

Reservoir Engineering 3 Day Training and Exercise Guide

Version 2017



Petrel

Shared earth—critical insight

NExT
A Schlumberger Company

Petrel Reservoir Engineering

3 Day

WORKFLOW/SOLUTIONS TRAINING
Version 2017

Software Integrated Solutions

August 16, 2017



Copyright notice

Copyright © 2017 Schlumberger. All rights reserved.

This work contains the confidential and proprietary trade secrets of Schlumberger and may not be copied or stored in an information retrieval system, transferred, used, distributed, translated or retransmitted in any form or by any means, electronic or mechanical, in whole or in part, without the express written permission of the copyright owner.

Trademarks & service marks

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

An asterisk (*) is used throughout this document to designate a mark of Schlumberger.

ABOUT THIS MANUAL

Prerequisites.....	2
Learning objectives.....	2
What you need.....	3
What to expect.....	4
Icons.....	5
More Petrel training courses.....	6
Petrel Reservoir Engineering course structure.....	7
Summary.....	7

INTRODUCTION TO PETREL

Learning objectives.....	9
Lesson 1: Seismic to simulation workflow with one application.....	10
Lesson 2: Simulators launched in Petrel.....	14
Lesson 3: Studio E&P knowledge environment.....	19
Review and summary.....	19
Review questions.....	20
Summary.....	20

Copyright © 2017 Schlumberger. All rights reserved.

PETREL RESERVOIR ENGINEERING FOR NEWCOMERS

Learning objectives.....	23
Lesson 1: Petrel user interface.....	24
Exercises — Use the Petrel user interface.....	84
Workflow.....	84
Data.....	85
Review and summary.....	101
Review questions.....	101
Summary.....	102

SIMPLE SIMULATION MODELS

Prerequisites.....	105
Learning objectives.....	105
Lesson 1: Simple simulation grid.....	107
Exercises — Make a simple grid with a simulation fault.....	132
Workflow.....	132
Data.....	132
Lesson 2: Rock physics functions.....	148

TABLE OF CONTENTS

Exercises — Create rock physics functions.....	165
Data.....	165
Lesson 3: Fluid models.....	175
Exercises — Make a fluid model.....	188
Data.....	188
Review and summary.....	191
Review questions.....	192
Summary.....	192

MODEL INITIALIZATION AND VOLUME CALCULATION

Learning objectives.....	195
Lesson 1: Initialization of the simulation model: Define simulation case.....	196
Exercises — Set up and initialize a simulation case	217
Workflow.....	217
Data.....	218
Lesson 2: Volume calculation.....	224
Exercises — Perform volume calculations.....	233
Workflow.....	233
Data.....	233
Review and summary.....	243
Review questions.....	243
Summary.....	243

Copyright © 2017 Schlumberger. All rights reserved.

SIMULATION RUN AND RESULTS VIEWING

Learning objectives.....	247
Lesson 1: Simulation case setup and output requirements.....	249
Lesson 2: Results charting and analysis.....	251
Exercises — Use the Results charting and analysis process.....	280
Workflow.....	280
Data.....	280
Lesson 3: 3D simulation results	296
Exercises — Analyze 3D simulation results.....	328
Workflow.....	328
Data.....	329
Lesson 4: Summary calculator and Geometrical modeling.....	338
Lesson 5: Dynamic saved searches.....	351
Review and summary.....	359
Review questions.....	359

Summary.....	359
--------------	-----

HISTORY MATCHING AND PREDICTION

Learning objectives.....	363
Lesson 1: Introduction to development strategy.....	364
Exercises — Create a history development strategy.....	376
Workflow.....	376
Data.....	376
Lesson 2: Fault transmissibility multipliers.....	395
Exercises — Assign transmissibility multipliers to faults.....	400
Workflow.....	400
Data.....	400
Lesson 3: Workflow editor and its applications.....	405
Exercise — Create a workflow	412
Data.....	412
Lesson 4: Prediction strategy.....	415
Exercises — Make prediction strategies.....	429
Workflow.....	429
Data.....	430
Review and summary.....	439
Review questions.....	440
Summary.....	440

Copyright © 2017 Schlumberger. All rights reserved.

SIMULATION STUDY

Prerequisites.....	443
Model description.....	443
Exercises — Simulation study.....	444
Workflow.....	444
Data.....	445

GLOSSARY

Terminology.....	455
------------------	-----

TABLE OF CONTENTS

About this manual

The overall objective of this course is to acquaint you with the interactive Petrel E&P software platform reservoir engineering tools used to construct simulation models.

In this course, you use Petrel to

- build a 3D simulation model based on geological input data
- add wells and well control rules
- create black oil fluid models and rock physics functions
- initialize the model and view the initial volumes
- view the simulation results using available Petrel tools for results analysis and visualization
- upscale a fine geological grid to a coarse simulation grid
- design well trajectory using two approaches, interactive and automated
- add aquifers and local grids to a simulation case

During the course, the practical application of most of the available tools in Petrel used for a typical reservoir engineering workflow is discussed and illustrated in the exercises. The learning contents are structured in modules with relevant lessons and exercises to help practice the lessons learned in each module.

Generally, preprocessing and postprocessing of simulation data is done in the Petrel modeling environment.

The vision for Petrel reservoir engineering is to encourage asset teams to work together using a single Petrel unified platform rather than working in silos. This approach requires close collaboration among the disciplines involved in a reservoir study and, ultimately, provides a means of incorporating feedback and knowledge sharing within the asset team.



Prerequisites

To complete this course successfully, you must have

- English proficiency
- basic Windows and practical computing skills
- familiarity with reservoir engineering fundamentals



Learning objectives

In this course, you prepare black oil simulation cases for ECLIPSE using Petrel Reservoir Engineering tools.

After completing this training, you will know how to:

- build a simulation grid in the Petrel modeling environment
- scale up grid structure and petrophysical properties using the available upscaling tools in Petrel
- use a correlations library in Petrel to make black oil fluid tables and rock physics functions or, alternatively, import existing files
- create initial conditions for model initialization
- create or import well trajectories using well engineering tools in Petrel
- import history data
- create a development strategy
- set up a simulation case, run the simulation, and view the results
- convert an existing ECLIPSE model into a Petrel case

You also are introduced to the Petrel user interface relevant to the Reservoir Engineering workflow.

What you need



To complete this course successfully, you must have this hardware and these applications to perform the workflows:

Minimum System Recommendations	
Operating system (recommended)	64-bit version of Microsoft Windows 7 with service pack 1 (Enterprise, Ultimate, or Professional edition), or 64-bit version Microsoft Windows 8.1 (Enterprise or Ultimate edition)
Processor	Quad-core processor (best with a fast clock speed and high cache)
Memory	16 GB RAM (32+ GB recommended)
Display	The quality of the viewing experience increases with the size and number of monitors
Graphics	NVIDIA Quadro K4200 or NVIDIA Quadro K2200
Primary storage	Fast rotational speed HDD (10Km 15K)
Preferred Hardware	
Operating system (recommended)	Microsoft Windows 7 SP1 - (64-bit) Professional, Enterprise, or Ultimate editions
Processor	Dual quad-core or hex-core processors (best with a fast clock speed and high cache)
Memory	64 GB RAM
Display	Two monitors, minimum viewing size of 24 inches (32 inches preferred)
Graphics	NVIDIA Quadro K5200 or K6000
Primary storage	SDD or SSD
Secondary storage	Fast rotation speed HDD (10K, 15K)
Other Required Software	
Microsoft .NET(R) Framework 4.5	

What to expect

In each module in this training material, you encounter these sections:

- Overview of the module
- Prerequisites to the module (if necessary)
- Learning objectives
- A workflow component (if applicable)
- Lessons that explain a subject or an activity in the workflow
- Procedures that show the steps needed to perform a task
- Exercises that allow you to practice a task by using the steps in the procedure with a dataset
- Scenario-based exercises
- Questions about the module
- Summary of the module

You also encounter notes, tips, and best practices.

Icons

Throughout this manual, you find icons in the margin representing various kinds of information. These icons serve as at-a-glance reminders of their associated text. See below for descriptions of what each icon means.



PREREQUISITES

Prerequisites required for the course or individual modules.



LEARNING OBJECTIVES

Learning objectives set for the course or individual modules.



WHAT YOU NEED

Applications, hardware, datasets, or other materials required for the course.



LESSON

Topic lessons in each module.



PROCEDURE

Steps required to perform a given task.



EXERCISE

Your turn to practice the procedure.



QUESTIONS

Questions at the end of each lesson.



REVIEW QUESTIONS

Review questions at the end of each module.



BEST PRACTICE

Best way to complete a given task when different options are available.



NOTE

Indicates important information.



TIP

Information to make your work easier.



WARNING

Proceed with extreme caution.

More Petrel training courses

This figure shows an overview of the Petrel courses and their assigned proficiency levels.

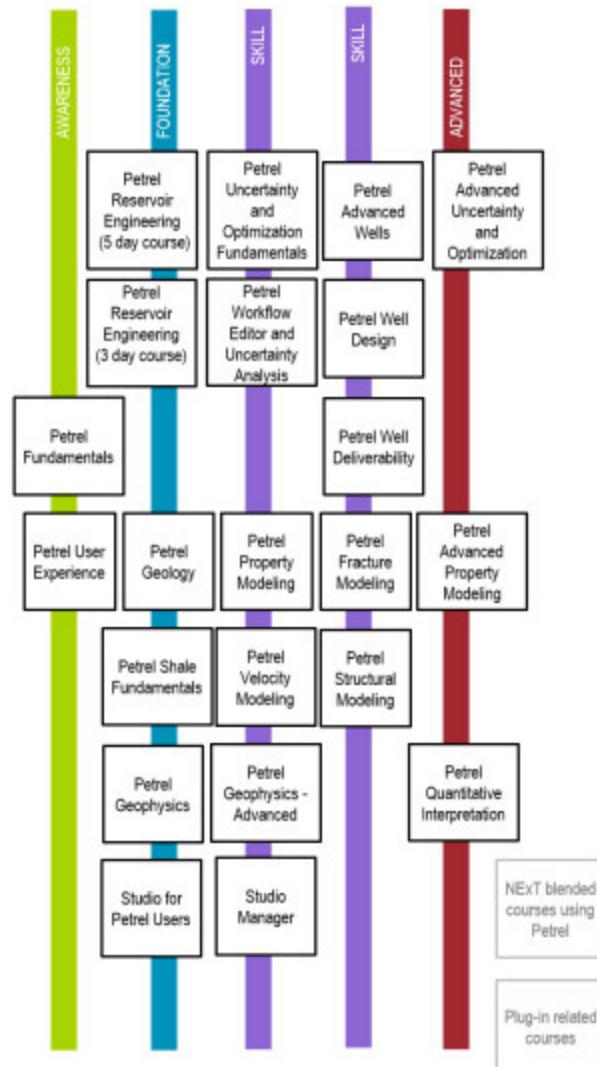


Figure 1. Petrel training courses

Petrel Reservoir Engineering course structure

This figure shows the course structure and contents for Petrel* Reservoir Engineering. The course starts with an introduction to the Petrel E&P software platform with a focus on the relevant reservoir engineering workflow interface. Subsequent learning modules then cover in detail the concepts and practices of the workflow. Finally, the Simulation study consists of scenario-based exercises designed to help you apply the knowledge that you acquire from the training.

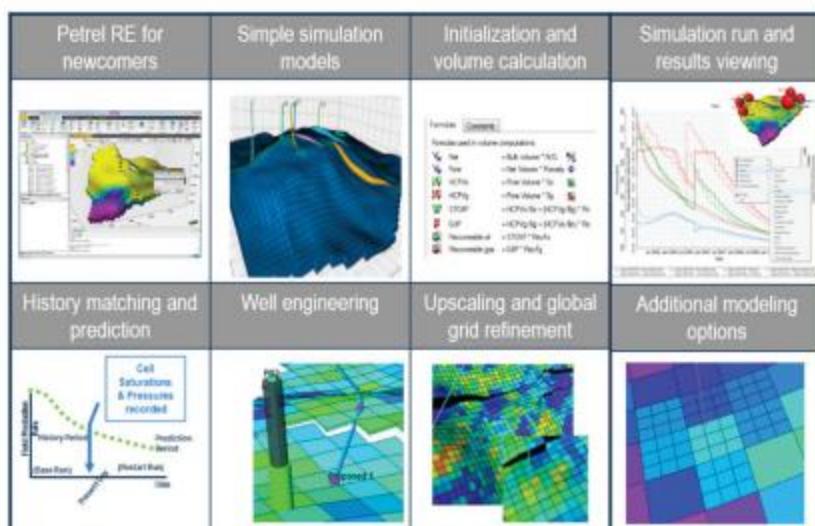


Figure 2. Petrel Reservoir Engineering course structure and content

Copyright © 2017 Schlumberger. All rights reserved.

Summary

This introduction:

- defined the learning objectives
- described the structure of the manual
- outlined the tools you need for this training
- discussed course conventions that you encounter in this material

NOTES

Copyright © 2017 Schlumberger. All rights reserved.

Module 1 Introduction to Petrel

This module introduces you to the Petrel* E&P software platform and provides an overview of the workflows. You learn about the added value of working in a unified modeling environment platform with domain driven workflows and the advantages of using one application within an asset team, especially in terms of data integration and team collaboration.

You also are introduced to the Studio* E&P knowledge environment features, which provide improved data and knowledge management.

Learning objectives

After completing this module, you will know about:

- the Petrel unified modeling environment and the value it brings to your workflow compared to other software tools
- the advantages of using one application within an asset team
- other existing Schlumberger simulation software
- Studio, a smart technology for improving productivity and knowledge management





Lesson 1 Seismic to simulation workflow with one application

In the classical reservoir modeling workflow, a geoscientist normally provides a completed static model to a reservoir engineer, which the reservoir engineer then uses to build a simulation model. In this scenario, the software platform design does not encourage asset team collaboration. The data flow between disciplines is difficult. The reservoir model cannot be updated quickly with new information. Basically, it is a one-way workflow that does not support the flow of data between different domains.

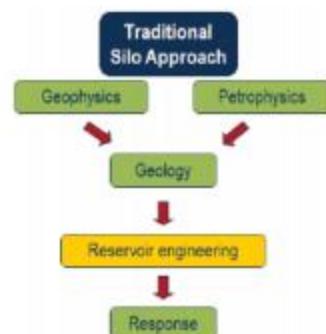


Figure 3. Traditional reservoir engineering workflow

The evolution of Petrel, with asset team-driven workflows, has changed how E&P engineers conduct their reservoir studies. The Petrel unified platform enables interaction between disciplines and collaboration within the asset team.

Copyright © 2017 Schlumberger. All rights reserved.

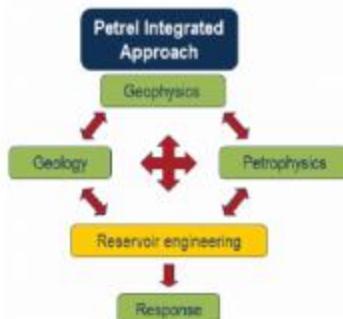


Figure 4. Petrel E&P software platform workflows

Petrel provides one common user interface for E&P workflows from seismic interpretation to reservoir simulation. Using ECLIPSE*, INTERSECT*, and FrontSim* with Petrel integrates the necessary simulation workflows so that the data flow is transparent and the interface is easy to learn.

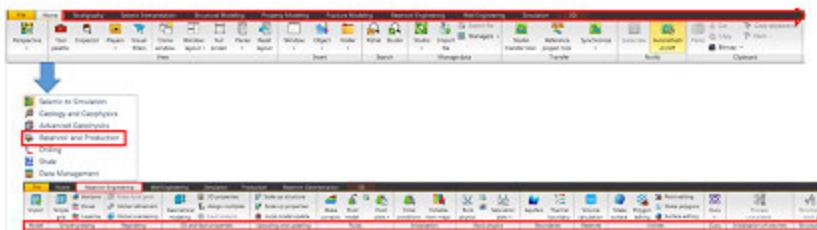


Figure 5. Ribbons in the Petrel interface are filtered based on the predefined domain driven workflow. To access these workflows, click Perspective

The Petrel environment supports most of the major workflows in ECLIPSE Office*, FloGrid*, FloViz*, PVTi*, and SCAL and Schedule. However, there are some operations that you must perform outside of Petrel. For example, Petrel functionality allows you to make compositional and thermal PVT models, but it does not support fluid characterization. For fluid characterization, use PVTi software. PVTi uses the equation of state (EOS) to simulate laboratory experiments. You then can import the results into Petrel.

Example of a workflow between a reservoir engineer and a geologist in Petrel

This scenario-based example highlights the importance of a unified software platform that incorporates modeling and simulation tools in a single application platform. It also demonstrates how this platform encourages communication and sharing of information within the asset team. In this example, a reservoir engineer receives a static model from a geologist and inputs the dynamic properties to run the simulation. The reservoir engineer observes that there is no flow between well C1 and C7 as shown in this figure.

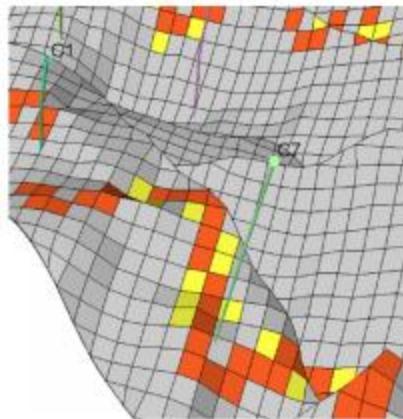


Figure 6. Fluvial system channels with no connection between well C1 and C7

However, based on an analysis of the observed pressure data, the reservoir engineer realizes that there is communication between the two wells (C1 and C7) that the geologist did not consider when building the static model (collaboration issue). To incorporate the new information from pressure analysis result into the static model, two options were proposed. Option 1 is to do a simple modification in the model to make flow possible, such as by altering the permeability model as shown in this figure. However, this type of change is not likely to represent real geological characteristics.

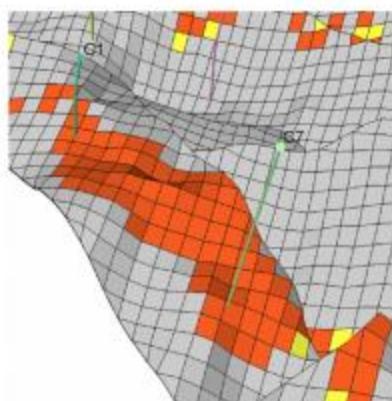


Figure 7. Simple modification to the fluvial system by the engineer

Option 2 is to follow the best practice of the reservoir engineer communicating the findings to the geologist. Then the geologist can update the model based on the feedback from the reservoir engineer so that it captures the real geologic characteristics (meandering sand channel) in the reservoir model as shown in this figure. The model can be updated easily and quickly in Petrel when the asset team uses the same application. This functionality is a unique value inherent in the Petrel workflow.

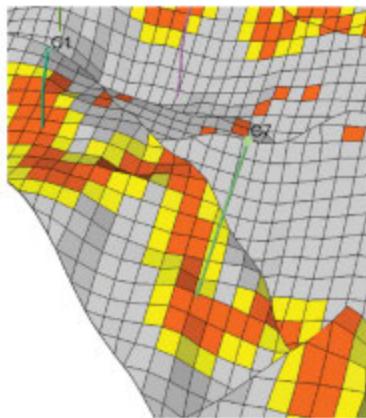


Figure 8. Geological modification: Meandering sand channel captured in the mode

You can track process activities in Petrel. This capability allows you to update or recalculate the model easily based on new information or process settings.



Lesson 2 Simulators launched in Petrel

The Schlumberger reservoir simulation software suite covers the whole spectrum of reservoir engineering workflows for all types of reservoirs and degrees of complexity: structure, geology, fluids, and development schemes. The suite specializes in black-oil, compositional, thermal, geomechanics, hydraulic fracture, and streamline reservoir simulation.



Figure 9. Simulators that can be launched in Petrel

Copyright © 2017 Schlumberger. All rights reserved.

All Schlumberger reservoir simulation software (ECLIPSE 100*, ECLIPSE 300*, FrontSim*, VISAGE*, and INTERSECT) can be launched in the Petrel environment. In addition, pre- and post-processing activities are possible in the same Petrel E&P software platform.

Define simulation case: Simulator selection options

The **Define simulation case** dialog box gives you the option to access different simulator types through the Petrel interface by selecting one of the enabled simulators. Petrel allows you to access five different

simulator types: ECLIPSE 100, ECLIPSE 300, FrontSim, VISAGE, and INTERSECT.

Select a simulator in the **Define simulation case** dialog box from the **Simulator** list based on your requirements.

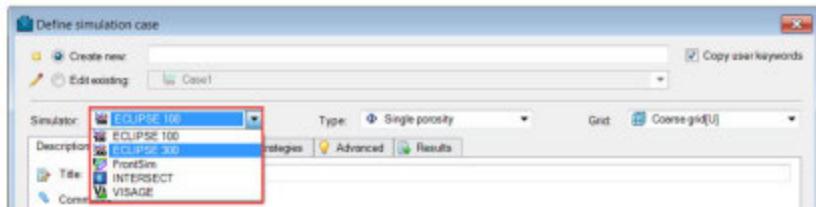


Figure 10. Simulator selection in the *Define simulation case* process

ECLIPSE Blackoil and Compositional

ECLIPSE 100 (ECLIPSE Blackoil) is a fully implicit, three-dimensional three-phase, and general purpose black oil simulator with gas condensate options. This simulator also supports extensive well controls, field operations planning, and comprehensive enhanced oil recovery (EOR) schemes.

ECLIPSE 300 (ECLIPSE Compositional) is a compositional simulator used to model reservoir fluid phase behavior and compositional changes associated with multi-component hydrocarbon flow. It uses a cubic equation of state (EOS), with the ability to define EOS regions.

FrontSim

FrontSim solves the same set of flow equations normally solved by a conventional finite difference simulator. The essential difference is that FrontSim models flow along streamlines rather than from grid block to grid block. FrontSim handles gravity segregation and compressible flow, but not capillary pressure.

FrontSim achieves greater performance by breaking large 3D problems into several 1D problems, interactively calculating pressure on the entire grid, and then updating saturation on each generated streamline independently.

FrontSim mainly is used for:

- Waterflood simulation
- Ranking and screening on geological models
- Upscaling validation
- Assisted history matching

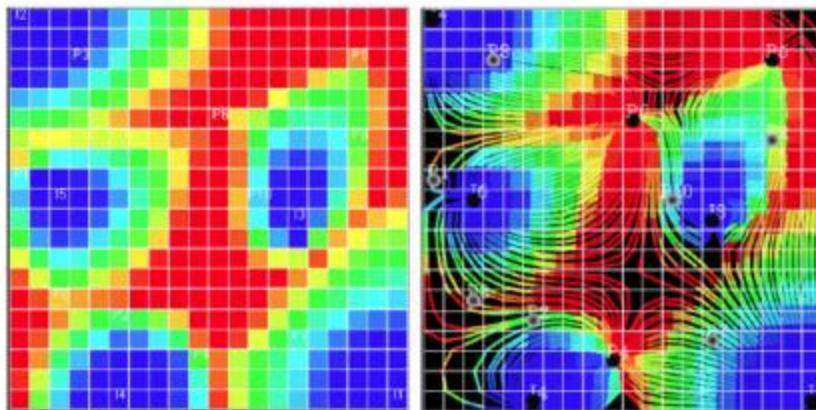


Figure 11. Finite difference (left) and Streamline (right)

INTERSECT high-resolution reservoir simulator

INTERSECT was developed with specific goals in mind. It goes beyond what the current generation of simulators offer.

- It runs high-resolution simulation models with millions of cells to capture reservoir geology with minimum or no upscaling.
- It accurately models reservoirs with high heterogeneity and permeability contrasts, natural fractured reservoirs, and highly faulted reservoirs.
- It can capture the shape of fluid fronts around wells using hundreds of structured or unstructured local grid refinements.
- It provides advanced functionality for wells, facilities, and field management.
- It provides integration with asset modeling, coupling wells, and surface production networks.

VISAGE

The VISAGE finite-element geomechanics simulator enables you to plan for and mitigate risks by modeling problems before they occur including:

- Compaction and subsidence
- Well and completion integrity
- Cap-rock and fault-seal integrity
- Fracture behavior
- Thermal recovery
- CO₂ disposal

The Petrel E&P software platform combines transparent data flows with an easy-to-learn graphical user interface (Petrel Reservoir Geomechanics) that supports the VISAGE simulation configuration and results visualization. This interface allows you to combine the powerful functionality of the VISAGE simulator seamlessly with other interpretation, modeling, and engineering workflows in the Petrel platform.

With the Petrel-enabled workflow, you can include multiple data types to create new 3D geomechanics property and stress models, or add geomechanics data to augment existing reservoir subsurface models. This seamless combination in the Petrel platform ensures that the geomechanics model is consistent and integrated with geophysics, geology, petrophysics, and reservoir data.

The initial structural and properties model, created using Petrel Reservoir Geomechanics, is submitted to the VISAGE numerical simulator, and controlled in the same familiar Petrel environment.

The VISAGE simulator also can be coupled to the ECLIPSE industry-reference reservoir simulator for both one-way and two-way coupling. In one-way coupling, the ECLIPSE simulator models the flow of fluids in the reservoir and calculates the pressure, temperature, and saturation changes that result.

The VISAGE simulator uses these calculations to perform 3D static or 4D flow-, pressure-, and temperature-coupled calculations for rock stresses, deformations, and failure. Two-way coupling between the ECLIPSE and VISAGE simulators allows you to update the permeability of the reservoir model at any selected timesteps.

You also can update mechanical properties in the geomechanics model to account for effects such as changing saturations and water softening.

For large models (many millions of cells), or those coupled to ECLIPSE reservoir simulation, you also are able to perform parallel geomechanics simulation runs using local or remote clusters. The entire process, for single computers or multicore clusters, is managed by the Petrel Reservoir Geomechanics platform, which allows the same seamless workflow to be maintained from project start to end.



Lesson 3 Studio E&P knowledge environment

The activities associated with exploration, development, and production produce large amounts of data, which you must be able to manage, access, and use effectively. The Studio environment provides three key capabilities that drive productivity for petrotechnical specialists.

- **Find** provides access to Petrel and Studio data, as well as other applications and data sources in the context of the model. Examples include GeoFrame*, Techlog*, IHS (North America), and any OpenSpirit enabled data (OW-R5000, Kingdom, Petra, EPOS, Finder*).

When searching, you can use a geographical context and filter the results of your search based on data-specific criteria (depth, user, project location, data type, dates). You also can apply spatial filters and graphically preview these results in your project to uncover previous work/interpretations or critical information instantly. When you find the data, you can load it into your project with a simple click.

- Share and collaborate with multidisciplinary asset team members to enhance the way they work together to explore, characterize, and develop reservoirs. The Studio environment enables you to publish interpretations and insights when they are ready for your team to access and use them. Team members can sign up for notifications of changes, so they are aware of updates made across the project and who made them.
- **Manage** in context of the asset to capture, retain, and deliver the necessary results to the organization. The Studio database is the engine that powers the collaboration, using a publish/subscribe model. With this approach, you publish interpretations and insights when they are ready to be shared. The Studio Manager tool gives data managers a quick understanding of the state of their Studio environment.

Review and summary

Review what you learned in this module.

The review and summary help you to reinforce the learning objectives for Introduction to Petrel.



Review questions

The review questions reinforce the learning objectives.

- Which simulators can be launched from Petrel?
- What makes the INTERSECT simulator different from the ECLIPSE simulator?
- Do you need FloGrid to build a simulation grid and PVTi to calculate fluid models based on correlations while working with Petrel?

Summary

In this module, you learned about:

- the Petrel unified modeling environment
- the advantages of using one application within an asset team
- existing Schlumberger simulation software
- Studio smart technology for improving productivity and knowledge management

NOTES

Copyright © 2017 Schlumberger. All rights reserved.

NOTES

Copyright © 2017 Schlumberger. All rights reserved.

Module 2 Petrel Reservoir Engineering for newcomers

In this module, you are introduced to the features in the Petrel user interface that are relevant to reservoir engineering workflows. The learning contents and exercises in this module are structured to help you get acquainted with the Petrel user interface focusing primarily on the reservoir engineering domain workflow.

Learning objectives

After completing this module, you will know how to:

- navigate through the typical Petrel Reservoir Engineering pre- and post-domain workflow
- use some of the Petrel simulation results analysis tools relevant to the reservoir engineering workflow
- display objects in **2D windows** and **3D windows** and use the **Inspector** to access displayed object information quickly
- review/edit object settings using the mini toolbar, tool palette, and shortcut menus
- access the Petrel Help Center and Guru Premium content





Lesson 1 Petrel user interface

The Petrel user interface is organized into four main elements: ribbon, panes, display window, and status bar.

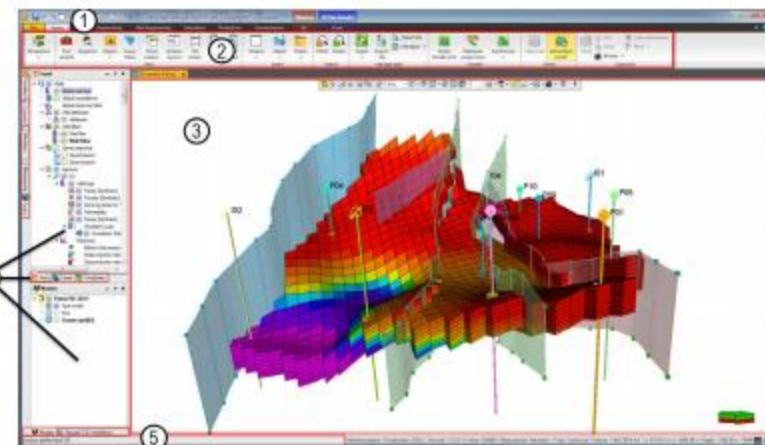


Figure 12. Petrel user interface

- 1 Quick Access Toolbar: Enables customization of your frequently used tools in the title bar
- 2 Ribbon: Main access point for tools and processes
- 3 Display window: Main area for visualizing and interactively working with data in 2D and 3D
- 4 Explorer panes: Organize pre- and post-modeling data (folders for data loading, browsing, and managing project content)
- 5 Status bar: Information about the progress of calculations

Copyright © 2017 Schlumberger. All rights reserved.

Petrel Explorer panes

Pre- and post-modeling data is managed in the Petrel **Explorer** panes, such as the **Input** and **Models** panes. You can unpin, drag, and dock panes in a new location or use them as floating panes that can be opened simultaneously by default.

The Petrel user interface has these panes:

- **Input**
- **Models**
- **Results**
- **Templates**
- **Cases**
- **Workflows**
- **Windows**
- **Favorites**
- **Tasks**
- **Message log**
- **Layouts**

When an item is bold (active) in a pane, Petrel recognizes it as the object to use. For example, if you have multiple 3D grid models in the **Models** pane and you want to create a property using Geometrical modeling for one of the 3D grid models, you must make that 3D grid model bold (active) for Petrel to recognize the object as the one to use. Data is stored in folders in the **Explorer** panes. To expand the folders, click the triangle. To collapse the folders, click the triangle again. The triangle is bold when the folder is expanded.

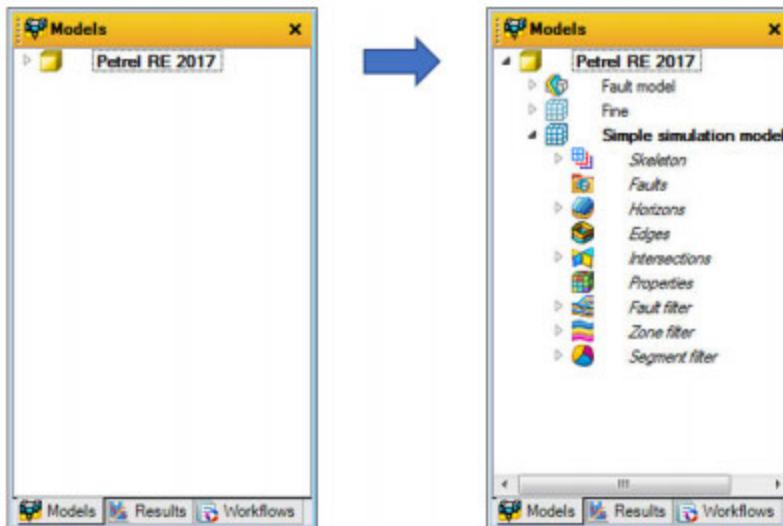


Figure 13. Petrel Explorer pane with data folders

Input pane

The **Input** pane stores imported data, such as wells, seismic, surfaces, lines, points, and SEG-Y seismic data. Output data of the same kind also is stored in this pane. For example, if a set of internally modeled faults is converted to polygons, the generated polygons are added automatically to the **Input** pane.

Models pane

The **Models** pane stores generated 3D, velocity, and fracture models, simulation grids, and the internally created data connected to them (faults, trends, and 3D grid properties). This pane also contains any imported 3D grids.

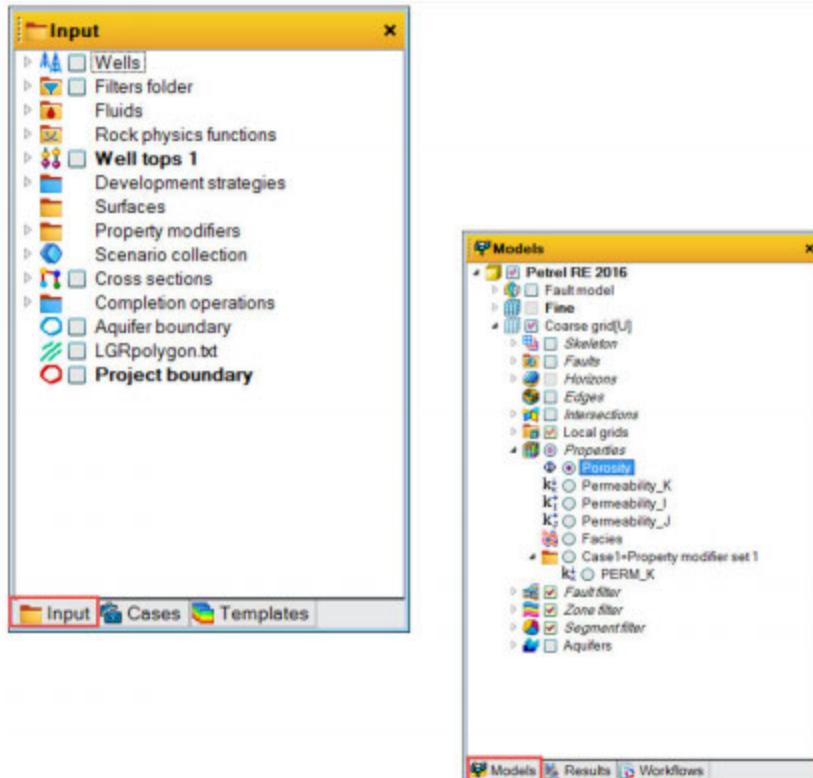


Figure 14. Input pane (left) and Models pane (right)

Copyright © 2017 Schlumberger. All rights reserved.

Cases pane

The **Cases** pane provides access to all cases defined for simulation and volume calculation.

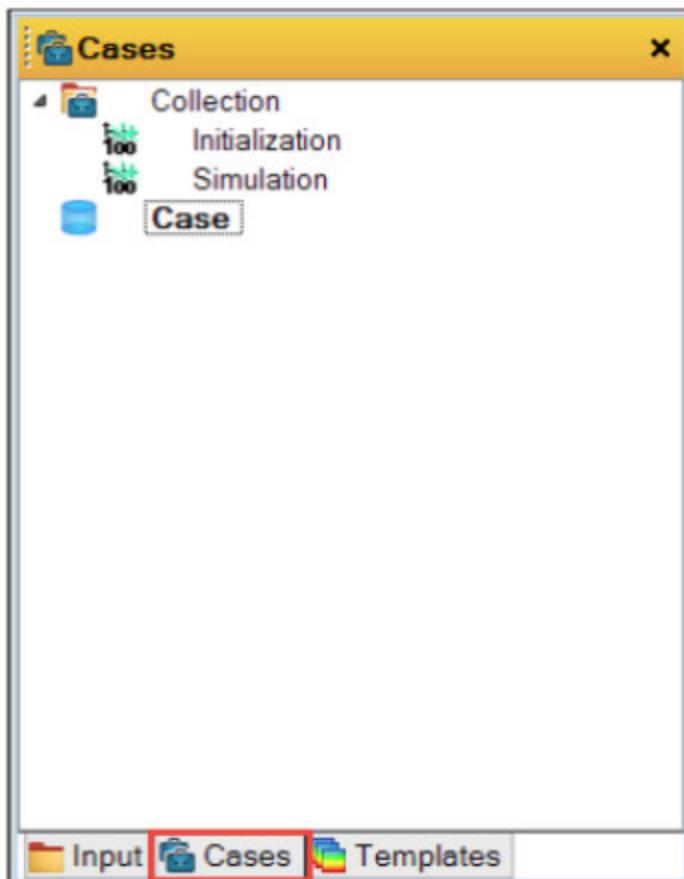
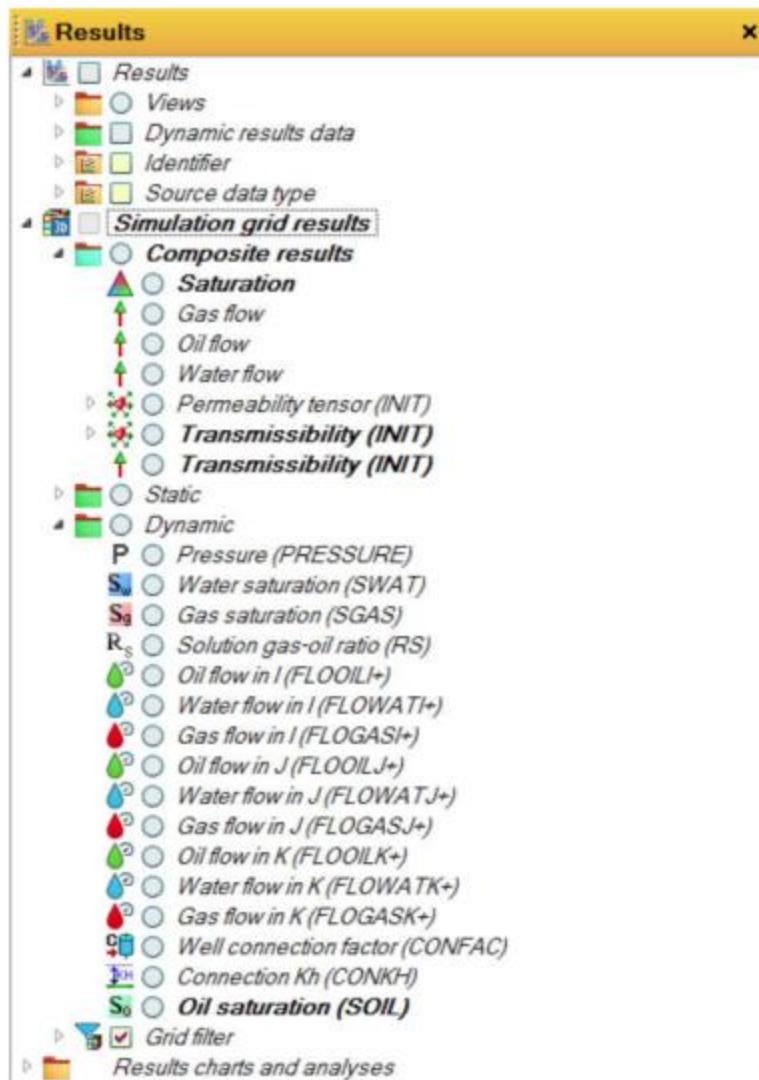


Figure 15. Cases pane

Results pane

The **Results** pane provides access to the numerical results from volume calculations and simulations.



Copyright © 2017 Schlumberger. All rights reserved.

Figure 16. Results pane

Workflows pane

The **Workflows** pane stores workflows created by the **Workflow editor** and the **Uncertainty and optimization** process. Workflows provide a programming-like user interface to Petrel. They allow you to automate tasks such as plotting or sensitivity studies.

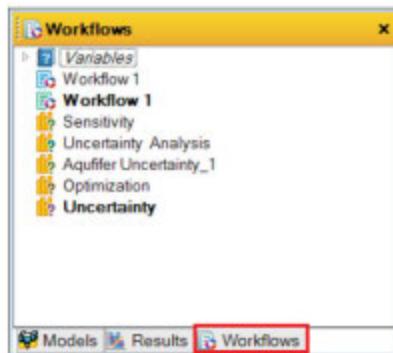


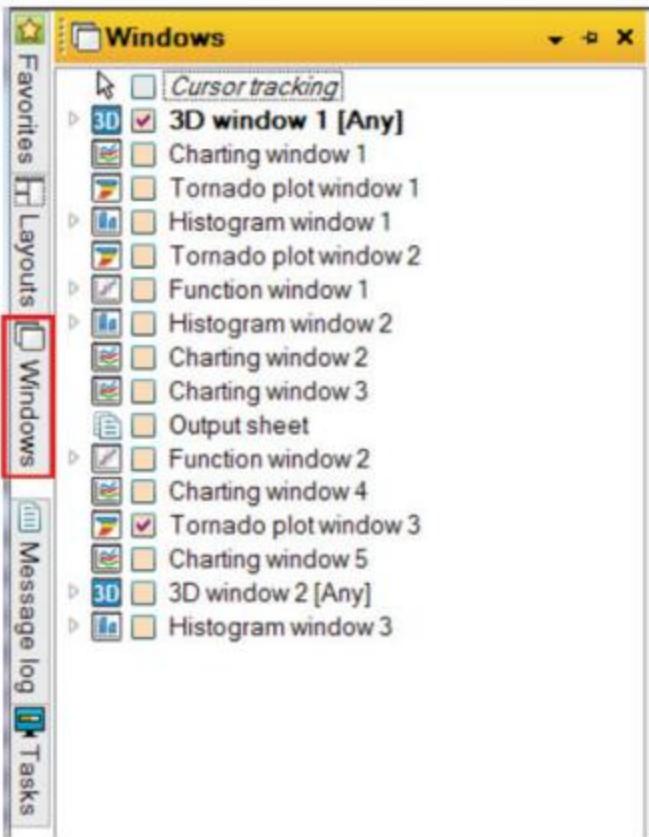
Figure 17. **Workflows pane**

Windows pane

The **Windows** pane stores all visualization windows in a project. If you close a window, it is still stored in the **Windows** pane. Windows created in the Petrel project are stored in this pane.

A selected check box next to a window indicates an open window. Having many windows open can slow down performance; therefore, it is recommended that you keep the number of open windows to a minimum. Each window is stored as a folder in the **Windows** pane.

In these folders are tools that you can use to visualize the legend and axis and to set the background color. These tools also are found on the toolbar.



Copyright © 2017 Schlumberger. All rights reserved.

Figure 18. Windows pane

Layouts pane

The **Layouts** pane allows you to create and save several display window layouts and easily switch between them. Use the layout feature to organize windows for specific workflows.

These multiple open windows can be tiled vertically, horizontal, and in grids. Double-clicking the window header allows you to minimize or maximize the window.

You can save layouts only if the **Enable tabbed windows** check box is selected on the **Effects** tab in the **System settings** dialog box. This option is selected by default.

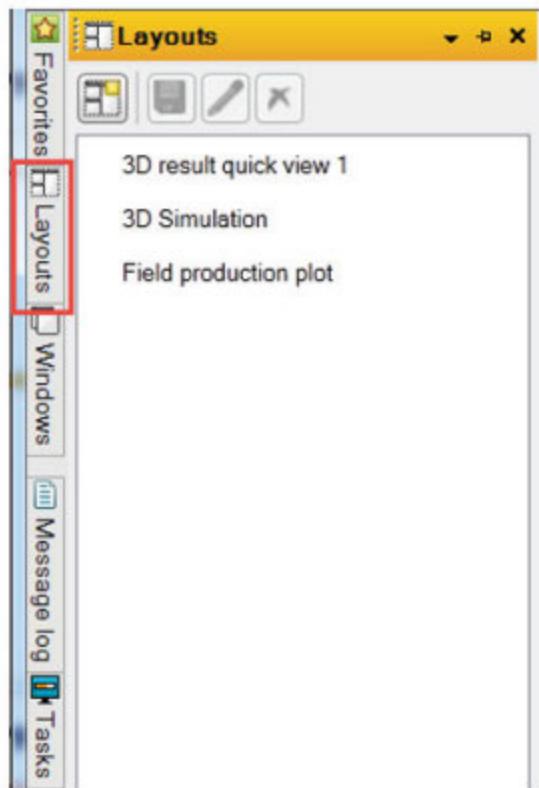


Figure 19. Layouts pane

Favorites pane

The **Favorites** pane allows you to build a list of shortcuts to the objects that you use most (for example, data, results, templates, and windows).

To add a shortcut, you can quickly drag an object into the **Favorites** pane. By default, this pane is docked to the left of the **Input** pane.

You can hide, float, and relocate this pane as you would any other pane. The location of the **Favorites** pane and the created shortcuts are saved when you close the project or close Petrel.

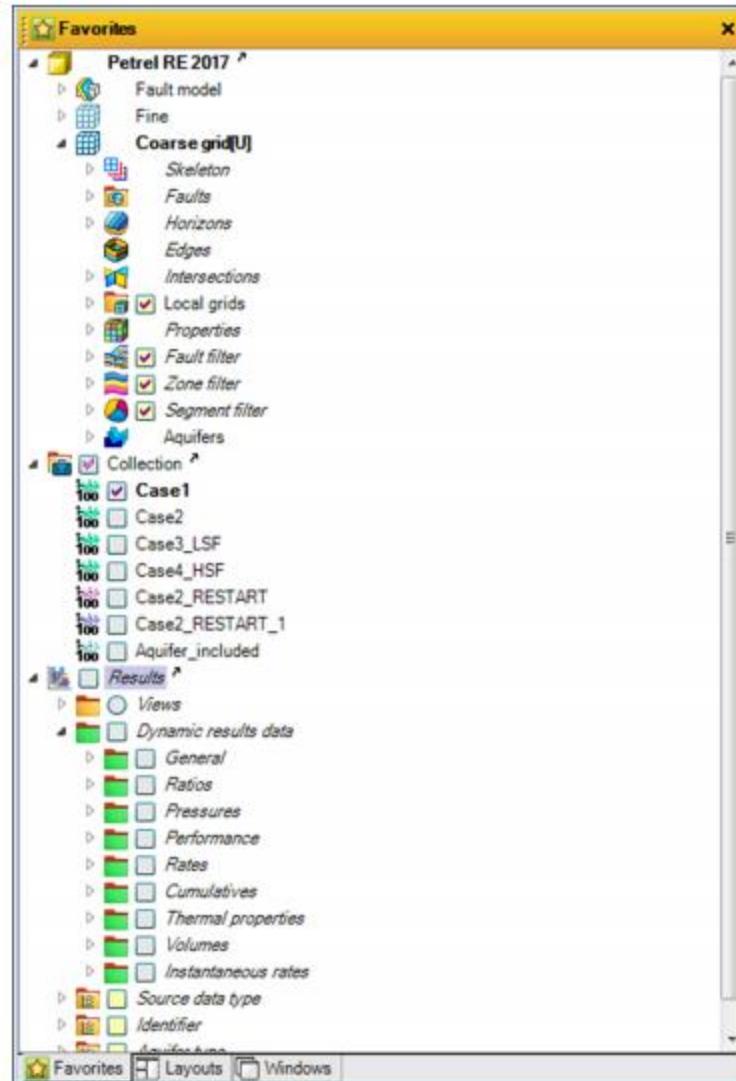


Figure 20. Favorites pane

Templates pane

Petrel objects are associated with data types. They are organized in folders in the **Templates** pane. You can modify all the templates, copy a continuous template to the other template folders, and add user-defined property templates to any of the property template folders in the **Templates** pane. The numeric precision for each template can be set, either in terms of the number of significant figures or the number of decimal places. A maximum precision of eight decimal places or eight significant figures can be set. The default precision for most templates is between two and four decimal places.

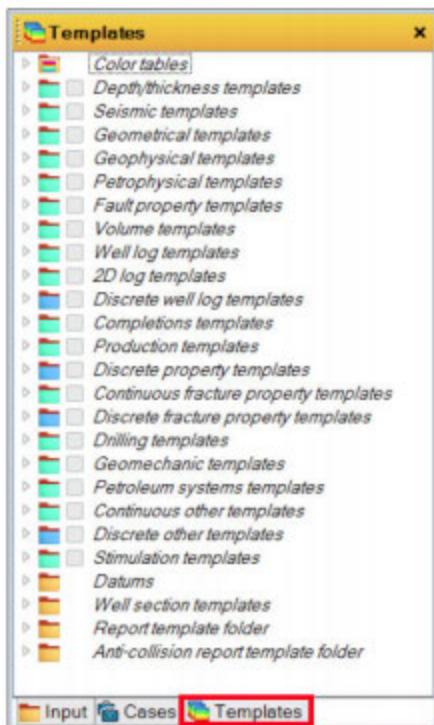


Figure 21. *Templates pane*

Typical pre- and post-processing reservoir engineering workflow

A typical reservoir engineering workflow starts with building a new 3D simulation grid and populating it with petrophysical properties. You also can upscale an existing geological model, add wells and well controls, generate fluid and rock physics functions, and then submit the model to the simulator using the **Define simulation case** dialog box. You can review and analyze simulation results using the available result analysis and visualization tools in Petrel.

If there is need for uncertainty assessment and optimization tasks, Petrel offers a set of uncertainty and optimization tools on the **Simulation** tab in the **History match and optimization** group.

Also, the well-defined and structured Reservoir and Production domain driven workflows in the **Perspective** tool give you access to all the available tools required for pre- and post-processing of your data in the Petrel modeling environment.

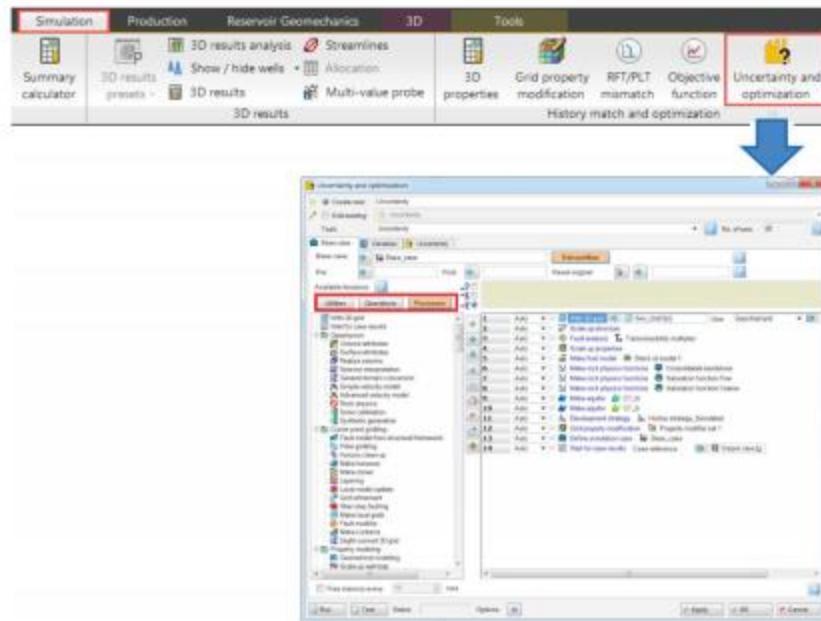


Figure 22. Accessing the Uncertainty and optimization process

These typical reservoir engineering workflow tasks can be performed using the tools available in the **Reservoir and Production** perspective:

1. Import or build a new simulation grid.
2. Define fluid and rock properties and set the boundary conditions.
3. Calculate reserves.
4. Import or create wells and set up completions.
5. Link your reservoir to facilities with the help of well deliverability tools and create a field development strategy for simulation.
6. Define and run a simulation case and analyze simulation results.
7. Perform history matching and manage uncertainty/optimization.
8. Convert the imported ECLIPSE* simulation deck to a Petrel case and create a restart case and sector model.



Copyright © 2017 Schlumberger. All rights reserved.

Ribbon and other interface elements

Petrel processes or workflow steps and utilities are organized on a ribbon. The Petrel ribbon has these elements:

- The **File** tab and **Home** tab make up the core tabs. The contents or tools on these tabs do not change. They are not domain-specific.
- The domain tabs are non-contextual tabs in the ribbon that lay out a distinctive workflow from left to right in the sequence in which it typically occurs.

- The **Home** tab contains the project management, visualization, and annotations tools. You also can access the **Perspective** tool that you use to filter the ribbon to display only domain tabs.
- The **Perspective** tool gives you access to the predefined domain workflows (Seismic to Simulation, Geology and Geophysics, Advanced Geophysics, Reservoir and Production, Shale, Drilling, and Data Management).



Figure 23. Predefined domain workflows accessed from the Perspective tool and the domain tabs

- 1 Domain tabs
- 2 Perspective tool



Figure 24. Workflows on the Reservoir Engineering domain tab with tools logically organized in groups from left to right

The **Status bar** displays information about the selected object in the active display window.

The **Inspector** is a floating window that provides detailed information about the objects that you click. You also can adjust style and other settings without leaving the display window.

The **Message log** shows a log of all actions done in the Petrel session.
You can pin and unpin the **Message log** like other Petrel panes.

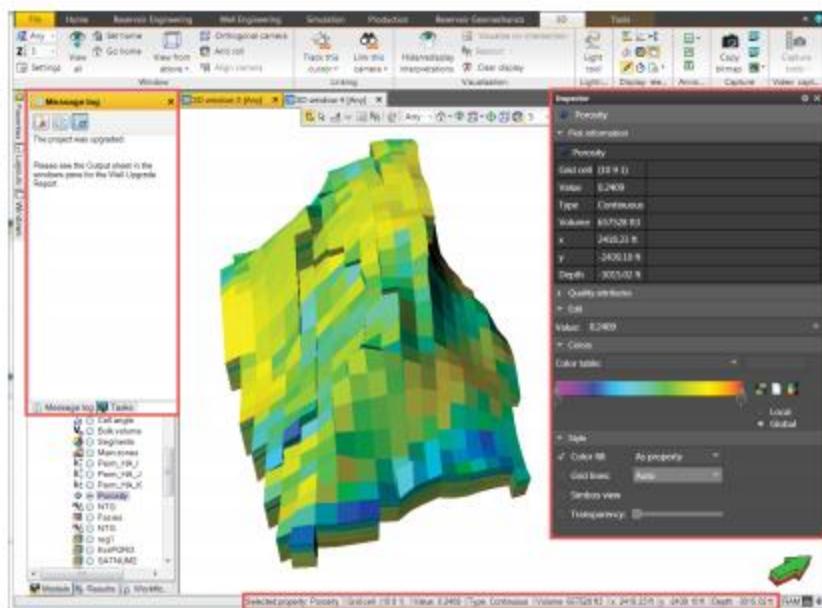


Figure 25. The Message log, Status bar, and Inspector

The mini toolbar and shortcut menu contain formatting tools for the displayed object in the window. The mini toolbar opens automatically with the shortcut menu when you right-click a displayed object.

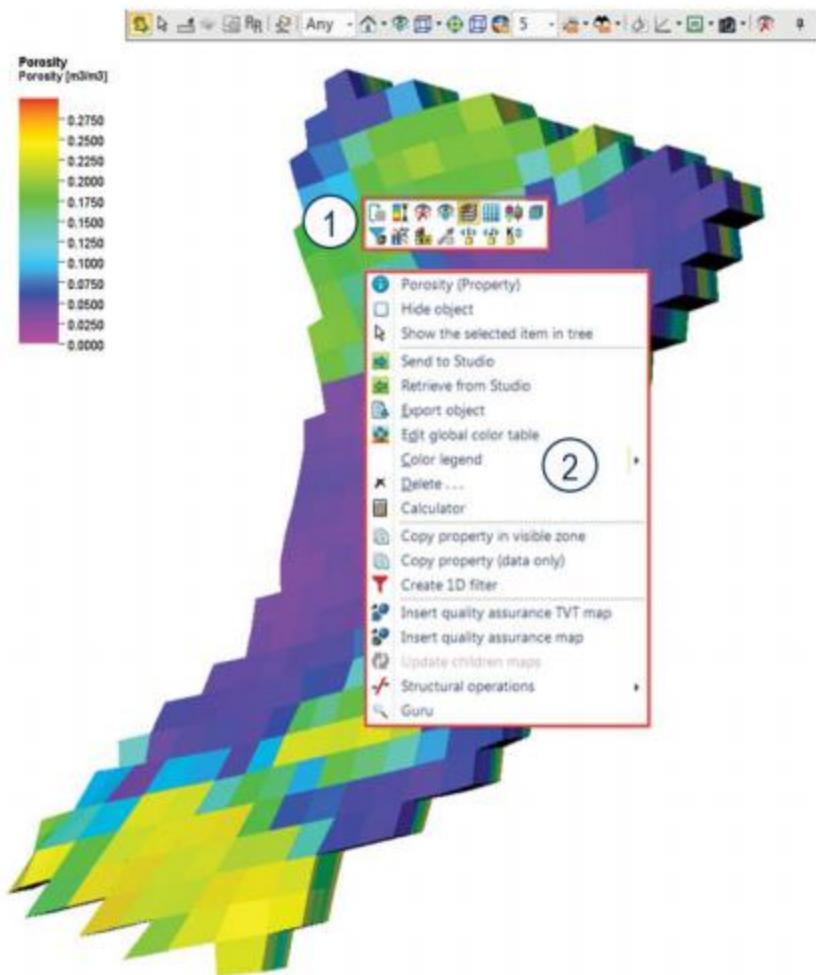


Figure 26. Shortcut menu and mini toolbar

- 1 Mini toolbar
- 2 Shortcut menu

Contextual tabs appear on the ribbon above the core domain tabs whenever you activate a window or select an object. They are colored differently from the core tabs.

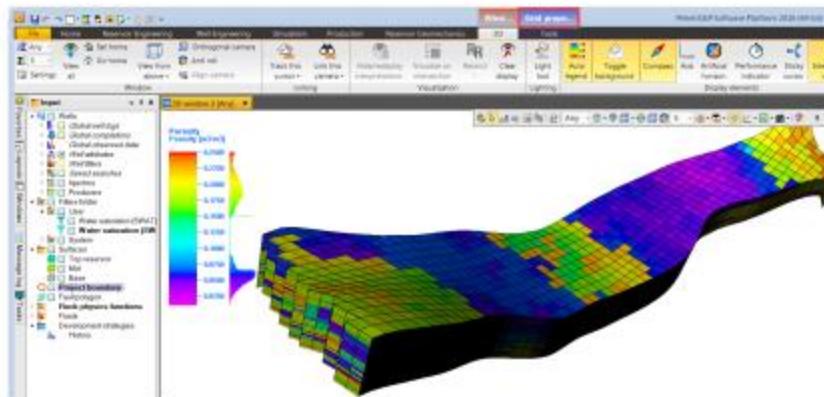


Figure 27. Contextual tabs

The **Window** toolbar shows the most frequently used interactive tools for the active window type. To hide the toolbar, click .

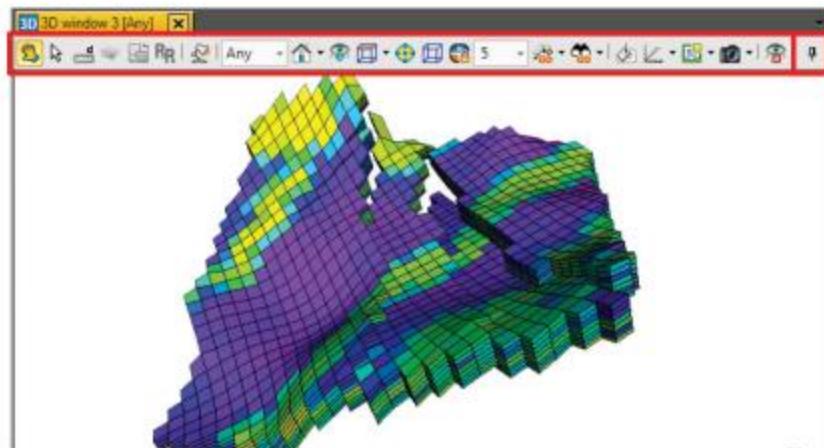


Figure 28. Window toolbar

The Quick Access Toolbar is a collection of frequently used tools in the title bar. By default it contains **Save**, **Undo**, **Redo**, **New window**, **Color table**, **Inspector**, **Players**, **Copy**, **Paste**, and **Customize** tools.

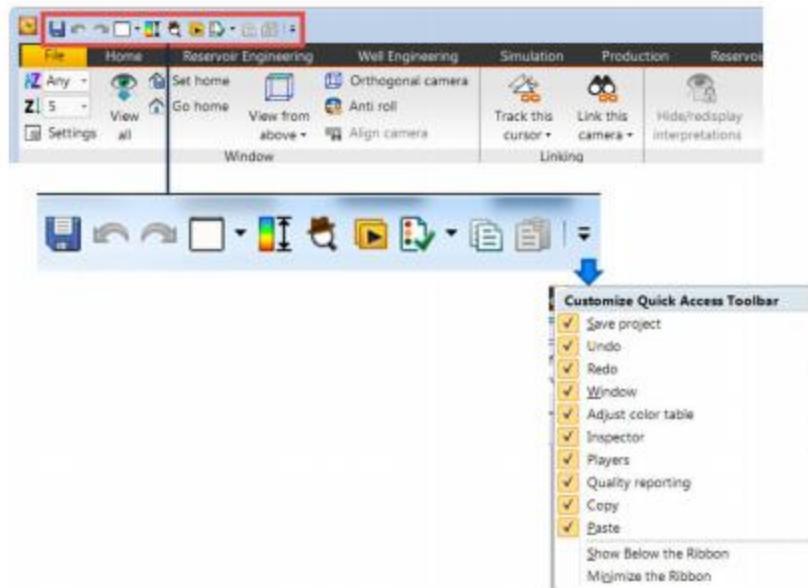


Figure 29. Quick Access Toolbar

You can add tools to the Quick Access Toolbar. Right-click any tool button and click **Include in Quick access toolbar**.

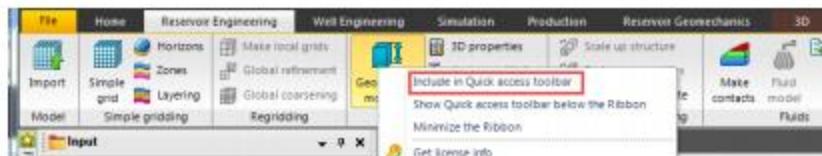


Figure 30. Command to add a tool to the Quick Access Toolbar

The **Tool Palette** is a floating window that indicates which interactive tool is active. It allows you to select the tools necessary to complete your workflow without leaving the display window.

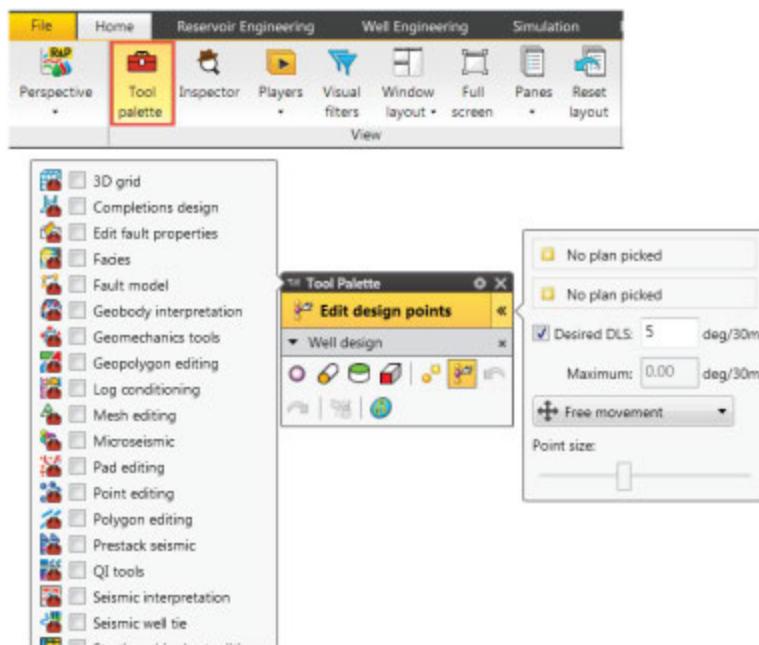


Figure 31. Tool Palette

The **Visual filters** pane provides a centralized view of the filters that are applied to the wells, 3D properties, and faults displayed in the active **2D window** or **3D window**.

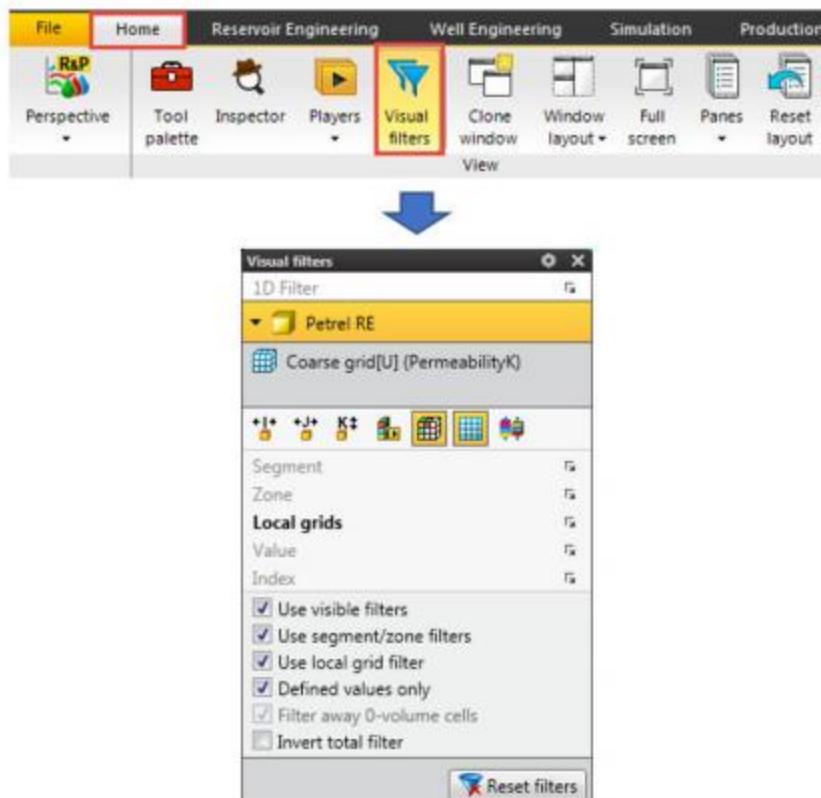


Figure 32. Visual filters pane

Searches in Petrel

The Ctrl+F search functionality provides quick searching, selection, and access to Petrel project data and processes found in the panes. It allows you to conveniently access project data and processes without using the **Explorer** panes.

You also can access the search feature by clicking **Search project** on the **Home** tab in the **Search** group.



Figure 33. Access search functionality on the Home tab

Selecting a result in the **Search** dialog box automatically opens the associated process dialog box and the data object.



Figure 34. Search functionality

Find (Ctrl + shift + F)

Search with Studio in Petrel , also known as **Find**, allows you to search, view, filter, and retrieve information natively in Petrel across data sources, without the need to know where the data is stored. A data source is any application in which information can be read and indexed to make it available to users in Petrel. Examples of data sources include a Petrel project, a Studio repository, and a Techlog project.



Figure 35. Search with Studio (Find) on the Home tab

Find works based on the Petrel Search settings. To open the Search settings, from the **File** menu, click **Options**.

By default, **Find** opens the **Search filters** pane, a **2Dwindow**, and the **Search results** pane. If a **3D window** already is open, the results appear in the active **3D window**. You use the filters in the **Search filters** pane to narrow your search. The results appear in the **2Dwindow** or **3D window** and the **Search results** pane.

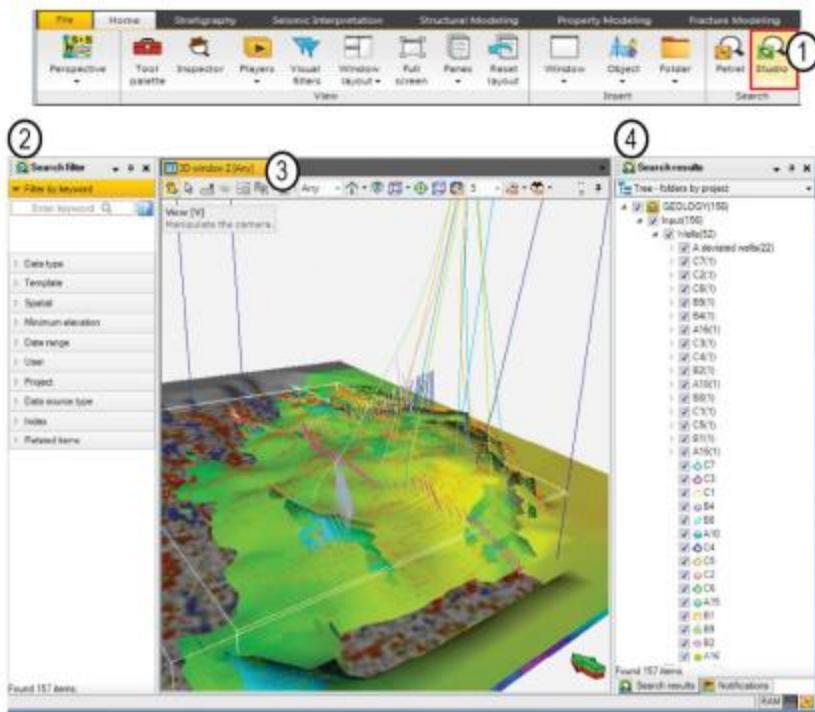


Figure 36. Find in Petrel

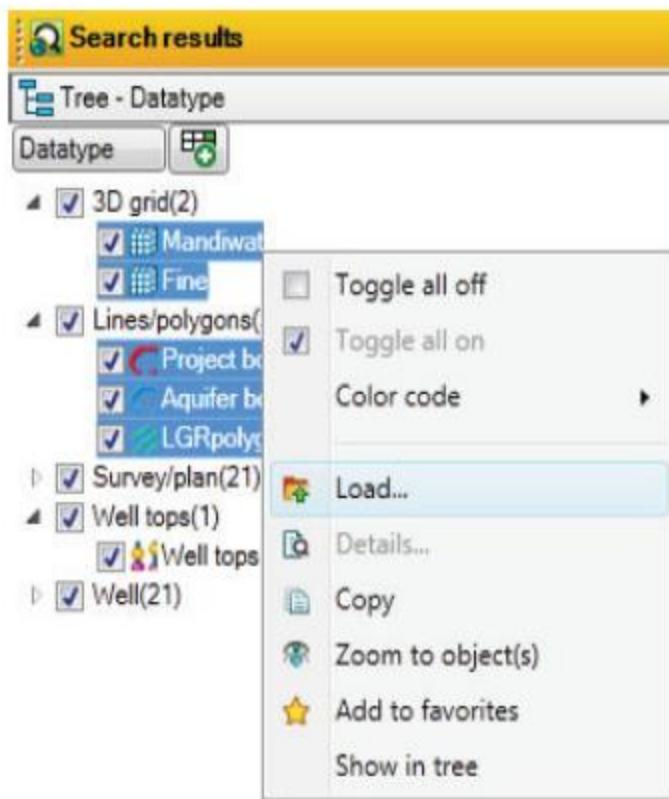
- 1 **Find**
- 2 **Search filter** pane
- 3 Search results in a **3D window**
- 4 **Search results** pane

When you find the data items that you need, you can load them directly into your Petrel project from the **Search results** pane. Because the data items belong to data stores such as a Studio repository, you are requested to log in to the data store. The data manager in your organization provides all the required information.

Procedure — Load data from Search results



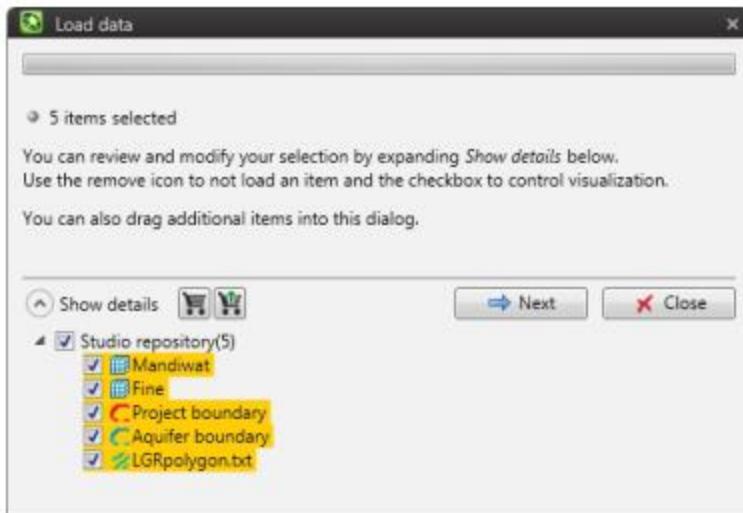
1. In the **Search results** pane, right-click the data items that you want to retrieve and click **Load**.



2. Optional: In the **Load data** dialog box, select **Show details** to see the details of the data being loaded.

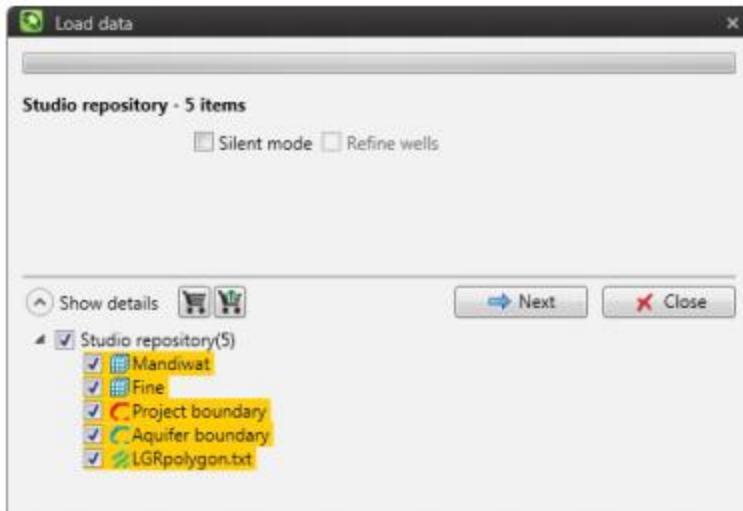
Copyright © 2017 Schlumberger. All rights reserved.

3. Click **Next**.



4. Optional: If you want to further refine the selection of data items to load, clear the **Silent mode** check box.

5. Click **Next**.



6. If the data items are stored in Studio, enter your login parameters and click **OK**.

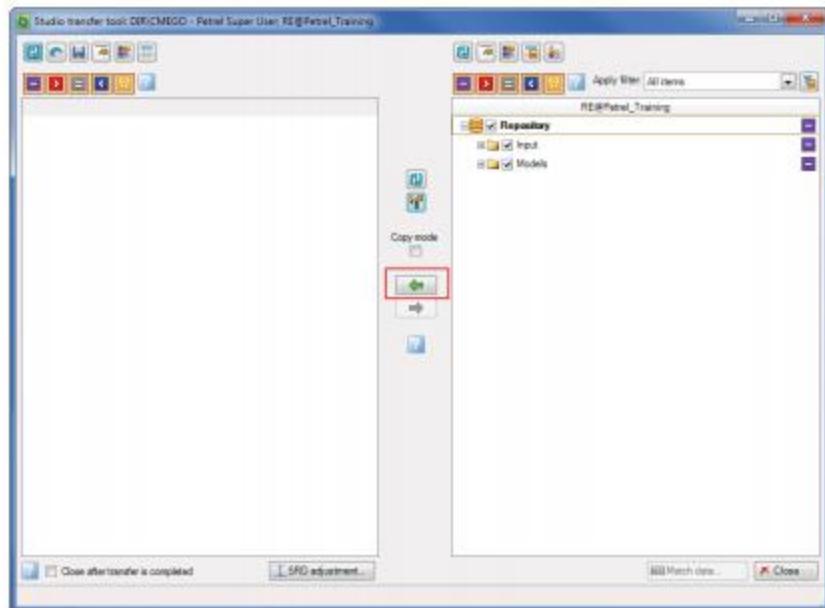
The login connection parameters are defined by the administrator.
Usually, you do not need to change anything in the **Login** dialog box.



If you cleared the **Silent mode** check box, the **Studio transfer tool** opens.

7. In the **Studio transfer tool**, review and refine your selection of data items as necessary.

8. To retrieve the selected items to your Petrel project, click .



9. Close the **Load data** dialog box.

Generic Windows and specific Petrel keyboard shortcuts

Shortcuts allow you to quickly navigate through Petrel without extensive use of the mouse, which helps you improve efficiency.

To access the Windows auto-generated keyboard shortcuts, press Alt.



Figure 37. Auto-generated keyboard shortcuts

Petrel also supports standard Microsoft Windows keyboard shortcuts. More keyboard shortcuts are listed in the Help Center. This figure shows examples of Petrel-specific keyboard shortcuts.

Key Shortcut	Function
Esc	(2D and 3D windows) Toggles between View  and the last selected action
V	Activates View 
Z	(2D and orthogonal 3D windows) Activates Magnify 
P	Activates Select 
Shift + Esc	(2D and 3D windows) Toggles between View  and Select 
arrow keys	In View  mode, scrolls the view of an item in the display window
Home	Brings the displayed item back to home position if the Set Home  tool was used
S	In View mode in a 3D window, activates Zoom 
Ctrl + U	Activates View all  and centers data in the middle of the display window
Ctrl + J	Activates the selected view position (for example View from above 
Ctrl+Tab	Opens a short cut menu for panes and display windows
F11	Hides the panes and minimizes the ribbon
Shift + F11	Minimizes the panes and the ribbon

Figure 38. Examples of keyboard shortcuts

Coordinate systems in Petrel

Petrel supports loading, exporting, and transferring point-based data and bin-grids in a spatial context. Objects defined in a specific Coordinate Reference System (CRS) can be converted or transformed to another CRS.

For work done at a reservoir level, spatial awareness is not a prerequisite. It is possible to set a project to be spatially unaware (for example, Undefined CRS). However, if you are working in a Studio environment, your Petrel project must have a CRS set. Setting a CRS is required when you transfer data between Petrel and Studio. When data is transferred, the CRS for the data item is converted to the CRS of the target Petrel project or Studio repository.

These data items are supported for coordinate conversion:

- Point-based data:
 - Points (points with attributes)
 - Polygons
 - 2D seismic lines
 - 2D seismic interpretation

- Faults
- Well data (Well Head, Deviation surveys (Grid/True north), Markers, Well point data)
- Surface images (cf. RPT)
- Bin-grids:
 - 3D seismic cube (SEG-Y and ZGY)
 - 3D seismic interpretation
 - Regular surfaces

Grid data always retains its original CRS (OCRS) during its life cycle. No CRS conversion is made for grid data transfers. However, when grid data is displayed in a **2D window** or **3D window**, the system performs on-the-fly conversion.

The Petrel software uses the Esri cartographic engine to perform the conversions/transformations. An external application named the Coordinate System Manager (CSM) enables you to create early bound CRSs. (For more details on the procedure, refer to the CSM topics in the Petrel Help Center.) Spatial enablement also introduces latitude and longitude support for read-outs, statistics, and maps with latitude and longitude lines.

Copyright © 2017 Schlumberger. All rights reserved.

Project settings

As soon as a new Petrel project is created, you must record project information and units. Ensure that the data is imported with the correct units. It is not possible to convert units of data already imported into the project, but you can convert data during the import and export processes.

NOTE: There is NO unit or coordinate conversion in Petrel when you change the unit/coordinate system; the conversion must be done before or when you import/export the data. However, you can set or customize the unit system used in dialog boxes and spreadsheets in the **Reservoir engineering unit settings** dialog box.



Reservoir engineering unit settings

The **Reservoir engineering unit settings** allows you to use three default reservoir engineering unit system (ECLIPSE - Metric, Field and Lab) for your simulation unit and also variable units by setting the default units system to "**Project**". You can use the **Reservoir engineering unit settings** dialog box to independently set or customize the unit system in supported processes and spreadsheets.

If you want to use the processes with variable units in the **Workflow editor** or Uncertainty and optimization workflows, set the Default reservoir engineering unit system to 'Project' and remove any customized measurements .

These dialog boxes and spreadsheets currently support flexible units:

- **Development strategy** dialog box
- **Field management** dialog box
- **Make fluid model** dialog box
- **Make rock physics functions** dialog box
- **VFP manager**
- **Make initial conditions** dialog box
- **Initial conditions from maps** dialog box
- **Settings for 'ECLIPSE network'** dialog box
- Rock physics spreadsheet
- Fluid spreadsheet
- Observe Data spreadsheet
- Make aquifer
- Separator modeling
- Define simulation case- Grid tab
- Thermal boundary condition process

Copyright © 2017 Schlumberger. All rights reserved.

Procedure — Set project settings and units



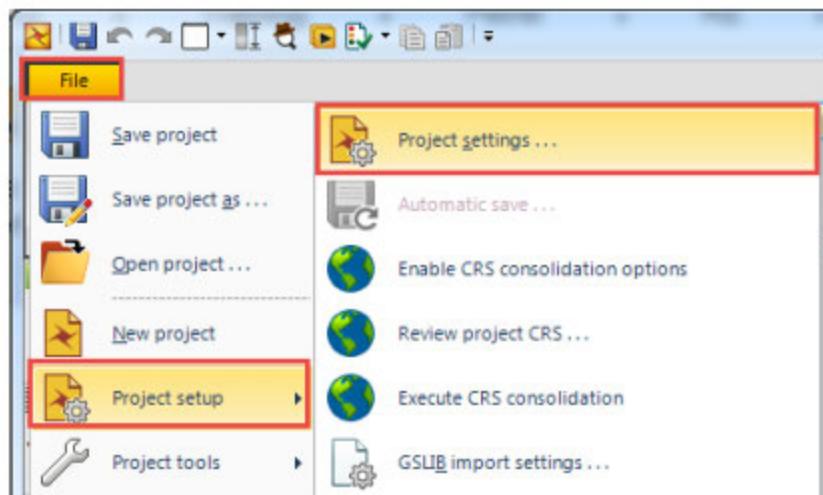
Follow these best practices when recording project information and setting units:

- Check your data before you import it and decide on the project unit to be used. Petrel allows you to use local coordinates, field, metric, or a combination.



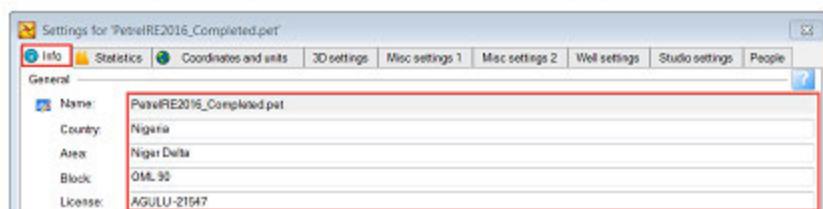
NOTE: The default setting for units is Metric.

- When you create a Petrel project, open the project **Settings** dialog box and select the units you want to use.
 - Before importing an object, verify the units of the data in the file and select a unit conversion in the **Import** dialog box, if necessary.
 - If the units of a data object are inconsistent with other data in the project, delete the object and re-import it with the correct conversion.
1. On the **File** tab, click **Project setup** to access the project settings.



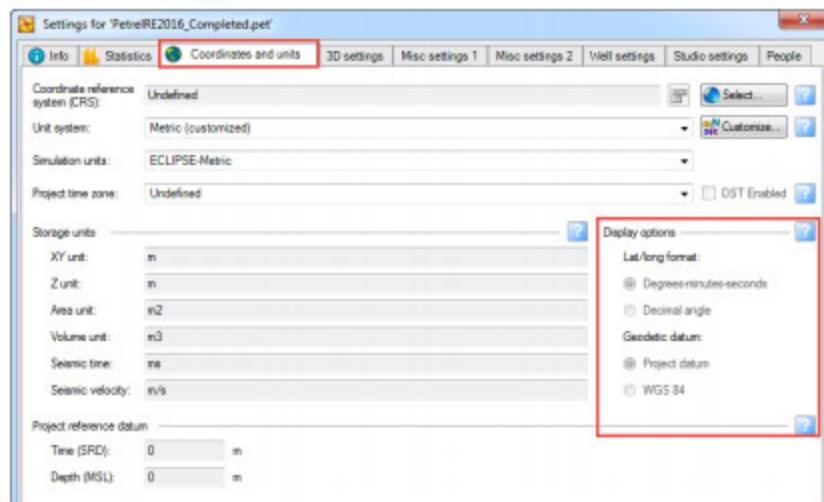
Copyright © 2017 Schlumberger. All rights reserved.

2. In the **Settings** dialog box, on the **Info** tab, enter project information.



3. On the **Coordinates and units** tab, set up the projection and units:
 - a. Click **Select** and select a coordinate system to use.
 - b. To activate the Petrel Spatial awareness, turn on lat/long format.

- c. Select a standard Unit system from the list (**Metric** or **Field**) or click **Customize** to set units from a mixed unit system.



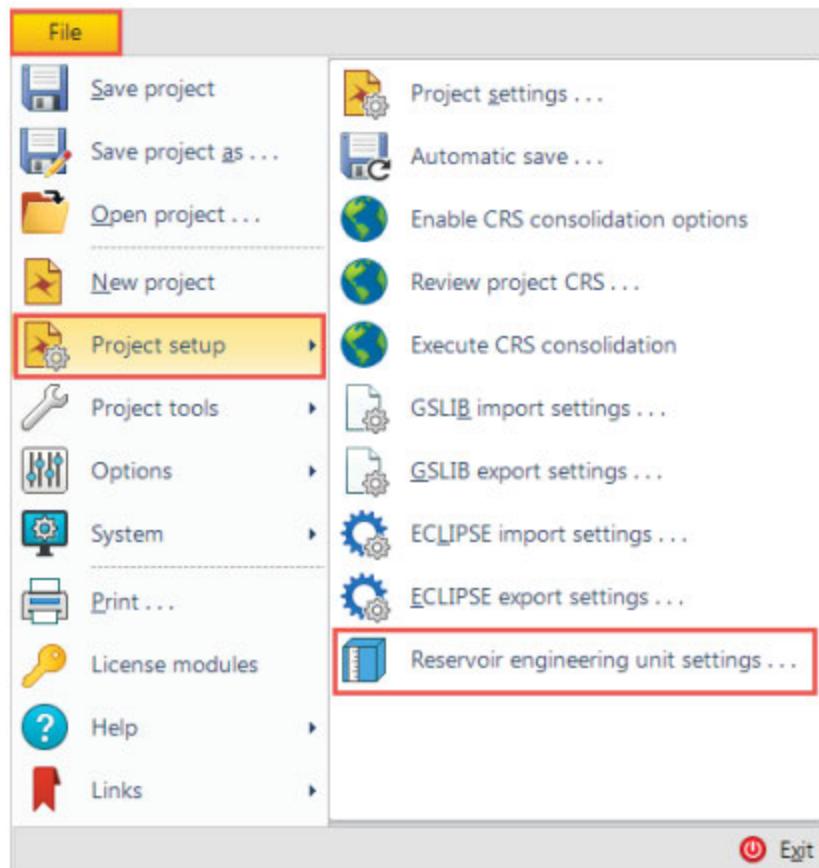
Copyright © 2017 Schlumberger. All rights reserved.



Procedure — Set the reservoir engineering unit system

This procedure describes the steps to customize the unit system in the supported dialog boxes and spreadsheet using the **Reservoir engineering unit settings** dialog box.

1. On the **File** menu, click **Project setup** and then click **Reservoir engineering unit settings** to open the **Reservoir engineering unit settings** dialog box.



Copyright © 2017 Schlumberger. All rights reserved.

- From the **Default reservoir engineering unit system** list, select the unit system that you want to use. The default unit system can be Project, ECLIPSE-METRIC, ECLIPSE- FIELD, or ECLIPSE- Lab.

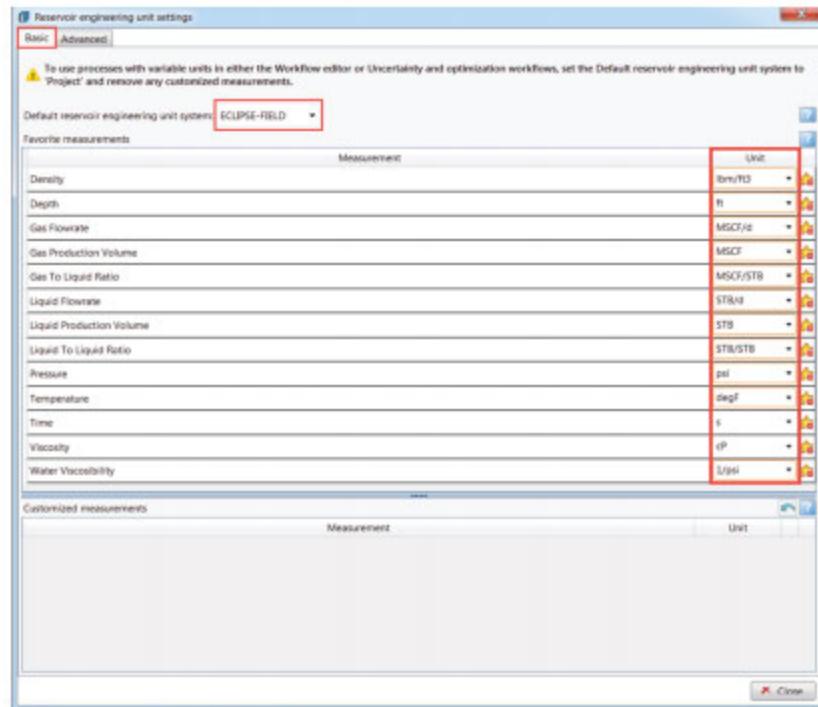


If the current unit system includes customized measurements, you are asked if you want to clear these measurements. Click **Yes** to clear all customizations, or **No** to retain your customized measurements.

Units displayed in the supported dialog boxes and spreadsheets are converted automatically to the selected unit system.

However, if you select a unit system that is different from the Petrel project unit, an orange border is displayed around unit lists in supported dialog boxes, spreadsheets, and in the **Favorite**

measurements section of the **Reservoir engineering unit settings** dialog box.



3. To save the changes and close the dialog box, click **Close**.

Procedure — Change the default unit for a measurement

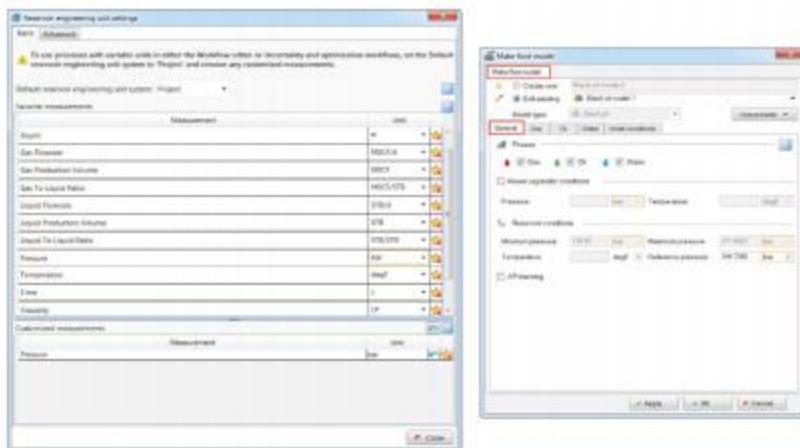
This procedure explains how to customize project units using the **Reservoir engineering unit settings** dialog box.



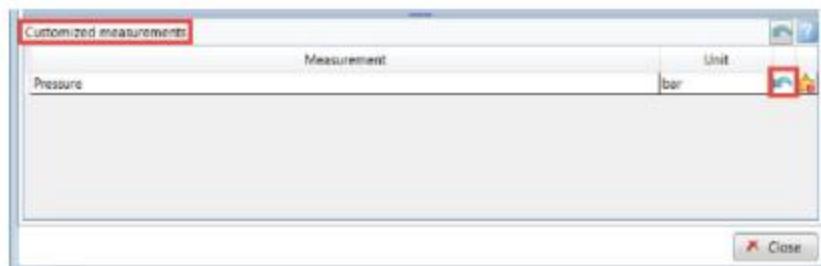
1. On the **File** tab, click **Project setup** and then click **Reservoir engineering unit settings** to open the **Reservoir engineering unit settings** dialog box.
2. Locate the unit, either in the Favorite measurements table on the **Basic** tab or on the **Advanced** tab.
3. Select the required default unit from the **Unit** list for a measurement.

The default unit is updated automatically in all supported dialog boxes or spreadsheets that use the selected unit.

For example, changing the default unit for Pressure to bar automatically changes the units for Pressure values to bar on the **General** tab in the **Make fluid model** dialog box.



Changing the default unit adds the changed measurement to the **Customized measurements** section of the **Customize reservoir engineering unit settings** dialog box. To restore this unit to its default unit for the selected unit system, click **Revert to default unit**.



4. To save the changes and close the dialog box, click **Close**.

You can change units locally in supported dialog boxes and spreadsheets by choosing a different unit from the list next to a measurement. These changes are saved only if **Synchronize default units with local unit changes** is selected on the **Advanced** tab in the **Customize reservoir engineering unit settings** dialog box.

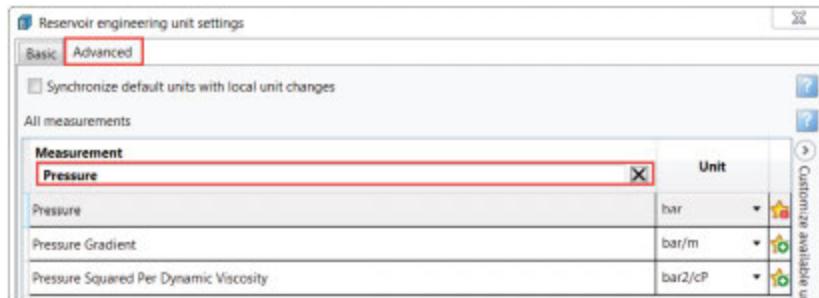
The default reservoir engineering units for those measurements in all other supported dialog boxes and spreadsheets also are updated.



Procedure — Define the available units for a measurement

Use the **Customize available units** option to define the choice of units available for selected measurements in dialog boxes and spreadsheets that support flexible unit settings.

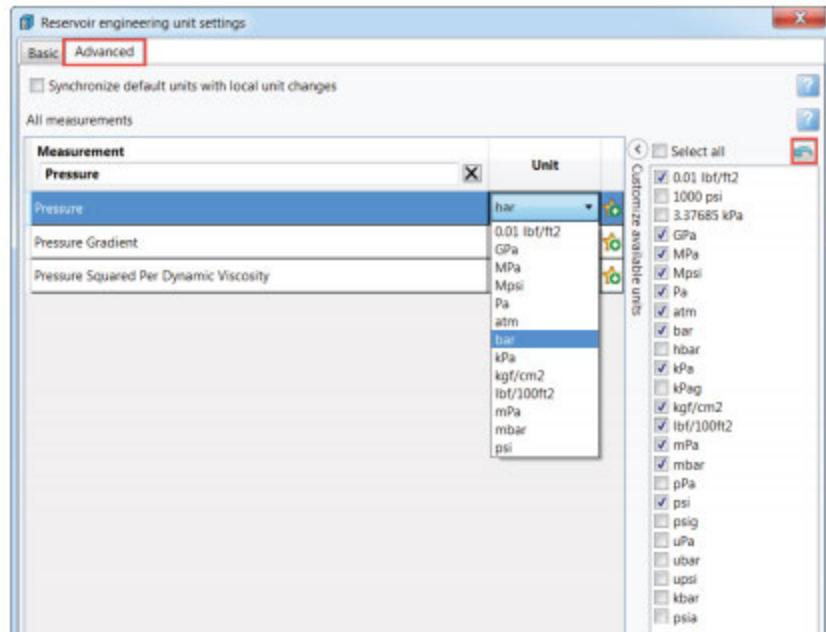
1. On the **File** menu, click **Project setup** and then click **Reservoir engineering unit settings** to open the **Reservoir engineering unit settings** dialog box.
2. On the **Advanced** tab, click **Customize available units**.
3. Select the measurement that you want to change. Use the search box at the top of the measurement column to locate a measurement of interest.



Changes to a selected measurement affect all equivalent measurements. Point to a measurement label in the table to display a list of all equivalent measurements.

4. Select the check box next to a unit to include it in the choice of units for the measurement available in supported dialog boxes and spreadsheets.

To remove a unit from the **Unit** list in a supported dialog box or spreadsheet, clear its check box. If a unit is set as the default unit of measure, you cannot remove it.



5. To restore default unit selections, click **Revert selected units to the original list** .
6. If you want to change the default unit for the selected measurement frequently, click **Add to favorites**  to add the selected measurement to the Favorite measurements table on the **Basic** tab.
7. Click **Close**.

Object settings

The **Settings** dialog box provides information about the object and options for altering the objects. The tabs vary by object type.

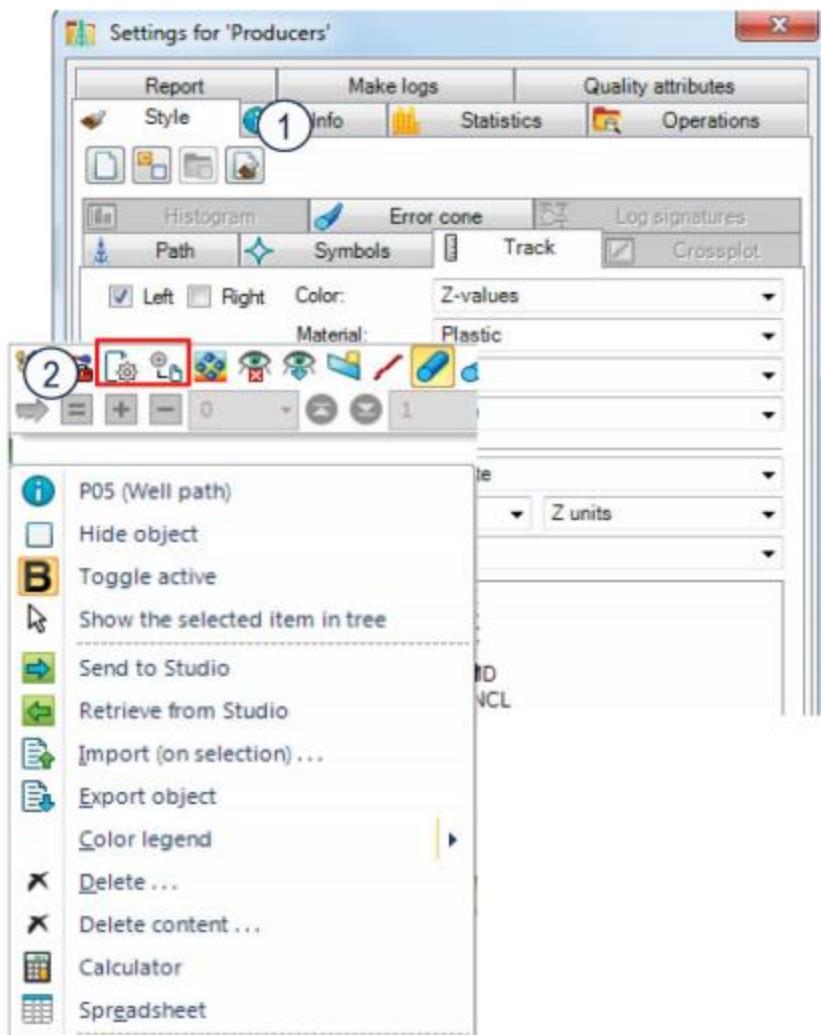


Figure 39. Accessing the object settings

- 1 Every object has a **Settings** dialog box.
- 2 To access the settings from the mini toolbar, right-click any displayed object.

Alternatively, you can right-click any object in the panes to access the settings.

The **Settings** dialog box displays different tabs and information. Depending on the type of object, more tabs are added for more functionality.

The **Settings** dialog boxes always include an **Info** tab and a **Statistics** tab.

Style tab

The **Style** tab is active only when a display window related to the Style options is active, for example, **2D** vs. **3D** display windows.

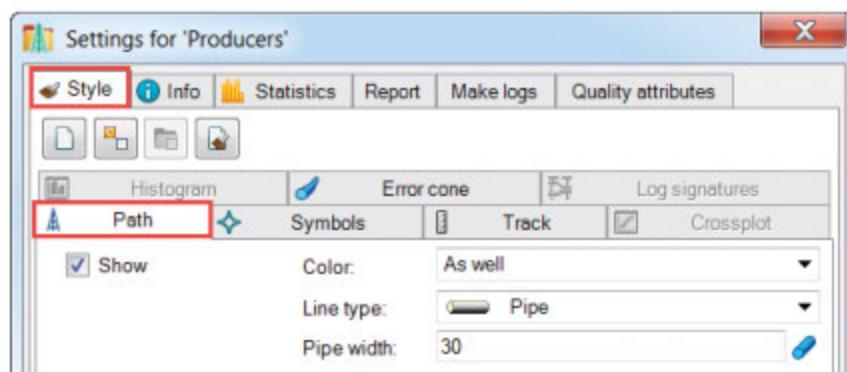


Figure 40. Style tab in the object settings

Statistics tab

It is important to quality check (QC) the statistics for accuracy and to verify that the items have the correct values.

By default, Z values are negative based on a reservoir that is below sea level. If your data is above sea level, select the appropriate options when importing your data. Petrel interprets increasing depth as increasing negative depth values, as opposed to GeoFrame* and ECLIPSE/INTERSECT*, which interpret increasing depth as increasing positive Z values.

The first list gives the X, Y, Z coordinates. If an attribute is available, it also is shown.

Generate a report from the **Statistics** tab by selecting the **Copy to output sheet** check box. Select one of the lists (for example, List 1) and click **output**. The contents of the list can be saved to a file or copied and pasted into an Excel spreadsheet.

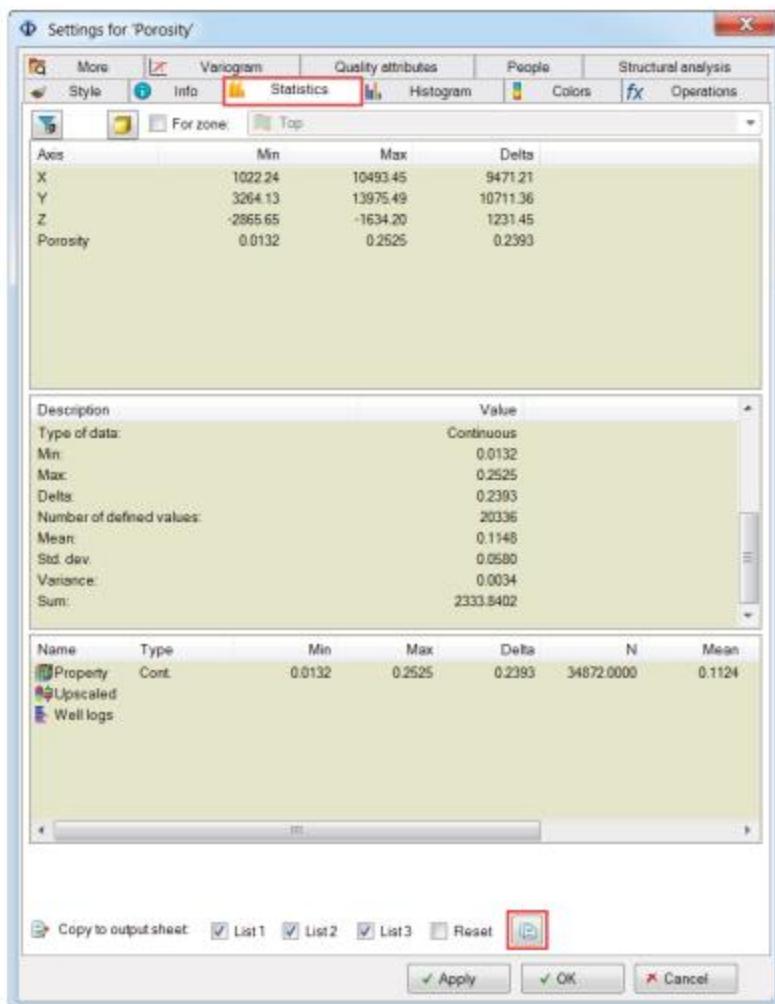


Figure 41. Statistics tab

Info tab

On the **Info** tab, you can rename the object and change the template (accessed by right-clicking the object and clicking **Settings**).

The **Comments** and **History** subtabs are on the **Info** tab of all **Settings** dialog boxes in Petrel. They provide a way to track edits made to an object and decisions made during the project.

The **Comments** subtab is an editable area where you can add information, such as the source of the data that was imported and its reliability. The name of the Petrel data file for the object and the name of the file that was imported (if applicable) are displayed beneath the **Comments** section.

The **History** subtab stores the history of the object. This history includes information about edits and operations performed on the object. By right-

clicking in this field, you can add your own date stamped comments or clear the history completely. Some objects do not have a **History** tab.

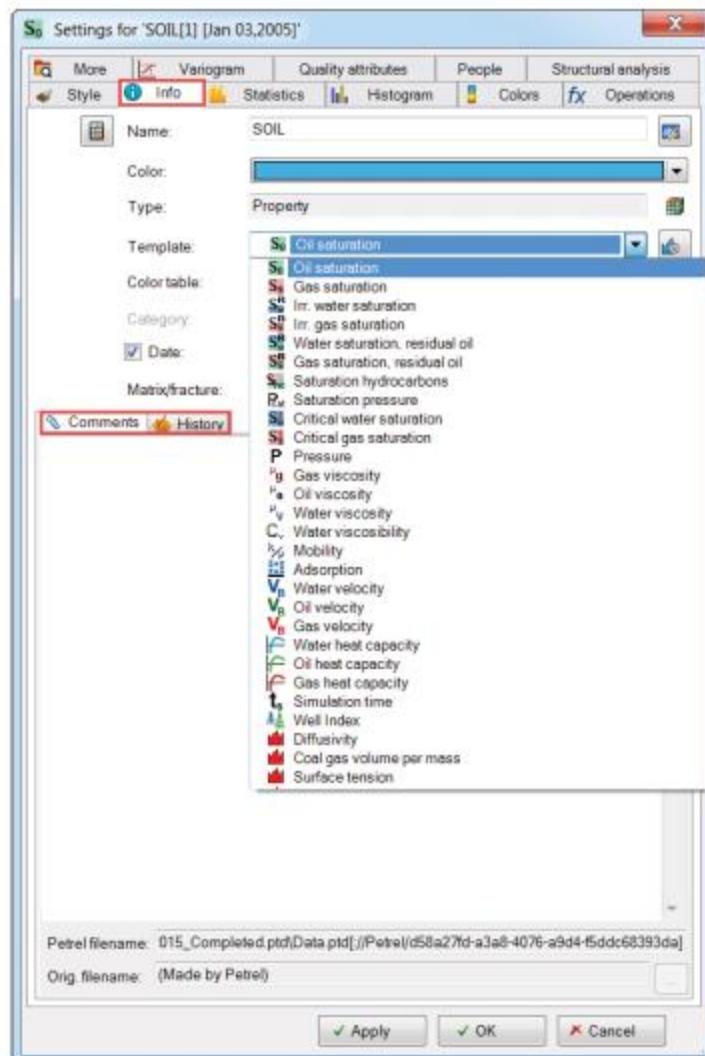


Figure 42. Info tab

Visualization: Windows

The most efficient way to QC the imported data or any other object is to visualize it.

Many types of windows are available.

- **3D window:** Visualize data in 3D.
- **2D window:** Visualize data in 2D. This window is useful when working with polygons and when you want to verify the object that you are viewing from overhead.
- **Map window:** Plot horizons and layers of the 3D property.
- **Intersection window:** Used in the **3D window** to visualize the faults with a 3D grid when analyzing a fault throw effect. It allows visualization of multiple data.
- **Interpretation window:** **2D window** for seismic interpretation.
- **Histogram window:** Normally used as a results analysis tool for plotting various data, such as volumetric results, 3D properties, and simulation results.
- **Function window:** Plot crossplots, variograms, and line plots.
- **Charting window:** Plot simulation results. Suitable for vector-type plots only.
- **Well section window:** Define Well correlation, log interpretation, and wellbore completions.
- **Plot window:** **2D window** used to display intersections, diagrams, functions, plots, maps, and 2D interpretation. Multiple viewports can be inserted (for example, function, intersection, and histograms together). A viewport is a limited rectangular area in which data objects are displayed.
- **Tornado plot window:** Allows you to compare the impact of each parameter in the sensitivity analysis.

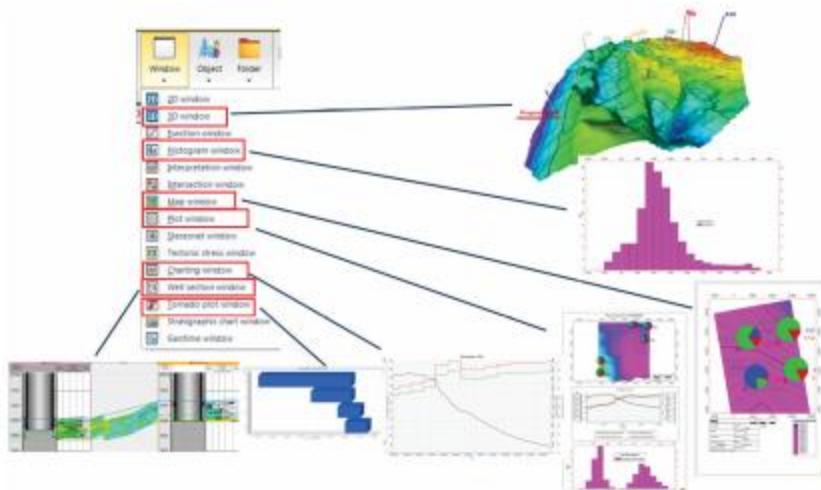


Figure 43. Visualization windows

Visualization: Display tools

There are many tools in Petrel in the **Window** toolbar. The figures show some important tools.



Figure 44. Tools for maneuvering in the Window toolbar

- 1 To move the view in any of the windows, use the left mouse button. However, to move anything, you must have the **View mode** (hand) active. When the **View mode** is active, a hand is shown in the position of the cursor. The shortcut key is V.
- 2 **Select/pick mode** (arrow) is used to select an item. In this mode, you can click any object and get information about it in the lower right corner of the Petrel window or in the **Inspector**. The shortcut key is P.



Figure 45. Tools for visualization in the Window toolbar

- 1 Go to home position (Home)
- 2 View all displayed data
- 3 View from specified position
- 4 Target zoom
- 5 Orthogonal camera

All windows that show a vertical scale can be set to show data in TWT (two-way time) or TVD (true vertical depth). The default is Any, which means that data from different domains can be mixed in the windows. (For example, it is useful to check if the wells are intersecting with the seismic data cube.)

However, if the vertical scale of a window is set to TWT, it is not possible to view depth data in the window. If you have trouble visualizing data in the selected window, always check that the vertical scale of the window is set to TVD (or Any).

Visualization: Check boxes

Select the gray check boxes to display the object in the active window.
Several objects can be displayed simultaneously.

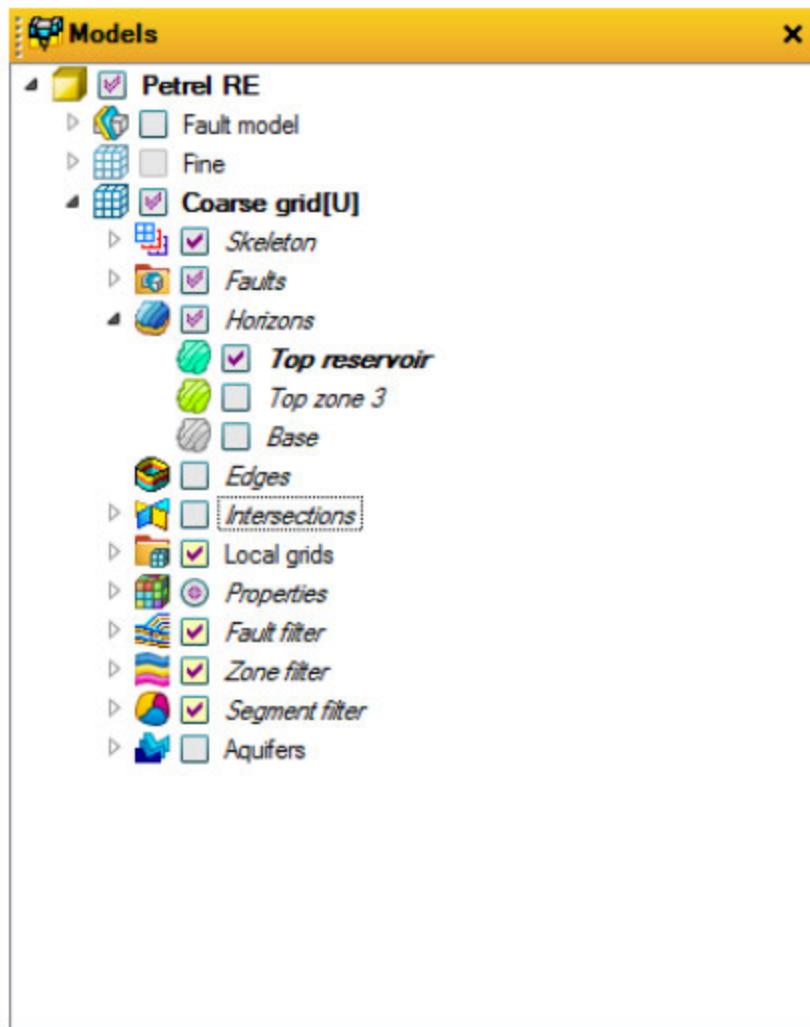


Figure 46. Gray check boxes

If only one object of a particular type can be displayed at a time in the active window (for example, grid properties), then the objects appear with option buttons next to them.

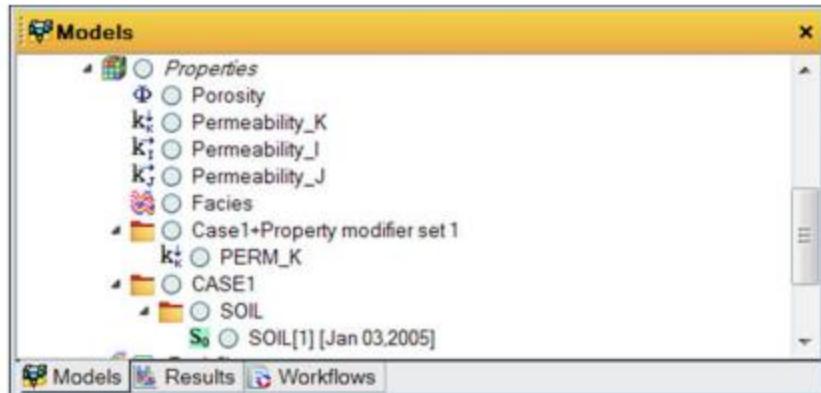


Figure 47. Gray option buttons

The yellow square boxes are filters. Use them to filter out different parts of a 3D grid.

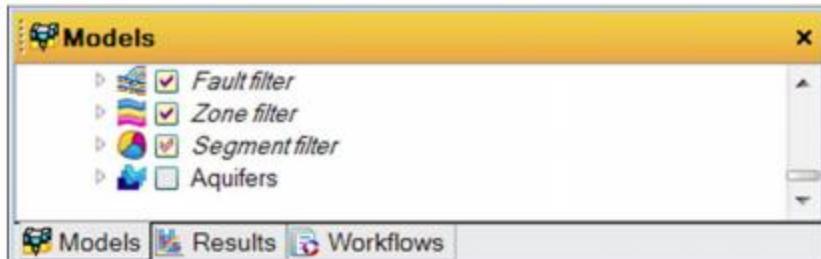


Figure 48. Yellow square boxes

Object insertion

Inserting a data object into a field in any dialog box in Petrel is done by selecting the object and then clicking .

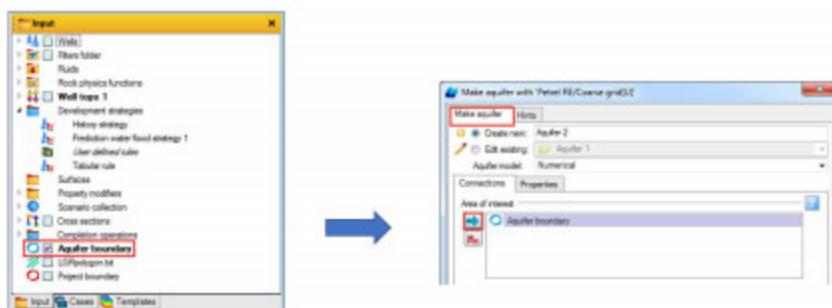


Figure 49. Inserting an object into a field

Options to open and save Petrel projects

A Petrel project contains all imported and generated data, as well as all dialog box and graphical settings.

When you save a project, a file and a folder are created. The file is named <ProjectName>.pet and contains a list of pointers to all data. The folder is named <ProjectName>.ptd and contains the data.

This organization ensures that memory is allocated for optimal performance, which is important when working with large datasets.

When you create a simulation case and export it from Petrel using the **Define simulation case** dialog box, a <ProjectName>.sim folder is created. This folder is saved in the same directory as the <ProjectName>.pet file. The simulation datadeck (with the same name as the case) also is saved in the <ProjectName>.sim folder.

The figure shows the options for opening and saving projects in Petrel.

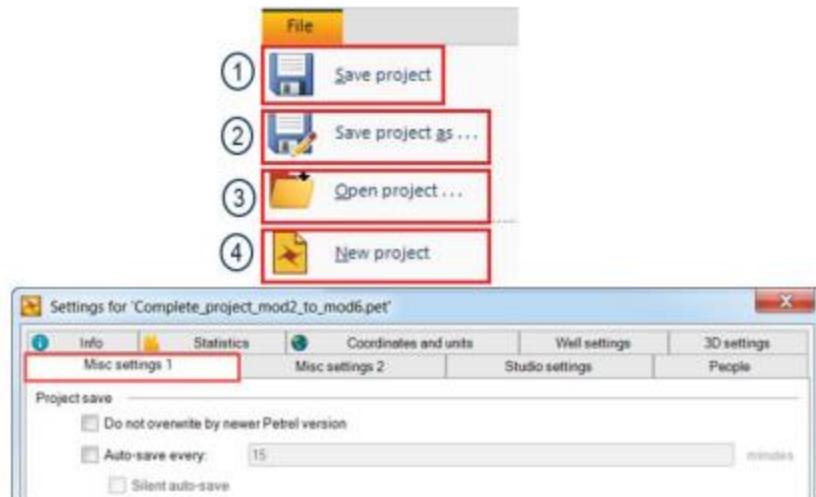


Figure 50. Options for opening and saving projects in Petrel

- 1 **Save project:** Saves a Petrel project to the existing .pet file.
- 2 **Save project as:** Saves a Petrel project to a new name. The .pet file, the .ptd data folder, and the .sim folder also are saved with the new project name.
- 3 **Open project:** Opens an existing Petrel project (.pet).
- 4 **New project:** Creates a project.
- 5 **Auto-save every:** (This option is not selected by default.) Automatically saves a Petrel project at the interval that you specify, overwriting the current saved version. Access this option on the **Misc settings 1** tab in the project **Settings** dialog box.

Copyright © 2017 Schlumberger. All rights reserved.



WARNING: Use the automatic save option with caution; it overwrites the currently saved project. If you select this option, you cannot revert to your original version if you make errors. Also, when working on large projects, saving the project at automatic intervals can take a long time.

File management

The first time you save a project, on the **File** tab, click **Save Project As**.

WARNING: When making a backup or sending your project to support, include all associated files, not just the **.pet** file. The **.pet** file contains only the index. It does not contain the data!

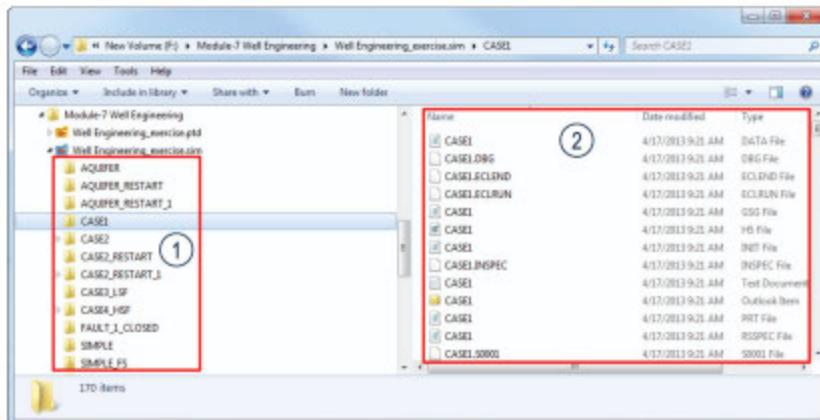


Figure 51. Example of the **.sim** folder and generated simulation files in the project directory

Copyright © 2017 Schlumberger. All rights reserved.

- 1 The **.sim** folder contains a subfolder for each case that you create.
- 2 Each simulation case folder contains the simulator keyword input files and result files.

It is recommended that you do not delete files in the generated folders while Petrel is running. Petrel might be using the files that you want to delete.

If you are using a network drive, make sure that no simulators are running on other computers and accessing the files.



NOTE: A folder named `ProjectName.sim/Cache` contains temporary XML files used by the keyword generator system to speed up random access to keywords. You can delete these files because Petrel regenerates them when required. Petrel writes simulator keyword input files based on the case definition.

If you did not make any edits directly to the simulator input files, you can delete any or all of these files. To rerun the simulation, re-export it from Petrel.

Simulation data management: .sim folder on an external drive

The simulation data folders can be stored on a drive/system that is different from where other project data is stored.

Enter the path for the `.sim` folder in the project **Settings** dialog box on the **Misc settings 1** tab.

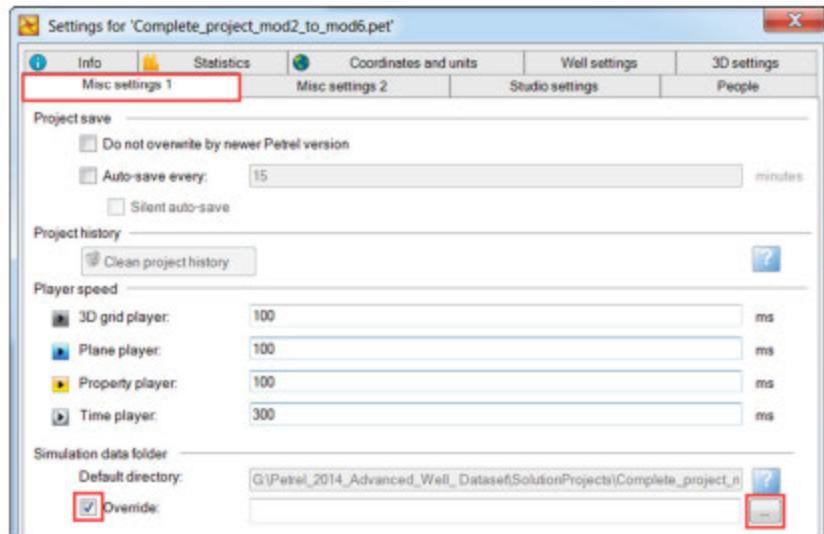


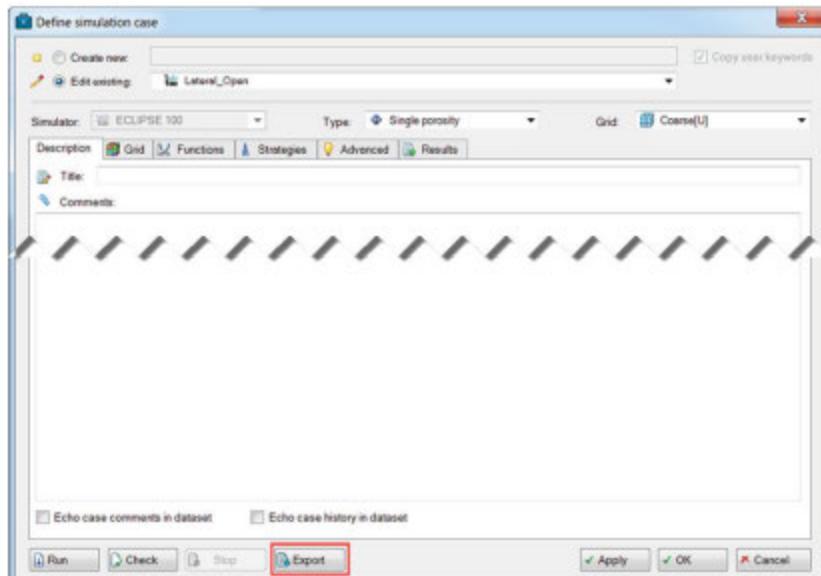
Figure 52. Settings dialog box where you enter the path for the `.sim` folder



Procedure — Export simulation keywords

To create and export a full simulation case, use the **Define simulation case** dialog box. The time required for export depends on the type of the grid used for the simulation. For example, a binary grid is smaller in size than an ASCII file, so it exports faster.

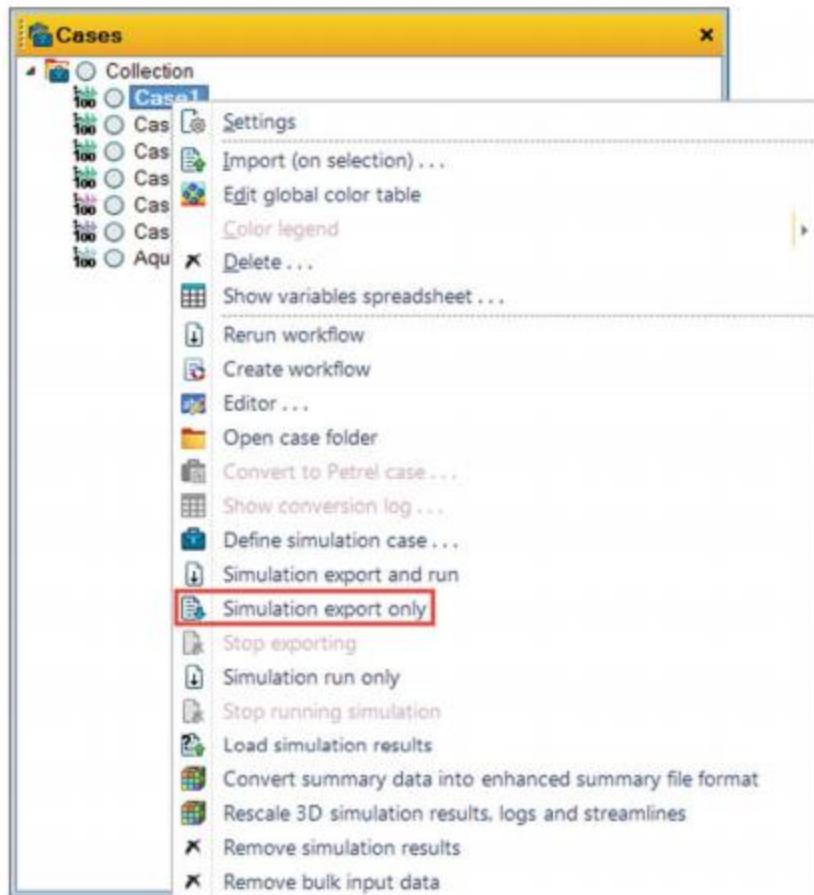
1. On the **Simulation** tab, in the **Simulation** group, click **Define case** to open the **Define simulation case** dialog box.
2. Click **Edit existing** and then select the desired case.



3. Click **Export**.

The simulation is exported, but not run. The consistency of the case is checked before the keywords are exported.

As a shortcut for exporting simulation keywords, right-click the simulation case in the **Cases** pane and click **Simulation export only**.



Copyright © 2017 Schlumberger. All rights reserved.

Team collaboration: Annotate

The **Annotate** tools provide a new and unique capability that allows you to have and share spatially aware knowledge in the context of Petrel content.

With the **Annotate** tools, you and your teams can share and manipulate content and knowledge in Petrel using intuitive workflows. This content and knowledge can be in the form of URLs, notes, files, photos, voice recordings, and any type of file-based content. Photos and hyperlinks can be added as attachments or embedded in the notes.



Figure 53. Annotate tools

Data in Studio can be written or updated by you or members of your team, provided that they have the appropriate access to the repository.

In a classical collaboration workflow, quality-tagging the data items allows easier identification of data.

When you work in a Studio environment, you can set up subscriptions to receive updates when information of interest to you is available or has changed in Studio.

Quality attributes

You can tag data items with quality attributes before sending them to Studio. Quality attributes allow you to add context to the data items in a Studio environment.

Items in Petrel can be tagged using these tools:

- Settings for the item (**Quality attributes** tab)
- **Inspector**
- Project data table

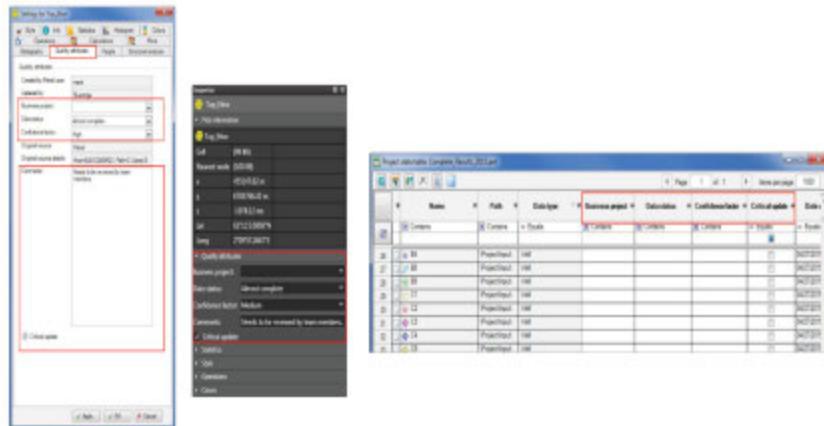


Figure 54. Quality tags in the Settings for the item (left), Inspector (center) and Project data table (right)

Copyright © 2017 Schlumberger. All rights reserved.

To send data from your Petrel project to Studio, in the **Input** pane or a window such as a **2D window** or **3D window**, right-click any item and click **Send to Studio**.

Subscriptions

When you work in a Studio environment, you can subscribe to receive alerts and notifications about data of interest to you. These alerts and notifications are called subscriptions.

You can subscribe to repository filters or folders. You subscribe to filters in the **Manage filters** tools and to folders in the **Input** pane. With folders, it is important to be aware that folders in Studio and Petrel are matched by GUID. To receive notifications on folder subscriptions, the folder in your Petrel project must have the same GUID as the folder in the Studio repository.

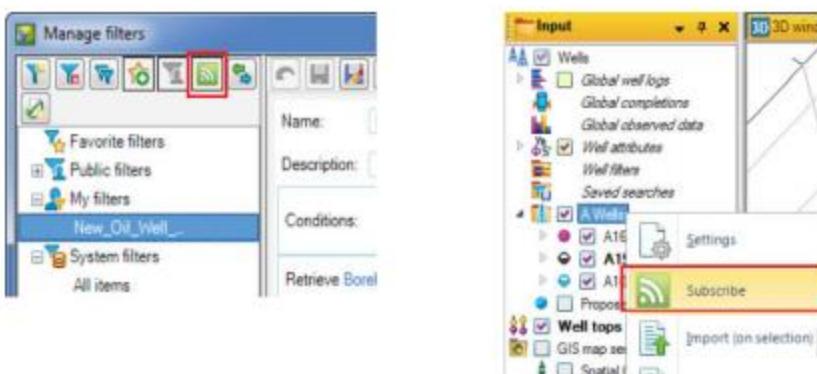


Figure 55. Subscription to folders (left) and filters (right)

Notifications and alerts work while you are connected to a repository. The alert messages that appear in the bottom right corner of your screen are associated with your filter or folder subscriptions.

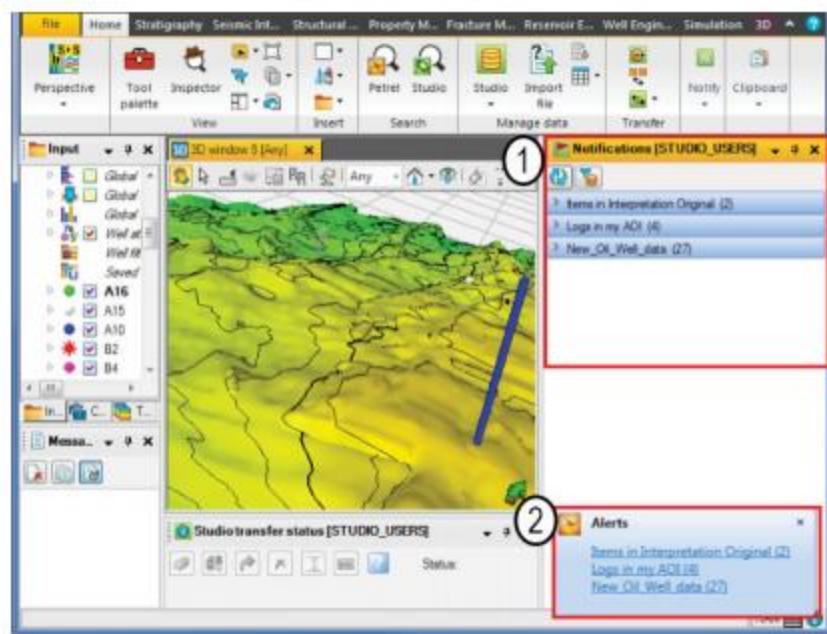


Figure 56. Notifications and alerts

1 **Notifications** pane

2 Alert messages

The **Notifications** pane shows more details and options related to the items of interest to you. For example, you can see more details, compare items to their counterparts in your Petrel project, and retrieve them normally or as copies.

Alert timing settings in Petrel

You can set the timing settings for an alert on the **Data settings** tab in the **Settings** dialog box for the project. On the **File** menu, click **Project setup** and then click **Project settings**. In the **Settings** dialog box, click the **Studio settings** tab, and then click the **Data settings** tab.

By default, while you are connected to Studio, Petrel checks for updates every five minutes, and alerts appear in the bottom right corner of your screen. You can change the settings for these alerts.

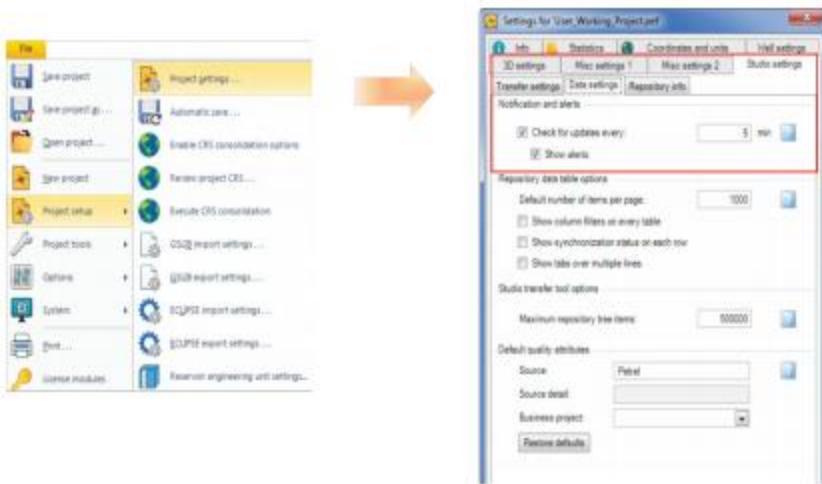


Figure 57. Settings for notifications and alerts

To enable or disable automatic notification updates and alert messages, on the **Home** tab, in the **Notify** group, click **Autorefresh on/off**.

To subscribe to a folder, select the folder in the **Input** pane and on the **Home** tab, in the **Notify** group, click **Subscribe**.



Procedure — Transfer data between projects using the Reference project tool

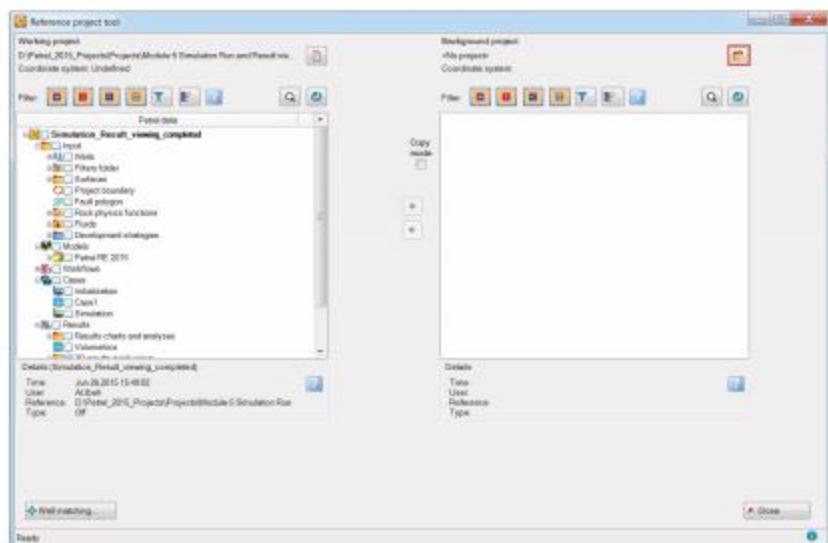
The **Reference project tool** encourages team collaboration by helping you share and compare data between different projects. You can filter out data and move subsets of data items between projects.

This procedure shows you how to use the **Reference project tool** to transfer data from the background project to the working project.

1. On the **Home** tab, in the **Transfer** group, click **Reference project tool**.

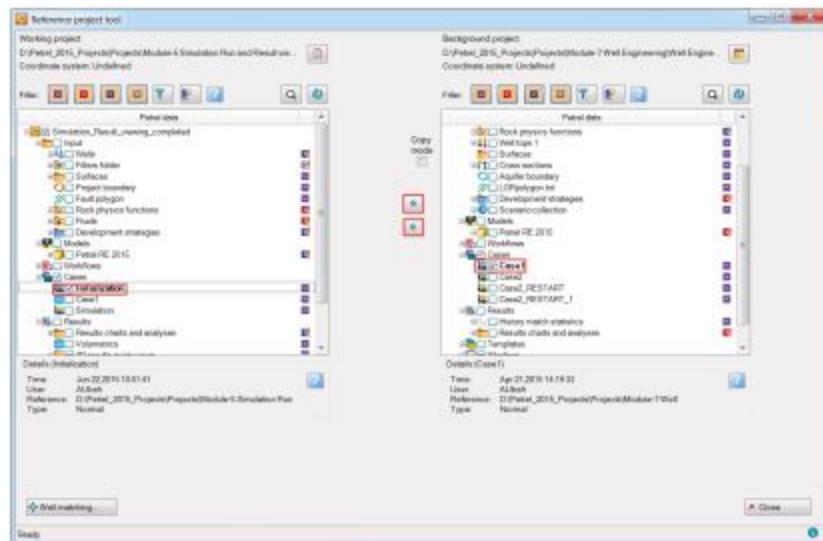


2. Open the background project. Click **Open a project**.



Copyright © 2017 Schlumberger. All rights reserved.

3. In the **Reference project tool** dialog box, select the check boxes for the objects that you want to transfer into the working project or background project, and then click **Receive from background project** or **Send to background project**.



Copyright © 2017 Schlumberger. All rights reserved.

Help Center

The Help Center provides access to Petrel technical documentation including the latest version of the Petrel Release Notes and the What's New document. You also can access an archive of product documents from the last two years, the Schlumberger Information Solution support portal and software, and technical training on the Schlumberger NExT training hub.

To access the Petrel Help Center, on the **File** tab, click **Help**, and then click **Help Center** or press F1.



Figure 58. Petrel Help Center

Copyright © 2017 Schlumberger. All rights reserved.

Exercises — Use the Petrel user interface

The purpose of these exercises is to get you familiarized quickly with the Petrel user interface by focusing on reservoir engineering domain workflows. You access these workflows by selecting **Reservoir** and **Production** in the **Perspective** tool on the **Home** tab.

Workflow

1. Start Petrel.
2. Navigate through the Petrel Reservoir Engineering domain workflow and user interface.
3. Display objects in **2D windows** and **3D windows** and use the **Inspector** for detailed objective view and style adjustment.

4. Review/edit object settings using the mini toolbar, Tool Palette, and shortcut menus.
5. Insert comments or attach a file directly to the displayed object using **Annotate** tools.

Data

In these exercises, you use the `PetrelRE2017_Completed.pet` project located in the `Dataset\Projects\Module_2_Petrel User Interface` folder.

Exercise 1 Start Petrel

In this exercise, you open Petrel, select license packages using a server license or remote dongle license key, and open an existing Petrel project.

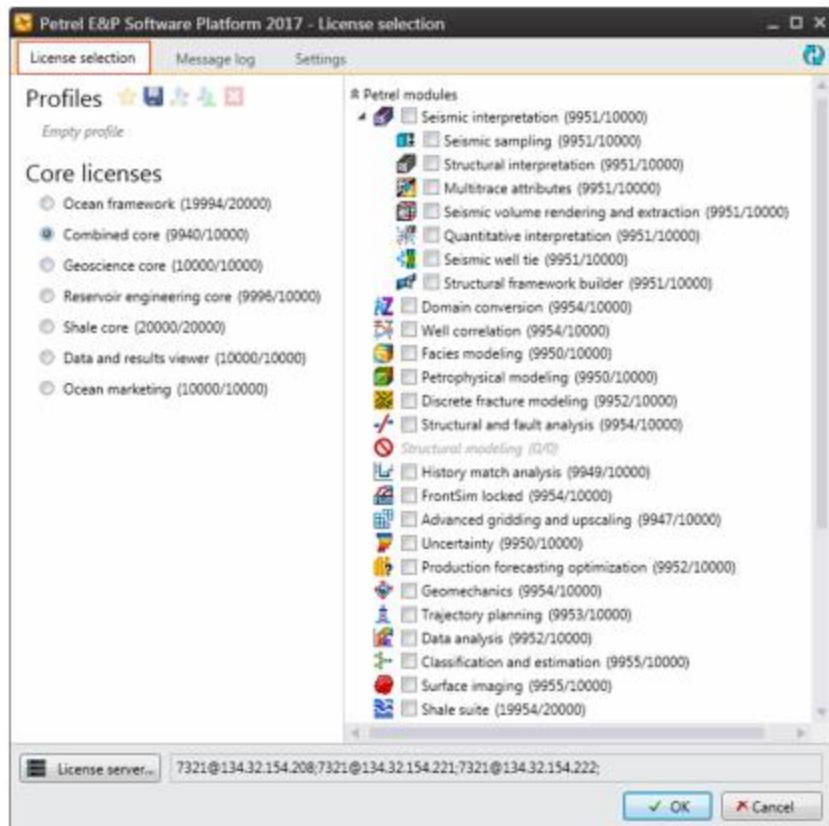


Copyright © 2017 Schlumberger. All rights reserved.

1. Start Petrel. Double-click the Petrel  icon on the desktop.
If you are using a dongle, an introduction window to Petrel appears before the Petrel window opens.
2. If you are using a server license, the first thing you see when you open Petrel is a **License Selection** window.

The license packages available on the selected license server are listed on the left side of the window. The modules available in this predefined package of modules are listed on the right.

Click the Package name in this list and click **OK** to start Petrel.



Copyright © 2017 Schlumberger. All rights reserved.

3. Open the training dataset. On the **File** tab, click **Open Project**.
4. Choose the **PetrelRE2017_Completed.pet** file from the **Dataset\Projects\Module 2_Petrel User Interface** folder.
5. Copy the project and save it to your student directory.
 - a. On the **File** tab, click **Save project as**.
 - b. Give the project a new name and click **Save**.



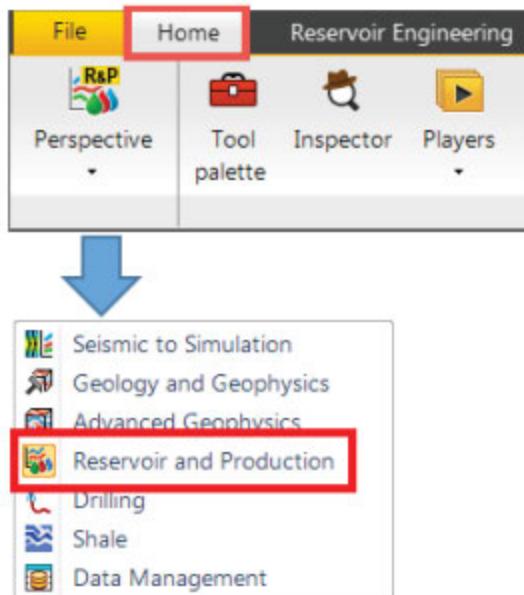
Exercise 2 Visualize data

There are various tools in Petrel for data analysis and visualization.

- **3D windows** and **2D windows** (for graphic display of objects)
- **Well section window** (for well correlation, visualization of well completions, or display of the simulation grid as a backdrop)
- **Function window** (for plotting fluid model and saturation functions)
- **Charting window** (for making line plots; this window is a good tool for field production analysis plots)
- **Inspector** (for quick access to the active displayed object information in the window).

In this exercise, you focus on how to use a **3D window** and **Inspector** to view and QC data.

1. On the **Home** tab, click **Perspective** and then click **Reservoir and Production**.



2. Open a **3D window**. On the **Home** tab, in the **Insert** group, click **Window** and then click **3D window**.

3. Change the **3D window** background color to white.
 - a. Under **Window**, on the **3D** tab, in the **Display elements** group, click **Settings** and then click **Window settings**.
 - b. In the **Color** list, select the white color. Click **OK**.
 - c. Under **Window**, on the **3D** tab, in the **Display elements** group, click **Toggle background**.



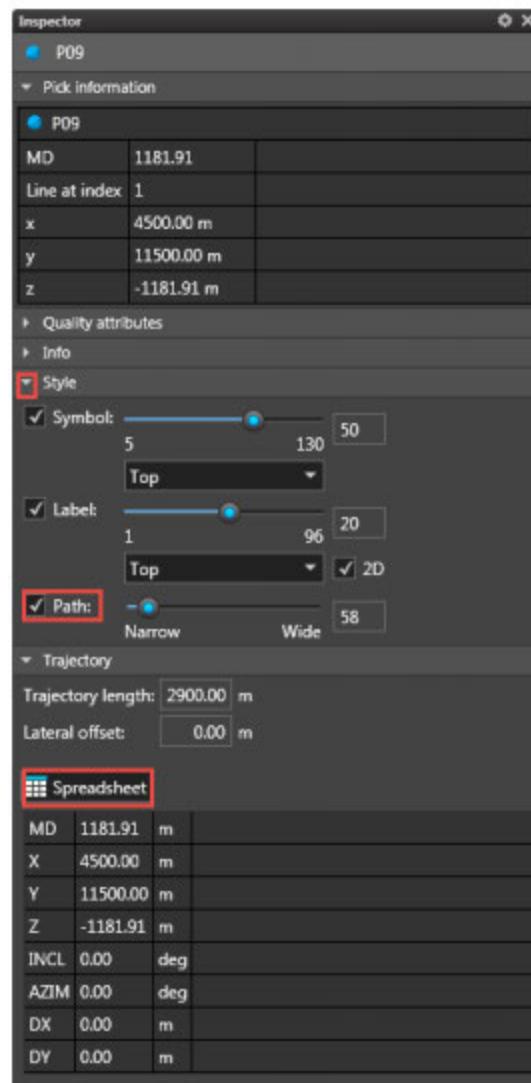
4. Display wells by selecting the check box next to the **Wells** folder in the **Input** pane.
5. Manipulate the object. Click **View** on the **Window** toolbar (at the top of the **3D window**).
6. Rotate the object in the **3D window**. Click and hold the mouse button and move the mouse.
7. Pan the object. Press Shift or Ctrl, then click and hold the mouse button and move the mouse.
8. Zoom in and out. Press and Shift + Ctrl, then click and hold the mouse button and move the mouse. With a three-button mouse, use the left and middle buttons to zoom.
9. Press Esc in viewing mode. The pointer changes to an arrow.
10. Optional: Change the pointer to an arrow by clicking **Select mode** on the **Window** toolbar.
11. Click any of the wells in the display.

Information appears in the status bar at the bottom of the window.

12. On the **Home** tab, in the **View** group, click **Inspector** and then click any of the displayed wells in the **3D window**.
13. Enlarge the **Inspector** dialog box and expand the **Style** section.

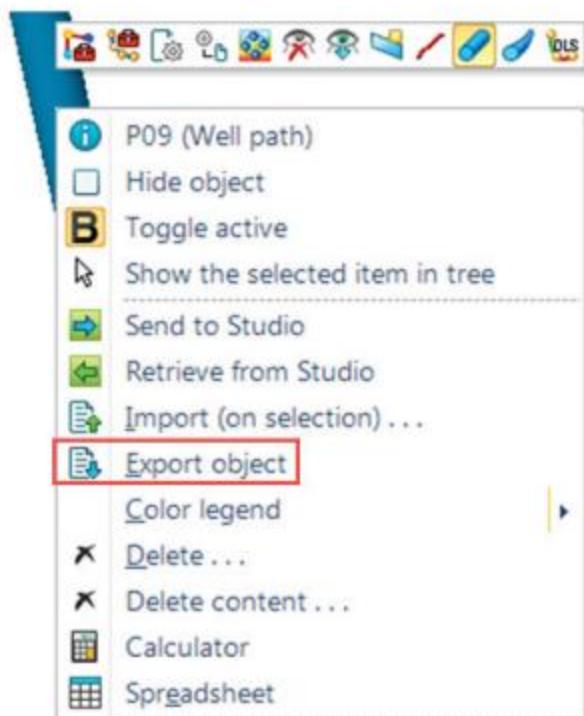
14. In the **Path** section, increase the size of the well path by moving the slider from left to right. If the labels disappear while increasing the size of the well path, click in the **3D window**.
15. Repeat the same process for Symbol.
16. Expand the **Info** section. Review well data such as X and Y coordinates.
17. Expand the **Trajectory** section and click **Spreadsheet**.

The well trace spreadsheet for the particular well opens.



18. Close the Inspector.

19. Right-click any of the wells and click **Export object**.



Copyright © 2017 Schlumberger. All rights reserved.

20. Enter the file name in the **File name** box and select **Well path/deviation (ASCII)(*.*)** as the type of file to save.
21. Create another subfolder in the Student folder and save the well file to that subfolder.

Exercise 3 Access and use the Input and Models panes

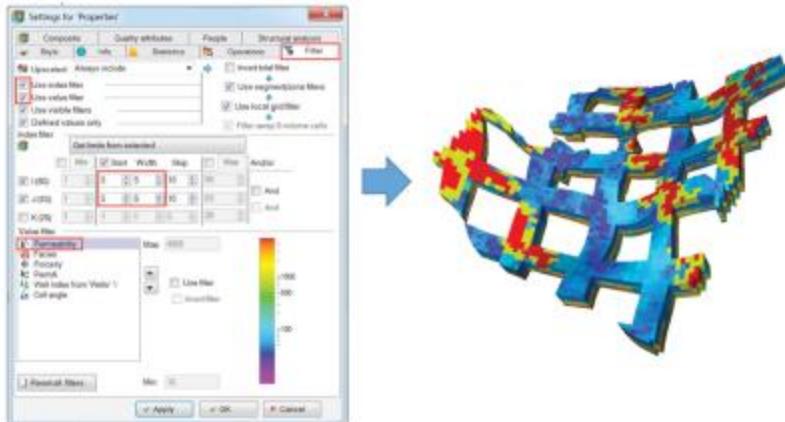


Data in Petrel is stored in different explorer panes, depending on the type of data. In this exercise, you explore the data that is stored in the **Input** and **Models** panes.

1. Click the **Models** pane.
2. Right-click the Petrel RE 2017 folder and click **Expand (recursive)** to see the subfolders and their contents.
3. Select **Permeability** in the Properties folder of the Fine grid.

4. On the **Window** toolbar, click **Zoom [S]**  and then click the displayed permeability property to zoom in.
5. Open the property filter dialog box. Right-click the displayed permeability grid property and click **Show property filter**  on the window mini toolbar.
6. In the **Settings** dialog box for the property, filter the property based on I, J indexes and values. Select the **Use index filter** and **Use value filter** check boxes and click **Apply**.
7. Change both **Start** and **Width** values in I and J directions to 5 and click **Apply**.
What do you observe?
8. Again change only the **Width** to 1 and click **Apply**.

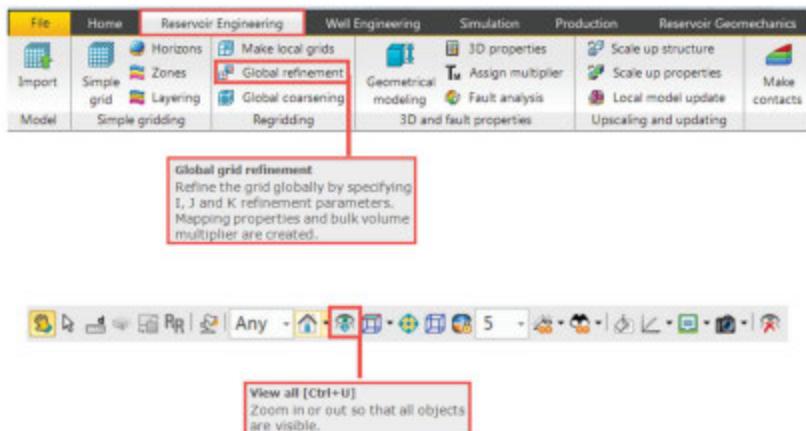
This filter feature gives you the flexibility to apply filters in the 3D grid in I, J, and K directions.



9. Clear the check boxes to reset the filters and click **Apply**.
10. Close the **Settings** dialog box.
11. Right-click the **Petrel RE 2017** folder and click **Collapse (recursive)**.
12. Open the **Input** pane and right-click the different folders to see which options are available. Different folders in the **Input** pane display different options.

Exercise 4 Access tooltips

Point to the tools on the ribbon and **Window** toolbar to see descriptions of each tool. This exercise is a quick way to learn the functions of the activated tools.



Exercise 5 Quality check objects on the Statistics tab

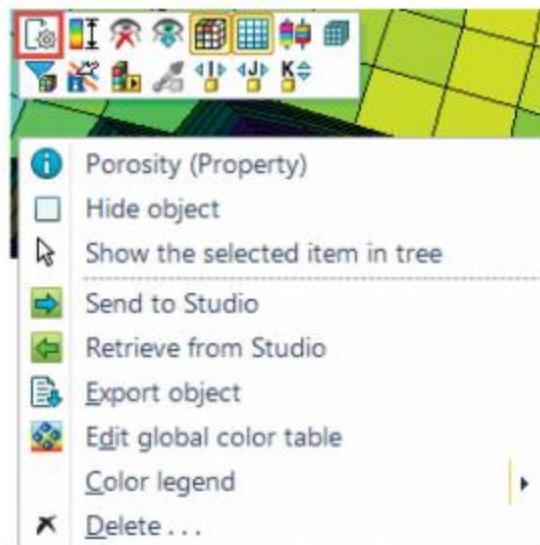
The **Statistics** tab in the **Object settings** dialog box provides you with valuable information about the data object in the Petrel project. The information displayed on the **Statistics** tab depends on the object type that you select. You also can use this tab to review a Petrel project from a colleague.

The **Statistics** tab is read-only. It normally is used for quick quality checking (QC) of the active object.

1. Clear all of the active objects in the **3D window**. Under **Window**, on the **3D** tab, in the **Visualization** group, click **Clear display**. Alternatively, click **Clear display** on the **Window** toolbar.



2. In the **Models** pane, expand the Petrel RE 2017 folder and select the **Porosity property** in the Properties folder of the Coarse grid[U].
3. Access the grid property settings. Right-click the displayed **Porosity property** and click **Settings [ALT+Return]** on the mini toolbar.

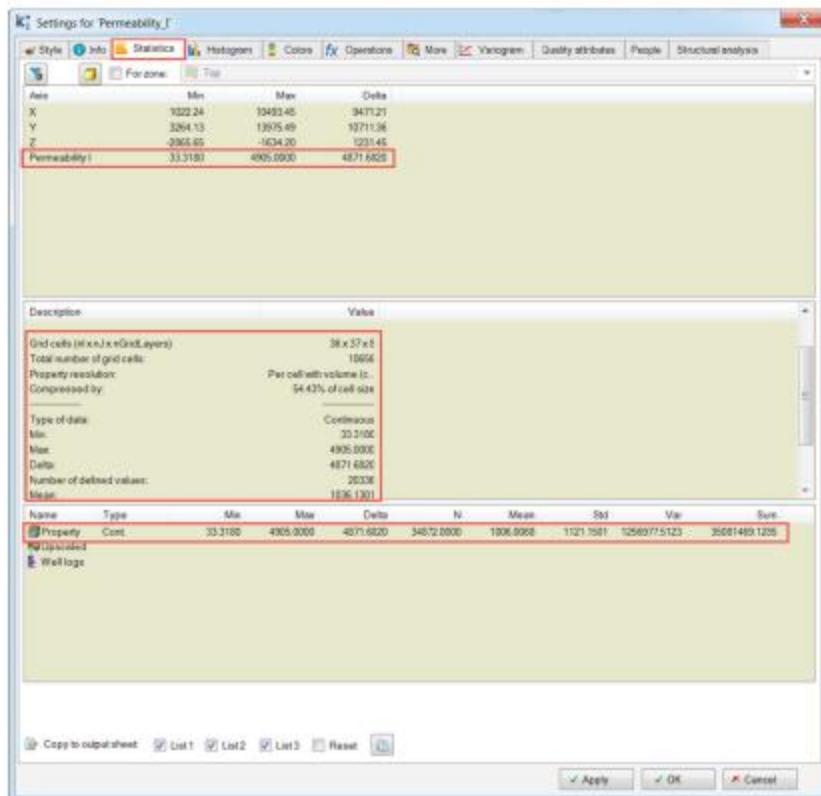


4. On the **Statistics** tab, check the size of the 3D grid. Alternatively, under **Grid Property**, on the **Tools** tab, in the **Analysis** group, click **Statistics**.

You can check the minimum and maximum values of the grid property. You also can find the number of cells and some statistics results, such as mean, standard deviation, and variance.



5. Open the settings of **Permeability_I** in the Properties folder.
 Check the statistics for other grid properties (for example, check the range of **Permeability_I** or **Permeability_K**).



Copyright © 2017 Schlumberger. All rights reserved.

6. Click **Copy to output sheet** to output the statistics of the Properties folder.

You can edit the output sheet, copy the information, and paste it into another spreadsheet. You also can save it as a file by using the available tools on the toolbar at the top of the active output sheet.

	A	B	C	D	E	F	G	H	I	J
Statistics for Permeability (Untitled-1)										
1	Axis		Min	Max	Delta					
2	X		1623.24	32492.41	30871.21					
3	Y		1284.12	12875.49	12751.38					
4	Z		-2855.65	-1834.20	321.45					
5	Permeability []		33.1080	4905.0000	4871.6820					
6	Description		Value							
7	Unit:		mD							
8	Is specified (t)		No							
9	Total number of defined cells in entire property, including local grids:		20336							
10	Grid cells (n x m x nGridLayers)		38 x 57 x 8							
11	Total number of grid cells		30656							
12	Property resolution:		Per cell with volume (20336 cells)							
13	Compressed by:		34.42% of cell size							
14	Type of data:		Continuous							
15	Min:		33.1080							
16	Max:		4905.0000							
17	Delta:		4871.6820							
18	Number of defined values:		20336							
19	Mean:		1096.1301							
20	Std. Dev.:		1133.9310							
21	Variance:		1288798.6246							
22	Sum:		21870781.3888							
23	Name	Type	Min	Max	Delta	N	Mean	Std	Var	Sum
24	Property	Const	33.1080	4905.0000	4871.6820	344872.0000	3066.0068	1325.1581	58977.5129	91468.1295
25	Uncoded									
26	Wet log:									

Copyright © 2017 Schlumberger. All rights reserved.

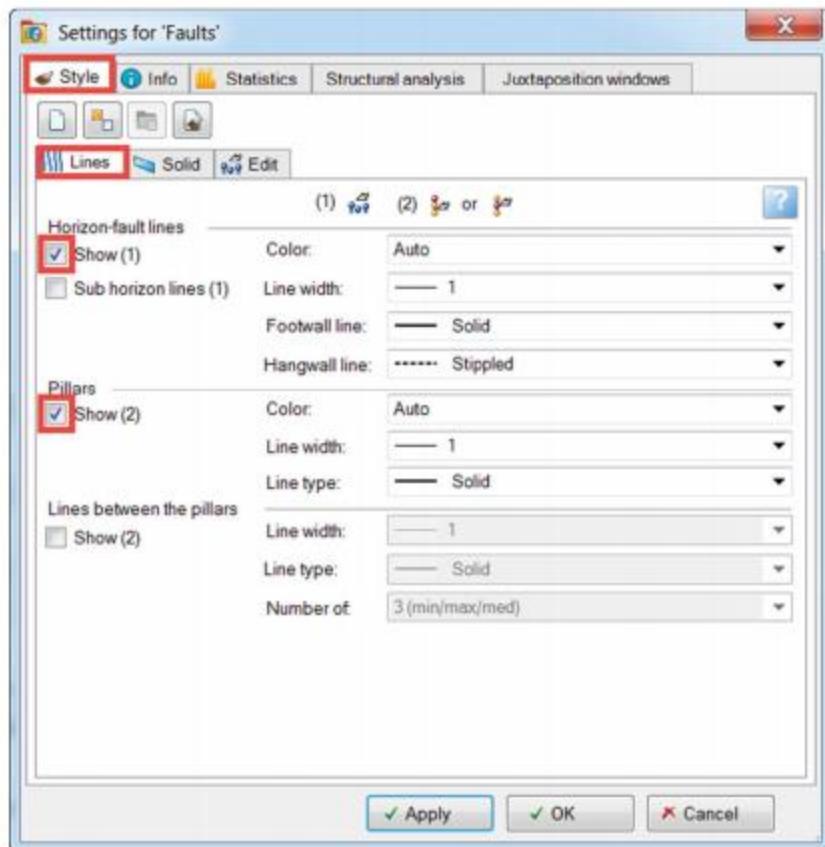


Exercise 6 View templates and set grid property styles on the Info and Style tabs

Objects in Petrel can be attached to templates that control their precision, unit, and color table. The style of each object also can be customized to the desired appearance. In this exercise, you view available property templates and change the style settings of a Horizon.

1. Open the grid property settings.
2. On the **Info** tab, open the **Template** list to see the available templates.
3. Clear the active displayed objects in the **3D window**.
4. Select to view one of the Horizons (Base) in the Coarse grid[U] in the **Models** pane.
5. Right-click the **Horizons** folder, click **Settings**, and then click the **Style** tab. This tab is where you can specify display style options.
6. On the **Grid lines** tab, select **Show** to view grid lines.

7. Click **Apply** and observe the changes. You also can specify the contour increment, the color of the contour/grid lines, and contour annotations on the same **Style** tab.
8. View the faults in the **3D window** along with the horizon. Select the check box next to the Faults folder in the Coarse grid [U] model.
9. Right-click the Faults folder, click **Settings**, and then click the **Style** tab.
10. On the **Lines** subtab, display fault pillars and a horizon fault line by selecting **Show (2)** in the **Pillars** section and **Show (1)** in the **Horizon-fault lines** section.



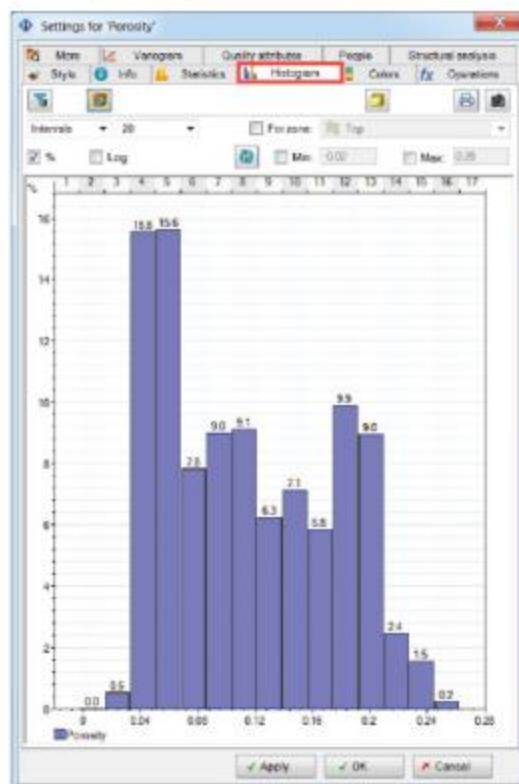
Copyright © 2017 Schlumberger. All rights reserved.



Exercise 7 Change histogram properties on the Histogram tab

This exercise explains how to view 3D grid properties distributions on the **Histogram** tab.

1. Clear all of the active displayed objects in the **3D window**.
2. Display the porosity property in the Properties folder of the Coarse grid[U].
3. Right-click the **porosity property** and click **Settings** on the Window mini toolbar.
4. On the **Histogram** tab, view the distribution of the property values.



Copyright © 2017 Schlumberger. All rights reserved.

Alternatively, to quickly open the **Histogram** tab, under **Grid property**, on the **Tools** tab, in the **Analysis** group, click **Histogram**.

- Similarly, click the **Histogram** tab of the permeability and view its distribution.

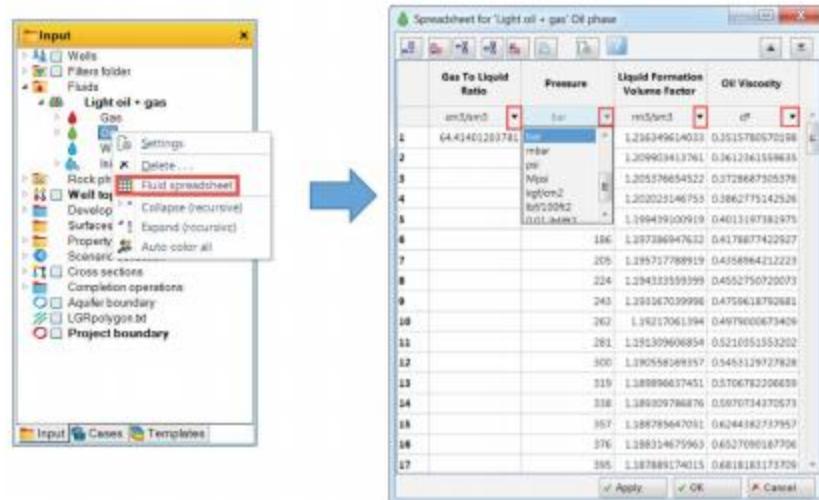
By default Petrel also shows the histogram along with the color legend in the **3D window**. It is the same histogram as shown on the **Histogram** tab. It provides a quick view of the distribution. To activate color legend in the **3D window** for a displayed object, open the **3D** contextual tab and click **Auto legend**. To display histogram along with color legend in the **3D window**, select **Show histogram** in the **Automatic legend settings** dialog box.

Exercise 8 Edit and quality check with the spreadsheet

There are many ways to review data in Petrel. Most of these methods are visual. One of these methods is using spreadsheets. In this exercise, you learn how to QC input data using a spreadsheet.

- In the Input pane, expand Fluids and then expand Light oil + gas.
- Right-click Oil and then click **Fluid spreadsheet**.

In the spreadsheet for the fluid, you can change the measurement units using the associated lists.



You can edit the data values in the spreadsheet and add rows, if necessary. You can copy the spreadsheet contents and paste them in Excel using options in the **Fluid Spreadsheet** dialog box.

Team collaboration and annotation tools

To reduce or eliminate rework, teams need easy and intuitive ways to collaborate. Teams can capture and share the generated information and knowledge to improve their productivity effectively. Studio provides this capability in Petrel.

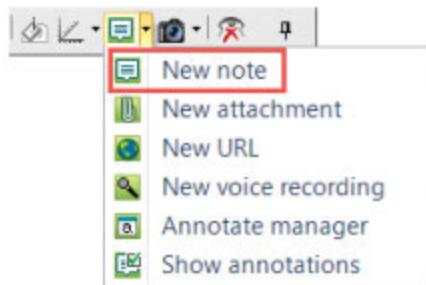
Knowledge sharing and capturing of information can be achieved by adding more context to the project. You can add attachments (notes, documents, video, voice) in the Petrel modeling environment by using sticky notes.



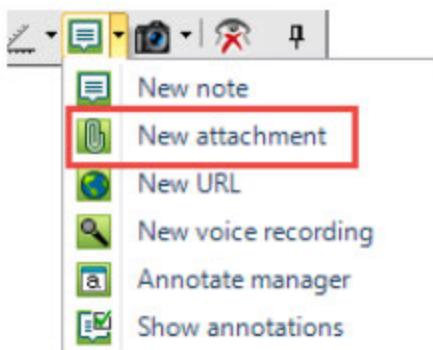
Exercise 9 Insert a note and attachment

In this exercise, you learn how to insert a simple note and attachment into the Petrel project.

1. Open a new **3D window**.
2. In the **Cases** pane, expand **Collection** and click **Case 1**.
3. Open the **Results** pane, expand **Simulation grid results**, expand **Dynamic**, and select **Oil saturation**.
4. In the **Input** pane, expand **Wells**, expand **Producers**, and select well **P09**.
5. On the **Window** toolbar, click the arrow next to **New note** and click **New note** in the list.



6. In the **3D window**, click the well P09 path or trajectory.
The **Note editor** opens automatically.
7. Enter New well plan trajectory and click **OK**.
A new note is added in the **3D window** on the same spot where you clicked the well path. Also, a folder named Annotate is added to the **Input** pane.
8. Click the arrow next to **New note** again and click **New attachment** in the list.



9. Click the well P09 path again.
10. Select the file named Observed.vol from the Dataset \ImportData\Observed data folder and click **Open**.
The document is attached to the well path. Also, a folder named Attachment is added in the **Input** pane.
11. In the Attachment folder, right-click the Observed.vol file and click **Open** to view the content of file.

Copyright © 2017 Schlumberger. All rights reserved.

Review and summary

Review what you learned in this module.

The review and summary help you to reinforce the learning objectives for the Petrel Reservoir Engineering for newcomers module.



Review questions

The review questions reinforce the learning objectives for this module.

- What pane do you use to build a list of shortcuts to the objects that you use the most, such as data, processes, and windows?
- What files and folders are created when a typical Petrel simulation project is saved for the first time?
- Input data, 3D models, and simulation cases are stored in which panes?
- Static and dynamic simulation results are stored in which panes?
- What tool do you use to access the Reservoir and Production engineering domain-tailored workflow and on what core tab?
- What are the **Inspector** and **Visual filters** pane used for?
- What are the basic Petrel user interface elements?
- What is the difference between the **Window** toolbar and **Window** tab?
- How do you access the mini toolbar and shortcut menu?
- How do you access the Windows auto-generated shortcut keys quickly?

Summary

In this module, you learned about:

- starting Petrel with a new or an existing project
- navigating through some of the user interfaces for reservoir engineering processes
- reservoir engineering unit settings
- reviewing and altering the settings for Petrel objects
- displaying data in a **3D window**
- adding a note and an attachment to an object

NOTES

Copyright © 2017 Schlumberger. All rights reserved.

NOTES

Copyright © 2017 Schlumberger. All rights reserved.

Module 3 Simple simulation models

In this module, you learn how to build a simple simulation model using the workflow tools available in Petrel. You start by creating a simple grid from scratch using two options: the Make simple grid process and an interpreted structural surface. In this workflow, you add a vertical simulation fault using the tools available on the **Polygon editing Tool Palette**.

You also learn how to import fluid model and saturation functions and how to use the Make fluid and Rock physics function processes to create PVT and saturation functions using the correlation library in Petrel.

The exercises in this module are structured to provide an overview of the typical workflows and tools available in Petrel that are required in a typical reservoir engineering workflow.

Prerequisites

To complete this module successfully, you must have knowledge of reservoir engineering.



Copyright © 2017 Schlumberger. All rights reserved.

Learning objectives



After completing this module, you will know how to:

- build a simple simulation grid from scratch or by using an interpreted structural surface
- create a vertical simulation fault without going through the Fault modeling and Pillar gridding processes
- create fault transmissibility properties
- create a correlation-based black oil fluid model and saturation functions
- create rock compaction functions based on correlations
- import laboratory fluid models and rock physics functions

- edit and visualize the imported fluid model and saturation functions in the Petrel environment
- use spreadsheets to enter functions data directly

Lesson 1 Simple simulation grid



The objective of this lesson is to understand how to create a simple simulation grid using a simple geometry and constant properties. By keeping everything simple, you can complete all of the necessary steps to set up a simulation run in just a few hours.

The main workflow steps in this process are

1. Create a grid using the Make simple grid process in the **Simple gridding** group.
2. Define grid properties using the **Property calculator** or Geometrical modeling.
3. Create a fault by digitizing the grid using the **Polygon editing Tool Palette**.
4. Convert the digitized fault to a simulation fault.
5. Assign a fault transmissibility multiplier by using the fault analysis tool in the **3D and fault properties** group.

The tools required for these processes are organized in groups on the **Reservoir Engineering** tab.



Copyright © 2017 Schlumberger. All rights reserved.

Figure 59. Groups for simple gridding, regridding, and 3D and fault properties

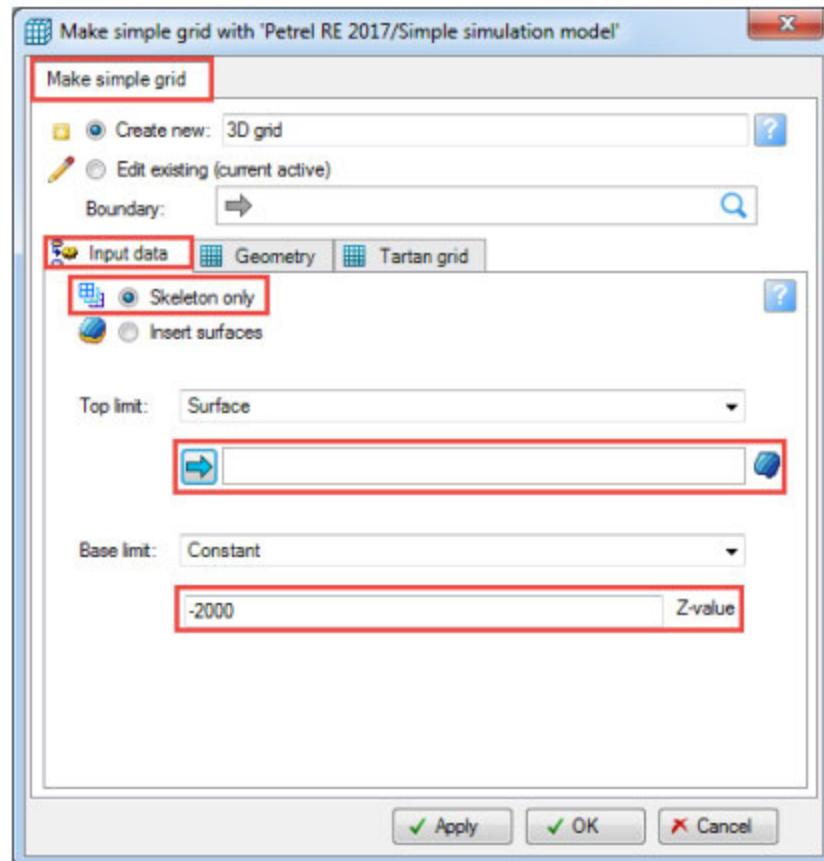
Procedure — Use a constant and surface in the Make simple grid process



Make simple grid is an alternative to Pillar gridding when you create 3D grids with no faults. The geometry of the grid is defined in the **Make simple grid** dialog box.

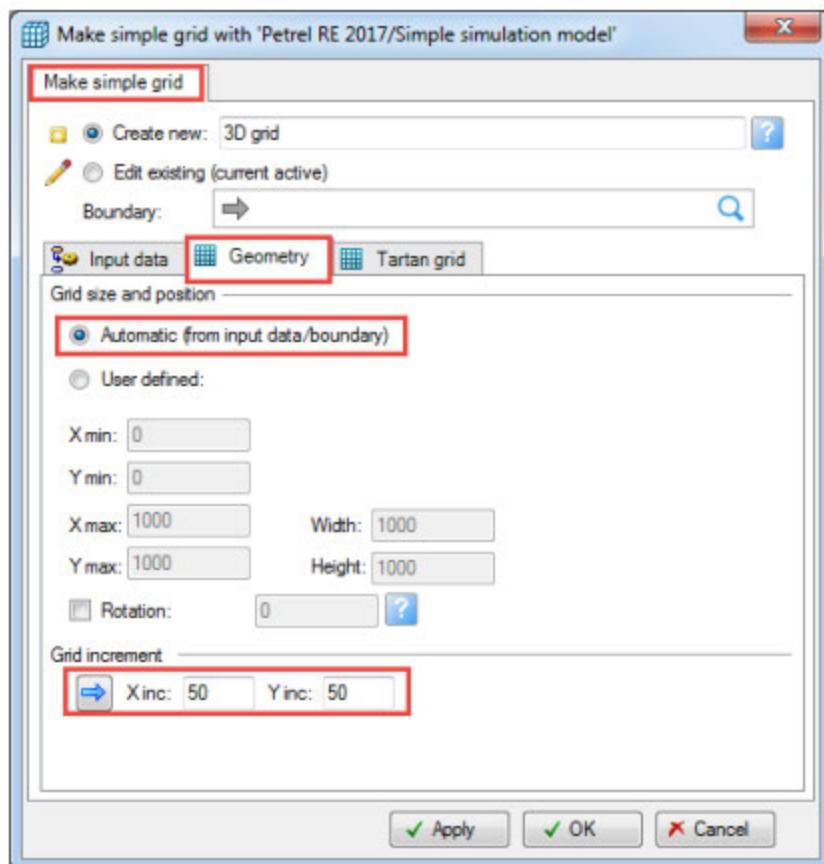
You can add a project boundary or enter X and Y limits. To define horizons in the 3D grid, you can insert surfaces.

1. To define top and bottom limits, use an existing interpreted structural surface or a constant.



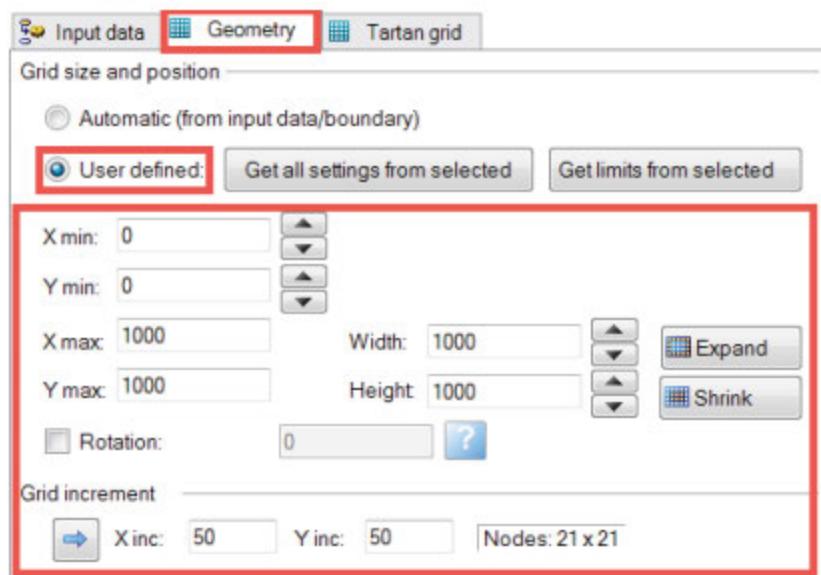
Copyright © 2017 Schlumberger. All rights reserved.

2. Define the grid geometry and cell size using the **Automatic (from input data/boundary)** option.



Copyright © 2017 Schlumberger. All rights reserved.

Alternatively, you can enter the X and Y limits manually using the **User defined** option.



Copyright © 2017 Schlumberger. All rights reserved.

Skeleton grid

A *Skeleton grid* is a Petrel term for the grid mesh framework that is made as the first step toward defining a 3D grid. A Skeleton grid consists of a top, middle, and bottom mesh defined by key pillars. The *pillars* define the lateral position of the corners in the three meshes. The Z-position is defined as the bottom, mid point, and the top of the pillars.



NOTE: The number of cells in X and Y directions in the skeleton usually defines the number of cells for the simulation grid.

After the Skeleton grid is generated, you can subdivide it further in the vertical direction by inserting surfaces. The topmost and bottommost inserted surfaces typically define the top and the bottom of the final 3D grid. Hence, the top and the bottom portions of the Skeleton grid usually are outside of the final 3D grid.

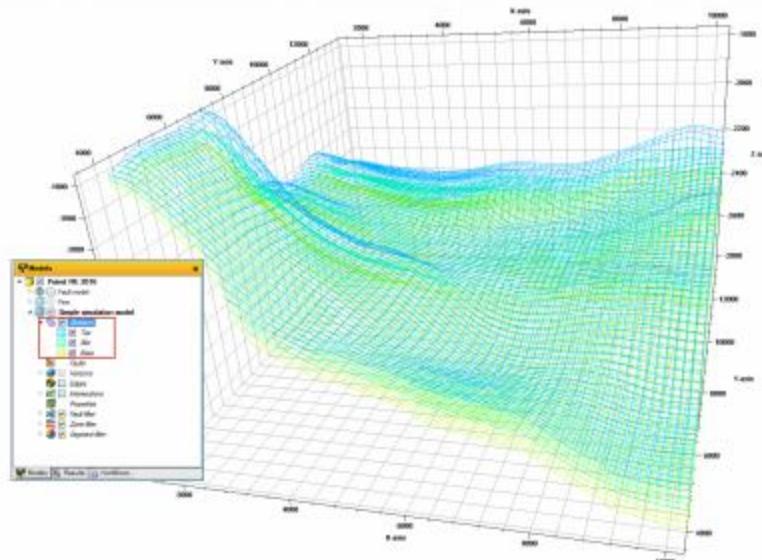
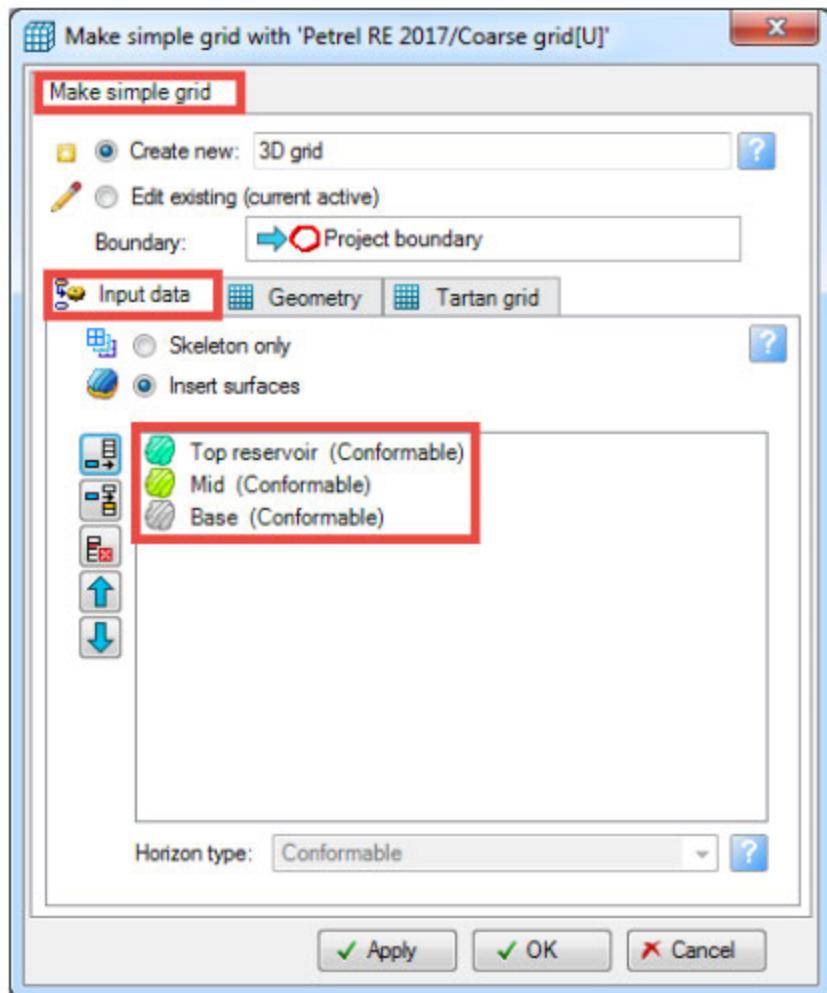


Figure 60. Skeleton grid

Insert horizons

To create horizons, you can insert structural surface data from the **Input** pane into the **Input data** tab of the **Make simple grid** dialog box.

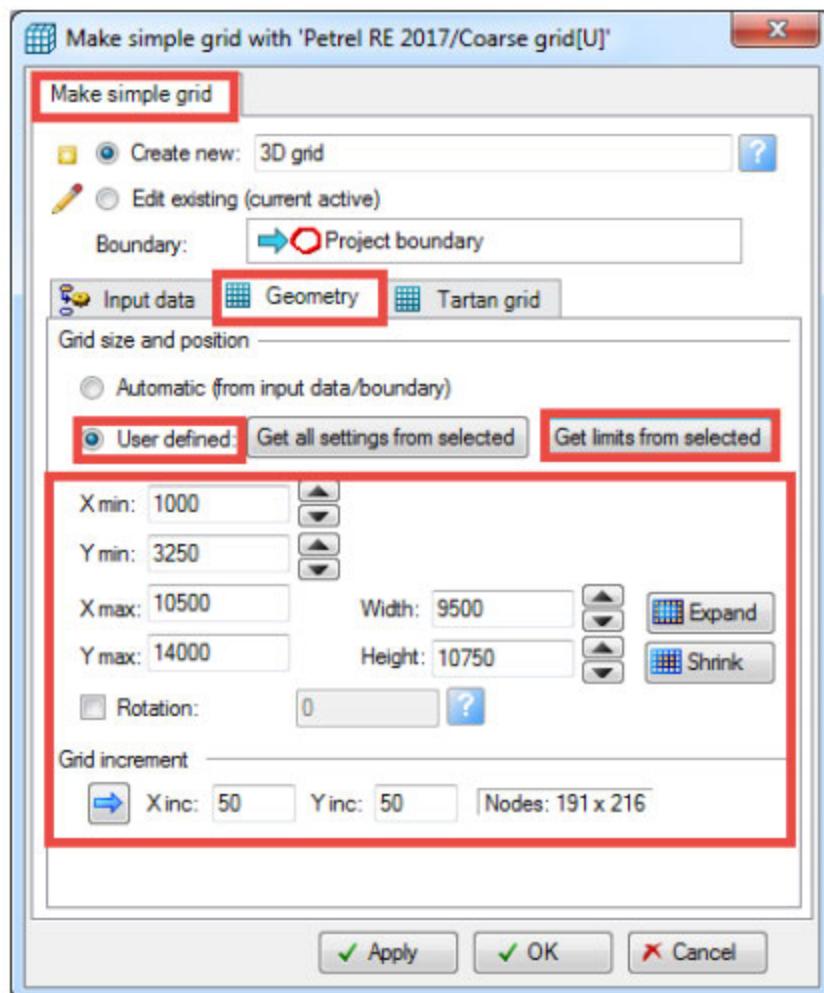


Copyright © 2017 Schlumberger. All rights reserved.

Figure 61. Inserting horizons in the Make simple grid dialog box

Set grid limits

Set the grid data limits by selecting one of the inserted structural surfaces and clicking **Get limits from selection** in the **Make simple grid** dialog box.



Copyright © 2017 Schlumberger. All rights reserved.

Figure 62. Defining grid limits using *Get limits from selected* option

The output is a skeleton grid with horizons.

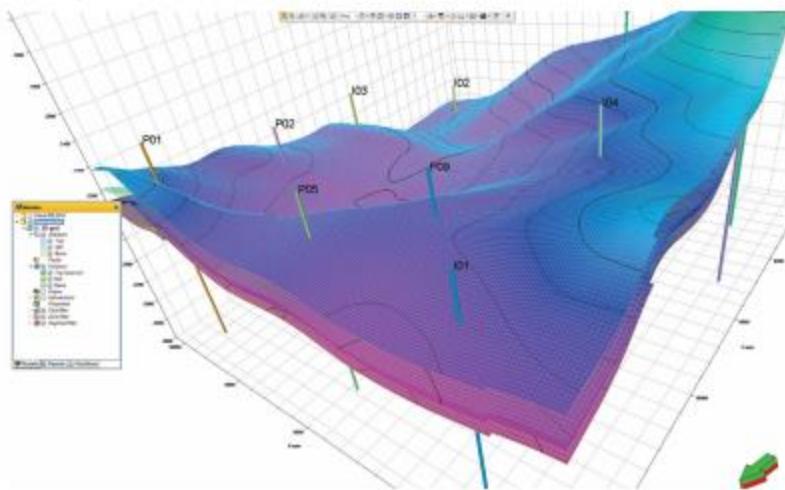


Figure 63. Skeleton grid with horizons

Copyright © 2017 Schlumberger. All rights reserved.

Vertical subdivision

The **Make horizons**, **Make zones**, and **Layering** tools are used to subdivide the reservoir based on the geological characterization or dynamic flow zones.

- Horizons: The **Make horizons** tool usually defines the main geological depositional units of the 3D grid. Usually, these units are the layers or flow zones identified and interpreted on seismic data. The **Make horizons** tool samples input surfaces into the 3D grid. A *horizon* in Petrel is a surface that is a part of the 3D grid.
- Zones: The **Make zones** tool defines the subunits of the 3D grid. These subunits are equivalent to reservoir flow zones. The **Make zones** tool inserts additional horizons (and zones) into the 3D grid by inserting isochores up or down from the horizons that were input previously. Isochores can be gridded thickness maps, or they can be calculated directly from well tops. You also can define zones as specific thickness intervals or percentages of the main zone.
- Layering: This step makes the final vertical resolution of the 3D grid. To keep the modeling simple, you can skip the **Make zones** tool and just do layering to obtain a suitable resolution for the simulation. The

assumption is that the surfaces that you insert define the stratigraphic layering.

The figure shows where these functionalities are located on the Petrel ribbon.

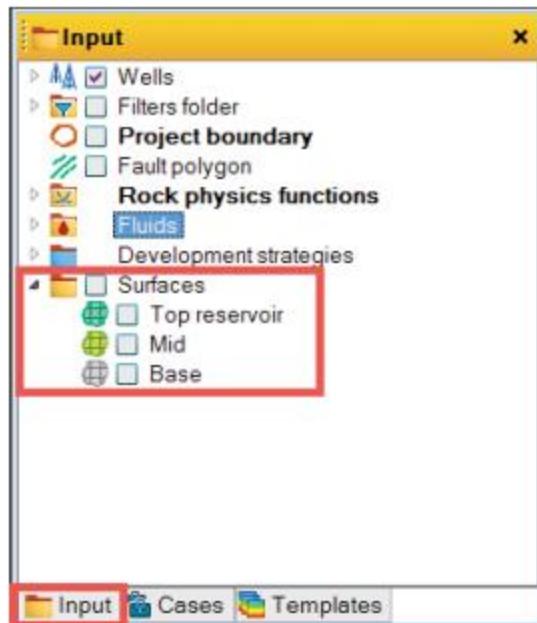


Figure 64. Horizon, Zones, and Layering functionality on the **ribbon**

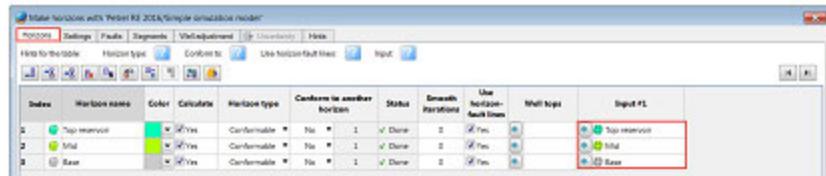
Procedure — Make horizons



1. On the **Reservoir Engineering** tab, in the **Simple gridding** group, click **Horizons** to open the **Make horizons** dialog box.
2. Insert the number of rows to match the number of surfaces required to subdivide the grid vertically. Click **Append items in the table**.



3. Select the surfaces and insert  interpretations or interpreted structural surfaces.



Multiple drop  allows you to insert multiple items into the **Input #1** column simultaneously. Make sure that your input surfaces are sorted in the correct stratigraphic order in the **Input** pane, then select the first item and click  in the **Input #1** column.

All surfaces are inserted in the order in which they appear in the **Input** pane. Horizon names are updated according to input names. If necessary, you can overwrite them.

Horizon types

To specify the type of horizon, choose one of these options in the Horizon type column.

- Erosional: Horizons below this horizon are truncated.
- Base: Horizons above this horizon are truncated.
- Conformable: Horizons are truncated by erosional, base, and discontinuous horizons. Lower conformable horizons are truncated by the upper conformable horizons in the Make horizons process.
- Discontinuous: The horizon is both a base and erosional type. Horizons below and above this horizon are truncated.

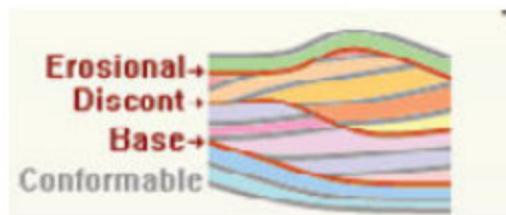


Figure 65. Horizon types

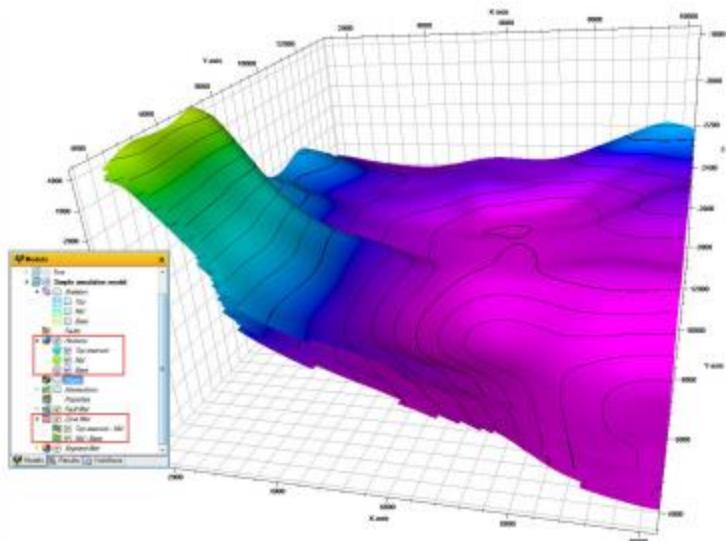
Make horizons output

The **Make horizons** dialog box is used to subdivide the 3D grid vertically using points, horizons interpretation, lines, and surfaces as inputs. The **Make zones** dialog box is used to subdivide the zones created using the **Make horizons** dialog box. It also can be used to create zones using existing thickness maps, bypassing the Make horizons process.

Horizons folder: The generated horizons from the Make horizons process are stored in the Horizons folder in the 3D grid model.

Fault filter: The fault filter allows you to visualize the fault throw for one horizon at a time. It is used mainly for mapping purposes.

Zone filter: A zone in Petrel is defined as the thickness between two horizons. It is created in the **Make horizon** dialog box or the **Make zones** dialog box. The zone filter allows you to visualize the edges and the I and J intersections of the 3D grid, zone-by-zone. The zone filter also can be applied to properties in the 3D grid.



Copyright © 2017 Schlumberger. All rights reserved.

Figure 66. Horizons displayed in a 3D window

Layering

After the **Make horizons** and the **Make zones** tools are run, use the **Layering** tool to vertically subdivide the reservoir.

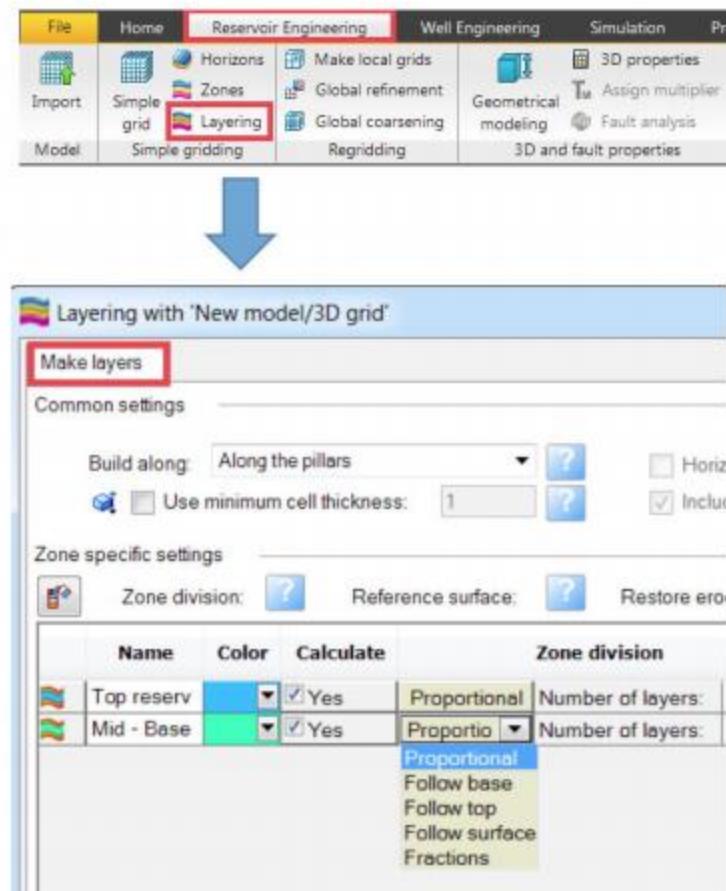


Figure 67. Layering options

Copyright © 2017 Schlumberger. All rights reserved.

There are five ways to apply layering in Petrel.

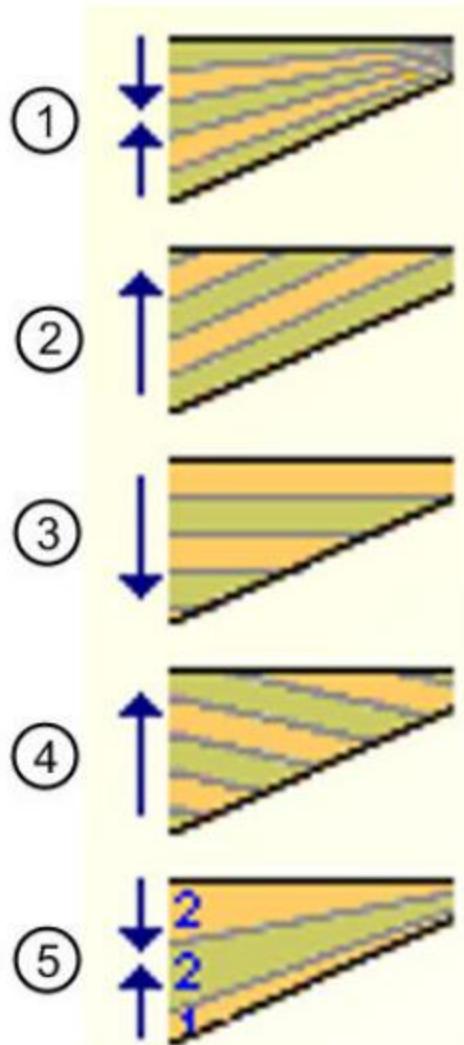


Figure 68. Five ways to apply layering

- 1 *Proportional*: Divides the zone into a given number of layers of the same thickness.
- 2 *Follow Base*: Divides the zone into cell layers with a constant user-controlled thickness. The cell layers are parallel to the base of the zone.

- 3 *Follow top*: Divides the zone into cell layers with a constant user-controlled thickness. The cell layers are parallel to the top of the zone.
- 4 *Follow surface*: Defines layers in the 3D grid parallel to a specific surface. This option preserves the correct layering in eroded zones and enables a more accurate property distribution.
- 5 *Fractions*: Divides each layer proportionally into smaller units.

The zone division should reflect the horizon type. For example, if a horizon is the Base type, the zone above the horizon should be one of these types: Follow base, Proportional, or Fractions.

Proportional and Fractions layering types give the least pinched-out layers; therefore, they usually are best for simulation grids.

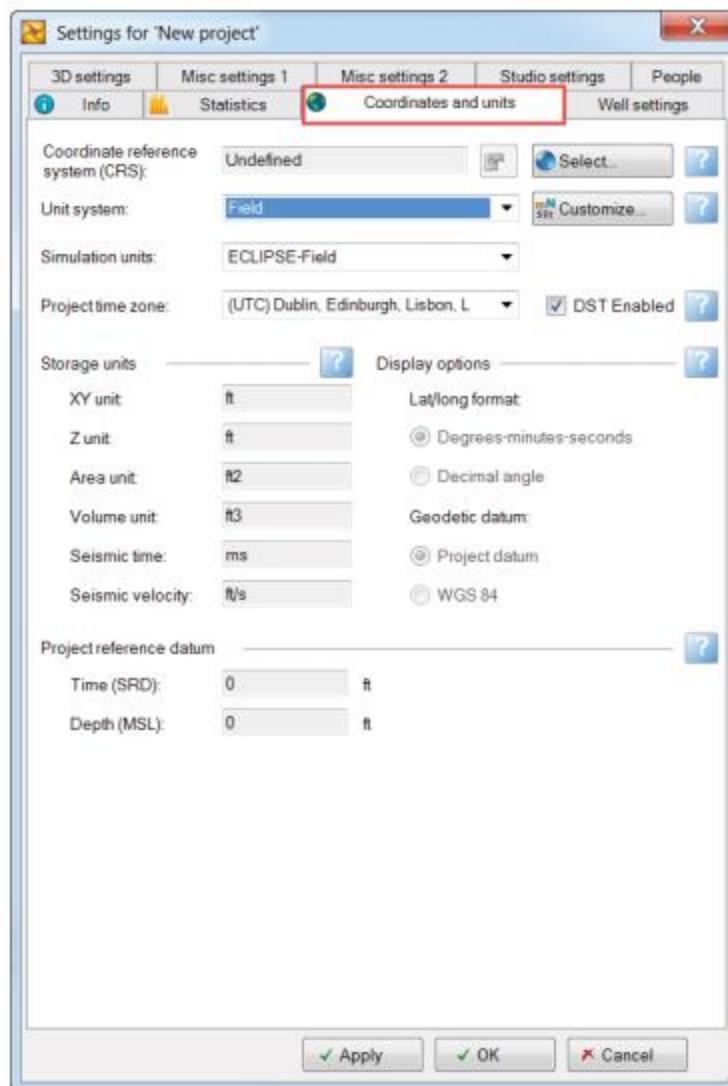


Procedure — Make a simple grid from scratch

This procedure explains the workflow steps for creating a simple grid.

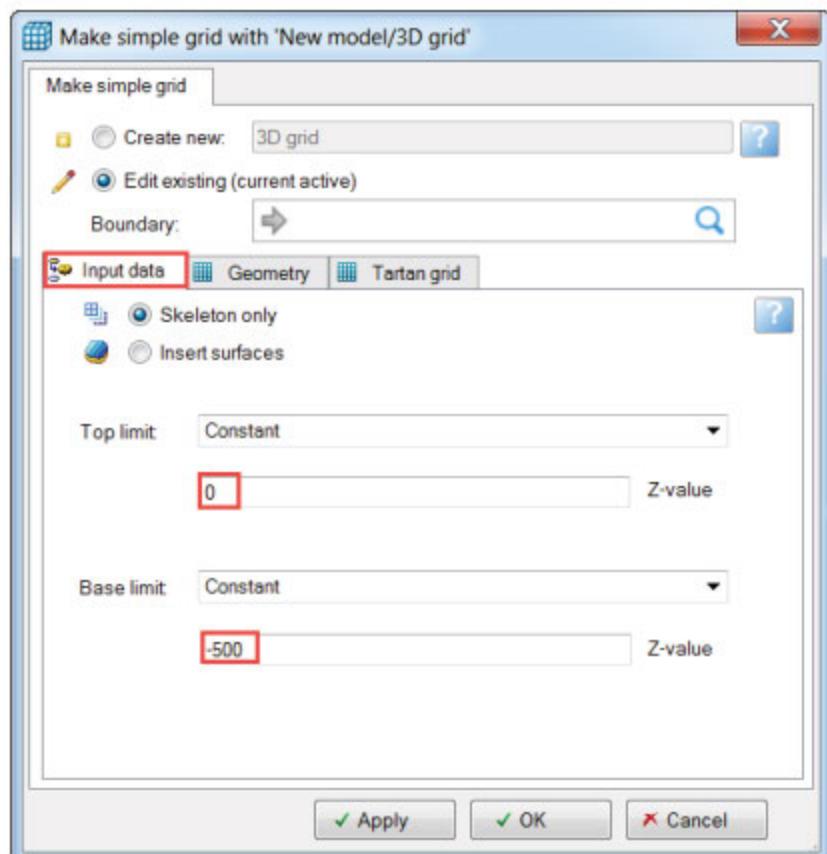
1. On the **File** menu, click **New project** to create a new Petrel project.
2. On the **File** menu, click **Project setup** and then click **Project settings** to open the project **Settings** dialog box.

3. On the **Coordinates and units** tab, set **Unit system** to **Field**. Click **OK**.



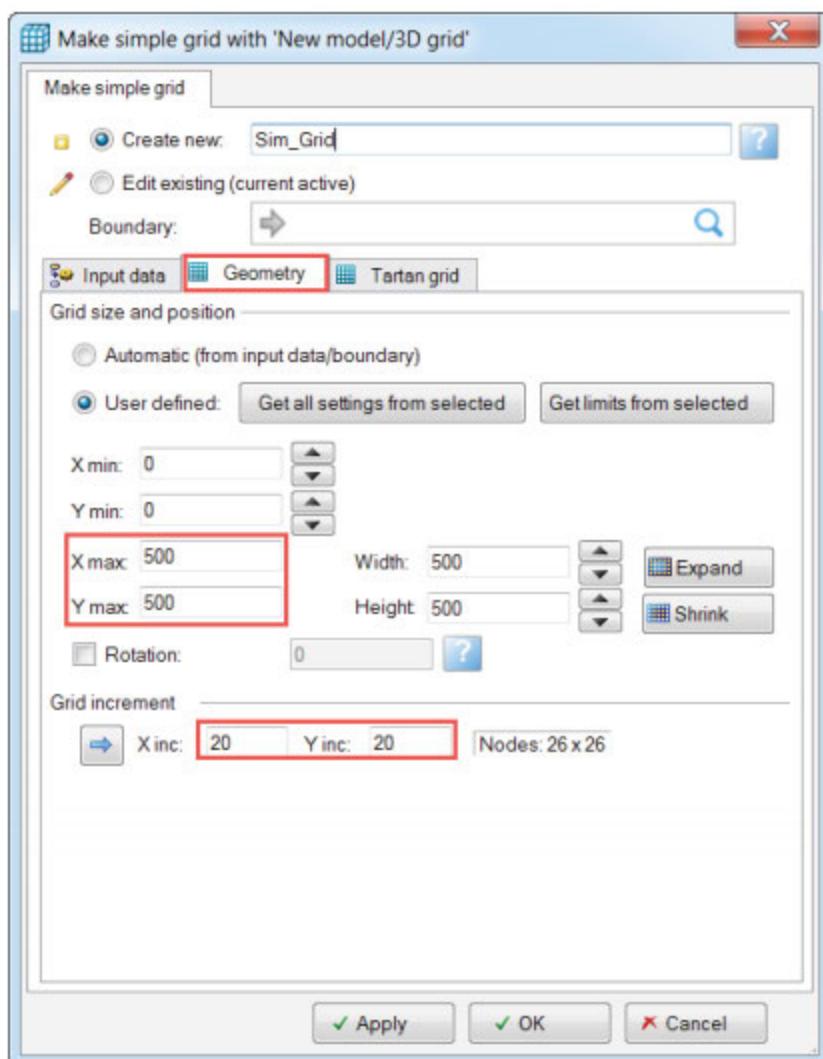
4. On the **Reservoir Engineering** tab, in the **Simple gridding** group, click **Simple grid** to open the **Make simple grid** dialog box. Ensure that you are in the Reservoir and Production engineering perspective.
5. In the **Make simple grid** dialog box, enter the grid name.

6. On the **Input data** tab, click **Skeleton only**. Enter a Top limit constant of 0 and a Base limit constant of - 500 ft, for example.



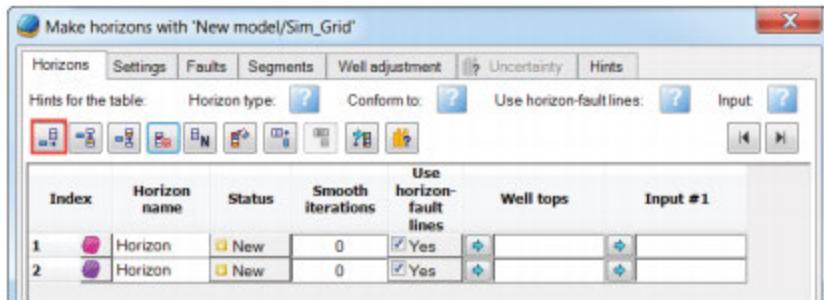
7. On the **Geometry** tab, select the **User defined** check box. Enter Xmin and Ymin, and then enter Xmax and Ymax.

8. Enter a **Grid increment** for both the X and Y directions. Click **OK**.



9. In the **Models** pane, right-click **New model** and click **Expand recursive** to open the folder.
10. Convert the skeleton grid to a surface by right-clicking the **Top** in the **Skeleton** folder and clicking **Convert to surface**.
11. Repeat Step 10 for the **Base skeleton**.

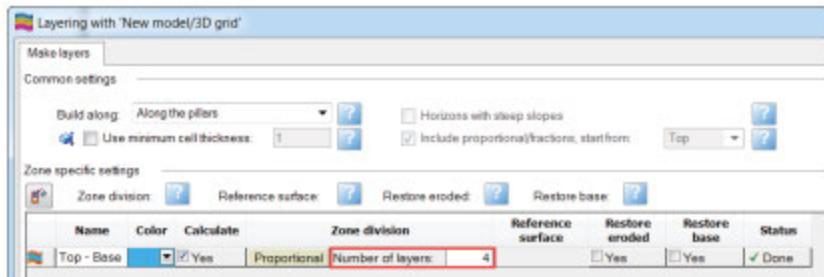
12. Open the **3D window**. On the **Home** tab, in the **Insert** group, click **Window** and then click **3D window**. In the dialog box that opens, click **Continue spatially unaware**.
13. Activate the Top and Base surfaces in the **Input** pane to display them in the **3D window**.
14. On the **Reservoir engineering** tab, in the **Simple gridding** group, click **Horizons** to open the **Make horizons** dialog box.
15. Add two rows by clicking the **Append items in the table** button.



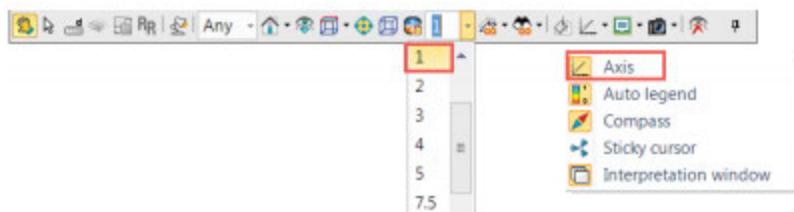
16. Highlight the Top surface and click under the **Input#1** column to insert the surface from the **Input** pane into the data box. Repeat this step for Base surface. Click **OK**.

Observe that the **Zones** and **Layering** buttons are enabled after you create the horizons.

17. Click the **Layering** button to open the **Layering** dialog box. Enter the number of layers in the data field under the **Zone division** column. Click **OK**.



18. On the **Window** toolbar, set the Z-scale to 1. Click **Show/hide axis** to display the axis.



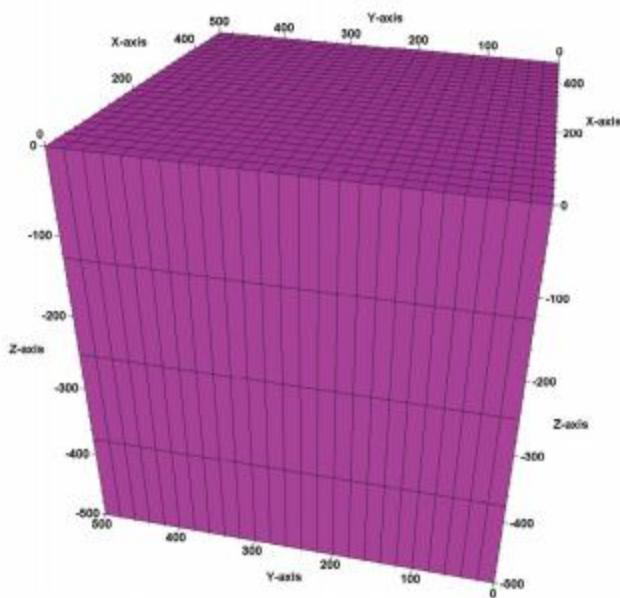
19. Open Geometrical modeling. Make these selections:

- **Create new**
- Settings method: Cell width
- Property template: Cell X dimension

20. Click **OK**.

Cell width is defined as the distance between the centers of opposing cell faces. It is equivalent to the lengths of the four edges that link the cell faces. The cell widths are referred to in ECLIPSE as DX and DY.

21. Activate the Cell width property under the Properties folder to display it in the **3D window**.



The next stage is to create grid properties, fluid models, and the rock physics function.

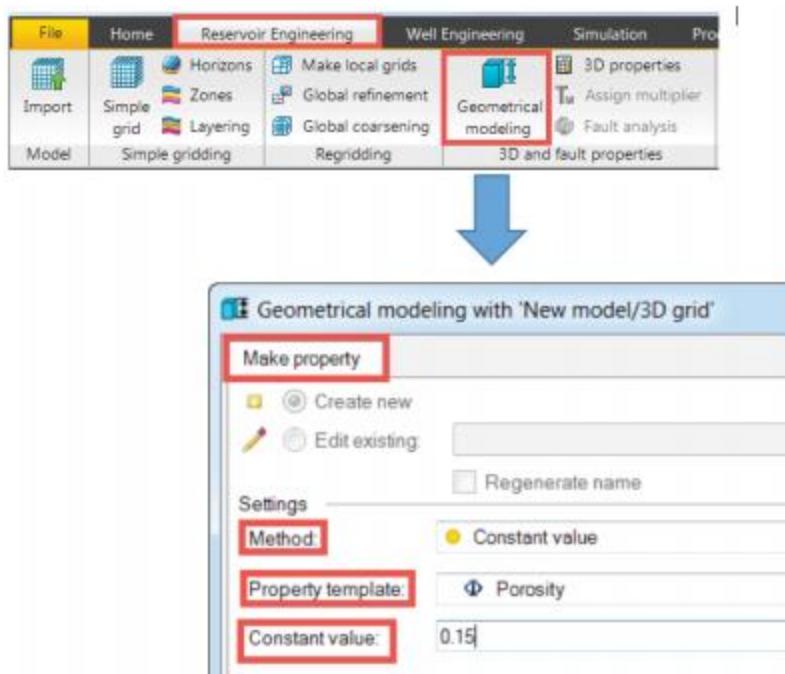
Copyright © 2017 Schlumberger. All rights reserved.



Procedure — Define grid properties

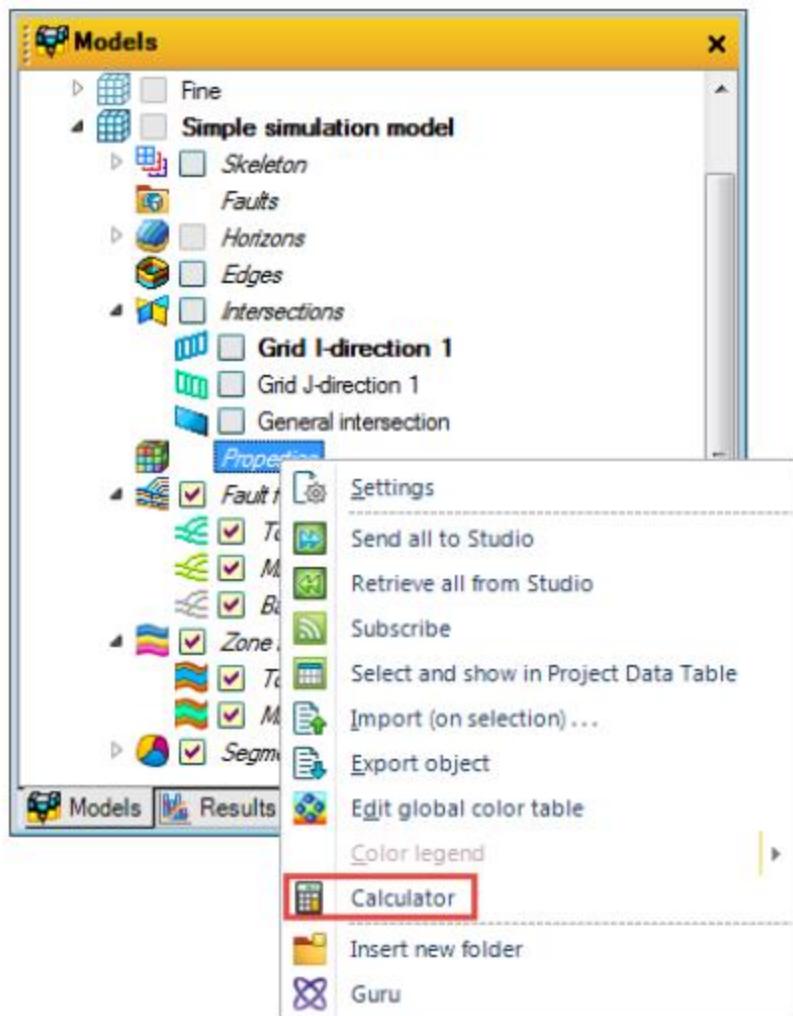
Property modeling based on well data is done using the Petrophysical modeling process, which is the recommended workflow. However, to assign a constant property or properties that can be defined geometrically, you can use the **Property calculator** or the Geometrical modeling process. Later in the course, you use the Geometrical modeling process to QC 3D grids by computing cell angles, cell inside out, and volumes.

- To create Constant properties, use the Geometrical modeling process. On the **Reservoir Engineering** tab, in the **3D and fault properties** group, click **Geometrical modeling** to open the **Geometrical modeling** dialog box.



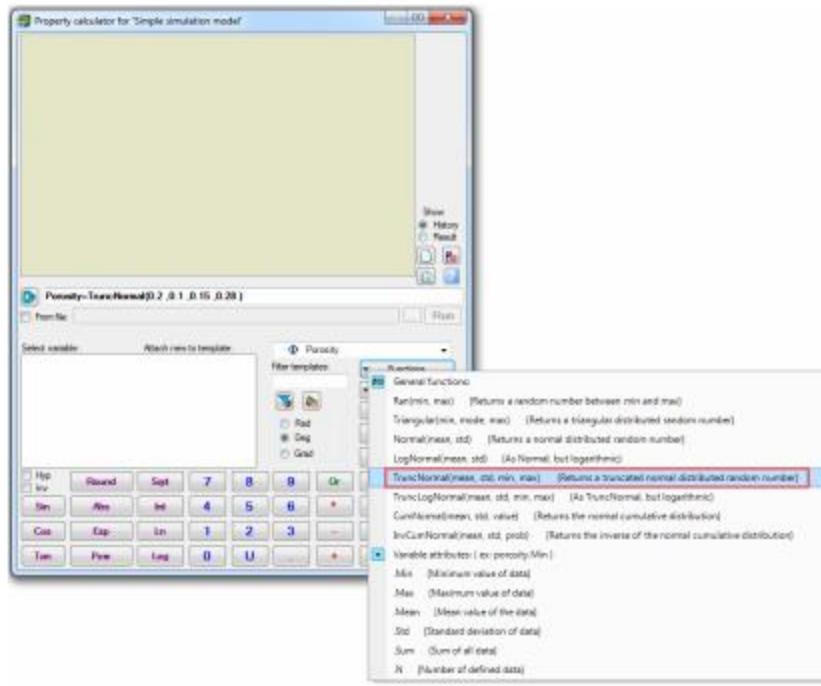
Alternatively, you can use the **Property calculator**, which gives the option of using some simple logic to assign properties to the grid. For example, you can use `TruncLogNormal(mean,std,min,max)` to quickly create porosity and also assign porosity values based on the layer or zones.

2. Right-click the Properties folder in the **Models** pane and click **Calculator**.



Copyright © 2017 Schlumberger. All rights reserved.

- Enter the expression Porosity =, click **Functions**, and select TruncNormal(mean, std, min, max) from the list.

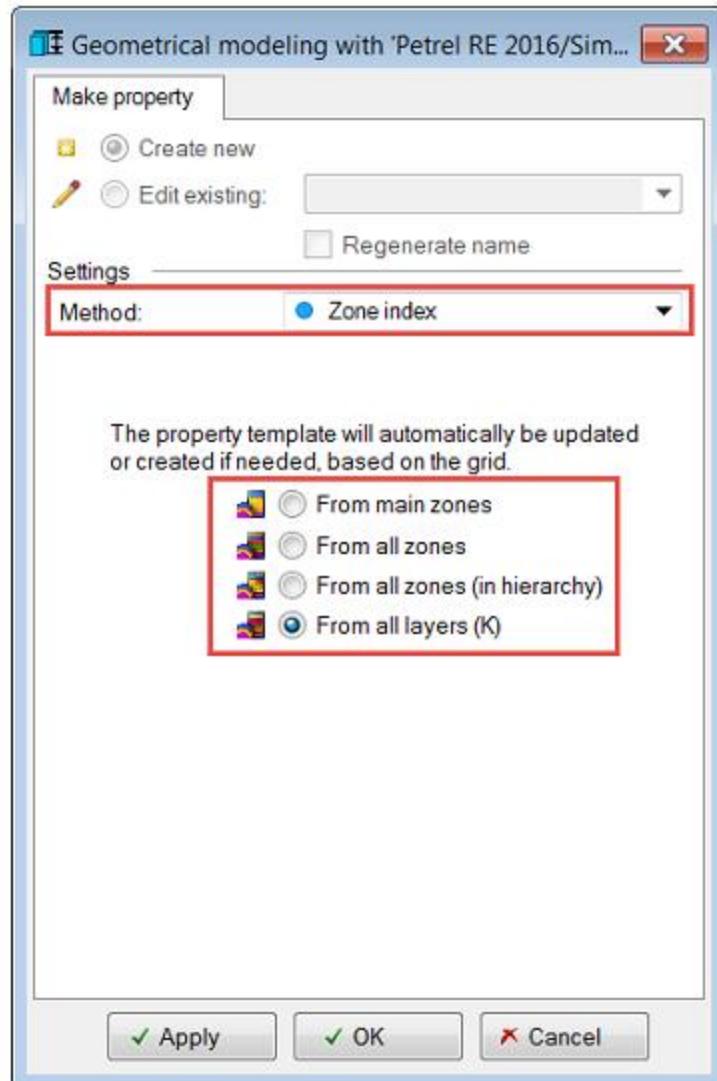


- Enter the values for the for mean, min, and max. Click **ENTER** to generate the porosity property.



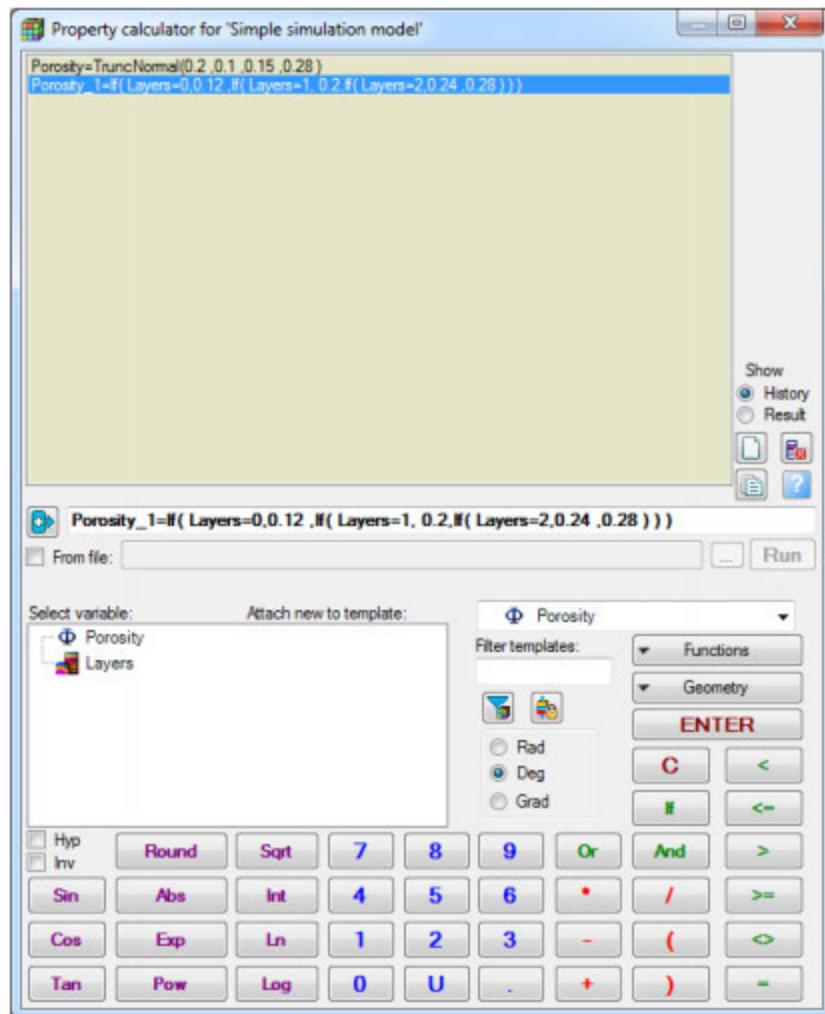
Procedure — Assign the Porosity property based on the reservoir zones or layers

1. Create Zone or Layer index property using Geometrical modeling.



Copyright © 2017 Schlumberger. All rights reserved.

2. Use a simple logic expression in the **Property calculator** to assign the porosity property based on a zone or layers. Alternatively, use K in the equation to replace Layers. K is the grid K index.



Copyright © 2017 Schlumberger. All rights reserved.

Exercises — Make a simple grid with a simulation fault

In these exercises, you create a simple simulation grid using interpreted structural surfaces.

You start by creating a simple simulation grid. You define the grid properties using the **Property calculator**. You then add a vertical fault on the grid using the Polygon editing tools and assign a fault property (transmissibility multipliers) to the fault.

Workflow

1. Make a simple grid using the existing interpreted surfaces.
2. Create porosity using a simple logic from the **Property calculator**.
3. Assign the porosity property based on the grid layers.
4. Make a simulation fault and assign a transmissibility multiplier to the simulation fault.

Data

Use the file `Simple_simulation_model_exercise.pet` in the `Dataset\Projects\Module-3 Simple simulation model` folder.

Copyright © 2017 Schlumberger. All rights reserved.

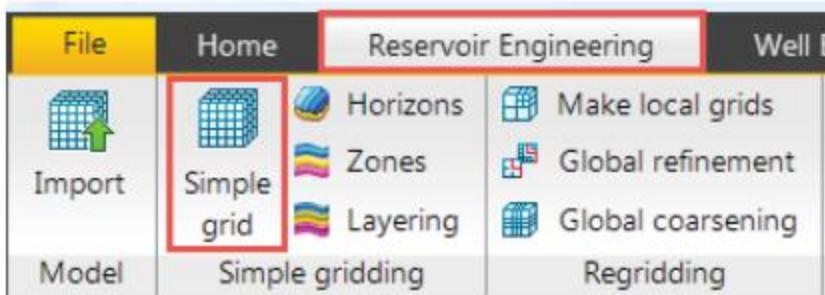


Exercise 1 Make a simple grid

In this exercise, you focus on creating a simple simulation grid using the Make simple grid process.

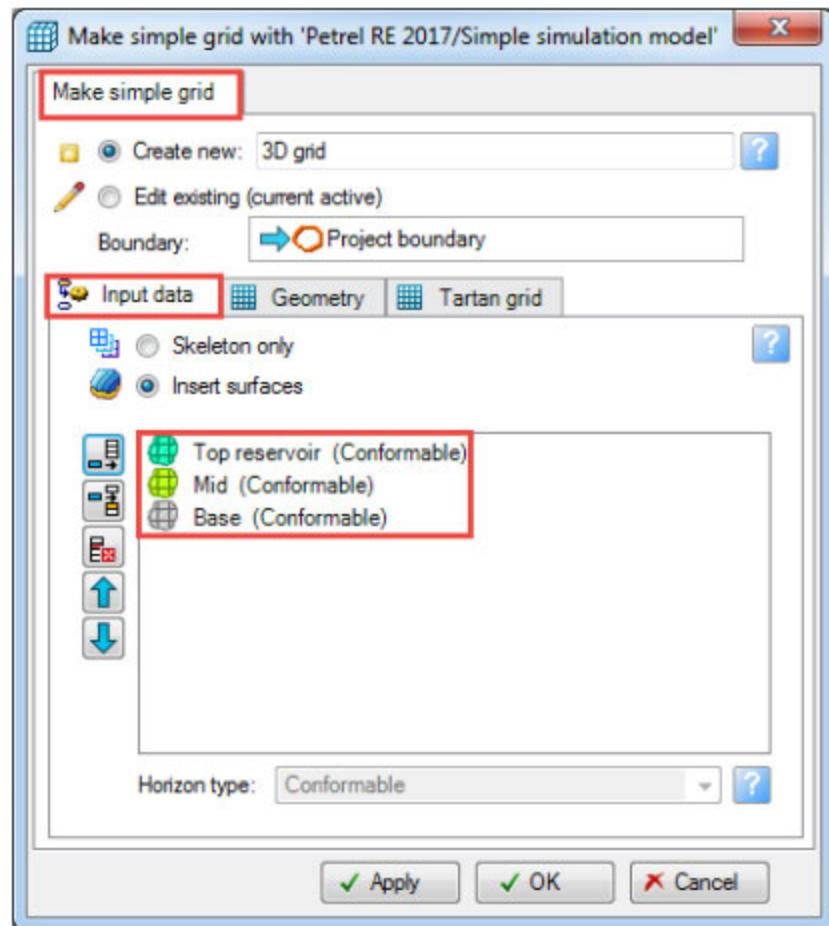
1. Open the `Simple_simulation_model_exercise.pet` project.

2. On the **Reservoir Engineering** tab, in the **Simple gridding** group, click **Simple grid**.



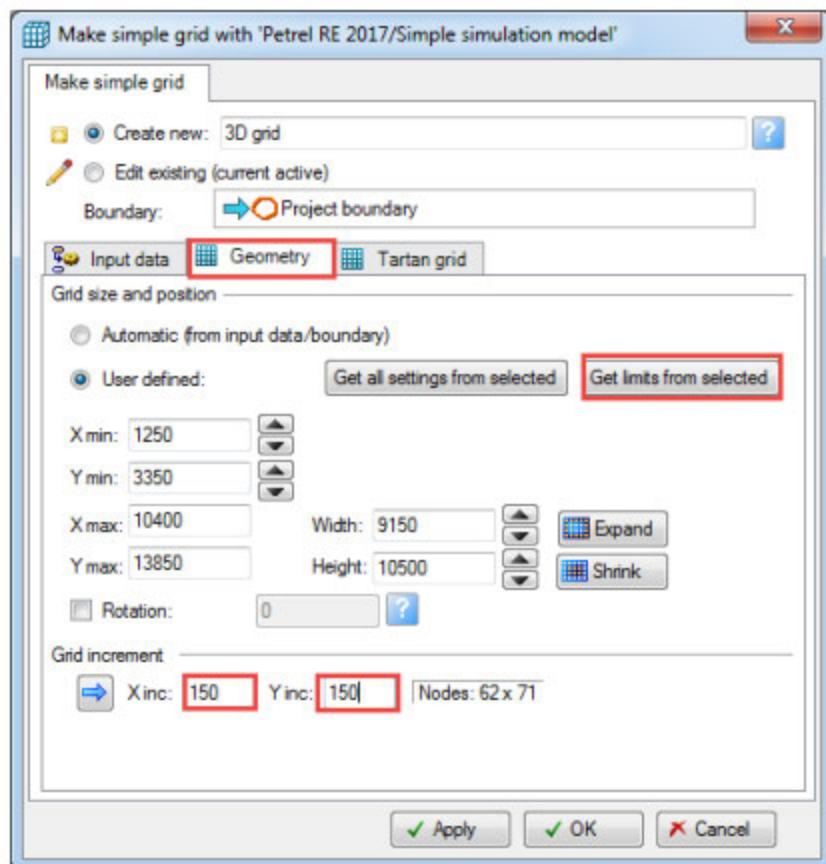
3. For this exercise, in the **Make simple grid** dialog box, rename the grid to Simple simulation model.
4. On the **Input data** tab, click **Insert surfaces**.

5. Use the **Append item in the table** button to insert Top reservoir, Mid, and Base surfaces from the Surfaces folder in the **Input** pane. Ensure that the structural surfaces are in increasing depth order.



6. Insert the Project boundary polygon from the **Input** pane into the **Boundary data** field.
7. On the **Geometry** tab, click **Get limits from selected**.

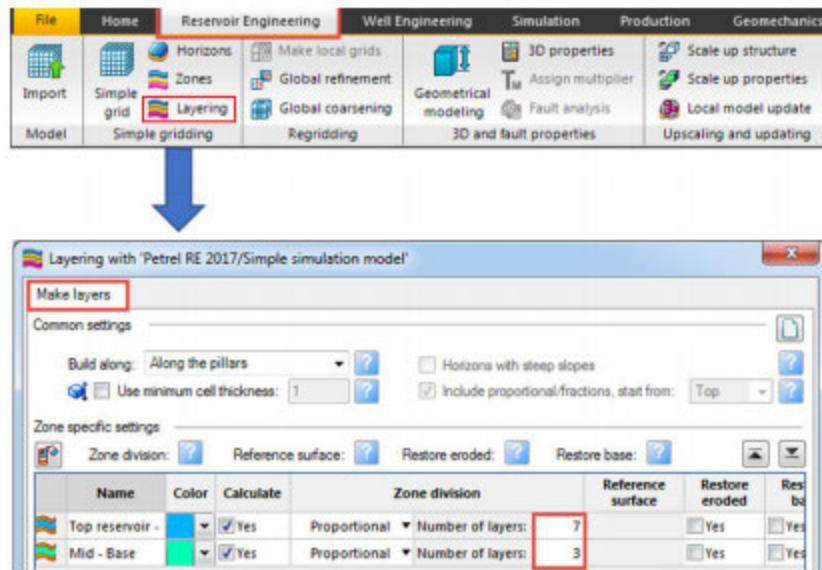
Because the project boundary is selected, Petrel uses it to define the grid limits.



8. Change **Grid increment** to 150 m in both the X and Y direction.
Click **OK**.

9. Your new grid is stored in the **Models** pane.
- View the horizons of the generated grid in a **3D window**.

- Subdivide the grid vertically. On the **Reservoir Engineering** tab, in the **Simple gridding** group, click **Layering** to open the **Layering** dialog box.



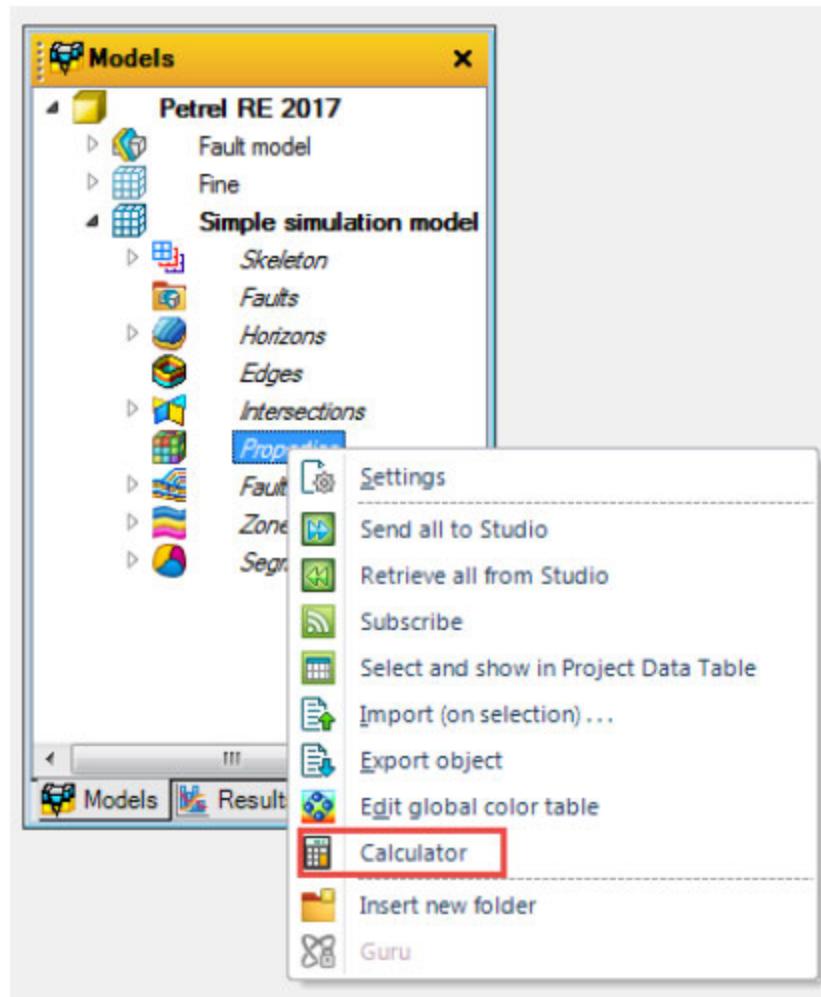
- Divide the zone between the Top reservoir and Mid horizons into seven layers and the zone between the Mid and Base horizons into three layers.
- Click **OK**.
- Display the Top reservoir and Base horizons. Select the **Edges** check box to view the generated layers.
- Open the **Settings** dialog box for the Horizons under the Simple simulation model in the **Models** pane. Change the contour lines to grid lines.
- Open the **Inspector**. Click any cell on the grid to view the cell information in the **Inspector**.



Exercise 2 Use simple logic to assign a porosity property to the grid using the Property calculator

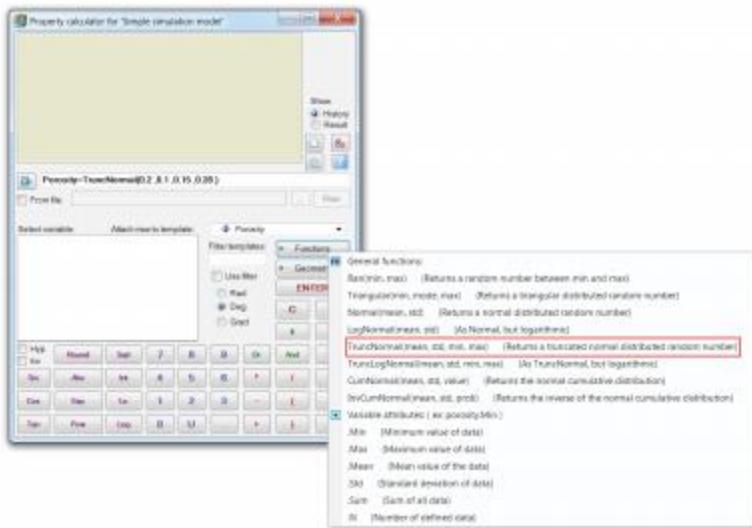
In this exercise, you learn how to assign a porosity property to the 3D grid using simple logic in the **Property calculator**. For the purpose of this exercise, you use TruncLogNormal (mean, std, min, max) to create porosity quickly and also assign porosity values to the simulation grid based on the grid layers.

1. In the **Models** pane, right-click the Properties folder and click **Calculator**.



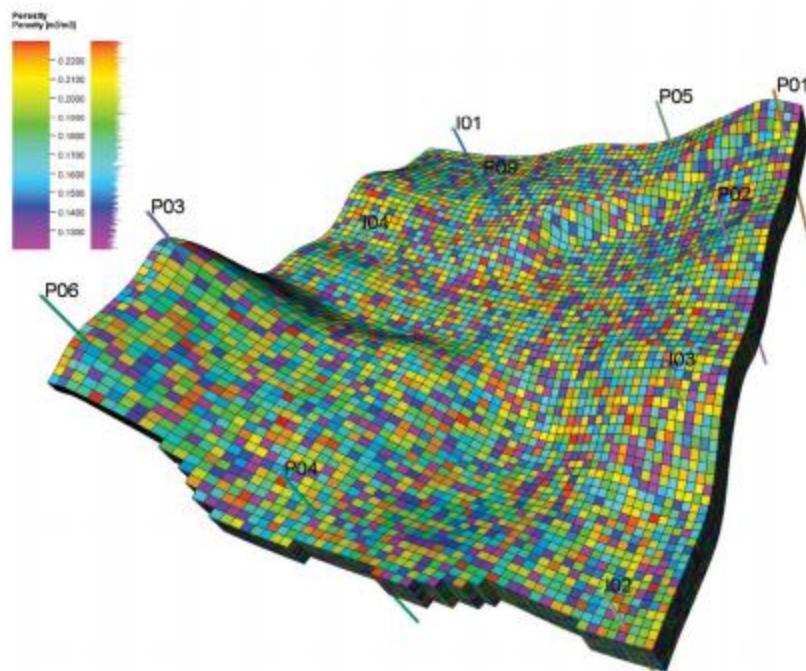
Copyright © 2017 Schlumberger. All rights reserved.

- Enter the expression Porosity =, click the **Functions** button, and select TruncNormal (mean, std, min, max) from the list. Ensure that the Porosity template is selected.



- Enter the values for mean = 0.2, std = 0.1, min = 0.12, and max = 0.23.
- Click **ENTER** in the **Property calculator** to generate the porosity property.
- Open the **3D window**. Activate the porosity property in the **Properties** folder in the **Models** pane to display it in the **3D window**.

- Right-click the displayed porosity and click **Adjust color table** from the mini toolbar. Click **Yes** in the **Petrel message** dialog box to scale the property values based on colors.



Copyright © 2017 Schlumberger. All rights reserved.

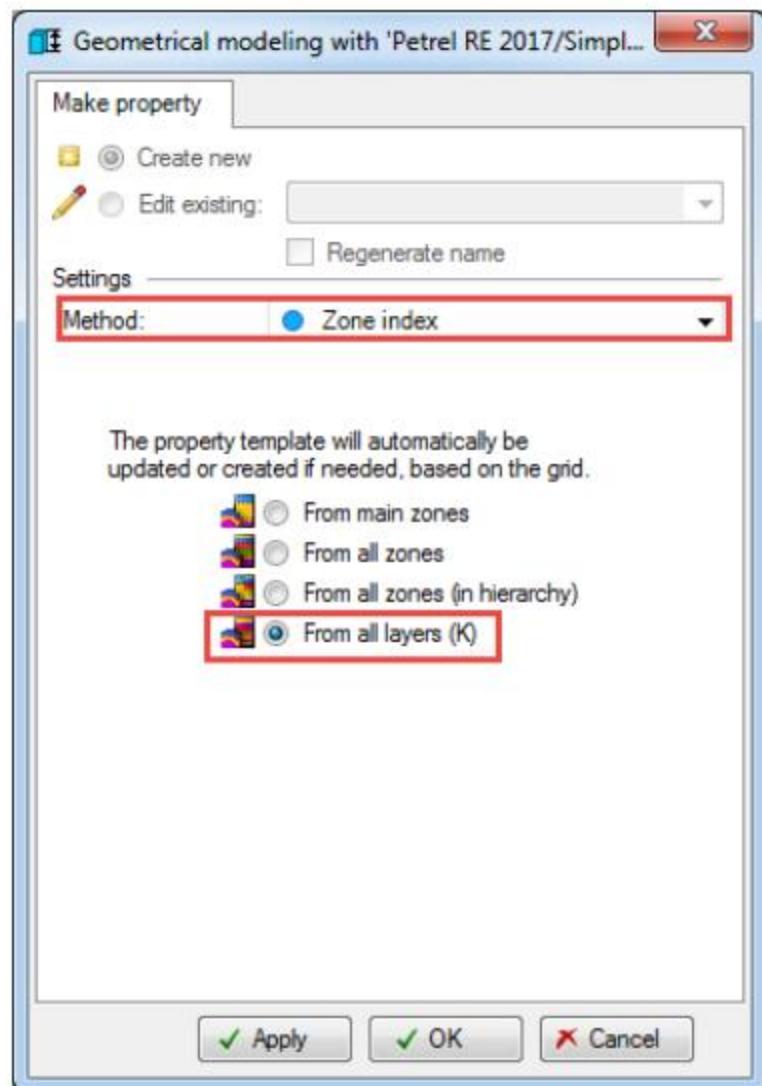
Exercise 3 Assign the porosity property based on the grid layers



In this exercise, you create a layer index property using the Geometrical modeling process. You then use a simple logic expression in the **Property calculator** to assign the porosity property to the simulation grid based on layers.

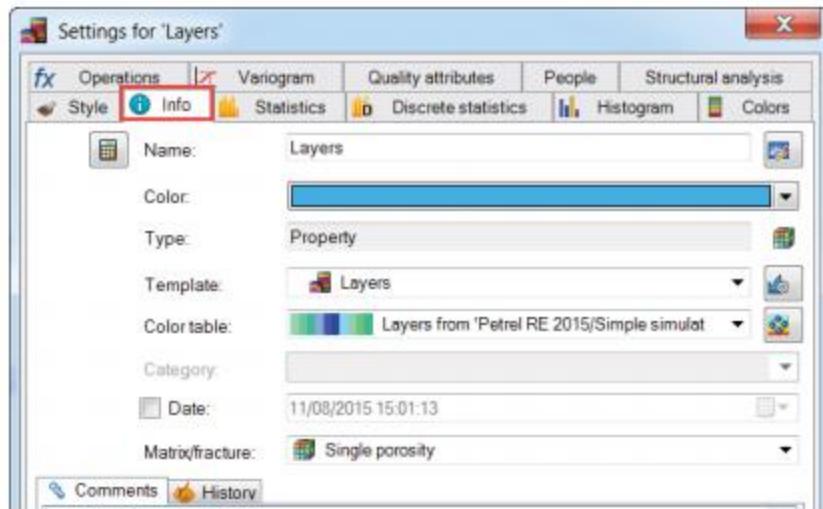
- On the **Reservoir Engineering** tab, in the **3D and fault properties** group, click **Geometrical modeling**. Ensure that you activate the Simple simulation model grid.
- In the **Method** list, click **Zone index**.
- Click the **From all layers (K)** option for this operation and click **OK**.

The layers property generated is stored in the **Properties** folder in the **Models** pane of the active grid (Simple simulation model).

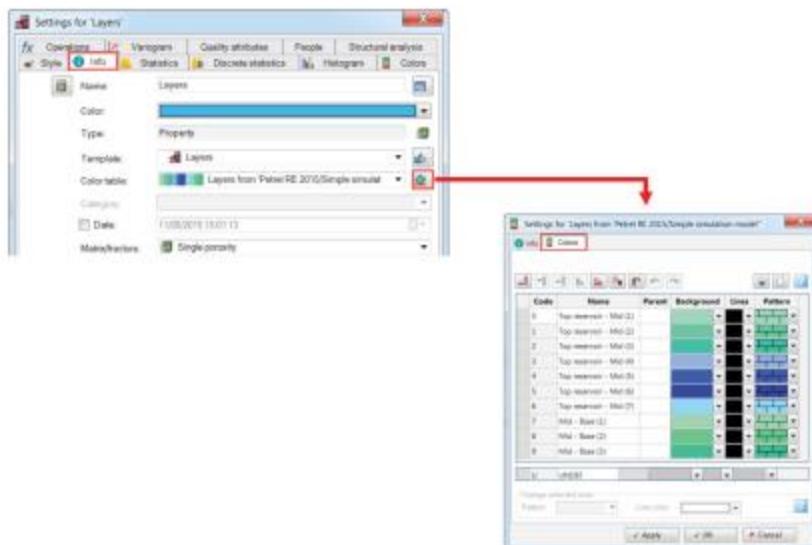


Copyright © 2017 Schlumberger. All rights reserved.

4. Right-click the Layers property and click **Settings** to open the **Settings** dialog box.

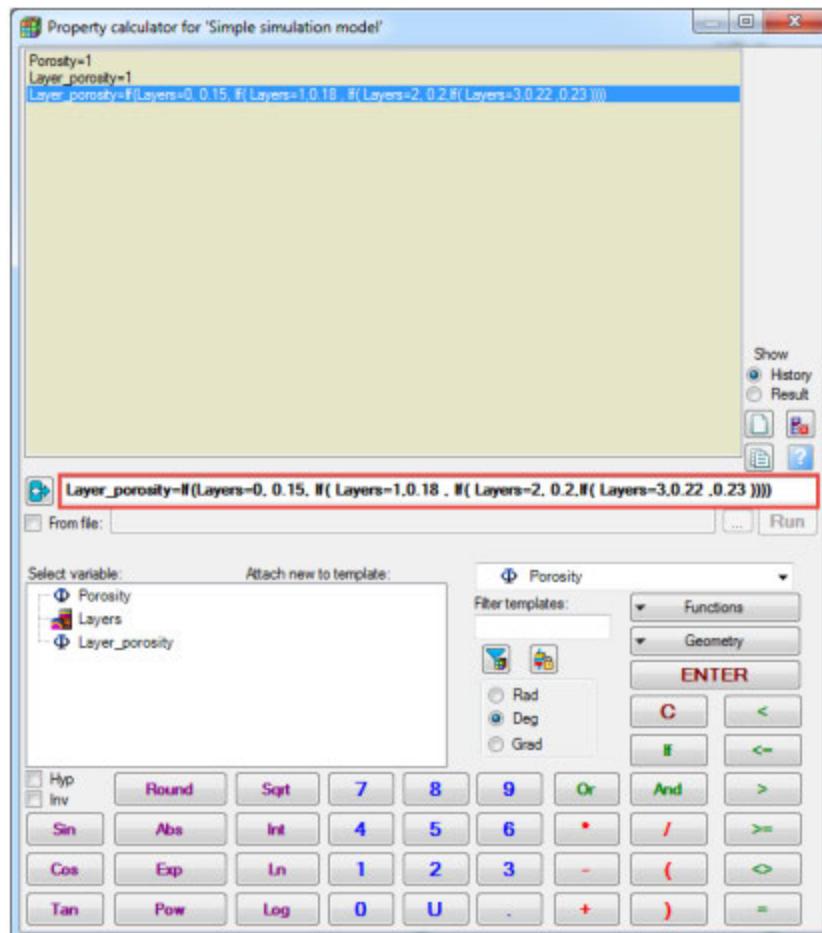


5. On the **Info** tab, click the button next to the **Color table** list as shown in the figure to open the **Settings for 'Layers'** dialog box. Review the settings and close the dialog box.



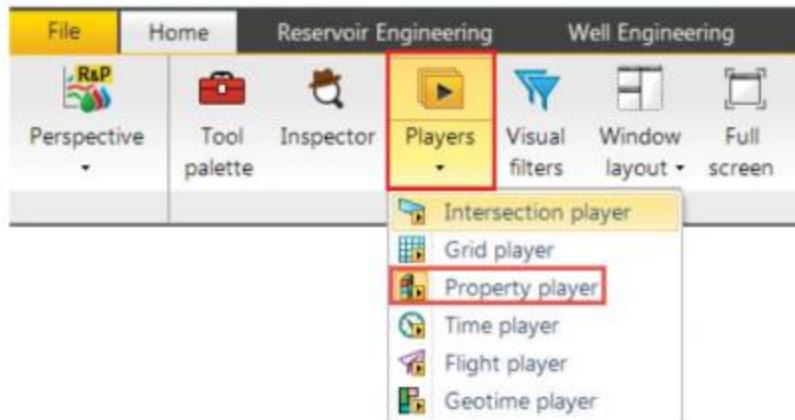
6. Right-click the Properties folder and click **Calculator**.

7. Enter a simple IF logic expression in the **Property calculator** to assign the porosity property to the simulation grid based on layers as shown in the figure: [Layer_porosity=If(Layers=0, 0.15, If(Layers=1,0.18 , If(Layers=2, 0.2, If(Layers=3,0.22 ,0.23)))). Make sure that the **Porosity** template is selected. Click **ENTER**. Close the **Property calculator**.

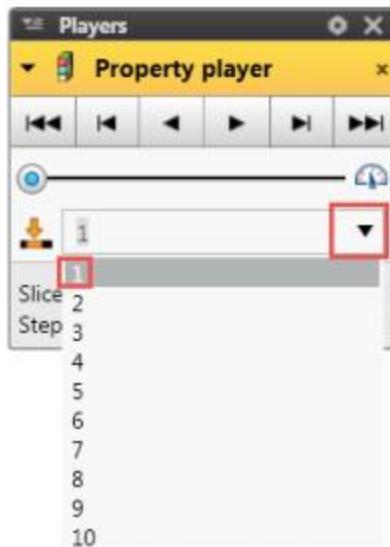


8. Activate Layer_porosity to display it in the **3D window**.

9. On the **Home** tab, in the **View** group, click **Players** and then click **Property player**.

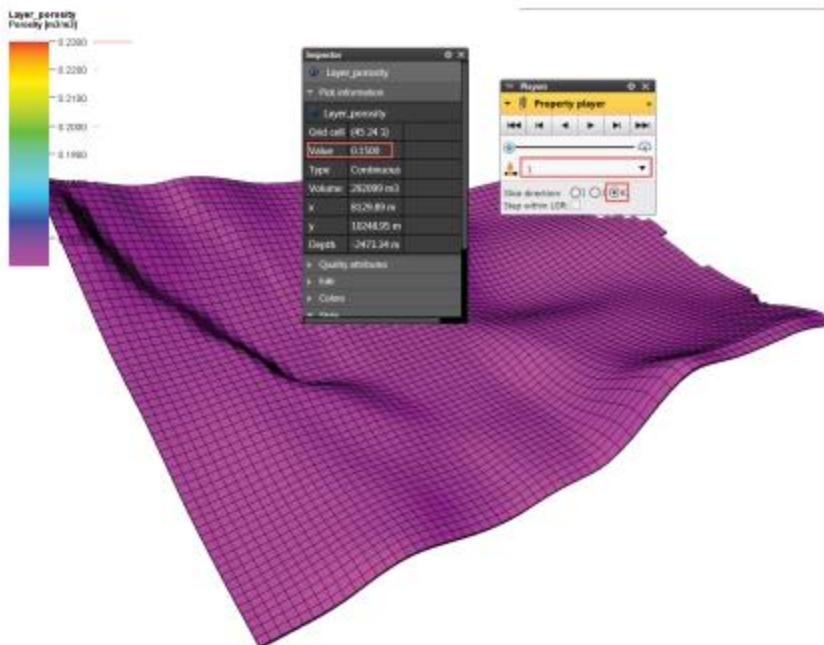


10. Activate **Slice direction: K**, click the **Index** button, and select layer 1.



Copyright © 2017 Schlumberger. All rights reserved.

- On the **Home** tab, in the **View** group, click **Inspector** and then click any cell in layer 1 to quickly validate the layer assigned porosity value.



- Save the project.

Copyright © 2017 Schlumberger. All rights reserved.



Exercise 4 Make a simulation fault

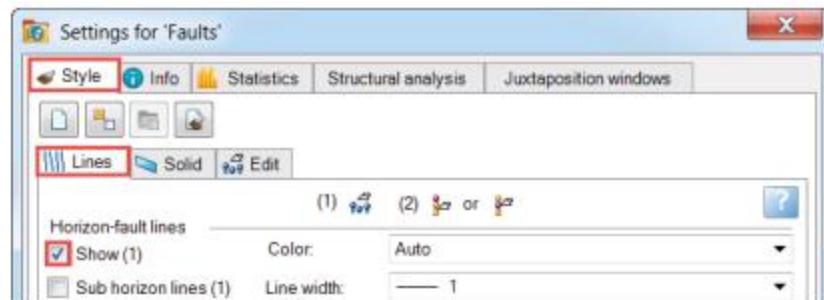
In this exercise, you add a simulation fault to the simple grid. You use the **Add points to polygon** tool on the **Polygon editing Tool Palette**. You digitize a polygon and convert the digitized polygon into a simulation fault. This exercise does not capture the throw of the fault. If you want to add a fault with throw, use the **Fault modeling** and **Pillar gridding** tools.

- Open a new **3D window**. Select to view the Top reservoir horizon along with faults from the Fine grid.
- Observe the structural displacement caused by the fault throws.

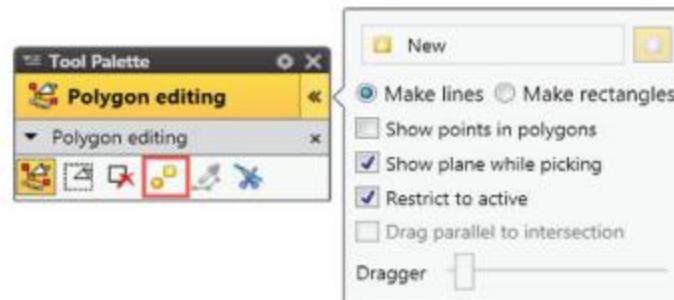
For example, across Fault5, the throw is so large that the structure is compartmentalized (no communication between the compartments). Your task is to add the vertical fault using the digitizing tool available

in Petrel to manually digitize the fault on the Simple simulation model grid to model this scenario.

3. Open a new **2D window**.
4. In the **2D window**, display these objects:
 - Base horizon from the Simple simulation model
 - Project boundary from the **Input** pane
 - Fault5 from the Fine grid
5. If you cannot see the fault, open the **Settings** dialog box for the **Faults** folder of the Fine grid and select **Show(1)** in the **Horizon-fault lines** section. Click **Apply** and zoom in to view the fault in the **2D window**. Close the dialog box.



6. Digitize the polygon that will be used to define the fault plane.
 - a. On the **Reservoir Engineering** tab, in the **Utilities** group, click to open the **Polygon editing Tool Palette**.
 - b. In the **Polygon editing Tool Palette**, click **Add points to polygon[A]**.
 - c. In the dialog box that opens, ensure that you are creating a new polygon. If not, click **Start new polygon**.



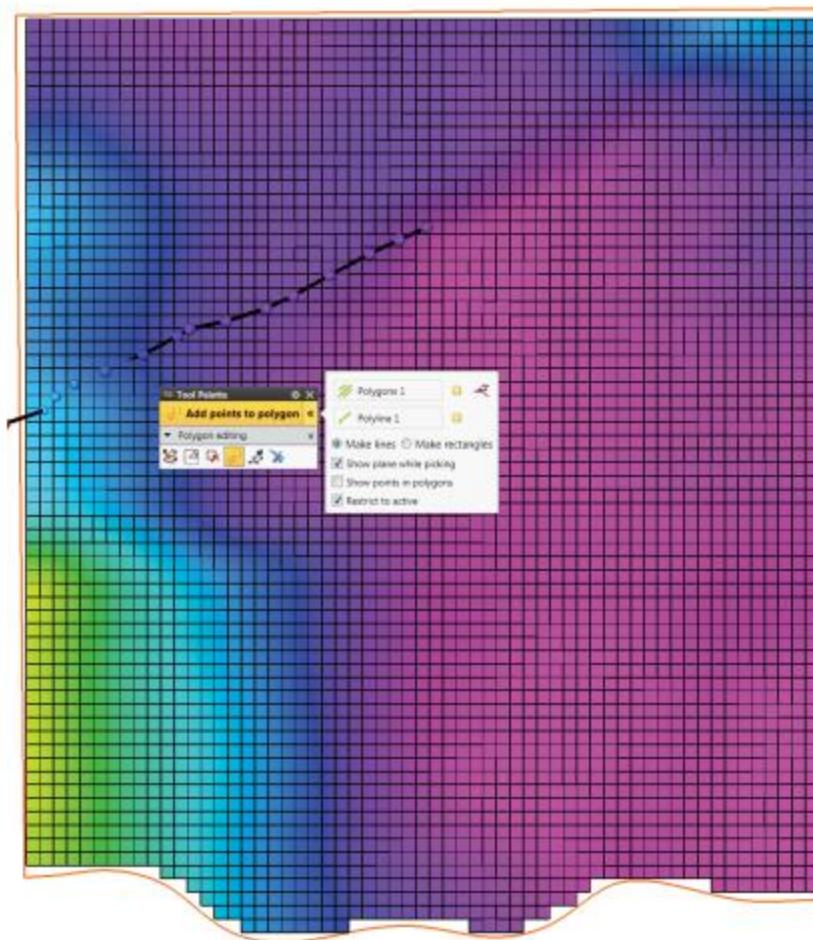
- d. Click in the active **2D window**. Digitize along the displayed Fault 5.

The distance between the digitized points should be small, so you can capture the curvature of the fault.

- e. Make sure that the fault is extended all the way to the boundary by adding a point in the grid cell closest to the project boundary.

The digitized polygon is stored in the **Input** pane.

- f. On the **Window** toolbar, click **View**  to deactivate the **Polygon editing** tool.



Copyright © 2017 Schlumberger. All rights reserved.

7. Click **Polygons 1**. Use shortcut key F2 to rename the polygon to Fault polygon.

8. Make sure that the Simple simulation model grid is active in the **Models** pane.
9. In the **Input** pane, right-click **Fault polygon** and click **Create simulation (grid) fault**.

The new fault is stored in the **Faults** folder in the Simple simulation model grid in the **Models** pane.

Exercise 5 Assign a transmissibility multiplier

In this exercise, you assign a transmissibility multiplier to the new fault you just created to model a barrier to flow. Assume that there is no flow through the fault (the fault is sealed).

1. On the **Reservoir Engineering** tab, in the **3D and fault properties** group, click **Assign multiplier**.
2. In the Faults folder in the Models pane, click **Fault polygon**.
3. In the **Fault analysis** dialog box, clear the **Use default** check box.
4. Enter a constant transmissibility multiplier of 0 (sealing fault) and click **OK**.
5. View the generated fault transmissibility multiplier in the Fault properties folder under the Faults folder of the Simple simulation model grid
6. Save the project.



Lesson 2 Rock physics functions

In this lesson, you learn how to make rock physics functions.

The Make rock physics functions process is used to create functions that represent the physics of the rock and the interaction between rock and fluids. This process allows you to make saturation functions and rock compaction functions.

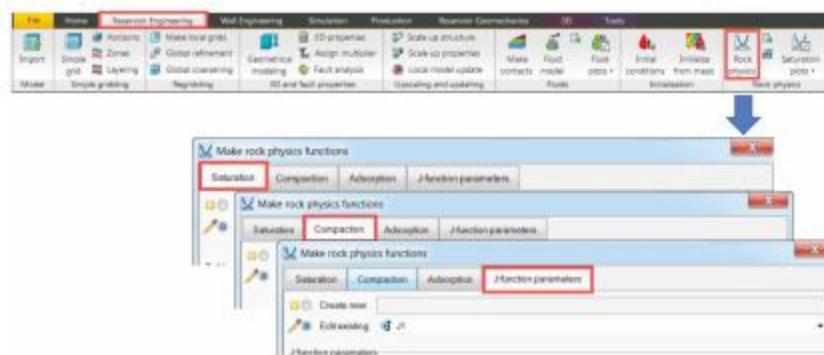


Figure 69. Make rock physics functions functionality

Copyright © 2017 Schlumberger. All rights reserved.

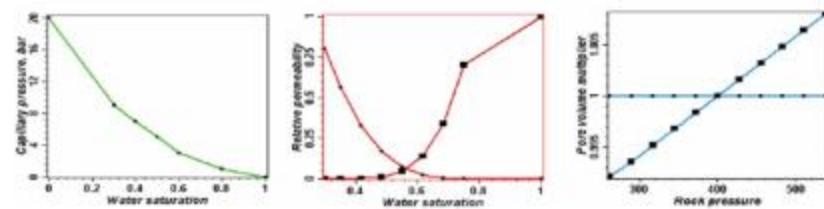


Figure 70. Capillary pressure curves (left), Relative permeability (middle), and Rock compaction (right)

Saturation functions: Purpose of saturation functions

Saturation functions are saturation-dependent inputs to the simulation, made up of two main elements: relative permeability and capillary pressure.

This data is used to calculate

- the mobility of a particular phase in the presence of other phase and to identify critical end-points that determines residual saturation
- the initial phase distribution and size of the transition zones
- the force driving fluid from the pore space

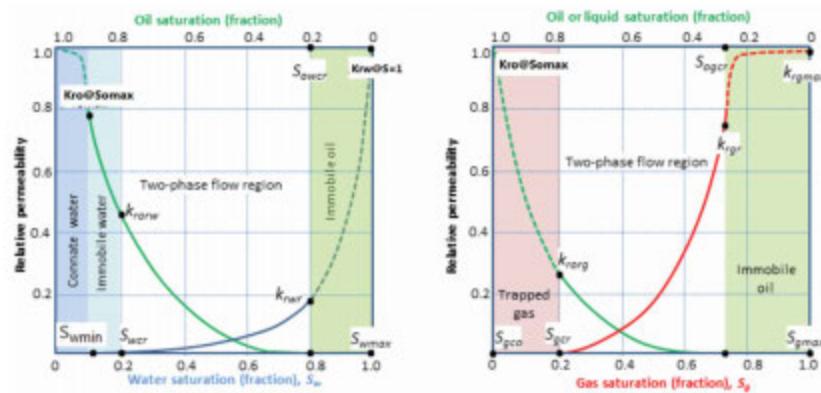


Figure 71. Saturation functions

Copyright © 2017 Schlumberger. All rights reserved.

Permeability

Permeability is a measure of the ease with which a formation permits a fluid to flow through it. To be permeable, a rock must have interconnected porosity (pores, vugs, capillaries, fissures, or fractures). Greater porosity usually corresponds to greater permeability, but not always. Pore size, shape, and continuity, as well as the amount of porosity, influence the formation permeability.

The permeability of a given rock to the flow of a single homogeneous fluid is a constant, provided the fluid does not interact with the rock. Permeability determined for a single homogeneous liquid is termed absolute, or intrinsic, permeability (k). The unit of permeability is Darcy.

One Darcy is the permeability that allows the flow of one cubic centimeter per second of a fluid of one centipoise viscosity through a cross-sectional area of one square centimeter under a pressure gradient of one atmosphere per centimeter. A Darcy is a large unit so, in practice, the millidarcy (md) is the most common unit.

Relative permeability

When two or more immiscible fluids (for example, oil and water) are present in the formation, their flows interfere. The effective permeability to oil flow (k_o) or water flow (k_w) is reduced.

Furthermore, the summation of effective permeability always is less than or equal to the absolute permeability (k). The effective permeability depends not only on the rock itself but also on the relative amounts and properties of the different fluids in the pores. In a given rock, k_o and k_w vary as oil and water saturations, S_o and S_w , vary.

Relative permeability is the ratio of the effective permeability to the absolute (single homogeneous fluid) permeability. Thus, for an oil-water system, the relative permeability to water, k_{rw} , is equal to k_w/k ; similarly, the relative permeability for oil, k_{ro} , is equal to k_o/k .

It is apparent that relative permeability usually is expressed in percent or fractions and never exceeds the unity (1 or 100%).

Oil-water relative permeability

The figure shows the relative permeability curves for a water-wet formation that contains only oil and water. The values of k_{rw} and k_{ro} vary with the saturation. The curves illustrate that at high oil saturation, k_{ro} is large and k_{rw} is small; the oil flows easily and little water flows.

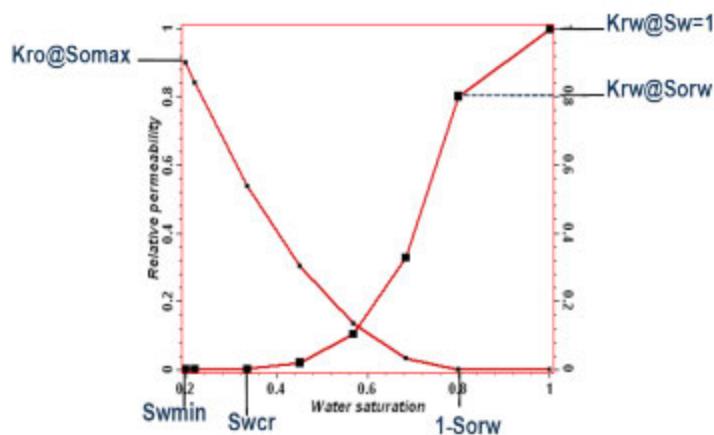


Figure 72. Relative permeability curves for a water-wet formation that contains only oil and water

At high water saturations, k_{ro} is small and k_{rw} is large; now the water flows easily and little oil flows. The shapes of the relative permeability diagrams depend on the formation and pore characteristics, as well as on the fluids present (water, oil, gas).

Make a rock physics function

This function allows you to create relative permeability curves using Corey correlations.

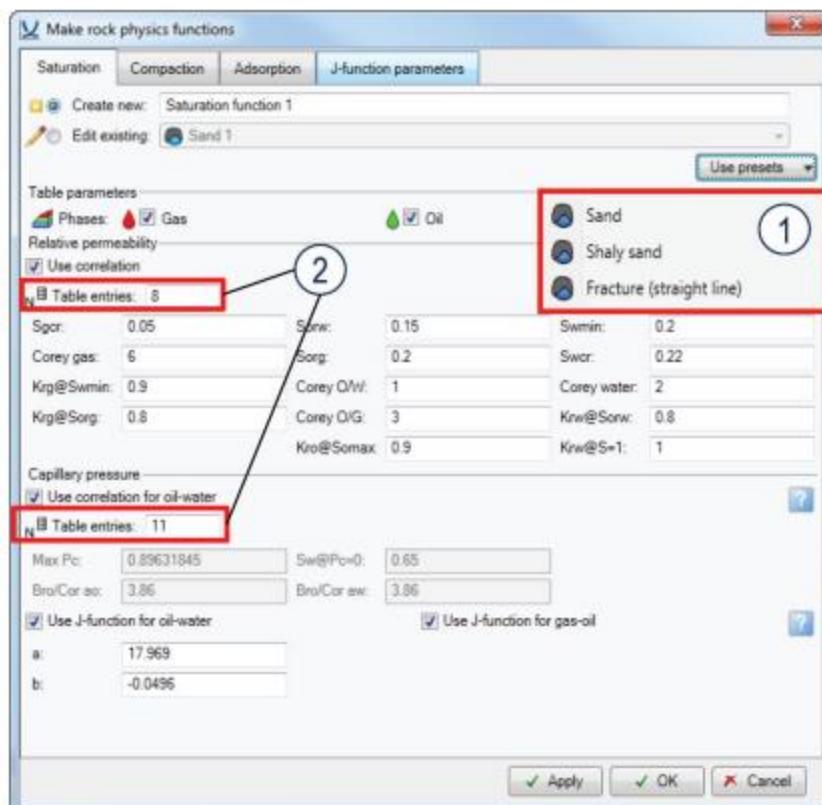


Figure 73. Make rock physics functions dialog box

- 1 Preset values are available for sand, shaly sand, and fracture (for dual permeability models).
- 2 The number of table entries controls the size of the tables passed to the simulator.

Make rock physics functions: Saturation tab

To create a saturation function or edit an existing one, you can use the **Use presets** list to insert endpoint values automatically or you can input endpoints manually if they available from the lab.

Click **Apply** or **OK** to create saturation functions based on correlation.

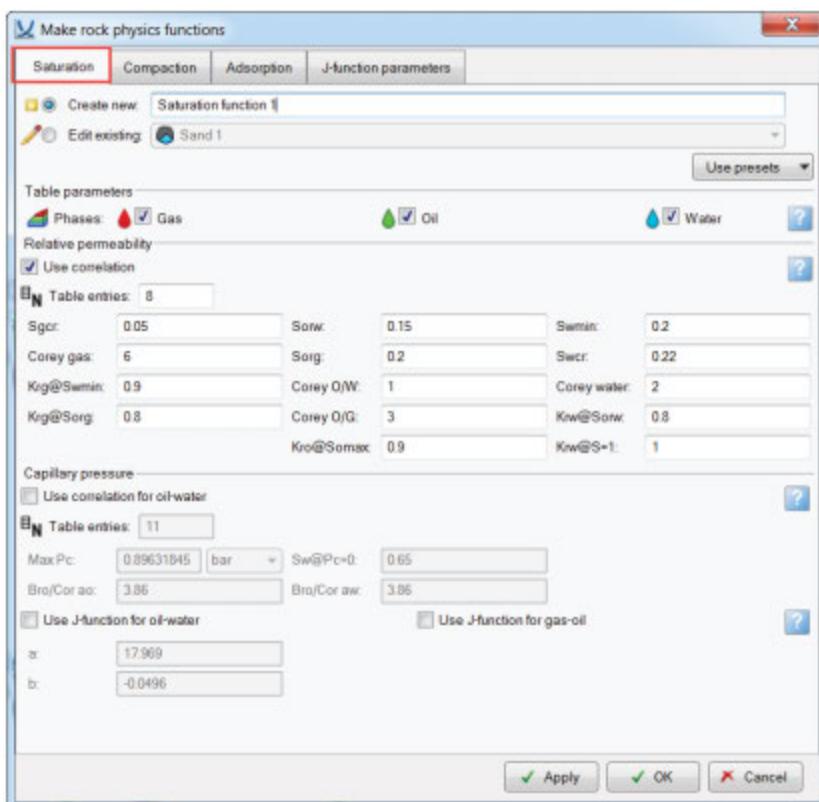


Figure 74. Saturation tab in the Make rock physics functions dialog box

The input to Corey correlations depends on the fluid phases you select to include in the model.

For gas, select

- Sgcr: The critical gas saturation
- Corey Gas: Corey gas exponent for values between the minimum water saturation and (1 - Sorg)

- $K_{rg@Swmin}$: Relative permeability value of gas at the minimum water saturation
- $K_{rg@Sorg}$: Relative permeability value of gas at the residual oil saturation

For oil, select

- S_{orw} : Residual oil saturation to water. Note that $(1 - S_{orw}) > S_{wcr}$.
- S_{org} : Residual oil saturation to gas. Note that $(1 - S_{org}) > S_{wcr}$.
- $Corey\ O/W$: Corey oil exponent for values between the minimum water saturation and $(1 - S_{orw})$
- $Corey\ O/G$: Corey oil exponent for values between minimum water saturation and $(1 - S_{org})$
- $K_{ro@Somax}$: Relative permeability of oil at the maximum value of oil saturation

For water, select

- S_{wmin} : Minimum water saturation
- S_{wcr} : Critical water saturation. This value must be greater than or equal to the minimum water saturation.
- $Corey\ Water$: Corey water exponent for values between S_{wcr} and $(1 - S_{orw})$
- $K_{rw@Sorw}$: Relative permeability of water at the residual oil saturation value
- $K_{rw@S=1}$: Relative permeability of water at a saturation value of unity

Irrducible saturations

When the K_{ro} value reaches zero, the oil remaining in the pore space is immovable. The corresponding value of oil saturation at which this event occurs is the residual oil saturation (S_{orw}).

The K_{rw} curve also becomes zero at an S_w value, indicated in the figures as S_{wcr} . At this saturation, only oil flows in the formation. The residual water is immobile.

In a water-wet formation, there always is a specific amount of water held in the pores by capillary forces. Oil at pressures encountered in formations cannot displace this water, so the water saturation does not reach zero.

$S_{w\min}$ usually is referred to as the irreducible water saturation S_{wi} . S_{wi} is a function of both porosity and permeability.

When oil is produced from a formation, the relative amounts of oil and water produced at a given level depend on the relative permeabilities at the current saturation.

Capillary pressure

In a thick reservoir that contains both water and hydrocarbon columns, the saturation can vary from 100% water at the bottom of the zone to a maximum oil saturation (and irreducible water saturation) at the top. There is a gradual transition between these two extremes in saturation. The transition interval might be short for porous and permeable formations or it might be long in formations of low permeability.

When both oil and water are present in the rock pores, the water, which is the wetting phase, coats the pore walls and fills the smaller pore channels. The oil tends to accumulate in globules in the larger pores.

The surface tension of the interface between water and oil causes the pressure within the oil globules to be greater than in the water. This difference in pressure is equal to the capillary pressure. Saturation at a particular elevation is a function of the capillary pressure and the densities.

Copyright © 2017 Schlumberger. All rights reserved.

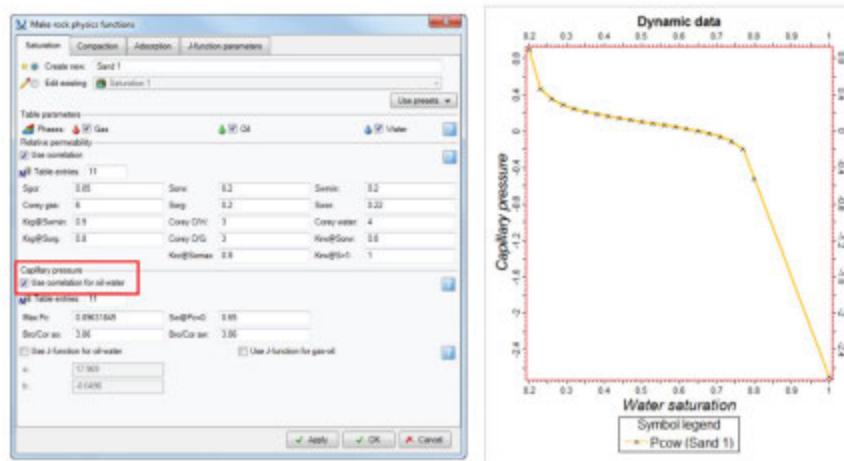


Figure 75. Rock physics function: Capillary pressure curve



Procedure — Generate a capillary pressure function for oil-water using a correlation

1. Select Use correlation for oil-water in the **Capillary pressure** section on the **Saturation** tab in the **Make rock physics functions** dialog box. This option generates a capillary pressure function using a correlation for mixed-wet reservoir rock from the simple power-law form of Brooks and Corey.
2. In Table entries, enter the total number of points required to create the capillary pressure tables. This option usually should be set to the same value as for the relative permeability tables, so that the combined table of relative permeability and capillary pressure versus saturation exported to a simulator matches the tables seen in Petrel.
3. Enter the values for the maximum capillary pressure (Max Pc), water saturation when the capillary pressure is zero (Sw@Pc=0), and pore-size distribution coefficients for oil and water (Bro/Cor ao and Bro/Cor aw, respectively).

- Max Pc: Maximum capillary pressure. Available only if **Use correlation for oil-water** is selected.
- Sw@Pc=0: Water saturation when the capillary pressure is zero. Available only if **Use correlation for oil-water** is selected.
- Bro/Cor ao: Pore size distribution for oil. Available only if **Use correlation for oil-water** is selected.
- Bro/Cor aw: Pore size distribution for water. Available only if **Use correlation for oil-water** is selected.

4. Click **Apply**.



NOTE: Petrel does not have a correlation for gas-oil capillary pressure data. If required, the data can be supplied elsewhere and input using the Rock physics spreadsheet.



NOTE: Creating saturation functions using Make rock physics functions also generates curves for gas-oil and water-oil capillary pressure versus saturation. These curves are set to zero by default.

Rock compaction tables

Petrel provides two rock compaction models: the Standard model, which has pore volume and transmissibility as a function of pressure, and the Palmer-Mansoori model, which models matrix shrinkage and swelling in coal. When the Standard model is selected, correlations and rock type determine the equation used to calculate rock compressibility.

The **Make rock physics functions** dialog box provides presets, which can be used to create rock compaction functions that are broadly representative of the rock type. Choosing a preset applies typical values for the parameters in the dialog box. Rock compaction functions are tables that show pore volume multipliers versus pressure or a single rock compressibility value used by the simulator to calculate the pore volume change. The compaction can be thought of as the reduction in the pore volume as a function of pressure.

Compaction drive can be modeled on the **Compaction** tab to make rock compaction tables from a choice of correlations.

Creating rock compaction functions using the **Make rock physics functions** dialog box also creates transmissibility multiplier versus pressure curves. These curves are set to 1.0 by default.

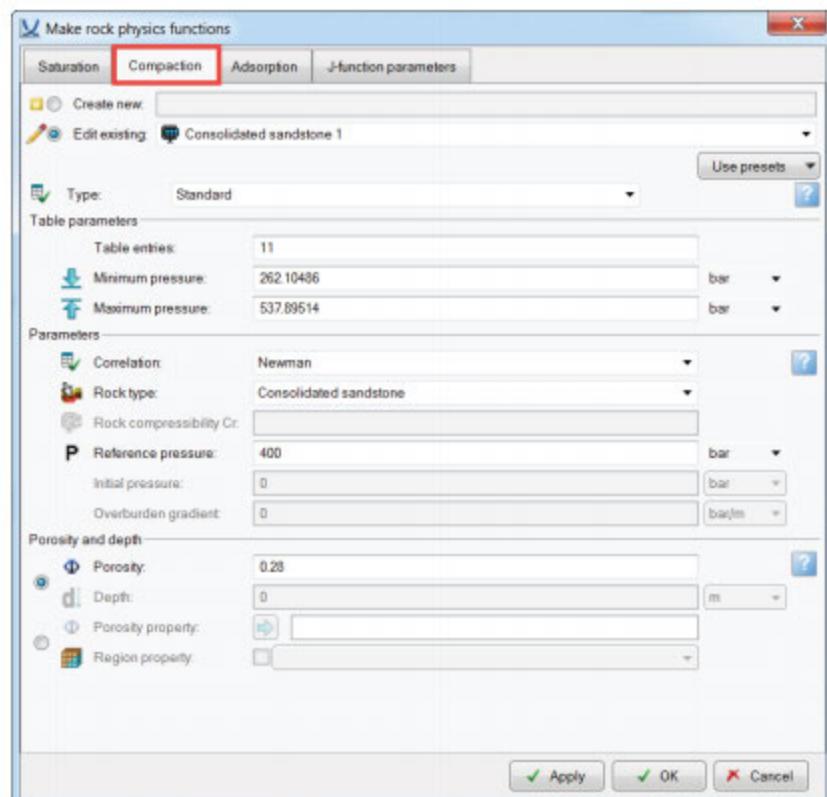


Figure 76. Compaction tab in the Make rock physics functions dialog box

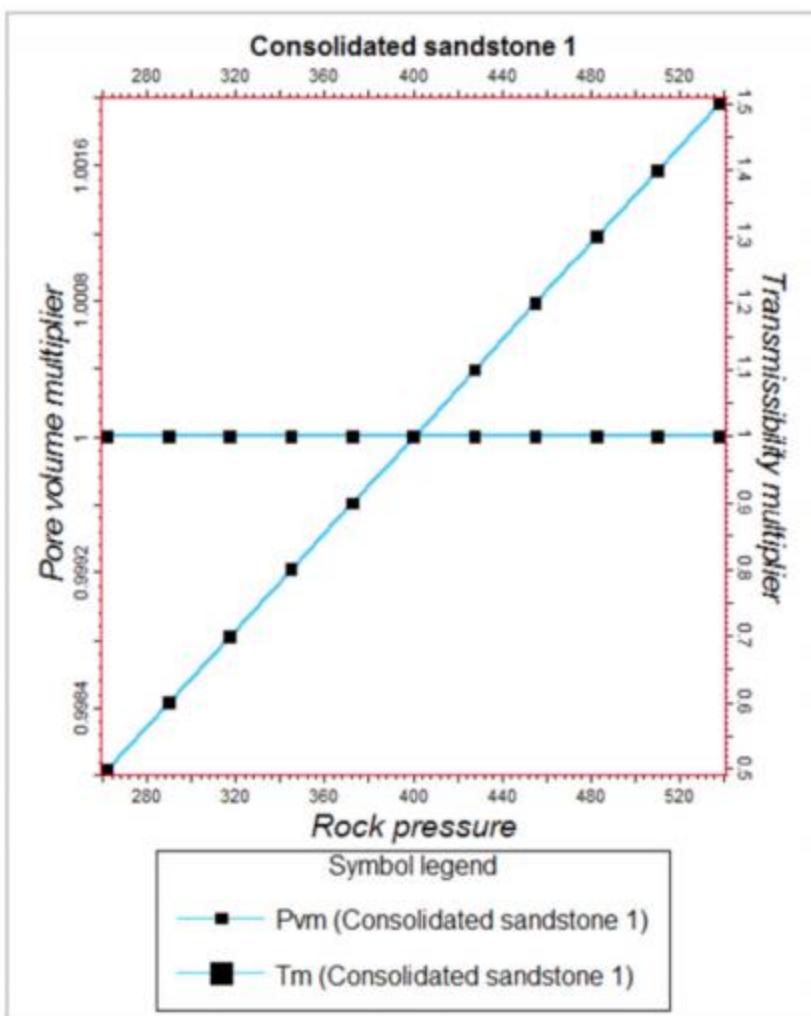


Figure 77. Pore volume and transmissibility multipliers vs. rock pressure plot

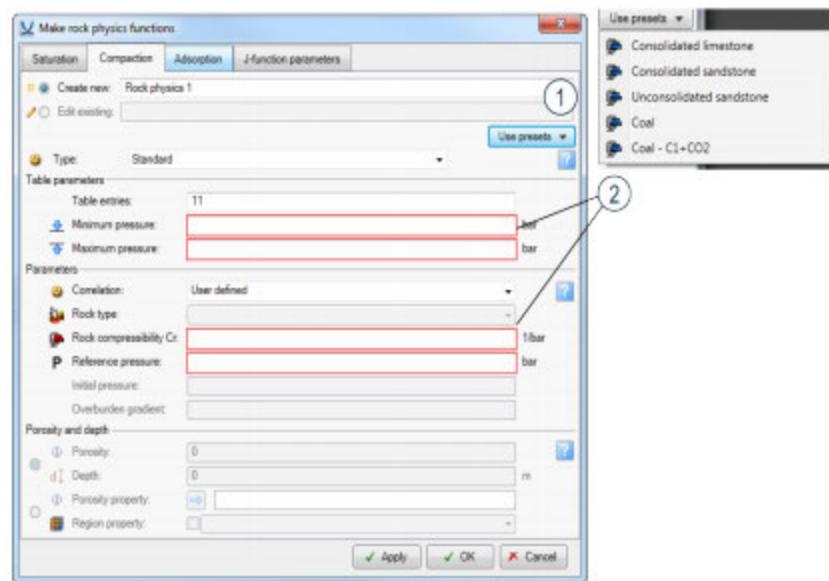


Figure 78. Compaction tab in the Make rock physics functions dialog box showing preset options and minimum required input data

- 1 Use presets: Five preset compaction functions
- 2 Red boxes: Minimum input data required for the selected correlation

By clicking the **Use presets** list, you can access five predefined compaction function options. Alternatively, you can fill in the fields manually.

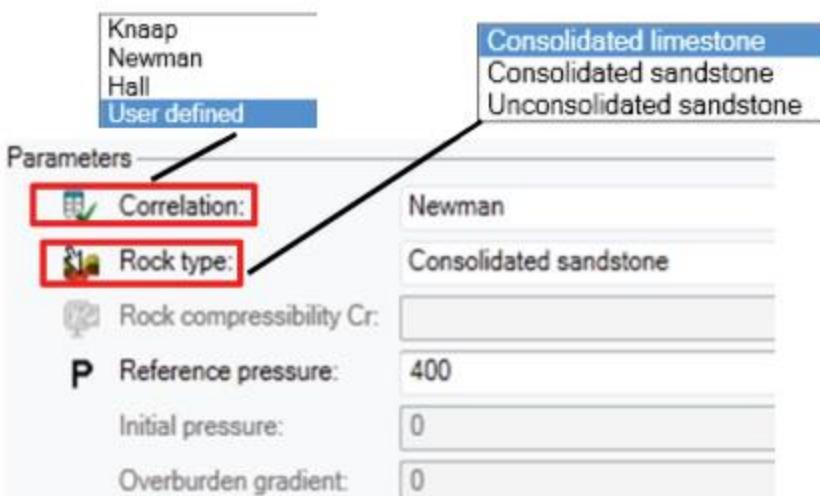


Figure 79. Available Correlations and Rock type options

Copyright © 2017 Schlumberger. All rights reserved.

If you select the correlation User defined, supply a compressibility and a reference pressure. Otherwise, the compressibility is computed using the correlation you select from the list.

In both cases, the result of the process is a curve that shows pore volume multipliers as a function of the reservoir pressure. The multiplier always is one for the reference pressure, which means that, at this pressure, the pore volume is equal to the initial pore volume.

When you add a rock compaction function to your simulation case, the simulator computes the (initial) pore volume to assign a pore volume multiplier. It takes the porosity property that you supply on the **Grid** tab of the **Define simulation case** dialog box as input.

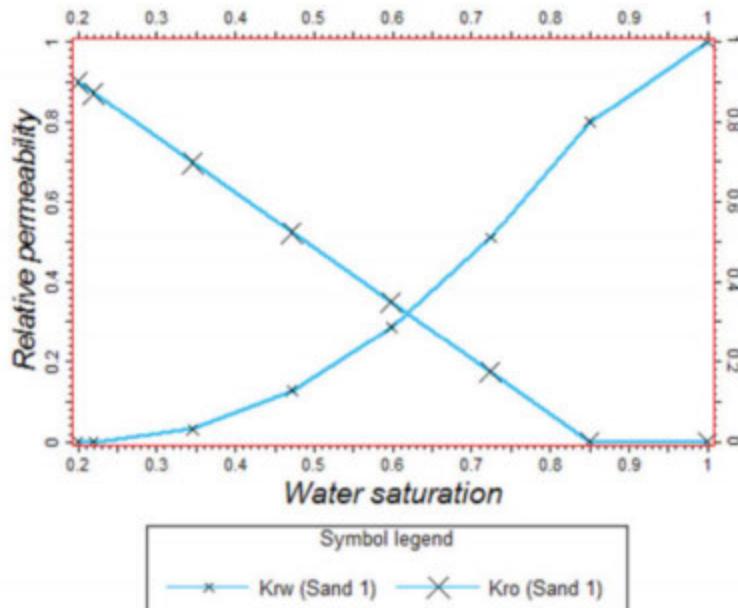
The simulator also uses the rock compaction curve. For each grid block, a pore volume is assigned by multiplying the original grid pore volume with the pore volume multiplier. So, if the pressure for a grid block is equal to the reference pressure, the volume is unaltered.

If the pressure is lower, the volume is reduced. However, if you want to model a reservoir that is compacted before the simulation begins, define a reference pressure that is higher than the initial cell pressures.



Procedure — Plot rock physics functions

To plot the rock physics function, activate the rock physics function in the **Input** pane. On the **Reservoir Engineering** tab, in the **Rock physics** group, click **Saturation plots** and then click plotting options from the list.



Copyright © 2017 Schlumberger. All rights reserved.

Spreadsheets for rock physics functions

The spreadsheet for rock physics functions has basic validation triggers. For saturation functions, the saturation values column must increase

downward. Depending on the relative permeability phase, it is constrained to be either increasing or decreasing down the column.

For rock compaction functions, the pressure must increase down the column. The pore volume and transmissibility multipliers must increase or decrease down the columns.

The simulator also runs a validation on the data ranges in the tables. If the endpoint in the tables exceeds 100% (1.0), the spreadsheet displays a warning.

To access the spreadsheet, right-click any of the plots in the active **Function** window and click **Spreadsheet**.

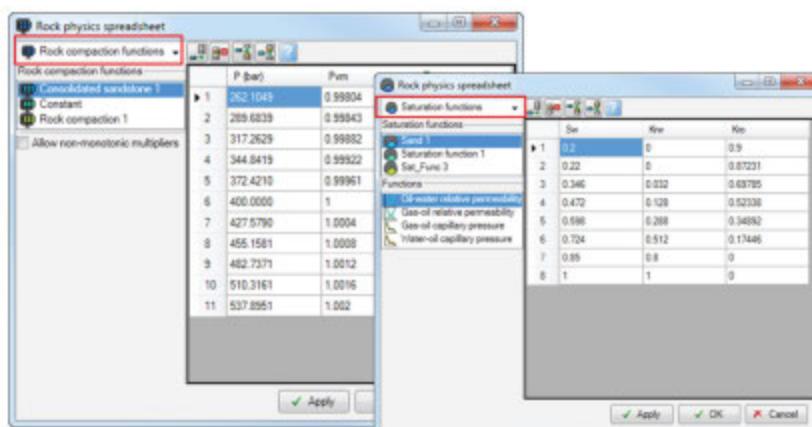


Figure 80. Rock physics spreadsheet

Copyright © 2017 Schlumberger. All rights reserved.

Endpoint scaling

The basic idea of endpoint scaling (EPS) is to generate saturation function curves with different endpoints based on a set of base input curves. With a small number of saturation tables as input in the **Property** section, different relative permeability and capillary pressure curves are derived by applying linear transformation to these input curves using different endpoints. The scaled curves are used in the simulation.

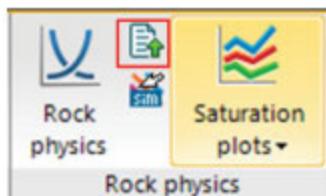
You can use endpoint scaling for different purposes. It can be a way to reduce the number of input saturation tables if they can be transformed to a smaller number of generic curves based on which of the original tables can be reproduced with different endpoints by scaling.



Procedure — Import rock physics functions

You can import functions into the Rock physics function folder.

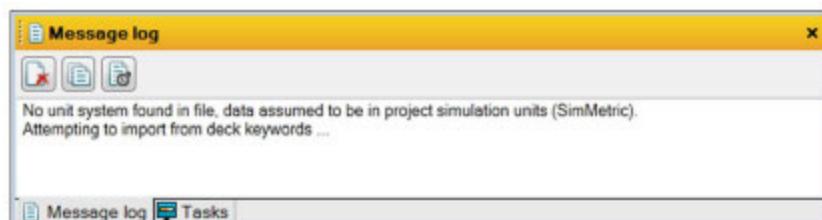
1. On the **Reservoir Engineering** tab, in the **Rock physics** group, click **Import**.



2. Browse to where the file to be imported is located.
3. Select the file and ensure that **Files of type selected** is **ECLIPSE SCAL and Rock (Rock physics)(Keywords) (*.*)**.
4. Click **Open** to import the file into the **Input** pane.



The status of the import is reported in the **Message log**.



Procedure — Manually insert rock physics functions

You also can insert rock physics functions manually into Petrel, but first, you must create a Rock physics functions folder.

1. To create this folder; on the **Home** tab, click **Folder** and select **Rock physics functions** from the list. Selection of Rock physics functions from the **Folder** list automatically inserts a Rock physics functions folder in the **Input** pane.

To organize the fluid functions models, you can create subfolders in the Rock physics functions folder. Creating subfolders allows you to use Petrel as a repository for the rock physics functions that you use regularly.

2. In the Input pane, right-click the Rock physics functions folder and click **Insert rock compaction function** or **Insert saturation function**. An empty function appears in the folder.
3. Right-click the function and select the spreadsheet from the list.
4. Select the fluid phases and enter function tables manually or copy data from an Excel spreadsheet.

Exercises — Create rock physics functions

There are three ways to create rock physics functions in Petrel:

- Make rock physics process
- Import existing data
- Use the spreadsheet

Data

Use the file `Simple_simulation_model_exercise.pet` in the `Dataset\Projects\Modules-3 Simple simulation model` folder.

Copyright © 2017 Schlumberger. All rights reserved.

Exercise 1 Make a saturation function

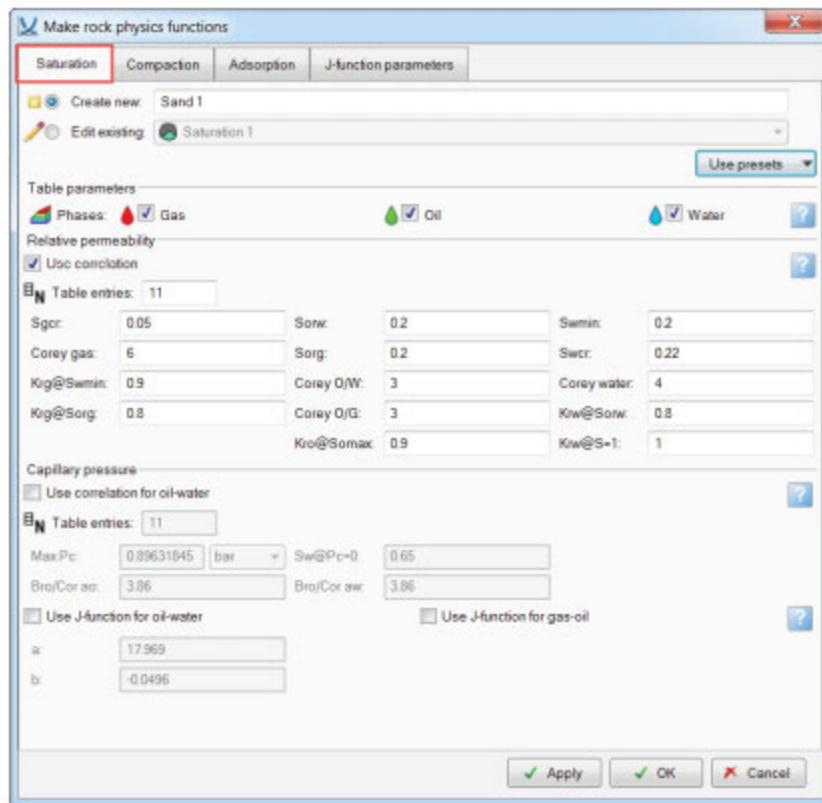
In this exercise, you create saturation functions based on Corey correlations using the Make rock physics functions process. The results are tables that you can export to the simulator through the Define simulation case process.



1. On the **Reservoir Engineering** tab, in the **Rock physics** group, click **Rock physics**.
2. In the **Make rock physics functions** dialog box, create a saturation function. On the **Saturation** tab, click **Create new**.
3. From the **Use presets** list, select Sand.

All dialog boxes are populated with default values for the selected preset option.

- Click **OK** to run the process and create the saturation function.



- In the **Input** pane, open the **Rock physics functions** folder. Click once on the rock physics function that you just created named **Sand 1** to make it active (**bold**).
- On the **Reservoir Engineering** tab, in the **Rock physics** group, click **Saturation plots** and then click **Saturation function**.



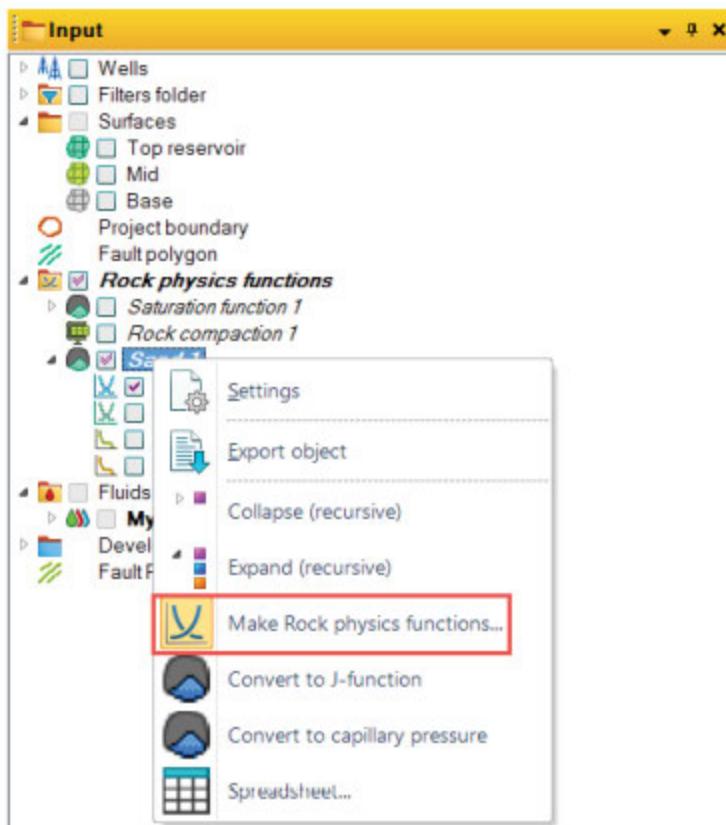
7. In the **Input** pane, complete these steps:
 - a. Expand **Sand 1** under the Rock physics functions folder.
 - b. Clear the **Sand 1** check box.
 - c. Select the **Oil-water relative permeability** check box to view the plot.

Exercise 2 Set the shape of the saturation curve and number of points



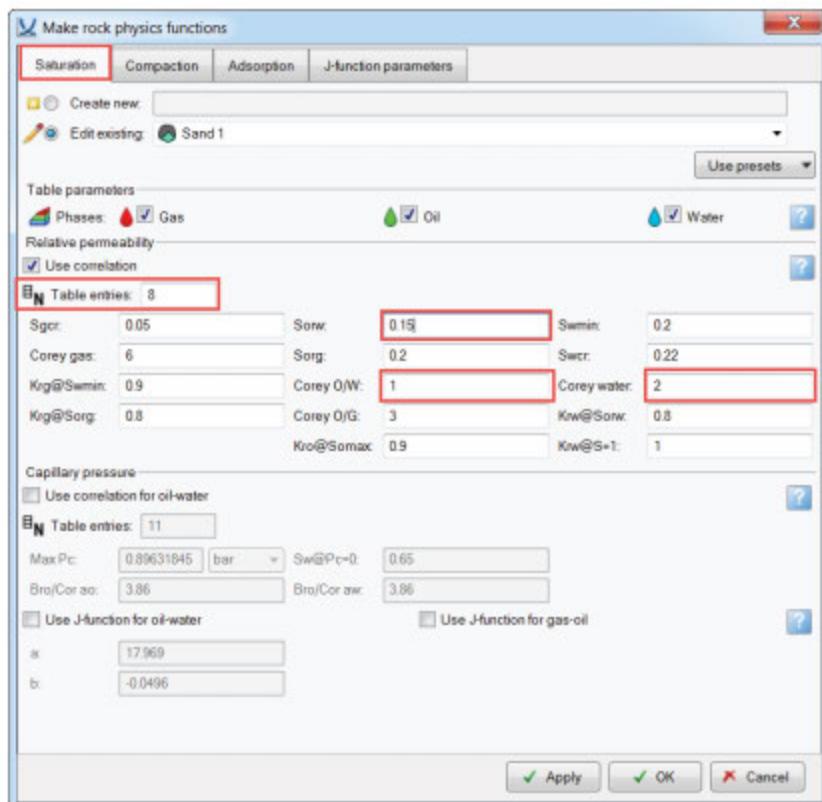
Set the shape of the relative permeability curve and the number of points in the table in the **Make rock physics functions** dialog box.

1. In the **Input** pane, right-click **Sand 1** and click **Make Rock physics functions**.



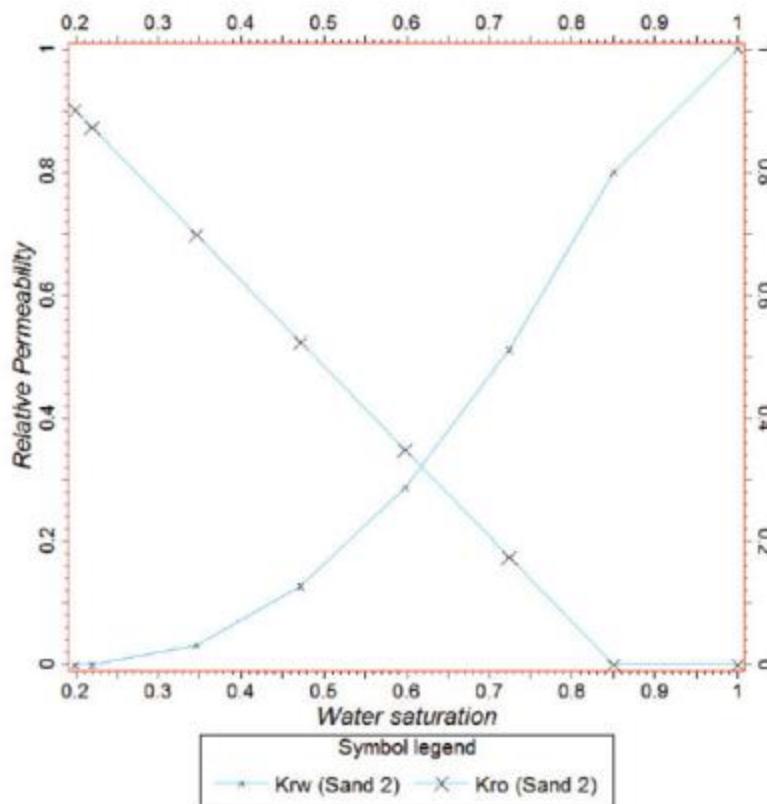
2. Change the number of **Table entries** to 8.

3. Change the **Corey O/W** to 1.
4. Change the **Corey water** to 2.
5. Change **Sorw** to 0.15 (refer to the figure).



6. Click **OK**.

7. Observe the plotted function updates.



Copyright © 2017 Schlumberger. All rights reserved.

8. Create one more saturation function using the preset Shaly sand and save your project.

Exercise 3 Make a rock compaction function

In this exercise, you use the Make rock physics functions process to create rock compaction functions based on correlations. The output from this process is rock compressibility tables that you export to the simulator through the Define simulation case process.

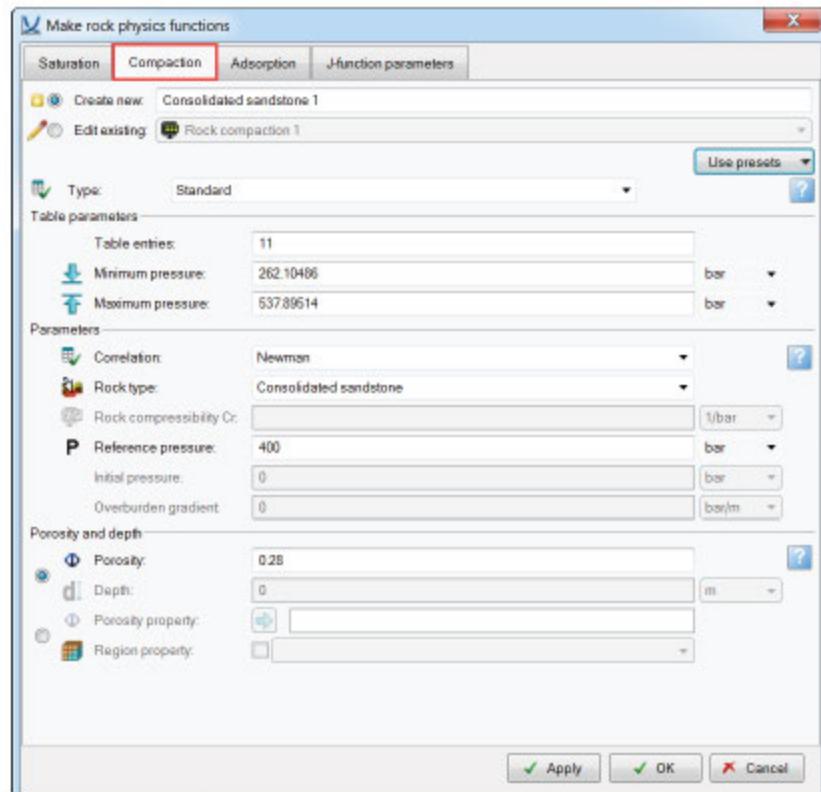


1. If you closed the **Make rock physics functions** dialog box, open it again.
2. Click the **Compaction** tab.
3. Click **Create new**.

- Click **Use presets** and select **Consolidated sandstone** from the list

Observe that all the boxes are populated with default values for the selected preset option. The default name of the function is **Consolidated sandstone 1**.

- To generate the function and close the dialog box, click **OK**.



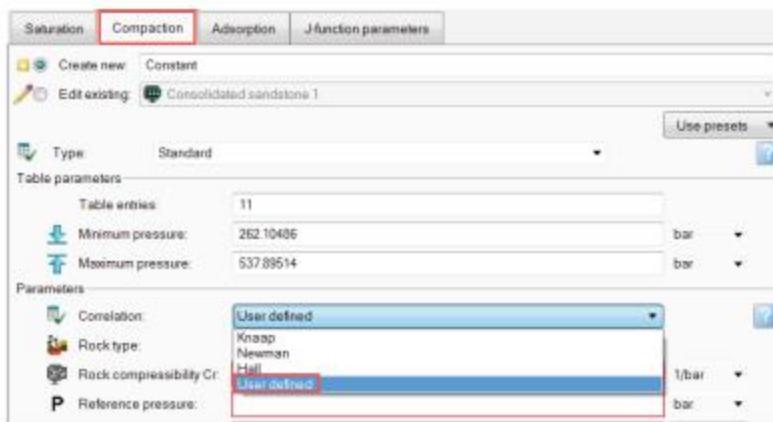
Copyright © 2017 Schlumberger. All rights reserved.

The new rock compaction function is stored in the Rock physics functions folder in the **Input** pane.

You also can assign a constant compressibility by completing the next steps.

- In the **Input** pane, right-click **Consolidated sandstone 1** and then click **Make Rock physics functions** to open the Make rock physics functions dialog box.
- Click the **Compaction** tab.
- Click **Create new** and name the new function **Constant**.

- From the **Correlation** predefined list in the **Parameters** section, select **User defined**.



- Enter these values and click **OK**.
 - Rock Compressibility Cr: 0 . 00001 1/bar
 - Reference pressure: 300 bar
- To plot the compaction functions, on the **Reservoir Engineering** tab, in the **Rock physics** group, click **Saturation plots** and then click **Rock compaction**.

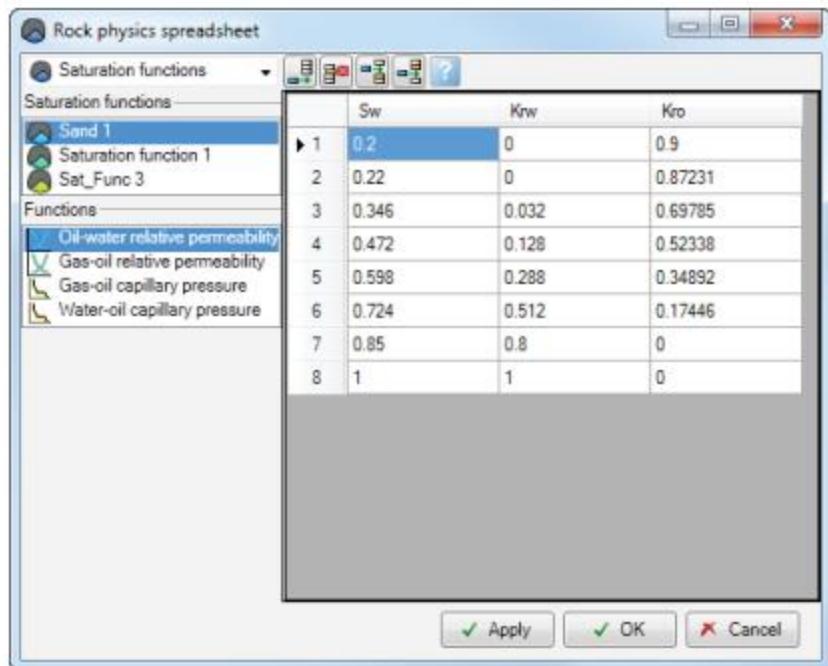
Exercise 4 Review rock physics functions data

In this exercise, you edit the rock physics functions that you created earlier. The graphical editing of the curve is done directly from the active plot window.

- View function values in table format. In the **Input** pane, right-click the **Rock physics** functions folder and click **Spreadsheet**.



- Select a function to view from the list on the left side of the window. Similarly, change between the relative permeability tables and the capillary pressure table by selecting them from the list.



- From the list at the top of the spreadsheet, choose between Saturation functions and Rock compaction functions.



- Click **Cancel** to close the spreadsheet.
- Use Saturation plots to plot one of the saturation functions that you created previously.
- There are two ways to edit the functions curve. You can edit the spreadsheet or edit the curves directly in the **Function** window using graphical tools. You cannot undo the changes.

NOTE: Make a copy of a function before you start editing.



7. Open the **Inspector** tool and click a plot point on one of the curves.

Information about this point appears in the **Inspector**.

8. On the **Window** toolbar, click **Edit function points [E]**
9. Click one of the points and drag it to another position to update the function.

WARNING: Remember, you cannot undo this operation!



10. On the **Window** toolbar, click **Shift function**
11. Click one of the points and move the functions up or down.

NOTE: Horizontal shift is not supported.



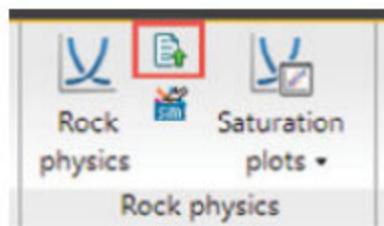
Exercise 5 Import a keyword rock physics function



In this exercise, you learn how to import rock physics functions from keyword files. The file can be an include file to the simulation deck or the keywords can be in the main .DATA file of the simulation deck.

1. On the **Reservoir Engineering** tab, in the **Rock physics** group, click **Import rock physics function**.

The **Import rock physics** dialog box appears.



2. Navigate to the Dataset folder and select the file ROCKPHYSICS.INC in the ImportData\Functions folder.
3. Click **Open**.
4. View the imported data by using both the spreadsheet and Saturation plots to plot the curves.

Lesson 3 Fluid models



To simulate reservoir fluid behavior during the production process and convert produced volume to reservoir conditions, you must have PVT properties. PVT data can be derived from laboratory experiments, correlations, or PVT modeling software.

In this lesson, you learn how to create a PVT model using Petrel built-in correlations and how to import an existing PVT model in ECLIPSE format.

Make fluid model process

The Make fluid model process allows you to create fluid models from the correlations library. In this course, you focus on black oil models only.

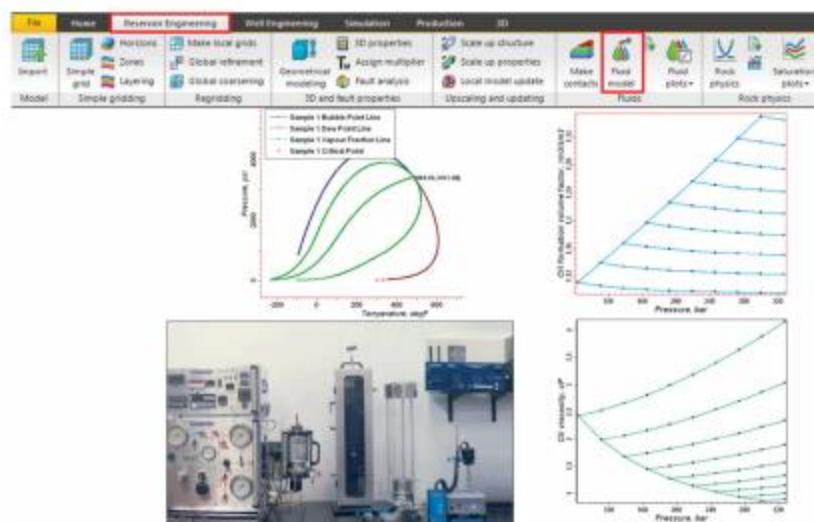


Figure 81. Fluid model button, PVT lab, and Fluid plots

Copyright © 2017 Schlumberger. All rights reserved.

Correlation library

The correlation library incorporates many published correlations. Some of these correlations use the separator conditions as input. All of the correlations have been tested against an extensive database of actual PVT (pressure-volume-temperature) experiments at the Schlumberger Reservoir Fluids Center in Edmonton, Canada.

Petrel selects which correlation to use based on the input data that you provided (the API gravity, the reservoir pressure).

The library contains about 70 black-oil correlations, including the most commonly used correlations.

Black oil and compositional models

There are always two, and often three, phases present in the reservoir during its producing life (oil, gas, and water). The proportions, compositions, and physical properties of the phases can change as production proceeds and pressures change. All phases are considered compressible, but to different degrees.

In a *black oil* model, the temperature is assumed to be constant. Typical temperatures at reservoir conditions are 350K-77 degC~171 degF. Because both the liquid hydrocarbon phase and the vapor phase are assumed to consist of mainly one component, it is customary to name them the oil phase and the gas phase, respectively.

The compositional behavior is modeled by allowing some of the gas component to be dissolved in oil and some of the oil component to be vaporized in gas.

A *compositional fluid* model represents the hydrocarbon fluid using a set of components (typically 6-12 for reservoir simulation). An equation of

state is used to determine the physical properties of the mixtures of these components as a function of pressure and temperature.

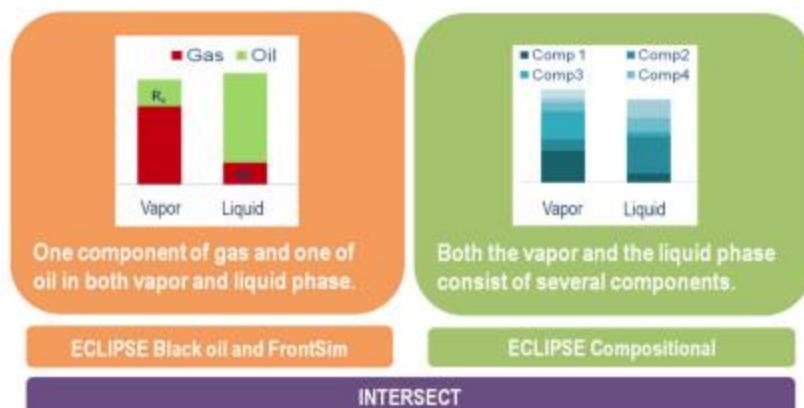


Figure 82. Fluid type and applicable simulator

Phase diagrams

Hydrocarbon behavior in a reservoir often is described in terms of a phase diagram, as shown in the figure. The phase diagram relates the fluid state to pressure and temperature in the reservoir. At pressures and temperatures to the left of the critical point (P_c , T_c), the line bounding the phase envelope is the *bubblepoint* line; on the right it is named the *dew point* line.

Copyright © 2017 Schlumberger. All rights reserved.

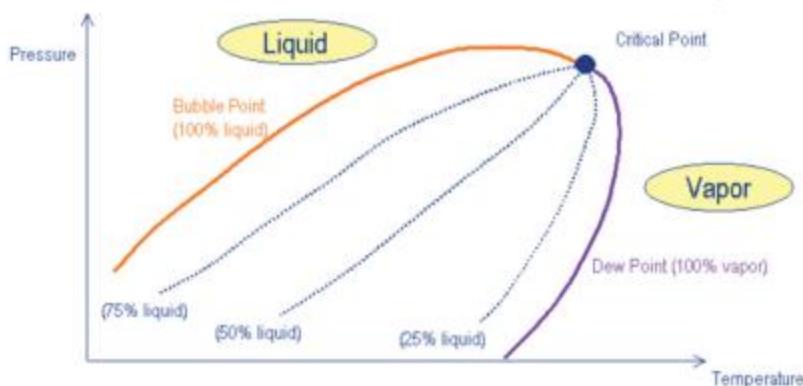


Figure 83. Phase diagram

The liquid region represents what often is termed *black oil*. It can cross the bubblepoint line and is distant from the critical point. The oil phase can, and usually does, contain a dissolved gas component. The strict definition of a black oil is, in fact, oil that contains no dissolved gas at stock tank conditions.

The vapor region represents a single-phase system that contains dry gas. As with black oil, the fluid is far from the critical point and does not cross the dew point line so the vaporized oil concentration (if any) remains fixed.

The area between the two lines represents the condition of pressure and temperature where both a liquid and a vapor phase are present simultaneously.

The *critical point* is where the bubblepoint line and the dewpoint line meet. At this pressure and temperature, the vapor and liquid properties are equal.

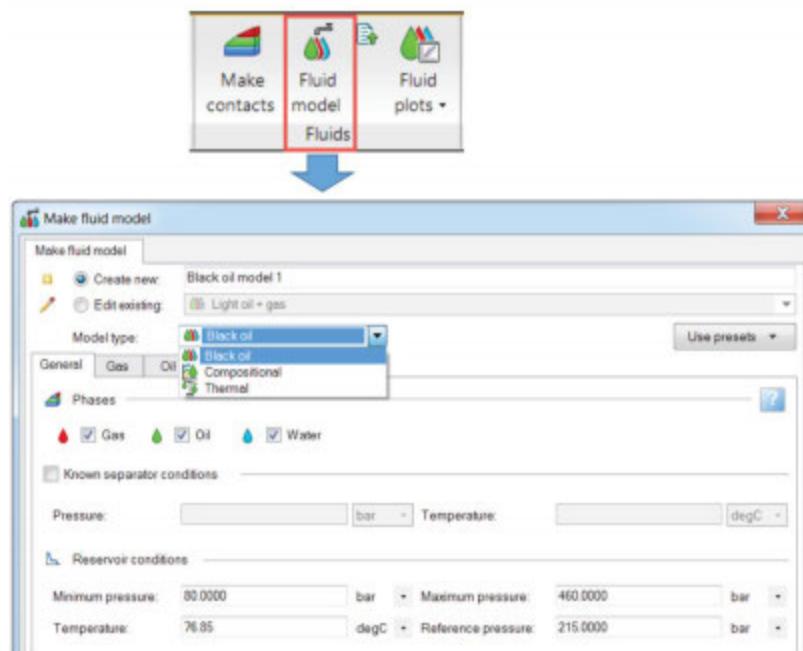


Procedure — Make a fluid model

In the **Make fluid model** dialog box, you have options to specify a black oil, compositional, or thermal fluid model. In this course, you focus on creating a black oil model.

1. On the **Reservoir Engineering** tab, in the **Fluids** group, click **Fluid model** to open the **Make fluid model** dialog box.

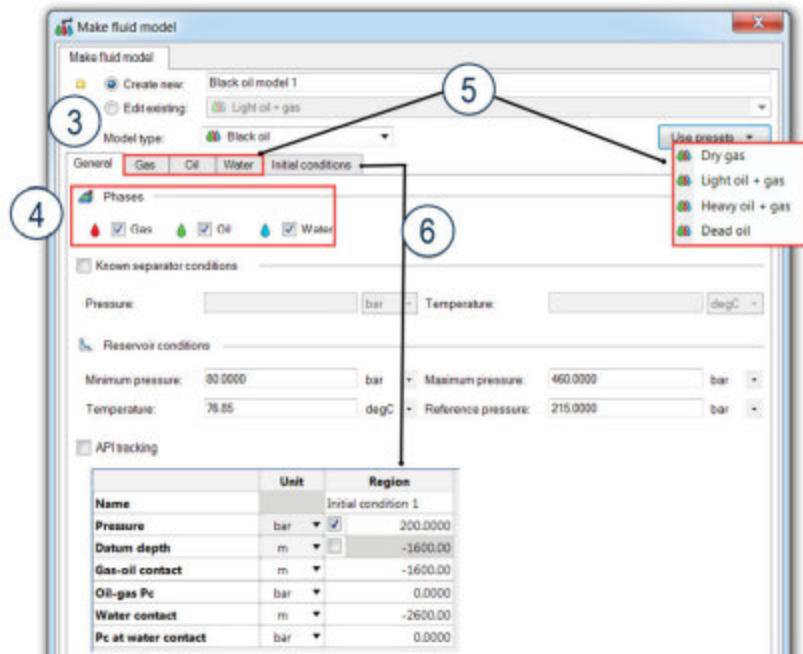
2. In the **Make fluid model** dialog box, click **Create new**.



3. Set Black oil as the **Model type**.
4. On the **General** tab, specify the required phases.
5. Enter the required properties on the **Gas**, **Oil**, and **Water** tabs.

Alternatively, use one of the presets to enter the required parameters.

6. Specify an initial condition.



Preset black oil models

There are four preset black oil models:

- Dead oil: Represents two phases – oil and water. Bubblepoint pressure is lower than minimum reservoir pressure, so no gas comes out of the oil.
- Heavy oil+gas: Represents three fluid phases – oil, gas, and water. The oil has an API gravity of 26.
- Light oil+gas: Represents three phases – oil, gas, and water. The oil is lighter with an API gravity of 45.
- Dry gas: Represents two fluid phases – gas and water.

In addition to the default fluid models, you can create black oil models by filling in the required data fields in the **Make fluid model** dialog box. Petrel selects a model from the correlations based on the input.

Make fluid model: General tab

The **General** tab allows you to specify which fluid phases are present along with the reservoir pressure and temperature.

Reservoir conditions: Specify the minimum and the maximum pressure in the reservoir. You also must enter the temperature in the reservoir and reference pressure.

Known separator conditions: Specify the pressure and the temperature at separator conditions. Some of the correlations need information on separator conditions.

Units: The volume unit of measurement in the industry is the stock tank unit, conventionally a barrel but, often, a cubic meter. Because stock tank oil is the result of a processing operation (gas separation), the volume that results from a unit volume of feed depends on the condition of processing.

As with gas, a volume is meaningless unless accompanied by a definition of the conditions of measurement. By convention, stock tank conditions used in the industry are 60 degrees Fahrenheit and 1 atmosphere pressure. The volumetric equivalence of one standard barrel of fluid (1 BBL) is stated as:

$$1 \text{ BBL} = 5.615 \text{ cubic feet} = 0.159 \text{ cubic meters}$$

Gas properties

On the **Gas** tab, you enter the density or the gravity of the gas phase (standard condition). The Vaporized gas/oil ratio option is available only if the model you are creating is dry gas. The value is used to set a constant concentration of vaporized oil. You also can select which correlations to use or you can allow Petrel to choose, based on your input data.

If you have information on the concentration of each component of the gas phase, you also can enter this value. This option is used only to select which correlations to use. It does not mean that you are defining a compositional model.

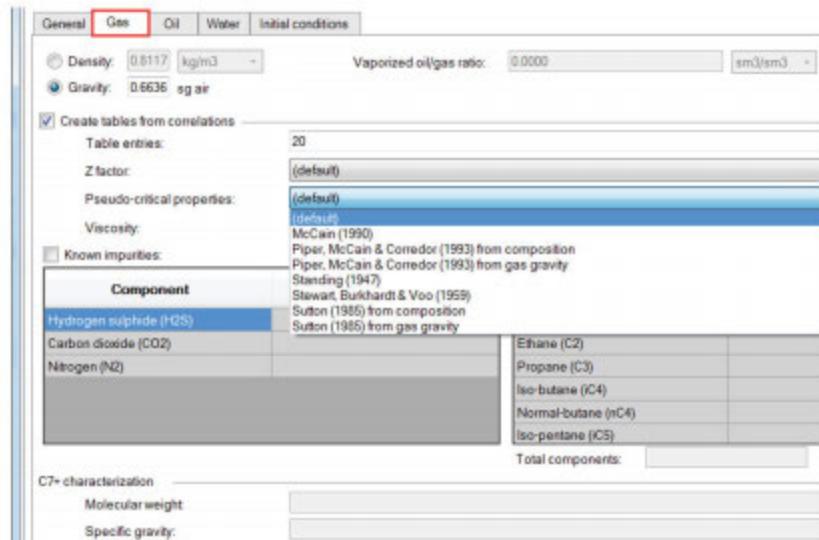


Figure 84. Gas tab in the Make fluid model dialog box

Copyright © 2017 Schlumberger. All rights reserved.

Oil properties

On the **Oil** tab , you specify the oil density or the oil gravity (API gravity). The usual range starts with water at 10 degrees and rises to volatile oils and straw colored condensate liquids at 60-70 degrees.

In addition, you must enter the Bubble point pressure or the Solution gas/oil ratio at the oil/gas contact. If the bubblepoint pressure that you use is lower than the minimum reservoir pressure, no gas comes out of the oil. As a result, the simulator models this oil as dead oil.

Oil gravity : In the petroleum industry, oil density has long been described using an expanded inverse scale authorized by the American Petroleum Institute called the API gravity.

Solution gas-oil ratio : In addition to the oil gravity or density, the volume of gas associated with unit volume of stock tank oil is an important property.

Expressed as gas-oil ratio, or GOR, the units at a reference condition of 60 degrees Fahrenheit and 1 atmosphere pressure are commonly standard cubic meter by standard cubic meter (SCM/SCM), or as standard cubic foot by stock tank barrel (SCF/STB). ECLIPSE uses MSCF (After J. S. Archer and C. G. Wall, Petroleum Engineering, Principle and Practice, 1986).

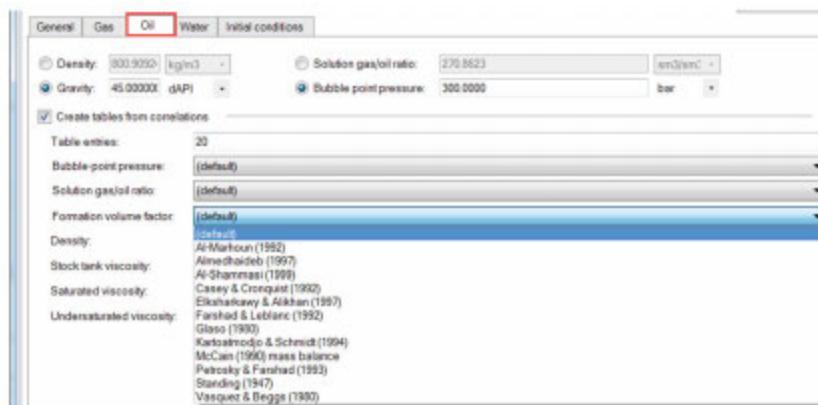


Figure 85. Oil tab in Make fluid model dialog box

You can select correlations manually or allow Petrel to select them, based on your input. The correlations used to make the fluid model are listed on the **Statistics** tab of the **Settings** dialog box for the fluid model as shown in the figure.

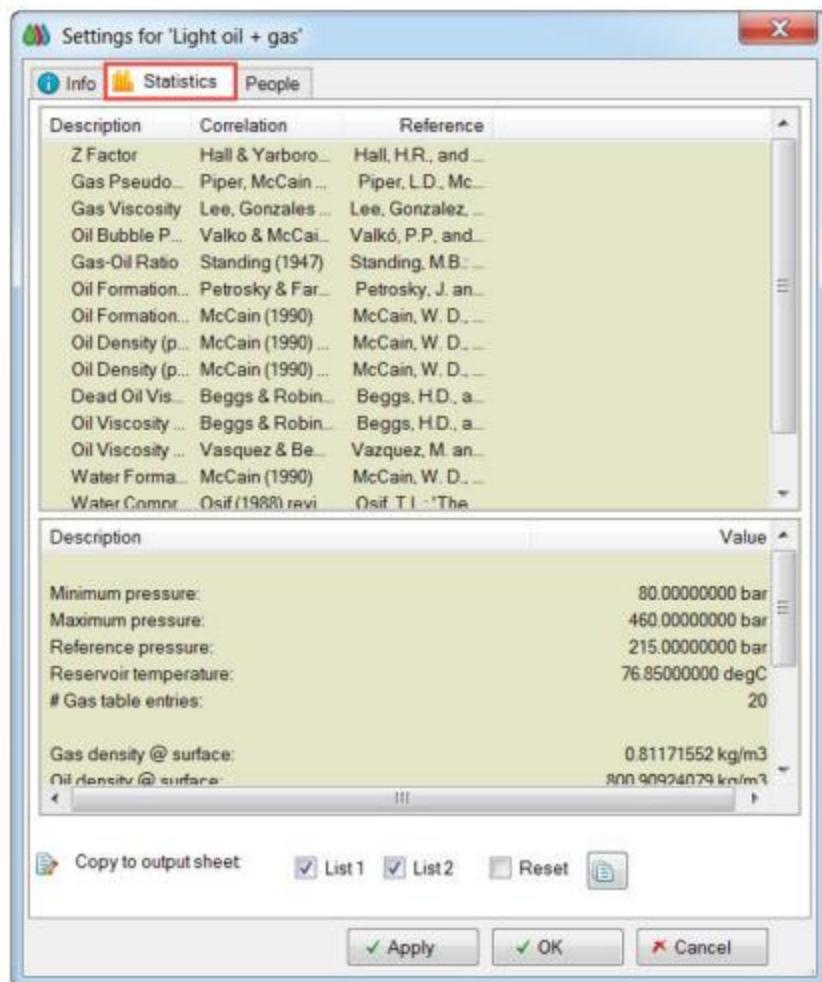


Figure 86. Correlations used as reported on the Statistics tab of the fluid model

Water properties

On the **Water** tab, enter the salinity of water present for the model. This value should be in pounds of solids per million of liquid (ppm). This option is available only if a water phase has been selected for the model.

Alternatively, the option to create tables from correlations can be cleared and you can enter directly the density, viscosity, formation volume factor, compressibility, and viscosibility.

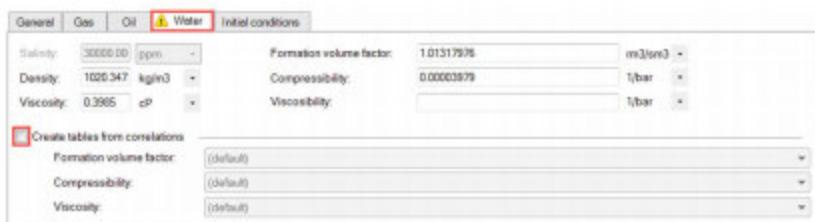


Figure 87. Water tab in the Make fluid model dialog box

Create tables from correlations option

If you want to create new water properties from correlations, enable this option on the **Water** tab. You can leave each setting at its default, in which case Petrel selects the correlation automatically, or you can choose a correlation from each list.

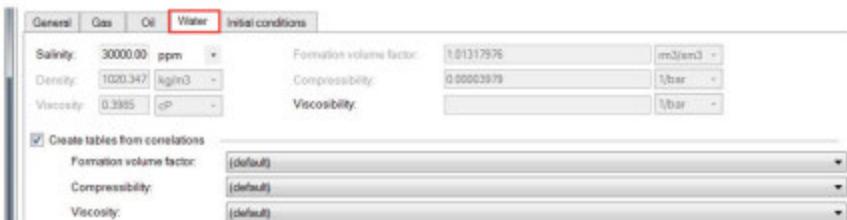


Figure 88. Create tables from correlations option on the Water tab in the Make fluid model dialog box

Fluid spreadsheet

Right-click the oil or gas phase of the fluid model to access data in a fluid spreadsheet.

You can copy data from Microsoft Excel and paste it into the spreadsheets and copy data from the spreadsheets and paste it into Excel.

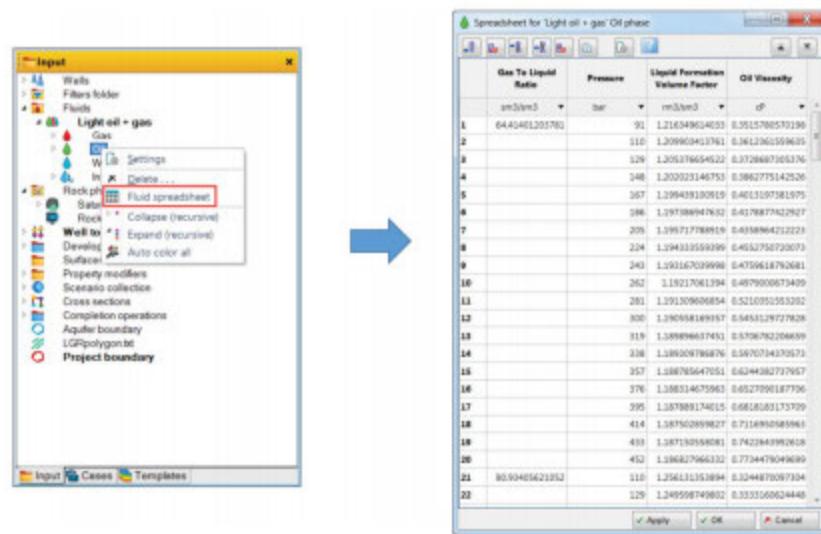


Figure 89. Accessing the fluid spreadsheet from the Input pane

Copyright © 2017 Schlumberger. All rights reserved.

Fluid model plots

You can plot fluid data in a **Function** window. The figure shows an example.

On the **Reservoir Engineering** tab, in the **Fluids** group, click **Fluid plots** and choose the type of fluid data from the list.

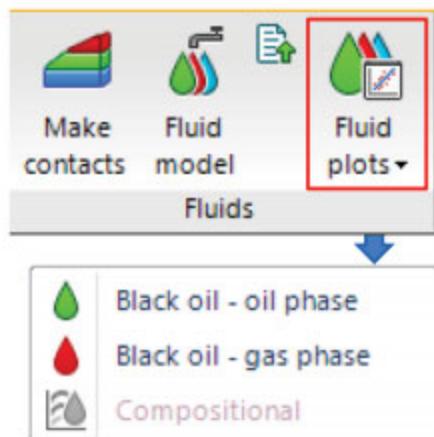


Figure 90. Create a fluid plot

Copyright © 2017 Schlumberger. All rights reserved.

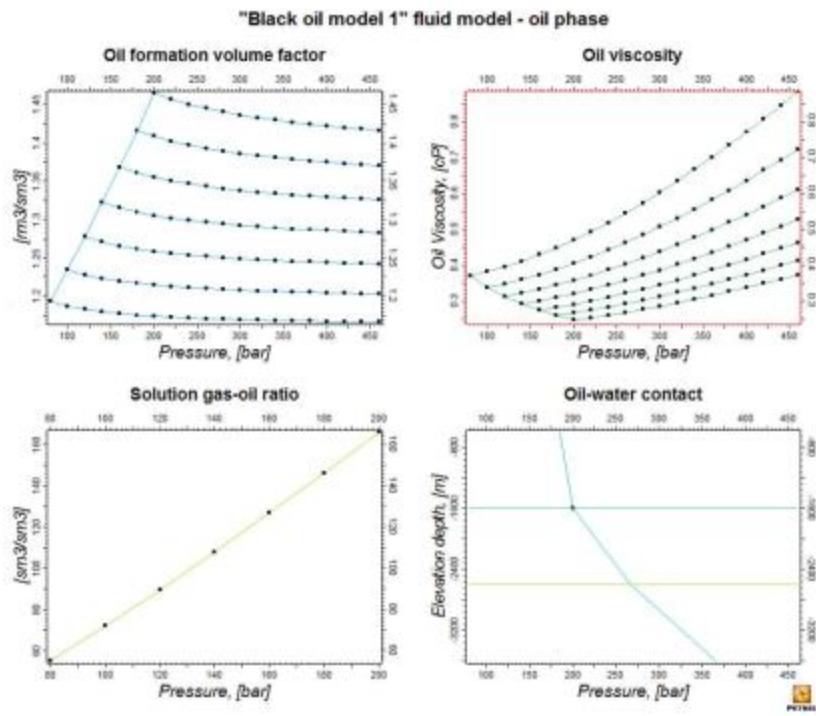


Figure 91. Examples of fluid plots for black oil models

Copyright © 2017 Schlumberger. All rights reserved.

Exercises — Make a fluid model

In this exercise, you practice how to create a fluid model using any of the three methods available in Petrel:

- Make fluid model process
- Import existing data
- Use the spreadsheet

Data

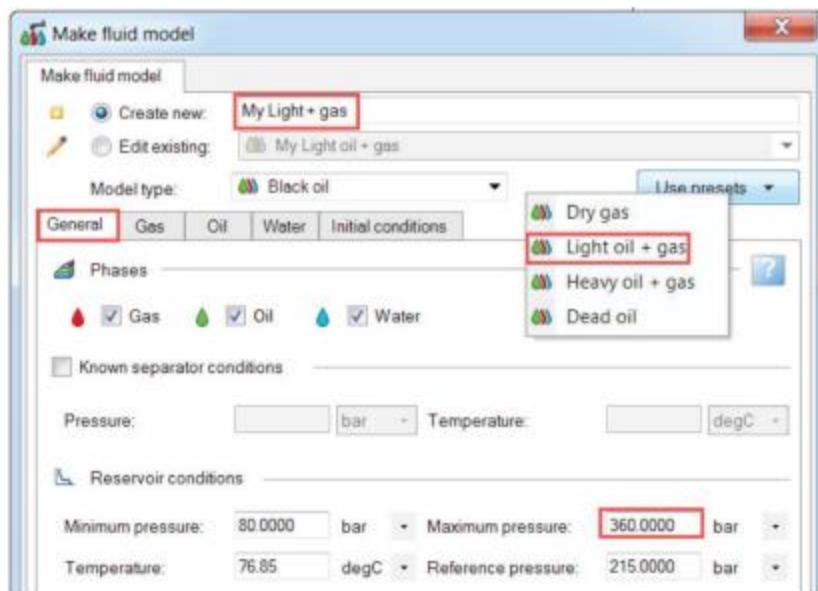
Use the file `Simple_simulation_model_exercise.pet` in the `Dataset\Projects\Modules-3 Simple simulation model` folder.



Exercise 1 Make a black oil model from the Petrel correlations library

In this exercise, you create a black oil fluid model based on correlations by using the Make fluid model process.

1. On the **Reservoir Engineering** tab, in the **Fluids** group, click **Fluid model** to open the **Make fluid model** dialog box.
2. Click **Create new**.
3. Click **Use presets** and select **Light oil + gas**. The dialog box is filled in with default values.
4. Enter a new name for the new model, for example **My Light oil + gas**.
5. On the **General** tab, change the Maximum pressure to 360 bar.



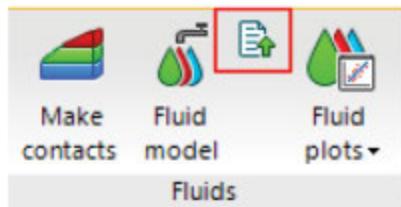
6. Examine the default values on the **Gas** tab and **Water** tab and accept the default settings for this exercise.
7. On the **Oil** tab, ensure that the bubblepoint pressure is 300 bar.
8. Leave the dialog box open. You will complete the exercise in the next module of the training manual; Model initialization and volume calculation.



Exercise 2 Import a keyword fluid model

In this exercise, you import a fluid model from an ECLIPSE keyword file. This file can be an include file to the simulation deck, or the main .DATA file of the simulation deck.

1. On the **Reservoir Engineering** tab, in the **Fluids** group, click **Import fluid model**.



2. Browse to the FLUID.INC file located in the Dataset \ImportData\Functions directory. Click **Open** to import the file.

A new fluid model named PVTNUM_1 is added to the Fluids folder in the **Input** pane.



NOTE: On import, you might get a warning or error message. You can find the details of the message in the **Message log**.

3. Review the imported data using fluid spreadsheets. Expand the imported PVTNUM_1 fluid model. Right-click **Oil 1** and click **Fluid Spreadsheet**.

Observe that the fluid model that you imported does not have an initial condition. Keep in mind that you must add an initial condition before you can use the fluid model in a simulation model.

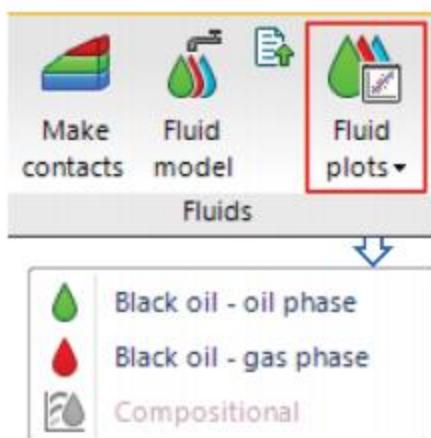


Exercise 3 Plot the imported fluid model

In this exercise, you plot the fluid model in a **Function** window.

1. Select the imported fluid model PVTNUM_1 to activate it.

2. On the **Reservoir Engineering** tab, in the **Fluid** group, click **Fluid plots** and then click **Black oil - oil phase**.



The plot oil formation volume factor, oil viscosity, solution gas-oil ratio versus pressure and fluid contact are plotted in a **Plot window**.

3. Again click **Fluid plots** and then click **Black oil - gas phase**.

The gas formation volume factor, gas viscosity, saturation pressure ratio (no data defined yet), and fluid contact are plotted in the **Plot window**.

4. Open the **Inspector**
5. Click anywhere on any curve.

The property name and values are displayed in the **Inspector**.

Copyright © 2017 Schlumberger. All rights reserved.

Review and summary

Review what you learned in this module.

The review and summary help you to reinforce the learning objectives for this module.



Review questions

The review questions reinforce the learning objectives for this module.

- Is it possible to add a vertical fault in a simulation grid without going through the Fault modeling and Pillar gridding processes? What are the downsides?
- What tool do you use to generate relative permeability, rock compaction, and capillary pressure based on correlation?
- When you have a non-faulted grid, which tool do you use to construct a 3D simulation grid quickly?

Summary

In this module, you learned about:

- building a simple simulation grid
- creating a vertical simulation fault without going through the Fault modeling and Pillar gridding processes
- creating fault transmissibility properties
- creating a correlation-based black oil fluid model and rock compaction and saturation functions
- using spreadsheets to enter function data directly
- creating and importing existing laboratory fluid models and rock physics functions from ECLIPSE* keyword files
- editing and visualizing the created model and functions in the Petrel environment.

NOTES

Copyright © 2017 Schäffer-Poeschl. All rights reserved.

NOTES

Copyright © 2017 Schlumberger. All rights reserved.

Module 4 Model initialization and volume calculation

This module shows you how to use the Define simulation case process to set up and initialize a simulation case. This process allows you to pull together the 3D grid, fluid model, rock physics functions, and development strategy to create a simulation case.

In this module, you learn about how to make static volumetric calculations. You also learn about some fundamental differences between geological and simulation grids. The disparity between volume calculations in static and simulation models also is discussed.

Learning objectives

After completing this module, you will know how to:

- define initial conditions
- initialize a simulation model
- calculate static volume
- explain why there is a disparity between static and dynamic volumes
- explain the fundamental difference between static and simulation grids





Lesson 1 Initialization of the simulation model: Define simulation case

In this lesson, you learn how to set up and initialize a simulation model. Relative permeability and a fluid model are required inputs for the simulator to calculate initial saturation and pressure.

Three ways of initializing the model

Initial gas, oil, and water saturations must be defined in the reservoir model. You do not need to define these values (the simulator calculates it for equilibration). Pressure data usually is given with reference to a datum depth. By default, the datum depth in Petrel is mean sea level.

There are three initialization options:

- Equilibration: The simulator computes initial phase pressures and saturations using the fluid model and capillary pressure curves if transition zones exist. The equilibration facility is a means of calculating the initial conditions based on capillary-hydrostatic equilibrium.

If necessary, the reservoir can be divided into separate equilibration regions in which hydrostatic equilibrium exists independently of the other regions. The number of equilibration regions is specified in the **Define simulation case** dialog box.

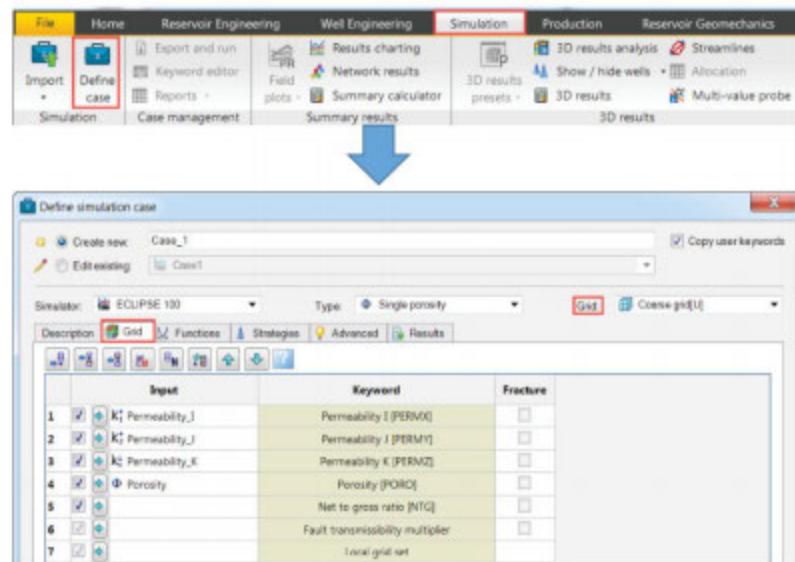
In each equilibration region, all grid blocks must use the same pressure table for their PVT properties. However, they can use different rock physics function tables, as specified in the **Make rock physics functions** dialog box.

- Enumeration: You set the initial value of pressure, saturation, and bubblepoint pressure explicitly in each grid block. You must insert the 3D grid property into the **Grid** tab of the **Define simulation case** dialog box.
- Restart: The initial conditions are read from a restart file of a previous run.

Procedure — Initialize the model: Grid properties



1. Open the **Define simulation case** dialog box.
2. Create a new case or edit an existing case.
3. Select the simulator type.
4. Choose a grid if you have multiple grids in your project.



Copyright © 2017 Schlumberger. All rights reserved.

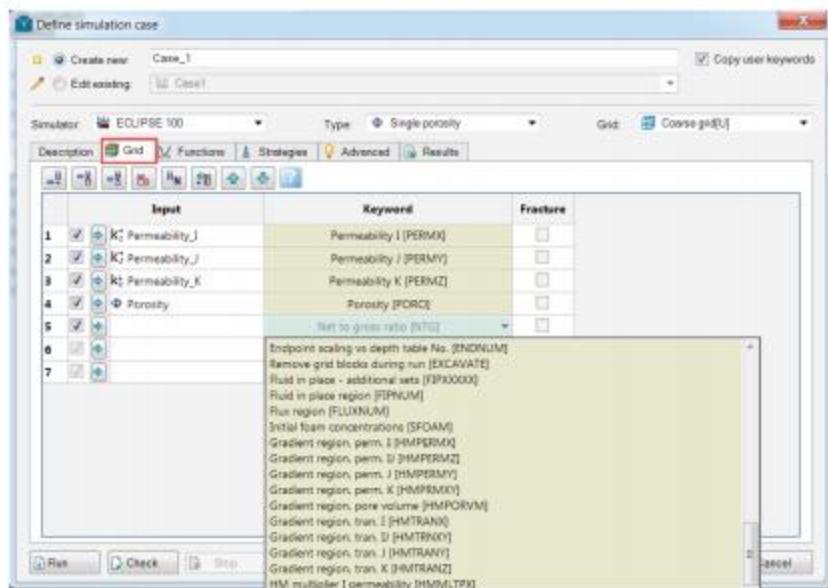
5. Add a row and use the lists to add more properties, if needed.

Procedure — Add more grid properties



1. Add a row by clicking .
2. Select a template.

3. Insert the 3D property. (For example, insert water saturation to be used for endpoint scaling and initial water saturation.)



In addition to permeability and porosity (which are required input), you can drop in other properties defined in the 3D grid. For example, you can select to use a local grid set, fault transmissibility, or endpoint scaling for saturation function.

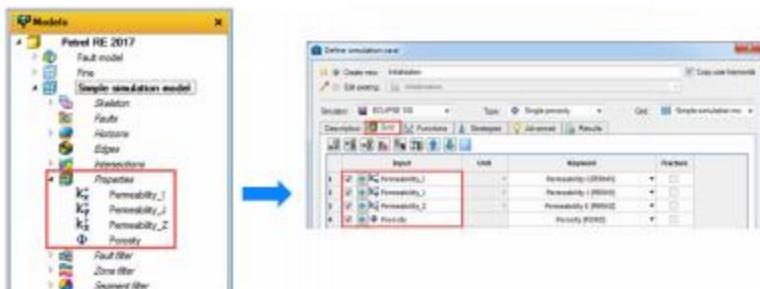
Copyright © 2017 Schlumberger. All rights reserved.



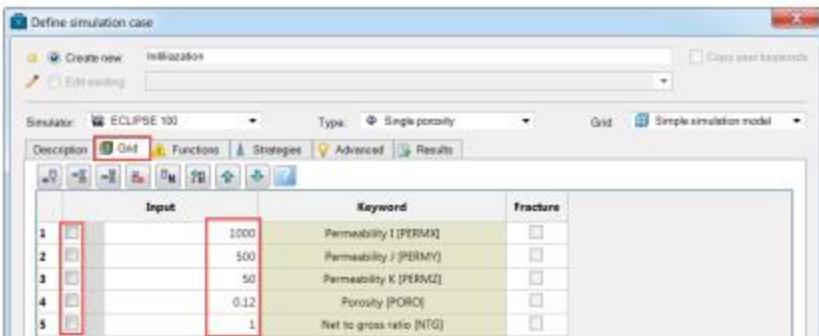
Procedure — Assign a 3D grid or constant property to a simulation case on the Define simulation case dialog box

1. To define 3D grid property to simulation case, take these actions:
 - a. On the **Simulation** tab, click the **Define case** button to open the **Define simulation case** dialog box.

- b. On the **Property** folder in the **Models** pane, insert the existing 3D grid property into the data drop-box.



2. To define the constant grid property to simulation case, take these actions:
 - a. On the **Simulation** tab, click the **Define case** button to open the **Define simulation case** dialog box.
 - b. Clear the check box of any grid property to enter constant grid property.



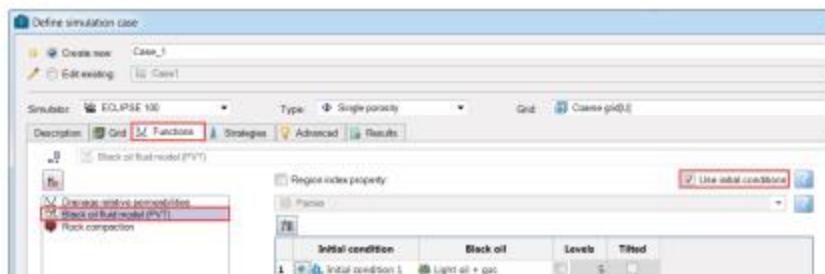
Copyright © 2017 Schlumberger. All rights reserved.

Procedure — Insert fluid model: Black oil (PVT)



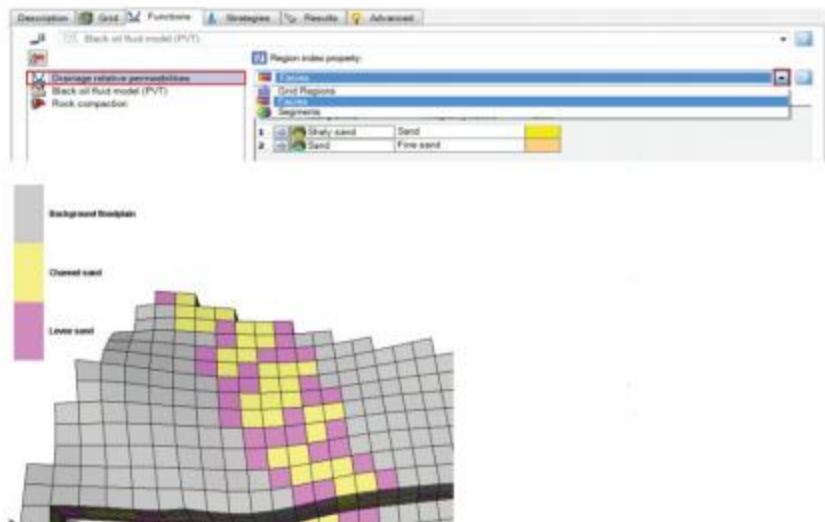
1. Click the Black oil fluid model (PVT).
2. Insert the initial condition of the fluid model.

3. If you use enumeration, clear the **Use initial conditions** check box.



Procedure — Insert a rock physics function

1. Insert :
 - Relative permeability curves
 - Rock compaction function
2. To assign different functions to different rock types, select the **Region index property** check box.



Copyright © 2017 Schlumberger. All rights reserved.

Initial conditions

The option Use initial conditions, used for setting initial pressures and saturations, is based on this data:

- Saturation function; set up in the Make rock physics functions process.
- Fluid contacts, datum depth, and pressure; set up on the **Initial conditions** tab in the **Make fluid model** dialog box.

When you use this option, the hydrostatic equilibrium is assumed.

Initial conditions can be defined by using a contact set or by entering the required data in table format.

If you are not using a contact set, you can enter the details of each initial condition in a table. The table consists of a column for each initial condition region. You can add or remove columns using the standard Petrel table manipulation tools.

NOTE: To keep the fluids in a tilted equilibrium, you must define the model accordingly (with an active aquifer and capillary pressure). If you enter only a tilted surface for the contact and run the simulator, the fluid gradually declines to a flat contact.



Copyright © 2017 Schlumberger. All rights reserved.

By default, the gas-oil contact is set as Datum. The pressure at datum defaults to a pressure gradient of 0.0981 bar/m over the depth given by (surface elevation - datum depth).

To enter a specific pressure or datum depth, select the check box in the column.

Name	Unit	Region
Pressure	bar	Initial condition 1 200.0000
Datum depth	m	-1600.00
Gas-oil contact	m	-1600.00
Oil-gas P _c	bar	0.0000
Water contact	m	-2600.00
P _c at water contact	bar	0.0000

Figure 92. Entering the required initial conditions data in table format

Existing contact set

If you have an existing contact set, you can select the **Use contact set** check box and insert the contact set from the **Models** pane.



Figure 93. Option to use an existing contact set

Alternatively, you can use the **Initial conditions** process to create an initial condition set for a reservoir. The Initial conditions process allows you to carry out your initialization in the context of the grid. It provides validation against the grid, fluid model, and simulator to create a valid initial condition set for simulation.

When a 3D model is built and the fluid models are created, you can use this process to define the equilibration regions and create initial condition sets. These sets then are used in the **Define simulation case** dialog box. You can create a single equilibration region for the entire reservoir that consists of a single fluid model.

You also can create an initial condition set for multiple equilibration regions by using the region index property (or contact set) to describe the equilibration regions in the reservoir model.

To open the **Initial conditions** dialog box, on the **Reservoir Engineering** tab, in the **Initialization** group, click **Initial conditions**.

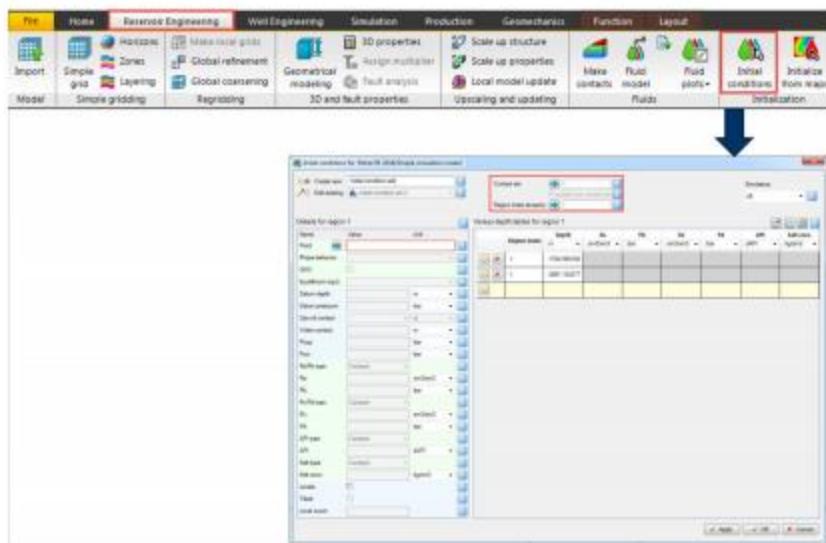


Figure 94. Access the Initial conditions dialog box

Petrel also allows you to create an initial condition set for a reservoir with areal variation in the depth of the oil-water and gas-oil contacts by using the Initialize from maps process. This process uses the spatially varying contacts and compositions (Rs/Pb, Rv/Pd, or both) as inputs. It then discretizes them into thousands of regions. It also associates the correct fluid model to each equilibration region by using a discrete PVT region index property.

The Initialize from maps process groups the cells in a reservoir into regions that require similar initial conditions based on the discretization. The discretization is the tolerance limit on the range of the spatially varying inputs provided. Discretization defines the level of tolerance, which is the interval of values that will be grouped together into a single equilibration region. The process then generates the appropriate equilibrium models for those cells.

The net result is that the reservoir is approximated by multiple equilibration regions mapped to appropriate fluid models. The process supports a single fluid model (when there is only one fluid model associated with the entire reservoir model) and a multiple fluid model (when the reservoir model has more than one fluid model associated with it, usually defined by a PVT region index property). You also can enable the Initialize from maps process in the **Workflow editor** and the Uncertainty and optimization process.

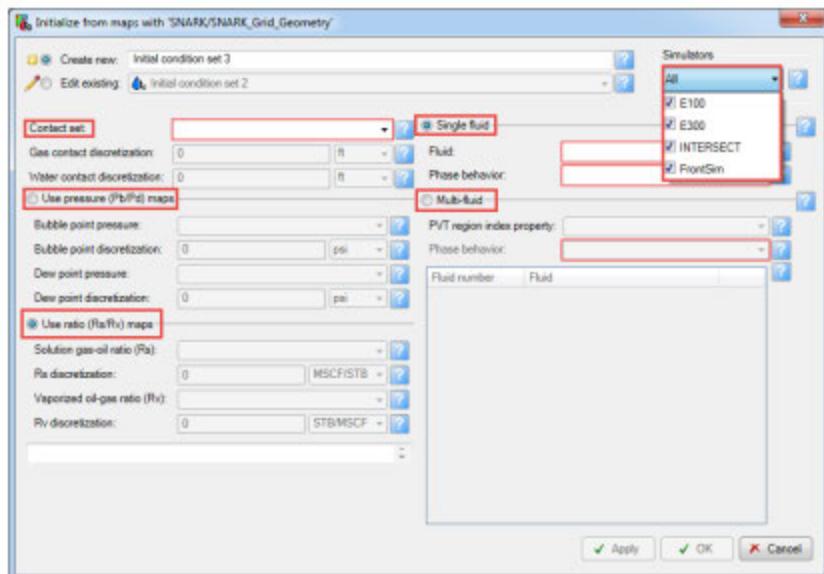


Figure 95. Initialize from maps dialog box

The **Initialize from maps** dialog box has these options:

- Contact set
 - Gas contact discretization
 - Water discretization
- Use pressure (Pb/Pd) maps
 - Bubble point pressure
 - Bubble point discretization
 - Dew point pressure
 - Dew point discretization
- Use ratio (Ra/Rv) maps
 - Solution gas-oil ratio (Ra)
 - Ra discretization: 0 MSCF/STB
 - VapORIZED oil-gas ratio (Rv)
 - Rv discretization: 0 STB/MSCF

- Use ratio (Rs/Rv) maps
 - Solution gas-oil ratio (Rs)
 - Rs discretization
 - Vaporized gas-oil ratio (Rv)
 - Rv discretization
- Single fluid
 - Fluid
 - Phase behavior
- Multi-fluid
 - PVT region index property
 - Phase behavior
 - Fluid number
 - Fluid

Contact set	<p>Select a pre-defined contact surface from the Contact set list. This process only supports contact sets with one areally-varying map per contact. If you have a single contact value, it is recommended to use the Initial conditions process instead of the Initialize from maps dialog box. To access the Initial conditions process, on the Reservoir Engineering tab in the Initialization group, click Initial conditions. To use the Initialize from maps dialog box, you must have a contact set with appropriate contacts for the active grid. The contact set must contain this input data:</p> <ul style="list-style-type: none"> • Areally-varying water contact • Areally-varying gas-oil contact • Contact datum depth • Constant datum pressure • Pcow and Pcog (optional; default to 0) <p>You create this data using the Make contacts process.</p>
Gas contact discretization	Use this option to enter the height that you want to use to discretize the gas-oil contact.
Water discretization	Use this option to enter the height that you want to use to discretize the water contact.

Use pressure (Pb/Pd) maps	<p>Select this option to use a pre-defined map that provides spatially-varying bubblepoint pressure across the model.</p> <p>This section has these options:</p> <ul style="list-style-type: none">• Bubble point pressure: Use this option to select the structured or regular surface map that specifies the oil bubblepoint pressure. This map is used to determine the fraction of dissolved gas in the oil phase in conjunction with the associated fluid model.• Bubble point discretization: Use this option to enter the pressure difference that you want to use to discretize the bubblepoint pressure map.• Dew point pressure: Use this option to select the structured or regular surface that provides spatially-varying dewpoint pressure across the model based on phase behavior.• Dew point discretization: Use this option to enter the pressure difference that you want to use to discretize the dewpoint pressure map.
---------------------------	---

Use ratio (Rs/Rv) maps:	<p>Select this option to use a map that provides a spatially-varying solution gas-oil ratio across the model.</p> <p>This section has these options:</p> <ul style="list-style-type: none">• Solution gas-oil ratio (Rs): Use this option to select the structured or regular surface map that specifies the solution gas-oil ratio. This ratio is used to determine the fraction of dissolved gas in the oil phase in conjunction with the associated fluid model.• Rs discretization: Enter the value that you want to use to discretize the solution gas-oil ratio map.• Vaporized gas-oil ratio (Rv): Use this option to select the structured or regular surface that provides a spatially-varying gas-oil ratio across the model based on phase behavior.• Rv discretization: Use this option to enter the value that you want to use to discretize the vaporized gas-oil ratio map.
-------------------------	---

Single fluid	<p>Select this option to use a single fluid model.</p> <p>This section has these options:</p> <ul style="list-style-type: none"> • Fluid: Select the fluid model that you want to use to create an initial condition set for a single fluid model. • Phase behavior: Use this option to select the required phase behavior. A black fluid model will fall under one of the seven phase behavior types: <ul style="list-style-type: none"> • Water (water phase only; this type is not supported by INTERSECT) • Dead oil (oil phase, water, or both) • Dry gas (gas phase, water, or both) • Live oil (oil phase with solution gas and gas phase, water, or both; this type is not supported by INTERSECT) • Wet gas (oil phase, gas phase with vaporized oil, water, or both) • Volatile oil (oil phase with solution gas, gas phase with vaporized oil, water, or both) • Gas condensate (oil phase with solution gas, gas phase with vaporized oil, water, or both) <p>Select Volatile oil if the fluid is bubblepoint. Select Gas condensate if the fluid is dewpoint. For INTERSECT, the fluid models must have a water phase defined, or you will get errors when you export the data.</p>
--------------	--

Multi-fluid	<p>Select this option to use a multi-fluid model.</p> <p>This section has these options:</p> <ul style="list-style-type: none"> • PVT region index property: Use this option to select the appropriate predefined discrete 3D property that distinguishes different PVT regions in the reservoir model. • Phase behavior: Use this option to select the phase behavior against which all the associated fluid models will be validated. A reservoir model should have all the associated fluid models that belong to one single-phase behavior for simulation. • Fluid number: Use this option in the table to associate the correct fluid model with each PVT region. Fluids can be mapped to numbers using the Fluid numbering dialog box. • Fluid: Use this option in the table to enter the name of the fluid.
-------------	---

Contact surface

For contact sets created from surfaces, the Make fluid model process discretizes the contact surfaces into regions with similar values and creates an initial condition for each unique combination of contact depths. The datum depth and pressure are defaulted, if they are not defined in the contact set. A discrete grid property, named after the contact set, is created that maps the initial conditions regions to the grid blocks.

You need to define the following:

- Target number of initial conditions

Enter the target number of initial conditions that you would like to represent the spatial variation in the contact surfaces. This parameter is ignored if the contact set was created from user entered contact depths.

If separate sloping contacts are specified for different zones or segments, then this target is applied separately to each. Therefore, for example, if the number of bins = 5, and there is a different surface specified for the OWC in, say, zones Upper and Lower, the process attempts to create 5 equilibration regions for zone Upper and 5 equilibration regions for zone Lower.



NOTE: This value is a target number of bins; it may not be matched exactly.

- Fill table from contact

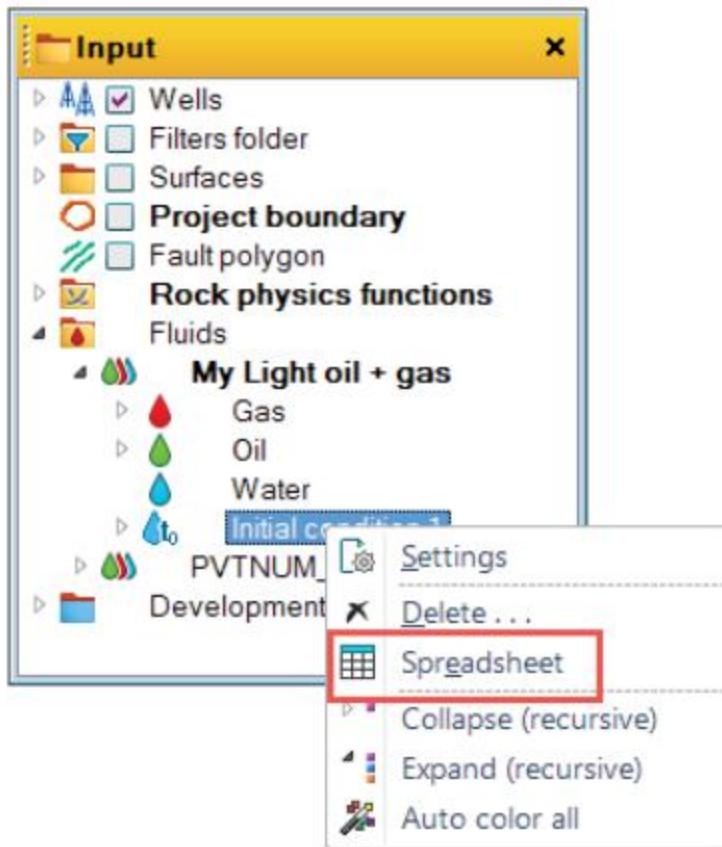
The calculation of contact values from multiple sloping contact surfaces can take some time. Select this option to do the calculation now. The calculated set of initial conditions (if less than 500) is displayed in the table; otherwise the initial conditions and region property are calculated along with the fluid models when you click **Apply** or **OK**.

Fluid variations with depth

Composition of oil frequently is a function of depth.

- Solution gas/oil ratio (R_s) or bubblepoint pressure (P_b)
- Vaporized oil/gas ratio (R_v) or dewpoint (P_d)

To model the variation with depth, fill in the initial conditions spreadsheet. You access the spreadsheet from the Initial conditions subfolder of the Fluid model folder in the **Input** pane.



Copyright © 2017 Schlumberger. All rights reserved.

Figure 96. Accessing the initial conditions spreadsheet from the Input pane

You can specify the bubblepoint or the R_s value at each depth. If you specify P_b , the R_s is calculated automatically. If you specify the R_s , the P_b is calculated automatically. Petrel uses the PVT table that was created from correlations or imported to look up P_b from R_s and R_s from P_b .

The screenshot shows a Microsoft Excel-like spreadsheet window titled "t₀ Spreadsheet for 'Initial condition 1'". The table has columns for Depth (m), Rs (sm³/sm³), and Pb (bar). The data rows are numbered 1 through 4, corresponding to depths of -1000, -1200, -1400, and -1600 meters respectively. The Rs value for row 3 is highlighted in blue.

	Depth m	Rs sm ³ /sm ³	Pb bar
1	-1000	1.1	16.4046
2	-1200	1.05	16.3458
3	-1400	0.95	16.2282
4	-1600	0.9	16.1693

Figure 97. Initial conditions spreadsheet

The Rs value on the **Oil** tab in the **Make fluid model** dialog box is used only for a dead oil initialization. For a live oil, it is used only for calculating the tables. The simulator calculates the constant Rs if it is not defined with datum depth set equal to GOC.

At any position in the reservoir, the Rs value derived from an Rs or Pb versus depth table is subject to an upper limit equal to the saturated value at the local pressure. The Rs value cannot exceed this limit.

Make contacts process

To use contacts as input to the Make fluid model and Initial conditions processes, you must define them in Petrel using the Make contacts process.

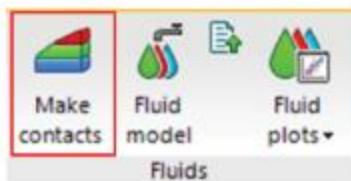
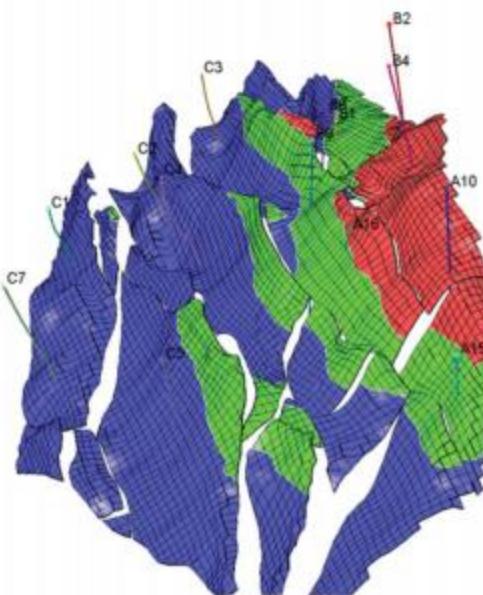


Figure 98. Accessing the Make contacts process in the Fluids group

The Make contacts process allows you to enter different types of contacts, such as constant values and surfaces. You also can use

different contacts for each zone and each segment or the same contacts for the entire 3D model.

You also can visualize contacts on a horizon. Visualizing contacts and horizons together shows the contact contour on the surface along with colored intervals for each hydrocarbon interval. Visualizing contacts is useful when displaying the areal extent of the hydrocarbon intervals.



Copyright © 2017 Schlumberger. All rights reserved.

Figure 99. Visualizing contacts in a 3D window

Procedure — Make a new contact



1. Open the Make contacts process. On the **Reservoir Engineering** tab, in the **Fluids** group, click **Make contacts**.

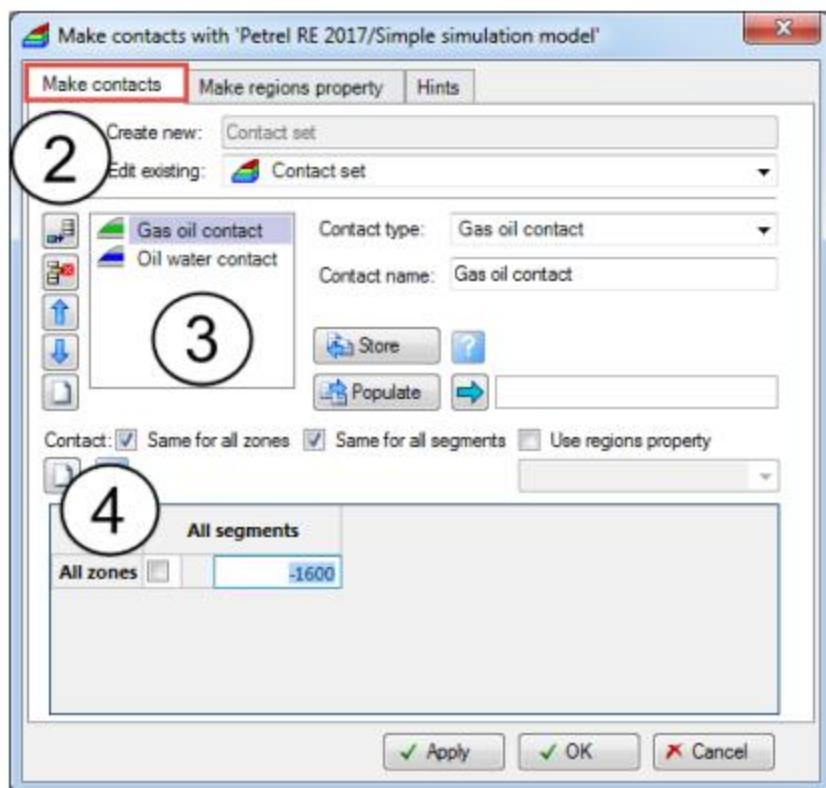


2. In the **Make contacts** dialog box, click **Create new** and name the new contact Contact set.

3. Select the contact type and, optionally, change the name.
4. Define multiple or flat contacts for all zones/regions.

The contact level can be a constant value or a surface. To enter a constant value, enter the value directly into the cell. If it is a surface, select the check box and insert the surface that represents the contact into the cell.

To use different contacts for each segment and zone, clear the options **Same for all zones** and **Same for all segments**. Region properties also can be used to define different contact depths.



Copyright © 2017 Schlumberger. All rights reserved.

Saturation in the water, oil, and gas zone

The **Use initial condition** option for setting initial pressures and saturations is based on the saturation functions, fluid contacts, and datum depth and pressure. On the **Initial conditions** tab in the **Make fluid model** dialog box, the contacts and a datum depth and pressure are specified and hydrostatic equilibrium is assumed.

The black oil equation of state (EOS) is used to calculate the hydrostatic pressure of each phase. Phase saturation in each zone is taken from the saturation functions.

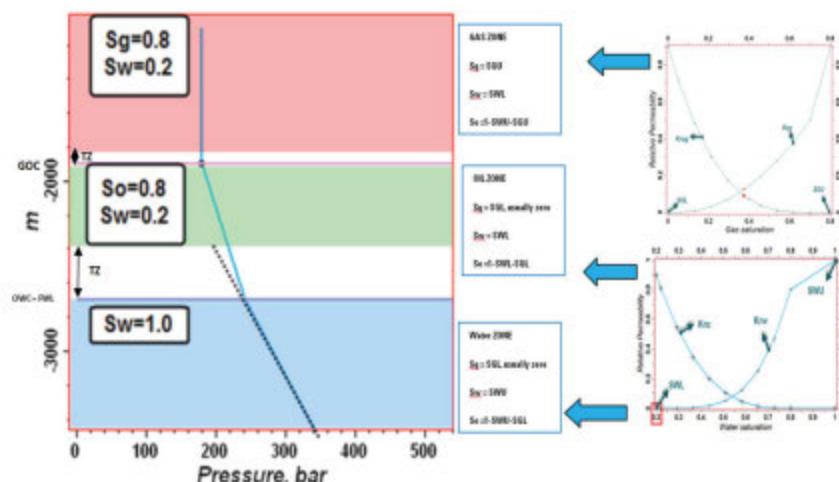


Figure 100. Equilibration calculation

After calculating the initial pressure, the simulator assigns phase saturations in each zone, except for the transition zones. The phase saturations are taken from the saturation endpoints.

- In the gas zone, S_g is at a maximum, which is SGU, the highest gas saturation in the input gas saturation function. S_w is at a minimum, which is SWL, the lowest water saturation in the input water saturation function table that is connate or irreducible water saturation S_{wco} . The oil saturation is then $S_o = 1 - SGU - SWL$.
- In the water zone, S_w is at a maximum, which is SWU, the highest water saturation in the input water saturation function. This value is often 1. S_g is at a minimum, which is SGL, the lowest gas saturation in the input gas saturation function table, that is connate gas saturation S_{gco} .

S_{gco} . This value is usually zero. The oil saturation is then $S_o = 1 - S_{WU} - S_{GL}$. Usually $S_o = 1 - S_{wo}$.

- In the oil zone, both water and gas are at minimum values. $S_g = S_{GL} = S_{gco}$ and $S_w = S_{WL} = S_{wo}$. The oil saturation is then $S_o = 1 - S_{WL} - S_{GL}$. Usually, $S_o = 1 - S_{wo}$.

Saturation in the transition zones

In the transition zone, the phase saturation is governed by capillary pressure. The simulator uses reverse lookup of the capillary pressure curves in the input saturation functions to determine the transition zone saturations.

You can set the capillary pressure (P_c) at the water-oil contact, the gas-oil contact, or both. If you use this option, the OWC and GOC shown on the **Initial conditions** tab are shifted relative to the value of P_c .

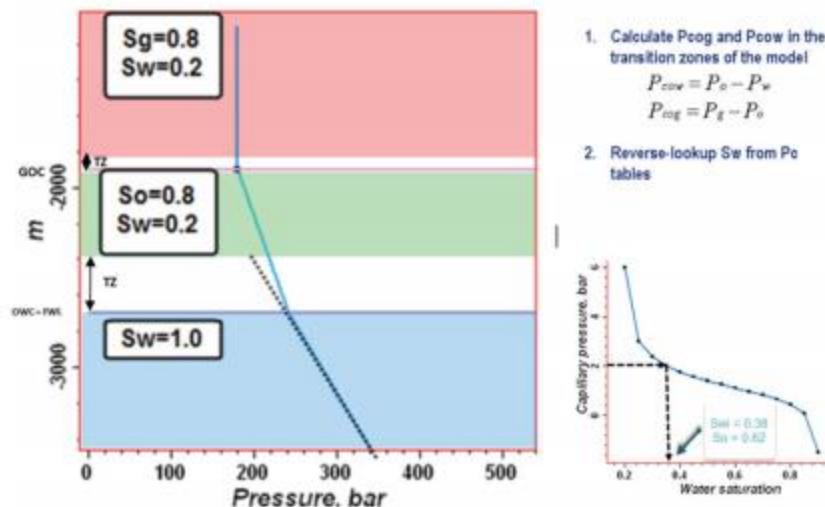


Figure 101. Setting saturation in the transition zones

Define simulation case: Strategies tab

The Development strategy is added to the case on the **Strategies** tab in the **Define simulation case** dialog box. However, you do not require a

development strategy in the model initialization stage, so you can leave the fields on the **Strategies** tab blank.

Development strategies are discussed later in the course.

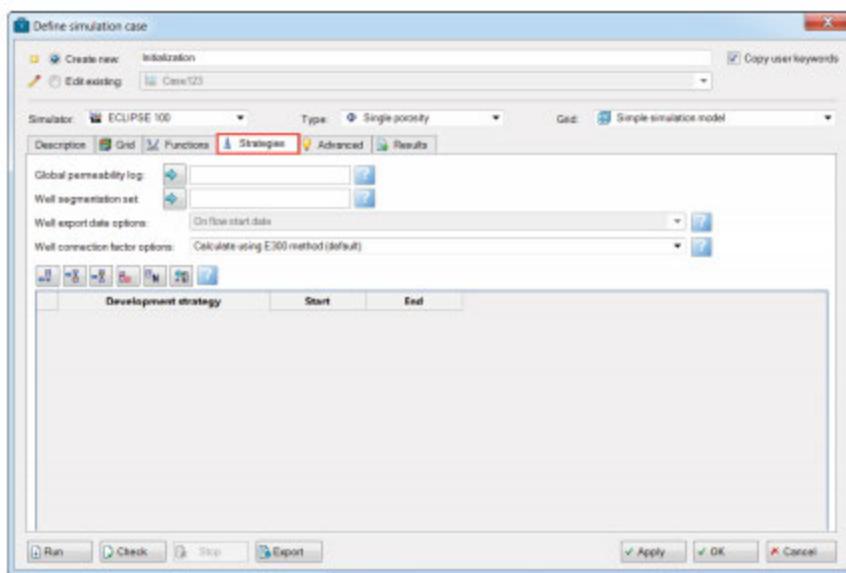


Figure 102. Strategies tab in the Define simulation case dialog box

Copyright © 2017 Schlumberger. All rights reserved.

Exercises — Set up and initialize a simulation case

In these exercises, you first complete the creation of the fluid model by entering the initial conditions. You then assemble all the required input data using the Define simulation case process to initialize the simple simulation case. A development strategy is not used at this stage because you are focusing on model initialization.

Workflow

1. Enter the initial conditions in the **Make fluid** dialog box and view the fluid model with the Fluid spreadsheet.
2. Set up and initialize a simple simulation case with the ECLIPSE Blackoil simulator using the Define simulation case process.
3. Check the original hydrocarbon in place in the generated .PRT file.

Data

Use the file Simple_simulation_model_exercise.pet in the Dataset\Projects\Module-3 Simple simulation model folder.



Exercise 1 Enter the initial conditions

In this exercise, complete the Make fluid model exercise (My Light oil +gas) by entering the initial conditions on the **Initial conditions** tab on the **Make fluid model** dialog box.

1. On the **Initial conditions** tab, use the table and enter -1600 m for the gas-oil contact and -2600 m for the water contact.
2. Enter a pressure at datum (the gas-oil contact) of 157 bar (as shown in the figure).

	Unit	Region
Name		Contact set
Pressure	bar	<input checked="" type="checkbox"/> 157.0000
Datum depth	m	<input type="checkbox"/> -1600.00
Gas-oil contact	m	<input type="checkbox"/> -1600.00
Oil-gas P _c	bar	0.0000
Water contact	m	<input type="checkbox"/> -2600.00
P _c at water contact	bar	0.0000

3. Click **OK**.



Exercise 2 View the fluid model with the spreadsheet

In this exercise, you view the fluid model created with correlation using the spreadsheet.

1. In the **Input** pane, expand the Fluids folder (where the fluid models are stored).
2. Expand My_Light+gas, right-click **Oil** and select **Fluid spreadsheet**.

The fluid spreadsheet opens and displays the oil properties that vary with pressure.

Spreadsheet for 'My Light oil + gas' Oil phase

	Gas To Liquid Ratio	Pressure	Liquid Formation Volume Factor	Oil Viscosity
	sm3/sm3	bar	rm3/sm3	cP
1	60.88717816879	86.84210526316	1.20789302437	0.3583644307213
2		101.0526315789	1.202846260778	0.3655350441635
3		115.2631578947	1.199057837617	0.3739166144425
4		129.4736842105	1.196109311977	0.3834060540736
5		143.6842105263	1.193749253153	0.3939248604276
6		157.8947368421	1.19181748013	0.4054116291439
7		172.1052631579	1.190207110059	0.4178170695773
8		186.3157894737	1.188844093128	0.4311005548699
9		200.5263157895	1.187675504513	0.4452276576836
10		214.7368421053	1.186662512772	0.4601683436493
11		228.9473684211	1.185775981529	0.4758956182705
12		243.1578947368	1.184993621552	0.4923844955768
13		257.3684210526	1.184298091	0.5096112010985
14		271.5789473684	1.183675694928	0.5275525496473
15		285.7894736842	1.183115474521	0.5461854565011
16		300	1.182608556287	0.5654865526161
17		314.2105263158	1.182147678571	0.5854318826491
18		328.4210526316	1.181726841472	0.6059966701936
19		342.6315789474	1.181341044212	0.6271551385781
20	73.07517390386	101.0526315789	1.237172801615	0.3365135375077
21		115.2631578947	1.232119963673	0.3432232401145
22		129.4736842105	1.228190628459	0.3509179523703
23		143.6842105263	1.225047584434	0.3595210608828
24		157.8947368421	1.222476291178	0.3689722279691

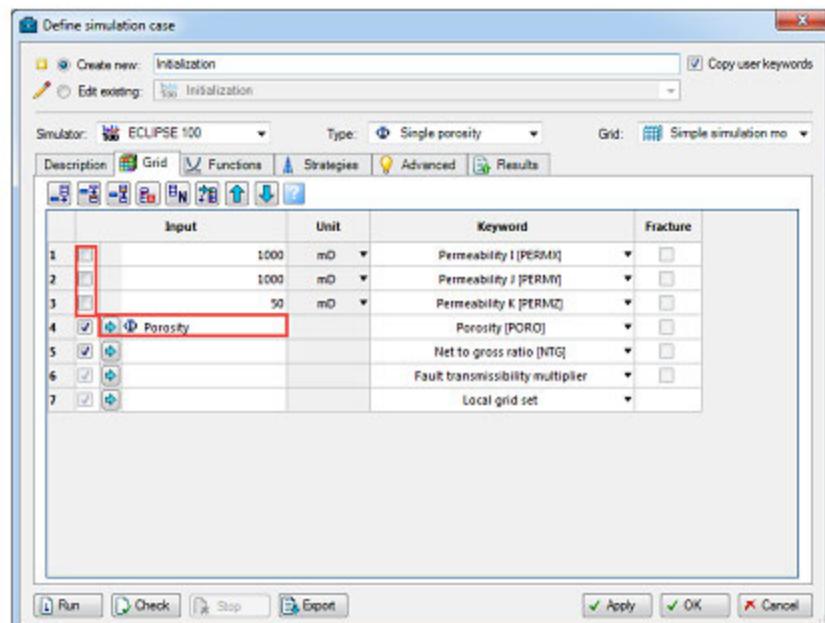
!!!

Apply OK Cancel



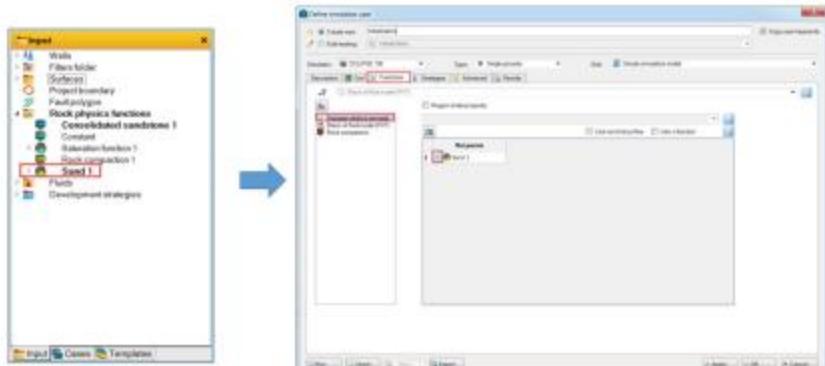
Exercise 3 Use the Define simulation case dialog box to initialize a simple simulation case

1. On the **Simulation** tab, open the **Define simulation case** dialog by clicking **Define case**.
2. Select **Create new** and name the new case **Initialization**.
3. Ensure that the Simulator is set to ECLIPSE 100 and the Grid selected is Simple simulation model.
4. On the **Grid** tab, clear the check boxes in front of Permeability (I,J,K).
5. Enter 1000 for the permeability in the I- and the J- direction and 50 in the K-direction.
6. Insert the porosity property you created in the previous exercise by using simple logic to assign a porosity property to the grid in the Property calculator into the **Define simulation case** dialog box.



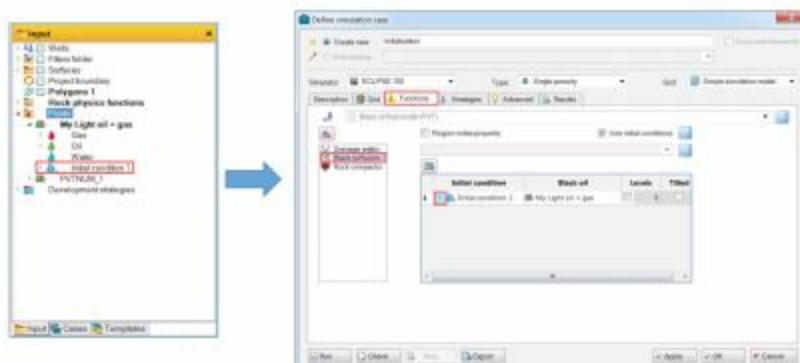
7. On the **Functions** tab, select the Drainage relative permeabilities in the left pane.

8. Insert Sand 1 from the Rock physics functions folder in the **Input** pane.



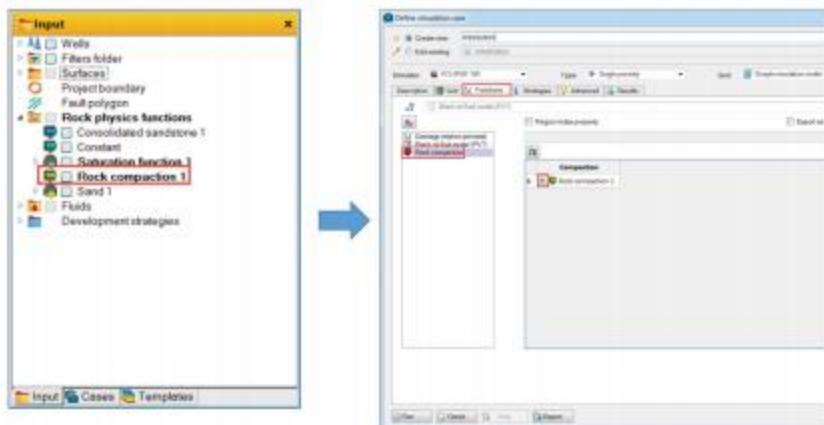
9. Select the Black oil fluid model from the list in the left pane.
 10. Ensure that the **Use initial conditions** option is selected.
 11. Insert the Initial condition 1 of the black oil fluid model, My Light + gas, that you created earlier.

Because you have only one region, there is no reason to use a region index property.



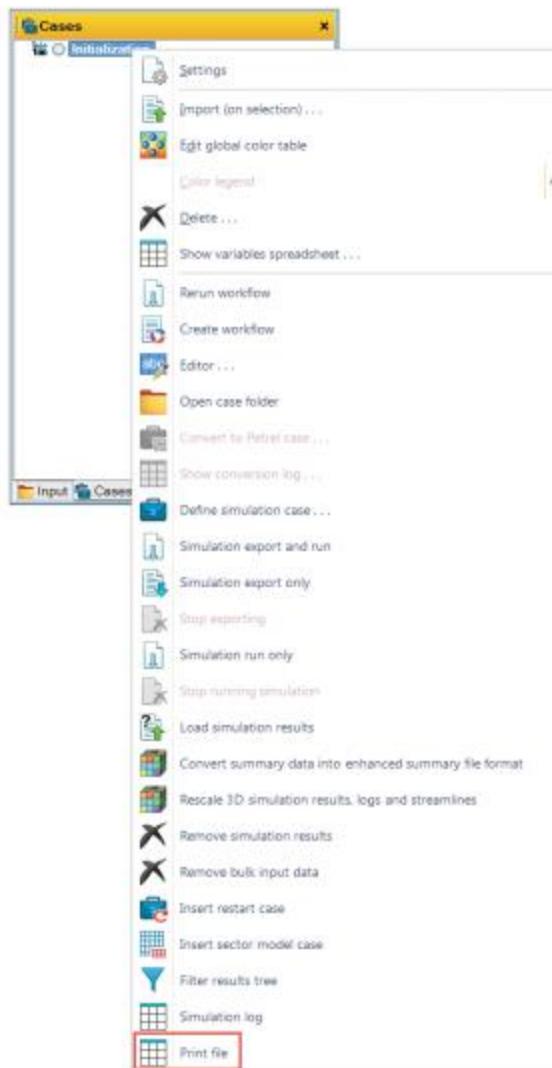
12. Select the Rock compaction function from the same list in the left pane.

13. Insert  the Rock compaction 1 from the Rock physics functions folder in the **Input** pane.



14. Leave the **Strategies** tab empty because you are only initializing the model at this stage.
15. To create the Initialization case, click **Apply**. The case is stored in the **Cases** pane.
16. Save the project while keeping the **Define simulation case** dialog box open.
17. To launch the simulator (ECLIPSE 100), click **Run**. Wait for the initialization of the case to finish.

18. When the run is finished, go to the **Cases** pane, right-click the initialization case, and click **Print file** to open the simulation data desk file using your default text editor.



Copyright © 2017 Schumacher. All rights reserved.

19. Search for BALANCE to view the Initial fluid in place report.



Lesson 2 Volume calculation

The Volume calculation process accurately calculates the volumes in a 3D grid (bulk, pore, and fluid). These figures often are used as a first indication of the economic viability of the field. Together with an uncertainty analysis, they can determine where efforts in reservoir evaluation should be concentrated.

One of the first tasks in a simulation workflow commonly is to validate the volume in place reported by the simulator with that calculated as part of a static modeling workflow.

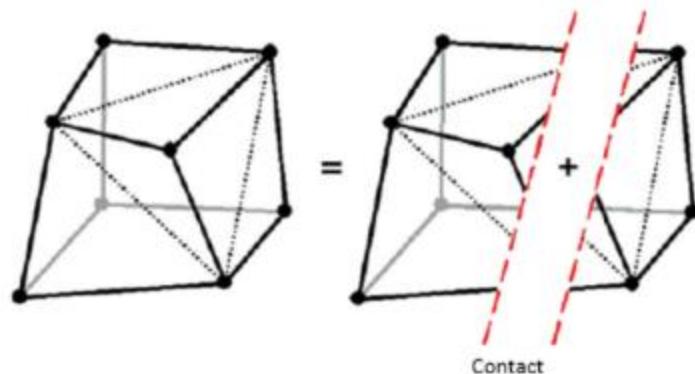


Figure 103. Volume calculation schematic

Copyright © 2017 Schlumberger. All rights reserved.

Volume calculation: Principle

Petrel uses the triangulation-based volume calculation method.

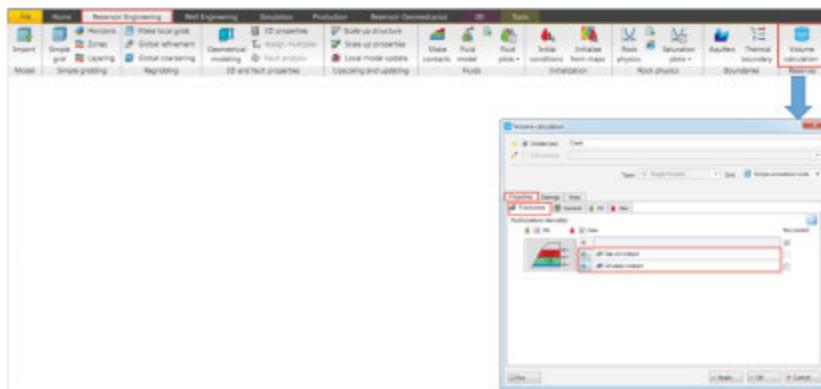
These inputs are needed for volume calculation:

- A 3D grid (if you want to calculate the volumes for each zone and each segment, the 3D grid must be divided into zones and segments).
- Properties, such as Porosity, N/G, Sw/So/Sg. If properties do not exist, a constant value that you enter is used.
- Fluid properties Bo,Bg,Rv, and Rs that are entered as constant values or grid properties.
- Contacts (must be created in the Make contacts process).
- Recovery factors (if available).
- Boundary, such as a license block, if available. (The boundary must exist as polygons.)

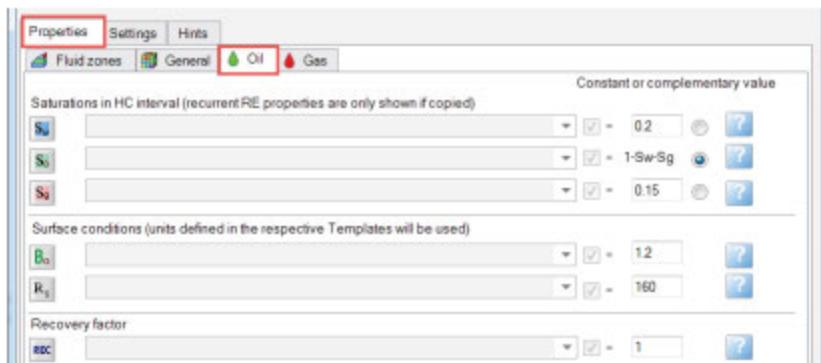
Procedure — Perform volume calculation using a 3D grid



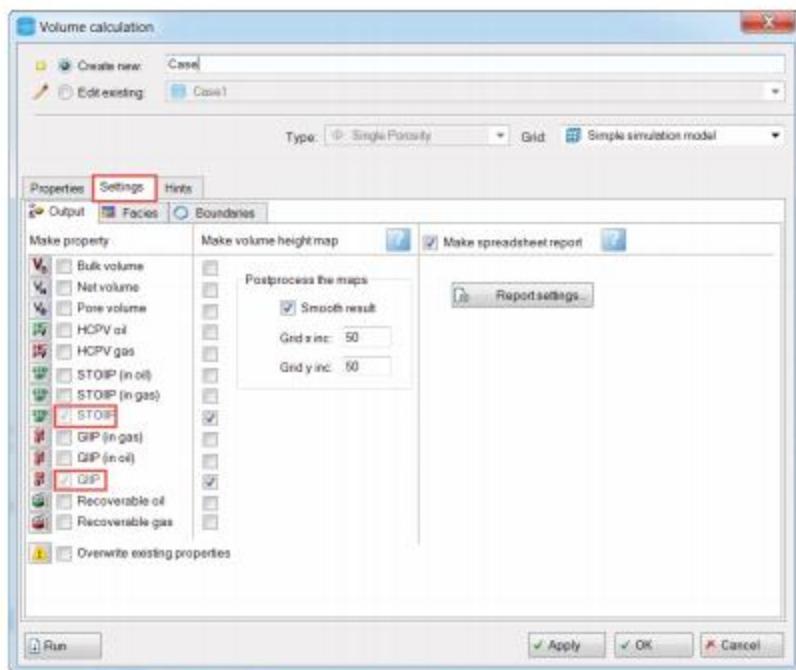
- On the **Reservoir Engineering** tab, in the **Reserves** group, click **Volume calculation** to open the **Volume calculation** dialog box.



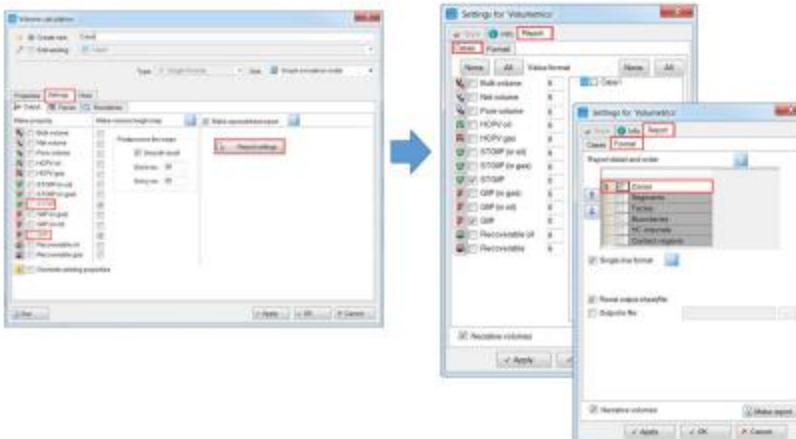
- Create a new case or update an existing case.
- Select a 3D grid from a list to run the Volume calculation.
- Define the Hydrocarbon intervals by inserting the Contacts from the Fluid contacts folder.
- On the **Properties** tab, select a grid block property or a constant value for each required input.



- On the **Settings** tab, specify the output to be generated.



- To specify the case and report format, click **Report settings**.



- Click **Apply** and close the **Settings** dialog boxes.
- In the **Volume calculation** dialog box, click **Apply** to store the data.
- Click **Run**.

A case is created and stored with the simulation cases in the **Cases** pane. Volume calculation results are stored in the same way as simulation results. If the case is deleted from the **Cases** pane, the associated results also are deleted.

The various calculated properties are shown in the **Results** pane under **Volumetrics**, along with the automatic filters (grid zone and segment) and any user-defined filters (polygons or facies filters).

You can view the results as a spreadsheet table of volumes. You can copy the results to any spreadsheet or text editing software.

	Zones	Segments	STOIP[sm ³]	GIIP (in oil)[sm ³]
43	Top reservoir - Mid	Segment 1	426135329	5.294190e+010
45	Mid - Base	Segment 1	173854394	2.159920e+010

Figure 104. Volume calculation results in a spreadsheet

Copyright © 2017 Schlumberger. All rights reserved.

Geologic and simulation grids

The table describes the fundamental differences between geologic and simulation grids.

Table 1. Differences Between Geologic and Simulation Grids

Geologic Models	Simulation Models
Detailed description of structure and petrophysical properties	Description of static properties (structure and petrophysics) is adequate to describe change in saturation and pressure with time.
Compatibility with Depositional model.	Must respect the assumptions taken by the chosen mathematical model.
Grid sizes limited more by structural complexity than strict number of cells.	Grid sizes limited by time and resources.

Volumes: Volume calculation vs. simulation case initialization

The disparity in the static and dynamic volume calculation is based on several factors. Some of the factors are enumerated in the table.

Table 2. Volume Calculation vs. Simulation Case Initialization

	Volume calculation	Initialization
Cells included	All cells above the oil-water contact	All active cells, with special consideration for transition zones
Contact	Computed accurately by triangulation of the cells	If default settings are used, saturations are computed in cell centers.
Fluid model	Constants are used.	Pressure tables are used.
Initial conditions	Constants/grid properties	Saturation endpoints are read from relative permeability. The effect of rock compaction also is taken into consideration. The effect of capillary pressure around the contacts

Copyright © 2017 Schlumberger. All rights reserved.

Volume calculation: Hydrocarbon intervals

To compare volumes computed by the Volume calculation process and the simulator, you must provide correct information on the **Fluid zones** tab in the **Volume calculation** dialog box .

Be aware that the Volume calculation process computes only volumes for cells above the oil-water contact whereas the simulators compute volumes for all cells.

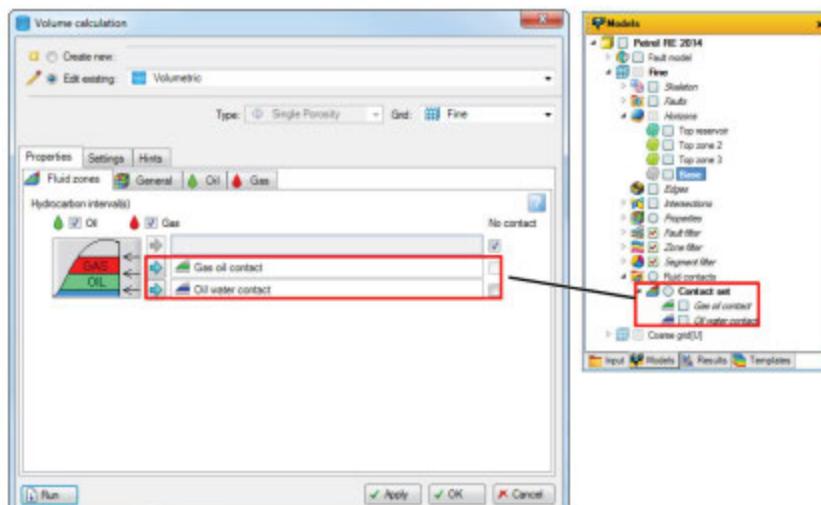
Include both oil and gas intervals in the calculation by selecting the options in the **Hydrocarbon interval(s)** section.

When using GOC with OWC, Petrel computes volumes for each fluid zone separately (cutting cells according to the contact level, if necessary, using the triangulation technique). Later, the bulk volume is reported as the sum of all hydrocarbon zones.

NOTE: Because the total bulk volume is reported as the sum of all hydrocarbon zones, you might see small differences in the total bulk volume for a case with only one zone as compared to a case with several zones.



Insert the contacts from the 3D grid into the drop boxes, as appropriate. The boxes are activated, depending on the selected hydrocarbon intervals.



Copyright © 2017 Schlumberger. All rights reserved.

Figure 105. Inserting the fluid contacts in the Volume calculation dialog box

The next step is to make sure that you use the appropriate porosity. You provide this information on the **General** tab in the **Volume calculation** dialog box. Select property models for each of the inputs or constant values.



Figure 106. Options for using an existing grid property or a constant on the General tab

Bulk volume along with porosity gives the pore volume. You now can run the process, disregarding the rest of the settings. If you want to compare the pore volume computed by the Volume calculation process and the simulator (.PRT file), recall that the simulator also reports volume below the oil-water contact.

Also keep in mind that if a compaction function was used with the simulation case, the pore volume is a function of pressure.

Fluid description: Use constants or existing grid properties

Depending on the number of hydrocarbon intervals you select on the **General** tab, provide the phase saturations S_w , S_g , S_o on the **Oil** tab, **Gas** tab, or both.

The initial water saturation distribution is one of the most significant factors in the calculation of hydrocarbon volumes.

The Volume calculation process allows you to specify the water saturation as a constant or as a 3D grid property. In addition, the formation volume factors strongly influence the results. The simulator gets all of this information from the saturation function and the fluid model.

To compare volumes calculated by the Volume calculation process and the simulator, review the inputs that were used for both processes, especially PVT.

Provide values for B_o , R_s for the oil zone and B_g , R_v for the gas zone.

After you make your selections and before you run the simulation, click the **Settings** tab in the **Volume calculation** dialog box to change the report settings or request the output of the properties from the Volume calculation process.

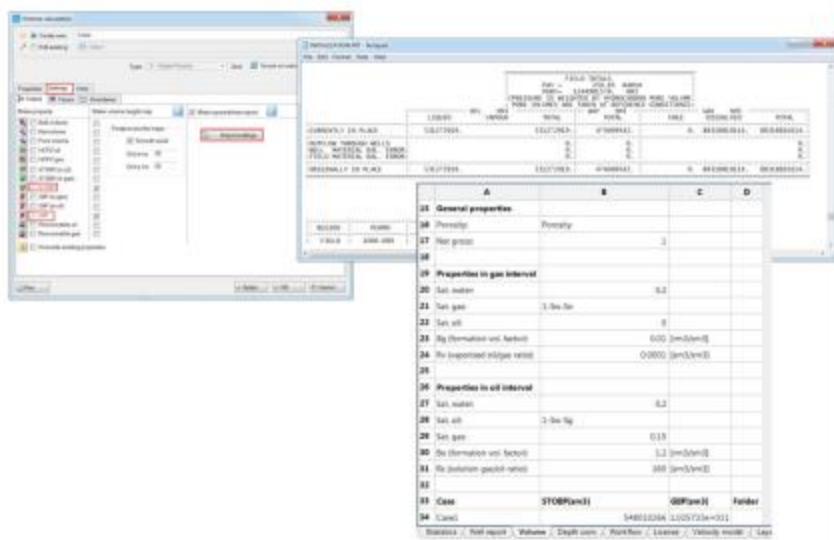


Figure 107. Report settings in the Volume calculation dialog box and PRT file report from the simulator

Copyright © 2017 Schlumberger. All rights reserved.

Exercise 1 Use the Make contacts dialog box to create a fluid contact set

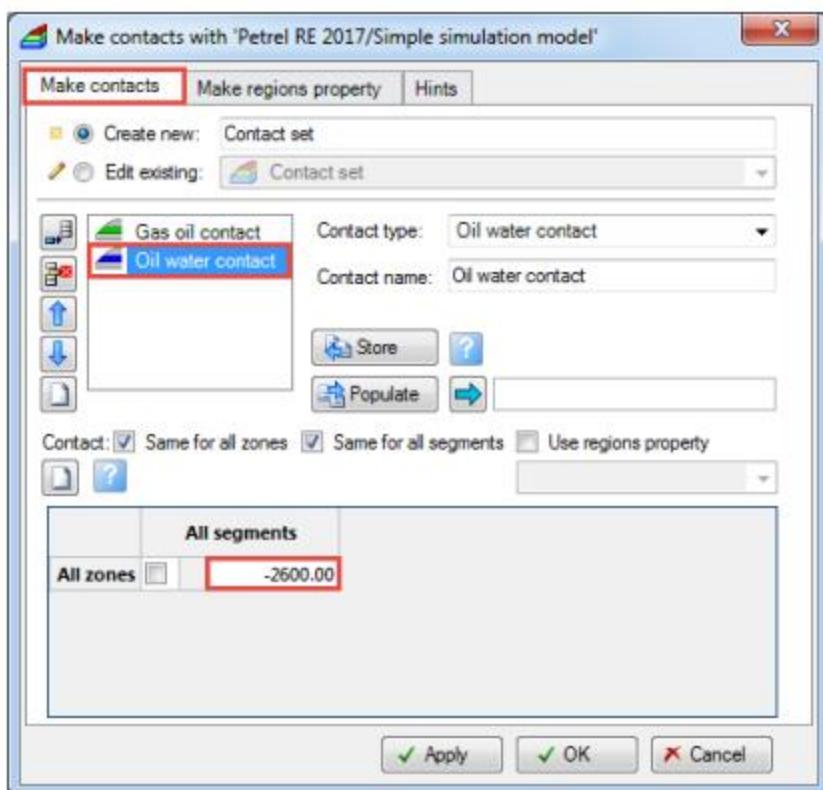
In this exercise, you create a new contact set that you use in volume calculation.

Use the file `Simple_simulation_model_exercise.pet` in the `Dataset\Projects\Modules-3 Simple simulation model` folder.

1. Ensure that the Simple simulation model grid is active.
2. On the **Reservoir Engineering** tab, in the **Fluids** group, click **Make contacts** to open the **Make contacts** dialog box.

A default setting is available on the **Make contacts** tab with two predefined contacts: **Gas oil contact** and **Oil water contact**.

3. Select **Oil water contact** and enter the Z-value of -2600m.



4. To create fluid contacts, click **OK**.

The generated fluid contacts set is stored in the **Fluid contacts** folder in the **Models** pane.

Exercises — Perform volume calculations

In this series of exercises, you learn how to calculate static volumes using the **Volume calculation** tool in Petrel. Volume can be calculated within zones, segments, and user-defined boundaries (for example, license boundaries). Compare the static volume with a volume in place for a model initialized in a simulator.

Workflow

1. Create 3D grid property for Bo and Rs.
2. Create a new volumetric case.
3. Enter the fluid contact by inserting the existing contacts into the contacts drop boxes.
4. Enter the grid and fluid properties.
5. Specify output properties and report format.
6. Run the volumetric calculation.

Data

Use the file `Simple_simulation_model_exercise.pet` in the `Dataset\Projects\Modules-3 Simple simulation model` folder.

Copyright © 2017 Schlumberger. All rights reserved.

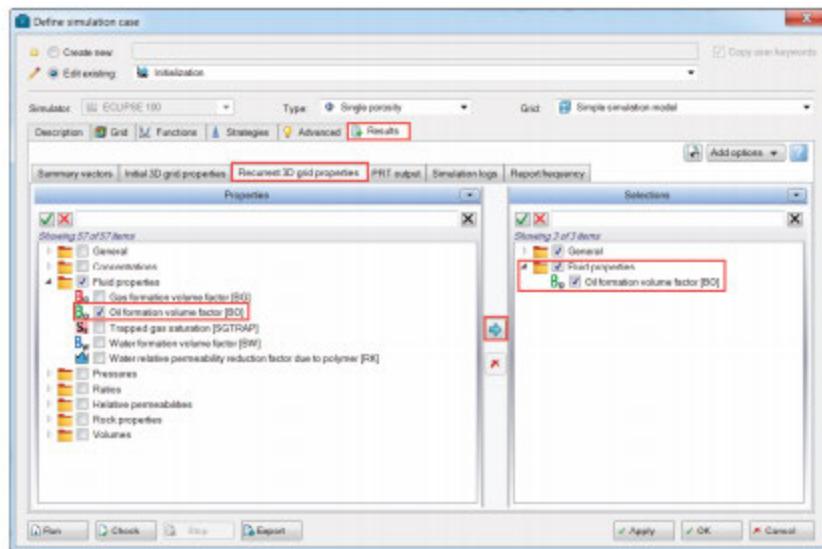
Exercise 1 Create 3D grid property for Bo and Rs

In this exercise, you learn how to create 3D grid properties such Bo and Rs. The 3D properties (Bo, Rs) will be used in volumetric calculation instead of constant parameters. This workflow helps to ensure consistency in the fluid properties used for both simulation and static volumetric calculations.



1. On the **Simulation** tab, in the **Simulation** group, click **Define case** to open the previously created Initialization case.
2. Select the **Results** tab on the **Define simulation case** dialog box to add additional simulation output.
3. On the **Recurrent 3D grid properties** tab in the **Properties** pane, open the fluid properties folder.

4. Activate Bo and insert it into the **Selection** pane.



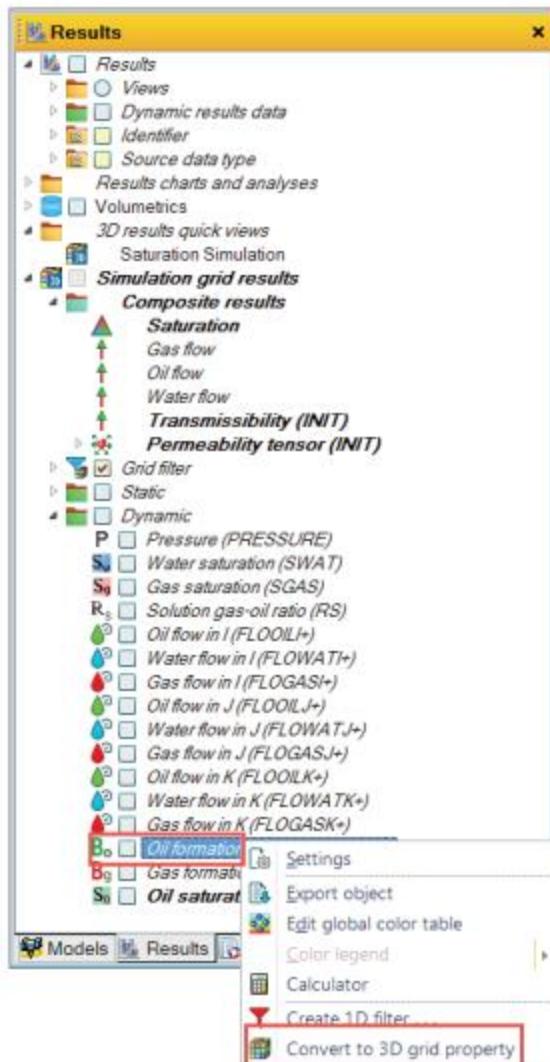
5. Rerun the Initialization simulation to generate the Dynamic 3D properties for the selected fluid properties.

The generated properties are stored in the Dynamic subfolder of the Simulation grid results folder.



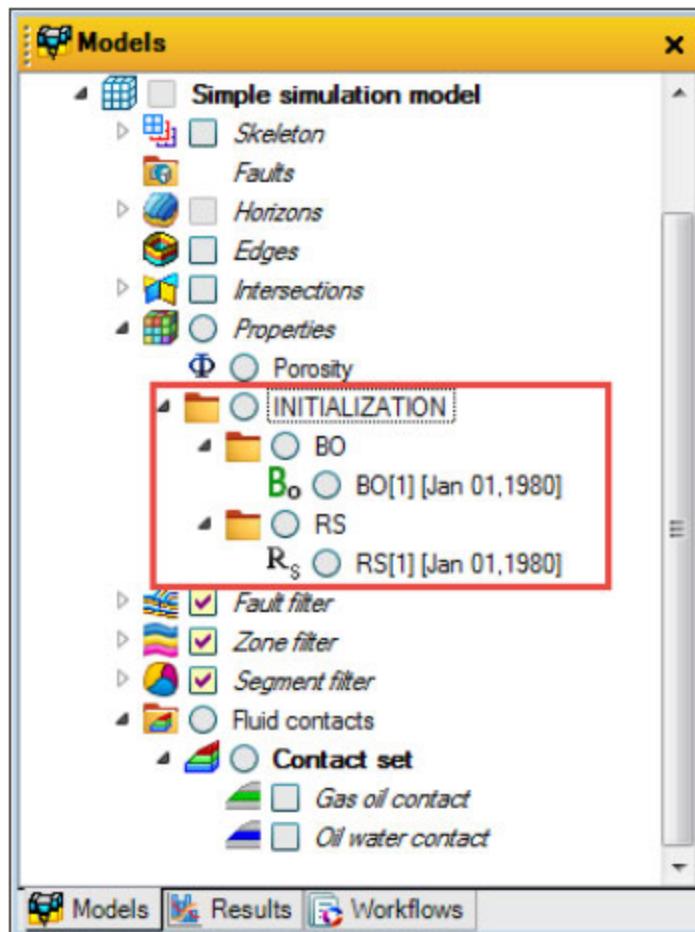
Copyright © 2017 Schlumberger. All rights reserved.

6. Turn on the Initialization simulation case in the **Cases** pane, right-click the Bo in the Dynamic folder and click **Convert to 3D grid property**.



Copyright © 2017 Schlumberger. All rights reserved.

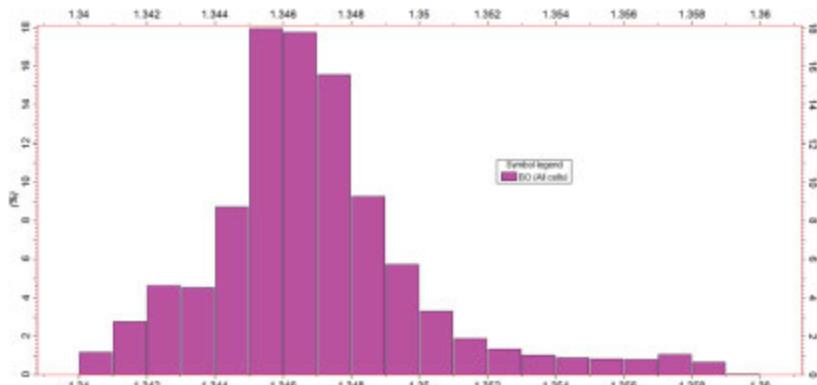
- In the **Settings for Simulation grid results** dialog box, select Bo, Rs, Timesteps and click **OK** to generate the 3D grid properties. They are stored in the **Properties** folder of the Simple simulation model in the **Models** pane.



Copyright © 2017 Schlumberger. All rights reserved.

- On the **Home** tab, in the **Insert** group, click **Window** and then click **Histogram window** to open a new **Histogram window**.

9. Activate the generated Bo 3D grid property to view the distribution.



10. Save the project.

Exercise 2 Create a new Volume calculation case



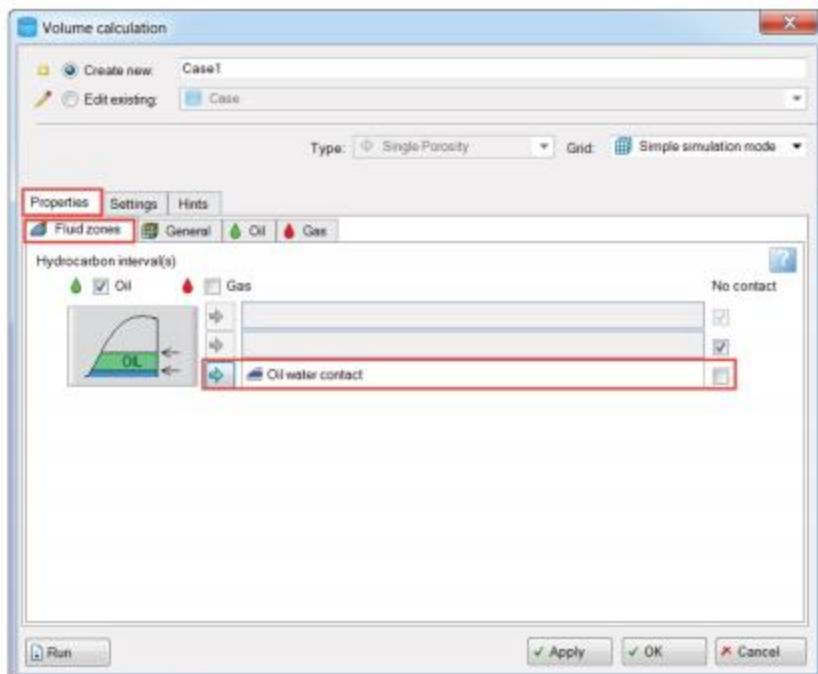
1. On the **Reservoir Engineering** tab, in the **Reserves** group, click **Volume calculation** to open the **Volume calculation** dialog box.



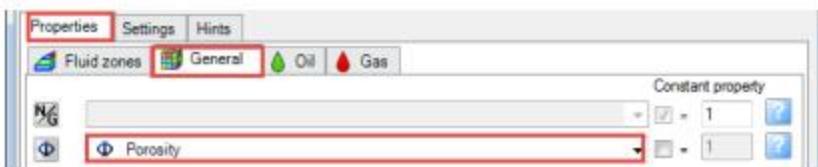
2. Click **Create new** and name the new case **Case1**.
 3. Select the grid named Simple simulation model from the **Grid** list.
 4. Specify the Hydrocarbon interval(s). Select the **Oil** check box to activate the contact boxes. Ensure that you clear the **No contact** check box for Oil on the right side of the dialog box.



5. In the **Models** pane, expand Simple simulation mode, expand Fluid contacts, and insert the Oil water contact into the contact box.



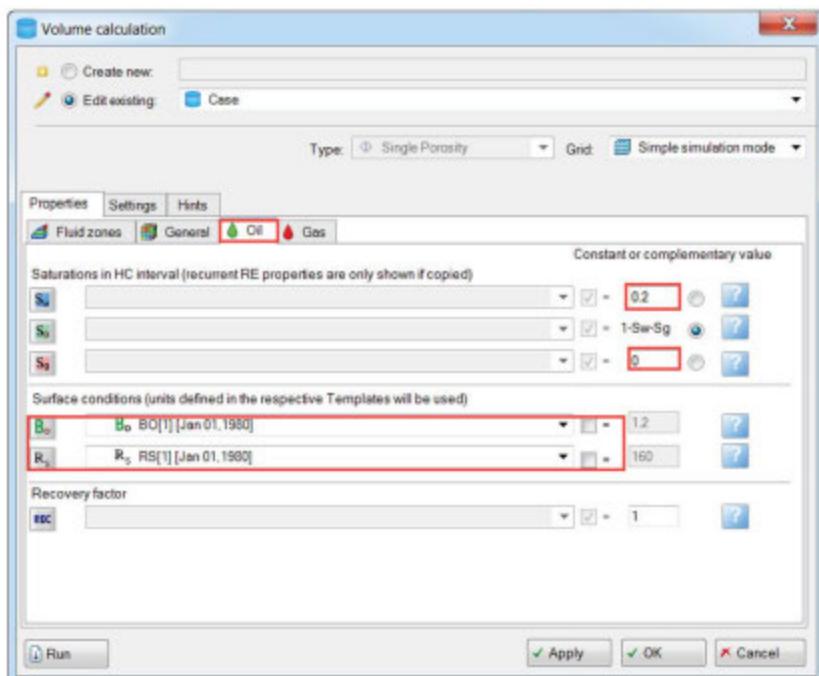
6. On the **General** tab, enter 1 in the **Constant property** field for Net/Gross and clear the check box for **Constant property** to use the existing porosity model that you created previously using simple logic in the **Property calculator**.



Copyright © 2017 Schlumberger. All rights reserved.

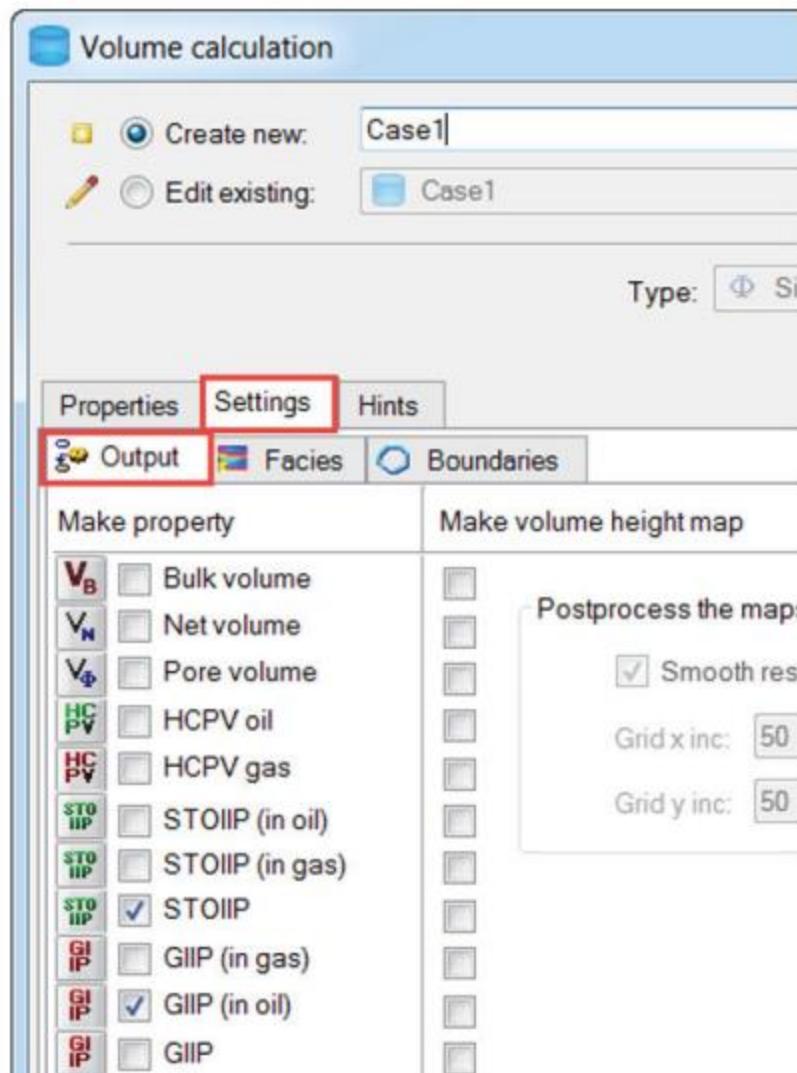
7. On the **Oil** tab, enter these values for **Constant**:

- S_w (water saturation) = 0.2 (same value used in the simulation. Open the spreadsheet for Oil-water relative permeability to confirm.)
- S_g (Gas saturation) = 0 (no initial free gas in the oil zone in the dynamic model).
- Use the B_o 3D grid property instead of a constant.
- Use the R_s 3D grid property instead of a constant. Ensure that you clear the check boxes in front of the B_o and R_s data field to enable the use of the 3D grid properties.



8. Click the **Settings** tab and then click the **Output** tab.
9. On the **Output** tab, select properties as shown in the figure.

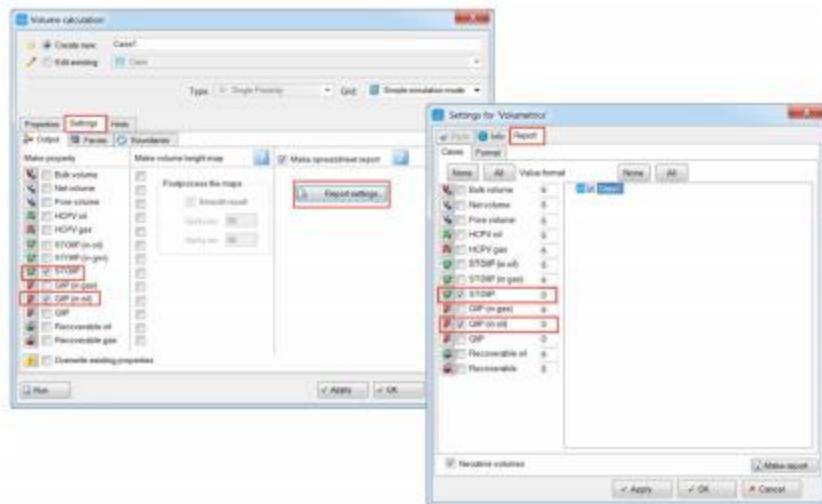
The properties you select are created and stored in the Properties folder of your Simple simulation model grid in the **Models** pane when you run the volume calculation case.



Copyright © 2017 Schlumberger. All rights reserved.

10. Click **Run**. A report is created in the **Output** sheet. All the properties that you previously selected are stored in the **Models** pane.
11. Click **Report settings**. In the **Settings** dialog box, click **None**.

12. Activate only STOIIP and GIIP(in oil) with a Value format equal to zero on the **Report** tab.



13. Click **Apply** and close the **Settings** dialog box.
14. In the **Volume calculation** dialog box, click **Run** to run the volume calculation.
15. To open the **Print** file of the Initialization case, activate the Initialization case in the **Cases** pane.
16. On the **Simulation** tab, in the **Case management** group, click **Reports**, and then click **Print file**.



17. Search for BALANCE to view the initial volume and compare it with the static volume calculated using the Volume calculation process. The percentage difference should be around 0.1%.

	A	B	C	D
1	Petrel 2015.1 044 040 Beta	In ModelSpace		
2	User name	Abdullah		
3	Date	Wednesday, July 06 2016 13:26:06		
4	Project	Simple_initialization_reservoir_exercise.pet		
5				
6	Model	Petrel RE 2016		
7	Grid	Simple simulation model		
8	Input XY units	m		
9	Input Z unit	m		
10				
11	WC settings	Includes saturation only		
12	Leave no contact	Contact cell/Cell water contact		
13				
14	General properties			
15	Press.ref	1013250		
16	Acq.grav	0		
17				
18	Properties in oil interval			
19	Oil formation vol. fac(%)	1.000000		
20	Oil suppressed storage ratio(%)	0 (medium)		
21				
22	Properties in oil interval			
23	Sat. water	0.2		
24	Sat. oil	0.99792		
25	Sat. gas	0		
26	Oil formation vol. fac(%)	0.0	(medium)	
27	Oil suppressed storage ratio(%)	0.0	(medium)	
28				
29	Cost	\$1000000000	1000000000	Folder
30	Cost			

Copyright © 2017 Schlumberger. All rights reserved.

Review and summary

This section provides a review and summary of what you learned in Model initialization and volume calculation.

Review questions

These questions help you review the major concepts covered in this module.



- What are the options available for initializing a reservoir model?
- Why is there a disparity between the static and dynamic volume calculations?

Summary

In this module, you learned:

- creating initial conditions
- initializing a simulation model using Petrel Reservoir Engineering workflow tools

- static volume calculation
- Make contacts process
- comparing the static volume with a volume in place for a model initialized in a simulator
- the possible pitfalls when comparing volumes calculated by the Volume calculation process (static volume) with volumes calculated by the simulator

NOTES

Copyright © 2017 Schlumberger. All rights reserved.

NOTES

Copyright © 2017 Schlumberger. All rights reserved.

Module 5 Simulation run and results viewing

In this module, you learn how to set up a simulation case with an existing development strategy and use the **Results** tab on the **Define simulation case** dialog box to define which computed results will be generated by the simulator. How to create a development strategy is discussed in detail in Module 6.

This module introduces you to several ways of displaying and analyzing simulation results in the Petrel environment using various results visualization and analysis tools. It also covers how to calculate new grid properties and simulation vectors from existing results using the **Summary calculator** and **3D result calculator**.

You also learn about the **Geometrical modeling** tool, which allows you to make geometrical properties that are relevant to the reservoir engineering workflow.

Copyright © 2017 Schlumberger. All rights reserved.

Learning objectives

After completing this module, you will know how to:



- include the development strategy in the **Define simulation case** dialog box and run a simulation
- make field/well performance analysis plots using the **Results charting and analysis** tool
- create thumbnail plots using the Split view functionality
- use the picking mode options and well player
- customize the data display by creating a chart theme and series styles
- export chart data to Microsoft Excel and chart images to the clipboard
- visualize 3D simulation grid results
- create grid properties and new summary vectors from existing results using the Summary and 3D results calculators
- create geometrical properties

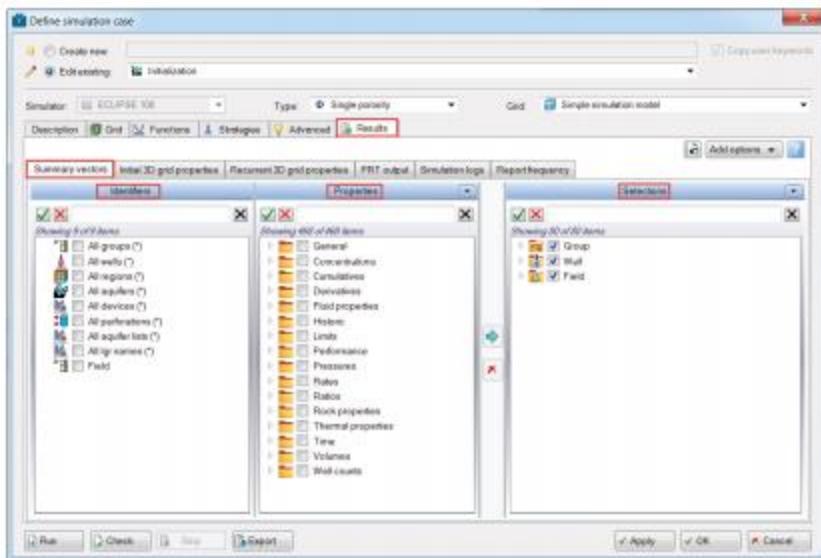
- create production bubble plots
- use filters in Petrel
- use the Multi-value probe as a quality checking tool

Lesson 1 Simulation case setup and output requirements



To set up a simulation case using the **Define simulation case** tool, you first must specify the input properties (the grid and its properties). You then select predefined initial conditions and fluids models (PVT), rock physics functions (Relative permeability and Rock compressibility), and development strategies (Wells and Rate control).

On the **Results** tab in the **Define simulation case** dialog box, select the outputs and results to be computed by the simulator. The completed simulation case appears in the list in the **Cases** pane.



Copyright © 2017 Schlumberger. All rights reserved.

Figure 108. Results tab in the Define simulation case dialog box

The time-based results that you select on the **Summary vectors** tab appear in the Dynamic results data folder on the **Results** pane. Use the identifier and property selections on the **Summary vectors** tab to define the dynamic data that is output from a simulation case.

When you select identifiers, incompatible property options are disabled in the **Properties** pane. To populate the **Selection** pane, you insert the identifiers and properties that you select.

You also can use the search box to locate the required properties while making selections. For example, enter Pressure to locate all properties that contain the word pressure, such as bottomhole pressure.

You can categorize properties to make selection easier. In the **Define simulation case** dialog box, on the **Summary vectors** tab, click **Properties**, click **Categorize**, and then click the required category. You also can categorize selections in a similar way so that you can check that the correct summary vectors are selected.

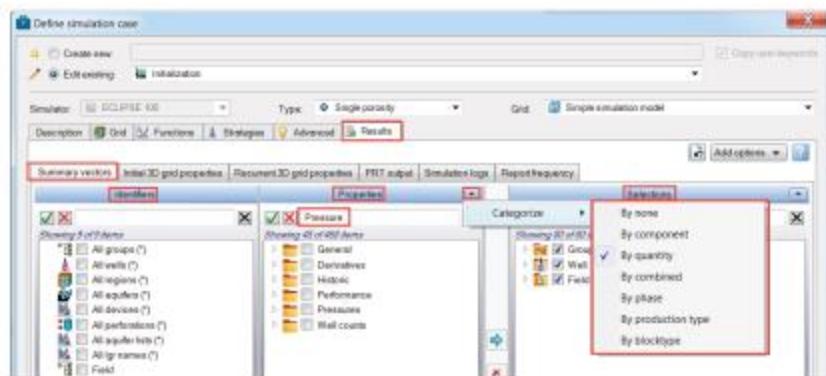


Figure 109. Summary vectors tab in the Define simulation case dialog box

Initial 3D grid properties, Recurrent 3D grid properties, and PRT 3D grid properties tabs

Use property selections on these tabs to specify initial and recurrent grid properties and to define which grid properties you want to appear in the print file output. Make your selections from the **Properties** pane and

insert them to populate the **Selections** pane.

You also can use the search box to locate the required properties. The categorization option is available for both properties and selections.



Lesson 2 Results charting and analysis

The Results charting and analysis process is an excellent tool for creating high-quality dynamic plots of vectors against time.

There are several ways to use the Results charting and analysis workflow to carry out common analyses of production data or simulation results:

- Field performance analysis: Analysis of field level information through field performance analysis. This type of workflow can be applied to reviewing multiple properties for a single identifier such as a well.
- Global well comparison: Analysis of the same type of information for multiple identifiers, such as viewing the oil production rate for all the wells in a field.
- Global case comparison: Testing multiple studies, such as history matching analysis where you view changes to multiple identifiers.

The Results charting and analysis process provides these benefits.

- Efficient selection of your results data:
 - Specialized process for efficient data selection.
 - Interactive searching to find and filter your data quickly.
 - Layouts and styles that you can customize and save for reuse or sharing.
- Highly customizable studies:
 - Quick access object information using the **Inspector** tool.
 - Lists of prebuilt themes and line templates.
 - Results can be used as a viewport plot to create multidisciplinary charts.
- Enhanced productivity:
 - Plotted results can be split to generate thousands of line graphs rapidly for easier comparison.
 - Save and reuse your work without saving Petrel windows so that you can preserve your work.
 - Share your work and deploy themes and line templates in your organization using the **Reference project** tool.
 - View and analyze results in the **Charting window Inspector**.

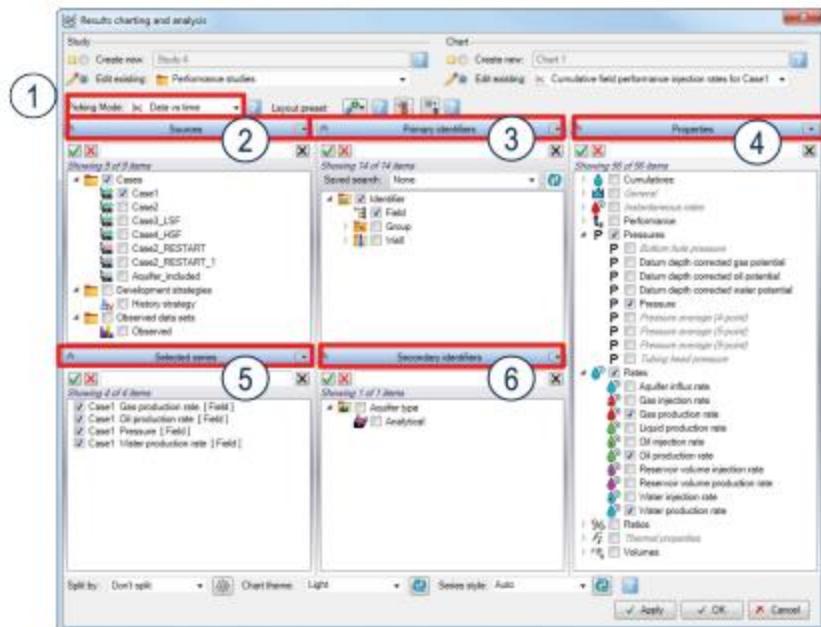


Figure 110. Results charting and analysis dialog box

- 1 **Picking modes:** Available methods for plotting data
- 2 **Sources** pane: Contains simulations, development strategies, and observed data
- 3 **Primary identifiers** pane: Contains the wells, groups, and field identifiers in the selected sources.
- 4 **Properties** pane: Contains the properties of the primary and secondary identifiers in the selected sources
- 5 **Selected series** pane: Shows the plotted vectors
- 6 **Secondary identifiers** pane: Contains a list of the secondary identifiers in the selected sources, such as Aquifer type, Fluid type (fluid in place set) and History match data (Objective functions).

Main components of Results charting and analysis

This section discusses the main components used in Results charting and analysis.

Results charting and analysis dialog box: This dialog box is used to choose the data required to plot a chart. This dialog box consists of selection panes from which you choose identifiers and properties to be plotted.

The figure shows the other main components that are part of the **Results charting and analysis** tool.



Figure 111. Main components of Results charting and analysis process

- 1 **Charting** window
- 2 **Inspector**
- 3 **Charting** window toolbar
- 4 **Charting** window tab
- 5 **Results** pane

Charting window

Displays only charts created by the Results charting and analysis process. You can modify the style of the charts in this window. Change chart content and axes, windows background, and the window legend by using the shortcut menus.

Inspector

In a **Charting** window, use the **Inspector** to examine values on the chart. The **Inspector** is available for all charts, but it operates differently, depending on the picking mode of the chart.

Charting window toolbar

Provides quick access to tools used to control the display in the **Charting** window. Tools include pan, zoom, and view all. You can move between identifiers when data for just one primary identifier, such as a well, is plotted. You also can open the **Results charting and analysis** tool from the toolbar, if it is not open already.

Charting window tab

Appears when a **Charting** window is open. The tab provides access to functionality that is specific to the chart. This functionality includes exporting graphics and Excel files, editing series styles and themes, and customizing splitting.

Results pane

Contains results from volume calculations and simulations runs. This pane also contains the customized settings that you enter in the **Settings** dialog box. Created charts are stored in the Results charts and analyses folder in the **Results** pane.

Preset options to create field performance plots

To quickly make an analysis of field level performance for the selected simulation case, use the **Field plots** tools (see the figure).

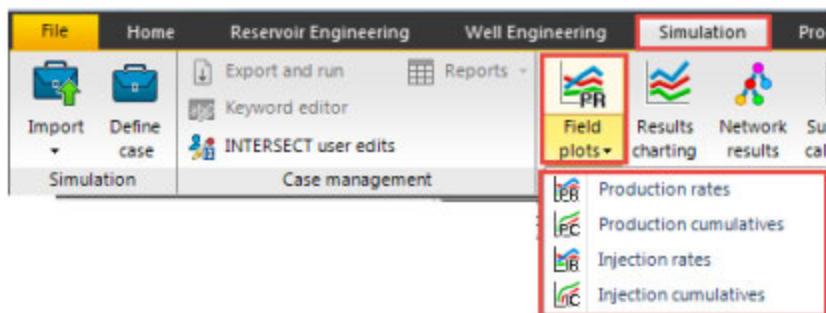


Figure 112. Field plot preset options

Picking modes

Picking modes are methods of selecting data in the **Results charting and analysis** dialog box. The available modes are

- Data vs. time: Plot 2D results and observed data on the Y-axis against time or date on the X-axis.
- Data vs. time (additive): An extension of Data vs. time. Individual series are added to the chart one at a time. This mode allows you to build more complex custom plots. This plot type displays these buttons:
 - **Add** Add the series determined by the selected sources, identifiers, and properties.
 - **Remove** Remove highlighted series in the **Select series** pane from the chart.
 - **Remove all** Remove all the selected series from the chart.
- Scatter by time: Plot two data series from the same source (case/observed dataset) and identifier (well/group/field) against each other. Each point represents a single identifier at a single time, with X and Y values representing the values of two different properties.

- Scatter by identifier: Plot multiple time-based data series from the same source (case/observed dataset) against each other. Each point represents a single identifier at the time selected in the **Time player**.
 - Simulation vs. observed: Plot data from one or more cases against an observed dataset to analyze any differences. The data is plotted at a particular timestep chosen with the **Time player**. Each point represents a single identifier for a single property at a single time.
- The Y value is chosen from a simulation case and the X value is chosen from the observed dataset. Any points on the X=Y line are exact matches. For points below the line, the observed quantity is larger than the simulated quantity. For points above the line, the observed quantity is smaller.
- Simulation event: Allow plotting of simulation events with summary vectors. Well and completion status changes (Open/Shut) including the display of the constraints for open wells are supported.

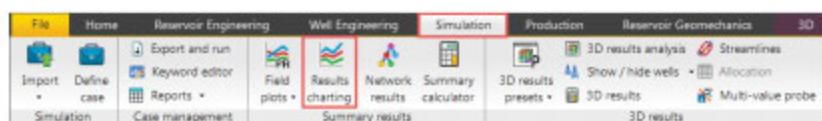


Procedure — Create a plot using the Results charting and analysis tool

View and chart simulation results.

1. On the **Simulation** or **Production** tab, in the **Summary results** group, click **Results charting**.

You also can access the tool from the **Charting** window toolbar.



2. In the **Results charting and analysis** dialog box in the **Study** section, click **Create new** and enter a name for the study.
3. In the **Chart** section, click **Create new** and enter a name for the chart.
4. In the **Sources** pane, choose the case that contains the results you want to view.
5. Choose the required primary identifiers and corresponding properties that you want to view.

Filter any list of results by entering search terms in the field at the top of the pane. You also can choose items by selecting the check box at the top of the pane.

A new folder that contains the study and chart is created in the **Results** pane.

Result charting and analysis: Settings

To access the settings, double-click in the active chart window.

Alternatively, right-click a chart in the **Results** pane (see the figure) and click **Settings**.

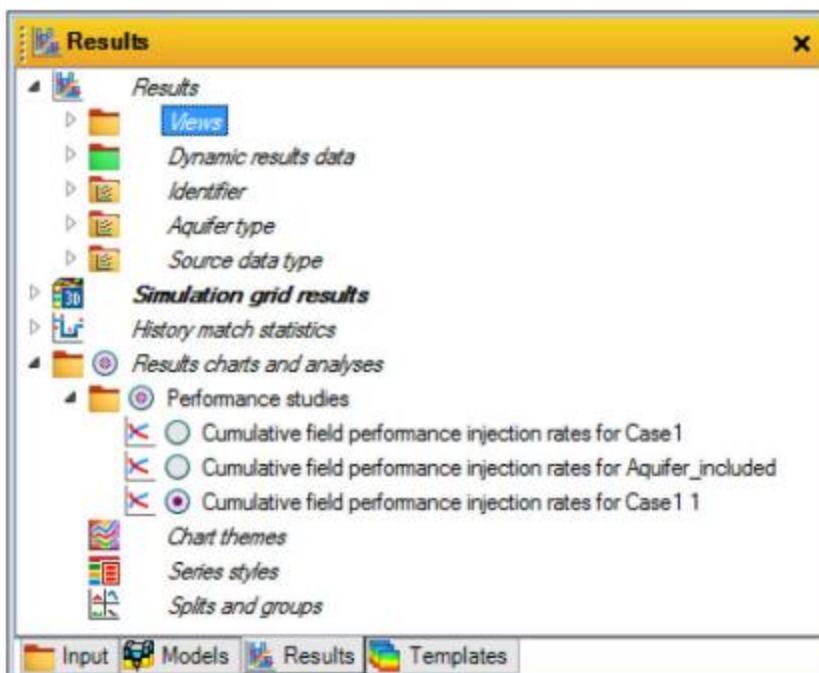


Figure 113. Results pane

You can modify all of the elements in the chart (header, legend style, title style, axis, line style, and color) on the **Style** tab in the **Settings** dialog box.

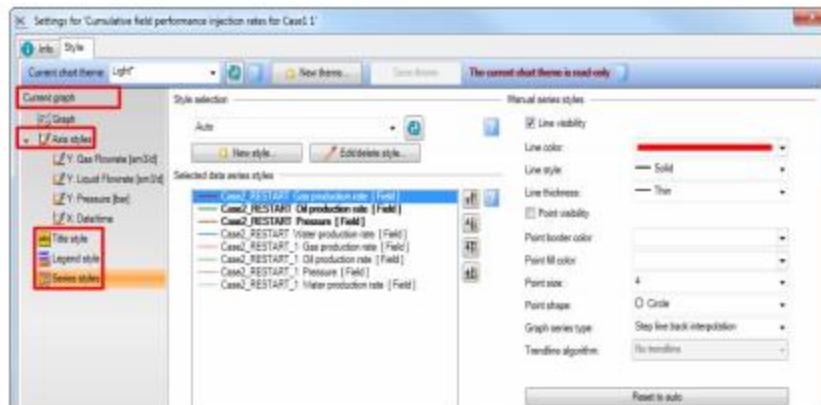


Figure 114. Style tab in the Settings dialog box

Split views

The **Charting** window initially groups all plotted lines onto a single graph. You can view this data differently using the **Split by** option.

For example, to split the graph into one plot per property, right-click the graph, click **Split by**, and then click **Property**. Alternatively, use the **Split by** menu in the bottom left of the **Results charting and analysis** dialog box.

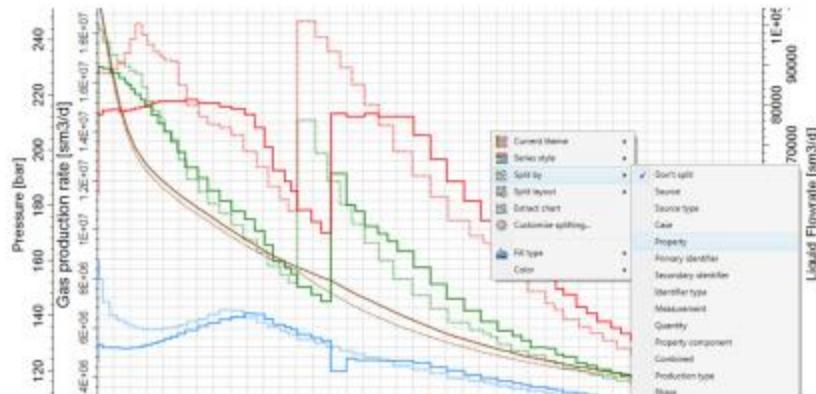


Figure 115. Procedure for splitting graphs into one property per plot

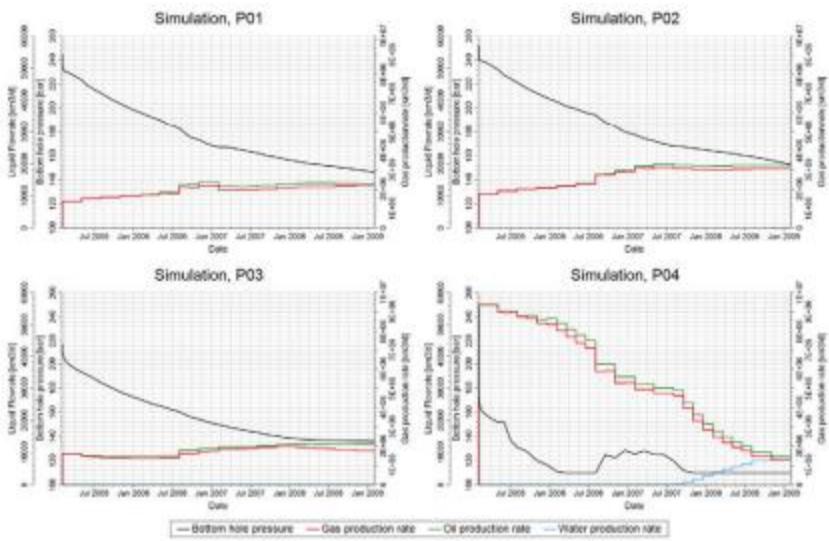


Figure 116. Plots per property (Well rate and Pressure)

You can change the color scheme used to identify different series on the plots.

For example, if properties from multiple wells have been plotted, then you can select to color each series by its identifier (in this case, Wells). Right-click the graph, click **Series style**, and then click **Identifier determines color family**.

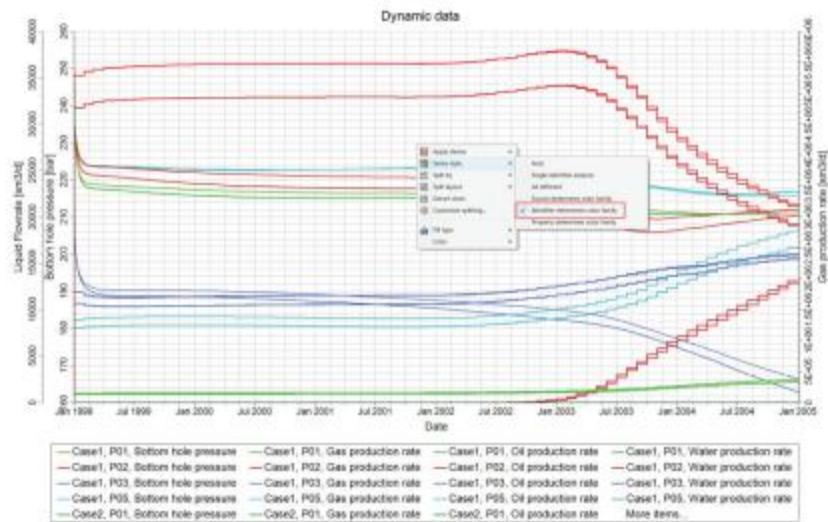


Figure 117. Procedure for changing the color on individual line plots

The **Split by** list in the **Results charting and analysis** dialog box specifies how to present your charts.

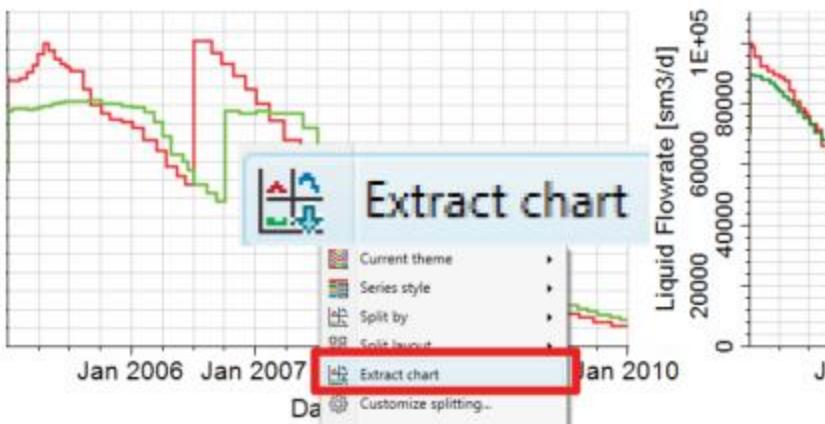


Procedure — Extract individual line plots from a group of thumbnails

Thumbnail is a term used in Petrel to represent a subset of a main chart. It usually is intended to make it easier and faster to look at or manage a group of plotted vectors. You can create multiple thumbnail charts based on different preset Split-by options (such as, Case, Properties, Source) available in the Results charting and analysis process.

After creating a group of thumbnail charts, you can extract individual line plots.

1. In the **Results** pane, choose the study from the **Results charts and analyses** folder to display the required chart.
2. Right-click the thumbnail you want to extract and click **Extract chart**.



The chart is added to the same study folder as the original, with a name based on the original chart, source result case, and data plotted. You can change the name, if necessary.

Define the appearance of the data elements on the charting window

Use the Series style on the **Charting window** to define the color of the data elements in the **Charting window** based on different color families:

- Sources
- Identifiers
- Property

Right-click the graph, click **Series style**, and then select the color family to use.

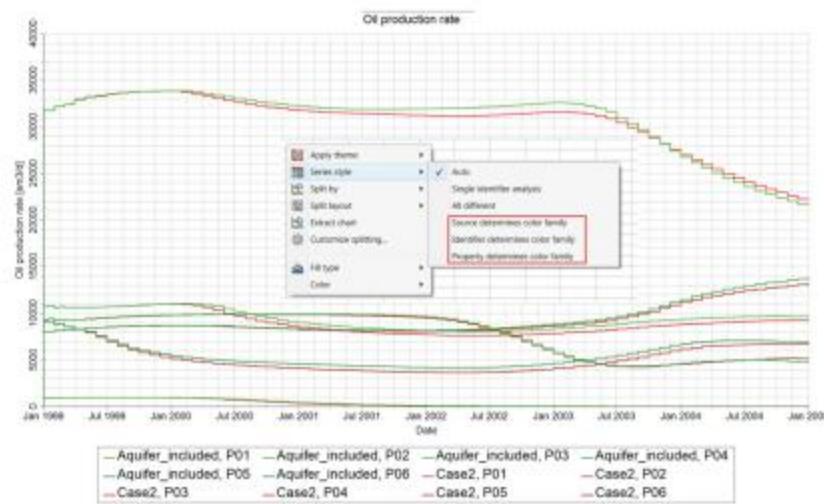


Figure 118. Defining the color of the data elements in the charting window based on different color families

Interactive searching and filtering

You can filter lists of items in a different pane so that you can locate items of interest more quickly. Searches and filters in the **Results charting and analysis** dialog box are not case sensitive.

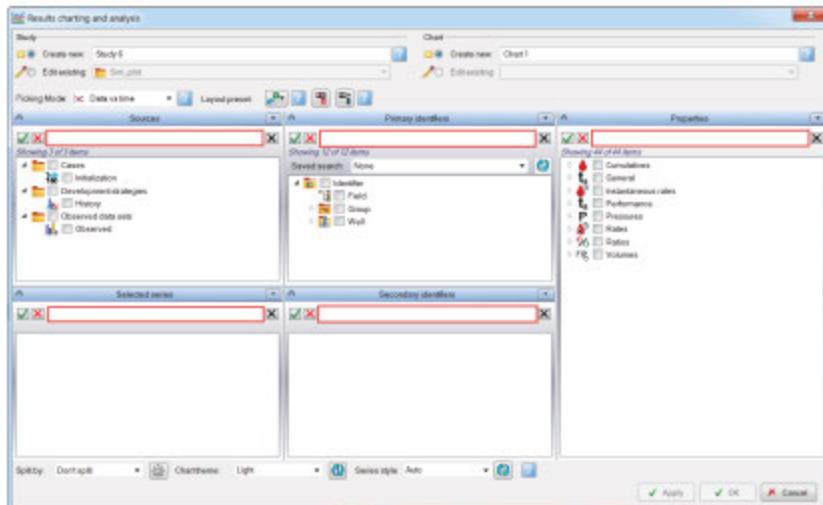


Figure 119. Filter items in the Results charting and analysis dialog box

To select all the visible items, click . To clear all the visible items, click . Visible items reflect an applied filter. To clear the text in the search field, click .

You can filter items using these methods.

- Contained text: Enter the text contained in the name of the item so that only those items appear.
- Wild cards: Filters support the wild cards (*) and (?). The (*) represents any number of characters, and the (?) represents a single character.

For example, the filter AH*THP finds all items that start with AH and end with THP.

- Initials: Filters support finding specific properties using their initials. For example, to find Oil Production Rate, enter OPR in the filter.

It is good practice to apply all filters before selecting the items to output as graphs. After you create a graph, applying more filters affects only the visibility of items in the **Results charting and analysis** dialog box but not the resulting graphs.

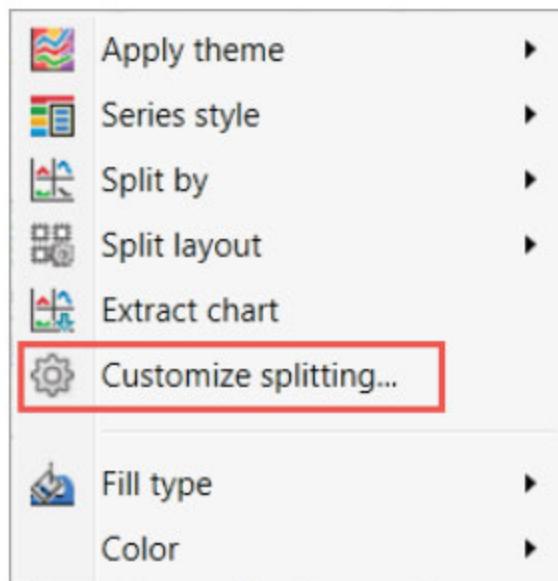


Procedure — Use the Customize splitting command

Custom property grouping allows you to assemble properties into collection sets that appear on separate charts.

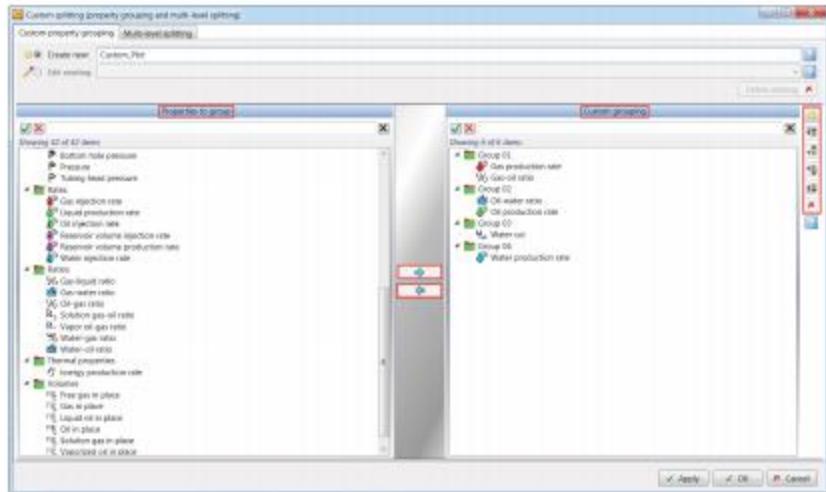
1. To open the **Custom splitting** dialog box, perform one of these actions:

- Click **Customize splitting**  at the bottom of the **Results charting and analysis** dialog box.
- Right-click the active chart window and click **Customize splitting**.



2. To define which properties appear on which charts, drag the properties from the **Properties to group** tree on the left of the **Custom property grouping** tab to the **Custom grouping** tree on the right.

Properties also can be dragged between groups in the **Custom grouping** tree, or into new groups. Each group represents one possible graph in the resulting split.



3. When you have defined a suitable property grouping, click **OK** to apply your changes and close the dialog box.
4. To apply the custom split, perform one of these actions:
 - Select the customized split from the **Split by** list in the **Results charting and analysis** dialog box.
 - Right-click a chart, click **Split by**, and then click the customized split.

Well player

The **Well player** is used to move between identifiers in a chart. It is enabled only in these picking modes:

- Data vs. time
- Data vs. time (additive)
- Scatter by time

This function allows you to move between identifiers when data for one primary identifier (a well) is plotted. Identifiers with no data are excluded.

Suppose that you have five wells (P01, P02, P03, P04, and P05), and your chart shows data for P01. To display data for P02, click **Move forward**; click it again to show data for P03, and so on. The **Well player** buttons are activated only when you select a well.

- **Move forward through identifiers**  : Moves forward one identifier. This button displays the next identifier (if one is available and has data to display). The display wraps around all available identifiers at the same level.
- **Move backward through identifiers**  : Moves back one identifier. This button displays the previous identifier (if one is available and has data to display). The display wraps around all available identifiers at the same level.
- **Jump to specific identifier**  : Shows a list of identifiers. Select an identifier and click **OK** to display it on the chart. To reduce the number of identifiers that you can select, use the **Filter** field. As you enter characters in this field, identifiers that do not match are removed from the list.

Chart theme

To create a new customized theme with the option to modify contents, use a theme template in the **Charting** window. After you apply a theme, you can modify the contents of the graphs by editing line templates.

Predefined theme templates cannot be modified, but you can create a new theme based on a predefined theme and modify the content of the new theme.

Copyright © 2017 Schlumberger. All rights reserved.

Procedure — Create a theme



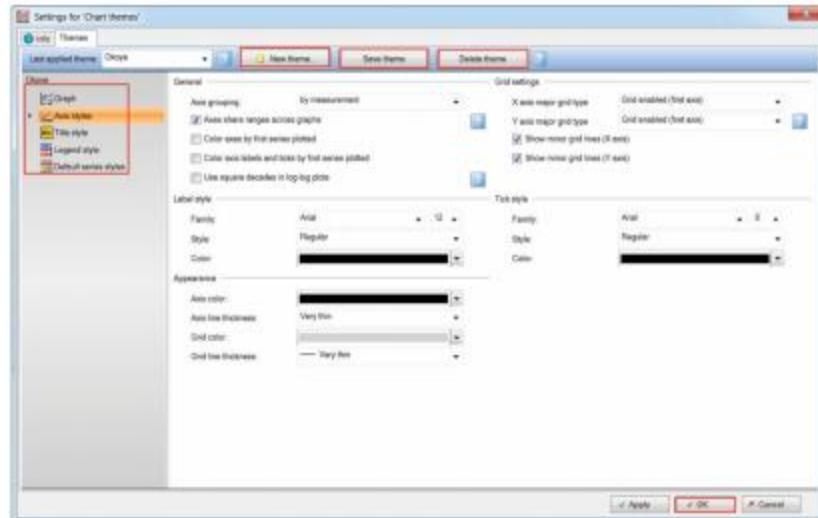
1. On the **Charting** tab, in the **Style** group, click **Chart themes**.



2. In the **Settings** dialog box, on the **Themes** tab, click **New theme**.
3. In the **New style name** dialog box, enter a name for the theme and click **OK**.
4. The left side of the **Settings** dialog box lists the different features that you can customize in this new theme.

Choose the group you want to modify and change the new theme.

- In the **Graph** group, modify general window chart styles.
- In the **Axis styles** group, modify settings that apply to axes. If you create a new theme from the Chart themes settings, the **Axis styles** group contains two subentries: one to apply styles to the X-axis and one to apply styles to the Y-axis. Also, if there are several properties displayed, then for each, a special subentry will be displayed.
- In the **Title style** group, modify general settings for the title of the charts.
- In the **Legend style** group, modify the settings for the **Charting** window legend for all graphs.
- In the **Default series styles** group, set the general styles used to display data on the charts.

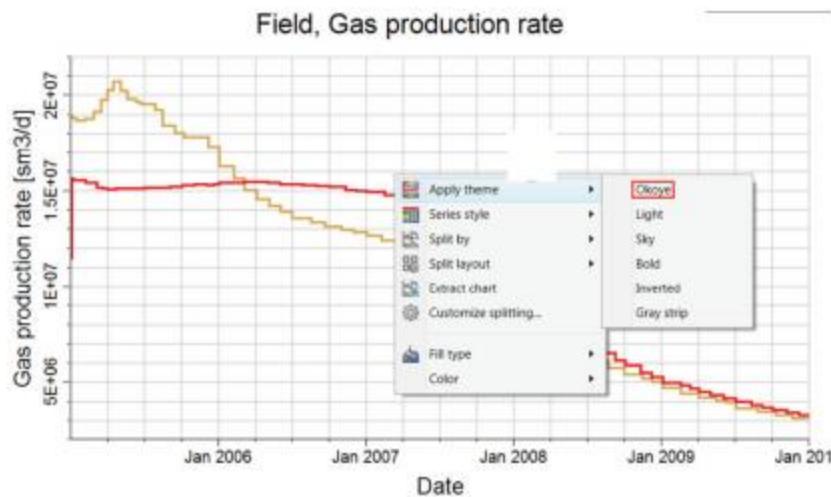


Copyright © 2017 Schäffer-Poeschl. All rights reserved.

5. To save the changes while you are updating the settings, click **Save theme**.
6. After you have modified the settings, click **OK**.

7. After you create a theme, right-click in the **Charting window**, click **Current theme**, and then click your newly created theme.

Your new theme applies to the current chart.



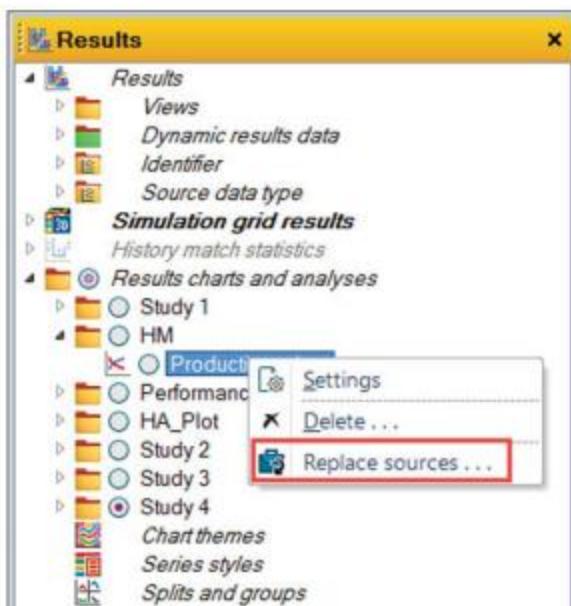
Copyright © 2011 Schlumberger. All rights reserved.



Procedure — Change the selected sources for a chart

You can change the selected source for a single chart or the sources for all of the charts in a study.

1. In the **Results** pane, right-click the chart and click **Replace sources**.



2. To replace the sources of the chart, select another source from the **Target** list.

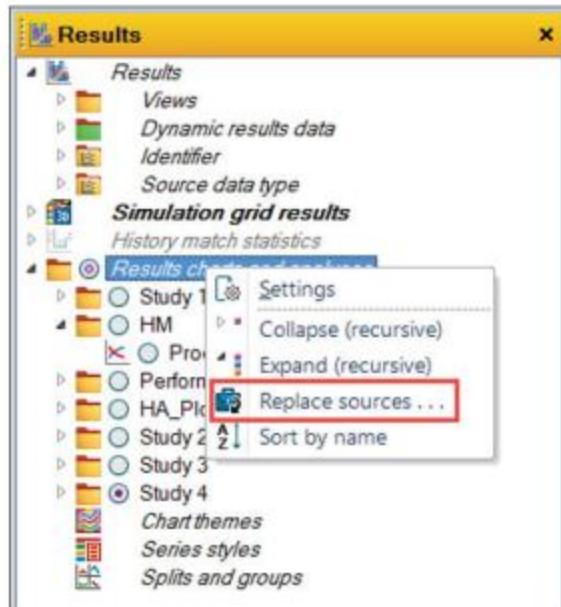


Copyright © 2017 Schlumberger. All rights reserved.



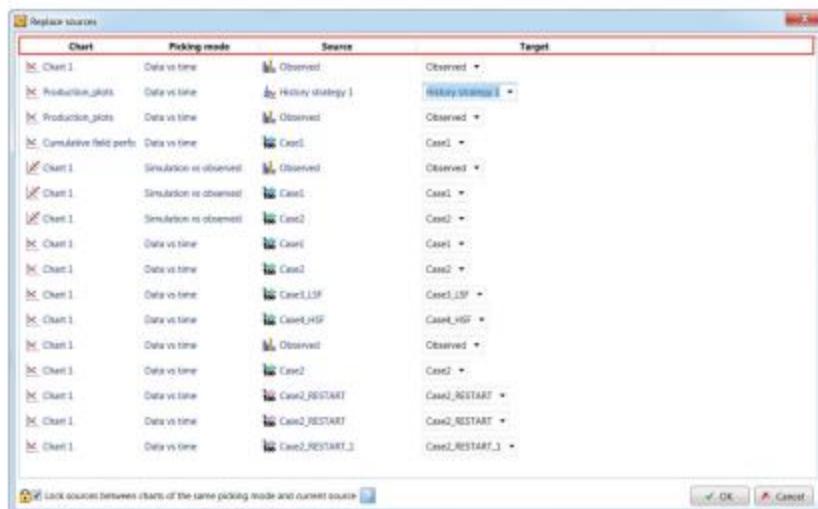
Procedure — Change the selected sources for all charts in a study

1. In the **Results** pane, right-click the Results charts and analyses folder and click **Replace sources**.



Copyright © 2017 Schlumberger. All rights reserved.

2. To replace the sources of all charts, click the **Target** list and then click another source.



With the **Lock sources between charts of the same picking mode and current source** check box selected, changing the target on one row automatically changes the target for any rows that use the named source and plot type.

Charting window export options

After you create a chart, you can export the results as images or export the plotted data to Excel. You also can send the captured image or plotted data to your colleague through e-mail within the Petrel environment. The figure shows the **Charting** window export options.

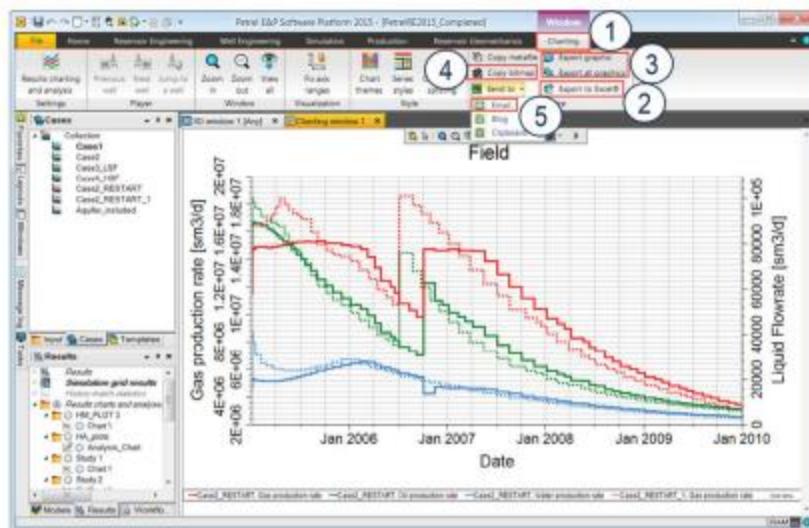


Figure 120. Charting window export options

- 1 To access chart-specific functionality on the **Charting** tab, open the **Charting** window that contains the plotted data.
- 2 To export chart data in table format to Excel, in the **Capture** group, click **Export to Excel**.
- 3 To capture the current page of the displayed charts as a graphic image, click **Export graphic** or **Export all graphics**.
- 4 To copy the current page of images to the clipboard, click **Copy bitmap**.
- 5 Send the captured image to your colleague through e-mail.

Plot window

To access plot windows, on the **Home** tab, in the **Insert** group, click **Window** and then click **Plot window**.

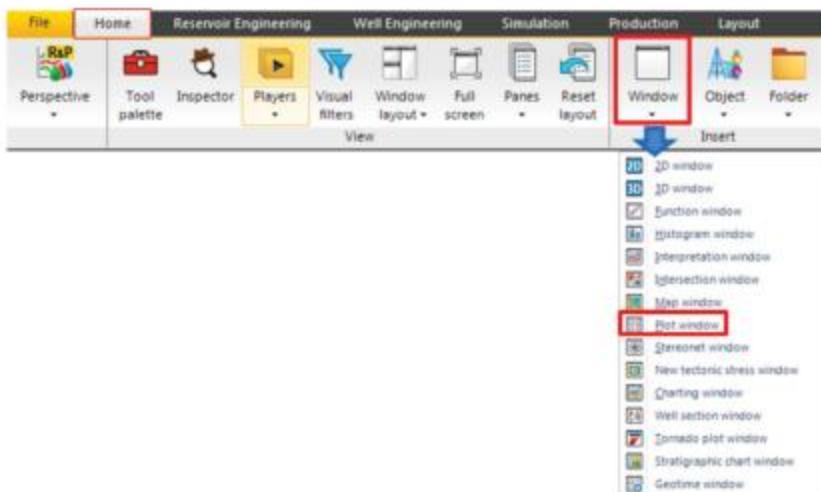


Figure 121. Accessing a Plot window

To add new viewports, click **New object** . You can combine several viewports of the same type or different types as shown in the figure.

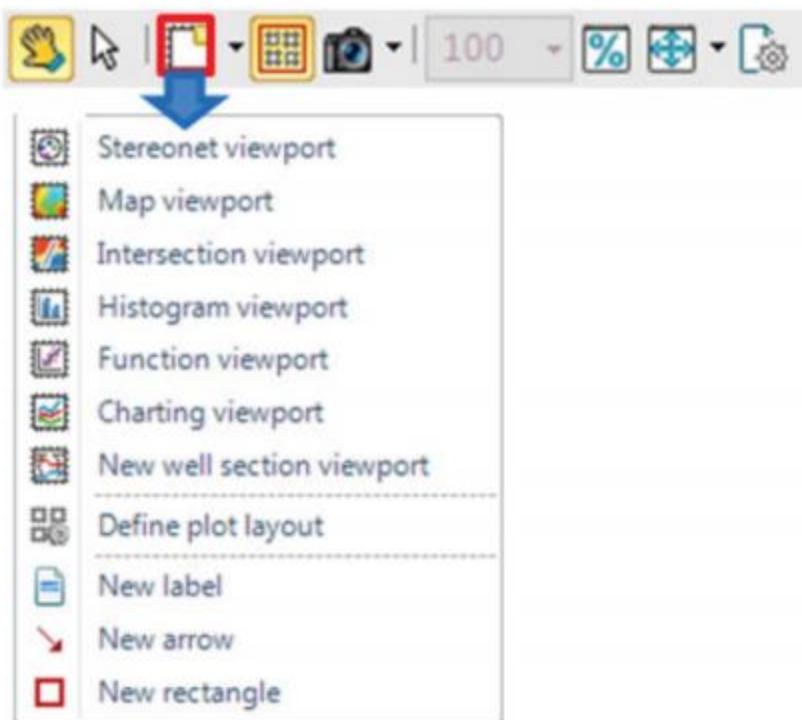


Figure 122. Viewport options

Copyright © 2017 Schlumberger. All rights reserved.

Different types of viewports have different tools available. When you switch between different viewports, the window toolbar changes.

When a viewport is active (red frame around it), the items that you can display in the viewport have a check box in the Petrel **Explorer** panes. If you select the check box, the item appears in the viewport.

Bubble maps: Time varying data

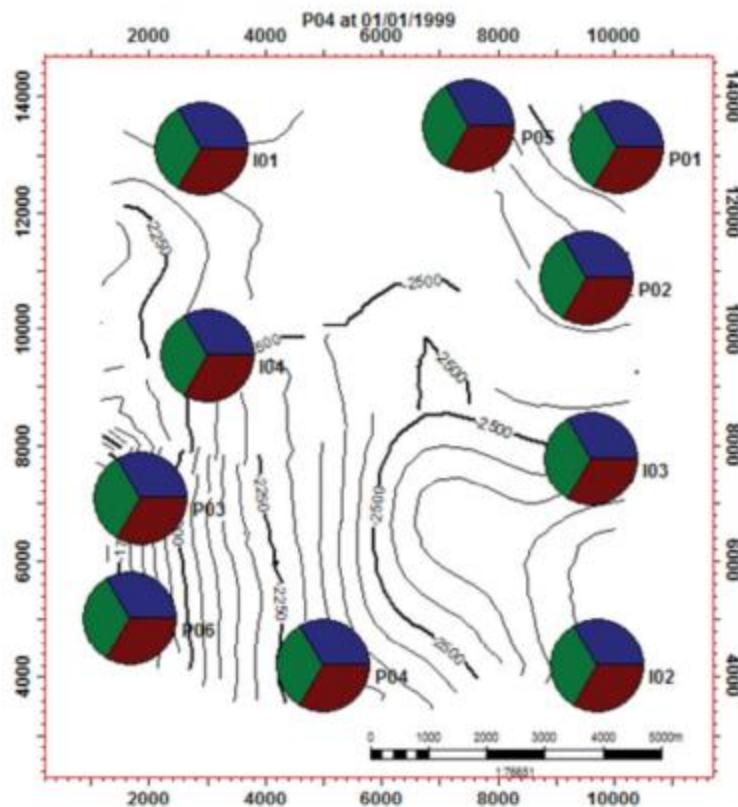
Bubble plots display observed data, development strategies, and simulation data. To go to the timestep that you want to view, use the **Time player**.



Procedure — Display time varying data

This procedure shows you how to display time varying data, such as production rates, as bubble maps.

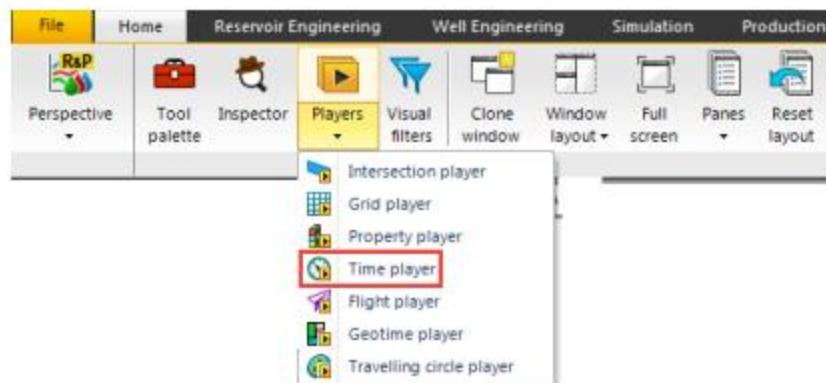
1. Insert a new map window.
2. In the Results folder on the **Results** pane, expand View and select **Dynamic data**.
3. From the Dynamic results data and Well from identifier folders, select **result vectors**.
4. From the **Cases** pane, select the simulation case.



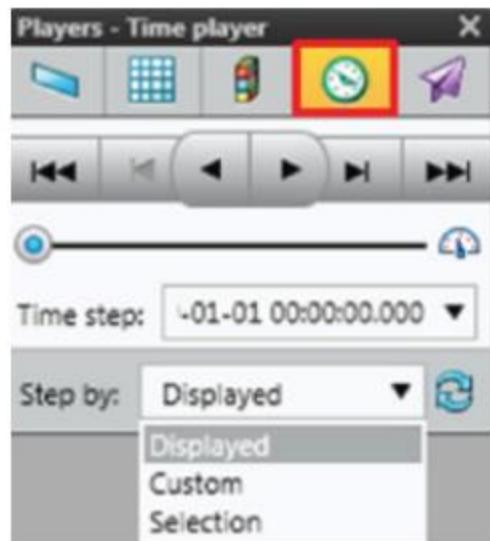
Copyright © 2017 Schlumberger. All rights reserved.

To remove wells with zero rates from visualization, you must select **Suppress irrelevant data** in **Dynamic data** settings.

5. Open the **Time player**. On the **Home** tab, in the **View** group, click **Player**, and then click **Time player**.



6. Play in time with the **Time player** toolbar.



Copyright © 2017 Schlumberger. All rights reserved.

File types to export dynamic results data

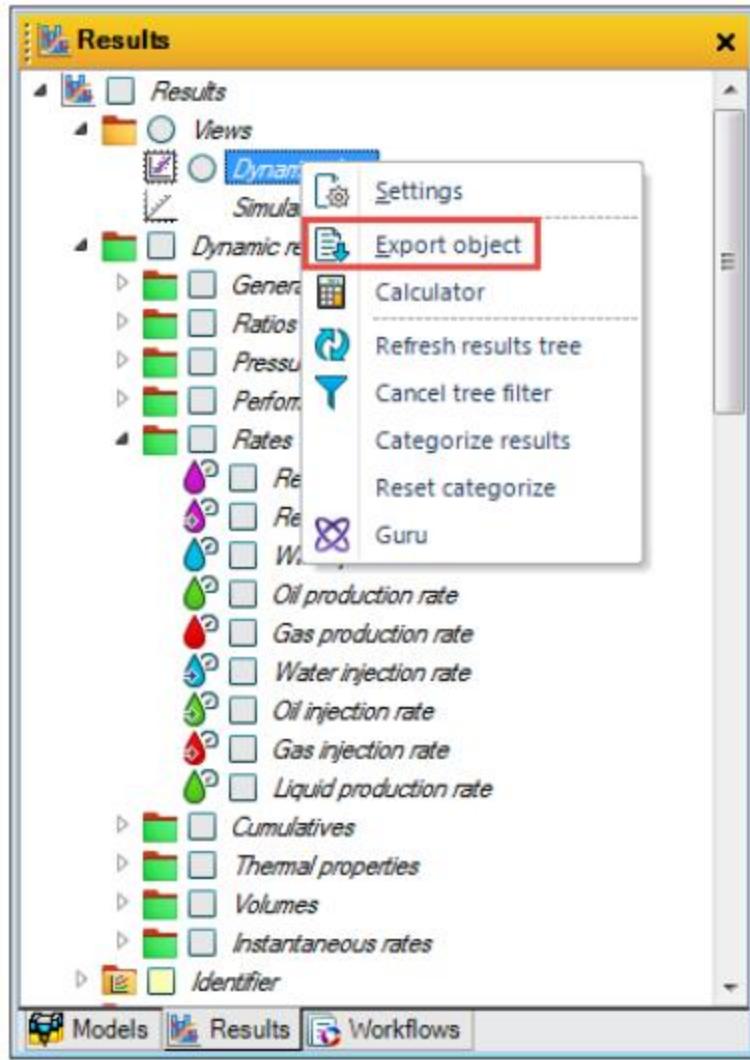
In the **Results** pane, you can export summary results files in two file formats: Well observed data (ASCII) (*.vol) and the Petrel summary data (ASCII) (*.*) formats.



Procedure — Export dynamic result data

This procedure shows you how to export summary data from the **Results** pane in Petrel.

1. Right-click **Dynamic data** or any results vector and click **Export object**.



Copyright © 2017 Schlumberger. All rights reserved.

2. In the **Export as** dialog box, enter a file name.
3. Click **Save**.

The **Export dynamic result data** dialog box opens.

4. Choose the file format from the **Save as type** list.

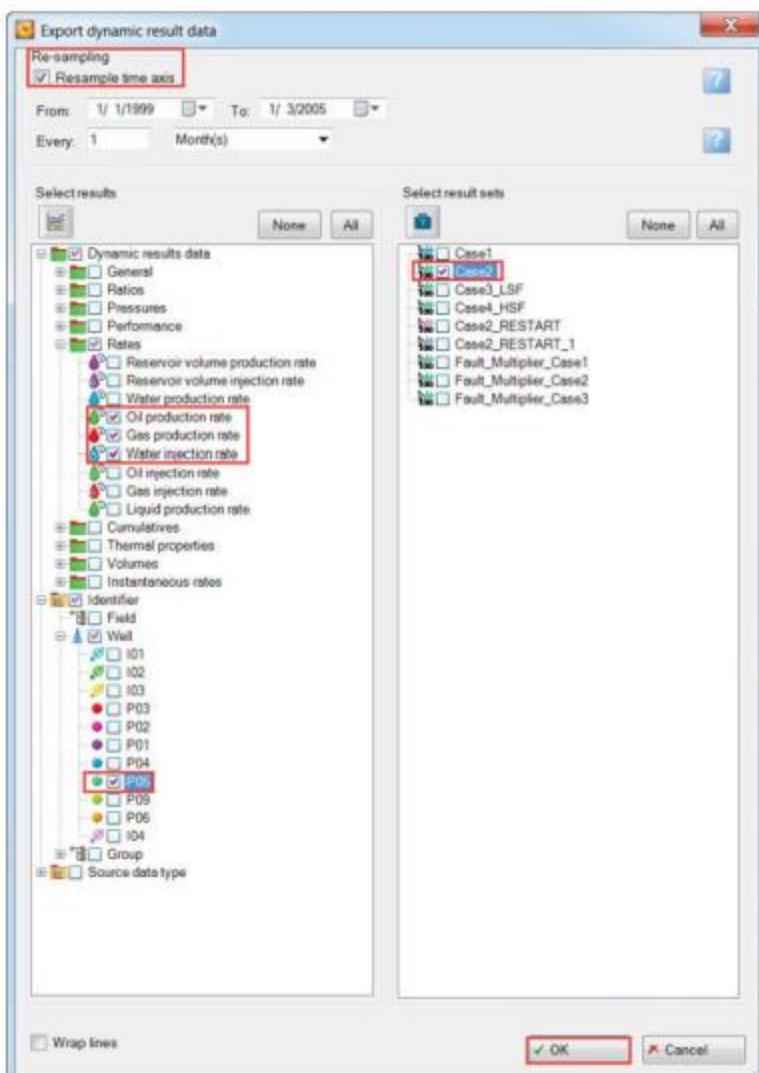
When you export the data in the Petrel summary data file format, the resampling options are enabled.

5. Enter the start and end date as well as the period of output.

The sampling method is predetermined based on the result type. For example, rates are backward sampled, observed data is forward sampled, and cumulatives are linearly sampled.

6. Select the desired vectors.
7. Select the identifier.
8. Select the case.

9. Click OK.



Copyright © 2017 Schlumberger. All rights reserved.

Exercises — Use the Results charting and analysis process

In these exercises, you learn how to include an existing development strategy in the **Define simulation case** dialog box and run a simulation. To plot and analyze your simulation results, you use the **Results charting and analysis** tool. You also learn how to edit and adjust all plot elements to fit your needs.

These exercises also illustrate how to export charts to Excel.

Workflow

1. Include the existing development strategy in the **Define simulation case** dialog box and run a simulation.
2. Create field performance analysis plots using the **Results charting and analysis** dialog box.
3. Create thumbnail plots using the Split view function and extract one of the split charts using the Extract chart option.
4. Export chart data to Excel and chart images to the clipboard.

Data

Use the project named

Simulation_Result_Viewing_exercise.pet in the
Dataset\Projects\Module-5 Simulation Run and
Result viewing folder.

Copyright © 2017 Schlumberger. All rights reserved.

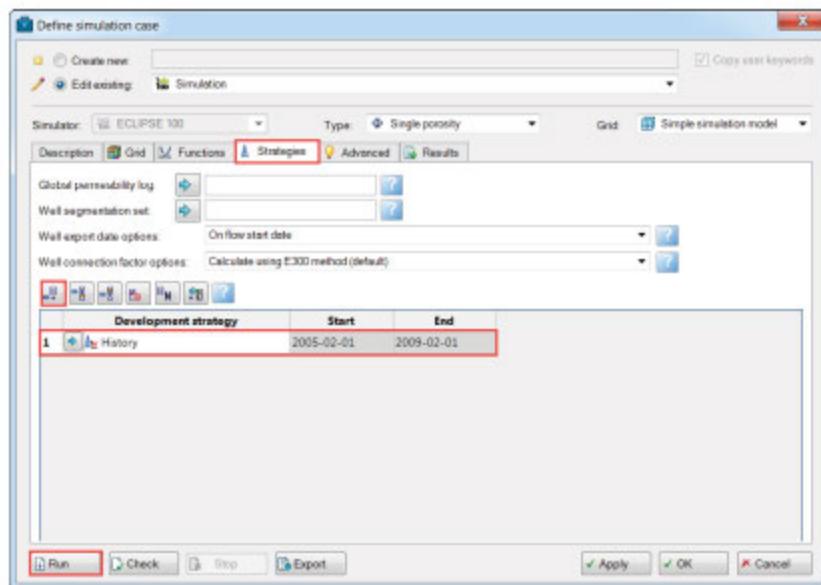


Exercise 1 Include the development strategy in the Define simulation case process and run the simulation

In this exercise, you include an existing development strategy from the Input pane in the Initialization case.

1. On the **Simulation** tab, in the **Simulation** group, click **Define case** to open the **Define simulation case** dialog box.
2. Click **Create new** and name the new case **Simulation**.
3. On the **Strategy** tab, add a data field row. Click **Append item in the table**

4. Expand the **Development strategies** folder in the **Input** pane and select **History**.
5. Insert  the selected History strategy into the development strategy data field and click **Apply**.
6. Save the project and click **Run**.



Copyright © 2017 Schlumberger. All rights reserved.

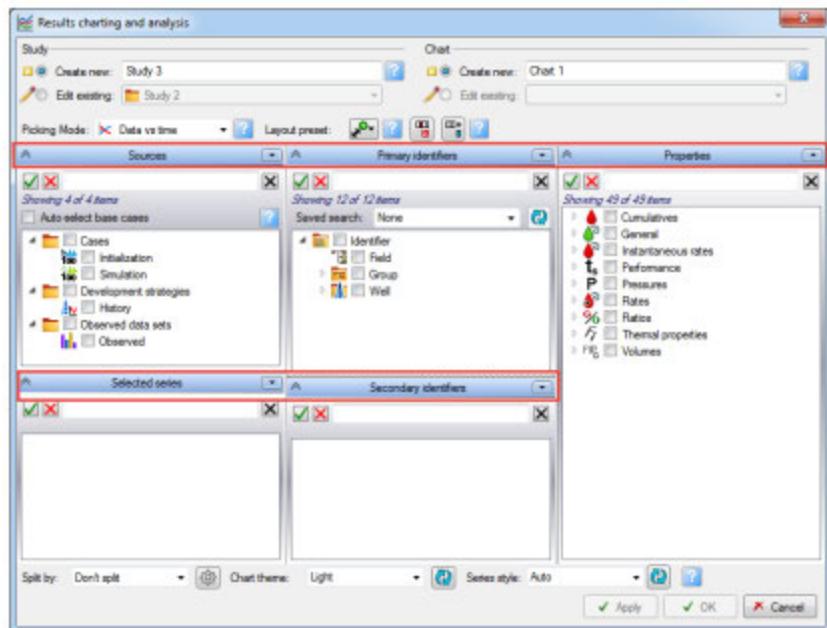
Exercise 2 Plot data versus time for field/well performance analysis



In this exercise, you learn how to use the Results charting and analysis process for field performance analysis.

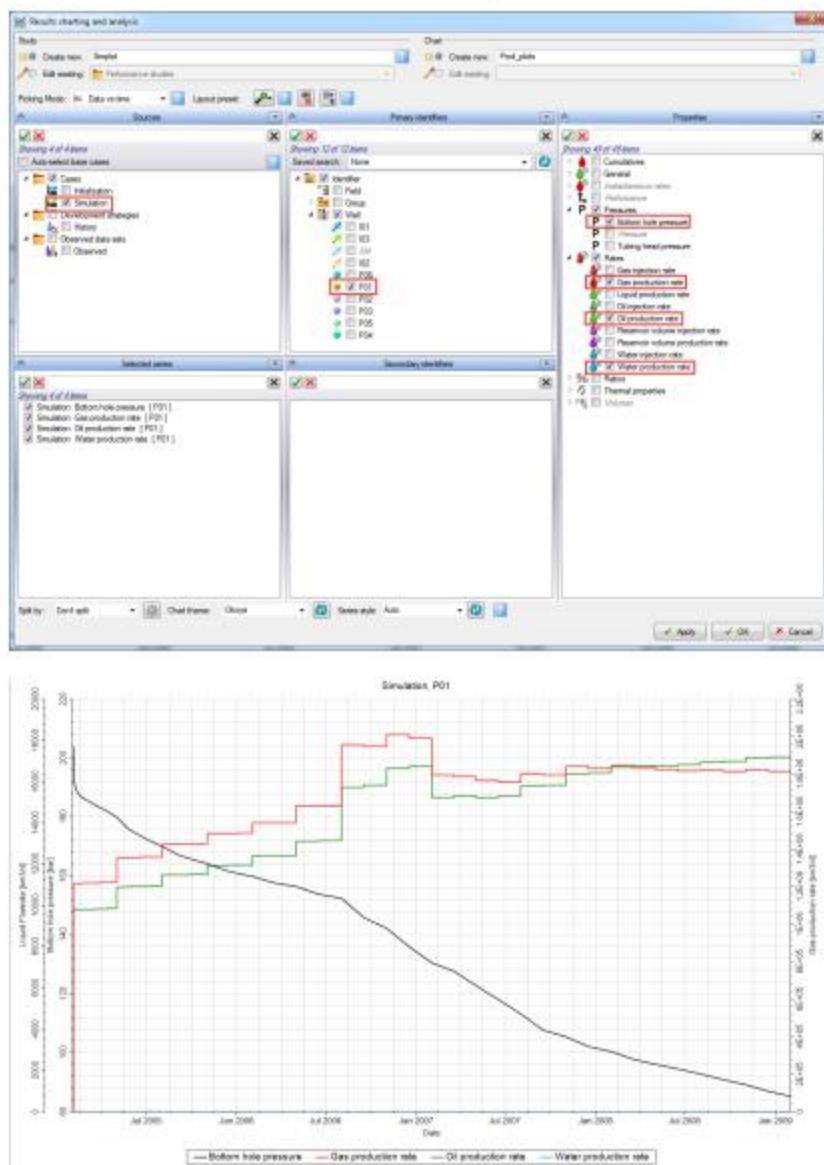
1. On the **Simulation** tab, in the **Summary results** group, click **Results charting** .

2. In the **Results charting and analysis** dialog box, take a few minutes to look at how the simulation results data is organized in the different panes.



3. In the **Study** section, click **Create new** and enter a name for the study. For this exercise, enter Simplot.
4. In the **Chart** section, click **Create new** and enter a name for the chart. For this exercise, enter Prod_plots.
5. In the **Sources** pane, select **Simulation**.
6. In the **Primary identifiers** pane, expand **Well** and select **P01**.
7. In the **Properties** pane, expand **Rates** and select **Oil production rate**, **Water production rate**, and **Gas production rate**.

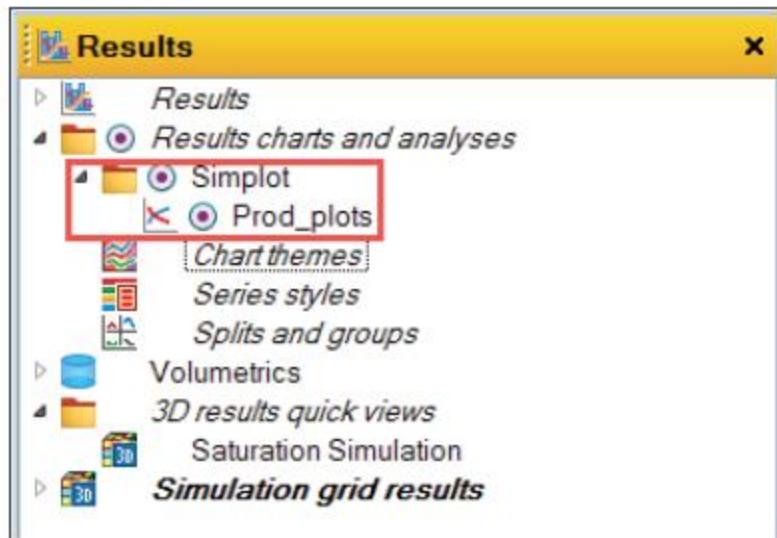
8. Expand **Pressures** and select **Bottom hole pressure**.



Copyright © 2007 Schlumberger. All rights reserved.

9. Confirm that all of your plotted results vectors in the graphs are listed in the **Series** pane.
10. To save the Simplot study and Prod_plot chart, click **OK**. To save the study and add more charts, click **Apply**.

A new Results charts and analyses folder is created in the **Results** pane. This folder contains the Simplot subfolder and the Prod_plots chart.



Copyright © 2017 Schlumberger. All rights reserved.

11. On the **Window** toolbar, click **Move backward through primary identifiers** and **Move forward through primary identifiers** .

Observe how moving from one identifier to the other in either direction changes the well that is displayed in the plot.

12. In the **Results charting and analysis** dialog box, change the **Picking Mode** to Data vs time (additive).
13. Select these check boxes:
 - In the **Sources** pane, select **Simulation**.
 - In the **Primary identifiers** pane, select **P05** in the Well folder and **Oil, Gas**, and **Water** production in the Rate folder.
14. Click **Add** .

The selected series are added to the existing plot created with the Data vs time picking mode.

15. Remove the series you just added. Select them in the **Selected series** pane and click **Remove** .

The selected series are removed from the existing plot created with the Data vs time picking mode.

16. Close the **Results charting and analysis** dialog box.
17. Make the case named **Simulation** active.
18. Quickly plot the field production rate. On the **Simulation** tab, in the **Summary results** group, click **Field plots** and then click **Production rates**.



Exercise 3 Create a chart theme template

This exercise demonstrates how to create a chart theme template.

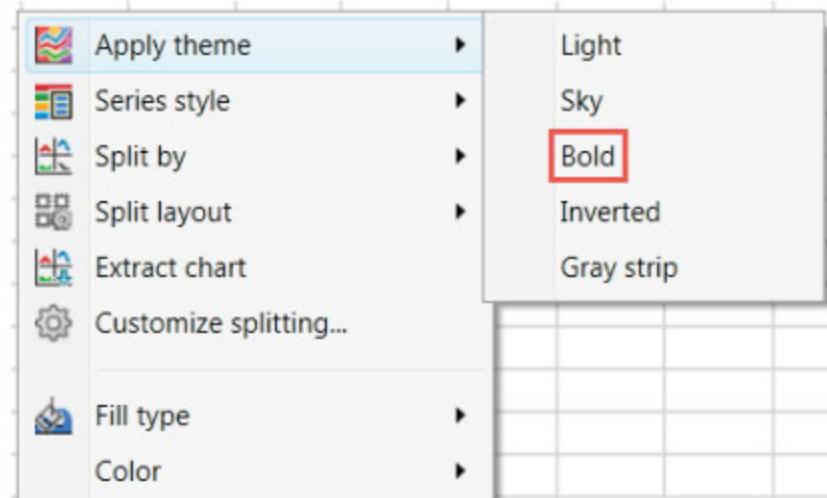
There are five existing predefined chart theme templates, each with a different specific design and layout in the **Result charting and analysis** dialog box:

- Light
- Bold
- Sky
- Inverted
- Gray strip

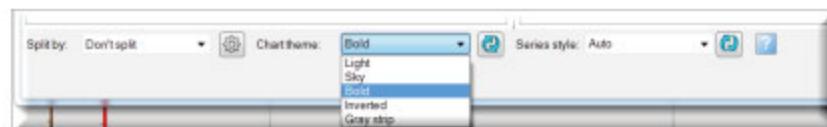
Continue to use the same project from the previous exercise.

1. Right-click the active **Charting** window that you created in the previous exercise, click **Apply theme**, and then click **Bold** or one of the other available templates.

The template changes to a different predefined chart theme.



Alternatively, use the **Chart theme** list at the bottom of the **Results charting and analysis** dialog box.



2. On the **Charting** window tab, in the **Style** group, click **Chart themes**



The **Settings** dialog box for Charting themes opens.

3. In the **Settings for Chart themes** dialog box, click the **Themes** tab and then click **New theme**.
4. In the **New style name** dialog box, enter a name for the theme template and click **OK**.
5. In the **Settings for Chart themes** dialog box, in the gray window, click **Axis style**

6. In the **Label style** section, in the **Family** list, change the font to Times New Roman and the size of the font to 14.
7. Click **Legend style**  and in the **Font style** section, increase the font to 14.
8. Click **Default series styles**  and change Default line to Thin/medium.
9. To save the changes, click **Save theme**. To close the dialog box, click **OK**.
10. Apply the chart theme that you created. Right-click in the active charting window, click **Apply theme**, and then click your theme.
11. Save the project.

Exercise 4 Create a split view

In this exercise, you learn how to split the graphs to show single lines that depend on identifiers that you choose using the Split view functionality.



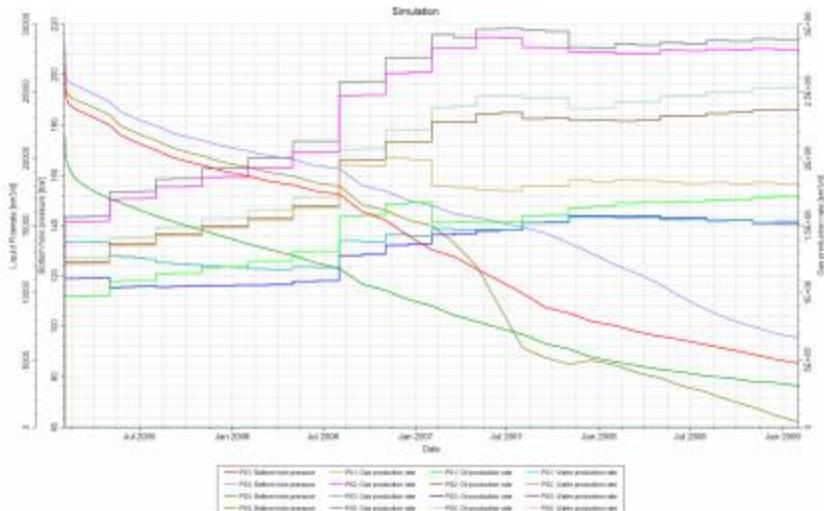
Copyright © 2017 Schlumberger. All rights reserved.

1. From the **Charting** window toolbar, open the **Results charting and analysis** dialog box.



2. In the **Study** and **Chart** sections, click **Create new**.
3. Name the study **Simulated** and the chart **H_plot**.
4. Set the **Picking Mode** to **Data vs time**.
5. In the **Sources** pane, select **Simulation** in the Cases folder.
6. In the **Well** folder in the **Primary identifiers** pane, select **P01, P02, P03, and P05**.

- In the **Properties** pane, in the Rates folder, select **Oil, Gas, and Water** production rates and in the Pressures folder, select **Bottom hole pressure**.

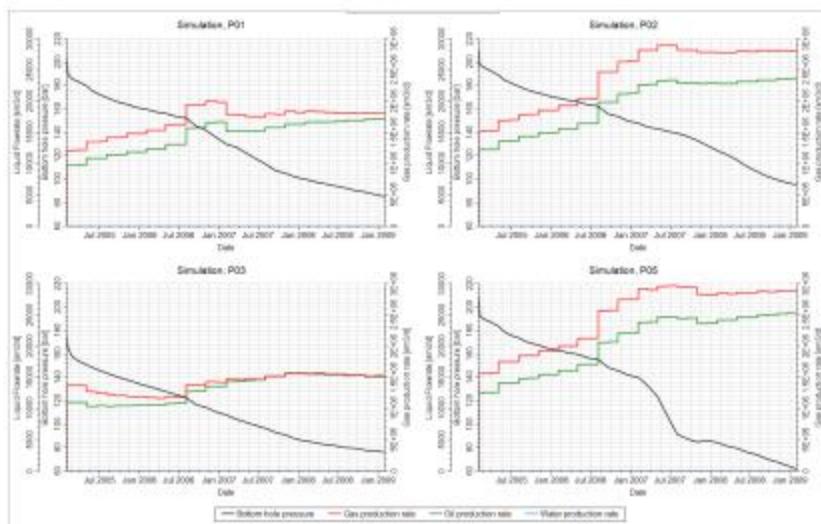


8. Set the split layout. Right-click the active chart, click **Split layout**, and then set the split layout to 2X2.
 9. Right-click the active plotted graph in the **Charting** window again, click **Split by**, and then click **Primary identifier**.

The graph splits into thumbnail plots based on the primary identifiers.

To go back to the previous graph, click **Split by** and then click **Don't split**.

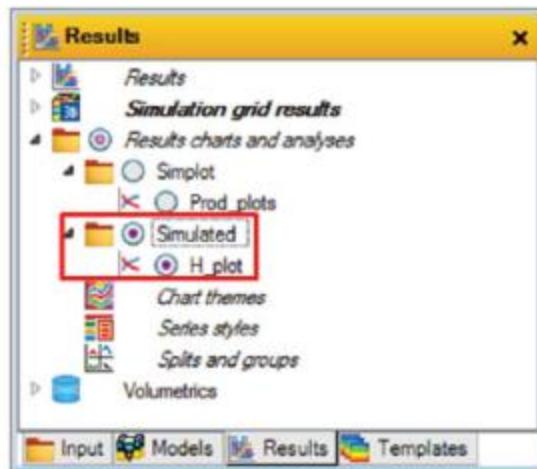
The same commands also are available in the menu in the bottom left portion of the **Results charting and analysis** dialog box.



Copyright © 2017 Schlumberger. All rights reserved.

10. Confirm that all of your plots are listed in the **Selected series** pane. Click **Apply**.

A new folder named **Simulated** and a new **H_plot** chart are created in the **Results charts and analyses** folder in the **Results** pane.

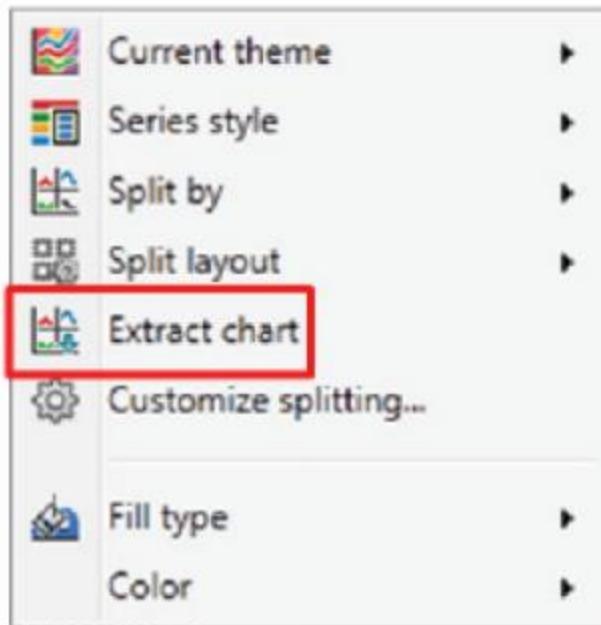




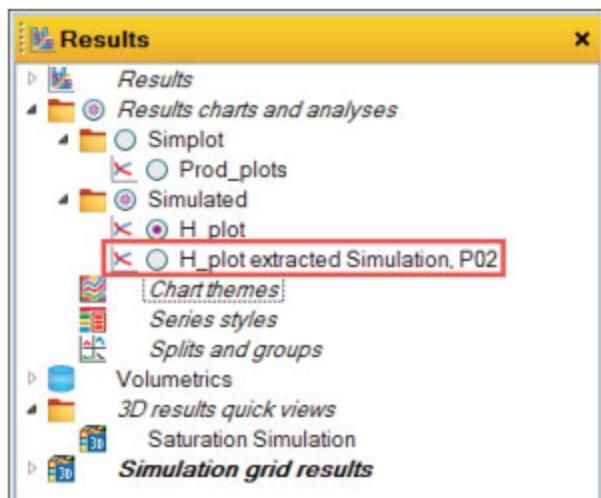
Exercise 5 Create an individual chart from a split view

After creating a group of thumbnail charts, you can extract individual thumbnail images.

1. In the active split view **Charting window** that you created in the previous exercise, right-click the thumbnail **P02** and click **Extract chart**.



The extracted chart named H_plot extracted Simulation, P02 is added to the Simulated folder in the **Results** pane.



Exercise 6 Export chart data to Excel

This exercise demonstrates how to export plotted data in table format to Excel. It also shows how to copy an active chart image to the clipboard.

- In the same active **Charting window** that you created in the previous exercise, on the **Charting window** tab, in the **Capture** group, click

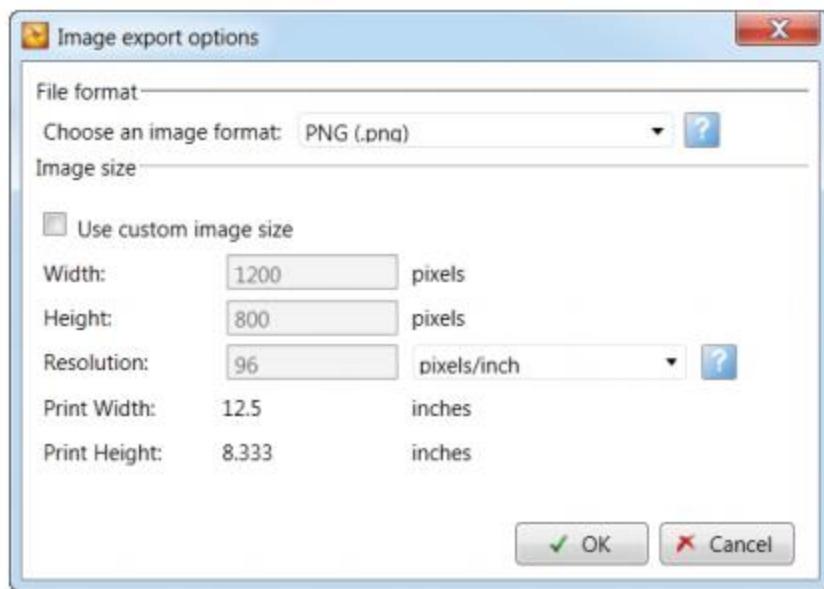


An Excel worksheet automatically opens and displays the exported plot data in table format.

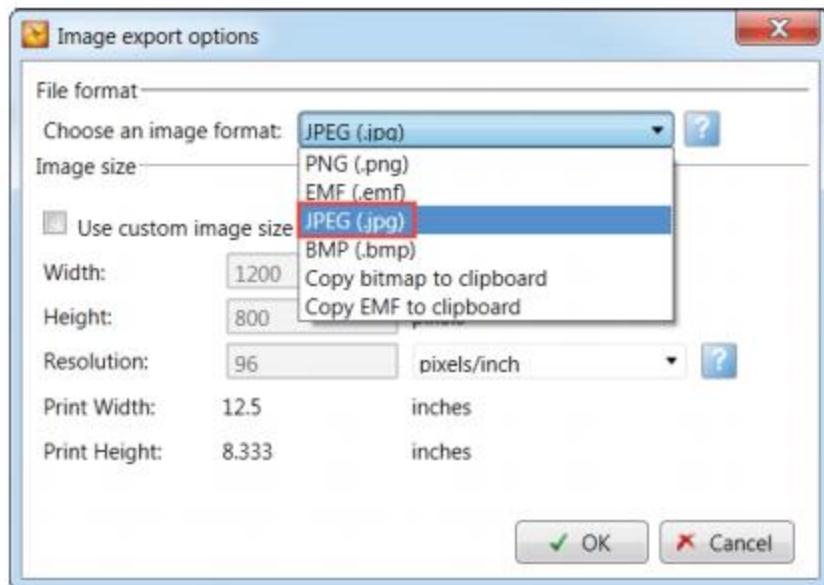
- To capture the active chart in the **Charting** window, click **Export graphic**



The **Image export options** dialog box opens.



3. Choose JPEG as the file format to export and click **OK**.



4. In the **Save as** dialog box, enter a name for the captured image and click **Save**.

To view the exported chart image, go the directory in which you saved the file and open it.

5. Send the active plotted chart through email. Click the **Send to** list and click **Email**.

You must have Microsoft Outlook 2007 or later installed.

Exercise 7 Plot window: Create multiple viewport plots

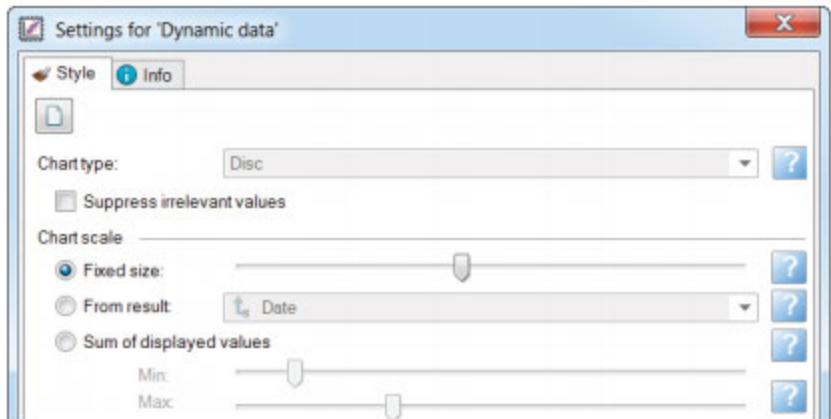
In this exercise, you learn how to combine several viewports in one plot window.



Copyright © 2017 Schlumberger. All rights reserved.

1. On the **Home** tab, in the **Insert** group, click **Window** and then click **Plot window**.
 2. On the **Window** toolbar, click **New object** and then click **Map viewport**.
 3. Drag a new map viewport into the **Plot** window.
 4. In the **Views** folder on the **Results** pane, select the **Dynamic** data subfolder.
 5. In the **Dynamic results** data folder in the **Results** pane, select to view **Oil**, **Gas**, and **Water** production rates for all producers (P01, P02, P03, P05, and P6). Remember to select these wells in the **Identifier** folder.
 6. In the **Window** toolbar, click **View all** to display all of the wells in the Map viewpoint.
 7. From your Simple simulation model in the **Models** pane, select to view a horizon (Top reservoir).
- A bubble map of the production appears in the **Map** window. Play through time if the bubble map does not appear (because the timestep is set to zero).
8. Use the **Time player** to go to the timesteps that you want to view.
 9. Optional: Change the size of the bubbles.
 - a. Double-click any of the bubbles on the map viewpoint or right-click the bubble and click **Settings**.

- b. In the **Settings for Dynamic data** dialog box, in the **Chart scale** section, select **Fixed size** and move the slider to the right to increase the size of the bubble radius. Click **Apply**.



10. On the **Window** toolbar, click **New object** and then click **Charting viewport**.
11. Drag a new Charting viewport into the **Plot** window.
12. In the **Results** pane, expand **Results charts and analyses**, expand **Simulated**, and select the **H_plot extracted Simulation P02** simulation.
13. Click the previously plotted map viewport.

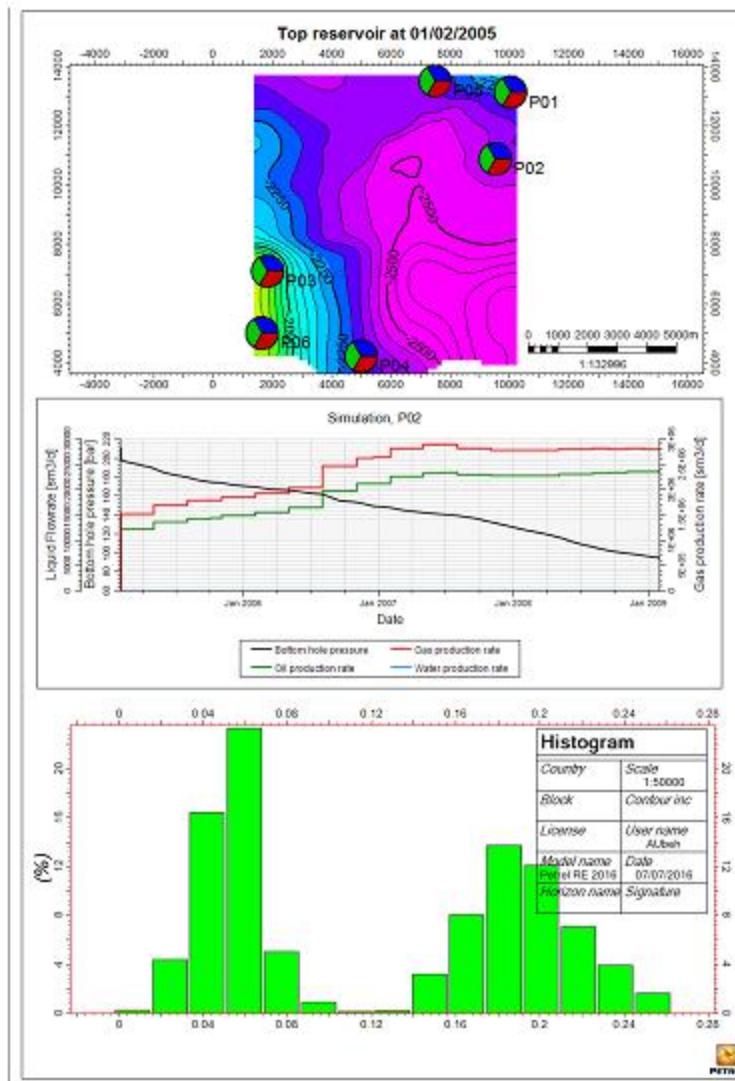
Notice that the type of active viewport determines the buttons that appear in the **Window** toolbar .

14. Click **New object** and then click the **Histogram viewport**.
15. Drag a new histogram viewport into the **Plot** window.
16. In the **Models** pane, expand **Properties** for your Fine grid and select to view the Porosity property.
17. (Optional) Move the viewports to the desired location and resize them in the **Plot** window.

- a. On the **Window** toolbar, click **Select mode** .
- b. Place the pointer on the axis lines in the viewport that you want to move/resize. When the pointer changes to a double-arrow, click and hold the mouse button. Then gradually drag the

viewport to resize it. When four arrows appear, drag the viewport to the new location.

18. Click the map viewport to activate it.
19. On the **Window** toolbar, click **Show/hide display elements**  to select any of the mapping elements (Info box, Auto legend, Header, Scalebar).



Copyright © 2017 Schlumberger. All rights reserved.



Lesson 3 3D simulation results

In this lesson, you are introduced to the powerful Petrel 3D postprocessing tools used to analyze and quality check 3D simulation grid results.

3D viewing

After the simulation runs, Petrel loads the results. A Simulation grid results folder is added to the **Results** pane.

This folder contains three subfolders:

- Composite results
- Static
- Dynamic

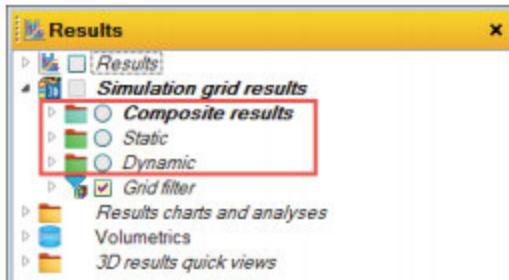


Figure 123. Subfolders in the Simulation grid results folder

Copyright © 2017 Schlumberger. All rights reserved.

The Composite results folder stores properties that have been created from multiple 3D results. These properties include ternary properties (saturation), vectors, and tensors.

The Static folder stores static input data from the simulators (INIT file).

The Dynamic folder contains data from the restart file. This data typically changes with time.



NOTE: You can convert the simulation grid results properties to grid properties and store them in the Properties folder in the **Models** pane.

Converting simulation grid results properties allows you to access more property operations. For example, you can create saturation and pressure properties that can be used as input in the **Define simulation case** dialog box when initializing with the enumeration option.

You access the 3D simulation results either by activating the 3D simulation grid property in the **Dynamic** subfolder of the Simulation grid results folder to display on **3D window** or you also can access



3D simulation results by clicking **3D results presets** on the **Simulation** tab in the **3D results** group. The **3D results presets** tool expedites the visualization and analysis of 3D simulation results. It also allows you to create quick plots of 3D simulation grid results, such as saturation and pressure, for the selected case in a **3D window**.

You can display other 3D simulation results. In the **Results** pane, select the check box for the **3D simulation grid result** and in the **Cases** pane, select the check box for the **simulation case**. If the property is time-dependent, you can use the **Time player** to animate the display through time.

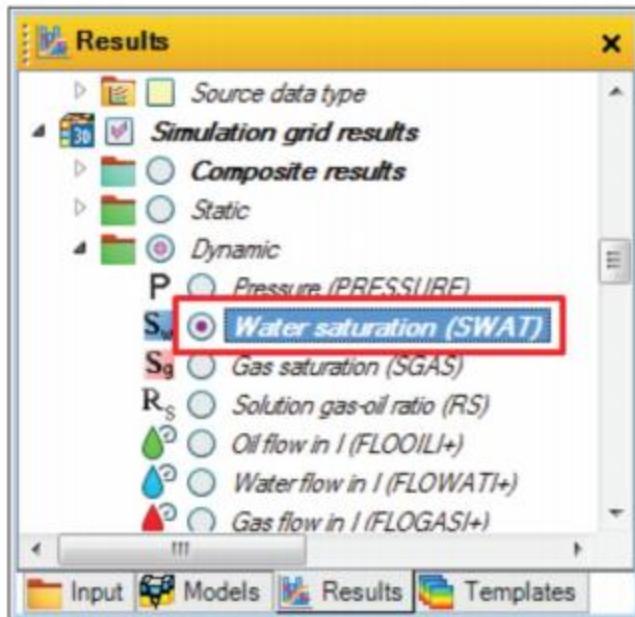
It is important to emphasize that you can manage the simulation results output in the Define simulation case process by customizing the output that you require.

Procedure — Display 3D results



1. Open a **3D window**.
2. In the **Cases** pane, select a case.

3. In the Simulation grid results folder in the **Results** pane, select a simulated property.



4. To play through time, use the **Time player** from the **Players** list or the window mini toolbar.

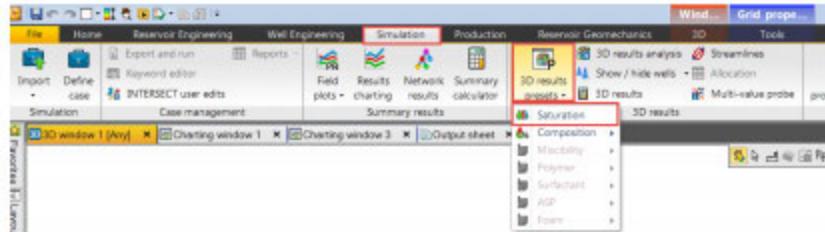


Copyright © 2017 Schlumberger. All rights reserved.

Procedure — Use presets to view 3D simulation results

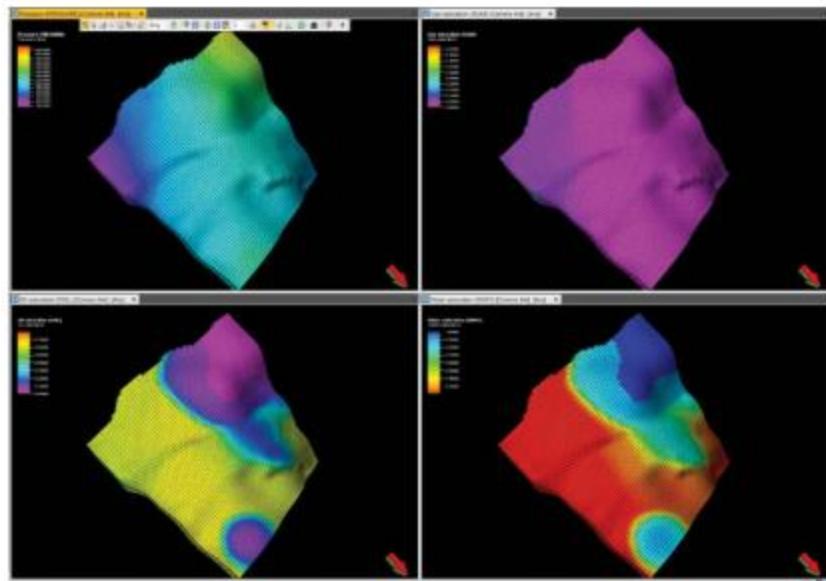


1. In the **Cases** pane, select the case that contains the results that you want to view.
2. On the **Simulation** tab, in the **3D results** group, click **3D results presets**, and then click the required preset that you want, for example, **Saturation**.



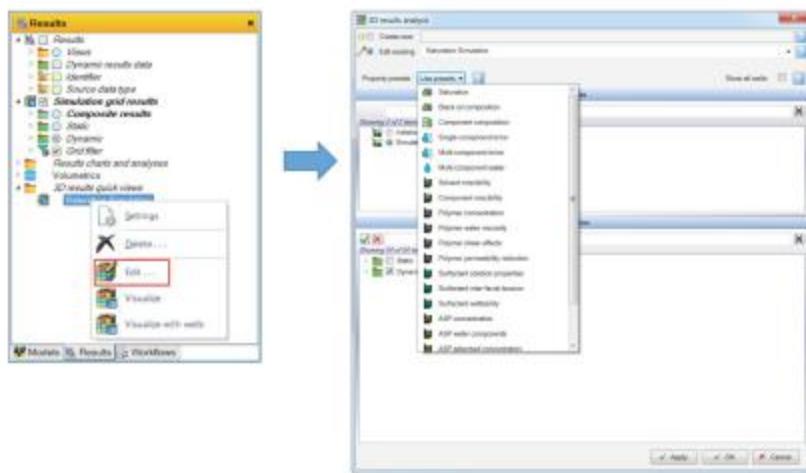
When you select a preset, all existing windows in the project are hidden and replaced with a tiled camera-linked **3D window**. This window displays the 3D results with legends and all properties scaled to their data range. The saturation preset selection shows 3D simulation results for pressure, oil, gas, water, and component saturation for the selected case.

The preset view is added to the 3D results quick views folder in the **Results** pane.



Copyright © 2017 Schlumberger. All rights reserved.

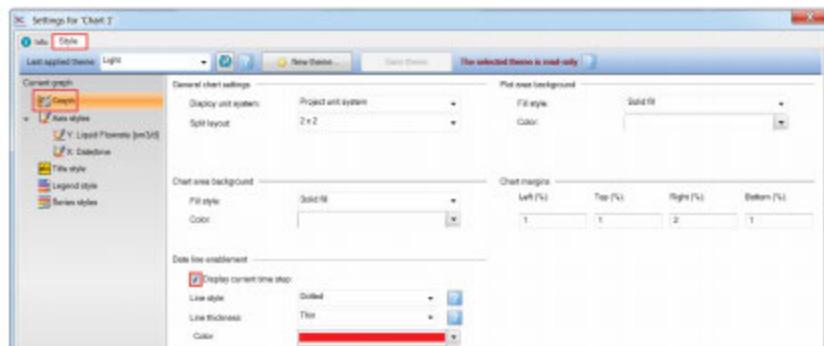
- To change the selections for a preset view for a particular case, right-click the preset view in the **3D results quick views** folder in the **Results** pane, and then click **Edit** to access the **3D results analysis** dialog box.



Procedure — View current timestep as a vertical line in the Charting window



1. In the **Cases** pane, select the case that contains results that you want to view.
2. On the **Simulation** tab, in the **3D results** group, click **3D results presets** and then click the required preset that you want, for example, **Saturation**.
3. Plot any simulation vector versus time using the **Results charting analysis** tool.
4. Display the current timestep as a vertical line in the **Charting** window. In the chart **Settings** dialog box, click the **Style** tab. In the **Graph** section, select **Display current time step**.



Copyright © 2017 Schlumberger. All rights reserved.

5. Click in any of the **3D windows**, open the **Time player**, and select the current date and time. You also can play through simulation time using the **Time player**.

Procedure — Link camera position



In **3D windows** you can synchronize the camera position, so that rotating and scanning your model in the active window will have the same effect in the other linked windows. This link is useful for plot windows as well as when your main display is tiled.

1. Click the first **3D window** in your series to make it active, then on the **Window** toolbar click **Link this camera** . The icon background is orange when turned on.

2. Activate the next **3D window** in your series and click  **Link this camera** in that window. Repeat as necessary.
3. Optional: Enable camera linking in all **3D windows**. Activate any **3D window**, click **Link camera**  and then click **Link all visible cameras** .
4. To disable linking in the active window, click  to turn background white (off), or click the **Link camera** button and then click  **Unlink all visible cameras**.

When windows are linked, any movement or rotation in one window will be mirrored in the linked windows.

Filters in Petrel

You can use filters in Petrel both for visualization and calculations.

There are two basic types of filters in Petrel:

- a simple on/off type, where an element is either turned on or off.
- more advanced filters, such as property filters, that are based on value ranges that you define

Check boxes act as the simple on/off filters in Petrel. There are three types of check boxes in Petrel:

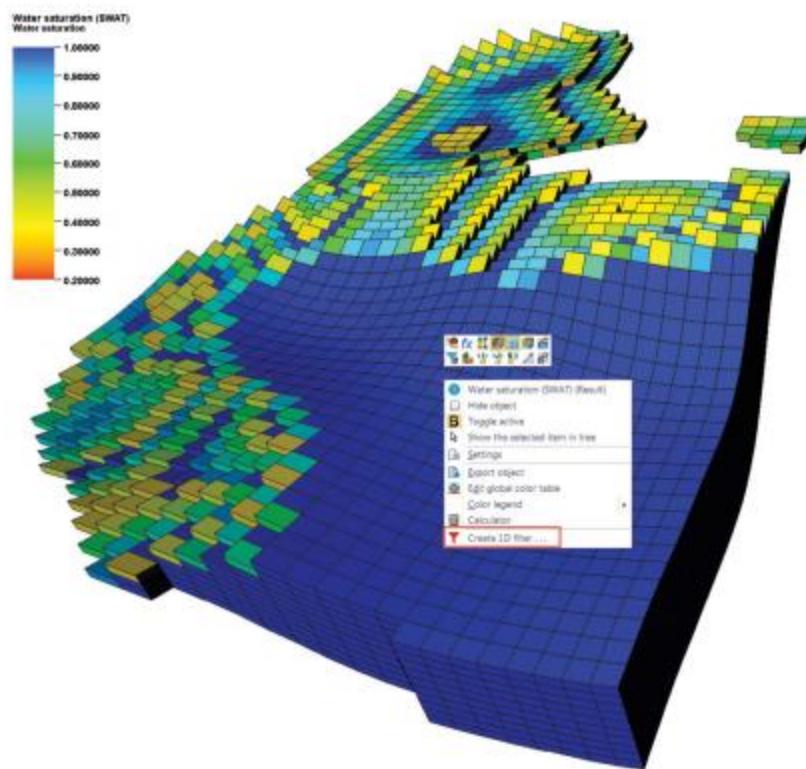
- **White** check boxes: Used only to switch a single object on or off
- **Yellow** check boxes: Used to filter the display of part of an object according to its structure (Segment, Zone, and Fault filters)
- **Blue** check boxes: Used to enable or disable visualization of an element in the active intersection plane

Procedure — Use 1D filters for 3D viewing



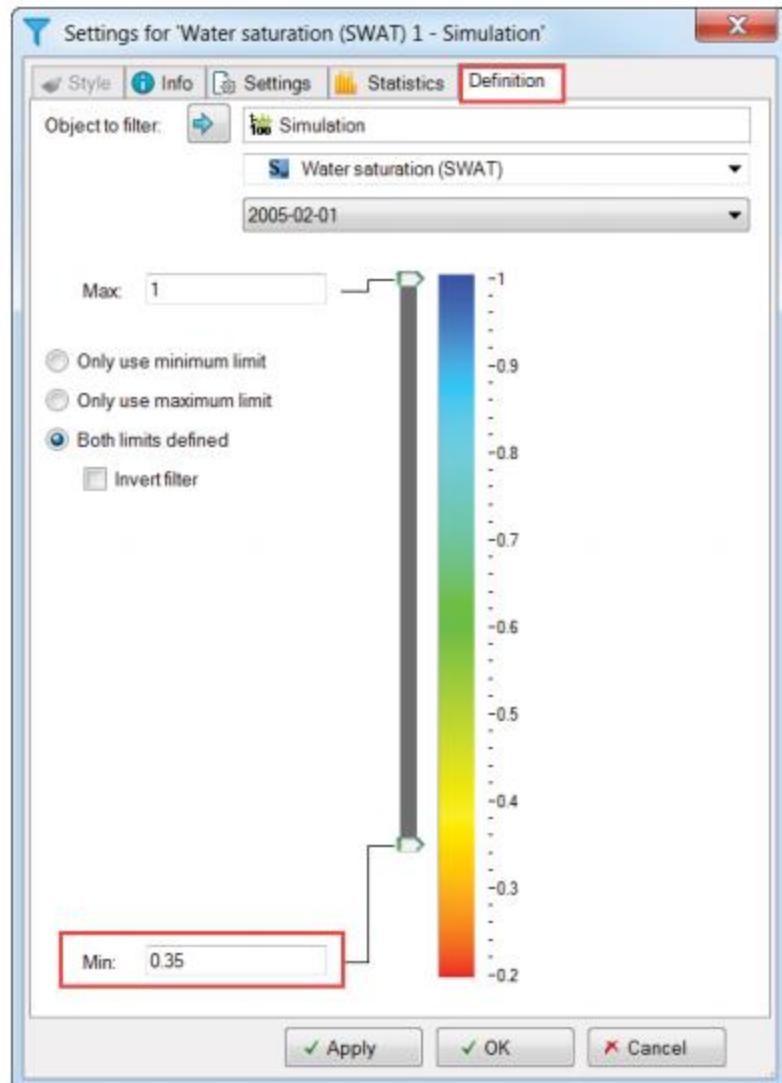
Use a 1D filter to filter a range of cells displayed for any property in the case.

1. Right-click a property in the Simulation grid results folder and click **Create 1D filter**.



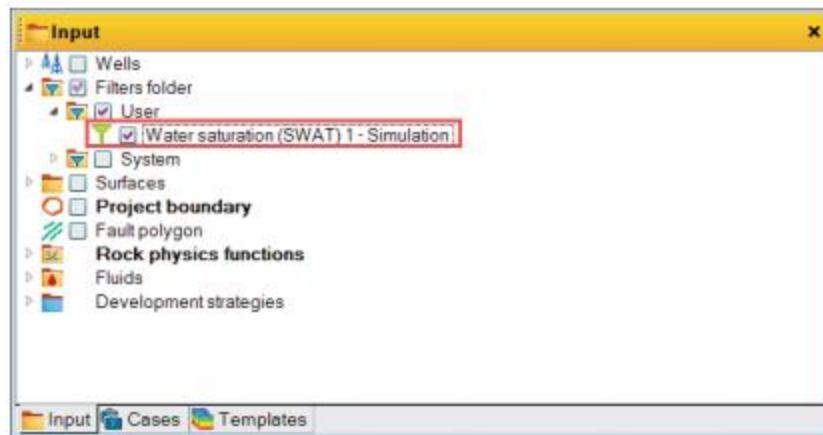
2. In the **Settings** dialog box, move the sliders to select the range of values you want to filter out.

You must specify the simulation case to which the property is referring and the report step to which the filter is applied.

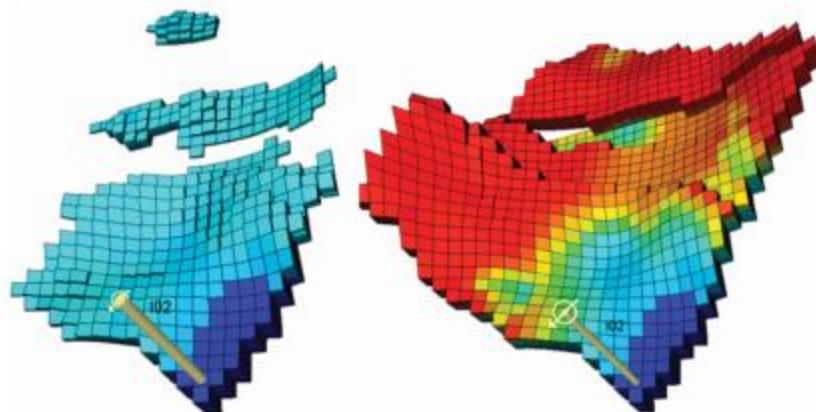


The filtered property is stored in the **Input** pane in the **Filters** folder.

3. Expand **Filters** folder and select the check box next to the filtered property to see the filter effect. Clear the check box to reset.



This figure shows the model with the 1D filter applied on the left and the model without the filter applied on the right.



Copyright © 2017 Schlumberger. All rights reserved.

Interactive viewing of the filter results in 3D, Function, and Histogram windows

You can create filters interactively from the **Function** and **Histogram** windows.

In a **Histogram** window, plot the distribution of a grid property, such as porosity or permeability, by selecting the check box next to the property.

To specify the filter, click  and drag a section of the distribution along the X-axis. The new filter appears in the Input pane. You can apply it to other properties and windows. In the figure the **Histogram** window is used to make a filter for high values of porosity. This filter is applied in a **3D window** to show oil saturation in high porosity cells.

It also is possible to create a filter from a **Function** window. For example, to make a crossplot of permeability against porosity in the **Function** window, select the check boxes for **permeability** and **porosity** in the **Models** pane and click **Create 1D range filters on the X-axis** to select a cloud of data points from the crossplot of the two properties.

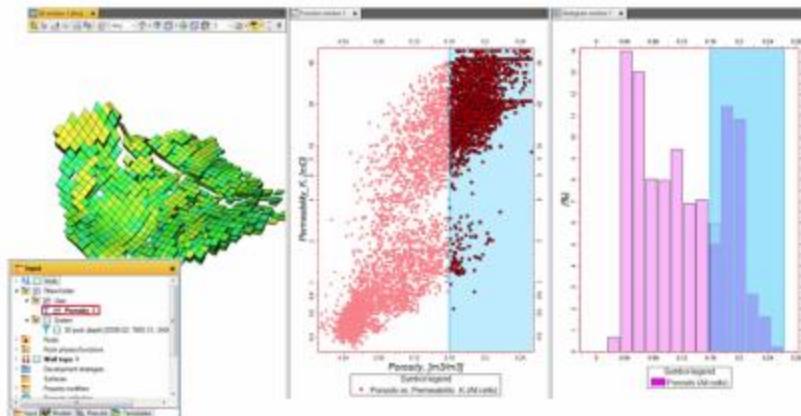


Figure 124. Interactively viewing filter results in 3D, Function, and Histogram windows

Freehand draw filter

The Freehand draw filter is one of the filters accessible in the **Function** window toolbar. The Freehand draw filter option allows you to select a cloud of data points with a closed polygon of any shape. Select the Freehand draw option from the available filters in the **Function** window toolbar and draw any polygon shape to specify the area to filter.

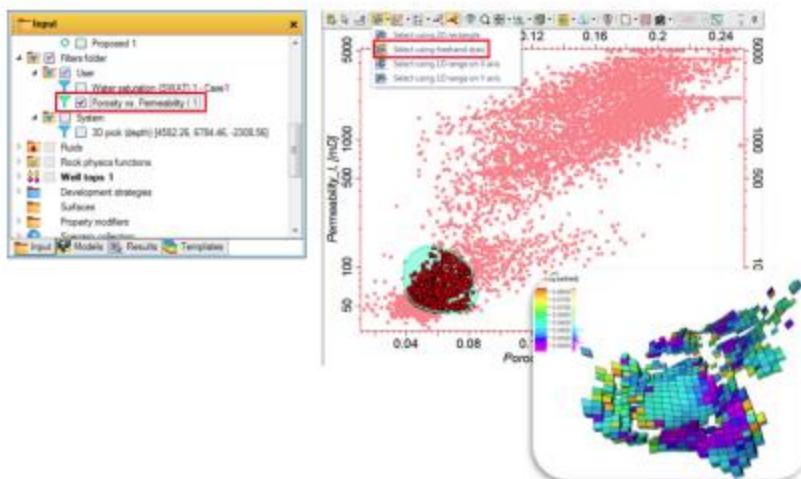


Figure 125. Freehand draw filter functionality

Copyright © 2017 Schlumberger. All rights reserved.

Filter by combining 1D filter range on X axis and 1D filter range on Y

The figure illustrates using the 1D filter in both the X and Y axes for data quality checking and analysis. It also is possible to view the results of the filtered property interactively in the **3D window**. These filter options are similar to the filters described previously.

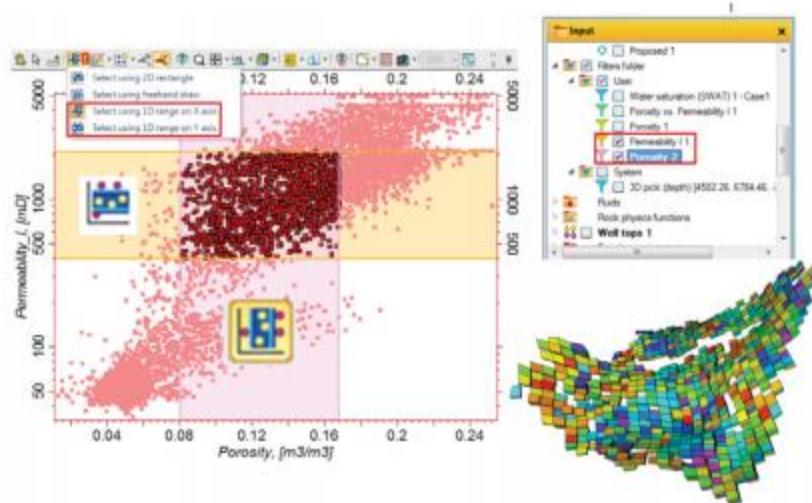


Figure 126. Using 1D filter in X and Y axes

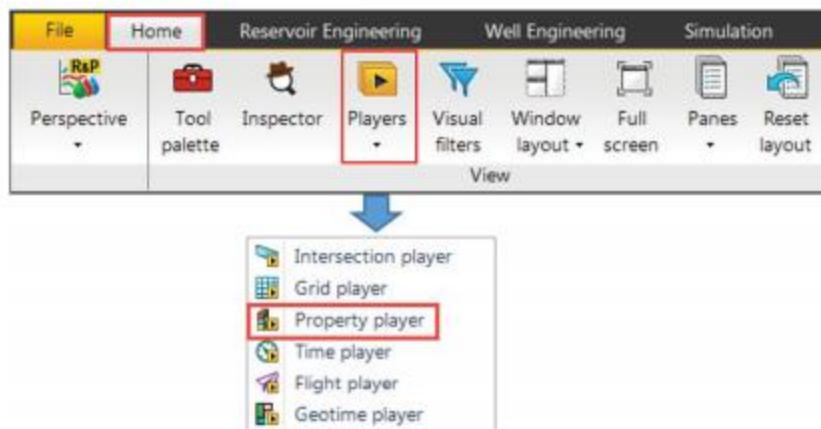
Copyright © 2017 Schlumberger. All rights reserved.



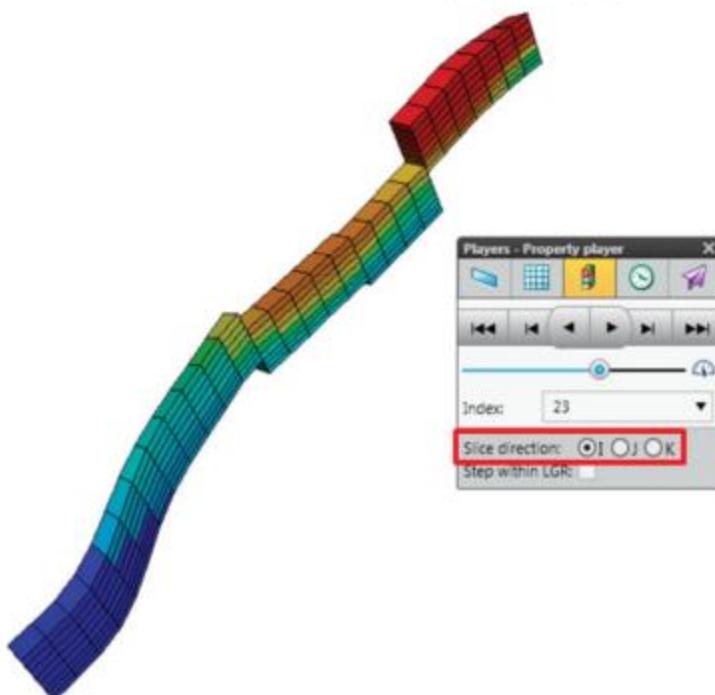
Procedure — View property modeling results

The **Property player** helps you view your property modeling results by moving through each layer, row, or column as a continuous animation or step by step. Use it to quickly browse through your 3D grid simulation results.

1. View all cells with the same I, J, or K index. On the **Home** tab, in the **View** group, click **Players**, and then click **Property player**.



2. To play through the model, use the **Property filter player**.



Copyright © 2017 Schlumberger. All rights reserved.

Filter tools also are accessible on the mini toolbar. Right-click any active displayed grid property in the **3D window** to display the mini toolbar.



Procedure — Create an index filter

The effect of using the Index filter is similar to creating a fence diagram. The filter is based on the indexing of the cells in the 3D grid.

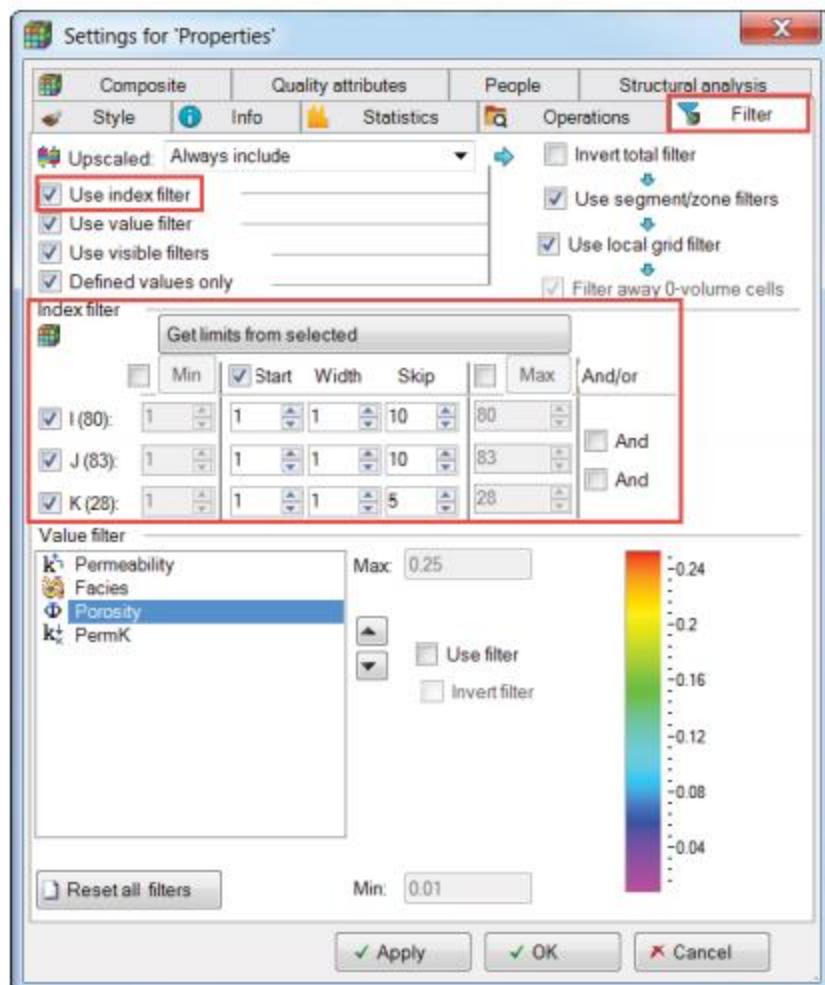
1. Open the **Settings** dialog box for the **Properties** folder in the **Models** pane.
Alternatively, right-click the displayed property in the 3D window and click **Show property filter** on the window mini toolbar.



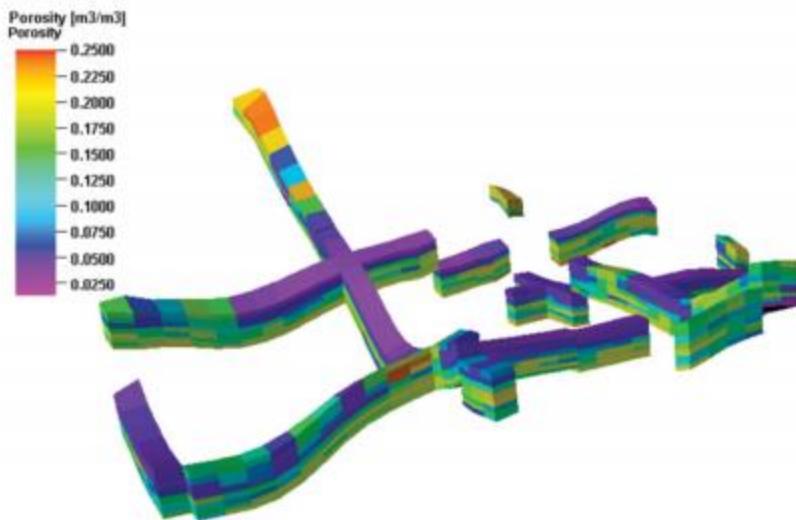
2. On the **Filter** tab in the **Settings** dialog box, select the **Use index filter** check box and specify how to filter in each direction (I, J, and K).

The Index filter is a combination of three filters, one for each main direction of the grid: I, J, and K.

The **I**, **J**, and **K** check boxes help you to select the ranges of indexes in each direction to display.



Copyright © 2017 Schlumberger. All rights reserved.



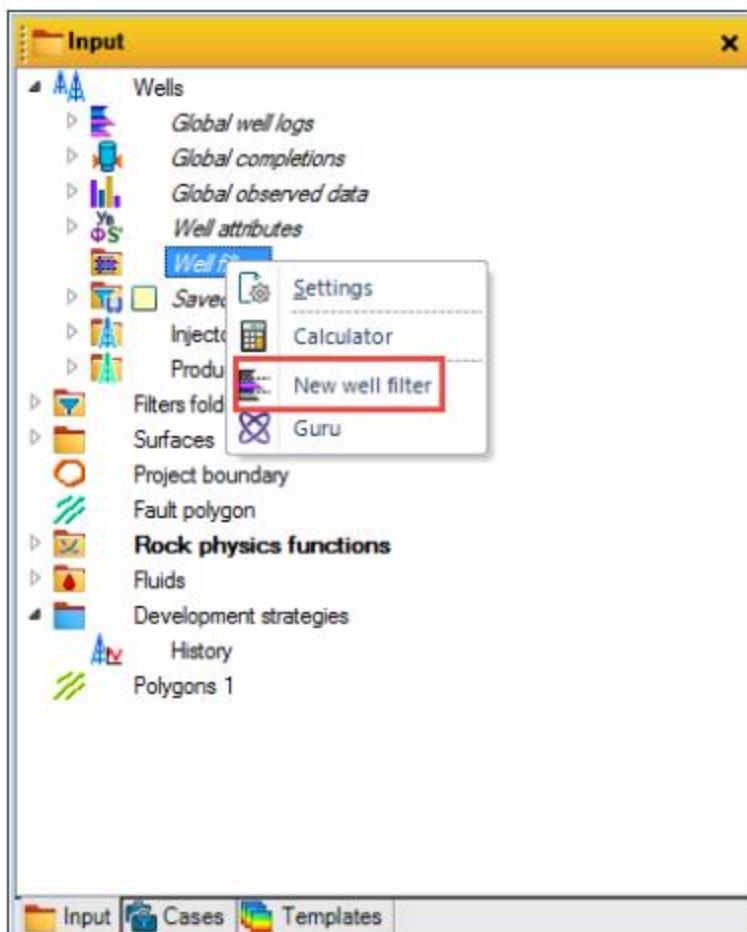
Copyright © 2017 Schlumberger. All rights reserved.



Procedure — Apply a well filter

Well filters limit visualization to the particular sections of a log or well path in a **Well section window**, **Map window**, or **Intersection window**. A well filter can be applied to all wells or a selection of wells. You can create multiple well filters and apply them simultaneously. They can be created based on the absolute Z value or based on well tops or surfaces.

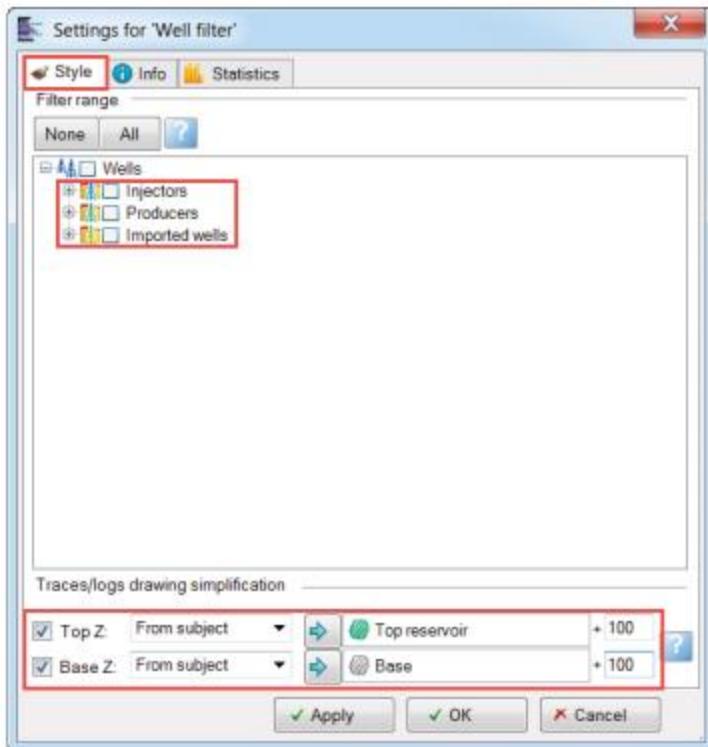
1. To insert a new filter, in the **Input** pane, expand **Wells**, right-click **Well filters**, and click **New well filter**.



2. In the **Settings** dialog box, select which interval of the well trace to display.

Copyright © 2017 Schlumberger. All rights reserved.

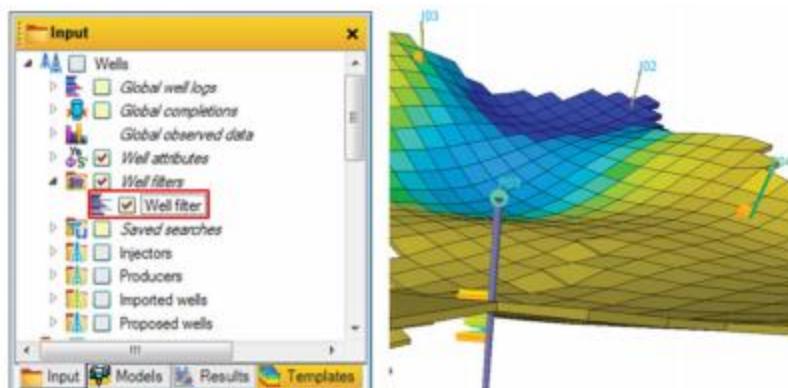
The filter applies to both well logs and traces.



Copyright © 2017 Schlumberger. All rights reserved.

The new filter is created in the **Input** pane in the **Well filters** folder.

3. Select the check box next to the new filter to apply it to the current display and clear the check box to disable its effect.



Saved searches

Saved searches functionality is restricted to well data. It allows you to display and access wells based on specific criteria, organize well data into different place holders without duplicating data, and automate coloring of wells, well paths, well labels, and symbols.

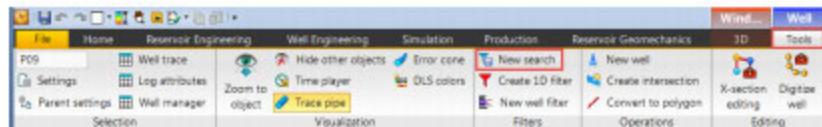
You can apply several types of search criteria and use each search in isolation or in combination with other searches.

You can use a dynamic saved search to filter and group wells based on input data from development strategies, observed data, and simulation cases.

Procedure — Create a user-defined well list Saved search

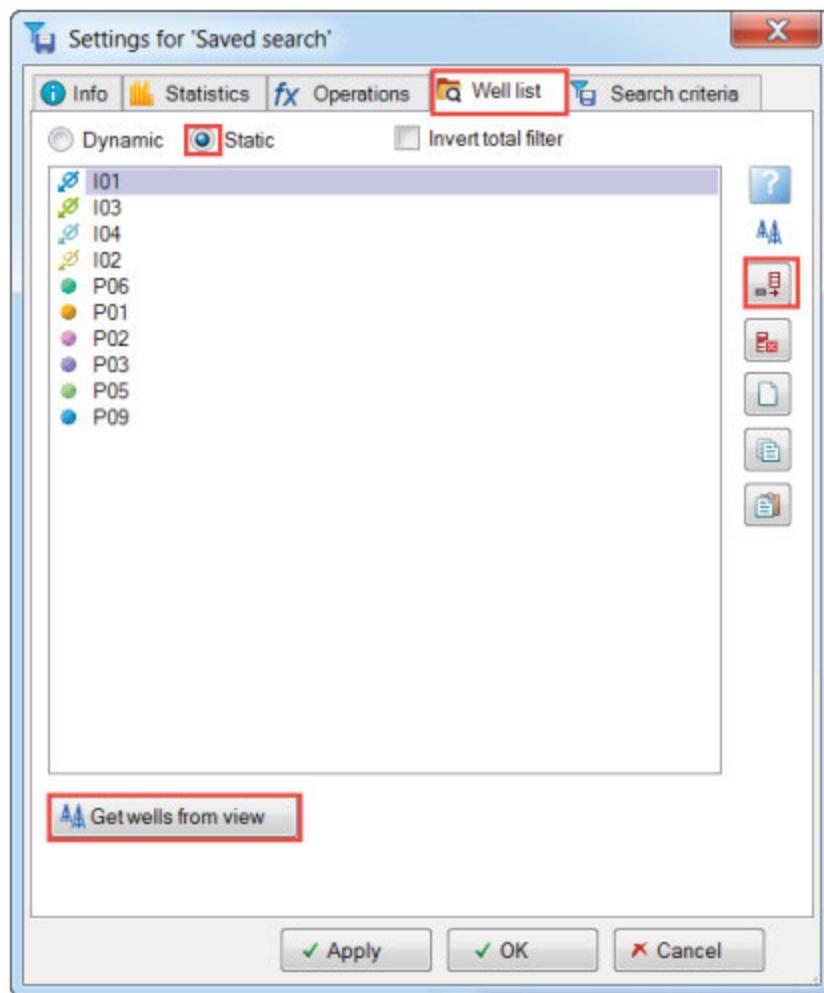


1. Display a well in a **3D window** or click a well name in the **Input** pane.
2. On the **Well** contextual tab, in the **Filters** group, click **New search**  (New search  also appears in the Saved searches folder in the **Input** pane).



3. In the **Settings for saved search** dialog box, on the **Well list** tab, select **Static**.

4. Select the wells you want to add and click **Append selected items (well or well folder in the Petrel explorer)** to the list to add the wells or alternatively, click **Get wells from view** to add all the displayed well to the **Saved search** dialog box.



5. Click **OK** to create a Saved search.

Visual filter

The **Visual filters** pane provides a centralized view of the filters that are applied to the wells, 3D properties, and faults displayed in the active **2D window** or **3D window**.

The filter helps you find details of the applied filters in the Petrel trees and dialogs boxes. It also helps you reset these filters to their initial states.

If the data filter is bold, it shows that the filter is applied. Click the small arrow that points to the right just after the filter object links to find the corresponding filter in the project.

To display or hide the **Visual filters** pane, on the **Home** tab, in the **View** group, click **Visual filters** or press **Ctrl+F8**.

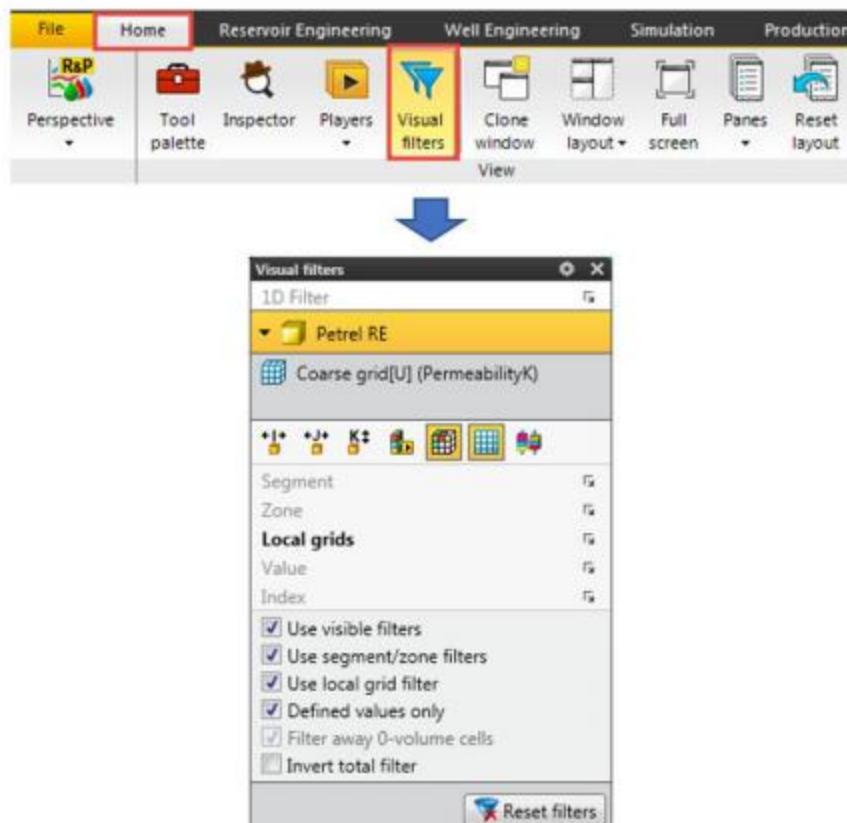


Figure 127. Accessing the *Visual filters*

General intersection

To create an intersection plane, right-click the **Intersections** folder in the **Models** pane and click **Insert intersection plane**. An intersection plane is added to the folder.

Displaying the intersection plane in the **3D window** activates the **Intersection** tab. Under the **Intersection** tab is the **Tools** tab.

To manipulate the active objects in the intersection plane, use the tools on the **Tools** tab.



Figure 128. Tools tab on the Intersection tab

To visualize objects on the intersection plane, on the **Tools** tab, in the **Visualization** group, click **Visualization on intersection** . When this tool is enabled, items with blue check boxes appear on the intersection.

To drag the plane along the displayed object, click **Manipulate plane** on the window toolbar.

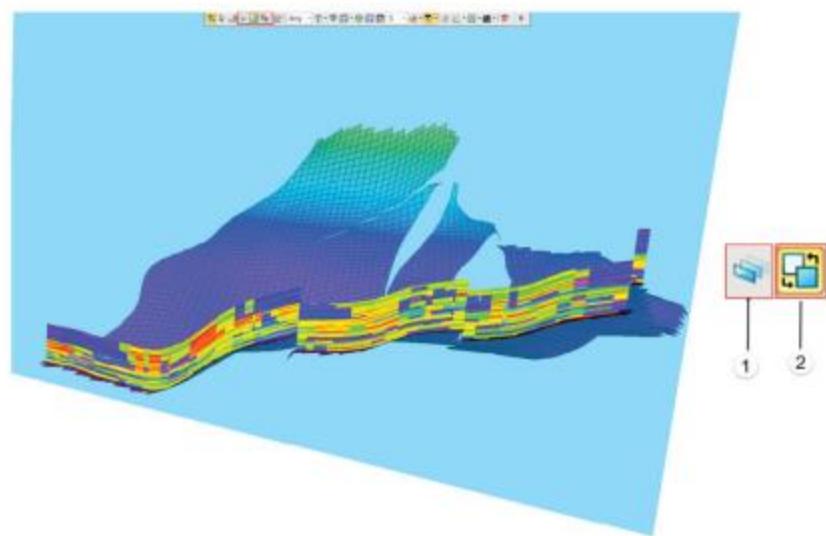


Figure 129. Toolbar with tools for the intersection

- 1 **Manipulate plane** tool
- 2 **Visualize on intersection plane** tool

Intersection window

To display your general intersection in an **Intersection** window, right-click the general intersection and click **Create intersection window**. Select the check box next to the items that you want to display in the **Intersection** window.

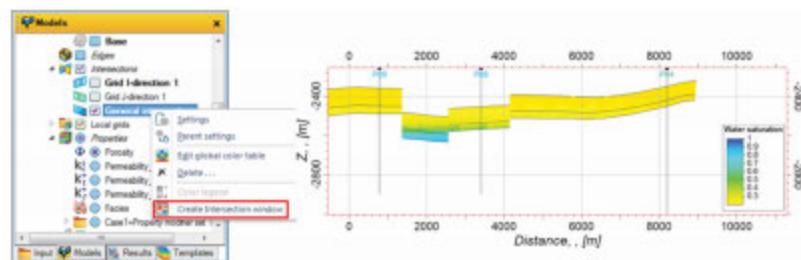


Figure 130. Creating an Intersection window

When you drag the general intersection into the **3D window**, the **Intersection** window automatically updates.

The **Intersection player** tool gives you the flexibility to play the inserted general intersection through the display object in the 3D window. To change the play speed and position and to insert the data object into the **Source** data box on the **Intersection** dialog box, use the options in the **Intersection player**.



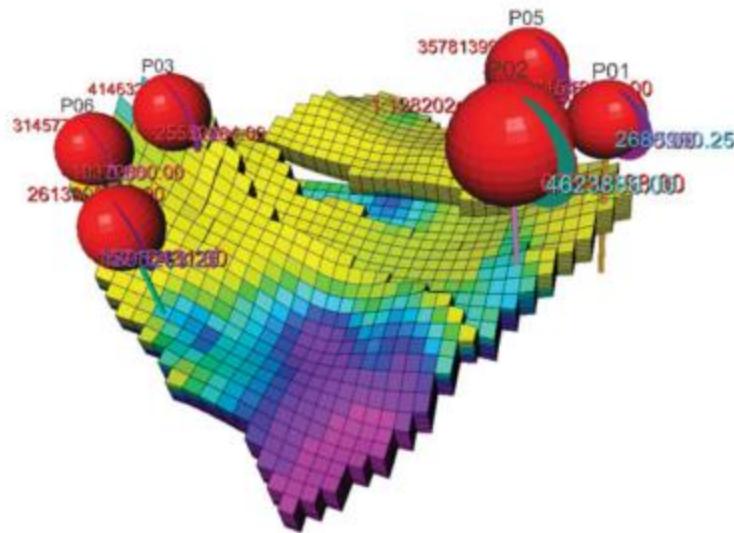
Figure 131. Intersection player



Procedure — Display summary data

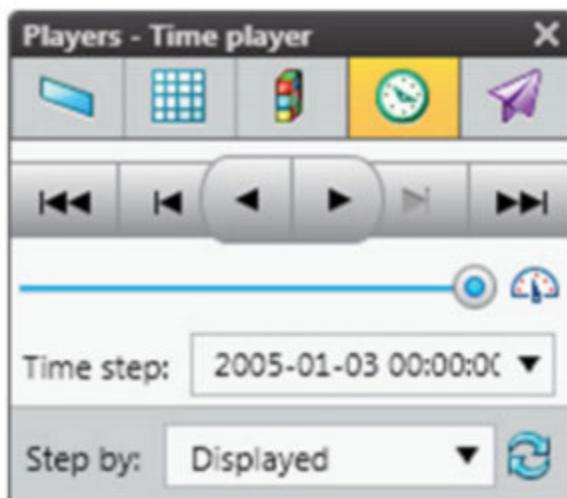
Simulation summary vector output can be displayed as Disc, Sphere, Stack or Bar on wells in conjunction with 3D simulation grid results. You also can play through the simulation timesteps using the **Time player**.

1. Open a **3D window** and display relevant wells.
2. On the **Results** pane, open the **Settings** dialog box for Dynamic data.
3. For chart type, select **Disc, Sphere, Stack, or Bars**.
4. From the Dynamic results data, specify vectors to display.
5. Select the case and identifier.



Copyright © 2017 Schlumberger. All rights reserved.

6. To play through the timesteps, use the **Time player**.

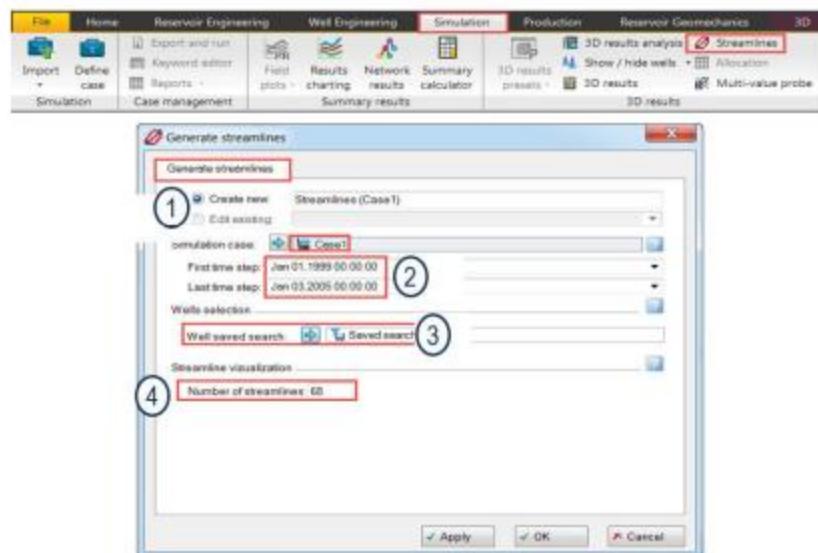


Procedure — Generate streamlines

The Generate streamlines process uses the FrontSim streamlines engine integrated into Petrel to calculate streamlines from ECLIPSE results. This process allows you to use the generated streamlines to quality check your simulation. It also helps you to understand your reservoir without running a separate simulation in FrontSim ((FLORES & FLOWS) needed by the process to run).

1. Insert a simulation case into the **Simulation case data** field.
2. Select the first timestep and last timestep.
3. To help restrict the start points for streamlines and define the first and last timesteps, insert a saved search filter into the **Well saved search** field.

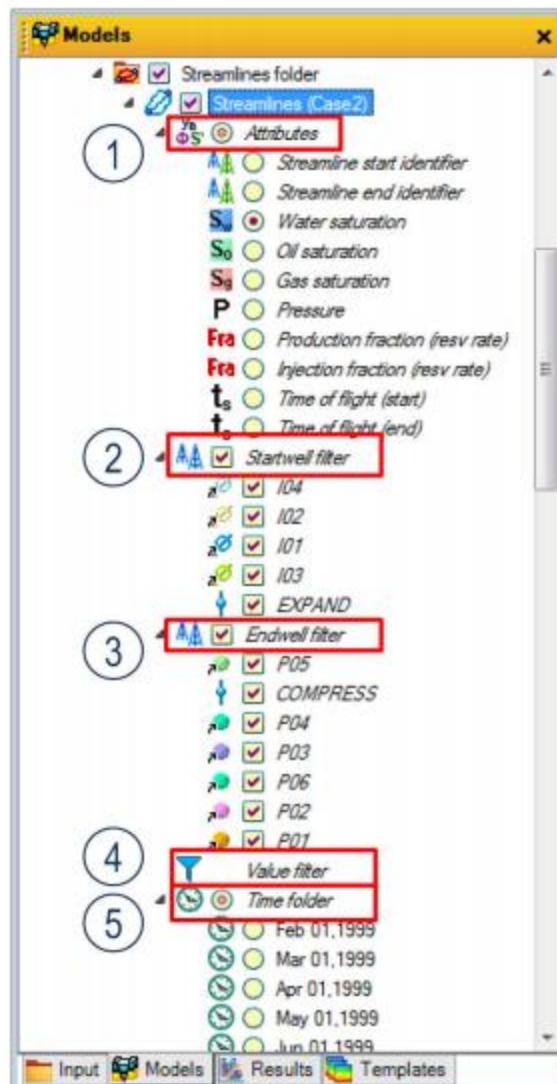
4. Enter the number of streamlines to output for visualization.



Copyright © 2017 Schlumberger. All rights reserved.

Streamlines output results

The generated streamlines are stored in the **Models** pane with the five types of filters shown in the figure.



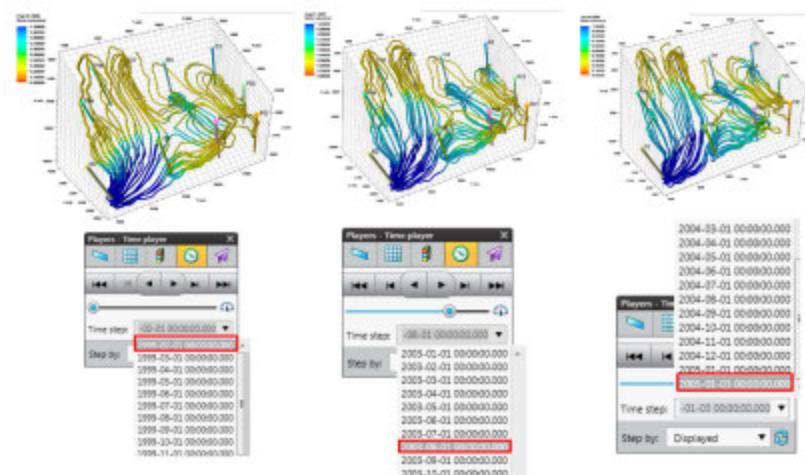
Copyright © 2017 Schlumberger. All rights reserved.

Figure 132. Models pane showing the five filter types from the Generate streamlines process

- 1 Attributes
- 2 Startwell filter
- 3 Endwell filter
- 4 Value filter
- 5 Time filter

Water saturation progression with time

To study the water saturation progression with time, use the **Time player**. For examples, refer to the figure.



Copyright © 2017 Schlumberger. All rights reserved.

Figure 133. Examples of water saturation progression with time

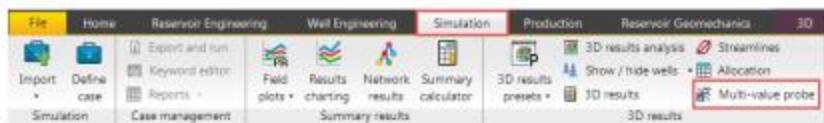
Procedure — Compare results with a multi-value probe

The **Multi-value probe** is a useful tool when comparing results from different simulation cases or checking upscaled values against properties from the fine grid.



1. Open the **Multi-value probe** dialog box. On the **Simulation** tab, in the

3D results group, click **Multi-value probe** 

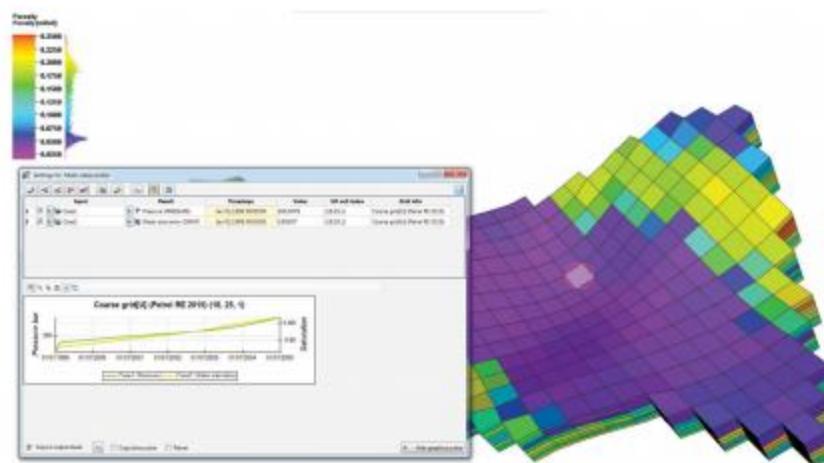


2. Add as many rows as necessary.
3. Insert  grid properties or use 3D simulation results from any grid. If you are using 3D simulation results, insert the simulation case from the **Cases** pane into the **Input** column first.



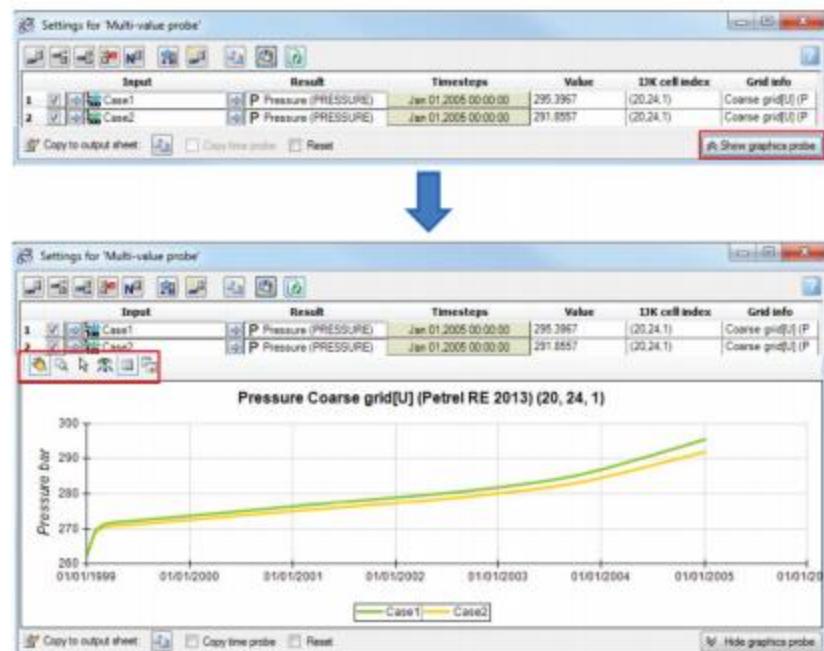
4. Pick a cell with the **Select/Pick mode** tool.

The cell values of the selected properties/results are reported for the picked cells.



The Multi-value probe helps you inspect various grid properties and the 3D simulation results of a grid cell for different timesteps. You also can use it to visualize the 3D simulation results of more than one simulation case.

Clicking **Show graphics probe** enables you to view the time-dependent results.



In the example in the figure, you can see how pressure in cell (20,24,1) increased with time for both simulation Case1 and Case2.

A basic set of tools appears in the toolbar above the chart.



To interact with the chart, use these tools.

Simulation table probe

Simulation table probe is a postprocessing tool that allows you to visualize any scaled curve used in a simulation for a particular cell picked from a 3D model.

The **Simulation table probe** consists of two panes: one for settings and one for plots. To activate the probe, on the **Reservoir Engineering** tab, in

the **Rock physics** group, click  to open the **Simulation table probe** dialog box. Display simulation grid results, then click any cell in the displayed simulation grid results to plot the curve used for that cell.

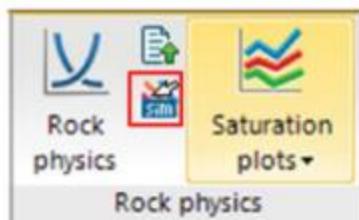
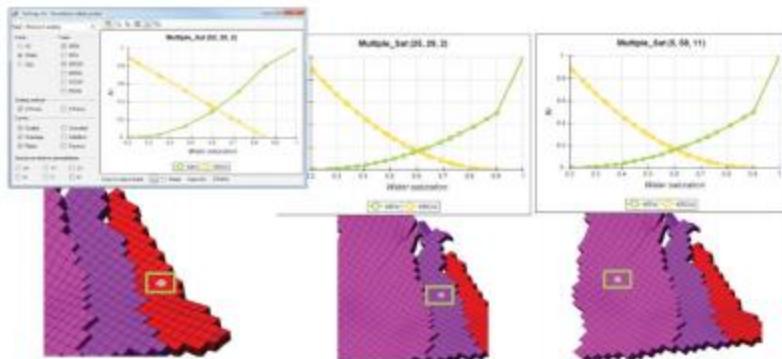


Figure 134. Access the Simulation table probe in the Rock physics group

To see the selected curves, pick a cell in the **3D window**.



Copyright © 2017 Schlumberger. All rights reserved.

Figure 135. Different relative permeability curves used for different reservoir compartments

Exercises — Analyze 3D simulation results

In these exercises, you learn how to analyze 3D simulation grid results using **3D windows**, filters, and other Petrel postprocessing tools.

Workflow

1. View 3D simulation data.
2. Use filters.
3. Use general intersection planes.

Data

Continue to use the same project from the previous exercises.

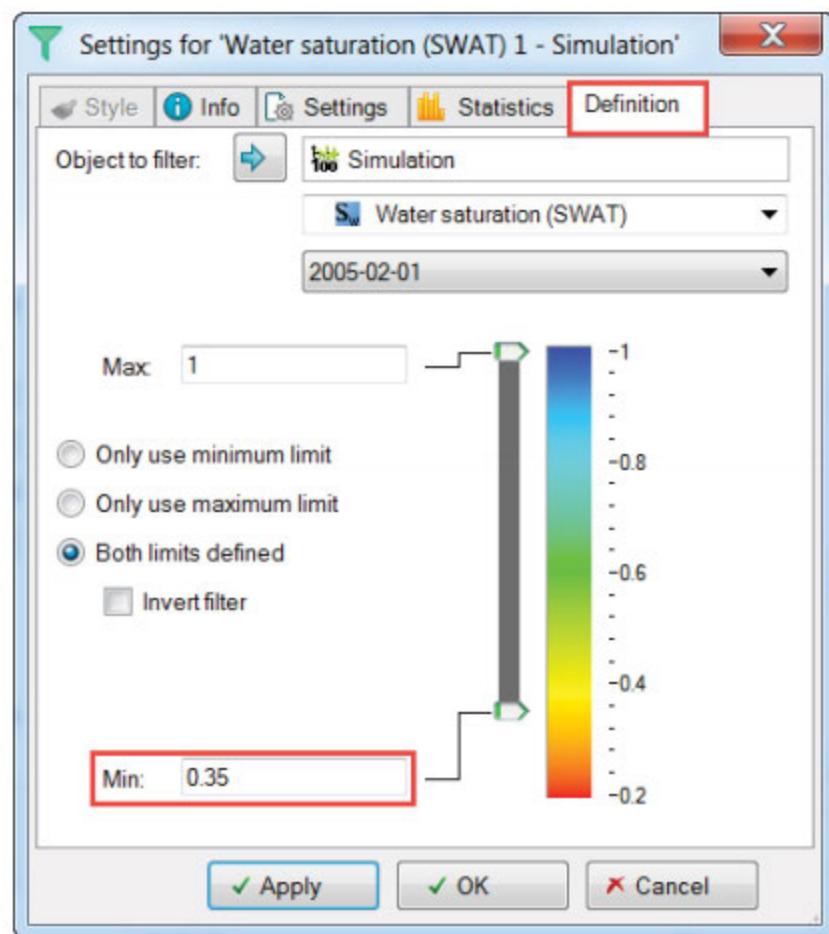
Exercise 1 View 3D results

After running a simulation, grid results are generated together with the result vectors. In this exercise, you display the 3D simulation results in **3D windows** and practice how to apply different filters based on your requirements.

1. Open a new **3D window**.
2. In the **Cases** pane, select the simulation.
3. On the **Simulation** tab, in the **3D results** group, click **3D results presets**, and then click **Saturation**.
4. To enlarge the **3D window** that displays the Water saturation (SWAT), double-click the window tab.
5. If the grid lines are not displayed already, click any cell on the displayed Water saturation grid and click **Show/hide grid lines** on the window mini toolbar  to display them.
6. To display the coordinate axis, click **Show/hide axis**  on the **Window** toolbar and select **Axis**.
7. To apply a filter, right-click the displayed Water saturation property and click **Create 1D filter**  Create 1D filter ...
8. In the **Settings** dialog box for the displayed water saturation property, use the slider to select a minimum water saturation of 0 . 35.
9. Click **Apply** and observe the change in the **3D window**.

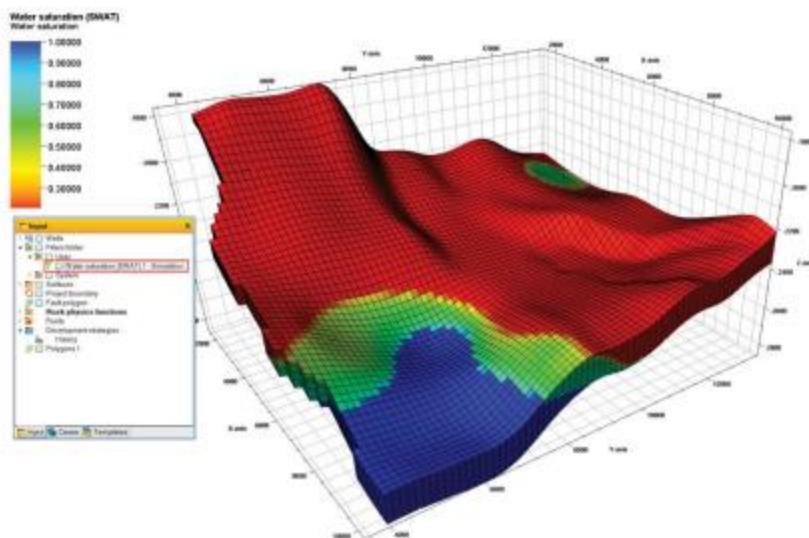


10. Click **OK** to close the dialog box.



Copyright © 2017 Schlumberger. All rights reserved.

11. In the **Input** pane, open the **Filters** folder and turn off the filter you just created.



Copyright © 2017 SCS Software. All rights reserved.

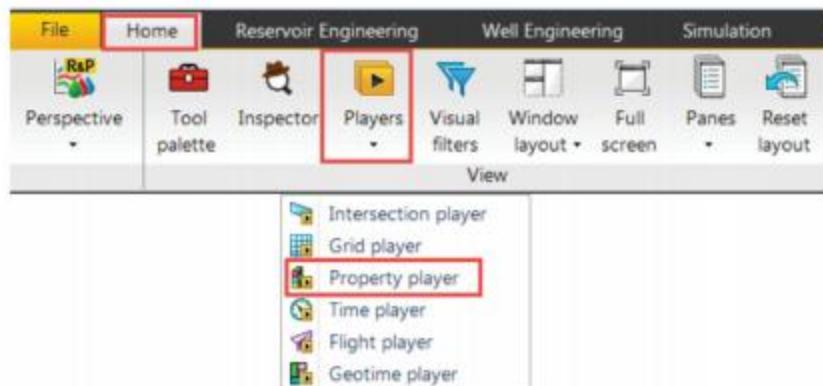
Exercise 2 Use I, J, and K filters in a 3D window

There is a wide range of available filters for viewing data in 3D. In this exercise, you learn how to use some of them. Continue to work with the **3D window** that displays the water saturation from the previous exercise.



1. On the **Home** tab, in the **View** group, click **Players** and then click **Property player**. Select slice direction: **K**.

Alternatively, right-click the displayed water saturation property and click **K intersection**.



- Using the **Property player**, view one K-layer at a time by moving the slider to the right or left. You also can select a specific K-layer using the **Index** list.



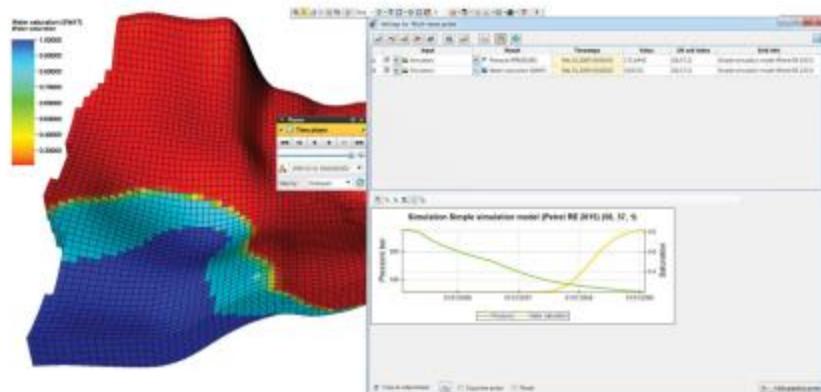
- Move to another K-layer. Click the step property **forward** and **backward** buttons. You also can play through the entire K-layer by clicking **Property plays forward** and **Property plays backward**.



4. Click again to cancel the K-layer filter.
5. Right-click the active displayed **Water saturation property** again and click **Multi-value probe** on the mini toolbar .

Alternatively, you can access the **Multi-value probe** tool on the **Simulation** tab in the **3D results** group.

6. Add two data field rows by clicking **Append item in the table** .
7. In the **Input** column, insert the Simulation case from the **Cases** pane for the first two rows.
8. In the **Results** column, insert **Pressure** and **Oil saturation** from the Dynamic subfolder in the Simulation grid results folder in the **Results** pane.
9. Click **Show graphics probe** and click any of the cells of the displayed Oil saturation grid property to see the progression of Pressure and Oil saturation with time.



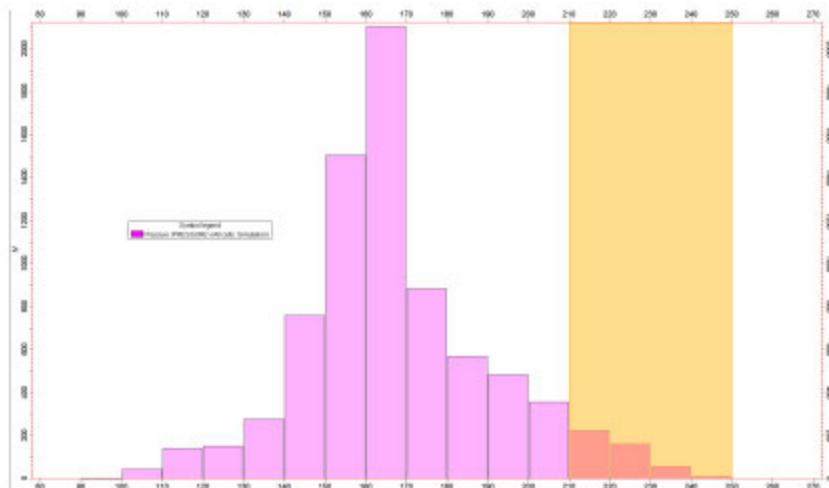
Copyright © 2017 Schlumberger. All rights reserved.

Exercise 3 Create filters using a Histogram window

This exercise illustrates how to integrate some of the Petrel windows used for data analysis. For example, you can create a filter from the data displayed in a **Histogram** window or a **Function** window and it is applied simultaneously to data displayed in a **2D window** or **3D window**.

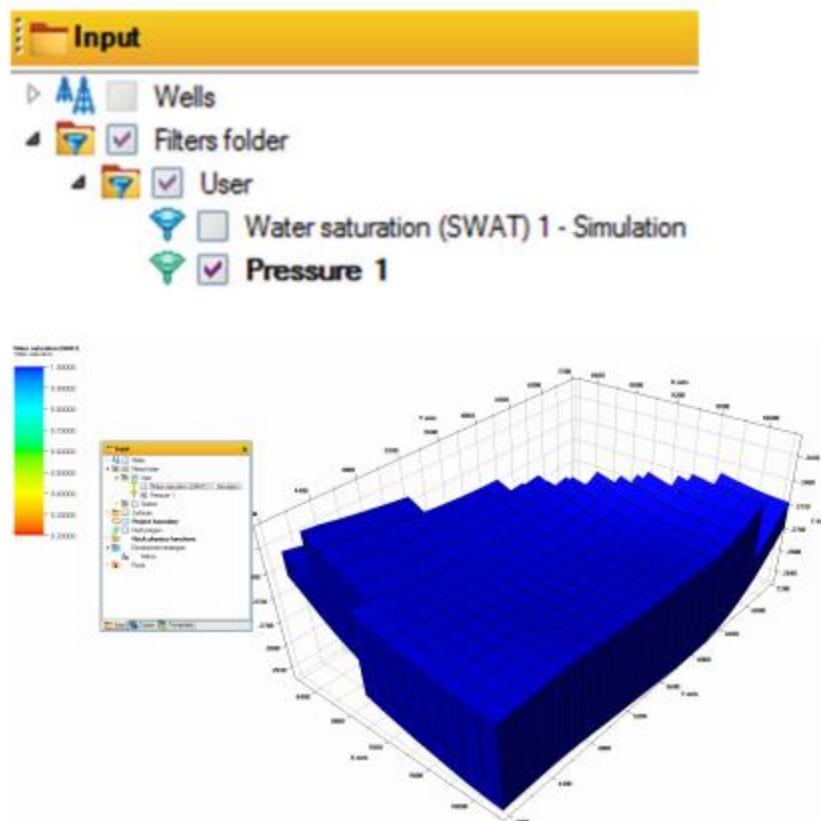


1. On the **Home** tab, in the **Insert** group, click **Window** and then click **Histogram**.
 2. In the **Cases** pane, select the case named Simulation.
 3. In the **Results** pane, expand **Simulation grid results**, expand **Dynamic**, and select to display **Pressure**.
 4. On the **Window** toolbar, click **Select using 1D range on x-axis** .
- You will create a filter that displays only cells with high-pressure values.
5. To define the filter, click in the **Histogram** window, drag the pointer, and select only the highest pressure values.



6. In the **3D window**, display water saturation from the same case named Simulation.
7. In the **Input** pane, expand **Filters**, expand **User**, and select the **Pressure**.

What effect do you observe in the **3D window**?



Exercise 4 View data on an intersection plane

In this exercise, you practice how to use the General intersection plane and **Intersection** window to view and QC data from a 3D model.

1. Open a new **3D window**, activate the Simulation case in the **Cases** pane, and display the water saturation. Remember to turn off all filters that you applied previously.
2. In your Simple simulation model, right-click **Intersections** and click **Insert Intersection plane**.

3. View data on the plane. On the **Tools** tab, in the **Visualization** group,

click **Visualize on intersection** 

Alternatively, click **Visualization on intersection** on the **Window** toolbar.

Observe that the items available for display on the plane now have a blue check box or blue option button.

4. In the **Results** pane, expand **Simulation grid results**, expand **Dynamic**, and turn on **Water saturation (SWAT)** to view the saturation on the plane.
5. On the **Tools** tab, in the **Visualization** group, click **Clip**, and then click

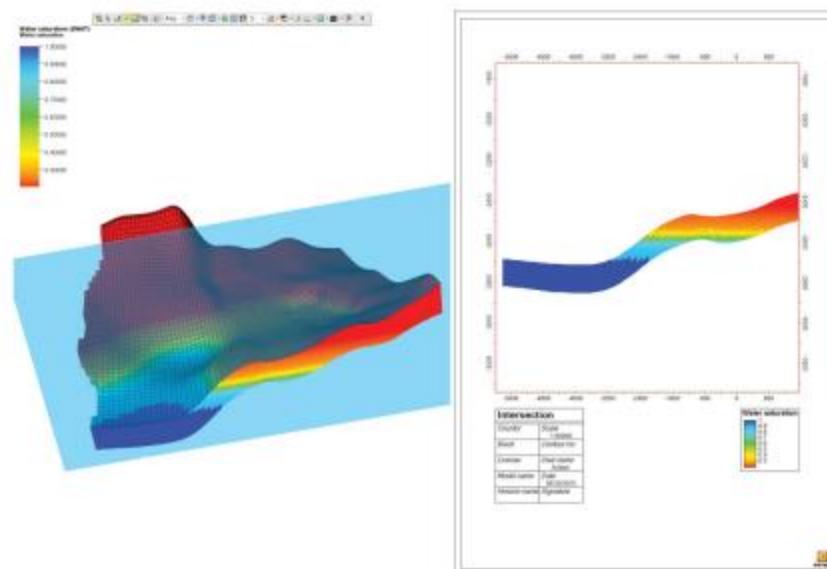
Clip in front 



6. On the **Window** toolbar, click **Manipulate plane**  to move the plane along the water saturation grid property.
7. Press and hold the Ctrl+Shift keys and click the plane to move it randomly in all directions, while maintaining its vertical position.
8. In the **Models** pane, right-click **General intersection** and click **Create intersection window**.
9. In the **Cases** pane, select **Simulation**.
10. In the **Results** pane, expand **Simulation grid results**, expand **Dynamic**, and select **Water saturation** to view the saturation in the **Intersection window**.
11. On the **Home** tab, in the **View** group, click **Window layout** and then click **Tile vertical** to tile the **Intersection window** vertically with the **3D window**. Close other windows except the **Intersection window** and **3D window** so you can focus on these two windows.
12. Move the intersection plane in the **3D window**.

What do you notice in the **Intersection window**?

13. Flip the **Intersection window**. On the **Window** toolbar, click **Flip window**.



Copyright © 2017 Schlumberger. All rights reserved.



Lesson 4 Summary calculator and Geometrical modeling

In this lesson, you learn how to:

- create new summary vectors from the existing simulation results using the **Summary calculator**
- use the **Geometrical modeling** tool to quality check the simulation grid and generate a Well index
- create new grid properties using the **3D results calculator**

Summary calculator

The **Summary calculator** allows you to create a new simulation vector from the existing simulation results. The created summary vector is stored in the **Results** pane. You can plot it with other results vectors using the **Results charting and analysis** tool.

Access the **Summary calculator** in one of these ways:

- On the **Simulation** tab, in the **Summary results** group, click **Summary calculator**.
- In the **Results** pane, right-click **Dynamic results data**, and click **Calculator**.

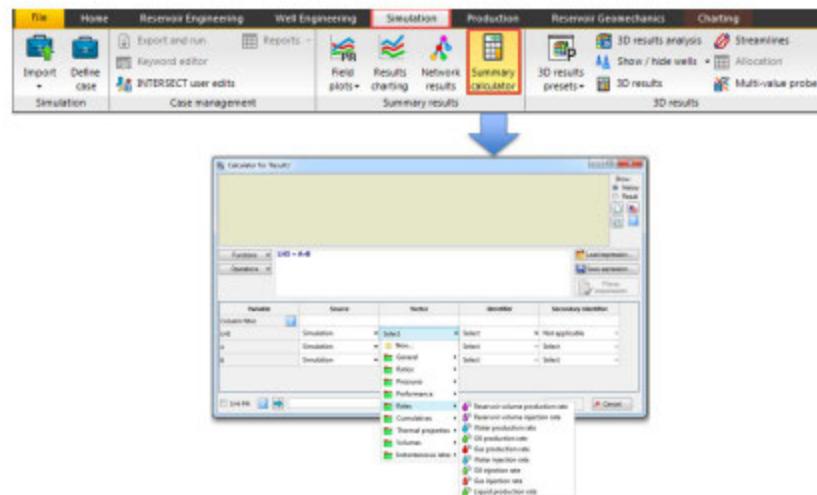


Figure 136. Accessing the *Summary calculator* from the *Simulation* tab

Summary calculator: Summary vectors workflow

The formula used to relate many variables is used, together with input data, by the **Summary calculator** to generate new data.

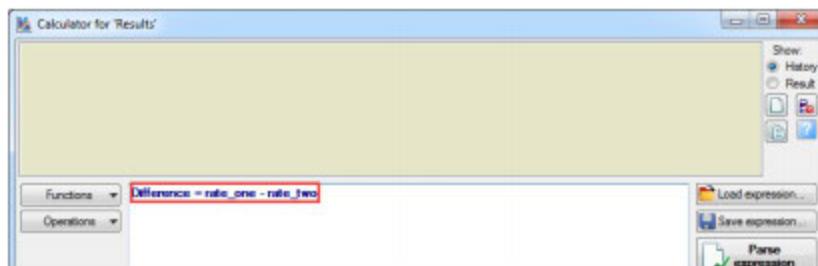
To create new data, the **Summary calculator** needs some input data and the relationship between this data. The process for generating this data is broken down into two parts.

1. A text expression describes the formula to be used to relate several variables. This expression is entered into the expression box.
2. This expression is parsed to extract the variables and then these variable names are bound to particular data items. Unlike the other calculators, names of variables are not important because they do not relate directly to the data. However, they must be bound to data in a separate step.

Procedure — Create a summary vector

These steps show the workflow to create a new summary vector from the existing simulation result vectors.

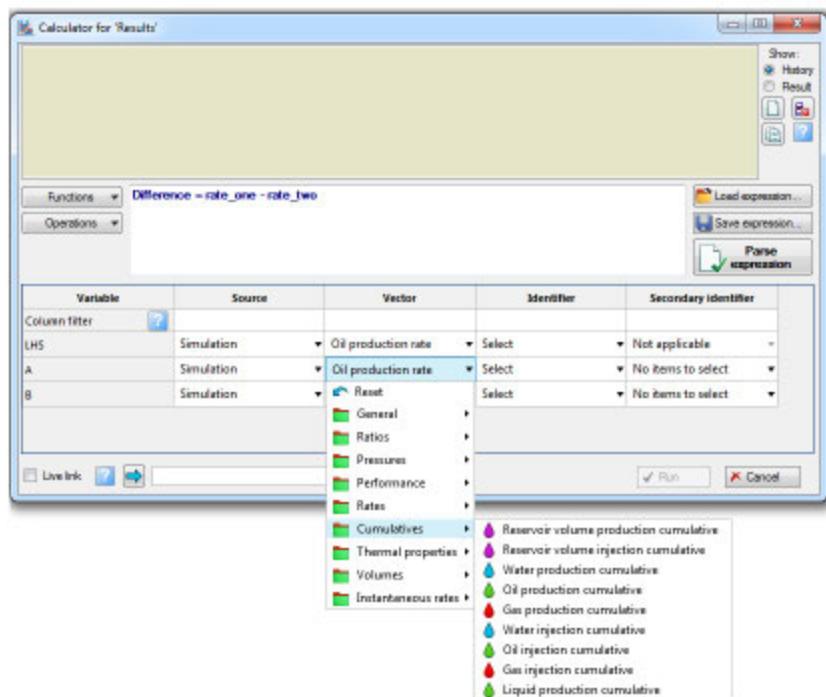
1. Open the **Summary calculator**. On the **Simulation** tab, in the **Summary results** group, click **Summary calculator**.
Alternatively, in the **Results** pane, expand **Views**, right-click **Dynamic data**, and then click **Calculator**.
2. Enter an expression, such as, `Difference = rate_one - rate_two`.



3. Click Parse expression.

Variable	Source	Vector	Identifier	Secondary Identifier
Difference	Select	Select	Select	Not applicable
rate_one	Select	Select	Select	Select
rate_two	Select	Select	Select	Select

4. Specify the data items (sources, vector, and identifier) to which the variables in the expression should be bound before running the calculator.



Copyright © 2017 Schlumberger. All rights reserved.

- **Source:** Specify this item first. The data can be taken from existing simulations, observed data, or data from development strategies. The Source is important because it affects how the items in the Vector and Identifier are filtered, so the Vector and Identifier columns (except for the output variable) are disabled until a Source is selected.
- **Vector:** The Vector determines the type of data to be taken from the Source defined in the Source column, for example, Oil production rate.

- The **Vector** list is taken from the **Results** pane and is organized the same. This organization allows you to have all the list items at the top level, or to categorize the items in the list. Only items in the **Results** pane appear in the **Vector** list, because these items are the only possible options that have data to select, including any user-created data.
- Identifier: The Identifier determines the object for which the data is used, for example, a particular Well or Group. The **Identifier** list is taken from the **Identifier** folder in the **Results** pane and is structured the same.
- Secondary identifier: Some dynamic data results require a second identifier to uniquely specify them, for example, compositional flow rates require a well and a component. In this instance, the Secondary identifier is enabled and displays the relevant secondary identifiers to uniquely find a result.

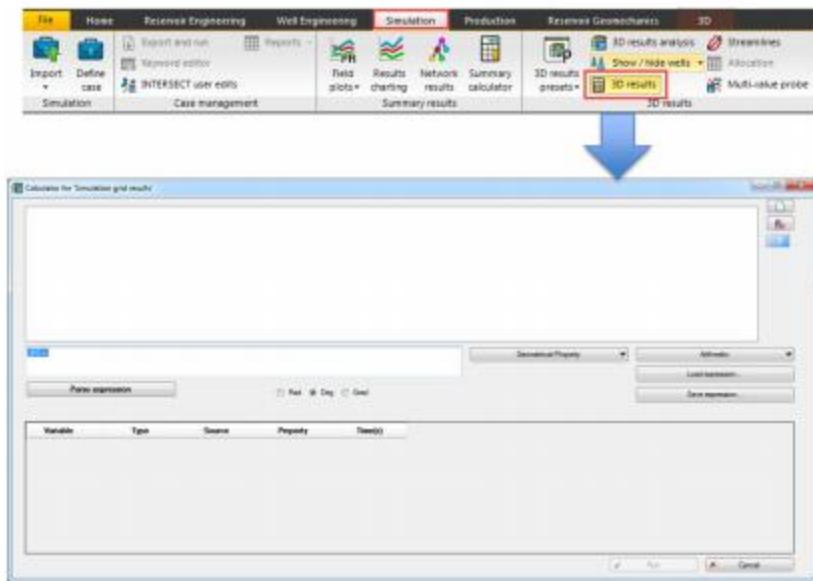
Procedure — Perform calculations on 3D simulation grid properties



You also can use the **3D results calculator** to perform operations on the grid properties in the Simulation grid results folder in the **Results** pane. The **3D results calculator** can be accessed on the **Simulation** tab, in the **3D results** group.

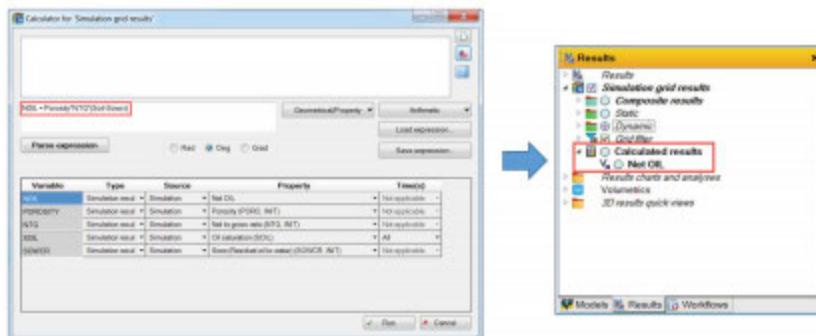
These steps show the workflow to perform calculations on the 3D simulation grid results.

1. Open the **3D results calculator**. On the **Simulation** tab, in the **3D results** group, click **3D results**.



2. Enter the formula that describes your calculation in the field above **Parse expression**, for example:
NOIL=Porosity*NTG*(Soil-Sowcr)
3. Click **Parse expression**.
4. Specify the Net Oil property and its template by clicking the **Property** column in the same row as the Net Oil variable.
5. Click **New property** to open the **New calculated result** dialog box.
6. Enter the Property name and select the appropriate template for the property by clicking the **Select** button on the dialog box.
7. Enter the 3D results or grid properties to associate with each variable used in the expression.

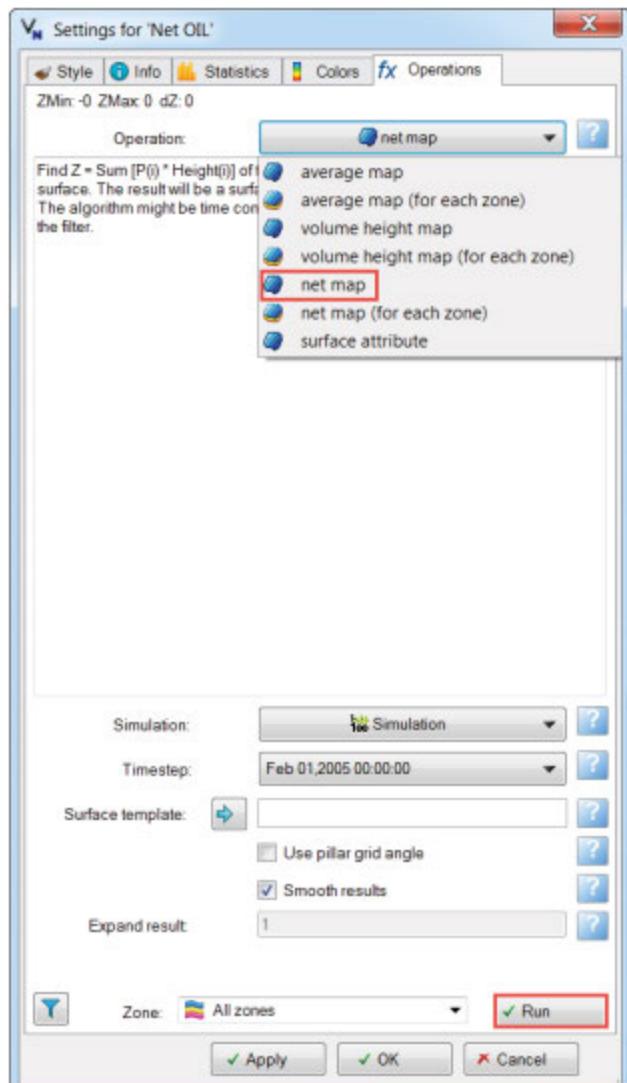
8. Click Run.



Procedure — Create a property map for calculated Net Oil

1. Right-click the calculated Net oil (Noil) and open the **Settings** dialog box.
2. On the **Operations** tab, expand **Make map from property**, click **Make net map**, and then click **Run** to create a Net Oil map.

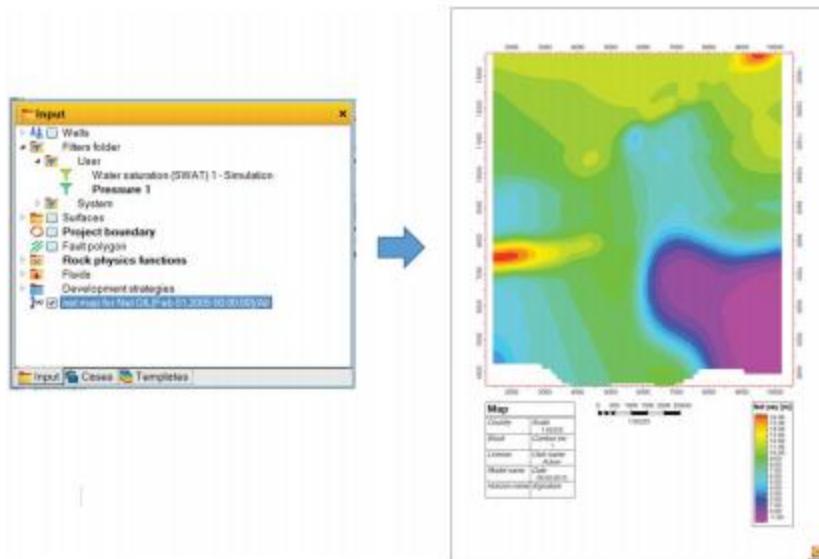
Copyright © 2017 Schlumberger. All rights reserved.



3. Open the **2D window**.

Copyright © 2017 Schlumberger. All rights reserved.

- In the **Input** pane, select the **Net oil map** check box to display the Net oil map.



Copyright © 2017 Schlumberger. All rights reserved.

Geometrical modeling

Geometrical modeling allows you to generate properties based on grid geometry and other inputs. Some of the geometry properties that can be generated are Cell angle, Cell height, Height above the contact, and Distance to object. You access the **Geometrical modeling** dialog box from the **3D and fault properties** group on the **Reservoir Engineering** tab.

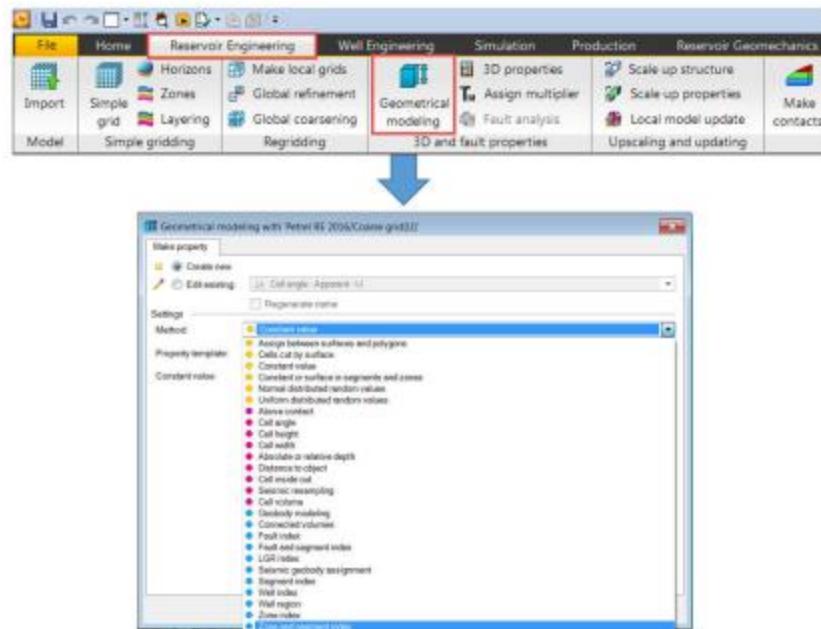


Figure 137. Geometrical modeling dialog box

Grid geometry

Three methods in the Geometrical modeling process are especially important for quality checking the simulation grid: Cell angle, Cell volume, and Cell inside out.

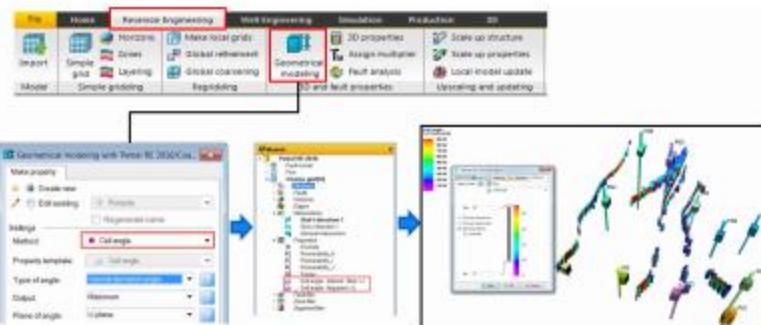


Figure 138. Procedure for generating a cell angle grid property

The geometry of the grid cells influences the flow computation in numerical simulators.

You can set the problematic cells as inactive (ACTNUM=0). If the problems are severe, rebuild the grid.

This process is not restricted to simple geometrical properties; it also covers more complex property distributions:

- Surface and polygons assignments
- Constant value
- Above contact
- Distance to object
- Connected volumes
- LGR index

Well index

You can use the Well index property to visualize all cells penetrated by wells.



Figure 139. Creating a Well index



Exercise 1 Create new vectors using the Summary calculator

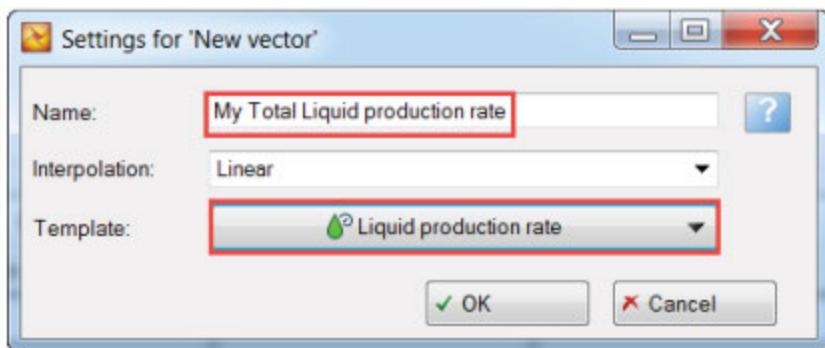
In this exercise, you learn how to create new vectors from existing simulation results.

Continue to use the same project from the previous exercises.

1. On the **Simulation** tab, in the **Summary results** group, click **Summary calculator**.
2. In the **Summary calculator**, enter the expression $LHS=A+B$ and click **Parse expression**.

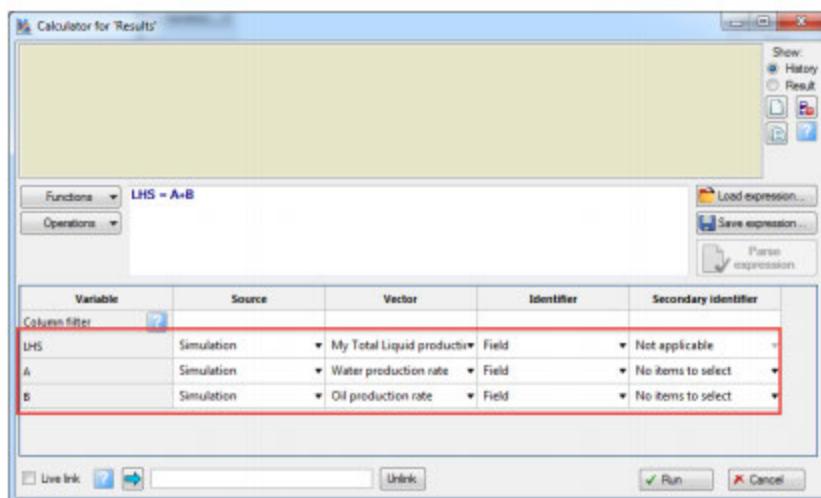
A table appears at the bottom of the calculator. Use this table to specify what the items in the expression represent.

3. For the LHS row:
 - a. Select **Simulation** as the Source.
 - b. In the Vector column, select New.
 - c. In the **Settings** dialog box for the new vector, name the vector **My Total Liquid production rate**.
 - d. Select the template **Production templates**, select **Liquid production rate** and click **OK**.



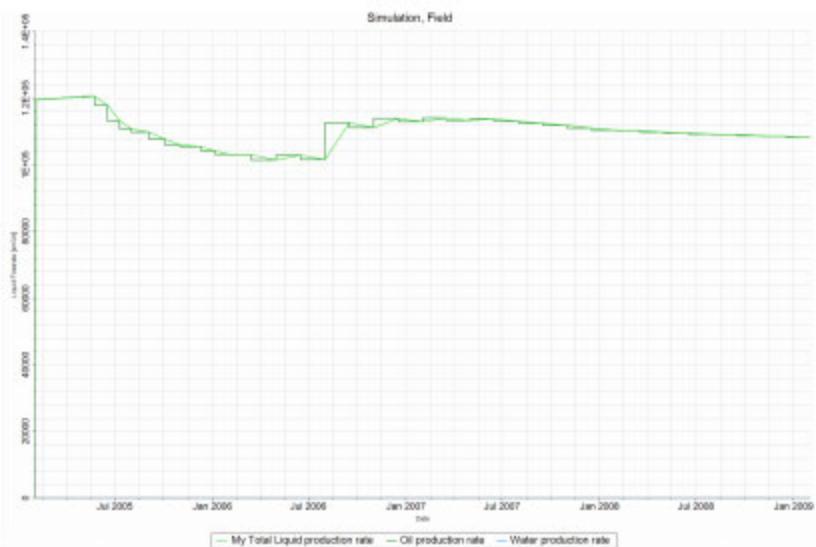
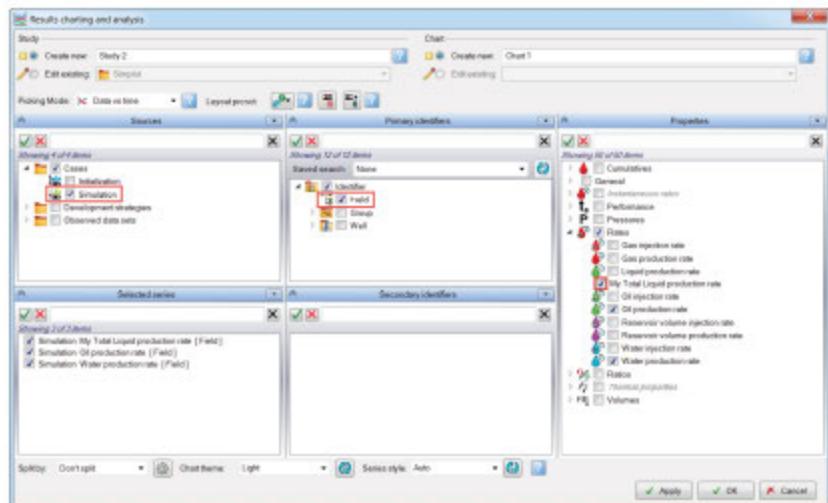
4. For the A row:
 - a. Select **Simulation** as the Source.
 - b. In the Vector column, select **Rates: Water production rate**.
 - c. In the Identifier column, select **Field**.

5. For the B row:
 - a. Select **Simulation** as the Source.
 - b. In the Vector column, select **Rates: Oil production rate**.
 - c. In the Identifier column, select **Field**.



6. Click **Run**.
7. Click **Cancel** to close the dialog box.
8. Open the **Results charting and analysis** tool.
9. In the **Sources** pane, select the **Simulation**.
10. In the **Primary identifiers** pane, select **Field**.

11. In the **Properties** pane, select the **Water production rate** and **Oil production rate** in the **Rate** folder, along with your new vector, **My Total Liquid production rate** in the **General** folder.



Copyright © 2017 Schlumberger. All rights reserved.

Lesson 5 Dynamic saved searches



A dynamic saved search is a type of extended saved search used to filter and group wells in the **Input** pane based on input data from development strategies, observed data, and simulation cases.

To filter the data for the dynamic saved search, you select a source, a property from that source, start and end dates, the value to compare against, the desired operator, and the frequency with which the comparison needs to meet the criteria. The units are given for reference only. They cannot be changed when creating a dynamic saved search.

Dynamic saved searches are listed in the Saved searches folder in the Wells folder in the **Input** pane. To apply a dynamic saved search, select the check box next to its name. Notice that [Search ON] is displayed next to Wells to indicate that a search is currently applied. To move wells filtered by a dynamic saved search to a new folder in the Wells folder, right-click the dynamic saved search and then click **Move wells to folder based on saved search**.

NOTE: Dynamic saved searches depend on calculated input data from defined sources. Any amendment of already defined saved search criteria affects the saved search output. In the **Input** pane, an amended saved search is indicated by an exclamation point. If you turn the search off and on, it recalculates its filtered wells.



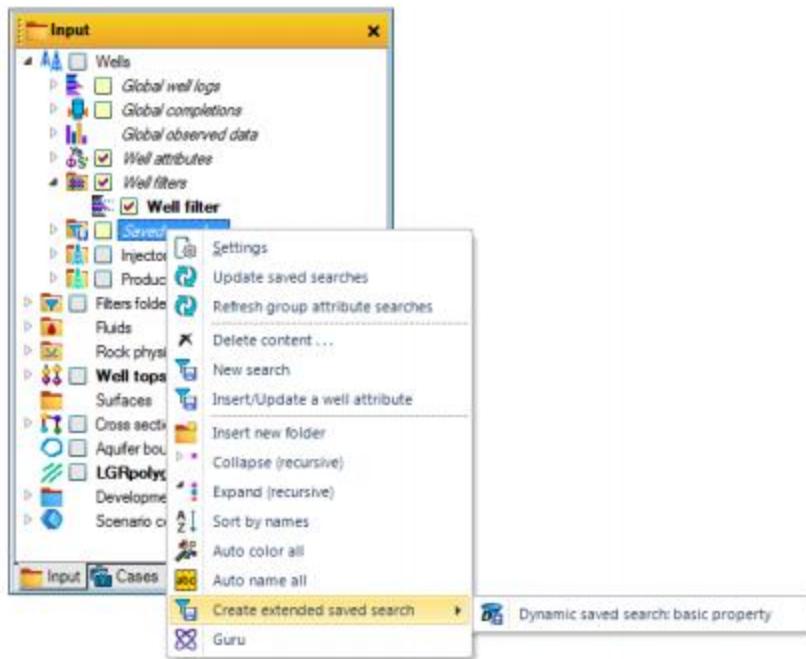
Copyright © 2017 Schlumberger. All rights reserved.

Procedure — Create a dynamic saved search



1. In the **Input** pane, expand Wells.

- Right-click **Saved searches**, click **Create extended saved search**, and then click **Dynamic saved search: basic property** to open the **Settings** dialog box.



- In the **Settings** dialog box, on the **Search criteria** tab, define your filter criteria for the search:
 - To add a new row to the table, click **Append comparison**.
 - In the **Source** column, select the input data source that will define the available properties for comparison.
 - In the **Property** column, select the property by which the comparison will be carried out, based on the selected input data source.
 - Select the **Start** and **End** dates over which the comparison is to be made.

These dates define the time period in which the comparison will be relevant. The values available depend on the selection of source and property. It is possible to search on only one timestep by setting the start and end date to the same date.

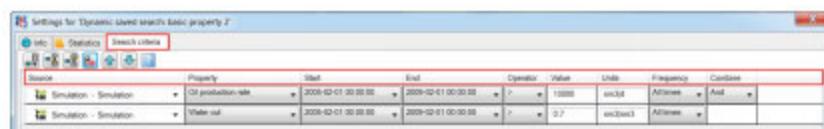
- Select the comparison **Operator** and **Value**.

The value that you enter is compared to the selected input data using the selected operator.

- Select the **Frequency of occurrence** over the selected time range.

If you select **All times**, the property value comparison is run for all times in the selected period. A well is included in the search if its input data satisfies the comparison options for all dates in the period.

If you select **Once**, a well is included in the search if its input data satisfies the comparison options for at least one time in the selected period.



4. Click **Apply** or **OK**.

Comparisons are combined in order according to the chosen **Combine** option. If you select **And**, the resultant wells of the current comparison (one or more rows) are combined with the resultant wells of the next comparison (row) as an intersection. If you select **Or**, the rows are combined as a union.

To modify an existing search, double-click the search to open the **Settings** dialog box. On the **Search criteria** tab, change the criteria used by the search.

Copyright © 2017 Schlumberger. All rights reserved.

Exercise 1 Use a dynamic saved search to filter wells based on the simulation results

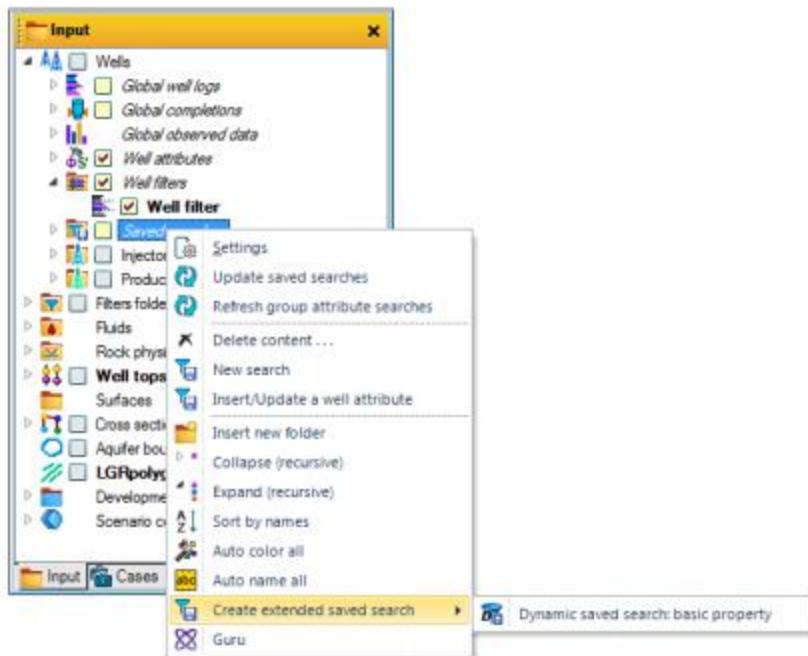


In this exercise, you use the simulation case results to filter the wells in the **Input** pane based on the defined search criteria using the extended Dynamic saved search option.

Exercise data: Continue to use the same project from the previous exercise

- In the **Input** pane, expand **Wells**.

2. Right-click **Saved search**, click **Create extended saved search**, and then click **Dynamic saved search: basic property** to open the **Settings** dialog box for Dynamic saved search.

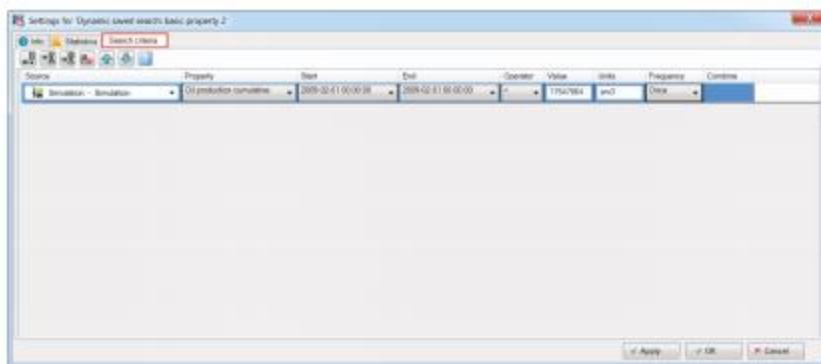


3. In the **Settings** dialog box, on the **Search criteria** tab, define the filter criteria for the search.



- Add a new row to the table. Click **Append comparison** .
- In the Source column, select **Simulation** as the input data source.
- In the Property column, select **Oil production cumulative** as the Property filter.
- Select the Start date of **2009-02-01** and End date of **2009-02-01** for a timestep.
- Select **> (more than)** as the comparison Operator.
- In the Value column, enter **17547864**.

- g. In the Frequency column, select **Once** as the frequency of occurrence over the selected time range.



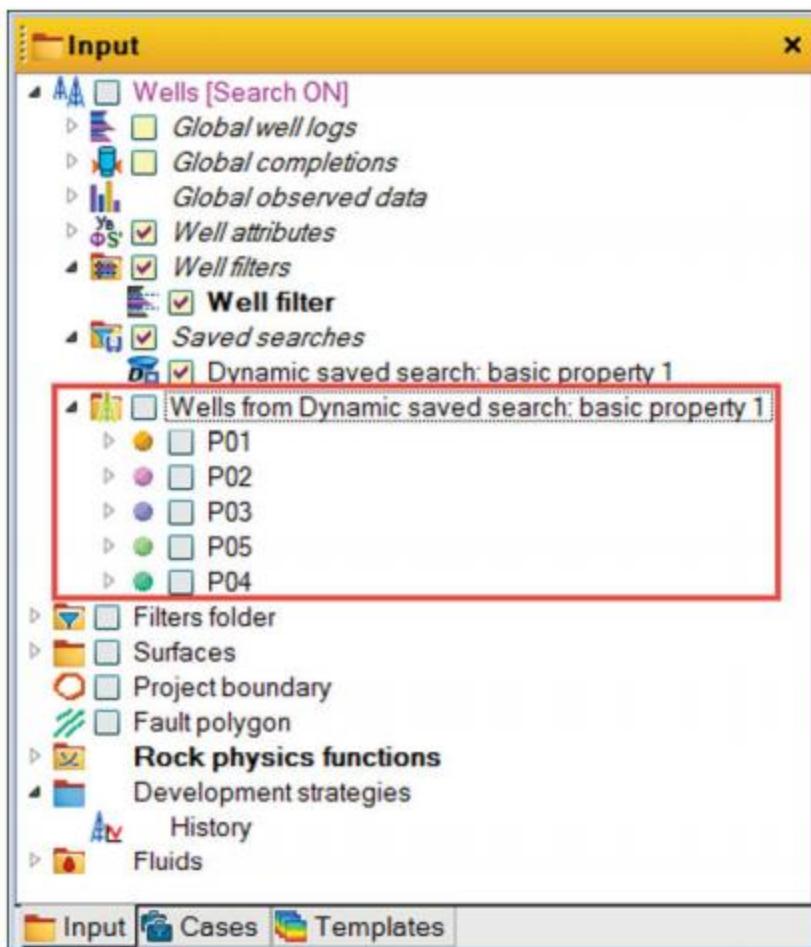
4. To generate the Dynamic saved search filter based on the defined search criteria, click **OK**.

The created Dynamic saved search: basic property is stored in the **Input** pane in the **Saved searches** folder.

5. To apply the Dynamic saved search filter, select the **Dynamic saved search: basic property** check box.

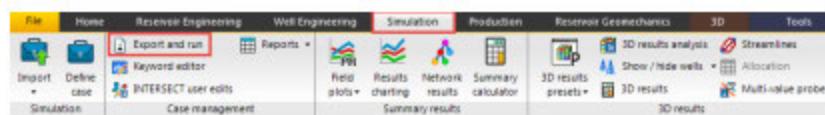
[Search ON] is displayed next to **Wells** folder to indicate that a search filter is currently applied.

6. Right-click the **Dynamic saved search: basic property** and click **Move wells to folder based on saved search**. This option allows you to move a well to a separate folder based on the saved search.



7. Export the Oil cumulative production for P01, P02, P03, and P05 to an Excel spreadsheet and validate the Dynamic saved search results.
8. Rerun the Simulation case.
 - a. In the **Cases** pane, select the Simulation case.

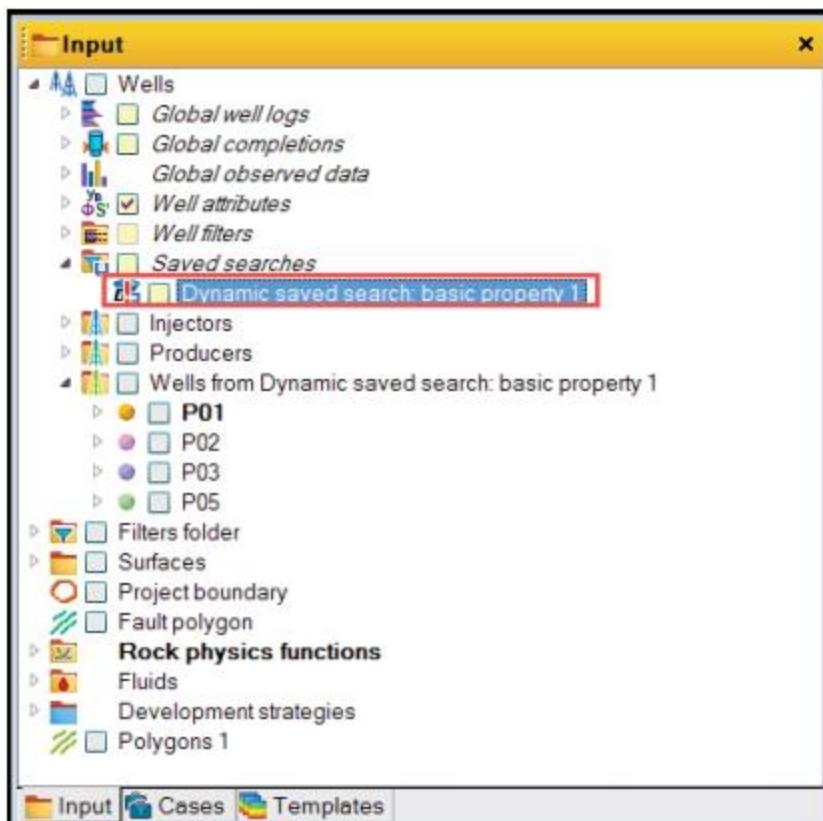
- b. On the **Simulation** tab, in the **Case management** group, click **Export and run**.



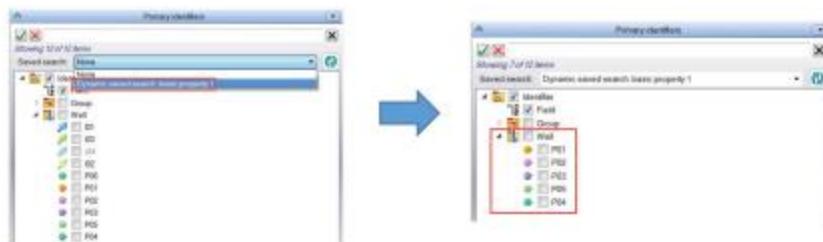
9. Turn on and off the **Dynamic saved search: basic property** check box to recalculate the filtered well.

NOTE: It is important to note that Dynamic saved searches depend on the calculated input data. Therefore, any change in the input data will affect the related saved search. In such cases, any potentially affected dynamic saved search is marked with an exclamation mark in the **Input** pane. Turning the search and on and off causes the dynamic saved search to recalculate the well filters.





10. Open the **Result charting and analysis** dialog box.
11. In the **Primary identifiers** pane, in the **Saved search** field, select **Dynamic saved search:basic property** from the list to filter for the Dynamic search that you previously created.
12. Activate the **Well** folder to select only the wells in the saved search, then display Oil production cumulative for these wells.



Review and summary

Review what you learned in this module.

The review and summary help you to reinforce the learning objectives for the Simulation run and results viewing module.

Review questions



Use these questions to review what you learned about viewing simulation results.

- To generate streamlines, do you need the FrontSim simulator? What process is available in Petrel to help you generate streamlines?
- What are the two basic filters in Petrel?
- What is the major difference between a **Plot window** and a **Map window**?
- What tools do you use to create a new vector from existing simulation results?
- What tools do you use to create new 3D grid results from existing 3D grid simulation results?
- What tool do you use to make high-quality dynamic plots of vectors against time?

Copyright © 2017 Schlumberger. All rights reserved.

Summary

In this module, you learned about

- including the development strategy in the Define simulation case process and running a simulation
- creating field/well performance analysis plots using the Results charting and analysis process
- creating thumbnail plots using the Split view functionality
- using the picking mode options and well player
- customizing the data display by creating a chart theme, series styles, and rules
- exporting chart data to Excel and chart images to the clipboard
- visualizing 3D simulation grid results

- creating grid properties and new summary vectors from the existing results using the **Summary calculator** and **3D results calculator**
- making geometrical properties
- creating production bubble plots
- using filters in Petrel
- using the Multi-value probe to quality check the grid static and dynamic properties

NOTES

NOTES

Copyright © 2017 Schlumberger. All rights reserved.

Module 6 History matching and prediction

In this module, you learn how to use the **Development strategy** tool to create history and prediction development strategies. The module exercises are structured to take you through all of the main elements of the **Development strategy** dialog box and its applications within the Petrel reservoir engineering workflow.

Learning objectives

In this module, you learn about the difference between history and prediction runs (strategies). After completing this module, you will know how to:

- make a history development strategy
- create and assign a fault transmissibility multiplier
- import from an OFM project
- import observed data (.vol file)
- make a prediction development strategy
- create a restart run from an existing base case





Lesson 1 Introduction to development strategy

Development strategies describe to the simulator how a field is developed. They specify which wells produce or inject, at what rates and pressures they flow, and what operations are carried out on the wells over time.

Development strategies make it easy to track how the control of a field evolves with time. For example, as new wells are drilled, the target field rates change, wells are converted from producer to injector, and new platforms and manifolds are added.

Development strategies also enable you to apply the same constraints to many wells by using well folders or different constraints for individual wells.

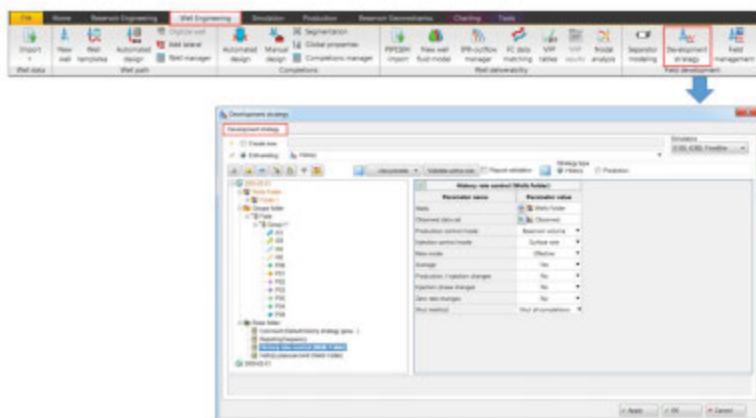


Figure 140. Development strategy dialog box

Simulation studies usually are in two phases: History match and Prediction. History matching allows you to simulate past performance of the reservoir and compare the results with actual historical data. Geological, geophysical, and petrophysical inputs are used to build a reservoir description and, from this data, you can build a simulation model. You then import actual production and pressure information along with fluid model and rock physics functions, run the model, and compare the simulated results with the history.

Sensitivity runs are used to identify which properties have the greatest effect on the simulation results.

Tuning runs are used to modify the properties of the model to improve the match between simulated results and the actual production. When you have an acceptable match, you can switch to a prediction forecast based on the matched history. At this point, you can use your matched model to predict the production response to new wells, new recovery techniques, or changes to existing well operations.

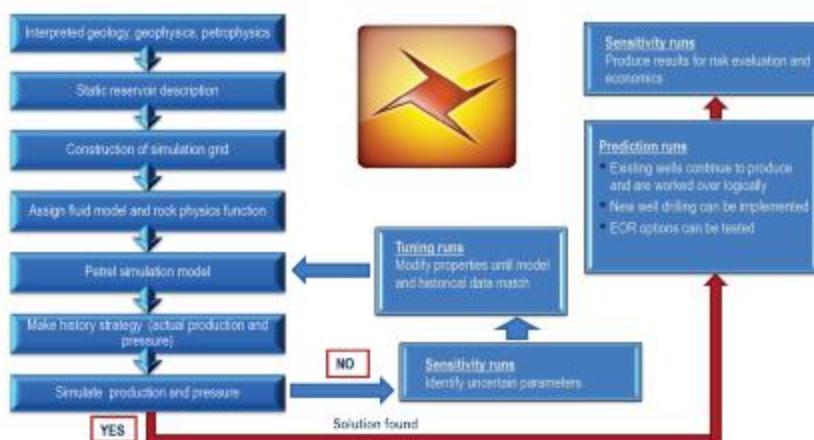


Figure 141. History matching and prediction workflow

There are no golden rules for conducting a history match, but the methodology is well known. (It is beyond the scope of this course to discuss it in detail.)

This course focuses primarily on how to apply Petrel software technology to create a history and prediction strategy as part of the reservoir engineering workflow. There are other courses provided by Schlumberger NExT that examine history matching methodology in more detail.

Copyright © 2017 Schlumberger. All rights reserved.

History and prediction

History:

- validates the model against history
- uses observed rates as well control data
- uses historic events/dates for perforations

Prediction:

- predicts future behavior
- specifies future operating rates or pressures of wells

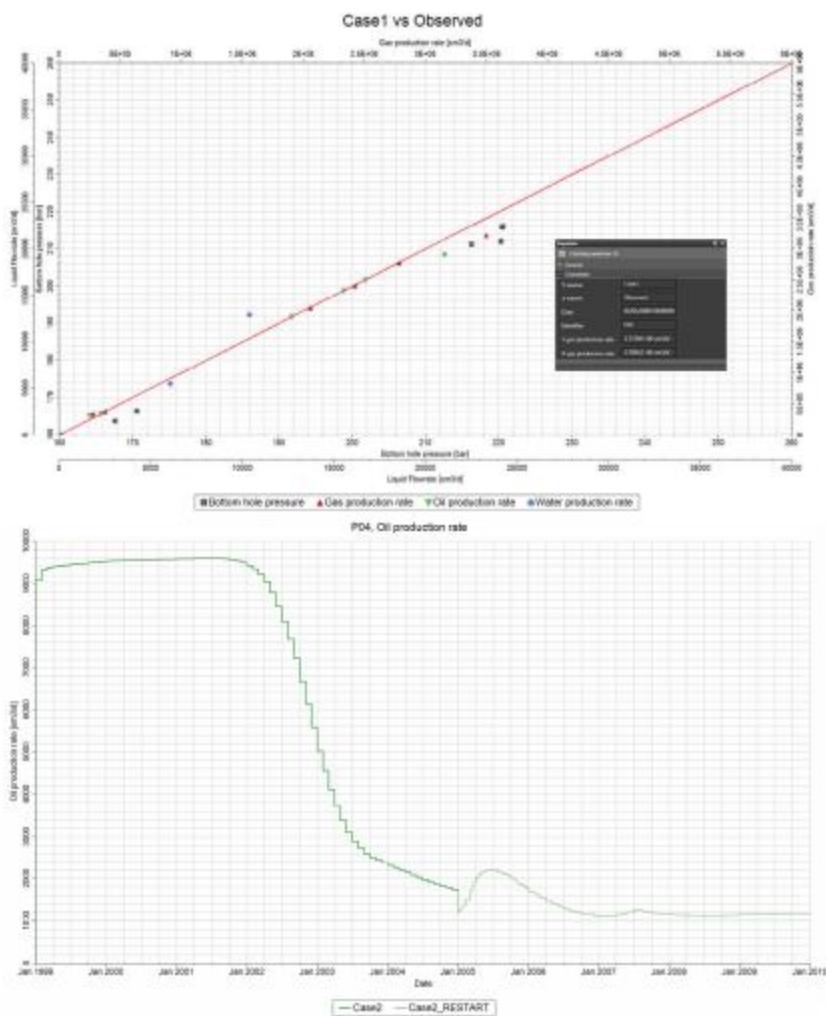


Figure 142. History match and Prediction plots

Input data required for history strategy

There are several types of information that you must have when you are creating a history development strategy:

- Well path (deviation surveys)
- Well historical data (completion intervals and workover events)
- Production/injection data

Well

To know where the wells are located in the simulation model and ensure communication between the reservoir and the surface facilities, the Petrel model must have the well paths (deviation surveys).

Completion events

Completion data typically is required input data to the simulator for a history development strategy. The completion events of each well must be specified in the simulation model by importing wellbore history records such as perforations, squeezes, plugs, and acidization into Petrel. Completion events also can be created in Petrel.

Production data

In a simulation study, production data (oil, gas, and water rates) is input to the history development strategy. Oil production in the simulation model normally is expressed as a monthly profile. However, there is an option in Petrel that allows you to smooth or reconfigure an oil production profile while still maintaining the value of the cumulative production.

You can import the production history directly from the OFM project database. On the **Production** tab, in the **Well data** group, click **Import** and then click **Observed data**. Use the OFM project data (*.ofm) format when you import the data.

Main elements of the Development strategy dialog box

This figure shows the main components of the **Development strategy** dialog box.



Figure 143. Development strategy dialog box

- 1 Toolbar: Used to add dates, wells, groups, and rules to the strategy
- 2 Strategy tree: Shows the timeline, the well folders and group hierarchy, and the rules active at each date
- 3 Rule table: Allows you to enter data for each rule
- 4 Status bar: Displays hints and error messages
- 5 Options to specify the simulator and strategy type
- 6 Strategies presets

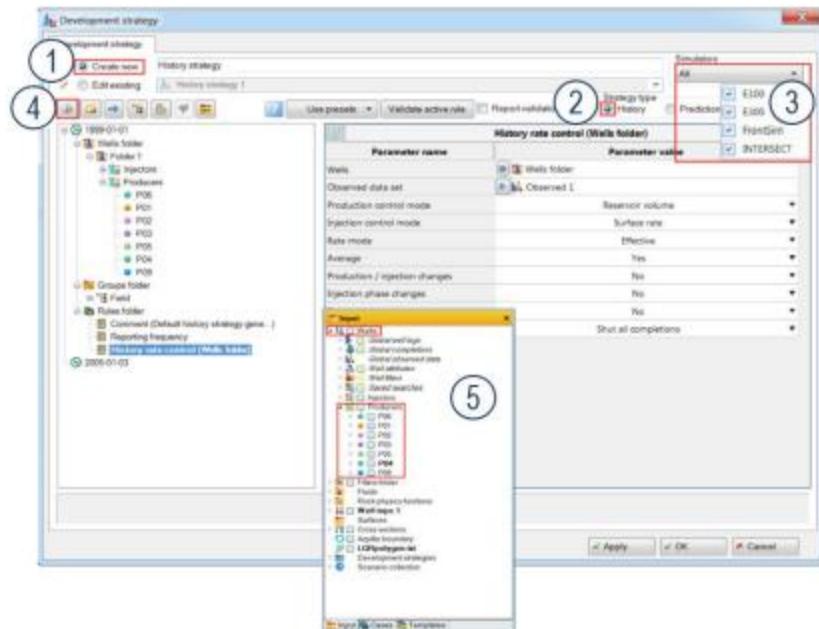
Copyright © 2017 Schlumberger. All rights reserved.

Procedure — Create a history development strategy

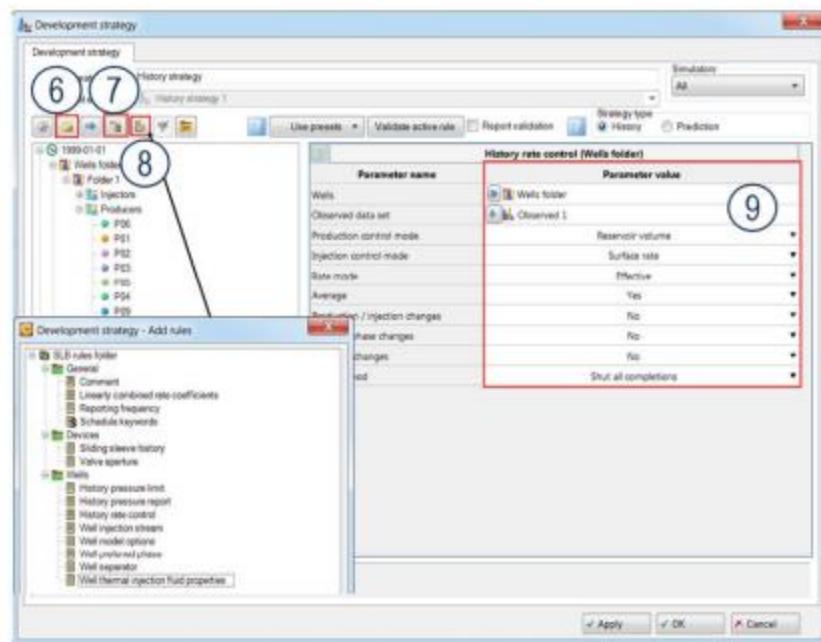


1. Click **Create new**.
2. Select a **Strategy type**, for example, **History**.
3. Select the simulator type.
4. Add control dates to the strategy tree and click **Add a new date** .

5. In the **Input** pane, select wells or a folder of wells and click **Add selected wells or well folders from the input tree** 



6. To organize wells with common controls into folders, click **Add a new user defined folder**  and drag the wells into the folders.
 7. Organize wells into groups for higher-level control.
 8. To add rules, click **Open add rules** 
 9. Edit rule parameters in the Rules table.



When you add wells to the strategy tree, their flow path is analyzed. If the well cannot flow (it is cased, but has no perforations), it is not added. If the well flows up both the tubing and the annulus, two well flows are added to the strategy tree.

Preset strategies

The **Use presets** list offers four defaulted strategies. These strategies are intended as starting points for creating strategies. Usually, they need further editing based on the requirements of the project.

- History strategy: Uses the first observed dataset listed in the Global observed data folder. All wells in the project are added to the strategy. In many cases, no further editing is necessary to make a strategy for reproducing reservoir volume rates for all wells with observed data.
- Empty prediction strategy: Provides a blank strategy, equivalent to when the process is opened the first time.

- Prediction depletion strategy: Sets up a field for production with no injection. All wells are added to the strategy and placed under production group control.

You must set a field group production target, as well as start and end dates. It is best to set the minimum bottomhole pressure and, optionally, the maximum rate limits.

- Prediction water flood strategy: Sets up a field for production with water injection. Group and well rules are set up for group production control and full voidage replacement. Petrel cannot detect which wells are producers and which are injectors, so you must drag producers to the PROD folder and injectors to the INJ folder.

In addition, you must set the field group production target and start and end dates. It is best to set bottomhole pressure limits and, optionally, maximum rate limits on both producers and injectors.

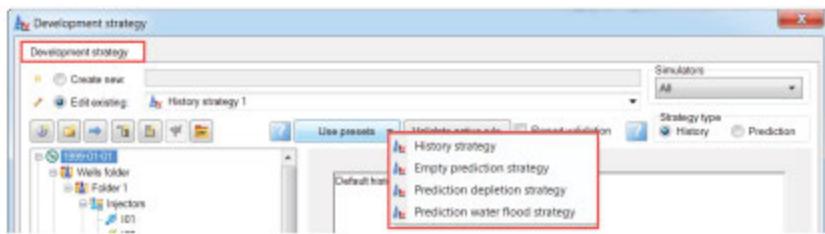


Figure 144. Preset strategies

Two rules normally are added under the default history strategy:
Reporting frequency and History rate control.

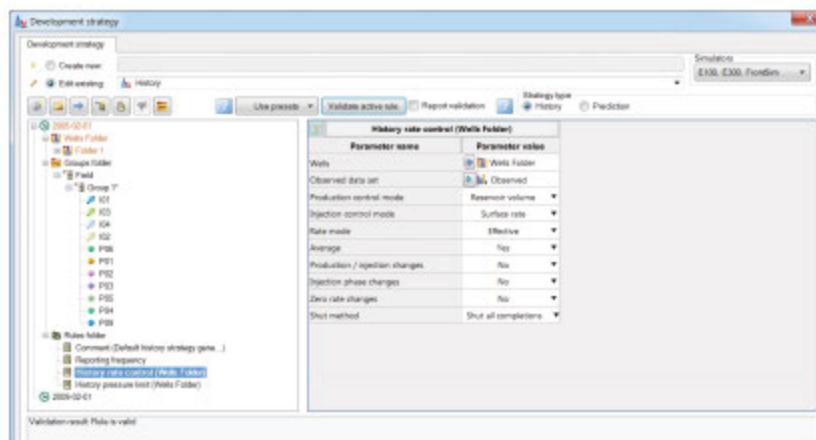


Figure 145. Two default history strategy rules

Default history strategy

A default history strategy:

- sets the start and end dates of the simulation from the first observed data in the project
- adds all wells automatically; wells without history are detected on export to the simulator and ignored
- sets the reporting frequency rule to monthly reports
- uses the recommended default History rate control rules for history matching

When the default strategy has been set up, any of the parameters can be modified.



Procedure — Edit default rules

1. Insert the observed data.
2. Change control modes as required.
3. Edit the settings for reporting frequency.

History rate control (Wells folder)	
Parameter name	Parameter value
Wells	Wells folder
Observed data set	Observed 1
Production control mode	Reservoir volume Reservoir volume
Injection control mode	Water Water
Rate mode	Gas Gas
Average	Liquid Liquid
Production / injection changes	Combined Combined
Injection phase changes	Reservoir volume Reservoir volume
Zero rate changes	Bottom hole pressure Bottom hole pressure
Shut method	Two Two
Zero rate changes	No No
Shut method	Shut all completions Shut all completions

Reporting frequency	
Parameter name	Parameter value
Reporting frequency	1 Months 1 Months
Add event times	No No

Between control changes, you can output regular reports from the simulator by changing the settings in the reporting frequency rule. By default, this rule is added to the first date of every strategy. It can be copied to later dates to change the settings. For example, you can report yearly in the early part of a history match and monthly in the most recent year.

Average of historical production data

If **Yes** is selected for the **Average** parameter in the History rate control rule, historical production data can be averaged to the specified reporting frequency .

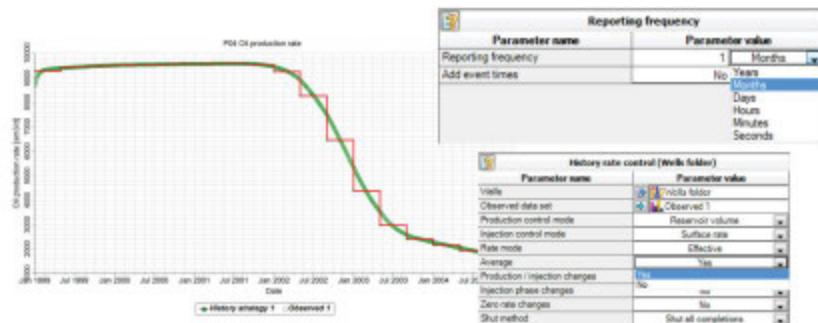


Figure 146. Averaging historical production data

Comparison of observed data and a created history strategy

Imported data is averaged over the report step length selected in the Reporting frequency rule in the Development strategy. To see how the averaged data compares with the observed data, plot the observed data and the history development strategy.

Open the **Result charting and analysis** dialog box and select to view:

- oil production rate vector in the **Properties** pane
- identifier (well) in the **Primary identifiers** pane
- development strategy in the **Sources** pane
- observed data in the **Sources** pane

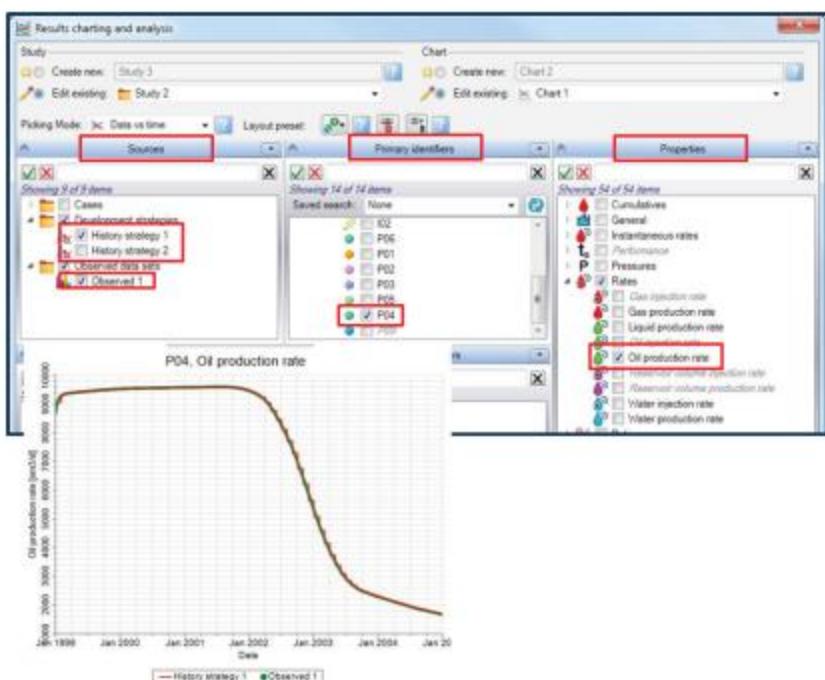


Figure 147. Comparing observed data and a created history development strategy using the Results charting and analysis dialog box

Copyright © 2017 Schlumberger. All rights reserved.

OFM Data Connector

OFM Data Connector allows you to import well data and completion information, as well as production and analytical forecasts from OFM to Petrel. You also can export simulation results, regular surfaces, and well models (deviation data and well configurations) to OFM and creates the data files needed for import back to Petrel.

Exercises — Create a history development strategy

In these exercises, you learn how to create a history development strategy using the **Development strategy** tool. You also practice importing some of the input data (completion event, observed data, wells) required for the History development strategy directly from the OFM database.

Workflow

1. Import observed data.
2. Create a history development strategy.
3. Visualize the imported observed data.
4. View the development strategy data.
5. Define a simulation case using the created history development strategy.

Data

Use the project `History_matching_and_Prediction_exercise.pet` in the `Dataset\Projects\Module - 6 History matching and Prediction` folder.



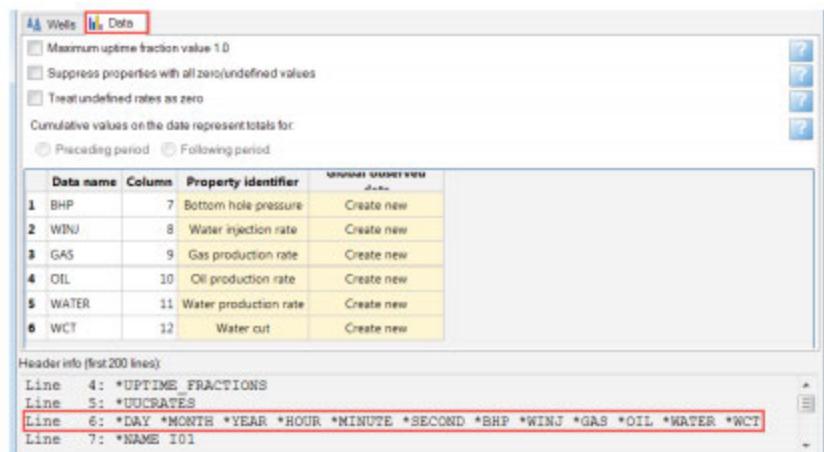
Exercise 1 Import observed data

In this exercise, you import historical production and injection data for the wells in the project.

1. On the **Well Engineering** tab, in the **Well data** group, click **Import** and then click **Observed data**.



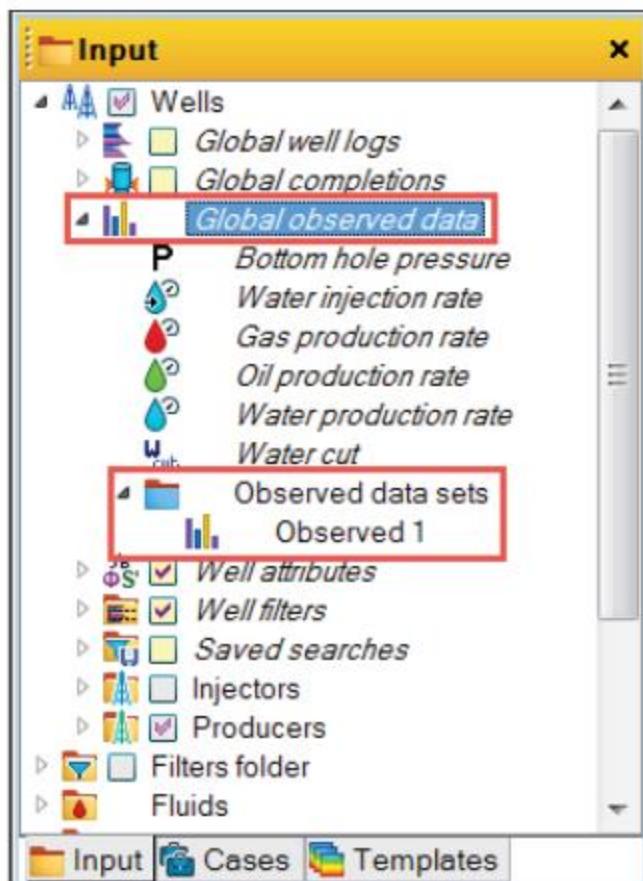
2. Select the file Observed.vol from the ImportData \Observed data folder. Make sure that the file type is Well observed data (ASCII)(*.vol). Click **Open**.
3. In the **Import observed data** dialog box, ensure that the well names in the file match the correct wells in the project.
4. Click the **Data** tab and verify that the column numbers are correct for the data that you are importing and that appropriate property identifiers are selected.



5. Click **OK**.

The observed data is stored in the Global observed data subfolder in the Wells folder.

6. In the **Input** pane, click **Global observed data** and then click **Observed data sets**.



7. Right-click **Observed 1** and click **Spreadsheet** to view the imported observed data.

Copyright © 2017 Schlumberger. All rights reserved.

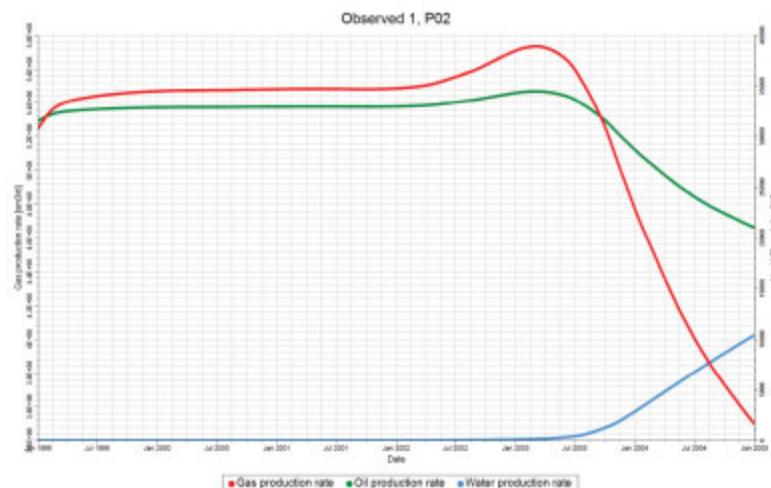
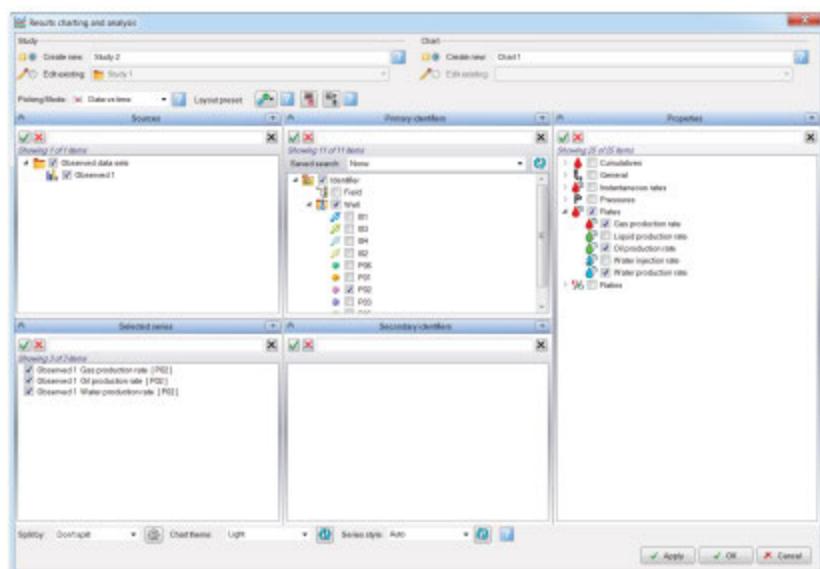
Exercise 2 View the imported observed data



In this exercise, you use the Results charting and analysis process to view the observed data that you imported.

1. Open the **Results charting and analysis** dialog box.
2. In the **Sources** pane, click **Observed data sets** and then click **Observed 1**.
3. In the **Primary identifiers** pane, click **Well** and select well P02.

4. In the **Properties** pane, select Oil, Water, and Gas production rates.



Copyright © 2017 Schlumberger. All rights reserved.

5. Select other wells in one of these ways:

- On the **Window** toolbar, click **Move forward through primary identifiers** .
- On the **Charting** tab, use the tools in the **Player** group. To navigate through wells in the Well folder, click **Previous well** and **Next well**.



6. Click **Jump to a well**.

7. In the **Jump to identifier** dialog box, enter **P** in the **Filter** field to filter for all the wells that start with the letter P.
8. Select any specific well and click **Apply** to display its observed data in the **Charting** window.
9. Click **Close** to close the **Jump to identifier** dialog box.



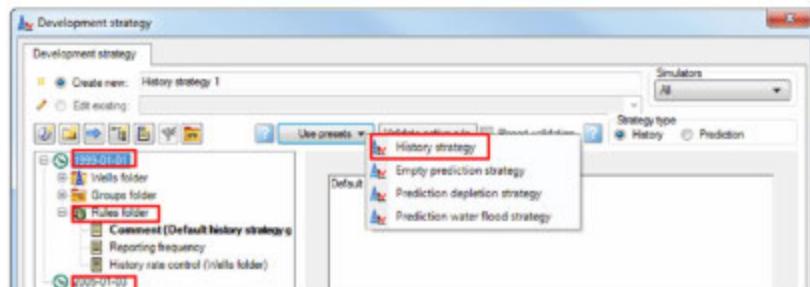
Copyright © 2017 Schlumberger. All rights reserved.

10. In the **Results charting and analysis** dialog box, perform these steps:
 - In the **Create new data** field in the **Study** section, change the name of the study to HM.
 - In the **Create new data** field in the **Chart** section, change the name of the chart to Production_plots.
11. Click **OK** to save the plot.

Exercise 3 Create a history development strategy

In this exercise, you create a history strategy using the default history strategy option in the **Development strategy** dialog box. The history strategy preset option automatically adds the well data and observed dataset into the dialog box.

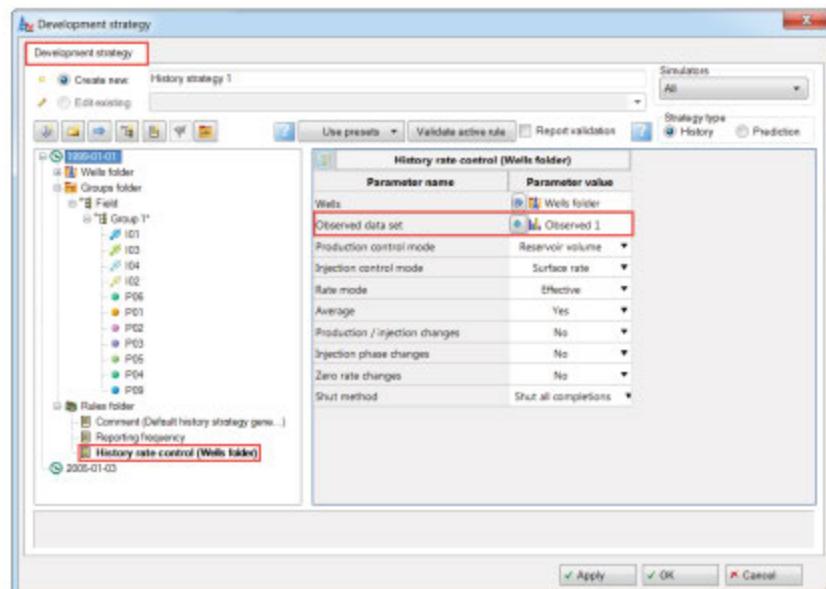
1. On the **Well Engineering** tab, in the **Field development** group, click **Development strategy**.
2. Click **Create new**.
3. In the **Use presets** list, click **History strategy**.



Observe that the historical dataset and the wells have been inserted automatically. The start date 1999-01-01 and end date 2005-01-03 are extracted from the observed dataset.

4. In the Rules folder in the **Strategy** tree, complete these steps:
 - a. Select the History rate control (Wells folder) rule.
 - b. Make sure that the **Production control mode** is set to **Reservoir volume**.

- c. Leave the report frequency for the Reporting frequency rule set to the default every (1) month.



5. Click **OK** to save the History strategy.

The new History strategy is saved in the new Development strategies folder in the **Input** pane.

Copyright © 2017 Schlumberger. All rights reserved.

Exercise 4 View the development strategy data



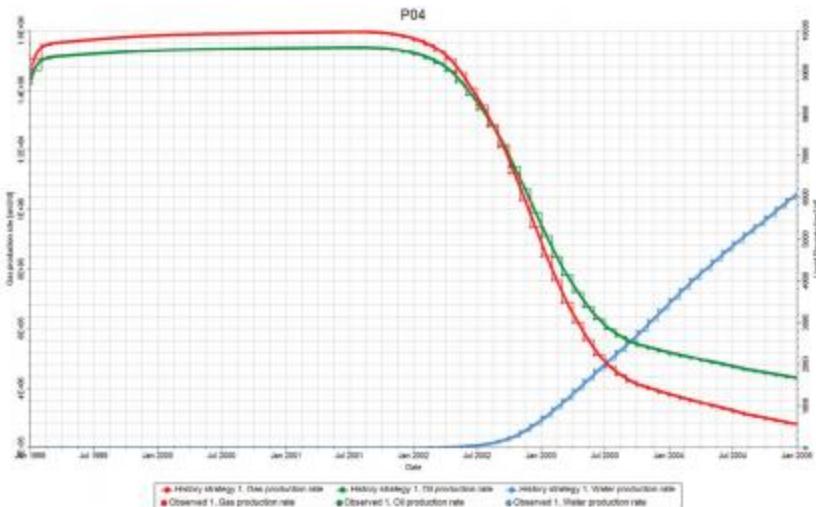
In this exercise, you examine and compare the History strategy that you just created with the observed data that you imported earlier.

1. Open the **Results charting and analysis** tool.



2. In the **Sources** pane, click **Development strategies** and select **History strategy 1**.

You now should see your History strategy data and your Observed data.

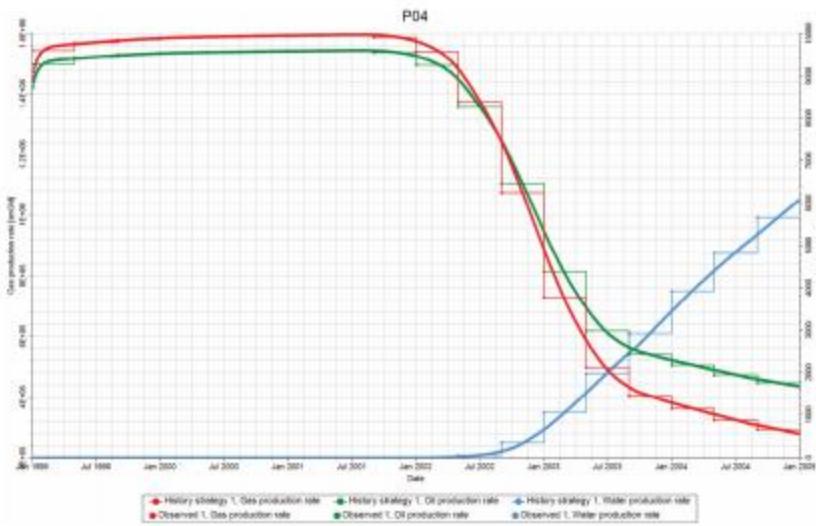


3. View the History strategy and Observed data for other wells. On the **Window** toolbar, click **Move forward through the primary identifiers**



4. Click **OK** to update the existing Production_plot study.
 5. Reopen the **Development strategy** dialog box, change the **Reporting frequency** rule to every four months, and click **Apply**.

What did you observe from the active plot?



6. In the **Reporting frequency** rule in the **Development strategy** dialog box, reset the previous reporting frequency to one month.
7. Save the project. You use it later.



Exercise 5 Import an OilField Manager (OFM) project into Petrel

The **OFM Data Connector** dialog box helps you manage the import of well data and completion information, as well as production and analytical forecasts from OFM to Petrel. The goal of this exercise is for you to practice importing an OFM project into Petrel.

1. Create a new Petrel project and name it **OFM_import**. Save the project in the **Student** folder.
2. On the **Well Engineering** tab, in the **Well data** group, click **Import** and then click **Observed data**.

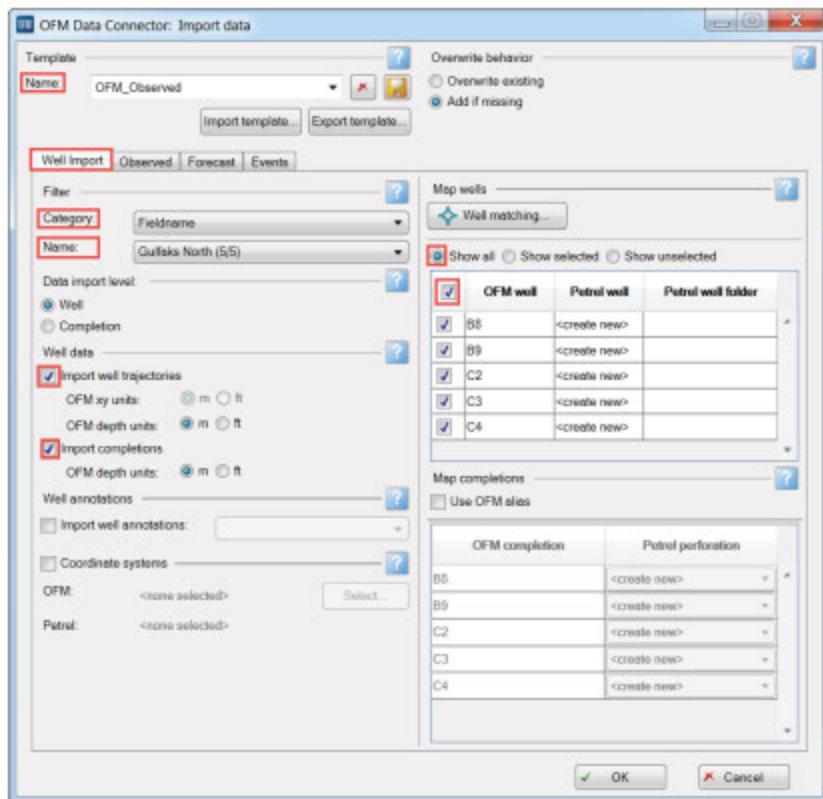


3. Select **OFM project data (*.ofm)** from the **File of type** list in the **Import file** dialog box.
4. Browse to the file **Gulfaks_OFM2012.ofm** in the **Dataset \ImportData\Observed_data\OFM_Project** folder and click **Open**.

The **OFM Data Connector: Import data** dialog box opens.

5. When the coordinate reference system warning message dialog box opens, click **Continue spatially unaware**.
6. In the **Template** section, enter **OFM_Observed** in the **Name** field.
7. On the **Well Import** tab, filter by **Category** and **Name** by selecting **Fieldname** as category and **Gulfaks North (0/5)** as the name.
8. In the **Well data** section, select the **Import well trajectories** and **Import completions** check boxes. Leave the units in meters.

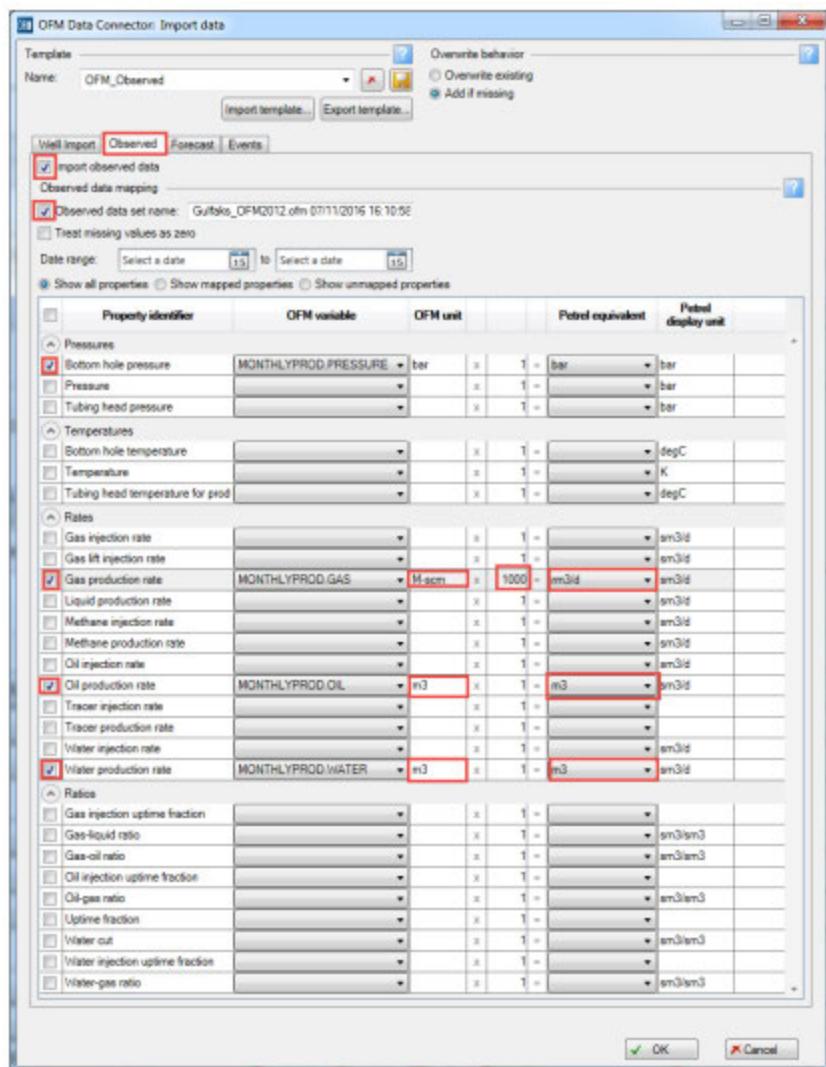
9. Skip the Coordinate system because it is not required for this exercise.
10. Ensure that **Show all** is selected.
11. Select the check box next to the **OFM well** column to select all wells to be loaded.



Copyright © 2017 Schlumberger. All rights reserved.

12. Click the **Observed** tab and complete these steps:
 - Select **Import observed data** and **Observed data set name** check boxes (`Gulfaks_OFM2012.ofm`).
 - Ensure that **Show all properties** is selected to show the type of properties to display in the grid.
13. In the **Property identifier** column, select the **Bottom hole pressure** check box.

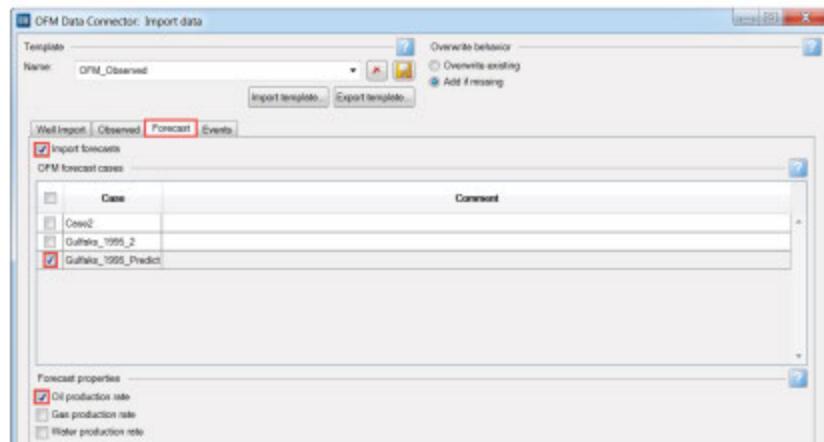
14. In the **OFM variable** column, select **MONTHLYPROD.PRESSURE** from the list.



15. In the **Property identifier** column, select the check boxes for Gas, Oil, and Water production rates.

16. In the **OFM variable** column, assign these variables to the corresponding selected rates.
 - Gas production rate: MONTHLYPROD.GAS (ensure that you select the Petrel equivalent unit to be sm3/d)
 - Oil production rate: MONTHLYPROD.OIL (m3)
 - Water production rate: MONTHLYPROD.WATER (m3)
17. Click the **Forecast** tab and complete these steps:
 - Select **Import forecasts**.
 - In the **Case** column, select **Gulfaks_1995_Predict**.
 - In the **Forecast properties** section, select **Oil production rate**.

This option allows you to import a production forecast from an Decline curve analysis. This analysis allows you to compare the imported analytical forecast with the simulation forecast.



18. Click the **Events** tab and select the **Import event data** check box.



19. Click **OK** to import well trajectories, completion, events, observed data, and OFM forecast data.

Copyright © 2007 Schlumberger. All rights reserved.

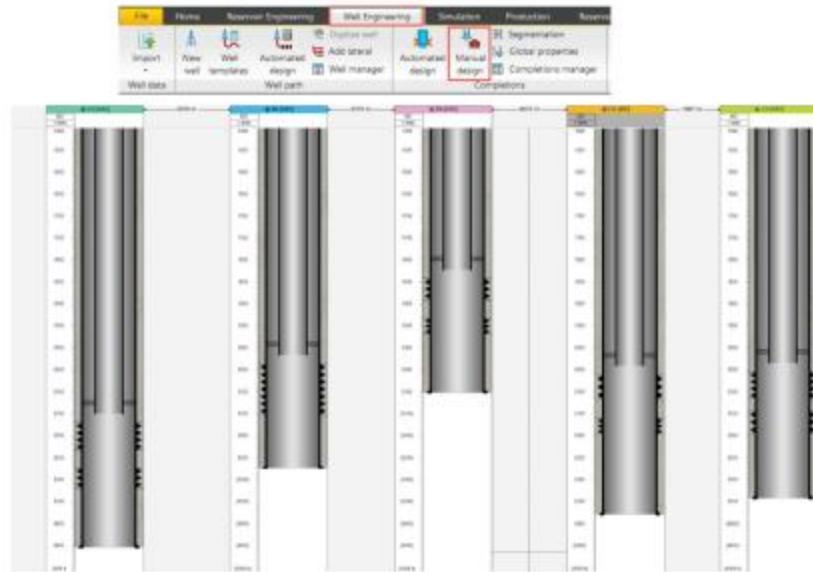
Exercise 6 View the OFM project imported in Petrel



In this exercise, you visualize all imported OFM data using visualization windows and results analysis tools in the Petrel environment.

1. Open a new **3D window**.
2. In the **Input** pane, click **Wells**.
3. Select the check box next to the OFM folder to display the wells in the **3D window**.
4. On the **Window** toolbar, click **View all** so that all of the displayed objects are visible in the active **3D window**.
5. Open the **Completion design Tool Palette**. On the **Well Engineering** tab, in the **Completion** group, click **Manual design**.

- To display the wells and completions in the **Well section window**, select the wells from the Well folder and completions from the Global completion folder.



Copyright © 2017 Schlumberger. All rights reserved.

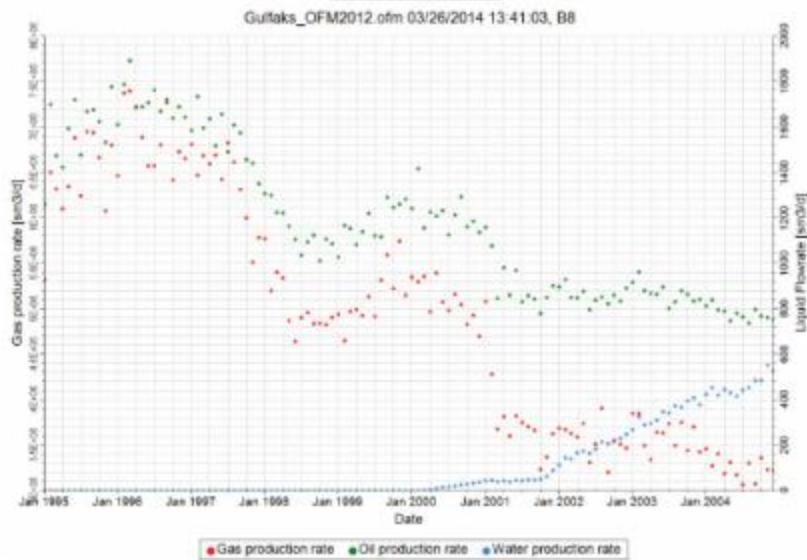
Exercise 7 View imported OFM observed data

In this exercise, you view the imported observed data using the Result charting and analysis process.



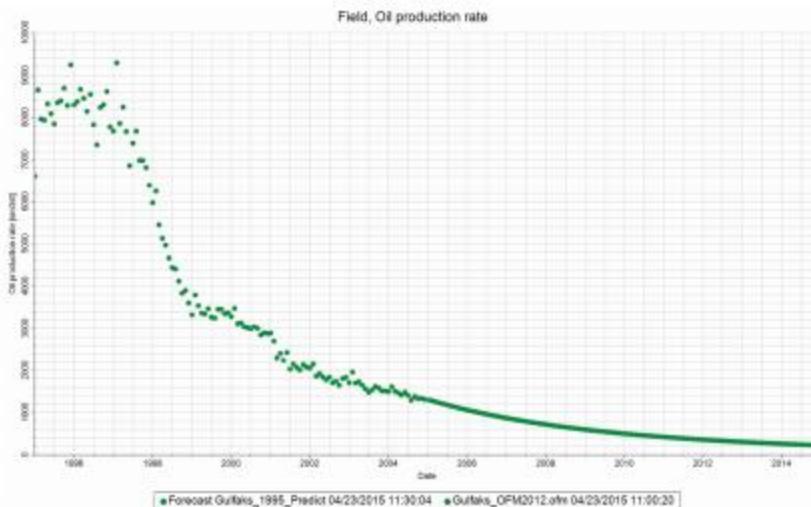
- Open the **Results charting and analysis** tool.
- In the Observed data sets folder in the **Sources** pane, select **Gulfaks_OFM2012.ofm**.
- In the **Primary identifiers** pane, click **Well** and select well B8.
- In the **Properties** pane, select Oil, Water, and Gas production rates.

You now should see the **Gulfaks_OFM2012.ofm** imported observed data displayed in your **Charting** window.



5. To view the imported Observed data for B9, on the **Charting** window tab, in the **Player** group, click **Next well**.
6. In the Rates folder, in the **Properties** pane, clear the Gas and Water production rates.
7. In the **Primary identifiers** pane, clear well B9 and then select **Field**. You now have the field oil production rate on the chart.
8. In the Observed data sets folder in the **Sources** pane, select **Gulfaks_1995_Predict** to display the imported field oil forecast from OFM.

Copyright © 2017 Schlumberger. All rights reserved.



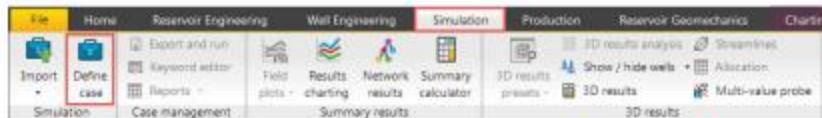
9. Click **OK** to save the chart and close the dialog box.
10. Save the project in the Student folder.

Exercise 8 Set up a simulation case and run the simulation

In this exercise, you create and run a simulation case using the history strategy that you created previously.

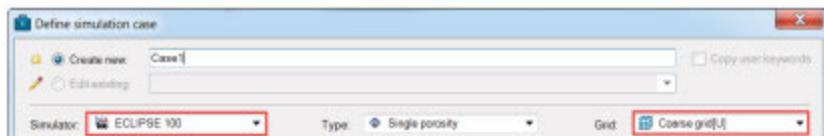
Continue to use the project History_matching_and_Prediction_exercise.pet in the Dataset\Projects\Module - 6 History matching and Prediction folder.

1. Open the **Define simulation case** dialog box.

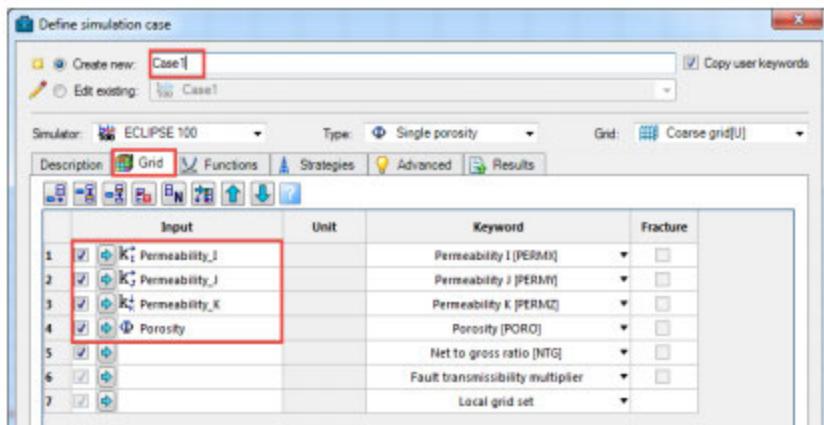


2. Click **Create new** and enter Case1 as the case name.
3. Select **ECLIPSE 100** as the **Simulator**.
4. Select **Coarse grid[U]** as the **Grid**.





- On the **Grid** tab, insert the Permeability_I, Permeability_J, Permeability_K, and Porosity properties from the **Models** pane for the Coarse grid[U] into the Input data fields in the **Define simulation case** dialog box.



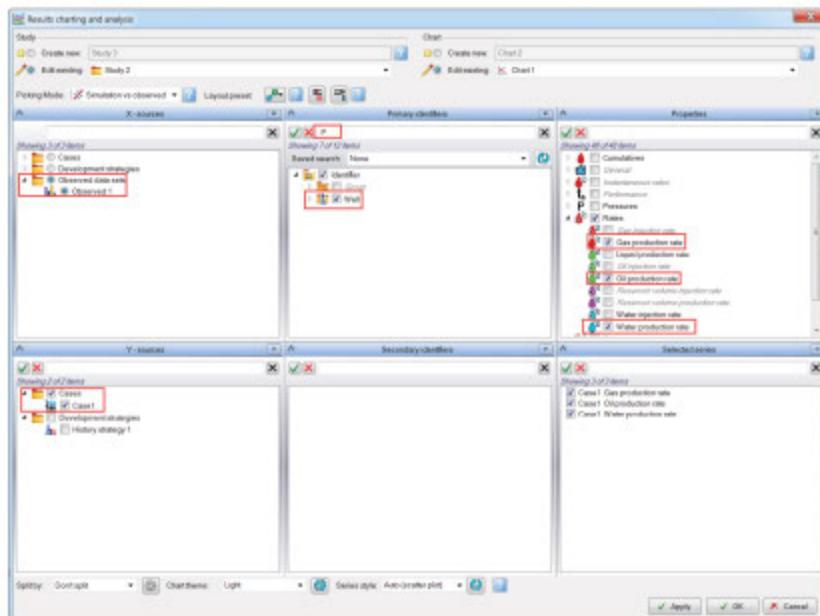
- Use Saturation 1 and Rock compaction 1 as inputs for the rock physics properties.
- Insert the initial condition from Light oil + gas from the **Input** pane in the Fluid folder.
- On the **Strategies** tab, add a row to the table. Click **Append item in the table**.
- In the **Input** pane in the Development strategies folder, select the History strategy that you created previously and insert it.
- Save the project and click **Run** to run your simulation.



Exercise 9 Plot a simulation versus observed data for the run

Petrel uses crossplots (Simulation vs. observed) to analyze the history match quality of your simulation cases. In this exercise, you learn how to use a Simulated vs. Observed plot to view simulated and observed results.

1. Open the **Results charting and analysis** tool.
2. In the **Picking mode** list, select **Simulation vs observed**.
3. In the **X-sources** pane, select **Observed 1** in the Observed data sets folder.
4. In the **Y-sources** pane, select the simulation case **Case1**.

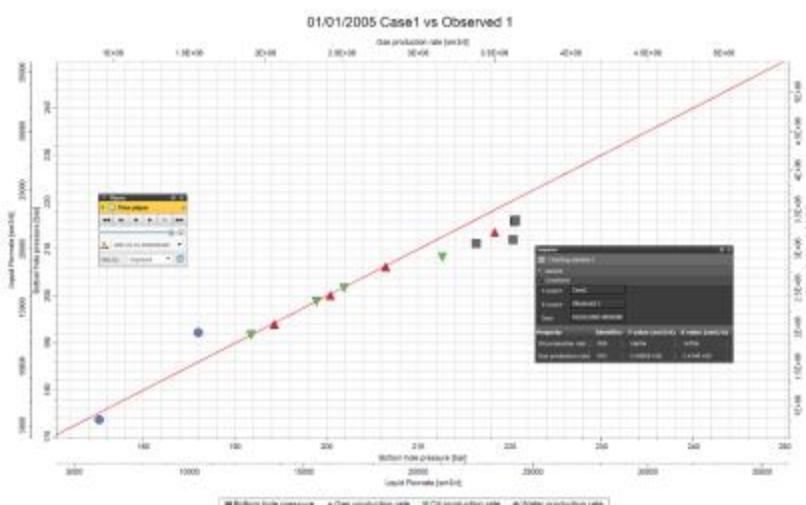


5. In the **Primary identifiers** pane, enter **P** in the **Filter** field to filter for all the wells that start with the letter P and turn on the **Well** folder.
6. In the **Properties** pane, complete these steps:
 - a. In the **Rates** folder, select Oil, Water, and Gas production rates.
 - b. In the **Pressure** folder, select **Bottom hole pressures**.

You now should see your simulated and observed data plotted in the **Simulation vs. observed Charting** window.

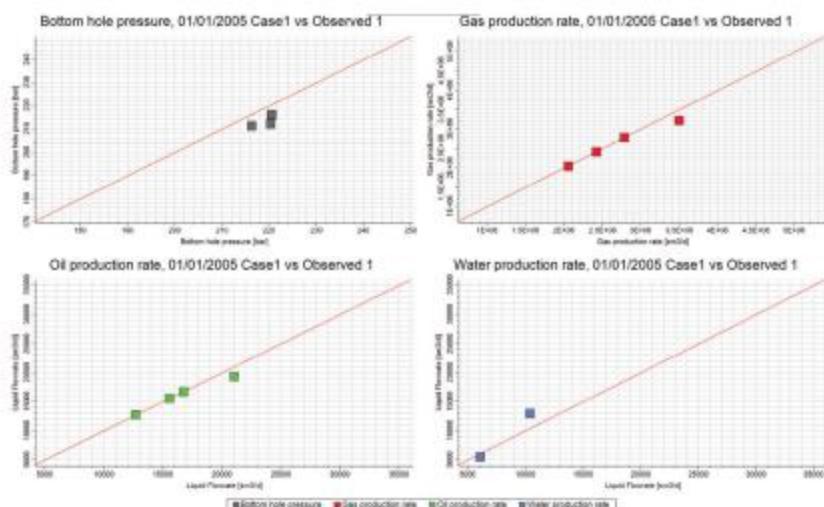
7. Open the **Inspector** and use it to analyze the plotted results. Click any data point and compare the X and Y values of the property.

To select different report steps, use the **Time player**.



The closer the data point is to the red line, the better the history match is for the property.

8. Split the plotted results by property. Right-click in the **Charting** window, click **Split by**, and then click **Property**.



Lesson 2 Fault transmissibility multipliers



The Fault transmissibility multiplier is another important parameter that you can modify to improve a history match. Petrel can assign a constant value to a fault transmissibility multiplier. Petrel also has a more advanced tool that allows you to model fault properties based on the geological grid properties. The added value of this functionality is that it enables you to assign variable fault transmissibility multipliers on a fault plane based on the geology instead of the traditional practice of assigning a constant across the whole fault face. You also can scale the fault transmissibility multipliers during history matching.

You assign only a fault transmissibility multiplier to the faults to model cross fault flow. You do not alter the geometry or throw of the faults.

Transmissibility Multipliers (TM)

Flow between grid cells in a simulation model is computed based on the grid cell transmissibility. The grid cell transmissibility depends on

permeability and the geometry of the cells. In a simulation model, a fault is represented as the cell face between two adjacent grid cells.

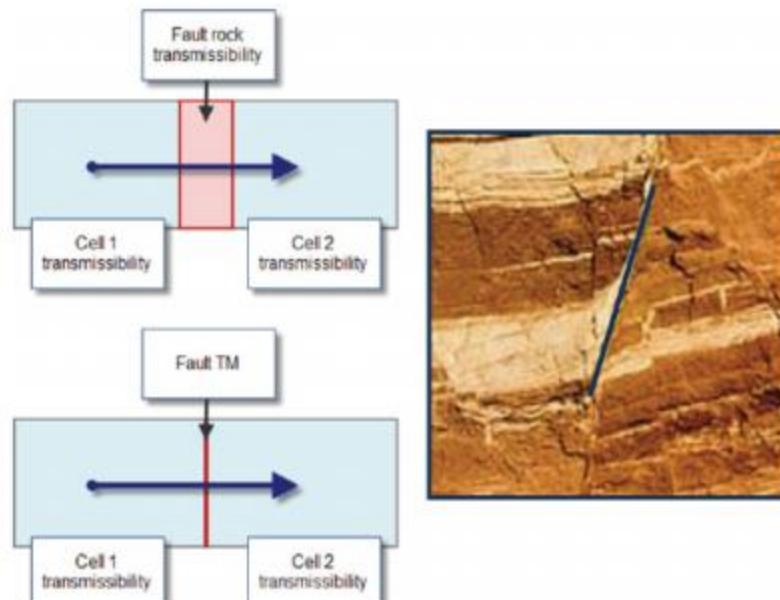


Figure 148. Fault transmissibility illustration

The flow between grid cells in a simulation depends on the transmissibility of the grid cell. To include the transmissibility of the fault, add a transmissibility multiplier to the grid cell face that represents the fault.

How the TM is computed

To compute the TM, the transmissibility between the centers of the two cells on either side of the fault is computed first without taking the fault into account. Then, the transmissibility between the two cells is recomputed, this time taking into account the permeability and thickness assigned to the fault face cell.

This second transmissibility is divided by the first transmissibility to give the transmissibility multiplier. The fault transmissibility multiplier usually is a number from 0 to 1. A fault transmissibility multiplier of zero means that the fault is sealing and there is no cross-fault flow.

There are two approaches to define the impact of fault-related flow retardation in reservoir simulation:

- Assign a constant value to a fault plane.
- Generate a variable fault transmissibility multiplier using geology-based calculation methods:
 - Clay prediction
 - Clay to permeability
 - Fault displacement

Structural and fault analysis dialog box

To access the **Structural and fault analysis** dialog box, on the **Reservoir Engineering** tab, in the **3D and fault properties** group, click **Fault analysis**

 **Fault analysis**. This dialog box gives you access to all settings necessary to compute fault transmissibility multipliers. There are several settings/input required.

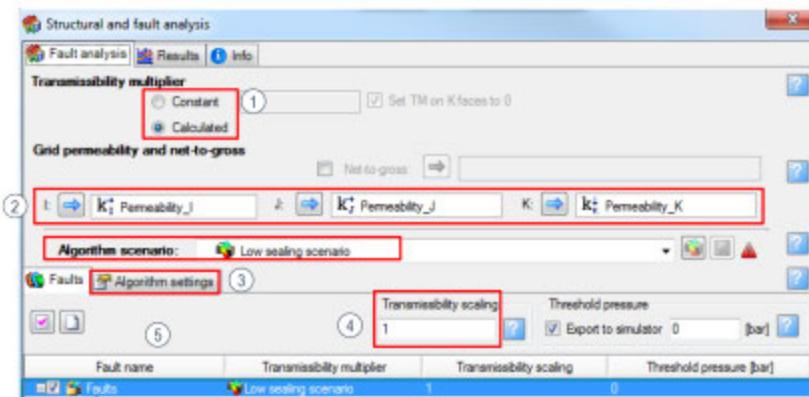


Figure 149. Structural and fault analysis dialog box

- 1 **Transmissibility multiplier:** The multipliers can be a constant number equal to or larger than zero (0 is closed and 1 is open to flow), or they can be computed based on fault properties. Click which option to use.
- 2 **Grid permeability and net-to-gross:** Insert  the 3D grid properties that you plan to use for your simulation model. The simulation grid properties are used to compute the permeability of the faults, which again is used to compute the TM.

- 3 **Algorithm settings:** This list is used to select different sealing scenarios.
- 4 **Transmissibility scaling:** The fault transmissibility is scaled by the number that you enter here.
- 5 **Faults tab:** Use this tab to select the faults for which to compute TM.

The computed fault transmissibility multiplier (TM) is added to the Fault Properties subfolder of the Faults folder. Also, the intermediate properties that were calculated to use as input to the TM calculations are stored in the same folder and can be visualized on faults in the **3D window**.

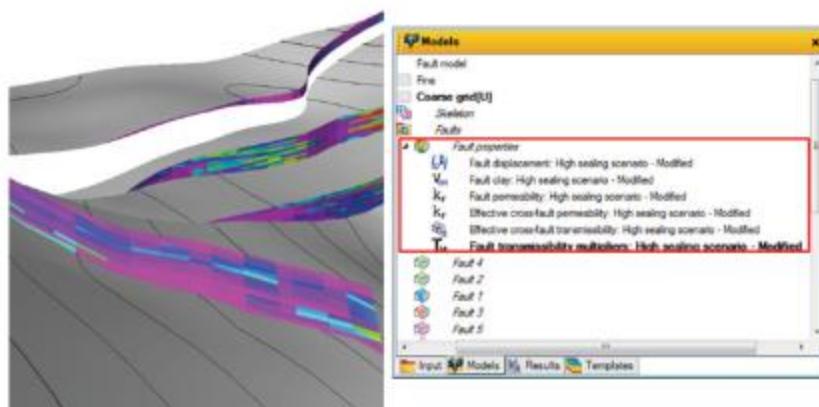


Figure 150. Computed fault transmissibility multipliers (TM) are stored in the Fault properties folder in the Models pane

Copyright © 2017 Schlumberger. All rights reserved.

TM tuning during history matching

The Transmissibility scaling field in the **Structural and fault analysis** dialog box allows you to scale the transmissibility without altering the geological property distribution. This feature can be useful when history matching a model.

You can use this feature to model a sealing fault ($TM=0$) or variable sealing fault using values between zero and one (where TM is not a

constant across the whole fault face). You can assign a different TM to different faults.

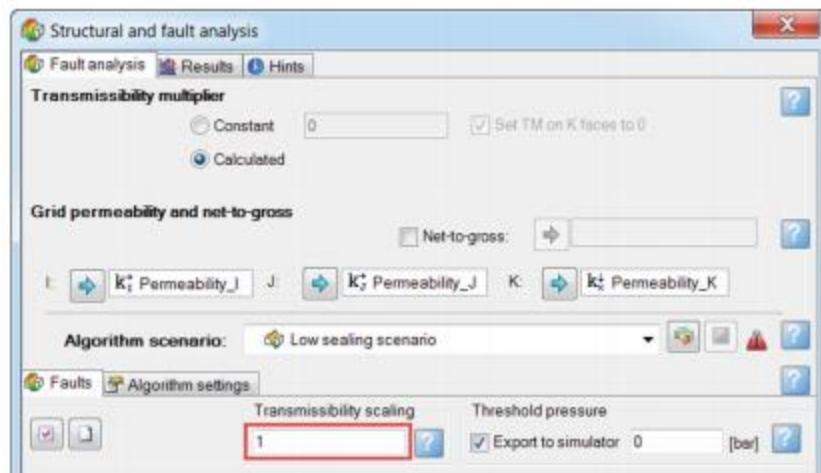


Figure 151. Fault Transmissibility scaling field in the Fault analysis dialog box

Copyright © 2017 Schlumberger. All rights reserved.

The transmissibility scaling:

- scales the value of the fault transmissibility that is applied to each fault
- can be applied only to calculated fault transmissibility multipliers
- scales the permeability and thickness of the faults proportionally
- is not a fault transmissibility multiplier

Fault transmissibility multiplier in a simulation case

The calculated fault transmissibility multiplier must be inserted into the **Define simulation case** dialog box on the **Grid** tab to be used in the simulation .

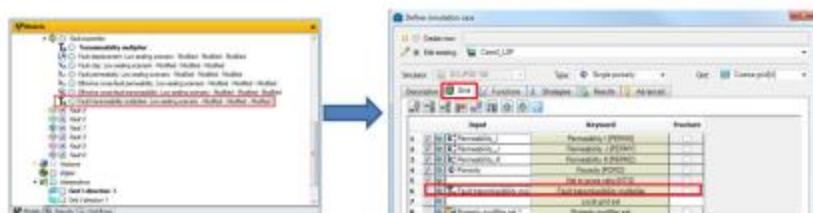


Figure 152. Define simulation case dialog box, Grid tab

Exercises — Assign transmissibility multipliers to faults

In these exercises, you practice how to use the **Structural and fault analysis** dialog box to compute transmissibility multipliers.

Workflow

1. Use Structural and fault analysis to assign transmissibility multipliers to faults.
2. Include the transmissibility multipliers in a simulation case.
3. View results of the simulation case.

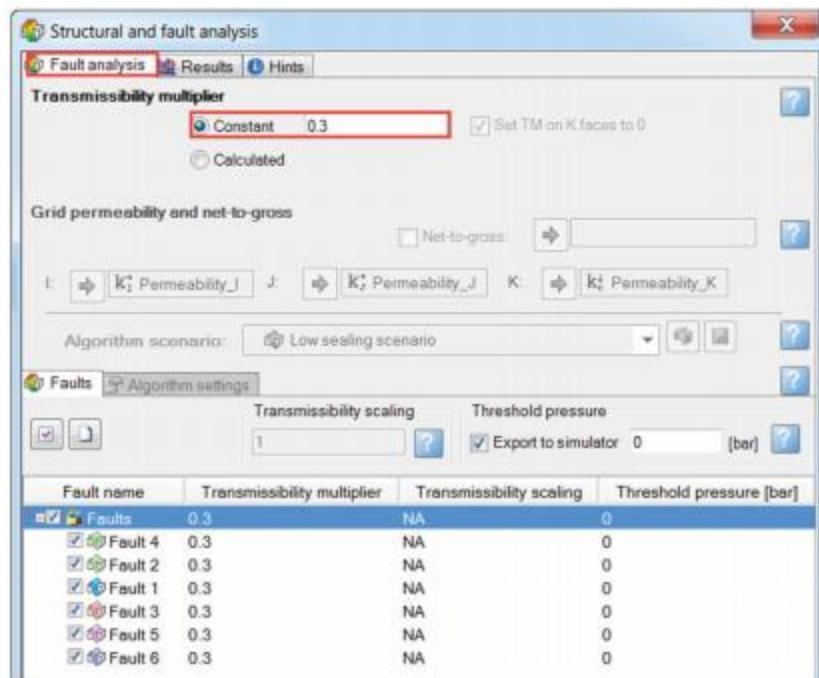
Data

Use the project History_matching and Prediction_exercise.pet in the Dataset\Projects \Module - 6 History matching and Prediction folder.

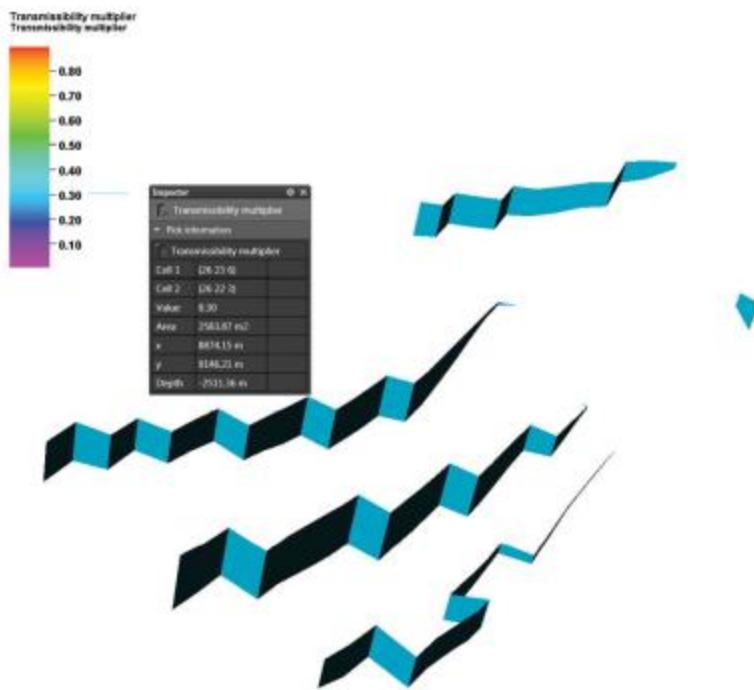


Exercise 1 Use the Faults analysis dialog box to assign transmissibility multipliers to faults

1. In the **Models** pane, activate the Coarse grid[U].
2. Open the **Structural and fault analysis** dialog box by clicking the **Fault analysis** button.
3. In the **Structural and fault analysis** dialog box, click **Constant** in the **Transmissibility multiplier** section and enter a constant multiplier of 0.3.
4. Click **OK**.



5. In the **Models** pane for the Coarse grid[U] grid, click **Faults** and then click **Fault properties**.
6. Open a **3D window** and select the **Fault properties** folder check box to display the fault transmissibility multipliers.
7. Use the **Inspector** to confirm that the assigned Fault transmissibility multiplier is 0.3 for all the faults.



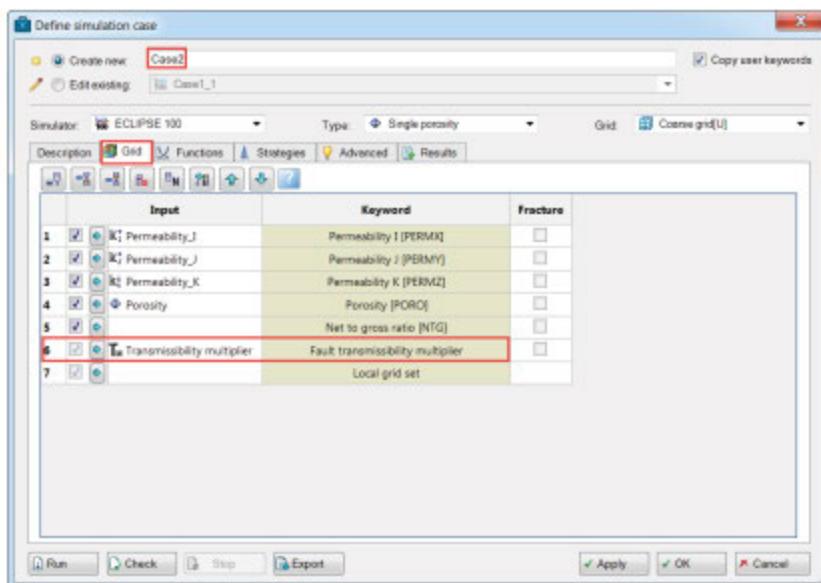
Copyright © 2017 Schlumberger. All rights reserved.



Exercise 2 Include fault transmissibility multipliers in a simulation case

In this exercise, you include the fault transmissibility multiplier in a simulation case.

1. Open the **Define simulation case** dialog box.
2. Click **Create new** and name the new case **Case2**.
3. Click the **Grid** tab and insert the Fault transmissibility multiplier from the **Fault properties** folder in the **Models** pane. Ensure that the corresponding option is selected in the Keyword column.



4. Click **Apply** to save the case.

5. Click **Run**.

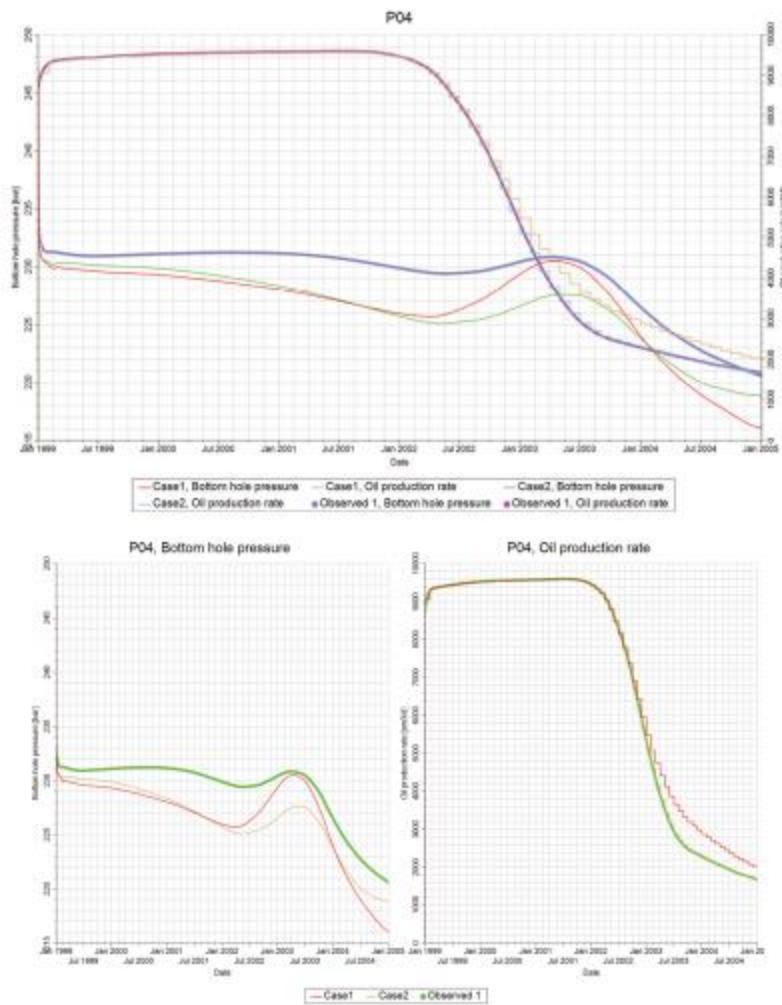
6. Close the dialog box.

Exercise 3 Create a plot of the oil rate for well P04 for the two simulation cases

In this exercise, you learn how to compare different simulation case results with the observed data using the Date vs. time picking mode. For this exercise, you compare the effect of including the fault transmissibilities multipliers in the simulation Case2 focusing on well P04.



1. Open the **Results charting and analysis** dialog box.
2. In the **Sources** pane, select to plot observed data and the simulation cases Case1 and Case2.
3. In the **Primary identifiers** pane, expand **Well** and select **P04**.
4. In the **Properties** pane, select **Oil production rate** and **Bottom hole pressure**.
5. Right-click in the chart window, click **Split by**, and then click **Property**.



Copyright © 2017 Schlumberger. All rights reserved.

Observe the effects of fault transmissibilities multipliers on the Bottom hole pressure and oil production match for Case2 where the TM is set to 0.3.

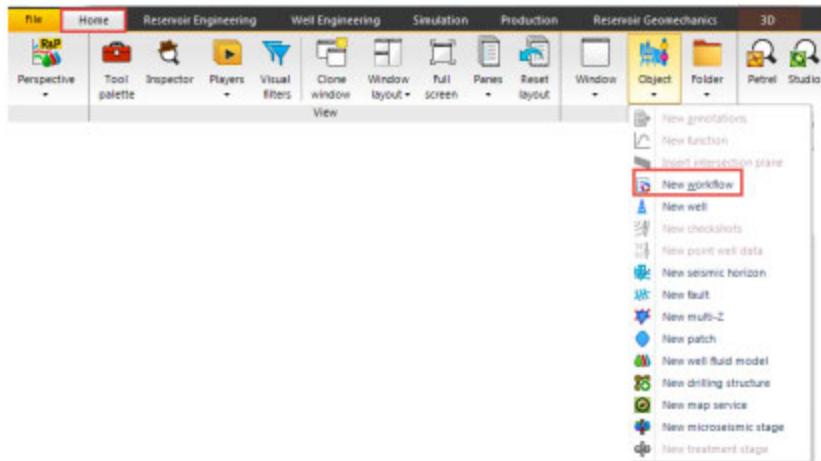
Lesson 3 Workflow editor and its applications



A workflow simply describes the sequential process steps of a project or task in the Petrel modeling environment. A workflow can be sequential, with each step contingent upon completion of the previous one. Building a workflow is a unique capability enabled in Petrel using the **Workflow editor**.

The **Workflow editor** is a feature in Petrel that allows you to create multiple workflows for a project and store them for later use. You can modify and rerun each workflow.

To access the **Workflow editor**, on the **Home** tab, in the **Insert** group, click **Object** and then click **New workflow**.



Copyright © 2017 Schlumberger. All rights reserved.

Figure 153. Accessing the Workflow editor

You can use the **Workflow editor** for a variety of tasks. It provides a simple interface for updating an existing 3D grid with properties or an existing simulation case. It also enables the coupling of the static and dynamic models during uncertainty and optimization studies.

When you create a workflow, you can store it in your project or transfer it between projects using the **Reference project tool**. You can rerun the workflow every time you need to perform the same sequence of operations.

With the **Workflow editor**, you can

- quantify uncertainties in a reservoir model by creating a workflow that computes multiple realizations of reservoir models by varying a range of input data
- update an existing model or simulation case with new data
- automate plotting tasks with a workflow

Workflow editor interface

This figure describes the different parts of the **Workflow editor** interface.

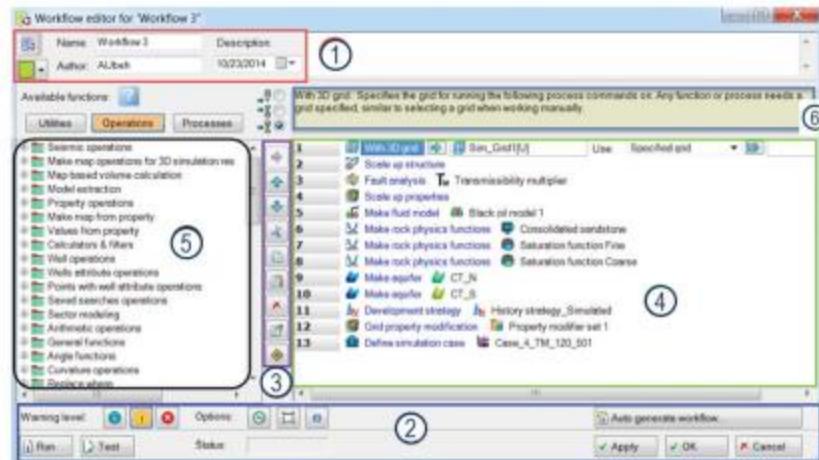


Figure 154. Workflow editor interface

- 1 Upper window: Contains fields for workflow name, author name, creation date, and general description
- 2 Lower window: Contains workflow control buttons
- 3 Toolbar: Provides buttons for inserting commands into a new workflow and adjusting the sequence
- 4 Right pane: Displays the current workflow
- 5 Left pane: Lists available workflow commands
- 6 Tip section

The **Workflow editor** dialog box is divided into five areas. It has similarities to the **Uncertainty and optimization** tool. You select statements from the left pane and drop them into the right pane to make your workflow.

Toolbox

The tabs shown in the figure are displayed in the left pane of the **Workflow editor**:

- **Utilities:** Contains programming functions and logic commands, for example, IF statements, loops, folder management commands, and variable commands
- **Operations:** Contains most of the operations available for objects in the **Input** pane. Selected operations from the **Models** pane, such as conversion of structural elements to input objects, also are available.
- **Processes:** Contains all available processes. Some processes, such as fault modeling, require manual input and are not available in the **Workflow editor**.

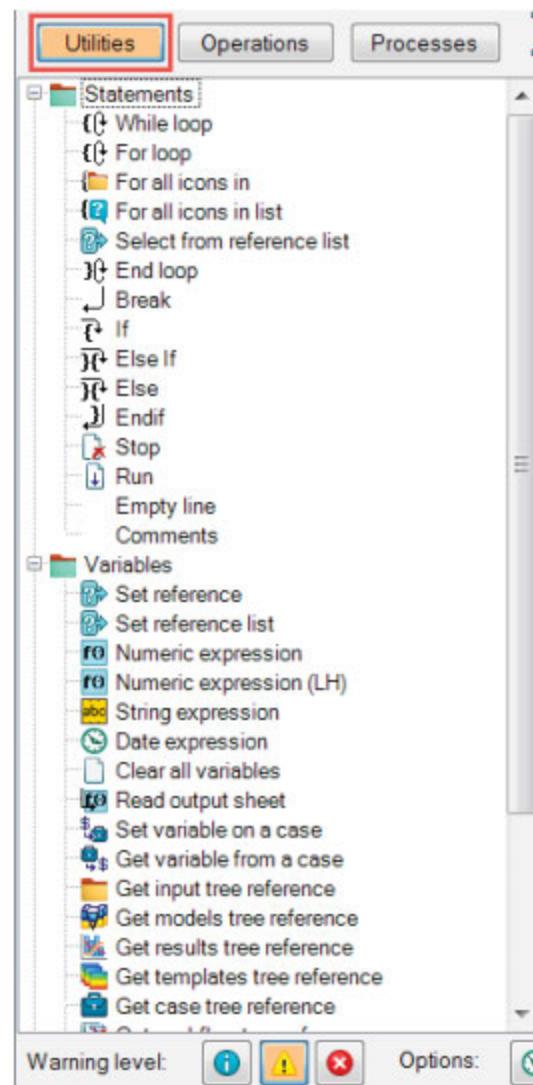


Figure 155. Utilities tab

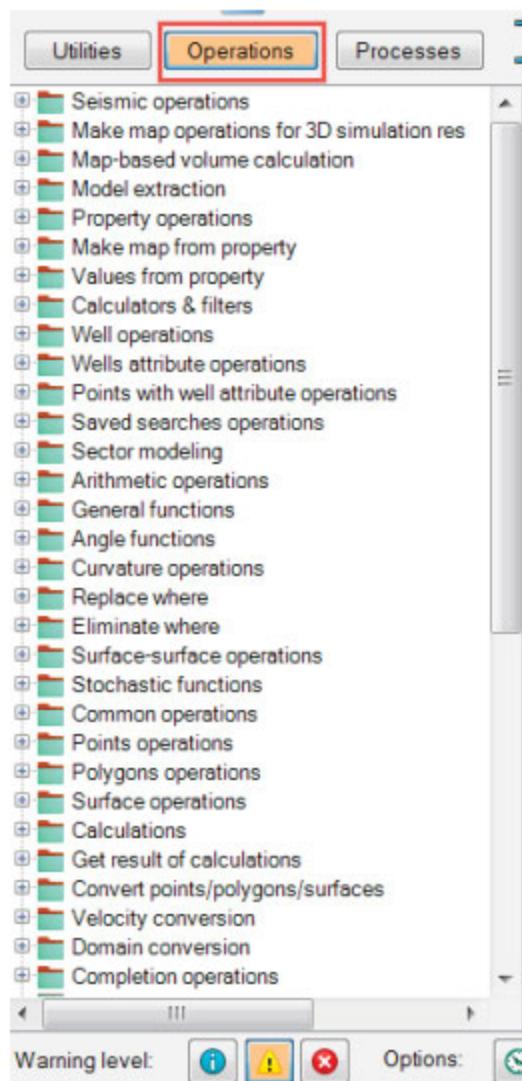


Figure 156. Operations tab

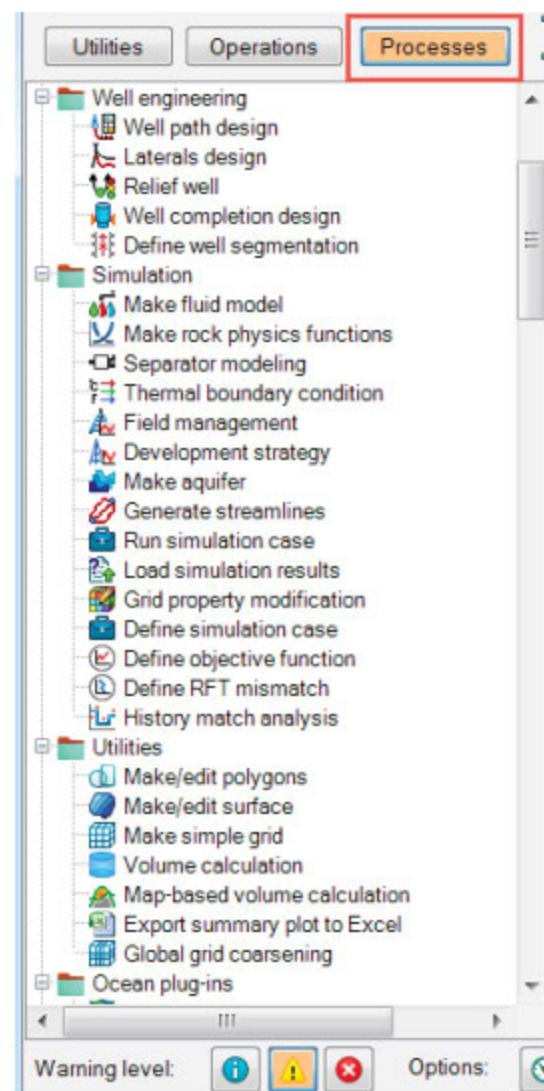


Figure 157. Processes tab

Commands to auto generate workflows

You have several options to generate workflows in Petrel based on the data or cases you have in your project (as shown in the figure).

- **Turn on visualization:** Makes a workflow that displays each of the objects in a selected folder. For example, you can edit the workflow to print the visualizations rather than just viewing them.
- **Run the calculator:** Sets up the syntax for automatically using a different property calculator expression for each of the zones in the active 3D grid. This option allows you to see the syntax for automatically setting up this type of expression and whether a filtering option needs to be used.
- **Repeat all calculations on the active 3D grid/case:** Makes a workflow that contains all process steps with the settings that were used to create the selected 3D grid or case. You can adjust the settings, or you can simply rerun the processes to update the model.
- **Make a scaled plot:** Sets up a workflow that plots each of the horizons in a selected 3D grid.

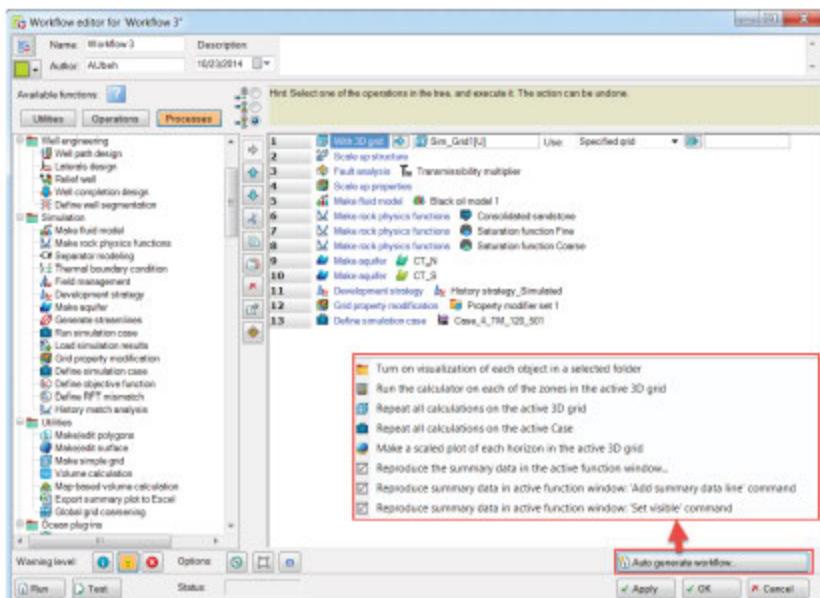


Figure 158. Commands to auto generate workflows

Exercise — Create a workflow

In this exercise, you create a workflow to regenerate the processes used in creating a simulation case and then use the workflow to do sensitivity on some of the simulation input parameters and view the results.

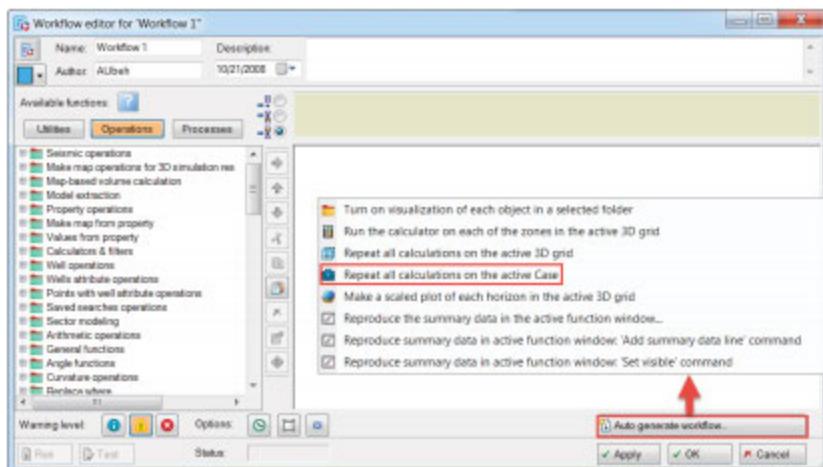
Data

Use the project History_matching and Prediction_exercise.pet in the Dataset\Projects \Module - 6 History matching and Prediction folder.



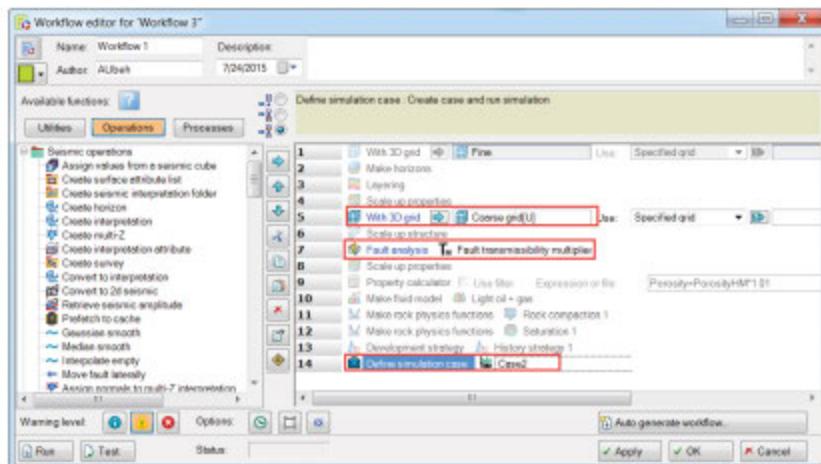
Exercise 1 Use the Auto generate workflow option to regenerate the processes used in creating a simulation case

1. Right-click in the **Workflow** pane and click **New workflow** to open a new workflow.
2. In the **Cases** pane, select **Case2**.
3. In the **Workflow editor** dialog box, click **Auto generate workflow** and then click **Repeat all calculations on the active Case**.

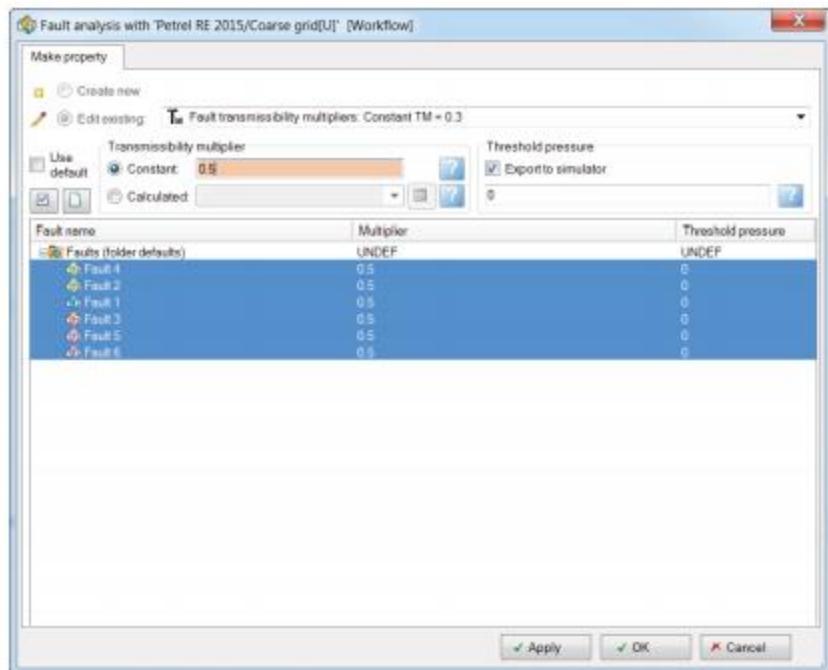


Observe that all the processes and settings used to build the active simulation case are regenerated because keeps track of the processes and settings that were used to build a model.

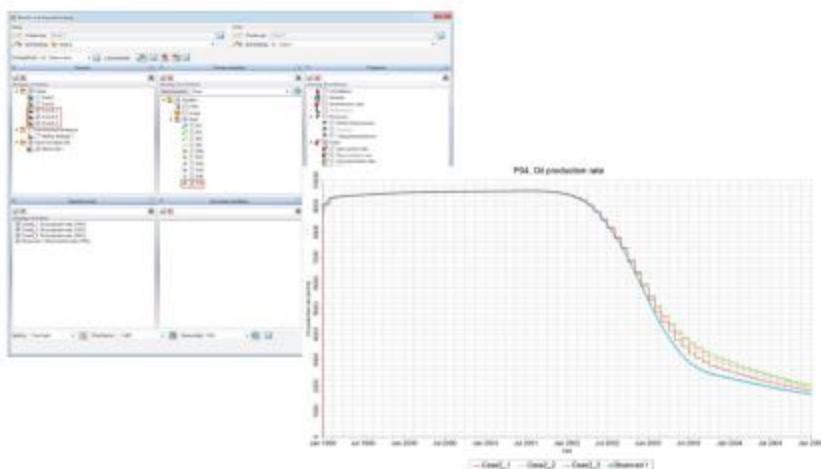
4. Disable all the processes except With 3D grid, Faults analysis, and Define simulation case to reduce the workflow run time. Note that the current process settings will be retained in the run, so there is no need to rerun all the processes. To disable processes in the **Workflow editor**, select all the processes to be disabled and click **Disable or enable selection [Ctrl+D]**.



5. Open the **Fault analysis** dialog box from the **Workflow editor**.
6. Select all the faults in the **Faults** folder, enter a constant of 0 . 5 in the **Constant** field, and click **OK**.



7. Click **Run** in the **Workflow editor** to run the simulation.
8. Change the fault transmissibility multiplier constant to 0 . 75 and 1 by repeating Step 6 to Step 8.
9. View the results with Result charting and analysis.



Copyright © 2017 Schlumberger. All rights reserved.

Lesson 4 Prediction strategy



In this lesson, you learn how to make a prediction development strategy using the **Development strategy** dialog box.

You also learn how to run a prediction strategy using a restart case from the previous history simulation run.

The purpose of a prediction development strategy is to improve the recovery or NPV from the field and determine the most appropriate and cost effective development scenarios for the future of the field.

To create the prediction development strategy, you must specify control mechanisms, new wells, an infill well, and economic limits.

It is important to apply settings that cause the simulator to treat wells in a manner similar to how the company operates the field.

For example, if the simulated field is offshore, there can be economic constraints and workovers might be limited. If an onshore waterflood has three workover rigs available, the number of workovers in a given time period might be set.

Similarly, artificial lift might or might not be plausible. Prediction runs are used to determine well flowstreams for economic calculations, so the simulator must be set up to mimic the operating procedures of the company.

Copyright © 2017 Schlumberger. All rights reserved.

NOTE: Planning for prediction simulation should begin in the early stages of a reservoir simulation study, even though predictions can be run in one of the last phases. Early planning ensures that all well completion design objects are included and that adequate reservoir modeling has been performed, which helps ensure that the study objectives can be met.

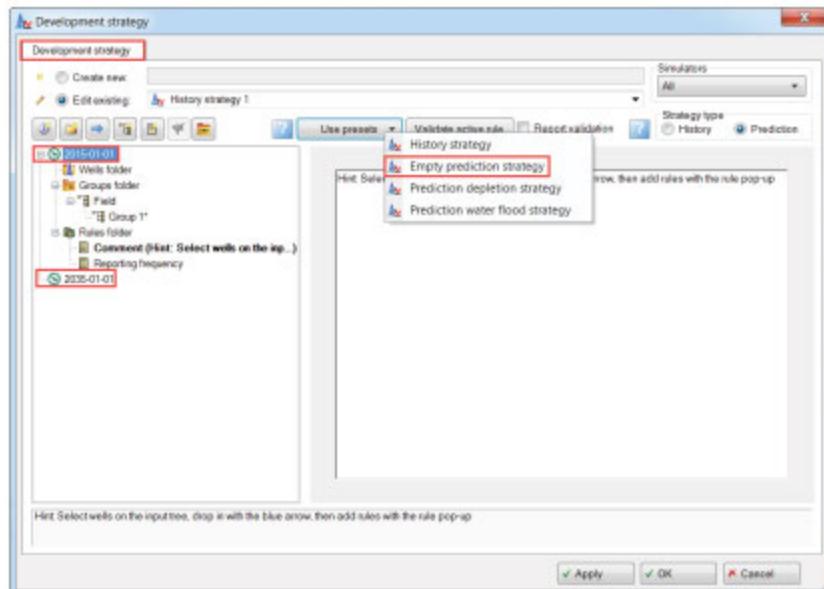


Procedure — Create a new empty prediction strategy



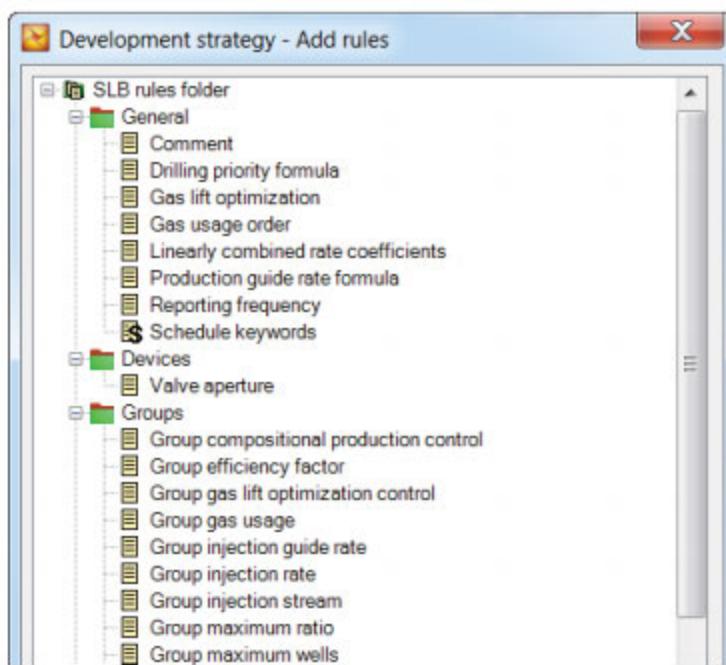
1. Open the **Development strategy** dialog box.
2. From the **Use presets** list, select **Empty prediction strategy**.

3. Edit the start and end date. Right-click the dates and click **Edit dates**.



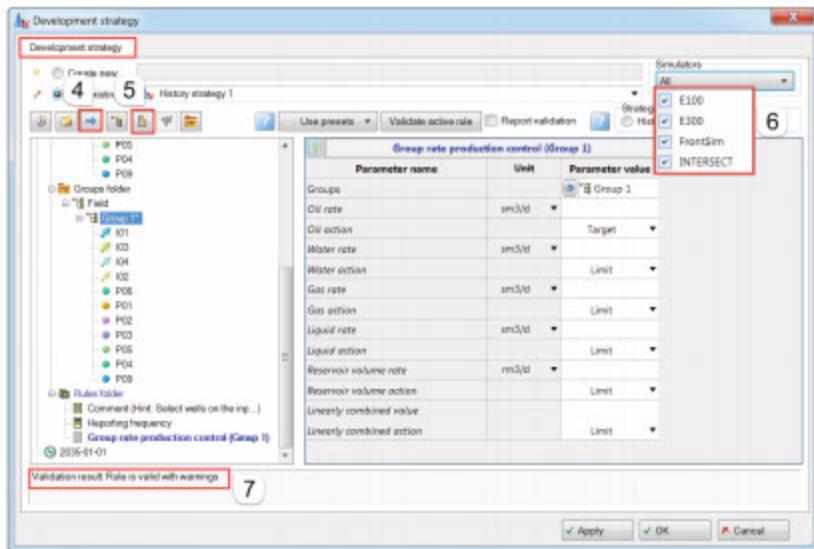
4. Insert a well or folder of wells from the **Input** pane.

5. Add control rules. Click **Open add rules dialog** .



6. Select simulators.

7. Check the rule validation result in the status bar.



Copyright © 2017 Schlumberger. All rights reserved.

Management of Well folders in the Development strategy dialog box

To apply the same rule to numerous wells in a single operation, use well folders. There are two types of folders: Linked folders and User folders.

- **Linked folders:** Add these folders by inserting a well folder from the **Input** pane into the **Development strategy** dialog box. The content is synchronized with the **Input** pane. You cannot edit the folders in the **Development strategy** dialog box. You can only delete the entire folder. So if you add a well to the folder in the **Input** pane, it is included the next time you export the development strategy to the simulator. You do not have to rerun the process.
- **User folders:** To add these folders, click **Add a new user defined folder** on the toolbar. To add wells to the folders, insert them from the **Input** pane. You also can copy them from the **Input** pane and paste them into the **Development strategy** dialog box, or drag them within the strategy tree in the **Development strategy** dialog box.

To delete a well or a user folder of wells from the strategy tree, select it and press **Delete**.

Group control

Groups are used to tell the simulator how to control several wells at a time. Often, a group corresponds to a physical structure in the field, such as a platform or a manifold. It also could be a logical grouping (for example, all of the wells that produce from Zone 2).

Group hierarchy

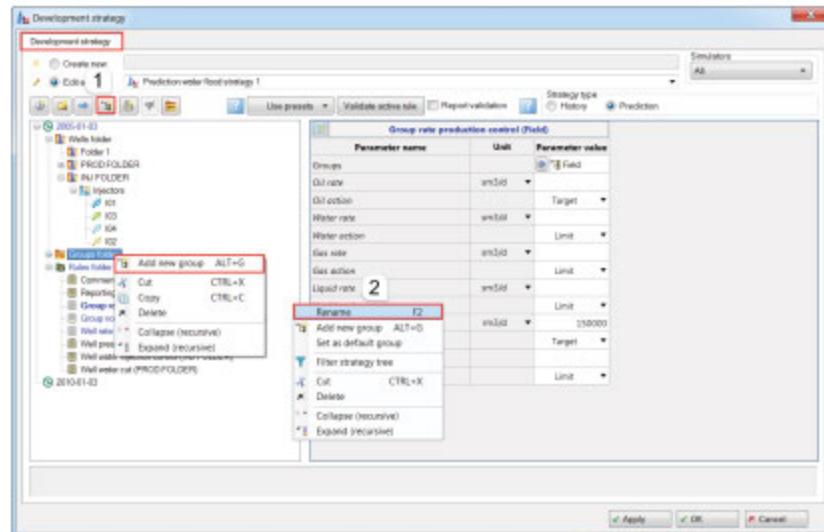
There are three levels in the group hierarchy within the strategy tree. From top to bottom, they are field, group, and well. Groups also can be children of groups. A well or a group can belong to only one group at a time. The top-level group is the **field**.

Petrel automatically adds wells to the default (first) group when they are added to the development strategy.

Procedure — Add a new well group



1. To add groups, click **Add new group**.
2. Rename a group. Right-click the selected group and click **Rename**.
3. Organize wells into groups by dragging them between groups in the **Development strategy** dialog box.



Management of Well and Group folders in the Development strategy dialog box

It is important to distinguish between well folders and groups.

- *Well folders* can be inserted into the Wells parameters of the rules. When exporting to the simulator, Petrel exports the rule to the simulator for every well in the folder. For example, set each well in a folder to produce 1,000 bbl/day.
- *Groups* can be inserted into the Groups parameters of the rules. When exporting to the simulator, Petrel exports the rule to the simulator one time for the group. The simulator works out how to apportion the rule to the members of the group.

For example, if the rule is that the field produces 10,000 bbl/day from 10 wells, it is up to the simulator to determine how much each well contributes to the field target.

Groups: Membership changing with time

You can change the group structure over time. For instance, you can move a well from the producers to the injectors group. Add the new

control date, copy the group folder to the new date, and edit the group structure accordingly (as shown in the figure).

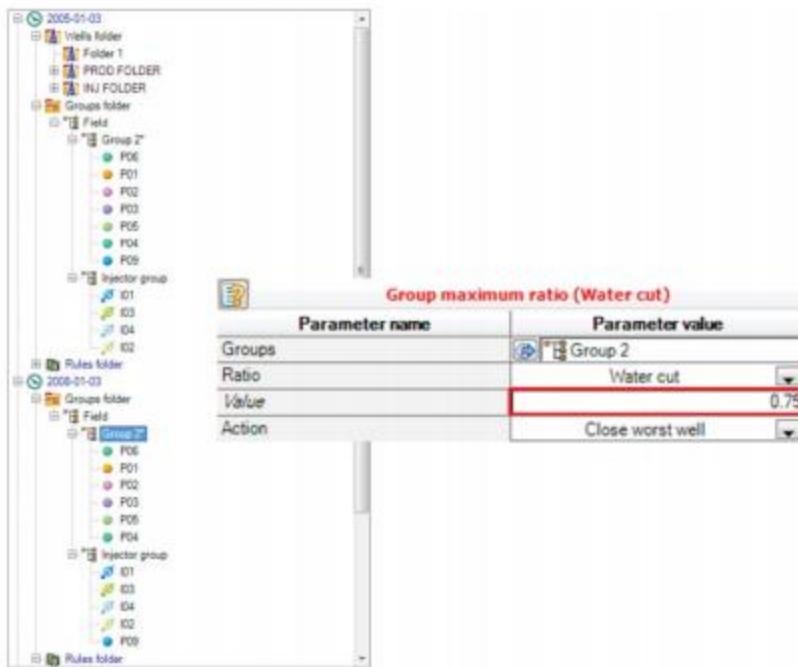


Figure 159. The conversion of well P09 to an injector in 2008-01-03 with a constraint of the group maximum ratio (water cut)

Copyright © 2017 Schlumberger. All rights reserved.

Rules

Rules provide the simulator control parameters and can generate one or more keywords.

To add rules, on the toolbar, click **Open Add rules dialog**. The **Development strategy - Add rules** dialog box opens. In this dialog box,

you can select one or more rules from a folder and add them to the strategy tree.

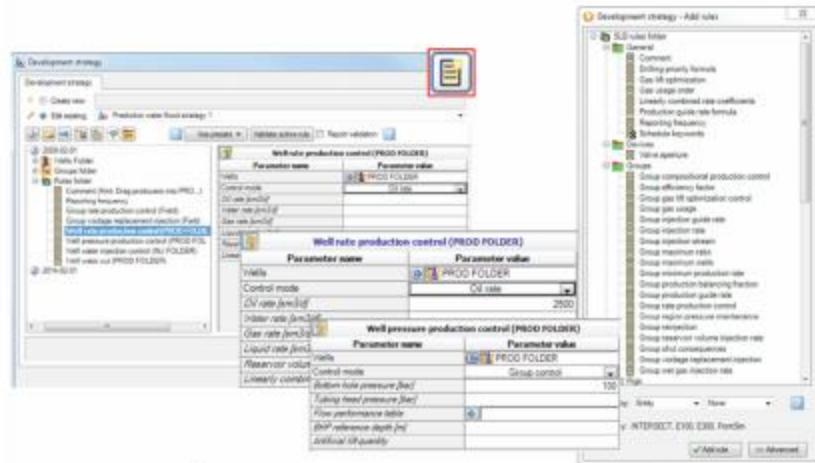


Figure 160. Adding rules in the Development strategy dialog box

The Rule selector shows only the rules that are valid for the current strategy type - history or prediction - and the currently enabled simulators. These choices appear at the top of the **Development strategy** dialog box.



NOTE: To change the way that the rules selector tree is categorized, use the **Ordered by** lists in the **Development strategy - Add rules** dialog box.

When you add a rule to the strategy, it appears in the rule table when selected. Enter numeric values and select among the choices from the lists.

There are two types of blue arrows for inserting rules. The conventional arrow is used just like the normal Petrel insert arrows. Use this type of arrow to insert data from the **Input** pane, including observed data and well flow performance (VFP) tables.

The other arrow is used to insert items from within the strategy tree, such as wells, well folders, and groups.

Targets and limits

When you add a Well rate production control rule to a well or a folder of wells, the first task is to specify the control mode. Should the rule be applied as a limit or a target?

If you select one of the rates as the control mode, enter the target rate in the corresponding field in the dialog box. You can leave other fields blank.

If you fill in an additional rate (for example, select oil rate as the target and enter a water rate), the secondary rate is implemented as a limit.

If you use Limits as the control mode, you can enter a limit for one or several rates. Notice that, in this example, you would need an additional rule to set the target for the wells.

The simulation begins by trying to meet the set target. If the set target is not met, the simulation changes control mode by imposing the set limits. However, if the set limits are not violated, the simulation runs to meet the set target.

Oil rate target example

If you set the control mode to target, all other rates you enter are implemented as limits.

Add a BHP limit in addition to the rate production control rule as shown in the first figure.

The second figure is an example of a production plot that shows the effect of applying target and limit control modes.

Well rate production control (W1)		Well pressure production control (W1)	
Parameter name	Parameter value	Parameter name	Parameter value
Wells	W1	Wells	W1
Control mode	Oil rate	Control mode	Limits
Oil rate [bm3/d]	1000	Bottom hole pressure [bar]	220
Water rate [bm3/d]	300	Tubing head pressure [bar]	
Gas rate [bm3/d]		Flow performance table	
Liquid rate [bm3/d]		BHP reference depth [m]	
Reservoir volume rate [bm3/d]		Artificial lift quantity	
Linearly combined value			

Figure 161. Oil and water rate production target constrained with BHP limit

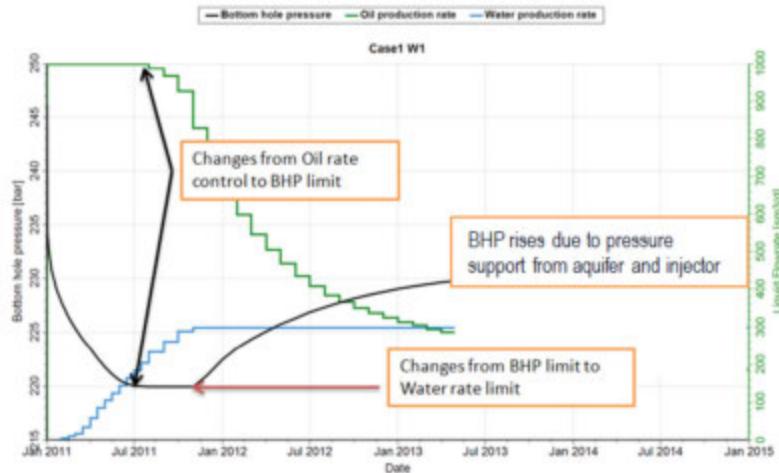


Figure 162. Production plot



Procedure — Create tabular rules

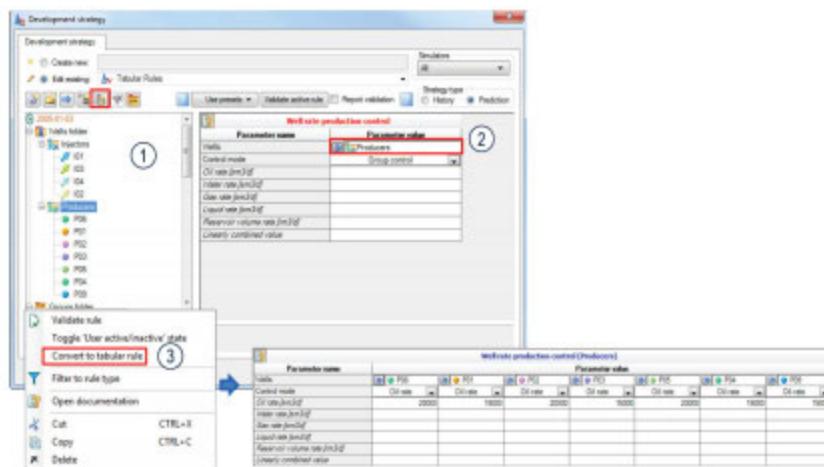
Sometimes, you must set individual parameters for several wells. For example, when you add a Well rate production control, you might want to specify different rates for some wells.

1. Start by creating a rule as usual.
2. Insert a folder that contains all of the wells into that rule, then enter all of the parameters.

These parameters will be the same for all the wells in the folder.

3. Right-click the rule and click **Convert to tabular rule**.

The tabular rule appears as a folder of rules in the strategy tree. When you select the tabular rule, it shows one column per well in the rule table, which allows you to set individual values for each well.



Rules: Validation

Rules can be valid, invalid, partially valid, unsupported, or inactive.

- A *valid* rule has all required parameters set. A valid rule (with warning) is valid for all selected simulators, but it has at least one parameter set that is not supported for one of the simulators. Unsupported parameters are shown in blue in the strategy tree.
- An *invalid* rule has one or more required parameters missing. These rules are shown with a cross overlaid on the icons in the strategy tree.

NOTE: Inactive or invalid rules are not written to the simulation dataset when you export a case.



- A *partially valid* rule has all of its required parameters set, but it is not supported by all enabled simulators. These rules are shown with an exclamation point over the icon in the strategy tree.
- An *Inactive* rule is not supported by any of the enabled simulators, or has been deactivated. These rules are shown as dimmed in the strategy tree.

The validation status and the icons are updated when you click **Apply** or **OK**. The active rule is updated when you click **Validate active rule**.



TIP: Point to the rule icon in the strategy tree to see a window with its validation report.

If you select the **Report validation** check box, all validation messages are copied to the Petrel **Message log** when you click **Apply** or **OK**.

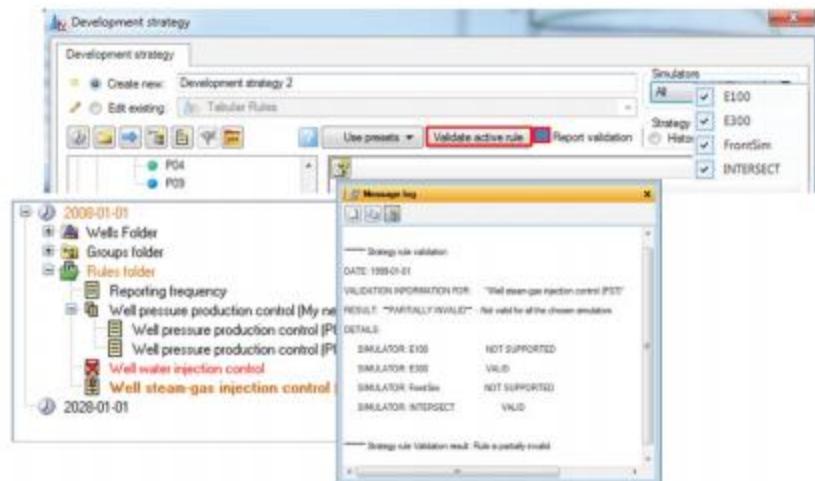


Figure 163. Rules validation

Prediction strategy in the simulation case

Insert  the prediction strategy into the data field on the **Strategies** tab of the **Define simulation case** dialog box.

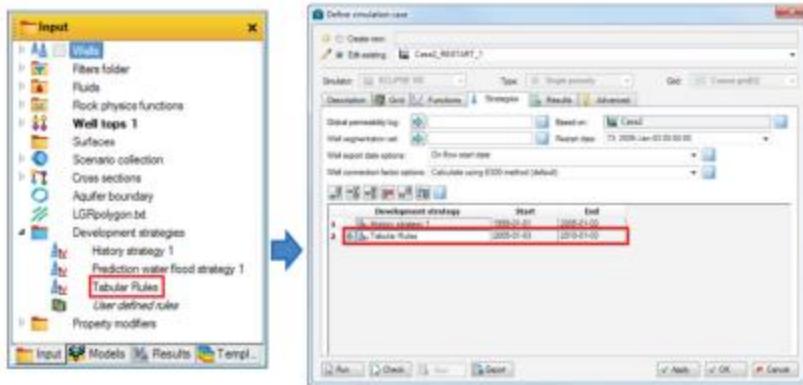
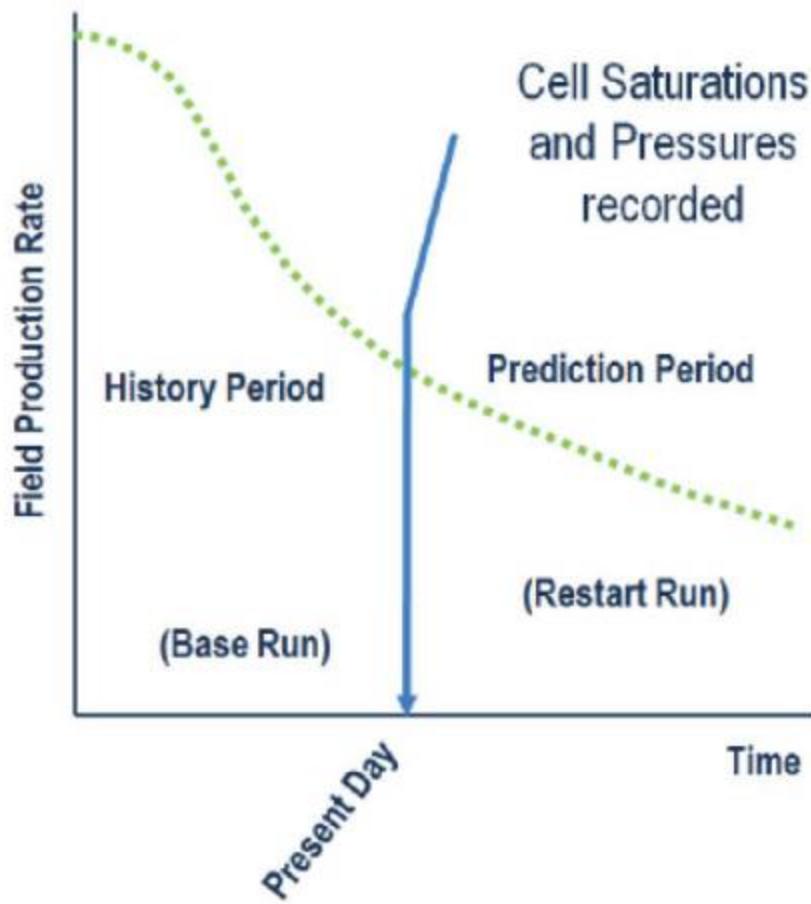


Figure 164. Insert a prediction strategy in a simulation case



Procedure — Create restart runs

Use the solution at the end of a history case as the start condition for a prediction run. Restarting runs in this way saves time because you do not recalculate pressure and saturation for the history period.

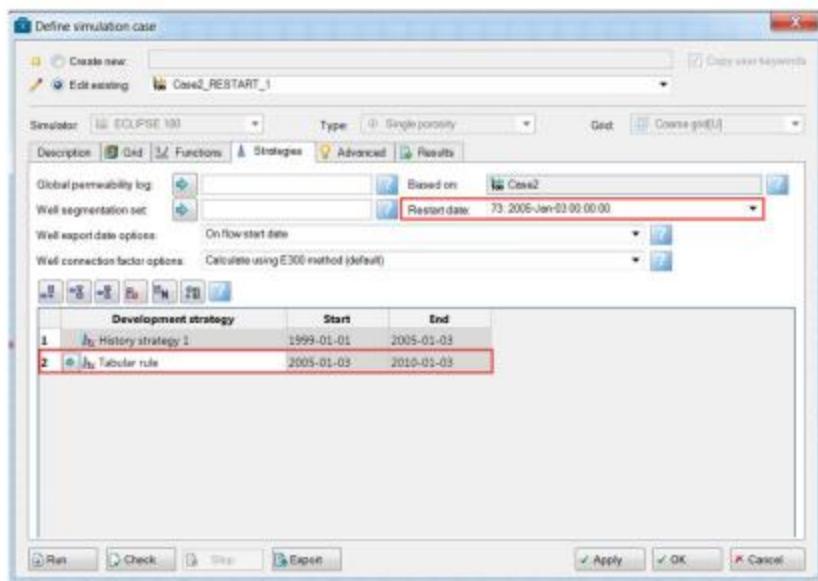


Copyright © 2017 Schlumberger. All rights reserved.

Figure 165. Application of a restart run in the prediction phase

1. On the **Simulation** tab, in the **Derive case** group, click **Restart**
2. Open the **Define simulation case** dialog box.
3. Click **Edit existing** and select the restart case.

4. Append a row to the development strategy table and insert  the prediction strategy.
5. Select the Restart date from the **Restart date** list.



Copyright © 2017 Schlumberger. All rights reserved.

Exercises — Make prediction strategies

In these exercises, you create prediction development strategies using the **Development strategy** dialog box. You then use the prediction development strategy for a production forecast for the previous history simulation case.

Workflow

1. Use a preset strategy.
2. Modify the strategy by adding a new rule to an existing strategy.
3. Set up a prediction simulation case and run the simulation.
4. View simulation results.

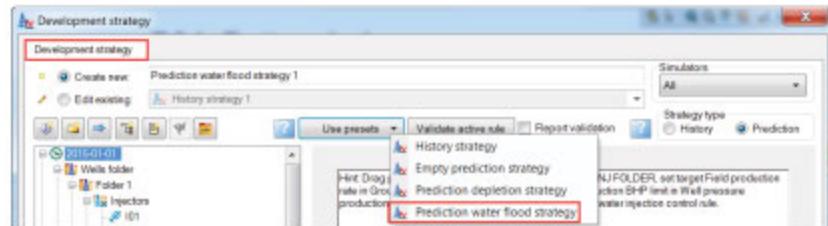
Data

Use the project `History_matching_and_Prediction_exercise.pet` that you used prior to importing the OFM project.



Exercise 1 Create a prediction strategy using a preset

1. Open the **Development strategy** dialog box.
2. From the **Use presets** list, select **Prediction water flood strategy**.



3. In the strategy tree of the dialog box, drag the **Producers** folder into the **PROD** folder and the **Injectors** folder into the **INJ** folder.
4. At the top of the left pane is the start date. Double-click it and change the start date to **2005-01-03 (YYYY-MM-DD)**.
The prediction case starts immediately after the history case ends.
5. Similarly, at the bottom of the left pane is the end date. To edit the end date, double-click it and select to run the simulation forecast for 5 years.
6. In the **Rules** folder, click the **Group rate production control** rule.
7. Enter a **Reservoir volume rate** rule of **150000 rm3/day**.
8. In the **Reservoir volume action** field, specify the **Reservoir volume rate** as **Target**.

Group rate production control (Field)		
Parameter name	Unit	Parameter value
Groups		Field
Oil rate	sm ³ /d	Target
Water rate	sm ³ /d	Limit
Gas rate	sm ³ /d	Limit
Liquid rate	sm ³ /d	Limit
Liquid action		Limit
Reservoir volume rate	rm ³ /d	150000
Reservoir volume action		Target
Linearly combined value		
Linearly combined action		Limit

9. Right-click the Groups folder and click **Add new group**.
10. Change the name of the new inserted group to **Injectors**.
11. Drag all of the injector wells (I01, I02, I03, and I04) from Group 1* in the Field folder to the new Injectors folder.
12. Click the group Voidage replacement injection rule and enter a Voidage replacement fraction of 1.

Group voidage replacement injection (Field, Water)	
Parameter name	Parameter value
Groups	Field
Phase	Water
Voidage replacement fraction	1
Control	Target
Voidage replacement group	

13. Click the **Well rate production control** rule. Ensure that the PROD FOLDER is inserted into the **Wells** field in the **Parameter value** column and that the **Control mode** is set to **Group control**.
14. Leave the remaining fields blank.

Well rate production control (PROD FOLDER)		
Parameter name	Unit	Parameter value
Wells		 PROD FOLDER
Control mode		Group control ▾
Oil rate	sm ³ /d ▾	
Water rate	sm ³ /d ▾	
Gas rate	sm ³ /d ▾	
Liquid rate	sm ³ /d ▾	
Reservoir volume rate	rm ³ /d ▾	
Linearly combined value		

15. Click the **Well pressure production control** rule.
16. Insert the PROD FOLDER into the **Wells** field.
17. In the **Control mode** field, select **Limits** from the list.
18. Specify a **Bottom hole pressure** limit of 100 bar.
19. Leave the remaining fields blank.

Well pressure production control (PROD FOLDER)		
Parameter name	Unit	Parameter value
Wells		 PROD FOLDER
Control mode		Limits ▾
Bottom hole pressure	bar ▾	100
Tubing head pressure	bar ▾	
Flow performance table		
BHP reference depth	m ▾	
Artificial lift quantity		

20. Click the **Well water injection control** rule.
21. Check that the INJ FOLDER is inserted into the **Wells** field.
22. Set **Control mode** to **Group** control.

23. Enter a **Bottom hole pressure** limit of 400 bar.

Well water injection control (INJ FOLDER)		
Parameter name	Unit	Parameter value
Wells		INJ FOLDER
Control mode		Group control ▾
Surface rate	sm ³ /d	▼
Reservoir rate	rm ³ /d	▼
Bottom hole pressure	bar	400
Tubing head pressure	bar	▼
Injection flow perf. table		
BHP reference depth	m	▼

24. Click the **Reporting frequency** rule and select to report every 3 months.
 25. Click **Apply** to save the prediction development strategy.
 26. Do not close the **Development strategy** dialog box.

Exercise 2 Add a new rule to an existing strategy

Continue this exercise from the previous exercise.



- In the **Development strategy** dialog box, click **Open Add rules dialog** .
- Select the Well water cut rule located in the **Wells** folder.
- Click **Add rule**.
- Close the **Add rules** dialog box.
- In the new Water cut rule, insert the PROD folder into the **Wells** field in the Parameter value column.
- Enter a **Water cut** limit of 0 . 9.
- In the **Water cut action** field, select **Close well** from the list.

Well water cut		
Parameter name	Unit	Parameter value
Wells		PROD FOLDER
Water cut	sm3/sm3	0.9
Water cut action		Close well
Secondary water cut	sm3/sm3	
Secondary action		

8. To check the validity of your new rule, click **Validate active rule**.
The color of the Well water cut rule changes to black.
9. Click **OK** to save your development strategy and close the dialog box.

Exercise 3 Set up a prediction simulation case

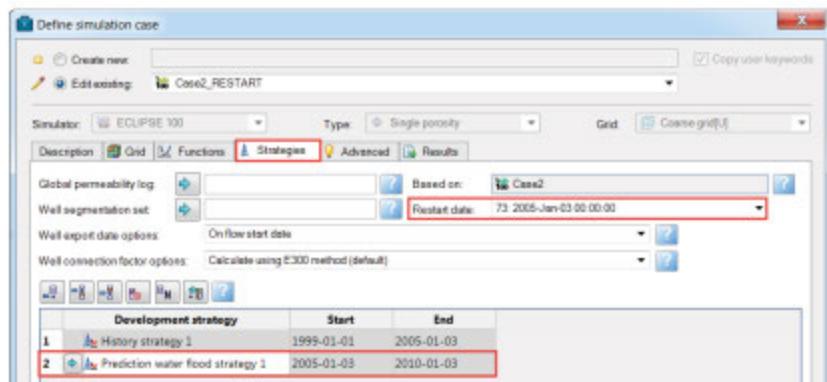


1. In the **Cases** pane, select **Case2**.
2. On the **Simulation** tab, in the **Derive case** group, click **Restart**.



3. Open the **Define simulation case** dialog box.
 4. Click **Edit existing** and select **Case2_RESTART**.
- By design, all of the settings on the **Grid**, **Functions**, and **Strategies** tabs are disabled to preserve the history case. These settings are not editable using the Restart functionality.
5. Click the **Strategies** tab.
 6. Add a row to the table. Click **Append item in the table**.
 7. In the **Input** pane, select **Prediction waterflood strategy 1** and insert it into the data field.
 8. Make sure that 3rd January 2005 is selected as the **Restart date**. If it is not, select it from the **Restart date** list.

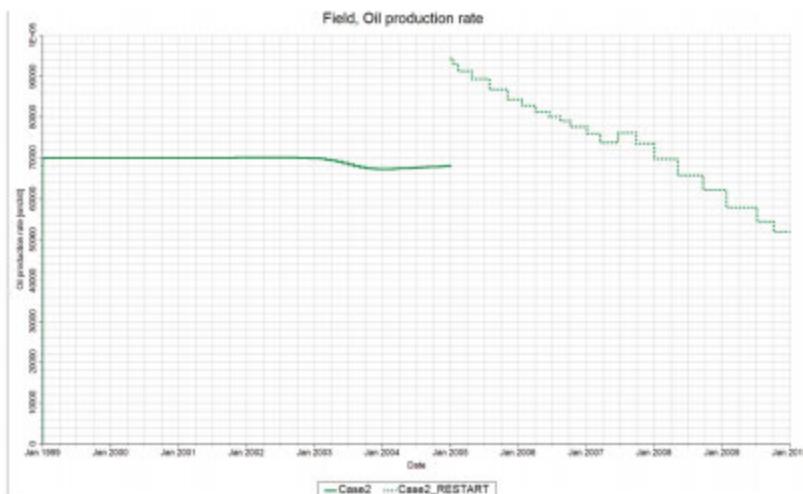
This selection helps reduce the simulation time by not going through the history period again, but starting at the last date of the history.



9. Click **Apply** to save your case.
10. Click **Run** to run the simulation.

Exercise 4 View the simulation forecast results

Plot the field Oil production for history (Case2) and Prediction (Case2_Restart) using the **Results charting and analysis** dialog box.



Exercise 5 Create a new prediction strategy using a tabular rule

1. Open the **Development strategy** dialog box.
2. Click **Create new**.

3. From the **Use presets** list, select **Empty prediction strategy**.
4. Rename the strategy Tabular rule.
5. Double-click the start date and change it to 2005-01-03.
6. Double-click the end-date and change it to 2010-01-03.
7. Click the Report frequency rule and select a report frequency of 6 months.
8. In the **Input** pane, select the Injectors and the Producers folders and insert  them into the **Development strategy** dialog box.
9. Click **Add rules**  to open the **Development strategy - Add rules** dialog box.
10. Add Group voidage replacement injection, Well water injection control, Well rate production control, and Well pressure production control rules.
11. Close the **Development strategy - Add rules** dialog box.
12. Right-click the Groups folder and click **Add new group**.
13. Change the name of the inserted new group to **Injectors**.
14. Drag all the injector wells (I01, I02, I03, and I04) from Group 1* in the Field folder to the new **Injectors** folder.
15. Click the Group voidage replacement injection rule and insert  the Field folder into the **Groups** field in the Parameter value column.
16. Enter a Voidage replacement fraction of 1.

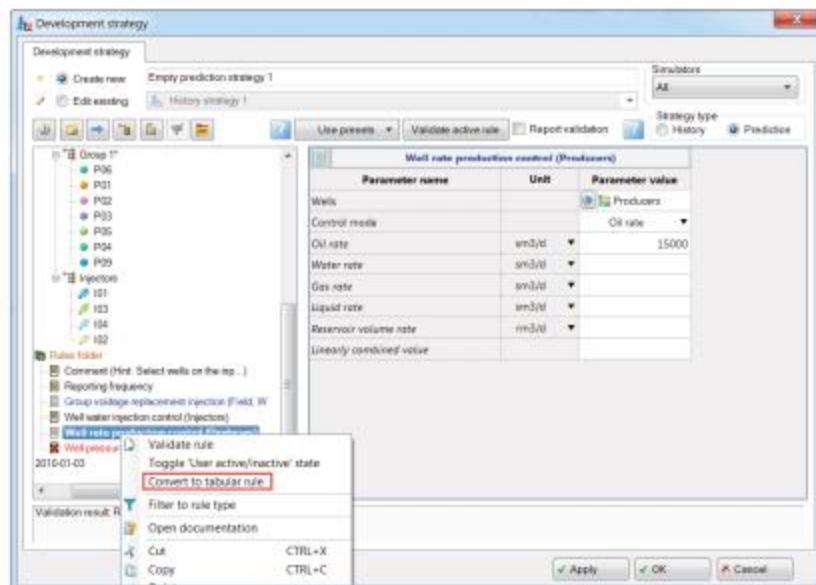
Group voidage replacement injection (Field)	
Parameter name	Parameter value
Groups	 "I04 Field"
Phase	Water ▾
Voidage replacement fraction	1
Control	Target ▾
Voidage replacement group	

17. Click the Well water injection control rule and insert  the **Injectors** folder into the **Wells** data field in the Parameter value column.
18. Select **Group control** as the **Control mode**.

19. Click the Well rate production control rule.
20. Insert  the Producers folder into the **Wells** field in the Parameter value column.
21. Select **Oil rate** as the **Control mode**.
22. Enter an oil rate of 15000 sm3/d.

Well rate production control (Producers)		
Parameter name	Unit	Parameter value
Wells		 Producers
Control mode		Oil rate ▼
Oil rate	sm3/d ▼	15000
Water rate	sm3/d ▼	
Gas rate	sm3/d ▼	
Liquid rate	sm3/d ▼	
Reservoir volume rate	rm3/d ▼	
Linearly combined value		

23. In the Rules folder, right-click the Well rate production control rule and click **Convert to tabular rule**.



24. In the tabular rule, change the **Oil rate** for wells P02, P05, and P06 to 20,000 sm3/d.

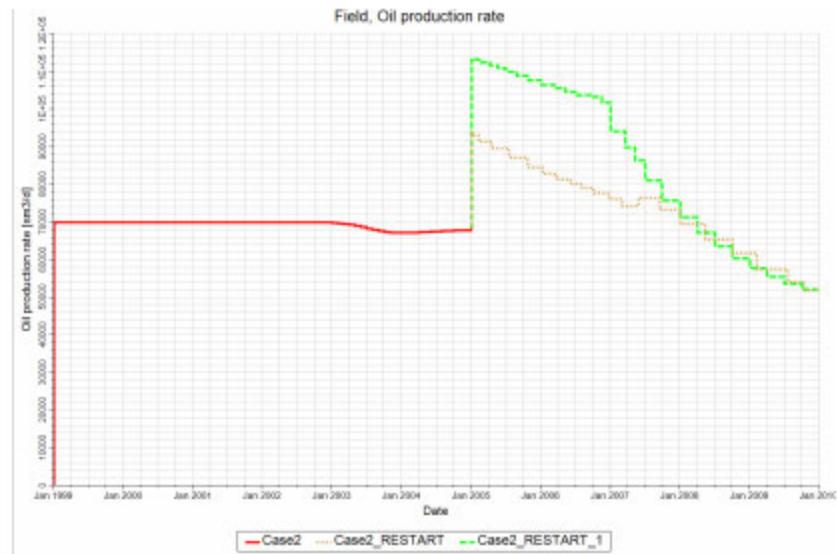
Parameter name	Unit	Parameter value							
Wells		P06	P01	P02	P03	P05	P04	P09	
Control mode		Oil rate		Oil rate		Oil rate		Oil rate	
Oil rate	sm3/d	20000	15000	20000	15000	20000	15000	20000	15000
Water rate	sm3/d								
Gas rate	sm3/d								
Liquid rate	sm3/d								
Reservoir volume rate	mm3/d								
Linearly combined value									

25. Click the Well pressure production control rule and insert the Producers folder into the **Wells** field.
 26. Select **Limits** as the **Control mode**.
 27. Enter a **Bottom hole pressure** limit of 100 bar.
 28. Click **OK** to save the new strategy.



Exercise 6 Include the tabular rule prediction strategy in a simulation case and run a simulation forecast

1. In the **Cases** pane, select Case2.
2. On the **Simulation** tab, in the **Derive case** group, click **Restart**.
3. Open the **Define simulation case** dialog box.
4. Click **Edit existing** and select Case2_RESTART_1.
5. On the **Strategy** tab, add a new data field row and insert the Tabular rule strategy.
6. Click **Apply** and run the simulation.
7. View the results using the **Results charting and analysis** dialog box and Spit by Restart case.



Copyright © 2007 Schlumberger. All rights reserved.

Review and summary

Review what you learned in this module.

The review and summary help you to reinforce the learning objectives for History matching and prediction.



Review questions

The review questions reinforce the learning objectives.

- What type of well data can you import using **OFM data connector** dialog box?
- What type of case must you create to prevent the simulator from recalculating pressure and saturation for the history period? On which domain tab can you access it?

Summary

In this module, you learned about:

- importing observed data with the .vol file format and from an OFM project

NOTES

Copyright © 2017 Schlumberger. All rights reserved.

NOTES

Copyright © 2017 Schlumberger. All rights reserved.

Module 7 Simulation study

The goal of these exercises is to give you the opportunity to apply most of what you have learned during this course in a more realistic scenario. Each exercise contains a description of the scenario and the main steps. The idea is to let you try to solve the task by yourself. Only a few steps are given in each exercise to encourage you to find your own way to complete the task.

Prerequisites

To get the full value from these exercises, it is recommended that you complete at least Modules 1 - 6 of this course successfully.



Model description

The Mandiwat reservoir is a variable thickness clastic structure with a minor fault that runs through the center. The reservoir covers an area of 16000 x 24000 ft (5 x 7 km) and is 700 ft thick at the thickest end. The reservoir is divided into 20 x 30 x 9 grid cells that yield a total of 5400 active cells. The reservoir properties are heterogeneous, with the porosity following an increasing trend with depth. This table lists the minimum and maximum values for the porosity and permeability.

Porosity	0.23-0.31
Horizontal Permeability (mD)	275-525
Kv/Kh ratio	0.3

The development scheme for the field uses a five-spot pattern. To cover the extent on the field, the development scheme has two additional injectors offset to the North and South. The wells have been completed

throughout the reservoir for maximum contact. The figure shows the Mandiwat model structure along with the well locations.

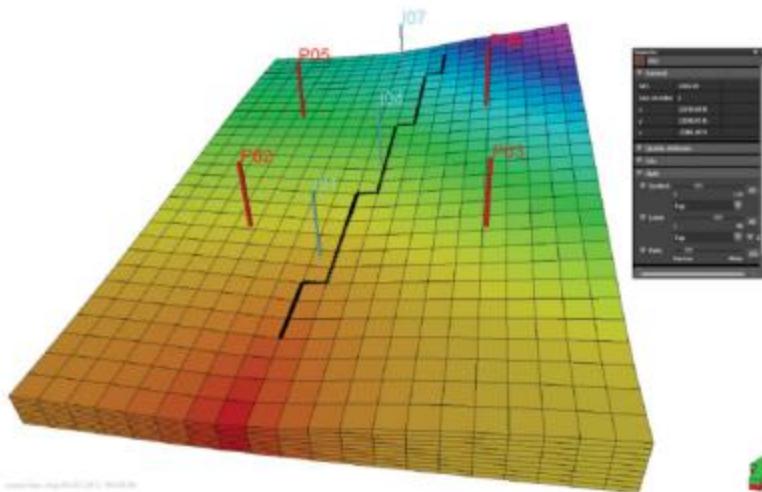


Figure 166. Mandiwat reservoir model with well locations

Copyright © 2017 Schlumberger. All rights reserved.

Exercises — Simulation study

In these exercises, you are expected to put into practice all you have learned during this training. These exercises give you the opportunity to apply the software skills that you have acquired into more practical reservoir engineering challenges.

Workflow

1. Prepare and review all of the input data required to assemble your base case.
2. Run the base case using the Mandiwat reservoir model.
3. Implement a new development strategy by converting one producer to an injector well.
4. Create a new injector well to maintain the oil production.
5. Implement another new development strategy that includes the new injector well.
6. Use the Petrel tools to analyze the results and determine the level of improvement between the cases.

Data

Use the project named Mandiwat_Start in the Dataset \\Projects\\Module-10 Simulation study folder.

Exercise 1 Prepare and review all the input data required to assemble your base case

The model contains only two phases, oil and water. The water contact is defined at -3330 ft.

A sample of the existing fluids in the Mandiwat reservoir has been analyzed in the laboratory. This table shows the results.

Datum depth	-2300 ft
Pressure at datum depth	3500 psi
Gas gravity	0.71 sg air
Oil gravity	27 deg API
Bubblepoint pressure	1500 psi
Water salinity	3000 ppm

This table shows the reservoir conditions.

Minimum pressure	1500 psi
Maximum pressure	5000 psi
Temperature	225 degF
Reference pressure	1500 psi

A different application created the saturation function for this reservoir. It is ready to be imported into your project.

A compaction test of the rock has been carried out in the laboratory, giving these results:

Minimum pressure	1500 psi
Maximum pressure	5000 psi
Compressibility	0.000001
Reference pressure	1500

1. Launch Petrel and open the project named `Mandiwat_start`.
2. Review the `Mandiwat_start` project. Pay attention to the existing data that you have in the project. Also, focus on the additional data that you might need to create or import to create your base case.
3. Create your fluid model. Use the data provided in the previous tables.
4. Create a user-defined Rock compaction function. Use the laboratory test results provided.
5. Import the saturation function named `RelPerms_Base` in the Input data Simulation Study folder.
6. Quality check the saturation functions and fluid model. Use the Rock physics plot in the **Rock physics** group and the Fluid model plot in the **Fluids** groups on the **Reservoir engineering** tab.



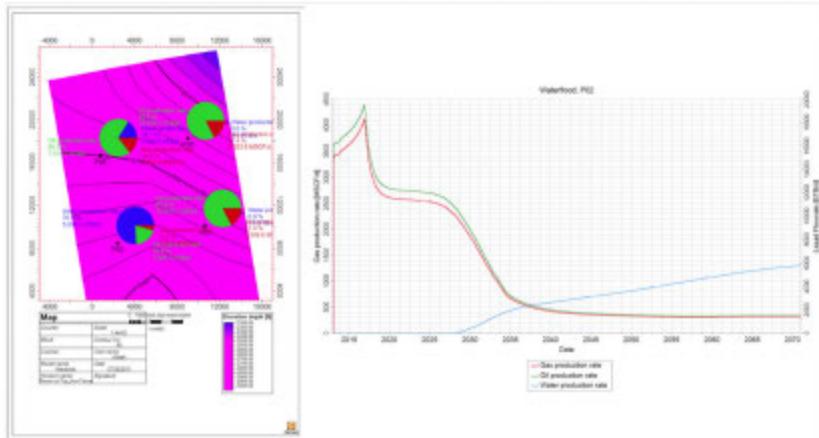
Exercise 2 Run the base case using the Mandiwat reservoir model

It is known that the oil rig used in this field has specific surface facilities constraints for handling the water injection (total injection must not exceed 80,000 STB/d). The development strategy contains a Well water injection control rule. This rule sets up a surface rate of 15,000 STB/d with an upper BHP limit of 5,000 psi.

To operate on a BHP target of 1,800 psi, a Well pressure production rule is used to set up the producers. Finally, a third Group rate production control rule is used to set an oil target of 80,000 STB/d for the group of producers. This strategy will be run for 60 years with a monthly report frequency. To simplify this exercise, the `Waterflood_Base` strategy has been created to meet these criteria.

It is important to remember as mentioned in the model description, that there is a minor fault running through the center in this reservoir model. This fault is partially sealing; therefore, it is crucial to make sure to include the transmissibility of the fault when you define the simulation case.

1. Open the **Define simulation case** dialog box and create your new Waterflood_base case (remember to include the transmissibility multiplier for the fault).
2. Use the existing development strategy (Waterflood_base strategy).
3. Run the simulation case. Use the ECLIPSE 100 simulator.
4. View and analyze the Oil production rate for the producer wells in a **Charting** window.
5. Create a General Intersection and align it along well P05, I04, and P03 using the **Snap to point** or **Snap to 2 points** options.
6. Display the water saturation simulation grid from the **Results** pane on the General intersection.
7. On the **Home** tab, in the **View** group, click **Players** and then click **Time player**. Play through time using the **Time player** to analyze the water progression to the producer with time. Remember to display the well completions.
8. Create a bubble map of the Oil, Water, and Gas production rate.



Copyright © 2017 Schlumberger. All rights reserved.



Exercise 3 Implement a new development strategy by converting one producer to an injector well

As seen in the plot of the oil, gas, and water production rate for all producers, the oil production rate for well P02 starts to decline dramatically in 2016. Around 2040, the well almost is not contributing to the total oil production of the field. If you review the simulation model in the **3D window**, you can see that because of the partial seal of the fault, most of the water is saturating the western part of the reservoir.

The 3D visualization of water saturation clearly shows how P02 receives a lot of the water injected in well I01. This water saturation causes the early water breakthrough around year 2027. The water breakthrough continues to increase throughout the production period.

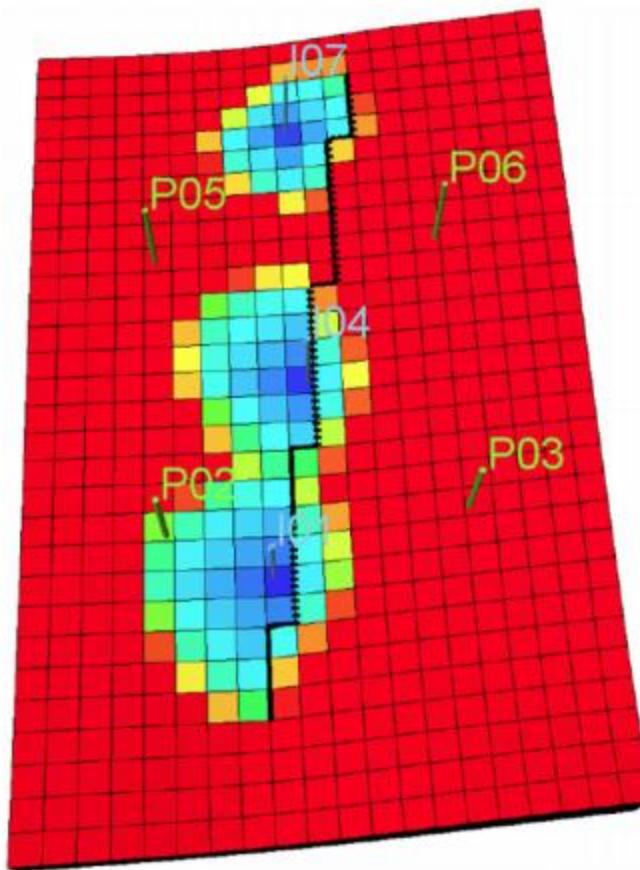


Figure 167. Water saturation simulation grid results at the last timestep

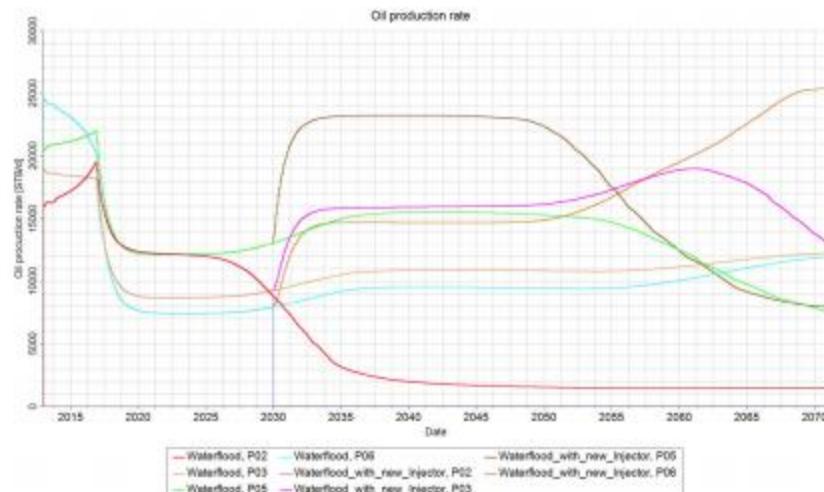
After analyzing the situation, the management decides to apply a different development strategy. Well P02 is converted to an injector after year 2030. In this new development strategy, the field operates as previously defined in the Waterflood_base strategy until year 2030. After 2030, a new timestep must be introduced and well P02 must be moved to the `Injector wells` folder. The same rules as previously described for the Waterflood_base strategy apply for this new part of the new development strategy.

1. Using the Waterflood_base development strategy as a reference, create a new strategy. In this new strategy, use well P02 as an injector after year 2030.



TIP: Introduce a new timestep at 2030-01-01. Copy the Waterflood_base strategy content and paste it into the new timestep. Next, drag well P02 from the Producers folder to the Injectors folder (these folders are in the Wells folder and the Groups folder).

2. Create a new simulation case. Use the same parameters as in the previous Waterflood_base case, but use the new development strategy that you just created.
3. Run the simulation of the new case.
4. Analyze the cases. Use the Result charting and analysis process.



Copyright © 2017 Schlumberger. All rights reserved.

As you can see in your plots, it seems that the well management has a positive impact on the oil production rate for the field after year 2030 where you decided to use well P02 as an injector. Think about other development strategy options that can help improve the oil production of the field.



Exercise 4 Create a new injector well to maintain the oil production

The wells P03 and P06 are located on the eastern part of the reservoir. These wells are on one side of the partial sealing fault and the rest of the wells are on the opposite side. In the previous simulation scenarios, you can see that the oil production rate for P03 and P06 is stable until around year 2061. In 2061, the oil production rate for well P03 starts to decline. Because these wells are on the other side of the sealing fault from the other wells, there is not enough energy support from the water injectors to continue to support the oil production.

To support those producers to keep the expected oil production rate, think about the best option to explore. In this scenario exercise, create a new injector well between well P03 and well P06.

1. In the **Input** pane, create a wells subfolder and name it **My Wells**.
2. In this subfolder, create a vertical well between the wells P03 and P06. Use the information in this table for the new well:

X location	9440.72 ft
Y location	14126.18 ft
KB	0 ft
Total MD	3222 ft
Well type	Injector

3. Complete the new injector with a casing and a perforation. Ensure that the casing covers the entire well. Place the perforation in the reservoir zone between the ReservoirTop and ReservoirBase surfaces. Use the timestep of Jan 01, 2061 for both completions.
4. Create a development strategy that includes the new injector well.

TIP: In this example, all you need to do is to insert the new well and add an injection control rule for it.



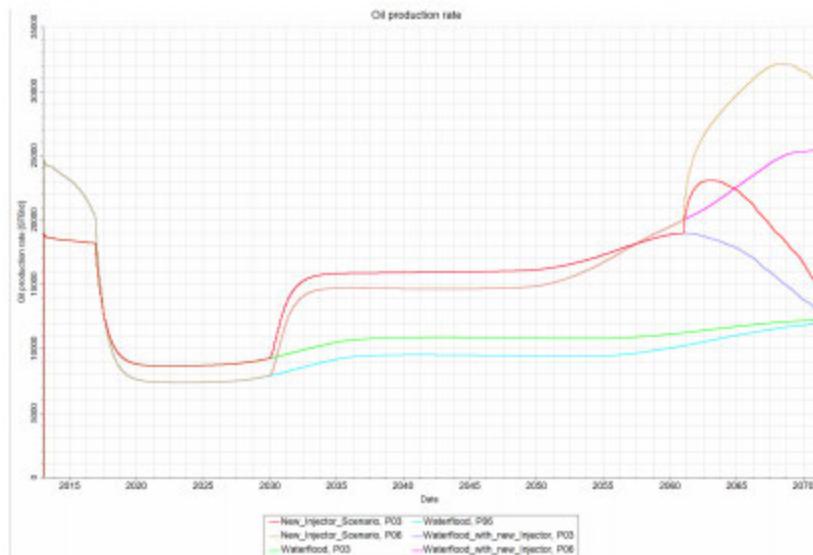
5. Create a new simulation case using the same parameters as the previous cases, but this time, use the new strategy that contains the new injector well.



TIP: If you are struggling with some parts of this simulation study, a solution project exists. It is named Mandiwat_Completed.pet.

6. Plot the simulation results. Compare the oil production rate based on the different development scenario simulation cases. .

This figure shows the effect of the injector well on the oil production.



NOTES

Copyright © 2017 Schlumberger. All rights reserved.

NOTES

Copyright © 2017 Schlumberger. All rights reserved.

Glossary

This section contains terminology and commonly used acronyms.

Terminology

<i>3D Grid</i>	A corner point 3D grid suitable for geological modeling and flow simulation.
<i>Boundary for the Pillar grid</i>	Defines the extent of the model. You can use a polygon as input, or you can digitize the boundary. Faults can be set as part of the boundary.
<i>Cases pane</i>	Shows volumetric cases and simulations cases as set up in the Define simulation case process.
<i>Cell</i>	The smallest volumetric element in the Pillar grid. Each cell has a unique I, J, K position in the Pillar grid. The I, J, K positions are zero-based integer locations of cells in a grid. A cell uses a right orientation with I, J as a horizontal surface and K increasing downward.
	The number of cells in a grid is equal to the number of I times the number of J times the number of K points. I, J, K indexing provides a way to identify the position of a cell in the grid.
<i>Charting window</i>	Plotting window used by the Results charting and analysis process.

Corner point grid

A flexible grid structure in which the eight corners of a cell (the nodes) can be moved to form irregular cell geometries. There are three techniques to build a Corner point grid: Make simple grid, Structural, and pillar gridding.

Contextual tabs

Contextual tabs appear on the ribbon above the core domain tabs whenever you activate a display window or select a displayed object. They are colored differently from core tabs.

They also support your Ocean plug-ins.

These are the two most frequently used contextual tabs:

- **Window:** Automatically updates with tools specific to the active display window.
- **Object:** Displays applicable commands and tools specific to the active object in the display window.

Directions (in the Pillar gridding process)

User-defined guidance for orientation of cells along faults. A fault can be given a direction, in which case, the grid is aligned with that fault.

Domain tabs

All of the dialog box launchers, tools, and actions associated with Petrel processes are on the domain tabs. Domain tabs can be filtered using the **Perspective** tool. To improve efficiency, some processes and utilities are on multiple tabs.

Fault

Vertical surface that segments the Pillar grid. It is represented by a set of pillars that join to define the fault. By default, an area enclosed by faults makes a segment of the model.

A fault can be given a direction, in which case, the grid is aligned with that fault.

Favorites pane

Allows you to build a list of shortcuts to the objects that you use most (for example, data, processes, and windows).

Filter

Petrel can display the whole model (all elements) or selected parts of it. Powerful filter techniques enable you to take full control of the 3D grid and visualize only the elements of interest for quality control. Filters in Petrel are used both for visualization and calculations. There are two basic types of filters in Petrel.

The first type is a simple On/Off type, where an element is either turned on or off. Other filters are more advanced (for example filtering of properties) and are based on values that you set.

File tab

This core tab contains project-level commands such as **Save**, **Open**, and **Setup**. You also access the online Help and licensing from this tab.

Function window

Plot window used to display functions, crossplots, sample variograms, and variogram models.

Histogram window

Plot window used to display histograms and cumulative distribution functions.

Horizon

The equivalent of a surface, except that a horizon is a surface in a 3D grid. It is an integrated part of the 3D model. A horizon in a Petrel 3D grid can have multiple Z values at a single XY value, whereas a surface cannot.

Home tab

Project management, interface setup, and annotation commands are grouped on the **Home** tab.

This tab also is where you find the **Perspective** tool for filtering the ribbon to display only tabs relevant to your daily workflow.

Input pane

Stores imported data such as lines, wells, fluid models, gridded surfaces, and SEG-Y data. Also stores data created by Petrel processes such as Development strategy and Rock physics functions.

<i>Intersection</i>	A plane along which data can appear. These planes can be positioned in any direction along model grid lines, seismic lines, well paths, or intersection fences. Intersections can be printed from 2D windows , 3D windows , or Intersection windows .
<i>Inspector</i>	Floating window that provides a detailed view of the objects that you select. The Inspector allows you to adjust style and other settings without leaving the display window.
<i>Intersection window</i>	This Plot window generates scaled plots of cross sections.
<i>Keyword editor</i>	Allows you to make edits to the case after it has been exported.
<i>Layers</i>	Fine scale division of a 3D grid that defines the cell thickness.
<i>LGR</i>	Local grid refinement that allows for a finer resolution of the grid in demanding areas. Local grids are defined using wells, surfaces, and polygons as input.
<i>Map window</i>	This Plot window generates scaled plots (2D maps) and display history match values and variogram maps created in Petrel.
<i>Mini toolbar and shortcut menu</i>	The mini toolbar contains formatting tools for the object selected in the display. Opening this toolbar also opens a shortcut menu with specialized commands related to that object. To open the mini toolbar and shortcut menu, right-click an object in the display.
<i>Model</i>	Complete set of data needed to describe a three-dimensional geological model. This set of data includes the 3D grid structure with faults and horizons and all cells with different properties. Each project can contain several models, and each model can contain several 3D grids.

<i>Models pane</i>	Data connected with a 3D model (such as faults, trends, and 3D grids) is stored here. Velocity models and discrete fracture network models also are stored here.
<i>Nodes</i>	Points in the 3D grid where horizons intersect pillars.
<i>Pillar gridding</i>	The process of building pillars between the Key Pillars (fault pillars) to create a 3D grid. Quality check the result using the skeleton grid and grid intersections. The grid holds no layering information at this point.
<i>Pillars</i>	Vertical lines that connect the corner points of 3D grid cells. The shape can be any of the four standards: vertical, straight, listric, or curved.
	There are two basic types of pillars in a 3D grid: faulted and non-faulted. After the pillar gridding process, the key pillars are replaced with faulted pillars. Non-faulted pillars are inserted in the non-faulted area of the 3D grid.
<i>Ribbon</i>	The ribbon consists of the core tabs that are always visible (File and Home), domain tabs that can be filtered with the Perspective button, and contextual tabs that appear as you work with objects in the display window.
<i>Polygon</i>	You can use polygons made in Petrel for many different purposes. For example, you can digitize contours for structure maps, isochores, or properties. Boundary polygons are used in the Make surface process or as a boundary in the Pillar gridding process or Make aquifer process.
<i>Processes pane</i>	All processes that you can apply to your data are located here. For each process, a new set of tools is available in the Function bar.

Results pane

The numerical results of volume calculations and simulations are stored in this pane. You can visualize the results or create reports from them.

Segments (Regions)

In Petrel, an area that is closed by faults, a grid boundary, segment boundaries, or any combination of these items. Segment is similar to the equilibration regions used in ECLIPSE if there is no communication.

Simulation grid

The 3D grid that is used for flow simulation in Petrel. This grid also can be exported to other simulation packages. This grid usually is a coarser, upscaled version of the geological grid.

Skeleton

Three 2D grids that represent the top, middle, and base points of the Key pillars in a pillar grid. These grids are used to quality check the pillars and, therefore, the 3D grid. The skeleton is not related to horizons in the grid in any way.

Simulation case

When you define a simulation case, you specify the input properties (grid and properties from the **Models** pane), then selecting predefined initial conditions and fluid models (PVT), rock physics functions (relative permeability and rock compressibility), and development strategies (wells and rate control).

Status bar

Information on processes and coordinates in the bottom corner of the user interface.

Surfaces

2D grids (imported or generated in Petrel). A surface is a simpler version of a horizon in Petrel. The major difference is that horizons are held in 3D grids (as opposed to 2D grids) and, therefore, can have multiple Z values at each XY point. Surfaces are stored in the **Input** pane. Horizons are stored in the **Models** pane.

Templates

Templates are linked to objects in Petrel and globally control their settings for color, units, and measurements. Petrel comes with several predefined templates: depth and thickness color tables, property templates, and seismic color tables.

Tool palette

This floating window indicates which interactive tool is active. It allows you to select the tools that you need to complete your workflow without leaving the display window.

Templates pane

Color tables and all of the different templates are stored in this pane.

Title bar

The file name (project name) and location is displayed in the Title bar at the top of the user interface.

Transmissibility multiplier

The transmissibility is a number assigned to cell faces that describes the ability of a fluid to flow from one cell to another. You can assign a transmissibility multiplier to faults to change the ability of a fluid to flow past the fault.

Trends (in the Pillar gridding process)

User-defined guidance for the grid cells orientation in a particular direction (I- and J-directions). This option also is used as a segment divider (when there are no faults).

Visual filters

The **Visual filters** pane in Petrel provides a centralized view of the filters that are applied to the wells, 3D properties, and faults displayed in the active **2D window** or **3D window**.

Window toolbar

Automatically changes to show the most frequently used interactive tools for the active window type. For example, the camera linking tool appears for **3D windows**.

The Window toolbar is designed to keep window actions close to where you are working in the display window, but far enough away so that you do not click the tools accidentally. The toolbar is centered at the top edge of the active window; it moves from one window to the next as you work across multiple windows and monitors. It can be pinned open (the default) or collapsed when you do not need to focus on these tools.

Well tops

Intersection points between well trajectories and structural surfaces. Sometimes referred to as well points, well picks, or tie points.

Well trajectories

Lines in space that represent well paths.

Well section window

This Plot window displays well sections.

Windows pane

Stores all opened and active plots and windows used in the Petrel project. In addition, it holds the Light sources and the Cursor tracker.

Workflows pane

Stores workflows made using the **Workflow editor** and the Uncertainty and optimization process. In addition, it contains a folder with predefined variables.

Zones

The volume between a top and a bottom horizon defines a zone.

