format.

Each PERMTAB statement holds a single data point. To complete the tabulation, at least four statements are required. PERMTAB does not define an IPR relationship, it should only be used in addition to the IFPPSSE maincode. If provided with any other IFP maincode it will be ignored.

Values should be provided without keywords in a strict positional order, as follows:

value 1 Water Saturation in the reservoir, as a ratio 0 to 1.

value 2 Oil relative permeability, as a ratio 0 to 1.

value 3 Water relative permeability, as a ratio 0 to 1.

Notes:

1. The PERMTAB table should be provided in the system profile immediately before the IFPPSSE maincode.
2. No other maincode should appear in the body of the table
3. Values should be provided in increasing order of Water Saturation
4. If your model contains multiple completions, 2 or more PERMTAB tables can be used to enter data for each completion.

Example

This example shows a table containing the same data as the example given for [PERMCRV](#) (p.668) :

	wsat	oil rp	water rp
permtab	0.0	0.9	0.0
permtab	0.1	0.9	0.0
permtab	0.2	0.9	0.0
permtab	0.3	0.6	0.05
permtab	0.4	0.43	0.1
permtab	0.5	0.35	0.125
permtab	0.6	0.2	0.2
permtab	0.7	0.13	0.33

```

permtab    0.8    0.07    0.44
permtab    0.9    0.0    0.44
permtab    1.0    0.0    0.44
layer      temp = 185 F use = oil44 inject=no label = 'strat bk'
ifppsse   pwstatic = 4269 psia perm = 200 md thickness = 50 ft &
          rade = 2000 ft skin = 2 diamwell = 5 in

```

5.5.19 HVOGEL (Optional)

Main-code: HVOGEL

PWSTATIC=

AOFP= Absolute open flow potential

RF= Recovery factor

5.5.20 FORCHHEIMER (Optional)

Main-code: FORCHHEIMER

see the [JONES \(p.655\)](#) maincode.

5.5.21 FRACTURE: Data for Hydraulic Fracture

Main-code: FRACTURE

This maincode specifies that the mechanical and dynamic skin factors are to be calculated from the details of the well fracture configuration provided. Note: The fracture options are only valid when used in conjunction with the Pseudo-Steady-State or Transient equations (as defined under the IFPPSSE and TRANSIENT main-codes).

The FRACTURE maincode is an alternative to the WCOPTION maincode, [WCOPTION \(p.657\)](#), and is exclusive with it.

PERMEABILITY= Average fracture permeability (md).

WIDTH= Average fracture width (feet or metres).

LENGTH= Fracture half-length (feet or metres).

RESAREA= Reservoir area (square feet or square metres).

TRANSIENT= Controls the Transient option. May be set to ON or OFF. Default OFF.

POROSITY= Reservoir porosity (dimensionless, a value between 0 and 1.) Ignored unless TRANSIENT=ON.

COMPRESS= Reservoir compressibility (1/psi or 1/Bar). Ignored unless TRANSIENT=ON.

TIME= Elapsed time since start of production (hours). Ignored unless TRANSIENT=ON.

CONDUCTIVITY=

See also [Hydraulic Fracturing](#).

5.5.22 TRANSIENT: Data for the Transient Inflow equation (Optional)

Main-code: TRANSIENT

PWSTATIC=	Static bottom hole pressure (bara or psia). This is the bottom hole pressure at zero flow rate.
PERMEABILITY=	Average formation permeability (md).
THICKNESS=	Average formation thickness (metres or feet) .
RADEXTERNAL=	Radius of external boundary of drainage area (metres or feet). Default = 609.6 m or 2000 ft.
SKIN=	Dimensionless skin factor. Default = 0.
DIAMWELLBORE=	Wellbore diameter (inches or mm).
FLOWTYPE=	Type of flow in well. LIQ Liquid flow GAS Gas flow
GASMETHOD=	PSEUDO Use pseudo pressure SQUARED Use pressure squared
SOURCE=	ST Use the stock tank flow formulation of the transient IPR equation (default) RES Use the reservoir flow formulation of the transient IPR equation
BPCORRECTION=	Allows user to choose whether to apply the Vogel correction below the bubble point. ON Applies Vogel correction. OFF Does not apply Vogel correction.
PICOEFFICIENT=	Vogel coefficient for Vogel correction. Default = 0.8.
DSKINLIQUID=	Dynamic skin for liquid phase. Default = 0.

DSKINGAS=	Dynamic skin for gas phase. Default = 0.
DRAINAGESKIN=	
POROSITY=	Porosity of the reservoir, fraction (0 - 1)
COMPRESSIBIL=	Total system compressibility (1/psi or 1/bar).
TIME=	Time since the well started to flow, hours.
SWAPTOPSS=	
	ON
	OFF

5.6 SYSTEM DATA

- [PIPE \(p.699\) Pipe Dimensions](#)
- [INLET \(p.623\) System Inlet Data](#)
- [EQUIPMENT \(p.443\) Equipment Data](#)
- [NODE \(p.698\) System Profile Data](#)
- [Changing Parameters within the System Profile \(p.606\)](#)
- [SLUG \(p.640\) Slug Calculation Options](#)
- [COMPRESSOR \(p.679\) Compressor](#)
- [CHOKE \(p.673\) Choke](#)
- [EXPANDER \(p.683\) Expander](#)
- [HEATER \(p.687\) Heater/Cooler](#)
- [PUMP \(p.701\) Pump](#)
- [ESP \(p.701\) Electrical submersible Pump](#)
- [PCP \(p.465\) Progressive Cavity Pump](#)
- [MPUMP \(p.696\) Multiphase Pump](#)
- [PUMPCRV \(p.695\) Pump Performance Curves](#)
- [COMPCRV \(p.677\) Compressor Performance Curves](#)
- [SEPARATOR \(p.705\) Separator](#)
- [EROSION \(p.639\) Erosion Rate and Velocity](#)
- [CORROSION \(p.639\) Corrosion Rate](#)
- [COMPLETION \(p.651\) Completion Profile Delimiter](#)
- [TUBING \(p.651\) Tubing Profile Delimiter](#)
- [FLOWLINE \(p.651\) Flowline Profile Delimiter](#)

[RISER](#) (p.651) Riser Profile Delimiter
[FMPUMP](#) (p.686)
[REINJECTOR](#) (p.704)
[MPBOOSTER](#) (p.695)

5.6.1 CHOKE (Optional)

Also refer to [Choke Theory](#) (p.444)

Main-code: CHOKE

All subcodes are optional unless noted otherwise:

DBEAN=	Required. Diameter of the choke bean (mm or inches)
DBEAN64=	An alternative to DBEAN=, allows the bean diameter to be specified in units of 1/64 in.
CCORR=	Selects the Critical flow correlation. May be one of:
GILBERT	Gilbert (p.451) correlation
ROS	Ros (p.451) correlation
BAXENDELL	Baxendell (p.451) correlation
ACHONG	Achong (p.451) correlation
PILEHvari	Pilehvari (p.451) correlation
ASHFORD	Ashford and Pierce (p.450) correlation
ASHFORDT	Sachdeva (p.450) correlation
POETBECK	Poetmann and Beck (p.450) correlation
OMANA	Omana (p.450) correlation
THEORY or MECHANISTIC	(default) The MECHANISTIC choke model is purely theoretical, based on a combination of Bernoulli's equation with an equation of continuity. Advanced users may wish to 'fine-tune' the model, or override some of the calculated values, by means of the sub-codes CD, CSP, CPCV, YCRIT, GASCP, and LIQCP.
USER	User-supplied (p.451) correlation. This uses the same equation as the Gilbert,Ros (p.451) and so on correlations, but with parameters supplied with the subcodes A=, B= C= and E=.

SCCORR=	Selects the required sub-critical flow correlation. Maybe one of:
ASHFORD	= Ashford and Pierce (p.450) correlation
MECHANISTIC	(default) The MECHANISTIC choke model (see above).
API14B	This is a special-case of the MECHANISTIC model, wherein the GAS and Liquid Csp values are preset to 0.9 and 0.85 respectively.
CPRATIO=	Pressure ratio at which flow through choke becomes critical. (Default = 0.53). (It is also possible to force PIPESIM to calculate the Critical Pressure Ratio; to do this, enter CPRATIO=0.0.)
TOL=	Percentage tolerance, for identification of critical flow conditions. (Default 0.5%)
CD=	Discharge coefficient (default = 0.6). This value is used to calculate the flow coefficient, CSP.
CSP=	Flow coefficient. This is normally calculated by PIPESIM, but can be overridden, if desired, by use of this sub-code. The valid range is 0 to 1.3, typically it is 0.6. It is used to calculate the pressure drop.
CPCV=	Fluid-specific heat ratio, $\gamma = C_P / C_V$. This is normally calculated by PIPESIM, but can be overridden if desired. The valid range is 0.7 to 2. Typically it is 1.26 for a natural gas, for a diatomic gas it is 1.4. It is used to calculate the critical pressure ratio, if CPRATIO=0.0 is specified.
YCRIT=	Gas expansion factor at critical flow. This is normally calculated by PIPESIM, but can be overridden if desired. The valid range is 0.5 to 1. It is used to modify the pressure drop equation to allow for gas compressibility.
GASCSP=	Flow coefficient for the gas phase. This is normally equal to the value calculated or input for CSP, but can be overridden if desired. For API14B compatibility, set it to 0.9.
LIQCSP=	Flow coefficient for the liquid phase. This is normally equal to the value calculated or input for CSP, but can be overridden if desired. For API14B compatibility, set it to 0.85.
CRITERION=	Allows the reasons for identification of critical, and supercritical, flow to be defined. This subcode will accept one or more of the following values, supplied in multiple value syntax (p.599) :

	PRATIO	Pressure ratio < critical pressure ratio (default)
	FLOWRATE	Flowrate flowrate at critical flow
	SONICUP	Upstream velocity sonic
	SONICDOWN	Downstream velocity sonic
	ALL	All of the above
	NONE	The value NONE will effectively prevent the identification of critical and supercritical flow, thus flow will always be subcritical. NONE should be used for API14B compatibility.
VERBOSE=	ON or OFF	(Default OFF). Allows detailed choke calculation output for the MECHANISTIC correlation. The detailed output appears on the user's terminal screen and on the primary output page. This output is intended primarily to aid the development and debugging of the choke model, but can also be of use to the advanced user.
SCADJUST ADJUSTSC=	ON or OFF	If ON, the selected sub-critical correlation is adjusted to ensure it predicts a flowrate at critical pressure ratio that matches that predicted by the critical correlation. Default is OFF.
A=, B=, C=, E=		Parameters for the USER critical correlation.
IDPIPE= or DPIPE= or PIPEID=		The diameter of the upstream pipe section (in or mm). It is used to calculate the Diameter Ratio Δ (For more information, refer to Choke Geometry (p.444) .) This subcode is required only if the choke is present in a branch and there is no other accompanying pipe equipment. If the subcode is provided, PIPESIM uses that value instead of the ID from any existing upstream pipe.
MAXMASS=		Maximum mass rate (lb/sec or Kg/sec).
MAXGAS=		Maximum gas rate (mmscfd or mmsm3d).
MAXLIQUID=		Maximum gross liquid rate (sbb/day or sm3/day).
MAXOIL=		Maximum oil rate (sbb/day or sm3/day).
MAXWATER=		Maximum water rate (sbb/day or sm3/day).

Note: Notes for rate limit subcodes:

- All rate limit values refer to the fluid at stock-tank pressure and temperature, 14.7 psia and 60F. There is no provision for limiting phase flowrates at flowing or in-situ pressure and temperature.

- Application of any rate limit will result in the choke bean diameter being reduced from its supplied value (in the DBEAN= subcode). This will cause an additional pressure drop in the system or branch.
 - Any combination of rate limits may be specified. The choke will be sized to ensure that none of them are exceeded.
 - In a single-branch model, the rate limits will be applied at the choke where they are specified.
 - In a network model, the limits will be "promoted" to the network branch level, and will be treated as though they were specified on the [BRANCH \(p.762\)](#) statement. The network solution algorithm will apply the rate limits at the inlet of the branch containing the choke.
 - In a network model, the rate limits will apply regardless of the direction of fluid flow in the branch
-

The choke model will calculate the pressure ratio across the choke for the current flow rate. The pressure ratio calculated is then categorized in one of 3 ways:

Subcritical

The pressure ratio (P_{out}/P_{in}) is higher than the critical pressure ratio. PIPESIM continues the case with the calculated pressure drop.

Critical

The pressure ratio is within the tolerance of the critical pressure ratio. PIPESIM continues the case with the calculated pressure drop, and writes an explanatory message to all output pages. Pressures calculated in the profile from this point on represent maximum values rather than true values; in reality, the pressures could be less than those reported. The reason for this is that in critical flow, the flow rate is independent of the system's downstream pressure.

Supercritical

The pressure ratio is lower than the critical pressure ratio. This represents a situation that cannot occur in reality, therefore PIPESIM will abort the case or iteration. In a non-iterative case, this will result in a CASE ABORTED message, but in an iterative case, a further iteration is started, at a higher inlet pressure or lower flow rate.

PIPESIM does not attempt to model the entire system analytically, rather it breaks it down into small elements, each of which are then analyzed in turn to achieve the desired answer. Because the system's heat balance is calculated rigorously at every element, it is not possible for PIPESIM to work backwards up the system profile from a known outlet pressure. Consequently, if the user fixes the outlet pressure, an iterative solution to the case is required. For non-iterative cases, PIPESIM starts its analysis of the profile with a fixed inlet pressure and flow rate. Iterative cases are made up of several passes down the profile, with the iteration routine taking informed guesses at the inlet pressure or flow rate; thus, for each separate iteration, the inlet pressure and flow rate are effectively fixed, as for the non-iterative case.

When PIPESIM encounters a choke in the profile, it evaluates the pressure drop across the choke, and labels it as one of critical, subcritical, or supercritical. The supercritical condition means that the current flow rate cannot pass through the choke with the current upstream pressure: i.e., it is a situation that cannot occur in reality. (Another example of such an impossible situation is a

negative pressure; PIPESIM must cope with this too). PIPESIM deals with this by aborting the case or iteration. If the case is iterative, the iteration routine will then guess a lower flow rate or higher inlet pressure, and another pass down the profile will begin. The critical condition is comparatively difficult to hit; it means that, at the current pressure upstream of the choke, the flow rate is very close to the maximum possible flow rate through the choke. When critical conditions exist, the flow rate is independent of the downstream pressure. The subcritical condition needs no special handling.

In an iterative case, it often happens that the user's specified outlet pressure cannot be met. This occurs when a choke in the profile is in critical flow. Any increase in the flow rate will result in supercritical flow through the choke, and so this sets an upper limit for the iteration routine. However, the outlet pressure for the flow rate that gives critical flow might be much higher than that required. Normally in this situation, the iteration routine would increase the flow rate and try again, but the presence of the choke in critical flow makes this pointless. Therefore, the iteration routine considers the case to have converged on a solution, and prints the case results. The pressure profile on the downstream side of the choke, while it does not represent the actual required solution, nevertheless represents the maximum pressure that can be achieved there; in reality, the pressures will be lower.

A wellhead choke or bean is used to control the production rate from a well. In the design of tubing and well completions one must ensure that neither the tubing nor the perforations control the production from the well. The flow capacity of the tubing and perforations always should be greater than the inflow performance behavior of the reservoir. It is the choke that is designed to control the production rate from a well. Wellhead chokes usually are selected so that fluctuations in the line pressure downstream of the choke have no effect on the well flow rate. To ensure this condition, flow throughout the choke must be at critical flow condition; that is, flow through the choke is at acoustic velocity. For this condition to exist, downstream line pressure must be approximately 0.55 or less of the tubing or upstream pressure. Under this conditions the flow rate is a function of the upstream or tubing pressure only.

Chokes are subjected to sand and gas cutting as well as asphalt and wax deposition, which changes the shape and size of the choke. This then could result in considerable error when compared to calculate values of choke for a standard choke size. A small error in the choke size caused by a worn choke can produce a much larger error in the predicted oil rate. Thus a 'cut' choke could result in estimated oil rates considerably lower than measured.

From the inflow performance relationship of a well and by knowing the tubing size in the well, the tubing pressure curve for various flow rates can be calculated.

5.6.2 COMPCRV and PUMPCRV: Compressor and Pump performance curves

Main-code: PUMPCRV or COMPCRV

Centrifugal pump and compressor performance curves are specified as a range of head and efficiency or power values versus volumetric flow rates. These values should be specified before the profile, and each curve is given a name so it can be referenced on a subsequent PUMP or COMPRESSOR statement.

NAME= Required: The name of the curve, for referencing on a subsequent PUMP or COMPRESSOR statement.

SPEED=	The speed for which this curve was generated. (rpm)
Q=	Values of flow rate, supplied as a multiple value set (p.599) . Flowrates are measured in volumetric terms at the flowing pressure and temperature at the inlet to the device. For a pump curve, units are bbl/day or m ³ /day; for a compressor curve, they are ft ³ /min or m ³ /sec. (See note below).
FLOWRATE=	Synonym for Q =.
HEAD=	Values of head, supplied as a multiple value set (p.599) . For a pump curve, units are feet or metres. For a compressor curve they are ft-lbf/lbm (foot-pounds force per pound mass) or Kj/kg. Note that the conversion factor between ft-lbf/lbm and feet is 1.
EFFICIENCY=	Values of efficiency (%), supplied as a multiple value set (p.599) . (Default is 100%). Exclusive with POWER=.
POWER=	Values of power (hp or kw), supplied as a multiple value set (p.599) . Exclusive with EFFICIENCY=.
STAGES=	For a Pump curve only, the number of stages for which this curve is defined (normally 1).
WHEELS=	For a compressor curve only, the number of compressor wheels for which this curve is defined.

The [multiple value sets \(p.599\)](#) supplied for Q=, HEAD=, and EFF= or POWER= sub-codes must contain at least 3, and no more than 30 values, separated by commas, and enclosed in parentheses. The values need not be entered in ascending or descending order, however there is a strict one-for-one correspondence between the values in each list, based on their position. Each list must contain the same number of values.

Since the multiple value lists can be quite lengthy, they may be supplied across more than one line in the input file. This can be achieved either by use of the continuation character &, or by repeating the maincode and curve name on each line. The examples below illustrate this, they both have the same effect:

Examples

Example 1

```
PumpCrv      name = GN7000      stages = 100      speed = 3600
PumpCrv      name = GN7000      Q      = (1250      ,3750      ,5800      ,7400      ,
9000      ,10666.7)
PumpCrv      name = GN7000      head = (4827.88  ,4176.29,3624.68,3160.3  ,
2338.76,971.734)
PumpCrv      name = GN7000      eff   = (23.6865  ,
55.0738,67.3463,71.3873,63.476 ,31.7383)
```

Example 2

```
PumpCrv      name = GN7000      stages = 100      speed = 3600      &
Q = (1250 , 3750 , 5800 , 7400 , 9000 , 10666.7) &
head = (4827.88 , 4176.29, 3624.68, 3160.3 , 2338.76, 971.734) &
eff = (23.6865 , 55.0738, 67.3463, 71.3873, 63.476 , 31.7383)
```

5.6.3 COMPRESSOR Compressor (Optional)

Main-code: COMPRESSOR

Sub-codes: TYPE, ROUTE, POUT, DP, PRATIO, POWER, EFF, NAME, SPEED, WHEELS, STONEWALL, VERBOSE

TYPE=	COMPRESSOR type.
	CENTRIFUGA Centrifugal compressor
ROUTE=	ADIABATIC Adiabatic compression is performed (default). For black oil models the heat capacity ratio (C_p/C_v) for the adiabatic exponent in the compression equations is assumed to be constant with a value equal to 1.26. For compositional models the heat capacity ratio is calculated using the relationship: $C_p = C_v - R$ The heat capacity is obtained at the average of the compressor suction and discharge conditions.
	POLYTROPIC Polytropic compression is performed. The heat capacity ratio (C_p/C_v) is calculated as outlined above for ADIABATIC compression. This is the default for Black Oil cases.
	MOLLIER Compression is based upon the Mollier method, that is isoentropic compression from suction to discharge pressures. This option is valid for compositional models only, where it is also the default.
POUT=	Discharge pressure from the compressor (bara or psia) (default 20,000 psia)
DP=	Pressure differential across the compressor (bar or psi) (default 10,000 psi)
PRATI=	Compressor pressure ratio (default 1000)
POWER=	Power available for compression (KW or hp) (default unlimited)
EFF=	Overall efficiency of the compression (default = 100%)
NAME=	The name of a previously specified Compressor Curve (see the COMPCRV (p.677) maincode)

SPEED=	Speed at which the compressor will run. This is only useful when a compressor curve name has been specified. (rpm) (default unlimited)
WHEELS=	
STONEWALL=	
VERBOSE=	

Notes:

1. At least one of the subcodes POUT, DP, PRATIO, POWER and SPEED should be supplied. If 2 or more are present, PIPESIM will treat them as upper limits, and will use whichever gives the smallest DeltaP. The others will be recalculated and displayed as answers on the output file.
 2. If a compressor curve name is supplied, the speed may also be specified. This is used to adjust the supplied curve against its specified speed (as set with [SPEED \(p.677\)](#) = on the [COMPCRV \(p.677\)](#) maincode). The adjustment is done using the so called affinity or fan laws, which state that "capacity is directly proportional to speed, head is proportional to square of speed, and power is proportional to cube of speed".
 3. In order to avoid confusion with the COMPOSITION main code, the minimum abbreviation acceptable for COMPRESSOR is COMPR.
-

5.6.4 RODPUMP: Rod- or Beam-pump

Main-code: RODPUMP or CVFMD

The RODPUMP statement allows the outline specification for a Rod- or Beam-pump to be supplied.

A rod-pump is an example of a Constant Volume Fluid Motive Device (CVFMD). CVFMDs are fixed-volume, positive-displacement pumps or compressors designed to move liquids, gases or 2-phase mixtures. Other notable CVFMD examples are: Progressive cavity pumps, Twin screw multiphase boosters, and reciprocating compressors. The statement allows a simplistic simulation of such a device to be performed.

NOMLIQRATE=	The flowing volume flowrate that the pump would produce, if it were pumping with no back-pressure at its discharge (m ³ /day or bbl/day).
SLIPCOEF=	A coefficient to specify the change in flowrate with respect to Delta pressure (m ³ /day/bar or bbl/day/psi). This is used to compute the pressure rise across the device when the actual flowrate is less than the specified nominal rate.
MAXDP=	Maximum pressure rise the device is allowed to exhibit (psi or bar). This is used to prevent excess rod loading.
MAXPOWER=	Maximum power the device is allowed to draw (hp or kw). This is used to prevent excess rod loading.

RODDIAMETER=	The Diameter of the drive rod (in or mm). The drive rod will be assumed to exist in the downstream pipe or tubing, and will stretch up to the wellhead or the end of the tubing. The fluid will flow in the annular space between the tubing ID and the rod OD. The rod diameter can be adjusted in downstream pipe sections by use of the RODDIAM= subcode on the PIPE (p.699) statement, this is useful to simulate taper rods.
VOLUME=	The swept volume of the pump cylinder, i.e. its cross-section area multiplied by the stroke length (bbl or m ³). In conjunction with SPEED=, this is an alternative to the nominal rate.
SPEED=	The pump speed, in strokes per minute. In conjunction with VOLUME=, this is an alternative to the nominal rate.
NOMINALRATE=	The flowing volume flowrate that the pump would produce, if it were pumping with no back-pressure at its discharge (m ³ /sec or ft ³ /min). This is the same information as NOMLIQRATE= but in different units, more suited to other types of CVFMD.

5.6.5 EQUIPMENT Generic Equipment

Main-code: EQUIPMENT

The EQUIPMENT maincode may be used to simulate a generic unit operation in which the pressure and/or temperature of the stream are modified.

DP=	Pressure gain (positive), or loss (negative). (Bar or psi). NB. In a network model, the DP is assumed to follow the flow direction in the branch, so if the branch flow reverses, the DP will change sign. This can be controlled with DPIFD=, see below. See note 2
DPIFD=	"DP is Independent of Flow Direction". Set this to YES to ensure the sign of the dp supplied with DP= is independent of branch flow direction in a network model. Default is NO, thus if the branch flow reverses, the dp will change sign. An example of a device whose DP is direction -independent is a choke. An example of a direction-dependent DP is a vertical section of pipe.
POUT= or SETP=	Equipment outlet set pressure (bara or psia). NB. In a network model, the imposition of a set pressure is likely to prevent the model from converging. In a single-branch model, if the outlet pressure and flowrate are fixed, the use of SETP= will cause an input data error. See note 2
PRATIO=	Pressure Ratio: the equipment outlet pressure is set to the specified multiple of the inlet pressure. Exclusive with MAXP=, MINP=, SETP= and DP=. See note 2.
MAXP=	Maximum pressure (psia or bara). If the pressure at the equipment is greater than the supplied limit, then it will be adjusted down to the limit. See note 3.
MINP=	Minimum pressure (psia or bara). If the pressure at the equipment is less than the supplied limit, then it will be adjusted up to the limit. See note 3.

ROUTE=	Allows the thermodynamic route to be specified, for calculation of fluid temperature change consequent upon changes in pressure. Exclusive with DT= and SETT=. can be used with DUTY=. See note 4. Available choices are: ISENTHALPIC or ADIABATIC: Constant enthalpy (default) ISENTROPIC or MOLLIER: Constant entropy (compositional cases only) ISOTHERMAL: Constant temperature
DT=	Temperature gain (+ve), or loss (-ve). (C or F). See note 1.
TOUT= or SETT=	Outlet set temperature ($^{\circ}\text{C}$ or $^{\circ}\text{F}$). See note 1.
DUTY=	Duty to be used to raise the temperature of the fluid (KW or Btu/hr). See notes 1, 4 and 6.
MAXT=	Maximum temperature (F or C). If the temperature at the equipment is greater than the supplied limit, then it will be adjusted down to the limit. See note 7.
MINT=	Minimum temperature (F or C). If the temperature at the equipment is less than the supplied limit, then it will be adjusted up to the limit See note 7.
NAME=	Defines the NAME of a user-supplied equipment entrypoint, as defined in a prior USERDLL statement. Note: The presence of this subcode causes all others (except OPTIONS=) to be ignored.
OPTIONS=	A character string that is supplied to the user-supplied equipment routine. Must be used with NAME=.
VERBOSE=	Controls the appearance of the one-line output in the report. may be set to ON or OFF, default ON.

Notes:

1. The subcodes SETT=, DT= and DUTY= are mutually exclusive.
2. The subcodes SETP=, DP= and PRATIO= are mutually exclusive.
3. If a MAXP= or MINP= is specified, the limit is applied AFTER any pressure change resulting from a SETP=, DP= or PRATIO=, and BEFORE any temperature or enthalpy change is applied or calculated.
4. If SETT= or DT is specified, the fluid outlet temperature will be set accordingly, otherwise it will be calculated using the selected (or defaulted) thermodynamic ROUTE= and DUTY=.
5. If SETP=, DP= or PRATIO= are specified in the absence of TOUT and DT, the choice of thermodynamic route is used to calculate the fluid outlet temperature. To simulate chokes and to predict Joule-Thomson cooling across pressure reduction valves etc. the most appropriate route is ISENTHALPIC (the default).

6. If a DUTY is specified, the corresponding fluid enthalpy change will be calculated, and added to that resulting from any pressure change using the selected ROUTE=. The outlet temperature is then adjusted accordingly.
7. If a MAXT= or MINT= is specified, the limit is applied AFTER any temperature change resulting from any other subcode.
8. All subcodes are optional.

Examples

Example 1

A pipeline compressor station (located at distance 120 Km and elevation 20 m) raises the pipeline pressure 35 bar and after coolers cool the compressed gas down to 40 C before it reenters the pipeline. Pipeline gas is withdrawn to power the compressors, so a RATE statement is used to subtract 2.5 kg/sec from the pipeline. The following three lines define the compressor station:

```
NODE      DIST= 120 km   ELEV= 20 m
EQUIPMENT DP = 35 bar   SETT = 40 C
RATE       ADDMASS = -2.5 kg/sec
```

Example 2

A wellhead choke is to be set to reduce the calculated wellhead pressure to 60 bara. The program will calculate the resulting temperature change across the choke (assuming an isoenthalpic expansion) :

```
EQUIPMENT SETP = 60 bar
```

5.6.6 EXPANDER Expander (Optional)

Main-code: EXPANDER

DP= Pressure differential across the expander (bar or psi) (default 10,000)

POUT= Discharge pressure from the expander (bara or psia) (default 20 psia)

PRATIO= Expander pressure ratio (Pin/Pout ; default 1000)

EFF= Overall efficiency of the expansion (%) (Default = 100%)

POWER= Power required from expansion (KW or hp) (default unlimited)

ROUTE=

ADIABATIC Adiabatic expansion is performed (default). For black oil models the heat capacity ratio (C_p/C_v) used as the adiabatic exponent in the expansion equations is assumed to be constant with a value

equal to 1.26 . For compositional models the heat capacity ratio is calculated using the relationship : $C_p = C_v - R$. The heat capacity is obtained at the average of the expander suction and discharge conditions.

POLYTROPIC Polytropic expansion is performed. The heat capacity ratio (C_p/C_v) is calculated in a similar manner to that outlined above for ADIABATIC expansion. This is the default for Black Oil cases.

MOLLIER Expansion is based upon the Mollier method, that is isoentropic expansion from suction to discharge pressures. This option is valid for compositional models only, where it is the default.

UNDEFINED Undefined

NAME=

SPEED=

WHEELS=

STONEWALL=

ON

OFF

VERBOSE=

ON

OFF

Note: At least one of the subcodes POUT, DP, PRATIO, and POWER should be supplied. If 2 or more are present, PIPESIM will treat them as upper limits, and will use whichever gives the smallest DeltaP. The others will be recalculated and displayed as answers on the output file.

5.6.7 FITTING : Valves and Fittings

Main-code: FITTING

Pipe fittings can be modeled in PIPESIM using the FITTING keyword. Either a resistance coefficient should be specified, or the type and dimensions of the fittings should be specified and the resistance will be calculated.

TYPE= "GLOBE-CONV" Globe Valve Conventional
"GLOBE-YPAT" Globe Valve Y-Pattern
"ANGLE-CONV" Angle Valve Conventional

"CHECK-SWING1"	Check Swing Valve Conventional
"CHECK-SWING2"	Check Swing Valve Clearway
"CHECK-LIFT1"	Check Lift Globe Valve
"CHECK-LIFT2"	Check Lift Angle Valve
"BALL-VALVE"	Ball Valve
"GATE-VALVE"	GateValve
"ELBOW-STD45"	Standard 45 degree Elbow
"ELBOW-STD90"	Standard 90 degree Elbow
"ELBOW-LR90"	Standard 90 degree Long Radius Elbow
"ELBOW-SR90"	Standard 90 degree Short Radius Elbow
"TEE-RUN"	Tee - Flow through run
"TEE-BRANCH"	Tee - Flow through branch
NOMINALD	Nominal diameter (mm or inches)
MINORD=	Minor internal diameter (mm or inches)
MAJORD=	Major internal diameter (mm or inches)
DANGLE=	Deflection angle
KVALUE =	Resistance coefficient

If the resistance K is not specified it will be calculated. In this case the [nominal diameter \(p.567\)](#) must be specified, together with the internal diameter of the fitting. Some fittings require two internal diameters to be specified, the minor diameter d_1 at a constriction and the major diameter d_2 .

If the resistance is specified, one of the internal diameters (d_1 or d_2) should also be specified. If both internal diameters are specified, the resistance is assumed to apply at the major internal diameter d_2 , that is $K = K_2$.

See the [Technical description \(p.452\)](#) for further details.

EXAMPLES

The fitting keyword can be inserted between node keywords in a pipe. The fitting is placed immediately after the preceding node.

```

...
node      dist = 0 elev = 0
node      dist = 1000 elev = 0
fitting kvalue = 1.6 majord = 4.815
node      dist = 1010 elev = 0
fitting type = "ANGLE-CONV" nominald = 5 majord = 5.047 minord = 4 dangle =
45
node      dist = 1200 elev = 0
...

```

5.6.8 FMPUMP (Optional)

Main-code: FMPUMP

5.6.9 FRAMO 2009 (Optional)

See also [Multiphase Boosting Technology \(p.470\)](#), [Framo 2009 Helico-Axial Multiphase Booster \(p.105\)](#).

Main-code: FRAMO2009

This keyword requests a Framo pump, modelled with the fram02009 dll provided by Framo.

POUT=	Discharge pressure from the pump (bara or psia)
DP=	Pressure differential across the pump (bar or psi)
PRATIO=	Pump pressure ratio
POWER=	Power available for pump (KW or hp)
LIMSPEED=	Maximum pump speed (<i>valid range 0.2 – 1.0</i>)
TUNE=	Tuning parameter (<i>valid range 0.7 – 1.5</i>)
QRECIN=	Flow in recirculation
NPARA=	Number of pumps in parallel (<i>valid range 1-7</i>)
NAME=	The name of the pump
FILE=	Framo file containing pump performance curves. This sub-code must be specified. Only the file name should be specified, not the path. The file must exist in the fram09 sub-directory of the PIPESIM data directory (default location “C:\Program files\Schlumberger\PIPESIM\data\fram09”).
PLOT	<p>Requests a pump performance plot. A plot file called “file_n.pfm” will be created in the model directory, where “file” is the pump name specified by the NAME sub-code and “n” is the case number.</p> <p>The pump name should be unique, otherwise plot files will over write each other.</p> <p>If the NAME sub-code is not given, the plot file will be called “framopump_n.pfm”.</p> <p>Pump performance plot files can be viewed by PSPlot.</p>

EXAMPLE

```
framo2009 file='framopump.dat' name='test' dp=100 plot
```

5.6.10 HEATER Heater/Cooler (Optional)

Main-code: HEATER

DP= Pressure drop across the heater (bar or psi) (Default =0)
 POUT= Discharge pressure from the heater (bara or psia)
 PRATIO= Pressure ratio across the heater
 TOUT= Discharge temperature from the heater ($^{\circ}$ C or $^{\circ}$ F)
 DT= Temperature drop/increase through the heater/cooler ($^{\circ}$ C or $^{\circ}$ F)
 VERBOSE=
 STATUS=

Notes:

1. The subcodes TOUT DT and DUTY are mutually exclusive only one should be supplied. Specification of the outlet temperature or DT will result in the calculation of the duty required to meet these conditions. If the DUTY is supplied PIPESIM will calculate the outlet temperature.
2. The subcodes POUT and DP are mutually exclusive and optional. Changes in pressure across a heater are modeled using an isoenthalpic route; if large pressure changes are required you are better served by modeling the DP with a separate EQUIPMENT maincode which gives a choice of route.

5.6.11 GASLIFT: Multiple Injection Ports in Gaslifted Wells

The GASLIFT statement is required to simulate the multiple gas lift valves resulting from a Gas Lift Design in the PIPESIM GUI. It is also used to calculate the Deepest point of Injection (DIP).

GASLIFT should be entered in the initial part of the input file, ie. before the profile or the first NODE statement. It specifies generic gas lift data such as casing head pressure (CHP), injection gas flow rate and properties, etc. The profile may then include one or more INJPORT statements to specify the position and properties of the gas lift valves.

Two modes of operation are available, viz. Deepest Injection Point (DIP), and Simulate; these are selected with the MODE= subcode.

In DIP mode, PIPESIM will calculate the deepest possible injection point for the system as specified. Any existing injection points and gas lift valves will be ignored. Required subcodes for this mode are INJGASRATE=, DP=, CHP= and TEMP=.

In SIMULATE mode, PIPESIM will accept a set of gas lift valves positioned at various depths in the system profile. Each valve has its associated settings such as Port diameter, test rack Pressure

Setting. etc. The program will simulate the entire system as specified, determining which valve(s) are operating, and calculate the injection gas flowrate through each valve. Required subcodes for this mode are MAXQ=, CHP=, and TEMP=.

For either mode, the recommended job will fix the reservoir pressure and outlet (wellhead) pressure, and calculate the system flowrate; i.e., use [ITERN \(p.621\)](#) TYPE=LFLOW. (Other iteration types are more difficult to use since the design of the gaslift system constrains the Injection valves to a narrow range of operating pressure, outside which no gas injection will occur, rendering meaningful results unlikely.)

Main-code: GASLIFT

MODE=	Required. Specifies the operational mode for the gaslift system. Available modes are: SIMULATE: requests a rigorous simulation of the system with a set of gas lift valves installed. The important feature of this mode is that it calculates the total flowrate of gas that is injected into all the valves, using the positions and settings of the installed gas lift valves. Be aware however, that this mode is very sensitive to the exact position and settings of the valves. Consequently its results are usually difficult to interpret due to the likelihood multiple solutions, or no solution, to the model cases. For this reason, PIPESIM uses one of the DIP modes instead (see below). Injection valves must be provided at appropriate depths with INJPORT (p.691) statements. DIP: requests a calculation of the Deepest Injection Point (DIP). In this mode, any existing Injection valves will be ignored. The program will simulate the injection of gas at the maximum possible depth for the given system parameters, and report the calculated DIP depth. In this mode the fixed flowrate of gas as specified with INJGASRATE= is injected at the deepest depth. DIP3: similar to DIP mode, but the calculated deepest injection depth is constrained to be coincident with the position of one of the existing injection valves specified with INJPORT statements. See note 2.
MAXFLOWRATE= or MAXQ=	Required for MODE=SIMULATE. The maximum available lift gas flow rate (MMscf/d or MMm ³ /d).
TEMPERATURE=	Required. The temperature of the lift gas (F or C) at the casing head. The program will adjust this by a formula that takes account of geothermal temperature gradient and production temperature to calculate the annulus temperature for each injection valve.
CHP=	Required. The casing head pressure (psia or bara) at the wellhead where the lift gas is supplied to the well. The program will adjust this by the pressure of the static head of gas in the annulus between the wellhead and the injection port to calculate the annulus pressure for each injection valve.

DP=	Required for MODE=DIP. The minimum allowable pressure difference across the injection valve. The valve will be positioned at a depth that ensures the DP between casing and tubing is at least this value. (psi or bar)
FLOWRATE= or INJGASRATE=	For MODE=DIP. The flow rate of lift gas (MMscf/d or MMm ³ /d) to be injected. See note 1.
FTEMPERATURE=	Optional. A factor (f) that allows the injection port temperature (Tp) to be interpolated between the casing gas temperature (Tc) and the production wellbore temperature (Tw) using the formula $T_p = T_c * (1-f) + T_w * f$. Can be set to a value between 0 and 1, default 1. The injection port temperature is important in gas-charged valves because it determines the dome pressure and hence the valve opening and closing pressures.
PLOT=	Optional, can be set to OFF or ON, default OFF. Produces a plot file representing the performance characteristics of each valve. One file is produced for each valve, these are named model.Vxx, where model is the model file core name, and xx is the valve number (shallowest being 01). The valve performance is exercised over a range of casing and tubing pressures, and the plot typically has tubing pressure on the X-axis against gas flowrate in the Y-axis, with the casing pressure giving a number of different lines.
MINFLOWRATE= or MINQ=	Optional, for MODE=SIMULATE. The lower limit of gas flowrate to be injected (MMscf/d or MMm ³ /d). If the simulation for any case predicts less than this gas flowrate, additional gas will be injected at the shallowest valve. This simulates the effect of the operator increasing CHP to inject more gas.
FLUIDNAME= or USE=	Optional. The name of the fluid (Black Oil or Compositional) representing the lift gas fluid specification, as specified with a BEGIN FLUID (p.634) block.
SGGAS=	Optional. For Black Oil fluids only, the lift gas specific gravity. (Default = SG of the produced gas).
KGAS=	Optional. For Black Oil fluids only, The lift gas thermal conductivity. (Default = K of the produced gas).
CPGAS=	Optional. For Black Oil fluids only, The lift gas heat capacity. (Default = CP of the produced gas).
METHOD=	Optional. Specifies the equation used to calculate gas flowrate across the valve. Allowable values are 1 for PIPESIM's standard mechanistic choke correlation, and 2 for the Thornhill-Craver equation.
PVTFILE=	Optional. For compositional fluids only, the name of the PVT file containing the composition of the lift gas.

GLR=	For MODE=DIP, injects the gas flowrate required to make the production fluid Gas Liquid Ratio (GLR) equal the supplied value. (scf/sbbl or sm3/sm3) See note 1.
GOR=	For MODE=DIP, injects the gas flowrate required to make the production fluid Gas Oil Ratio (GOR) equal the supplied value. (scf/sbbl or sm3/sm3) See note 1.
INJGLR=	For MODE=DIP, injects the gas flowrate required to increase the production fluid Gas Liquid Ratio (GLR) by the supplied value. (scf/sbbl or sm3/sm3) See note 1.
INJGOR=	For MODE=DIP, injects the gas flowrate required to increase the production fluid Gas Oil Ratio (GOR) by the supplied value. (scf/sbbl or sm3/sm3) See note 1.
MAXDEPTH=	Optional, for MODE=DIP, limits the injection depth to the specified maximum.
FRICTION=	Optional, For Mode=DIP. Requests a rigorous treatment of injection gas pressure profile resulting from friction in the annulus. This can make a significant difference to the calculated DIP when gas rate is high or annulus cross-section area is small. Can be set to ON or OFF, default OFF.
U=	Optional, for FRICTION=ON, specifies the overall Heat Transfer Coefficient to be used for calculating injection gas temperature changes due to heat exchange with the tubing and casing. (btu/hr/ft ² /F or W/m ² /C). Default 20 btu/hr/ft ² /F.
IFACTOR=	Optional, for FRICTION=ON, A factor (f) that allows the ambient temperature (Ta) used in gas friction heat transfer calculations to be interpolated between the ambient rock temperature (Tr) and the production wellbore temperature (Tw) using the formula Ta = Tr*(1-f) + Tw*f. Can be set to a value between 0 and 1, default 0.9.
PRINTF=	Optional, for FRICTION=ON, requests detailed output from the simulation of gas flow in the annulus. The resulting output pages are similar to those produced for the production wellbore. Can be set to ON or OFF, default ON.
SIP=	Optional. Specifies the Surface Injection Pressure used for Alhanati Gas Lift Instability Criteria calculation . (psia or bara)

Note:

1. The subcodes FLOWRATE=, GLR=, GOR=, INJGLR= and INJGOR= are mutually exclusive.
 2. Any subcode valid for MODE=DIP is also valid for MODE=DIP3.
-

5.6.12 INJPORT Gas Lift Injection Valve

INJPORT statements specify the position of Gas Lift Injection Valves. In conjunction with the [GASLIFT \(p.687\)](#) statement they allow a well with multiple injection valves to be simulated. PIPESIM will calculate the production flowrate and the quantity of gas injected through each valve.

INJPORT statements should be positioned in the system profile immediately following the NODE statement representing the desired position for each valve. In a typical gas lift design, the profile will contain between 4 and 10 injection valves.

Main-code: INJPORT

DPORT=	The diameter of the valve port (in. or mm). Required.
PTR=	The Test Rack Pressure Setting for the valve, measured at test rack conditions, that is 60 °F and 14.696 psia. This is the pressure (applied to the casing side of the valve, with the tubing side open to atmosphere) required to just open the valve. Required.
MODE=	The valve's mode of operation. Required.
	DUMMY
	ORIFICE
	TUBING
	IPO
	PPO
	CASING
AP2AB=	The ratio of the Port Area to the Bellows Area, AP/AB, for the valve. Required.
CV=	The flow coefficient for the valve port. This is a value normally in the range 0.4 to 1.6, used to characterize the port in the equation for gas flow. The valve manufacturer usually measures this in the laboratory. Optional, default 0.6.
TYPE=	The construction type of the valve. (Optional,)
	DUMMY
	ORIFICE
	BELLOWS Default
	SPRING
LABEL=	An identifier for the valve.

TF=	Throttling factor for the valve (mscfd/psi or msm ³ d/bar). Optional, if not supplied it will be calculated using an assumed value of (P open-Pclose).
PCO=	Experimental: the Casing Opening pressure for the valve. If supplied this value will be used instead of the calculated value.
PTC=	Experimental: the Tubing Closing pressure for the valve. If supplied this value will be used instead of the calculated value.
PDT=	Experimental: the Dome Pressure of the valve at operating temperature. If supplied this value will be used instead of the calculated value.
TEMPERATURE=	Experimental: The valve operating temperature. If supplied this value will be used instead of the calculated value.

The values of TYPE= and OPMODE= determine the characteristics of the valve.

A Bellows valve has a dome and bellows, which is charged with gas (usually nitrogen) in the test rack to provide the required closing force on the valve plunger. The force exerted by the gas charge depends on its pressure, which increases with temperature. Since the temperature where the valve is installed in the tubing is much higher than test rack temperature, this pressure correction must be done using an accurate value for valve operating temperature if the valve simulation is to be relied upon.

A Spring valve has a spring instead of a bellows to provide the closing force on the valve plunger. The spring force is relatively insensitive to temperature variation.

An Orifice valve has no plunger, and is equivalent to a normal choke. It will always be open, regardless of tubing and casing pressure, thus in theory not only can gas flow from casing to tubing, but production fluid can also flow from tubing to casing. In practice this is not a problem as static head limits fluid buildup in the casing. An orifice valve is sometimes specified as the deepest injection valve, because it will not suffer from unwanted closure if the gas lift design no longer matches the system operating parameters.

A dummy valve is a plug that passes no gas.

Bellows and spring valves are sensitive to both tubing and casing pressure to a greater or lesser extent, depending on their construction and the way they are installed in the tubing string. The objective of valve design, placement and test rack pressure setting is to achieve a desired response to changes in tubing and casing pressure. These are called Modes of Operation, and have the following names and meanings:

IPO:

Injection Pressure operated. Valve will respond only to changes in injection gas pressure.
Response is off-on rather than proportional.

PPO:

Production Pressure Operated. Valve will respond only to changes in tubing pressure.
Response is off-on rather than proportional.

TUBING:

Tubing-sensitive proportional response. Valve will respond predominantly to changes in tubing pressure, exhibiting a proportional response.

CASING:

Casing-sensitive proportional response. Valve will respond predominantly to changes in casing pressure, exhibiting a proportional response.

5.6.13 INJGAS: Injection Gas (Optional) and INJFLUID: Fluid Injection

INJGAS and INJFLUID allow the injection and mixing of a side-stream fluid. They both work for Blackoil and Compositional fluids.

INJGAS simulates a single gas injection point, and allows a specified quantity of gas to be added to the production fluid at any position. This quantity can be expressed in volume or ratio terms. It differs from [GASLIFT \(p.687\)](#) in that it always injects the specified gas quantity, whereas [GASLIFT \(p.687\)](#) will calculate how much gas is injected.

INJFLUID simulates a single fluid injection point, and allows the mixing of the main stream with a side-stream of any required composition and phase. It allows a base pressure and temperature to be specified for the side stream, this is used to fix the volume flowrate if specified as a gas or liquid rate.

INJGAS will only allow the injected fluid to be gas, it will apply certain sanity limits when working in PIPESIM-NET mode, and it will calculate values for Alhanati Gas Lift Stability criteria. It also allows injected gas properties such as SG, Cp and K to be defined for a black oil fluid.

Main-code: INJGAS or INJFLUID

GASRATE=	Defines the flow rate of injection gas in volumetric terms (mmscf/d or mmsm ³ /d).
MASSRATE=	Defines the flow rate of injected fluid in mass terms (lb/sec or kg/sec).
LIQRATE=	INJFLUID only. Defines the flow rate of injected fluid in volumetric terms of its stock-tank liquid phase (sbb/d or sm ³ /day).
FLOWRATE= or RATE=	INJGAS only, a synonym for GASRATE=.
GOR= or GLR=	Defines the flowrate of injection gas in terms of Gas Liquid Ratio or Gas Oil Ratio. Sufficient gas will be injected to adjust the produced fluid's GLR or GOR to the specified value. If the fluid currently has a higher value, no gas will be injected.
INJGLR= or INJGOR=	Defines the flowrate of injection gas in terms of an increase in Gas Liquid Ratio or Gas Oil Ratio. Sufficient gas will be injected to increase the produced fluid's GLR or GOR by the specified value.
TEMP= or IPTEMP=	Temperature of injection fluid at the point of injection (F or C). If omitted, defaults to the temperature of the produced fluid at the injection point

SG=	INJGAS only, optional. For black oil fluids only, Specific gravity of the injection gas (default = Value of gas SG from produced fluid)
CP=	INJGAS only, optional. For black oil fluids only, Heat capacity of injection gas (default = Value of gas Cp from produced fluid)
KGAS=	INJGAS only, optional. For black oil fluids only, Thermal conductivity of injection gas (default = Value of gas K from produced fluid)
CHP=	INJGAS only, Optional. Casing head pressure to be used for Alhanati Gas Lift Stability Criteria calculation. (psia or Bara)
DSIC=	INJGAS only, Optional. Diameter of the Surface Injection Choke. Used for Alhanati Gas Lift Stability Criteria calculation (in. or mm). Exclusive with SIP=.
IDCT=	INJGAS only, optional: the presence of the IDCT= subcode signals that gas injection is occurring through coiled tubing. The value supplied is the Internal Diameter of the Coiled Tubing (in. or mm.). The Cullinder & Smith correlation is used to calculate the DP in the injection string, this is compared with the available Casing Head Pressure and tubing pressure, and insufficient CHP will trigger an informative message. NB: Subsequent flow up the tubing should be specified as ANNULAR, using an appropriate PIPE statement that supplies the correct annulus dimensions with AID= and AOD=. Coiled tubing being used as a 'velocity string' (i.e. with no injected gas) should be specified simply as annular flow with an appropriate PIPE statement as above, there is no need for any INJGAS statement.
SIP=	INJGAS only, optional. Surface Injection Pressure. Used for Alhanati gas Lift Stability Criteria calculation. (psia or Bara).
DP=	INJGAS only, Optional.. Injection port delta pressure (psia or Bara). used to calculate casing pressure for Alhanait Gas Lift Stability Criteria calculation. (psi or Bar). Exclusive with DPOR=.
DPOR=	INJGAS only, Optional. Injection port diameter. Used for Alhanati gas Lift Stability Criteria calculation. (in. or mm)
PVTFILE=	Optional. For compositional fluids only, the name of a PVT file containing the composition of the injected fluid. Exclusive with FLUIDNAME=, USE=, and STREAMNAME=.
FLUIDNAME= or USE=	Optional. The name of the fluid (Black Oil or Compositional) representing the injected fluid specification, as specified with a BEGIN FLUID (p.634) block. (A special-case fluid name of *SEP_DISCARD specifies that the injected fluid specification and flowrate is obtained from the discard stream of a separator (p.705) located somewhere upstream in the same branch. See also STREAMNAME= below.). Exclusive with PVTFILE= and STREAMNAME=.

HANDLE=

STREAMNAME= Optional, INJFLUID only. The name of the fluid stream representing the injected fluid specification, as specified on the DISCARDNAME= subcode of an upstream separator. The injected stream definition includes its fluid definition (Black Oil or Compositional), flowrate, and enthalpy. This feature provides the same functionality as *SEP_DISCARD described above, but in addition ensures the fluid enthalpy is conserved. Multiple separated streams may be re-injected within the same branch by ensuring they are defined with unique names. Exclusive with PVTFILE=, FLUIDNAME= and USE=.

PRINT= Optional. Requests a verbose printout of the specification of the injected fluid.

CHTEMP= INJGAS only, Optional. Injection gas temperature can be provided at the casing head (wellhead) as an alternative to TEMP=. This will be corrected to a temperature at the injection point by use of a formula that depends on geothermal gradient and production temperature. (F or C)

LIMIT= or
LIMITMR=

GLRLIMIT= or
LIMITGLR=

The quantity of gas or fluid to be injected must usually be specified, either as a volumetric or mass rate, or as a ratio with the liquid phase of the produced fluid. Thus one of the subcodes GASRATE=, MASSRATE=, LIQRATE= GOR=, GLR=, INJGOR= or INJGLR= should be specified. However, if STREAMNAME= or *SEP_DISCARD is used, the flowrate is obtained from the upstream separator..

Mixing of the injected and the produced fluid is assumed to occur at the pressure which pertains at the injection point during the simulation. The temperature of the fluid after mixing is calculated by a heat balance around the mixing point.

Alhanati gas-lift stability criteria are calculated for a gas lift system and added to the Plot file for use by the Well Optimization feature, if sufficient information is provided on the INJGAS statement. Any two of the subcodes SIP=, DPORT=, CHP=, DP= and DSIC= should be provided, preferably the first two. The others will then be calculated.

5.6.14 MPBOOSTER (Optional)

Main-code: MPBOOSTER

TYPE= Multiphase pump type.

GENERIC

TWINSCREW

VENDOR

NAME=	The pump name.
ROUTE=	
ADIABATIC	Adiabatic compression is performed (default). For black oil models the heat capacity ratio (C_p/C_v) for the adiabatic exponent in the compression equations is assumed to be constant with a value equal to 1.26. For compositional models the heat capacity ratio is calculated using the relationship: $C_p = C_v - R$. The heat capacity is obtained at the average of the compressor suction and discharge conditions.
POLYTROPIC	Polytropic compression is performed. The heat capacity ratio (C_p/C_v) is calculated as outlined above for ADIABATIC compression.
MOLLIER	Compression is based upon the Mollier method, that is isoentropic compression from suction to discharge pressures. This option is valid for compositional models only
POUT=	Discharge pressure from the multiphase pump (bara or psia) (default 20,000 psia)
DP=	Pressure differential across the pump (bar or psi) (default 10,000 psi)
PRATIO=	Multiphase pump pressure ratio (default 1000)
POWER=	Power available (KW or hp) (default unlimited)
PUMPEFF=	Efficiency of compression of the gas phase (default 100%)
SPEED=	Pump speed
VISCORR=	
STATUS=	Status of the pump.
	GENERIC
	TWINSCREW
	VENDOR

5.6.15 MPUMP Multiphase Pump (Optional)

Main-code: MPUMP

TYPE= Multiphase pump type.

CENTRIFUGA Centrifugal pump.

NAME= The pump name.

ROUTE=

ADIABATIC Adiabatic compression is performed (default). For black oil models the heat capacity ratio (C_p/C_v) for the adiabatic exponent in the compression equations is assumed to be constant with a value equal to 1.26. For compositional models the heat capacity ratio is calculated using the relationship: $C_p = C_v - R$. The heat capacity is obtained at the average of the compressor suction and discharge conditions.

POLYTROPIC Polytropic compression is performed. The heat capacity ratio (C_p/C_v) is calculated as outlined above for ADIABATIC compression.

MOLLIER Compression is based upon the Mollier method, that is isoentropic compression from suction to discharge pressures. This option is valid for compositional models only

POUT= Discharge pressure from the multiphase pump (bara or psia) (default 20,000 psia)

DP= Pressure differential across the pump (bar or psi) (default 10,000 psi)

PRATIO= Multiphase pump pressure ratio (default 1000)

POWER= Power available (KW or hp) (default unlimited)

COMPEFF= Efficiency of pumping the liquid phase (default 100%)

PUMPEFF= Efficiency of compression of the gas phase (default 100%)

SPEED= Pump speed

STAGES=

STONEWALL=

ON

OFF

VERBOSE=

ON

OFF

TARGETGAS=

TARGETLIQUID=

TARGETMASS=

Note: At least one of the subcodes POUT, DP, PRATIO, and POWER should be supplied. If 2 or more are present, PIPESIM will treat them as upper limits, and will use whichever gives the smallest DeltaP. The others will be recalculated and displayed as answers on the output file.

5.6.16 NODE System Profile Data (Required)

Main-code: NODE

The physical geometry of the pipeline system is defined by entering the distance and elevation coordinates of each system node. A minimum of two nodes are required to define a system and there is a maximum limit of 1,000 nodes.

Note: For the calculation of temperature and pressure profiles, PIPESIM internally subdivides the section of pipe between each node into a number of segments. Normally 4 segments are created, but this can be controlled from 1 to 50 if desired (see [Options \(p.609\)](#)).

DISTANCE= Horizontal distance coordinate of the node (km or feet).

ELEVATION= Vertical elevation coordinate of the node (m or feet).

MD= Measured Depth (m or feet)

TVD= True Vertical Depth (m or feet)

TEMP= Ambient temperature at the node (°C or °F). If no value is entered, the value will be calculate: see below.

U= Overall heat transfer coefficient relative to the pipe outside diameter (W/m²/K or Btu/hr/ft²/ °F). If no value is entered the value from the previous node is assumed. The U sub-code is not required if the heat transfer coefficients are to be calculated by the program (if specified they will be ignored)

LABEL= Node labels are for information only and can appear on any node card. There is a maximum length limitation of 12 characters. The label should be included in quotes if it contains delimiter characters (for example blanks). If a node is labelled, it will appear on the Summary Output at the end of the job.

MP=

MT=

Notes:

1. Each node may have either specification of DISTANCE= and ELEVATION= , or TVD= and MD=, but not both. It is also possible to join together pipe sections with either specification, in which case the nodes where the sections join are assumed to occupy the same position.
 2. The ambient temperature is optional.
 3. If it is omitted on MD/TVD nodes, it is assumed to be a point on a geothermal temperature gradient, and its value is calculated by linear interpolation against TVD between known values on either side. On the DIST/ELEV nodes however, the value from the previous node will be used.
 4. It is possible to place separate sections of pipe within the PIPESIM input file exactly as they were measured, i.e. with their own particular X/Y datums, effectively a datum reset feature. Datums are reset whenever a change of NODE card specification occurs (change from using DISTANCE= and ELEVATION= to MD= and TVD=), and when a supercode is used (COMPLETION, TUBING, FLOWLINE, RISER NAPOINT, see [Completion \(p.651\)](#) and [NAPOINT \(p.743\)](#)).
 5. As with all other maincodes and subcodes, the node data keywords can be abbreviated down to the minimum number of letters required to make them unique.
 6. In addition, if distance and elevation is used, the subcodes can be omitted, as long as the data is supplied in the correct order, viz distance, elevation, temperature, U, label. Blank fields should be delimited by commas.
 7. Zero length pipe sections can be defined, that is NODE cards with DISTANCE= and ELEVATION= sub-codes the same as the previous one can be defined.
-

5.6.17 PIPE: Pipe or Tubing cross-section dimensions (Required)

Main-code: PIPE

ID=	Pipe Internal diameter (mm or inches).
WT=	Pipe wall thickness (Default = 12.7 mm or 0.5 inches)
ROUGHNESS=	Pipe roughness (Default = 0.025 mm or .001 inches).
AID=	Annulus inside Diameter (mm or inches). (Default = 0). See notes below.
AOD=	Annulus Outside Diameter (mm or inches). See notes below.
FLOWTYPE=	Specifies the flowpath for a tubing/annulus system. May be: TUBING Flow is in the normal tubing or pipe, whose internal diameter is specified with ID=. This is the default. ANNULUS Flow is in the annular space between tubing and casing, dimensions set with AID= and AOD=

BOTH or TUBANN	Flow is in both tubing and annulus: tubing internal diameter is set with ID=, casing dimensions set with AOD=, ID= and WT=; the AID= subcode will be ignored.
CONDUCTIVITY= or KPIPE=	Pipe thermal conductivity (W/m/K or Btu/hr/ft/ °F)
WAXTHIKNESS=	The thickness of a coating of wax that exists on the inside of the pipe (in or mm) (default zero).
WAXK=	The thermal Conductivity of the wax (W/m/K or Btu/hr/ft/ °F)
RODDIAM=	Specifies that a pump drive rod exists in the center of the pipe or tubing, and supplies its diameter (in or mm). If flowtype is set to TUBING, Fluid will flow in the annular space between the rod and the pipe internal diameter specified with the ID= subcode.
AXID= or AAREA= or ANNULUSAREA=	Optional. The cross-section area of the flowpath (ft ² or m ²). If specified, this value is used preference to the area that would otherwise be calculated from ID=, AID=, flowpath and so on to compute the fluid velocity. This is useful for modeling flow in ducts that do not have a circular or annular shape.
AWP=	Optional. The total Wetted Perimeter WP (in or mm). If specified, this value is used in preference to the sum of the relevant diameters specified with ID=, AID- and AOD=, depending on the flowpath. This is useful for modeling flow in ducts that do not have a circular or annular shape.
AEHD=	Optional. The Equivalent Hydraulic Diameter (in or mm). This is normally calculated from the relation EHD = 4 AXID / WP . It is used to obtain the friction factor. This is useful for modeling flow in ducts that do not have a circular or annular shape.
ILH= or ILHPOWER= or ILHMAXPOWER=	In-Line Heater. This subcode allows a fixed power or duty to be specified, that is used to transfer heat to the fluid flowing in the pipe. The value is interpreted as power per unit length (BTU/hr/ft or Kw/m). See notes below.
ILHMINTEMP=	This subcode allows a fixed minimum temperature to be maintained across the pipeline by assigning required variable heating power per unit length of pipe (BTU/hr/ft or Kw/m.) See notes below.
RODDIAM=	Drive rod diameter. This subcode allows the pipe cross-section area to be reduced, to simulate the presence of a drive rod for a pump. The diameter of the rod must be provided (inches or mm). Note, if the RODPUMP or CVFMD statement is used to place a pump in the system, the drive rod diameter can also be

set there, in which case it is not necessary to supply it here as well.

Note:

1. The AID= and AOD= subcodes should only be used if annular flow is desired. For normal pipe or tubing flow, ID=should be used. The FLOWTYPE subcode should always be used to confirm the desired type of flow.
2. Both AID= and AOD= refer to the dimensions of the annulus, that is the space between tubing and casing, or between successive casings. For example, if annular flow between tubing and casing is to be modeled, the AOD= is the casing inner diameter, and the AID= is the tubing outer diameter.
3. Most of the published multiphase flow correlations have been developed assuming normal pipe flow, not annular flow. Whilst Schlumberger have taken every care in the coding and validation of these correlations, you should carefully examine the results of annular flow simulation to ensure the selected correlation behaves as expected. We recommend that the results from a number of correlations be compared when annular flow is modeled.
4. When ILHMINTEMP= and ILH= (or ILHPOWER= or ILHMAXPOWER=) subcodes are used together, the supplied power is treated as the maximum limit. The specified minimum temperature is maintained along the pipeline as long as the required power does not exceed the available maximum power. When the required temperature cannot be maintained without exceeding the power limit, the available power will be used as fixed power and resultant temperature will be calculated.

5.6.18 PUMP Pump (Optional)

Main-code: PUMP

POUT=	Discharge pressure from the pump (bara or psia) (default 20,000 psia)
DP=	Pressure differential across the pump (bar or psi) (default 10,000 psia)
PRATIO=	Pump pressure ratio (default 1000)
POWER=	Power available for pump (KW or hp) (default unlimited)
EFF=	Overall efficiency of the pump (%) (Default = 100%)
NAME=	The name of the pump. Used to specify which pump curve defined before the profile under the PUMPCRV main-code is to be used.
SPEED=	The pump impeller speed (rpm) (default unlimited)
STAGES=	The number of stages for this particular pump. (default: number of stages as specified on the PUMPCRV maincode.)

ROUTE=	The thermodynamic route of operation of the pump
ADIABATIC	No heat transfer
ISENTHALPIC	Constant enthalpy
MOLLIER	Mollier
ISENTROPIC	Constant entropy
ISOTHERMAL	Constant temperature
ACF	
UNDEFINED	Undefined
VISCCORR=	Viscosity correction
NONE	
CENTRILIFT	
REDA	
TURZO	
USER	
VERBOSE=	
ON	
OFF	
STAGECALCS=	Perform the calculations through the pump on a stage-by-stage basis. The details are then reported in the output file.
ON	
OFF	
SEPEFF=	
EQUILIBRIUM=	
ON	
OFF	
CALCNSTAGES=	
MAXWCUT=	
MINSSU=	

VISCFLUID=

OIL

WATER

LIQUID

MIXTURE

VISCFATOR=

STATUS=

Notes:

1. At least one of the subcodes POUT, DP, PRATIO, POWER and SPEED should be supplied. If 2 or more are present, PIPESIM will treat them as upper limits, and will use whichever gives the smallest DeltaP. The others will be recalculated and displayed as answers on the output file.
 2. If a pump curve name is supplied, the speed and/or number of stages may also be supplied. These are used to adjust the supplied curve against its specified speed and number of stages (as set with SPEED= and STAGES= on the PUMPCRV maincode). The adjustment for speed is done using the so called affinity or fan laws, which state that "capacity is directly proportional to speed, head is proportional to square of speed, and power is proportional to cube of speed".
-

5.6.19 COMPCRV and PUMPCRV: Compressor and Pump performance curves

Main-code: PUMPCRV or COMPCRV

Centrifugal pump and compressor performance curves are specified as a range of head and efficiency or power values versus volumetric flow rates. These values should be specified before the profile, and each curve is given a name so it can be referenced on a subsequent PUMP or COMPRESSOR statement.

NAME= Required: The name of the curve, for referencing on a subsequent PUMP or COMPRESSOR statement.

SPEED= The speed for which this curve was generated. (rpm)

Q= Values of flow rate, supplied as a [multiple value set \(p.599\)](#). Flowrates are measured in volumetric terms at the flowing pressure and temperature at the inlet to the device. For a pump curve, units are bbl/day or m³/day; for a compressor curve, they are ft³/min or m³/sec. (See note below).

FLOWRATE= Synonym for Q =.

HEAD= Values of head, supplied as a [multiple value set \(p.599\)](#). For a pump curve, units are feet or metres. For a compressor curve they are ft-lbf/lbm (foot-pounds force

per pound mass) or Kj/kg. Note that the conversion factor between ft-lbf/lbm and feet is 1.

EFFICIENCY= Values of efficiency (%), supplied as a [multiple value set \(p.599\)](#). (Default is 100%). Exclusive with **POWER=**.

POWER= Values of power (hp or kw), supplied as a [multiple value set \(p.599\)](#). Exclusive with **EFFICIENCY=**.

STAGES= For a Pump curve only, the number of stages for which this curve is defined (normally 1).

WHEELS= For a compressor curve only, the number of compressor wheels for which this curve is defined.

The [multiple value sets \(p.599\)](#) supplied for Q=, HEAD=, and EFF= or POWER= sub-codes must contain at least 3, and no more than 30 values, separated by commas, and enclosed in parentheses. The values need not be entered in ascending or descending order, however there is a strict one-for-one correspondence between the values in each list, based on their position. Each list must contain the same number of values.

Since the multiple value lists can be quite lengthy, they may be supplied across more than one line in the input file. This can be achieved either by use of the continuation character &, or by repeating the maincode and curve name on each line. The examples below illustrate this, they both have the same effect:

Examples

Example 1

```
PumpCrv      name = GN7000      stages = 100      speed = 3600
PumpCrv      name = GN7000      Q      = (1250      ,3750      ,5800      ,7400      ,
9000      ,10666.7)
PumpCrv      name = GN7000      head = (4827.88  ,4176.29,3624.68,3160.3  ,
2338.76,971.734)
PumpCrv      name = GN7000      eff   = (23.6865  ,
55.0738,67.3463,71.3873,63.476 ,31.7383)
```

Example 2

```
PumpCrv      name = GN7000      stages = 100      speed = 3600      &
Q      = (1250      ,3750      ,5800      ,7400      ,9000      ,10666.7)  &
head = (4827.88  ,4176.29,3624.68,3160.3  ,2338.76,971.734)  &
eff   = (23.6865  ,55.0738,67.3463,71.3873,63.476 ,31.7383)
```

5.6.20 REINJECTOR (Optional)

Main-code: REINJECTOR

5.6.21 RODPUMP: Rod- or Beam-pump

Main-code: RODPUMP or CVFMD

The RODPUMP statement allows the outline specification for a Rod- or Beam-pump to be supplied.

A rod-pump is an example of a Constant Volume Fluid Motive Device (CVFMD). CVFMDs are fixed-volume, positive-displacement pumps or compressors designed to move liquids, gases or 2-phase mixtures. Other notable CVFMD examples are: Progressive cavity pumps, Twin screw multiphase boosters, and reciprocating compressors. The statement allows a simplistic simulation of such a device to be performed.

- NOMLIQRATE= The flowing volume flowrate that the pump would produce, if it were pumping with no back-pressure at its discharge (m^3/day or bbl/day).
- SLIPCOEF= A coefficient to specify the change in flowrate with respect to Delta pressure ($\text{m}^3/\text{day}/\text{bar}$ or $\text{bbl}/\text{day}/\text{psi}$). This is used to compute the pressure rise across the device when the actual flowrate is less than the specified nominal rate.
- MAXDP= Maximum pressure rise the device is allowed to exhibit (psi or bar). This is used to prevent excess rod loading.
- MAXPOWER= Maximum power the device is allowed to draw (hp or kw). This is used to prevent excess rod loading.
- RODDIAMETER= The Diameter of the drive rod (in or mm). The drive rod will be assumed to exist in the downstream pipe or tubing, and will stretch up to the wellhead or the end of the tubing. The fluid will flow in the annular space between the tubing ID and the rod OD. The rod diameter can be adjusted in downstream pipe sections by use of the RODDIAM= subcode on the [PIPE \(p.699\)](#) statement, this is useful to simulate taper rods.
- VOLUME= The swept volume of the pump cylinder, i.e. its cross-section area multiplied by the stroke length (bbl or m³). In conjunction with SPEED=, this is an alternative to the nominal rate.
- SPEED= The pump speed, in strokes per minute. In conjunction with VOLUME=, this is an alternative to the nominal rate.
- NOMINALRATE= The flowing volume flowrate that the pump would produce, if it were pumping with no back-pressure at its discharge (m^3/sec or ft^3/min). this is the same information as NOMLIQRATE= but in different units, more suited to other types of CVFMD.

5.6.22 SEPARATOR Separator (Optional)

Main-code: SEPARATOR

- TYPE= Type of separator: required, and may be set to one of:

	GAS	Defines a gas separator: the Gas phase will be wholly or partly discarded.
	LIQUID	Defines a liquid separator: the liquid phase(s) will be wholly or partly discarded.
	WATER	Defines a water separator: the Aqueous Liquid phase will wholly or partly discarded.
EFFICIENCY=		The volumetric efficiency of separation expressed as a percentage, 0 to 100. At 100% efficiency, all of the discarded phase will be removed; at lower values, some of the discard phase will remain in the kept stream. Note that the efficiency applies only to the discarded phase; no portion of the kept phase(s) will be discarded at any efficiency.
DISCARDNAME=		Optional: the name of the discarded fluid stream. The discarded stream can be re-injected in the branch, in a downstream fluid injector, if it is given a name. This is an alternative to *SEP_DISCARD, as described in note 3 below.
VERBOSE=		Optional: allows control over the detailed output for the separator, written to the report file (.out).
	ON	Separator output is written to report file.
	OFF	Separator output is not written to report file.

Notes:

1. In a Black Oil case, the separator will result in a redefinition of the fluid's stock-tank Gas to Liquid Ratio (originally supplied on the RATE maincode as GLR, GOR, OGR or LGR) to a GLR or LGR. In a compositional case, a rigorous flash is performed at the separator pressure and temperature, and the molar composition re-defined in terms of the flowrates of the components in the kept phase(s).
 2. If the efficiency is 100% and the flowrate is defined in terms of the discarded phase, the flowrate basis will be changed to that of the kept phase.
 3. Use of the SEPARATOR statement normally results in the separated phase being discarded from the system. In a single branch model, or within a given branch of a network model, the discarded phase can be recovered and re-injected into the branch further downstream, by use of the special fluid name [*SEP_DISCARD \(p.693\)](#) on the [INJFLUID \(p.693\)](#) statement. In a network model, a [Network Separator \(p.769\)](#) can be used (at the network level) to ensure both separator outlet streams are kept.
-

5.6.23 WELLHEAD Wellhead Profile Delimiter

Main-code: WELLHEAD

This maincode marks the position of the wellhead in the system profile. It is required when modeling multiple injection ports in gas lifted wells.

5.7 HEAT TRANSFER DATA

- [HEAT \(p.707\)](#) Heat Balance Options
- [COAT \(p.710\)](#) COAT Pipe Coat and Annular Space Medium Data
- [TCOAT \(p.711\)](#) Pipe Coat Thickness Data
- [KCOAT \(p.712\)](#) Pipe Coat Thermal Conductivity Data
- [KFLUID \(p.715\)](#) Fluid Thermal Conductivity Data
- [CONFIG \(p.715\)](#) Configuration Data
- [Pipeline burial depth examples \(p.716\)](#)

5.7.1 Notes on Heat Transfer Output Printing

1. Normally heat transfer input and output data is invoked by using: [PRINT \(p.624\)](#) HINPUT HOUTPUT (See Section 1)
2. The radial temperature profile of coatings surrounding the pipe can be printed by invoking: [PRINT \(p.624\)](#) EXTRA=TGRAD
3. Details of convective heat transfer calculations can be printed by invoking: [PRINT \(p.624\)](#) EXTRA=CONVx where 'x' can be either 1, 2, 3, or 4 representing coats 1 to 4 respectively.
Note that details can only be obtained for 1 coat in each PIPESIM case performed.

5.7.2 HEAT Heat Balance Options (Optional)

Main-code: HEAT

BALANCE=	OFF	No heat balance is performed and the fluid temperature is set equal to the ambient (or local ground) temperature as specified in the system profile under the NODE main-code.
	ON	A heat balance is performed (default).
U=	CALC	Calculate heat transfer coefficients from data supplied in this section.
	INPUT	Heat transfer coefficients are to be read from the NODE cards (default).
HTCRD=		Heat Transfer Coefficient Reference Diameter (in or mm). This value will be used as the reference diameter (instead of the current Pipe Outside Diameter) for all Heat Transfer Coefficients printed in the Heat Transfer Output Data (p.624) page.

IFCMODE=	INPUT	Controls whether a separate Inside Film Coefficient (IFC) is calculated when U values are supplied on NODE statements (U=INPUT, see above). May be set to INPUT or CALC. INPUT means the U supplied on the Nodes is assumed to include the IFC; CALC means it does not, so IFC will be calculated separately and added (using the correct reciprocal formula) to the supplied U.(If INPUT is used, the IFC is calculated, but not added; the calculated value is compared to the supplied U, and if grossly smaller, will trigger the production of a warning message.) Applies only if U=INPUT. Default=INPUT.
	CALC	IFC is calculated and added to the supplied U value.
SPIFCMETHOD=		Single phase fluid Inside Film Coefficient (HFI) correlation. To be used in conjunction with MPIFCMETHOD=. If the flow conditions are single phase then the PIPESIM engine uses this correlation. The default method is BJA.
	SEIDERTATE	Seider and Tate
	ORIGINAL	Original
	VOLAVERAGE	Kreith eqn 8-20 (Vol Ave.)
	KREITH10	Kreith eqn 10-6
	GH9	Groothuis & Hendal eqn 9
	GH10	Groothuis & Hendal eqn 10
	BJA	BJA (Kreith eqn 8-20)
	SHELL	Shell
	BP	Bp
MPIFCMETHOD=		Multiphase Inside Film Coefficient (HFI) correlation method that is used in conjunction with SPIFCMETHOD=. If the flow conditions are multiphase then the PIPESIM engine uses this correlation. The default method is BJA.
	KAMINSKY	Kaminsky
	ORIGINAL	Original
	VOLAVERAGE	Kreith eqn 8-20 (Vol Ave.)
	KREITH10	Kreith eqn 10-6
	GH9	Groothuis & Hendal eqn 9

	GH10	Groothuis & Hendal eqn 10
	BJA	BJA (Kreith eqn 8-20)
	SHELL	Shell
	BP	Bp
HOLDUP=		IFC depends on holdup
	ON	
	OFF	Default
TRMIN=		
TRMAX		
WAX=		Wax calculation mode.
PARTBURYMETH=		Partial burial calculation method
	2009	2009 Method
	2000	2000 Method
	1983	1983 Method
MASTER=		Enthalpy is master.
	TEMPERATURE	
	ENTHALPY	
UVALUE=		
RAMEYMETHOD=		The Ramey heat transfer calculation method for tubings.
	OFF	Heat transfer coefficients for tubing are to be read from the NODE cards (default).
	LARGETIME	Heat transfer coefficients are to be calculated for tubing using the Ramey model (p.81) (for times greater than 168 hr). Note that the time is specified via OPTIONS subcode RAMEYTIME.
GRNDCP=		Ground specific heat capacity (default = 837.4 J/kg/K or 0.2 Btu/lb/°F) used in the Ramey model.
GRNDDEN=		Ground density (default = 2242.6 kg/m ³) or 140lb/ft ³) used in the Ramey model.

DHEQUATION=	Energy equation to calculate temperature changes in pipe flow.
2009	Rigorous energy equation taking into account elevation, friction and heat loss.
1983	Simplified energy equation.

Heat transfer mode

There are three heat transfer modes to select from:

1. No heat balance is performed and the fluid temperature is set equal to the ambient (or local ground) temperature as specified in the system profile under the NODE main-code.
2. A heat balance is performed using overall heat transfer coefficients input under the NODE main-code. Note: This is the default option. So, if the HEAT main-code is omitted, the program will perform a heat balance and expect heat transfer coefficients to be entered under the NODE maincode. If the program fails to find a 'U' value on the first node card an input data error will be generated.
3. A heat balance is performed using heat transfer coefficients calculated by the program from the data specified in this section.

5.7.3 COAT Pipe Coat and Annular Space Medium Data (Optional)

Main-code: COAT

The COAT statement allows the data for pipe coating or annular space thickness, conductivity and medium to be specified for a single coat or annular space. For multiple coats, additional COAT statements may be provided, up to a maximum of 26. COAT is an alternative to the TCOAT and KCOAT statements, they both allow the same information to be entered.

NUMBER=	The coat or annular space number to be specified. Coat 1 is the innermost coat. Must be an integer between 1 and 26
THICKNESS=	Coating or annular space thickness (ins or mm, default zero). All coats are assumed to be of zero thickness until specified with a positive thickness
CONDUCTIVITY= , K=	Coat thermal conductivity (W/m/K or Btu/hr/ft/ °F, default infinite). Exclusive with MEDIUM=.
MEDIUM=	The name of the fluid medium contained in the annular space. This must match the core portion of a filename with the extension .APF in the PIPESIM installation's data directory. Default available filenames to specify typical annular fluids are provided, these are as documented for the KCOAT statement. Exclusive with CONDUCTIVITY=.
PRESSURE=	The average pressure in the annular space (psia or Bar, default 1000 psia).
LABEL=, NAME=	The name of the coat or space.

U=	The overall heat transfer coefficient for this coat (W/m ² /K or Btu/hr/ft ² / °F). This value will be used instead of any calculated value. The reference diameter for this will be the pipe outside diameter, that is the junction between the pipe and the first coat, this can be changed using the RD= subcode below.
RD=	The Reference Diameter to be used for the U= subcode provided above (in or mm).
RESETALL	Specifies that all previous coatings information be reset to zero. This allows new coatings information to be supplied with subsequent COAT (or TCOAT and KCOAT) statements without risk that earlier higher-numbered coats will be remembered.

Note: As with other data, any coat thickness data specified at a particular node will automatically be carried forward to subsequent nodes unless altered or reset to zero thickness. A coat can be removed (effectively) by specifying its thickness as zero. This will not affect the properties of higher-numbered coats. All coats can be removed by use of the RESETALL subcode.

Example

```
PIPE ID=5.25 thickness =.375 K = 56.4
COAT resetall
COAT num = 1 thickness = 3 medium=brine name='brine-filled annulus'
COAT num = 2 thickness = 0.5 K = 50 name='casing 1'
COAT num = 3 thickness = 2 medium=gas65 name='gas filled annulus'
COAT num = 4 thickness = 0.5 K = 50 name='casing 2'
COAT num = 5 thickness = 3 K = 3.5 name='cement'
```

Note: All keywords can be entered using the [EKT \(p.94\)](#).

5.7.4 TCOAT Pipe Coat Thickness Data (Optional)

Main-code: TCOAT

The TCOAT and KCOAT main-codes are used to specify up to ten concentric pipe coatings for use in the heat transfer calculations. This data is only required if U=CALCULATE is specified with the [HEAT \(p.707\)](#) main-code. **Note** that the pipe thickness is specified under the [PIPE \(p.699\)](#) main-code.

TPIPE= Thickness of pipe (default = 0.0 mm or in).

TWAX= Thickness of wax (default = 0.0 mm or in).

T1= Thickness of coat 1 (default = 0.0 mm or in).

T2= Thickness of coat 2 (default = 0.0 mm or in).

- T3= Thickness of coat 3 (default = 0.0 mm or in).
 - T4= Thickness of coat 4 (default = 0.0 mm or in).
 - T5= Thickness of coat 5 (default = 0.0 mm or in).
 - T6= Thickness of coat 6 (default = 0.0 mm or in).
 - T7= Thickness of coat 7 (default = 0.0 mm or in).
 - T8= Thickness of coat 8 (default = 0.0 mm or in).
 - T9= Thickness of coat 9 (default = 0.0 mm or in).
 - T10= Thickness of coat 10 (default = 0.0 mm or in).
-

Note: As with other data, any coat thickness data specified at a particular node will automatically be carried forward to subsequent nodes unless altered or reset to zero.

5.7.5 KCOAT Pipe Coat Thermal Conductivity Data (Optional)

Main-code: KCOAT

KPIPE= Pipe thermal conductivity (default = 60.6 W/m/ °K or 35 Btu/hr/ft/ °F).

- K1= Coat 1 thermal conductivity (W/m/K or Btu/hr/ft/ °F).
- K2= Coat 2 thermal conductivity (W/m/K or Btu/hr/ft/ °F).
- K3= Coat 3 thermal conductivity (W/m/K or Btu/hr/ft/ °F).
- K4= Coat 4 thermal conductivity (W/m/K or Btu/hr/ft/ °F).

KWAX=

Note: The default value for coat thermal conductivities is infinity (effectively), which means that the default thermal resistance is effectively zero.

The sub-codes described here are utilized in the modeling of convective heat transfer within fluid filled annuli.

- F1= The name of the data file in which annular fluid properties for Coat No. 1 are stored.
- F2= The name of the data file in which annular fluid properties for Coat No. 2 are stored.
- F3= The name of the data file in which annular fluid properties for Coat No. 3 are stored.
- F4= The name of the data file in which annular fluid properties for Coat No. 4 are stored.

P1= Average pressure in Coat No. 1 (default = 1,000 psia / 70 bara)

P2= Average pressure in Coat No. 2 (default = 1,000 psia / 70 bara)

P3= Average pressure in Coat No. 3 (default = 1,000 psia / 70 bara)

P4= Average pressure in Coat No. 4 (default = 1,000 psia / 70 bara)

Convective (Fx) and conductive (Kx) coatings can be specified in any sequence desired. The example below illustrates the modeling of a riser within a gas filled caisson which is insulated on the outside with 0.5" neoprene. Physical properties for the gas contained within the caisson will be read from the file 'GAS65.APF'.

Note: The Fx and Kx subcodes for any coating are mutually exclusive. Calculated convective heat transfer coefficients are printed to the HOUTPUT page as normal. In addition an equivalent thermal conductivity is calculated from the convective heat transfer coefficient and printed to the relevant position in the HINPUTpage.

Files

BJA have prepared data files for the most common annular fluids such as natural gas, brine, mud and so on. These files are located in the PIPESIM data directory. The files currently available are listed below together with a brief description of the file contents.

File Name Contents

wbm.apf Properties of a typical water based mud

obm.apf Properties of a typical oil based mud

gas65.apf Properties of a 0.65 S.G. gas

brine.apf Properties of a typical brine

Annular property data files are written in a simple to understand text file format to enable users to edit files or create their own if required.

Example

The example below illustrates the table format.

Note: A column containing line numbers is not part of the file.

Line 1 Specification of file format. Should not be modified

Line 2 Title line. May be modified as necessary

Line 3 Specifies the number of properties to be read

- Line 4 Format in which lines are to be read. Should not be modified
- Lines 5 - 10 Identifies the properties contained in the file and the order in which they are arranged. Note that this order is fixed and cannot be changed by the user. The two columns of numbers contain the conversion factors necessary to convert the property from the units specified in the file to standard PIPESIM SI units. Column 1 contains the additive conversion factor whilst column 2 contains the multiplying factor. This specification allows users to specify data in different units as required.
- Line 11 Should not be modified.
- Line 12 - 29 Contains the necessary physical property data in the sequence specified in Lines 5 - 10. Columns 1 & 2 contain the pressure and temperature to which the physical property data relates. Note that the maximum number of pressure and temperature points permitted within a file is 20.
- Line 30 Identifies the end of file.

Example File: BRINE.APF

```

ANPROP-1.0 LINEAR TABLE-1 ,A, 0, 0, NOGO
JOB : WATER TEST-1
6
(7f11.0)
Pressure PSIA 0. 6.894730
Temperature F 255.3722 .5555555
Liquid Density LB/FT3 0. 16.01800
Liquid Viscosity cP 0. 1.000000
Liquid conductivity BTU/hrftF 0. 1.730104
Liquid Heat Capacity BTU/LBF 0. 4.186800
NEWLINE, SCALE, CONT
14.7 50 62.4 0.880 .332 1.000
14.7 60 62.3 0.760 .340 0.999
14.7 70 62.3 0.658 .347 0.998
14.7 80 62.2 0.578 .353 0.998
14.7 90 62.1 0.514 .359 0.997
14.7 100 62.0 0.458 .364 0.998
14.7 150 61.2 0.292 .384 1.000
14.7 200 60.1 0.205 .394 1.000
14.7 400 53.8 0.001 .394 1.000
1000 50 62.4 0.880 .332 1.000
1000 60 62.3 0.760 .340 0.999
1000 70 62.3 0.658 .347 0.998
1000 80 62.2 0.578 .353 0.998
1000 90 62.1 0.514 .359 0.997
1000 100 62.0 0.458 .364 0.998
1000 150 61.2 0.292 .384 1.000
1000 200 60.1 0.205 .394 1.000
1000 400 53.8 0.001 .394 1.000
composition

```

Typical [Thermal Conductivities \(p.572\)](#) in W/m/K (Solids) [Thermal Conductivities \(p.572\)](#) in W/m/K (Liquids and Gases)

Note: All keywords can be entered using the [EKT \(p.94\)](#).

5.7.6 FLUID Fluid Thermal Conductivity Data (Optional)

Main-code: KFLUID

The fluid thermal conductivities are used in the calculation of the pipe internal film heat transfer coefficients. Default values are supplied and should be adequate for most cases.

OIL= Oil thermal conductivity (default = 0.138 W/m/K or 0.08 Btu/hr/ft/ °F).

GAS= Gas thermal conductivity (default = 0.035 W/m/K or 0.02 Btu/hr/ft/ °F).

WATER= Water thermal conductivity (default = 0.605 W/m/K or 0.35 Btu/hr/ft/ °F).

Note: All keywords can be entered using the [EKT \(p.94\)](#).

5.7.7 CONFIG: Pipe Heat Transfer Configuration Data (Optional)

Main-code: CONFIG

This data is only required if U=CALCULATE is specified under the HEAT main-code and if near-surface horizontal or near-horizontal pipe is being considered. For pipe or tubing situated away from the influences of air and seawater the CONFIG card is not required.

DEPTH= The burial depth (in mm or inches) as measured from the ground surface or mudline to the center-line of the pipe. A negative burial depth implies that the pipe center-line is above the surface and the pipe is therefore partially buried or fully exposed. Default is Fully Buried at depth of 800 feet.

KGROUND= Ground thermal conductivity (W/m/K or Btu/hr/ft/F).

VAIR= Ambient air velocity. Used to calculate the outside film heat transfer coefficient (Default = 0.033 m/s or 0.1 ft/s). Exclusive with VWATER=.

VWATER= Ambient water velocity. Used to calculate the outside film heat transfer coefficient. (Default = 0.033 m/s or 0.1 ft/s). Exclusive with VAIR=.

WDENS= Ambient water density. (lb/ft³ or kg/m³).

WVISC= Ambient water viscosity. (cP)

WCP= Ambient water specific heat capacity. (btu/lb/F or KJ/Kg/C)

WK= Ambient water thermal conductivity. (btu/ft²/ft/F or KJ/m²/m/C)

WBETA= Ambient water coefficient of thermal expansion.

ADENS= Ambient air density. (lb/ft³ or kg/m³)

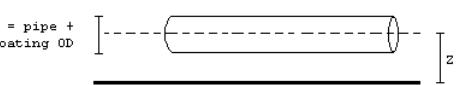
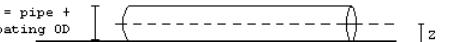
AVISC= Ambient air viscosity.(cP)
 ACP= Ambient air specific heat capacity. (btu/lb/F or KJ/Kg/C)
 AK= Ambient air thermal conductivity. (btu/ft²/ft/F or KJ/m²/m/C)
 ABETA= Ambient air coefficient of thermal expansion.
 TOPDEPTH= The burial depth (in mm or inches) as measured from the surface to the top of the pipe.

5.7.8 Pipeline burial depth examples

For a horizontal pipeline

Z: burial depth.

D: outside diameter of pipe and coatings

Burial depth	Pipe and coatings	Exposed to air or water	Schematic
$-D/2 > Z$	above ground / seabed	yes	$D = \text{pipe + coating OD}$ 
$Z = -D/2$	resting on ground/ seabed	yes	$D = \text{pipe + coating OD}$ 
$0 > Z > -D/2$	partially (less than half) buried	yes	$D = \text{pipe + coating OD}$ 
$D/2 > Z > 0$	partially (more than half) buried	yes	$D = \text{pipe + coating OD}$ 

$Z > D/2$	completely buried	no	
-----------	-------------------	----	--

5.8 Fluid Models

5.8.1 BLACK OIL DATA

[BLACKOIL \(p.717\)](#) Black Oil Correlation Options

[PROP \(p.719\)](#) Fluid Property Data

[LVIS \(p.721\)](#) Liquid Viscosity Data

[CPFLUID \(p.726\)](#) Fluid Heat Capacity Data

[RATE \(p.619\)](#) Flow Rate Data

[ITERN Iteration Data \(Optional\) \(p.621\)](#)

[TPRINT \(p.726\)](#) Black Oil Table Printing

[CALIBRATE \(p.727\)](#) Black Oil Property Calibration

[INJGAS and INJFLUID \(p.693\)](#) Injection Gas and side stream fluid injection

[WELLHEAD \(p.706\)](#) Wellhead Profile Delimiter

[GASLIFT \(p.687\)](#) Multiple Injection Ports in Gaslifted Wells

[INJPORT \(p.691\)](#) Gas Lift Injection Port

[CONTAMINANTS \(p.728\)](#) Gas contaminants data

BLACKOIL: Black Oil Fluid definitions

Main-code: BLACKOIL

Fluid properties can be generated internally by so called "black oil" correlations which have been developed by correlating gas/oil ratios for live crudes with various properties, such as oil and gas gravities. The selected correlation is used to predict the quantity of gas dissolved in the oil at a particular pressure and temperature.

The black oil correlations have been developed specifically for crude oil/gas/water systems and are therefore most useful in predicting the phase behavior of crude oil well streams. When used in conjunction with the calibration options, the black oil correlations can produce accurate phase behavior data from a minimum of input data. They are particularly convenient in gas lift studies where the effects of varying GLR and water cut are under investigation.

A Blackoil fluid must define its stock-tank volume phase split using one of: GLR=, GOR=, OGR=, or LGR=, and one of WCUT=, WGR= and GWR=.

Gas saturation (RS) correlations

RSCORR= ELSHARK	Elsharkawy
GHETTO	de Ghetto
GLASO	Glasø (p.510) correlation.
KART	Kartoatmodjo
LASATER	Lasater (p.511) correlation (default for bubble point pressure and solution gas).
PETROSK	Petrosky-Farshad
STANDING	Standing (p.511) correlation (default for oil formation volume factor at the bubble point)
VAZBEG	Vazquezand Beggs (p.511) correlation (default for oil formation volume factor above the bubble point).

Gas densities are calculated using a Z-factor correlation developed by [Katz \(p.588\)](#) and Standing and so the black oil correlations can also be used for single phase gas systems and gas/condensate systems with more or less constant gas/liquid ratios. However, if the accurate phase behavior prediction of light hydrocarbon systems is important, it is recommended that the more rigorous compositional models is employed.

Gas Compressibility correlation choices

GASZCORR= DPR	The Dranchuk, Purvis and Robinson correlation for curve fitting the Standing- Katz (p.588) reduced pressure-reduced temperature Z-Factor chart.
GOPAL	The Gopal correlation for curve fitting the Standing- Katz (p.588) reduced pressure-reduced temperature Z-Factor chart.
HALLYAR	Hall & Yarborough correlation for curve fitting the Standing- Katz (p.588) reduced pressure-reduced temperature Z-Factor chart.
STANDING	The Standing modification to the Brill and Beggs correlation for curve-fitting the Standing- Katz (p.588) reduced pressure-reduced temperature Z-Factor chart.
TSTANDING	

Oil Formation Volume Factor correlation choices

OFVFCORR= ELSHARKAWY Elsharkawy

KART Kartoatmodjo

PETROSKY	Petrosky
STANDING	Standing
VAZBEG	Kartoatmodjo

Gas dissolution and saturation in water

WRSCORR= HPPAC

KATZ

NONE

Gas viscosities are calculated using the [Lee et al. \(p.589\)](#) correlation.

Blackoil fluids that have been previously defined with [BEGIN FLUID \(p.634\)](#) can be selected with the FLUIDNAME= or USE= subcode.

Stock-tank Phase Ratios

A Blackoil fluid must define its stock-tank volume phase split using one of: GLR=, GOR=, OGR=, or LGR=, and one of WCUT=, WGR= and GWR=. (For historical reasons, these subcodes are also available on the RATE statement, but you are strongly encouraged to use them here instead. A compositional fluid can also define them but they must be supplied on the COMPOSITION statement.).

GWR= Gas Water ratio at stock-tank conditions. (sm^3/sm^3 or scf/STB)

WGR= Water/gas ratio at stock-tank conditions. ($\text{sm}^3/\text{mmsm}^3$ or STB/mmscf).

WCUT= Watercut, i.e. the volume % aqueous phase in the total liquid phase at stock tank conditions.

GLR= Gas/liquid ratio at stock tank conditions. (sm^3/sm^3 or scf/STB).

GOR= Gas/oil Ratio at stock tank conditions. (sm^3/sm^3 or scf/STB).

LGR= Liquid/gas ratio at stock-tank conditions. ($\text{sm}^3/\text{mmsm}^3$ or STB/mmscf).

OGR= Oil/gas ratio at stock-tank conditions. ($\text{sm}^3/\text{mmsm}^3$ or STB/mmscf).

Care must be exercised in combining these subcodes, as it is possible to specify a value for one of them that renders the use of the other one meaningless or illegal: such as with WCUT=100, any value for GOR= is meaningless; for example, GLR=0 conflicts with any non-zero value for GWR=. It is however always possible to re-state the desired definition correctly by well-chosen alternative subcodes.

PROP Fluid Property Data (Optional)

Main-code: PROP

API=	Dead oil API gravity at stock tank conditions (see note 1). Default = 30 API. The API gravity is defined as follows: - API = (141.5/sg) - 131.5 where sg is the oil specific gravity relative to water. Exclusive with DOD=.
DOD=	Dead oil density (kg/sm ³ or lb/ft ³) at stock tank conditions (see note 1). Default = 876 kg/sm ³ . Exclusive with API=.
GASSG=	Associated gas specific gravity relative to air (MW/28.964) at stock tank conditions (see note 1). Range 0.55 < GASSG 1.2. Default = 0.64.
CONEDGASSG=	Coned gas specific gravity (default: same as associated gas SG as defined with GASSG=). The coned gas SG will only be used if a coning relationship has been defined for the completion with CONETAB (p.663) or IPRCRV (p.660) . Coning will result in a mix of associated and coned gas, resulting in a produced gas SG somewhere between these 2 values.
WATERSG=	Water specific gravity at stock tank conditions (see note 1). Default = 1.02.
STENSION=	Specifies the method for calculating Liquid/gas interfacial tension. This allows for the possibility of three phase (gas/oil/water) flow where the liquid hydrocarbon (oil or condensate) flows as a segregated layer on top of the aqueous phase. Some 2-phase flow correlations (such as BJA and Duns and Ros) take account of the interfacial surface tension when calculating such parameters as the liquid wave height. Can be set to MIXED or SEGREGATED, meaning: MIXED: Surface tension is calculated based on the average properties of the water and hydrocarbon liquid mixture. This is the default value. SEGREGATED: Surface tension is calculated based on properties of the hydrocarbon liquid only. This option should only be used when it is expected that the liquid hydrocarbon and aqueous phases will be segregated, for example, long pipelines operating in the stratified flow regime.
GSAT=	The quantity of gas which would dissolve in the oil, and saturate it, at a given pressure and temperature (sm ³ /sm ³ or scf/bbl) (see note 2).
PSAT=	The saturation pressure for GSAT= (bara or psia) (see note 2).
TSAT=	The saturation temperature for GSAT= (°C or °F) (see note 2).
PSEP=	The separator pressure used by Kartoatmodjo (etc) correlations
TSEP=	The separator temperature used by Kartoatmodjo (etc) correlations

Note:

1. The oil, water and gas properties should be entered at stock tank conditions, that is 14.696 psia and 60 °F .

-
2. The oil saturated gas content at a known temperature and pressure (for example at reservoir conditions) should be entered to allow calibration of the black oil model. Such calibration will significantly improve the accuracy of the predicted gas/liquid ratios. If the calibration data is omitted the program will calibrate the correlation on the basis of oil and gas gravity alone and there will be a consequent loss in accuracy. Note, the value of GSAT= is independent of any GLR= or GOR= supplied on the BLACKOIL statement
-

LVIS: Liquid Viscosity Data (Optional)

Main-code: LVIS

DOVCORR= Specifies the choice of Dead Oil Viscosity correlation.

USER or BEAL Dead oil viscosity will be calculated by fitting two measured values of viscosity versus temperature to Beal's chart. Use with TEMP1=, TEMP2=, VIS1= and VIS2=. PIPESIM will fit a curve of the form

$$\log(\mu) \propto \frac{1}{T}$$

through the two given data points and then interpolate (or extrapolate) to find dead oil viscosities at given temperatures. Normally you would enter viscosity data at temperatures close to the expected maximum and minimum operating temperatures. Entering two identical temperatures or viscosities will cause an error.

TABLE Dead oil viscosity will be calculated by fitting a table of viscosity vs. temperature to Beal's chart. Use with TEMPS= and VISCS=. See note 1 below

BEGROB Beggs and Robinson (1975). This correlation is constrained to work within limits as published by the API (Petroleum Engineering handbook, Page 22-16), that is Temperature between 70 and 295 °F, and oil API gravity between 16 and 58.

GLASO Dead oil Viscosity will be calculated using the correlation of Glasø (1980)

KARTOATMODJO Kartoatmodjo and Schmidt correlation

GHETTO De Ghetto correlation

HOSSAIN Hossain correlation

ELSHARKAWY Elsharkawy correlation

PETROVSKY Petrovsky correlation

LOVCORR=		Specifies the choice of Live (gas-saturated) Oil Viscosity correlation. May be set to one of:
	CHEWCON	Chew and Connally correlation
	BEGROB	Beggs and Robinson correlation
	KARTOATMODJO	Kartoatmodjo and Schmidt correlation
	KHAN	Khan correlation
	GHETTO	De Ghetto correlation
	HOSSAIN	Hossain correlation
	ELSHARKAWY	Elsharkawy correlation
	PETROVSKY	Petrovsky correlation
UOVCORR=		Specifies the choice of Undersaturated Oil Viscosity correlation. May be set to one of:
	VAZBEG	Vasquez and Beggs correlation (default)
	KOUZEL	Kouzel correlation. Coefficients are supplied with the KA= and KB= subcodes.
	KARTOATMODJO	Kartoatmodjo and Schmidt correlation
	KHAN	Khan correlation
	GHETTO	De Ghetto correlation
	HOSSAIN	Hossain correlation
	ELSHARKAWY	Elsharkawy correlation
	BERGMAN	Bergman and Sutton correlation
	PETROVSKY	Petrovsky correlation
	NONE	Undersaturated oil viscosity calculation will be omitted, oil viscosity will be set to the result of the Live Oil viscosity correlation at the minimum of (actual and bubble point) pressure.
EMULSION=		Used to select one of various options for the calculation of oil-water mixture viscosities. The water viscosity data is generated internally by PIPESIM, using the van Wingel correlation. Generally speaking, at water cuts less than approximately 60% water (by volume), the oil phase is continuous with the water phase distributed. Under these

		conditions some oil-water mixtures can form highly viscous water-in-oil emulsions, particularly at water cuts in the range of 30-50%. Emulsion viscosities many times higher than that of either the oil or water are not uncommon. At water cuts above 60%, water is usually the continuous phase and the resulting oil-in-water emulsion has a viscosity similar to that of water. Used in conjunction with the CUTOFF= subcode.
SWAP		The mixture viscosity equals the oil viscosity at water cuts less than or equal to cutoff % and equals the water viscosity at water cuts greater than cutoff % (default).
VOLRATIO		The mixture viscosity equals the volume ratio of the oil and water viscosities.
WLOSE		Use Woelflin "Loose Emulsion" correlation at watercuts below the cutoff, set to water viscosity above.
WMEDIUM		Use Woelflin "Medium Emulsion" correlation at watercuts below the cutoff, set to water viscosity above.
WTIGHT		Use Woelflin "Tight Emulsion" correlation at watercuts below the cutoff, set to water viscosity above.
WORIG		Use PIPESIM original Woelflin loose emulsion correlation at watercuts below the cutoff, set to water viscosity above.
KENMONROE		Liquid viscosities are calculated from oil and water viscosities using the Kendal & Monroe equation. This is the option used when Emulsions are set to None for a Compositional fluid.
TABLE		Emulsion viscosities are interpolated from the table supplied with the EWCUTS= and EVISCS= subcodes. See note 1 below.
BRINKMAN		Use the Brinkman correlation. This generally predicts elevated liquid viscosities on either side of the cutoff.
VAND		Use the Vand correlation, using Vand's coefficients. This generally predicts elevated liquid viscosities on either side of the cutoff.
VANDBARNEA		Use the Vand correlation, using Barnea and Mizrahi coefficients. This generally predicts elevated liquid viscosities on either side of the cutoff.
VANDUSER		Use the Vand correlation, using coefficients supplied with the K1= and K2= subcodes. This predicts liquid

	viscosities on either side of the cutoff. Suitable choice of coefficients can yield elevated or depressed viscosities.
RICHARDSON	Use the Richardson correlation, using coefficients supplied with the RKOIW= and RKWIO= subcodes. This predicts liquid viscosities on either side of the cutoff. Suitable choice of coefficients can yield elevated or depressed viscosities.
LEVITON	Use the Leviton and Leighton correlation. This generally predicts elevated liquid viscosities on either side of the cutoff.
REDAOIW	
REDAWIO	
REDASWAP	
ABPCORR=	
ON	
OFF	
ORDER=	
UCORR=	
TEMP1=	Temperature 1 for DOVCORR=USER. (Default = 93.3 °C/200 °F)
VIS1=	Oil viscosity at temperature 1 for DOVCORR=USER. (Default = 0.5 centipoise)
TEMP2=	Temperature 2 for DOVCORR=USER. (Default = 15.6 °C / 60 °F).
VIS2=	Oil viscosity at temperature 2 for DOVCORR=USER. (Default = 10 centipoise).
VISCS=	List of oil viscosities for DOVCORR=TABLE, in Multiple Value format. See note 1.
TEMPS=	List of temperatures for DOVCORR=TABLE, in Multiple Value format. See note 1.
CUTOFF= or BOUNDARY=	The value of watercut where Phase Inversion occurs. In an oil-water mixture at low watercuts, water droplets are carried dispersed in a continuous oil phase. At much higher watercuts, water is the continuous phase and oil droplets are dispersed in it. The CUTOFF is the watercut

where the continuous phase changes. It is used by the Emulsion subcode, see above. (%, default 60)

PRESSURES=

FLTYPES=

LIVEOIL

DEADOIL

EWCUTS=

List of Watercuts for EMULSION=TABLE, in Multiple Value format. The first watercut value in the table must be zero. See note 1.

EVISCS=

List of oil viscosities for EMULSION=TABLE, in Multiple Value format. The first viscosity value in the table is used to divide into all the others to yield multipliers. See note 1.

K1= or VANDK1=

Coefficient k1 for use in the VANDUSER correlation, default 2.5

K2= or VANDK2=

Coefficient k2 for use in the VANDUSER correlation, default 0.609

RK= or

KRICHARDSON=

RKOIW= or
KROIW=

Coefficient k for the RICHARDSON correlation, used when oil-in-water conditions expected (watercut > cutoff).

RKWIO= or
KRWIO=

Coefficient k for the RICHARDSON correlation, used when water-in-oil conditions expected (watercut < cutoff).

KA=

Value of the A parameter for the Kouzel UOV correlation, default 0.0239

KB=

Value of the B parameter for the Kouzel UOV correlation, default 0.01638

TPVT=

Controls three-point viscosity tuning.

ON

OFF

Note: When using the user-supplied table options (DOVCORR=TABLE and EMULSION=TABLE), at least 3 and no more than 30 viscosity and (temperature or watercut) values must be supplied. The values need not be entered in any particular order, but there is a strict one-to-one correspondence between the values in the subcode pairs. Once read, the values will be sorted in

order of increasing temperature/watercut for use by the engine. Viscosity must never increase with temperature, but may vary with watercut as desired.

CPFLUID: Fluid Heat Capacity Data (Optional)

Main-code: CPFLUID

Fluid specific heat capacity data is required for the calculation of fluid enthalpies.

OIL= Oil heat capacity. Default = 1.89 kJ/kg/K or 0.45 Btu/lb/ °F.

GAS= Gas heat capacity. Default = 2.31 kJ/kg/K or 0.55 Btu/lb/ °F.

WATER= Water heat capacity. Default = 4.3 kJ/kg/K or 1.0 Btu/lb/ °F.

HVAP= Latent heat of vaporization. Default = 0.0 kJ/kg or 0.0 BTU/lb

CPFLUID is only used for black oil fluids: compositional fluids use heat capacities calculated by the selected physical properties package.

Note: All keywords can be entered using the [EKT \(p.94\)](#).

TPRINT Black Oil Table Printing (Optional)

Main-code: TPRINT

The TEMPERATURE and PRESSURE sub-codes have been added to the TPRINT main-code to permit the generation of tables of black oil properties.

Note that the property table is printed for information only and cannot be used as input for subsequent PIPESIM jobs.

TEMPERATURE= Temperature points at which properties should be tabulated (°C or °F). A maximum of 20 points may be specified

PRESSURE= Pressure points at which properties should be tabulated (bara or psia). A maximum of 20 points may be specified.

The format of the table is similar to that generated for a compositional table, however the properties tabulated in the table do differ from the compositional case.

Example

In the following example, a table of black oil properties at 5 temperatures and pressures is specified:

```
TPRINT temp = (400,300, 250, 200, 150)
TPRINT pressure =(3000,2500,2000,1500,1000)
```

Note: All keywords can be entered using the [EKT \(p.94\)](#).

CALIBRATE: Black Oil Property Calibration (Optional)

In many cases, actual measured values for some properties show a slight variance when compared with the value calculated by the black oil model. In this situation it is useful to calibrate (or match, or tune) the property using the measured point. PIPESIM uses the known data for the property to calculate a calibration constant K_c as noted below:

$$K_c = \frac{\text{Measuredproperty}_{(T,p)}}{\text{Calculatedproperty}_{(T,p)}} \quad \text{Eq. 5.1}$$

This calibration constant is then used to modify all subsequent calculations of the property in question, that is:

$$\text{CalibratedValue} = K_c \times \text{PIPESIMCalculatedValue} \quad \text{Eq. 5.2}$$

Properties which may be calibrated in this manner are

- Saturated Oil formation volume factor
- Saturated oil viscosity
- Undersaturated oil formation volume factor
- Undersaturated oil viscosity
- Gas viscosity
- Gas compressibility factor

Main-code: CALIBRATE

FVFRN= Measured value for Saturated oil formation volume factor.

TFVFRN= Temperature to which the figure given for FVFRN= refers ($^{\circ}\text{C}$ or $^{\circ}\text{F}$).

PFVFRN= Pressure to which the figure given for FVFRN= refers (bara or psia).

LOVIS= Measured value for saturated oil viscosity (cP).

TLOVIS= Temperature to which the figure given for LOVIS= refers ($^{\circ}\text{C}$ or $^{\circ}\text{F}$).

PLOVIS= Pressure to which the figure given for LOVIS= refers (bara or psia).

UFVFRN= Measured value for undersaturated oil formation volume factor.

TUFVFRN= Temperature to which the figure given for UFVFRN= refers ($^{\circ}\text{C}$ or $^{\circ}\text{F}$).

PUFVFRN= Pressure to which the figure given for UFVFRN= refers (bara or psia).

- UOVIS= Measured value for undersaturated oil viscosity (cP).
- TUOVIS= Temperature to which the figure given for UOVIS= refers ($^{\circ}\text{C}$ or $^{\circ}\text{F}$).
- PUOVIS= Pressure to which the figure given for UOVIS= refers (bara or psia).
- GASZ= Measured gas compressibility factor.
- TGASZ= Temperature to which the figure given for GASZ= refers ($^{\circ}\text{C}$ or $^{\circ}\text{F}$).
- PGASZ= Pressure to which the figure given for GASZ= refers (bara or psia).
- GVIS= Measured gas viscosity (cP).
- TGVIS= Temperature to which the figure given for GVIS= refers ($^{\circ}\text{C}$ or $^{\circ}\text{F}$).
- PGVIS= Pressure to which the figure given for GVIS= refers (bara or psia).

For each property being calibrated all three subcodes that is property, temperature and pressure must be specified.

Example

The example below supplies calibration values for the saturated oil formation volume factor and viscosity:

```
CALIBRATE FVFRN = 1.4 TTVFRN = 250 PFVFRN = 4000
CALIBRATE LOVIS = 0.85 TLLOVIS = 275 PLLOVIS = 2200
```

CONTAMINANTS Gas phase contaminants data (optional)

Main-code: CONTAMINANTS

Gas Compressibility Factor may be corrected for the presence of non hydrocarbon impurities. These must be measured as mole fractions (i.e. a value between 0 and 1) of the gas phase at stock-tank conditions.

CO2= Mole fraction of CO2 (default zero)

H2S= Mole fraction of H2S (default zero)

N2= Mole fraction of N2 (default zero)

H2= Mole fraction of H2 (default zero)

CO= Mole fraction of CO (default zero)

The CO2 and H2S values are used to modify the pseudo-critical pressure and temperature (used to calculate gas compressibility factor) as described in Beggs pages 30-31. The N2 value is used to adjust the gas compressibility factor as described in McCain page 120.

The Black Oil model treats contaminants as part of the gas phase, and assumes they dissolve in the oil as pressure increases in the same manner as the hydrocarbon gas components. Thus for any given fluid, the mole fractions will not vary with pressure, temperature or RS.

5.8.2 COMPOSITIONAL DATA

[COMP \(p.732\)](#) Fluid Data File Specification

[TPRINT \(p.738\)](#) Tabular Data Print Options

[RATE \(p.619\)](#) Flow Rate Data

AQUEOUS: Aqueous Component Specification

Also refer to [Compositional Modeling \(p.141\)](#), [COMPOSITION \(p.732\)](#), [LIBRARY \(p.736\)](#), [PETROFRAC \(p.737\)](#) and [MODEL. \(p.736\)](#)

A compositional fluid is usually defined with the PIPESIM GUI, and is written to a PVT file. The name of this file can be referenced with the [COMPOSITION \(p.732\)](#) keyword. Another way to define a composition is with the LIBRARY, PETROFRAC, AQUEOUS and MODEL statements.

The composition consists of a set of component names, their respective mole fractions, and a collection of modeling parameters for the phase behavior package. As such it specifies the composition of the total stream, regardless of any phase split the composition may exhibit at any pressure and temperature. The AQUEOUS keyword specifies the unit system in which the aqueous components will be specified.

Main-code: AQUEOUS

BASIS= Selects the desired units. Can be one of:

MOLAR: Aqueous components are specified in moles

GAS: Aqueous components are specified as WGR in m³/m³s³

LIQUID: Aqueous components are specified as watercut percent in vol/vol

CEMULSION Compositional Liquid Emulsion Data (Optional)

Main-code: CEMULSION

INVERSION= or

Specifies the “inversion point,” the value of watercut where Phase Inversion occurs. In an oil-water mixture at low watercuts, water droplets are carried dispersed in a continuous oil phase. At much higher watercuts, water is the continuous phase and oil droplets are dispersed in it. The inversion point is the watercut where the continuous phase changes. (%), default 60). The inversion point can also be calculated using the Brauner and Ullman correlation, this can be selected by supplying the special value “*CALC”.

CUTOFF= or

BOUNDARY=

TR= or

TRANSITION=

The width of a transition region, which is assumed to exist immediately above the inversion point. When the fluid

METHOD= or MODEL=		watercut falls within the transition region, the liquid viscosity is calculated by interpolation between that at the inversion point and the water viscosity.
SWAP	Selects the required emulsion model. Available models are:	
VOLRATIO	The mixture viscosity equals the oil viscosity at water cuts less than or equal to the inversion point, and equals the water viscosity at water cuts greater than the inversion point (default).	
WLOSE	The mixture viscosity equals the volume ratio of the oil and water viscosities.	
WMEDIUM	Use Woelflin "Loose Emulsion" correlation at watercuts below the inversion point, set to water viscosity above.	
WTIGHT	Use Woelflin "Medium Emulsion" correlation at watercuts below the inversion point, set to water viscosity above.	
WORIG	Use Woelflin "Tight Emulsion" correlation at watercuts below the inversion point, set to water viscosity above.	
KENMONROE	Use PIPESIM original Woelflin loose emulsion correlation at watercuts below the inversion point, set to water viscosity above.	
TABLE or USER	Liquid viscosities are calculated from oil and water viscosities using the Kendal & Monroe equation. This is the option used when Emulsions are set to None for a Compositional fluid.	Emulsion viscosities are interpolated from the table supplied with the WCUTS= and VISCS= subcodes. The table is applied at watercuts below and at the inversion point, set to water viscosity above.
BRINKMAN	Use the Brinkman correlation. This generally predicts elevated liquid viscosities on either side of the inversion point.	
VAND	Use the Vand correlation, using Vand's coefficients. This generally predicts elevated liquid viscosities on either side of the inversion point.	
VANDBARNEA	Use the Vand correlation, using Barnea and Mizrahi coefficients. This generally predicts elevated liquid viscosities on either side of the inversion point.	

VANDUSER	Use the Vand correlation, using coefficients supplied with the K1= and K2= subcodes. This predicts liquid viscosities on either side of the inversion point. Suitable choice of coefficients can yield elevated or depressed viscosities.
RICHARDSON	Use the Richardson correlation, using coefficients supplied with the RKOIW= and RKWIO= subcodes. This predicts liquid viscosities on either side of the inversion point. Suitable choice of coefficients can yield elevated or depressed viscosities.
LEVITON	Use the Leviton and Leighton correlation. This generally predicts elevated liquid viscosities on either side of the inversion point.
REDAOIW	Use Reda Oil-in-Water correlation for all watercuts. (Inversion point is ignored.)
REDAWIO	Use Reda Water-in-Oil correlation for all watercuts. (Inversion point is ignored.)
REDASWAP	Use Reda Oil-in-Water at watercuts below the inversion point and Reda Water-in-Oil correlation at or above the inversion point.
WCUTS= or WATERCUTS=	List of Watercuts for METHOD=TABLE, in Multiple Value format. The first watercut value in the table must be zero. Between 3 and 40 values must be supplied.
VISCS= or VISCOSITIES=	List of oil viscosities for METHOD=TABLE, in Multiple Value format. The first viscosity value in the table is used to divide into all the others to yield multipliers.
K1= or VANDK1=	Coefficient k1 for use in the VANDUSER correlation. The default value is 2.5
K2= or VANDK2=	Coefficient k2 for use in the VANDUSER correlation. The default value is 0.609
RK= or KRICHARDSON=	Coefficient k for the Richardson correlation. Use throughout the watercut range. (Inversion point is ignored.)
RKOIW= or KROIW=	Coefficient k for the RICHARDSON correlation, used when watercut is below the inversion point.
RKWIO= or KRWIO=	Coefficient k for the RICHARDSON correlation, used when watercut is at or above the inversion point.

COMPOSITION: Compositional Fluid Specification

Also refer to [Compositional Modeling \(p.141\)](#), [LIBRARY \(p.736\)](#), [PETROFRAC \(p.737\)](#), [AQUEOUS \(p.729\)](#) and [MODEL. \(p.736\)](#)

A compositional fluid is usually defined with the PIPESIM GUI, and is written to a PVT file. The name of this file can be referenced with COMPOSITION as documented here. (Another way to define a composition is with the [LIBRARY \(p.736\)](#), [PETROFRAC \(p.737\)](#), [AQUEOUS \(p.729\)](#) and [MODEL \(p.736\)](#) statements.)

PVT files can also be written by other programs, and can conform to a number of different formats. PVT files may contain a table of fluid physical properties and phase split at various pressures and temperatures; this table may be in addition to, or instead of, a fluid composition. (In the absence of a composition certain PIPESIM features will not be available; these include fluid mixing ([INJGAS](#), [INJFLUID \(p.693\)](#)), separation ([SEPARATOR \(p.705\)](#)) and the transformation subcodes described below.)

The composition will consist of a set of component names, their respective mole fractions, and a collection of modeling parameters for the phase behavior package. As such it specifies the composition of the total stream, regardless of any phase split the composition may exhibit at any pressure and temperature.

The supplied composition may be transformed to match a given stock tank phase split. The subcodes GLR=, WCUT= and so on allow this predefined composition to be transformed so as to match a specified phase ratios; this of necessity will change the components' mole fractions. The original composition must exhibit a phase split at stock-tank conditions that includes some of the phase(s) to be adjusted.

An example of few example of composition are below

```
library name='CARBON DIOXIDE' comp=0.020600
library name=NITROGEN comp=0.005060
library name=METHANE comp=0.760200
library name=ETHANE comp=0.078500
library name=PROPANE comp=0.039500
library name=ISOBUTANE comp=0.005960
library name=BUTANE comp=0.014400
library name=ISOPENTANE comp=0.004670
library name=PENTANE comp=0.006058
petro name=BPC6 comp=0.007548 tboil=129.29 molwt=0.860E2 sg=0.676
petro name=BPC7 comp=0.010000 tboil=167.09 molwt=0.930E2 sg=0.725
petro name=BPC8 comp=0.010600 tboil=228.29 molwt=0.105E3 sg=0.760
petro name=BPC9 comp=0.006852 tboil=271.49 molwt=0.117E3 sg=0.780
petro name=BP19 comp=0.010400 tboil=359.69 molwt=0.148E3 sg=0.805
petro name=BP27 comp=0.009832 tboil=519.89 molwt=0.212E3 sg=0.835
petro name=BP37 comp=0.001986 tboil=717.89 molwt=0.325E3 sg=0.864
```

```
petro name=BP47 comp=0.000794 tboil=876.29 molwt=0.477E3 sg=0.883
model eos=rks bip=oil1 vis=pedersen
aqueous basis=molar
library name=WATER comp=0.007014
```

Example of fluid using the default package Multiflash and the RKS equation of state

```
library name=N2 comp=0.005060
library name=C1 comp=0.760200
library name=C2 comp=0.078500
library name=C3 comp=0.039500
library name=IC4 comp=0.005960
library name=NC4 comp=0.014400
library name=IC5 comp=0.004670
library name=NC5 comp=0.006058
petro name=BPC6 comp=0.007548 tboil=129.29 molwt=0.860E2 sg=0.676
petro name=BPC7 comp=0.010000 tboil=167.09 molwt=0.930E2 sg=0.725
petro name=BPC8 comp=0.010600 tboil=228.29 molwt=0.105E3 sg=0.760
petro name=BPC9 comp=0.006852 tboil=271.49 molwt=0.117E3 sg=0.780
petro name=BP19 comp=0.010400 tboil=359.69 molwt=0.148E3 sg=0.805
petro name=BP27 comp=0.009832 tboil=519.89 molwt=0.212E3 sg=0.835
petro name=BP37 comp=0.001986 tboil=717.89 molwt=0.325E3 sg=0.864
petro name=BP47 comp=0.000794 tboil=876.29 molwt=0.477E3 sg=0.883
model eos=E300PR2
comp package = PVT_E300
aqueous basis=molar
library name=H2O comp=0.007014
```

Example of fluid using the Eclipse 300 flash package and the Peng Robinson 2 equation of state

Main-code: COMPOSITION

FILENAME= or PVTFILENAME= or FILE=	The name of the data file in which the fluid definition and/or property data tables are stored. The name should be enclosed in quotes if it contains delimiter characters or spaces.
USE=	Optional. The name of the fluid as specified with a BEGIN FLUID (p.634) block.
LVISFACTOR=	A multiplier for adjusting the tabular liquid viscosity data. (Default = 1.0).

CRICONDENBAR=	Upper pressure limit (bara or psia) or the two-phase region which is used to decide the method of interpolation between 100% liquid and 100% vapor data points. Above this value PIPESIM will assume dense phase and interpolate the tabular data appropriately. (Default = 0.0)
WCUT=	Watercut, that is the volume % aqueous phase in the total liquid phase at stock tank conditions. See note 1.
GLR=	Gas/liquid ratio at stock tank conditions (sm ³ /sm ³ or scf/stb). See note 1.
GOR=	Gas/oil Ratio at stock tank conditions (sm ³ /sm ³ or scf/stb). See note 1.
LGR=	Liquid/gas ratio (sm ³ /mmsm ³ or stb/mmscf) .See note 1.
OGR=	Oil/gas ratio at stock tank conditions (sm ³ /sm ³ or stb/mmscf). See note 1.
GWR=	Gas/water ratio at stock tank conditions (sm ³ /sm ³ or scf/stb). See note 1.
WGR=	Water/gas ratio (sm ³ /mmsm ³ or stb/mmscf) .See note 1.
PPMETHOD=	Flashing method for Physical Properties prediction: may be 1, 2, or 3. See note 5 below.
THMETHOD=	Flashing method for Temperature-Enthalpy-Entropy balance: may be 1, 2, or 3. See note 5 below.
PRINT	Prints a verbose printout of the fluid composition and stock-tank phase split.
ONECOMPONENT=	Controls "one component" behavior. Can be set to ON or OFF (default is OFF). If enabled, the fluid is assumed to consist entirely of one component molecule, and hence does not exhibit a classical phase envelope when graphed on axes of pressure versus temperature. Salient Examples of such systems are pure water or steam, pure Carbon Dioxide, pure methane, and so on. Special algorithms must be employed to ensure accurate results in such systems.
PACKAGE=	Selects the desired PVT code package name. It can be one of: MULTIFLASH: The third party company Infochem supplies the Multiflash package SHELL: Shell oil company's proprietary package PVT_E300: Eclipse 300 PVT package PVT_DBR: DBR PVT 2-Phase package PVT_GERG: GERG PVT package

PVT_NIST: NIST REFPROP PVT package

Note:

1. The presence of any of the subcodes GLR=, GOR=, LGR=, OGR=, WCUT=, WGR= or GWR= causes the supplied composition to be transformed match the specified phase ratios. The fluid is flashed at stock-tank pressure and temperature, and the resulting phases are re-combined to yield a new composition.
2. The subcodes GLR=, GOR=, OGR= and LGR= are optional, and mutually exclusive..
3. The subcodes WCUT=, WGR= and GWR= are optional and mutually exclusive
4. Care must be exercised in combining these subcodes, as it is possible to specify a value for one of them that renders the use of the other one meaningless or illegal: for example with WCUT=100, any value for GOR= is meaningless; for example GLR=0 conflicts with any non-zero value for GWR=. It is however always possible to re-state the desired definition correctly by well-chosen alternative subcodes.
5. The PPMETHOD= and THMETHOD subcodes control the manner in which Physical properties are computed. The balance is between speed and accuracy. Each of these subcodes can be set to the value 1, 2 or 3, which have the following meanings:
 - 1: Always Interpolate (fastest). This option uses linear interpolation between physical properties stored on a predefined grid of temperature and pressure points (default).
 - 3: Always Rigorous Flash (slowest). Interpolation never occurs: properties are obtained by flashing at the required pressure and temperature. This is the slowest, but most accurate, method.
 - 2: Rigorous Flash when close to the Phase Envelope, interpolation elsewhere. This is a compromise between speed and accuracy, which assumes that properties will change more rapidly when close to a phase boundary. Interpolation is performed whenever the grid points comprising a rectangle all show the presence of the same phases. For example if all 4 points in the rectangle have some oil, some gas, and no water, then we assume the rectangle lies entirely within the 2-phase region of the hydrocarbon phase envelope, so interpolation is appropriate. If however one, two or three of the points have no oil, then clearly the hydrocarbon dew point line crosses the rectangle, so a rigorous flash is required.

PPMETHOD= controls determination of transport Physical properties (PP) These are the values required to perform the multiphase fluid flow and heat transfer calculations, and include phase volume fractions, densities, viscosities, heat capacities and surface tensions.

THMETHOD= controls the Temperature-Energy Balance These values are used to maintain the temperature/enthalpy/entropy balance of the fluid.

In most simulations, for every PP flash that is performed, there are about 5 to 10 TH flashes, thus the TH flashes will have the greatest effect on speed and run time. The inaccuracies of TH interpolated flashes are usually minimal.

The speed impact of each choice will obviously depend on the composition, and the phase behavior in the PT region of interest. As a rough guide, taking the base case as interpolation, swapping just the PP flashes to "rigorous" will multiply your run time by about 4. With TH

flashes also "rigorous", run time will probably increase at least 20 fold. Use of the 'compromise' choices will be faster.

For those requiring more accuracy, we have found the "most useful" setting (that is the greatest increase in accuracy for the smallest effect on performance) to be PPMETHOD=2, THMETHOD=1..

LIBRARY: Library Component Specification

Also refer to [Compositional Modeling \(p.141\)](#), [COMPOSITION \(p.732\)](#), [PETROFRAC \(p.737\)](#), [AQUEOUS \(p.729\)](#) and [MODEL. \(p.736\)](#)

A compositional fluid is usually defined with the PIPESIM GUI, and is written to a PVT file. The name of this file can be referenced with the [COMPOSITION \(p.732\)](#) keyword. Another way to define a composition is with the LIBRARY, PETROFRAC, AQUEOUS and MODEL statements.

The composition consists of a set of component names, their respective mole fractions, and a collection of modeling parameters for the phase behavior package. As such it specifies the composition of the total stream, regardless of any phase split the composition may exhibit at any pressure and temperature. The LIBRARY keyword specifies the name and the composition of the library component..

Main-code: LIBRARY

NAME= The name of the component in the library

COMPOSITION= The composition (in moles for non aqueous elements and in the unit specified by the [AQUEOUS \(p.729\)](#) keyword for aqueous elements)

MODEL: Model Properties Specification

Also refer to [Compositional Modeling \(p.141\)](#), [COMPOSITION \(p.732\)](#), [LIBRARY \(p.736\)](#), [PETROFRAC \(p.737\)](#), [AQUEOUS \(p.729\)](#) and [MODEL. \(p.736\)](#)

A compositional fluid is usually defined with the PIPESIM GUI, and is written to a PVT file. The name of this file can be referenced with the [COMPOSITION \(p.732\)](#) keyword. Another way to define a composition is with the LIBRARY, PETROFRAC, AQUEOUS and MODEL statements.

The composition consists of a set of component names, their respective mole fractions, and a collection of modeling parameters for the phase behavior package. As such it specifies the composition of the total stream, regardless of any phase split the composition may exhibit at any pressure and temperature. The MODEL keyword specifies the fluid property model in terms of EOS, Viscosity model, BIP set, Physical properties method and Temperature-Enthalpy balance method. In addition, the flash package can be specified. This is a requested package and the engine will try to honor it if another package has not yet be loaded. To enforce the loading of a specific package, use the package subcode of the main code [COMPOSITION \(p.732\)](#).

Main-code: MODEL

EOS= optional – RKS or PR or CSM or RKSS or CPA or BWRS or CSMA or PVTIPRC or PVTIPRC3P or PVTIPRS or PVTIPR3P or E300PR2 or E300PR2C or

E300PR3C or E300SRK2 or E300SRK3 or DBR2PR2C or DBR2PR3C or DBR2SRK2 or DBR2SRK3 or GERG-2008 or NIST-DEFAULT.

VISCOSITY= optional – PEDERSEN, PEDERSEN-E, PED-E, LBC, LBC-E, LBC-D, LBCSE, LBCVR, LBCWOEL. PEDSE, PEDVR, PEDWOEL, SHELL_MODEL

BIP= optional – PVTIDEFAULT, FILE (BIP File), OIL1, OIL2, OIL3, OIL4, E300_DEFAULT, E300_USERFILE, DBR2_DEFAULT, DBR2_USERFILE, GERG_DEFAULT, NIST_DEFAULT. See [BIP \(p.147\)](#).

PACKAGE= optional –
MULTIFLASH: The third party company Infochem supplies the Multiflash package
SHELL: Shell oil company's proprietary package
PVT_E300: Eclipse 300 PVT package
PVT_DBR: DBR PVT 2-Phase package
PVT_GERG: GERG PVT package
PVT_NIST: NIST REFPROP PVT package

PPPACKAGE= optional –
MULTIFLASH: The third party company Infochem supplies the Multiflash package
SHELL: Shell oil company's proprietary package
PVT_E300: Eclipse 300 PVT package
PVT_DBR: DBR PVT 2-Phase package
PVT_GERG: GERG PVT package
PVT_NIST: NIST REFPROP PVT package

PPMETHOD= optional – 1, 2 or 3

THMETHOD= optional – 1, 2 or 3

PETROFRAC: Petroleum Fraction Specification

Also refer to [Compositional Modeling \(p.141\)](#), [COMPOSITION \(p.732\)](#), [LIBRARY \(p.736\)](#), [AQUEOUS \(p.729\)](#) and [MODEL. \(p.736\)](#)

A compositional fluid is usually defined with the PIPESIM GUI, and is written to a PVT file. The name of this file can be referenced with the [COMPOSITION \(p.732\)](#) keyword. Another way to define a composition is with the LIBRARY, PETROFRAC, AQUEOUS and MODEL statements.

The composition consists of a set of component names, their respective mole fractions, and a collection of modeling parameters for the phase behavior package. As such it specifies the composition of the total stream, regardless of any phase split the composition may exhibit at any pressure and temperature. The PETROFRAC keyword specifies the name, composition and properties of the petroleum fraction..

Main-code: PETROFRAC

NAME= The name of the petroleum fraction
COMPOSITION= The composition (in moles) — default 0
BPOINT= optional — Boiling Point
MW= optional — The molecular weight
SG= optional — The specific gravity
TCRIT= optional — The critical temperature
PCRIT= optional — The critical pressure
ACENTRIC= optional — The acentric factor
VISCOSITY= optional — The reference viscosity

Notes:

- Minimum data requirements for a petrofraction component are for the Multiflash package:
 - a. Either MW and SG
 - b. BPOINT and SG
 - c. PCRIT, TCRIT and ACENTRIC
 - There is no petroleum fraction supported in GERG and NIST. In E300 and DBR, the minimum data required is the molecular weight (MW)
-

TPRINT Tabular Data Print Options (Optional)

Main-code: TPRINT

FILE= The name of the fluid data file to be printed (12 characters maximum) which should be entered in quotes if the string contains delimiter characters. Up to five different files can be specified. Once a file has been specified it will be printed at the beginning of each case in the job until table printing is switched off using the NONE sub-code. To print the main fluid, use the wildcard * INLINE.

NONE= Turns the table printing option off. Table printing can produce large amounts of output, so it is common practice to print the data files in the first case of a job and then insert a TPRINT, NONE command in the second case to suppress table printing in the subsequent cases.

5.9 PIPESIM OPERATIONS OPTIONS

[MULTICASE Introduction and Summary \(p.743\)](#)

[Explicit Subcodes \(p.744\)](#)

[General PurposeSubcodes \(p.747\)](#)

[Combining MULTICASE and CASE/ENDCASE \(p.748\)](#)

[Multiple Case and PS PLOT \(p.749\)](#)

[Reservoir Simulator Tabular Data Interface \(p.750\)](#)

[Changing Profile Data by Assignment \(p.751\)](#)

[ITERN Iteration Data \(Optional\) \(p.621\)](#)

[Wax Deposition \(p.205\)](#)

5.9.1 NAPLOT: Nodal Analysis

Main-code: NAPLOT

This maincode, in conjunction with PIPESIM graphics processor PS-PLOT, allows the generation of a graph of inflow/outflow curves about the Nodal Analysis point specified with the NAPOINT maincode.

The '?' - delimiter symbols are like the general purpose ("greek") sub-codes on the MULTICASE statement. They can be used anywhere in the subsequent profile, and be equated to multiple values in the same way as the sub-codes on MULTICASE. For more information see [MULTICASE \(p.743\)](#).

All subcodes are optional.

- ?INFLOW= The inflow sensitivity values. Each value will produce one inflow curve. If omitted, a single inflow curve will be generated. See note 6.
- ?INFLOW2= These sub codes may be equated to a range of values, the number of values provided must equal the number provided in the ?INFLOW subcode. The values provided are selected in step with those on ?INFLOW. See note 6.
- ?INFLOW3=
- ?INFLOW4=
- ?INFLOW5=
- ?OUTFLOW= The outflow sensitivity values. Each value will produce one outflow curve. If omitted, a single inflow curve will be generated. See note 6.
- ?OUTFLOW2= These sub codes may be equated to a range of values, the number of values provided must equal the number provided in the ?OUTFLOW subcode. The values provided are selected in step with those on ?OUTFLOW. See note 6.
- ?OUTFLOW3=
- ?OUTFLOW4=
- ?OUTFLOW5=
- NINPTS= The number of points to be used to generate each inflow curve (default 20, maximum 200).
- NOUTPTS= The number of points to be used to generate each outflow curve (default 20, maximum 200).
- POUT= The system outlet pressure. This is used to generate the outflow curves. If this subcode is omitted the system outlet pressure will be obtained from the POUT= subcode of the ITERN statement. See note 3. (Psia or Bara.)

LIMITIN=	This subcode controls the application of any flowrate limit to the inflow curves. (Flowrate limits are supplied on the MAXLIQ=, MAXGAS= or MAXMASS= subcodes, or are assumed implicitly from the maximum value on GAS=, LIQ= or MASS= subcodes.) Can be set to YES or NO, the default being NO. If YES, the limit is applied, so the inflow curves will extend to that flowrate limit or to each curve's natural AOFP (Absolute Open Flow Potential, i.e. the rate at which the operating point pressure falls to zero), whichever is smaller. If NO, the limit is not applied so all inflow curves will extend to their natural AOFP..
LIMITOUT=	This subcode controls the application of a calculated pressure limit to the outflow curves. (The pressure limit will be calculated from the maximum pressure occurring on any of the inflow curves. Note: an explicit pressure limit can also be provided with the MAXP= subcode, which will take priority.) Can be set to YES or NO, the default being NO. If the limit is applied, then the outflow curves will extend to the maximum rate limit supplied or calculated, or to 20% above the maximum pressure calculated for any of the inflow curves.
MAXP=	The maximum pressure to be used when generating the outflow curves. (Psia or Bara.) Default is double the maximum pressure in any of the inflow curves.
MINP=	The minimum pressure to be used when generating the inflow curves. (Psia or bara). Default is none, so the inflow curves will extend to their AOFP or the specified flowrate limit.
MAXLIQ=	The maximum liquid flow rate to be used when generating the outflow curves. See notes 2 and 4. (m^3/d or STB/D).
MAXGAS=	The maximum gas flow rate to be used when generating the outflow curves. See notes 2 and 4. (MMm^3/d or MMscf/d).
MAXMASS=	The maximum mass flow rate to be used when generating the outflow curves. See notes 2 and 4. (kg/s or lbs/s).
PRINT=	Sets the number of cases for which detailed output will be generated in the output file: default is 1. This number is applied separately to the inflow, outflow and operating points, so you actually get 3 times as many cases printed as the value you supply. Eg. at its default of 1, you will get detailed output for the first inflow point, the first outflow point, and the first operating point; set it to 5 and you get the first 5 cases of inflow, the first 5 of outflow, and the first 5 operating points.
OPPOINTS=	Controls the explicit generation and display of Operating Points. Can be set to YES or NO, default is YES. The intersection of one inflow curve and one outflow curve is known as an Operating Point. Whilst it is possible to infer the system flowrate geometrically from the line intersections alone, it is more accurate and far safer to calculate the flowrate by simulating the system end-to-end, which PIPESIM is well designed to do. The resulting pressure and flow rate is displayed on the Nodal Analysis graph as an Operating Point. This explicit calculation ensures the inflow and outflow

fluid properties and temperature are identical, thus eliminating the possibility of a mismatch and consequent error in answer interpretation.

Operating points are generated for each permutation from the lists of inflow and outflow sensitivity variables, as supplied in the ?INFLOW= and ?OUTFLOW= subcodes. However, it is possible to set up the sensitivities so that some combinations are invalid, and these do not result in operating points being generated and displayed. For example, if you set both inflow and outflow sensitivity to the fluid watercut, most of the permutations will be invalid, because the fluid at the intersection cannot have 2 different values for watercut. With Operating point generation enabled, the valid intersections are clearly distinguishable from the invalid ones: operating points will only be generated for "valid" combinations.

Sometimes it will happen that the displayed operating point does not coincide with the geometric intersection. The cause of this will always be that the outflow fluid properties or temperature do not match that of the operating point. The fact that the mismatch is evident should be regarded as a feature, not a bug, and should alert the user to a problem or condition that requires particular caution and attention.

With operating point generation enabled, the profile plot file will contain valid profile plots for each operating point: these can be viewed by selecting "Reports > Profile Plot" in the PIPESIM GUI.

MATCH=

This subcode selects the method by which the fluid temperature and composition is matched between the inflow and outflow curves. Can be set to:

MAXFL: The inflow curve with the maximum AOFP rate is used. All outflow curves will use the values interpolated from this single inflow curve. (This was the behaviour in release 2009.1 and earlier, before the operating points were available).

OP: The generated Operating Points are used, along with the appropriate inflow curve on the low flowrate side. At flowrates higher than the operating points, the temperature and composition from the highest appropriate operating point is used. This is the default. If operating point generation is suppressed however (see OPPOINT= below), the MAXFL method will be used.

OP2: The generated Operating Points are used, along with the appropriate inflow curve on both sides. This option can cause the outflow curves to exhibit marked changes in slope at high flowrates, caused by the use of unrealistically low temperatures interpolated from the high rate inflow curve close to its AOFP.

OFF: matching is turned off. The outflow curves will use the system-defined fluid properties and inlet temperature.

Matching is important to ensure the fluid temperature/enthalpy and composition are consistent across inflow and outflow curves. Without it, the intersections or operating points between the curves may bear little or no resemblance to physical reality. The matching is achieved by using the temperatures and composition(s) from the correct inflow curve or operating point(s) to generate the ones used for the outflow curves. For example, if the inflow curves are generated

from multiple completions, each of which has a different reservoir fluid, the resulting mixed fluid composition at the NA point will change at each value of flowrate. The matching algorithm ensures the temperature and composition are interpolated from the correct inflow curves and operating points, so as to produce outflow curves that use an appropriate fluid composition and temperature. Thus each point on the outflow curve will usually have a unique temperature and composition. Matching is applicable to both black oil and compositional fluids.

MATCHENTH=	Allows the use of Enthalpy, instead of temperature, in the matching (see MATCH= above). Can be set to YES or NO. The default is NO.
LIQ=	A set of stock-tank liquid flow rates to be used when generating the outflow curves. A maximum of 200 flow rates may be specified. If omitted, the program will generate the set of flowrates at run-time. See notes 1, 2, 5 and 6. (m^3/d or STB/D).
GAS=	A set of stock-tank gas flow rates to be used when generating the outflow curves. A maximum of 200 flow rates may be specified. If omitted, the program will generate the set of flowrates at run-time. See notes 1, 2, 5 and 6. (MMm $^3/d$ or MMscf/d).
MASS=	A set of mass flow rates to be used when generating the outflow curves. A maximum of 200 flow rates may be specified. If omitted, the program will generate the set of flowrates at run-time. See notes 1, 2, 5 and 6. (kg/s or lbs/s).

Notes:

1. If a set of flow rates are supplied with GAS= LIQ= or MASS= subcodes, they will be used to generate the outflow curves. A maximum of 200 flow rates may be supplied, and the Range Format can be used. If omitted, then the program will choose the rates for the outflow curves using an algorithm designed to distribute the points on the curve to best effect. This will result in rates being clustered close together in areas where the pressure is changing fastest, i.e. in regions of maximum slope. Rates will also be generated at the operating points, to make the validity (or otherwise) of the curve intersections evident. (The rates used for the inflow curves are always generated with this algorithm.)
2. The subcodes LIQ=, GAS=, MASS=, MAXLIQ=, MAXGAS=, and MAXMASS= are mutually exclusive.
3. The pressure iteration is the only valid iterative option when doing nodal analysis, and is only applicable to the outflow curves. The outlet pressure and the flow rate are known which requires the calculation of an inlet flowing pressure. POUT can be specified either on the NAPLOT statement, or the ITERN statement.
4. MAXLIQ=, MAXGAS= and MAXMASS= will also apply to the inflow curves if LIMITINFLOW=YES.
5. The special value "none" can be used on the LIQ=, GAS= and MASS= subcodes. If used, its effect is to remove or cancel an existing list of flowrates supplied on a previous statement. An example of why this might be useful is to override a list of rates supplied by PIPESIM.

6. The multiple values should be supplied enclosed in parenthesis, and separated by commas. A multi-value range can also be specified. For more information, see [Multiple value Data Sets.](#) ([p.603](#))
-

5.9.2 NAPOINT System Analysis Point

Main-code: NAPOINT

Use this main-code to specify the required system (nodal) analysis point. NAPOINT divides the profile into two halves, and effectively runs separate jobs on each half. NAPOINT can be placed anywhere in the profile.

LABEL=

RESETDATUM=

ON

OFF

5.9.3 MULTICASE Introduction and Summary

The MULTICASE card is available to allow the user to set up many PIPESIM cases without having to enter many CASE and ENDCASE cards. By use of the MULTICASE card, it is possible to specify multiple values for various flow parameters on one card, rather than repeating cases.

The central idea behind MULTICASE is that its sub-codes can accept more than one value. So if, for example, you want to run 8 cases at various different flow rates, then instead of having to append an extra 7 explicit cases to your input file, a single MULTICASE card can be used to specify all 8 flow rates, and PIPESIM will execute the 8 separate cases automatically.

If a second multiple-valued subcode is provided, PIPESIM will execute as many separate cases as are required to combine all the values in each multiple subcode. So, for example, if we had:

Example

```
MULTICASE LIQ=(10,20,30,40,50,60,70,80) WCUT=(30,60,90)
```

The result would be 24 cases, representing the combination of all specified flow rates with all specified water cuts.

There are two distinct classes of sub-code available:

1. Explicit sub codes, such as LIQ=, WCUT, and IPRES are simply duplicates (or duplicate the function of) sub codes that appear on other main codes, such as RATE and INLET. The important difference is that they only accept multiple values on the MULTICASE card.
2. General purpose sub codes, such as ?ALPHA and ?BETA, which accept multiple values on the MULTICASE card and are then used in place of a sub code value further down the input data.

The provision of MULTICASE has allowed other sophisticated PIPESIM features to be built alongside it, for example the [Reservoir Table Interface \(p.750\)](#) and Well Performance Curve Generation.

General Rules for use with MULTICASE

The following notes apply to all subcodes on MULTICASE. Further restrictions exist for particular sub-codes and combinations of sub-codes, and these are documented where they arise.

1. The MULTICASE main-code must appear before the first NODE card in the job.
2. Each multiple-valued sub-code can be equated to a maximum of 20 values, separated from one another by commas, and the whole group enclosed in parentheses.
3. In any job, the maximum number of subcodes containing multiple values is 5.
4. The MULTICASE card cannot be continued: however, 2 or more MULTICASE cards can appear sequentially in the input file, thus allowing many sub-codes to be specified.
5. Each sub-code containing multiple values must appear on a single MULTICASE card. A maximum of 80 characters is allowed for all values enclosed in parentheses. The maximum input line length is 140 characters.
6. The MULTICASE card(s) should appear immediately before the first NODE card in the job, except when greek symbols are used, when the card(s) using the greeks should appear between MULTICASE and NODE.
7. MULTICASE was designed to be used instead of explicit extra cases (the CASE and ENDCASE cards), however both can be used in combination as long as no MULTICASE cards appear in subsequent explicit cases.
8. When subsequent explicit cases are used with MULTICASE, each subsequent explicit case will result in another complete set of multiple cases (see Section 8.4 for an example of this). The LIMIT subcode applies only to the set of multiple cases defined by the MULTICASE card(s), not to the total number of cases in the job.
9. MULTICASE jobs contain an implied 'loop' structure in the input data. Every line of input between the MULTICASE card(s) and the beginning of the system profile is scanned at the start at the beginning of every case, to ensure that any Greek symbols are assigned the correct values. Only the symbolic information is processed, and any other input is ignored except on the first case.

Multiple Case Specification Card

Main-code: MULTICASE

The subcodes available on the MULTICASE card can be divided into two distinct categories as outlined in [Explicit subcodes \(p.744\)](#) and [general purpose subcodes \(p.747\)](#).

5.9.4 Explicit Subcodes

LIQ= (,)	Gross liquid flow rate values at stock tank conditions. A maximum of 20 flow rates may be specified (sm ³ /d or STB/D).
------------	--

GLR= (,)	Gas/liquid ratio values at stock tank conditions. A maximum of 20 values may be specified (sm ³ /sm ³ scf/STB/D). Default = 0.
GAS= (,)	Gas flow rate values at stock tank conditions. A maximum of 20 flow rates may be specified. (mmsm ³ /d or mmscf/d).
LGR= (,)	Liquid/gas ratio values at stock tank conditions. A maximum of 20 values may be specified. (sm ³ /d or STB/scf). Default= 0.
GOR= (,)	Gas/oil ratio values at stock tank conditions. A maximum of 20 values may be specified. (sm ³ /sm ³ or scf/STB). Default = 0.
OGR= (,)	Oil/gas ratio values at stock tank conditions. A maximum of 20 values may be specified. (sm ³ /d or STB/scf). Default = 0. Note that the flow rate may be expressed either on the basis of the stock tank liquid or gas flow rate. The LIQ+GLR and GAS+LGR options are therefore mutually exclusive. An error will be reported if an invalid combination is entered and program execution will be terminated.
MASS= (,)	Mass flow rates for use with compositional cases. A maximum of 20 flow rates may be specified. (Kg/s or lbs/s).
WCUT= (,)	Water cut values that is the volume % water in the liquid phase at stock tank conditions. A maximum of 20 values may be specified. Default = 0.
WTYPE= (,)	Alphanumeric sub-code. This describes the type of well under consideration that is Injector (INJ) or Producer (PRD). This sub-code is only required if the WTHP sub-code is used.
WTHP= (,)	Tubing head pressure values. The definition of tubing head pressure is dependent on the physical configuration of the well. In the case of a producer it is the system outlet (last node) pressure, that is tubing head pressure for a well only, or the separator pressure in the case of a well and flowline. In the case of an injector it is the system inlet (first node) pressure, (bara or psia). A maximum of 20 values may be specified. The WTYPE sub-code must accompany this sub-code.
IPRES= (,)	Inlet set pressure (bara or psia). This provides a means of specifying the inlet pressure of a system. The maximum number of values which may be specified is 20.
OPRES= (,)	Outlet set pressure (bara or psia). Similar usage to the IPRES sub-code above. The maximum number of values which may be specified is 20.
XEST=	This sub-code does not take multiple values under the MULTICASE option, but may take a value for iterative cases as defined previously under the ITERN card (see Section 2.4). For cases where the inlet or bottom hole pressures are to be calculated an estimate of the parameter may be made using the following formula.

Default = well (pipe) vertical length x pressure gradient + WTHP (OPRES)

Pressure 0.0679 bar/m or 0.3 psi/ft for liquid) = 0.0113 bar/m or 0.05 psi/ft (for gas).
gradient
=

ITYPE=	Iteration Type see ITERN card description.
PRINT=	If the MULTICASE option is specified then all output except titles will be suppressed after the first case. In order to override this 'auto-noprint' procedure the sub-code PRINT must be included. Care should be exercised here if a large number of cases are set up as very large quantities of output can be generated.
LIMIT=	This sub-code sets a limit on the number of cases which will be run, and will abort the job at the start of execution if the number of cases to be run is greater than this. The default value is ONE, and therefore a limit must be set by you as a guard against an excessive number of cases being run.
LINE=	When using 3 or more Multicase options, this sub-code allows you to specify which loop controls the line structure. In the absence of a LINE= sub-code, the innermost loop controls this. Every time the innermost loop resets to its first value, a new line is started in the Job Plot file. You can specify the depth of the required controlling loop with the LINE= sub-code.

Notes:

1. Some sub-codes on the MULTICASE card are duplicates, or equivalents of sub-codes on the ITERN, RATE, and INLET cards (for example the IPRES sub-code is equivalent to the PRES sub-code on the INLET card and the XEST sub-code serves the same purpose as the one appearing on the ITERN card). If such duplicate or equivalent sub-codes are used on both the MULTICASE card and elsewhere in the same case, then the values supplied on the MULTICASE card will override the values supplied elsewhere. For example, Here the LASTANSWER sub-code (see Section 9.1) has been specified along with the MULTICASE card. Outlet and Inlet Pressures have been specified under both the ITERN and INLET cards in addition to being specified under the MULTICASE card. In such an example PIPESIM will ignore the POUT, TYPE, PRESS and XEST sub-codes specified under the ITERN and INLET sub-codes and will use IPRESS, OPRESS and XEST values specified under the MULTICASE card. The LASTANSWER option will be in operation even though the rest of the sub-codes specified under the ITERN card will be ignored.
2. The sub-codes can be entered in any order.
3. The use of certain sub-codes excludes the use of other sub-codes
 - a. Flow rate must be defined (with LIQ, GAS or MASS sub-codes) when the WTHP sub-code is used.
 - b. The well type must be defined with the WTYPE sub-code when the WTHP sub-code is used.
 - c. ITYPE, IPRES and OPRES sub-codes exclude the use of the WTHP sub-code .

- d. Iteration type must be defined with the ITYPE sub-code when the OPRES sub-code is used and outlet pressure must be defined under OPRES when the ITYPE sub-code is used.
- e. If LIQ, GAS or MASS sub-codes appear on the MULTICASE card then either IPRES or OPRES may be used but not both.

5.9.5 General Purpose Subcodes

Sub-codes : ?ALPHA, ?BETA, ?GAMMA, ?DELTA, ?EPSILON

The subcodes described here greatly increase the power and flexibility of MULTICASE.

The purpose of the MULTICASE maincode is to allow the user to execute a PIPESIM case for every combination of a set of input variables. For example, suppose we specify 3 water cuts, 4 flow rates, 5 GLR's and 6 outlet pressures: all possible combinations of these values will result in PIPESIM executing 360 individual cases, since $3 \times 4 \times 5 \times 6 = 360$. The process of selecting every possible combination of a series of variables is called permutation, and so we often use the verb permute when describing what the MULTICASE maincode can do.

Five new general-purpose symbolic subcodes have been added, namely: ?ALPHA, BETA, ?GAMMA, ?DELTA, and ?EPSILON. They are collectively known as Greeks. They can be equated to multiple values in the same way as other subcodes on MULTICASE. The symbols can then be used further down the input data in place of any other value. Thus, the greeks behave similarly to symbols created by the [ASSIGN \(p.751\)](#) maincode.

Examples

In the following example, 3 values of inlet temperature are permuted with 2 values of Gas-Lift GLR in a well:

Example 1

Example

```
RATE LIQ=45 GLR=180
MULTICASE ?ALPHA =(210,250,290)
MULTICASE ?BETA=(200, 220)
INLET PRESS=4200 TEMP=?BETA
NODE DIST=0 ELEV=-4000 TEMP=?BETA
NODE DIST=0 ELEV=-3000 LABEL='GAS LIFT'
RATE GLR=?ALPHA
NODE DIST=0 ELEV=-2000
```

The symbol ?BETA is set to 2 values on the MULTICASE card. ?BETA is then used on the INLET card in place of the value of Inlet Temperature. Note that, while it is possible to control Inlet Pressure with the existing IPRES subcode on the MULTICASE card, Inlet Temperature is not available as an explicit subcode on MULTICASE. However, because of the 'general-purpose' nature of the Greek symbols, it is now possible to control it (and, in principle, almost anything else) from the MULTICASE card.

The symbol ?ALPHA is set to 3 values on the MULTICASE card. ?ALPHA is then used on the RATE card in the profile, in place of the value for GLR. Thus, like the ASSIGN card, the greek

symbols provide a convenient way to change values within the system profile. Unlike the ASSIGN card however, the values equated on the MULTICASE card will be permuted to result in a number of cases being executed.

Example 2

The values equated to the greeks can be any appropriate numeric value or character string, depending on the use to which it is put further down the input data. For example, it is possible to permute a range of flow correlations:

```
MULTICASE ?DELTA=(BBO,BJA1,BJA2) ?GAMMA=(BB,TD)
HCORR PLOSS=BBO HOLDUP=?DELTA MAP=?GAMMA
```

Notes:

1. The new greek subcodes can be used in conjunction with the existing MULTICASE subcodes, but the maximum number of multiple value specifications on any MULTICASE card set remains 5. One 'multiple value specification' consists of a keyword equated to a number of values in parentheses.
2. PIPESIM scans its input data once, starting at the top, so any greek symbols must be equated on the MULTICASE card before they are used in place of a value elsewhere.

5.9.6 Combining MULTICASE and CASE/ENDCASE

The MULTICASE card is a way of achieving a large number of PIPESIM cases for comparatively little input data. It can be called a 'shortcut', for the alternative is to provide explicit extra cases in the input data, with the ENDCASE and CASE cards. In general, it is always possible to 'expand' a job containing MULTICASE cards into one containing a (large?) number of explicit cases, with CASE and ENDCASE cards (except where certain maincodes require the presence of MULTICASE, viz. TABLE and WPCURVE). However, it is not always possible to 'compress' a job consisting of a number of explicit cases into a MULTICASE job. The choice is therefore open to you, depending on the application, whether to use MULTICASE or explicit cases. It is also possible (but tricky) to combine the two.

Why would anyone want to use MULTICASE and explicit cases? There are a number of things that are just not possible with MULTICASE, where explicit extra cases are the only way to achieve the desired result. For example, suppose you want to see the effect of changing pipe diameter and flow rate on the resulting outlet pressure from a flowline. You might set up something like this:

Example

```
MULTICASE LIQ=(200,300,400,500,600,700,800)
MULTICASE ?ALPHA=(4.2,5,5.5,6.2)
PIPE ID=?ALPHA
NODE DIST=0 ELEV=0
```

This would result in 28 cases, one for each flow rate/diameter combination. However, it is incomplete. The pipe wall thickness has not been specified on the PIPE card. And it's when we try to add the wall thickness that the problems arise, because each pipe diameter has its own particular wall thickness. We can't use another greek on the MULTICASE card to specify the wall

thicknesses, because to do so would result in PIPESIM permuting all combinations of diameter and wall thickness and running 112 cases, which is definitely not what we want! The only sensible solution to this problem is to remove the pipe diameter from the MULTICASE, and add 3 more explicit cases to do what we want. However, there is a catch. To see what the catch is, look at the modified input data:

Example

```

PIPE ID=4.2 WT=.6
MULTICASE LIQ=(200,300,400,500,600,700,800)
MULTICASE ?ALPHA=(4.2,5,5.5,6.2)
NODE DIST=0 ELEV=0
...
ENDCASE
CASE Second size
PIPE ?ID=5 WT=0.8
ENDCASE
CASE Third size
PIPE ID=6.2 WT=0.8

```

Now we have complete control over what is with what. The first explicit case will consist of 7 multicase cases, one for each of the flow rates on three MULTICASE card, all at the first pipe diameter and wall thickness. The second explicit case changes the diameter and wall thickness, and because it changes nothing else, it too will consist of 7 multiple cases for each flow rate. And so for the third and fourth explicit cases. What, therefore, is the catch?

The catch in the above example concerns the position of the first PIPE card. Notice that, in the first example, the PIPE card appeared after the MULTICASE cards. It had to appear there because it contained a greek symbol which was defined on the MULTICASE card. Why, therefore, have we moved it? The reason : All input data between the MULTICASE cards and the first NODE card is scanned by PIPESIM on each case. Therefore, values supplied in these cards will override any values supplied in subsequent explicit cases.

This, then is the 'MULTICASE/Explicit Case Combination Catch': it is perfectly possible to mix MULTICASE and explicit cases in the same job, but take care not to put any cards between the MULTICASE cards and the first NODE card unless they really need to be there, that is if they use greek symbols defined on the MULTICASE cards.

There is something else which, although perfectly possible and legal, you are advised not to do: do not put further MULTICASE cards in subsequent cases. You almost certainly will not get the result you expect if you do.

5.9.7 Multiple Case and PS-PLOT

The order in which the subcodes appear on the MULTICASE card determines the order in which the cases will be executed, which in turn determines which points make up a single 'line' on the finished graph. Therefore, care should be exercised to ensure that the subcodes appear in an appropriate order.

An example will serve to clarify this point:

```
MULTICASE WCUT=(0,25,50,75,90) LIQ=(20,50,100,150,200,250)
```

This multicase card will result in 30 cases being executed. Since the LIQ subcode appears last, it forms the 'inner' loop of the execution process: PIPESIM will take the first WCUT value and execute 6 cases, one at each of the liquid flow rates. Then it will take the second WCUT value, and execute another 6 cases, one for each of the liquid flow rates. This loop will be repeated until all 5 water cut values have been executed. Since the plot file is written to at the end of every case, the first 6 points will represent 6 different flow rates at the first water cut, the next 6 will represent 6 flow rates at the second water cut, and so on. Thus the graph that PS-PLOT will draw will contain 5 curves, one for each water cut. Each curve will consist of 6 points, corresponding to the flow rates. Now consider the following MULTICASE card:

```
: MULTICASE LIQ=(20,50,100,150,200,250) WCUT=(0,25,50,75,90)
```

The only difference between this card and the previous card is the order of the subcodes. Now, the WCUT subcode appears last and so will form the 'inner' loop of execution. Thus the graph that PS-PLOT will draw will contain 6 curves, one for each flow rate; each curve will consist of 5 points, one for each water cut.

5.9.8 Reservoir Simulator Tabular Data Interface

Main-code : TABLE

This main-code is used to write tabular performance data to a file for input into another model (such as a reservoir simulator). The effects of variations of one or more (up to four) parameters are investigated. A tabular data file is created in a format as specified under the TYPE sub-code accordingly.

The TABLE main-code should appear before the first [NODE \(p.698\)](#) card in a job and for PIPESIM versions 2.4+ should appear after the [MULTICASE \(p.743\)](#) card.

FLOWTYPE= Type of flow by definition specified by the Reservoir Simulator. This must be specified by the user and may be LIQ or GAS for multiphase flow or OIL, WATER, or GAS for single phase flow

LIQ Liquid

GAS Gas

OIL Oil

WATER Water

TYPE= Type (or format) of data file to which the results of the calculations are written.

PORES PORES

ECLIPSE ECLIPSE

VIP VIP

WEPS WEPS

MORES	MORES
COMP4	COMP4
ADDTEMP=	Type of variable to write to the data file (Default = NO)
NO	Write out only the BHP data
YES	In addition to the BHP data, also write out the Temperature data in a separate VFP Table file
USERELEV=	User specified bottom hole datum depth, in default system unit. If given, it overrides the engine computed default value.
NUMBER=	Table number (between 1 and 10000) which forms part of the name of the interface data file to be created and appears within the file itself (for example, if input file is fred.psm and NUMBER=2, for a production well, the BHP data file name is fred.VFPPROD.BHP.02.txt). Default = 1.
ALQ=	Artifical Lift Quantity. Sensitivity values can be specified for one of the following quantities: INJGAS, GLR, GOR, INJGLR, INJGOR, INJMAS, INJLIQ, PUMPDP, PUMPPR, PUMPPO, PUMPPW, PUMPST and PUMPSP.

Note:

1. Any [RATE \(p.619\)](#) or [ITERN \(p.621\)](#) card in the job input is ignored once the [MULTICASE \(p.743\)](#) and TABLE option have been selected.
 2. The sub-codes can be entered in any order.
-

5.9.9 ASSIGN Changing Profile Data by Assignment

Main-code: ASSIGN

In PIPESIM Versions 2.70 and higher, ASSIGN can be used to supply "parallel values" in MULTICASE'd jobs. Any ASSIGN card defining a multiple-valued symbol, which appears after a MULTICASE card defining a Greek symbol or explicit multi-value subcode, will be treated as parallel to the multicase symbol or sub-code immediately preceding it. Please note, a multi-value ASSIGN card must not appear before the first MULTICASE card.

Suppose you want to set up a MULTICASE job to permute a range of PIPE IDs against something else, e.g. water cut. To make the analysis more rigorous the correct wall thickness (WT) for each ID should be used. If a 3rd Greek on the MULTICASE card is included, this will permute all the combinations of ID and WT, not what is required. However, by using an ASSIGN card, containing the extra data, at the correct point, will result in the required result.

Notes:

1. In PIPESIM Versions 2.41 and higher it is now possible to change variable values within a system profile without repeating the whole profile. Previously, if a value within the profile was changed in a second or subsequent case, then the complete system profile had to be re-entered. It is possible for you to invent one or more symbols and then assign different values to the symbol in subsequent cases. The symbol is typically used within the profile in place of a numeric value. The value can then be assigned outside the profile, thus obviating the need to repeat the entire system profile in subsequent cases.
 2. All symbols must begin with a question mark "?" and are limited to 12 characters. The value assigned can be any appropriate numeric value or alphanumeric string. If delimiters are included, the string must be enclosed in quotes. Up to 30 symbols can be defined and assigned.
 3. ASSIGN may be used to update data within the profile but it can not be used to introduce new main-codes or sub-codes within the profile. If a new main-code or sub-code is introduced within the profile (that is after the second NODE card) then the whole profile must be repeated.
-

Example

```
MULTICASE ?BETA = (0,20,50,80)
MULTICASE ?ALPHA= (3,4,4.5,5,6)
ASSIGN ?THICK=(0.4,0.5,0.5,0.6,0.6)
INLET PRESS=900 TEMP=70
RATE LIQ=3000 WCUT=?BETA
PIPE ID=?ALPHA WT=?THICK
```

5.9.10 OPTIMIZE

Allows the PIPESIM single branch engine to calculate optimal values of parameters to match measured pressure and / or temperature data.

Main-code: OPTIMIZE

- ?OPT01= (....,...) Minimum and maximum values for 1st optimization variable.
- ?OPT02= (....,...) Minimum and maximum values for 2nd optimization variable.
- ?OPT03= (....,...) Minimum and maximum for 3rd optimization variable.
- ?OPT04= (....,...) Minimum and maximum for 4th optimization variable.
- ?OPT05= (....,...) Minimum and maximum for 5th optimization variable.
- PMATCH= Weighting factor for pressure match.
- TMATCH= Weighting factor for temperature match.
- TOL= Accuracy (default 0.02). Optimization converges when the fractional change in the RMS is less than the specified accuracy.

Note: To ensure good convergence, PIPESIM automatically scales the single branch pressure tolerance (PTOL) to be tighter/smaller than the RMS tolerance (TOL). However, if either of these tolerances are manually altered using keywords in such a way that PTOL > TOL, this could result in convergence problems for the data matching operation.

MAXIT=	Maximum number of iterations (default 200). Optimization finishes without convergence if the number of PIPESIM iterations needed exceeds this limit. The actual number of PIPESIM runs may be less than the reported number of iterations. This is because the optimizer may call PIPESIM with the same inputs as an earlier iteration. In this case PIPESIM is not re-run — the results are read from memory.
VERBOSE= ON	Outputs details of optimizer iteration runs
OFF	Outputs details of the initial and optimized runs. (Default)

Examples

Example 1: optimizing flow correlation parameters to match measured pressure data

In this example the friction factor and hold up factor for the vertical flow correlation are set equal to the first two optimization variables. The OPTIMIZE keyword is used to set the range for these two variables (0.2 to 5 in both cases) and to select measured pressure data matching. The OPTIMIZE keyword is used to set the range for these two variables (0.2 to 5 in both cases) and to select measured pressure data matching.

```
optimize ?opt01(0.2,5) ?opt02=(0.2,5) pmatch=1
vcorr      type=HBR ffactor=?opt01 hfactor=?opt02
```

Example 2: optimizing heat loss rate to match measured temperature data

In this example the u value multiplier is set equal to the first optimization variable. The OPTIMIZE keyword is used to set the range (0.01 to 100) and to select measured temperature data matching.

```
optimize ?opt01(0.01,100) tmatch=1
vcorr      ufactor=?opt01
```

Example 3: matching measured pressure and temperature data simultaneously

In this example the two previous examples are combined to match both pressure and temperature data simultaneously. This can be important in cases when the heat loss affect the pressure calculations. The relative weightings of the pressure and temperature matches have been set equal in this example. The MULTICASE keyword is also used to allow multiple flow correlations to be used.

```
multicase ?beta=(ANSARI,DR,HBR)
optimize ?opt01(0.2,5) ?opt02=(0.2,5) ?opt03=(0.01,100) pmatch=1 tmatch=1
```

```
| vcorr      type=?beta ffactor=?opt01 hfactor=?opt02
| options    ufactor=?opt03
```

Example 4: controlling the optimization

To control the optimization you can set the accuracy and maximum number of iterations:

```
| optimize tol=0.001 maxit=500
```

The keyword can be supplied using [Engine Options \(p.172\)](#) to control the [Data Matching \(p.198\)](#) operation.

5.9.11 Wax deposition and Time Stepping modeling options

Main-code: WAX or TIME

The deposition of wax from a fluid on to the walls of the pipe or tubing can be modeled as a function of time. Data must be provided to specify the required wax properties, the required time parameters, and timestep calculation criteria. Since these properties overlap to a considerable degree they can all be provided on either the WAX or TIME maincode. All times are currently assumed to be in HOURS.

Wax deposition can also be modeled on an instantaneous basis. The rate of wax deposition can be calculated, and used to produce a graph of (for example) wax deposition rate against distance. Multiple sensitivity cases can then be used in the usual way to sensitize on variables of interest, so as to observe their effect on wax deposition rate. To do this, ensure your job omits any of the following time-based subcodes, and specify the desired sensitivity variable values with MULTICASE.

Time subcodes

Subcodes concerned only with setting time-based data and options:

DURATION=	Duration of the simulation: provides an alternative to ENDTIME=. To simulate the system over a period of time the duration must be positive: if it is zero, the simulation will consist of a normal PIPESIM steady-state run, valid for an arbitrary instant in time (which is useful for investigating the factors that contribute to wax deposition).
STEP SIZE=	The size of each timestep: only used if OPTION is 1. Timestep size can also be computed automatically during the run by selecting a suitable OPTION.
STARTTIME=	Time at which simulation is to start. (Default zero)
ENDTIME=	Time at which simulation is to finish: see also DURATION= below. (Default zero)
UNITS=	Units of time to be used in simulation. Can be any of YEARS, MONTHS, WEEKS, DAYS, HOURS, MINUTES, or SECONDS.
MINSTEP SIZE=	The minimum allowable time step size that can be computed from OPTIONS 2 through 5.

REPINTERVAL=	The interval between reporting steps. This can be set independently of the timestep size to allow a number of timesteps to occur with no reported output, if desired. The timestep size will be adjusted to ensure that one ends at each report interval, in order to allow the report to be written.
PRINT=	Specifies the number of timesteps for which the detailed wax deposition output page will appear. This value will override the CASES= subcode of PRINT.
RESTART=	Time at which to restart from a previous simulation. If a restart time is specified it overrides any supplied STARTTIME. The wax profile to restart from is obtained from the restart file, see READRESTART.
READRESTART=	Specifies the name of the restart file to read (default model name.WRS). Has no effect unless accompanied by RESTART=. The file will be searched for a profile representing the specified restart time. If necessary, 2 existing profiles will be interpolated to create a profile representing the required time.
WRITERESTART=	Specifies the name of the restart file to write to (default model name.WRS) A restart file is always written if the run is stepping through time (that is has a positive duration, see above). If the run is restarting and the read and write restart filenames are identical, the new profiles will be written at the file position corresponding to the time of restart, thus any pre-existing profiles for later timesteps will be overwritten and lost. If this is not the desired behavior, this or the previous subcode can be used to specify alternate file names, which can be copies or new files as appropriate. In addition the model name may be changed (with the File/Save As) menu.

Termination subcodes

Subcodes concerned with or terminating the timestepping simulation, as a result of simulation conditions:

MAXPIGDP=	The maximum Delta Pressure available to push a wax removal scraper pig through the line. The simulation will terminate early when sufficient wax has deposited to cause the specified DP to occur.
MAXSYSDP=	An upper limit on the Delta Pressure between system inlet and outlet (psi or bar). In order to take effect, the simulation operation must have specified that inlet pressure or outlet pressure be the calculated variable.
MAXWAXTHICK= or MAXTHICKNESS=	An upper limit in the thickness of the wax deposit anywhere in the system (in or mm).
MAXVOLUME= or MAXWAXVOLUME= or MAXPIGVOLUME=	The maximum volume of wax allowed to accumulate in the system. (ft ³ or m ³)

MINLIQRATE=	A lower limit for system stock-tank liquid flowrate. (bbl or m ³). In order to take effect, the simulation operation must have specified that flowrate be the calculated variable.
MINGASRATE=	A lower limit for system stock-tank gas flowrate. (mmscf ³ or mmm ³). In order to take effect, the simulation operation must have specified that flowrate be the calculated variable.
MINMASSRATE=	A lower limit for system total mass flowrate. (lb/sec or kg/sec). In order to take effect, the simulation operation must have specified that flowrate be the calculated variable.
MINID= or MINWAXID=	A lower limit on the internal Diameter of the wax deposit anywhere in the system (in or mm).

Wax subcodes

Subcodes concerned with Setting wax properties, deposition properties, and modelling options:

METHOD= or CLIENTMODEL=	The chosen Wax Deposition method. May be: DBR or DBRS: D. B. Robinson and Associates, single-phase DBRM: D. B. Robinson and Associates, multi-phase SHELL: Shell oil Company proprietary method BP: British Petroleum Company proprietary method Note that all methods require an explicit license. D .B. Robinson and Associates is a wholly-owned subsidiary of Schlumberger.
DENSITY= or RHOWAX= or WAXRHO=	Wax density (lb/ft ³ or kg/m ³).
CONDUCTIVITY= or WAXCONDUCTIVITY=	Wax thermal conductivity (BTU/hr/ft/F or W/m/C).
ROUGHMODE =	Specifies whether wax wall roughness is to be calculated. Set to INPUT to use the roughness supplied with ROUGH= below (or on the PIPE statement), or CALC to have it calculated.
ROUGHNESS= or WAXROUGHNESS= or ROUGH=	Surface roughness of the wax (in or mm) .
WAXYIELDSTR = or TAUWAX= or WAXTAU= or YIELDSTRENGTH=	the Yield strength of the deposited wax (psi or/bar). Used to calculate DP during pigging.
IFCMETHOD=	Inside Film Coefficient method. This is provided for backwards compatibility, and accepts the same values as SPIFCMETHOD= and MPIFCMETHOD= subcodes on the HEAT statement.

BP, DBRS or DBRM method subcodes

Subcodes that are specific to the BP, DBRS or DBRM methods:

FILENAME= or FNAME= or INPUTFNAME=	The name of the Wax properties file. the BP and DBR methods require a separate file to hold wax thermodynamic and deposition properties, the format of which is proprietary to each method.
BPFILENAME= or BPFNAME= or BPINPUTFNAME=	Same as FILENAME=, and sets METHOD=BP.
DBRFILENAME= or DBRFNAME= or DBRINPUTFNAME=	Same as FILENAME=, and sets METHOD=DBRS.
OILFRAC=	Oil fraction in the wax (0 to 0.99).
SHEARCOEF= or SRMULT=	Shear coefficient to simulate wax stripping. Also known as Shear reduction Multiplier. Note: Each keyword has different ranges: SHEARCOEFF= is intended to be used with the BP method (0 to 1); SRMULT= is intended to be used with the DBR method (-10 to +10).
DIFFCO= or MDMULT= DIFCOEFACTOR=	Molecular Diffusion coefficient multiplier. Note: Each keyword has different ranges: Both DIFFCO= and DIFCOEFACTOR= are intended to be used with the BP subcode (0.01 to 1); MDMULT= is intended to be used with the DBR subcode (-10 to +10).
COEFWAXK=	Multiplier for the oil thermal conductivity, to simulate the thermal conductivity of the wax deposit. Must be in the range 1 to 2. BP method only
DCMETHO= or FLAGDIFFCOEF=	Diffusion Coefficient method, may be: WILKECHANG: Wilke & Chang HAYDUMINHAS: Hayduk & Minhas USER: user-supplied with DIFFCO= BP method only.
ROUGHCOEF=	Roughness multiplier (0 to 1). BP method only.
BPFFMETHOD=	BP Friction factor method. Can be set to ON or OFF. Controls which Friction factor is used for calculating the Inside Film Coefficient with BP method (IFCMETHOD=BP). If set to ON, then the friction factor is calculated using the BP internal flow correlation. if set to OFF, the friction factor is calculated by the PIPESIM selected flow correlation. BP method only.

COEFWAXK=	Wax Thermal Conductivity Coefficient (0 to 100). BP method only.
LHCOEF=	Coefficient on Liquid Holdup for two-phase scaling (-5 to +5). DBRM method only.
S2YCOEFF=	Coefficient on ratio of shear stress to yield stress (-5 to +5). DBRM method only.
SFCOEF=	Coefficient on shear factor used in porosity calculation (-5 to +5). DBRM method only.
SWCOEFF=	Coefficient on Surface Wetting for two-phase scaling (-5 to +5) DBRM method only.
T2RCOEFF=	Coefficient on ratio of wax thickness to radius (-5 to +5).DBRM method only.
TFCOEF=	Coefficient on temperature factor used in oil fraction calculation (-5 to +5). DBRM method only.
TRANSTEMPRANGE= or WAXTRANTEMP=	Wax transition temperature range / region (F or C). DBRM method only.

Shell subcodes

Subcodes specific to the SHELL method:

OPTION=	Options to control how the timestep size is computed. An integer in the range 1 through 5, meaning: 1: Fixed timestep using the user's specified step size. 2: Auto timestep, all constraints: Wax DX, HTC , DP. 3: Auto timestep, wax DX and DP constraints only. 4: Auto timestep, wax DX and HTC constraints only. 5: Auto timestep, wax DX constraint only.
MINDX=	The minimum allowable increase in wax ID. This sets a lower limit on the timestep size computed from OPTIONS 2 through 5.
SETDX=	The maximum increase in wax ID. This is used to compute the timestep size from OPTIONS 2 through 5.
HTCLIMIT=	Controls the application of the Heat Transfer Coefficient limit on the timestep size. Set to ON or OFF.
RELAX=	The relaxation factor for automated timestep adjustment computed from OPTIONS 2 through 5. Must be a real number between 0 and 1; higher values favour the new value, lower the old.

DFRACTION=	Fraction of the pressure drop change allowed with the new timestep (0.01 == 1%) computed from OPTIONS 2 through 5.
CWDPRES=	Critical Wax Deposition Pressures. A vector of pressures, up to 30 may be provided, which must be in ascending order (psia or bara). Values are separated by commas and enclosed in parentheses.
CWDTEMP=	Critical Wax Deposition Temperatures. A vector of temperatures, up to 30 may be provided, to correspond with the values for CWDPRES= (F or C). Values are separated by commas and enclosed in parentheses.
MPTEMP=	Modeling Parameter temperatures. A vector of temperatures, up to 30 may be provided, which must be in ascending order (F or C). Values are separated by commas and enclosed in parentheses.
MPA=	Modeling Parameter A values. A vector of coefficients, up to 30 may be provided, to correspond with the values for MBTEMP=. Values are separated by commas and enclosed in parentheses.
MPB=	Modeling Parameter B values. A vector of coefficients, up to 30 may be provided, to correspond with the values for MBTEMP=. Values are separated by commas and enclosed in parentheses
MPC=, MPD=, MPE=, MPF=, MPG=, MPH=, MPI=, MPJ=	Additional subcodes to specify modelling parameters C through J. Each requires a vector of coefficients, up to 30 may be provided, to correspond with the values for MBTEMP=. Values are separated by commas and enclosed in parentheses
RATEMODEL= or MODEL=	Deposition rate model number. Currently there is only one rate model, number 1.
CWRPRES=	Critical Wax Removal Pressures. A vector of pressures, up to 30 may be provided, which must be in ascending order (psia or bara). Values are separated by commas and enclosed in parentheses.
CWRTEMP=	Critical Wax Removal Temperatures. A vector of temperatures, up to 30 may be provided, to correspond with the values for CWDPRES= (F or C). Values are separated by commas and enclosed in parentheses.
MODE=	Controls whether to model wax deposition or removal: set to DEPOSITION or REMOVAL.

5.10 PIPESIM-Net keywords

[SETUP \(p.760\)](#)

[BRANCH \(p.762\)](#)

[SOURCE \(p.764\)](#)

[SINK \(p.767\)](#)

[JUNCTION \(p.769\)](#)[NSEPARATOR \(p.769\)](#)

Note: The preferred order of statements is: [SETUP \(p.760\)](#) , [SOURCE \(p.764\)](#) , [JUNCTION \(p.769\)](#) , [SINK \(p.767\)](#) , [BRANCH \(p.762\)](#) , [NSEPARATOR \(p.769\)](#).

See [Input Files and Data Conventions \(p.599\)](#) for more detailed information on formatting.

5.10.1 SETUP

SETUP is a [network keyword \(p.759\)](#), used to define various network options.

Subcodes

TITLE=	The model title. Can include spaces if enclosed in quotes.
TOLERANCE=	The overall tolerance of the converged network solution. Must be between 0.5 and 1e-6, default 0.01.
MAXITER=	The maximum allowable number of overall network iterations. Must be between 3 and 1000, default 100.
FLUIDMODEL= or COMPOSITION=	An override on the type of fluid model to use. This is not normally required, as it is obtained from the fluid definitions supplied in the branch files, but can be supplied here if desired. Can be set to: BLACKOIL: Fluid type is set to black oil COMPOSITION: Fluid type is set to compositional STEAM: fluid type is set to steam
UNSTABLEWELL=	How to treat unstable wells. If the converged network solution results in a well operating in its unstable, or liquid-loaded region, you may wish it be automatically shut in. Can be set to SHUT or FLOW, meaning: SHUT: Shut in any well that is operating in its unstable region. This is the default. FLOW: allow unstable wells to remain in operation.
RECIPBYPASS=	How to treat redundant Reciprocating Compressor (recips). The network solution can converge with a recip in a so-called “redundant” state, where pressure actually reduces across it instead of increasing. Clearly, a recip in this state is not doing the job it was intended to do, and the network would be better off without it. If this subcode is set to ON, then any redundant recip will be bypassed, i.e. effectively removed from the model solution. Can be set to ON or OFF, default ON.

ALGORITHM=	The choice of network solution algorithm. Can be set to WEGSTEIN or JACOBIAN, default WEGSTEIN.
WOFLMODE=	Global settings for Wells Off Line Mode. May be set to: OFF: Disable WOFL mode. All pressure-specified production wells and source branches are modelled ON-line. CREATE: Enable WOFL mode, and unconditionally create WOFL files for all pressure-specified sources and production wells at the start of the simulation. CREATE?: Enable WOFL mode. Read and validate any existing WOFL files, comparing the fluid definition, pressure boundary condition, and branch geometry in them to the corresponding values in the current model for the branch. If they match, use the file, otherwise re-create it. USE: Enable WOFL mode. Unconditionally read any existing WOFL files and use them, despite possible mismatch between them and the current model settings. No new files will be created.
ECHOBRANCH=	Allows the contents of all well and branch geometry files to be echoed to the network output file. Can be set to YES or NO, default NO.
SKIPINACTIVE=	Controls the “skipping” (i.e., omission of processing) of geometry files for inactive branches: can be set to YES or NO. In a coupled PIPESIM/Eclipse/IAM simulation, it is common practice to start the simulation with a number of branches turned OFF. The timestepping simulation then turns them ON at a later timestep, maybe after many hours of simulation CPU time. If at this time it is found that the branch geometry files concerned contain syntax or logic errors, the simulation has to be aborted, resulting in much time and work lost. Clearly, for this type of simulation, it is preferable to check and validate all branch geometry files at the start of the run, so as to diagnose such errors as early as possible. Thus the default state for a coupled simulation is NO. By contrast, in a normal uncoupled PIPESIM run, a branch's ON or OFF state will not change during the run, so any syntax or logic errors caused by the contents of inactive geometry files can be ignored. Thus the default state for an uncoupled simulation is YES. This flag can also be set with the -E command line switch, which has the same effect as setting it to YES.
RESTARTINTERVAL=	Specifies the time interval between writing restart files: a value in seconds, default 1800. The interval is measured in real (i.e. wallclock, not CPU) seconds.

Restart files are written at the end of every simulation, and at the end of any as-yet-unconverged network iteration after the specified interval. The purpose of the restart file is to allow the simulation to be restarted, which is useful to allow a new simulation to re-use the converged results of a previous one, and/or to recover from a simulation that terminated abnormally. However, the file writing can take considerable time, and so impose a speed penalty on the overall simulation. So there is strong incentive to minimise the frequency of writing them.

By default the files are written every half-hour, the idea being that if the program is interrupted or fails abnormally, you can restart it, having lost at most half an hour's work. If you would prefer to lose less work in this event, set the interval to a value smaller than 1800 seconds, but by doing so you accept the extra overhead of writing the files more often.

You can also increase the interval to reduce the restart file writing. Setting it to a very large value (eg 1e10) will result in the files being written only when the simulation converges, or when it hits iteration limit.

ECHONET= Controls the echo of the network input data to the output file. can be set to YES or NO, default YES.

5.10.2 BRANCH

BRANCH is a [network keyword \(p.759\)](#), used to define a branch and associated network topology.

Subcodes

NAME= The name of the branch. Can include spaces if enclosed in quotes.

FILENAME= The file name containing the Branch's input data, as formatted for a Single-branch PIPESIM model. Can include spaces if enclosed in quotes. See note 1.

START= The branch start Network Node name. See note 1.

END= The branch end Network Node name. See note 1.

BLOCK= Specifies a direction in which flow is “blocked”, i.e. not allowed to go. Can be set to:

NONE: No flow block exists, so flow may go in either direction

FORWARD: Flow is blocked in the forward direction, so it may only go in reverse.

REVERSE: Flow is blocked in the reverse direction, so it may only go forward,

	BOTH: flow is blocked in both directions, so the branch is effectively inactive.
ON	Specifies that the branch is “active” and that no flow block exists in it, so flow can go in either direction. has the same effect as BLOCK=NONE.
OFF	Specifies that the branch is “inactive” and flow is blocked in both directions. has the same effect as BLOCK=BOTH.
ESTLIQUID= or EST_LIQUID=	An estimate of the flowrate in the branch, as a stock-tank liquid rate (sbbl/d or sm3/d). The iterative network solution algorithm will commence with this as the branch flowrate.
ESTGAS= or EST_GAS=	An estimate of the flowrate in the branch, as a stock-tank gas rate (mmscf/d or mmsm3/d). The iterative network solution algorithm will commence with this as the branch flowrate.
ESTMASS= or EST_MASS=	An estimate of the flowrate in the branch, as a mass rate (lb/sec or Kg/sec). The iterative network solution algorithm will commence with this as the branch flowrate.
UPPERMASS= or MAXMASS= or LIMITMASS=	Upper limit of mass flowrate for the branch (lb/sec or Kg/sec).. See note 2.
UPPEROIL= or MAXOIL= or LIMITOIL=	Upper limit of oil flowrate for the branch(sbbl/d or sm3/d). See note 2.
UPPERLIQ= or MAXLIQ= or LIMITLIQ=	Upper limit of liquid flowrate for the branch (sbbl/d or sm3/d). See note 2.
UPPERGAS= or MAXGAS= or LIMITGAS=	Upper limit of gas flowrate for the branch (mmscf/d or mmsm3/d). See note 2.
UPPERWAT= or MAXWAT= or LIMITWAT=	Upper limit of water flowrate for the branch (sbbl/d or sm3/d). See note 2.
USERRESTART=	A per-branch override on the use of solution data from restart files. When a model run is restarted (p.212) , by default, the solution information for all branches is extracted from the restart file, and used as the start point for the run. This subcode allows the restart information for this branch to be ignored, so the run will use default information for the branch. Can be set to YES, to use the restart data, or NO, to ignore it. Default is YES.

Notes:

1. The names of the network nodes which adjoin the branch must be specified with the START= and END= subcodes. Network nodes are defined with the statements [SOURCE](#), ([p.764](#)) [SINK](#) ([p.767](#)) , [JUNCTION](#) ([p.769](#)) and [NSEPARATOR](#) ([p.769](#)) statements. The pipeline geometry as detailed in the file supplied with the FILENAME= subcode, is assumed to start at the network node named with the START= subcode, and end at the node named with END=. Note, this does not specify the direction of fluid flow: the network solution will determine if the branch actually flows forward, i.e. with the geometry direction, or reverse, i.e. against the geometry direction.
 2. Any combination of Maximum Flowrate limits may be specified, the simulation will enforce whichever turns out to be most limiting. The limits are enforced by the addition of a choke at the branch outlet. The choke bean diameter is calculated so as to enforce the limit, so a pressure drop will occur across the choke. Flowrate limits may be applied to all branches, except for (a) any branch connected to the outlet of a network separator, and (b) any branch draining a source with a fixed flowrate specification.
-

5.10.3 SOURCE

SOURCE is a [network keyword](#) ([p.759](#)), used to define conditions at a network inlet.

Subcodes

NAME=	The name of the source. Can include spaces if enclosed in quotes.
PRESSURE=	Source pressure specification (psia or bara). See note 1.
TEMPERATURE=	Temperature of fluid flowing from the source (F or C). If absent, this is obtained from the data in the branch geometry file.
LIQUIDRATE= or LIQ=	Source flowrate specification, as a stock tank liquid rate (sbbl/d or sm3/d). See note 1.
GASRATE= or GAS=	Source flowrate specification, as a Stock tank gas rate (mmscf/d or mmSCM/d) . See note 1.
MASSRATE= or MASS=	Source flowrate specification, as a Stock tank mass rate (lb/sec or Kg/sec). See note 1.
REBC=	“Remove Existing Boundary Condition” for the source. This is used when multiple SOURCE statements refer to the same named source, and you want this statement to remove all boundary conditions for this source specified with earlier statements.
FCLIQUID=	Source flowrate specification, as a Flowing liquid rate (bbl/d or m3/d). See notes 1 & 2.

FCGAS=	Source flowrate specification, as a Flowing gas rate (mmscfd or sm3/d). Note the units of this are at stock-tank conditions.
FCPRESSURE=	The pressure for the accompanying flowing rate specified with FCLIQ= or FCGAS= (psia or Bara)
FCTEMPERATUR=	The temperature for the accompanying flowing rate specified with FCLIQ= or FCGAS= (F or C)
CURVEP=	For curve specified source, an array of pressures (psia or bara). See note 3. Example: CURVEP=(20,1000,2000) psia
CURVEL=	For curve specified sources, an array of liquid rates (bbl/d or m3/d). See note 3. Example: CURVEL=(20,1000,2000) bbl/d
CURVEG=	For curve specified sources, an array of gas rates (mmscfd or mmsm3d). See note 3. Example: CURVEG=(20,1000,2000) mm scfd
CURVEM=	For curve specified sources, an array of mass rates (lb/s or Kg/s). See note 3. Example: CURVEM=(20,1000,2000) lb/s
CURVET=	For curve specified sources, an array of temperatures (F or C). See note 3. Example: CURVET=(20,40,60) F
ON	Specifies that the source is active, or switched ON.
OFF	Specifies that the source is inactive, or switched OFF.
CURVEFILE=	Specifies that the source and adjoining branch has already been simulated in Wells Off-Line (WOFL) mode, and that the results of this are available in the named file. Example: CURVEFILE='Curve1.PWH' A number of additional special values may be supplied instead of the filename, these are distinguished by the first character being an asterisk, '*', namely: CURVEFLIE=*USE: this has the same effect as above, but the filename to read from is assumed from the default source and branch names.

	CURVEFLIE=*CREATE : Specifies that a WOFL file for the source and adjoining branch are to be created at the start of the network simulation. The network simulation will then use the results in this file.
	CURVEFLIE=*CREATE?: Same as above, except that any pre-existing file will be used if its specifications match the current network's.
QUALITY=	For steam systems, the quality (fraction gas) of the steam flowing into the source. If absent, this will be obtained from the branch geometry file.
CURVESENS_P=	For WOFL specified sources, supplies sensitivity information in units that match the data in the WOFL file.
CURVESENS_T=	For WOFL specified sources, supplies sensitivity information in units that match the data in the current (.TNT) file.
CURVESENS_S=	For WOFL specified sources, supplies sensitivity information in Strict SI units.
UPPERMASS= or MAXMASS= or LIMITMASS=	Upper limit of mass flowrate for the source (lb/sec or Kg/sec).. See note 4.
UPPEROIL= or MAXOIL= or LIMITOIL=	Upper limit of oil flowrate for the source (sbb/d or sm3/d). See note 4.
UPPERLIQ= or MAXLIQ= or LIMITLIQ=	Upper limit of liquid flowrate for the source (sbb/d or sm3/d). See note 4.
UPPERGAS= or MAXGAS= or LIMITGAS=	Upper limit of gas flowrate for the source (mmscf/d or mmsm3/d). See note 4.
UPPERWAT= or MAXWAT= or LIMITWAT=	Upper limit of water flowrate for the source (sbb/d or sm3/d). See note 4.
ELEVATION=	Absolute elevation of the source (ft or m). If supplied, this will be used as a datum for plotting branch elevations. If more than one junction has an elevation, they will be used to cross-check with other source, sink and junction elevations, to help identify where loop elevation mismatch error(s) have occurred. N.B. It is useful to supply the absolute elevations of any number of Network nodes (Junctions, Sources, and Sinks). This allows the elevation data in the connecting branches to be checked. Note that the same node name may appear in multiple SOURCE statements: the data on each statement is additive for the overall node specification. Thus for example,

a series of SOURCE statements can be supplied under **Setup ➤ Engine options** to set the node elevations for sources already specified in the GUI's model. Same goes for JUNCTION and SINK statements.

ESTPRESSURE=	Estimate of pressure to be used as a starting point for the network solution (psia or bara).
--------------	--

Notes:

1. A source may have a pressure specification, or a flowrate specification, or a curve specification. These are known as [Hydraulic Boundary Conditions. \(p.41\)](#)
2. A “flowing flowrate” may be specified as an alternative to a stock-tank flowrate. It must be accompanied by FCPRES= and FCTEMP= .
3. A source may be specified with a curve of flowrate against pressure, as an alternative to a fixed pressure or flowrate. The subcodes CURVEP=, and one of (CURVEG=, CURVEL=, or CURVEM=) are used for this, they all accept [Multiple Value Data Set \(p.603\)](#). The curve may be accompanied by a temperature array with CURVET=. All subcodes so specified must have the same number of values. Between 3 and 30 values may be supplied.
4. Any combination of Maximum Flowrate limits may be specified, the simulation will enforce whichever turns out to be most limiting. The limits are enforced in the adjoining branch, by the addition of a choke at the branch outlet. The choke bean diameter is calculated so as to enforce the limit, so a pressure drop will occur across the choke.

5.10.4 SINK

SINK is a [network keyword \(p.759\)](#), used to define conditions at a network outlet.

Subcodes

NAME=	The name of the sink. Can include spaces if enclosed in quotes.
PRESSURE=	Sink pressure specification (psia or bara). See note 1.
LIQUIDRATE= or LIQ=	Sink flowrate specification, as a stock tank liquid rate (sbb/d or sm3/d). See note 1.
GASRATE= or GAS=	Sink flowrate specification, as a Stock tank gas rate (mmscf/d or mmscm/d). See note 1.
MASSRATE= or MASS=	Source flowrate specification, as a Stock tank mass rate (lb/sec or Kg/sec). See note 1.
REBC=	“Remove Existing Boundary Condition” for the sink. This is used when multiple SINK statements refer to the same named sink, and you want this statement to remove all boundary conditions for this sink specified with earlier statements.

ON	Specifies that the sink is active, or switched ON.
OFF	Specifies that the sink is inactive, or switched OFF.
UPPERMASS= or MAXMASS=	Upper limit of mass flowrate for the sink (lb/sec or Kg/sec).. See note 2.
UPPEROIL= or MAXOIL=	Upper limit of oil flowrate for the sink (sbb/d or sm3/d). See note 2.
UPPERLIQ= or MAXLIQ=	Upper limit of liquid flowrate for the sink (sbb/d or sm3/d). See note 2.
UPPERGAS= or MAXGAS=	Upper limit of gas flowrate for the sink (mmcf/d or mmsm3/d). See note 2.
UPPERWAT= or MAXWAT=	Upper limit of water flowrate for the sink (sbb/d or sm3/d). See note 2.
ELEVATION=	Absolute elevation of the sink (ft or m). If supplied, this will be used as a datum for plotting branch elevations. If more than one network node has an elevation, they will be used to cross-check with other source, sink and junction elevations, to help identify where loop elevation mismatch error(s) have occurred. N.B. It is useful to supply the absolute elevations of any number of Network nodes (Junctions, Sources, and Sinks). This allows the elevation data in the connecting branches to be checked. Note that the same node name may appear in multiple SINK statements: the data on each statement is additive for the overall node specification. Thus for example, a series of SINK statements can be supplied under Setup » Engine options to set the node elevations for sinks already specified in the GUI's model. The same goes for JUNCTION and SOURCE statements.
ESTPRESSURE=	Estimate of sink pressure to be used as a starting point for the network solution (psia or bara).

Notes:

1. A sink may have a pressure specification, or a flowrate specification. These are known as [Hydraulic Boundary Conditions. \(p.41\)](#)
 2. Any combination of Maximum Flowrate limits may be specified, the simulation will enforce whichever turns out to be most limiting. The limits are enforced in the adjoining branch, by the addition of a choke at the branch outlet. The choke bean diameter is calculated so as to enforce the limit, so a pressure drop will occur across the choke.
-

5.10.5 JUNCTION

JUNCTION is a [network keyword \(p.759\)](#), used to define a junction, or to supply additional or override data for an existing junction.

Subcodes

NAME=	Name of the junction. Can include spaces if enclosed in quotes.
ESTPRESSURE=	Estimate of pressure to be used as a starting point for the network solution (bara or psia)
ESTTEMPERATURE=	Estimate of fluid temperature to be used as a starting point for the network solution (F or C)
ELEVATION=	Absolute elevation of the junction (ft or m). If supplied, this will be used as a datum for plotting branch elevations. If more than one Network node has an elevation, they will be used to cross-check with other node elevations, to help identify where loop elevation mismatch error(s) have occurred. N.B. It is possible, and useful, to supply the absolute elevations of any number of Network nodes (Junctions, Sources, and Sinks). This allows the elevation data in the connecting branches to be checked. Note that the same node name may appear in multiple JUNCTION statements: the data on each statement is additive for the overall node specification. Thus for example, a series of JUNCTION statements can be supplied under Setup » Engine options to set the node elevations for junctions already specified in the GUI's model. Same goes for SOURCE and SINK statements.

5.10.6 NSEPARATOR

NSEPARATOR is a [network keyword \(p.759\)](#), used to define a network separator.

Subcodes

NAME=	Name of the separator
FEEDBRANCH=	Name of the branch feeding the separator
DISCARDBRANCH=	Name of the branch to receive the “discarded” stream. See note 1.
TYPE=	The phase of the “discarded” stream: may be GAS, LIQUID, or WATER. See note 1.
PRESSURE=	Separator pressure (psia or bara). This is optional: see note 2.
EFFICIENCY=	Percentage efficiency of the separation process: see note 3. Must be in the range 10 to 100.

Notes:

1. A network separator causes a feed stream to be separated into 2 outlet streams, as specified by the TYPE= subcode. They are known as the “discard” stream and the “kept” stream, for compatibility with the [single-branch separator \(p.705\)](#). In a network model however, the term “discard” is misleading, because the discard stream is not “discarded”, it is separated and made to flow into the branch named with the DISCARDBRANCH= subcode.
 2. The requirements of the network solution dictate that a pressure discontinuity must occur at the outlets of a network separator. If a pressure has been specified, then both of the outlet branches will exhibit a pressure discontinuity, calculated to ensure that the separated streams' flowrates are maintained in the downstream network(s). If a pressure is not specified, then only the “discard” branch will exhibit a discontinuity. These discontinuities represent the necessary pressure control valves and/or pumps that are required to maintain liquid level control in the separator.
 3. The efficiency term refers to how much of the “discard” phase is separated from the feed stream. For example, an efficiency of 90% in a gas separator will cause 90% of the gas phase to be sent down the discard branch; the remaining gas, plus ALL of the liquid, will go down the keep branch.
 4. Separators work at flowing, or in-situ, pressure and temperature. The flowing phase split as predicted by the selected fluid PVT model will usually be very different from the stock-tank phase split, please bear this in mind when you look at the resulting branch flowrates. In particular, PIPESIM allows you to display branch and node flowrates using the “report tool”, but alas this shows only stock-tank rates, which are not useful to understand separator performance.
 5. If the phase to be separated does not exist, then clearly the separator cannot function as expected. In this case all flow will go down the keep branch. Inlet and outlet compositions will be identical.
 6. If the phase to be separated is the only phase present, then clearly the separator cannot function as expected. In this case the “efficient” fraction of the flow will go down the discard branch, with the remainder going down the keep branch. Inlet and outlet compositions will be identical.
-

5.11 Keyword Index

[Input Files and Input Data Conventions \(p.599\)](#)

[General Data \(p.606\)](#)

[Compositional Data \(p.729\)](#)

[Blackoil Data \(p.717\)](#)

[Heat Transfer Data \(p.707\)](#)

[Flow Correlation Data \(p.638\)](#)

[System and Equipment Data \(p.672\)](#)

[Well Performance Modeling \(p.650\)](#)

[PIPESIM Operations \(p.738\)](#)

[PIPESIM-Net Keywords \(p.759\)](#)

5.11.1 Keyword List

A (p.595) B (p.595) C (p.596) D (p.596) E (p.596) F (p.596) G (p.596) H (p.597) I (p.597) J (p.597)
K (p.597) L (p.597) M (p.597) N (p.598) O (p.598) P (p.598) Q (p.598) R (p.598) S (p.598) T
(p.599) U (p.599) V (p.599) W (p.599) X (p.599) Y (p.599) Z (p.599)

A

- [ASSIGN \(p.751\)](#)

B

- [BACKPRES \(p.664\)](#)
- [BEGIN \(p.634\)](#)
- [BLACKOIL \(p.717\)](#)
- [BRANCH \(p.762\)](#)

C

- [CASE \(p.608\)](#)
- [CALIBRATE \(p.727\)](#)
- [CHOKE \(p.673\)](#)
- [COAT \(p.710\)](#)
- [COMP \(p.732\)](#)
- [COMPCRV \(p.677\)](#)
- [COMPLETION \(p.651\)](#)
- [COMPRESSOR \(p.679\)](#)
- [CONE TAB \(p.663\)](#)
- [CONFIG \(p.715\)](#)
- [CONTAMINANTS \(p.728\)](#)
- [CORROSION \(p.639\)](#)
- [CPFLUID \(p.726\)](#)

D E

- [END \(p.634\)](#)
- [ENDCASE \(p.634\)](#)
- [ENDJOB \(p.634\)](#)

- EROSION (p.639)
- EQUIPMENT (p.443)
- ESP (p.701)
- EXPANDER (p.683)

F

- FETKOVICH (p.654)
- FITTING (p.684)
- FLOWLINE (p.651)
- FMPUMP (p.686)
- FORCHHEIMER (p.670)
- FRACTURE (p.670)
- FRAMO2009 (p.686)

G

- GASLIFT (p.687)

H

- HCORR (p.645)
- HEATER (p.687)
- HEADER (p.607)
- HEAT (p.707)
- HORWELL (p.664)
- HVOGEL (p.670)

I

- IFPPSSE (p.655)
- IFPTAB (p.662)
- IFPCRV (p.660)
- INLET (p.623)
- INJGAS (p.693)
- INJFLUID (p.693)
- INJPORT (p.691)
- IPRCRV (p.660)
- ITERN (p.621)

J

- [JOB](#) (p.607)
- [JONES](#) (p.655)
- [JUNCTION](#) (p.769)

K

- [KCOAT](#) (p.712)

L

- [LAYER](#) (p.666)
- [LVIS](#) (p.721)

M

- [MPBOOSTER](#) (p.695)
- [MPUMP](#) (p.696)
- [MULTICASE](#) (p.743)

N

- [NAPLOT](#) (p.739)
- [NAPOINT](#) (p.743)
- [NODE](#) (p.698)
- [NOPRINT](#) (p.634)
- [NSEPARATOR](#) (p.769)

O

- [OPTIONS](#) (p.609)
- [OPTIMIZE](#) (p.752)

P

- [PCP](#) (p.701)
- [PERMCRV](#) (p.668)
- [PERMTAB](#) (p.669)
- [PETROFRAC](#) (p.737)
- [PIPE](#) (p.699)
- [PLOT](#) (p.631)
- [PRINT](#) (p.624)
- [PROP](#) (p.719)

- PUMP (p.701)
- PUMPCRV (p.677)
- PUSH (p.636)

Q R

- RATE - Compositional (p.738)
- RATE - Blackoil (p.619)
- REINJECTOR (p.704)
- RISER (p.651)

S

- SEPARATOR (p.705)
- SETUP (p.760)
- SINK (p.767)
- SLUG (p.640)
- SOURCE (p.764)
- SPHASE (p.648)

T

- TABLE (p.750)
- TCOAT (p.711)
- TIME (p.205)
- TPRINT - Compositional (p.738)
- TPRINT - Blackoil (p.726)
- TRANSIENT (p.671)
- TUBING (p.651)

U

- UNITS (p.608)
- USERDLL - Flow Correlations (p.650)
- USERDLL - Equipment (p.638)

V

- VCORR (p.641)
- VOGEL (p.654)

W

- Wax Deposition (p.205)

- WELLHEAD (p.706)
- WELLPI (p.653)
- WCOPTION (p.657)
- WPCURVE (p.654)

XYZ

6

Open Link

Open Link is a set of API calls that can be made by any application to control PIPESIM or elements of PIPESIM.

Two sets of controls exist:

1. Controlling the GUI.
2. Controlling the calculation engines directly.

Please refer to the Open Link documentation installed with the software.

Third party applications

PIPESIM has been designed with "Openness" in mind. Therefore key modules can be "driven" from 3rd party applications, for example, Microsoft Excel, VB, C++, and so on.

Schlumberger have collected these open modules together into a single product, called Open Link. As the development of Open Link components are an ongoing process the latest documentation is available with the installed documentation.

Examples of using the Open Link technology, using Microsoft Excel are provided in the Case Studies\Open Link directory. Documentation is also provided in the form of PDF files (AdobeAcrobat reader is required for this; a copy of which can be found on the PIPESIM CD) from the start menu Schlumberger\PIPESIM\Documentation\Open Link or from the directory Program Files\Schlumberger\pipesim\programs Folder.

Index

0-9

- modeling
 - sand modeling
 - scale prediction
 - solids appearance
 - wax modeling

A

- Abort Run 195
- Alhanati 232
- Annular Gas Pressure Design Method 234
- Ansari 377
- API14B critical flow correlation 450
- Aqueous Components — cubic EoS 548
- AQUEOUS keyword 729
- Artificial Lift analysis
 - basic steps 213
 - creating curves 213
 - performance curves 205
- Asphaltene 551
- ASSIGN keyword 751

B

- Babu and Odeh 55
- Back Pressure equation
 - 4 point test 60
 - BACKPRES keyword 664
 - data required 60
 - details 402
- Batch mode 265
- Beggs and Brill 377
- Beggs and Robinson
 - Dead oil viscosity 516
 - live oil viscosity 518
- BEGIN keyword 634

- Benedict-Webb-Rubin-Starling 539
- Bergman and Sutton
 - undersaturated oil viscosity 522
- binary interaction parameter 134
- Biot number 499
- BIP 147, 148
- Black oil
 - CALIBRATE keyword 727
 - calibration 136
 - correlations 506
 - data sets 135
 - enthalpy 526
 - fluid modeling 134
 - fluid modeling theory 505
 - mixing 528
 - options 136
 - surface tension 525
 - thermal data 140
 - TPRINT Table printing 726
- black oil 127
- BLACK OIL DATA keyword 717
- BLACKOIL keyword 717
- Boundary conditions 41, 118
- Bracketing 215
- Branch 37
- BRANCH keyword 762
- Brauner and Ullman correlation 555
- Brill and Minami 377
- Brinkman correlation 555
- Bubble point
 - calibration data 137
 - correction 414
 - solution gas-oil ratio 508
- C**
- CALIBRATE keyword 727
- Calibration

bubble point calibration data	137	COMPLETION keyword	651
solution gas-oil ratio	512	Completion options	68
CASE/ENDCASE keyword	748	Compositional	
CASE keyword	608	flashing	164
Casing		fluid: AQUEOUS keyword	729
standard sizes	562	fluid: LIBRARY keyword	736
Casing head pressure	204	fluid: MODEL keyword	736
CEMULSION keyword	729	fluid modeling	141
Character Input	602	fluid specification	732
Check model	195	importing composition	149
Chew and Connally		petroleum fraction specification	737
live oil viscosity	518	property models	145
Choke		selecting components	142
Achong critical flow correlation	450	specifying composition	148
API 14-B subcritical flow correlation	446	template	141
Ashford and Pierce critical flow correlation	450	viscosity models for compositional fluids	550
Ashford and Pierce critical pressure ratio	449	COMPOSITIONAL DATA keyword	729, 738
Ashford and Pierce subcritical flow correlation	446	compositional fluid	127
Baxendall critical flow correlation	450	COMPOSITION keyword	732
critical flow correlations	450	Compressor	
critical pressure ratio	449	curve	184, 185
Gilbert critical flow correlation	450	data entry	242
Mechanistic critical flow correlation	450	entering details	91
Mechanistic subcritical flow correlation	446	reciprocating	185
Omania critical flow correlation	450	theory	455
Pilehvari critical flow correlation	450	user curves	186
Poetmann-Beck critical flow correlation	450	COMPRESSOR keyword	679
Ros critical flow correlation	450	condensate pipeline	
Sachdeva critical flow correlation	450	tutorial	269
subcritical flow correlations	446	Conduction shape factor	499
theory	443	CONETAB keyword	663
user-defined critical flow correlation	451	CONFIG keyword	715
CHOKE keyword	673	Coning	442
Clear Dictionary	261	Coning Relationship Tabulation	663
Coating	710	CONTAMINANTS keyword	728
COAT keyword	710	Convective heat transfer	712
Coiled tubing	87	Conversion factors	581
Comment Statements	603	Copy	257
COMPCRV keyword	677, 703	Correlation	

black oil	134	Design Parameters (Gas lift)	223
comparison	122	Detailed tubing model	79
matching	197, 198	de Waard	390
options	717	Distributed Productivity Index Method	441
selecting a flow correlation	120	Drainage radius	574
Corrosion	176	Dry gas data entry	129
CORROSION keyword	639	Duns and Ros	377
corrosion model)		Dynamic skin	416
de Waard's equation	390		
CPA	536, 539		
CPFLUID keyword	726		
Critical unloading gas rate	201	E	
CSMA	539	E300 Flash	536
Cubic Equations of State	544	Editing	636
Cunliffe's Method	393	Elbows	575
CVFMD keyword	680, 705	Electrical Submersible Pumps — see ESP	235, 458
D		Elsharkawy and Alikhan	
Damaged zone	57	dead oil viscosity	517
Darcy		live oil viscosity	518
data required	61	undersaturated oil viscosity	521
equation	403	Emulsion viscosities	146
Data files	413, 603	END keyword	634
Data matching		Energy equation	485
procedure	198	Engine	
tutorial	344	options	172
DBR flash	536	preferences	178
Dead oil viscosity	516	Engine Keyword Tool	94
Deepest Injection Point	215	Equations of State	
De Ghetto		components of cubic	544
correlations	506	components of non-cubic	549
dead oil viscosity	516	Equations of State — Cubic	536
live oil viscosity	519	Equations of State — non-cubic	539
solution gas-oil ratio	508	Equipment	99
undersaturated oil viscosity	521	EQUIPMENT keyword	681
Delimiters	600	Erosion	176
Design	227	EROSION keyword	639
Design Control (Gas lift)	218	ESP	
Design method	218	curve	184
		data entry	187, 242
		design	241

overview	235, 458	horizontal multiphase	372
selection/design	189, 240, 464	matching	198
system components: pumps	236, 460	single phase	385
user curves	186		
EXCEL	264	Flow correlations	
EXECUTE keyword	637	valid horizontal combinations	646
Expander		valid vertical combinations	642
curve	185	vertical multiphase	377
equations	468	vertical options	641
EXPANDER keyword	683	Flowing Conditions	166
parameters	94	Flowline	95
Expert mode	264, 266, 595, 770	Flowrate	120
Explicit Subcodes	744	Flowrate Operator details	112
External PVT tables		Flow regimes	370
see Fluid Property Tables	160	Fluid	
F		black oil calibration	136
Facilities	91	black oil options	136
Fetkovich	59, 401	calibration	134
FETKOVICH keyword	654	calibration to match lab data	136
File conversion	32	compositional fluid modeling	141
Find	29	fluid models dialog	181
First law of thermodynamics	485	modeling	134
Fittings		property data	719
equations	452	property table files	554
FITTING keyword	684	specific heat capacity data	726
Flash		typical values of properties	571
flash/separation	151	viscosity models for compositional	550
Flashing		fluid files with same template of components	134
compositional	164	FLUID keyword	715
in-line	165	Fluid model calibration	138
Flow	648	Fluid Property Tables	160
Flow control valves		FMPUMP keyword	686
mechanistic theory	451	Forchheimer equation	64, 402
purpose and how to add	194	FORCHHEIMER keyword	670
FLOW CORRELATION DATA keyword	638	FRACTURE keyword	670
Flow correlations		Framo	
comparison/matching	197	FRAMO2009 keyword	686
defining	121	multiphase booster 1999	108
		multiphase booster 2009	105
		Friction	121

Friction and Holdup factors	
Moody and AGA Friction factors	385
Moody Friction factor	386
Friction pressure drop	
Cullender and Smith	388
Hazen-Williams	389
Panhandle A	388
Panhandle B	388
Weymouth	388
G	
Gas compressibility	522
Gas condensate reservoirs	398
Gas condensate systems	129
Gas lift	
Annular Gas Pressure Design method	234
Deepest Injection Point operation	215
design	217
design parameters	223
design procedure	218
Design Summary screen	228
diagnostics	231
Diagnostics operation	230
Diagnostics operation data	231
injection valve	691
instability	232
in the riser	114
Lift Gas Response Curves	216
minimum data set	227
multiple injection ports in wells	687
overview	214
properties	83
rate vs Casing head pressure	204
safety factors	224
setting injection points	84
theory	232
valve list	193
valves	84
Gas lift	
bracketing	215
GASLIFT keyword	687
Gas phase contaminants	140, 728
Gas reservoirs	398
Gas viscosity	525
gas well performance analysis	
tutorial	304
Gas wells	438
General Data	606
General Purpose Subcodes	747
Generic Pump	185
GERG	539
Glasø	
correlations	506
dead oil viscosity	516
solution gas-oil ratio	510
Glossary of symbols	576
GLR Matching	152
Govier and Aziz	377
Graphical reports	256
Gravel pack	57
Gray	377
H	
Hagedorn and Brown	377
HCORR keyword	645
HEADER keyword	607
Heat Balance	707
Heat configuration data	715
HEATER keyword	687
Heat Exchanger	98
HEAT keyword	707
Heat transfer	
between flowline and ground	499
between well and rock	503
coefficient	486
conductive coefficient calculations	495
convective coefficient calculations	497
IFC correlations	489
input or calculate U	100

options	171
pipe coat thermal conductivity	712
wellbore	81
HEAT TRANSFER DATA keyword	707
Holdup factor	121
Holdup factors	385
Horizontal completion	
data entry	53
determining length	52
Inflow Performance options	664
IPR model	53
IPRs	427
length optimization	202
optimal length	202
overview	51
properties options	56
Horizontal flow	
correlation abbreviations	646
regimes	371
Horizontal flow	
valid correlations	646
Horizontal flow	
correlation options	645
Horizontal gas wells	438
Horizontal Multiphase Flow Correlations	372
HORWELL keyword	664
Hossain	
dead oil viscosity	517
live oil viscosity	518
undersaturated oil viscosity	521
Hot Keys	22
HVOGEL keyword	670
Hydrate	
inhibitors	552
model details	553
overview	552
sub-cooling calculation	172
Hydraulic fracture	670
I	
Ice prediction	553
IFPCRV keyword	660
IFPPSSE keyword	655
IFPTAB keyword	662
Inflow Performance Curve	660
Inflow Performance Relationships (IPRs)	
Back Pressure equation	402
Back pressure equation data	60
Bubble point correction	414
data file	413
Fetkovich's equation	401
Fetkovich's equation data	59
Forchheimer equation	402
Forchheimer equation data	64
horizontal completions	427
Jones' equation	401
Jones equation data	59
multilayer reservoir	74
Multi-rate data	64
oil/water relative permeability	441
Productivity Index (PI)	399
Pseudo steady state/Darcy	403
Pseudo steady state/Darcy data	61
solution gas-drive reservoirs	438
summary for vertical completions	398
Transient IPR	408
Transient IPR data	66
vertical completion	57
Vogel's equation	400
Vogel's equation data	58
Inflow Performance Tabulation	662
Inflow Production Profiles	431
Injection performance	89
Injection Point	101
Injection well	39, 89
INJFLUID keyword	693
INJGAS keyword	693
INJPORT keyword	691

INLET keyword	623	Kouzel undersaturated oil viscosity	520
Inlet pressure	119	Kreith	489
In-line flashing	165	L	
Input data	599	Laplace equation	500
Input files	599, 604	Lasater correlations	506
Inside Film Coefficient method	172	solution gas-oil ratio	511
Inside Fluid Film Heat Transfer Coefficient	489	LAYER keyword	666
Inside forced convection	470, 489	LBC	550
IPRCRV keyword	660	Lee method	525
IPR keyword	654	Leviton and Leighton correlation	558
IPR — see Inflow Performance		Library components — cubic EoS	544
Relationships (IPRs)	438	Library components — non-cubic EoS	549
Iteration Data	621	LIBRARY keyword	736
J		Lift Gas Response Curves	216
JOB keyword	607	Limits	561
Jones data	655	Liquid by sphere	394
Jones equation	59	Liquid-gas surface tension	560
Jones Equation	401	Liquid holdup	394
JONES keyword	655	Liquid loading displaying point	201
Joshi	53	equations	396
JUNCTION keyword	769	implementation	90
K		inclination angle	90
Kartoatmodjo and Schmidt correlations	506	Liquid properties	555
dead oil viscosity	516	Liquid viscosity and oil/water emulsions	555
live oil viscosity	518	correlations	555
Oil Formation Volume Factor	514	data	721
solution gas-oil ratio	510	Live oil viscosity	518
undersaturated oil viscosity	520	Load Dictionary	261
KCOAT keyword	712	Local Fluid Data	133
Keyword abbreviations	601	looped gas gathering network case study	324
Keywords	759	LVIS keyword	721
KFLUID keyword	715		
Khan live oil viscosity	519		
undersaturated oil viscosity	520		

M	
Measured data	197, 198
Mechanical skin	415
MFL File	127
Minimum data set (Gas lift)	227
Mixing (Black oil)	528
Model building	33
modeling	161
Modeling systems	128
Modeling wells	89
MODEL keyword	736
MOODYCALC keyword	649
MPBOOSTER keyword	695
Mukherjee and Brill	377
MULTICASE keyword	743, 748
Multiflash	536, 539
multiflash models	154
Multilayer reservoir	74
Multiphase	377
Multiphase booster	
efficiencies tables	484
Framo 1999	108
Framo 2009	105
generic	102
twin screw boosters	103
Multiphase pump	
MPUMP keyword	696
vendor performance curve format	110
Multiple case	749
Multiple completion well	90
Multiple injection ports	687
multiple MFL files	154
Multiple Value Data Sets	603
Multi-rate	64
Multi-stage separation	151
N	
NAPLOT keyword	739
NAPOINT keyword	743
Network	
constraints	42
estimates	42
iterations	180
separator	38
Network models	35
Network Option Control	173
Network Report tool	262
Network source	35
NIST recommendation for pure fluid / mixture	539
Nodal Analysis	
NAPLOT keyword	739
NAPOINT keyword	743
performing	200
point	88
NODE keyword	698
Non-cubic Equations of State	549
NOPRINT keyword	634
NOSLIP	377
NSEPARATOR keyword	769
Numeric data	602
O	
Oil/water emulsions	555
Oil/water relative permeability	441
Oil Formation Volume Factor (OFVF)	
defined	512
saturated systems	513
undersaturated systems	514
Oil reservoirs	398
Oil viscosity	
black oil	515
undersaturated	520
oil well performance analysis	
tutorial	286
OLGAS	377
Open Link	776
Operations	738

OPTIMIZE keyword	752	Pipe coating	710, 711
optimizing PIPESIM network simulation performance tutorial	43	Pipe dimensions	699
Options	182	pipe inline heating tutorial	
OPTIONS keyword	609	PIPE keyword	699
Orkiszewski	377	Pipeline	
Outlet pressure	119	burial	716
Output options	173	burial method	171
		operations	91
		pipeline and facilities model	91
		selecting size	87
		standard sizes	567
		tools	91
P		PLOTFILEDATA keyword	637
Parallelized Network		PLOT keyword	631
solver	179	Plotting options	631
Parameters	606	Pressure	119
Paths used by tools	170	Pressure/Temperature profile	196
PCP		Pressure drop	
data entry	192	effect on productivity	427
standard curves	191	for gas	388
PCP curve	190	multiphase	431
Pedersen	550	single phase	430
Peng-Robinson	536	Principle of conservation of energy	485
Perforation	57	Print	257
PERMCRV keyword	668	PRINT keyword	624
Permeability	574	Production well	38
PERMTAB keyword	669	Progressive cavity pump — see PCP	465
PETROFRAC keyword	737	Project Data	170
Petroleum fractions		Property calibration	727
for Cubic Equations of State	549	PROP keyword	719
setting up	143	Pseudo steady state	
Petrosky and Farshad		data for equation	655
correlations	506	equation and data	61
dead oil viscosity	517	IPR	403
live oil viscosity	518	productivity	435
solution gas-oil ratio	511	PS-PLOT keyword	749
undersaturated oil viscosity	521	PsPlot utility	256
Phase Behavior package	142	Pump	
Phase envelope	149		
PI			
data required	58		
equations	399		

curve	184
data entry	242
entering parameters	117
user curves	186
PUMPCRV keyword	677, 703
PUMP keyword	701
PUSH keyword	636
PVT File	127
Q	
Quality lines	150
R	
Ramey model	503
RATE keyword	619
Reciprocating compressor	
adding to database or model	93
entering details	93
overview	458
Red box (missing data)	34
References	582
REFPROP	539
REINJECTOR keyword	704
Relative permeability	441
Remedial coiled tubing	87
Remote action editing	636
Report	262
Reports	256
Report Tool	115, 261
Reservoir	
creating look-up tables	203
layer properties	666
simulation data	750
tables	202
RESERVOIR keyword	668
Restart Model	212
Richardson correlation	555
Riser Details	112
S	
Rod pump	
module	242
RODPUMP keyword	680, 705
Roughness	572
Run Model	195
Running multiple models	266

perforated well	420	SYSTEM DATA keyword	672
Pseudo Steady State equation	63	System Profile	606
Transient IPR data	67		
Slug	641	T	
Slug calculation	640	TABLE keyword	750
Slug catcher	641	TCOAT keyword	711
SLUG keyword	640	Tees	576
Soave-Redlich-Kwong	536	Temperature profile	196
solids appearance	162	Terminate Run	195
solids modeling		Thermal conductivities	572
sand modeling	161	Thermal conductivity	715
scale prediction	163	Thermal data (black oil)	140
solids appearance	162	Third party applications	777
wax modeling	161	TIME	754
Solution gas-oil ratio	508	Tolerance equations	561
Source	118	Toolbox	25
SOURCE keyword	764	TPRINT keyword	726, 738
SPHASE keyword	648	Transient IPR	
Standard curves	191	data	66
Standing		equations	408
correlations	506	TRANSIENT keyword	671
Oil Formation Volume Factor	513	Tubing	
solution gas-oil ratio	511	coiled tubing/velocity string	87
Statements	600	detailed profile model	79
Steady State Heat Transfer	486	selecting size	87
Steady-State productivity	432	standard sizes	562
Steam	166	summary table	82
Stock tank conditions	166	tool	76
Stock tank properties	136	Turner's equation	396
Suggested flow correlations	122, 383	tutorials	268
Summary tubing table	82	Twin screw boosters	103
Surface tension		Twin screw pumps	110
black oil	525	Typical values	571
liquid gas	560		
water-gas	526	U	
Swap angle	390	Undersaturated oil viscosity	520
Symbols glossary	576	Units	
System analysis	196	description string	602

selecting a set	170	VOGEL keyword	654
UNITS keyword	608	Vogel IPR	403
User curves	186	Volume ratio	555
userdll.dat	125		
USERDLL keyword	638, 650		
User DLL tutorial	334	Water-gas surface tension	526
User variable	211	Water systems	128
U Value Multiplier	172	Wax	
		BP method and deposition limits	208
		deposition	205
		overview	554
		Shell method and deposition limits	206
		WAX keyword	754
		wax modeling	161
		WCOPTION keyword	657
		Well	
		completion data	657
		horizontal completion	51
		optimization	213
		performance curves	203
Valve	193	Wellbore	
database	193	heat transfer between well and rock	503
FITTING keyword	684	heat transfer in	81
typical values	574	Well Curves	40
Vand correlations	555	WELLHEAD keyword	706
Vasquez and Beggs	506	Well Performance analysis	50
correlations	506	WELL PERFORMANCE MODELING keyword	650
Oil Formation Volume Factor	513	WELLPI keyword	653
solution gas-oil ratio	511	Well Productivity	653
undersaturated oil viscosity	520	Wizard	28
VCORR keyword	641	Woelflin correlation	555
Vertical completion	57	WPCURVE keyword	654
data required	57		
IPRs	398		
Vertical flow	370		
correlation abbreviations	643		
regimes	370		
valid correlations	642		
VCORR keyword	641		
Vertical flow	377		
multiphase flow correlations	377		
View options	182		
Viscosity	516		
dead oil	516		
emulsion	146		
models for compositional fluids	550		
Vogel	400		
equation	400		
equation data	58		