
User's Guide

Builder

Pre-Processing Applications

Version 2012

By Computer Modelling Group Ltd.



This publication and the application described in it are furnished under license exclusively to the licensee, for internal use only, and are subject to a confidentiality agreement. They may be used only in accordance with the terms and conditions of that agreement.

All rights reserved. No part of this publication may be reproduced or transmitted in any form or by any means, electronic, mechanical, or otherwise, including photocopying, recording, or by any information storage/retrieval system, to any party other than the licensee, without the written permission of Computer Modelling Group.

The information in this publication is believed to be accurate in all respects. However, Computer Modelling Group makes no warranty as to accuracy or suitability, and does not assume responsibility for any consequences resulting from the use thereof. The information contained herein is subject to change without notice.

Copyright © 1987-2012 Computer Modelling Group Ltd.

All rights reserved.

The license management portion of this program is based on:

Reprise License Manager (RLM)

Copyright © 2006-2012, Reprise Software, Inc.

All rights reserved

Builder, CMG, and *Computer Modelling Group* are registered trademarks of Computer Modelling Group Ltd. All other trademarks are the property of their respective owners.

Computer Modelling Group Ltd.

Office #150, 3553 - 31 Street N.W.

Calgary, Alberta Canada T2L 2K7

Tel: (403) 531-1300

Fax: (403) 289-8502

E-mail: cmgl@cmgl.ca

Contents

| | |
|---|-----------|
| Introduction and Overview | 1 |
| Introduction..... | 1 |
| What's New in Builder 2012 | 2 |
| What's New in Builder 2011 | 3 |
| What's New in Builder 2010 | 4 |
| Starting Builder..... | 5 |
| Specifying "Except" Units in STARS | 6 |
| Converting Simulator Type for a Dataset | 6 |
| Importing Partial Simulator Data..... | 7 |
| Saving Your Work to a Simulator Input File (Dataset) | 8 |
| Saving Comments | 9 |
| Selecting the Main File | 9 |
| Choosing the Array Saving Method | 9 |
| Grid Display Selection..... | 10 |
| Organizing the Data | 10 |
| Organizing Include Files..... | 10 |
| Validating the Dataset..... | 12 |
| Viewing Model Initialization Generated by Simulator..... | 13 |
| Simulating Current Dataset from Within Builder..... | 14 |
| Simulating Current Dataset to a Job Queue from Within Builder | 14 |
| Customizing Toolbars, Menus and Theme | 15 |
| Docking Panes | 15 |
| Exiting Builder..... | 15 |
| Reservoir Description | 17 |
| Overview..... | 17 |
| Creating a Simulation Grid Using Structure Maps | 17 |
| Top Maps - Elevation or Depth Map | 19 |
| Units for the X and Y Coordinates of Maps | 20 |
| Opening and Displaying a Contour Map File or Mesh Map File | 20 |
| Creating a Variable Depth or Thickness Cartesian Grid or Orthogonal Corner Point Grid..... | 21 |
| Creating a Radial Grid | 22 |
| Moving and Rotating a Grid | 23 |
| Moving Grid Lines..... | 24 |
| Adding and Deleting Grid Lines..... | 24 |

| | |
|---|----|
| Undoing a Change..... | 25 |
| Creating a Non-Orthogonal Corner Point Grid with Vertical Faults..... | 25 |
| Zooming and Panning While Creating or Editing a Grid..... | 26 |
| Creating a Grid for a Pattern Flood Model..... | 26 |
| Creating a Non-orthogonal Grid with Sloping Faults Using Structure Maps | 28 |
| Contour Maps and Sloping Faults | 28 |
| Steps for Constructing Sloping Fault Grids with Builder | 29 |
| Step By Step Instructions | 30 |
| Useful Tips | 33 |
| Importing 3D Simulation Grids and Grid Properties..... | 33 |
| Importing from File Output by a Geological Program..... | 33 |
| Importing Simulation Grid, Properties and Well Locations from RESCUE Model | 34 |
| Importing Partial Datasets from <i>Petrel</i> | 37 |
| Resolving Problems with Axis Direction in Imported Grids | 38 |
| Importing Grid and Grid Properties from a CMG Dataset File..... | 40 |
| Creating a Simulation Grid Using 3D Surfaces from RESCUE Model | 41 |
| Building Simulation Grid Using the Geological Modelling Task Manager..... | 43 |
| Import Well Data..... | 45 |
| Select Tops | 46 |
| Edit Markers | 46 |
| Create Horizons..... | 48 |
| Create 2D Grid | 50 |
| Create Boundary..... | 51 |
| Create 3D Grid | 53 |
| Create Properties | 54 |
| Editing an Existing Simulation Grid | 56 |
| Property Modifications (MODs) | 57 |
| Adding Refined Grids Using the Refinement Wizard..... | 59 |
| Adding Refined Grids around a Well Using the Refinement Wizard | 64 |
| Splitting Grid Planes | 66 |
| Join Blocks Option | 67 |
| Combining Layers | 70 |
| Extracting Sub-sections of Grids..... | 82 |
| Editing Cartesian, Radial or Orthogonal Grids | 83 |
| Editing Non-orthogonal Corner Point Grids Created Using Structural Maps | 83 |
| Editing Grids Created Using RESCUE Surfaces | 84 |
| Converting an Existing Simulation Grid to a Node Based Format | 84 |
| Specifying, Calculating and Adjusting Structure and Rock Property Values | 85 |
| Specifying Property Interpolations and Calculations | 85 |
| Performing Interpolations and Calculations | 90 |
| Deleting a Property..... | 92 |
| Defining Custom Property..... | 92 |

| | |
|---|-----|
| Importing Spatial Properties | 93 |
| Importing Properties from CMG Dataset Files without Importing the Grid..... | 93 |
| Importing Properties from CMG Simulation Results Files | 94 |
| Importing Properties from a Text File Containing an Array of Values | 95 |
| Defining/Editing Geological Units | 95 |
| Converting to Dual Porosity | 97 |
| Pinching Out Thin Grid Blocks | 97 |
| Fixing Overlapping Blocks | 98 |
| Fixing Net-to-gross Ratio Problems | 100 |
| Setting Transmissibility Multipliers at Sealing or Partially Sealing Faults | 101 |
| Entering Rock Compressibility..... | 103 |
| Editing Property Values of Selected Blocks | 103 |
| Editing Property Values around Wells | 105 |
| Reviewing Property Modifications | 106 |
| Editing Grid Structure Values..... | 106 |
| Defining and Editing Sectors..... | 116 |
| Create/Edit/Delete Lease Planes | 117 |
| Create/Edit/Delete Aquifers..... | 118 |
| Locating and Adding Wells in the Simulation Grid | 120 |
| Wells in Map Data | 121 |
| Opening a 3D Well Trajectory File and Adding Well Completions..... | 121 |
| Manually Adding or Modifying Well Completions..... | 121 |
| Wellbore Diagram..... | 122 |
| Rock Compaction/Dilation Regions (IMEX and GEM Only)..... | 123 |
| Selecting Compaction/Dilation Regions..... | 123 |
| Adding a New Compaction/Dilation Region..... | 123 |
| Removing Compaction/Dilation Regions | 124 |
| Compaction/Dilation Tables | 124 |
| Adding a New Compaction/Dilation Table | 124 |
| Removing Compaction/Dilation Tables | 125 |
| Setting Thermal Properties for the Rock Type (GEM Only)..... | 125 |
| Other Reservoir Properties (STARS Only)..... | 126 |
| General Editing of a Rock Type | 127 |
| Adding a New Rock Type | 127 |
| Removing a Rock Type | 127 |
| Entering Rock Compressibility..... | 128 |
| Setting Thermal Properties for the Rock Type | 129 |
| Entering Overburden Heat Loss Parameters..... | 131 |
| Specifying Dilation Recompaction..... | 132 |
| Specifying Compaction Rebounding | 134 |
| Specifying Variable Permeability | 135 |
| Reservoir Options (IMEX and GEM Only)..... | 136 |

| | |
|---|------------|
| Curvature Analysis | 136 |
| Flux Sector (IMEX Only)..... | 138 |
| Miscellaneous Grid Calculations..... | 139 |
| Fluid Model - IMEX | 143 |
| Overview | 143 |
| Viewing and Editing Data | 143 |
| Creating a New Fluid Model | 144 |
| Creating Quick Models | 144 |
| Using the Detailed Fluid Model Dialog Box..... | 145 |
| Importing a WinProp Generated Model..... | 145 |
| Changing an Existing Model..... | 146 |
| Working with Multiple PVT Regions | 146 |
| Creating/Editing PVT Tables | 147 |
| Creating a New Table..... | 147 |
| Editing a Table | 147 |
| Generating a PVT Table Using Correlations..... | 148 |
| Entering Other PVT Region Properties..... | 149 |
| Generating Water Properties Using Correlations | 150 |
| Entering Undersaturated Data | 150 |
| Entering Data for the Pseudo-miscible Option..... | 151 |
| Entering Data for the Polymer Option..... | 152 |
| Entering Data for the API Tracking Option | 152 |
| Launching WinProp with Current Fluid Model Data..... | 153 |
| Displaying Component Properties Partial Dataset | 154 |
| Data Validation..... | 154 |
| Fluid Model - GEM | 155 |
| Overview | 155 |
| Library and User Components | 155 |
| Specifying a Fluid Model..... | 156 |
| Adding/Editing an Equation of State..... | 156 |
| Adding/Editing Components | 157 |
| Adding/Editing a Component..... | 158 |
| Deleting a Component..... | 159 |
| Adding/Editing Comments..... | 159 |
| Entering Binary Interaction Coefficients..... | 160 |
| Adding/Editing Comments..... | 160 |
| Entering Viscosity Data..... | 161 |
| Adding/Editing Comments..... | 161 |
| Entering Aqueous Phase Solubility | 162 |
| Entering Single Phase Fluid and Gas Diffusivity..... | 163 |

| | |
|---|------------|
| Entering Reservoir Temperature..... | 164 |
| Entering Water Properties..... | 164 |
| Displaying Component Properties Partial Dataset..... | 165 |
| Data Validation..... | 165 |
| “Quick” Coal Bed Methane (CBM) Setup..... | 165 |
| New Case | 166 |
| Existing Case | 171 |
| Fluid Model - STARS | 173 |
| Overview..... | 173 |
| Adding/Editing Components | 173 |
| Adding/Editing a Component | 174 |
| Entering K Value Data..... | 175 |
| K value Correlations | 176 |
| K Value Tables | 177 |
| Entering Liquid Density Data | 180 |
| Liquid Density Nonlinear Mixing Option..... | 181 |
| Entering Solid Phase Density Data..... | 182 |
| Entering Liquid Phase Viscosity Data | 182 |
| Viscosity Correlations..... | 183 |
| Viscosity Tables..... | 184 |
| Liquid Viscosity Nonlinear Mixing Option..... | 184 |
| Entering Gas Phase Viscosity Data..... | 185 |
| Working With Multiple Viscosity Property Types | 186 |
| Entering Fluid Enthalpy Data | 186 |
| Entering Other Properties | 187 |
| Working with Reactions | 189 |
| Advanced Options..... | 190 |
| Launching WinProp with Current Fluid Model Data | 191 |
| STARS Import Black Oil PVT Wizard | 192 |
| Introduction..... | 192 |
| PVT Wizard vs. Winprop Fluid Model Generation..... | 192 |
| Theoretical Background..... | 192 |
| Using the STARS Import Black Oil PVT Wizard | 194 |
| Displaying Component Properties Partial Dataset..... | 208 |
| Data Validation | 208 |
| Rock Fluid - General | 209 |
| Overview..... | 209 |
| Rock Fluid Options | 209 |
| Rock Fluid Options: IMEX..... | 209 |
| Rock Fluid Options: GEM..... | 210 |
| Rock Fluid Options: STARS | 211 |

| | |
|--|------------|
| Rock Fluid – Diagnostic Plots..... | 211 |
| Show Blocks ← Minimums | 213 |
| Rock Fluid Properties - IMEX | 215 |
| Overview | 215 |
| Modifying Existing Rock Types | 216 |
| Adding a New Rock Type | 216 |
| Deleting an Existing Rock Type..... | 216 |
| Averaging Laboratory Data into an Average Rock Type..... | 217 |
| Setting Rock Type Properties..... | 218 |
| Defining Relative Permeability Tables | 219 |
| Tool: Generate Tables Using Correlations..... | 220 |
| Tool: Smooth Table..... | 222 |
| Tool: Curve Scaling..... | 223 |
| Modifying Tables from the Plot View..... | 224 |
| Setting Hysteresis Modelling | 225 |
| Seawater Scale Deposit and Damage Tables..... | 225 |
| Rock Fluid Properties - GEM | 227 |
| Overview | 227 |
| Modifying Existing Rock Types | 228 |
| Adding a New Rock Type | 228 |
| Deleting an Existing Rock Type..... | 228 |
| Averaging Laboratory Data into an Average Rock Type..... | 229 |
| Setting Rock Type Properties..... | 231 |
| Defining Relative Permeability Tables | 231 |
| Tool: Generate Tables Using Correlations..... | 233 |
| Tool: Smooth Table..... | 234 |
| Tool: Curve Scaling..... | 235 |
| Modifying Tables from the Plot View..... | 236 |
| Setting Hysteresis Modelling | 237 |
| Rock Fluid Properties - STARS | 239 |
| Overview | 239 |
| Modifying Existing Rock Types | 240 |
| Adding a New Rock Type | 240 |
| Deleting an Existing Rock Type..... | 240 |
| Averaging Laboratory Data into an Average Rock Type..... | 241 |
| Setting Rock Type Properties..... | 243 |
| Defining Relative Permeability Tables | 244 |
| Tool: Generate Tables Using Correlations..... | 246 |
| Tool: Smooth Table..... | 247 |

| | |
|---|------------|
| Tool: Curve Scaling | 248 |
| Modifying Tables from the Plot View | 249 |
| Setting Hysteresis Modelling | 250 |
| Setting Relative Permeability End Points | 251 |
| Setting Interpolation Set Parameters..... | 252 |
| Working with Interpolation Sets..... | 253 |
| Modifying an Interpolation Set..... | 254 |
| Adding an Interpolation Set..... | 255 |
| Deleting an Interpolation Set | 255 |
| Initialization - IMEX | 257 |
| Overview..... | 257 |
| Tree View Items and Menu..... | 258 |
| Standard Interface | 258 |
| Advanced Interface | 259 |
| Calculation Methods tab | 260 |
| PVT Region Parameters tab..... | 261 |
| Advanced Parameters tab..... | 263 |
| Data Validation | 263 |
| Initialization – GEM | 265 |
| Overview..... | 265 |
| Tree View Items and Menu..... | 265 |
| Entering and Editing Initial Conditions | 266 |
| Calculation Methods | 266 |
| Init. Region Parameters..... | 267 |
| Advanced Parameters | 269 |
| Separators..... | 270 |
| Gas Plant Tables | 271 |
| Specification of Non-EOS Stream Density Calculations..... | 272 |
| Data Validation | 272 |
| Initialization – STARS | 273 |
| Overview..... | 273 |
| Tree View Items and Menu..... | 273 |
| Entering and Editing Initial Conditions | 274 |
| Data Validation | 275 |
| Well and Group Control | 277 |
| Overview..... | 277 |
| Time-line View of Recurrent Data | 278 |

| | |
|---|-----|
| Date/Time Information..... | 279 |
| Adding/Removing Dates | 279 |
| Adding Dates..... | 280 |
| Deleting Dates | 281 |
| Well Data..... | 281 |
| Adding New Wells..... | 281 |
| Deleting Wells..... | 282 |
| Well Events | 283 |
| Adding Well Constraints | 284 |
| New Well Constraint Definition..... | 285 |
| List of Well Constraints | 285 |
| Seawater Model (IMEX Only)..... | 286 |
| Copying Well Events..... | 287 |
| Deleting Well Event | 287 |
| Changing Events Settings..... | 288 |
| Using the Well and Date Filter..... | 288 |
| Well Event Comments..... | 290 |
| Well Completions (PERF)..... | 291 |
| Calculating the Horizontal Well Skin Factor | 292 |
| Adding New Well Completions | 293 |
| Joining Well Completions (Multilaterals) | 295 |
| Additional Data Columns..... | 295 |
| Copying Data Values to Other Well Completions | 296 |
| Relative Permeability Options..... | 297 |
| Well Trajectories | 297 |
| Well Trajectory Import..... | 298 |
| Create Trajectories from Well Completions..... | 300 |
| Trajectory Perforation Intervals | 301 |
| Trajectory Properties | 303 |
| Trajectory Smoothing..... | 306 |
| Background | 307 |
| Using Trajectory Smoothing | 310 |
| Trajectory Smoothing Example..... | 316 |
| Well Logs Import | 318 |
| Wellbore Diagram | 319 |
| Group Settings..... | 321 |
| Adding New Groups..... | 321 |
| Deleting Groups | 322 |
| Group Events..... | 323 |
| Adding Wells to Group | 324 |
| Group Apportionment Methods | 332 |
| Group ON-TIME..... | 334 |
| Cycling Group | 334 |
| Copying Group Events | 336 |

| | |
|---|-----|
| Deleting Group Events..... | 336 |
| Using the Group and Date Filter..... | 337 |
| Group Event Comments..... | 340 |
| Group and Well Connection Display..... | 340 |
| Reporting Groups..... | 344 |
| Gas Lift Optimization Options..... | 346 |
| Adding New Gas Lift Optimization Options | 346 |
| Tubing Performance Tables..... | 347 |
| Adding a New Tubing Table | 348 |
| Using Tubing Files..... | 349 |
| Using the Tubing Pressure Calculator | 350 |
| Fixing Well Definition Dates..... | 352 |
| Validating Well Completions | 352 |
| Production Data Wizard..... | 353 |
| Overview..... | 353 |
| Files Required by Production Data Wizard | 353 |
| Using the Production Data Wizard | 354 |
| Switch Well Constraints Based on Production Data..... | 359 |
| Average Production/Injection Data..... | 360 |
| Trigger | 361 |
| Multiple Trigger Conditions | 364 |
| Well List Shut/Open | 368 |
| Heater Well (STARS Only)..... | 369 |
| Copy Well Wizard | 370 |
| Creating Pattern Based Wells | 373 |
| Hydraulically Fractured Wells (IMEX & GEM) | 376 |
| Working with Hydraulic Fractures | 379 |
| Specifying Non-Darcy Information for Hydraulic Fractures..... | 382 |
| Creating a Stimulated Reservoir Volume (SRV) Stage | 383 |
| Microseismic Events..... | 389 |
| Importing Microseismic Data | 389 |
| Exporting Microseismic Data | 394 |
| Deleting Microseismic Data | 395 |
| Viewing the Original File | 395 |
| Display Properties..... | 395 |
| Microseismic Event Data..... | 398 |
| Animation | 399 |
| Create Grid..... | 399 |
| FlexWells (STARS)..... | 403 |
| About FlexWells | 403 |
| Managing the Annulus..... | 408 |
| Managing Tubing Strings | 416 |
| Managing Concentric Tubing Strings..... | 422 |
| Managing Instrument Tubing Strings | 425 |

| | |
|---|------------|
| FlexWells Validation Errors..... | 427 |
| Modifying Well Perforations with the FlexWells Dialog Box..... | 430 |
| Modifying Well Names, Definition Dates, and Completion Dates..... | 431 |
| Using Sink/Source Wellbore Models..... | 432 |
| Managing FlexWell and Well Events..... | 432 |
| Managing FlexWell Reporting Groups | 451 |
| Input/Output Control | 453 |
| Overview | 453 |
| Titles and Case ID..... | 453 |
| Run Time Dimensioning..... | 454 |
| Restart..... | 455 |
| Simulation Results Output | 457 |
| Text Output..... | 461 |
| Miscellaneous Options | 462 |
| Numerical Control | 465 |
| Overview | 465 |
| Setting and Editing Numerical Controls (IMEX & GEM)..... | 465 |
| Setting and Editing Numerical Controls (STARS) | 466 |
| Dynamic Grid Regions..... | 469 |
| PARASOL Class Partitioning Pattern (PPATTERN) | 470 |
| Geomechanics – GEM and STARS | 471 |
| Overview | 471 |
| Data Validation..... | 472 |
| Calculation and Numerical Options | 472 |
| Geomechanical Rock Types..... | 474 |
| Geomechanical Domain | 475 |
| Rigid NULL Blocks and Rigid Cap Rock..... | 477 |
| Boundary Stress Unloading..... | 478 |
| Displacement Boundary Conditions..... | 479 |
| Changing Display Content and Settings | 485 |
| Overview | 485 |
| Understanding the Parts of the Main Window | 486 |
| Displaying the X,Y Coordinates of a Point in the Reservoir..... | 487 |
| Displaying Crosshairs Centered on the Cursor | 487 |
| Displaying the IJK Grid Block Address..... | 488 |
| Viewing Properties | 488 |
| Selecting a Property..... | 488 |
| Selecting the View Type and Plane Number..... | 488 |

| | |
|---|-----|
| Selecting Plane Number within a Refined Grid..... | 489 |
| Selecting Block Fill, Contour Lines or Contour Fill..... | 489 |
| Selecting a Time Display Format | 490 |
| Selecting a Time | 490 |
| Changing Builder Properties..... | 490 |
| Builder Properties Dialog Box..... | 490 |
| Selecting a Font Name, Style, Size and Color | 491 |
| Selecting a Line Style, Thickness and Color | 492 |
| Changing Displayed Titles..... | 493 |
| Toggling the Display of Grid, Map and Wells | 493 |
| Showing Fault Faces | 493 |
| Results 3D and Builder User Preferences..... | 494 |
| Moving and Resizing Display Objects | 496 |
| Moving the Title, Info Box, Color Legend, and Map Scale | 496 |
| Resizing the Titles and Info Box | 496 |
| Resizing the Color Legend | 496 |
| Resizing the Map Scale..... | 496 |
| Resizing the Reservoir Viewport..... | 496 |
| Toggling the Color Gradient Background | 497 |
| Changing the Aspect Ratio and Scale..... | 498 |
| Zooming, Panning, and Rotating the Reservoir..... | 499 |
| Zooming..... | 499 |
| Panning (Moving) the Reservoir..... | 499 |
| Undoing a Zoom or Pan..... | 499 |
| Rotating, Panning and Zooming the Reservoir in 3D..... | 500 |
| Improving 3D Graphics Performance | 500 |
| Using Stereoscopic 3D..... | 501 |
| Selecting IJK Slabs and Regions | 502 |
| Cutting Away Part of a Reservoir in 3D..... | 507 |
| Modifying the Color Scale..... | 509 |
| Modifying a Linear or Logarithmic Color Scale | 509 |
| Setting or Modifying a General Color Scale..... | 511 |
| Saving Color Scale Settings in Preferences..... | 512 |
| Viewing and Editing Tabular Data | 513 |
| Printing and Saving Images | 515 |
| Changing the Selected Printer and Paper Size | 515 |
| Printing the Current View..... | 516 |
| Specifying Bitmap Resolution in 3D Printing | 516 |
| Saving the Current View as an Image File | 516 |
| Improving the Quality of Saved Images | 517 |
| Working With Multiple Views and Documents | 517 |
| Opening a Second View | 517 |
| Synchronizing Views..... | 518 |
| Interactive Block Selection (3D Only) | 518 |

| | |
|---|------------|
| Docking Tools (3D Only)..... | 520 |
| Selection | 520 |
| 3D Properties..... | 520 |
| Wells..... | 521 |
| Statistics | 522 |
| Opening and Closing Docking Tools | 522 |
| Formula Manager | 523 |
| Overview | 523 |
| Formula Manager Dialog Box..... | 523 |
| General Rules and Guidelines | 525 |
| Terms..... | 525 |
| Operators | 525 |
| Formula Syntax | 527 |
| IF Blocks | 528 |
| Other..... | 528 |
| Using Formula Manager..... | 528 |
| To create a formula..... | 528 |
| To apply a formula to a property..... | 531 |
| To view a formula | 536 |
| To edit a formula..... | 536 |
| To view the new property..... | 536 |
| To copy a formula | 536 |
| To delete a formula..... | 537 |
| To delete a property..... | 537 |
| Saving and Restoring Formulae and Formula-based Properties..... | 537 |
| Calculating Grid and Property Statistics | 537 |
| Importing Geological and Well Trajectory Data | 539 |
| Overview | 539 |
| Required Geological Data | 539 |
| Importing Hard Copy Contour Maps | 540 |
| Supported Atlas Boundary File Format..... | 541 |
| WinDig Contour Format | 542 |
| Creating Quick Contour Maps Using Builder..... | 545 |
| CMG Mesh Format | 545 |
| Creating Mesh Data Files | 545 |
| Sample CMG Mesh Files | 552 |
| Importing Data from a History Match Run Done by a CMG Simulator | 553 |
| Supported ZMAP Formats | 558 |
| Supported EarthVision Formats | 560 |
| Supported CPS-3 Formats | 562 |
| Newer CPS-3 Files | 564 |

| | |
|---|------------|
| Production Analyst Well Trajectory File Format | 565 |
| XY File Description..... | 565 |
| Deviated File Description..... | 566 |
| Well Trajectory Table File Format | 567 |
| Well Perforation File Format | 569 |
| Table Format for Well Trajectory Perforation Intervals | 569 |
| Old Perforation Format..... | 570 |
| Well Trajectory and Logs in RMS Format | 571 |
| Well Trajectory in GOCAD Format | 571 |
| Well Log Formats | 573 |
| LAS Well Log Format | 573 |
| Single Well Log File..... | 575 |
| Multiple Well Log File | 576 |
| Formation Top File Format..... | 577 |
| Table Format for Formation Tops..... | 577 |
| Importing from a PPDM database | 578 |
| Overview..... | 578 |
| Prerequisites..... | 578 |
| Launching Database Import..... | 579 |
| PPDM Coverage in Database Import..... | 592 |
| Merging Data Sets | 595 |
| Creating Maps and Geostatistical Property Calculations | 601 |
| Overview..... | 601 |
| Points Tab | 602 |
| Faults Tab | 603 |
| Methods Tab | 604 |
| Inverse Distance Estimation | 605 |
| Ordinary Kriging (OK) Estimation..... | 605 |
| Ordinary Kriging with Secondary Variable..... | 606 |
| Trend Method | 607 |
| Gaussian Geostatistical Simulations..... | 608 |
| Number of Realizations | 609 |
| Use Uniform Grid Assumption for Speed Up..... | 609 |
| Gaussian Geostatistics and Secondary Variable | 609 |
| Unconditional Gaussian Geostatistical Simulation..... | 609 |
| Object Modeling | 610 |
| Mesh Tab and Region Tab..... | 617 |
| Mesh Creation..... | 617 |
| Region Selection..... | 617 |
| Extend By | 617 |
| Map Value Limits or Value Limits (3D) | 617 |
| Run and Apply Buttons..... | 617 |

| | |
|---|------------|
| Variogram Matching | 618 |
| Lack of Sufficient Pairs..... | 618 |
| Number of Pairs | 618 |
| Lag Tolerance..... | 619 |
| Instability..... | 621 |
| Trend | 622 |
| Search Radius | 622 |
| Normal Score Transform..... | 623 |
| Adjusting Variogram Parameters | 624 |
| Weighting Examples | 625 |
| Inverse Distance Method..... | 625 |
| Trend Estimation | 626 |
| Variogram Range (Kriging Method)..... | 627 |
| Variogram Anisotropy (Kriging Method) | 628 |
| Variogram Nugget (Gaussian Geostatistical Method) | 629 |
| Search Range (Gaussian Geostatistical Method) | 630 |
| Object Modelling – Inclined Shales | 630 |
| References | 630 |
| Property Calculation Scripting | 631 |
| Overview | 631 |
| Repeat and End Repeat Actions | 633 |
| Geostatistical Calculation Action..... | 633 |
| Calculation from Specification Action | 633 |
| Set Number Substitution Action..... | 634 |
| Save Dataset Action | 635 |
| Simulator Initialization Action | 636 |
| Queue for Simulation Action | 637 |
| Advanced Options | 638 |
| Running the Script..... | 639 |
| Setting Up Datasets for CMOST | 641 |
| Overview | 641 |
| Open CMOST Task File..... | 641 |
| Open CMOST Parameters Selection Window | 642 |
| Add CMOST Parameters..... | 644 |
| Edit CMOST Parameters..... | 648 |
| Delete CMOST Parameters | 649 |
| Delete Candidate Values from a CMOST Parameter..... | 650 |
| Add CMOST Parameter Associations..... | 651 |
| Delete CMOST Parameter Associations | 651 |
| Switch CMOST Parameter Associations..... | 652 |
| Export CMOST Master Dataset | 652 |

| | |
|---|------------|
| Save CMOST Task file..... | 653 |
| Save As CMOST Files..... | 654 |
| Merge Master Dataset Files | 654 |
| Parameterizing Relative Permeability Correlations..... | 655 |
| Parameterizing Formulas | 656 |
| Exporting Data | 659 |
| Overview..... | 659 |
| Exporting Grid Coordinates and Property Values | 659 |
| Exporting Properties to a RESCUE Model..... | 661 |
| Appendix A: Units and Unit Strings | 663 |
| Overview..... | 663 |
| Appendix B: Technical Notes - Tubing Head Pressure Calculator | 667 |
| Overview..... | 667 |
| PVT Correlations | 667 |
| Pressure-Gradient Prediction Methods | 668 |
| References..... | 669 |
| Appendix C: Correlation Equations | 671 |
| Equations for Prediction of Relative Permeability in Sandstone and Conglomerate..... | 671 |
| Equations for Prediction of Relative Permeability in Limestone and Dolomite | 672 |

Introduction and Overview

Introduction

Builder is a Microsoft Windows based software tool that you can use to create simulation input files (datasets) for CMG simulators. All three CMG simulators – IMEX, GEM and STARS – are supported by Builder. Builder covers all areas of data input, including creating and importing grids and grid properties, locating wells, importing well production data, importing or creating fluid models, rock-fluid properties, and initial conditions. Builder contains a number of tools for data manipulation, creating tables from correlations, and data checking. It allows you to visualize and check your data before running a simulation.

Brief Abstract of the Chapters in this Manual

[Reservoir Description](#) describes in detail how to use Builder to create and modify a simulation grid, interpolate grid structure and other properties, and add well locations. The chapter shows how to import 3D static models from several modelling packages, specify aquifers, sectors, and lease-planes, refine the grid structure, import well trajectory data in several formats, combine layers, and extract a grid subsection. Finally, the chapter shows how to use Builder to enter rock compressibility and to specify compaction/deletion regions.

[Fluid Model - IMEX](#), [Fluid Model - GEM](#), and [Fluid Model - STARS](#) discuss creating and editing fluid models, PVT regions and tables, and components used by each of the IMEX, GEM and STARS simulators. You can view XY plots of tabular properties, and also use WinProp to create fluid models.

[Rock Fluid Properties - IMEX](#), [Rock Fluid Properties - GEM](#) and [Rock Fluid Properties - STARS](#) discuss creating and editing rock-fluid properties, including relative permeability tables, interpolation sets, and adsorption. Again, XY plots of tabular data will be displayed by Builder.

[Initialization - IMEX](#), [Initialization – GEM](#) and [Initialization – STARS](#) discuss setting up the initialization options used by the simulators.

[Well and Group Control](#) discusses importing and editing the well and well group operation controls, including rates, constraints and targets. The Production Data wizard is used to simplify importing historical well data into Builder. A complete interface to all well controls is available. Builder also produces a Time-Line view, showing the changes in well controls versus time.

[Input/Output Control](#) discusses the Builder interface to the simulator controls for output from the simulator, and how to set up restart simulation runs.

[Numerical Control](#) discusses the Builder interface to numerical control options for the simulators.

[Geomechanics – GEM and STARS](#) discusses Builder interface to Geomechanics section of GEM and STARS.

[Changing Display Content and Settings](#) discusses modifying, saving and printing the displayed image. You can control fonts, line colors, title locations, etc., in a manner similar to CMG's Results 3D.

[Formula Manager](#) discusses how to calculate array properties from other array properties using formulas in a general calculator.

[Importing Geological and Well Trajectory Data](#) describes the formats of geological and well trajectory data that can be read by Builder.

[Creating Maps and Geostatistical Property Calculations](#) describes tools in Builder to create 2D geological maps, and to perform 2D and 3D geostatistical property calculations.

[Property Calculation Scripting](#) describes the feature that is designed to help you automate some of the workflow procedures and iterative calculations.

[Appendix A: Units and Unit Strings](#) includes two tables: a table of the expected units, by unit system, and a table of allowed unit strings for each unit dimension.

[Appendix B: Technical Notes - Tubing Head Pressure Calculator](#) provides an outline of the correlations used for the evaluation of PVT properties and pressure gradients used in the tubing head pressure calculator.

[Appendix C: Correlation Equations](#) provides the formulas used to predict relative permeability for various rock types.

What's New in Builder 2012

1. Improved support for visualization of geomechanics boundary conditions has been added. Refer to [Geomechanics – GEM and STARS](#), [Displacement Boundary Conditions](#).
2. A trajectory smoothing feature has been added. Through this feature you can highlight and resolve problems with trajectories, insert and delete trajectory nodes, eliminate kinks, interpolate between nodes, replace an existing trajectory with a smoothed trajectory, and export the smoothed trajectory to a Builder-formatted trajectory file for later import. Refer to [Trajectory Smoothing](#).
3. FlexWells now supports the REPLACE keyword, which allows a given FlexWell to change a PRODUCER for an INJECTOR and vice versa. This is supported in the **Events** table. FlexWells is now consistent with the PHWELLBORE model so if an annulus uses a wellbore model, all tubing wells must also use it. FlexWells now supports WP (withdrawal point) through the user interface. If a withdrawal point is present, FlexWells checks that the wellbore model is not used. Refer to [FlexWells \(STARS\)](#) for further details.

4. Formula Manager provides a centralized location for managing formulas. Refer to [Formula Manager](#).
5. The Import Black Oil PVT wizard now supports multiple PVT regions, and improvements have been made to the user interface. Refer to [STARS Import Black Oil PVT Wizard](#).
6. Builder supports datasets with grids specified in node based format. If the grid is not specified in node based format, Builder can convert (most of) the corner points. Refer to [Converting an Existing Simulation Grid to a Node Based Format](#).

What's New in Builder 2011

1. A wizard has been added to allow the user to more easily add or remove grid refinements. Support for non-uniform refinements has also been added. See [Adding Refined Grids Using the Refinement Wizard](#).
2. The uplayering feature has been improved significantly. An interface has been developed to provide an interactive method of creating layer boundaries. You may also now divide the reservoir into different sections to improve performance of the automatic layer generator. See [Combining Layers](#).
3. A wizard has been added to allow the user to more easily fix overlapping blocks where, for example, the contour data for two grid tops and the thickness of the layer or layers in between are not consistent. See [Fixing Overlapping Blocks](#).
4. The interface available for filtering well or group events (for copying or deletion) has been enhanced to allow the user to more easily create new dates and to set search conditions. See [Using the Well and Date Filter](#).
5. In the interface for attaching wells to groups, the operation has been enhanced to facilitate the attachment of wells from sectors. See [Adding Wells to Group](#) and [Using the ByFilter Button](#).
6. The trigger function has been enhanced (GEM only) to allow the triggering of an action based on a logical statement of multiple trigger conditions. See [Trigger](#).
7. The FlexWell feature has been enhanced to provide support for reporting groups, instrument tubings, packers, and specification of the maximum Nusselt number. A FlexWell event scheduler is now available that provides an interface for controlling well status from within the FlexWell. See [FlexWells \(STARS\)](#).
8. In certain edit operations, Builder provides the option of retaining existing MOD keywords instead of overwriting them with the newly calculated values. See [Property Modifications \(MODs\)](#).
9. Builder now supports High-DPI screen configurations.

What's New in Builder 2010

1. An interface has been added to STARS datasets for supporting the new FLX_WELBORE keywords. Please see the [FlexWells \(STARS\)](#) section for more information.
2. An interface has been added to IMEX datasets for supporting the new Flux Sector keywords. See the [Flux Sector \(IMEX Only\)](#) section for more information.
3. The interface for Saving Builder datasets has been updated to improve the customization of how datasets are saved. The user can now save most keywords in their own include file / binary file, or group together multiple adjacent keywords (as opposed to being limited to saving entire sections of the dataset). The updated version of the **Save As** dialog box will provide better tailoring for CMOST template files. See [Saving Your Work to a Simulator Input File \(Dataset\)](#).
4. The liquid phase viscosity tables in STARS datasets have been updated to allow the user to create multiple pressure based viscosity tables. See [Entering Liquid Phase Viscosity Data](#).
5. The **Hydraulically Fractured Wells** dialog box has been updated to support the creation of Stimulated Reservoir Volume (SRV) stages. These stages can be created with the mouse or by incorporating microseismic data into Builder. See the [Hydraulically Fractured Wells \(IMEX & GEM\)](#) and [Microseismic Events](#) sections for more information.
6. Builder now supports microseismic data. Using the Microseismic Data Manager, you can import and export files, change how microseismic events are displayed, as well as animate events in both 2D and 3D. The data manager also enables you to use microseismic events as a guide to creating a new Cartesian grid. Microseismic data can also be used in modeling a Stimulated Reservoir Volume (SRV) stage via the **Hydraulically Fractured Wells** dialog box. See the [Hydraulically Fractured Wells \(IMEX & GEM\)](#) and [Microseismic Events](#) sections for more information.
7. In Builder, it is possible to create or modify Parameters in CMOST Master Datasets (CMM files). The ability to do this for formula related parameters and relative permeability end point parameters has been added. See the [Parameterizing Relative Permeability Correlations](#) and [Parameterizing Formulas](#) sections for more information.
8. Builder now supports the ability to have multiple Numerical sets for STARS datasets. These numerical sets are time dependent, so different dates can use different numerical settings. See the [Setting and Editing Numerical Controls \(STARS\)](#) section for more information.
9. The Compaction/Dilation section for GEM datasets has been updated to support some new thermal based keywords. See the [Setting Thermal Properties for the Rock Type \(GEM Only\)](#) section for more information.
10. In Builder and Results 3D, new functionality has been added in the **Formulas** dialog box which allows spatial properties to be calculated based on the values in

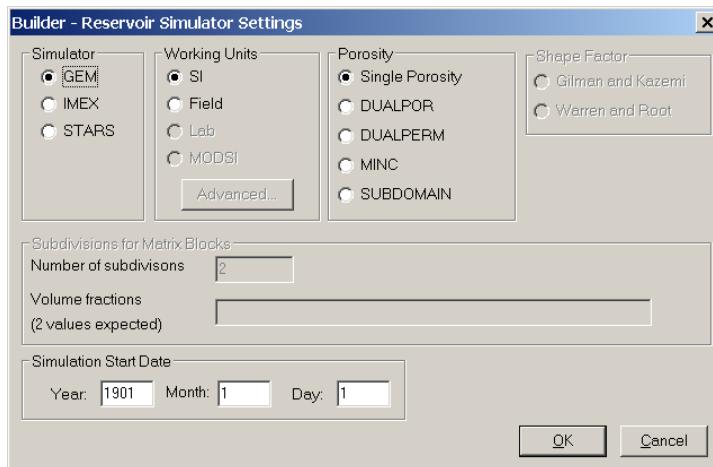
neighboring blocks. The list of functions available is discussed in the [Formula Manager](#) section.

Starting Builder

Prior to starting Builder, you should set up a project directory within the CMG Technologies Launcher. If you need to convert any contour map or mesh map files, this should also be done prior to starting Builder.

To start Builder from CMG Technologies Launcher for a new case:

1. In the CMG Technologies Launcher, double-click the Builder icon. Builder starts then the **Reservoir Simulator Settings** dialog box is displayed.



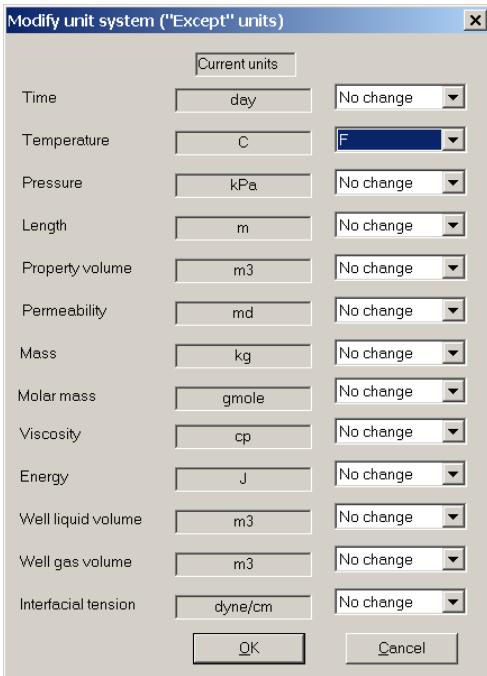
2. Under **Simulator**, select the simulator that you will be using.
3. Under **Working Units**, select the unit system to use.
4. If you were creating a STARS dataset, the **Advanced...** button will be enabled. Currently this is used for specifying the “Except” units (see the *EXCEPT keyword in the *STARS User’s Guide*). See [Specifying “Except” Units in STARS](#).
5. Under **Porosity**, select Single Porosity, or one of the dual porosity options. Some of the dual porosity options will enable input of **Shape Factor** or **Subdivisions for Matrix Blocks** input.
6. Enter the **Simulation Start Date**. This is usually the date of the start of production or injection in the earliest well.
7. Click **OK** to apply your settings.

To start Builder with an existing data set:

1. In the CMG Technologies Launcher, click the main file of the existing data set then drag it onto the Builder icon.
2. Release the mouse button. Builder will start up and open the existing data set.

Specifying “Except” Units in STARS

To specify “Except” units in STARS, click **Advanced** in the **Builder – Reservoir Simulator Settings** dialog box (which is displayed when you create a new dataset). The **Modify unit system (“Except” units)** dialog box is displayed:

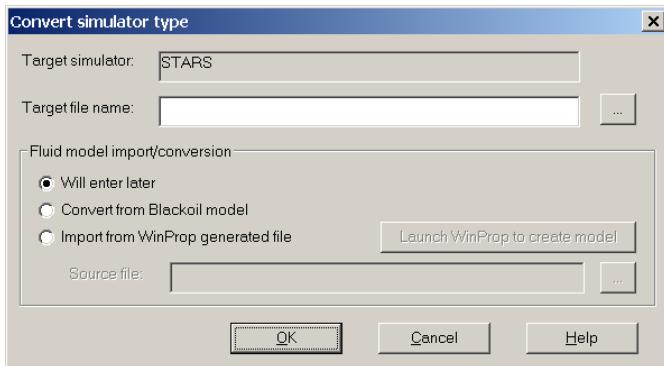


Select the alternate units for one or more of the properties listed and then click **OK**. The **Builder – Reservoir Simulator Settings** dialog box becomes active. Once you click **OK** in the **Builder – Reservoir Simulator Settings** dialog box, the “Except” units cannot be changed. If you enter a dataset with “Except” units, Builder honors them, but you cannot change them.

Converting Simulator Type for a Dataset

You can use Builder to convert an IMEX dataset to a STARS or GEM dataset. From the **File** menu select **Convert simulator type for dataset** and then **To STARS** or **To GEM**. If you have just closed a dataset or cancelled out of a new dataset dialog box (the **Builder – Reservoir Simulator Settings** dialog box), select **Convert simulator type for dataset** and then **IMEX to STARS** or **IMEX to GEM**. In this case, specify the IMEX dataset file that you want to convert.

Builder will launch the **Convert simulator type** dialog box. Either type in the name of the target dataset file or use the browse button to specify the new dataset file.



You can specify the WinProp generated file from which to import the Fluid Model in the new dataset file by selecting **Import from WinProp generated file** in the **Fluid model import/conversion** area. If you have not already generated a file, you could do so by launching WinProp from the dialog box. When you are done with WinProp, specify its name in the **Source File** text box.

If you are converting to STARS, you could convert the existing Black oil PVT data into STARS fluid model by selecting **Convert from Blackoil model** option button.

Builder will convert the data in the Reservoir, Components, Rock-Fluid, Initial and Recurrent Data sections that are common between the two simulators (IMEX and the target). The data that could not be converted is listed in the messages that come up while the conversion is taking place. It may be a good idea to save and examine these messages to further complete the new dataset.

Builder also imports all the relevant metadata – RESULTS keywords – that it stores in the dataset to facilitate restoring these objects.

Unit conversion is not done in this feature. You cannot perform the conversion if:

1. The IMEX dataset were using the MODSI units system.
2. The IMEX dataset were using the LAB units and you want to convert to GEM.

Importing Partial Simulator Data

You can import the following data into Builder:

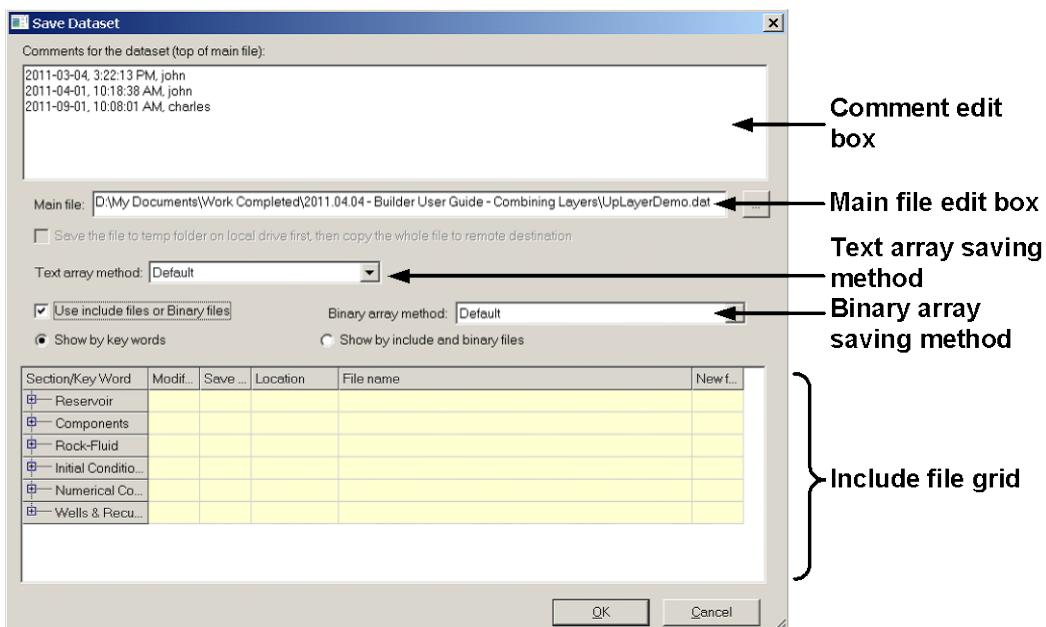
1. Simulation grid and spatial properties from another dataset file. See [Importing Spatial Properties](#).
2. Simulation grid and spatial properties exported from a geological program. The exported data could be a RESCUE model or a partial dataset for a CMG simulator. See [Importing 3D Simulation Grids and Grid Properties](#).
3. Spatial properties from another dataset or from an ASCII text file. See [Importing Spatial Properties](#).
4. Spatial properties from CMG simulation results files. See [Importing Spatial Properties](#).

5. Component Properties data from a file generated using CMG's WinProp. See the individual sections for the different simulators on Fluid Model in this manual.
6. Black oil PVT into a STARS model. See [STARS Import Black Oil PVT Wizard](#).
7. Component Properties data from another dataset file. From the **File** menu select **Import from another file** and then select **Component Properties**. Specify the dataset file from which you want to import the property.

Saving Your Work to a Simulator Input File (Dataset)

Builder has both **File | Save** and **File | Save As** menu items. You can save the file as a single file, or as a set of “*INCLUDE” files.

When you select the **Save As** menu item (or the first time you select **Save** for a new dataset), you will bring up the **Save As** dialog box:



The dialog box is split into the following sections:

- Comment edit box
- Main file edit box
- Text array saving method combo box
- Use include files and Binary files check box
- Binary array saving method combo box
- Show by key words select button
- Show by include and binary files select button

- Include File Grid

These controls define how the dataset is saved.

Saving Comments

The comments for the **Save As** dialog box are stored at the top of the main file of the dataset. By default, a timestamp and user ID are appended to these comments every time the **Save As** dialog box is used. You can edit these comments in the **Comment** edit box.

Selecting the Main File

The **Main File** edit box shows the currently defined main file. You can change this file by clicking the browse  button to the right of the box. This opens a standard Windows **Save File** dialog box which can be used to change the filename and directory where the dataset will be stored. Include files (if there are any) are stored in the same directory.

Choosing the Array Saving Method

Datasets include a large amount of numeric data that is organized in arrays. The manner in which these arrays are organized in the dataset makes a real difference on the size of the files and the speed with which they save and load. The **Array Saving Method** combo box handles this facet of the dataset. There are two ways for saving arrays, one is text format, and the other is binary format.

There are two methods in text format:

- Optimize for Size & Speed
- Optimize for Readability

There are two methods in binary format as well:

- Binary File Format (*.cmgbin)
- Compressed Binary File Format (*.cmgbin)

If you chose **Optimize for Size & Speed**, then all of the arrays will be written compressed (i.e., 2345*0.0) and with long line lengths. If you choose **Optimize for Readability**, then all of the arrays will be written for *readability* (uncompressed, and with J,K annotations) when you save the dataset.

The **Binary File Format** method outputs grid and spatial property arrays in binary format in a special file with extension .cmgbin. This file is saved in the same folder as the main dataset file. The reading of binary data is much faster than the reading of data in ASCII format.

Therefore this selection can speed up the loading of the data in Builder for large models. The simulators also read this file. If an array is “simple”, for example, *CON, the array will NOT be written to the binary format even if you select this option. You can switch between the binary and ASCII formats whenever you want.

From the user’s perspective, the **Compressed Binary File Format** method is identical to the **Binary File Format**. The difference is handled internally. This saving option usually generates smaller .cmgbin file size and results in faster reading speed.

Grid Display Selection

There are two ways to show the include file name with key words:

- Show key words followed by file names.
- Show file names followed by key words list.

When **Show by include and binary files** is selected, the following grid format is displayed:

| File name | Modifi... | Save ... | Key words |
|-----------------|-----------|-------------------------------------|------------|
| WATFLD_FAULT... | No | <input checked="" type="checkbox"/> | FAULTARRAY |
| WATFLD_POR.inc | No | <input checked="" type="checkbox"/> | POR |

Organizing the Data

There are two ways to save the data:

- All data in a single file.
- Some data in include and binary files.

When **Use include files and Binary files** is cleared, the entire dataset will be stored in the main file. The **Include File Grid** is not required and will not be displayed.

When **Use include files and Binary files** is selected, the dataset is stored in a number of different files (include files) as well as in the main file. The **Include File Grid** controls these include files so it is required and displayed.

Organizing Include Files

The **Include File Grid** allows the user to organize the include files. Each row describes a segment of the dataset and where it will be stored. The columns describe how each segment is stored. You can modify this information within the grid.

The **Section/Key Word** column describes the set of information being saved. It lists the six sections. The key word list will be displayed under the section when is clicked:

| Section/Key Word | Modifi... | Save ... | Location | File name | New f... |
|------------------|-----------|--------------------------|-----------|------------|----------|
| Reservoir | | | | | |
| Components | | | | | |
| MODEL | No | <input type="checkbox"/> | Main file | WATFLD.DAT | No |
| PVT | No | <input type="checkbox"/> | Main file | WATFLD.DAT | No |
| BWI | No | <input type="checkbox"/> | Main file | WATFLD.DAT | No |
| CO | No | <input type="checkbox"/> | Main file | WATFLD.DAT | No |
| CVO | No | <input type="checkbox"/> | Main file | WATFLD.DAT | No |
| CWW | No | <input type="checkbox"/> | Main file | WATFLD.DAT | No |
| CW | No | <input type="checkbox"/> | Main file | WATFLD.DAT | No |

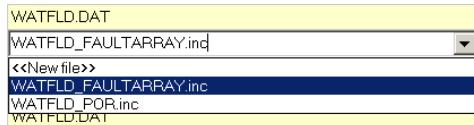
The **Modified** column indicates if that portion of the data has been modified, and will therefore be written out.

The **Save file** column indicates if the file will be saved. If a new file is added or a file name is changed, the **Save file** check box will be automatically checked. When a check box is clicked all **Save file** check boxes with the same file name will be updated also.

The **Location** column indicates whether the data will be saved in the *Main file*, in its own *Include file*, in the *Binary file*, or in the *Binary file in include file*. This affects how the corresponding **Filename** entry will behave. There is a combo box in the cell of this column which users can use to select where to save the key word with data. If the key word can be saved as binary format, it will have four selections; otherwise, it has two selections (the *Main file* or *Include file*).

| Section/Key Word | Modif... | Save ... | Location | File name |
|------------------|----------|-------------------------------------|-----------------------------|---|
| NULL | No | <input type="checkbox"/> | Main file | WATFLD.DAT |
| PINCHOUTA... | No | <input type="checkbox"/> | Main file | WATFLD.DAT |
| FAULTARRAY | No | <input checked="" type="checkbox"/> | Include file | WATFLD_FAULTARRAY.inc |
| POR | No | <input checked="" type="checkbox"/> | Binary file in include file | Include File: WATFLD_POR.inc Binary File: WATFLD_Pi |
| PERMI | No | <input type="checkbox"/> | Main file | WATFLD.DAT |
| PERMJ | No | <input type="checkbox"/> | Main file | WATFLD.DAT |
| PERMK | No | <input type="checkbox"/> | Main file | WATFLD.DAT |
| TRANSI | No | <input type="checkbox"/> | Main file | WATFLD.DAT |

The **File name** column indicates the name of the file where the data is saved. All of the files will be saved in the same hard disk as the *main file*. For *Include files*, they can have the relative path and the names can be changed by selecting the <<New file>> in the combo box.



The Copy and Paste popup menu can be used to copy an include file name to multiple neighboring rows. To access this feature, use the mouse to select the row and right-click to select either **Copy** or **Paste**.

| Section/Key Word | Modif... | Save ... | Location | File name | Newf... |
|------------------|----------|-------------------------------------|--------------|----------------|---------|
| Reservoir | | | | | |
| Components | | | | | |
| MODEL | No | <input type="checkbox"/> | Main file | WATFLD.DAT | No |
| PVT | No | <input checked="" type="checkbox"/> | Include file | WATFLD_PVT.inc | Yes |
| BWI | No | <input type="checkbox"/> | Main file | WATFLD.DAT | No |
| CO | No | <input type="checkbox"/> | Main file | WATFLD.DAT | Copy |
| CVO | No | <input type="checkbox"/> | Main file | WATFLD.DAT | Paste |
| CW | No | <input type="checkbox"/> | Main file | WATFLD.DAT | No |
| CW | No | <input type="checkbox"/> | Main file | WATFLD.DAT | No |

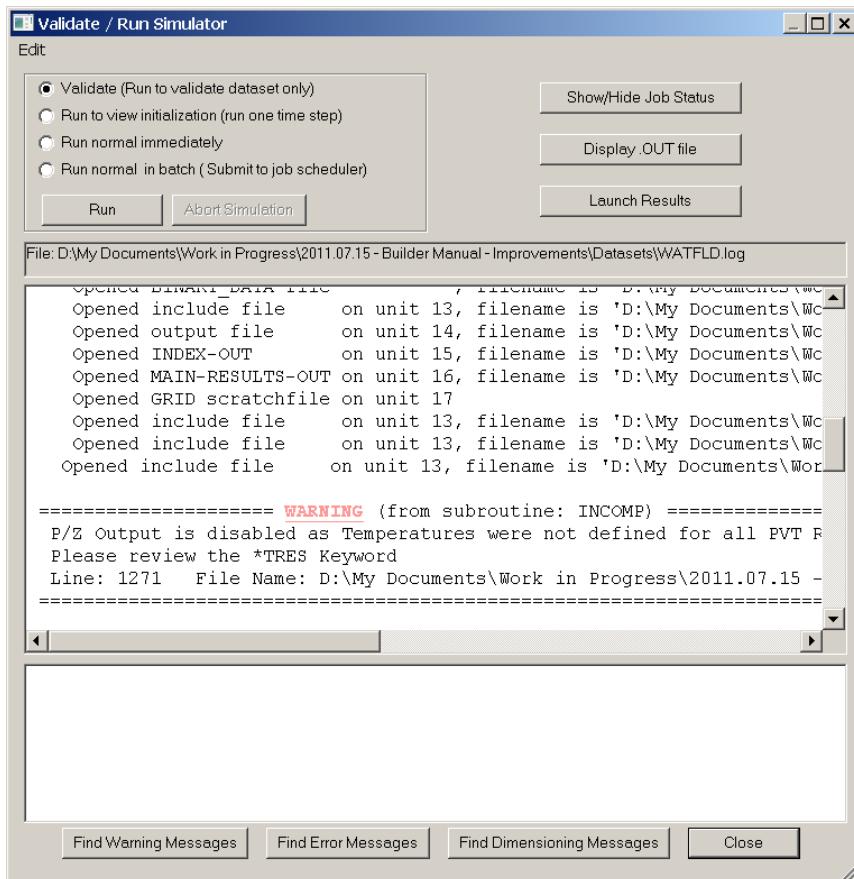
For the *main file*, the cell is read-only – users can only change the name/location of the *main file* in the **Main File** text box at the top of the dialog box. If the users do change the main file in the **Main File** text box, the *main file* name in the read-only cells will be updated to match. Only the adjacent key words can be saved into the same include file. For each dataset only one binary file is supported.

The **New file** column indicates if this file exists in the current directory. If it does not exist, this will read *yes* and the new file will be created. This gives the users feedback in case there are files they do not want to create (because they already exist).

Validating the Dataset

Builder validates your data when it reads in a file. It also validates the data dynamically as you enter or generate new data. The results of validation are shown in green (ok), red (error) and yellow (warning) bitmaps with individual items of the tree view. To view the details of the validation result, right-click in the tree view and then select **Validate**.

In addition, you can validate the dataset with the simulator. Click the **Validate with <simulator>** button or select **Validate with simulator** menu on the **Tools** menu. If you have made changes to the dataset, choose to save it when prompted. The **Validate/ Run Simulator** dialog box is displayed. Click **Run**.

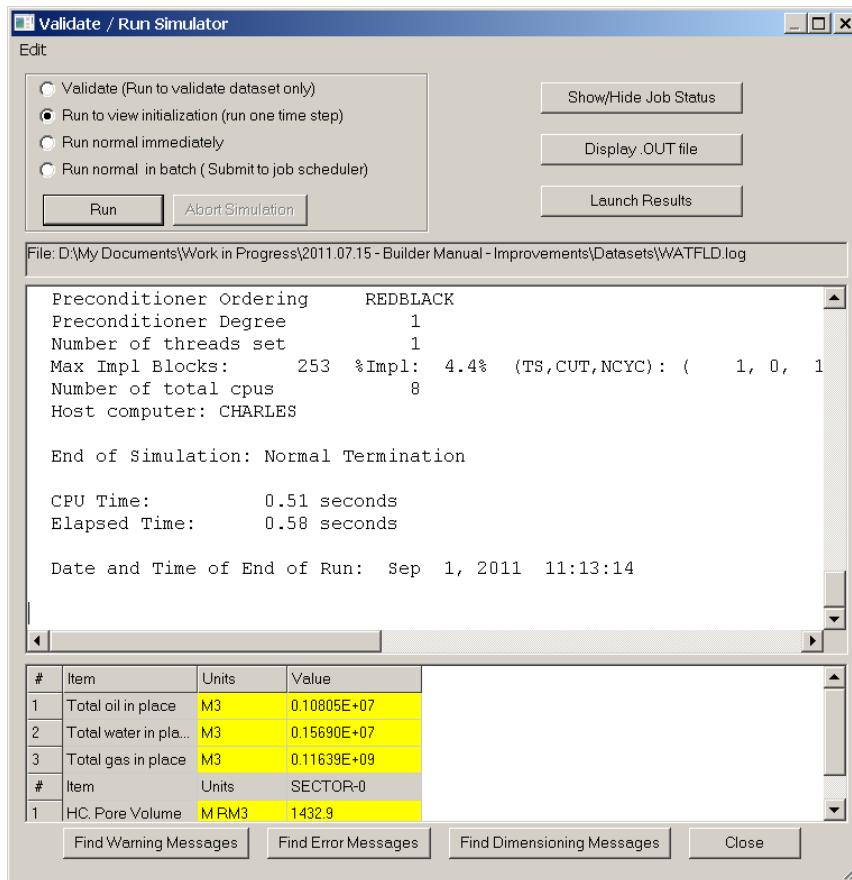


The log file generated by the simulator is displayed. When validation is complete, if there are errors or warnings, they are highlighted. You can switch between the log and the out files using the **Display .OUT file** button. You can also view errors and warnings by clicking **Find Error Messages** and **Find Warning Messages** at the bottom of the dialog box. To find runtime dimensioning messages from the simulator, click **Find Dimensioning Messages**.

Viewing Model Initialization Generated by Simulator

You can view the total and hydrocarbon pore volumes, and fluids initially in place in Builder. Select **Run simulator to view initialization** on the **Tool** menu. The **Validate / Run Simulator** dialog box appears. This is the same dialog box as that used for validating with the simulator. See [Validating the Dataset](#).

Click **Run simulator**. This causes the simulator to run for one time step. The log file generated by the simulator is displayed:



When the simulation is complete, the pore volume and the fluids in place information is extracted from the .out file and displayed in the grid at the bottom of the dialog box. You can also view the .out file by clicking **Display .OUT file**.

You can also launch Results 3D by clicking **Launch Results** and viewing the initialized values of saturations and other results.

Simulating Current Dataset from Within Builder

To run the simulation from within Builder, select **Run simulation** on the **Tools** menu. The **Validate / Run simulator** dialog box is displayed. This is the same dialog box as that used for viewing the initialized values generated by the simulator. See [Viewing Model Initialization Generated by Simulator](#).

Click **Run simulator**. The log file generated by the simulator is displayed.

When simulation is complete, the pore volume and the fluids in place information is extracted from the .out file and displayed in the grid at the bottom of the dialog box. You can also view the .out file by clicking **Display .OUT file**.

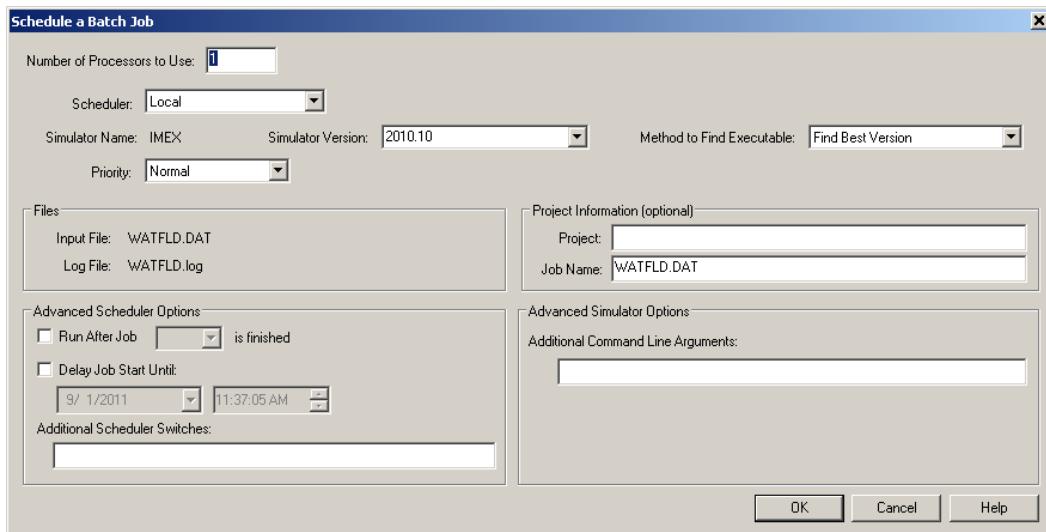
You can abort the simulator anytime by clicking **Abort simulation**.

You can also launch Results 3D by clicking **Launch Results** and viewing the initialized values of saturations and other results.

Simulating Current Dataset to a Job Queue from Within Builder

You can queue up several simulation jobs to be executed at a later time from within Builder by selecting **Run normal in Batch (Submit to job scheduler)** and then clicking **Submit**.

The **Schedule a Batch Job** dialog box is displayed:



You can submit a job to be executed at later time or run after a certain job is done. If you have Platform LSF software installed, **Submit to Platform LSF job queue** will be enabled. Refer to the file ‘readme_lsf.txt’ under Launcher’s installation folder for more information regarding the use of LSF with CMG’s software. If you have access to Microsoft Compute Cluster, then you can submit jobs to it. See the *Launcher User’s Guide* for information on configuring this feature.

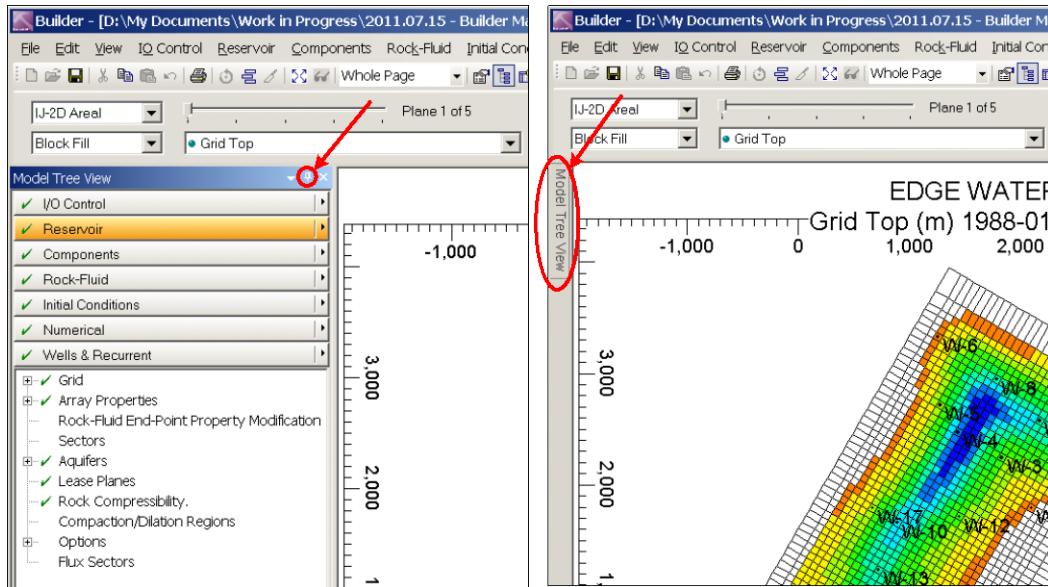
Customizing Toolbars, Menus and Theme

Builder allows you to customize your work environment. You are now able to drag all toolbars and menus of Builder and position them anywhere within your working environment. This permits you to have more screen real estate when viewing your reservoir. Builder remembers the location of the toolbars and menus between sessions. The next time you open Builder, items will appear in their last position when Builder was closed.

You are also able to change the theme used to color the Builder main window frame. To select a theme, click **View** in the menu then select **Theme**. The selected theme is also remembered between sessions.

Docking Panes

Builder provides a feature called docking panes. By clicking the pin in the tree view, the **Model Tree View** can be hidden when it is not in use. The **Model Tree View** can be displayed if you mouse over the tab hooked to the main window frame, as shown below:



Exiting Builder

To exit Builder, select **Exit** on the **File** menu.

Reservoir Description

Overview

Builder can create or import the simulation grid and grid properties describing the volume of a reservoir. Builder has tools for the creation of 3D models from 2D maps of top of structure, gross thickness, and other properties (see [Creating a Simulation Grid Using Structure Maps](#)). Builder can also import 3D “static” models from several geological modeling packages, such as EDS Gocad, DGI EarthVision, Roxar RMS, and SIS Petrel (see [Importing 3D Simulation Grids and Grid Properties](#)).

Once the grid structure has been created or imported, there are a number of grid editing operations that Builder can perform. Grids can be refined. Builder can split grid blocks in each of the I, J, and K directions. A sub-section of a grid can be extracted from a model. Finally, grid layers can be combined, to reduce the number of vertical layers.

To build a simulation model, you must locate wells in the grid. Builder imports a number of different well trajectory formats, then can locate the grid blocks that the well trajectory intersects.

This chapter also provides instructions for specifying aquifers, lease planes and sectors in your model. You can enter rock compressibility and specify compaction/dilation regions for your model. Some of these options will depend on the simulator you are using.

Creating a Simulation Grid Using Structure Maps

Constructing a simulation grid from 2D maps requires three main steps. First, you describe the grid geometry in aerial (or plan) view and position it over your geological maps. Second, you create your three-dimensional grid by interpolating the reservoir structure and rock properties from your geological maps. Finally, you specify the well completion locations in the grid, from well positions on the geological maps, from 3D well trajectories, or by entering them manually.

A more detailed process is as follows:

1. **Obtain map files:** Once you receive contour maps from a geologist you can either:
 - Digitize the maps with a digitizer tablet and the program Didger 3, or
 - Export ASCII output file from a geological modelling package to a file format that Builder can read (see [Importing Geological and Well Trajectory Data](#)). The data may either be in the form of contour maps or

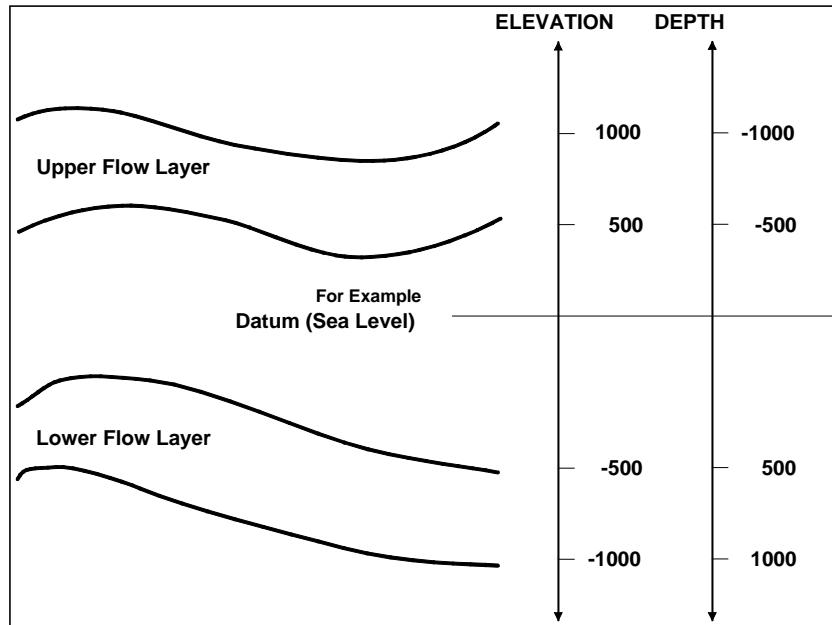
- a mesh of points with the property value at each point. Builder can read several contour and grid files formats, or
- Create a map from known data points (for example, well picks) using the map creation routines included in Builder (see [Creating Maps and Geostatistical Property Calculations](#)).
2. **Create a working directory:** Create a working directory, and place the contour map files in that directory. Start the CMG Technologies Launcher, click on “Manage Projects” and add a project for your directory.
 3. **Start Builder:** Using the Technologies Launcher, double-click the Builder icon to start it. By default, a “New” case will be started, and the **Reservoir Simulator Settings** dialog box will appear. This dialog box allows you to select the simulator, working units, single or dual porosity options, and a simulation start date.
 4. **Open a map and create a preliminary grid:** First, open one of the maps that you will be using. Define the grid in the aerial or plan view and position it on the grid. You can translate, rotate and resize the grid, move, add, and delete grid lines, and add refined grids.
 5. **Compare contour maps:** View the grid with other contour maps for the same reservoir to see if the grid adequately overlays all geological features of the reservoir for all maps and layers.
 6. **Specify calculations:** Specify how grid properties (porosity and permeabilities, for example) are to be calculated, either as:
 - Constant per layer,
 - Interpolated from contour maps, or
 - Calculated from other property values using a formula
 7. **Calculate:** Interpolate, assign or calculate the specified properties.
 8. **Add Wells:** Wells indicated on your contour maps will be automatically positioned on the simulation grid during the interpolation process. You can also add wells by reading in 3D well trajectory and perforation files. If you do not have perforation information, you can ask Builder to calculate the completion blocks. You can also manually enter well names and click on grid blocks to define completions.
 9. **Write out, review and adjust values, add other properties:** To save a partial simulation input data set, select **Save As** on the **File** menu. You can use Builder to add fluid property data, well rates, and other data required by the simulator.
 10. **Run the simulation.**
 11. **View the results of the simulation:** You can view the results of the simulation run using Results 3D and Results Graph.

Builder supports the creation of four types of grids: variable depth and thickness Cartesian, radial (cylindrical) grids, orthogonal corner point, and non-orthogonal corner point. Cartesian

and orthogonal corner point grids have rectangular grid blocks in map view. Non-orthogonal corner point grid blocks are four sided quadrilaterals, but need not be rectangular, in map view.

If you are creating a grid for a field study, you must open a contour file or mesh map file for the field prior to creating the grid. If you are doing an idealized pattern study, without any maps, it is not necessary to open a map prior to creating a grid.

Top Maps - Elevation or Depth Map



The inputs to the simulator are either *DTOP for Cartesian grids, defined as "the *depth* to the center of the top face of each grid block in the top layer of the grid" or *ZCORN, defined as "an array of corner point *depths* for the corner point grids". These are depths, measured positively increasing downward. The simulator would have gas percolate upward from a block with a greater depth (say, k=2, top=4800) to a block with a less depth (k=1, top=4700).

Over the past few years, more and more companies are changing to use elevation maps for formation top maps. In these maps, the measurement is positively increasing upward. The above diagram compares depths and elevations for two flow units of a formation.

Assume we have an elevation map for the top of "Upper Flow Layer". The interpolated value from the map would be 1000, and multiplying by -1 would give a depth of -1000. If the Upper Flow Layer is split into five simulation layers, then the tops (as depths for the simulator) would be (k=1 top = -1000), (k=2 top = -900), (k=3 top = -800), (k=4 top = -700), and (k=5 top = -600).

Now assume we have an elevation map for the top of the Lower Flow Layer. The interpolated value from the map would be -500, and multiplying by -1 would give a depth of 500. Again, split the reservoir into five simulation layers. The tops (as depths for the simulator) would be at (k=1 top = 500), (k=2 top = 600), (k=3 top = 700), (k=4 top = 800), and (k=5 top = 900).

One thing to be careful about if using elevation maps is well trajectories. In trajectory data, TVD represents depth and TVDSS represents the elevation.

Units for the X and Y Coordinates of Maps

It is not uncommon to have map files where the X and Y coordinates are in meters, while the Z value is in feet. Builder can accommodate this combination of units. The simulation input data can be constructed for either SI Units (distance in meters) or Field Units (distance in feet). CMG's simulators do not allow mixed distance units in the simulation input data.

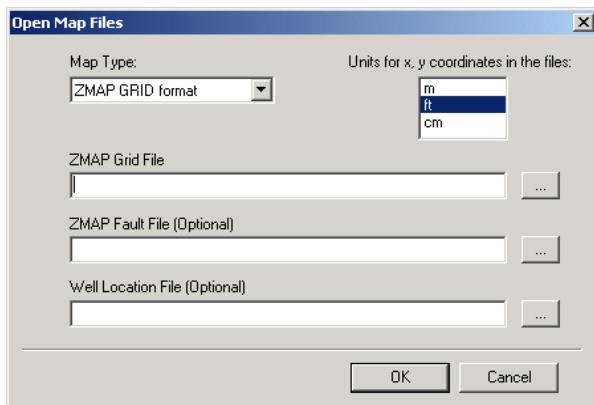
If you want to work in Field Units, your map X, Y coordinates are in meters, and depth and thickness values on the maps are in feet, then select Field Units. When opening map files or specifying them for interpolation, make sure that the **Units for x,y coordinates in the files**, on the **Open Map Files** dialog box, are specified correctly as meters. No further action is required.

Opening and Displaying a Contour Map File or Mesh Map File

The supported contour map and mesh map file formats are discussed in the chapter [Importing Geological and Well Trajectory Data](#).

To open a contour map or mesh map file:

1. From the **File** menu, select **Open Map File**. The **Open Map Files** dialog box is displayed:



2. From the **Map Type** drop down selection box, select the format corresponding to your files. For a description of the supported formats, see the chapter [Importing Geological and Well Trajectory Data](#).
3. Depending on the **Map Type**, one or more file name text boxes will appear. (Some formats require separate files with fault or well information.) Click the browse button, next to the file name box, and a standard **Open File** dialog box will appear. Select the file to open, then click **OK** or **Cancel**.
4. If the format for the Map Type contains unit information, the units used in the file will be selected. Otherwise, you will need to select the **Units for x,y coordinates in the files**.

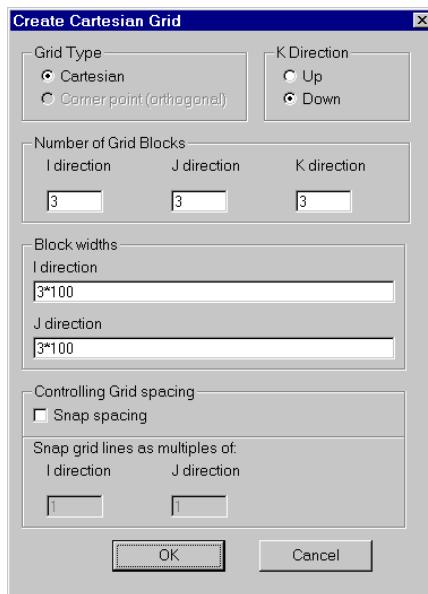
- Click **OK** to open the map, or **Cancel**.

To open a different map:

- Builder supports the simultaneous display of multiple maps. Select **File | Open Map File** to open a map and to open additional maps. Maps are displayed in different colours for visual differentiation.
- To close one or more maps, select **File | Close Map File**. The **Select Map Files to Close** dialog box is displayed. Select the map or maps you want to close then click **OK**.

Creating a Variable Depth or Thickness Cartesian Grid or Orthogonal Corner Point Grid

- To create a Cartesian grid, select **Cartesian** from the **Reservoir** menu. To create an orthogonal corner point grid, select **Orthogonal Corner Point** from the **Reservoir** menu. The **Create Cartesian Grid** dialog box is displayed:



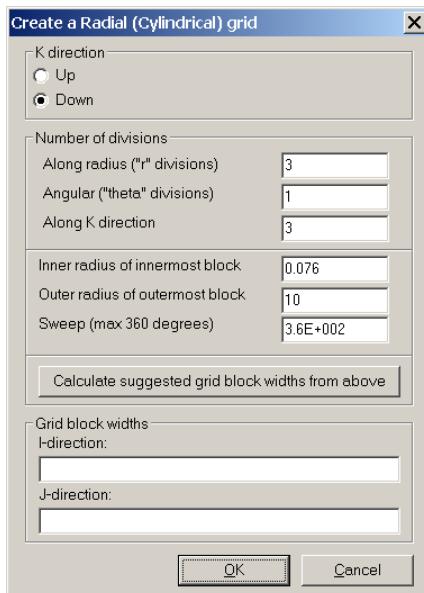
- For Cartesian grids, select the **K Direction** as either **Down** or **Up**. With K Direction Down, the K layers are numbered increasing downward. Corner point grids must be numbered with K increasing downward.
- In **Number of Grid Blocks**, specify the number of blocks for each direction.
- In **Block widths**, enter specific block widths. Separate the width notation with commas or spaces. You can use notation to quickly specify block widths. For example, to specify a 5 block model where the first block is 2000 m, the next three blocks are 500 m and the last block is 1500 m, you can enter the following:

2000, 3*500, 1500

- To force the grid spacing (grid block widths) to be round numbers when moving grid lines, select **Snap spacing** in the **Controlling Grid spacing** panel. Enter values for the intervals to which the moved grid line will snap. For example, suppose you have two blocks, each of width 50 m. Further, suppose that you want to move the grid line between the blocks. Without snap spacing, if you use the mouse to move the grid line, the grid block widths after the move may be numbers with a large number of digits after the decimal (say, 41.9231 and 58.0769). If you set the snap spacing to 5 then moved grid lines would be spaced multiples of 5 apart.
- Click **OK** to accept your selections, or **Cancel** to cancel your selection. If you click **OK**, the grid appears (overlapping the contour map if one is open).

Creating a Radial Grid

- To create a radial grid, select **Radial (Cylindrical)** from the **Reservoir** menu. The **Create a Radial (Cylindrical) grid** dialog box is displayed.



- Select the **K direction** as either **Up** or **Down**.
- Enter the number of grid blocks or divisions along each of the radial, angular and K directions.

4. To automatically calculate the grid block widths in the I (radial) and J (angular) directions, enter the **Inner radius of innermost block**, the **Outer radius of outermost block**, and the **Sweep** of the grid, then click on **Calculate suggested grid block widths from above**.
5. Alternatively, you can directly enter a set of block widths for the I (radial) and J (angular) directions in the **Grid block widths** fields.
6. Click **OK** to create the grid, or **Cancel**.

Moving and Rotating a Grid

To move a Cartesian grid, radial grid, or an orthogonal corner point grid:

1. You must be in **Edit Grid** mode to perform this operation. To enter **Edit Grid** mode, select **Edit Grid** from the mode selection box, or right-click to pop up the context menu and select **Edit Grid** by left-clicking on it.
2. Either:
 - Hold down the SHIFT key and click on a point of the grid with your cursor. While holding the mouse button down, drag the grid to the desired position.

OR

 - Use the **Modify Grid Position** dialog box by selecting **Modify Position** from the **Reservoir** menu. Enter the X and Y locations (in map coordinates) of the origin of the grid.

To rotate a Cartesian grid, radial grid, or an orthogonal corner point grid:

1. You must be in **Edit Grid** mode to perform this operation. To enter **Edit Grid** mode, select **Edit Grid** from the mode selection box, or right-click to pop up the context menu and select **Edit Grid** by left-clicking on it.
2. Either:
 - Hold down the CTRL key, and click on a point on the grid with your cursor. While holding down the mouse button, drag the grid around to the desired angle.

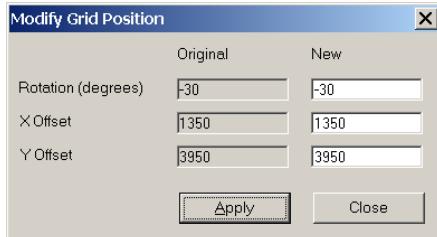
OR

 - Use the **Modify Grid Position** dialog box by selecting **Modify Position** from the **Reservoir** menu. Enter the angle of the grid, in degrees, measured counter-clockwise from the X-axis.

To see or enter exact X and Y offsets of the grid origin from the contour map origins:

1. You must be in **Edit Grid** mode to perform this operation. To enter **Edit Grid** mode, select **Edit Grid** from the mode selection box, or right-click to pop up the context menu and select **Edit Grid** by left-clicking on it.

2. Select **Modify Position** from the **Reservoir** menu. The **Modify Grid Position** dialog box is displayed:



3. The current values for **Rotation**, **X Offset** and **Y Offset** are shown.
4. To move or rotate the grid, enter new values in the **New** column and click **Apply**, otherwise click **Close** to dismiss the dialog box.

Moving Grid Lines

Note: You cannot move grid lines in radial or non-orthogonal corner point grids.

To move a grid line:

1. You must be in **Edit Grid** mode to perform this operation. To enter **Edit Grid** mode, select **Edit Grid** from the mode selection box or right-click to pop up the context menu and select **Edit Grid** by left-clicking on it.
2. As you move the mouse cursor over the grid, Builder displays the location of the cursor in the status bar at the bottom of the main window. In **Edit Grid** mode, it will display the grid line or grid block under the cursor.
3. Select the line you want to move by placing your cursor on the line and clicking. The line you select will appear in the highlight color (default – red). For finely spaced grids, you may need to zoom in on the region of the grid on which you are working. For details on zooming, see [Zooming and Panning While Creating or Editing a Grid](#).
4. Drag the grid line to the new position, and release the mouse button.

Adding and Deleting Grid Lines

Note: You cannot add or delete grid lines in radial or non-orthogonal corner point grids.

To add a grid line:

1. You must be in **Edit Grid** mode to perform this operation. To enter **Edit Grid** mode, select **Edit Grid** from the mode selection box, or right-click to pop up the context menu and select **Edit Grid** by left-clicking on it.
2. Click on a line that is similar to the one you want to add. The line you select appears in the highlight color (default – red).

3. Select **Copy** from the **Edit** menu, or press CTRL+C.
4. Click on the grid block in which you want the line inserted.
5. Select **Paste** from the **Edit** menu, or press CTRL+V.

To delete a grid line:

1. You must be in **Edit Grid** mode to perform this operation. To enter **Edit Grid** mode, select **Edit Grid** from the mode selection box, or right-click to pop up the context menu and select **Edit Grid** by left-clicking on it.
2. Click on the line you want to delete. The line you select appears in the highlight color (default – red).
3. Select **Cut** from the **Edit** menu, or press CTRL+X.

Undoing a Change

The Undo feature only works when you are moving or rotating a grid, moving grid lines or adding refined grids. Only one level of Undo is saved. Changes made with the Create Grid dialog box cannot be undone.

To undo a change:

- Select **Undo** from the **Edit** menu. This returns your grid to the state it was before your last change.

Creating a Non-Orthogonal Corner Point Grid with Vertical Faults

Builder includes a “wizard” to step you through the creation of a non-orthogonal corner point grid. The wizard leads you through several steps in a linear fashion. The steps are as follows:

1. **Begin:** Open a contour or mesh map if one is not already open.
2. **Outer Boundary:** Define the outer boundary of the grid region by clicking on map locations.
3. **Inner Control Lines:** Add control lines in the interior of the outer boundary. These control lines can be used to force grid block boundaries to follow faults, run parallel to horizontal wells, or to create regions of varying grid density.
4. **Adjustment of points on boundaries and control lines:** Make fine adjustments on the location of points on the control lines.
5. **Number of grid blocks:** Input the number of grid divisions along each segment of the control lines.
6. Grid creation.

The wizard will allow you to return to a previous step if you wish to change your inputs. Online help is available in the wizard to explain each step in detail.

To start the non-orthogonal corner point grid creation wizard:

- Select **Non-orthogonal corner point grid** from the **Reservoir** menu. The toolbar will change to a wizard control bar. Read the instructions, and click the **Help** button for more information.

Zooming and Panning While Creating or Editing a Grid

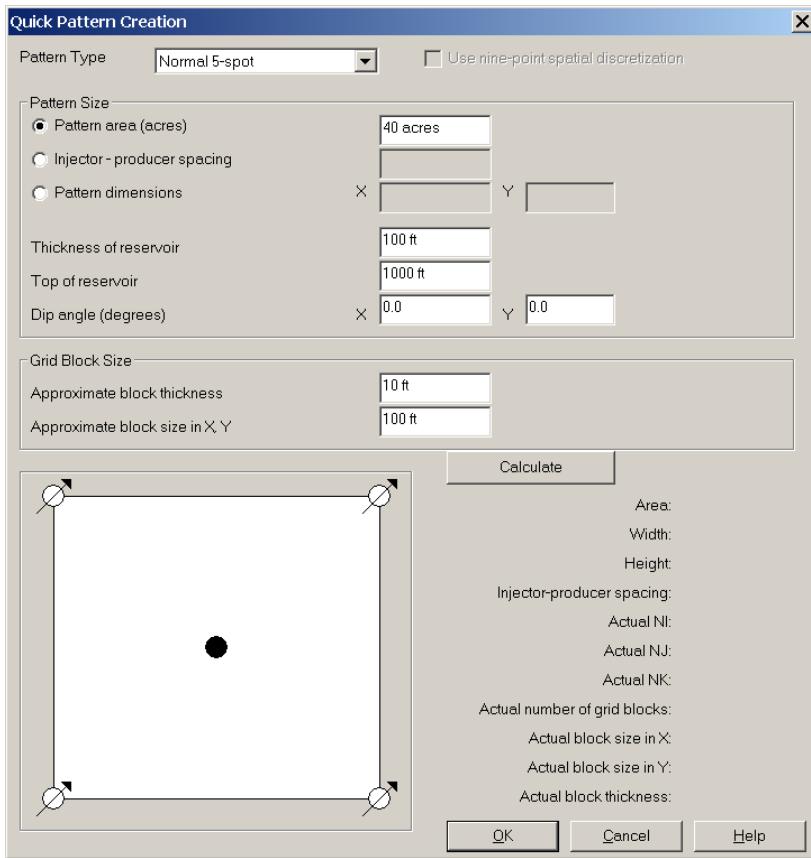
You can zoom in on a part of the grid, or pan across the grid while zoomed-in, during grid construction. This is done as follows:

1. Right-click in the reservoir viewport. A pop-up (context) menu appears.
2. Select **Zoom Reservoir** or **Pan Reservoir** by clicking on the menu item with the left mouse button.
3. Perform the zoom or pan operation(s).
4. Right-click in the reservoir viewport. A pop-up (context) menu appears. Select **Edit Grid** from the pop up (context) menu.
5. Continue with editing operations.

Creating a Grid for a Pattern Flood Model

Builder can help you to quickly create a Cartesian (*VARI) grid for modeling a pattern flood problem. This option allows you to select one of the available pattern types. After you enter the grid geometry values (area, depth, thickness, block size, and so on), a grid with properly placed producer and injector wells is created. The area and volume modifiers for the corner and edge well blocks are calculated automatically to model the flow at boundary.

This dialog box can be accessed from the **Reservoir** menu, by choosing the **Create Grid** and **Quick Pattern Grid** menu item.



You start by choosing a desired pattern from the **Pattern Type** combo box. Normal pattern places the producer in the center and injectors at the corners and edges. Inverted patterns reverse the placement of producers and injectors. 1/8 and 1/6 patterns are the minimum symmetry elements. The line drawing at the bottom shows the selected pattern.

Next, if you are working on a STARS dataset, you may check **Use nine-point spatial discretization** to turn on this grid option. This option is not available for IMEX or GEM.

Now specify the pattern dimensions - area, thickness, top and dip angles. The pattern area can be specified by the surface area of the pattern, by the well spacing or by specifying the width and height. (The last option is not available for seven-spot patterns.) For the grid block size, you only need to enter approximate thickness and size and the program will calculate the actual exact values to create a grid of the specified area. Once all values are entered, click **Calculate** to show the actual values that will be used to create the grid.

You must specify the depth to the top of the middle of the pattern. In addition, you may specify a non-zero dip angle in the X or Y directions. If you do so, the depth of the tops of the grid blocks will be adjusted to account for the dip angle.

Click **OK** to exit the dialog box and you should see a Cartesian grid created with the injectors and producers placed at the center of proper grid blocks.

The volume and area or transmissibility modifiers keywords are automatically created. Refer to VAMOD and VATYPE keyword description if you are using STARS simulator, and VOLMOD, TRANI, TRANSJ and TRANSK keyword descriptions if you are using IMEX or GEM simulator.

Well perforations will also be created. The entire thickness of the reservoir will be perforated. If the wells are only to be perforated in part of the thickness, you will need to edit the well perforations.

You can save the dataset now, but it is not ready to be used with the simulator. You still have to create well constraints, porosity, permeability, rock-fluid properties, initial conditions and other values as required by the simulator. Refer to other sections of the manual for instructions on entering these values.

Creating a Non-orthogonal Grid with Sloping Faults Using Structure Maps

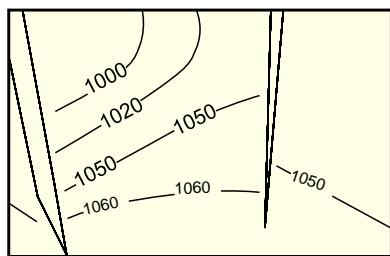
When generating conventional non-orthogonal corner point grids (vertical blocks), the x, y values used for each grid block corner are assigned independent of the top and thickness of the reservoir. The z values are interpolated from the contour maps at each x, y coordinate. Thus, general grid structure is defined starting with a flat plane and assuming some arbitrary constant thickness for each layer. Values of elevation can be computed at a later stage.

Where sloping faults are needed, however, the x, y coordinates for the grid will change as you progress down the fault through the grid. Consequently, the x, y location needed to determine the z value of a particular grid block corner would depend on the location of the grid block above. Since the grid structure is so tightly defined by the actual surface and layer thickness, grid structure cannot be predetermined without input data defining elevation and thickness.

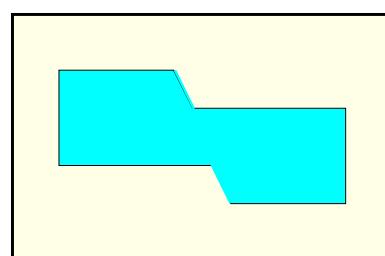
Contour Maps and Sloping Faults

Normal Faults (or dropped faults) are the most common. These faults are typically expressed on a contour map with two dashed lines - one line for each edge of the separated surface. These paired lines are often referred to as bifurcating fault lines. The slope of the fault is implied from the separation of the lines and the map elevation contours on either side of the fault.

Figure 1: Normal (Dropped) Faults



a) Aerial View (Contour Map)



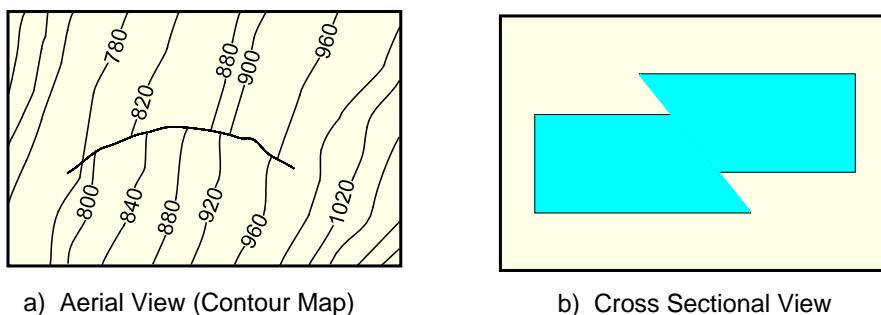
b) Cross Sectional View

In theory, a normal sloping fault grid could be constructed from data obtained from a single surface contour map (together with a specification for thickness). However, the distance between the two surfaces separated by the fault, is typically very small relative to the scale of the contour map. Hence, the resulting fault face may not accurately reflect the geology of the reservoir. Errors can be introduced when contour and fault data are digitized or during the process used to permit grid block boundaries to follow fault lines.

If maps are used for both upper and lower surfaces, the resulting sloping fault is more likely to be accurate. Note that there may still be some discrepancy between the fault slope implied by the bifurcating fault lines on each map and the relative displacement of these fault lines between the two maps. Consequently, the two surfaces separated by the fault may not be tangent to one common fault face. Adjustments are necessary to correct blocks at the fault face.

Thrust Faults are much less common. These faults are more difficult to determine from a contour map. Since the edge of one surface hangs over the other, the edge of the upper surface is normally drawn to indicate the fault. The only way to distinguish a thrust fault from a simple vertical fault is by examining a second contour map for a lower horizon of the reservoir. If the fault line is displaced between the two maps, then there is a sloping thrust fault. The degree of slope is determined by the degree of displacement of the fault lines, together with the difference in the contours between the two maps.

Figure 2: Thrust Faults



a) Aerial View (Contour Map)

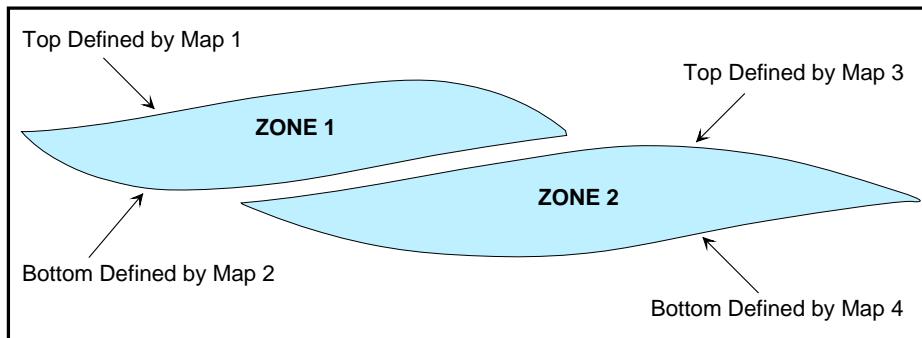
b) Cross Sectional View

To construct a grid for a reservoir having thrust fault lines, two maps are required. Since the upper surface is displayed on the maps, the upper surface would define the fault face. Values for thickness and elevation for blocks along the lower surface can be determined for the visible part of the block and must be estimated for the part of the block that is hidden by the thrust fault surface.

Steps for Constructing Sloping Fault Grids with Builder

Builder requires that you divide the reservoir into distinct horizontal zones. The elevation and thickness for each zone is defined by an upper and a lower contour map. Since a zone is defined by two contour maps, thickness throughout the zone can be variable. Each zone is divided into a number of layers. The thickness of each layer is initially a fraction of the total thickness of the horizontal zone. However, after grid creation, thickness can be specified and calculated for each layer. Zones may or may not be in contact with each other - this completely depends on the contour maps used.

Figure 3: Sample Reservoir with Two Distinct Zones



To construct a grid, you define a set of control points breaking the reservoir into regions separated by control lines. This is similar to the process used in the current non-orthogonal corner point wizard. However, control lines can be separated to conform to bifurcating fault lines.

Once the control points are defined, a duplicate set of control points is assigned to the top and bottom of each horizontal zone. You examine each contour map in turn adjusting the corresponding set of control points to follow the fault lines. As you descend through each zone, the control lines from the bottom of the previous zone are used as the starting values for the top of the next zone. This will assist you in maintaining grid block alignment where the two consecutive zones are tangent.

Since the process of defining the sloping non-orthogonal corner point grid results in a grid that is structurally complete (that is, both top and thickness are defined), separate specification and calculation of structural properties is not required. The “Specify” and “Calculate” functions will exclude Grid Top, Grid Bottom, and Grid Paydepth properties for this grid type. Grid Thickness will still be available; however, the total thickness within a reservoir zone will be defined by the top and bottom maps specified during grid construction. Calculated thickness values will be used to determine the proportional subdivision of the zone into layers.

As mentioned earlier, should user control lines or map data result in overlapping or illogically disjoint fault blocks, Builder will make adjustments to the grid whenever possible.

Note: Due to errors in data, errors in defining control points, inconsistencies between bifurcation control lines and top/bottom maps, grid faces on either side of a fault line may not meet when the grid is constructed. Builder will make adjustments to grid blocks along faults defined by control lines so these blocks meet up along a common fault face.

Step By Step Instructions

1. Starting the non-orthogonal corner point wizard for sloping fault grid creation:

To start the non-orthogonal corner point grid creation wizard, select **Non-orthogonal Corner Point** from the **Reservoir** menu. Note that the wizard will always display four basic navigation buttons: **Prev**, **Next**, **Cancel**, and **Help** on the right hand side of the toolbar at the top of the window. Click **Help** at any time for detailed instructions on the current step.

Once the wizard has launched, click the check box labeled **Sloping Faults**. Click **Next** to proceed to the next step. The wizard will remind you to open an initial contour map if you have not done so already.

2. Defining four outer boundary lines:

At this stage of the wizard, it is necessary to define four outer boundary lines defining the grid. Begin with the top most boundary line. Click the mouse at the appropriate location on the contour map to specify control points along the top most boundary line. As the mouse is clicked, points will be drawn and joined to form a line. When you have finished defining the top most line, click the toolbar button labeled **Next Line**. Continue to define points for the right most, bottom most, and left most boundary lines. When you have completed the last line (the left most line), click the **Next** button in the wizard toolbar to proceed. The wizard makes the final connection required to connect the top most line with the left most line before updating the wizard toolbar for the next step.

3. Defining internal control lines:

At this stage of the wizard, it is necessary to define internal control lines. Internal control lines are used to indicate faults within the area defined by the four outer boundary lines. To begin defining an internal control line, click the button labeled **Add Internal Line**. To define a control line that is parallel to the left and right most boundary lines start by clicking points near the top most boundary line. To define a control line that is parallel to the top and bottom most boundary lines, start by clicking points near the left most boundary line. Click the mouse at the appropriate location to define the first point. After the second point is clicked, the wizard will join the new internal line to the top or left most boundary line. Each new point will be connected to the previous point to form the control line. The final connection to the right or bottom most boundary line will be done by the wizard when you begin defining another line or when you move on to the next wizard step. Click **Add Internal Line** to define another internal line or click **Next** to proceed to the next wizard step.

4. Specifying reservoir zones and modifying control lines for bifurcating faults:

There are two tasks to complete at this stage of the wizard. First, you must enter the number of distinct reservoir zones in the text box labeled **Reservoir Zones**. Second, you must edit internal control lines to conform to bifurcating faults.

‘Bifurcating faults’ refer to fault lines on the contour map representing normal (or dropped) faults. These fault lines are typically drawn on the contour map as a pair of dashed lines representing the upper and lower side of the fault. An internal control line can be separated to conform to these paired fault lines. Click the mouse on a control line intended to conform to a bifurcating fault. The wizard will indicate that the line has been selected for editing by highlighting it in red. Then click and drag individual points along the control line to separate the control line. Once the control line has been separated, each side of the control line can be selected for editing points. Click **Next** to proceed.

5. Indicating the number of divisions for each grid region:

At this stage of the wizard you must indicate the number of grid divisions required between each set of control lines. The region between each set of control lines can be highlighted by using the wizard toolbar buttons labeled **Previous Section** and **Next Section**. Two opposing line segments are highlighted to show the active section. Visit each section in turn, and specify the number of grid divisions by entering a number in the text box labeled **Divisions**. Click **Next** to proceed.

6. Defining contour maps, zone divisions, and adjusting control points for each map:

This is the most involved step of the wizard. Several tasks must be completed for the top and bottom of each reservoir zone. The up/down arrows in the wizard toolbar are used to navigate from the top to the bottom of each zone in turn. In the previous steps of the wizard, you defined a set of control points that will be used as a template for a set of control points for the top and bottom of each reservoir zone.

As you navigate down through the reservoir, start by defining the contour map used to define that surface of the reservoir. Click **Define Map** to specify the contour map. Then indicate the number of k-layers required in the current reservoir zone by entering a number in the text box labeled **K-Layers**. Finally, proceeding through the reservoir, adjust control lines to conform to the fault lines of the contour map at that level. Note that further separation of control lines defining bifurcating faults cannot be done in this step (the next step will permit the selection and editing of control lines on either side of bifurcating faults). Click **Next** to proceed.

7. Making further adjustments to control lines for bifurcating faults:

This stage of the wizard allows you to make further adjustments to control lines defining bifurcating faults at any level of the reservoir. The up/down arrows on the wizard toolbar can still be used to navigate through the reservoir. If you wish to adjust a control line, highlight the line by clicking any part with your mouse. Then click and drag control points to make adjustments. Click **Next** to proceed.

Note that a number of advanced grid construction options can be selected at this stage of the wizard. Click the **Advanced** button in the wizard toolbar to view these options. The default settings are recommended for construction of typical sloping fault grids.

8. Creating the grid, adding grid block refinements:

As you enter this stage of the wizard, the initial grid is constructed. Before selecting **Finish**, you make grid block refinements. To refine a number of grid blocks, select the grid blocks in question by clicking and dragging your mouse over the grid blocks in question. Then select **Cartesian refined** from the **Reservoir** menu. Click **Finish** when you are ready to exit the wizard.

Useful Tips

- Orient the boundary lines such that the majority of the reservoir faults are perpendicular/parallel to the four boundary lines. This will minimize non-orthogonality resulting from the internal control lines drawn to conform to the faults.
- If you find that the resulting grid is significantly non-orthogonal due to meandering control lines (highly irregular faults) – use additional control lines on either side of the fault in question. If these control lines become straighter as you move towards the boundary, fewer irregular blocks are likely.
- Block faces gradually verticalize as you move from the sloping fault to the next control line. Add control lines, on each side of a sloping fault control line, to control how rapidly the blocks become vertical.
- Navigate back through the wizard to make adjustments to control lines if grid structure is unsatisfactory. Once the grid has been constructed, it is not too late to make changes. You can enter the wizard from the last step by selecting Edit Grid. The wizard will warn you if any information will be lost by navigating backwards. To navigate backwards, click the button labeled **Prev**.

Importing 3D Simulation Grids and Grid Properties

You can import a simulation grid and grid properties in two ways:

1. From files output by a geological program.
2. From another CMG dataset file.

Importing from File Output by a Geological Program

There are two ways of importing 3D static reservoir models, created by geological modeling programs, into Builder. First, you can import data into Builder using the RESCUE model exchange format. Second, several geological modeling programs have “Export to CMG” functions, which write out the CMG grid and grid property keywords in ASCII files.

The first option is usually preferred, as more information is included in a RESCUE model, and the import is easier. Builder has a wizard to lead you through the importing of a RESCUE model. The wizard will start up automatically if you open a RESCUE model (from the **Reservoir** menu, select **Open RESCUE model**). For more details, see [Importing Simulation Grid, Properties and Well Locations from RESCUE Model](#).

A number of geological modeling software packages can export to CMG ASCII format (simulator input format), complete with keywords. You may need to specify the output units to use, as well as other items, depending on which software you are using. This manual contains some specific instructions for exporting from Petrel. See [Importing Partial Datasets from Petrel](#).

Once you have imported the grid (and properties), limited grid editing capabilities are available. You may add refined grids, split grid planes, extract a sub-model, and combine layers (uplayer). To find out more about each of these operations, see the relevant manual sections, such as [Adding Refined Grids Using the Refinement Wizard](#).

Importing Simulation Grid, Properties and Well Locations from RESCUE Model

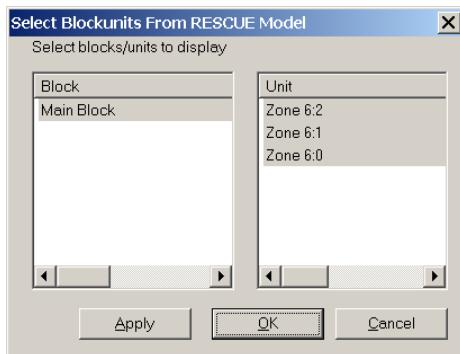
RESCUE is a Joint Industry Project managed by the Petrotechnical Open Software Corporation (POSC). The acronym 'RESCUE' stands for *REServoir Characterization Using Epicentre*. At its inception the purpose was to provide a forum for the development of an open standard for the transfer of data from geomodels to upscalers. A RESCUE format model can define "block units" (formed by 3D surfaces representing horizons, boundaries and fault surfaces), 3D grids, and properties such as porosity, permeability, and so on. Fault surfaces, wellbore trajectories, and well logs can also be included in the model.

Using Builder you can import a "global grid" from a RESCUE model, or the all the units of a single RESCUE block.

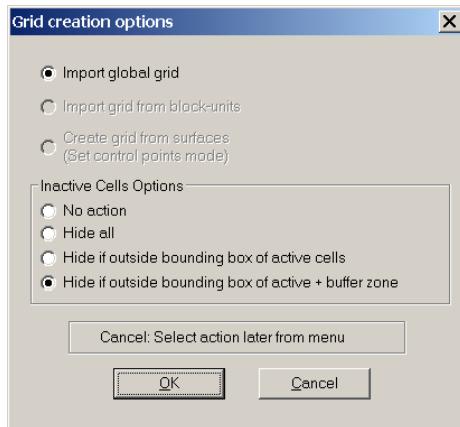
To import a three-dimensional grid saved in RESCUE model:

1. Select **Open RESCUE file** from the **Reservoir** menu. An **Open RESCUE file** dialog box will be displayed.

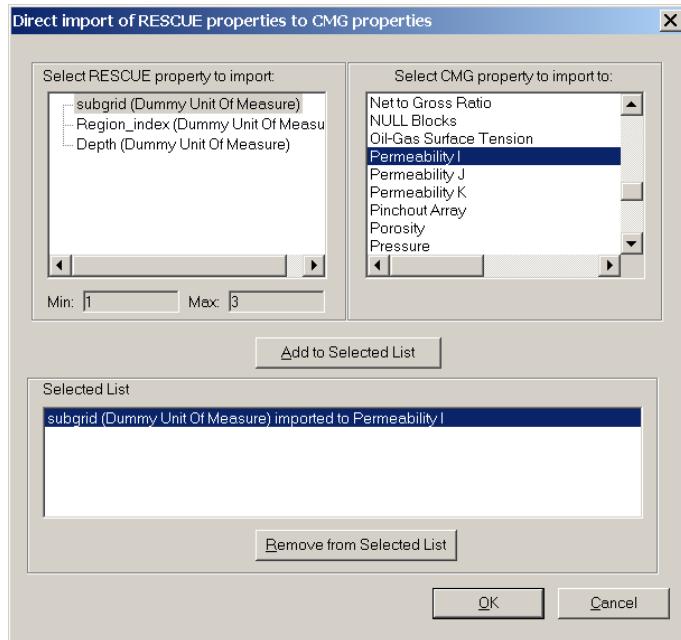
After you select the file to open, the main file will be read, and then a **Select Blockunits From RESCUE Model** dialog box will be displayed. Select the geological units that you wish to import. By default, all units will be selected. After you have made your selection, click on **OK**.



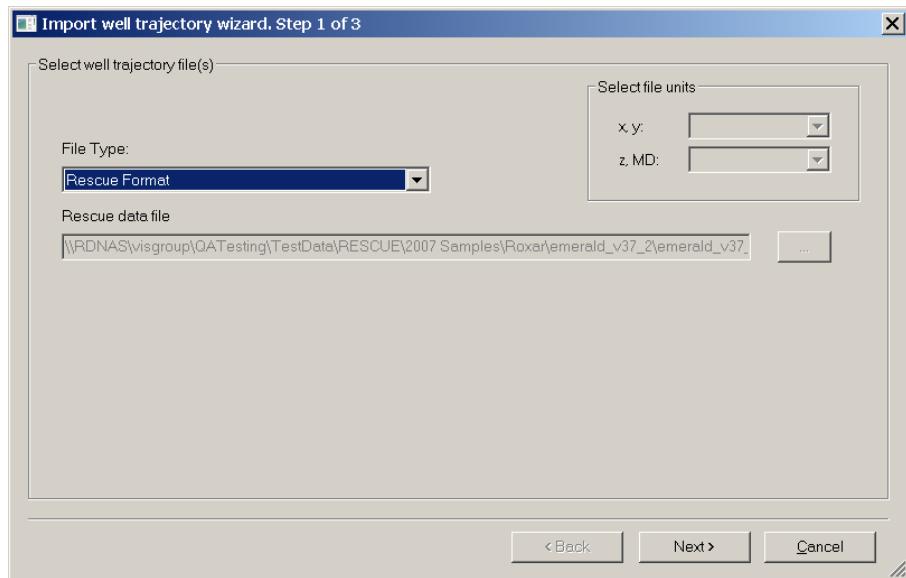
- The **Grid creation options** dialog box will be displayed. Usually it is best to accept the default option selected. Most RESCUE models allow the importation of a “global grid” or a “block unit grid”. The *Inactive Cells Options* is used to avoid drawing unused cells. The following discussion assumes that one of these was selected. Click **OK**.



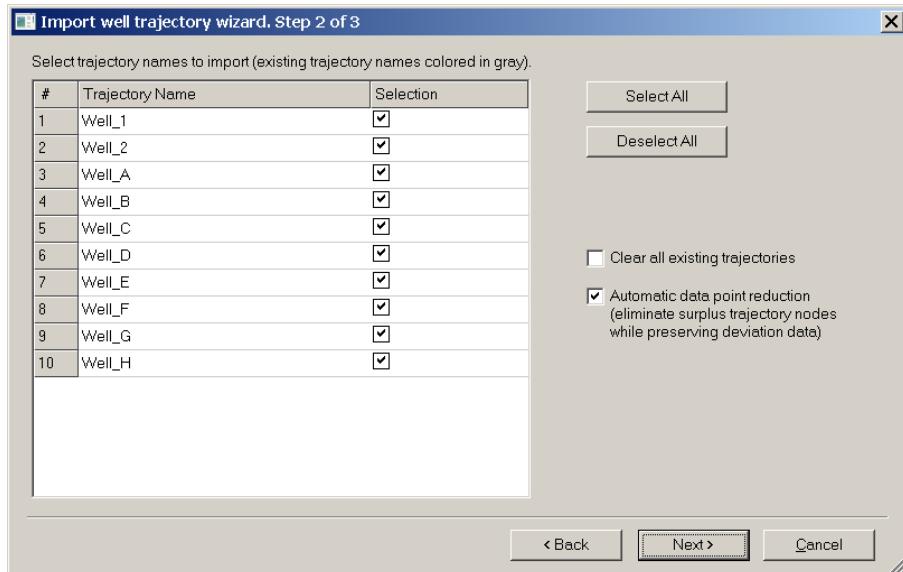
- If the grid in the geological model has the lower most layer as K=1, you may be asked if you wish to renumber the layers so that K=1 is the topmost layer. You can answer *Yes* or *No* to the question. If you answer *No*, then in your simulation K=1 will be the lower most layer, and K=nk will be the upper most layer.
The grid data will be read from the RESCUE file, and the grid drawn.
- If the RESCUE model contains grid properties, such as porosity or permeability, a **Direct import of RESCUE properties to CMG properties** dialog box will display. Select one item from the list of RESCUE properties, and the corresponding CMG property name, and then click **Add to Selected List**. Once you have added all the properties you wish to import, click on **OK**. If you did not want to import any properties at this stage, click **Cancel** on the **Direct import of RESCUE properties to CMG properties** dialog box. You can import properties later by selecting **Direct import of RESCUE property** under the **Reservoir, Advanced Rescue** menu.



5. If the RESCUE model contains well trajectories, the **Import Well Trajectory Wizard Step 1 of 3** dialog box will be displayed. The format and data file name will be filled in. Click **Next**.



- The **Import well trajectory wizard, Step 2 of 3** dialog box displays a list of all the well trajectories available in the RESCUE model. Click the check box under **Selection** to clear any wells you do not wish to import. Click **Next**.



- The **Import well trajectory wizard Step 3 of 3** dialog box is displayed. Edit the contents of the table as appropriate then click **Finish**. The data in the RESCUE model is imported into Builder.

Note: When you save the dataset, the RESCUE filename, names of selected blocks and units, specification of the imported properties, and any imported well trajectories are saved in the dataset. This information is available the next time the dataset is read back into Builder.

Importing Partial Datasets from Petrel

You can export partial datasets from *Petrel* in a format suitable for CMG simulators. However, some special care needs to be taken to make the export successful.

To export from Petrel, once you have created the simulation model:

- Under **Project**, select **CMG Export Settings**. In the **CMG Export Settings** dialog box, change **Undefined Property Value** to 0.0. Click **OK**.
- In the Petrel Explorer, under the **Models** tab, click **Simulation Model to Export**. Right-click then select **Export**. In the **Save As** dialog box, change **Save as type** to **CMG grid**, enter a file name such as *my_sim.dat* then click **OK**.
- The **CMG ASCII Export Grid** dialog box is displayed. You can select which properties you want to export with the grid. Typically, porosity, permeability, and net to gross are exported. After you have made your selection, click **OK**.
- The **Export Units** dialog box will be displayed. Change the **Output XY** units and **Output Z** units so that they are in the same units (that is, *m* for simulator SI units, *ft* for simulator Field units), then click **OK**.

Note: You will not be able to correctly run the simulation unless these are the same.

5. With Petrel v. 3.2 (Feb. 16, 2001), the output CMG keyword for the net to gross property is incorrect. If you have included the Net/Gross property in your export, open the exported file in an editor and search for “NET/GROSS”. Replace this keyword with the correct CMG keyword “NETGROSS”.
6. Petrel does not currently indicate the output units. While you have the dataset open in the editor, add the following line at the top of the file if you export with distances in feet.

*INUNIT *FIELD

7. If you edited the file, save it and exit.

To bring the exported data into Builder, follow these steps:

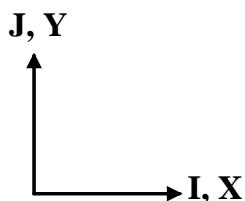
1. Start Builder for an existing dataset (if you just want to replace the grid and properties with those exported from the geological model), or for a new case (you will need to specify the simulator and units).
2. From the **File** menu, select **Import from another file**, then select **Grid and Spatial Properties from dataset**.
3. Select the file that was imported from Petrel (or other geological program) to read the grid and properties from. You will be asked to select which CMG simulator (IMEX, GEM or STARS) you want to create a dataset for. Next, you will be given a list of the properties available in the file, and asked to select which you want to import. After the file reading is complete, proceed with adding other input data required by the simulators.

Resolving Problems with Axis Direction in Imported Grids

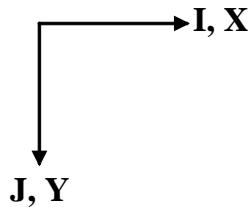
If you import a simulation grid from a non-CMG simulator Builder and Results 3D may display the grid “flipped” or “mirrored” from the way you expect it to appear. If necessary, select **Flip grid in I direction** and **Flip Grid in J direction** under the **Reservoir** menu. Use these menu items to flip or mirror the grid until it appears the way you expect it.

Using these menu items will accomplish the same as modifying the RESULTS AXES-DIRECTIONS keyword in the data file. The following discussion is gives more details on the axes direction problem.

Traditionally, reservoir simulation has used a coordinate system as follows (2D map (areal) view)
KDIR UP (k=1 is bottom layer) (possible for CART and VARI grids)

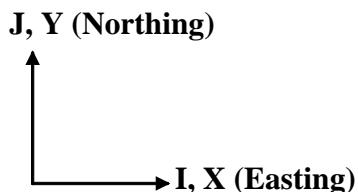


KDIR DOWN (k=1 is top layer) (possible for CART, VARI, and CORNER grids)



We refer to this as “traditional simulation grid coordinates” or “grid coordinates” for short.

In the past few years, geological modelling programs have used coordinate systems, such as UTMs, where the axes directions are:



We refer to this as “reservoir map coordinates” for short. Almost all maps are now created with these coordinates. When a grid is created by a geological modelling program directly, it is most often created in reservoir coordinates.

To display the reservoir correctly, Results 3D and Builder need to know whether “traditional simulation grid coordinates” or “reservoir map coordinates” are being used for the grid block corners in the simulation. Results 3D will translate “traditional simulation grid coordinates” into “reservoir map coordinates” so that the display on the screen will always be in “reservoir map coordinates”, as this is now the way that most geologists and engineers expect the display.

For a number of years, Results has tried to detect if reservoir map coordinates are used, and to ask the engineer for confirmation with a question box. With new versions of Builder, the simulators and Results 3D, the information is stored in dataset and SR2 files, to avoid asking the question.

Builder may add a line in the dataset to indicate that the grid is in reservoir map coordinates.

```
RESULTS AXES-DIRECTIONS 1. -1. 1. ** grid in reservoir map  
                           coordinates  
RESULTS AXES-DIRECTIONS 1. 1. 1. ** grid in traditional simulation  
                           grid coordinates
```

If you are using a dataset where the grid was created with a version of Builder prior to 2002.10, or if for some reason the keyword was deleted, or if you have imported a grid produced by some other means, adding the correct one of the above two lines may solve your problem. This line is added immediately after the RESULTS XOFFSET, YOFFSET, and ROTATION keywords (if they exist) and before the simulator GRID keyword in the dataset.

When the above feature was added, we maintained backwards compatibility, so older datasets and SR2 files would act as before. If the display in Results is incorrect, you may need to add the keyword, then re-run the simulation.

If running the simulation takes a long time, you may wish to try the following.

1. Save a backup copy of the simulation output SR2 files (*.irf, *.mrf).
2. Open the *.irf file in a text editor.
3. Remove the line with "INDEXED" or "INDEX64", if it exists. If the line exists it will be the first line in the file. Remove the entire line.
4. find the following lines

```
GRID-XOFFSET      0.0000
GRID-YOFFSET      0.0000
GRID-ROTATION     0.0000
```

Immediately after the lines, add the following line:

```
GRID-AXES-DIRECTIONS 1.0 -1.0 1.0
```

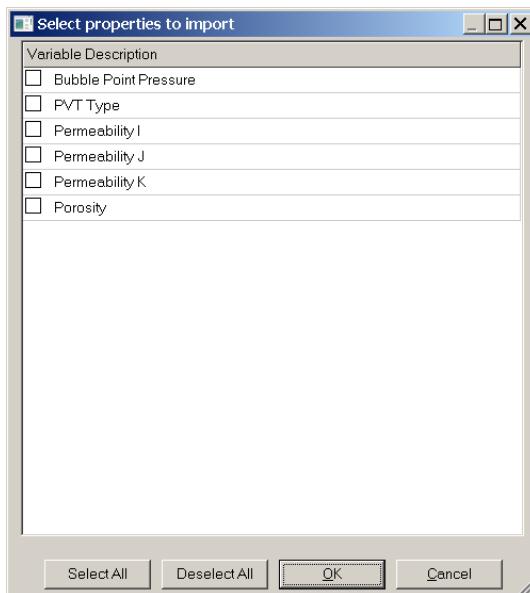
where the signs of the 1.0's match those in the RESULTS AXES-DIRECTIONS keyword used in the dataset.

5. Save the file then try opening in Results 3D.

If the RESULTS AXES-DIRECTIONS is in the dataset, but not in the *.irf (after the simulation is re-run), let CMG know, and send us the dataset and the simulator version that were used. If the keywords are in both the dataset and the simulation SR2 file, but the displays are still different, also send CMG the dataset.

Importing Grid and Grid Properties from a CMG Dataset File

From the **File** menu select **Import from another file** and then **Grid & spatial properties from dataset**. Specify the dataset file from which to import the grid. Builder will import the grid and then display a dialog box listing the importable properties in the source dataset:



Structural properties (Grid Top, Grid Thickness, and so on) are not importable. Those properties that are supported by the simulator in the source dataset but not by the simulator in the target (current) file are not importable.

If a grid already exists in the current dataset, it and all the existing grid-dependent information (spatial properties, wells, and so on) are deleted.

Grid and property values are recalculated to take into account different unit systems.

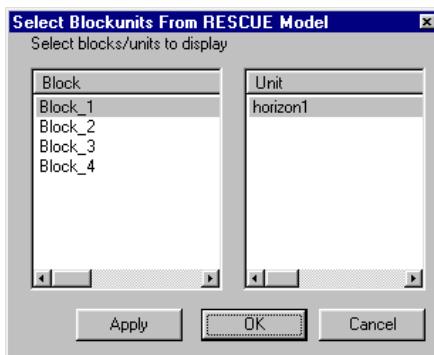
Builder also imports all the metadata – RESULTS keywords – that it stores in the dataset to facilitate restoring these objects.

Creating a Simulation Grid Using 3D Surfaces from RESCUE Model

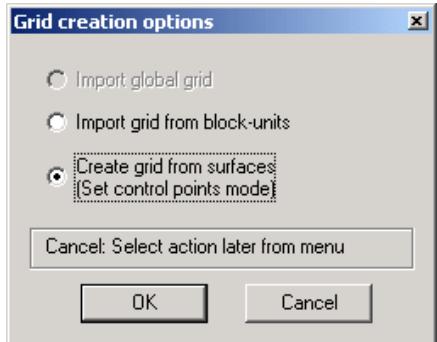
It is possible to create a grid from the horizon and fault surfaces in the RESCUE model. However, CMG's current grid creation algorithm is limited in its ability to handle complex faulted systems, which limits the usefulness of this feature. In addition, this feature cannot honour RESCUE's interior (fault) surfaces. Therefore, use this feature only if the RESCUE model is designed such that each block unit is bounded on the sides by fault surfaces.

To construct a simulation grid in volumes represented by RESCUE block units:

1. Start Builder for a new case as explained in [Starting Builder](#).
2. Select **Open RESCUE File** from the **Reservoir** menu. Select and open the file from the **Open RESCUE File** dialog box.
3. The **Select Block Units from RESCUE Model** dialog box is displayed. Select the blocks and units you want to work with, and then click **OK**.



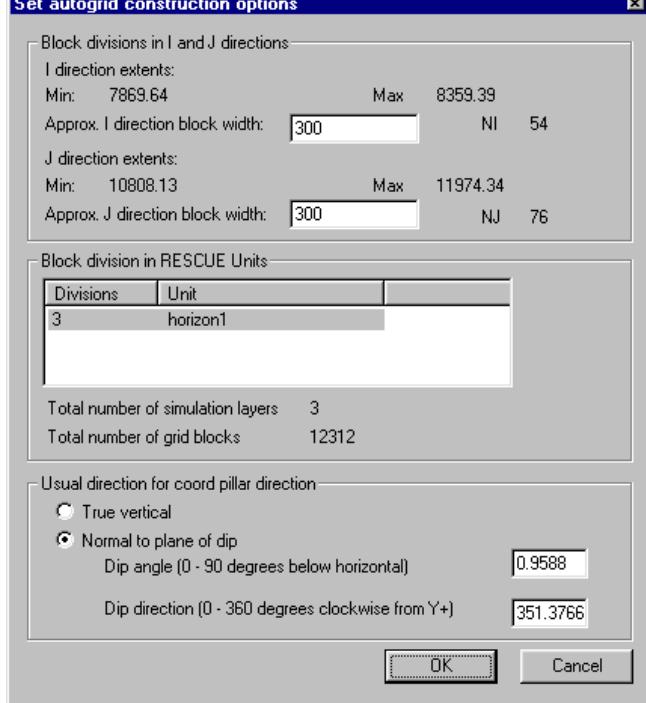
4. The **Grid creation options** dialog box is displayed. Make your selection and then click **OK** or click **Cancel** to select action later from menu.



- Builder displays the top horizon surfaces of the topmost block units in the XY plane. You can change the view type to **3D View** to inspect the model as described in [Overview](#).

Note: If you leave the control points mode at any time, you can return to it by selecting **Set control points mode** from the **Reservoir | Advanced RESCUE** menu.

- Left-click four control points to define the extent of the grid in the XY plane. The points do not have to fall on the surfaces. Grid blocks that are not located in any of the block units are set to inactive (NULL).
- The **Set autogrid construction options** dialog box is displayed:

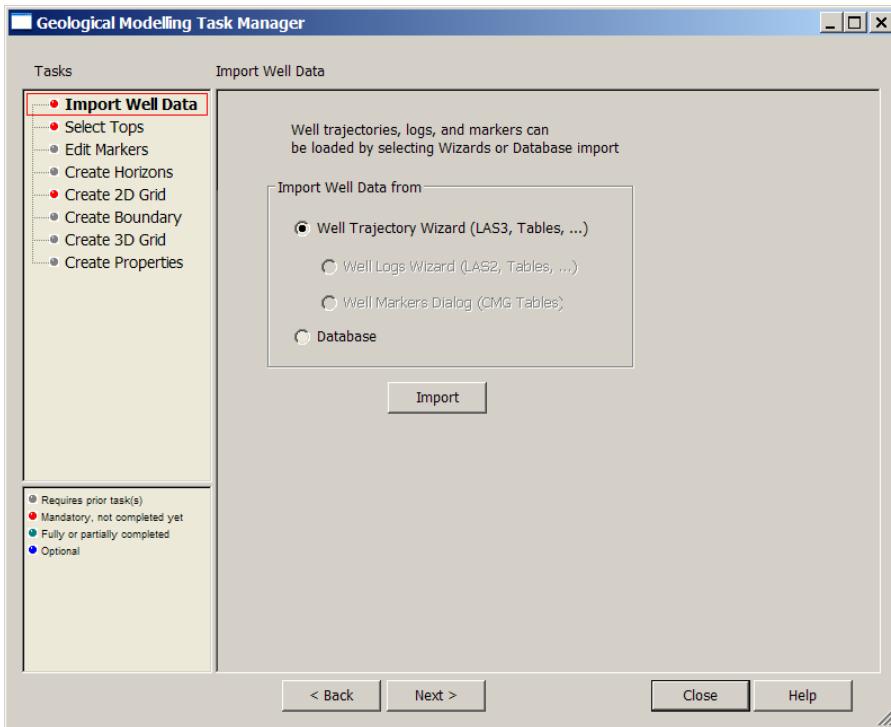


8. Set the block widths in I- and J directions. As you enter these values, Builder calculates the number of grid blocks in the I and J directions (NI and NJ).
9. Set the number of divisions in each of the selected units by double-clicking the value in the **Divisions** column in the list control. Builder will calculate the total number of simulation layers as you enter these values (NK).
10. Select the “usual” direction of the “coord pillars”. These pillars represent the line segments joining the grid nodes with identical (I, J) values in the topmost horizon and the bottommost horizon. The direction you choose will be used for all the pillars that need not be rotated to honour a fault surface. The pillars along fault surface will be adjusted automatically.
11. Click **OK**. Builder displays the mesh created using the NI and NJ values calculated in step 7. If you wish to change the control points and/or the information input in steps 7 – 9, select **Set control points mode** again and repeat steps 5 onwards.
12. Select **Construct Grid from surfaces** from the **Reservoir | Advanced RESCUE** menu. Builder will create the simulation grid and display it in the **IJ View**. Note that at this stage, Builder is in **Edit Grid** mode. You could add grid block refinement at this stage. You could also add refinement later as explained in [Editing an Existing Simulation Grid](#).
13. Click the **Probe mode**  icon on the modes toolbar to exit the **Edit Grid** mode.

Note: When you save the dataset, the RESCUE filename, names of selected blocks and units and the control point locations are saved in the dataset. This information is available the next time the dataset is read back into Builder.

Building Simulation Grid Using the Geological Modelling Task Manager

It is possible to create a grid from well data when well markers are available from geological interpretations of the reservoir tops. The Task Manager, found under **Reservoir | Build Static Model with Task Manager** menu item, can be used for building a grid and populating static properties on it in one integrated workflow. It also allows for building grids with pre-existing structure meshed maps of the reservoir tops. The **Geological Modelling Task Manager** dialog box is shown below.



The Geological Modelling Task Manager is designed as a wizard dialog capable of guiding you through the main tasks for building simple geological grids. The different tasks are presented as a vertical tree on the left window pane. Selecting a task will change data controls and options seen on the right window pane. The **<Back** and the **Next >** buttons found at the bottom of the dialog box allow you to navigate forward (down the task tree) or backward (up the task tree) in the workflow. Some tasks are dependent on other tasks and cannot be accessed before the other tasks are completed. Some tasks are always required and some are optional.

The tasks are color-coded to help you visualize the status of the workflow, as follows:

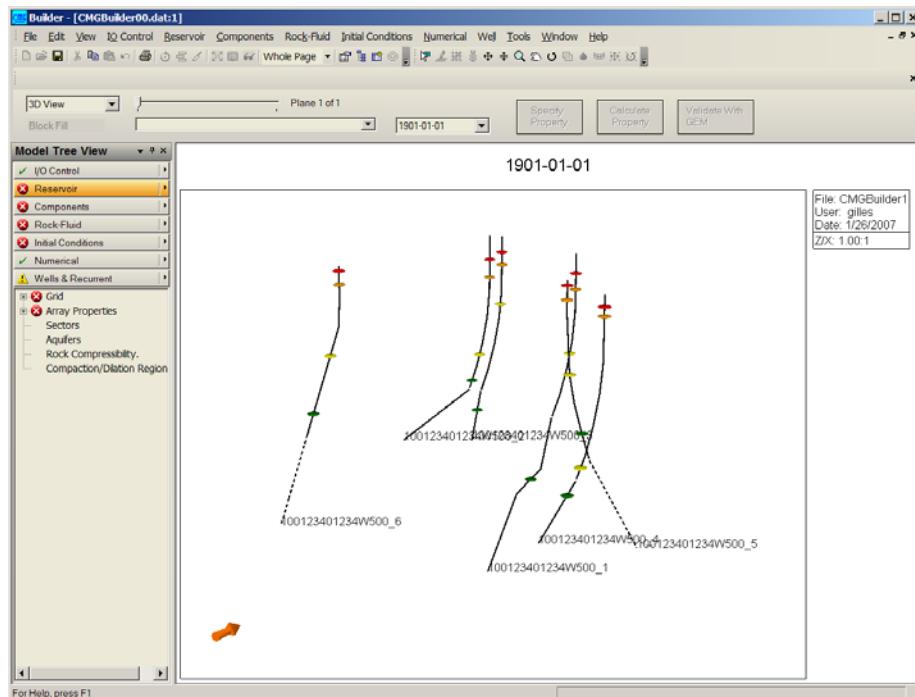
| Colour | Meaning |
|--------|--|
| Red | Required task |
| Blue | Optional task |
| Grey | Task is dependent on a not-yet-completed prior task (task that is higher up in the tree) |
| Green | Task is completed |

Navigation in the workflow can also be performed by directly selecting the task with a mouse click without the need for using the **<Back** and the **Next >** buttons. This allows you to jump directly forward or backward to a specific task in the workflow. Builder's views (IJ-2D Areal or 3D View) will be automatically updated with new data or objects after each task is completed. The different tasks are described in the following sections.

Import Well Data

The Import Well Data task is the first task to be completed (see previous picture) if no well data are already loaded in Builder. This is where well trajectories, well logs, and well markers can be imported from different data sources. Various file formats and an optional PPDM database connection is available for data import. The well trajectory wizard (see [Well Trajectory Import](#)) or the database wizard (see [Importing from a PPDM database](#)) can be launched by selecting the appropriate option and clicking the **Import** button. The [Importing Geological and Well Trajectory Data](#) chapter describes various file formats supported by Builder.

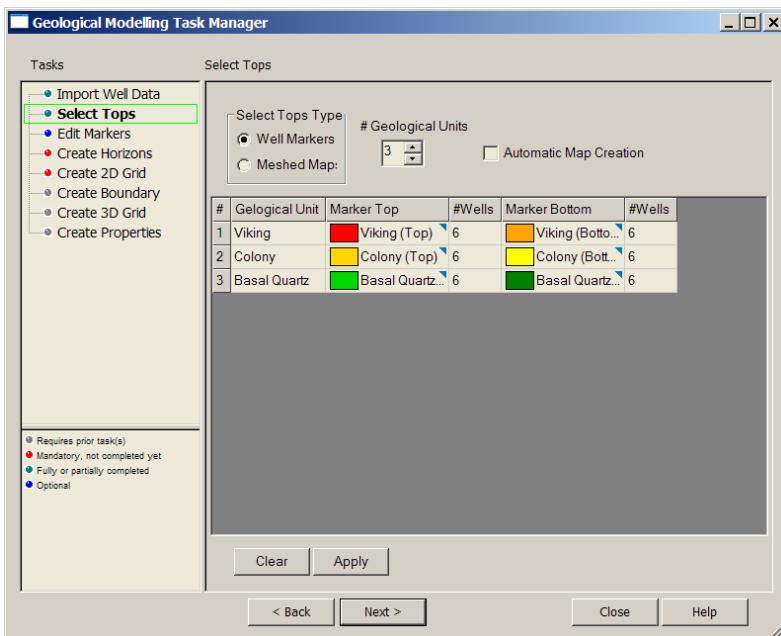
After this task is completed, well trajectories and well markers will be displayed in the Builder 3D View as shown in the following example. Note that the **Geological Modelling Task Manager** dialog box is a modeless dialog and you still have controls on the Builder's options for controlling the view and changing the display properties. When completed, the colour of the Import Well Data task will change to green to indicate its new status.



Select Tops

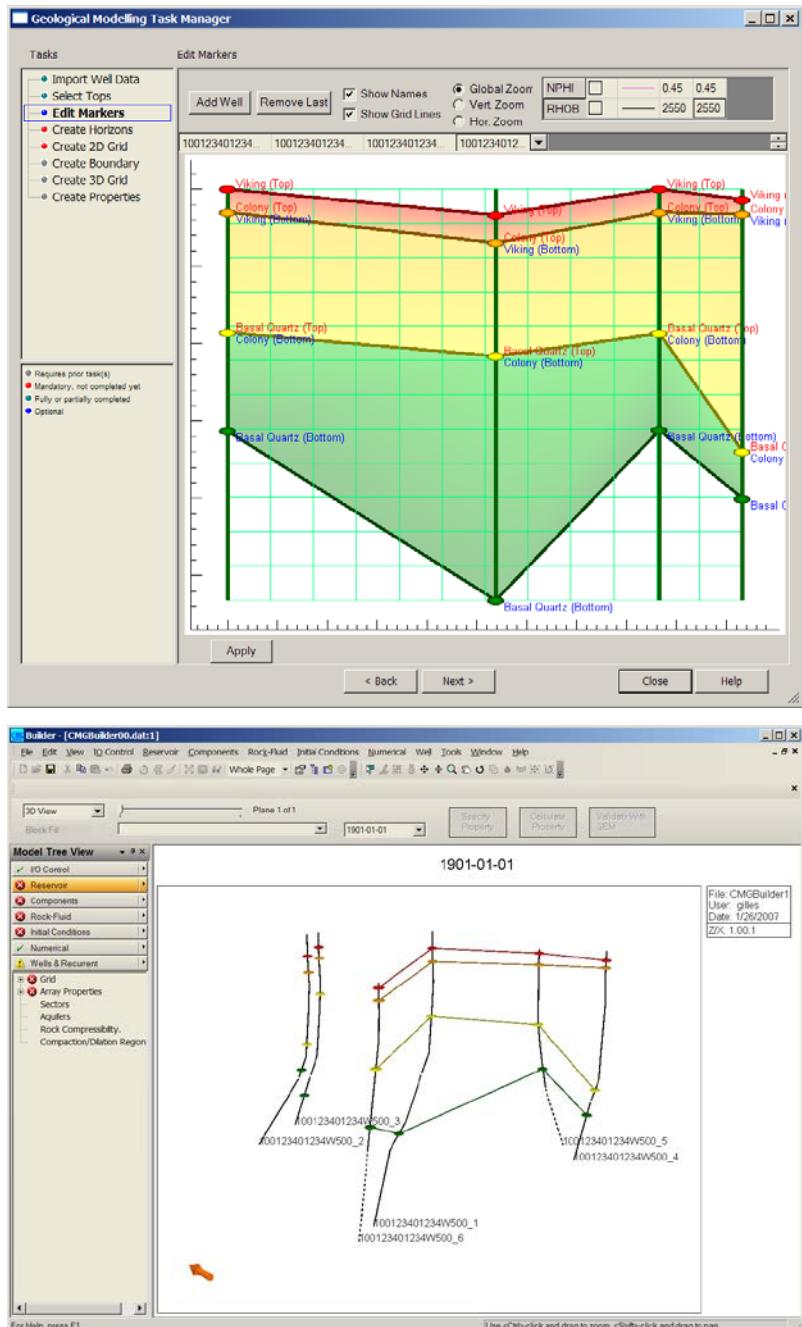
The Select Tops task is the second mandatory task to be completed (shown in the following screen). If no well markers are available from the wells, only the Meshed Maps option will be available. You can set the number of geological units you want to include in the grid. You need to select a marker for the top and a marker for the bottom of each geological unit.

Alternatively, with the Meshed Maps option you need to select a pair of maps instead of a pair of markers. You can combine elevation/depth maps with thickness maps. When markers are available in a relatively small number, they are automatically selected (as shown). You can name the geological units to match the markers pair name or give it a name of your choice such as Pay Zone for example. The number (#) of wells column indicates the number of wells associated with the selected marker is present.



Edit Markers

The Edit Markers task is optional and only available if markers are selected instead of structure meshed maps. You are building a fence diagram when using this option. You can use the **Add Well** button to add wells in the fence diagram. As wells are selected, they are connected by lines joining the markers. The connection lines are also displayed in the 3D View of Builder as shown in the following examples.

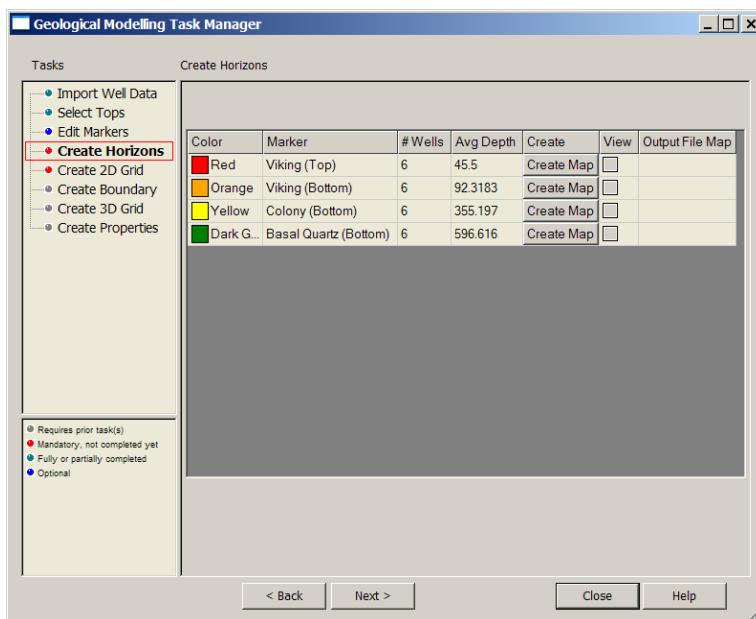


On the Task Manager, the Edit Markers view is a projection of the wells on a vertical plane. The well locations are determined by their surface locations and they appear as vertical lines. Therefore, the fence diagram is useful for comparing TVD values for markers at the selected wells. If a marker appears to be miss-located, it can be moved up and down from its position along the well path. You can select any marker on the fence diagram with a mouse click and

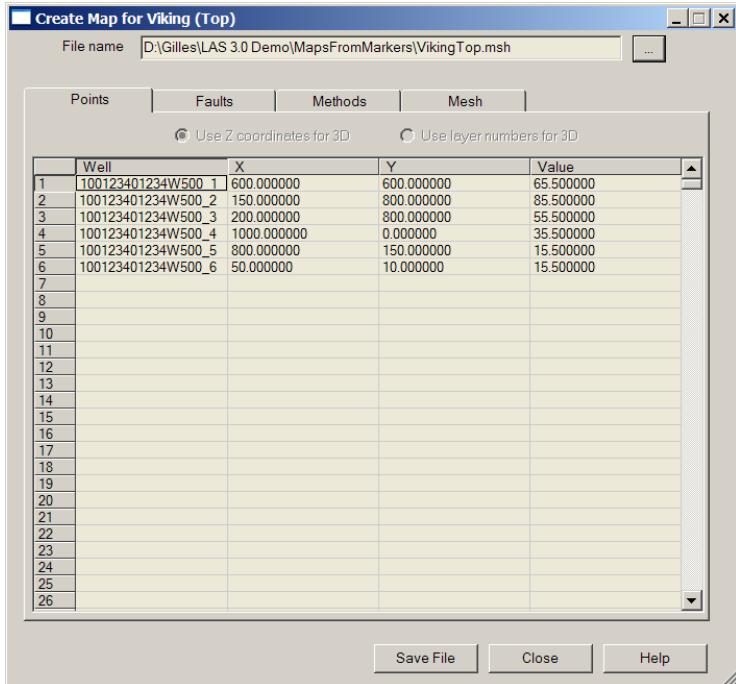
drag it along the well to edit its TVD position. You click **Apply** to update the well data when you are done editing the markers. This will also update the 3D view. Optionally well logs can be displayed close to the wells to assist with the marker's editing.

Create Horizons

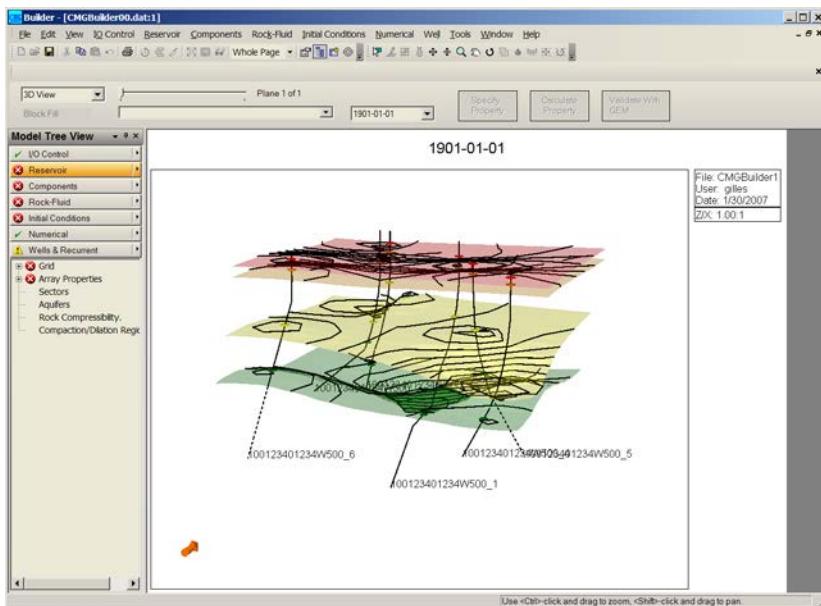
The Create Horizons task is mandatory if markers are selected instead of structure meshed maps (see [Select Tops](#)). Here you will be creating structure meshed maps by interpolating the selected markers. When you select this task, each selected marker will be listed in a spreadsheet row where you have access to a **Create Map** button. You can open the **Create Map** dialog box (see [Creating Maps and Geostatistical Property Calculations](#)) by clicking this button (see the examples below).



When the **Create Map** button is clicked, the **Create Map** dialog box is automatically filled with the appropriate marker data. You can use all options available in the Create Map dialog box for mesh map interpolation (see [Creating Maps and Geostatistical Property Calculations](#) for more details).



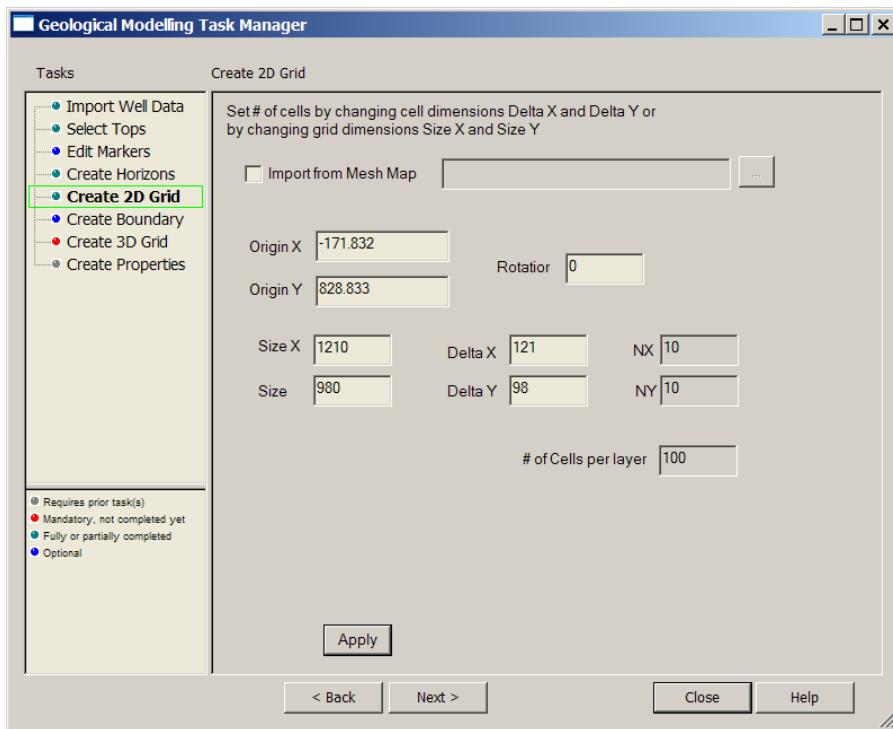
Through the **Create Map** dialog box, you can interpolate each selected marker and create a meshed map file for it. The meshed map filenames (with their path) will be reported back to the Output File Map column in the Create Horizons task view when created. Each meshed map is automatically loaded and displayed in Builder 3D View as soon as it is created, as shown below:

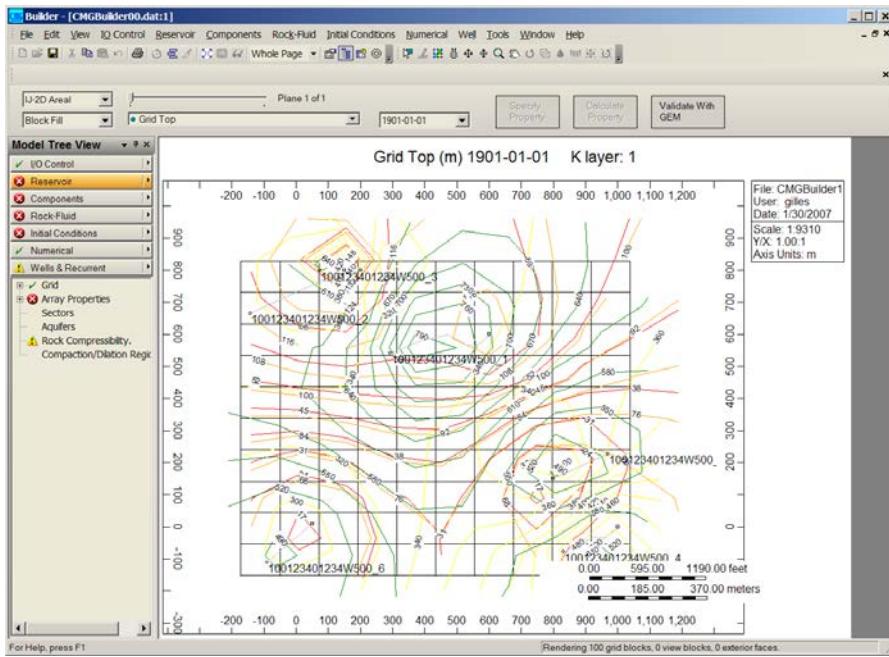


Create 2D Grid

The Create 2D Grid task is always mandatory. The 2D grid is the rectangular horizontal definition of the 3D grid (yet to be defined in a subsequent task). Before we can create the 3D grid, we need to lay out the meshed maps (created for the marker tops at the Create Horizons step) on a common horizontal grid definition. This will give vertical structure to the grid.

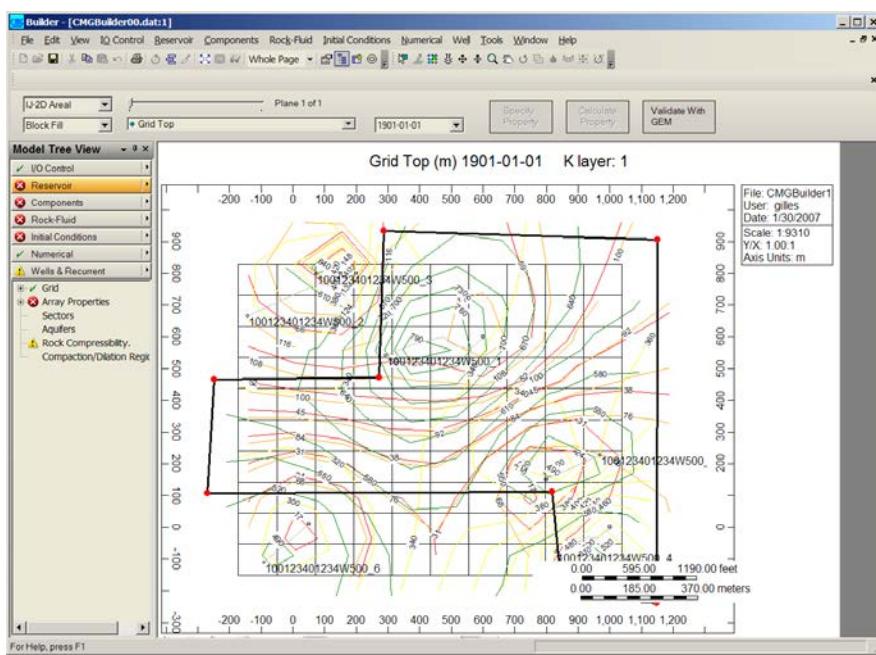
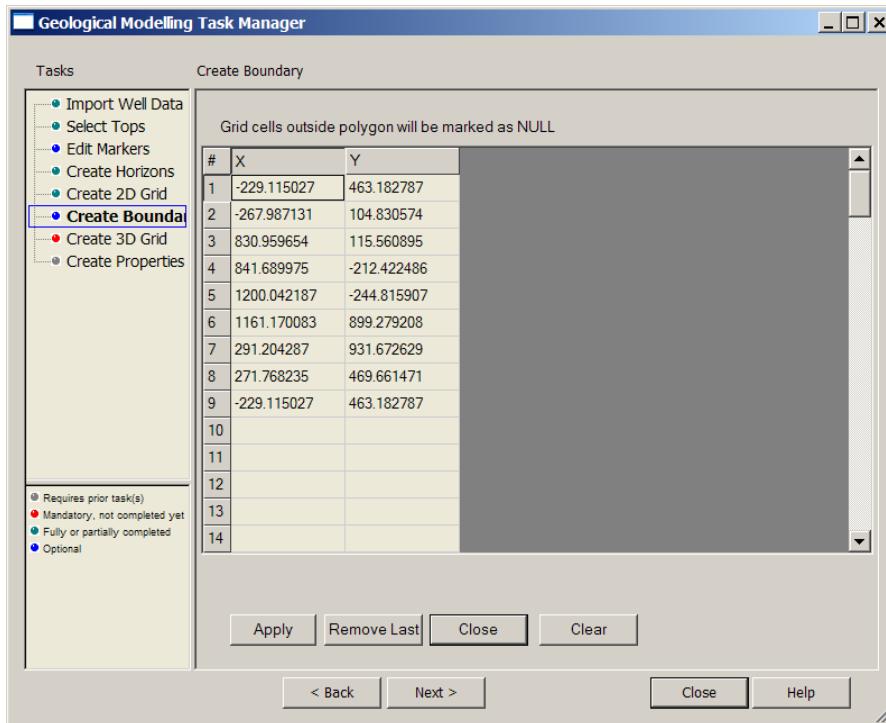
When you select this task, the default is to create a grid with only one cell represented by the bounding box around all the well surface locations. You can shift, rotate, and further discretize this bounding box for creating the final 2D grid. Alternatively, you can import the 2D grid definition from any meshed map file. For example, a map created at the previous step, Create Horizons, could be used for the 2D grid definition. You click the **Apply** button to accept the 2D Grid definition. This will display the 2D grid in the Builder II-2D Areal view, as shown in the following examples.





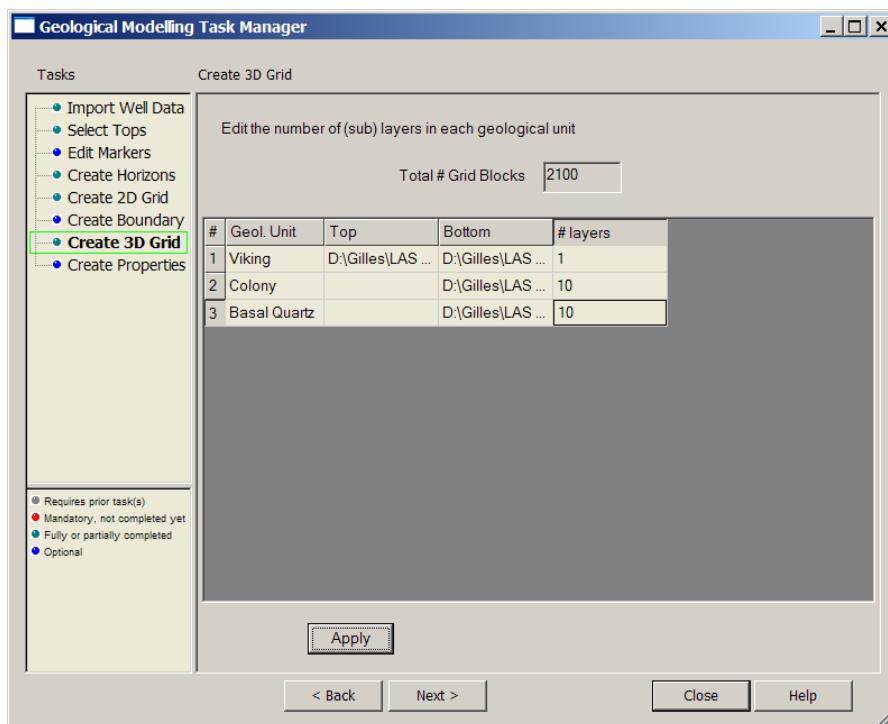
Create Boundary

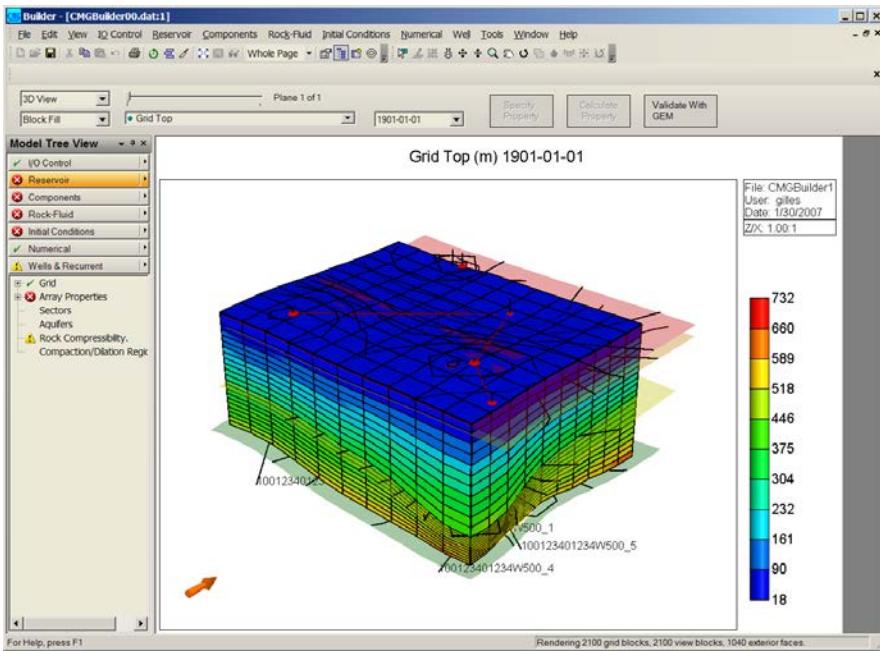
The Create Boundary task is optional. It is used to null out some parts of the 2D grid. Here, you can define a polygon by clicking points on the Builder IJ-2D Areal view. The points (red dots) will be connected into a closed polygon. Click **Close** to close the polygon. Click **Remove Last** to undo the last point. Grid cells outside the polygon will be marked as inactive for the flow simulator. The polygon will be used to create the NULL property array when creating the 3D grid.



Create 3D Grid

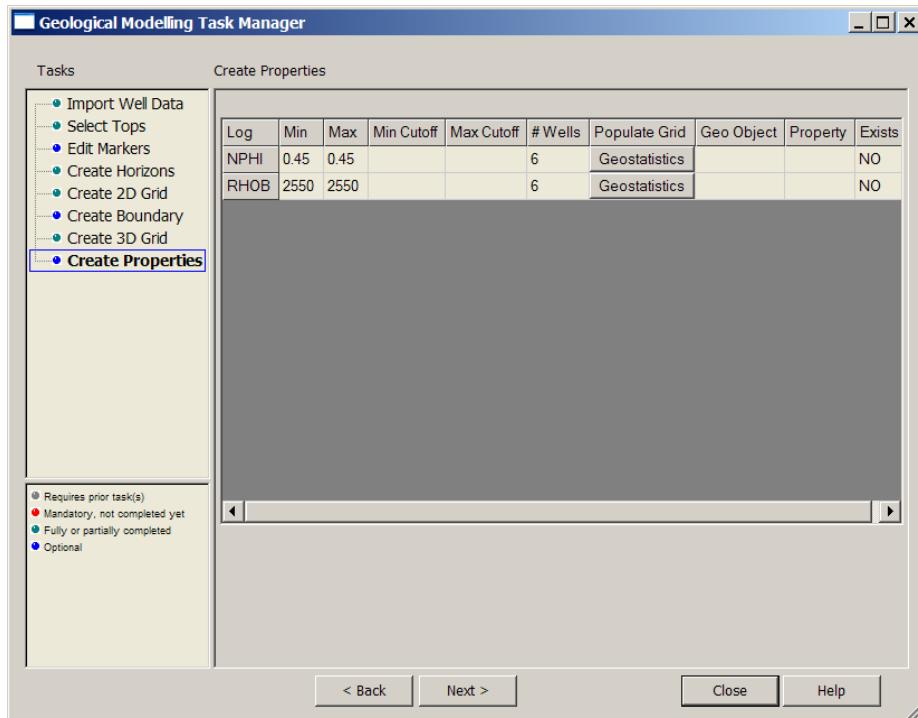
The Create 3D Grid task is mandatory. It is the last step for completely defining the simulation grid. Here, you can define the vertical grid according to the structure meshed maps representing the reservoir tops. Each geological unit will be bounded by the meshed maps from the Create Horizons task as indicated in the Top and Bottom columns (as shown below). You need to define the number of layers required in each geological unit. Click **Apply** to update the Builder 3D view with the newly created 3D grid (see second picture below).





Create Properties

The Create Properties task is optional. Now that the 3D grid has been built, it can be populated with static properties. At this point you could close the **Task Manager** dialog box and use the **Reservoir | Geostatistics** menu option (see [Creating Maps and Geostatistical Property Calculations](#)). Instead, you can use this option within the Task Manager by clicking the **Geostatistics** button for a selected well log (see the first example below). This will select the well log data and transfer them into the **Create Geostatistical Objects** dialog box (see the second example below). The well log data will be averaged (upscaled) on a grid block basis. Cutoff values can be used to filter out some log data. Through the **Create Geostatistical Objects** dialog box, you can select the methods and parameters for interpolating the well log data on the 3D grid (see [Creating Maps and Geostatistical Property Calculations](#) for more detail).



Create Geostatistical Objects for NPHI

| Name | Output Property | Porosity |
|------|-----------------|----------|
| NPHI | | |

Points | Methods | Regions |

Use Z coordinates for 3D Use layer numbers for 3D

| | Well | X | Y | Z (3D Only) | Layer# | Value |
|----|--------------------|------------|------------|-------------|--------|----------|
| 1 | 100123401234W500_1 | 614.668000 | 593.833000 | 89.532336 | 1 | 0.450000 |
| 2 | 100123401234W500_1 | 614.668000 | 593.833000 | 127.486101 | 2 | 0.450000 |
| 3 | 100123401234W500_1 | 614.668000 | 593.833000 | 473.451149 | 15 | 0.450000 |
| 4 | 100123401234W500_2 | 130.668000 | 779.833000 | 142.598750 | 2 | 0.450000 |
| 5 | 100123401234W500_2 | 130.668000 | 779.833000 | 368.600127 | 12 | 0.450000 |
| 6 | 100123401234W500_2 | 130.668000 | 779.833000 | 487.761950 | 17 | 0.450000 |
| 7 | 100123401234W500_3 | 251.668000 | 779.833000 | 151.183497 | 3 | 0.450000 |
| 8 | 100123401234W500_4 | 977.668000 | -4.167000 | 48.576822 | 1 | 0.450000 |
| 9 | 100123401234W500_4 | 977.668000 | -4.167000 | 369.610933 | 9 | 0.450000 |
| 10 | 100123401234W500_4 | 977.668000 | -4.167000 | 562.760430 | 19 | 0.450000 |
| 11 | 100123401234W500_5 | 856.668000 | 191.833000 | 43.061793 | 1 | 0.450000 |
| 12 | 100123401234W500_5 | 856.668000 | 191.833000 | 183.735345 | 6 | 0.450000 |
| 13 | 100123401234W500_5 | 856.668000 | 191.833000 | 374.547195 | 14 | 0.450000 |
| 14 | 100123401234W500_6 | 9.668000 | -4.167000 | 40.243991 | 1 | 0.450000 |
| 15 | 100123401234W500_6 | 9.668000 | -4.167000 | 168.778117 | 6 | 0.450000 |
| 16 | 100123401234W500_6 | 9.668000 | -4.167000 | 466.013840 | 20 | 0.450000 |
| 17 | | | | | | |
| 18 | | | | | | |
| 19 | | | | | | |
| 20 | | | | | | |
| 21 | | | | | | |
| 22 | | | | | | |
| 23 | | | | | | |
| 24 | | | | | | |
| 25 | | | | | | |
| 26 | | | | | | |

Run OK Cancel Apply Help

Editing an Existing Simulation Grid

You can edit a grid in the **Edit Grid** mode. You can carry out several operations to edit a grid that was created in Builder:

- Redefine grid parameters (NI, NJ, NK and so on)
- Rotate, translate the grid
- Add, delete, move gridlines (orthogonal grids)
- Add, delete, move, redefine control points (non-orthogonal grids)

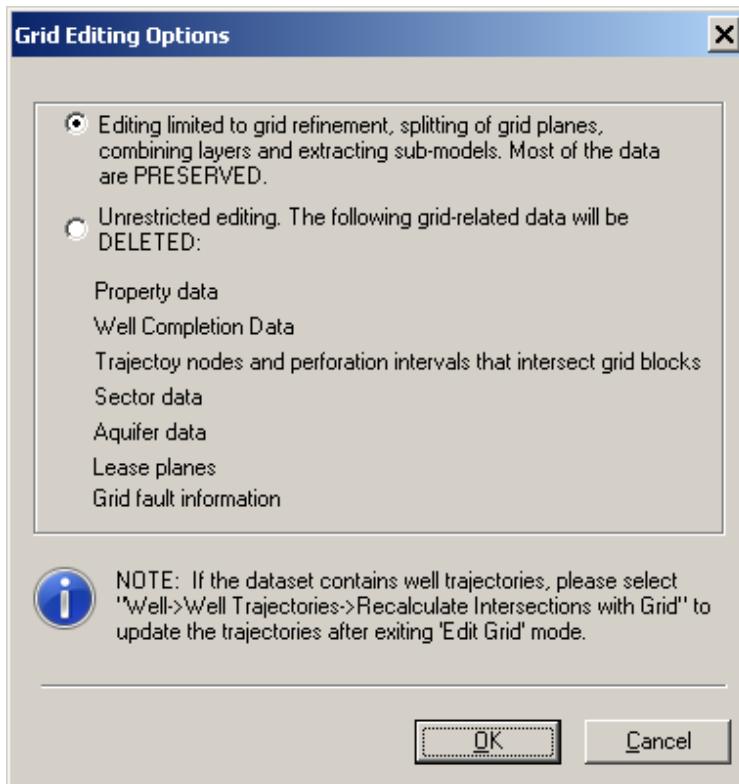
For all grids, you can:

- Add, remove refined grids
- Split I, J, K “planes” of the grid
- Extract a sub-section of the grid

You must be in **Edit Grid** mode to perform the above operations. To enter **Edit Grid** mode, click the **Edit Grid**  button on the modes toolbar, or right-click to pop up the context menu and select **Edit Grid** by clicking it. If spatial property and well data do not exist at this stage, you will be put into the edit grid mode immediately.

Sometimes you may wish to alter the grid in a dataset that already has grid properties and well locations. For example, you may have a grid from a previous simulations study where you don't have the original map files, or where grid properties have been modified during a history match. Builder allows several operations where the properties and well perforations are preserved, but the grid is altered. You can add refined grids, split existing grid blocks along I, J, or K “planes”, or extract a rectangular sub-section of the simulation model.

If spatial property and well data currently exist, the **Grid Editing Options** dialog box is displayed:



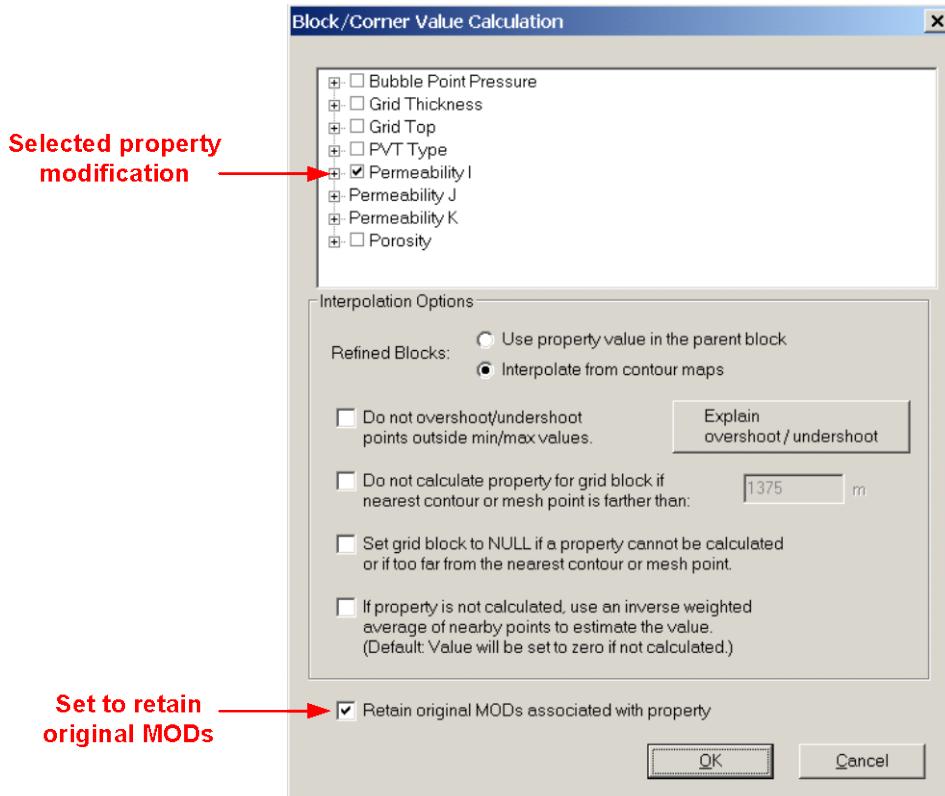
Note: If you select the unrestricted editing option, all of the grid-related data will be destroyed. You will have to recreate or re-import them. You will normally use this option only in the initial stages of model development or for certain situations that may develop later, as explained above.

Property Modifications (MODs)

MODs are property modifications which can be applied to specific sections (blocks, areas or layers, for example) of a grid. When viewing a dataset with MODs in a text editor, MODs will be listed after the array properties they modify.

In previous Builder releases, when re-calculating an array property, the modification (keyword MOD) would be applied to the grid but the record of the change would not be saved to the dataset.

In the current release, a check box is provided in the **Block/Corner Value Calculation** dialog box, used in several edit operations, to provide MOD retention options:



The **Retain original MODs associated with property** check box is used as follows:

- If the check box is not checked when you click **OK**, the modifications associated with the selected property (Permeability I in the above example) are not retained and the MOD information is removed. The property will be re-calculated based solely on the information in the **General Property Specification** dialog box.
- If the check box is selected, the MOD will be retained in the dataset, immediately following the array properties.

When MODs exist in a dataset, **Edit grid** operations are affected as follows:

- **Refine Blocks/Refine Wells:** If you refine a block or well, the MOD is applied, as appropriate, to the resulting refinement. If a block with an applicable MOD is refined into smaller blocks, for example, the MOD is applied to each of the new sub-blocks.
- **Split Grid Plane:** If you split a grid plane, applicable MODs are applied to the resulting grid planes.

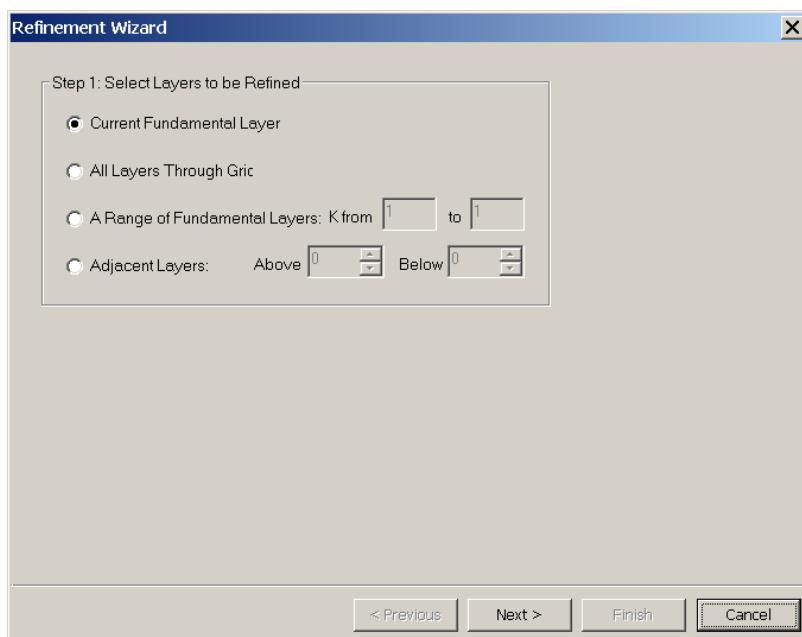
- **Combine Layers:** If you combine layers with an applicable MOD, the MOD keyword will be retained if it applied to all layers being combined. Otherwise, the modification is incorporated into the property array.
- **Extract SubModel:** If you extract a submodel, applicable MODs will also be retained in the submodel dataset.

Adding Refined Grids Using the Refinement Wizard

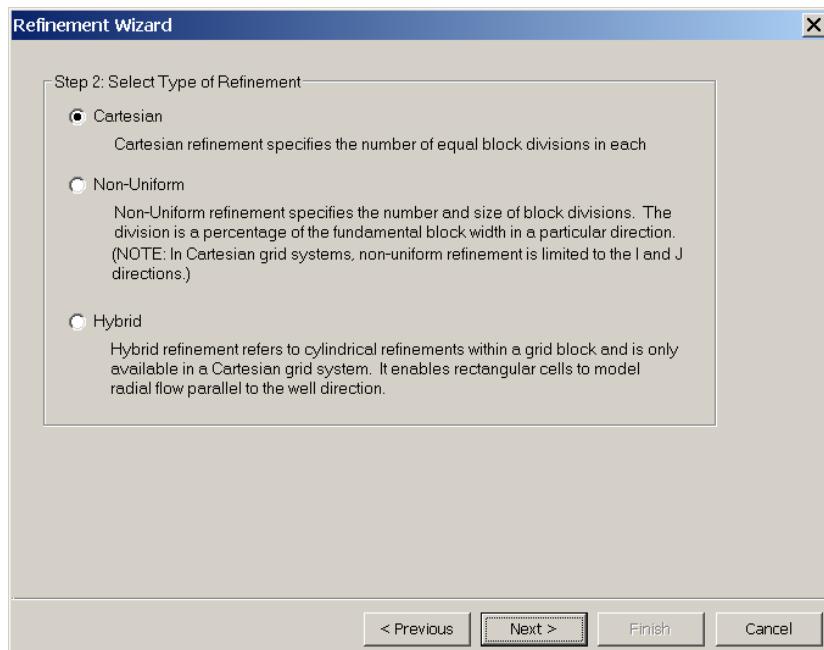
Note: For information about the processing of property modifications, refer to [Property Modifications \(MODs\)](#).

To add a refined grid:

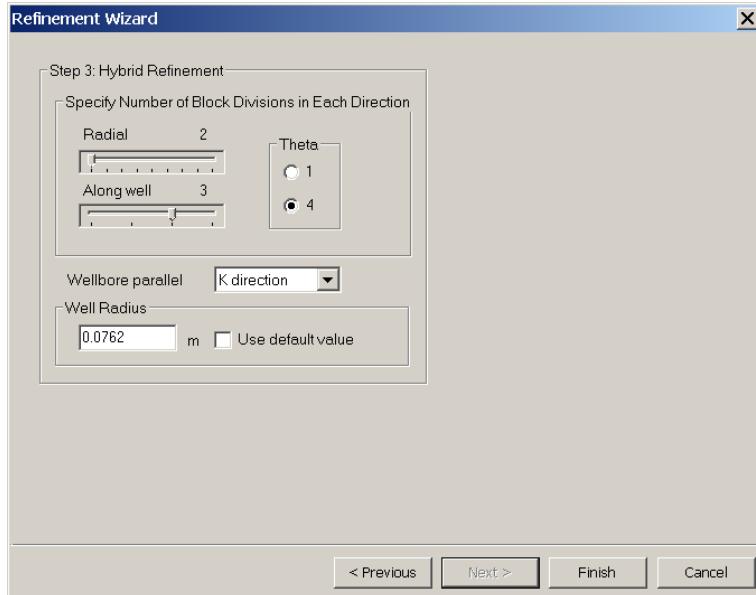
1. You must be in **Edit Grid** mode to perform this operation. To enter **Edit Grid** mode, click the **Edit Grid**  button on the modes toolbar, or right-click to pop up the context menu and then click **Edit Grid**.
2. Select the grid block you want to refine. If you want to refine a range of grid blocks, click on a block and hold the left mouse button. Drag the mouse to the desired range. As you drag the mouse, the selected grid blocks are highlighted.
3. Activate the **Refinement Wizard** via the menu. Select **Reservoir | Edit grid | Refine Blocks**.



4. In **Step 1** of the **Refinement Wizard**, select the layers you want to refine. **Current Fundamental Layer** is limited to the layer in which you selected blocks. **All Layers Through Grid** will refine your selection in every layer throughout the reservoir. You can pick just a subset of layers by specifying **A Range of Fundamental Layers** or by choosing the number of **Adjacent Layers Above** or **Below** the current layer. When you are finished, click **Next**. The **Step 2: Select Type of Refinement** dialog box is displayed:



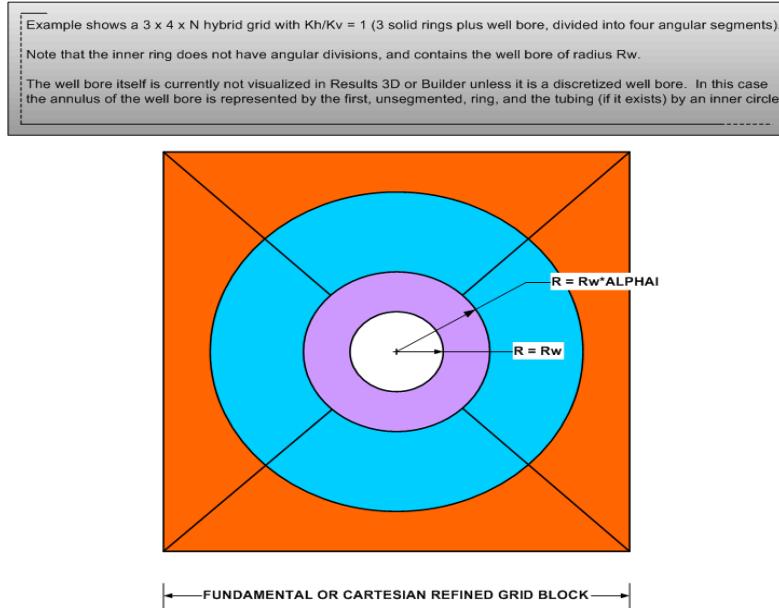
5. In **Step 2** of the **Refinement Wizard**, you choose how you would like the selected blocks to be refined. **Cartesian** refinement allows you to specify the number of block divisions in the I, J, and K directions. Each block division will be equal. In **Non-Uniform** refinement, you not only can specify the number of block divisions, but also the size of these divisions. In Cartesian grid systems, non-uniform refinement is limited to the I and J directions. **Hybrid** refinement allows you to model radial flow parallel to the well direction and is only available in Cartesian grid systems. After you have made your choice, click **Next**. If you selected **Hybrid** refinement, the **Step 3** of the **Refinement Wizard** is displayed.
6. The dialog box that is displayed depends on your selection in the previous step. If you chose **Hybrid** refinement, you will be presented with a series of options:



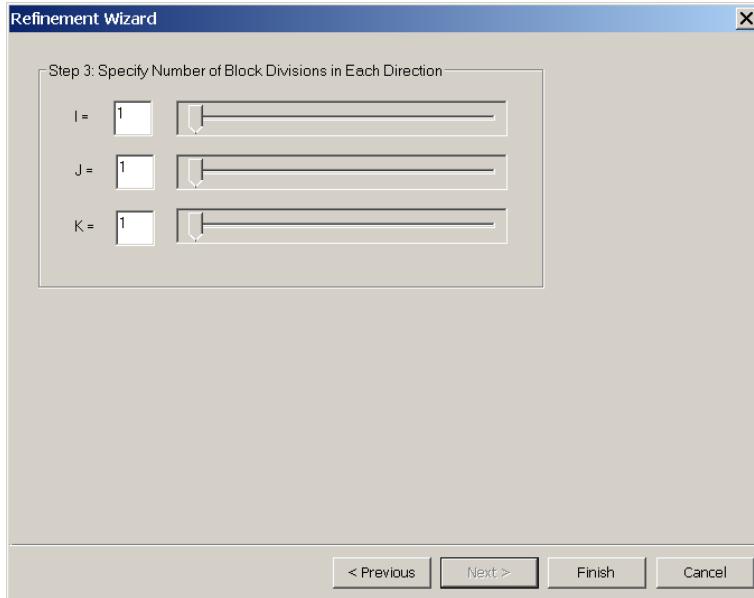
7. You can select the number of divisions in the Radial, Theta, and Along well directions. Make sure to correctly specify the parallel direction of the wellbore. You can also specify the Well Radius or use a default value. When you are satisfied with your choices, click Finish.

The following diagram provides further clarification on **Hybrid** refinement.

Hybrid Grid Definition

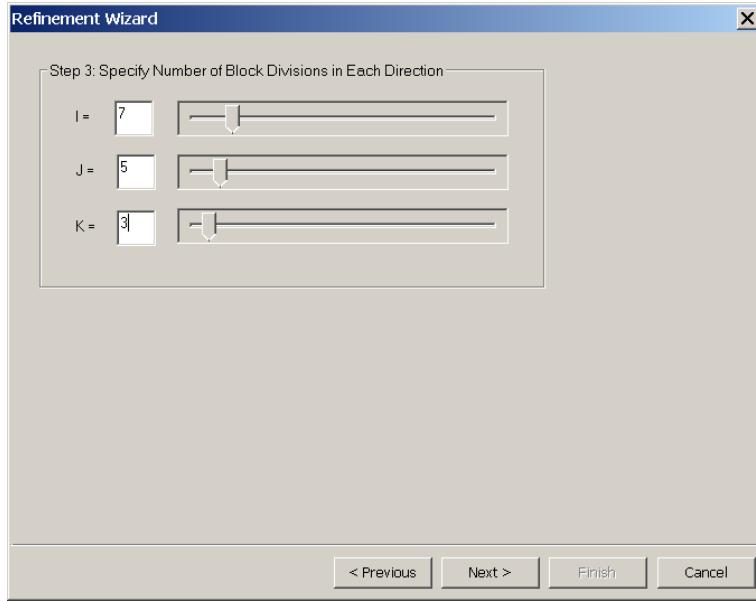


8. If you chose **Cartesian** refinement in **Step 2**, you will be presented with the option to **Specify Number of Blocks Divisions in Each Direction**:



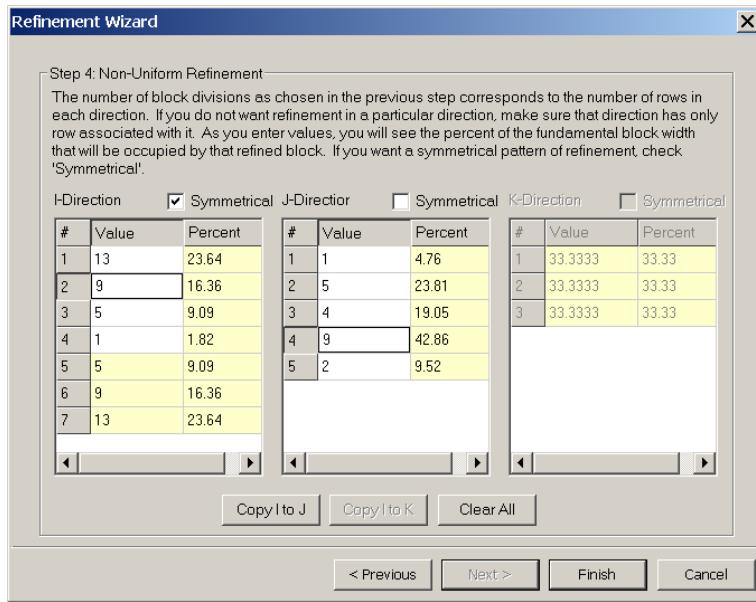
You can indicate this number by using the sliders or by directly typing into the edit boxes. This is the last step of the **Refinement Wizard** if you specified **Cartesian** refinement; if that is the case, click **Finish**.

9. If you specified **Non-Uniform** refinement in **Step 2**, you will be presented with the option to **Specify Number of Blocks Divisions in Each Direction**. Unlike the case of Cartesian refinement, the **Next** button is enabled:



As with the Cartesian refinement, specify the number of block divisions in each direction using the sliders or by directly typing into the edit boxes, and then click **Next**.

Step 4 of the Refinement Wizard is displayed, allowing you to specify refinement values for **Non-Uniform Refinement**.



The number of rows in each direction corresponds to the number of block divisions entered in **Step 3**. If you do not want to refine in a particular direction, make sure that there is only one row associated with it.

As you enter numbers in the **Value** columns, you will see the corresponding percent fundamental block width in the **Percent** column that will be used to calculate refinement. For instance, in the above example, a block will be divided five times in the **J-Direction**. The first division will encompass 4.76% of that fundamental block's width in the **J-Direction**. The second division will encompass 23.81%, and so on until five divisions have taken place. This will be repeated for every block selected in the reservoir. If you want every division to be the same width, just enter the same value in each cell of the **Value** column, as shown above for the **K-Direction**.

If you want a symmetrical pattern of non-uniform refinement for a particular direction, make sure to check **Symmetrical**. This will set certain rows in the **Value** column as read only. As you fill up the **Value** column, the remainder of the cells will fill up in a symmetrical fashion.

If you want to use values you have entered in the **I-Direction** for the **J-Direction** or **K-Direction**, click the **Copy I to J** or **Copy I to K** buttons. Clicking the **Clear All** button will remove all values for all directions. When you are satisfied with your choices, click **Finish**.

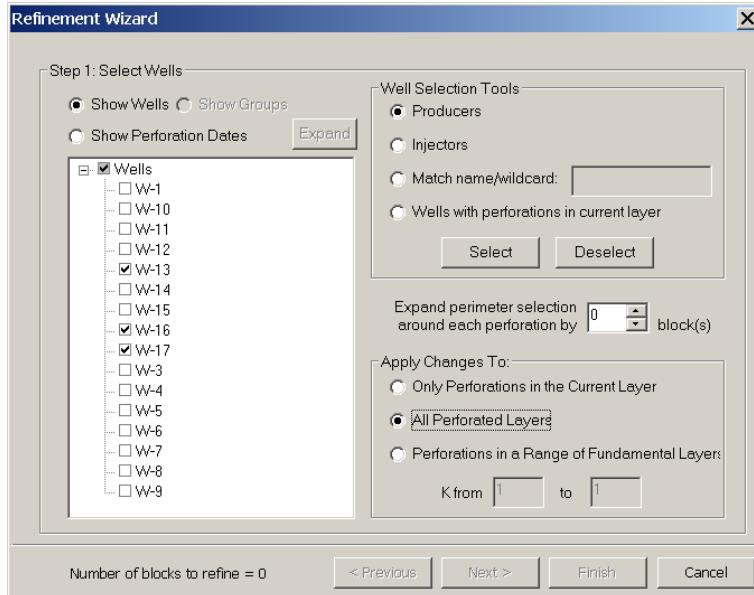
Note: If you want to add **Non-Uniform** refinement to a Cartesian grid, you will not be able to specify values for the **K-Direction**. You can still specify the number of divisions in the **K-Direction** in **Step 3**, but every division will automatically be equal.

Adding Refined Grids around a Well Using the Refinement Wizard

Note: For information about the processing of property modifications, refer to [Property Modifications \(MODs\)](#).

To add refined grids around a well:

1. You must be in **Edit Grid** mode to perform this operation. To enter **Edit Grid** mode, click the **Edit Grid**  button on the modes toolbar, or right-click to pop up the context menu and click **Edit Grid**.
2. Activate the Refinement Wizard via the menu, by selecting **Reservoir | Edit grid | Refine Wells**.
3. If you want to refine blocks around a well or several wells, you will be presented with a different **Step 1** of the **Refinement Wizard**:



Here you can select which perforations to refine via the tree on the left-hand side. The default view is to select via **Show Wells**. You also have the option of selecting via groups in which case you would select **Show Groups**. If you want to see and select via specific perforation dates, select **Show Perforation Dates**. For the latter two options, you can **Expand** or **Collapse** the tree to suit your needs.

To aid in your selection process, you can use the features under **Well Selection Tools**. Here you can **Select** or **Deselect** all **Producers**, **Injectors**, wells matching a certain name or wildcard, or **Wells with perforations in current layer**.

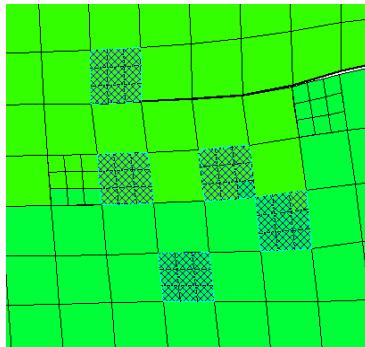
If you want to not only refine perforations but adjacent blocks, you can indicate how many surrounding blocks to include by changing the value for **Expand perimeter selection around each perforation**.

In the **Apply Changes To** section, you can choose to apply refinement to **Only Perforations in the Current Layer**, **All Perforated Layers**, or just **Perforations in a Range of Fundamental Layers**.

The **Number of blocks to refine** in the lower left corner will help you to see the effect of your choices. This number will change as you make different selections in the tree, the number of surrounding blocks, or the layers in which to apply refinement. As soon as the number is greater than zero, the **Next** button will become activated and you can proceed with the remaining steps of the wizard which are identical to what has already been described above. For more information, see [Adding Refined Grids Using the Refinement Wizard](#).)

To remove a refined grid:

1. You must be in **Edit Grid** mode to perform this operation. To enter **Edit Grid** mode, click the **Edit Grid**  button on the modes toolbar, or right-click to pop up the context menu then select **Edit Grid**.
2. Click on the refined grid. If you want to select more than one contiguous block, click on the upper left block, hold the left-mouse button and drag until a range of refined blocks is selected. As you do this, you will see the refined blocks become highlighted. If you have several ranges of refined blocks in the current layer that you want to remove but are not adjacent, start with the range of blocks in the upper left, repeat the same process but drag the mouse to the lower right of the reservoir. You should see several refined blocks become highlighted as shown below:



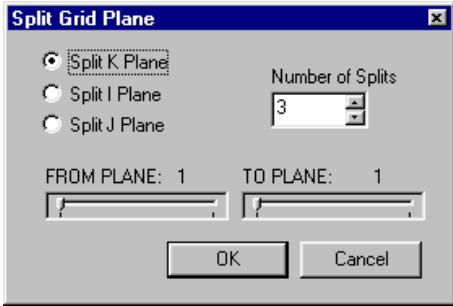
3. Once you have finished selecting the blocks in which to remove refinement, select **Reservoir | Edit grid | Remove Refinement**.

Splitting Grid Planes

Note: For information about the processing of property modifications, refer to [Property Modifications \(MODs\)](#).

To split grid planes:

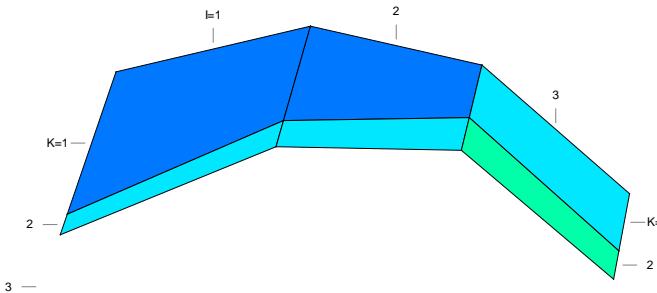
1. Enter **Edit Grid** mode as described earlier.
2. Select **Split Grid Plane** from **Reservoir, Edit Grid** menu. The **Split Grid Plane** dialog box is displayed:



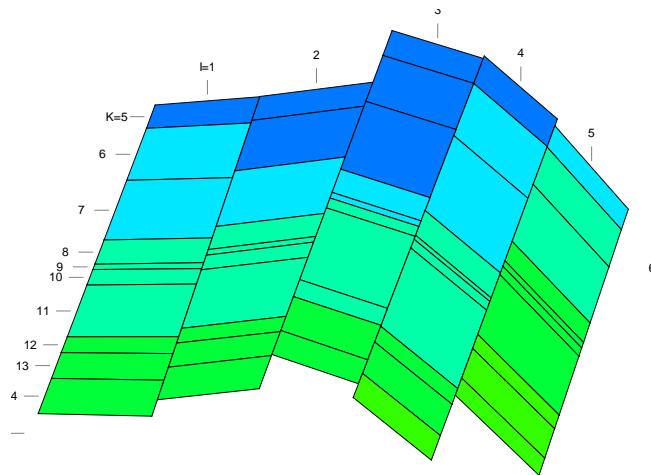
3. To split planes in “I plane” select **Split I Plane** option. This option will split all grid in the columns from $I=I_1$ to $I=I_2$ where I_1 and I_2 are selected using the **FROM PLANE:** and **TO PLANE:** sliders in the dialog box.
4. Click **OK**.
5. Repeat steps 2 and 3 for splitting along J rows and K layers.
6. When you are finished modifying the grid, exit **Grid Edit** mode. It is recommended that you examine the well perforation locations, sector definitions and other features of the grid to ensure that the grid modifications have been performed as you expected.

Join Blocks Option

A corner point grid is normally constructed so that adjacent blocks that are located in the same vertical layer have corners that have the same x,y,z coordinates. For example, the grid below the top right corner for block 1,1,1 (left top block) has the same x,y,z coordinates as the top left corner for block 2,1,1 (middle top block).



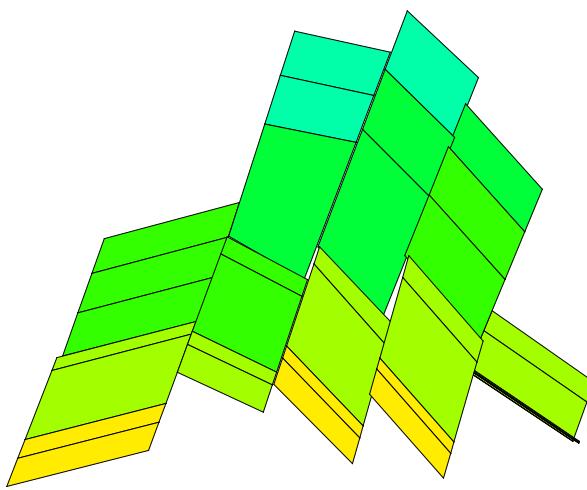
The grid module within each CMG simulator compares the corners from adjacent blocks to determine if the distance between these corners is less than *CORNER-TOL. If true, then the grid module creates a connection between these two adjacent blocks. If the distance between these corners is greater than *CORNER-TOL, then the grid module attempts to determine which blocks are contacting each other. If two blocks are in contact, then connections are made between these blocks. An example of this behavior is shown below:



Note that in the above example, almost all of the corners for adjacent blocks are not the same. However, the blocks are still connected because they are contacting each other. For example, block 1,1,5 (top left block) is contacting blocks 2,1,5 and 2,1,6. Therefore, the grid module will create connections to these blocks. To view the actual block connections, open Results 3D, select the menu **View | Probe Display** and check **Connections**.

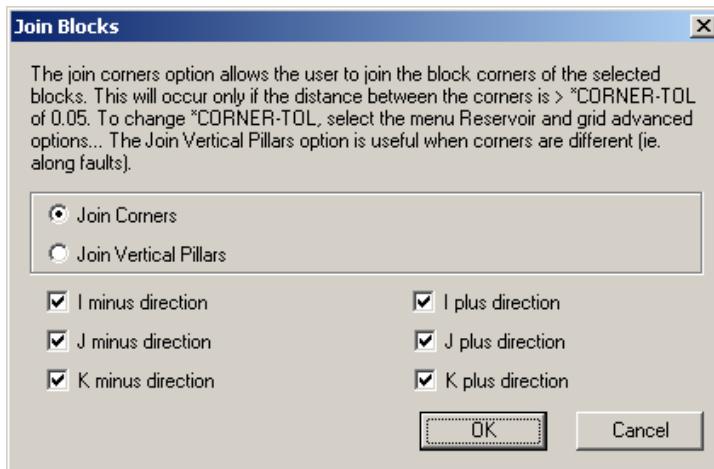
Poorly Constructed Grids

The Join Blocks option was created to help you fix certain grid problems that can be created in some geological modelling software packages. The example below shows a poorly constructed grid that has spaces between blocks. With this grid, there will be no connections created when the blocks are not in contact with each other. This absence of connections will create a barrier to fluid flow in the model that may not be correct.



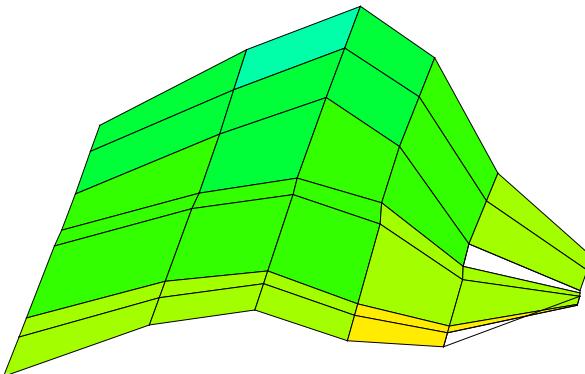
Join Corners Option

The join corners option can be invoked in 2D or 3D views by selecting the menu item **Reservoir | Edit Grid | Join Blocks**. You can then select some blocks with the mouse and when the mouse is released, a dialog box will be displayed. If you go out of the join blocks mode by selecting another mode such as probe mode, or pan mode, then you must re-select the Join Blocks menu item to get back into the join blocks mode. The **Join Blocks** dialog box is displayed, as follows:



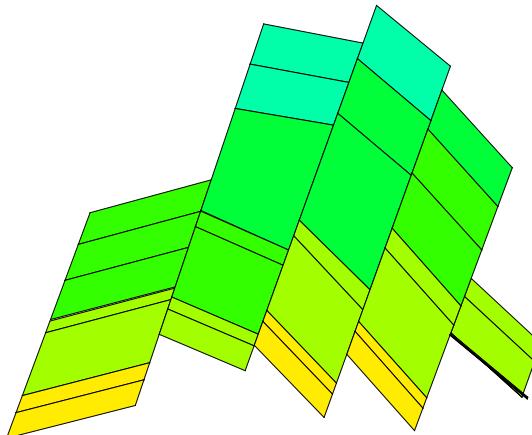
Join Corners is the first (default) option. You can also check or cancel boxes to determine which direction is searched from the selected block. Click **OK** to search the requested directions and calculate the distance between corners of adjacent blocks. If the distance between two adjacent block corners is greater than *CORNER-TOL, then the corner coordinates are averaged and both corners are assigned these coordinates.

For example, if the **Join Corners** option is used for the grid in the above poorly constructed grid example, the grid becomes as shown below:



Join Vertical Pillars Option

Sometimes grid problems exist at fault locations and it is not appropriate to simply join the corners between blocks because the action of joining the corners will remove the fault. In this case, the **Join Vertical Pillars** option should be used. This option will allow block corners to be different, but the vertical pillars of the grid will be preserved. Below is the result of this option for the poorly constructed grid example:



Combining Layers

Introduction

Combining layers (or uplayering) is a technique that provides you with optimal geologic layer-grouping schemes for simulation model construction. It will reduce many (often hundreds) geologic fine layers to fewer coarse simulation layers yet at the same time, maintain the heterogeneity of the reservoir in the K (vertical) direction.

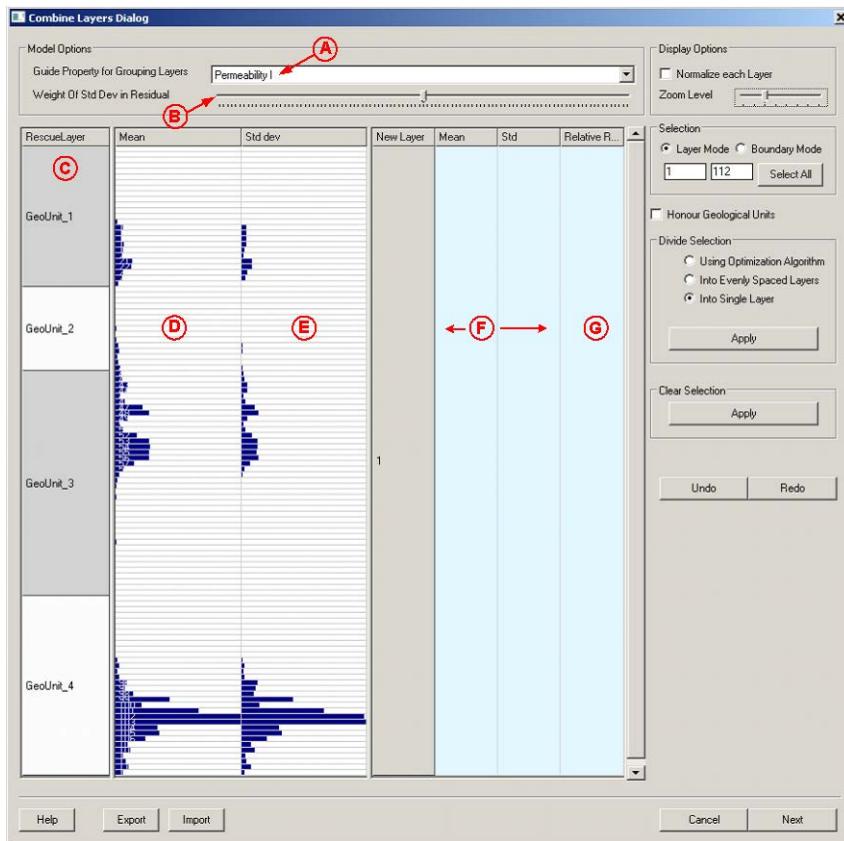
The grouping of the layers can be done manually or automatically in Builder. In manual mode, Builder provides the average and standard deviation of the selected guide property, to help you decide which layers to combine. The guide property is typically porosity, K direction permeability, or a combination of both. In automatic mode, Builder can suggest the optimal combination of the layers given the desired number of simulation layers. The method is based on the ‘residue optimization’ method presented by Li and Beckner in SPE paper #57273. A table of ‘residues’ (the difference between the defined property of the fine- and coarse-layer models) is produced from an exhaustive analysis of all possible layering combinations. Using this table, engineers can determine the number of simulation layers needed based on their tolerance of possible loss of fine-layer geologic features.

Note that all refined grid blocks will be deleted during the combining operation. These may be manually added back in after the combining is complete. Well perforations, sectors, aquifers and lease planes will be preserved during combining.

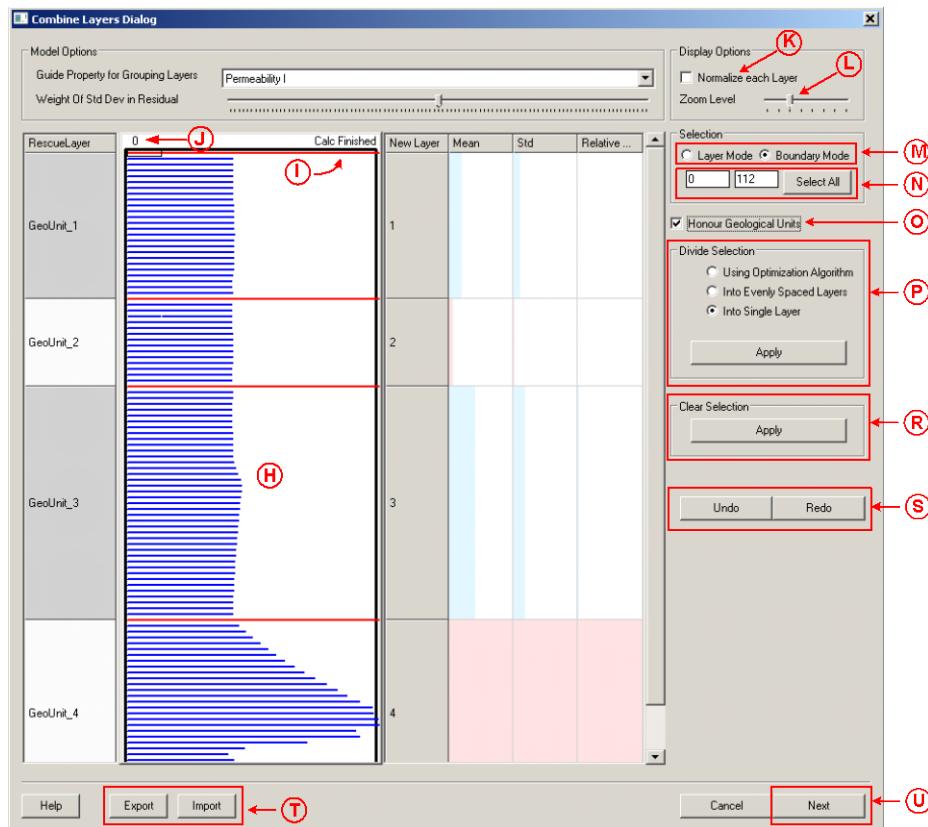
Note: For information about the processing of property modifications, refer to [Property Modifications \(MODs\)](#).

Combine Layers Dialog

The following figures and tables illustrate the purpose and use of the **Combine Layers Dialog** screen elements. After this section, in [To combine layers](#), we will walk you through the process of using Builder to combine layers in a new model:



| Ref | Description |
|-----|--|
| A | Select the property used to facilitate the combining. |
| B | Determines the weighting of standard deviation and mean in the calculation of the residual. For information about this feature, go to page 75. |
| C | Geological units. These can be set using Reservoir Geological units , or brought in from RESCUE. |
| D | Graph of the mean of the selected property across all I, J for each K in the original model. |
| E | Graph of the standard deviation of the selected property across all I, J for each K in original model. |
| F | Mean and standard deviation of the selected property in the post-uplayering (new) model. |
| G | Relative residual is a graph showing the heterogeneity of each post-uplayering in the new mode. A high value means that the layer should be further divided. |



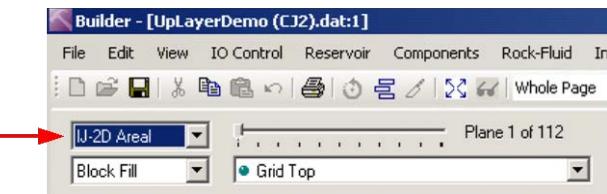
| Ref | Description |
|-----|---|
| H | Boundaries are displayed if you select Boundary Mode . Each light blue (dark blue, if selected) line corresponds to a boundary in the original model. Light red (dark red, if selected) lines are boundaries that will remain in the model after combining. The dialog displays the results of calculations after each step of the combining process. The length of the line corresponds to the improvement in the residual calculation, if the boundary is preserved in the new model. The optimal boundary spans the width of the plot. This value can act as a guide for manual editing. Double-click any boundary in the original model to either include it in the new model or, if the boundary already exists, remove it. |
| I | Progress of single optimization step. In the example shown, the calculation is finished. |
| J | Mouse over boundary (or layer) number, boundary ‘0’ in this case |
| K | If not selected, Builder uses the maximum of the entire model to normalize the graph. If selected, Builder uses the maximum value within each layer to normalize the graph for that layer. Select Normalize each Layer when you want to see how the optimization algorithm will act on a subset of the model. The plot will be adjusted to display the optimization results for each layer in the new model, rather than for the global optimization. Use this option when you are editing a subset of the model. |
| L | Adjust the displayed height of layers by zooming in or zooming out. |
| M | Choose between displaying “layers” and displaying “boundaries”. |
| N | Display/choose selected boundaries (type in or select with mouse). |
| O | When selected (checked), if there are separate geological units in the model, Builder will add geological boundaries and these boundaries will not be removable by other methods. When not selected (not checked), Builder does nothing with the geological boundaries. These boundaries will be treated as regular boundaries and can be deleted. |
| P | Using Optimization Algorithm: Puts a boundary at the top and bottom of selection, erases all boundaries inside (except for geological boundaries if Honour Geological Units is checked), and brings up a dialog box to add boundaries inside the selection. Into Evenly Spaced Layers: Puts a boundary at the top and bottom of the selection, erases all boundaries inside (except for geological boundaries if Honour Geological Units is checked), and brings up a dialog box to add evenly spaced boundaries inside the selection. If Honour Geological Units is checked, and the selection bridges a geological boundary, the dialog box will split the layers into two or more ranges, as necessary. Into Single Layer: Puts a boundary at the top and bottom of selection, and erases all boundaries inside (except for geological boundaries if Honour Geological Units is checked). |
| R | Will remove all boundaries in the selected range, with the exception of geological boundaries if Honour Geological Units is checked. |

| Ref | Description |
|-----|--|
| S | Allows you to undo (and subsequently redo) the previous ten changes made to the model. |
| T | Allows you to save (export) the uplayering scheme in a tab-delimited text file, which can be run on files with similar geometry (e.g., different geostat realizations). |
| U | If you accept the model without making changes (i.e., clicks Next then Finish), the Combine Layers Dialog will combine all of the original layers into a single layer. |

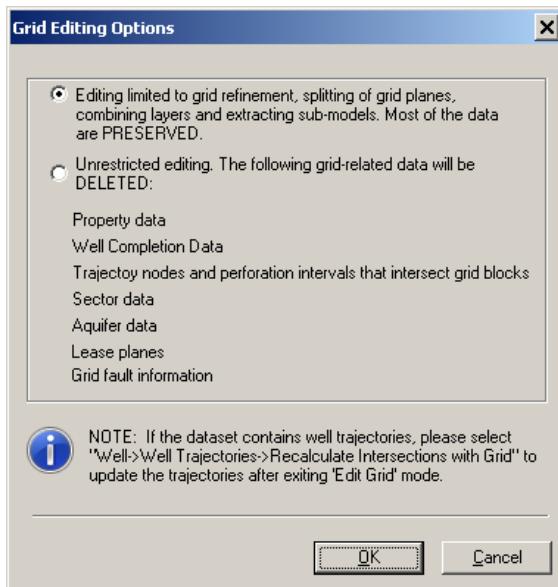
To combine layers:

The following steps illustrate the procedure for combining layers. As necessary, refer to [Combine Layers Dialog](#) for more information on the purpose and operation of the elements in the **Combine Layers Dialog** screen.

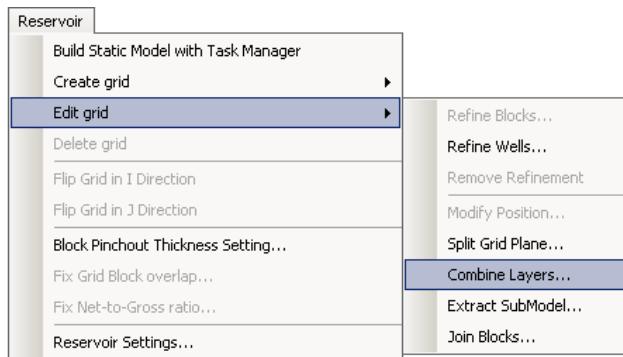
1. In the main Builder screen, select **IJ-2D Areal** view:



2. Click **Edit Grid** in the toolbar, or right-click in the grid to display the context menu then select **Edit Grid**. The **Grid Editing Options** dialog box will be displayed:



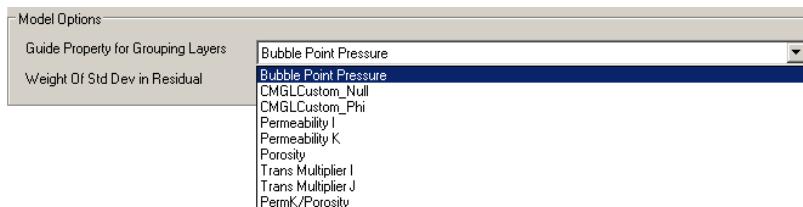
3. Select **Editing limited to grid refinement** (the default) then click **OK**. You are now in grid edit mode.
4. In the menu bar, select **Reservoir | Edit grid | Combine Layers**:



The **Combine Layers Dialog** is displayed. For information about the purpose and use of the elements in the **Combine Layers Dialog** screen, refer to [Combine Layers Dialog](#).

5. In the **Model Options** area:

- a) Select the **Guide Property for Grouping Layers**:



By viewing the mean and standard deviation display in the **Combine Layer Dialog**, choose a property that has appropriate and adequate variation and is not constant throughout the model. A common approach is to set **Guide Property for Grouping Layers** to **PermK/Porosity**, if both properties have values in the current model. If only one of Porosity or Permeability K is available, then set **Guide Property for Grouping Layers** to the one that is available. Other properties may also be used, but their use is less common.

- b) Set the weight of the standard deviation and mean terms in computing the ‘residue’ by clicking and dragging the **Weight of Std Dev in Residual** slider. Moving the slider all the way to the left means “use mean only” and all the way to the right means “use standard deviation only”. The default is in the middle, which corresponds to a 50/50 weighting.

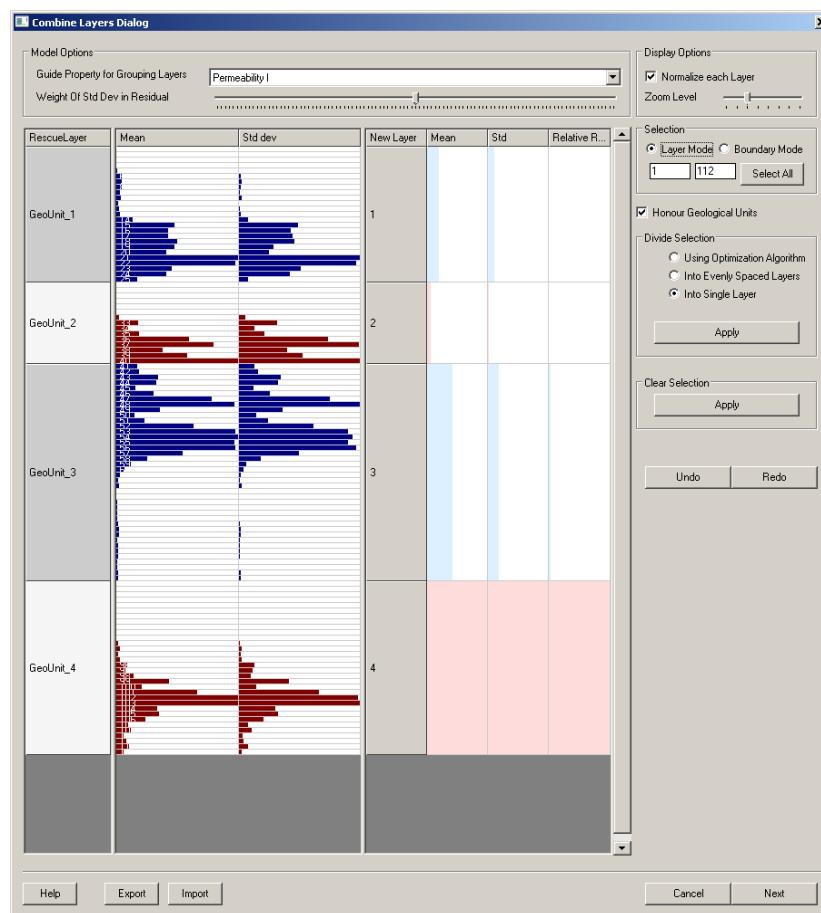


Refer to SPE paper #57273 for more detailed information about this setting.

6. In the **Display Options** area:

- Select **Normalize each Layer** to normalize the levels of the values within each layer based on the maximum value of the property within that layer.
- Using the **Zoom Level** slider to increase or decrease the range of layers shown in the display; i.e., to the zoom level that best displays the number of layers in the model.

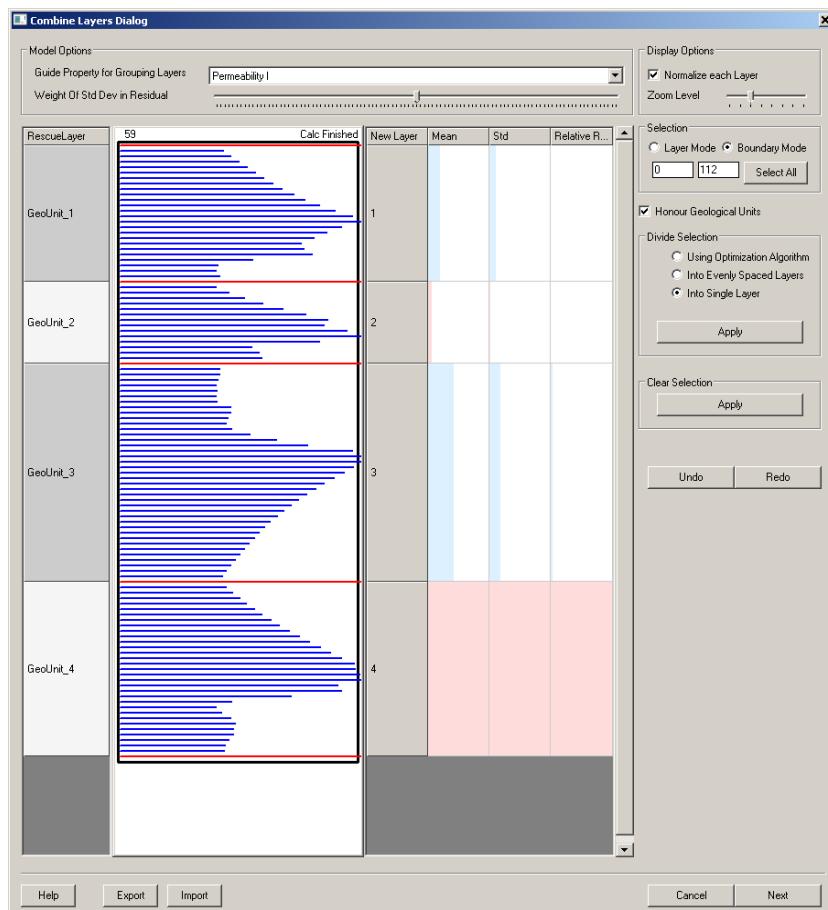
In our example, we selected **Normalize each Layer**, zoomed out to show all layers, and selected **Honour Geological Units**, as shown below:



As shown above, by selecting **Honour Geological Units**, the right pane is now divided into four layers, with boundaries aligned with the geological units.

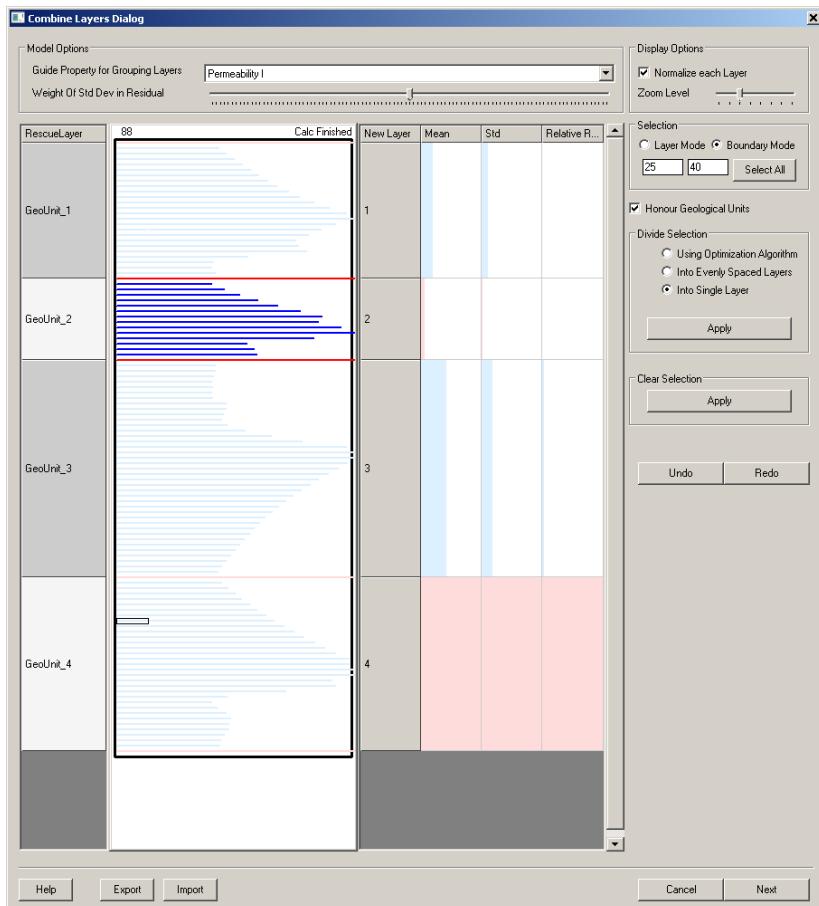
7. In the **Selection** area:

- Select **Layer Mode** to display the mean and standard deviation of the property for each layer, as shown in the above example. When you select layers in this view, the colours of the layers will change from light blue and light red to dark blue and dark red (an alternating two-colour scheme has been chosen to help delineate the layers in the display).
- Select **Boundary Mode** to show the boundaries between the levels.



As shown above, boundaries between the layers of the original model that have not been carried forward to the new model are displayed in blue. Boundaries that have been carried forward to the new model are shown in red.

If you click a cell in the **New Layer** table, such as Layer 2, the boundaries of the original model that are in that layer will be highlighted in dark blue, and the boundaries in other layers will be displayed in light blue, as shown below:



You can select a range of layers or boundaries in several ways. As discussed above, you can select all of the imported layers that are physically inside a new layer by clicking the new layer (for example, New Layer '2' in the above example). As well, in the left side, you can click the layer or boundary at the high or low end of a range then, holding the button down, drag the mouse to define the other end of the range.

Alternately, enter the layer or boundary numbers at the beginning and end of the range in the edit boxes provided in the **Selection** area, near the top right side of the display. When you do this, you need to tab out of the text box for the selection to take effect. If you click **Select All**, all layers in the model will be selected.

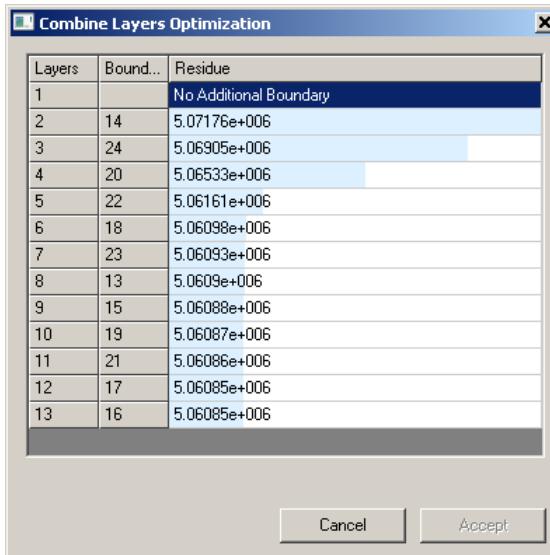
If you select a range in the new model, the selection will be constrained by the boundaries defined for the new model.

8. If the geological unit associated with each grid layer is available (for example, when the grid was directly imported from a RESCUE model), then the option **Honour Geological Units** will be available. If you check this option, then Builder will not combine layers from two different geological units when searching for the optimal combination scheme.
9. Once you have selected a range of imported levels, you can use the options in the **Divide Selection** area to divide the range into new levels, as follows:

Note: In all of the following, clicking **Apply** will remove all boundaries in the selected range and place boundaries at the top and bottom of the range. If **Honour Geological Units** is checked, the corresponding boundaries will not be erased.

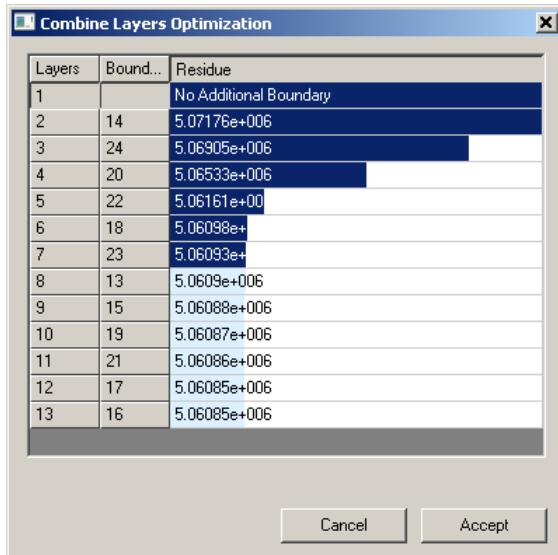
- a) Using Optimization Algorithm: Select **Using Optimization Algorithm** then click **Apply**. The Builder optimization algorithm will determine the optimum division. The **Combine Layers Optimization** dialog box will be displayed.

As the optimization algorithm proceeds, a list of boundaries is generated, starting with the geological boundaries, if applicable. This will be followed by a list of all of the possible boundaries, from most to least optimal. For very large datasets, this calculation may take some time, in which case a progress bar will be displayed. When the calculation for each iteration is complete, the optimal layer number, and a bar graph showing the total remaining residual, are displayed, as shown in the following example:



The residue values are normalized. They are the square root of the residues (as defined in the SPE paper) divided by the average property value for the whole reservoir.

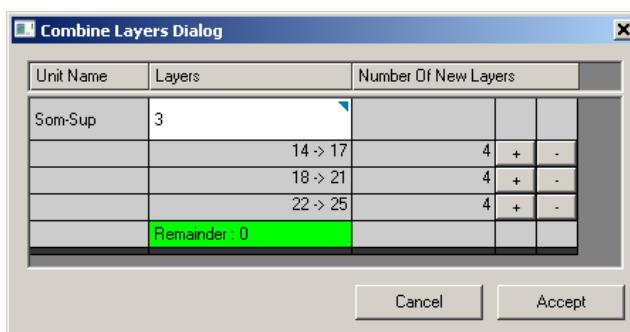
In the above example, you may choose to divide the selection into seven layers by clicking ‘7’ in the **Layers** column of the table. The new layers for the selected range will be highlighted, as shown below:



Note: As you select the number of layers, this selection will immediately be displayed in the **Boundary Mode** display; however, until you click **Accept**, the division will not be applied to the new model.

Click **Accept** to accept the division. This will be reflected in the **Combine Layers Dialog**.

- b) Into Evenly Spaced Layers: Select **Into Evenly Spaced Layers** then click **Apply**. The **Combine Layers Dialog** will be displayed, as shown in the following example:

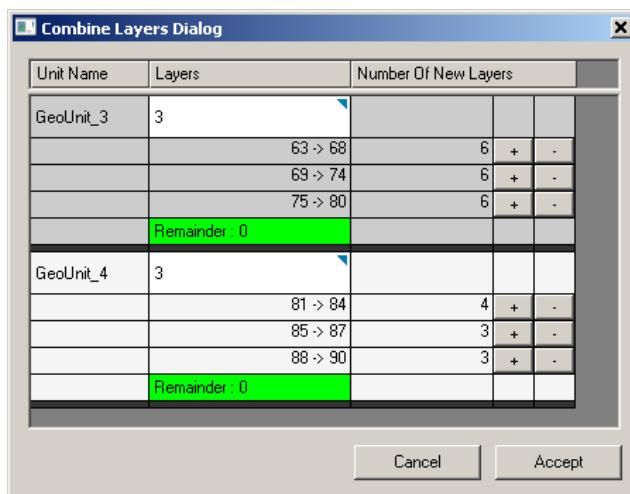


You will not be able to **Accept** the results of the dividing process unless all of the layers have been accounted for. If all of the layers have been accounted for, the remainder showing will be ‘0’, and the cell at the bottom of the table will be green, as shown above. If the number of new layers does not add up to the number of selected layers in the unit, the

remainder box will be red, informing you of the number of layers that will need to be added.

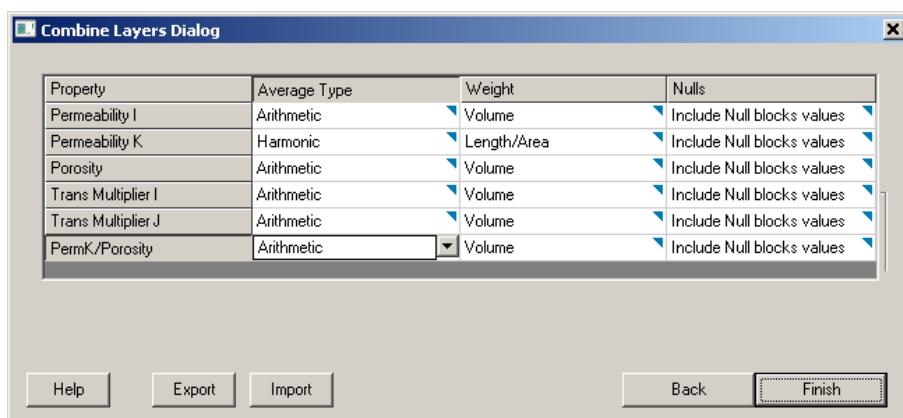
You can click the blue arrow ▲ in the upper right hand corner to increase or decrease the number of layers into which you want to divide the selection, or click + or - to move the boundary between the new layers up or down. Click **Accept** to accept the division. This will be reflected in the **Combine Layers Dialog**.

If you have selected **Honour Geological Units** and you select a range that bridges geological boundaries, Builder will split the range into two or more ranges, each separated by a boundary, as shown in the following example:



- c) **Into Single Layer:** Select **Into Single Layer** then click **Apply** to immediately combine the selected layers into one layer. No further action is taken.

10. Once you have divided the new model into the desired layers, click **Next**. The following **Combine Layers Dialog** is displayed:

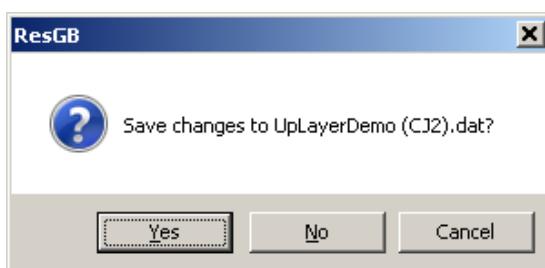


The **Combine Layers Dialog** contains the table of property names and averaging methods to be used for the properties. Initially, the default method is displayed for each property. You can change the average method, weight and treatment of nulls by clicking ▾ in the associated cell to open a drop-down list of options then selecting the desired one.

Note: Through the **Combine Layers Dialog**, you can apply an EQUALSI specification or formula, where available, to the new grid. If you uplayer a property that has a formula using another average type, the specification will be removed but the formula will be retained.

Geometric properties, such as grid top, bottom, thickness and net pay, are not shown in the table because they are handled internally with their most appropriate uplayering methods. The values of integer properties for combined blocks are based on the ‘volume majority’ rule; i.e., the combined blocks are assigned the integer value with the greatest volume. NULL block combination uses a different rule – the resulting block is NULL only if all contributing blocks are NULL.

11. Once the desired average type, weight and nulls are chosen for each property, click **Finish** to start the uplayering process. There is no undo for this operation. If you do not like the results of the uplayering and want to try a different layer combination scheme, you will have to re-read the original dataset and start the Combine Layers process again.
12. To save your results, click **Yes** when prompted on exit:

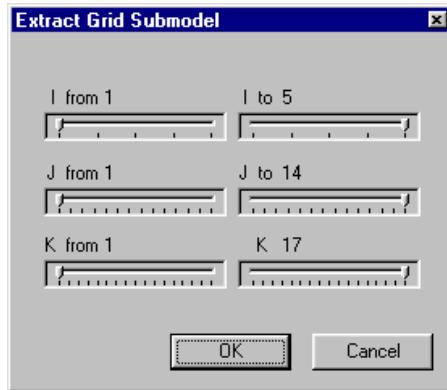


Extracting Sub-sections of Grids

Note: For information about the processing of property modifications, refer to [Property Modifications \(MODs\)](#).

To extract a sub-section of a grid:

1. Enter **Edit Grid** mode as described earlier.
2. Select **Extract SubModel** from **Reservoir, Edit Grid** menu. The **Extract Grid Submodel** dialog box is displayed:



3. Adjust the beginning and ending **I**-, **J**-, and **K**- **from** and **to** sliders until the desired sub-section of the grid is highlighted.
4. Click **OK**.
Note: There is no “Undo” for this operation, because of the potentially large memory required to back up the grid and all the properties.
5. When you are finished modifying the grid, exit **Grid Edit** mode. It is recommended that you examine the well perforation locations, sector definitions and other features of the grid to ensure that the grid modifications have been performed as you expected.

Editing Cartesian, Radial or Orthogonal Grids

1. Enter the **Edit Grid** mode as described earlier.
2. The grid rotation, translation, grid line editing and grid refinement operations are described in [Creating a Simulation Grid Using Structure Maps](#).
3. When done, click the **Probe Mode** button on the modes toolbar to exit **Edit Grid** mode.

Editing Non-orthogonal Corner Point Grids Created Using Structural Maps

1. Enter the **Edit Grid** mode as described earlier. This puts you in last stage of the non-orthogonal corner point grid creation wizard.
2. You can add grid block refinement at this stage without destroying the property and well data. If you only intended to add refinement, click **Finish**.
3. If you want to modify the control points, click the **Previous** button. A message warning that if you proceed further all your property and well data will be destroyed comes up. Click **Yes** in the message box and the instructions in [Creating a Non-orthogonal Grid with Sloping Faults Using Structure Maps](#).

Editing Grids Created Using RESCUE Surfaces

1. Enter the **Edit Grid** mode as described earlier.
2. You can add grid block refinement at this stage without destroying the property and well data. If you only intended to add refinement, choose the first option in the **Grid editing options** dialog box that comes upon entering **Edit Grid** mode. Proceed as described in *Adding and Removing Refined Grids*.
3. If you selected the second option in **Grid editing options** dialog box, the **Editing grid created from RESCUE model** dialog box is displayed. Select one of the options and then click **OK**.
4. If you selected the first option, the **Set autogrid construction options** dialog box appears and you can proceed from step 6 of *Creating a Simulation Grid Using 3D Surfaces from RESCUE Model*.
5. If you selected the second option, you can proceed from step 5 of *Creating a Simulation Grid Using 3D Surfaces from RESCUE Model*.
6. If you selected the third option, you can proceed from step 3 of *Creating a Simulation Grid Using 3D Surfaces from RESCUE Model*.
7. If you selected the last option, you can proceed from step 2 of *Creating a Simulation Grid Using 3D Surfaces from RESCUE Model*.

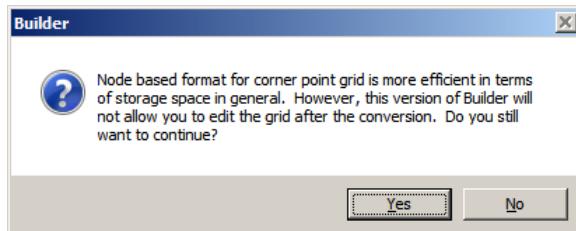
Converting an Existing Simulation Grid to a Node Based Format

Builder supports the conversion of an existing simulation grid to a node based format if the existing grid can be converted; that is, if it meets the following criteria:

- It is a corner point grid.
- It is specified using one of the following:
 - *CORNERS keyword
 - *COORD keyword
 - *XCORN/*YCORN keywords

To convert a grid that meets the above criteria to a node based format:

1. In the Builder menu bar, select **Reservoir | Convert to Node Based Format**. A message will be displayed, informing you that although the node based format is more efficient in terms of storage space, this version of Builder may not allow you to edit the grid after the conversion.



Note: The **Convert to Node Based Format** menu item will only be enabled if the grid can be converted.

2. Click **Yes** to proceed with the conversion. A message will be displayed informing you that the conversion is complete. Click **OK**.

Specifying, Calculating and Adjusting Structure and Rock Property Values

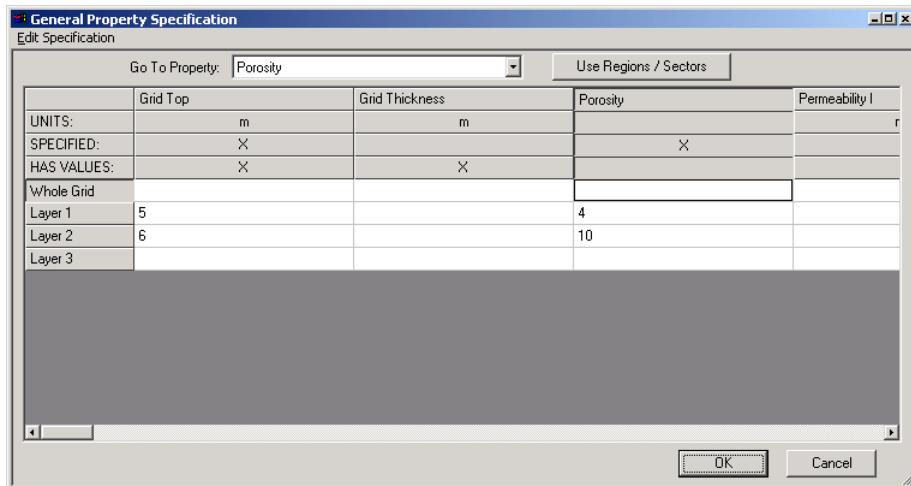
Once you have finished creating or modifying your simulation grid, you can specify how to assign, calculate, or interpolate values for structural and rock properties for each grid block. After you have specified how to calculate one or more properties, you command Builder to do the calculations. After the calculations are complete, the calculated property values can be displayed and modified.

Properties may be assigned (by layer or region), interpolated from maps, imported from a RESCUE model, or calculated using geostatistical routines. Builder has a number of map creation and geostatistical routines available. For further information, refer to [Creating Maps and Geostatistical Property Calculations](#). The routines are available through **Create map** under the **File** menu, and **Geostatistics** under the **Reservoir** menu.

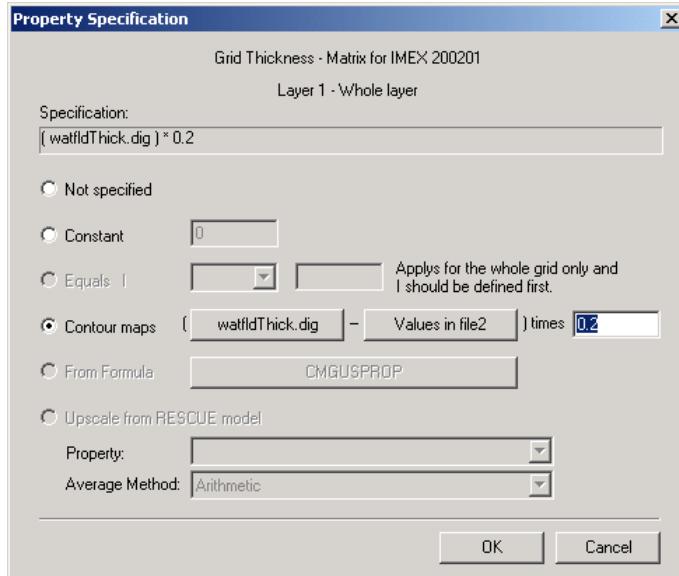
Specifying Property Interpolations and Calculations

To specify a property calculation:

1. You must be in **Probe Mode** to perform this operation. To enter Probe Mode, select **Probe Mode**  from the mode selection box, or right-click to display the context menu then click **Probe Mode**.
2. Select **Specify Property** from the **Reservoir** menu, or click the **Specify Property** button. The **General Property Specification** dialog box is displayed:

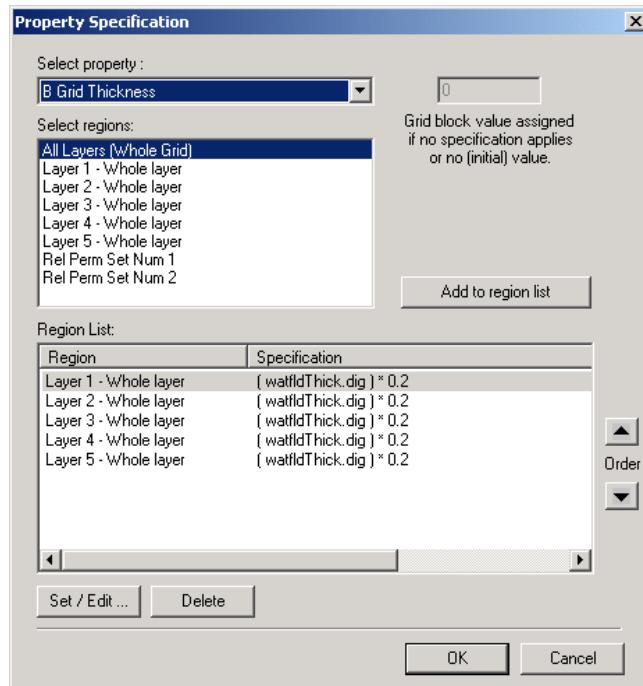


3. The dialog box lists the property names in columns and the simulation layers in rows. **Whole Grid**, which indicates an identical specification for all the layers, is also included. If a property is specified and/or has values, it is indicated in the top rows which are grayed out. Also displayed are the units of the property.
 4. Enter specifications for the property. Use the **Go To Property** combo box to select the property. This will bring the property column in the view. Or use the scroll bar at the bottom to bring the column into view.
 5. If you want to specify a constant property for a layer, simply type in the value. You can also use the **Cut**, **Copy** and **Paste** commands on the context menu (right-click in a cell) to delete, move or copy a specification from one cell to one or more cells. These and other menu items appearing on the context menu are also available in the **Edit Specification** menu at the top of the dialog box.
- Note:** Copy and Paste from Excel can also be done. This is restricted to numerical values only and only a single column at a time. If multiple columns are selected only the first column will be pasted.
6. To specify a geological map or formula, or to import/upscale from a RESCUE model, use the corresponding menu items. If appropriate, you can also specify EQUALSI for a property. The **Property Specification** dialog box is displayed:



7. Click the **Constant**, **Equals I**, **Contour maps**, **From Formula** or the **Upscale from RESCUE model** radio button.
8. For **Constant**, enter a single value to be applied to all grid blocks in the selected region.
9. For **Contour map**, click on the **Values in file 1** button and an **Open** dialog box will appear. Select the first map file. If you wish to calculate the property as a difference between values from two maps, specify the second map file. Enter a multiplier applied to interpolated values in the **times** text box.
10. For **From Formula**, see [Formula Manager](#) to learn how to create formulae to calculate one array property from one or more previously calculated array properties. This option is enabled only if at least one formula is defined.
11. If the grid was created using 3D surfaces from RESCUE and if properties are available in the RESCUE model, the **Upscale from RESCUE model** option will be enabled. To upscale from the RESCUE model
 - Select the **Upscale from RESCUE model** option.
 - Select the name of the RESCUE property you want to upscale from the **Property** drop-down list box.
 - Select the averaging method from the **Average Method** drop-down list box. Currently four methods are available: **Arithmetic**, **Geometric**, **Harmonic**, and **Harmonic/Arithmetic**. **Direct Import** should only be used if the grid was directly imported from a RESCUE model.

12. Click **OK** or **Cancel** to close the **Property Specification** dialog box.
13. If you want to enter specifications by sector or region, click the **Use Regions/Sectors** button. The **Property Specification** dialog box will be displayed:



14. Select the property to be specified from the **Select property:** list. The left column of the property list will contain one of the following codes:

| If you see... | Then the property is... |
|---------------|------------------------------------|
| — | Neither specified nor interpolated |
| B | Both specified and interpolated |
| S | Specified only |
| V | Values only – not specified |
15. Select regions to be specified by clicking on a region in the **Regions List**, then clicking on **Add to region list**. Regions can be the whole grid, individual layers in the grid, sectors, or PVT and rock type regions. You can select multiple regions at once using the usual Windows CTRL and SHIFT mechanisms.
16. Once you have added regions to the **Region List**, you can re-order the list using the **Order** **▲** and **▼** buttons. The calculations are done in the order of the list. If a grid block is in two regions, a value calculated in a later region calculation will override the earlier calculation done in the first region.

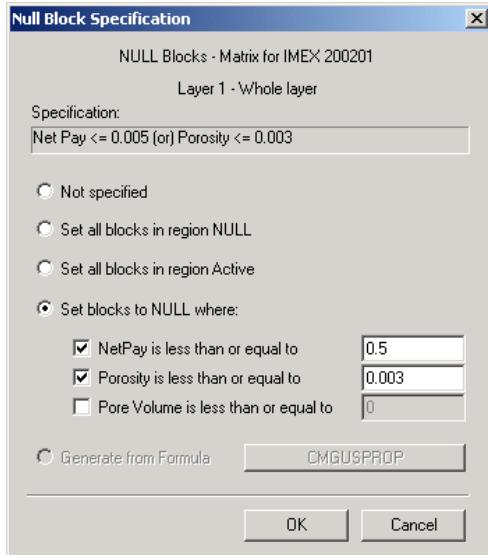
17. Next, specify how to calculate each region. Click on a particular region in the **Region List**, and click on the **Set/Edit...** button. The **Property Specification** dialog is displayed, as shown above.
18. Follow steps 7 – 12 to input a specification.
19. Select the next region and perform steps 6 through 11. Continue until all regions are specified.
20. You can select and specify additional properties. When you have specified all the properties you want to at this time, click **OK** to apply the specifications, or **Cancel**.

Notes:

1. **Grid Thickness:** You can define grid thickness as the difference between interpolated values of two contour maps times a multiplier. For example, you may wish to divide a single geological layer into two or more simulation layers. If you only have contour maps for the top and bottom of the geological layer, you can specify the thickness of each simulation layer as a fraction of the distance between two contour maps. In this case, select on map for **Values in file 1** and the second map for **Values in file 2**.
2. **Grid Top or Grid Bottom:** You do not need to specify all the layers for Grid Top or Grid Bottom. Layers that you do not specify are stacked below or above the specified layers. If you have not specified the top most layers, these layers will stack above the specified layers.
3. **Multipliers:** The multiplier, specified in the **times** text box, has several possible uses. As previously mentioned, it can be used to divide a single geological layer thickness over several simulation layers. It can also be used to do unit conversion. For example, the simulators expect porosity to be expressed as a fraction. If your contour map has porosity as a percentage, you can multiply by 0.01 to convert to a fraction. Finally, if you have an elevation map (Z direction measured positive upwards) you can convert this to a depth map (Z direction measured positive downwards) by multiplying by -1.

To specify NULL block calculations:

1. NULL block specification is similar to other property specification, described above, except for the dialog box used to specify each layer or region:



2. Select the desired specification. You may also need to enter a cut-off value for some of the specifications.
3. Click **OK** to save the specification, or **Cancel**.

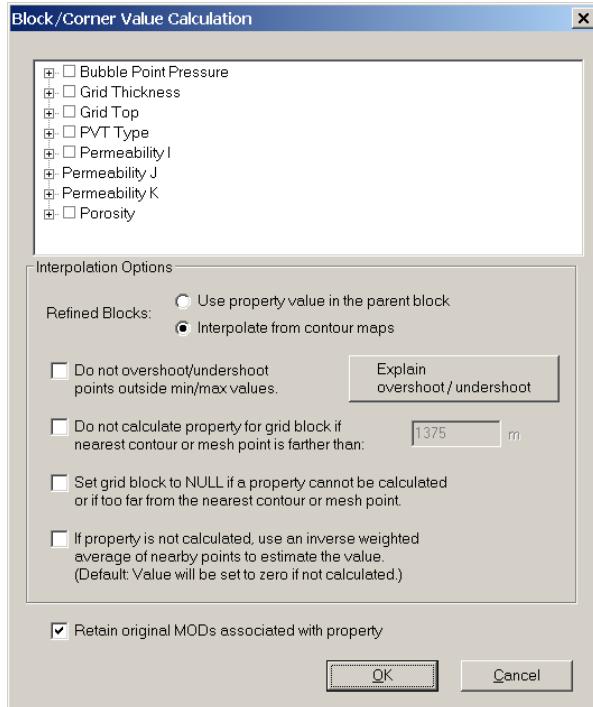
Performing Interpolations and Calculations

After you have specified one or more properties, you can command Builder to perform the calculations. You do not need to specify all the properties at one time. Instead, you can specify and calculate the properties one at a time.

To perform interpolations and calculations:

Note: For information about the use of the **Retain original MODs associated with property** check box, refer to [Property Modifications \(MODs\)](#).

1. You must be in **Probe Mode** to perform this operation. To enter Probe Mode, select **Probe Mode** from the mode selection box, or right-click to display the context menu then click **Probe Mode**.
2. Select **Calculate Properties** from the **Reservoir** menu, or click on the **Calculate Properties** button. The **Block/Corner Value Calculation** dialog box is displayed:



3. At the top is a list of all the properties that you have specified. Properties that need to be calculated are indicated with a check mark. You can force the recalculation of a specified property or defer the calculation of a property by checking or cancelling individual properties. Clicking on the icon beside a property will show the list of regions and specifications for the property. You can check individual regions to calculate only specific regions for a property. For example, if you have specified a property with a map interpolation for each grid layer, and you receive a new map for one particular layer, you can specify that only that layer be recalculated.
For the properties specified with EQUALSI, the check box will not be displayed.
4. In Property Values in **Refined Blocks**, select either **Use property value in parent block** or **Interpolate from contour maps**. Note that structural properties (Grid Top and Grid Thickness) always use the parent block values to determine refined block values. **Interpolate from contour maps** only applies to rock properties such as porosity and permeability.
5. Occasionally, the interpolation routine is unable to interpolate a value for a grid block because there are no nearby contour lines, the interpolation point is surrounded by faults, or all the mesh points in the vicinity are set to the “NULLFLAG” value. If this occurs when interpolating the “Grid Top” property, the top value at the interpolation point is set to zero. If you want these blocks to be set to NULL, select **Set grid block to Null**.
6. Click **OK** to begin the calculation, or **Cancel**.

While the interpolation and calculation is being carried out, progress messages will be displayed on the status bar at the bottom of the main Builder window.

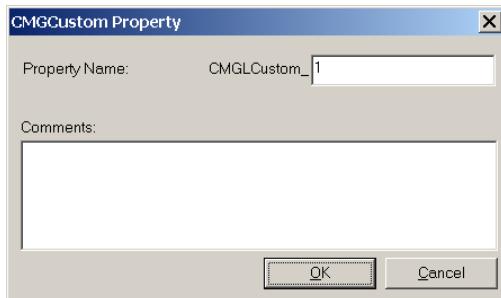
Once the property calculation is complete, one of the calculated properties will be displayed in view. For information on changing view settings and content, see [Changing Display Content and Settings](#).

Deleting a Property

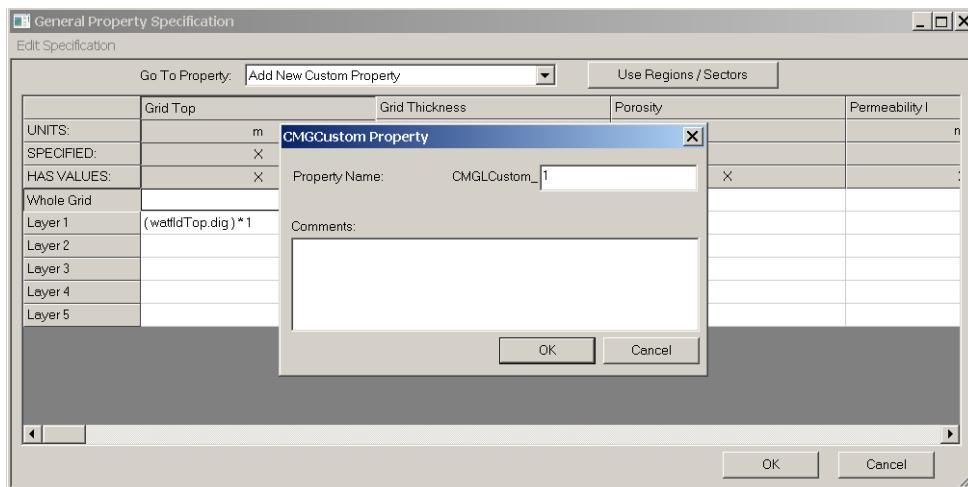
If you wish, you may delete a calculated or imported property. From the **Reservoir** menu, select **Delete Properties...** The **Delete Properties** dialog box will be displayed with a list of the calculated properties (not including structural properties like Grid Top and Grid Thickness). Select a property (or properties) to delete, and click **OK**.

Defining Custom Property

To define custom properties, from the **Reservoir** menu, select **Add New Custom Property**. The **CMGCustom Property** dialog box will be displayed, allowing you to type in the name of the custom property and comments lines:



The new custom property can also be defined in a number of places when the property drop-down combo-box is available. It is done by selecting the very first item, **Add New Custom Property**, in the combo-box. The following graph shows that a new custom property is to be created from within the **General Property Specification** dialog box, as shown below:



The existing six temporary properties, CMGLTemp PropX (where X = 1 to 6), if defined in an existing dataset, will be automatically converted to the corresponding custom properties, CMGLCustom_TempX (where X = 1 to 6), when the dataset is read in.

Importing Spatial Properties

You can import spatial properties in a number of ways:

1. Import the properties along with the simulation grid from files output by geological packages (refer to [Importing 3D Simulation Grids and Grid Properties](#)).
2. Import the properties along with the simulation grid from CMG dataset files (refer to [Importing 3D Simulation Grids and Grid Properties](#)).
3. Import properties from CMG dataset files without importing the grid.
4. Import properties from a text file containing an array of values.

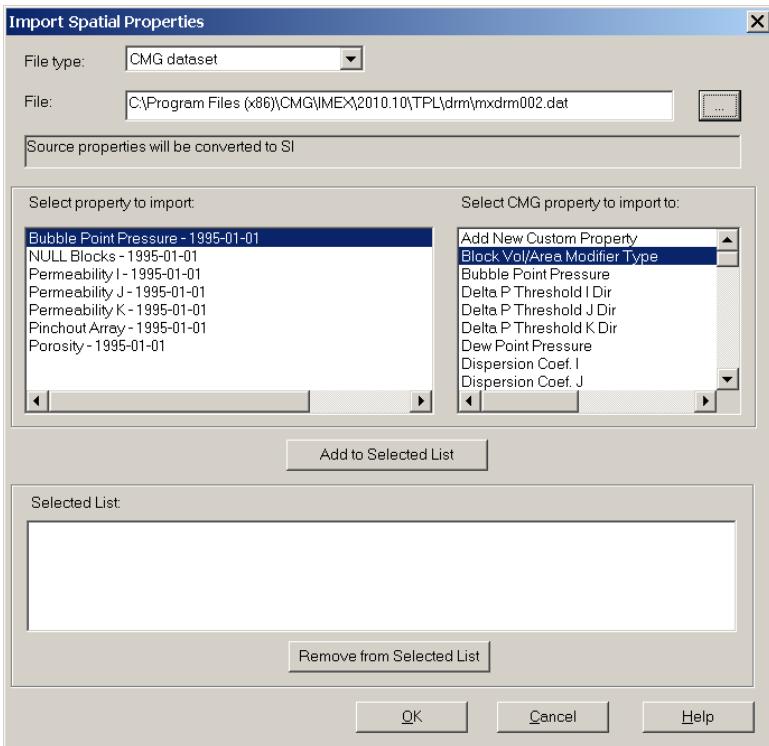
Importing Properties from CMG Dataset Files without Importing the Grid

From the **File** menu select **Import from another file** and then select **Spatial properties from dataset**. Specify the dataset file from which to import the property. The **Import spatial properties** dialog box is displayed. The importable properties in the source file are listed in the list on the left. Select a property name from the list on the right. Click **Add to Selected List** button. The source property will be imported as the property you specified in the list on the right.

Values will be converted to account for different unit systems in the two dataset files.

Properties will not be imported unless the numbers of fundamental blocks in the two files are identical. If the target dataset has local grid refinement but the source dataset does not, only the fundamental grid block values will be imported and the refined blocks will inherit from the parent blocks. If both the datasets have local grid refinement, the refined block values are imported only if the total numbers of grid blocks – fundamental and the refined – in the two files are identical.

Note: Builder does not validate the location and size of each refined grid; that is, it assumes that each refined grid is located in the same parent block in the two files and that it is of the same size in the two files.



Importing Properties from CMG Simulation Results Files

From the **File** menu select **Import from another file** and then select **Spatial properties from CMG simulation results**. Specify the dataset file from which to import the property. The **Import spatial properties** dialog box is displayed. The importable properties in the source file are listed in the list on the left. Select a property name from the list on the right. A property may be listed a number of times if it was output more than once in the simulation output file. Click **Add to Selected List** button. The source property will be imported as the property you specified in the list on the right. This selection enables you to initialize a property with values at times later than the zero time from another simulation run.

Values will be converted to account for different unit systems in the two dataset files.

Properties will not be imported unless the numbers of fundamental blocks in the two files are identical. If the target dataset has local grid refinement but the source dataset does not, only the fundamental grid block values will be imported and the refined blocks inherited from the parent blocks. If both the datasets have local grid refinement, the refined block values are imported only if the total numbers of grid blocks – fundamental and the refined – in the two files are identical.

Note: Builder does not validate the location and size of each refined grid; that is, it assumes that each refined grid is located in the same parent block in the two files and that it is of the same size in the two files.

Importing Properties from a Text File Containing an Array of Values

This feature reads one or more array values from an ASCII text file. If the file contains more than one value array, each array should begin with a keyword. The keyword could be a valid array keyword but is not required to be so. A grid must be defined in the current dataset.

The number of elements in the value array must be identical to the number of fundamental or fundamental + refined grid blocks.

From the **File** menu select **Import from another file** and then **Spatial properties from stand-alone arrays**. Specify the dataset file from which to import the property. The **Import spatial properties** dialog box is displayed. The importable properties in the source file are listed in the list on the left. Select a property name from the list on the right. A property may be listed a number of times if it was output more than once in the simulation output file. Click **Add to Selected List**. The source property will be imported as the property you specified in the list on the right. Values are assumed to be in Builder's current unit system.

If the target dataset has local grid refinement but the source file does not, only the fundamental grid block values will be imported and the refined blocks inherit from the parent blocks. If both have local grid refinement, the refined block values are imported only if the total numbers of grid blocks – fundamental and the refined – in the two files are identical.

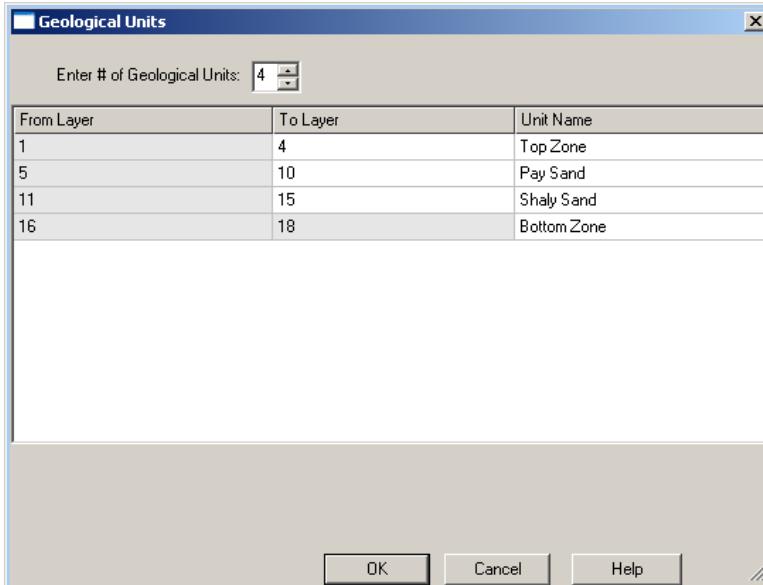
Note: Builder does not validate the location and size of each refined grid; that is, it assumes that each refined grid is located in the same parent block in the two files and that it is of the same size in the two files.

Defining/Editing Geological Units

Builder allows you to define geological units. A geological unit is defined as a group of contiguous grid layers all belonging to the same geological formation or reservoir unit. A geological formation is typically bounded by a top and a bottom geological surface. In such a case, the grid layers included between the top and bottom surface will form a natural definition. In any case, one should strive for homogeneity when defining a geological unit. Homogeneity in the porosity and/or permeability values is especially important because all the Builder's methods for geostatistical property estimation operate within a geological unit. Using geological units is a way to avoid mixing very different data when using the geostatistical methods.

To define geological units:

1. Select **Geological Units** from the **Reservoir** menu. The **Geological Units** dialog box is displayed:



If the geological units are defined, the dialog box will show the current definition as in the example above. Note that the geological units are usually defined when importing a RESCUE model (refer to [Importing 3D Simulation Grids and Grid Properties](#)). If no geological units are defined, the dialog box will show only one unit including all grid layers. By default, all layers belong to the same geological unit.

2. Select the **Number of geological units** with the spin box. The number of geological units that can be defined is limited between 1 and N where N is the total number of grid layers. The geological units must be contiguous, non-overlapping, and no gaps are allowed. Each grid layer must belong to one and only one geological unit. The **Geological Units** dialog box is designed to enforce those rules.

The **From Layer** cell of the first geological unit is always 1 and cannot be edited. Similarly, the **From Layer** cell for all other geological units cannot be edited. Instead, they are automatically updated to **To Layer + 1** when the **To Layer** cell of the previous geological unit is edited.

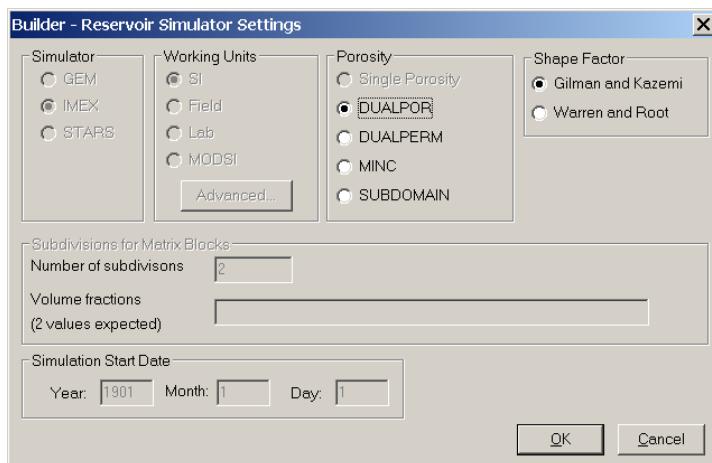
The **To Layer** cell of the last geological unit cannot be edited. It is always defined as N, the total number of grid layers.

The **Unit Name** can be changed at will.

3. Click **OK** to save the newly defined geological units. The **Cancel** button allows you leave the dialog box without saving.

Converting to Dual Porosity

To convert a single porosity dataset to one of the Dual Porosity simulator options, select **Convert to/edit fracture reservoir** from the **Reservoir** menu. The **Builder - Reservoir Simulator Settings** dialog box will be displayed:



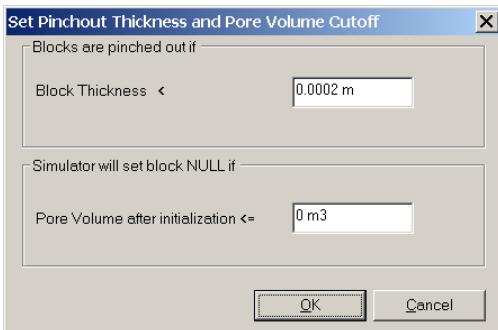
Select the desired dual porosity option to use then click **OK** to perform the conversion. After the conversion is done, you will need to specify and calculate grid properties for the fractures, as well as enter other required data.

Pinching Out Thin Grid Blocks

Very thin grid blocks may slow down a simulation run, while only containing an extremely small fraction of the total reservoir volume. You can pinch-out grid blocks (remove them from the active simulation grid, while maintaining reservoir flow in a vertical direction through the blocks). By default, only extremely thin grid blocks are pinched out (thickness less than 1.0e-4 units).

To automatically pinch-out thin reservoir blocks:

From the **Reservoir** menu, select **Block Pinch-out thickness setting**. The **Set Pinchout Thickness and Pore Volume Cutoff** dialog box is displayed:



Enter a new value for the pinchout thickness then click **OK**. Builder will mark all blocks less than the specified thickness as pinchout blocks.

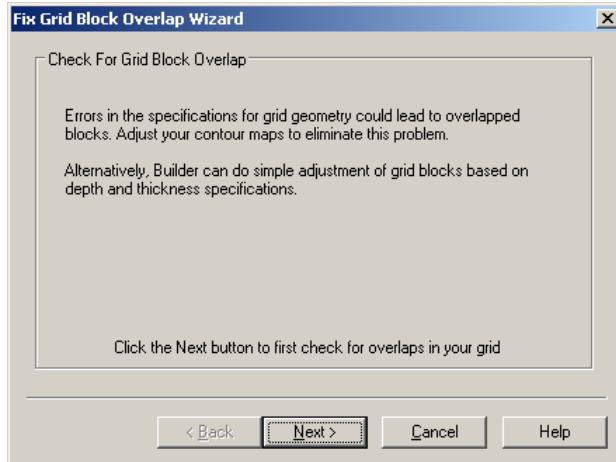
Fixing Overlapping Blocks

Whenever the grid structure is changed by an interpolation, the grid is checked to see if grid blocks overlap. Such an overlap could occur if the contour data for two grid tops and the thickness of the layer(s) in between are not consistent. For example, the map you give for “Grid Top” in layer 1 may, at points, dip below the map you give for “Grid Top” of layer 3. Builder includes a feature that allows you check for overlaps in the grid. If overlaps are found, an option is provided to select the grid depth specification (property and layer) that you consider most reliable and to “fix” the grid block overlap based on this map.

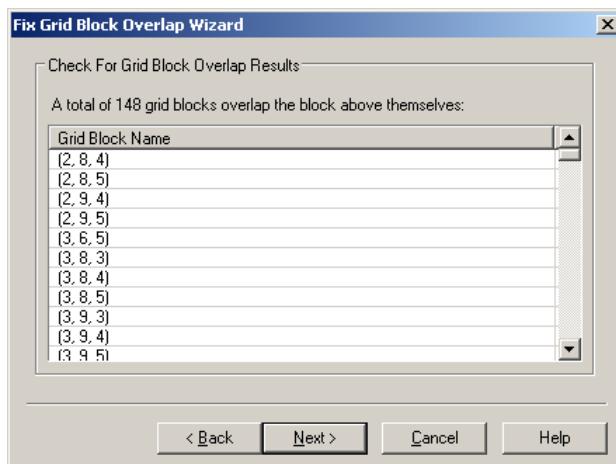
The property and layer indicated as the most reliable data will be left unaltered, and other properties will be adjusted to eliminate grid block overlap. For each column of overlap, Builder will first fix the location of the block in the selected layer using the selected depth property specification, and then alter the depths of other blocks in the column by adding or subtracting the grid block thickness.

To fix grid block overlap:

1. Select **Fix Grid Block overlap** from the **Reservoir** menu. The **Fix Grid Block Overlap Wizard** dialog box is displayed:



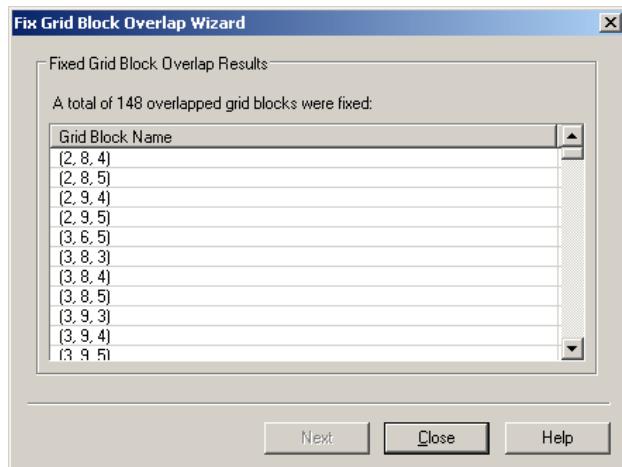
2. Click **Next** to perform the check for overlaps in your grid. If overlaps are found in your grid then a list of overlapped blocks will be listed, as shown in the following example:



3. Click **Next** again to proceed to the Fix Grid Block Overlap options page. Select the property that you consider most reliable (that is, which data is better, the data used to calculate the Grid Top, or the data used for the Grid Thickness) and which layer you consider most reliable, as illustrated in the following example:



4. Click **Next** to apply the fix, or **Cancel** to proceed without fixing any overlaps. Once the fix is complete, the final wizard page lists the grid blocks that were fixed.

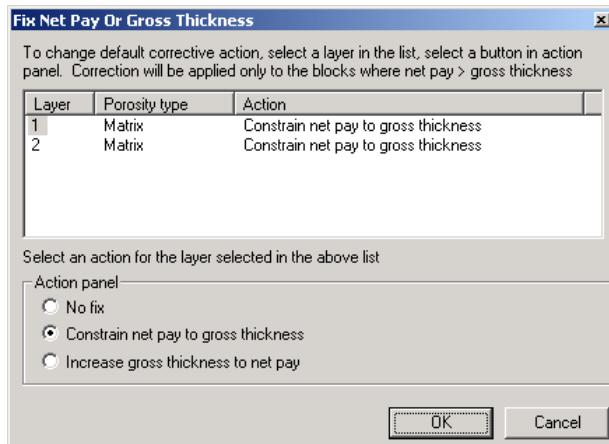


Fixing Net-to-gross Ratio Problems

If you calculate both Grid Thickness (that is, gross thickness) and Net Pay, Builder will check that the net-to-gross ratio is one or less. If the net pay is greater than the gross pay in any grid block, a message box will inform you.

To fix net-to-gross ratio problems:

1. Select **Fix Net-to-Gross ratio** from the **Reservoir** menu. The **Fix Net Pay Or Gross Thickness** dialog box is displayed:



2. The simulation layers with problems will be listed. Select a **Layer** and **Porosity type** by clicking on the layer number, and then choose one of the actions listed in the **Action panel** area.
3. After the action for all problem layers has been specified, click **OK** to adjust values, or **Cancel**.

Setting Transmissibility Multipliers at Sealing or Partially Sealing Faults

CMG simulators allow you to set transmissibility multipliers in the I, J and K directions, via the *TRANSI, *TRANSJ, and *TRANSK keywords. These array properties have been used to stop or reduce fluid flow across faults, and for sealing or partially sealing faults. Builder had a convenient way of setting these arrays through the **Set/Edit Transmissibility Multiplier** dialog box.

With the 2005 release, a new simulator keyword (*TRANSF) has been developed to specifically handle setting transmissibility multipliers across faults. A separate *TRANSF keyword gives the transmissibility multiplier and fault locations for each fault.

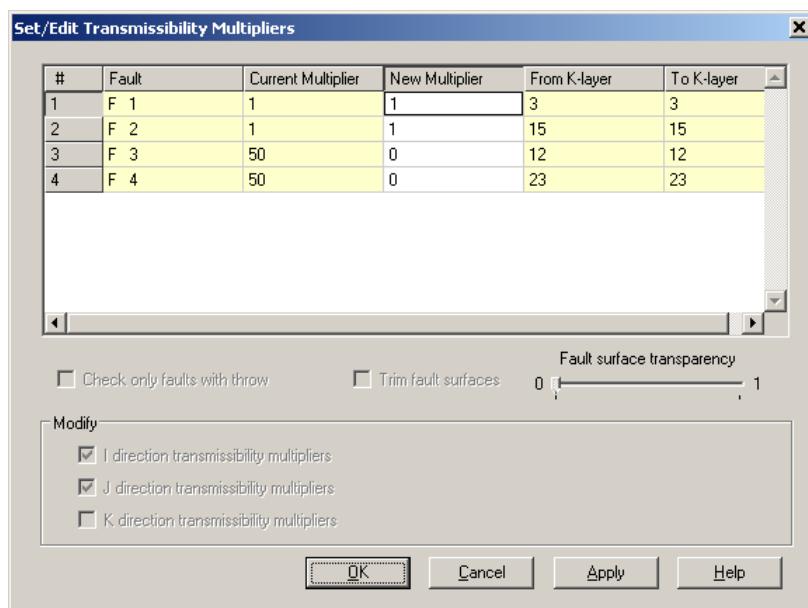
For continuity, if you have an existing dataset with *TRANSI, *TRANSJ, or *TRANSK, then Builder 2005 and later will modify these arrays to set transmissibility multipliers across the faults. Builder 2005 and later create *TRANSF keywords to communicate fault information to the simulator. Builder's method of setting the transmissibility multipliers across the faults is the same as in previous versions.

If you have a Grid Top map with faults that was used to create the grid, or the grid was imported from a RESCUE model with fault surfaces, then you will be able to set transmissibility multipliers separately for each fault. If you are using a grid created directly by a geological modelling program and you do not have the top map, then you will only be able to apply a single, common transmissibility multiplier across all faults

To set transmissibility multipliers across faults:

1. You must be in **Probe Mode** to perform this operation. To enter Probe Mode, select **Probe Mode** from the mode selection box, or right-click to display the context menu then select **Probe Mode**.
2. Open the Grid Top map with fault lines if you have one, or open the RESCUE model that the grid was imported from. If a grid is imported from a RESCUE model by Builder 2004.15 or later, the fault locations may be stored in the dataset, and re-opening the RESCUE model may not be required.

Select **Set Transmissibility Multiplier Across Faults** from the **Reservoir** menu. The **Set Transmissibility Multiplier Across Faults** dialog box is displayed:



3. As you select different faults from the fault list, grid blocks adjacent to the selected fault(s) will be highlighted. If you do not have the top map with fault information open, then only one fault will appear on the list, and all grid blocks adjacent to faults will be selected.
4. Enter a value for the transmissibility multiplier in the **New Multiplier** column. A value of 0.0 indicates a completely sealing fault (no flow across the fault), and a value of 1.0 indicates a non-sealing fault.

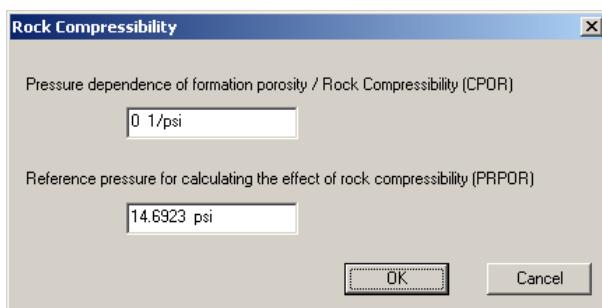
Note: The simulator will take into account the fault throw when calculating flows and connections across the fault. With sufficient throw, layer 1 on one side of the fault may connect to layers 2 and 3 on the other side of the fault. Consult the simulator manual for details.

5. By default, the new transmissibility multiplier will be applied to both I and J directions. However, you may choose otherwise in the **Modify** area.
6. Click **OK** or **Apply** to apply the changes, otherwise **Cancel**.

Note: The **Trim fault surfaces** and **Fault surface transparency** options become active for RESCUE models. Both options are useful for enhancing the faults display in Builder's view.

Entering Rock Compressibility

To specify rock compressibility, select **Rock Compressibility** from the **Reservoir** menu. Fill in the required values, and then click **OK**.

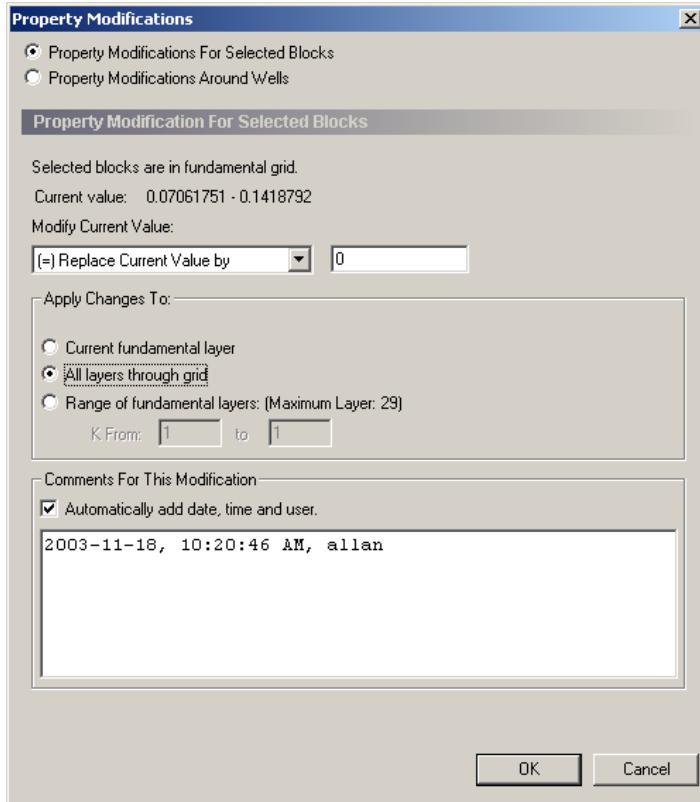


Editing Property Values of Selected Blocks

After property calculations, most property values may be modified.

To edit (modify) a calculated property value:

1. You must be in **Edit Reservoir Property** mode to perform this operation. To enter Edit Reservoir Property mode, select **Edit Reservoir Property** from the mode selection box, or right-click to display the context menu then select **Edit Reservoir Property**.
2. Select the property you want to edit from the **Property Selection** list.
3. Click on the grid block to modify, use CTRL+click to select a set of grid blocks, or drag out a rectangle of grid blocks. When you release the mouse button, the **Property Modifications** dialog box is displayed:



4. You can replace the value in the selected blocks or apply a simple arithmetic operation, with a constant operand, to the blocks. Select the operation and the operand value under **Modify Current Value**.
5. You can apply the operation to the **Current fundamental layer**, to **All layers through grid**, or to a **Range of fundamental layers**. You may choose **Visible blocks only** if there are refined grid blocks in the selection. You can add comments about the modification.

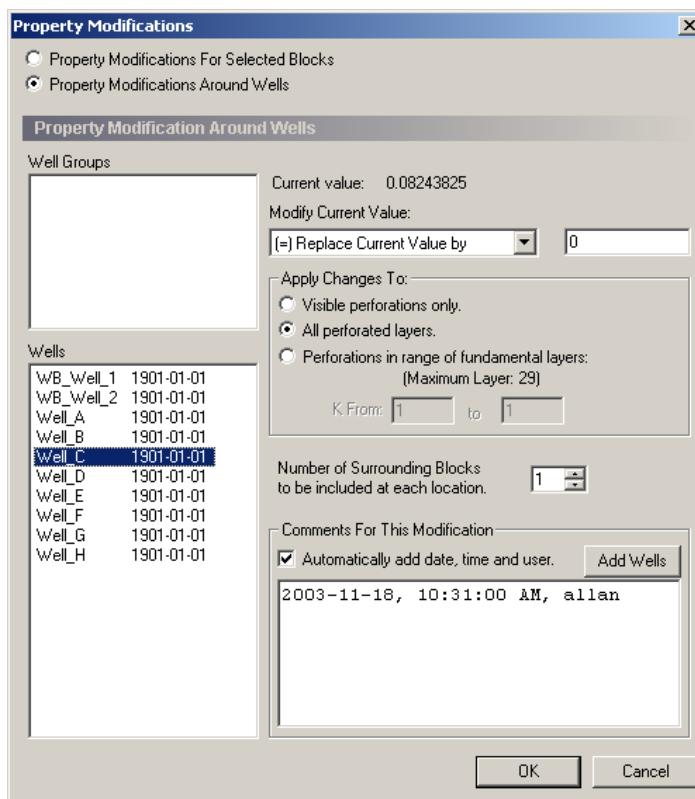
Note: Builder 2004 and later store *MOD keywords in the simulator dataset. The comments you add will be stored with the *MOD.
6. Click **OK** to apply the change, or **Cancel**.

Editing Property Values around Wells

You can select a well or all the wells in a well group, and Builder will select all of the blocks around the well(s) to apply a property modification.

To edit (modify) a calculated property value:

1. You must be in **Edit Reservoir Property** mode to perform this operation. To enter Edit Reservoir Property mode, select **Edit Reservoir Property** from the mode selection box, or right-click display the context menu and then select **Edit Reservoir Property**.
2. Select the property that you want to edit from the **Property Selection** list.
3. Click the grid block containing the well. When you release the mouse button, the **Property Modifications** dialog box is displayed:



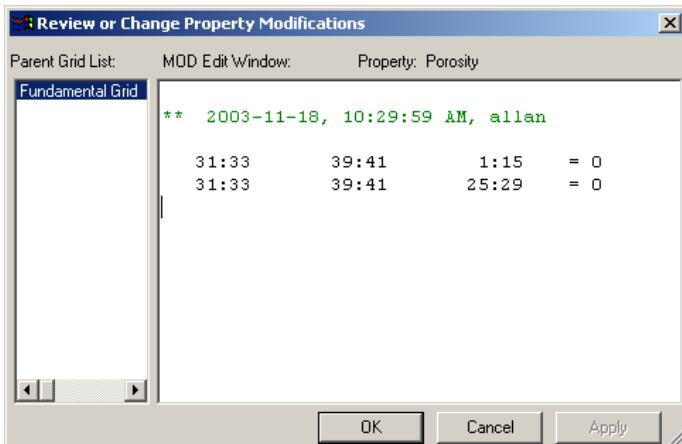
You can select more than one well. If you have well groups defined in your dataset, you can click a well group to select all the wells in that group.

4. You can replace the value in the selected blocks or apply a simple arithmetic operation, with a constant operand, to the blocks. Select the operation and the operand value under **Modify Current Value**.

5. You can apply the operation to the **Visible perforations only**, to **All perforated layers**, or to **Perforations in a range of fundamental layers**.
6. You can change the **Number of Surrounding Blocks** to be included in the selection along with the perforated blocks. You can add comments about the modification.
Note: Builder 2004 and later store *MOD keywords in the simulator dataset. The comments you add will be stored with the *MOD.
7. Click **OK** to apply the change, or **Cancel**.

Reviewing Property Modifications

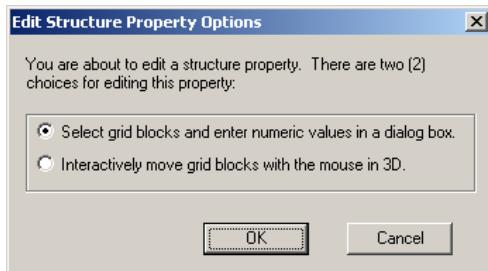
To review the modifications that you have made to a particular property, click on the **Reservoir** button of the tree view. Open the property list if it is not already open (click beside **Array Properties**). Scroll to the property that you want to review, and open the specification list (click beside the property name). If the property has been modified, you will see a line “Property Edited (*Mod keyword)”. Double-click this line. The **Review or Change Property Modifications** dialog box will be displayed.



You can edit, comment out, or delete modifications as necessary.

Editing Grid Structure Values

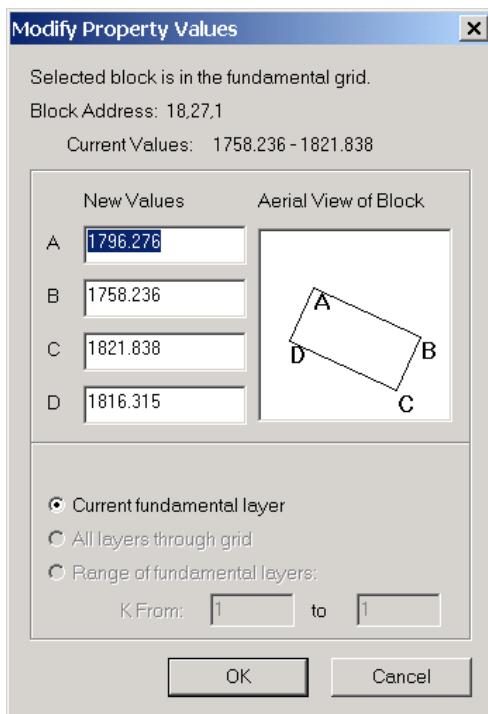
If the property to be edited is a grid structure property (Grid Top, Grid Bottom, Grid Paydepth, or Grid Thickness), the **Edit Structure Property Options** dialog box, providing you with two options for editing the property, is displayed:



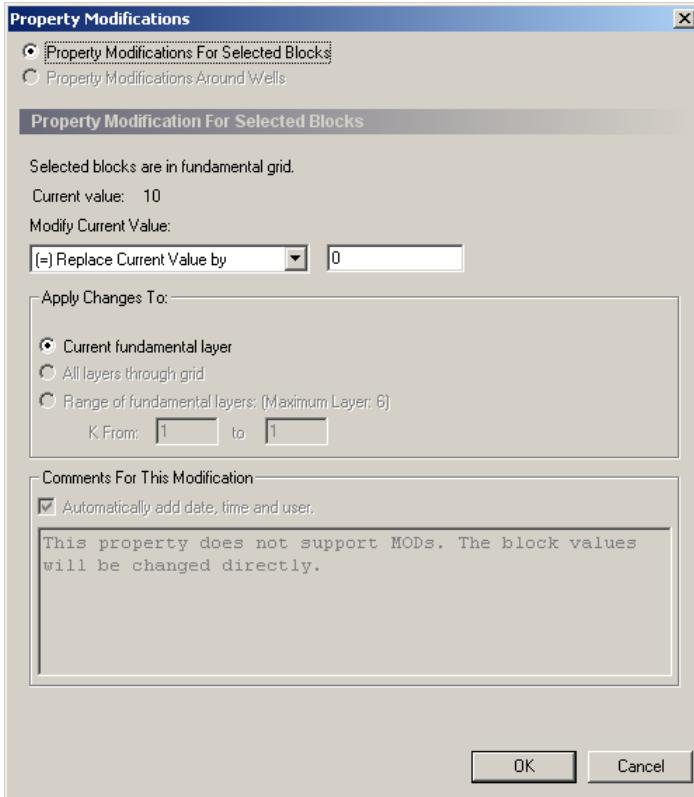
Enter grid structure property values in a dialog box:

If **Select grid blocks and enter numeric values in a dialog box** is selected, when the **OK** button is clicked, you must select one or more grid blocks. When the mouse button is released, depending on the type of grid used (Cartesian, Corner Point, Radial, and so on) one of the following dialog boxes will appear.

For Corner Point grid the following dialog box appears. You must enter numeric values for each block corner in the **Modify Property Values** dialog box:



For Non Corner Point grid the Property Modifications dialog box is displayed. You enter a single block value for the property:

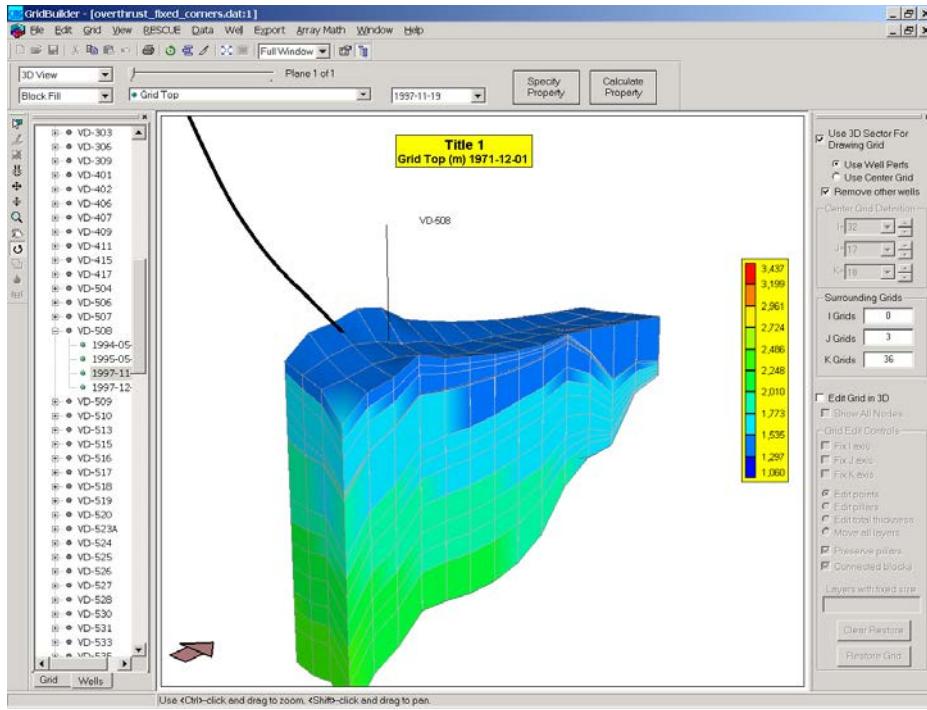


3D Editing Options

If you selected **Interactively move grid blocks with the mouse in 3D** in the **Edit Structure Property Options** dialog box, a dockable window will appear on the left side of the screen. This dockable window can be moved and/or docked in alternate locations by clicking and dragging the top part of this dockable window.

Use a 3D sector for drawing the grid:

Often it is desired to view only a portion of the 3D grid to simplify grid editing and to aid in quality control of the model. The top portion of the dockable window (shown on the left of the picture below) contains controls for viewing 3D sectors of the grid:



If **Use 3D Sector for Drawing Grid** is selected, two sub-options are enabled - **Use Well Perfs** and **Use Center Grid**. When **Use Well Perfs** is selected, you must select a well and/or date from the tree view shown on the left side of the above example. Each time a new well or date is selected the window will change to be the grid blocks surrounding the well perforations on that particular date.

The number of surrounding grids is controlled by the values entered in the **Surrounding Grids** area. The window will not change unless a valid well and date is selected from the tree view. When the grid structure is changed using the option, the selected well will be reset and you must click on a well and/or date again to change the 3D display. If at any time it is desired to display the entire grid, activate the **Selecting IJK Slabs, Quick Slabs or Region Slabs** option and then click **Reset**. If the **Remove other wells** box is selected, then all other well perforations and trajectories will not be displayed. To display all wells again, enter the **Properties** form, select **Wells** in the tree view, and then select **Show All Wells**. If trajectories are displayed, select **Well Trajectories** from the tree view and then select **Show All Wells**.

When the **Use Center Grid** option is selected, you must define the center grid in the **Center Grid Definition** frame. Each time a new center grid is selected, the display will change to be a 3D sector around the center grid block. The number of surrounding grids is controlled by the values in the text boxes in the **Surrounding Grids** frame.

Interactively edit grid in 3D:

When **Edit Grid in 3D** is selected, the display will change to 3D with grid lines and null blocks shown, and block fill turned on. You can click and select grid nodes that are located at the grid block corners. When a grid node is selected, it will become a red sphere, and the probe panel will display information about this node. If you want to view all grid nodes, select **Show All Nodes**; however, for large models, the drawing of all grid nodes may slow Builder substantially. To change the size of the grid nodes, change the width of the grid lines in the **Properties** panel under the **Grid**.

You can select more than one grid node by holding down the CTRL key and clicking the left mouse button. A grid node may be unselected by holding down the CTRL key and clicking the selected node again. You may toggle between selected and unselected by clicking the node multiple times. Once all desired grid nodes are selected, you can hold down the left mouse button and drag the grid nodes to the new locations. The probe panel will show information about how far the nodes have been moved in all three directions.

The modifications to the grid structure are applied to the fundamental grid only. Refined block structures will be inherited from the fundamental grid. If the refined block structures are defined explicitly, then these definitions will be lost using this option, and the refined structures will change to become inherited from the fundamental grid.

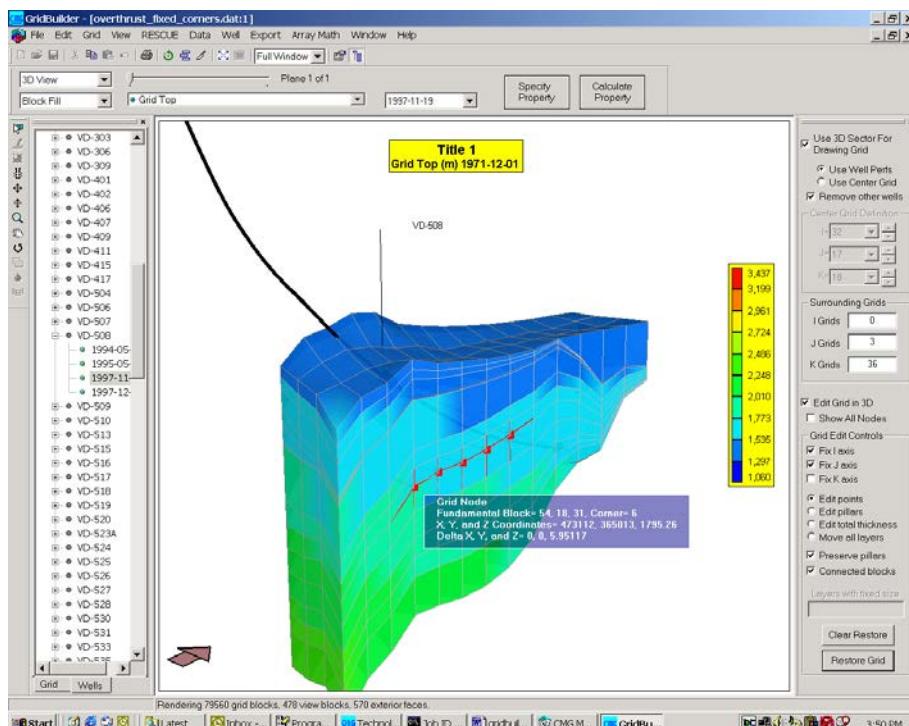
As there are some safeguards built into this option, you can only move the block or point so those blocks do not become overlapped. However, it is still possible for you to create grid blocks that are overlapped, inside out, or generally poorly constructed. When this occurs, you will be informed about the poorly constructed blocks (a **Block is non convex** message), and it is recommended that these blocks be fixed before running the simulator. Poorly constructed blocks may slow down the simulator, or even stop the simulator if the block is extremely bad.

Fix I, J, or K Axis

To edit a 3D picture on a 2D flat surface, one of the axes must be fixed. To change the axis that is fixed, check or uncheck these check boxes. If none of these check boxes are selected, then you will not be able to edit any corner points or pillars. For grids in COORD format, only Z or K coordinates may be edited. For complicated grids, allow only one axis to move to reduce the number of variables changing at any one time.

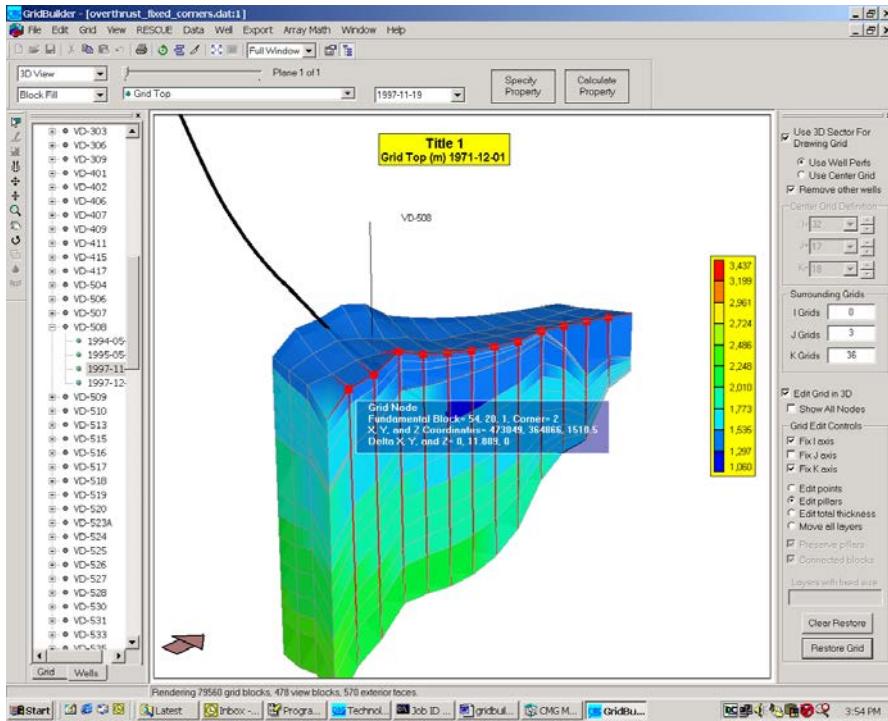
Edit Points

Selection of this option will allow editing of points for each grid block. If the grid system is Cartesian, then the entire grid block will be moved when an edit point is clicked upon and dragged. If the grid system is corner point, then only the one grid corner location will be changed. If the **Connected Blocks** option is selected, then the neighboring block point will also be changed unless the neighboring block is already disconnected (the block corner locations are different by more than 0.1 m/ft). Below is an example picture of five grid nodes that were selected and the information that is displayed on the probe panel after these nodes were moved.



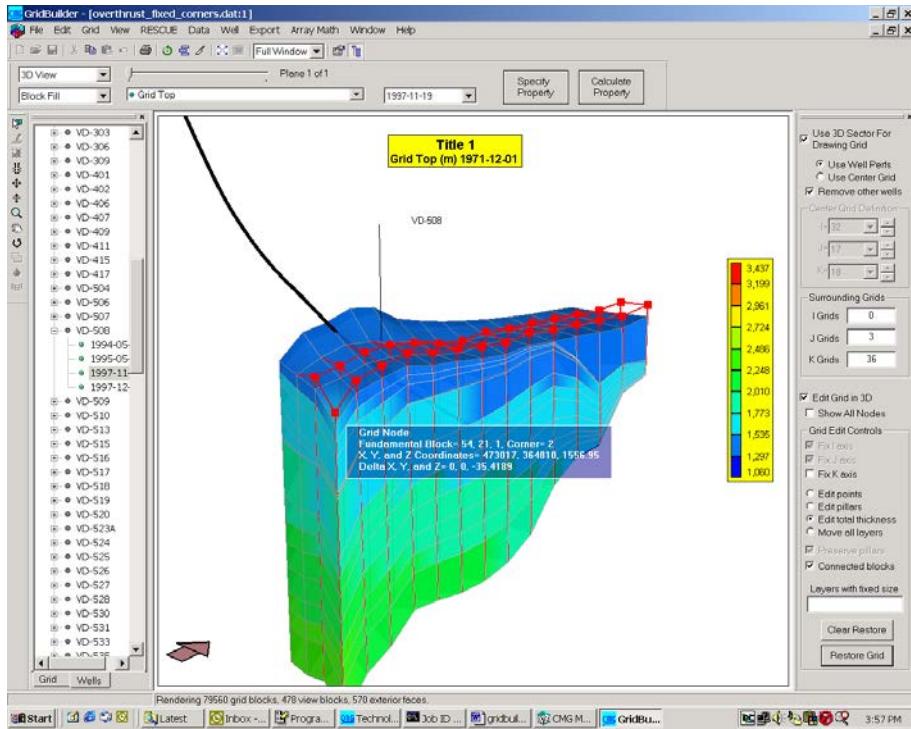
Edit Pillars

This option is only enabled when the grid is defined using either the COORD or CORNERS keywords. To edit the pillars click on this option, and select the pillars by clicking either the top layer 1 grid node, or the bottom layer grid node, as shown in the picture below. If any grid nodes do not lie exactly on the pillar that is being moved, this node will be moved to lie on the pillar, as is the case when the option is checked.



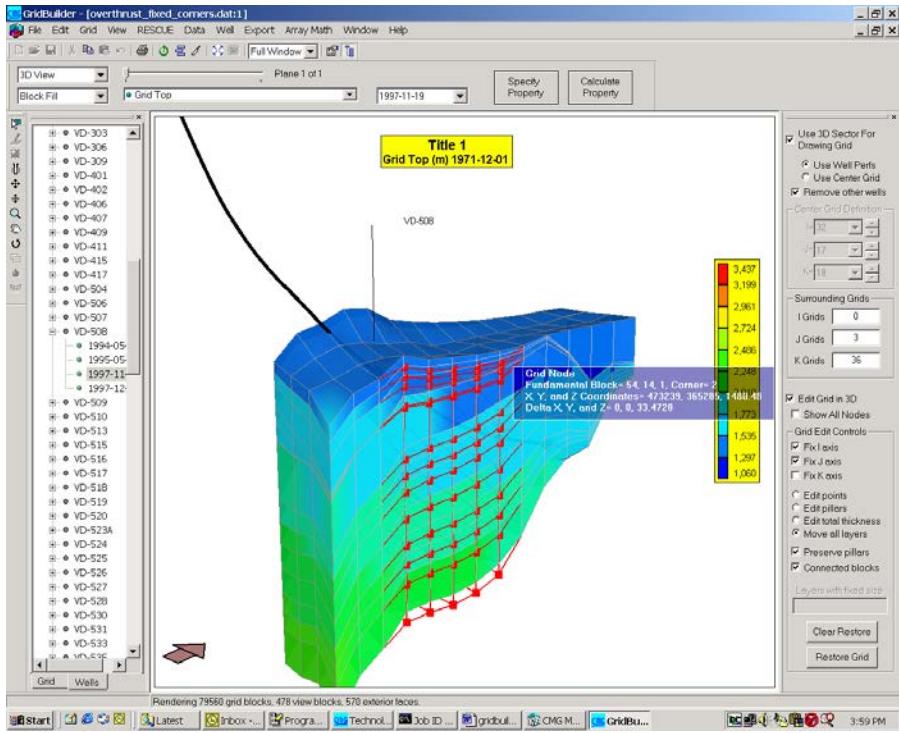
Edit Total Thickness

If this option is selected, then all blocks in the column will have their thickness changed simultaneously so that the relative thickness between layers is preserved. If the top grid node is moved, then the bottom grid node of the bottom layer will be fixed. If the bottom grid node is moved, then the top grid node of the top layer will be fixed. You can also fix some layers so that their thickness will not change (see the **Layers with Fixed Size** below). Below is an example picture of multiple top grid nodes selected and the total thickness increased.



Move All Layers

If this option button is clicked, then all blocks in the column will be moved simultaneously. This option is useful for moving all layers together, while preserving the original layer thicknesses. The following shows an example of multiple grid nodes moved down:



Preserve Pillars

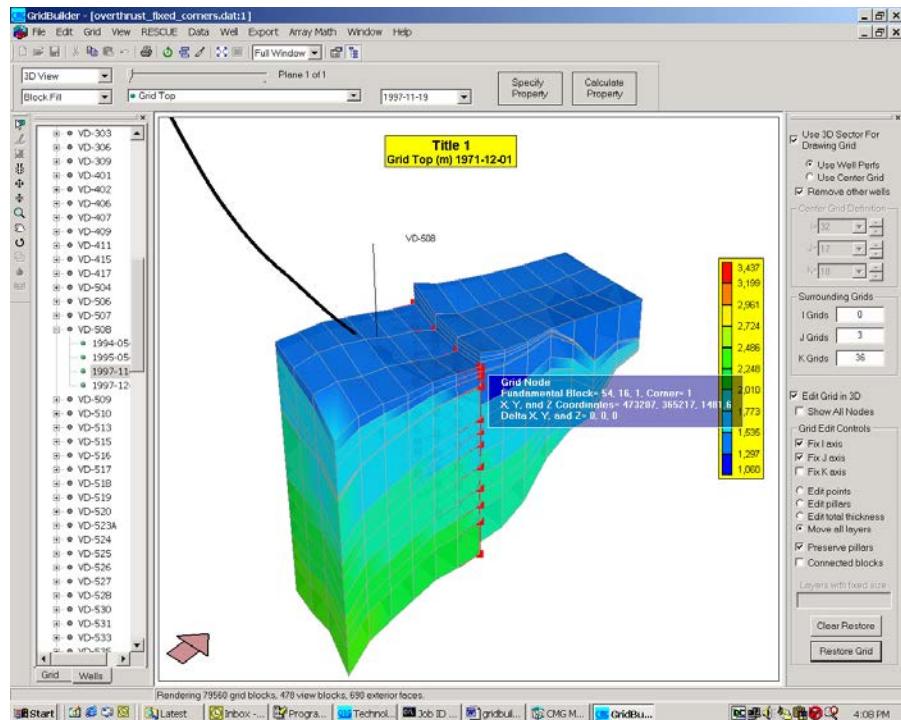
This option is only enabled when the grid is constructed using the CORNERS keyword. If this check box is selected, then grid block corners can be moved only in the directions set by the pillars. This option is useful for fault modeling. Note that if this option is selected, and the **Connected Blocks** option is cancelled, you can create faults where no fault existed previously. You can also modify the throw of the faults in this manner. If the **Preserve pillars** option is selected, then the **Fix K axis** option must be cancelled to permit editing.

Connected Blocks

If this check box is selected, then all of the blocks that share the corner will be moved even if some of these blocks are not currently displayed, unless the neighboring block is already disconnected (the block corner locations are different by more than 0.1 m/ft). If the blocks are disconnected, the blocks can be reconnected by dragging the grid node so that it lines up with the neighboring grid node. When reconnection occurs, red grid lines will appear in the neighboring block indicating that it has occurred.

If this check box is cancelled, then none of the neighboring blocks will be modified. The latter option may be useful to create faults where none existed before. It may also be useful in a Cartesian grid when there is space between layers, and you wish to change the thickness of one block without changing the thickness of the connecting block.

When this option is cancelled, you should ensure blocks do not overlap other blocks, as the safeguards to prevent this will be disabled. In any case, if you create grid blocks that are poorly constructed (non-convex blocks), warning messages will be issued. Below is an example of a fault that was created with this method.



Layers with Fixed Size

If this option is selected, then the text box becomes enabled. You can enter fundamental layer numbers (separated by spaces) in this text box that will become fixed in size. Thus, this option is useful if you wish to change the total reservoir thickness, but does not want to change one or more layers.

Clear Restore

Once the grid has been changed to your satisfaction, this button should be clicked to clear the restore array. When **Restore Grid** is selected, the grid nodes will be restored to the state of the previous **Clear Restore**, or back to the state when the grid was loaded into Builder.

Restore Grid

If the changes to the grid structure are to be discarded, then you should click this button to restore the grid that was present before any changes to the grid structure were made. This option will restore the grid that existed the last time the **Clear Restore** button was pressed.

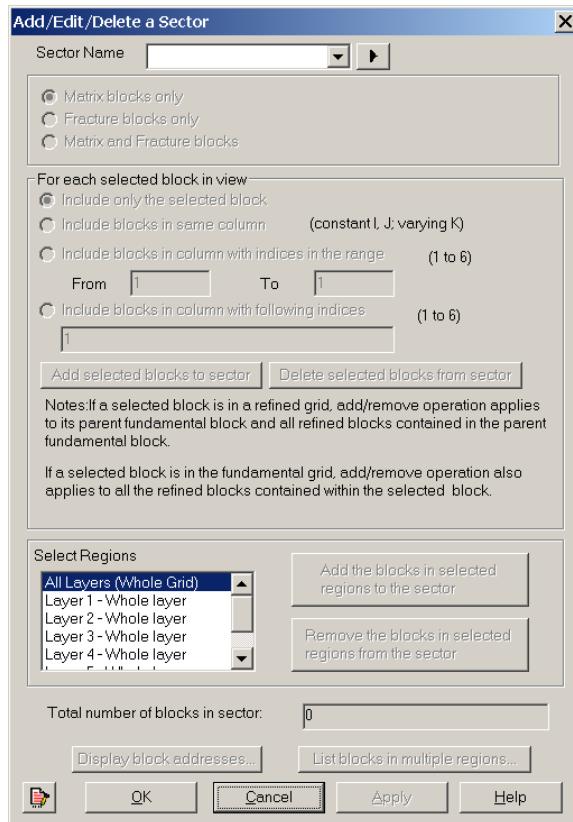
Defining and Editing Sectors

Sectors are arbitrary groupings of grid blocks that are used by Builder, the simulators, and Results 3D to calculate display statistics on different regions of the field. You can create sectors and assign grid blocks to sectors using Builder.

In general, a grid block can be in more than one sector. To facilitate conversion of datasets from other simulators, we have added a new keyword *ISECTOR where each grid block is assigned to a single sector number. If you are using *ISECTOR, and you wish to add new overlapping sectors, Builder will convert the information in the *ISECTOR property to the more general sector definition.

To add or edit (modify) a sector:

1. You must be in **Probe** mode to perform this operation. To enter **Probe** mode, select **Probe Mode**  from the mode selection toolbar, or right-click to display the context menu and the select **Probe Mode**.
2. Select **Create/Edit Sectors** from the **Reservoir** menu. The **Add/Edit/Delete a Sector** dialog box is displayed:



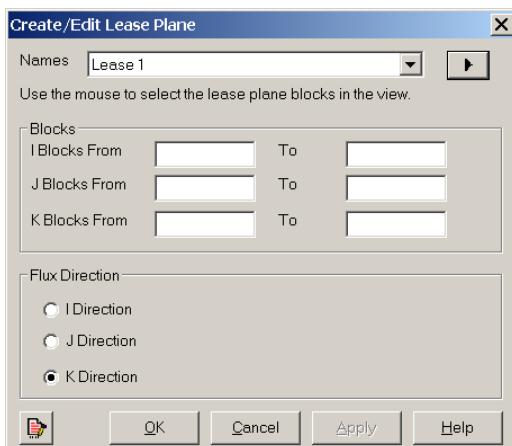
- To add a new sector, click the right arrow  button beside the **Sector Name** combo box and select **New Sector**. To edit an existing sector, select the sector from the list of existing sectors in the **Sector Name** combo box.
- If you are creating a new sector, enter a sector name in the **Input Name** dialog box:



- Select grid blocks by clicking on them, and using CTRL+click and drag to select a number of grid blocks at once. Then click on either **Add selected blocks to sector** or **Delete selected blocks from sector**. The blocks currently in the sector will be outlined.
- You can add or remove whole layers, rock type regions or PVT regions to a sector by selecting the region from the **Select Regions** list, and clicking the **Add the blocks in selected regions to the sector** or **Remove the blocks in the selected regions from the sector**.
- When you are finished adding or removing blocks, click **Apply** to confirm your changes.
- When you are finished adding or editing all the sectors of interest, click **OK** to close this dialog box. Click **Cancel** to cancel the changes to the last sector that you worked on.

Create/Edit/Delete Lease Planes

The **Lease Planes** dialog box can be accessed from **Create/Edit Lease Plane of Reservoir** menu or the **Create/Edit Lease Planes** icon in the **Modes** tool bar, or **Edit Lease Plane** menu item in the popup menu when you right click in the reservoir view or double-click any lease plane object in the tree view.



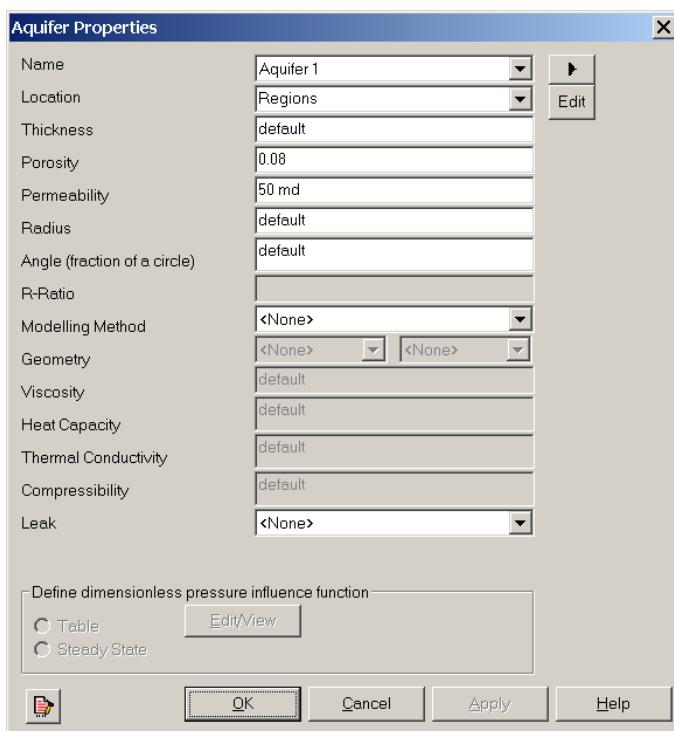
Initially, the first lease plane in the list box is selected and the lease plane blocks are highlighted in the reservoir view with magenta color. If you want to delete the selected lease plane, click the right arrow  button beside the **Names** text box, and then select **Delete Lease Plane**.

You can manually change any value in the dialog box or go to reservoir view to select a range of grid blocks. The values in the dialog box will be updated as soon as you release the button. You can change the lease plane name subject to the limitation of 16 characters maximum. Click **OK** to keep the change or **Cancel** to cancel the change.

Creating new lease plane is similar to editing a lease plane except the values are initially blank.

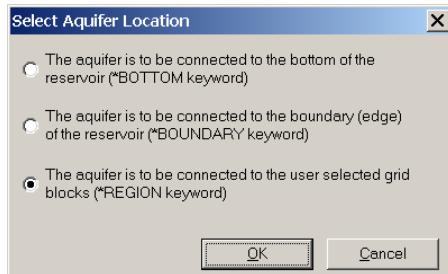
Create/Edit/Delete Aquifers

The **Aquifer Properties** dialog box can be accessed from **Create/Edit Aquifers** under the **Reservoir** menu or the **Create/Edit Aquifers** icon in the Modes tool bar, or **Create/Edit Aquifer** menu item in the popup menu when you right click in the reservoir view or double click any aquifer object in the tree view. Some dialog box items are disabled depending on the simulator you are working on. Refer to the proper simulator manual for the meaning of each value. You can select an item, or place the mouse cursor in a text field, then press the **F1** key to get help for that item.

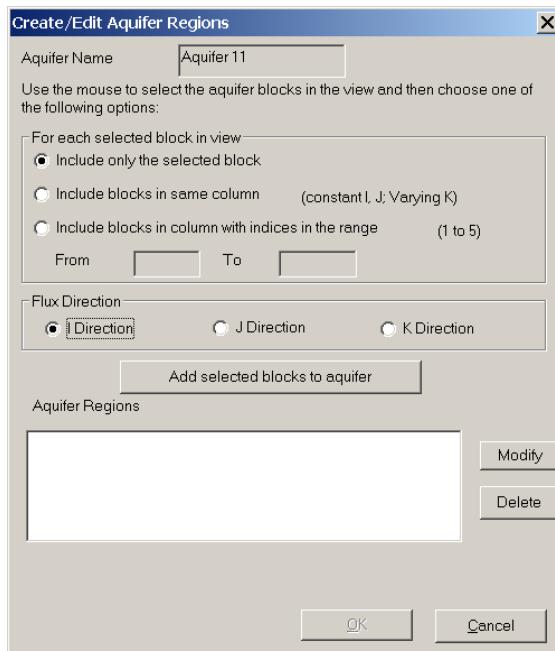


Adding an Aquifer

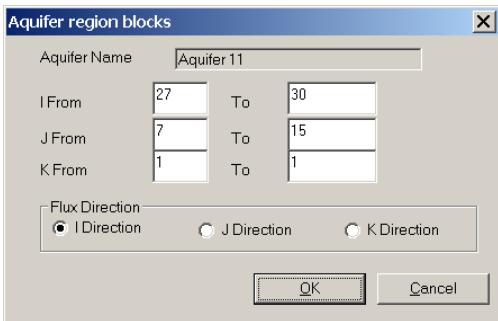
To add an aquifer, click the right arrow button next to the **Name** box and then select **New Aquifer**. The **Select Aquifer Location** dialog box is displayed.



If you select the third (*REGION) option, the **Create/Edit Aquifer Regions** dialog box (shown below) will be displayed, through which you can select the aquifer blocks; otherwise, the **Aquifer Property** dialog box will be displayed immediately.



Use the mouse to select the desired aquifer blocks in the reservoir view and then click one of the options in the **For each selected block in view** frame. Select proper flux direction. Click **Add selected blocks to aquifer**. Repeat this process to select more aquifer blocks for this aquifer. The selected aquifer blocks will be shown in the **Aquifer Regions** list. You can delete or modify any aquifer region by selecting it from the list and the click **Delete** or **Modify** button. If you click **Modify** button, the **Aquifer region blocks** dialog box displays. Through this dialog box, you can change the aquifer block range:

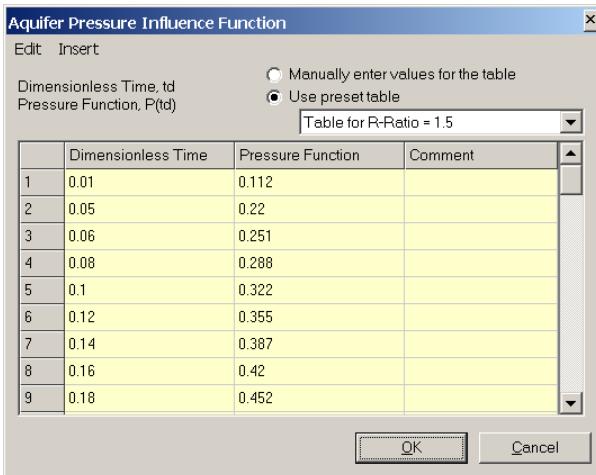


Deleting an Aquifer

Initially, when opening the **Aquifer Properties** dialog box, the first aquifer in the name list box is selected and the aquifer blocks are highlighted in the reservoir view with magenta color. To delete an aquifer, select the aquifer from the **Name** list. Then click on the right arrow button next to the name list and then select **Delete Aquifer**.

Aquifer Influence Function

Click **Edit/View** in the **Define dimensionless pressure influence function** frame or select **Carter-Tracy (limited extent)** from the **Modelling Method** combo box to display the **Aquifer Pressure influence Function** dialog box. You can select values from the preset values entered in the tables.



If you enter your own table, the values in each column must be listed in increasing order.

Locating and Adding Wells in the Simulation Grid

There are three ways to specify or determine the grid blocks in which wells are completed:

1. Give the X,Y well locations on the contour or mesh maps used for interpolation

2. Import X,Y,Z well trajectory and perforation depth information and have Builder determine which grid blocks contain perforation intervals
3. Manually specify the grid blocks for a well.

Any combination of these may be used.

A detailed description of how to locate well completions in the simulation grid is given in the [Well and Group Control](#) chapter. An overview is provided below.

Wells in Map Data

When Builder completes an interpolation, any wells located on the contour map or mesh map within the boundaries of the grid are created and their completion locations are computed. These well locations will be automatically displayed after the interpolation is complete. After the interpolation is completed, you can add additional wells or modify the computed completion locations.

If a well is located in a fundamental (coarse) grid block, Builder assumes that the well is at the center of the grid block for the well index calculation. If the well is located in a fundamental grid block that contains a hybrid-refined grid, the well is assumed to be at the center of the refined grid. If the well is located in a fundamental grid block that contains a Cartesian refined grid, the well is assumed to be at the center of the refined block containing the map well location. For information about calculating well indices, see the STARS, GEM or IMEX User's Guides.

Well location information does not need to be in all maps used for interpolation but should be in at least one map used in each simulation layer. For this reason, it is often convenient to put the well location information in the maps used for Grid Thickness calculations.

Opening a 3D Well Trajectory File and Adding Well Completions

After creating and interpolating the structural properties of a grid, you may open a well trajectory file and have Builder determine which grid blocks are intersected by perforated well trajectory segments. The chapter [Importing Geological and Well Trajectory Data](#) describes the file formats supported by Builder. After reading a 3D well trajectory file, Builder will display the well trajectories over the simulation grid that you have created, and can determine which grid blocks contain part of a well trajectory perforation. To create simulation model wells (that is, the *PERF keywords), you must specify the perforation interval along the trajectories. The [Well Trajectories](#) section of the [Well and Group Control](#) chapter gives more details on loading trajectories and using them to create well completions for the simulator data.

Manually Adding or Modifying Well Completions

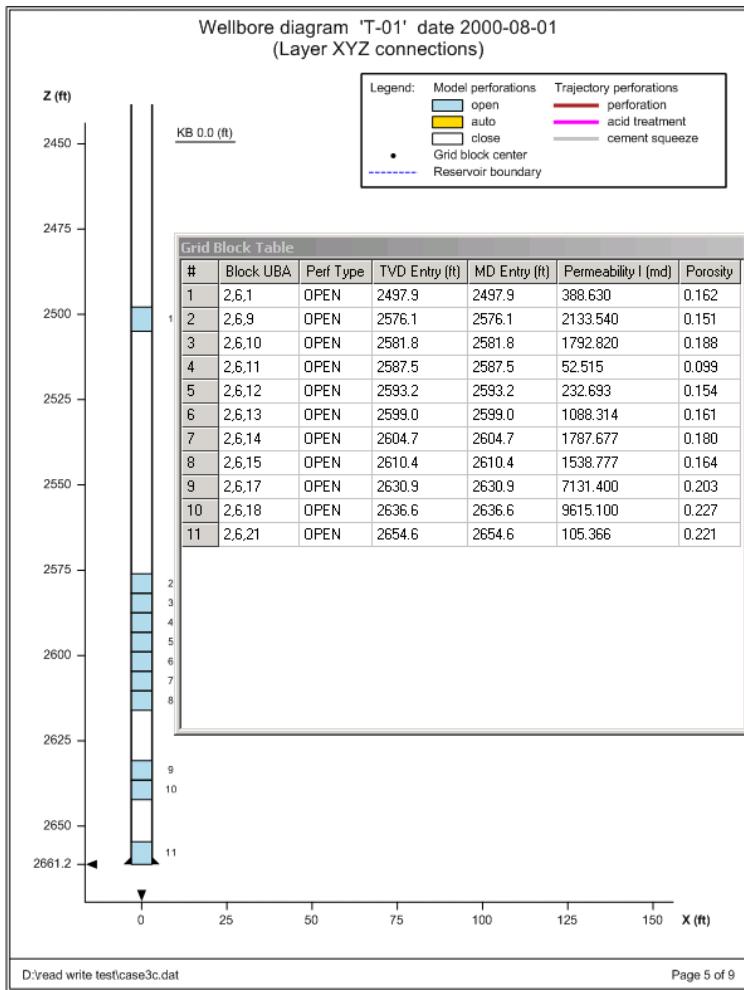
You can input specific grid blocks for a well completion. A description of this is given in the [Well Completions \(PERF\)](#) section of the [Well and Group Control](#) chapter.

Wellbore Diagram

To display a wellbore diagram for a well, click **Well & Recurrent** in the tree view, and then expand the list of wells. From the **Well** menu, select **Open Wellbore Diagram**. Click on any of the well PERF items on the tree view to see the wellbore diagram for that well and date.

There are a number of controls for the layout of the wellbore diagram. Once you are viewing a wellbore diagram, move the mouse cursor over the diagram and right-click to display a context menu. Select **Properties** to open a dialog box controlling how the wellbore diagram is drawn.

You can also view the entire wellbore diagram, or zoom in on the part of the diagram that intersects the reservoir, as shown in the following example:



Rock Compaction/Dilation Regions (IMEX and GEM Only)

For IMEX and GEM, you can edit, add, or delete rock compaction/dilation regions. In the main menu, select **Reservoir | Create/Edit Compaction/Dilation Regions**. Alternatively, the dialog box can be accessed directly from the main tree view for the reservoir section by double-clicking **Compaction/Dilation Regions**.

The **Compaction/Dilation** dialog box displays data for a specified compaction/dilation region. The dialog box also displays a drop-down list of existing regions together with a button for adding, copying and deleting regions. This dialog box allows you to build and edit a set of different regions, and to enter related information.

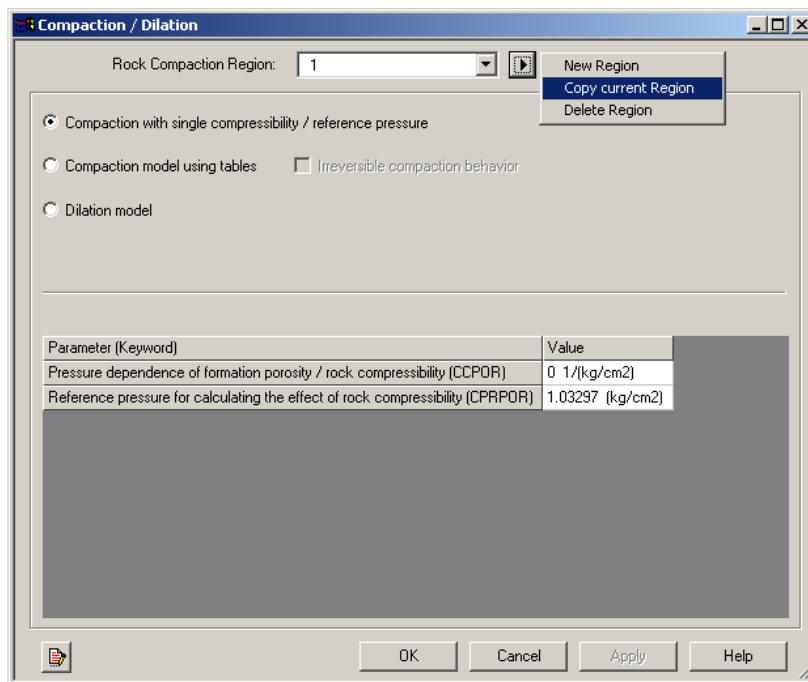
Selecting Compaction/Dilation Regions

From the drop-down list, select a particular region to edit. The information for the region will be displayed on the dialog box. The data displayed in the dialog box is updated according to the region selected from the list. When editing a particular rock type, changes are not permanently written to the data file until you select **OK** or **Apply**.

Adding a New Compaction/Dilation Region

A new region may be added to the list of available regions by clicking the right-arrow  button next to **Rock Compaction Region** and then selecting from the drop-down menu.

Data from the currently displayed region can be copied to the new region by selecting the menu option **Copy Current Region**. Alternatively, an uninitialized new region can be created by selecting the menu option **New Region**.



Removing Compaction/Dilation Regions

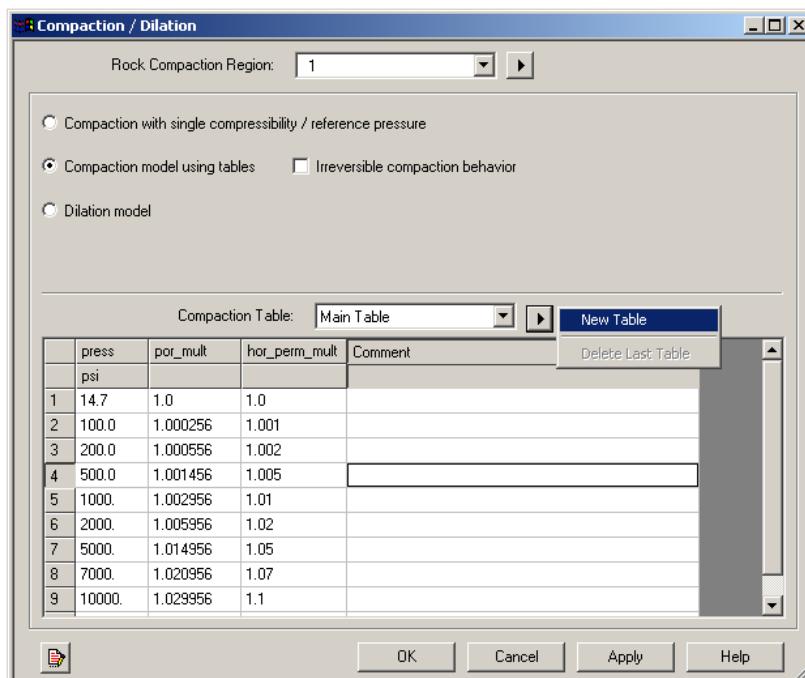
An existing region can be deleted by first selecting the desired region from the drop-down list, then clicking the right-arrow  button and selecting **Delete Region**. The region will be removed from the drop-down list.

As each region is removed, the remaining regions are renamed so that the region numbers are sequential. For example, if there are three regions (Region 1, Region 2 and Region 3) and Region 2 is deleted then Region 3 is renamed to Region 2. This leaves a list of two regions (Region 1 and Region 2).

For the above reason, deleting a region might affect other properties. When deleting regions review the properties affected by the change to make sure the values are properly entered.

Compaction/Dilation Tables

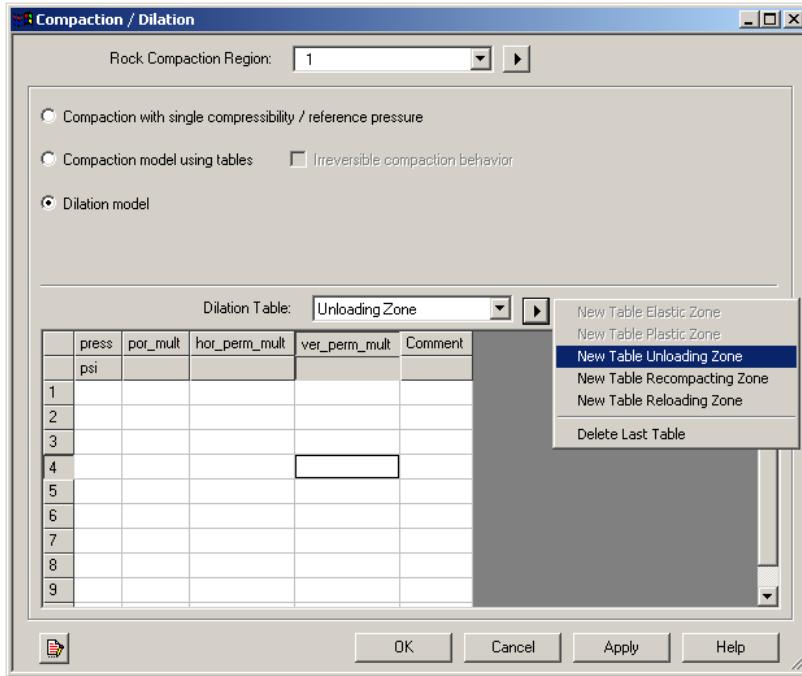
When you define your region with compaction or dilation tables, the **Compaction/Dilation** dialog box will allow you to add, edit, and delete tables accordingly. A drop-down list can be used to select which table will be displayed at any given time.



Adding a New Compaction/Dilation Table

A new table can be added to the list of available tables by clicking on the right-arrow  button beside **Compaction Table** and selecting from a drop-down menu.

For the Dilation model, only particular table definitions may follow in sequence; therefore, the dialog box will prevent you from defining the wrong table type at any step.



Removing Compaction/Dilation Tables

Only the last table in the list can be deleted. For the Compaction model, this is to preserve the integrity of specific data requirements for successive tables. For the Dilation model, this is also to preserve the permitted order of allowed tables in sequence. Refer to the appropriate section of the simulator manual to review these requirements.

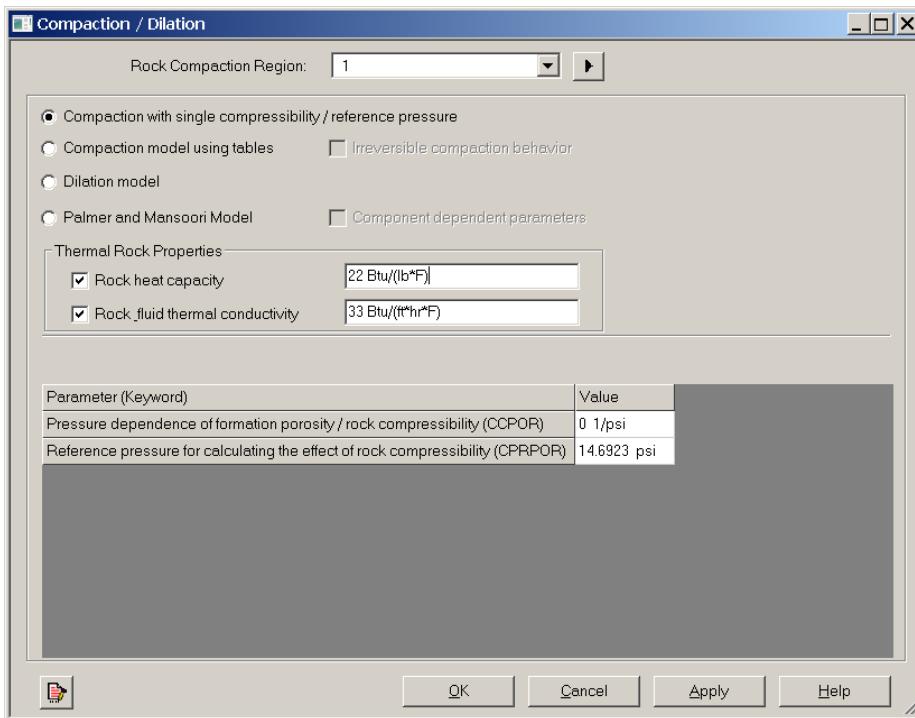
To delete the last table in the sequence, click on the right-arrow button then select **Delete Last Table** from the drop-down menu. The table will be removed.

Setting Thermal Properties for the Rock Type (GEM Only)

Through the **Compaction/Dilation** dialog box, you can also set the following thermal rock properties for GEM:

- Rock heat capacity
- Rock fluid thermal conductivity

These properties can be set for each rock compaction region that is defined.



Other Reservoir Properties (STARS Only)

STARS “Other Reservoir Properties” can be accessed from the main menu by selecting **Reservoir | Other Reservoir Properties**. Alternatively, the dialog box can be accessed directly from the main tree view for the reservoir section by double-clicking **Thermal Rock Types**.

The **Other Reservoir Properties** dialog box will hold a set of six tabs and a drop-down list of the different rock types together with a menu button for adding, copying, and deleting rock types. This dialog box allows you to build and edit a set of different rock types, and to define their related information.

Each rock type and its related information are entered through the following tabs:

- Rock Compressibility
- Thermal Properties
- Overburden Heat Loss
- Dilation Recompaction
- Compaction Rebounding
- Variable Permeability

General Editing of a Rock Type

From the drop-down list, select a given rock type to edit. The information for the given rock type will be displayed on the interfaces found on the tabs. The data displayed in the tabs is updated according to the rock type selected from the list.

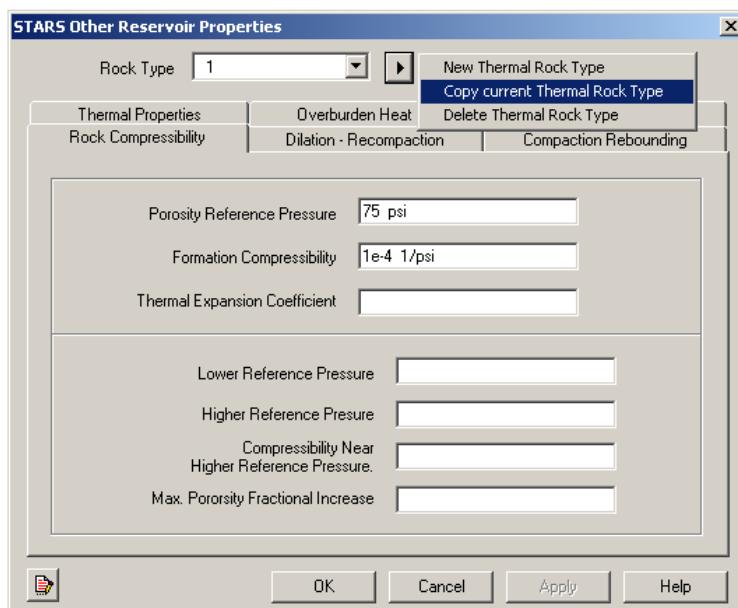
When editing a given rock type, changes made in the interface will immediately update the current working data set. However these changes are not permanently written to the data file until you select **OK** or **Apply**.

While editing a given rock type, you may encounter messages warning you that continuing may result in some data being deleted. It is important that you read these messages carefully, to understand the implications of your choice. Further details will be given below.

Adding a New Rock Type

A new rock type may be added to the list of available rock types by clicking the right-arrow  button and selecting from the drop-down menu.

Data from the currently displayed rock type can be copied to the new rock type by selecting the menu option **Copy Current Thermal Rock Type**. Alternatively, an uninitialized new rock type can be created by selecting the menu option **New Thermal Rock Type**.



Removing a Rock Type

An existing rock type can be deleted by first selecting the desired rock type from the drop down list, then clicking the right-arrow  button and selecting **Delete Thermal Rock Type** from the drop-down menu. The rock type will be removed from the drop-down list.

As each rock type is removed, the remaining rock types are renamed so that the rock type numbers are sequential. For example if there are three rock types (RockType 1, RockType 2 and RockType 3) and RockType 2 is deleted, then RockType 3 is renamed RockType 2. This leaves a list of two rock types (RockType 1 and RockType 2.)

For the above reason, deleting a region may affect other properties. When deleting regions, review properties affected by the change to make sure that the values are properly entered.

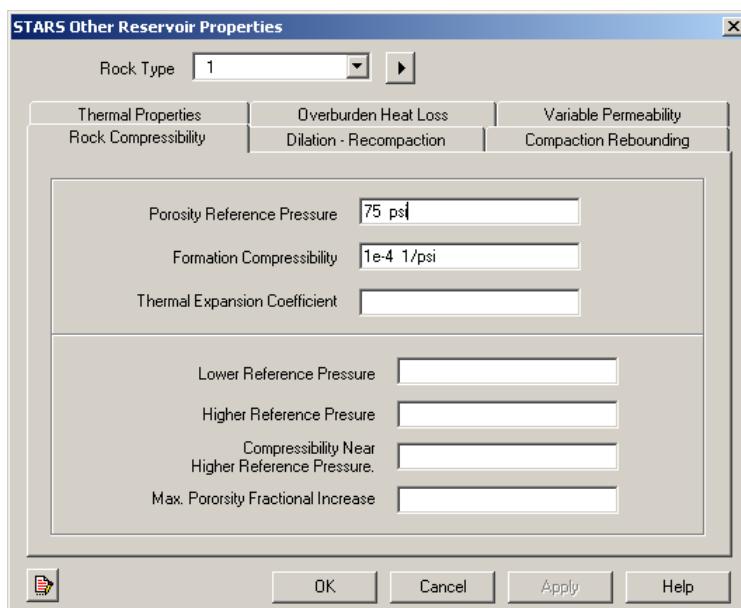
The last remaining rock type cannot be deleted, since the simulator requires a minimum of one rock type defined for the dataset.

Entering Rock Compressibility

The **Rock Compressibility** tab allows you to enter values for:

- Porosity Reference Pressure
- Formation Compressibility
- Thermal Expansion Coefficient
- Lower Reference Pressure
- Higher Reference Pressure
- Compressibility Near Higher Reference Pressure
- Maximum Porosity Fractional Increase

This section corresponds to the **Other Reservoir Properties | Rock Compressibility** section of the *STARS User's Guide*.

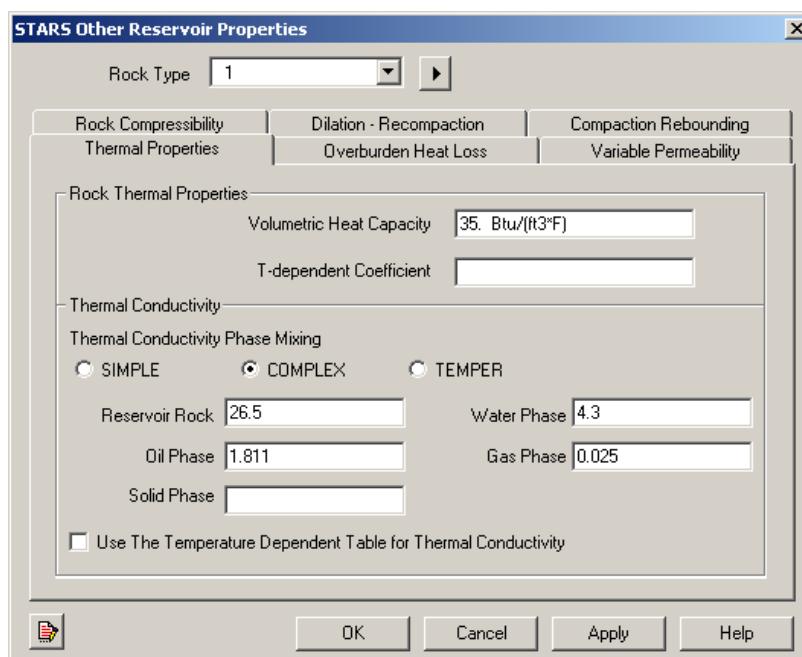


Setting Thermal Properties for the Rock Type

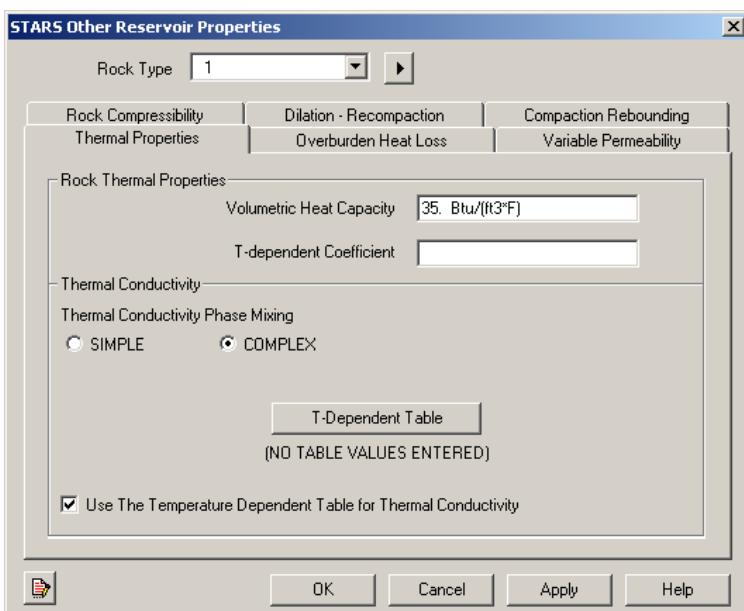
The **Thermal Properties** tab allows you to enter values for:

- Volumetric Heat Capacity
- T-dependent Coefficient of the Volumetric Heat Capacity
- Thermal Conductivity of Reservoir rock
- Thermal Conductivity of the Solid Phase
- Thermal Conductivity of the Water Phase
- Thermal Conductivity of the Oil Phase
- Thermal Conductivity of the Gas Phase
- Temperature Dependent Table for Thermal Conductivity
- Thermal Conductivity Phase Mixing – SIMPLE | COMPLEX | TEMPER

This section corresponds to the **Other Reservoir Properties | Rock Thermal Properties** section of the *STARS User's Guide*.



In this tab you can enter the thermal conductivity data either using the five fields shown above (that is, Reservoir Rock, Water Phase, Oil Phase, Gas Phase and Solid Phase) or a temperature-dependent table. For the temperature-dependent table only the SIMPLE or COMPLEX thermal conductivity phase mixing can be used. To select the temperature-dependent table for thermal conductivity the check box must be selected. Any thermal conductivity data entered in the five fields above will be ignored if you choose to use the temperature-dependent table for thermal conductivity. On selecting the check box, the five fields for entering thermal conductivity data will be replaced by a button that will permit table data to be entered. The modified tab interface to enter the temperature-dependent table for thermal conductivity is shown below.



Canceling the check box will display a warning that any data entered in the table will be deleted. If you choose to continue, the previous interface will reappear.

To access the actual form where the table values are entered, you must click the **T-Dependent Table** button. Once the button is pressed, a window is displayed (see below) for entering table data. Note that the column for entering thermal conductivity of the solid phase is optional.

Table

| | Temp | Rock | Water | Oil | Gas | Solid | Con |
|---|--------|--------------------------------|---------------------------------|---------------------------------|---------------------------------|--------------------------------|-----|
| 1 | 50 F | 1 Btu/(ft ² hr°F) | 0.36 Btu/(ft ² hr°F) | 0.07 Btu/(ft ² hr°F) | 0.08 Btu/(ft ² hr°F) | 2 Btu/(ft ² hr°F) | |
| 2 | 550 F | 1.2 Btu/(ft ² hr°F) | 0.4 Btu/(ft ² hr°F) | 0.08 Btu/(ft ² hr°F) | 0.09 Btu/(ft ² hr°F) | 2.1 Btu/(ft ² hr°F) | |
| 3 | 1050 F | 1.4 Btu/(ft ² hr°F) | 0.5 Btu/(ft ² hr°F) | 0.09 Btu/(ft ² hr°F) | 0.1 Btu/(ft ² hr°F) | 2.2 Btu/(ft ² hr°F) | |
| 4 | 1550 F | 1.6 Btu/(ft ² hr°F) | 0.6 Btu/(ft ² hr°F) | 0.1 Btu/(ft ² hr°F) | 0.12 Btu/(ft ² hr°F) | 2.4 Btu/(ft ² hr°F) | |

Entering Overburden Heat Loss Parameters

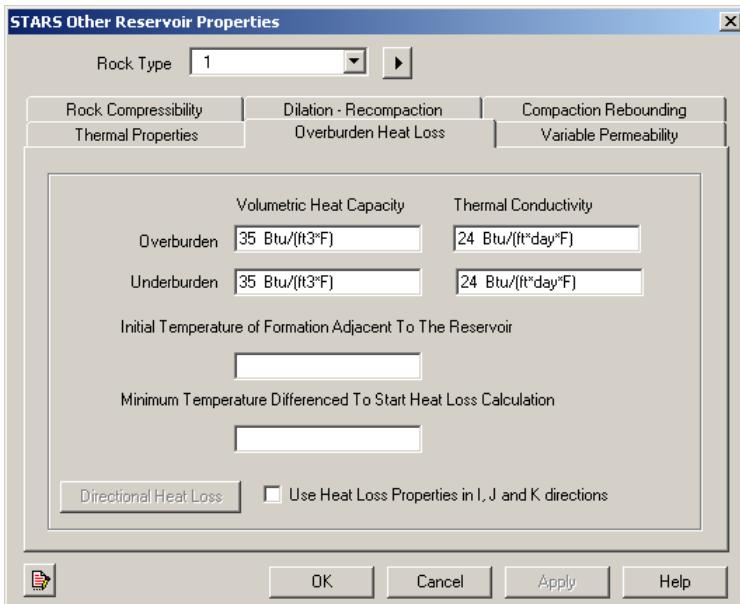
The **Overburden Heat Loss** tab allows you to enter the values for the semi-analytical infinite-overburden heat loss model. The values included are:

- Overburden heat loss properties to the outer grid block faces at the top of the reservoir.
- Underburden heat loss properties to the outer grid block faces at the bottom of the reservoir.
- Heat loss to the outer grid block faces in the indicated direction.
- Initial temperature of formation adjacent to the reservoir, used by the heat loss calculation.
- Minimum temperature difference needed between block temperature and thf to start heat loss calculation.

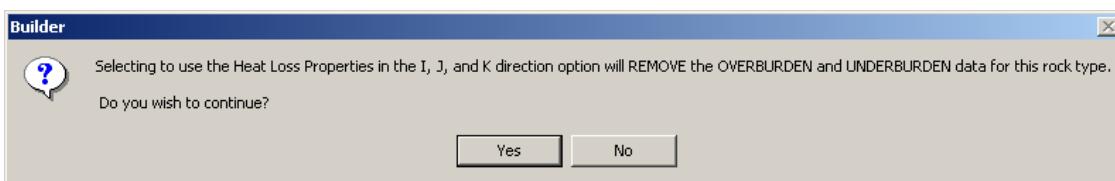
Note: The heat loss properties include the values for:

- Volumetric heat capacity of formation, adjacent to the reservoir in the indicated direction and
- Thermal conductivity of formation, adjacent to the reservoir in the indicated direction.

This section corresponds to the **Other Reservoir Properties | Overburden Heat Loss** section of the *STARS User's Guide*.



In this interface you can enter Volumetric Heat Capacity and Thermal Conductivity by using either overburden and underburden or heat loss properties in the I, J and K directions but not both. Due to this requirement you choose the variables you wish to use by selecting or canceling the **Use Heat Loss Properties in I, J and K directions** check box. If the check box is selected then the **Directional Heat Loss** button will be enabled. Clicking this button will display the interface for entering the values for the heat loss properties in the I, J and K directions. A warning message will also appear, notifying you that any values entered for overburden and underburden will be deleted. When canceling a check box, a similar message is displayed, warning you that the data for the heat loss properties in the I, J and K directions will be deleted.



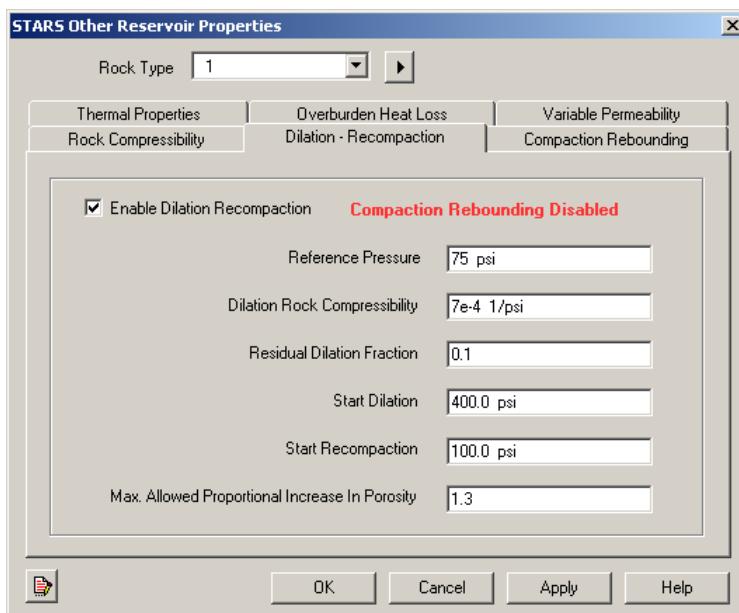
Specifying Dilation Recompaction

The **Dilation Recompaction** tab allows you to enter the values for:

- Reference Pressure
- Dilation Rock Compressibility
- Residual Dilation Fraction, that is, the fraction of total dilation not recovered on recompaction.
- Start Dilation – pressure at which dilation begins.

- Start Recompaction – pressure at which recompaction begins.
- Maximum allowed proportional increase in porosity, applied individually to each block's base porosity.

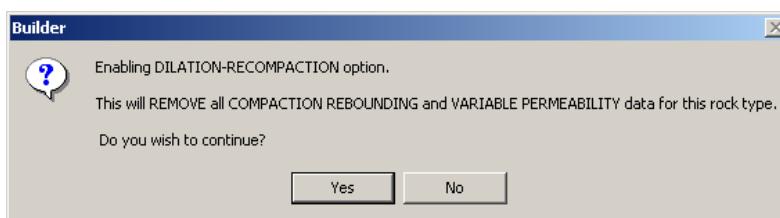
This section corresponds to the **Other Reservoir Properties | Reservoir Dilation-Recompaction** section of the *STARS User's Guide*.



If you select the **Enable Dilation Recompaction** check box, data entered in the **Compaction Rebounding** tab or the **Variable Permeability** tab will be removed. This is because each rock type can only have data for one of the three tabs:

- Dilation Recompaction,
- Compaction Rebounding or
- Variable Permeability.

Further details can be found in the *STARS User's Guide*. Before data is deleted, a message is displayed to warn you of possible consequences. The warning message allows you to continue (Yes) or cancel (No). A similar message is displayed when the check box is cancelled, warning you that any data entered on the tab will be deleted.

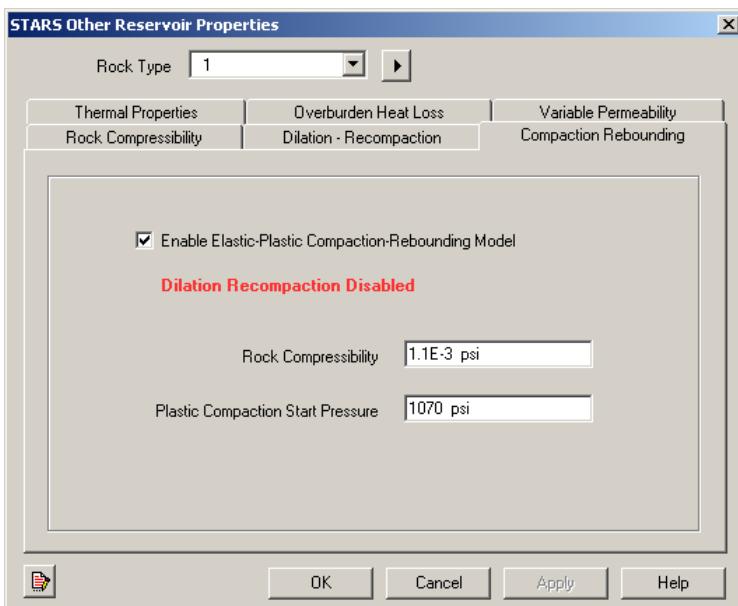


Specifying Compaction Rebounding

The **Compaction Rebounding** tab allows you to enable the compaction-rebounding model and enter values for:

- Rock compressibility for plastic compaction
- Threshold pressure at which plastic compaction begins

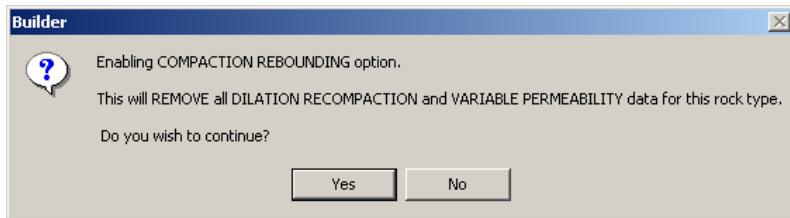
This section corresponds to the **Other Reservoir Properties | Reservoir Compaction Rebounding** section of the *STARS User's Guide*.



If you select the **Enable Elastic-Plastic Compaction-Rebounding Model** check box, data entered in the **Dilation Recompaction** tab or the **Variable Permeability** tab will be removed. This is because each rock type can only have data for only one of the three tabs:

- Dilation Recompaction,
- Compaction Rebounding or
- Variable Permeability.

Further details can be found in the *STARS User's Guide*. Before any data is deleted, a message is displayed to warn you of the possible consequences. The warning message allows you to continue (Yes) or cancel (No). A similar message is displayed when the check box is cancelled, warning you that any data entered into this tab will be deleted.

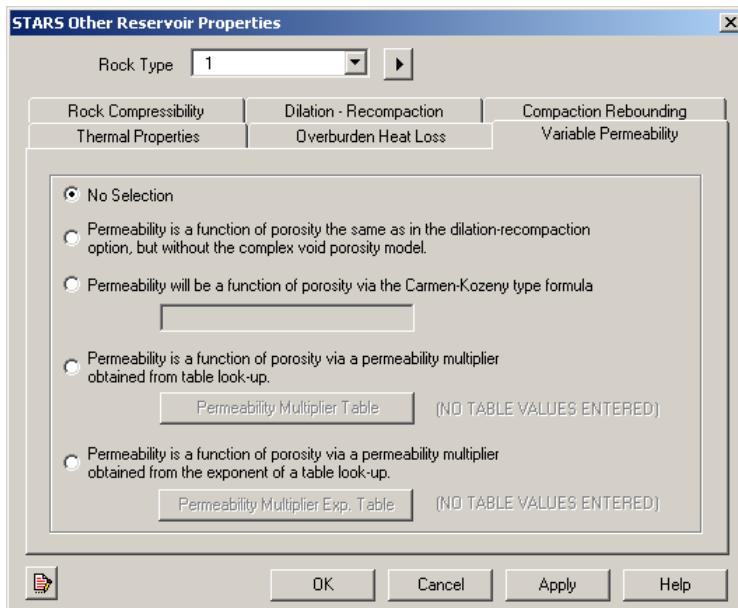


Specifying Variable Permeability

The **Variable Permeability** tab allows you to specify the dependence of permeability on porosity. The options are:

- Permeability is a function of porosity the same as in the dilation-recompaction option, but without the complex void porosity model.
- Permeability is a function of porosity via the Carmen-Kozeny type formula.
- Permeability is a function of porosity via a permeability multiplier obtained from table look-up.
- Permeability is a function of porosity via a permeability multiplier obtained from the exponent of a table look-up.

This section corresponds to the **Other Reservoir Properties | Variable Permeability** section of the *STARS User's Guide*.



By selecting one of the options from the **Variable Permeability** tab, any data entered in the **Dilation Recompaction** and **Compaction Rebounding** tabs will be removed. This is because each rock type can only have data for only one of the three tabs:

- Dilation Recompaction,
- Compaction Rebounding or
- Variable Permeability.

Further details can be found in the *STARS User's Guide*. Before any data is deleted, a message is displayed to warn you of the possible consequences. The warning message allows you to continue (Yes) or cancel (No).

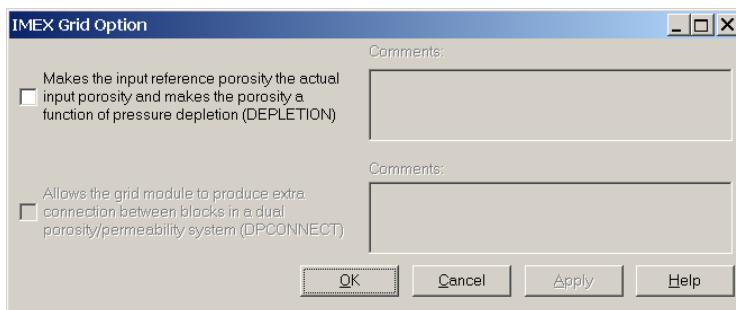


By selecting one of the last two options a button will be enabled which when clicked will bring up the interface to enter tabular data for permeability as a function of porosity via a permeability multiplier obtained from table look-up or from the exponent of a table look-up.

Reservoir Options (IMEX and GEM Only)

IMEX Reservoir Options can be accessed from the main tree view for the reservoir section by double-clicking **Options**.

The **IMEX Grid Option** dialog box allows you to turn on or turn off the reference porosity option (IMEX and GEM only) and extra block connection option for a dual porosity and/or permeability system (IMEX only). The **IMEX Grid Option** dialog box is shown below:

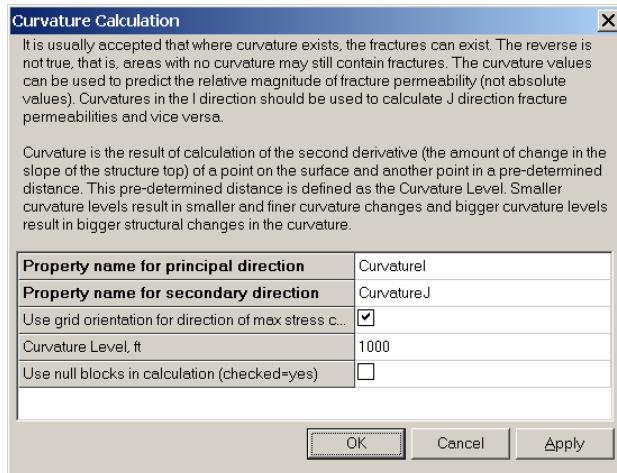


Curvature Analysis

To achieve fracture characterization for a reservoir you will need to consider various parameters, one of which is the kinematic history of the reservoir. While we may not know the exact sequence of events that a reservoir might have gone through, it is possible to evaluate the present day structure in detail and use this information to estimate fracture

presence. It is usually accepted that where curvature exists, fractures can exist. The reverse is not true; that is, areas with no curvature may still contain fractures.

This curvature analysis feature calculates and displays the curvature property in each I and J direction for each layer using the structure top property for each layer. In Builder, the curvature calculation is valid for corner point grids only. It is available through **Tools | Fracture Analysis | Curvature Calculation**:



Curvature is a unitless parameter which has relative values. It is the result of calculation of the second derivative (the amount of change in the slope of the surface structure values) of a point (centroid of the block top) on the surface and another point at a pre-determined distance. This pre-determined distance is defined as the curvature level. For example, a curvature level of 2000 ft means that the curvature calculations are carried out between a point on the surface and other points which are 2000 ft from that point. We can conclude that smaller curvature levels would result in smaller and finer curvature changes and larger curvature levels would result in larger structural changes in the curvature map displays.

If you enter a curvature level that is less than a grid block length, then the grid block length is used as the curvature level and the curvature calculation will be done from block to block neighbor. If you enter a curvature level that is larger than the entire span of the grid, then the curvature level is set to be the span of the entire grid. Negative curvature values are generally thought to be compressive, and therefore do not contribute to increased fracture permeability. Therefore, only positive curvature values should be used for the calculation of fracture properties.

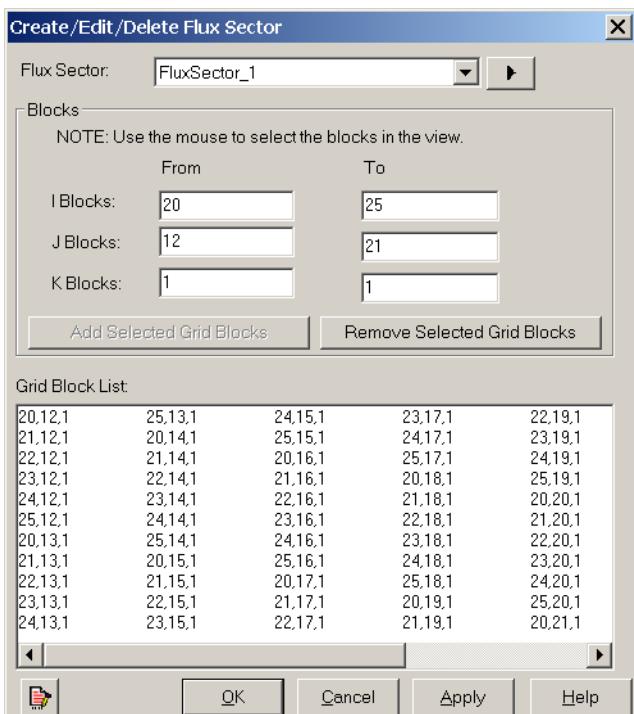
The directions of maximum and minimum stress are calculated from the grid orientation if **Use grid orientation for direction of max stress** is selected. If this option is unchecked, then you can enter the direction of the maximum stress, and this direction will be used for the entire grid. However, the use of this option is not recommended as normally the grid should be constructed in a way so that it is oriented along the direction of maximum stress so that permeability anisotropy can be modeled correctly.

The usage of the curvature calculation has one clear advantage over methods that use properties measured in a well in that the fracture properties can be estimated in areas where no well control exists. The only input required is the structure top. The curvature calculations in the two directions assure that the fracture permeability anisotropy can be accounted for. Even though the absolute scale of the fracture permeability cannot be determined with this method, the permeability modifier distribution in two directions can be evaluated. Therefore, a multiplication factor should be applied to the curvature property in both directions to directly calculate fracture permeability that can be used in the simulator. This multiplication factor should be considered a history matching variable. Other types of data such as well tests, FMI logs, production logs, and/or rock fracturability can be used in combination with curvature to determine fracture properties.

Flux Sector (IMEX Only)

The **IMEX Flux Sector** dialog box can be accessed from the main tree view for the **Reservoir** section by double-clicking on the **Flux Sectors** node or any of its child nodes, or through **Reservoir | Create/Edit Flux Sectors**.

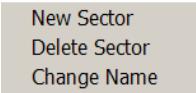
The **IMEX Flux Sector** dialog box allows you to create, edit and delete flux sectors:



To add or remove grid blocks from the current flux sector:

- Use the mouse to select a grid block region from the current view. Note that you can only perform simple selection, that is , only a single rectangular region can be selected at a time. The SHIFT and CTRL keys cannot be used to perform the selection. You can also specify the block region by entering the **From** and **To** IJK-indexes. This is very useful if you want more than one layer of the current plane selected, for example, the depth direction for the current 2D view.
- Click **Add Selected Grid Blocks** or **Remove Selected Grid Blocks**, as necessary.

The dialog box has tool menus for creating a new flux sector, deleting the current flux sector, and renaming the current flux sector. The tool menus are accessed by clicking the right-arrow  button. The menus are:



A flux sector can also be deleted from the tree view by right-clicking the corresponding flux sector node and selecting **Delete Sector**.

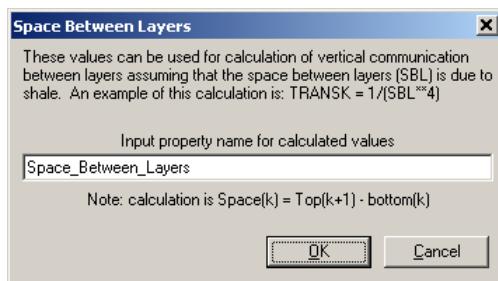
Miscellaneous Grid Calculations

Builder supports the calculation of:

- **Space Between Layers (SBL):** The space between adjacent layers is calculated as the top of layer $(k+1)$ less the bottom of layer k . These values can be used for the calculation of the vertical communication between layers.
- **Connected Net Pay:** A measure of the communication between reservoirs, layers or geological units. A cell is considered connected vertically when it is active and connected above or below with another active cell. The success of some processes depends heavily on vertical connectivity, and a larger connected net pay will usually mean higher oil recovery and/or a higher NPV.

To carry out these calculations:

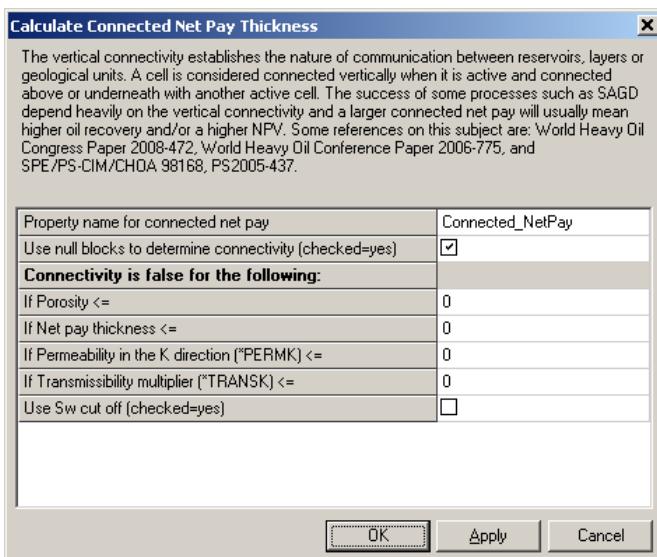
1. Select **Well | Miscellaneous Grid Calculations**.
2. If you select **Calculate Space Between Layers**, the **Space Between Layers** dialog box is displayed, with a default name entered (*Space_Between_Layers*):



Edit the property name as necessary and then click **OK**. The **Space Between Layers** property is calculated then displayed in the main view. It is available for use in other property calculations (refer to [Formula Manager](#) for further information):



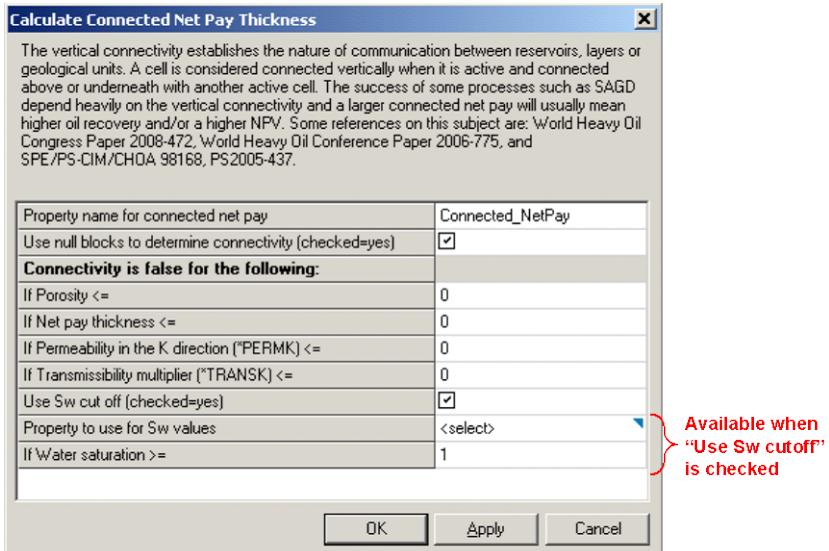
3. If you select **Calculate Connected Net Pay**, the **Calculate Connected Net Pay Thickness** dialog box is displayed:



Edit the cells in the dialog box as appropriate:

- **Property name for connected net pay:** Edit the default name (*Connected_NetPay*), if necessary.
- **Use null blocks to determine connectivity (checked=yes):** If checked, then blocks separated by a null block will be considered not connected. If it is not checked, connection will be determined based only on the settings of the fields under **Connectivity is false for the following:**
- **Connectivity is false for the following:** If any of the “if” statements are true, or if **Use Sw cut off (checked=yes)** is checked and the **Sw cut off** is exceeded, then connectivity will be false; that is, the blocks will be considered not connected.

If you check **Use Sw cutoff (checked=yes)**, additional conditions are opened, as shown below:



Once you have completed the configuration, click **OK**. The **Connected_NetPay** property is calculated then displayed in the main view. It is available for use in other property calculations (refer to [Formula Manager](#) for further information):



Fluid Model - IMEX

Overview

Using Builder you can:

- Set or change the fluid component model
- Create a “quick” BLACKOIL or GASWATER model using correlations
- Import a fluid component model created using CMGL’s PVT/phase behavior application WinProp
- Launch WinProp with the current component data in Builder
- Create and edit multiple PVT regions
- Create and edit PVT tables
- Set and edit PVT Region properties such as fluid densities and water properties
- Generate water properties using correlations
- Create and edit tables of undersaturated data (VOT, COT, BOT tables)
- View and edit comments for individual data items

Viewing and Editing Data

Apart from the usual dialog boxes, fluid model data can also be viewed and edited interactively in the plot view. Refer to [Viewing and Editing Tabular Data](#) in [Changing Display Content and Settings](#).

You can enter property values in any valid units. A list of the valid unit strings is available in [Appendix A: Units and Unit Strings](#) later in the manual. Builder will convert the value to the current units and display the converted value and unit strings.

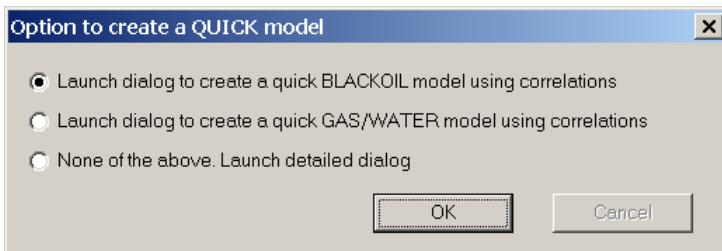
Note: There is no automatic conversion of tables. Tables have to be entered in the units.

Creating a New Fluid Model

To create a model,

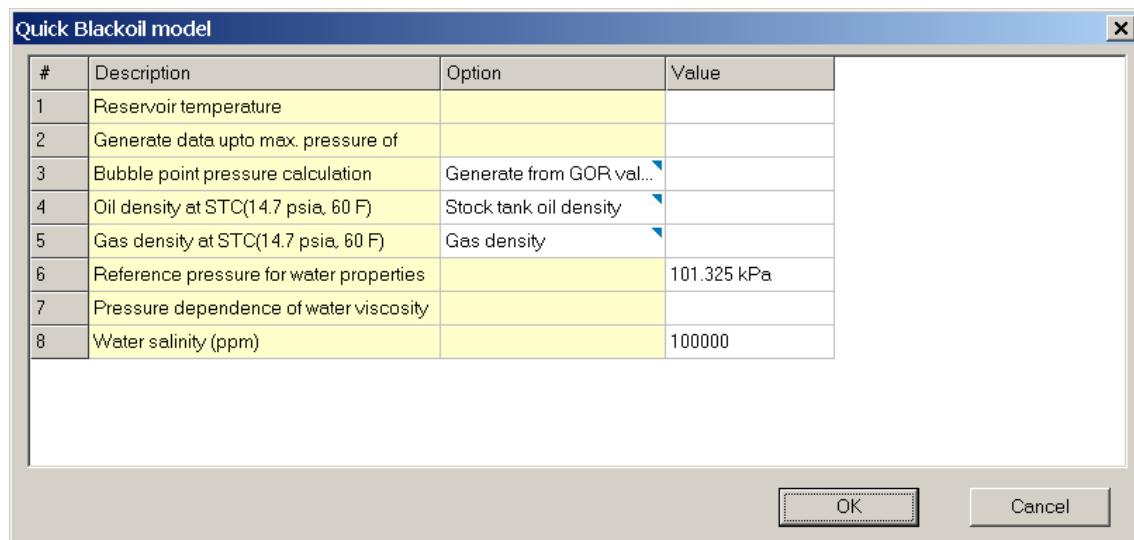
- In the main tree view, double-click the **MODEL** node; or
- Select **Component | Quick Fluid Model** menu item; or
- Select **Component | Model** menu item.

The **Option to create a QUICK model** dialog box is displayed:



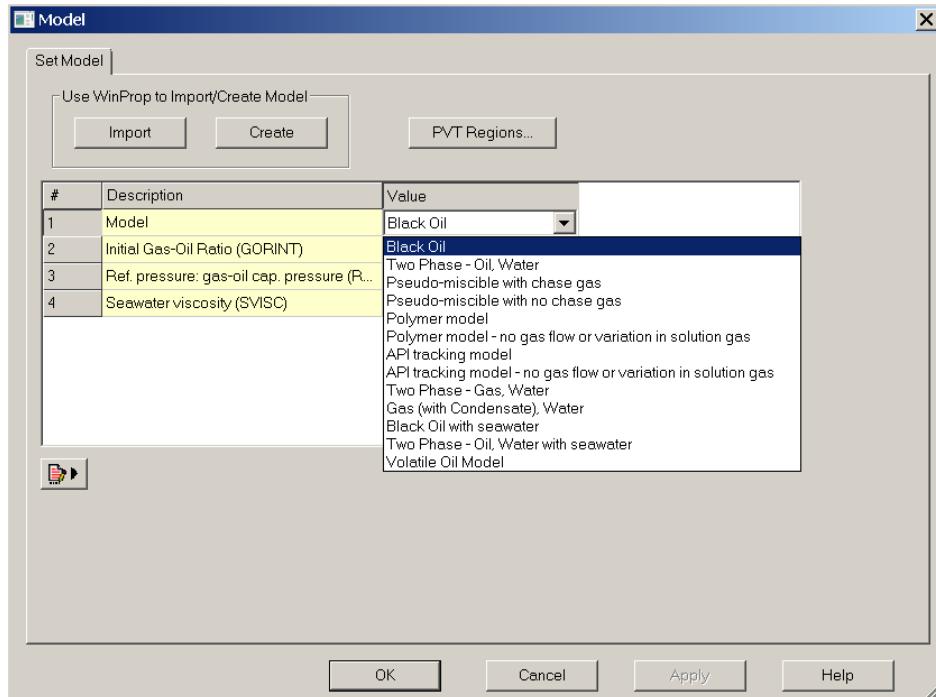
The quick model options will create all the fluid model data required by IMEX. The detailed dialog box will simply set the model, leaving it to you to create other data such as PVT table and densities. However, here you have an option to select any one of the models available in IMEX.

Creating Quick Models



The dialog boxes for BlackOil and GasWater models are similar to the above. You can enter bubble point pressure directly or let the program calculate it from GOR. Enter oil density directly or as API or gravity. Enter gas density directly or as gravity.

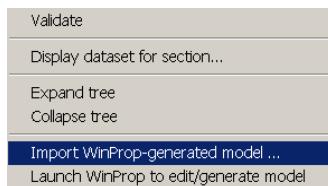
Using the Detailed Fluid Model Dialog Box



Click in the **Value** cell of the **Model** row. This will enable a drop-down box in the cell. Select the model and click **Apply**. You can then go directly to the **IMEX PVT Regions** dialog box to add other data (PVT table, densities, and so on) or quit.

Importing a WinProp Generated Model

Select **Component | Import WinProp generated model** and specify the WinProp generated file (generally with extension imx). Builder will read the file and create the model. You can also select the same menu from the main tree-view context menu:



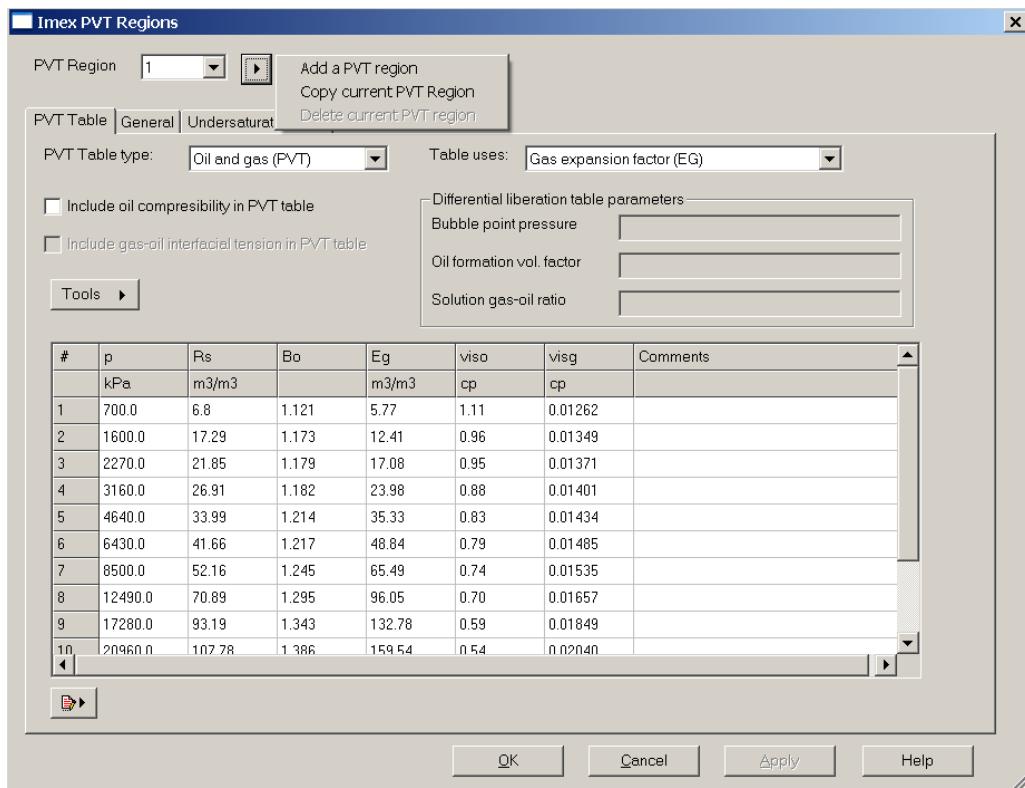
Changing an Existing Model

Launch the **Model** dialog box either by double-clicking the **Model** node in the main tree-view or selecting **Component | Model** menu. Select the new model in the **Value** cell of the **Model** row. Depending on the existing model, Builder may display a list of the data in the existing model that will be deleted if you proceed with the new model. It may also display a list of the required data that need to be entered.

You can also change an existing model by importing a WinProp generated model as described earlier.

Working with Multiple PVT Regions

The data for each PVT region is entered in the **Imex PVT Regions** dialog box. This dialog box has several tabs. The number of tabs may change depending on the fluid model. The **PVT Table**, **General** and the **Undersaturated** Data tabs will always be there.



One PVT region is always defined. The current PVT region number is displayed in the PVT Region drop down box which lists all the regions currently defined. To create a new PVT region, click the right-arrow button next to the PVT Region drop-down box and select **Add a PVT Region** menu. To copy an existing region, select the **Copy current PVT Region** menu.

To delete the current region, select **Delete current PVT Region** menu. This menu will be disabled if there is only one PVT region.

Creating/Editing PVT Tables

The PVT table is located on the **PVT Table** tab of the **Imex PVT Regions** dialog box. For API-INT and API-INTOW models, this tab is called **Gas PVT Table (API)**. You can launch the dialog box by double-clicking on the **PVT Table** node of the main tree view or by selecting **Components | Add/Edit PVT table** menu.

Use the  button to edit or enter comments for the PVT table. Enter comments for individual rows of the table in the **Comments** column.

Creating a New Table

Enter values in a new table in the following manner:

- Type in the values. If you type in with a unit string, Builder will convert the values and re-display.
- Generate a new BlackOil or GasWater type table using correlations.
- Copy and paste from a spreadsheet program.

Editing a Table

Simply type to edit values in the table. If you type in with a unit string, Builder will convert the values and re-display.

You can also do the following:

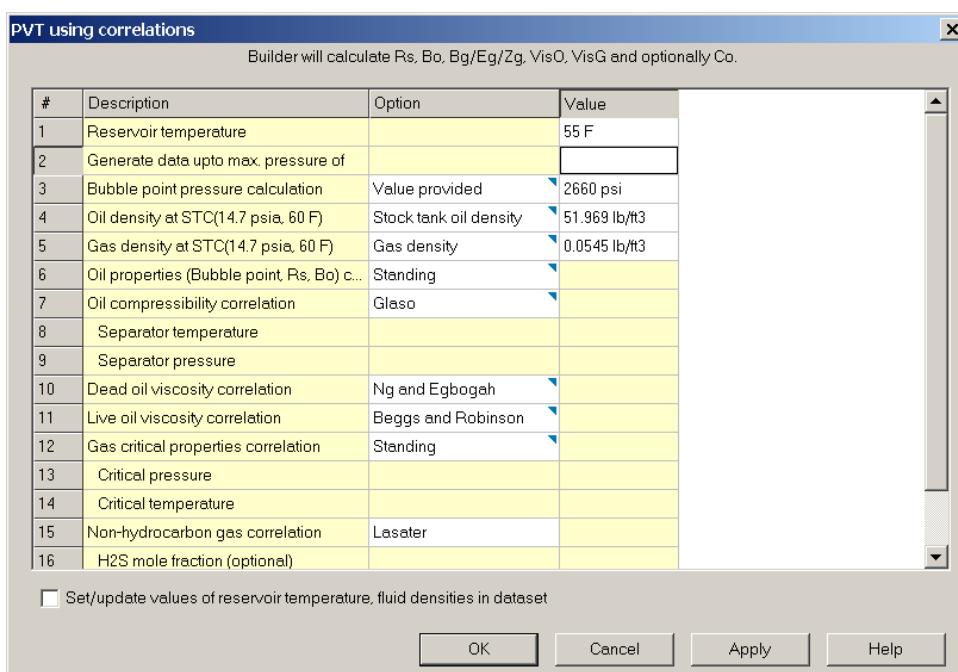
- Add or delete the compressibility and/or interfacial tension columns in the table.
- Set how the gas column is displayed – Gas expansion factor, Gas formation volume factor or Gas compressibility factor.
- Edit differential liberation table parameters – bubble point pressure, oil formation volume factor and solution GOR.
- Copy and paste from a spreadsheet program.
- Perform the following operations using the tools (click the  button):
 - **Generate PVT Table Using Correlations:** Replace existing table by generating a new table using correlations. Refer to [Generating a PVT Table Using Correlations](#) for further information.
 - **Fix Negative Total Hydrocarbon Compressibility:** This check is made by IMEX when it reads in a file.
 - **Shift Column Values to Match:** Change an entire column after fixing a value in a column (and optionally the pressure).
 - **Temperature Shift PVT Table Values:** Shift the entire table from one temperature to another.

- You can extrapolate and interpolate the IMEX PVT table one row at a time. Add a row to the table, enter the pressure value which you want use for interpolating/extrapolating other columns, and then select the **Interpolate Values** or **Extrapolate Values** menu items.

Generating a PVT Table Using Correlations

You can generate a PVT table for BlackOil and GasWater type of models using correlations.

Click on the **Tools** button on the **PVT Table** tab of the **Imex PVT Regions** dialog box. Select the **Generate PVT Table using correlations** menu.



Enter the missing values on the dialog box. If the value for bubble point is available, its maximum value is entered automatically in the **Bubble point pressure calculation** value box. You can change the value displayed. Alternatively, enter an initial GOR value which is used by Builder to calculate the initial bubble point.

Select correlations for computing the oil properties, compressibility, oil viscosity and gas critical properties.

If you would like Builder to automatically enter or replace the values for reservoir temperature and density, check the **Set/update values of reservoir** check box at the bottom before clicking **Apply** or **OK**.

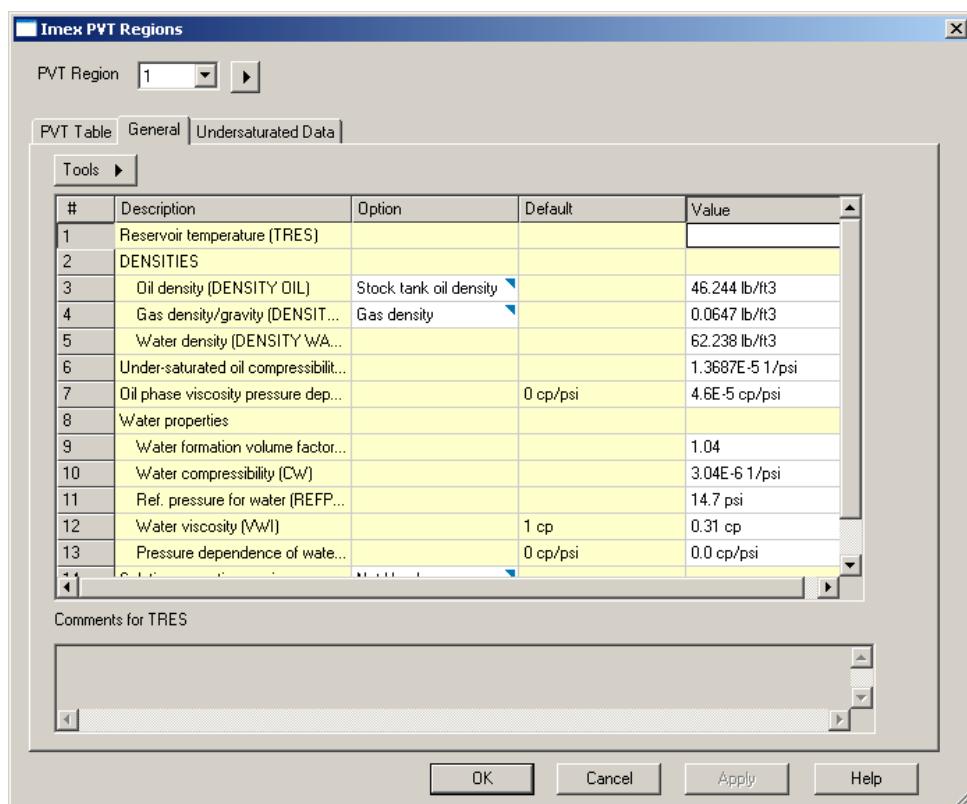
When Builder finishes the calculations, it will offer to include computed oil compressibility values in the generated table.

It will also offer to set or replace the bubble point pressure for all the grid blocks in the PVT Region to the value calculated.

The black oil and related properties for IMEX are generally carried out using the well-known correlations reviewed by McCain [McCain, W.D. Jr., "Reservoir-Fluid Property Correlations - State of the Art", SPE Reservoir Engineering (May 1991)] and the SPE monograph on Phase Behavior. The actual calculations depend on the choices made on the detailed dialog box. For the quick PVT generation feature, most of the correlations default to those by Standing and co-authors. The compressibility factor (z-factor) calculation for gas uses the Standing-Katz correlation. The gas viscosity is calculated using the Lee-Gonzalez correlation.

Entering Other PVT Region Properties

These are entered on the **General** tab of the **Imex PVT Regions** dialog box.



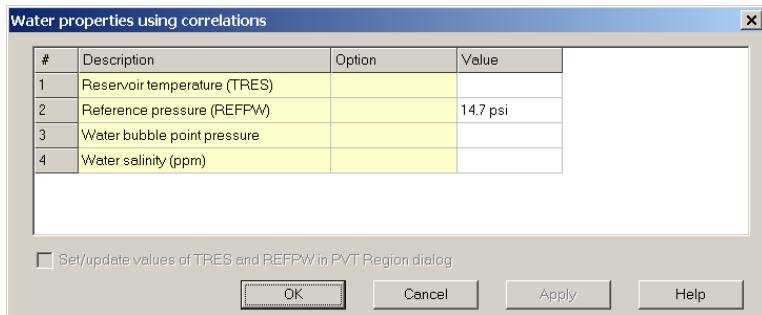
You need not enter unit strings if the values are already in the correct units. Builder will always display the unit strings in the boxes. For oil density three options are available: mass density, gravity or API gravity. For gas, you can enter as density or gravity.

Not all data are required. If you do not enter a required item, Builder will point that out when you click **Apply** or **OK**.

Comments for each data item in the grid are displayed and edited in the text window at the bottom of the dialog box.

Generating Water Properties Using Correlations

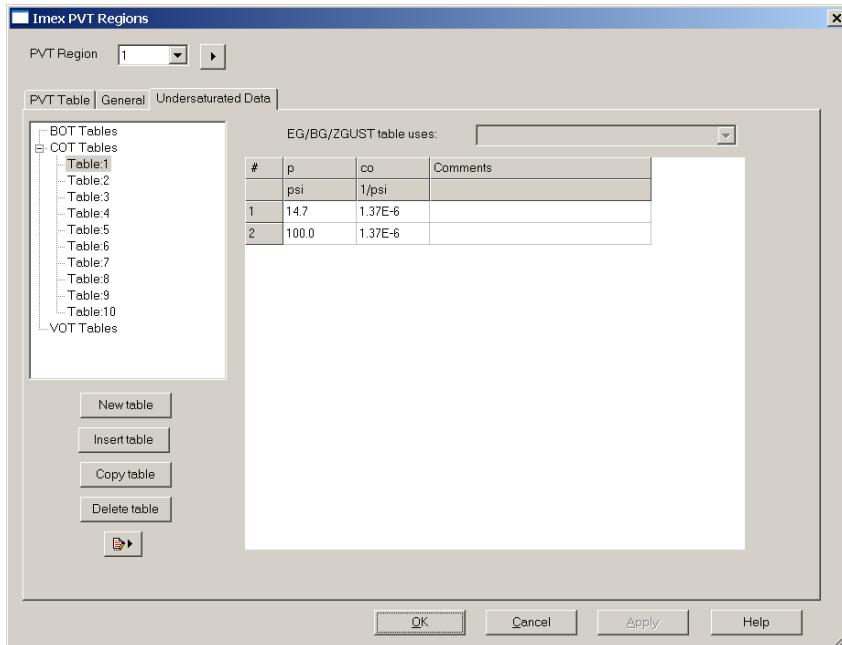
To generate water properties using correlations, click the button and select **Generate water properties using correlations** menu.



Enter the data and click **Apply** or **OK**. You can set or replace the reservoir temperature and/or reference pressure values by checking the **Set/update values of TRES** check box at the bottom.

Entering Undersaturated Data

These are entered on the **Undersaturated Data** tab of the **Imex PVT Regions** dialog box:



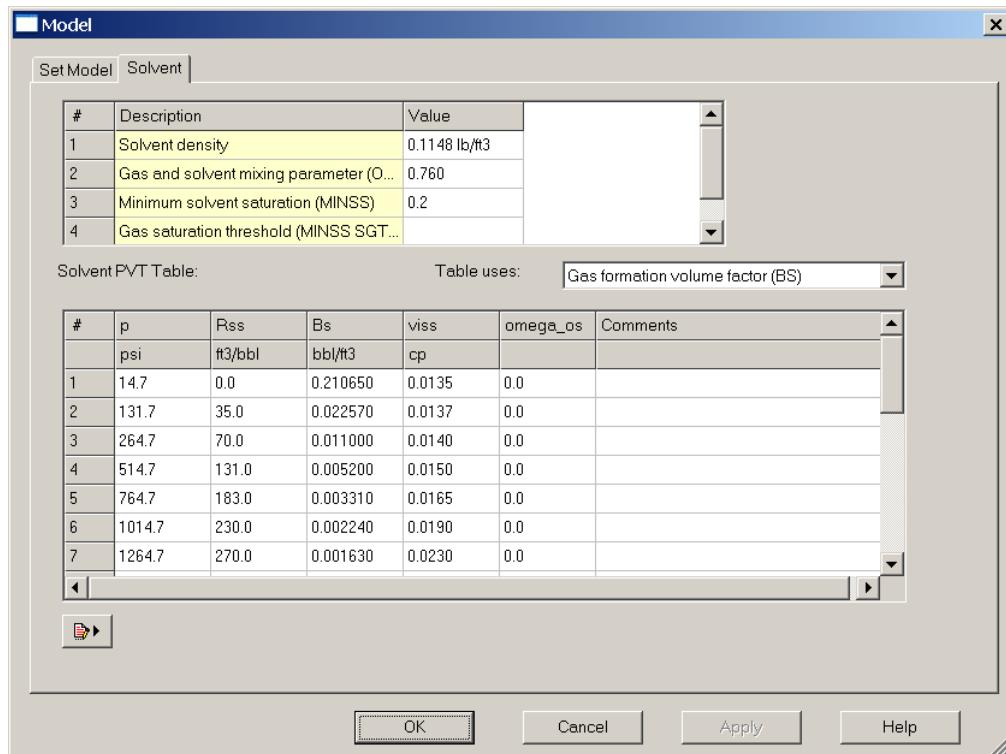
Depending on the fluid model, you can work with the following tables on this tab:

- Oil compressibility as a function of pressure (COT table)
- Oil formation volume factor (B_o) above the bubble point as a function of pressure (BOT table)
- Oil viscosity above the bubble point as a function of pressure (VOT table)
- Condensate model – Undersaturated Eg, B_g , Z_g table
- Condensate model – Undersaturated gas viscosity table

Use the appropriate buttons to add, copy or delete tables.

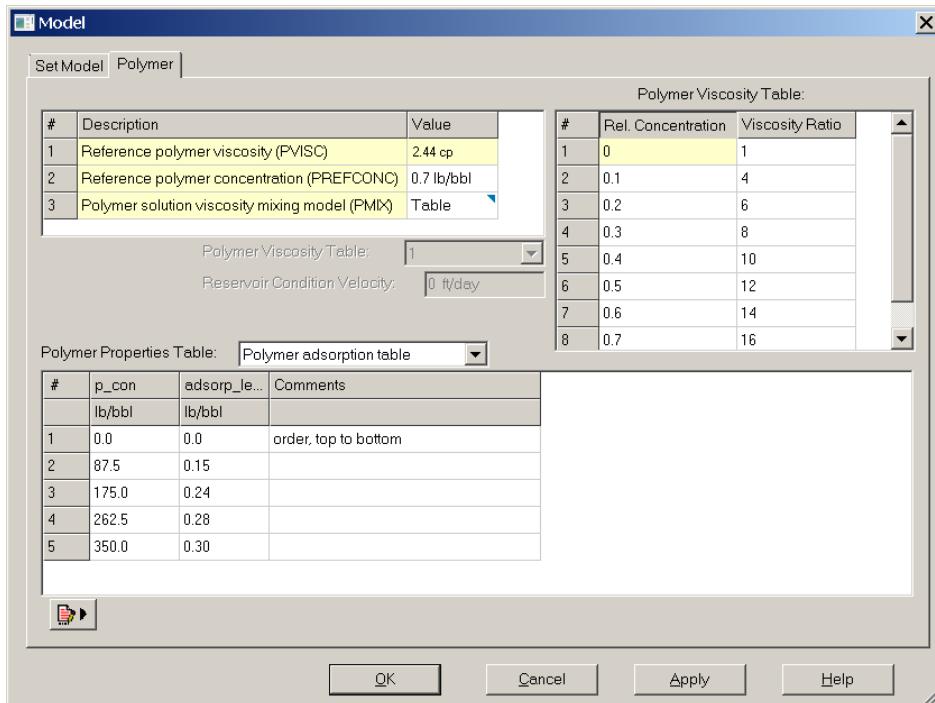
Entering Data for the Pseudo-miscible Option

Enter the PVT table for the hydrocarbons on the **PVT Table** tab of the **Imex PVT Regions** dialog box. Enter the solvent properties and its PVT table on the **Solvent** tab of the **Model** dialog which can be launched by left double-clicking the **Solvent PVT Table (PVTS)** node on the main tree view or by selecting **Components | Model** menu.



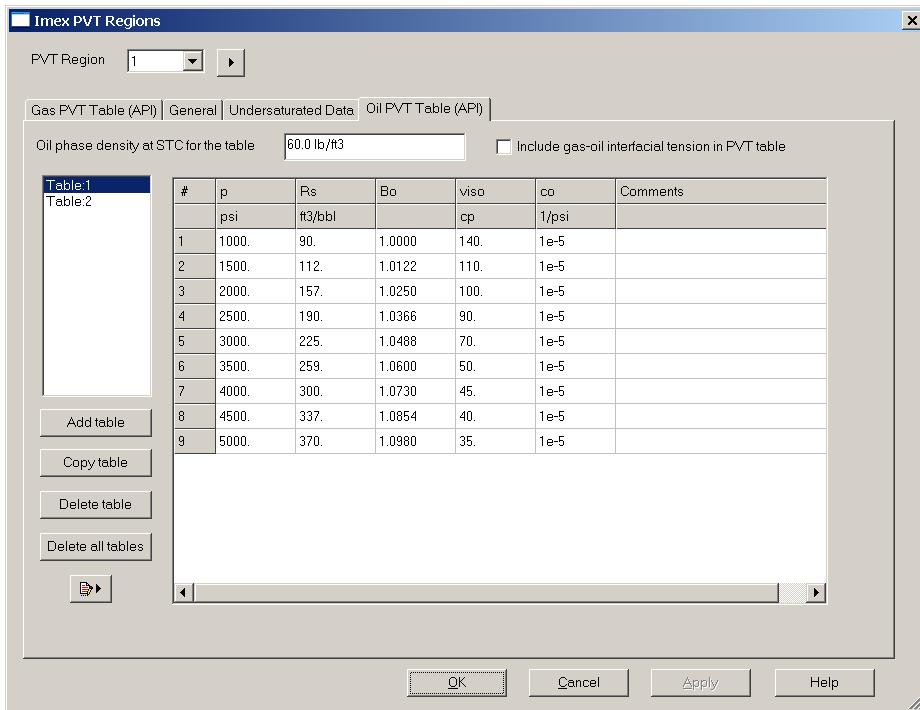
Entering Data for the Polymer Option

Enter the PVT table for the hydrocarbons on the **PVT Table** tab of the **Imex PVT Regions** dialog box. Enter the polymer properties, polymer adsorption and permeability tables, and polymer viscosity table on the **Polymer** tab of the **Model** dialog box which can be launched by left double-clicking the polymer related nodes on the main tree view or by selecting **Components | Model**.



Entering Data for the API Tracking Option

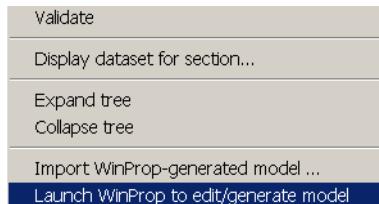
The gas PVT table for the API model is located on the **Gas PVT Table (API)** tab of the **Imex PVT Regions** dialog box. Enter the oil PVT tables at different oil phase density values on the **Oil PVT Table (API)** tab.



The oil tables must be entered in the order of decreasing oil density at STC.

Launching WinProp with Current Fluid Model Data

You can launch WinProp with the current fluid model data so that you work with the model in WinProp. Select the **Components | Launch WinProp to edit/generate model** menu. You can also select the same menu from the main tree view context menu:



After you have worked with the model in WinProp, you can import it back into Builder as explained earlier.

Displaying Component Properties Partial Dataset

You can view the Component Properties section data in keyword format by selecting the **Display dataset for section** menu from the main tree view context menu.

Data Validation

Builder automatically performs validation of the data when the following occur:

- You read in a dataset
- You add or modify data in the fluid model

You can also force validation by selecting the **Validate** menu from the main tree view context menu.

Fluid Model - GEM

Overview

You can use Builder to:

- Specify a model, reservoir temperature and water properties
- Create Equation of State sets
- Create components and specify:
 - Pure component properties
 - Interaction coefficients
 - Viscosity of components
 - Aqueous phase solubilities
- Import fluid model data generated using WinProp
- Launch WinProp with the current fluid model data

Builder will read and validate fluid model data in a dataset.

You can use the Quick CBM Setup to quickly create partial data required for Coal Bed Methane simulation. You can subsequently add grid and well data to complete the dataset.

To export the current data to WinProp, select **Components | Launch WinProp to edit/generate model**. Or you can launch the **Model** dialog box and click the **Create** button. To launch the **Model** dialog box select **Components | Model**. Alternatively, double-click the **Model** node in the tree view to launch the **Model** dialog box.

To import model from a WinProp file, select **Components | Import WinProp-generated model** and specify the file, or you can launch the **Model** dialog box and then click **Import**.

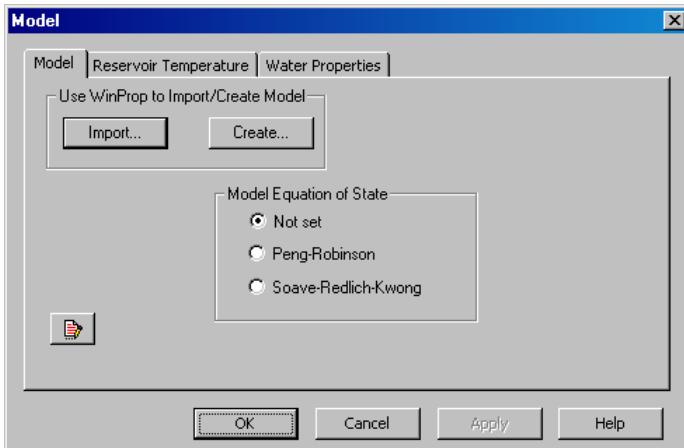
Library and User Components

The syntax for the GEM fluid model allows you to distinguish between a library component and a user component. When Builder reads a dataset, it converts all the library components to user components by bringing in the pure component data. If you save the dataset, Builder will write out the component section with all the components specified as user components.

In view of the above, all the user interfaces for the GEM fluid model in Builder are designed for user components only.

Specifying a Fluid Model

Launch the **Model** dialog box. If no model has been specified, the **Not set** option will be selected. To specify or change the model, make the appropriate selection.

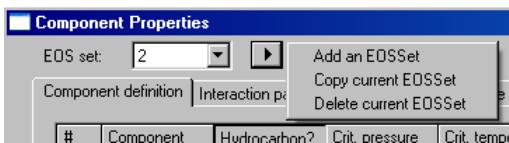


Use the button to edit or enter comments for MODEL keyword.

Adding/Editing an Equation of State

Use the **Component Properties** dialog box to add, edit or delete an Equation of State.

To launch the dialog box, select **Components | Add/edit components**. Alternatively, if components have already been defined, click the **Components** button on the main tree view. Double-click any node to bring up the dialog box.



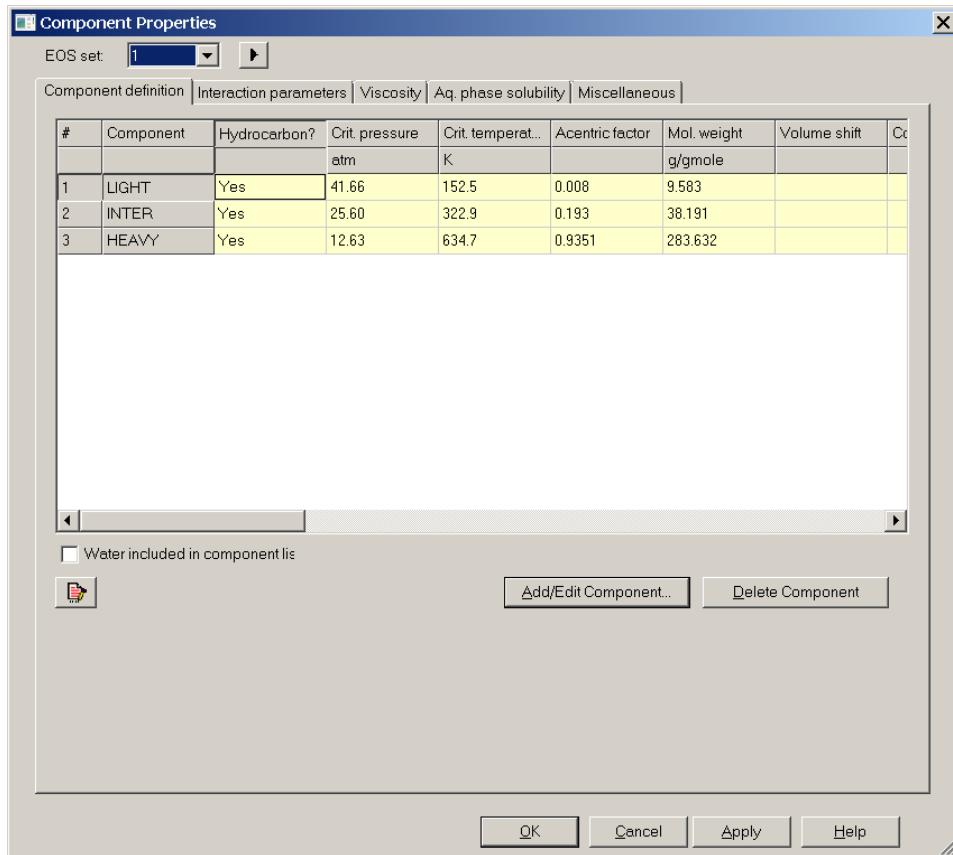
Use the right-click button next to **EOS set** to add, copy or delete an equation of state. The **Add an EOSSet** works similar to **Copy current EOSSet** except when none is defined.

Adding/Editing Components

The component definition appears on the **Component definition** tab of the **Component properties** dialog box.

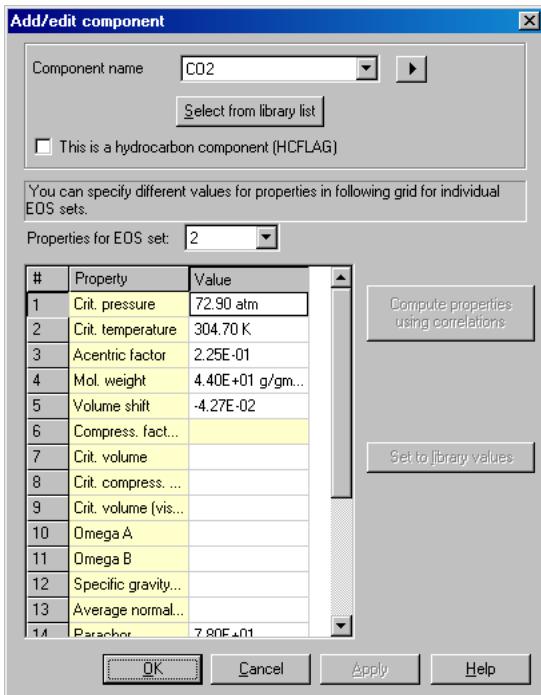
Before you can add components, an EOSSet has to be created.

The **Component definition** tab displays the component/phase table along with pure component properties.

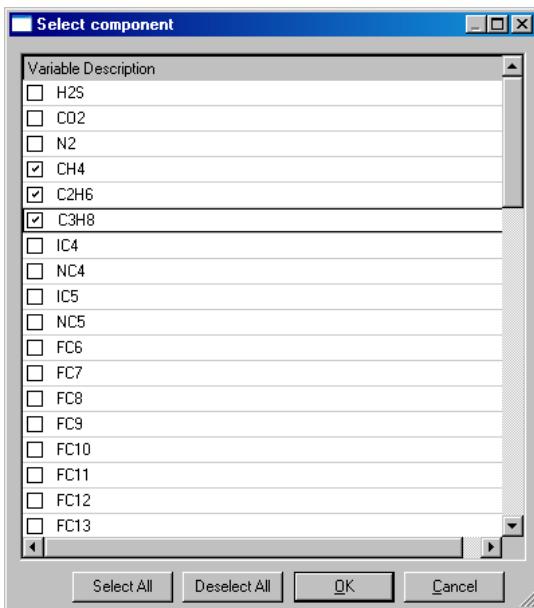


Adding/Editing a Component

Click the **Add/Edit component** button to bring up the **Add/edit component** dialog box.



To add a component from the library, click **Select from library list**:



You can select multiple components from the list. On exit, Builder will read from the library all the available properties and display the first selected property in the **Add/edit component** dialog box. You can edit the properties in the dialog box. Some of the properties, which are not required in GEM, are displayed as read-only values.

To add a non-library component, use the  button next to the **Component name** drop-down box in the **Add/edit component** dialog box and then select **Add a Component** from the pop-up menu.



Enter the name of the component in the **Input Name** dialog box:



You can select **Copy Current Component** from the pop-up menu to copy the currently displayed component in the **Add/edit component** dialog box. Builder will ask you for a name for the new component.

You can also change the name of an existing component by selecting the **Change Name of Current Component** item from the pop-up menu.

To specify a component as a hydrocarbon, select the **This is a hydrocarbon component (HCFLAG)** check box. Cancel the check box to specify a non-hydrocarbon.

To change the properties of a component for a particular EOS Set, select the set in the **Properties for EOS Set** drop-down box, change the properties and then click **Apply**.

Deleting a Component

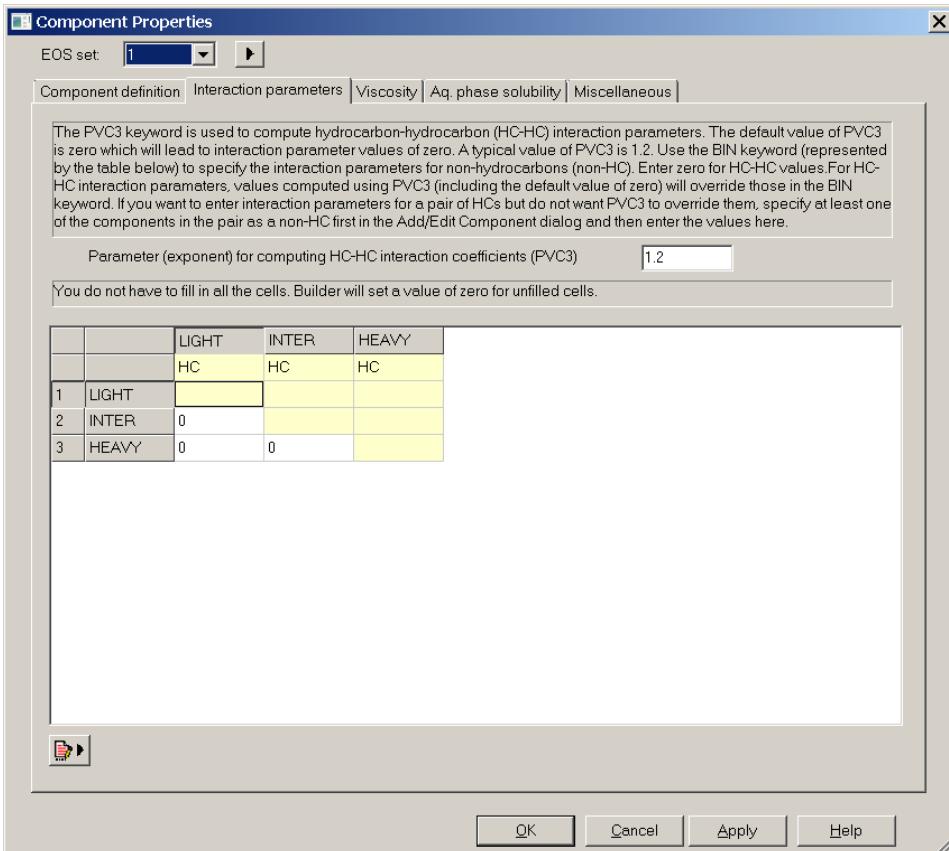
Select the row corresponding to the component in the grid on the **Component definition** tab of the **Component Properties** dialog box and then click **Delete Component**.

Adding/Editing Comments

Select the column (click in any cell in the column) in the grid on the **Component definition** tab of the **Component Properties** dialog box corresponding to the property for which you want to add/edit comments. Click the  button to display the **View/Edit Comments** dialog box.

Entering Binary Interaction Coefficients

You can add or edit the coefficients on the **Interaction parameters** tab of the **Component Properties** dialog box.



The interaction coefficients for each pair of hydrocarbon components are overwritten in GEM by the value of PVC3. If you do not specify a value for PVC3, GEM uses the default value of 0.0 (zero). To specify interaction coefficients for a hydrocarbon pair, at least one of the pair should be a non-hydrocarbon.

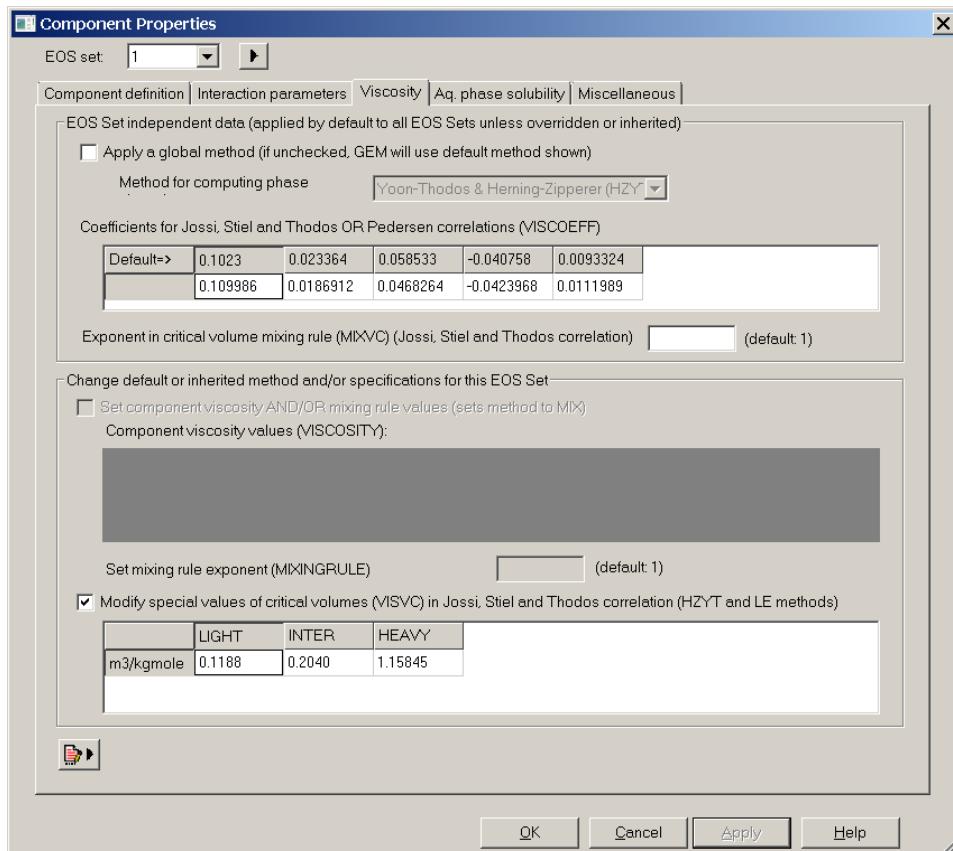
Coefficients can be set individually for EOS sets. Select the desired EOS Set from the **EOS Set** drop-down box at the top of the dialog box.

Adding/Editing Comments

Use the button to add or edit comments for the BIN and the PVC3 keywords.

Entering Viscosity Data

Enter the viscosity data on the **Viscosity** tab of the **Component Properties** dialog box.



To override the default global method, check the **Apply a global method** check box and select the desired method from the **Method for computing phase viscosity** drop-down box. To apply the default method, cancel the check box.

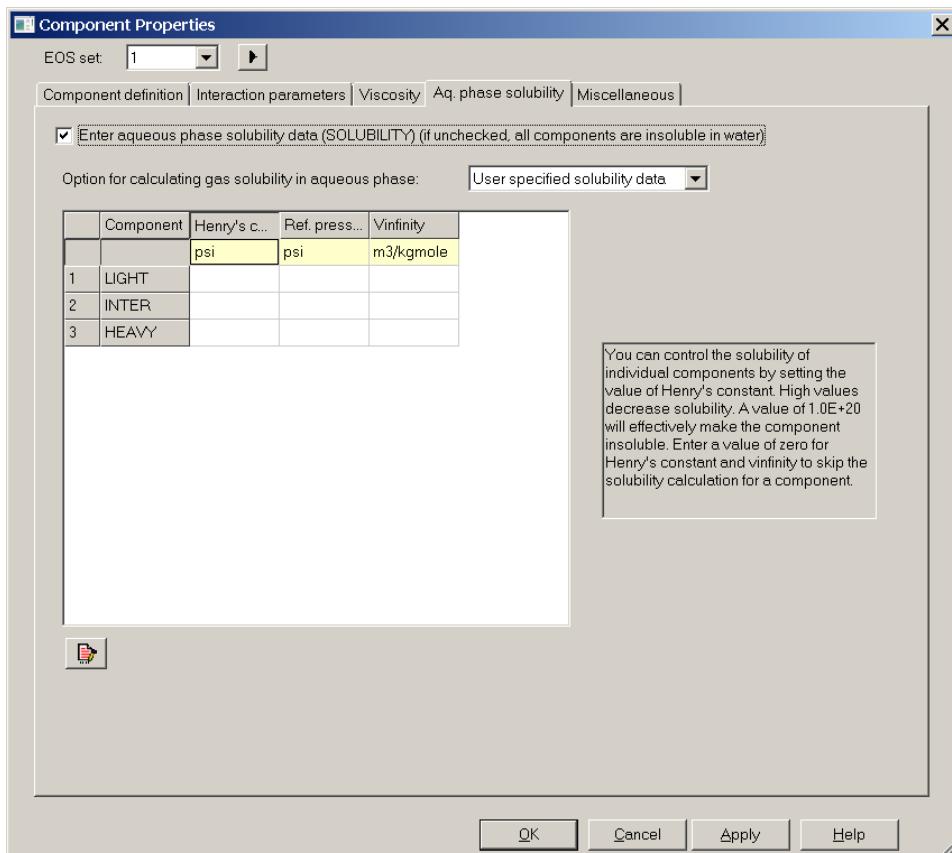
Viscosity data can be specified individually for EOS sets. Select the desired EOS Set from the **EOS Set** drop down box at the top of the dialog box.

Adding/Editing Comments

Use the button to add or edit comments for the various keywords associated with viscosity data.

Entering Aqueous Phase Solubility

Solubility data are entered on the **Aq. Phase solubility** tab of the **Component Properties** dialog box.

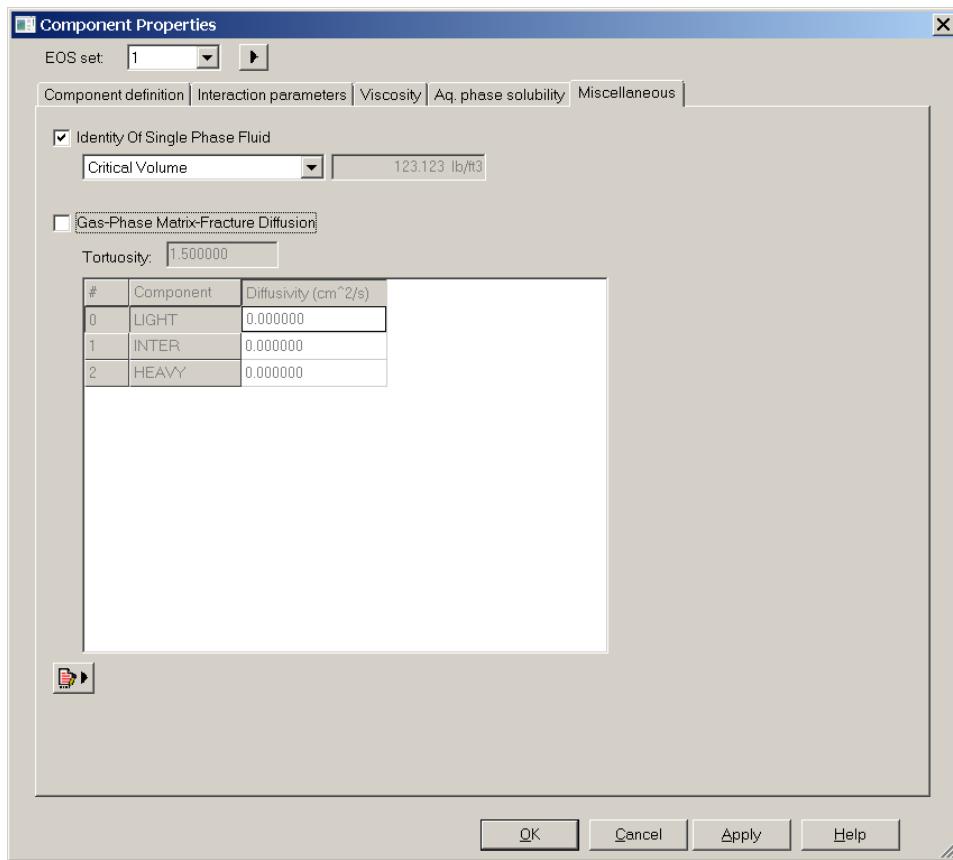


Select **Enter aqueous phase solubility data (SOLUBILITY)** to enter the data. Cancel the check box if all the components are insoluble in water. Use the **Option for calculating gas solubility in aqueous phase** drop-down box to specify computation method.

Viscosity data can be specified individually for EOS sets. Select the desired EOS Set from the **EOS Set** drop-down box at the top of the dialog box.

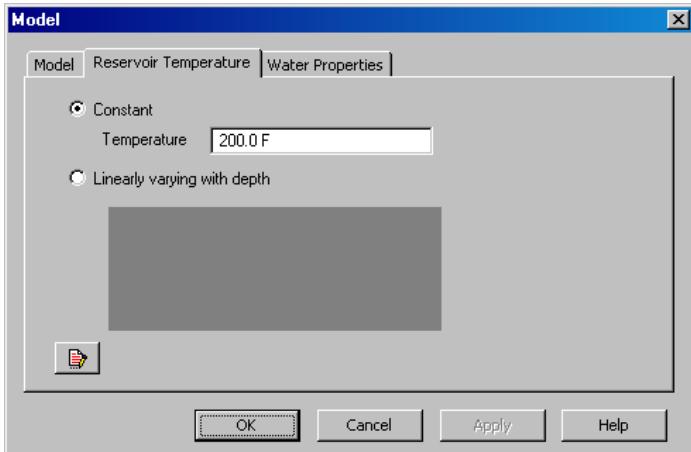
Entering Single Phase Fluid and Gas Diffusivity

Identification of single phase fluid and gas diffusivity are entered on the **Miscellaneous** tab of the **Component Properties** dialog box.



Entering Reservoir Temperature

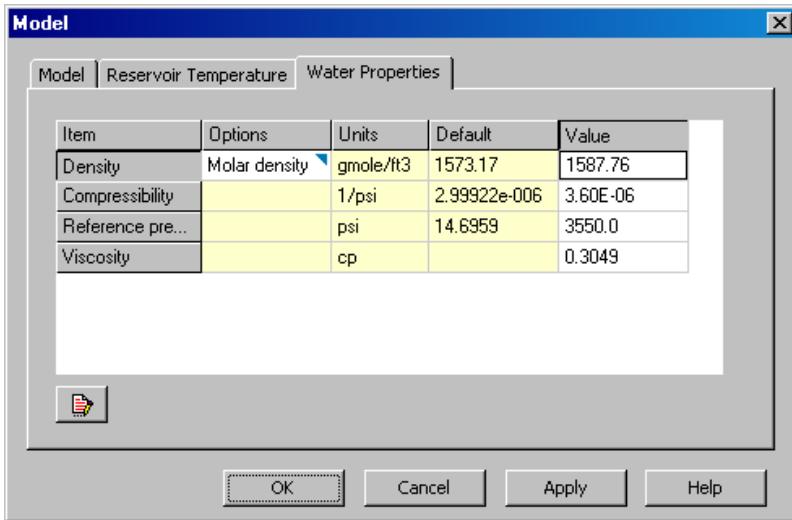
Use the **Reservoir Temperature** tab on the **Model** dialog box to enter either a constant value or a gradient for reservoir temperature.



Use the button to enter comments for TRES.

Entering Water Properties

Water properties are entered on the **Water Properties** tab of the **Component Properties** dialog box.



Select the row (click in any cell in the row) in the grid on the **Water Properties** tab corresponding to the property for which you want to add or edit comments. Click the button to display the **View/Edit Comments** dialog box.

Displaying Component Properties Partial Dataset

You can view the Component Properties section data in keyword format by selecting the **Display dataset for section** menu from the main tree view context menu.

Data Validation

Builder automatically performs validation of the data when the following occur:

- You read in a dataset
- You add or modify data in the fluid model

You can also force validation by selecting the **Validate** menu from the main tree view context menu.

“Quick” Coal Bed Methane (CBM) Setup

Builder provides an interface to quickly create data required for fluid, rock-fluid and initial conditions sections.

This feature has been enhanced in the 2007 version. In the previous releases:

- One Langmuir curve for each component was applied to the whole reservoir
- Only two initialization schemes were handled: *USER_INPUT and *VERTICAL *BLOCK_CENTER *COMP
- Most of the data were added to the dataset by Builder. You could, of course, modify them outside the CBM interface

Due to the above characteristics, the feature was a “Quick” method. The enhanced feature gives you more control in terms of specifying the data:

- You can now specify separate curves for different regions of the reservoir:
 - Sectors
 - Compaction/dilation regions
 - Rock-fluid regions
- You can now also work with *VERTICAL *BLOCK_CENTER *WATER_GAS and *VERTICAL *DEPTH_AVE *WATER_GAS options for initializing the reservoir
- You can specify the following properties in the CBM interface itself (Matrix and Fracture, whole grid only):
 - PERMI, PERMJ and PERMK
 - POR
 - SW
 - DIFRAC, DJFRAC, DKFRAC (Matrix only)

- In addition, you can also specify the following Fracture properties if you are working with *USER_INPUT initialization scheme and multiple regions:
 - ZGLOBALC
 - PRES

Note: The “Quick” method is still available and in fact that is the view presented to you first. To access the advanced method, click the **Advanced CBM modeling** button at the top of the dialog box. If you are opening an existing dataset containing CBM data and if you had previously worked in the advanced mode, Builder will automatically put you in the advanced mode. If you are in the advanced mode you can still work in the “Quick” mode by selecting the options that corresponds to the “Quick” method.

Before you access this feature, create or import the simulation grid. In addition, depending on the type of region you want to specify the Langmuir curves for, other properties must be defined. If they are not defined, the wizard gives you a chance to define them before launching the dialog where you can input the Langmuir model parameters. The following table summarizes the required properties.

| Region | Required properties and comments |
|---------------------|--|
| Sectors | Sectors. You can create these from within the wizard (graphical input of the member blocks is not enabled). |
| Compaction/dilation | Compaction Rock tables (*CROCKTYPE) and related properties. Refer to the <i>GEM User’s Guide</i> . If you have not already created these tables, you could launch the dialog to create them from within the wizard. Also, an array assigning the tables to grid blocks (*CTYPE) is required. This can be created from within the wizard (constant values only for each k-layer). |
| Rock-fluid | Relative permeability curves (*RPT, *SWT, etc.). There is no provision to create the relative permeability curves from within the wizard. Also, an array assigning the curves to grid blocks (*RTYPE) is required. This can be created from within the wizard (constant values only for each k-layer). |

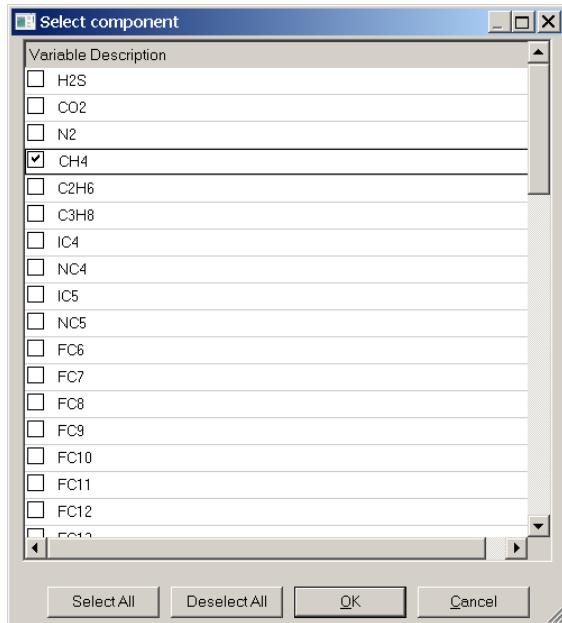
You will also need to create or import the well data to complete the model.

To launch the quick CBM setup, select **Components | Quick CBM setup**.

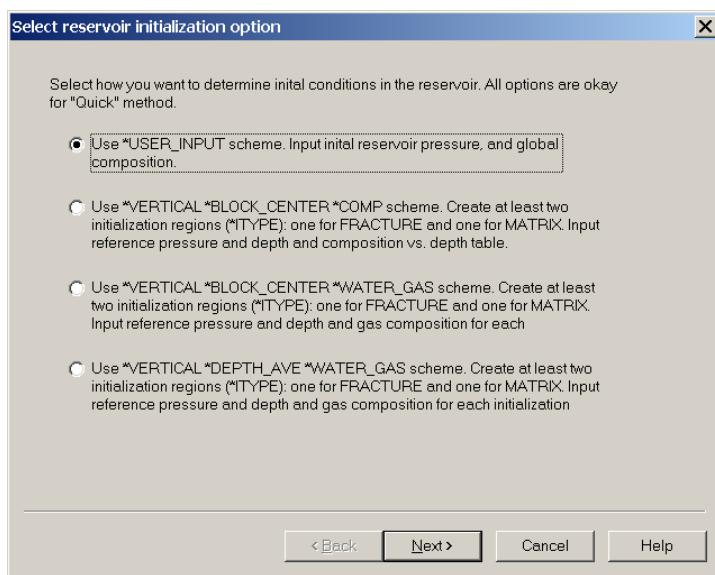
New Case

If you started off with a single-porosity model, Builder will offer to convert to a dual porosity model. If you decline, the feature would end there since CBM requires fracture data.

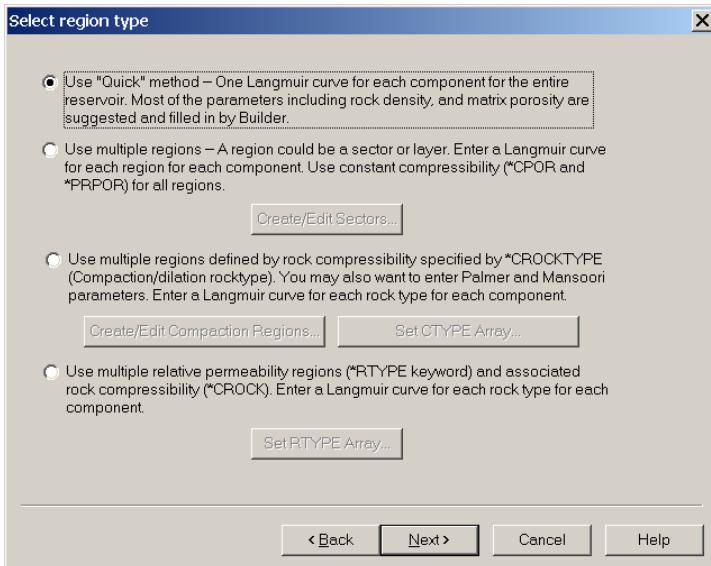
The **Select Components** dialog box is displayed. Select the names of the library components.



If you have not already specified how the initial reservoir conditions are to be determined (in the **Initial Conditions** section), the **Select reservoir initialization option** dialog box is displayed:



As indicated earlier, you now have more options to select from. Select a scheme and then click **Next**. The **Select region type** dialog box is displayed:



For “Quick” method, accept the default setting. Otherwise make a selection from the list of options.

Click **Next** to launch the **Coal bed methane model data** dialog box.

If you are working with Sectors and have not defined any sectors or want to add or delete sectors, click the **Create/Edit Sectors** button to launch the **Add/Edit/Delete a Sector** dialog box.

If you have specified *USER_INPUT as the initialization scheme and have not specified a single sector for the Fracture blocks, Builder will prompt you to add one or more sectors for the Fracture blocks. This is because the initial reservoir pressure (*PRES) and global composition (*ZGLOBALC) for both Matrix and Fracture spatial property arrays are required with *USER_INPUT. Note that the **Sectors** dialog box lets you create sectors for the Matrix and Fracture blocks simultaneously.

If you have not specified any relative permeability curves and want to select the option for multiple relative permeability options, you cannot proceed. You will have to exit the wizard (click **Cancel**), create the curves and then come back.

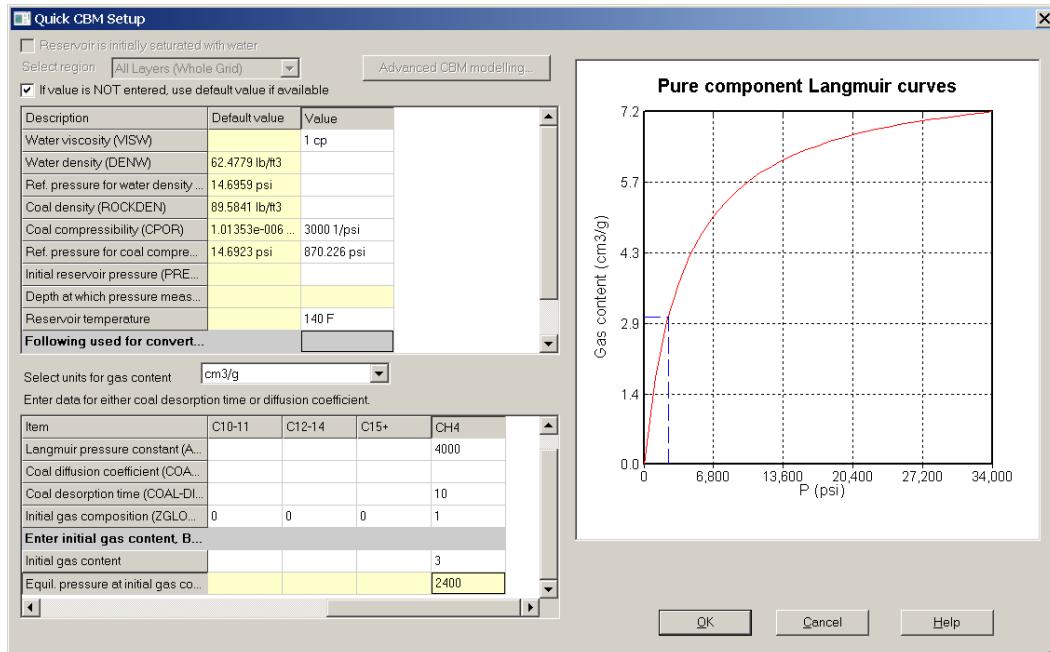
If you have not created compaction/dilation data (or want to edit them) and want to proceed with multiple rock compressibility regions, you can create them by clicking the **Create/Edit Compaction Regions** button (this brings up the **Compaction/Dilation** dialog box). You can also enter *Palmer-Mansoori parameters* using the **Compaction/Dilation** dialog box.

If you have not created the *CTYPE array or want to modify them, click the **Set CTYPE Array** button.

If you have not created the *RTYPE array or want to modify them, click the **Set RTYPE Array** button.

With Compaction/dilation tables or with multiple relative permeability curves, it is recommended that you also specify *CTYPE or *RTYPE for fracture blocks.

If you are using the "Quick" method, the following dialog box, which is almost identical to the one in the previous versions, is displayed:



Enter values for **Maximum gas content**, **Langmuir pressure**, **Initial gas content**, **Initial gas composition**, **Initial reservoir pressure**, **Coal density** and **Coal compressibility**.

Enter either **Coal desorption time** or **Coal diffusion coefficient**.

Builder displays the pure component Langmuir curves in the plot view. It also displays the equilibrium pressure calculated from the curve and the Initial gas content for each component.

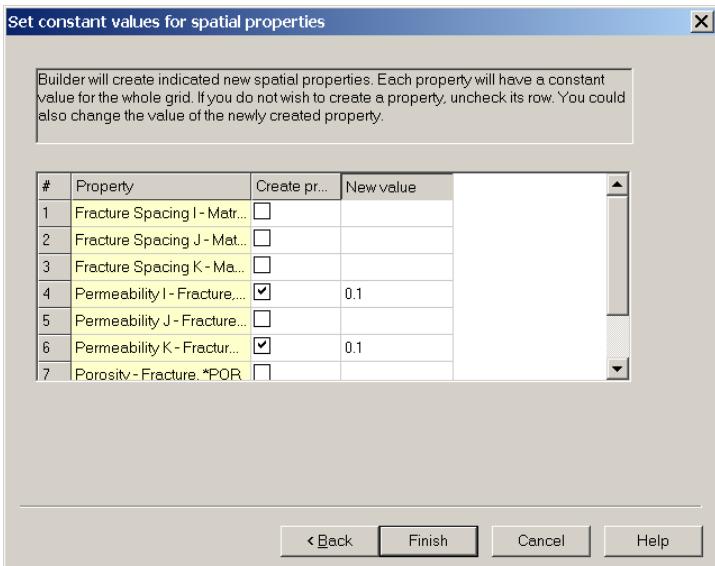
You could select one of the following units to enter gas content: SCF/ton, SCF/long-ton, ft³/lb, m³/kg, cm³/g and m³/tonne.

A value for reservoir pressure is required.

The rest of the data are optional – Builder may enter values for some of these optional data.

The Standard pressure and Standard temperature are required for converting the gas content to mole-basis (GEM requires these data in mass/g mole units).

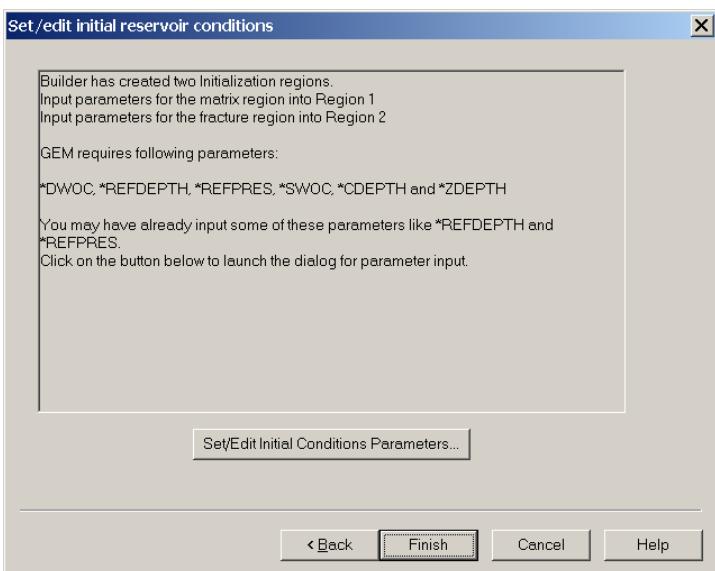
When you click **OK**, Builder will launch the **Set constant values for spatial properties** dialog box:



You can specify constant values for the fracture spacing, permeability in I, J and K directions, porosity and water saturation. Permeability and porosity fields will be filled in by suggested values since they are required properties.

If you are using *USER_INPUT initialization scheme with the “Quick” method, this is the last step in the wizard. Otherwise the **Finish** button in the above dialog box is replaced by the **Next** button.

When you click the **Next** button, Builder will display the **Set/edit initial reservoir conditions** dialog box:



Click **Set/Edit Initial Conditions Parameters** to launch the **Initial Conditions** dialog box. When you return to the CBM wizard, click **Finish** to exit.

Builder will create a number of data objects that are required by GEM (refer to the *GEM User's Guide* for keyword descriptions in the following):

- Components
- Water properties (if you entered them), *TRES (Component Properties)
- Sets the reservoir initialization option
- *ADGCSTC, *ADGMAXC, *ROCKDEN, *COAL-DIF-TIME or *COAL-DIF-COMP
- Two rock types with permeability tables (Rock Fluid section) unless they were already specified
- *PRES and *ZGLOBALC arrays for matrix and fracture (Initial Conditions) if you selected *USER_INPUT
- Fracture spacing and permeability in the I, J and K directions, porosity and water saturation if you entered values for these in the **Set constant values for spatial properties** dialog box.

You can modify the properties by re-entering the wizard. You could also change the spatial properties through the **General Property Specification** and the **Block/Corner Value Calculation** dialog boxes.

You can view or edit the data by re-launching the dialog box.

Existing Case

This feature is available only if the reservoir initialization is specified using one of the following keywords (refer to the *GEM User's Guide* for a description of the keywords):

- *USER_INPUT
- *VERTICAL *BLOCK_CENTER *COMP
- *VERTICAL *BLOCK_CENTRE *WATER_GAS
- *VERTICAL *DEPTH_AVE *WATER_GAS

If the water-oil contact depth (*DWOC) is not specified when you launch the dialog box, the **Reservoir is initially saturated with water** check box will be enabled.

Fluid Model - STARS

Overview

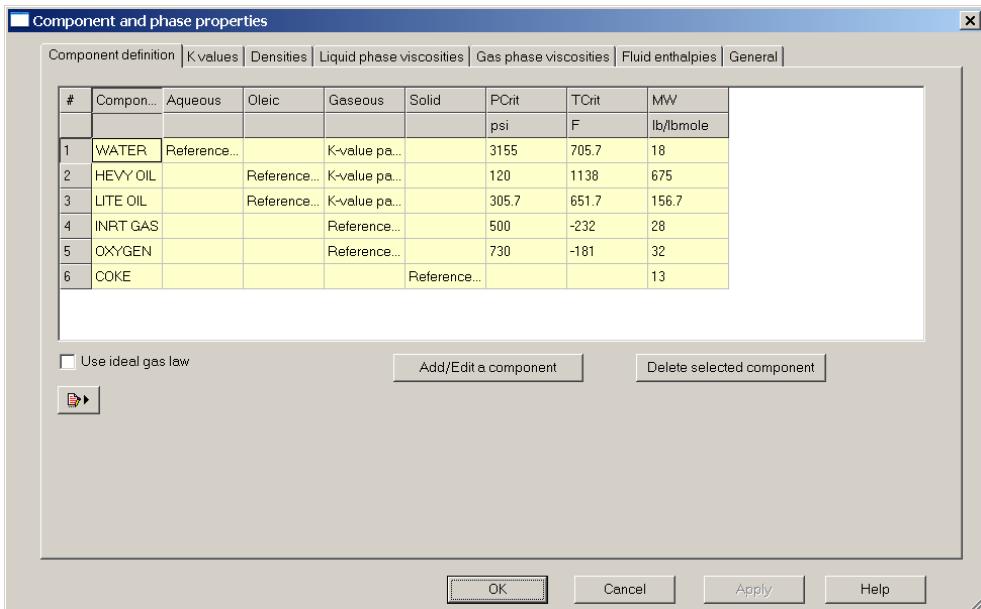
You can use Builder to create components and specify phases in which each component can appear:

- Specify:
 - Pure component properties
 - Gas-liquid and liquid-liquid K values
 - Liquid and solid phase densities
 - Liquid and gas phase viscosities
 - Fluid enthalpies
 - Reference and surface pressure and temperature conditions
 - Component/phase distribution for well production reporting
 - Reactions
- Import fluid model data generated using WinProp
- Import fluid model data generated from BlackOil PVT data
- Launch CMGL's WinProp application with the current fluid model data

Adding/Editing Components

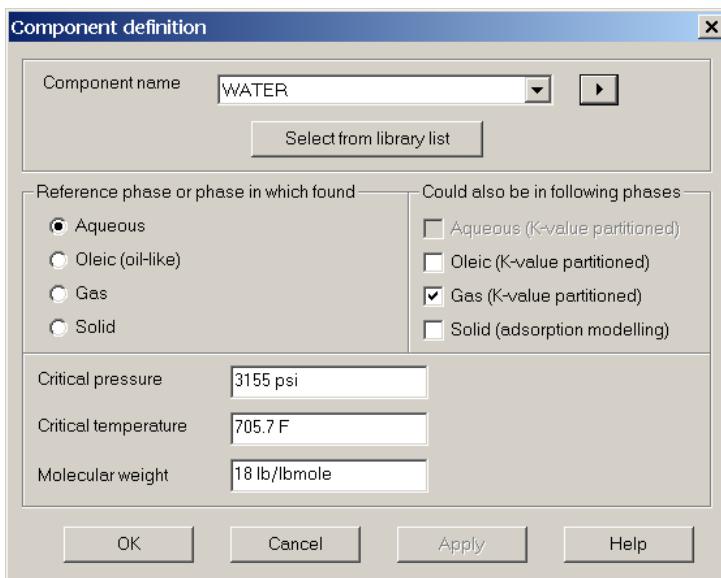
The component definition appears on the **Component definition** tab of the **Component and phase properties** dialog box. To launch the dialog box, select **Components | Add/edit components**. Alternatively, click the **Components** button on the main tree view. Double-click any node to display the dialog box.

The **Component definition** tab displays the component/phase table along with pure component properties.

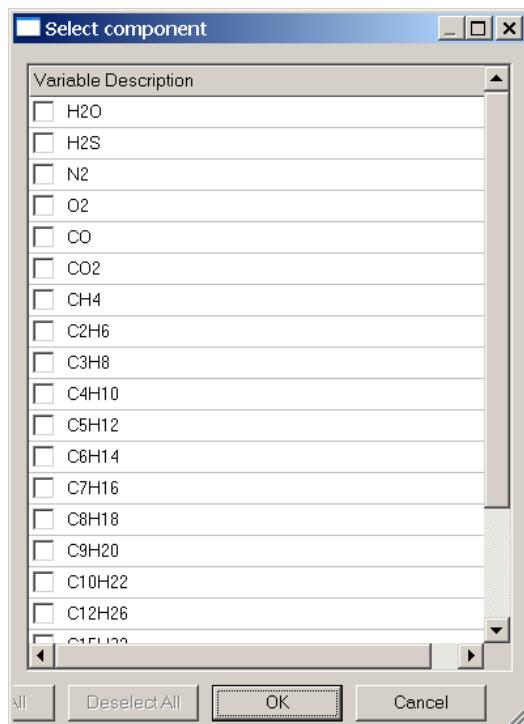


Adding/Editing a Component

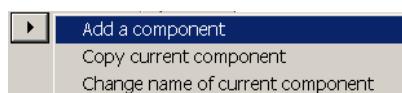
Click the **Add/Edit a component** button to display the **Component definition** dialog box:



You can add a component from a library which is created from the information given in the **Tables** section of the *Stars User's Guide*. Click the **Select from library list** button. The **Select component** dialog box is displayed:



To add non-library components, click the button next to the **Component name** drop-down box and then select **Add a component**:



Specify the reference phase and other phases in which the component may be present. You can change the component name and/or the pure component property data.

You can also copy an existing component. You will be prompted to enter a name for the new component.

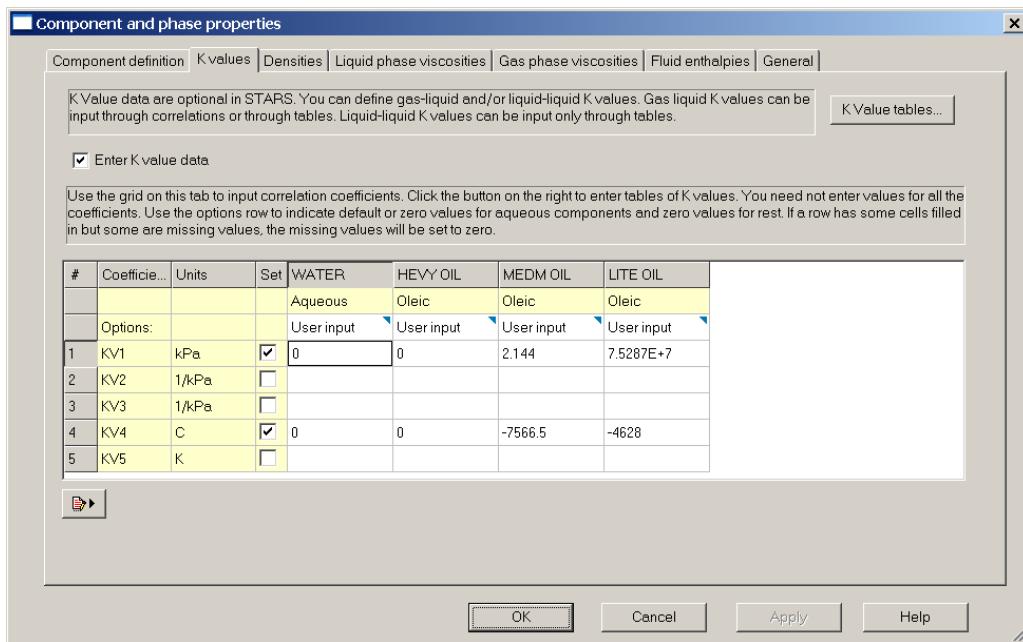
Entering K Value Data

K value data are optional. Gas-liquid K values can be input through correlation coefficients or through tables. Liquid-liquid K value data can be input through tables only.

For gas-liquid K values, if you enter both correlations and tables, STARS will ignore the correlations.

K value Correlations

You enter the K value correlations coefficients on the **K values** tab of the **Component and phase properties** dialog box.

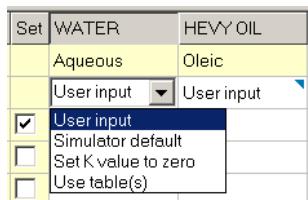


Check Boxes

The check box for a row indicates whether the keyword corresponding to the row will be output to the dataset. The check boxes are read-only, that is, you cannot check or cancel them. Builder sets each one of them depending on the data entered for a row.

Options Row

The cells in the **Options** row in each component column contain a drop-down box.



The following table gives an explanation of the options.

| Options | Comments |
|---|--|
| User Input | Data already entered or can be entered. Not to be used as a command. Set automatically by Builder. |
| Simulator default (for aqueous components only) | Value for this coefficient should be defaulted (command). |
| Set value to zero | Sets all coefficients except KV4 to zero for aqueous components. For oleic, sets the value to zero (command). |
| Use table(s) | Builder has detected tabular data for this component. Not to be used as a command. Set automatically by Builder. |

You do not need to enter all of the coefficients. If a row is completely blank, the keyword corresponding to that row will not be output to the dataset (the check box in the **Set** column for that row will be cancelled).

If a row has a value for one component, the rest of the cells in that row will be filled in with zeroes. This happens when you read in a dataset or click on **Apply** after making changes to the data.

If you select the **Use table(s)** option for a component, that column will be disabled.

If you wish to remove a keyword from a dataset:

- Set the option for all aqueous components to **Simulator default** and
- Set the option for all oleic components to **Set K value to zero**.

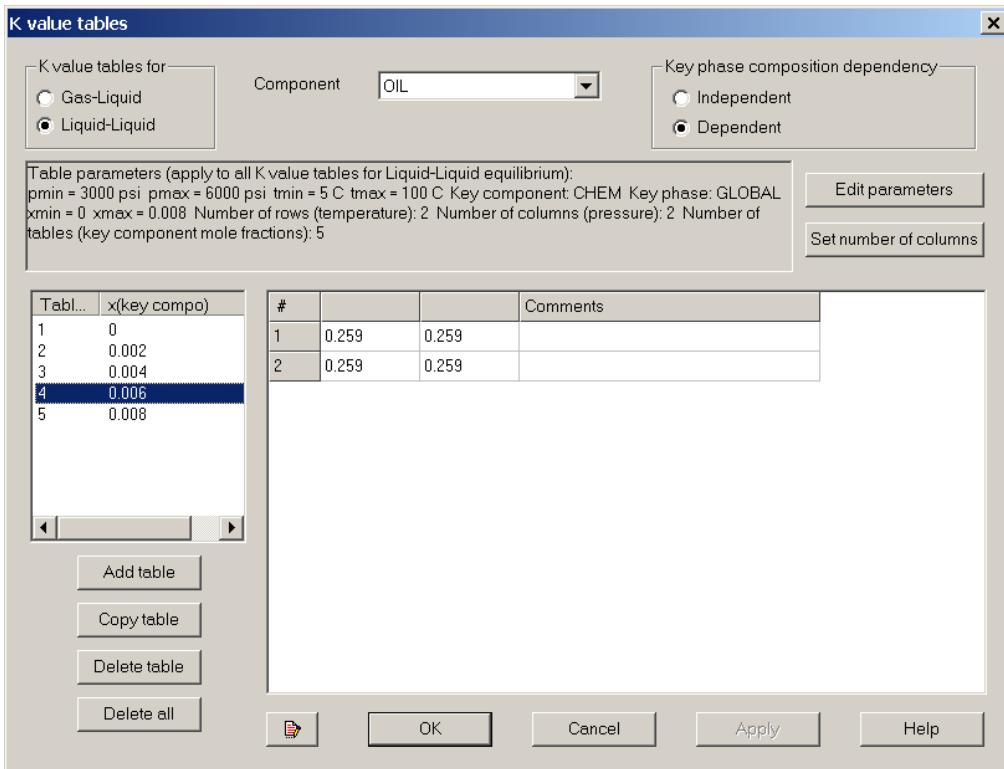
Comments for the correlation coefficients:

Use the  button to edit or enter comments.

K Value Tables

To launch the **K value tables** dialog box, click the **K Value tables** button on the **K values** tab of the **Component and phase properties** dialog box.

If you want to directly go to tables for a particular component, click in the column for that component in the K value correlations tab grid before clicking on the **K Value tables** button.



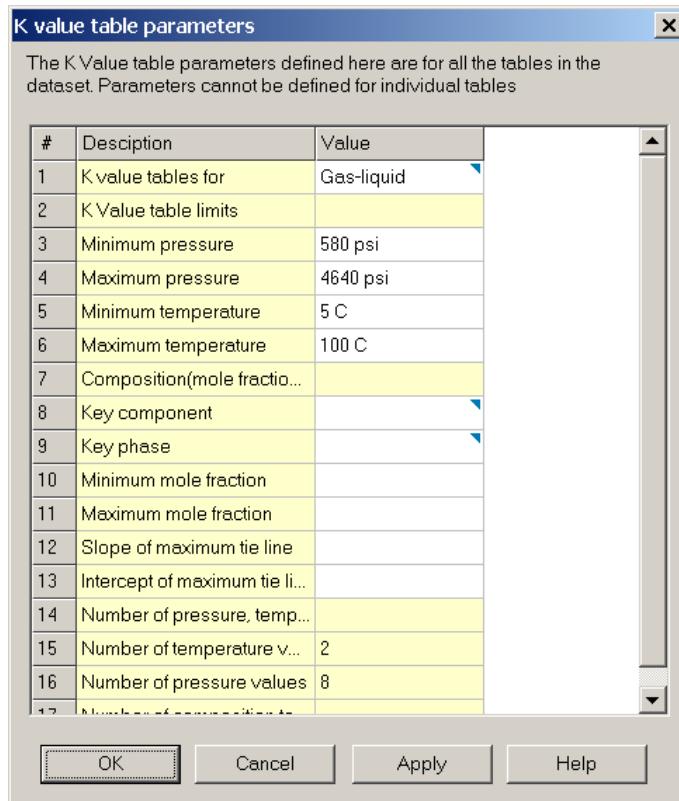
Use the **Component** drop-down box to navigate through components.

Use the selections in the **K value tables for** frame in the left top corner to change between gas-liquid and liquid-liquid type tables for a component.

The list on the left shows all the tables for a component for the selected table type (gas-liquid or liquid-liquid). If composition-dependent tables are defined, the list box will also display the key component concentration for the table in the **x(key comp)** column. Selecting a row in the list displays the table in the grid on the right.

The information panel above the list displays the table parameters: pressure and temperature minima and maxima, name of the key component, key component concentration minimum and maximum, number of temperature values (rows) in each table and the number of pressure values (columns) in each table. These parameters are “global” in the sense they apply to all tables (for all components) for a given type of table (gas-liquid or liquid-liquid).

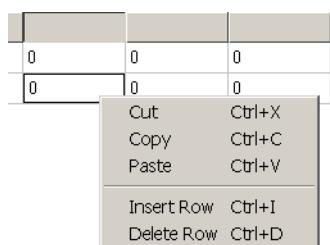
If no tables are defined and the table parameters are not defined, Builder will prompt you to enter values for pressure and temperature minima and maxima.



You can also define the parameters for composition-dependent tables in the **K value table parameters** dialog box.

If the table parameters are not defined or you are switching from composition-independent to composition-dependent tables for the first time, you can also edit the number of temperature (rows) and number of pressure values (columns) for all the tables.

You can also change the number of temperature values (rows) in a table using the context menu in the grid.



You can also change the number of pressure values (columns) by clicking the **Set number of columns** button.

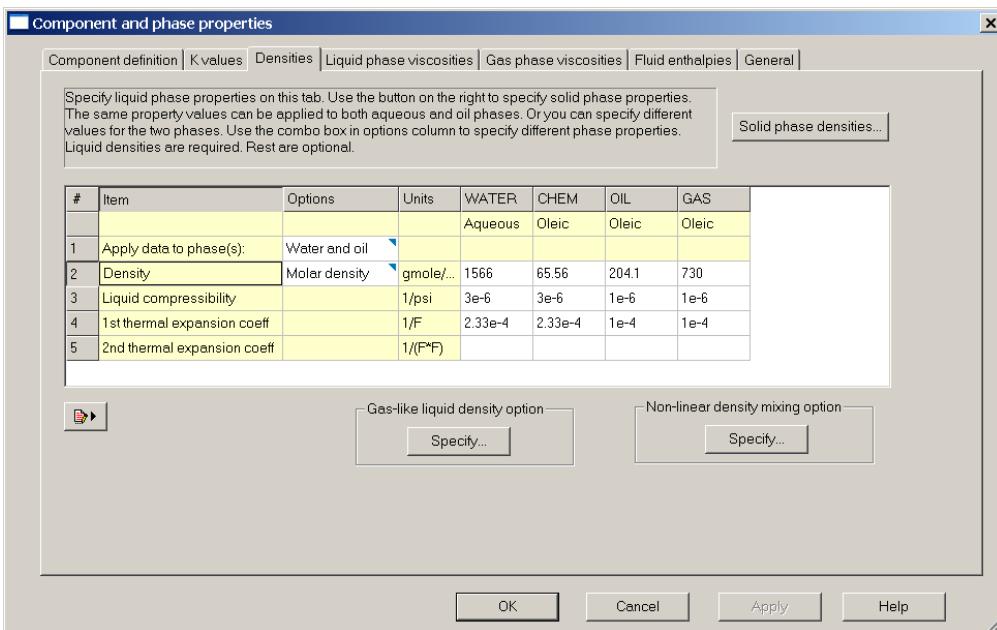


The selection in the **Key phase composition dependency** frame in the top right corner is automatically set by Builder. You can change the current selection. If the current selection is **Independent** and you click on **Dependent**, Builder will launch the **K value table parameters** dialog box if the parameters for composition-dependent tables are not already defined.

Use the button to add or edit comments for the current tables.

Entering Liquid Density Data

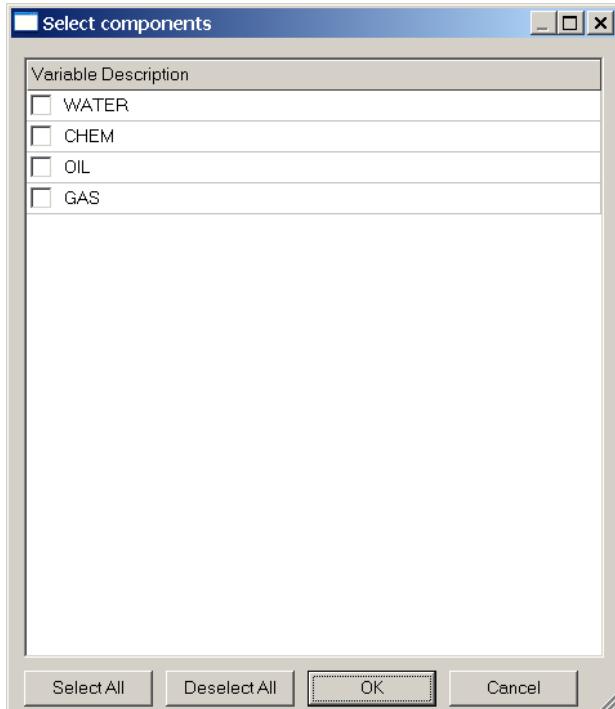
Enter the liquid density, compressibility and thermal expansion coefficients on the **Densities** tab on the **Component and phase properties** dialog box:



You can enter the data separately for the water and oil phases or you can apply the same data to both phases. Use the drop-down box in the **Apply data to phase(s)** cell of the **Options** column.

Use the drop-down box in the **Density** row of the **Options** column to enter molar density, mass density or molar volume.

Use the **Specify** button in the **Gas-like liquid density option** frame to input components for which this option (*GASSYLIQ) applies.



Liquid Density Nonlinear Mixing Option

Use the **Specify** button in the **Non-linear density mixing option** frame on the **Densities** tab on the **Component and phase properties** dialog box to input non-linear density mixing data.

Nonlinear mixing

Apply data to phase(s)

| # | Item | Options | Water | Bitumen | CO2 | Naphtha |
|----|-----------------------|---------|---------|---------|-------|---------|
| | | | Aqueous | Oleic | Oleic | Oleic |
| 1 | Key component min... | | | | 0.05 | 0.07 |
| 2 | Key component maxi... | | | | 0.25 | 0.37 |
| 3 | f(1) | | | | 0.05 | 0.07 |
| 4 | f(2) | | | | 0.07 | 0.10 |
| 5 | f(3) | | | | 0.09 | 0.13 |
| 6 | f(4) | | | | 0.11 | 0.16 |
| 7 | f(5) | | | | 0.13 | 0.19 |
| 8 | f(6) | | | | 0.15 | 0.22 |
| 9 | f(7) | | | | 0.17 | 0.25 |
| 10 | f(8) | | | | 0.19 | 0.28 |
| 11 | f(9) | | | | 0.21 | 0.31 |
| 12 | f(10) | | | | 0.23 | 0.34 |
| 13 | f(11) | | | | 0.25 | 0.37 |

Entering Solid Phase Density Data

Click the **Solid phase densities** button on the **Densities** tab of the **Component and phase properties** dialog box:

Solid phase densities

| # | Item | Units | COKE |
|---|----------------------|--------------------|------|
| 1 | | Solid | |
| 2 | Density | lb/ft ³ | 57.2 |
| 3 | Isothermal compr... | 1/psi | 0 |
| 4 | Isobaric thermal ... | 1/F | 0 |

Entering Liquid Phase Viscosity Data

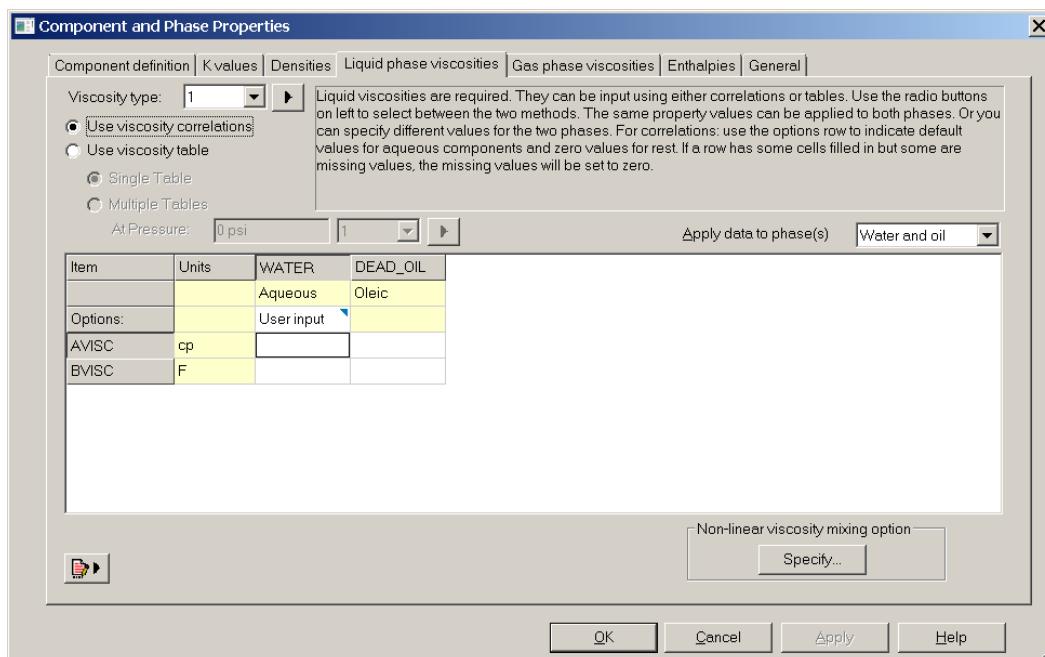
Enter the liquid phase viscosity data on the **Liquid phase viscosities** tab on the **Component and Phase Properties** dialog box.

Liquid viscosities data are required. You can input the data as viscosity correlations *OR* as viscosity-component tables. Select the type of input using **Use viscosity correlations** and **Use viscosity table**. If the table option is selected, you have the further choice of using either a **Single Table** or defining **Multiple Tables**. The **Multiple Tables** option will allow you to enter a different viscosity table for different pressure values. Consequently, the **At Pressure** value is required for the viscosity tables specified.

You can define one set of data for both the liquid phases (water and oil). You can also define viscosity separately for both the phases. Or you can define common data for both the phases but override for one or both liquid phases. Use the **Apply data to phase(s)** drop-down box to select the phase type.

The data you enter will be applied to the current viscosity-type displayed in the **Viscosity type** drop-down box.

Viscosity Correlations



Enter values for AVISC and/or BVISC. Missing values will be set to zero. However, AVISC cannot be zero for non-aqueous components.

Options Row

The cells in **Options** row in each *aqueous* component column contain a drop-down box. Non-aqueous components do not have default values for AVISC and BVISC.

| | WATER | HEVY OIL | LITE OIL |
|----------|------------|----------|----------|
| Aqueous | Oleic | Oleic | |
| Options: | User input | | |
| AVISC | 4.02e-4 | | |
| BVISC | 6121.6 | | |

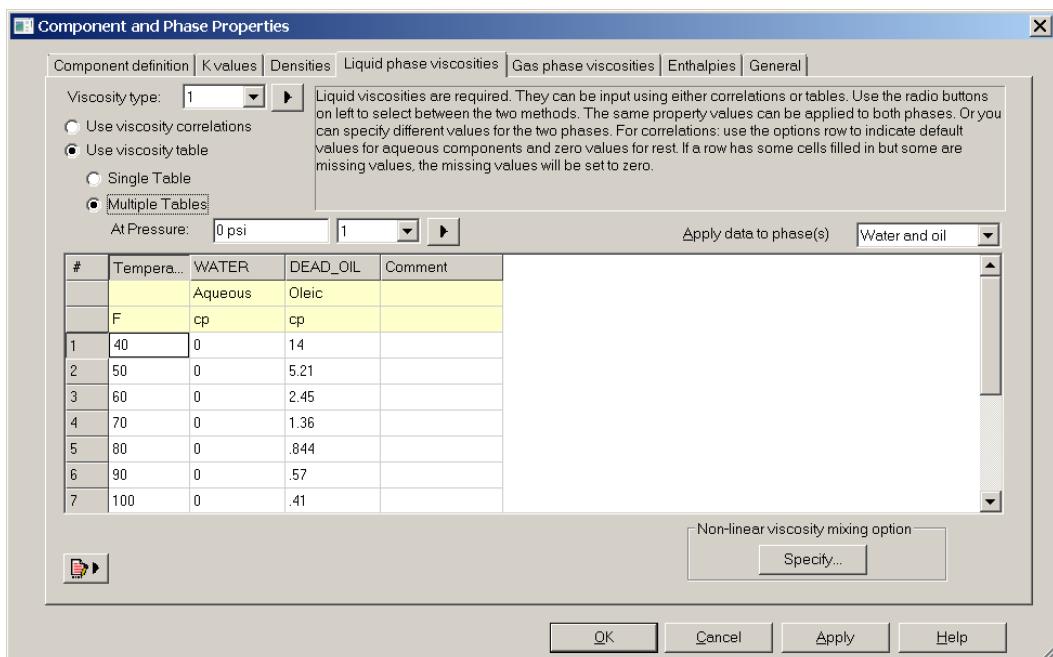
The following table gives an explanation of the options:

| Options | Comments |
|-------------------|---|
| User Input | Data already entered or can be entered. Not to be used as a command. Set automatically by Builder. |
| Simulator default | Value for this coefficient should be defaulted (command). Builder will output the values of 0.0 for both AVISC and BVISC. |

Viscosity Tables

The first column is the temperature column, followed by columns for all of the aqueous or oleic based components. You can increase the number of rows using the context menu of the grid, and selecting the **Insert Row** option. Cut, copy and paste functionality is also available.

To delete information in a table, select all of the rows that should be removed and press the **DELETE** key (or use the context menu and select the **Delete Row** option). If the **Multiple Tables** option is chosen, the tool menu button (with a solid right triangle shape) at the **At Pressure** row will allow tables to be created, copied or deleted. Different pressure values can be entered for each of these tables.



Liquid Viscosity Nonlinear Mixing Option

Click the **Specify** button in the **Non-linear density mixing option** frame on the **Liquid phase viscosities** tab on the **Component and phase properties** dialog box to input non-linear density mixing data.

Nonliner mixing

Apply data to phase(s) **Water and oil**

| # | Item | Options | WATER | POLYMER | NAOH | DEAD OIL | |
|----|-----------------------|---------|---------|---------|---------|----------|--|
| | | | Aqueous | Aqueous | Aqueous | Oleic | |
| 1 | Key component mini... | | | 0 | | | |
| 2 | Key component maxi... | | | 4E-6 | | | |
| 3 | f(1) | | | 0 | | | |
| 4 | f(2) | | | .075 | | | |
| 5 | f(3) | | | .16 | | | |
| 6 | f(4) | | | .25 | | | |
| 7 | f(5) | | | .35 | | | |
| 8 | f(6) | | | .45 | | | |
| 9 | f(7) | | | .56 | | | |
| 10 | f(8) | | | .67 | | | |
| 11 | f(9) | | | .78 | | | |
| 12 | f(10) | | | .89 | | | |
| 13 | f(11) | | | 1 | | | |

OK **Cancel** **Apply** **Help**

Entering Gas Phase Viscosity Data

Enter the gas phase viscosity data on the **Gas phase viscosities** tab on the **Component and phase properties** dialog box.

Component and phase properties

Component definition | K values | Densities | Liquid phase viscosities | **Gas phase viscosities** | Fluid enthalpies | General

Viscosity type: **1**

Gas viscosities are optional. If not input, or if both AVG and BVG are zero for a component, STARS uses internal default correlations end/or values. Use the options row to indicate default values for a component. If a row has some cells filled in but some are missing values, the missing values will be set to zero.

Enter gas phase viscosity data

| # | Item | Units | WATER | HEVY OIL | LITE OIL | INRT G... | OXYGEN |
|---|----------|------------|-----------|----------|------------|------------|-----------|
| | | | Aqueous | Oleic | Oleic | Ges | Gas |
| 1 | Options: | User input | Simulato. | User ... | User input | User input | |
| 1 | AVG | cP/F | 8.822e-6 | 0 | 2.166e-6 | 2.1267e-4 | 2.1960e-4 |
| 2 | BVG | | 1.116 | 0 | 0.943 | 0.702 | 0.721 |

Enable correction to gas phase viscosity for high gas density

OK **Cancel** **Apply** **Help**

Options Row

The cell in **Options** row in each component column contains a drop-down box.

| | |
|-------------------|-------------|
| WATER | HEVY OIL |
| Aqueous | Oleic |
| User input | Simulato... |
| User input | |
| Simulator default | |
| 1.116 | 0 |

The following table gives an explanation of the options.

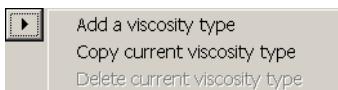
| Options | Comments |
|-------------------|---|
| User Input | Data already entered or can be entered. Not to be used as a command. Set automatically by Builder. |
| Simulator default | Value for this coefficient should be defaulted (command). Builder will output the values of 0.0 for both AVG and BVG. |

You do not need to enter all the coefficients. If a row has a value for one component, the rest of the cells in that row will be filled with zeroes. This happens when you read in a dataset or click **Apply** after making changes to the data.

If you enter gas viscosity data, both AVG and BVG coefficients must be input.

Working With Multiple Viscosity Property Types

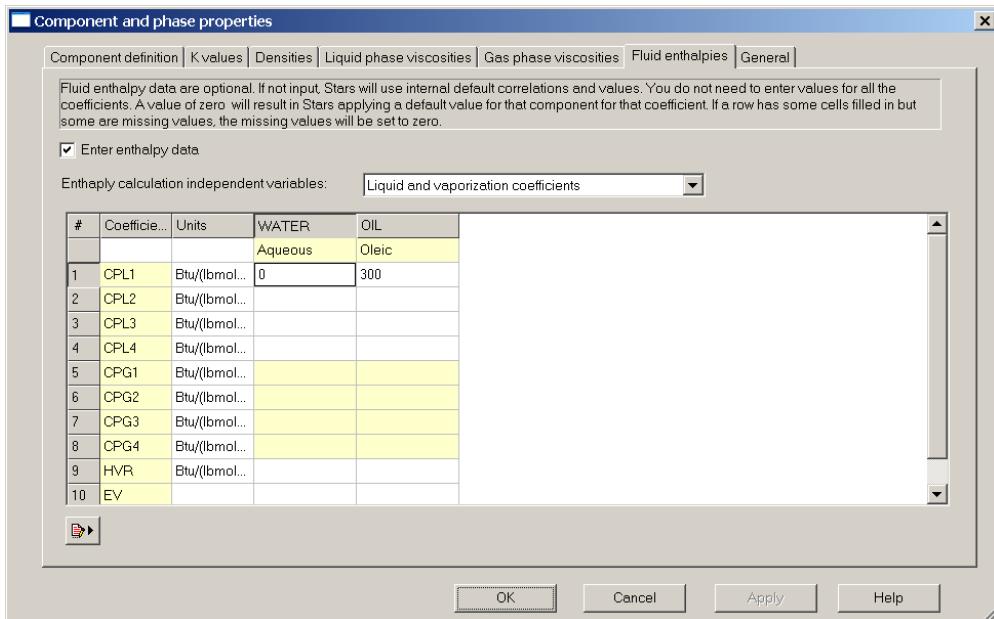
One viscosity type is always created and is persistent. To add, copy or delete viscosity types click the  button next to the **Viscosity type** drop-down box on the **Liquid phase viscosities** and the **Gas phase viscosities** tabs on the **Component and phase properties** dialog box.



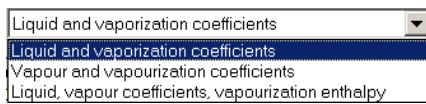
Enter the gas phase viscosity data on the **Fluid enthalpies** tab on the **Component and phase properties** dialog box.

Entering Fluid Enthalpy Data

Enter the gas phase viscosity data on the **Fluid enthalpies** tab on the **Component and phase properties** dialog box.



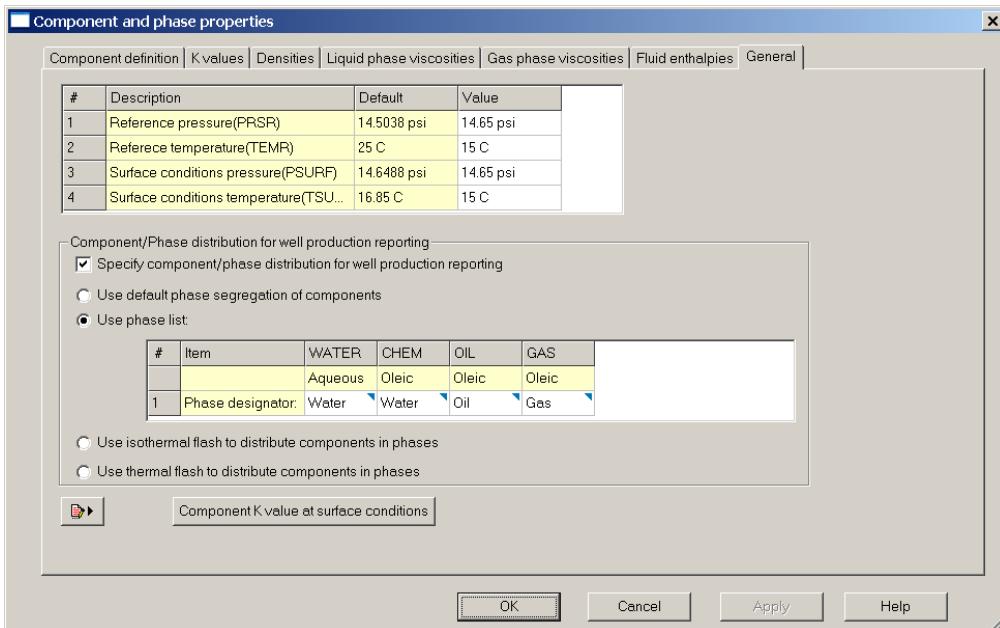
Select the type of data using the **Enthalpy calculation independent variables** drop-down box:



Depending on your selection, some of the rows may be disabled.

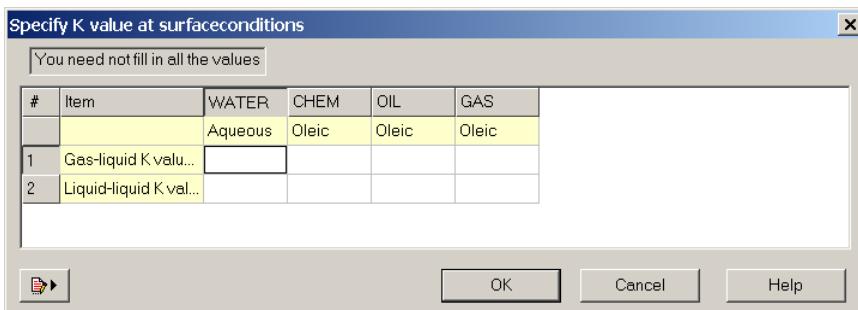
Entering Other Properties

Enter reference and surface pressure and temperature conditions on the **General** tab of the **Component and phase properties** dialog box.



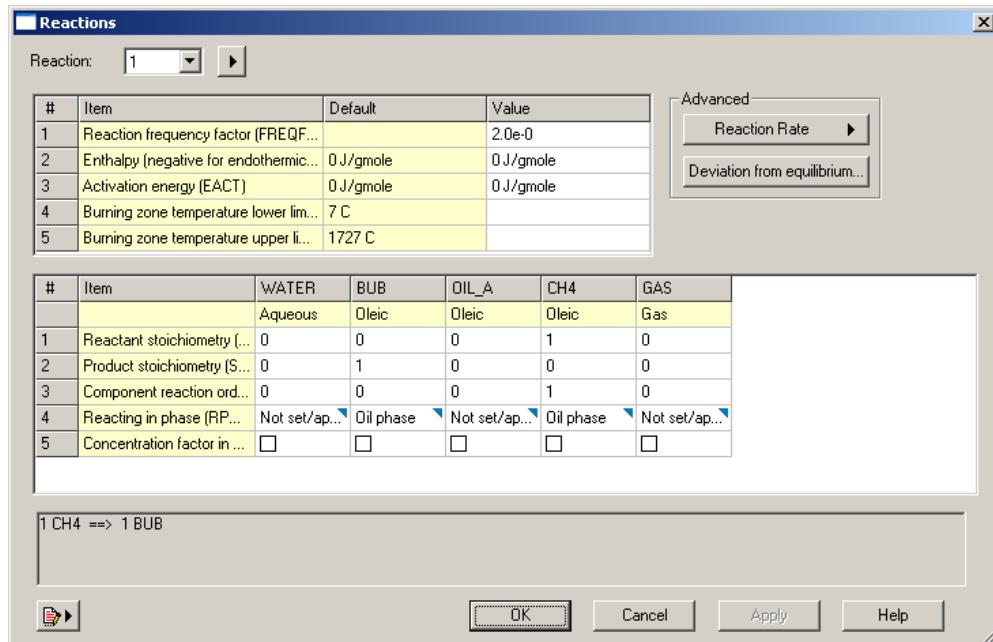
You can also select options for component/phase segregation for production reporting. If you select **Use phase list**, you must set the **Phase designator** for all the components. Use the drop-down box for this purpose.

To specify overriding surface K values for a component, click the **Component K value at surface conditions** button.

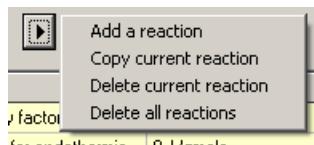


Working with Reactions

To add new or edit existing reactions, select **Components | Reactions** or double-click on a Reaction node in the **Components** tree view.

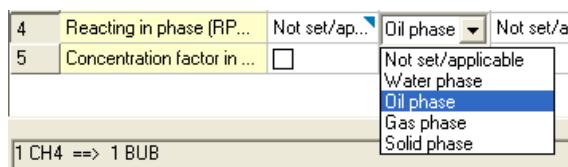


To edit a particular reaction, select it from the **Reactions** list box. To add, delete or copy reactions, use the button to the right of the list box:



Component-independent data are entered in the upper grid and component-dependent data in the lower grid.

To specify the phase in which a reactant reacts, use the drop-down list in the **Reacting in phase (RPHASE)** row:



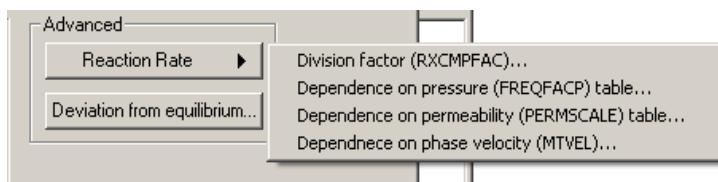
The stoichiometry of the reaction is displayed in the read-only text window at the bottom of the dialog box.

This window also displays errors in mass balance, if any.

Use the  button to edit or enter comments.

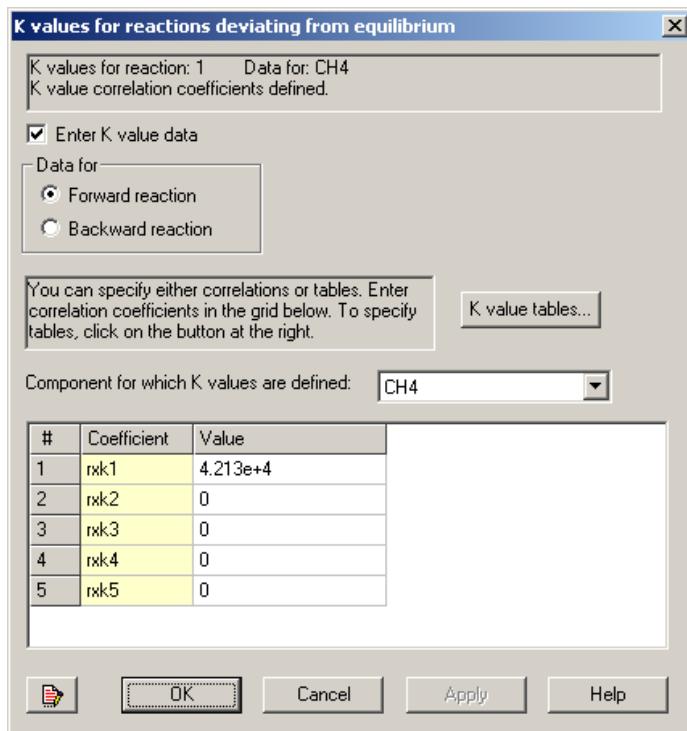
Advanced Options

Options for rate modifications and reactions deviating from equilibriums are available through the **Reaction Rate** and **Deviation from equilibrium** buttons in the **Advanced** area:

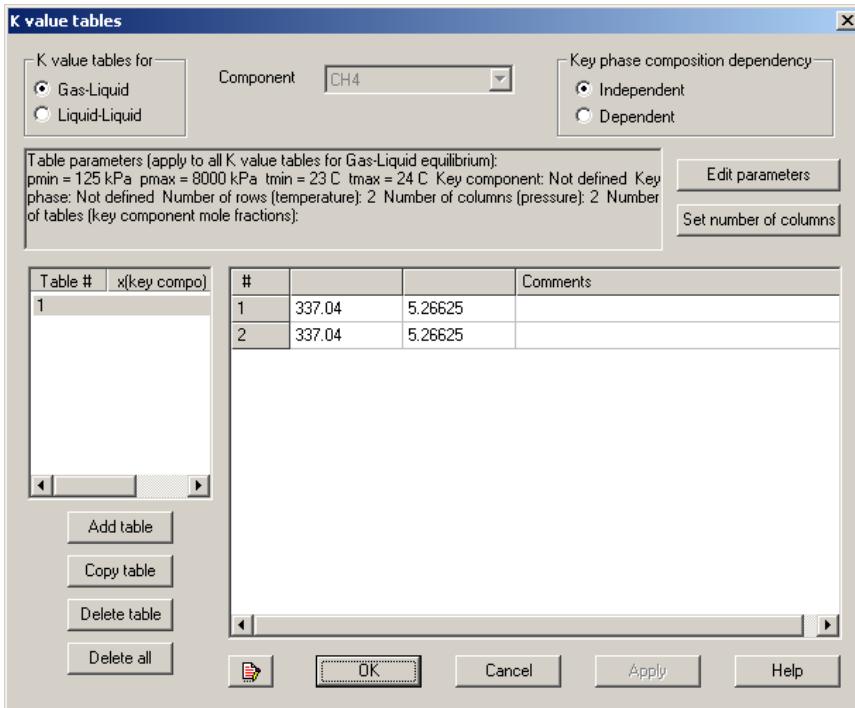


Reactions that deviate from equilibrium:

Modifying data in the form of K values is needed to use this option. You can either specify the correlation coefficients:



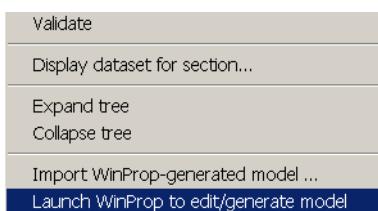
or as tables:



To launch the **K value tables** dialog box, click the **K value tables** button in the **K values for reactions deviating from equilibrium** dialog box. You can specify composition-independent or composition-dependent (KEYCOMP) tables. Select the component to which the K value tables apply in the **K values for reactions deviating from equilibrium** dialog box before launching the **K values table** dialog box. You cannot change the component in the **K value tables** dialog box.

Launching WinProp with Current Fluid Model Data

You can launch WinProp with the current fluid model data so that you work with the model in WinProp. Select the **Components | Launch WinProp to edit/generate model** menu. You can also select the same menu from the main tree view context menu:



After you have worked with the model in WinProp, you can import it back into Builder as explained earlier.

STARS Import Black Oil PVT Wizard

Introduction

The STARS Import Black Oil PVT wizard will create a completely new STARS fluid model using carefully calculated parameters that are matched to the black oil PVT data. Since STARS uses K value and component-based formulations for the fluid model, we recommended you use this wizard any time major parameters are to be changed in the fluid model. Changing parameters manually in the data set without the help of this wizard will have unpredictable results, and will usually result in a fluid model that no longer matches the black oil PVT data.

PVT Wizard vs. Winprop Fluid Model Generation

The main difference between the PVT wizard and Winprop is that with the PVT wizard, the STARS equations are used directly on the PVT data to be matched, which results in the following advantages:

- It is fast and easy to use.
- The wizard can handle conversions from IMEX (or ECL) to STARS and multiple PVT regions better than Winprop.
- The accuracy can be better than Winprop as the equations are solved directly (important when matching the original oil in place from IMEX).

The Winprop workflow is to match the PVT data with an equation of state (EOS), then match the STARS equations from the EOS (that is, export this data to STARS).

- Winprop's advantage is that very complicated PVT can be matched and then exported to STARS.
- One disadvantage is that this PVT matching can be quite involved.

Theoretical Background

Four matches are performed by the STARS Import Black Oil PVT wizard:

Density Match

The density match will affect the oil density (D_o), gas density in liquid (D_g), oil compressibility (C_o), gas compressibility in liquid (C_g), and the oil formation volume factors. If a match appears to be poor, the most common values to change are the oil compressibility and stock tank gas density. Both of these values can be changed if the **Launch the Black Oil PVT Graphical User Interface** button is clicked in Step 1 of the wizard.

The density match routine uses different points in the black oil PVT table for matching. The equations that it uses are:

$$V_{OilST} = \frac{e^{[C_{t1}(T-T_{emr}) + C_{t2}(T^2 - T_{emr}^2) - C_{Oil}(P-P_{sr})]}}{D_{OilSt}}$$

$$VoGasST = \frac{e^{[Ct1(T-T_{mr}) + Ct2(T^2 - T_{mr}^2) - CoGas(P - P_{rsr})]}}{\text{DenGasSt}}$$

$$\frac{1}{\text{DenOil}(P)} = [VoOilST \times X_2 + VoGasST \times X_3]$$

A three-point match uses the following equation to calculate compressibilities of Dead_Oil and Soln_Gas:

$$\begin{aligned} Co(\text{input value above bubble point}) \\ = Co \times \text{mole_fraction_oil} + Cg \times \text{mole_fraction_gas} \end{aligned}$$

The density equation is solved for Dead_Oil density and Soln_Gas density using three data points in the black oil PVT table, resulting in three equations. Note that Soln_Gas density and compressibility in the liquid phase do not resemble these same values when it is in the gas phase. Therefore, gas density and compressibility in the liquid phase will resemble liquid values, rather than normal gas values.

A two point match uses the input value of oil compressibility from the black oil PVT data, and solves for Dead_Oil and Soln_Gas density.

The automatic matching option will check all match types and all points in the PVT table to obtain the match with the minimum least squares error. The automatic matching routine will also vary the thermal expansion coefficient (Ct) to obtain the best fit of the data if the **Automatically Match Value** option is chosen in wizard Step 5.

You can change the emphasis of the matching routine by changing the weighting factors for oil density, oil compressibility, or oil formation volume factor.

GOR Match (or K value Match)

The K value match will affect the gas oil ratio of component 3 (Soln_Gas) in the oil phase. The K value match routine uses three different points in the black oil PVT table for matching. The three point match will solve three equations simultaneously to obtain the best fit of the black oil data.

Liquid Viscosity Match

The liquid viscosity match will use the dead oil viscosity at atmospheric pressure and reservoir temperature that was input on the black oil PVT tables, and calculate a solution gas viscosity in liquid that will match the live oil viscosity values in the PVT table. If the nonlinear viscosity option is used, the live oil viscosities will be matched exactly up to and including the bubble point.

Gas Match

The gas match includes the gas compressibility factor (z-factor) and the gas viscosity. The gas z-factor is matched by varying the critical pressures and temperatures through the range published for methane and propane, until a satisfactory match is obtained.

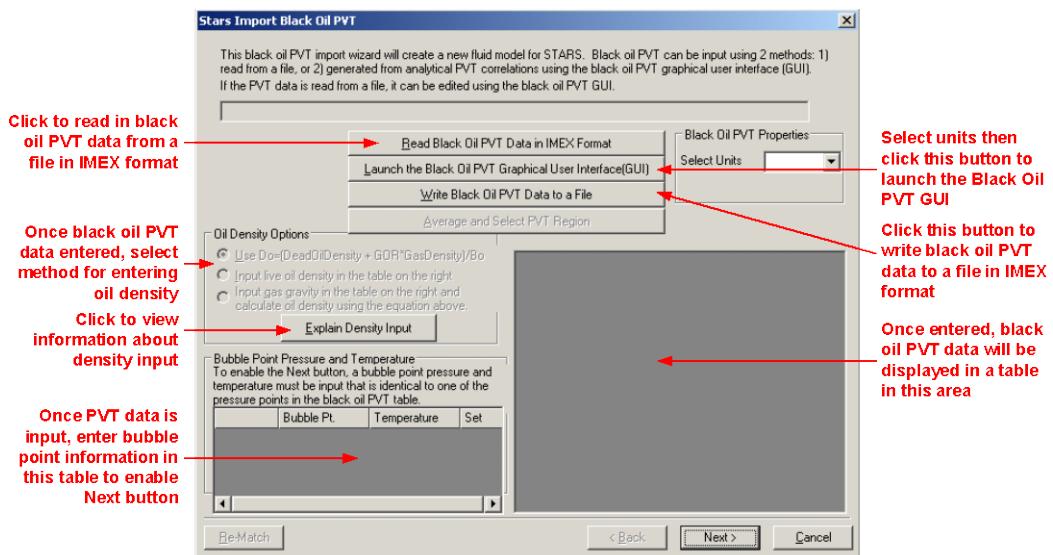
If the differences of z-factor between the PVT table input and the results from the cubic equation are not reasonable, one of the significant reasons may be that the assumption of pure component in gas phase is not the best representation of real gas. Multi-components are required to represent the real gas. The z-factors from the PVT table may be used to back-calculate the gas properties used in STARS.

For gas viscosity, if the **Internal correlation for gas viscosity** option is selected in wizard Step 5, no match of the black oil gas viscosity is performed. If the **Composition dependent gas viscosity** option is selected, the gas viscosity match is done by calculating a weighted averaging gas viscosity from the black oil data, then calculating the constant A (assuming the constant B=1.0) that will duplicate the average gas viscosity with the equation used in STARS. If **Use high gas density correction for gas viscosity** is selected, then the correlation described in the *STARS User's Guide* will be used to correct the gas viscosity at high density.

Using the STARS Import Black Oil PVT Wizard

To use the **Import Black Oil PVT** wizard:

1. Click Components | Import Blackoil PVT in the Builder menu bar. The **STARS Import Black Oil PVT** dialog box will be displayed:

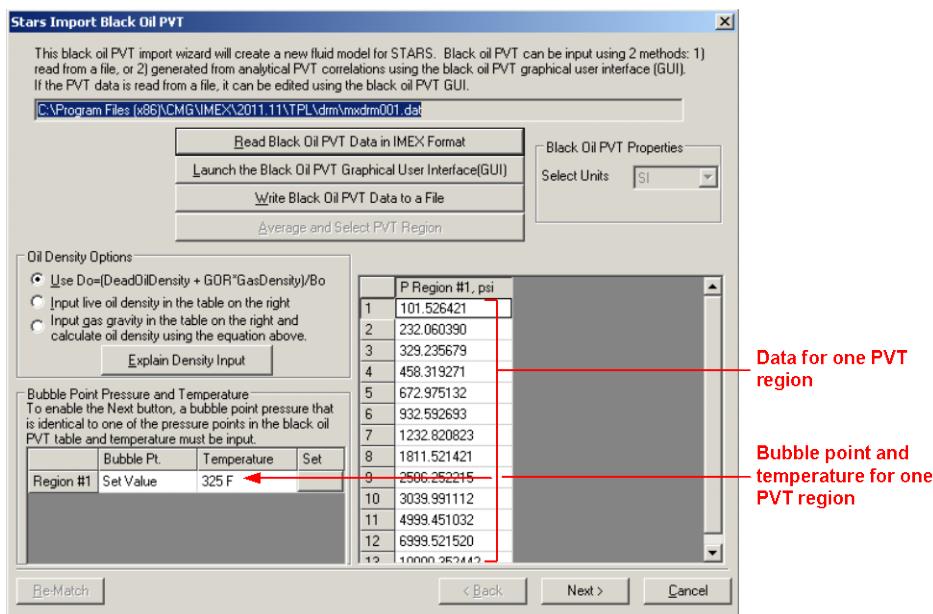


Through this dialog box, you can import or create black oil PVT data using one of the following methods:

- **Read from file:** If you click **Read Black Oil PVT Data in IMEX Format**, a dialog box will be displayed allowing you to select the file. Select the file and then click **Open**.

- **Import From GUI:** If you click **Launch the Black Oil PVT Graphical User Interface (GUI)**, you can create PVT data using analytical correlations, or create/edit the PVT from existing tables. Refer to [Working with Multiple PVT Regions](#), [Creating/Editing PVT Tables](#) and [Generating a PVT Table Using Correlations](#) for directions on how to do this.

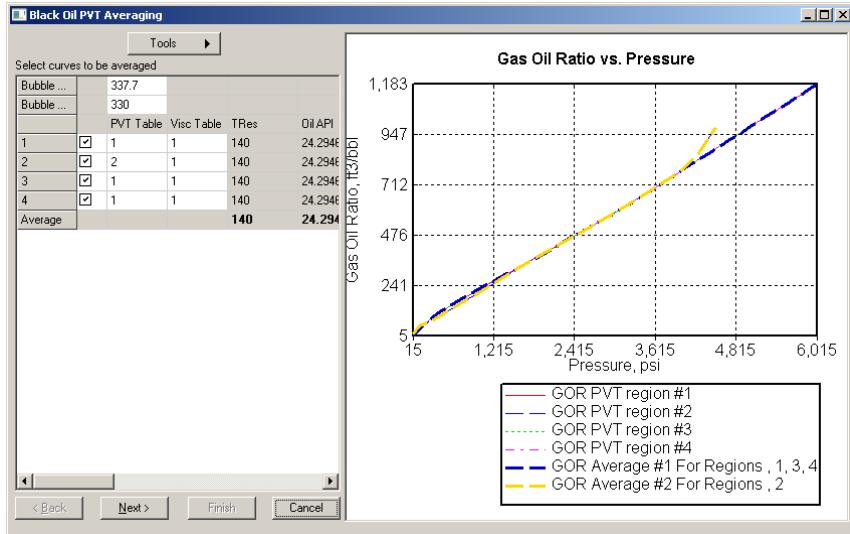
2. If you clicked **Read from file** and then selected a dataset with only one PVT region, then the **STARS Import Black Oil PVT** dialog box will be populated with the PVT data for one region, as shown below:



Proceed to step 5 of this procedure.

3. If you clicked **Read from file** and then selected an IMEX dataset with multiple PVT regions, the **Black Oil PVT Averaging** dialog box will be displayed:

Note: The **STARS Import Black Oil PVT** dialog box will also appear if you are converting an IMEX dataset to a STARS dataset through **File | Convert Simulator Type for Dataset | To STARS**. If the IMEX dataset contains multiple PVT regions, this dialog box will appear but you will need to click **Average and Select PVT Region** to average the appropriate regions.



As shown in the above example, bubble point entry is provided for each PVT table. In the above example, there is a PVT table that averages PVT regions 1, 3 and 4, and one that has only PVT region 2. If we had defined a third table, a third bubble point entry would be provided, as shown in the following example:

Third bubble point cell displayed because three PVT tables are defined →

| Select curves to be averaged | | | | | |
|------------------------------|-------------------------------------|------------|-------|-----------|------------|
| Bubble ... | 422.125 | Bubble ... | 337.7 | Set Value | |
| Bubble Pt 3, psi | | | | | |
| | | | | PVT Table | Visc Table |
| 1 | <input checked="" type="checkbox"/> | 1 | 1 | 140 | 24.294 |
| 2 | <input checked="" type="checkbox"/> | 2 | 1 | 140 | 24.294 |
| 3 | <input checked="" type="checkbox"/> | 3 | 1 | 140 | 24.294 |
| 4 | <input checked="" type="checkbox"/> | 1 | 1 | 140 | 24.294 |
| Average | | | | 140 | 24.294 |

Click the **Tools ▶** button and then select one of **Plot Using a Fixed Bubble Point** or **Plot Using a Variable Bubble**. **Plot Using a Fixed Bubble Point** only works when a valid bubble point has been entered. When this occurs, the plot will change and you will see the slope changing at the bubble point. This is done to help users visualize the under-saturated behavior (that is, the behavior above the bubble point) and how this region will be averaged.

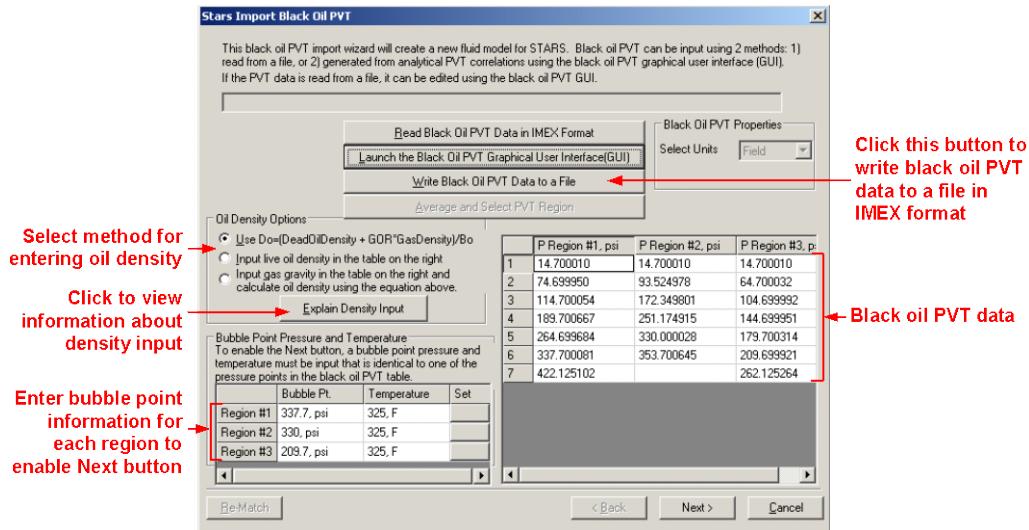
Note: If there is only one PVT region in the “averaged” region, you can enter any bubble point pressure. The wizard will populate the cell with the nearest available bubble point pressure in the imported or generated IMEX PVT table.

Click **Next** as necessary to review the curves. The **Finish** button will be enabled once:

- Averaging regions for the PVT Table and Visc Table are identical, or
- There is one averaged PVT region with multiple averaged Visc regions.

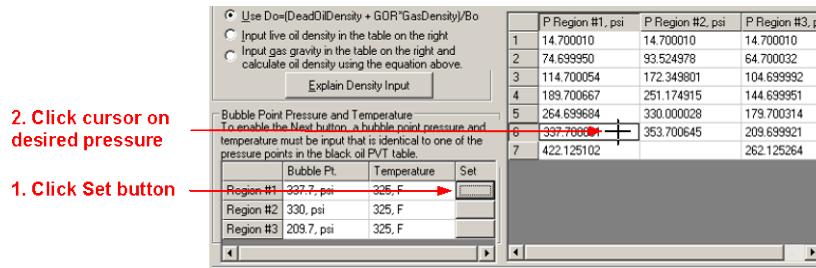
Click the **Finish** button. The number of regions specified and the averaged PVT data will be entered in the tables in the **STARS Import Black Oil PVT** dialog box.

- Once you have entered the data for each of the PVT regions, the **STARS Import Black Oil PVT** dialog box will be similar to the following:

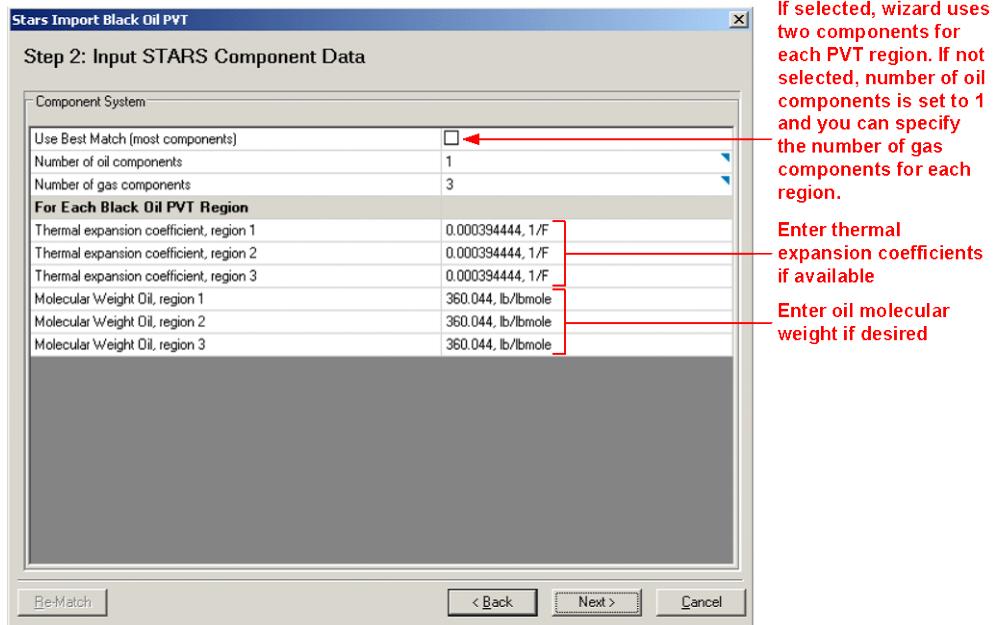


Note: After going through the **Black Oil PVT Averaging** dialog box, PVT regions cannot be added, copied or deleted through **Launch the Black Oil PVT Graphical User Interface (GUI)**.

- Enter the bubble point data for each PVT region shown. This data must be entered before the **Next** button will be enabled. To enter a bubble point in the **Bubble Point Pressure and Temperature** table, click the **Set** button for the region then use the cross-hair cursor to click the pressure in the table, as shown below:



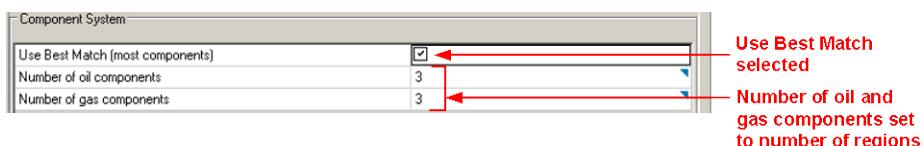
6. Click **Next**. The Step 2: Input STARS Component Data dialog box will be displayed:



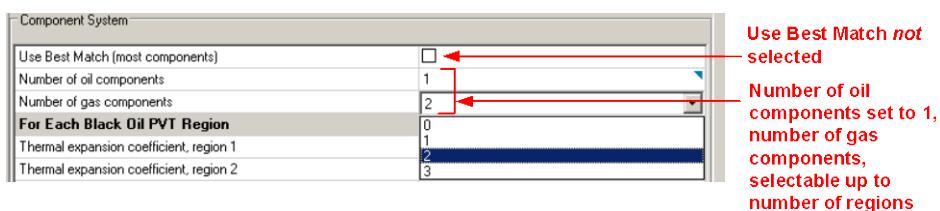
The main purpose of this step is to define parameters specific to STARS, but which are not normally used in a black-oil simulation.

The primary objective is to minimize the number of components, thereby reducing the simulation runtime, while at the same time maintaining modelling accuracy.

- If you select **Use Best Match**, the wizard will use two components (Oil and Soln_Gas) for each PVT region, for example, six components for three PVT regions, as shown below:



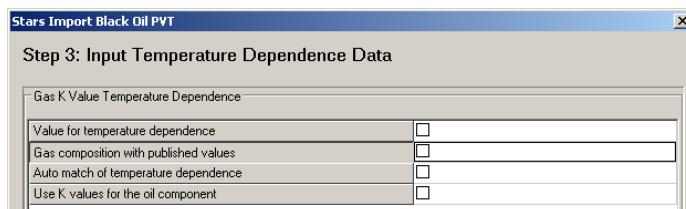
- If you do not select **Use Best Match**, the number of oil components is set to 1, and the number of gas components can be set to 1, 2...n, where n is the number of regions.



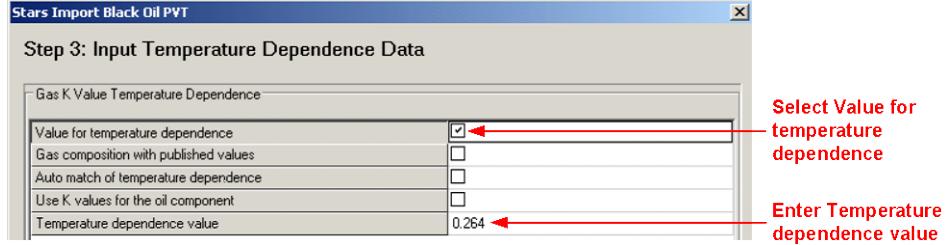
Example 1: If there are three PVT regions and you select one oil component plus one gas component for each PVT region, this will result in 1 oil + 3 gas = 4 components. This method matches each PVT region with two separate components then averages the oil components into one component. Some adjustments during wizard Step 4 are required to better match the under-saturated properties.

Example 2: If there is one oil component and less than one gas component for each PVT region (for example, three PVT regions could consist of 1 oil + 2 gas = 3 components), this will result in the fewest number of components. The wizard begins by matching the lowest and highest GOR region with two components (Oil and Soln-Gas). The two oil components are then averaged to become the one oil component. For the other PVT regions, iterations are performed to determine the composition that produces the best match. Finally, viscosity matches are performed for each PVT region (each is assigned a VISCTABLE) using these optimal compositions.

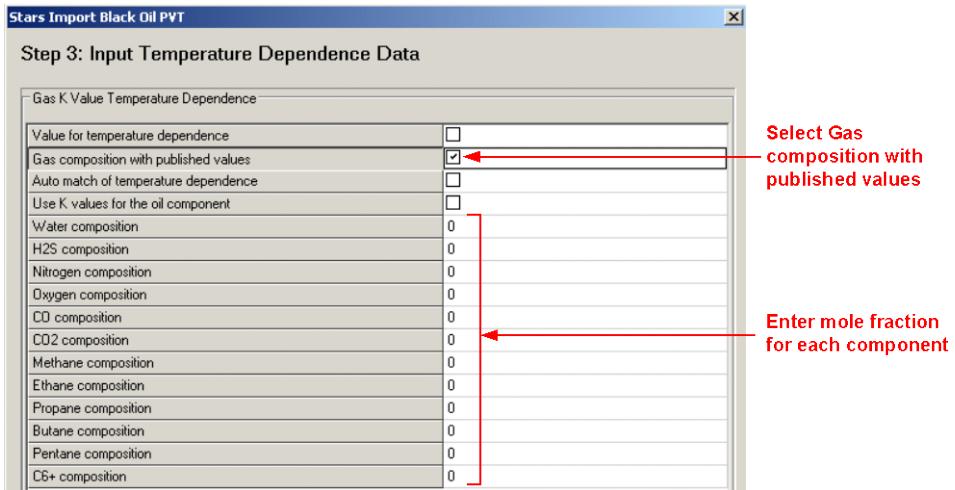
- c. For each PVT region, you can enter a **Thermal Expansion Coefficient** if it is available from laboratory measurements; otherwise, an automatic match of the thermal expansion coefficient will be carried out.
 - d. **Molecular Weight Oil** can be entered for each region, if available; however, changing this value will have minimal effect on the simulation. The only requirement is that all calculations of parameters for the STARS fluid model use the same value of molecular weight; therefore, changing this value manually in the data set without going through this wizard again will have an unpredictable effect and it is strongly not recommended.
7. Once you have configured the STARS component data, click **Next**. The **Step 3: Input Temperature Dependence Data** dialog box, through which you can configure the dependence of the gas K value on temperature, will be displayed:



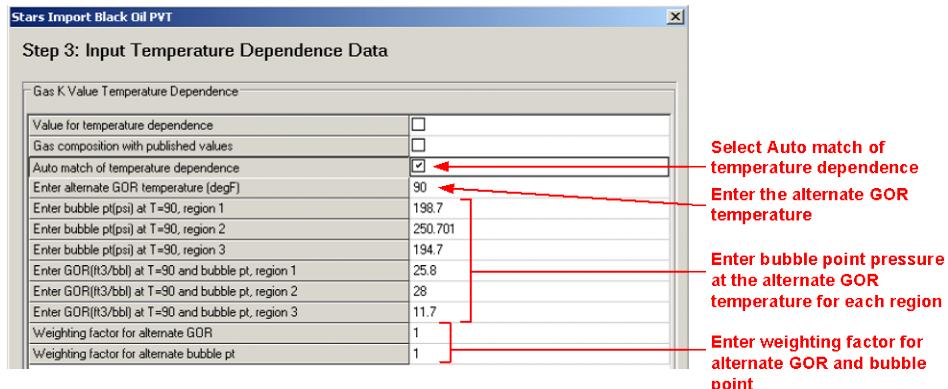
- a. If you select **Value for temperature dependence**, you can enter the temperature dependence value, as follows:



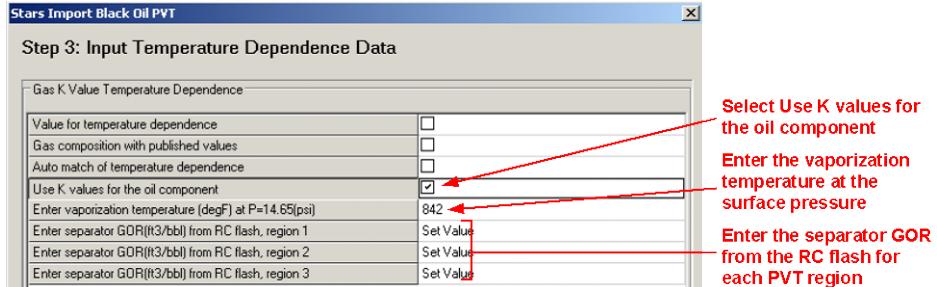
- b. If you select **Gas composition with published values**, you can enter the mole fraction of each component, as follows:



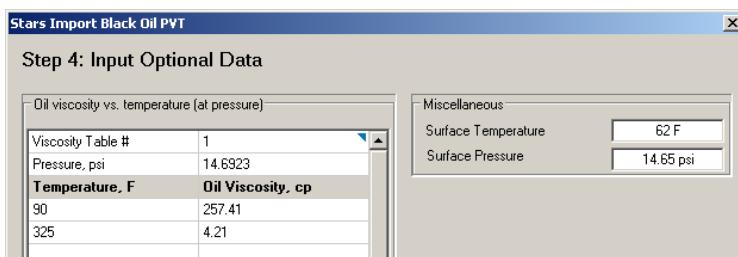
- c. If you select **Auto match of temperature dependence**, then iterations for matching the lower temperature bubble point and GOR will be performed:



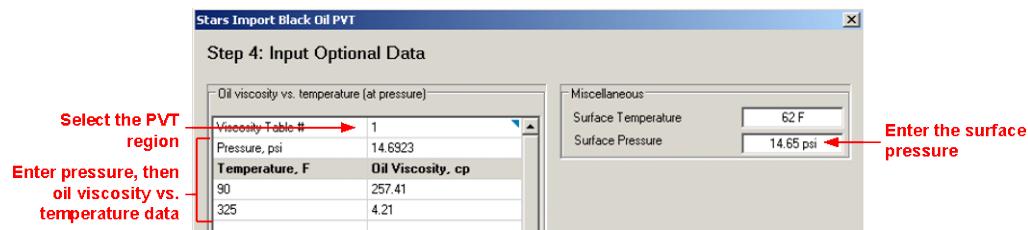
- d. If you select **Use K values for the oil component** then a match of the vaporization temperature will be performed. If a separator GOR is entered, then both vaporization temperature and separator will be matched:



8. Once you have selected and configured the gas K value temperature dependence, click **Next**. The **Step 4: Input Optional Data** dialog box will be displayed:

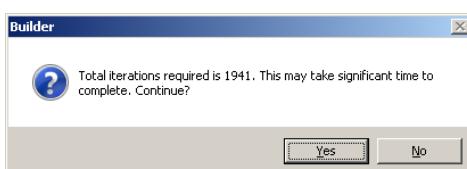


Enter the **Surface Temperature** and **Surface Pressure**, and then input a table of dead oil viscosities versus temperature for each region at the specified pressure if they are available:



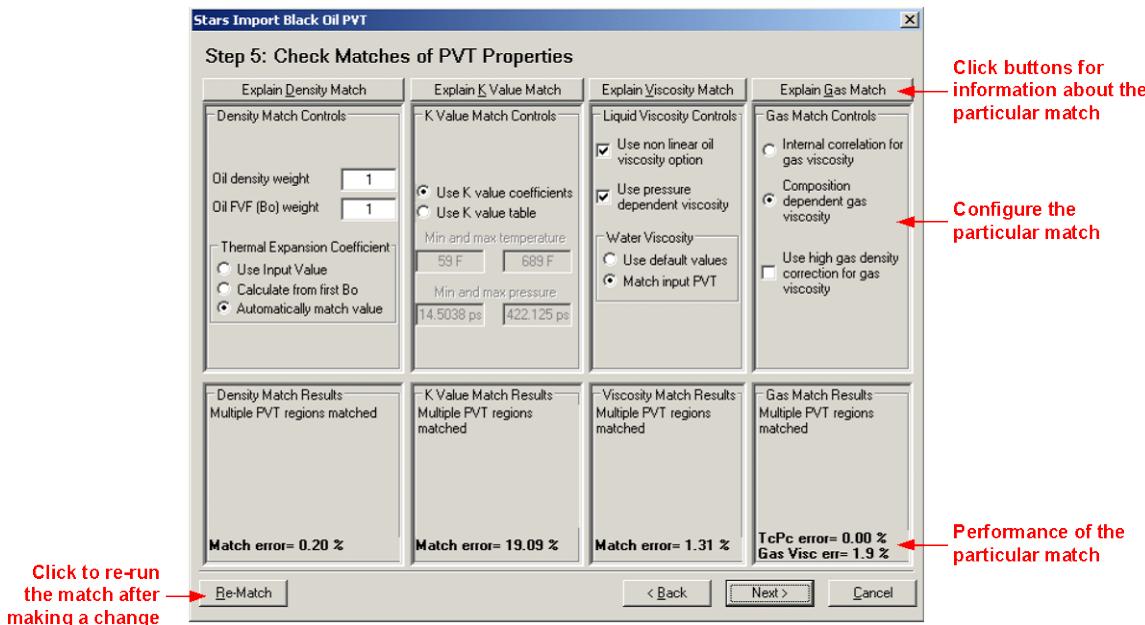
If these values are not available, an internal correlation will be used to generate the temperature effect on viscosity.

9. Once you have filled in the desired parameters in the **Step 4: Input Optional Data** dialog box click **Next**. The total iterations required will depend on the number of regions and components. If the anticipated total number of iterations exceeds 500, you will be asked if you want to continue:

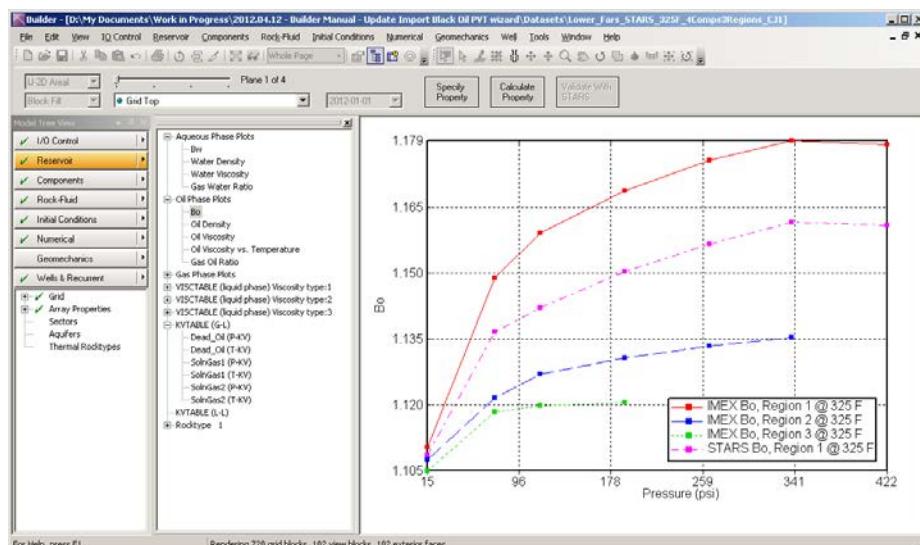


If you click **No**, you will be able to click **Back** to return to earlier wizard steps to adjust the match parameters. If you click **Yes**, the match will proceed, with progress indicated at the bottom of the screen.

Once the match is complete, the **Step 5: Check Matches of PVT Properties** dialog box will be displayed, as well as graphs to show how good the match is:



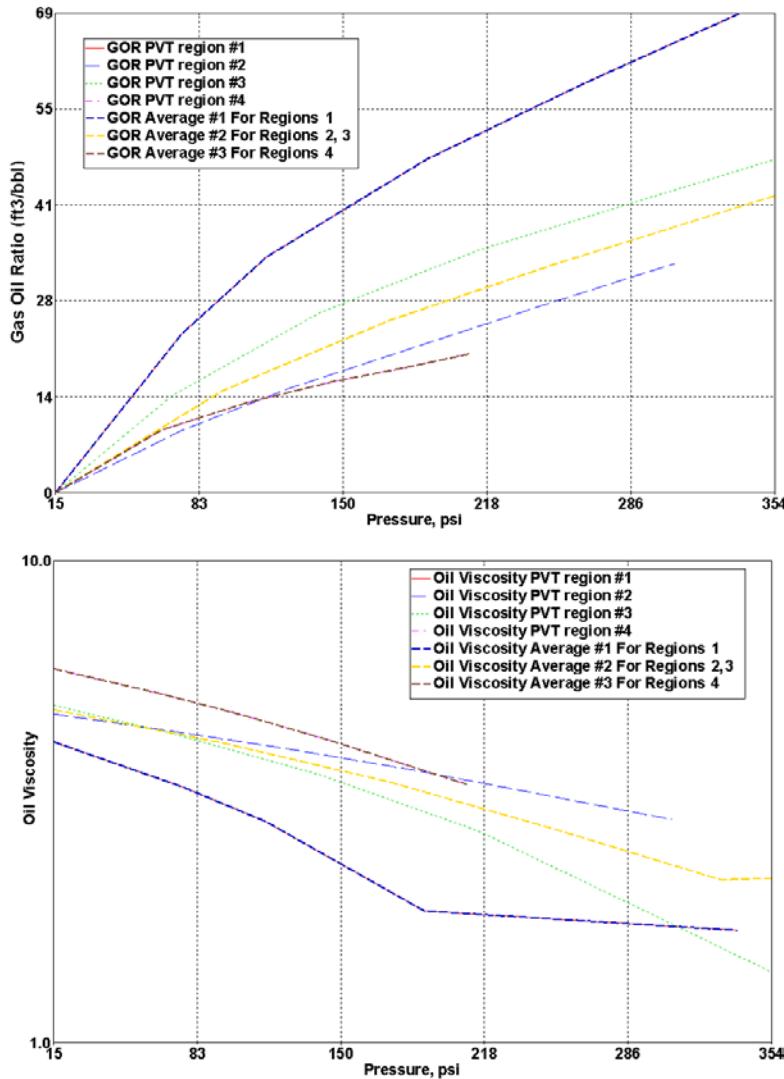
A list of available graphs will be displayed in the tree view, for example:



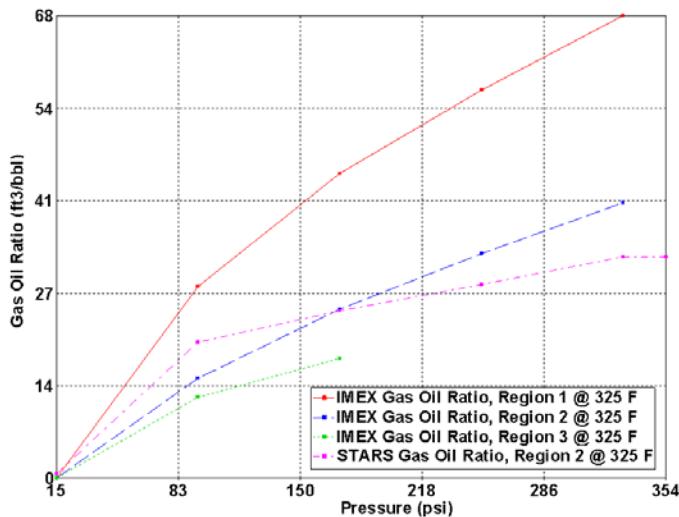
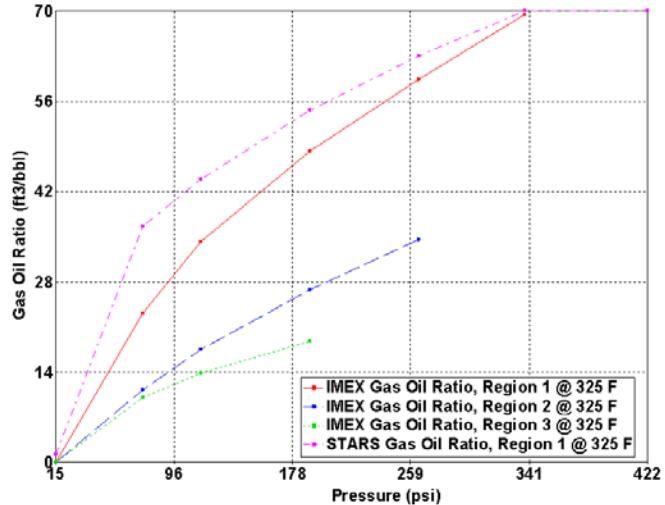
The purpose of wizard Step 5 is to refine the match of the black oil PVT data. The water, oil, and gas matches are plotted in the main Builder view, as shown above,

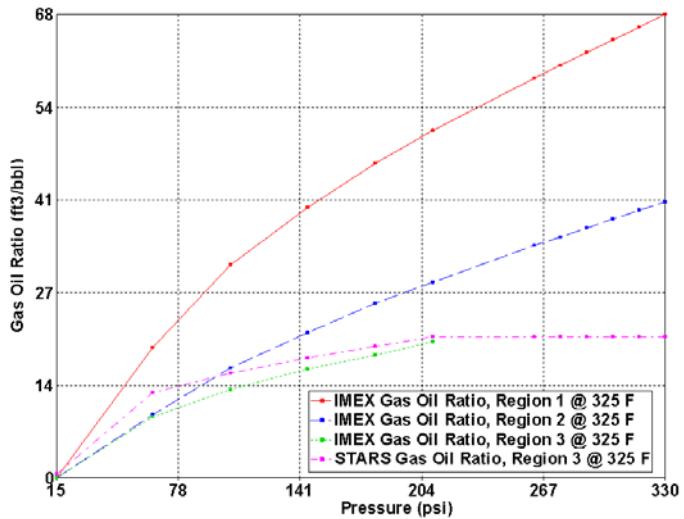
and you can select from the tree view to view the matches. Each time you click the **Re-Match** button in the **Step 5: Check Matches of PVT Properties** dialog box, the match errors and graphs will be updated based on the changes you make.

The graphs that are important will depend on your reservoir and the process you are simulating. The number of curves will depend on the number of PVT regions. To illustrate, consider the following graphs of GOR and Oil Viscosity, which show the averaging of PVT regions 2 and 3:

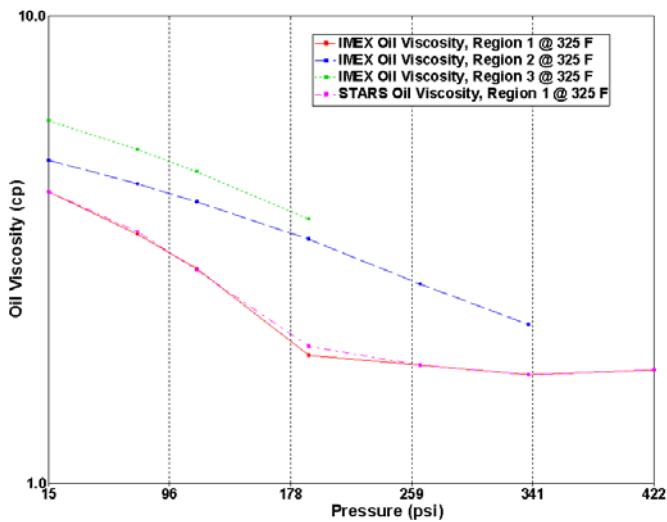


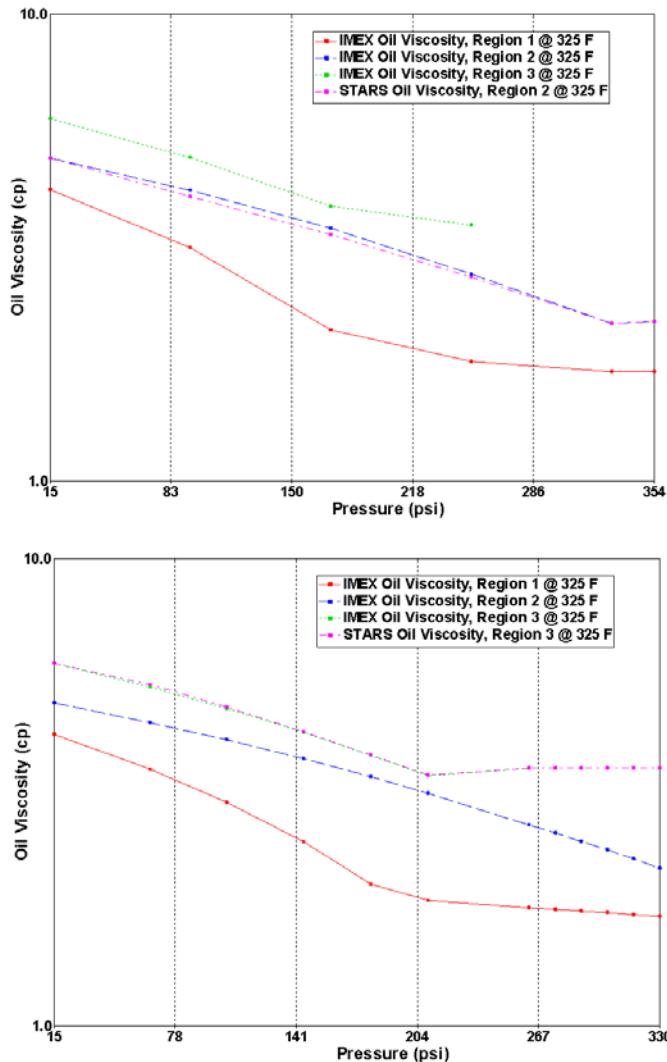
The following graphs of the Gas/Oil ratio show a good match with four components, including water, at 325°F:





The following graphs show an excellent match between viscosity and pressure at 325°F for three PVT regions:

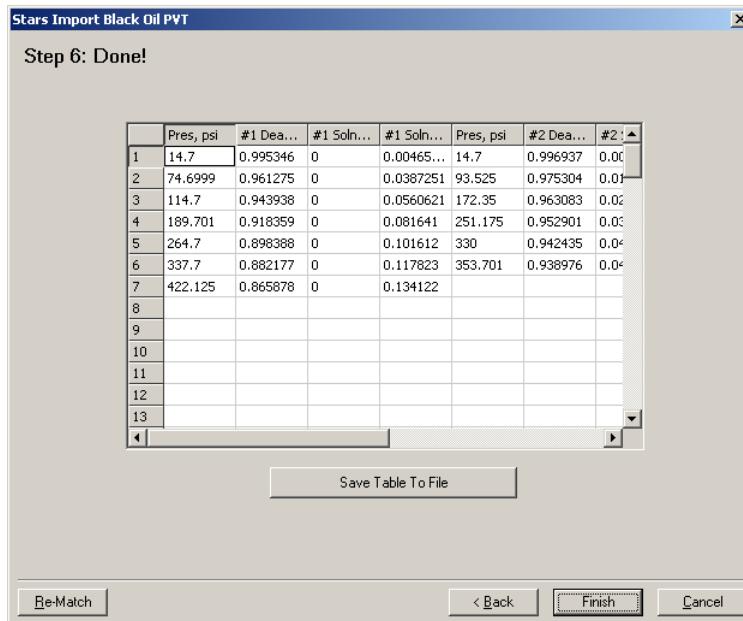




If the matches are not satisfactory, you can try the following:

- Change the settings in the **Step 5: Check Matches of PVT Properties** dialog box and then click **Re-Match**.
- In the **Step 5: Check Matches of PVT Properties** dialog box, click **Back** to return to the **Step 2: Input STARS Component Data** dialog box and try one of the following:
 - Select and configure **Auto match of temperature dependence** if you had not done this on the earlier match.
 - Change the number of components or select **Use Best Match**.

10. Click **Next**. The **Step 6: Done** dialog box will be displayed:



The main purpose of wizard Step 6 is to allow advanced users to look up mole fractions for different bubble points, which may be useful if you want to change the bubble point or change the composition in other parts of the reservoir.

Note: If you advanced to wizard Step 6 before you had clicked **Re-Match**, you can access it in Step 6.

Click **Save Table To File** to save the information in the table to a text file.

11. Click **Finish** to copy the new fluid model data into the data set. At this stage, the calculated mole fractions for the new fluid model will be copied into the data set in different ways, depending on the data supplied:

- If there is only one PVT region, or if the grid in the IMEX data set is different from the grid in the STARS data set, or if the grids are the same but there is no information in the IMEX data set on the PVT region locations (keyword *PTYPE), then the STARS initialization keyword MFRAC_OIL is set constant for all grid blocks, and is calculated from the composition at the input constant bubble point.
- If the IMEX and STARS grid sizes are identical and bubble point or GOR arrays were present in the IMEX data sets, then the STARS MFRAC_OIL values are calculated from the input black oil bubble point array (*PB or *PBT) or the GOR vs. depth array (*RST). If multiple PVT regions are input and the PVT region location (*PTYPE) data is missing, the MFRAC_OIL values will be calculated from the first PVT region data.

Displaying Component Properties Partial Dataset

You can view the Component Properties section data in keyword format by selecting the **Display dataset for section** menu from the main tree view context menu.

Data Validation

Builder automatically performs validation of the data when the following occur:

- You read in a dataset
- You add or modify data in the fluid model

You can also force validation by selecting the **Validate** menu from the main tree view context menu.

Rock Fluid - General

Overview

The Rock-Fluid section is used primarily to specify relative permeability and information for various rock types in the reservoir. IMEX, GEM, and STARS include the ability to define multiple rock types. Relative permeabilities of each rock type are entered using tables. Builder provides dialog boxes for the entry of relative permeability tables and related options.

Rock Fluid Options

Rock Fluid Options: IMEX

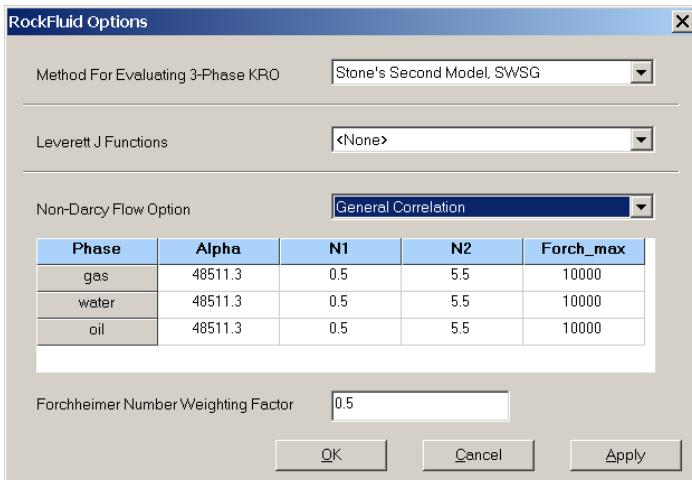
Builder provides a **Rock Fluid Options** dialog box through which you can select features available in IMEX that apply globally to all Rock Types. Through this dialog box, you can set the following:

- Method for evaluating 3-Phase KRO (KROIL)
- Leverett J functions option (JFUNC)
- Non Darcy flow option (NONDARCY)
- Trapped oil hysteresis option

There are two ways to access the **Rock Fluid Options** dialog box:

- Select the menu item **Rock-Fluid | Rock Fluid Options**.
- In the tree view, under **Rock-Fluid**, double-click **Rock Fluid Options**

The **Rock Fluid Options** dialog box is shown below:



For the Non-Darcy Gas Flow Option, there are a few different correlations available for selection. They include **Geertsma's Correlation**, **Frederick and Graves First Correlation**, **Frederick and Graves Second Correlation**, and a **General Correlation**. The **General Correlation** allows you to define how non-Darcy flow is a function of saturation, porosity and permeability for each phase.

When using the Geertsma, or Frederick and Graves Correlation options (NONDARCY GEERTSMA, FG1 or FG2), the non-Darcy flow coefficient (β) for all phases except gas is assumed to be zero. For gas, the parameters for α_g , $N1_g$ and $N2_g$ are displayed but cannot be modified.

When using the General Correlation option (NONDARCY GENERAL) you can enter a table of α , $N1$, $N2$ and $Forch_{max}$ for each phase in the reservoir and modify the Forchheimer Number Weighting Factor.

It is required that a Non-Darcy Gas Flow Option be set before creating hydraulically fractured wells (see [Hydraulically Fractured Wells \(IMEX & GEM\)](#)).

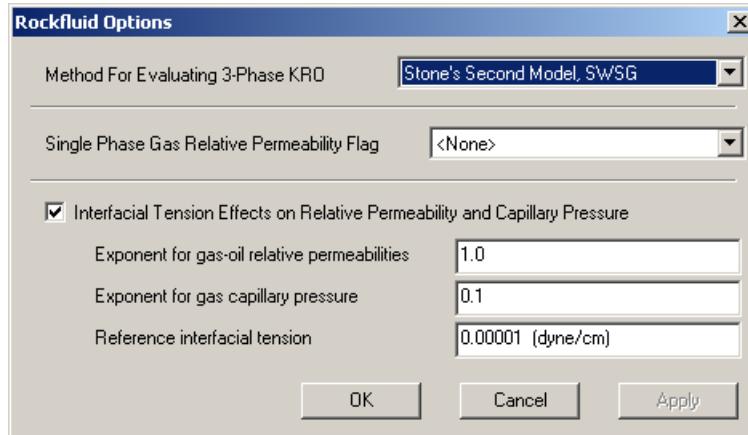
Rock Fluid Options: GEM

Builder provides **Rock Fluid Options** dialog box for selecting features available in GEM that apply globally to all Rock Types. This dialog box includes simple controls to set:

- Method for evaluating 3-Phase KRO (KROIL)
- Single-phase gas relative permeability calculation flag (KRGAS)
- Interfacial tension effects (SIGMA)

There are two ways to access the **Rock Fluid Options** dialog box:

- Select menu item **Rock-Fluid | Rock Fluid Options**
- In the tree view, under **Rock-Fluid**, double-click **Rock Fluid Options**



Rock Fluid Options: STARS

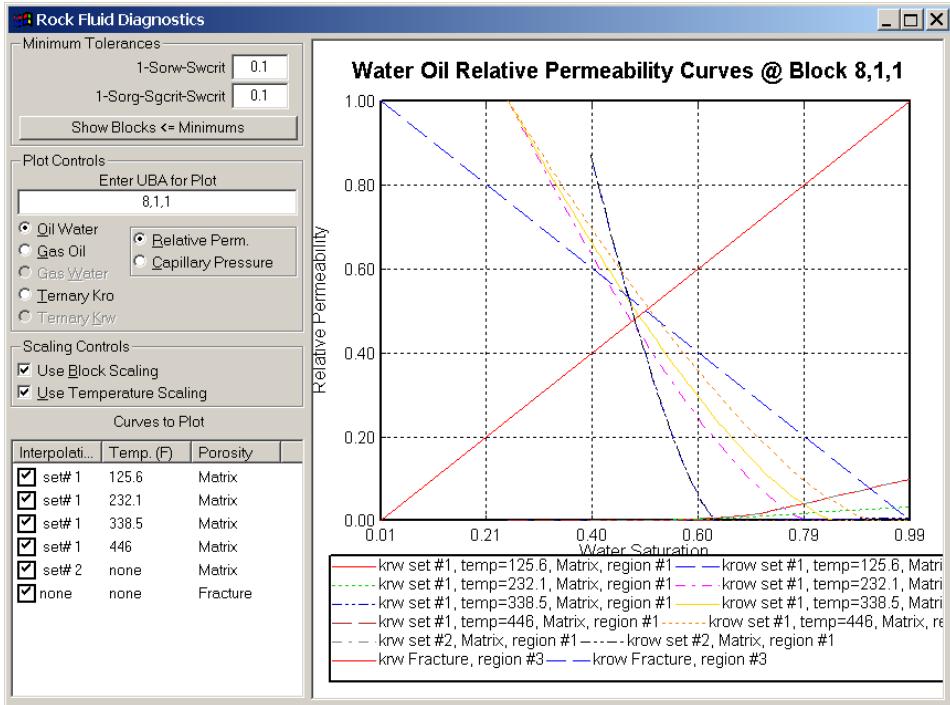
Parameters for rock types in STARS apply to individual rock types and the possible interpolation sets. There are no global settings.

Rock Fluid – Diagnostic Plots

The rock fluid diagnostic plots feature can be accessed either from the top **Rock-Fluid** menu, or from the context menu available by right-clicking the mouse on the reservoir grid display. The purpose of this feature is to allow you to display the relative permeability relationships that exist at each grid block. The rock fluid diagnostic plots will include the following items if they are used in the simulator data file:

- Block dependent relative permeability end point scaling values
- Temperature dependent relative permeability end point values
- Relative permeability interpolation values

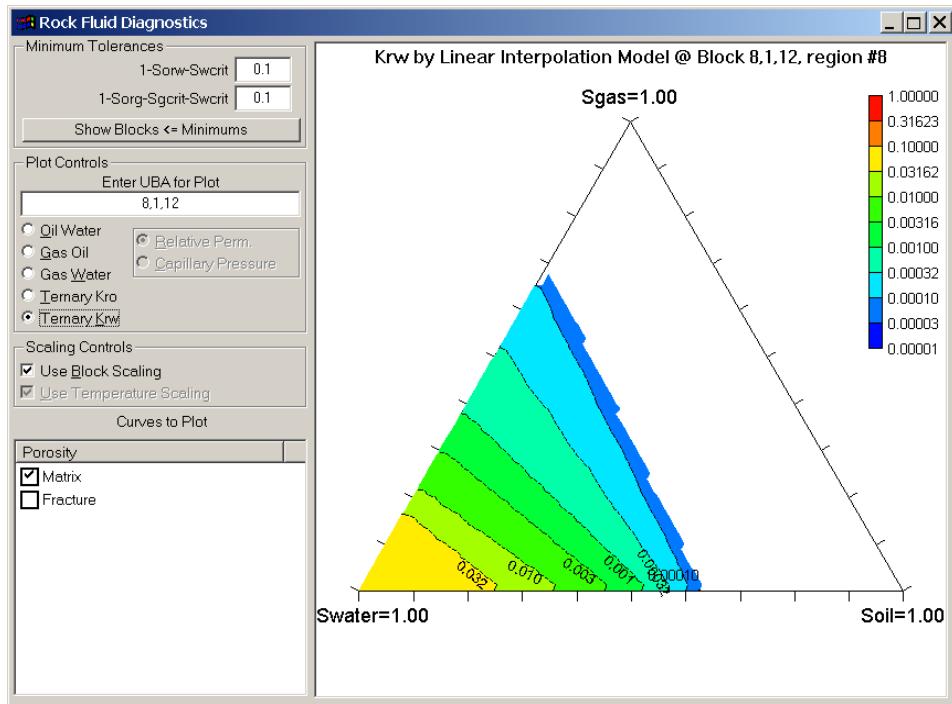
An example of the rock fluid diagnostic plots is shown below:



Since the relative permeability relationships can be block dependent, you must either enter a UBA (block 8,1,1 in the above example) or click on a grid block. If the block has both matrix and fracture properties, then both of these curves will be displayed. If the block has a rock region assigned to it that has relative permeability interpolation, then all of the interpolation sets will be displayed. Similarly, if the rock region has end point temperature dependence, then these temperature dependencies will also be displayed. The above example has all of these features turned on for block 8,1,1.

The rock fluid diagnostic plots can also display ternary diagrams showing 3-phase relationships that exist at each block. If the block is assigned a rock region that uses the normal water wet option, then you can display the 3 phase Kro relationship existing at the block. If the rock region uses the oil wet option, or any of the intermediate wettability options, then you can display the 3 phase Krw relationship existing at the block.

An example is shown below:



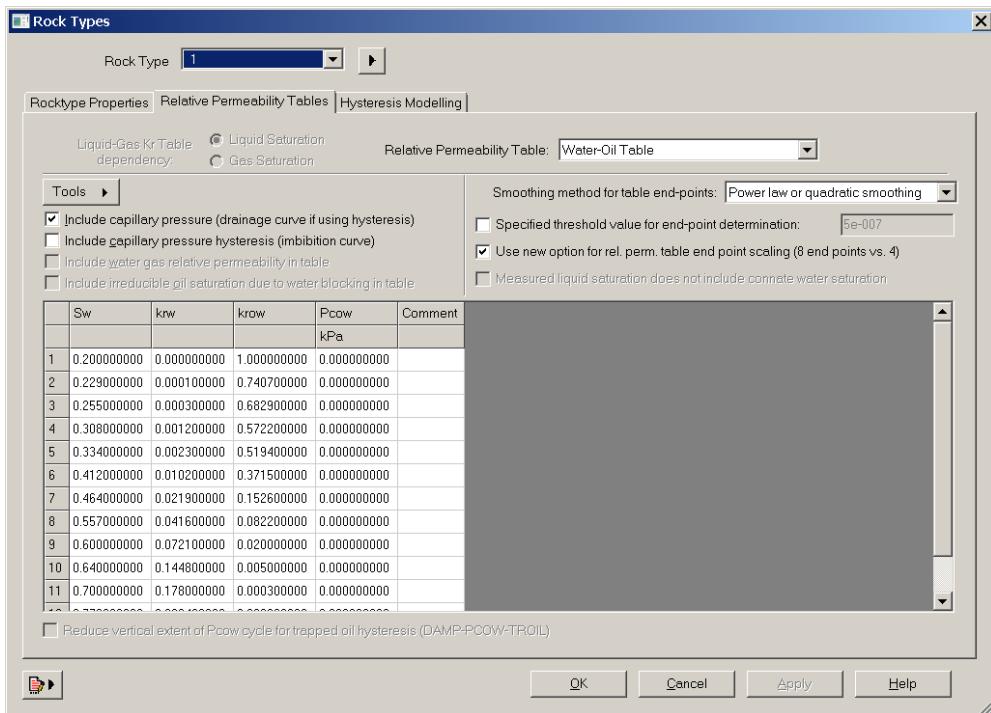
Show Blocks <= Minimums

This sub-feature of the rock fluid diagnostic plots is intended to allow you to locate blocks that may cause poor simulator performance because of a poor choice of end points. If the range between end points is too narrow, then the relative permeability curves become very steep and the simulation may become very slow with very small time-steps. Therefore, if any type of end point scaling option is used in the data set, it is recommended to check each block using this option. You can change the default minimum tolerances of **1-Sorw-Swcrit** and **1-Sorg-Sgcrit-Swcrit** if desired and then click on the **Show Blocks <= Minimums** button. If any blocks are less than the minimum tolerances, then a text box will be displayed that lists these blocks. It is up to you to fix these blocks to prevent slow simulation runs.

Rock Fluid Properties - IMEX

Overview

IMEX rock types and rock fluid properties are entered through the **Rock Types** dialog box, shown below:



As shown above, the **Rock Types** dialog box has three tabs and a drop-down list of the different rock types together with a menu button for adding, copying and deleting rock types.

Each rock type, with its related information, is entered through the following tabs:

- Rocktype Properties
- Relative Permeability Tables
- Hysteresis Modelling

There are several ways to access the **Rock Types** dialog box:

- In the main Builder window, select **Rock-Fluid | Create/Edit Rock Types**
- In the tree view, double-click **Rock Fluid Types**
- Double-click in the tree item for an existing rock type
- Select from the tree context menu item **Create/Edit Rock Types**

Note: When the **Rock Types** dialog box is opened, Builder will correspondingly open the plot view window and display relative permeability curves for the selected rock type.

Modifying Existing Rock Types

To modify an existing rock type, open the **Rock Types** dialog box and select the desired rock type from the drop down list at the top of the dialog box. The data displayed in the tabs will reflect settings for the selected rock type.

When editing a given rock type, changes made to the data will not be applied until you select **OK** or **Apply**. When changing the selected rock type, you will be asked if you want to apply the changes before continuing.

While changes to the relative permeability tables are not permanently applied to the dataset until you save, Builder's plot view window will display relative permeability curves derived from the table data in the selected rock type from the **Rock Types** dialog box. This will permit you to experiment with the table data and cancel at any time.

Adding a New Rock Type

A new rock type can be added to the list of available rock types by clicking the  button then selecting **New Rock Type** from the drop-down menu.

Alternatively, a new rock type can be created as a copy of an existing rock type. To do this, select the rock type you want to copy from the drop-down list then click the  button and select **Copy Current Rock Type**.

Deleting an Existing Rock Type

An existing rock type can be deleted by first selecting the desired rock type from the drop-down list then clicking the  button and selecting **Delete Rock Type** from the drop-down menu.

Note: You cannot cancel from this operation therefore Builder will issue a message to confirm that you want to delete the currently selected rock type.

As each rock type is removed, the remaining rock types are renamed so that the rock type numbers are sequential. For example, if there are three rock types (RockType 1 through 3) and RockType 2 is deleted, then RockType 3 is renamed RockType 2. Therefore, keep in mind that deleting a rock type might affect another property such as when assigning multiple rock types to the grid. When deleting rock types, review related array properties (such as RTYPE and KRTYPE) to make sure the values correctly reflect the available rock types.

Note: Builder will prevent you from deleting the last remaining rock type, since the simulator requires a minimum of one rock type defined for the dataset.

Averaging Laboratory Data into an Average Rock Type

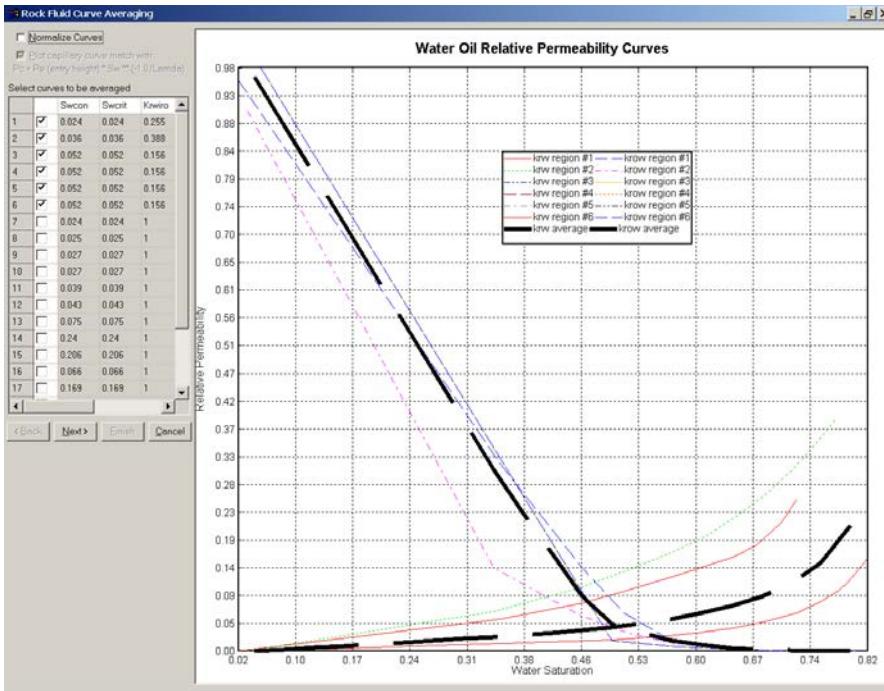
The feature to average rock types is designed to help you select and average relative permeability and capillary pressure curves measured in the laboratory. This feature is available by selecting **Average Rock Type** from the **Rock Types** dialog box by clicking the  button at the top of the page.

If laboratory data is available, it is necessary to convert this information into a format compatible with CMG's relative permeability and capillary pressure data input format (see the simulator keywords *SWT, *SGT, and *SLT), and insert this information into a normal simulator data set. For this Average Rock Type option, it is not necessary to make sure that the endpoints are correct (that is, Krocw is the same for both *SWT and *SGT) as is required by the simulators. The rock type created when you click **Finish** will be compatible with all of CMG's simulators by making sure that any endpoint rules required by the simulators are honored.

There can be up to seven steps in this wizard, depending on the content of the data set:

1. Water-Oil Relative Permeability Averaging
2. Gas-Liquid Relative Permeability Averaging
3. Gas-Water Relative Permeability Averaging (for oil wet or mixed wettability options)
4. Water-Oil Imbibition Capillary Pressure Averaging
5. Water-Oil Drainage Capillary Pressure Averaging
6. Gas-Liquid Imbibition Capillary Pressure Averaging
7. Gas-Liquid Drainage Capillary Pressure Averaging

For each of the steps, there is a grid control on the left side of the screen that contains a user-controllable column of check boxes, as shown in the following example:

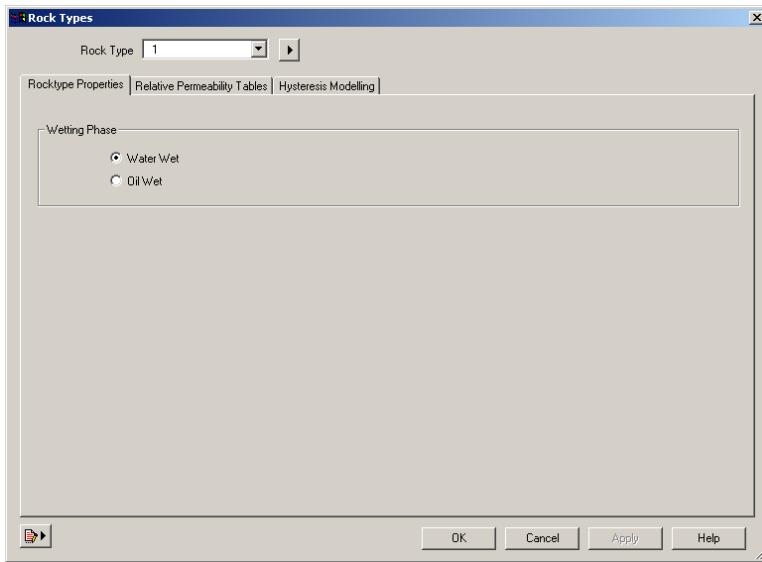


You can select or cancel any of the curves by checking or clearing the associated check boxes. In all cases, the heavy dashed black line will be the averaged curve. You can check the shape of these curves both in normal display mode, and normalized display mode by checking or clearing the **Normalize Curves** check box at the top left of the dialog box. The state of this check box does not affect any calculations of the averaged curve.

If capillary pressure curves are loaded into the data set, then these curves will also be averaged in the same fashion as the relative permeability curves. In addition, each capillary pressure curve is matched using the equation $P_c = P_e \cdot S_w^{**} (-1.0/\Lambda)$, where P_c is capillary pressure, P_e is entry height, S_w is irreducible water saturation, and Λ is the slope of the P_c versus S_w curve when plotted as a Log-Log plot. These coefficients are useful for some fracture characterization methods that require a calculation of S_w from capillary pressure curves.

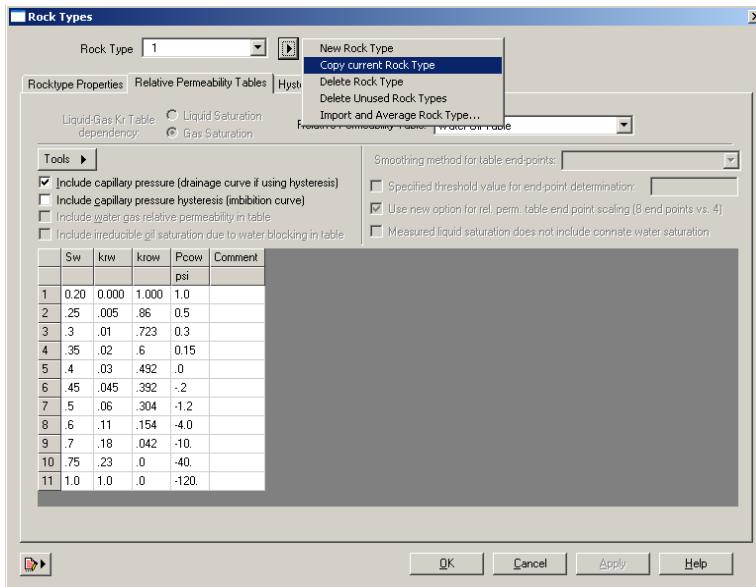
Setting Rock Type Properties

In IMEX the wetting phase can be set for each rock type. This setting is found on the **Rock Types** dialog box, on the **Rocktype Properties** tab:

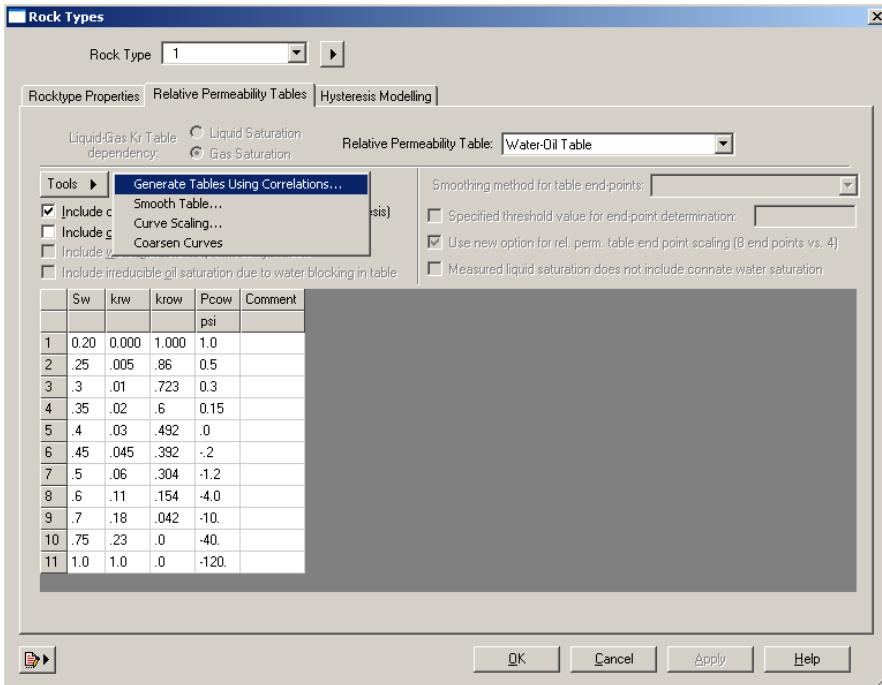


Defining Relative Permeability Tables

The **Relative Permeability Tables** tab is important for entering information about each rock type. The tabular interface allows you to copy and paste data from spreadsheet programs. Aside from the convenience of a spreadsheet-like interface, this tab allows you to access a number of convenient tools for working with relative permeability tables. There is a tool for generating relative permeability tables from analytical equations, a tool for smoothing curve data, and a tool for scaling curve data.

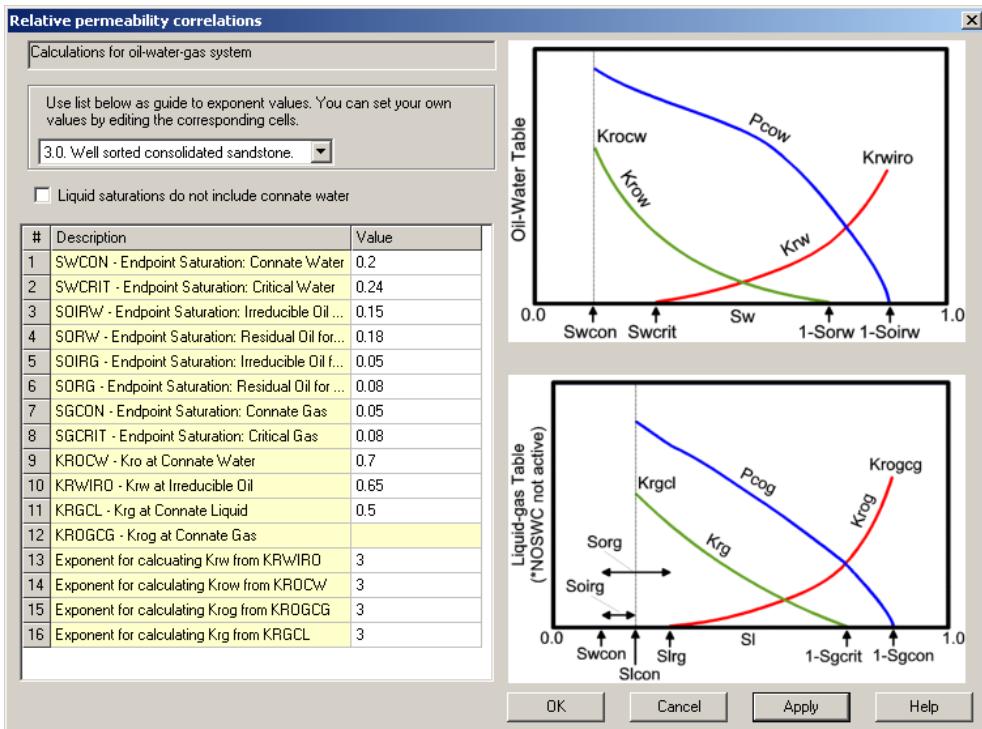


The **Tools**  button in the upper left corner of the **Relative Permeability Tables** tab is used to access the available tools. When this button is clicked, a drop-down menu with a list of the available tools will be displayed:



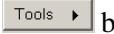
Tool: Generate Tables Using Correlations

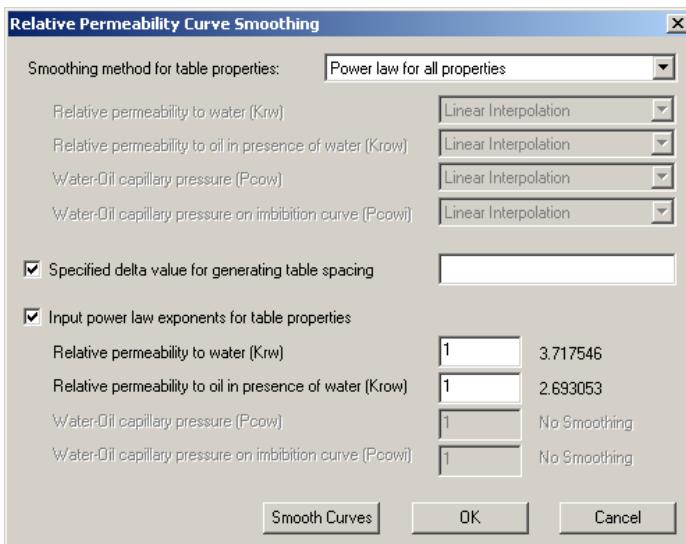
To open the dialog box for generating tables from analytical equations – correlations, click the  button then select **Generate Tables Using Correlations**. The **Relative permeability correlations** dialog box is displayed:



The **Relative permeability correlations** dialog box displays relative permeability curves to help you identify the meaning of the required fields. Builder will verify that entered values are appropriate for the analytical equations. After entering required values and clicking **OK** or **Apply**, Builder will generate the appropriate relative permeability tables. The curves will be displayed in Builder's plot view. The equations used for these correlations are given in Appendix A of the *IMEX User's Guide*.

Tool: Smooth Table

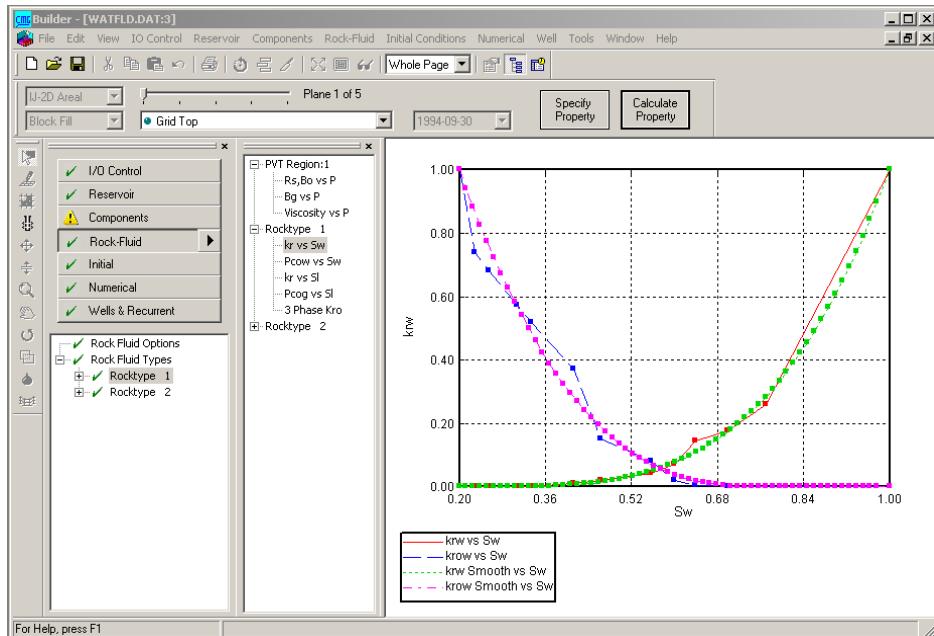
To open the **Relative Permeability Curve Smoothing** dialog box, click the  button and then select **Smooth Table**:



Note: Relative permeability curves can be smoothed with all of the smoothing capabilities provided in CMG's IMEX simulator.

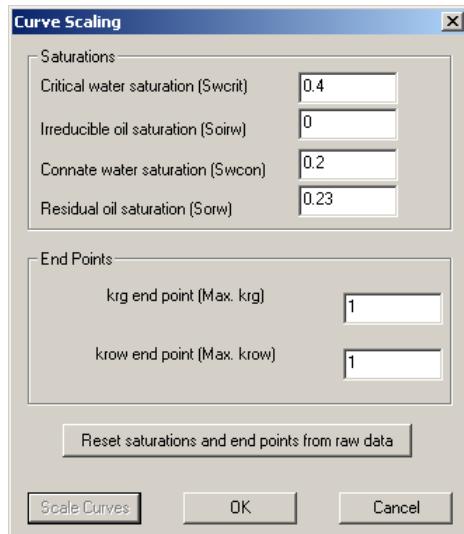
When the **Relative Permeability Curve Smoothing** dialog box is opened, the plot view will display relative permeability curves for both before and after the smoothing operation. You can modify the default settings in the **Relative Permeability Curve Smoothing** dialog box at any time. After you click the **Smooth Curves** button, the plot view will be updated to display the smoothed curves.

When you are satisfied with the smoothed curves, click **OK** to exit the dialog box and apply the changes to the table in the **Rock Types** dialog box. Alternatively, click **Cancel** to cancel the changes in the current **Smooth Curves** session.



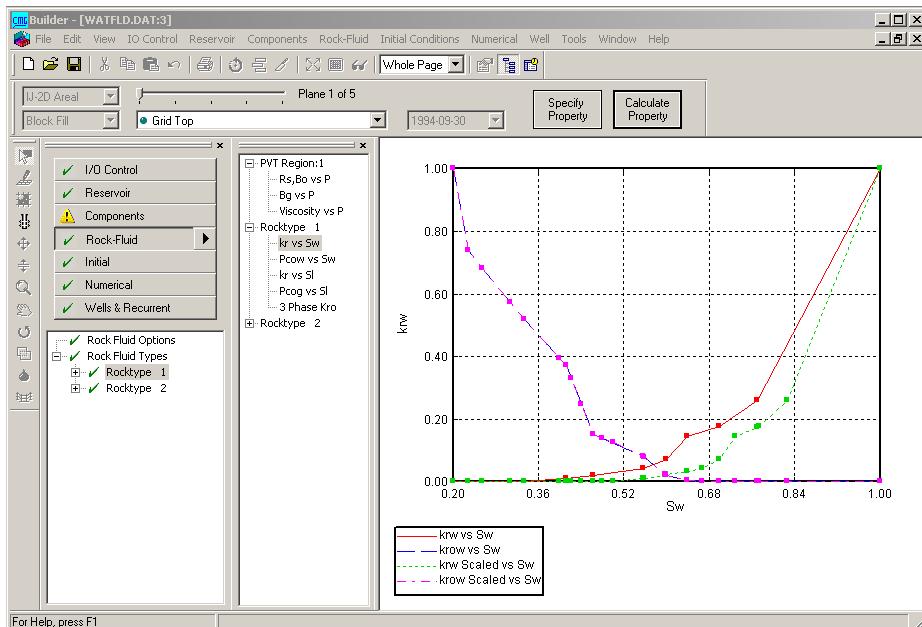
Tool: Curve Scaling

To open the **Curve Scaling** dialog box, click the button then select **Curve Scaling**:



When the **Curve Scaling** dialog box is opened, the plot view will display relative permeability curves both before and after the curve scaling. You can modify the default settings in the **Curve Scaling** dialog box at any time. After clicking the **Scale Curves** button, the plot view will be updated to display the scaled curves.

When you are satisfied with the scaled curves, click **OK** to exit the dialog box and apply the changes to the table in the **Rock Types** dialog box. Alternatively, click **Cancel** to cancel the changes in the current **Curve Scaling** session.



Modifying Tables from the Plot View

Rather than modifying numerical values in a table, Builder allows you to change your relative permeability curves directly from the plot view, by clicking and dragging the points up or down.

To edit the curves, the **Rock Types** dialog box must be open. The **Rock Types** dialog box can be opened directly from the menu or main tree view, as previously discussed. Alternatively, you can double-click on a curve directly on Builder's plot view. This will open the **Rock Types** dialog box with the appropriate rock type preselected. You can then edit points on the curve by clicking and dragging them.

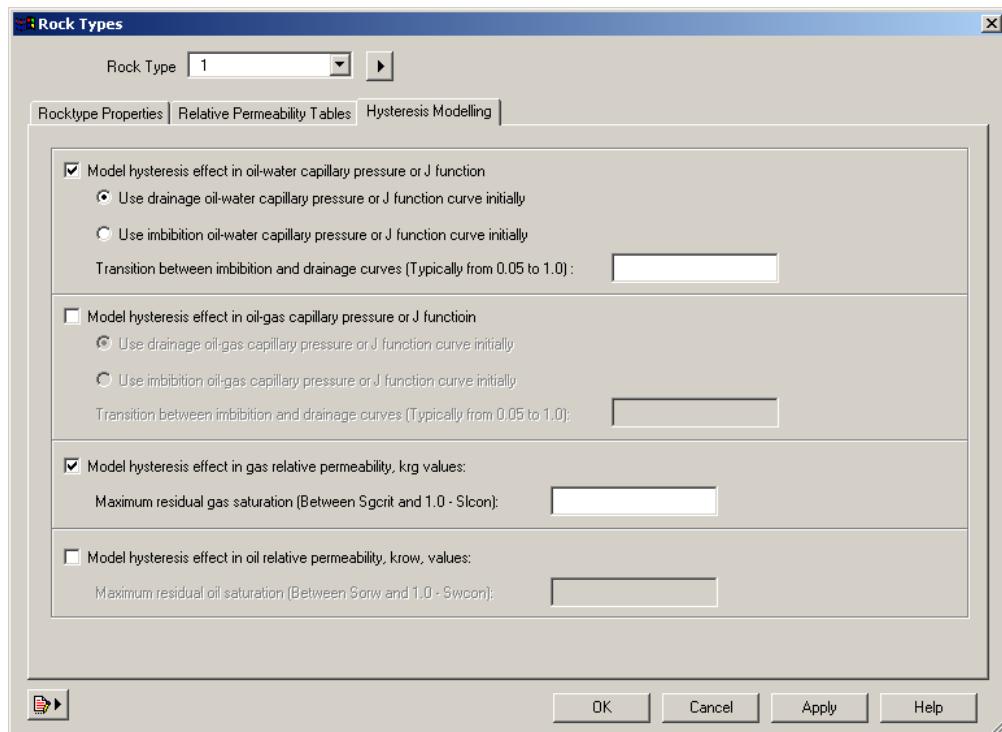
You can click **OK** or **Apply** in the **Rock Types** dialog box at any time to permanently apply the changes to the dataset. Alternatively, you can click **Cancel** if you decide that your changes are not appropriate.

Setting Hysteresis Modelling

IMEX supports hysteresis for both capillary pressure and gas relative permeability. For capillary pressure hysteresis, this requires the specification of the transition between the imbibition and the drainage curves. For gas relative permeability hysteresis, this requires the specification of the maximum gas saturation (this saturation in practice is an adjustable parameter which determines the imbibition krg curve as a function of the given drainage curve).

To enable a hysteresis effect, select the appropriate check box on the **Hysteresis Modelling** tab then enter the values in the relative permeability table on the **Relative Permeability Tables** tab.

To cancel a hysteresis effect, clear the check box on the **Hysteresis Modelling** tab. The imbibition column will be removed from the relative permeability table.



Seawater Scale Deposit and Damage Tables

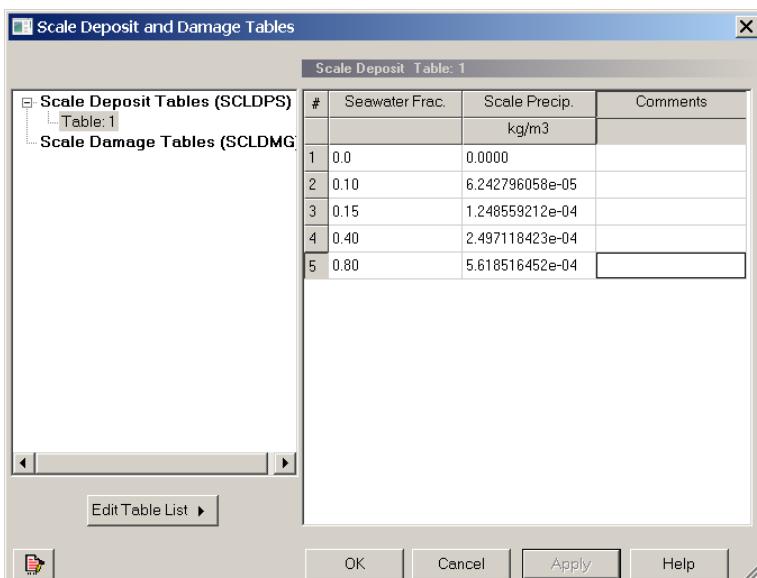
The IMEX **Scale Deposit and Damage Tables** dialog box allows you to enter tables for both scale deposition and scale damage. The dialog box can be opened by selecting **Rock-Fluid | Seawater Scale Deposit/Damage Tables** or by double-clicking on one of the **Seawater Scale Buildup** tree view items.

The dialog box provides a tree view which lists existing tables. By selecting a table from the list, the grid on the right is updated to reflect the selection. A graph of the tables can also be viewed in the plot view, which is synchronized with the dialog box.

The values for the tables can be modified by entering values directly in the grid or by clicking on a point in the plot view graph and dragging the point. You can also edit the table by right-clicking on the table. This action will bring up a menu which allows you to cut, copy, paste, insert row and delete row. Both the dialog box and graph are synchronized – any changes made in one will immediately be reflected in the other.

The **Edit Table List** button allows you to manage the table list. When the button is clicked a menu appears which allows you to add new, copy, insert and delete tables.

Comments can be entered individually for each row in a table or as single comment for the whole selected table. The row comments are entered in the last column of the grid. The single table comment can be entered and viewed by clicking the comment  button. Each table can still have its own comment.



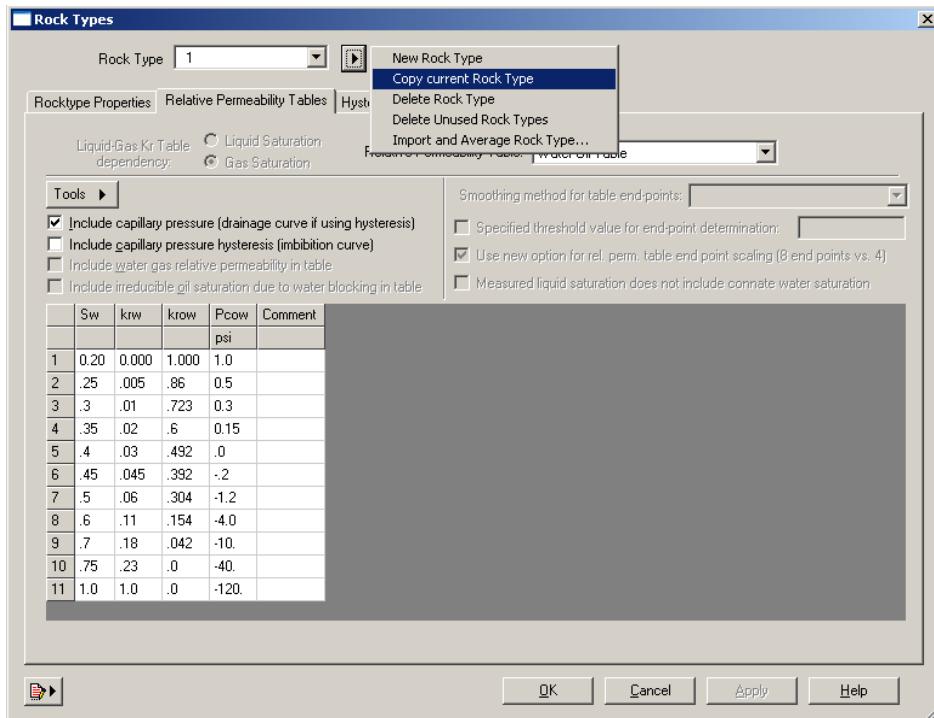
The following buttons are displayed at the bottom of the **Scale Deposit and Damage Tables** dialog box:

- **OK:** Closes the interface and saves any changes that have been made.
- **Cancel:** Closes the interface but will not save any changes made unless **Apply** was clicked first.
- **Apply:** Saves any changes that have been made but will not close the interface.
- **Help:** Brings up simulator help information about the item on the interface with the current focus. Clicking the **F1** key will also bring up the respective simulator help.

Rock Fluid Properties - GEM

Overview

GEM rock types and rock fluid properties are entered through the **Rock Types** dialog box, shown below:



As shown above, the **Rock Types** dialog box has three tabs and a drop-down list of the different rock types together with a menu button for adding, copying and deleting rock types.

Each rock type and its related information are entered through the following tabs:

- Rocktype Properties
- Relative Permeability Tables
- Hysteresis Modelling

There are a number of ways to access the **Rock Types** dialog box:

- In the main Builder window, select **Rock-Fluid | Create/Edit Rock Types**
- In the tree view, double-click **Rock Fluid Types**
- Double-click in the tree item for an existing rock type
- Select from the tree context menu item **Create/Edit Rock Types**

Modifying Existing Rock Types

To modify an existing rock type, open the **Rock Types** dialog box and select the desired rock type from the drop-down list at the top of the dialog box. The data displayed in the tabs will reflect settings for the selected rock type.

When editing a given rock type, changes made to the data will not be applied until you select **OK** or **Apply**. When changing the selected rock type, you will be asked if you want to apply the changes before continuing.

While changes to the relative permeability tables are not permanently applied to the dataset until you save, Builder's plot view window will display relative permeability curves derived from the table data in the selected rock type from the **Rock Types** dialog box. This will permit you to experiment with the table data and cancel at any time.

Adding a New Rock Type

A new rock type can be added to the list of available rock types by clicking the  button then selecting **New Rock Type** from the drop-down menu.

Alternatively, a new rock type can be created as a copy of an existing rock type. To do this, select the rock type you want to copy from the drop-down list then click the  button and select **Copy Current Rock Type**.

Deleting an Existing Rock Type

An existing rock type can be deleted by first selecting the desired rock type from the drop-down list then clicking the  button and selecting **Delete Rock Type** from the drop-down menu.

Note: You cannot cancel from this operation therefore Builder will issue a message to make sure that you wish to delete the currently selected rock type.

As each rock type is removed, the remaining rock types are renamed so that the rock type numbers are sequential. For example, if there are three rock types (RockType 1 through 3) and RockType 2 is deleted, then RockType 3 is renamed RockType 2. Therefore, you must keep in mind that deleting a rock type might affect another property such as when assigning multiple rock types to the grid. When deleting rock types review related array properties (such as RTYPE and KRTYPE) to make sure the values correctly reflect the available rock types.

Note: Builder will prevent you from deleting the last remaining rock type, since the simulator requires a minimum of one rock type defined for the dataset.

Averaging Laboratory Data into an Average Rock Type

The feature to average rock types is designed to help you select and average relative permeability and capillary pressure curves measured in the laboratory. This feature is available by selecting

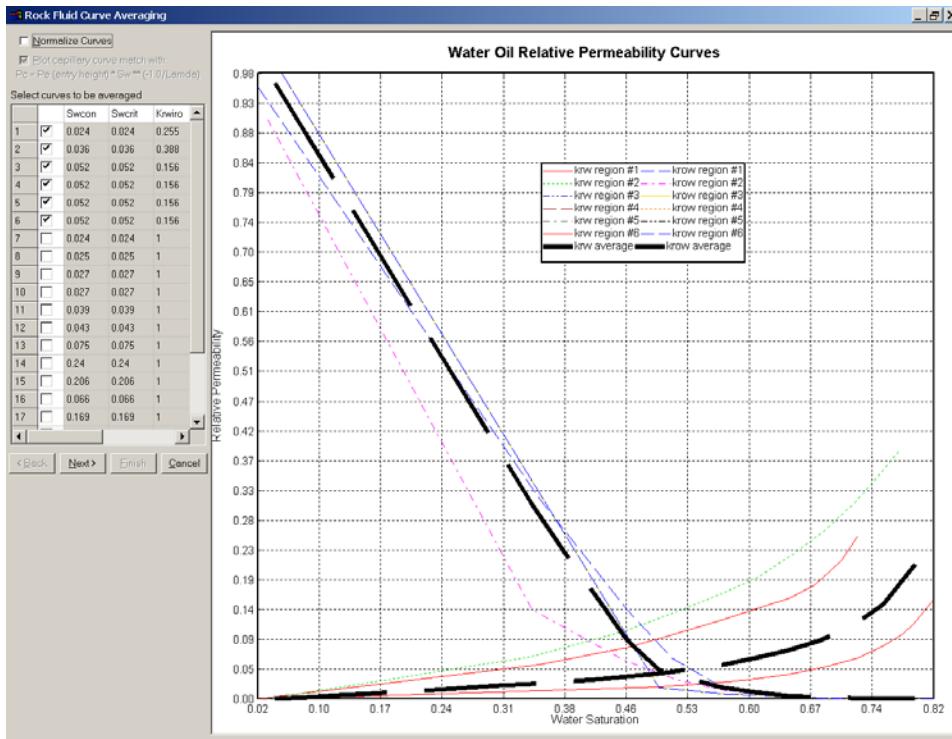
Average Rock Type from the **Rock Types** dialog box by clicking this button  at the top of the page.

If laboratory data is available, it is necessary to convert this information into a format compatible with CMG's relative permeability and capillary pressure data input format (see the simulator keywords *SWT, *SGT, and *SLT), and insert this information into a normal simulator data set. For this Average Rock Type option, it is not necessary to make sure that the endpoints are correct (that is, Krocw is the same for both *SWT and *SGT) as is required by the simulators. The rock type created when you click **Finish** will be compatible with all of CMG's simulators by making sure that any endpoint rules required by the simulators are honored.

There can be up to seven steps in this wizard, depending on the content of the data set:

1. Water-Oil Relative Permeability Averaging
2. Gas-Liquid Relative Permeability Averaging
3. Gas-Water Relative Permeability Averaging (for oil wet or mixed wettability options)
4. Water-Oil Imbibition Capillary Pressure Averaging
5. Water-Oil Drainage Capillary Pressure Averaging
6. Gas-Liquid Imbibition Capillary Pressure Averaging
7. Gas-Liquid Drainage Capillary Pressure Averaging

For each of the steps, there is a grid control on the left side of the screen that contains a user-controllable column of check boxes, as shown in the following example:



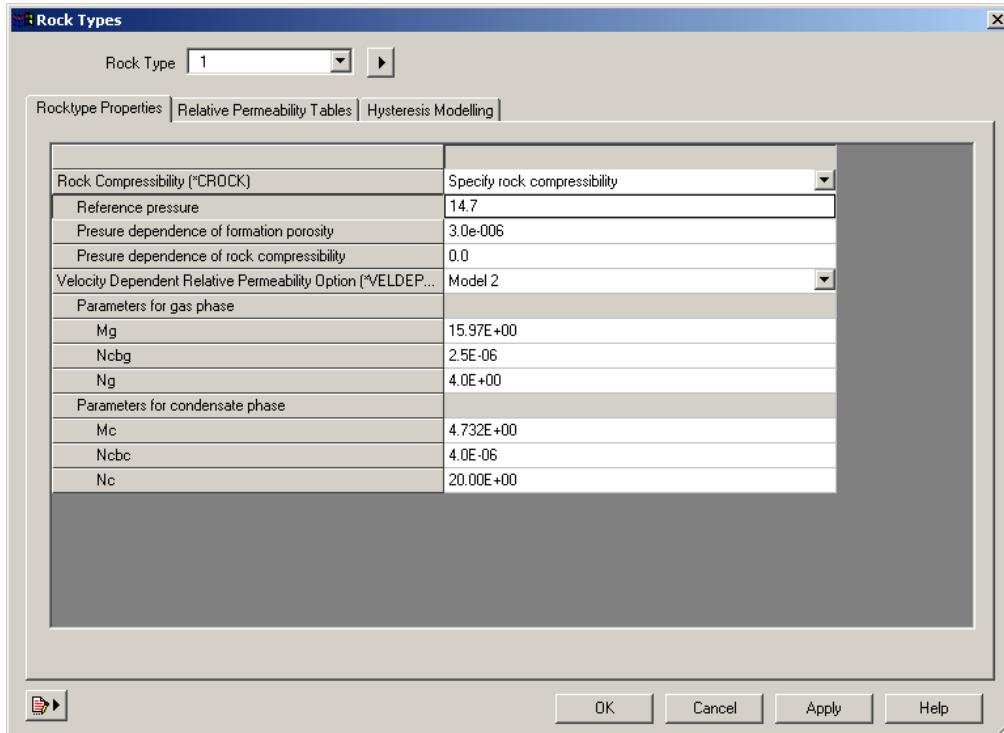
You can select or cancel any of the curves by checking or clearing the associated check boxes. In all cases, the heavy dashed black line will be the averaged curve. You can check the shape of these curves both in normal display mode, and normalized display mode by checking or clearing **Normalize Curves** at the top left of the dialog box. The state of this check box does not affect any calculations of the averaged curve.

If capillary pressure curves are loaded into the data set, then these curves will also be averaged in the same fashion as the relative permeability curves. In addition, each capillary pressure curve is matched using the equation $P_c = P_e * S_w^{**} (-1.0/\text{Lamda})$, where P_c is capillary pressure, P_e is entry height, S_w is irreducible water saturation, and Lamda is the slope of the P_c vs. S_w curve when plotted on a Log-Log plot. These coefficients are useful for some fracture characterization methods that require a calculation of S_w from capillary pressure curves.

Setting Rock Type Properties

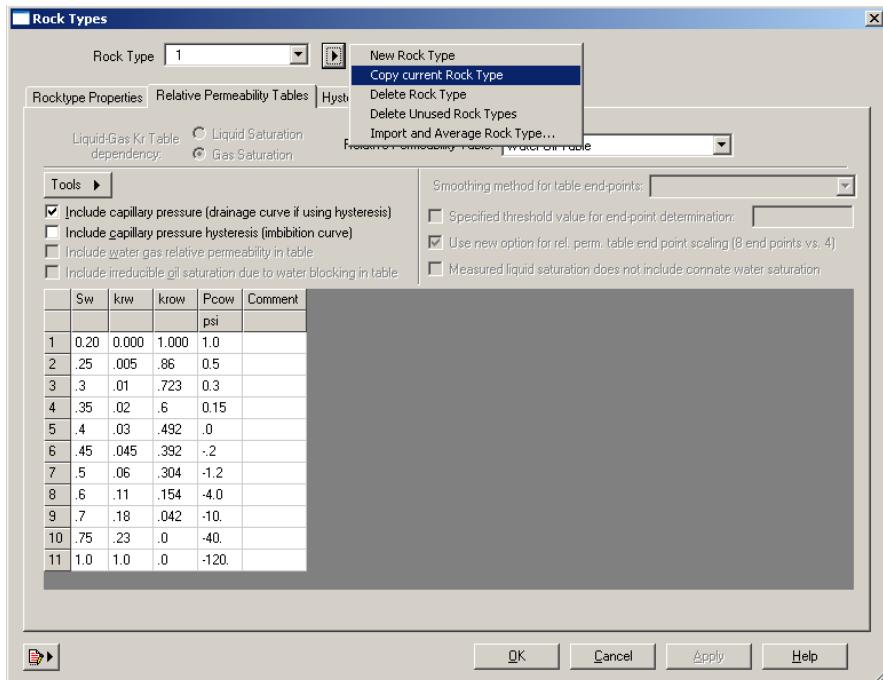
In GEM rock compressibility can be set independently for each rock type. This setting can be found on the Rock Types dialog box on the **Rocktype Properties** tab.

Settings for **Velocity Dependent Relative Permeability Option (VELDEPRP)** can also be found on the **Rocktype Properties** tab:

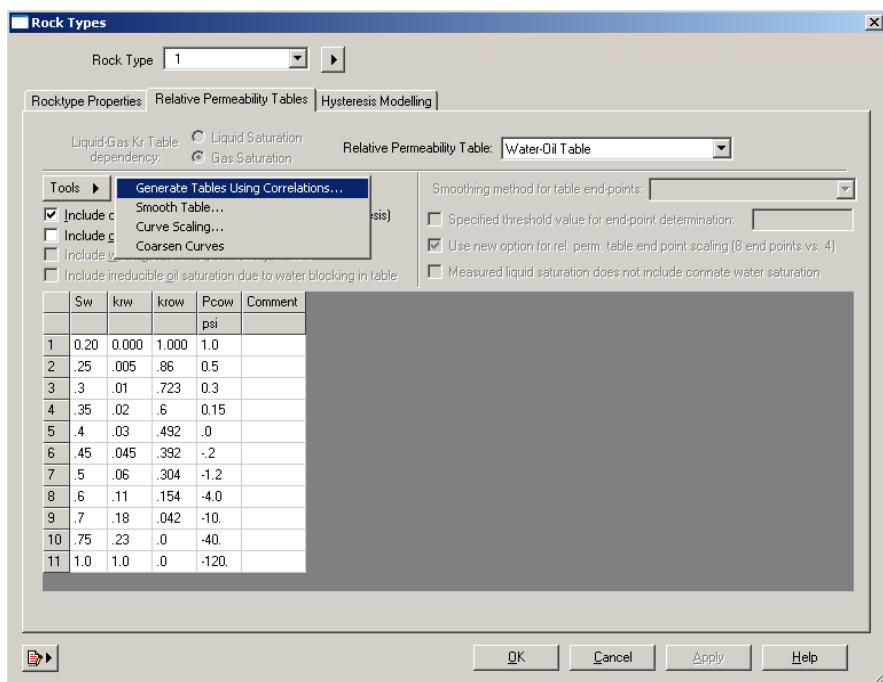


Defining Relative Permeability Tables

The Relative Permeability Tables tab is important for entering information about each rock type. The tabular interface allows you to copy and paste data from spreadsheet programs. Aside from the convenience of a spreadsheet-like interface, this tab allows you to access a number of convenient tools for working with relative permeability tables. There is a tool for generating relative permeability tables from analytical equations, a tool for smoothing curve data, and a tool for scaling curve data.

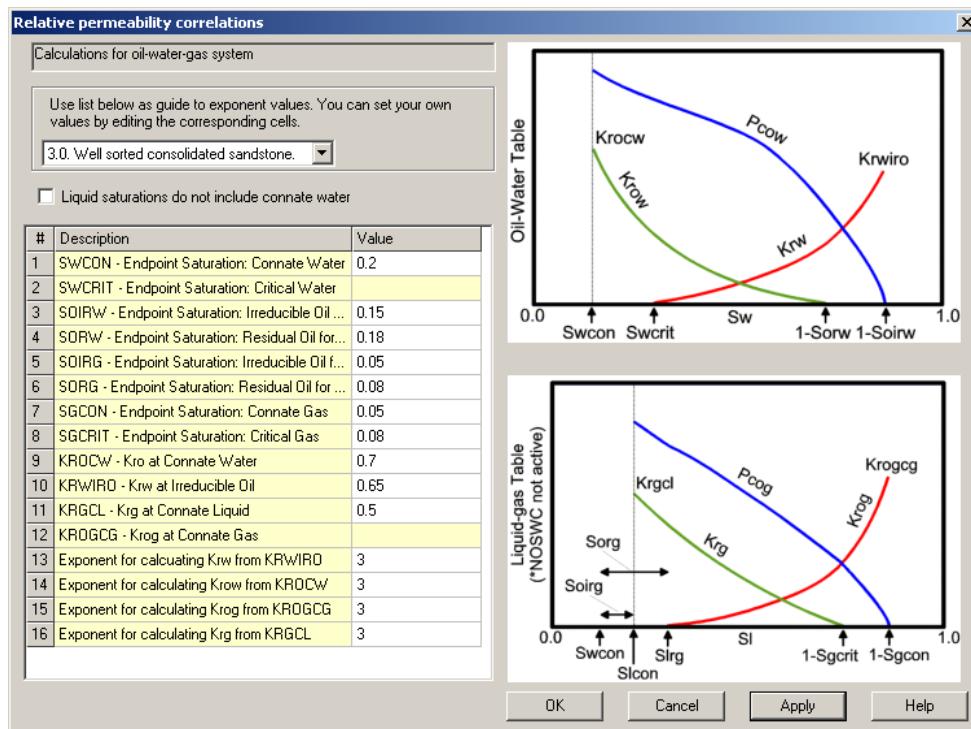


The **Tools** button in the upper left corner of the **Relative Permeability Table** tab is used to access the available tools. When this button is clicked, a drop-down menu with a list of the available tools will be displayed:



Tool: Generate Tables Using Correlations

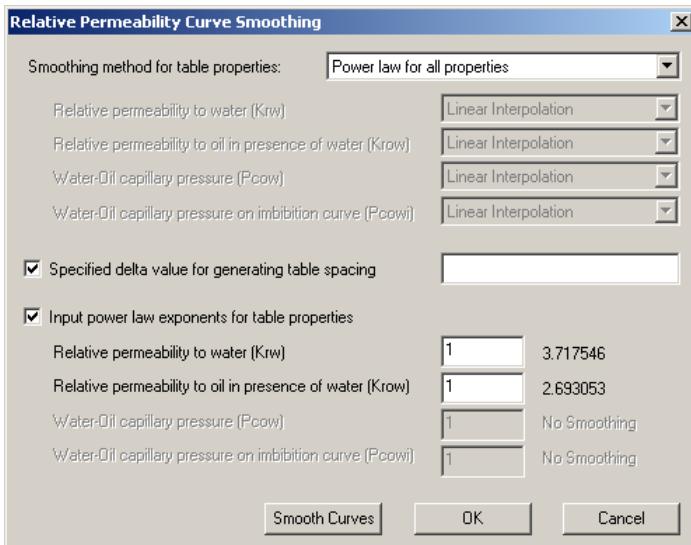
To open the dialog box for generating tables from analytical equations – correlations, click the **Tools** button then select **Generate Tables Using Correlations**. The **Relative permeability correlations** dialog box is displayed:



The **Relative permeability correlations** dialog box displays relative permeability curves to help you identify the meaning of the required fields. Builder will verify that your entered values are appropriate for the analytical equations. After entering required values and clicking **OK** or **Apply**, Builder will generate the appropriate relative permeability tables. The curves will be displayed in Builder's plot view. The equations used for these correlations are given in Appendix A of the *GEM Users' Guide*.

Tool: Smooth Table

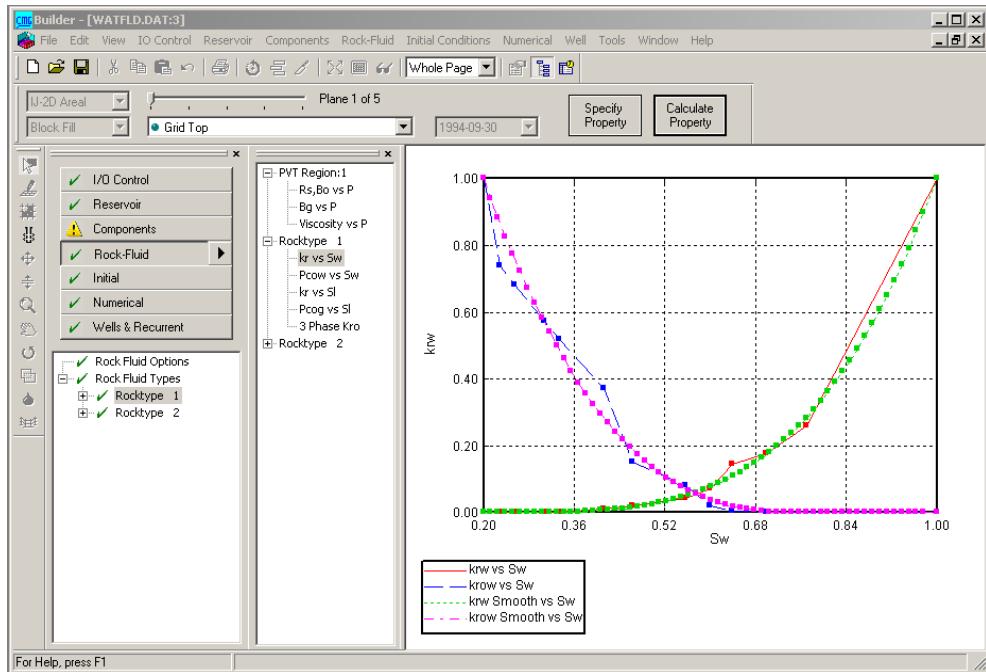
To access the **Relative Permeability Curve Smoothing** dialog box, click the button and then select **Smooth Table**:



Note that relative permeability curves can be smoothed with all the smoothing capabilities build in CMG's GEM simulator.

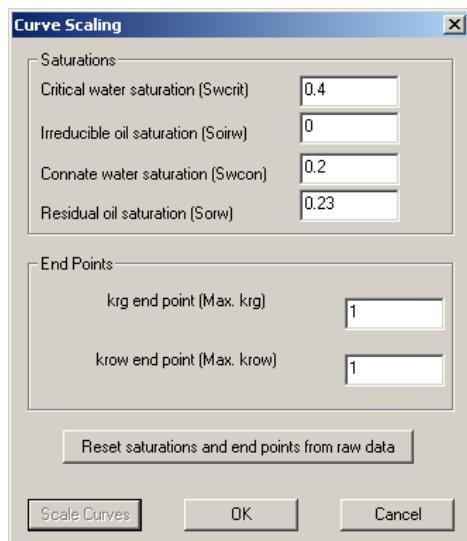
When the **Relative Permeability Curve Smoothing** dialog box is opened, the plot view will display relative permeability curves for both before and after the smoothing. You can modify the default settings in the **Relative Permeability Curve Smoothing** dialog box at any time. After clicking the **Smooth Curves** button, the plot view will be updated to display the smoothed curves.

When you are satisfied with the smoothed curves, click **OK** to exit the dialog box and apply the changes to the table in the **Rock Types** dialog box. Alternatively, click **Cancel** to cancel the changes in the current **Smooth Curves** session.



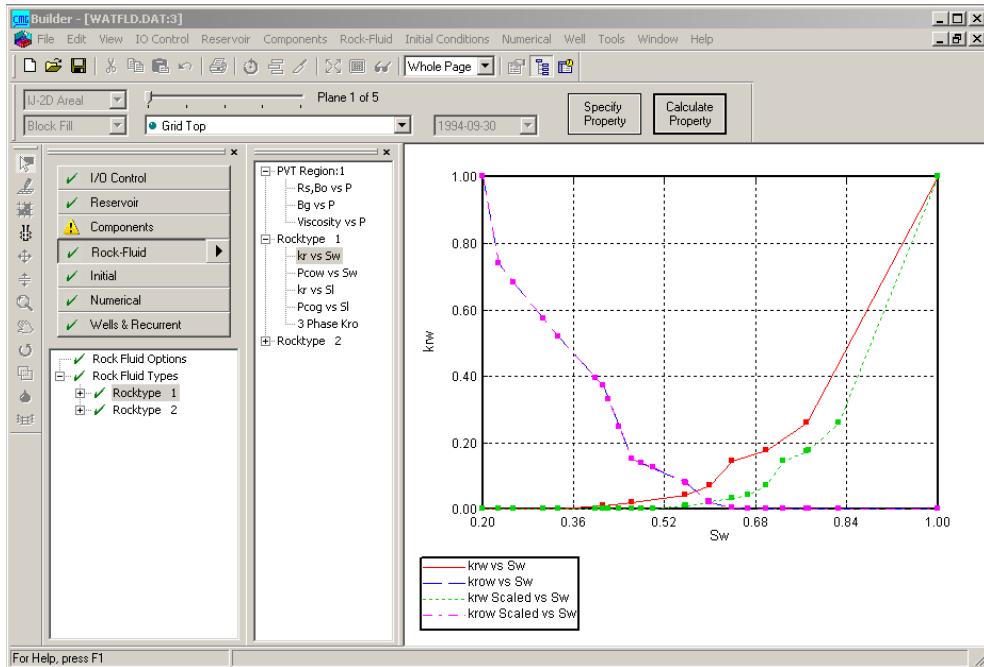
Tool: Curve Scaling

To open the **Curve Scaling** dialog box, click the button then select **Curve Scaling**:



When the **Curve Scaling** dialog box is opened, the plot view will display relative permeability curves both before and after the curve scaling. You can modify the default settings in the **Curve Scaling** dialog box at any time. After clicking the **Scale Curves** button, the plot view will be updated to display the scaled curves.

When you are satisfied with the scaled curves, click **OK** to exit the dialog box and apply the changes to the table in the **Rock Types** dialog box. Alternatively, click **Cancel** to cancel the changes in the current **Curve Scaling** session.



Modifying Tables from the Plot View

Rather than modifying numerical values in a table, Builder allows you to change your relative permeability curves directly from the plot view, by clicking and dragging the points up or down.

To edit the curves, the **Rock Types** dialog box must be open. The **Rock Types** dialog box can be opened directly from the menu or main tree view, as previously discussed.

Alternatively, you can double-click on a curve directly on Builder's plot view. This will open the **Rock Types** dialog box with the appropriate rock type preselected. You can then edit points on the curve by clicking and dragging them.

You can click **OK** or **Apply** in the **Rock Types** dialog box at any time to permanently apply the changes to the dataset. Alternatively you can click **Cancel** if you decide that your changes are not appropriate.

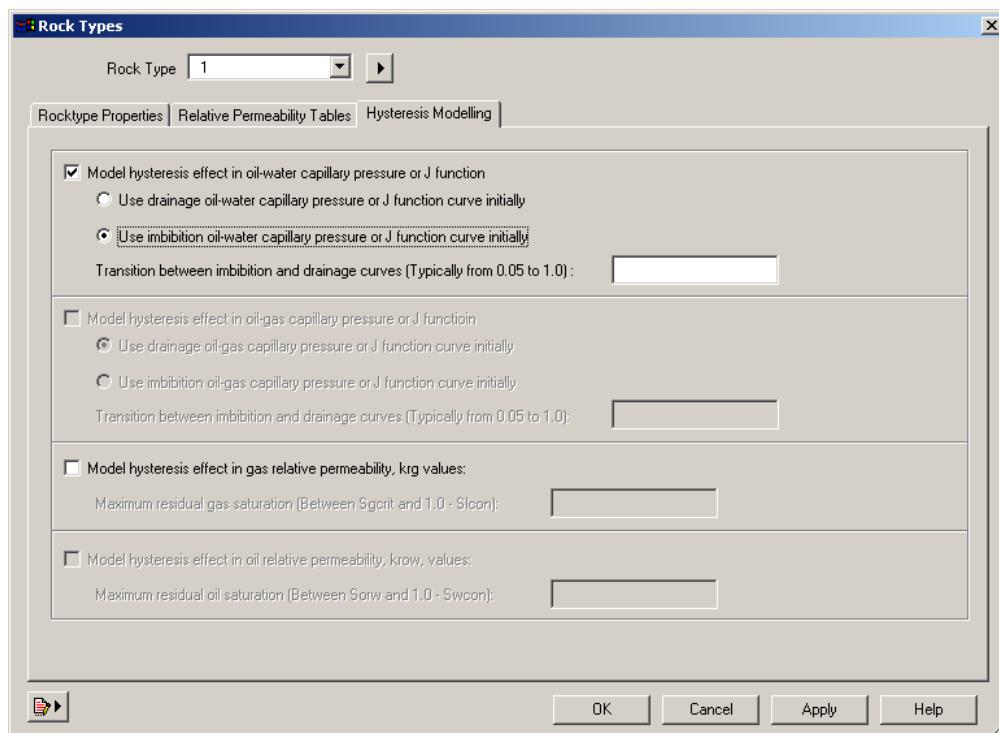
Setting Hysteresis Modelling

GEM supports hysteresis for both capillary pressure and gas relative permeability. Unavailable options appear grey and the corresponding controls cannot be accessed.

For capillary pressure hysteresis, this requires the specification of the transition between the imbibition and the drainage curves. For gas relative permeability hysteresis, this requires the specification of the maximum gas saturation (this saturation in practice is an adjustable parameter which determines the imbibition krg curve as a function of the given drainage curve).

To enable a hysteresis effect, select the appropriate check box on the **Hysteresis Modelling** tab then enter the values in the relative permeability table on the **Relative Permeability Tables** tab.

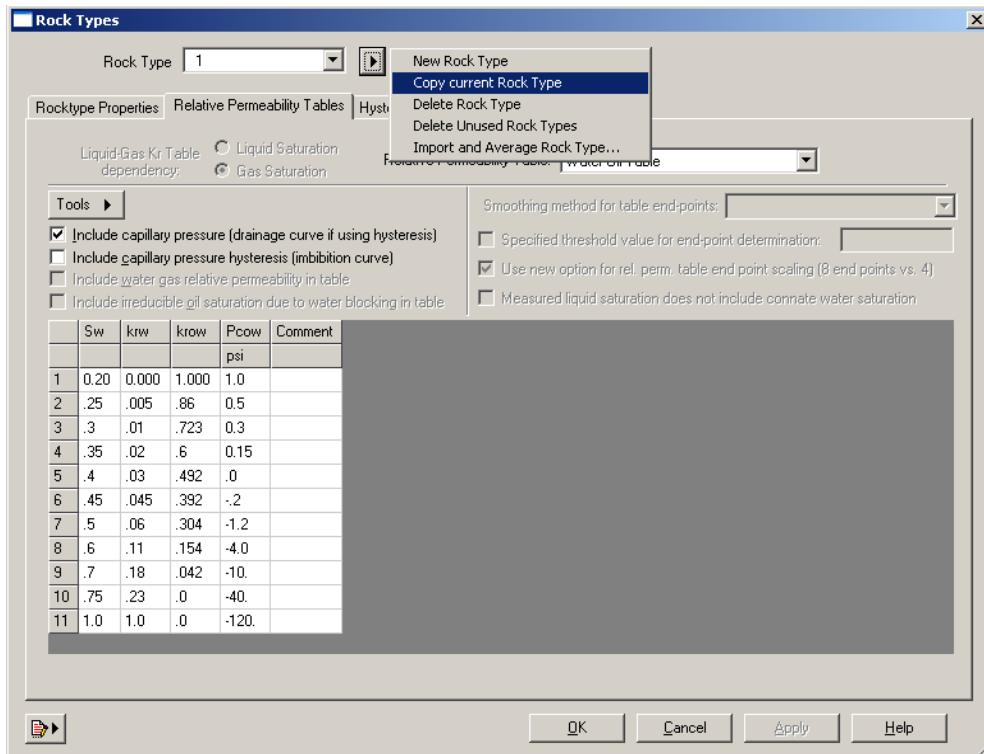
To cancel a hysteresis effect, clear the appropriate check box on the **Hysteresis Modelling** tab. The imbibition column will be removed from the relative permeability table.



Rock Fluid Properties - STARS

Overview

STARS rock types and rock fluid types are entered through the **Rock Types** dialog box, shown below:



As shown above, the **Rock Types** dialog box has three tabs and a drop-down list of the different rock types together with a menu button for adding, copying and deleting rock types.

Each rock type, with its related information, is entered through the following tabs:

- Rocktype Properties
- Relative Permeability Tables
- Hysteresis Modelling

- Relative Permeability End Points
- Interpolation Set Parameters

There are several ways to access the **Rock Types** dialog box:

- In the main Builder window, select **Rock-Fluid | Create/Edit Rock Types**
- In the tree view, double-click **Rock Fluid Types**
- Double-click in the tree item for an existing rock type
- Select from the tree context menu item **Create/Edit Rock Types...**

Modifying Existing Rock Types

To modify an existing rock type, open the **Rock Types** dialog box and select the desired rock type from the drop-down list at the top of the dialog box. The data displayed in the tabs will reflect settings for the selected rock type.

When editing a given rock type, changes made to the data will not be applied until you select **OK**, or **Apply**. When changing the selected rock type, you will be asked if you want to apply the changes before continuing.

While changes to the relative permeability tables are not permanently applied to the dataset until you save, Builder's plot view window will display relative permeability curves derived from the table data in the selected rock type from the **Rock Types** dialog box. This will permit you to experiment with the table data and cancel at any time.

Adding a New Rock Type

A new rock type may be added to the list of available rock types by clicking the  button then selecting **New Rock Type** from the drop-down menu.

Alternatively, a new rock type can be created as a copy of an existing rock type. To do this, select the rock type you want to copy from the drop-down list then click the  button and select **Copy Current Rock Type**.

Deleting an Existing Rock Type

An existing rock type can be deleted by first selecting the desired rock type from the drop-down list then clicking the  button and selecting **Delete Rock Type** from the drop-down menu.

Note: You cannot cancel from this operation therefore Builder will issue a message to make sure that you want to delete the currently selected rock type.

As each rock type is removed, the remaining rock types are renamed so that the rock type numbers are sequential. For example, if there are three rock types (RockType 1 through 3) and RockType 2 is deleted, then RockType 3 is renamed RockType 2. Therefore, keep in mind that deleting a rock type might affect another property such as when assigning multiple rock types to the grid. When deleting rock types please review related array properties (such as RTYPE and KRTYPE) to make sure the values correctly reflect the available rock types.

Note: Builder will prevent you from deleting the last remaining rock type, since the simulator requires a minimum of one rock type defined for the dataset.

Averaging Laboratory Data into an Average Rock Type

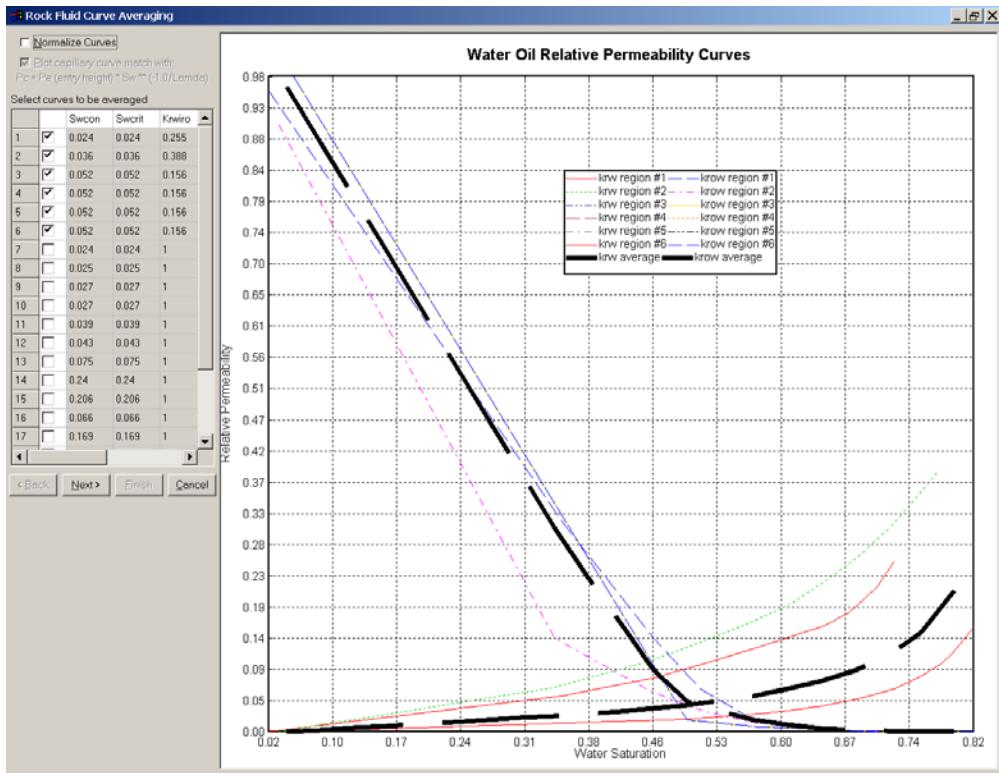
The feature to average rock types is designed to help you select and average relative permeability and capillary pressure curves measured in the laboratory. This feature is available by selecting **Average Rock Type** from the **Rock Types** dialog box by clicking the  button at the top of the page.

If laboratory data is available, it is necessary to convert this information into a format compatible with CMG's relative permeability and capillary pressure data input format (see the simulator keywords *SWT, *SGT, and *SLT), and insert this information into a normal simulator data set. For this Average Rock Type option, it is not necessary to make sure that the endpoints are correct (that is, Krocw is the same for both *SWT and *SGT) as is required by the simulators. The rock type created once you click **Finish** will be compatible with all of CMG's simulators by making sure that any endpoint rules required by the simulators are honored.

There can be up to seven steps in this wizard, depending on the content of the data set:

1. Water-Oil Relative Permeability Averaging
2. Gas-Liquid Relative Permeability Averaging
3. Gas-Water Relative Permeability Averaging (for oil wet or mixed wettability options)
4. Water-Oil Imbibition Capillary Pressure Averaging
5. Water-Oil Drainage Capillary Pressure Averaging
6. Gas-Liquid Imbibition Capillary Pressure Averaging
7. Gas-Liquid Drainage Capillary Pressure Averaging

For each of the steps, there is a grid control on the left side of the screen that contains a user-controllable column of check boxes, as shown in the following example:



You can select or cancel any of the curves by checking or clearing the associated check boxes. In all cases, the heavy dashed black line will be the averaged curve. You can check the shape of these curves both in normal display mode, and normalized display mode by checking or clearing the **Normalize Curves** check box at the top left of the dialog box. The state of this check box does not affect any calculations of the averaged curve.

If capillary pressure curves are loaded into the data set, then these curves will also be averaged in the same fashion as the relative permeability curves. In addition, each capillary pressure curve is matched using the equation $P_c = P_e * S_w^{**} (-1.0/\text{Lamda})$, where P_c is capillary pressure, P_e is entry height, S_w is irreducible water saturation, and Lamda is the slope of the P_c versus S_w curve when plotted as a Log-Log plot. These coefficients are useful for some fracture characterization methods that require a calculation of S_w from capillary pressure curves.

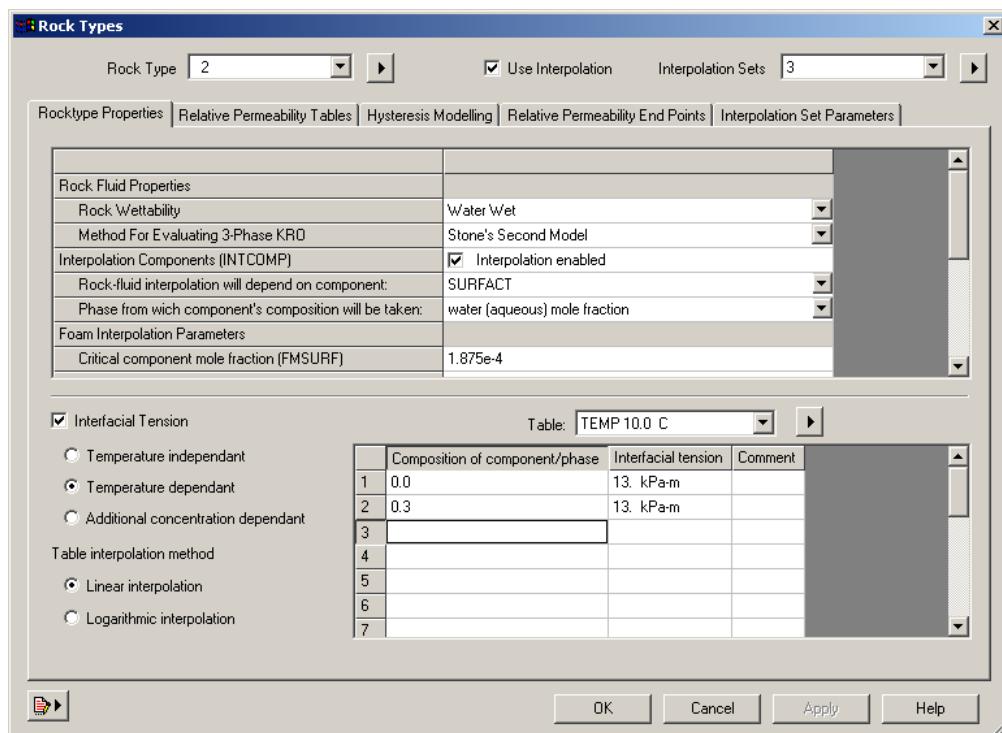
Setting Rock Type Properties

The **Rocktype Properties** tab, shown below, holds controls for parameters that can be set independently for each defined rock type. The tab holds settings for:

- Rock Wettability
- Method for Evaluating 3-Phase KRO

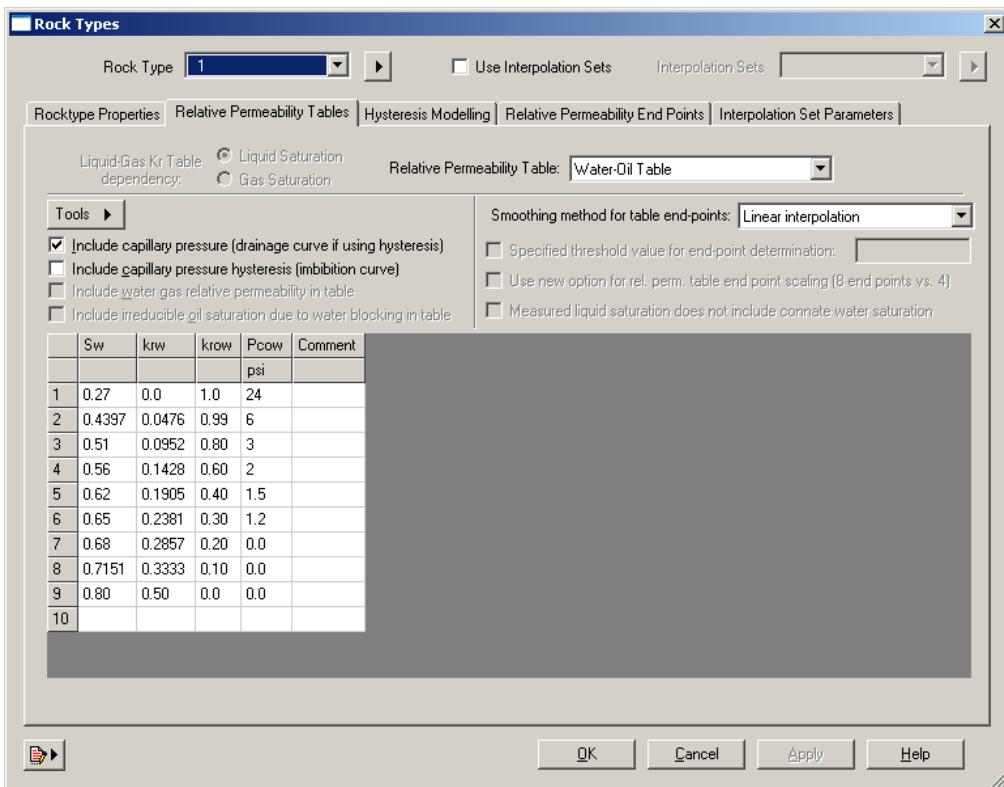
When multiple interpolation sets are defined, the **Rocktype Properties** tab also holds settings for:

- Interpolation Components (INTCOMP)
- Foam Interpolation Parameters
- Interfacial Tension

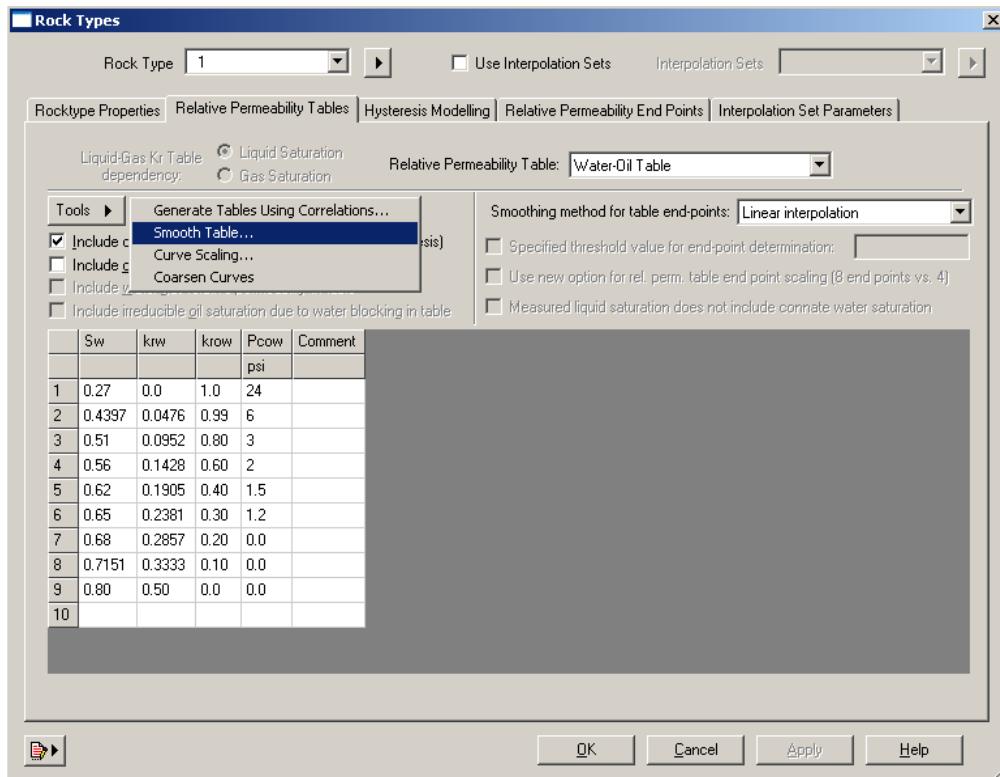


Defining Relative Permeability Tables

The **Relative Permeability Tables** tab is important for entering information about each rock type. The tabular interface allows you to copy and paste data from spreadsheet programs. Aside from the convenience of a spreadsheet-like interface, this tab allows you to access a number of convenient tools for working with relative permeability tables. There is a tool for generating relative permeability tables from analytical equations, a tool for smoothing curve data, and a tool for scaling curve data.

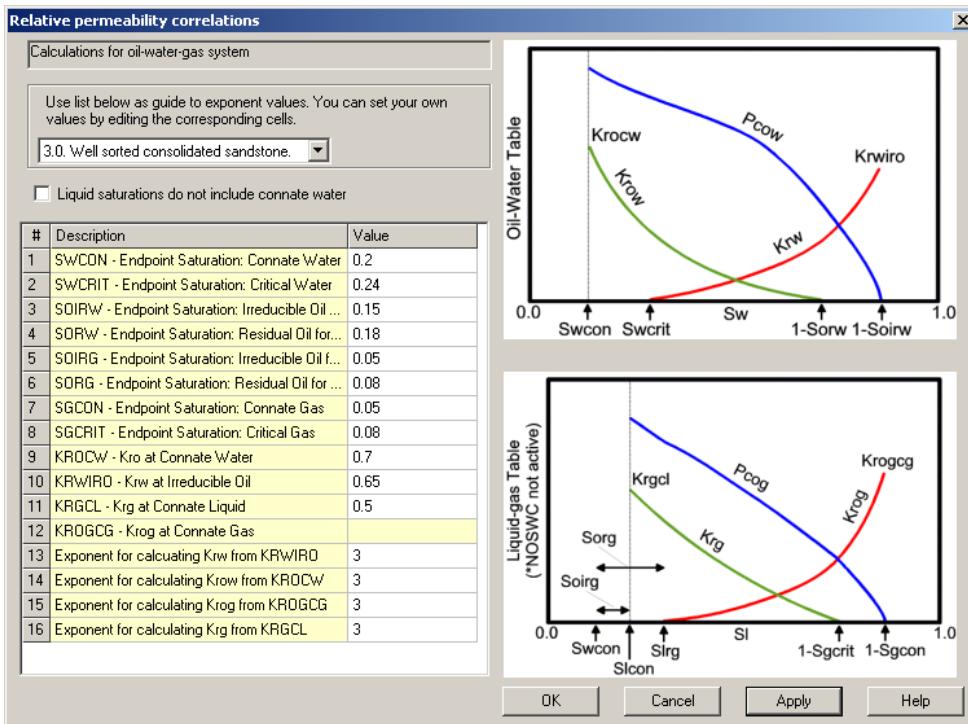


The **Tools** button in the upper left corner of the **Relative Permeability Tables** tab is used to access the available tools. When this button is clicked a drop-down menu with a list of the available tools will be displayed:



Tool: Generate Tables Using Correlations

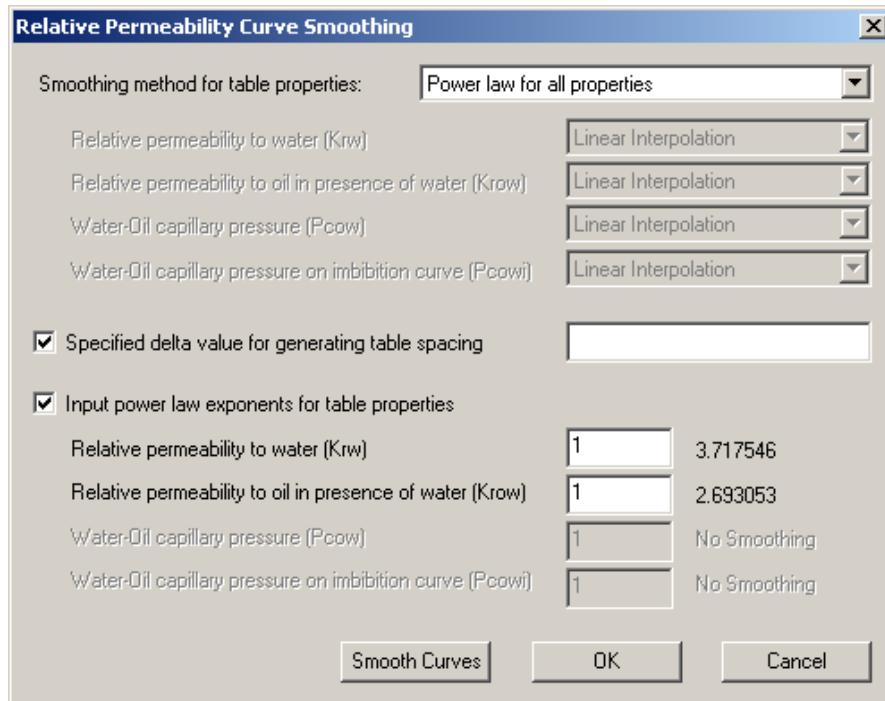
To access the dialog box for generating tables from analytical equations – correlations, click the **Tools** button then select **Generate Tables Using Correlations**. The **Relative permeability correlations** dialog box is displayed:



The **Relative permeability correlations** dialog box displays relative permeability curves to help you identify the meaning of the required fields. Builder will verify that entered values are appropriate for the analytical equations. After entering required values and clicking **OK** or **Apply**, Builder will generate the appropriate relative permeability tables. The curves will be displayed in Builder's plot view. Appendix F of the *STARS Users' Guide* shows the equations for these correlations.

Tool: Smooth Table

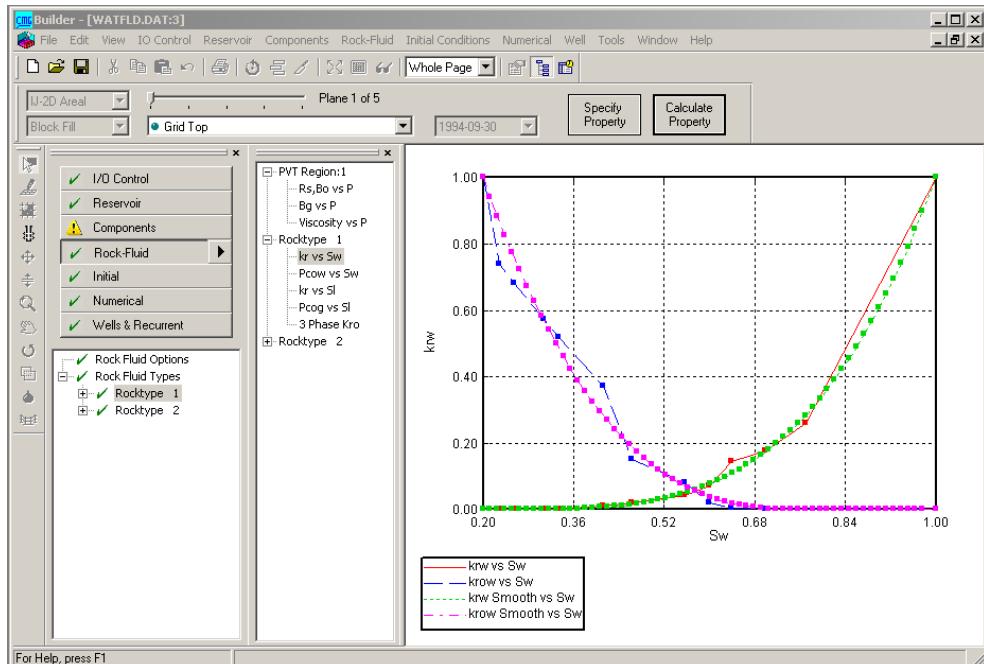
To open the **Relative Permeability Curve Smoothing** dialog box, click the button then select **Smooth Table**.



Note: Relative permeability curves can be smoothed with all the smoothing capabilities build in CMG's STARS simulator.

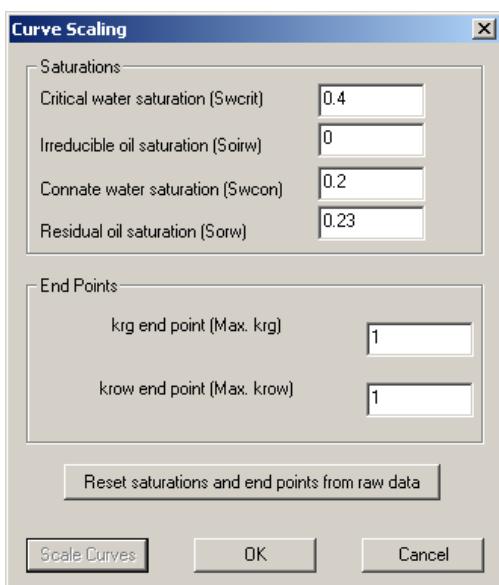
When the **Relative Permeability Curve Smoothing** dialog box is opened, the plot view will display relative permeability curves for both before and after the smoothing operation. You can modify the default settings in the **Relative Permeability Curve Smoothing** dialog box at any time. After clicking the **Smooth Curves** button, the plot view will be updated to display the smoothed curves.

When you are satisfied with the smoothed curves, click **OK** to exit the dialog box and apply the changes to the table in the **Rock Types** dialog box. Alternatively, click **Cancel** to cancel the changes in the current **Smooth Curves** session.



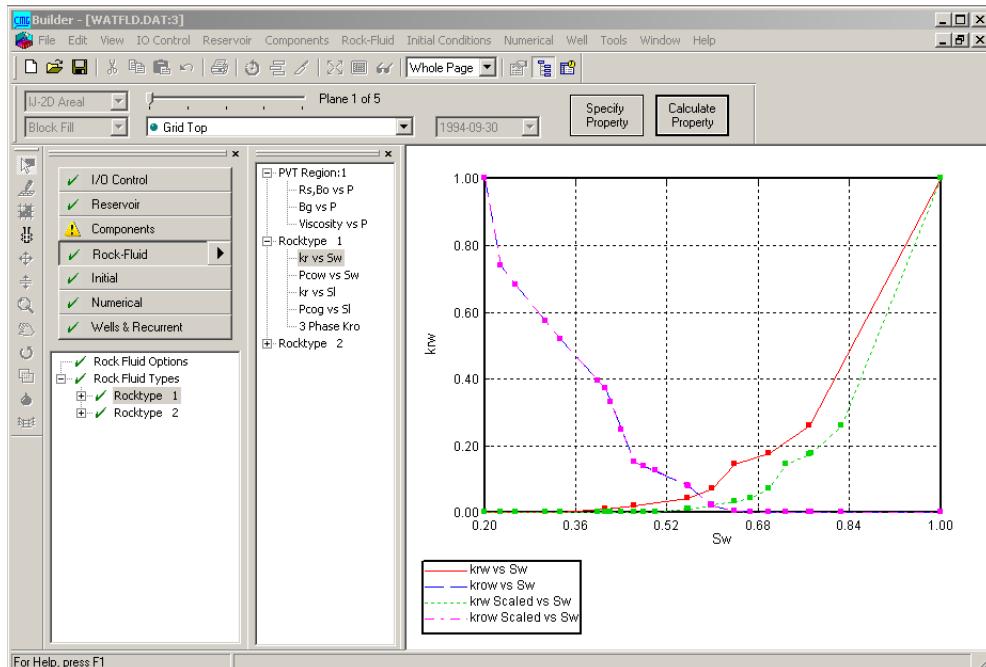
Tool: Curve Scaling

To open the **Curve Scaling** dialog box, click the button then select **Curve Scaling**.



When the **Curve Scaling** dialog box is opened, the plot view will display relative permeability curves both before and after the curve scaling. You can modify the default settings in the **Curve Scaling** dialog box at any time. After clicking the **Scale Curves** button, the plot view will be updated to display the scaled curves.

When you are satisfied with the scaled curves, click **OK** to exit the dialog box and apply the changes to the table in the **Rock Types** dialog box. Alternatively, click **Cancel** to cancel the changes in the current **Curve Scaling** session.



Modifying Tables from the Plot View

Rather than modifying numerical values in a table, Builder allows you to change your relative permeability curves directly from the plot view, by clicking and dragging the points up or down.

To edit the curves, the **Rock Types** dialog box must be open. The **Rock Types** dialog box can be opened directly from the menu or main tree view, as previously discussed.

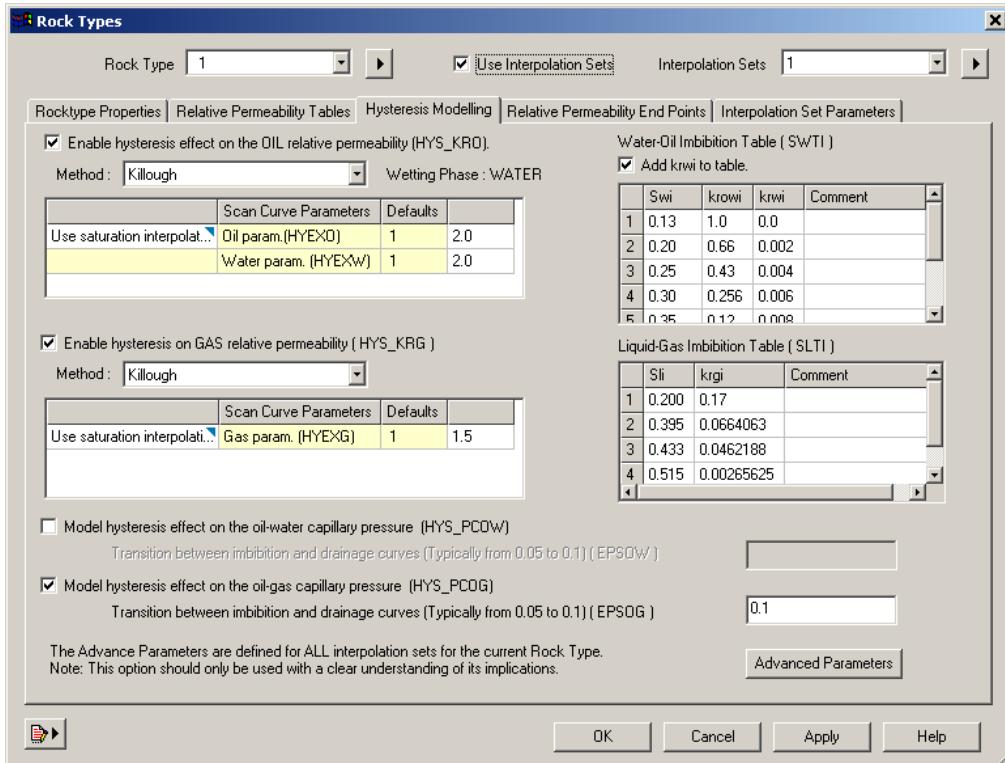
Alternatively, you can double-click on a curve directly on Builder's plot view. This will open the **Rock Types** dialog box with the appropriate rock type preselected. You can then edit points on the curve by clicking and dragging them.

You can click **OK** or **Apply** in the **Rock Types** dialog box at any time to permanently apply the changes to the dataset. Alternatively, you can click **Cancel** if you decide that your changes are not appropriate.

Setting Hysteresis Modelling

In STARS, it is possible to apply hysteresis to either of the relative permeabilities, capillary pressures, or both simultaneously. See **Hysteresis Parameters** section in the “Rock-Fluid Data” chapter of *STARS User’s Guide* for details.

The hysteresis option and its related parameters can be entered on the **Hysteresis Modelling** tab:



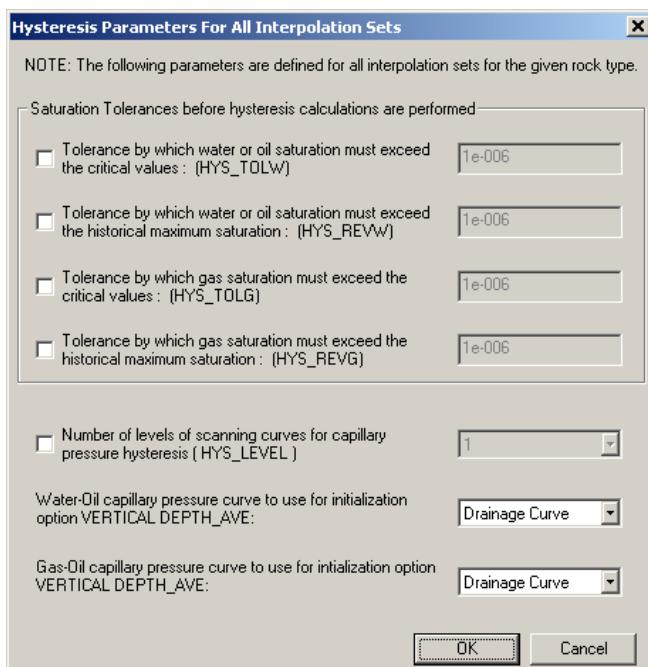
The water-oil hysteresis is only available when the wetting phase is either “Water Wet” or “Oil Wet” which can be set on the **Rocktype Properties** tab. Depending on the options selected, the interface will enable or disable controls appropriately.

Once you have selected to use the hysteresis option, you can then select which method to use. Selecting different methods will change the corresponding information grid and the values which can be entered. Refer to the *STARS User’s Guide* for specific meaning of the different variables.

The imbibition tables entered on the right hand side can be viewed in the plot view. They are displayed together with the drainage curves entered in the **Relative Permeability Tables** tab. The values of the imbibition tables can be modified by changing the values on the table grid or by clicking and dragging the desired point in the plot view.

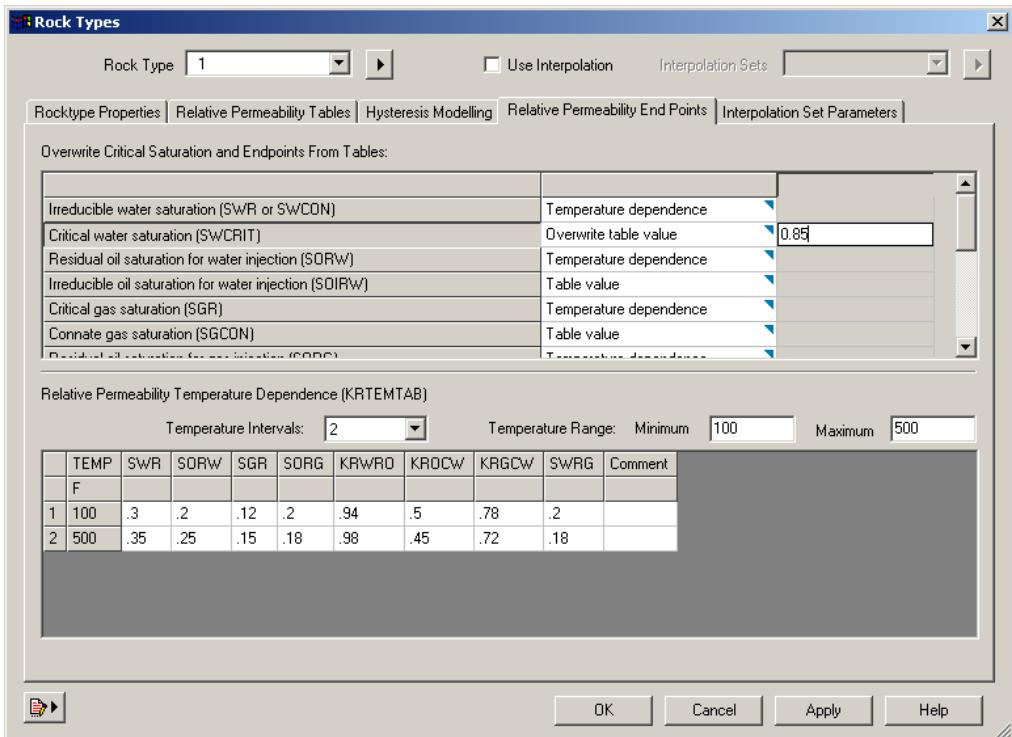
The fields available in the **Hysteresis Modeling** tab will vary with **Rock Type** and **Interpolation Set**; however, the parameters available in the **Hysteresis Parameters For All Interpolation Sets** dialog box, which is displayed by clicking the **Advanced Parameters** button, will only vary with **Rock Type**.

Note: The values in the **Hysteresis Parameters For All Interpolation Sets** dialog box should only be modified if you are certain of their implications. Refer to the **Hysteresis Parameters** section of the “Rock-Fluid Data” chapter in the *STARS User’s Guide* for further details.



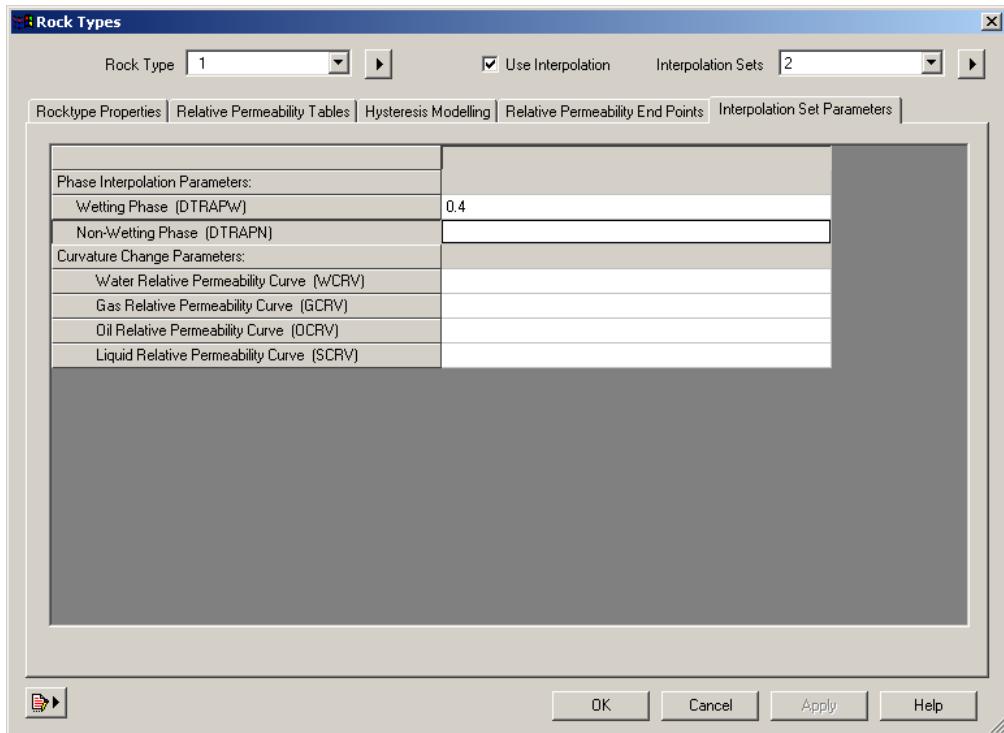
Setting Relative Permeability End Points

In STARS, you can overwrite critical and connate saturations and endpoints from the tables. Alternatively, you can specify temperature dependence for critical saturations and endpoints. These settings can be found on the **Relative Permeability End Points** tab in the **Rock Types** dialog box:



Setting Interpolation Set Parameters

When working with multiple interpolation sets, interpolation parameters can be set for each interpolation set, specifically phase interpolation parameters and curvature change parameters. For further details, refer to the **Interpolation Set Number and Parameters** section in the “Rock-Fluid Data” chapter of the *STARS User’s Guide*. These settings can be found on the **Interpolation Set Parameters** tab of the **Rock Types** dialog box:

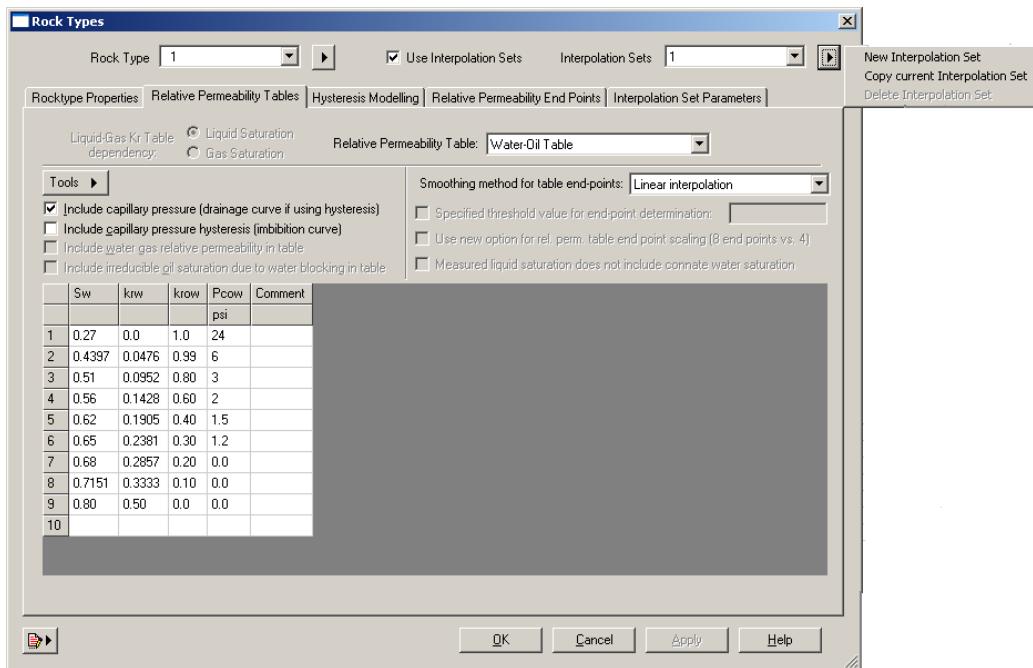


Working with Interpolation Sets

The STARS **Rock Types** dialog box includes controls for working with multiple interpolation sets. If you select **Use Interpolation**, you can create, copy, or modify multiple interpolation sets. The data fields on the tabs of the **Rock Types** dialog box will then include fields for parameters that only apply when multiple interpolation sets are defined.

Note: When a rock type has multiple interpolation sets, the **Rocktype Properties** tab will continue to display parameters that apply to all interpolation sets for the selected rock type. The other tabs will reflect relative permeability data and interpolation set parameters for the selected interpolation set. If you make any changes, Builder will require you to save changes if you try to select a different interpolation set, or rock type.

Also, when working with multiple interpolation sets, Builder's plot view will display relative permeability curves for the currently displayed interpolation set in the **Rock Types** dialog box:



Modifying an Interpolation Set

To modify an existing interpolation set, open the **Rock Types** dialog box then select the desired rock type and interpolation set from the two drop down lists at the top of the dialog box. The data displayed in the tabs will reflect settings for the selected rock type and selected interpolation set.

When editing data for a particular interpolation set, changes made to the data will not be applied until you select **OK** or **Apply**. On changing the selected rock type or interpolation set, you will be asked if you wish to apply the changes before continuing.

While changes to the relative permeability tables are not permanently applied to the dataset until you direct Builder to do so, the plot view window will display relative permeability curves derived from the table data in the selected rock type and selected interpolation set from the **Rock Types** dialog box. This will permit you to experiment with the table data and cancel at any time.

Adding an Interpolation Set

A new interpolation set may be added to the list of available interpolation sets by clicking on the  button then selecting **New Interpolation Set** from the drop-down menu.

Alternatively, a new interpolation set can be created as a duplicate copy of an existing interpolation set by selecting the interpolation set you want to copy from the drop-down list then clicking the  button and selecting **Copy Current Interpolation Set**.

Deleting an Interpolation Set

An existing interpolation set can be deleted by first selecting the desired interpolation set from the drop-down list, then clicking the  button and selecting **Delete Interpolation Set** from the drop-down menu.

Note: You cannot cancel from this operation; therefore, Builder will issue a message to make sure that you wish to delete the currently selected interpolation set.

If an interpolation set is removed, the remaining interpolation sets are renamed so that the interpolation set numbers are sequential.

Note: Builder will prevent you from deleting the last remaining interpolation set, since the simulator requires a minimum of one rock type defined for the dataset. Also, on exiting and returning to the **Rock Types** dialog box, Builder will assume that interpolation is not being used, in the event that only one interpolation set has been defined for the currently displayed rock type.

Initialization - IMEX

Overview

The Initial Conditions section allows you to enter information regarding the state of the reservoir at initial time. To some extent the required information for this section depends on the information entered in the Component Properties section (see [Fluid Model - IMEX](#)).

Before data for the Initial Conditions section can be entered, a fluid component model must be selected.

Additional information that can be entered in this section includes the capillary-gravity method of calculating vertical equilibrium, bubble point and dew point pressure, light oil volume fraction, reference depth and pressure, three phase contact depths, and datum depth.

The initialization interface can be accessed in two ways, either through the main menu **Initial Conditions | Initialization Settings** or through **Initial Conditions** in the tree view. When accessing initial conditions interface from the tree view, double-click the desired tree item and the appropriate interface will be displayed:

The **Initial Conditions** section provides two main interfaces:

- **Standard Interface:** This interface is a single form which provides a quick and easy way of entering the most commonly used initialization parameters (see [Standard Interface](#)).
- **Advanced Interface:** This interface consists of three tabs covering all reservoir initialization-relevant parameters to utilize the full capabilities of IMEX (see [Advanced Interface](#)).

Depending on the initialization information that is entered, one of the two interfaces will display; however, the **Advanced Interface** can always be accessed from the **Standard Interface** by clicking the **Advanced** button.

Both the Standard and Advanced interfaces have four buttons at the bottom of their dialog boxes:

- **OK:** Closes the interface and saves any changes that were made.
- **Cancel:** Closes the interface but does not save any changes that were made during the session unless the **Apply** button was clicked.

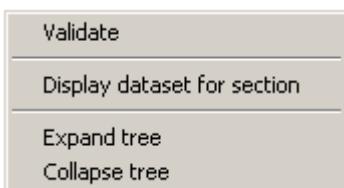
- **Apply:** Saves any changes that were made but does not close the interface.
- **Help:** Opens help information related to the current focus. Pressing the **F1** key will also bring up the help information.

Selecting the comments  button at the bottom far left will allow you to view and edit comments related to specific interface items.

The current status of the Initial Conditions information is indicated by icons under the tree view button labeled **Initial** and on the different items on the tree view.

Tree View Items and Menu

Right-clicking the **Initial Conditions** area in the tree view will display the following menu:



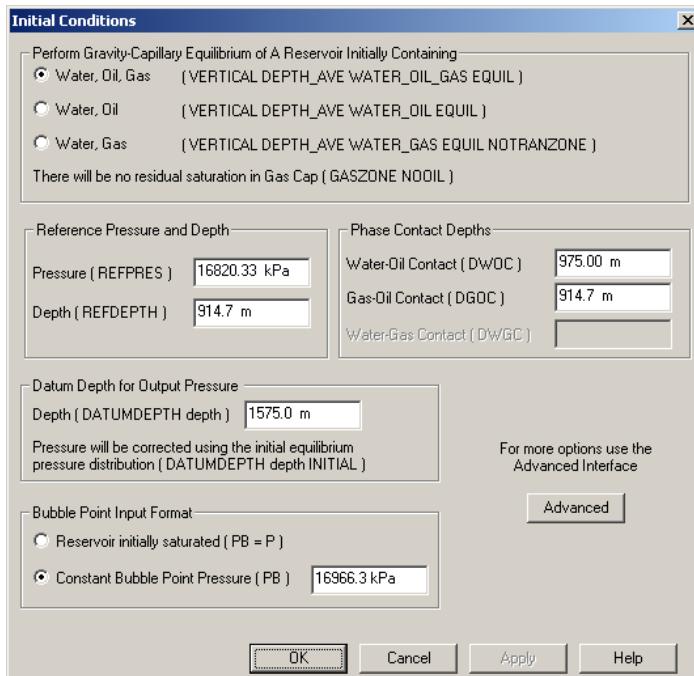
- **Validate:** Displays a list of all the warning and error messages.
- **Display dataset for section:** Displays the Initial Condition information as it will be stored in the dataset.
- **Expand/Collapse tree:** Expands or collapses the branch for the different items on the tree.

Double-clicking an item on the tree will display the relevant interface.

Standard Interface

The **Standard Interface** is displayed when a new dataset is created or if the following conditions are satisfied:

- For the capillary-gravity method of calculating vertical equilibrium, the saturation assigned to a given grid block is the average over the grid block volume of the saturations.
- A single PVT Region is defined.
- Datum Depth specification used the option that the initial equilibrium pressure distribution in the reservoir will be used to calculate the corrected datum pressures.
- Bubble Point Pressure is a constant define by a grid property array.



Only the constant **Bubble Point** pressure array format can be entered using this interface. The radio buttons for the **Bubble Point** pressure provide short cut to defining the constant bubble point pressure array. This property can also be entered in the Reservoir tree view button under the Array Properties tree item (see [Reservoir Description](#)).

To access other properties such as **Dew Point** pressure, **Light Oil Volume** fraction or depth dependent tables of these properties, including **Bubble Point**, the Advanced interface needs to be used.

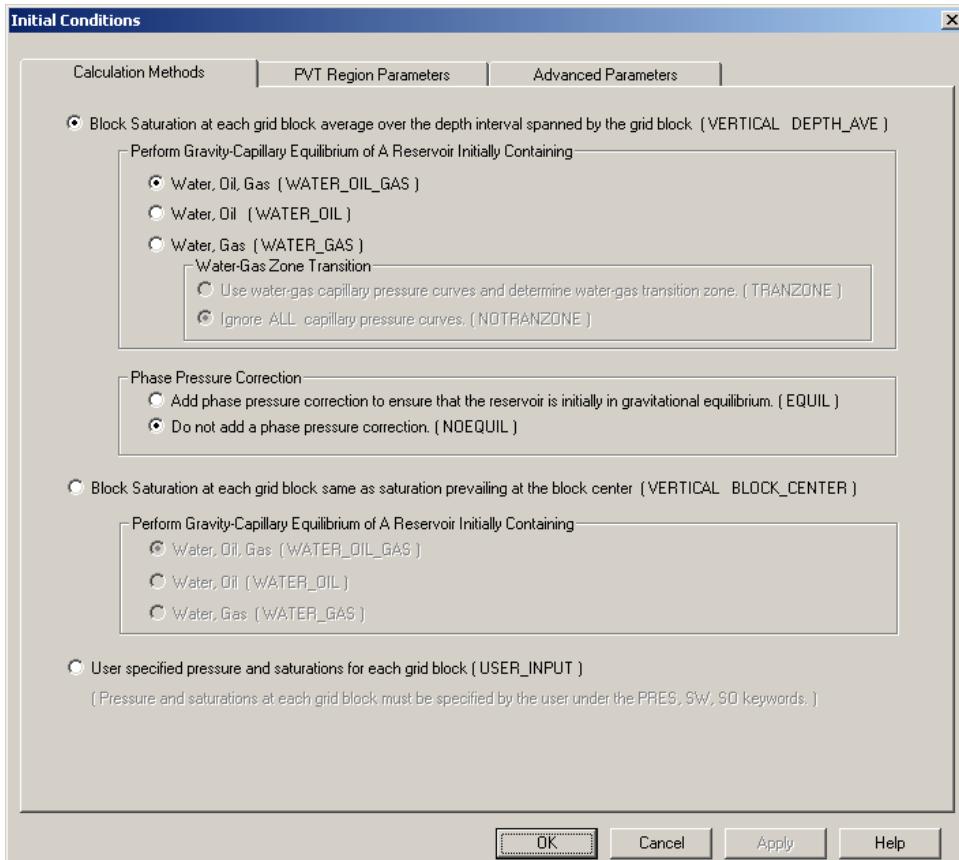
The **Advanced Interface** can be accessed from the **Standard Interface** by clicking the **Advanced** button.

Advanced Interface

The **Advanced Interface** provides access to all available initialization parameters, including vertical equilibrium calculation using the block center option, use of multiple initialization regions, and the use of dew point pressure and light oil volume fraction.

The **Advanced Interface** has three tabs – **Calculation Methods**, **PVT Region Parameters** and **Advanced Parameters**.

Calculation Methods tab



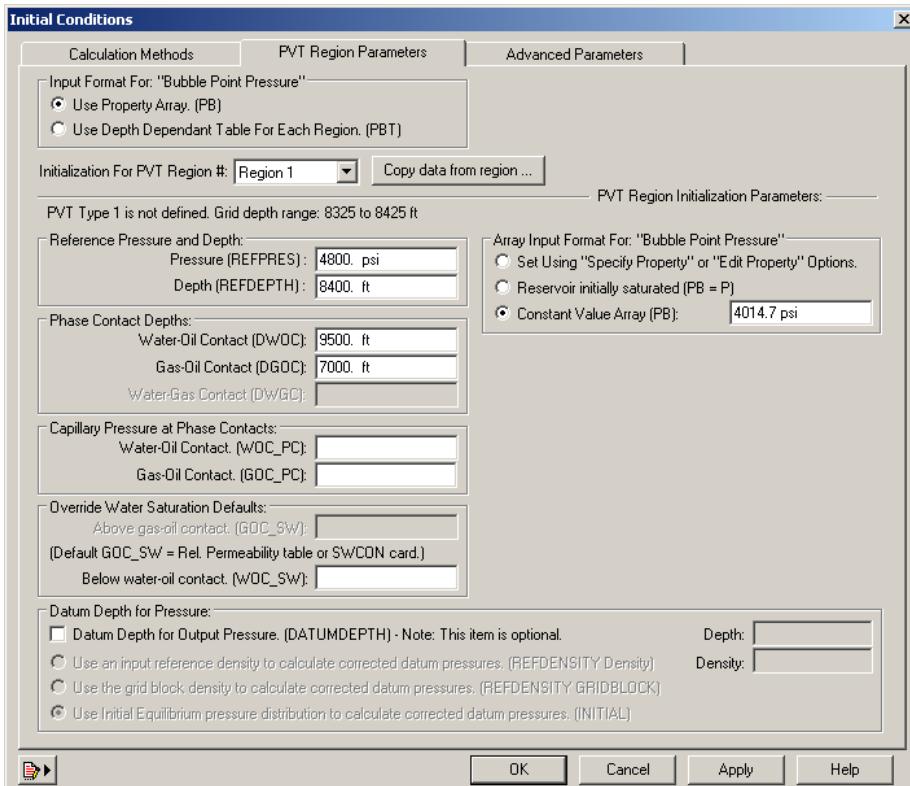
The **Calculation Methods** tab allows:

- Selecting the capillary-gravity method for calculating vertical equilibrium, whether block centered, block averaged or user-defined.
- Selecting the initial phases (water and oil, water and gas, or water, oil and gas) needed to perform gravity-capillary equilibrium initialization. Which of these options are enabled will depend on the fluid model selected in the **Component Properties** section (see [Fluid Model - IMEX](#) for further information).

Depending on the options selected on the **Calculation Methods** tab, parameters on the other tabs will be enabled or disabled based on what is allowed or not allowed for the different options.

If you select **User specified pressure and saturations for each grid block**, specification of initial pressures and saturations is required. These properties must be entered in the Reservoir tree view button under the **Array Properties** tree item, using the **Specify Property** and **Calculate Property** options (see [Reservoir Description](#) for further information).

PVT Region Parameters tab



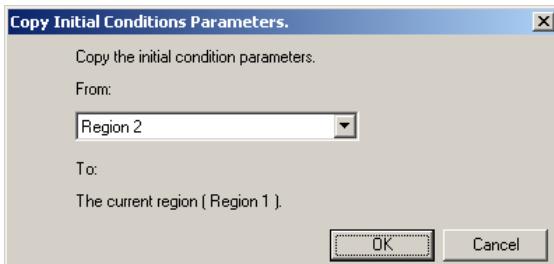
The **PVT Region Parameters** tab holds most of the initialization parameters for the different PVT Regions. The PVT region information includes a reference depth, a reference pressure and three phase contact depths, datum depth information, the capillary pressure at the phase contacts, and overriding water saturation default. The parameters for the different regions can be viewed by selecting the desired region in the **Initialization For PVT Region #** box.

For reference, the sentence just below the **Initialization For PVT Region #** box describes the number of blocks defined for the selected region and also the depth range of the region. This notice is related to the **PVT Type** grid array property, which can be viewed in the **Reservoir** tree view tab. If a PVT Type property is not defined for a PVT region then the notice will be updated to reflect that.

The number of PVT initialization region information that needs to be entered depends on the number of PVT regions defined in the Fluid Model section.

Note: The initialization regions cannot be added or deleted through this interface. To add or delete PVT regions, you must use the IMEX PVT Regions interface, found in the **Components** tree view button (see [Fluid Model - IMEX](#)). Adding or deleting PVT Regions will appropriately update the list of initialization regions found in this interface.

Initialization information can be copied from a different region to the current region being edited. To copy the information from a different region, click the **Copy data from region** button. The **Copy Initial Conditions Parameters** dialog box is displayed. This interface allows you to select the region from which you want to copy information.



The **PVT Region Parameters** tab also lets you enter information for **Dew Point** and **Bubble Point** pressure as well as **Light Oil Volume Fraction** either for depth-dependent tables or constant grid array property format. The properties that are available will depend on which Fluid Model was selected (for further information, refer to the “Component Properties” chapter of the *IMEX User’s Guide* and [Fluid Model - IMEX](#) in this manual).

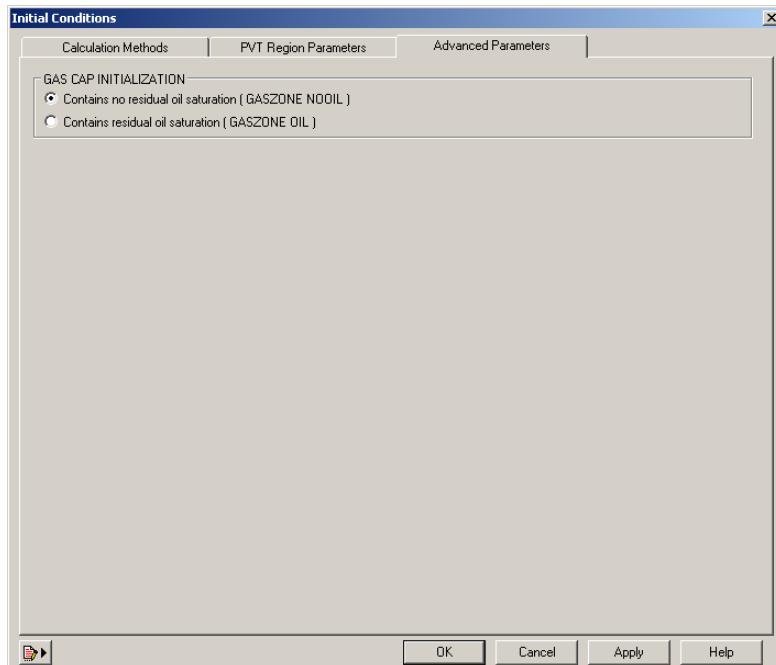
The input format for these properties, whether table or array format, can be selected at the top of the tab by making the appropriate selection. This will automatically update the options displayed on the left side of the tab.

When using the property array format, the corresponding property will be updated only as a constant property. This update will be done only to the corresponding region defined in the **PVT Type** property. If no corresponding **PVT Type** property has been defined for a given region then the parameters **Dew Point Pressure**, **Bubble Point Pressure** or **Light Oil Volume Fraction** cannot be properly updated. If the array format for these parameters is more complicated than the constant format, the property arrays must then be specified through the **Reservoir** tree view button under the **Array Properties** tree item, using the **Specify Property** and **Calculate Property** or **Edit Property** options (see [Reservoir Description](#)).

Notes:

1. Under certain circumstances when entering either the Standard or Advanced interface and there are no PVT Type define, if there is only one PVT region then a PVT Type which encompasses the entire grid, will automatically be created.
2. Only one format (either array or table) is allowed for each of **Dew Point** and **Bubble Point** pressure and **Light Oil Volume Fraction**. Selecting one format and saving the changes will cause the other information to be deleted. Thus selecting array format will delete the table information entered, and selecting the table format will delete the array information.

Advanced Parameters tab



The **Advanced Parameters** tab holds parameters and options that are less often used, for example, residual oil saturation.

Data Validation

When a dataset is initially read in, any errors encountered in this section will be noted by error or warning messages. As well, the corresponding status icon will be displayed on the **Initial Conditions** button of the tree view. You can then enter the **Initial Conditions** interfaces and review the information. Required changes can then be made and saved.

The information entered in the **Standard Interface** or the **Advanced Interface** is once again validated when the **Apply** or **OK** buttons are selected.

Under certain circumstances if there is information needed which has not been entered, you are not allowed to exit the interface until the required information is entered. In these cases, a list of relevant error messages is given.

As well there is some basic validation done when values are entered into edit boxes; for example, if a numerical value is expected and a non-numerical value is entered, an error message is displayed. Also, values entered will be displayed with their respective current working units. Values may be entered as a number only or as a value followed by a space and the units used. Values entered with units different than the current working units will be converted and the converted value will be displayed with the current working units.

Initialization – GEM

Overview

The Initial Conditions section allows you to enter information regarding the state of the reservoir at initial time. To some extent the required information for this section depends on the information entered in the Component Properties section (see [Fluid Model - GEM](#)).

Additional information that can be entered in this section includes the capillary-gravity method of calculating vertical equilibrium, bubble point and dew point pressure, light oil volume fraction, reference depth and pressure, three phase contact depths, and datum depth).

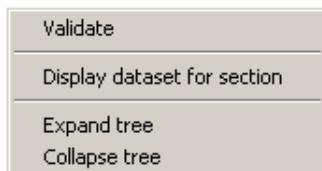
Other information includes gas plant tables, specification of non-EOS stream density calculations and well separators.

The initialization interface can be accessed in two ways, either through the main menu **Initial Conditions | Initialization Settings** or through **Initial Conditions** in the tree view. When accessing initial conditions interface from the tree view, double-click the desired tree item and the appropriate interface will be displayed.

The current status of the Initial Conditions information is indicated by the icons under the tree view button labeled **Initial** and on the different items on the tree view.

Tree View Items and Menu

Right-clicking the **Initial Conditions** tree view section will display the follow menu.



- **Validate:** Displays a list of all the warning and error messages.
- **Display dataset for section:** Displays the Initial Condition information as it will be stored in the dataset.
- **Expand/Collapse tree:** Expands or collapses the branch for the different items on the tree.

Double-clicking an item on the tree will display the relevant interface.

Entering and Editing Initial Conditions

The **Initial Conditions** dialog box provides access to all the non-array initialization parameters. Some of the parameters and options include the different capillary-gravity methods of calculating vertical equilibrium, definition of multiple initialization regions, oil zone and gas cap compositions, and variation of composition with depth.

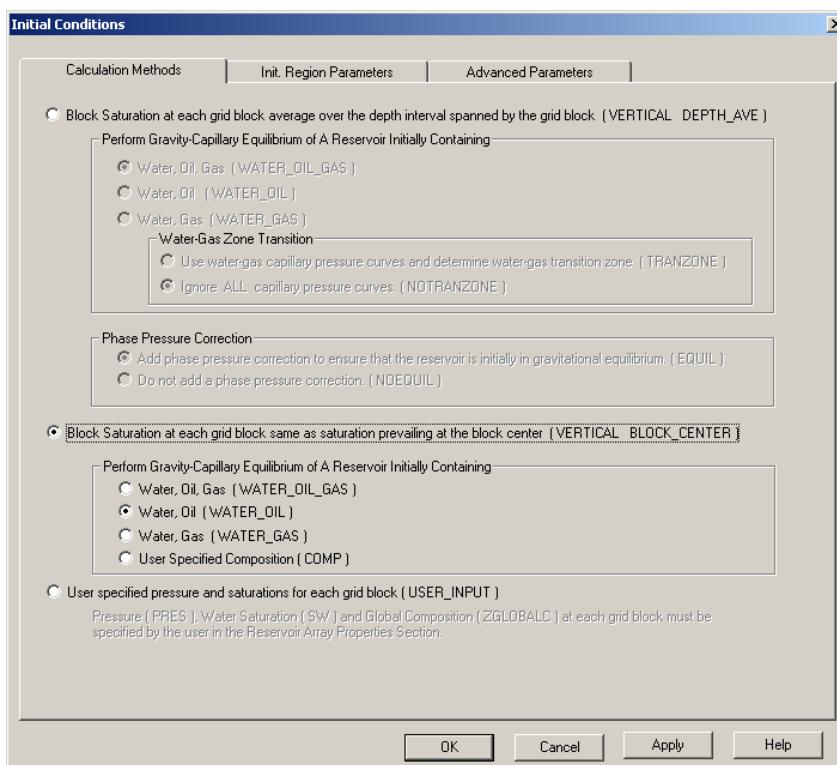
The **Initial Conditions** dialog box has three tabs – **Calculation Methods**, **Init. Region Parameters**, and **Advanced Parameters**.

The **Initial Conditions** dialog box has four buttons at the bottom:

- **OK**: Closes the interface and saves any changes that were made.
- **Cancel**: Closes the interface but does not save any changes made unless **Apply** was clicked first.
- **Apply**: Saves any changes that were made but does not close the interface.
- **Help**: Displays help information about the item with the current focus. Pressing the **F1** key has the same effect.

Use the **Comments**  button at the bottom far left to view and edit comments related to specific interface items.

Calculation Methods



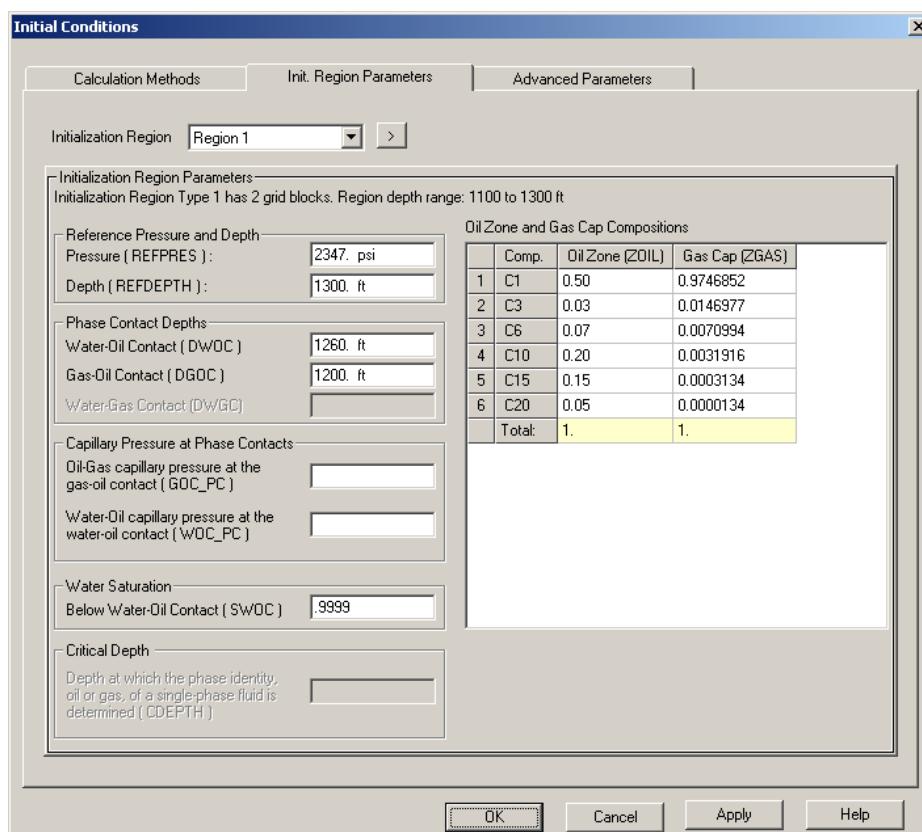
The **Calculation Methods** tab allows:

- Selection of the capillary-gravity method of calculating vertical equilibrium, whether block centered, block averaged or user-defined.
- Selection of the initial phases (water and oil, water and gas, or water, oil and gas, or user-specified composition) needed to perform gravity-capillary equilibrium initialization.

Depending on the options selected in the **Calculations Method** tab, parameters on the other tabs will be enabled or disabled based on what is allowed of not allowed for the different options.

If you select **User specified pressure and saturation for each grid block**, specification of initial pressures and saturations, and the initial grid block global composition on a component-by-component basis, are required. These properties must be entered in the **Reservoir** tree view button under the **Array Properties** tree item, using the **Specify Property** and **Calculate Property** options (see [Reservoir Description](#)).

Init. Region Parameters



The **Init. Region Parameters** tab holds most of the initialization parameters for the different initialization regions. The initialization region information includes reference depth, reference pressure and three phase contact depths, capillary pressure at phase contacts, water saturation, critical depth, oil and gas capillary composition tables, and a variation of composition with depth table (only when using the COMP option).

The parameters for the different regions can be viewed by selecting the desired region in the **Initialization Region** box.

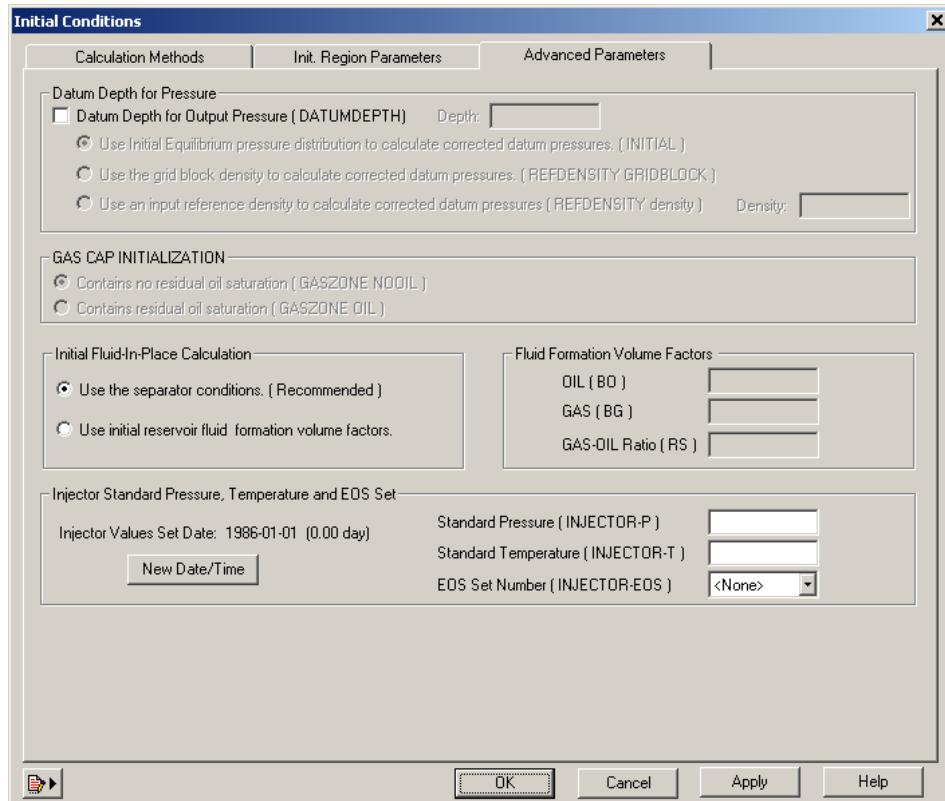
For reference, a sentence just below the **Initialization Region** box describes the number of blocks defined for the selected region and also the depth range of the region. This notice is related to the **Initialization Region Type** grid array property, which can be viewed in the **Reservoir** tree view tab. If an Initialization Region Type property is not defined for an Initialization Region then the notice will be updated to reflect that.

Clicking the right- arrow key  will display a menu with the following selections.

- a) **Add New Region:** Creates a new region with all fields empty.
- b) **Add New Region By Copying:** Creates a new region by copying information from an existing region.
- c) **Copy Values From A Region:** Copies information from another region to the region that is currently displayed.
- d) **Delete Current Region:** Deletes the region that is currently displayed.

Note: When adding or deleting Initialization Regions make sure to properly update the **Initialization Region Type** property array, which can be found in the **Reservoir** tree view button under the **Array Properties** tree item. Use the **Specify Property**, **Calculate Property** or **Edit Property** options to change the values (see [Reservoir Description](#)).

Advanced Parameters



The **Advanced Parameters** tab holds parameters and options that are less often used, including the capillary pressure at the phase contacts, residual oil saturation, initial fluid-in-place calculation methods, and injector standard pressure, temperature and EOS set.

Separators

There are two types of separators.

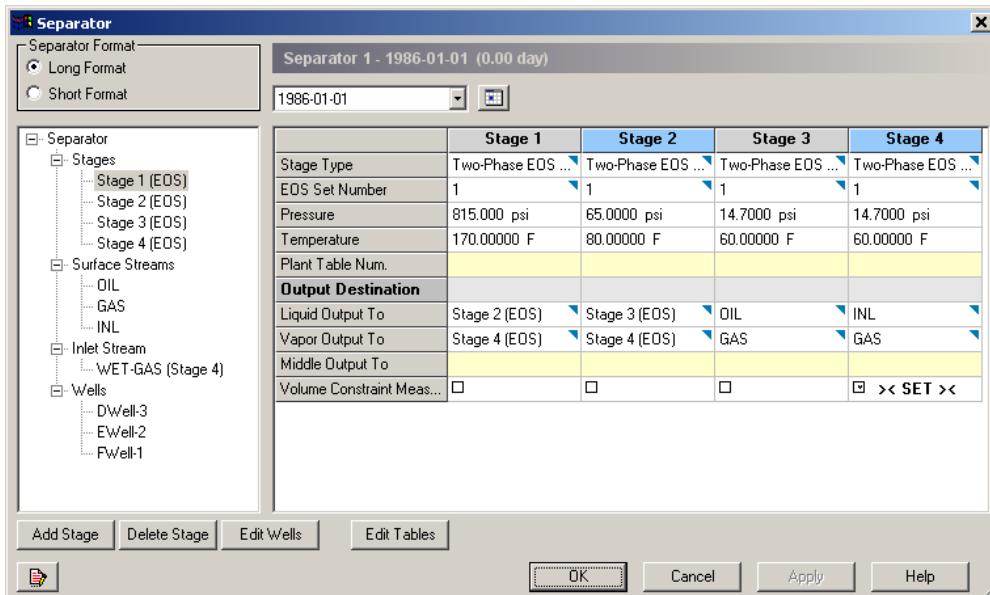
- **Default Separator:** Defined in the initial conditions section and used for initial fluid-in-place calculations.
- **Well Separators:** Defined in the well and recurrent section. These are separators through which the produced fluids are assigned surface stream compositions, molar rates, and volumetric rates.

To access any of these separator options, select the appropriate tree view button. For the Default Separator select **Initial Conditions**. For the Well Separators, select **Well & Recurrent**. On the tree view, you will then find the corresponding tree view items.

To create a new separator, double-click on the **Default Separator** tree item for **Initial Condition** or **Separators** for the **Well & Recurrent** section. To edit an existing separator, double-click the specific separator. In either case, the **Separator** dialog box is displayed.

Note: Right-clicking the tree view item will display a menu that allows you to add, delete, edit and validate the separators.

The **Separator** dialog box is shown below:



By clicking different items on the tree, the right side will change to allow the appropriate items to be edited. **Separator Format** (that is, long or short) can be selected in the top left.

With the Long Format, different stages can be added or deleted by clicking the **Add Stage** or **Delete Stage** buttons.

The separator streams can be modified through the **Separator** dialog box, as follows. Select a stream from the tree view and the right side of the dialog box will change, revealing the different streams available and allowing you to edit them.

Modify the well list by clicking the **Edit Wells** button. This will display a well-selection dialog box.

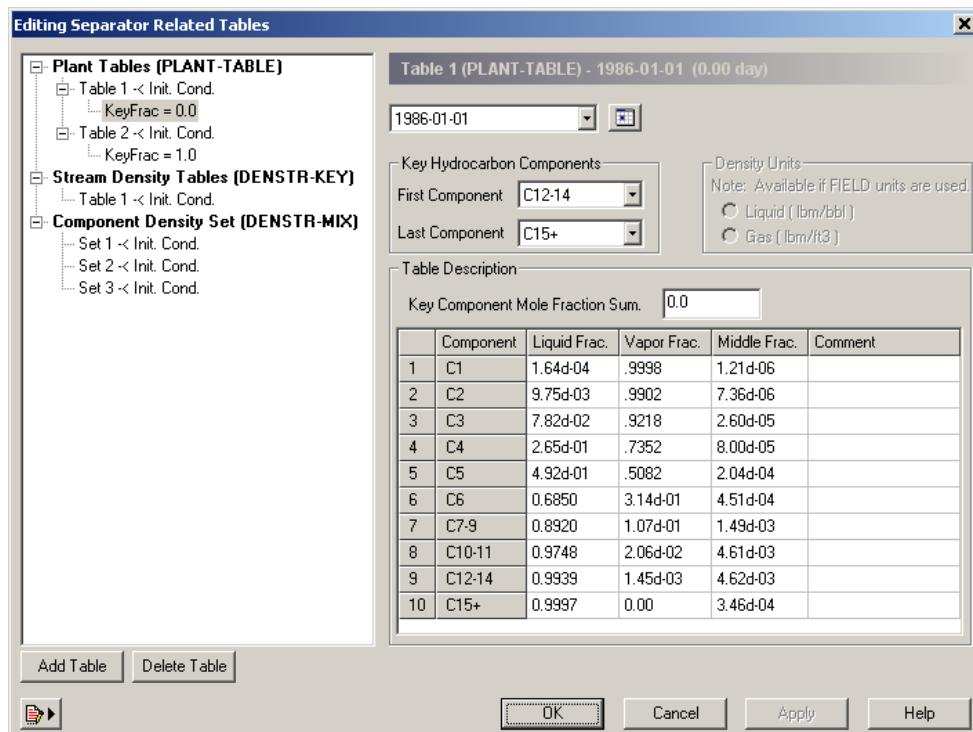
Click **Edit Tables** to create or edit **Plant Tables** for the stages, and **Density Tables** and **Component Mass Density Sets** for the stream. The **Editing Separator Related Tables** dialog box, described below, is displayed.

The date for which the separator is set may be selected through date combo box. If the date does not appear in the list, a new date can be entered through the date  button.

Note: For the **Default Separator**, the date selection combo box and date button are not enabled since this separator belongs only in the **Initial Conditions** section.

Gas Plant Tables

Click **Edit Table** in the **Separator** dialog box to create or edit **Plant Tables** for the stages and **Density Tables** and **Component Mass Density Sets** for the stream interface (refer to [Separators](#)). The **Editing Separator Related Tables** dialog box is displayed:



Selecting different items on the tree will update the **Editing Separator Related Tables** dialog box, allowing you to modify the item.

To add or delete tables or sets, select the appropriate item on the tree then click the **Add Table** or **Delete Table** button as appropriate.

The date for which the table is set may be selected through date combo box. If the date does not appear in the list, a new date can be entered through the date  button.

Specification of Non-EOS Stream Density Calculations

The table and component density sets, which allow the computation of the mass density of surface streams of separators, can be edited using the above interface, in the **Gas Plant Tables** section. The interface allows you to edit the **Stream Density Tables** and **Component Density Sets**.

Data Validation

When a dataset is initially read in, errors that are encountered in the section are indicated by error or warning messages. As well, the corresponding status icon will be displayed on the **Initial** button of the tree view. You can enter the **Initial Conditions** interfaces and review the information. Required changes can be made and saved.

The information entered in the **Standard** or **Advanced** interfaces is again validated when the **Apply** or **OK** buttons are selected.

Under certain circumstances, if there is information needed which has not been entered, you are not allowed to exit the interface until the required information is entered. In these cases, a list of relevant error messages is given.

Basic validation is performed when values are entered; for example, if a numerical value is expected and a non numerical value is entered, an error message will be displayed. As well, values entered will be displayed with their respective current working units. Values may be entered as a number only or as a value followed by a space and the units used. Values entered with units different than the current working units will be converted and the converted value will be displayed with the current working units.

Initialization – STARS

Overview

The **Initial Conditions** section allows you to enter information regarding the state of the reservoir at initial time.

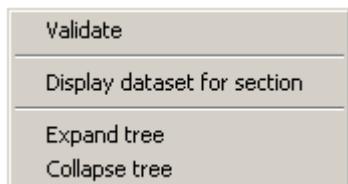
Additional information that can be entered in this section includes the capillary-gravity method of calculating vertical equilibrium, initial reservoir saturations, reference depth and pressure and three phase contact depths.

The initialization interface can be accessed in two ways, either through the main menu **Initial Conditions | Initialization Settings** or through the tree view button labeled **Initial**. When accessing initial conditions interface from the tree view, double-click on the desired tree item and the appropriate interface will be displayed.

Tree View Items and Menu

The current status of the Initial Conditions information is indicated by icons under the tree view button labeled **Initial** and on the different items on the tree view.

Right-clicking the **Initial Conditions** tree view section will display the follow menu:



- **Validate:** Displays a list of all the warning and error messages.
- **Display dataset for section:** Displays the Initial Condition information as it will be stored in the dataset.
- **Expand/Collapse tree:** Expands and collapses the branch for the different items on the tree.

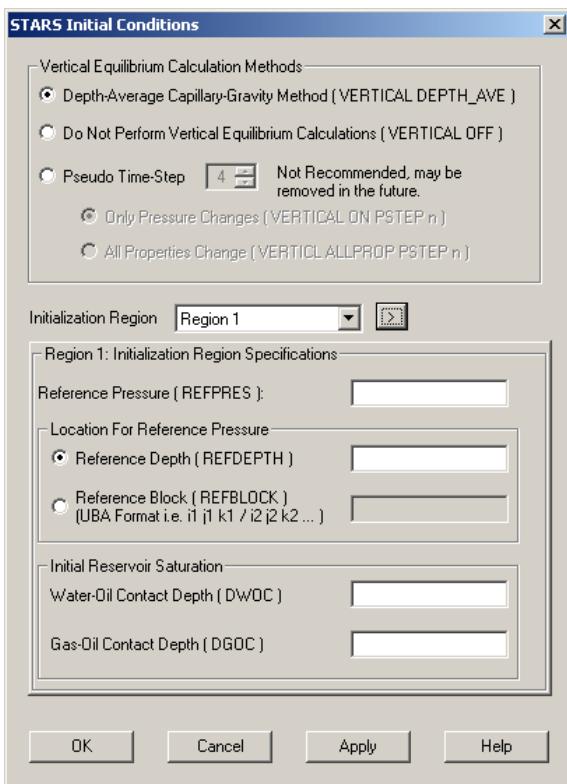
Double-clicking an item on the tree will bring up the relevant interface.

Entering and Editing Initial Conditions

Through the **Stars Initial Conditions** dialog box, you can edit non-property-array initialization parameters such as capillary-gravity methods of calculating vertical equilibrium, initial reservoir saturations, reference depth and pressure, and three phase contact depths.

To enter or edit property arrays such as initial saturation (water, oil, gas), phase mole fractions, bubble point pressure, solid concentration, pressures and temperature, you must add the properties in the **Reservoir** tree view tab under the **Array Properties** tree item, using the **Specify Property**, **Calculate Property** or **Edit Property** options (see [Reservoir Description](#)).

To view the non-array parameters for a given region, select the desired region in the **Initialization Region** combo box:



Clicking the right-arrow key  displays a menu with the following choices:

- **Add New Region:** Creates a new region with all fields empty.
- **Add New Region By Copying:** Creates a new region by copying the information from an existing region.
- **Copy Values From A Region:** Copies information from another region to the currently displayed region.
- **Delete Current Region:** Deletes the currently displayed region.

Note: When adding or deleting Initialization Regions make sure to properly update the property array **Initialization Set Number**, which can be found in the **Reservoir** tree view button under the **Array Properties** tree item. Use the **Specify Property**, **Calculate Property** or **Edit Property** options to change the values (see [Reservoir Description](#)).

There are four buttons at the bottom of the **Initial Conditions** dialog box:

- **OK:** Closes the dialog box and saves any changes that were made.
- **Cancel:** Closes the dialog box but does not save any changes unless the **Apply** button was clicked first.
- **Apply:** Saves any changes that were made but does not close the dialog box.
- **Help:** Displays information about the function in the current focus. Selecting **F1** key will also displays this information.

The addition comments  button, displayed on the bottom left, can be used to view and edit comments related to specific items in the dialog box.

Data Validation

When a dataset is initially read in, any error encountered in this section will be indicated by an error or warning message. As well, the corresponding status icon will be displayed on the **Initial** button of the tree view. You can then enter the **Initial Conditions** interface, review the information, and then make and save the necessary changes.

The information entered in the **Initial Conditions** interface is again validated when the **Apply** or **OK** buttons are selected. If there are any errors or warnings a list of messages will be given.

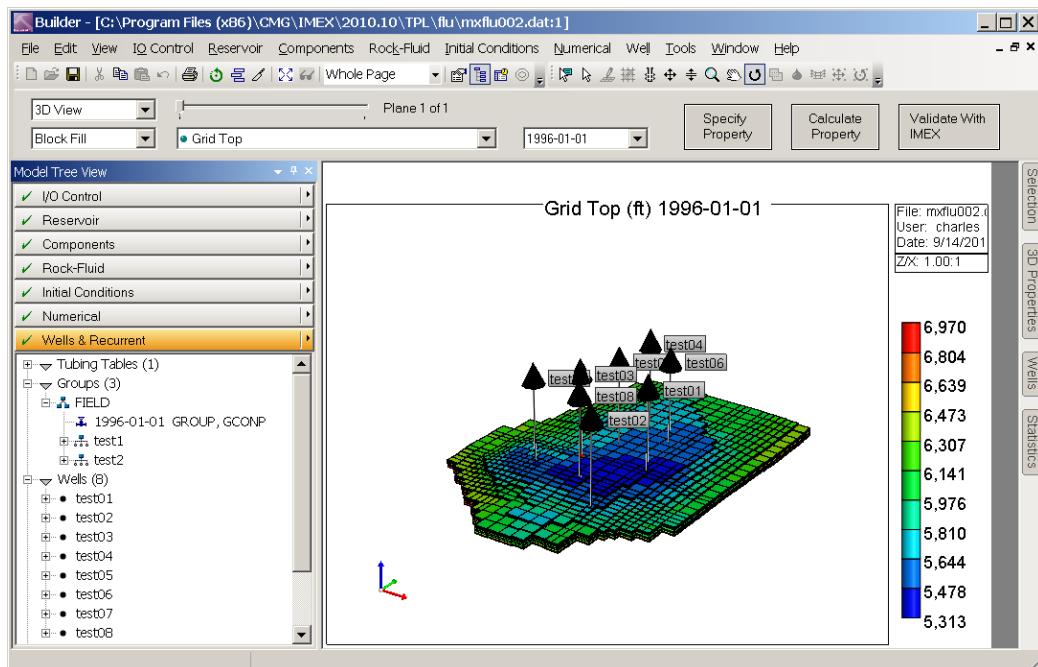
As well, basic validation is carried out when values are entered; for example, if a numerical value is expected and a non numerical value is entered, an error message is displayed. As well, values entered are displayed with their current working units. Values may be entered as a number only or as a value followed by a space and the units used. Values entered with units that are different from the current working units will be converted, and the converted value will be displayed with the current working units.

Well and Group Control

Overview

Through the **Well and Recurrent** data section you can:

- Define wells and groups
- Set production/ injection constraints
- Define well completions and other properties as a function of time.



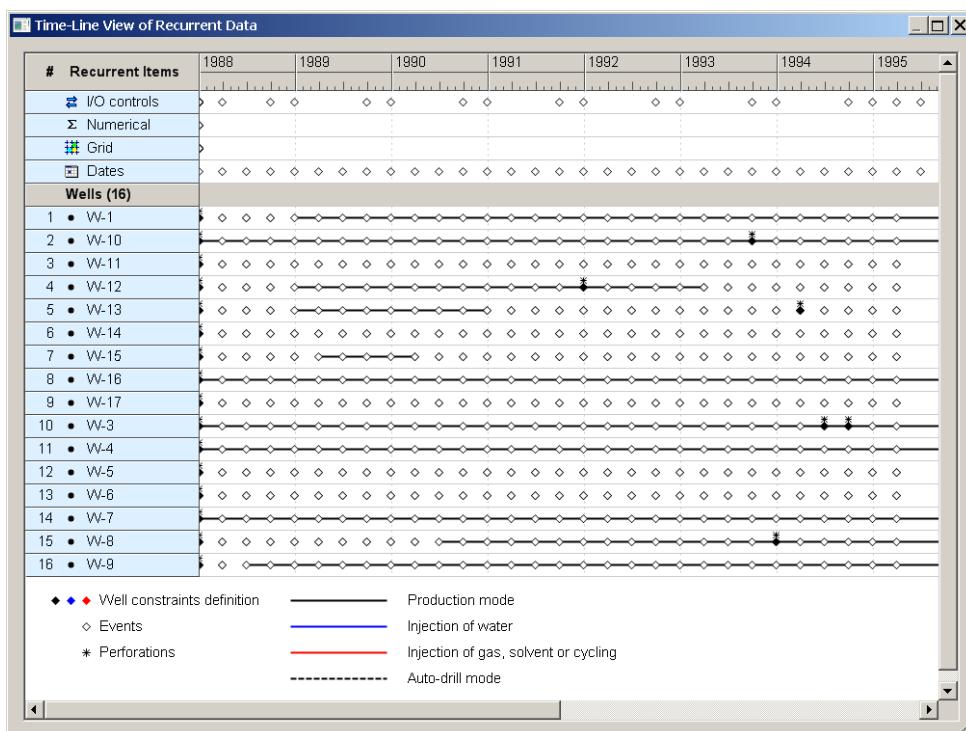
The tree view in the **Well and Recurrent** section displays all data items related to wells, groups and dates available in the dataset. Each data item has an icon. Validation status is shown as an overlay image on the symbols when warnings or errors are detected. Right-clicking a tree item will display a context menu presenting options to add new, delete, validate and change properties. Double-clicking a tree-view item opens a property control panel for that item.

The Time-Line View provides a summary of when different well and other events occur in time.

If you have large amounts of injection and production data, Builder provides a **Production Data Wizard** to assist you in importing this data. Data can be averaged to reduce the number of well changes, and hence reduce the time required for a simulation run. The Production Data Wizard will set injection and production constraints for wells, based on the imported data.

Time-line View of Recurrent Data

The time-line view shows well activities and other recurrent data on a time scale. It provides a quick and easy way to understand well status, to perform overall data quality checking and to access data controls. To open this view, select **Open Time-Line View** in the **Well** menu or click **Show/Hide Time Line View** in the tool bar.



The time-line view is in a floating window which can remain open while you are using other controls. If your computer has a second monitor, it may be convenient for you to drag the time-line view to the second monitor, so you can keep the view open while using the first monitor to interact with Builder. The time-line view has several useful features and functions:

- A time scale with variable length and units
- Accessing data property controls

- Plotting well and group constraints
- Probing data with a press of the left mouse button
- Displaying fixed notes
- Exporting image files and copying images to the Windows clipboard
- Printing the whole image on one page of any available size
- Options to hide/display chosen data items
- Legend information at the bottom of the view

All functions and controls of time-line view are available from the context menu that pops up when you right-click in the view area. Double clicking on a particular data item, event or time scale will open the corresponding property control panel.

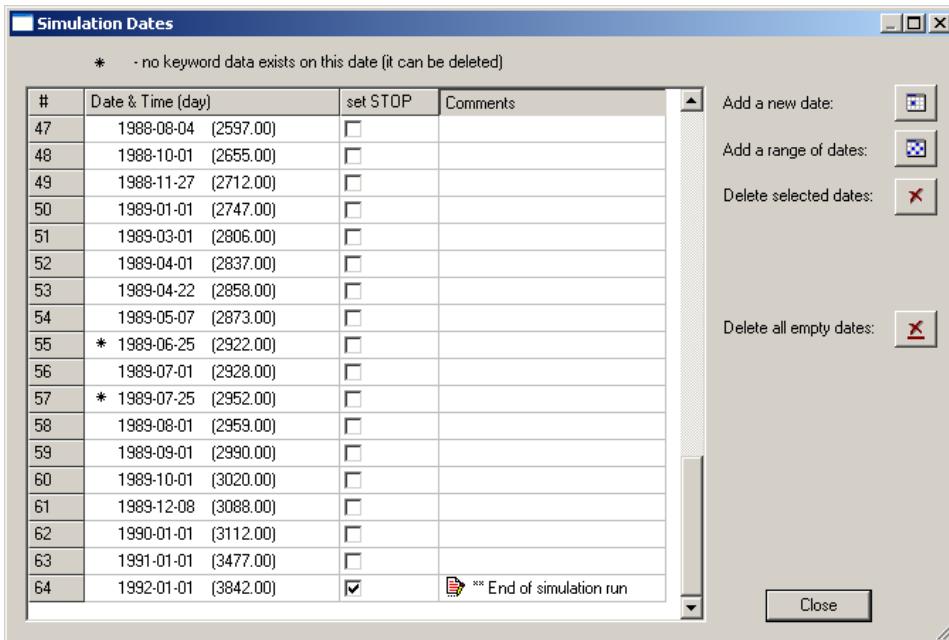
Date/Time Information

The **Well and Recurrent Data** section must contain at least one date (start date) at the beginning of this section. In Builder, the start date is set when the new dataset is created. The simulators will use a default starting date if one is not found in this section. Given a start date, subsequent recurrent data is set by entering a new date or time and then entering the simulation information for that date or time.

Note: If time is used, it refers to the elapsed time (for example in days) from the simulation start date. Dates and times are expected to follow in chronological order. You can enter a single date or time, or a series of dates or times. Builder always uses date format to display and write out recurrent data.

Adding/Removing Dates

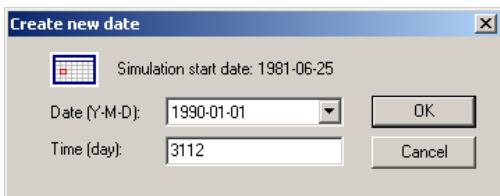
Builder automatically creates date entries for all recurrent data items whenever they are added to the dataset. Having empty dates is not a requirement, unless you need to increase simulator output frequency. The **Simulation Dates** dialog box provides tools for adding new or removing dates that contain no recurrent data items. The **Simulation Dates** dialog box is available by selecting **Dates** from the **Well** menu or by double-clicking **Dates** in the tree-view. While the **Simulation Dates** dialog box is opening, Builder updates all recurrent data items to get existing dates and determine whether or not they are empty.



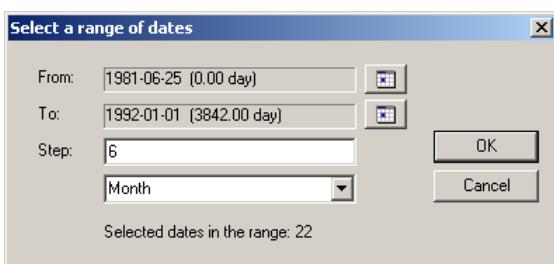
This **Simulation Dates** dialog box shows all dates that exist in the dataset. Through this dialog box, you can set simulation STOP dates, and enter comments for each date.

Adding Dates

Open the **Simulation Dates** dialog box. Press the **Add a new date** button to open a calendar dialog box through which you can enter a new date or time from the simulation start. The dialog box checks that the new date will not repeat an existing one.



In the **Simulation Dates** dialog box, click **Add a range of dates** to open the **Select a range of dates** dialog box, through which you can create a range of new dates with a selected step.



Deleting Dates

Deleting a date is possible only if the date contains no recurrent data items. If it does, the data must be removed prior to deleting the date. You can use the time-line view to see what data exists on a particular date. Click **Delete selected dates** to remove empty dates that you have selected in the grid control. Click **Delete all empty dates** to automatically find empty dates and remove them (after confirmation).

Having a large number of empty dates in a dataset may significantly impact simulator run time, by increasing the number of time steps and output operations. You should check that your dataset has a minimum of redundant date entries.

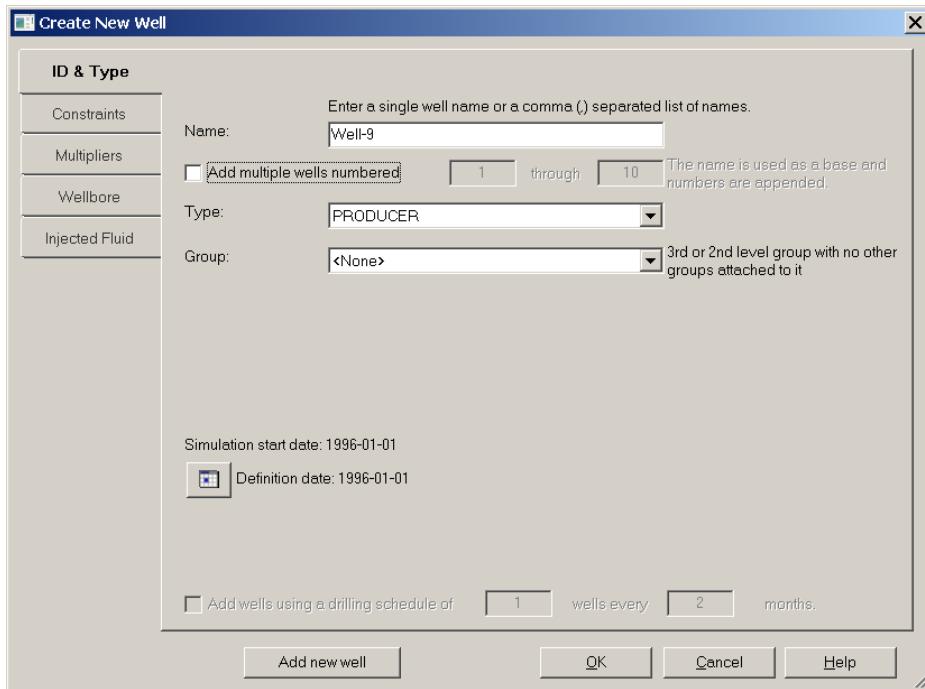
Well Data

The well data is found on the section tree-view under the root item Wells(#). Each well is defined with:

- Unique name with 1 to 40 characters (except * and ?).
- Type PRODUCER or INJECTOR (constant in time).
- Group affiliation (optional).
- Trajectory data (optional) that may have one main branch and several multilateral legs. Each trajectory branch may have perforation intervals on various dates.
- Completions, also called model wells or PERF cards.
- Events that set constraints and other properties in chronological order.

Adding New Wells

Select **Well New** from the **Well** menu or **New** from the context menu after right-clicking a well item in the tree-view. The **Create New Well** dialog box will be displayed. Through this dialog box, you can enter well definition information such as name, type, group name, first definition date, set constraints and injected fluid properties. Multiple wells can be created with the same properties if you put a comma-separated list of names in the **Name** edit box. The **Add new well** button allows you to create new well(s) and immediately reset the dialog box for the next new one, so that you do not have to keep reopening the dialog box. Only **ID & Type** data is required to define a new well; other properties, such as constraints, can be entered later.



The group attaching the well must be defined in the dataset as a 2nd- or 3rd-level group. The top level group cannot attach wells. One group can attach wells or groups but not both. To view the current grouping status, select **Group & Well Connections** from the **Well** menu. This will open a window that shows, in chronological order, all attach-to commands and affiliations by group.

Deleting Wells

A well can be deleted by right-clicking the chosen well item in the tree-view and selecting **Delete** from the context menu. You will be asked to confirm this action. The group status is updated automatically when a well is deleted.

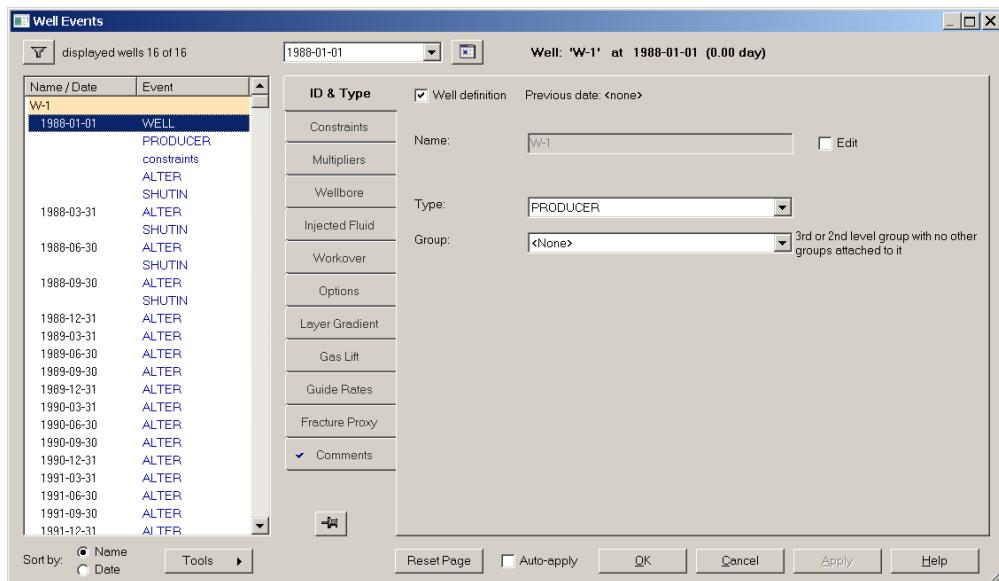
Multiple (or all) wells can be deleted if you right-click **Wells** in the root tree and select **Delete** from the context menu. This command will open a well selection dialog box through which you can select the wells you want to delete.

Well Events

Well Events is a collection of data items related to a particular well. It includes well definition, constraints, constraint multipliers, status and other properties set in time. Each well event has a corresponding keyword in the simulator dataset, so the order and content of well events is a reflection of dataset syntax.

In Builder, the **Well Events** dialog box provides controls for adding new, or editing or deleting existing events. There are several ways to open this window:

- Select **Well Events** from the **Well** menu
 - Double-click a particular event item on the tree or time-line view
 - Select **Properties** from context menu after right-clicking a particular event item on the tree or time-line view



The list on the left side shows all well events defined by keywords in the dataset. The list can be sorted by well name (default mode) or by date, depending on the selection below the list. If the list is too long and inconvenient for scrolling, use the **Filter**  button at the top of the list to display a selected subset of wells.

The right side of the **Well Events** dialog box shows the controls for entering event data, arranged on several tabs. Some tabs show controls for multiple event types and some show controls for one particular event only. When you select a well event in the list control, the corresponding tab is displayed. The current well name and date are displayed in bold at the top of the dialog box.

To add well events:

1. Open the **Well Events** dialog box.
2. Scroll down the list on the left side of the dialog box and select the well name.
3. Select an existing date in the combo box at the top of the window or use a calendar button next to the combo box to enter a new date.
4. Select the tab that has setting for corresponding event and select the appropriate check boxes to enable data input.
5. Enter the event data settings.
6. Click **Apply** or **OK** to validate and enter the new settings. If, after the **Apply** button has been clicked, the dialog box stays open, the event list will have been updated. If the **OK** button is clicked, the dialog box will close immediately.

Multiple wells can be selected in the list using the CTRL or SHIFT keys. In this case, the control changes will apply to all selected wells on the single current date. This is a very convenient way for defining wells with similar properties.

Adding Well Constraints

According to the dataset syntax, well constraints are grouped together with the PRODUCER or INJECTOR keywords. These events combine several data items:

1. Well type: PRODUCER or INJECTOR, cannot change in time.
2. Operate, monitor and penalty constraints: At least one operate constraint is required. The first operate constraint in the list is the primary. It is common to have two operate constraints, the first for flow rate and the second for bottom-hole pressure control. Monitor and penalty constraints are optional.
3. Well-bore definition: Required for pressure drop calculations where the well-head pressure (WHP) constraint has been set or you need to print well-head pressure in the output file.
4. Injection fluid composition: Required for all injections wells.
5. Injected steam parameters: Required for injectors in the STARS simulator.

On the tree view, these items are displayed in lower-case letters under the PRODUCER and INJECTOR keywords.

New Well Constraint Definition

To define the constraints for a new well:

1. Open the **Well Event** dialog box.
2. Select the well name in the event list.
3. Select an existing date from the combo box or add a new one.
4. Select **Constraint definition** at the top of **Constraints** tab then enter your data into the well constant list. Press **Apply**.
5. If **Wellbore** and **Injected Fluid** are required then set them on the corresponding tabs for the same well and date. The controls for each of the data items may change depending on well and simulator types.

Each new constraint definition completely overwrites the previous one. This includes wellbore and injection fluid data. If only the primary constraint value needs to be changed after the constraints have been defined then you can use event ALTER to set a new value. Event TARGET can also be used to set a new or to change an existing constraint.

List of Well Constraints

The list of well constraints must have one or more operate constraints and any number (including zero) of monitor and penalty constraints. The first operate constraint is called primary. To add a new entry in the constraint list, select a constraint type from the drop-down box “select new” at the end of the list. Fill in the other parameters on the new line.

| ID & Type | Constraint definition previous date: <none> | | |
|----------------|---|------------|---------------------------------|
| Constraints | # | Constraint | Parameter |
| Multipliers | * 1 | OPERATE | STO surface oil rate |
| Wellbore | 2 | OPERATE | DWN draw-down pressure |
| Injected Fluid | 3 | OPERATE | WHP IMPLICIT well-head pressure |
| Workover | 4 | OPERATE | BHP bottom hole pressure |
| Options | 5 | MONITOR | GOR gas-oil ratio |
| Layer Gradient | | MONITOR | WOUT |

The following tool buttons allow you to control the number and order of constraints:



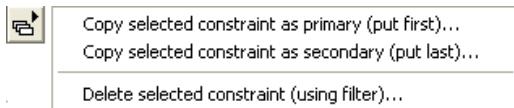
Delete selected constraint



Move selected constraint up and down on the list



Plot well constraints



Commands to copy or delete a single selected constraint to other wells

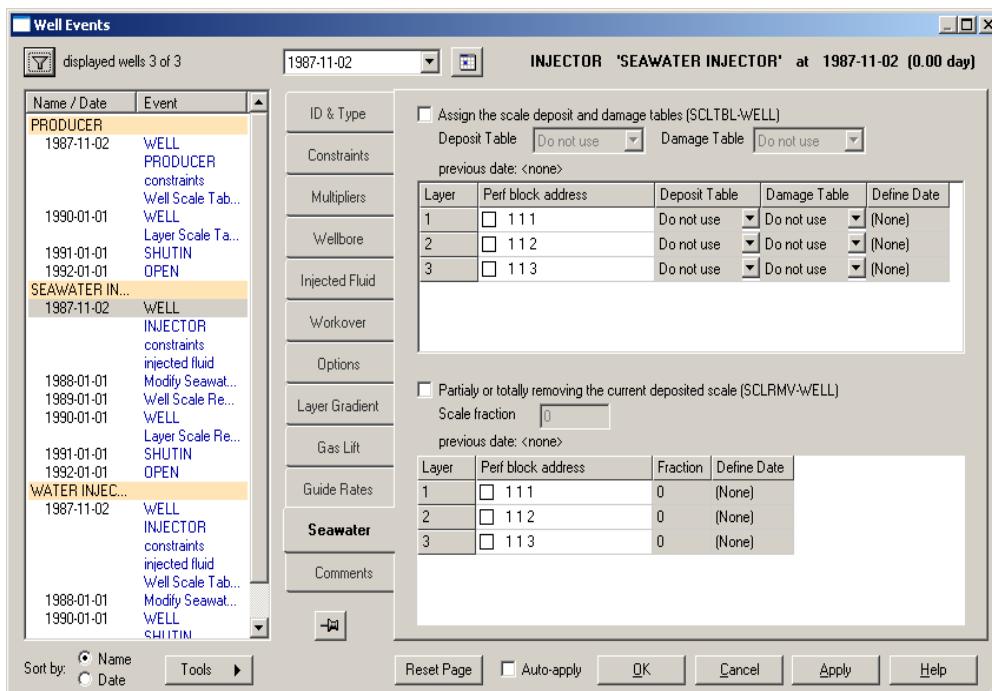
The following rules apply when copying selected constraints:

- Destination wells must be of the same type and have event PRODUCER or INJECTOR set on the selected date(s).
- If destination well already has this constraint type (like STO), than the old constraint is deleted and the new one is added.

The global MXCNRPT value can be specified in this tab-page as well.

Seawater Model (IMEX Only)

The top part of this dialog box, shown below, is for specifying the scale deposit and damage tables for the IMEX sea water simulation model. The bottom part is for specifying the scale removal options.



The global set-ups, **Deposit Table**, **Damage Table**, and **Scale fraction**, are applied to the entire well. To do this, first select **Assign the scale deposit and damage** and/or **Partially or totally removing the currently deposited scale**. You can modify the global set-ups for individual layers via the two grids.

Copying Well Events

New events are first created for a selected well and date. It is common that some events need to be repeated for several wells and/or dates. In this case, you can use the event copy function:

1. Open the **Well Event** dialog box.
2. Select the events that need to be copied (as a source) in the list control. The events to be copied must belong to a single well and date.
3. Right-click in the list control or press **Tools** button to bring the command menu.



4. Select **Copy events using filter**.
5. Set up search conditions and create a list of well and date pairs that will be the destination for the new copies (see [Using the Well and Date Filter](#) for further information).
6. Press **OK**.

Deleting Well Event

Deleting a single well event:

1. Open the **Well Event** dialog box.
2. Scroll the list control and select a particular well event. The corresponding tab page will display the event settings.
3. Clear the check box for the selected event then click **Apply** or **OK**.

Quick way to delete selected events:

1. Open the **Well Event** dialog box.
2. Select one or more events in the list control using the mouse and the SHIFT or CTRL keys.
3. Right-click in the list control or press the **Tools** button to pop up the context menu.
4. Select **Delete events selected in the list**.
5. Click **Yes** in the confirmation dialog box.

Deleting events of selected type using filter:

1. Open the **Well Event** dialog window
2. Select events in the list control.
3. Right-click in the list control or press the **Tools** button to pop up the context menu.
4. Select **Delete events using filter**.

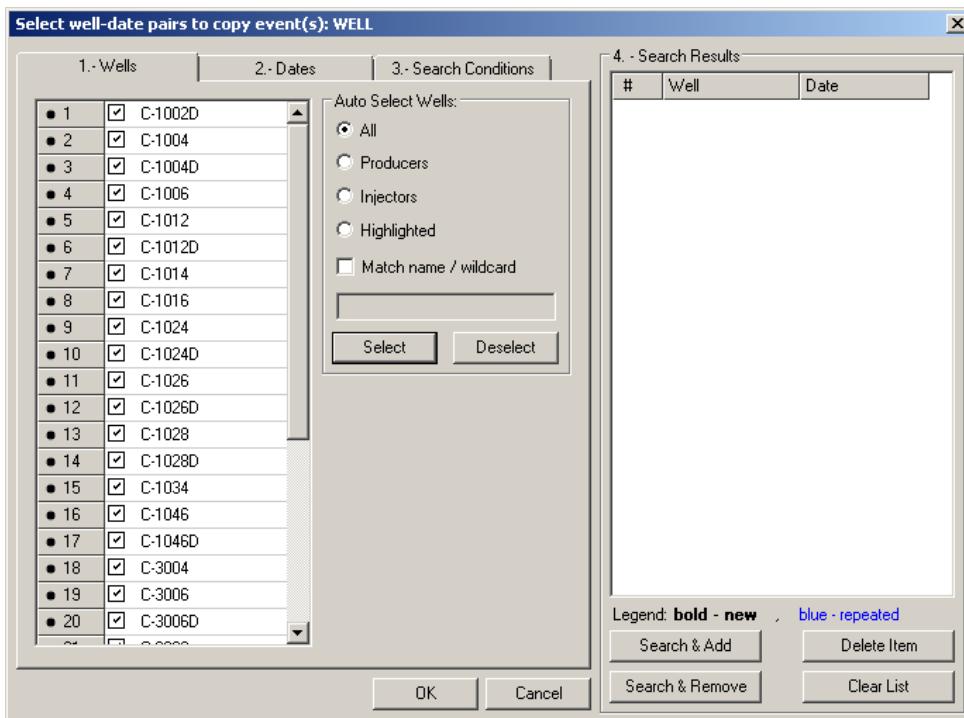
5. Set up the search conditions then create a list of well and date pairs existing in the dataset that may have the events to be removed (see [Using the Well and Date Filter](#) for further information).
6. Press **OK**.

Changing Events Settings

1. Open the **Well Events** dialog box from the **Well** menu, or double-click, or right-click then select **Properties**, on any event item of the tree or time-line view.
2. Scroll the list control and select a particular well event. The corresponding tab will display the event settings.
3. Enter new data values in the corresponding fields then press **Apply** or **OK**.

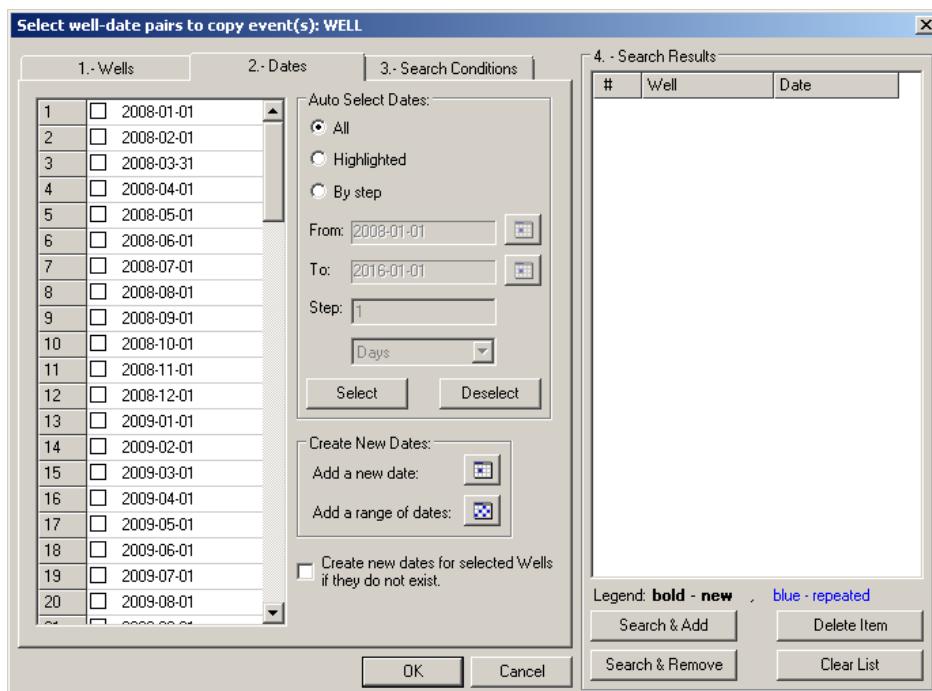
Using the Well and Date Filter

Well & Date Filter is a dialog window used to set up a list of well-date pairs for copying or removing well events. This dialog window is opened when you select **Copy** or **Delete** using the filter in the **Well Event** dialog box. The following example demonstrates the use of the filter for a copy operation. It works similarly in delete mode with the difference being that it does not have an option of creating new dates.



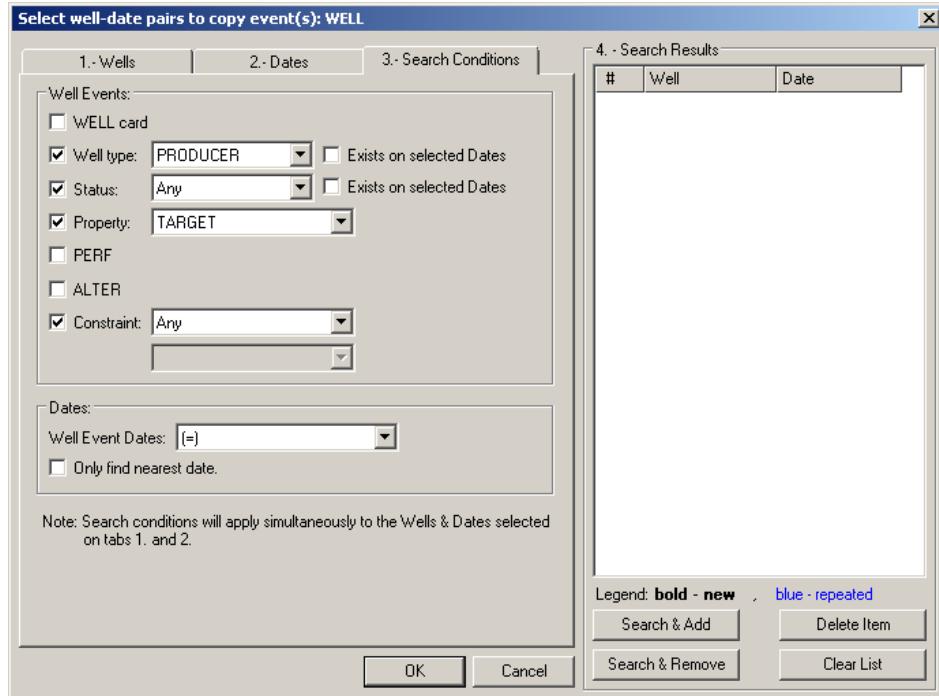
The list control on the right side of the window contains the selected well & date pairs, according to the applied search conditions. This list is preserved when you close and re-open the window. Initially this list is empty.

1. Select well names on the **Wells** tab. You can do this manually by setting the check marks in the table boxes, or you can use the auto-select controls.
2. Select the **Dates** tab.



The table on the left side has a list of all existing well dates that you can select for filtering well & date pairs. Selecting **Create new dates for selected Wells if they do not exist?** will add any missing selected dates from the **Dates** tab to the selected wells from the **Wells** tab when the **Search & Add** button is clicked. The **Add a new date** and the **Add a range of dates** buttons enable you to add new dates to the date list individually or as a range.

3. On the **Search Conditions** tab you can set optional conditions to further filter out the names and dates selected in steps 1 and 2 above. If no search conditions are selected then only the well & date pairs matching the selections from the **Wells** and **Dates** tabs will be included. Multiple search conditions are applied together with AND type logic. The **Exists on selected Dates** check box is used to filter **Well type** and **Status** events that are defined either on the selected dates (check ON), or prior to the selected date (check OFF). The **Only find nearest date** check box will only select a single well event date that is closest to the selected dates. The **Well Event Dates** combo box inside the **Dates** group box further refines the date search by returning only those well event dates which are either : =, <, >, <=, >= to the dates selected from the **Dates** tab.

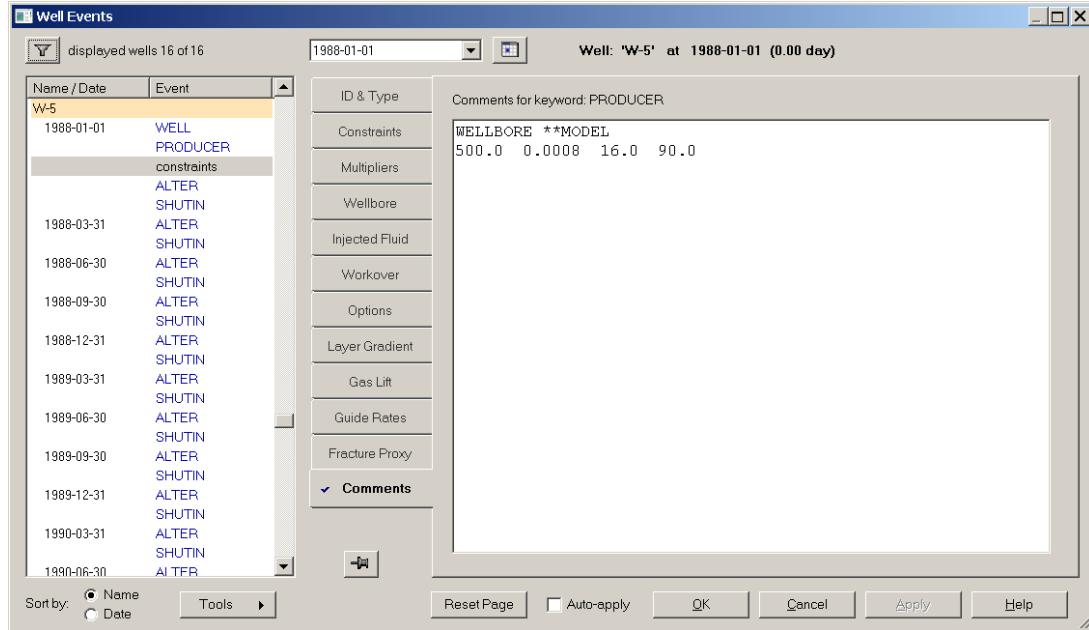


4. Click the **Clear List** button (at the bottom-right corner) to empty the previous search results and click **Search & Add** button to run the filter and create a new well & date list.
5. Finally, click **OK** to copy the selected well events from the **Well Events** tree view to the selected well & date pairs shown in the **Search Results**.

Well Event Comments

1. Open the **Well Event** dialog box from the **Well** menu, or double-click, or right-click then select **Properties**, on any event item of the tree or time-line view.
2. Scroll the list control on the left side of the dialog box then select a well event.
3. Select the **Comments** tab to read or write new comments.

Note: If comments are available for the selected event, then there is check mark on the **Comments** tab as an indication. By default, the current tab is automatically changed for each event selected in the list. If you click the push-pin  button (below the **Comments** tab), you can fix the current tab. This may help you to view the comments while you are browsing the event list.

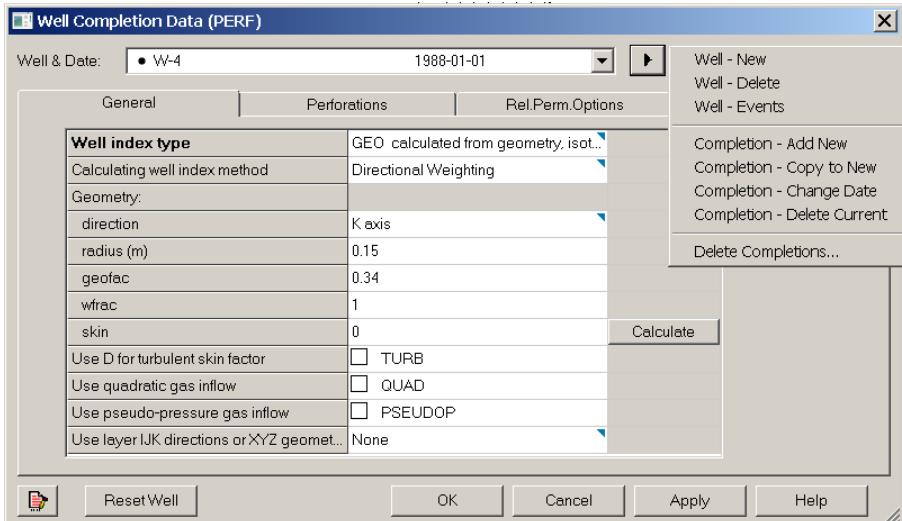


Well Completions (PERF)

Well completions define well locations and flow connections to the grid blocks in the simulation grid. At least one completion is required for each well. Well completions include data used in several dataset keywords: GEOMETRY, PERF, LAYERXYZ, LAYERIJK and KRPERF. Well completion data can also be referred to as PERF cards or model wells.

To open the **Well Completion Data (PERF)** dialog box, use one of:

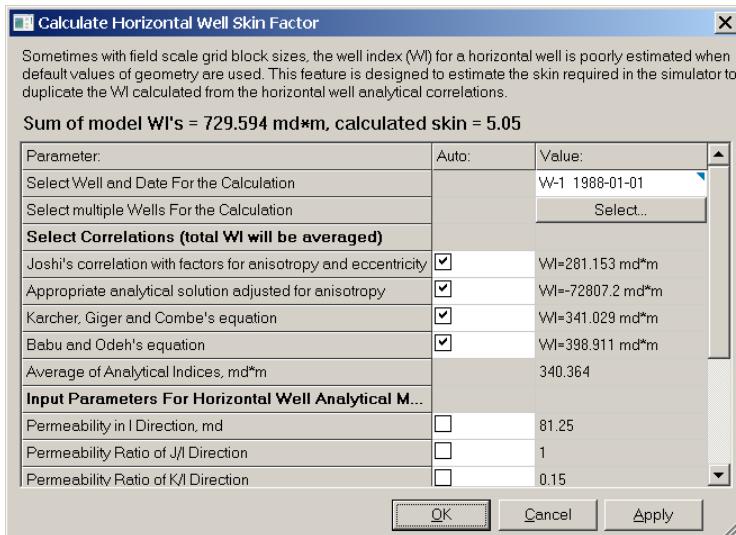
- Select **Well Completions (PERF)** from the **Well** menu.
- Double-click on a completion item on the tree-view.
- Right-click a completion item on the tree-view then select **Properties** from the context menu.



The **Well & Date** drop-down list (at the top of the dialog box) contains all available well completions. The **right-arrow** button brings up a context menu with the main commands for creating and deleting wells and completions. If you have several completion dates for one well the blue color is used to indicate those items that have different values from the previous date.

Calculating the Horizontal Well Skin Factor

It is possible to provide a better estimate of the well index (WI) for horizontal wells. The feature can be accessed by pressing the **Calculate** button next to the skin factor. The **Calculate Horizontal Well Skin Factor** dialog box is displayed. As indicated at the top of the dialog box, the WI value is quite often poorly estimated when using the default geometry values for field scale grid block sizes. This feature will re-calculate the skin value required in order to duplicate the WI calculated from the horizontal well analytical calculations.



Adding New Well Completions

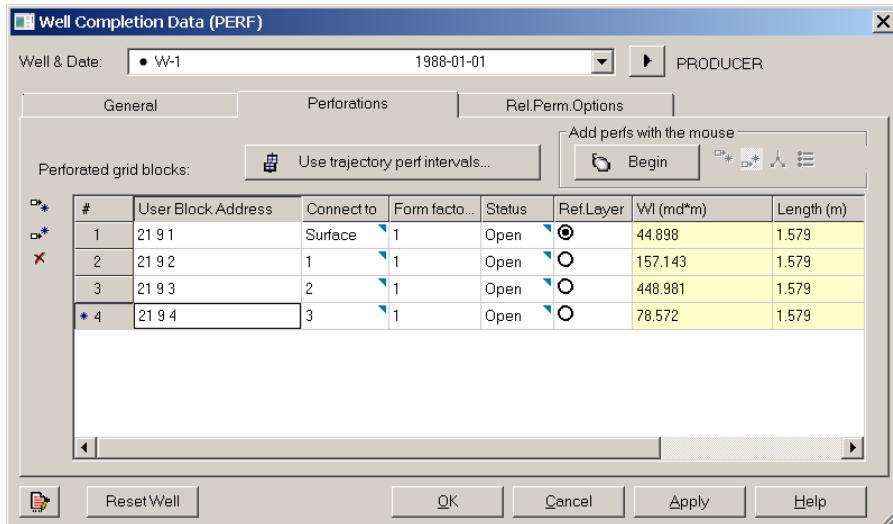
Well completions can be created in two ways:

- By entering the geometry data and block addresses in the **Well Completion** dialog box, as described below
- Have it automatically calculated after importing data of actual well trajectories and perforation intervals (look in the in corresponding trajectory section).

To enter new well completion data:

1. Open the **Well Completion Data (PERF)** dialog box.
2. Press right arrow button at the top of the dialog box to display the context menu.
3. Select **Completion - Add New** then select a date.
4. On the **General** tab, select the appropriate well index type and enter the well geometry data.
5. Select the **Perforations** tab.
6. Press the **Begin** button to start the mouse input mode.
7. Click on grid blocks in the 2D or 3D reservoir view to add block addresses for well perforations.
8. Click the **Stop** button to terminate the mouse input mode.
9. Click **Apply** or **OK**.

As well as using the mouse, you can manually add new completed blocks or change the existing address. Just add a perforation line (using the tool bar buttons to the left of the table) and type in a user block address UBA and look on the reservoir view for the new node location. A range of grid blocks is allowed when typing a UBA address like 3,3,1:6 for vertical well section or 10,1:20,3 for horizontal one. UBA string with the range subscript will be automatically expanded into a number of perf layers.



The following additional features are available through tool buttons:

This button enables you to set perforation intervals by measured depth then automatically calculates completed grid blocks and fills in the table. It works only when trajectory data is available for the selected well.

Button inserts a new perforated grid block line above the selected row.

Button inserts a new perforated grid block line below the selected row.

Button deletes the selected perforated grid block row.

Start and stop mouse input of grid blocks from the 2D or 3D reservoir view.

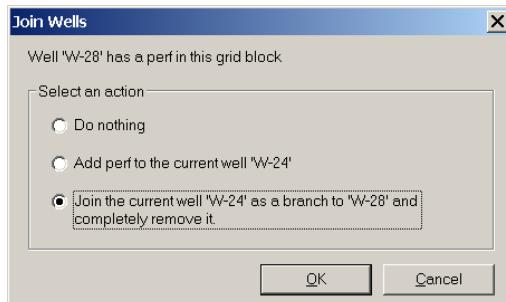
Buttons control insertion of new grid blocks from mouse input. These buttons control if the new block is inserted above the selected block, the new block is inserted below the selected block, or if a new branch is added from the selected block.

Button opens the **Perf Options** dialog box which presents options for inserting multiple perforation nodes in a line between two mouse clicks. Depending on which button control is selected, multiple perforations will be inserted above the selected block, multiple perforations will be inserted below the selected block, or multiple perforations will be added as a new branch from the selected block.

Joining Well Completions (Multilaterals)

Builder enables you to setup multilateral (branching) completions. It is achieved by changing the layer that the branch is attach-to. If you use mouse input then

1. Open the well completion dialog window.
2. Select the well completion that is going to be the leg branch.
3. Select the **Perforations** tab and then select the layer which is going to be connected
4. Press the **Begin** button
5. In the 2D or 3D view, click the attach-to block in the well completion that is going to be that main branch. The **Join Wells** dialog box is displayed:



6. Select the third option and then click **OK**.
7. Click the **Stop** button to finish the mouse input.
8. Click **Apply** or **OK** to apply the change.

Note: A more effective way to create multilaterals is by using trajectory data (refer to [Well Trajectories](#) for further information) which calculates multilateral completions automatically.

Additional Data Columns

The **Perforated Grid Blocks** table constrains a few columns with yellow background, indicating information that is not a part of the well completion but which has been calculated for display purpose only:

- **WI** column shows the estimated well index not including the fluid mobility part. The general formula for this value is:

$$WI = \frac{2 \pi kh wfrac}{\ln(R_c/R_w) + S} \text{ (md * m) or (md * ft)}$$

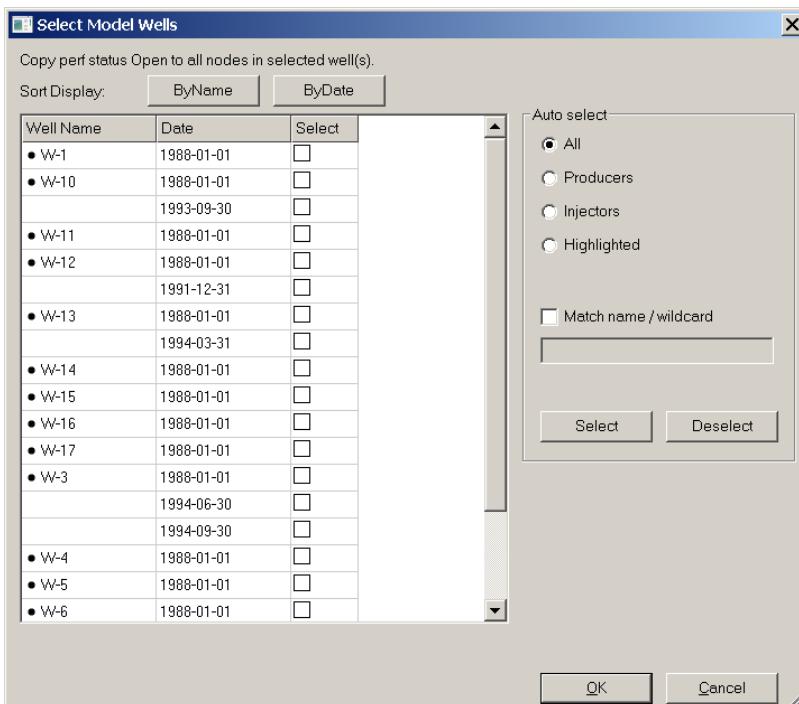
$$R_e = \text{geofac} * \sqrt{\frac{(\text{areap})}{(\pi * wfrac)}}$$

The specific values for effective permeability and external block radius depend on the grid properties, well direction and user inputs on the **General** tab. See the simulator manual for complete formulas of well index calculations.

- **Length** column shows estimated length of perforated well section in the grid block.
- **Block Top** and **Block Bottom** columns show average depth or elevation of grid for block top and bottom side respectively.

Copying Data Values to Other Well Completions

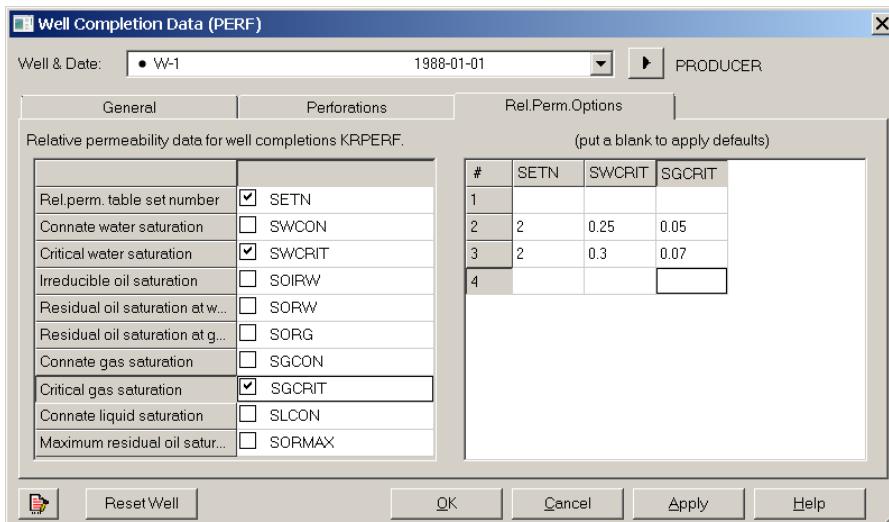
There is a quick way to copy data values such as well radius or form factor from one well completion to another. Right-click the data cell that contains the source value then select **Copy data item**. The **Select Model Wells** dialog box will be displayed, through which you can select the destination well completions:



The display can be sorted by well name or completion date. Select the target wells then click **OK** to paste the values.

Relative Permeability Options

The **Rel Perm Options** tab provides several options for modifying rock type and relative permeability end points in the completed grid blocks.

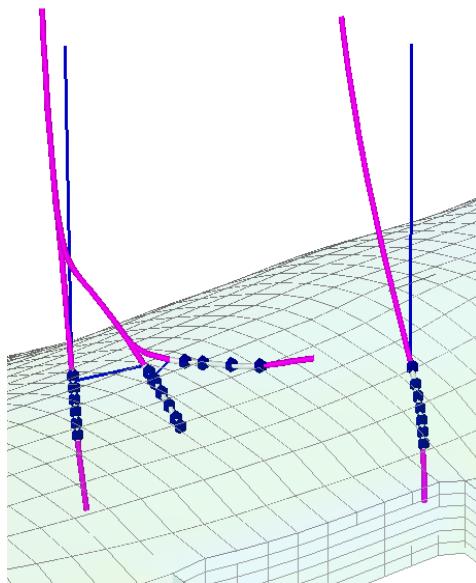


Modeling of water or gas coning around the well may be one of the applications for these controls.

1. In the left side, select the relative permeability data that you want to modify.
2. Input the corresponding values for one or more grid blocks. Leaving a blank cell retains the default values.
3. Click **Apply** or **OK** to apply the change.

Well Trajectories

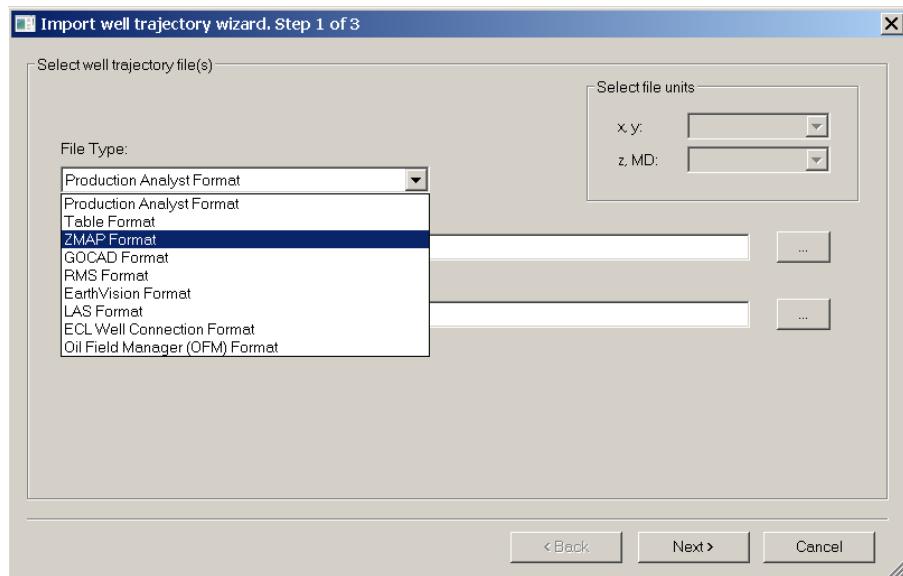
A well trajectory is a description of a well path through the earth. Pragmatically, it is a list of the nodes (vertices) that give a location of the well path in space. Each node has xyz coordinate and measured depth from the well top (KB elevation). A trajectory may include multiple perforation intervals, each defined by the measured depth of the start and the end of the perforated interval and the date of the perforation. Builder supports multilateral trajectories with several levels of branching for one well. It is recommended that you use well trajectory data, whenever it is available, to create well completions in your dataset, for more accurate completions and a better visual representation.



Well Trajectory Import

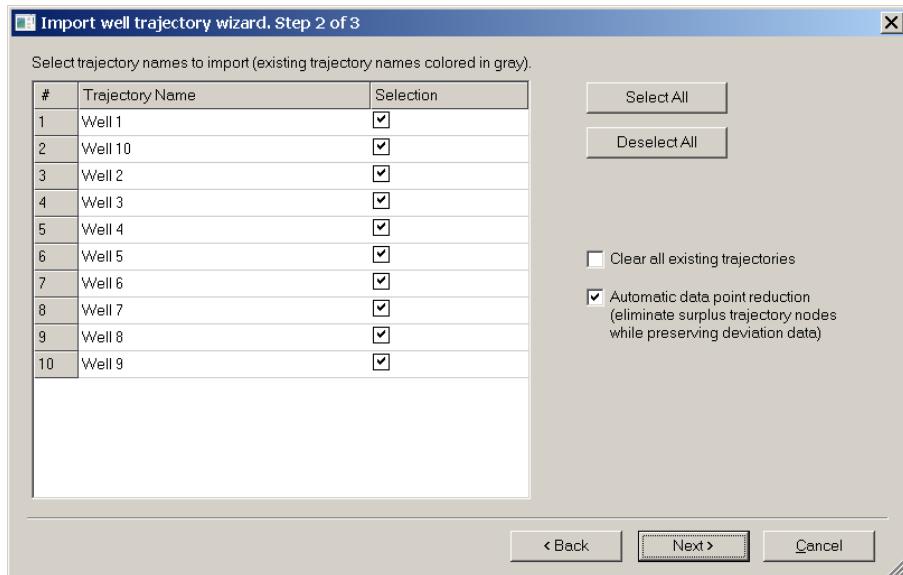
Builder supports a number of file formats for reading in well trajectory data, and provides automatic unit conversion.

1. Select the menu item **Well | Well Trajectories | Well Trajectories** to open Step 1 of the **Import well trajectory wizard**:

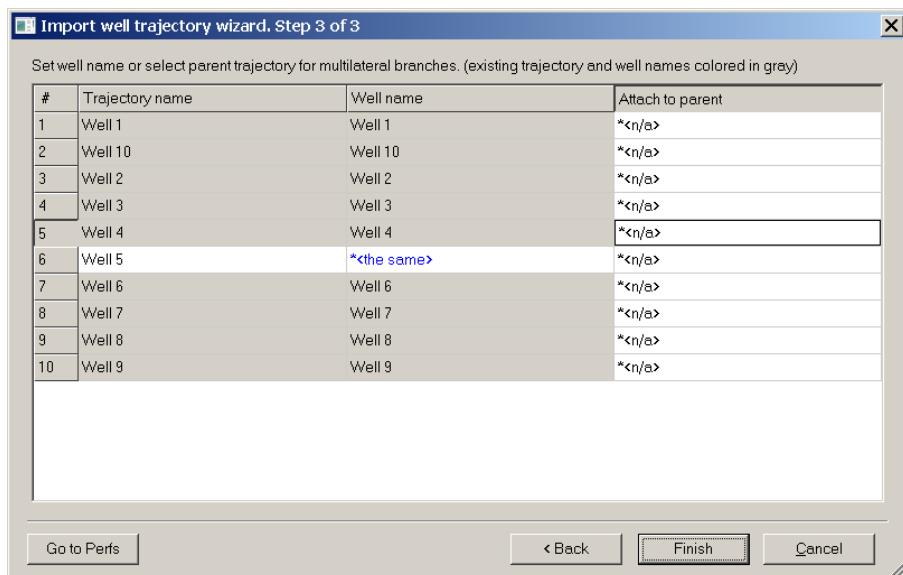


2. Select the appropriate file format, units and location.

3. Click **Next**. Step 2 of the **Import well trajectory wizard** is displayed:



4. Select the trajectory names that you wish to import. Check on or off the option to remove all existing trajectories. It is recommended to use automatic reduction of trajectory data points (the default) to speed up well calculations and to avoid inflating the data file. The trajectory names that already exist in the dataset are highlighted in gray color and can be overwritten.
5. Press **Next** button. Step 3 of the **Import well trajectory wizard** is displayed:



- Step 3 provides you with the following options:

- Edit a trajectory name
- Set a new or existing well for imported trajectory (creating a main branch)
- Set parent trajectory (creating a multilateral leg branch)

Note: Entry “*<the same>” indicates that a new well will be created with exactly the same name as trajectory.

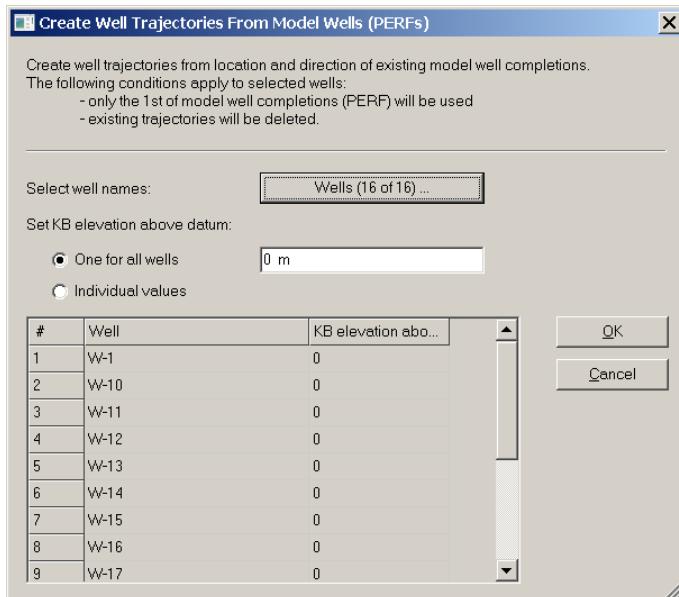
- Press **Finish** to complete and return to the main view or click **Go to Perfs** to complete the trajectory import and immediately go to the **Trajectory Perforation Intervals** dialog box to set or import perforation intervals.

The imported well trajectories are saved at the end of the dataset file.

Create Trajectories from Well Completions

This function is useful where you have no files with trajectory survey data for importing but the current dataset has existing well completions (PERF cards) and you wish to quickly add trajectories and set different perforations intervals. Provided that the wells are nearly vertical or horizontal, the created trajectories will be quite representative.

- Select **Well | Well Trajectories | Create Trajectories from Completions (PERFS)** to open the **Create Well Trajectories From Model Wells (PERFS)** dialog box:

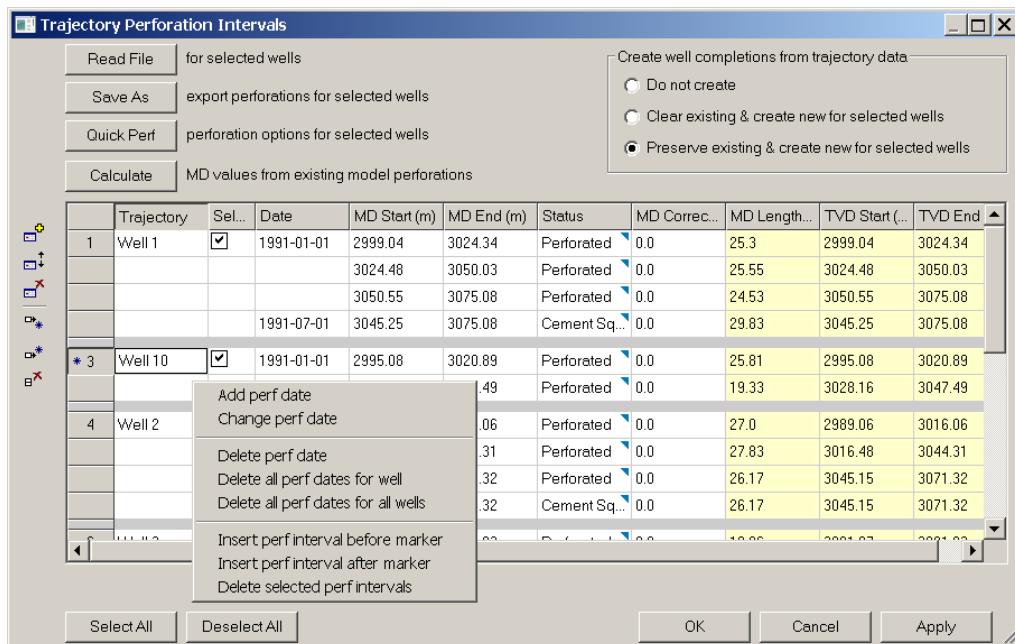


- Click the **Wells** [Wells (16 of 16) ...] button to select wells to add trajectories for.
- Set KB elevation for all wells at once or for each one individually
- Press **OK**.

Trajectory Perforation Intervals

A trajectory perforation is a set of perforation intervals with a measured depth, starting and ending at a particular date. A well trajectory must be imported or created before the perforations can be added. It is important to understand the difference between trajectory perforations and the well completions used by simulators. A trajectory perforation is a part of raw data used by Builder to create the grid-based well completions (PERF card) for simulator input.

To open the **Trajectory Perforation Intervals** dialog box, select **Well | Well Trajectories | Trajectory Perforation Intervals**:



Tool bar commands:

- Adds new perforation date for selected well
- Changes perforation date
- Deletes perforation date
- Inserts new perforation interval above the marker
- Inserts new perforation interval below the marker
- Deletes selected perforation intervals

As shown above, right-click anywhere in the grid area to bring up a context menu with the same commands as the tool bar, and additional commands for managing the table.

Reading a perforation file:

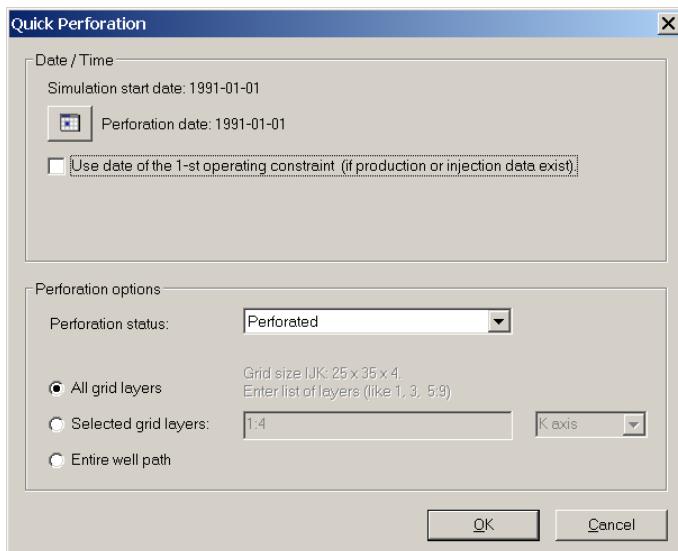
Set the **Select** check mark for the well trajectories that you want to read new perforations for. Use the **Read File** button to browse for the file location and select file depth units. The format of the perforation file can be found in [Well Perforation File Format](#) on page 569.

Saving a perforation file:

Set the **Select** check mark for well trajectories for which you want to save perforations. Use the **Save File** button to browse for the file location. The format of perforation file can be found in [Well Perforation File Format](#) on page 569.

Quick Perforation

The **Quick Perf** button provides an easy way to create new perforation intervals in selected grid layers. These intervals can be created for the whole grid or for a list or range of grid layers.



Creating well completions from trajectory data:

The group box at the top of the **Trajectory Perforation Intervals** dialog box provides three options for creating well completions for simulator input from trajectory perforations. The selected option is applied when you click **OK** or **Apply**.

- Select **Do not create** to completely avoid changing well completions
- Select **Clear existing & create new for selected wells** to remove all completions for all wells and then create new completions for selected (checked) wells.
- Select **Preserve existing & create new for selected wells** to keep existing completions and create new for selected (checked) wells.

Note: If the new completion has the same well and date as the existing one then the old will be replaced.

Trajectory Properties

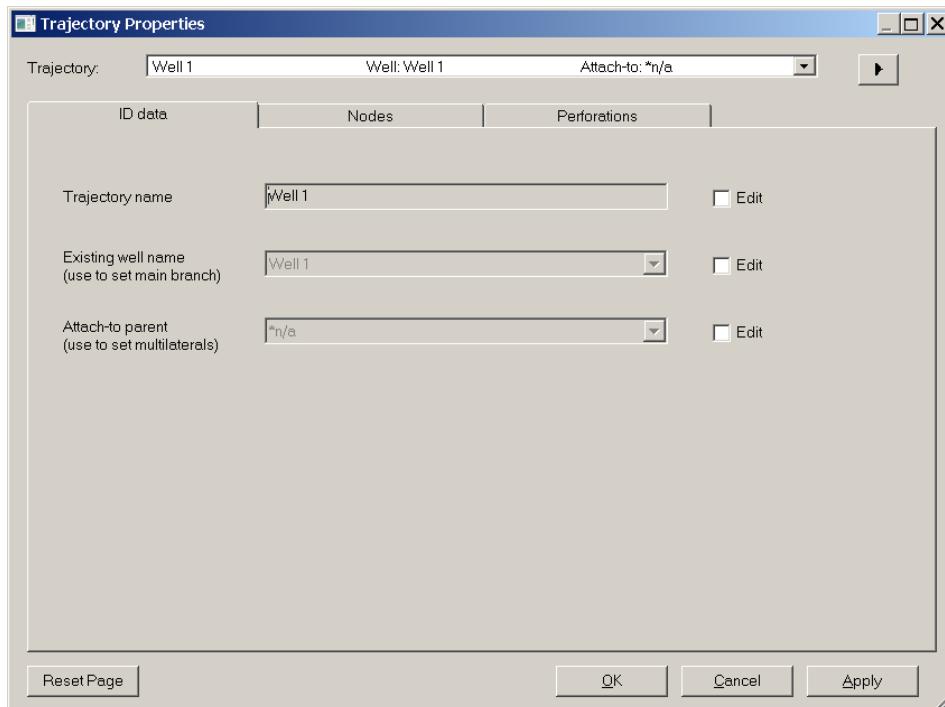
Well trajectory properties combines several data items:

- Own id with “attach-to parent” information
- Well path nodes (list of xyz locations)
- Trajectory perforation intervals

Through the **Trajectory Properties** dialog box, you can view and modify the above information for a selected trajectory. This feature is intended for advanced users.

There are several ways to open the **Trajectory Properties** dialog box:

- Select menu item **Well | Well Trajectories | Trajectory Properties**
- Double-click on a trajectory in the tree view
- Right-click a trajectory in the tree view then select Properties from the context menu



Press the right-arrow button to the right of **Trajectory** drop-down box to open a context menu with commands for creating new trajectories and deleting existing ones:



Select **Create new trajectory** to create a new blank trajectory object. Select **Create new trajectory copy** to create a copy of current trajectory with a different name.

The **ID data** tab shows the trajectory name and parent, which can either be a well (for the trajectory main branch) or another trajectory (for multilateral leg). You can change these settings after checking **Edit** for the corresponding item. Press **Apply** or **OK** to save the changes.

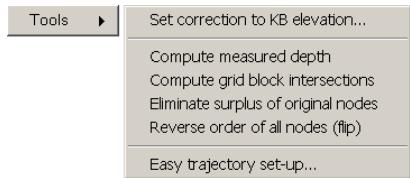
The **Nodes** tab shows the table of trajectory nodes that make up the whole well path, with added grid block intersections and boundaries of perforation intervals. The **Node Type** column shows what each node represents.

Note: Only nodes of the *Original* type are preserved through calculation. Other node types are added or removed when calculating intersections with grid blocks.

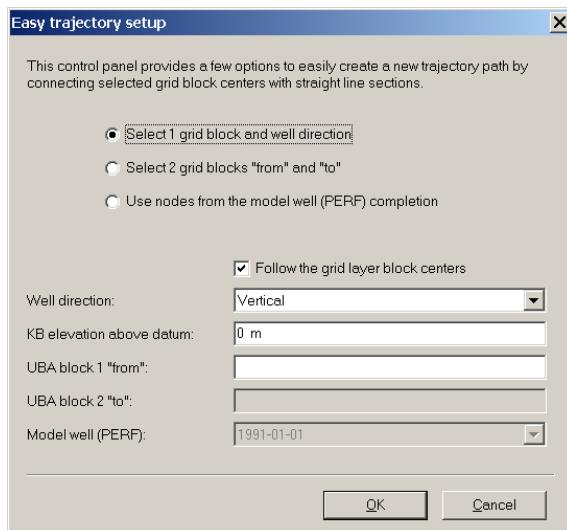
| # | x (m) | y (m) | z (m) | Measured de... | Node type | Perforation | Grid block | U |
|----|---------|---------|---------|--------------------------------|-------------|----------------|------------|----|
| 1 | 1680.84 | 2838.81 | 0.00 | 0.00 | Original | No | | |
| 2 | 1680.84 | 2838.81 | 2987.10 | 2987.10 | Original | No | | |
| 3 | 1680.84 | 2838.81 | 2998.54 | 2998.54 | Block entry | No | 239 | 15 |
| 4 | 1680.84 | 2838.81 | 2999.04 | 2999.04 | Perforation | Yes | 239 | 15 |
| 5 | 1680.84 | 2838.81 | 3024.29 | 3024.29 | Block exit | Yes | 239 | 15 |
| 6 | 1680.84 | 2838.81 | 3024.29 | 3024.29 | Block entry | Yes | 1114 | 15 |
| 7 | 1680.84 | 2838.81 | 3024.34 | 3024.34 | Perforation | No | 1114 | 15 |
| 8 | 1680.84 | 2838.81 | 3024.48 | Insert a number of nodes... | | Yes | 1114 | 15 |
| 9 | 1680.84 | 2838.81 | 3045.25 | Append a number of nodes... | | Cement sque... | 1114 | 15 |
| 10 | 1680.84 | 2838.81 | 3050.03 | Set a total number of nodes... | | Cement sque... | 1114 | 15 |
| 11 | 1680.84 | 2838.81 | 3050.05 | Delete selected nodes | | Cement sque... | 1114 | 15 |
| 12 | 1680.84 | 2838.81 | 3050.05 | Delete incomplete nodes | | Cement sque... | 1989 | 15 |
| 13 | 1680.84 | 2838.81 | 3050.55 | Delete non-original type nodes | | Cement sque... | 1989 | 15 |
| 14 | 1680.84 | 2838.81 | 3075.08 | 3075.08 | Perforation | No | 1989 | 15 |

Right-click in the table to open a context menu with commands for controlling the number of nodes, as shown above.

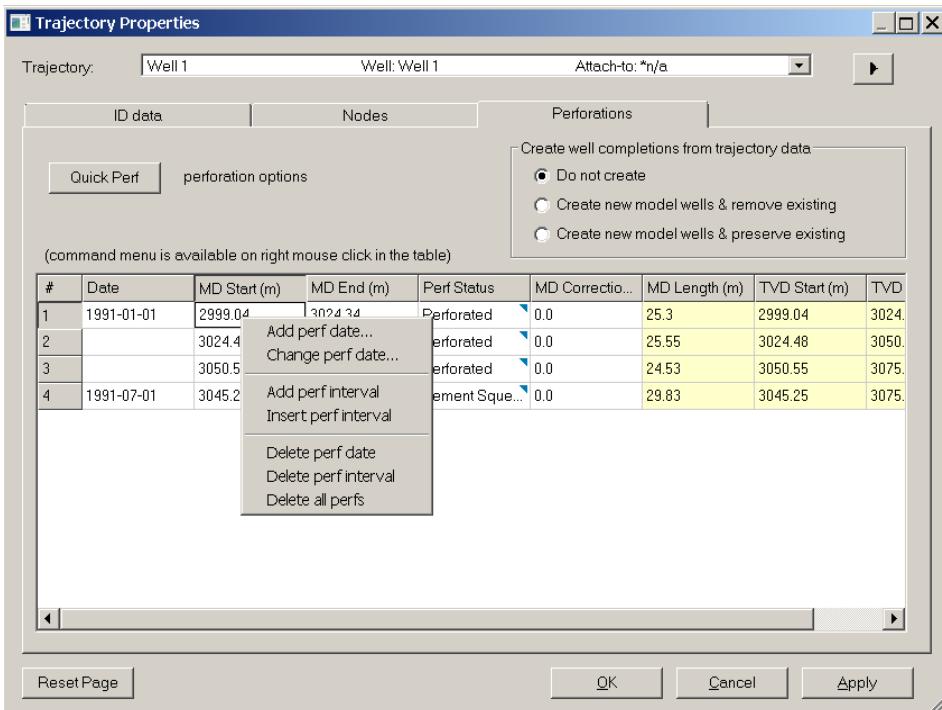
In most cases, trajectory nodes are imported from a file, but you have the option to directly input a new xyz location by typing, or by copy and paste, or by editing existing values. In addition you can click the **Tools** button to open a menu with the following commands:



- **Set correction to KB elevation** allows you to shift all nodes in the z (vertical) direction by changing KB elevation above datum. Grid block intersections will be recalculated.
- **Compute measured depth** allows recalculating measured depth values for trajectory nodes. Two options are available, one to preserve existing non-blank values, the second to recalculate all starting from zero depth at the top.
- **Compute grid block intersections** invokes a function to find grid block intersections of the trajectory path, then update them in the table.
- **Eliminate surplus of original nodes** finds and removes extraneous nodes from the well trajectory, while still preserving the deviation information. This may speed up calculation and reduce file size, particularly with near-vertical wells.
- **Reverse order of all nodes (flip)** basically turns the trajectory upside-down. It is an easy way to create some production-injections well pair for advanced recovery processes.
- **Easy trajectory set-up** opens the **Easy trajectory setup** dialog box, through which you can create a new trajectory path. Choose the appropriate options then click **OK** to create the new trajectory path.



The **Perforations** tab shows the trajectory perforation intervals. The command options available on this tab are exactly the same as they are in the standalone dialog Trajectory Perforations Intervals, described above. The only difference is that here all changes happen for single well and trajectory. For more information on **Quick Perf** and context menu commands, refer to [Trajectory Perforation Intervals](#) on page 301.



Trajectory Smoothing

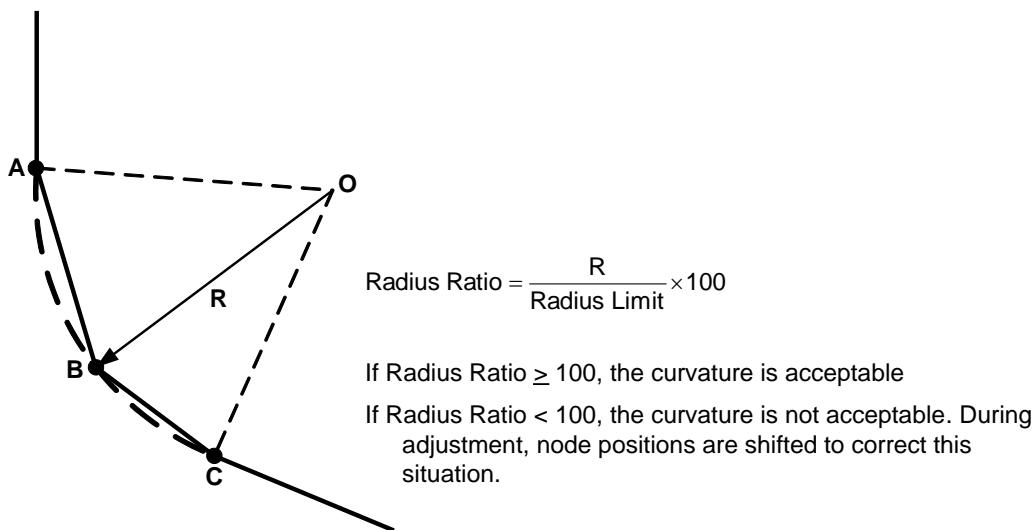
Using the Trajectory Smoothing feature you can:

- Highlight and resolve problems with a trajectory
- Clean redundant trajectory nodes
- Manually insert and delete points in the trajectory
- Eliminate kinks
- Interpolate between trajectory nodes
- Replace an existing trajectory with a smoothed trajectory
- Export the smoothed trajectory to a Builder-formatted trajectory file for later import

Background

Adjusting the Radius ratio

You can limit the well curvature of a segment to be greater than or equal to a defined minimum bending radius. This is illustrated in the following diagram:

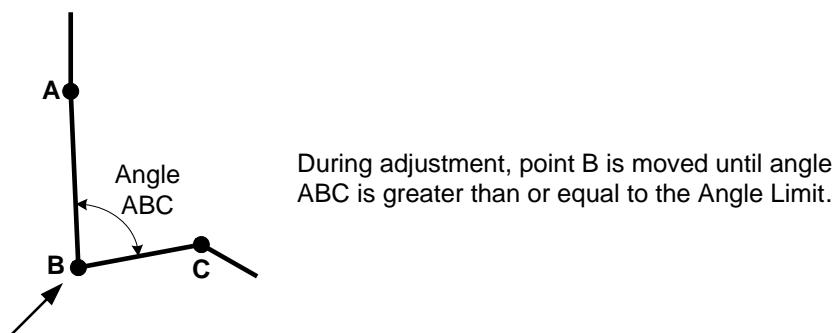


In the above example, the trajectory passes through points A, B, and C. If the radius ratio is greater than or equal to 100, the trajectory between these points is equal to or greater than the minimum bending radius and the curvature is acceptable. If the radius ratio is less than 100, then the radius of the curve is less than the minimum bending radius, which is not acceptable. In this case, the node positions will need to be adjusted.

Note: If Radius Ratio ≥ 100 , Builder will display it as 100 in the **Smooth Trajectory** table.

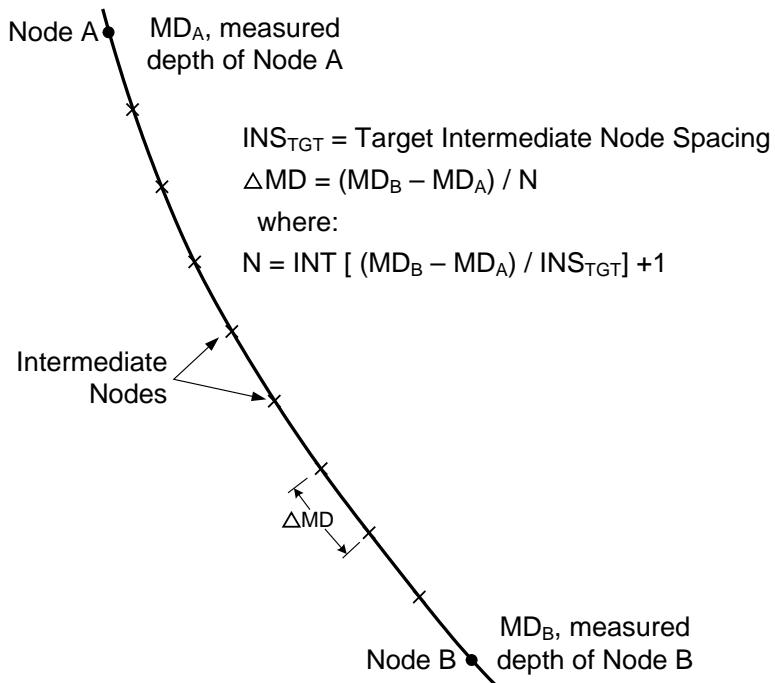
Adjusting the Angle ratio

You can limit the curvature of a well segment by defining the minimum angle allowed between three consecutive points along the trajectory. This is illustrated in the following diagram:



Interpolating between nodes

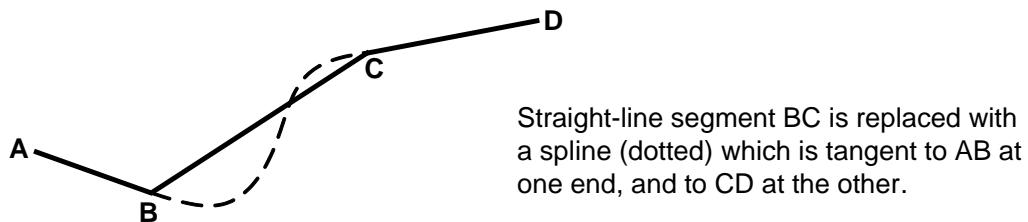
When you interpolate between two nodes, Builder divides the curve between adjacent nodes into equal-length segments, as illustrated below:



For example, if the distance between two nodes is 97.2 m and you want the curve between the nodes to be interpolated into sections of (about) 10 m, then $N = \text{INT} (97.2/10) + 1 = \text{INT} (9.72) + 1 = 10$, and the interpolation spacing will be $97.2/10 = 9.72$ m.

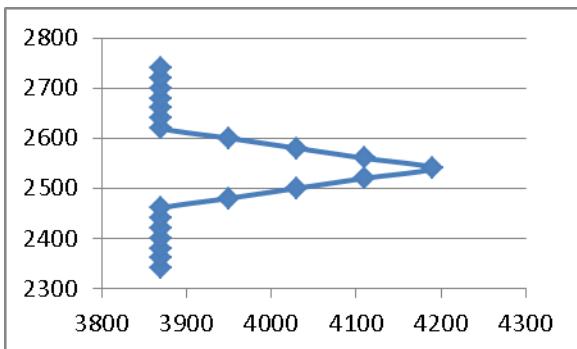
Note: Interpolation calculation between two nodes is independent of the interpolation calculation between other nodes.

When you interpolate the trajectory between two nodes, Trajectory Smoothing inserts a Hermite spline as shown below:

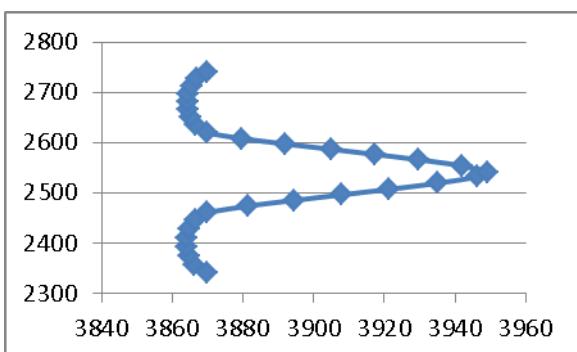


As illustrated above, the measured depth interval will increase when you insert the spline. The greater the tangent factor, the more pronounced the curve, as shown in the following examples.

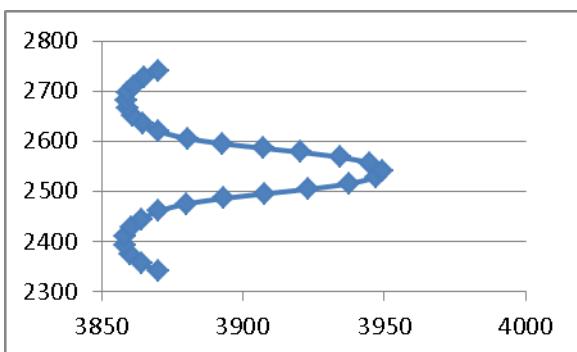
Raw Data:



Tangent Factor = 0.5:

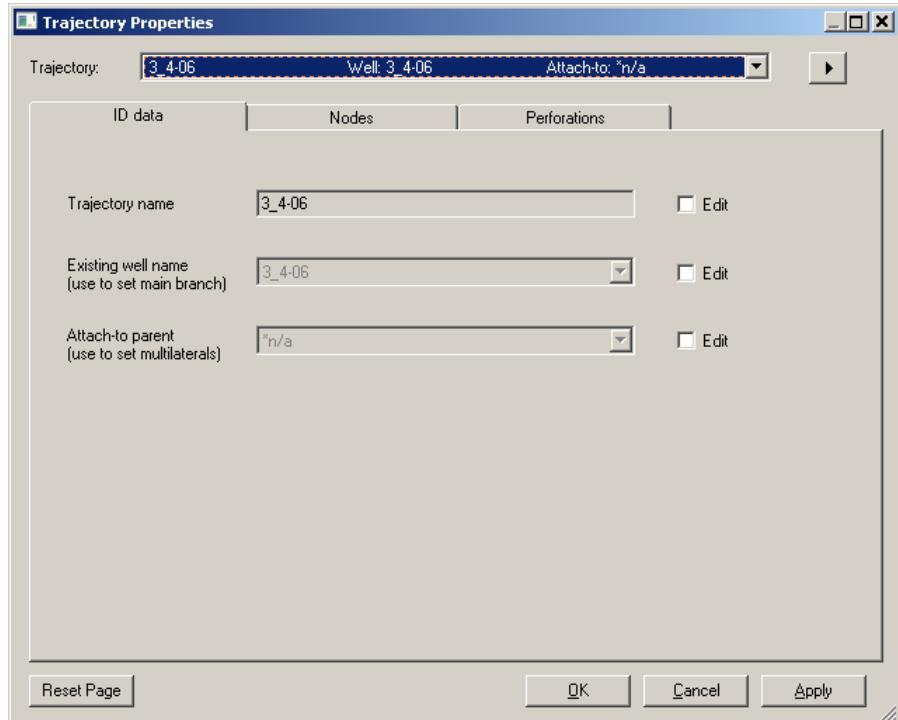


Tangent Factor = 1.0:



Using Trajectory Smoothing

1. Click Well | Well Trajectories | Trajectory Properties in the Builder menu bar. The **Trajectory Properties** dialog box will be displayed with the **ID data** tab selected:



2. In the **Trajectory** box, select the trajectory that you want to smooth.

3. Select the **Nodes** tab. The raw trajectory data will be displayed, as shown in the following example:

| # | x (m) | y (m) | z (m) | Measured depth ... | Node type | Perf |
|----|-----------|------------|---------|--------------------|-------------|------|
| 1 | 430511.72 | 6743018.58 | 3255.46 | 0.00 | Original | Yes |
| 2 | 430515.71 | 6743018.54 | 3258.60 | 5.07 | Block exit | No |
| 3 | 430515.71 | 6743018.54 | 3258.60 | 5.07 | Block entry | No |
| 4 | 430519.96 | 6743018.51 | 3261.94 | 10.48 | Original | Yes |
| 5 | 430476.83 | 6742984.36 | 3272.31 | 66.46 | Block exit | No |
| 6 | 430476.83 | 6742984.36 | 3272.31 | 66.46 | Block entry | No |
| 7 | 430472.21 | 6742980.70 | 3273.43 | 72.46 | Block exit | No |
| 8 | 430472.21 | 6742980.70 | 3273.43 | 72.46 | Block entry | No |
| 9 | 430433.49 | 6742950.06 | 3282.74 | 122.71 | Original | Yes |
| 10 | 430482.14 | 6742984.29 | 3276.51 | 182.53 | Block exit | No |
| 11 | 430482.14 | 6742984.29 | 3276.51 | 182.53 | Block entry | No |
| 12 | 430530.62 | 6743018.41 | 3270.30 | 242.14 | Original | Yes |
| 13 | 430489.08 | 6742984.14 | 3281.98 | 297.24 | Block exit | No |
| 14 | 430489.08 | 6742984.14 | 3281.98 | 297.24 | Block entry | No |
| 15 | 430480.53 | 6742977.08 | 3284.39 | 308.59 | Original | Yes |
| 16 | 430480.53 | 6742977.08 | 3284.39 | 308.59 | Block exit | No |
| 17 | 430447.38 | 6742949.73 | 3293.71 | 352.56 | Block entry | No |

4. Click the **Tools** button and then select **Smooth trajectory**. The **Smooth Trajectory** dialog box is displayed:

| # | Type | Protected | x (m) | y (m) | z (m) | Measured de... | Radius r... | Angle rat... |
|----|------|--------------------------|-----------|------------|---------|----------------|-------------|--------------|
| 1 | ORI | <input type="checkbox"/> | 430511.72 | 6743018.58 | 3255.46 | 0.00 | 100 | 100 |
| 2 | EXI | <input type="checkbox"/> | 430515.71 | 6743018.54 | 3258.60 | 5.07 | 100 | 100 |
| 3 | ENT | <input type="checkbox"/> | 430515.71 | 6743018.54 | 3258.60 | 5.07 | 100 | 100 |
| 4 | ORI | <input type="checkbox"/> | 430519.96 | 6743018.51 | 3261.94 | 10.48 | 51 | 67 |
| 5 | EXI | <input type="checkbox"/> | 430476.83 | 6742984.36 | 3272.31 | 66.46 | 100 | 100 |
| 6 | ENT | <input type="checkbox"/> | 430476.83 | 6742984.36 | 3272.31 | 66.46 | 100 | 100 |
| 7 | EXI | <input type="checkbox"/> | 430472.21 | 6742980.70 | 3273.43 | 72.46 | 100 | 100 |
| 8 | ENT | <input type="checkbox"/> | 430472.21 | 6742980.70 | 3273.43 | 72.46 | 100 | 100 |
| 9 | ORI | <input type="checkbox"/> | 430433.49 | 6742950.06 | 3282.74 | 122.71 | 94 | 6 |
| 10 | EXI | <input type="checkbox"/> | 430482.14 | 6742984.29 | 3276.51 | 182.53 | 100 | 100 |
| 11 | ENT | <input type="checkbox"/> | 430482.14 | 6742984.29 | 3276.51 | 182.53 | 100 | 100 |
| 12 | ORI | <input type="checkbox"/> | 430530.62 | 6743018.41 | 3270.30 | 242.14 | 56 | 8 |
| 13 | EXI | <input type="checkbox"/> | 430489.08 | 6742984.14 | 3281.98 | 297.24 | 100 | 100 |
| 14 | ENT | <input type="checkbox"/> | 430489.08 | 6742984.14 | 3281.98 | 297.24 | 100 | 100 |
| 15 | EXI | <input type="checkbox"/> | 430480.53 | 6742977.08 | 3284.39 | 308.59 | 100 | 100 |
| 16 | ENT | <input type="checkbox"/> | 430480.53 | 6742977.08 | 3284.39 | 308.59 | 100 | 100 |
| 17 | ORI | <input type="checkbox"/> | 430447.38 | 6742949.73 | 3293.71 | 352.56 | 38 | 70 |

The row colors are interpreted as follows:

| Colour | Meaning |
|---------------|---|
| Yellow | In the case of a single cell, this means a read-only cell. |
| Blue | A warning that the node is one or more of the following: <ul style="list-style-type: none"> • Far from its neighbors • Out of order • Exit/Entry/Repeat node |
| Yellow to Red | If the node has a problem, the entire row will be shaded from yellow through orange to red, with yellow being the least problematic and red being the most problematic. |

The node types are as follows:

| Node Type | Meaning |
|-----------|---|
| ENT | Block entry node, calculated and added by Builder |
| EXI | Block exit node, calculated and added by Builder |
| INP | Row entered by interpolation (see step 11) |
| INS | Row manually inserted (see step 7) |
| ORI | Original-type node which represents the actual trajectory path and which is preserved through all calculations. Other types of nodes can be added or removed to accommodate grid block intersections and varying perforation intervals. Users need to provide at least two original-type nodes to run calculations. |
| PER | Node is perforated |

Notes:

- a. You can select **Show problem nodes (with adjacent)** at the lower left to show only the nodes that have problems. Nodes adjacent to problem nodes will also be displayed.
- b. If **Radius ratio** or **Angle ratio** exceeds 100, they will be shown as 100.
- c. If you select the **Protected** check box in a row, then the x, y and z coordinates of that trajectory node will not be changed or deleted during smoothing operations.

5. Click the **Clean** button. The **Smooth Trajectory – Clean** dialog box will be displayed:



Notes:

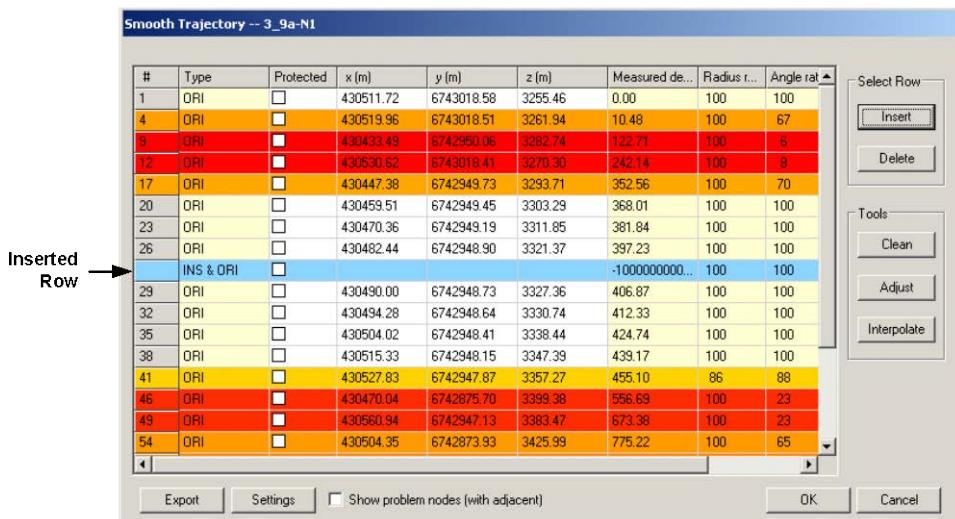
- a. At least one of **Entry nodes** or **Exit nodes** has to be checked.
 - b. If you select **Linear nodes**, this will delete all types of linear nodes.
 - c. If you select **Linear nodes** and then select **Keep original nodes**, even if an original node is a linear node, it will not be deleted.
 - d. If you are smoothing a multilateral trajectory, entry and exit nodes should not both be cleaned or the origin will be shifted. In this case, select either **Entry nodes** or **Exit nodes**.
6. Configure the clean settings. Click **Clean** to remove the selected node types. If you click **Cancel** you will lose the changes you made to the **Smooth Trajectory – Clean** dialog box and will be returned to the **Smooth Trajectory** table without cleaning. If you clicked **Clean**, the clean settings will be applied to the **Smooth Trajectory** table, as shown in the following example:

| # | Type | Protected | x [m] | y [m] | z [m] | Measured de... | Radius r... | Angle rat... |
|----|------|--------------------------|-----------|------------|---------|----------------|-------------|--------------|
| 1 | ORI | <input type="checkbox"/> | 430511.72 | 6743018.58 | 3255.46 | 0.00 | 100 | 100 |
| 4 | ORI | <input type="checkbox"/> | 430519.96 | 6743018.51 | 3261.94 | 10.48 | 100 | 67 |
| 9 | ORI | <input type="checkbox"/> | 430433.49 | 6742950.06 | 3282.74 | 122.71 | 100 | 6 |
| 12 | ORI | <input type="checkbox"/> | 430530.62 | 6743018.41 | 3270.30 | 242.14 | 100 | 8 |
| 17 | ORI | <input type="checkbox"/> | 430447.38 | 6742949.73 | 3293.71 | 352.56 | 100 | 70 |
| 20 | ORI | <input type="checkbox"/> | 430459.51 | 6742949.45 | 3303.29 | 368.01 | 100 | 100 |
| 23 | ORI | <input type="checkbox"/> | 430470.36 | 6742949.19 | 3311.85 | 381.84 | 100 | 100 |
| 26 | ORI | <input type="checkbox"/> | 430482.44 | 6742948.90 | 3321.37 | 397.23 | 100 | 100 |
| 29 | ORI | <input type="checkbox"/> | 430490.00 | 6742948.73 | 3327.36 | 406.87 | 100 | 100 |
| 32 | ORI | <input type="checkbox"/> | 430494.28 | 6742948.64 | 3330.74 | 412.33 | 100 | 100 |
| 35 | ORI | <input type="checkbox"/> | 430504.02 | 6742948.41 | 3338.44 | 424.74 | 100 | 100 |
| 38 | ORI | <input type="checkbox"/> | 430515.33 | 6742948.15 | 3347.39 | 439.17 | 100 | 100 |
| 41 | ORI | <input type="checkbox"/> | 430527.83 | 6742947.87 | 3357.27 | 455.10 | 86 | 88 |
| 46 | ORI | <input type="checkbox"/> | 430470.04 | 6742875.70 | 3399.38 | 556.69 | 100 | 23 |
| 49 | ORI | <input type="checkbox"/> | 430560.94 | 6742847.13 | 3383.47 | 673.38 | 100 | 23 |
| 54 | ORI | <input type="checkbox"/> | 430504.35 | 6742873.93 | 3425.99 | 775.22 | 100 | 65 |
| 57 | ORI | <input type="checkbox"/> | 430447.15 | 6743068.74 | 3436.65 | 978.53 | 100 | 57 |

On the right side of the table are buttons for 'Select Row', 'Insert', 'Delete', 'Tools', 'Clean', 'Adjust', and 'Interpolate'. At the bottom are 'Export', 'Settings', and 'OK' buttons.

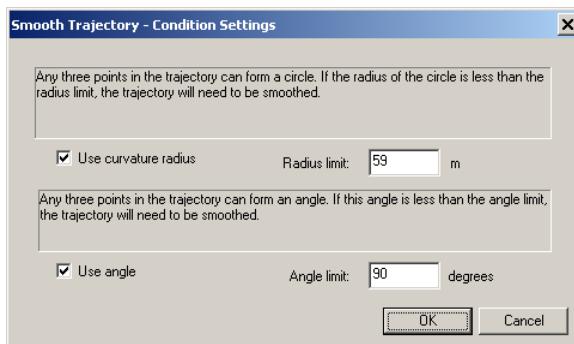
7. You can manually delete or insert rows in the table. Select a row by clicking its number in the left side and then clicking **Delete** to remove it from the table. Select a row and then click **Insert** to insert a row above it. The inserted row will appear, populated as shown in the following example. You will need to manually enter values for the x, y and z coordinates.

Note: Hold down the CTRL or SHIFT keys then click to define multiple nodes to delete or the number of rows to insert.



The screenshot shows a table titled "Smooth Trajectory -- 3_9a-N1" with columns: #, Type, Protected, x (m), y (m), z (m), Measured de..., Radius r..., Angle rat. The table contains 26 rows of data. A row labeled "INS & DRI" is highlighted in blue and is positioned between rows 26 and 27. An arrow on the left points to this row, labeled "Inserted Row". To the right of the table are buttons for "Select Row" (with "Insert" and "Delete" options), "Tools" (with "Clean", "Adjust", and "Interpolate" options), and buttons for "Export", "Settings", and "OK/Cancel".

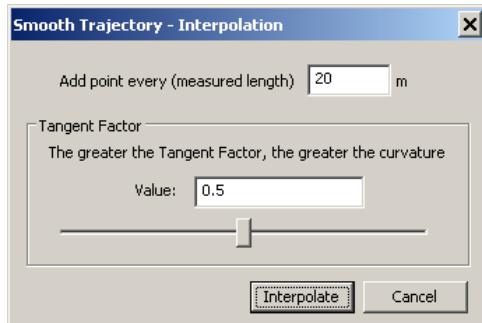
8. Click the **Settings** button at the lower left to open the **Smooth Trajectory - Condition Settings** dialog box:



9. Select the smoothing mechanism or mechanisms, enter the associated limits, and then click **OK**.
10. Click **Adjust** to smooth the trajectory as specified in the **Smooth Trajectory - Condition Settings** dialog box. The x, y and z coordinates of the trajectory nodes will be adjusted and all problematic rows addressed as shown in the following example:

| # | Type | Protected | x (m) | y (m) | z (m) | Measured de... | Radius r... | Angle rat... |
|----|------|--------------------------|-----------|------------|---------|----------------|-------------|--------------|
| 1 | ORI | <input type="checkbox"/> | 430511.72 | 6743018.58 | 3255.46 | 0.00 | 100 | 100 |
| 2 | ORI | <input type="checkbox"/> | 430501.80 | 6743012.65 | 3255.37 | 10.48 | 100 | 100 |
| 3 | ORI | <input type="checkbox"/> | 430487.83 | 6743001.87 | 3258.36 | 122.71 | 100 | 100 |
| 4 | ORI | <input type="checkbox"/> | 430477.03 | 6742989.67 | 3265.57 | 242.14 | 100 | 100 |
| 5 | ORI | <input type="checkbox"/> | 430470.34 | 6742977.19 | 3276.29 | 352.56 | 100 | 100 |
| 6 | ORI | <input type="checkbox"/> | 430468.35 | 6742965.54 | 3289.46 | 368.01 | 100 | 100 |
| 7 | ORI | <input type="checkbox"/> | 430471.29 | 6742955.51 | 3303.74 | 381.84 | 100 | 100 |
| 8 | ORI | <input type="checkbox"/> | 430476.98 | 6742945.82 | 3319.45 | 397.23 | 100 | 100 |
| 9 | ORI | <input type="checkbox"/> | 430484.59 | 6742945.74 | 3330.50 | 406.87 | 100 | 100 |
| 10 | ORI | <input type="checkbox"/> | 430488.30 | 6742942.99 | 3339.25 | 412.33 | 100 | 100 |
| 11 | ORI | <input type="checkbox"/> | 430491.35 | 6742939.67 | 3348.85 | 424.74 | 100 | 100 |
| 12 | ORI | <input type="checkbox"/> | 430493.88 | 6742935.73 | 3359.37 | 439.17 | 100 | 100 |
| 13 | ORI | <input type="checkbox"/> | 430496.56 | 6742931.72 | 3370.64 | 455.10 | 100 | 100 |
| 14 | ORI | <input type="checkbox"/> | 430498.43 | 6742926.85 | 3384.18 | 556.69 | 100 | 100 |
| 15 | ORI | <input type="checkbox"/> | 430499.99 | 6742920.55 | 3400.85 | 673.38 | 100 | 100 |
| 16 | ORI | <input type="checkbox"/> | 430500.66 | 6742917.76 | 3421.81 | 775.22 | 100 | 100 |
| 17 | ORI | <input type="checkbox"/> | 430458.73 | 6743025.75 | 3438.78 | 978.53 | 100 | 100 |

11. To interpolate data between nodes, click **Interpolate**. The **Smooth Trajectory – Interpolation** dialog box will be displayed:



Set the spacing for the interpolation points and the tangent factor, as described in [Interpolating between nodes](#). You can type the **Tangent Factor** directly in the **Value** box or use the slider to adjust it. Once you have adjusted it, click **Interpolate**. Data will be interpolated in accordance with the interpolation settings, and you will be returned to the **Smooth Trajectory** table. If you click **Cancel** in the **Smooth Trajectory – Interpolation** dialog box, you will lose your interpolation settings and the trajectory will not be interpolated.

Note: If you have resolved all problem nodes, the **Show problem nodes (with adjacent)** check box will be disabled:

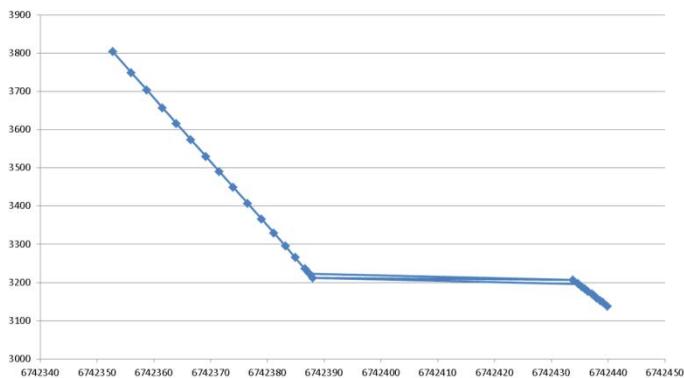
The dialog box shows the "Show problem nodes (with adjacent)" checkbox, which is currently unchecked. A red arrow points to this checkbox.

12. If you are finished with smoothing the trajectory, click **OK** to return to the **Trajectory Properties** dialog box. Before you click **OK** or **Apply**, you can click **Reset Page** to restore the original trajectory to the **Trajectory Properties** table. If you want to replace the original trajectory with the smoothed trajectory, click **OK** or **Apply**.
13. If you want to save the smoothed trajectory to a Builder-formatted trajectory file (.wdb) for later import, click **Export**. You may want to do this if, for example, you are not ready to commit the smoothed trajectory to the dataset.

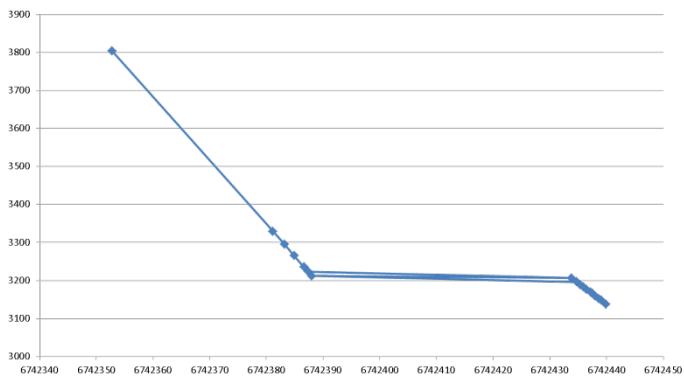
Trajectory Smoothing Example

To illustrate the trajectory smoothing process, consider the following example.

Trajectory before cleaning:



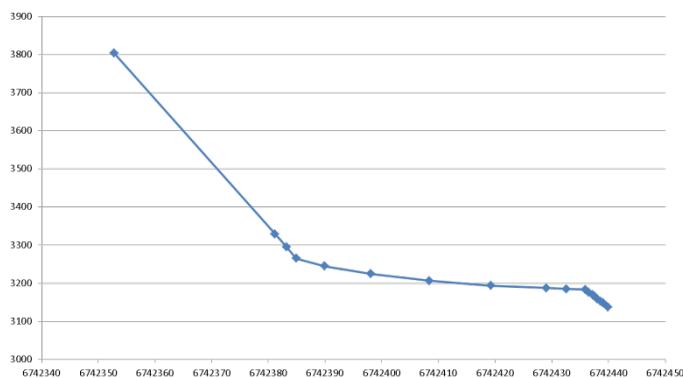
Trajectory after cleaning:



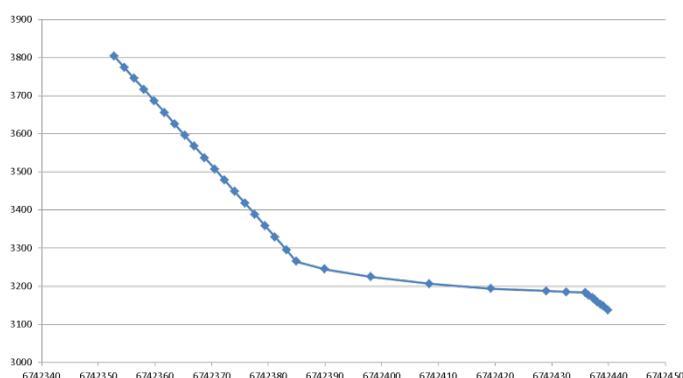
Trajectory after cleaning, zoomed in:



Trajectory after adjusting:



Trajectory after interpolation:

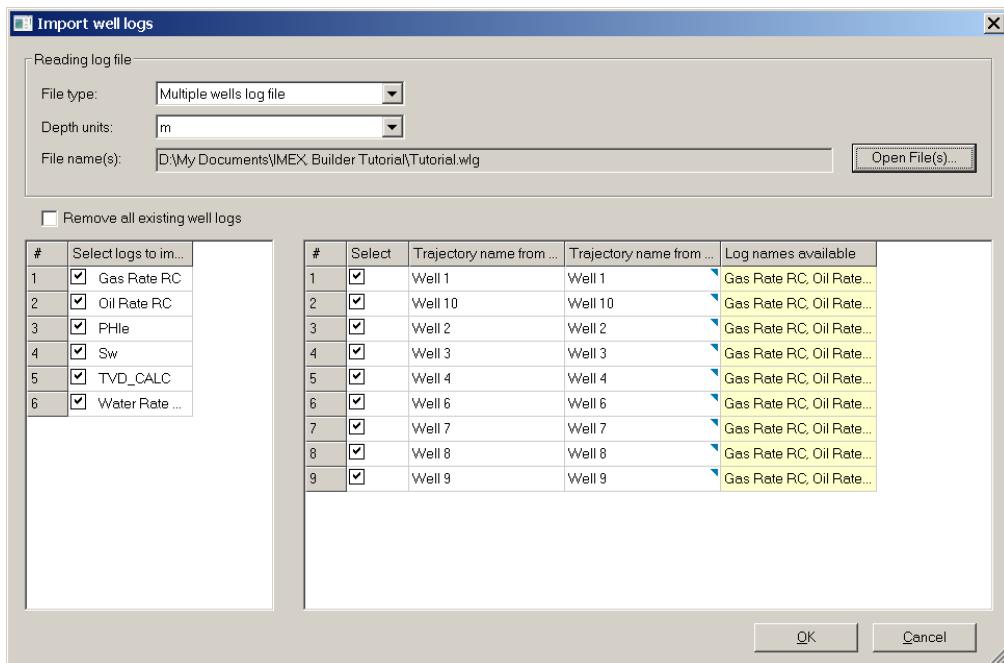


Well Logs Import

Builder has an option to import well log files. Currently log information can be displayed only on the probe window when moving cursor along the well trajectory in 2D or 3D view. Well logs can also be used when creating contour maps. To import wells logs:

1. Select menu item **Well | Well Trajectories | Import Well Logs** to open the Import well logs dialog box.
2. Select the type of well log file and depth units.
3. Use **Open File** button to browse to the well log file(s). Multiple log files can be opened at once. You can repeat this step a number of times to read in different files.
4. Select the names of the well logs you want to import in the left side table.
5. Select the names of the trajectories to receive the imported logs.
6. Press **OK**.

The supported file formats for well log information are described in [Well Log Formats](#) on page 573.



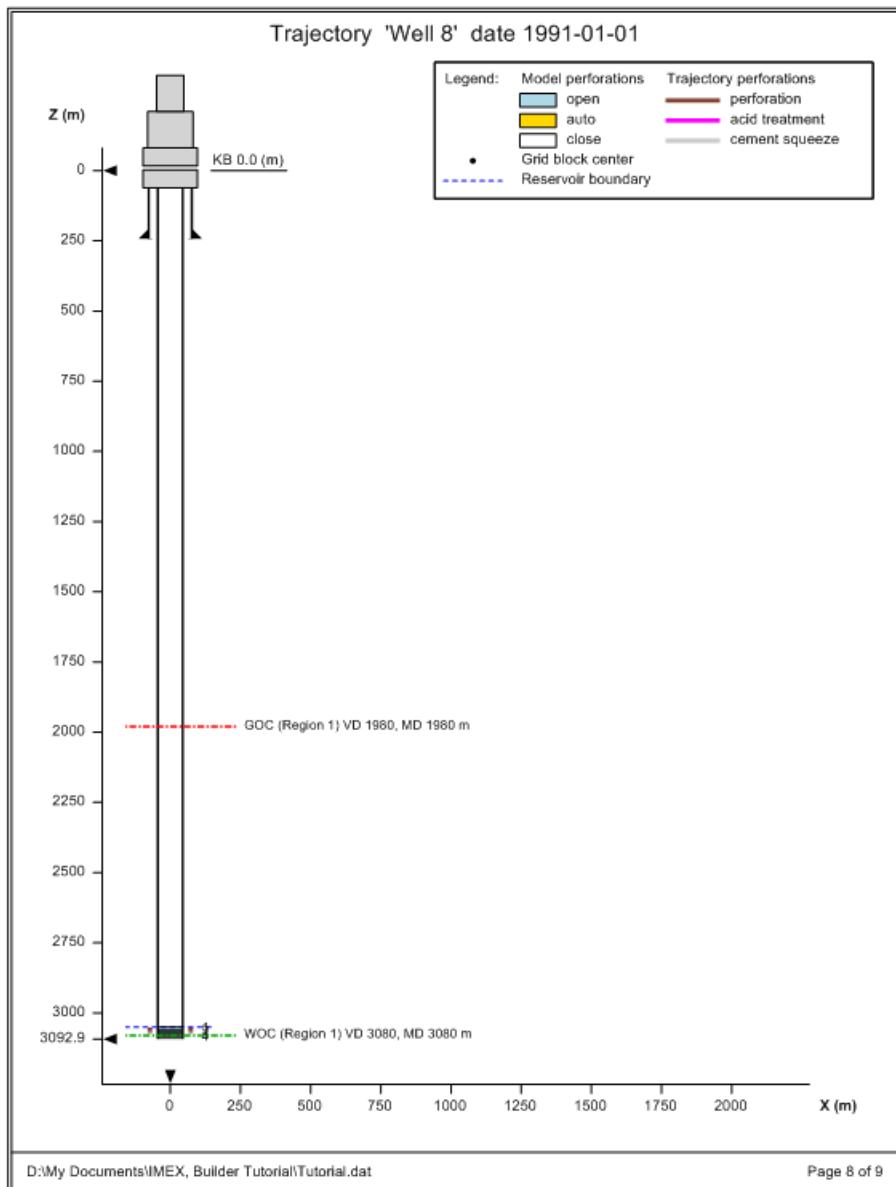
Wellbore Diagram

The Wellbore diagram displays well trajectories and completions in a simple 2D view for quick visual understanding and quality checking of well data. The Wellbore diagram has the following display features:

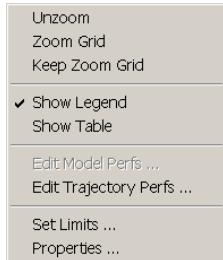
- Well path along the axis with true depth and deviation
- Trajectory perforation intervals (as lines along the outer sides of the well path)
- Intersected and completed grid blocks (as filled sections inside the well path)
- Intersected fluid contacts (GOC, WOC, WGC)
- Floating table that shows values of intersected grid blocks and perforation intervals
- Title, legend, page header and footer
- Context menu with control commands available on right mouse click
- Property control panel to change display settings
- Printing, image file export, copying metafile image on clipboard
- Zooming

Open the wellbore view by selecting **Open Wellbore Diagram** from the **Well** menu or from the context menu after right-clicking a trajectory or well completion in the tree view. To change the well in the wellbore diagram, select a different trajectory or completion item in the tree view.

Note: For multilateral well completions, only the main branch is displayed on this diagram.



The Wellbore view context menu is shown below:



Group Settings

Groups are used to simulate gathering centers, groups and platforms, set production injection or cycling constraints and receive output for multiple wells. Certain rules apply for creating group structure according to the simulator syntax:

- Groups must be arranged in 2 or 3 levels
- Top level group (field) can attach only other groups and no wells
- Second level group can attach wells or groups but not both at the same time
- Third level group can attach wells only

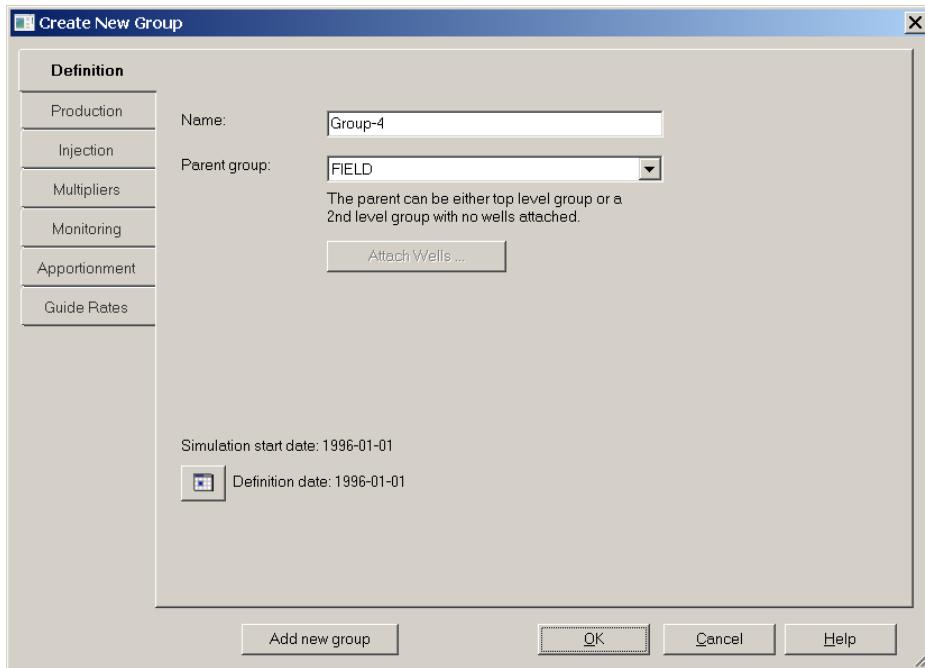
Note: When all groups have been defined using GROUP cards and all group constraints have set, any later specification of a previous or new group using a GROUP keyword requires you to completely re-specify all group actions, properties and constraints. For ease of use it is recommended that all groups be defined on one date.

Group structure is shown on the tree view under Groups root items.

- icon: Icon for groups attaching other groups
- icon: Icon for groups attaching wells
- icon: Icon for groups with no attachments

Adding New Groups

To add a new group to the dataset, select **Group New** from **Well** menu or select **New** from the context menu after right-clicking on Groups item on the tree view. The **Create New Group** dialog box will be displayed:



1. Enter a unique group name.
2. Select parent group.
3. Use the **Attach Wells** button to select attached well names.
4. Set the group definition date.
5. Enter group production, injection and monitor constraints, if needed.
6. Press **OK** or **Add new group**.

Clicking **Add new group** creates the new group and immediately resets all controls for adding next one so that you do not have to reopen this window multiple times.

Deleting Groups

1. Select a group item on the tree view
2. Right-click on the group item to open the context menu
3. Select **Delete**.
4. Click **OK** in confirmation box

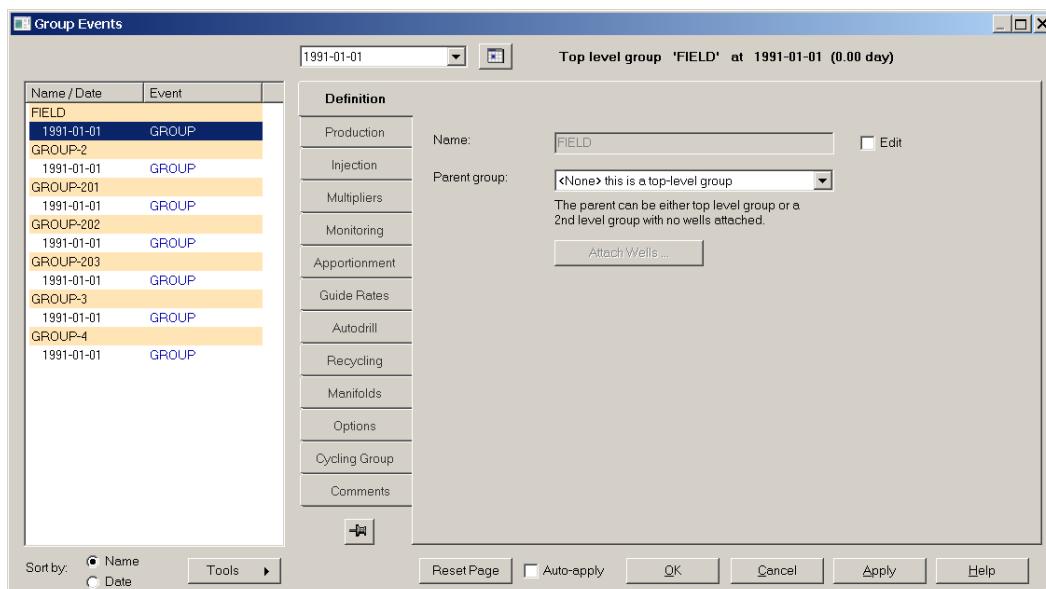
Note: If the deleted group has other groups attached to it, the attached groups will also be deleted. No wells are removed in this process.

Group Events

A Group event is a collection of data items related to a particular group that includes group definition, constraints, constraint multipliers, recycling and other properties, at a particular time. Each group event has a corresponding keyword in the simulator dataset; thus, the order and content of group events is a reflection of the dataset syntax.

Builder has a single dialog window called Group Events that provides controls to add new events or the edit and delete existing events. There are several ways to open this window:

- Select **Group Events** from the Well menu.
- Double-click the particular event item on the tree or time-line view.
- Right-click on a particular event item on the tree or time-line view then select **Properties** from the context menu.

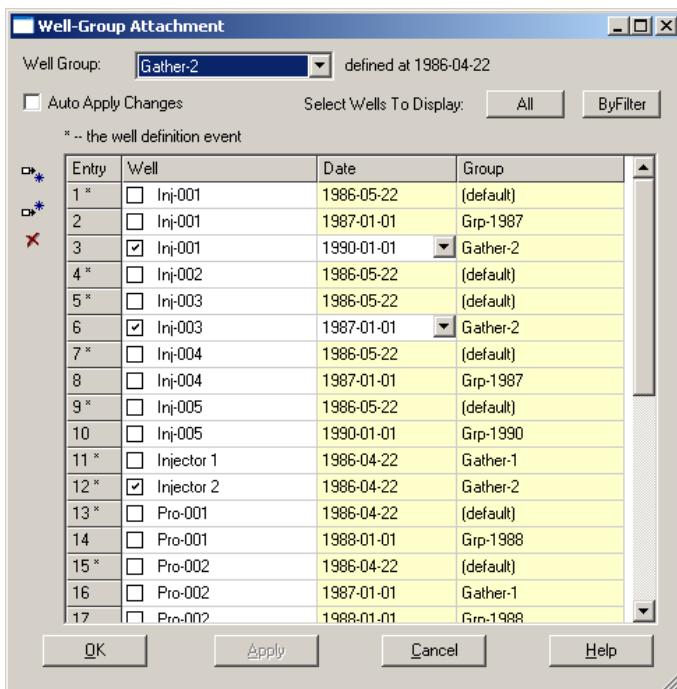


The list control on the left side of the **Group Events** dialog box contains all group events defined by the keyword existing in the dataset. The list can be sorted by group name (default mode) or by date depending on the selection below the list.

The right side of the **Group Events** dialog box shows controls to set the event data arranged on several tabs. When you select a group event in the list control the corresponding tab page is shown bringing the event settings to display. Current group name and date is printed at the very top of the window in bold font.

Adding Wells to Group

Open the **Well-Group Attachment** dialog box by clicking the **Attach Wells** button on the **Definition** tab of the **Group Events** dialog box:



The **Well Group** drop-down box displays the group currently selected (*Gather-2* in this case). The main grid area will update accordingly if the current group selection is changed. The group's definition date is displayed on the right side of the drop-down box (1986-04-22 in the above figure).

There are two buttons, **All** and **ByFilter**, in the **Select Wells To Display** area. Clicking **All** will select all available wells and **ByFilter** will open the **Well Selection** dialog box through which you can select the wells based on different criteria.

If **Auto Apply Changes** is checked, changes made to the current selected group will be applied automatically after the group selection changes or one of the **All** or **ByFilter** buttons is clicked. Otherwise, you will be asked to save the changes or discard them.

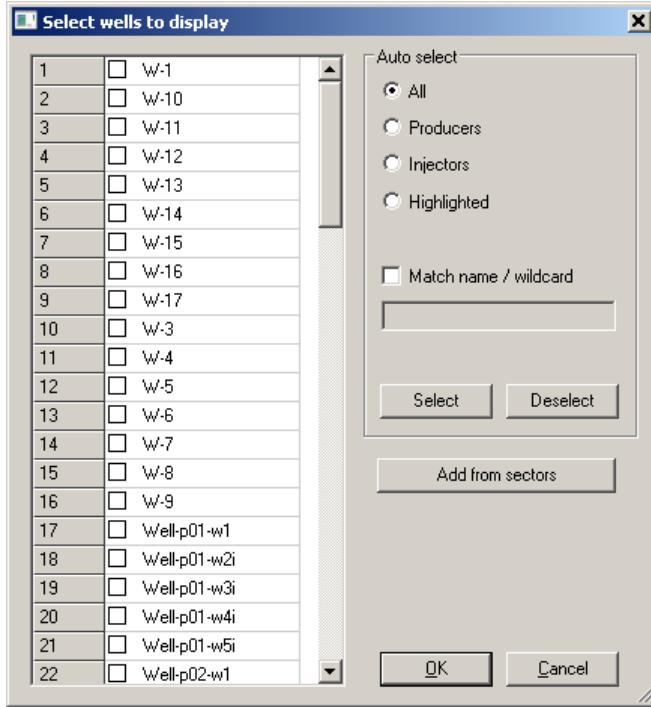
The grid has four columns:

- **Entry** – Entry ID. Read-only. The entry ID with * on the right side indicates that this entry is a well definition. This entry cannot be deleted from this dialog box. Its date cannot be changed either. You can only check or uncheck this entry (via the 2nd column).
- **Well** – Well name. The check box will be checked if the well already belongs to the current selected group at the date shown in the **Date** column.
 - Select the check box to add this well to the current group at the date shown in the **Date** column.
 - Clear the check box to remove this well from the current group at the date shown in the **Date** column.
 - If the date in the **Date** column is not a well definition date, you can also change the date at which you want the well added to the current group.
 - If the current selected group is the default group you cannot clear the check box to remove it from the default group.
- **Date** – The date at which the well is added to the group indicated by the **Group** column cell. You can select date if this entry is not a well definition entry and the **Well** column cell is checked.
- **Group** – Group name. Read-only.

The three tool buttons on the left side of the grid – , , and  – are used, respectively, to insert a new entry before the currently selected entry, to insert a new entry after the currently selected entry, and to delete the currently selected entry.

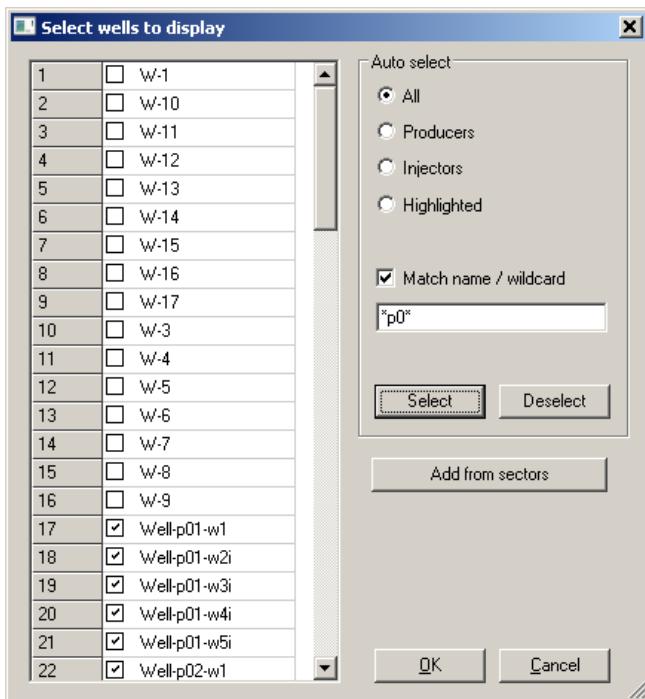
Using the ByFilter Button

1. In the **Well-Group Attachment** dialog box, click the **ByFilter** button to open the **Select wells to display** dialog box:



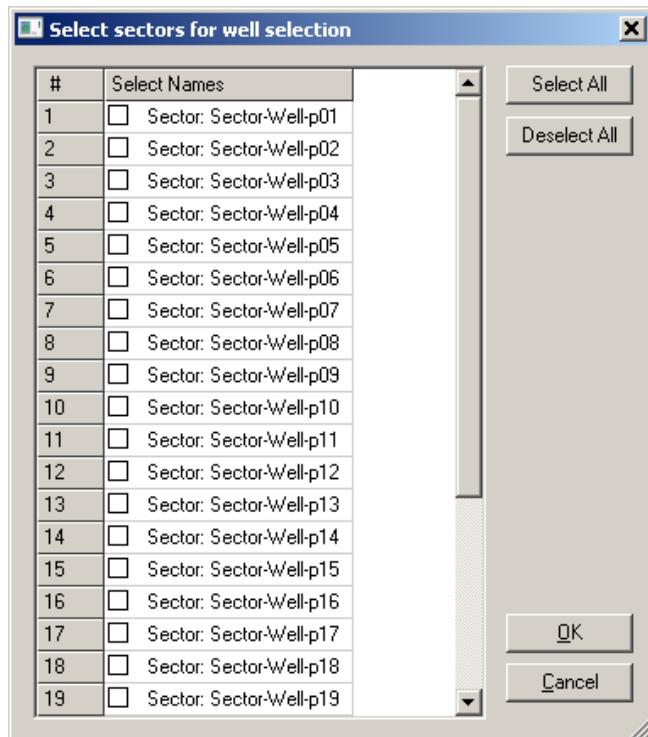
2. You can select wells from the table in the following ways:
 - a) Click wells individually.
 - b) Use the radio buttons in the **Auto select** pane to select wells. For example, click **All** then **Select** to select all wells. All wells will be selected in the table. Alternately, to select Injectors (or Producers), check **Injectors** (or **Producers**) then click **Select**. The injector (or producer) wells will be selected in the table.
 - c) Similarly, select **Highlighted** in the **Auto select** pane, click and drag your mouse over the well names in the table to define a range (or hold down CTRL or SHIFT and click the right mouse button to define the range) then click **Select**.
 - d) Click the **Add from sectors** button to select wells from sectors, as shown below.

- e) Select **Match name / wildcard** then enter the match name or name fragment, using the “*” wildcard as necessary, then click **Select**, to select wells on the basis of their names. This is illustrated in the following example, in which cells with names containing the fragment “p0” have been selected:



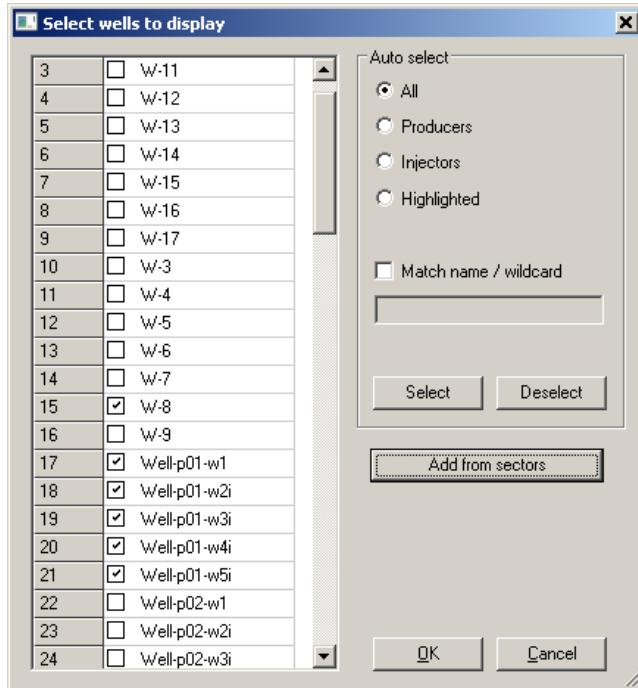
You can use the **Deselect** button to cancel your selections, once again based on the settings in the **Auto select** pane. For example, if you have selected **All**, clicking **Deselect** will cancel all of the selections. If you have selected **Highlighted** and then dragged your mouse to define a range of wells, clicking **Deselect** will cancel all selections in that range. Finally, if you have selected **Match name / wildcard** and have entered a name fragment, clicking **Deselect** will cancel selections where the well name contains this name fragment.

- To add wells from sectors, click **Add from sectors**. The **Select sectors for well selection** dialog box, which shows the sectors that have been defined for the field, will be displayed for selection. For example:



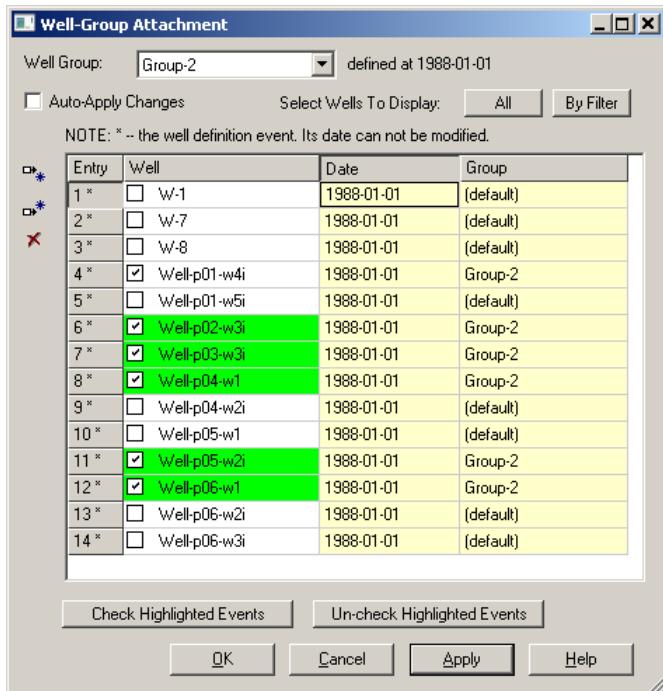
4. Click the sectors that contain the wells that you want to add to the Group. You can use **Select All** to select all of the sectors, and **Deselect All** to cancel all sector selections. Once you have completed your selection, click **OK**. You will be returned to the **Select wells to display** dialog box.

The wells in the sectors selected above will be checked, as shown in the following example:



- Once you have completed your well selection, click **OK**. The **Well-Group Attachment** dialog box will be displayed and all of the wells that you have selected, including those selected through the **ByFilter** button, will be displayed. Finally, select the wells that you want to attach to the Group.

As you select these wells, the cell background will change to green, as shown in the following example:



As shown above, the **Group** column displays the Group to which the well has been attached or to which it will be attached. You can select a range of wells for attachment by dragging the mouse over their names then clicking the **Check Highlighted Events** button. Likewise, you can hold down CTRL or SHIFT and click the right mouse button to select multiple wells before clicking **Check Highlighted Events**. These same operations apply to un-checking wells using the **Un-check Highlighted Events** button.

6. Click **OK** (or **Apply**, if you want to save the changes but keep the **Well-Group Attachment** dialog box open for further operations) to attach the wells to the Group.

Adding Group Events

1. Open the **Group Event** dialog box from the main **Well** menu or by using a double click or the Properties context menu on any event item of the tree or time-line view.
2. Select a group name in the list control on the left side of the window.
3. Select an existing date in the combo box at the top of the window or use the calendar button next to combo box to set a new date.

4. Select a tab that has the setting for the corresponding event and set the event check box ON to enables input of data.
5. Fill in event data settings.
6. Click **Apply** or **OK** to validate and enter new settings. If the **Apply** button is clicked then the dialog window stays open and the event list is updated. If **OK** button is clicked the dialog box will be closed immediately.

Changing events settings:

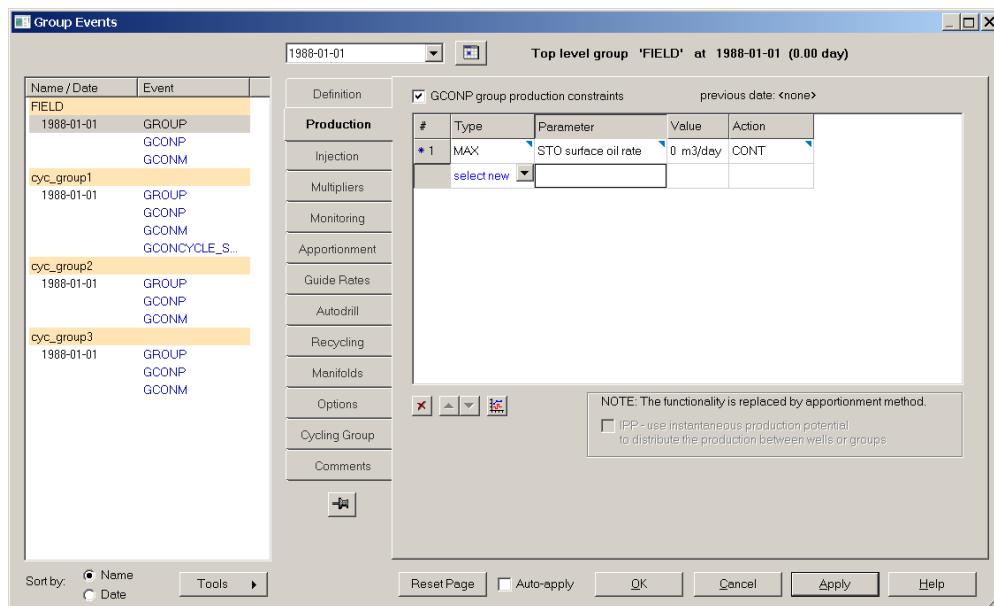
1. Open the **Group Event** dialog box.
2. Scroll the list control and select a particular group event. The corresponding tab page will display the event settings.
3. Enter new data values in the corresponding fields and then click **Apply** or **OK**.

Deleting group event:

1. Open the **Group Event** dialog box.
2. Scroll the list control and select a particular group event. The corresponding tab page will display the event settings.
3. Set check box OFF for selected event and then click **Apply** or **OK**.

List of Group Constraints

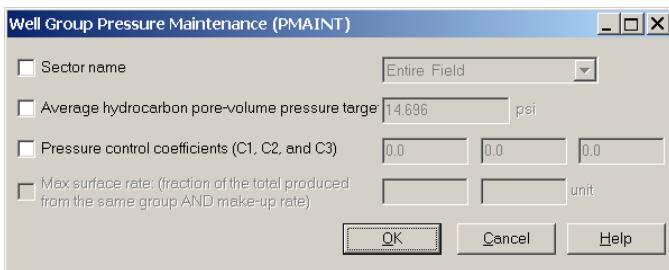
Type and number of group constraints is completely optional for production, injection or monitoring purpose. To add a new entry in the constraint list select a type of new constraint from the **select new** drop-down list box at the end of the list and enter other parameters on the new line.



Tool buttons below the list enable you to control the number and order of constraints:

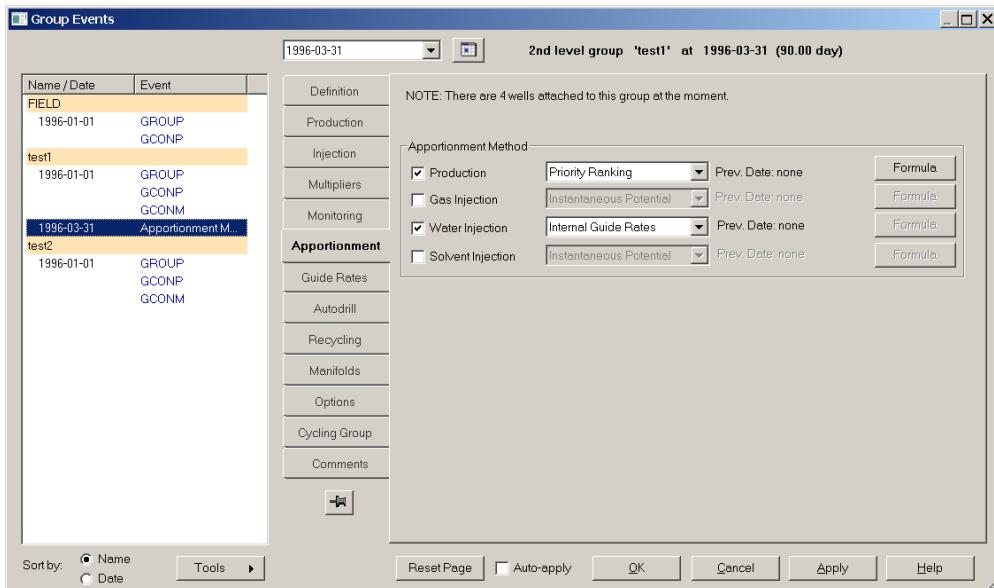
-  Delete selected constraint
-  Move selected constraint up on the list
-  Move selected constraint down on the list
-  Plot group constraints

The **PMAINT** (pressure maintenance) group constraint is a complex constraint. It is for both production and injection. The **Well Group Pressure Maintenance (PMAINT)** dialog box is used to define and/or modify this constraint. The dialog box can be launched by clicking the **Edit** button in the **PMAINT** constraint value column:



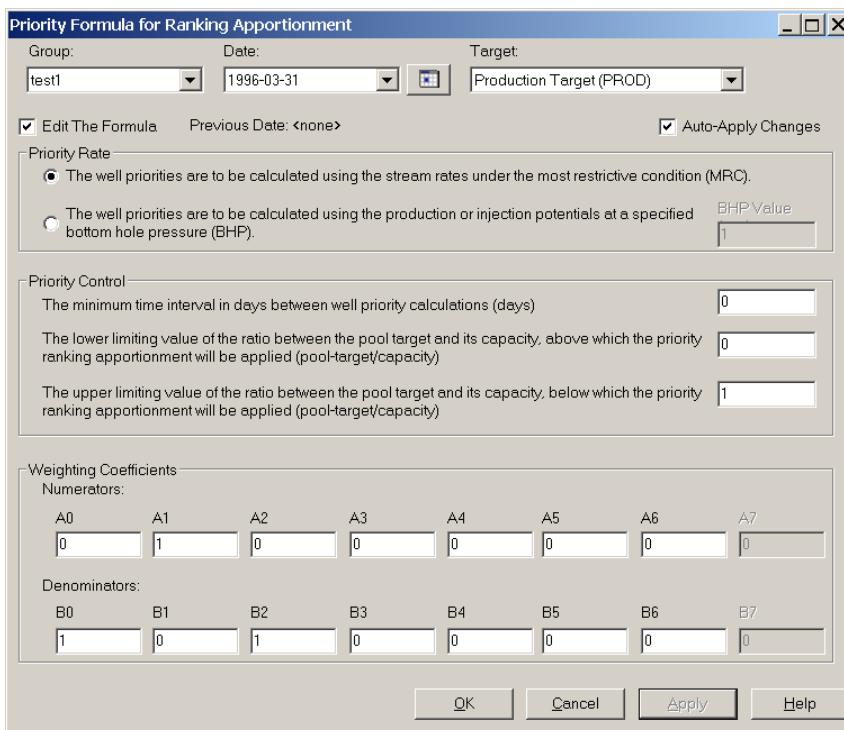
Group Apportionment Methods

For a well group there are four possible (one production and three injection) apportionment targets. For each target, you can specify one of the four available apportionment methods. If the Priority Ranking method is selected you need to enter the priority formula. In the dialog box, the group apportionment targets are displayed as check boxes and apportionment method is selected via the drop-down combo-box for this target. The required priority formula for priority ranking method will be specified through the **Priority Formula for Ranking Apportionment** dialog box, which is opened by clicking the **Formula** button for this target. The **Apportionment** tab is shown below:



You can also define a priority ranking formula for later use, by clicking **Formula** in the **Apportionment** tab.

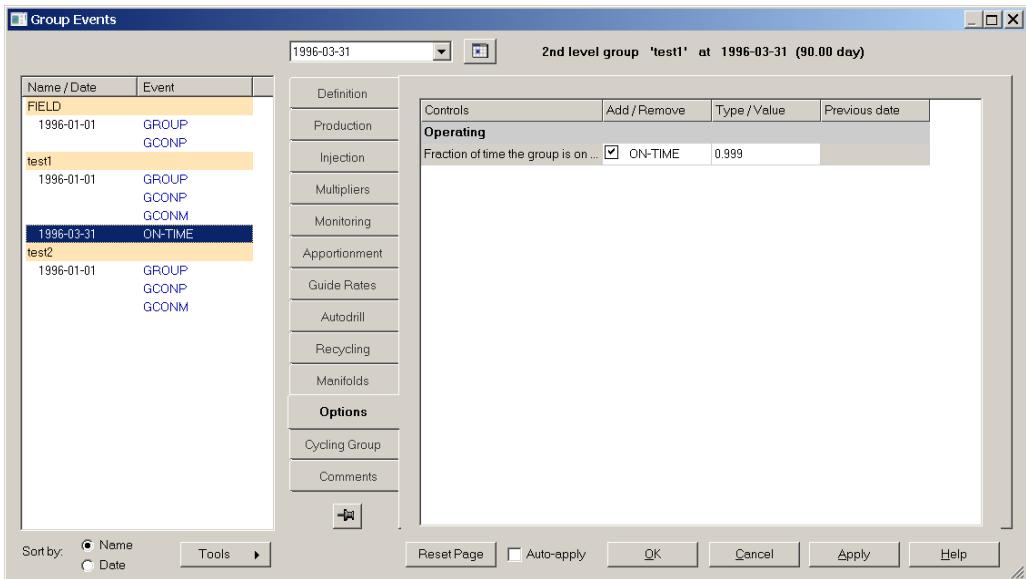
The priority formula can be defined through the **Priority Formula for Ranking Apportionment** dialog box, as shown in the following example:



Priority formulae are different for different targets and simulators. Refer the appropriate simulator manual for details.

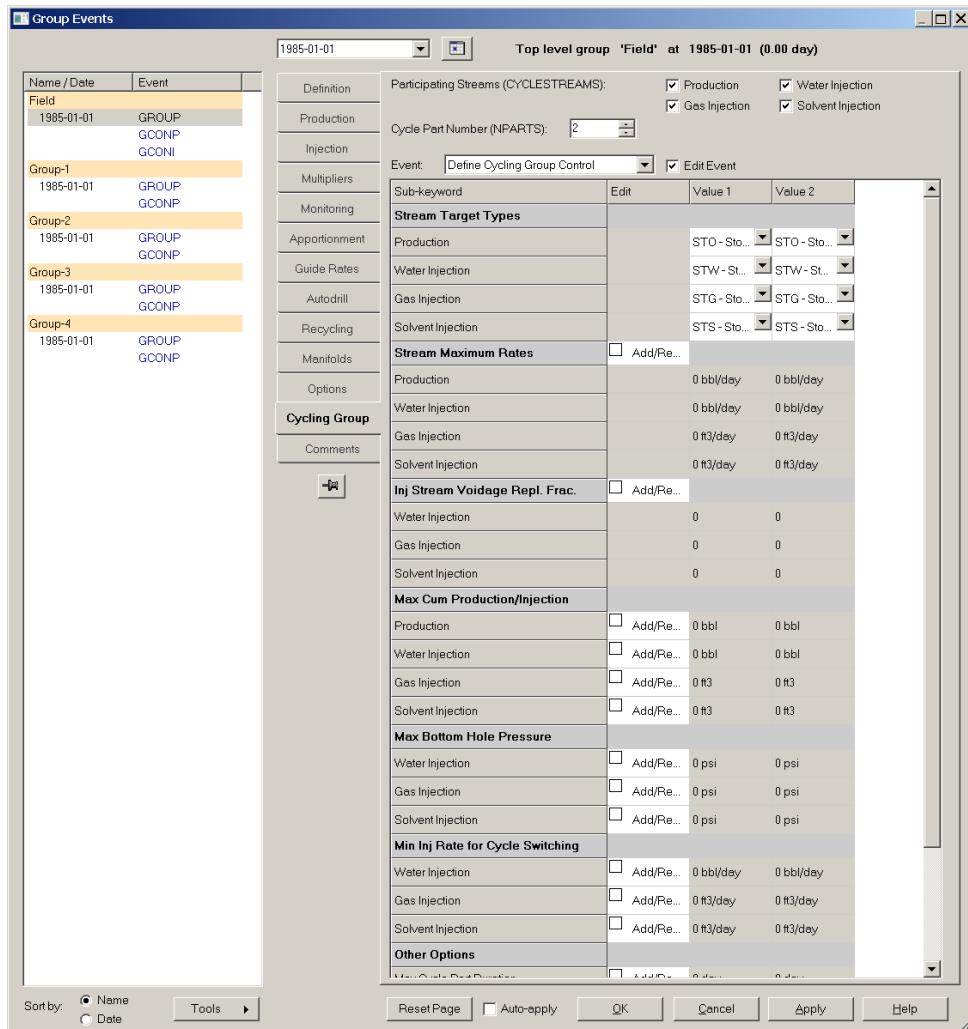
Group ON-TIME

This event can be defined via the **Options** tab, as shown in the following example:



Cycling Group

The purpose of the **Cycling Group** tab is to allow you to define and/or modify cycling group properties. Refer to the simulator manuals for a detailed description of cycling groups. The **Cycling Group** tab is shown in the following example:



For a cycling group, up to four types of fluid streams can be manipulated. They are production, water injection, gas injection, and solvent injection (IMEX only), and can be selected via the four check boxes on the upper-right corner of the **Cycling Group** tab.

The operation cycle of a cycling group consists of from 1 to 10 cycle parts. It is configured via the control **Cycle Part Number (NPARTS)**.

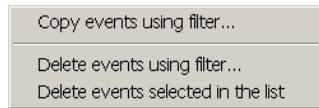
The **Event** combo-box is for you to select the cycling event to edit. There are five kinds of cycling events available.

The main part of the **Cycling Group** tab allows you to specify and/or modify various cycling operation quantities for each of the defined cycle parts. The number of available operation quantities will change based on the number of cycle parts specified, the number of participating streams selected and the simulator being used. Some of the settings are required, but they vary based on the simulator used. Refer to the appropriate simulator manual for more information.

Copying Group Events

New events are first created for one selected well group and date. It is very common that some events need to be repeated for other well groups and/or dates. In this case you can use group event copy function.

1. Open the **Group Event** dialog box.
2. Select the events that need to be copied (as a source) in the list control. The events to be copied must belong to a single well group and date.
3. Right-click in the list control or press the **Tools** button to bring the command menu:



4. Select **Copy events using filter**.
5. Set up search conditions and create a list of well group & date pairs that will be the destination for the new copies (for further information, refer to [Using the Group and Date Filter](#) on page 337).
6. Press **OK**.

Deleting Group Events

Deleting a single group event:

1. Open the **Group Event** dialog box.
2. Scroll the list control and select a particular well group event. The corresponding tab page will display the event settings.
3. Clear the check box for the selected event then click **Apply** or **OK**.

Deleting selected events quickly:

1. Open the **Group Event** dialog box.
2. Select one or more events in the list control using mouse and SHIFT or CTRL keys.
3. Right-click in the list control or click **Tools** to display the context menu.
4. Select **Delete events selected in the list**.
5. Click **Yes** in the confirmation box.

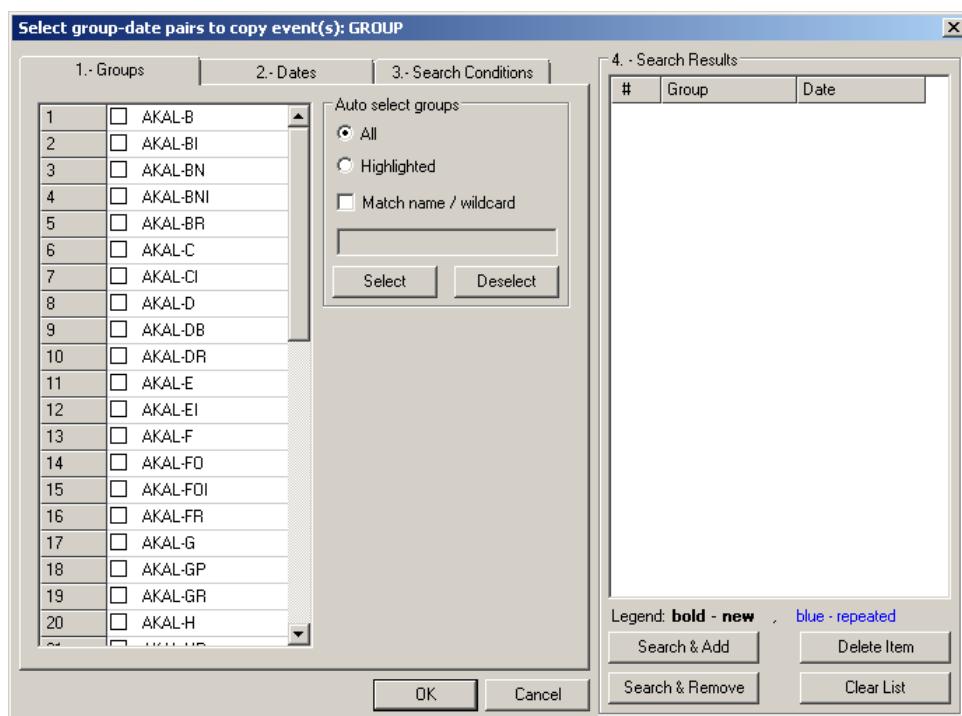
Deleting events of selected type using filter:

1. Open the **Group Event** dialog box.
2. Select events in the list control.
3. Right-click in the list control or click the **Tools** button to display the context menu.
4. Select **Delete events using filter**.

5. Set up the search conditions and create a list of well group & date pairs existing in the dataset that may have events to be removed (refer to [Using the Group and Date Filter](#) below).
6. Click **OK**.

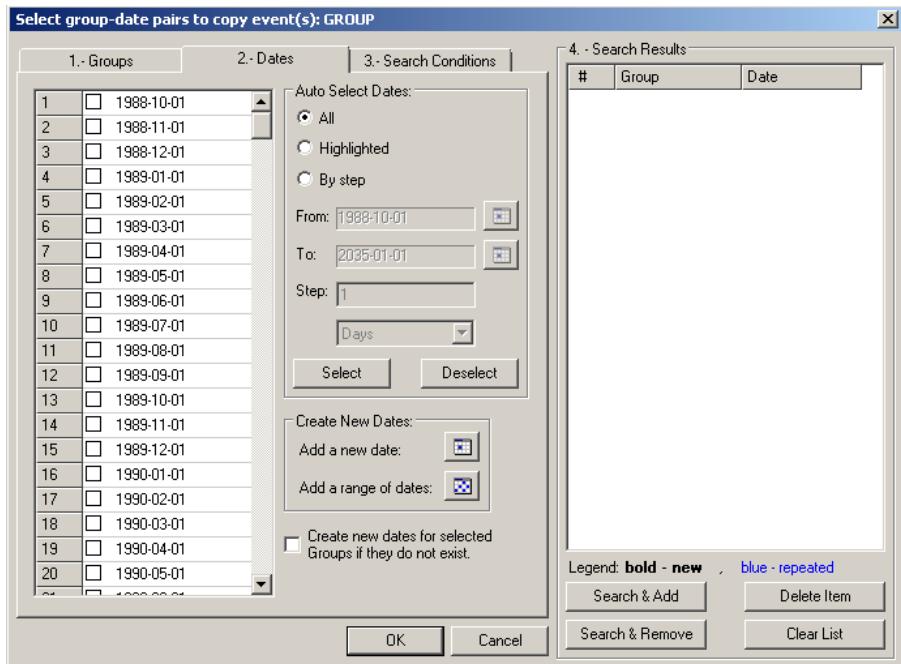
Using the Group and Date Filter

The **Group & Date Filter** dialog box is used to set up a list of group & date pairs for copying or removing well events. The dialog box opens when you select **Copy** or **Delete** using the filter in the **Group Event** panel. The following example illustrates the use of the filter for a copy operation. It works similarly in delete mode with the difference that it does not have an option of creating new dates.



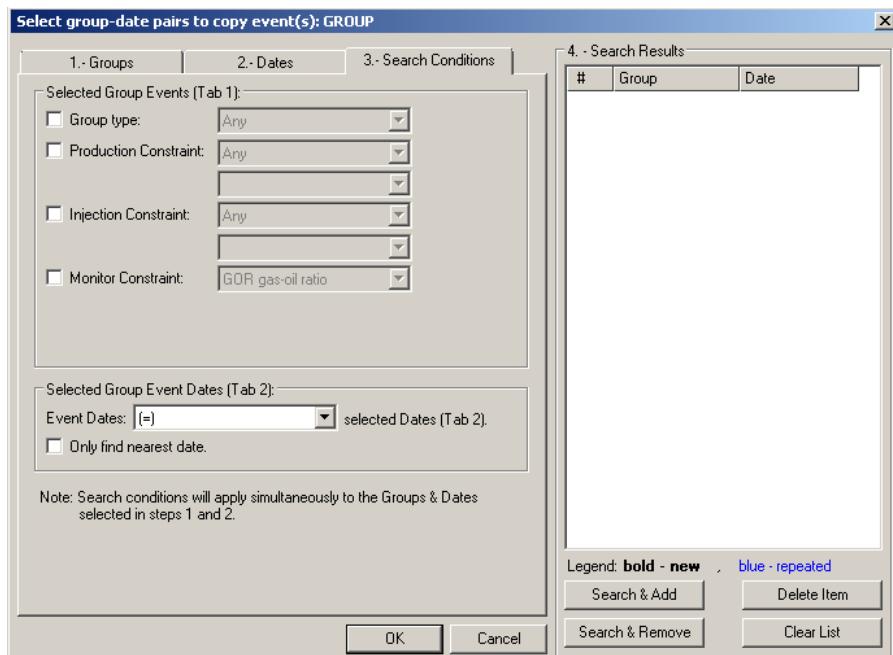
The list control on the right side of the window contains the selected group & date pairs, based on the applied search conditions. This list is preserved when you close and re-open the window. Initially this list is empty.

1. Select group names on the **Groups** tab. You can do this manually by checking the boxes or you can use the auto-select controls.
2. Select the **Dates** tab.



The table on the left side has a list of all existing group dates that you can select for filtering group & date pairs. If selected, the **Create new dates for selected Groups if they do not exist** check box will add any missing selected dates from the **Dates** tab to the selected groups from the **Groups** tab when the **Search & Add** button is clicked. The **Add a new date** and **Add a range of dates** buttons enable you to add new dates to the date list individually or as a range.

- The Search Conditions tab is shown below:



Through the **Search Conditions** tab you can set optional conditions to further filter the names and dates selected in steps 1 and 2. If no search conditions are set then all group & date pairs matching the selections from the **Groups** and **Dates** tabs will be included. Multiple search conditions apply together with AND type logic. On the **Dates** tab, the **Set on selected Dates** check box is used to select dates that have this property defined exactly on the date (check box selected), or at any time prior (check box cleared). The **Only find nearest date** check box will only select the first group date that matches the search conditions. The **Event Dates** combo box inside the **Selected Group Event Dates (Tab 2)** area further refines the date search by returning only those group event dates which are either : =, <, >, <=, >= to the dates selected through the **Dates** tab.

- Press the **Clear List** button (near bottom-right corner) to empty the previous search results and press **Search & Add** button to run the filter and create a new well & date list.
- Finally, press **OK** to copy the selected group events in the **Group Events** dialog box tree view to the selected group & date pairs shown in the **Search Results**.

Group Event Comments

1. Open the **Group Event** dialog box.
2. Select the group event in the list control on the left side of the dialog box.
3. Select the **Comments** tab to read or write new comments.

Note: If comments are available for the selected event, then there will be a check mark on the **Comments** tab. The current tab is automatically changed for each event selected in the list. A small push-pin  button below the **Comments** tab allows you to pin (fix) the current tab. This may help you to more efficiently view the comments while you are browsing the event list.

Group and Well Connection Display

The **Group & Wells Connections** dialog box, accessible through the **Well** menu, provides a tabular summary of all group and well connections.

Note: Group and well attachments are set by well and group definition events with WELL and GROUP keywords.

The **Group & Wells Connections** dialog box has two display modes:

- **Attach-to commands:** Shows a map of all attachment commands.
- **Group affiliation:** Shows all well and groups arranged by the parent.

Group & Well Connections

Table of group affiliation for groups and wells

Display mode:

- 'Attach-to' commands
- Group affiliation

Properties Print Grid Close

D:\temp\DataSet\Gem\GEM_ManyGWDTs.dat

Group - Well Connection (1983-12-01)

Legend: Group (orange), Producer (green), Injector (red), Undefined (grey)

```

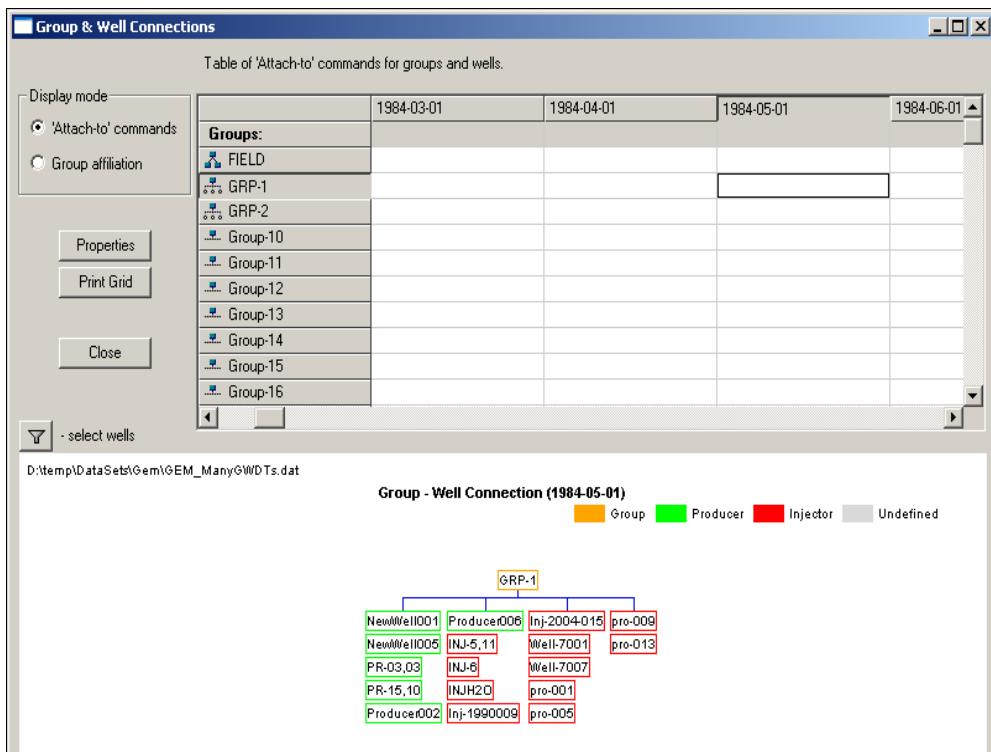
graph TD
    FIELD --> GRP1[GRP-1]
    FIELD --> GRP2[GRP-2]
    FIELD --> DefaultGroup[Default-Group]
    GRP1 --> INJ511[INJ-5,11]
    GRP1 --> INJ6[INJ-6]
    GRP2 --> INJH20[INJH20]
    GRP2 --> INJH20_2[INJH20]
    DefaultGroup --> INJ511_2[INJ-5,11]
    DefaultGroup --> INJ6_2[INJ-6]
    DefaultGroup --> INJH20_3[INJH20]
    DefaultGroup --> INJH20_4[INJH20]
    DefaultGroup --> INJH2C[INJH2C]
    DefaultGroup --> INJ1990009[INJ-1990009]
  
```

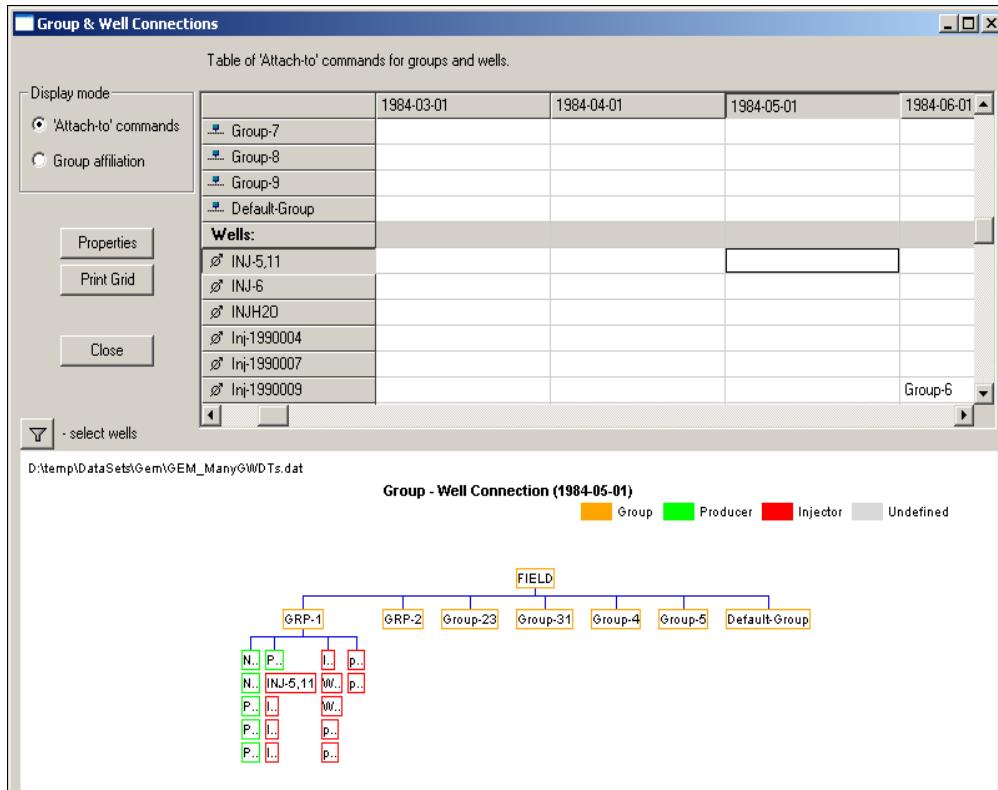
You can change the attachment setting for the item selected in the table by pressing the **Properties** button or context menu after a right-click. The corresponding well or group event dialog box will be opened. Click **Print** to make a printed copy of the table.

Note: “Default-Group” is added to the table as a virtual owner for wells that do not have explicit group attachment.

The **Group-Well Connection** graph is the visual representation of the group-well hierarchy structure. From the graph you can access then modify well or group properties. You can print the graph on a single page or save it as an image file.

From the grid control you can select a single group, a single well, and/or a particular date to display in the hierarchy plot. If a group is selected, this group and all its children will be displayed in full. However, if a well is selected, its entire group hierarchy and its parent's group's siblings will be displayed. The two example displays are shown below:

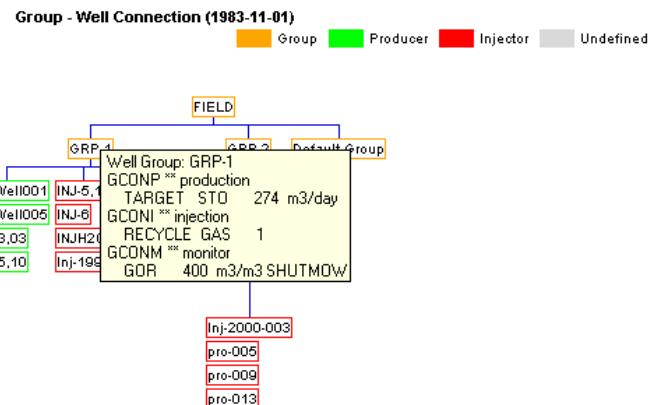




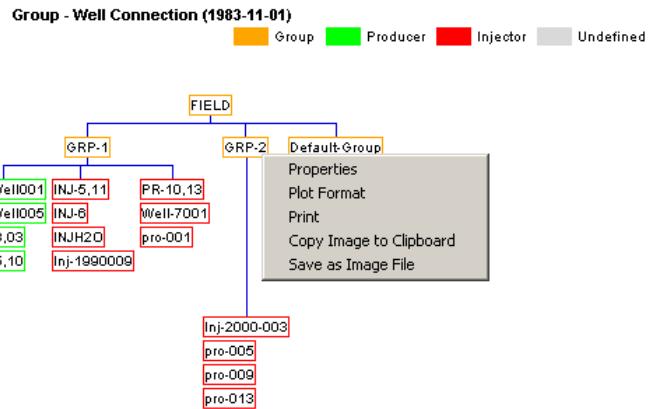
You can use the **select wells**  button to select and display multiple wells.

From the graph, you can launch the **Well Events** or **Group Events** dialog box by double-clicking a name box or right-clicking a name box then selecting the **Properties** menu item.

To get key information about a well or a group, click and hold the name box. A small pop-up window with the key information will be displayed, as shown in the following example:



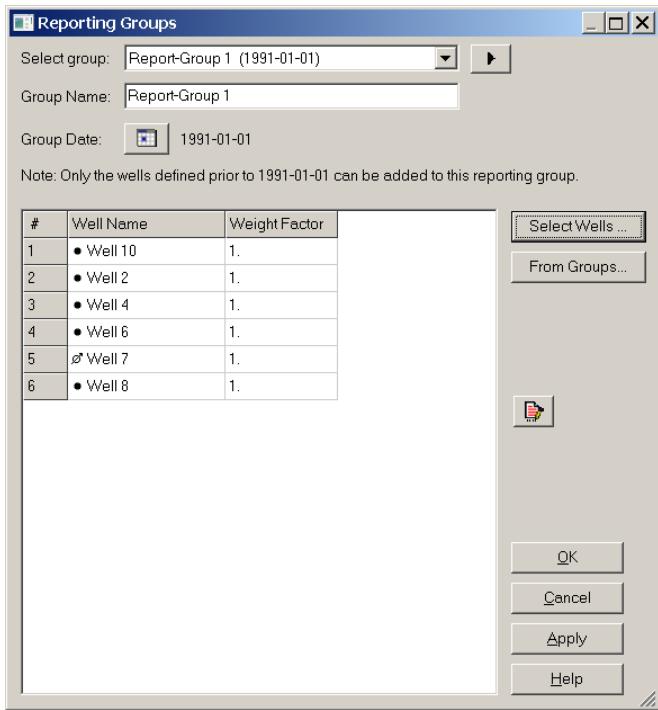
The context menu has up to five menu items (**Properties** is only available if you click on a name box). You can modify view properties, print the graph, and save the graph as image files.



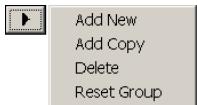
Reporting Groups

The Reporting Groups feature allows you to define a set of wells, with differing membership weights, then have the data for those wells reported to output files (as is done for the hierarchical groups). No group controls can be specified for reporting groups, but there are no restrictions upon well membership in reporting groups. A well may be a member of an arbitrary number of reporting groups, and a well can have any non-negative membership weight in a reporting group.

To open the **Reporting Groups** dialog box, select **Reporting Groups** from the **Well** menu:



The right arrow button contains a pop-up menu with commands to create and delete reporting groups.



To add a new reporting group:

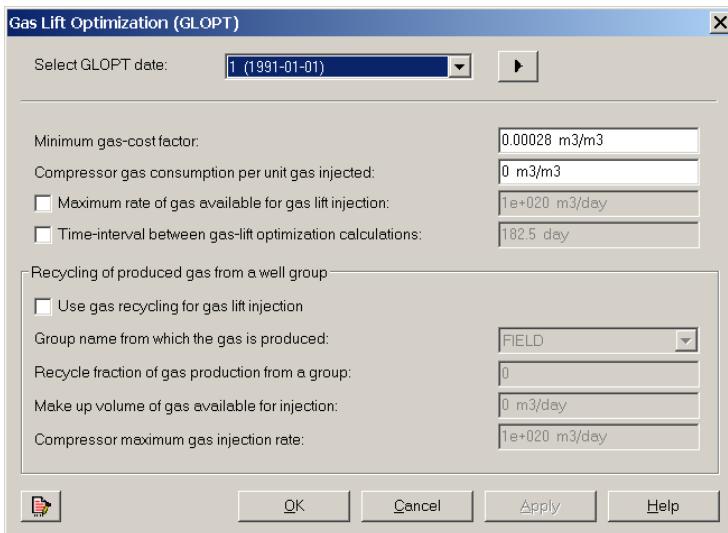
1. Select **Add New** from the pop-up menu, shown above.
2. Enter a unique group name.
3. Set a date for group definition.
4. Select well members using the **Select Wells** and **From Groups** buttons.
5. Enter weight factors for the selected wells
6. Click **Apply** or **OK**.

Note: Wells must be properly defined prior the reporting group date.

Gas Lift Optimization Options

Gas lift optimization options enable to control parameters of automatic allocation of lift gas to production wells. To open the Gas Lift Optimization dialog box:

- Select **Gas-Lift Optimization** from the **Well** menu.
- Right-click the corresponding tree item then select **Properties**.
- Double-click the corresponding tree view item



Clicking the right-arrow button to display a pop-up menu with commands for creating and deleting gas lift optimization on a selected date:



Adding New Gas Lift Optimization Options

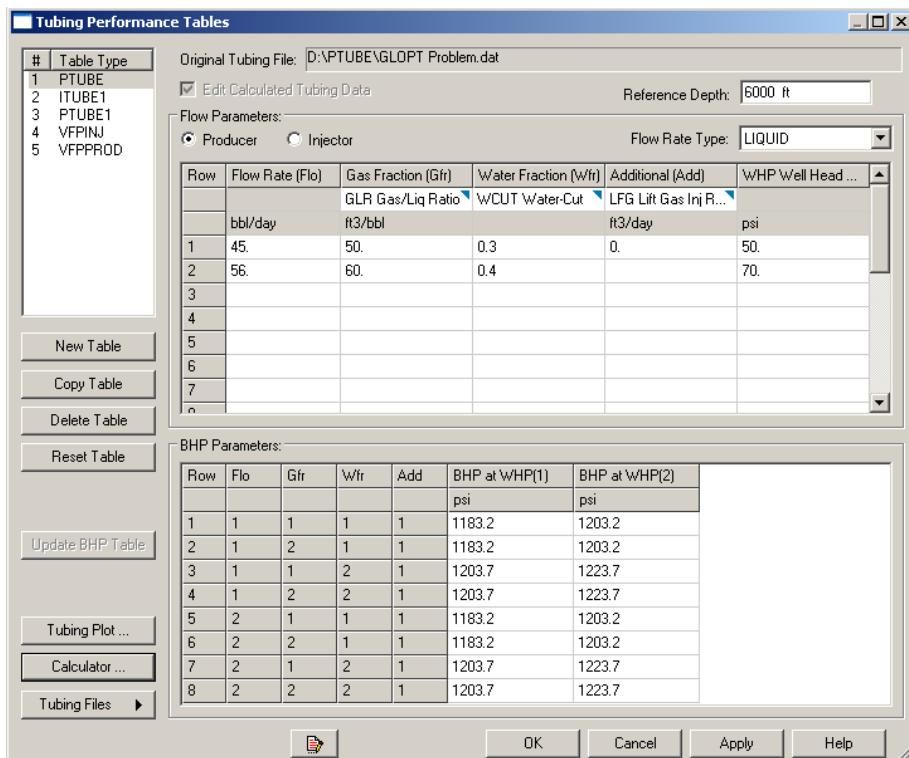
1. Select **New** from the pop-up menu.
2. Choose a date from the calendar.
3. Use the check boxes to select desired options.
4. Enter the control values.
5. Click **Apply** or **OK**.

Tubing Performance Tables

Tubing performance tables provide bottom-hole pressure (BHP) values for production and injection wells over a given range of gas fractions, water fractions, independent variables, flow rates and well-head pressures (WHP).

To open the **Tubing Performance Tables** dialog box:

- Select **Tubing Tables** from the **Well** menu.
- Select the **Wells & Recurrent** portion of the main tree view and then double-click on a corresponding Tubing Tables item.
- Select the **Wells & Recurrent** portion of the main tree view, right-click on a corresponding Tubing Tables item and then select **Properties** from the context menu.



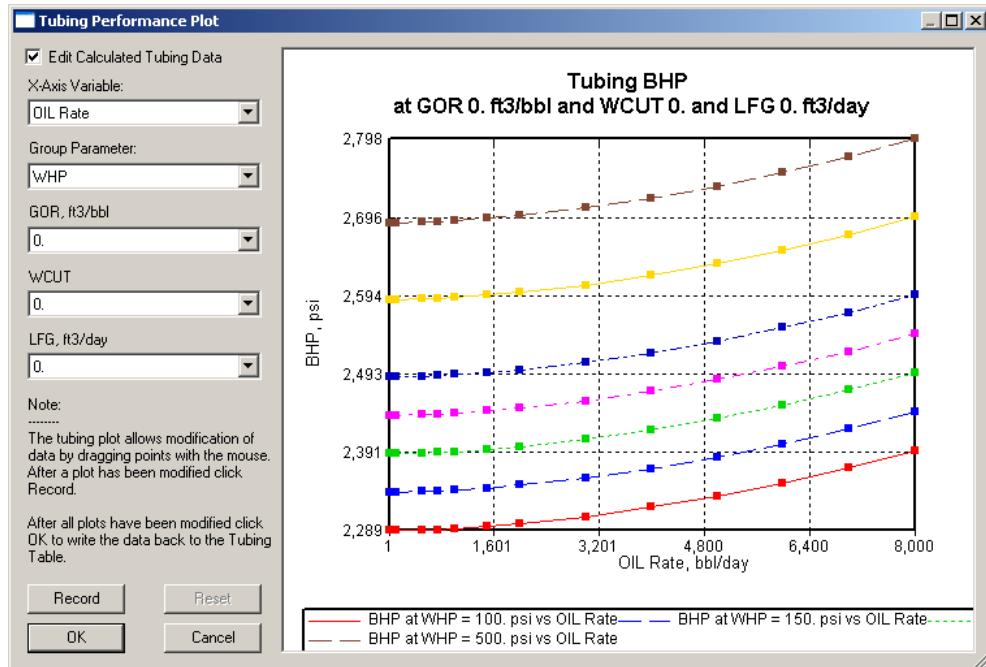
The list control in the top left corner of the window contains all available tubing tables. Select one in the list to display its information. The following control buttons are provided:

| | |
|------------------|--|
| New Table | Creates a new blank tubing table |
| Copy Table | Copies a selected tubing table |
| Delete Table | Deletes a selected tubing table |
| Reset Table | Cancels all changes made since the last OK or Apply command |
| Update BHP Table | Sets the number of rows and columns in the BHP Parameters table according to the number of Flow Parameters specified |
| Tubing Plot ... | Opens the Tubing Performance Plot window for the selected tubing table |
| Calculator ... | Opens the Tubing Pressure Calculator window to compute bottomhole pressure values for the selected tubing table |
| Tubing Files ▶ | Reads and saves data files for the selected tubing table. See the Using Tubing Files section. |

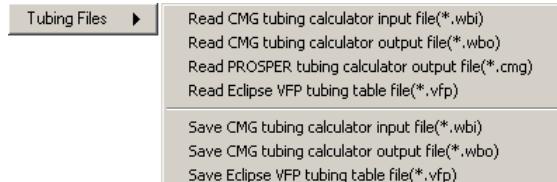
Adding a New Tubing Table

1. Open the **Tubing Performance Tables** dialog box.
2. Click the **New Table** button.
3. Select the Producer or Injector well type.
4. Select the Flow Rate Type.
5. Set the Reference Depth.
6. Use the **Flow Parameters** table to select the Gas Fraction (Gfr), Water Fraction (Wfr) and Additional (Add) independent lookup variables. Enter one or more values for Flow Rate (Flo), WHP Well Head Pressure, Gfr, Wfr and add variables into their corresponding columns. Notice that column values must stand in ascending order.
7. Press the **Update BHP Table** button to set the number of rows and columns in the **BHP Parameters** table according to the number of **Flow Parameters** set in previous step.
8. Enter values into the **BHP Parameters** table or use copy & paste to bring this data from another source. You can also use the **Calculator** button to compute **BHP Parameters** from the Flow Parameters, Reference Depth, PVT Methods, Pressure Computation Methods, Fluid Gravity, Tubing Conditions and Tubing Segment Data entered on the **Tubing Pressure Calculator** window.
9. Press the **Tubing Plot** button to view a plot of the BHP values for visual quality checking. Notice that the plot window enables you to adjust any BHP value by dragging its data point with the mouse. You need to press the **Apply** button to retain any change that you make on the plot.

The list-control in the top-left corner of the **Tubing Performance Tables** dialog box contains all available tubing tables. Select one in the list to display its information.



Using Tubing Files



The **Tubing Files** button contains commands to read and save tubing table data in the following formats:

Wellbore calculator output file format (*.wbo)

File contains formatted tubing data specified by the **PTUBE1** and **ITUBE1** keywords. This format is used by the **Tubing Pressure Calculator** to create **BHP Parameters** data, and is also used by the **Builder** dataset file to read and save tubing data.

Eclipse VFP file format (*.vfp)

File contains formatted tubing data specified by the Eclipse **VFPPROD** and **VFPINJ** keywords. This format can be read from the **Builder** dataset file.

| | |
|--|---|
| Wellbore calculator input file format (*.wbi) | File contains formatted Flow Parameters, Reference Depth, PVT Methods, Pressure Computation Methods, Fluid Gravity, Tubing Conditions and Tubing Segment Data. This format is used by the Tubing Pressure Calculator to create BHP Parameters data, and is also used by the Builder dataset file to read and save calculator data. |
| PROSPER calculator output file format (*.cmg) | File contains PROSPER tubing data which is formatted to be read by Builder. See Wellbore calculator output file format (*.wbo) . |

The Original Tubing File: control on the **Tubing Performance Table** window shows the name of the file where the selected tubing table data originated from. If tubing data is read or saved to a *.wbi, *.wbo or *.vfp formatted file, the Original Tubing File: location will be updated to reflect the file name used. The selected tubing table name will also be updated to reflect the current type of keyword contained in the file.

When the **Builder** dataset is saved, **PTUBE** and **VFPProd** tubing data gets converted to the **PTUBE1** format and **VFPINJ** tubing data gets converted to the **ITUBE1** format. The **PTUBE**, **VFPProd** and **VFPINJ** tubing table names will not exist after the dataset has been saved and reloaded.

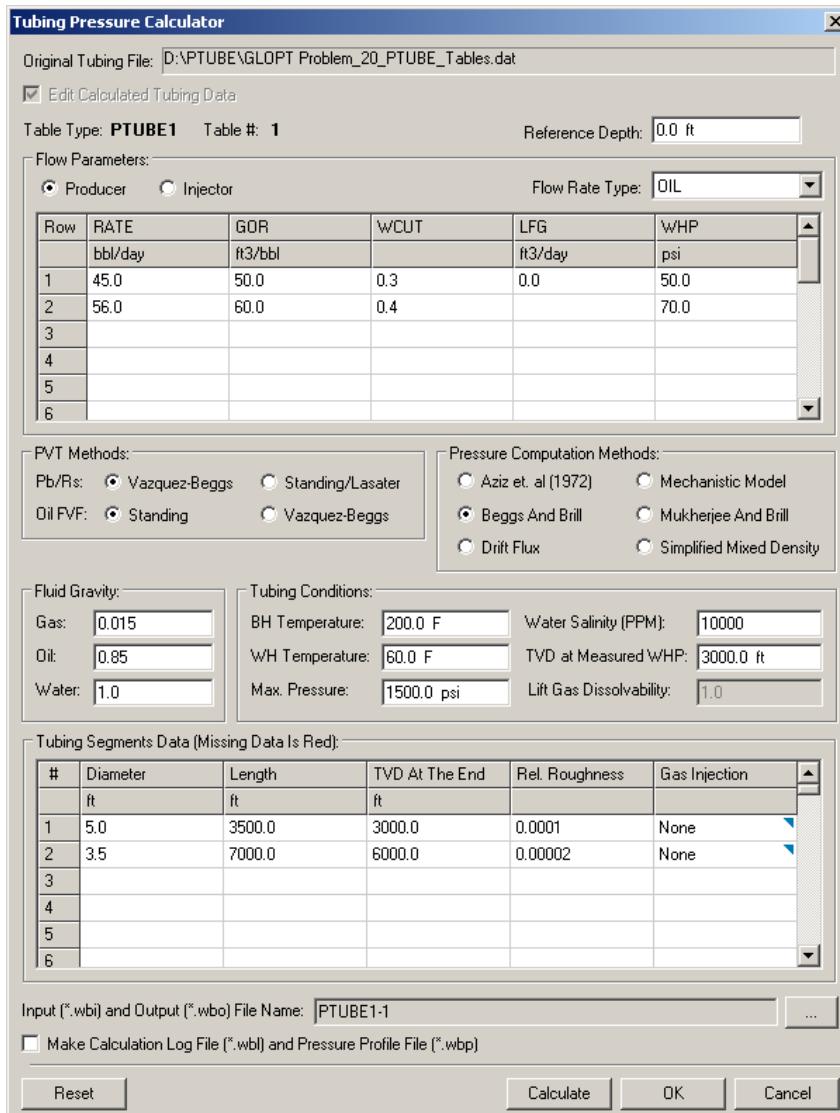
Using the Tubing Pressure Calculator

The **Tubing Pressure Calculator** provides built-in correlations to compute bottomhole pressures for the selected tubing table. The tubing calculator is available by pressing the **Calculator** button on the **Tubing Performance Tables** dialog box. The calculator can only use certain Flow Rate Type, Gfr, Wfr and Add variables to calculate BHP values.

1. Press the **Calculator** button on the **Tubing Performance Tables** dialog box.
2. Select a PVT Method for Pb/Rs and Oil FVF.
3. Select a Pressure Computation Method.
4. Set Fluid Gravity and Tubing Conditions.
5. Enter values for one or more tubing sections including Diameter, Length, TVD At The End, Relative Roughness and Gas Injection.
6. Set the name of the intermediate calculator input (*.wbi) and output files (*.wbo). These files will be saved to the same directory as the Builder dataset file.
7. Select the Make Calculation Log File (*.wbl) and Pressure Profile File (*.wpb). These files will be saved to the same directory as the Builder dataset file.
8. Press the **Calculate** button and wait for the results text window. It may take from a few seconds to several minutes for the calculator run depending on the dimensions of the table and the computer capability.
9. Press **OK** to accept the calculated results or **Cancel** to reject.

Note: Tubing calculator input and output data will be saved to the dataset file.

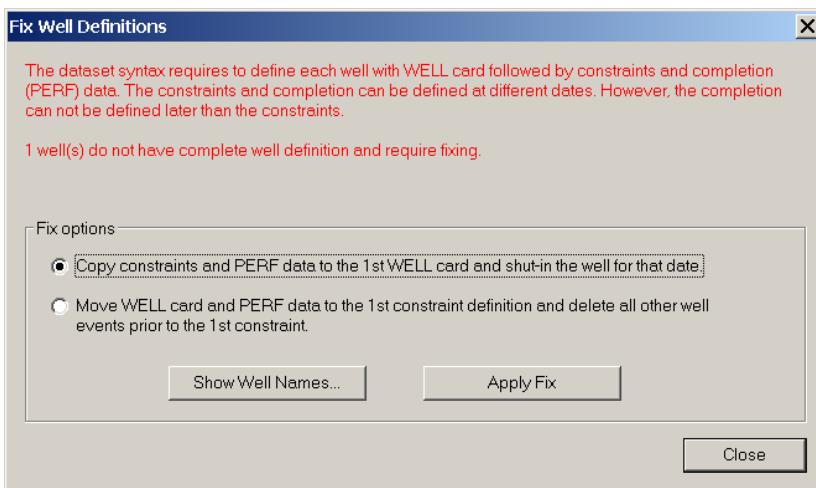
Refer to [Appendix B: Technical Notes - Tubing Head Pressure Calculator](#) in this manual for more information on the use of this feature.



After the Tubing Pressure Calculator has computed BHP values and the data has been OK'd the Tubing Performance Table window will update and disable access to all calculated tubing data. This is done so the Flow Parameters and BHP Parameters tables remain in synch. If you wish to modify any of the tubing data they must select the **Edit Calculated Tubing Data** check box.

Fixing Well Definition Dates

The current dataset syntax requires that initial well definition must include keywords WELL, PRODUCER or INJECTOR, and PERF set on the same date. It is not uncommon that in the process of building a dataset these events get set on different dates. If you get a validation error of this type you can use command **Fix Well Definition Dates** from the **Well** menu. It will open the dialog box shown below. Select the type of fix you want to use and then click **Apply Fix**.



Validating Well Completions

If some well completions have perforation nodes in the NULL or inactive blocks (having zero porosity, thickness, permeability, and so on), this may create problems for the simulation run. Through the **Validate Null Completions** dialog box, you can validate completions and removed these types of NULL perforations:

1. Use menu item **Wells | Validate Well Completions for NULL Blocks**
2. Select the type of action. Option *Ignore* will just show validation results without making any changes.
3. Click **OK**.



Production Data Wizard

Overview

The Production Data Wizard is designed to help you to import well production, injection and pressure data into Builder, which then builds the well operating constraint information (using *ALTER keywords in the well data section of the simulation input file) for the CMG simulators. This data can optionally be saved in field history files for display in Results Graph, which allows you to compare simulation results with actual field production history.

Files Required by Production Data Wizard

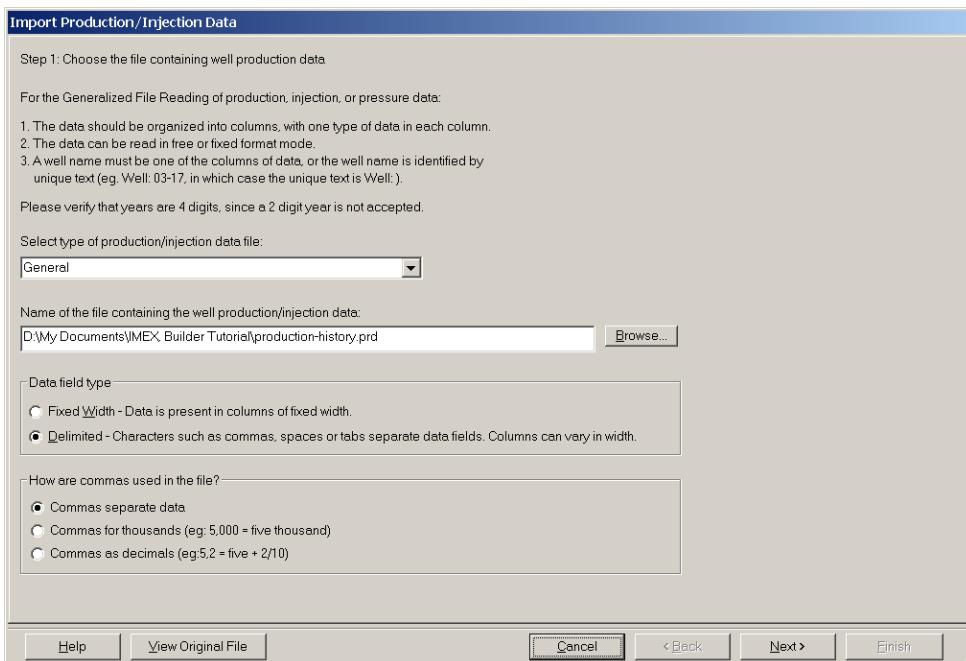
Before using the Production Data Wizard, you need to have the input file ready. Currently, three file formats are supported:

- General table file (General) format
- CMG Field History File (FHF) format
- Peep 2002 Edition 1 Data Transfer File (Merak) format

For the general table file format, production, injection, and pressure data should be organized into columns, with one type of data in each column. The data can be in either free format mode or fixed format mode. For the free format mode, the data columns are separated by delimiter characters, such as comma, space or tab. For fixed format mode, the data columns begin and end in fixed columns. The well names can be one of the data columns, or can be in rows, optionally preceded by a unique string, such as “*NAME:”, “WELL:” or “POZO:”. If the data are for a well group, then the well group name should be after “*NAME:”.

Using the Production Data Wizard

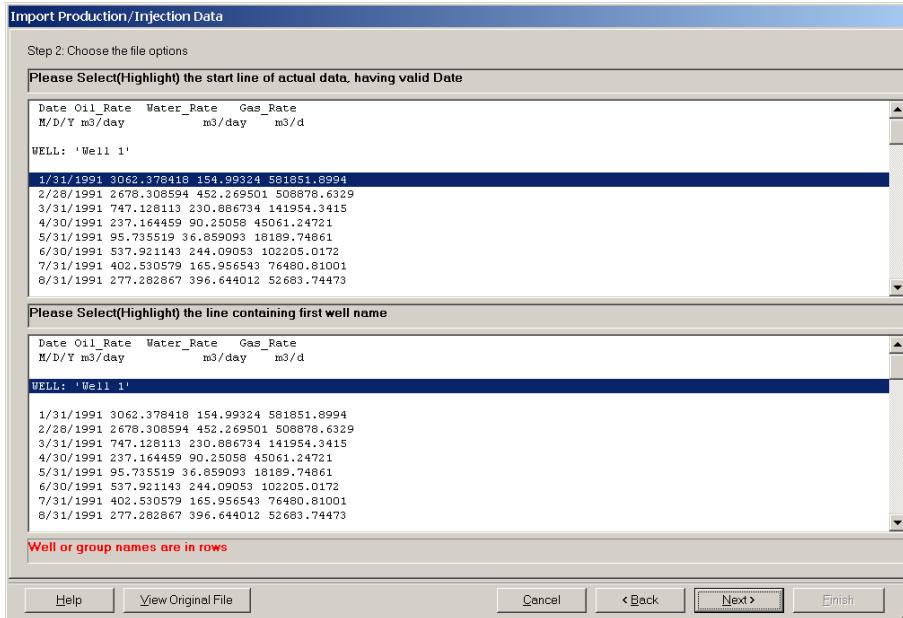
To access the Production Data Wizard, select **Import Production/Injection Data** from the **Well** menu. The **Import Production/Injection Data (Step 1)** dialog box will be displayed, as shown in the following example:



Choose an input file and format. The data field type can be fixed width or delimited. Select how commas are used in the file. The default is to use a comma to separate data fields. You can view the contents of the original file at any time by clicking **View Original File**.

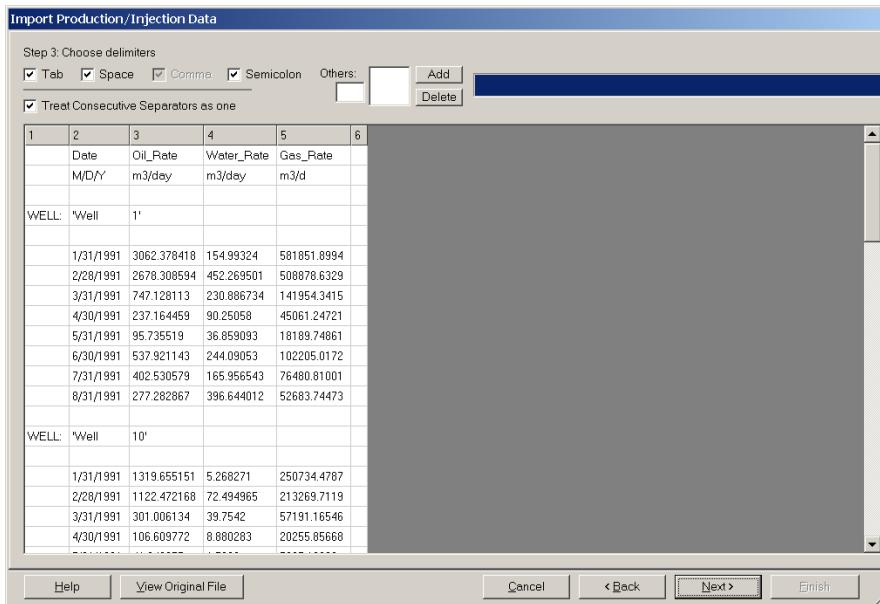
Click **Next**. The dialog box that is displayed will depend on the file format you chose in this step. For FHF and Merak formats, you will go directly to step 5, where you will choose the primary constraints for wells and groups.

For general table file format, the **Import Production/Injection Data (Step 2)** dialog box will be displayed and you will continue through steps 2, 3 and 4 to instruct the program how to read the file.



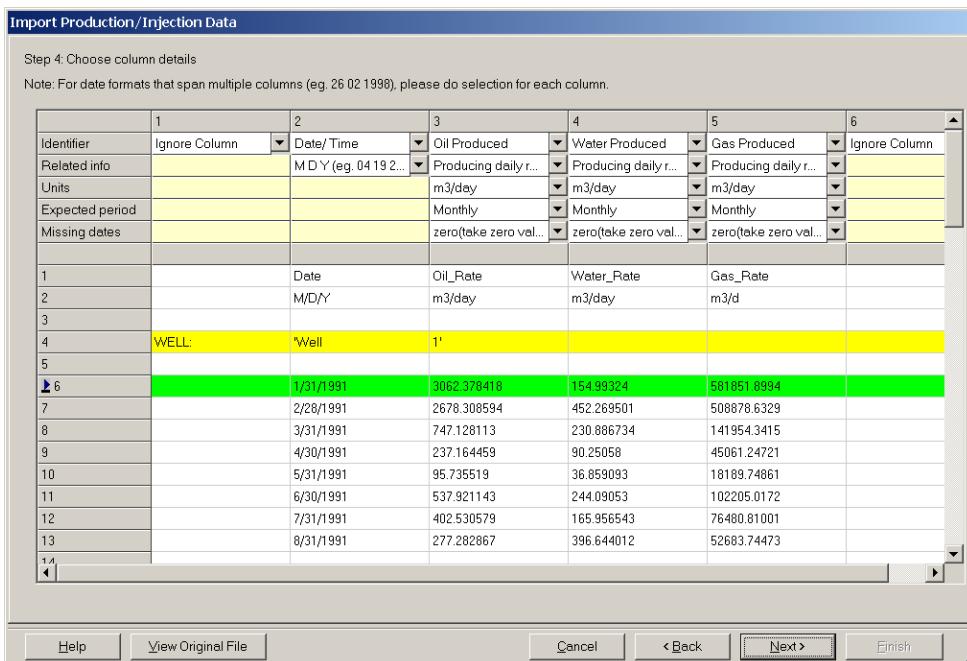
As shown above, specify the locations of the well or group names and the start of the data row, which must contain a valid date string. You select the start of the data row in the top edit box, and the well or group name row in the bottom edit box. These two rows can be the same if well or group names appear as one of the data columns. The red text at the bottom of the dialog box indicates the results of your selection.

Click **Next**. The **Import Production/Injection Data (Step 3)** dialog box is displayed:



In step 3, you tell the program how to separate the data columns. If you chose **Delimited** in step 1, you will specify the delimiters here. They can be one or more of tab, space, comma, semicolon or custom character. If you chose **Fixed Width** in step 1, you specify the start and end of each data column by clicking inside the frame containing the file. A vertical line will be drawn to indicate the column break. If you make a mistake and want to delete it, double-click on the line.

Click **Next**. The **Import Production/Injection Data (Step 4)** dialog box is displayed:



In step 4, you define the meaning of each data column. The top five rows instruct the program how to interpret the data in each column.

Identify the date column or columns, and select **Date/Time** from the combo box at the top of that column. Once this is done, the combo box in the **Related info** row will display the program's best guess of the date format. Change it if necessary.

Continue to specify the other columns by selecting items from the **Identifier** combo box.

Note: The **Producing daily rate** and **Calendar daily rate** in the **Related info** combo box are related as follows:

$$\text{Producing daily rate} = \frac{\text{Calendar daily rate}}{\text{On-time fraction}}$$

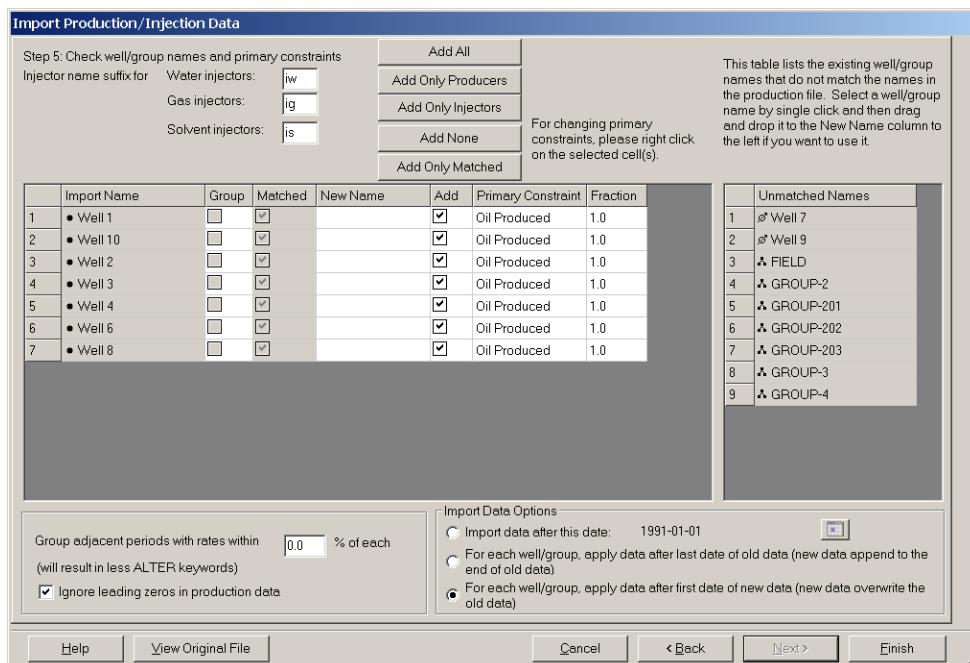
where:

$$\text{On-time fraction} = \frac{\text{Period of time when the production/injection is on}}{\text{Total period of time}}$$

This value is calculated by Builder based on the **Days or hours on** property and should have a value between 0 and 1.

The **Unit** combo lets you specify the units for the selected property. The other two combo boxes will be enabled or disabled accordingly. Always check if the items in the combo boxes selected by the program are correct. If they are not, change them. At least one production, injection or pressure information along with date information must be specified before you proceed to the next step.

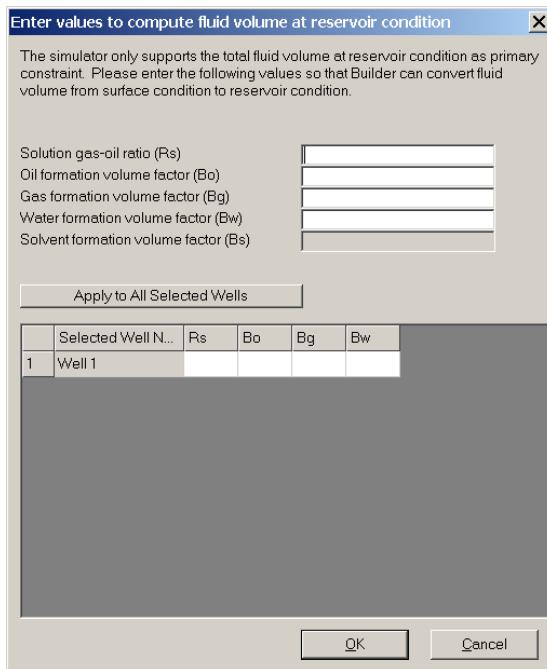
Click **Next**. The **Import Production/Injection Data (Step 4)** dialog box is displayed:



In step 5, specify the primary constraints for each well or group imported by the file.

- **Group:** If checked, indicates the name represents a well group. If you check this cell and the group name does not exist in the original dataset, you will be asked to create the well group with that name through the **Create New Group** dialog box.
- **Matched:** Indicates if the well or group name matched existing well or group names in the dataset.
- **Add:** Indicates if data for this well or group will be added to Builder.
- **New Name:** Provided for you to change the well or group name. If you want to change the well or group name to an existing well or group name displayed in the table at right, just click the name, then drag and drop that name to the **New Name** cell.
- **Primary Constraint:** Right-click on a cell under the **Primary Constraint** column to bring up a list of available primary constraints for that well or group. You can do multiple selections by using SHIFT or CTRL with a left-mouse click, and then right-click to bring up the menu.

If you choose **Total Fluid Produced** as the primary constraint, you will be prompted to provide values for solution gas to oil ratio and volume formation factors for oil, water and gas:



This option specifies the total fluid volume (oil, water, gas and/or solvent) at the reservoir condition as the operating constraint. Production volume data from the production file are for surface condition. That is why Builder needs additional information to convert volume at surface condition to volume at reservoir condition. Enter the required information and then click **OK** to return to the **Import Production/Injection Data (Step 4)** dialog box.

Before you click **Finish**, you can choose one of the **Apply Date Options** to overwrite the existing production data with the newly imported data or append only the new data to the existing wells. If your production data is in more than one file, you can invoke this wizard as many times as you want to import all the production data you have. Once all the data are in Builder, you can examine the *ALTER keywords by clicking the **Well & Recurrent** button in the tree view, right-clicking the blank area in the tree view, and then selecting **Display dataset for section**. If you think there are too many *ALTER keywords, which may impact the simulation run time, you can invoke **Average production/injection data** from the **Well** menu to reduce the number of this keyword.

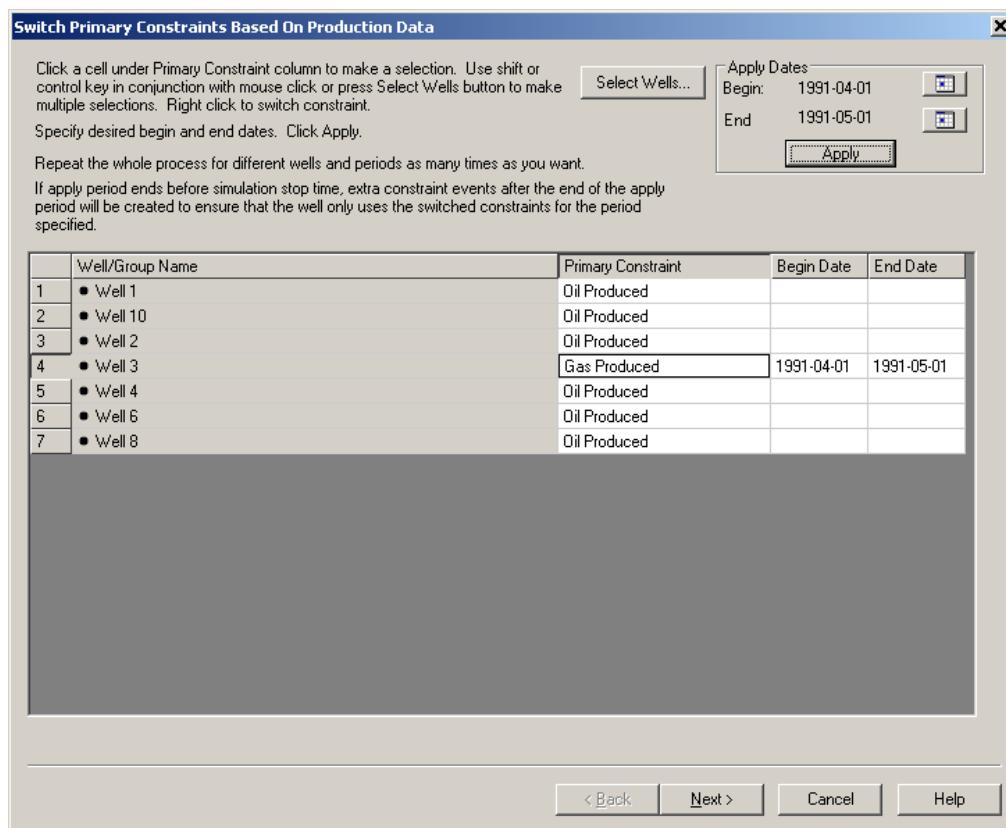
All data imported by the wizard can be saved to a CMG Field History File by selecting **Create Field History File** in the **Well** menu. Production, injection and pressure data are saved to different files. These files can later be used in Results Graph to compare with simulation results, or in Results Report program to generate custom reports.

The meanings of date/time in the FHF created by Builder are different for different properties. For all production properties, such as oil, water and gas rates, the date/time means the end of a period and the associated value is applied to the period previous to this date/time. Therefore, valid production history data should start with a date/time with a property value of zero to establish the first period. For non-production properties such as pressure and temperature, the value corresponding to the date/time is considered to be an instantaneous value at this point in time. A -99999 value for any property means the data is missing for the property at that time.

Switch Well Constraints Based on Production Data

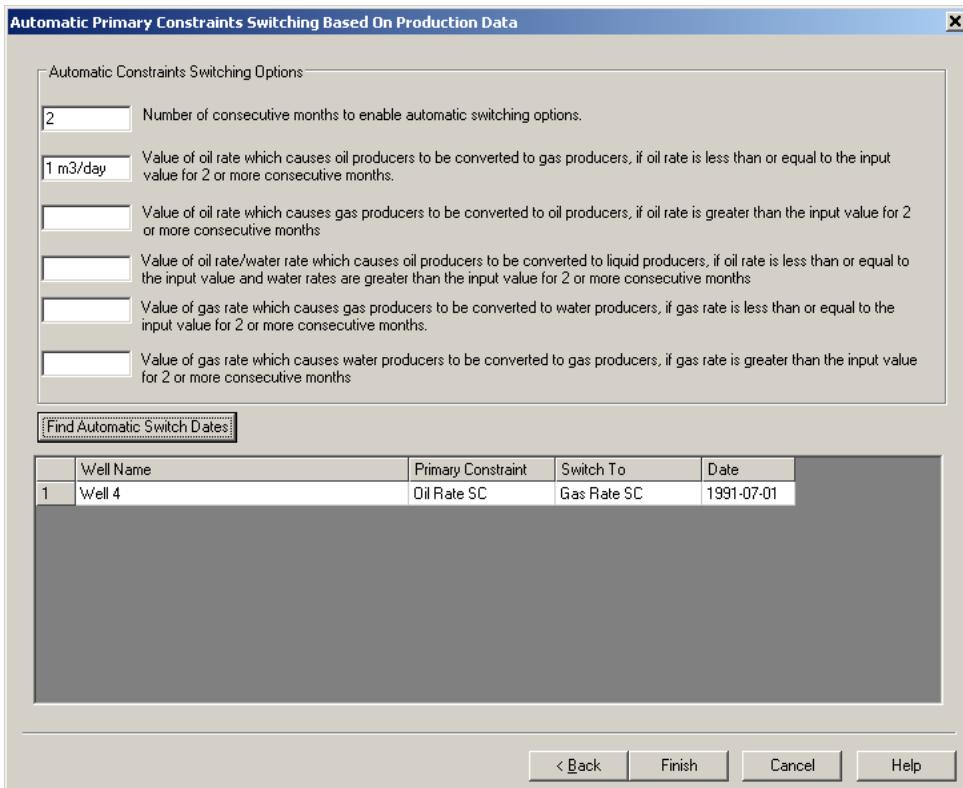
After you finish importing well production data through the Production Data Wizard, you can still change the primary constraints through **Well | Switch Constraints** from the main menu. You can also instruct Builder to automatically switch the primary constraints based on the actual production rates in the production data file. For example, if a well starts with oil rate as the primary constraint, but the oil rate drops to almost zero after some time, Builder can automatically switch to gas rate constraint when this happens.

The **Switch Primary Constraints** dialog box is shown below:



When the dialog box opens, the table displays the current constraint. You can change the period for which the constraint switch will apply.

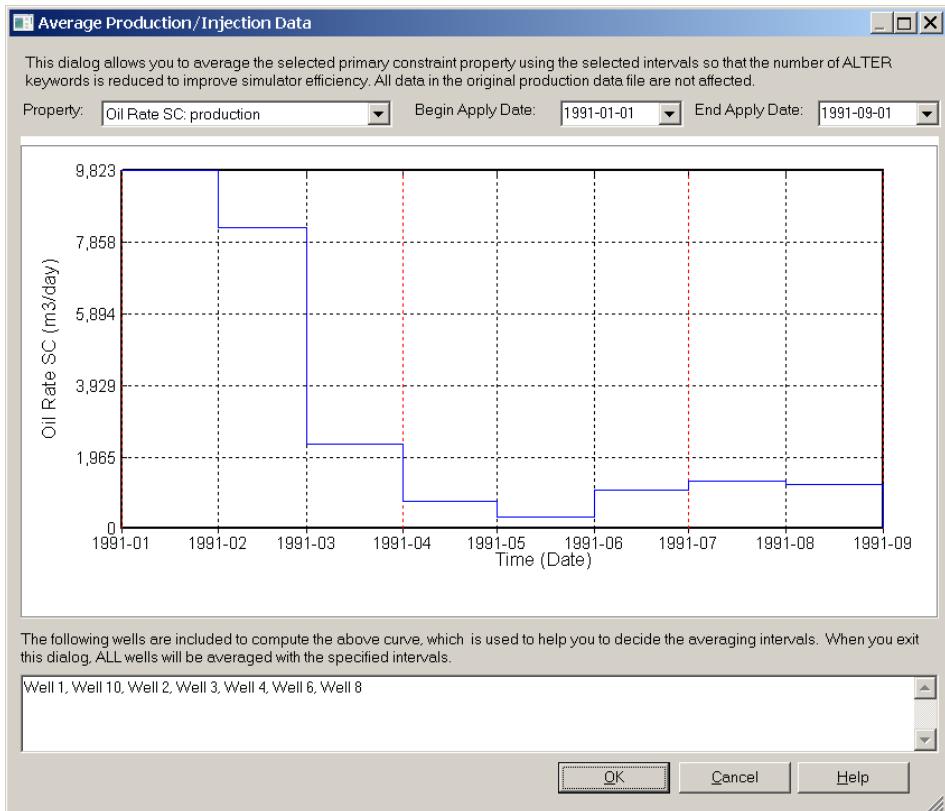
If you do not need to switch any constraint, click **Next**. The **Automatic Primary Constraints Switching Based on Production Data** dialog box is displayed:



After entering the desired tolerance values, click the **Find Automatic Switch Dates** button to display the dates when the constraint switching occurs. If you do not want to use this option, do not enter any value and click **Finish** to exit the dialog box.

Average Production/Injection Data

This option allows you to reduce the number of ALTER keywords generated by the Production Data Wizard. Too many unnecessary ALTER keywords will reduce the running efficiency of the simulator. The **Average Production/Injection Data** dialog box, shown below, is opened through **Well | Average Production/Injection Data**.



The dialog box displays a curve representing the sum or average of the property used as the primary constraint. This is a visual aid to help you decide how to average the data. You can average the data on a weekly, monthly or quarterly basis. Right-click on the vertical line in the graph to select the averaging period. Repeat as many times as necessary.

The plot shows the averaging periods as red vertical lines. The default is 3 months. To change the averaging interval, right-click on the plot at the desired date, and from the pop-up menu, select the new interval. After this is done, the averaging interval will change for all dates after the selected date.

Trigger

This dialog box allows you to define well management triggers. The general format for trigger is:

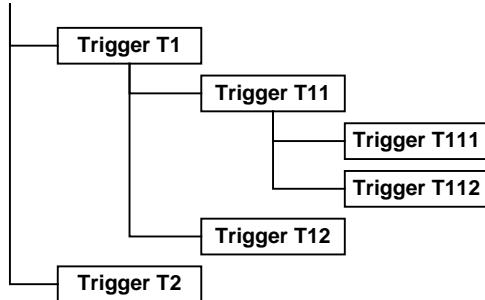
trigger-name (trigger-type trigger-condition).

For example for the trigger named GroupTrigger (ON_GROUP STO-RP), GroupTrigger is the trigger name, ON_GROUP the trigger type, and STO-RP the trigger condition.

To create a new top-level trigger, you can

- Double-click the **Triggers** node in the **Wells & Recurrent** menu in the tree view
- Right-click the **Triggers** node then select **New**
- Right-click an existing top-level trigger item then select **New**

Triggers can be nested to define an order of testing, as shown below:



In the above example, the order of testing is T111, T112, T11, T12, T1, T2.

To create a new nesting trigger:

- Right-click an existing top-level or nesting trigger item then select **New Child** to add a new nesting trigger to the selected trigger. In the above example, T111 is a child of T11.
- Right-click an existing nesting trigger then select **Sibling New** to add a nesting trigger as the sibling trigger to the selected nesting trigger. In the above example, T112 is a sibling of T111.
- Double-click an existing trigger (top-level or nesting) and use the action area to type in the trigger you want.

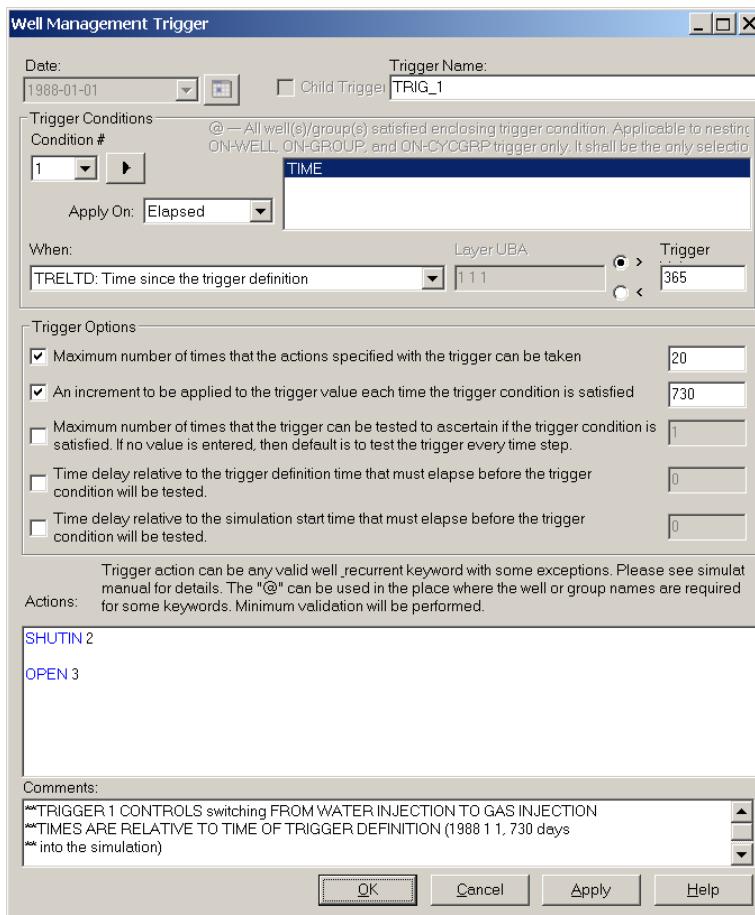
To remove trigger(s):

- Right-click a trigger item then select **Delete** to delete a single trigger.
- Right-click **Triggers** then select **Delete** to delete all triggers. Both remove trigger operations require user confirmation.

To modify an existing trigger:

- Double-click a trigger item to open the **Well Management Trigger** dialog box.
- Right-click a trigger item then select **Properties**.

An example of the **Well Management Trigger** dialog box is shown below:



- **Date** combo box: Select the date on which the trigger is defined.
- **Child Trigger** check box: Valid for nested trigger only; i.e. the trigger defined as an action item for another trigger.
- **Trigger Name**: Name used to identify the trigger.
- **Trigger Conditions** area: In this area, define the trigger condition or trigger conditions (in the case of GEM), that will cause the actions defined in the **Trigger Options** area.
- **Condition #** drop-down list: Supports multiple conditions in a trigger. It is available for GEM only and is grayed out for STARS and IMEX. Refer to [Multiple Trigger Conditions](#) below.
- **Apply On** drop-down list: Use to specify whether trigger applies to wells, groups, layers, sector, field, elapsed time, or cycling groups.

- **Subject name** combo box (the one next to the **Apply On** box): Available subject names (wells, groups, sectors, “field” for ON-FIELD trigger, “time” for ON_ELAPSED trigger, and cycling groups for ON-CYCGRP trigger), to select to apply this trigger on. For ON-WELL, ON-GROUP, and ON-CYCGRP types of triggers, multiple selections are allowed. Only single selection is allowed for the other four types of triggers. For a nesting ON-WELL or ON-GROUP or ON-CYCGRP trigger @ can be used to represent all wells or groups or cycling groups specified by its enclosing trigger. If @ is selected it shall be the only selection, i.e. any other selections additional to @ will be ignored.
- **When** drop-down list: Available trigger conditions to select.
- **Layer UBA:** Type in the layer (i, j, k) for ON_LAYER trigger.
- **Trigger Options** area: A total of five options are available for no ON-CYCGRP trigger. The last two options are mutually exclusive, i.e. you can specify only one of them at the same time.
- **Actions** area is where you can enter the trigger actions. They are basically any valid well & recurrent statements including triggers, and more. Please refer to the simulator manual for details. Only minimum syntax checking will be performed on the typed in trigger actions.
- **Comments** area: Use this area to record comments. Please note all the comments lines will be written at the beginning of its enclosing root-level trigger.

Multiple Trigger Conditions

This feature, which is available for GEM only, allows you to trigger an action based on a logical statement of multiple trigger conditions; for example, you can trigger an event to take place after an elapsed period of time OR [after the pressure of one well exceeds one level AND the pressure of a second well exceeds a different level].

Two logical operators are supported, AND and OR. Each of these operators can process two inputs. With the AND operator, both input conditions must be evaluated as TRUE for the logical statement to be considered TRUE. In the case of the OR operator, if either input condition is evaluated as TRUE then the logical statement is considered to be TRUE.

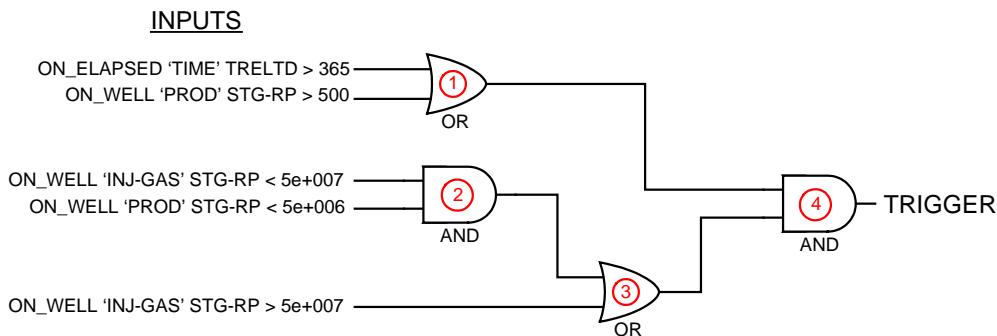
Every AND and OR operator in the trigger condition statement is assigned a precedence. The precedence assigns a priority ranking to the operator. For example, an AND operator with a precedence of 1 will be evaluated first; similarly, an AND operator with a precedence of 3 will be evaluated third.

The logic of the trigger conditions is entered in tabular form, as shown below:

| Logic ... | Prece... | Trigger Condition |
|-----------|----------|-----------------------------------|
| | | ON_ELAPSED 'TIME' TRELTD > 365. |
| OR | 1 | ON_WELL 'PROD' STG-RP > 500. |
| AND | 4 | ON_WELL 'INJ-H2O' STO-RP < 5e+007 |
| AND | 2 | ON_WELL 'PROD' STG-RP < 5e+006 |
| OR | 3 | ON_WELL 'INJ-H2O' STO-RP > 5e+006 |

GEM will evaluate operations in order of the specified precedence from the lowest number to the highest.

The above table of logic statements represents the following logic diagram:

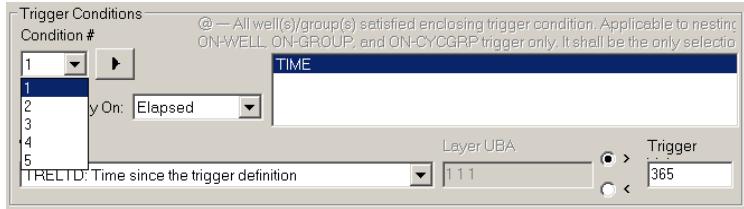


In the logic diagram:

- AND and OR gates represent the logical operations specified in the table
- Trigger conditions (inputs) are shown on the left, the trigger output is shown on the right
- Circled numbers indicate the order in which the gates are tested

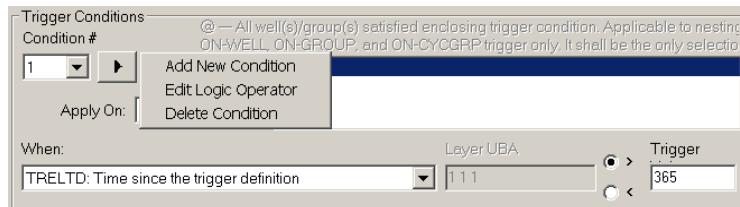
To view or edit an existing trigger condition:

Select the trigger condition number in the **Condition #** drop-down list. All values for the selected trigger condition will be displayed for viewing or editing in the **Trigger Conditions** area:



To add a new trigger condition:

1. Click the button then select **Add New Condition** in the pop-up menu:

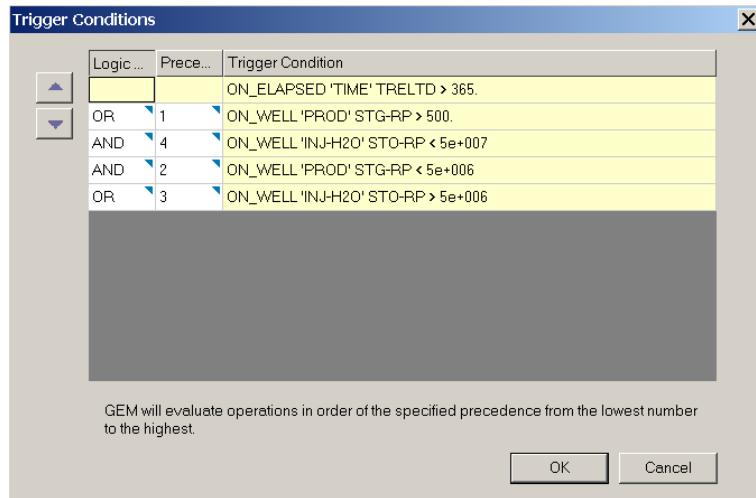


2. Enter the parameters for the new trigger condition.
3. The trigger condition will be saved when you have selected a different trigger condition (through the **Condition #** drop-down list), selected **Edit Logic Operator**, clicked the **Apply** button, or clicked the **OK** button.

To edit a logic operator:

After adding or editing conditions, you can edit the logic operators and add precedence to conditions, as necessary. To do this:

1. Click the button then select **Edit Logic Operator** in the pop-up menu. The **Trigger Conditions** dialog box will be displayed:



The first column of the **Trigger Conditions** table is **Logic Operator**. There are two choices - AND or OR. The second column is **Precedence**. If there are only two trigger conditions, no precedence is needed. If there are n conditions, these numbers must be from 1 to $n-1$. Precedence numbers can only be used once and must not be repeated.

To change a logic operator:

1. Click on the cell (first column and expected row) then select AND or OR as necessary:

| Logic ... | Prece... | Trigger Condition |
|-----------|----------|-----------------------------------|
| OR | 1 | ON_ELAPSED 'TIME' TRELTD > 365. |
| AND | 4 | ON_WELL 'PROD' STG-RP > 500. |
| OR | 2 | ON_WELL 'INJ-H2O' STO-RP < 5e+007 |
| AND | 3 | ON_WELL 'PROD' STG-RP < 5e+006 |
| OR | 3 | ON_WELL 'INJ-H2O' STO-RP > 5e+006 |

To change the precedence:

1. Click on the cell in the **Precedence** column. The **Precedence** drop-down list will be displayed, as shown below:

| Logic ... | Prece... | Trigger Condition |
|-----------|----------|-----------------------------------|
| OR | 1 | ON_ELAPSED 'TIME' TRELTD > 365. |
| OR | NONE | ON_WELL 'INJ-H2O' STO-RP > 5e+007 |
| AND | 1 | ON_WELL 'PROD' STG-RP < 5e+006 |
| OR | 3 | ON_WELL 'INJ-H2O' STO-RP > 5e+007 |
| | 4 | |

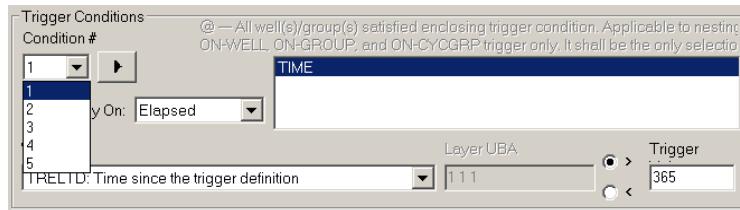
2. Select the appropriate precedence for the trigger condition.

To move the position of a trigger condition:

1. Select a trigger condition by clicking on the **Trigger Condition** column. The **Up** or **Down** buttons will be enabled. If you have selected the first trigger condition, you will not be allowed to move it up. Similarly, if you select the last, you will not be allowed to move it down.
2. Click the **Up** button to exchange the selected row with the one above it.
3. Click the **Down** button, to exchange the selected row with the one below it.
4. Click **OK** to return to the **Well Management Trigger** dialog box.

To delete a trigger condition:

1. Select the condition that you want to delete by selecting its **Condition #**:

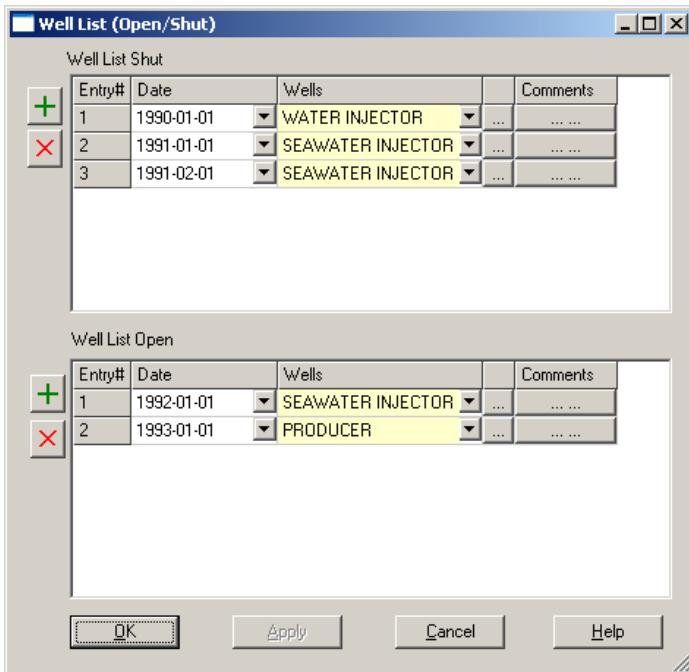


2. Click then select **Delete Condition**.
3. Click **Yes** to confirm.

Well List Shut/Open

The **Well List (Open/Shut)** dialog box allows you to define the well management WLISTSHUT and WLISTOPEN keywords. It can be launched by:

- Through the main menu, selecting **Well | Well List (Open/Shut)**.
- Through the **Wells & Recurrent** tree view item, double-click **Well List (Open/Shut)**.
- Right-click **Well List (Open/Shut)** in the **Wells & Recurrent** tree view then select **Properties**.



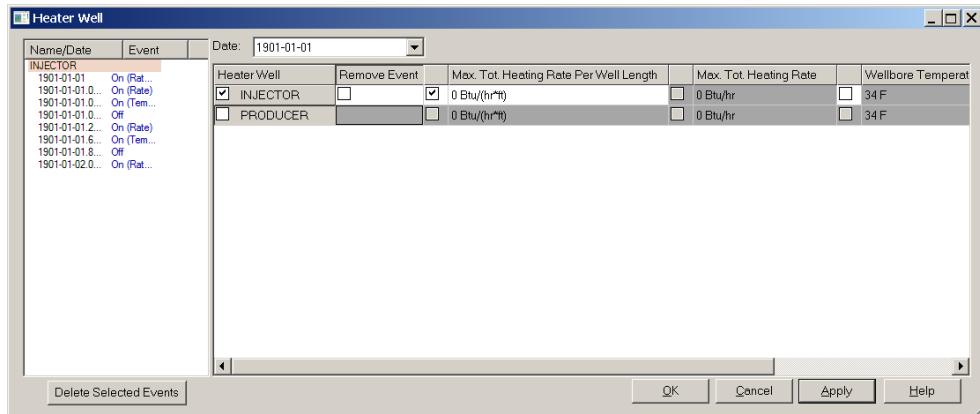
The top grid is for WLISTSHUT and the bottom for WLISTOPEN. They both have five columns. In the fourth column from the left, click  in a cell to launch the **Well Selection** dialog box and select an available well. The other columns are as follows:

- **Entry#:** Entry ID. This is a read-only cell.
- **Date:** Definition date.
- **Wells:** Well names that have been selected for this WLISTSHUT or WLISTOPEN entry. Read-only. To change the selected well, click  beside the column to open the **Well Selection** dialog box.
- **Comments:** Click  to open the **View/Edit Comments** dialog box.

Use the  and  buttons on the left side of the grids to add or delete the respective entries.

Heater Well (STARS Only)

The **Heater Well** dialog box allows you to define and modify STARS heater wells. It can be launched by double-clicking the **Well & Recurrent** tree view item **Heater Well** or right-clicking the same item and selecting the **Properties** menu.



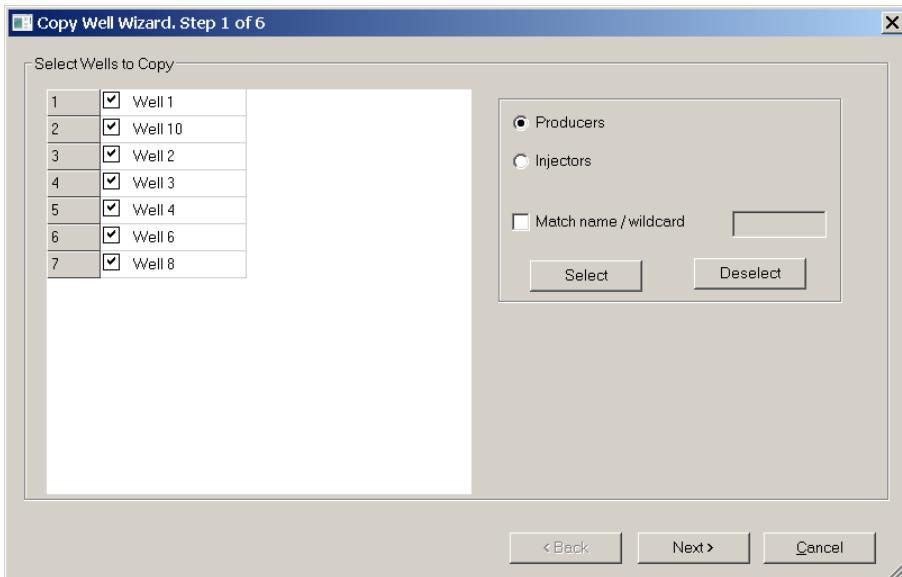
Either the wellbore temperature and/or one of the total heating rate values can be entered for a heater well. It is also possible to turn a heater well off. To do this, a heater well has to be first created and shown as "On". To turn it off, select a later date and then clear the check box next to the name of the heater well that you want set to "Off".

To delete or remove heater wells that have been created, the **Remove Event** box can be checked for a particular well. Alternatively, select the items you want to delete in the **Name/Date** list and then click the **Delete Selected Events** button.

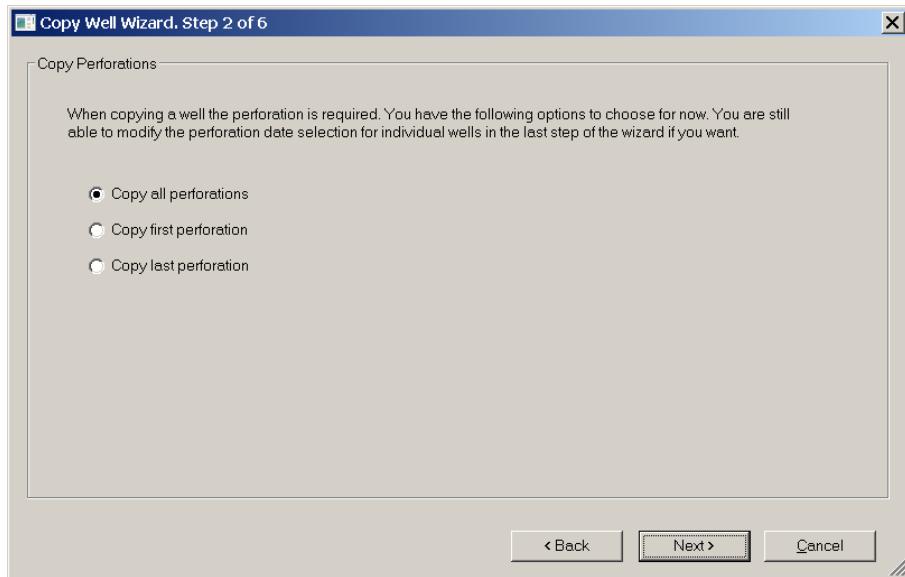
Copy Well Wizard

This wizard allows you to copy multiple wells of the same type (producer or injector) to the same number of the wells of the different type (that is, to injector or producer) with the specified well geometry, perforations, and trajectories.

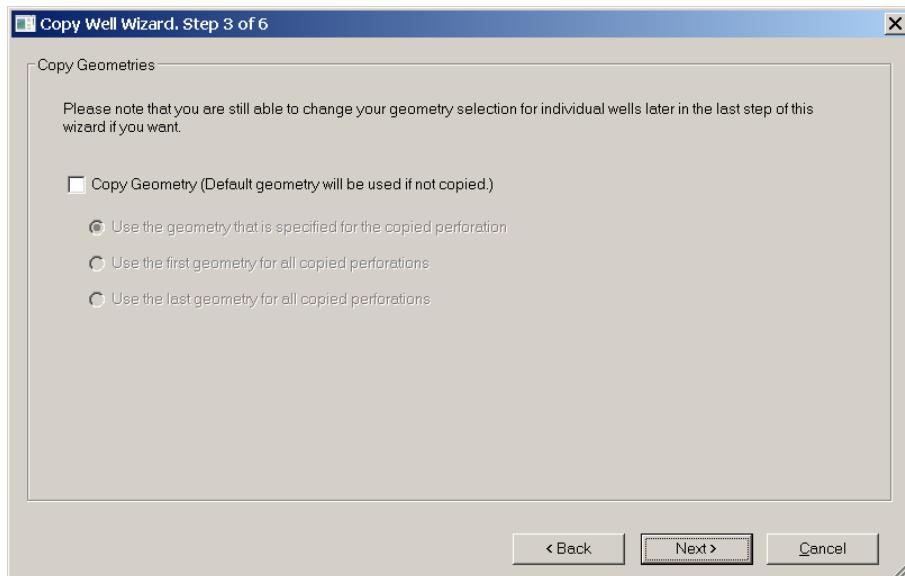
In the **Well** menu select **Copy Well** to open the wizard. The first step is to select the wells that will be copied using the selection options:



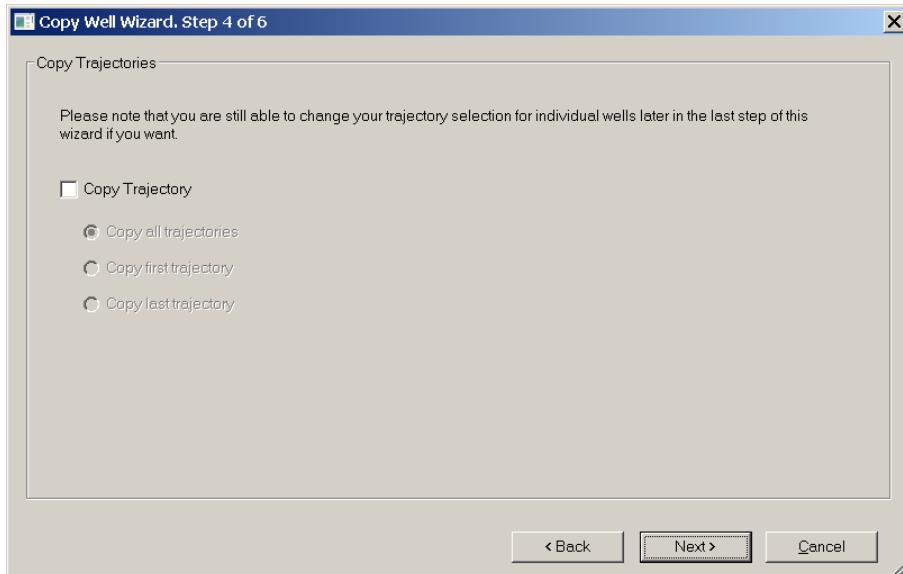
Click **Next**. The **Copy Well Wizard Step 2** dialog box is displayed. Through this dialog box, which you select the well perforation copying option. A well has to have perforation, so the perforation copying is required for the operation:



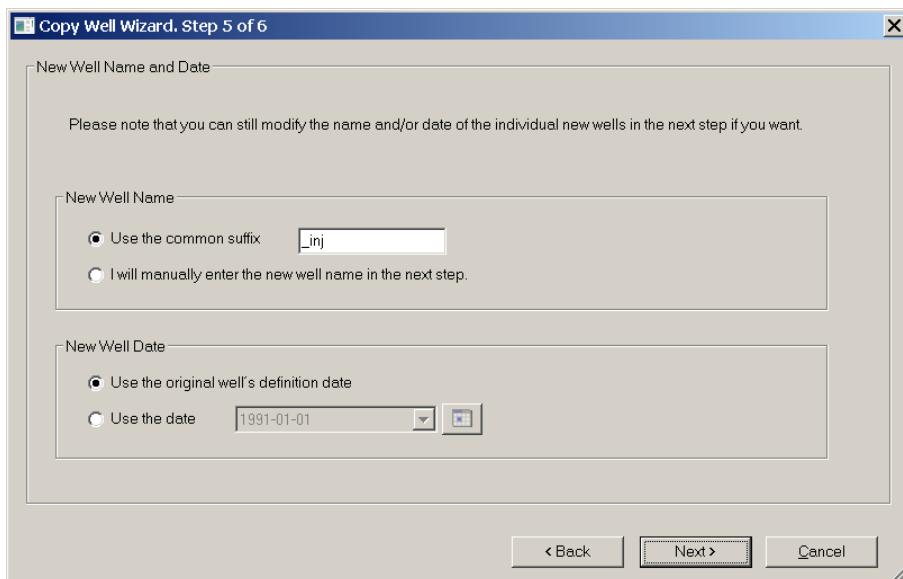
Click **Next**. The **Copy Well Wizard Step 3** dialog box is displayed. Through this dialog box, select the well geometry copying option. It is optional. Check the **Copy Geometry** check box to copy well geometry:



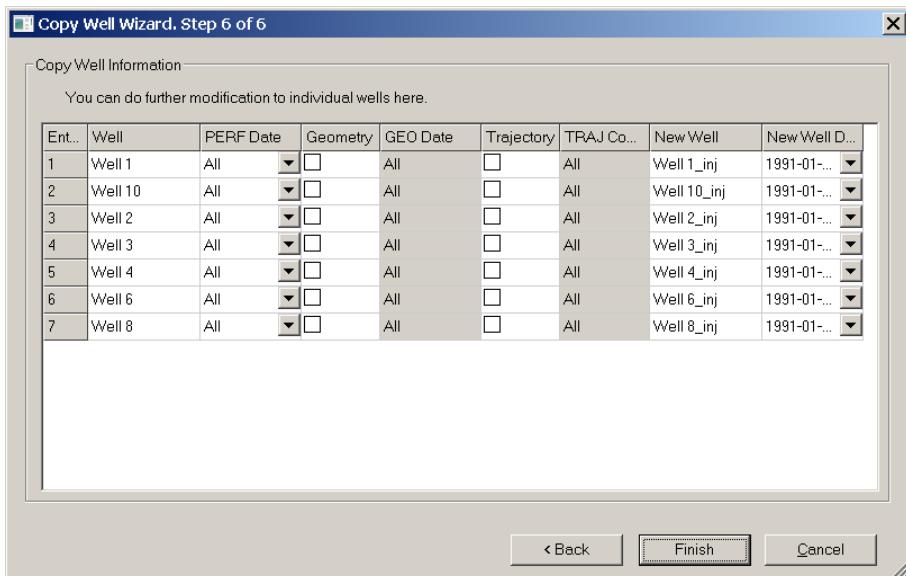
Click **Next**. The **Copy Well Wizard Step 4** dialog box is displayed. Through this dialog box, you can select the trajectory copying option. It is optional and enabled only if any of the selected wells have trajectories. Check the **Copy Trajectory** check box to copy the well trajectory:



Click **Next**. The **Copy Well Wizard Step 5** dialog box is displayed. Through this dialog box, you can specify the resulted well name suffix and definition date:



Click **Next**. The **Copy Well Wizard Step 6** dialog box is displayed. This final page shows all of the information you have entered regarding your copy well selections. At this stage, you can adjust the selections before completing the process:



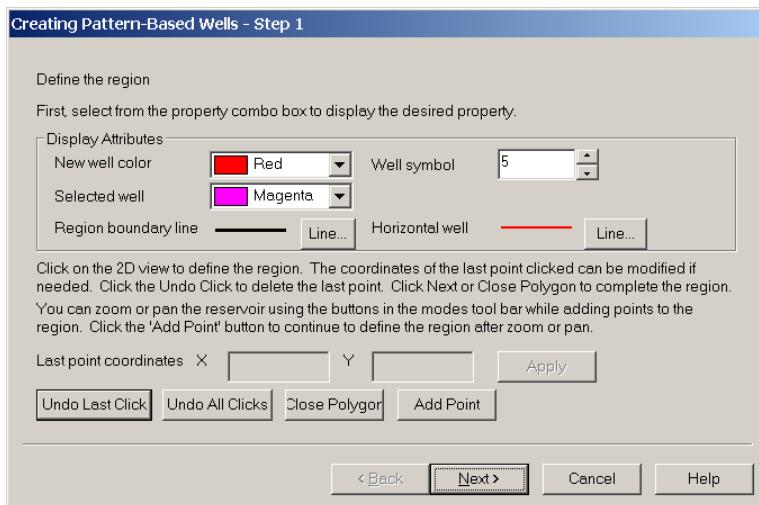
Click **Finish** or **Cancel** to complete the operation.

Creating Pattern Based Wells

The purpose of this option is to allow you to quickly create pattern-based vertical wells or producers (vertical or horizontal) in a specified region of the reservoir using a dialog wizard. The pattern can be any one of the following: 5-spot, 7-spot, 9-spot, or line drive. The pattern center can have producer or injector well, i.e., normal or inverted. You can specify the conditions for creating the perforations of the wells. The conditions can be layer number, block center depth or block property being greater or less than certain value.

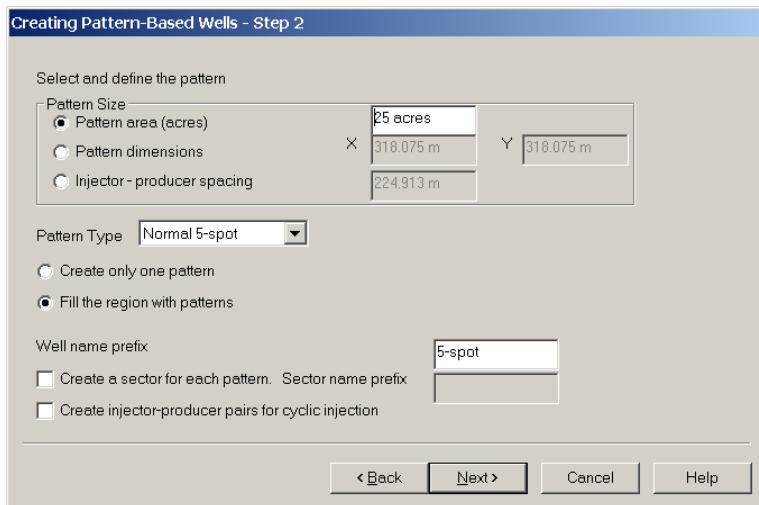
Creating Pattern-Based Wells Wizard

The wizard is accessed through **Well | Pattern-Based Wells**. The **Creating Pattern-Based Wells – Step 1** dialog box is displayed. Through this dialog box, you can define a region to contain the pattern based wells:



Note: There are four steps in this wizard. You can invoke other Builder functions while the wizard dialog box is active.

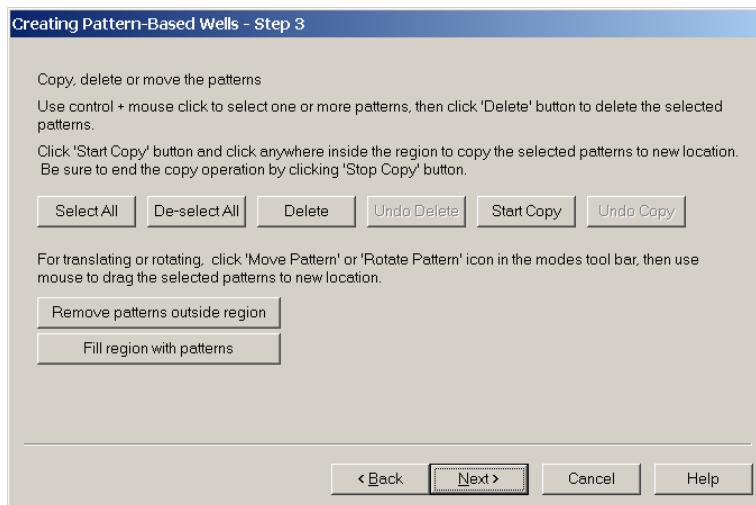
Follow the directions on the dialog box. Click **Next** or **Close Polygon** to complete the region. The **Creating Pattern-Based Wells – Step 2** dialog box is displayed:



In Step 2, you define the pattern area and select a pattern type. You can create one pattern or fill the whole region with patterns. If you select one pattern, then the next step will let you copy and paste new patterns to other locations. There is an option here to create a sector for each pattern

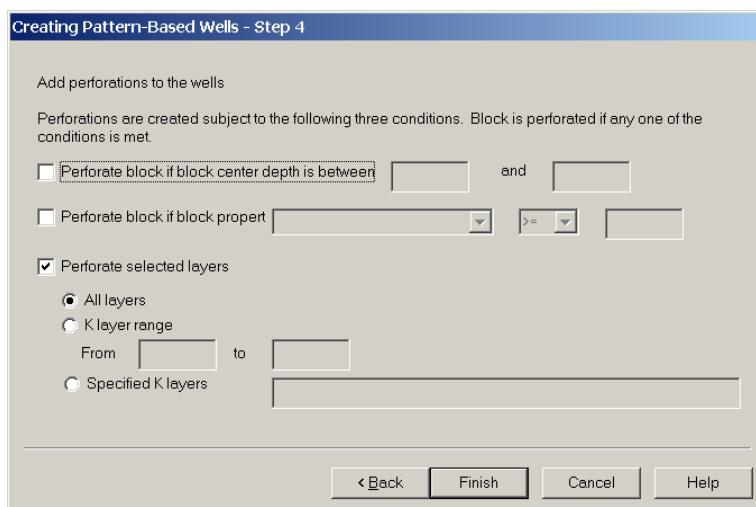
and specify a name prefix for the sector. If you choose **Create injector-producer pairs for cyclic injection**, then an injector well is created for every producer well and vice versa.

Click Next. The **Creating Pattern-Based Wells – Step 3** dialog box is displayed:



In step 3, you can copy and delete selected patterns. If you have selected **Fill the region with patterns** in step 2, then all the patterns will be selected automatically for you. You can move or rotate the selected patterns by clicking **Move Patterns** or **Rotate Patterns** button in the modes tool bar. To select multiple patterns, use the CTRL key in conjunction with mouse click. If you use the **Start Copy** button to start a series of copying, remember to click **Stop Copy** button when you are done with copying.

Click Next. The **Creating Pattern-Based Wells – Step 4** dialog box is displayed:



In step 4, you conditionally add perforations to the wells. The default is to add perforations to all layers of the wells. Note that these conditions are OR'ed to produce the final result; that is, if any condition is met, then the grid block will be perforated. The block property can be any property including a custom one.

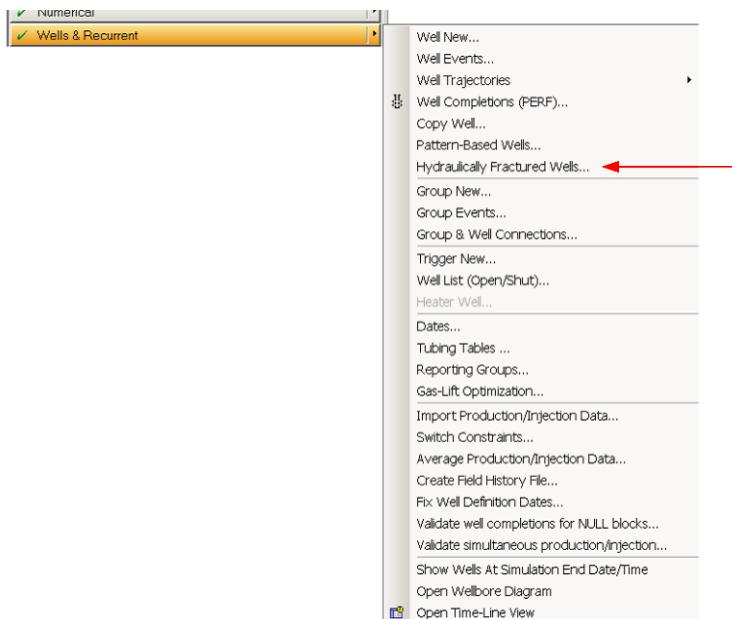
Click **Finish** to complete the wizard.

Hydraulically Fractured Wells (IMEX & GEM)

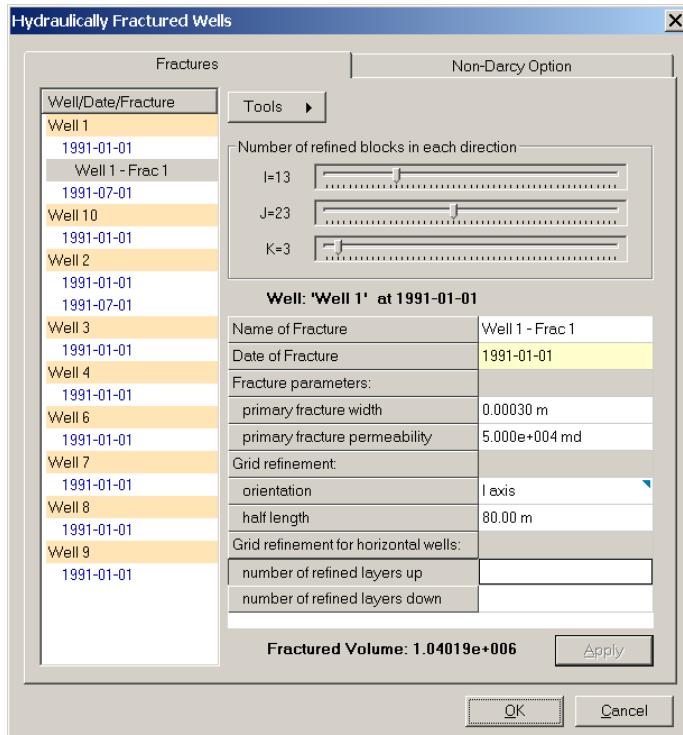
To model a fracture more explicitly than using the skin factor, you need to perform a local grid refinement and create a fairly small grid cell width that is perpendicular to the direction of the fracture. The **Hydraulically Fractured Wells** dialog box automates this task and performs Local Grid Refinement (LGR) to bring the grid-block size close to the actual fracture width.

Since in simulation the minimum grid block width cannot be less than the well radius, the smallest grid width (that is, the fracture grid) is typically set to 1 foot and then other fracture parameters are properly adjusted to arrive at a dimensionless fracture conductivity of the desired value. You can create a single-plane vertical fracture in a vertical well or multiple-plane vertical fractures in a horizontal well in Cartesian or corner point grids.

You can open the **Hydraulically Fractured Wells** dialog box through **Well | Hydraulically Fractured Wells** or through the tree view:



The **Hydraulically Fractured Wells** dialog box, shown below, contains two tabs: **Fractures** for managing fracture creation/copying/deleting and **Non-Darcy Option** for selecting correlation for non-Darcy flow calculations.

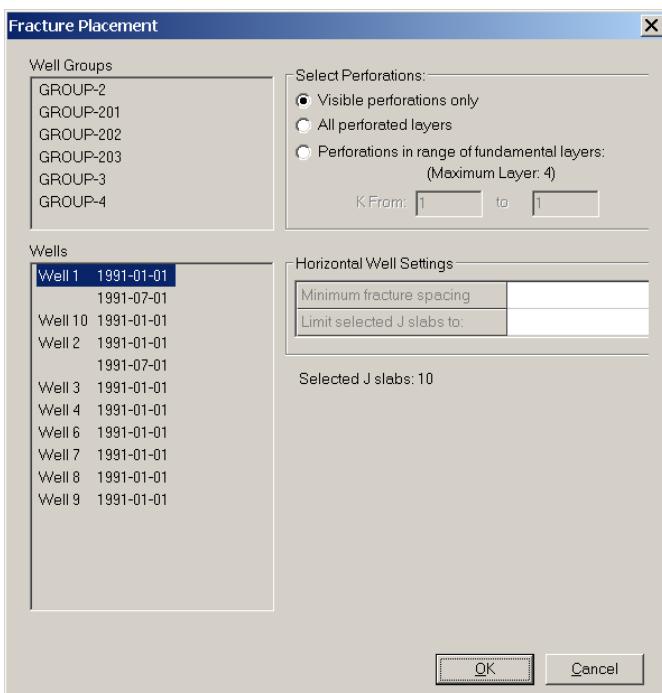


You can see all well names in the **Well/Date/Fracture** list control on the left side of the **Fractures** tab. If there are any fractures associated with a particular well, they are listed beneath that well name. If you select a fracture from the **Well/Date/Fracture** list control, the name of the selected well/fracture and all relevant fracture parameters will be displayed on the right side of the **Fractures** tab.

You can change the default **Number of refined blocks** in each direction (e.g., 13, 23, 3) by sliding the I, J and K sliders. Notice that only an odd number of refinements is allowed in I and J directions.

You can select X (I axis) or Y (J axis) **orientation** for the fracture. You also have to specify the fracture **half length**, **width** and **permeability**. You can also specify a **secondary fracture width** and **secondary fracture permeability**. For horizontal wells, you need to also specify a **number of refined layers up** and **down** from the layer perforated by the well (listed under **Grid refinement for horizontal wells**). For vertical wells, those numbers are determined from perforations.

If you want to place several fractures for horizontal wells at the same time, you can specify **Minimum fracture spacing** or select **I** or **J slabs** in the **Fracture Placement** dialog box, which is displayed when you click **Apply** in the **Fractures** tab:



Builder will try to set up several fractures spaced as closely as possible to the selected distance (starting from the end of the horizontal well), or will put fractures in the selected slabs.

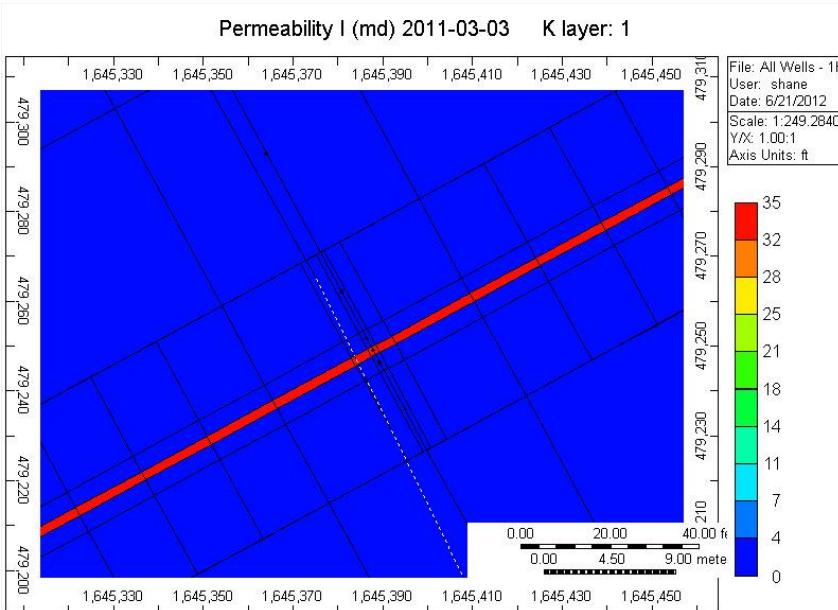
Builder will modify keywords based on the porosity type of the model and whether or not non-Darcy correlation is specified, as shown below:

| Model Porosity Type | Non-Darcy Correlation Specified? | Simulator Keywords Modified |
|----------------------------|---|--|
| Single Permeability | No | PERMI, PERMJ, PERMK |
| | Yes | PERMI, PERMJ, PERMK, NDARCYCOR |
| Dual Permeability | No | PERMI, PERMJ, PERMK MATRIX |
| | Yes | PERMI, PERMJ, PERMK MATRIX, NDARCYCOR MATRIX |
| Dual Porosity | No | PERMI, PERMJ, PERMK FRACTURE |
| | Yes | PERMI, PERMJ, PERMK FRACTURE, NDARCYCOR FRACTURE |

where:

| Simulator Keyword | Builder Property |
|------------------------------|---|
| PERMI, PERMJ, PERMK | Permeability x |
| PERMI, PERMJ, PERMK FRACTURE | Permeability x - Fracture |
| NDARCYCOR | Forchheimer Equation Beta Correction |
| NDARCYCOR FRACTURE | Forchheimer Equation Beta Correction - Fracture |

The following illustrates permeability modification in a single-permeability model:



Working with Hydraulic Fractures

The **Tools** button at the top of the **Fractures** tab displays the following selections:



To start the Hydraulically Fractured Wells Wizard:

1. Select **Hydraulically Fractured Wells** from the **Well** menu to start the wizard.

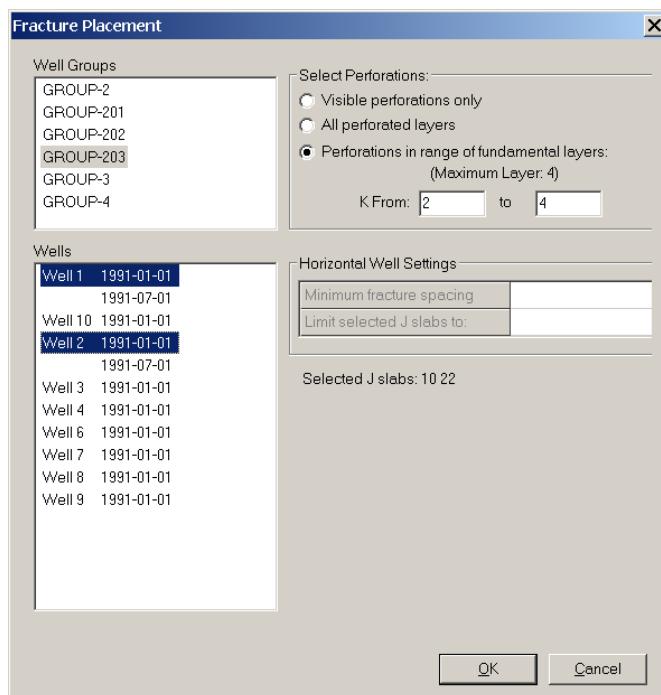
To add a new fracture:

1. Select a date for a well from the **Well/Date/Fracture** list control.

2. Click the **Tools** button and select **Add new fracture** from the popup menu.
3. A new blank (no parameters) fracture will be added to the selected well.
4. You have to specify all necessary parameters and click the **Apply** button after it is enabled.
5. The **Fracture Placement** dialog box will be displayed. Through this dialog box, you can select perforations for the selected wells.
6. You can apply the refinement to all layers in the K direction or in the wellbore direction by selecting **All perforated layers**. Alternatively, you can select **Visible perforations only** or specify the K values in **Perforations in range of fundamental layers** option. You can also specify **Minimum fracture spacing** or select **I or J slabs** for horizontal wells.
7. Click **OK** to start the calculations and create a fracture.

To copy a fracture to wells:

1. Select a fracture from the **Well/Date/Fracture** list control. All relevant fracture parameters will be displayed on the right side of the **Fractures** tab. You can modify any of these parameters before copying to other wells.
2. Click the **Tools** button and select **Copy fracture to wells** from the pop-up menu.
3. The **Fracture Placement** dialog box will be displayed. Through this dialog box, you can select wells and perforations for the selected wells. You can also specify **Minimum fracture spacing** or select **I or J slabs** for horizontal wells.



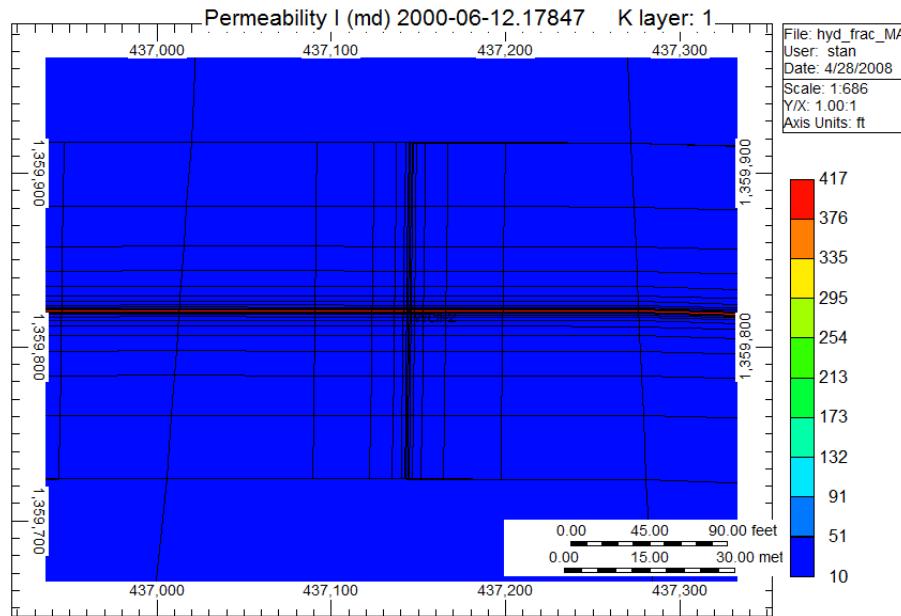
4. Select the desired wells in the **Wells** list (multiple selections are allowed). You can also select all the wells in a Group by selecting the desired group in the **Well Groups** list.
5. You can apply the refinement to all layers in the K direction or wellbore direction by selecting the **All perforated layers** radio button. Alternatively, you can choose **Visible perforations only** or specify the K values in **Perforations in range of fundamental layers** option.
6. You can also specify **Minimum fracture spacing** or select **I** or **J slabs** for horizontal wells.
7. Click **OK** to copy the fracture(s).

To delete selected fractures:

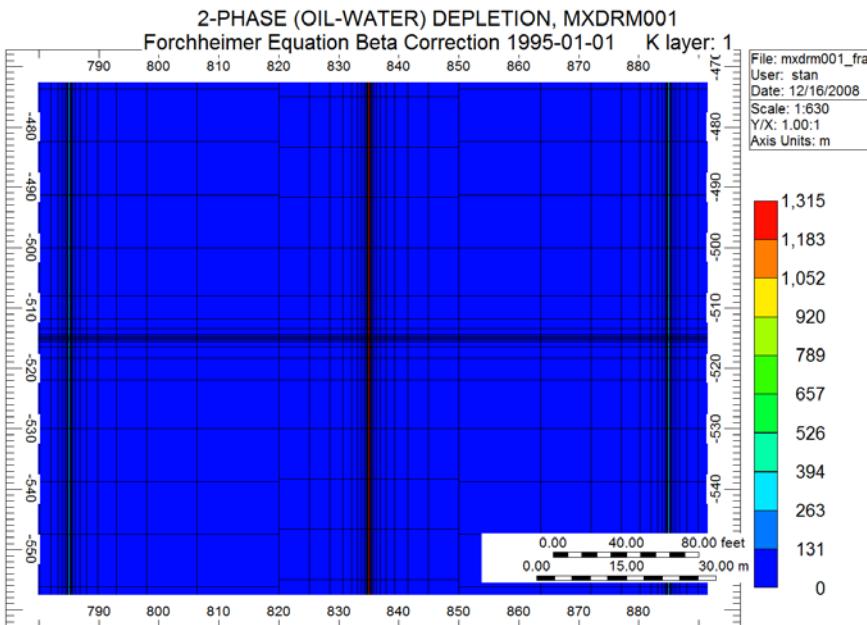
1. Select a fracture from the **Well/Date/Fracture** list control.
2. Click the **Tools** button and select **Delete selected fractures** from the pop-up menu.
3. The selected fracture will be deleted from the list, and any associated grid refinements will be undone.

You can zoom in to see the refinement for the fractures and how the **Permeability** and **Forchheimer Equation Beta Correction** properties have been modified for the grid blocks corresponding to the fractures.

IJ-2D Areal View Showing the Permeability Changes in the I Direction due to Fractures:



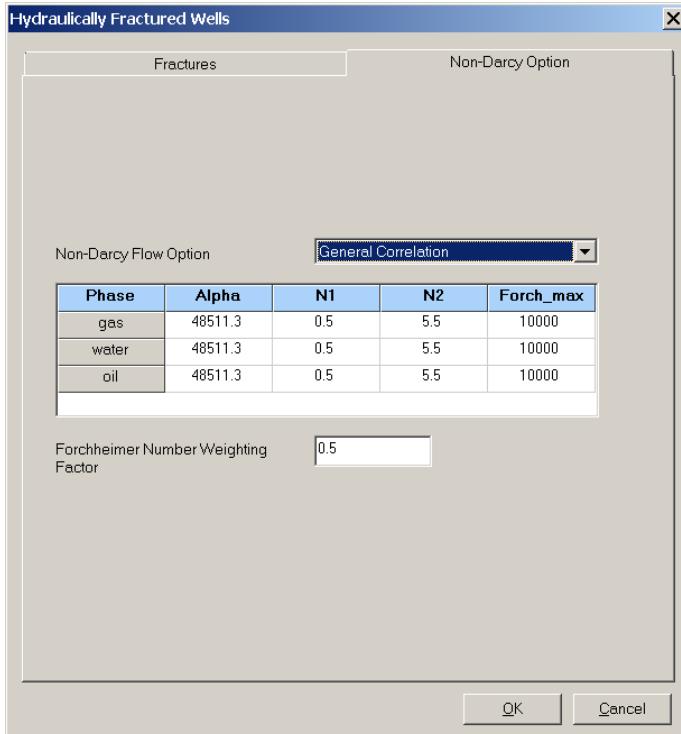
IJ-2D Areal View Showing the changes in the Forchheimer Equation Beta Correction Factor:



You can also view the fracture in 3D View by selecting the fracture in the **Well/Date/Fracture** list control and switching to 3D View in **Builder**. If you have multiple fractures for a particular date, click on the associated date to see them in 2D or 3D. If you want to see every fracture for a particular well in 2D or 3D, click that well in the **Well/Date/Fracture** list control. The **Fractured Volume** will also be displayed at the bottom of **Hydraulically Fractured Wells** dialog box based on your selection.

Specifying Non-Darcy Information for Hydraulic Fractures

The **Non-Darcy Option** tab in the **Hydraulically Fractured Wells** dialog box allows for quick selection of correlation for non-Darcy flow calculations. Note that this information can also be specified through **Rock-Fluid | Rock Fluid Options** in the **Builder** menu when you are working with IMEX datasets.



When using the **General Correlation** option (NONDARCY GENERAL) you can enter a table of Alpha, N1, N2 and Forch_max for each phase in the reservoir, and modify the Forchheimer Number Weighting Factor.

When using the **Geertsma or Frederick and Graves Correlation** options (NONDARCY GEERTSMA, FG1 or FG2) non-Darcy flow coefficient (β) for all phases except gas is assumed to be zero. For gas, the parameters for α_g , $N1_g$ and $N2_g$ are displayed but cannot be modified.

Creating a Stimulated Reservoir Volume (SRV) Stage

The **Hydraulically Fractured Wells** dialog box enables you to create an SRV Stage based on imported microseismic data (for further information, refer to [Microseismic Events](#) on page 389) or through interactive block selection. Both methods are facilitated by the **Create SRV Stage Wizard** accessible through the dialog box.

To create an SRV stage:

In the **Hydraulically Fractured Wells** dialog box, under the **Well/Date/Fracture** list control, select the date to be used for creating an SRV stage.

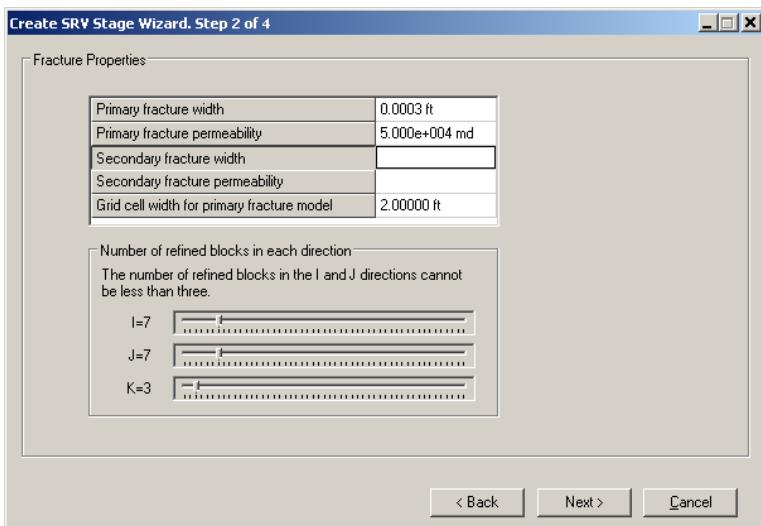
Click the **Tools** button and then select **Add new SRV stage** in the pop-up menu. This will display the **Create SRV Stage Wizard Step 1** dialog box.

Step 1: Choose the mechanism by which to create the SRV stage



In step 1, you decide whether you are going to use microseismic data or the mouse to create the SRV stage. The microseismic option is only available if you have loaded microseismic data (refer to [Microseismic Events](#)). If you choose microseismic data, you will see a list of the stages associated with the well you chose in the **Well/Date/Fracture** list control. These stages are listed in the **Well / Stage** drop-down box and are derived from the file containing microseismic information. The reservoir blocks containing microseismic events associated with the chosen stage will automatically be selected but you will have the option of adding or removing blocks interactively. If you want to only use the mouse, choose the second option. Once you make your selection click **Next**.

Step 2: Set Fracture properties for the SRV stage



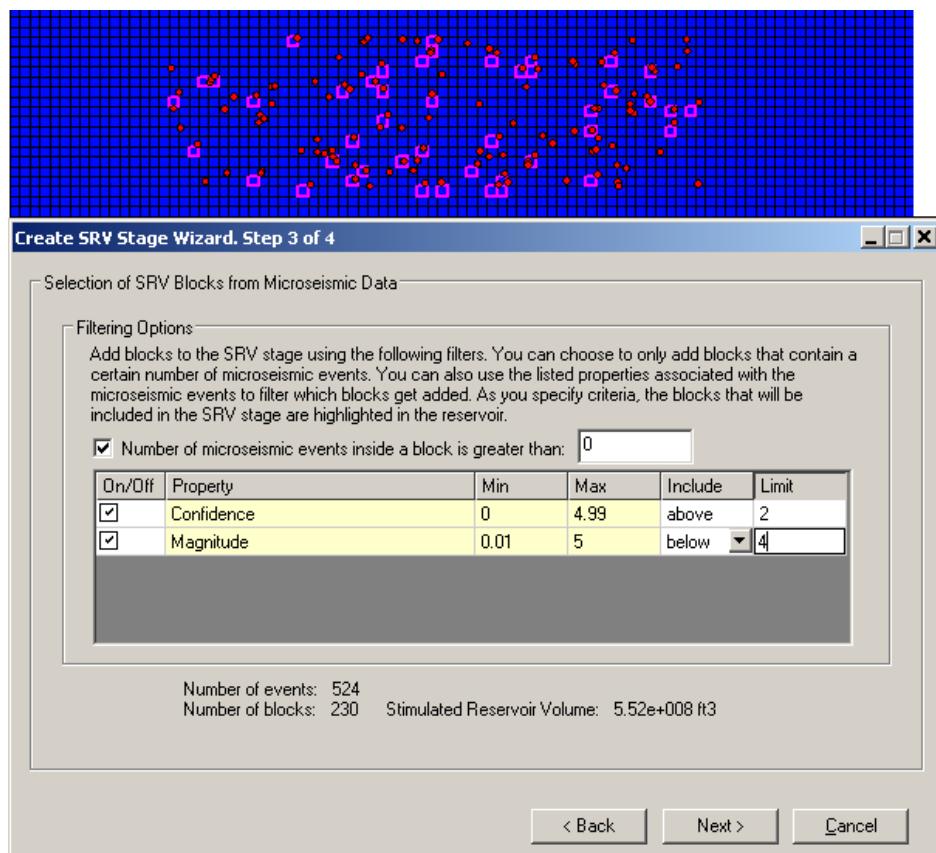
In Step 2, set the **Primary fracture width** and **Primary fracture permeability** for the fracture. You can supply values for the **Secondary fracture width** and **Secondary fracture permeability**. By default, **Grid cell width for primary fracture model** is set at 2 ft or 0.6096 m.

Primary fracture width is modeled as a set of center narrow blocks in a refined fundamental block. **MATRIX** permeability in these blocks is modified to account for the values you enter. You can usually see this permeability modification in the grid if you zoom into the fracture.

The secondary fracture properties are used only if you have a dual perm model. In this case, **FRACTURE** permeability is modified for all remaining refined blocks (the fundamental block minus the set of center narrow blocks that model the fracture).

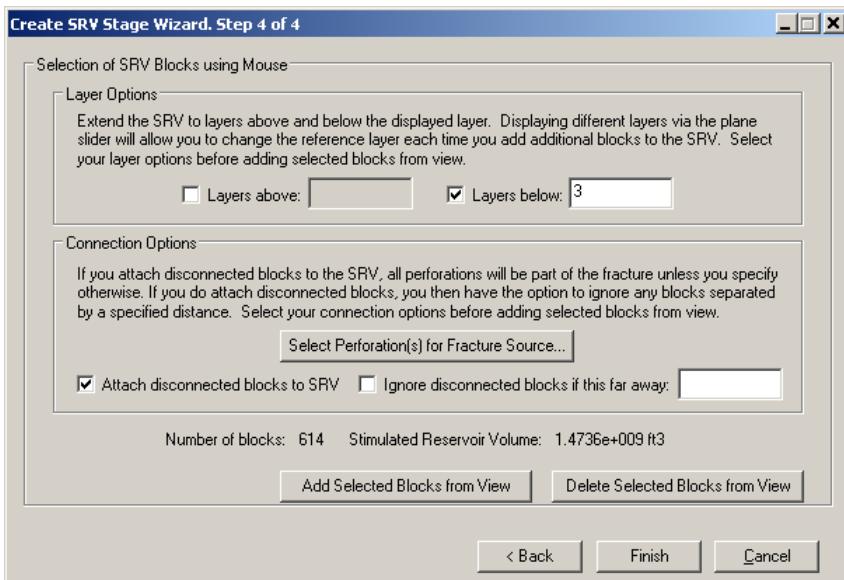
Note: Permeabilities are only modified if the fundamental block size in I/J direction is greater than **DIFRAC/DJFRAC**.

Step 3: Filter blocks by Microseismic Properties



Step 3 of the wizard will only appear if you chose to incorporate microseismic data. This step enables you to filter which blocks get added to the SRV stage based on particular microseismic properties. In this example, microseismic events contain a **Confidence** and **Magnitude** property. You can see the lower and upper bound for each. Using the filter, only blocks containing microseismic events that have a **Confidence** above 2 and a **Magnitude** below 4 will be added. In addition, only blocks that contain a microseismic event will become part of the SRV stage. As you make changes, the corresponding blocks will be highlighted in the reservoir. At the bottom of the dialog box, you can see the total **Number of events**, the **Number of blocks** that will form the SRV stage, and the **Stimulated Reservoir Volume** based on your selection. When you are done, click **Next**.

Step 4: Layer and Connection Options

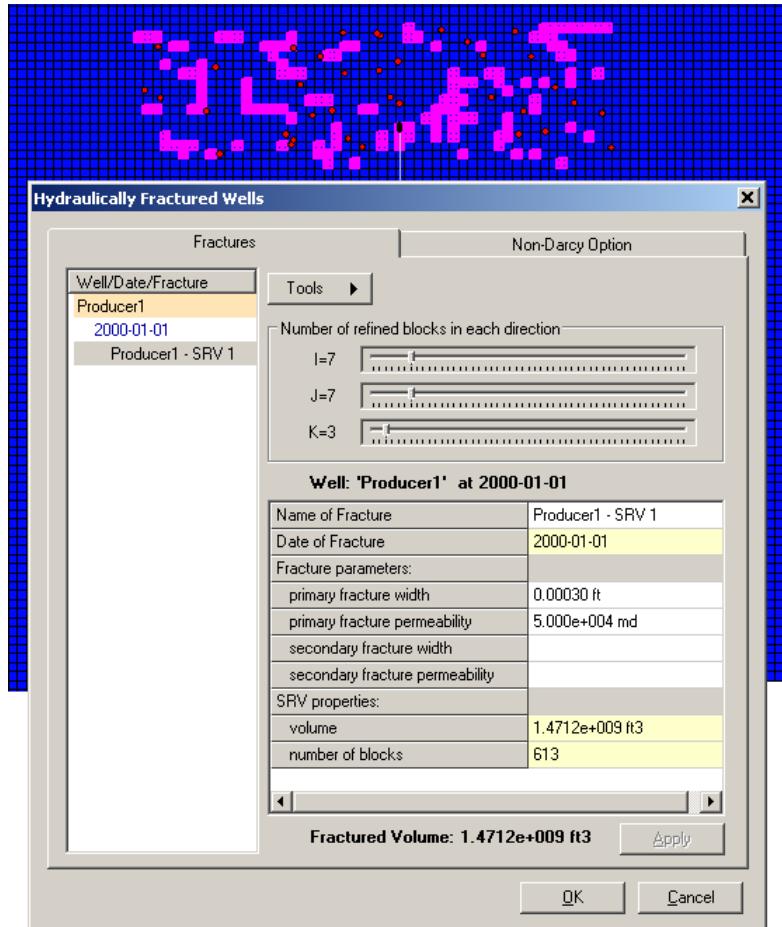


This is the last step of the wizard. Set your layer and connection options before adding additional blocks with the mouse.

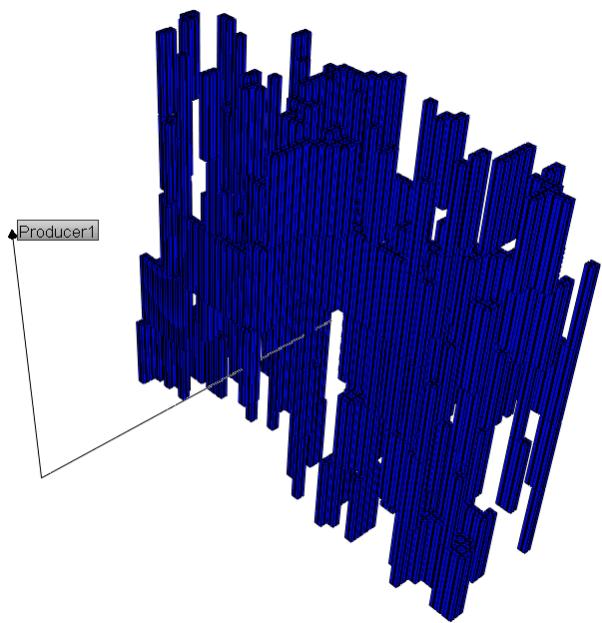
For **Layer Options**, extend the SRV stage to layers above and below the currently displayed layer by entering values for **Layers above** and **Layers below**. Using Builder's plane slider, you can change the reference layer each time you add additional blocks to the SRV.

For **Connection Options**, all perforations will be included in the SRV stage unless you specify otherwise by clicking on **Select Perforation(s) for Fracture Source**. This will bring up a small dialog box through which you can view and choose perforations. After you have chosen layers and/or perforations, select **Attach disconnected blocks to SRV**. You can ignore disconnected blocks if they are too far away by specifying a distance. You will see the reservoir display update, and the new blocks that have become part of the SRV stage will be highlighted. In addition, the **Number of blocks** and **Stimulated Reservoir Volume** will be updated.

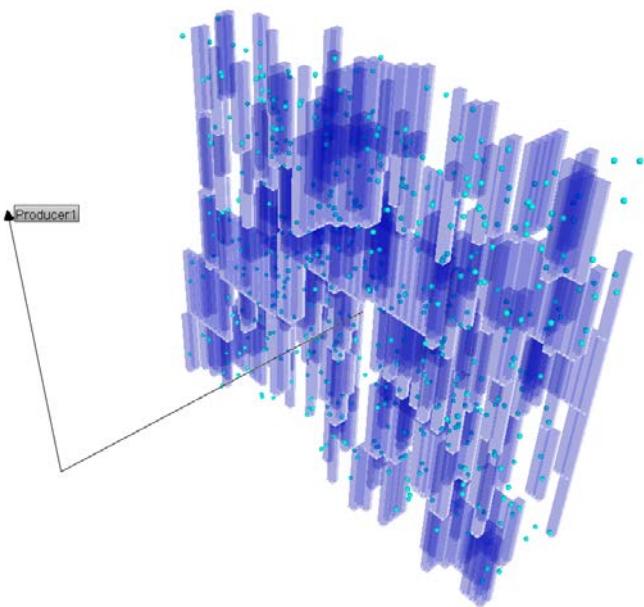
Once your layer and connection options are set, add additional blocks to the SRV via the mouse by selecting blocks in the reservoir. When you are done selecting blocks, click **Add Selected Blocks from View**. You can repeat this process as many times as you want. When you are done, click **Finish**. The status bar will update as the SRV stage is created. Once it is created, the SRV stage will be included in the **Well/Date/Fracture** list control of the **Hydraulically Fractured Wells** dialog box.



The **Hydraulically Fractured Wells** dialog box will update to indicate the parameters associated with the newly created SRV stage. In addition, the blocks corresponding to the SRV stage will be highlighted in the reservoir. With the dialog box open, select the SRV in the **Well/Date/Fracture** list control and switch to **3D View** to view the SRV in 3D. Click on a date to see all SRV stages associated with that date. Click on a well to see all SRV stages associated with that well.



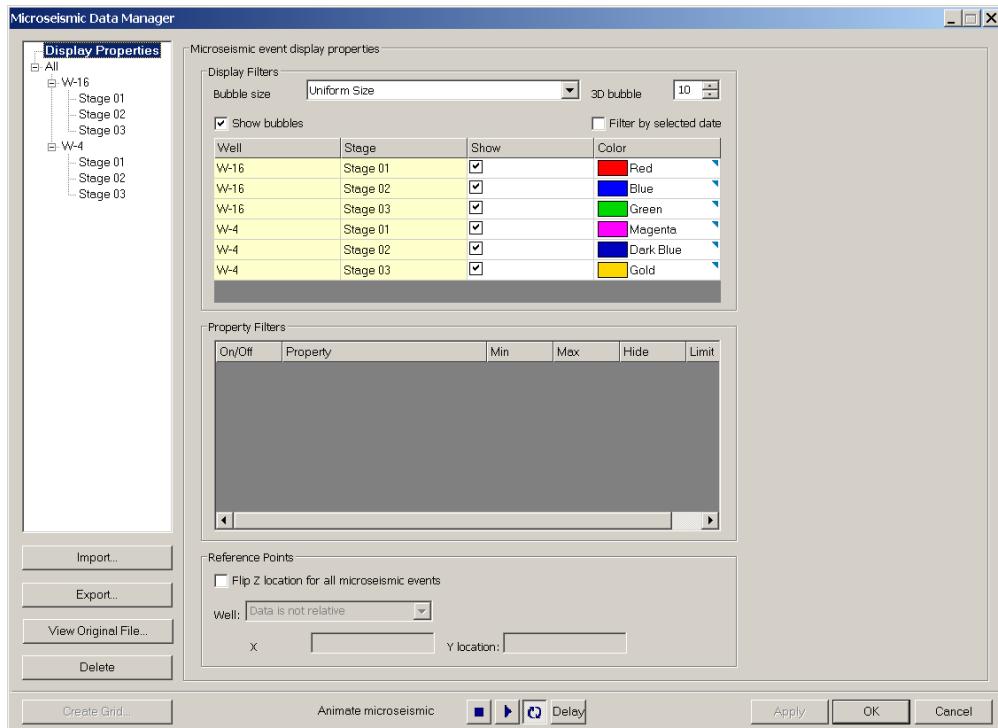
Use the **3D Properties** pane to modify the display to better view specific properties such as **Permeability I**. Generally speaking, permeability is best viewed by turning off **Show Grid** and turning on **Show Outlines**. You can then zoom in to see permeability



If your SRV stage was created via microseismic data, you can view the microseismic data by reducing the **Transparency**. If you want to change the display properties of the microseismic events, close the **Hydraulically Fractured Wells** dialog box and then select **Tools | Microseismic Data Manager** (refer to [Display Properties](#) in the Microseismic Events section for information about manipulating the appearance of microseismic events and which ones are displayed).

Microseismic Events

Builder allows you to import microseismic data to help in modeling your Stimulated Reservoir Volume (SRV). All microseismic related functionality is maintained in the **Microseismic Data Manager** available under the **Tools** menu.



Importing Microseismic Data

Builder supports microseismic data in fixed width, delimited, or in a CMG-specific XML format.

Data Format

Before being able to use an XML file, you must load fixed width or delimited data (usually a .csv file created in Microsoft Excel). The data must be organized into columns, with one type of data in each column. Column names or headers must be the first line of the file. Required columns include a reference to the well, stage, date, x-coordinate, y-coordinate, and z-coordinate. However, the column headers can have any name in the original file. The x and y

coordinates can be relative to a reference point but the z coordinate must be in absolute or reservoir coordinates. Time and additional microseismic properties are optional. (**NOTE:** If Time exists in addition to Date, they must be in separate columns.) The following is a simple example of the format.

| | A | B | C | D | E | F | G | H |
|----|------|----------|----------|-----|------|------|------|------|
| 1 | Well | Stage | Date | X | Y | Z | Mag | Conf |
| 2 | W-16 | Stage 01 | 1/1/1988 | 385 | 1282 | 1927 | 1.83 | 2.45 |
| 3 | | Stage 01 | 1/1/1988 | 285 | 1270 | 1929 | 2.65 | 2.15 |
| 4 | | Stage 01 | 1/1/1988 | 390 | 1060 | 1925 | 3.85 | 2.35 |
| 5 | | Stage 01 | 1/1/1988 | 519 | 1147 | 1921 | 2.01 | 2.01 |
| 6 | | Stage 01 | 1/1/1988 | 209 | 1013 | 1930 | 2.45 | 3.58 |
| 7 | | Stage 01 | 1/1/1988 | 396 | 1212 | 1926 | 4.85 | 2.87 |
| 8 | | Stage 01 | 1/1/1988 | 314 | 1182 | 1928 | 2.82 | 2.98 |
| 9 | | Stage 01 | 1/1/1988 | 221 | 1165 | 1930 | 2.87 | 2 |
| 10 | | Stage 01 | 1/1/1988 | 273 | 1124 | 1928 | 2.96 | 3.47 |
| 11 | | Stage 01 | 1/1/1988 | 256 | 1065 | 1929 | 3.33 | 2.44 |
| 12 | | Stage 01 | 1/1/1988 | 133 | 1141 | 1932 | 2.14 | 2.33 |

To load a file containing microseismic data, click on the **Import** button in the **Microseismic Data Manager**. You will be presented with a wizard that will guide you through the import process.

Step 1: Choosing the file and format

Import Microseismic Data

Step 1: Choose the file containing microseismic event data.

For non-xml files, please verify the following:

- The data must be organized into columns, with one type of data in each column.
- Column names must be the first line of the file followed by actual data.
- Columns must include Well Name, Stage, Date, X, Y, Z. (Time, Magnitude, and Confidence are optional.)
- Years must be four digits since a two digit year is not accepted. Time must be formatted as HH:MM:SS.
- The data can be fixed width or delimited.

File selection

Name of the file containing microseismic event data:
\\wsgroup\QATesting\TestData\Microseismic_Files\Watfld Samples\watfld_microseismic.csv

Data field type for non-XML files

Fixed Width - Data is present in columns of fixed width
 Delimited - Characters such as commas, spaces or tabs separate data fields. Columns can vary in width

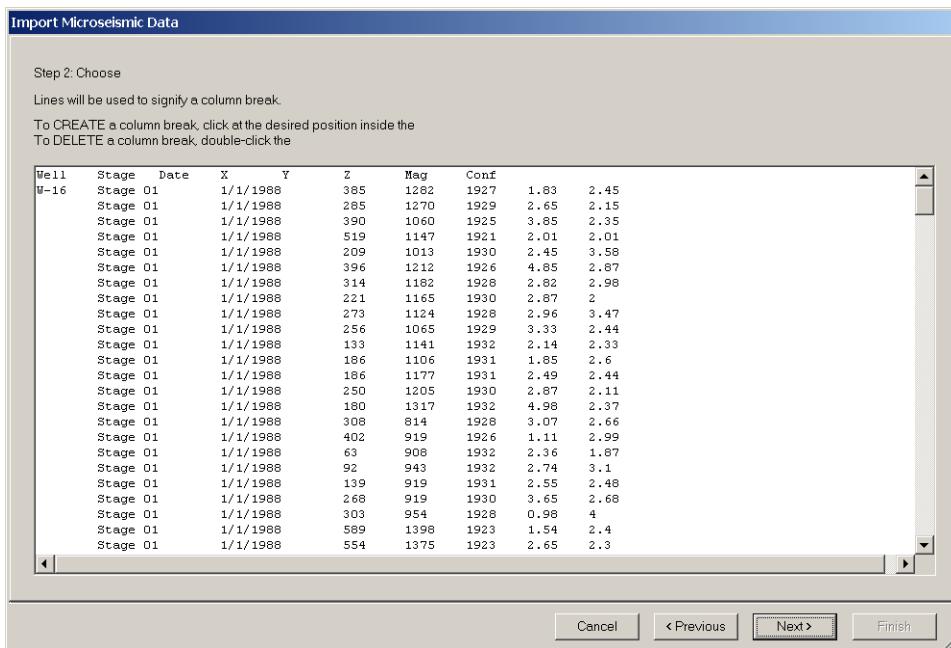
Number format for delimited files

Commas separate data
 Commas for thousands (Example: 5,000 = five thousand)
 Commas as decimals (Example: 5,2 = five + 2/10)

In step 1, you are presented with a reminder of the required data format for non-xml files. Once your data is suitable, click **Browse** to select your file. The file filter automatically defaults to XML format so select **All Files**. Once you have selected your file, indicate whether it is **Fixed Width** or **Delimited** by selecting the appropriate radio button. If it is delimited, indicate how commas are used in the file. After you are done, click **Next**.

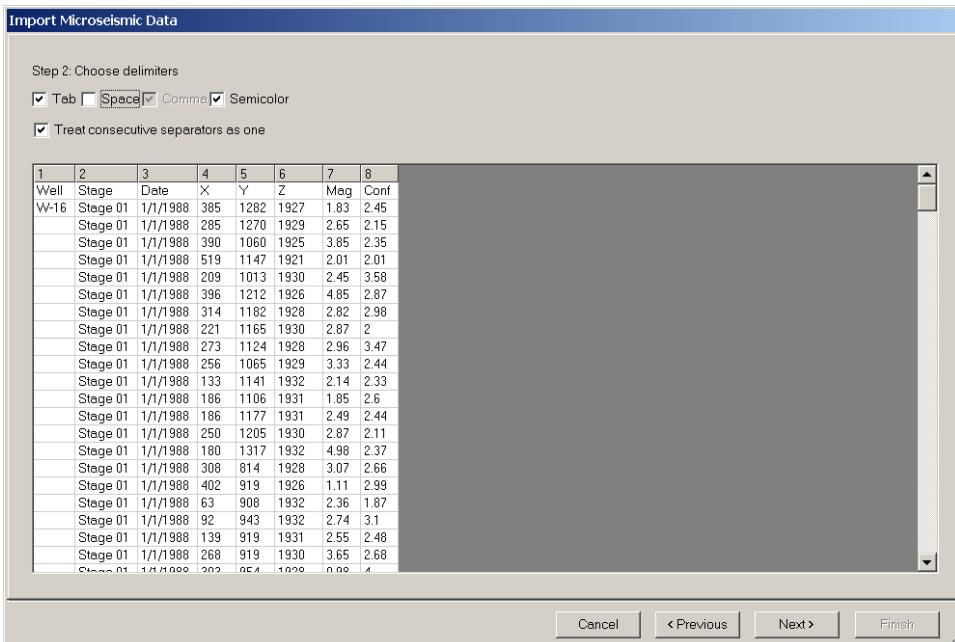
Step 2: Choose spacing or delimiters

If your file is fixed width, you will be presented with the following screen:



Follow the instructions on the screen to create columns with the mouse. When you are done, click **Next**.

If your file is delimited, you will instead be presented with the following screen:



At this point, make sure the delimiters are behaving as expected and that all data correctly corresponds to a column heading. In the above screen, note how the stage number is incorrectly listed under the **Date** column. This is because the **Space** delimiter is checked. Once it is unchecked, the columns are lined up correctly with the data. When you are done, click **Next**.

Note: When using XML format, you will not have to visit either of these two screens.

Step 3: Set or verify column details

Import Microseismic Data

Step 3: Choose column details and date format
Well Name, Stage, Date, X-Coordinate, Y-Coordinate, and Z-Coordinate must be selected.
Valid date delimiters include forward slash (/), dash (-), dot (.), and a space.

| Well Name | Stage | Date | X-Coordinate | Y-Coordinate | Z-Coordinate | Ignore Column | Ignore |
|-----------|----------|----------|--------------|--------------|--------------|---------------|--------|
| Well | Stage | Date | X | Y | Z | Mag | Conf |
| 1 W-16 | Stage 01 | 1/1/1988 | 285 | 1282 | 1927 | 1.83 | 2.45 |
| 2 | Stage 01 | 1/1/1988 | 285 | 1270 | 1929 | 2.65 | 2.15 |
| 3 | Stage 01 | 1/1/1988 | 390 | 1060 | 1925 | 3.85 | 2.35 |
| 4 | Stage 01 | 1/1/1988 | 519 | 1147 | 1921 | 2.01 | 2.01 |
| 5 | Stage 01 | 1/1/1988 | 209 | 1013 | 1930 | 2.45 | 3.58 |
| 6 | Stage 01 | 1/1/1988 | 396 | 1212 | 1926 | 4.85 | 2.87 |
| 7 | Stage 01 | 1/1/1988 | 314 | 1182 | 1928 | 2.82 | 2.98 |
| 8 | Stage 01 | 1/1/1988 | 221 | 1165 | 1930 | 2.87 | 2 |
| 9 | Stage 01 | 1/1/1988 | 273 | 1124 | 1928 | 2.96 | 3.47 |
| 10 | Stage 01 | 1/1/1988 | 256 | 1065 | 1929 | 3.33 | 2.44 |
| 11 | Stage 01 | 1/1/1988 | 133 | 1141 | 1932 | 2.14 | 2.33 |
| 12 | Stage 01 | 1/1/1988 | 186 | 1106 | 1931 | 1.85 | 2.6 |
| 13 | Stage 01 | 1/1/1988 | 186 | 1177 | 1931 | 2.49 | 2.44 |
| 14 | Stage 01 | 1/1/1988 | 250 | 1205 | 1930 | 2.87 | 2.11 |
| 15 | Stage 01 | 1/1/1988 | 200 | 1017 | 1922 | 4.00 | 2.07 |

Event coordinates are relative to a reference point Flip Z location for all microseismic events

Date format: Month Day Year

Cancel < Previous Next > Finish

At this point with non-XML files, you must select what each column represents via the drop down list. (This information is already populated when using an XML file.) The **Next** button will only become activated when the required columns are picked and there are no duplicates. For additional microseismic properties such as **Magnitude** and **Confidence**, indicate that you would like to use the associated data by selecting **Include Column**. Make sure the **Date format** selected correctly corresponds to your **Date** column.

If your x and y coordinates are relative to a specific location, you can indicate this by selecting **Event coordinates are relative to a reference point**. If your z coordinates are inverse to those used in the dataset, you can also choose **Flip Z location for all microseismic events**. It is important to set this information correctly otherwise microseismic events may not appear in the display. However, you can still change this with the **Microseismic Data Manager** if you make a mistake during import. Once you are satisfied, click **Next**.

Step 4: Well and microseismic data options

Import Microseismic Data

Step 4: Well and microseismic data

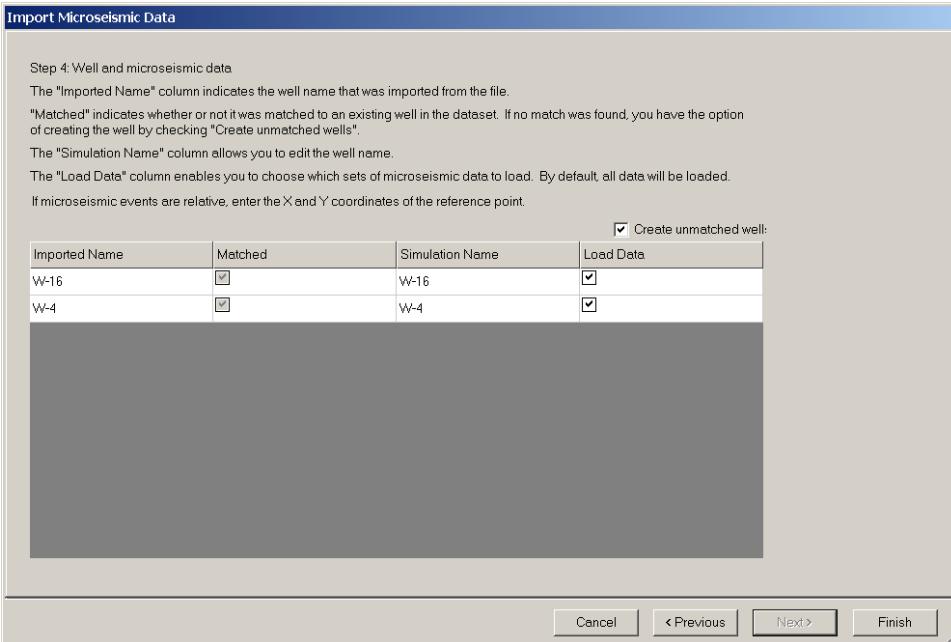
The "Imported Name" column indicates the well name that was imported from the file. "Matched" indicates whether or not it was matched to an existing well in the dataset. If no match was found, you have the option of creating the well by checking "Create unmatched wells". The "Simulation Name" column allows you to edit the well name. The "Load Data" column enables you to choose which sets of microseismic data to load. By default, all data will be loaded.

If microseismic events are relative, enter the X and Y coordinates of the reference point.

Create unmatched well:

| Imported Name | Matched | Simulation Name | Load Data |
|---------------|-------------------------------------|-----------------|-------------------------------------|
| W-16 | <input checked="" type="checkbox"/> | W-16 | <input checked="" type="checkbox"/> |
| W-4 | <input checked="" type="checkbox"/> | W-4 | <input checked="" type="checkbox"/> |

Cancel < Previous Next > Finish



This step acts like a summary and presents you with some options. First, the **Imported Name** column tells you the name of the wells in your microseismic data. The **Matched** column indicates whether this name matches the name of a well in the dataset. If it does not, you have the option of creating it at this point by selecting **Create unmatched wells**. The **Simulation Name** column indicates the name of the well used for simulation purposes and is editable. The **Load Data** column enables you to indicate whether the microseismic data associated with the well should be loaded. By default, all microseismic data is loaded. Last, if you indicated in the previous step that x and y coordinates were relative to a reference point, you must enter the reference point here. Click **Finish** when you are done and you will be taken back to the **Microseismic Data Manager**.

Exporting Microseismic Data

Now that you have loaded microseismic data via a fixed width or delimited file, you can expedite future loading by saving the file in XML format.

In the **Microseismic Data Manager**, the tree view selection determines what data is saved to XML. For instance, if you select **All**, every microseismic event for every stage and well will be saved. If you click on a particular well, only the stages for that well will be saved. Last, if you click on a single stage, only that stage will be saved. Once you have selected an item in the tree view, click on **Export** to save the file in XML format. If your data uses relative coordinates, you are presented with the option to save them as coordinates relative to the reference point or as absolute coordinates.

Deleting Microseismic Data

As with exporting, the tree view also governs what is deleted. If you click on **All**, a well, or a stage, you can then delete that item by clicking on the **Delete** button. Alternatively, you can right-click on any particular item to bring up a context menu which will give you the option to delete.

Viewing the Original File

At any point in the **Microseismic Data Manager**, you can see from where the microseismic data for a particular stage originated. To do this, click on a particular stage and select **View Original File**. A new window will appear showing you the contents of the file

Display Properties

Display Filters

The **Microseismic Data Manager** provides many tools to manipulate not only how microseismic events look but also which microseismic events are displayed. To access these features, click on **Display Properties** in the tree view. Here you can manage both your 2D and 3D display. Each of these tools is described below.



Bubble size

In 2D, every microseismic event is represented by a small circle of uniform size. In 3D, you can control not only the size of the event but how this size is determined.

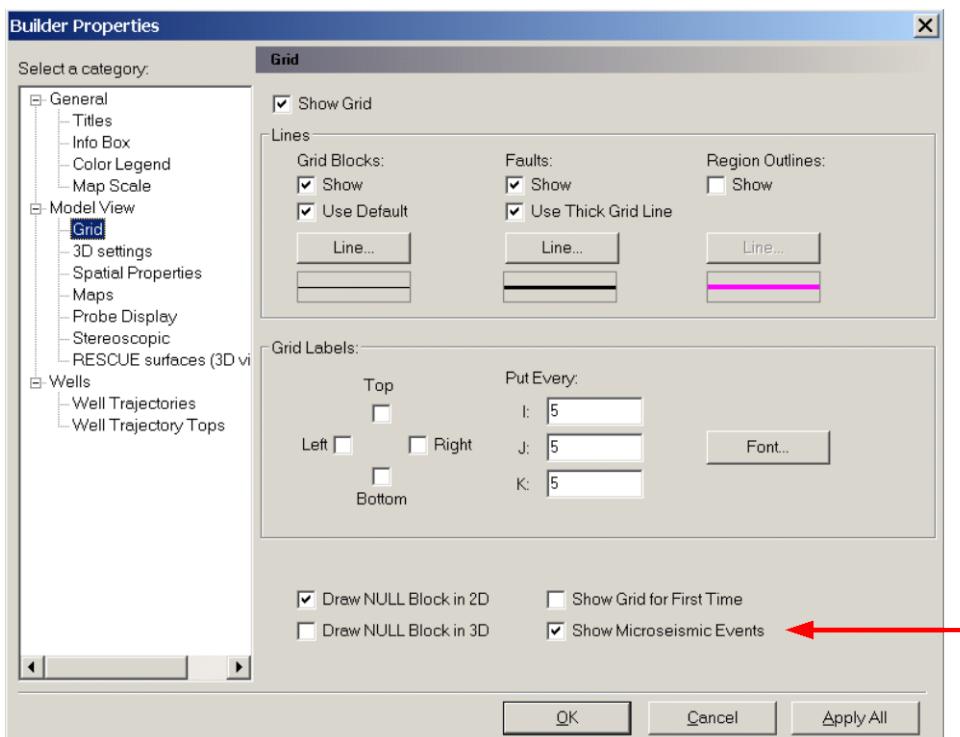
In the above example, you can see the properties available in the drop down box. If you did not load any additional properties during import (see [Step 3: Set or verify column details](#) on page 393) then **Uniform Size** will be your only option. However, if you did import a property like **Magnitude**, for example, then the bubble size is drawn relative to the actual magnitude data for the event. To change which property governs the size of the microseismic event in 3D, select the property and then click **Apply**.

Sometimes due to the property being used or the size of the reservoir, the microseismic events will be extremely large or too small to see easily. As a result, you can use the **3D bubble size** option to change the size to something more suitable. Unlike the property which governs the actual size of an event, this number will simply scale all events to make them larger or smaller. Once you have increased or decreased this number, click **Apply** to see the results.

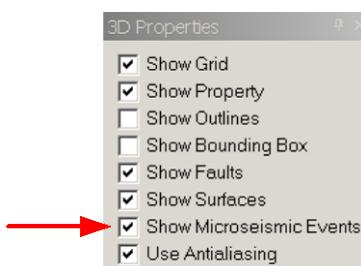
Turn bubbles on/off or change color

To turn off all bubbles, uncheck **Show bubbles**. If you just want to turn off an individual stage, uncheck the corresponding check box under the **Show** column. To change the color of a particular stage, use the drop down list under the **Color** column to select a new color. After any of these actions, click **Apply** to see the changes.

If you do not have the **Microseismic Data Manager** open and simply want to turn off all microseismic events, there are a couple of convenience options available. If you right-click on the reservoir and select **Properties**, the **Builder Properties** dialog box is displayed. Under the **Grid** category in the tree view, you can check or uncheck **Show Microseismic Events**.



Alternatively, if you are in 3D view, you can use the **3D Properties** window pane to turn microseismic events on or off by toggling the **Show Microseismic Events** check box.

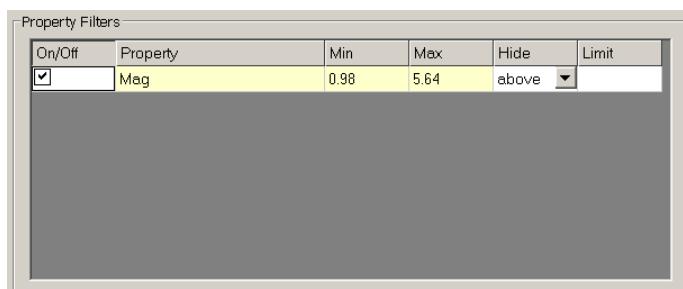


Filter by selected date

Another way to change which microseismic events are displayed is to turn on **Filter by selected date**. When you load microseismic data, dates associated with the events are added to the date drop down box in **Builder**. If you turn on this filter, then only events associated with the selected date in the drop down box will be displayed. Each date is typically associated with one stage.

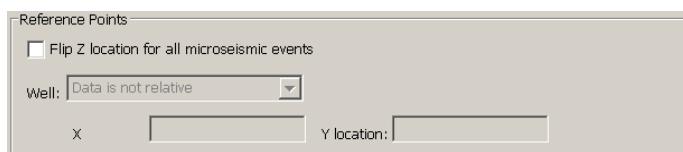
Property Filters

If you load additional properties during import (*see Step 3 described earlier*), then you have the option to filter by those properties to control which microseismic events are displayed.



In this particular example, the microseismic events have only a **Mag** property. The **On/Off** column indicates whether you want to use this property filter. The **Property** column shows the name of the property. The **Min** column shows the lower bound while the **Max** column shows the upper bound. The **Hide** column indicates whether you want to hide events above or below a certain value. The **Limit** column is where you enter this value for the property. For the example shown, events with a Magnitude below 1.5 will be hidden from the display.

Reference Points



If during import you indicate that microseismic event locations in the x and y directions are relative to a reference point, that information is shown here.

Note: You are not required to manually enter a relative location if you load an XML file saved through **Builder** because that information is already stored in the file.

If the microseismic events are not displayed correctly, you can alter the x and y reference points here to better line them up with a particular well. In addition, if the z coordinates are off by a factor of -1, you can **Flip Z location for all microseismic events**.

Note: While x and y coordinates can be relative to a reference point, the z coordinate must correspond to the actual range shown in **Builder** or the microseismic events will not be displayed.

Microseismic Event Data

In the tree view, if you click on All, a well, or a stage, you will see the imported data for every microseismic event. Even if your data is relative to a reference point, all information presented here for x and y location is in absolute or reservoir coordinates.

Note: Your data has not been modified and you still can save the events using relative coordinates when you click on **Export**. Refer to [Exporting Microseismic Data](#) above. You are also shown a summary indicating the minimum and maximum for the x, y and z coordinates.

| Microseismic event data | | | | | | |
|-------------------------|----------|------------|--------|---------|---------|-----------|
| Well | Stage | Date | X | Y | Z | Magnitude |
| W-16 | Stage 01 | 1988-01-01 | 385.00 | 1282.00 | 1927.00 | 1.83 |
| W-16 | Stage 01 | 1988-01-01 | 285.00 | 1270.00 | 1929.00 | 2.65 |
| W-16 | Stage 01 | 1988-01-01 | 390.00 | 1060.00 | 1925.00 | 3.85 |
| W-16 | Stage 01 | 1988-01-01 | 519.00 | 1147.00 | 1921.00 | 2.01 |
| W-16 | Stage 01 | 1988-01-01 | 209.00 | 1013.00 | 1930.00 | 2.45 |
| W-16 | Stage 01 | 1988-01-01 | 396.00 | 1212.00 | 1926.00 | 4.85 |
| W-16 | Stage 01 | 1988-01-01 | 314.00 | 1182.00 | 1928.00 | 2.82 |
| W-16 | Stage 01 | 1988-01-01 | 221.00 | 1165.00 | 1930.00 | 2.87 |
| W-16 | Stage 01 | 1988-01-01 | 273.00 | 1124.00 | 1928.00 | 2.96 |
| W-16 | Stage 01 | 1988-01-01 | 256.00 | 1065.00 | 1929.00 | 3.33 |
| W-16 | Stage 01 | 1988-01-01 | 133.00 | 1141.00 | 1932.00 | 2.14 |
| W-16 | Stage 01 | 1988-01-01 | 186.00 | 1106.00 | 1931.00 | 1.85 |
| W-16 | Stage 01 | 1988-01-01 | 186.00 | 1177.00 | 1931.00 | 2.49 |
| W-16 | Stage 01 | 1988-01-01 | 250.00 | 1205.00 | 1930.00 | 2.87 |
| W-16 | Stage 01 | 1988-01-01 | 180.00 | 1317.00 | 1932.00 | 4.98 |
| W-16 | Stage 01 | 1988-01-01 | 308.00 | 814.00 | 1928.00 | 3.07 |
| W-16 | Stage 01 | 1988-01-01 | 402.00 | 919.00 | 1926.00 | 1.11 |
| W-16 | Stage 01 | 1988-01-01 | 63.00 | 908.00 | 1932.00 | 2.36 |
| W-16 | Stage 01 | 1988-01-01 | 92.00 | 943.00 | 1932.00 | 2.74 |
| W-16 | Stage 01 | 1988-01-01 | 139.00 | 919.00 | 1931.00 | 2.55 |
| W-16 | Stage 01 | 1988-01-01 | 268.00 | 919.00 | 1930.00 | 3.65 |

| Summary | | |
|--|---------|---------|
| | MINIMUM | MAXIMUM |
| Fracture Start Date: | X-Value | 63.00 |
| Fracture End Date: | Y-Value | 294.00 |
| | Z-Value | 1901.40 |
|  Stage Simulation | | 1939.90 |

If you click on a stage, you can see the **Fracture Start Date** (the earliest date) and **Fracture End Date** (the latest date). The **Stage Simulation Date** corresponds to the **Fracture Start Date** but you can change this date (and time) by clicking on the calendar icon. Dates and times for every microseismic event will be changed relative to this new **Stage Simulation Date**. Be sure to **Export** and save this file if you require this new date for future use.

Animation



You can animate chronologically through microseismic events in both 2D and 3D. Which events are animated depends on the selection in the tree view.

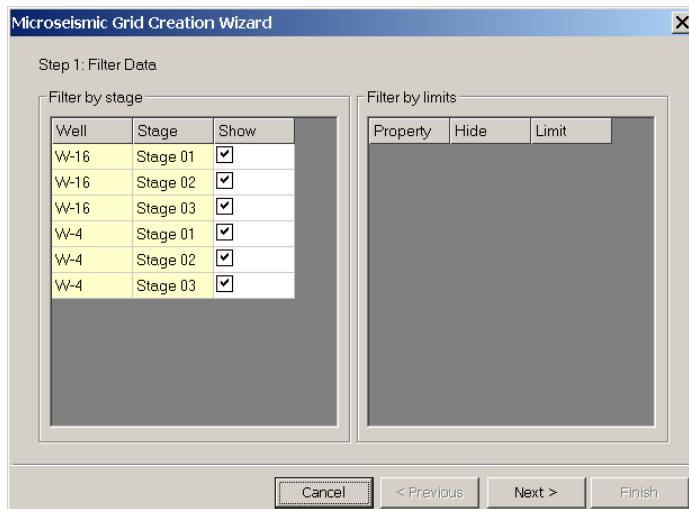
Note: You cannot animate if your events do not have an associated time).

Set the delay time between microseismic events by clicking **Delay**. You can toggle whether to loop repeatedly through the events by clicking on the button. As soon as you click , the **Microseismic Data Manager** will minimize and animation will begin. Stop animation by restoring the data manager and pressing .

Create Grid

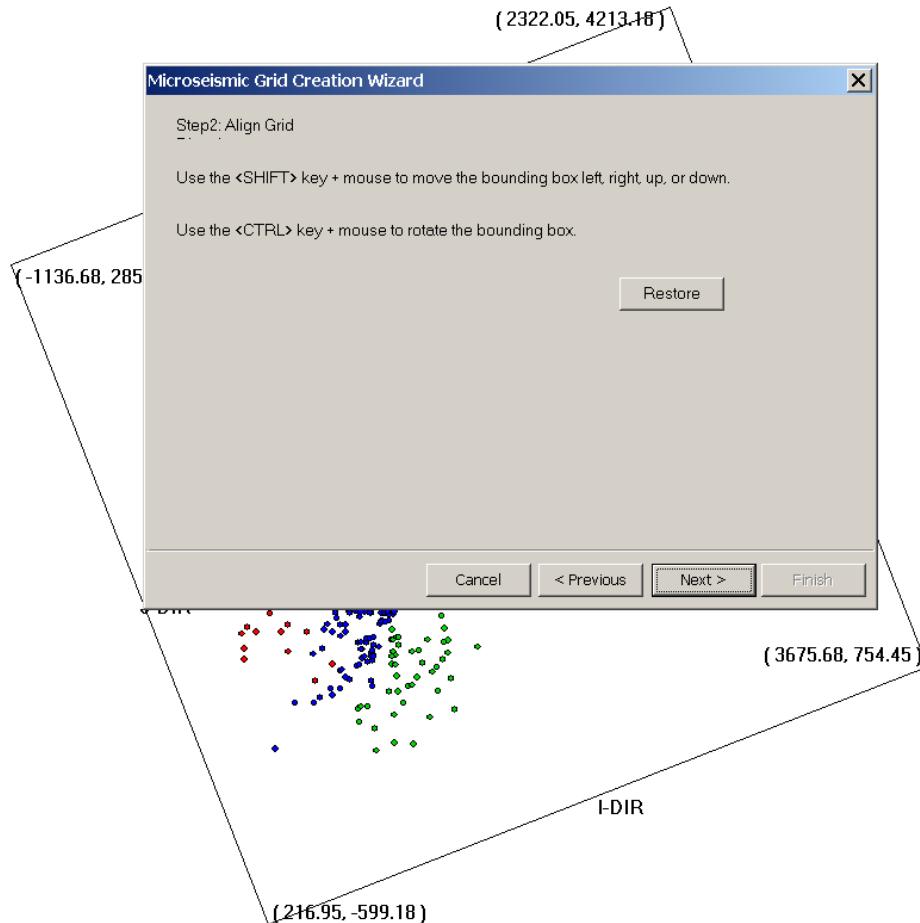
If you import microseismic data into a dataset that lacks a reservoir, you can use the microseismic events to guide you in creation of a grid by clicking on **Create Grid** in the **Microseismic Data Manager** to launch the **Microseismic Grid Creation Wizard**.

Step1. Filter Data



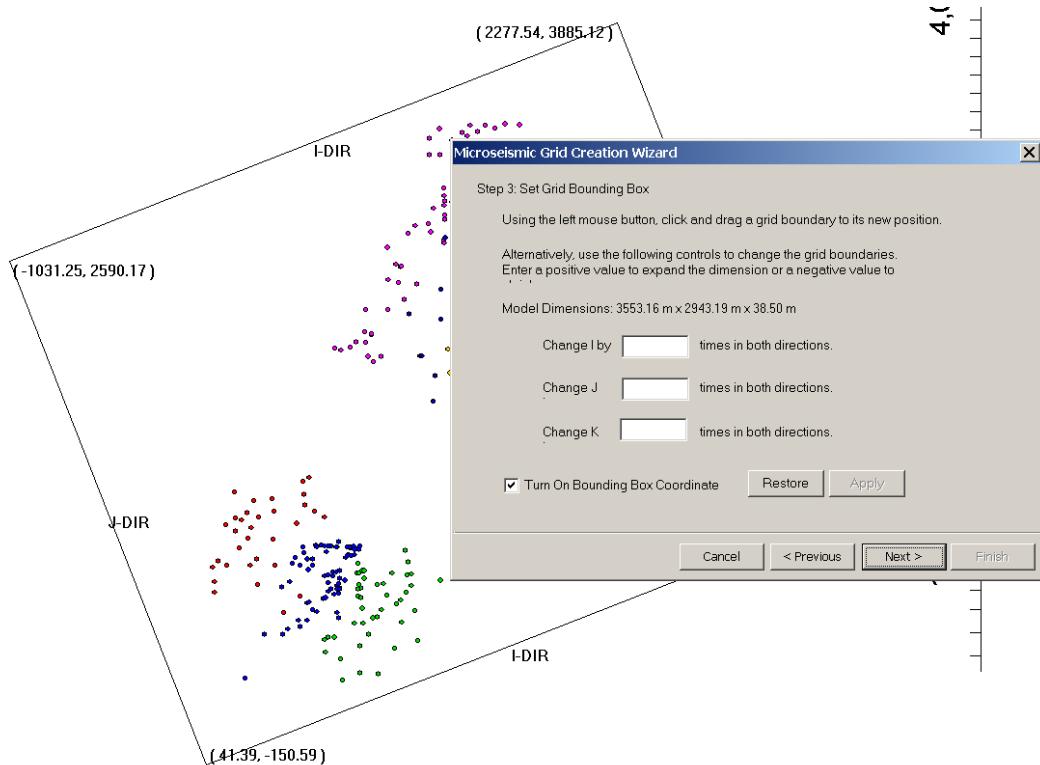
Since grid creation is guided by which microseismic events are displayed, in step one you are presented with another opportunity to filter out microseismic events, similar to what exists in the **Microseismic Data Manager**. You can **Filter by stage**, which will turn off individual stages. You can also **Filter by limits**, based on additional properties that may have been imported. When you are done, click **Next**.

Step 2: Align Grid Direction



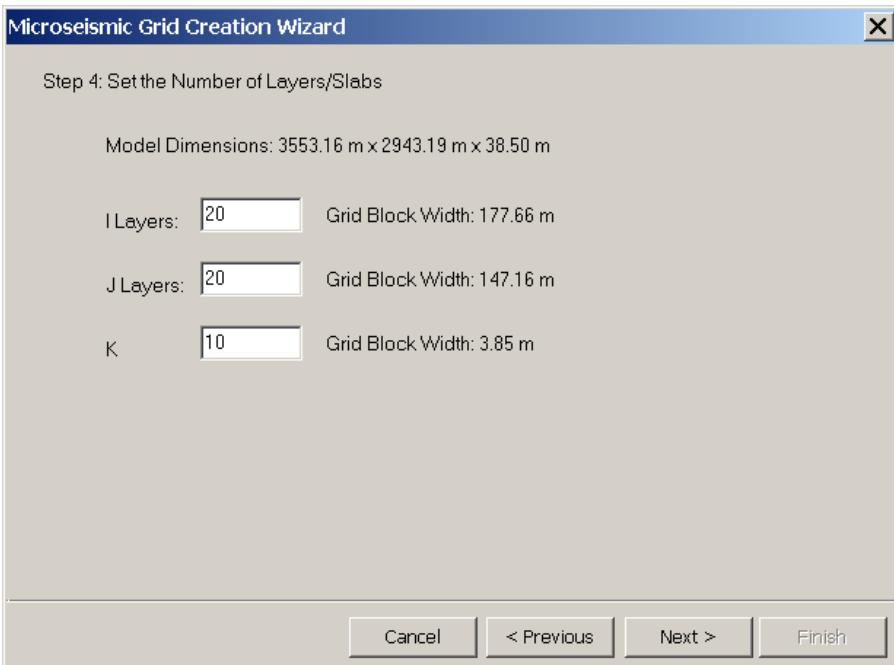
In step two, you can orient the grid as you see fit by either panning or rotating the bounding box as described in the dialog box. The I and J directions, as well as bounding box coordinates, are there to help you. If at any time you are dissatisfied with the changes, click **Restore**. When you are happy with the result, click **Next**.

Step 3: Set Grid Bounding Box

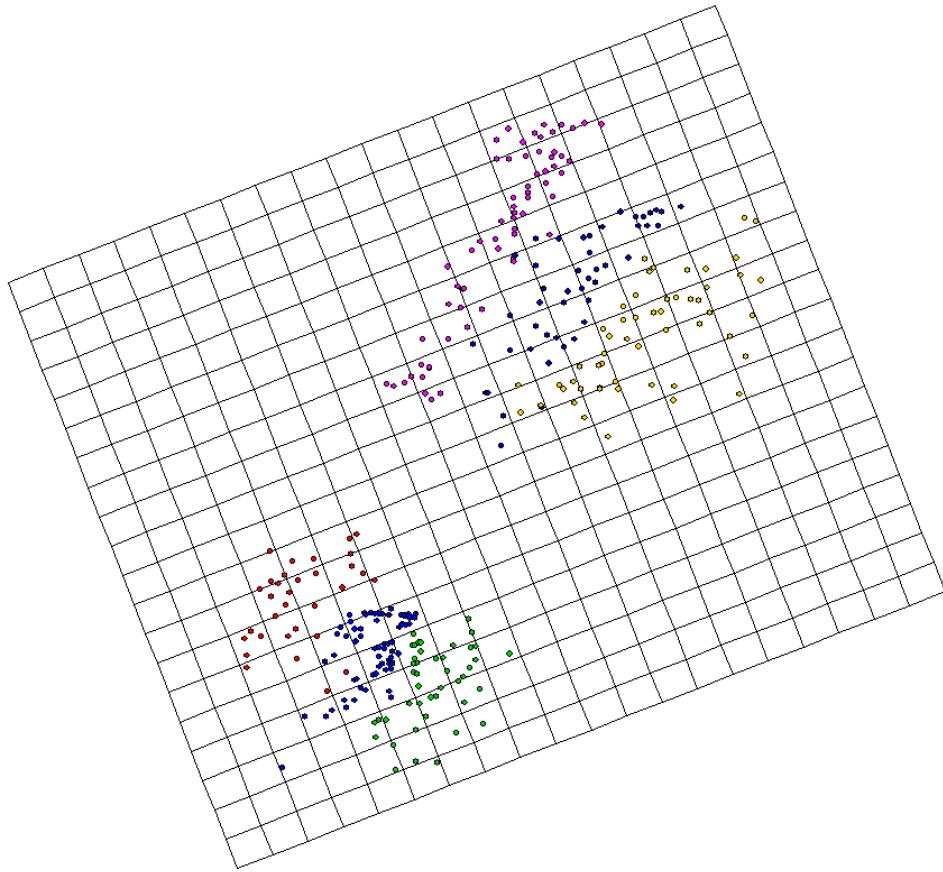


In step three, you can change the dimensions of your model in one of two ways. The first option is to use your mouse and drag a grid boundary to its new position. The second is to use a positive number to expand or a negative number to shrink either the I, J, or K directions. To see the change in the K direction, switch to 3D view. The model dimensions, as well as bounding box coordinates, are there to help you. If at any time you are dissatisfied with the changes, click **Restore**. When you are done, click **Next**.

Step 4: Set Number of Layers/Slabs



Set the number of layers in the I, J, and K directions. Until the number of layers in each direction is defined, the **Next** button will not appear. Once you are satisfied, click **Next** and then **Finish**. You will be returned to the **Microseismic Data Manager** and your grid will be created.



At this point, you can add properties and interact with the grid to develop your reservoir model.

FlexWells (STARS)

About FlexWells

FlexWells allows you to model:

- Complex wellbore behavior which cannot be modeled by a sink/source well. In particular, you can model a collection of up to three tubing strings in an annular space that runs in any direction (vertical, horizontal, slanted or undulating). The annular space can also have multilaterals.
- Annulus with fully or partially cemented casing and varying diameter along its length.
- Tubing strings and instrument tubing strings with various lengths, fully or partially insulated, and varying diameter along their length. Tubing strings can contain concentric tubing.

For additional information refer to the *STARS User's Guide*.

Compared with Sink/Source Well

Refer to the itemized comparison in the Explanation section of *FLX_WELLBORE in the *STARS User's Guide*.

Compared with Discretized Well

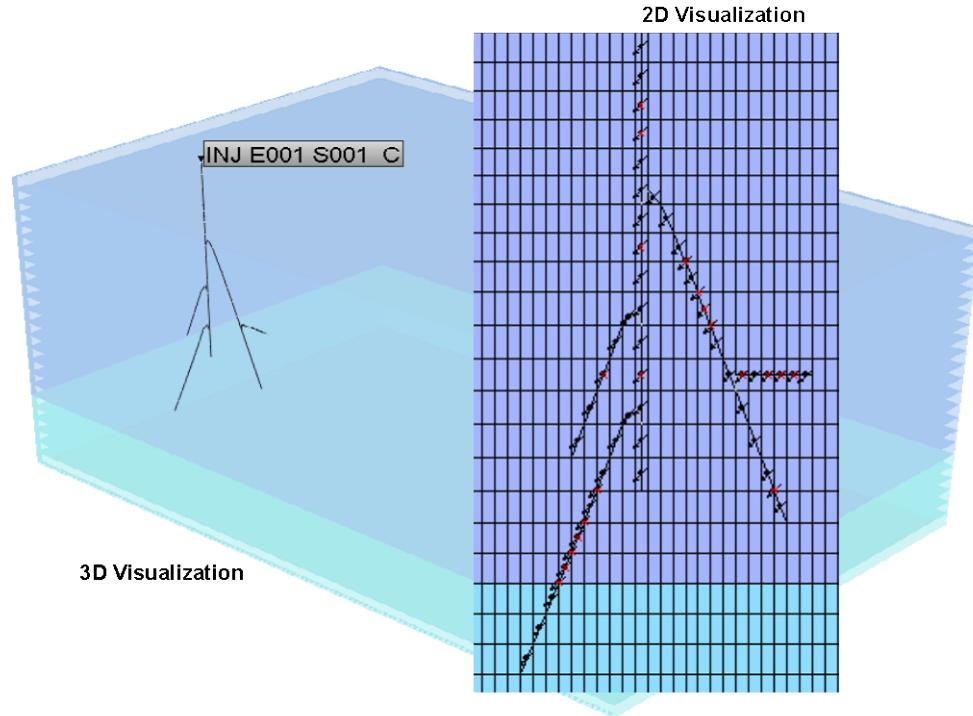
Like a discretized wellbore, a Flexible Wellbore models the accumulation and flow of fluids and heat in the various wellbore volumes, and radial heat conduction accounts for the specified amounts of metal, insulation and cement. However, Flexible Wellbore is not restricted by the principle limits of discretized wellbore:

- Flexible Wellbore has the same powerful trajectory capability as source/sink well, whereas discretized wellbore is restricted to one or two grid columns.
- Flexible Wellbore can account for up to three tubing strings plus one concentric inside an annulus, whereas discretized wellbore is restricted to one tubing string.
- Flexible Wellbore isolates the numerical solution of the coupled well streams from the reservoir grid. In contrast, discretized wellbore cells are fully coupled with the reservoir grid so slow wellbore convergence results in additional full-grid iterations.

Benefits of Using Builder

Builder provides an easy-to-use interface for creating, configuring, viewing, modifying, copying, and deleting FlexWells. Benefits of using Builder to manage FlexWells include:

- **Time Saving:** Builder supports easy entry of complex well annulus and tubing geometries.
- **Visualization:** Builder allows you to view two- and three-dimensional representations of the FlexWell, including the main wellbore, multilaterals and perforations, for example:



- **Well Assignment:** Builder provides a simple interface for assigning existing or new sink/source wells to a FlexWell annulus or tubing string.
- **Validation:** As FlexWell data is entered, Builder validates it. If rules are broken, Builder presents error messages and suggestions for correcting the error. Key validation checks are outlined in [Data Validation Checks](#).
- **Branches:** Using Builder, it is straightforward to change the annulus branch to which a tubing string is assigned, or to assign a new well to an annulus branch. It is easy to extend branches by adding perforations, or to shorten branches by removing perforations from the end.
- **Events Table:** Builder provides a chronologically organized table of FlexWell and well events, which can be used, for example, as a scheduling tool. An example is shown below. Through the table, you can manage the FlexWell, such as changing a well's status from SHUTIN to OPEN. The table cells and text are colour-coded to clearly indicate status and errors.

| Event Date | FlexWell Event Type | Annulus Well Status "INJ E001 5001 C" | Tubing Well Status "Well-3" |
|---------------|---------------------|--|--------------------------------|
| 1980-03-18 50 | Definition/ACTIVATE | (SHUTIN) | OPEN* |
| 1980-03-19 50 | DE-ACTIVATE | | SHUTIN |
| 1980-03-20 | ACTIVATE | | |
| 1980-03-20 50 | | | |
| 1980-03-20 50 | REPLACE | | |
| 1980-03-21 50 | | | |
| 1980-03-21 50 | | | |
| 1980-03-21 50 | REPLACE | | |
| 1980-03-21 50 | | | |

Through the **Events** table, you can shift the dates of all events on, or after an event, by a fixed amount. You can easily replace one well with another; for example, you can replace an injection well with a production well or a production well with an injection well to model an injection/production cycle. You do not need to define a separate FlexWell for this purpose.

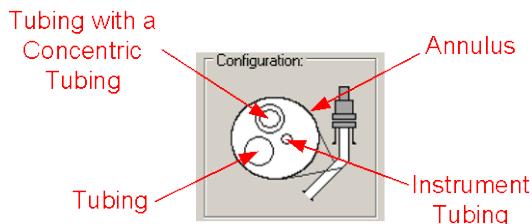
You can take a snapshot of the **Events** table for one FlexWell for visual comparison with the **Events** table of another FlexWell.

FlexWell Configurations

A FlexWell can be configured with a variety of annulus/tubing string combinations:

- One annulus
- Up to three tubing strings
- Up to one instrument tubing
- Up to one concentric tubing string

The following shows the cross-section of a FlexWell with one annulus, two tubing strings, one concentric tubing string and one instrument tubing string:



FlexWell Status

The status of a FlexWell is either of the following:

ACTIVATED Wells attached to the FlexWell can have flow.

DE-ACTIVATED Wells attached to the FlexWell cannot have flow, unless the well is also assigned to another FlexWell that is ACTIVATED

The status of a sink/source well that has been assigned to a FlexWell is one of the following:

SHUTIN Well is not open to flow.

OPEN Well is open to flow.

The status of a perforation is one of the following:

OPEN Perforation is open to flow.

CLOSED Perforation is closed to flow.

AUTO Perforation is open or closed to flow based on the status of a user-defined condition. Refer to the *STARS User's Guide* for further information.

Process

The process for defining a FlexWell is summarized as follows:

1. Create the FlexWell.
2. Configure the FlexWell (including name and definition date).
3. Assign an existing or new well to the FlexWell annulus.
4. Configure the annulus (casing properties and perforations, for example).
5. Add tubing strings, as required.
6. For each tubing string, assign an existing or new well.
7. Configure the tubing strings (annulus branch, tubing diameter and tubing length, for example).
8. Add and configure concentric and instrument tubing strings as necessary.
9. Manage the FlexWell through the FlexWell **Events** table.

Data Validation Checks

As you record and modify FlexWell data, Builder carries out validation checks, some of which are shown below. When configuration rules are broken, messages are presented to the user explaining the problem and suggesting corrections.

- **FlexWell Name**: The name of the FlexWell must be unique.
- **Well Selection**: When you are selecting a well to be assigned to the annulus or to a tubing string, you can filter the wells so that only wells meeting the validation criteria will be displayed for selection.
- **Alignment**: A tubing string must align with one and only one annulus completion branch; that is, it can only have one path to the surface.

- **Definition Dates:** Sink/source well definition dates must occur on or before the FlexWell definition date. Wells must be SHUTIN before the FlexWell definition date unless they are assigned to another FlexWell that is active on the same date.
 - **Perforations:** The annulus must have at least two perforations. If two perforations are not specified, you will need to select another well or add the required perforations. Perforations must use LAYERXYZ geometry with a well index type GEO or GEOA. Builder can be directed to modify the annulus and tubing string wells so that they use LAYERXYZ geometry.
- If **Auto-complete PERFS** is checked, then perforations must be continuous; that is, all connecting grid blocks between the starting and ending perforations must be perforated. Builder can be directed to complete the perforations in all grid blocks. When missing perforations are added, Builder sets their status to CLOSED. If a grid block perforation is OPEN then that grid block will be included in both material and heat flow calculations for the annulus and tubing string well. If the grid block perforation is CLOSED, then that grid block will only contribute to heat flow calculations for the annulus and tubing string well.
- **Tubing:** Tubing wells cannot extend beyond the end of the annulus branch they occupy.
 - **Concentric Tubing:** Concentric tubing must align with its outer tubing. Concentric tubing can extend beyond its outer tubing, but not beyond the annulus branch it occupies.
 - **Well Diameter:** The tubing diameters must be less than the diameters of the annulus branch in which they reside. ID must be less than OD, and OD must be less than the insulation/cement diameter.

For more detailed information about data validation checks, refer to [FlexWells Validation Errors](#) on page 427.

Note: To view validation errors at any time, right-click **FlexWells** or **FlexWell-n** in the Builder tree view, and then select **Validate**. A list of validation errors will be displayed. Newly entered or modified data will only be validated after you have applied the changes.

This section has provided an overview of FlexWells. The next sections will provide an overview of the main **FlexWells** dialog box, and detailed procedures for managing FlexWells.

Managing the Annulus

To open the **FlexWells** dialog box

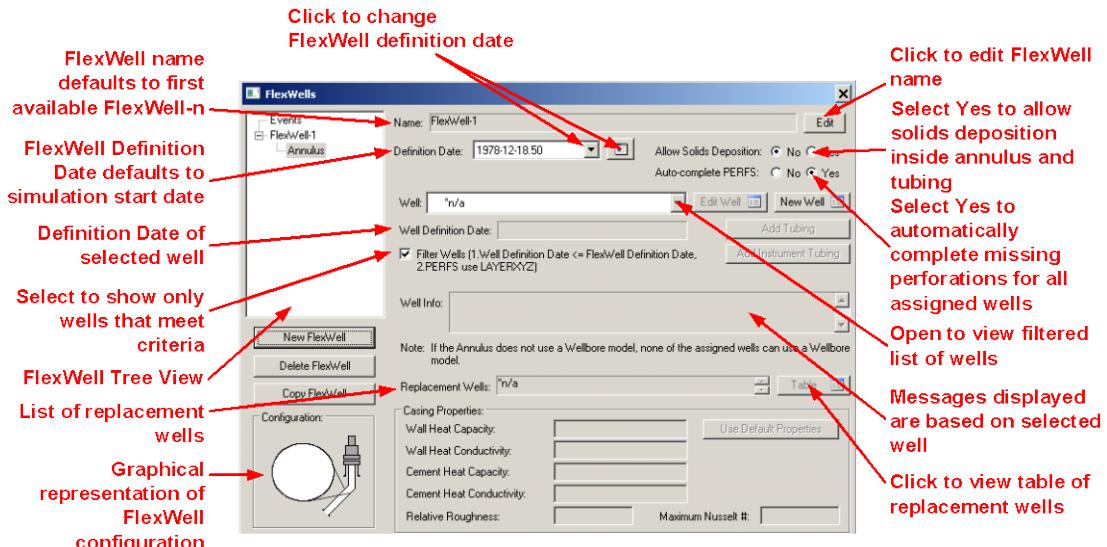
To open the **FlexWells** dialog box:

- Click **Well | FlexWells** in the Builder menu bar or
- Double-click the FlexWells entry in the **Wells & Recurrent** section of the **Model Tree View**.

The **FlexWells** dialog box is displayed.

To create a new FlexWell

1. In the **FlexWells** dialog box, click the **New FlexWell** button. The **FlexWells** dialog box will be displayed as follows:



As shown above, Builder activates a new FlexWell, and assigns it the following:

- Annulus
 - FlexWell definition date, defaulted to the simulation start date.
 - Generic “FlexWell- #” name, starting from “FlexWell-1” and increasing sequentially.
2. To change the name of the FlexWell, click **Edit** beside the **Name** field, enter the desired name in the **Input Name** dialog box, and then click **OK**.
 3. When a new FlexWell is created, you can modify the FlexWell definition date in one of two ways:
 - By selecting a new date from the list of available dates displayed inside the **Definition Date** box.
 - By creating a new date by clicking the button and entering the new date through the **CMG Calendar** dialog box.

Note: If you have assigned an annulus well and some tubing wells to a FlexWell, and then decide to modify the FlexWell definition date, Builder will remove the annulus and tubing wells if their definition dates occur after the new FlexWell definition date.

4. You can choose to allow solids deposition inside the annulus and tubing by selecting one of the options. Solids can be deposited by reaction and if allowed, this will reduce the hydraulic diameter of the annulus and tubing over time.
5. You can also select the option to direct Builder to complete missing perforations for all wells that are assigned to the FlexWell.

Note: At any time, you can click **Apply** to apply the changes without closing the **FlexWells** dialog box. Click **OK** to both apply the changes and close the dialog box.

To assign an existing sink/source well to the annulus

When a new FlexWell is created it is automatically activated and assigned an annulus. You will need to assign a sink/source well to the annulus, as outlined in the following procedure. The procedure assumes no validation errors. Validation errors are discussed in [FlexWells Validation Errors](#) on page 427.

1. Select a sink/source well from the list that Builder has automatically compiled and displayed in the **Annulus | Well** box. This list will only contain sink/source wells that meet the following criteria:
 - They are not already assigned to this FlexWell.
 - Their definition date is the same as or earlier than the FlexWell definition date.
 - They use LAYERXYZ geometric data with a well index type of GEO or GEOF for their completion data.
 - They contain at least two perforations. Two perforations are the minimum required to determine the well's direction.

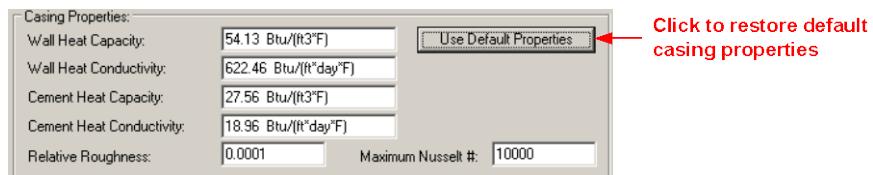
Notes:

- a. To view validation errors at any time, right-click **FlexWells** or **FlexWell-n** in the Builder tree view, and then select **Validate**. A list of validation errors will be displayed. Newly entered or modified data will only be validated after you have applied the changes.
 - b. If you clear the **Annulus | Filter Wells** check box, all sink/source wells from the dataset will appear in the **Annulus | Well** box.
 2. If the annulus well perforations are not continuous, and **Auto-complete PERFS** is set to **Yes**, Builder will ask you if it can modify the annulus well to fill in the missing perforations.
- If you click **Yes**, the well will be assigned to the annulus and Builder will auto-complete the PERFS. If you click **No**, the well will be assigned to the annulus with incomplete perforations, and **Auto-complete PERFS** will be changed to **No** for all subsequent FlexWell operations.

If **Auto-complete PERFS** was already set to **No**, the well will be assigned to the annulus and **Auto-complete PERFS** will not be changed.

When Builder adds the missing perforations, it sets their status to CLOSED. If a grid block perforation is OPEN, then that grid block will be included in both material and heat flow calculations for the annulus well. If the grid block perforation is CLOSED, then that grid block will only contribute to the heat flow calculations for the annulus well.

3. Configure the **Casing Properties**, as necessary.

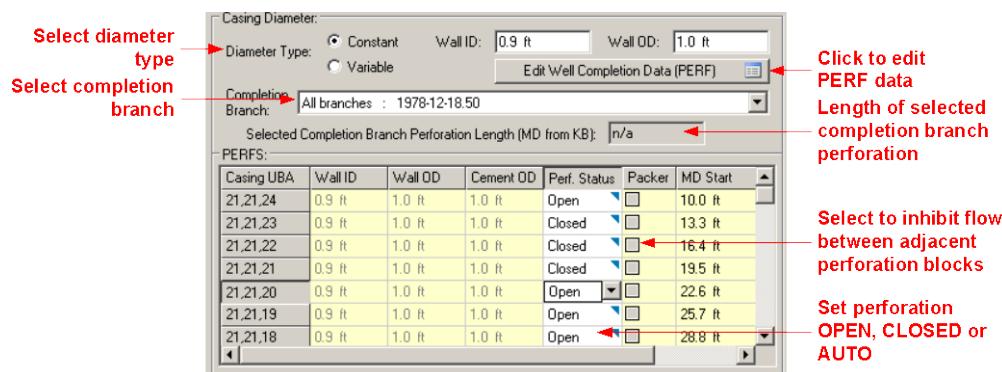


The **Casing Properties** area will initially contain default values. These default values are defined by the FLX_WELLBORE keyword.

Click **Use Default Properties** to restore the default casing properties, which are shown above.

4. You can modify the annulus well perforations at any time by clicking the **Edit Well Completion Data (PERF)** button in the **Casing Diameter** area. Each time the annulus well perforations are modified through the **Well Completion Data (PERF)** dialog box, Builder checks the annulus well to make sure it contains at least two perforations using LAYERXYZ geometric data and, if **Auto-complete PERFS** is set to **Yes**, that all perforations are continuous. After the annulus well perforations are modified, Builder automatically re-calculates the annulus completion branches and determines if the existing tubing wells still align. If tubing wells no longer align they are removed and a message displays stating this.

5. Configure the **Casing Diameter** settings.



Note: If **Variable** diameter type is selected, data can be entered in the **Wall ID**, **Wall OD**, **Cement OD**, and **Packer** columns.

When an annulus well is assigned to the FlexWell, Builder automatically constructs completion branches from the annulus well completion data. These are selectable through the **Completion Branch** list. The number of completion branches in the list depends on the number of multi-laterals in the selected annulus well. There will be one completion branch for each multi-lateral, and one completion branch for the main, or parent branch.

6. Select a **Constant** or **Variable** diameter type. If you select the **Constant** diameter type, all completion branches will have the same diameter. If you select the **Variable** diameter type, all Casing UBA diameters can have different values. The default selection is **Constant**.

For a **Constant** casing diameter type, the minimum requirement is that the **Wall ID** must be specified. The default for the annulus **Wall ID** is 1 ft. Builder will automatically assign the **Wall OD** and **Cement OD** equal to the **Wall ID**. You will be given a warning if you do not change this. The annulus **PERFS** table is disabled, except for the **Perf. Status** column. Through this column, you can modify the status of any annulus perforation.

Note: When tubing is assigned to a constant-diameter annulus, Builder checks to make sure that the combined tubing **Insulation Wall ODs** do not exceed the annulus **Wall ID**.

For a **Variable** casing diameter type, the minimum requirement is that each **Casing UBA** row specifies a **Wall ID**, **Wall OD** and **Cement OD**. The default value for the annulus **Wall ID** is 1 ft. Builder automatically sets the **Wall OD** equal to **Wall ID**, and the **Cement OD** equal to the **Wall OD**.

Note: When tubing is assigned to a variable-diameter annulus, Builder checks to make sure that the combined tubing **Insulation ODs** do not exceed the annulus **Wall ID** at each **Casing UBA** perforation block.

7. To edit the **PERFS** table for a particular completion branch, select it above the **Annulus | PERFS** table, as shown in the following example:

| Casing UBA | Wall ID | Wall OD | Cement OD | Perf. Status | Packer | MD Start |
|------------|---------|---------|-----------|--------------|-------------------------------------|----------|
| 21.21.24 | 0.9 ft | 1.0 ft | 1.0 ft | Open | <input checked="" type="checkbox"/> | 10.0 ft |
| 21.21.23 | 0.9 ft | 1.0 ft | 1.0 ft | Closed | <input checked="" type="checkbox"/> | 13.3 ft |
| 21.21.22 | 0.9 ft | 1.0 ft | 1.0 ft | Closed | <input checked="" type="checkbox"/> | 16.4 ft |
| 21.21.21 | 0.9 ft | 1.0 ft | 1.0 ft | Closed | <input checked="" type="checkbox"/> | 19.5 ft |
| 21.21.20 | 0.9 ft | 1.0 ft | 1.0 ft | Open | <input checked="" type="checkbox"/> | 22.6 ft |
| 21.21.19 | 0.9 ft | 1.0 ft | 1.0 ft | Open | <input checked="" type="checkbox"/> | 25.7 ft |
| 22.21.19 | 0.9 ft | 1.0 ft | 1.0 ft | Open | <input checked="" type="checkbox"/> | 32.96 ft |

You can display all perforations for all branches (the default), or only those perforations for a specific completion branch, by making a selection from the **Annulus | Completion Branch** box. No completion branch perforation length will be displayed if you select **All branches** in the **Completion Branch** box.

Edit the PERFS table for the selected completion branch as necessary.

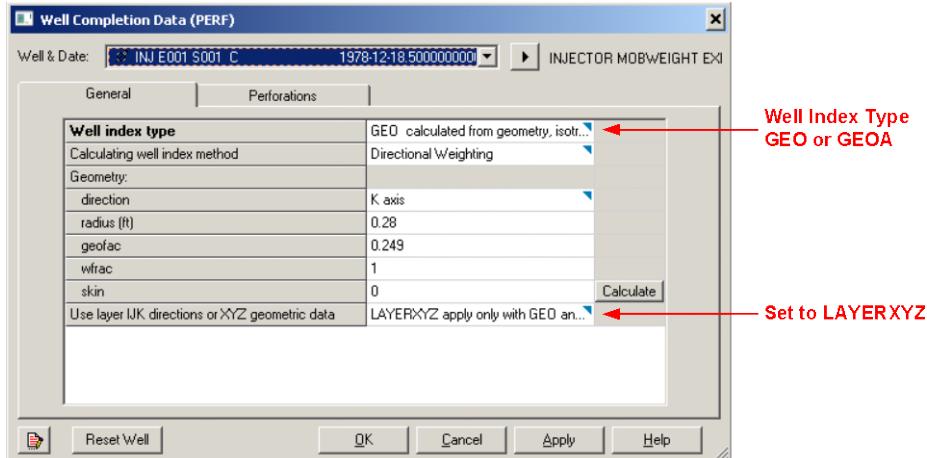
Note: Each time you change the **Completion Branch** selection, the list of associated perforations will change inside the **PERFS** table. Each time you click a **Casing UBA** row in the **PERFS** table, the corresponding perforation will be highlighted in the reservoir view.

8. For a **Variable** casing diameter type, you can inhibit lateral flow between adjacent perforation blocks by checking the associated **Packer** box in the **PERFS** table.
9. Click **Apply** to apply the changes to the dataset without closing the FlexWells dialog box. Click **OK** to both apply the changes and close the dialog box.

To modify perforations

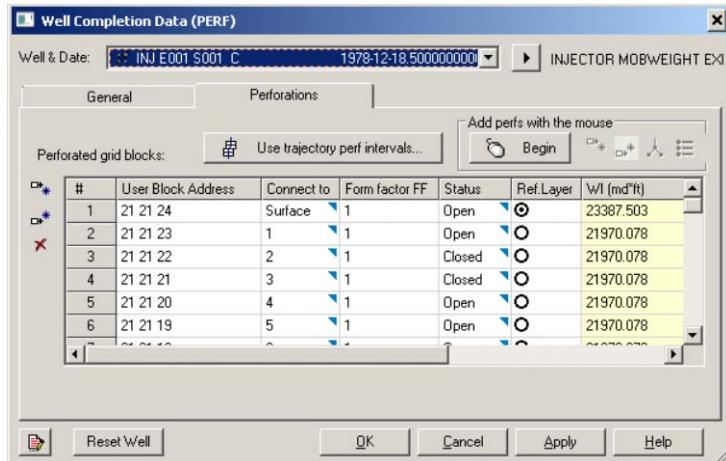
You can use the annulus branch's **Edit Well Completion Data (PERF)** button to modify the perforations of the assigned sink/source well:

1. Click the **Edit Well Completion Data (PERF)** button on the **Annulus** branch. The **Well Completion Data (PERF)** dialog box will be displayed, with the **General** tab selected:



Note: The **General** tab has been modified so that the **Well index type** box only contains GEO and GEOA, and the **Use layer IJK or XYZ geometric data** box only contains LAYERXYZ. Builder requires LAYERXYZ data to calculate the annulus completion branches and the MD of each perforation.

2. Select the **Perforations** tab:



Notes:

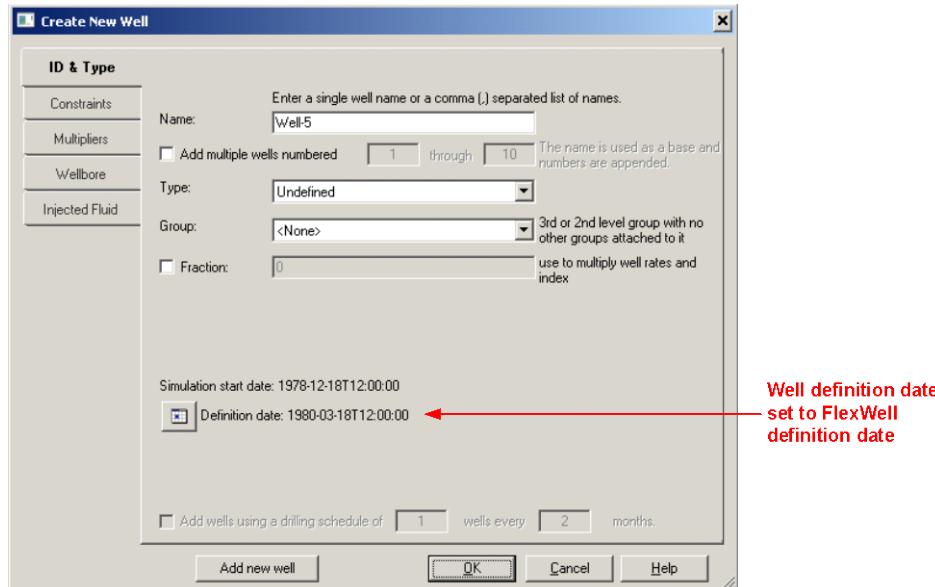
- a. The **Perforations** tab can be modified if viewed using the **FlexWells** dialog box; however, if the **Well Completion Data (PERF)** dialog box is displayed without using the **Edit Well Completion Data (PERF)** button, the **Perforations** tab will be disabled. This is done to enforce the alignment relationship between the annulus and its tubing wells.
- b. If you have assigned tubing inside the annulus, and then modify the annulus perforations, Builder may remove the tubing wells if the modified annulus completion branches no longer align with the tubing completion branches.
3. Edit the perforation nodes as necessary, and then click **OK**. Builder checks that the completions are valid, as per the validation errors.

To assign a new sink/source well to an annulus

1. To assign a new sink/source well to an annulus, click the **New Well** button:



The **Create New Well** dialog box will be displayed, with the new well **Definition date** set to the FlexWell definition date, as shown below:



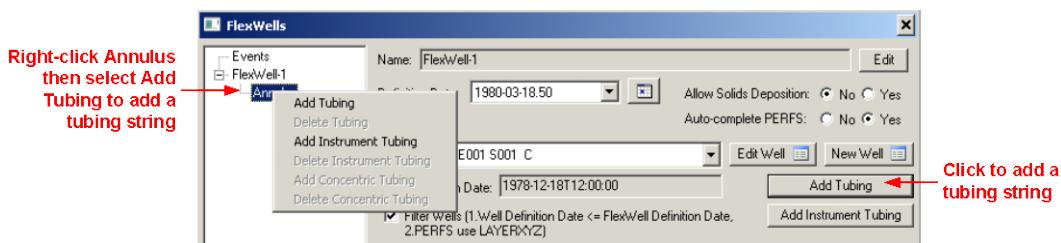
2. If necessary, click to modify the new well **Definition date**. Note the following:
 - a. If you modify the **Definition date** on the **Create New Well** dialog box so that it occurs *after* the FlexWell **Definition Date**, and then click **OK**, a message will be displayed in the annulus **Well Info** panel, informing you that the well cannot be used, and to select another well.
 - b. If you modify the **Definition date** on the **Create New Well** dialog box so that it occurs *before* the FlexWell **Definition Date**, and then click **OK**, because the new well is automatically OPEN when it is created, the well status will be displayed in bold red in the **Events** table. For further information about the **Events** table, refer to [Managing FlexWell and Well Events](#).
 - c. If you do not modify the **Definition date** on the **Create New Well** dialog box, and then click **OK**, a message will be displayed in the **Annulus | Well Info** panel informing you that tubing cannot be assigned until annulus well perforations have been added.
3. Once you have made the necessary changes click **OK** to return to the **FlexWells** dialog box.
4. The **Casing Properties** area of the **Annulus** page will initially contain default values. These initial values are defined by the **FLX_WELLBORE** keyword. Edit these values as necessary.

5. You can add new well perforations by clicking the **Edit Well Completion Data (PERF)** button. Each time the annulus well perforations are modified using the **Well Completion Data (PERF)** dialog box, Builder checks that completions are valid as outlined in [FlexWells Validation Errors](#) on page 427.
6. Click **Apply** to save your edits without closing the **FlexWells** dialog box. Click **OK** to apply the changes and close the **FlexWells** dialog box.

Managing Tubing Strings

To add a tubing string to an annulus

To add a tubing string to the annulus click the **Add Tubing** button or right-click **Annulus** on the **FlexWells** dialog box tree view to open the context menu, and then select **Add Tubing**.



To assign an existing sink/source well to a tubing string

After the tubing string is added, Builder will compile a list of valid sink/source wells and display it in the **Tubing | Well** box. You will need to assign a sink/source well to the tubing, as outlined below. The procedure assumes no validation errors. Validation errors are discussed in [FlexWells Validation Errors](#) on page 427.

1. Select the sink/source well from the list that Builder has automatically compiled and displayed in the **Annulus | Well** box. This list will only contain sink/source wells that meet the following criteria:
 - They are not already assigned to this FlexWell.
 - Their definition date is the same as or earlier than the FlexWell definition date.
 - They use LAYERXYZ geometric data with a well index type of GEO or GEOA for their completion data.
 - They contain at least two perforations.
 - Their perforation paths align with one, and only one, of the annulus completions branches.

Notes:

- a. To view validation errors at any time, right-click **FlexWells** or **FlexWell-n** in the Builder tree view and then select **Validate**. A list of validation errors will be displayed. Newly entered or modified data will only be validated after you have applied the changes.
 - b. If you clear the **Tubing | Filter Wells** check box, all sink/source wells from the dataset will appear in the **Tubing | Well** box.
 - c. If you clear **Filter Wells** and then try to assign a well with multiple completion branches or one that is not aligned with an annulus completion branch, Builder will take you through the steps of configuring the well so that it is aligned properly.
2. If the selected sink/source well *does* contain at least two perforations, and if **Auto-complete PERFS** is set to **Yes**, Builder checks to make sure that these perforations are continuous, and that they align with one, and only one annulus completion branch. Perforations are continuous when all grid blocks between the starting and ending perforation are perforated.

Note: Unlike the annulus well, you cannot change the tubing well completion data through the **Well Completions Data (PERF)** dialog box. Access to the tubing well's **Well Completions Data (PERF) | Perforations** tab is blocked, and you can only select a well index type of GEO or GEOA on the tubing well's **Well Completions Data (PERF) | General** tab. This ensures the tubing well will stay aligned and always use LAYERXYZ geometric data.

3. As necessary, modify the **Tubing Properties** and **Tubing Diameter** data as was done for the annulus (refer to [To assign an existing sink/source well to the annulus](#)). The **Tubing | PERFS** table is not editable unless **Diameter Type** is set to **Variable**. There are differences between the **Annulus | PERFS** table and the **Tubing | PERFS** table. The **Tubing | PERFS** table has an **Insulation OD** column instead of a **Cement OD** column. Since flow into and out of the tubing occurs only at the end of the tubing string, there is no **Perf. Status** column. The table includes a column for selecting a **WP** (withdrawal point), but this column is disabled for injector wells.

Note: A producer well cannot have a withdrawal point at the toe of the tubing string.

The following screen shows the **Annulus | PERFS** table with the **Diameter Type** set to **Variable**. As shown, in this case, the **PERFS** table includes **Cement OD** and **Perf. Status** columns:

The screenshot shows the 'Casing Diameter' configuration for an annulus completion. The 'Diameter Type' is set to 'Variable'. The 'Completion Branch' is listed as 'All branches : 1978-12-18.50'. The 'Selected Completion Branch Perforation Length (MD from KB)' is 'n/a'. The 'PERFS' table lists seven completion branches (21.21.24 to 21.21.18) with the following data:

| Casing UBA | Wall ID | Wall OD | Cement OD | Perf. Status | Packer | MD Start |
|------------|---------|---------|-----------|--------------|--------------------------|----------|
| 21.21.24 | 1.0 ft | 1.0 ft | 1.0 ft | Open | <input type="checkbox"/> | 10.0 ft |
| 21.21.23 | 1.0 ft | 1.0 ft | 1.0 ft | Open | <input type="checkbox"/> | 13.3 ft |
| 21.21.22 | 1.0 ft | 1.0 ft | 1.0 ft | Closed | <input type="checkbox"/> | 16.4 ft |
| 21.21.21 | 1.0 ft | 1.0 ft | 1.0 ft | Closed | <input type="checkbox"/> | 19.5 ft |
| 21.21.20 | 1.0 ft | 1.0 ft | 1.0 ft | Open | <input type="checkbox"/> | 22.6 ft |
| 21.21.19 | 1.0 ft | 1.0 ft | 1.0 ft | Open | <input type="checkbox"/> | 25.7 ft |
| 21.21.18 | 1.0 ft | 1.0 ft | 1.0 ft | Open | <input type="checkbox"/> | 28.8 ft |

Annotations on the right side of the screenshot explain the configuration:

- 'Producer well assigned, Diameter Type set to Variable'
- 'Perf. Status column'
- 'Cement OD column'

The following screen shows a **Tubing | PERFS** table with the **Diameter Type** set to **Variable**. As shown, in this case, the **PERFS** table does not include a **Cement OD** or **Perf. Status** column, but does include **Insulation OD** and **WP** columns:

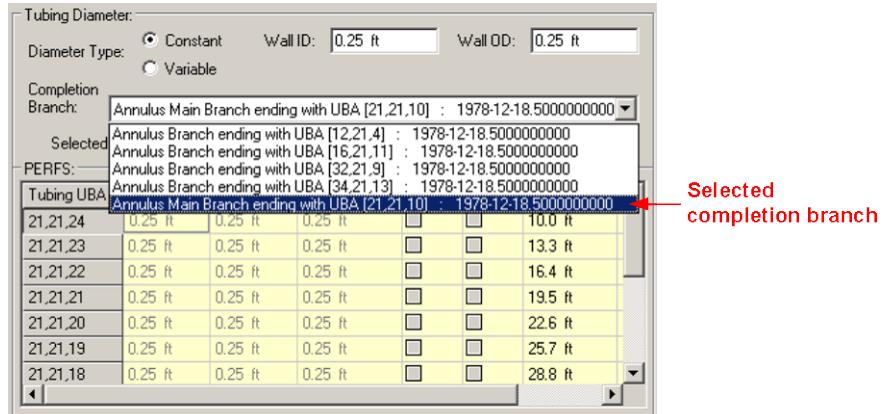
The screenshot shows the 'Tubing Diameter' configuration for a tubing completion. The 'Diameter Type' is set to 'Variable'. The 'Completion Branch' is listed as 'Annulus Main Branch ending with UBA [21.21.10] : 1979-02-18.50'. The 'Selected Completion Branch Perforation Length (MD from KB)' is '60.0 ft'. The 'Edit' button is visible. The 'PERFS' table lists seven completion branches (21.21.24 to 21.21.18) with the following data:

| Tubing UBA | Wall ID | Wall OD | Insulation OD | Packer | WP | MD Start |
|------------|---------|---------|---------------|--------------------------|--------------------------|----------|
| 21.21.24 | 0.25 ft | 0.25 ft | 0.25 ft | <input type="checkbox"/> | <input type="checkbox"/> | 10.0 ft |
| 21.21.23 | 0.25 ft | 0.25 ft | 0.25 ft | <input type="checkbox"/> | <input type="checkbox"/> | 13.3 ft |
| 21.21.22 | 0.25 ft | 0.25 ft | 0.25 ft | <input type="checkbox"/> | <input type="checkbox"/> | 16.4 ft |
| 21.21.21 | 0.25 ft | 0.25 ft | 0.25 ft | <input type="checkbox"/> | <input type="checkbox"/> | 19.5 ft |
| 21.21.20 | 0.25 ft | 0.25 ft | 0.25 ft | <input type="checkbox"/> | <input type="checkbox"/> | 22.6 ft |
| 21.21.19 | 0.25 ft | 0.25 ft | 0.25 ft | <input type="checkbox"/> | <input type="checkbox"/> | 25.7 ft |
| 21.21.18 | 0.25 ft | 0.25 ft | 0.25 ft | <input type="checkbox"/> | <input type="checkbox"/> | 28.8 ft |

Annotations on the right side of the screenshot explain the configuration:

- 'Producer well assigned, Diameter Type set to Variable'
- 'WP (Withdrawal Point) column'
- 'Insulation OD column'

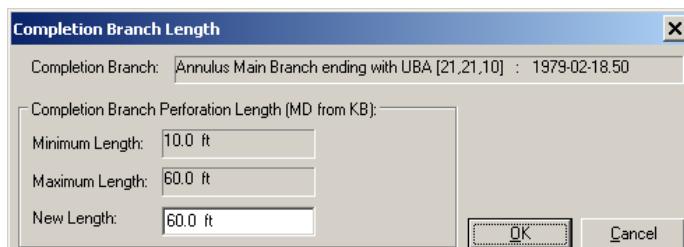
4. You can align the tubing well with a different annulus completion branch at any time by selecting another branch from the **Tubing | Completion Branch** box:



When a new annulus completion branch is selected for the tubing well, all existing tubing well perforations are deleted, and replaced with copies from the selected annulus completion branch. The **Tubing | Selected Completion Branch Perforation Length (MD from KB)** value and the **PERFS** table values will be updated with the new completion branch data.

Note: If the tubing string contains a concentric tubing string, changing the tubing completion branch may cause its concentric tubing to no longer align; as a result, Builder may remove the concentric tubing well.

After the new annulus perforations have been added to the tubing well, you can still modify the length of the tubing completion branch while the **Edit** button inside the **Tubing Diameter** area is enabled. This will display the **Completion Branch Length** dialog box:

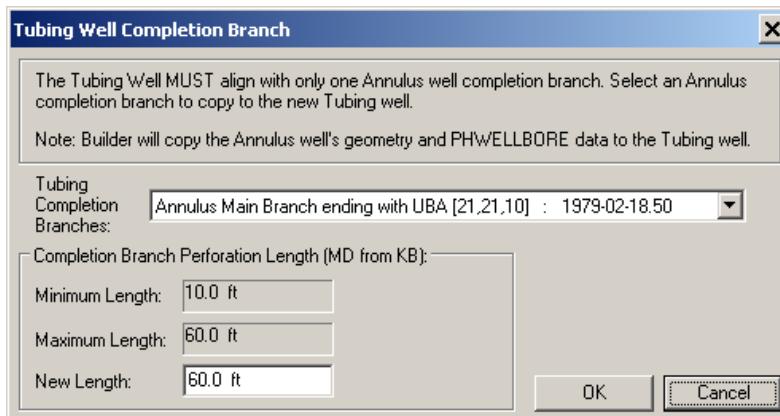


5. Click **Apply** to save your edits without closing the **FlexWells** dialog box. Click **OK** to apply the changes and close the **FlexWells** dialog box.

To assign a new sink/source well to a tubing string

To assign a new sink/source well to a tubing string:

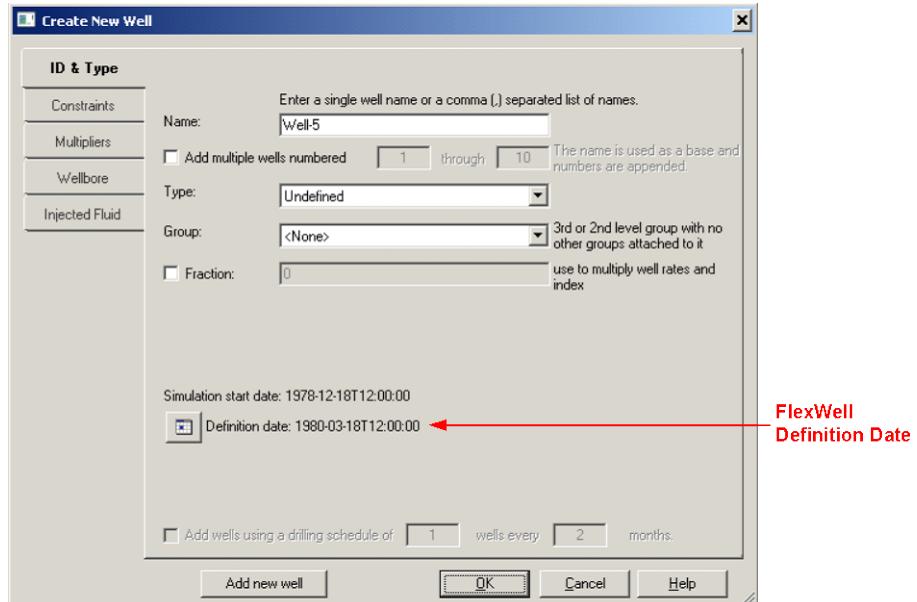
1. Click the **New Well** button in the **FlexWells** dialog box. The **Tubing Well Completion Branch** dialog box will be displayed:



Since the tubing string must align with one, and only one, of the annulus completion branches, copying one of the annulus completion branches to the new tubing well guarantees that the tubing and annulus perforations will align.

2. Select the annulus completion branch in which the tubing well will reside. Most tubing strings only allow flow from the end of the tube, so all of the copied annulus completion branch perforations will have a CLOSED perforation status except for the ending perforation, which will be OPEN. The dialog box also allows you to specify the tubing completion branch perforation length.
3. Specify the tubing completion branch length. The tubing well completion branch cannot extend beyond the ends of the annulus completion branch in which it resides; therefore, the length of the tubing well completion branch must be less than or equal to the length of the annulus completion branch.

If you select **OK**, the **Create New Well** dialog box will be displayed, allowing you to define the new tubing well:



As shown above, Builder automatically assigns the FlexWell **Definition Date** to the **Create New Well | Definition date**.

If you click **Cancel**, the new tubing well will not be created and the **Tubing | Well** box will revert back to the previous selection.

4. Configure the new well as required and then click **OK**.

Notes:

- If you modify the **Definition date** on the **Create New Well** dialog box so that it occurs *after* the FlexWell **Definition Date** and then click **OK**, a message will be displayed in the **Tubing | Well Info** panel, informing you that the well cannot be used, and to select another well.
- If you modify the **Definition date** on the **Create New Well** dialog box so that it occurs *before* the FlexWell **Definition Date** and then click **OK**, because the new well is automatically OPEN when it is created, an error will be displayed in the **Events** table. For more information about the **Events** table, refer to [Managing FlexWell and Well Events](#). If you apply these changes to the FlexWells dialog box, then an error icon will be displayed beside the FlexWell name in the Builder tree view, which you can display by right-clicking the FlexWell name and then selecting **Validate**.

5. As necessary, modify the **Tubing Properties** and **Tubing Diameter** data as outlined for the annulus (refer to [To assign an existing sink/source well to the annulus](#)). As explained in [To assign an existing sink/source well to a tubing string](#), the **Tubing | PERFS** table is not editable unless **Diameter Type** is set to **Variable**. There are differences between the **Annulus | PERFS** table and the **Tubing | PERFS** table. The **Tubing | PERFS** table has an **Insulation OD** column instead of a **Cement OD** column. Since flow into and out of the tubing occurs only at the end of the tubing string, there is no **Perf. Status** column. The table includes a column for selecting a **WP** (withdrawal point), but this column is disabled for injector wells.

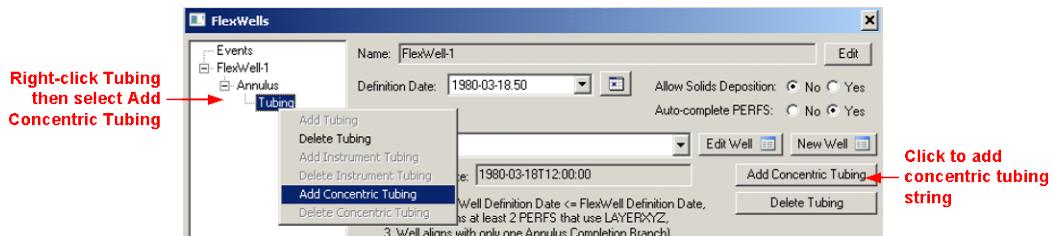
Note: A producer well cannot have a withdrawal point at the toe of the tubing string.

6. Click **Apply** to save your edits without closing the **FlexWells** dialog box. Click **OK** to apply the changes and close the **FlexWells** dialog box.

Managing Concentric Tubing Strings

To add a concentric tubing string to a tubing string

To add a concentric tubing string to an existing tubing string click the **Add Concentric Tubing** button, or right-click **Tubing** on the **FlexWell** tree view and then select **Add Concentric Tubing**:



To assign an existing sink/source well to a concentric tubing string

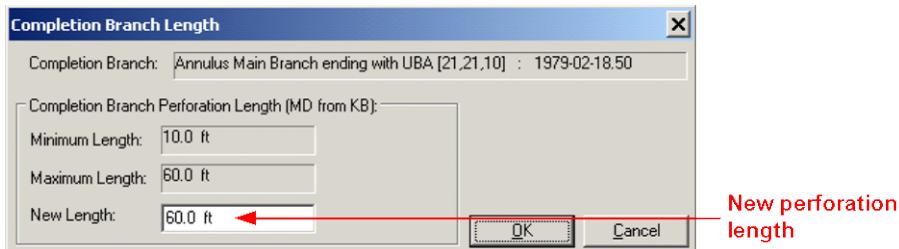
After the concentric tubing string is added, Builder will compile a list of valid sink/source wells to populate the **Concentric Tubing | Well** box. You will need to assign a sink/source well to the concentric tubing, as outlined below. The procedure assumes no validation errors. Validation errors are discussed in [FlexWells Validation Errors](#) on page 427.

1. Select a sink/source well from the list that Builder has automatically compiled and displayed in the **Concentric Tubing | Well** box. This list will only contain sink/source wells that meet the tubing criteria (see page 416) as well as the following:
 - Its perforation path aligns with the outer tubing completion branch.

Notes:

- a. To view validation errors at any time, right-click **FlexWells** or **FlexWell-n** in the Builder tree view and then select **Validate**. A list of validation errors will be displayed. Newly entered or modified data will only be validated after you have applied the changes.
 - b. If you clear the **Concentric Tubing | Filter Wells** check box, all sink/source wells from the dataset will appear in the **Concentric Tubing | Well** box.
 - c. If you clear **Filter Wells** and then try to assign a well with multiple completion branches or one that is not aligned with the outer tubing completion branch, Builder will take you through the steps of configuring the well so that it is aligned properly.
2. If the selected sink/source well *does* contain at least two perforations, and if **Auto-complete PERFS** is set to **Yes**, Builder checks to make sure that these perforations are continuous, and that they align with the outer tubing string. Perforations are continuous when all grid blocks between the starting and ending perforation are perforated.
- Note:** Unlike the annulus well, you cannot change the concentric tubing well completion data through the **Well Completions Data (PERF)** dialog box. Access to the concentric tubing well's **Well Completions Data (PERF) | Perforations** tab is blocked, and you can only select a well index type of GEO or GEOA on the tubing well's **Well Completions Data (PERF) | General** tab. This ensures that the concentric tubing well will stay aligned and always use LAYERXYZ geometric data.
3. As necessary, modify the **Concentric Tubing Properties** and **Concentric Tubing Diameter** data as was done for the annulus (refer to [To assign an existing sink/source well to the annulus](#)). The **Con. Tubing | PERFS** table is not editable unless **Diameter Type** is set to **Variable**. There are differences between the **Annulus | PERFS** table and the **Con. Tubing | PERFS** table. The **Con. Tubing | PERFS** table has an **Insulation OD** column instead of a **Cement OD** column. Since flow into and out of the concentric tubing occurs only at the end of the concentric tubing string, there is no **Perf. Status** column. The table includes a column for selecting a **WP** (withdrawal point), but this column is disabled for injector wells.
- Note:** A producer well cannot have a withdrawal point at the toe of the tubing string.
4. The concentric tubing well inherits its completion branch and perforations from the outer tubing in which it resides. The concentric tubing well completion branch length can be longer than, or shorter than the outer tubing, but it cannot be greater than the annulus completion branch in which they both reside.

After the concentric tubing well has been added, you can still modify the length of the concentric tubing completion branch whenever the **Edit** button inside the **Concentric Tubing Diameter** area is enabled. This will display the **Input Completion Branch Perforation Length (MD from KB)** dialog box:



Enter the new perforation length and then click **OK**.

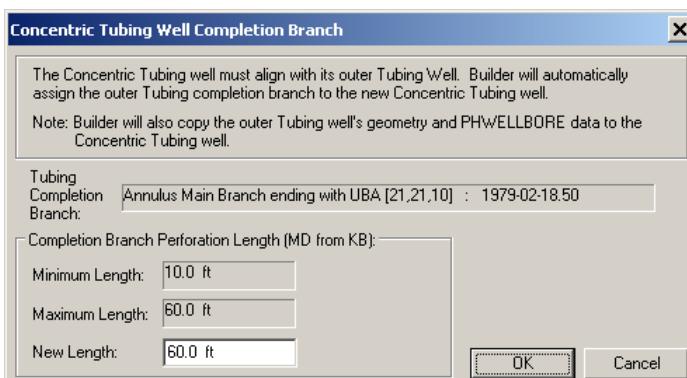
- Click **Apply** to save your edits without closing the **FlexWells** dialog box. Click **OK** to apply the changes and close the **FlexWells** dialog box.

Note: If you delete a tubing string that contains a concentric tubing string, this will also delete the concentric tubing.

To assign a new sink/source well to a concentric tubing string

To assign a new sink/source well to a concentric tubing string:

- Click the **New Well** button. The **Concentric Tubing Well Completion Branch** dialog box will be displayed:



The concentric tubing string must align with the outer tubing completion branch, so copying the outer tubing completion branch to the new concentric tubing well guarantees that they will align.

Most tubing strings only allow flow from the end of the tube, so all of the selected outer tubing completion branch perforations will be copied to the concentric tubing well with a CLOSED perforation status, except for the ending perforation which will be OPEN. This dialog box also allows you to specify the concentric tubing completion branch perforation length.

2. Specify the concentric tubing completion branch length. The concentric tubing well completion branch length can be longer or shorter than the outer tubing in which it resides, but it cannot be greater than the annulus completion branch in which they both reside.

If you click **Cancel**, the new concentric tubing well will not be created and the concentric **Tubing | Well** box will revert back to its previous selection.

3. If you select **OK**, the **Create New Well** dialog box will be displayed, allowing you to define the new concentric tubing well. The steps for creating the new well are the same as steps 3 to 5 for creating a new well for a tubing string, as outlined in [To assign a new sink/source well to a tubing string](#).
4. Click **Apply** to save your edits without closing the **FlexWells** dialog box. Click **OK** to apply the changes and close the **FlexWells** dialog box.

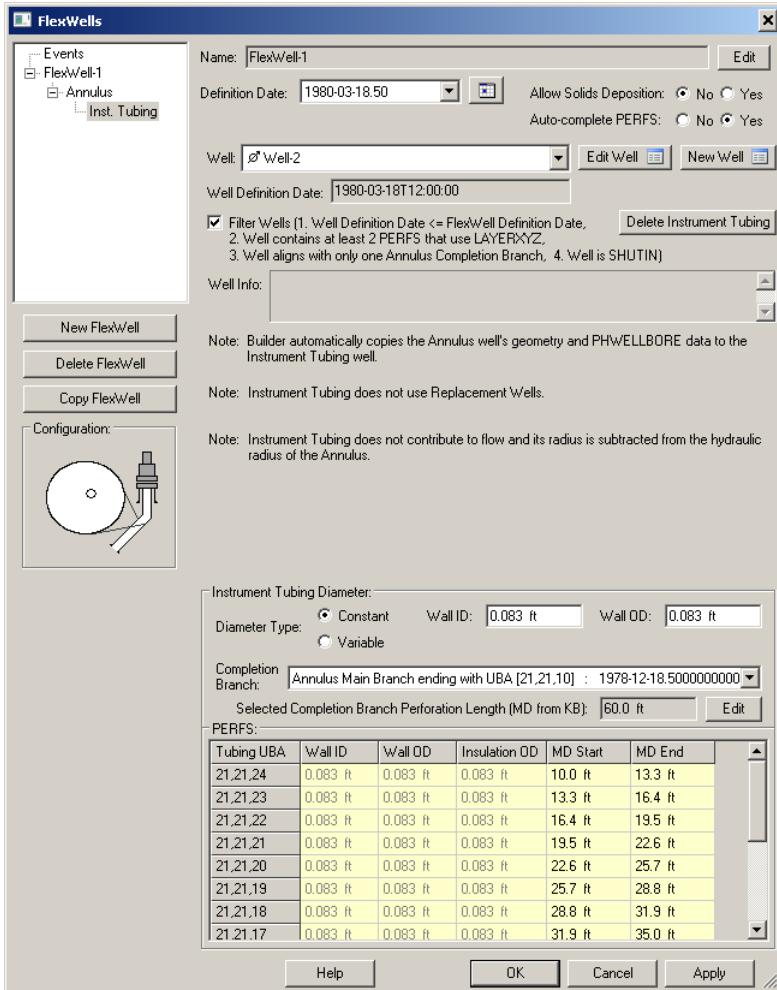
Note: If you delete a tubing string that contains a concentric tubing string, this will also delete the concentric tubing.

Managing Instrument Tubing Strings

To assign an existing or new sink/source well to an instrument tubing string

The procedure for assigning an existing or new sink/source well to an instrument tubing string is similar to the procedures outlined in [To assign an existing sink/source well to a tubing string](#) and [To assign a new sink/source well to a tubing string](#). An instrument tubing string does not require **Tubing Properties** or **Packers** and the sink/source well that is assigned to the instrument tubing must always be SHUTIN.

The **FlexWells | Instrument Tubing** dialog box should be similar to the following:



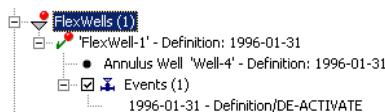
Note: The instrument tubing sink/source well does not contribute to flow, and its radius is subtracted from the hydraulic radius of the annulus.

FlexWells Validation Errors

The following table provides a list of errors you may encounter when performing FlexWell operations on an annulus, tubing string, concentric tubing string, or instrument tubing string.

Note: If you open a dataset that contains FlexWell validation errors, these errors will be displayed in a **Builder** message box after the file is loaded. Error icons will also be displayed next to the appropriate FlexWells in the Builder tree view, and errors will be indicated in the cells of the corresponding **Events** table.

| Error | Detail | Applies to |
|--|---|---|
| 1 No wells meet the selection criteria | If Builder does not find any wells that meet the selection criteria (well definition date equal to or before the FlexWell definition date, a minimum of two perforations, and LAYERXYZ geometry), an error message will be displayed in the Well Info panel, informing you that there are no available sink/source wells that meet these criteria. | Annulus Tubing Concentric Tubing Instrument Tubing |
| 2 Selected well definition date is later than FlexWell definition date | If the Filter Wells check box is not selected, and you select a sink/source well with a Well Definition Date that is later than the FlexWell Definition Date , a message will be displayed in the Well Info panel, informing you that the well cannot be used, and directing you to select another well. | Annulus Tubing Concentric Tubing Instrument Tubing |
| 3 Selected well must be SHUTIN until used by an ACTIVATED FlexWell | If the FlexWell is ACTIVATED on its Definition Date and you assign a sink/source well with an OPEN status and a Well Definition Date that is earlier than the FlexWell Definition Date , then the well must be SHUTIN until it is used by the FlexWell. While the error is not resolved, if you click Apply or OK in the FlexWells dialog box, an error indicator will be displayed beside FlexWells and FlexWell-n in the tree view, as shown in the following example: | Annulus Tubing Concentric Tubing Instrument Tubing |



An error will also be displayed in the FlexWell **Events** table.

| Error | Detail | Applies to |
|--|--|---|
| | <p>Note: If the well is assigned to another ACTIVATED FlexWell, it is allowed to be OPEN.</p> <p>For information about modifying FlexWell events, refer to Managing FlexWell and Well Events.</p> | |
| 4 Selected well must be SHUTIN if assigned to a DE- ACTIVATED FlexWell | <p>There are two variants of this error:</p> <p>Variant a: If the FlexWell is DE- ACTIVATED <i>on</i> its Definition Date and you assign a sink/source well with an OPEN status and a Well Definition Date that is the same as the FlexWell Definition Date, then the well must be SHUTIN until the FlexWell is ACTIVATED.</p> <p>Variant b: If the FlexWell is DE- ACTIVATED <i>after</i> its Definition Date and you assign a sink/source well with an OPEN status, the well must be SHUTIN until the FlexWell is ACTIVATED.</p> <p>While the error is not resolved, if you click Apply or OK in the FlexWells dialog box, an error indicator will be displayed beside FlexWells and FlexWell-n in the Builder tree view, as shown in the following example:</p> <p>An error will also be displayed in the FlexWell Events table.</p> <p>Note: If the well is assigned to another ACTIVATED FlexWell, it is allowed to be OPEN.</p> <p>For information about modifying FlexWell events, refer to Managing FlexWell and Well Events.</p> | Annulus Tubing Concentric Tubing Instrument Tubing |

| Error | Detail | Applies to | |
|-------|---|---|---|
| 5 | Selected well does not have perforations | If you clear the Filter Wells check box and then select a sink/source well that <i>does not</i> contain perforations, a message will be displayed in the Well Info panel, informing you that tubing and instrument tubing cannot be assigned until annulus well perforations are added. For information about adding perforations to the annulus well, refer to To modify perforations . | Annulus Tubing Concentric Tubing Instrument Tubing |
| 6 | Selected well does not have at least two perforations | If you clear the Filter Wells check box and then select a sink/source well that <i>does not</i> contain a minimum of two perforations, a message will be displayed in the Well Info panel, informing you that tubing cannot be assigned until more well perforations are added. For information about adding perforations refer to To modify perforations . | Annulus Tubing Concentric Tubing Instrument Tubing |
| 7 | Selected well does not have continuous perforations | This error occurs when a FlexWell using Auto-complete PERFS is saved to a dataset and then you manually delete some of the FlexWell perforations using a text editor and then save the file. When you load this file in Builder, the FlexWell perforations will be checked to make sure they are continuous. If they are not, a message will be displayed directing you to either clear Auto-complete PERFS or click Edit Well Completion Data (PERF) to add the missing perforations. | Annulus Tubing Concentric Tubing Instrument Tubing |
| 8 | Selected well does not use LAYERXYZ geometric data | If you clear the Filter Wells check box and then select a sink/source well that contains perforations but does not use LAYERXYZ geometric data, Builder will ask you if it can modify the well so that it does use LAYERXYZ geometric data. If you click No , a message will be displayed in the Well Info panel, informing you that the well cannot be used, and to select another well. If you click Yes , Builder will modify the well so that it uses LAYERXYZ geometric data, with a default well index type of GEO. | Annulus Tubing Concentric Tubing Instrument Tubing |

| Error | Detail | Applies to |
|---|--|---|
| 9 Selected well has been modified and the completion branches must be updated | If the dataset was manually edited, a problem can occur where the constructed FlexWell perforation branches may not match the actual well perforations. In this case, a message will be displayed instructing you to click the annulus Edit Well Completion Data (PERF) button. When the Well Completion Data (PERF) dialog box is opened then closed, Builder recalculates the annulus completion branches and determines if the existing tubing wells still align with the newly calculated annulus completion branch. | Annulus |
| 10 Selected well type is invalid | Builder checks to make sure that the selected well is either an INJECTOR or a PRODUCER. If it is an INJECTOR, it must use MOBWEIGHT. | Annulus Tubing Concentric Tubing Instrument Tubing |

Modifying Well Perforations with the FlexWells Dialog Box

After a sink/source well is assigned to a FlexWell annulus, you can only modify the annulus well perforations by clicking the  button on the **Annulus** tab. If the **FlexWells** dialog box is not open, you will not be able to modify the annulus well perforations by accessing the **Well Completion Data (PERF)** dialog box through the Builder **Well** menu or through the **Wells & Recurrent** tree view. Also, you will not be able to modify the annulus well perforations by accessing the **Trajectory Perforation Interval** dialog box through the Builder **Well** menu. Both of these dialog boxes have been modified to detect if the selected well is assigned to a FlexWell, and if it is, will disable all of its perforation-related controls.

These dialog boxes are disabled to prevent users from inadvertently modifying an assigned well such that its perforations are no longer continuous, or modifying an assigned well in such a way that its completion branch no longer aligns with the annulus.

A message is displayed at the bottom of these dialog boxes that:

- The well is assigned to a FlexWell.
- You must use the FlexWell's **Edit Well Completion Data (PERF)** button to modify FlexWell perforations.
- Only annulus perforations can be modified.

After a sink/source well is assigned to a FlexWell tubing, concentric tubing or instrument tubing string, you cannot directly modify its perforations. You can only modify the well completion branch length by clicking the **Edit** button next to the **Selected Completion Branch Perforation Length (MD from KB)** value.

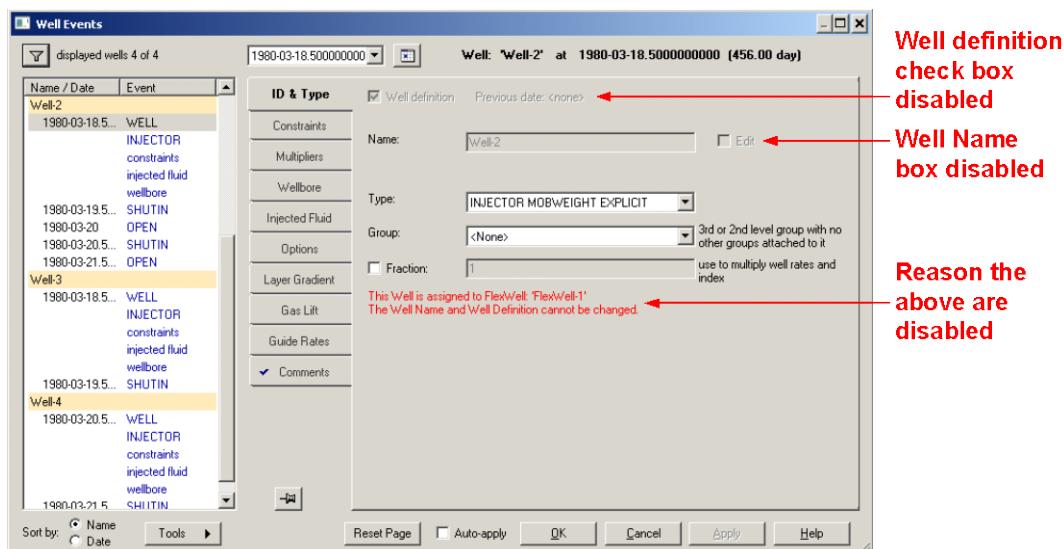
Modifying Well Names, Definition Dates, and Completion Dates

A sink/source well that is assigned to a FlexWell cannot be deleted. The well must be removed from the FlexWell before it can be deleted. If you try to delete a well using the **Wells & Recurrent** tree view, a **FlexWell Warning** will be displayed, informing you that the well cannot be deleted.

Similarly, any well assigned to a FlexWell cannot have its completions deleted. The well must be removed from the FlexWell before the well completions can be deleted. If you try to delete the well completions using the **Well Completion Data (PERF)** dialog box, a **FlexWell Warning** will be displayed, informing you that the well completions cannot be deleted.

As well, if you try to delete multiple well completions, you will not be able to select well completions that are assigned to a FlexWell on the **Select Model Wells** dialog box.

Finally, any sink/source well assigned to a FlexWell cannot have its name or definition date changed. As shown below, if the well is assigned to a FlexWell, both the **Name** box and the **Well definition** check box will be disabled on the **Well Events** dialog box.



Similarly, any sink/source well assigned to a FlexWell cannot have its completion date changed. If you try to modify the well completion date using the **Well Completion Data (PERF)** dialog box, a **FlexWell Warning** will appear, informing you that the well completion date cannot be changed.

Using Sink/Source Wellbore Models

If a sink/source well is not assigned to a FlexWell, the well's wellbore model (PHWELLBORE) casing and tubing properties apply to the entire well. If the sink/source well is assigned to a FlexWell, the well's wellbore model (PHWELLBORE) casing and tubing properties apply from the Kelly Bushing to the first perforation, after which FlexWell casing and tubing data is used.

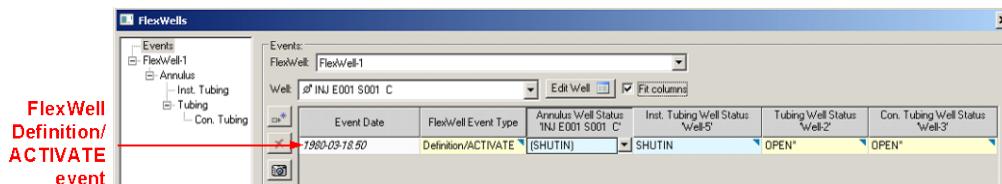
Notes:

1. If the annulus uses a wellbore model (PHWELLBORE), all wells assigned to the FlexWell must use a wellbore model (PHWELLBORE). If the annulus does not use a wellbore model (PHWELLBORE), all wells assigned to the FlexWell cannot use a wellbore model (PHWELLBORE).
2. If any of a FlexWell's assigned tubing wells specify a WP (withdrawal point) in their PERFS grid then none of the assigned wells can use a wellbore mode.

Managing FlexWell and Well Events

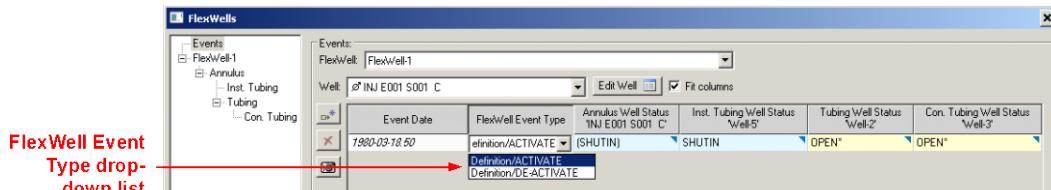
To access the FlexWell Events table, click the **Events** node in the **FlexWell** tree view. The last selected FlexWell will automatically have its information displayed in the **Events** table.

When a new FlexWell is created, it is automatically in the ACTIVATED state. For this reason, Builder assigns a **Definition/ACTIVATE** event to each new FlexWell. This event is not written to the dataset, and is only displayed to notify users that the FlexWell was activated when it was defined. The **Definition/ ACTIVATE** event can be changed to a **Definition/DE-ACTIVATE** event, but it cannot be deleted or have its date modified from the **Events** view. To change the FlexWell **Definition/ ACTIVATE** event date, users must select a different date from the **Definition Date** box or create a new date by clicking the **Calendar** button on either the **Annulus**, **Tubing**, **Concentric Tubing** or **Instrument Tubing** tabs. Refer to page 435 for information about changing the FlexWell **Definition Date**.



Basic Operations

To change a **Definition/ACTIVATE** event to a **Definition/DE-ACTIVATE** event, open the drop-down list in the **FlexWell Event Type** cell and then select **Definition/DE-ACTIVATE**:



Note: An italicized Event Date, such as the one shown above, cannot be modified.

Entries in the **Events** table are text-formatted as outlined in the **Legend** area below the table:

| | | |
|---|--------------------------------|---|
| <u>Legend:</u> | 'Well Status' | - Explicit Well Status event (OPEN and SHUTIN only). |
| | 'Well Status*'* | - Implicit Well Status event (Well events such as WELL, PRODUCER, INJECTOR, ALTER, WTMULT, TARGET and some types of Constraints can cause a Well to be OPEN or SHUTIN). |
| | '(Well Status)' | - Explicit/Implicit Well Status event occurs prior to FlexWell definition date or REPLACE event date. |
| | 'Well Status:(#)' | - Well assigned to multiple active FlexWells on this date. |
| | 'Well Status:(<*)' | - Well must be SHUTIN until used. |
| | 'Well Status' or '----' | - Well must be SHUTIN on this date. |
| | '.....' | - No event or replacement Well on this date. |
| | 'FlexWell Event Type' | - Consecutive FlexWell events. |
| | 'Well name:(+)' | - Well already assigned to this FlexWell. |
| | 'Well name:(WT)' | - Invalid Well Type. |
| | 1991-07-07 | - Cannot change date. |
| | | <input checked="" type="checkbox"/> View Legend |
| | | <input type="button" value="Clean Events"/> |
| Note: The replaced well's PERFS, geometry and PHWELLBORE data are automatically copied to the new REPLACE well. | | |

Notes:

1. If you clear **View Legend** (in the lower right of the **Legend** area), the legend will be removed and the **Events** table will be extended into the vacated area.
2. If you move the pointer over an event cell, a short message will be displayed if there is an explicit or implicit error associated with it.

A **Well Status** column is added to or removed from the **Events** table each time a well is assigned to or removed from the FlexWell. If the assigned well is defined before the FlexWell definition date, the table will only show the well's status on the FlexWell **Definition/ACTIVATE** or **Definition/DE-ACTIVATE** event date, not before. FlexWell does not use well status on dates before the FlexWell was defined. If the assigned well status date is defined before the FlexWell definition date, its status text will be shown in brackets.

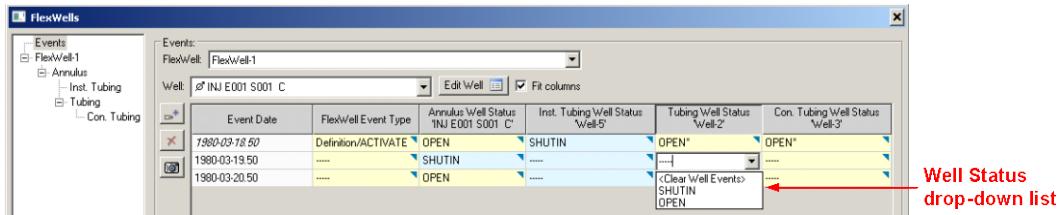
In the above example, an annulus, tubing string, instrument string, and concentric tubing string have been added to the FlexWell.

The **Events** table shows all well status events that occur on or after the FlexWell definition date. Initially, the **Definition/ACTIVATE** event will be the only FlexWell Event Type in the **Events** table. In the following example, the annulus well has been SHUTIN then OPENed:

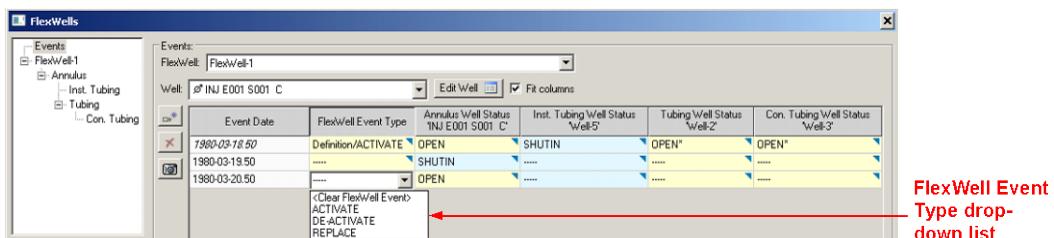
| Event Date | FlexWell Event Type | Annulus Well Status | Inst. Tubing Well Status | Tubing Well Status | Con. Tubing Well Status |
|---------------|---------------------|---------------------|--------------------------|--------------------|-------------------------|
| 1990-03-18 50 | Definition/ACTIVATE | OPEN | SHUTIN | OPEN* | OPEN* |
| 1990-03-19 50 | | SHUTIN | | | |
| 1990-03-20 50 | | OPEN | | | |

Annulus well
SHUTIN then
OPENed

To add a new well status event or modify an existing well status event, make a selection from the appropriate well status list.



To add a new FlexWell event to an existing **Well Status** date, make a selection from any **FlexWell Event Type** box except in the first row.



Event Dates

Adding Event Dates

To add a new **Event Date** to the table, select an existing date in the **Event Date** column and then click the button. If the selected event date has no succeeding event date, the new event date will be 1 day after the selected event date. If the selected event date has a succeeding event date after it, then the new event date will be half the time between the selected and succeeding dates.

Note: The new date will be removed from the table if no **FlexWell Event Type** or **Well Status** type is added to the date before the **Events** table is closed.

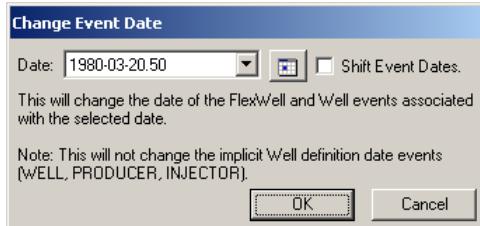
Deleting Event Dates

To delete a selected event date, click the button. All **FlexWell Event Type** events and **Well Status** events, both explicit and implicit, for that event date will be deleted.

Note: You will not be able to delete the FlexWell or well **Definition Date** events through the **Events** table.

Changing Event Dates

To change any FlexWell or well event date, select the date in the table and then click beside it. The **Change Event Date** dialog box will be displayed.



Select the new date in the **Date** drop-down list, or click  to open the **Calendar** dialog box and then enter the new date. You can change the date forward or backward, as long as it does not overlap with the date of the adjacent event. All FlexWell and well events on the changed date are moved to the new date.

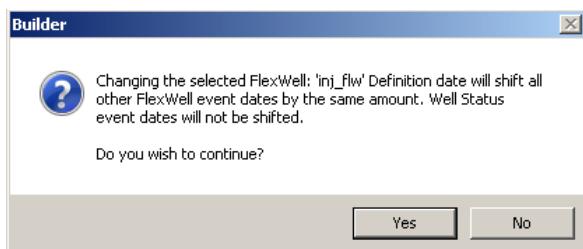
If you select **Shift Event Dates**, as shown in the following example, then all dates after the adjusted date will be similarly adjusted; for example, if you add one week to the selected date, then all dates after it will be increased by one week.



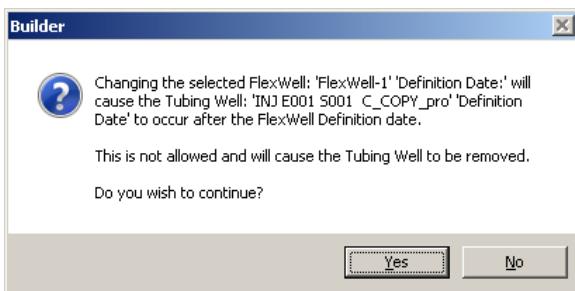
You can shift dates forward but not backward, since you are not allowed to change the FlexWell **Definition Date**, or the well **Definition Date**, through the **Events** table.

Note: You cannot change the **Definition/ACTIVATE** or **Definition/DE-ACTIVATE** date through the **Events** table. To change the FlexWell **Definition/ ACTIVATE** event date, users must select a different date from the **Definition Date** box or create a new date by clicking the **Calendar**  button on either the **Annulus**, **Tubing**, **Concentric Tubing** or **Instrument Tubing** tabs.

Before the date will be changed, you will be notified that all FlexWell event dates will be shifted to correspond with the new FlexWell **Definition Date** and that well status event dates will not be changed:



If the FlexWell definition date is shifted backwards, such that its assigned wells are defined *after* the FlexWell definition date, the following message will be displayed, informing you that these wells will be removed if you continue:



Replace Wells

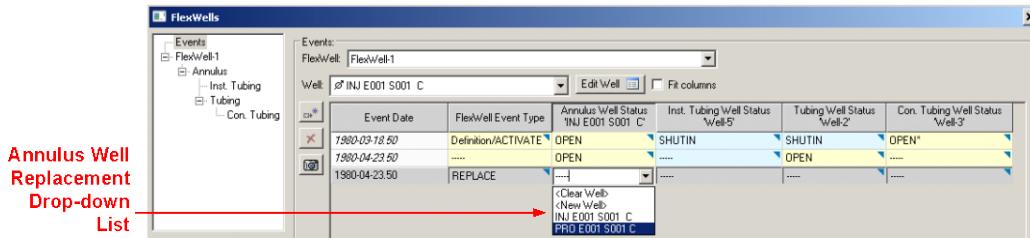
You can use the REPLACE FlexWell Event Type to replace one well with another; for example, to replace an injector well with a producing well. To do this, on the date row where you want to make the replacement, select REPLACE in the **FlexWell Event Type** cell drop-down list. Two rows will be entered, with the same event date. The row above the REPLACE row allows you to change the well status of the well that will be replaced (its date cannot be changed). The REPLACE row allows you to select a replacement well for the currently assigned well, as shown below:

| Event Date | FlexWell Event Type | Annulus Well Status | Inst. Tubing Well Status | Tubing Well Status | Con. Tubing Well Status |
|---------------|---------------------|---------------------|--------------------------|--------------------|-------------------------|
| 1980-09-18 50 | OPEN | INJ E001 S001 C | SHUTIN | SHUTIN | OPEN* |
| 1980-04-23 50 | OPEN | | | OPEN | |
| 1980-04-23 50 | REPLACE | | | | |

Notes:

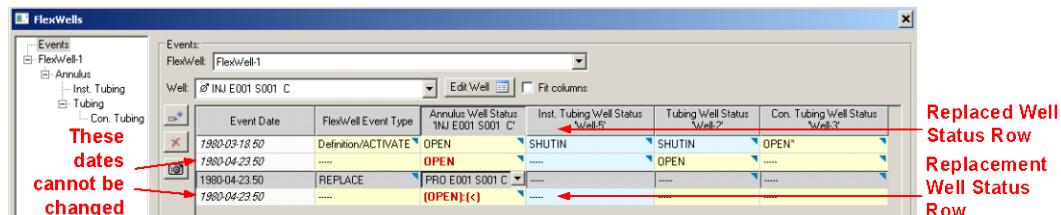
- The instrument tubing well cannot be replaced.
- When you replace a well, the name of the well is added to the **Replacement Wells** box in the appropriate **Annulus**, **Tubing** or **Concentric Tubing** tab. To view a table of replacements, click the tab's **Table** button beside the **Replacement Wells** box.

In the REPLACE row, in the column for the well you want to replace, open the drop-down list of wells, for example:

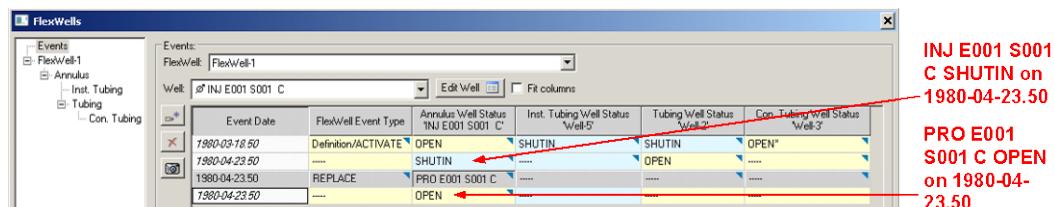


Note: The drop-down list will contain only the names of wells that are defined on or before the REPLACE event date and that have the same completions as the replaced well.

Select the replacement well. In the above example, if you select PRO E001 S001 C, a new row will be inserted after the REPLACE row. The new row below the REPLACE row allows you to change the status of the replacement well if required.

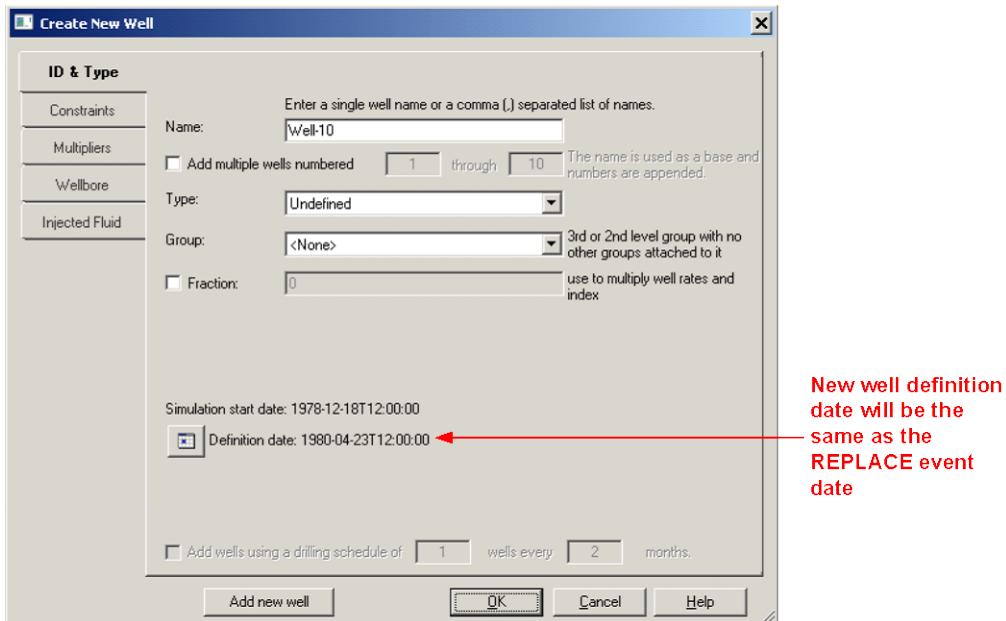


In the above example, you need to SHUTIN well INJ E001 S001 C on 1980-04-23.50 and OPEN well PRO E001 S001 C on 1980-04-23.50, as shown below:



In the above example, you can replace the annulus well multiple times to simulate cycling between injection and production.

If there is no appropriate annulus replacement well available in the drop-down list, you can select <New Well> in the Annulus Well Status column to open the **Create New Well** dialog box:

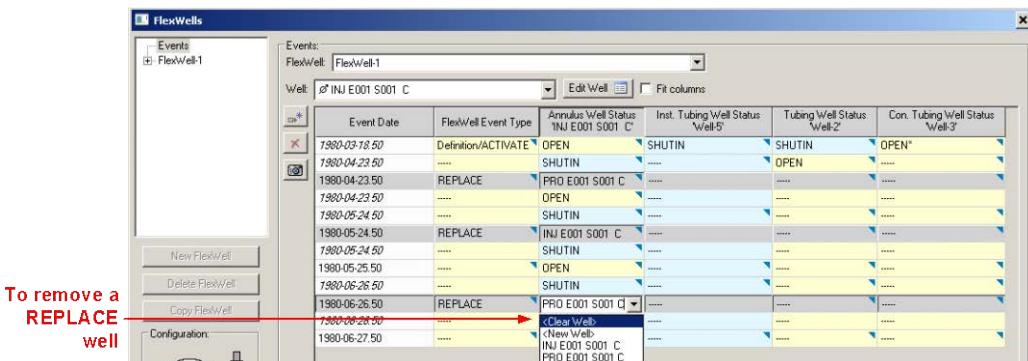


Modify the well parameters as necessary. You will not be able to change the completions or the completion length, which must be the same as those of the replaced well. Click **OK** to return to the **Events** table.

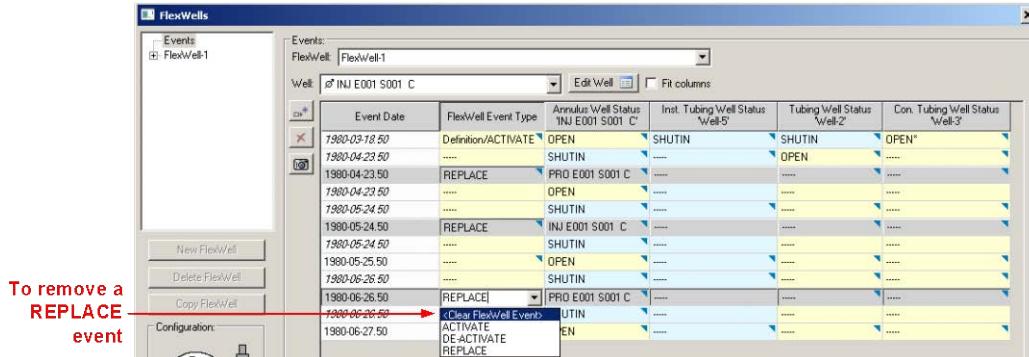
Notes:

1. The new well definition date will be the same as the REPLACE event date.
2. To insert an event after a REPLACE event, select the REPLACE event and then click the button.

To remove a REPLACE well, select <Clear Well> in the drop-down list in the well column:



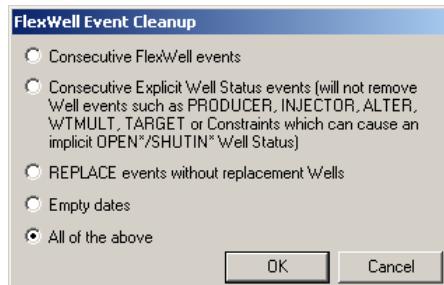
To remove a REPLACE event, select <Clear FlexWell Event> from the drop-down list in the **FlexWell Event Type** cell.



<Clear FlexWell Event> removes the REPLACE rows and the table reverts to the earlier state; that is, the state before the REPLACE.

Clean Events

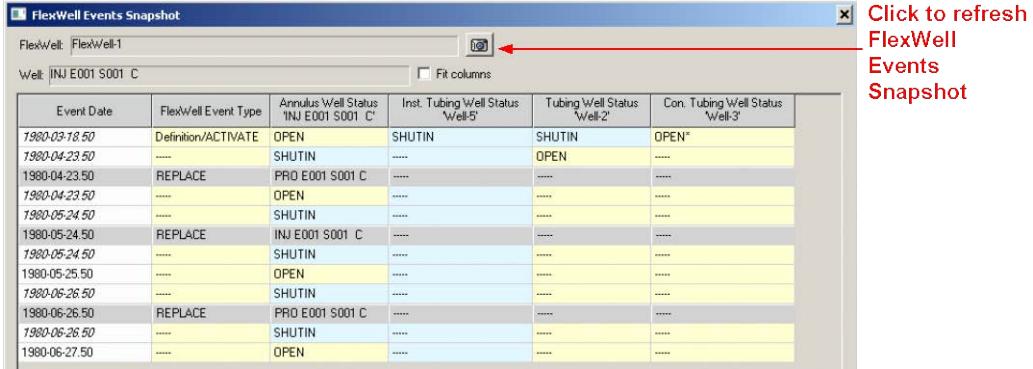
You can use the Clean Events feature to remove various problem events. Click the **Clean Events** button in the lower right corner of the Legend to open the **FlexWell Event Cleanup** dialog box:



Select the desired option and then click **OK**. If you select **Empty dates**, for example, Builder will remove all rows in the **Events** table that have no entry in the **FlexWell Event Type** and **Well Status** columns. If you select **All of the above**, Builder will carry out all of the cleanup tasks.

Events Snapshot

To take a “snapshot” of the current **Events** table, click the **Take Snapshot of Events Grid** button. A snapshot of the **Events** table will be displayed in a separate **FlexWell Events Snapshot** dialog box, as shown below. This dialog box can be used for reference purposes, while you are making additional changes to the **Events** table.



Click the **Refresh Snapshot** button in the **FlexWell Events Snapshot** dialog box to refresh the snapshot or click **OK** to close it.

Events Errors

As outlined above, the **Events** table displays errors by highlighting them in red text, or bold red text. Each error will be explained in the validation messages for that FlexWell. Event errors can be any of the following:

1. Instrument tubing sink/source well must be SHUTIN for the life of the FlexWell.
2. Consecutive FlexWell events of the same type are not allowed.
3. If an assigned well is defined before its FlexWell (FlexWell-1, for example), it must be SHUTIN until the FlexWell-1 definition date, unless the assigned well is also assigned to another ACTIVATED FlexWell (FlexWell-2, for example) which has a definition date on or before the assigned well.
4. If the assigned well is defined before its REPLACE event, it must be SHUTIN until the REPLACE event date, unless the assigned well is also assigned to another ACTIVATED FlexWell (FlexWell-2, for example) which has a definition date on or before the REPLACE event date.
5. During the DEACTIVATE time of a FlexWell, all assigned wells must be SHUTIN, unless the assigned wells belong to another FlexWell which has an ACTIVATE event during that time.
6. If a well is concurrently assigned to multiple FlexWells, only one FlexWell may be ACTIVATED.
7. A well can be shared by multiple tubing strings as long as it is only used by a single tubing string at any given time. This also applies to the annulus. Consider the case where Well-1 is assigned to a tubing string, then replaced by Well-2. After it is replaced, Well-1 can be assigned to another tubing string in the same FlexWell.
8. If a well is an INJECTOR type, it must be MOBWEIGHT.

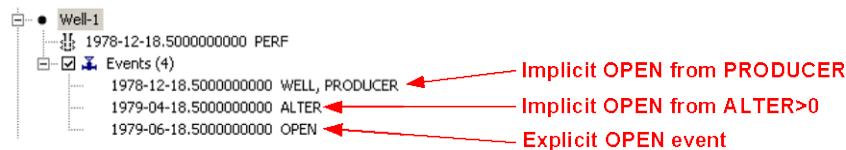
To return to the **Annulus** or any of the **Tubing** tabs, select the appropriate item in the **FlexWell** tree view.

Examples

The following examples provide a sampling of the operations that are available through the **Events** table.

Example 1: Effect of changing FlexWell definition date on well events.

1. Create a new well, Well-1, configured as follows:



2. Create a new FlexWell, FlexWell-1, with its definition date the same as the Well-1 definition date. Assign Well-1 as the annulus. In the **FlexWells Events** table, the following events are displayed:

| Event Date | FlexWell Event Type | Annulus Well Status |
|---------------|---------------------|---------------------|
| 1978-12-18.50 | Definition/ACTIVATE | OPEN* |
| 1979-04-18.50 | ALTER | OPEN* |
| 1979-06-18.50 | OPEN | OPEN |

3. Change the FlexWell definition date to 1979-02-16.50, which is before the ALTER and OPEN events, but after the WELL, PRODUCER definition date:

| Event Date | FlexWell Event Type | Annulus Well Status |
|---------------|---------------------|---------------------|
| 1979-02-16.50 | Definition/ACTIVATE | (OPEN*):< |
| 1979-04-18.50 | ALTER | OPEN* |
| 1979-06-18.50 | OPEN | OPEN |

As shown above, the Well-1 status on the FlexWell-1 definition date changes to **(OPEN*):<**. The opening () brackets indicate that the well status even occurs before the FlexWell definition date. The * asterisk indicates that the status event is implicit. The :< indicates that the well must be SHUTIN until it is used and finally, it is bold red because it is an error condition.

4. Change the FlexWell definition date to 1979-05-16.50, which is between the ALTER and OPEN events. The **Events** table will change as follows:

| Event Date | FlexWell Event Type | Annulus Well Status |
|---------------|---------------------|---------------------|
| 1979-05-16.50 | Definition/ACTIVATE | (OPEN*):(<) |
| 1979-06-18.50 | | OPEN |

As shown above, the Well-1 status on the FlexWell-1 definition date remains as **(OPEN*):(<)** which indicates its OPEN status is set implicitly and that the Well-1 definition date is before the FlexWell-1 definition date and is not SHUTIN.

5. Change the FlexWell definition date to 1979-07-19.50, which is after all of the well events. The **Events** table will change as follows:

| Event Date | FlexWell Event Type | Annulus Well Status |
|---------------|---------------------|---------------------|
| 1979-07-19.50 | Definition/ACTIVATE | (OPEN):(<) |

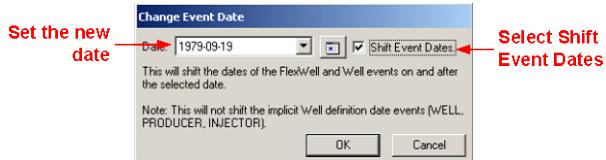
As shown above, the Well-1 status on the FlexWell-1 definition date changes to **(OPEN):(<)** which indicates that its OPEN status is set explicitly, and that the Well-1 definition date is before the FlexWell-1 definition date and must be SHUTIN.

Example 2: Effect of shifting an event date on other FlexWell and well event dates.

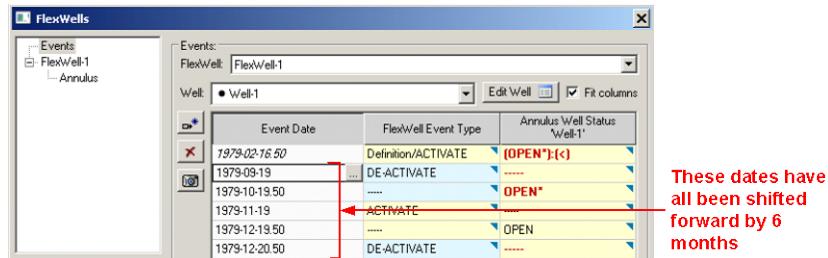
1. Start with the configuration used in Example 1. Reset the FlexWell definition date to 1979-02-16.50. Add a few FlexWell events to the **Events** table as shown below:

| Event Date | FlexWell Event Type | Annulus Well Status |
|---------------|---------------------|---------------------|
| 1979-02-16.50 | DE-ACTIVATE | (OPEN*):(<) |
| 1979-03-19 | DE-ACTIVATE | |
| 1979-04-18.50 | | OPEN* |
| 1979-05-19 | ACTIVATE | |
| 1979-06-18.50 | | OPEN |
| 1979-06-19.50 | DE-ACTIVATE | |

2. Select Event Date 1979-03-19 by clicking beside it. The **Change Event Date** dialog box will open. As shown below, change the date to 1979-09-19, which is 6 months after the original date. Select **Shift Event Dates**, and then click **OK**.

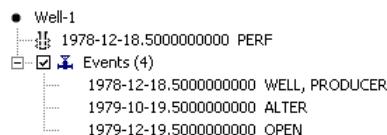


3. All of the well and FlexWell event dates after the shifted date will be shifted forward by 6 months as shown below:



Note: If there had been well definition dates (i.e., not well event dates) after the shifted date, these dates would not have been shifted.

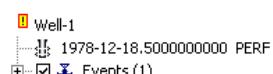
4. The well nodes in the **Wells & Recurrent** tab in the **Model Tree View** will also be shifted, as shown below (all dates are shifted except for the well definition date):



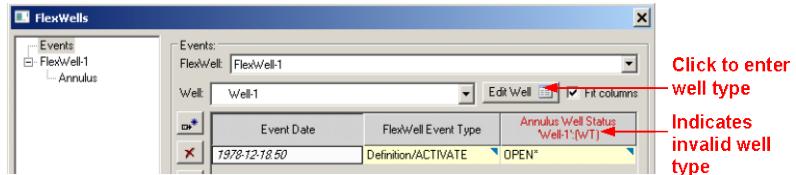
5. In this example, if you had not selected **Shift Event Dates**, then only the selected event would be shifted. If the shift causes the shifted event to overlap with the next (adjacent) event, you would get a message that this is not allowed and to select another date.

Example 3: Well defined on FlexWell Definition Date but no Well Type defined.

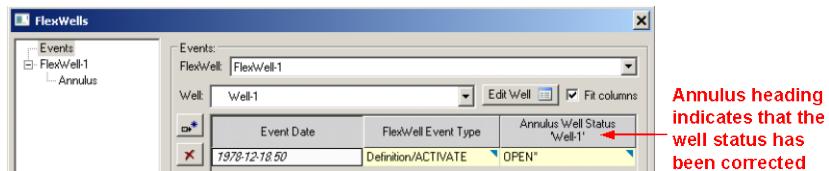
1. Create Well-1 with a well definition date of 1978-12-18.50 but do not define the well type. In the Model Tree View, the well will have a warning which you can view by right-clicking Well-1 and then selecting **Validate**.



2. Create FlexWell-1 with a definition date the same as Well-1. Assign Well-1 to the annulus.
3. Select **Events** in the **FlexWells** tree view:

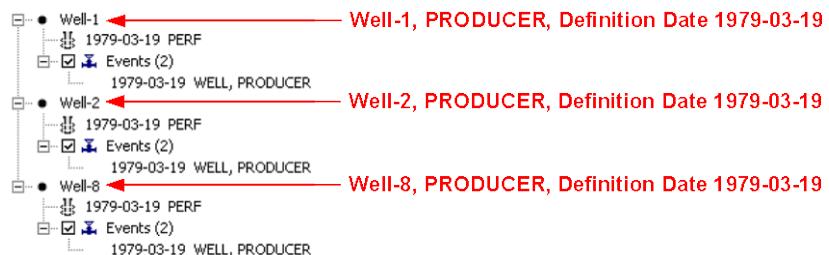


4. As shown above, the **Annulus** heading indicates that the assigned well type is invalid. Click **Edit Well** to open the **Well Events** dialog box and then set the well type through the **ID & Type** tab.
5. Select the well type and then click **OK**. You will have to set constraints, for example, to OPERATE. The warning is removed from Well-1 in the Model Tree View, and the error condition is removed from FlexWell-1.
6. In the **Events** table, the **Annulus** heading will change as follows, indicating that the well type is now valid:

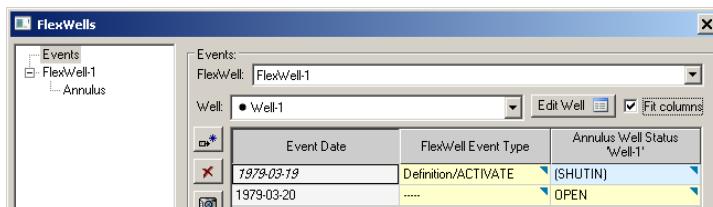


Example 4: Well used by more than one tubing string at the same time.

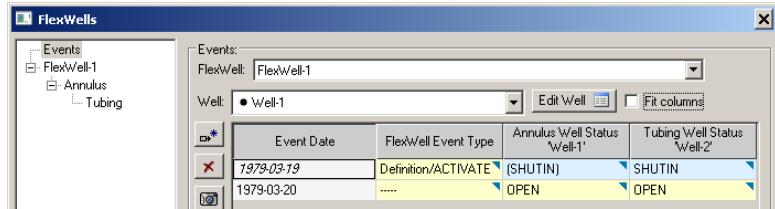
1. Create wells Well-1, Well-2 and Well-8, as follows:



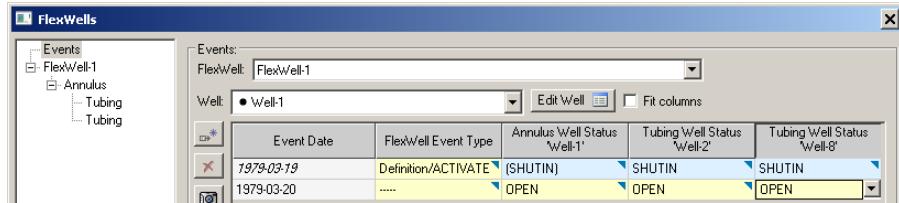
2. Create FlexWell-1 and assign Well-1 to the annulus. The **Events** table will appear as follows:



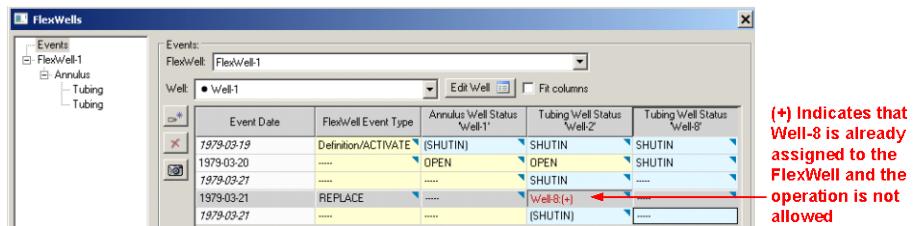
3. Create a tubing string and assign Well-2 to it. Resolve errors and warnings. The **Events** table now appears as follows:



4. Create a second tubing string using a third well, Well-8 for example:

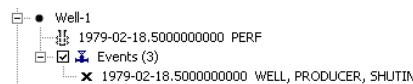


5. Replace Well-2 with Well-8. You will get the following, indicating that Well-8 is already being used in FlexWell and cannot be used again:

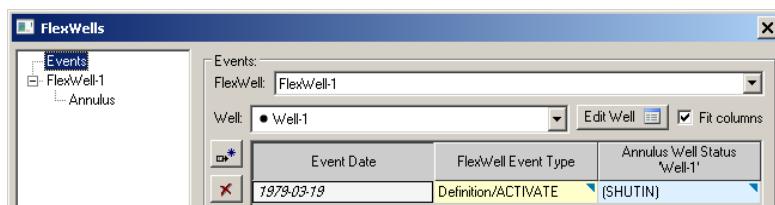


Example 5a: Assign a well which is already assigned to another FlexWell.

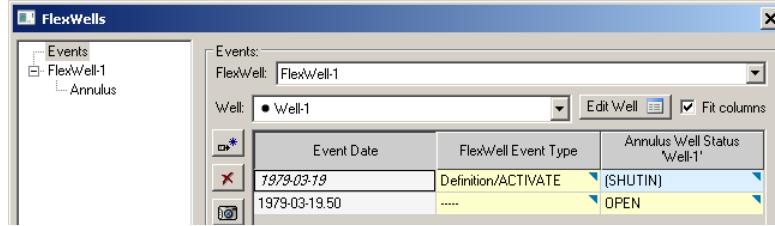
1. Create Well-1 as follows:



2. Create FlexWell-1 and assign Well-1 to its annulus, as shown in the following example:



3. In the **Events** table, add an event to change **Annulus Well Status** to OPEN and then click **Apply**:

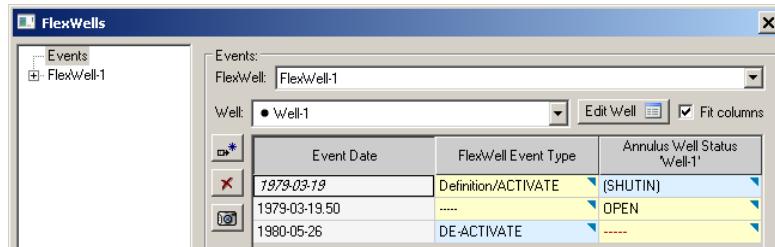


The Well-1 entry in the Model Tree View will be as follows:

```

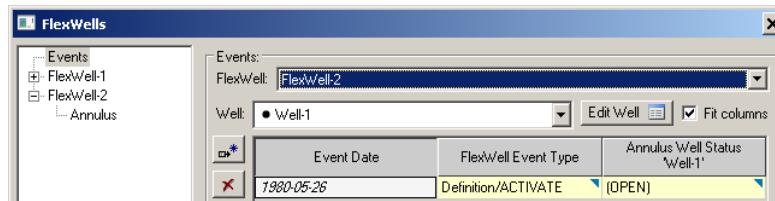
● Well-1
  |- 1979-02-18.5000000000 PERF
  |- Events (4)
    |- 1979-02-18.5000000000 WELL, PRODUCER, SHUTIN
    |- 1979-03-19.5000000000 OPEN
  
```

4. DE-ACTIVATE FlexWell-1; in our example, on 1980-05-26, as shown below:

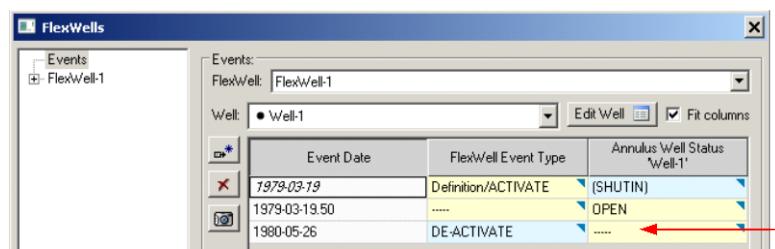


The “----” Annulus Well Status indicates that Well-1 must be SHUTIN because FlexWell-1 is DE-ACTIVATED and the well is not assigned to another ACTIVATED FlexWell on this date.

5. Create FlexWell-2 on 1980-05-26 and assign Well-1 to its annulus, as shown in the FlexWell-2 Events table:



The FlexWell-1 Events table will change as follows:

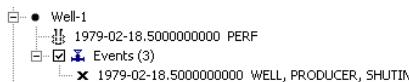


Status has
changed from red
bold to black
normal

The reason Well-1 no longer shows an error for FlexWell-1 or FlexWell-2 is because only FlexWell-2 is ACTIVATED on 1980-05-26.

Example 5b: Assign a well which is already assigned to an ACTIVATED FlexWell to another ACTIVATED FlexWell.

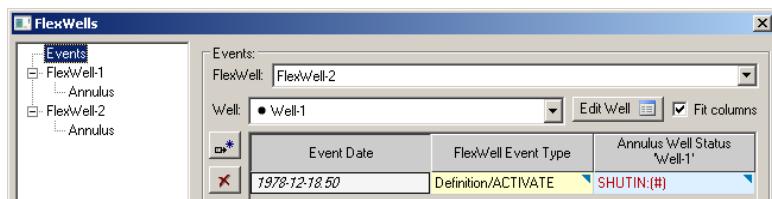
1. Create Well-1 as follows:



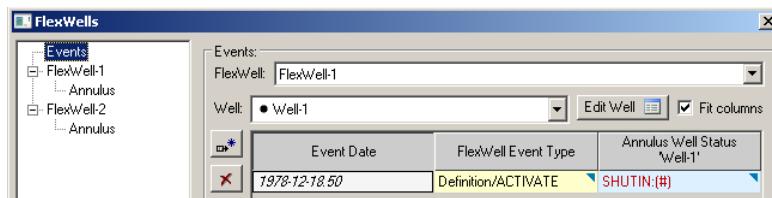
2. Create FlexWell-1 and assign Well-1 to its annulus, as shown in the following example:



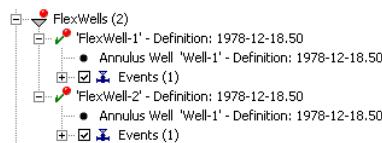
3. Create FlexWell-2 and assign Well-1 to its annulus, as shown in the following example:



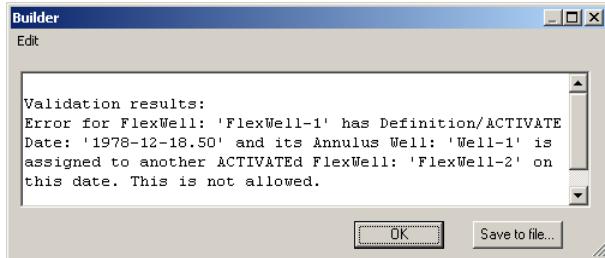
As shown, **SHUTIN:(#!)** displayed in normal red indicates that Well-1 is assigned to multiple active FlexWells on this date. If we open the FlexWell-1 **Events** table, it will now show the same:



4. The FlexWells area of the **Wells & Recurrent** section of the Builder tree view will be displayed as follows, indicating there are errors with both FlexWell-1 and FlexWell-2:

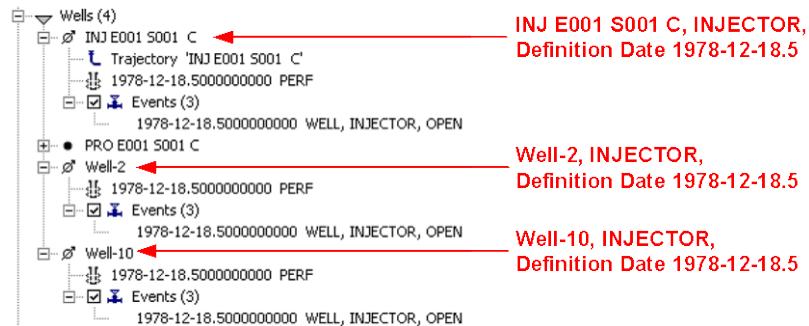


5. Right-click **FlexWell-1** and then select **Validate**. The following error message will be displayed:

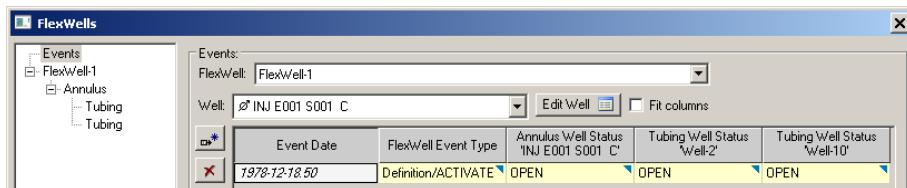


Example 6: Well can be used by more than one tubing string at different times (i.e., when there is no overlap).

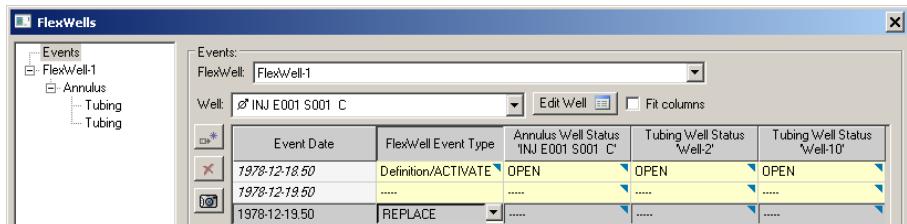
1. Create wells Well-2, Well-10 and INJ E001 S001 C, as follows:



2. Create a FlexWell (FlexWell-1) and assign a well to its annulus (INJ E001 S001 C in the following example). Add two tubing strings and assign wells to both (Well-2 and Well-10 in the following example):



2. Add an event date, then select REPLACE in the **FlexWell Event Type** column:



- Replace tubing Well-2 with a new well, in our example, Well-11. Then replace tubing Well-10 with Well-2. You will need to SHUTIN Well-10 before the REPLACE. The result is shown below:

| Event Date | FlexWell Event Type | Annulus Well Status | Tubing Well Status Well-2 | Tubing Well Status Well-10 |
|---------------|---------------------|---------------------|---------------------------|----------------------------|
| 1978-12-18 50 | Definition/ACTIVATE | OPEN | OPEN | OPEN |
| 1978-12-19 50 | | | | SHUTIN |
| 1978-12-19 50 | REPLACE | | Well-11 | Well-2 |
| 1978-12-19 50 | | | OPEN* | (OPEN) |

As shown above, Well-2 does not need to be SHUTIN after it is replaced by Well-11 because it is being used as a replacement well for another tubing string on the same date. The procedure outlined in this example can also be used to:

- Replace the well assigned to the annulus with a well that was previously assigned to a tubing string.
- Replace the well assigned to a tubing string with one that was previously assigned to the annulus.

The only constraints are:

- A well cannot be used in more than one place at the same time; for example, it cannot be assigned to two tubing strings at the same time.
- The completion branch of the replaced well must be the same as the completion branch of the replacing well.

Example 7: Demonstrate use of ‘SHUTIN until used’.

- Create a FlexWell, FlexWell-1, and assign a well to it, Well-1:

| Event Date | FlexWell Event Type | Annulus Well Status |
|---------------|---------------------|---------------------|
| 1978-12-18 50 | Definition/ACTIVATE | Well-1 |

- Add several ACTIVATE and DE-ACTIVATE events, and OPEN/SHUTIN as required. On the final DE-ACTIVATE event, set the Annulus Well Status to OPEN:

| Event Date | FlexWell Event Type | Annulus Well Status 'Well-1' |
|---------------|---------------------|------------------------------|
| 1978-12-18.50 | Definition/ACTIVATE | SHUTIN |
| 1978-12-19.50 | DE-ACTIVATE | SHUTIN |
| 1978-12-20.50 | ACTIVATE | OPEN |
| 1978-12-21.50 | DE-ACTIVATE | SHUTIN |
| 1978-12-22.50 | ACTIVATE | OPEN |
| 1978-12-23.50 | DE-ACTIVATE | OPEN |

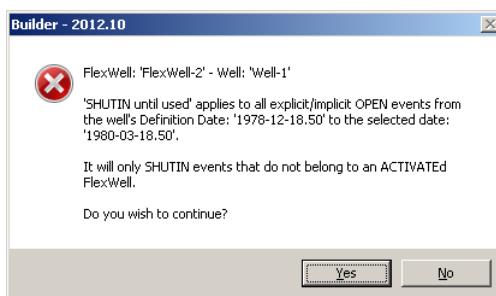
As shown, the well status on the final DE-ACTIVATE event is shown in bold red, indicating that the well is not assigned to any other FlexWell, and must be SHUTIN on this date.

3. Create a second FlexWell, FlexWell-2, with a definition date that is later than the last DE-ACTIVATE date for FlexWell-1, and assign Well-1 to it:

| Event Date | FlexWell Event Type | Annulus Well Status 'Well-1' |
|---------------|---------------------|------------------------------|
| 1980-03-18.50 | Definition/ACTIVATE | (OPEN):(<) |

As shown, the status of Well-1 is **(OPEN):(<)**, indicating that Well-1 is OPEN before it is used by FlexWell-2.

4. In the FlexWell-2 Events table, change the status of Well-1 to “SHUTIN until used”. The following message will be displayed, informing you that there are other Well-1 OPEN events besides the OPEN event on 1980-03-18.50 that need to be SHUTIN:



5. Click **Yes** to continue. The FlexWell-2 Events table will change as follows, indicating that Well-1 has been SHUTIN before the FlexWell-2 definition date.

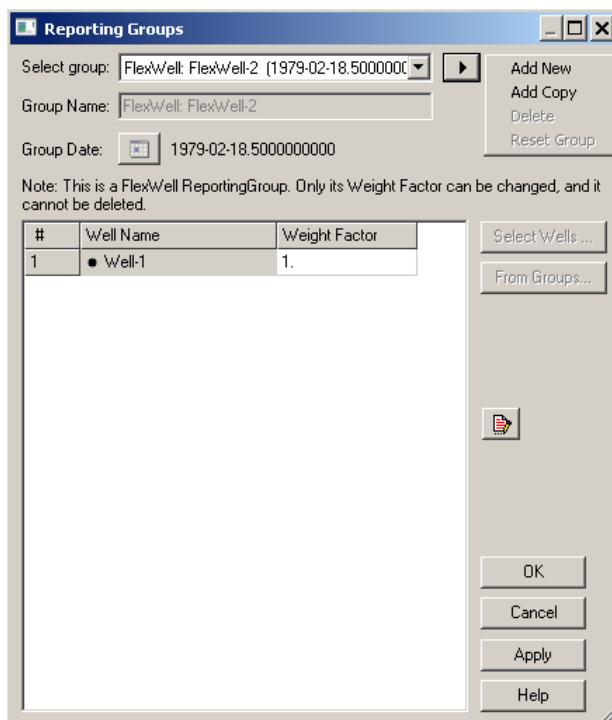
| Event Date | FlexWell Event Type | Annulus Well Status 'Well-1' |
|---------------|---------------------|------------------------------|
| 1980-03-18.50 | Definition/ACTIVATE | (SHUTIN) |

The Events table for FlexWell-1 will also have changed, as follows, indicating Well-1 is now SHUTIN on 1978-12-23.50. The status of other events in the table has not changed:

| Event Date | FlexWell Event Type | Annulus Well Status 'Well-1' |
|---------------|---------------------|---------------------------------|
| 1978-12-18 50 | Definition/ACTIVATE | SHUTIN |
| 1978-12-19 50 | DE-ACTIVATE | SHUTIN |
| 1978-12-20 50 | ACTIVATE | OPEN |
| 1978-12-21 50 | DE-ACTIVATE | SHUTIN |
| 1978-12-22 50 | ACTIVATE | OPEN |
| 1978-12-23 50 | DE-ACTIVATE | SHUTIN |

Managing FlexWell Reporting Groups

After a new FlexWell is created, click **OK** on the **FlexWells** dialog box to automatically create a new reporting group for it. The new reporting group is automatically named after the FlexWell to which it belongs. You cannot directly modify a FlexWell reporting group name, date, or list of assigned wells using the **Reporting Groups** dialog box, which you can access through **Well | Reporting Groups**. You can only modify this data using the **FlexWells** dialog box. After a FlexWell is modified, its reporting group is automatically updated with the modified data. An example of a FlexWell **Reporting Groups** dialog box is shown below.



For more information about reporting groups, refer to [Reporting Groups](#).

Input/Output Control

Overview

The Input/Output (I/O) Control section controls a number of aspects of simulator data: Titles and Case ID, Run Time Dimensioning, Restart, Simulation Results Output, Text Output, and Miscellaneous. While all of these can be set to default values, you should, as a minimum, enter simulation run titles and a case id.

Normally, you will not need to enter anything in the Run Time Dimensioning section; however, if your simulation run terminates with a report of a dimensioning error, you may need to enter dimensioning values to override the simulator defaults.

Restart runs are used to break a simulation run into a sequence of (shorter) simulation runs. For example, you could run one simulation for the history portion of a simulation, and then run several forecast runs, each for a different development scenario, without having to repeat the simulation of the historical period.

There are many options for controlling the information that is saved from the simulation run. Saving all information leads to very large simulation results files which may fill large hard drives. The Simulation Results Output section allows you to choose the appropriate variables to output to the SR2 file. The data in SR2 files may be viewed and analyzed using CMG Results 3D and Graph.

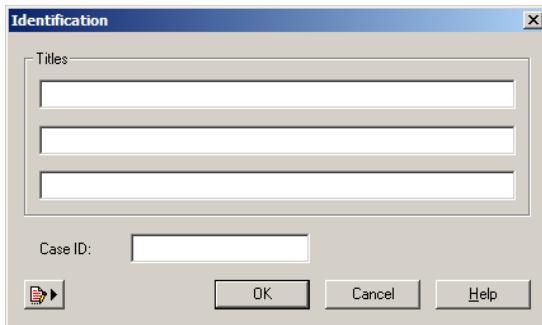
The Text Output section has controls for the variable and information output to the ASCII output and log files. These files may be opened and read in a text editor.

The Miscellaneous section has a few controls that don't fit in any of the other sections.

Titles and Case ID

To bring up the **Identification** dialog box, click on the **I/O Control** button above the tree view, then double-click **Titles and Case ID** on the tree view. Alternatively, select **Titles and Case ID** from the **IO Control** menu.

The **Identification** dialog box will be displayed:

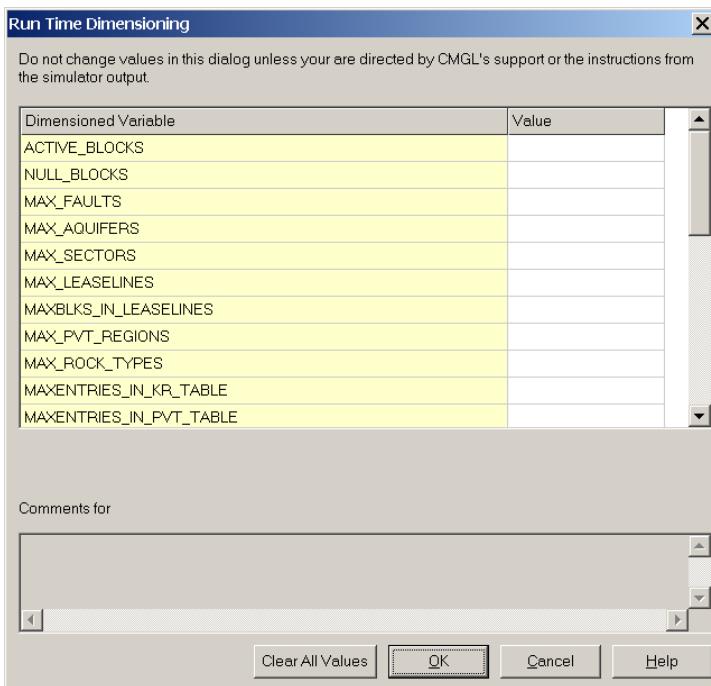


In this dialog box, enter text in the text entry fields. Some the fields are limited in the number of characters that they can accept – when the limit is reached, characters entered will no longer be shown in the text entry field. When you have finished the text entry, click **OK** to accept your changes, or **Cancel**.

Run Time Dimensioning

The **Run Time Dimensioning** dialog box should only be used if you are an experienced simulator user who knows how to read the dimensioning information in the simulator ASCII output, or if you are directed to enter values in the dialog box by CMG support personnel.

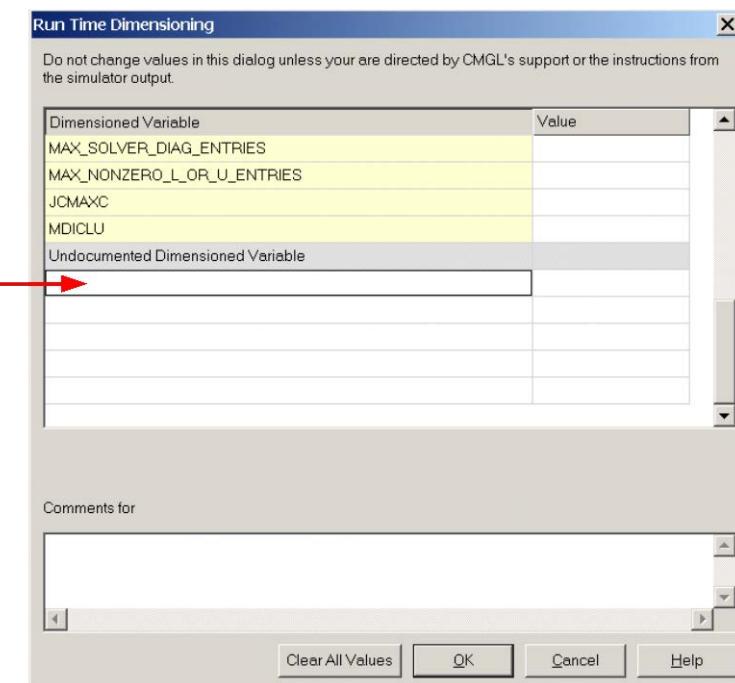
To open the **Run Time Dimensioning** dialog box, select **IO Control | Run Time Dimensioning**. The **Run Time Dimensioning** dialog box will be displayed:



Find the row of the dimension that needs to be changed, and enter a numerical value in the **Value** column.

If you get Runtime Dimensioning errors when you run the simulator, the simulator also suggests values that you need to input for the above keywords. You can view these suggestions in the .out or .log files generated by the simulator. If you have to run initialization using the **Validate/Run Simulator** dialog box, you can view the suggestions in this dialog box also.

The simulator may suggest that you enter a specific keyword – value pairs that are not documented in the manual. You can enter these in the lower part of the grid in the above dialog box, below the **Undocumented Dimensioned Variables** row:



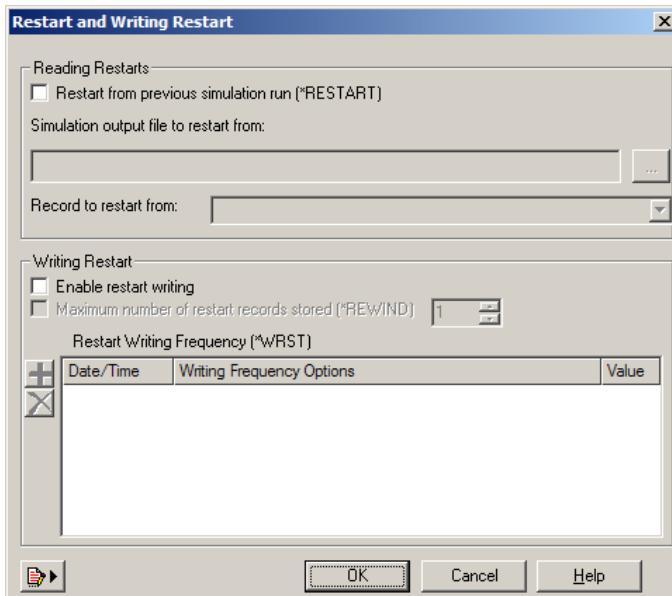
These pairs are preserved in the dataset. You can also edit previously entered values.

Restart

Restarts are used to break a simulation run into two or more shorted simulations. There are two aspects to Restart control:

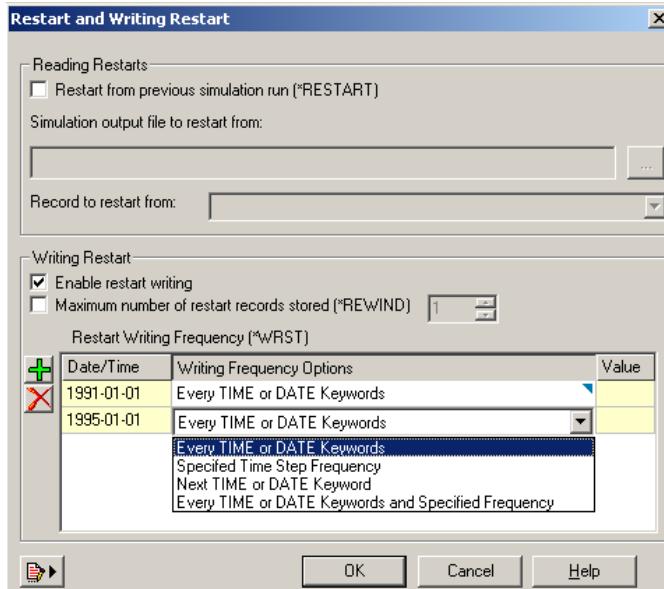
- Writing restart information to a file (so you can start a later run from this run)
 - Reading a restart file that was written by earlier simulation runs.

Open the **Restart and Writing Restart** dialog box by selecting **IO Control | Restart** through the Builder menu.



The **Restart and Writing Restart** dialog box has two sections. Through the **Reading Restarts** section you can restart a simulation run from a point in a simulation run that was completed previously. The **Writing Restart** section controls writing information to a restart file so you can later restart from this simulation run.

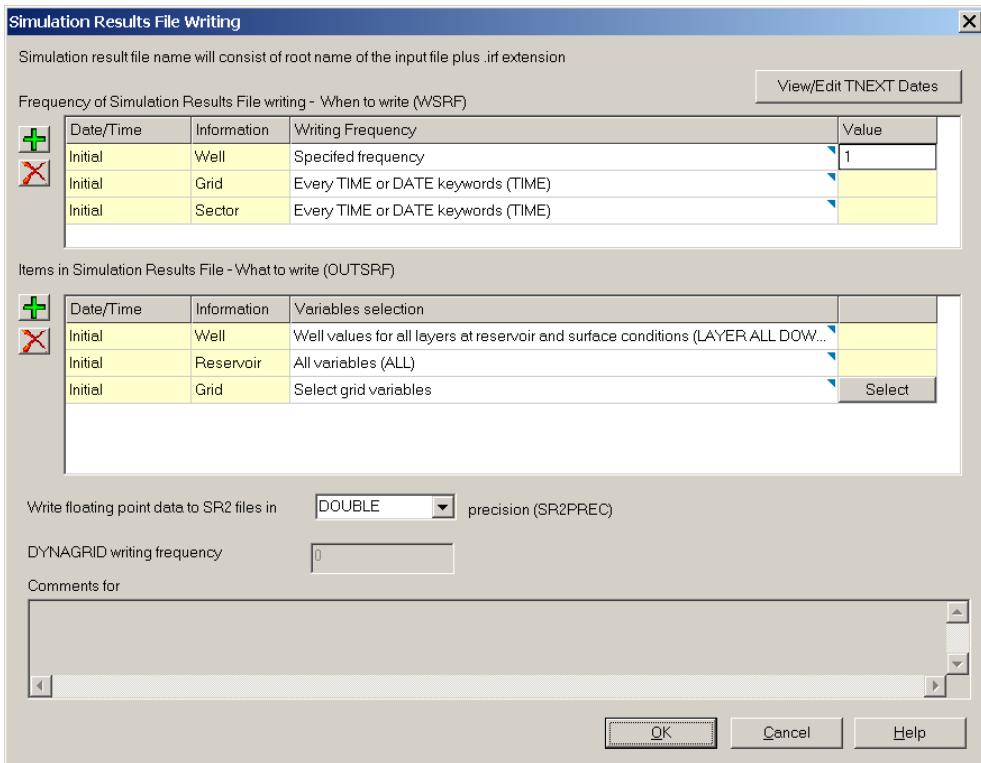
To write restart information from a simulation run, select **Enable restart writing**. And then click to add a restart writing control. You will be prompted to enter a date or time when the control is activated. The control will be added to the list. There are a number of options for controlling how frequently a restart record is written.



After you have run a simulation where restart records are written, you may restart a subsequent simulation from the restart. This is controlled by the **Reading Restarts** section of the dialog box. Select **Restart from previous simulation run** and then browse to select the file containing the restart records from the previous run. After the file is opened, the list of available restart time/dates will be filled in. Select the time/date from which you want to restart.

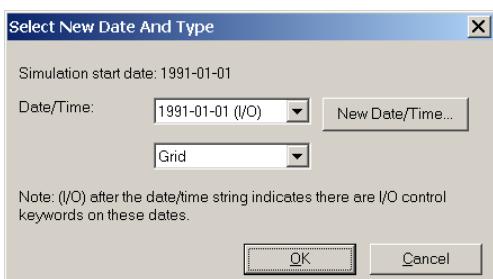
Simulation Results Output

The simulation results file is used to control the information that is written to the SR2 file, for later view and analysis with CMG Results 3D and Results Graph. The SR2 file is actually a pair of files with the same root file name and the file extensions *.irf and *.mrf. To open the **Simulation Results Output** dialog box, click **I/O Control** in the tree view then double-click **Simulation Results Output**, or select **Simulation Results Output** from the **IO Control** menu:



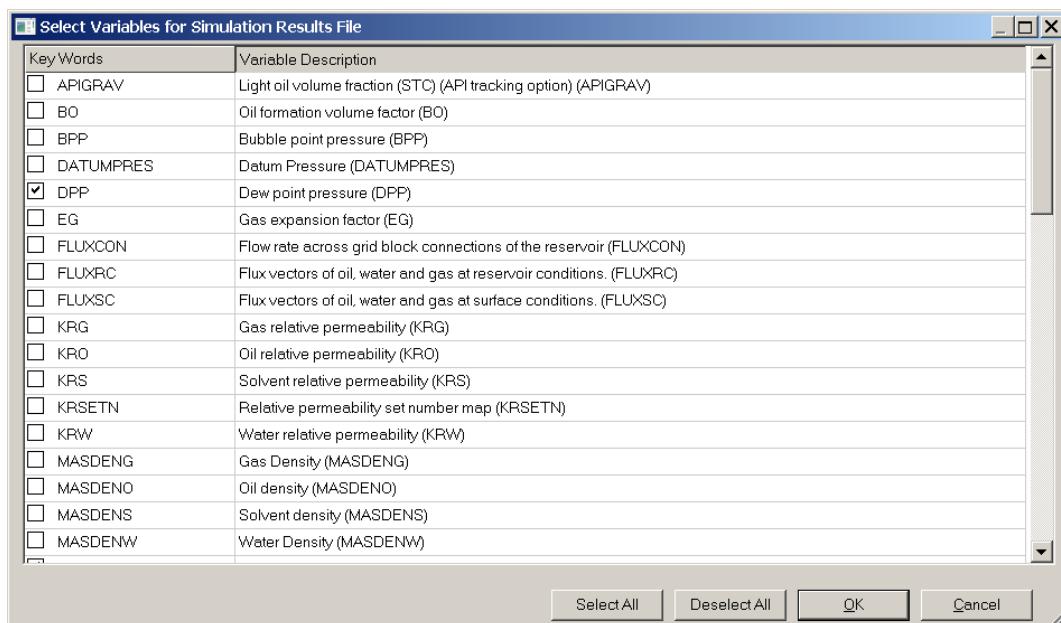
The **Simulation Results File Writing** dialog box is divided into two parts. The **Frequency of Simulation Results File Writing** section controls how often, in simulation time, different types of information are written. The **Items in Simulation Results Files** controls what is written. Each control has a simulation Date/Time when it applies, and the control remains in effect until it is overwritten by a later control. The Date/Time when the control comes into effect is indicated in the **Date/Time** column of the control.

To add a control, click on the left of one of the two grid controls to start the **Select New Date And Type** dialog box:



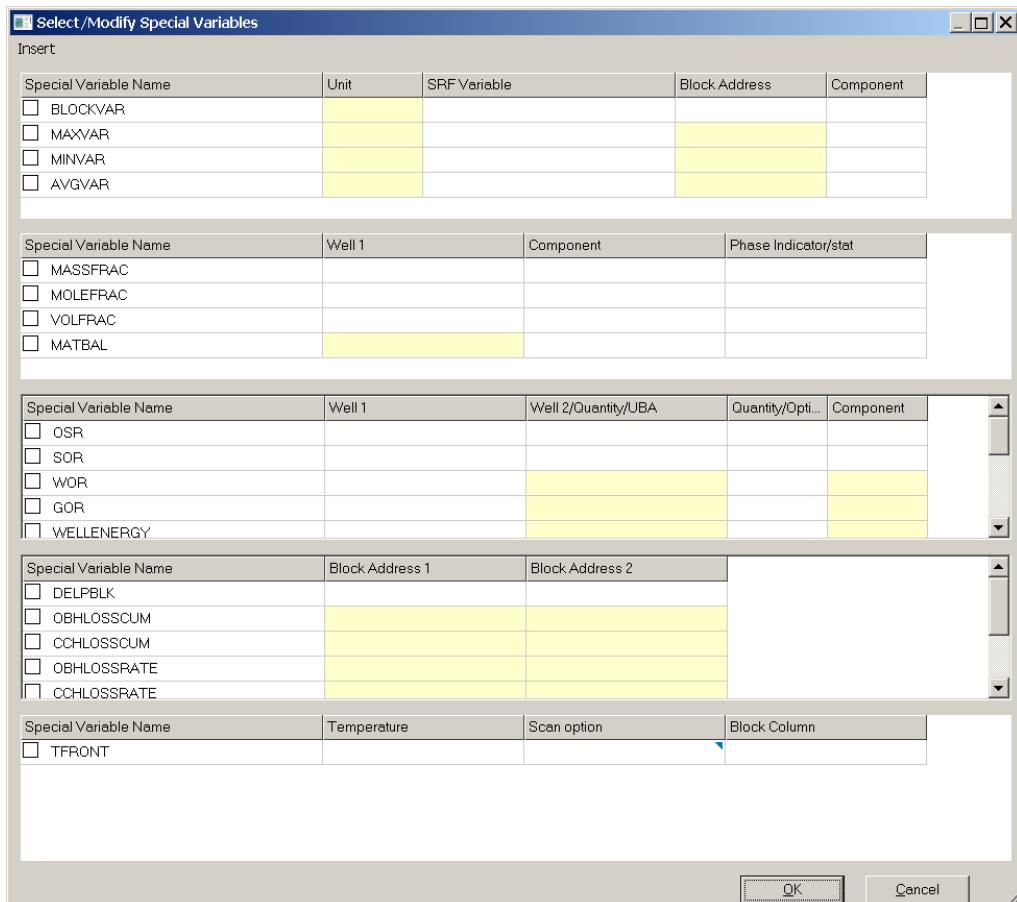
Through this dialog box, you need to select the date and time when the control is activated and the information type that the control applies to. The selectable options depend on the simulator types. They include reservoir properties, grid properties, well injection/production, FlexWell properties, sector properties, flux-sector properties, lease plane, and special history. The **Select New Date And Type** dialog box is slightly different for different simulators.

After you select the Date/Time and the Information Type, and click **OK**, a new control will be added to the list. Depending on the **Information Type**, you can select from drop-down lists in the grid control, or click on a **Select** button to open the **Select Variables for Simulation Results File** dialog box:

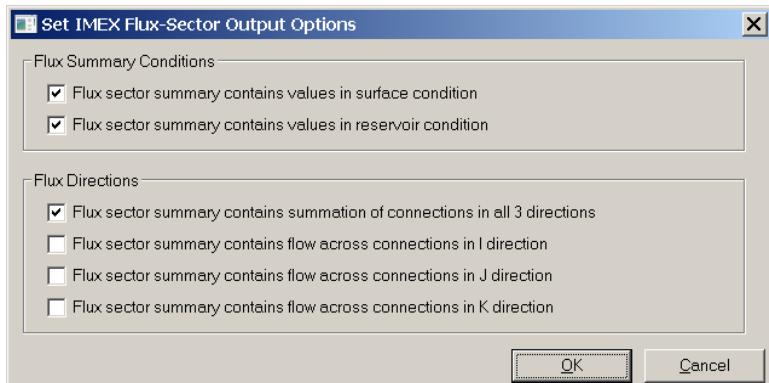


From the **Select Variables for Simulation Results Files** dialog box, you can select or clear the properties being outputted. The dialog box will be slightly different for different simulators and/or properties.

If you are viewing a STARS dataset and you click **Select** beside a property of type *Special* in the **Simulation Results File Writing** dialog box, the **Select/Modify Special Variables** dialog box will be displayed:



If you are viewing an IMEX dataset and you click **Option** beside a property of type *Flux-Sector* in the **Simulation Results File Writing** dialog box, the **Set IMEX Flux-Sector Output Options** dialog box will be displayed:



To delete a control from the **Simulation Results File Writing** dialog box, click on the control and then click to the left of the grid of controls.

Once you have finished adding or removing controls in the dialog box, click **OK** to apply the changes you have made. If you do not wish to retain the changes, click **Cancel**.

Text Output

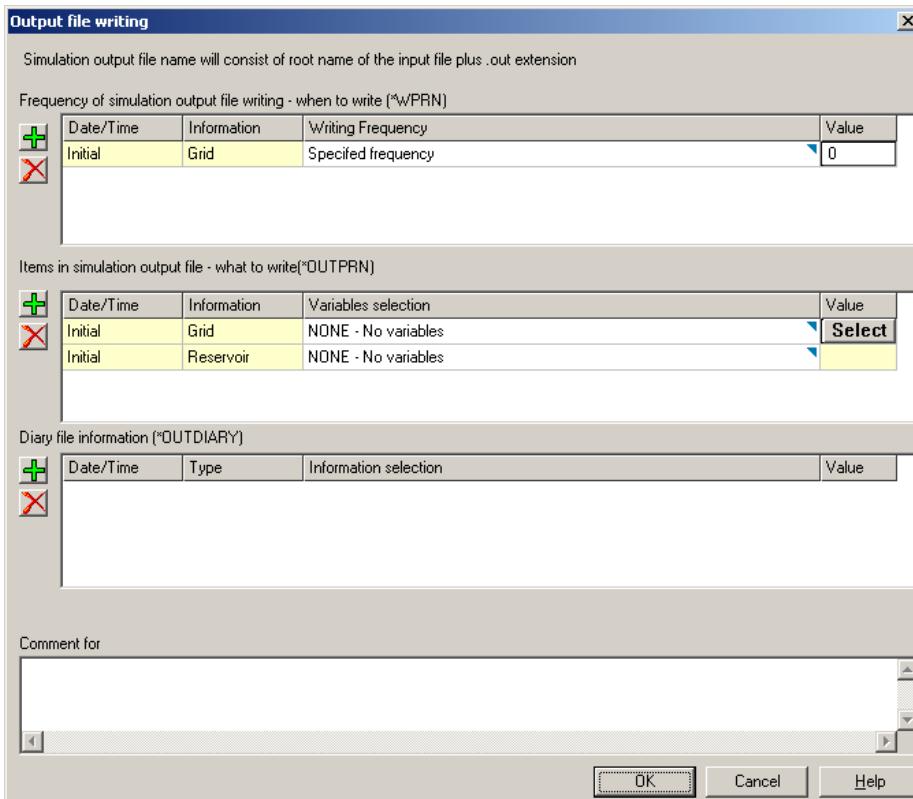
In addition to the binary information output in the SR2 file (described in [Simulation Results Output](#) above, and used by Results 3D and Results Graph), the simulators can output information to text (ASCII) files. There are two main text files produced by the simulators:

- a log (or diary) file, showing a small amount of information indicating the progress of the simulation run
- *.out file, which can contain a large amount of information.

If the simulation is run using Launcher, the diary file can be conveniently viewed using Launcher. Information output to text files can be read in a text editor program, or printed out.

You can control how much information is output to text files; for example, when running a large simulation (several hundred thousand or more grid blocks), text output of grid variables, such as oil saturation, are of limited usefulness. This information is more usefully viewed in Results 3D, and a selected property and time may be exported from Results 3D into text files if required. However, some other information, like debugging information on numerical method convergence, is only available in the text output file.

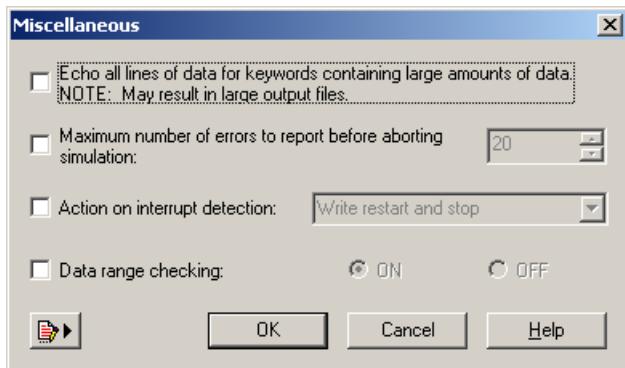
To change the frequency and types of information being written to the text output file or the diary file, click **I/O Control** in the tree view then double-click **Text Output**, or select **Text Output** from the **IO Control** menu. The **Output file writing** dialog box will be displayed:



The **Output file writing** dialog box works similarly as the **Simulation Results File** dialog box. Controls are added or removed using the or buttons to the left of the grid controls. When a control is added, you are required to enter the Date/Time at which the control becomes active, and the Information Type. Once the control is in the grid, you can select different options from drop-down lists or by clicking on a **Select** button, then selecting properties to be output from a list.

Miscellaneous Options

The **Miscellaneous** dialog box in the I/O Control section controls a number of miscellaneous options. This dialog box is opened by double-clicking **Miscellaneous** under **I/O Control** in the tree view, or selecting **Miscellaneous** from the **IO Control** menu. The **Miscellaneous** dialog box will be displayed:



Help information on any of the options can be obtained by clicking on the option, and then pressing the F1 key. The simulator help will be displayed opened to the appropriate section.

Numerical Control

Overview

Normally, you do not need to enter any values in the Numerical Control section, as defaults are supplied for all values. It is recommended that you only override the defaults if you are an experienced reservoir simulation user, or directed to do so by CMG support personnel.

Numerical controls are provided in three subsections: time step control, solution method control, and linear solver control. For STARS you can define multiple sets of time step and solution method control parameters. Builder provides an interface for setting these controls. Detailed information on each control keyword is provided by the simulator user manuals. Selecting a control and pressing the F1 key will open the simulator user manual at the appropriate page.

Setting and Editing Numerical Controls (IMEX & GEM)

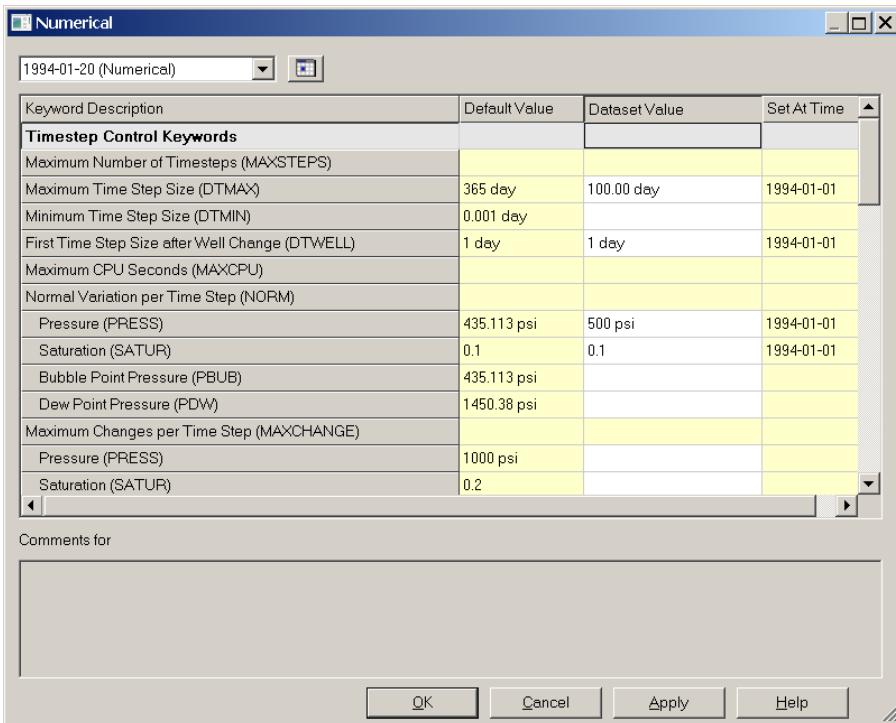
To access the **Numerical** dialog box, use one of the following methods:

- Click on the **Numerical** button in the tree view then double-click the numerical control of interest
- Click on the **Numerical** button in the tree view then click the desired option



- Select the desired option from the **Numerical** menu.

The **Numerical** dialog box will be displayed, with the selected control or section highlighted, for example:



To understand more about a particular control keyword, select the keyword by clicking on it, and then press F1. This action should open the simulator help to the appropriate keyword.
(CONFIRM)

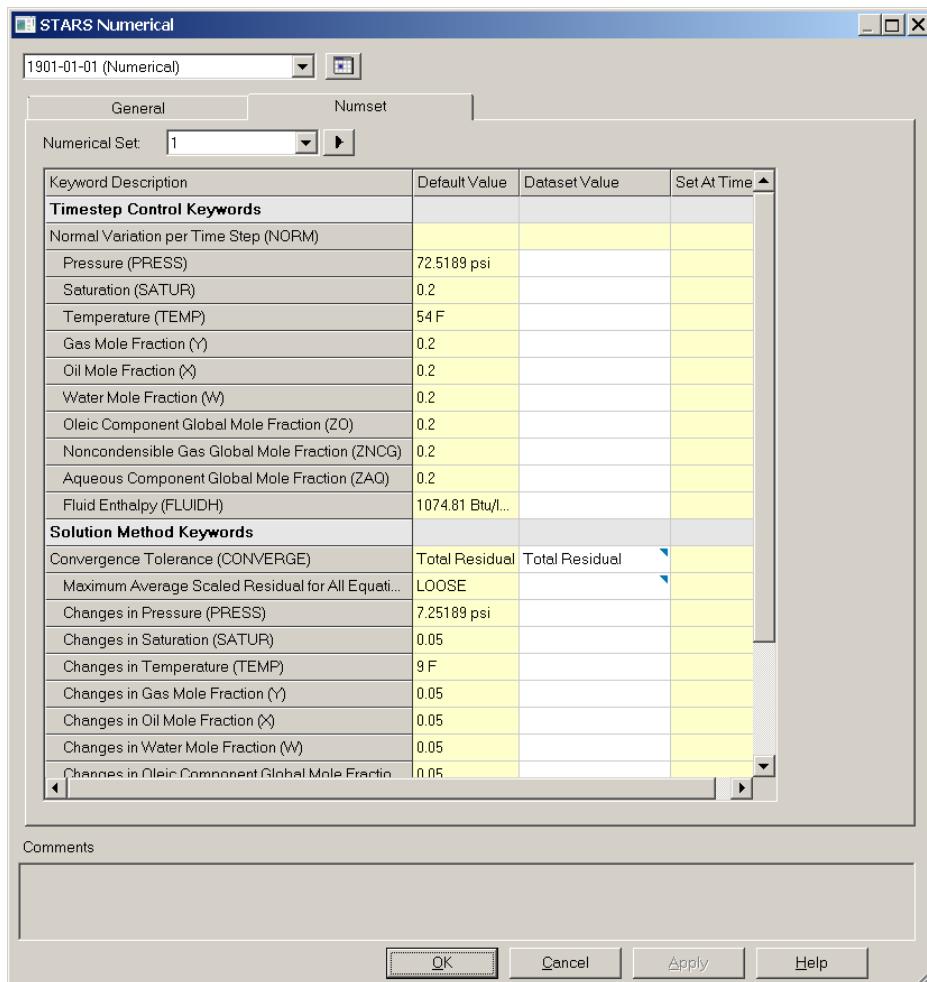
Some control keywords are allowed to be set once, in the initial section or at the simulation start date/time. Other keywords are allowed to be set or changed in the recurrent data section. Initially, the **Date/Time** in the **Numerical** dialog box will be set to the simulation start date (time=0). You can add dates and times by clicking the calendar icon next to the **Date/Time** selection box. If a control keyword cannot be changed after the initial date/time, then the grid control in the **Dataset Value** column will be set to yellow indicating that it cannot be edited if the Date/Time is not the simulation start time. The **Set At Time** column indicates if a numerical control has been set to a non-default value at a Date/Time that is earlier than the Date/Time currently selected.

Setting and Editing Numerical Controls (STARS)

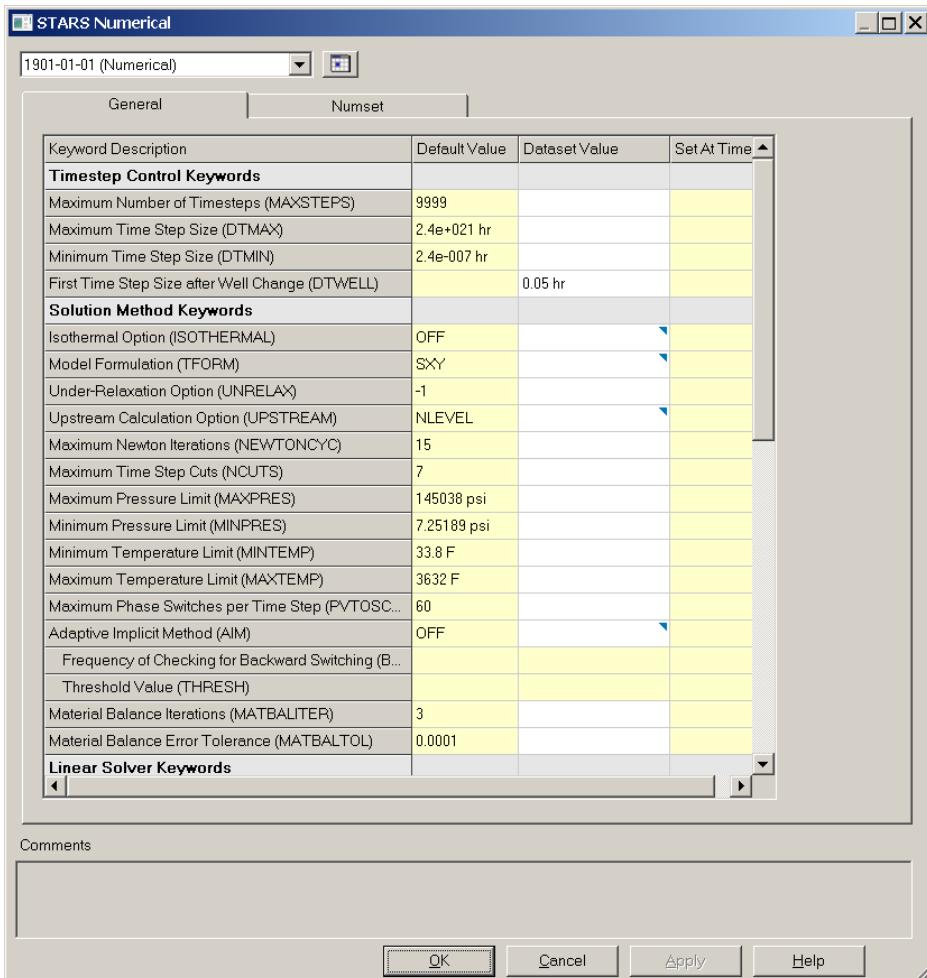
For the STARS simulator, you can configure multiple sets of time control and/or solution method parameters. Refer to the *STARS User's Guide* under the title, NUMSET, for details. To open the **STARS Numerical** dialog box, click on the **Numerical** button in the tree view then double-click on the numerical control of interest, or select one of the options from the **Numerical** menu.

The **STARS Numerical** dialog box will be displayed, with the selected control or section highlighted.

From the **Numset** tab, shown below, you can modify these time control and/or solution method parameters for a defined NUMSET, which is selected via the NUMSET selection box. To create a new NUMSET, copy an existing one as a new one, or delete an existing NUMSET, click the arrow button next to the **Numerical Set** selection box and select the corresponding context menu item to proceed.



From the **General** tab you can configure any other numerical control parameters:

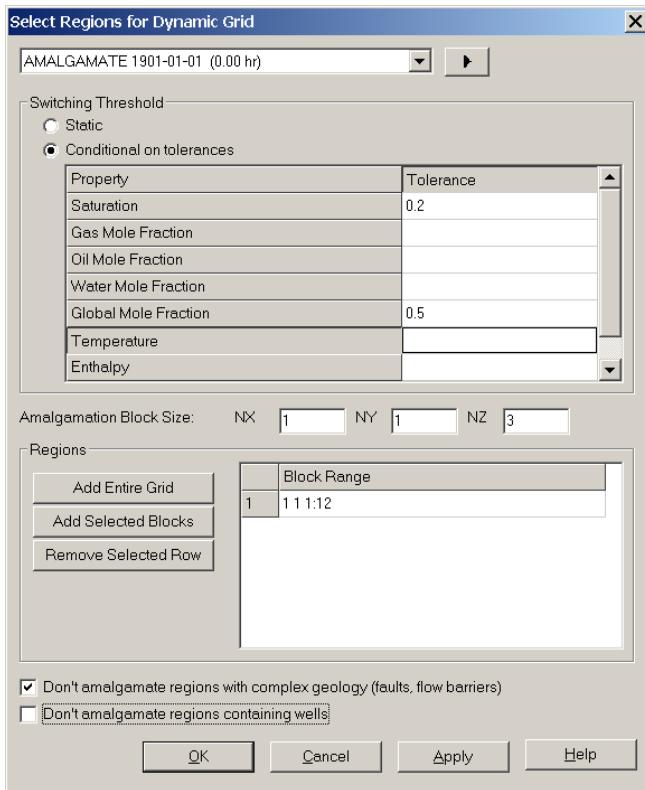


To understand more about a particular control keyword, select the keyword by clicking on it in the grid, then press F1. This action should start the simulator help, opening it to the appropriate keyword. (CONFIRM)

Some control keywords can only be set once, in the initial section or at the simulation start date/time. Other keywords are allowed to be set or changed in recurrent data. Initially, the Date/Time in the **Numerical** dialog box will be set to the simulation start date (time=0). You can add dates and times by clicking the calendar icon next to the **Date/Time** selection box. If a control keyword cannot be changed after the initial date/time, then the grid control in the **Dataset Value** column will be set to yellow indicating that it cannot be edited if the Date/Time is not the simulation start time. The **Set At Time** column indicates if a numerical control has been set to a non-default value at a Date/Time that is earlier than the Date/Time currently selected.

Dynamic Grid Regions

The **Select Regions for Dynamic Grid** dialog box can be accessed by selecting **Dynamic Grid Regions** either from the **Numerical** menu or through **Numerical** in the tree view. Through this dialog box, you can amalgamate, de-amalgamate, de-refine or re-refine regions of a reservoir at any time steps. The basic idea is to group blocks with similar property values into coarser blocks to reduce the computation time for the simulation. Refer to the simulator user manual for the four sub keywords handled in this dialog box – **AMALGAMATE**, **DEREFINE**, **DEAMALGAMATE** and **REREFINE**. An example of the **Select Regions for Dynamic Grid** dialog box is shown below:



The combo box at the top displays the current action keyword appended by the date of the action. If multiple keywords have the same action and date, then the duplicated ones are appended by a sequence number, for example, (1) and (2). Using the arrow button to the right, you can add a new keyword at a certain date, delete an existing keyword or copy the current keyword to a new date.

The conditions for amalgamation or de-refine can be static or dynamic. ‘static’ means the amalgamation or de-refine will take place at the specified time step, while ‘dynamic’ means the action will happen if the property values (for example, temperature or mole fractions) are within the specified tolerance.

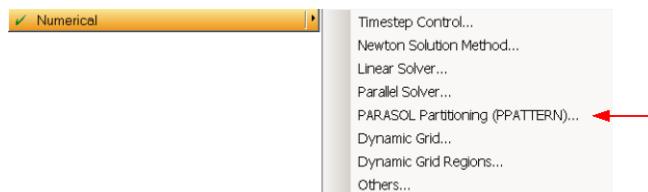
Amalgamation Block Size allows you to specify how many cell blocks ($NX*NY*NZ$) in the selected regions will be grouped into a single block. You can specify the selected regions by entering the block ranges in the table, or by dragging and highlighting a range of blocks in the reservoir view then clicking **Add Selected Blocks**. The two options (checkboxes) at the bottom correspond to EVEN-IF-CMPLX and EVEN-IF-WELL sub-keywords of the DYNAGRID keyword. Once you have finished entering the desired information, click **Apply** to save the changes.

PARASOL Class Partitioning Pattern (PPATTERN)

To enter PPATTERN information, the SOLVER keyword must first be set to PARASOL through the **General** tab of the **Numerical** dialog box, accessible through **Numerical | Timestep Control:**

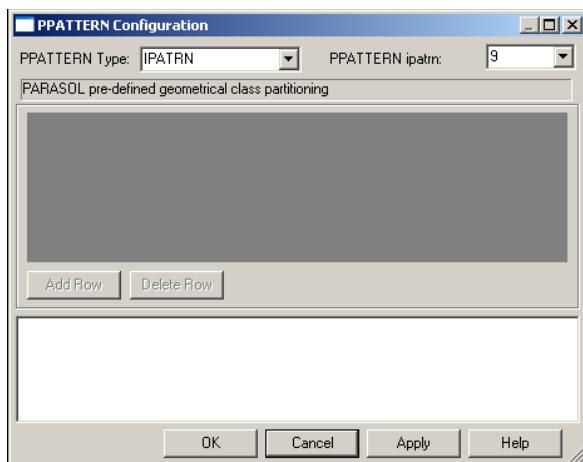
| Parallel Processing Keywords | |
|--|----------------|
| Target number of planes per Jacobian domain (DPL...) | 4 |
| Number of threads to be used (PNTHRDS) | 2 |
| AIMSOL/PARASOL Switch (SOLVER) | AIMSOL PARASOL |

PPATTERN is now available through the Builder menu or tree view:



PPATTERN sets the basic partitioning of the reservoir into non-connected regions and separators that make possible the parallelization of the linear solution. Refer to the appropriate simulator user manual for the six sub keywords handled in this dialog box – IPATRN, AUTOPSLAB, PARTITION, APARTITION, GPARTITION and PPARTITION.

An example of the **PPATTERN Configuration** dialog box is shown below:



Geomechanics – GEM and STARS

Overview

Currently, geomechanics is used only by GEM and STARS. Builder will read, validate and modify geomechanics data in a GEM or STARS dataset.

Using Builder, you can enter and modify:

- Calculation and Numerical Options
- Geomechanical Rock Types
- Boundary Conditions

To get an overview of available geomechanic options, click the **Geomechanics** section in the Builder tree view, and then right-click on the tree view. Select **Expand tree** from the pop-up menu. All of the tree view elements under **Geomechanics** will be displayed.

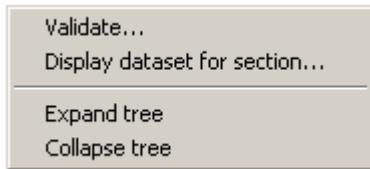
Builder provides several interfaces for modifying geomechanics information. These interfaces are accessible by:

- Selecting **Geomechanics** in the Builder main menu.
- Clicking on the tree view button labeled **Geomechanics** and then clicking on the **Right Arrow**  button.
- Double-clicking on the appropriate tree view item.

The current status of geomechanics information is indicated by the icons next to the elements in the tree view. There are four possible states for a tree view item:

- Valid : Information has been entered for that option and no errors have been found.
- Warning : Information has been entered for that option but the information entered is not recommended and may lead to incorrect results.
- Error : Information is either missing or the information that has been entered is incorrect. The simulator will not accept this data or incorrect results will be obtained.
- No data: No icon next to the tree item means that no data has been entered for that option.

Right-clicking on the Geomechanics tree view section will open the following menu:



Validate

Displays a list of all warning and error messages.

Display dataset for section

Displays the Geomechanical information as it will be stored in the dataset.

Expand/Collapse tree

Expands and collapses the branch for the items in the tree.

Double-clicking an item on the tree will open the relevant interface.

Data Validation

When a dataset is initially read in, any error encountered in this section will be noted by error or warning messages. As well, the corresponding status icon will be displayed on the **Geomechanics** button in the tree view. You can then enter the relevant interface and review the information. Required changes can then be made and saved.

The information entered in the different geomechanics interfaces is again validated when the **Apply** or **OK** buttons are selected. If there are any errors or warnings, a list of the messages will be displayed.

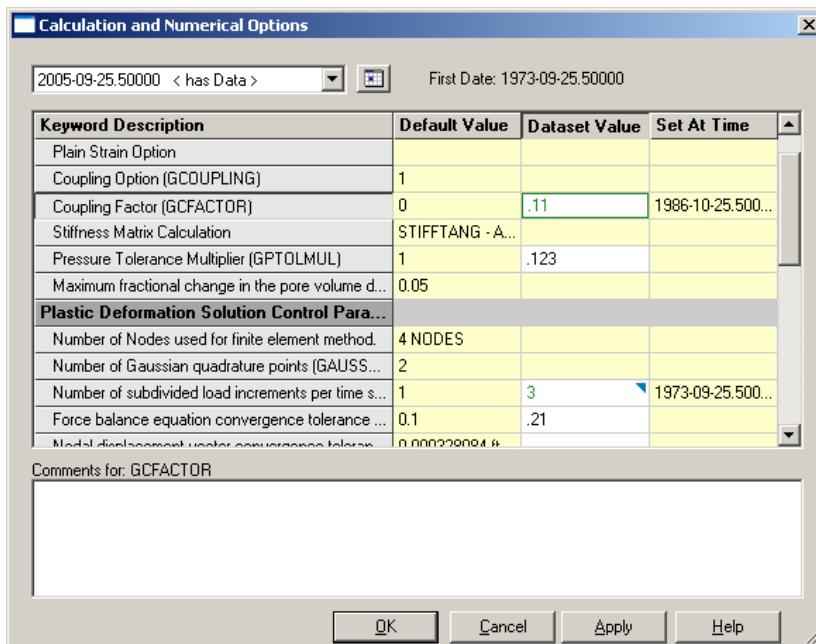
As well, basic validation is done when values are entered into edit boxes. For example, if a numerical value is expected but a non-numerical value is entered, an error message is displayed. Also, values entered will be displayed with their respective current working units. Values may be entered as a number only or as a value followed by a space then the units used. Values entered with units different than the current working units will be converted and the converted value will be displayed with the current working units.

Calculation and Numerical Options

The **Calculation and Numerical Options** dialog box allows you to modify parameters related to general calculation options, plastic deformation solution control parameters, and matrix solver control parameters.

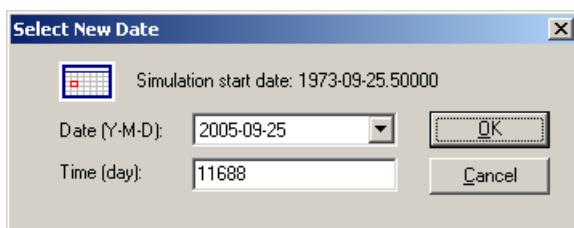
The **Calculation and Numerical Options** dialog box can be opened by selecting the main menu items **Geomechanics | Calculation Options** or **Geomechanics | Numerical Parameters**.

Alternatively, you can double-click one of the relevant geomechanics tree view items (that is, tree view items under **Calculation Options** or **Numerical**). Once you make the selection, the **Calculation and Numerical Options** dialog box will be displayed:



Since some of the parameters are date dependent, a date box is provided at the top left corner of the dialog box. This date box lists currently used dates. If calculation or numerical data has been entered for the date, this is indicated by the phrase “< has Data >”.

Next to the date box there is a **Date**  button which can be clicked to add new dates. Clicking the date button opens the **Select New Date** dialog box:



Once a date is chosen, you can view and edit the different parameters. The **Calculation and Numerical Options** dialog box has a grid table with four columns:

- **Keyword Description:** Lists the descriptions of the different parameters.
- **Default Value:** Default values used by the simulator. If you do not enter a value for a given parameter then the simulator will use the given default value.
- **Dataset Value:** Value entered for the given parameter. If the parameter has units, then entering a value with a given set of units will automatically be converted to the appropriate units.
- **Set At Time:** The last date at which a given parameter had a value entered for it. If a value for a parameter is not entered at the current selected date then the last date at which a value was entered for the parameter will appear in this field. As well, the last actual value entered will appear light green in the **Dataset Value** column. If you enter a value, the date in the **Set At Time** column will disappear, and the colour of the parameter value will be black.

Comments for the different parameters may also be viewed and modified by using the edit box provided at the bottom of the dialog box.

The **Calculation and Numerical Options** dialog box has four buttons at the bottom. The functions of these buttons are as follows:

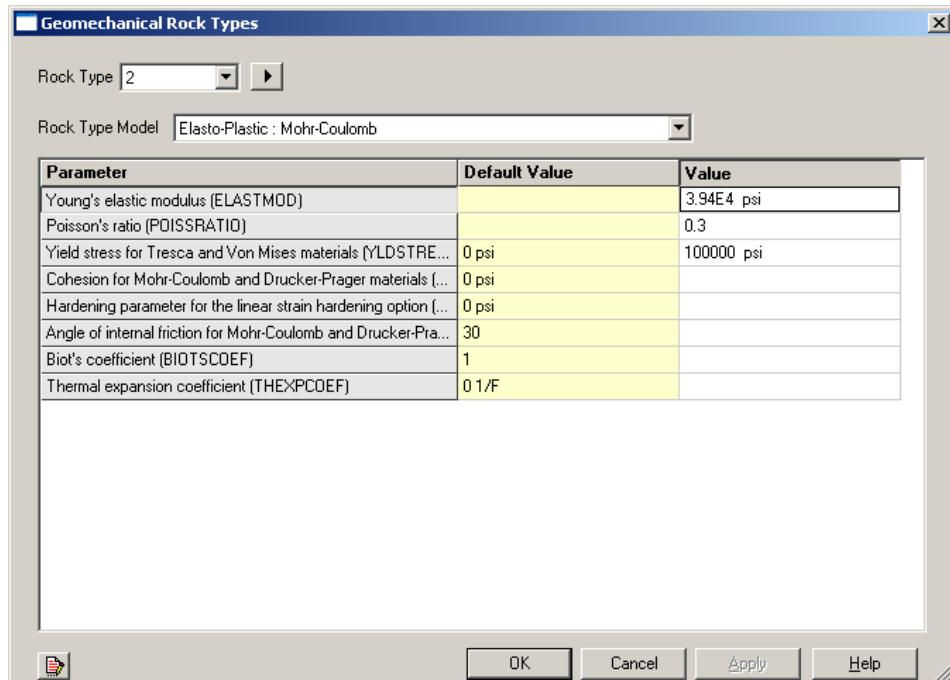
- **OK:** Closes the dialog box and saves any changes that were made.
- **Cancel:** Closes the dialog box but will not save any changes that were made unless **Apply** was clicked first.
- **Apply:** Saves any changes that were made but does not close the dialog box.
- **Help:** Displays help information about the active dialog box item. Pressing **F1** will also display this help information.

Geomechanical Rock Types

Through the **Geomechanical Rock Types** dialog box, you can define the geomechanical rock types that will be used in the simulation and enter the relevant parameters.

The **Geomechanical Rock Types** dialog box can be opened by selecting **Geomechanics | Geomechanical Rock Types** on the main Builder menu.

Alternatively, you can double-click one of the relevant tree view items (that is, tree view items such as **Geomechanical Rock Type** and **GeoRock 1**). Once you make the selection, the **Geomechanical Rock Types** dialog box will be displayed:



You can select the rock type information you want to view or edit in the **Rock Type** box at the top left of the dialog box. The rock type model for the selected rock type can then be selected from the **Rock Type Model** list.

Depending on the rock type model you select, a list of parameters will appear in the grid table. The grid table has three columns – the first column for the parameter description, the second column for default values used by the simulator, and the third column for the current values entered for that parameter.

You can view and edit the comments associated with a specific parameter, by first clicking the specific parameter and then clicking the **Comments**  button.

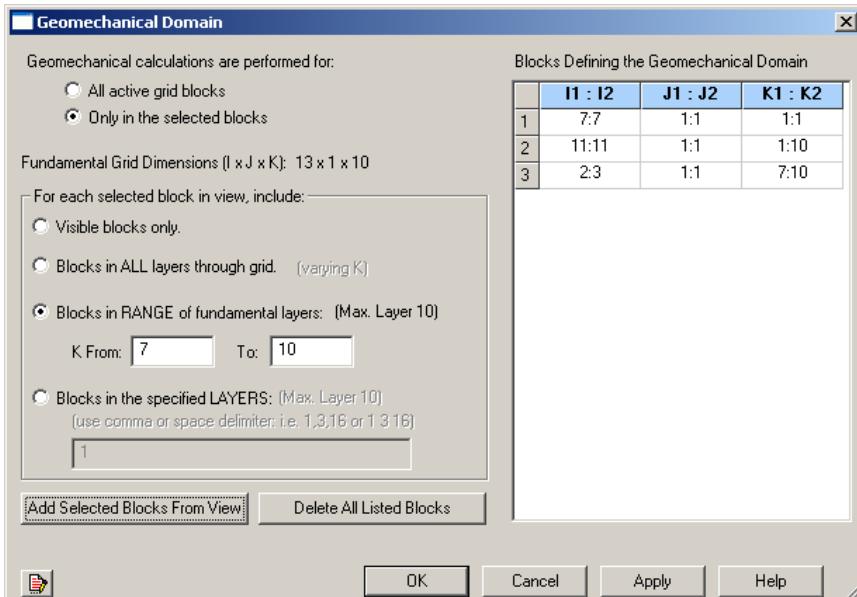
You can also view help information for a specific parameter by first clicking on the specific parameter and then clicking **Help** or pressing the **F1** key.

Geomechanical Domain

Through the **Geomechanical Domain** dialog box, you can define the blocks that will be used for the geomechanical calculations.

The **Geomechanical Domain** dialog box can be opened by selecting **Geomechanics | Geomechanical Domain** on the main Builder menu.

Alternatively, you can double-click **Geomechanics Domain** under **Boundary Conditions** in the tree view. Once you make the selection, the **Geomechanical Domain** dialog box will be displayed:



If you select **All active grid blocks**, then none of the other options are available and the dialog box controls are disabled.

If you select **Only in the selected blocks**, then the other options in the dialog box are made available and selected grid blocks can be specified.

The block selection can be made by either entering the block values manually in the grid table or by selecting the grid blocks using the mouse to highlight them in the grid.

If you select the grid blocks using the mouse, you can then choose one of the options listed under **For each selected block in view, include:**. The option you select will then be applied to all selected blocks. Once you have selected the blocks using the mouse, you can click **Add Selected Blocks From View**, which will add the selected block after applying the specified option to the grid table.

You can clear the table by clicking **Delete All Listed Blocks**. You can view or edit comments associated with the **Geomechanical Domain**, by first clicking the specific parameter and then clicking the **Comments**  button.

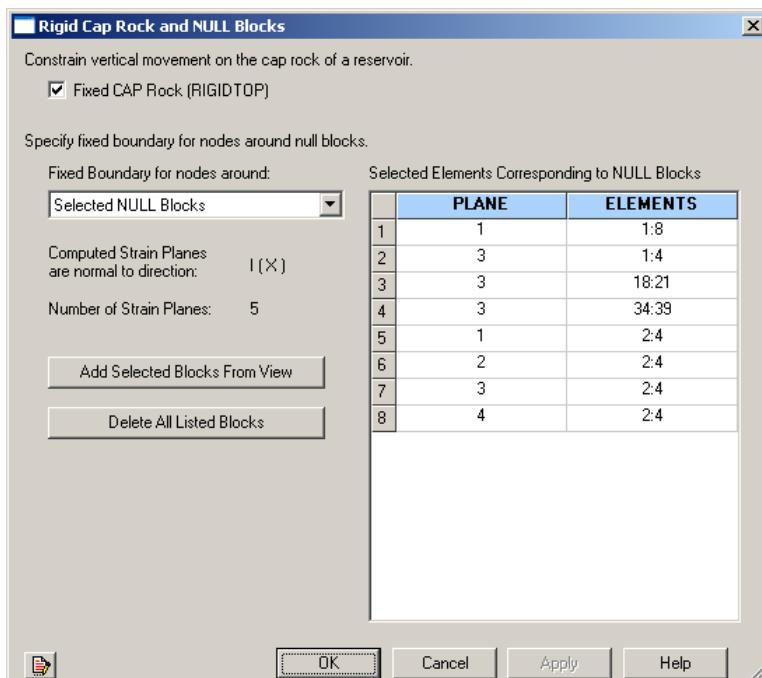
You can also view help information for the **Geomechanical Domain**, by clicking **Help** or by pressing the **F1** key.

Rigid NULL Blocks and Rigid Cap Rock

Through the **Rigid Cap Rock and NULL Blocks** dialog box, you can set whether there is vertical movement constrain on the cap rock of the reservoir, and you can select which NULL blocks will have fixed boundaries at the block nodes.

The **Rigid Cap Rock and NULL Blocks** dialog box can be opened by selecting **Geomechanics | Rigid NULL Blocks** through the main Builder menu.

Alternatively, you can double-click the relevant tree view items under **Boundary Conditions** (that is, tree view items **Rigid Null Blocks** or **Rigid Cap Rock**). Once you make the selection, the **Rigid Cap Rock and NULL Blocks** dialog box will be displayed:



The rigid cap rock can be set by selecting **Fixed CAP Rock (RIGIDTOP)**.

The option you want to use for fixed boundary nodes around null blocks can be selected through the **Fixed Boundary for nodes around** box. Both the empty option and the **All NULL Block** are equivalent since the simulator default, if this option is not used, is the **All NULL Block**. If you do not want any rigid NULL block option to appear in the dataset then select the empty option.

If the **Selected NULL Blocks** option is chosen, then specific NULL blocks can be selected. Specific NULL blocks are selected by either entering the blocks in the grid table or by selecting the grid blocks using the mouse and highlighting them in the grid.

If you enter the blocks using the grid table, the blocks must be specified using the plane number and the element number. These two parameters are described in the *STARS User's Guide* under the RIGIDNULL keyword.

If you select the grid blocks using the mouse, click **Add Selected Blocks From View**. This button will then add the selected block after applying the specified option to the grid table.

You can clear the table by clicking **Delete All Listed Blocks**. You can view and edit comments associated with the **Rigid NULL Blocks** by clicking the **Comments**  button.

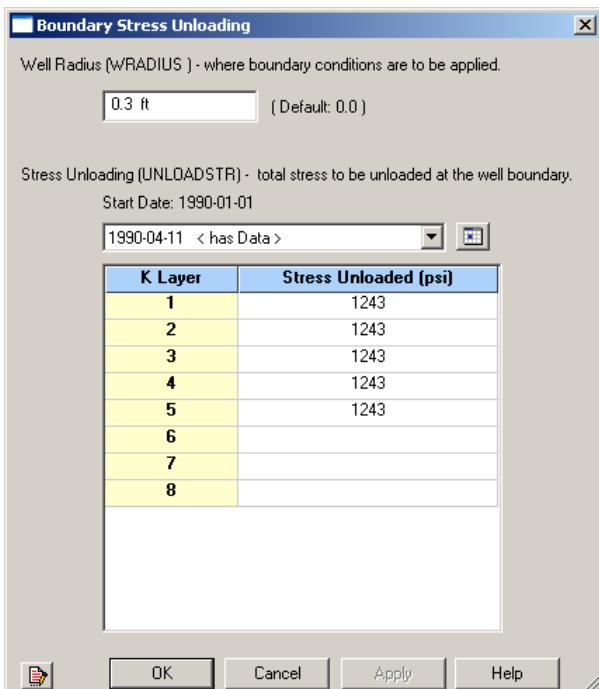
You can also view help information for the rigid NULL blocks and fixed cap rock by clicking **Help** or by pressing the **F1** key.

Boundary Stress Unloading

Through the **Boundary Stress Unloading** dialog box, you can set the well radius where the boundary conditions are to be applied and the total stress to be unloaded at the well boundary.

The **Boundary Stress Unloading** dialog box can be opened by selecting **Geomechanics | Boundary Stress Unloading** through the main Builder menu.

Alternatively, you can double-click the relevant tree view items under **Boundary Conditions** (that is, tree view items **Well Radius**, **Boundary Stress Unloading** or a date under **Boundary Stress Unloading**). Once you make the selection, the **Boundary Stress Unloading** dialog box will be displayed:



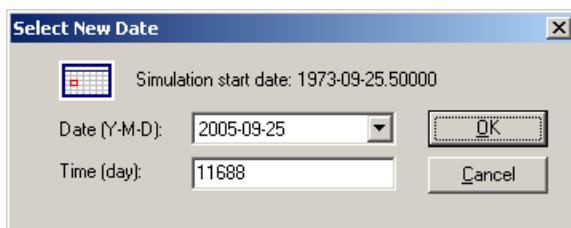
The well radius where the boundary conditions are to be applied can be entered in the edit box at the top of the dialog box. It is date independent, so only one value is needed.

The stress unloading is date dependent and as such, the appropriate date to be viewed and edited must be selected from the date's combo box. The values for the stress to be unloaded

are then entered in the grid table. The grid table lists the different layers in the first column. The stress unloaded is entered in the second column.

In the list of dates, the phrase “< has Data >” signifies that data has been entered for that date.

Next to the date combo box there is a **Date**  button, which is used to add new dates. Clicking the date button will open the **Select New Date** dialog box:



You can view and edit the comments associated with the well radius, by first clicking on the well radius edit box and then clicking the **Comments**  button. To view comments for stress unloading, first click on the grid table and then click the **Comments**  button.

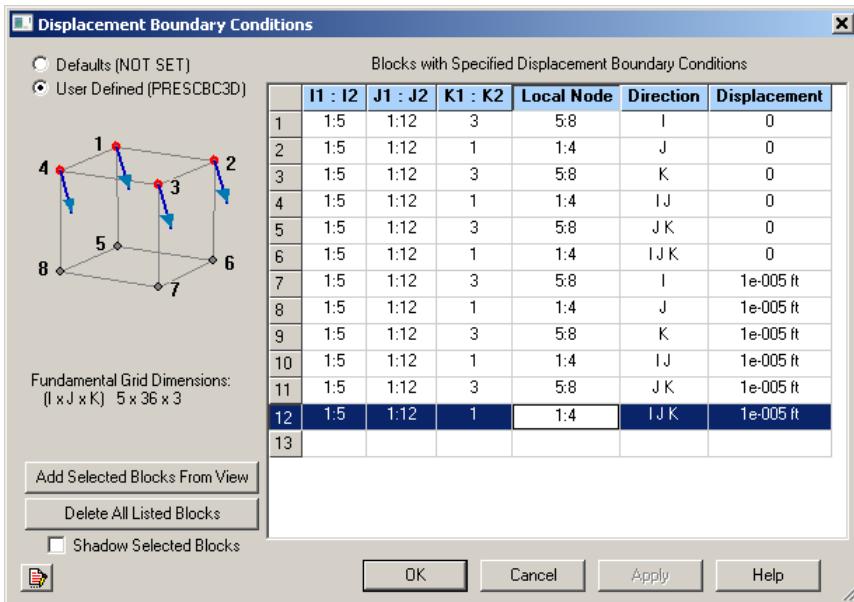
You can also view help information for a specific parameter by first clicking on the parameter field (such as the edit box for well radius or the grid table for stress unloading) and then clicking **Help** or pressing the **F1** key.

Displacement Boundary Conditions

Through the **Displacement Boundary Conditions** dialog box, you can assign displacement boundary conditions at any node on any finite element.

The **Displacement Boundary Conditions** dialog box can be opened through the main Builder menu by clicking **Geomechanics | Displacement Boundary Conditions**.

Alternatively, you can double-click on the relevant tree view items under **Boundary Conditions** (that is, tree view item **Displacement Boundary Conditions**). Once you make the selection, the **Displacement Boundary Conditions** dialog box will be displayed:

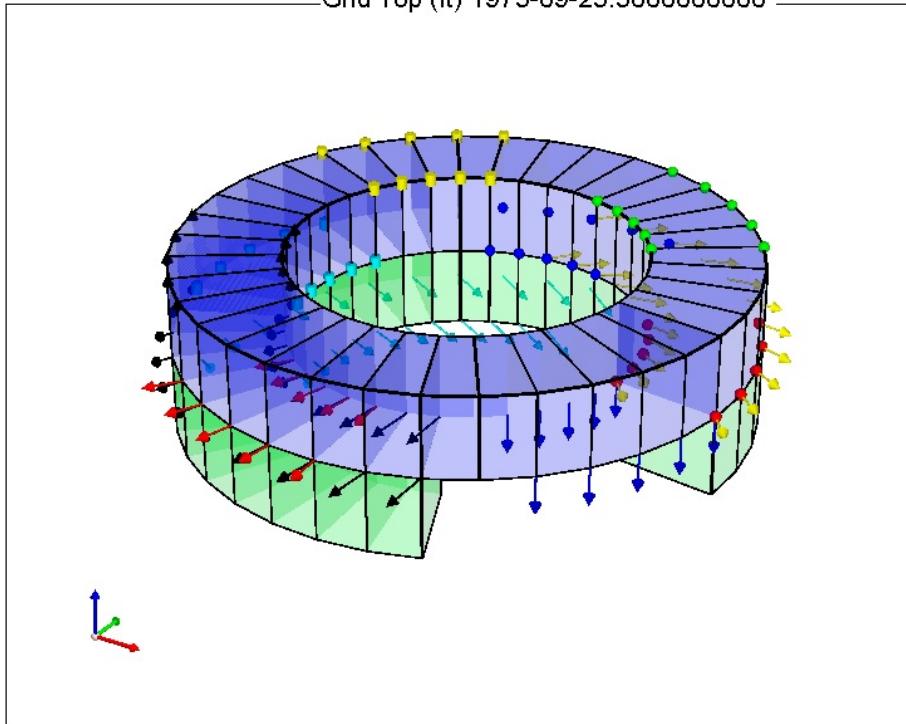


In the above case, the PRESCBC3D keyword was defined in the data set and **Finite Element Method Dimension 3D (GEOM3D)** was selected in **Calculation and Numerical Option** view. Notice that the User Defined (PRESCBC) option is not shown since only one model (2D or 3D) can be chosen at a time, not both.

A schematic representation of a grid block is displayed on the left of the dialog box as a square when using the 2D option or a cube when using the 3D option. The diagram is displayed in the local node system.

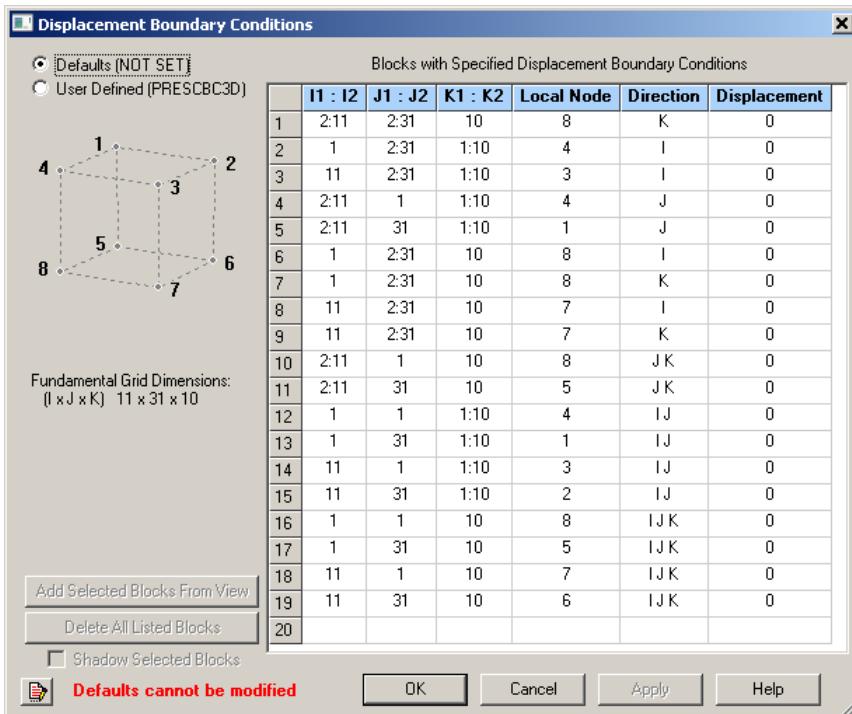
You can select the local nodes, set the direction (I, J, or K), and enter displacement values in the table. Appropriate symbols or vectors are displayed in the view for the current row in the table. If displacement is set to zero and the node is fixed in one direction, a circle is displayed; in two directions, a square is shown; and in three directions, a triangle is shown. You have to select only one row in the table to see the symbols or vectors on the schematic diagram of a grid block. But you can select any rows or all rows to see the symbols or vectors on the reservoir grid. If you select all rows in the above example (by clicking outside the grid in the table area) the following will be seen in 3D view. Note that only selected blocks are displayed; the entire grid is not displayed.

STARS Geomechanics Template #17
Grid Top (ft) 1973-09-25.5000000000



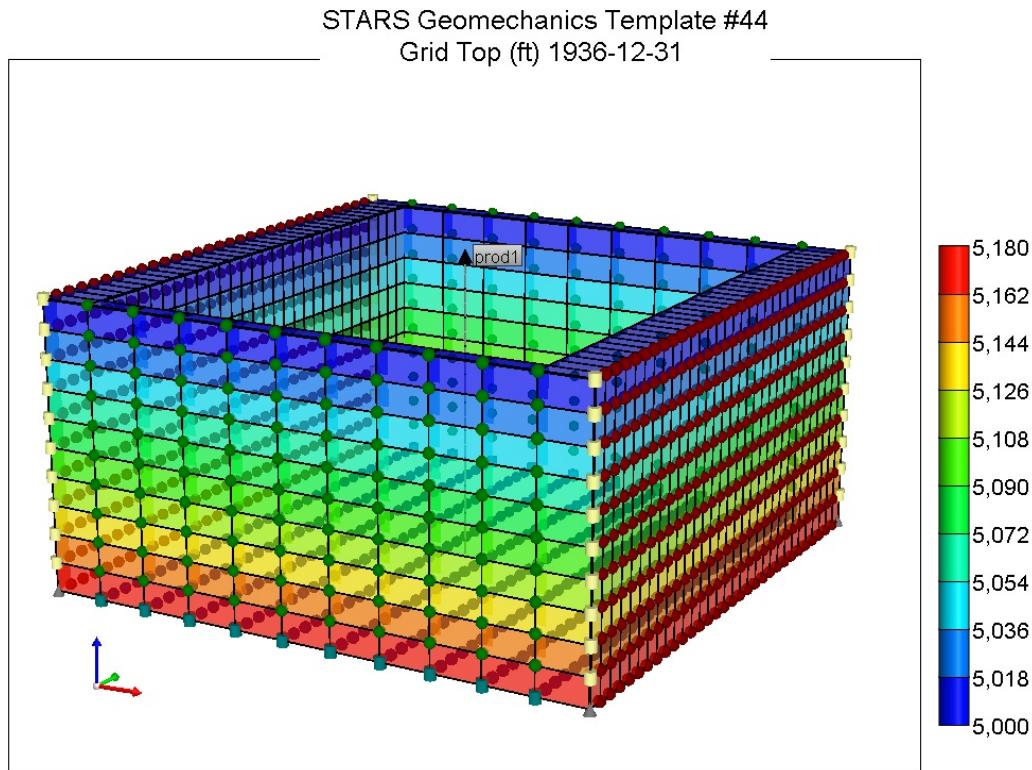
In 3D view, if the node is fixed in one direction, a sphere is displayed; if it is fixed in two directions, a cylinder is shown; and in three directions, a cone is shown. Colors of the symbols or vectors correspond to the directions they are fixed or point in and match the colors of the axes or their mixtures. The example given above is only to illustrate features of the dialog box and all types of anchors and vectors, but does not have any real geomechanical meaning.

If the option NOT SET is chosen then the keyword PRESCBC and PRESCBC3D are absent in the data set and default essential boundary conditions are applied; for example, left, right, front, back and bottom sides of a reservoir are constrained. You can see the defaults in the table and on the grid by selecting the **Defaults (NOT SET)** option.



Notice that if the **Defaults (NOT SET)** option is selected, some buttons are disabled and you can select but cannot change any entry in the table. This option is only for display. A warning message "Defaults cannot be modified" is displayed in red at the bottom of the dialog box. If you now click **Apply** or **OK** no PRESCBC or PRESCBC3D key word will be written to the data set (or will be deleted if it was there before) and the simulator will use defaults for the boundaries. If you did not define anything for PRESCBC3D and selected the **User Defined (PRESCBC3D)** option, the defaults will be copied to your keyword. You will be able to modify it and then to save it into the data set.

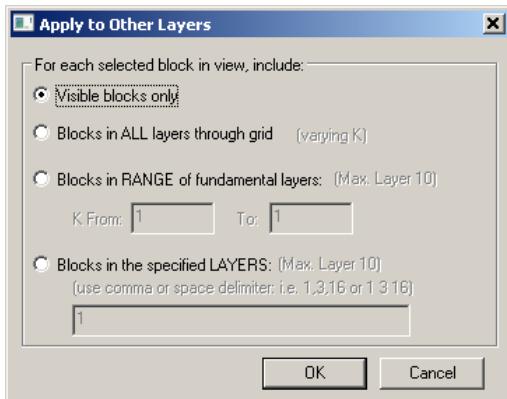
Also notice that for defaults, slightly different display colors are used for anchors and vectors on the reservoir grid, as shown in the following 3D view:



If the **User Defined (PRESCBC)** or **User Defined (PRESCBC3D)** options are selected, then other options in the dialog box become available, allowing you to select and add extra grid blocks to the table.

The block selection can be made either by entering the block values manually in the grid table or by selecting the grid blocks using the mouse and highlighting them in the reservoir grid displayed in 2D. Once you have selected the blocks using the mouse, you can click **Add Selected Blocks From View**.

The **Apply to Other Layers** dialog box will be displayed:



You can then choose one of the options listed in the dialog box. The option you select will be applied to all selected blocks. Click **OK** to add the blocks to the grid table.

If your reservoir grid is displayed in 2D, you can select **Shadow Selected Blocks** to be able to see the blocks you selected in the previous layer when you move one layer up or down in the grid. The selected blocks are outlined in pink and the shadow blocks in dark blue. This feature can be useful when your boundary shape is more complicated and difficult to set up manually.

You can clear the table by clicking **Delete All Listed Blocks**. You can view and edit the comments associated with the **Displacement Boundary Conditions**, by first clicking on the specific parameter and then clicking the **Comments**  button.

You can also view help information about **Displacement Boundary Conditions**, by clicking **Help** or by pressing the **F1** key.

Changing Display Content and Settings

Overview

Both Builder and Results 3D let you select either a two-dimensional (2D) or three-dimensional (3D) view of a reservoir for onscreen viewing and printing. Both of these programs have the same capabilities and user interface to control the display. Builder and Results 3D can display grid, reservoir property, simulation wells, sectors, aquifers, lease planes and geological map information. Information on file opening is discussed in the Builder or Results 3D specific parts of the User's Guides. The current section discusses the parts of the interface that are common to the two programs.

You can view any reservoir property for which there is data in the open file. Builder and Results 3D each begin with a default property selection – the first property in your file. You can alter the current property, and, in Results 3D, select a different time for the displayed data, and animate the data for a series of times. See [Viewing Properties](#).

You can control a large number of display properties, such as the font and size used for text information, the colors of grid lines, and the position of titles. The **Display Settings** tabbed dialog box controls these - see [Changing Builder Properties](#). There is also an interactive method for moving the titles, color scale, info box and map scale. See [Moving and Resizing Display Objects](#).

You can alter the Y/X and Z/X aspect ratios and the map scale if desired. See [Changing the Aspect Ratio and Scale](#).

In the 2D views, you can choose which layer and cross-section to view. You can zoom in to view only part of the reservoir. In the 3D view, you can interactively zoom, pan, and rotate to display the region of interest. See [Zooming, Panning, and Rotating the Reservoir](#). In addition, you can remove portions of the reservoir to see the interior region. See [Selecting IJK Slabs and Regions](#) and [Cutting Away Part of a Reservoir in 3D](#).

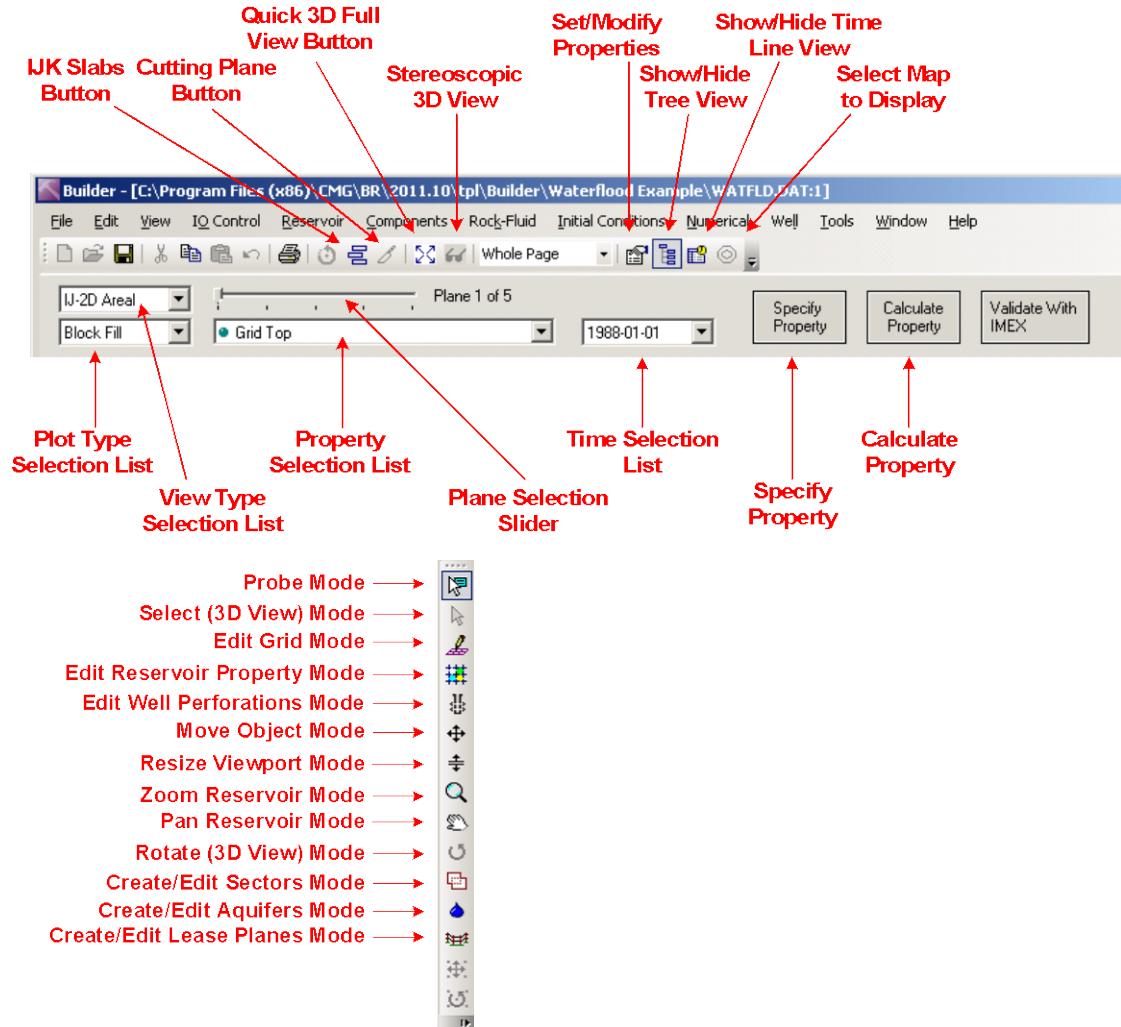
In 2D and 3D, you can also use color to distinguish between ranges of data values in your reservoir view. See [Modifying the Color Scale](#).

You can print or save all on-screen views as image files (in a number of formats) that you can later insert into word processor or presentation software. See [Printing and Saving Images](#).

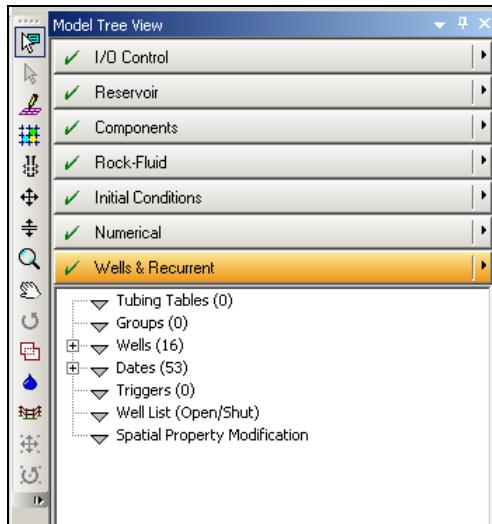
Finally, you can simultaneously view different properties for the same file or look at properties from different files. See [Working With Multiple Views and Documents](#).

Understanding the Parts of the Main Window

Builder and Results 3D have many controls and dialog boxes in common, but they also have differences. The Builder toolbar is as follows:



The Tree View, shown below, lists the spatial (grid) properties, sectors, aquifers, lease planes and wells. It marks the properties that have specifications and/or values, displays quick-review strings for specifications; and dates at which well changes (perforations) take place for a well. It offers a convenient way to change displayed properties and launch **General Property Specification** or the **Well Selection** dialog box (double-click on a property name, specification or date for a well). You hide or show the Tree View using the **Show/Hide Tree View** button on the main tool bar.



Displaying the X,Y Coordinates of a Point in the Reservoir

To display the X,Y coordinates of a point in the reservoir:

When in **IJ-2D Areal** view type, move the cursor over the reservoir, and the X,Y coordinates (in map units) of the cursor hot spot are displayed on the left side of status bar at the bottom of the main window.

When in **IK-2D X-Sec** or **JK-2D X-Sec** view types, move the cursor over the reservoir, and the X,Z or Y,Z coordinates are displayed on the left side of the status bar at the bottom of the main window.

Displaying Crosshairs Centered on the Cursor

It is also possible to turn on “crosshairs”, which draw lines vertically and horizontally from the cursor hot spot.

To toggle the display of crosshairs in 2D views:

1. While pointing at the reservoir, click using the rightmost mouse button. A pop up (context) menu will appear.
2. Click with the leftmost mouse button on **Show X-hair in Probe Mode** to toggle the display of crosshairs on or off.

Displaying the IJK Grid Block Address

To display the IJK block address:

In Probe mode, using the leftmost mouse button, click and hold the mouse cursor anywhere in the drawing area over an active reservoir image. The IJK grid block address of the grid block and the current property value is displayed.

To change the items shown in the probe display, or the font used, choose **Probe Display** from the **View** menu.

Viewing Properties

Selecting a Property

All the properties available in the current file are available. The default selection is always the first property in the list. Currently all the spatial properties are defined in Builder at “zero time”. However, if you have defined wells or well changes at later times, then the **Time Selection List** will show the latest time.

To select a reservoir property to use:

1. Click on the drop-down button of the properties selection list. A list of available properties appears, with scroll bars if the list is longer than the list box length.
2. Click on the property you want to see.

OR

1. Scroll the Tree View to the property name. Click on the property name string.

If a property has specifications and/or values, the name will be marked by a sign. If a property has an EQUALSI specification, it will be marked by a sign.

Selecting the View Type and Plane Number

The default view type is **IJ-2D Areal** with the K layer set to 1. You can select any layer in the K direction from all K layers in the file.

To view another plane:

1. Move the **Plane** slider.

You can move the Plane slider by clicking and dragging the slider. Once you have clicked on the slider (and thus made it the selected control), you can move through the planes using the right and left arrow keys on the keyboard.

To select another view type:

1. Click on the drop-down button of the view type selection list.
2. Click on the desired view type.

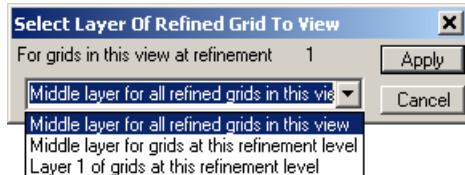
The available 2D view types are aerial or map view (**IJ-2D Areal**) and two cross-sectional views (**IK-2D X-Sec** and **JK-2D X-Sec**). The last choice is 3D view (**3D View**).

Selecting Plane Number within a Refined Grid

The default for refined grids is to show the “middle” plane of the refined grid. For example, suppose you are looking at an **IJ-2D Areal** view of plane K=3. If block 1,1,3 contains a Cartesian refined grid with 3x3x3 refinement, probing a refined grid block will show that plane K=2 within the refined grid is being displayed.

To change the displayed plane with a refined grid:

1. In **Probe** mode, move the cursor over the refined grid of interest, and then right-click.
2. From the pop up (context) menu, select **Select Refinement Level**. The **Select Layer of Refined Grid to View** dialog box is displayed:



3. Click on the pop down button of the selection list, and choose a layer to view.
4. Click **Apply**.
5. To close the dialog box, click **Cancel**.

Selecting Block Fill, Contour Lines or Contour Fill

The plot type selection list allows you to choose from **Block Fill**, **Contour Lines** and **Contour Fill**. To select a plot type, click on the pop down button on the plot type selection list and then click the desired type of plot.

Block Fill colors each grid block with the color associated with the value of the property at the center of the block.

Contour Lines are calculated for block centered properties by first interpolating the values for block corners from the block center property values. The corner values are then contoured. The number and value of contour lines is controlled by the settings for the color scale for the property being displayed.

Contour Fill uses data interpolation between grid block centers values to smooth data. First values for block corners are interpolated from the block center property values. The corner values are then contoured. For Cartesian (Variable depth and thickness) grids, the corners of the grid blocks are also smoothed (in cross-section and 3D views) to better show the flow connections and reservoir shape. By default, Contour Fill also shows the contour lines. The **View | Properties Spatial Properties** dialog box has controls to **Draw lines in Color Fill mode** and **Show Values in Color Fill mode** which control whether lines and values are shown in Contour Fill mode.

Some properties, such as Ternary, cannot be contoured. They will only be displayed in **Block Fill** mode.

Selecting a Time Display Format

Times can be displayed as Date or Time (time since simulation start, in day, hours, and so on, determined by the simulation output units of the open file). The default time format is *Date*.

To change the time format:

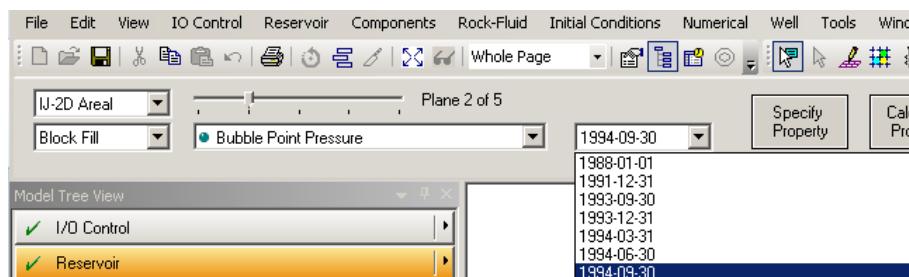
1. Select **Properties** from the **View** menu, or right-click in the reservoir viewport and then select **Properties** from the pop-up (context) menu. The **Builder Properties** dialog box is displayed.
2. Click the **General** tab.
3. Select the desired time format in the **Preferred Time Display** area.

Selecting a Time

The simulation output file (SR2) may contain property data for several simulation times. In Builder, the time selection list gives a list of all times with well perforation changes.

To view a property at a different time, either:

1. Click the drop-down button of the time selection list. A list of the times for which the selected property is available will appear.



2. Click the desired time or date to select it.

Changing Builder Properties

Builder Properties controls the details of the Builder view display. For example, one of the many display settings is the font type, size and color used for the first title. Each view window within a main Builder window has its own set of display settings.

Builder Properties Dialog Box

To open the **Builder Properties** dialog box:

Either

- Select **Properties** from the **View** menu.

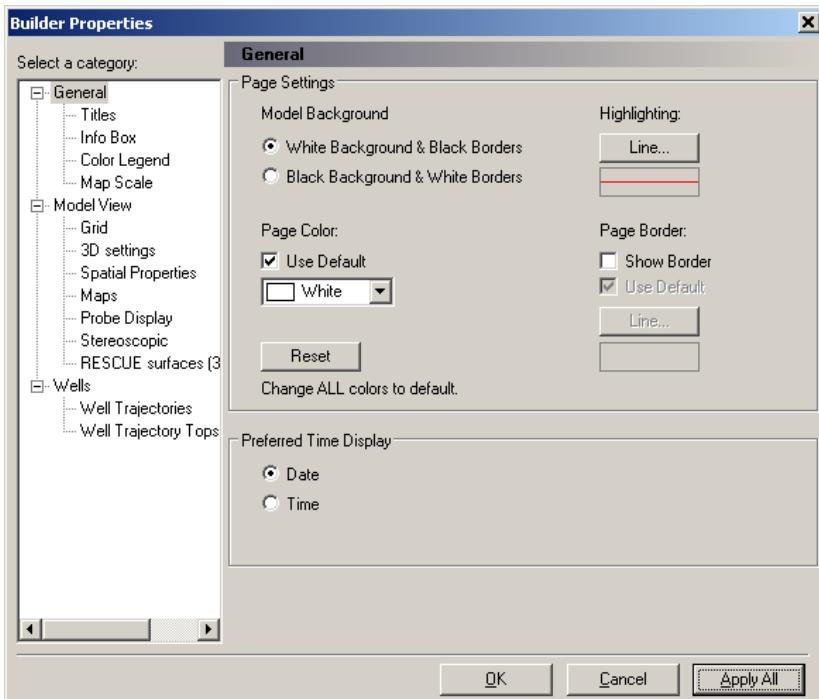
OR

- Position the cursor in the current view and then right-click. A pop up (context) menu appears. Click on the **Properties** menu item.

OR

- Click the Set/Modify Properties  button on the main toolbar.

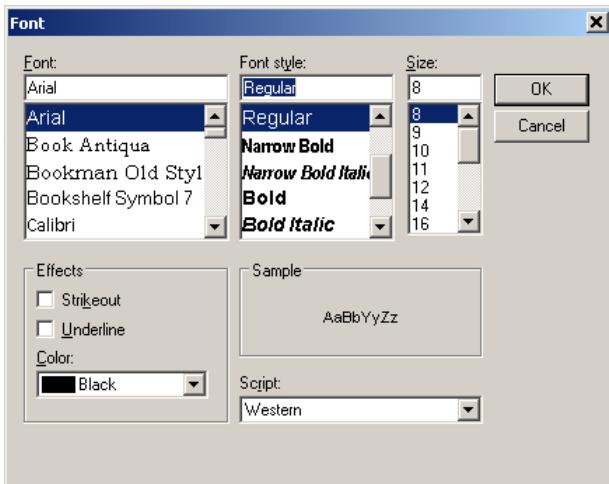
The **Builder Properties** dialog box is displayed:



The **Builder Properties** dialog box is organized by a tree view, with each heading controlling one section of the total display. To select a particular section on the tree view, click on it. Each section contains a number of controls that allow you to customize the display of a particular item.

Selecting a Font Name, Style, Size and Color

Many of the **Builder Properties** dialog box tabs contain buttons to set the font to use for particular text output. For example, the **Titles** tab contains a **Line 1 Font** button, and the Contour Map tab contains a **Set Values Font** button. Clicking on one of these buttons will display the common **Font** dialog box:

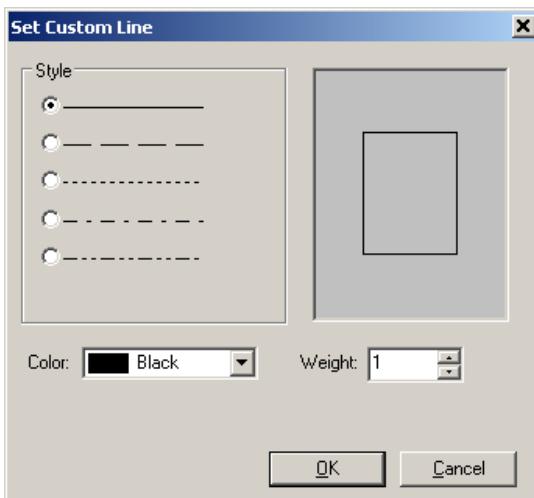


The **Font** dialog box allows you to select a font, style, size and color from the fonts that are installed on your computer.

Selecting a Line Style, Thickness and Color

Many of the **Builder Properties** dialog box tabs contain buttons to set a custom line type. For example, the **Grid** tab contains several **Line** buttons as does the **Spatial Properties** tab.

Clicking any one of these buttons will open the **Set Custom Line** dialog box:



Through the **Set Custom Line** dialog box, you can select a line style, color and weight. Note that only a solid style can have a weight greater than 1 (this is a Windows limitation).

Changing Displayed Titles

To change titles that appear in the view:

1. Select **Properties** from the **View** menu.
2. Click on the **Titles** tab.
3. In **Line 1** and **Line 2**, specify the title that should appear. You can enter the following to automate specific entries in the titles. OR Select from a list by clicking on the  button to the right of the text box.

| Enter... | To include this in the titles... |
|----------|---|
| \$file | Current data filename |
| \$prop | Current grid property |
| \$title1 | String specified by “*Title1” keyword in the simulator dataset |
| \$title2 | String specified by “*Title2” keyword in the simulator dataset |
| \$title3 | String specified by “*Title3” keyword in the simulator dataset |
| \$date | Today’s date |
| \$time | Current simulation time, date, or time step of the data being displayed |
| \$layer | Current layer information (this is empty if you are in 3D mode) |

4. Click **OK** to accept your changes and close the dialog box.
5. Click **Cancel** to close the dialog box. If you click **Cancel** before you click **OK**, you will lose all your changes.

Toggling the Display of Grid, Map and Wells

The options in the **Builder Properties** dialog box allow you to toggle on or off the display of all the items in the view. For example, on the **Info Box** tab, there is a **Show Info Box** check box, which is used to turn the display of the **Info Box** on or off.

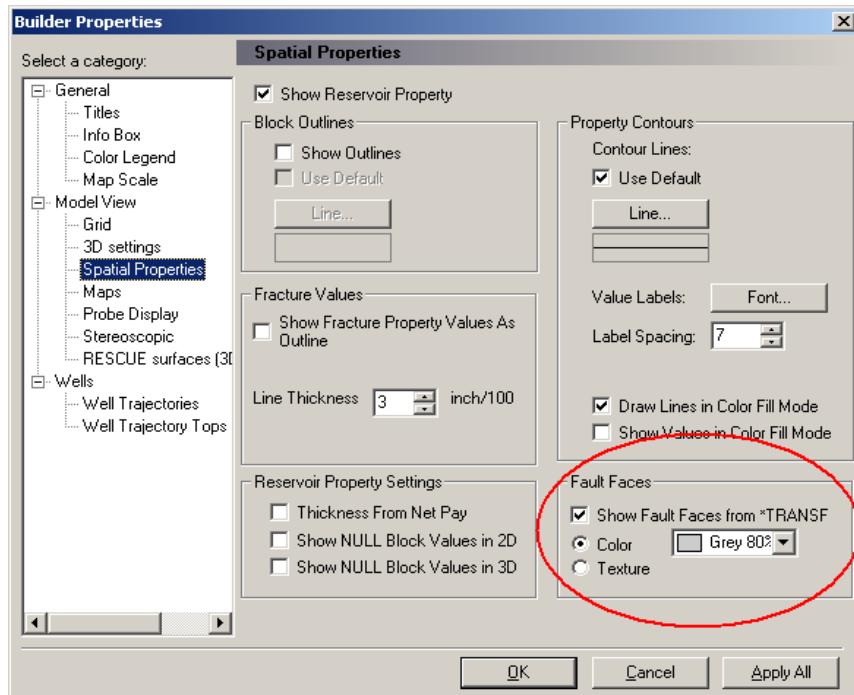
To toggle the display of grid lines and axes on and off:

1. Select **Properties** from the **View** menu. The **Builder Properties** dialog box is displayed.
2. Click on the **Grid** tab.
3. Click on the **Show Grid** check box to toggle grid display on or off.
4. Click **OK** or **Apply All** to apply your change. Click **Cancel** to cancel your change.

Showing Fault Faces

To show fault faces with solid color or using texture:

1. Bring up the **Properties** dialog box as described earlier.
2. Click on the **Spatial Properties** section of the tree view.



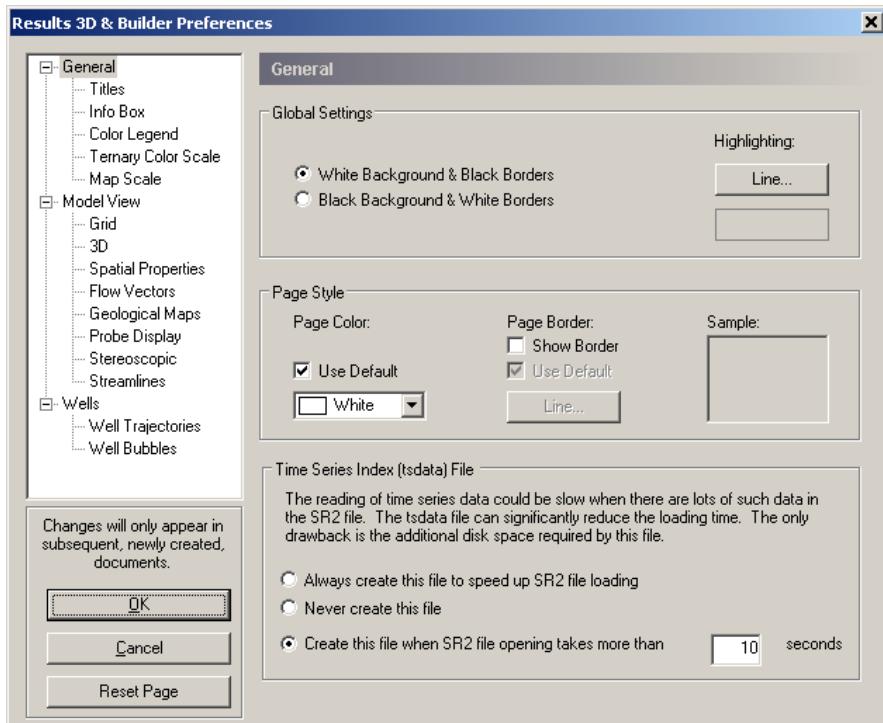
3. Select the **Show Fault Faces From *TRANSF** check box.
4. Select **Color** to show fault faces with solid color. The color used for fault faces can be selected through the color combo box.
5. Or click on **Texture** radio button to show fault faces with texture pattern.
6. Click **Apply All** or **OK**.

Use **Preferences** feature to save the **Fault Faces** settings as default values.

Note: When texture patterns are used to show fault faces thinner lines on the pattern correspond to greater permeability of the fault.

Results 3D and Builder User Preferences

User Preferences feature Results 3D Builder settings that can be saved as default values in Windows registry under the current user name. Select **File | Results 3D & Builder Preferences** in the Builder menu to open the **Results 3D & Builder Preferences** dialog box.



Select desired plot layout, fonts, lines, colors and so on, and then click **OK** to save it in the registry. Click **Reset Page** to return factory defaults on the current preference page. User preferences apply to newly created plots and do not change the existing ones.

Under the **General** page of the **Results 3D & Builder Preferences** dialog box, there is a section labeled **Time Series Index (tsdata) File**. This section is used to manage the way large files are handled when opened in Results 3D. When the SR2 file is opened, a file with the same base file name and the extension of tsdata will be created automatically if the file loading takes longer than the default of 10 seconds and **Create this file when SR2 file opening takes more than** is selected. This file will greatly reduce the subsequent loading time of the same SR2 file; however, this option slightly increases the loading time the first time you open the file and it occupies additional disk space. If you never open the same SR2 twice, then this option is not beneficial and can be disabled through **File | Results 3D & Builder Preferences** menu. You can also change the minimum loading time required to create the tsdata file in the same **Results 3D & Builder Preferences** dialog box.

Moving and Resizing Display Objects

A view consists of a number of items: title block, info box, color scale, map scale, and reservoir viewport. These items can be moved about on the view.

Moving the Title, Info Box, Color Legend, and Map Scale

To move an item:

1. Move the cursor inside the view, and then right-click. A pop up (context) menu appears.
2. Click **Move Object**. The cursor will change shape to the move cursor.
3. Click on the item you want to move and, holding the mouse button down, drag the item to the desired location. An outline of the item's shape will move as you drag, then the item will be redrawn at the new location when you release the mouse button. Note that you cannot drag an item outside the view margins.

Resizing the Titles and Info Box

The sizes of the titles and the info box are determined by the fonts used by these items and the length of the text to be displayed. To increase or decrease the size, use the **Builder Properties** dialog box, and the **Titles** or **Info Box** tabs, to change the font size.

Resizing the Color Legend

The **Builder Properties** dialog box, on the **General | Color Legend** tab, contains text entry fields for **Color Bar Width** and **Color Bar Height**. The width of the color scale is determined by the value of **Color Bar Width**, the **Color Legend Font**, and the **Numeric Format** used. The height is determined by the **Color Bar Height**.

Resizing the Map Scale

The **Builder Properties** dialog box, on the **General | Map Scale** tab, contains a text entry field for **Approximate Width (% page)**. This controls the width of the Map Scale. The number of scale bars displayed and the **Map Scale Font** sets the height of the map scale box.

Resizing the Reservoir Viewport

To resize the reservoir viewport:

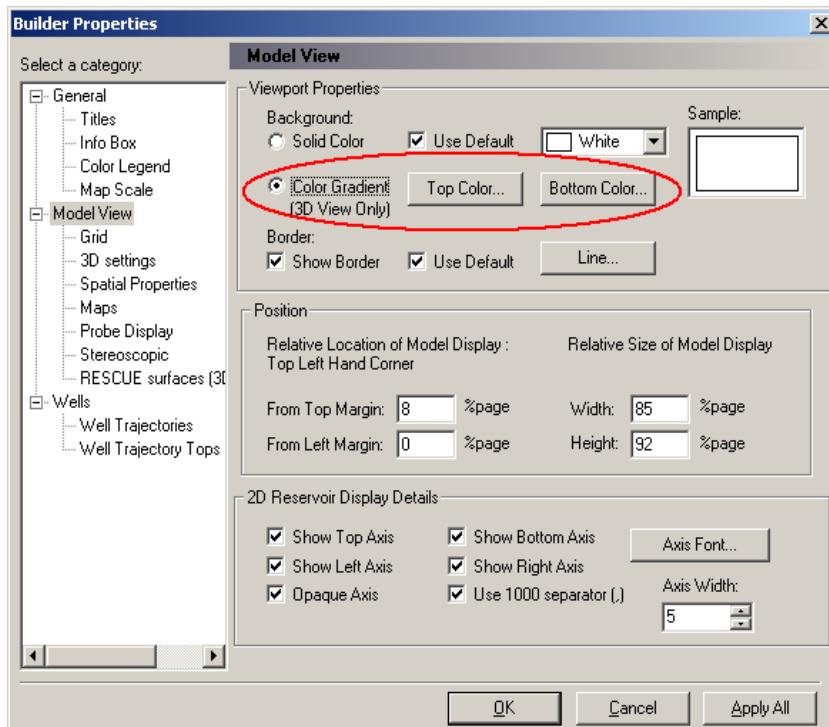
1. Move the cursor over the reservoir viewport, and then right-click. A pop-up (context) menu will appear.
2. Click **Resize Viewport**. The cursor will change to the resize cursor and small rectangle handles will appear at the corners and sides of the viewport.
3. Move the cursor over a handle, and the cursor will change shape to indicate the direction that the handle can be dragged in.
4. Click and drag the handle to the desired position.

- Release the left mouse button, and the reservoir viewport will be redrawn in the new size.

Toggling the Color Gradient Background

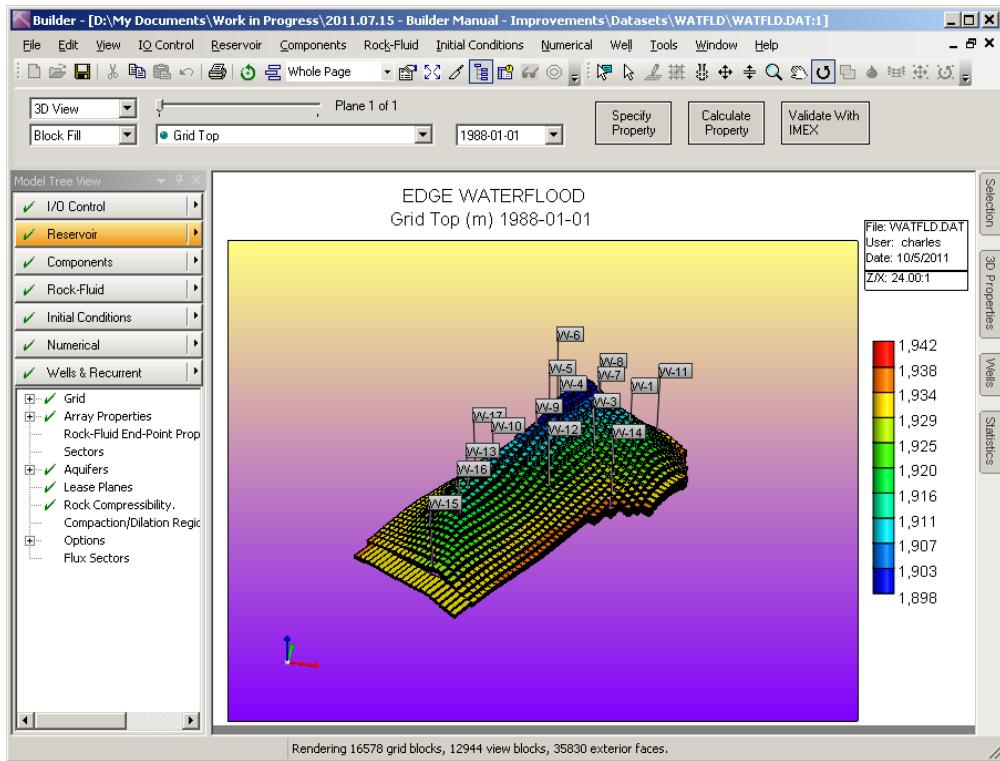
To toggle color gradient background of the reservoir viewport:

- Open the **Builder Properties** dialog box as described earlier.
- Click the **Model View** section of the tree view.



- Select the **Color Gradient** option in the **Viewport Properties** area.
- Click the **Top Color** button to open the **Color** dialog box.
- Select the desired color for the reservoir viewport top.
- Click the **Bottom Color** button and repeat the steps to select the reservoir viewport bottom color.
- Click **Apply All** or **OK**.

A smooth color gradation will be displayed over the reservoir viewport background.



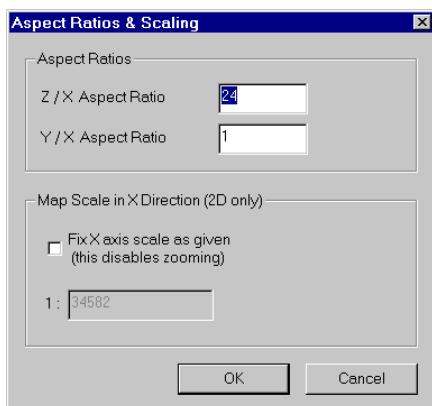
Use **Results 3D & Builder Preferences** to save the color gradient settings as default values.

Note: Color gradient background is provided for 3D view only.

Changing the Aspect Ratio and Scale

To change the aspect ratio or scale used to draw the reservoir:

1. Select **Aspect Ratios and Scale** from the **View** menu. The **Aspect Ratios & Scaling** dialog box is displayed:



2. Enter new values for **Z/X Aspect Ratio** and/or **Y/X Aspect Ratio** to change the aspect ratios.
3. To set a fixed scale, click on the check box for **Fix X axis scale as given**, and then enter a value for the scale.
4. Click **OK** to confirm your changes, or **Cancel**.

Zooming, Panning, and Rotating the Reservoir

Zooming

To zoom in on a part of the reservoir:

1. Move the cursor over the reservoir viewport, and then right-click. A pop-up (context) menu will appear. Click **Zoom Reservoir**. Alternatively, click the **Zoom**  button on the mode toolbar.
2. The cursor will change to a magnifying glass.
3. Move the cursor to the top left corner of the area that you want to zoom in on, and drag out a rectangle of the zoom area. The height to width aspect ratio of the zoom rectangle will always match the height to width ratio of the viewport.

Panning (Moving) the Reservoir

Once you have zoomed in on a portion of the reservoir, you may no longer see the entire reservoir in the reservoir viewport part of the view.

To pan the reservoir within the reservoir viewport:

1. Move the cursor over the reservoir viewport, and then right-click. A pop-up (context) menu will appear. Click **Pan Reservoir**. Alternatively, click on the **Pan**  button on the mode toolbar.
2. The cursor will change to a hand.
3. Click on a point in the reservoir, and then drag it to a different position within the viewport. Release the mouse and the reservoir will be redrawn at the new position.

Undoing a Zoom or Pan

To undo your last zoom or pan:

1. Move the cursor over the reservoir viewport, and then right-click. A pop-up (context) menu will appear.
2. Click **Undo Zoom or Pan**. Your view will change back to showing the reservoir as it was prior to the last zoom or pan operation that you preformed.

To return to viewing the entire reservoir:

1. Move the cursor over the reservoir viewport, and then right-click. A pop up (context) menu will appear.
2. Click **Full Reservoir View**.

Rotating, Panning and Zooming the Reservoir in 3D

When you are in a 3D view, you can use the zoom and pan features as discussed above. In addition, you can rotate, pan and zoom the reservoir interactively by direct manipulation of the mouse. If you have a large and complex reservoir, you may require a fast graphics card to have good control while doing these operations.

To rotate the reservoir in 3D:

1. Move the cursor over the reservoir viewport, and then right-click. A pop-up (context) menu will appear.
2. Click **Rotate (3D View)**. The cursor will change to the rotate cursor.
3. Click in the reservoir viewport, and drag the mouse left or right, up or down to rotate the reservoir.
4. Hold down the CTRL key, click in the reservoir viewport and drag the mouse up or down to increase or decrease the size of the model.
5. Hold down the SHIFT key, click in the reservoir viewport and drag the mouse to pan the reservoir.

Note: Panning the reservoir may also change the center of rotation. While you are panning the reservoir, a small cross hair will appear in the center of the screen. When you finish the pan operation, the grid block under the cross hair will become the new center of rotation.

If you have a complex reservoir, and/or a slow (non-OpenGL) graphics card, you may wish to rotate in bounding box mode.

To rotate in bounding box mode:

1. Move the cursor over the reservoir viewport, and then right-click. A pop-up (context) menu will appear.
2. Click **3D Settings**. The **Builder Properties** dialog box will be displayed, open to **Model View | 3D settings**.
3. Select **Rotate bounding until mouse button released** check box.
4. Click **OK** to apply the change, or **Cancel**.

Improving 3D Graphics Performance

One way to improve 3D graphics performances is to have a graphics card that supports OpenGL in hardware. The card must come with OpenGL ICD files and have enough memory to support double buffering. Video Card performance has improved significantly in the past couple of years, and most cards currently for sale include OpenGL support. A “professional” OpenGL graphics card can significantly improve 3D performance for large simulation models.

There are two ways to indicate to Results 3D and Builder to use hardware OpenGL acceleration. The first is in the Technologies Launcher by using a command line option of “-s”. This is normally set automatically by the installation program. The second method is to check the **Hardware acceleration for 3D** check box on the **3D** tab of the tabbed property sheet.

To add or remove the **Hardware acceleration for 3D** option, do the following:

1. In the CMG Technologies Launcher, select the Results 3D icon.
2. Select **Modify Icon** from the **Programs** menu.
3. In the text box **Additional command line switches**, add or remove a “-s” after the “-3”. The entire line should read “-3 -s”.
4. Click on **OK** to save the modified settings.
5. Start up Results 3D, open a simulation output file (SR2) and change to a 3D view.
6. If adding the -s command line option does not improve performance, or if your display is now incorrect, remove the -s using the Technologies Launcher **Modify Icon** dialog box.

Other tips on improving 3D graphics performance include:

- Turn off the display of the grid prior to switching to 3D View.
- For Cartesian (variable depth and thickness) grids, switch to Contour Fill mode prior to switching to 3D view.

Both of these steps are done automatically if you use the **Quick 3D Full View** button on the toolbar.

Using Stereoscopic 3D

Stereoscopic 3D works by alternately displaying “left eye” and “right eye” images, with slightly different perspectives, while a pair of shutter glasses cover or uncover the appropriate eye in sync with the images being displayed. Your mind’s vision center will use the different perspective information from the two images to compose a 3D composite – the image will “pop” out of the screen appearing to be a true 3D object.

To use stereoscopic 3D, you must have a graphics card that supports OpenGL stereo buffers, a pair of shutter glasses, and a display that is capable of a high refresh rate (>100 Hz). The normal CMG software installation procedure should set Results and Builder to use Stereoscopic 3D if the graphics card is capable.

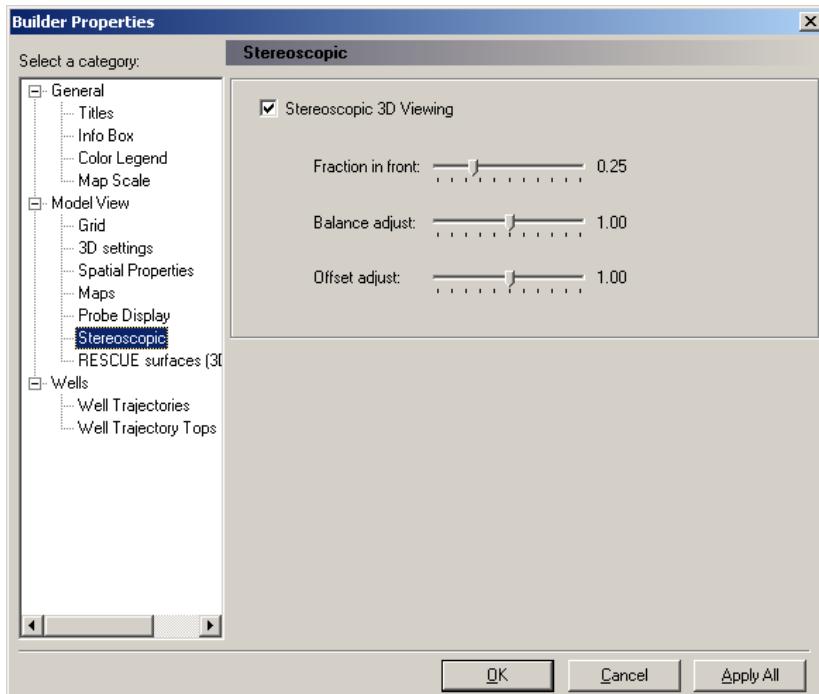
To add or remove stereoscopic 3D display:

1. In the CMG Technologies Launcher, select the Builder icon.
2. Select **Modify Icon** from the **Programs** menu.
3. You must be using hardware acceleration (“-s”) in the text box **Additional command line switches** to use stereoscopic 3D. The entire line should read “-g -s”.
4. Click on **OK** to save the modified settings.
5. Start up Builder, open a data set file and change to a 3D view.
6. You can set stereoscopic 3D default value enabled or disabled in **Preferences** dialog box on **Stereoscopic** page.

Note: if you are not wearing shutter glasses, or if they are not working correctly, you will see two superimposed images on the screen when in Stereoscopic 3D mode.

To change the setting options for stereoscopic 3D:

1. Select **Stereoscopic View Settings** from the **View** menu. The **Model View | Stereoscopic** page of the **Builder Properties** dialog box is displayed:

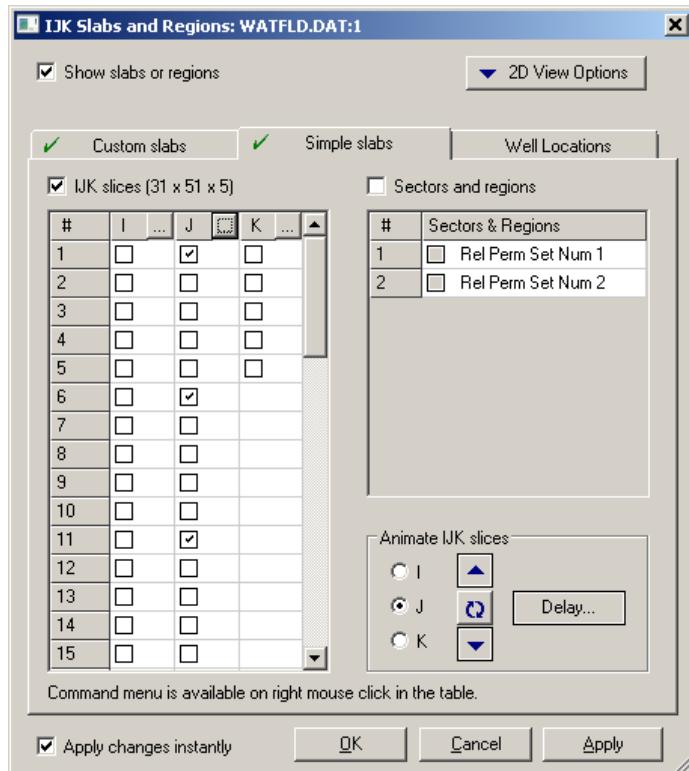


2. Select **Stereoscopic 3D Viewing** to enable stereoscopic 3D, or cancel to disable.
3. Move the **Fraction in front** slider to pull the image more “out” of the screen.
4. If you significantly change the **Fraction in front** setting, you may need to adjust the **Offset adjust** and **Balance adjust** to get a comfortable image. It is suggested that you start with small changes.
5. Click **Apply** or **OK** to apply your changes. Click **Cancel** to dismiss the dialog box.

Selecting IJK Slabs and Regions

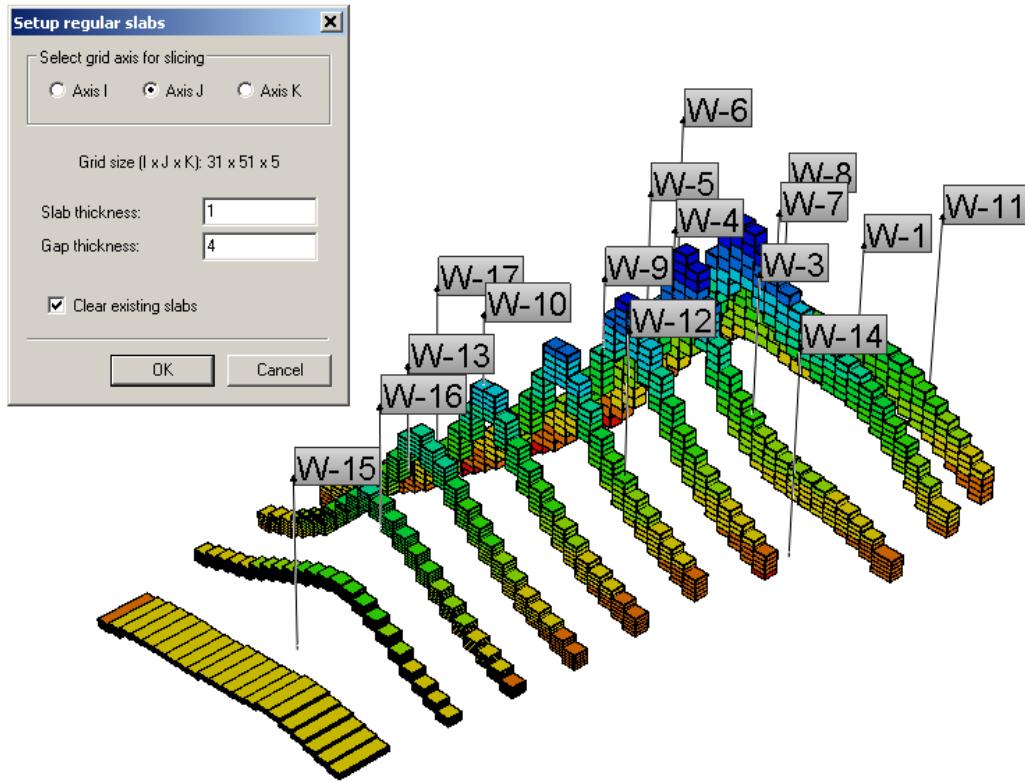
You can define rectilinear volumes of grid blocks in IJK coordinates; then, only those volumes are displayed in 2D and 3D. There are four methods for defining slabs. All methods can be selected at the same time, to have a combined effect on the view. Selecting **Apply changes instantly** makes the 3D view update immediately without clicking **Apply**. Select **IJK Slabs and**

Regions from the View menu, or click on the IJK Slabs  button on the toolbar. The IJK Slabs and Regions dialog box appears. Check Show slabs or regions.



Simple Slabs - IJK Slices

The left table contains three columns with check boxes for each of the IJK coordinates. Selecting the check box adds a corresponding slice of grid to the view. The top row has three buttons next to I, J, K names. Clicking these buttons opens the **Setup regular slabs** dialog box, through which you can set slices with a constant thickness and step along one of the axes.



Right-click in the IJK slices table to display the following context menu, through which you can turn selected slices ON or OFF, clear all slices, or set regular slices. The table cells can be selected as in any spread sheet.



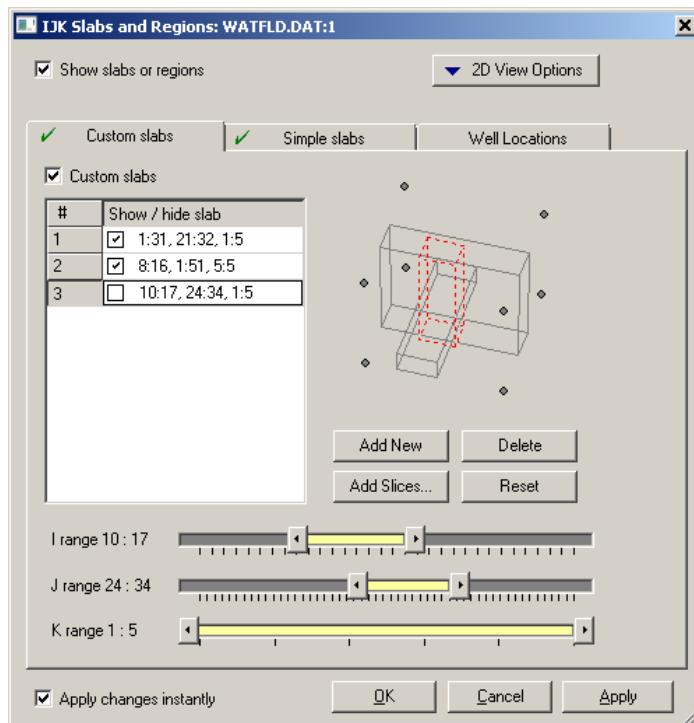
The **Animate IJK slices** area of the **IJK Slabs and Regions** dialog box contains options to start continuous shifting of selected IJK slices along one of the axis. Simply press down the **Loop** button and click the **Up** or **Down** arrow to start the animation. The **Delay** button controls the time break in milliseconds between the animation steps. To stop animation you can click on **Loop** button once again or close the control panel.

Sectors and Regions

This table contains names for all available sectors in regions that have been used in the simulation run and saved in the SRF file. If none of them have been created in the dataset, then the table is empty.

Custom Slabs

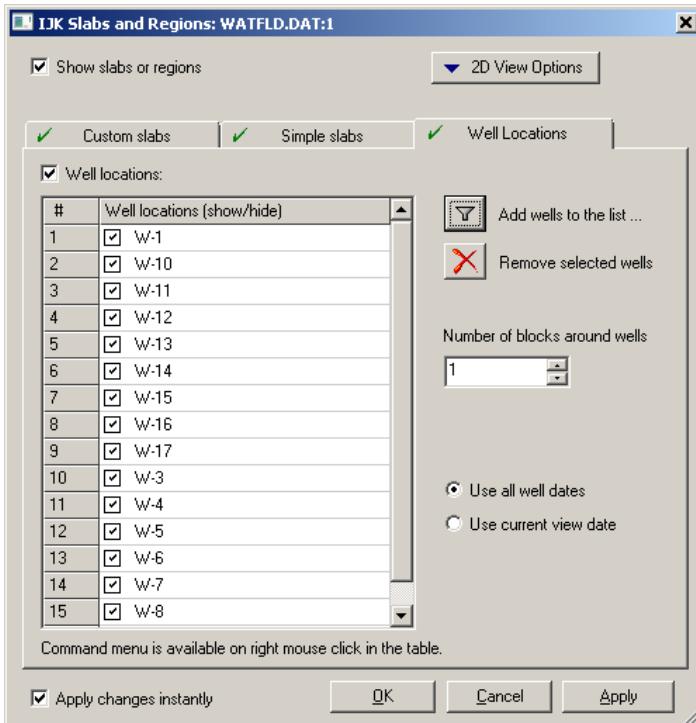
Through the **Custom slabs** tab you can cut a part of the grid model in terms of IJK ranges and show or hide it on display:



1. Select the **Custom slabs** check box.
2. Click the **Add New** button to append a new slab covering the entire grid size.
3. Use IJK range sliders to adjust the boundaries of the current slab. The current slab is shown in red color in the diagram on this tab. The current slab can be changed by clicking on a different row in the table.
4. Clearing the **Show/hide slab** check box in the table hides or effectively makes a hole in the grid on display. It is a handy feature for creating cutoffs around wells. Make sure that some visible slabs are added otherwise the screen will be empty.
5. Use the **Reset** button to return to the maximum range for the current slab and the **Delete** button to remove the current slab from the view.

Well Locations

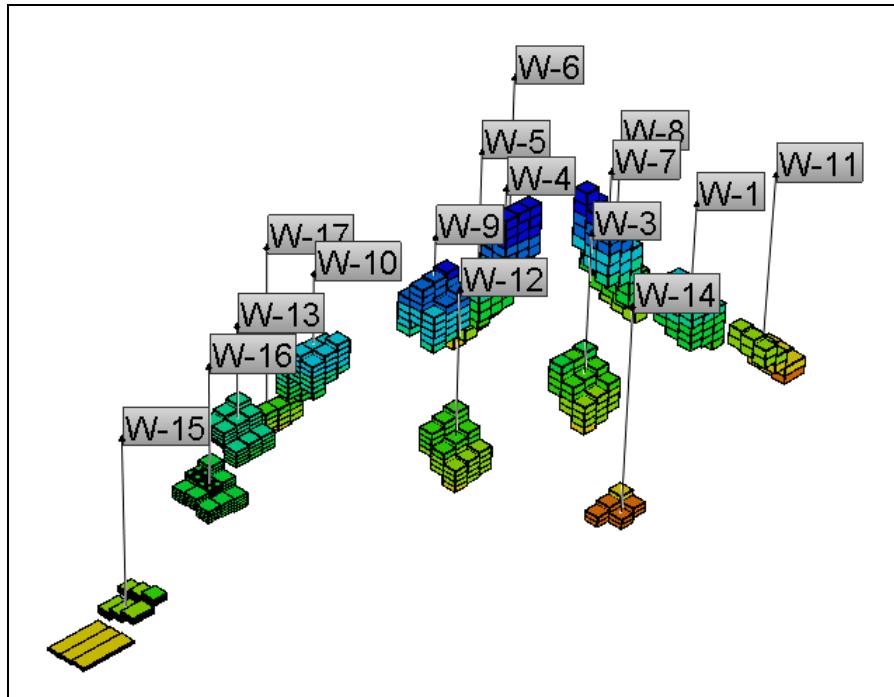
The **Well Location** tab provides controls to display fundamental grid blocks around the well completions.



The **show/hide** check boxes allow you to add or hide these grid blocks from viewing.

1. Select the **Well location** check box.
2. Use the **Add wells to the list** button to select new well names.
3. Adjust number of grid block around wells to show bigger or smaller area at well completions.
4. Select **Use current view date** to ensure well locations will be synchronized with the current view date (in case when well completions are changing with time) otherwise completed grid blocks from all times are shown together.

5. Use button **Remove selected wells** to delete selected names from the table and view the display:

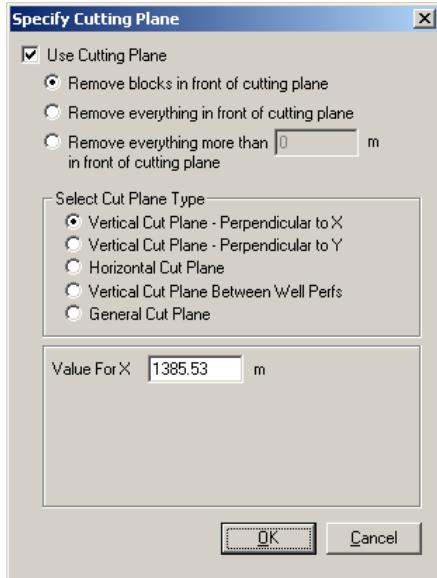


Cutting Away Part of a Reservoir in 3D

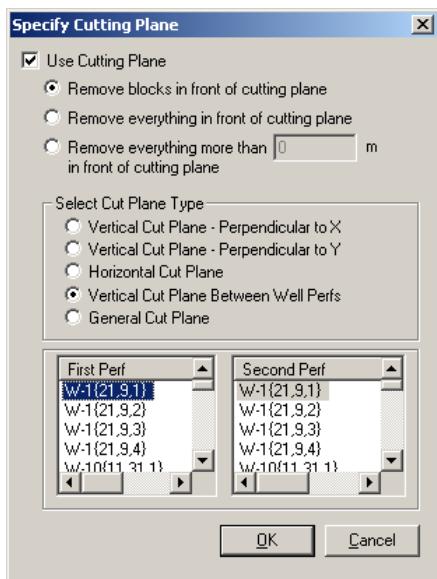
You can use a cutting plane to remove part of the reservoir while in 3D view. This allows you to see the interior of the reservoir. Cutting planes are particularly useful when using non-orthogonal corner point grids, where IJK slabs do not give you planar surfaces. Cutting planes can also be used between wells and through refined grids.

To use a cutting plane:

1. Select **Cut Plane** from the **View** menu, or click the **Cut Plane** icon. The **Cutting Plane** dialog box appears.



2. There are five options on how to specify the cutting plane. The first three specify vertical or horizontal cutting planes, where you give one value to specify the plane location (default to center of the reservoir). Click **OK** to set the cutting plane.
3. If you select **Vertical cut plane between well perfs**, the dialog box changes as follows:



4. Select two well perforations of interest, and then click **OK** to set the cutting plane.
5. For a **General Cut Plane**, you must enter a point and a normal direction to specify the cutting plane. For example, a value of NX = 0.5 and NZ = 0.5 will give you a

plane at a 45° angle from the horizontal if your Z/X aspect ratio is 1/1. If you have a high Z/X aspect ratio, the cutting plane may appear more vertical than you expect. To see what your Z/X aspect ratio is, use **View | Aspect Ratios and Scaling**.

6. Once you have set the initial cutting plane, and the dialog box is closed, you can move the cutting plane as follows. Make sure you are in Probe mode by clicking the **Probe mode**  button on the **Mode** toolbar. Hold down SHIFT key, click the left mouse button while the cursor is in the view, and drag the mouse up and down, moving the cutting plane back and forth along its normal direction. For vertical cutting planes, you can rotate the cutting plane by holding down the CTRL key and clicking and dragging the mouse left and right.
7. To turn off the cutting plane, clear **Use Cutting Plane** and then click **OK**.

Modifying the Color Scale

Color is used to represent ranges of values. The color scale is only available (active) when a property is displayed. You can select from three types of color scales: **Linear**, **Logarithmic**, and **General** and three types of color systems: **RGB** (red, green, blue), **HSV** (hue, saturation, value) and **Gray** (black and white).

Modifying a Linear or Logarithmic Color Scale

The Color Scale shows you the value of a reservoir property in a specific grid block in both the 2D and 3D views. You may change the colors used, whether the scale is linear or logarithmic, and the range of values covered by the scale. Grid blocks with values outside the range are not displayed.

To set a linear or logarithmic color scale for each reservoir property:

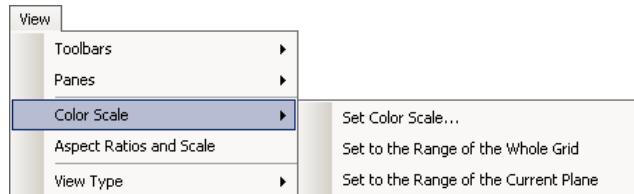
1. Select **Color Scale** from the **View** menu. The following menu items appear for 3-D view:



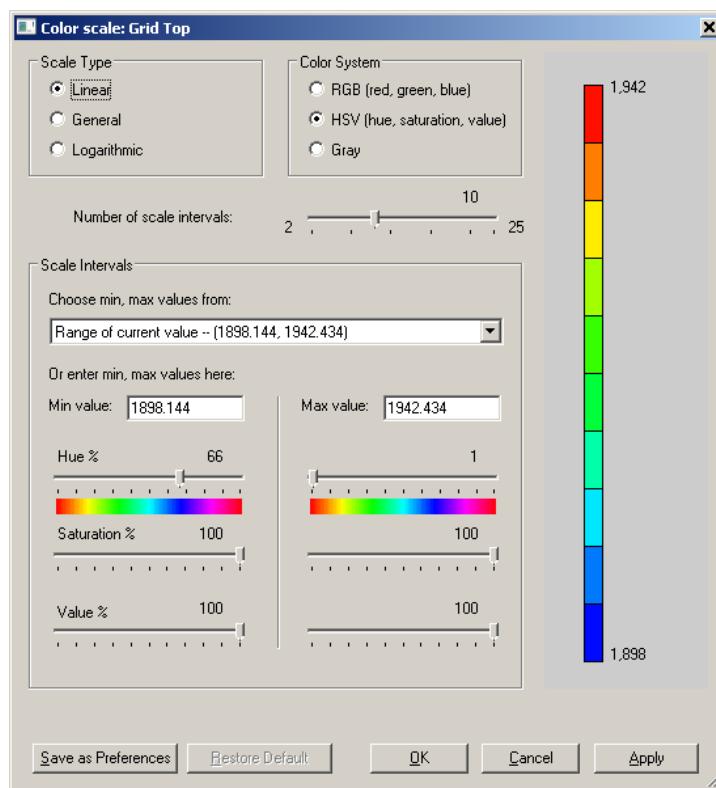
2. Select **Set to the Range of the Whole Grid** to set the color scale minimum and maximum values to the range of the whole grid for the current reservoir property.
3. Select **Set to the Range of the Cutting Plane** to set the color scale minimum and maximum values to the minimum and maximum value of the current reservoir property on the cutting plane.
4. Select **Set to the Range of the Slabs** to set the color scale minimum and maximum values to the minimum and maximum value of the current reservoir property on all the selected slabs.

5. Select **Set to the Range of the User Selections** to set the color scale minimum and maximum values to the minimum and maximum value of the current reservoir property on all the selected grid blocks.

In a 2-D view the following menu items appear:



6. Select **Set to the Range of the Whole Grid** to set the color scale minimum and maximum values to the range of the whole grid for the current reservoir property.
7. Select **Set to the Range of the Current Plane** to set the color scale minimum and maximum values to the minimum and maximum value of current reservoir property in the current plane.
8. Select **Set Color Scale** to open the **Color Scale** dialog box, through which you can set specific values for the minimum and maximum, or change other aspects of the color scale (Linear is the default).



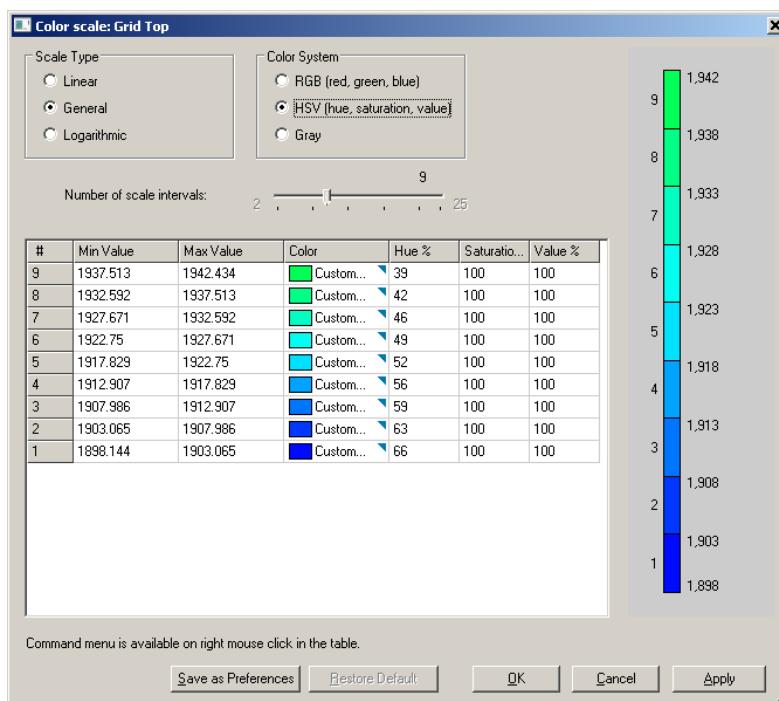
9. **Min Value** and **Max Value** default to the current selected lowest and highest values for the current reservoir property. You can change these to specific values.
10. You can also set the range of the scale to the range of the whole grid, to the range of the current cutting plane, or to the range covered by the currently selected blocks.
11. In **Number of scale intervals**, click and drag the slider to the right to get more intervals and to the left to get fewer intervals.
12. In **Color System**, select the system you want to use: **HSV** (Hue Saturation Value), **RGB** (Red-Green-Blue), or **Gray** (gray scale).
13. You can move the sliders to change the endpoints of the color components on the scale, and to adjust the colour values.
14. Click **OK** or **Apply** to accept your selections and redraw the image with your specifications.

Note: In the **Logarithmic** scale, interval division is done logarithmically and **Min value** must be greater than zero.

Setting or Modifying a General Color Scale

To set a general color scale for each reservoir property:

1. Select **Color Scale** from the **View** menu to open the **Color scale** dialog box and then select **General** in the top-left corner. Adjust the width of the dialog box to see all table columns. The **General Color Scale** allows customizing color and Min-Max values for each interval in the scale.



2. The **Number of sale intervals** slider is disabled for the **General** color scale to avoid accidental resetting the color and interval division to defaults. There are a number of commands available from the context menu to change the number of General scale intervals (as described below) without resetting the defaults. If you still prefer to use this slider than switch to Linear or Logarithmic scale type, adjust the slider and switch back to the General scale.
3. In **Color System**, select the system you want to use: **HSV** (Hue Saturation Value), **RGB** (Red-Green-Blue), or **Gray** (gray scale).
4. Each row in the table represents a color scale interval. You can edit the Min and Max values and select any color you wish by changing data in the cells. Make sure the **Max** value is always greater than **Min** value and that scale intervals do not overlap.
5. Right-click in the table to open a context menu with commands for modifying the current settings:



Add Interval: Adds a new interval at the top of the table (end of color scale).

Insert interval: Inserts a new interval above the select grid row.

Delete interval(s): Deletes selected table rows and scale intervals.

Split interval(s): Splits the selected intervals into a number of smaller ones.

Combine intervals: Combines selected adjacent intervals into a single interval inheriting the color of the lower one.

Interpolate intervals: Recalculates colors for all intervals using linear interpolation between the first and last colors on the scale to get even color spread. With this command, you can set only the top and bottom colors and quickly interpolate the rest.

6. Click **OK** or **Apply** to accept your selections and redraw the image with your specifications.

Saving Color Scale Settings in Preferences

You can save your settings for a property in the preferences by clicking **Save as Preferences**. You can restore the default settings by clicking the **Restore Default** button.

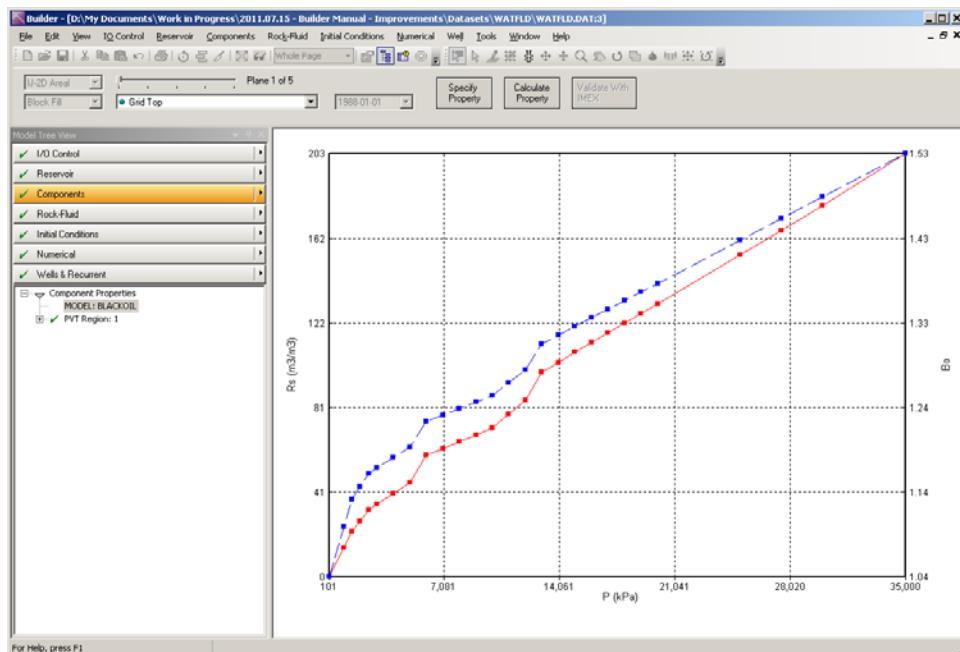
Viewing and Editing Tabular Data

In addition to the dialog boxes used for viewing and editing data, Builder provides plots of tabular data (PVT, VOT, COT, BOT tables, etc.) in a special view window called the plot view. The plot view consists of a plot tree view which lists the plots you can display. The rest of the plot view displays the currently selected plot in the plot tree view.

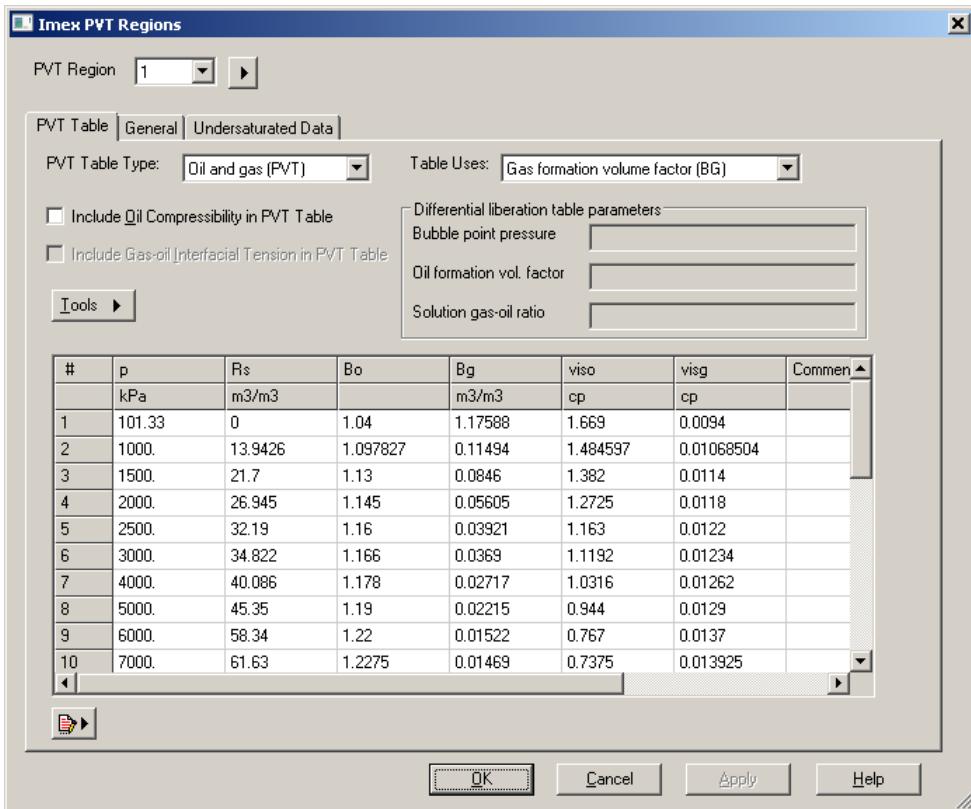
The plots are automatically created as soon as data is available, which takes place when you open a file or create new data.

The plot view becomes the active view when you:

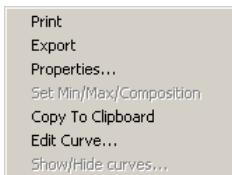
- Click **Components** or **Rock-Fluid** on the main tree view; or
- Select one of the correspondingly named menu items



If you double-click on the plot, a dialog box containing the data in the plot is launched, as shown below. You can now interactively modify the plot or the dialog box data. The data in the plot and the dialog box are synchronized.



A context menu opens up if you right-click the plot:



You can print the plot, export it to a file or copy to clipboard. Limited display properties editing capabilities are available through the Properties sub-menu. The legend in the plot can be moved around.

If you have closed the plot view, you can reopen it by selecting **View | Open Plot view**.

If you have closed the plot tree view, you can reopen it by selecting **View | Show/hide plot tree view**.

You can also go directly to the plot view by selecting its window from the list under the Window menu.

Printing and Saving Images

In general, all views are in “what you see is what you get” mode; that is, the printed output should look like the screen display. This is true for both 2D and 3D displays. Prior to printing, you should set the display as you want your printed output to look (i.e., select fonts, change or move titles, and so on). Also prior to printing, you should make any necessary changes to your page setup and printer selection.

If you are printing to a printer that does not support color (that is, black and white only), you may wish to change your color scale to use a gray shade scale. See [Modifying the Color Scale](#) for more information.

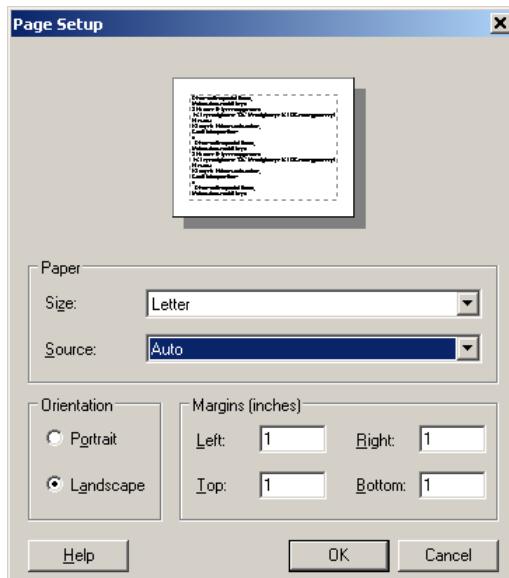
You can save the current view as an image file in one or a number of supported formats. Many office applications such as Microsoft Word and PowerPoint can import and display images. Using this facility, you can include images produced by Results or Builder directly into reports or presentations.

The supported image formats are Enhanced Windows Metafile (*.emf), bitmap (*.bmp), JPEG file interchange format (*.jpg), tagged image format (*.tif), and portable network graphics (*.png).

Changing the Selected Printer and Paper Size

To change the selected printer or plotter and paper size:

1. Select **File | Page Setup**. The **Page Setup** dialog box will appear.



2. When printing, a dialog box will appear allowing you to select from the printers that have been installed on your computer. Select the desired printer, and then click **OK**.

3. In the **Page Setup** dialog box, select the **Size** and **Source** from the drop down selection lists in the **Paper** frame.
4. In the **Orientation** frame, choose **Portrait** or **Landscape**.
5. Enter new values to the **Margins** if you wish to change these.
6. Click on **OK** to confirm your changes, or **Cancel**.

Printing the Current View

To print the current view:

1. Select **Print** from the **File** menu. The **Print** dialog box appears.
2. The **Print** dialog box will allow you to change printers; however, if you also want to change the paper size, you should use the **Page Setup** dialog box prior to using the **Print** dialog box.
3. Click **OK** to print, or **Cancel**.

Specifying Bitmap Resolution in 3D Printing

Printing 3D views is accomplished by rendering (displaying) the 3D image to a bitmap, then copying the image to the printer. For small paper sizes, the bitmap can have the same resolution as the printer, yielding the best image possible. However, for larger plots at full plotter resolution, a full resolution bitmap would require a significant amount of memory, and the drawing routines would take significant time to complete. To reduce this resource requirement, the maximum bitmap size is limited by default. If you need to improve the print quality for large 3D printout, you will have to increase the maximum bitmap size.

To increase the maximum 3D bitmap size:

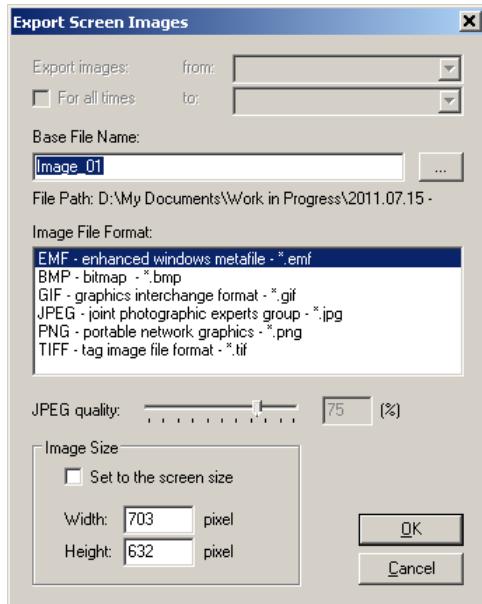
1. Move the cursor over the view to be printed and right-click. A pop-up (context) menu appears.
2. Click **3D Settings** from the menu. The **Builder Properties** dialog box opens to the **Model View | 3D Settings** page.
3. In the **Print Resolution** area, enter a value for **Maximum Pixels in X**.
4. Click **OK** to confirm your changes, or **Cancel**
5. Use **File | Print** to print with the new resolution and see if the printed image is improved.

Saving the Current View as an Image File

To save the current view as an image file:

1. Display the view you want to capture.

2. Select **Tools | Export Image** from the menu bar. The **Export Screen Images** dialog box appears:



3. Enter the file name for the saved image. Select the desired image format from the **Image File Format** selection list.
4. As necessary, change the size of the saved image.
5. Click **OK** to save the image, or **Cancel**.

Improving the Quality of Saved Images

Images are saved at the current screen resolution of the view. To produce higher quality saved images, you can increase the view size. The quickest way to do this is to make the Builder or Results 3D main window full screen and to make the view to be saved full screen. In addition, you can select **View | Paper Zoom | 200%** (or other zoom setting) to increase the resolution of the view, prior to saving the image.

Working With Multiple Views and Documents

Opening a Second View

To create and control a second or subsequent view:

1. Select **New View** from the **View** menu. A second view window will appear within the main Results 3D or Builder window.
2. To see both windows at once, select **Tile Horizontally** or **Tile Vertically** from the **Window** menu.

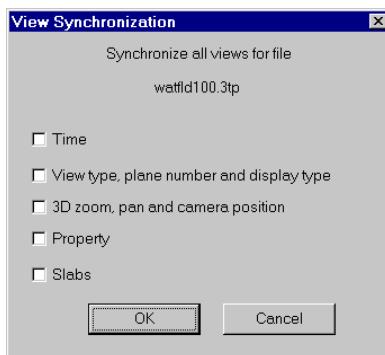
- To change the **Display Settings**, property, or time of one of the views, click on the view to make it the current view, then use the menus and dialog boxes as you would if you only had one view. The changes you make will only affect the current view unless you have synchronized the views (see next section).

Synchronizing Views

You can open two or more views, then synchronize the views so that certain types of changes made in one view will also be made in other views.

To synchronize views:

- Select **Synchronize Views** from the **View** menu, or right click to pop up the context menu, and then select **Synchronize Views**. The **View Synchronization** dialog box is displayed:



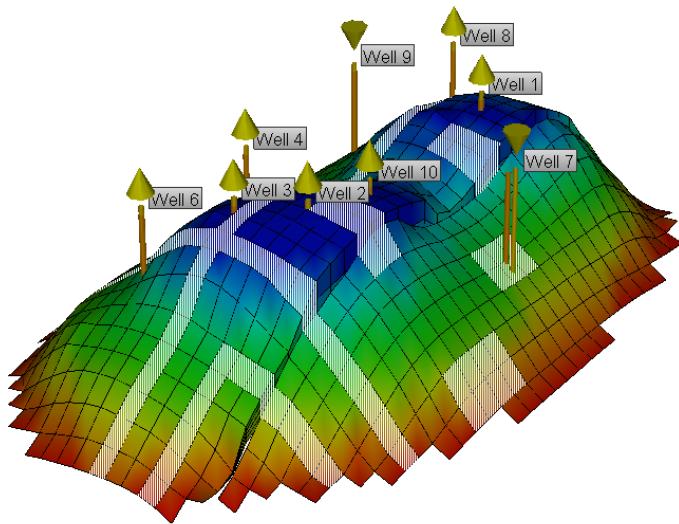
- Select the items you want to synchronize between views. For example, if you want all views to show properties and wells at the same time, select **Time**.
- Click **OK** to confirm your changes, or **Cancel**.

Interactive Block Selection (3D Only)

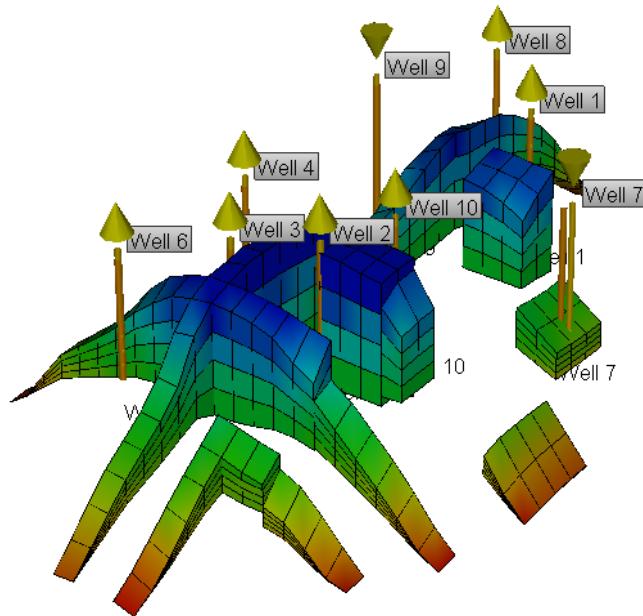
Select mode can be set from the **Modes Tools** toolbar or the 3D context menu by clicking on the **Select** menu item.



When this mode is set you can select a single block or various block combinations. To define the combination of block(s) you want to select, use the options in the **Selection** docking tool (which are described in next section of this manual). The options include selection by Block, Pillar, Layer, Slab_I, Slab_J, Stripe_I, Stripe_J or Region. Put the mouse cursor over a block and then drag the mouse while holding down the left mouse button.



After grid blocks have been selected you can use the **Hide Selected** or **Hide Unselected** buttons on the **Selection** docking tool to set only those blocks visible which you currently want to analyze. Subsequent set of blocks can be selected from the current selection following the same steps. To cancel the current selection, press the ESC key. The highlighting will disappear from any blocks that were selected. To show all grid blocks click **Show All** in the **Selection** docking tool.



The above shows an example of the 3D Reservoir View after hiding unselected blocks.

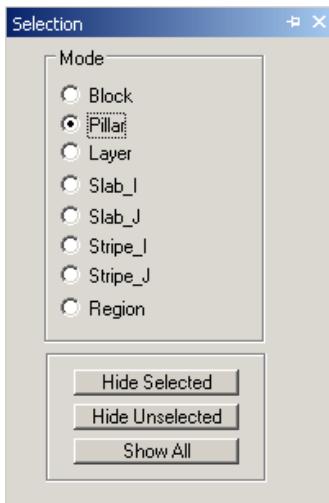
Docking Tools (3D Only)

The Docking Tools enable you to quickly access parameter settings which are used more often. You can show or hide different parts of the grid, change property settings or choose to display well/trajectory information.

There are four Docking Tools: Selection, 3D Properties, Wells and Statistics.

Selection

The **Selection** docking tool is used to set the grid block selection type and also to hide selected or unselected blocks. The **Show All** option can be used to undo the selections made.



3D Properties

The **3D Properties** docking tool allows you to show or hide different parts of a grid, and change 3D graphics properties.



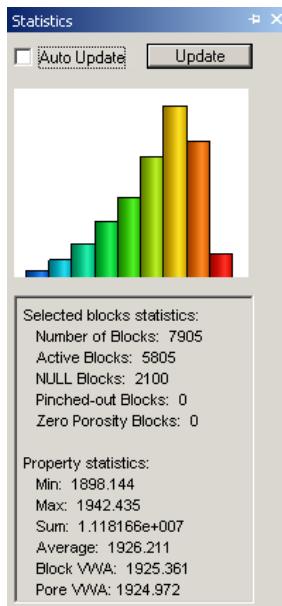
Wells

The **Wells** docking tool is used to show or hide wells and well trajectories and set different controls related to wells. This feature allows you to easily turn off wells in large models to make Builder more responsive.



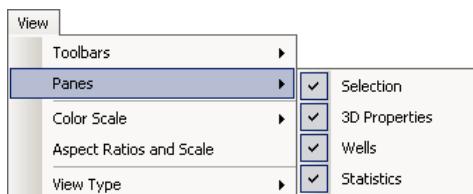
Statistics

In the Statistics docking tool you can see histogram and summary information for all of the blocks displayed for the current property. The histogram displays the number of blocks for each property value range. Colors of histogram bars correspond to colors of the color scale. When a grid property is changed, Statistics can be update manually by clicking on the **Update** button. Select the **Auto Update** check box to update statistical information automatically each time a new block selection has been made, or when the property shown has changed.



Opening and Closing Docking Tools

All of the Docking Tools can be opened or closed by accessing the Panes options through the View menu.



Formula Manager

Overview

You can calculate array properties based on other properties in the dataset. Builder provides a dialog box (refer to [Formula Manager Dialog Box](#)) through which you can create the formulae that describe these functional relationships. Refer to [To create a formula](#) and [To edit a formula](#).

Once a formula has been created, it can be assigned to an existing or custom property. Refer to [To apply a formula to a property](#).

Once you have applied the formula to a property, you can view the property in the Builder view. Refer to [To view the new property](#).

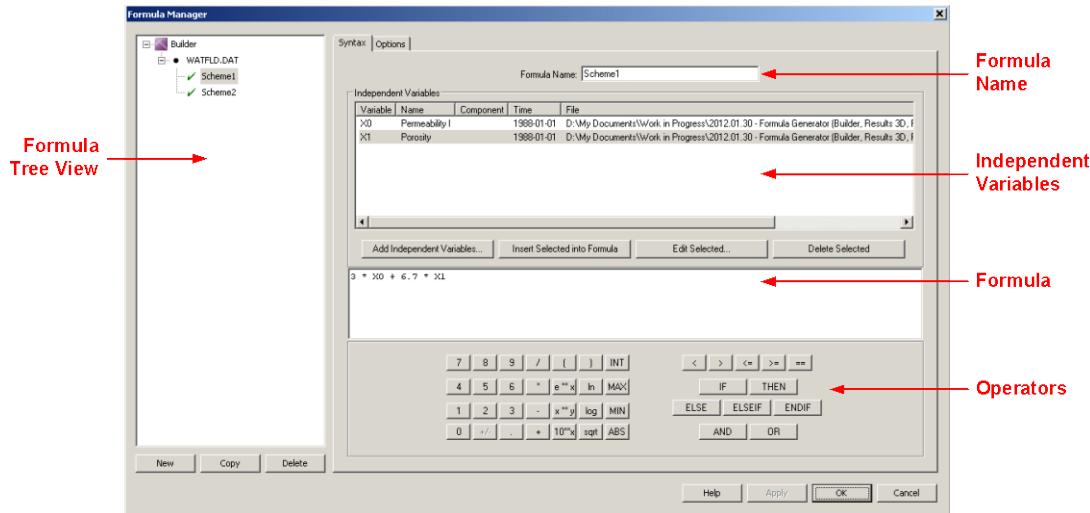
You can copy a formula to a new formula, in which case the formulae are maintained independently. Refer to [To copy a formula](#) for more information.

If you delete a formula, the property values that were generated are still saved with the dataset. Refer to [To delete a formula](#).

Simple averages and pore volume weighted averages can be calculated for any displayed property. For information refer to [Calculating Grid and Property Statistics](#).

Formula Manager Dialog Box

Open the Formula Manager dialog box through **Tools | Formula Manager** in the menu bar:



Formula Tree View: Shows the Builder dataset (*.dat file) that is currently open and the formulae that have been created. A ✓ check mark indicates that a formula is ready for use, and a ✘ indicates that it is not.

Formula Name: The default name is *SchemeN* (*Scheme1*, for example), where *N* is chronologically assigned. The formula name has to be unique. You can change the formula name to one that is suitable for your analysis.

Independent Variables: This table shows a list of the independent variables selected for use in the formula. You can add, edit and delete independent variables. For each independent variable, the table includes the following information:

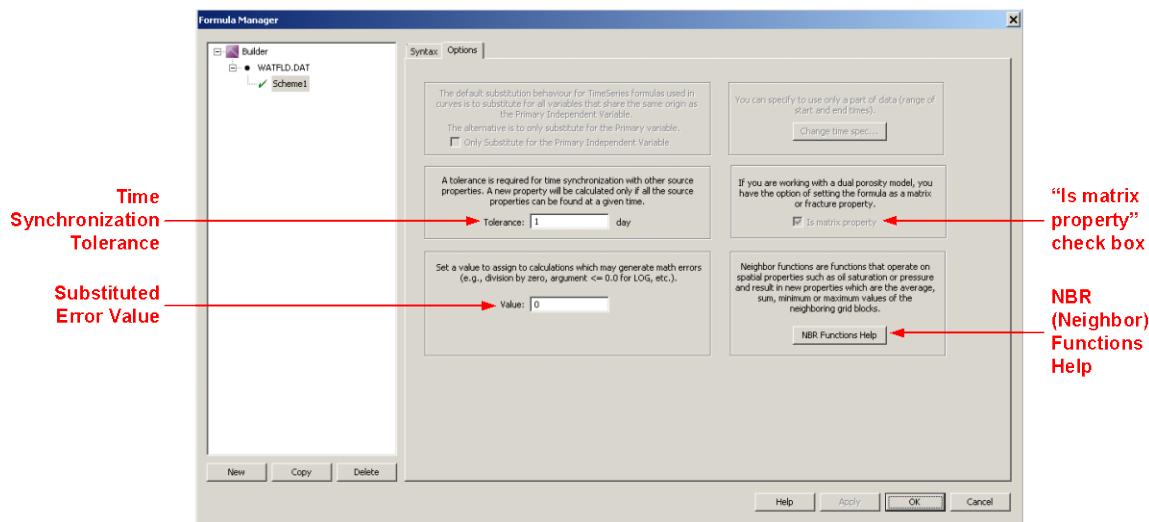
- **Variable:** Indicates the variable symbol, *X0* for example.
- **Name:** This name is assigned from the dataset.
- **Component:** If the dataset contains components for the independent variable, these will be displayed in the column.
- **Time:** The time at which the independent variable is taken.
- **File:** This is the folder and name of the file containing the independent variable data used in the formula.

Note: In the above example, *Permeability I* data at time *1988-01-01* is used for *X0*.

Formula: The formula is entered by typing the formula directly or by using the buttons below the **Formula** area.

Operators: Operators that can be used to define the formula. For further details, refer to [General Rules and Guidelines](#).

The **Options** tab displays several Formula Manager option settings:



The following options are available:

- **Time Synchronization Tolerance:** Enter the tolerance required for time synchronization of source properties. A new property will be calculated only if all the source properties can be found at a given time. The default value is 1 day. In this case, if a derived property D is a function of independent variables A , B and C , then D will be calculated at time $t = t_n$ if and only if there are values for A , B and C within 1 day of t_n .
- **Substituted Error Value:** Set the value that will be assigned to calculations that generate mathematical errors, such as division by zero or logarithm arguments less than or equal to zero. The default is 0.
- **Is matrix property:** This check box is selected and disabled in Builder, which means a MATRIX (not a FRACTURE) property is calculated.
- **NBR Functions Help:** Click this button for information about this option. Refer to [Neighbor functions](#) for further information.

General Rules and Guidelines

Terms

Formula: The functional relationship expressed using symbols representing arithmetic, logical and logarithmic operators and other special quantities, and independent variables.

Source property: An existing property in the current or other simulation output file. It can be a property previously defined and calculated using Formula Manager.

Independent variable: Representation of the source property in the formula, denoted by X_n , $n = 0 \dots 99$

Dependent variable: A calculation which results in the creation of a temporary property with the name you input for the formula.

Operators

Arithmetic operators:

+, -, / (divide), * (multiply), ** (power), **SQRT** (square root)

Logarithmic operators:

LOG, **LN**, **EXP10** (10^{**x}), **EXP** (e^{**x})

Logical operators:

IF, **THEN** (as a delimiter), **ELSE**, **ENDIF** (as a delimiter), **AND**, **OR**, <, <=, >, >=,
==

Neighbor functions:

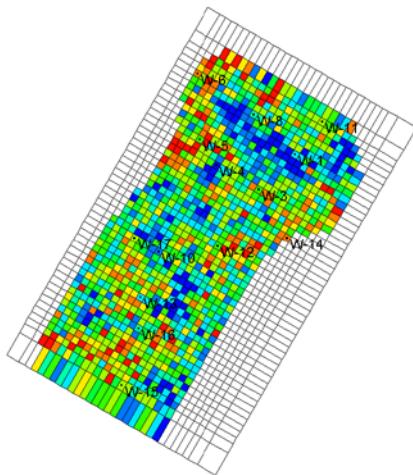
These are functions that operate on spatial properties, such as oil saturation or pressure, and result in new properties which are the average, sum, minimum or maximum values of the neighboring grid blocks. The function names and required arguments are as follows:

```
Average value function : NBRAVG( X, Method, Weight, i1, i2, j1, j2, k1, k2 )
Minimum value function : NBRMIN( X, i1, i2, j1, j2, k1, k2 )
Maximum value function : NBRMAX( X, i1, i2, j1, j2, k1, k2 )
Sum value function     : NBRSUM( X, i1, i2, j1, j2, k1, k2 )
```

| | | |
|----------|---|--|
| X | : | any spatial property or term evaluated to a spatial property |
| Method | : | ARI GEO HAR |
| Weight | : | NW BVW PVW |
| i1,i2 | : | range in negative/positive I direction (any integer or NI) |
| j1,j2 | : | range in negative/positive J direction (any integer or NJ) |
| k1,k2 | : | range in negative/positive K direction (any integer or NK) |
| NI,NJ,NK | : | number of fundamental blocks in I/J/K direction |
| ARI | : | arithmetic average |
| GEO | : | geometric average |
| HAR | : | harmonic average |
| BVW | : | block volume weighted |
| PVW | : | pore volume weighted |
| NW | : | no weight |

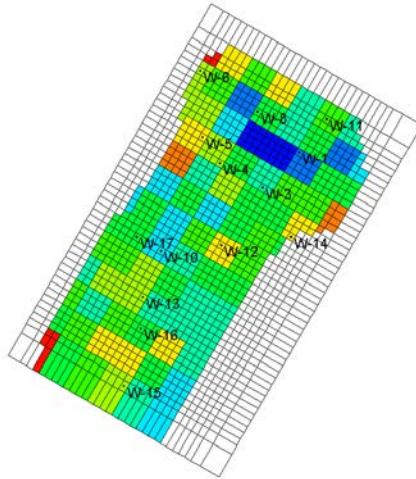
If the function name has a suffix of ‘S’, for example NBRAVGS, then the function is in stride mode, meaning the property values of all the grid blocks within the requested range are the same. This is equivalent to property upscale without grid upscale. If stride mode is used to calculate a property value, the resultant reservoir will typically look like a coarser quilt as a number of blocks will have the same value.

To illustrate, consider the following example. The original property values are shown below:



The Stride mode option could use these property values (X0) for calculating a new reservoir property which looks like the following:

Formula used: NBRAVGS (X0 , ARI , BVW , 1 , 2 , 1 , 2 , 0 , 0)



Other Functions:

(and) as delimiters for function arguments

INT (rounding of floats to ints; values up to and including i.5 will be rounded to i, those greater than i.5 to [i+1])

MAX, MIN, ABS (absolute value)

Xn (n = 0 ... 99) for independent variable

Formula Syntax

Only enter the right side of an equation as the formula. For example, if the formula is:

PERMK = PERMI * 0.01

enter it as:

X0 * 0.01

where X0 is the independent variable Permeability I.

Precedence and order of evaluation

Calculations are always carried out from left to right.

Precedence is defined as follows: * and / have equal precedence but are higher than + and - which have equal precedence. An expression within parentheses is evaluated completely and the result is treated as the single value in a binary operation.

The following have equal precedence

<, <=, >, >=, ==

IF Blocks

A formula can have only one IF block. An IF block completely defines a formula; that is, a formula containing an IF block can only have the **IF ... THEN ... (ELSEIF ... THEN) ... ELSE (... ENDIF)** statements.

An IF block must have an **ELSE** statement. An **IF** statement must be terminated by **THEN**. An **ELSE** statement may be terminated by **ENDIF**. An **ELSE**, **ELSEIF**, **AND** and **OR** statements cannot be present without an **IF** statement.

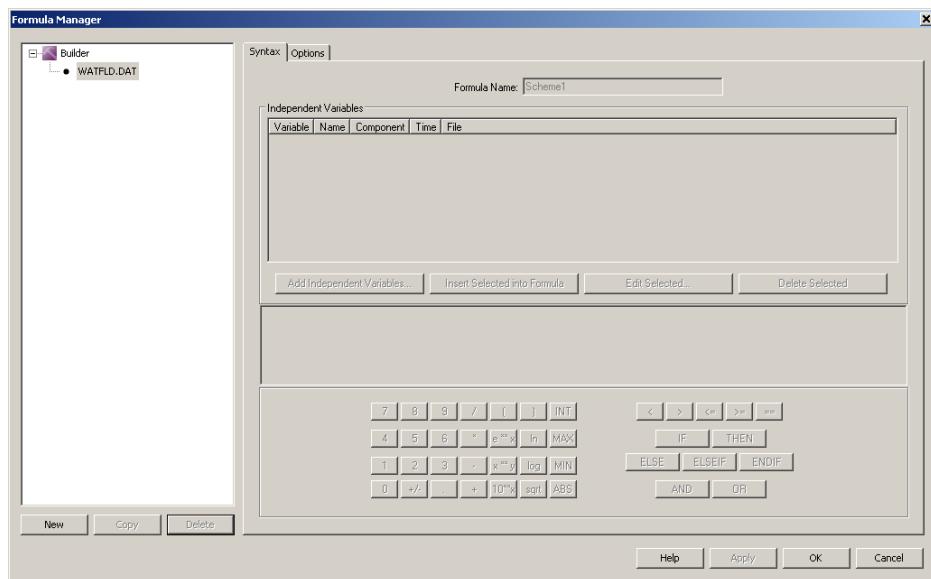
Other

- The following operators must be followed by an expression enclosed in parentheses: **LOG**, **LN**, **EXP10**, **EXP**, **SQRT**, **INT**, **ABS**, **IF**, **ELSE**, **ELSEIF**, **AND**, and **OR**.
- **MAX** and **MIN** operators are followed by two expressions in parentheses, separated by a comma, for example, **MAX (A, B)**.
- Spaces can be inserted between operators, functions and parentheses to make the formula easier to read and understand.
- A formula can span more than one line.

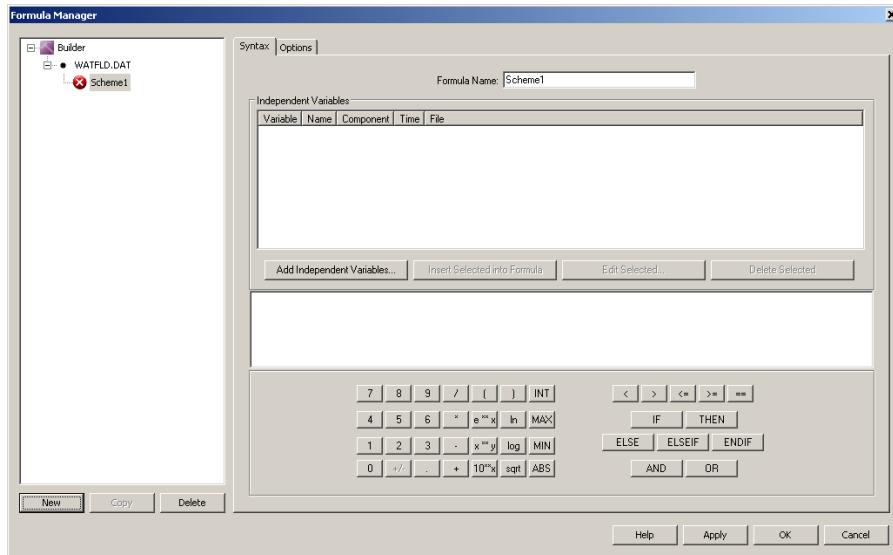
Using Formula Manager

To create a formula

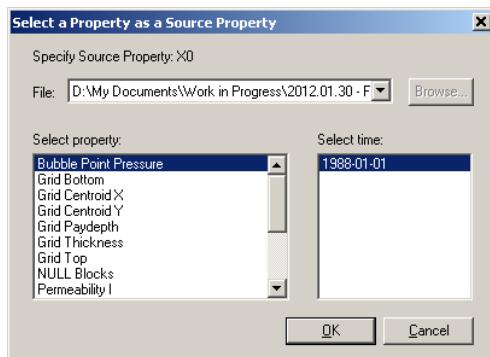
1. Select **Tools | Formula Manager** from the menu bar. The **Formula Manager** dialog box is displayed:



- Click **New** in the lower left. The controls in the **Formula Manager** dialog box are activated and a default **Formula Name** entered (*Scheme1* in the following example):



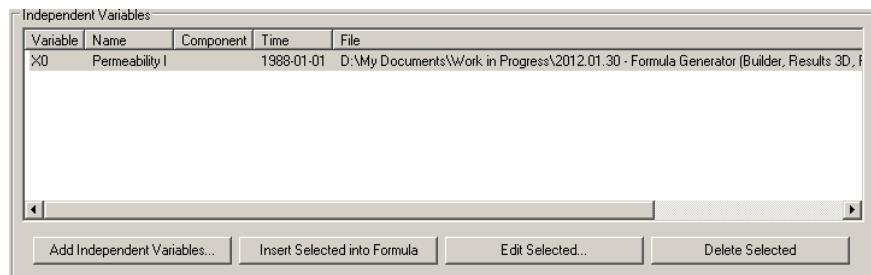
- As desired, change the name of the formula in the **Formula Name** box.
- Click **Add Independent Variables**. The **Select a Property as a Source Property** dialog box, listing available properties, is displayed. Through this dialog box, you add the properties that you want to use in the formula:



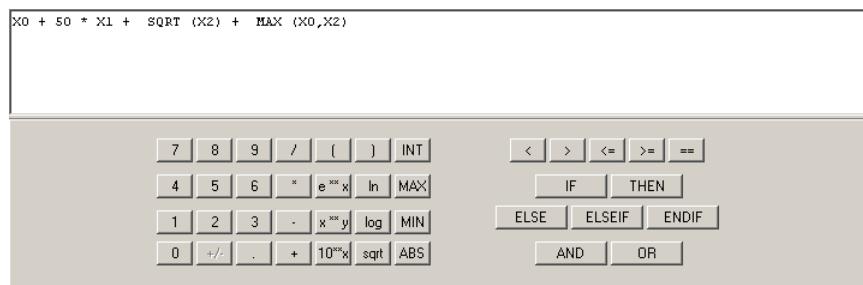
Specify the property as follows:

- File:** The file is the dataset and the **Browse...** button is disabled.
- Select Property:** Select the property that you want to use as the independent variable.
- Select Time:** For each property, the times at which data values are available are displayed. If multiple times are available, select the desired one.

5. After you have specified the required property, click **OK**. The independent variable will be assigned a symbol ($X_0, X_1, \dots X_n$) and will be displayed in the **Independent Variables** table of the **Formula Manager** dialog box as shown in the following example:



6. Repeat steps 4 and 5 for other independent variables, as necessary.
7. Once you have entered the necessary independent variables, enter the formula, as shown in the following example:



You can add source properties to the formula any number of times by either typing its variable symbol (X_0, X_1 , and so on) or by selecting the property in the list and clicking **Insert Selected into Formula**. You cannot delete a source property from the list unless you remove all references to it in the formula. You can edit the source property (change from one property to another) without removing references to it in the formula.

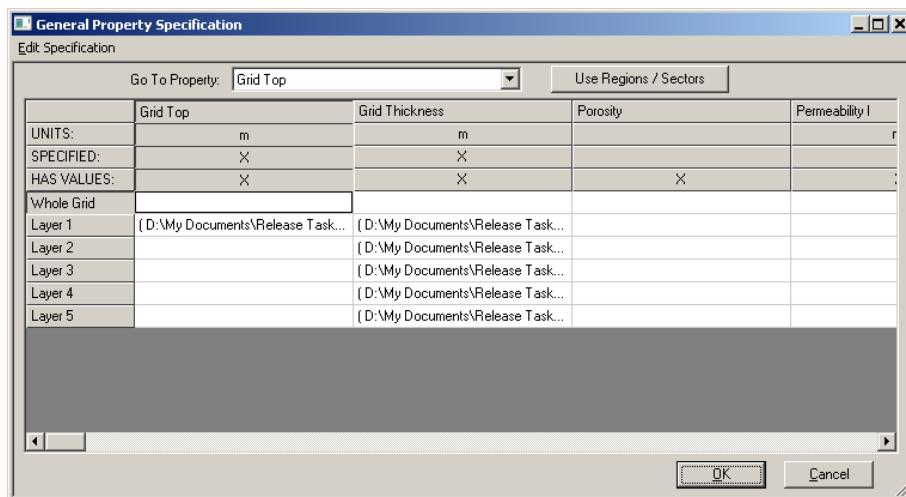
Examples of formulae

- Example 1: $1.23 * (X_0 ** 0.2)$
 Example 2: $\text{IF}((X_0 < 0.1) \text{ AND } (X_1 < 623)) \text{ THEN}(\text{INT}(X_1 / 10))$
 $\text{ELSEIF}((X_0 < 0.1) \text{ AND } (X_1 < 1200)) \text{ THEN}(\text{INT}(X_1 / 100))$
 $\text{ELSE}(\text{INT}(X_0 * X_1))$
 Example 3: $\text{MAX}((\text{LOG}(X_0 * X_1 * X_2)), \text{ABS}(X_0 - (X_1 / X_2)))$

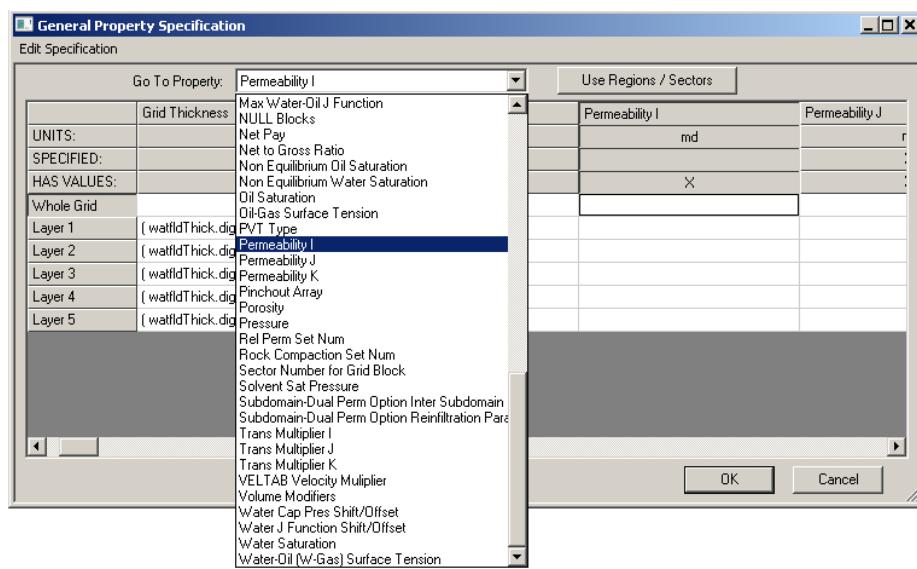
- Click **OK** or **Apply** to save the formula with the dataset. You will be prompted if you do not use all of the independent variables in the formula. If you proceed, the unused independent variables will be deleted from the table. You will be prompted with additional messages if criteria for creating the formula are not met.

To apply a formula to a property

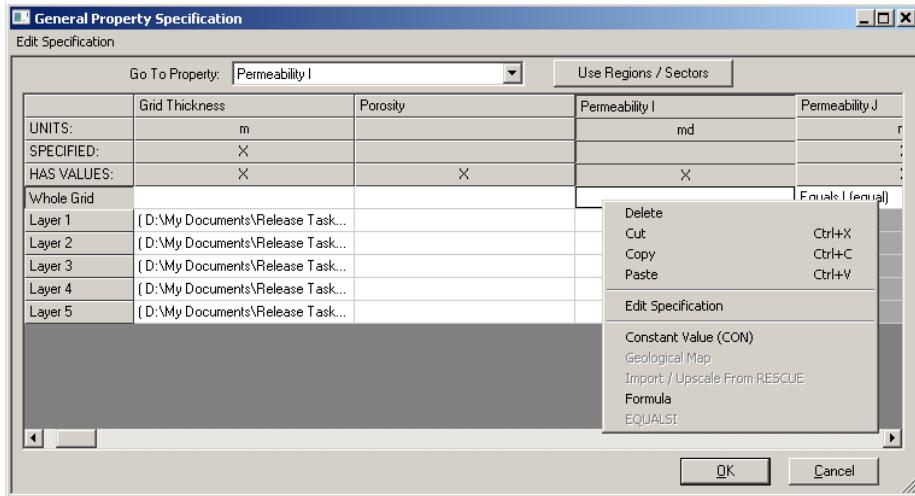
- Click the **Specify Property** button. The **General Property Specification** dialog box will be displayed. Through this dialog box, you can specify the formulae, maps, and constant values to be applied to individual layers or regions of the property:



- Through the **Go To Property** list, select the property for which you want to specify a formulae. In the following example, Permeability I is selected in the list:



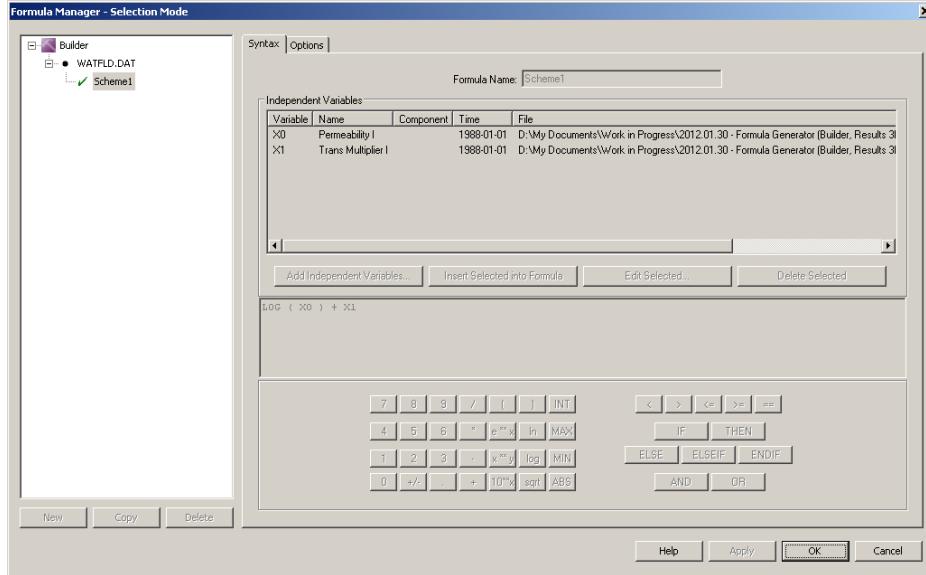
3. Right-click the appropriate cell in the table. In the following example, the **Permeability I | Whole Grid** cell has been right-clicked:



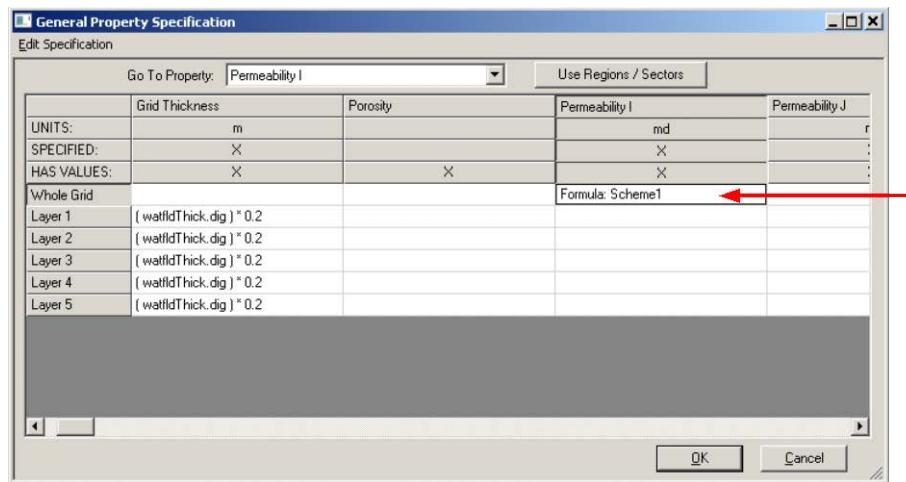
4. Select **Formula** in the list. The **Property Specification** dialog box is displayed, with the property described in the heading, as shown in the following example:



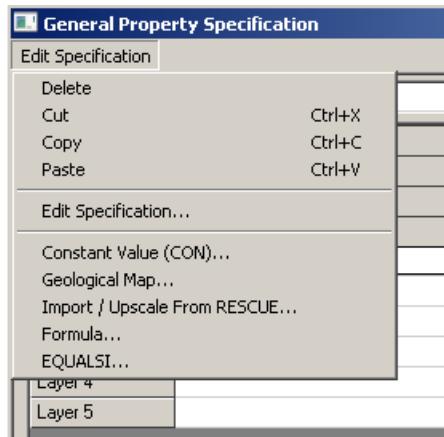
5. Click the **Formula** button. The **Formula Manager – Selection Mode** dialog box is displayed:



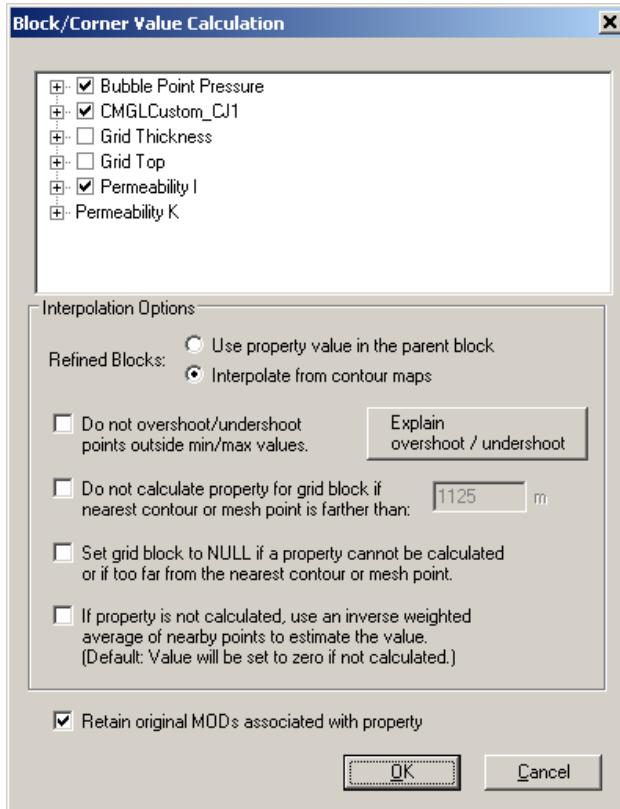
6. Select the formula in the tree view and then click **OK**. In the **Property Specification** dialog box, the name on the **Formula** button will be updated with the formula name. Click **OK**. The formula is inserted in the cell, as shown in the following example:



Note: At any time you can select a property in the **General Property Specification** dialog box and then click **Edit Specification** in the upper left to open a list operations for editing the property:



7. Click **OK**. The **Block/Corner Value Calculation** dialog box is displayed:

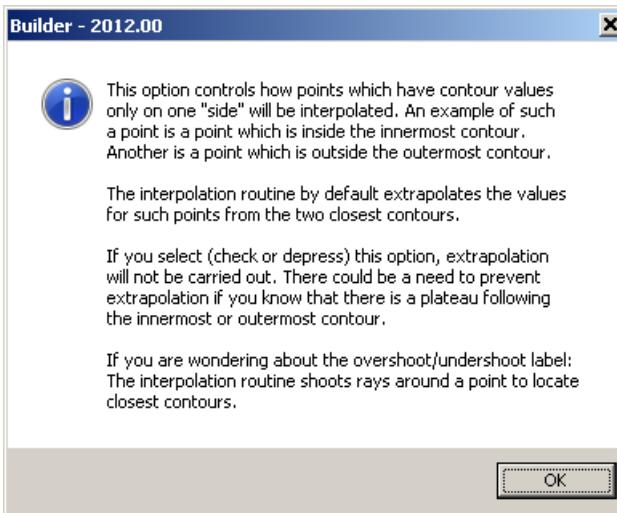


8. Configure the **Block/Corner Value Calculation** dialog box as follows:

- **Property Selection Area:** This area contains a list of properties that may need to be recalculated, because a formula has changed, for example. Select the properties that you want to be recalculated.
- **Interpolation Options:**

Select **Use property value in the parent block or Interpolate from contour maps**, as appropriate.

Click **Explain overshoot/undershoot** to view information about how points that have contour values on one “side” will be interpolated. Configure as necessary.



- For information about the use of the **Retain original MODs associated with property** check box refer to [Property Modifications \(MODs\)](#).

9. Click **OK** in the **Block/Corner Value Calculations** dialog box. Values for the selected properties will be calculated and stored when you save the dataset.

Notes:

- Values for properties based on formulas are determined when the formula is applied to the property. When you save the dataset, the formulas and the calculated values are both saved.
- At any time, you can click the **Calculate Property** button in the main Builder screen to open the **Block/Corner Values Calculation** dialog box. You may be prompted to do this if, for example, you have changed a formula.

To view a formula

To view a formula select **Tools | Formula Manager** from the menu bar. The **Formula Manager** dialog box will be displayed. In the tree view, click the formula name.

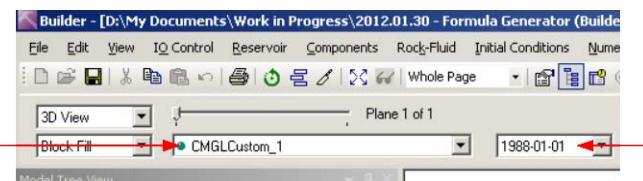
To edit a formula

To edit a formula select **Tools | Formula Manager** from the menu bar. The **Formula Manager** dialog box will be displayed. In the tree view, click the formula name. Make the necessary changes and then click **OK** or **Apply**. You will be prompted to click the

Calculate Property button in the main Builder screen to generate new values for the property.

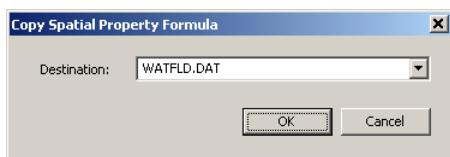
To view the new property

To view the new property, select the property and date as shown in the following example:

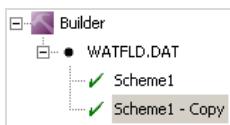


To copy a formula

1. In the tree view, select the formula that you want to copy.
2. Click **Copy** at the bottom of the tree view. The **Copy Spatial Property Formula** dialog box will be displayed:



3. Click **OK**. The formula is copied. The name of the new formula will be the same as the original formula, appended with “- Copy”, as shown in the following example:



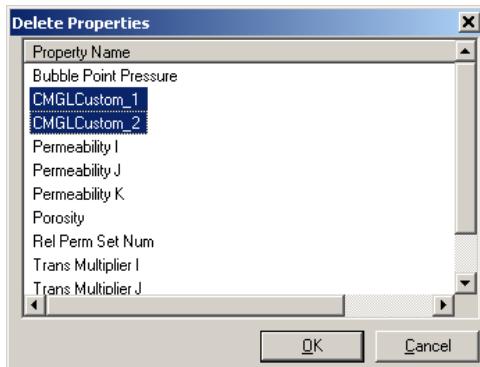
The new formula can be modified (its name can be changed, for example) without changing the original formula and vice versa.

To delete a formula

1. In the **Formula Manager** tree view, select the formula you want to delete.
2. Click **Delete** at the bottom of the tree view. You will be asked to confirm the deletion.

To delete a property

To delete a property, right-click **Reservoir** in the main Builder tree view and then select **Delete**. The **Delete Properties** dialog box is displayed:



Select the custom properties (or properties) you want to delete, as shown in the above example, and then click **OK**. When you save, the properties will be removed from the dataset.

Saving and Restoring Formulae and Formula-based Properties

- Each formula that is used to specify a property (regular as well as custom properties) is saved when you save the dataset.
- A regular property that is calculated using a formula specification is saved in the dataset.
- The values of custom properties are saved in the RESULTS section of the dataset. You can export the values of regular or custom properties to an ASCII format file through **Tools | Export Property Values**.

Calculating Grid and Property Statistics

Builder can calculate pore volumes, property averages, sums over all blocks and histograms for the currently displayed properties. A report is generated and displayed in a dialog box. The report can be saved to a text file.

To view statistics for the current property:

1. Select **Tools | Grid and Property Statistics**. The status bar will indicate the progress of the calculations. When complete, the **Builder - Property Statistics** report will be displayed.
2. To save the report, click **Save to file**. A file definition dialog will be displayed. Enter or confirm the file name, and then click **OK**.
3. Click **OK** to close the **Builder - Property Statistics** report.

Note: The histogram of the property uses the property ranges of the color scale divisions for categories. If you want to use a logarithmic histogram for permeability, change the color scale to logarithmic before running the property statistics.

Importing Geological and Well Trajectory Data

Overview

Builder can import geological data from two types of data sources: data files and data stored in a Public Petroleum Data Model (PPDM) compatible database.

Builder reads two types of geological data files: files giving x,y points along contour lines (contour maps) and files giving a property value at each node in a regular mesh of points (mesh maps). Both types of maps can contain fault lines (as sets of connected x,y points) and well names and locations. When you open a contour map, the contours will be displayed, and interpolation from the map will use the nearest contour lines to each interpolation point to determine the value at the interpolation point. When you open a mesh map, Builder will create contours for display purposes from the mesh but will use the nearest mesh nodes for the interpolation.

Builder supports a number of file formats for geological information as described in the following sections. The supported contour formats include WinDig format, Simgen format, CPS-3, and ZMAP CNTR format. The supported mesh formats include CMG mesh format, CPS-3, ZMAP GRID format, and EarthVision mesh format. You will need to ensure that your data is in one of these formats.

Builder can also import 3D well trajectories and compute the grid blocks containing perforated sections of these wells. A number of different formats are accepted, as described below. Coupled with the well trajectories, you can enter well log, picks, or other data versus measured depth along the trajectory.

Required Geological Data

If you are conducting a full field or regional study, geological data describing the structure and rock properties of the field must be available for an accurate simulation. This data can be in the form of hand-drawn or hard-copy maps that must be digitized into electronic form on the computer, or the data may already be in electronic form as files on the computer. You use the geological data describing the reservoir structure (formation tops, thickness, possibly bottoms) to position the simulation grid and to ensure that the number and size of grid blocks that you have specified will create a grid large enough to cover the area of the study. Once you are satisfied with the aerial position and coverage of the grid, you interpolate from structural and rock property maps to assign values to each simulation grid block. Typically, you need to have maps for top, thickness, porosity, permeability, and net pay for each geological layer in the reservoir. If you are working with a flow unit conceptualization, you

will need these maps for each flow unit. If your reservoir is dual porosity, you will need only one top and thickness map for each layer but separate (matrix and fracture) maps for other properties, as well as a fracture spacing map. If you do not have maps for one or more of the properties or layers, you can assign constant values to any layer of any property.

When doing a pattern or pilot study, you may wish to use a typical or idealized geology with uniform geometric and rock properties for each layer. You can construct a grid without opening a map file. Instead of interpolating grid block properties from maps, you assign a constant value to each simulation layer.

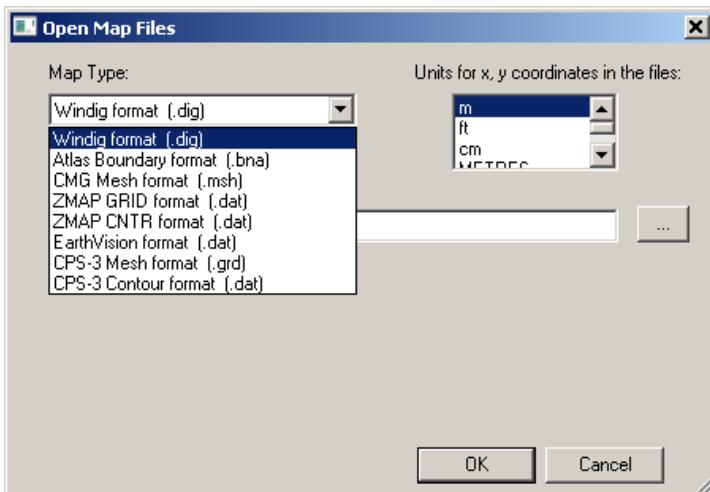
Importing Hard Copy Contour Maps

Builder reads exported files from the digitizing package: Didger from Golden Software, Inc., and WinDig. You can also create contour map files in the CMG Mesh format from Builder through **File | Create Map File**.

Didger is a digitizing program that can be downloaded from the Golden website at www.goldensoftware.com. You will need to register and pay a license fee to Golden Software before you can use Didger. Builder will import files exported in the Atlas Boundary format (extension “.bna”).

With WinDig, you can digitize your hard copy contour maps and create WinDig format contour maps that can be directly imported into Builder.

The Didger and WinDig files can be directly imported into Builder. After starting Builder, select **File | Open Map File**. The **Open Map Files** dialog box will be displayed:



In the **Map Type** drop-down list, select the appropriate format.

Supported Atlas Boundary File Format

Builder can import maps using the Atlas Boundary file format, containing contour, fault and well information. Atlas Boundary file (with extension “bna”) stores special information including polygons, polylines and points. The file entries are identified by primary and secondary IDs. No information on map units is included.

The general format of Atlas Boundary file is:

```
"Primary ID", "Secondary ID", type/length  
x1,      y1  
x2,      y2  
...  
xn,      yn
```

An example contour map in Atlas Boundary file format is:

```
"Contour",    "2900",      -23  
1346.70948,   1034.67152  
1346.20296,   1034.69648  
1345.7058,    1035.21584  
...           (total of 23 X and Y coordinates)  
1900.79736,   1128.47024  
"Fault",      "1",        -4  
1936.82184,   4165.40192  
1937.83452,   4165.35152  
1897.332,     -273.95536  
1897.84752,   -273.4864  
"well",       "OP1",      1  
1558.72428,   1293.3224  
"well",       "WI2",      1  
1605.21252,   2196.40304
```

Primary ID and Secondary ID are text strings. The quotes around the text strings are required if the text contains spaces or other delimiting signs. In Builder, the Primary ID must be:

- “Contour” or just first letter “C” for contour objects.
- “Fault” or just first letter “F” for fault objects.
- “Well” or just first letter “W” for wells.

IDs for Builder are not case sensitive.

The type/length is an integer which identifies the object and also the number of X, Y coordinate pairs following the identifier:

| | |
|-----|------------------------------------|
| 1 | point |
| > 2 | polygon |
| <-1 | polyline |
| 2 | ellipse (not supported by Builder) |

Each coordinate can be a number with or without a decimal. Each pair should start on a new line.

WinDig Contour Format

Most geological modelling programs have commands to output contour maps into ASCII text files. The output text file must then be reformatted into the WinDig file format to be read by Builder or Results 3D. For more information on the WinDig file format, see [Contour Map Entries](#) below.

When you create contour files directly from geological modelling packages, you may encounter the following issues:

1. Duplicate contour lines: Some packages may output duplicate contours (that is, they may repeat the same set of X, Y points and contour values). Although Builder attempts to remove duplicate lines when processing the contour map, duplicate lines lead to some increase in the processing time.
2. Gaps in contour lines: Some contour lines produced from geological modelling packages have “gaps”. A contour line that should be continuous is actually made to consist of a number of shorter segments with small spaces between segments. These gaps between contour lines can lead to anomalous interpolated values.
3. Lines consisting of a single point: Occasionally, a contour file produced from a geological modelling package is observed to contain some contour lines that consist of a single point or an identical point repeated a number of times. While code has been added to Builder to handle these cases, ideally, only true contour “lines” should be in the contour file.

Syntax

Each record in the contour file must end with a new line character. Within a record, you can enter arbitrary white-space characters between items. A decimal point is not mandatory in floating point numbers. If you use an exponent in a floating-point number, use E, e, D, or d in the exponent.

In contour maps, some contours are closed. If the first and last points in a contour are “close”, the contour is automatically closed. After the entire map is read, the bounding box of all points specified in the map is computed. A quantity called the map scale is computed. Two points are essentially “close” if the distance between them is less than the map scale $\times 10^{-4}$.

Contour Map Entries

A WinDig format contour file consists of a line indicating the coordinate units (“Meters” or “Feet”) and two lines of arbitrary text, followed by a separator record followed by sequences of contour, fault, and well records. You must delimit sequences of contour, fault, and well records with separator records. You do not need a separator record at the end of the file. The arbitrary text at the beginning of the file is ignored. Each separator, contour, fault, and well record must be on a line by itself.

A sample contour map appears at the end of this section.

There are four types of records in a contour file:

- Separator
- Contour
- Fault
- Well

Separator Record

A separator record looks like this:

1E20, 1E20, <integer>

You must enter the character string "1E20"; "1e20" does not work.

<integer> can be any valid integer.

Contour Record

A contour record specifies a point on a contour line and appears as follows:

<float>, <float>, <float>

<float> is any valid floating point number. The first float is the X coordinate, the second is the Y coordinate and the third is the contour value. The coordinates are always measured from the lower-left-corner. All contour records in a given sequence must have the same contour value. The third number cannot have the value -22; -22 is used in fault records.

Fault Record

A fault record specifies a point on a fault line and appears as follows:

<float>, <float>, -22

<float> is any valid floating point number. The first value is the X coordinate and the second value is the Y coordinate. The third value must be “-22”. If it is “-22.” (i.e., ending with a decimal point), then it will be interpreted as a contour line with value -22.

Well Record

A well record specifies the location of a completion and appears as follows:

<float>, <float>, <character string>

<float> is any valid floating point number. The first value is the X coordinate and the second is the Y coordinate. <character string> is a string of one to 16 characters giving the well name delimited by quotation marks. The characters can be anything except a new line character.

Well records in the same sequence can have the same or different names. All completions with the same name are collected in the same well trajectory.

Sample Contour Map

```
"Meters"
"T-left-X", "T-left-Y", "B-left-X", "B-left-Y", "B-right-X", "B-right-Y"
-4.6331E-05, 3965.09, -1.452141E-05, -3.630352E-06, 4793.34, -9.58161E-06
1E20, 1E20, 1
1905.352, 3264.652, 1005
2029.305, 3103.384, 1005
2146.144, 2903.98, 1005
2209.058, 2754.711, 1005
2265.23, 2601.082, 1005
2322.337, 2444.148, 1005
2424.756, 2252.355, 1005
2607.315, 1997.372, 1005
2747.745, 1784.885, 1005
2869.451, 1629.062, 1005
1E20, 1E20, 2
1502.037, 3140.076, 1010
1727.288, 2796.821, 1010
1866.314, 2584.361, 1010
1939.994, 2348.265, 1010
1962.182, 1955.597, 1010
1930.631, 1772.525, 1010
1953.85, 1623.257, 1010
2059.359, 1463.099, 1010
2722.144, 1322.82, 1010
2593.187, 1085.788, 1010
1E20, 1E20, 3
2.562778, 0, -22
394.626, 0, -22
802.856, 0, -22
1209.739, 0, -22
1601.801, 0, -22
1997.906, 0, -22
2412.873, 0, -22
2821.103, 0, -22
3218.555, 0, -22
1E20, 1E20, 4
2.562778, 388.0218, -22
394.626, 386.8077, -22
802.856, 388.2938, -22
1209.739, 389.7792, -22
1601.801, 389.9125, -22
1997.906, 388.6997, -22
2412.873, 388.8408, -22
2821.103, 391.6743, -22
3218.555, 391.8093, -22
1E20, 1E20, -1
1536.815, 440.7855, "well1"
1624.046, 777.6982, "well2"
2768.67, 1343.382, "well2"
1E20, 1E20, -2
2471.65, 1946.688, "well3"
```

Creating Quick Contour Maps Using Builder

You can create contour map files in the CMG Mesh format from Builder through **File | Create Map File**. The input data consists of property values (porosity, permeability, and so on) at the well locations (Well name, X and Y coordinates). You can do one or more of the following:

- Import the well locations from an existing map file
- Import the well locations from the current dataset in the view
- Type in the well names and locations
- Input locations using the mouse
- Import tops from trajectories

You can also output Fault locations to the map file. Fault information can be input using the mouse or by typing it in.

Three methods are available for creating contours using the data: Inverse Distance, Kriging and Trend.

Once the map file is created, the map files can be imported or used for property specification like any other map file. For additional information refer to [Creating Maps and Geostatistical Property Calculations](#).

CMG Mesh Format

Builder allows you to specify files containing data at the nodes of a mesh. The data can be produced by:

- Geological modeling packages
- Results Report - extracting the property values from a simulator output file (SR2)

Builder will contour the mesh points for display; however, the original data at the closest four non-NULL nodes are used for interpolation of properties for each simulation grid block. This is true even if the mesh is finer than the simulation grid (that is, there is no averaging of mesh nodes falling within the boundaries of a grid block).

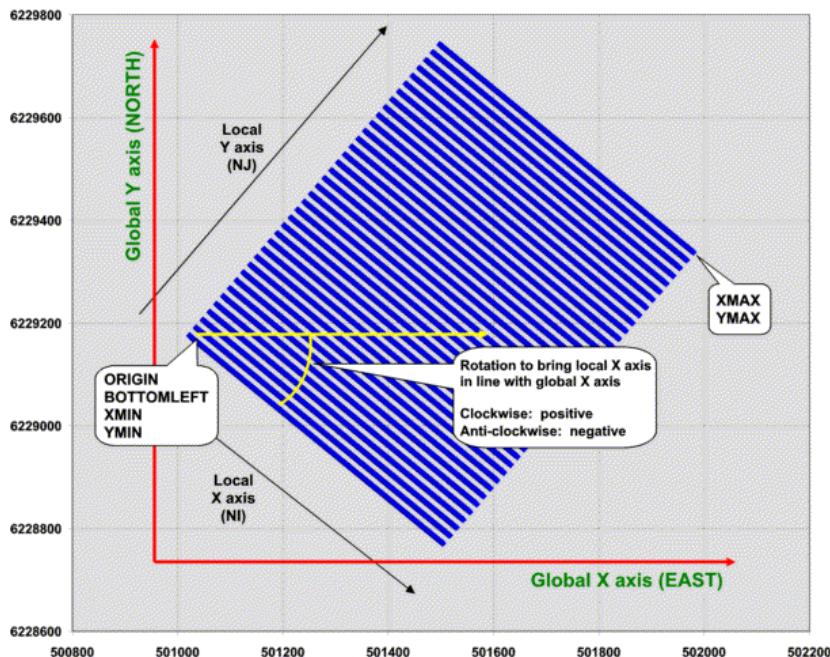
Creating Mesh Data Files

You can create a mesh data file in one of two formats:

- ASCII file format
- XYZ file format

To create a mesh data file in ASCII format, you must give information on the locations of the nodes and the order of the node data, and then provide a value for each node. A special value, called the NULLFLAG, indicates that there is no data for a particular node. NULL nodes are ignored in the interpolation and contouring routines.

To create a mesh data file in XYZ file format, you must provide information on the mesh node locations, and then provide data as triplets of x-coordinate, y-coordinate, and node value. The node on the mesh, closest to the given x-coordinate and y-coordinate, is found and the value is assigned to that node. Coordinates that fall outside of the mesh are ignored, and the value is not assigned to any node. You do not need to enter data for every node in the mesh. The XYZ file format handles unassigned data by assigning a NULL value to unassigned nodes. NULL nodes are ignored in the interpolation and contouring routines.



MESHTYPEFILE

Purpose: Signal the type of data input file.
Format: RESULTS MESHTYPEFILE ASCII XYZ
Definitions: ASCII: ASCII format
XYZ: XYZ format
Defaults: No defaults
Conditions: This must be in the first line in the file.

NI

Purpose: Number of nodes in the I (X) direction (i.e., number of columns).
Format: NI num_nodes
Definitions: Num_nodes: number of nodes
Defaults: No defaults
Conditions: Required keyword
Explanation: NI does not indicate the number of blocks. The number of blocks in the I direction is equal to NI-1.

NJ

Purpose: Number of nodes in the J (Y) direction (i.e., number of rows).
Format: NJ num_nodes
Definitions: Num_nodes: number of nodes
Defaults: No defaults
Conditions: Required keyword
Explanation: NJ does not indicate the number of blocks. The number of blocks in the J direction is equal to NJ-1.

ORIGIN

Purpose: Specifies the location of the first node (point) of the mesh relative to other points of the mesh.
Format: ORIGIN: TOPLEFT
BOTTONLEFT
Definitions: TOPLEFT: The first node is at the top left corner of the mesh
BOTTONLEFT: The first node is at the bottom left corner of the mesh
Defaults: No defaults
Conditions: Required keyword
Explanation: XMIN, YMIN, XMAX, YMAX, and ORIGIN fix the location, size, and order of the mesh nodes.

XMIN

Purpose: x (world) coordinate of first node (point) of the mesh
Format: XMIN value
Definitions: Value: coordinate value
Defaults: No defaults
Conditions: Required keyword
Explanation: This, along with YMIN, fixes the position of the first node of the mesh.

YMIN

Purpose: y (world) coordinate of the first node (point) of the mesh.
Format: YMIN value
Definitions: Value: coordinate value
Defaults: No defaults
Conditions: Required keyword
Explanation: This fixes the position of the first node (point) of the mesh.

XMAX

Purpose: The x (world) coordinate of the last node (point) of the mesh.
Format: XMAX value
Definitions: Value: coordinate value
Defaults: No defaults
Conditions: Required keyword
Explanation: This, along with YMAX, fixes the location of the last node (point) of the mesh.

YMAX

Purpose: y (world) coordinate of the last node (point) of the mesh.
Format: YMAX value
Definitions: Value: coordinate value
Defaults: No defaults
Conditions: Required keyword
Explanation: This, along with XMAX, fixes the location of the last node (point) of the mesh.

ORDER

Purpose: Specifies the order in which the data values should be read (for ASCII format only).
Format: ORDER: NEXTROW
 NEXTCOLUMN
Definitions: NEXTROW: The order of the data array is such that the next value in the array will be for the node in the same column but next row, unless the end of the column is reached.
 NEXTCOLUMN: The order of the data array is such that the next value in the array will be for the next node in the same row but next column, unless the end of the row is reached.
Defaults: No defaults
Conditions: Required with ASCII format
Explanation: The ORIGIN and ORDER keywords work together to indicate the order in which the data values are placed on the nodes. For example, suppose we have a mesh with NI=3 (three columns) and NJ=4 (four rows). Suppose that the data values are given as 1.0, 2.0, 3.0, 4.0, ... 12.0.

Explanation: If ORIGIN=TOPLEFT and ORDER=NEXTROW, then the values are placed on the mesh as follows:

1.0 5.0 9.0
2.0 6.0 10.0
3.0 7.0 11.0
4.0 8.0 12.0

If ORIGIN=BOTTOMLEFT and ORDER=NEXTCOLUMN, then the values are placed on the mesh as follows:

10.0 11.0 12.0
7.0 8.0 9.0
4.0 5.0 6.0
1.0 2.0 3.0

SPACING

Purpose: Specifies the type of spacing between nodes in I- and J- directions.

Format: SPACING: EQUAL
 VARI

Definitions: EQUAL: The nodes in the I-direction and J-direction are equally spaced.
The distance in the I direction = (length of I-direction side)/(NI-1).

VARI: The distance between nodes in the I- and J- directions is variable and is specified by the DI and DJ arrays.

Defaults: If DI and DJ are specified, the spacing type is set to VARI, otherwise it is set to EQUAL.

Conditions: Optional

DI

Purpose: Specifies the variable spacing between nodes in the I-direction (i.e., between columns of node points). This distance is constant along a column formed by two neighboring nodes.

Format: Array of values following the keyword

Definitions: Value: coordinate value

Defaults: No defaults

Conditions: Required with SPACING VARI. (NI-1) values expected.

DJ

Purpose: Specifies the variable spacing between nodes in the J-direction (i.e., between rows of node points). This distance is constant along a row formed by two neighboring nodes.

Format: Array of values following the keyword

Definitions: Value: coordinate value

Defaults: No defaults

Conditions: Required with SPACING VARI. (NJ-1) values expected.

ROTATE

| | |
|--------------|---|
| Purpose: | Specifies the angle in degrees by which the mesh is rotated with respect to the axes of the map. |
| Format: | ROTATE value |
| Definitions: | Value: The angle of rotation in degrees, e.g. 0.0 (-90 < value < +90) |
| Defaults: | 0.0 |
| Conditions: | Optional |
| Explanation: | This keyword is required if the mesh was rotated in the geological program with respect to the X and Y axis directions that you want to use in Builder. This keyword is distinct from RESULTS ROTATION keyword which is used in combination with RESULTS XOFFSET, RESULTS YOFFSET and RESULTS KDIR (refer to “Importing Data from a History Match Run Done By a CMG Simulator” in the manual). If these latter keywords are specified, ROTATE keyword is ignored. |

NULLFLAG

| | |
|--------------|--|
| Purpose: | Specifies the value to be assigned to a NULL node. |
| Format: | NULLVAL value |
| Definitions: | Value: A value to be interpreted as the corresponding node having no valid value (i.e., it is a NULL node) |
| Defaults: | 0.0 |
| Conditions: | Optional |
| Explanation: | NULL nodes are ignored in interpolation and calculating contours. |

MESHDATA

| | |
|-------------|--|
| Purpose: | Specifies the beginning of the data. |
| Format: | Array of values following the keyword |
| Defaults: | No defaults |
| Conditions: | Required. There can be only one set of data per mesh file. |

FAULT

| | |
|----------|---|
| Purpose: | Specifies the beginning of a fault definition. |
| Format: | FAULT number name 'name with spaces' x1 y1 x2 y2 . . xn yn |

| | | |
|--------------|--|---|
| Definitions: | number: name: name with spaces: x1 y1 etc.: | fault number alpha-numeric name alpha-numeric name including spaces. Only one of number, name or "name with spaces" is allowed world coordinates of the points on the fault |
|--------------|--|---|

| | |
|----------------|---|
| Defaults: | No defaults |
| Conditions: | Optional. One keyword required for each fault. |
| Explanation: | The fault is constructed by joining together successive points to form segments. For example, first segment is (x1, y1) → (x2,y2), second from (x2,y2) → (x3,y3)... |
| WELLS | |
| Purpose: | Specifies the beginning of well information |
| Format: | WELLS NAMEFIRST followed by one or more lines of the NAMEFIRST format: number x1 y1 name x2 y2 'name with spaces' xn yn WELLS XYFIRST Followed by one or more lines of the XYFIRST format: x1 y1 number x1 y1 number x2 y2 name xm ym 'name with spaces' |
| Definitions: | NAMEFIRST: Each record begins with a name/number/'name with spaces' followed by the x and y locations of the well. The name/number/'name with spaces' could be repeated. The repeated records do not have to be consecutive. XYFIRST: Each record begins with the coordinates of the well followed by a name/number/'name with spaces'. The name/number/'name with spaces' could be repeated. The repeated records do not have to be consecutive. name: alpha-numeric name name with spaces: alpha-numeric name including spaces. Only one of number, name, or 'name with spaces' is allowed. x1 y1 etc.: World coordinates of location of the well. |
| Defaults: | No defaults |
| Conditions: | Optional. One keyword required for each fault. |
| Explanation: | Optional |
| INCLUDE | |
| Purpose: | Specifies the file to be "included" at this location in the main file. |
| Format: | INCLUDE 'filename' |
| Definitions: | filename: Name of the file to be included. |
| Defaults: | No defaults |
| Conditions: | Optional |

Explanation: When the reader comes across this keyword, it opens filename file and continues reading that file. When it comes to the end of the file, it closes filename and reverts back to reading from the main file immediately following the INCLUDE filename words. This is a handy way of keeping the main file short and readable, while "including" files containing long arrays of data. You can have as many "include" files as you want (for data, faults, wells, etc.). Nesting of include files is not permitted.

Sample CMG Mesh Files

Format 1: ASCII File Format

The following is an example of an ASCII format mesh data file:

```
RESULTS MESHTYPEFILE ASCII
NI 36
NJ 60
SPACING EQUAL
ORDER NEXTCOLUMN
ORIGIN BOTTOMLEFT
XMIN 0
YMIN 0
XMAX 7200
YMAX 18000
** ROTATE 60
NULLFLAG -99999.9
MESHDATA
-99999.9 -99999.9      4.00   6.00   8.00 10.00 12.00 14.00 16.00 18.00
    20.00    22.00    24.00   26.00   28.00 30.00 32.00 34.00 34.00 32.00
    30.00    28.00    26.00   24.00   22.00 20.00 18.00 16.00   4.00 12.00
    10.00     8.00     6.00    4.00 -99999.90 -99999.90
-99999.90 -99999.90 4.00 6.00 8.00 10.00 12.00 14.00 16.00 18.00
20.00 22.00 24.00 26.00 28.00 30.00 32.00 34.00 34.00
32.00 30.00 28.00 26.00 24.00 22.00 20.00 18.00 16.00
14.00 12.00 10.00  8.00   6.00    4.00 -99999.90 -99999.90
...
(continue to a total of 36x60 data values)
```

Format 2: XYZ File Format

The following is an example of an XYZ format mesh data file:

```
RESULTS MESHTYPEFILE XYZ
NJ 135
NI 122
XMIN 464843.0
YMIN 5566479.
XMAX 468473.0
YMAX 5570499.
NULLFLAG .1000000E+11
SPACING EQUAL
ORIGIN BOTTOMLEFT
MESHDATA
INCLUDE 'xyz.inc'
```

```
FAULT 1
465923.9 5566479
466434.34 5567283.96
466981.24 5568146.43
467436.99 5568865.13
467747 5570499
WELLS NAMEFIRST
Well-1 466798 5568294
Well-2 465578 5567239
Well-3 464978 5570344
```

The following (x, y, value) triplet array is extracted from XYZ file xyz.in:

```
4.64843222e+05 5.56647899e+06 1.00000000e+10
4.64873222e+05 5.56647899e+06 1.00000000e+10
4.64903222e+05 5.56647899e+06 567.9
4.64933222e+05 5.56647899e+06 574.2
...

```

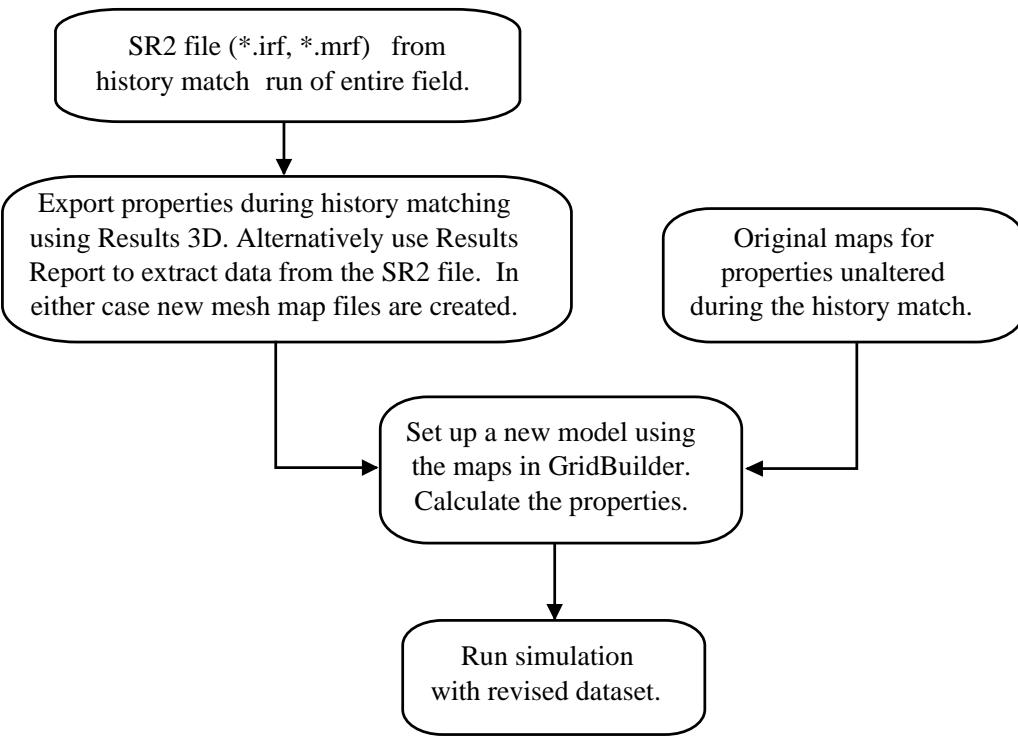
The number of values in the array is not fixed. Builder will create a mesh rectangle using XMIN, XMAX, YMIN, YMAX, and ROTATE. It will then create the mesh nodes using NI and NJ. Ideally, the (x and y) values in triplet array would match with the mesh node locations. If a point in the triplet array does not coincide with a mesh node, it will be assigned to the nearest node. If the number of number of triplets is greater than the number of mesh nodes, some of the triplets will remain unassigned and therefore unused. If the number of triplets is less than the number of nodes, some of the mesh nodes will remain unassigned. Unassigned nodes will carry the null value and will not participate in calculations.

Importing Data from a History Match Run Done by a CMG Simulator

Occasionally, a reservoir engineer may wish to conduct a simulation study covering a small portion of an existing field study. For example, the engineer may wish to study a new recovery process. This may require a fine scale study on only a small portion (say one five spot well pattern) due to the high cost of simulating the entire field with a finer grid.

In setting up the fine scale grid, the engineer may choose to redo the history match using only the region of interest. However, if there is an existing history match of the entire field, the engineer may wish to start the new simulation study from the ending point of the history match. If the new study uses a finer grid than the history match does, some work is necessary in obtaining the input grid properties and initial conditions to use in the new study.

The **Export** menu item in CMG's Results 3D can be used to extract information from a history match SR2 file and writing out the information in a format that can be read into Builder to prepare the grid properties and initial conditions for the new study.



Data Flow during Creation of New Grid

Alternatively, the same information can be written out using Results Report. The process described below gives details about how to extract the information using Results Report. To learn how to use the **Export** feature of Results 3D, refer to the *Results User's Guide*.

1. Use Results Report to extract information from the history match output SR2 file. To do this, it is first necessary to create a Results Report command file. (For more information on using Results Report, see the *Results User's Guide*.) The keyword to extract grid property information is PROPERTY-FOR. Suppose we wish to extract pressure, oil saturation and water saturation for the first two layers from the edge water flood example in the Results templates directory. The history match portion ends at a simulation time of 2586.0 days.

The Results Report command file would look like this:

```
*FILE 'watfld1.irf'    ** open the SR2 file for the history match
*OUTPUT 'pres1.dig'    ** assign the output file name
*PROPERTY-FOR 'PRESS'  2586.0 *MESHAYER 1
*OUTPUT 'pres2.dig'
*PROPERTY-FOR 'PRESS'  2586.0 *MESHAYER 2
*OUTPUT 'sol.dig'
*PROPERTY-FOR 'SO'     2586.0 *MESHAYER 1
*OUTPUT 'so2.dig'
*PROPERTY-FOR 'SO'     2586.0 *MESHAYER 2
*OUTPUT 'sw1.dig'
*PROPERTY-FOR 'SW'      2586.0 *MESHAYER 1
*OUTPUT 'sw2.dig'
*PROPERTY-FOR 'SW'      2586.0 *MESHAYER 2
```

This will create six files in the current directory. Each file will contain the grid block values for the specified properties, together with grid position information, so that these can be read into Builder in a similar manner to contour maps.

2. Adjust the offset and rotation of the extracted mesh files. The SR2 file may or may not contain information on how the history match simulation grid was positioned relative to the origin of the contour maps used to create the history match grid. If the extracted mesh files are being used in conjunction with the original contour maps for structure top and so on, it may be necessary to add in the offset and rotation information.

In each mesh map file produced by Results Report, you will see three lines as follows:

```
RESULTS ROTATION      0
RESULTS XOFFSET        0
RESULTS YOFFSET        0
```

If the history match simulation data file was produced with Builder, you should find three similar lines in your data file. In each mesh map file, change the rotation value of "0" to the rotation value found in your simulator data file, the X offset value of "0" to the X offset value found in your simulator data file, and the Y offset value of "0" to the Y offset value found in your simulator data file. Once you do this, the extracted mesh map contours will be positioned properly with respect to the contours in the original structure maps.

3. Start up the Builder program. You can now open the extracted mesh map files in a similar manner to any digitized contour map. These can be displayed or used for interpolating values (such as the initial conditions for pressure, oil saturation, and water saturation) for a new grid created with Builder.

The following keywords are used with mesh data produced from a simulation results file (SR2).

XOFFSET

Purpose: x (world) offset of the simulation grid origin. To be used if XMIN etc. are not available in world coordinates

Format: RESULTS XOFFSET value

Definitions: value: coordinate value

Defaults: 0.0

Conditions: Conditional keyword

Explanation: This keyword is required only to convert the simulation grid coordinate values into world coordinates and its meaning is specific to a combination of RESULTS generated values of XOFFSET, YOFFSET, ROTATION, and KDIR.

YOFFSET

Purpose: y (world) offset of the simulation grid origin. To be used if XMIN etc. are not available in world coordinates.

Format: RESULTS YOFFSET value

Definitions: value: coordinate value

Defaults: 0.0

Conditions: Conditional keyword

Explanation: This keyword is required only to convert the simulation grid coordinate values into world coordinates and its meaning is specific to a combination of RESULTS generated values of XOFFSET, YOFFSET, ROTATION, and KDIR.

ROTATION

Purpose: The rotation of the simulation grid in degrees. To be used if XMIN etc. are not available in world coordinates.

Format: RESULTS ROTATION value

Definitions: value: rotation in degrees

Defaults: 0.0

Conditions: Conditional keyword

Explanation: This keyword is required only to convert the simulation grid coordinate values into world coordinates and its meaning is specific to a combination of RESULTS generated values of XOFFSET, YOFFSET, ROTATION, and KDIR.

KDIR

Purpose: The K- or Z- axis direction of the simulation grid. To be used if XMIN etc. are not available in world coordinates.

Format: RESULTS KDIR UP
 DOWN

Definitions: UP: In the simulation grid, the first layer was at the bottom and the last layer was at the top.
DOWN: In the simulation grid, the first layer was at the top and the last layer was at the bottom.

Defaults: DOWN

Explanation: This keyword is required only to convert the simulation grid coordinate values into world coordinates, and its meaning is specific to a combination of RESULTS generated values of XOFFSET, YOFFSET, ROTATION, and KDIR.

The following is an example of mesh data extracted from an SR2 file:

(continued for 100x50 values)

Supported ZMAP Formats

Builder and Results 3D support four ASCII (text) formats, all of which can be exported from Landmark Graphics Corporation's Z-Map *Plus* program. Some other geological modelling packages also support these formats. The supported formats are the *default* formats of the ZMAP GRID file, ZMAP CNTR file, ZMAP FALT file, and ZMAP DWEL file, with the units for all X,Y coordinate in either "meters" or "feet". A simple well location file format may be used to locate wells on each layer if the DWEL file is not available. Note that a separate GRID or CNTR map is needed for each layer of each property. All the maps for a layer should share a common fault (FALT) file. As Builder currently only handles vertical faults, the fault file associated with the topmost layer of 'Grid Top' will be used to locate the faults on the simulation grid, however, you can input different fault files for each layer to control the interpolation for the maps in that layer.

The ZMAP GRID file will have a header similar to the following:

```
!
!      ZIMS FILE NAME : Brown Top 11
!      FORMATTED FILE CREATION DATE: APR 3 98
!      FORMATTED FILE CREATION TIME: 16:45
!
@Brown Top 11 HEADER , GRID, 5
15, 0.1000000E+31, , 7, 1
31, 27, 20750.00, 27250.00, -28500.00, -21000.00
500.0000, 0., 0.
@
0.1000000E+31 0.1000000E+31 0.1000000E+31 0.1000000E+31 0.1000000E+31
0.1000000E+31 0.1000000E+31 0.1000000E+31 0.1000000E+31 0.1000000E+31
0.1000000E+31 0.1000000E+31 0.1000000E+31 -2831.180 -2872.764
-2902.100 0.1000000E+31 0.1000000E+31 0.1000000E+31 0.1000000E+31
```

The ZMAP CNTR format will have a header similar to the following:

```
!
!      ZIMS FILE NAME : Brown Top 11
!      FORMATTED FILE CREATION DATE: APR 3 98
!      FORMATTED FILE CREATION TIME: 16:44
!
@Brown Top 11 HEADER , CNTR, 80, 1
X (EASTING) , 1, 1, 1, 1, 14, 7, 0.1000000E+31, , 14, 7, 0
Y (NORTHING) , 2, 2, 1, 15, 28, 7, 0.1000000E+31, , 14, 7, 0
@
0.1000000E+31 0.1000000E+31
-2900.000 1.400000
20750.00 -24731.65
20774.10 -24750.00
0.1000000E+31 0.1000000E+31
-2900.000 1.400000
21000.00 -25079.29
21038.94 -25250.00
```

The ZMAP FALT file contains (vertical) fault trace information and will have a header similar to the following:

```
!
!      ZIMS FILE NAME : mergflts-maintroncoso
!      FORMATTED FILE CREATION DATE: JAN 8 98
!      FORMATTED FILE CREATION TIME: 9:55
!
@mergflts-maintroncoso HEADER , FALT, 80, 1
X (EASTING) , 1, 1, 1, 1, 15, 7, 0.1000000E+31, , 15, 7, 0
Y (NORTHING) , 2, 2, 1, 16, 30, 7, 0.1000000E+31, , 15, 7, 0
SEG I.D. , 3,35, 1, 31, 40, 1, 0.1000000E+31, , 10, 1, 0
@
2442342. 5884844. 1.0
2442466. 5884454. 1.0
2442661. 5883656. 1.0
2442802. 5882699. 1.0
2442962. 5881848. 1.0
```

If you do not have the deviated well trajectory information, you can put the well location information for each layer in a separate file with a simple format. The simple format is either:

Well_name X Y
or
X Y well_name

where X and Y are the coordinates of the well location on the surface for the corresponding map. If the well name contains spaces, the well name must be enclosed in single quotes. If the well names contain only numbers, then a WELLNAMEFIRST or XYFIRST keyword must be the first line in the file.

The ZMAP DWEL file contains well trajectory information. When using a DWEL file, the well information is input separate from the maps, as the information applies to all layers. The DWEL file will have a header similar to the following:

```
!
!      ZIMS FILE NAME : TRAJECTORIES 11/02
!      FORMATTED FILE CREATION DATE: JAN 8 98
!      FORMATTED FILE CREATION TIME: 10: 5
!
@TRAJECTORIES 11/02 HEADER , DWEL, 80, 2
DEVIATED WELL NAME , 1,27, 1, 1, 7, 7
measured depth , 2, 3, 1, 8, 21, 8, 0.1000000E+31, , 14, 8, 0
drift angle , 3, 3, 1, 22, 34, 7, 0.1000000E+31, , 13, 7, 0
drift azimuth , 4, 3, 1, 35, 48, 8, 0.1000000E+31, , 14, 8, 0
TVD , 5, 3, 1, 49, 62, 8, 0.1000000E+31, , 14, 8, 0
TVDSS , 6, 3, 1, 63, 76, 8, 0.1000000E+31, , 14, 8, 0
Y (NORTHING) , 7, 2, 2, 1, 13, 7, 0.1000000E+31, , 13, 7, 0
X (EASTING) , 8, 1, 2, 14, 26, 7, 0.1000000E+31, , 13, 7, 0
SEG ID , 9,35, 2, 27, 43, 8, 0.1000000E+31, , 17, 8, 0
@
FDT01 0.0000000E+000.0000000E+000.0000000E+000.0000000E+00 929.00000
5879738. 2439425. 1.0000000
FDT01 84.000000 3.000000 296.00000 84.000000 845.00000
5879739. 2439423. 1.0000000
FDT01 106.00000 2.500000 290.00000 105.90000 823.09998
5879739. 2439422. 1.0000000
```

Supported EarthVision Formats

Builder and Results 3D support Dynamic Graphics, Inc., EarthVision scattered data default export format. The projection must be either “Local Rectangular” or “Universal Transverse Mercator”, and Units must be either “meters” or “feet”. The Format could be “free” or “fixed”.

An example of the mesh file header is as follows:

```
# Type: scattered data
# Version: 4
# Description: Exported from grid uncf.2grd (mike, 04/21/98)
# Format: free
# Field: 1 x
# Field: 2 y
# Field: 3 z
# Field: 4 column
# Field: 5 row
# Projection: Local Rectangular
# Units: meters
# End:
# Information from grid:
# Grid_size: 49 x 49
# Grid_X_range: 30000 to 50000
# Grid_Y_range: 70000 to 90000
# Scattered_data: case.dat
# Z_field: UNCF
30000 70000 -7935.12548828125 1 1
30416.6666666667 70000 -7922.39404296875 2 1
30833.3333333333 70000 -7909.3388671875 3 1
31250 70000 -7896.59033203125 4 1
31666.6666666667 70000 -7884.5322265625 5 1
32083.3333333333 70000 -7873.51806640625 6 1
```

The EarthVision vertical fault file is also supported. It will have a header similar to the following:

```
# Type: vertical faults
# Version: 1
# Description: VERT FAULTS
# Format: fixed
# Field: X 1 10
# Field: Y 11 20
# Field: FLTNUM 21 30
# Projection: Universal Transverse Mercator
# Zone: 11
# Units: meters
# Ellipsoid: Clarke 1866
```

```

# End:
265113. 3970926.      1.
265582. 3970764.      1.
266961. 3970631.      1.
267474. 3970495.      1.
269186. 3970677.      2.
269913. 3970542.      2.
271748. 3970441.      2.
272338. 3970269.      2.
263965. 3964144.      3.

```

A sample well trajectory file is as follows. Note that the required Fields are:

- If Format: free - "wellid" or "wellname", "x", "y", tvdss, "md" or "vdblMDs".
- If Format: fixed - "wellid" or "wellname", "x", "y", "z" or "tvdss (z)\", "md" or "vdblMDs".

The first point of a well trajectory should give the surface location.

```

# Type: scattered data
# Version: 4
# Description: Exported paths data from database:offshore5.dwd
# Format: fixed
# Field: wellid 1 15 non-numeric
# Field: md 16 25
# Field: "tvdss (z)" 26 35
# Field: x 36 45
# Field: y 46 55
# Field: linecol 50 53 non-numeric
# Field: symbol 54 57 non-numeric
# Field: commonid 59 62 non-numeric
# Projection: Transverse Mercator
# Units: meters
# Ellipsoid: Hayford International 1924
# Scale_Factor_at_Central_Meridian: 0.999600
# Central_Meridian: 45 30 15.000000 E
# Latitude_of_Origin: 15 30 10.000000 N
# False_Easting: 100000.000000
# False_Northing: 500000.000000
# End:
5-337-14R      0.00      854.10    59239.20   116551.90
5-337-14R     4070.00    -3216.00    59238.70   116551.30
5-337-14R     4080.00    -3226.00    59238.60   116551.20
5-337-14R     4090.00    -3236.00    59238.60   116551.00
5-337-14R     4100.00    -3246.00    59238.50   116550.90
5-337-14R     4110.00    -3256.00    59238.40   116550.70
5-337-14R     4120.00    -3266.00    59238.30   116550.60
5-337-14R     4130.00    -3276.00    59238.20   116550.50

```

| | | | | |
|-----------|---------|----------|----------|-----------|
| 5-337-14R | 4140.00 | -3286.00 | 59238.10 | 116550.40 |
| 5-337-14R | 4150.00 | -3296.00 | 59238.00 | 116550.20 |
| 5-337-14R | 4160.00 | -3306.00 | 59237.90 | 116550.10 |
| 5-337-14R | 4170.00 | -3316.00 | 59237.80 | 116550.00 |
| 5-337-14R | 4180.00 | -3326.00 | 59237.80 | 116549.90 |
| 5-337-14R | 4190.00 | -3336.00 | 59237.70 | 116549.80 |
| 5-337-14R | 4200.00 | -3346.00 | 59237.60 | 116549.60 |
| 5-337-14R | 4210.00 | -3356.00 | 59237.50 | 116549.50 |
| 5-337-14R | 4220.00 | -3366.00 | 59237.40 | 116549.40 |

If you do not have the deviated well trajectory information, you can put the well location information for each layer in a separate file with a simple format. The simple format is either:

Well_name X Y
or
X Y well_name

where X and Y are the coordinates of the well location on the surface for the corresponding map. If the well name contains spaces, the well name must be enclosed in single quotes. If the well names contain only numbers, then a WELLNAMEFIRST or XYFIRST keyword must be the first line in the file.

Supported CPS-3 Formats

Builder and Results 3D support CPS-3 Grid Data (mesh data) and Contour file format. Newer versions of CPS-3 output comments rather than keywords. These newer files require manual editing before they can be read by Builder and Results 3D, refer to the instructions that follow this section.

A sample Grid Data file is as follows:

```
FSASCI 0 1 "Computed" 0 1000000015047466219876688855040.000000 0
FSATTR 0 0
FSLIMI 1272000.0 1291000.0 230000.00 263000.00 448.54349 3685.0830
FSNROW 166 96
FSXINC 200.00000 200.00000
 0.168304E+04 0.1718736E+04 0.1754464E+04 0.1790438E+04 0.1826841E+04
 0.186319E+04 0.1899145E+04 0.1934789E+04 0.1970341E+04 0.2006256E+04
 0.204292E+04 0.2080046E+04 0.2117406E+04 0.2154524E+04 0.2190654E+04
 0.222484E+04 0.2256072E+04 0.2285586E+04 0.2314498E+04 0.2343419E+04
 0.237284E+04 0.2402762E+04 0.2432462E+04 0.2461459E+04 0.2489767E+04
 0.251781E+04 0.2546487E+04 0.2575592E+04 0.2604711E+04 0.2633801E+04
 0.266300E+04 0.2692280E+04 0.2721548E+04 0.2750791E+04 0.2780021E+04
 0.280925E+04 0.2838482E+04 0.2867689E+04 0.2896782E+04 0.2925824E+04
 0.295488E+04 0.2983842E+04 0.3012370E+04 0.3039971E+04 0.3066538E+04
```

| | |
|---------------|---|
| FSASCI | Used to identify the CPS-3 mesh format |
| FSATTR | Not used |
| FSLIMI | The first 4 numbers are X-Minimum, X-Maximum, Y-Minimum and Y-Maximum respectively. The last two numbers are not used. |
| FSNROW | The first number identifies the number of rows, the second number the number of columns. |
| FSXINC | X and Y increments. However, they are not used in the program because these can be calculated from the limits and number of rows and columns. |

All the numbers after the "→Default" line give the mesh values at the mesh points. These values can represent any grid properties, such as grid tops, porosities, and so on.

CPS-3 fault file is also supported, as shown in the following example:

```
FSASCI 0 1 "Computed" 0 1000000015047466219876688855040.000000 0
FFATTR 3 0
(2E15.7)
->f1
 0.6101402E+04  0.2260635E+04
 0.6037690E+04  0.2552553E+04
 0.5958145E+04  0.2881542E+04
 0.5867917E+04  0.3162905E+04
 0.5905114E+04  0.2886946E+04
 0.5989938E+04  0.2520760E+04
 0.6101402E+04  0.2260635E+04
->f2
 0.6398598E+04  0.3401793E+04
 0.6387917E+04  0.3629874E+04
 0.6387917E+04  0.3667197E+04
```

| | |
|---------------|--|
| FSASCI | Used to identify the CPS-3 mesh format |
| FSATTR | Not used |
| (2E17.7) | Fortran style format statement to indicate there are two numbers in each line with width of 17 and 7 digits after the decimal point. |
| →f2 | "→" indicates the start of a fault line with the name "f2" |

Two numbers in each line indicate the x and y coordinates of each fault point in the fault line.

Builder also supports the generic contour format exported from the CPS-3. The first line is a Fortran style statement to declare the number of values and format for all the lines. Each line defines a contour point and the value at that point. Therefore, only three numbers representing X and Y coordinates and contour value are required in each line. Here is a sample file:

```
(F18.2,1X,F18.2,1X,F10.0)
 425685.59      9578175.00      1430.
 425695.47      9578170.00      1430.
 425700.00      9578168.00      1430.
 425714.34      9578161.00      1430.
 425728.12      9578153.00      1430.
 425733.59      9578150.00      1430.
 425744.28      9578144.00      1440.
 425750.00      9578142.00      1440.
 425761.25      9578136.00      1440.
 425765.25      9578135.00      1440.
 425775.00      9578131.00      1440.
```

The following type of fault file is also supported:

```
3    VARIABLES
NORTH   N 12  3
EAST    N 12  3
STRING   C 12  0
9574720.541 420947.983  FAULT
9574694.011 420979.182  FAULT
9574633.647 421036.825  FAULT
9574412.333 421234.327  FAULT
9574325.651 421296.639  FAULT
9574264.791 421333.198  FAULT
9574090.160 421420.126  FAULT
9578014.717 421798.850  FAULT
9578014.709 421807.073  FAULT
9578015.209 422065.966  FAULT
9577976.720 422163.306  FAULT
```

The first line specifies how many variables in each line. The second line indicates the first field, in the data line, is Y coordinate increasing in the north direction. The third line indicates the second field in the data line is X coordinate increasing in the east direction. The last field in the data line is the fault name.

Newer CPS-3 Files

In order for Builder or Results3D to read newer CPS-3 format files, some manual editing is required. These newer files do not have the required keywords in the file header. They have comments containing some of this information.

FSASCI must be added to the start of the file. The critical information is the second last value which indicates the “null” value used in the file. The value is usually 1E30 but you may need to examine the data values for confirmation.

```
FSASCI 0 1 "Computed" 0 1E30 0  
FSATTR 0 0
```

Next, FSLIMI, FSXINC, and FSNROW must be added. The original file comments contain the equivalent information.

```
!Grid Lattice: Generic Binset  
!VOI Box XMIN: 46550.0 m  
! XMAX: 46760.0 m  
! YMIN: 108650.0 m  
! YMAX: 108910.0 m  
!Lattice XINC: 100.0 m  
! YINC: 100.0 m  
! NCOL: 22  
! NROW: 27  
FSLIMI 46550.0 46760.0 108650.0 108910.0 1.0 1.0  
FSXINC 100.0 100.0  
FSNROW 27 22
```

Finally, add the line “→Default:” just before the start of the data.

```
->Default:
```

Production Analyst Well Trajectory File Format

One of the supported well trajectory file formats is from Production Analyst, a program used to record and analyze well data. Two files from Production Analyst program are required to completely describe the well trajectories: an XY file, and a DEVIATED file. The XY file contains the well names, surface locations and creation date, and DEVIATED file gives the coordinates of the nodes of the trajectories. Both files can be generated from the Production Analyst by choosing PA_DATA from the PA_TOOLS menu. They are both ASCII files. “.xy” is the file extension for XY Files and “.dev” for DEVIATED Files.

If the node coordinates are not directly available, you can enter the inclination and azimuth at points versus measured depth, and Builder will calculate the deviations.

XY File Description

Sample XY File

```
*metric  
*well *type *xcoor *ycoor *kbel *tdepth *inter *region *cdate *gcorr *alias  
OGCI_01:A20 Gas 9.235e+003 7.485000e+003 32.61 3447.59 0.83330 1 770124 1.01500 W-1  
OGCI_02:A20 Gas 9.235e+003 7.435000e+003 32.61 3163.52 0.62500 1 770228 1.01500 W-2  
OGCI_03:A20 Gas 9.285e+003 7.435000e+003 32.61 3286.05 0.62500 1 770307 1.01500 W-3  
OGCI_04:A20 Gas 9.310e+003 7.435000e+003 32.61 3582.92 0.62500 1 770414 1.01500 W-4
```

If all the numbers in the file are in metric units, then the first line must contain *METRIC (or *metric) keyword and nothing else. Keywords are case insensitive. If this keyword is not present, then the units for all the numbers are assumed to be in English units. When metric unit is used, the length is in meters; otherwise, the length is in feet.

The rest of the file defines a table with the first line containing column titles. Each column title is a word preceded by “*”. Although there are many keywords output by Production Analyst - Builder only needs the following keywords to define a surface well:

- *well Well name with optional completion name separated by a “.”. The completion name is not used by Builder. Can consist of any printable ASCII characters and numbers. No space is allowed within the name. However, if the name is enclosed in a pair of single quote, then space(s) are allowed. Well names are case sensitive
- *xcoor X coordinate of the well surface location. Any valid number. Scientific notation is allowed.
- *ycoor Y coordinate of the well surface location. Any valid number. Scientific notation is allowed.
- *kbel Kelly bushing elevation, which is used as Z coordinate of the well surface location. Any valid number. Scientific notation is allowed.
- *cdate Well creation date. Must be in YYMMDD or YYYYMMDD format. For example, 881230 or 19881230.

The keywords and numbers are separated by at least one blank. A well name with different completion names is allowed in the same file. However, the well creation date (*cdate) is taken from the first one encountered.

Deviated File Description

Sample Deviated File

```
*METRIC
*WELL *DEPTH *XDELTA *YDELTA *TVD
OGCI_01 0.00      0.00      0.00      0.00
OGCI_01 30.48     0.00      0.00      30.48
OGCI_01 609.60    -358.14   -67.36    510.54
OGCI_01 1219.20   -716.28   -134.72   993.04
OGCI_01 1828.80   -1074.42  -202.08   1475.84
OGCI_01 2438.40   -1432.56  -269.44   1958.34
OGCI_01 3048.00   -1790.70  -336.80   2419.20
OGCI_01 3462.83   -2035.45  -381.91   2747.47
OGCI_02 0.00      0.00      0.00      0.00
OGCI_02 30.48     0.00      0.00      30.48
OGCI_02 609.60    -275.54   -127.41   528.22
OGCI_02 1219.20   -551.08   -254.81   1052.47
OGCI_02 1828.80   -826.31   -382.22   1576.43
OGCI_02 2438.40   -1101.85  -509.63   2100.38
OGCI_02 3048.00   -1377.39  -637.03   2624.33
OGCI_02 3178.76   -1436.52  -671.78   2736.80
```

If all the numbers in the file are in metric units, then the first line must contain *METRIC keyword and nothing else. Keywords are case insensitive. If this keyword is not present, then the units for all the numbers are assumed to be in English units. When metric unit is used, the length is in meters; otherwise, the length is in feet.

The rest of the file is a table with the first line containing the column titles. Each column title is a word preceded by “*”. Each of the rest of the lines defines a 3D point of the trajectory for a well. The meaning of each keyword is as follows:

| | |
|---------|---|
| *WELL | Well name. Can consist of any printable ASCII characters and numbers. No space is allowed within the name, unless the name is enclosed in a pair of single quotes. The well names are case sensitive. |
| *DEPTH | Numeric, measured depth in wellbore, positive downward and the first depth is zero. |
| *XDELTA | Numeric, distance the wellbore at a specific measured depth deviated in the X direction (east or right is positive) from the well surface location. Any valid number. |
| *YDELTA | Numeric, distance the wellbore at a specific measured depth deviated in the Y direction (north or up is positive) from the well surface location. Any valid number. |
| *TVD | True vertical depth of the wellbore at a specific measured depth. Any valid number |

Angular well trajectory is supported by the following keywords:

| | |
|--------------|--|
| *AZIMUTH | The angle between the wellbore and the y axis (north), in degrees Should be between 0 - 360 degree |
| *INCLINATION | The angle between the wellbore and a vertical line (z axis), in degrees Should be between 0 - 180 degrees |

When *AZIMUTH keyword is present, *INCLINATION and *DEPTH should be present too. *TVD is optional. However, if there is no *TVD data, then the INCLINATION angle of the first point should be zero so that the measure depth and true vertical depth of the first point are equal.

Well Trajectory Table File Format

This is a generic format which is generated from querying most of the database. It contains all information required to describe the well trajectories.

The first line contains the column titles (keywords) - the order of which is not important. The column titles can be optionally preceded by a “*”. Any line that starts with “**” is treated as a comment. The rest of the lines define 3D points of the trajectory for a well whose name appears under UWID or WELL column. Column titles can be specified using upper or lower case and are separated by at least one blank. The meaning of each column title (keyword) is as follows:

| | |
|----------------------|--|
| UWID(or WELL) | Unique well name, any text string without blanks. If blanks are necessary, then the name must be enclosed in a pair of single quote, i.e., 'Well name 1'. Also, well names are case sensitive. |
| X-SURFACE | Well surface location, x coordinate (optional). |
| Y-SURFACE | Well surface location, y coordinate (optional). |
| ELEVATION | Well surface location, z coordinate, positive upward. |
| X-COORD | Well trajectory node, x coordinate. |
| Y-COORD | Well trajectory node, y coordinate. |
| TVDS | True vertical depth subsea. z coordinate, negative downward from the sea level. |
| TVD | True vertical depth, z coordinate positive downward from the surface location. |
| DEPTH-MD | Measured depth along the wellbore (optional). If not present, then measured depth will be calculated using coordinates information. |

A sample file is shown below:

| UWID | TVD | X-COORD | Y-COORD | X-SURFACE | Y-SURFACE | ELEVATION |
|-------------|------------|----------------|----------------|------------------|------------------|------------------|
| 316-23R | 0.0 | 57848.0 | 12128.0 | 57848.0 | 12128.0 | 1128. |
| 316-23R | 600.5 | 57848.0 | 12128.0 | 57848.0 | 12128.0 | 1128. |
| 316-23R | 650.3 | 57848.0 | 12127.6 | 57848.0 | 12128.0 | 1128. |
| 316-23R | 660.5 | 57848.2 | 12127.2 | 57848.0 | 12128.0 | 1128. |
| 316-23R | 670.4 | 57848.4 | 12126.9 | 57848.0 | 12128.0 | 1128. |

All the lines that belong to the same well must be grouped together and sorted with decreasing TVDSS or increasing TVD order. The X-SURFACE, Y-SURFACE and ELEVATION for the lines that belong to the same well should have the same values. The X-SURFACE, and Y-SURFACE columns are optional. If they are absent, then the first point of the well trajectory is taken as the surface location. ELEVATION column is ignored if TVDSS is present. It is used in conjunction with TVD to compute z coordinate of the trajectory node. When TVD column is present and ELEVATION is not present, then it is assumed to be zero.

The following is a sample file that does not contain the surface location:

| WELL | SKIP | SKIP | DEPTH-MD | TVDSS | Y-COORD | X-COORD | SKIP |
|-------------|-------------|-------------|-----------------|--------------|----------------|----------------|-------------|
| FDT01 | 0.00E+00 | 0.000E+00 | 0.000E+00 | 929.00000 | 5879738. | 2439425. | 1.0000000 |
| FDT01 | 3.000000 | 296.00000 | 84.000000 | 845.00000 | 5879739. | 2439423. | 1.0000000 |
| FDT01 | 2.500000 | 290.00000 | 105.90000 | 823.09998 | 5879739. | 2439422. | 1.0000000 |
| FDT01 | 2.500000 | 296.00000 | 116.90000 | 812.09998 | 5879740. | 2439422. | 1.0000000 |
| FDT01 | 2.750000 | 293.00000 | 134.89999 | 794.09998 | 5879740. | 2439421. | 1.0000000 |
| FDT01 | 2.750000 | 290.00000 | 139.89999 | 789.09998 | 5879740. | 2439421. | 1.0000000 |
| FDT01 | 2.000000 | 288.00000 | 156.89999 | 772.09998 | 5879740. | 2439420. | 1.0000000 |
| FDT01 | 2.750000 | 298.00000 | 166.89999 | 762.09998 | 5879740. | 2439420. | 1.0000000 |

A SKIP keyword in the first line indicates the program should ignore the corresponding column. The units for the column names are specified through the **Trajectory File Open** dialog box. The X_COORD and Y_COORD can be in different units from the rest of the columns; for example, UTM (meters) for X and Y and field unit (feet) for the measured depth and elevation.

Well Perforation File Format

The well perforation file is used to specify which section of the well trajectory is perforated for production or injection purposes. This file does not exist by itself. It has to be associated with some well trajectory file in PA, table format or other formats because perforations are specified using measured depth along the wellbore. Builder supports two different formats.

Table Format for Well Trajectory Perforation Intervals

In this format, the well name, date, perforation interval, and activity are given in a table, with a small amount of header information at the start of the file.

```
INUNIT SI
DATE_FORMAT 'YYYY/MM/DD'
WELL DATE MD_START MD_END STATUS MD_CORR SKIP
OP1A 2000/01/01 5500 5600 PERF 0.0 0.
'OP 12C' 2000/10/15 6000 6200 CEM_SQZ 20.5 1.
```

The perforation date may be delimited by ‘/’ or ‘-’. Well names and dates can be in any order.

The header keyword INUNIT must be followed by either SI or FIELD. SI units require measured depths in meters. Field units require measured depths in feet, and well diameters in inches.

The header keyword DATE_FORMAT must be followed by a string specifying the date format. Use Y to specify a year digit, M to specify a month digit and D to specify a day digit. Four digit years are required.

Each column in the table is identified with a keyword in the header line, immediately above the table in the file. The keywords that identify columns are:

| | |
|----------|---|
| WELL | well name, must be in single quotes if it contains spaces |
| DATE | date of the perforation, in the format given be DATE_FORMAT |
| SKIP | ignore this column |
| DIAM | well diameter |
| STATUS | specifies the type of perforation event. Allowed options are: PERF or ACID – causes a perforation segment to be created CEM_SQZ – causes a perforation segment to be closed off |
| MD_START | measured depth to start of the perforation interval |
| MD_END | measured depth to the end of the perforation interval |
| MD_CORR | depth correction, added to MD_START and MD_END |

Old Perforation Format

This file format is the same as the perforation file generated from CMG's Builder program. The following is a sample file:

```
DATE_FORMAT 'M D Y'
'WELL0' 1193 5.5 100
    2300    2302    12/18/1983  PERF   18  ** comment
    2305    2306    12/18/1983  PERF   18
    2307    2308    12/18/1983  PERF   18
    2309.5  2310.3  12/18/1983  PERF   18
'WELL1' 1193 5.5 -500
    2300.0  2305.5  12/18/1984  PERF   18
    2310.0  2312.6  12/18/1984  PERF   18
```

Builder ignores all the keywords except DATE_FORMAT. If present, it has to be followed by one of the following strings, which specify the order of year, month and day in the date string:

'M D Y' means month, day, year. (example: 1/31/2001, Jan 1 2001)

'M Y D' means month, year, day. (example: 1/2001/1)

'D M Y' means day, month, year. (example: 31/01/2001, 31-JAN-2001, 31.01.2001)

'D Y M' means day, year, month. (example: 31/2001/01)

'Y M D' means year, month, day (example: 2001/01/31, 2001-January-31)

'Y D M' means year, day, month. (example: 2001/31/01)

When the month is specified, it can be a 1-2 digit number, the first 3 letters of the month, or the complete month name. The month name can be in English or Spanish. The delimiter between month, day and year can be slash('/'), dot('.'), dash('-') or space. A four-digit year is preferred. If a two digit year is encountered, 1900 is automatically added to the year; for example, 96 will become 1996.

If there is no DATE_FORMAT keyword, the default is 'M D Y' or integer format.

Integer date format is an 8 digit integer in the form 'YYYYMMDD' (for example, 20010131). Note that there must be no delimiter between year, month and day in this format.

The well name starts the definition of the perforations. It has to start a new line and be included in a pair of single quotes. Three numbers follow the well name, which are KB elevation, well hole size and off depth correction. "off depth correction" is a number added to KB elevation to correct the trajectory depth (z coordinate) such that perforation is in the desired grid block. Well hole size and KB elevation are not used by the Builder perforation file reading routine. KB elevation should have been specified in the file that defines the well trajectories, such as Production Analyst format, CMG Table Format and Zmap format.

Every line, after the well name line, specifies a perforation segment. The first number is the measured depth of the start of the perforation segment and second number the end of the segment. The third word (or 4th and 5th word if date is space separated) is the date the perforation job is done. The fourth (or 6th) word can be PERF, CEM_SQZ, or ACID. The PERF or ACID keyword will cause a perforation segment to be created. Builder ignores the rest of the line. Comments can be inserted anywhere in the file as long as they are preceded by "##".

The unit for the measured depths of the start and end of the perforation segment in this file is assumed to be the same as the unit of the measured depth or z coordinate in the well trajectory file(s) that the file is associated with.

Well Trajectory and Logs in RMS Format

This format can contain both well trajectory and multiple well logs for one well in one file. The first two lines are ignored by the Builder. The third line contains the well name the rest of the line is ignored. The fourth line indicates the number of well logs this file contains, followed by the same number of lines containing the well log names. Everything after the well log names is ignored. Finally, there is a table of well trajectory nodes and logs. The first three columns are X, Y and Z (Elevation) coordinates of the nodes, the rest of the columns are the well logs.

A sample file is shown below:

```
1.0
Oil
Well-14 0.0 0.0
2
Porosity % linear
Permeability % linear
455550.000000 6784224.500000 -1818.000000 0.1 1
455549.906250 6784224.500000 -1818.199997 0.3 2
455549.843750 6784224.500000 -1818.300003 0.4 2
455549.781250 6784225.000000 -1818.399994 0.3 2
455548.125000 6784228.000000 -1821.500000 0.3 4
```

Well Trajectory in GOCAD Format

The GOCAD is a very flexible file format. It is object oriented and can supply all the information needed by any graphics program. A sample file containing well trajectory (WellPath) data is as follows:

```
GOCAD Well 0.01
HEADER{
name:wt-12
color:red
}
WREF 1440    1715    0.0

DPLN 0
TVSS_PATH    0.0      0.0    0.0      0.0
TVSS_PATH   1920.0   -1920    0.0      0.0
TVSS_PATH   1925.0   -1925    0.00     0.00
TVSS_PATH   1926.0   -1926    0.00     0.00
TVSS_PATH   1927.0   -1927    0.00     0.00
TVSS_PATH   1928.0   -1928    0.00     0.00
TVSS_PATH   1930.0   -1930    0.00     0.00
```

```

MRKR marker 1 2 meters
NORM 0 0 1
ZONE zone 4 7 1
END
GOCAD Well 0.01
HEADER{
name:wt-14
}
#WREF 1440 1715 0.0
DPLN 0
#      *X      *Y      *Z
VRTX 1440 1715 0
VRTX 1440 1715 -1920
VRTX 1440 1715 -1925
VRTX 1440 1715 -1926
VRTX 1440 1715 -1927
VRTX 1440 1715 -1928
VRTX 1440 1715 -1930

```

```

MRKR marker 1 2 meters
NORM 0 0 1
ZONE zone 4 7 1
END

```

The WellPath is composed of a header and a coordinate section. The header defines the reference point and the datum plane. The coordinate section defines the well path by giving either a series of PATH definition or VRTX definition.

```

WREF X Y Z
DPLN datum_plane
PATH Zm Z dX dY
VRTX X Y Z

```

WREF gives the X and Y coordinates of the reference point for the well path - usually the surface location of the well. The Z value of the WREF is used as the depth for drawing the derrick and the 0 position of the Well Name. It is NOT the KB or RT of the well.

PATH keyword can be replaced by TVSS_PATH (depth in sub-sea) or TVD_PATH (depth in true vertical depth).

TVSS_PATH describes a point of the WellPath, giving a measured depth, Zm, a real depth (with KB or RT elevation already taken out; the "real world" Z coordinate), Z and x, y deviations, dX and dY relative to the X and Y of the WREF point. The Z and dX, dY of the PATH is used to draw the well path, while the Zm is not used by the Builder.

Well path can be given by TVD_PATH keyword, which is the true vertical depth. To find the sub-sea depth, GOCAD subtracts the WREF Z value from the TVD_PATH Z.

VRTX describes a point of the WellPath given in absolute (real world) coordinates.

Builder only deals with the following keywords:

```
GOCAD Well
WREF
VRTX
TVSS_PATH
TVD_PATH
HEADER
END
```

Within the HEADER block, only “name” is used.

The above is a brief description of GOCAD format. For complete references, see the *GOCAD Developer’s Guide*, prepared by Jean-Claude Dulac.

Well Log Formats

LAS Well Log Format

Canadian Well Logging Society’s LAS (Log ASCII Standard) Well Log format version 3.0 is supported by the Builder. This format can have “wrap” and “no wrap” option, and both options are supported. Contact the following address for detailed format description:

CWLS Committee
Suite 229, 640-5 Avenue S.W.
Calgary, Alberta
CANADA T2P 0M6

Builder only read the following information sections from the file:

```
~VERSION INFORMATION
~Well Information
~Curve Information
~ASCII | CURVE
~Inclinometry_Definition
~Inclinometry | Inclinometry_Definition
~Perforations_Definition
~Perforations | Perforations_Definition
~Tops_Definition
~Tops | Tops_Definition
```

The well surface location is given by UTM in the Well Information section. Builder does NOT read the location given in longitude and latitude format. The well log data are given by Curve Information and ASCII | CURVE sections. The Inclinometry_Definition and Inclinometry sections describe the 3D well trajectory. The Perforations_Definition and Perforations sections specify the perforation interval along the trajectory. The information in these two sections, if present, will be read by the Builder.

The default file extension for this format is ".las". This type of file is self-explanatory. So the sample file below should explain most of the format:

~VERSION INFORMATION

VERS. 3.0 : CWLS LOG ASCII STANDARD -VERSION 3.0
WRAP. NO : ONE LINE PER DEPTH STEP
DLM . COMMA : DELIMITING CHARACTER BETWEEN DATA
COLUMNS
Acceptable delimiting characters: SPACE (default), TAB, OR COMMA.

~Well Information

| #MNEM.UNIT | DATA | DESCRIPTION |
|------------|--------------------------|-------------------------|
| STRT .M | 1670.0000 | : First Index Value |
| STOP .M | 713.2500 | : Last Index Value |
| STEP .M | -0.1250 | : STEP |
| NULL . | -999.25 | : NULL VALUE |
| COMP . | ANY OIL COMPANY INC. | : COMPANY |
| WELL . | ANY ET AL 12-34-12-34 | : WELL |
| FLD . | WILDCAT | : FIELD |
| LOC . | 12-34-12-34W5M | : LOCATION |
| PROV . | ALBERTA | : PROVINCE |
| SRVC . | ANY LOGGING COMPANY INC. | : SERVICE COMPANY |
| DATE . | 13/12/1986 | : LOG DATE {DD/MM/YYYY} |
| UWI . | 100123401234W500 | : UNIQUE WELL ID |
| API . | 12345678 | : API NUMBER |
| LAT .DEG | 34.56789 | : Latitude {DEG} |
| LONG.DEG | -102.34567 | : Longitude {DEG} |
| UTM . | 1234587 3489875 | : UTM LOCATION |

~CURVE INFORMATION

| #MNEM.UNIT | API CODES | CURVE DESCRIPTION |
|------------|--------------|-------------------------|
| DEPT .M | | : 1 DEPTH |
| DT .US/M | 60 520 32 00 | : 2 SONIC TRANSIT TIME |
| RHOB .K/M3 | 45 350 01 00 | : 3 BULK DENSITY |
| NPHI .V/V | 42 890 00 00 | : 4 NEUTRON POROSITY |
| SFLU .OHMM | 07 220 04 00 | : 5 RXORESISTIVITY |
| SFLA .OHMM | 07 222 01 00 | : 6 SHALLOW RESISTIVITY |
| ILM .OHMM | 07 120 44 00 | : 7 MEDIUM RESISTIVITY |
| ILD .OHMM | 07 120 46 00 | : 8 DEEP RESISTIVITY |

~PARAMETER INFORMATION

| #MNEM.UNIT | VALUE | DESCRIPTION |
|------------|-----------|----------------------------|
| MUD . | GEL CHEM | : MUD TYPE |
| BHT .DEGC | 35.5000 | : BOTTOMHOLE TEMPERATURE |
| BS .MM | 200.0000 | : BIT SIZE |
| FD .K/M3 | 1000.0000 | : FLUID DENSITY |
| MATR . | SAND | : NEUTRON MATRIX |
| MDEN . | 2710.0000 | : LOGGING MATRIX DENSITY |
| RMF .OHMM | 0.2160 | : MUD FILTRATE RESISTIVITY |
| DFD .K/M3 | 1525.0000 | : DRILL FLUID DENSITY |

```

~OTHER
    Note: The logging tools became stuck at 625 metres
          Causing the data between 625 metres and 615 metres to be invalid.
~ASCII | CURVE
    1670.000 123.450 2550.000  0.450 123.450 123.450 110.200 105.600
    1669.875 123.450 2550.000  0.450 123.450 123.450 110.200 105.600
    1669.750 123.450 2550.000  0.450 123.450 123.450 110.200 105.600
~Inclinometry_Definition
MD. M                                : Measured Depth      {F}
TVD. M                               : True Vertical Depth {F}
AZIM. DEG                            : Borehole Azimuth   {F}
DEVI. DEG                            : Borehole Deviation {F}
~Inclinometry | Inclinometry_Definition
0.00,0.00,290.00,0.00
100.00,100.00,234.00,0.00
200.00,198.34,284.86,1.43
300.00,295.44,234.21,2.04
400.00,390.71,224.04,3.93
500.00,482.85,224.64,5.88
600.00,571.90,204.39,7.41
~Perforations_Definition
PERFT.M                                : Perforation Top Depth {F}
PERFB.M                                : Perforation Bottom Depth {F}
PERFD.SHOTS/M                          : Shots per meter     {F}
PERFT.                                 : Charge Type        {S}
~Perforations | Perforations_Definition
545.50,550.60,12,BIG HOLE
551.20,554.90,12,BIG HOLE
575.00,595.00,12,BIG HOLE

```

Single Well Log File

This file format stores the well logs for one well per file. The first line is the well name and an optional date and/or null value. The next line may optionally contain log units which must be input using the keyword LOGUNITS. The next line contains the log names (column headings). One of the log names must be DEPTH. The other log names can by any name desired. The rest of the lines are the log data. A sample file is shown below:

| Well-14 | | | | | | | | | |
|---------|-----|--------|--------|--------|-----|-------|--------|--------|--------|
| DEPTH | ASN | LLS | LLD | ILD | Rt | SP1 | SP2 | GR1 | GR2 |
| 2510.0 | 7.8 | -999.0 | -999.0 | -999.0 | 8.7 | -19.9 | -999.0 | -999.0 | -999.0 |
| 2511.0 | 8.6 | -999.0 | -999.0 | -999.0 | 9.8 | -20.2 | -999.0 | -999.0 | -999.0 |
| 2512.0 | 8.7 | -999.0 | -999.0 | -999.0 | 9.9 | -20.3 | -999.0 | -999.0 | -999.0 |
| 2513.0 | 8.6 | -999.0 | -999.0 | -999.0 | 9.7 | -19.9 | -999.0 | -999.0 | -999.0 |
| 2514.0 | 8.1 | -999.0 | -999.0 | -999.0 | 9.2 | -19.4 | -999.0 | -999.0 | -999.0 |
| 2515.0 | 7.8 | -999.0 | -999.0 | -999.0 | 8.7 | -19.3 | -999.0 | -999.0 | -999.0 |

You can also enter a depth range using the log names DEPTH and DEPTH2 (see example in the Multiple Well Log File below). When this type of file format is selected in the **Well Trajectory** dialog box, you can select multiple files from the **File Open** dialog box.

Multiple Well Log File

The file format permits multiple wells in a single file. It is similar to the Single Well Log File format except the first line is the log names (column headings). The next line may optionally contain log units which must be input using the keyword LOGUNITS. The next line contains the well name and an optional date and/or null value, followed by well log data. Another well name and log data can exist in the same file. A sample file is shown below:

| DEPTH | ASN | LLS | LLD | ILD | Rt | SP1 | SP2 | GR1 | GR2 |
|---------|----------|--------|--------|--------|-----|-------|--------|--------|--------|
| "wt-14" | 19880725 | | | | | | | | |
| 1936.0 | 7.8 | -999.0 | -999.0 | -999.0 | 8.7 | -19.9 | -999.0 | -999.0 | -999.0 |
| 1936.1 | 7.9 | -999.0 | -999.0 | -999.0 | 8.7 | -19.9 | -999.0 | -999.0 | -999.0 |
| 1936.4 | 8.0 | -999.0 | -999.0 | -999.0 | 8.7 | -19.9 | -999.0 | -999.0 | -999.0 |
| 1937.0 | 8.6 | -999.0 | -999.0 | -999.0 | 9.8 | -20.2 | -999.0 | -999.0 | -999.0 |
| 1938.0 | 8.7 | -999.0 | -999.0 | -999.0 | 9.9 | -20.3 | -999.0 | -999.0 | -999.0 |
| 1939.0 | 8.6 | -999.0 | -999.0 | -999.0 | 9.7 | -19.9 | -999.0 | -999.0 | -999.0 |
| 1940.0 | 8.1 | -999.0 | -999.0 | -999.0 | 9.2 | -19.4 | -999.0 | -999.0 | -999.0 |
| 1941.0 | 8.1 | -999.0 | -999.0 | -999.0 | 9.2 | -19.4 | -999.0 | -999.0 | -999.0 |
| 1942.0 | 8.1 | -999.0 | -999.0 | -999.0 | 9.2 | -19.4 | -999.0 | -999.0 | -999.0 |
| "wt-12" | | | | | | | | | |
| 1923.0 | 7.8 | -999.0 | -999.0 | -999.0 | 8.7 | -19.9 | -999.0 | -999.0 | -999.0 |
| 1924.0 | 8.6 | -999.0 | -999.0 | -999.0 | 9.8 | -20.2 | -999.0 | -999.0 | -999.0 |
| 1925.0 | 8.7 | -999.0 | -999.0 | -999.0 | 9.9 | -20.3 | -999.0 | -999.0 | -999.0 |
| 1925.2 | 8.8 | -999.0 | -999.0 | -999.0 | 9.9 | -20.3 | -999.0 | -999.0 | -999.0 |
| 1925.5 | 8.9 | -999.0 | -999.0 | -999.0 | 9.9 | -20.3 | -999.0 | -999.0 | -999.0 |
| 1926.0 | 8.6 | -999.0 | -999.0 | -999.0 | 9.7 | -19.9 | -999.0 | -999.0 | -999.0 |
| 1927.0 | 8.1 | -999.0 | -999.0 | -999.0 | 9.2 | -19.4 | -999.0 | -999.0 | -999.0 |
| 1928.0 | 8.1 | -999.0 | -999.0 | -999.0 | 9.2 | -19.4 | -999.0 | -999.0 | -999.0 |
| 1929.0 | 8.1 | -999.0 | -999.0 | -999.0 | 9.2 | -19.4 | -999.0 | -999.0 | -999.0 |

You can also enter a depth range using the log names DEPTH and DEPTH2. If both of these log names are found, additional log points will be automatically added so that the log values remain constant for the depth interval specified. If depth intervals are missing between points, additional log points will be added with zero values. In the example below, since depth ranges are missing between the first and second points (16296 – 16338), zero values will be added for this depth range.

```
DEPTH      DEPTH2 'Water Rate RC' 'Oil Rate RC' 'Gas Rate RC' Production
LOGUNITS 'ft' 'ft'   'bbl/day' 'bbl/day' 'bbl/day'      ''
"WELL-112" 19880101
16276.0 16296.0      0       8.91      0       .0278
16338.0 16357.9      0      213.59      0       .6669
16363.9 16384.0      0      97.75      0       .3052
```

```

"WELL-112" 19890101
16276.0 16296.0      0     0.63      0     .0021
16338.0 16357.9      0    227.94      0     .7694
16363.9 16384.0      0     67.69      0     .2285
"WELL-94" 19880101
16240.0 16246.4      0   3562.2      0     .4922
16247.4 16253.9      0   948.27      0     .1310
16255.2 16260.1      0   1648.13      0     .2277
16263.5 16283.4      0  1078.99      0     .1491
"WELL-94" 19880201
16240.0 16246.4      0   2305.11      0     .5428
16252.5 16278.7      0   1937.84      0     .4563
16281.5 16304.0      0     3.91      0     .009
"WELL-94" 19880301
16240.0 16246.4      0   3984.2      0     .5601
16252.5 16278.7      0   3087.82      0     .4341

```

The well name must be enclosed in single or double quotes in this format. The default file extension is ".wlg".

Formation Top File Format

If you have loaded LAS format (version 3.0) files to the application, they might have already top data. For other file formats, you can load formation top data using CMG's **Table Format**.

The formation top file is used to specify formation or geological unit tops associated with the well trajectories by name. Builder supports the following file format:

Table Format for Formation Tops

In this format, the well name, geological unit name, formation top measured depth, formation bottom measured depth, are given in a table, with a small amount of header information at the start of the file.

| INUNIT | SI | | | |
|----------|---------|---------|-----------|--|
| WELL | GEOUNIT | MD_TOP | MD_BOTTOM | |
| OP1A | Basal | 5600.50 | 5690 | |
| 'OP 12C' | Quartz | 5690 | 5820 | |

Well names and geounit names can be in any order.

The header keyword INUNIT must be followed by either SI or FIELD. SI units require measured depths in meters. Field units require measured depths in feet. If INUNIT keyword is absent in the file, it is assumed that the measured depth is given in the dataset working units.

Each column in the table is identified with a keyword in the header line, immediately above the table in the file. The keywords that identify columns are:

| | |
|---------|---|
| WELL | well name, must be in single quotes if it contains spaces. The name must match the name as given in the trajectory file |
| GEOUNIT | geological unit or formation name, must be in single quotes if it contains spaces |
| MD_TOP | measured depth to top of the formation |
| MD_END | measured depth to the bottom of the formation |

Note: Value -99999 has to be used for missing values of MD_TOP and MD_BOTTOM columns.

Importing from a PPDM database

Overview

Builder supports the import of well data from a PPDM compatible database. The following types of well data can be imported:

- General well data (well name, UWI, surface and bottomhole location, and so on)
- Directional surveys
- Perforated and completion intervals
- Formation tops
- Well logs
- Well core analyses (for example, permeability and porosity)
- Monthly production data

In addition to importing the well data, Builder can optionally create a simple orthogonal corner-point grid and create grid properties (tops and thicknesses) from selected formation tops.

Successive dialog boxes of the **Database Import** wizard guide the import process, from selecting wells to be imported, through selecting specific categories of well data to be imported, to matching production data to wells.

You can import data into a new Builder file, or you can add data to an existing Builder file. If imported data is being added to an existing file, then an additional data merge step will be required (see [Merging Data Sets](#) for further information).

Prerequisites

For Builder to import well data from a database, the following must be satisfied:

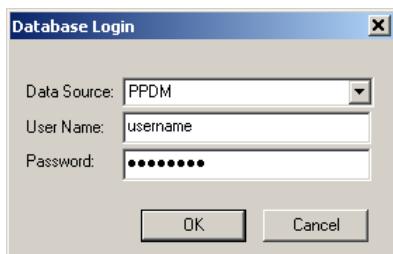
1. Data must be available in a PPDM-compliant database. Builder database import requires a PPDM version 3.7 (or later) database.

2. The data source must be configured for ODBC. Builder uses Open Database Connectivity (ODBC) to connect to and retrieve data from the PPDM database. Note that the configuration of an ODBC data source may require installation of database platform-specific software onto each computer that will access the PPDM database.

Contact local network and database administrators for assistance with creating an ODBC data source or to connect to an available PPDM database. The minimum information required is usually a valid database user id and password, and the name of the ODBC data source.

Launching Database Import

The database import is accessible through **File | Import from Database**. Alternatively, select **Reservoir | Build Static Model with Task Manager** to open the **Geological Task Manager** dialog box, select **Database**, and then click the **Import** button. Either will start the database import. The first step in the import is establishing a connection with the PPDM database. This requires the entry of a valid database user ID and password, as well as the selection of the appropriate ODBC connection. This information is entered in the **Database Login** dialog box, which will be displayed immediately after the database import menu item has been selected:



The **Database Login** dialog box will allow up to three attempts to connect to the database. After each unsuccessful attempt, the following failure message will be displayed:



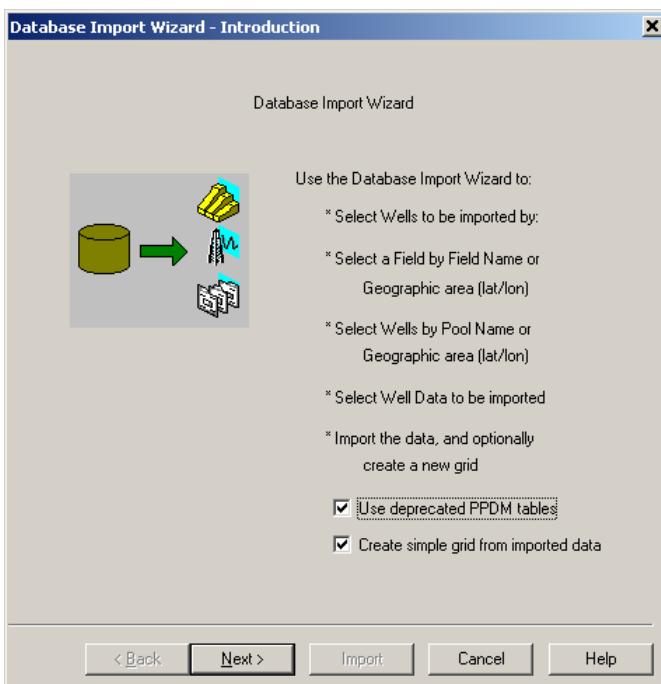
After three unsuccessful attempts, the login process will be terminated, and will need to be restarted.

Possible reasons for an unsuccessful attempt to connect to the database include:

- Incorrect database user ID or password has been entered.
- Incorrect ODBC data source has been selected.
- Database access permission has not been granted.
- Database is not available at the present time.

Contact the local network/database administrator if connection problems persist.

Once a database connection has been established, the **Database Import Wizard - Introduction** dialog box is displayed:



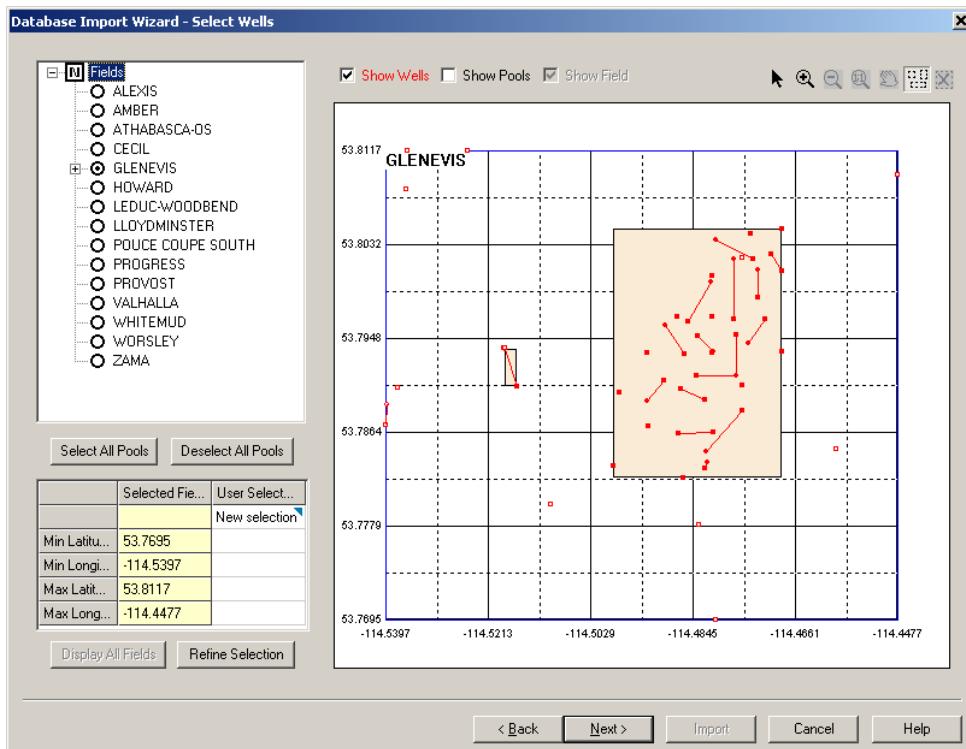
This first dialog box of the **Database Import Wizard** displays general information about the import process, and allows some import options to be changed. The import options that can be changed at this point are:

1. **Use deprecated PPDM tables:** PPDM has evolved. Support for new types of data has been added, and support for existing data has changed. In this case, the option to use deprecated tables refers to the PPDM support for well production data.

Prior to PPDM version 3.7, well production data was stored in table PDEN_VOL_BY_MONTH. In PPDM version 3.7 and later, PDEN_VOL_BY_MONTH is deprecated in favor of PDEN_VOL_SUMMARY. Consult the local database administrator to determine which PPDM table contains production data, and set this option accordingly.

2. **Create simple grid from imported data:** The **Database Import** wizard can create a simple orthogonal corner point grid after a successful import; however, if data is being imported into an existing file (with existing data), or if more advanced grid creation is anticipated, then this option should not be selected.

Click **Next**. The **Data Import Wizard - Select Wells** dialog box is displayed, with fields stored in the PPDM database displayed in the tree view:



Before well selection, the field must be selected on the tree. After selecting the field, the well name, UWI and well location of all wells belong to the field will be imported from the database. The names of all pools in the field will be added to the tree under the field node, and all wells will be drawn on the view. The field limitations will be shown in middle column of the grid in the lower left. You can switch to any other field by selecting it in the tree view. If more than one field has been clicked, the **Display All Fields** button will be enabled. If you click this button, fields with well data will be drawn on the view. In this situation, a field can be selected by clicking a field area on the view or by selecting the field on the tree view. To help with field selection, you can zoom into the view.

To do this, click the **Zoom In** button, then use the mouse to select the area that you want to zoom in on. In the following example, four of 15 fields are selected. The second image shows the four fields after zooming in. You can zoom in until only one field is in the view.

Note: The field name will be shown when the field area is large enough to write the field name.

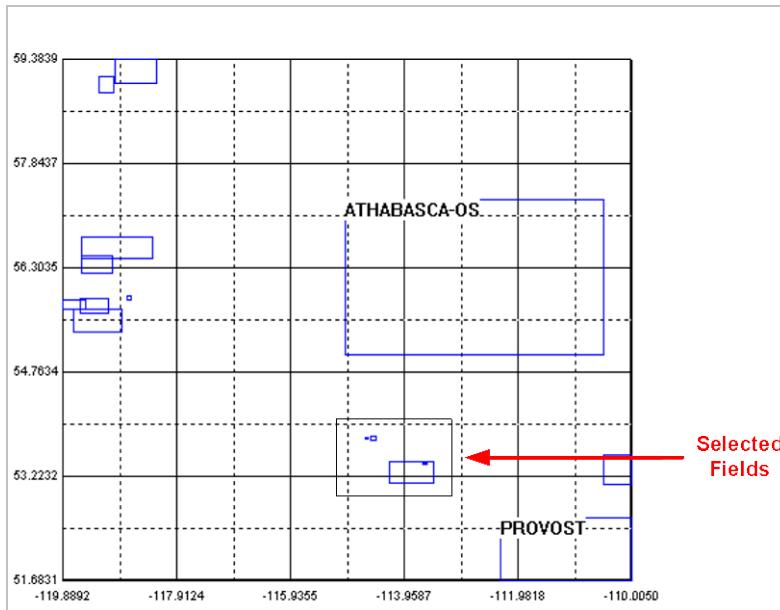
To zoom out, click the **Zoom Out**  button (this button will not be enabled unless you have zoomed in before). The view will be zoomed out. This button can be clicked multiple times until the view becomes its original size. Alternately, you can click the **Display Original View**  button to return the view to its original size.

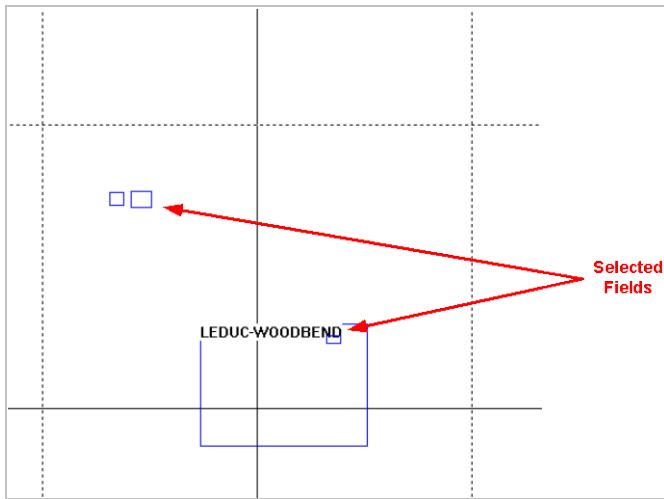
To pan a picture, the view must be zoomed in. Click the **Pan**  button. The view can be panned by holding the left mouse button down and moving the cursor.

To select a field for view, the single select mode must be set. To do this, click the **Select a Field**  button then click inside of a field to be selected. The field will be displayed. Wells in the field will be displayed as

To select wells in the field, click the **Select Wells**  button, and then use the mouse to define areas that contain the wells you want to select. You can define multiple areas. Click the desired areas. You can use the SHIFT or CTRL keys to select multiple areas.

To undo an action, click the **Undo**  button if it is enabled. If a field is selected by clicking the field area on the graph view or by selecting it from the tree, then the view will show all wells in the selected field. If you click the **Undo**  button, the view will be returned to the previous view. While wells are being selected (that is, the **Select Wells**  button is active and one or more selected area is highlighted), if you click the **Undo**  button, the selection of wells in these highlighted areas will be cancelled.



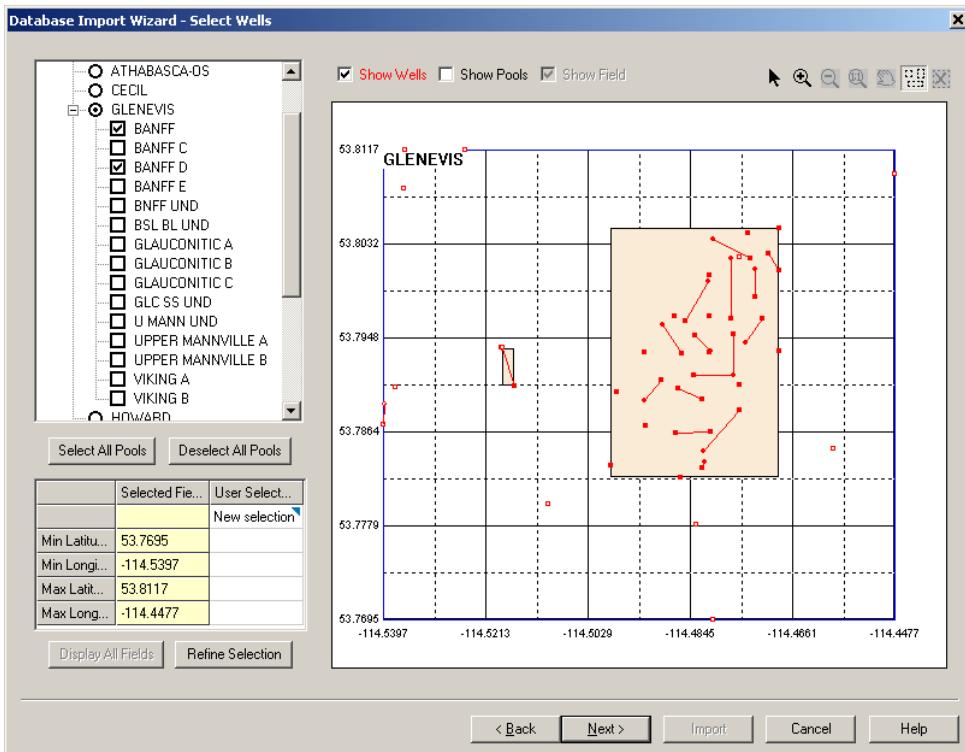


Wells to be imported can be selected by three means:

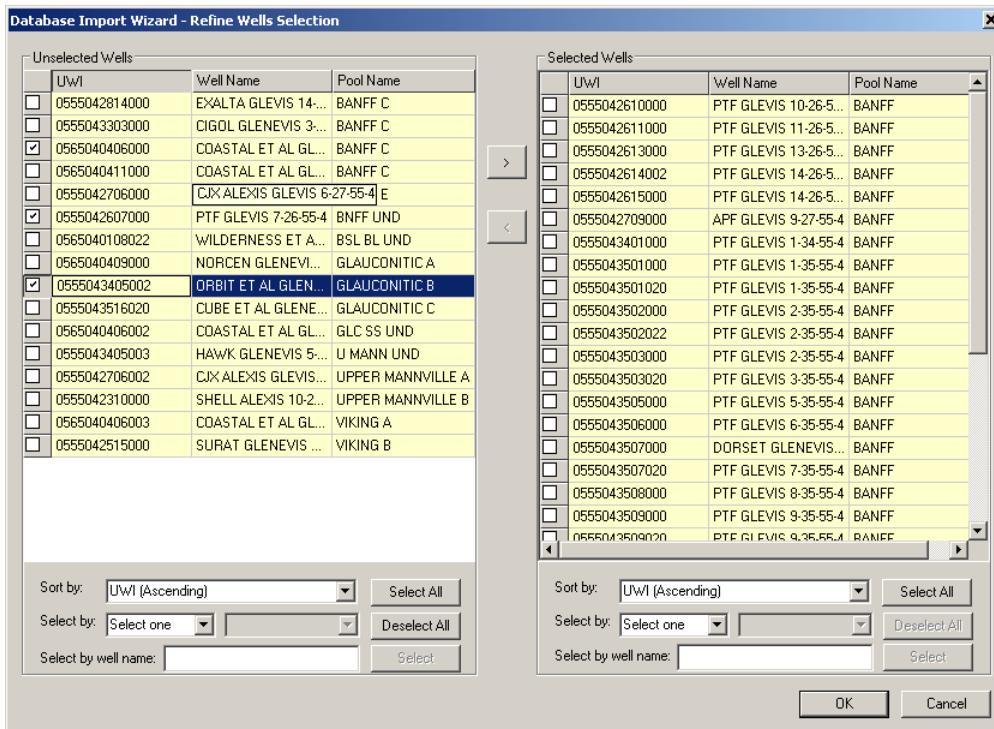
- By specifying the latitudes and longitudes of the area of interest (AOI)
- By specifying a producing pool
- By specifying well names

To select wells by the area of interest, select **New selection** from the list on first row and right column of the grid in the lower left. Enter the lower left and upper right latitude and longitude in the grid. Multiple selections can be made by repeating the above steps. Note that all entries are in decimal degrees, and westerly longitudes and southerly latitudes must be negative values. You can do same thing by using the mouse to select a rectangular area in the view. The icons of the selected wells will change from an empty box □ to a filled box ■.

To select wells by specifying a producing pool, click the check box beside the pool name. All wells that belong to the pool will be selected after the pool box is checked. Multiple pools can be selected by checking their boxes, as shown in the following example:



To select wells by specifying well names, click the **Refine Selection** button. The **Data Import Wizard - Refine Wells Selection** dialog box is displayed:



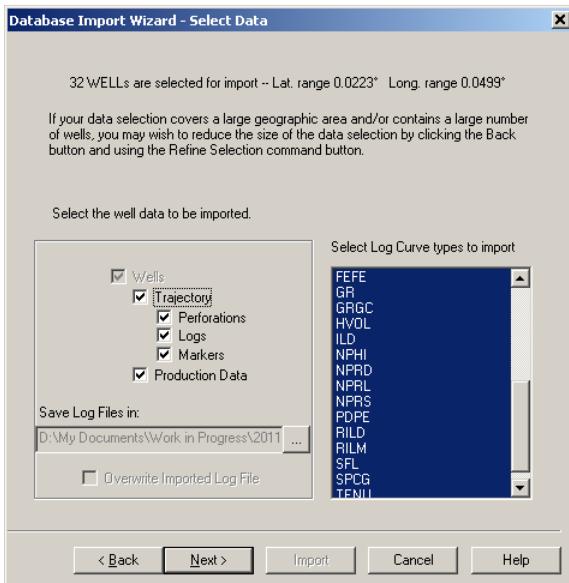
Unselected wells are listed in the left grid and selected wells are listed in the right grid. Wells in the list can be sorted by **UWI**, **Well Name**, **Pool Name** or checked/unchecked status. To add unselected wells to the selected wells list, check wells to be added in the unselected wells list, and then click the **>** button. To remove wells from selected wells list, select the wells to be removed from the selected wells list, and then click the **<** button.

There are four ways to select wells in the **Data Import Wizard - Refine Wells Selection** dialog box:

- Click the associated check box.
- Click **Select All** button to select all check boxes.
- Select by pool names or user selections (in the boxes in the lower left of the dialog box).
- In the **Select by well name** box, you can use a wildcard string (using “*”, for example) and then click the **Select** button.

Click **OK** to save your changes and return to the **Data Import Wizard - Select Wells** dialog box. If you click **Cancel**, you will be returned to the **Data Import Wizard - Select Wells** dialog box and your changes will not be saved.

Click **Next**. The **Database Import Wizard - Select Data** dialog box is displayed:



Through the **Database Import Wizard - Select Data** dialog box, you select the well data that will be imported. Well data includes trajectories, perforated intervals, logs, and so on. The total number of wells to be imported and the geographic range covered are displayed at the top of the dialog box.

If a large number of wells are being imported, or if the geographic range is large, it may be desirable to refine the data retrieval for the following reasons;

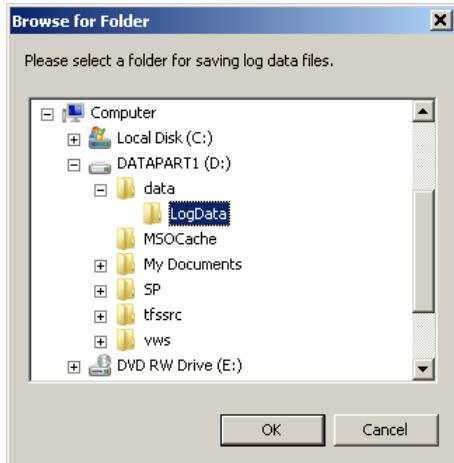
- Retrieving a large number of wells can result in a lengthy database connection time.
- A large geographic range can result in a very coarse grid.

To refine the data retrieval, click **Back** to return to the **Database Import Wizard - Select Wells** dialog box, through which you can reduce the area of interest or the number of selected wells.

It is possible to limit the well data being imported; for example, you can limit the import to production data by excluding trajectory-based data such as perforated intervals, well logs, and formation markers. To do this, cancel the **Trajectory** check box in the **Database Import Wizard - Select Data** dialog box. Note that the check boxes for Perforations, Logs and Markers will be cancelled when you do this.

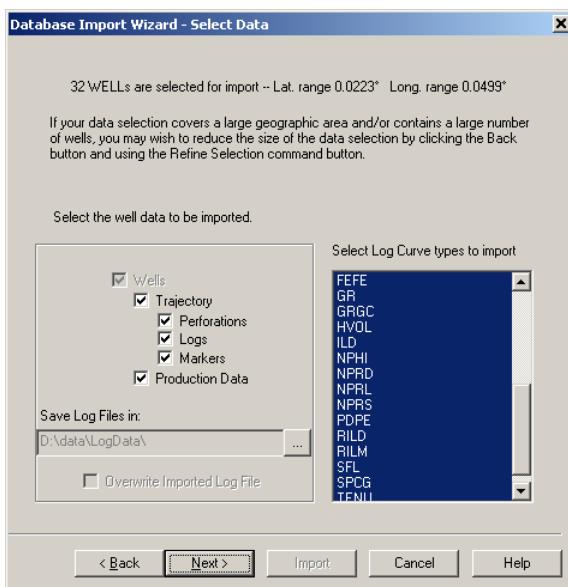
Note: Canceling **Wells** prevents any data from being imported.

If the **Logs** box is checked, you will need to select a writable directory for Builder to save the log data files to zip and “las” format. To do this, click the **Browse**  button to select a directory. After you click the **Browse**  button, the **Browse for Folder** dialog box will be displayed:

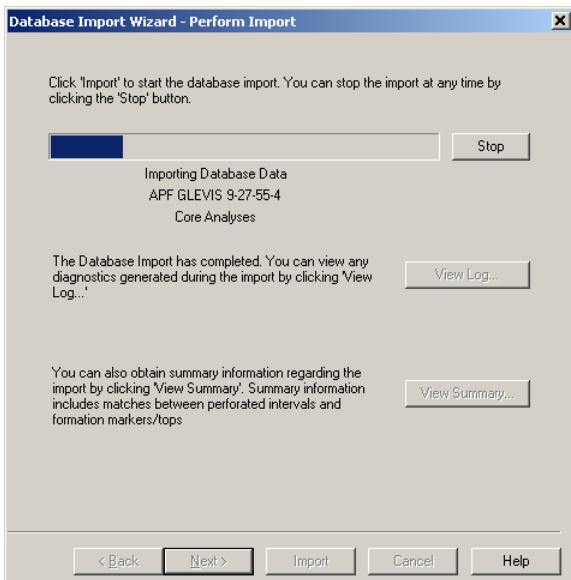


If there are any “las” format files in the selected path, the **Overwrite Imported Log File** check box is enabled. If you select the **Overwrite Import Log Files** check box, the log data file will be overwritten if there is a file with the same name in the selected directory; otherwise, Builder will read log data from the file in the selected directory without retrieving log data from the database (this will save time, but may result in the wrong log data being saved).

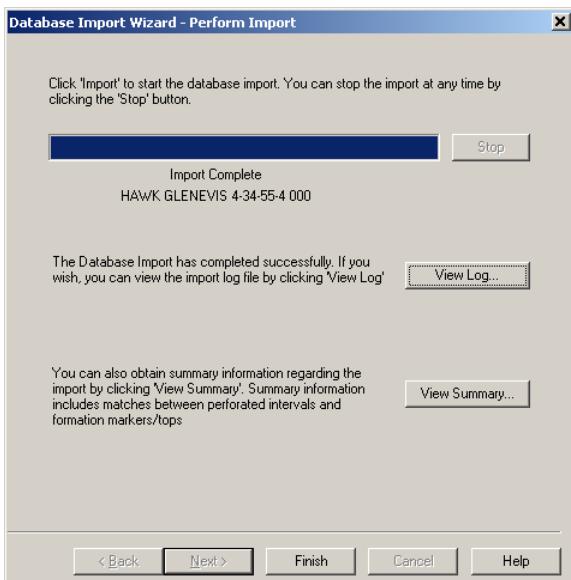
Select a writable directory (example: D:\data\LogData) and then click **OK**. The **Next** button will be enabled, as shown below:



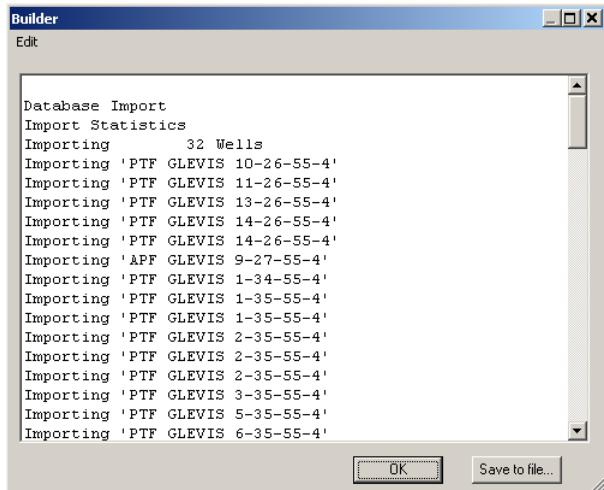
After the well data to be imported has been selected, click **Next** to begin the import from the database. The **Database Import Wizard - Perform Import** dialog box is displayed, providing progress status of the import:



The database import can be halted by clicking the **Stop** button. If the import is stopped, no changes will be made to the Builder file. If the import is allowed to proceed to completion, a summary of the import will be displayed, as shown in the following example:



If errors occur during the database import, error messages can be read by clicking **View Log**, as shown in the following example:



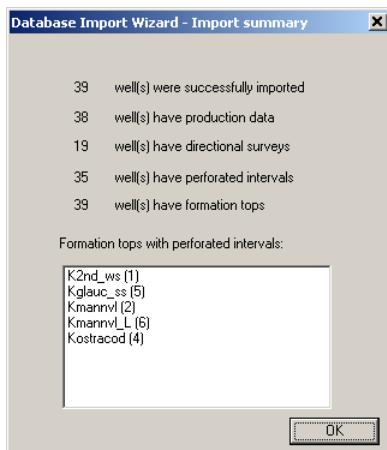
The screenshot shows a window titled "Builder" with a menu bar "Edit". The main area displays a log of database import operations. The log entries are as follows:

```
Database Import
Import Statistics
Importing      32 Wells
Importing 'PTF GLEVIS 10-26-55-4'
Importing 'PTF GLEVIS 11-26-55-4'
Importing 'PTF GLEVIS 13-26-55-4'
Importing 'PTF GLEVIS 14-26-55-4'
Importing 'PTF GLEVIS 14-26-55-4'
Importing 'APF GLEVIS 9-27-55-4'
Importing 'PTF GLEVIS 1-34-55-4'
Importing 'PTF GLEVIS 1-35-55-4'
Importing 'PTF GLEVIS 1-35-55-4'
Importing 'PTF GLEVIS 2-35-55-4'
Importing 'PTF GLEVIS 2-35-55-4'
Importing 'PTF GLEVIS 2-35-55-4'
Importing 'PTF GLEVIS 3-35-55-4'
Importing 'PTF GLEVIS 5-35-55-4'
Importing 'PTF GLEVIS 6-35-55-4'
```

At the bottom of the window are two buttons: "OK" and "Save to file...".

Click **Save to file** to save this information for diagnostic purposes.

A summary of the imported data can be viewed by clicking **View Summary** in the **Database Import Wizard - Perform Import** dialog box. The **Import summary** report is displayed, as shown in the following example:

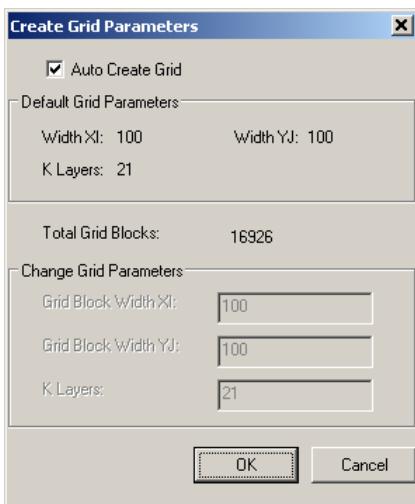


The **Import summary** report displays the number of wells that were imported, and the number that have production data, trajectories (directional surveys), perforated intervals, and formation tops. Also, the formations that are associated with perforated intervals are shown. This may be useful in selecting which geological formations to model for gridding and geostatistical analysis. The geological formation code name is displayed, as is the number of associated perforated intervals.

Note: The number of wells that were imported can be greater than the number that were originally selected. The **Database Import** wizard can recognize when wells are combined producers and injectors. In these cases, Builder effectively imports these wells twice, but assigns a slightly different name to the repeated wells. The repeated wells will be identical to the original wells, except that injection data will be retrieved instead of production data. Injection wells have well names with the suffix “iw” appended.

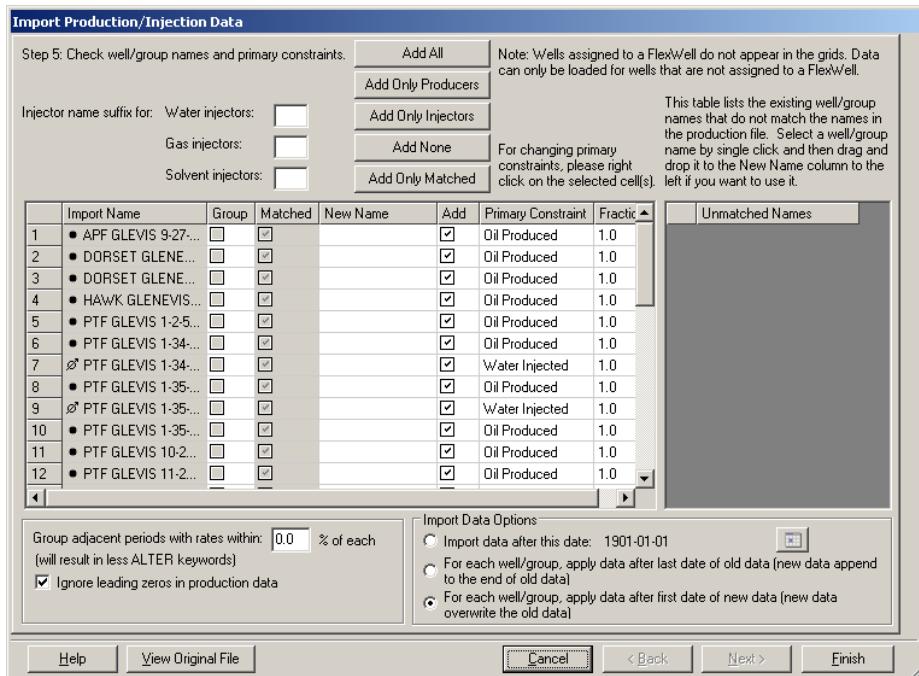
Click **Finish** to complete the data import phase. Builder will save the imported data then will perform post-retrieval processing on the data. The following sections discuss this post-retrieval processing.

If the option to create a simple grid was selected (on the **Database Import Wizard - Introduction** dialog box), then the **Create Grid Parameters** dialog box will be displayed:



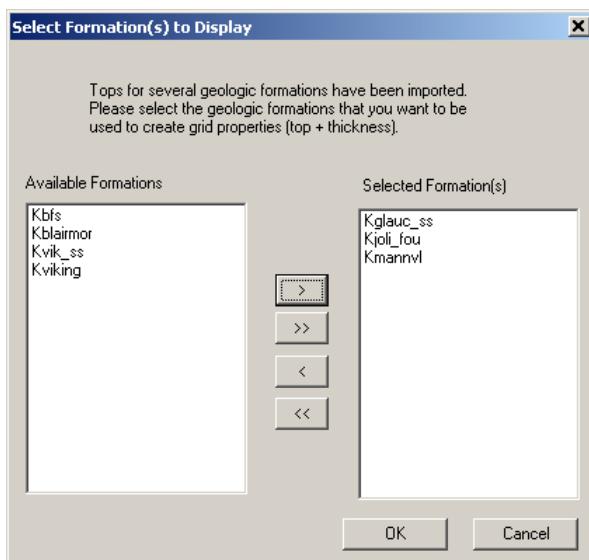
The **Database Import** wizard calculates the default gridding parameters. If these values are acceptable, click **OK** to continue; otherwise, cancel **Auto Create Grid**, and then change the Grid Block Widths and/or K Layers, as necessary. Click **OK**.

Another aspect of the post-import processing is that Builder will attempt to create events from the production data. This is accomplished by its invoking step 5 of the Production Data Wizard. Consult the [Production Data Wizard](#) section for information on using the **Import Production/Injection Data** dialog box, which is shown below:



Click **Finish**. Builder will process production data and create events.

The final phase of post-import processing is the creation of grid properties from geological formations. This is an optional phase, and will only occur if the **Create simple grid** option was selected at the start of the database import. If it was selected, the **Select Formation(s) to Display** dialog box will be displayed:



Select one or more formations to be used in the creation of grid properties. Move formation names from **Available Formations** to **Selected Formation(s)** by either double-clicking on the formation name, or by selecting the name and then clicking the button. All available formations can be selected by clicking the button. You can remove selected formations by selecting the name in the **Selected Formation(s)** table and then clicking the button, or by clicking the button to remove all of them.

Once the desired formation tops have been selected, click **OK**. Builder will create mesh maps for the selected formations then grid properties (top and thickness) for each of the selected formations.

This completes the database import process.

PPDM Coverage in Database Import

The purpose of this section is to describe the PPDM entities that are retrieved during a database import.

The Builder PPDM import retrieves selected entities from the PPDM Wells, Stratigraphy and Production Reporting groups. The following table shows the high-level mapping between PPDM entities and equivalent Builder objects. Table names in **green text** represent navigation tables, which are used to locate values in other tables, and which are not necessarily represented as Builder objects.

| PPDM Table | Builder Object/Description |
|---------------------------------|----------------------------|
| Well General Information | |
| WELL | |
| well_name | Well |
| uwi | |
| parent_uwi | |
| surface_longitude | |
| surface_latitude | |
| bottom_hole_longitude | |
| bottom_hole_latitude | |
| drill_td | |
| drill_td_ouom | |
| max_tvd | |
| max_tvd_ouom | |
| WELL_NODE | |
| latitude | Well |
| longitude | |

Well General Information

| | |
|------------------------------|------------|
| WELL_DIR_SRVY | Trajectory |
| WELL_DIR_SRVY_STATION | TrajNode |
| x_offset | |
| x_offset_ouom | |
| y_offset | |
| y_offset_ouom | |
| station_tvd | |
| station_tvd_ouom | |
| station_md | |
| station_md_ouom | |
| WELL_DIR_SRVY | Trajectory |
| WELL_DIR_SRVY_STATION | TrajNode |
| x_offset | |
| x_offset_ouom | |
| y_offset | |

Well Perforations, Completions and Cores Information

| | |
|------------------------------|-------------------------------------|
| WELL_PERFORATION | TrajPerf, TrajPerfInterval |
| perforation_date | |
| top_depth | |
| top_depth_ouom | |
| base_depth | |
| base_depth_ouom | |
| WELL_CORE_SAMPLE_ANAL | Used to synthesize well logs for... |
| bulk_density | Core Bulk Density |
| effective_porosity | Core Effective Porosity |
| k90 | Core K90 |
| kmaz | Core KMax |
| kvert | Core KVert |
| oil_sat | Core Oil Saturations |
| pore_volume_gas_sat | Core Pore Vol. Gas Sat. |
| pore_volume_oil_sat | Core Pore Vol. Oil Sat. |
| pore_volume_water_sat | Core Pore Vol. Water Sat. |
| porosity | Core Porosity |
| water_sat | Core Water Saturation |

Well Logs Information

WELL_LOG

| | |
|----------------------|----------------------------------|
| WELL_LOG_CURVE | WellLog |
| reported_mnemonic | |
| RM_INFO_ITEM_CONTENT | |
| RM_DATA_CONTENT | |
| RM_DATA_STORE | |
| long_location | Contains LAS filename (ZIP file) |
| WELL_LOG | |
| WELL_LOG_CURVE | WellLog |
| reported_mnemonic | |
| RM_INFO_ITEM_CONTENT | |
| RM_DATA_CONTENT | |

Well Formation Markers Information

STRAT_NAME_SET

| | |
|--------------------|--------------------------|
| STRAT_UNIT | |
| long_name | |
| ordinal_age_code | |
| STRAT_WELL_SECTION | TrajTop, TrajTopInterval |
| pick_depth | |
| pick_depth_ouom | |
| STRAT_NAME_SET | |
| STRAT_UNIT | |
| long_name | |
| ordinal_age_code | |
| STRAT_WELL_SECTION | TrajTop, TrajTopInterval |

Well Production Data

PDEN_WELL

| | |
|-------------------|---|
| PDEN | |
| PDEN_VOL_BY_MONTH | ProductionData |
| year | |
| product_type | (‘OIL’, ‘GAS’, ‘WTR’, ‘INJHRS’) |
| activity_type | ‘INJECTION’ or ‘PRODUCTION’ |
| jan_volume | Monthly production or injection volumes |
| ... | |
| dec_volume | |

PDEN_WELL

| | |
|-------------------|---------------------------------|
| PDEN | |
| PDEN_VOL_BY_MONTH | ProductionData |
| year | |
| product_type | (‘OIL’, ‘GAS’, ‘WTR’, ‘INJHRS’) |

Field names that end with “_ouom” contain unit-of-measure strings (for example, “ft.”, “m”), and are used by the database import to perform appropriate unit conversion.

Merging Data Sets

If data is being imported from a database or text files into an existing data set, data merging will usually be required. Data merging involves the combining of well data from one data set (the source) with matching well data in a second data set (the destination).

Data merging is performed using the **Data Merge** wizard. The **Data Merge** wizard will automatically open whenever Builder needs to merge data from two data sets, typically after a data import has been performed.

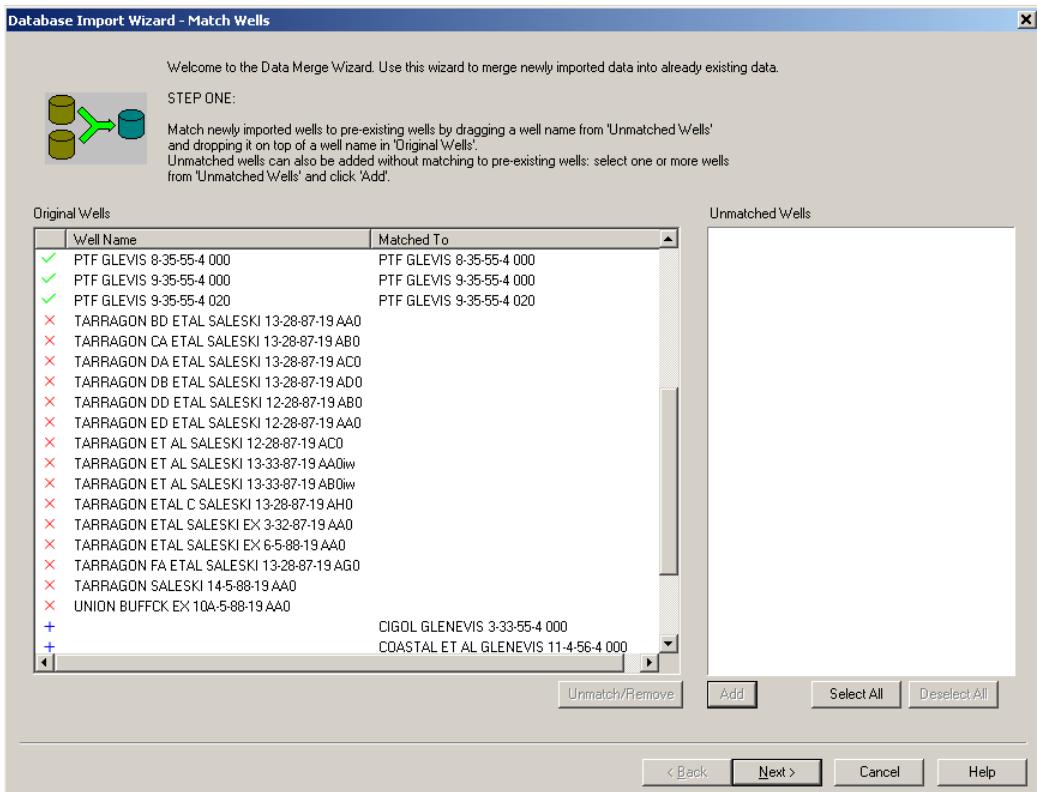
The goal of data merging is either to replace data in the destination well with data from the source well, or to insert data from the source well into the destination well. If data from the source well corresponds to data in the destination well (for example, trajectory nodes or perforated intervals), the data from the source well will replace the equivalent data in the destination well; however, if there is no correspondence, the data from the source well will be inserted into the destination well.

The basic data merge workflow consists of two steps:

1. Match each well in the source data set with a well in the destination data set.
2. Match each trajectory in each source data set well with the appropriate trajectory in the matched well in the destination data set.

The first step involves matching wells between the two data sets. The wizard will attempt to match as many wells as possible, comparing well names and recognizing a match when a well name from the source data set is identical to a well name in the destination data set. If the wizard cannot automatically match a well in the source with a well in the destination, you must attempt to manually match the wells. If there is no match for the well, you can add the unmatched well directly to the destination data source.

When the wizard starts, the **Database Import Wizard - Match Wells** dialog box is displayed, for example:



Automatically matched wells are displayed with a green check mark, and both destination and source well names are displayed. Unmatched wells have a red mark beside their names.

Wells that require a manual match are displayed in the list on the right. To manually match an unmatched well, drag its name from the list on the right and drop it on an unmatched well in the left listing. Note that it is not permissible to match a well that has already been matched.

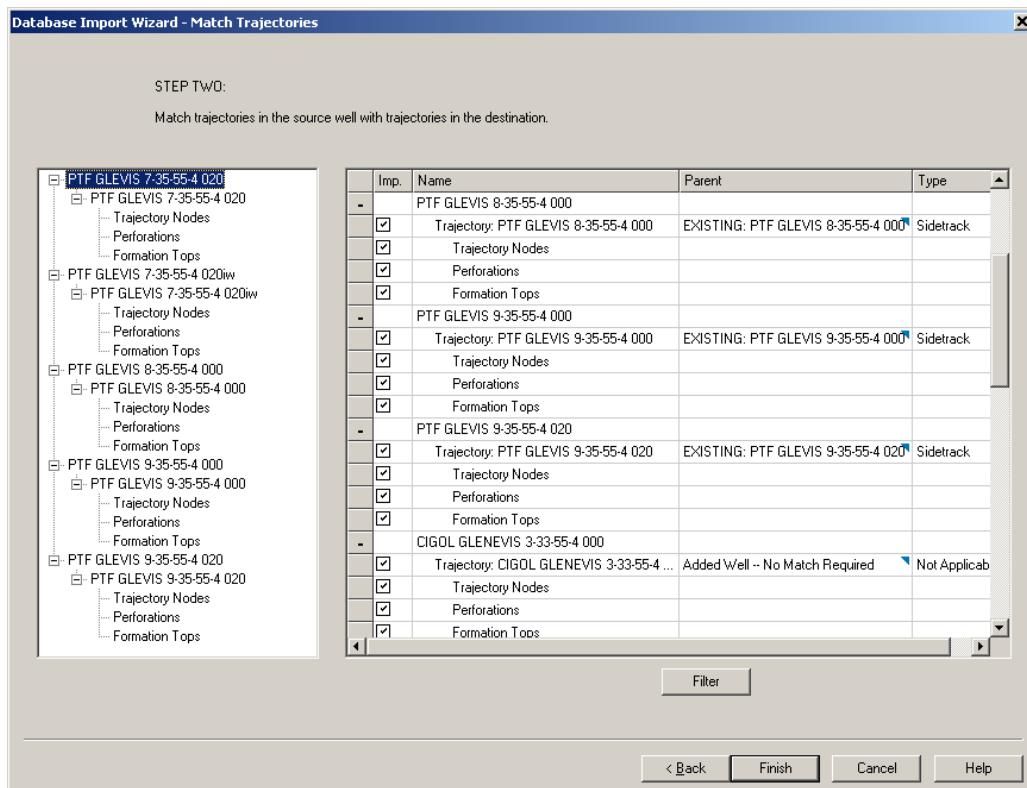
If a manual well match is successful, the red mark will be replaced with a green check mark, the matching well name will be displayed in the **Matched To** column, and the name will also be removed from the right list.

To add unmatched wells to the destination database, select the wells in the **Unmatched Wells** list and then click **Add**. The wells will be moved to the **Original Wells** table, displayed with a blue mark.

It is possible to cancel an existing match. Highlight the appropriate row in the list and click **Unmatch/Remove**. The green check mark is replaced with a red mark and the well name appears in the **Unmatched Wells** list.

Note: Matched wells do not have to have the same well name. There are several reasons why the same well might have different names. One reason is that the well rights were sold and purchased by a different company than the original operator of the well. Matching these two wells requires that you possess additional information (for example, comparing well locations) in order to manually match the wells.

Click **Next** to proceed to step 2 of the **Database Import** wizard:



The **Database Import Wizard - Match Trajectories** dialog box contains two lists. The left hand list shows all wells from the destination data set, with well data that is available for each well (trajectory nodes, perforations, formation markers, well logs). The right hand list shows all wells from the source data set. If a well from the source data set has been matched to a well in the destination data set, the well name from the destination will be displayed in the **Parent** column. If a well from the source data set has been directly added to the destination data set (that is, it has not been matched to a well in the destination data set), the **Parent** column will show *Added Well -- No Match Required*.

The **Match Trajectories** dialog box has two purposes:

1. To specify the parent-child relationship between a trajectory in the source well and a trajectory in either the source well or the destination well.
2. To specify, on a per-well basis, the well data to be merged

If a source well contains multiple trajectories (that is, it is a multilateral well), there will be multiple entries for that well in the right hand list. Similarly, a multilateral well in the destination data set will have multiple entries in the left hand list. Alternately, it might be that the source well/trajectory pair actually represents a specific trajectory in the destination well. Regardless, the purpose of trajectory matching is to make the relationships explicit.

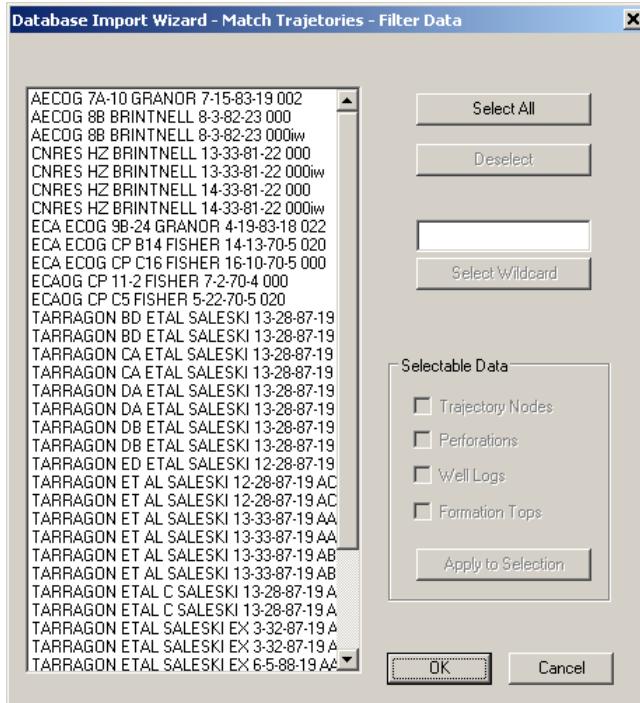
To match a trajectory to its parent (which could be in either the source well or the destination well), click the appropriate cell in the **Parent** column, and expand the drop-down list. The list contains candidate trajectory names from both the source and destination wells. Trajectories from the source well are labeled as “**NEW**”, while trajectories from the destination well are labeled “**EXISTING**”. Make the appropriate selection.

It is possible to specify the well data (trajectories, perforations, and so on) that will be merged on a per-well basis. The grid on the **Database Import Wizard - Match Trajectories** dialog box shows two columns that can be used to specify which well data is to be merged. The **Imp** column shows a series of check boxes, while the **Name** column shows well and trajectory names, in addition to the well data that is available. The check boxes in the **Imp** column correspond to entries under the **Name** column.

To merge specific types of well data, ensure that the check boxes under **Imp** are correctly selected. For example, to merge trajectory nodes and perforations, but not formation tops or well logs, make sure that the check boxes associated with trajectory nodes and perforations are checked, and the check boxes for well logs and formation tops are not checked.

Note: Not every well will contain every type of well data. Only well data that actually occurs in the well will appear under the **Name** column.

It is possible to perform large scale (bulk) changes to the well data to be merged. This is achieved through the **Filter Data** dialog box:



The **Database Import Wizard - Match Trajectories - Filter Data** dialog box shows the wells that will be merged into the destination data set (either matched or added).

To change the well data that will be merged, make a selection in the list box, select the appropriate well data in **Selectable Data**, and then click **Apply To Selection**. Wells can be selected by:

1. Clicking on individual wells in the list box.
2. Clicking **Select All**.
3. Entering a selection string using wildcard characters to select matching well names.

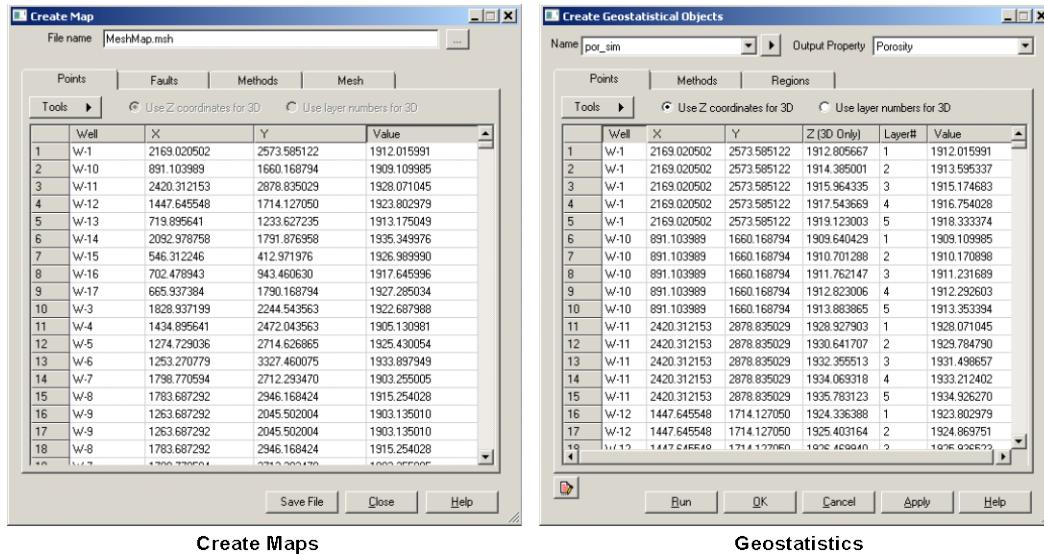
The changes to the well data to be merged are made permanent after **OK** is clicked.

Complete the **Data Merge** wizard by clicking **Next**.

Creating Maps and Geostatistical Property Calculations

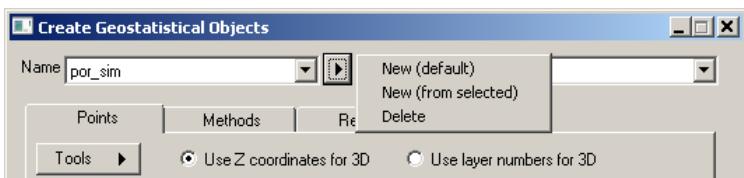
Overview

This feature is designed to help you create quick maps or grids and geostatistical realizations. The input to this feature is well and/or fault data points (tops, thickness, porosity, permeability, and so on), and the program will use one of the calculation methods to calculate values at other locations. This option can be invoked in 2D from either Results 3D or Builder by selecting the menu item **File | Create Map**, or in 3D from Builder by selecting the menu item **Reservoir | Geostatistics**. The option **File | Create Map** is always available while the option **Reservoir | Geostatistics** is available only if a grid is already defined. The purpose of the **File | Create Map** option is to give you a quick tool for creating reservoir maps (from well marker points) which can be used in building grids. The purpose of the **Reservoir | Geostatistics** option is to populate existing grids with reservoir properties. The dialog boxes for each option are shown below:



The **Create Map** dialog box has an input field for entering the name of the file to which you want to save the results of your calculations. The filename is defaulted to *MeshMap.msh* and the results are written in the Builder mesh ASCII format (see [Creating a Simulation Grid Using Structure Maps](#)).

The **Geostatistics** dialog box has two pull down selection lists at the top. The **Name** list is used to select existing geostatistical calculation objects and the **Output Property** list is used for choosing the Builder property to which the geostatistical calculations will apply. To the right of the **Name** list, a button opens more options for creating new geostatistical calculation objects or for deleting existing ones.



The two dialog boxes are otherwise similar. The tabs displayed in the dialog boxes are described in the following sections.

Points Tab

The table on the **Points** tab is used to input data points. The table will display four predetermined columns in 2D (Create Map) and six in 3D (Geostatistics). The **Well** column is reserved for an optional well name. The **X**, **Y**, and **Z** columns (only in 3D) are reserved for the data point coordinates, and the **Value** column is reserved for the parameter value at the corresponding location. The **Layer#** column only shows in 3D and is used only when the **Use layer numbers for 3D** option is selected. The data can be copied from other documents and directly pasted into the cells of the table or alternatively, the **Tools** button can be used. The **Tools** button provides five methods for inputting well locations and parameters:

| Points | | | | Faults | Methods | Mesh |
|--------|------|---|-------------|-------------|-------------|---|
| Tools | | Import wells and faults from a contour map... | | | | |
| | | | | | | Import wells and faults from a contour map... |
| | | | | | | Import wells from the view... |
| | | | | | | Import logs or tables of measured depth values... |
| | | | | | | Add points with mouse clicks |
| | | | | | | Import well test permeabilities... |
| | | | | | | Import tops from trajectories... |
| Tools | | Import wells and faults from a contour map... | | | | |
| 1 | W-1 | 1447.645548 | 1714.127050 | 1912.015991 | 1909.109985 | 1928.071045 |
| 2 | W-10 | 719.895641 | 1233.627235 | 1923.802979 | | 1913.175049 |
| 3 | W-11 | | | | | |
| 4 | W-12 | | | | | |
| 5 | W-13 | | | | | |

1. The first method is to extract them from an existing map file. The **Import wells and faults from contour map** option is available on the **Points** tab. If a map is available that contains well locations and fault points, this map can be opened with this button, and the well locations and fault points will be extracted from this map and placed in the appropriate tables for this option. Then, you can input parameter values for these well locations to create a new map.

2. The second option is to extract the well locations from the simulator data file. These well locations may be different from the actual locations because the simulator data file well locations have been moved into the center of a grid block. The parameter values will be extracted from the currently displayed grid property in the Builder main view. They will be added into the table at the currently selected cell. Thus, existing values in these cells in the table may be overwritten. For the **Create Map** option (2D), only data from the Builder main view selected layer (K-plane) will be imported.
3. The third method is to input well X,Y,Z coordinates and values from logs and trajectories. In 3D, the option to import only the coordinates that lie within the model grid will be enabled. If this option is chosen, then if more than one log point is located within a particular grid block, these log points will be averaged (simple average method) and only one value will be imported for each block. If the option to import all points is selected, then when the log values are imported from the view, all log points will be imported.
4. The fourth method is to input points by clicking the mouse on the currently displayed grid property. This method will add locations and parameters into the table after the selected cell in the table, but well names will not be provided. Therefore, these points will be used in the calculation, but will not appear as wells on the view.
5. The fifth method is to import values from well test data and is available only in 3D.
6. The sixth method is to import values from the trajectory top data that was loaded from an LAS V3.0 or CMG formation top file (.tdb).

Faults Tab

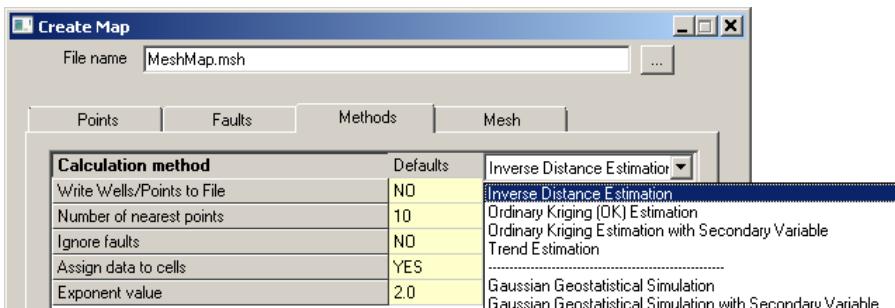
The **Faults** tab is only available for the 2D option. There are three methods available for inputting fault points.

1. The first method is to extract them from an existing map file. This option is available on the **Points** tab.
2. The second method is to input points by clicking the mouse on the currently displayed grid. This method will add fault points for the particular fault that is selected, and will place them into the table after the currently selected cell. Note that this method will overwrite any existing data points in the cell.
3. The third method is to paste values in the table from a spreadsheet such as Excel.

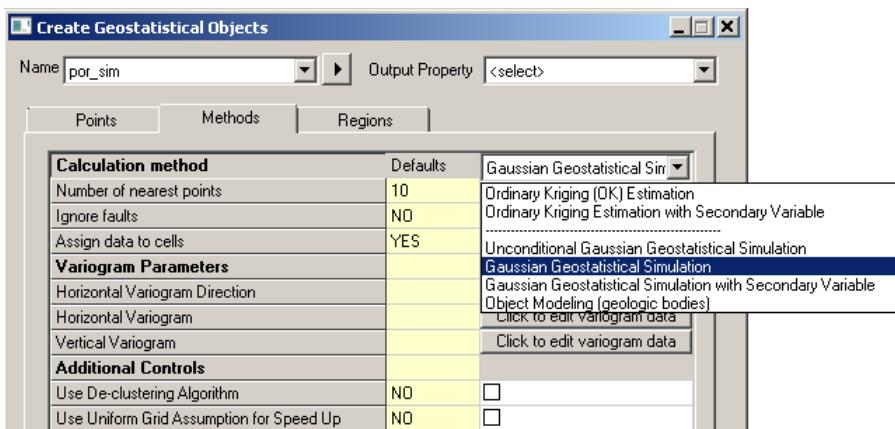
The use of faults in the generation of maps or geostatistical realizations can be disabled by selecting **Ignore faults** on the **Methods** tab. If **Ignore faults** is not selected, the 2D calculation of maps will be based on nearby points that do not cross any fault lines. For 3D geostatistical realizations, the calculations will use fault planes that are already built into the structure of the 3D model grid.

Methods Tab

Geostatistical methods can be divided into four categories: two main categories, *estimation* and *simulation*, which can be further divided into methods for *continuous* (for example, porosity) or *categorical* data (for example, rock types). Use estimation when you need a smooth interpolation across the data. In contrast, use simulation when you need to reproduce the variability actually seen in the data. As opposed to estimation methods which generate only one result, simulation methods can generate multiple cases. For this reason, simulations are important for uncertainty analysis. The 2D option offers estimation and simulation for continuous data only. The 3D option offers estimation and simulation for continuous data, and simulation for categorical data. Normally, estimation methods require at least two wells for data input. There are currently six estimation and simulation methods available for creating maps, as shown below:



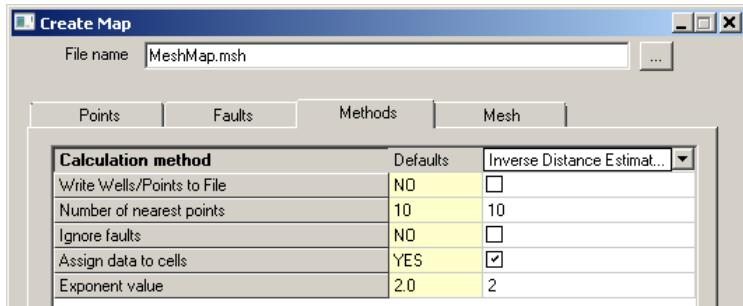
There are currently six estimation and simulation methods available for creating geostatistical calculation objects, as shown below:



The parameters used in the calculations will vary with the method selected. These methods are described in the following sections.

Inverse Distance Estimation

In 2D, the first and default method is to use an inverse distance calculation:



This calculation method uses the following procedure for calculating values at mesh cells that do not contain a well point:

- For a mesh cell (i,j), a search for the **Number of nearest points** will be done. The default value of 10 can be changed on the **Methods** tab.
- The value at mesh cell (i,j) is calculated by:

$$\frac{1}{\sum_i \frac{1}{d_i^{\text{exp}}}} \cdot \sum_i \frac{V_i}{d_i^{\text{exp}}}$$

where:

- V Nearby data value
D Distance from data to mesh cell
exp Exponent value.

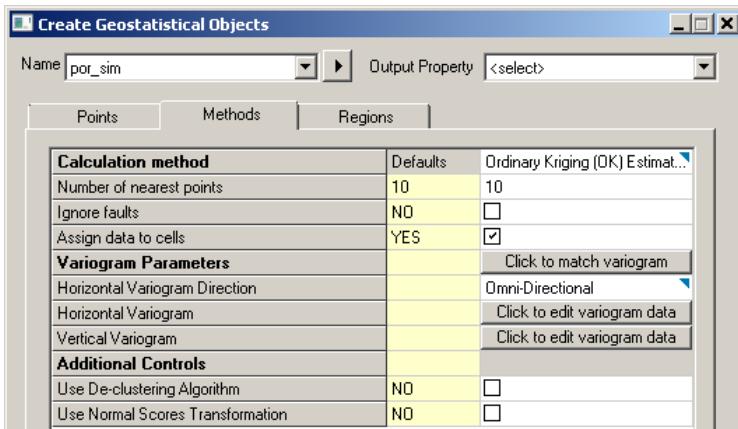
The option to **Write Wells/Points to File** is available for all 2D estimation methods. Use it to save the data along with the results. The option to **Assign data to cells** is also available for all 2D methods. It is used to force a data value at a grid node as oppose to an estimated value. The data point closest to the node will be used.

Ordinary Kriging (OK) Estimation

Another method, available in 2D and 3D, is to use the Kriging method, which is the method normally used when it is important to account for spatial correlation and anisotropy among the data. The Kriging method requires the use of variograms and search areas. The variogram is a function of the data variance against the distance between data locations. It describes the spatial structure and is used to model the spatial correlation in the data. Each dataset is characterized by its own spatial structure, and thus has its own variogram. This option has an automatic variogram matching utility that creates and matches the variogram using a non-linear least squares type of fitting routine. This makes map creation very easy. Usually, the map is substantially improved when compared to the inverse distance or trend methods. The map can

be improved further by improving the variogram match, or by changing the anisotropy parameters. More information on variograms can be found in [Variogram Matching](#).

As with inverse distance, the Kriging estimator is also a linear combination of the nearest data points: $\sum_i w_i \cdot V_i$ with $\sum_i w_i = 1$. The weights w depend on the variogram used.



Ordinary Kriging estimation has additional controls such as **Use De-clustering Algorithm** and **Use Normal Scores Transformation** of the data.

Declustering is used to reduce possible bias in the data histogram. For example, declustering will help to reduce the influence of over-sampling in sweet spots. The clustered data will have lesser weights than isolated data points. If not used, the data are all equally weighted ($1/n$ where n is the number of the data points). The declustering option is used in conjunction with the normal scores transformation. A normal scores transformation is usually used when the data histogram is strongly skewed. Normalizing the data histogram may help to obtain a better estimation because Kriging algorithms are optimum when the data histogram is normal (experience has shown that Kriging is robust against departure from normality). The results are transformed back into the original histogram or into the declustered histogram after estimation.

Ordinary Kriging with Secondary Variable

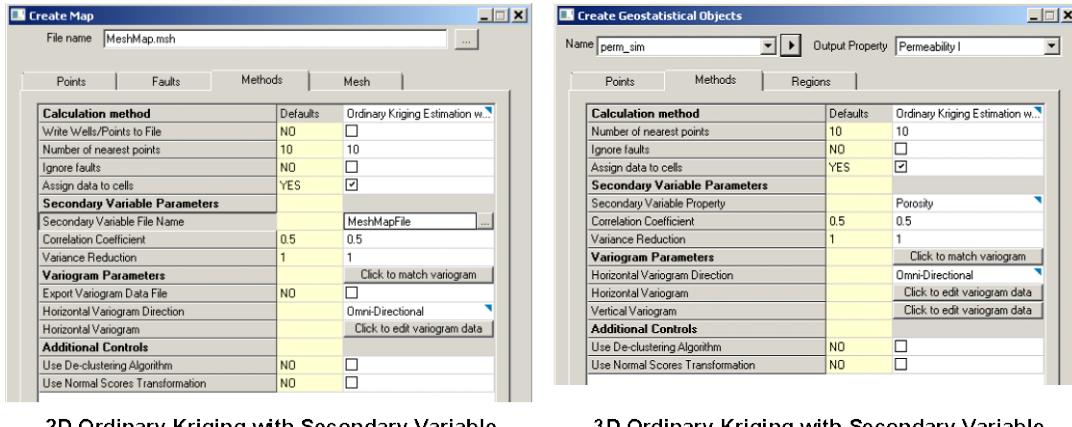
This option can be used when you have a secondary attribute for estimating the primary variable. The values of the secondary attribute will be combined with the data input values for a particular property. Note that the secondary attribute needs to be known at the grid cells in 2D or at the grid blocks in 3D. The Kriging estimator is a linear combination of the nearest data points of the primary variable V and the secondary variable U at the grid cell or grid block being estimated:

$$\sum_{i=1} w_i \cdot V_i + w_0 \cdot (U_0 - m_U + m_V) \text{ where } \sum_{i=0} w_i = 1$$

The weights w_i depend on the variogram used and on the correlation between variables V and U . To avoid bias, the values of the secondary variable U are shifted so that they have the same mean (m) as the values of the primary variable V . An example of the use of this option in 2D would be to calculate structure top from well and seismic data together. Another

example, in 3D, would be to calculate permeability values by taking into account the correlation with the previously calculated porosity values.

The following show the options for Ordinary Kriging with Secondary Variable, for both 2D and 3D:



2D Ordinary Kriging with Secondary Variable

3D Ordinary Kriging with Secondary Variable

In the 2D option, the secondary variable input is from a mesh file. In the 3D option, the secondary variable input is from an existing Builder property, such as Porosity.

A **Correlation Coefficient** must be specified. Its value can range from -1 to +1. Using a correlation coefficient of 1 (or -1) will give all the Kriging weights to the secondary variable and none to the primary attribute. Conversely, using a correlation of 0.0 will give all the Kriging weights to the primary variable and none to the secondary attribute. Therefore, the coefficient of correlation is used to give more or less weight to the secondary variable.

Usually, the coefficient of correlation is obtained by cross plotting the collocated values of the primary and the secondary variable.

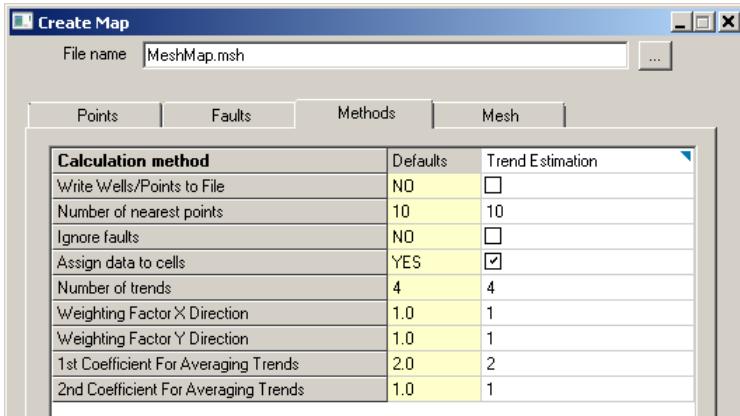
The **Variance Reduction** parameter is used to reduce the variance of the secondary variable. This is sometimes necessary when the variance of the secondary variable is much larger than the variance of the primary variable. This situation may lead to an overestimation of the Kriging variance. The variance reduction parameter is generally used in conjunction with the **Normal Scores Transformation** option. The **Normal Scores Transformation** always creates transformed variables with a variance equal to 1.0. It may be necessary to adjust the variance of the transformed secondary variable so that it is smaller than the transformed variance of the primary variable (1.0).

The work flow required to do this is very similar to the work flow example given in the section [Gaussian Geostatistics and Secondary Variable](#).

Trend Method

Another estimation method is the trend method, where the trend between two data points (wells) is calculated and applied at different locations. The default is to calculate four such trends, and then to average the trends based on an inverse distance calculation (the exponent for this inverse distance calculation is calculated from the values input on the **Methods** tab:

1st Coefficient For Averaging Trends, 2nd Coefficient For Averaging Trends, and the mesh cell sizes).



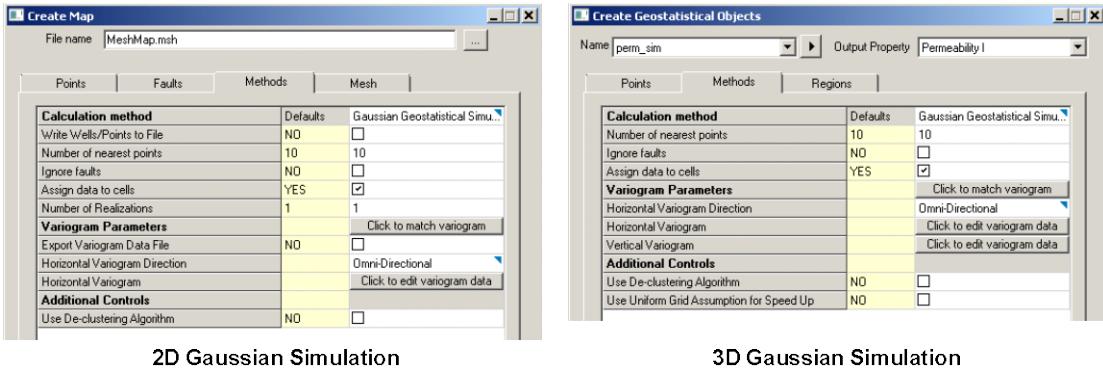
The trend method requires you to input **Number of trends**, **Weighting Factor X Direction**, **Weighting Factor Y Direction**, **1st Coefficient For Averaging Trends**, and **2nd Coefficient For Averaging Trends**.

The trend method usually predicts smooth contours which follow field trends properly. However, if two data points which are close to each other have parameter values that are very different, the trend method will predict a very sharp trend which rises or falls very rapidly. In this case, a zone of high or low values will be predicted in a region where no data exists. If you want to remove this region of high or low values, you can input a pseudo well close to the other two wells to prevent the sharp trend from being predicted.

The trend method is only available in 2D.

Gaussian Geostatistical Simulations

The implementation of these simulation options is based on GsLib², enhanced to handle rotated grids, and variable grid sizes, including non-orthogonal corner point grids. Gaussian simulations are used to generate multiple versions (called realizations) of grid values that all reproduce the data, reproduce the histogram of the data, and reproduce the variogram of the data. Contrary to the Kriging routines described earlier, they add an element of randomness to account for the uncertainty associated with some reservoir properties. If you average multiple realizations, you would obtain results that are very close to those obtained with the Kriging estimation methods. These simulation methods are generally used for calculation of properties such as porosity, permeability, and so on. They can also be used for the calculation of integer indicator type properties such as rock type because allowances have been made to handle integer properties. The more correct method to handle indicator type properties is to use the SISIM routines available in GsLib; however, until this method is implemented in Builder, the Gaussian Geostatistics method can be used.



2D Gaussian Simulation

3D Gaussian Simulation

Number of Realizations

This option is only available with Create Map in 2D (in 3D the number of realizations is set into a script. See [Property Calculation Scripting](#) for further details). If the number of realizations is set to be greater than 1, Builder calculates and writes mesh maps (*.msh) using the input base file name, appended with a number that corresponds to the realization number.

Use Uniform Grid Assumption for Speed Up

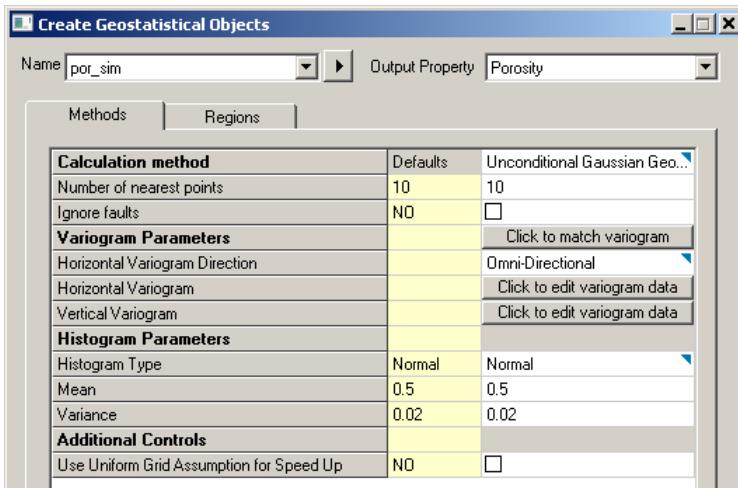
This option is only available in 3D where the grid is allowed to have blocks of varying size. Selecting this option will speed up the calculations. It allows the pre-calculation of the variogram values needed for the Gaussian simulation. This is done by assuming the grid is uniform. The averaged grid block dimension is used to build a lookup table for fast reading of the variogram values. These values may not be accurate when the size and location of the grid blocks vary greatly. Use this option in cases where you can trade some precision for speed.

Gaussian Geostatistics and Secondary Variable

This option is similar to Kriging using a Secondary Variable. The most common use of this option is to calculate permeability from core permeability and log porosity. The primary variable would be the core permeability. Since permeability is related to porosity in some fashion, porosity would be the secondary variable. If seismic data was available, it could also be used as a secondary variable.

Unconditional Gaussian Geostatistical Simulation

This option is similar to the Gaussian Geostatistical Simulation, described above. The main difference is that there is no need to define data points. Therefore, data conditioning is not required for this method, and the **Points** tab is not displayed on the **Create Geostatistical Objects** dialog box:



There are four options for controlling the distribution (histogram) of the simulated values. The histogram of the simulated values can be Uniform, Triangular, Normal, or LogNormal. Uniform and Triangular histograms are controlled by a minimum and a maximum value, whereas the Normal and the LogNormal histograms need a mean and a variance value. Sometimes, a combination of mean and variance values can generate negative simulated values. In such cases, the negative values will be truncated at 0.0 if the output property cannot be negative. This may generate abnormal proportions in the lower tail of the histogram.

Unconditional simulations are useful for testing assumptions on certain properties when no sampling data is available.

Object Modeling

The Object Modeling method is available only for 3D geostatistics. This means that:

- **Reservoir | Geostatistics** menu item must be selected
- Currently displayed property must not be a structure property
- Mesh creation option is set to **Use model grid and populate the grid directly**
- Grid type is set to 3D

The Object Modeling option uses the Marked Point Simulation method to define objects within the 3D grid system. The major steps in this technique are as follows:

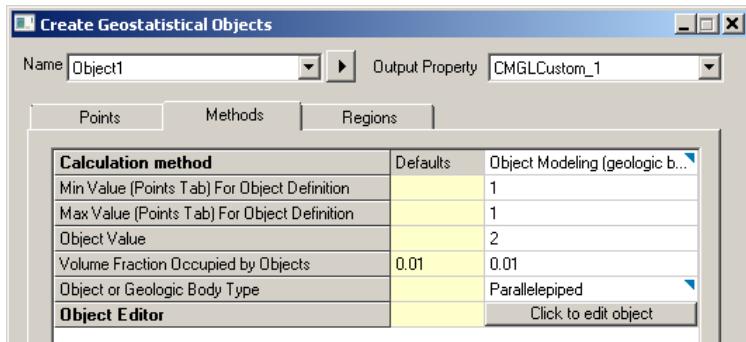
1. Select a location from the conditioning data set to insert an object. The object locations are drawn one at a time from the conditioning data.
2. Insert an object at the location of the conditioning data selected in step 1. Draw the dimensions and orientation of the object, and place its centroid at the conditioning data location.
3. Check if objects have been inserted at all conditioning data locations. If there are conditioning data locations without an object, return to step 1. If all conditioning data locations have been populated with objects, proceed to step 4.

4. Select the location of an object away from the conditioning data locations. Draw, at random, the coordinates for the centroid of the object.
5. Insert an object at the location selected in step 4. Draw the dimensions and orientation of the object, and place its centroid at the location from step 4.
6. Calculate the volume fraction of the objects. Return to step 4 if the volume fraction is below the target value. Stop the simulation when the volume fraction reaches the target volume fraction.

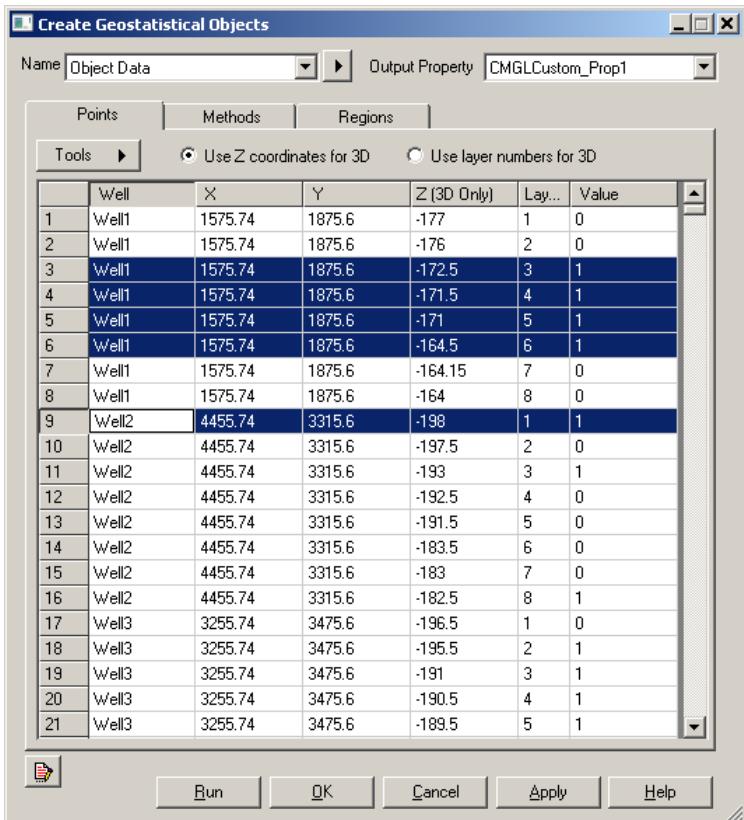
This simulation process can be repeated for other types of geologic facies that are represented with objects of different properties and volume fractions.

Data Input Requirements

There are four numeric values required to define the object locations within the 3D grid system. The first two values (min and max values for object definition) are used to define objects that have fixed locations determined from the conditioning data. The next required numeric input value is the Object Value. This value will be assigned to grid blocks that are located within the object limits. Normally, the displayed property should be an integer or region type property such as Relative Permeability Set Number, or Sector Array, so that special properties can be assigned to the grid blocks located within the object limits. The last numeric input value required is the volume fraction occupied by the objects. When this volume fraction is exceeded, the Marked Point Method will stop execution (see description of this method in the preceding section. Currently, the type of objects can be chosen as parallelepiped or channel.



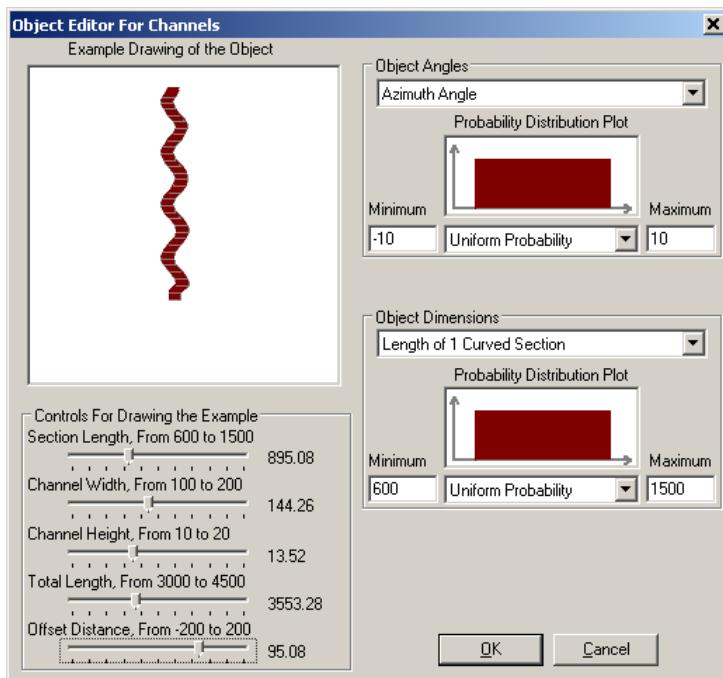
A conditioning data set is shown below. If the minimum and maximum values for object definition are set to 1, then the first object will be placed at the 3D point (1575.74, 1875.6, -172.5) because the data value is greater than or equal to 1 (min value), and less than or equal to 1 (max value). The height of this object will be equal to $(-164.15) - (-172.5) = 8.35$. The second object will be placed at the 3D point (4455.74, 3315.5, -198) and the height of the object at this point will be $(-197.5) - (-198) = 0.5$.



Criteria for Choosing Object Modeling

The most accepted conditional simulation techniques to describe geologic facies are object modeling and Sequential Indicator Simulation (SIS) (not yet available in Builder) technique for categorical variables. Two groups of geostatistics practitioners that advocate the use of either object modeling or SIS have emerged in recently published field cases. Although, there are no general rules for choosing one technique over the other, a few guidelines, for deciding the conditions that make object modeling more appropriate can be derived, based on the type of information available.

In Builder, the shape parameters and the dimensions of the objects are selected with the Object Editor. Through **Create Geostatistical Objects | Methods** tab, select the **Object Modelling** calculation method, select the **Geologic Body Type (Parallelepiped or Channel)**, then click the **Click to edit object** button. If you select *Channels*, the **Object Editor For Channels** dialog box will be displayed. An example is shown below:



Depositional Environment. Object modeling is recommended for fluvial, deltaic and deep marine depositional environments. The shape of most facies in these environments, like channels, mouth bars, levees and different types of shale, are often reasonably preserved in the subsurface and can be represented by discrete objects with unique shapes. Usually, geologists can estimate dimensions and volume fractions of facies based on modern river analogs, outcrops, ground-penetrating radar and seismic data. Object modeling is less applicable to carbonate environments because facies have gone through post-depositional processes, such as dolomitization. Therefore, it is difficult to approximate the shape of facies and estimate their dimensions in carbonate environments. Still, a few applications of object modeling have been found attractive for quantifying the distribution of laterally continuous, but thin, sedimentary bodies in a carbonate environment.

Conditioning Data. Object modeling is recommended during early appraisal stages of a reservoir and when limited wells with conditioning data are available. It is difficult for object modeling to honor conditioning data in several wells that are closely spaced and when facies extend across multiple wells. Data conditioning is not a problem for SIS.

Proportion of Objects. Object modeling works better when volume fractions of facies represented by objects are small and volume fractions of the background facies are large. Examples include a reservoir that has 5% volume fraction of shale objects within a sand background or a reservoir that has about 30% volume fraction of channel objects within a shale background. Object modeling can fail to reproduce the specified dimensions, in cases where volume fractions of facies are large and of about the same magnitude. Excessive eroding among objects can make reproduction of target volume fractions difficult and can result in abnormal shapes.

Object Dimensions. Object modeling can encounter difficulties, while generating descriptions when object dimensions are large, relative to the well spacing and simulation area. The problems arise when the marked point simulation process attempts to place an object at a conditioning well, and because of its large dimensions, the object extends over to a nearby well which does not have conditioning data at this location. The simulation process attempts to iterate around these conflicts with nearby wells; however, because the object's dimensions are large, it will not converge, even after several iterations.

Seismic Data. Object modeling is not well suited to condition descriptions with seismic data. The marked point process does not have a mechanism that can easily use the volume of reservoir attributes, generated with recently developed seismic inversion techniques. Seismic inversion provides reservoir attributes, which can be correlated to geologic facies or convey direct information about trends and lateral variations of facies. One approach proposed to incorporate the seismic attribute data into the marked point process, which indicate that channels tend to have lower seismic impedance than non-reservoir facies. However, only limited results were presented, and more evaluations of this technique are needed to prove the effectiveness of incorporating seismic attribute data.

Available Data. Object modeling requires information external to the data available from well logs and cores. The data about dimensions, shapes and orientations of various objects can be obtained from outcrop analogues, modern analogues, and literature data. However, if such information is not readily available, object modeling could render results that are meaningless and far removed from reality.

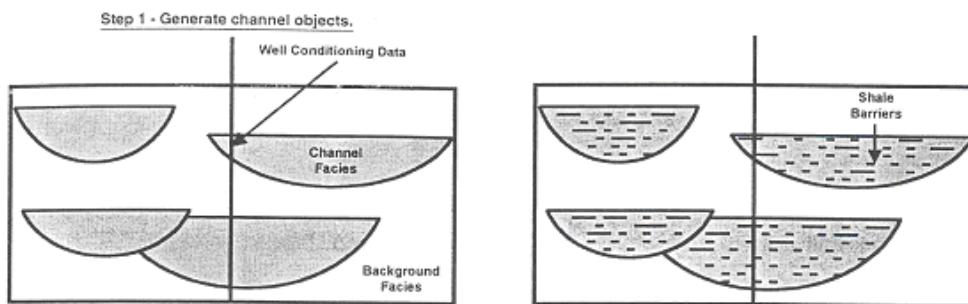
Hybrid Conditional Simulation Approaches

This chapter, as well as previous ones, has shown that individual conditional simulation techniques suffer from some shortcomings because these techniques are not applicable for all scenarios, even within the same depositional environment. To overcome these deficiencies, geostatisticians have proposed different hybrid techniques that are more flexible and generate more realistic reservoir descriptions. Hybrid approaches combine two or more conditional simulation techniques to take advantage of the best features of each technique.

Multi-step Approach Using Object Modeling. The multi-step object modeling approach applies the marked point process multiple times in several stages. At each stage, the simulation process generates geologic objects at different scales. In the first stage, large-scale objects are simulated to fill the reservoir volume with major geologic facies or facies associations that extend throughout the simulation area. In subsequent stages, facies at smaller scales or lithotypes, which occur within major facies, are simulated within each of the

objects from the previous stage. Several stages of this process can be repeated to describe the necessary level of geologic detail. This iterative approach can be written as a script and run with Builder scripting tool (see chapter [Property Calculation Scripting](#)).

The following example considers the description of channels and shale in a meandering river environment. The following figure illustrates the major steps of the two-step approach. In the first stage, channels with 30% volume fraction are distributed with object modeling within the floodplain shale background. In the second stage, drape shale with 5% volume fraction are described within each of the channel objects.



Multi-step Hybrid Approaches Using Object Modeling and Pixel-Based Techniques. It is unrealistic for a single geostatistical technique to describe reservoir heterogeneities at all scales. Therefore, multi-step approaches, using more than one technique, were introduced to characterize distinct styles of heterogeneity at different reservoir scales. For example, in a fluvial environment, the character and degree of correlation are different for major geologic facies, such as channels, and for beds of sand grain within each channel, which control porosity and permeability. Channels are correlated across the field along the paleoflow direction, while, at smaller scales within individual channels, the arrangement of sand grain is highly influenced by the distance from the source of the river. For some reservoir characterization applications, it is important to properly describe the heterogeneities at the scale of major facies and at the smaller scales within the facies.

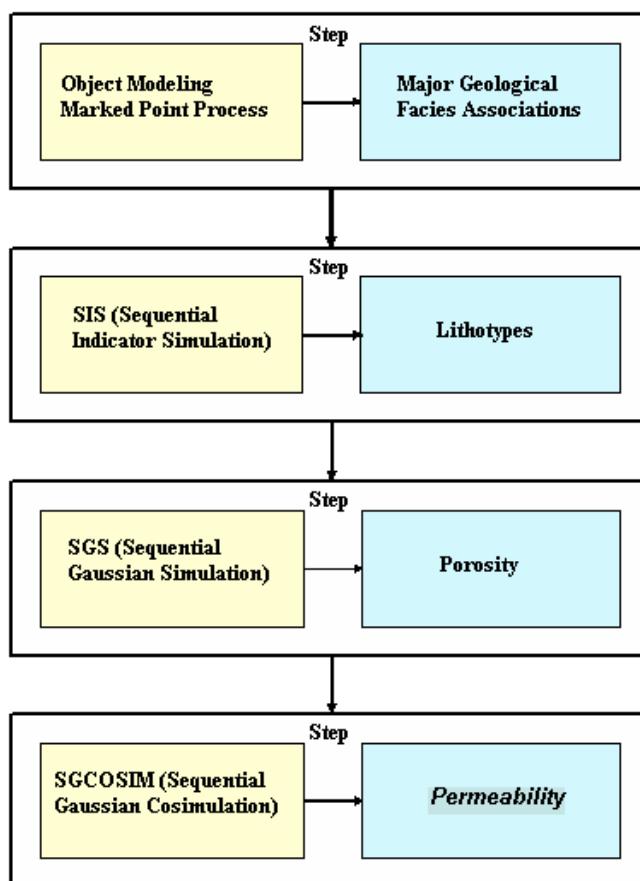
The number of steps required in a hybrid approach depends on the variation of correlation at different scales and the type of reservoir properties being described. A hybrid approach, consisting of four steps that describe major facies and reservoir properties in a fluvial depositional environment, is depicted in the following figure. Other approaches with different steps can be designed to fit other modeling purposes. Steps are executed sequentially, so the descriptions from previous steps can be used in subsequent steps. This can be automated by using Builder scripting tool. The four steps are:

Step 1: Describe major geologic facies associations with object modeling. This includes facies that extend across a significant region of the field, such as channels, mouth bars, levees and floodplain shale. The dimensions, type of objects, volume fractions, and conditioning data are specified for each facies. (Currently, Builder offers parallelepiped and channel objects).

Step 2: Describe lithotypes with Sequential Indicator Simulation (SIS). Lithotypes include beds of different lithologies, such as fine and coarse grain sandstones, mudstones and calcite concretions, which are present in different proportions within each geologic facies described in step one. The variograms, volume fraction and conditioning data are specified for each lithotype within each facies. (Currently, Builder does not include SIS).

Step 3: Describe porosity with Sequential Gaussian Simulation (SGS). The variograms, porosity mean value and conditioning data are specified for each lithotype described in step 2. This approach allows us to assign different properties to the same lithotype present in more than one facies.

Step 4: Describe permeability with Sequential Gaussian Cosimulation (SGCOSIM). The variograms, permeability mean value, conditioning data, and cross-correlation between porosity and permeability are specified for each lithotype described in step 2.



Mesh Tab and Region Tab

The **Mesh** tab is available only in 2D for the Create Map option. In 3D, the **Mesh** tab is replaced with a **Regions** tab (see [Region Selection](#) below).

Mesh Creation

There are currently two methods of specifying the mesh for map or geostatistical realization creation:

1. Specify number of mesh cells between data points. With this option, the program will calculate the average distance between data points (wells), then divide by the user input value of the “number of mesh cells between data points” to result in the mesh size. The program will center this mesh over the input data points and faults before calculation of the map will commence.
2. Specify number and size of mesh cells. You must input both the number of mesh cells and the size in both the X and Y directions. The program will center this mesh over the input data points and faults before calculation of the map will commence.

Region Selection

This option is enabled for 3D. You may specify the entire grid or select a region from a list of defined regions. You may add to the list of regions by adding new sectors in Builder.

Extend By

This option is enabled for 2D. Values entered in these boxes will result in maps that extend outwards from the limits defined by the input data points (wells). If the entered values are zero, then the resulting map will be limited to the area defined by the data points (wells). There is no limit to the maximum value for these parameters, but you should be cautious when using values outside of the data point region because of extrapolation errors.

Map Value Limits or Value Limits (3D)

If you wish to limit the map or geostatistical realization to some range, then you can input a range in these boxes. If this option is not used, the maximum and minimum values calculated from the input data points will be used.

Run and Apply Buttons

As shown below, the **Run** and **Apply** buttons are available at the bottom of the **Create Geostatistical Objects (3D)** dialog box. The **Create Geostatistical Objects** dialog box is used to create, run, and save geostatistical objects, which in turn can be selected at a later time for execution in a script (see the [Property Calculation Scripting](#) chapter). The **Run** button will become available when you start entering data and setting geostatistical parameters. Note, that you have to create a new geostatistical object before you can input data and set parameters.



When you are done entering data and setting all the geostatistical parameters, you can click **Run** to run the object or/and you can use the **Apply** button to save it. The **OK** button will also save a newly created geostatistical object or a modified one. The **OK** button will not run the geostatistical object. Any new geostatistical objects will be permanently saved when saving the dataset. Comments associated with a geostatistical object can be entered by clicking the **Comments**  button.

Variogram Matching

If the **Export Variogram Data File** option is selected on the **Methods** tab, then if you click the **Click to match variogram** button or when the map is created, the automatic variogram matching routine is started. You can take the data in this file and load it into Excel and plot the actual versus the model variogram data. The following description of variograms has been extracted from reference #2.

In practice, several difficulties are encountered in estimating the variogram. The goal is to capture the spatial relationship in the best way possible, so the original variogram equation is modified to capture the spatial relationship. This section discusses common problems encountered in variogram estimation and the proposed solutions implemented in practice. While many of these proposed solutions may not have a strong theoretical foundation, they have been proven to work in practice.

Lack of Sufficient Pairs

As defined, the variogram is a statistical property. We calculate the average squared difference between two values located a certain lag distance apart. For this average to be truly representative of a given lag distance, sufficient data pairs are needed for that lag distance. Although defining a precise number of pairs as sufficient is difficult, at least 25 to 30 pairs are needed for a reliable estimate of the variogram for a given lag distance. Obviously, the more pairs you have for a given lag distance, the more precise the estimate of the variogram. Also, it is important to remember that the minimum number of pairs is only relevant to the extent that a reliable, interpretable experimental variogram can be observed. For example, if 25 pairs do not provide an interpretable structure, alternative means of obtaining a more reliable variogram need to be explored.

Number of Pairs

A commonly used method to ensure sufficient pairs for a given lag distance is to restrict the maximum distance at which the variogram is computed. For a given region of stationarity, the possible number of pairs decreases as lag distance increases. Eventually, as the lag distance corresponds to the maximum possible distance between any two sample points, only one data pair may be available at that lag distance. Obviously, we may not be able to obtain a reliable estimate of the variogram at that lag distance. To avoid this problem, a rule of thumb used in the geostatistical literature is to use half the maximum possible distance within a region of interest as the maximum lag distance at which the variogram is calculated. For example, if

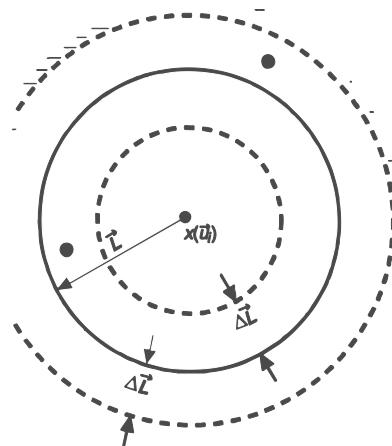
10,000 ft is the maximum distance between any two sample points within the region of interest, the variogram estimation is restricted to a maximum lag distance of 5,000 ft.

This rule of half the maximum distance also serves another important purpose. Recall the definition of the estimated variogram. Because the variogram is computed as the average of the squared difference, it is symmetric about the lag distance. That is, the following squared differences as $[x(u) - x(u + L)]^2$ or $[x(u + L) - x(u)]^2$ or $[x(u) - x(u - L)]^2$ or $[x(u - L) - x(u)]^2$ are all combined to provide a variogram estimate for lag distance L. Using the half-the-maximum-distance rule effectively ensures that representative pairs are collected on both sides of a given location.

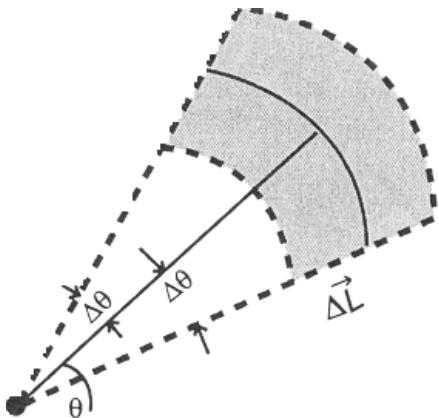
Lag Tolerance

The second alternative to ensure sufficient pairs for a given lag distance is to define the tolerance with respect to the distance as well as with respect to the direction. This is especially useful for areal data sets. Typically, examination of areally distributed data shows that the samples are not distributed at uniform intervals; therefore, a sufficient number of pairs for a precise lag distance cannot be obtained. For example, to estimate the variogram at a lag distance of 330 ft, we may have one data pair at precisely 330 ft. However, we may have three additional pairs between 310 and 330 ft and five additional pairs between 330 and 350 ft. Obviously, one pair, precisely at 330 ft, does not provide a reasonable estimate of the variogram. However, we can capture additional pairs for a better estimate of the variogram by defining a tolerance with respect to the lag distance.

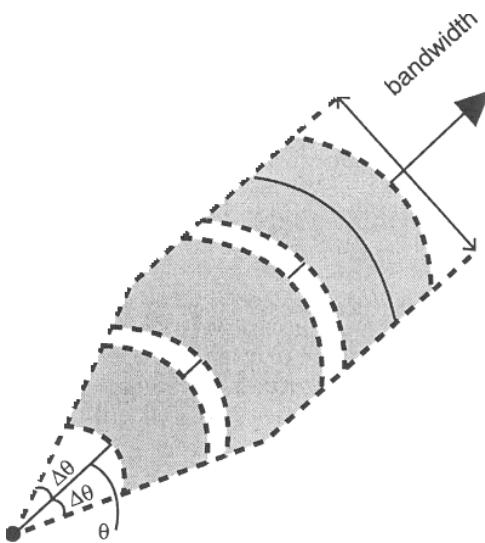
To estimate the variogram at a lag distance of L, we collect all the pairs within $L \pm \Delta L$ lag distance. The following figure illustrates tolerance with respect to distance for an isotropic search, which is concerned only with distance for all the angles. When searching for sample points away from $x(u)$ at a given distance L, we can search for all the data points within $L \pm \Delta L$ distance. In the following figure, no sample point is precisely at a distance L; however, by using a tolerance of ΔL , we can obtain two pairs at that lag distance.



In a similar fashion, the next figure shows a tolerance with respect to distance and direction in a 2D domain. A tolerance of ΔL is defined with respect to distance, and $\Delta\theta$ is defined with respect to direction, or the tolerance angle.



In addition to tolerance with respect to direction, another restriction can be placed on picking appropriate pairs. We can define a bandwidth, which is a distance perpendicular to the direction in which we are computing the variogram. This distance defines the maximum width of the area to be used to select the data pairs. As the following figure shows, at higher lag distances, once the area reaches the maximum width, the same width is used to select appropriate pairs for a given lag distance and to select the direction. In the absence of the bandwidth restriction, the width of the areal coverage increases as the lag distance increases. Bandwidth allows better control of the directionality of the estimated variogram.



In 3D data sets, if we define tolerances with respect to distance and two directions, the tolerance volume looks like a flattened cone. The bandwidth has to be defined for both directions.

The tolerance that is appropriate with respect to distance and direction is a subjective decision. The guiding principle should be to use the tolerance that is appropriate to achieve a clearly interpretable variogram structure. As Deutsch and Journel¹ state, in general, the spatial structure cannot be created by manipulating the tolerance and direction if it does not exist within the data. However, a poor choice of tolerance may mask the existing spatial structure. Therefore, a tolerance should be chosen so that the most interpretable variogram is obtained.

In practice, estimation of the variogram should begin with a small tolerance. If the variogram does not exhibit a clearly defined structure or shows too many fluctuations, the tolerance should be increased. The smallest possible tolerance at which an interpretable structure can be observed should be used to estimate the variogram. This technique applies whether the tolerance is defined with respect to distance or direction. If the tolerance is too large, the estimated variogram may not be as crisp as desired; that is, some details in structures may be lost that may be important from a spatial relationship point of view. Specifically, if the spatial structure indicates anisotropy (different spatial relationships in different directions) then a large tolerance with respect to direction may camouflage the anisotropic structure. For example, if the ratio of the range in the variogram in the x direction to the range of the variogram in the y direction (which is perpendicular to the x direction) is 5, using a tolerance of $\pm 45^\circ$ with respect to the angle, may reduce the ratio to 2.4° . It is, therefore, important to use the smallest possible tolerance to capture both the true structure and the anisotropy, if present.

From an estimation point of view, it is always preferable to start with an isotropic variogram structure before investigating the presence of anisotropy. The isotropic variogram structure assumes that the variogram structure is a function only of distance, not of direction. Estimation of the isotropic variogram allows adjustment of some of the parameters (such as tolerance with respect to distance) before proceeding with estimation of the anisotropic variogram. Also, in almost all instances, some type of spatial structure should be evident in the isotropic variogram. If no apparent structure is observed in an estimated isotropic behavior, it is difficult to imagine that some type of structure can be captured by investigating the anisotropic behavior. It is important to remember that the isotropic variogram, by considering all the pairs at a given lag distance independent of direction, uses a lot more pairs at a given lag distance compared with the anisotropic estimated variograms, which are restricted by direction as well as distance. Therefore, the isotropic variogram should provide a more stable structure than any of the anisotropic variograms. If interpretable structure cannot be captured with an isotropic variogram, it is almost impossible to observe an interpretable structure in anisotropic variograms.

Instability

The estimated variogram represents the arithmetic average of the squared differences of variable pair values at a particular lag distance. Because it uses the square of the difference, any large difference between a given pair is magnified. If pairs exhibit a large difference, the squared difference may have a significant impact on the arithmetically averaged variogram value. This effect may change the variogram value disproportionately at a particular lag distance, resulting in instability of the estimated variogram. This instability may prevent capturing the underlying variogram structure that may be present and also causes fluctuations in the estimated variogram as lag distance increases. The instability must be minimized to model the variogram. A normal

scores transform (see [Normal Score Transform](#) below) can help to reduce large fluctuations in the variogram.

The two methods commonly used to minimize fluctuations are to increase the possible number of pairs for a given lag distance. The previous section discussed the first possibility: increasing the possible number of pairs for a given lag distance by use of appropriate tolerance values with respect to the distance and the direction. That discussion showed that increasing the number of pairs for a given lag distance does improve stability of the variogram.

An alternative for improving the stability of the estimated variogram is to examine the possible pairs used for estimation of the variogram for a given lag distance. The difference between the two point values in a pair is what affects the variogram. If the difference is very large, the squared difference can have a significant impact on the estimated variogram. If we can eliminate certain "extreme" pairs that have a significant impact on the variogram computations, we may be able to obtain a better estimate of the variogram that is less affected by these extreme pairs.

Scatter plots are one way to examine these extreme pairs. Plotting one data point of a pair versus the other data point from the same pair may reveal the differences between the two data points. If the match between the two points is exact, the point falls on a line. On the basis of the scatter plots, certain pairs can be removed, and the variogram can be recomputed for a given lag distance. Alternatively, a certain percentage of the pairs showing the maximum deviation can be removed to create more uniformity in the analysis and to eliminate subjectivity in deciding which pairs should be removed. For example, for every lag distance, 10% of all pairs in the order of showing the maximum deviations can be removed. Under these circumstances, the variogram represents a truncated mean of the differences squared for a particular lag distance. Such truncated means are often used in statistics to reduce the adverse effect of erratic values (for example, in figure skating in the Olympic games, the two extreme scores are removed from the final tally). This procedure also has the advantage of being objective.

Trend

The variogram should level off at the data variance for large lag distances. Sometimes it keeps increasing a way above the data variance line. This is often symptomatic of a trend in the data and should be investigated. Note that trends can be present in the local average of the data and/or in their local variance. If a trend is present, it should be removed before proceeding with estimation or simulation. If the trend is not too pronounced, it might still be possible to use the raw data if the search window (see [Search Radius](#) below) is small enough that only the beginning of the variogram will be used in any of the Kriging systems.

Search Radius

When a mesh cell value is to be calculated using either the Kriging or Gaussian Geostatistical methods, a search is done on nearby mesh cells to determine if the cell already has a value assigned to it (mesh cells that contain wells always have values assigned to them). If the neighbor mesh cell has a value, then this cell is used for the conditioning of the mesh cell to be calculated. The size of the search neighborhood is determined by the user input search radius in each direction, and is limited by the **Maximum Nearby Cells For Search**. Generally, if the search neighborhood is small, the resulting map or geostatistical realization will be more disconnected than if the search neighborhood is large because the mesh values will be more

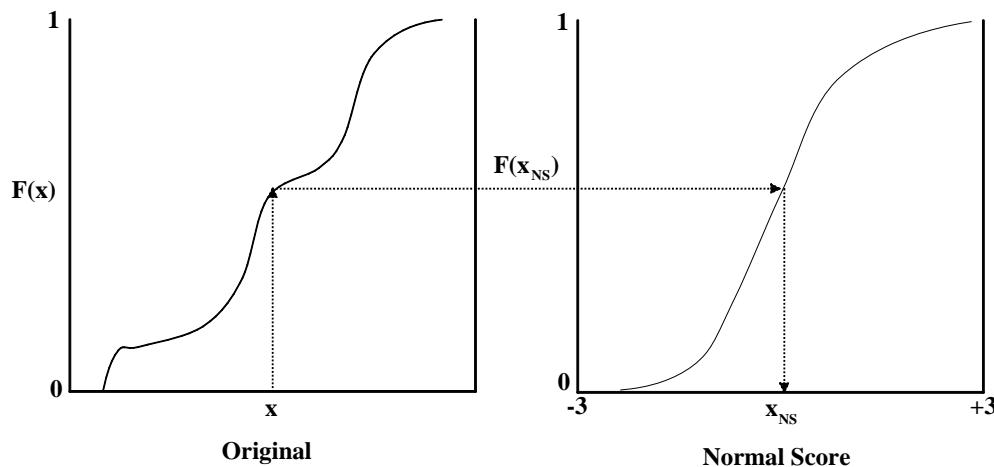
independent of the neighboring mesh values. If the search radius is set to be very large, the problem may require a large amount of memory and long execution times to establish the search neighborhood for each mesh cell. If **Auto** is checked next to the input for maximum nearby cells for search, then the program will estimate the number of nearby cells using the input search radius and the current cell size.

After the search neighborhood for a particular mesh cell is calculated, the program will sort the neighbors by variogram closeness (covariance) to this mesh cell. When the search neighborhood is actually used for assigning values, only the first of the **Number of Nearest Points** will be used for the calculations. If the number of nearest points is large, the resulting realization will appear more averaged than if this value is small. See the [Weighting Examples](#) section for an illustration of this effect.

If **Geological Units** are defined (see [Defining/Editing Geological Units](#) in the Reservoir Description chapter), the search will be limited within the geological unit of the estimated/simulated grid block.

Normal Score Transform

The normal score transform allows transformation of sample data into equivalent data that follows a normal (Gaussian) distribution. The following figure shows this transformation schematically.

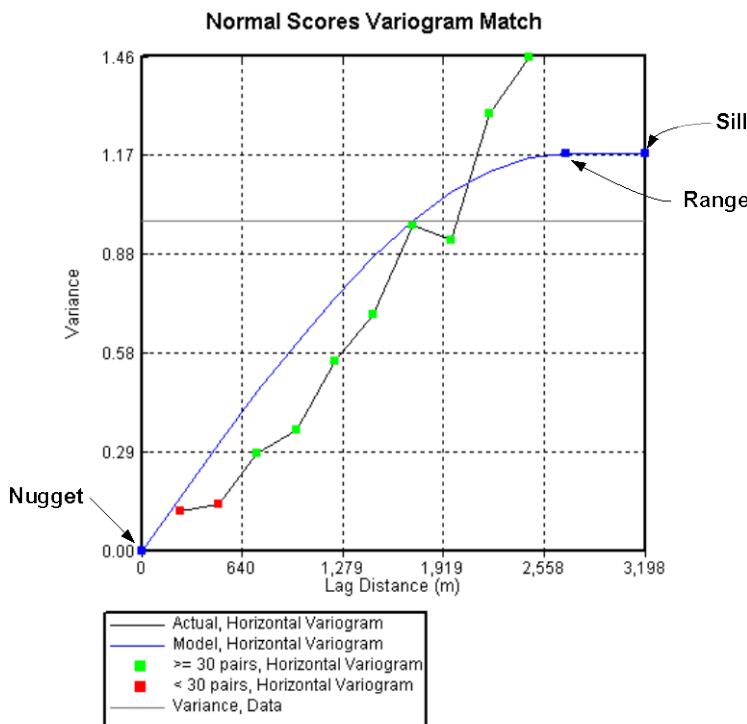


A cumulative distribution function can be constructed for any sample data. The cumulative distribution function has a minimum value of zero and a maximum value of one. For the normal distribution function, the cumulative distribution function has the same range. Each sample value is associated with a corresponding cumulative distribution function. For the same value of the cumulative distribution function, we can define an equivalent normal-score-transformed value. Typically, the normal-score-transformed values have a mean of zero and a variance of one; therefore, they range from -3 to +3. Using such a transform for every sample value allows transformation of all the sample values to a corresponding normal score. There is a one-to-one correspondence between the sample value and the transformed value; that is, if we know the transformed value, we can back-calculate the corresponding sample value.

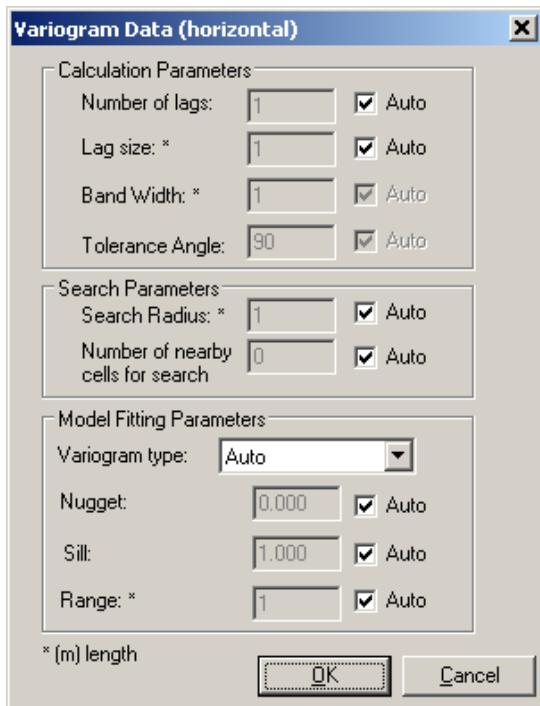
With transformation of the sample data to a normal score, the variability of the data set is restricted to -3 to +3. Effectively, there are no order-of-magnitude variations within the transformed data set. The effect of extreme data on the variogram should be minimized with the transformation. Another advantage of the normal-score transform is that certain estimation techniques work better with normal-score-transformed data. After obtaining the estimation with the transformed data, we can back transform the data to original variable values.

Adjusting Variogram Parameters

Where the method uses a variogram, you can manually adjust the variogram, as illustrated in the following example:



You adjust the variogram parameters either directly on the graph or through the **Variogram Data** dialog box which you can open by clicking the **Click to edit variogram data** button on the **Methods** tab. There is a dialog box for the horizontal variogram and one for the vertical variogram:

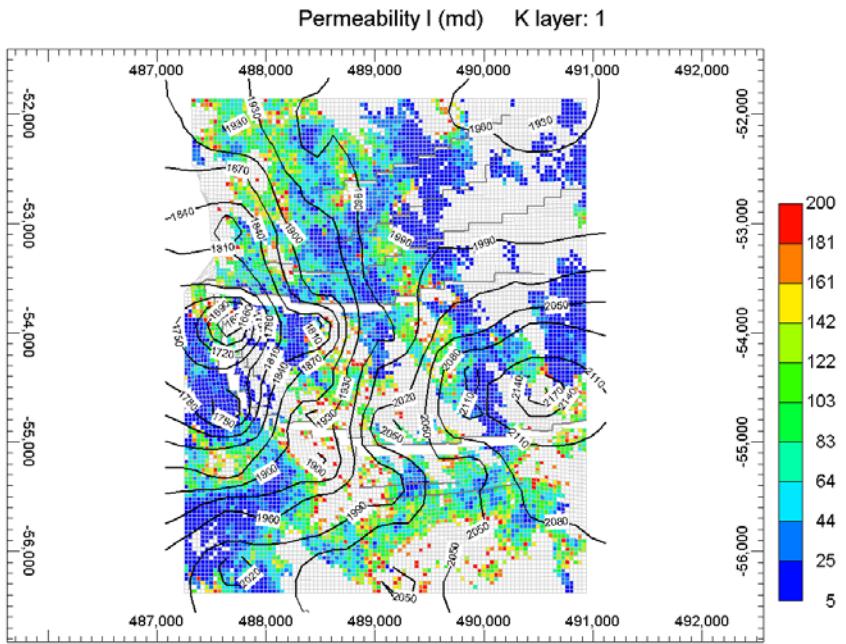


To adjust a variogram parameter, you must first cancel **Auto** beside the parameter name. You can also adjust the model fitting parameters directly through the variogram curve, by clicking the **Click to match variogram** button on the **Methods** tab, then dragging the parameters for the desired fit using your mouse. The values in the **Variogram Data** dialog box will be adjusted appropriately.

Weighting Examples

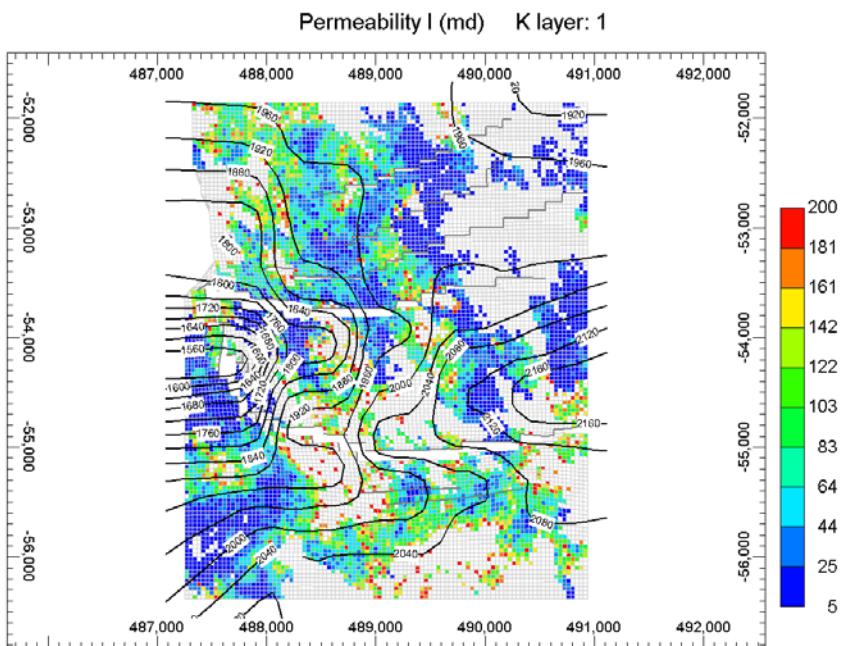
Inverse Distance Method

The following is an example of a map created with the inverse distance method. Note that the map has many circles (or bulls eyes) that indicate an area that has a value much different from its neighbors. This behavior is typical of the inverse distance method, and is frequently the reason why other methods may be preferable.



Trend Estimation

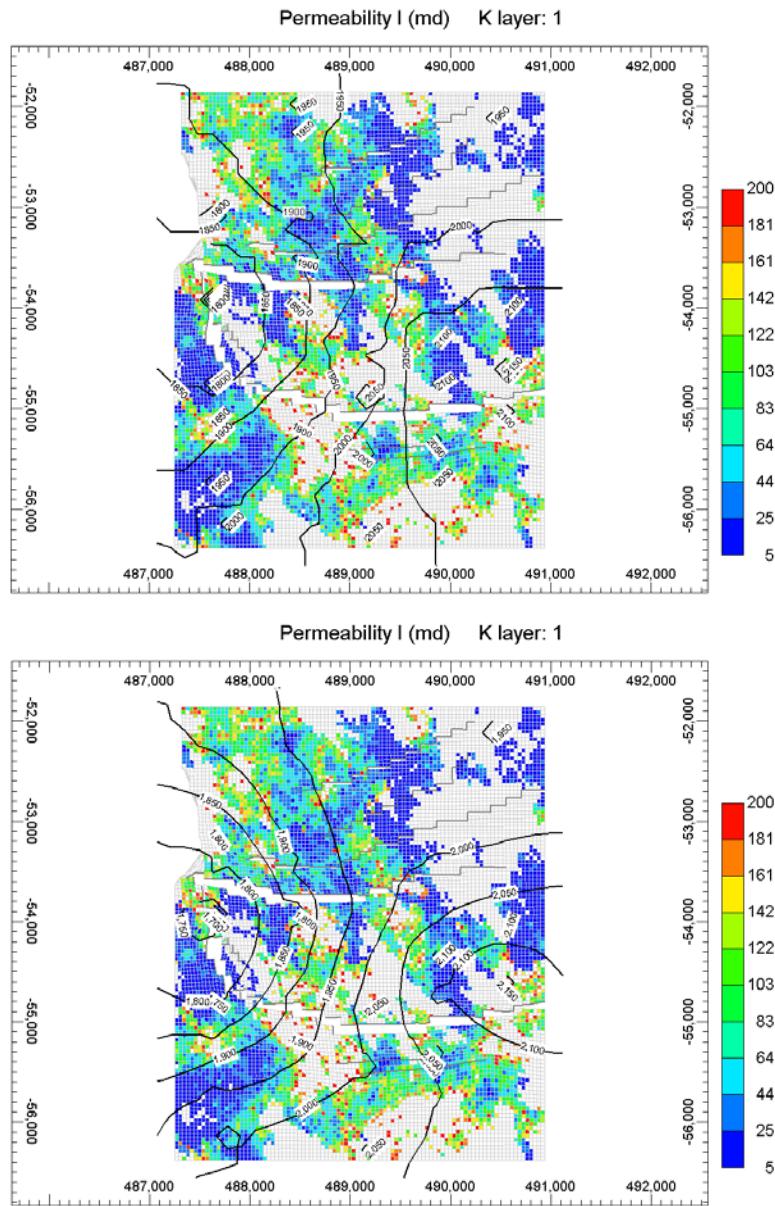
An example of a map created using the trend estimation method is shown below:



As shown in the above map, the bulls eyes evident in the map created using the **Inverse Distance** method have been eliminated.

Variogram Range (Kriging Method)

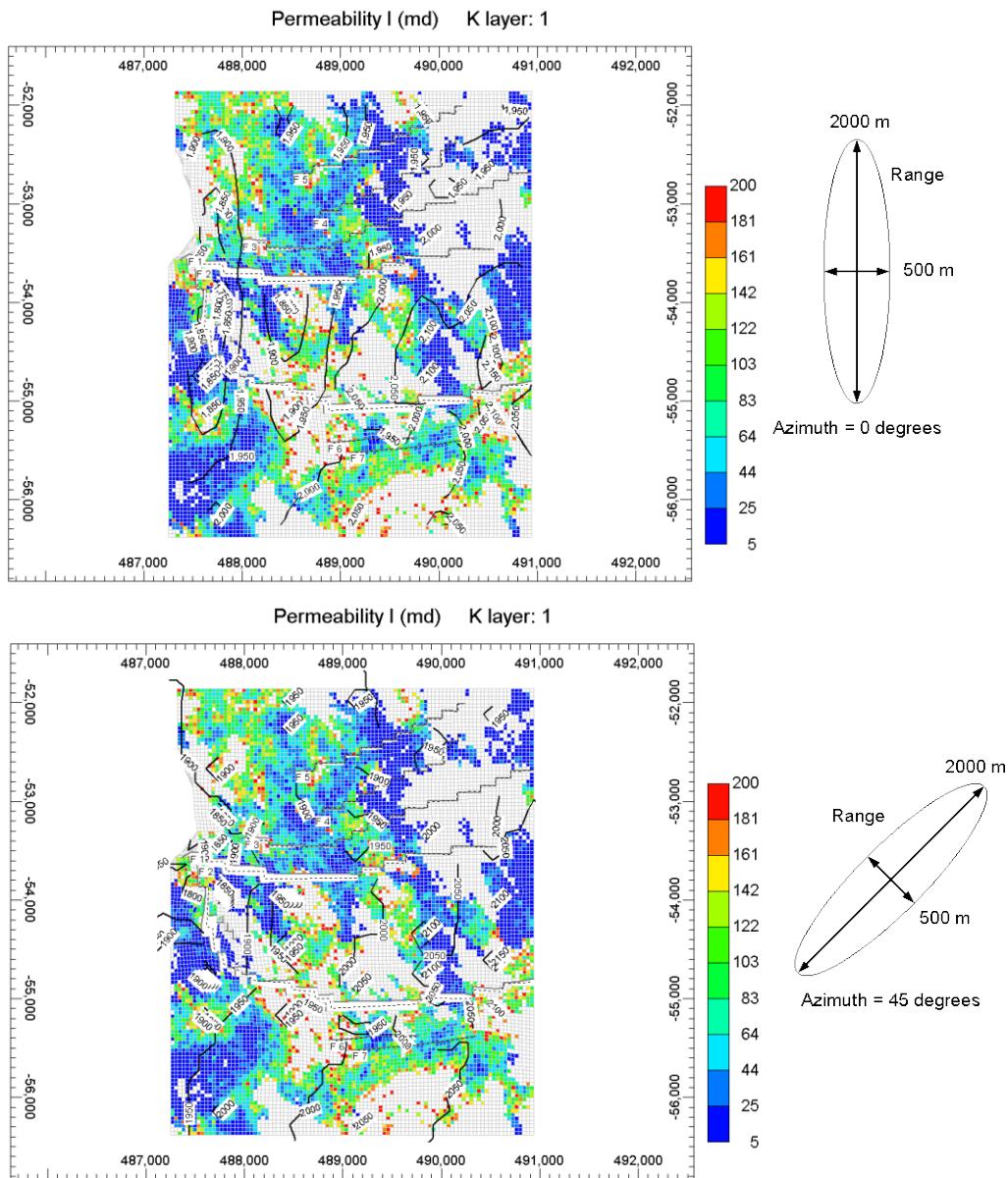
The following maps were created with the same data except the variogram range was set to 500 m for the top map then to 2000 m for the bottom map.



By increasing the range, more points are used in the calculation. Increasing the range increases the relative weight assigned to points that are further out.

Variogram Anisotropy (Kriging Method)

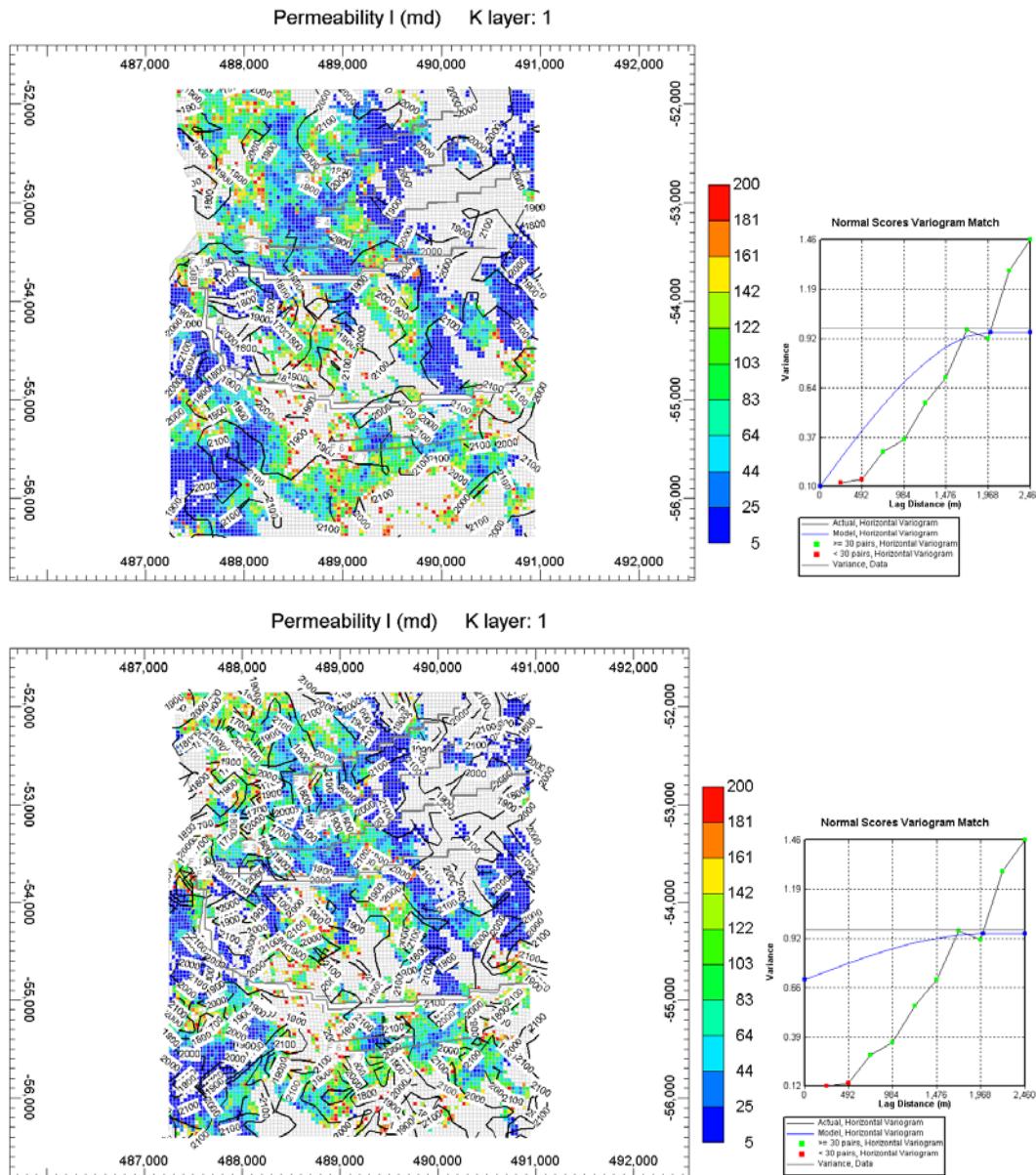
If the variogram is changed to be anisotropic (**Bi-Directional** option on the **Methods** tab) and the ranges are 2000 m and 500 m in the principal and minor directions respectively, the maps are shown for azimuth angle = 0.0 (top) and for azimuth angle = 45 (bottom):



The above maps show the effect of forcing a stronger relationship with points in the azimuthal direction. In general, this method is appropriate when the geologic structure is directional.

Variogram Nugget (Gaussian Geostatistical Method)

As the nugget value increases greater than zero, the amount of short-wavelength randomness increases. Examples of geostatistical realizations with nuggets of 0.1 (top) and 0.7 (bottom) are shown below (in both cases, the sill is 0.95 and the range is 2000 m):



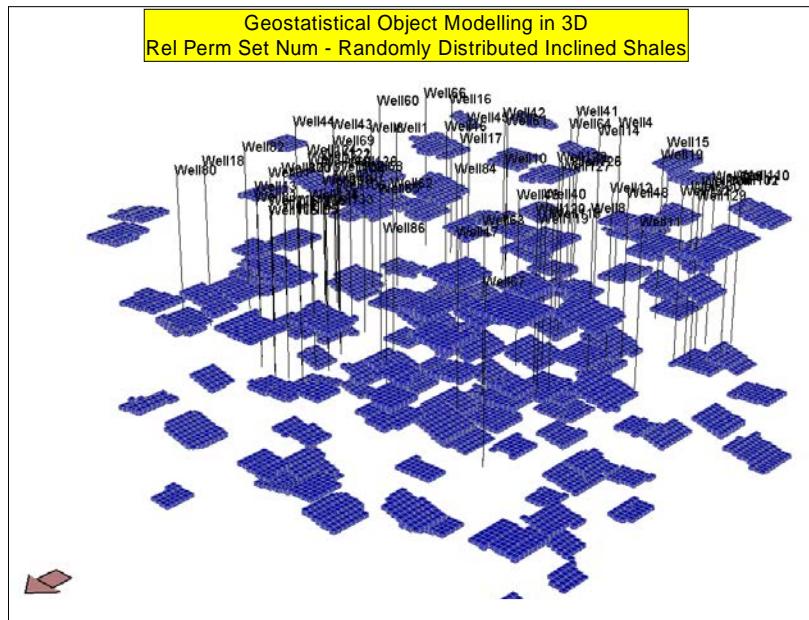
In the above example, increasing the size of the nugget has increased the “noisiness” of the contour map.

Search Range (Gaussian Geostatistical Method)

With the Gaussian Geostatistical method, the size of the search neighborhood can be increased by rejecting data outside a radius (using) or by limiting the number of points used in the calculation (using **Number of Nearest Points**).

Object Modelling – Inclined Shales

The following was created using the **Object Modelling** method and the **Parallelepiped** object type.



References

1. Deutsch, C.V. and Journel, A.G.: "Geostatistical Software Library and Users Guide", Oxford U. Press, New York City, (1992) Chap 3.
2. Mohan Kelkar and Godofredo Perez, "Applied Geostatistics for Reservoir Characterization", Society of Petroleum Engineers Inc., Richardson, Texas 2002.

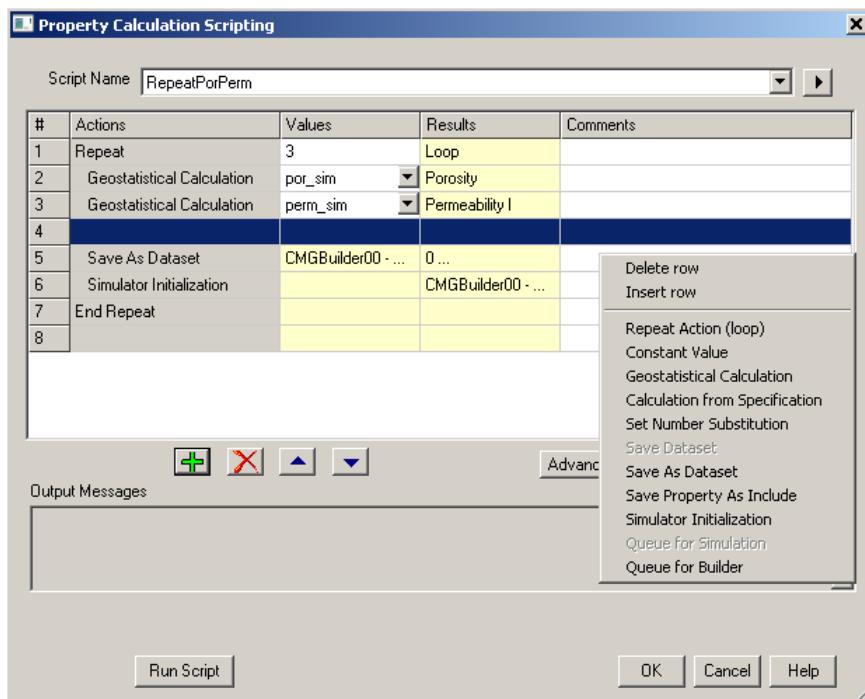
Property Calculation Scripting

Overview

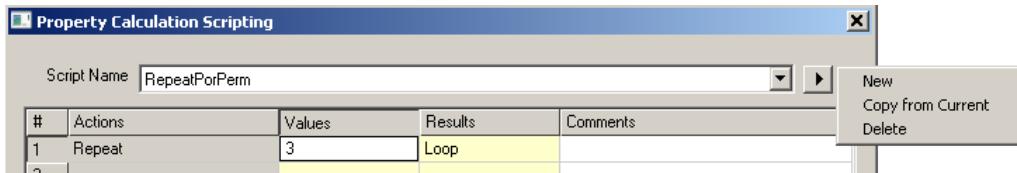
This feature supports the automation of workflows and iterative calculations. It involves the selection of actions, and sequencing them for execution. The actions are a subset of the calculation methods available in Builder. Scripts can be used for many purposes. An example is the case where you need to:

1. Run many geostatistical realizations of porosity values.
2. Use the multiple versions of the porosity values to generate multiple versions of permeability values.
3. Save and initialize all the resulting datasets with the flow simulator. Multiple versions of a reservoir dataset can then be used for uncertainty analysis.

The **Property Calculation Scripting** dialog box is available through **File | Scripting**:



The **Script Name** box presents a list of existing script calculation objects. To the right of the **Script Name** box, the button opens a list of options for creating new script objects or deleting existing ones. If a script is selected, its actions will appear in the table, one action per row. If no scripts exist, you will need to create one before you will be able to insert actions.



Actions can be selected from a context menu displayed when you right-click on the rows of the table. The context menu of actions will be active only when an empty row is selected.

You can add, delete and move actions using the buttons at the bottom of the table, as follows:

- Adds a row below the selected row
- Deletes the selected row
- Swaps the selected row with the row above it
- Swaps the selected row with the row below it

To replace an action, you need to delete it first.

A row can be selected by clicking one of its cells or selecting them all by right-clicking in any of its cell. A selected row will be highlighted in blue.

The table is divided into five columns:

- | | |
|-----------------|---|
| # | Automatic number assigned to the rows. Each row corresponds to a script action. |
| Actions | Identifies the type of action. |
| Values | Depends on the type of action. Further details are given below. |
| Results | Result of the action. More details are given below. |
| Comments | Comments can be entered by selecting this cell of an action. |

Actions can be edited by double-clicking in the **Actions** cell (second column). The sequence of actions in the script is important because some calculations may depend on the results of previous actions.

The following sections describe the actions available for scripting.

Repeat and End Repeat Actions

The **Repeat** and the **End Repeat** actions are created in pairs by selecting **Repeat Action (loop)** in the context menu of actions. Deleting one of them will automatically delete the other but not the actions in between. A **Repeat Action** is a loop for repetitive calculations. The number of loop iterations is given by the action's Value. The default value is one iteration. This can be changed by selecting the **Value** cell of the **Repeat** action. Any other actions added between a pair of **Repeat** and **End Repeat** will be part of that loop. Indentation is automatically added for improved readability of the script. Multiple levels of Repeat loops can be used for the creation of embedded loops.

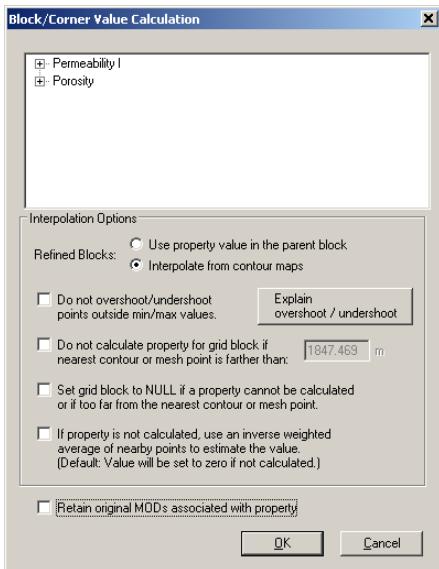
Geostatistical Calculation Action

Selecting a **Geostatistical Calculation** action will display a selection list of geostatistical objects in the **Values** cell of its corresponding row. The drop-down list contains all the currently defined geostatistical objects. These existing objects would have been defined using the **Create Geostatistical Objects** dialog box (see [Creating Maps and Geostatistical Property Calculations](#) section). If no such geostatistical objects are currently defined, the list will be empty which will invalidate the script. The list shows *<not selected>* by default. For a script to be valid and saved, you need to select a geostatistical object for each geostatistical calculation action. When a geostatistical object is selected, the **Results** cell will display the Builder property to which the calculation applies. This property is known by the geostatistical object and was chosen at the time of its creation.

Calculation from Specification Action

Note: For information about the use of the **Retain original MODs associated with property** check box, refer to [Property Modifications \(MODs\)](#).

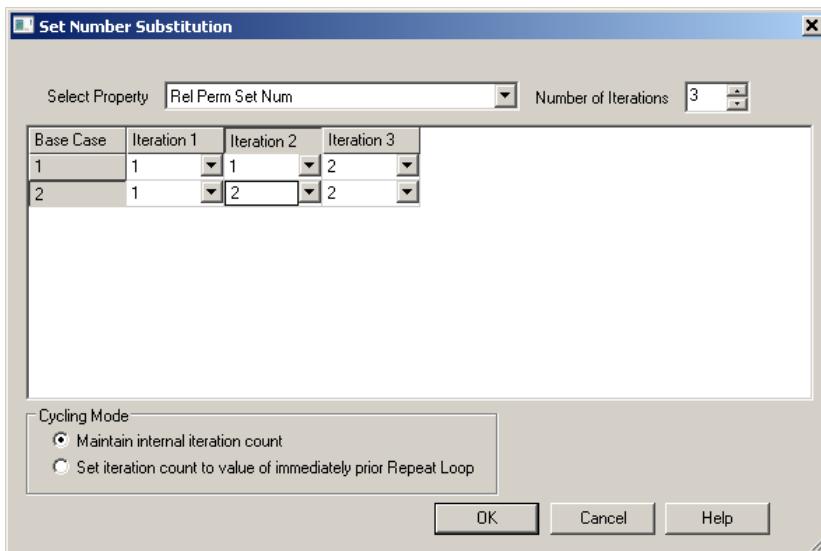
This action reproduces the functions available with the **Calculate Property** button found on Builder main view (see [Performing Interpolations and Calculations](#) in the **Reservoir Description** chapter for further information).



A series of calculations can be selected with this action. The list of properties that have been selected will appear in the Results cell for this action.

Set Number Substitution Action

The **Set Number Substitution** action is used to replace or substitute the index number of a property of type *integer*. Therefore, only properties such as Relative Permeability Set Number, PVT, and Rock Type can be selected for this action. The selectable properties are limited to integer properties. In addition, property sets need to already have been defined in the dataset, and the property needs to exist on the simulation grid.

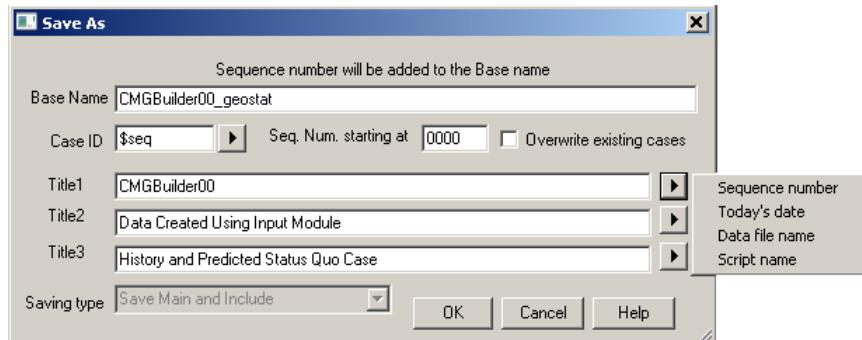


Each time an integer property is selected, the dialog box will update the spread sheet control with all integer values of this property currently in use on the simulator grid. These values will be listed in the **Base Case** column. You can define cases for substituting or replacing the **Base Case**. The number of substitution cases is controlled by setting **Number of Iterations**. For each iteration column, the cells will be filled with drop-down lists, allowing the selection of a different integer value to replace the corresponding **Base Case** value. Each drop-down list is populated with all set numbers existing in the dataset for the selected property. For each of the iterations, you assign a new integer value to change the set number of the **Base Case**. Each time the **Set Number Substitution** action is executed, its internal iteration count number is incremented by one and the corresponding substitution is applied to the property on the simulation grid. The internal iteration counter will be reset to 1 each time the number of executions exceeds the number of iteration cases. Alternatively, the internal counter can be controlled by the immediate prior **Repeat** loop in the script. It should be noted that the substitution always applies to the **Base Case** and not to the previous substitution.

When a **Set Number Substitution** action is defined, the **Values** cell will display the number of iteration cases and the **Results** cell will display the property to which it applies.

Save Dataset Action

The **Save Dataset** action reproduces some of the functionality found in the **File | Save As** menu option. It is used to save the dataset in its current state in the script.



A **Base Name** must be chosen, the default being the name of the currently open dataset. The **Base Name** will be appended with a sequence number which will be automatically incremented by one each time the script executes the **Save As** action. The default value is 0 if no previously saved dataset exists with the same base name and sequence number. Otherwise, the default value will be one plus the largest sequence number found in the current working directory. If the **Overwrite existing cases** check box is selected, Builder will start the sequence at the number provided, and will overwrite any existing datasets.

Macros (identified with the \$ sign) can be used to add the sequence number, the date, the file name, and the script name in the dataset case ID, title1, title2, and title3.

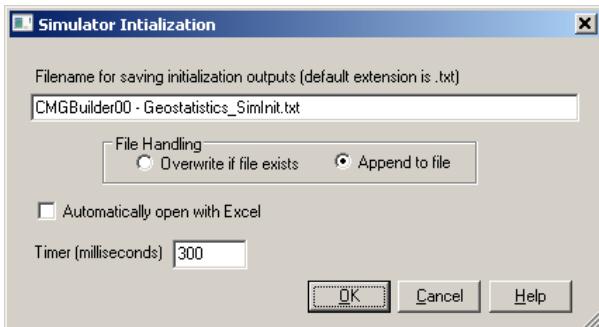
You can choose how Builder saves the dataset during this action:

- **Save Main and Include:** All files, including any include files in the dataset, are saved to disk.
- **Save Main Only:** Only the main dataset is saved, and any include files are left in their original state.
- **Save Include Only:** Only include files are saved, and the main dataset is left in its original state.

When a **Save As** action is defined, the **Values** cell in the script dialog will display the dataset base name and the **Results** cell will display the first sequence number that will be used. Generally a **Save As** action will be placed at the end of the innermost loop and/or before a simulator action.

Simulator Initialization Action

The **Simulator Initialization** action is used to run the simulator one time step and extract initial values such as **Total Oil in Place** and **Volumetrics**.

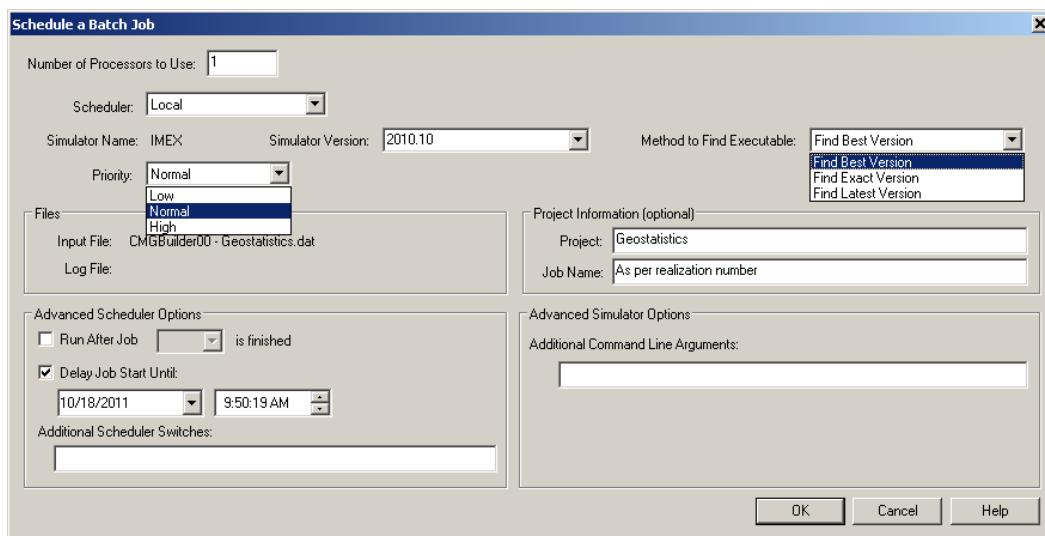


A filename must be given to save the initial values output by the simulator. This file will have a .txt extension and will be formatted (tab separated) for easy loading in a spreadsheet application such as Excel. An option is provided to automatically open the file with Excel when the script has finished with this action. If the file already exists, the new values can be appended to the file, or the file can be overwritten. This action will wait for the simulator to finish (one time step) before it retrieves the initialization outputs. **Timer (milliseconds)** is used to force Builder to periodically go into short sleeping periods to give the simulator the most CPU cycles to complete its initialization. Note that a running script can be cancelled by pressing the **ESC** key.

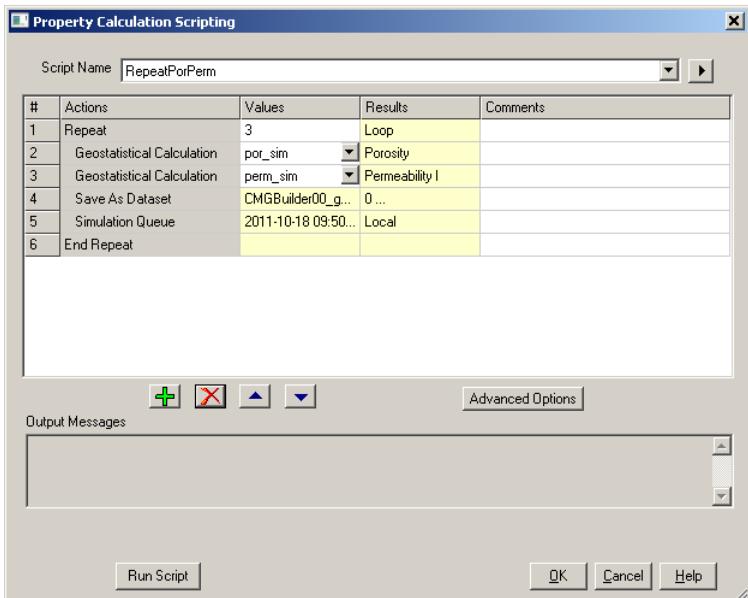
When this action is defined, the **Results** cell will display the path and filename to which the simulator outputs will be saved. If simulator initialization is performed within a loop, the multiple outputs will allow you to assess uncertainty on volumetrics and material in place by computing statistics such as mean and variance.

Queue for Simulation Action

The **Queue for Simulation** action reproduces the functionalities found in the **Schedule/Run a Simulation Job** dialog box that is displayed when you submit a dataset file to a simulator from Launcher. Refer to **Scheduling a Simulation Job** in the *Launcher User's Guide* for further information. This script action is used to schedule a previously saved dataset for execution in batch mode by the appropriate simulator. Therefore, a **Queue for Simulation** action should be inserted in the script just after a **Save Dataset** action.



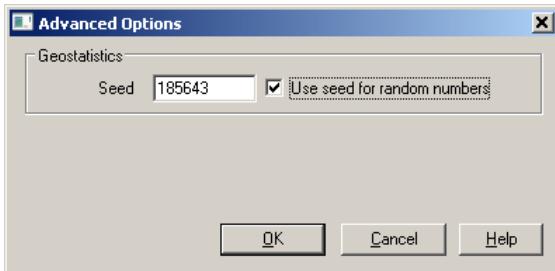
The dataset can be submitted for running with the simulator on your local computer or on a remote computer that is part of a computer farm. This second option is available only if third-party software for sharing computer resources, such as Platform LSF, is installed. Scheduling a job is accomplished by setting a date and time to start, and a priority for execution.



When a **Queue for Simulation** action is defined, the **Values** cell will display the date/time at which the first saved dataset will be executed. Subsequent saved datasets that have been generated by the script will be put in the execution queue in a sequential manner, to be run after the previously scheduled job. The **Results** cell will indicate if the jobs will be run locally or remotely.

Advanced Options

The **Advanced Options** dialog box can be used to set a unique random seed for all geostatistical simulations.



Using a predefined seed for generating random numbers means you can produce the same results when running the same script multiple times. You could delete all multiple geostatistical realizations and keep only the seed, knowing that the geostatistical realizations could be reproduced at a later time. The seed is saved with the script.

If you select **Use seed for random numbers**, the pseudorandom numbers used in the geostatistical simulations will be generated using the GSLib routine (Deutsch, C.V. and Journel, A.G.: “Geostatistical Software Library and Users Guide”, Oxford U. Press, New York City, 1992); otherwise, the operating system pseudorandom number generator is used.

Running the Script

The property calculation script is run by clicking **Run Script**. While the script is running, progress messages are sent to the **Output Messages** window to inform you about the sequence of actions being executed, and their current status. A timer bar will also appear below the message window. The running script can be aborted by pressing the **ESC** key.

Builder can be loaded to run a script automatically in the background from the command line. When started this way, Builder will automatically exit when the script is completed. This may be convenient if you need to do a lot of repetitive Builder actions on a remote computer, for example.

To start Builder in this manner, use the *-script* command line switch to indicate which script Builder should run when started. Multiple script names may be provided after the *-script* switch if they are separated by a + character.

You must also specify the *-g* switch when starting Builder. The *-f* switch is required to instruct Builder which dataset to load. Other optional switches may be provided if required.

For example, the following command line loads Builder to run a script called *PorSim* in the dataset directory:

```
C:\path to resgb\resgb.exe -g -f c:\data\porsim\dataset.dat -script PorSim
```


Setting Up Datasets for CMOST

Overview

This feature is designed to help you create a master dataset for CMOST or to edit the Parameters section of an existing CMOST task data file. In order to run CMOST tasks, the CMOST task data file and CMOST master dataset are needed. You must create the CMOST task file by using CMOST studio first, and then they can make the CMOST master dataset manually or create it by using this feature in Builder.

The CMOST parameter association is the link between CMOST parameters and the locations of data values that are associated with keywords in the base dataset. According to these CMOST Parameter Associations, Builder can write out the CMOST strings into the master dataset that CMOST can read and decode.

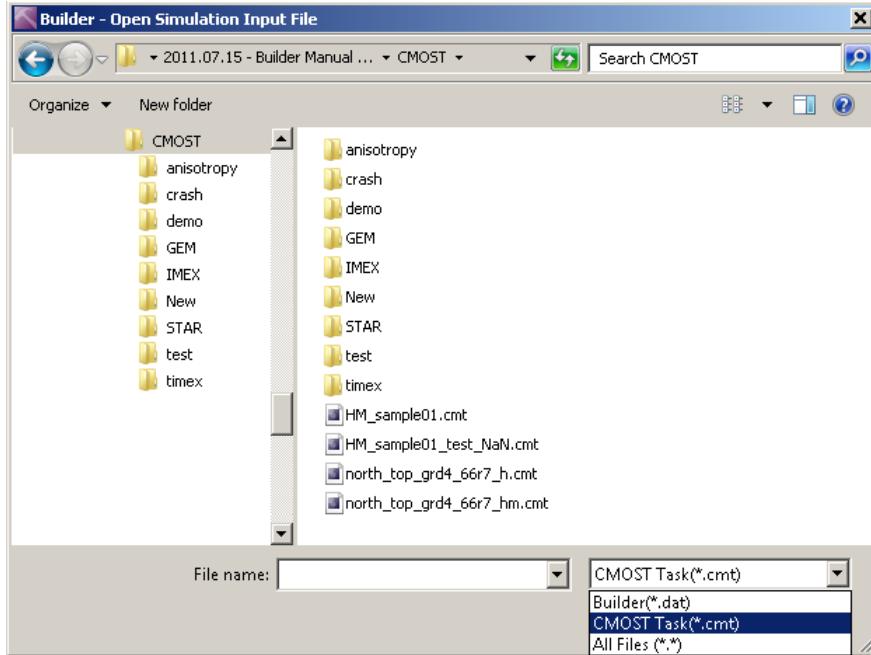
The following tasks can be performed when using this feature.

- Open CMOST task file (filename.cmt) in Builder
- Open CMOST Parameters Selection window
- Add CMOST parameters
- Edit CMOST parameters
- Delete CMOST parameters
- Add CMOST parameter associations
- Delete CMOST parameter associations
- Switch CMOST parameter associations
- Export CMOST master dataset
- Save CMOST task file

Open CMOST Task File

In order to export the CMOST master data file or edit a CMOST task file, an existing CMOST task file has to be opened first. The steps to do this are as follows:

1. Start Builder.
2. From the **File** menu, select **Open** or click the **Open**  icon on the tool bar. The **Builder – Open Simulation Input File** is displayed:



3. Select **CMOST Files(*.cmt)** in the **File Type** box.
4. Select the CMOST task file.
5. Click **Open**.

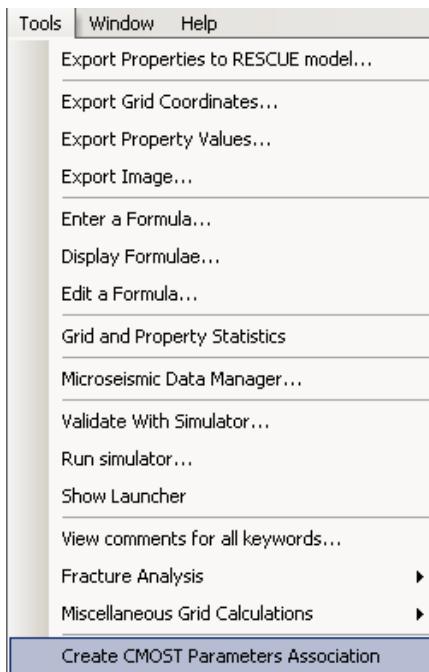
If there is no base dataset file name in the task file, Builder will ask you to choose a dataset file.

Note: If a CMOST task file does not exist, use CMOST Studio to create one and then perform the above steps again.

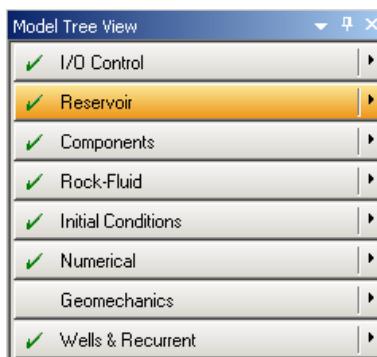
Open CMOST Parameters Selection Window

To add new parameters, edit parameters, delete parameters and export the information into a master dataset, the **CMOST Parameters Selection** dialog box must be used. This dialog box will automatically appear if a CMOST task file is opened through Builder.

To manually open the **CMOST Parameter Selection** dialog box, select **Create CMOST Parameter Association** in the Tools menu:

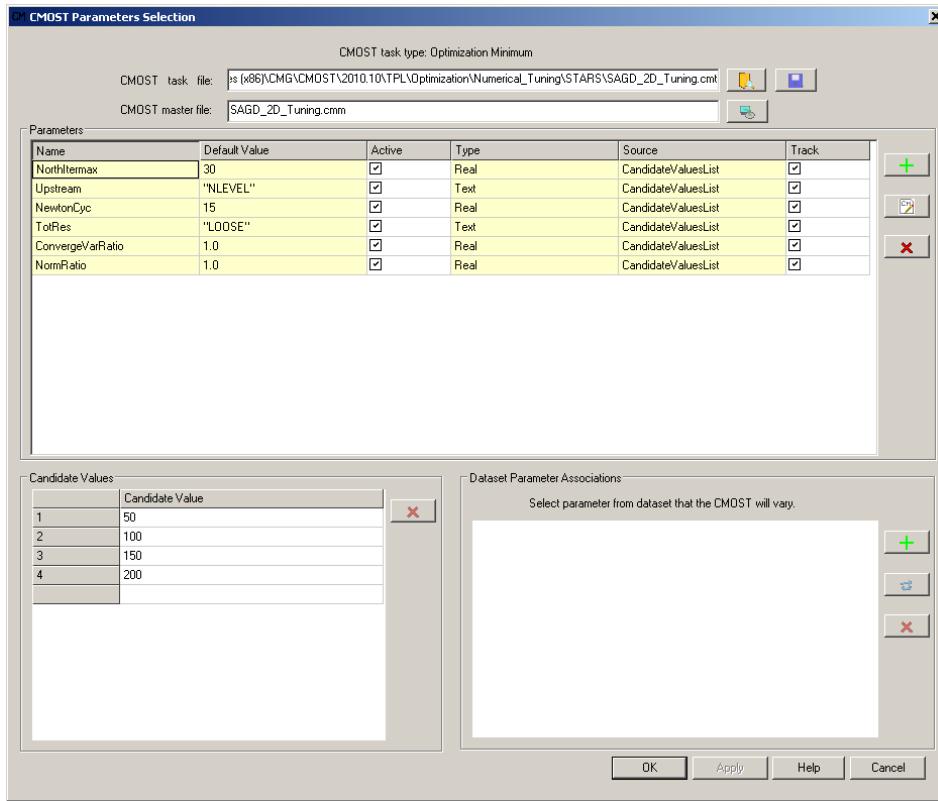


Note: If the **Create CMOST Parameter Association** menu item is not enabled, click **Reservoir** in the **Model Tree View**. This item will not be enabled unless a task file is open in Builder.



Add CMOST Parameters

After you have opened the CMOST task file in Builder, the following **CMOST Parameters Selection** dialog box will be displayed:

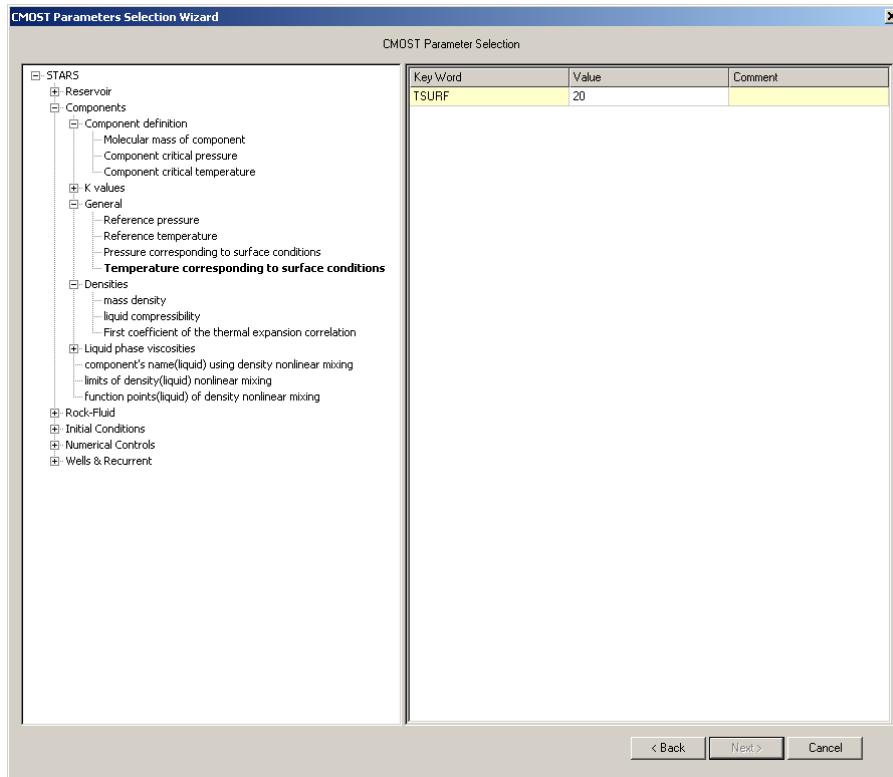


1. In the **Parameters** area, click the **Add Parameter** button. The **CMOST Parameters Selection Wizard** dialog box is displayed:



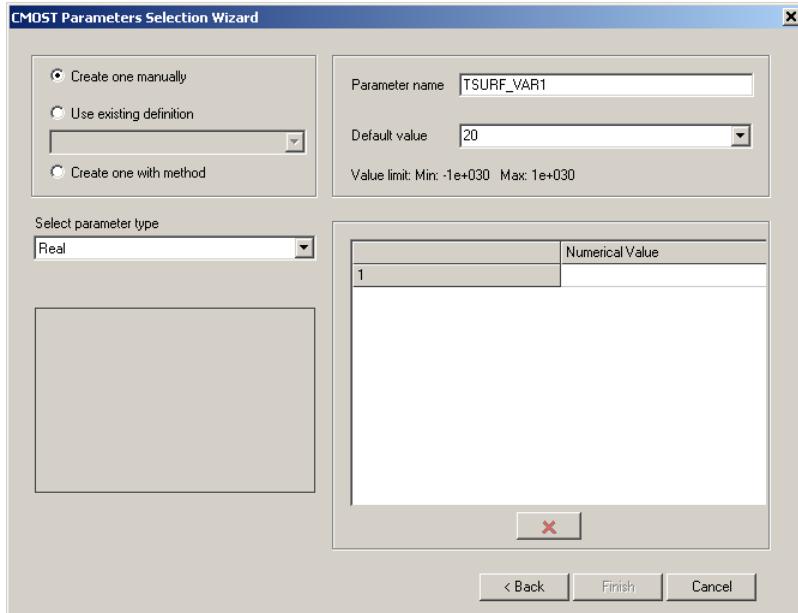
2. Select the **Select parameter from dataset** option to add a CMOST parameter and an association.

3. Select **Create a working parameter** to add a CMOST parameter in the same way as the CMOST Studio.
4. Click **Next**. The next dialog box in the **CMOST Parameters Selection Wizard** dialog box is displayed:

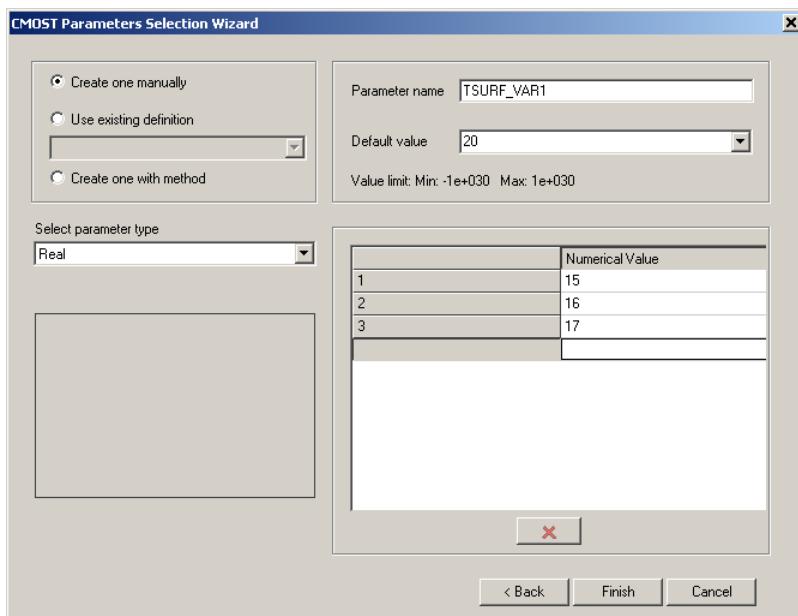


5. Select a parameter from the tree view on the left. The associated data will be displayed in the right pane.
6. Click the data cell that contains the keyword value for the desired parameter. This will enable the **Next** button.

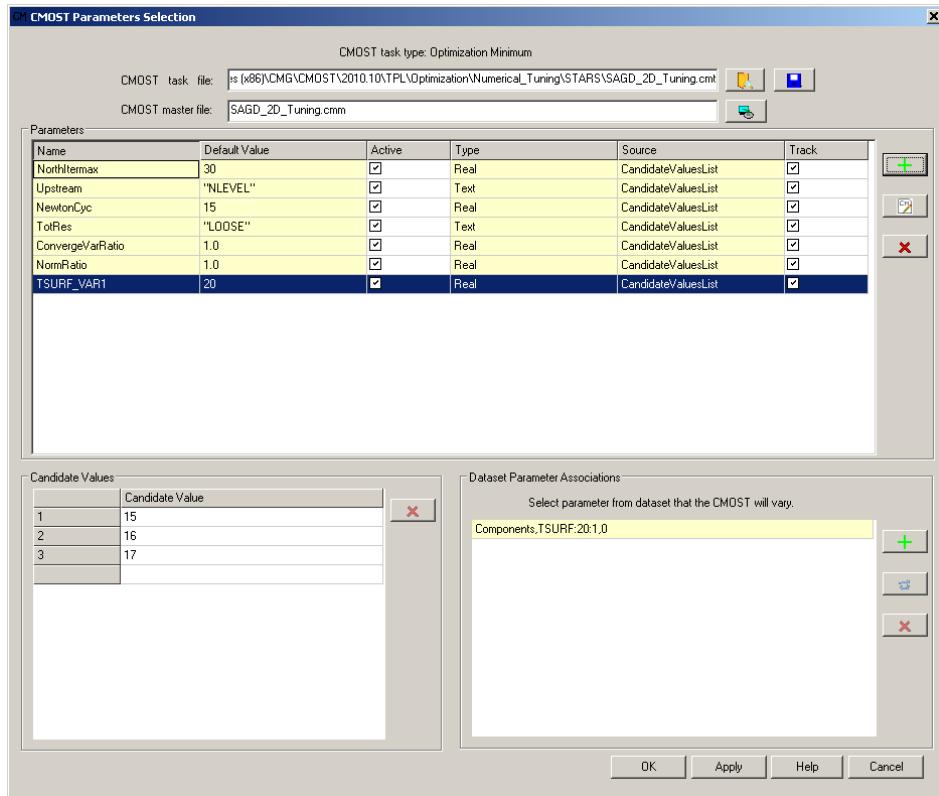
7. Click **Next**. The next dialog box in the **CMOST Parameters Selection Wizard** dialog box is displayed:



8. As required, in the upper right, select one of **Create one manually**, **Use existing definition**, or **Create one with method**.
9. If you select **Create one manually**, values can be entered in the grid on the right, as shown in the following example, where we have entered 15, 16 and 17:



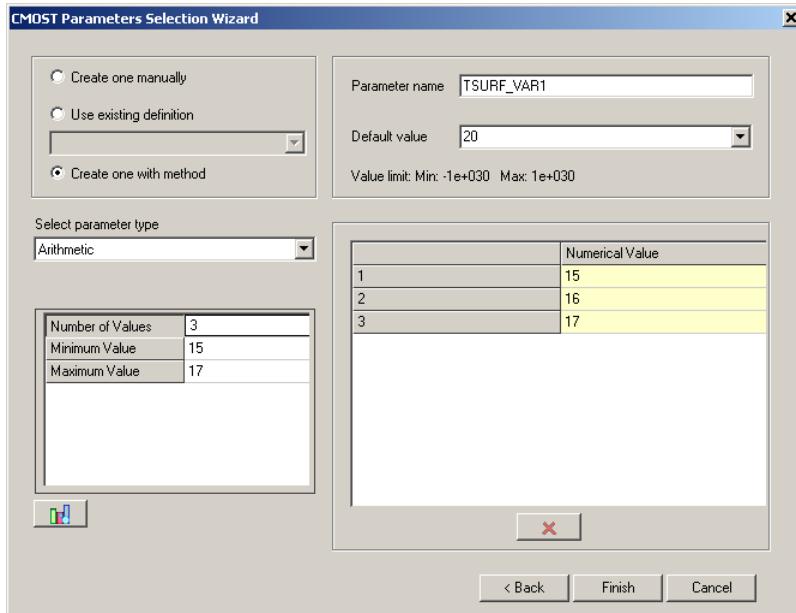
Click **Finish**. The CMOST parameter and CMOST parameter association will be added, as shown in the following example:



The general information for the CMOST parameter is displayed in the top grid. The candidate values of the CMOST parameter is shown on the left bottom grid. The CMOST parameter association is shown on the right bottom grid.

- If you select **Use existing definition**, a CMOST parameter association will be added to an existing CMOST parameter (refer to [Add CMOST Parameter Associations](#) for additional information).
- If **Create one with method** is selected, the candidate values can be created by one of the following methods (for further information, refer to the *CMOST Studio User's Guide*).
 - Arithmetic
 - Geometric
 - Fixed
 - Uniform
 - Triangle
 - Normal
 - Lognormal
 - Universal

Select the parameter type, input the values in the left grid, and then click the **Generate**  button. The **Finish** button will be enabled:

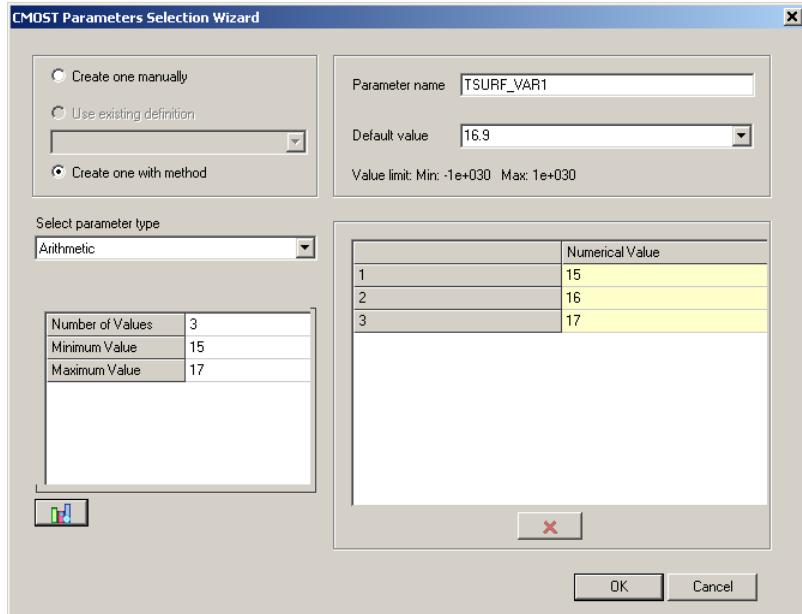


Click the **Finish** button to add the CMOST parameter and CMOST parameter association.

Edit CMOST Parameters

On the **CMOST Parameters Selection Wizard** dialog box:

1. Select the CMOST parameter on the left top grid.
2. Click the **Edit Parameter**  button.



3. The following values can be changed:

- **CMOST Parameter name**
- **Default value**
- Candidate values. There are three ways to change the candidate values:
 - Change the value or add new values in the above dialog box.
 - Do the same in the CMOST Parameters Selection dialog box.
 - Change values by using a method.

4. After you have made the changes, click **OK**.

Delete CMOST Parameters

You can delete one or more CMOST parameters at the same time. All CMOST associations belonging to the deleted parameters will also be deleted.

On the **CMOST Parameter Selection** dialog box:

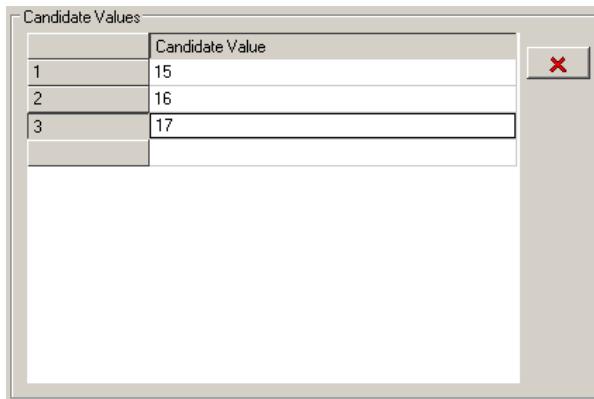
1. Select the CMOST parameter or parameters you want to delete in the **Parameters** grid.
2. Click the **Delete**  button on the right side of the dialog box.
3. When you click **OK** or **Apply**, the selected parameters are deleted.
4. If you click **Cancel**, the selected parameters will be retained.

Delete Candidate Values from a CMOST Parameter

One or more selected values can be deleted at one time. There are two ways to delete the candidate values.

On the **CMOST Parameters Selection** dialog box:

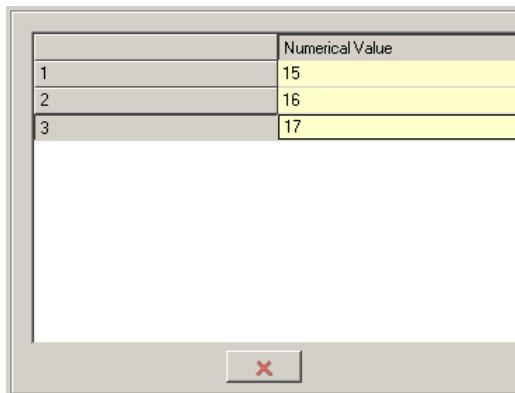
1. In the **Parameters** pane, select the CMOST parameter for which you want to delete one or more values.
2. Select the values in the **Candidate Values** grid.



3. Click the **Delete** button on the right side of the grid.
4. Once you click **OK** or **Apply**, the selected values are deleted.

On the **CMOST Parameters Selection** dialog box:

1. In the **Parameters** pane, select the CMOST parameter for which you want to delete one or more values.
2. Click the **Edit Parameter** button.
3. Select the values in the right bottom grid.



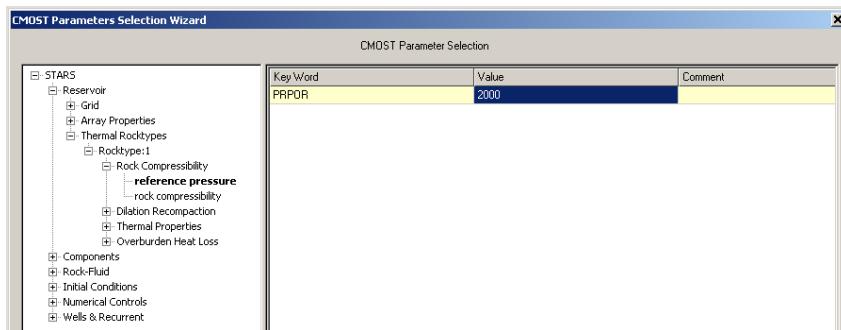
4. Click the **Delete** button at the bottom of the grid.

Add CMOST Parameter Associations

To create the CMOST master dataset, the CMOST parameter associations must be added to the CMOST parameters. The CMOST parameter associations tells Builder which data is to be replaced by the CMOST variables.

To add a CMOST parameter association:

1. On the **CMOST Parameter Selection** dialog box, select a CMOST parameter on the **Parameters** grid.
2. Click the **Add Dataset Parameter Association**  button beside the **Dataset Parameters Association** pane. The **CMOST Parameters Selection Wizard** dialog box is displayed, as follows:

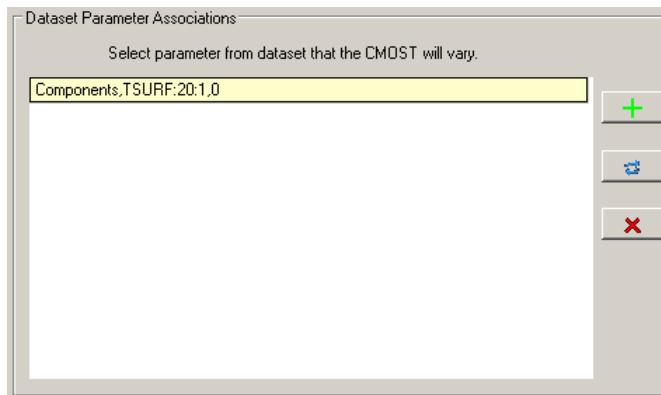


3. Select a tree node from the tree view.
4. Click a data value in the grid.
5. Click **OK** to add the parameter association.

Delete CMOST Parameter Associations

To delete a CMOST parameter association:

1. On the **CMOST Parameter Selection** dialog box, select the CMOST parameter associations in the **Dataset Parameter Associations** grid at the bottom right.

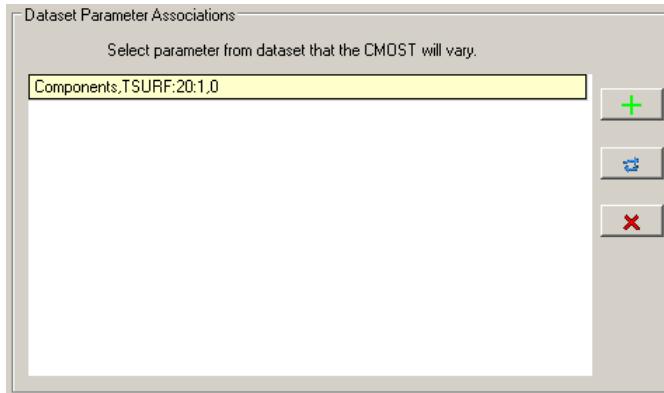


- Click the **Delete Dataset Parameter Association**  button on the right bottom group box.

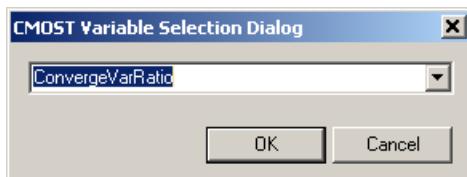
Switch CMOST Parameter Associations

To switch a CMOST parameter association from one parameter to another:

- On the **CMOST Parameter Selection** dialog box, select the CMOST parameter associations in the **Dataset Parameter Associations** grid at the bottom right.



- Click the **Switch Dataset Parameter Association**  button to the right of the **Dataset Parameter Associations** pane. The **CMOST Variable Selection Dialog** is displayed:



- Select the parameter to which you want to switch.
- Click **OK**.

Export CMOST Master Dataset

After adding CMOST associations to parameters, the CMOST master dataset can be exported as follows:

- On the **CMOST Parameters Selection** dialog box, the names of the task and master files are displayed:



Master files are usually saved to the same folder as the task file. If there is already a master file in the task folder, that name will be displayed as the **CMOST master file**. If there is no CMOST master file in the task folder, Builder will use the task file name, with the extension “cmm” appended, as shown above. If this is acceptable, click the **Export**  button. If there is a CMOST master file with the same name in the task folder, the **Export CMM File Option** dialog box will be displayed:



If you select **Merge** then click **OK**, Builder will generate a table of differences between the existing file and the new file, as outlined in [Merge Master Dataset Files](#). If you select **Overwrite** then click **OK**, Builder will overwrite the existing master file with the new master file.

2. Alternately, click **OK** to close the **CMOST Parameters Selection** dialog box, and then use **File | CMOST Save As** in the main Builder menu to export the CMOST master dataset. You can also click **Save**  in the Builder toolbar, in which case you will be prompted to confirm the saving of both the task and the master files.

Note: Refer to [Save As CMOST Files](#) for information about using the **CMOST Save As** menu.

Save CMOST Task file

After any changes are made, the CMOST task file can be saved, as follows:

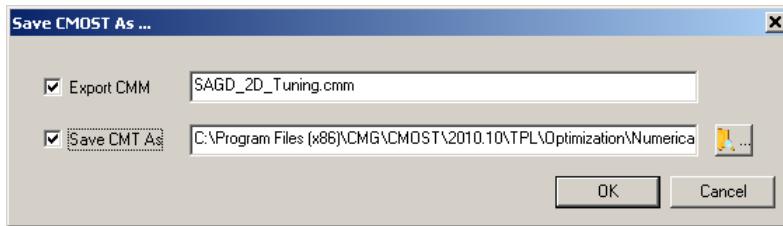
1. In the **CMOST Parameter Selection** dialog box, click **Save**  to the right of **CMOST Task File** box to save the CMOST task file. You can change the task file name by typing it directly or by clicking the **Browse**  button.
2. Alternatively, you can click **OK** to close the **CMOST Parameters Selection** dialog box then save the CMOST task file through **File | CMOST Save As**. You can also click **Save**  in the Builder toolbar, in which case you will be prompted to confirm the saving of both the task and the master files.

Note: Refer to [Save As CMOST Files](#) for information about using the **CMOST Save As** menu.

Save As CMOST Files

After you have edited the CMOST parameters, the CMOST task file, master file and base dataset can be saved. If you want to save changes to the default files, click **OK** on the **CMOST Parameters Selection** dialog box to close it, and then select **Save** from the File menu or click the **Save**  button on the toolbar. It is also possible to save the data into different files by performing the following steps:

1. Click **File | CMOST Save**. The **Save CMOST As** dialog box is displayed:



2. Select the **Export CMM** and **Save CMT File** options.

- You can change the directory and file name by clicking the **Browse**  button. Since the files should be saved in the same directory, the **Browse** button applies for all of the file types.
- You can choose to export the CMOST master file (*.cmm file) only.

3. Click **OK** to carry out the selected options.

Merge Master Dataset Files

When you are saving or exporting the CMOST master file and one with that name already exists in the target same directory, then you will be presented with the option to overwrite or merge the master files, as follows:

1. Select an option from the **Export CMM File Option** then click **OK**:



2. If you select the **Merge** option, the **CMOST File Mergence** dialog box will be displayed:



The left pane contains the contents of the existing file, the middle pane contains the contents of the new file, and right pane contains the merged file. Text that is colored red represents text that is new. Text that is colored blue represents text that has been deleted. Text that is colored yellow represents text that has been changed.

3. Use the arrow buttons at the top left to move the cursor.
4. Use the options Keep Old Keep New Existing then New New then Existing to merge the old file with the new file:
 - Select **Keep Old** if the original CMOST keywords need to be retained.
 - Select **Keep New** if the new CMOST keywords should be used.
 - If the **Keep Old** and **Keep New** options are checked, the **Existing then New** and **New then Existing** options will be enabled. Select the desired ordering using these options.
5. When you have completed the merge, click **OK** in the **CMOST File Mergence** dialog box. You will be returned to the **CMOST Parameter Selection** dialog box through which you can save your changes.

Parameterizing Relative Permeability Correlations

In CMOST, you may want to see how changes in their relative permeability tables will affect their reservoir properties. When working in Builder, relative permeability tables are typically modified by changing the end points associated with the table then getting Builder to recalculate the table values based on available correlations. The following steps show how Builder can be used to modify the end point values for CMOST parameterization.

1. Open a CMT file with Builder, and make sure there is at least one relative permeability table in the main dataset.
2. Click the **Add CMOST Parameters**  button, select the **Select parameter from dataset** option, and then click **Next**.



3. Expand the **RESULTS** section by clicking it on the tree view. Select the table that will be modified. Select an endpoint value that you want to change.

The dialog box shows a tree view on the left with sections like STARS, Reservoir, Components, Rock-Fluid, Initial Conditions, Numerical Controls, Wells & Recurrent, and RESULTS. The RESULTS section is expanded, showing Rel. Perm. Tables, Rock Type 1, Set 1, Set 2, and Rock Type 2. On the right is a table titled "CMOST Parameter Selection" with columns "Key Word" and "Value". The table lists various parameters and their values, with the "SORG" row highlighted.

| Key Word | Value |
|-------------|------------|
| SWCON | 0.2 |
| SWCRIT | 0.2 |
| SORW | 0.2 |
| SORW | 0.2 |
| SOIRG | 0.2 |
| SORG | 0.2 |
| SGCON | 0.2 |
| SGCRIT | 0.2 |
| KROCW | 0.2 |
| KRWIRO | 0.2 |
| KRIGCL | 0.2 |
| NW | 2 |
| NOW | 2 |
| NOG | 2 |
| NG | 2 |

4. Perform the same steps as you would for adding a normal CMOST parameter association.
5. Export to a CMM file.

Note: CMOST Studio will call Builder to recalculate the relative permeability table in the background and save the new dataset. For additional information, refer to the *CMOST Studio User's Guide*.

Parameterizing Formulas

It is possible to use CMOST parameter settings to change a property that is used within a formula. The steps for doing this are as follows:

1. Open a CMT file in Builder and make sure there is at least one property that is calculated by a formula in the main dataset.
2. Click the **Add CMOST Parameters**  button, select the **Select parameter from dataset** option, and then click **Next**.



3. Expand the **RESULTS** part of the tree view, and then select the formula you would like to change.

A screenshot of the CMOST software interface. On the left is a tree view of project components: IMEX, Reservoir, Components, Rock-Fluid, Initial Conditions, Numerical Controls, Wells & Recurrent, and RESULTS. Under RESULTS, there are Formulae, Formula1, and Scheme2. To the right is a table with three columns: Column 1, Column 2, and Column 3. Row 1 contains X0, *, and 1.2 respectively. The 'Scheme2' node in the tree view is highlighted.

4. Perform the same steps as you would for adding a normal CMOST parameter association.
5. Export to a CMM file.

Note: CMOST Studio will call Builder to recalculate the relative permeability table in the background and save the new dataset. For additional information, refer to the *CMOST Studio User's Guide*.

Exporting Data

Overview

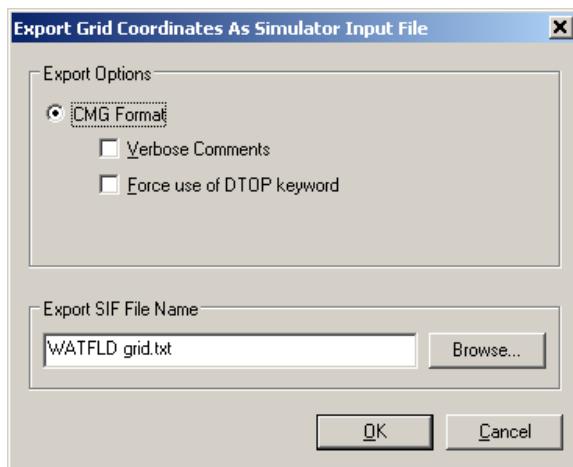
Grid coordinates, and the value of the currently displayed property, can be exported to ASCII text files. The grid coordinates and property values exported in this fashion can be used directly in the simulator input file (dataset file) for CMG simulators. The exported property values can be used as initial values in the dataset. Alternatively, you could use them as map files for property calculations. For this purpose, export the properties in the Mesh or XYZ formats.

You could also optionally export well perforation locations in terms of x and y along with the property values in the Mesh and XYZ formats. If you exported Grid Top with the well information, Builder will add the well names and perforations within the grid to the dataset when the property is calculated.

Exporting Grid Coordinates and Property Values

Exporting grid coordinates:

1. Select Tools | Export Grid Coordinates. The **Export Grid Coordinates As Simulator Input File** dialog box is displayed:

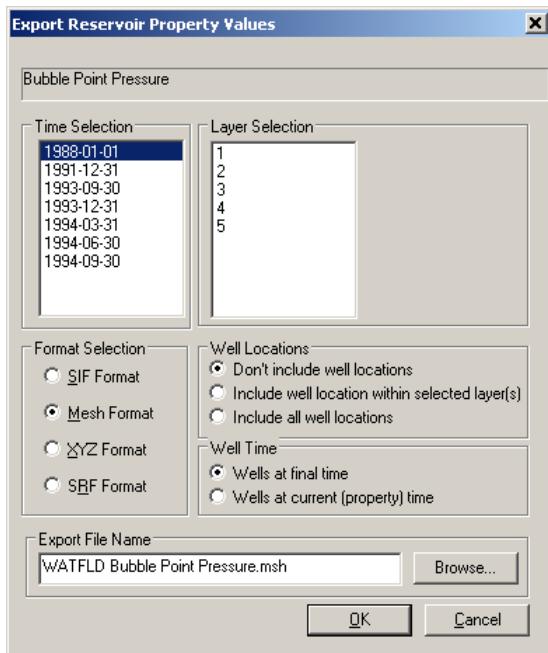


2. Select the options. Results will write the depth values with the *PAYDEPTH keyword (depth specified for more than one layers). To output *DTOP (depth for only one layer) select the **Force use of DTOP keyword** option.

3. Specify a file name. You can change the file name and/or path with the **Browse** button.
4. Click **OK**.

Exporting property values:

1. Select the desired property in the Builder main screen.
2. Select **Tools | Export Property Values**. The **Export Reservoir Property Values** dialog is displayed.



3. Select the times for which you want to export the property. You can use the SHIFT and CTRL keys to select multiple times.
4. Select the export format.
5. Use **SIF Format** to include the property directly in a CMG dataset.
6. Use **Mesh Format** or **XYZ Format** to export the data in map formats. Select one the options under **Well Locations** and **Well Time**.
7. Depending on the format you have selected, you may be able to select layers. Again, you can use the SHIFT and CTRL keys to select multiple layers.
8. Select the desired **Well Locations** and **Well Time** options.
9. Specify the file name. You can change the file name and path using the **Browse** button.
10. Click **OK**.

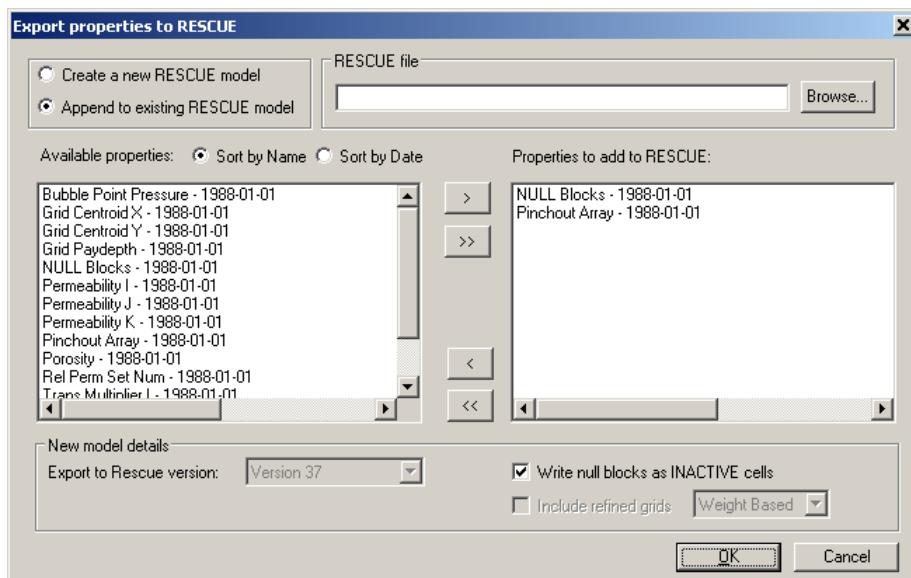
Exporting Properties to a RESCUE Model

RESCUE is a Joint Industry Project managed by the Petrotechnical Open Software Corporation (POSC). “RESCUE” stands for *REServoir Characterization Using Epicentre*. At its inception, the purpose was to provide a forum for the development of an open standard for the transfer of data from geomodels to upscalers. A RESCUE format model can define “block units” (formed by 3D surfaces representing horizons, boundaries and fault surfaces), 3D grids, and properties such as porosity and permeability. Fault surfaces, wellbore trajectories, and well logs can also be included in the model.

Using Builder, you can import a “global grid” from a RESCUE model, or all the units of a single RESCUE block. Once you have built a complete dataset using this imported grid and have run the simulation, you can export (or append) simulation calculated properties (such as saturations changing with time) back to the RESCUE model. Some geological modelling software packages can then use this data for well planning or 4D seismic applications.

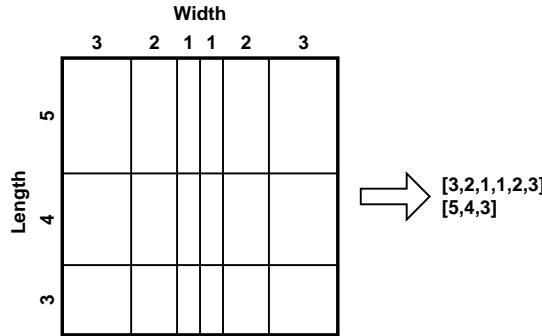
To export properties back to a RESCUE model:

1. Select **Tools | Export Properties to RESCUE model**. The **Export properties to RESCUE** dialog box is displayed:

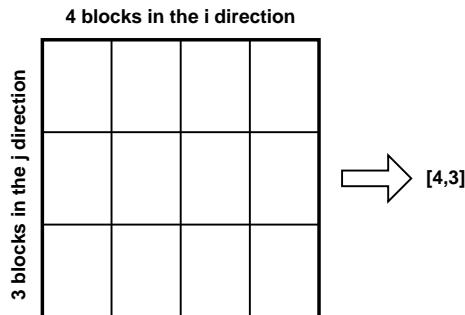


2. To append to an existing RESCUE model, click the **Browse** button, and then browse to and select the RESCUE model to which you want to append. Results will verify that the simulation grid and the RESCUE model grid match.
3. Select properties from the list of available properties and times in the left column, and then add them to the right column (**Properties to add to RESCUE**) by clicking the **Right Arrow** button.
4. If you have made grid refinements, **Include refined grids** is checked and you will be able to select one of the following options:

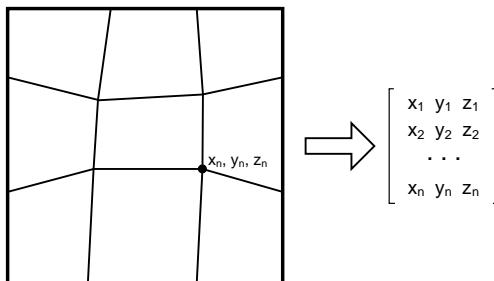
- **Weight Based (default):** All refinement blocks are regularly divided into rectangles, but their dimensions are weighted to add detail to, for example, a hydraulic fracture. In this case, only the widths and lengths of the rectangles need to be exported, as illustrated below:



- **Rule Based:** All blocks have the same dimensions. In this case, only the grid array size $[n \times m]$ needs to be exported, as shown in the following example:



- **Unconstrained:** The coordinates of the grid are not constrained and each node is exported as $[x \ y \ z]$. An unconstrained grid is illustrated below:



5. Click **OK** to add the selected properties to the RESCUE file.

Appendix A: Units and Unit Strings

Overview

In many of the Builder dialog boxes, you enter numerical values in text entry boxes. In most cases, it is possible to indicate that the numerical value entered is not in the working units (that is, SI, Field, or Lab, chosen when you create a new dataset). In these text entry boxes, you can type the units after the numerical value (separated by a space). Builder will attempt to interpret the unit string, and then convert the numerical value entered into the working units that the simulator requires.

There are two tables in this chapter. The first is a table of the expected units, by unit system, and the second is a table of allowed unit strings for each unit dimension.

Strings for compound units are formed by combining simple unit strings. For example, to form a string for a well liquid rate, take a string from the dimension [well liquid volume] and a string from [well rate time]. For this example, the units string would be “m³/day” or “MBBL/day”. If a unit string for a single dimension contains a “/”, then enclose the string in parenthesis when forming a compound unit string [for example, 1/(kg/cm²) for inverse pressure in modified SI].

Expected Units

| Dimension | SI | Field | Lab | Mod. SI | GEM SI | GEM Field |
|-----------------------------|--------------------|--------------------|--------------------|--------------------|--------------------|--------------------|
| Diffusion/Dispersion Coeff. | cm ² /s |
| Electrical Conductivity | S/m | S/m | S/m | S/m | S/m | S/m |
| Electrical Current | A | A | A | A | A | A |
| Electrical Energy | kW-hr | kW-hr | kW-hr | kW-hr | kW-hr | kW-hr |
| Electrical Potential | V | V | V | V | V | V |
| Electrical Power | kW | kW | W | kW | kW | kW |
| Electrical Resistance | ohm | ohm | ohm | ohm | ohm | ohm |
| Energy | J | Btu | J | J | J | Btu |
| Interfacial Tension | dyne/cm | dyne/cm | dyne/cm | dyne/cm | dyne/cm | dyne/cm |
| Length | m | ft | cm | m | m | ft |
| Mass | kg | lb | g | kg | kg | lb |
| Molar Mass | gmole | lmole | gmole | gmole | gmole | gmole |
| Permeability | md | md | md | md | md | md |
| Pressure | kPa | psi | kPa | kg/cm ² | kPa | psi |
| Property Volume | m ³ | ft ³ | cm ³ | m ³ | m ³ | ft ³ |
| Temperature | C | F | C | C | C | F |
| Temperature Difference | C | F | C | C | C | F |
| Time | day | day | min | day | day | day |
| Viscosity | cp | cp | cp | cp | cp | cp |
| Well Gas Volume | m ³ | ft ³ | cm ³ | m ³ | m ³ | ft ³ |
| Well Liquid Volume | m ³ | bbl | cm ³ | m ³ | m ³ | bbl |
| Well Rate Time | day | day | min | day | day | day |

Allowed Unit Strings

| Dimension | Allowed Unit Strings |
|-----------------------------|---|
| Diffusion/Dispersion Coeff. | cm ² /s |
| Electrical Conductivity | S/m |
| Electrical Current | A, mA, kA |
| Electrical Energy | kW-hr, GJ, J |
| Electrical Potential | V, mV, kV |
| Electrical Power | kW, W, J/day |
| Electrical Resistance | ohm |
| Energy | J, Btu, MBTU |
| Interfacial Tension | dyne/cm, N/m, kPa-m, mN/m |
| Length | m, ft, cm, inches, mm |
| Mass | kg, lb, g, ton, tonne |
| Molar Mass | gmole, lbmole, kgmole |
| Permeability | md, darcy, m ² , micro-m ² |
| Pressure | kPa, psi, atm, bar, kg/cm ² , MPa |
| Property Volume | m ³ , ft ³ , cm ³ |
| Temperature | C, F, K, R |
| Temperature Difference | C, F, K, R |
| Time | day, hr, min, yr, s |
| Viscosity | cp, kPa-day, kPa-hr |
| Well Gas Volume | m ³ , ft ³ , cm ³ , bbl, Mcf, MMcf, E3m ³ |
| Well Liquid Volume | m ³ , ft ³ , cm ³ , bbl, STB, MBBL, MMBBL |
| Well Rate Time | day, hr, min, yr |

Appendix B: Technical Notes - Tubing Head Pressure Calculator

Overview

The tubing pressure table calculator (ptube.exe) was developed to generate tubing pressure tables for producers using the PTUBE1 keyword or for injectors using ITUBE1 keyword.

For producers, liquid-gas two-phase tubing flow is handled within the calculator. Basically, the production flow is upwards; however, if the appropriate pressure predication method is selected, tubing segments with horizontal or even downward flow are acceptable. Optionally, gas lift is applicable to producers.

For injectors, only single phase flow, dry gas or water, is handled; therefore, no specification of pressure method is required and tubing segments with any angle are acceptable.

The input wellbore profile consists of a user-defined number of segments. Within the tubing calculator, each of these segments is divided into sections to apply Runge-Kutta integration. The length of sections is limited to a maximum of 32 meters and may dynamically be changed based on the convergence characteristics of the integration. When the gas phase is not negligible, a section's length is also limited to ensure the pressure drop within the section is less than 100 psi. Within a section, equal Runge-Kutta steps are applied. Local fluid PVT properties and pressure gradient are evaluated in each of these steps.

When lifting gas is injected, a user-provided lifting gas dissolvability is applied for all the integration steps above the injection valve. The dissolvability is a volume fraction, which is the volume of the lifting gas allowed to dissolve into the under-saturated oil.

The following sections provide an outline of the correlations used for the evaluation of PVT properties and pressure gradients.

PVT Correlations

A black oil, gas and water PVT package based on the summary by McCain (McCain 1991) is used to predict the local properties of the fluid. Additional correlations are added for calculations of volumetric properties.

Oil bubble point pressure - solution GOR methods:

1. Vazquez-Beggs (Vazquez and Beggs 1980)
2. Lasater's correlation combined with the Standing's correlation (Chierici et al 1974, Beggs 1987)

If API > 15, Lasater's correlation is used; otherwise, Standing's correlation is used.

Oil formation volume factor methods:

1. Standing (McCain 1991)
2. Vazquez-Beggs (Vazquez and Beggs 1980)

Vazquez-Beggs and Standing are set to be the default correlations for Oil Pb/Rs and FVF calculations respectively.

The interfacial tension (IFT) is calculated internally using the correlation of Baker-Swerdloff (Beggs 1987).

Pressure-Gradient Prediction Methods

Six selectable methods are provided for predicting the pressure loss in the wellbore. Three of these methods were developed for flow at any inclination. The remaining methods are strictly validly only for upward flow.

Beggs-Brill (Beggs and Brill 1973, Brill and Beggs 1984) is a correlation, which was the first one to predict the flow for all inclinations. The method is believed to be suitable for pipeline system for the method tends to over predict pressure drop both at upward and downward inclinations. However, in the tubing calculator, Payne et al's modifications (Petalas and Aziz 2000) are applied to improve these errors.

Mukherjee-Brill (Mukherjee and Brill 1985, Mukherjee and Brill 1985 [two references]) is a correlation in attempt to overcome some limitations of Beggs-Brill method. Except for the downward stratified flow, the liquid-holdup correlation is expressed by separate equations for upward and downward flow, which are continuous across flow pattern regimes.

Petalas-Aziz mechanistic model (Petalas and Aziz 1996, Petalas and Aziz 2000) is a method applicable for all pipe inclinations. The model was tuned against 5,951 data points from the Stanford Multiphase Flow Database. The method was tested extensively and proved to be robust and exhibited few discontinuities.

The **Aziz-Govier** (Aziz, Govier and Fogarasi 1972) is a correlation for vertical producer, which is included as it is the default used in the analytical pressure drop model used in CMG simulators.

The **Drift-flux** (Shi et al 2003) is a method developed at Stanford, which uses the mechanistic approach for bubble or slug flow and introduces a transition zone to and equations for annular flow. The method employs one equation system for all flow regimes, so that it is characterized by simple, naturally continuous and smooth behavior. The method is currently valid for upward flow.

A very simple **Simplified mixed density** model is also included. In this model, all flow is assumed to occur in the bubble regime.

In the tubing calculator, the Moody frictional factor is calculated using the original Colebrook's equation and Zigrang et al's explicit approximation (Zigrang and Sylvester 1985) [for Petalas-Aziz mechanistic model] is also used.

When a small rate (for example, 3 inch tube with 10 bbl oil per day) is given for a producer combined with the gas lift option, tables made by the Beggs-Brill method may contain jump points at these small rates. This is because the huge gas/liquid ratio the method encounters is out of the smooth range of the correlation. The Mukherjee-Brill method can also suffer from similar problems. If the table has to cover small rates and uses gas lift, the Petalas-Aziz mechanistic model or drift-flux method is recommended.

References

- Aziz, K., Govier, G.W., and Fogarasi, M. "Pressure Drop in Wells Producing Oil and Gas," J. Cdn. Pet. Tech., July-September 1972, 11, pp. 38-48
- Beggs, H.D. and Brill, J.P.: "A Study of Two-Phase Flow in Inclined pipes," J. Pet. Tech., May 1973, pp. 607-617
- Brill, J.P. and Beggs, H.D.: "Two-Phase Flow in Pipes," University of Tulsa, 1984.
- Brill, J.P. and Mukherjee, H.: "Multiphase Flow in Wells," SPE. Monograph, Vol. 17, 1999, Richardson Texas
- Chierici, G.L. et al.: "Two-Phase Vertical Flow in Oil Wells – Prediction of Pressure Drop," J. Pet. Tech., Aug. 1974, pp. 927-938
- Beggs, H.D.: "Oil System Correlations," Petroleum Engineering Handbook, SPE, 1987, Richardson Texas, Chap. 22
- McCain, W.D. Jr.: "Reservoir-Fluid Property Correlations – State of the Art," SPERE, May 1991, pp. 266-272
- Mukherjee, H. and Brill, J.P.: "Pressure Drop Correlations for Inclined Two-Phase Flow," J. Energy Res. Tech., Dec. 1985, Vol. 107, pp. 549-554
- Mukherjee, H. and Brill, J.P.: "Empirical Equations to Predict Flow Patterns in Two-Phase Inclined Flow," Int. Multiphase Flow, Vol. 11, No. 3, pp. 299-315, 1985.
- Petalas, N. and Aziz, K.: "Development and Testing of Mechanistic Models for Multiphase Flow in Pipes," Petroleum Engineering Department, Stanford University, April, 1996.
- Petalas, N. and Aziz, K.: "A Mechanistic Model for Multiphase Flow in Pipes", J. Cdn. Pet. Tech, June 2000, pp. 43-55
- Shi, H. et al.: "Drift-Flux Modeling of Multiphase Flow in Wellbores," SPE 84228, 2003.
- Vazquez, M. and Beggs, H.D.: "Correlations for Fluid Physical Property Prediction," JPT (June 1980) 968
- Zigrang, D.J., Sylvester, N.D.: "A review of Explicit Friction Factor Equations", J. Energy Res. Tech., June 1985, pp. 280-283

Appendix C: Correlation Equations

Equations for Prediction of Relative Permeability in Sandstone and Conglomerate

$$k_{rw} = 0.035388 \frac{(Sw - Sw_i)}{(1 - Sw_i - So_{rw})} - 0.010874 \times \left[\frac{(Sw - So_{rw})}{(1 - Sw_i - So_{rw})} \right]^{2.9} + 0.56556 (Sw)^{3.6} (Sw - Sw_i) \quad (water-wet)$$

$$k_{rw} = 1.5814 \left(\frac{Sw - Sw_i}{(1 - Sw_i)} \right)^{1.91} - 0.58617 \frac{(Sw - So_{rw})}{(1 - Sw_i - So_{rw})} \times (intermediately-wet) \\ (Sw - Sw_i) - 1.2484 \phi (1 - Sw_i) (Sw - Sw_i)$$

$$k_{ro*sw} = 0.76067 \left[\frac{\left(\frac{So}{1 - Sw_i} \right) - So_r}{1 - So_{rw}} \right]^{1.8} \left[\frac{So - So_{rw}}{1 - Sw_i - So_{rw}} \right]^{2.0} \quad (any\ wettability) \\ + 2.6318 \phi (1 - So_{rw}) (So - So_{rw})$$

$$k_{ro*rg} = 0.98372 \left(\frac{So}{1 - Sw_i} \right)^4 \left[\frac{So - So_{rg}}{1 - Sw_i - So_{rg}} \right]^2 \quad (any\ wettability)$$

$$k_{rg} = 1.1072 \left(\frac{Sg - Sg_c}{1 - Sw_i} \right)^2 k_{rg(So_{rg})} + 2.7794 \times \frac{So_{rg} (Sg - Sg_c)}{(1 - Sw_i)} k_{rg(So_{rg})} \quad (any\ wettability)$$

Equations for Prediction of Relative Permeability in Limestone and Dolomite

$$k_{rw} = 0.0020525 \frac{(Sw - Sw_i)}{\phi^{2.15}} - 0.051371 (Sw - Sw_i) \left(\frac{1}{k_a} \right)^{0.43} \quad (water-wet)$$

$$k_{rw} = 0.29986 \left(\frac{Sw - Sw_i}{1 - Sw_i} \right) - 0.32797 \left(\frac{Sw - So_{rw}}{1 - Sw_i - So_{rw}} \right)^2 x \\ (Sw - Sw_i) + 0.413259 \left(\frac{Sw - Sw_i}{1 - Sw_i - So_{rw}} \right)^4 \quad (intermediately\ wet)$$

$$k_{ro^*w} = 1.2624 \left(\frac{So - So_{rw}}{1 - So_{rw}} \right) \left(\frac{So - So_{rw}}{1 - Sw_i - So_{rw}} \right)^2 \quad (any\ wettability)$$

$$k_{ro^*g} = 0.93752 \left(\frac{So}{1 - Sw_i} \right)^4 \left(\frac{So - So_{rg}}{1 - Sw_i - So_{rg}} \right)^2 \quad (any\ wettability)$$

$$k_{rg} = 1.8655 \frac{(Sg - Sg_c)(Sg)}{1 - Sw_i} k_{rg(So_{rg})} + 8.0053 x \\ \frac{(Sg - Sg_c)(So_{rg})^2}{(1 - Sw_i)} - 0.025890 (Sg - Sg_c) x \\ \left(\frac{1 - Sw_i - So_{rg} - Sg_c}{1 - Sw_i} \right)^2 x \\ \left(1 - \frac{1 - Sw_i - So_{rg} - Sg_c}{1 - Sw_i} \right)^2 \left(\frac{k_a}{\phi} \right)^{0.5} \quad (any\ wettability)$$