Petrel E&P Software Platform 2014

WHAT'S NEW GUIDE

Petrel

Shared earth—critical insight

Schlumberger

Trademarks & Service Marks

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

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

Security Notice

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

Contents

Welcome to Petrel 2014

Finding, characterizing, and exploiting new and existing reservoirs is increasingly complex. To overcome these challenges, we must continually improve and innovate the way we work. With the Petrel* E&P Software Platform, Schlumberger revolutionized the oil and gas industry by bringing disciplines together with best-in-class science in an unparalleled productivity environment.

Our commitment to Petrel resulted in dramatic strides forward in the way we develop and deliver a software platform, and with the Petrel 2014 Platform and Update releases, we continue to deliver on our promise of better integration, deep science, and productivity. Today we support an engineering team unrivalled in size and expertise, empowered by the Ocean software development framework. More than ever before, we are positioned to help you develop critical insights into reservoirs throughout the oilfield lifecycle.

This document is updated for the 2014.6 release. The following sections are organized, as far as possible, by domain. Within each section you will find the New Features and any significant behavioral changes listed, grouped by the release in which they first appeared; with the newest release appearing first. Note that not all domains will have New Features in every release.

Refer to the companion documents, *Petrel E&P Software Platform 2014 Release Notes* and *Petrel 2014 Installation Guide*, for information on licensing and system requirements.

Documentation for all prior Petrel releases is available from the Software Integrated Solutions (SIS) Support Portal.

Petrel 2014 User Experience

Petrel 2014 delivers a step change in User Experience, keeping focus on relevant workflows, and providing quick access to the data in a productive and ergonomic working environment. Based on the widely accepted user interface methodologies from Microsoft, Petrel 2014 uses the Ribbon concepts to support a logical flow of activities for each individual user. Workflows are driven from in-context activities, where the user selects interactive tasks, interacting with and operating on the data directly. Petrel gives total freedom to access tools across the whole workflow by lifting the concept of an active process. Workflows productivity improvements have been measured and preliminary results show:

- Reduction of long mouse travel between 50-80%
- Reduction of mouse clicks by 30-40%
- Increase of the time spent on the data/interpretation tasks by 30-40%

The clean, aesthetic and logical design makes Petrel easier to adopt and learn, and makes the available technology more easily accessible. *Petrel 2014 enables you to truly focus on your workflows and on your data, delivering you a step change in productivity.*

User Experience: General

Process freedom

A significant improvement in workflow flexibility and productivity in Petrel 2014 is the removal of the concept of an active process, allowing dialogs to be opened and closed whenever you want while, at the same time, maintaining full access to all other Petrel tools.

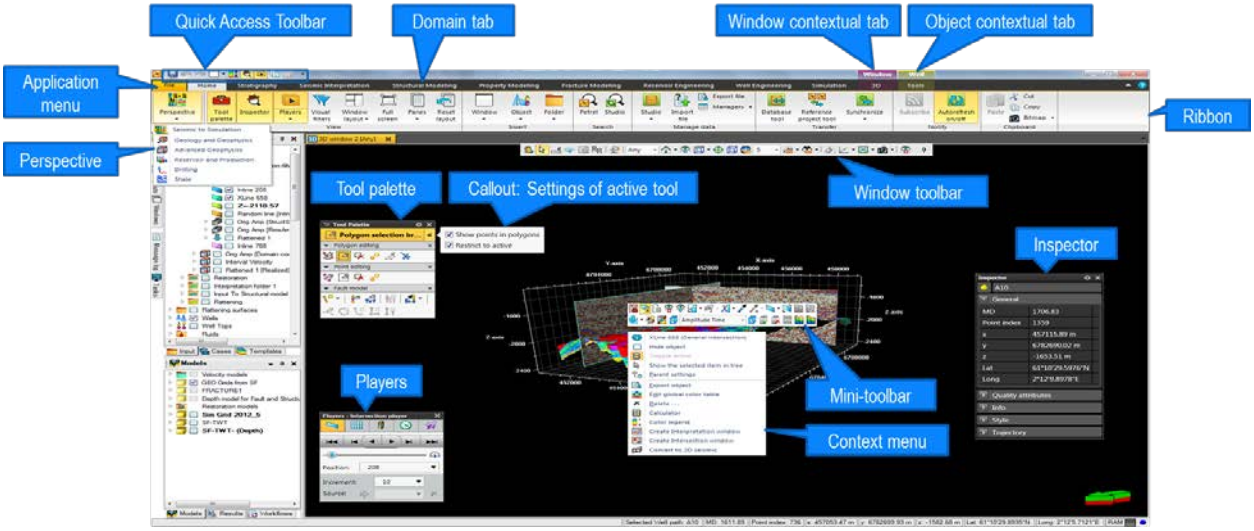
A long standing constraint to workflow productivity has been lifted by disconnecting the availability of tools from the concept of active process. The tools and dialogs are instead re-distributed in the interface and made available where and when you need them, either in larger scale workflows or close to the objects they apply to.

Not only are the tools disconnected from the processes, we are removing process concept in its entirety. The combined effect allows tools to be available all the time:

- You can interpret seismic and adjust polygons without the need to switch processes
- You can interpret well tops in the Well section window, edit polygons in the 2D window, and de-spike surfaces without the need to switch processes

New User Experience/Interface Elements

Figure 1: Overview and terminology of the user interface components contributing to the new Petrel user experience



Ribbon

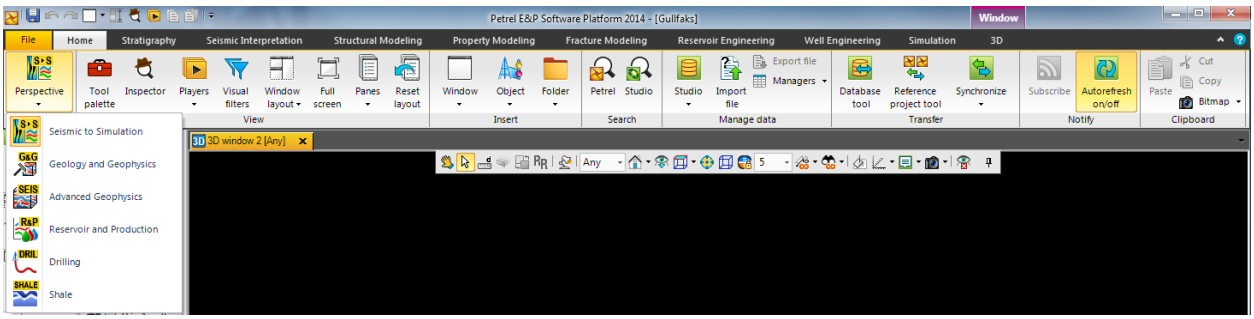
The most visibly noticeable aspect of Petrel 2014 is the user interface redesign, part of which is an implementation of the ribbon concept known from Microsoft's "Fluent UI" technology. Petrel exploits this technology by laying out industry relevant workflows in domain tabs, where each tab re-organizes and -distributes the familiar Petrel dialogs and tools in left-to-right sequences, guiding you through the work steps necessary to accomplish these end-to-end industry workflows.

Perspectives

Providing a targeted set of workflows and tools tailored to a key set of user profiles and themes, perspectives focus the user interface by only displaying a subset of the available tabs, workflows and tools in Petrel 2014.

Petrel as a platform contains and enables a large number of domains and sciences, each with a specific set of tools and workflows. Providing users with a mechanism to modify the user interface in line with their user type, domain or workflow, perspectives allow Petrel to offer a targeted working environment for the heterogeneous work force all the way down to the individual user.

Figure 2: Using Perspectives, the Petrel user interface can be targeted to your task at hand

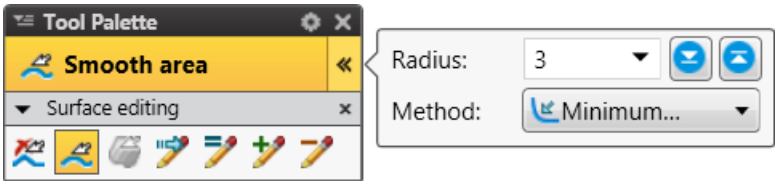


Tool palettes

Interactive tools and their associated settings are logically organized into tool sets that are available in a dedicated tool palette that can be launched in context of the current workflow or data object.

The tool palette is a floating window that indicates which interactive tool is active. It allows you to select the tools you need to complete your workflow without leaving the display window. Many tools have settings that refine the way they work, and when active those settings are also available through the tool palette in a dedicated callout section.

Figure 3: Interactive tools are now gathered in a common location with easy access to settings influencing their operation

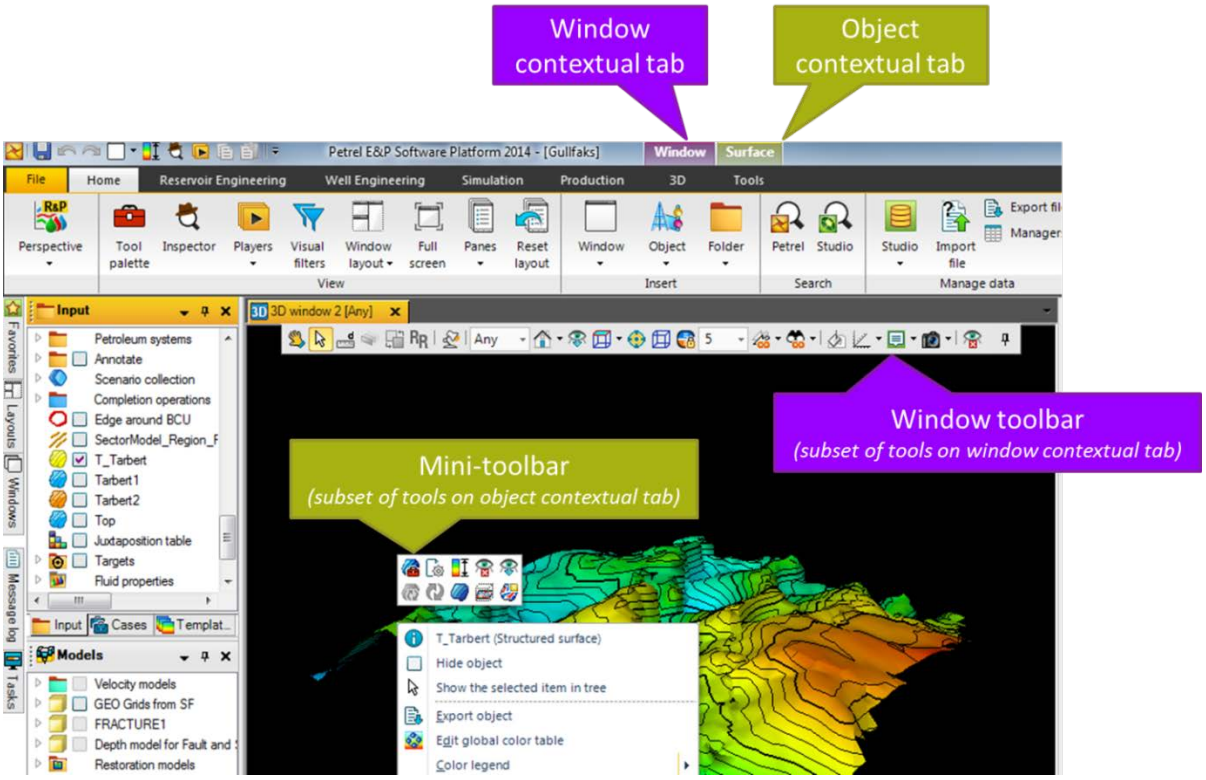


Contextual tool access

Petrel now provides the tools you require to complete your current task through a more intuitive interface using contextual tabs and mini-toolbars.

Contextual tabs provide access to common tools that you can use to modify the selected object, or alter the view in the selected window. The contextual tabs are automatically added to the end of the ribbon when a context is set, meaning that whenever a window is active, a window contextual tab is added, and whenever an object is selected an associated object contextual tab is added.

Figure 4: Tools that only apply in the context of windows or objects are gathered in contextual tabs and toolbars that surface only when the tools apply



Window contextual tab

Tools, dialogs, and settings that target how a Petrel window is set up, how it displays data, and how it is annotated and exported in terms of visual captures, are all collocated and presented in a window contextual tab.

Each window has its own contextual tab with tools that are specific to the window type (e.g., 3D window, Interpretation window, etc.). This is where you find functionality to set up the view, link windows, add display elements, annotations, and more. The window contextual tab is automatically appended to the ribbon when a window becomes active.

Figure 5: Example of a (3D) window contextual tab

Window toolbar

The most frequently used tools for each type of Petrel window are made available for quick and easy access inside windows themselves; thereby, significantly reducing the amount of mouse travel needed when working with multiple windows.

Window toolbars are displayed along the top edge inside the window and move from window to window as they are activated. The window toolbars are closely linked to the associated window contextual tabs. Whereas the contextual tabs contain the full set of tools applying to the active window, the window toolbar brings a subset of that functionality into the display window, close to where you are working.

Object contextual tab

Tools, dialogs, and settings that target a specific object type, either by operating on it or consuming it as input, are collocated and presented in an object contextual tab.

Each object has its own contextual tab with tools that are specific to the object type (e.g., surface, polygon, 3d grid, etc.). This is where you find functionality that acts on or consumes the selected object, such as setting color tables, editing tools, operations, database synchronization, and more. The object contextual tab is automatically appended to the ribbon when an object becomes selected in the active window or pane.

Figure 6: Example of an (surface) object contextual tab

Mini-toolbar

The most frequently used tools for each object type are made available for quick and easy access close to the data, displaying above the familiar context menu that appears when clicking with the right mouse button on objects displayed inside a window.

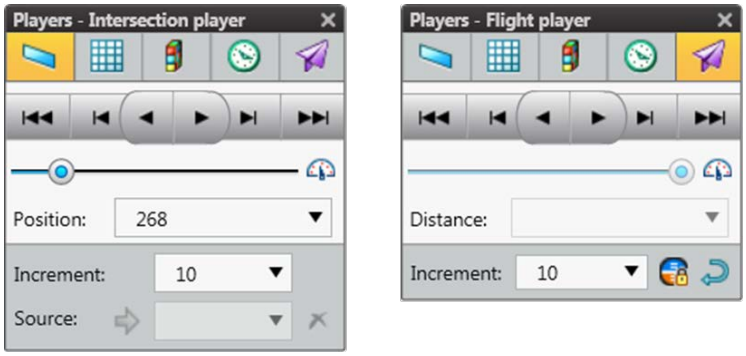
Mini-toolbars only appear in the context of objects and are closely linked to the associated object contextual tabs. Whereas the contextual tabs contain the full set of tools applying to the object in context, the mini-toolbar brings a subset of that functionality into the display window, close to the object and where you are working.

Players

All players in Petrel are now available in a common dialog, sharing the same general look and feel, and when being displayed are visible irrespective of active window and active object.

Gathering the Petrel players in a single control interface that you can explicitly display and hide allows for a more stable user interface. The players can be accessed when and where needed by invoking context dependent launchers on contextual tabs and mini-toolbars, or from the **Home** tab.

Figure 7: Players in Petrel have been consolidated in a common location with a consistent look-and-feel shared by all the players

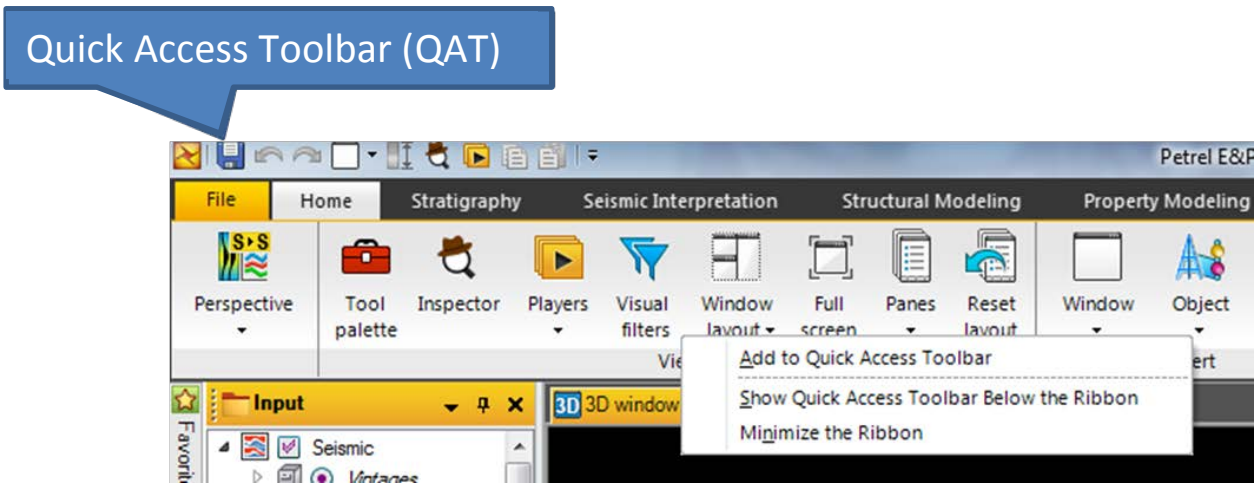


Quick Access Toolbar

All tools available anywhere on the ribbon can be added to the Quick Access Toolbar which sits at the very top left of the application, is available irrespective of the chosen perspective and tab, and has efficient shortcut access via KeyTips.

Right-clicking with the mouse on any tool or tab group in the ribbon gives the option to add to the Quick Access Toolbar.

Figure 8: The Quick Access Toolbar is a user customizable area where you can add tools and tab groups by right-clicking these elements in the ribbon

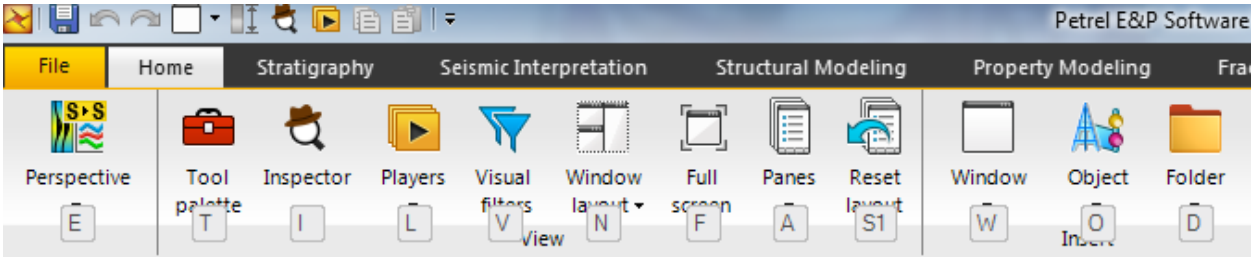


KeyTips

The keyboard can be used to access all tools available anywhere on the ribbon by simply hitting the Alt key and following the subsequent sequence of auto-generated key tips appearing on the screen.

The KeyTips are provided as a pure usability feature for easy keyboard access to functionality available in the ribbon. The auto-generated nature of these KeyTips makes them different from the standard Petrel shortcuts.

Figure 9: KeyTips offer easy keyboard access and navigation of the ribbon, and is consistent with similar functionality in Microsoft Office products



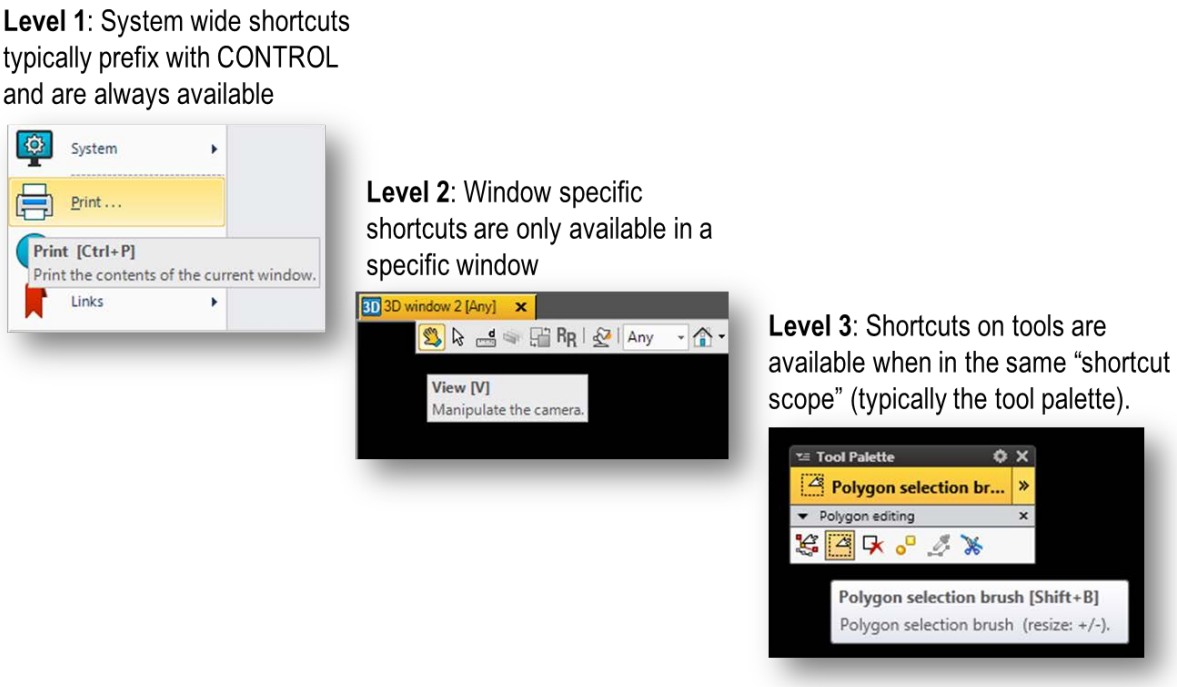
Shortcuts

Greater flexibility and re-use of shortcuts between interactive tasks is now possible due to a more explicit focus on the various levels of shortcuts available in the system.

The tool sets found in the Tool Palette have in Petrel 2014 been introduced as a new shortcut scope. Previously scoped to active process, shortcuts for interactive tools are now reused in accordance with the tool set they belong to. The new shortcut scope works in cooperation with the application and window level scopes.

This means that when in polygon editing, shortcuts will refer to tools within polygon editing, the window, or the system. Shortcuts relating to seismic interpretation for example are not accessible, until the moment you switch to any of the seismic interpretation tools (and thus change the "shortcut scope").

Figure 10: Petrel shortcuts are logically organized in various levels (or scopes), offering flexibility and re-use of shortcuts between interactive tasks



Layout

Petrel 2014 comes with a new default pane layout offering a more logical organization of panes that better support user profiles and associated work and pane access patterns.

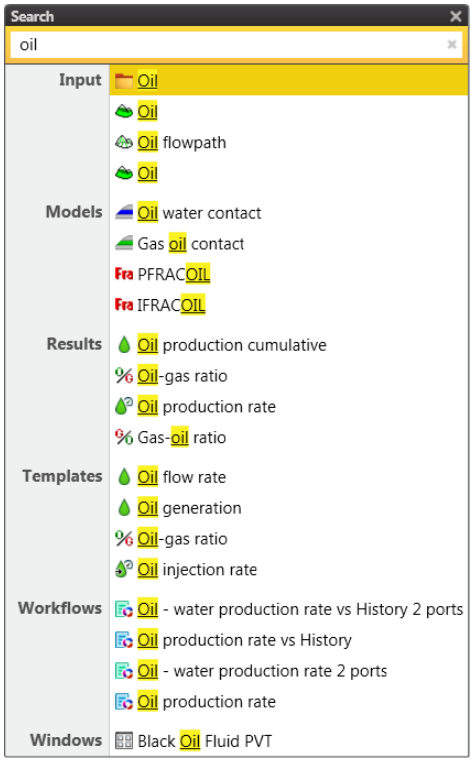
The new organization optimizes data access patterns by distributing commonly accessed panes in non-conflicting locations. For instance, geologists and geophysicists commonly switch between the Input and Models panes, whereas reservoir and production engineers often access the Cases and Results panes. Hence, to minimize pane tab switching, the panes belonging to each pair are now split and put into the upper and lower pane sections respectively. The **Favorites** pane will now, when pinned, appear to the right of the Input/Models or Cases/Results pane pairs for easy dragging and dropping of favorite data entries. In addition, the process pane is no longer part of the default layout since Petrel does not use the concept of processes anymore, as explained in the section, General, Process freedom on page 19.

Searching

New **Ctrl+F** search functionality provides quick searching, selection and execution of Petrel project data and dialogs found in panes.

The search tool traverses all panes in Petrel and presents matches in a list tagged with the panes where they appear. Selecting any of the results causes the entry to become selected and the pane brought forward.

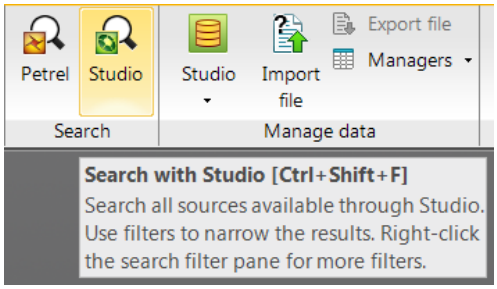
Figure 11: A new and powerful Ctrl+F search functionality is introduced, providing you a quick and easy way of locating data in your Petrel project



Tool Tips

Improved description of tools and their shortcuts are now available in Tool Tips that appear when hovering over buttons in the interface.

Figure 12: Petrel 2014 comes with richer and more elaborate set of tool tips



Mini-toolbar and Context Menu access in View mode

It is now possible to bring up the object mini-toolbar and context menu while in View mode by simply clicking with the right mouse button on objects. This allows you to more efficiently navigate and interact with the data in the display window.

Enrich Your Workflow

Enrich Your Workflow is a new feature in Petrel 2014 that gives you the information you need to get the most out of your Petrel experience. See "Enrich Your Workflow" on page 250 for more information.

User Experience: Geophysics

General

Seismic Interpretation tab

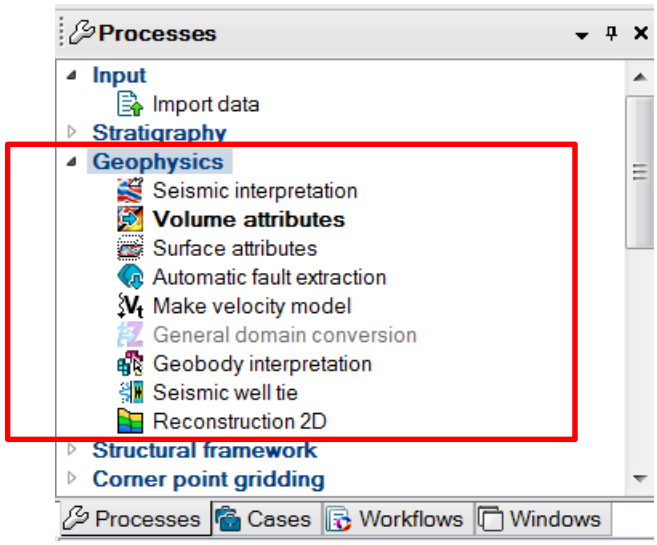
In the new 2014.1 User Experience, key Geophysics features and tools can be accessed through any of the following workflow perspectives: **Seismic to Simulation**, **Geology and Geophysics** or **Advanced Geophysics**. These perspectives give access to the **Seismic Interpretation** tab, which provides these features and tools in the following main sequence grouping starting from the left:

- Setup
- Seismic-well calibration
- 2D/3D interpretation
- Attributes
- Volume interpretation
- Structural framework
- Depth
- Utilities

Since processes are not used in the new 2014.1 User Experience, some of the Geophysics features and tools are located in a different location. The following screen shots demonstrate the differences between the Classic view in Petrel 2014.1 and the new Petrel 2014.1 User Experience.

Figure 13: New Petrel 2014.1 User Experience view – Geophysics Tools/Features

Figure 14: Petrel 2014.1 Classic View – Geophysics Tools/Features



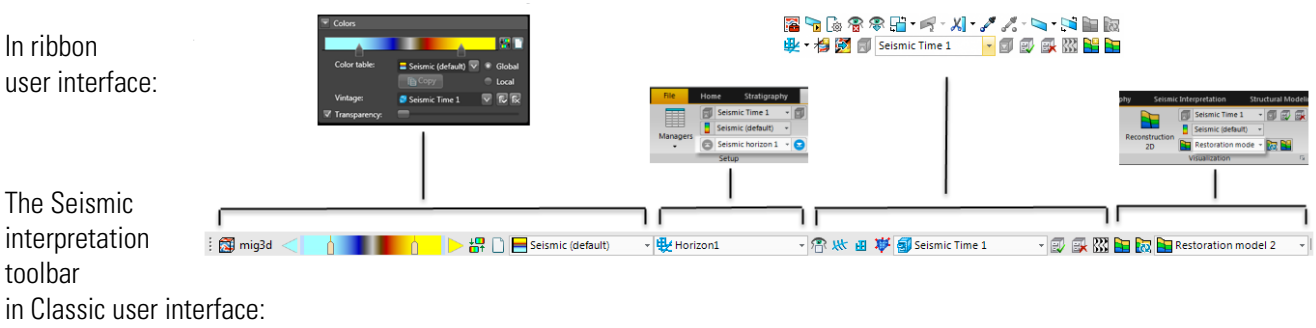
Features/tools are grouped logically within each group of tasks located under the **Seismic Interpretation** domain tab so you can find tasks quickly based on the context of your work, for example Volume attributes and Surface attributes can be found in the Attributes group, while Geobody interpretation is found in the Volume interpretation group. In a typical project, you may start your work path from the left of the tab, with Setup tasks, and progress towards the right of the tab, to Depthing tasks.

Alternatively, key features and tools can be accessed directly from and in context of selected data.
As you work interactively with your seismic data, you can use the following ribbon features, most of which automatically adapt their tools and information for the type of data object being manipulated:

- Tool palette (well tie editing, seismic interpretation, geobody interpretation, polygon editing)
- Toolbar and contextual tabs for windows (Interpretation window)
- Mini-toolbar, contextual tabs and context menu for objects (seismic, horizon interpretation, fault interpretation, fault patch, probe, geobody, multi-Z interpretation)
- Inspector (seismic, horizon interpretation, fault interpretation, multi-Z interpretation)
- Player (Intersection)

The tools located on the Seismic Interpretation toolbar that exist in the Classic user interface are regrouped separately to be in closer reach from the different objects of interest. As in the Classic user interface, you can access a selection of the seismic tools readily via the Inspector. You can now access the remaining seismic and interpretation tools via the new object interfaces e.g., the Seismic mini-toolbar by right-clicking anywhere on a seismic section. The current active interpretation objects are indicated towards the left corner of the ribbon.

Figure 15: The Classic Seismic interpretation tool has been regrouped for easy access in the 2014.1 ribbon user interface



Modeless access

The new modeless access to features and tools allows multiple dialogs to be opened simultaneously for a productive workflow, for example in the following combination from:

- Make/edit surface
- Make velocity model
- Seismic interpretation
- Geobody interpretation

This allows for continuous seismic interpretation tasks while engaged in the following example simultaneous activities, without the need to change a process:

- Seismic interpretation
- Creating polygons
- Generating surfaces
- Generating volume attributes.

Focused object and process activation

When working in a display window, objects and process modes are now activated and focused automatically. For example:

- Clicking an object in Select mode, automatically activates the object for subsequent use.
- Creating a new horizon, fault or multi-Z object, automatically puts you in the appropriate Interpretation mode.
- Inserting a crop volume, automatically puts you into Select mode.
- When generating a surface, choose the option to automatically display it upon completion.

Quantitative Interpretation

Quantitative Interpretation tab

In the new 2014.1 User Experience, key Geophysics features and tools can be accessed through any of the following workflow perspectives: **Seismic to Simulation**, **Geology and Geophysics** or **Advanced Geophysics**. The Advance Geophysics perspective gives access to the **Quantitative Interpretation** tab, which provides these features and tools in the following main sequence grouping starting from the left:

- Setup
- Rock physics
- Seismic-well calibration
- 2D/3D interpretation
- AVO
- Seismic inversion
- Crossplot

Features/tools are grouped logically, with each group of tasks located under the **Quantitative Interpretation** tab, so you can find tasks quickly based on the context of your work.

Figure 16: New Petrel 2014.1 User Experience view – Quantitative Interpretation tools and features

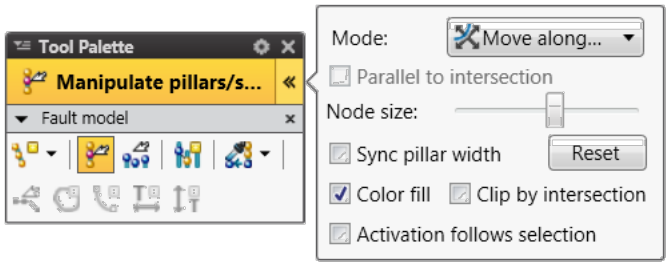
User Experience: Geology

General

Corner Point Gridding is available in two ribbon perspectives: **Seismic to Simulation** and **Geology and Geophysics**. These processes now work concurrently, for example you can have the **Make Horizons** dialog open while updating the input in **Make/Edit Surfaces**. The **Make/Edit Surface** process, which is fundamental to many workflows, can be found in most domain tabs.

Tools for the **Pillar Gridding** and **Fault Modeling** processes are combined into a single tool palette. Tool palettes are also available for Surface editing and 3D Grid editing. The tools to create and edit a proposed well are available in the Well Design tool palette.

Figure 17: The Fault model tool palette

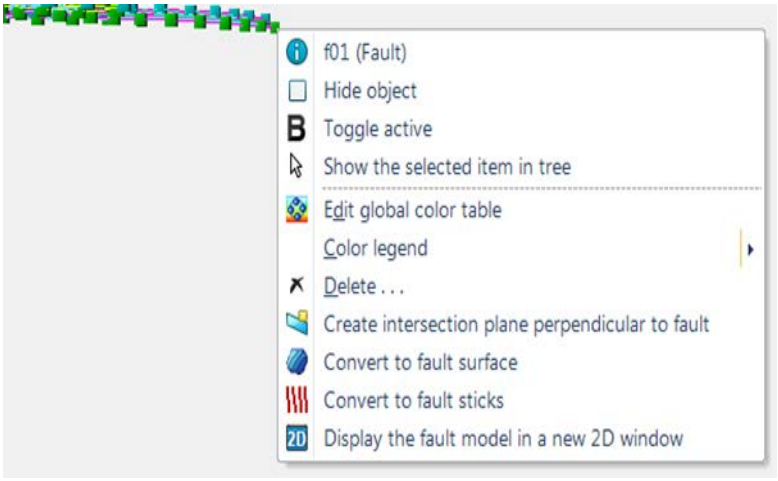


Other usability improvements include:

- Option to display fault models in a new 2D window using the right mouse button
- Option to display surfaces when running Make/Edit surfaces

Mini-toolbars provide immediate and localized access to object specific functionality. These are available for Wells, Well Tops Logs, Fault Models, Surfaces, and 3D Grids.

Figure 18: Mini toolbar for a fault.



Structural Framework

Process conversion into dialogs

The workflow steps of the construction of a Structural Model, going from the creation of a Fault Framework to the calculation of a Depospace will no longer block anymore the user from being able to use other tools simultaneously.

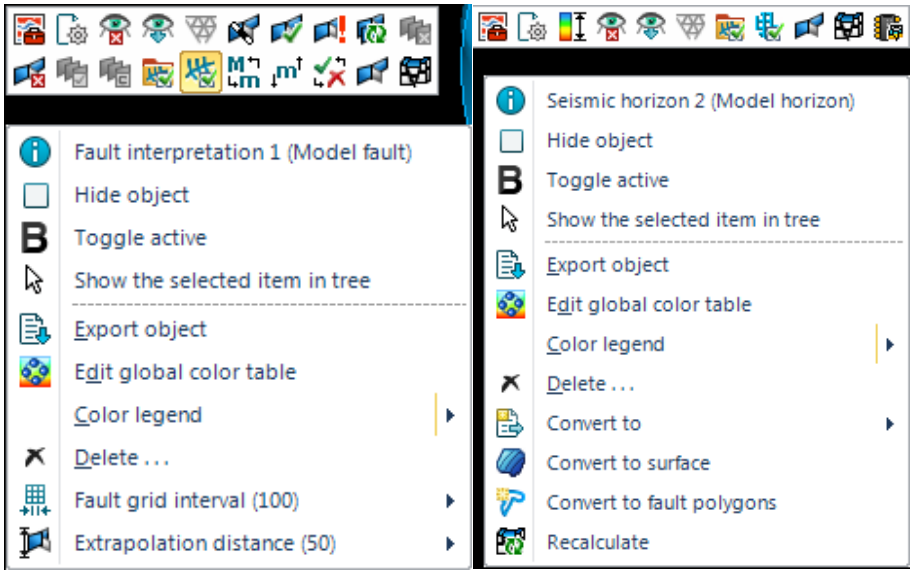
Several processes in Petrel have been update so that they can be used in combination with others. For example, you can have Horizon Modeling and Fault Framework open while still being able to edit a seismic horizon or a fault interpretation.

Fault and Horizons Mini-Toolbars

Globally in Petrel, mini-toolbars have been introduced to allow easy access to tools related to displayed objects in any given window. Faults and Horizons of the Structural Model are adhering to the rule.

Bringing the QC and editing tools in context with the workflow motivated the creation of the mini-toolbars.

Figure 19: Illustration of the mini-toolbars for Faults and Horizons of the Structural Model



For Faults and Horizons, the set of tools are logically outlined to guide the user though the QC workflow. You will always be able to open the Fault and Horizon Modeling dialogs directly from the object, as well as call open the interpretation tool palette to get access to the interpretation editing tools.

Modeling

Property Modeling is available in two ribbon perspectives: **Seismic to Simulation** and **Geology and Geophysics**. The Property Modeling ribbon contains all the functionalities for performing Facies and Petrophysical modeling. The ribbon tools are classified by function groups and organized in a logical order from left to right, to capture typical reservoir modeling workflows. The tool dialogs now work concurrently, for example you can have both the **Facies** and **Petrophysical modeling** dialogs open while updating the input in **Data Analysis**.

The new Petrel user experience increases productivity by keeping the focus on your workflow, and more importantly, on your data. In addition to the context menu, the mini toolbar gives you access to key interactive tools for the displayed property. For example, the mini toolbar provides quick access to the Facies tool palette, property settings, and the most useful tools for window display options such as the Property player.

Figure 20: Property mini toolbar and context menu, opened from the data in the window display

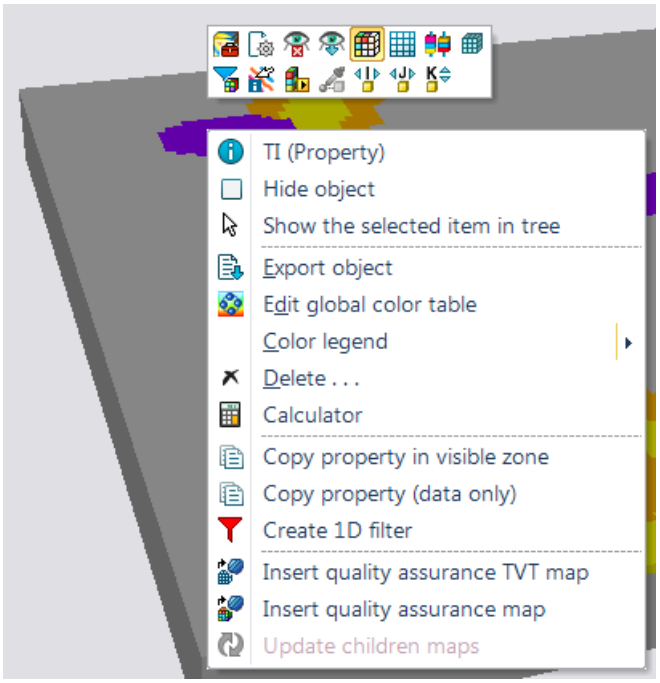
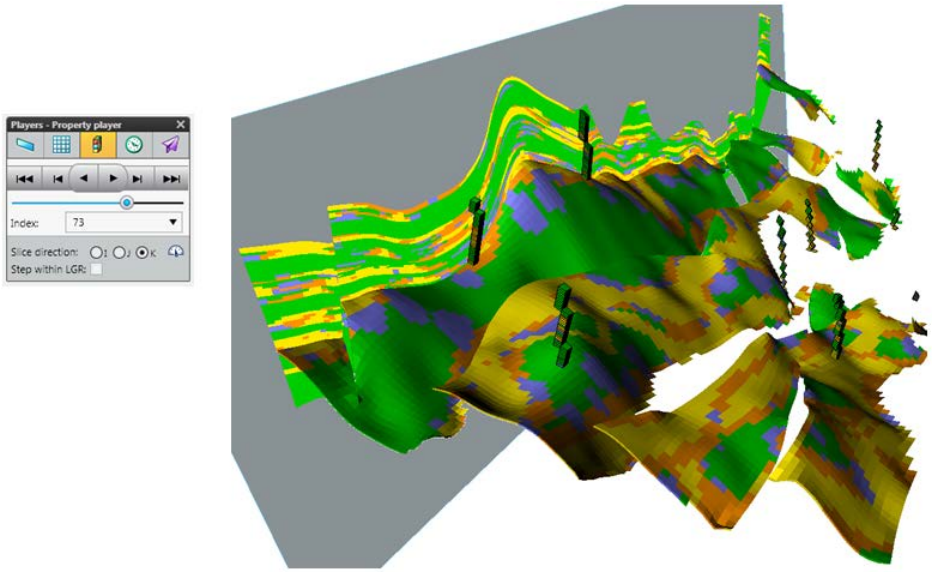
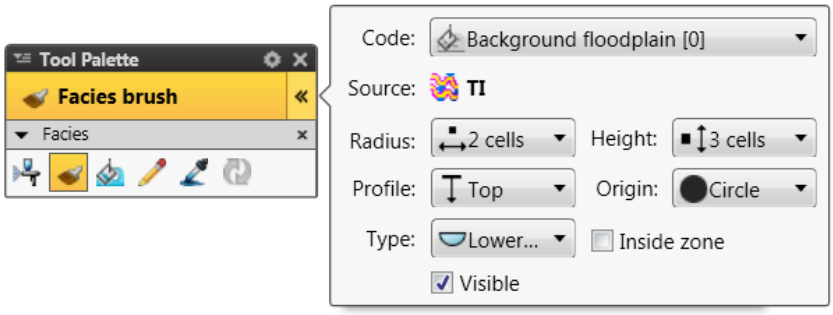


Figure 21: Property player, opened from mini-toolbar



The Facies tool palette combines tools for interactive facies editing into one location. When you select a tool, the tool palette expands the settings callout so that you can easily change the tool settings.

Figure 22: Facies tool palette, expanded to show Brush tool options



User Experience: Reservoir Engineering

The new Petrel 2014 user experience brings a clearer, easier to learn, more logical Petrel for each user. Reservoir engineering users particularly benefit from the new 2014 interface which can be configured in domain perspectives that present, for example, reservoir and production functionality in logical left to right order for petroleum engineering workflows. New UI widgets such as the mini-toolbars, players and the Inspector greatly improve your productivity by enabling you to access the functionality you want when and where you need it.

A selection of videos to help users migrate to 2014, illustrating reservoir engineering and simulation workflows being carried out in the new user experience is available from the SIS software download center.
In the **Reservoir and Production** perspective, there are five tabs: **Home**, **Reservoir Engineering**, **Well Engineering**, **Simulation** and **Production**.

1. In the **Home** tab, common to all perspectives, you set up your project, arranging the windows and managing data through Studio or Reference project tool.
2. The **Reservoir engineering** tab takes you left to right from data import to a reservoir model ready for reserves calculation and simulation initialization.
3. The **Well engineering** tab is for well and completions design and well performance calculation, leading to the field development strategy built either in Development Strategies for ECLIPSE or in Field Management for INTERSECT
4. The **Simulation** tab provides the tools to import or create, manage, and analyze single and multiple dynamic reservoir models, including history matching, uncertainty and production optimization studies.
5. The **Production** tab provides the tools for production analysis and production interpretation workflows.

Generic usability and performance enhancements

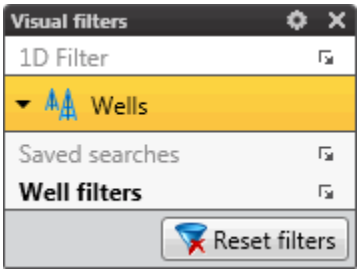
Visual filters tool

The **Visual Filters** tool on the **Home** tab allows you to see at a glance the filters that are applied to wells and grids displayed in the **2D** or **3D window**, and then reset or change applied filters.

The **Visual filters** tool can be displayed or hidden by clicking **Visual filters** in the **View** group on the **Home** tab or by using the **Ctrl+F8** shortcut. Currently the **Visual filters** panel supports **2D** or **3D windows** only.

Filter display

The **Visual filters** panel shows the filters that are applicable to the wells, 3D properties or faults currently displayed in the active display window. For example, if a well is displayed in the active window then **Saved searches and Well filters** are visible.



When a filter is applied and the filter's settings deviate from their default state, the caption of the corresponding filter item in the **Visual filters** panel is shown in bold to provide you with a visual cue that the filter is in use in the active display window.

Every displayed filter contains the shortcut link on the right side of the filter name. Click the shortcut to navigate quickly to the filter in the Petrel tree. For some filters, a dialog with the filter's setting will open.

Supported filters

The following filters are supported and displayed by the **Visual filters** panel:

Generic

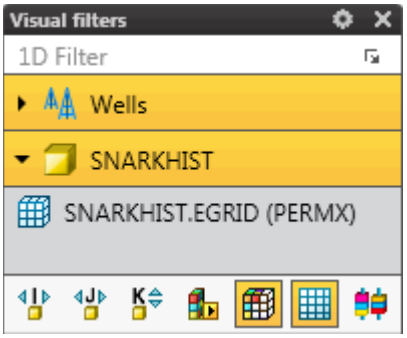
- 1D filter

Wells

- Saved searches
- Well filters

3D model and simulation properties

In the caption of the Grid section of the filters, 3D property name is displayed next to the grid name. If a simulation property is displayed, case name is also displayed.



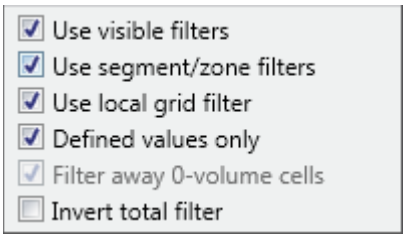
Buttons under the **Grid name** provide you with quick access to:

- I, J, K slicing
- Property slice player
- Solid and/or grid rendering
- Upscale filter

Shortcuts are provided to the following filter types:

- Segment filter
- Zone filter
- Local grid filter
- Value filter - high level control for the Value filter
- Index filter - high level control for the Index filter

In addition to the Value and Zone high level filter controls, the **Visual filters** panel also displays filter settings defined on the **Filter** tab of the corresponding 3D property folder.



Faults

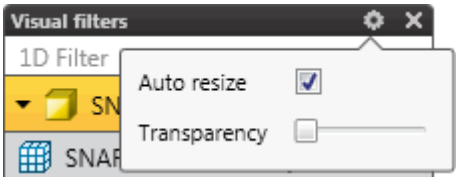
- Fault filter

Reset filters

To reset all applied filters to their default state, click **Reset filters**. The default state depends on the filter type. Only the filters that are applied to the objects displayed in the active display window are reset.

Settings

You can change the transparency and re-size behavior of the Visual filters panel by modifying its properties. To do this, click **Settings**. By default, the resizing of the **Visual filters** panel occurs automatically based on the size the individual filters displayed.



Global search

The global search tool (CTRL-F) is a new general Petrel tool to find data or dialogs throughout your Petrel project.

Large model performance

The whole Petrel platform, all domains, has addressed aspects of the performance of Petrel for large well count, large grid-cell count projects in the 2014.1 release. Performance covers the interactivity of Petrel when a large project is in use, as well as the time taken for calculation intensive tasks or an overall user workflow. Projects with tens of thousands of wells, and containing grids with hundreds of millions of cells have been targeted. Large simulation case export performance improvements have targeted INTERSECT simulation cases, but improvement has also been achieved for ECLIPSE simulation cases. The interactivity of the Petrel 3D window when visualizing 15,000 wells, or using an intersection plan through a grid of over 200 million cells has been improved by 1-2 orders of magnitude.

User Experience: Exploration Geology

Exploration Geology processes on Petroleum Systems and Risk tab

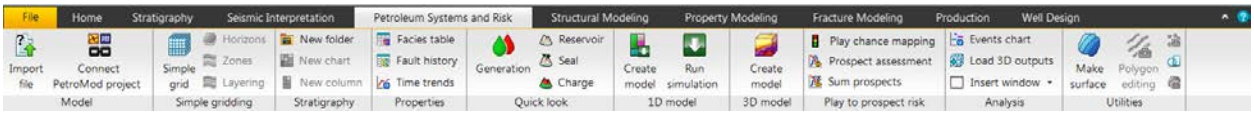
Exploration Geology processes within the context of Geology and Geophysics are now part of Petrel's ribbon structure and are accessed on the **Petroleum Systems and Risk** tab.

The position of the processes on the tab guides you through the workflow of screening a petroleum system, preparing petroleum system models for simulation in PetroMod, and performing risk assessments. Access to PetroMod 3D petroleum systems model simulation results has been simplified. When connected to a PetroMod project, all 3D model data can be found on the Results pane, a pre-selection of properties has become unnecessary.

In the new 2014.1 User Experience, key Exploration Geology features and tools can be accessed on the **Petroleum Systems and Risk** tab, which provides these features and tools in the following main sequence grouping starting from the left:

- Model
- [Simple gridding]
- [Stratigraphy]
- Properties
- Quick look
- 1D model
- 3D model
- Play to prospect risk
- Analysis
- [Utilities]

Figure 23: Petroleum Systems and Risk tab in Ribbon view



This setup guides you through the entire workflow of screening a petroleum system, preparing petroleum system models for simulation in PetroMod, and performing risk assessments.

User Experience: Drilling

General

In the Petrel 2014.3 ribbon mode, the **Drilling** perspective gives access to the **Well Design**, **Well Positioning**, **Offset Well Analysis** and **Real-time** tabs.

- The **Well Design** tab consists of the key well design tool groups, such as **Target**, **Well path**, and some other groups which could be used in the Well design workflow such as **Well**, **Trajectory uncertainty**, **Drilling simulation, Completion** and **Utilities** groups.

- The **Well Positioning** tab consists of the **Trajectory uncertainty**, **No-go zone** and **Anti-collision** tool groups and some other groups which could be used in the Well positioning workflow such as **Well** and **Well path** groups.

- The **Offset Well Analysis** tab gives accesses to import offset well data, manage EDM data index items and manage the drilling risks. You can also use the **Well**, **Well tops and zones,** and **Utilities** tool groups to help you manage wells and make marks.

- With the **Real-time** tab, you can stream real-time data, such as logs, events, and trajectories from InterACT*, as well as other third-party WITSML data and files servers.

User Experience: Production

Well Deliverability

Well Deliverability now supports the new ribbon design. It co-exists under both perspectives **Production and Reservoir**, and **Seismic to Simulation**. This brings easier, clearer, and organized steps to follow a workflow in a logical order from left to right. Having a reservoir and well model, you start by quality checking the well model, retrieving fluid properties, associating fluids to the well and reservoir, and optimizing the algorithms for an enhanced Well performance simulation. The following features are available from the **Well Deliverability group** ribbon.

Option Name	Description
PIPESIM import	Importing fluid and well data from PIPESIM
New fluid properties	Creating a new fluid property, and performing fluid property data matching
IPR manager	Associating the fluid-well and reservoir mode, managing IPR properties for perforations, and generating IPR curves
FC data matching	Creating or editing a flow correlation data matching case for single or multiple wells, and optimizing flow correlation for vertical and deviated wells
VFP tables	Creating or editing a VFP table collection for single or multiple wells
VFP Results	Displaying Performance Table Results, and applying quality control rules to prepare the table for Predictive Development Strategy
Nodal analysis	Creating a nodal analysis case for single or multiple wells, running a nodal analysis case, and creating a nodal analysis chart

As part of the Microsoft Fluent UI elements, Well Deliverability functionalities contain contextual tabs for all of the objects created after a simulation is run.

The following features are available from the **Nodal Analysis Tools** contextual tab.

Option Name	Description
Run for all wells 	Running a nodal analysis case
Settings 	Viewing and editing the general appearance settings for a nodal analysis case
Reload results 	Loading nodal analysis results for a different nodal analysis case
Spreadsheet 	Viewing nodal analysis results in a spreadsheet, and creating nodal logs
Summary and details 	Viewing, searching, and exporting the output summary and details log file for a nodal analysis case

The following features are available from the **FC Data Matching Tools** contextual tab.

Option Name	Description
Settings 	Viewing and editing the general appearance settings for a flow correlation data matching case
Reload results 	Loading simulation results for a different flow correlation data matching case
Chart 	Displaying flow correlation data matching results graphically
Spreadsheet 	Viewing flow correlation data matching results in a spreadsheet, and committing the results
Summary and details 	Viewing, searching, and exporting the output summary and details log file for a flow correlation data matching case

Well Deliverability also provides the following Custom element.

Inspector	Description
Nodal analysis	Viewing a nodal chart X/Y values of an inflow or outflow curve
Flow correlation data matching	Viewing the X/Y values of a curve displayed on a flow correlation data matching chart

Production Analytics

Production Analytics Group is part of the Production domain tab. The Production domain tab exists under the **Reservoir and Production** perspective. All Production Analytics now supports the new ribbon design. The following features are available from the **Production** ribbon.

Option Name	Description
Import (Well data group) 	Importing OFM well data
Results charting 	Creating or editing a crossplot chart for analyzing well information in time
Production mapping 	Creating or editing a production grid map to identify geospatial trends and anomalies in production data
Production bubble mapping 	Creating or editing a production bubble map to display large amounts of data to help identify trends, anomalies, or potential problems in an area of interest
New split set 	Creating a new split set to back allocate wellbore production to the completion level
Split set manager 	Managing split sets by reallocating wellhead production rates to completion levels

Production Analytics also provides the following Custom element.

Inspector	Description
Bubble map	Viewing bubble map attributes (property rates and associated percentages, values and units), and changing the bubble map style
Production grid map	Viewing production grid map attributes (cell value, nearest node, oil production rate, X/Y values), and changing the grid map style

Geophysics

Geophysics: General

2014.5

Horizon 3D snap (behavioral change)

The "**3D snap**" has been updated with a new fixed-size sinc interpolator to produce more accurate results when snapping. Additionally, the amplitude attribute stored with the 3D tracked horizon will be updated accordingly.

In the case where a 3D horizon is autotracked on low frequency (e.g., broadband) seismic data, subtle stair-stepping artifacts may be observed. Applying the renewed "3D snap" option with "Max vertical delta" set to 0 will remove these artifacts by correctly snapping the horizon to the relevant signal feature.

Performance

The following enhancements include:

- Improved robustness in **saving** projects when working with horizon interpretations.
- **Closing** a project with large regional seismic and temporary interpretation now takes seconds.

2014.4

Synthetic seismogram (behavioral change)

The synthetic seismogram bitmap now displays with smooth interpolation in the 3D and Interpretation window.

Horizon Interpretation (behavioral change)

An option to switch off the **live update** display of the 3D autotracker has been added to the Style tab of the Seismic Interpretation Settings dialog. This setting is valid for the duration of a session only; however, it will also be saved with the project from version 2015.1 onwards.

Geobody probes (behavioral change)

The option to use the **Opacity: sum** mode when co-rendering triple volumes has been added.

Performance (behavioral change)

The speed of amplitude **display for a random line** in the Interpretation window has been improved in the case when the random line with restricted horizons is rotated in a linked 3D window.

2014.3

New file import format “General 3D interpretation lines”

Horizon interpretation data defined in columns of inline, crossline and z can now be imported via **File > Import** and by using the new file format “General 3D interpretation lines”.

Seismic Overlay in the Interpretation window

Seismic Overlay allows co-visualization of seismic attributes/vintages within the same survey, in the Interpretation window, which is advantageous for numerous seismic interpretation workflows. Seismic Overlay is available in the Interpretation window through the Inspector. Overlay vintages are automatically filtered, based on the geometry of the background vintage to avoid a long list of vintages. It is available for both 2D and 3D seismic data. You can interpret horizons, faults and also run reconstruction for QC and validation of interpretation, while co-visualizing two vintages.

Figure 24: Seismic Overlay display in the Interpretation window: Seismic Amplitude vintage with wiggle as background and Seismic Velocity vintage as foreground

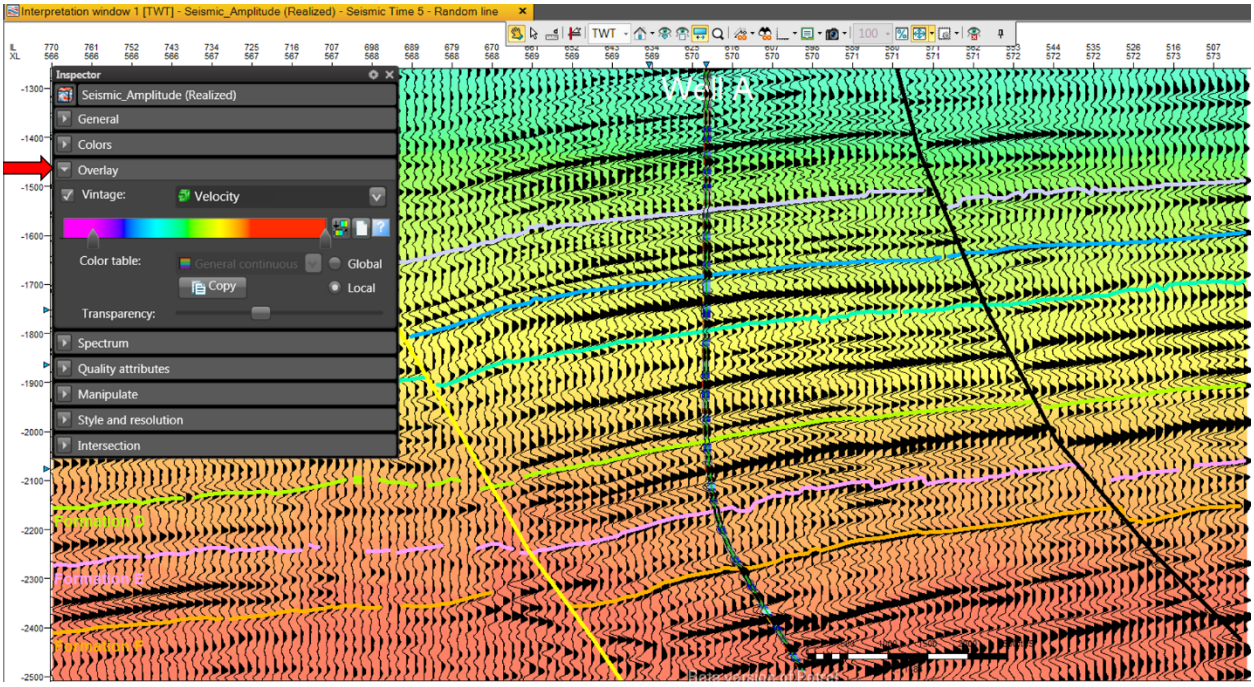
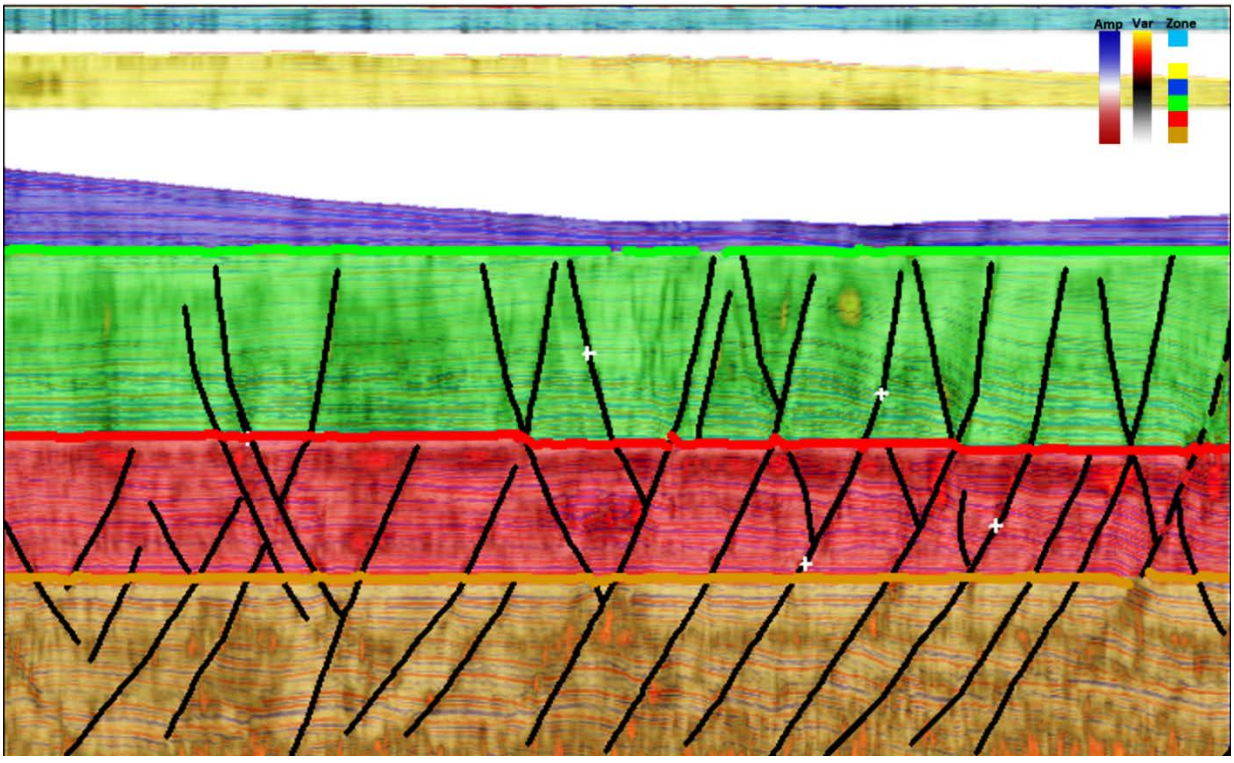


Figure 25: Seismic amplitude vintage with variance vintage as overlay and zones from the reconstruction process, for picking and validation of seismic interpretation in the Interpretation window. (Data courtesy of Geoscience Australia)



Volume attributes (behavioral change)

Updated and improved "Consistent Dip" attribute

The updated Consistent Dip provides a very precise volumetric Dip estimation.

The Consistent Dip estimation is based on an iterative global optimization method to calculate the dip and honors global consistency constraints of reciprocity, causality, consistency, and continuity.

Consistent dip properties:

Reciprocity
This principle states that there should be symmetry in the estimate of seismic dips between neighboring traces (both in inline and crossline directions and increasing and decreasing trace index numbers).

Causality
The causality principle describes the relation between a cause and an effect, where the effect is the consequence of the cause. In sedimentary layers, the law of superposition states that the sedimentary layers are deposited in a time sequence, with the oldest on the bottom and the youngest on the top. This is extended to seismic reflections unless layers are disturbed due to tectonic or other reasons. The principle of causality is violated if seismic reflectors (e.g., a peak and a trough) swap places from one trace to the next (i.e., they have flipped) and if this happens, the dip calculation should be discarded.

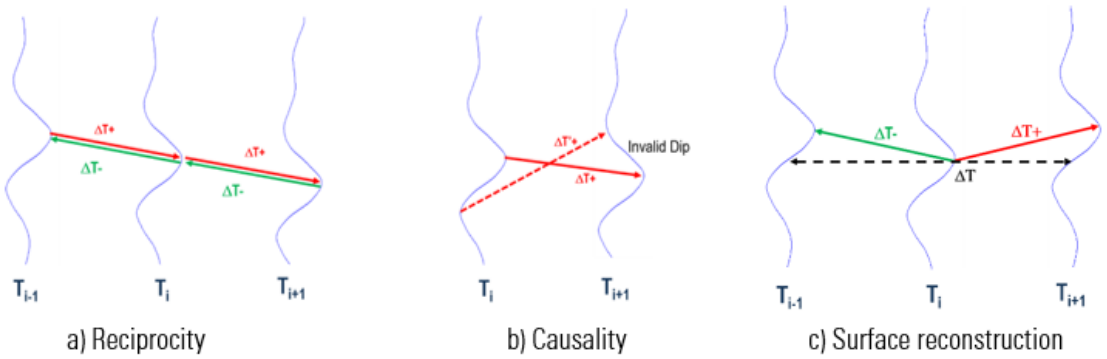
Consistency
If the correct dips have been estimated at each location both in inline and crossline directions, any chosen path from one sample at an arbitrary location (x, y) should lead to the same sample at another arbitrary location (x_i, y_i).

Continuity
In presence of discontinuities due to faults/channel cuts or due to poor signal-to-noise ratio, a robust method should not try to estimate the dip, but instead the method should generate a byproduct (residual dip or dip quality attribute) highlighting the area where the dip estimation is not valid.

XYZ versus IJK Space
Consistent Dip estimation operates in XYZ space (Cartesian coordinate system) for the correct computation of seismic dips rather than IJK space (Image coordinate system), which works on a basic assumption that seismic samples are equidistant in all three dimensions. Dip estimation methods that do not honor the true bin spacing will amplify the apparent dip in one direction (with more spacing, in general crossline direction). Dip estimation in the XYZ space imply that the dip fields are stored in the correct (signed) unit, which is in 'ms m^{-1}' for seismic data in the time domain, and 'm m^{-1}' for seismic data in the depth domain.

Surface reconstruction
For proper reconstruction of a local surface from a dip model, negative and positive dips need to be estimated at each seismic sample location, both in inline and crossline directions instead of estimating the average dip, which measures a single slope value extending over two traces.

Figure 26: Shows the schematic of global consistency constraints, which Consistent Dip honors



Instead of the nine Consistent Dip outputs, you can now generate only two outputs based on the selected output method type: Dip and azimuth, Inline and crossline Dip, High resolution Dip, or Residual dip and Dip RMS.

Figure 27: a) Consistent Dip attribute now provides two outputs based on the method selected
b) You have the choice of four different output methods

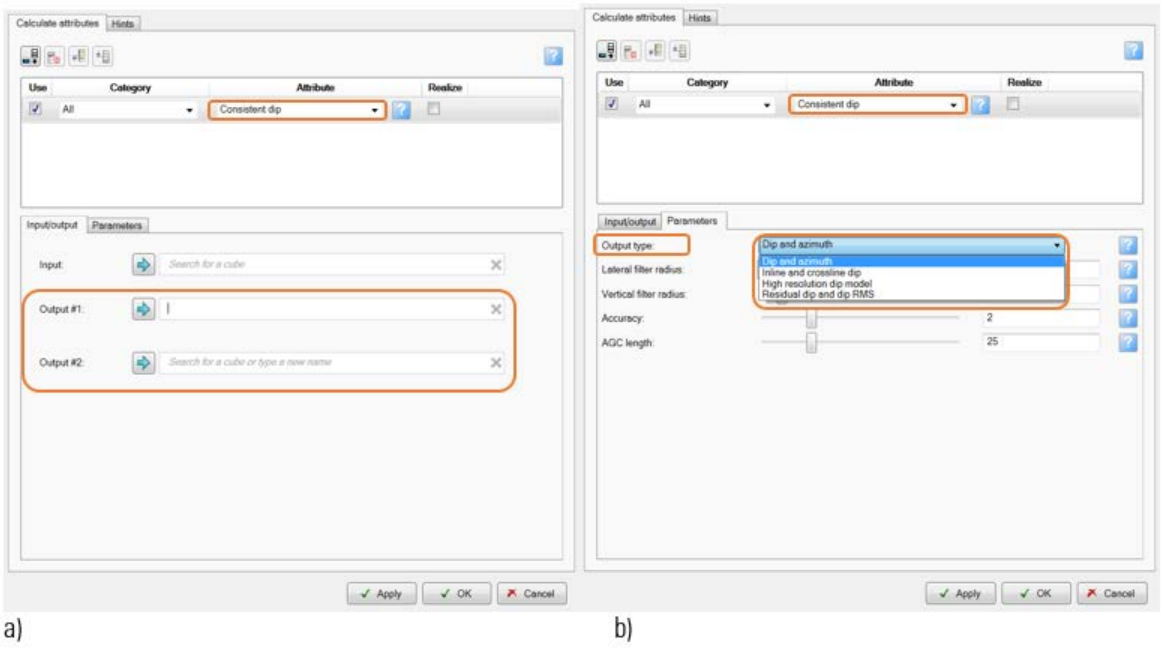


Figure 28: Example of high resolution inline Consistent Dip attribute, highlighting the salt domes and fault system

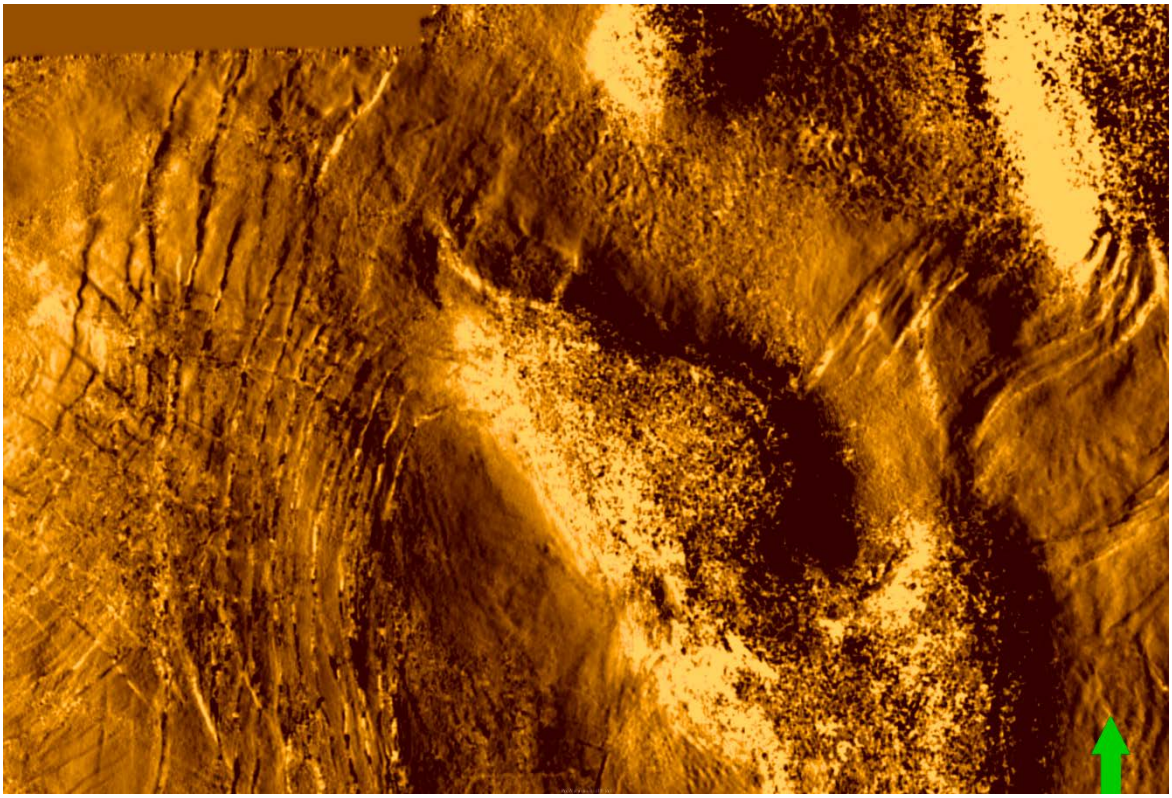
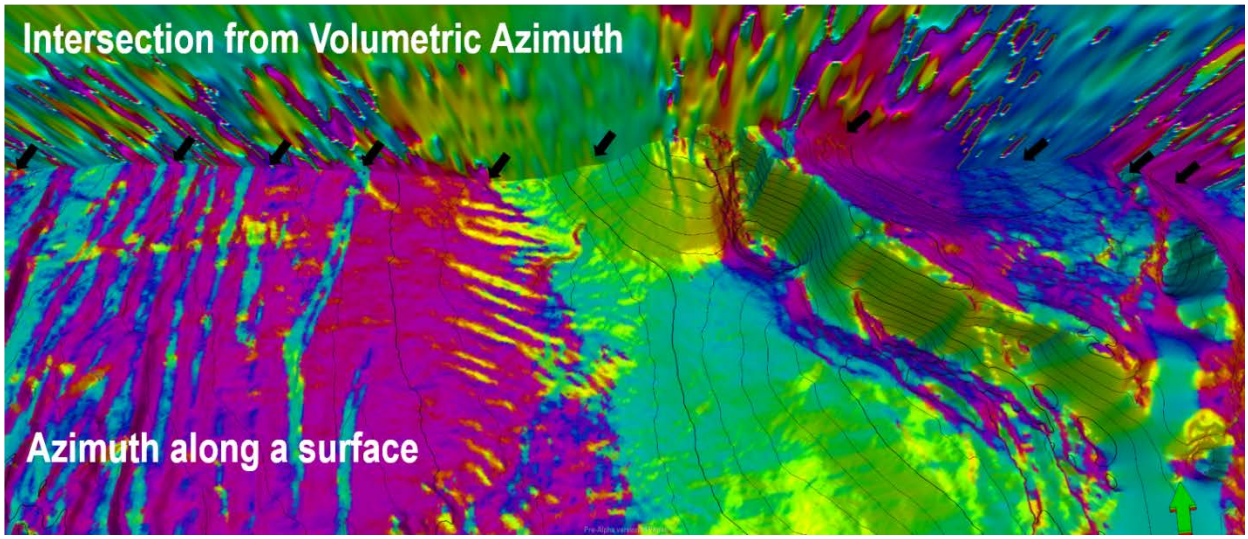


Figure 29: Example of Volumetric Azimuth and azimuth calculated along the surface, black arrows show the correlation between the volumetric azimuth and the azimuth calculated along the surface



Volume attributes

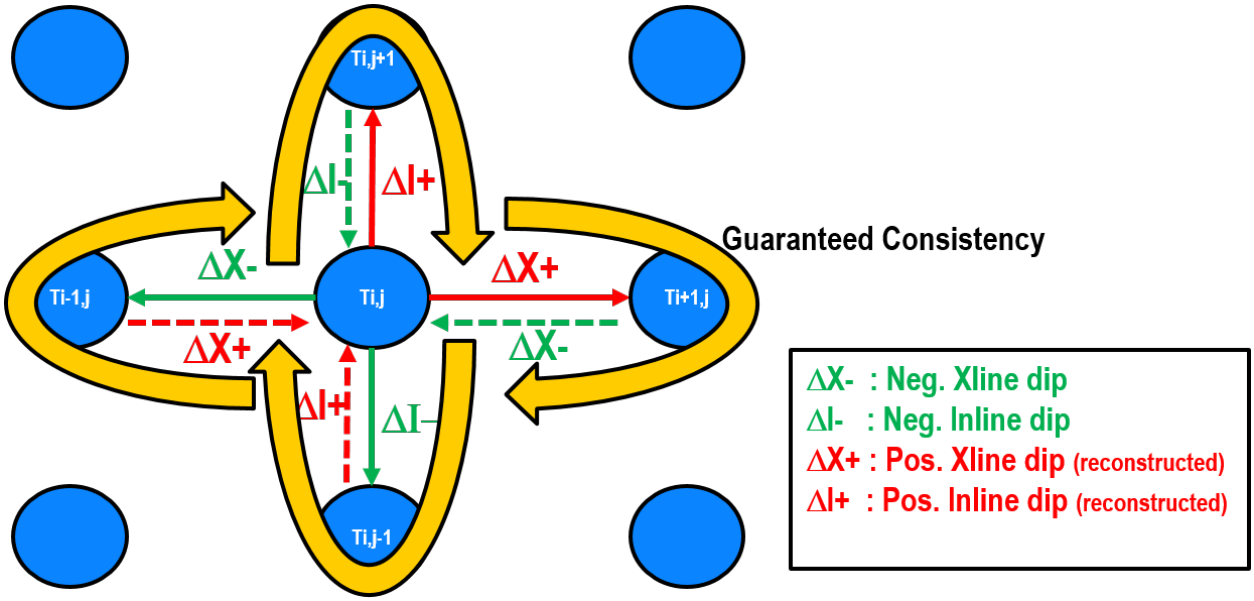
New "Consistent Curvature" attribute

Consistent curvature provides the most accurate curvature estimation for precise interpretation of geologic structures or subtle stratigraphic features from seismic data.

Consistent curvature properties:

- Consistent curvature takes the inline and crossline dip or the high resolution consistent dip as an input for various curvature attribute generation. This is unlike the traditional volumetric curvature calculations where the dip estimation is done on the fly from the seismic amplitude and then, the curvature is computed from the dip. Consistent curvature is inherently benefited by the input consistent dip, as consistent dip is the most accurate dip estimated by iterative global optimization and honors the global consistency constraints of reciprocity, causality, consistency, and continuity.
- The Consistent curvature algorithm requires four consistent dip cubes, namely negative and positive dip in the inline and crossline direction for all the samples. Positive inline and crossline dips are reconstructed by the algorithm from the user input, negative inline and cross line dip (for example, inline and crossline high resolution dip models) while honoring the consistency.
- Calculations of Consistent curvature are done in XYZ space, honoring the true bin spacing of the seismic; hence, avoiding exaggeration in the curvature estimation of seismic lineaments in one direction (inline or crossline) as in the IJK space curvature calculation.

Figure 30: A schematic showing the inline and cross line negative and positive (reconstructed on demand by algorithm) for a sample (Tij) required for the calculation of consistent curvature



The following are different Consistent Curvature outputs types:

- K_1 and K_2 Curvature
- K_{max} and K_{min} Curvature
- Inline and crossline K_{max} curvature
- Shape index and curvedness
- Most positive and most negative
- Azimuth of maximum and minimum curvature

Figure 31: Consistent Curvature attribute dialog showing different Output type

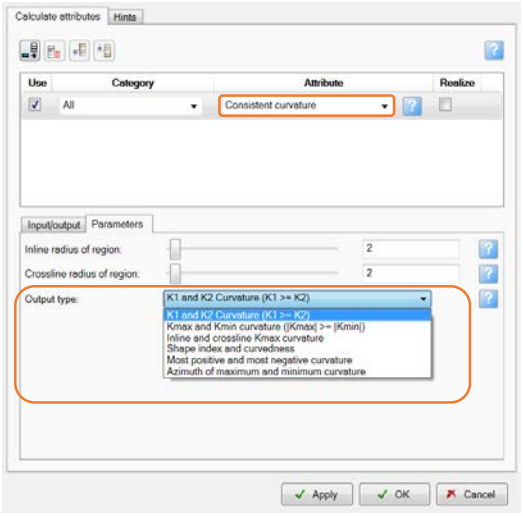
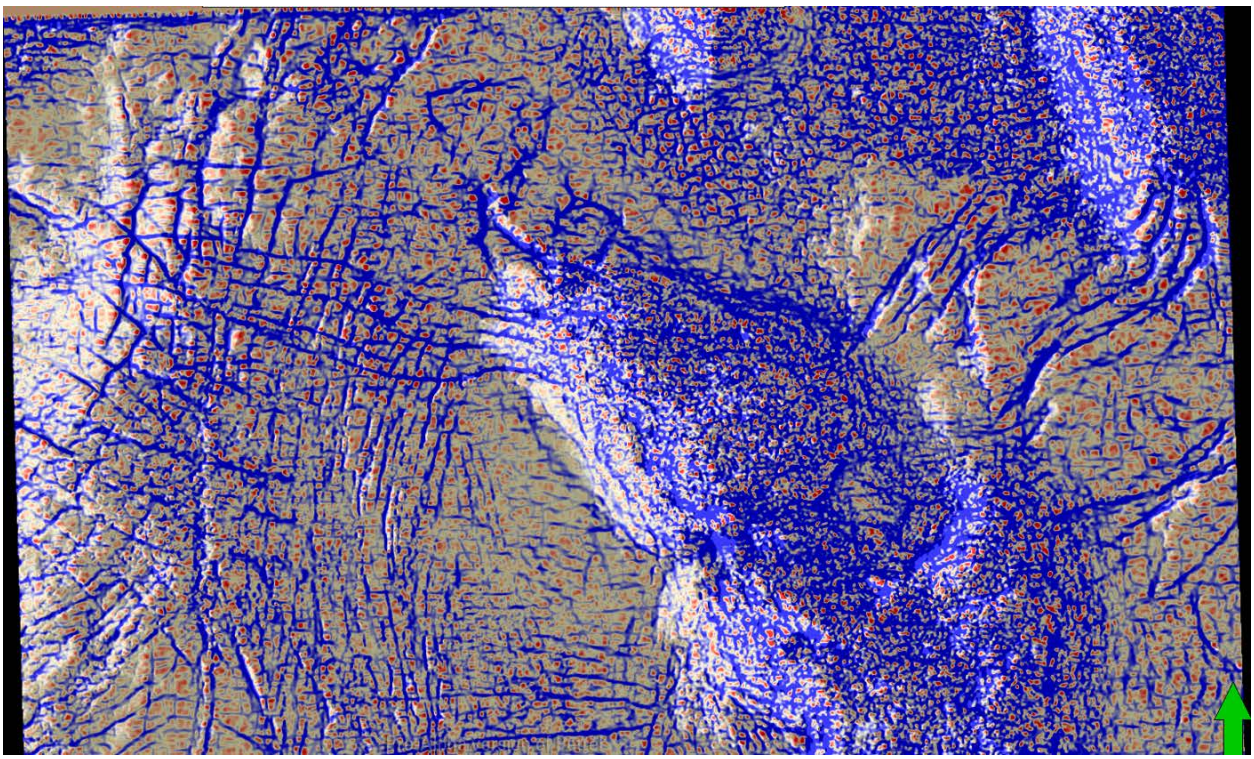


Figure 32: Example of Consistent Most Negative Curvature, highlighting the salt domes and fault system



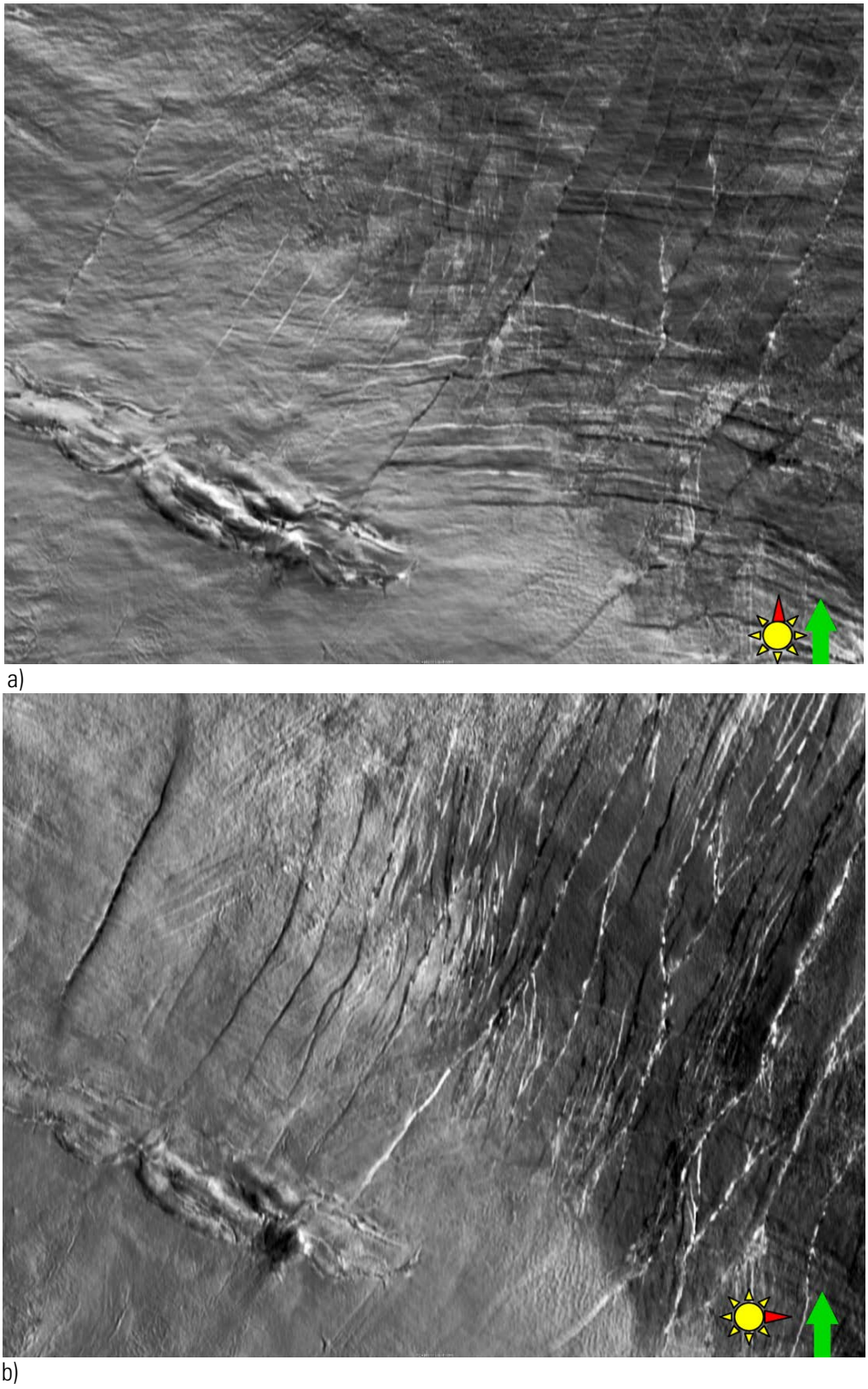
New "Directional Blending" attribute

The directional blending attribute generates a directional dip or curvature attribute highlighting the geologic features in a specific azimuthal direction.

A directional dip or curvature attribute generated in a desired azimuthal directions can provide an enhanced image of deformation or sweet spots, which could help in correlation with borehole, microseismic, and other production measurements.

The directional blending volume attribute requires two inputs (inline and crossline components of a chosen vector attribute), and provides one single (blended) output volume at a user-defined azimuthal direction. You can choose the azimuth, relative to survey North direction or relative to inline direction and change the azimuth interactively.

Figure 33: Example of Dip Directional Blending a) Highlighting the discontinuities in W-E direction b) Highlighting the faults system within N-S orientation



a)

b)

Notes on Consistent Dip, Consistent Curvature, or Directional blending volume attributes:

Case 1: If virtual Consistent Dip, Consistent Curvature, or Directional blending volume attributes are created in Petrel 2014.3 and accessed in earlier Petrel 2014 versions, the following applies:

- Virtual Consistent Dip, Consistent Curvature (with exception of Azimuth of Maximum and minimum curvature), or Directional Blending volume attributes can be visualized, interpreted on, and realized. However, attribute parameter settings cannot be accessed through the Settings, the Inspector, and the Interactive Parameter Control.
- Virtual Consistent Dip, Consistent Curvature, or Directional Blending volume attributes visualized or realized in earlier Petrel 2014 versions may differ from the same attributes visualized or realized in Petrel 2014.3.

Case 2: If a virtual Consistent Dip volume attribute is created in Petrel 2014.2 or earlier versions and opened in Petrel 2014.3 or later, the following applies:

- Virtual Consistent Dip volume attributes can be visualized, interpreted on, and realized. However, attribute parameter settings cannot be accessed through the Settings, the Inspector, and the Interactive Parameter Control.
- A virtual Consistent Dip volume attribute visualized or realized in earlier Petrel 2014 versions may differ from the same attribute visualized or realized in Petrel 2014.3.

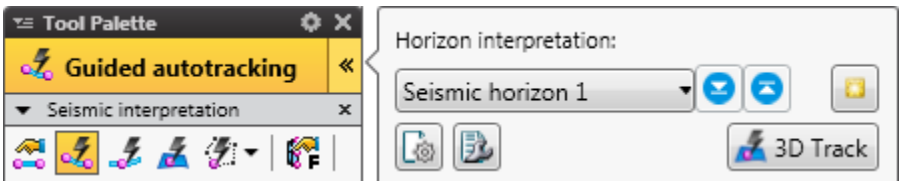
It is recommended to create a Consistent Dip volume attribute in Petrel 2014.3, as it is the most accurate.

2014.2

Settings callout for key interpretation tools

As part of the 2014 User Experience, productivity is further increased for key workflows in seismic interpretation. Via the Seismic interpretation tool palette callouts, interpretation objects can be easily activated and horizon, fault and multi-Z interpretation tools have easy access to their Settings.

In horizon interpretation, click any of the four horizon tracking tools to activate the selected tracking mode, and also open a callout to immediately identify the active horizon interpretation. Two settings buttons below it give you direct access to the Autotracking settings tab of the active horizon, and to the Autotracking process defaults for parameter tuning. With pre-tuned parameters, you can fluidly initialize new horizons, provide seed interpretations and run 3D track, all within close reach of the cursor.

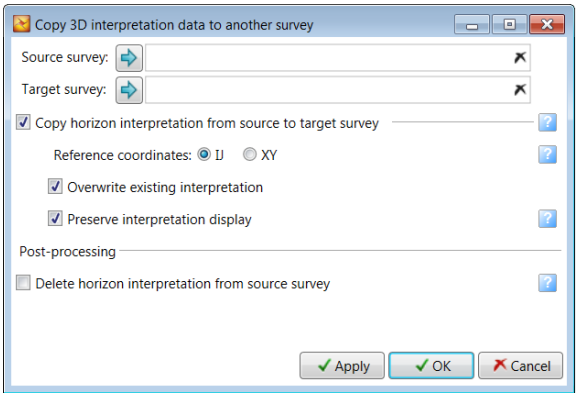


Similar tool settings callouts are also enabled for Fault interpretation and multi-Z interpretation.

Copy 3D horizon interpretation across surveys

Through a single dialog you can now easily copy all 3D horizon interpretations associated with one survey to another 3D survey within the same project. The target survey can be of different extent, orientation, annotation, interval spacing and coordinate reference system. If you have a new survey with a larger areal extent, you can use the copy function to 'extend' the containing survey of your existing interpretation data.

Using the copy function, you can also transfer interpretation data to a new survey defined with different XY coordinates. The copy function includes associated horizon attribute data and information about the 3D interpretation modes as used in the '3D interp inclusion' filters.



Geobody probes

Since the removal of the 4GB memory limitation in the Seismic Texture Cache setting in Petrel 2014, further optimization has been done on how Petrel utilizes GPU memory beyond 4GB when handling large seismic data sets in geobody interpretation workflows. Petrel now fully supports GPU memory allocation above 4GB, utilizing all the available hardware and the latest graphics cards from NVIDIA®. With the latest Kepler cards, seismic volumes of tens of GBs are handled with high performance and high quality display.

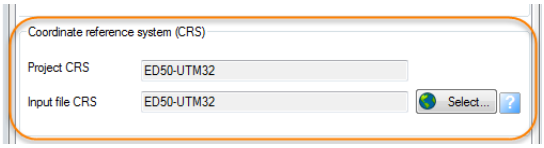
2014.1

SEG-Y 2D toolbox (behavioral changes)

Original CRS preserved

When loading 2D SEG-Y files into a project defined with a different CRS, and merging with navigation information external to the project (that is using the options "SEG-Y trace coordinates" or "Separate navigation file"), the edited/output SEG-Y file now preserves the original file CRS. Previously, such edited SEG-Y files were output in the CRS of the project.

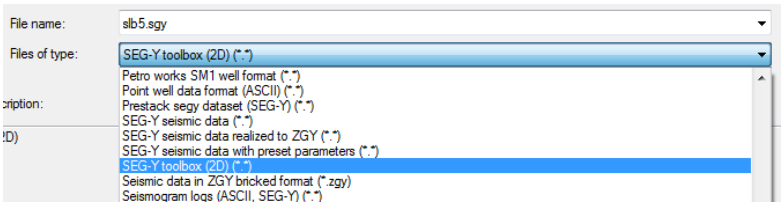
Figure 34: Updated CRS selection dialog



Access

To access the SEG-Y 2D toolbox in the new User Interface, right-click anywhere in the background of the Input tab and select Import file. Under Files of type, select SEG-Y toolbox (2D)(*.*). Alternatively, on the **Home** tab, in the **Manage data** group, click Import file.

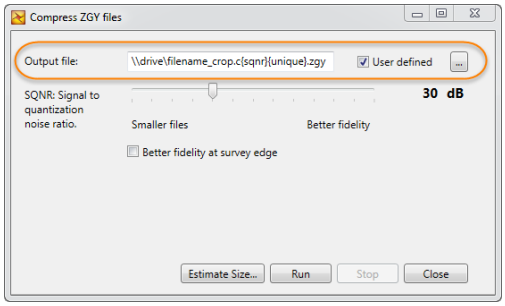
Figure 35: Access to SEG-Y 2D toolbox is now via the common Import file dialog



Seismic compression

There is now the ability to specify an output path for the compressed ZGY file.

Figure 36: Option to specify an output path for the compressed ZGY file

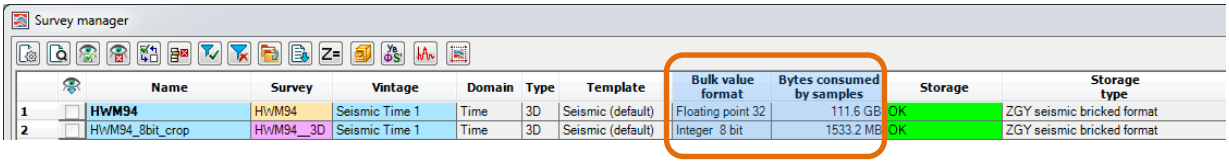


Survey Manager (behavioral change)

The following column headers have been renamed and now reflect the consistent values reported in the **Statistics** tab of the seismic data **Settings** dialog.

- The previous column “File size” is now correctly labeled “Bytes consumed by samples”. Note that file sizes on disk may be a smaller value due to compression or presence of null traces, or larger due to multi-resolution data handling. The values have been reformatted to be more readable and readily comparable in MB and GB units. This improvement also allows for better sorting results.
- The previous column “Format” is now correctly labeled “Bulk value format”.

Figure 37: Survey manager renamed column headers "Bytes consumed by samples" and "Bulk value format"



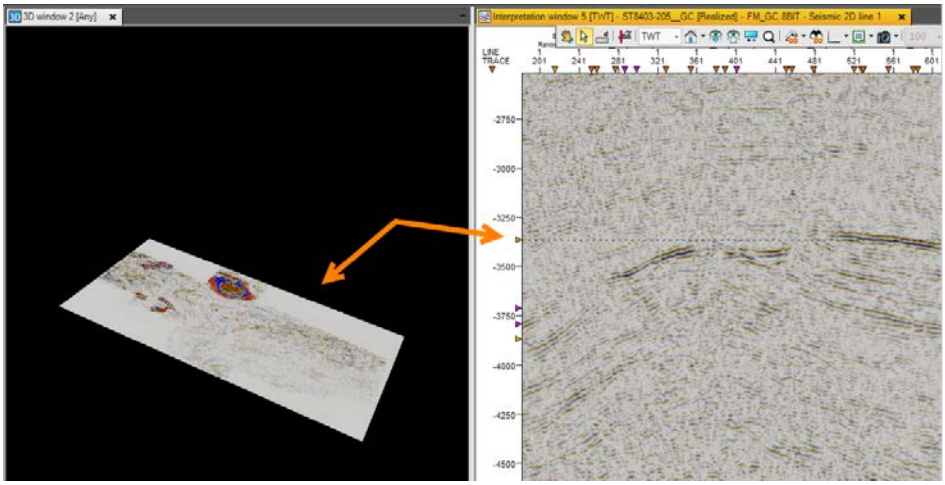
Interpretation window

Horizontal slice indicator

In the Interpretation window, you can find horizontal slice indicators along its vertical axis. These represent available time slices and their positions are synchronized across display windows, allowing for fast and easier slice manipulation.

Manipulating a time slice in a 3D window for example, updates the vertical position of the corresponding slice indicator providing relative orientation. Conversely, manipulating a slice indicator within vertical zones of interest identified in the Interpretation window will update its time-slice positioning in the 3D window.

Figure 38: New horizontal slice indicators in the Interpretation window are linked with other display windows to easily view and manipulate time slices



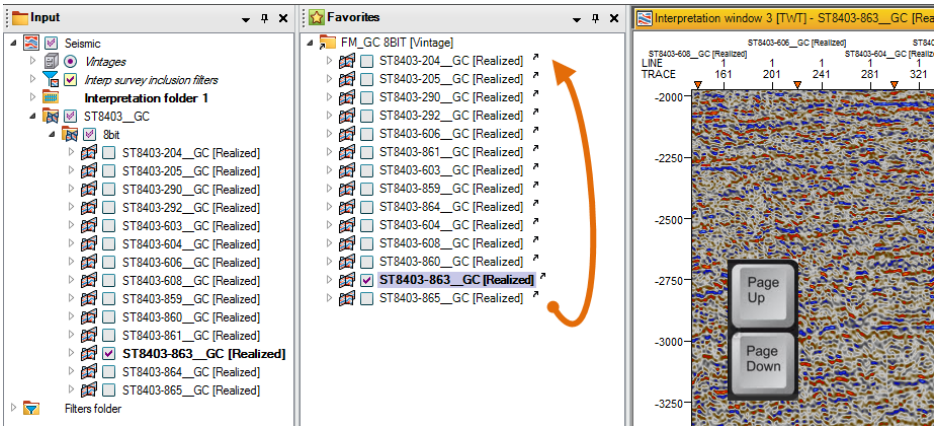
2D seismic line selection (PgUp/PgDn)

In an active Interpretation Window, easily review 2D lines in the sequence listed in the Input tree or **Favorites** pane, by using the new keyboard shortcuts **Page Up** and **Page Down**.

For a selected vintage, use **Page Up**/**Page Down** to step through the individual 2D lines. Also, to step through virtual, calculated and other child objects of each 2D line, use **Shift+Page Up** and **Shift+Page Down.**

Compared to the **Up** and **Down** arrow keys, using the **Page Up/Page Down** keys provide the advantage of circular selection within a seismic folder.

Figure 39: Use of keyboard shortcuts Page Up/Page Down in an Interpretation window

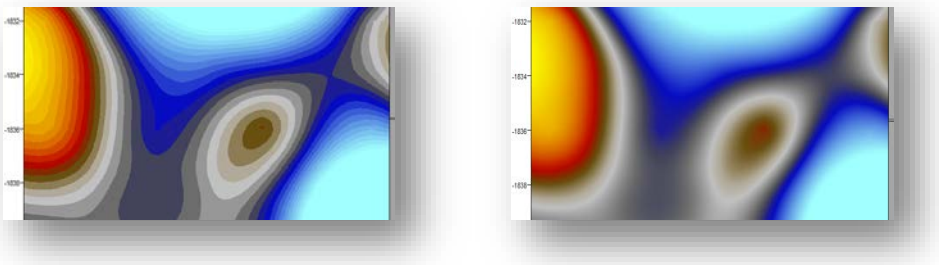


Color Interpolation

The Interpretation window uses RGB interpolation to get full true color (24bit) displays of seismic data.

Improvements have been made to the color interpolation of seismic data in the Interpretation window. The same technique as already used for seismic data displayed in the 3D window is now also applied for seismic data displayed in the Interpretation window. It uses RGB interpolation to get full true color (24bit) displays of seismic data. This enhancement is specifically noticeable for seismic color tables that are significantly compressed.

Figure 40: Color interpolation comparison between Petrel 2013 and Petrel 2014



Petrel 2013 Petrel 2014

Seismic Inspector

Frequency spectrum

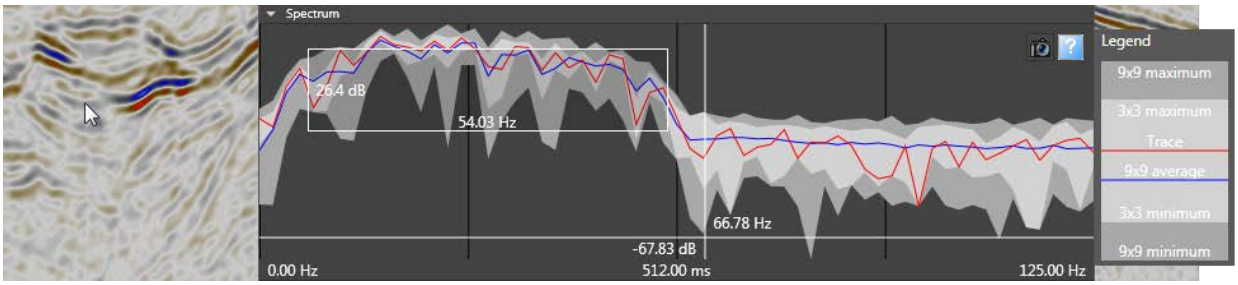
Understanding the frequency spectrum of seismic data is one of the fundamental tasks of any geoscientist working with seismic data. The Inspector enables quick and interactive interrogation of the frequency spectrum as you select an area of interest.

Moving the pointer inside the graph display, you can read-out the dB level for each frequency. A fixed window covering lengths of 64,128, and 256 samples and a maximum of 9x9 neighboring traces centered on the picked sample is analyzed. Graphs are shown for the selected trace and several averages of neighboring traces.

The frequency spectrum display can be used to interrogate 3D and 2D seismic sections.

The graph can be exported as a bitmap or a table of numbers for external reporting.

Figure 41: Frequency spectrum display in the Seismic Inspector

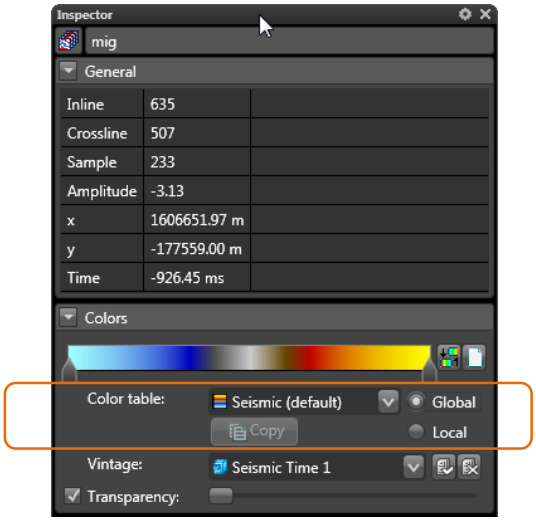


Colors

It is now possible to set Global (shared) or Local (private) color table for seismic data from the Inspector.

In the Inspector, you can set the Global option, which allows sharing the same color table by many seismic objects at a time; or the Local option, which applies a private color table for a selected object. This local color table will only be used by this object. A local copy of the global color table can be created and applied for a selected object.

Figure 42: Global and local color tables settings in the Seismic Inspector



In the 3D window, the **Colors** tab is now accessible for Composite lines.

Performance improvements

Various targeted improvements have been made to Petrel Geophysics:

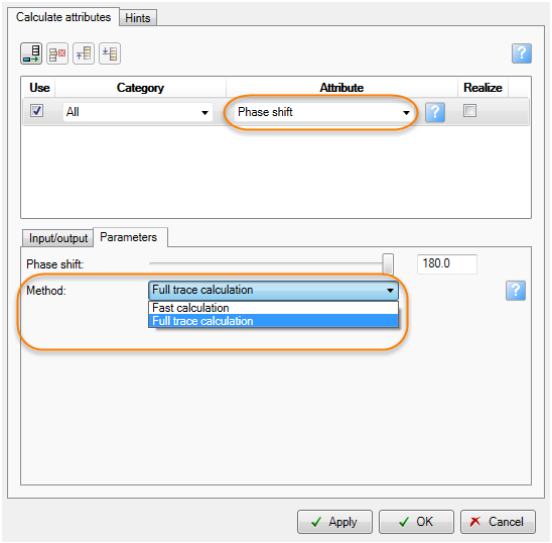
- Random line visualization: There is performance improvement in panning of random line sections compared to 2013. Improvements have been made for faster data access and faster interpolation rendering when working with different levels of detail. The benefits depend on the hardware specification of the machine used, mainly the memory, graphic cards and disk access speed.
- Horizon interpretation transfer: a 60% improvement in transfer speed between Petrel and Studio.

Volume attributes (behavioral changes)

Phase shift

In 2013.5, a new volume attribute Phase shift (full trace) was introduced to provide enhanced support for low frequencies, typical in broadband data. In 2014.1, the Phase shift volume attribute now defaults to Phase shift (full trace) calculation. Both algorithms (full trace and fast calculation) can be accessed from the single Phase shift volume attribute).

Figure 43: Phase shift attribute now provides two methods "Full trace calculation" using the full trace' and "Fast calculation", which is the same as the 'Phase shift' in earlier releases



For more information on Phase shift (full trace calculation), see the 2013.6 What's New document and the online Help.

Notes on virtual volumes for phase shift:

- Virtual "Phase shift (full trace)" attribute volume using both "Full trace calculation" and "Fast calculation" methods created in Petrel 2013.5 and later releases will continue to work in the same manner, except when going to the **Settings > Virtual attribute** tab, where the Attribute is now shown as "Phase shift."
- Virtual "Phase shift" attribute volumes created in Petrel 2013.5 and earlier releases will not be fully accessible. These deprecated virtual cubes can be viewed and used as normal, but, when going to the **Settings > Make virtual attribute** tab, their existing parameters are unavailable.
- To access the new Phase shift parameters, create a new virtual attribute by choosing the type "Phase shift". Similarly, to run an existing Workflow containing virtual attributes, first update the workflow by creating a new "Phase shift" virtual attribute.

Relative Acoustic Impedance

A new parameter is enabled to allow control of the low-cut frequency applied for the Butterworth filter.

Previously, this value was hard-coded at 10Hz. The current implementation allows for preservation of ultra-low frequencies, typically observed within broadband seismic data.

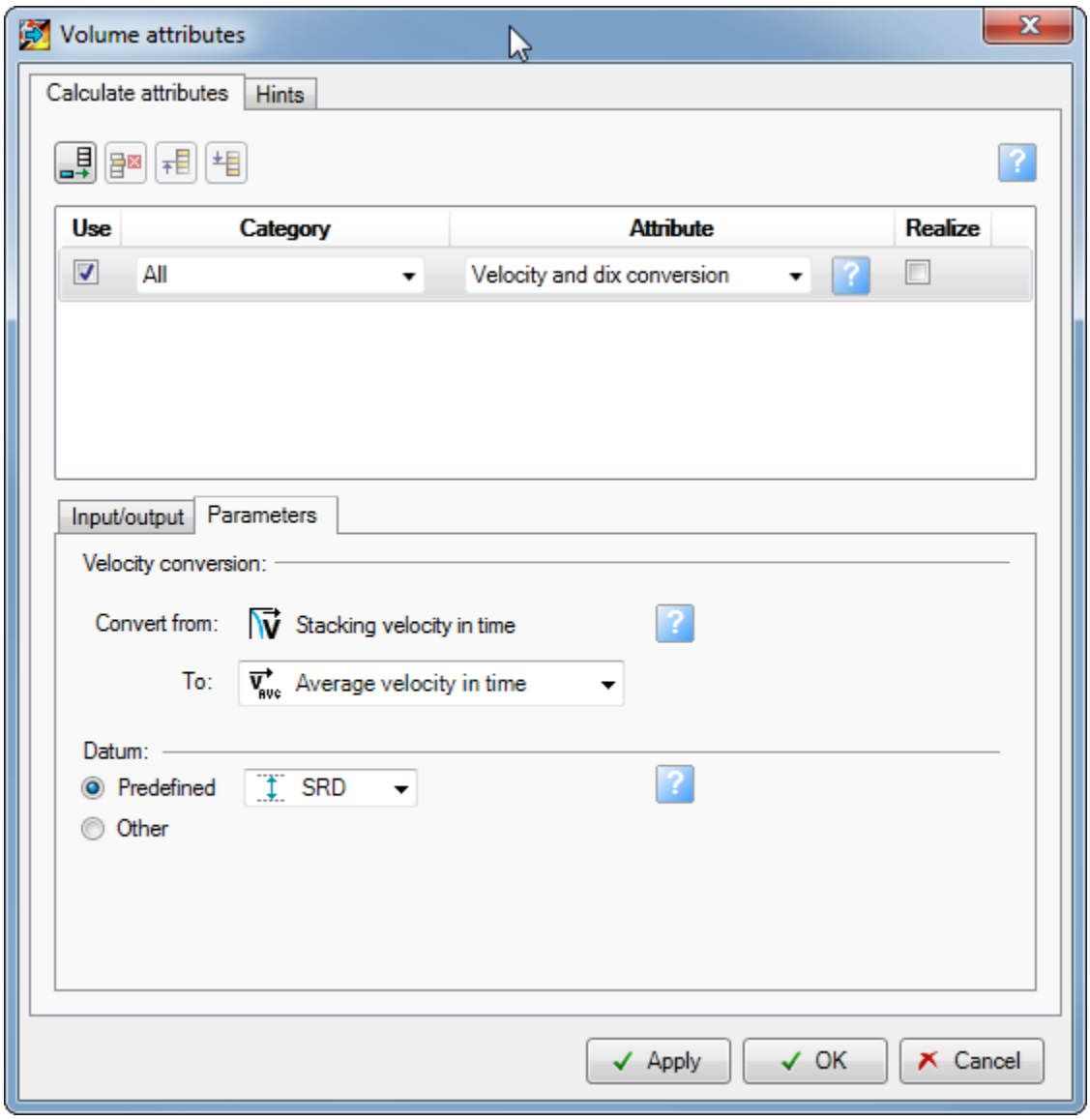
Notes on virtual volume:

- Virtual "Relative acoustic impedance" attribute volumes created in Petrel 2013 and earlier releases will not be fully accessible. These deprecated virtual cubes can be viewed and used as normal, but, when going to the **Settings > Make virtual attribute** tab, their existing parameters are unavailable.
- To access the new low-cut frequency parameter, create a new virtual attribute by choosing the type "Relative acoustic impedance". Similarly, to run an existing Workflow containing virtual attributes, first update the workflow by creating a new "Relative acoustic impedance" virtual attribute.

Velocity and dix conversion

Starting from Petrel 2014.1, "Velocity cube" attribute and "Dix conversion" attribute are combined in one "Velocity and dix conversion" attribute, which preserves functionality existing in earlier Petrel versions.

Figure 44: Combined Velocity and Dix conversion attribute in Volume Attributes



Notes on virtual volume:

- Virtual "velocity cube" and "Dix conversion" attribute volumes created in Petrel 2013 and earlier releases will not be fully accessible. These deprecated virtual cubes can be viewed and used as normal, but when going to the **Settings > Make virtual attribute** tab, their existing parameters are unavailable.
- To access the new velocity parameters, create a new virtual attribute choosing the type "Velocity and Dix conversion". Similarly, to run an existing Workflow containing virtual attributes, first update the workflow by creating a new "Velocity and Dix conversion" virtual attribute.

Horizon interpretation

Autotracker: live visual update while tracking

3D autotracked points are now visualized live in the 2D or 3D window as the tracker progresses.

The live update allows you to do visual QC as the autotracking progresses, and if deemed necessary, the tracker can be stopped as soon as the tracker tracks incorrectly. Confidence in tracking results can be achieved from an early stage, thereby reducing the amount of corrective work later.

Autotracker: arbitrary polyline intersection through parent points

The 'Select parent points' tool has been enhanced to help identify and validate wrongly autotracked points easily, by enabling the creation of an arbitrary polyline intersection through selected parent points.

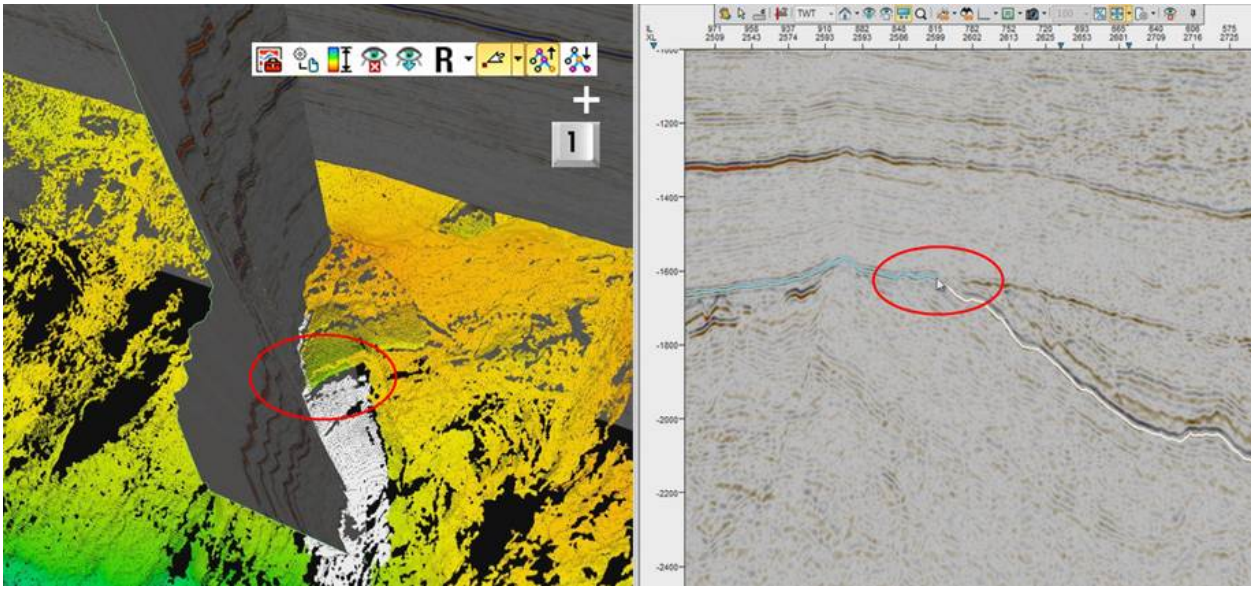
When working with challenging seismic datasets, it can be difficult to determine the tipping point when the autotracker is set off onto a wrong track. You can now automatically generate an arbitrary polyline intersection through selected parent points on a 3D horizon interpretation to help identify and validate these points easily.

As normally done to select all parents points associated with a point, use "Select parent points' together with the 'Single point select' selection option, and pick a point of interest on the off-track section of an auto-tracked horizon.

Next, press the number **1** key, to create an arbitrary polyline intersection through these points.

Within the 3D window (or an additional Interpretation window), it is easy to see the path tracked by the horizon, visualized on the newly created polyline intersection. In this profile, you can quickly identify and validate potential mis-picks.

Figure 45: Autotracker - generate a polyline intersection through selected parent points to quickly identify mispicks and validate tracking

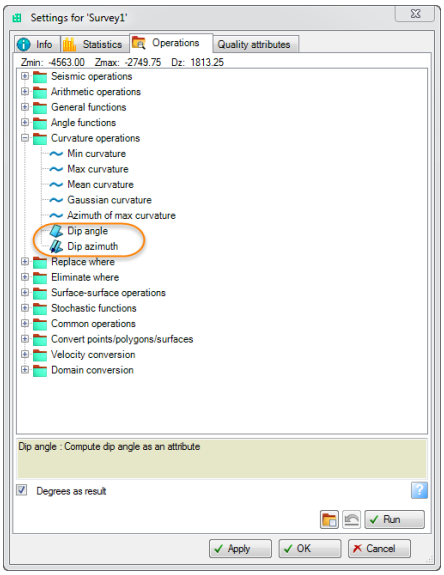


Dip angle and azimuth

Dip angle and dip azimuth attributes can now be generated directly from horizon interpretations.

In the Settings dialog of the 3D interpretation grid, go to the Operations tab, Curvature operations folder. Dip angle and Dip azimuth computation are available, with the option to output in degree or radian units.

Figure 46: Dip angle and azimuth computation is accessed from the Settings dialog of the 3D interpretation grid



Multi-Z

Interpretation upgrade

Once a 2013 project is open in Petrel 2014, multi-Z interpretation objects do not have normals assigned. In order to assign normals, go to Settings of the multi-Z object > Operations > Seismic Operations > Assign normals to multi-Z interpretation operation. After this, the multi-Z object(s) will be displayed with the respective normals.

RPT/Studio

To be visible within RPT, multi-Z interpretation objects must have normals assigned. Assign normals first, then open Reference project tool to transfer the data. The same behavior persists within the Studio environment.

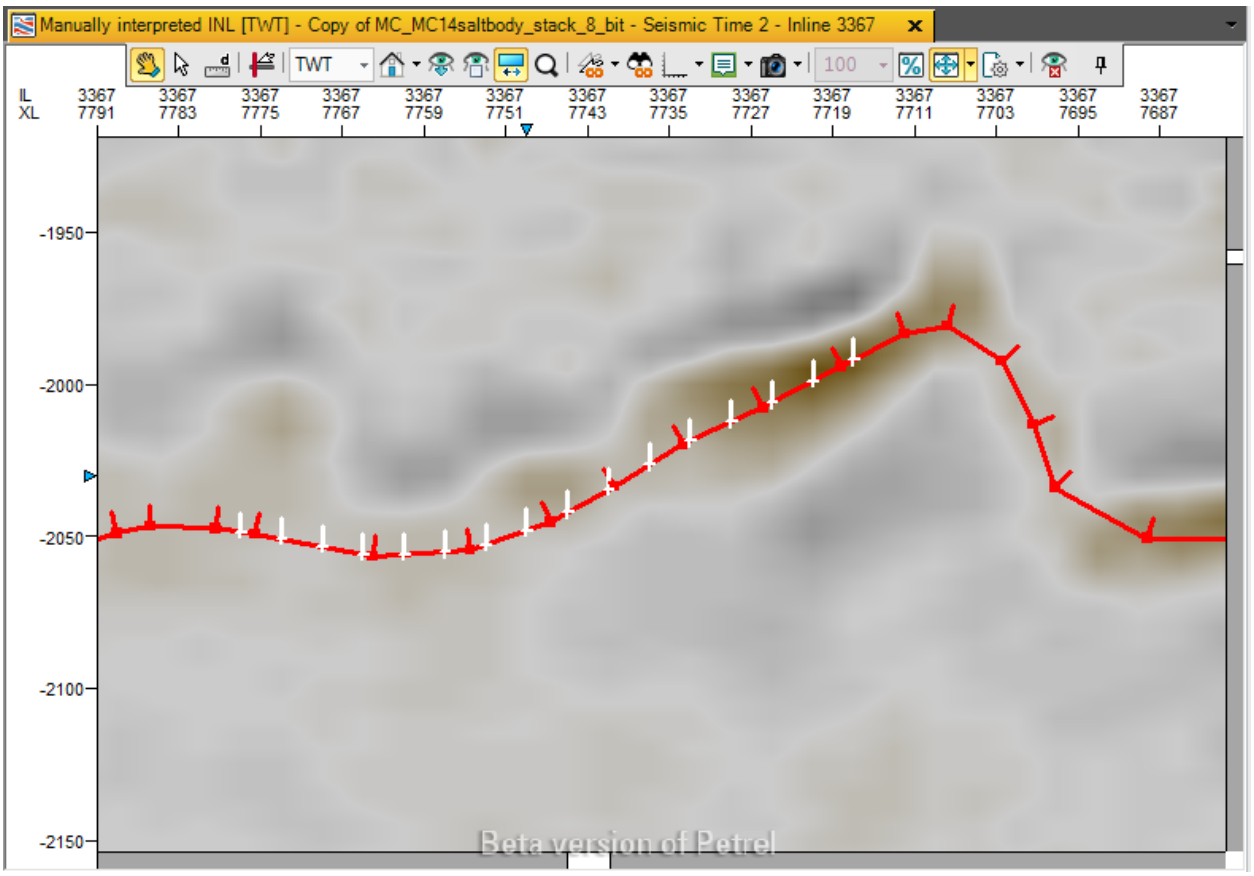
New styles for interpretation normals

You can change Style settings for normals in 3D/2D/Interpretation/Intersection windows.

In the Style tab for the multi-Z interpretation object, you can toggle on/off the normals' display in 3D/2D/Interpretation/Intersection windows, as well as change the normals parameters (color, thickness and length).

In Interpretation/Intersection windows, you can set different styles for points and corresponding normals interpreted on the same section vs. points and corresponding normals interpreted on crossing seismic sections. These points and normals are called crossing points and crossing normals (defined by the white color in the following example).

Figure 47: Example showing crossing points and crossing normals



Create new interpretation in the Workflow editor

In Petrel 2014, it is now possible to create a new empty multi-Z interpretation by applying the "Create multi-Z" operation. This new multi-Z interpretation can be used in other workflow processes/operations.

The new command is located in the Seismic operations folder of Operations. It allows the ability to specify the name of a new object, location (Interpretation folder), and domain (time or depth).

Regenerate triangle mesh

The Regenerate option can be used to update the existing triangle mesh while interpreting.

After an output triangle mesh is already created but new segments are interpreted for the same multi-Z interpretation, this existing output triangle mesh can be updated with new information. To apply the change, from the multi-Z interpretation Mini-toolbar for the multi-Z interpretation select Regenerate or use the **Shift-V** shortcut to update the existing triangle mesh.

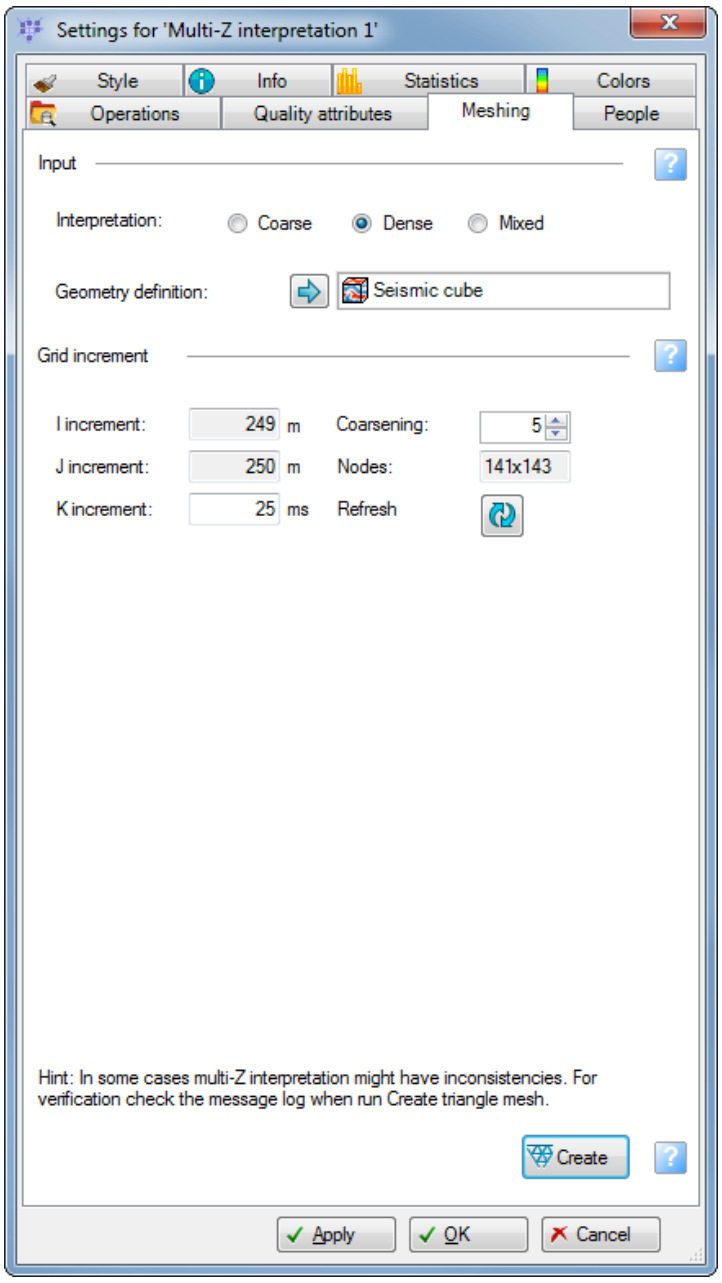
Figure 48: Multi-Z interpretation mini-toolbar - Regenerate triangle mesh

Refresh triangle mesh

The Refresh option  allows rapid update of an existing triangle mesh after the resolution has been changed.

Before getting the final triangle mesh with high resolution, you can run Create triangle mesh with bigger coarsening factor and k increment. By setting low resolution for the output, you will reduce the time spent to create an output triangle mesh. After the QC of the output is done and you have gotten the expected result, set new (higher) resolution for the output, and by using the Refresh option, update the existing triangle mesh. It will take less time to run the process as it refreshes the resolution of the existing output only.

Figure 49: Settings for Multi-Z interpretation - Refresh triangle mesh option



Geobody interpretation

Geobody extraction performance

Automated geobody extraction performance has been improved by parallelization.

Improvements have been made to the automatic geobody extraction algorithm. The improvements consist of parallelizing the segmentation algorithm. The extraction is now also handling large volumes better than it was previously.

Geobody extraction from seismic intersections

Geobodies can now be extracted directly from seismic intersections at full resolution of the seismic data without the use of a probe.

For quick assessment of amplitude anomalies observed in the seismic data, it is now possible to extract geobodies directly from seismic intersections without the use of a probe. Using the 'Seeded geobody extraction' mode, a geobody will be extracted at full resolution of the seismic data and based on 100% opacity.

Seismic Texture Cache

Due to limitations in the graphic frameworks used by Petrel, previously, there was a memory limitation for the Seismic Texture Cache setting of 4GB. This limitation has now been removed for Petrel 2014. Note that to take advantage of this change, an updated driver from NVIDIA with a release number no older than 332.76 is needed.

- Geobody interpretation - Geobody workflows are the most frequent feature using Seismic Texture Cache. This setting will decide how much data from the seismic used in your Geobody probe you will load into your graphic card memory. Setting this number to a high number (recommended no more than 75% of your available Graphic card memory) will then render as much as possible in LOD0 (Highest resolution) and give you the best display quality of your probes. However, allocating this much memory to the data itself, will affect your interactivity performance. Setting this to a low number might increase your interactivity working with Geobody probes.
- Seismic data - Displaying regular seismic is also affected by the Seismic Texture Cache, but only when displaying seismic using "Volume rendering" techniques. For more information, see the section Volume Visualization in 3D in the Help Center. When using this feature, be aware of the same memory tradeoffs mentioned in the information above on Geobody interpretation.

Retirement of SEG-Y Utility (behavioral change)

The SEG-Y Utility module is retired. Functionalities to merge 2D navigation information into SEG-Y trace headers have been superseded by enhancements available in the SEG-Y 2D toolbox. Also, see Geophysics: General, "SEG-Y 2D toolbox (behavioral changes) - Access" on page 55 for more information.

Retirement of Immersion mode (behavioral change)

With the introduction of the new User Interface in 2014.1, the Immersion mode is retired. Future improvements on an immersive working environment with full navigation interactivity and real-time display will be channeled and focused on the new User Interface.

Geophysics: Seismic Well Tie

2014.3

Extended Statistical Wavelet extraction Method

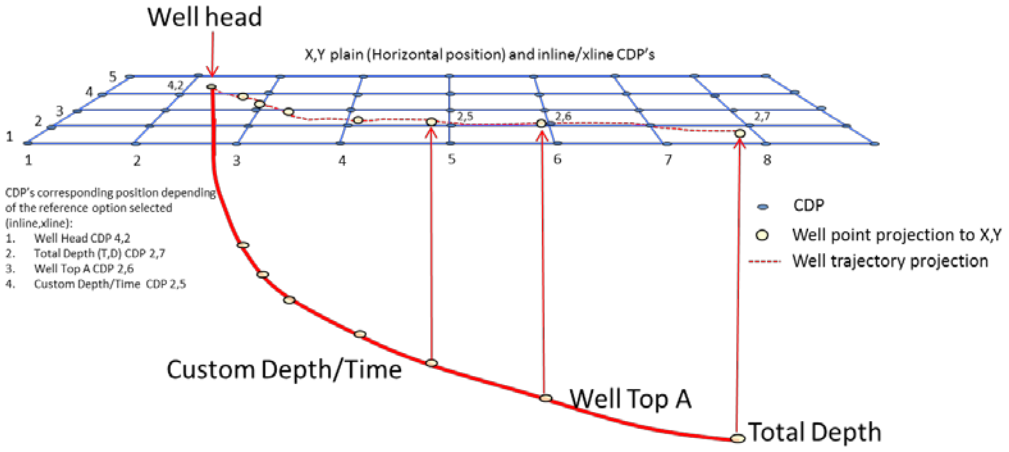
Following the same theory from the statistical method, but with no limitation on the selection range and being independent from the well, the extended extraction gives a selection range of inline, xline, and vertical (the limit will be the seismic selected).

The seismic inputs for the extended extraction could be 3D volumes (realized, segy compressed, and virtual) and 2D lines. All of them are in time domain.

In the region section, a X,Y reference trace selection tool helps to select a position in the horizontal plane (X,Y) in different ways:

- Well head: The reference will be the closest CDP to the X,Y corresponding to the well head position of the well.
- TD (Total depth): The reference will be the closest CDP to the X,Y corresponding to the total depth position of the well.
- Well Top: The reference will be the closest CDP to the X,Y corresponding to the well top position in the horizontal plane. This option is very useful at the time to perform analysis within the reservoir. In the case where a marker is repeated (Well tops held in the stratigraphy folder should be organized in the correct geological order and with the correct hierarchy and the repetition is not a recommended practice), the shallowest one will be selected.
- Custom depth/Time: The reference will be the closest CDP to the X,Y corresponding to the position of and specific depth along the well trajectory. The options in time will be possible just when the well has a time depth relationship assigned.

Those options above will give you a reference of the specific CDP position (inline and xline) format, especially for more localized wavelet extractions.



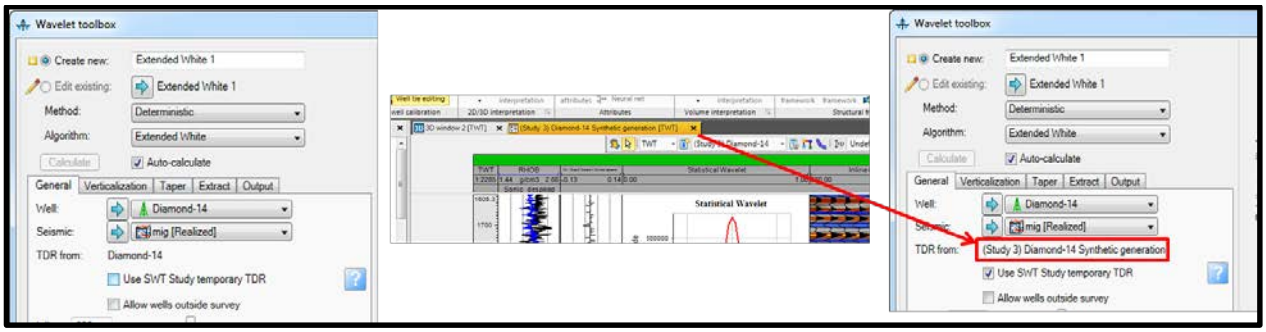
Reflectivity coefficient sampling interval

In Petrel 2014.3, the sample rate (SR) will be defined from the wavelet SR for new synthetic generation and integrated seismic well tie studies. The default SR for new studies will be 1 ms. Other SRs can be selected, depending from the interpreter criteria. Old studies will keep the original SR.

Well Section Window's temporary Time Depth Relationship for wavelet deterministic extraction

In Petrel 2014.3, a temporary Time Depth Relationship (TDR) from an active Well Section Window (WSW) can be used to extract deterministic wavelets. In the older versions, the TDR has to be assigned to the well in order to perform the deterministic extraction.

If a study WSW is active, the temporary TDR from that window will be considered for the deterministic wavelet extraction. Any change from the bulk shift or stretch and squeeze process in the study will be listen by the WTB and used for the extraction. The name of the TDR or temporary TDR used for the extraction will be reported at "TDR from" in the interface.



2014.1

Wavelet average: No normalization option

Before Petrel 2014.1, the product from the wavelet average feature was a normalized wavelet with amplitude values between -1 and 1. In Petrel 2014.1, the default option is still the normalized option, but if the normalize option is deactivated, the amplitude will be the resulting average from the input wavelet amplitudes.

Gardner's patching: auto-complete reflection coefficient inputs

In Petrel 2014.1, a new option to autocomplete missing sections of Density or Sonic logs using Gardner's equation is available. This option fills the gaps where one of the input logs (Sonic or density) is missing.

One of the logs has to exist to compute the other one:

$$\rho = aZ_n^{pb}$$

Where;

a= The Constant unit (imperial/metric) will be shown next to the constant text box. For the "Exponent" parameter the default value is 0.25. The user can change values for these two fields and these values will be used as parameters for Gardner's equation

b= Exponent

The default values will come as 0.23 for the "Constant" parameter, if the project is in Imperial system. If the project is in metric system it will convert the default imperial value to metric using the equation:
a (in metric) = (a in imperial) * (1000 / (0.3048 ^ exponent)).

Gardner's equation usually works fine for sedimentary rocks with velocities above around 1500 m/s to around 6000 m/s and densities from 2x103 Kg/m3 to around 2.8x103 Kg/m3 (for more information refer to Boore, D.M. 2007).

Correlation track enhancement

In Petrel 2013.x, the correlation track was limited by the input logs. The window for the correlation is defined from the top and base of the synthetics generated from those input logs. In some cases when the data from the checkshots is not available or synthetics (derivate from velocities or share from other boreholes), the correlation into the window may be not the ideal. In Petrel 2014.1, the correlation interval is defined by the default as top and base from the seismic volume selected for the study.

This allows us to detect other good correlation points that were not detected if the correlation interval is defined by the Reflection Coefficient interval (input logs).

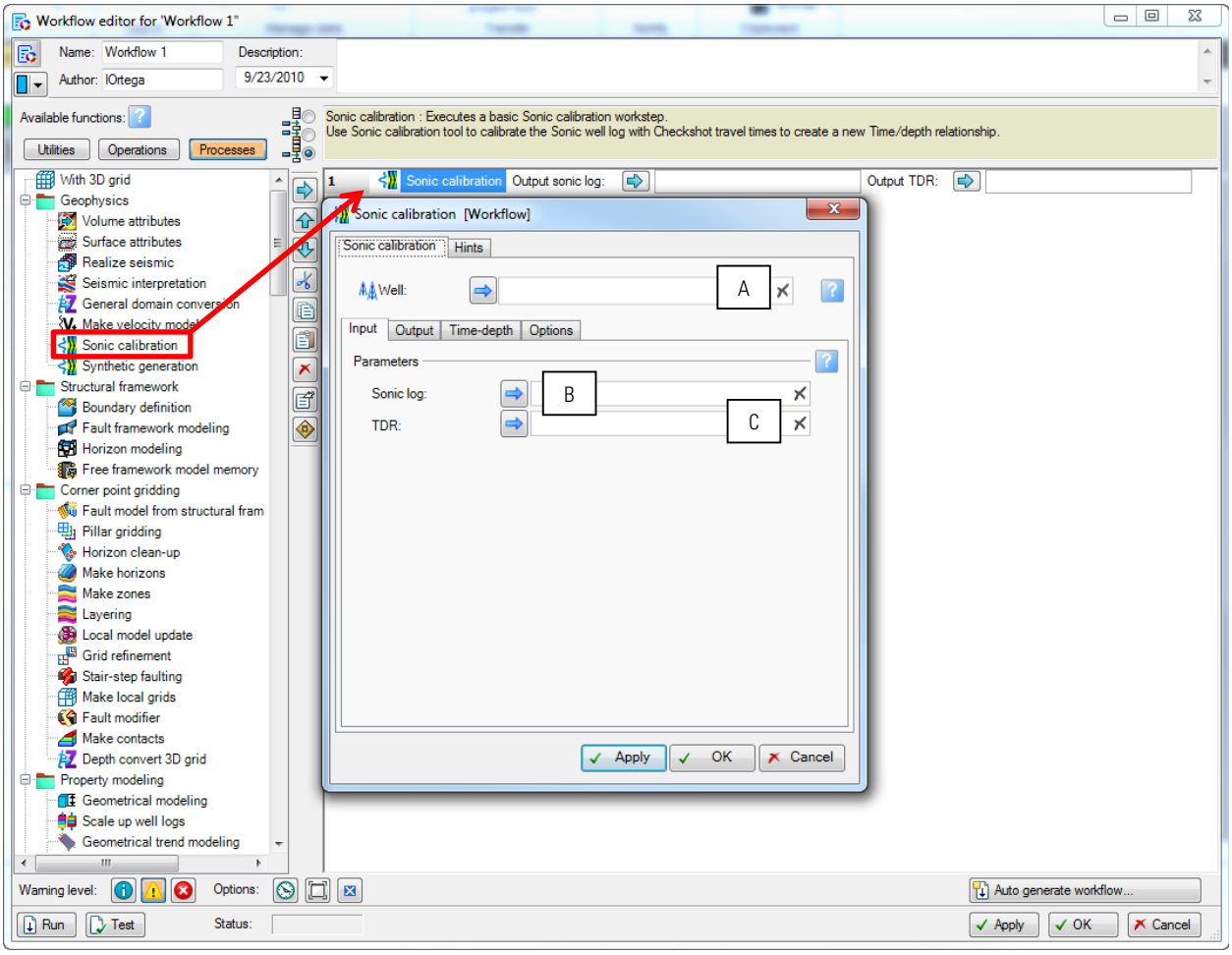
The same phenomenon happens when a window is defined from the correlation tab at the synthetic generation. For Petrel 2013.x, the correlation for the synthetics and the seismic inside the windows were defined. The new way that it works for Petrel 2014.1, is that the synthetic is clipped by the windows defined at the correlation tab of the synthetics generation study; however, the whole seismic interval will be always be considered for the correlation.

Retirement Synthetics (behavioral change)

The synthetic tool, under Stratigraphy, is retired from Petrel 2014.1. It is replaced by Multiwell Sonic Calibration (**Workflow editor**) and Multiwell Synthetics Generation (**Workflow editor**).

Multiwell Sonic Calibration (Workflow editor):

The process for Sonic Calibration is now present under the Geophysics process, the **Workflow editor**. This process can be applied to one well or a workflow can be configured to apply the process to different wells, using the loop function.



Inputs:

A: The well or a variable can be dropped as input from the **Input** pane.

B: The Sonic calibration from the Global well logs folder can be added as input (the input log template has to be Sonic and can be from the global well logs).

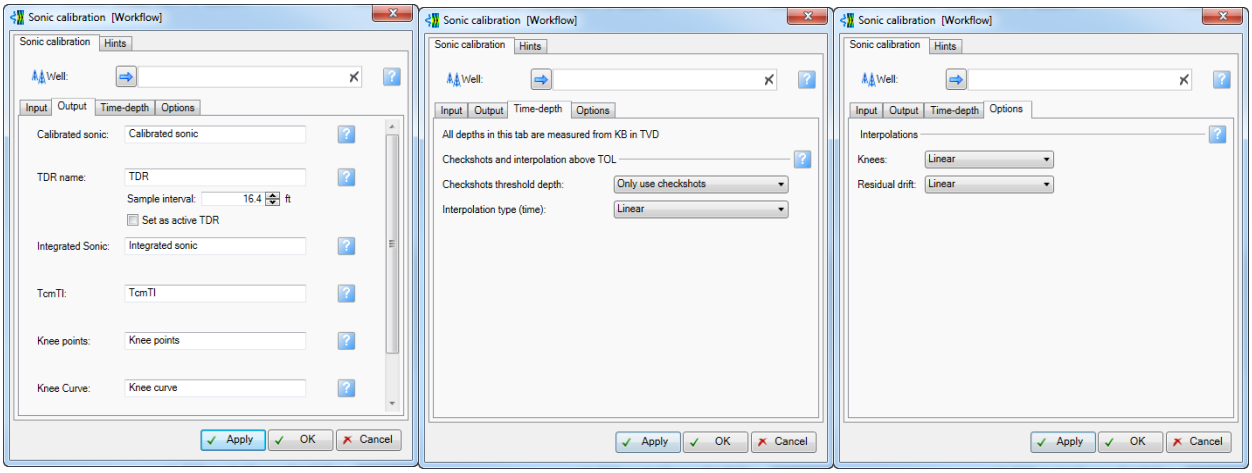
C: The checkshot folder can be dropped from the Input pane.

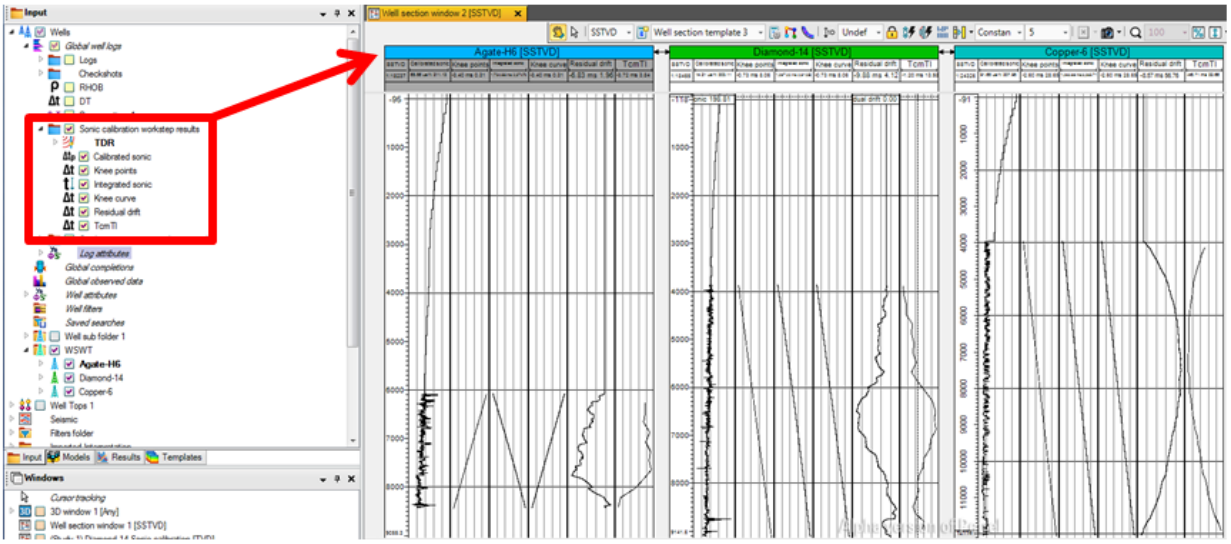
If the external outputs are set up, the data will overwrite the dropped objects (Sonic log or TDR).

In the **Output** tab, the name of the calibrated sonic, TDR, integrated sonic TcmTI, knee points and knee curve can be defined. Also, the resultant TDR can be set as active for the wells and the sample interval can be assigned for that TDR. All those outputs will be saved under the Global well logs folder.

In the **Time-depth** tab, as is normally done in a seismic well tie study, the way the checkshot will be treated can be defined.

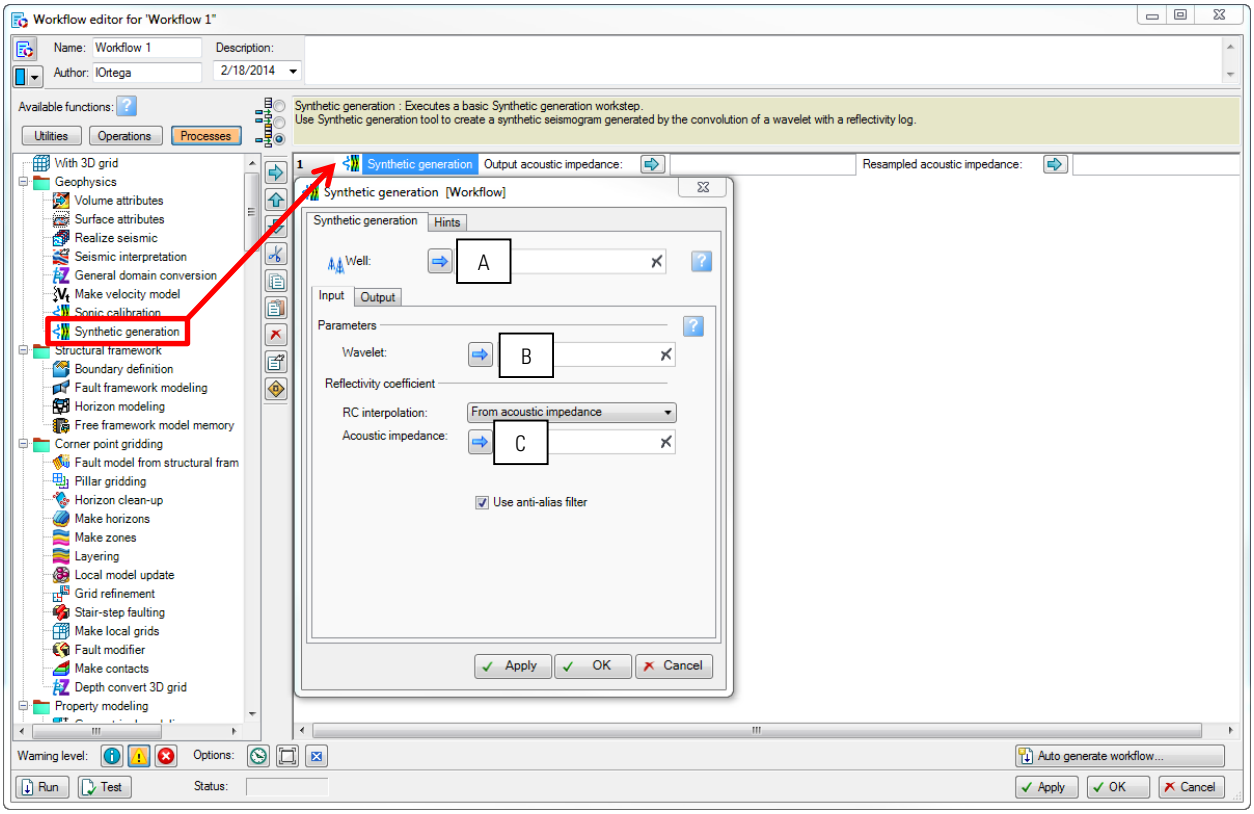
In the **Options** tab, the knees interpolation can be defined.





Multiwell Synthetics Generation (Workflow editor)

The process for Synthetics Generation is present now under Geophysics process, the **Workflow editor**. This process can be applied to one well or a workflow can be configured to apply the process to different wells using the loop function.



Inputs:

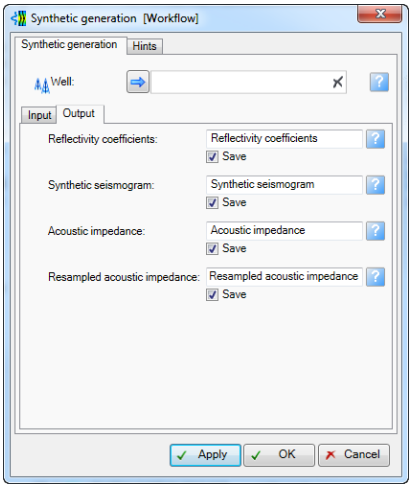
A: The well or a variable can be dropped as input from the **Input** pane.

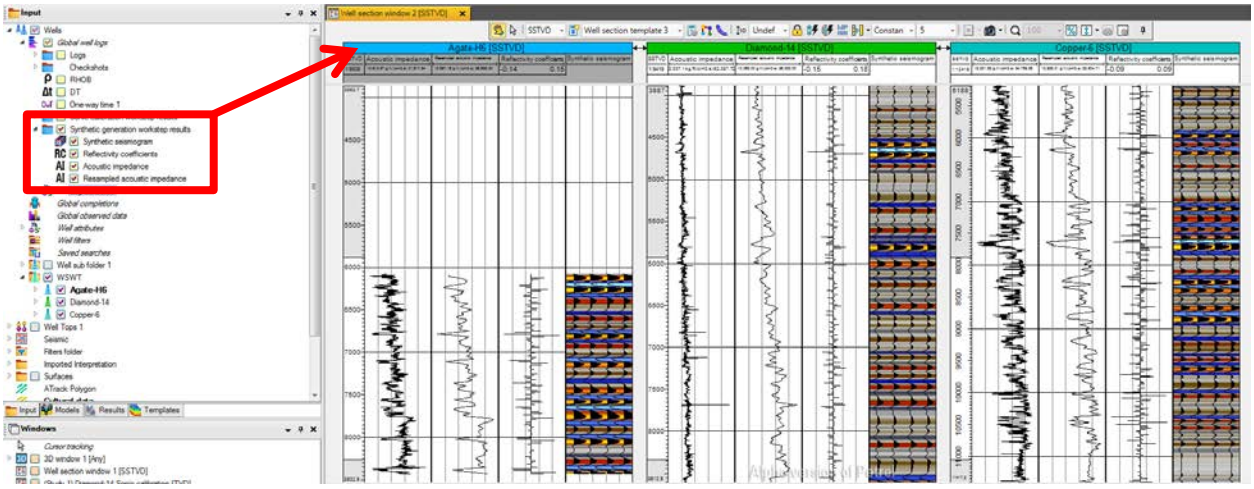
B: The Wavelet or a variable can be dropped from the **Input** pane.

C: The input logs for the reflection coefficient computation must be dropped from the Global well logs folder. (Two RC interpolations are available: “From acoustic impedance” and “From sonic velocity and density”.

If the external outputs are set up the data will overwrite the dropped objects (acoustic impedance and resampled acoustic impedance).

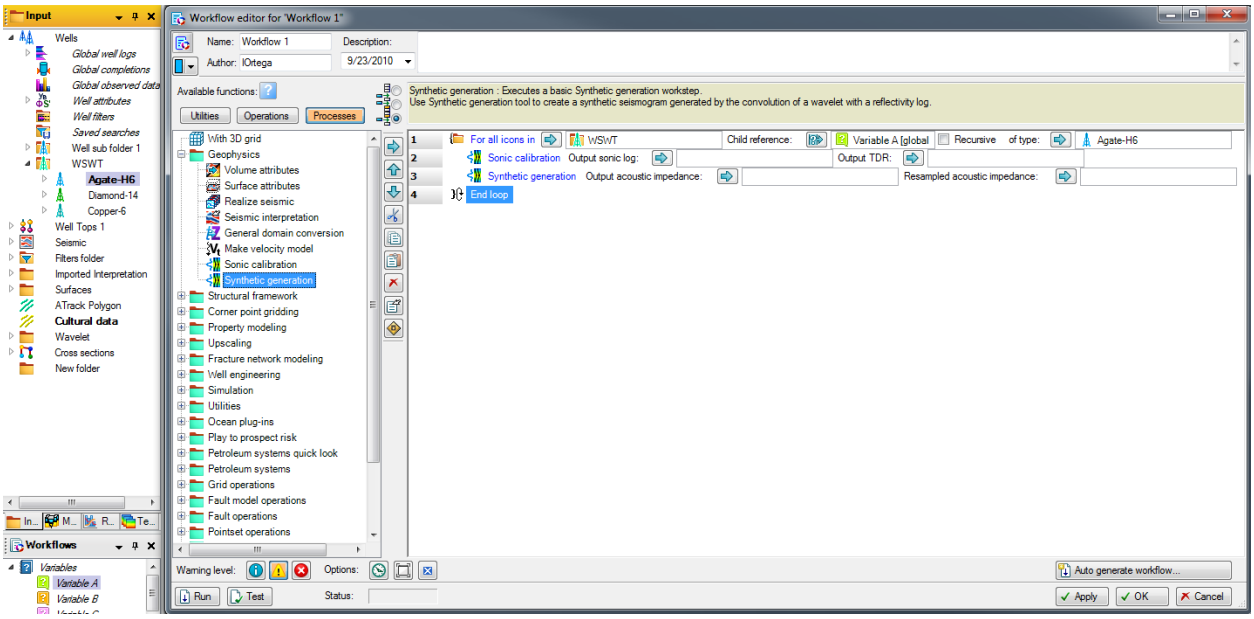
From the **Output** tab, the user has the option to save or not to save the Reflectivity coefficients, Synthetic seismogram, Acoustic impedance and Resampled acoustic impedance. The option to save the logs is activated by default. All these results will be saved under the Global well logs folder.

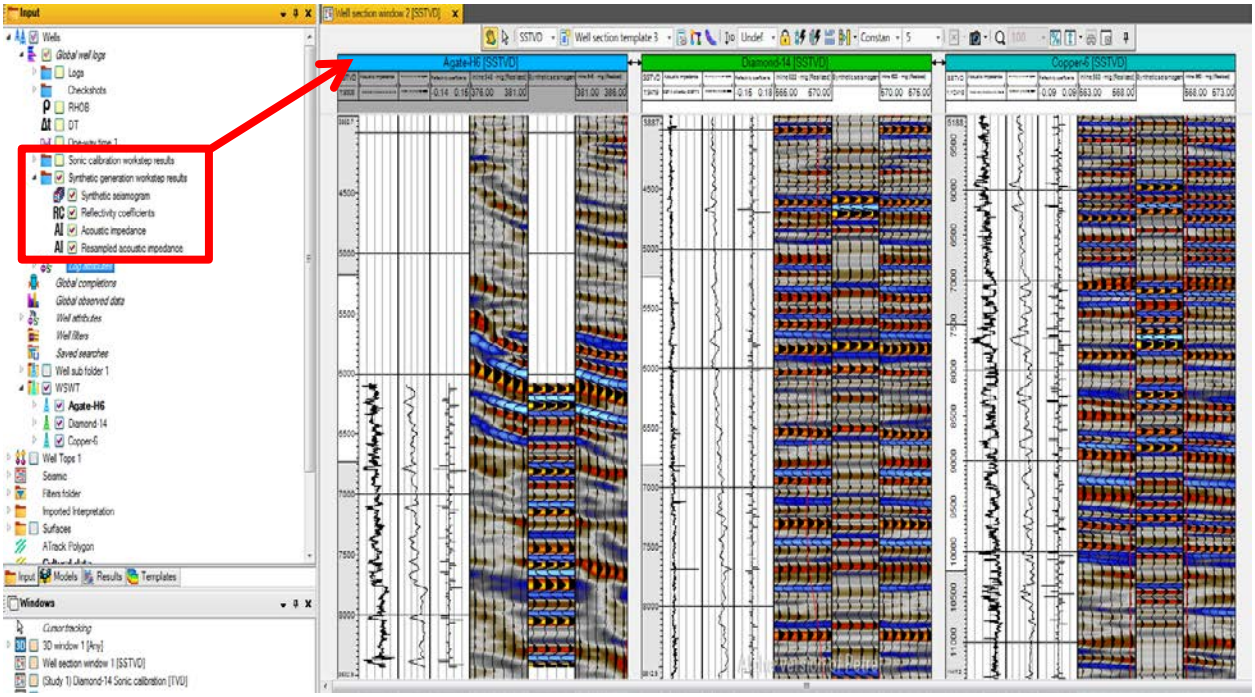




Basic workflow for multiwell sonic calibration and synthetics generation

In the attached workflow, the sonic calibration and the synthetics generation will be performed for all wells in the WSWT folder (Agata-H6; Diamond-14 and Copper-6).





Log Conditioning - Trend analysis and frequency filter license dependency

In Petrel 2014.1, the trend analysis and frequency filter is under the Geoscience core or Combined core.

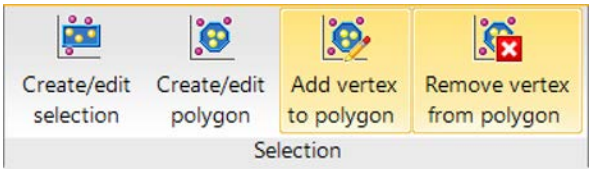
Geophysics: Quantitative Interpretation

2014.3

Add or Remove vertex from a polygon selection in the Crossplot window

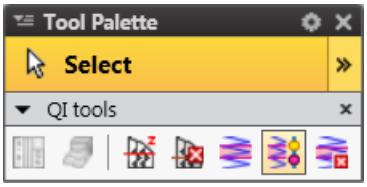
As part of the continuous improvement of the Crossplot window, new functionalities have been introduced into the selection mechanisms. Via the Crossplot window toolbar or in the Crossplot window tab at the ribbon, it is now possible to add or remove a vertex (inflexion point) from an active polygon selection.

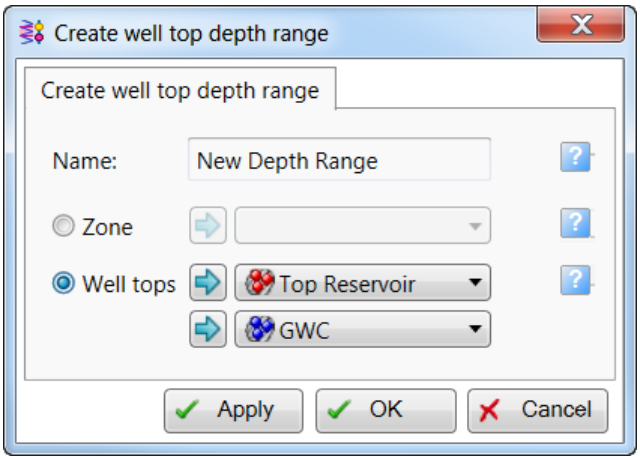
This is a basic set of functionalities that enables the ability of refining the set of data to be analyzed in the Quantitative Interpretation Workflows.



Create well depth range using Well tops

The Well Top depth range tool helps you to define the depth interval for the Rock Physics calculation based on the depths of the Well Tops. With this feature, the Rock Physics calculations can be expanded for multiple wells at the same stratigraphic intervals.

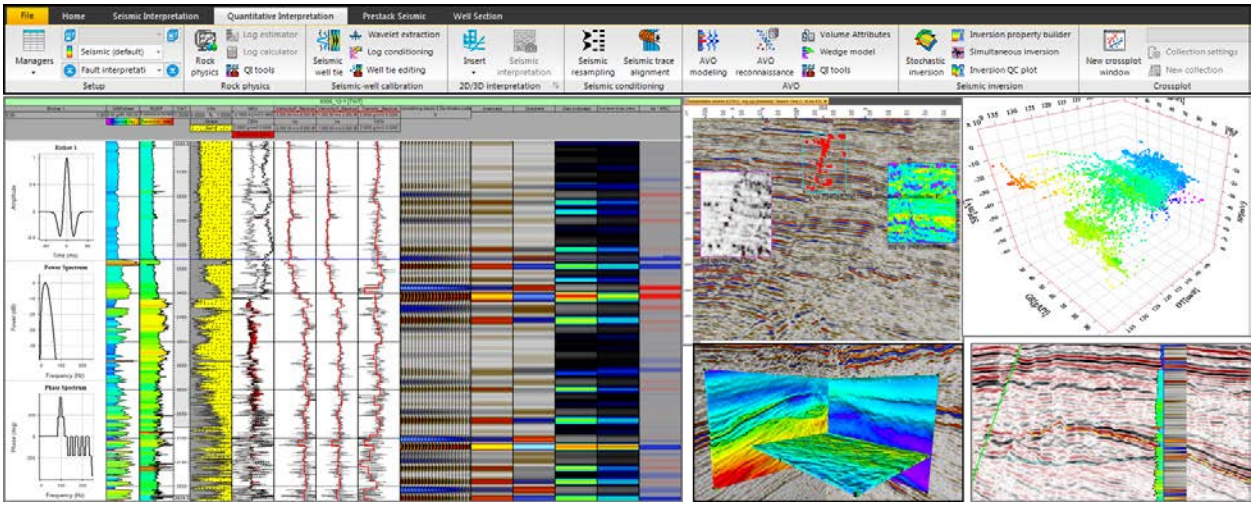




2014.1

The Quantitative Interpretation tools are aimed at helping the Geological and Geophysical interpretation and characterizing the reservoir. They will extract additional information from the input data (logs and seismic data) through different processes:

- Rock Physics
- AVO Modeling
- AVO Reconnaissance
- Seismic Inversion
- Cross-plot Analysis



Quantitative Interpretation builds the bridge between seismic and rock properties and helps to interpret the lithology and the pore fluids. It also helps to identify hydrocarbon anomalies by generating different AVO attributes and providing some powerful tool for cross-plot analysis. These techniques, for the seismic interpreter, will provide a quantitative approach to mitigate the risk by reducing the uncertainty.

Rock physics

Despite good seismic quality and well control, the correct interpretation requires quantifying the connection between geology and seismic data. Qualitative seismic interpretation is limited to identifying geologic features from seismic reflection data. However, quantitative seismic interpretation techniques seek to extract extra information about the subsurface rocks and their pore fluids from the reflection amplitudes, i.e., rock physics models that can guide the interpretation.

The Rock physic process enables the ability to establish the connection between the seismic data and the geology based on the input logs. The following are the different Rock Physic operations that are available:

- Elastic parameters estimation by using elastic constants relations for isotropic media
- Shear velocity estimation, an operation that allows generating S-wave velocity log from a P-wave velocity log, by using either the Arco Mudrock estimation or the Greenberg-Castagna estimation
- Log blocking Backus, a method of blocking the properties of a stack of thin isotropic layers so they are similar to the average properties of a single thick layer to tile to seismic frequency
- Fluid substitution, a process that uses Gassmann's relations (1951) to model the effects of changing fluid properties on elastic logs; Gassmann's relations are used to predict saturated rock properties from dry rock properties and vice versa

AVO modeling

The AVO modeling workflows have key elements, which take the interpreter to understand how the subsurface lithology and fluid content impact the elastic properties behavior, such as P-wave velocity, S-wave velocity, and Density, which directly affect the seismic response.

This module computes AVA responses creating synthetic gathers that explain the rock physics properties in terms of seismic amplitudes, applying a variety of convolutional algorithms for AVA modeling:

- Zoeppritz: Describes the reflection coefficient for plane elastic waves as a function reflection angle at a single interface
- Aki & Richards: A well-known approximation to the Zoeppritz equations, which assumes weak layer contrasts
- Shuey: Approximation of the Aki & Richards equations, which can describe the variation of amplitude at different angular ranges (near, mid and far angles near to the critical angle)

AVO reconnaissance

The AVO reconnaissance workflow helps to identify and classify AVO anomalies in an exploration context. It generates primary and secondary AVO attributes from Post and Pre-stack data. The Volume of Interest coupled with the new functionality to display the gathers on seismic lines in an interpretation window allows quality control, and using the cross plot tool allows a quick AVO analysis.

Crossplot window

The new Crossplot window canvas is used to display data as points plotted against each other. This is used by:

- Continuous well logs
- Discrete well logs
- Seismic logs
- Post-stack seismic data
- Pre-stack seismic data

2D or 3D crossplots can be created to analyze the mentioned data, and it brings the big advantage of analyzing different types of data by adding different collections in the same canvas.

The Crossplot window is completely useful when it participates in the Quantitative Interpretation workflows, allowing the generation of:

- Useful Geophysical Crossplots which provide clues as presence of hydrocarbons:
  - P-wave impedance vs S-wave impedance
  - P-wave impedance vs Vp/Vs ratio
  - Lambda*Rho vs Mu*Rho
  - Acoustic impedance vs Poisson's ratio
  - Poisson's ratio vs Young modulus
- AVO or AVA crossplots, using the Z-level tool, which helps to understand the amplitude variations with offset or angle by reading the information contained in gathers displayed in the Well section window.
- Intercept versus Gradient crossplots, to recognize the AVO responses for different types of gas sands according to the classification scheme suggested by Rutherford and Williams (1989).

Wedge Model

The wedge model tool creates synthetic seismic angle stacks from hybrid models consisting of a reservoir model embedded in an elastic property volume. Alternatively, the synthetics can be generated from 3D high-resolution inverted elastic properties. A synthetic wedge model can be really helpful in generating a synthetic AVO response for resolution and survey design studies. It helps in scenario testing for quantitative seismic interpretation and reservoir characterization.

Seismic Conditioning

Quantitative interpretation is highly sensitive and dependent on the quality of seismic data, which often require conditioning before seismic inversion to elude misleading reservoir property estimations. The seismic conditioning tool allows seismic resampling and seismic trace alignment, which provide optimum quality input for QI workflows. Seismic resampling allows seismic resampling of seismic cubes or 2D lines, both spatially and temporally. Seismic trace alignment is based on non-rigid matching (NRM) and optimally aligns angle stacks or time-lapse seismic cubes (4D inversion workflow).It also generates and saves displacements fields for analysis and interpretation.

Simultaneous Seismic Inversion

Based on the proven ISIS suite of reservoir characterization technology used for long by Schlumberger for its Inversion Consulting Services, Simultaneous Seismic Inversion is an industry-leading model-based pre- and poststack deterministic inversion, which jointly inverts multiple inputs to compute elastic properties. It relies on simulated annealing, a global optimization mechanism to find the best subsurface model and it can effectively handle AVA seismic, converted waves (PP/PS AVA), time-lapse full-stack seismic, and time-lapse prestack seismic data in inversion workflows. The seismic inversion can be run by using fixed wavelet(s), or an attenuation model can be derived and applied to the wavelet(s) to allow for temporal changes over the time window of inversion.

Inversion Property Builder

The Inversion Property Builder allows interactive generation of the low frequency prior model of elastic properties, which in general cannot be obtained from the seismic data. Well logs are interpolated or extrapolated throughout the volume using the seismic horizons, with the well log data stretched and squeezed between the horizons, according to the layer thickness given by the horizons. An additional guide model may be used to add spatial information between wells and also depth trend control might be specified. The Inversion Property Builder also allows the blind well test, which is a very critical QC step of inversion property building.

Inversion QC Plot

The Seismic Inversion QC Plot tool provides a quick and intuitive access to quality control generated inversion outputs, property datasets, and synthetic seismic. These results must be evaluated against other data, such as well properties and prior models.

Stochastic Seismic Inversion

The Stochastic Seismic Inversion tool addresses the challenges of vertical resolution as pre- and poststack seismic data are inverted directly into high-resolution Petrel 3D geological models, so that inversion results are immediately available at the appropriate scale for integration into reservoir characterization and dynamic flow simulation workflows. Unlike deterministic seismic inversion, the stochastic method accounts for non-uniqueness of the inversion process by delivering multiple realizations consistent with the available well and seismic data. It helps provide better constrained reservoir properties and a better understanding of the uncertainties in the reservoir model.

Geology

Geology: General

2014.3

Pillar gridding

The option to set the number of cells by the Pillar gridding process, is now supported in the **Fault contextual** tab and **Fault mini toolbar** when a fault with a trend is selected.

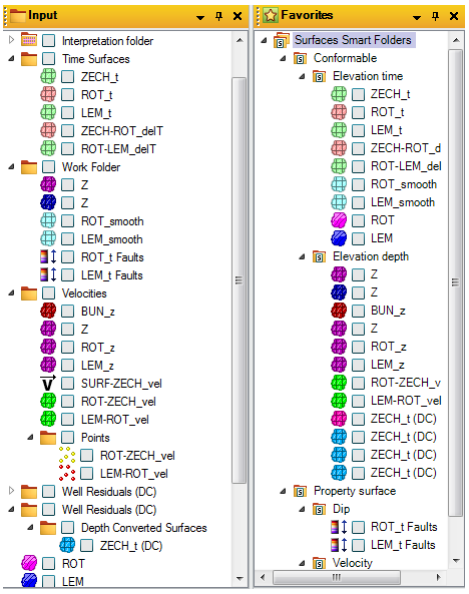
2014.2

Smart folders for surfaces

You can now create different instances of a hierarchical tree in the **Favorites** pane to give you quick access to individual or group of surfaces by using a few common sorting attributes.

Surfaces are typically generated continuously in the input tree during the entire life of a Petrel project.
The **Input** pane (on the left) shows a typical tree structure as the project is populated with an increasing number of surfaces. The **Favorites** pane (on the right) shows the same inventory of surfaces contained in the project, which are sorted in a user-preferred hierarchy. The tree structure is dynamic and responds to your selected preferred attributes in their sorting order.

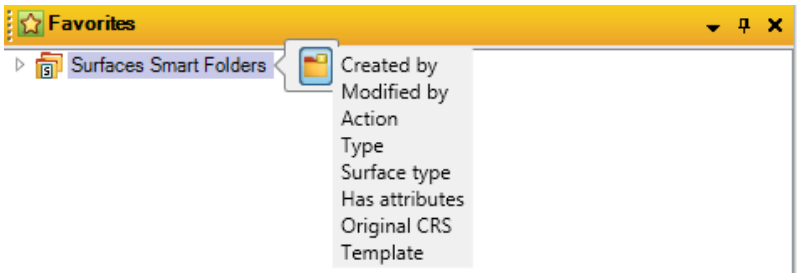
Figure 50: Favorites pane showing instance of the surfaces smart folders hierarchical tree



How to create surfaces smart folders and group surfaces with selected attributes:

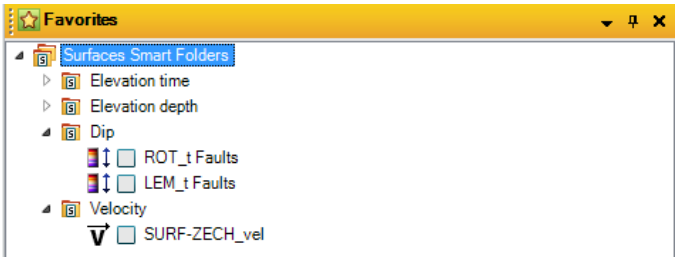
- From the **Home** tab, in the **View** group, click **Panes** and select **Favorites** from the split button. The Favorites display in Petrel Explorer.
- Right-click in the **Favorites** pane and select **Insert surfaces smart folder** from the context menu
- Right-click **Surfaces Smart Folders** and select **Select attributes**
- Click the attribute selector icon to select the criteria to group the surfaces

Figure 51: Attributes selector on surfaces smart folders



Expand the surfaces smart folder to visualize the hierarchical tree.

Figure 52: Surfaces grouped by template



- To group surfaces on more than one attribute, it is possible to select additional criteria by clicking on the attribute selector icon.
  - Attributes can be reordered using drag and drop functionality
  - Attributes can be removed by clicking the X icon in the attribute selector

Attribute selector descriptions:

- The attribute selector panel has eight attributes available. All these criteria can be combined.
- **Created by**: As defined in the surface settings history of the Info tab. It looks in the history section and picks the user who created the surface to group and make smart folders of the surfaces
- **Modified by**: As defined in the surface settings history of Info tab. It looks in the history section and picks the user who modified the surface to group and make smart folders of the surfaces.
- **Action**: This method is used to create the surface in Petrel as defined in the Info tab under the action section of history (make/edit surface, Import CPS-3 grid, convert horizon to surface, residual from well correction, velocity model process, faults in fault model converted to surface, etc.)
- **Template**: Defined in the Info tab, such as time / depth/ thickness/residual/velocity (Vo) /porosity, etc.
- **Type**: Surface could be structured (use bitmap or grayscale contour image as input, faults in fault model converted to surface), regular (2D grids created by make/edit surface process or imported), triangulated (created

from multi-z values interpretation). Type, as a surface attribute, is picked by looking at the 'type'. It gets the information from the Info tab and the **Statistics** tab of the description section to group surfaces in to folders.

- **Surface type**: Groups surface into smart folders, using the surface type in the Info tab of the surface, namely conformable, erosional, base, discontinuous, and intermediate. Other surface types include: thickness, property surface (surface created by velocity modeling process, Residual).
- **Has attributes**: As defined in the action and description tabs of the parent surface history in the Info tab. This is used to group surfaces into folders containing those with and without attributes and use the criteria to group surfaces into folders.
- **Original CRS**: It looks at the Original CRS information in the Info tab of the surface and uses it to group surfaces into smart folders. The original CRS name, e. g., UTM84-33N is used without the CRS description. Surfaces without defined original CRS will be grouped into an 'undefined' smart folder.

Tips and limitations of Smart Folders are listed below:

- In order to preserve different structures defined by the Smart folders criteria, it is possible to generate shortcuts folders in the **Favorites** pane by right-clicking a smart sub-folder and selecting "Create shortcuts".
- The Surfaces Smart Folders are sensitive to most updates made to a Surface object (e.g., via the **Settings** tab). To sort on updated surfaces that have been added, edited, or deleted on the Input tab, right-click **Surfaces Smart Folder**s and select **Load/refresh smart folders**.
- It is possible for an object to be listed more than once, for example, when sorting under the attribute 'modified by' when multiple users have edited the object.

Limitations:

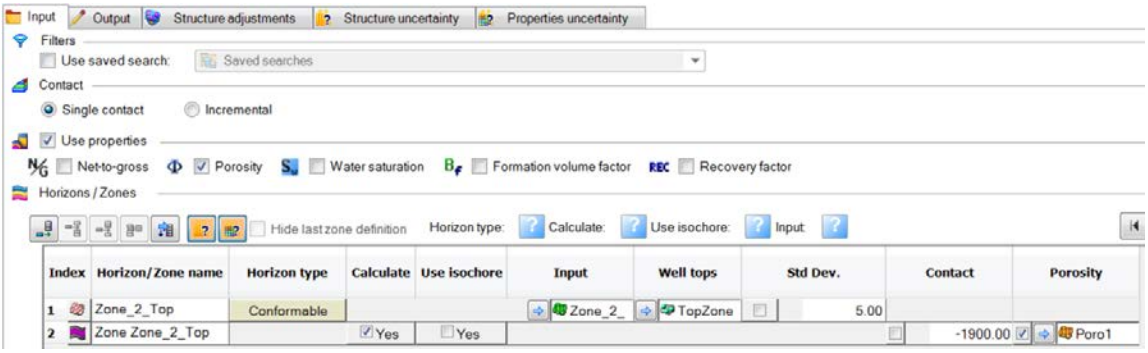
- Surfaces Smart Folders and their converted favorites folder equivalent do not have a toggle check box to select all folder content items for visual display. To toggle on/off the display of all items within a single folder level, multi-select the folder content and press the spacebar.
- Surfaces Smart Folders and their converted favorites folder equivalent are not recognized in the **Workflow Editor**.
- Surfaces Smart Folders content objects and their converted short-cut favorites equivalent may not be recognized as input to certain processes.
- Surface Smart Folders need to be refreshed when reopening a project by right-clicking **Surfaces Smart Folders** and selecting **Load/refresh smart folders**.

2014.1

Map-based volume calculation – uncertainty

When conducting Uncertainty and Optimization (U&O) analysis, this functionality allows quantification of volume uncertainty based on stochastic realization of the surfaces used to create the structure and properties in the volumetrics case.

Figure 53: Map-based volume calculation process window



The two tabs are activated by toggling the buttons on the **Input** tab. In the Std Dev. column, you can specify the uncertain inputs. Then specify the relevant settings on the **Structural uncertainty** tab.

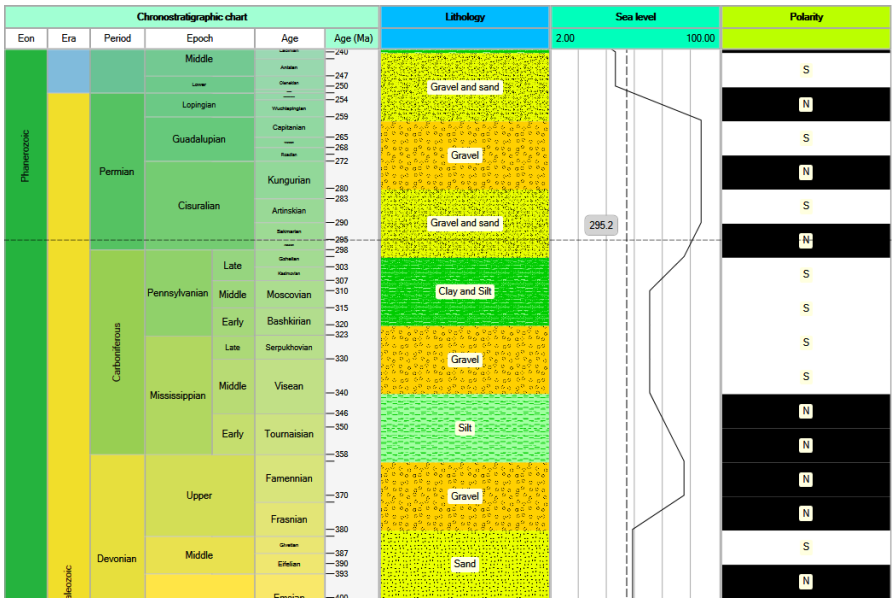
Follow the same procedure for property surfaces by using the **Properties uncertainty** tab. Set the uncertain property surfaces by specifying the Std Dev value and the relevant variogram settings.

Stratigraphic charts window enhancements

Many usability enhancements were made to the **Stratigraphic charts** window. Right-click the chart to see options for zooming and turning the cross line on and off.

An auto coloring option has been added to the context menu of columns in the tree. When using this option, you will be able to apply coloring schemes using three different color table templates (Figure 54).

Figure 54: New stratigraphic chart look in 2014.1



Surface Inspector – 2D Volumetric

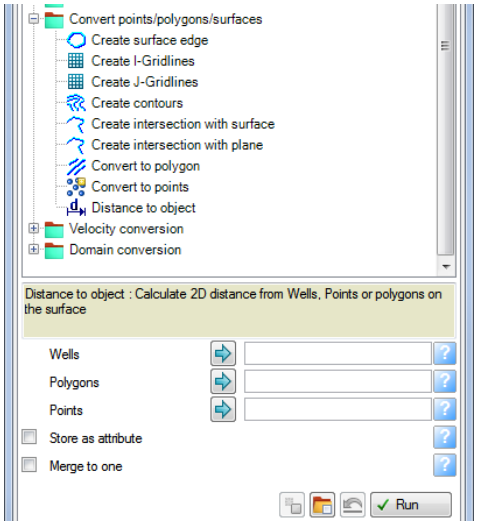
For regular surfaces, the Surface Inspector now shows volumetric parameters for Porosity, Bo, GRV, HCPV and STOIIP. A list of units is available from the drop-down menu for each volume reading. Changing the unit will result in a conversion of the volume, which is also recorded on the message log. Use the **Inputs** and **Results** buttons to place information onto the clipboard.

Figure 55: 2D volumetric section on the surface inspector in 2014.1

Surface operations – distance to object

This new operation generates surface attributes based on the 2D distance of objects on a given surface. You can use a point set, a polyline, and a set of wells as objects. You can find this operation in the Convert points/polygons/surfaces folder (Figure 56).

Figure 56: Distance to object surface operation



Enhanced contouring

When displaying surfaces in a Map window, the default contouring method in the Settings dialog is now the **Enhanced** method. This uses the **Contours** operation for coloring instead of the other options available on the **Solid** tab. The **Contours** operation also works if the solid is different from the source.

If the **Enhanced** method cannot be used, it will fall back to the **Classic** method and send an alert to the message log.

Contour attribute coloring

When displaying a surface attribute, you can control the solid color increment, independently of the increment for the contour lines. This is possible with both **Classic** and **Enhanced** contouring.

When using the Classic method can you choose **Smooth**, **Bold**, or **Contours** for the Solid coloring operation.

Algorithmic improvements to Enhanced contouring

There is now better handling of the coloring along the faults, when the solid is different from the contour source, and solid contour coloring, as illustrated in the following figures.

Figure 57: Contouring in Petrel 2013 (left) compared to 2014 (right)

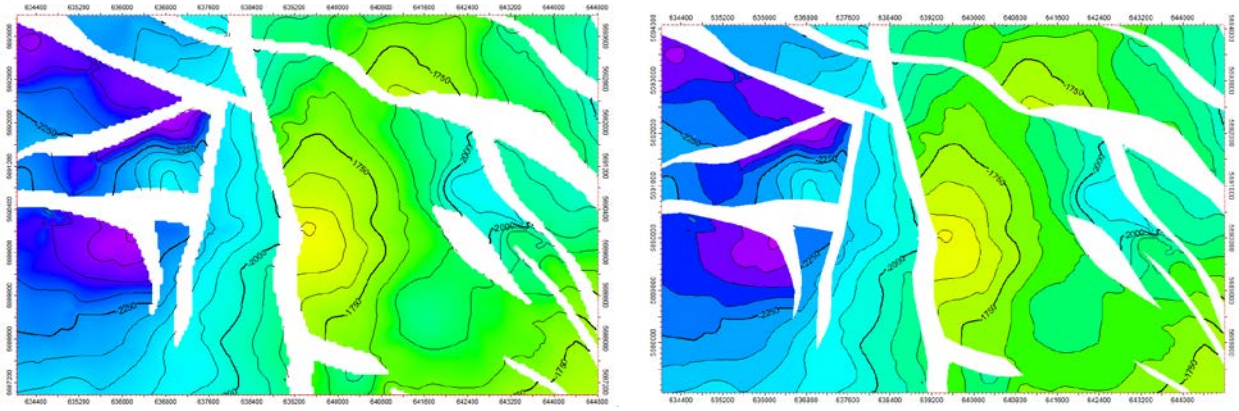
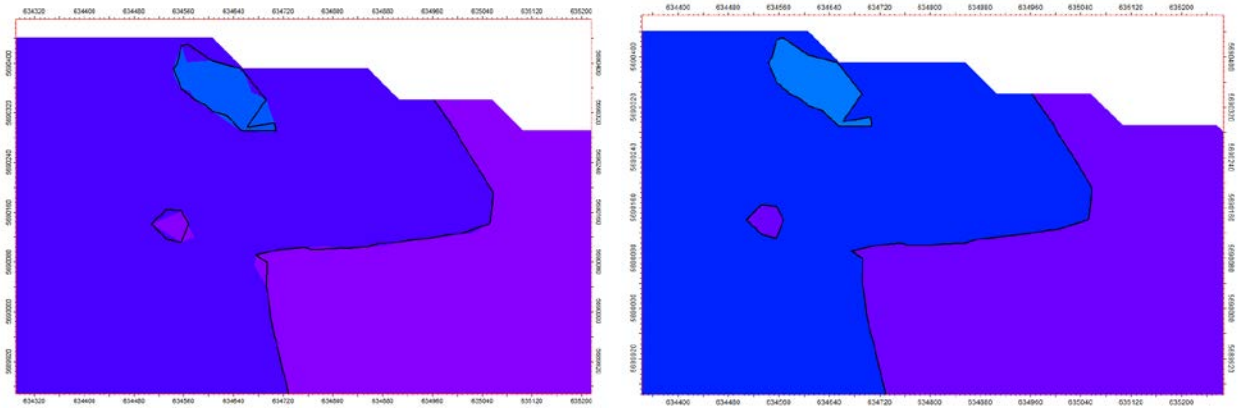


Figure 58: Solid contour coloring in Petrel 2013 (top) and 2014 (bottom)



Geology: Well Section Window

2014.5

Well Correlation Ghost Curve

The new well correlation ghost curve allows you to create a single curve, multi-log single track, and multi-track ghost curves. You can display the well name, well log name, and index tracks with a ghost curve. You can also deactivate and reactive ghost markers, and add additional control points to a ghost curve.

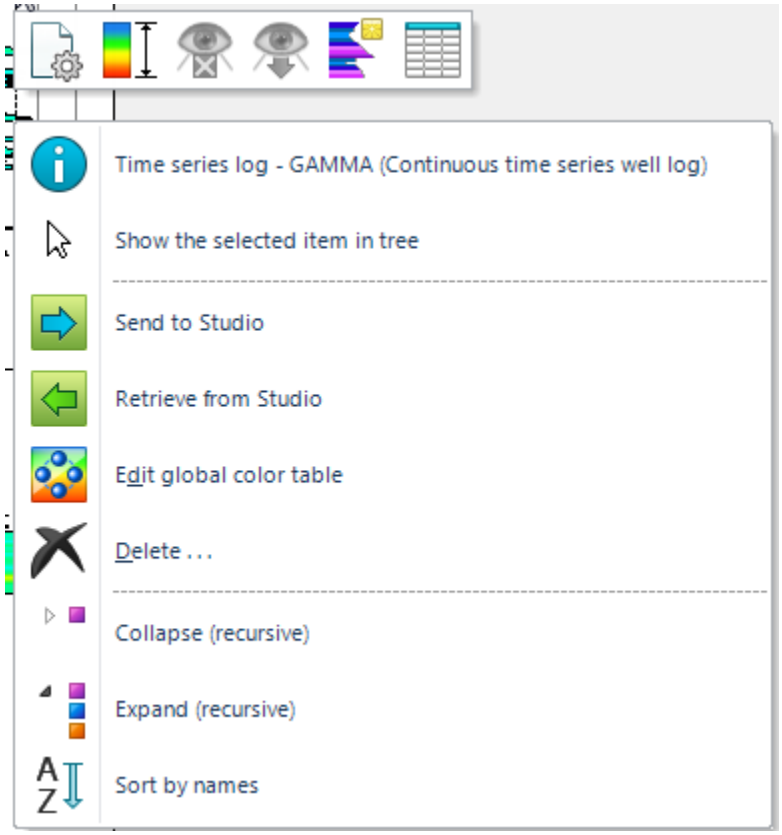
2014.3

Mini Toolbar

The mini toolbar is available for continuous and discrete logs, time series logs, the well header, point attributes, and index tracks for wells displayed in the Well section window. The mini toolbar contains tools and functions associated with the well log, well path, point attributes, and time series data.

You can right-click a continuous or discrete time series log to display the mini toolbar and context menu.

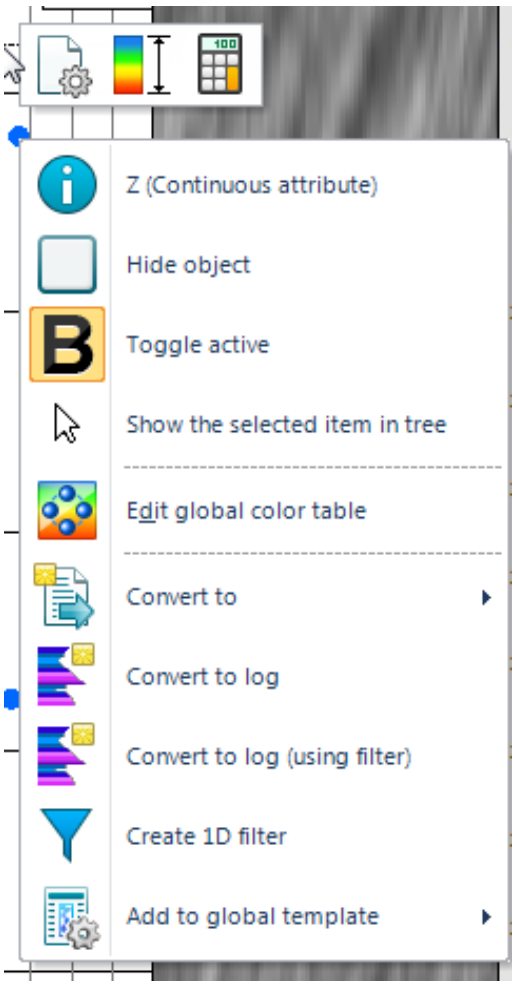
Figure 59: Time series log mini toolbar and context menu



The time series mini toolbar will be enabled after the time player has been activated in the 3D window.

You can right-click a point attribute to display the mini toolbar and the context menu.

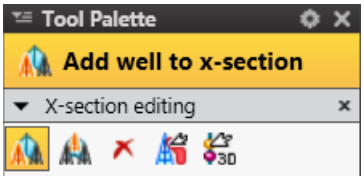
Figure 60: Point attribute mini toolbar and context menu



User Interface

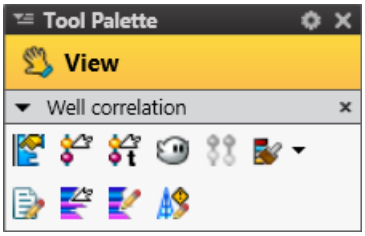
- The X-section editing Tool Palette is now divided into two tool palettes: X-section editing and Well correlation.
- The X-section editing Tool Palette is available for creating and editing cross sections in a 2D, 3D, or Map window. The X-section editing Tool Palette is accessible from the Stratigraphy tab, in the Cross-section group.

Figure 61: X-section editing Tool Palette



- The Well correlation Tool Palette contains the tools and functions for well correlation in the Well section window. This tool palette is available in the Well section tab, and can be accessed from the Stratigraphy tab by opening the Edit well tops Tool Palette in the Well tops and zones group.

Figure 62: Well correlation Tool Palette



Cross Section

The following enhancements have been added to enhance interactivity editing cross sections:

- X-section add/edit well mode in 2D and Map windows: selecting a displayed well anywhere along the well path adds the well between existing well hinges if the selected well is orthogonal to the cross section. Pressing CTRL and selecting a well adds the well to the cross section after the closest well hinge to the selected well.
- X-section delete well: selecting a displayed well anywhere along the well path removes the selected well from the cross section.
- X-section add well mode: pressing SHIFT and selecting the well adds the well to the beginning of the cross section rather than the end.
- In any of the three modes mentioned above, when the cursor is moved over a displayed well path it will change to a cross hair. Otherwise, the cursor is a pointing hand.
- It is also possible to perform a single undo of the last cross section editing operation by pressing CTRL+Z.
- When in X-section add well mode, a pointer displays to help you track where the cursor is and where the next selected item will be added.

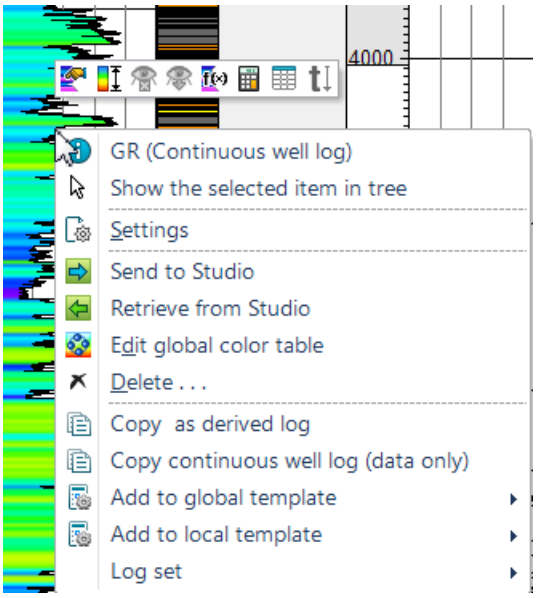
2014.2

Mini Toolbar

The mini toolbar is available for continuous and discrete logs, the well header, and index tracks for wells displayed in the Well section window. The mini toolbar contains tools and functions associated with the well log and well path.

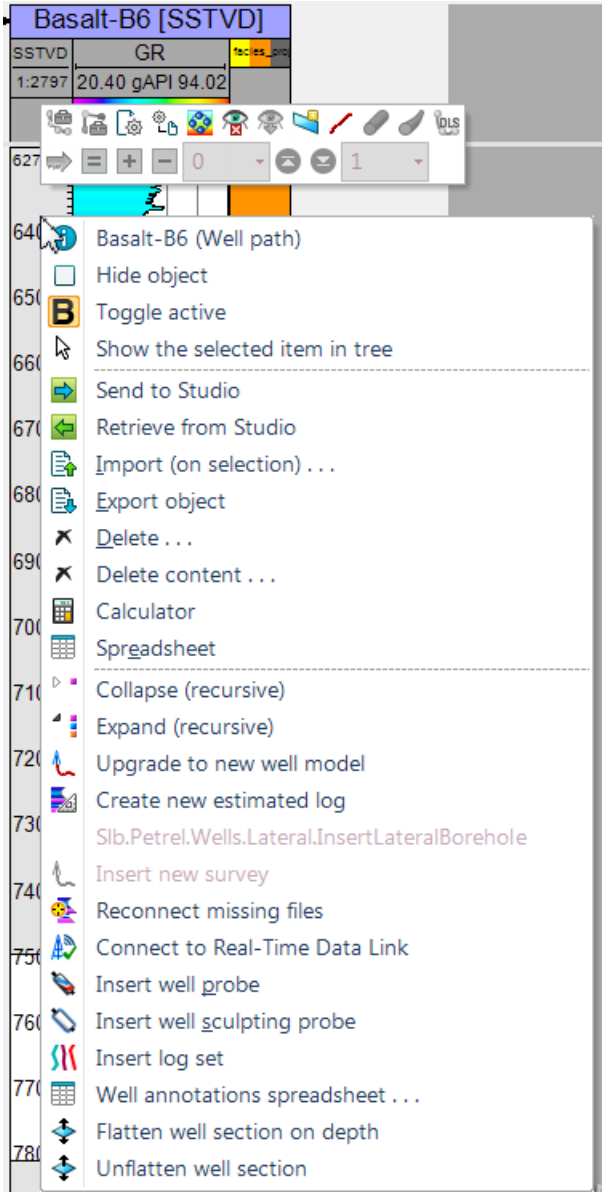
You can right-click a continuous log curve to display the shortcut menu and mini toolbar.

Figure 63: Continuous log curve mini toolbar and shortcut menu



You can right-click the vertical well header or index track to display the mini toolbar or shortcut menu.

Figure 64: Well header mini toolbar and shortcut menu



Polysection and Intersection Plane Cross Sections

It is now possible to create a cross section by interactively drawing an area of interest covering specific wells in a 2D window with the new polysection tool . The area of interest coverage can be interactively adjusted from the **Settings for x-section creation via polysection** dialog box, and the wells that are within the area of interest will be used to create a new cross section. A new cross section can also be created from an existing intersection plane by defining a distance limit for wells in proximity to the plane.

Figure 65: Settings for x-section creation via polysection dialog box

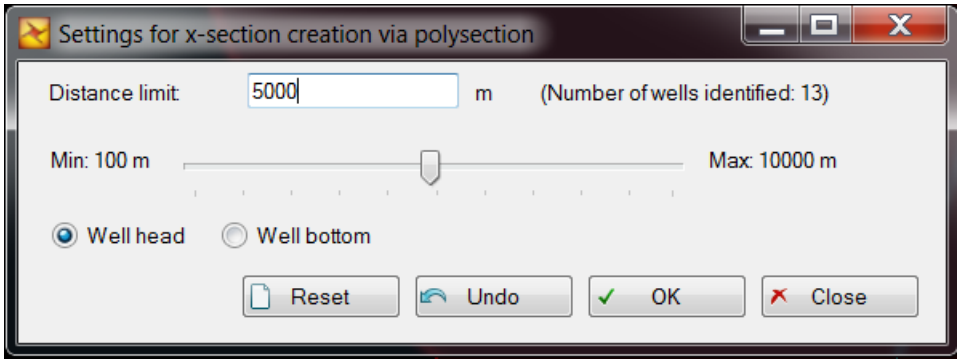
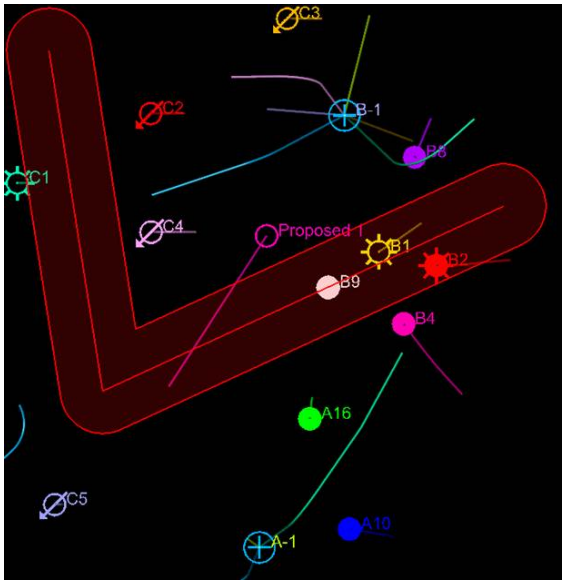


Figure 66: An area of interest identifying wells to be included in the cross section



2014.1

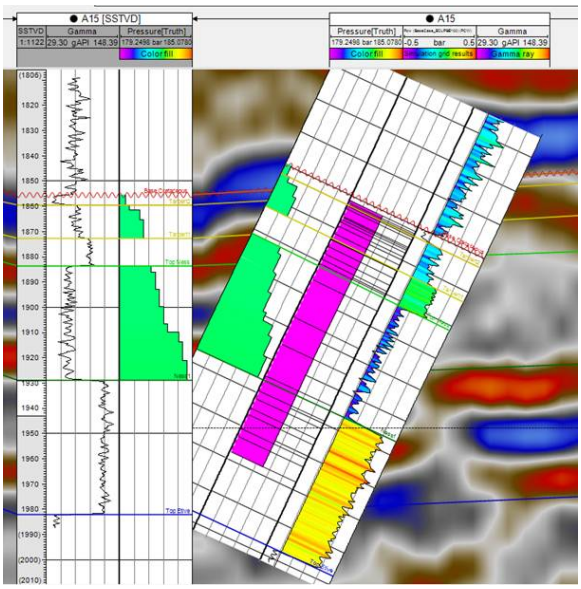
Deviated Display

The well section deviated template has been expanded to include additional tracks for Points attribute, Simulation grid results, Simulation log, Summation track, and Time series logs. The tracks can be added in the **Well section template settings** dialog box or in the **Input** pane.

Well tops can be created and edited interactively on the deviated display.

You can apply a color fill by clicking **Curve fill** on the **Tool Palette** or by using the shortcut menu.

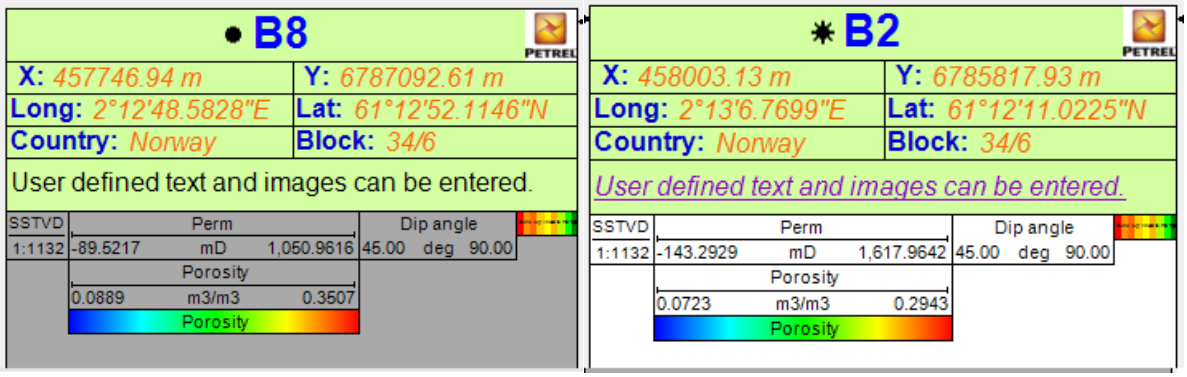
Figure 67: Deviated Tracks with Newly Supported Tracks



Customized Headers

The header is divided into a well header, a sub-header, and a log header. You can have different settings for vertical and deviated headers, or you can keep the same settings for both headers. The well header and sub-header are customizable and can be configured in the **Cross sections settings** dialog box for the global header or in an individual **X-section settings** dialog box for the local header. The well header consists of the well name, domain, symbol, and an image. The sub-header contains well attributes, user defined attributes, project settings, and user defined text. The log header is always displayed and cannot be edited. Customized headers can be copied and pasted between cross sections or shared via the Reference project tool.

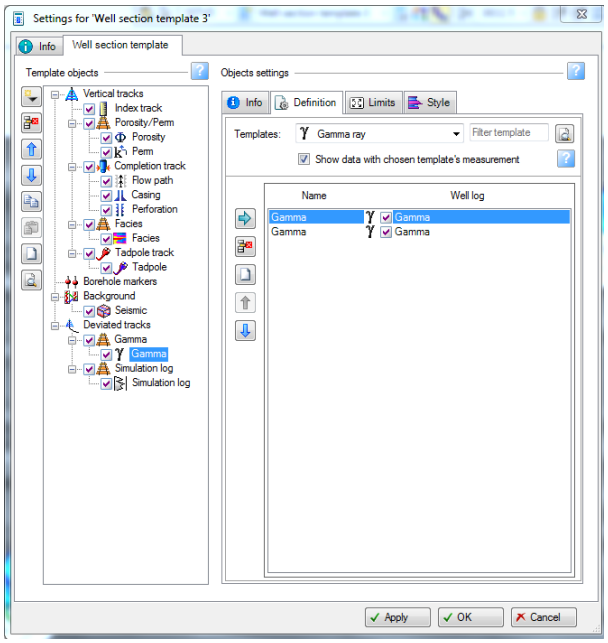
Figure 68: Customized Well Headers



Showing and Hiding Template Objects

You can show or hide tracks, logs, and individual objects, which allows you to turn the objects on and off in the Input pane without losing their style settings and position in the Well section template. The check boxes of the objects can be selected or cleared in the **Input** pane or in the **Well section template settings** dialog box to show or hide the objects in the Well section window.

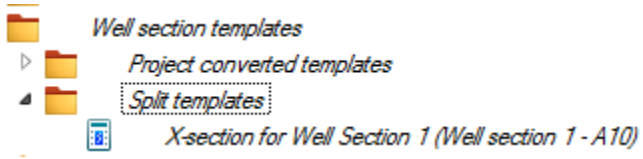
Figure 69: Settings for Well Section Template with Check Boxes



Local Template

You can now generate a local template by adding a log from an individual well, the log can be added to a vertical or deviated track. A new template will be created and named with the x-section, original template, and well name that were used to create the template. The local template is stored in the Templates pane > Well section templates > Split templates folder. The local template can also be used as a global template.

Figure 70: Example of a Local Template Structure

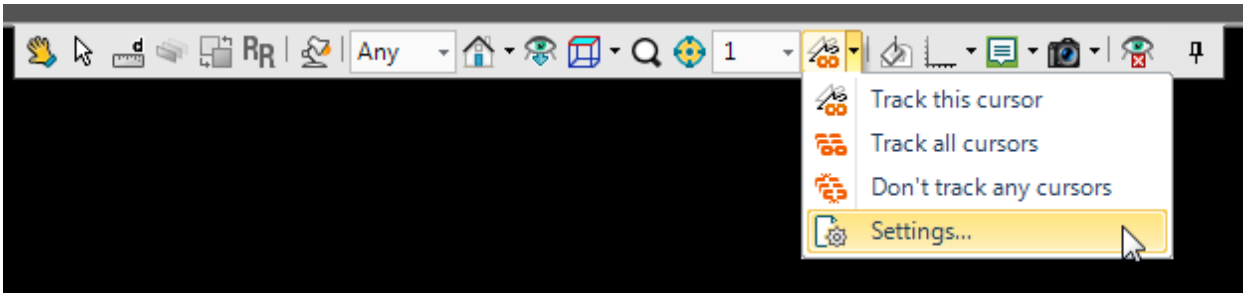


Global templates can be selected from the Select template list on the window toolbar. You can identify if a local template is used by a well in the **Definition tab**, in the **X-section settings** dialog box. The template name can also be displayed by enabling the customized well sub-header.

Well Section Window Interaction with 2D, 3D, and Map windows

Cursor tracking allows you to mirror your cursor location from the Well section window to other enabled windows when you hover your mouse over a vertical track using the Select/Pick mode. The cursor tracking can be enabled or disabled from the window toolbar when you are working with a 2D, 3D, or Map window.

Figure 71: Cursor Tracker



Cross Section Label

When a cross section is displayed in a 2D, 3D, or Map window, the cross section name will be posted at the end of the cross section. You can configure the position, color, and font size for the cross section label in the **X-section settings** dialog box.

Figure 72: X-section Settings Dialog Box with Show Labels

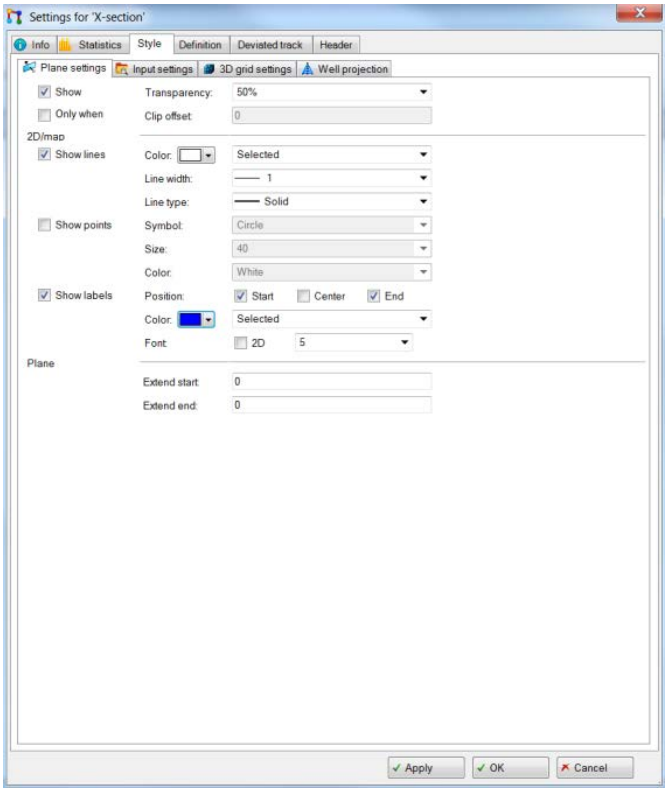
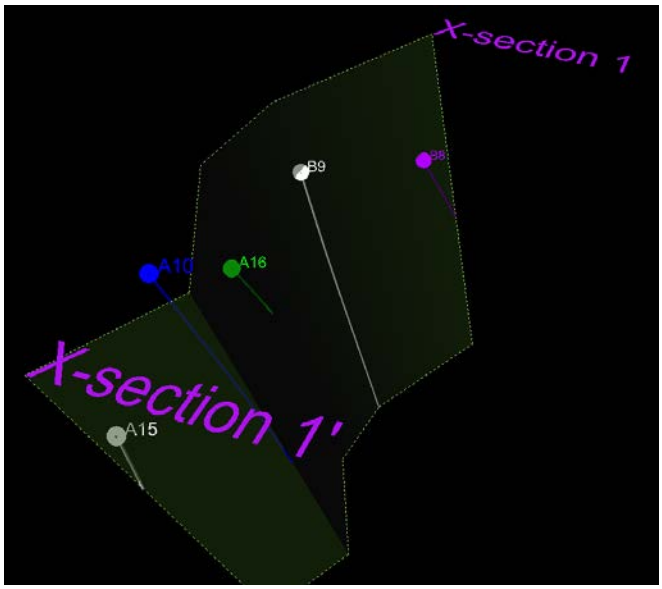


Figure 73: Example of an X-section with a Label

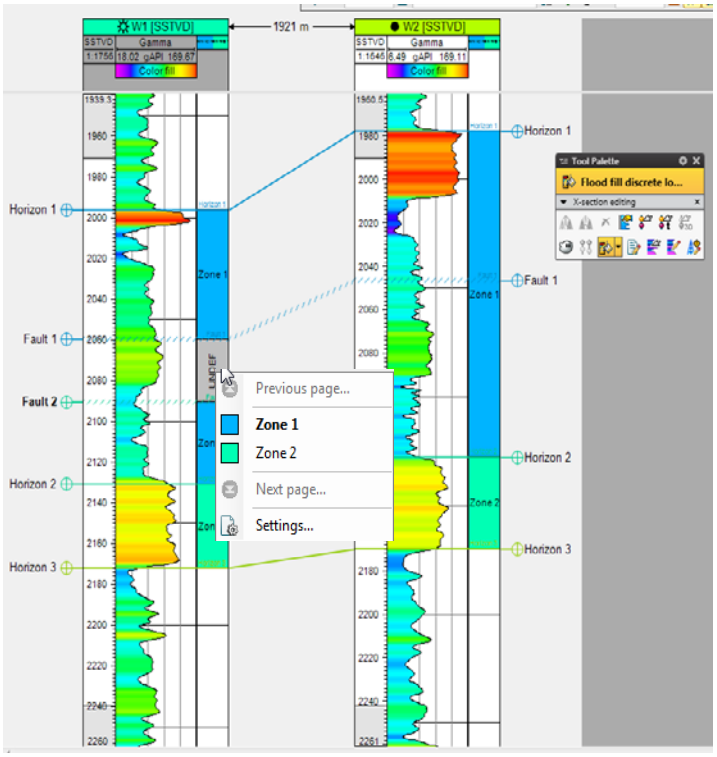


Zone Log Editing

You can now overwrite invalid zones by using the Flood fill discrete log icon.

- Right-click the zone log, select the zone you want to apply, and then click the zone. The zone will be reassigned.
- To undo the reassignment of the zone, the top-most well top needs to be deleted and re-picked.

Figure 74: Reassigning a Zone Log



Viewport

The Well section viewport allows for higher resolution plotting, which allows for a better quality plot from the Plot window. The creation and manipulation of the Plot window and the Viewport remains the same.

Figure 75: Creating a New Well Section Viewport

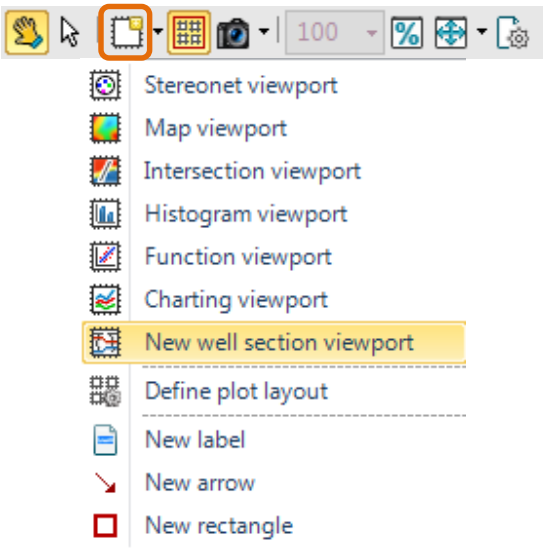
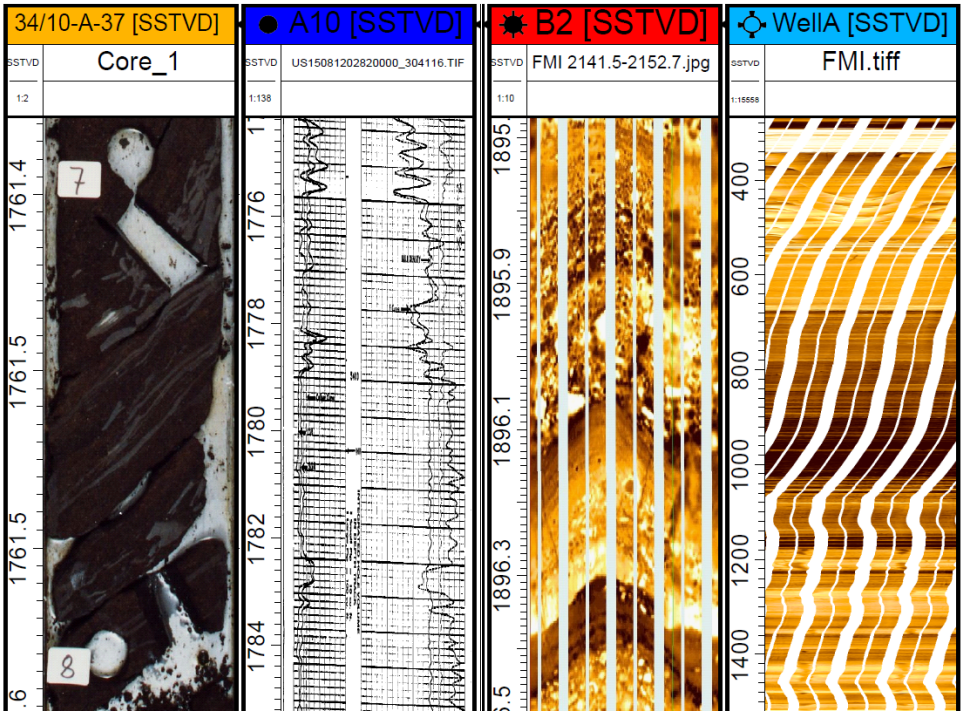


Figure 76: High Resolution Printing

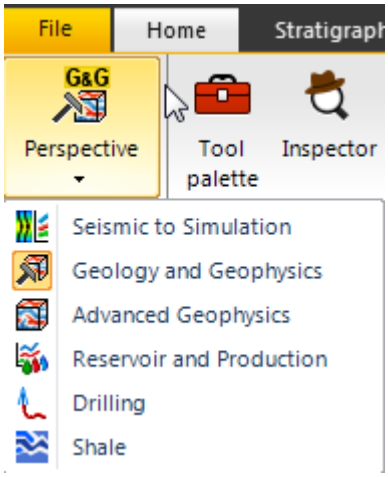


Stratigraphy Tab

The **Stratigraphy** tab contains all the relevant tools for the well section workflow from stratigraphic charts, log creation and manipulation, to creating well tops and surfaces. The **Stratigraphy** tab is available only when you select the **Seismic to Simulation** or **Geology and Geophysics** perspective.

Figure 77: Stratigraphy Tab

Figure 78: Perspective Selection

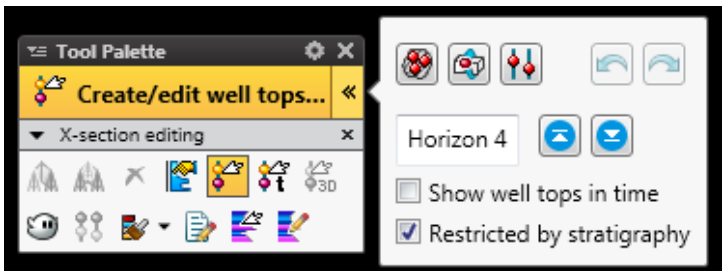


Tool Palette

Click **X-section editing** to open the Tool Palette from the **Stratigraphy** tab when you have a 2D, 3D, or Map window active.

Click **Edit well tops** to open the Tool Palette from the **Stratigraphy** tab when you have a Well section window active. When Edit well tops icon is activated, additional icons are available to create and edit markers.

Figure 79: Edit Well Tops with additional tools



The Tool Palette can be resized, repositioned, and expanded to include other domain's tool palettes.

Figure 80: Tool Palette in a 2D Window

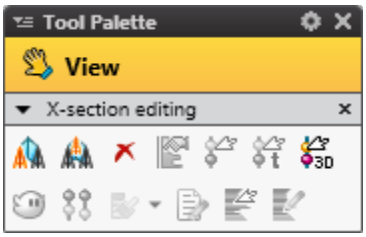


Figure 81: Tool Palette in a Well Section Window

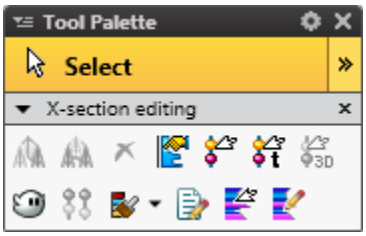
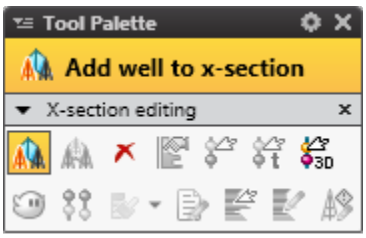


Figure 82: Tool Palette in a 3D Window



The window toolbar can also be added to the Tool Palette by clicking the **Settings** icon on the **Tool Palette.**

Well Section Tab

The **Well Section** tab contains tools that allow you to configure the settings for cross sections, well section templates, and well section windows. The Well section tab is available only when you have an active Well section window.

Figure 83: Well Section Tab

Geology: Structural Framework

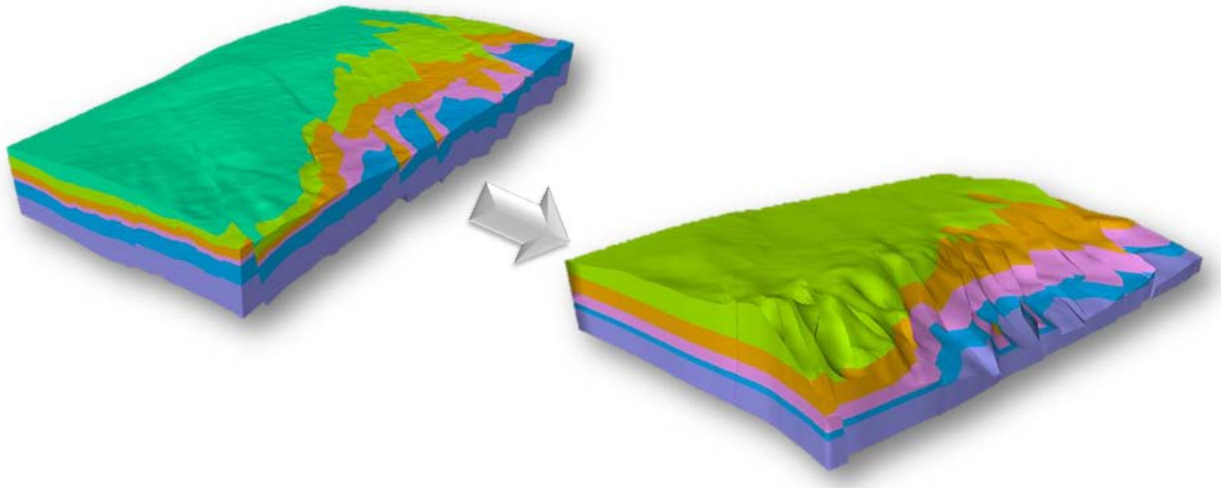
2014.1

Depospace construction

The objective of Depospace is the construction of a virtual and physically consistent depositional space controlled by the chronostratigraphic environment at the time of deposition of geological terrains as they are described in the geometry of the associated structural model (VBM). This computed depositional space can be used for property modeling (in case of depositional properties, such as: facies, porosity, permeability, etc.) targeting more accurate calculations, proportions, distributions, and further volumetrics.

Depospace is the representation of a Structural Model in an "Unfaulted and Unfolded" state. It is equivalent to a Pseudo-3D-Chronostratigraphic view of the model (Pseudo as the vertical axis does not represent absolute age). The construction of the latter will also provide some transformation to bring any point from the "present day" 3D representation (Geospace) to its corresponding location in Depospace (and vice-versa). This will be used subsequently by other processes. In Petrel 2014, property population within Structural Grids can be done using the Depospace transforms (see Geology – Modeling section). The construction of the Depospace relies on a geo-mechanical engine that will ensure proper reconstruction to a flat-space. The usage of geo-mechanics is quite important, as it will honor some contact and mass preservation constraints, making sure the reconstructed model in its 3D – Pseudo-Chronostratigraphic space has a consistent shape. Distances between points in a given stratigraphic layer will, for example, be coherent with how they were initially deposited: a property that is quite important for later use with data analysis and variogram construction.

Figure 84: The upper left part characterizes the 3D Structural Model in its present day geometry, whereas the bottom right corresponds to its Depospace representation (Modified from Geosciences Australia Datasets)



Since no prior knowledge of "geo-mechanicals" concepts is needed in order to build the Depospace, user interactions are quite limited. A quick QC from the eye of a geologist will be sufficient to validate that the process concluded successfully. You simply have to specify the pieces of the model to be included within the Depospace calculation protocol.

New tree for structural framework node (behavioral change)

The new tree structure creates a split between the fault framework and the structural model. The fault framework can now be commonly used by several structural models.

A new hierarchy has been introduced to help better structure the workflow, as well as enhance productivity. The steps for building the structural model are now, as follows:

1. Define the "Structural Folder" and its domain (time or depth)
2. Build the Fault Framework
3. Define the VOI (Volume of Interest) for the Structural model (can be only a portion of the fault framework)
4. Build the Structural Model via *Horizon Modeling*
5. Finally, build the Depospace

You will then be able to QC the faults initially built during the Framework generation process against the ones outputted from the horizon modeling stage, where they have been re-meshed as part of the volume building process.

Retirement of Surface Based Algorithms for Structural Model building (behavioral change)

Previous means of building Structural Models using "Fault Center Grid Dislocation (FGD)" and "Multi-Patch Interpolator (MPI)" are being retired and will no longer be exposed.

Volume Based Structural Models are the only ones that you can create from within Petrel going forward. Previously built Structural Models using the legacy algorithms will be upgraded and can still be used further within the grid construction workflow, but you will not be able to regenerate them in case, for example, new interpretation has been made. Conversion to a Volume Based model will then be needed in order to update it.

Geology: Modeling

2014.4

Depospace enhancements

Various enhancements have been implemented for **stair-stepped grids** with **Depospace**:

- Additional options are now available for the target space definition when creating property in Property Calculator and in Operations on several properties when using both Geospace and Depospace properties.
- The method "Absolute or relative depth" in Geometrical modeling dialog now correctly handles Geospace and Depospace.
- Statistics for reporting Simbox cells are now correct.
- Local model update for properties is now enabled.
- Interactive facies editing is now enabled: in Depospace for [D] properties and in Geospace for non-[D] properties.

Use of Free memory on Grid Properties

The use of **Free memory** is now preserving the integrity of the **Property filter**. In previous Petrel versions, using Free memory (in the Workflow editor, the Uncertainty and Optimization or at the project level) would corrupt, momentarily, the Property filter and could produce unexpected results.

Use of non-zero based discrete color table for Property modeling

When the first code is strictly positive in a discrete color table, it now behaves as when using a zero code: it correctly enables the use of the Property filter and interactive facies modeling; it also correctly displays and supports the picked value for stair-stepped grids. Negative values are no longer allowed in the code field.

Stair-stepped grid enhancements

Various enhancements have been implemented for **stair-stepped grids**:

- **Custom property modeling algorithms** were not correctly implemented in the shadow zone of reverse faults for stair-stepped grids. This is now fixed. No change is needed in the Ocean plugin code. Existing algorithms will now work correctly on stair-stepped grids.
- In **Simbox** mode, the picked cell volume is now correctly reported in the **Status bar** for all **stair-stepped grids**.
- The option to **Flip property values** in the I, J or K direction is now correctly disabled for all stair-stepped grids.

Output part of a 3D grid for a single subzone

When using the **Output** tab of a 3D grid to extract a subzone of the grid, the properties and contacts settings are now correctly preserved for the subzone, as well as the name of the subzone.

Surface default color when running Make/Edit Surface dialog

When running the **Make/Edit Surface** dialog on an existing surface, the color shown on the **Info** tab of the surface is now assigned the specified color of the Main input object.

Fluvial channel object modeling

Since Petrel 2014.2, re-running a **legacy property**, using **Fluvial channel** objects in the **Object modeling** method (Facies modeling dialog), provides a different stochastic result with an identical seed due to a fix that modified the pseudo-randomness of the algorithm.

LGR filter

The LGR filter in **Property calculator** is now enabled. In Petrel 2014.3, the LGR filter was not properly applied in the Property calculator.

Well index method in Geometrical modeling dialog

The **Well index template** is now correctly assigned when using the **Well index method** with the option "Use visible wells only".

Property statistics operation in Workflow editor

The Workflow editor operation **Property statistics to output sheet** now has a new "Weighting" toggle available in the command line to specify whether the statistics report should use a weighting: length-weighting for discrete property or volume-weighting for continuous property. When toggled off, no weighting is used.

2014.2

Use of Depospace in Facies and Petrophysical modeling

The depositional space, or Depospace, represents the geological model at the time of deposition. This is the most accurate space to use for modeling the facies and properties that are syn-depositional (shape and position of objects, facies fractions or the properties variograms). This is the simbox that is used if the Use Depospace option has been selected during the Structural gridding.

By ticking the option 'Use the depospace for property modeling' in the structural gridding process, the resulting grid receives [IJK] and [D] tags in its name:

- [IJK] indicates that the grid has been built using the structural gridding process

- [D] indicates that it is using the depospace simbox as a simbox. (There is an internal depositional space mapping, or transform, that links the two spaces – Geospace and Depospace).

  Subsequently, any property computed with the Facies, Petrophysical, and Trend modeling dialogs will also be [D] tagged.

Properties created using geometrical modeling, or the property calculator, will NOT get the [D] tag because they are not computed in the depospace simbox, but in the real grid (representing the Geospace, or present day space). However, properties made in Depospace can be updated using the Calculator and still maintain the correct [D] tag.

The mapping between the geospace and the Depospace is not bijective. Single cells in geospace can be mapped to several cells of the depospace simbox. Therefore the numbers of cells in the Geospace and Depospace are not identical. These numbers are reported in the **Statistics** tab of the Properties folder.

Grid properties smart folders

You can create different instances of a hierarchical tree to get a quick and easy access to individual or groups of grid properties by using specified sorting attributes.

The **Models** pane typically contains a tree structure as the project is populated with an increasing number of grid properties. Grid properties are continuously generated in the **Models** pane during the entire life of a project. The **Favorites** pane allows displaying the same inventory of grid properties contained in the project sorted in a user-preferred hierarchy through the use of grid properties smart folders.

This tree structure is dynamic and responds to your selected preference of attributes and their sorting order.
In the example below, the same properties are now listed in both the **Models** pane and the **Favorites** pane. In the **Favorites** pane, the user listed the properties per grid, per last action performed, per property type and per template.

Figure 85: Favorites pane showing an instance of the Grid properties smart folders hierarchical tree.

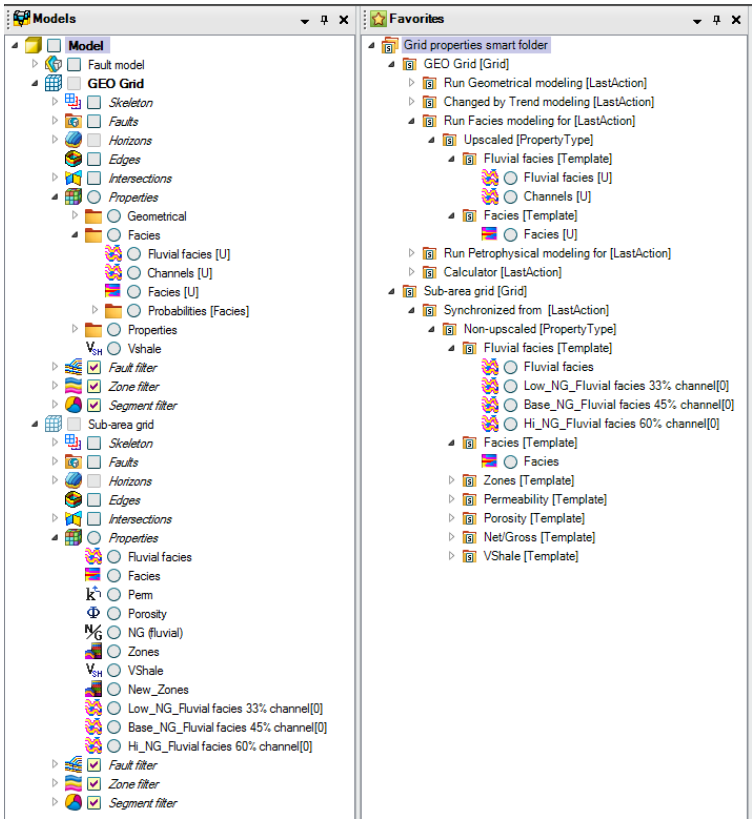
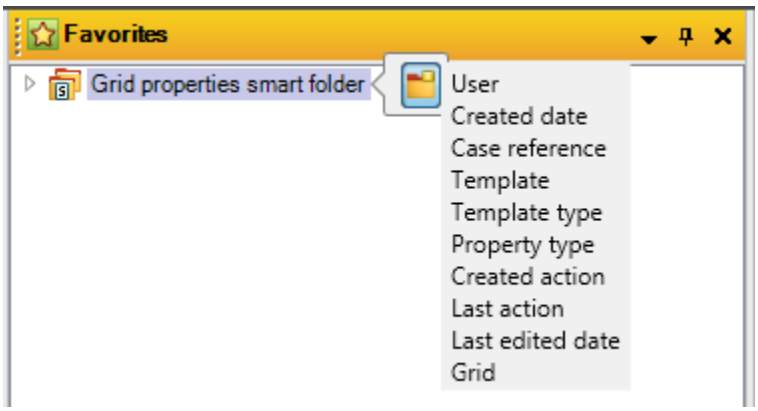


Figure 86: Attributes selector on Grid properties smart folder



2014.1

Quality Assurance Maps

Quality Assurance (QA) maps are attribute maps computed from 3D grid properties and the 3D grid.

Various attribute maps are often used to help assess the quality of reservoir models, as well as the horizontal and vertical continuity of the reservoir. Examples of QA maps include Isochore (TVT), net thickness, net reservoir, pay proportion, permeability thickness, hydrocarbon thickness, average porosity, etc.

The QA maps generated in Petrel are dynamic, which means that:

- They can be automatically updated when their parent property model is changed.
- A QA map displayed in an active window will be recalculated and updated immediately upon selection of single or combined zones.

In order to Quality Control (QC) the modeled property against the input well data, you can also display the equivalent attribute values calculated from the upscaled cells and well logs used in upscaling the QA maps. At each well location, the following attribute values will be available for display:

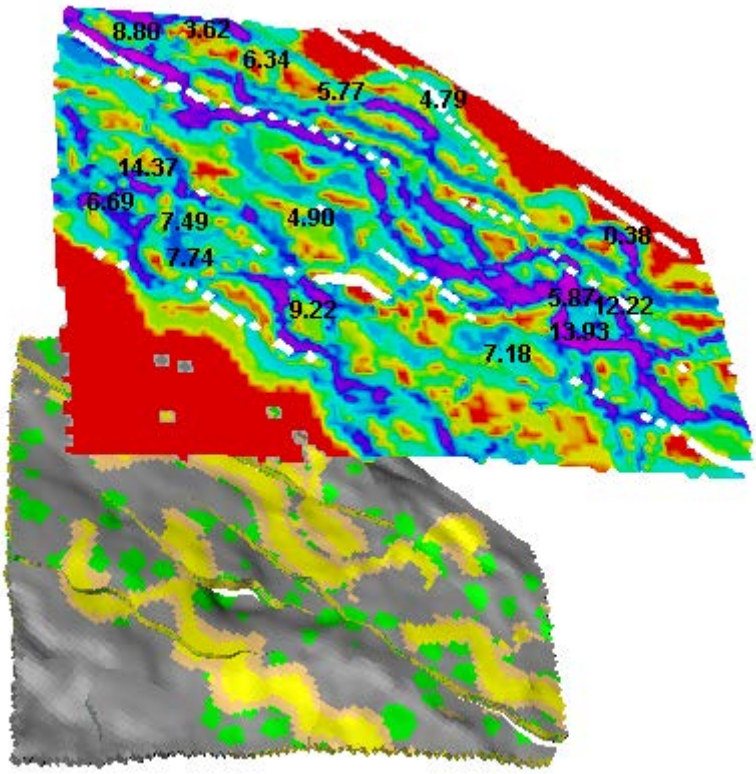
- The attribute map value extracted at the XY locations of the intersection of the input wells with the Top horizon of the shallowest selected zone currently displayed.
- The equivalent attribute value computed on the upscaled cells along each input well.
- The equivalent attribute value computed on the input well log data used in upscaling.

Depending on whether the property is discrete (for example, facies) or continuous (for example, porosity) the attribute map can be one of two types: thickness or average.

Depending on the settings selected in the QA map dialog, QA maps will be one of the following:

- For discrete properties-
  - Thickness/Proportion: Vertical thickness or proportion maps for the different facies/discrete codes of a discrete property (for example, net sand/reservoir maps or net pay thickness/proportion maps from a discrete reservoir/net flag property).

Figure 87: Fine sands proportion map and Facies model. Proportions computed from well log data in overlay



- For continuous properties-
  - Property thickness: A surface representing the sum of the property multiplied by the height at each point. With volume proportion properties (L3/L3), such as Net to Gross, this represents the effective thickness of the unit (for example, porosity/permeability thickness maps).
  - Average: A surface with the average value of the property at each XY location (for example, average porosity maps).
  - Volume height: A surface representing the integral of a volume property with respect to elevation divided by the cell bulk volume at each XY location. When performed on volume properties (HCPV, STOIIP, etc.) the resulting map represents the volume per unit area in the chosen zone at that point or, to put it another way, the fraction of the height of each column occupied by fluids (for example, Hydrocarbon thickness maps). HCPV volume height maps give a powerful visual rendering of the evolution of the height of the hydrocarbon column across the field as if the rock matrix had been removed.

It is possible to perform automatic recomputation of QA maps and "on the fly" visualization/ computation.

All the property-derived QA maps will be updated when an updated property is RPT transferred or a property is re-run or modified in any way (such as, in the following processes: Upscale well logs, Facies modeling, Petrophysical modeling, Geometrical modeling, Trend modeling, Geometrical trend modeling, Property calculator, and Property operations).

The Automatic recomputation command in the right-click context menu of the QA folder allows you to select or deselect the automatic recomputation of QA attribute maps:

- If Automatic recomputation is deselected (default) and a grid property is changed, a [*] flag is added to all the children QA attribute maps. A QA map can then be recomputed either by pressing the **Run** button in the Attribute computation tab of the QA map settings, or by using the Update map command in the right-click context menu of the QA map.
- If the automatic update is selected and a grid property is changed, all existing children QA maps are automatically recomputed.

If a QA map is generated with the filter option selected, and any setting in the property filter is changed, only the maps generated with a filter will be flagged. If Automatic recomputation is selected and the property filter settings are changed, only the maps with a filter [ON] will be recalculated.

The Zone computation folder allows you to compute and display attribute maps "on the fly" on selected zones:

- By default, when a QA attribute map is displayed in an active window, all the zones in the Zones computation folder are selected and the QA map is displayed for the combination of ALL the zones and sub-zones existing in the grid.
- The QA map displayed in an active window will be recalculated/updated on the fly for the combination of all the zones selected in the Zone computation folder.
- Selecting a parent zone will automatically select all children zones.
- A QA map can be simultaneously displayed for different zones selection on different visualization windows.

Modeling Input Parameters for continuous properties

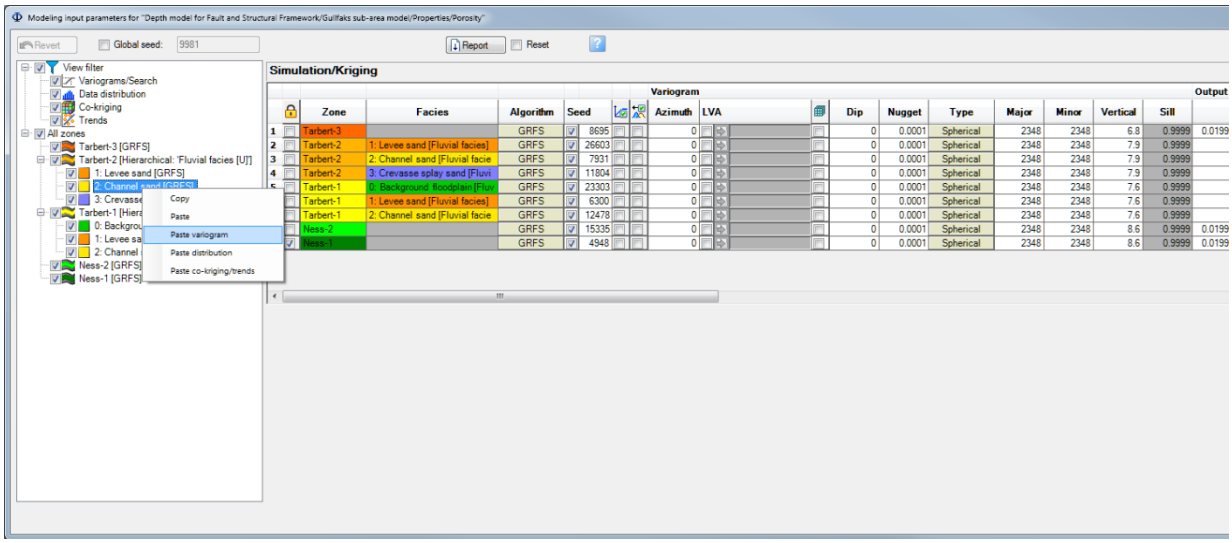
The new Modeling Input Parameters (MIP) editor for petrophysical modeling allows you to view, edit, exchange, and report full sets or sub-sets of Petrophysical parameters between zones inside a flexible unified tabular interface.

Similar to the Facies modeling MIP, the general user interface for the MIP for continuous properties is a standalone dialog divided into a zones tree and a parameters pane. The zones tree is a hierarchical representation of the structure of the property model. The parameters pane displays the modeling parameters of the selected (ticked on) nodes in the tree in a series of tables. The table structure caters to the specifics of PM:

- Hierarchical representation of the structure of the property model in a zones tree
- Parameters pane displaying the modeling parameters of the selected (ticked on) nodes in the tree in a series of tables
- The parameters for all zones or zone/facies using the same type of algorithms are displayed in the same table
- Copy to common clipboard and paste operations allow you to exchange parameters between nodes of the same MIP or of different MIPs
- A report button allows you to create a spreadsheet output of the currently displayed parameters in Petrel's output sheet

The new MIP for petrophysical properties brings a significant usability enhancement for modelers.

Figure 88: The Modeling Input parameters editor for continuous properties

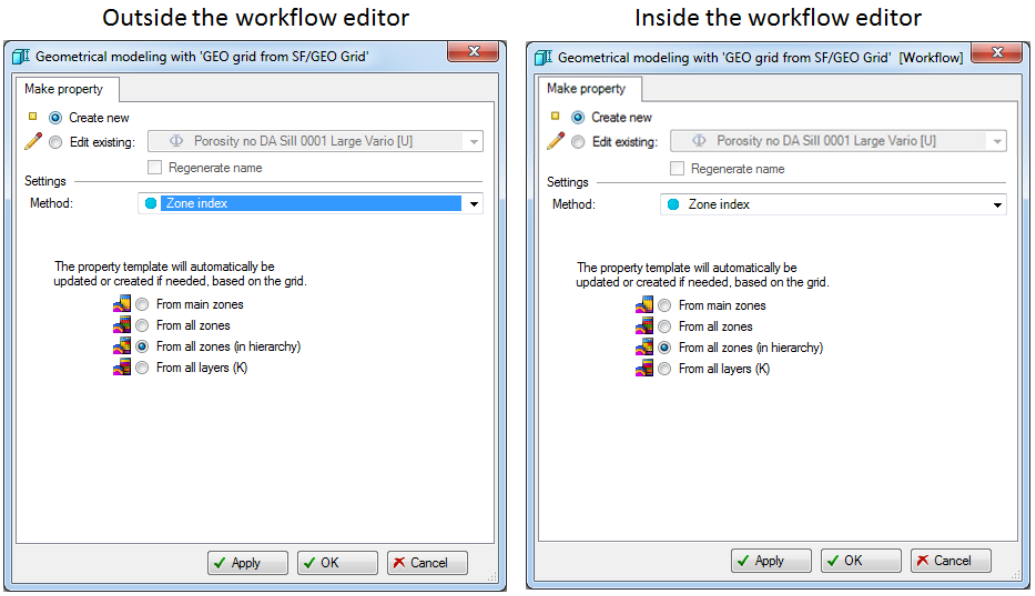


Creation of new properties with Geometrical modeling in a workflow

Prior to Petrel 2014, properties had to be created outside the workflow and edited in the **Workflow editor.** It is now possible to create new properties with the Geometrical modeling dialog in a workflow. Any property created within a workflow is re-usable in other following workflow processes/operations. Using the "Output" variable, it is now possible to retrieve the newly created property after the process is run and use it in further processes/operations if needed.

Enabling this capability for Geometrical modeling is a major improvement in usability and quality control of models run in the **Workflow editor.**

Figure 89: The Geometrical modeling dialog: Creating a new property is now enabled in the Workflow editor



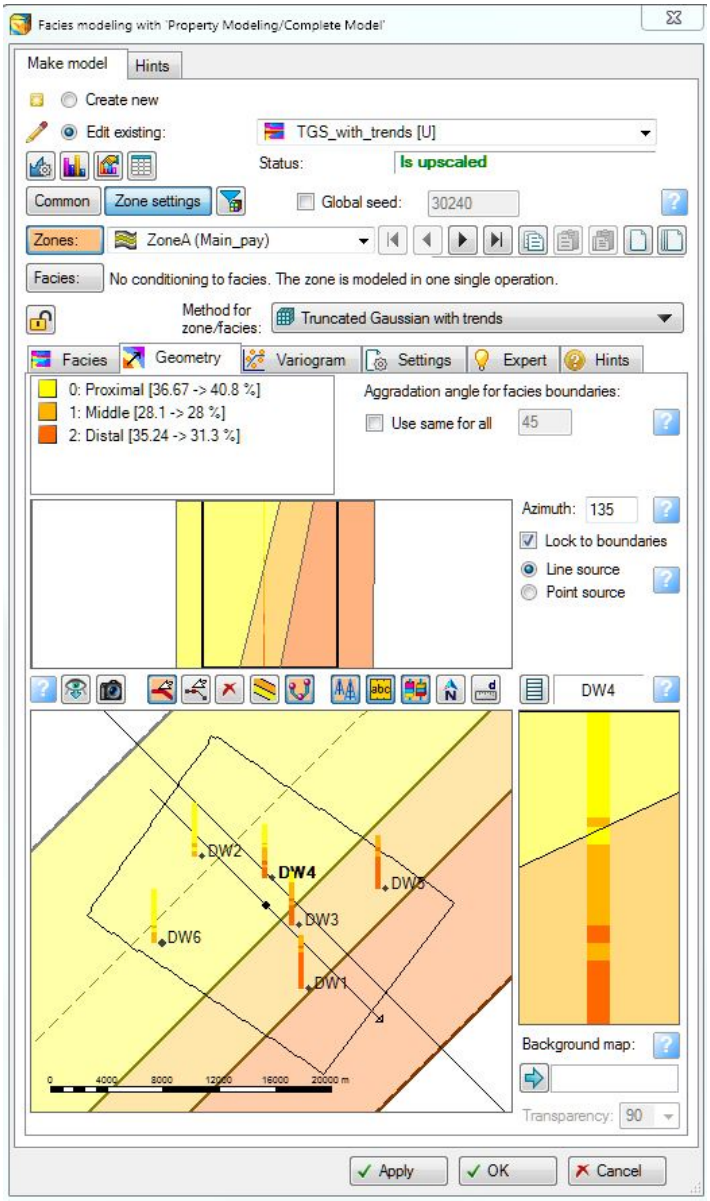
Facies modeling – Truncated Gaussian with trends

The geometry tab of the Truncated Gaussian with trends algorithm in Facies modeling has been enhanced to improve its usability (learnability, efficiency of use, and acceptability) and extend its modeling capabilities for creating and editing the deterministic trend models.

The new, intuitive interface offers several easy-to-use features:

- New cross-section displaying a diagrammatical view of a cross-section of the trend model along the line intersecting the well in the azimuth direction
- New facies table with estimated facies global fractions from the deterministic trend model. These are re-calculated on the fly while editing the trend model.
- Enhanced zooming capabilities in map view allows you to model aggradation angles lower than 1 degree
- The modeling domain of interest is now displayed in the map view
- Ability to rotate the azimuth line independently from the facies boundaries allows you to model oblique facies transitions
- Display user-defined maps (surfaces, images) as background in map view as well as transparency capabilities

Figure 90: Enhanced Geometry tab of Truncated Gaussian with trends algorithm



Enhanced Vertical Proportion Curves functionality

The usability of the Vertical Proportion Curve (VPC) manipulation and use of the Data analysis process has been significantly improved.

The improved interface offers several easy-to-use features:

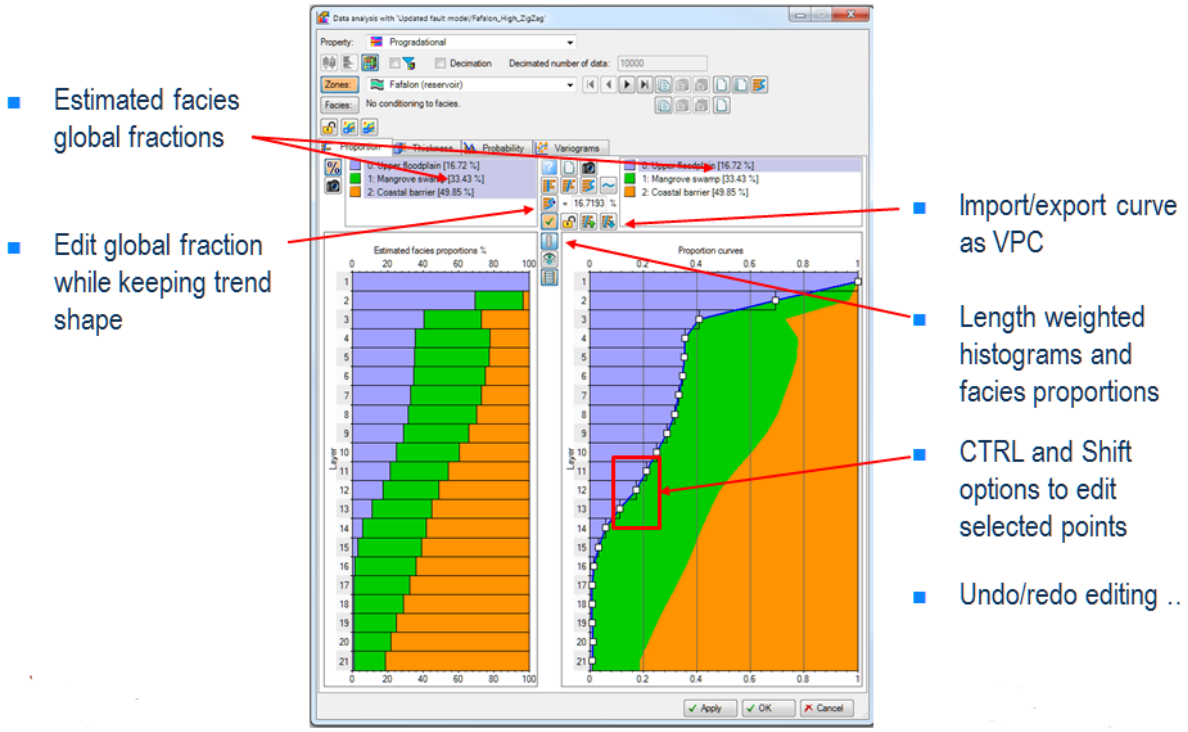
- A new length weighting option is now available for upscaled data and well logs. This allows you to compute the histograms and the input data global fractions using the facies segments lengths as weights
- Display of the global fractions in both facies lists:
  - a fraction for each facies computed from the input data
  - a fraction for each facies computed from the VPC
- Display of the facies fraction in a specific layer for both the estimated facies proportions and the vertical proportion curves. Values are updated on the fly as the cursor moves
- Shift a VPC by typing a target global fraction: the whole VPC is shifted according to the specified target
- Shift all points between two selected points in the VPC to edit them simultaneously
- Select and edit simultaneously contiguous or isolated multiple points
- Import/export selected VPC on selected facies code
- Shortcuts such as, **Ctrl+Z** for undo and **Ctrl+Y** for Undo/Redo, for all operations performed since the most recent save
- Automatically fit all VPC curves to histogram for all zones and all facies at once, and for all hierarchical facies
- Synchronized zooming in/out histograms and VPC graph

Additional improvements related to Data analysis:

- In the Transformations tab, there is an automatic refresh button for the histogram, in addition to the currently existing manual refresh

In combination with the Data analysis scenarios, the improvement of the usability of curves in the Data analysis process will drastically improve the main usability of Modeling.

Figure 91: Enhanced Proportions tab for Data analysis



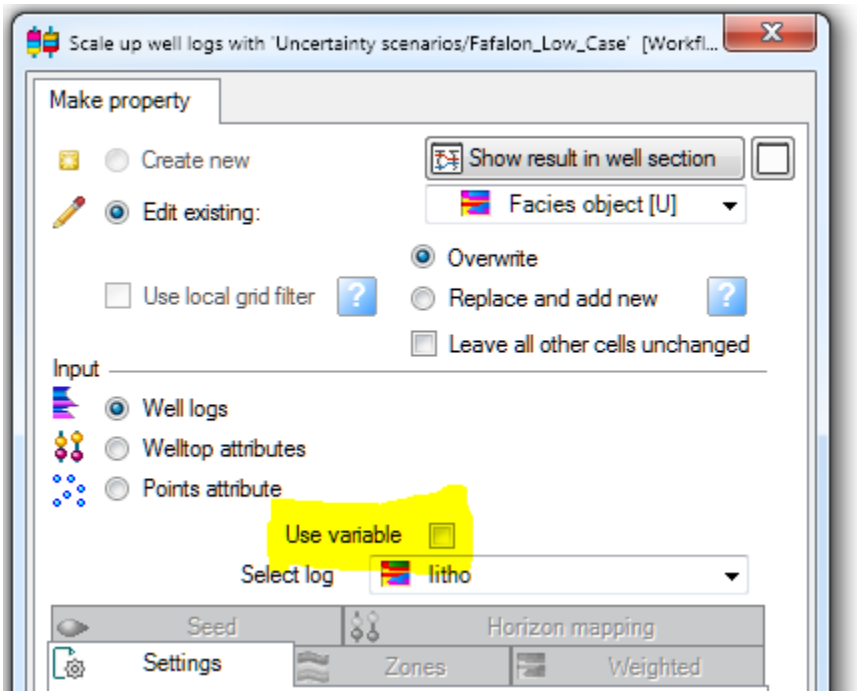
Scale Up well logs variable enablement in Workflow editor

A "Use variable" toggle has been added to the Upscale well logs dialog in the **Workflow editor**. This allows you to use an object variable as global well log and therefore loop through a list of input logs in an uncertainty workflow while preserving the drop-down selection of the process. This is critical to enable petrophysical uncertainty in the **Workflow editor** or Uncertainty & Optimization process.

Also, it is now possible to upscale multiple attributes of the same point set or the same attribute for multiple point sets.

- The settings for the currently selected property type will be used for the object variable
- For point attributes, you can now drop the attribute to be upscaled itself instead of the parent point set
- The variable will represent the attribute and can be used to upscale the same attribute for multiple point sets in a folder loop or multiple attributes in the same point set by looping on the attributes folder

Figure 92: Workflow editor instance of the Upscale well logs dialog with variable enablement



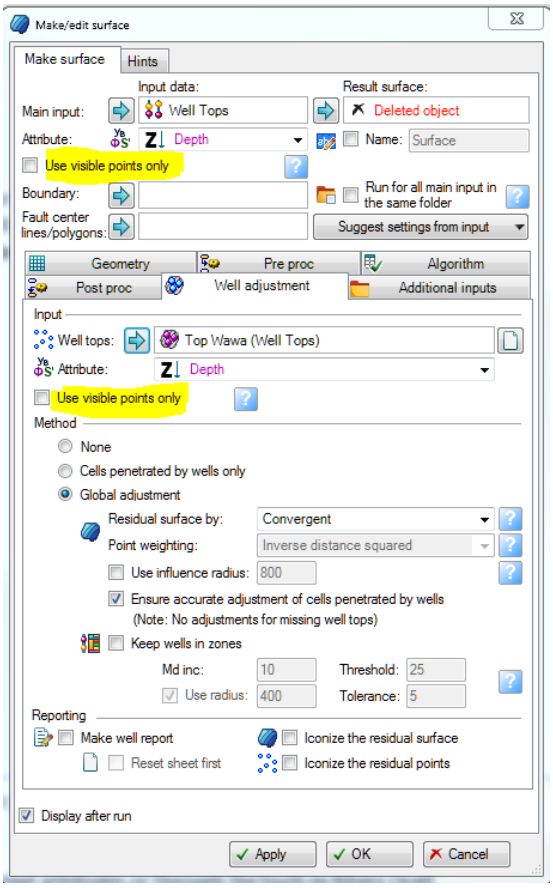
Make surface visible points filter

The new "Use visible points only" toggle in the Make surface dialog allows you to generate maps with a user-defined sub-set of points.

It is now possible to generate maps with a user-defined sub-set of points without having to create new data. A "Use visible points only" toggle has been added in the Make surface dialog for both the main input and the well adjustment. The toggle is only active when data from either point data or well tops is dropped as input or well adjustment control. Otherwise, the "Use visible points only" toggle is inactive or hidden. When ON, only the visible points (for the current input stratigraphic pick or pointset) in the currently active window will be used to create the surface.

Points may be filtered through 1D generic filters on the pointset attributes, the built-in filters (well filter, interpreter, etc.), or the additional filters that can be created on any string, discrete, or Boolean filter in a pointset.

Figure 93: New 'Use visible points only' toggle in the Make/edit surface dialog



Statistics to output sheet command

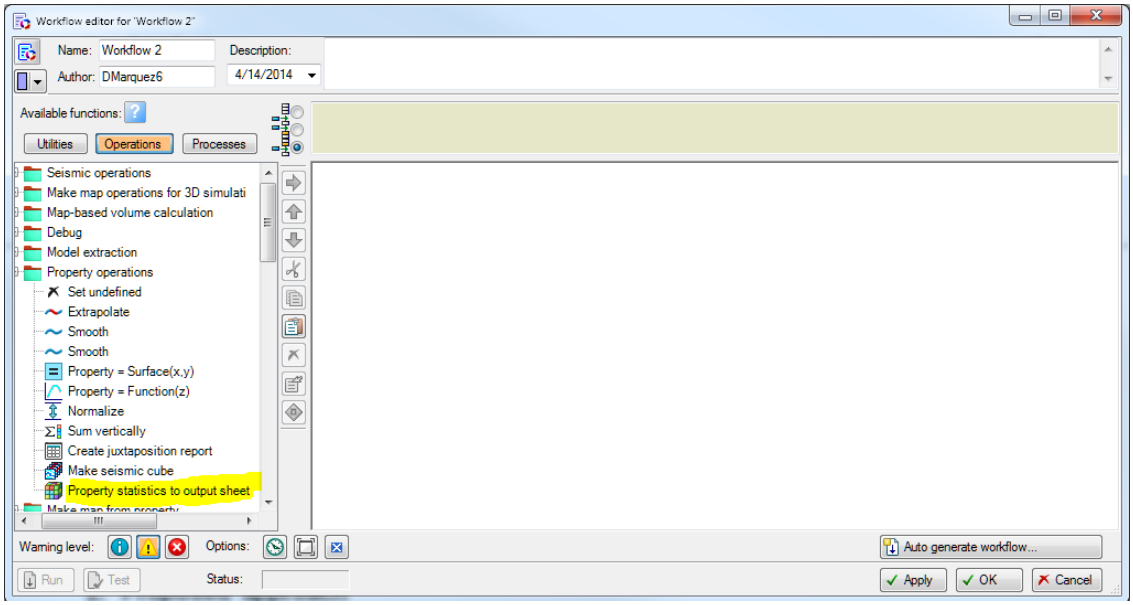
A new **Property statistics to output sheet** command has been added to the **Workflow editor**. This makes it possible to extract filtered statistics inside the **Workflow editor**, which allows you to enable feedback loops on property modeling (e.g. discard realizations if result fraction is lower than a threshold).

The new command is located in the Model extraction folder of the Operations. It has the same options as in the standard statistics command but it also includes:

- A "For zone:" box (object variable enabled) where users can drop a zone from the model Zone filter
- A "Use filter" tick box replicating the behavior of the "Show statistics for the filtered cells only" button in the property settings

The new command processes the content of the Statistics tab when the input property is continuous and of the Disc. Stats tab when the input property is discrete.

Figure 94: New Property statistics to output sheet command in the Workflow editor



Geology: Fractures

2014.1

Estimate fracture density

The fracture density process aims at improving and simplifying the first phases of the fracture modeling workflow. In general, it does the following:

- Makes it easier to properly compute P32 fracture density along wells.
- Properly accounts mechanical zonation while computing the density logs (i.e. mechanical boundaries should correspond to discontinuities in the density log).
- Estimates uncertainties of computed density and discards values which are too uncertain.
- Properly distinguishes between “zero fracture density” and “no measured data” cases along the wells.

The main motivations behind this process are layering preservation, interpreted interval definition, and uncertainty calculation (P10, P90), in that order.

The principle of fracture density calculation stays unchanged compared to the Insert global intensity log usually used. Additionally, the calculation can be performed independently for the “mechanical zones” specified.

Options that will remain from the current “compute global fracture intensity log” are:

- Window length
- Sampling distance

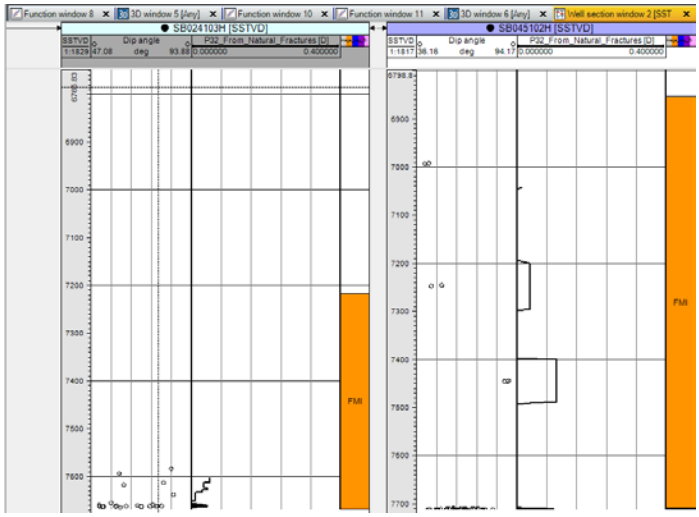
Note that the new process will only allow computing P32 fracture densities (corresponds to option *"Correct for borehole deviation"* turned *on*).

There are options to:

- Discard (i.e., replace by UNDEF values) portions of logs for which the uncertainty (P85-P15) is greater than a given proportion *p* (default: 60%) of the calculated P50 value. See Uncertainty calculation for more information.
- Stop the log calculation at given upper and lower boundaries
- Split the log calculation in between several "mechanical zones"

Accounting for "Interpreted interval"

- Allows (optionally) specifying the interpreted interval(s) using a well log *L* of any kind so that the "interpreted interval" (i.e., portion that has been imaged and interpreted) corresponds to portions of that log *L* for which values are "defined" (i.e., different from UNDEF).
- Within the "interpreted interval", the fracture density values will be computed as described above, outside the "interpreted interval" output intensity values (and associated logs) will be set to "UNDEF".



In orange, log used for defining "interpreted interval".

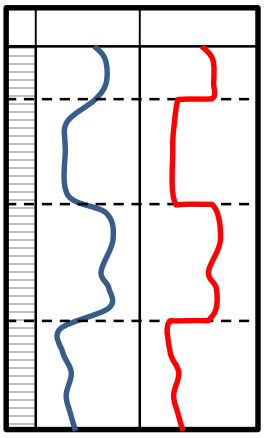
Separation by "mechanical zone"

It is possible to split the well trajectory in several zones prior to the calculation of density logs. Each zone is computed independently from the other zones located above/below.

At zone boundaries, the log should be C0 discontinuous (see the following figure).

Zone boundaries will be specified by the following:

Crossing grid horizons. A mechanical boundary will be inferred from the crossing of the well and of a grid horizon.



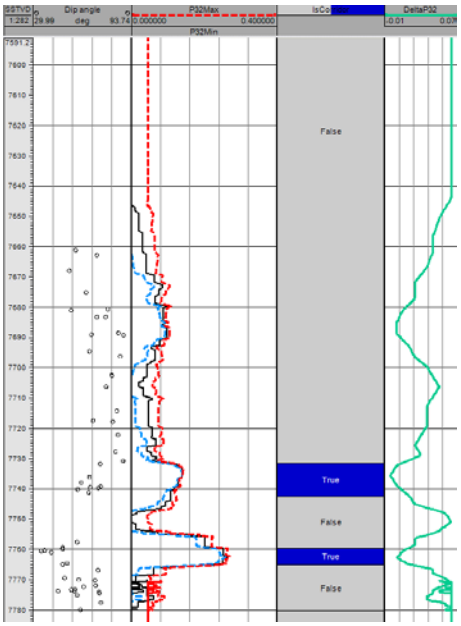
Effect of mechanical boundaries (in dashed lines) on computed fracture density log. Left (blue): intensity log implementation which cannot account for mechanical boundaries. Right (red): curve discontinuous across mechanical boundaries, produced by the process.

Uncertainty calculation

The P32 intensity traditionally computed from interpreted fractures is actually the expected value of a probability distribution, for which characteristics can be recovered from the distribution of fracture samples along the well. Quite intuitively, when the density estimate is based on a small number of samples (i.e. number of fractures within a "window length"), the uncertainty on this estimate is higher.

These values will be computed at each MD of the output intensity log. It is now possible to produce

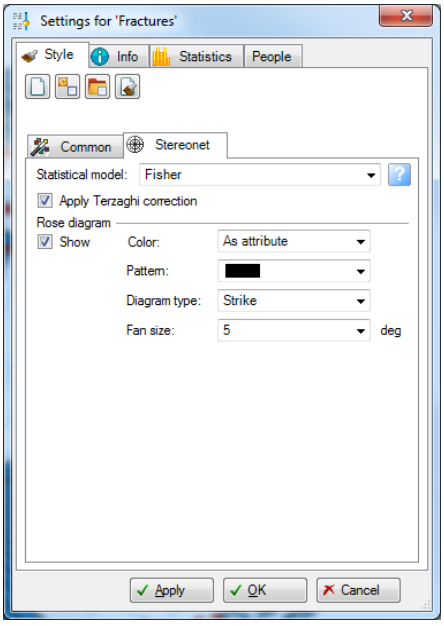
- A "main" P32 fracture density log (i.e. expected value of the P32)
- A "P90 percentile" of P32 density log
- A "P10 percentile" of P32 density log
- A standard-deviation of P32 log

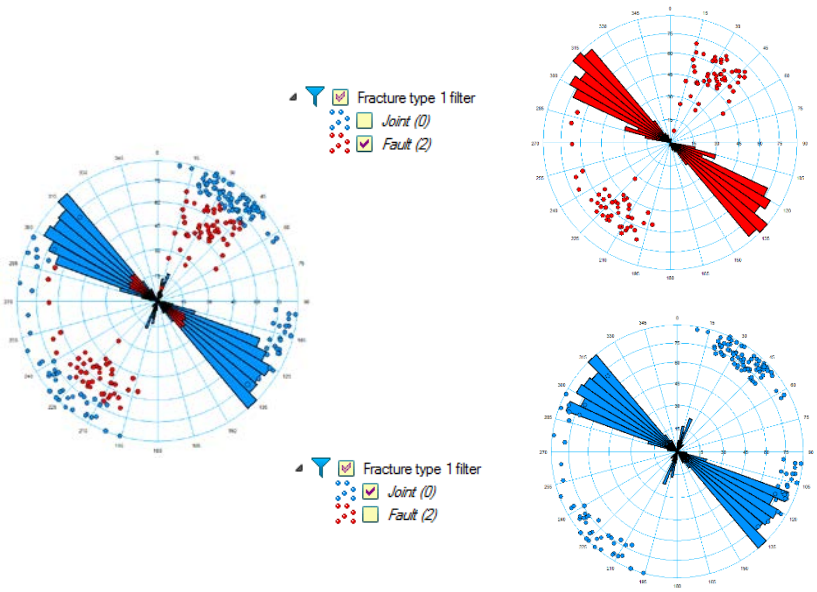


Rose diagram on Stereonet Window

Fracture data analysis and visualization as rose diagrams are currently linked to the Well section window (WSW) because of the nature of the fracture data file (Point well data). The stereonet window is not linked to a WSW and combines the rose diagram and stereonet in a same Fracture data analysis and visualization tool for non WSW.

When you want to make a rose diagram in a Well section window, on the strike of fractures, you have to select dip azimuth and Dip angle, which are attributes taken from the point well data. But there is no way to differentiate in the newly created rose diagram, which fan belongs to which fracture type of the fracture set. This is now possible in the stereonet as you directly take the attribute in the tree. Using filtering for the fracture type, you can display rose diagrams by fracture type. The fan of the rose diagram can be colored by attributes and you can apply the Terzaghi correction (Correct for borehole deviation).





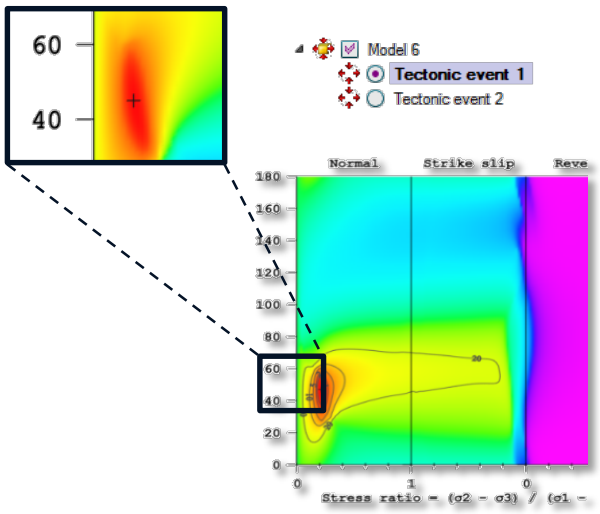
Rose diagram data filtered by fracture type

Rose diagram data versus Computed fractures

Tectonic stress window

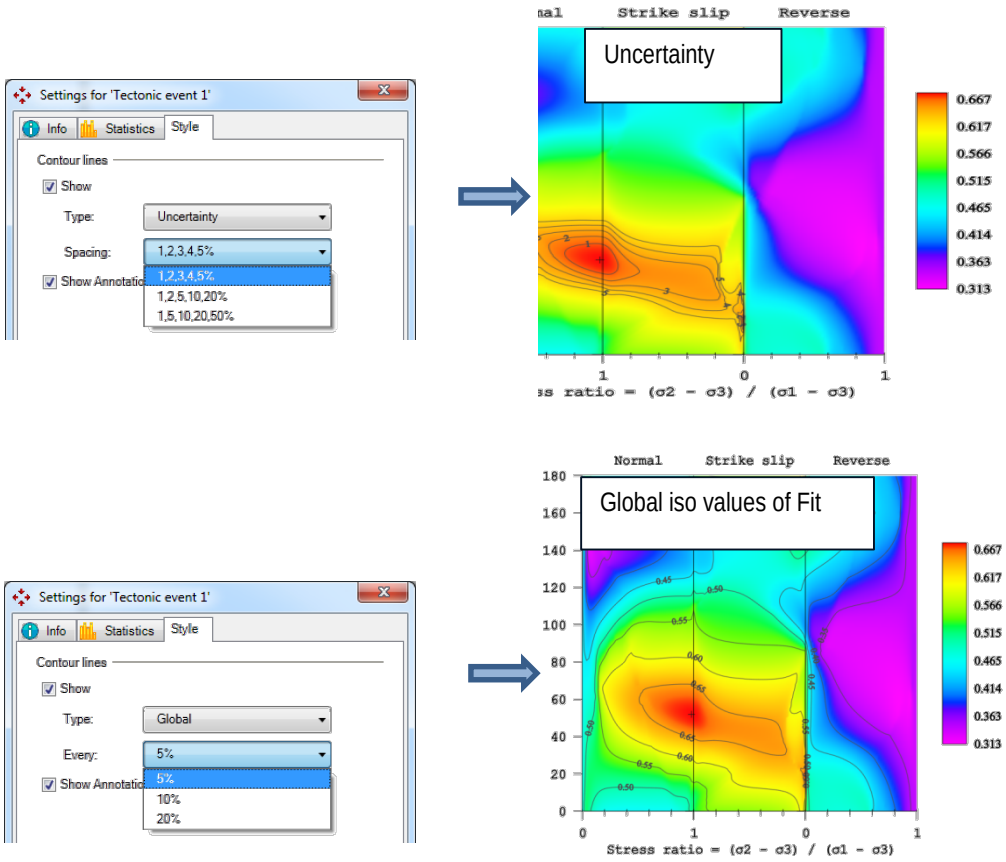
The results of the stress inversion are given numerically by the settings window of the Tectonic event item, on the statistic pane, but also visually with the tectonic stress window. A cross gives the position of the best Fit in the graph. By default, contour lines of the Fit values are visible.

The black cross gives the position of the best Fit.



Iso values contour lines and position of the Best Fit (Cross)

By default, the tectonic stress window is open with the contour lines set to uncertainty interval on the Best Fit to [1,2,5,10,20 %]. The user has the choice between 3 intervals, either for uncertainty or for global fit on the window.



Choice between uncertainty and global iso values of the Fit, with 3 several intervals

Scalability improvements of tectonic inversion

The performance of the forward modeling geomechanical solver (Ibem3D, Iterative Boundary Elements Method) used to compute paleo-tectonic stress field has been improved and in the current version of Natural Fracture Prediction (NFP), the total maximum number of triangles per model is increased from 6000 to approximately 25,000. This implies that a single fault with complex geometry (irregular tipline and curved shape) would not need to be oversimplified in the re-meshing process.

The improvements to the geomechanical solver now make it possible for you to handle models with hundreds of faults without any significant over-simplification of their geometry. The maximum number of triangles that can be used for the re-meshing phase of the fault framework has increased from 6000 to 25,000.

Additionally, optimization of the computations performed during the "Generate fracture driver" process allows the use of 3D grids with higher cell counts.

Discrete Fracture Network (DFN)

DFN upscaling has been improved to avoid memory swapping and crashes on big models. Sigma upscaling calculation has been improved. The properties are now saved in a Folder.

Geology: Wells

2014.2

Saved searches enhancement

An enhancement has been made within the wells saved search dialog box. Two additional icons are available when, within the Search criteria pane, the user selects the filter type: User specified well list.

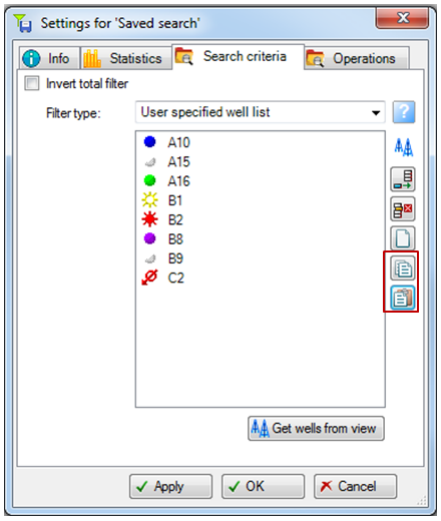
The two new icons enable you to copy and paste well names. The copy icon is used to copy all or selected well-names from the list of wells shown on the User specified well list. These copied wells can be subsequently pasted into other software, such as Notepad, Word, Excel, etc. The second icon is a paste functionality, which is used to paste well-names copied from elsewhere (i.e., not from within Petrel). The paste occurs directly into the saved search dialog box, and onto the User specified well list.

The well names which are copied from outside Petrel software must match alphabets/strings of the wells contained within the Petrel project. The file must be in a single vertical format.

Figure 95: Well names arranged in single vertical order

A10
A15
A16
B1
B2
B8
B9

Figure 96: Copy and paste icons in red border of user well list dialog

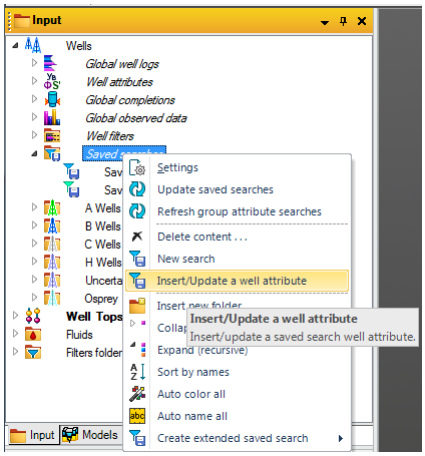


A message log guides the user by indicating if the wells were found in the project. Well names are listed which have been rejected, because the well name was not recognized as an existing well within the project. Also listed are names of wells which already exist within the User specified well list. Those wells which meet the criteria, and do not already exit on the User specified well list, are added to the filer list.

Color by saved searches:

To color a well path by a saved search, right-click the saved searches folder in the tree and select insert/update a well attribute.

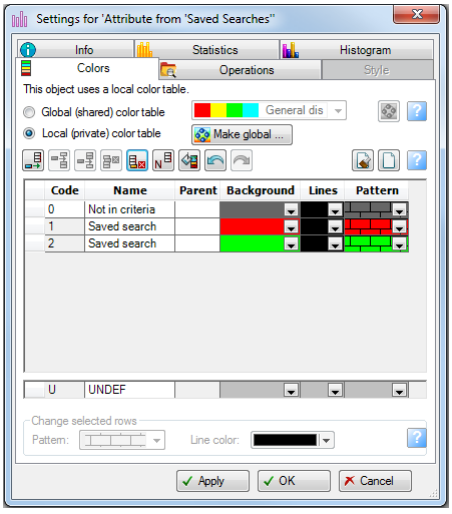
Figure 97: Create/update a well attribute from saved searches folder



In the pop up dialog, a 'yes' assigns color to only active saved search and a 'no' uses all saved searches. Wells not within the search criteria is assigned a 'Not in criteria' code.

This operation creates a discrete attribute for saved searches in the well attributes folder. To alter the colors assigned to each saved search, from the well attributes folder, right-click on 'Attribute from 'Saved Searches'' to open the settings and amend the colors.

Figure 98: Saved searches attribute settings for color selection



In the well manager, the wells are updated with the discrete color code assigned to each saved search containing the well.

The well path can be colored by this attribute to show the different saved searches, in 2D, 3D, Map, and Intersection windows. This is achieved within the parent well folder settings style tab. In the path tab, select show color 'As attribute' and select the Attribute from Saved Searches to visualize the path in the assigned colors.

Note: Only one saved search attribute can exist in the well attributes folder. It gets overwritten when the attribute is updated.

Smart folders for wells

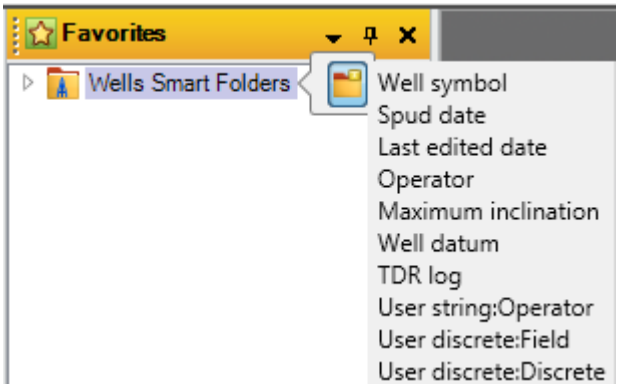
You can now create different instances of a hierarchical tree to give users quick access to individual or group of wells by using a few common sorting attributes.

Wells Smart Folders in the **Favorites** pane show the same inventory of wells in the project, which is sorted in a user-preferred hierarchy. The tree structure is dynamic and responds to your selected preferred attributes and their sorting order.

How to create Wells Smart Folders and group wells in the Favorites pane with selected attributes:

- From the **Home** tab, in the **View** group, click **Panes** and select **Favorites** from the split button. The **Favorites** pane displays in Petrel Explorer.
- Right-click in the **Favorites** pane and select **Insert wells smart folder** from the context menu
- Right-click **Wells Smart Folders** and select **Select attributes**
- Click the attribute selector icon to select the criteria to group the wells

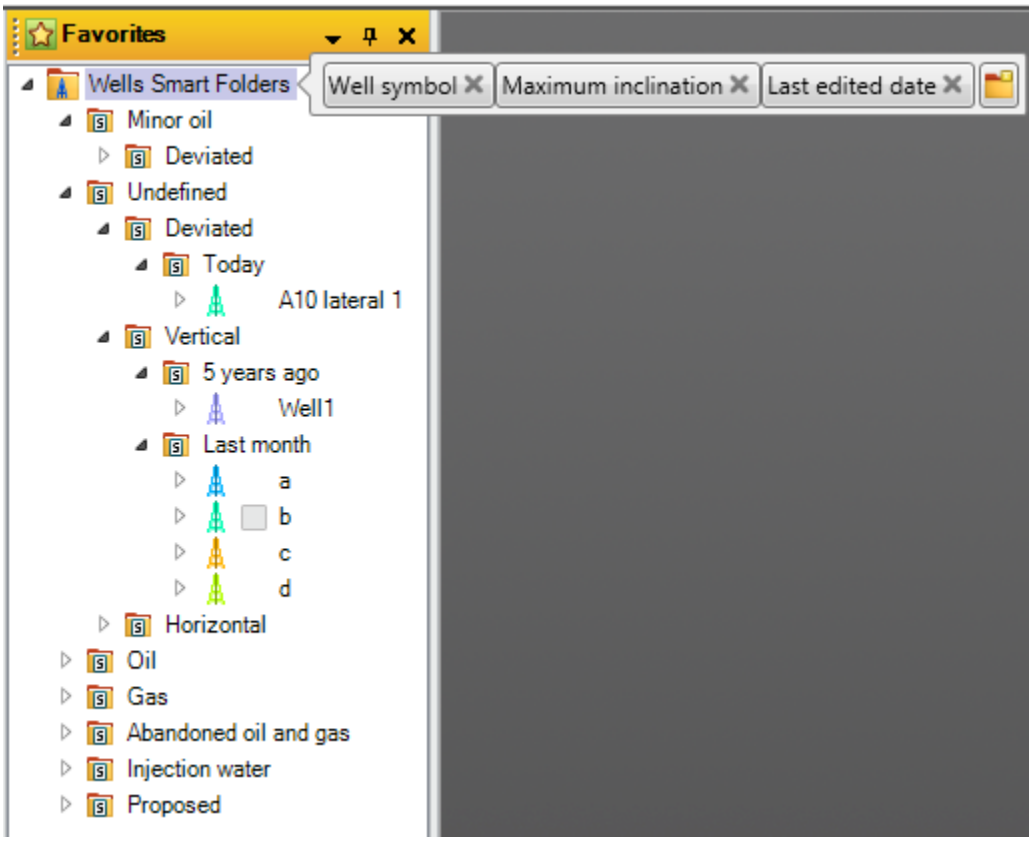
Figure 99: Attributes selector on wells smart folders



- Expand the wells smart folders to see the result.

The wells are listed by the selected attributes (e.g., well symbol, maximum inclination, and last edited)

Figure 100: Favorites pane showing instance of well smart folders hierarchical tree.



- To group wells on more than one attribute, it is possible to select additional criteria by clicking on the attribute selector icon.
    - Attributes can be reordered using drag and drop
    - Attributes can be removed by clicking the X icon in the attribute selector
    - Short cuts can be created from right on smart folder to preserve the hierarchy in the **Favorites** pane

- To sort Wells Smart Folders with data added, edited or deleted in the **Input** pane, requires the user to select load/refresh smart folder from the right-click menu.

Attribute selector descriptions:
Attribute Selector Sorting

The attribute selector panel has both default attributes and user attributes available.

All these criteria can be combined.

Symbol: The 'symbol', as defined in the well settings and well manager can be used to group wells into folders in the **Favorites** pane.

It looks at the well symbol to group and make smart folders for Wells
The symbol folder shows all wells with a symbol 'Oil', 'Gas', 'proposed', abandoned well etc.

Spud date: The 'spud date' for each well, as contained in the well manager, can be used to group wells into folders in the **Favorites** pane.

The spud date format should match the well settings default date. (MMM/dd/yyyy). The date format can vary depending on the region.

Last edited date: The 'last edited date', as defined in the well manager can be used to group wells into folders based on the modified date in the well history tab.

The date format should match the well settings default date (MMM/dd/yyyy) for consistency. The date format can vary depending on the region.

Operator: The 'Operator', as defined in the well manager can be used to group wells into folders based on the string 'operator'.

Well datum: The 'Well datum name' as defined in the well settings page and in the well head position section (e.g., KB, GL, R, etc.) and available in the well manager can be used to groups wells into folders.

The well datum name is used to group wells into smart folder using the string defined in the well settings and well manager.

User defined discrete attribute: A 'user defined discrete attribute' 'as defined in the well attribute tree and available in the well manager can be used to group wells into fields in the well manager.

User defined string attribute: A 'user defined string attribute' as defined in the well attribute tree and available in the well manager can also be used to group wells into folders in the well manager.

Max inclination: 'Maximum inclination can be used as the attribute to group wells into vertical, deviated and horizontal based on the max dip angle of the well path. The well could be classified as vertical with max dip angle range of (0 – 5 degrees), deviated of max dip range of (5 – 80 degrees) and horizontal of max dip angle range of (Over 80 degrees).

TDR log: 'TDR log' criteria groups wells into smart folders based on the type of data used to calculate the active time log on the well.

Limitations:

- Wells Smart Folders and their converted favorites folder equivalent do not have a toggle check-box to select all folder content items for visual display. To toggle on/off the display of all items within a single folder level, multi-select the folder content and press the spacebar.
- Wells Smart Folders and their converted favorites folder equivalent are not recognized in the **Workflow Editor**.
- Wells Smart Folders content objects and their converted short-cut favorites equivalents may not be recognized as input to certain processes.
- Wells Smart folders need to be refreshed when reopening a project by -click the Wells Smart Folders and select Load/refresh smart folders. This was done to improve the start up time for projects with large data.
- New user discrete and string attributes appear with default names in the Smart Folders attribute selector until the project is saved and re-opened (User: Discrete 1 or User: String 1)

2014.1

Multiple trajectories in a well

Surveys and plans folder

Now you can create multiple trajectories in a new model well by inserting a surveys and plans folder.
Insert new surveys and plans and set them to definitive and active in the well. The definitive survey takes precedence over the active plan in the well. Use Inspector to reflect the general status of the point picked on the different trajectories namely: MD, INCL and AZIM, XYZ and Explicit (survey from old model well trace on upgrade to new model well). The input data and interpolated points are displayed in the new trajectory spreadsheet. The multiple trajectories in the new well model participate in RPT and Petrel Studio workflows.

This implementation supports different domain workflows, such as drilling, geosteering, real-time, and reservoir engineering. You can create a surveys and plans folder multiple ways:

- in the Well head importer, select the **Create "surveys and plans" folder for each well** option
- in the **Create new well** dialog, select the **Create surveys and plans folder** option
- for the Wells folder that contains legacy wells, right-click and select **Upgrade to new model well**

Figure 101: Creating Surveys and plans folder using the Well head importer

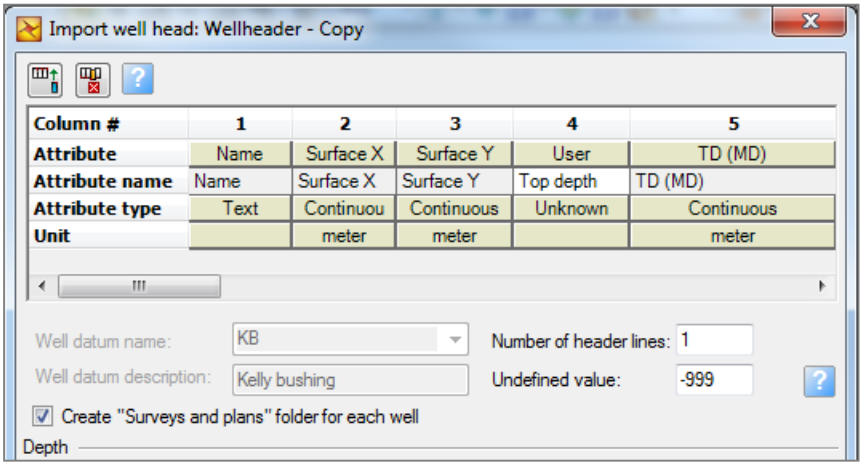
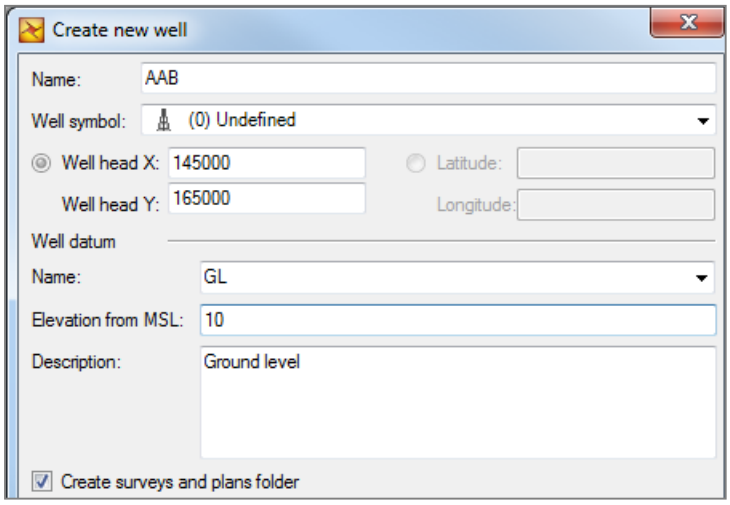


Figure 102: Creating the Surveys and plans folder using the Create new well dialog



The survey type can be in MD, Inclination, Azimuth, X,Y,Z or Explicit. The explicit trajectory is the trace converted on upgrade from old model to new multi-trajectory model well.

Surveys and trajectories can be created multiple ways:

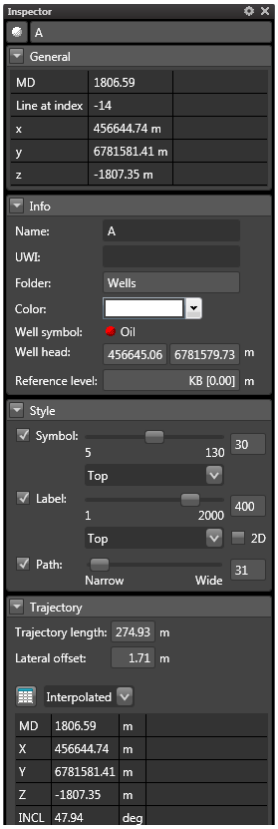
- by inserting a new survey from the **Surveys and plans** folder
- by using the new survey import format, well path/deviation for surveys (ASCII)(*.*)
- by upgrading from the old model to the new model (right-click the Wells folder and select **Upgrade to new well model**)

A survey is not overwritten, but can be appended to the **Surveys and plans** folder.

Inspector for surveys and trajectories

When working with surveys and trajectories the generic Inspector window adapts to reflect the general status of a point picked on the survey. The info gadget allows you to edit survey and well names, color (but not well symbol), well head position, and well datum reference/value. The commonly used well style settings can be modified using the slide bar, for example to adjust the symbol, label, and path. The trajectory gadget allows you to access the trajectory spreadsheet, trajectory length, and lateral offset (distance of point picked from the well head).

Figure 103: Inspector display showing different gadget settings

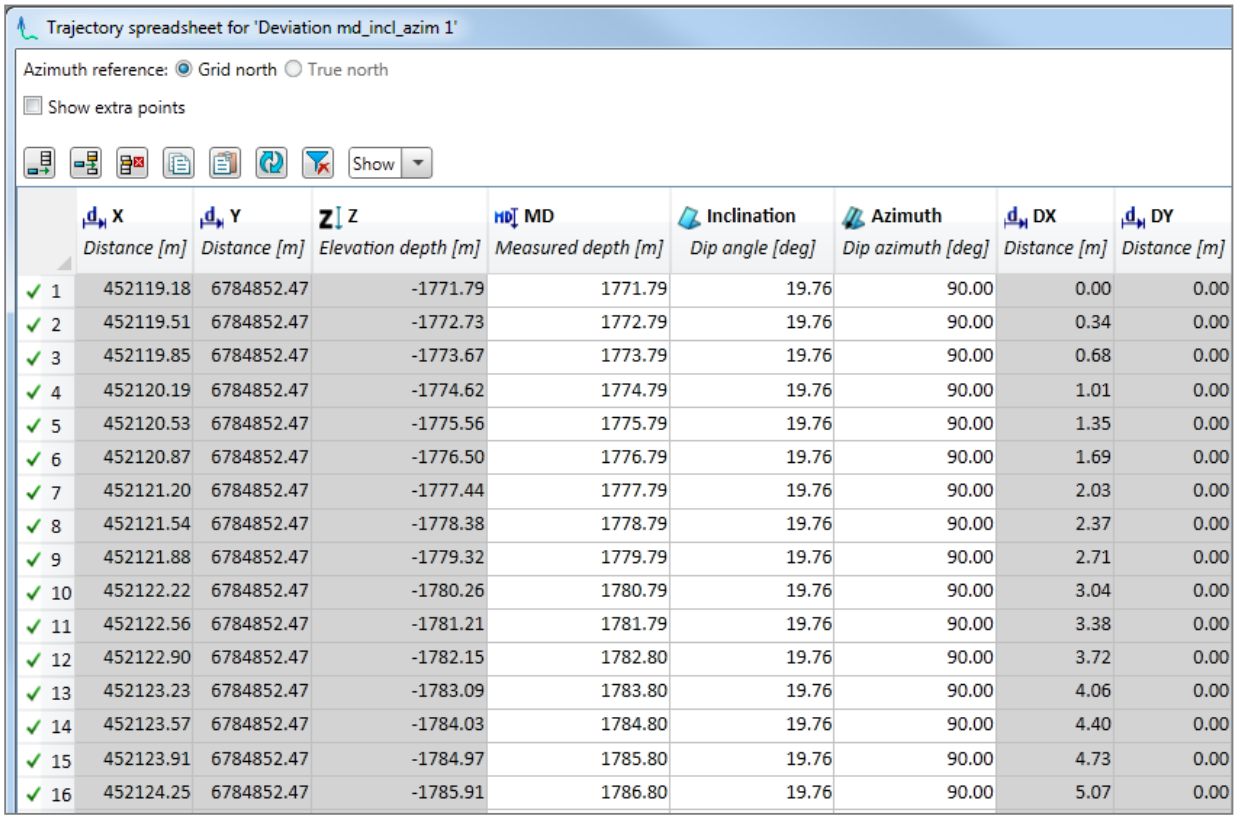


Trajectory spreadsheet

The survey/trajectory input data and calculated results are displayed in the new trajectory spreadsheet. Launch the spreadsheet using one of the following methods:

- from the Inspector, on the Trajectory tab, click the **Spreadsheet** icon
- in the Input pane, right-click a survey and click **Spreadsheet**
- in the display window, right-click a trajectory, and in the mini toolbar, click **Trajectory spreadsheet**

Figure 104: Trajectory spreadsheet showing input (white columns) and computed (gray columns).



	X *Distance [m]*	Y *Distance [m]*	Z *Elevation depth [m]*	MD *Measured depth [m]*	Inclination *Dip angle [deg]*	Azimuth *Dip azimuth [deg]*	DX *Distance [m]*	DY *Distance [m]*
1	452119.18	6784852.47	-1771.79	1771.79	19.76	90.00	0.00	0.00
2	452119.51	6784852.47	-1772.73	1772.79	19.76	90.00	0.34	0.00
3	452119.85	6784852.47	-1773.67	1773.79	19.76	90.00	0.68	0.00
4	452120.19	6784852.47	-1774.62	1774.79	19.76	90.00	1.01	0.00
5	452120.53	6784852.47	-1775.56	1775.79	19.76	90.00	1.35	0.00
6	452120.87	6784852.47	-1776.50	1776.79	19.76	90.00	1.69	0.00
7	452121.20	6784852.47	-1777.44	1777.79	19.76	90.00	2.03	0.00
8	452121.54	6784852.47	-1778.38	1778.79	19.76	90.00	2.37	0.00
9	452121.88	6784852.47	-1779.32	1779.79	19.76	90.00	2.71	0.00
10	452122.22	6784852.47	-1780.26	1780.79	19.76	90.00	3.04	0.00
11	452122.56	6784852.47	-1781.21	1781.79	19.76	90.00	3.38	0.00
12	452122.90	6784852.47	-1782.15	1782.80	19.76	90.00	3.72	0.00
13	452123.23	6784852.47	-1783.09	1783.80	19.76	90.00	4.06	0.00
14	452123.57	6784852.47	-1784.03	1784.80	19.76	90.00	4.40	0.00
15	452123.91	6784852.47	-1784.97	1785.80	19.76	90.00	4.73	0.00
16	452124.25	6784852.47	-1785.91	1786.80	19.76	90.00	5.07	0.00

In the spreadsheet:

- Read-only columns are gray, editable input data columns are white.
- Use the tool buttons to ad, insert, and delete rows, copy and paste, and reset and clear filters.
- In the angle column, an inclination or azimuth angle greater than 360 will be recalculated by subtracting 360 from the value. For example, if you enter 500 by mistake, 360 will be subtracted from 500, and the cell value will be 140 degrees.
- For MD, inclination, and azimuth, the **Grid north** and **True north** options switch the azimuth column header, but values are not recalculated. The **True north** option is only activated if the project is spatially aware.

For other spreadsheet calculation settings, see the online Help topic, "Well trajectory spreadsheet."

Multi-trajectory import and export

When importing the new model well and survey:

- A new multi-trajectory well can only revert to the legacy model by using the RPT workflow.
- A new model well participates in RPT and Petrel Studio data sync across projects. (See the online Help topics for RPT and surveys and plans.)
- There is no option to import an old model well trace on a new well model.
- Log import and deviation loaders from the Well folder create an old model well.
- Survey import in the new well model requires you to select the survey format (MD, inclination, azimuth, or XYZ from the input data setting), but cannot use both during multiple files import. Other survey data types XYTVD and DX, DY, TVD are not available. (See the Survey import settings in the online Help).

- Set survey as 'definitive' if the 'surveys and plans' folder in the well is empty.
- Append new survey, even if the surveys and plans folder is not empty and no survey is definitive or plan is active.

When exporting surveys, you can use the new export format implemented for multiple surveys, and on the well the old export format is still available. From the well, you can export in **well path/deviation (ASCII)** and it will export both the input and interpolated points. From the survey, you can export in **well path/deviation for surveys (ASCII)** and it exports only the input points.

Upgrading the well model

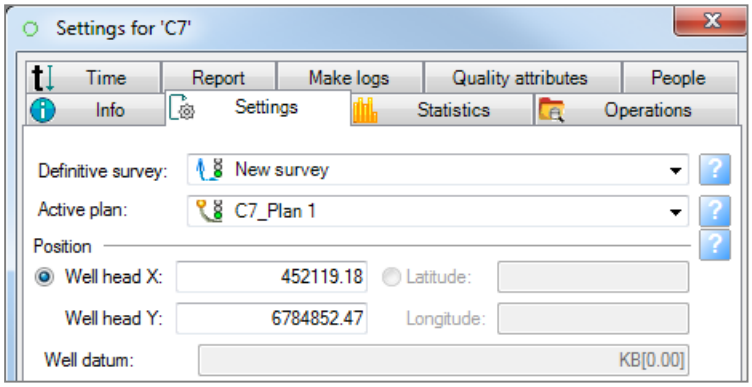
The upgrade command is available from the old well context menu. This creates an explicit survey (read-only) from the old well trace in the surveys and plans folder. The explicit survey is set as definitive and promoted to the well.

If **MD, INCL, AZIM**, or **XYZ** or any other survey format type (for example DX, DY, TVD or XYTVD) was the original input data, it is converted to explicit trajectory. A warning message appears before you click **OK** and commit the changes.

If a well has a surveys and plans folder in a 2012.2 Petrel project, it is treated as a new well model when the project is opened in Petrel 2014.

In the Input pane, the definitive survey is marked with the icon. Changing the definitive survey will automatically move MD loaded data (well logs and tops). For details, see the online Help topic, "Definitive surveys and active plans." The definitive survey and active plan are updated in the new model well settings.

Figure 105: Showing definitive and active plan in the well settings



Note: A definitive survey is read-only in the tree and cannot be deleted. Proposed wells do not participate in the upgrade to a new model well operation.

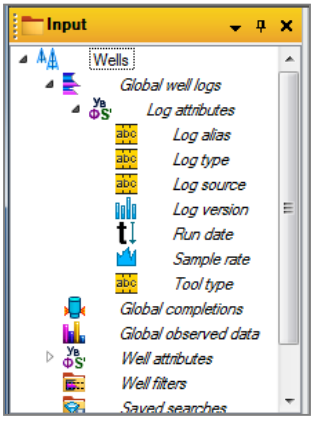
Predefined log attributes

Petrel 2013 introduced user-defined log attributes. Petrel 2014 delivers pre-defined standard attributes to help you properly manage log curves in large well projects. The user-defined attributes (Boolean, continuous, discrete, date/time, string) are still available.

The following predefined log attribute names appear in italics in the Input pane, and their names, colors, and templates cannot be edited or deleted:

- *Log alias* – Short name used to identify a log curve
- *Log source* – Name of company or organization that provides log data or processing for logs
- *Log type* – Log data type as it appears in RECALL (edited, analysis, composite, raw, etc.)
- *Log version* – Discrete number to identify number of times the same log is loaded
- *Run Date* – Date the log was recorded (same as time step attribute)
- *Tool type* – Logging tool used to acquire the log
- *Sample Rate* – Log sample interval steps

Figure 106: Well log attributes in the Input pane



Log attributes can be displayed in the Log attributes spreadsheet, where their values can be modified.

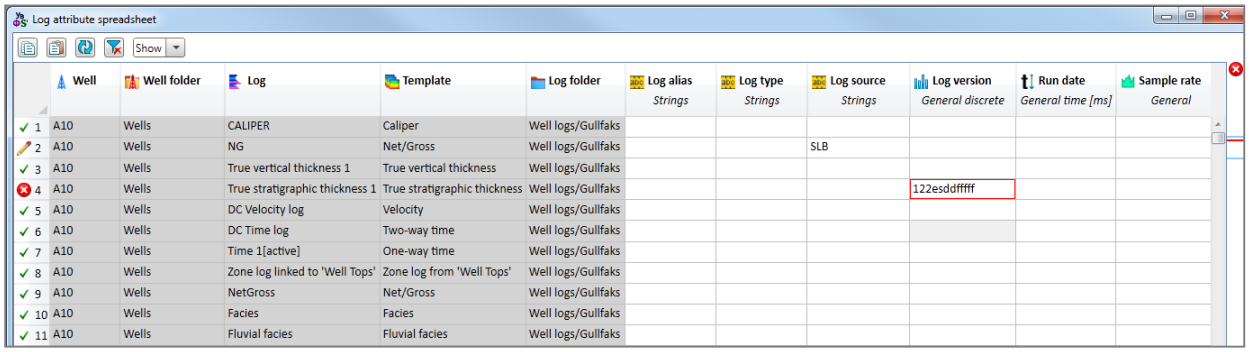
In the tree, the time step attribute is merged with the run date attribute. On upgrade, the time step attribute in prior Petrel releases is merged with the run date attribute in the log attribute spreadsheet.

On the LAS importer, the time step attribute used to create the time series log is replaced with Run date. See the log attributes LAS importer for details. The log attributes are not auto selected. You must check the attributes to be loaded in the log attributes tab of the LAS importer.

You can launch the Log attributes spreadsheet from multiple locations:

- On **Home** tab, in the **Manage data** group, click the **Managers** split button and click **Log attributes**.
- On the **Stratigraphy** tab, in the **Well** group, click the Well data managers split button.
- In the Input pane, right-click the main Wells folder, the *Global well logs* folder, or the *Log attributes* folder, then click **Log attributes**.

Figure 107: Log attributes spreadsheet with row edit tracking indicators



	Well	Well folder	Log	Template	Log folder	Log alias *Strings*	Log type *Strings*	Log source *Strings*	Log version *General discrete*	Run date *General time [ms]*	Sample rate *General*
1	A10	Wells	CALIPER	Caliper	Well logs/Gullfaks						
2	A10	Wells	NG	Net/Gross	Well logs/Gullfaks			SLB			
3	A10	Wells	True vertical thickness 1	True vertical thickness	Well logs/Gullfaks						
4	A10	Wells	True stratigraphic thickness 1	True stratigraphic thickness	Well logs/Gullfaks				122esddfffff		
5	A10	Wells	DC Velocity log	Velocity	Well logs/Gullfaks						
6	A10	Wells	DC Time log	Two-way time	Well logs/Gullfaks						
7	A10	Wells	Time 1[active]	One-way time	Well logs/Gullfaks						
8	A10	Wells	Zone log linked to 'Well Tops'	Zone log from 'Well Tops'	Well logs/Gullfaks						
9	A10	Wells	NetGross	Net/Gross	Well logs/Gullfaks						
10	A10	Wells	Facies	Facies	Well logs/Gullfaks						
11	A10	Wells	Fluvial facies	Fluvial facies	Well logs/Gullfaks						

Spreadsheet tools allow you to copy, paste, reset, and clear filters on the spreadsheet. The Log attributes spreadsheet allows you to copy data from outside Petrel Explorer panes into the white, editable cells, although you cannot use copy and paste to add rows to the spreadsheet. Data in gray cells is fixed (read-only). There is one column per log attribute and one row per global well log.

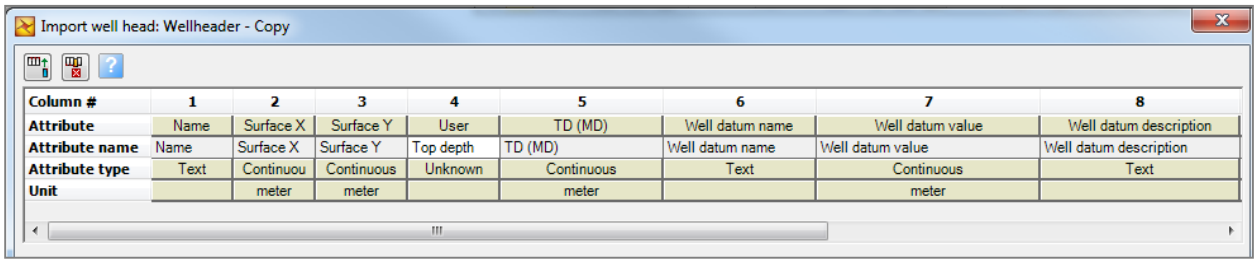
Log attributes and log curves can be synced across projects using RPT and Petrel Studio workflows.

Well reference datum

Well datum reference previously referred to as KB/offset is changed to the well datum value in the well attributes folder. This implementation supports different well datum name and value on well heads import format.

Well heads import format is modified to include three additional columns: well datum name, well datum value, and well datum description, as shown in Figure 108. The well datum name can be KB, GL, RT, DF or user defined. The well datum template is changed from KB to well datum. Well datum value appears in the tree under *Well attributes* folder.

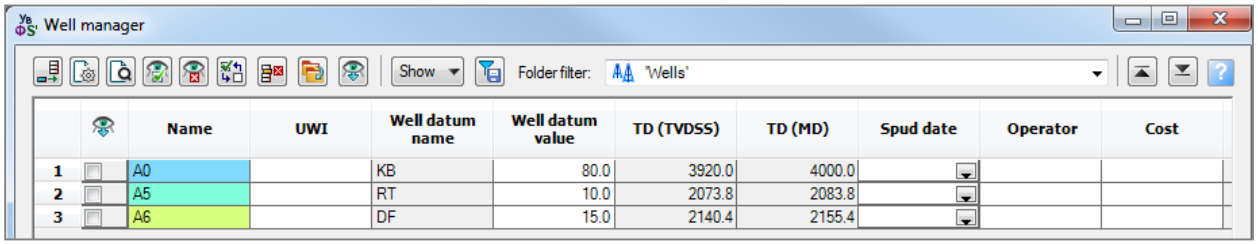
Figure 108: Well head import settings showing the well datum name, value and description



Column #	1	2	3	4	5	6	7	8
Attribute	Name	Surface X	Surface Y	User	TD (MD)	Well datum name	Well datum value	Well datum description
Attribute name	Name	Surface X	Surface Y	Top depth	TD (MD)	Well datum name	Well datum value	Well datum description
Attribute type	Text	Continuou	Continuous	Unknown	Continuous	Text	Continuous	Text
Unit		meter	meter		meter		meter	

In the well manager, the well datum name and value columns are displayed as part of the default attribute. The well datum value can be edited, but the well datum name is read-only. See the online Help for more details.

Figure 109: Well manager showing the well datum name and value



	Name	UWI	Well datum name	Well datum value	TD (TVDSS)	TD (MD)	Spud date	Operator	Cost
1	A0		KB	80.0	3920.0	4000.0			
2	A5		RT	10.0	2073.8	2083.8			
3	A6		DF	15.0	2140.4	2155.4			

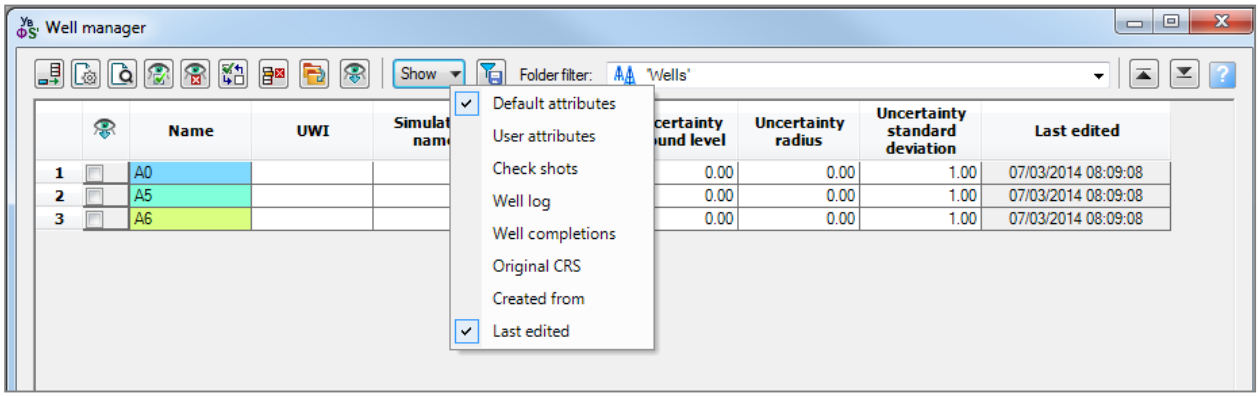
Last edited attribute

In the Well manager, a new **Last edited** attribute appears on the **Show** button pull-down list. The **Last edited** spreadsheet column allows you to display and sort by the time stamp (the last time the well was edited in the Well manager). This **Last edited** attribute is not visible in the well attributes folder.

The time stamp date format corresponds to Petrel project default date in the well settings. The time stamp is read-only and cannot be edited.

Use the **Show** button to hide or display the last edited column. Double-click the **Last edited** column header to sort in ascending or descending order.

Figure 110: Last edited column in the well manager

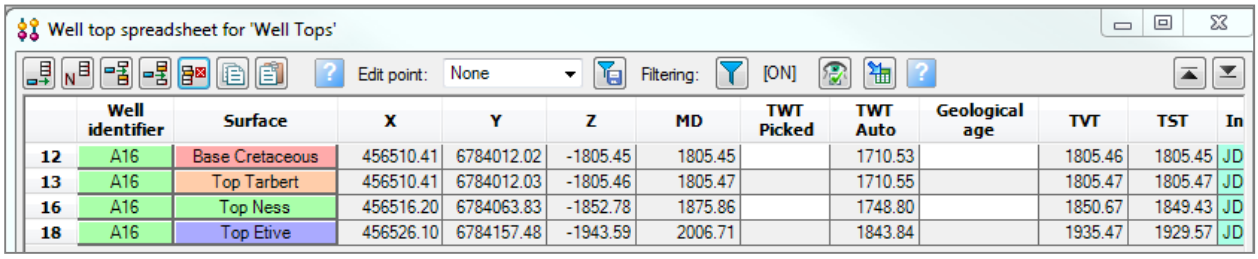


	Name	UWI	Uncertainty radius	Uncertainty standard deviation	Last edited
1	A0		0.00	1.00	07/03/2014 08:09:08
2	A5		0.00	1.00	07/03/2014 08:09:08
3	A6		0.00	1.00	07/03/2014 08:09:08

Locked well tops

Locked well tops are grayed out (not editable) in the Well top spreadsheet. If the Well top spreadsheet is open when the lock operation is performed from the top of the context menu, the Well top spreadsheet is updated interactively.

Figure 111: Locked well tops for Top Tarbert and Top Etive are not editable in the Well top spreadsheet

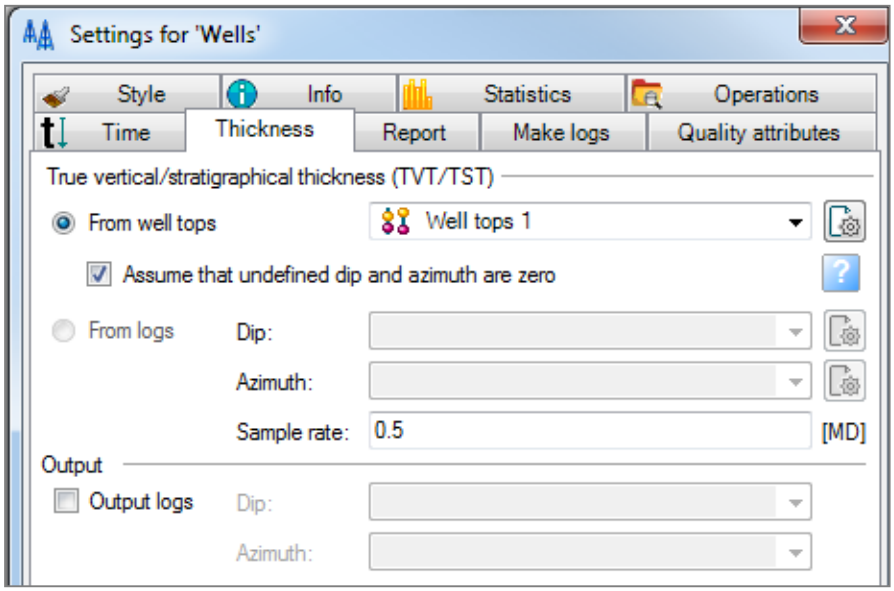


	Well identifier	Surface	X	Y	Z	MD	TWT Picked	TWT Auto	Geological age	TVT	TST	In
12	A16	Base Cretaceous	456510.41	6784012.02	-1805.45	1805.45		1710.53		1805.46	1805.45	JD
13	A16	Top Tarbert	456510.41	6784012.03	-1805.46	1805.47		1710.55		1805.47	1805.47	JD
16	A16	Top Ness	456516.20	6784063.83	-1852.78	1875.86		1748.80		1850.67	1849.43	JD
18	A16	Top Etive	456526.10	6784157.48	-1943.59	2006.71		1843.84		1935.47	1929.57	JD

Well tops thickness calculation

A well's Settings dialog now has an option for well tops thickness calculation. This allows well tops with undefined dip and azimuth values to have dip and azimuth treated as zero when the option is selected. When this checkbox is cleared, undefined dip and azimuth values are ignored in the thickness calculation.

Figure 112: Thickness calculation option in the well Settings dialog



Geology: Structural & Fault Analysis

2014.2

SFA operations on VBM faults

In 2014.1, SFA operations were not available for use on faults generated by VBM. With the 2014.2 release, the full suite of SFA becomes available.

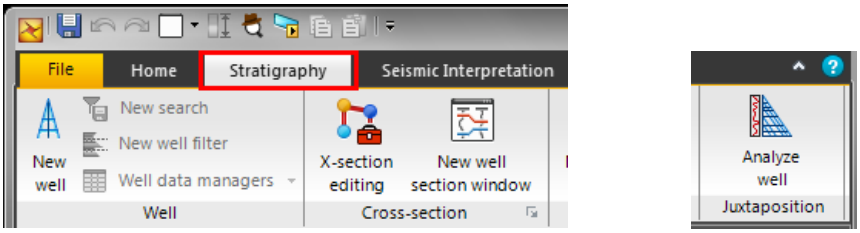
2014.1

Well juxtaposition analysis

The new 1D juxtaposition analysis process allows 'juxtaposition' or 'triangle' diagrams to be generated directly from well data. This is one of the classic methods for de-risking faulted traps in exploration settings.

The well juxtaposition analysis process allows rapid 1D seal analysis of a faulted reservoir, without pre-creating a 3D grid. The process can be used to generate multiple realizations to understand the predicted seal behavior of a fault, with the synthetic grids and faults generated, allowing access to Petrel property calculator and filters, and RDR's existing seal analysis tools.

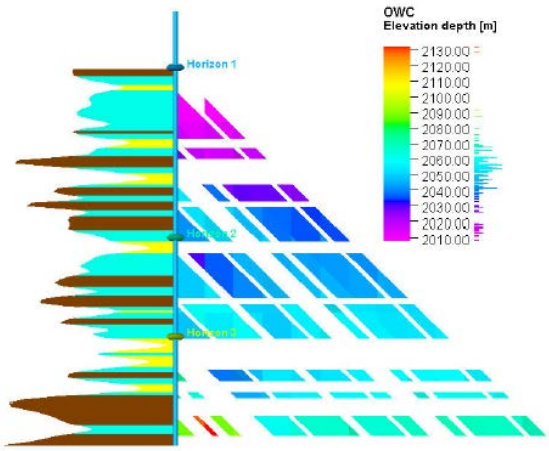
Figure 113: Accessing the well juxtaposition analysis tools



The 1D juxtaposition analysis tools allow for:

- The rapid risking of faults in exploration settings.
- Multiple uncertainty realizations that can be generated quickly to assess the likely seal behavior of a fault.
- Generated synthetic grids and faults that allow access to Petrel property calculators/filters and all the wide array of existing seal analysis tools (e.g., reservoir windows, SGR's, column heights).

Figure 114: Exploration well with associated juxtaposition diagram highlighting potential key fault juxtaposition windows that are likely to leak and seal

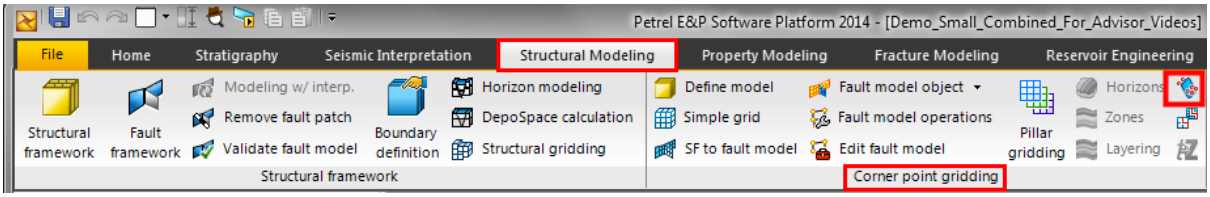


Horizon clean-up process

The horizon clean-up process allows you to quickly clean data. This enables more rapid and accurate mapping and the creation of more robust geomodels.

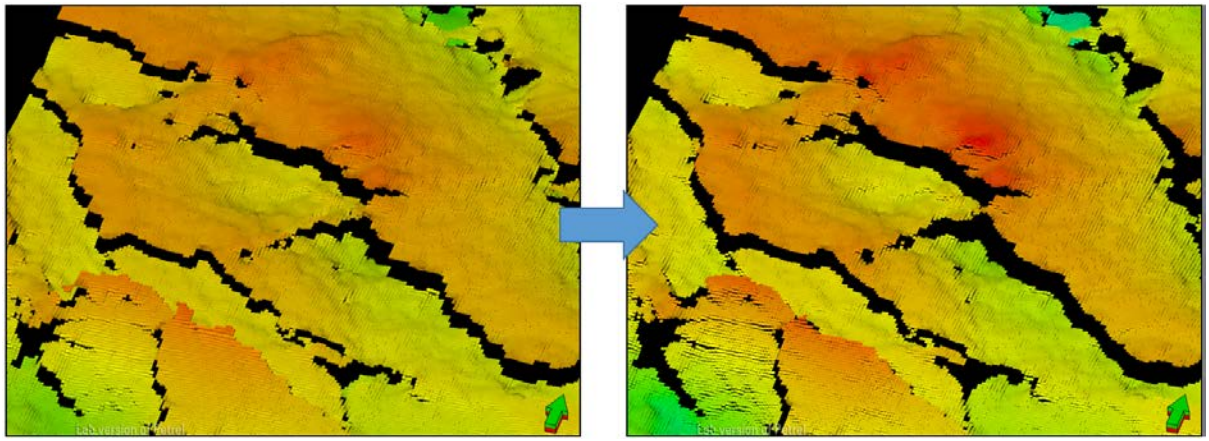
The horizon clean-up tool is a major new process in Petrel. The tool allows you to intelligently and rapidly clean up horizon interpretation data.

Figure 115: Accessing the horizon clean-up process tools



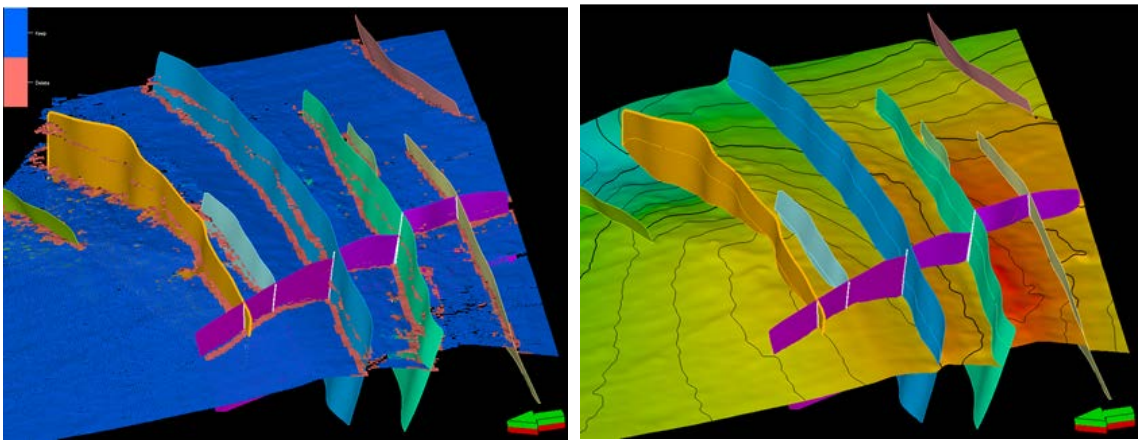
You can perform general cleaning of the horizon or specifically clean around faults with a range of different algorithms.

Figure 116: General cleaning of a seismic horizon prior to any fault picking/modeling



This ability to efficiently clean-up data allows for more rapid and robust geomodels to be built using the pillar gridding/corner point gridding tools and the structural framework tools. The tool should aid geophysicists, geologists, and geomodelers to produce better interpretations and models more rapidly.

Figure 117: Automatically selected areas to delete around faults (left) and the resulting high quality geomodel constrained by the good quality horizon data (right)



Reservoir Engineering

Reservoir Engineering: General

2014.3

Petrel 2014.3 introduces support for INTERSECT 2014.1 through supporting dual porosity and unstructured local grid refinements (the INTERSECT Grid Edits workflow); the ability to use the INTERSECT MR license when running simulation cases; and the addition of options to import networks into Petrel and use these networks in a strategy created using the Field management process to run reservoir-network coupled cases.

You can import ECLIPSE Network Simulator (ENS) network data, generate network coupled INTERSECT cases and visualize the network results in the new network result visualization window. You can also import coupled ENS-INTERSECT simulation cases running outside of Petrel and visualize the network results.

ENS is designed to be used by a reservoir engineer to evaluate pressure losses in both production and injection networks; an ENS model can be thought of as a proxy for a more sophisticated network model, but one which may better suited to full field simulations.

Grid Property Edits

- Support for dual porosity and unstructured local grid refinements (LGRs) with INTERSECT 2014.1.
- Properties exported from Petrel can be edited within INTERSECT using IXF. In this case, transmissibility and pore volume are recalculated from the Petrel pillar grid geometry imported using the GSG file.
- The ECLIPSE Migrator now converts edits in the GRID and EDIT section to equivalent edits in IXF rather than applying them within the Migrator and writing the edited properties into the GSG file. This means it is now possible to modify the edits within IXF without running the Migrator again.

ECLIPSE Network Simulator

The ECLIPSE Network Simulator (ENS) can be used to evaluate pressure losses in both production and injection networks. ENS functions in two modes:

- A standalone mode in which the network can be modeled and tested with various boundary conditions and constraints,
- A coupled mode in which the network can be balanced with the INTERSECT reservoir simulator.

The ENS data file can be imported into Petrel to visualize the network topology, modify network settings and generate reservoir-network coupled simulation cases. The modified ENS network settings can be exported for a standalone ENS run.

Settings dialog (ENS network object)

You can view and alter various properties besides the main network structure in the **Settings** dialog for an ENS object.

To open the **Settings** dialog for an ENS object, either double-click on the ENS object in the **ECLIPSE networks** folder in the **Input** pane, or right-click the ENS object, and then click **Settings**.

In addition to the standard Petrel tabs for **Info** and **Statistics**, the **Settings** dialog contains a **Style** tab (defining Node and Branch colors and Layout) and a **Settings** tab.

The **Settings** tab comprises the following sub-tabs:

Pipes tab

The **Pipes** tab displays information for the ENS pipes. Each pipe consists of a name, a start node and an end node. Each pipe can be associated with a VFP table and if so with an artificial lift quantity (ALQ).

You can filter the table of pipes by the **Branch name**, the **Start node** name and the **End node** name. Click to clear any filter text.

To modify the associated VFP table, either click **Launch search dialog** and choose the required Flow performance table, or select a VFP table in the **Input** pane and insert into the selected row in the pipes table. To disassociate a pipe from a VFP table, click the currently selected VFP table and press the **Delete** key on your keyboard.

Boundary conditions tab

The **Boundary conditions** tab provides information on the network boundary nodes and their conditions. You can filter boundary conditions by the **Node** name. Click to clear any filter text.

The coupling between the boundary nodes and the project flows determined at import time can be modified. Boundary conditions can be one of three types "Productivity index", "Constant Rate" and "Inflow performance relationship". By selecting the appropriate boundary condition type, the relevant table cells are enabled and irrelevant ones are disabled so that you may only make relevant changes. The **Terminal node pressure**, which is the fixed pressure of the terminal sink node in the production network, may also be changed.

Node constraints tab

The **Node constraints** tab is used to view and edit the constraints set on the network nodes. You can filter node constraints by the **Node** name. Click to clear any filter text.

To add a node constraint to the table, click **Add new node constraint**. Click **Launch search dialog** to select the required node and enter the constraint values. A constraint will only export if **Active** is selected. To remove a constraint, select the row and click **Remove selected node constraint**.

Gas & water removal tab

The **Gas & water removal** tab is used to set the gas and water removal rates and fractions on the network nodes. It is only displayed for production networks. You can filter gas or water removal by the **Node** name. Click to clear any filter text.

To add a row to the table, click **Add new gas and water removal definition**. Click **Launch search dialog** to select the required node, and enter the removal values. Removals will only export if **Gas removal active** and/or **Water removal active** are selected. To remove a removal, select the row and click **Remove selected gas and water removal**.

Network tab

The **Network** tab displays an overview of the network in a tree form. Network nodes can be expanded and collapsed by clicking on a node name. The **Title** can be modified, if required.

Network results visualization

Use the **Network results visualization** process to select the network results that you want to display.

Once a coupled simulation is complete, the network results can be superimposed on the network topology in a **Charting window**, as bubbles or pie charts. Use the **Time player** together with network results visualization to identify trends, anomalies, or potential problems during the simulation period.

Import an ENS model

Import an ENS model into Petrel to visualize the network and change its properties. To import an ENS model:

1. On the **Home** tab, in the **Manage data** group, click **Import file**.
2. From the **Files of type** list, select 'ECLIPSE Network Simulator (*.DATA)' and select the file that you want to import.

   The Petrel importer validates the data in the file and tries to match any boundary nodes with existing Tubing or Annular flows in the currently opened project. The **Matching confidence** column in the **Import ECLIPSE Network Simulator model** dialog is green if there is a perfect match in names, or yellow if there is a partial or failed match.

   For a partial/failed match, click to open the **Search for Tubing or Annular Flow** dialog. Select the matching well and click **OK**.
3. Click **OK** to finalize the import.

   If multiple files are imported, the **Import ECLIPSE Network Simulator model** dialog appears for each network.

   The importer finalizes the import by reading in any referenced VFP tables. A **VFP (Hydraulic tables)** folder and an **ECLIPSE networks** folder containing the ENS object are created in the **Input** pane.

   Note: Failing to import a VFP table results in a failed ENS import.

View an ENS network

After importing an ENS data set, view the network structure from the **Settings** dialog.

1. In the **ECLIPSE networks** folder in the **Input** pane, right-click the ENS object, and then click **Settings**.
2. In the **Settings** dialog, click the **Settings** tab and then click the **Network** tab.

   This tab displays an overview of the network in a tree form. Network nodes can be expanded and collapsed by clicking on a node name.

   To change the network name, type the new name into the **Title** text box and click **Apply**.

Associate a network pipe with a VFP table

A network pipe can be associated with a VFP table and artificial lift quantity (ALQ) from the **Settings** dialog.

1. In the **ECLIPSE networks** folder in the **Input** pane, right-click the ENS object, and then click **Settings**.
2. In the **Settings** dialog, click the **Settings** tab and then click the **Pipes** tab.

   The **Pipes** tab displays information for the ENS pipes. Each pipe consists of a name, a start node and an end node.

   You can filter the table of pipes by typing the text that you want to search for into the **Branch name**, **Start node** or the **End node** text boxes. Click to clear any filter text.
3. Select a row in the pipes table and click **Launch search dialog**. Choose the required flow performance table and click **OK**.

   Alternatively, select a VFP table in the **Input** pane and insert into the row in the pipes table.
4. If required, set the artificial lift quantity (ALQ) for the pipe.
5. To disassociate a pipe from a VFP table, click the currently selected VFP table and press the **Delete** key on your keyboard.
6. Click **Apply**.

Change network boundary conditions

The coupling between the boundary nodes and the project flows is determined at import time but can be modified from the **Settings** dialog.

1. In the **ECLIPSE networks** folder in the **Input** pane, right-click the ENS object, and then click **Settings**.
2. In the **Settings** dialog, click the **Settings** tab and then click the **Boundary conditions** tab.

   This tab displays the network boundary nodes and their conditions.

   You can filter boundary conditions by typing the text that you want to search for into the **Node** text box. Click to clear any filter text.
3. Make the required changes to the boundary conditions. The **Terminal node pressure**, which is the fixed pressure of the terminal node in the network, may also be changed.
4. Boundary conditions can be one of two types: Productivity index or Constant Rate. By selecting the appropriate boundary condition type, the relevant table cells are enabled and irrelevant ones are disabled so that you may only make relevant changes.

   Note:

   - In a standalone ENS run, Petrel does not support an Inflow Performance Relationship (IPR) type boundary condition.
   - In a coupled simulation case, when boundary nodes are coupled with wells, the coupler (INTERSECT Field Management) sets an IPR from the reservoir wells as the boundary condition on the coupled nodes in the network.
5. To change the coupled well or choke node, click **Launch search dialog** and select the required ENS node.
6. Click **Apply**.

Define constraints on network nodes

Define network flow and pressure operational limits for an ENS network from the **Settings** dialog.

1. In the **ECLIPSE networks** folder in the **Input** pane, right-click the ENS object, and then click **Settings**.
2. In the **Settings** dialog, click the **Settings** tab and then click the **Node constraints** tab.

   This tab displays the constraints set on the network nodes.

   You can filter existing node constraints by typing the text that you want to search for into the **Node** text box. Click to clear any filter text.
3. Click Add new node constraint to add a new row to the table.
   a. Click **Launch search dialog** and select the required node for the new constraint.
   b. Enter constraint values into the appropriate fields.
   c. Select **Active** to activate the node constraint.

      A constraint will only export if **Active** is selected.
4. To remove a node constraint, select a row and click **Remove selected node constraint**.
5. Click **Apply**.

Define gas or water removal from a network node

Gas or water removal from a network node can be defined from the **Settings** dialog.

1. In the **ECLIPSE networks** folder in the **Input** pane, right-click the ENS object, and then click **Settings**.
2. In the **Settings** dialog, click the **Settings** tab and then click the **Gas & water removal** tab.

   This tab displays the gas and water removal rates and fractions on the network nodes.

   You can filter existing gas or water removal by typing the text that you want to search for into the **Node** text box. Click to clear any filter text.
3. Click Add new gas and water removal definition to add a new row to the table.
   a. Click **Launch search dialog** and select the required node for the new removal.
   b. Enter the removal values into the appropriate fields.
   c. Select **Gas removal active** and/or **Water removal active** to activate the removal.

      Removals will only export if **Gas removal active** and/or **Water removal active** are selected.
   4. To remove a removal, select a row and click Remove selected gas and water removal.
5. Click **Apply**.

Export an ENS model

ENS models can be exported from Petrel and saved as a .DATA file.

1. In the **Input** pane, select the ENS model that you want to export.
2. On the **Home** tab, in the **Manage data** group, click **Export file**.

3. Type a file name in the **Export as** dialog and click **Save**.
4. Select the **File unit system** and click **OK**.

   The model and any dependent data (VFP tables and so on) are exported to the selected location. Petrel displayed any export messages in the **Message log**.

Create a study to view network results

The network result study is created using the **Network results visualization** process. You can share a study between projects using the **Reference project tool**.

1. On the **Simulation** tab, in the **Summary results** group, click **Network results** to open the **Network results visualization** dialog and a **Charting window** to display the selected results.
2. Click **Create new** and enter a name for the study.
3. Select the sources, identifiers and properties that you want to view. When making selections, use the search boxes at the top of the **Sources**, **Identifiers** and **Properties** panes to locate the required sources, identifiers or properties. Click to clear search text.
4. Click to select all sources, identifiers or properties or click to clear all selections.

   You can also categorize identifiers and properties to make selection easier. To do this, click next to **Identifiers** or **Properties**, click **Categorize**, and then click the required category.
5. Choose the **Layout method**.
6. Choose the **Chart type**.

   Properties can be displayed as nested bubbles on a **Bubble chart**, with the area indicating the quantity, or as **Pie charts**, with the segment size indicating the quantity.
7. To hide labels on the chart, clear **Show labels**.
8. Click **Apply**.

   The study is saved in the **Network result** folder in the **Results** pane and the selected results are displayed in the **Charting window**.
9. To change the layout of the results, select a different **Layout method** and click **Re-layout** to update the **Charting window**.

View network results in a study

The network results in a study can be displayed in a **Charting window**. Use the **Time player** to identify the trend or anomalies in the network results.

1. Open a **Charting window**.
2. Select a study from the **Network result folder** in the **Results** pane.

   The results from the study display in the **Charting window**.
3. To edit the study or change the layout method or chart type, click **Network results visualization** on the **Charting window** tool palette to open the **Network results visualization** dialog.
4. To check how the results change with time, on the **Home** tab, in the **View** group, click **Players** and then click **Time player**.

Common workflows for network coupling cases

These examples illustrate how the network coupling cases are supported in Petrel.

The first example describes how to create a reservoir-network coupled case. It covers importing networks, adding the network with network balance actions in a Field management strategy, and defining the simulation case using the created Field management strategy.

The second example demonstrates various processes to display network results in bubble map and line plot formats.

Generate a reservoir-network coupled case

As pre-processor, Petrel imports ENS network data and displays network topology.

After import, you can make the necessary modification to the network settings. The imported networks are then used in the **Field management** process to create a field development strategy for inclusion in a coupled simulation case.

Import and visualize ENS network data

Visualize imported ENS network data in a **Charting window**.

1. In the **Input** pane, right-click and then click **Import file**.
2. In the **Import file** dialog, choose ECLIPSE Network Simulator (*.DATA) from the Files of type list, select the required file and click **Open**.

   Petrel matches the surface node with existing Tubing or Annular flows using their names. If the match is perfect, the **Matching confidence** is green; any partial or failed match is yellow.

   For a partial/failed match, click to open the **Search for Tubing or Annular Flow** dialog.

   In the **Input** pane, the imported networks are listed under the **ECLIPSE Networks** folder. Branches and associated VFP table are listed in a sub-folder in the **VFP (Hydraulic tables)** folder.
3. Open a **Charting window**.
4. In the **Input** pane, double-click on the network to open its **Settings** dialog.
5. On the **Style** tab of the **Settings** dialog:
   d. The color of node and branch can be modified.
   e. The layout of display can also be changed. To do this, select the method from the **Method to use** list and click **Re-layout**.
6. On the **Settings** tab of the **Settings** dialog:
   f. On the **Pipes** sub-tab, the VFP table and ALQ can be modified.
   g. On the **Boundary conditions** sub-tab, the terminal node pressure and boundary conditions can be changed. The value has to be provided for the later simulation run.
   h. On the **Node constraints** sub-tab, the pressure and rate constraints are set.
   i. On the **Gas & water removal** sub-tab (only for a production network), the gas and/or water removal rate or fraction can be set or modified.
   j. The **Network** sub-tab shows the tree structure of network.

7. In the **Input** pane, right-click on the network and click **Export object**.

   The ENS data and network associated VFP tables are exported. The exported ENS model can be run standalone in ENS.

Create and run a coupled simulation case

Use imported networks to create a field development strategy for use in a coupled simulation case.

1. On the **Well Engineering** tab, in the **Field development** group, click **Field management**.
2. In the **Input** pane, click the network in **the ECLIPSE Networks** folder.
3. In the **Field management** dialog, click **Import selected network from input tree** to add the networks into the strategy.
4. In the **New Network balance action** dialog, enter the name for the network balance action and its associated strategy and click **OK**.

   In the **Field management** dialog:

   a. The selected networks are added under the **Networks** folder.

   b. The new action is listed in a **Network Balance Actions** sub-folder in the **Actions** folder.

   c. The new strategy and included action are listed under the **Strategies** folder.
5. Click **OK** to close the **Field management** dialog.
6. Open the **Define simulation case** dialog and create a new INTERSECT simulation case.
7. On the **Strategies** tab:

   a. Select **Use field management strategies**.

   b. Click in the strategies table to open the **Search for Field management strategy** dialog.

   c. Select the field management strategy (with networks) and click **OK**.
8. On the **Results** tab, click the **Summary vectors** sub-tab.

   a. Select all vectors related to networks.

   b. Click to add the vectors to **Selections**.
9. Save the simulation case.
10. Click **Run** to export and run the coupled simulation case.

View network results

Petrel can display network results from a Petrel-generated simulation case or any ENS network coupled cases running outside of Petrel.

Display network results as a line plot

View network results as a line plot using **Results charting and analysis**.

1. On the **Simulation** tab, in the **Summary results** group, click **Results charting**.
2. In the **Result charting and analysis** dialog, toggle on appropriate network summary vectors.
3. Click **OK** to close **Result charting and analysis**.
4. To display more plots, right-click on the **Charting window** and select **Split layout**.

Display network results as bubble maps

View network results as bubble maps in a Charting window. To display network results as bubble maps:

1. On the **Simulation** tab, in the **Summary results** group, click **Network results**.
2. In the **Network results visualization** dialog:
   a. Enter a name for the new plot.
   b. Toggle on appropriate network results.
3. On the **Home** tab, in the **View** group, click **Players** and then click **Time player**.
4. Play with time and observe the bubble size changes with time at network nodes. The area of each bubble represents the value of the result for that node. If multiple result properties have been selected, then stacked bubbles are displayed, one bubble for each property.
5. Alternatively, select **Pie Chart** from the **Chart type** list. Results are grouped into compatible properties and displayed as pie segments. Compatible properties share the same unit measurement, for example oil flow rate and water flow rate. Other selected properties are displayed as stacked regular bubbles.
6. To modify the arrangement of the network nodes, select a display option from the Layout method to use list in the Network results visualization dialog, and click **Re-layout**.

   The node names shown for each bubble can be hidden by clearing the **Show Labels** check box.
7. Click **OK** to close the **Network results visualization** dialog.

   To re-open the **Network results visualization** process dialog, click **Network results visualization** on the **Charting window** tool palette.
8. In **Pick/select mode**, select nodes in the **Charting window**.
9. Right-click on selected nodes and click:
   - **Show chart**, to display a 'stacked' plot at selected nodes in a **Charting window**.
   - **Show chart split by identifier**, to display 'thumbnail' plots at selected nodes in a **Charting window**.
   - **Show stacked chart to terminal**, to display a 'stacked' plot from the selected node all the way to terminal in a **Charting window**.
   - **Show stacked chart to well boundaries**, to display a 'stacked' plot from the selected node all the way to well boundaries on that branch in a **Charting window**.
10. To change the layout, right-click on the chart and select a **Split by** or **Split layout** option.
11. To change the properties display, click **Results charting and analysis** and make appropriate modifications.

Field Management

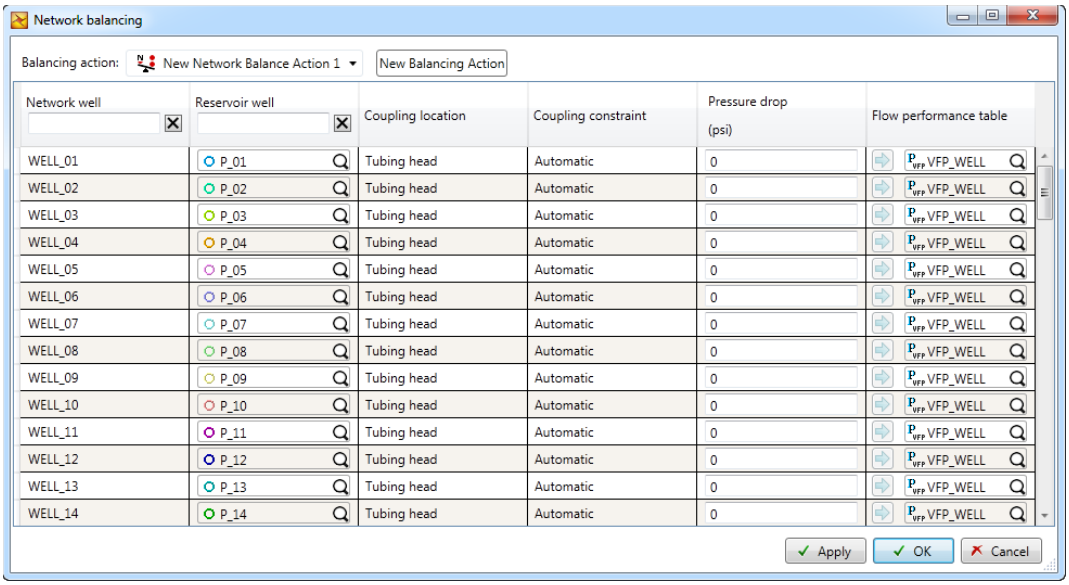
Petrel 2014.3 includes new tools within the Field management process to help you define a strategy more easily including:

- Network balancing to help you create and edit network balancing actions.
- Linking options to help you set up the correct links between actions, instructions, strategies and balancing actions.

Network balance

This dialog assists in the creation and editing of network balancing actions.

To open the **Network balance** dialog, right-click the **Networks** node in the **FM Strategy** tree, and then click **Network balance**.



The components of the dialog are as follows:

Balancing action

The name of the balance action being created or edited; click this to display a list of the available balance actions.

New Balancing Action button

Use this to create a new network balance action. If you already have a network balance action open, you are prompted to save any changes made to it before creating a new one of the specified type.

Network well

This displays the network well for which the values in the other columns apply.

Reservoir well

This column sets up the coupling between the reservoir well and the network well boundaries. This column and the network well column are used to set the boundary node map property on the network balance action.

Coupling location

This column sets up the coupling location between the specified network well and reservoir well. This column and the network well column are used to set the coupling location map and default coupling location properties on the network balance action.

Coupling constraint

This column sets up the coupling constraint on the specified network well. This column and the network well column are used to set the well constraint map and default well constraint properties on the network balance action.

Pressure drop

This column sets up an artificial pressure drop on the specified network well and reservoir well. This column and the reservoir well column are used to set the pressure drop map property on the network balance action.

Flow performance table

This column provides an easy way for the flow performance table on the given reservoir well to be specified at the same time. When coupling at tubing head, this is required to be set to a valid flow performance table.

Linking options

In a valid Field management strategy, there are a number of links between created nodes that need to be set up correctly.

Actions need to be contained in **Instructions** to run, and in turn these instructions need to be listed in the **Strategy** node to be executed during the INTERSECT run. If group constraints and/or coupled networks are to be used, then the corresponding balance action(s) also need to be linked to the **Strategy** node. To assist in the correct formation of these links, whenever an action or instruction is created in the **Field management** process a linking dialog is displayed containing all the linking options appropriate for the type of node that is being created.

Instruction linking

Instructions are created and linked to actions and strategies using the **New Instruction** dialog.

The **Instruction to action(s) link** allows you to choose options for linking to actions.

- No linking.
- Create a new action of the specified type and name along with the instruction and link to its actions property.
- Link to any of the existing actions in the strategy.

The **Instruction to Strategy link** allows you to choose options for linking the strategy node.

- No linking.
- Create a new strategy node with the instruction and add to its "instructions" property.
- Select an existing strategy for the instruction.

Action linking

Any action other than a balancing action is created and linked to instructions and strategies in the **New Configure well action** dialog.

The **Configure well action to instruction link** allows you to choose options for linking to instructions.

- No linking.
- Create a new instruction with the specified name along with the action and link to its actions property.
- Create a link to an existing instruction in the strategy.

The **Instruction to Strategy link** allows you to choose options for linking the instruction to the strategy node if the option to create a new instruction has been selected.

- No linking.
- Create a new strategy node with the specified name along with the instruction. The instruction is added to its "instructions" property.
- Create a link to an existing strategy for the instruction.

Balance action linking

Balancing actions are created and linked to actions and strategies using the **New LP balance action** dialog.

The **Balance action to strategy link** allows you to choose options for linking to the strategy node.

- No linking.
- Create a new strategy node of the specified name along with the instruction and add to its balancing action property.
- Link to an existing strategy using one of the following options:
  - Directly linked. The new balance action is set on the selected strategy's balancing action property, replacing any currently-linked action.
  - Using an action sequence. The new balance action and any existing linked balance actions are wrapped in an action sequence, which is set on the selected strategies balancing action property.
  - A combined network balance action. The new balance action and any existing linked balance actions are linked into a new combined network balance action, which is set on the selected strategies balancing action property.

2014.2

Hydraulic fractures in the Local grid process (behavioral change)

If cells identified for refinement were non-contiguous (for example, if the region contained pinch outs) a local grid refinement around a given hydraulic fracture was not created. This check has been relaxed to allow refinements in regions containing non-contiguous cells to be made.

Hydraulic fracture transmissibility multipliers in history mode (behavioral change)

If a hydraulic fracture was specified with a start date earlier than the export date of a well, then it would not be exported at the first export date as expected. The export date of the well is, by default, calculated as the later of the date at which it can first flow and the start date of observed data. From Petrel 2014.2, hydraulic fractures specified earlier than this export date will be exported at first export date as expected.

2014.1

The **Reservoir Engineering** tab in the new Petrel 2014 user interface takes you left to right from data import to a reservoir model ready for reserves calculation and simulation initialization. It includes the **Make surface** and **Make polygon** utilities, which from the 2014 platform are all available on the Reservoir Engineering core license. You'll find many familiar dialogs and tools on this tab, and also new tools such as the one-click QC plots for fluids and saturation functions.

You can **build a simple simulation grid** with properties from scratch, or **refine or upscale** the geological grid and properties, then **import or create the fluids**, **rock physics functions** and **aquifers**. The one-click plots enable you to QC the input data as it is imported, and **CTRL-F** makes it easy to find the data you've created. Once your reservoir model is built then you calculate reserves using the static **volume calculation** or by moving onto the **Simulation** tab and using a simulator to initialize your reservoir.

The dual scale modelling workflow introduced in 2013.3 uses the **Global refinement** dialog to create the matching refined grid for property population from the initial simulation resolution grid.

QC workflow enhancements

Fluid model plots

Single-click fluid model plots, available from the **Fluids** group on the **Reservoir Engineering** tab, are designed to streamline the quality control process.
The following fluid model plots are available:

- **Black oil - oil phase**

  For a Black oil fluid model, a plot with four function viewports is created displaying oil formation volume factor, oil viscosity, solution gas-oil ratio and fluid contacts.
- **Black oil - gas phase**

  For a Black oil fluid model, a plot with four function viewports is created displaying gas formation volume factor, gas viscosity, saturation pressure ratio and fluid contacts.
- **Compositional**

  For a Compositional fluid model, a plot with two function viewports is created displaying phase envelope and fingerprint plot (fraction versus molar mass).

Display fluid model plots

Use fluid model plots to display quickly the data for the active fluid model in multiple viewports.

To display a fluid model plot:

1. In the **Fluids** folder in the **Input** pane, select a fluid model.
2. On the **Reservoir Engineering** tab, in the **Fluids** group, click Fluid p**lots**, and then click the plot you want.

   A **Function window** opens, displaying the selected plot.

Note: If there is no active fluid function, the first function listed in the **Fluids** folder is displayed. If a **Function window** is already active, the plot will overwrite the data currently shown in the window.

Rock physics plots

Single-click rock physics plots, available from the **Rock physics** group on the **Reservoir Engineering** tab, are designed to streamline the quality control process.
The following rock physics plots are available:

- **Saturation function**

  For a saturation function, the plot displays relative permeabilities and capillary pressures (both for 3-phase situations).
- **Rock compaction**

  For a rock compaction function, the plot displays pore volume multiplier and transmissibility multiplier versus rock pressure.

Display rock physics plots

Use rock physics plots to display quickly the data for the active saturation function or rock compaction object.
To display a rock physics plot:

1. In the **Rock physics function** folder in the **Input** pane, select a rock compaction or saturation function.
2. On the **Reservoir Engineering** tab, in the **Rock physics** group, click **Saturation plots**, and then click the plot you want.

   A **Function window** opens displaying the selected plot.

Note: If there is no active function, the first function listed in the **Rock physics functions** folder is displayed. If a **Function window** is already active, the plot will overwrite the data currently shown in the window.

Dual scale workflow

This is an alternative to the traditional method for preparing a simulation grid.

Dual-scale reservoir modeling stimulates collaboration between sub-surface experts, notably Reservoir Engineers and Geologists, and across the domains in the general Petrel workflow. The process starts with the collaborative production of a simulation-friendly coarse-scale grid, which provides a physical model of geological structures including faults and surfaces. This grid is then refined, without changing the geometry of the structures, and populated with properties. The properties are then upscaled to the original coarse grid ready for use in simulation. The different scales in the coarse and fine models provide the resolution required by the domain experts, but with a common view of physical features.

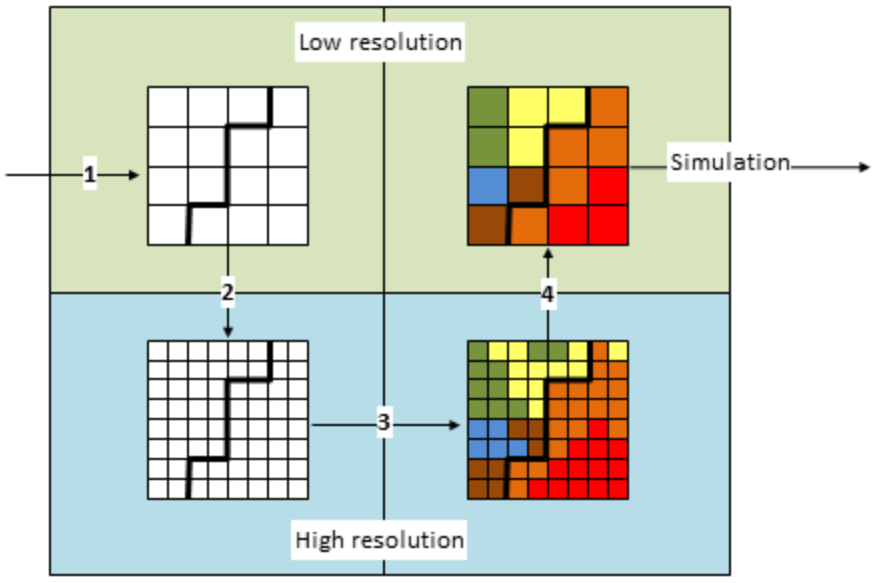
There are a number of technical benefits inherent in the dual-scale workflow:

- There is rigorous alignment between the fine and coarse grids, determined by the mapping function.
- There is no need for geometrical comparisons between the two grids as the geometries of the grid structures are not changed by refinement and upgridding.
- The bulk rock mismatch between the coarse and fine models is minimized by the mapping.
- Each fine cell contributes to no more than one coarse cell, minimizing sampling errors and maximizing property resolution.
- There is a well-defined mapping from the cells in the fine grid to the cells in the coarse grid, which is required for transmissibility-based upscaling.

One of the features of the workflow is that all structures such as faults and horizons are at the same resolution in both the coarse and fine grids, that is the refinement and upscaling do not change any of these structures. When you analyze the volumes at the two different grid scales, you may find that the differences in them are due to the original grid being too coarse, with insufficient structural detail for the reservoir features. In this case, you need to refine parts of the original grid to provide a more appropriate level of detail for the reservoir structures and start the workflow again.

Note: The following grids are currently supported by the dual-scale reservoir modeling process: structured pillar grids, vertical, straight, listric, curved and multi-linear pillars, pillar faults, stair-step faults.

Figure 118: Dual scale workflow



The differences in approach between dual-scale reservoir modeling and the traditional method for preparing a simulation grid are:

Dual-scale reservoir modeling	Traditional simulation grid building
Create/import a coarse-scale grid (**1**). Create a simulation-friendly, coarse grid with sufficient detail to show features such as faults and horizons.	
Create a fine-scale grid (**2**). Use the refinement process to produce a fine-scale grid. The process does not alter any structures and creates mapping parameters that define the relationships between fine and coarse cells.	Create a fine-scale grid. Use the fine-scale grid to define the physical and structural features of the geological model.
Populate properties in the fine grid (**3**). Populate the fine-scale grid properties including rock types, porosity, permeability and fluid saturations.	Populate properties. Populate properties in the grid using the same processes as dual-scale reservoir modeling
	Create the coarse-scale simulation grid.
Upscale properties to the coarse grid (**4**). Use the mapping parameters in the mapped upscaling process to average properties to the original coarse grid.	Upgrid and upscale from the fine to the coarse grid. Use the upgridding process to coarsen the grid geometry and to recalculate the physical descriptions of structures in the model. Use the upscaling process to average and recalculate property values for the coarser cell sizes of the coarse-scale simulation grid.
Quality check (QC) the grids. Compare the bulk volumes, net bulk volumes, pore volumes and fluid volumes of the fine and coarse grids.	QC the grids. Perform the same QC as for dual-scale reservoir modeling.

The mapping function is used during refinement and upscaling to maintain the relationship between the coarse and fine cells. During refinement, the mapping function creates a relationship between each coarse cell and the fine cells created from it. During upscaling/upgridding, which are combined in dual-scale reservoir modeling, the mapping ensures that each refined cell is mapped back to the original coarse cell that produced it.

Grid refinement

Use the Grid refinement process to refine an existing grid.

The input to the Grid refinement process is a simulation-friendly 'coarse' grid which is refined without changing the structure.

Note: The following grids are currently supported by the dual-scale reservoir modeling process: structured pillar grids, vertical, straight, listric, curved and multi-linear pillars, pillar faults, stair-step faults.

To refine a grid:

1. On the Reservoir Engineering tab, in the Regridding group, click Global refinement to open the Grid refinement dialog.
2. In the **Models** pane select the coarse-scale grid and insert ➪ as the **Input grid**.

Note that a name is automatically allocated to the new refined grid based on the Input grid. This name can be changed later in the **Settings** dialog of the new refined grid, if required.

3. Specify the **I**, **J** and **K** refinements.
4. If you want to apply different **K** refinements for each zone (and subzone) in the grid, select the **Use per zone refinement** check box and enter the required refinement for each zone (and subzone) in the zone refinement table. This table is automatically populated when an input grid is selected.
5. To use the same K refinement for all zones, enter the value in the **K refinement** box and click  .
6. Click **OK**.

   The grid is refined and appears on the Models pane. The grid name is appended with 'Refinement of' for easier identification.

Note: If you right-click the refined grid in the **Models** pane, and then click **Settings**, a record of the refinement is shown on the **Info**, **History** tab.

Mapping properties

In addition to a refined grid, the **Grid refinement** process creates three properties in a **Refinement properties** folder in the **Properties** folder for the refined grid on the **Models** pane. These properties, **I mapping**, **J mapping and K mapping**, supply a mapping from cells in the refined grid to their corresponding cell in the input grid. The **I mapping property**, for example, will have values from 1 to the maximum I index in the input grid.

The I, J and K indices are interpreted using the current **Petrel cell origin** settings on the input grid found on the **Export to simulator and visualization** sub-tab of the **Mapping** tab in the **Settings** dialog for the grid. The mapping properties are used in the **Scale up properties** process to upscale properties from the refined grid to the input grid by selecting the **Mapping with properties** sampling method.

Refinement statistics

The **Grid refinement** process also creates a **bulk volume multiplier** property in the **Refinement properties** folder. The value of this property represents the relative difference in total volume of a coarse cell and its corresponding refined cells. For example, if the value equals 1.0, there is a perfect match. If the value equals 0.5, the combined volume of the fine cells is twice the volume of the original coarse cell. Note that the value is constant across all fine cells belonging to the same coarse cell.

Well Engineering

The **Well engineering** tab guides the user through **Well and completions** design and **Well deliverability** calculation, leading to building your **Field development strategy** either in Development Strategies for ECLIPSE or in Field Management for INTERSECT. 2014.1 introduce several significant usability and productivity enhancements to the Field Management dialog.

The new interface presents the existing range of options within Petrel for well path design and for completion design in a logical left to right order. For example, within the Well path group, the interface offers left to right the simplest vertical well using New well, then the use of predefined drilling-friendly templates, then to automated design option suited to optimization algorithms, and finally a fully interactive manual option. The new enhanced explanatory tooltips to help the user to choose the best option for their task for well path and completion design.

The well deliverability group guides the user through the steps to define and use single well simulation using the PIPESIM engine leading to a Nodal analysis.

Completions import and export enhancements

Several enhancements have been made to the import and export of completions, particularly for the sliding sleeve completions introduced in 2013.1 release.

Sliding sleeve

Two new rules apply if you choose to build a functional well completion when importing a device control data file :

- The well is cased to Total Depth (TD).
- A perforation is placed between each packed interval.

Event file export

When exporting an event file you can now export completion depths and attributes in either Metric or Field unit systems. By default, the unit system most closely matching the Project system is selected. A UNITS keyword in the file maintains knowledge of which unit system was selected.

Import completion device control data for sliding sleeves and AFCVs

If you choose to build well completions from the **SlidingSleeve** section of the file, Petrel will create a completion around the sliding sleeve/AFCV. This will consist of a casing to the well's total depth (TD), tubing extending to the most upstream depth from the 'SlidingSleeve' data, packers at the given 'SlidingSleeve' depths, the required completion device (Sliding Sleeve or AFCV) at the 'SlidingSleeve' downstream depth and perforations between each pair of packers. This will use default equipment. Upstream and downstream are defined in terms of the flow path of a producing well. Note that the **Build well completion from Sliding Sleeve data** option will delete any existing completions before creating the completions.

Field management dialog

Petrel 2014.1 introduces several significant usability and productivity enhancements to the **Field Management** dialog designed to reduce the setup time for users. The well constraints table gives users clear overview and quick access to all wells and their constraint values in a single filterable dialog. The syntax and validation required for expressions in Field Management has been simplified to reduce mouse clicks .A dialog has been introduced to link actions, instructions and strategies. This dialog guides the users to create or link the three objects in one go, and hence it boosts users' confidence in understanding the INTERSECT FM logic framework.

Well constraints table

Management of well constraints is simplified with the introduction of a **Well constraints** table.

Manage well constraints in a Field management strategy

The following workflow example illustrates how to create, edit or delete constraints for groups of wells in a Field management strategy.

Constraints can be applied to individual wells in a Field management strategy by selecting the well and setting the appropriate properties. If you have grouped wells within your strategy, you can view, create, edit or delete constraints for wells within a group using the Well constraints option.

To manage well constraints:

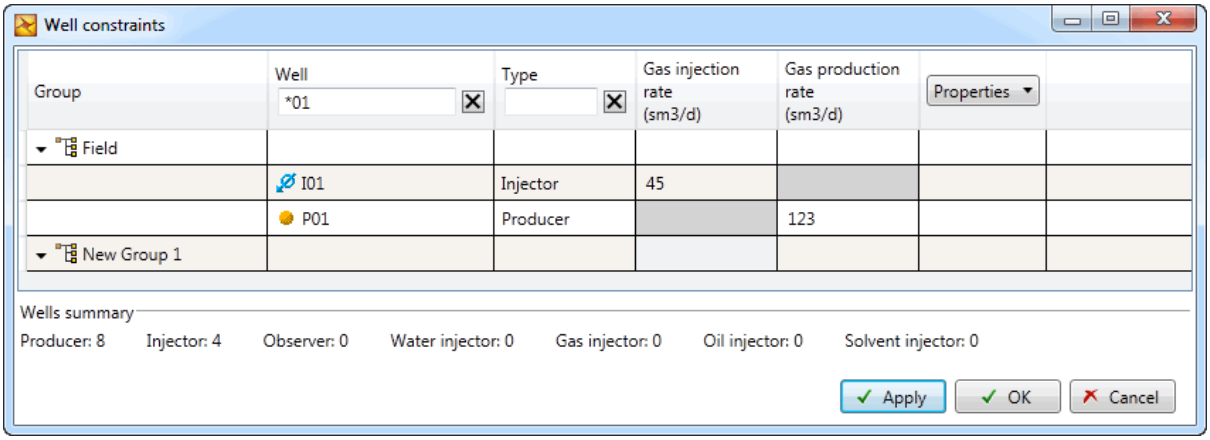
1. Open the **Field management** dialog.
2. Either create a new strategy or select an existing strategy from the **Edit existing** list.
3. If you are creating a new strategy, add and group your wells.
4. Right-click the **Wells** node in the **FM Strategy** list, and then click **Well constraints.**

   The **Well constraints** dialog appears listing existing groups, their contained wells and any well constraints that have been defined.

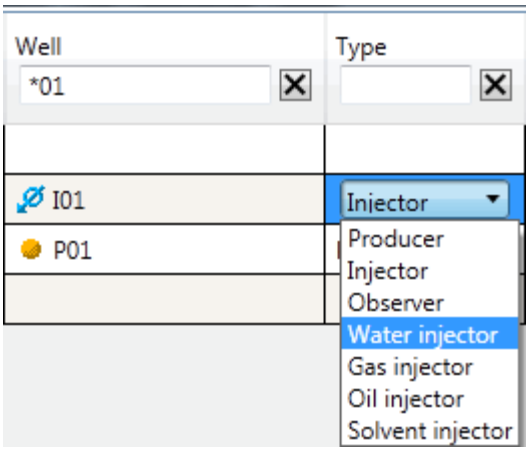
   Note: The wells for a group can be displayed or hidden by clicking the triangular 'expand/collapse' icon in the **Groups** column. Wells can also be filtered from view using the well or type filters at the top of the **Well** and **Type** columns respectively. Filters interpret trailing whitespace as a wildcard (*).
5. To add a constraint, click **Properties** and select the required constraint property.

   A new empty column for the selected property is added to the table; no constraints are created.
6. Click inside the cell in the table corresponding to the constraint that you want to define or change and type the required value.

   Inappropriate constraint properties for a well type are displayed in gray and cannot be edited. An invalid value for a constraint is indicated by a red border around the cell.



7. To delete a constraint, clear the value in the appropriate table cell.
8. To change a well type, select the table cell corresponding to the well in the **Type** column. Click and select the new type from the list.



9. When all constraints are defined, click **Apply** and then **OK** to close the **Well constraints** dialog.
10. Click **Apply** to save your strategy.

Actions, instructions and strategies

Enhancements have been introduced to make it easier to create actions and instructions and link them to new or existing instructions, actions or strategies.

Create an action and link to an instruction

When you create new actions, you can link them to new or existing instructions.

1. In the **FM Strategy** pane, right-click **Actions**, click **New**, and then click and action type.
2. In the **New 'action'** dialog box, optionally change the **Name** of the action.
3. To link the action to a new instruction, click **Create new instruction** and optionally change the **Name** of the new instruction.
4. To link the action to an existing instruction, click **Link existing instruction** and select the instruction from the list.
5. Click **OK**.

If you select an instruction in the **FM Strategy** pane, the linked actions are displayed in the **Properties** pane. If you chose to create a new instruction to link to the action, you should configure the instruction's trigger and success properties, if applicable.

Create and link an instruction to an action or strategy

When you create new instructions, you can link them to new or existing actions and/or strategies.

1. In the **FM Strategy** pane, right-click Instructions, click **New**, and then click **Instruction**.
2. In the **New Instruction** dialog box, optionally change the **Name** of the instruction.
3. To link the instruction to a new action:
   a. Click Create new Action.
   b. Select the action **Type**.
   c. Optionally, change the action **Name**.
4. To link the instruction to an existing action, click **Link existing Action(s)** and select the actions that you want to link.

5. To link the instruction to a new strategy, click **Create new Strategy** and optionally change the **Name** of the new strategy.
6. To link the instruction to an existing strategy, click **Link existing Strategies** and select the strategies that you want to link.
7. Click **OK**.

If you select an instruction in the **FM Strategy** pane, the linked actions are displayed in the **Properties** pane. Select a strategy in the **FM Strategy** pane to display the linked instructions in the **Properties** pane.

Simulation

The **Simulation** tab provides the tools for single and multiple simulation case workflows. You can **import** or **create**, **manage** and **analyze** dynamic reservoir models. You can move on to do **history matching**, analyze **sensitivities** and **uncertainties** across multiple cases. Finally you can **derive** new cases from existing ones when converting an imported case, or creating RESTART cases and sector models.
A number of new productivity enhancements have been delivered in 2014.1 that are applicable to all simulators from making it easier to find tools such as the multi-value probe, and removing the need to select the simulator type when importing a simulation deck into Petrel, to one-click access to line plots and 3D results displays, and the new custom splitting functionality in results charting and analysis.

The simulation case context tab presents all the operations and tools relevant to an individual simulation in one easy to see tab, which will generally act on the *selected* case – whose name is also displayed on the left hand of the tab. Note though that the dialogs for Petrel processes, such as Results Charting or Define simulation case, continue to open on the *active* simulation case

Significant new uncertainty and optimization functionality is introduced with 2014.1; this is the first step on the multi-year roadmap to bring the best of the MEPO uncertainty and optimization technology onto the Petrel platform.

The import of INTERSECT cases and results, and the user control over the output of INTERSECT cases created within Petrel have both been enhanced in Petrel 2014.1. NB These enhancements require the use of INTERSECT 2013.2.

Simulation analysis

Productivity enhancements

Single-click plots performance reports have been introduced to enhance productivity when analyzing simulation results.

Field plots

Use **Field plots** to create different field performance charts for the active case and display these in a **Charting window**.

Field plot are stored in the Performance studies folder under Results charting and analyses on the Results pane. You can adjust field plots to suit the needs of the project using the Result charting and analysis process.

The following field plots are available:

- **Production rates**

Oil, gas and water production rates along with pressure at the field level.

- **Production cumulatives**

  Oil, gas and water production cumulatives at the field level.
- **Injection rates**

  Oil, gas and water injection rates at the field level.
- **Injection cumulatives**

  Oil, gas and water injection cumulatives at the field level.

To display a field plot:

1. In the **Cases** pane, select a case.
2. On the **Simulation Case Tools** tab, in the **Quick view** group, click **Field plots**, and then click the plot that you want.

   A **Charting window** opens displaying the selected field performance chart.
3. To adjust the chart, on the **Charting window tool palette**, click  to open the **Results charting and analysis** process.

Dynamic well filters

Fundamental to simulation and production workflows is identifying wells of interest from amongst all the wells in your field, and then analyzing them, for example for identifying work-over candidates or the reasons for a poor history match. The well saved search feature in Petrel has been extended as a great way to define a filter for a group of wells based on their dynamic production or simulation data. With 2014, these well saved searches can now be based on dynamic observed or simulation production data at a given date, or a range of time. So you can for example, define a saved search to filter all the wells that have high oil production but also high water cut at the end of the history. Another example is that you can quickly group all the wells into groups based on different well types (producers, injectors, injectors converted from producers, etc.). If you're using assisted history matching in Petrel, all those mismatch summary vectors can be used here to quickly filter out the badly history matched wells.

Extended saved searches

Extended saved searches expand the existing saved searches functionality to group wells in the **Input** pane based on various criteria.

Dynamic saved searches

A dynamic saved search is a type of extended saved search that uses input data to filter wells in the **Input** pane. Input data, in the case of the dynamic saved searches, are results from a number of different types of sources. These include development strategies, observed data and simulation cases. The filtering is done with the selection of a source, a property from that source, start and end dates, the value to compare against, the desired operator and finally the frequency with which the comparison needs to meet the criteria. The units are given for reference only and cannot be changed when creating a dynamic saved search.

Dynamic saved searches are listed under the **Saved searches** folder in the **Wells** folder on the **Input** pane. To apply a dynamic saved search, select the check box next to its name – '[Search ON]' is displayed next to **Wells** to indicate that a search is currently applied. To move wells filtered by a dynamic saved search to a new folder in the **Wells** folder, right-click the dynamic saved search, and then click **Move wells to folder based on saved search**.

Note: Dynamic saved searches depend on calculated input data and when these data change any search that includes this data in one or more of its comparisons will potentially be affected. When this happens, any dynamic saved search that is potentially affected is marked with an exclamation mark in the **Input** tree. Toggling the search off and on will cause it to recalculate its filtered wells.

Create a dynamic saved search

Use a dynamic saved search to filter and group wells based on input data from development strategies, observed data and simulation cases.

To create a dynamic saved search:

1. In the **Input** pane, expand the **Wells** folder.
2. Right-click the **Saved searches** folder, click **Create extended saved search**, and then click **Dynamic saved search: basic property**.
3. On the **Search** criteria tab of the **Settings** dialog, define the filter criteria for the search.
   To do this, add, modify, reorder and remove comparison rows that, in combination, form the filter criteria of the search.
   a. Click **Append comparison** to add a new row to the table.
   b. Select the **Source** for the data.
      This is the selection of the input data source that will define the available properties for comparison.
   c. Select the **Property**.
      Depending on the selected source, this will be the property by which the comparison will be carried out.
   d. Select the **Start** and **End** dates over which the comparison is to be made.
      These are the dates that define the time period in which the comparison will be relevant. The values available depend on the selection of source and property. It is possible to search on only one time step by setting the start and end date to the same date.
   e. Select the comparison **Operator** and **Value**.
      Available operators are 'more than', 'more than or equal', 'equal', 'less than' and 'less than or equal'. The value that you enter will be compared using the selected operator with the input data selected.
   f. Select the **Frequency** of occurrence over the selected time range.
      'All times' will cause the property value comparison to be executed for all times in the selected period. A well will be included in the search if its input data satisfies the comparison options for all dates in the period. 'Once' will include a well in the search if its input data satisfies the comparison options for at least one time in the selected period.
   g. Add new rows to the table, as required. You can either **Append** a new row, or **Insert above** or **Insert below** existing comparison rows. Change the order of rows using the **Move up** and **Move down** buttons.
      Comparisons are combined in order according to the chosen **Combine** option. 'And' will combine the resultant wells of the current comparison (one or more rows) with the resultant wells of the following comparison (row) as an intersection. 'Or' will combine rows as a union.
4. Click **Apply** or **OK**.

To modify an existing search, double-click the search to open the **Settings** dialog. Click the Info tab to change the name of the search and its display color; to change the criteria used by the search, click the **Search criteria** tab.

Customized charting

The results charting and analysis for production and simulation data has two key enhancements that enable our users to now create and of course share and re-use through the Reference project tool just about any set of split charts they can imagine. The first of these is the custom property grouping. By using it, you have the flexibility to generate plots for any combination of properties, based on reporting or analysis purpose. The custom groups can be set up interactively from the right mouse context menu on a chart, and are saved with the settings for the chart. Hence regularly used custom groupings can be reused in the current project for new simulation studies, and can be shared to other projects through Reference project tools and reused in other projects.

The second major enhancement is the multi-level splitting that enables for example a mix of field and wells level charts to be created in one split. By using it, you can quickly generate all the plots for selected wells, selected properties, and multiple cases in an expected combination. Custom property grouping can be used here as well.

Customize chart splitting

You can create new ways of splitting charts.

To define customized splits for your charts, click the **Customize splitting** button  at the bottom of the **Results charting and analysis** dialog to open the **Custom splitting** dialog.

Note: You can also open the **Custom splitting** dialog:

- By right-clicking a chart and selecting **Customize splitting**.
- From the **Style** group on the **Window Charting** tab.
- From the **Settings** dialog for the **Splits and groups** subject on the **Results** pane.

There are two ways to customize splitting, represented by the two tabs on the **Custom splitting** dialog:

- **Custom property grouping**, which allows you to group properties into groups that will appear on separate charts.
- **Multi-level splitting**, which allows you to define multiple levels of splitting, and introduce exceptions to each level that will be distributed over multiple charts.

Custom property grouping

To define which properties should appear on which charts, drag and drop properties from the **Properties to group** tree on the left of the **Custom property grouping** tab to the **Custom grouping** tree on the right. Properties can also be dragged between groups in the **Custom grouping tree**, or into new groups. Each group represents one possible chart in the resulting split.

When you have defined a suitable property grouping, click **OK** to apply your changes and close the dialog. To apply the custom split, either select from the **Split b**y list on the **Results charting and analysis** dialog, or right-click a chart, point to **Split by** and select the custom split.

Multi-level splitting

Here you can define to split by more than one property of the selected data. For instance, you could split by 'Source' then by 'Primary identifier'. This would result in all series for each well and for each source being placed in a single plot.

You can also define exceptions. For example, if you want to compare the results from multiple simulation cases to one observed data set, you could split by 'Source', but add an exception for Source type, 'Observed'. This would result in the observed data being plotted alongside the data for each selected case.

To add a multi-level split, open the **Custom splitting** dialog and click the **Multi-level splitting** tab. Click **Add level** and select the required property from the drop-down list. A level is only considered valid if it has an assigned property. Add additional levels, as required. To define an exception, click **Manage exceptions** to open the **Edit splitting component and select exceptions** dialog. Select the exceptions and click **OK**.

Once you have defined a suitable multi-level split, click **OK** to apply your changes and close the dialog. To apply the custom split, either select from the **Split by** list on the **Results charting and analysis** dialog, or right-click a chart, point to Split by and select the custom split.

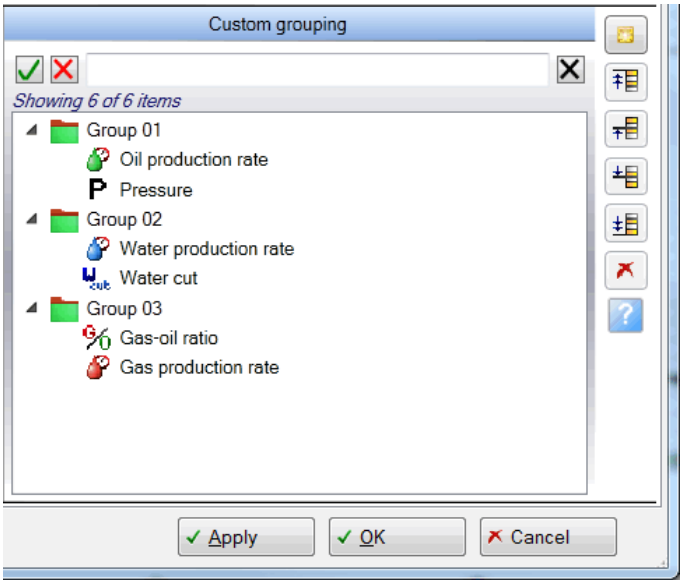
Sharing customized splits

Customized splits can be shared between multiple projects by copying the **Splits and groups** subject on the **Results** pane via the **Reference project tool**.

Use custom property grouping to create a chart with customized splitting

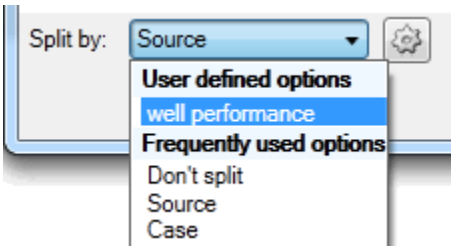
How to use custom property grouping to split charts.

1. On the Simulation tab, in the Summary results group, click Results charting.
2. If required, create a new study or chart.
3. Click to open the **Custom splitting** dialog.
4. On the **Custom property grouping** tab, create a property grouping called 'well performance'.
5. In the **Custom grouping** panel, create three groups containing the following properties, and then click **OK**.



The groups appear in the order shown starting with Group 01 on the left-hand side of your graph, Group 02 on the right-hand side, and so on. Note that the appearance also depends on the selected split layout, which by default is 2 X 2. To change the layout, right-click the chart, point to **Split layout**, and then click the required layout.

6. Select the split that you just created from the **Split by** list.



Now any data selections you make will be split into separate charts according to your custom grouping.
Note: All properties that were not explicitly placed into a custom group will be grouped together into an additional chart.

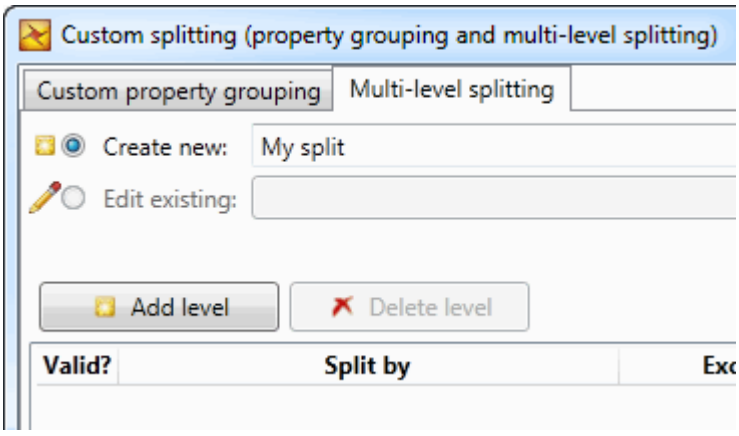
Once you have reviewed the content, you can extract information for reporting and sharing purposes.

Use multi-level splitting to create a chart with customized splitting

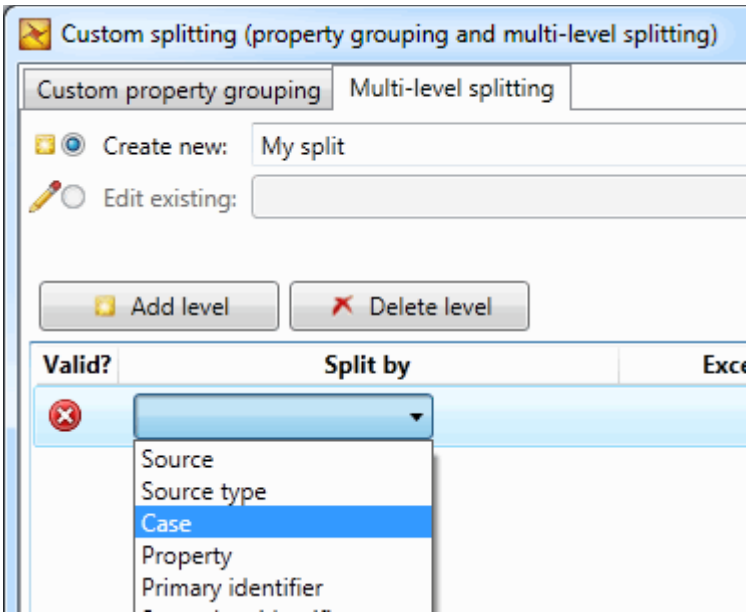
How to use multi-level splitting to create a chart to compare a base case with a number of other comparison cases.

This example assumes you have a project with a base case and multiple comparison cases.

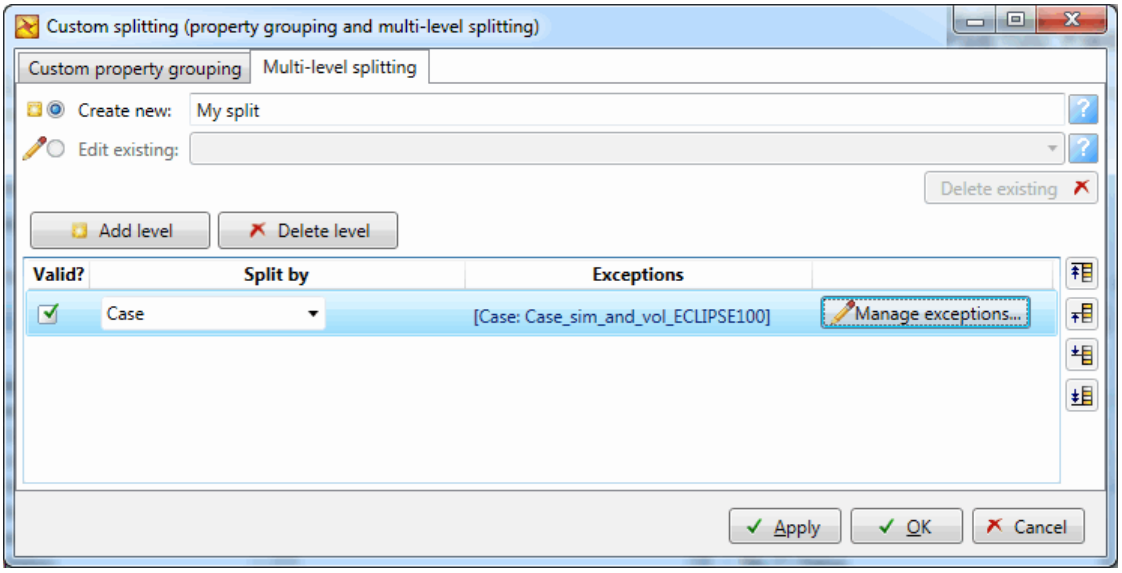
1. On the Simulation tab, in the Summary results group, click Results charting.
2. If required, create a new study or chart.
3. Click ⚙ to open the **Custom splitting** dialog.
4. Click the Multi-level splitting tab.
5. Type a name for the new split in the box next to **Create new**.



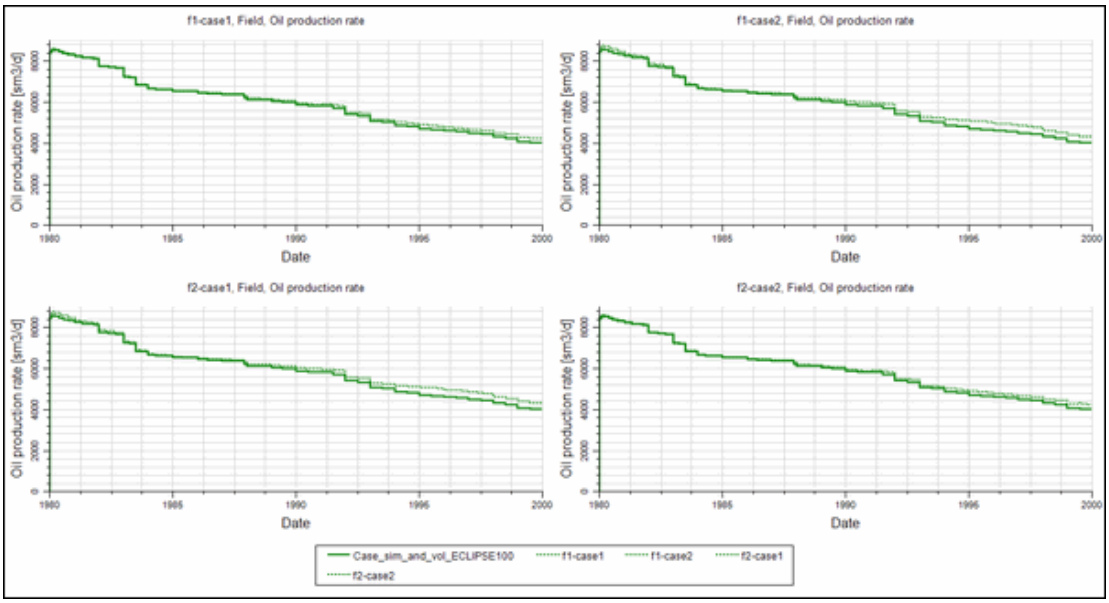
6. Click **Add level** and select 'Case' from the **Split by** list.



7. Click **Manage exceptions** to open the **Edit splitting component and select exceptions** dialog. Click **Case** and then select your 'base case'. Click **OK**.
   You should now have something similar to the following:



8. Click **OK** to close the **Custom splitting** dialog.
9. Select your new split from the **Split by** list in the **Results charting and analysis** dialog.
10. Select Field OPR from the base case and a number of comparison cases. You should see a chart similar to the following in which each plot contains the base case and one comparison case:



Once you have reviewed the content, you can extract information for reporting and sharing purposes.

Interactive filtering and saved searches

When a simulation case viewed in the **Results charting and analysis** dialog contains large numbers of items in its selection panes, you can filter the lists to help locate items of interest more rapidly.

- **Interactive searches**

  You can filter items using the following interactive search methods: contained text, wildcards and initials. The filters support the use of '*' and '?' wildcards, where '*' represents any number of characters and '?' represents a single character.
  Note: Apply all filters before selecting the items to output as graphs. Once you have created a graph, applying further filters only affects the visibility of items in the dialog, not in the resultant graphs.

- **Saved searches**

  If the project contains well saved searches (defined in the **Saved searches** folder in the **Input** pane), they will appear in a **Saved search** list below the interactive filter for the **Primary** identifiers tree. Selecting a saved search filters the visibility of wells in the tree to include only those wells that match the chosen search. Selecting "None" resets the visibility of all the wells.

Pre-set multi-level split options

Three pre-set multi-level split options are included in the context menu within the **Charting window**:

- **Field-well comparison**
  One chart for each selected well as the primary identifier. All selected series having the same well as the primary identifier will appear on the same chart. To aid comparison between the field and a well, the field data for each selected property (if selected) is also plotted on each chart.

- **HM Multi-case comparison**
  One chart for each selected primary identifier (for example, well) with each selected property (such as "oil production cumulative"). All series having the same primary identifier and the same property will appear on the same chart.

- **HM observed data**
  One chart for each selected source with each selected primary identifier (for example, field or well) and with each selected property (such as "oil production cumulative"). All series having the same source, the same primary identifier and the same property will appear on the same chart. To aid comparison between the simulated data and the observed data, the observed data (if selected) for each selected primary identifier with each property is also plotted on each chart.

Saturation functions and fluid model must now be added to Numerical Aquifers (behavioral change)

In previous releases of Petrel, it was possible to clear the **Fluid model** and **Saturation function** fields in the **Properties** tab of the **Make Aquifer** process. Doing this meant that the aquifer used the fluid model and saturation function of the cell in the model to which the aquifer was connected.

With this release of Petrel, it is no longer possible to create a Numerical aquifer without providing a fluid model and saturation function for the aquifer.

Although possible, it is **not recommended** to reproduce previous behavior by extracting the fluid model and saturation function from the ECLIPSE dataset. This approach it is likely to be inaccurate – the fluid model and saturation function in the grid cell are likely very different from the desired aquifer fluid model and saturation function.

Reservoir Engineering: Uncertainty and Optimization

2014.1

Multiple case analysis

Single-click tornado plots

Use single-click tornado plots to compare quickly the relative importance of the uncertain variables to field cumulative production or objective function.

Single-click tornado plots are designed to streamline the quality control process. The plot is built for all cases in the selected case folder. A new **Tornado plot window** is created if no existing window is currently active.

The following tornado plot options are available:

- **Production** – used to compare the relative importance of the uncertain variables to field cumulative production.
- **Objective function** – used to compare the relative importance of the uncertain variables to objective function.
- **Production - cumulative view –** used to compare the relative importance of the uncertain variables to field cumulative production for all timesteps at once.

To display a single-click tornado plot:

1. In the **Cases** pane, select a case folder.
2. On the **Simulation** tab, in the **Multiple case analysis** group, click **Tornado plot**, and then click the plot that you want.

   A **Tornado plot window** opens, displaying the selected tornado plot.

Correlation analysis

Correlation analysis, which is typically is used to understand dependency between uncertainty variables and the project objective – or to reveal conflicts appearing due to a history matching set-up – like the water cut match steadily becoming poorer as the pressure match in the same well improves. This identifies that the model may not be matched with the current use of uncertainty variables and guides the user to reconsider the setup.

The *Uncertainties vs. Responses* analysis displays the impact of parameters; this is similar analysis that a tornado plot provides, but is more flexible as any simulation database can be analyzed.

The *Responses vs Responses* analysis displays potentially conflicting objectives; strong red colors in this view would indicate problems requiring inspection
Data types supported by correlation analysis are the same as those in cluster analysis: case variables, objective function partials and RFT mismatches.

To open the **Correlation analysis** dialog, on the **Simulation** tab, in the **Multiple case analysis** group, click **Analysis** and then click **Correlation analysis**.

The **Correlation analysis** dialog is divided into two main sections:

- The upper part of the dialog provides tools for selecting data to be used in analysis.
- The area at the bottom of the dialog shows the correlation analysis result. Results are presented either as a table or in a list. Cells are automatically colored according to the correlation coefficient values.

It is recommended that you select the cases first to filter the list of variables accordingly. If you do this, you avoid building up a combination of variables which are not available in the shown cases. Data attributes can be added to and removed from the **Rows** and **Columns** lists on the right side using the buttons between the data attributes list and the **Rows** and **Columns** panes. Data grouping and filtering is also supported.

To run the analysis, two different methods can be defined, the 'Pearson' or the 'Spearman' algorithm. The Pearson method is a linear correlation while the Spearman correlation is used when the data has many outliers. The second correlation is more stable since it first ranks the values and then correlates those ranks. The **Calculate** button will run the analysis process.

Cluster analysis

The cluster analysis technique is useful to find models that are fundamentally different. Cluster analysis makes it easier for users to find for instance the models that are of most interest after a history matching study. The analysis may reveal which models of good quality that are fundamentally different with respect to use of uncertainty variables and which models of equal overall match quality that are most different when it comes to matching of different data sets – like some matching pressures best and another are better on GOR and WCT.
For prediction purposes the models that have similar match quality, but fundamental differences are the most interesting to evaluate. Simply because they have the largest likelihood of producing substantially different forecasts to account for in the decision making process.

To open the **Cluster analysis** dialog, on the **Simulation** tab, in the **Multiple case analysis** group, click **Cluster analysis**.

The **Cluster analysis** dialog is divided into two main sections:

- The upper part of the dialog provides tools for selecting data to be used in analysis.
- The area at the bottom of the dialog shows the cluster analysis result, where all simulation cases are designated to a cluster and cluster representatives are marked.

For cluster analysis it is recommended that you select simulation cases first (in the **Select cases** pane) since this will have an impact on the parameter selection. The **Select data** pane only contains parameters which are available in all selected cases. If no cases are selected in the **Select cases** pane (by default), all available attributes will be shown in the selection list.

There are three different data types available for the cluster analysis:

- Case variables - typically uncertainty/control parameters and global values of objective functions.
- Partial mismatches calculated by the **Define objective function** process (stored in the results tree as history match quantities).

- Partial mismatches calculated by the **Define RFT mismatch** process (stored separately in the project).

To make data selection easier, it is possible to create custom groups of these data types.

Data attributes that will be used by cluster analysis are shown in the **Selected** pane. To add parameters to the cluster analysis, select them in the **Select data** list and insert into the **Selected** pane.

When the data selection is complete, the number of clusters need to be specified before the analysis method can be run using the **Calculate** button. It is recommended to start with 3 or 4 clusters and increase or decrease the number of clusters, if needed.

The result is presented in a table format at the bottom of the **Cluster analysis** dialog. The cluster number is shown for each selected case as well a check box marking the representative of the individual clusters.

Group and filter data for cluster or correlation analysis

Instead of using individual data values, the data attributes for cluster or correlation analysis can be grouped together. Typical examples of groupings are a sum of all mismatches for a particular quantity, for a particular well or for a particular date.

Data attributes can also be filtered to reduce the number of presented attributes. Both the grouping and the filtering are achieved using data selections which are available in cluster and correlation analysis.

To create groupings and selections:

1. In the **Correlation analysis** or **Cluster analysis** dialog, click to open the **Data identifier selection** dialog.
   All available parameters are shown on the left side of the dialog.
2. Enter a name for the data selection in the box next to **Create new**.
3. To add data to the selection, select an item in the mismatches list and insert into the **Selected** pane.
4. To create a group, highlight the items to be grouped in the original list, right-click and choose the required grouping option.
   a. **Create group** will create one single parameter equal to sum of all those currently selected.
   b. **Create groups by boreholes**, **Create groups by properties** or **Create groups by dates** will create one group per borehole, per property or per date from the currently selected items.
   Groups appear in the list on the right and can be renamed there, as required.
5. Click **OK** to save the selections or groups and close the **Data identifier selection** dialog
6. To use a specific data selection in one of analysis methods, select the group or selection from the list in the **Select data** pane.
   When a data selection is used, only the items defined in the group are available for analysis.

Data selections are stored in the project unless you delete the selections. Groups are not created anywhere in the project and are just declared in the data selection object. They are taken into account by respective analysis methods. Data selections **do not** modify any information associated with simulation cases.

History matching and production optimization

RFT data may now be included in the history matching mismatch calculations. This is a new process that may be included in the existing objective function calculations. Note that the RFT mismatch dialog requires its observed MDT/RFT logs as Petrel time stamped point well data

Create an RFT mismatch

An RFT mismatch object contains definition of parameters and input data to be used in a workflow for calculating numeric mismatches between observed and simulated RFT.

To create an RFT mismatch object:

1. On the **Reservoir Engineering** tab, in the **History match and optimization** group, click **Define RFT mismatch**.
2. Insert a simulation case into **Base case**.

   The base case should have RFT logs associated with it. These logs are created by simulator and can be found in the **Simulation Logs** folder under **Global well logs** in the **Input** pane.
3. Insert an RFT input dataset into **Observed RFT**.

   The process accepts only one observed RFT input. Therefore, all related RFT data - different attributes (pressure, saturations) for all time steps and all wells which you want to use in calculation, should be imported to the same data object.

   Once base case and RFT input data are defined, the **Data mapping** tab is automatically populated with appropriate observed and simulated attributes. On this tab, you should decide which attributes should be used in calculation and ensure that they are matched correctly.
4. If you want to override an automatic mapping, insert the correct simulation log into the appropriate fields in the **Data mapping** tab.
5. Clear the **Use** check box next to an attribute if you want to exclude it from mismatch calculations.
6. Click the **Dates and weights** tab.

   Each row on the **Dates and weights** tab represents a property for a specific well in a specific time step from the observed object. A simulated date can be set so that the desired RFT report can be matched with the proper observed data.
7. Insert a **Boolean Global well log** into **Filter**.

   If used, the Boolean log will exclude particular points from calculations at the depths corresponding to the points to be excluded.
8. Insert a **General Global well log** into **Weight**.

   This is a continuous log and is used as a multiplier to individual mismatch points. It can be used to define the impact individual points will have on the total mismatch value
9. Define the settings for each line in the **Dates and weights** table.
   a. To exclude a line from calculations, clear the Enabled check box.
   b. Ensure the **Simulated Date** is correct. To override the auto-mapped value, select the required date from the list.
   c. Adjust the **Measurement error**.

      This sets a standard deviation for the calculation of the partial RFT mismatch between the simulated and observed data.

d. Set the **Weight**.

   This allows you to set a multiplier for the partial mismatch prior to the summation in the global RFT mismatch.

e. Select the **Apply filter** and/or **Apply weight** check boxes to apply the specified filter / weight to individual lines in the table.

   The **Base Value** shows a preview of the partial mismatch values computed using the base case data.

10. Click **OK**.

Calculate RFT mismatches in a workflow

Describes how to add a mismatch calculation of RFT data to an optimization process and how to use the RFT mismatch value in an objective function.
Before continuing, ensure that you define an RFT mismatch function outside the **Uncertainty and optimization** process using the **Define RFT mismatch** process.

To include an RFT mismatch in a workflow:

1. Open the **Uncertainty and optimization** process and select an existing uncertainty workflow.
2. On the **Base case** tab, click **Edit workflow**.
3. From the list of **Processes**, select **Define RFT mismatch** and insert it before the **Define objective function** process.

   Define RFT mismatch has to be executed once simulation has finished, so make sure it is called after **Wait for case results**.
4. Edit the **Define RFT mismatch** process.
   a. Select an existing **RFT mismatch** function.
   b. Define a numeric variable where the results of the calculation will be stored.
   c. After the process is finished, the workflow can access the RFT mismatch value using this variable.
   d. Use an **Output case (global)** workflow variable to reference the case.
5. Edit the **Define objective function** process to use include the variable defined in the **Define RFT mismatch** process.

   To combine RFT mismatch with mismatch based on observed data such as production rates or history matching optimization, select "Add item" in the **Composite function** panel in the **Define objective function** process and enter the name of the numeric mismatch variable. It is also possible to apply scale factor to harmonize the impact of RFT on the total mismatch. If there is no observed production data available, it is still possible to perform optimization on RFT mismatch. To do this, you should use a Production optimization (not History matching) Objective function and select the **User defined term** option.

   Note: It is also possible to evaluate an RFT mismatch on existing cases in the **Workflow editor**. Mismatch values can be reported to the Message log, Output spreadsheet, attached to the case as case variable, and so on.
6. Click **Apply**.

Genetic Algorithm Optimizer

The Genetic Algorithm included has stronger global search capabilities that the existing Evolution Strategy supported and is more applicable for forward-optimization problems. The new GA is capable of handling both discrete and continuous parameters, so-called mixed-integer capabilities, which is important for most Petrel Uncertainty and optimization workflows.

Mixed-integer Proxy Optimizer

The Mixed-integer Proxy Optimizer operates fairly similarly as the Neural Optimizer that has been in Petrel for some time. It is, however, based on a better performant proxy modeling technique and has true mixed-integer capabilities. This new optimizer combines full simulation runs and optimization on automatically built proxies. It starts with a set of runs and builds a proxy which is then sequentially tested and improved. This method has the potential to significantly improve optimization efficiency.

Reservoir Engineering: INTERSECT simulation

2014.1

INTERSECT case export

Our INTERSECT user feedback has guided the usability and functional enhancements to the **Define simulation case** dialog for INTERSECT cases, which now enables you to flexibly control exactly what 3D and summary results are output from your simulation runs, both speeding up export, reducing the amount of unwanted output on disk and hence speeding up results import.

We anticipate introducing a similar interface for ECLIPSE simulation in a future release.
INTERSECT export in Petrel has two different modes.

In the first mode, an E300 keyword deck is exported first, with additional files, and the SLB Migration tool is run in the background to convert this keyword deck to an IXF INTERSECT input deck. This is how Petrel has traditionally supported the INTERSECT simulator.

In Petrel 2014.1, a second mode has been added. In this mode *.IXF and grid data files are written directly from Petrel. This export mode is faster and produces a cleaner input model for the INTERSECT simulator than the traditional method.

The type of export is chosen dynamically based on features of the case. Currently the following types of cases will default to using the older traditional form of export:

- Targeting INTERSECT 2013.1 or lower
- Using development strategies
- Dual porosity
- Dual permeability
- Containing unstructured local grid refinements

- Containing adsorption functions

Flexible reporting from INTERSECT cases

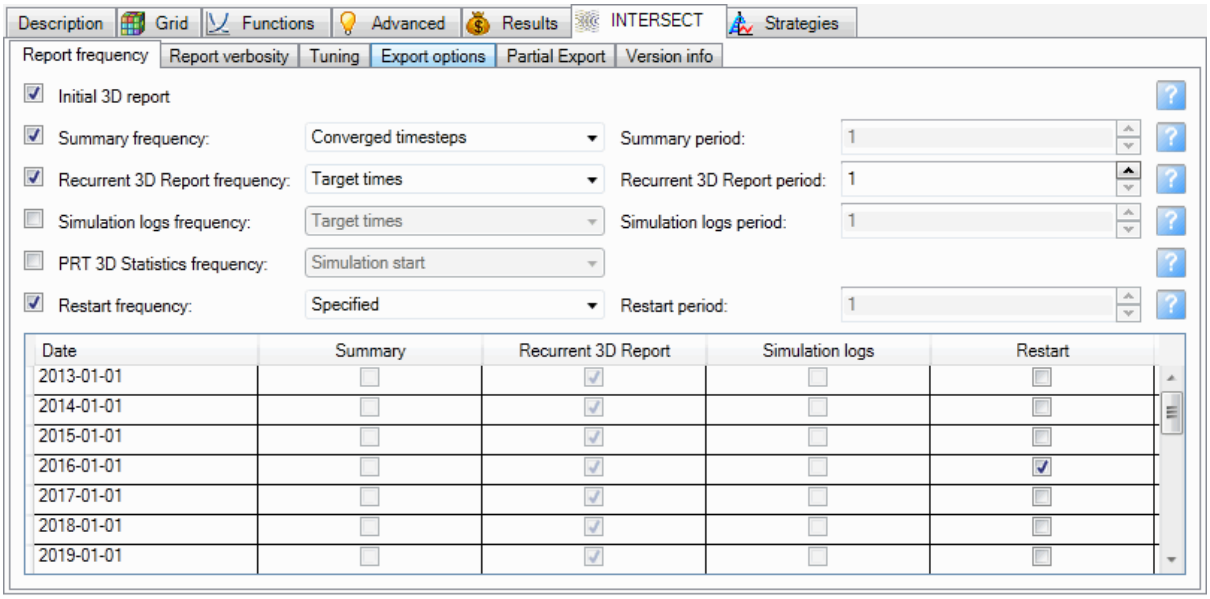
The **Results** tab for INTERSECT simulation cases now comprises a series of sub-tabs that allow you to more easily select the outputs for an INTERSECT case.

Report frequency

The new **Report frequency** tab allows you have a generic control on all the simulation reports, on whether you need a specific report (like INTERSECT restart) or not, and what frequency you need. Most of the report types can have a different frequency from each other. Typically, you would like a high frequency for summary result, lower frequency for recurrent 3D report for 3D visualization, and restart report for only the last step.

For INTERSECT cases, greater flexibility is given to allow you to control the output and frequency of the reports generated by the simulator.

From the **Report frequency** tab you can select which reports, statistics and logs to generate from the simulation. You can choose to generate INIT and recurring 3D grid property reports, 3D property statistics to the print file, summary and simulation logs. For cases using Field Management strategies you can also control the frequency of the simulator generate restart files.



Summary vectors

With this new **Summary vectors** selection tab, all the summary vectors you selected will be listed in the selection tree, which makes the selection much more transparent than before and there is a complete coverage on the summary properties available for INTERSECT cases. You get the flexibility to select or delete any summary you need for field, group, well, region, aquifer, while before the summary vectors supported and the control given you to change it was limited in Petrel. A default summary vector can automatically be set up for the INTERSECT simulation case, and you can see exactly which vectors are included in this default selection.

Select summary vectors for an INTERSECT case

Use identifier and property selections to define the result data that is output when an INTERSECT simulation case is run.

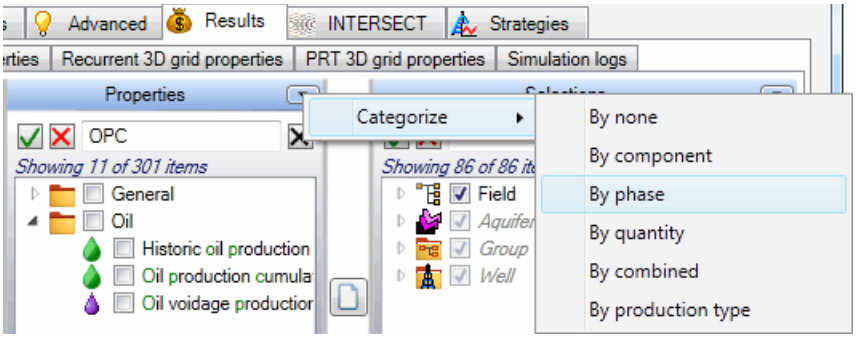
The time-based results that you select here will appear in the **Dynamic results data** folder on the **Results** pane. You can also use the property selection procedures described here to select the output properties on the **Initial 3D grid properties**, **Recurrent 3D grid properties** and **PRT 3D grid properties** tabs.

To select summary vectors:

1. Open the **Define simulation case** process and create a new INTERSECT case or choose an existing case to edit.
2. Click the **Results** tab.
3. To create a list of summary vectors, select the **Identifiers** and **Properties** and then generate **Selections**.
4. (Optional) Click to clear the current selections and generate a number of default selections from your simulation case, then use steps below to modify the selections.
5. In the **Identifier** pane on the **Summary Vectors** tab, select the required identifiers. Click to select all identifiers or click to clear all selections.

   Different identifiers appear in the **Identifiers** pane based on other simulation case input. For example, All Well segment and All Device appear if a well segmentation set is defined on the **Strategies** tab.
   Note: When you select identifiers, incompatible property options are disabled in the **Properties** pane.
6. In the **Properties** pane, select the required properties.

   You can categorize properties to make selection easier. To do this, click the **Properties** drop-down button, click **Categorize**, and then click the required category.



   Tip: Use the search box to locate the required properties. For example, type OPC to locate all properties containing 'opc', such as oil production cumulative.
7. Click to populate the **Selections** pane with the chosen identifier and properties.

   The time-based results that you select here appear in the **Dynamic results data** folder in the **Results** pane when the simulation case is run.

   Note: Selections can also be categorized. To do this, click the **Selections** drop-down button, point to **Categorize** and select the required category.
8. Click **Apply** to save your selections.

Supported identifiers

The following identifiers are supported: All Aquifer (*), All Device (*), All Group (*), All Heater (*), All Region (*), All Well (*), All Well segment (*), Field, Aquifer, Group, Region and Well.

Note: The All (*) identifiers provide a convenient way to select all identifiers of a particular type. This is not equivalent to selecting all of the same type of identifier; the identifiers corresponding to the chosen type are disabled. For example, if the 'All Wells (*)' identifier is selected, 'Well' is grayed out in the **Identifiers** pane and a wildcard '*', rather than well names is exported in the simulation case. This improves efficiency when exporting a large number of summary vectors from an INTERSECT case.

Initial 3D grid properties, recurrent 3D grid properties, and PRT 3D grid properties

The separate recurrent 3D and PRT 3D properties tabs enable you to select exactly those 3D properties that you are interested in reporting. For a large model being able to deselect the non-essential ones saves much disk space and simulation time.
Use property selections on these tabs to specify initial and recurrent grid properties and to define which grid properties you want to appear in the print file output. Make your selections from the **Properties** pane and click  to populate the **Selections** pane. A search box is available to locate the required properties and both properties and selections can be categorized.

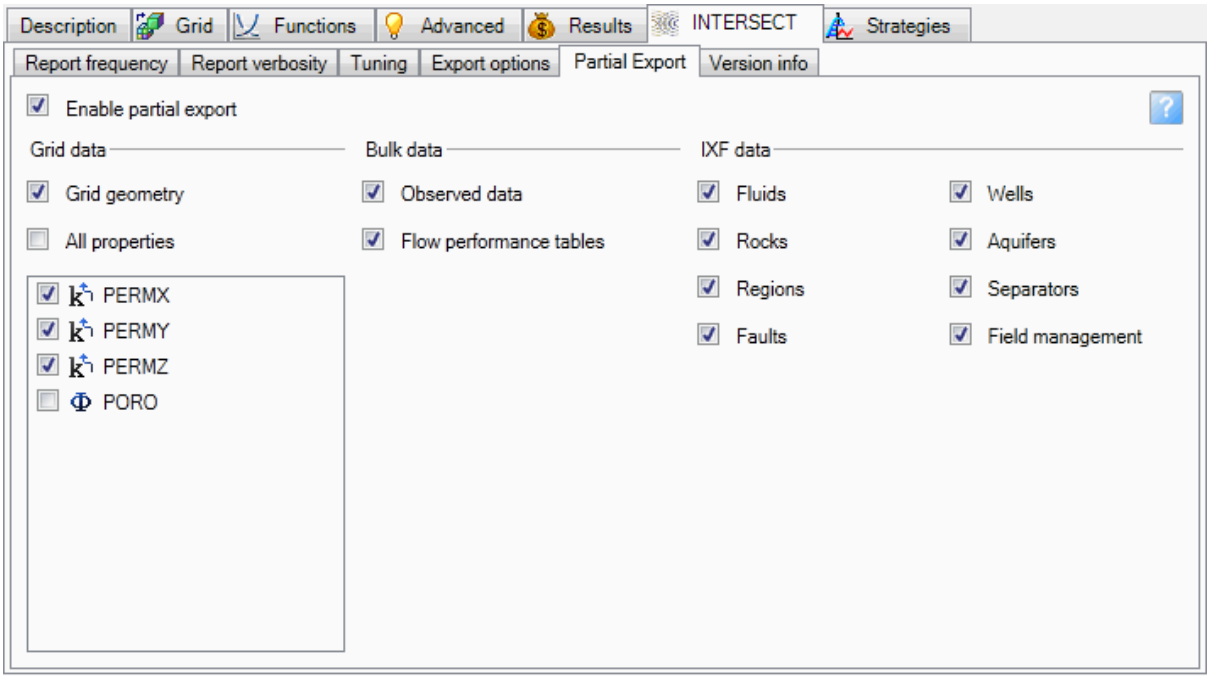
Simulation logs

Simulation logs are generated by the simulator when you select one or more of the **Well RFT**, **Well PLT**, **Well segment** and **Burden temperature** options on the **Simulation logs** tab. Recording the burden temperature output from numerical heat loss models in the under/overburden areas, is only valid for thermal cases.

Partial export of data from an INTERSECT case

Exporting simulation cases for large models can be very time-consuming, but you often make specific modifications to certain parts of the model that do not require the entire model to be re-exported. To speed up simulation case export, you may choose to suppress re-export of specific elements in the simulation case (for example, grid, 3D properties, observed data, Field Management strategies, well definitions or other IXF files) that have not changed in the Petrel model since the previous export. Turning off items means that they are not recalculated or exported; instead, the files from the previous export remain unchanged.

This option should be used with extreme care – if you disable parts of the export, you may create inconsistent data sets.



Note: This option is not available for all types of INTERSECT export, only those using Field Management strategies can allow partial export as these do not require the Migrator to run on an exported ECLIPSE keyword deck.

Partial export is disabled and all the check boxes are all selected by default, meaning that the full data set will be exported. Enabling partial export and clearing any a check box will prevent the identified part of the model from being exported.

Imported case workflows

The support for imported case workflows, that is those where the initial creation (or migration) of the INTERSECT case has been performed outside of the Petrel environment, has been greatly enhanced. The support requires the latest INTERSECT 2013.2 format files, the “grid edit” format GSG files for the grid geometry.

With these, the INTERSECT input and results files can be imported directly into Petrel 2014.1 as an imported INTERSECT case, including in 2014.1 the 3D grid geometry and3D results. This removes the need for the user to generate an EGRID file during the migration step when preparing the INTERSECT simulation case.

The imported INTERSECT cases can be copied and edited, and the edited copies run directly from Petrel. The imported case can be used within the **Uncertainty and optimization** process, where the user can variabilize the parameters within the IXF files for use within sensitivity or optimization.

Exploration Geology

Exploration Geology: Petroleum Systems

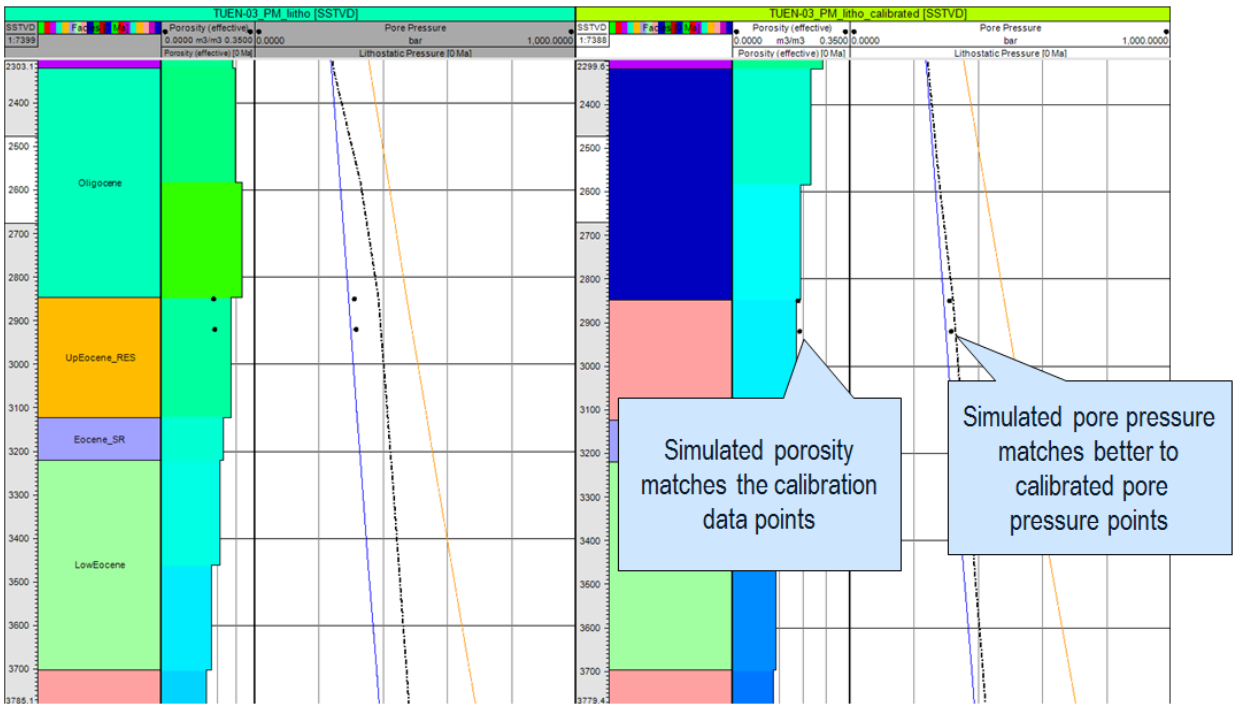
2014.1

Calibration using petroleum systems lithology editor

Petroleum systems modeling

Calibrate the parameters for the mechanical compaction, permeability, and seal properties by modifying the lithology settings in the Lithology editor of the Settings box of an individual lithology and then comparing the results of a 1D simulation with the measured data.

Figure 119: Compare the results of simulated and measured porosity and pore pressure data after calibration using a Petroleum systems 1D simulation



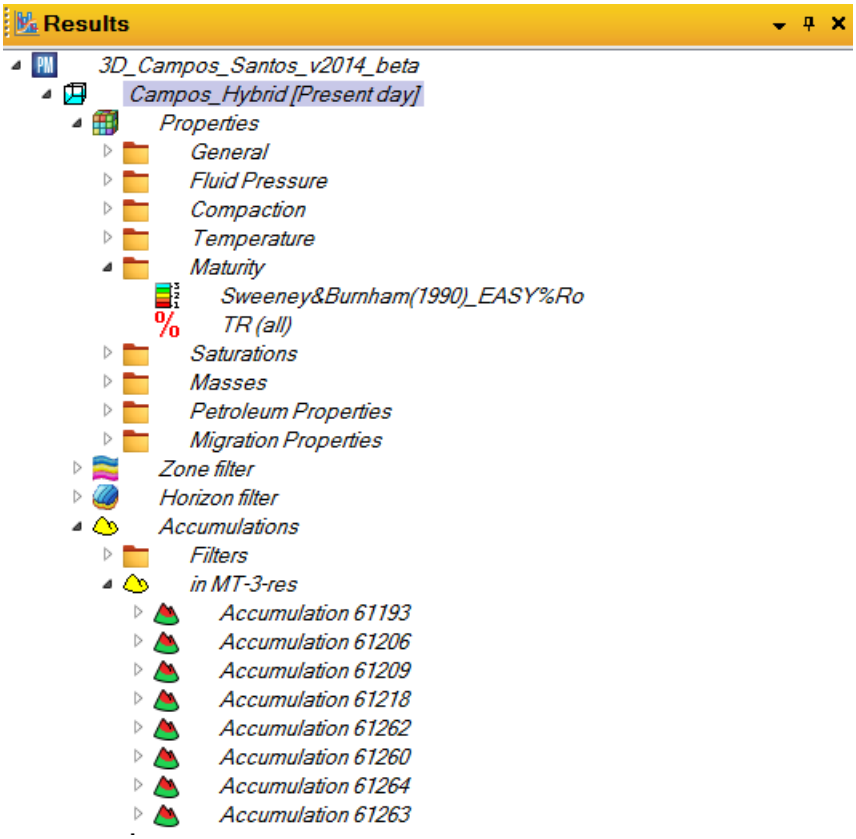
Increased usability of 3D petroleum systems model simulation results

Petroleum systems modeling

You can now access petroleum systems simulation results directly on the Results pane of Petrel – this significantly reduces the amount of mouse movement and clicking.

When you connect your Petrel project to a PetroMod project, you have all 3D model data available on the Results pane. Without having to pre-select certain properties, you can simply load all properties calculated during simulation. The speed of data access and of switching between the results at different periods has been greatly improved. Your general Reservoir core, Geoscience core or Combined core licenses are sufficient to access PetroMod simulation results in Petrel, you do not need a specific Exploration Geology license.

Figure 120: Access to the results of a Petroleum systems 3D simulation on the Results pane

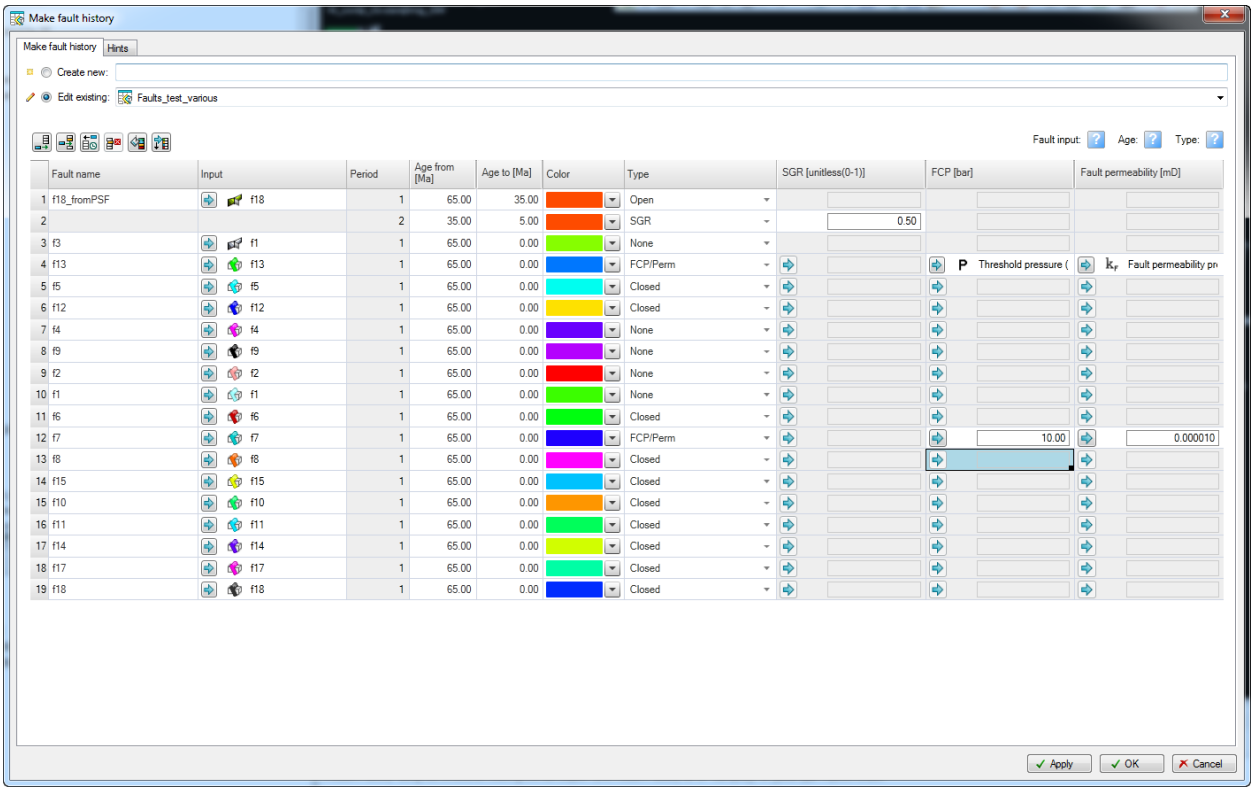


Fault data for Petroleum Systems

You can now collect fault data from structural models and interpretations and transfer this information to a petroleum systems simulation.

The fault data can contain geometry and property information like shale gauge ratio or fault capillary pressure derived from structural fault analysis workflows in Petrel. In the workflow this data is combined with geologic age information. Fault data can now be added to a petroleum systems model and transferred to PetroMod together with a simple grid.

Figure 121: Fault history process



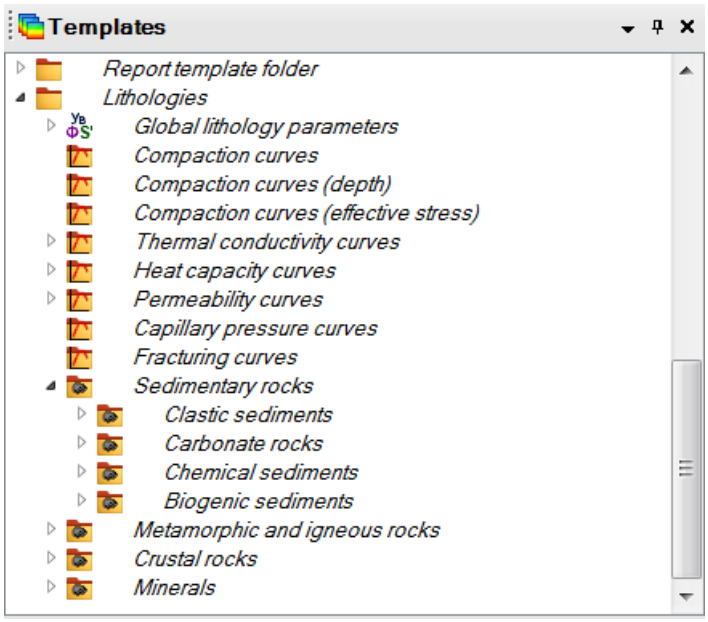
Lithology editor

Petroleum systems modeling

Manage and create custom lithologies directly in Petrel and use them for petroleum systems modeling.

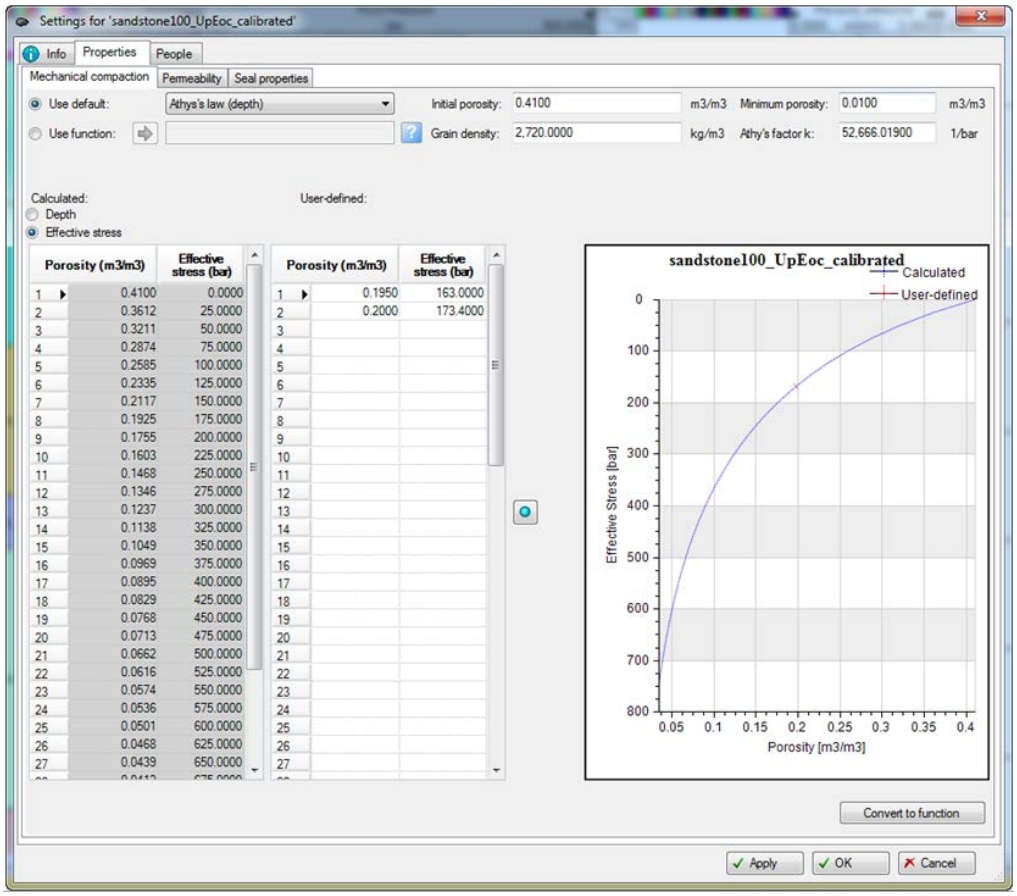
All Petrel lithologies can be accessed on the Templates pane in Petrel. More than 150 default lithologies are available. Open once the Facies table process to add the lithologies to the Templates pane.

Figure 122: Standard lithologies on the Templates pane



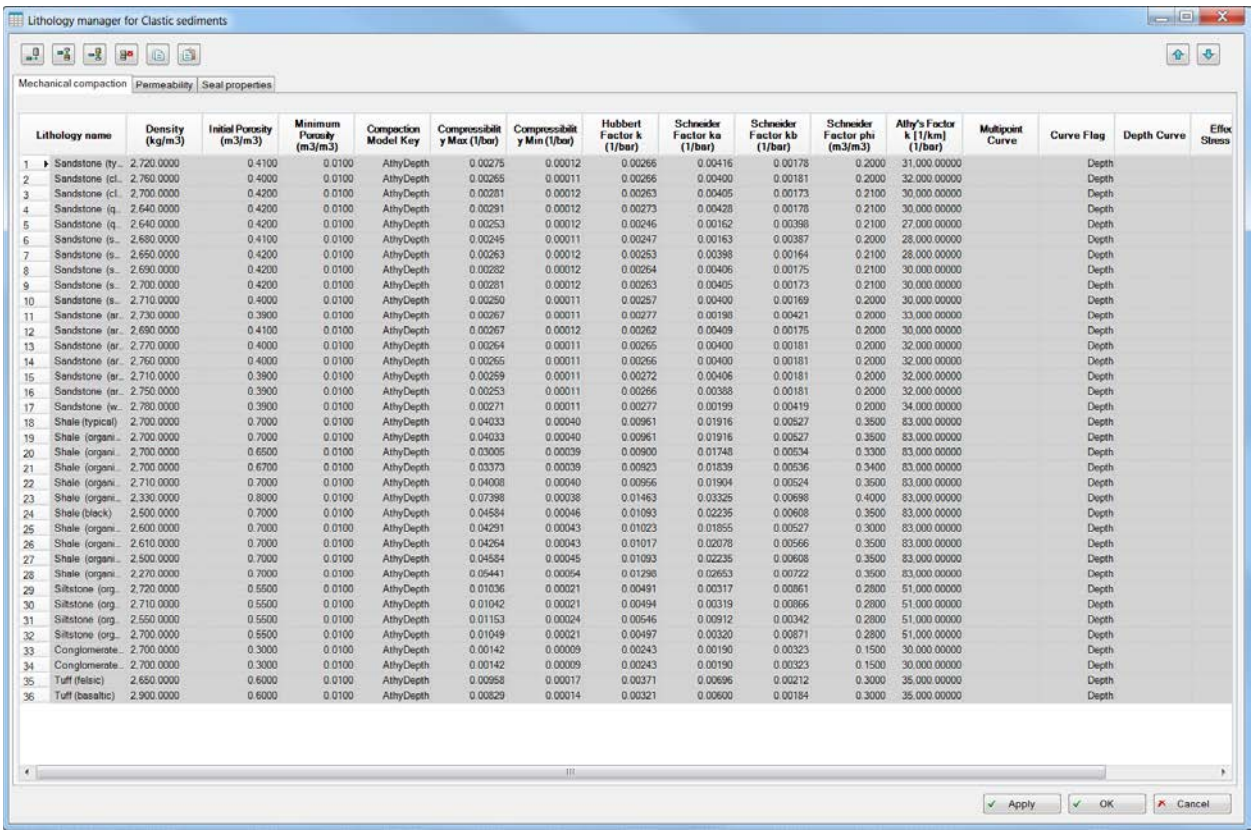
Lithologies created in Petrel can be merged with a PetroMod project. User-defined lithologies can be generated by copying default lithologies, mixing lithologies from existing or defining them from scratch based on your data. Compaction-related properties of the lithologies can now be modified for user-defined lithologies in a lithology editor and a lithology manager.

Figure 123: Modify an existing lithology in the Lithology editor



This editor can be accessed via the Settings box of each individual lithology. Modifying the compaction behavior can be used to calibrate the pressure calculation of petroleum systems models in Petroleum Systems 1D and 3D.

Figure 124: Access the settings of all lithologies of a group in the Lithology manager



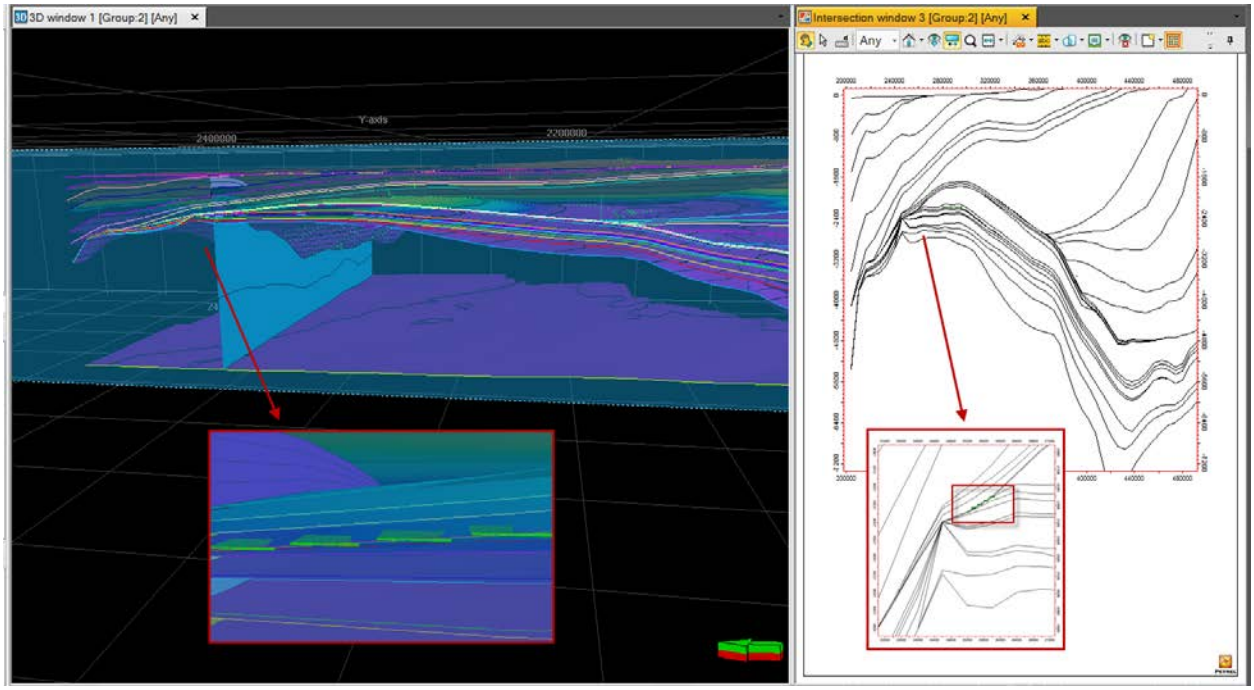
	Lithology name	Density (kg/m3)	Initial Porosity (m3/m3)	Minimum Porosity (m3/m3)	Compaction Model Key	Compressibility Max (1/bar)	Compressibility Min (1/bar)	Hubbert Factor k (1/bar)	Schneider Factor ka (1/bar)	Schneider Factor kb (1/bar)	Schneider Factor phi (m3/m3)	Athy's Factor k [1/km] (1/bar)	Multipoint Curve	Curve Flag	Depth Curve	Effe Stress
1	Sandstone (ty...	2,720.0000	0.4100	0.0100	AthyDepth	0.00275	0.00012	0.00266	0.00416	0.00178	0.2000	31,000.00000		Depth		
2	Sandstone (cl...	2,760.0000	0.4000	0.0100	AthyDepth	0.00265	0.00011	0.00266	0.00400	0.00181	0.2000	32,000.00000		Depth		
3	Sandstone (cl...	2,700.0000	0.4200	0.0100	AthyDepth	0.00281	0.00012	0.00263	0.00405	0.00173	0.2100	30,000.00000		Depth		
4	Sandstone (q...	2,640.0000	0.4200	0.0100	AthyDepth	0.00291	0.00012	0.00273	0.00428	0.00178	0.2100	30,000.00000		Depth		
5	Sandstone (q...	2,640.0000	0.4200	0.0100	AthyDepth	0.00253	0.00012	0.00246	0.00162	0.00398	0.2100	27,000.00000		Depth		
6	Sandstone (s...	2,680.0000	0.4100	0.0100	AthyDepth	0.00245	0.00011	0.00247	0.00163	0.00387	0.2000	28,000.00000		Depth		
7	Sandstone (s...	2,650.0000	0.4200	0.0100	AthyDepth	0.00263	0.00012	0.00253	0.00398	0.00164	0.2100	28,000.00000		Depth		
8	Sandstone (s...	2,690.0000	0.4200	0.0100	AthyDepth	0.00282	0.00012	0.00264	0.00406	0.00175	0.2100	30,000.00000		Depth		
9	Sandstone (s...	2,700.0000	0.4200	0.0100	AthyDepth	0.00281	0.00012	0.00263	0.00405	0.00173	0.2100	30,000.00000		Depth		
10	Sandstone (s...	2,710.0000	0.4000	0.0100	AthyDepth	0.00250	0.00011	0.00257	0.00400	0.00169	0.2000	30,000.00000		Depth		
11	Sandstone (ar...	2,730.0000	0.3900	0.0100	AthyDepth	0.00267	0.00011	0.00277	0.00198	0.00421	0.2000	33,000.00000		Depth		
12	Sandstone (ar...	2,690.0000	0.4100	0.0100	AthyDepth	0.00267	0.00012	0.00262	0.00409	0.00175	0.2000	30,000.00000		Depth		
13	Sandstone (ar...	2,770.0000	0.4000	0.0100	AthyDepth	0.00264	0.00011	0.00265	0.00400	0.00181	0.2000	32,000.00000		Depth		
14	Sandstone (ar...	2,760.0000	0.4000	0.0100	AthyDepth	0.00265	0.00011	0.00266	0.00400	0.00181	0.2000	32,000.00000		Depth		
15	Sandstone (ar...	2,710.0000	0.3900	0.0100	AthyDepth	0.00259	0.00011	0.00272	0.00406	0.00181	0.2000	32,000.00000		Depth		
16	Sandstone (ar...	2,750.0000	0.3900	0.0100	AthyDepth	0.00253	0.00011	0.00266	0.00388	0.00181	0.2000	32,000.00000		Depth		
17	Sandstone (w...	2,780.0000	0.3900	0.0100	AthyDepth	0.00271	0.00011	0.00277	0.00199	0.00419	0.2000	34,000.00000		Depth		
18	Shale (typical)	2,700.0000	0.7000	0.0100	AthyDepth	0.04033	0.00040	0.00961	0.01916	0.00527	0.3500	83,000.00000		Depth		
19	Shale (organi...	2,700.0000	0.7000	0.0100	AthyDepth	0.04033	0.00040	0.00961	0.01916	0.00527	0.3500	83,000.00000		Depth		
20	Shale (organi...	2,700.0000	0.6500	0.0100	AthyDepth	0.03005	0.00039	0.00900	0.01748	0.00534	0.3300	83,000.00000		Depth		
21	Shale (organi...	2,700.0000	0.6700	0.0100	AthyDepth	0.03373	0.00039	0.00923	0.01839	0.00536	0.3400	83,000.00000		Depth		
22	Shale (organi...	2,710.0000	0.7000	0.0100	AthyDepth	0.04008	0.00040	0.00956	0.01904	0.00524	0.3500	83,000.00000		Depth		
23	Shale (organi...	2,330.0000	0.8000	0.0100	AthyDepth	0.07398	0.00038	0.01463	0.03325	0.00698	0.4000	83,000.00000		Depth		
24	Shale (black)	2,500.0000	0.7000	0.0100	AthyDepth	0.04584	0.00046	0.01093	0.02235	0.00608	0.3500	83,000.00000		Depth		
25	Shale (organi...	2,600.0000	0.7000	0.0100	AthyDepth	0.04291	0.00043	0.01023	0.01855	0.00527	0.3000	83,000.00000		Depth		
26	Shale (organi...	2,610.0000	0.7000	0.0100	AthyDepth	0.04264	0.00043	0.01017	0.02078	0.00566	0.3500	83,000.00000		Depth		
27	Shale (organi...	2,500.0000	0.7000	0.0100	AthyDepth	0.04584	0.00045	0.01093	0.02235	0.00608	0.3500	83,000.00000		Depth		
28	Shale (organi...	2,270.0000	0.7000	0.0100	AthyDepth	0.05441	0.00054	0.01298	0.02653	0.00722	0.3500	83,000.00000		Depth		
29	Siltstone (org...	2,720.0000	0.5500	0.0100	AthyDepth	0.01036	0.00021	0.00491	0.00317	0.00861	0.2800	51,000.00000		Depth		
30	Siltstone (org...	2,710.0000	0.5500	0.0100	AthyDepth	0.01042	0.00021	0.00494	0.00319	0.00866	0.2800	51,000.00000		Depth		
31	Siltstone (org...	2,550.0000	0.5500	0.0100	AthyDepth	0.01153	0.00024	0.00546	0.00912	0.00342	0.2800	51,000.00000		Depth		
32	Siltstone (org...	2,700.0000	0.5500	0.0100	AthyDepth	0.01049	0.00021	0.00497	0.00320	0.00871	0.2800	51,000.00000		Depth		
33	Conglomerate...	2,700.0000	0.3000	0.0100	AthyDepth	0.00142	0.00009	0.00243	0.00190	0.00323	0.1500	30,000.00000		Depth		
34	Conglomerate...	2,700.0000	0.3000	0.0100	AthyDepth	0.00142	0.00009	0.00243	0.00190	0.00323	0.1500	30,000.00000		Depth		
35	Tuff (felsic)	2,650.0000	0.6000	0.0100	AthyDepth	0.00958	0.00017	0.00371	0.00696	0.00212	0.3000	35,000.00000		Depth		
36	Tuff (basaltic)	2,900.0000	0.6000	0.0100	AthyDepth	0.00829	0.00014	0.00321	0.00600	0.00184	0.3000	35,000.00000		Depth		

Invasion percolation migration results

You now have access in Petrel to invasion percolation migration results from PetroMod.

Invasion percolation is a fast migration simulation method which uses capillary entry pressure to distribute fluids in a petroleum systems simulation. This method is predominantly used for structurally complex models and models including faults.

Figure 125: Invasion percolation results in Petrel



Geotime window improvements

The Geotime window was improved in performance and visualization capabilities.

You have now improved zooming capabilities, e.g. synchronized zooming over several charts, zooming and panning in horizontal and vertical direction.

Drilling

Drilling: General

2014.1

Relief well path template

You can now quickly create a relief well plan with the **Relief well path** template.

A relief well is a directional well, designed to intersect and communicate with a blowout well in order to kill the uncontrolled flow or conduct abandonment operation.

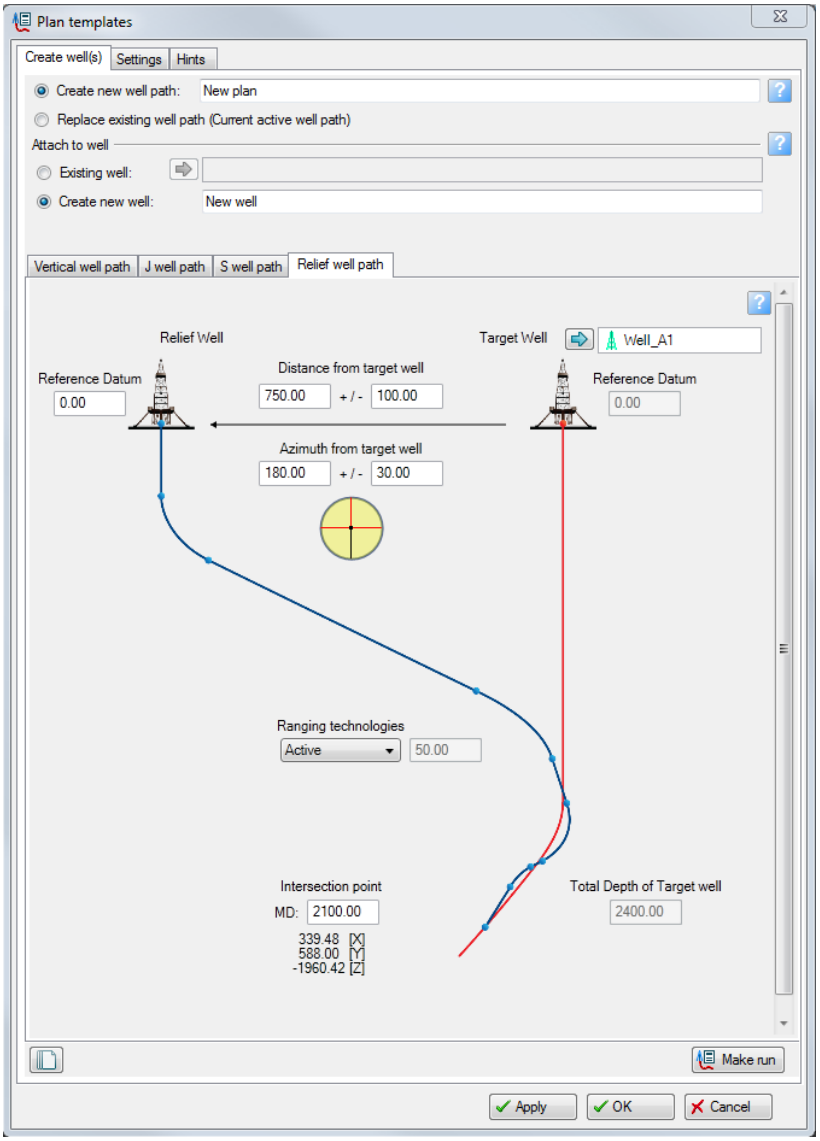
Relief well planning may be required for any well operations that will enter the pressure envelope of a well that is capable of natural flow.

The relief well path template allows you to design a well plan intersecting another target well.

To access the relief well path template, do one of the following:

- In Ribbon mode, on the **Well design** tab, in the **Well path** group, click **Well templates**.
- In Classic mode, double-click **Well path design (new)** in the **Processes** pane.

The **Well templates** or **Well path design (new)** dialog box will appear. In this dialog box, you can design the relief well plan on the **Relief well path** tab.



On the relief well path template, you can select or input the following:

- Options to create a new well path plan or replace the existing well path plan.
- Options to create a new well or attach the new plan to an existing well.
- Target well to be intersected: you can select it in the **Input** pane or **3D** window and add it to the **Target well** field on the **Relief well path** tab.
- Preferred distance and allowable range between the target well and the relief well.
- Preferred azimuth and allowable range from the target well to the relief well.
- Reference datum of the relief well.
- Depth of the intersection point in the target well.
  In most circumstances the preferred interception point should be at the top of the reservoir where the blowout well penetrates the flowing reservoir. For all relief wells, it is essential that there is ferrous material (casing or drill string) at the point of interception as this is required for the ranging tools to function. Generally, the first approach point should have casing or drill pipe extending for at least 300m above the intersection point. In Petrel, the MD of the intersection point must be at least 600m.

- Ranging technology which will be used during drill the relief well.
  The ranging instruments are used to search and detect the target blowout well while drilling. It can determine the position of a nearby well relative to the position of the well being drilled. There are two types of instruments available as options: **Active** and **Passive**. You can also select the **Custom** option and input a range for the ranging tool.

After input of all the design data, click **Make run**. The relief well will be created in the **Input** pane. You can edit the relief well with Well path designer or Inspector.

In the following cases, Petrel might fail to compute out a relief well trajectory with your input.
- The MD of the intersection point is less than 600m.
- The inclination of the target well between the intersection point and ranging point is more than 60 deg.
- The DLS of the target well between the intersection point and ranging point is too high. It is recommended to select an intersection point in the smooth section. For example, the average DLS between the intersection point and ranging point is less than 1deg/30m.
- The relief well location is too far away from the target well.

When Petrel cannot generate the relief well, a warning message will appear. This message will suggest that you edit the intersection point or the distance range between the target well and the relief well.

Trajectory uncertainty

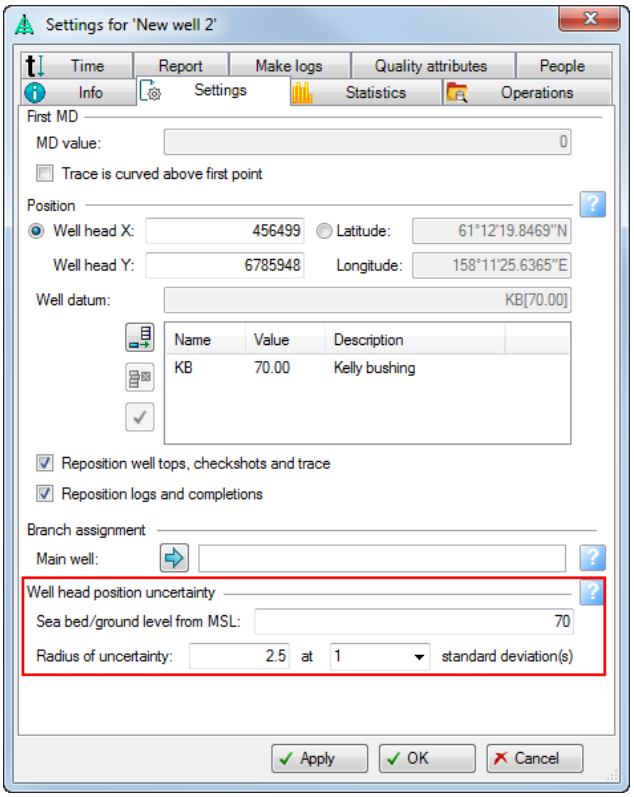
You can now define the survey program and calculate the Ellipsoid of uncertainty (EOU) for a plan or survey.

Whenever a well is surveyed, measurement errors from a variety of sources introduce some degree of uncertainty about the actual position of the borehole.

The **Trajectory uncertainty** function allows you to define the survey program for a plan or a survey, and then the EOU will be calculated and created for this trajectory.

Well head position uncertainty
Before calculating the well uncertainty or analyzing well anti-collision, now you can set the well head position uncertainty in the well settings. If this information is not set, default values will be used that may affect the accuracy of the well positioning calculation.

Double-click the well in the Input pane. The **Settings** dialog box for this well appears.



Go to the **Settings** tab. Input the following for well head position uncertainty:

- **Sea bed/ground level from MSL**: The vertical offset to the sea bed or ground level from the mean sea level (MSL) at the well head location. Positive value indicates sea bed or ground level is above the MSL. The EOU calculation will start from the sea bed/ground level.
- **Radius of uncertainty**: The well head position uncertainty expressed by radius at specified confidence level. This uncertainty includes the structure uncertainty of the offshore platform location and the slot uncertainty (usually the slot radius).
- **Standard deviation**: The confidence level of the radius uncertainty. This value indicates how much variation or dispersion from the average exists. A low standard deviation indicates that the data points tend to be very close to the mean; a high standard deviation indicates that the data points are spread out over a large range of values.

EOU

Ellipsoids of uncertainty (EOUs) were originally proposed by Wolff and deWardt to describe the uncertainty in a wellbore position arising from asymmetric systematic errors (SPE 9223).

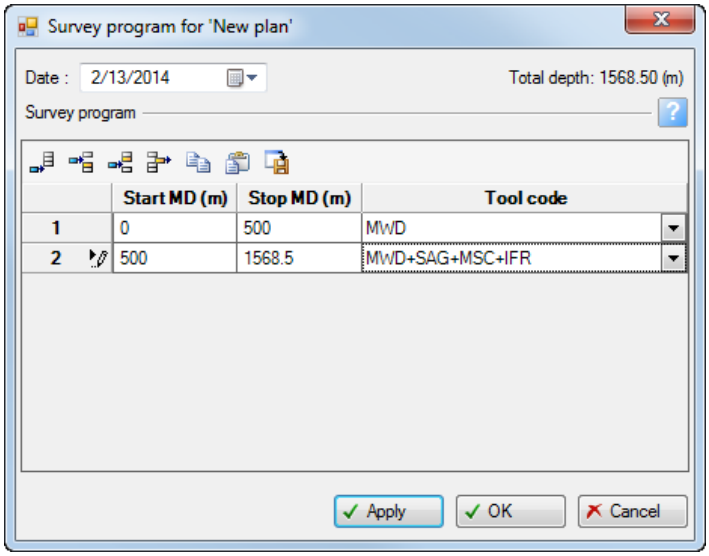
The EOUs of Wolff and deWardt were ellipsoids projected onto the horizontal plane, plotted on positional maps (Top view maps) to depict the relative positional uncertainties between wells. The vertical section could represent the projection of the ellipsoid on the vertical plane.

The Industry Steering Committee for Wellbore Survey Accuracy (ISCWSA) has built a survey instrument error model for solid-state magnetic instruments (e.g., MWD, W&DEW, & EMS). The model is based on a paper published by H. Williamson "Accuracy Prediction for Directional MWD" (SPE 56702). The model vastly extends the work started with the systematic error model and incorporates the experience of the many participating parties.

The extensible ISCWSA error modeling has configurable error terms and weighting functions.
In Petrel, ISCWSA model is used to account for the survey tool errors. To calculate the EOU, you must select the correct tool code which has been assigned the predefined parameters for the survey tool based on the information from the tool manufacturer.

After calculating the EOU, you can display the uncertainty in an ellipsoid or a volume around a well that is built by connecting all the EOUs together.

Survey program
The survey program is used to calculate the Ellipsoid of uncertainty (EOU) and Volume of uncertainty (VOU) of the well trajectory. The survey program for a survey or a plan is the sequence of the survey tools used to measure the well path trajectory for each survey section. For plan, select the survey tools to be run. As drilling proceeds, surveys are taken of the new hole section and sections of well may be re-surveyed using more accurate survey tools.



In the survey program, you can input the date for plan or survey. In the Survey program table, you can define the depth range and survey tool for each survey section.

1. **Date:** Petrel will select the geomagnetic model based on the input date. The geomagnetic models are built with data from worldwide observatories, satellite missions, and surveys from land, sea, and air. Because the data is constantly changing, the magnetic models are updated periodically, for example once per year.
   - For survey, input the date of the survey or drilling. If there are a few survey dates while drilling, use the last date.
   - For plan, input the design date.
   - The earliest date you can input is Jan 1, 1900, and the latest date is Dec 31, 2014.

2. **StartMD**: The top depth of the survey section.

3. **StopMD**: The bottom depth of the survey section.

4. **ToolCode**: Select the survey instrument used to measure this survey section from the list. Each tool code has been assigned the predefined error model based on the information from the tool manufacturer. The following predefined survey tool codes are available:

   - **MWD**: The standard ISCWSA error code for Measurement While Drilling (MWD) tools without any correction technologies.
     MWD is a widely used magnetic survey instrument in the industry. It is run inside a nonmagnetic drill collar as part of the BHA during drilling operations. It measures the inclination and azimuth (Magnetic North reference) at the specific measurement depth of the well. The measurements are taken in downhole and transmitted to the surface as mud pressure pulses for real-time survey measurements. These pressures may be positive, negative or continuous sine waves.

     The MWD tools which measure formation parameters (for example, gamma ray, resistivity, porosity, and so on) are referred to as Logging While Drilling (LWD) tools.

   - **MWD+SAG**: The tool error code for MWD tools with SAG correction.
     In surveying, SAG (typically capitalized) refers to the mechanical misalignment between the survey package and the borehole axis in the vertical plane due to gravity and borehole curvature. The magnitude of SAG depends on BHA type, stiffness, geometry, stabilizer position, sensor spacing, hole size, mud weight and other factors.

     SAG correction can improve the inclination accuracy in measurement, and directly related to TVD error. On average, non-calculated SAG can create up to 80% of TVD errors.

   - **MWD+SAG+MSC+IFR**: The standard Schlumberger MWD tool error code for MWD tools with SAG, MSC, and IFR corrections.
     Multi-station correction (MSC) is a technique for dealing with drilling magnetic interference to MWD. Drillstring magnetism is a cause of errors in magnetometer readings in MWD. MSC can improve the azimuth accuracy in measurement.

     In-Field Referencing (IFR) is a technique for reduction in the uncertainty associated with the estimates of geomagnetic field values, like magnetic declination, dip angle and total strength, which generally come from geomagnetic model. IFR performs a magnetic survey closer to the earth, which can be done using a ship in the sea or an airplane, flying or sailing inside a small grid area. This survey allows a local magnetic map of the crustal field to be built, which can improve knowledge about magnetism for a particular area and indirectly improve the azimuth accuracy of MWD.
     This is the state of the art MWD tool error code in the industry that represents the most accurate measure of MWD.

   - **MWDGyro**: The preliminary tool error code for the Gyroscope MWD tool.

     Note: It can be used in inclination up to 70 degrees and low latitude. If the Gyroscope MWD is used out of application restrictions, large measurement error might be generated.

     Gyroscope MWD is a gyroscope survey tool running as part of the BHA during drilling operations, which measures the inclination and azimuth (True North reference) at the specific measurement depth of the well. Gyroscope uses the geographic North Pole as the azimuth reference and is not interfered by the magnetic objects around it. Therefore, it has a higher accuracy than magnetic MWD, but more expensive.

     The application environment of the gyroscope survey instrument varies from model to model. Mostly it is used in low latitudes and low inclination.

     Further information of a specific model is recommended to obtain from the gyro vendors.

- **Continuous Gyro**: The preliminary tool error code for Continuous NorthSeeking Gyroscope tool. Continuous NorthSeeking Gyroscope (CNSG) is a widely used survey tool in the industry, which measures the inclination and azimuth (True North reference) at the specific measurement depth of the well. It is a high-accuracy northseeking gyroscope, which is run with wireline cable in a cased hole. CNGS can always be used to recheck the MWD measurement and cased borehole re-entry.

  Further information of a specific model is recommended to obtain from the gyro vendors.

5. **Apply/OK:** After clicking this button, a survey program and an EOU will be created under the selected plan or survey in the **Input** pane.

   Note:
   - In the Survey program table, if there is depth overlap between the sections in the sequence table,
     - For survey, the later tool error code will be used for EOU calculation.
     - For plan, the earlier tool error code will be used for EOU calculation.
   - If the TVD of the well does not start from 0, you cannot directly create survey program and EOU for it in Petrel. For example, you cannot create survey program and EOU for a sidetrack wellbore.

   The workaround is to manually combine the survey data of the parent well and the sidetrack well into one well and create the survey data which starts from 0.

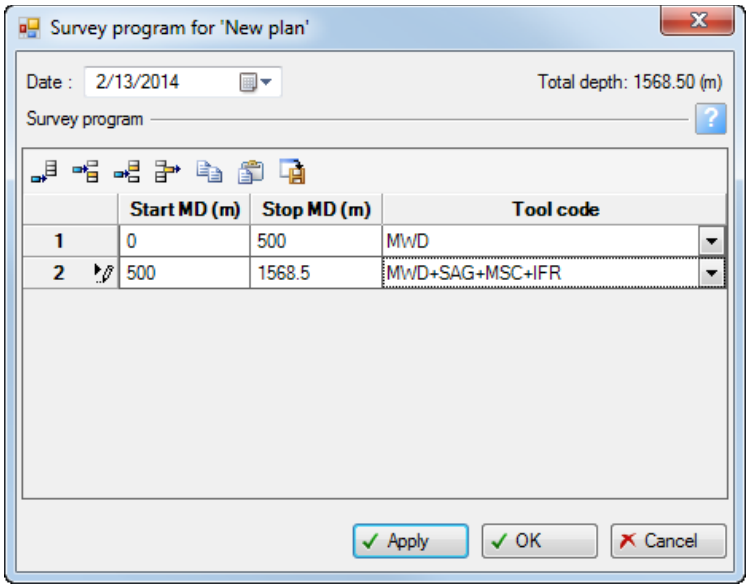
   - If the start point of the plan trajectory is NOT at the same XYZ location as the well head, you cannot create survey program and EOU for it in Petrel.

   The workaround is to create a new well using the start point of the trajectory as the well head location, and then copy the plan to the new well.

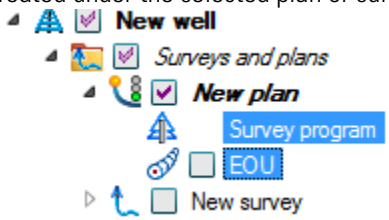
Creating survey program and EOU

You can define the survey program for a plan or a survey. After a survey program is created, the EOU for the plan or survey will also be created.

1. Select a plan or survey in the **Input** pane.
2. Open the **Survey program** dialog box. To do it, do one of the following
   - If you are using the Petrel ribbon mode, on the **Well design** tab, in the **Well positioning** group, click **Trajectory uncertainty**.
   - If you are using the Petrel classic mode, click **Borehole uncertainty** on the toolbar.



3. In the **Date** field, input the date for the survey or the date for the plan.
4. In the Survey program table, define the depth range and survey tool for each survey section.
5. Click **Apply** or **OK**.
   A survey program and an EOU are created under the selected plan or survey in the Input pane.



Showing EOU and VOU

You can now show the Ellipsoid of uncertainty (EOU) and Volume of uncertainty (VOU) for the selected surveys, plans, and wells in 3D windows.

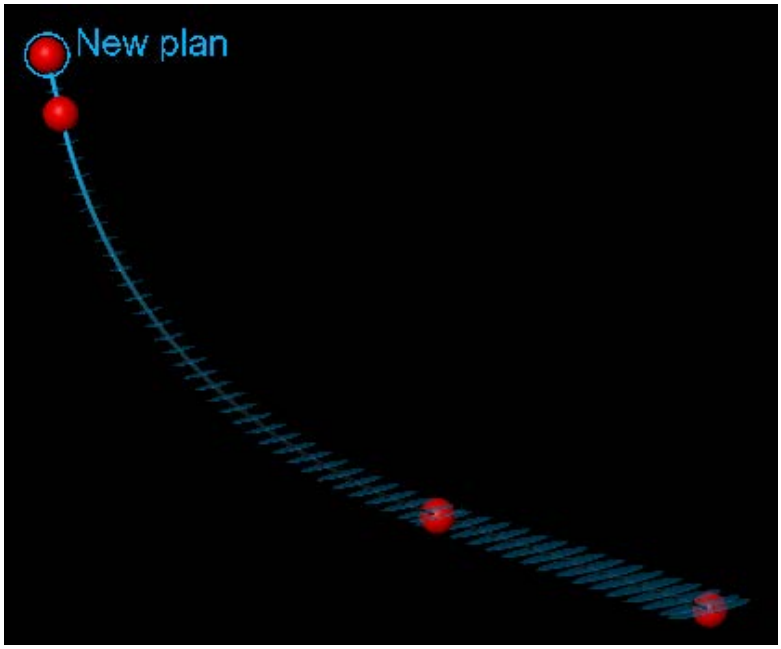
After the survey programs are created for the plans or surveys, the EOU and VOU can be displayed in 3D windows. EOU uses the ellipsoid volume to indicate the magnitude of the wellbore position uncertainty at a particular depth. VOU shows the volume around a well that is built by connecting all the EOUs together.

To display EOU:

1. Activate the 3D window.
2. In the **Input** pane, click the EOU object, or the plan, survey or well with EOU object. You can also select multiple objects together.
   Note: When a well is selected, the EOU of the definitive survey or active plan will be displayed.

3. Do one of the following:
   - If you are using the Petrel ribbon mode, in the **Well position** group, toggle on the **Show EOU** button.
   - If you are using the Petrel classic mode, toggle on the **Show EOU** button on the toolbar.

   The EOU is displayed in the 3D window.

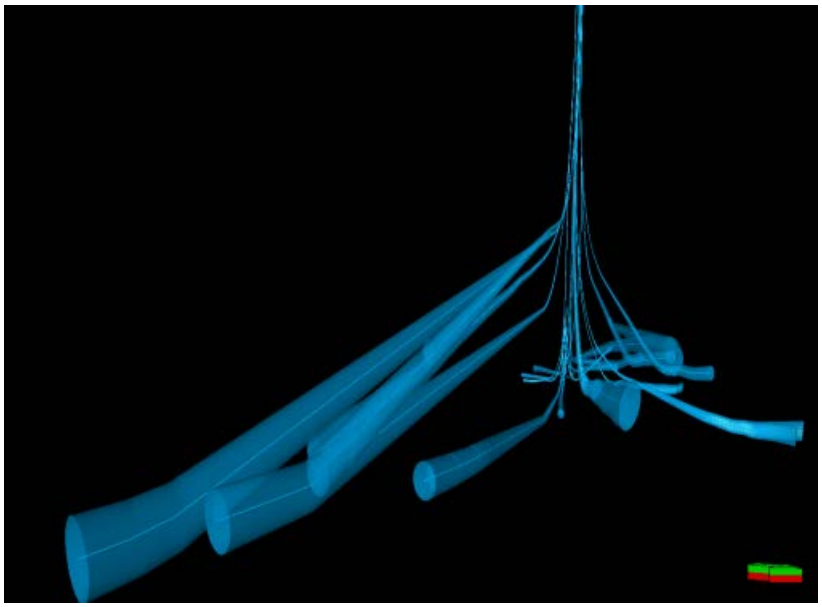


   To hide the EOU display, toggle off the **Show EOU** button.

To display VOU:

1. Activate the 3D window.
2. In the **Input** pane, click the EOU object, or the plan, survey or well with EOU object. You can also select multiple objects together.
   Note: When a well is selected, the VOU of the definitive survey or active plan will be displayed.
3. Do one of the following:
   - If you are using the Petrel ribbon mode, in the **Well position** group, toggle on the **Show VOU** button.
   - If you are using the Petrel classic mode, toggle on the **Show VOU** button on the toolbar.

The VOU is displayed in the 3D window.



To hide the VOU display, toggle off the **Show VOU** button.

Limitations and issues

There are some limitations and known issues related to display of EOU and VOU.

- VOU and EOU can be displayed together. However, when they are displayed together, there is a known issue of visualization. To correctly visualize them together, you need to toggle on the **Orthogonal camera** mode on the 3D window toolbar.

- Displaying a lot of EOU or VOU might cause performance issues. To avoid performance issues, you can hide the EOU or VOU display by toggling off the **Show EOU** or **Show VOU** button.
- If **MWD Gyrotool** error code is used for the well over 70 degree inclination, large measurement error might be generated.
- Sometimes you might see misalignment between the well trajectory center and the center of EOU or VOU.

In rare cases, the edge of the VOU might not be tangent to the edge of the EOU for the same well.

Drilling: Well Design

2014.3

Geological target

Making geological targets

The dialog box for making the geological targets has been enhanced. Now you can use the **Lock** and **Unlock** buttons to define how to shift the target point and geometry center.

You can make a new geological target by defining the target point value or from a source object in the **Make/edit geological target** dialog box.

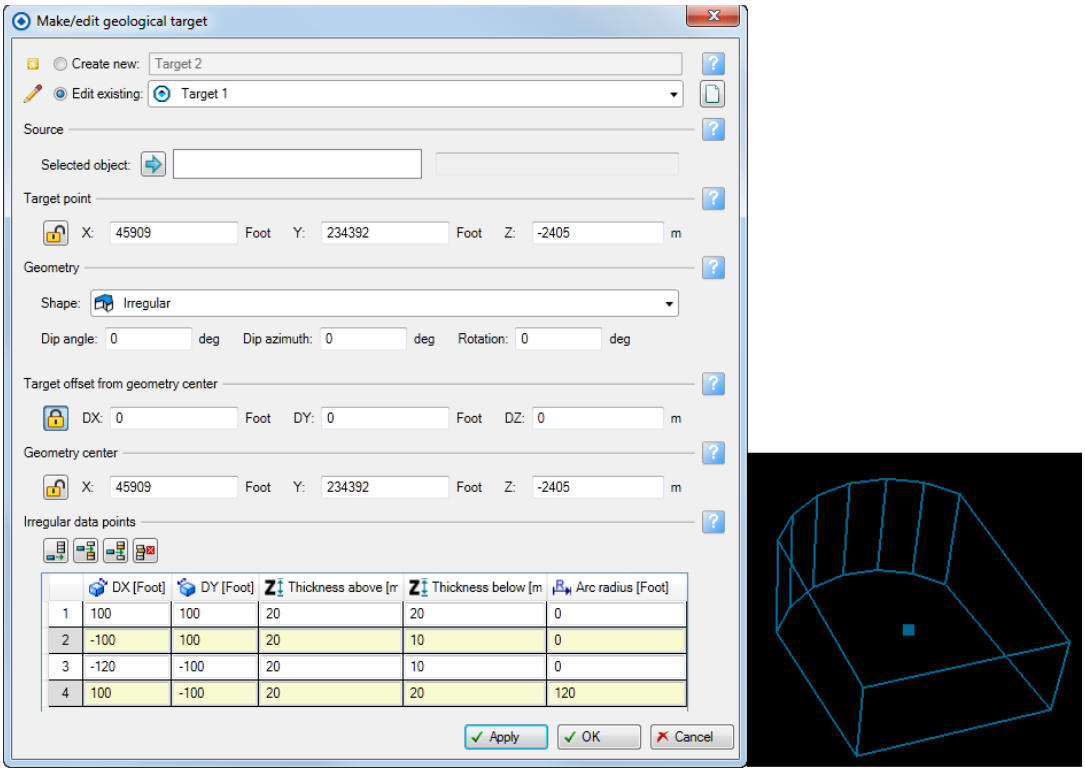
On the **Well Design** tab, in the **Target** group, click **Make geological target**. The **Make/edit geological target** dialog box opens.

In the **Make/edit geological target** dialog box, you can select to create a new or edit an existing target.

With the Lock and Unlock buttons, you can shift the target point or geometry center.

- To shift the target point offset from the geometry center, lock the Geometry center field and input the DX, DY, and DZ values in the Target offset from geometry center field. The target point will be shifted accordingly.
- To shift the geometry center, lock the Target center field and input the DX, DY, and DZ values in the Target offset from geometry center field. The geometry center will be shifted accordingly.
- To shift the target point and the geometry center together, lock the Target offset from geometry center field and edit the X, Y and Z values in either the Target center field or Geometry center field. Both the target point and the geometry will be shifted accordingly.

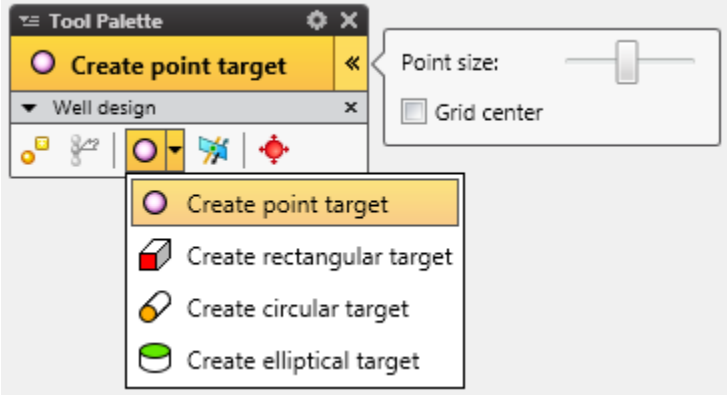
For the Irregular target, now you can input the **Thickness above** and **Thickness below** at each offset point. You can also input the **Arc radius** between the two connected points.



Digitizing geological targets

The **Digitize target** tool is available on the **Tool palette** of Well design. You can use the **Digitize target** tool to create geological targets in the 3D window.

1. On the **Well design** tab, in the **Target** group, click **Digitize target**. The **Tool palette** for **Well design** appears.
2. Activate the create target tool on the **Tool palette** and select the target type from the drop-down list. Depending on the target type, the target variables appear in the extended pane.



3. Define the target variables in the extended pane.
   To create target at the grid center, select the **Grid center** check box.
4. Click the interest target point on an object in the 3D window. A geological target is created at that interested point and shown in the 3D window.

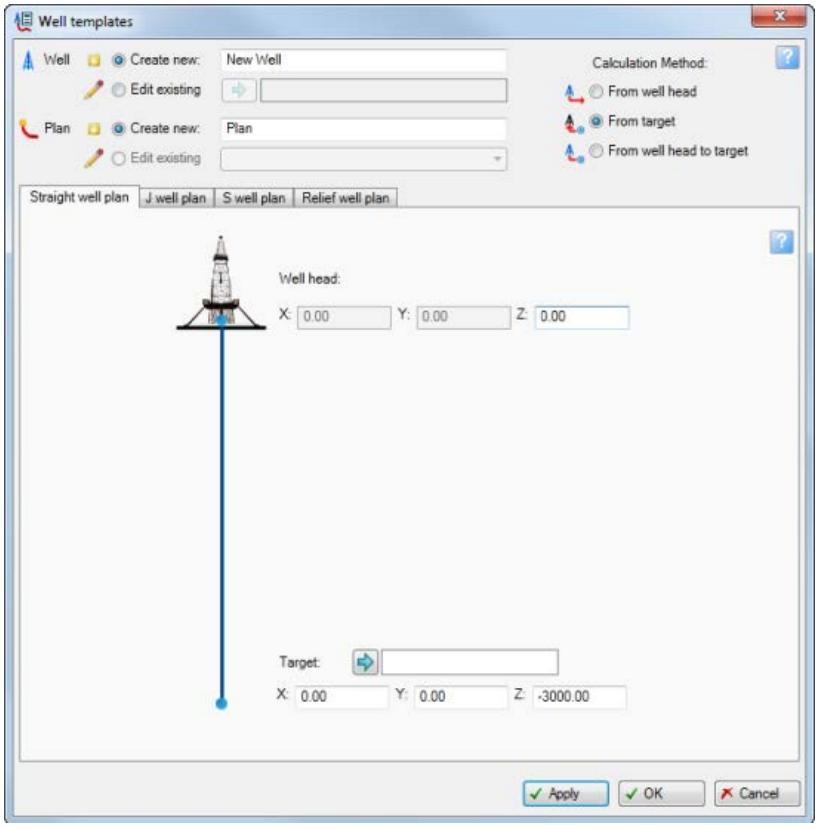
Well templates

The **Well templates** interface has been enhanced. Now three calculation methods are provided for calculating the well plan: **From well head**, **From target**, and **From well head to target**. The **Make run** button has been removed. Now you can click **Apply** or **OK** to create the well plan.

1. In the **Well** area of the Well template, click **Create new** or **Edit existing**.

   In the **Plan** area, click **Create new** or **Edit existing**.

   - If you select to create a plan, a new well plan will be added to the well after you click **Apply** or **OK**.
   - If you select to edit an existing plan, select the existing plan to be replaced from the drop-down list. After you click **Apply** or **OK**, the selected plan will be replaced with the plan that is calculated using the template attributes.



2. Select a calculation method:
   - **From well head**: This is to create a well plan that has specified well head location.
   - **From target**: This is to create a well plan that has specified target location.
   - **From well head to target**: This is to create a well plan that has specified well head and target location.

   The calculation method will constrain your input. The From target calculation method is not available for the Edit existing well option to prevent unintentional changes to the existing well head location.

3. Select the sub tab of the predefined well template.
   On the sub template tab, input the required well data.
   - **Well head**: When Create new well option and From well head or From well head to target calculation method are selected, you can type in the well head location data manually or drop the predefined point set or target with the blue arrow as the well head (X,Y) and reference level for the well plan.
   - **Target**: You can type in the target location data manually or use the blue arrow to drop in the selected target or point set.
   - Attributes such as DeltaMD, INCL and AZIM.
4. Click **Apply** or **OK** to create the well plan.

A new well plan is generated, based on the template displayed in this dialog box.

Target inspector

You can now use the Inspector to quickly view the geological target data or edit some of its style settings in the 2D or 3D window.

Open the Inspector and click a geological target in the 2D or 3D window. The target data appears in the Inspector window.

- The **General** tab lists the X, Y, Z coordinates of the selected target center.
- On the **Quality attributes** tab , you can view the target modification information such as Business projects, Data status, Confidence factor and Comments.
- On the **Info** tab, you can view the shape geometry data of the target. To edit the target, click the edit button on the Info tab.
- On the **Style** tab, you can change the target style settings, such as Color, Size and Label. The style settings on the Inspector Style tab is synchronized with the settings in the Target settings dialog box.
- On the **Association** tab, you can view which well or wells the target is associated with.

Link to well head

In the **Well path designer**, the **Link to well head** option is provided for the first row. When this option is selected, the start point of the well plan is set to the same as the well head location.

Drilling: Real Time

2014.4

Real-time connect

In previous releases, the Real-time connect could only accept data in the WITSML 1.3.1.1 format. In Petrel 2014.4, the Real-time connect can now accept WITSML in the 1.3.1.1 and 1.4.1.1 formats.

2014.3

Streaming real-time survey (behavioral change)

The streamed real-time survey data is now created as a **Survey** object under the **Plans and surveys** folder.

Drilling: Well Positioning

2014.3

Anti-collision analysis

In Petrel 2014.3, you can now perform an Anti-collision analysis for a planned subject well against the existing definitive surveys and the active well plans of the offset wells.

To protect safety of people at the rig site and the environment, every well design and execution must avoid collision with neighboring wells, especially when adjacent wells are producing. Therefore, Anti-collision analysis is important during the well design process.

In Petrel, you can run **Anti-collision project scan** to analyze all the offset wells in the Petrel project; or you can use the **Anti-collision windows scan** to analyze all the wells displayed in the active 3D, 2D, Intersection or Map window.

Each offset well is analyzed based on the surface location, the subsurface distance between the subject well and the offset well, and the trajectory uncertainty.

After using Anti-collision analysis, you can view the risky offset wells and their least distance line to the subject well in the 3D window. You can also view the collision risk level of the offset well in the Anti-collision spreadsheet and generate the Anti-collision reports and plots for analyzing collision risks.

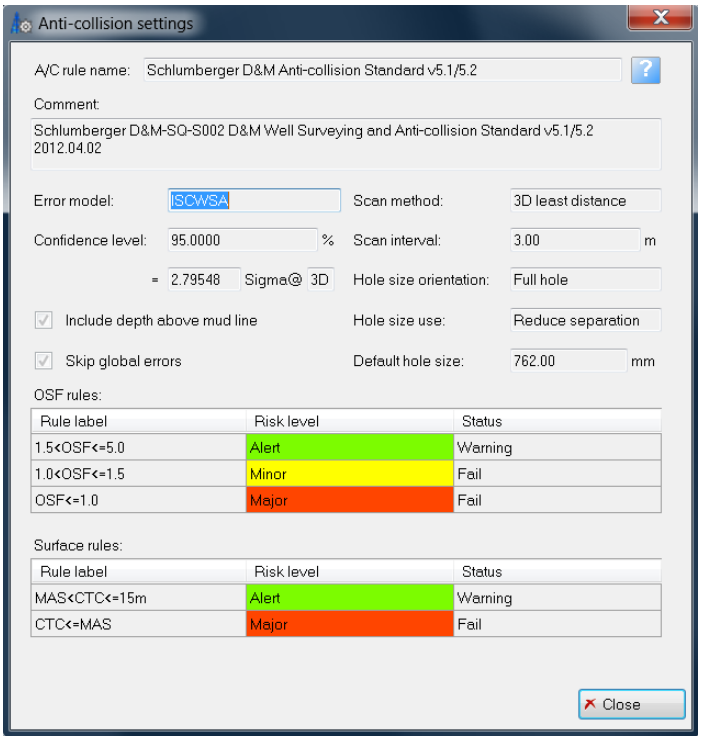
If you modify your subject plan or any of the offset wells, the previous analysis will become invalid. You need to perform the Anti-collision analysis again.

Anti-collision settings

You can view the settings, including the rules used for Petrel Anti-collision analysis.

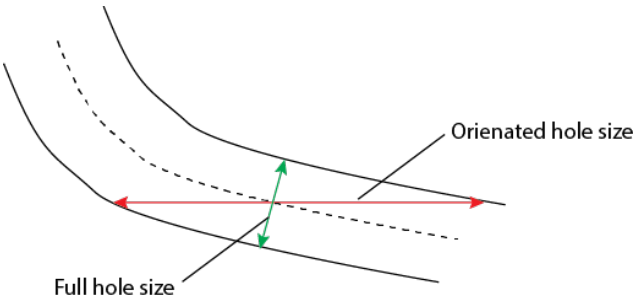
On the **Well Positioning** tab, in the **Anti-collision** group, click the settings button.

The **Anti-collision settings** dialog box appears.

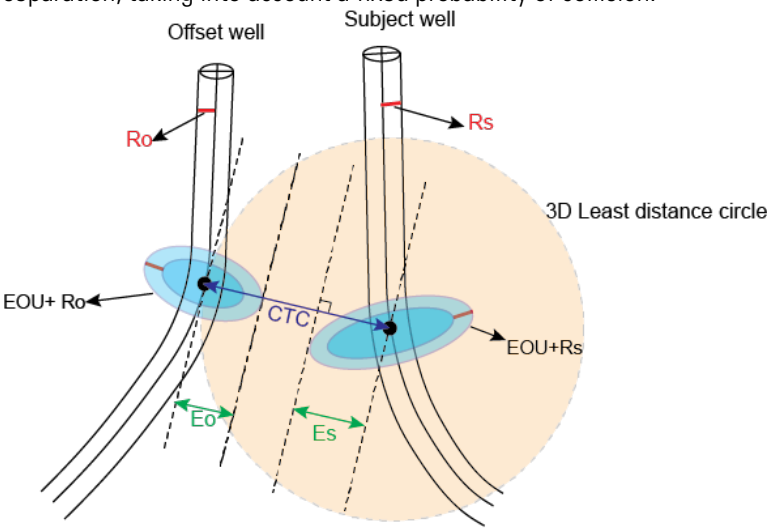


The following settings and rules are used for the Anti-collision proximity scan.

- **A/C rule name**: The Anti-collision rule name.
- **Comment**: The current Anti-collision rule is based on Schlumberger DM Anti-collision standard.
- **Error model**: The survey instrument error model is built by Industry Steering Committee for Wellbore Survey Accuracy (ISCWSA).
- **Confidence level**: All EOU computations are computed at 95% confidence level in three dimensions.
- **Scan method**: The scan method used for proximity scan. The **3D least distance method** calculates the nearest distance to each offset well by stepping down the subject trajectory at a depth interval.
- **Scan interval**: The depth interval for the proximity calculation scan.
- **Hole size orientation**: The same default hole size is used for the full hole.



- **Hole size use**: When calculating the OSF ratio with the Reduce separation method, the denominator of the equation is equal to the center-to center distance subtracting the hole radii.
- **Default hole size**: A default hole size (762 mm, 30 inch) is used for all wells during the A/C calculation.
- **Include depth above mud line**: The trajectory uncertainty above the mud line is included in the Anti-collision calculation.
- **Skip global errors**: The global errors such as magnetic declination are not included in the EOU computation.
- **OSF rules**: The oriented separation factor (OSF) rule is used to analyze the collision risk level. The OSF is defined as the ratio of the center to center separation between wells and the ellipsoid of uncertainty separation, taking into account a fixed probability of collision.



$$OSF = \frac{CTC - Ro - Rs}{Eo + Es}$$

OSF = (CTC-Ro-Rs)/(Eo+Es)

Where

Eo: the ellipsoid radius offset well

Es: the ellipsoid radius subject well

CTC: the center-to center distance

Ro: the hole radius of offset well

Rs: the hole radius of subject well

- **Surface rules**: At or near surface, the OSF values are unrealistically high, so a minimum allowable separation (MAS) rule is applied to the collision risk level. For the Surface rule: MAS = (Ro + Rs) + 0.8(CTC at Wellhead – (Ro + Rs)).

Note: In the current Petrel implementation, these rules are shown in the setting for reference only.

Anti-collision project scan

You can now run the **Anti-collision project scan** to analyze the collision risk for all the offset wells in the Petrel project.

Do one of the following to run the Anti-collision project scan:

- Right-click the subject well plan in the **Input** pane and click **Anti-collision project scan**.
- In the Ribbon mode, click the subject well plan in the **Input** pane and click the **Anti-collision project scan** button in the **Anti-collision** group on the **Well Positioning** tab.
- In the classic mode, click the subject well plan in the **Input** pane and click the **Anti-collision project scan** button on the toolbar.

The Anti-collision result for project scan is created under the selected subject well plan in the **Input** pane. The result name is **AC results by project scan**.

Alternatively, you can also click or right-click a well and run **Anti-collision project scan**. By this way, the Anti-collision result is created under the well trace trajectory (either the definitive survey or active plan) in the **Input** pane.

Anti-collision window scan

You can now run the **Anti-collision window scan** to analyze the collision risk for all the offset wells in the active **3D**, **2D**, **Map**, or **Intersection** window.

1. Activate a 3D, 2D, Map or Intersection window.
2. Display the offset wells to be analyzed in the active window.
3. Do one of the following:
    - Right-click the subject well plan in the **Input** pane and click **Anti-collision window scan**.
    - In the Ribbon mode, click the subject well plan in the **Input** pane and click the **Anti-collision window scan** button in the **Anti-collision** group on the **Well Positioning** tab.
    - In the Classic mode, click the subject well plan in the **Input** pane and click the **Anti-collision window scan** button on the toolbar.

The Anti-collision result for window scan is created or updated under the selected subject well plan in the **Input** pane. The result name is **AC results by window scan**.

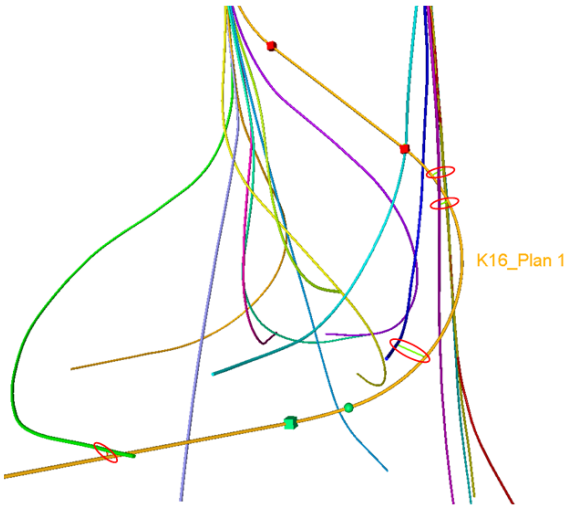
Alternatively, you can also click or right-click a well and run **Anti-collision window scan**. By this way, the Anti-collision result is created under the well trace trajectory (either the definitive survey or active plan) in the **Input** pane.

Showing Anti-collision result

After running the **Anti-collision project scan** or **Anti-collision window** scan, you can show the risky offset wells and their least distance line to the subject well in the 2D or 3D window.

1. Activate a 2D or 3D window.
2. Click the subject well plan in the **Input** pane and toggle on the **Show Anti-collision result** button in the **Anti-collision** group on the **Well Positioning** tab.

The Anti-collision result is shown in the window.



- For the project scan result, the risky offset wells identified by the Anti-collision project scan are shown in the window.
- For the window scan result, the offset wells used during the Anti-collision window scan are shown in the window.
- The least distances between the subject well and each risky offset well are visualized as lines in the window.

3. To hide the Anti-collision result, toggle off the **Show Anti-collision result** button.

Anti-collision spreadsheet

After running the **Anti-collision project scan** or **Anti-collision window** scan, you can view the detailed Anti-collision calculation results for each risky well in a spreadsheet.

1. To open the **Anti-collision spreadsheet**, do one of the following:
   - Right-click the Anti-collision result in the **Input** pane and click **Anti-collision spreadsheet**.
   - In the Ribbon mode, click the Anti-collision result in the **Input** pane and click the **Anti-collision spreadsheet** button in the **Anti-collision** group on the **Well Positioning** tab.

The **Anti-collision spreadsheet** window opens.

	Zoom to	Offset well	Subject well MD (Measured depth [m])	Offset well MD (Measured depth [m])	CtC distance (Distance [m])	MAS (Distance [m])	OSF
1		M03	1539	1257.87	65.98	30.12	3.35
2		M04	1659	1374.66	42.42	27.14	2.37
3		M06	1626	1334.8	7.48	26.16	0.38
4		M07	1626	1334.8	7.48	26.16	0.38
5		M09	2277	2177.09	99.18	38.25	3.94
6		M10	1371	1166.9	37.36	29.06	1.94
7		K01	0	15	10.58	8.61	
8		K01	3264	3156.46	20.29	53.67	0.55
9		K02	201	216.01	6.53	5.38	295.94
10		K04	201	216.01	4.13	3.45	666.09
11		K05	228	243.19	10.04	8.26	67.82
12		K06	219	234.07	7.14	5.93	80.58
13		K07	0	15	13.46	10	
14		K08	0	15	6.96	5.72	
15		K09	0	14.99	10.28	9.18	162225.79
16		K10	0	14.97	12.84	10	205417.89

By default, all the **Critical** Anti-collision results are displayed in the spreadsheet. The **Critical** risk levels are classified as **Major**, **Minor** and **Alert**.

2. Click the **Critical** or **Detail** view option.
   - In the **Critical** view, only the critical stations with collision risks are displayed in the spreadsheet. The **Critical** risk levels are classified as **Major**, **Minor** and **Alert**.
   - In the **Detail** view, the Anti-collision data for the selected offset well are listed in the spreadsheet per every 3-meter interval of the subject well MD.

   The critical stations are highlighted with colors.
3. In the **Offset wells** area, the offset wells calculated by the proximity scan are listed. Click each offset well. The critical or detail data for the selected well are listed in the spreadsheet.
4. To view a critical station clearly in the 3D window, in the **Critical** view, click the **Zoom to** button of that critical station row. The critical station is zoomed in the 3D window.

Anti-collision report

After running the **Anti-collision project scan** or **Anti-collision window** scan, you can generate the Anti-collision report in PDF format.

1. Do one of the following:
   - Right-click the Anti-collision result in the **Input** pane and click **Anti-collision report**.
   - Click the Anti-collision result in the **Input** pane and click the **Anti-collision report** button in the **Anti-collision** group on the **Well Positioning** tab.

   The **New report** dialog box appears.
2. Select a report template in the **Template selection** area. The **Report settings** area displays the contents of the template. A non-editable default template is provided with the installation. You can use this default template to generate the Anti-collision report, or you can create a new Anti-collision report template.

3. Select the **Generate report** check box to create the PDF report.
   If this check box is not selected, report generation will only create a report object under the **Report Folder** in the **Input** pane. You can create the PDF report with the report object later. To do it, right-click the report object in Petrel **Input** pane and click **Create report**.
4. Click **OK**. The **Save as** dialog box appears.
5. In the **Save as** dialog box, give the file name and click **Save**. The PDF Anti-collision report is generated.

No-go zone analysis

You can now visualize the No-go zones around the offset wells for a subject well plan. The No-go zone visualization is very helpful for improving the anti-collision analysis efficiency. The Petrel 3D No-go zone not only indicates the anti-collision failure information, but more importantly, displays the 3D space where the well plan can go.

In Petrel, you can run **No-go zone project scan** to analyze all the offset wells in the Petrel project and visualize the No-go zone around the offset wells nearby the subject plan.

You can also use the **No-go zone windows scan** to show No-go zones around all the offset wells which are displayed in the active 3D window.

When calculating the No-go zone for each offset well, Petrel steps down each EOU station of the offset well trajectory. At each EOU station of the offset well, Petrel computes the perpendicular cross-section, and computes the minimum allowable separation distance for each direction. This forms the No-go zone on this cross section. The three-dimensional No-go space is constructed by connecting all of the No-go cross sections on each offset well, from wellhead to TD.

No-go zone project scan

You can now run **No-go zone project scan** for a well plan against all the offset wells in the Petrel project.

To run the **No-go zone project scan** for a well plan against all the offset wells in the Petrel project, do one of the following:

- Right-click the subject well plan in the **Input** pane and click **No-go zone project scan**.
- Click the subject well plan in the **Input** pane and click the **No-go zone project scan** button in the **No-go zone** group on the **Well Positioning** tab.

The No-go zone result for project scan is created under the selected subject well plan in the **Input** pane. The result name is **No-go zone by project scan**.

No-go zone window scan

You can now run **No-go zone window scan** for a well plan against all the offset wells in the active **3D** or **2D** window.

1. Activate a **3D** or **2D** window.
2. Display the offset wells to be analyzed in the active window.

3. Do one of the following:
    - Right-click the subject well plan in the **Input** pane and click **No-go zone window scan**.
    - Click the subject well plan in the **Input** pane and click the **No-go zone window scan** button in the **No-go zone** group on the **Well Positioning** tab.

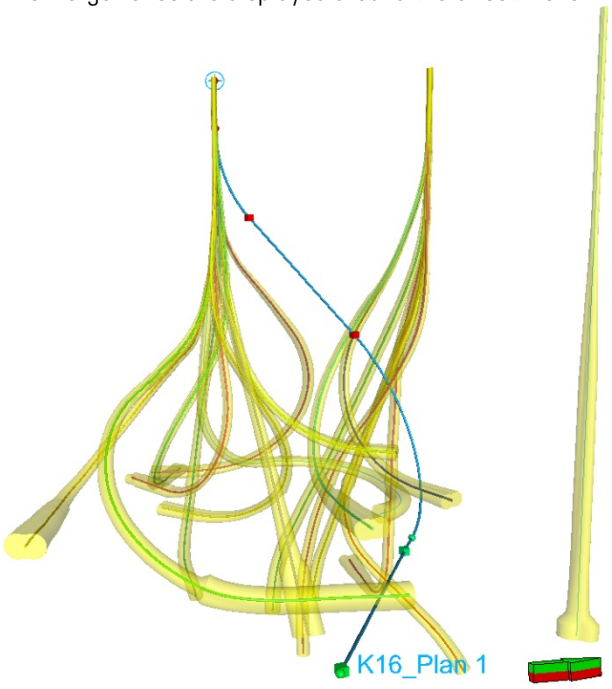
    The No-go zone result for window scan is created or updated under the selected subject well plan in the **Input** pane. The result name is **No-go zone by window scan**.

Showing No-go zone

After running the **No-go zone project scan** or **No-go zone window scan**, you can show or hide the No-go zone result in the 3D or 2D window.

1. Activate a 3D or 2D window.
2. To show the No-go zone, do one of the following:
    - Click the subject well plan or the No-go zone object in the **Input** pane and toggle on the **Show No-go zone** button in the **No-go zone** group on the **Well Positioning** tab.
    - Toggle on the No-go zone in the **Input** pane.

    The No-go zones are displayed around the offset wells in the 3D or 2D window.



3. To hide the No-go zone, toggle off the **Show No-go zone** button or toggle off the No-go zone object in the **Input** pane.

Refreshing No-go zone

After running the **No-go zone project scan** or **No-go zone window scan**, if any underlying data has been changed since the computation was performed, the **No-go zone refresh** button will become available to remind you to refresh the No-go zone result in the window.

Click the subject well plan or the No-go zone result in the **Input** pane. The **No-go zone refresh** button becomes available.

The No-go zone result is recomputed and updated in the window.

Alternatively, you can re-run the **No-go zone project scan** or **No-go zone window scan** for the subject well.

In the following condition, the **No-go zone refresh** button will become available to remind you to refresh the No-go zone result:

- The trajectory of the subject well or any offset well has been changed.
- The Survey program and EOU of the subject well or any offset well has been changed.
- Any offset well is removed.

Note: If any new offset well has been added in the Petrel project, the **No-go zone refresh** button will not be available for refreshing. You can re-run the **No-go zone project scan** to include the new offset wells.

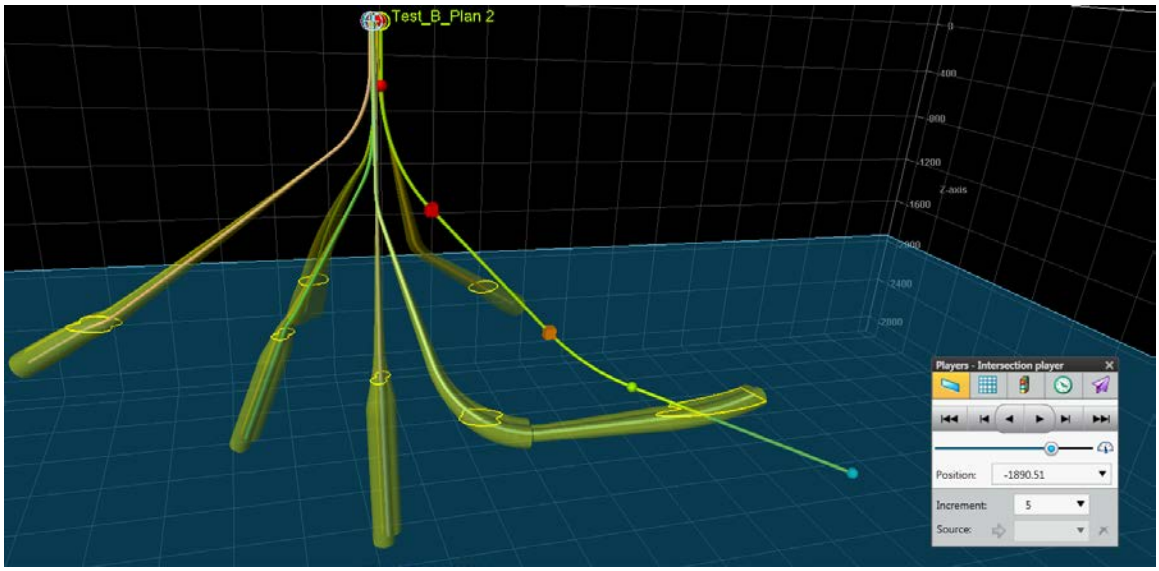
Viewing No-go zone slice

You can now view the intersection slices of No-go zone from different directions in a 3D or 2D window.

The No-go zone can be displayed in a 3D window or 2D window. However when there are many wells in the window, it might be difficult for the users to see the location of the No-go zone to the subject well. The No-go zone intersection slice can be viewed using the **General Intersection plane** function.

To view the No-go zone slice on the Intersection plane:

1. Toggle on the No-go zone result in the **Input** pane to display it in a 3D or 2D window.
2. On the **Home** tab, in the **Insert** group, click **Object** > **Insert random line**.
   A **General intersection** is created under the **Input** pane.
3. Toggle on the **General intersection** to display it in the window.
4. To display the intersection in horizontal, on the **Intersection** tab, in the **Alignment** group, click the **Align Horizontally**.
5. Click the **Snap to point** button and click the subject well at a specified depth in the 3D window. The intersection plane will be snapped to specified depth.
6. Click the **Visualize on intersection** button**.** The check boxes of the items which can be displayed on the General intersection plane become blue color.
7. Toggle on the No-go zone result. The No-go zone intersection slice is highlighted on the General intersection plane.
8. Click **Intersection player** in the **Position** group to review the No-go zone on the General intersection plane. The Players window appears.
9. In the **Players-Intersection player** window, adjust the increment and play forward or backward the intersection plane to observe the distance between the subject well plan and No-go zone intersection slice.



If the well plan is crossing the No-go zone intersection line, run the Anti-collision scan for further checking.

Note: The No-go zone only gives you a view of collision hit, but the actual collision risk level must be confirmed using the Anti-collision scan.

Trajectory uncertainty

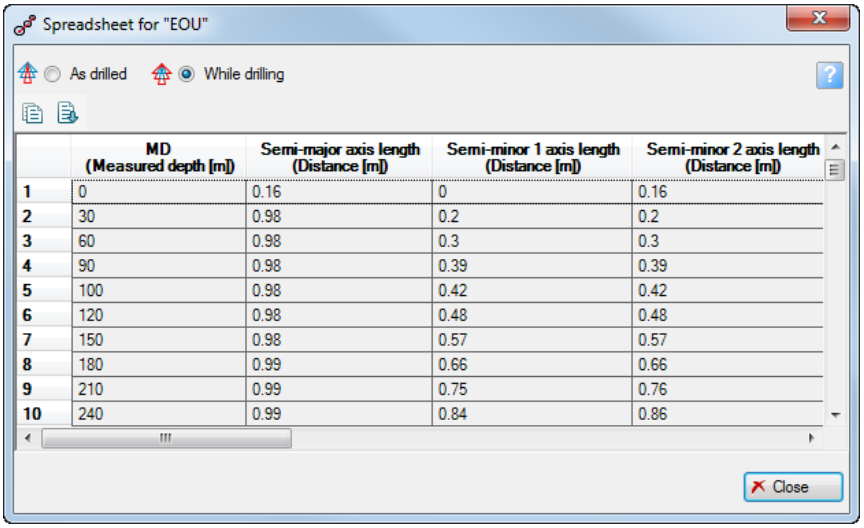
EOU spreadsheet

After an EOU is created, you can view the detailed EOU results at each station in the EOU spreadsheet.

To open the EOU spreadsheet, do one of the following:

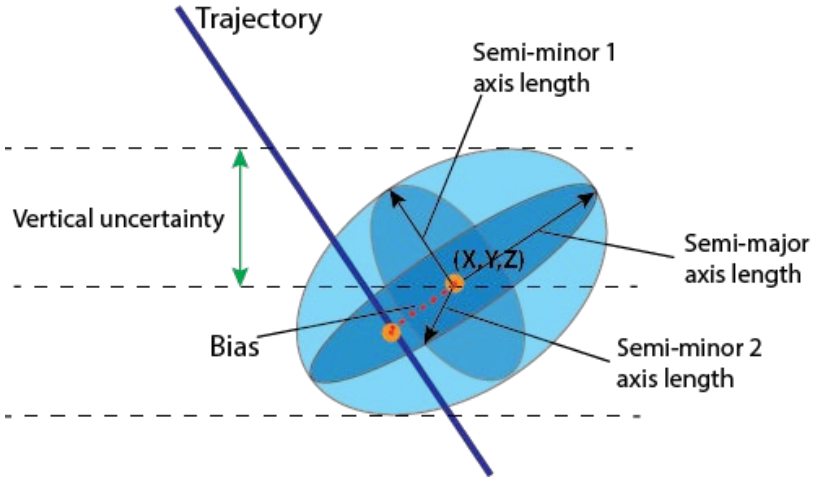
- Right-click the EOU result in the **Input** pane and click **EOU spreadsheet**.
- Click the EOU result in the **Input** pane and click the **EOU spreadsheet** button in the **Trajectory uncertainty** group on the **Well Positioning** tab.

The EOU spreadsheet window opens.



	MD (Measured depth [m])	Semi-major axis length (Distance [m])	Semi-minor 1 axis length (Distance [m])	Semi-minor 2 axis length (Distance [m])
1	0	0.16	0	0.16
2	30	0.98	0.2	0.2
3	60	0.98	0.3	0.3
4	90	0.98	0.39	0.39
5	100	0.98	0.42	0.42
6	120	0.98	0.48	0.48
7	150	0.98	0.57	0.57
8	180	0.99	0.66	0.66
9	210	0.99	0.75	0.76
10	240	0.99	0.84	0.86

The EOU spreadsheet lists the detail EOU data at each measured depth, including the length and direction of the semi axes of the ellipsoid, the center of the ellipsoid and its bias to the trajectory station point.



- Semi-major axis is the longest axis for the ellipsoid.
- Vertical uncertainty is the half length of the maximal vertical projection.
- Centers of X, Y, and Z are the coordinates of the ellipsoid center.
- Biases of X, Y, and Z are the offset distances from the trajectory station to the ellipsoid center in the X, Y, Z directions.

If the EOU result is for a well plan, you can select show the EOU spreadsheet calculated by **As drilled** logic or **While drilling** logic.

You can copy or export the EOU spreadsheet data.

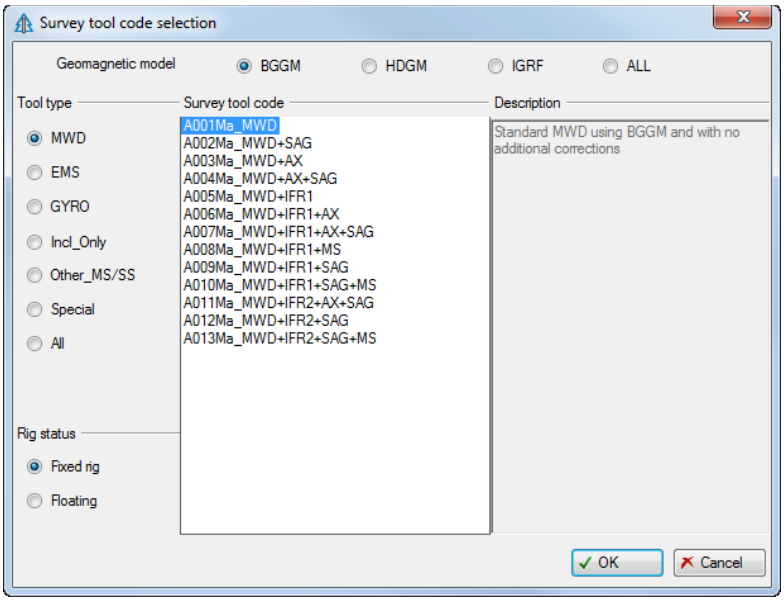
Geomagnetic model for Survey program

The default geomagnetic model for survey program is IGRF-11. Now you can select other geomagnetic models which can be downloaded from the **Ocean** store and installed as Petrel plug-ins. In the **Trajectory uncertainty settings**, you can change the default geomagnetic model and view its valid period.

OWSG tool code for Survey program

A new **OWSG** tool code group is available in the Survey program. This tool group includes additional tool codes from the Operator's Wellbore Survey Group (OWSG).

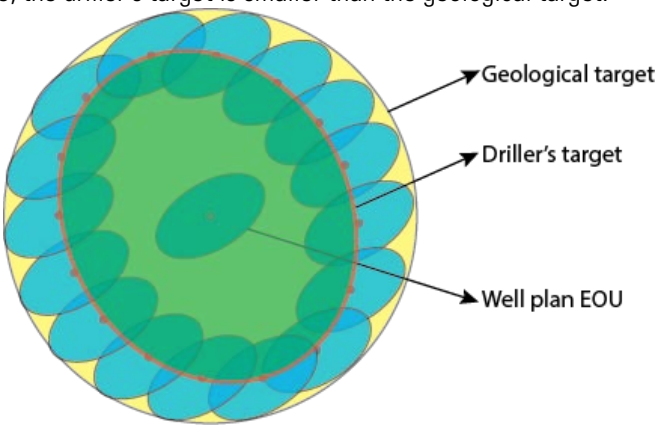
When this option is selected, the Survey tool code selection dialog box appears. You can select the **Geomagnetic model type**, **Tool type** and **Rig status** to filter the OWSG tool codes.



Driller's target

You can now create and visualize a driller's target for a well trajectory and its associated geological target.

A driller's target is derived from a geological target considering the uncertainty of the well plan or survey. Although the well trajectory is planned to intersect the 3D space of the geological target, due to the presence of inaccuracies of survey tools, it is still possible that the actual trajectory might miss the geological target. To analyze this, you can create and visualize the driller's target for a well plan and its geological target.
Driller's target is defined as a target within which the survey will not miss considering the uncertainty of the survey. Therefore, the driller's target is smaller than the geological target.



If the geological target is too small or the trajectory uncertainty is too big, the driller's target will become very small, or even you cannot generate a driller's target for the well trajectory and the geological target. In this case, you can consider to make applicable changes, such as:

- Extending the geological target.
- Choose a more accurate survey tool for the survey program to reduce the trajectory uncertainty.
- Modify the well plan to lessen the approach angle.

- Reduce the confidence level of the target penetration. Currently, you cannot change the confidence level for EOU in Petrel. A fixed confidence level of 95% at 3D is applied.

Now in Petrel, you can create a driller's target for a well plan and its geological target.

Before you create the driller's target, you must have:

- A geological target which has been attached to the well.
- A well plan or survey which intersects the associated geological target under the same well.
- The Survey program and valid EOU under the well plan or survey.

To create the driller's target for the well plan or survey, do one of the following:

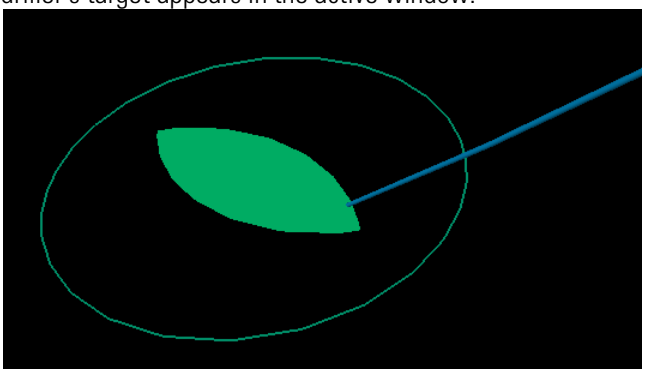
- Right-click the plan or survey and click **Make driller's target**.
- Right-click the Survey program or EOU and click **Make driller's target**.
- Click the plan or survey and click **Make driller's target** in the **Target** group on the **Well Design** tab.

A driller's target object is created under the associated target folder.

The created driller's targets are only for visualization. You cannot manually edit the driller's target. If any of the underlying data is changed, the related driller's targets will be updated automatically.

A driller's target can be displayed in a 3D, 2D or Map window.

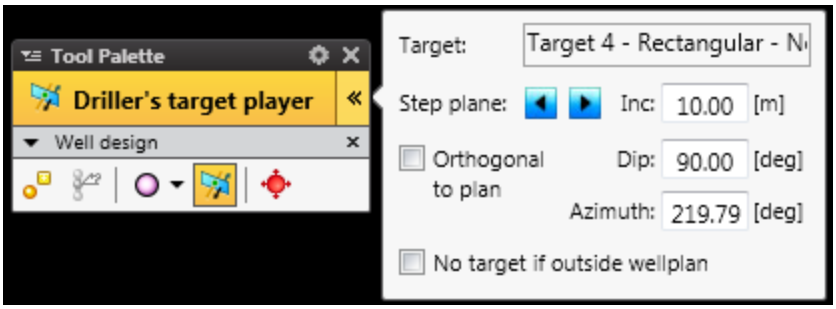
1. Activate the 3D, 2D or Map window.
2. (Optional) Display the associated geological target and well trajectory in the window.
3. Toggle on the driller's target in the **Input** pane.
   The driller's target appears in the active window.



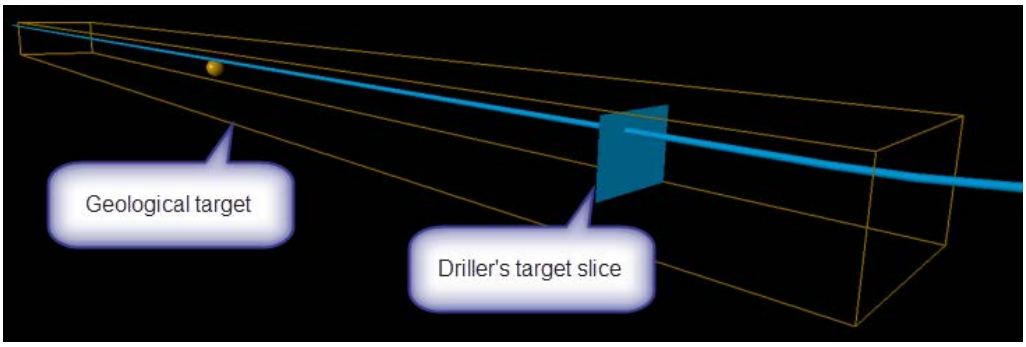
4. To zoom the driller's target, right-click the driller's target and click **Zoom to object**. The driller's target is zoomed in the window.
5. To change the display parameters, double-click the driller's target to open its Settings dialog box. Similar to editing the settings for geological target, you can edit the Style settings for driller's target on the **Style** tab and change the color or name on the **Info** tab.
6. To hide driller's target display in the window, toggle off the driller's target in the **Input** pane.

You can also view the driller's target slices using the **Driller's target player**.

1. On the **Well design** tab, in the **Target** group, click **Digitize target**. The **Tool palette** for **Well design** appears.
2. Activate the **Driller's target player** tool on the **Tool palette.**
3. Click the driller's target.
   The name of the target appears in the extended pane.



4. Set player settings such as the increment, dip and azimuth of the slice.
   To set the plane direction orthogonal to the well plan, select the Orthogonal to plan check box.
   To hide the target if the well plan is outside the well plan, select the No target if outside well plan check box.
5. Play forward or backward the driller's target plane to observe the driller's target slice.



Drilling: Well Construction

2014.2

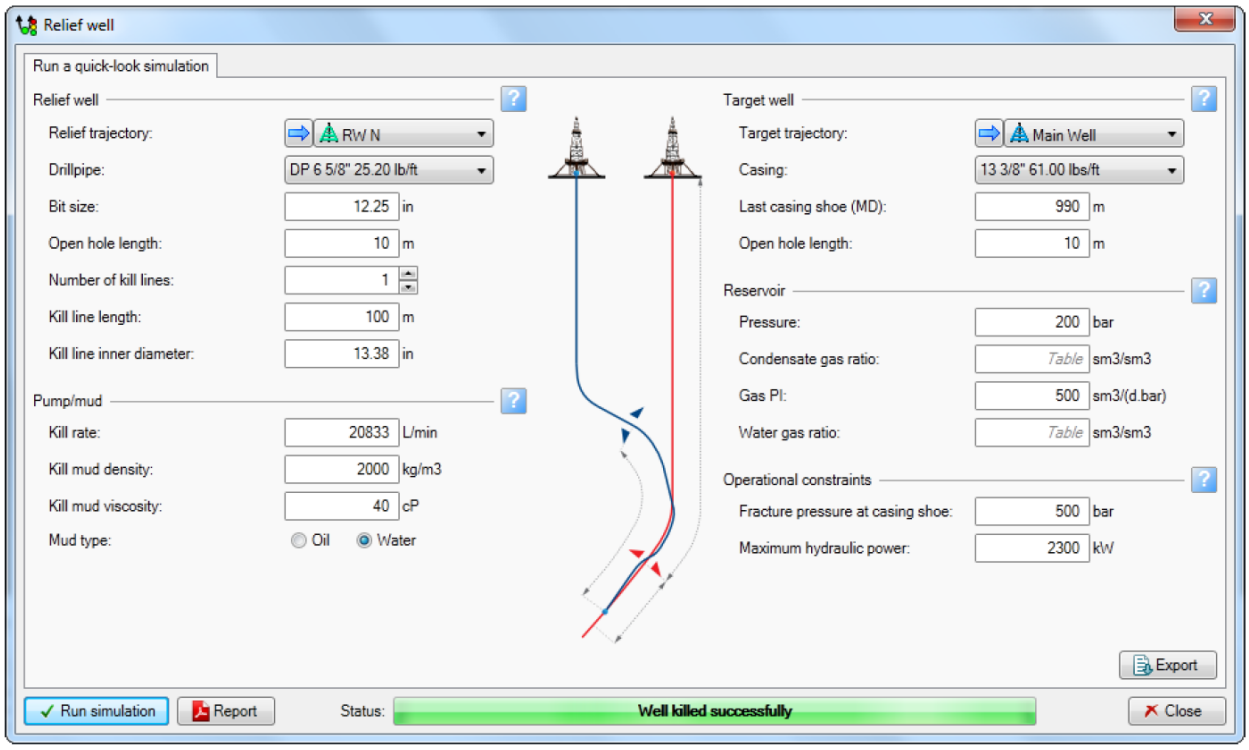
Well Construction

This is a new module in Petrel that provides relief well dynamic simulation for well contingency planning. Relief well geometries can quickly be evaluated and the requirements to kill any proposed well plan are simulated using the OLGA* dynamic multi-phase simulator. This enables rapid screening of new well designs for contingency planning purposes. Reports are produced as an output to conform with regulatory requirements in many countries, including quantification of "worst-case discharge rate" for the proposed well.

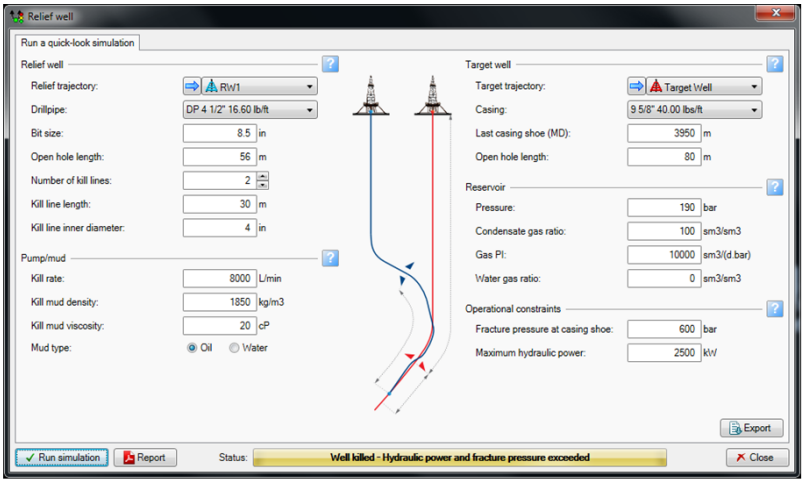
Well Construction is the module name of a suite of drilling engineering tools. For Petrel 2014, Well Construction will include only Petrel Relief Well.

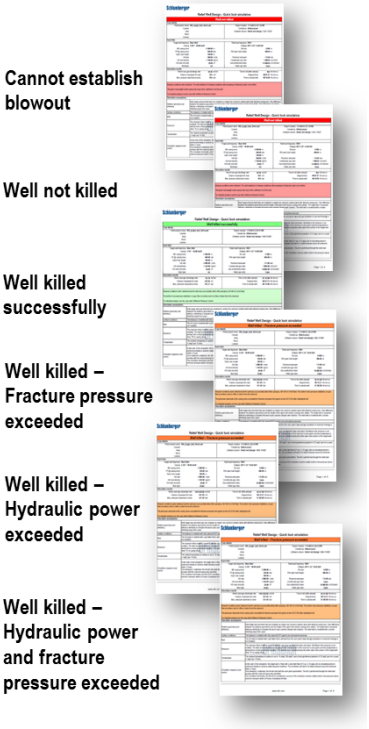
Petrel Relief Well is a quick screening tool to evaluate the feasibility of a relief well. This is achieved through:

- A simple workflow based tool that incorporate full physics blowout simulation
- Meaningful and easy to understand results



From this simple-to-use tool, some key parameters are given and a full Drillbench Blowout Control simulation is run to provide accurate results highlighting whether or not the well can be killed. The entire input process only takes a few minutes and the results are presented in a concise report.





The Petrel relief well tool is designed as an integral part of the Petrel process and will use the target and relief well trajectories in the Petrel project directly without the need to exchange data through error prone import and export. Typically, a blowout study will require a number of inputs that can be time consuming to collect and enter. These data may also be highly uncertain in the early stages of a well design. This is why the input has been scaled down to key parameters that have a critical impact on the analysis. The rest of the parameters to the underlying simulator are automatically set based on conservative assumptions, so that a "successful kill" obtained through the Petrel relief well simulation will remain successful if a more detailed study is performed.

The only pre-requisite to run a Petrel relief well simulation is to have two trajectories defined in the Petrel project that intersect. After opening and entering the required input into the Petrel relief well interface, the simulation will be performed as a background process. The process window can be closed during the simulation. The simulation process can take some time (from a few minutes to an hour). Run time depends on the complexity of the trajectory and the simulated flow rates. When the simulation is completed, the results will be given in the form of a PDF report.

Production

Production: General

2014.1

OFM Data Connector

Well Matching

When importing OFM wells into your Petrel project, you can now match OFM wells to Petrel wells based on UWI, name, or well head location.

The well matching feature provides the following options:

- Defining well matching criteria
- Auto-matching OFM and Petrel wells that have a 1:1 match
- Manually matching an OFM well that has multiple Petrel well matches
- Manually matching unmatched wells (for OFM wells that do not have a Petrel well match based on your matching criteria)

Production: Well Deliverability

2014.1

Fluid Data Matching

The Fluid Properties feature provides new options for viewing and modifying Viscosity, Calibration, and Thermal properties to match your field fluid data. The software also provides correlation mappings for data imported from PIPESIM.

The Fluid Properties feature provides the following options for entering field fluid data:

Option	Description
Viscosity	• Oil viscosity correlations: undersaturated oil, live oil, dead oil temperature and viscosity • Emulsion viscosity method • Watercut

Option	Description
Calibration	• Fluid data • Pressure • Temperature • Correlations
Thermal	• Specific heat capacity: gas, oil, and water • Thermal conductivity: gas, oil, and water • Enthalpy calculation method and specific latent heat of vaporization

Flow Correlation Data Matching

The Flow Correlation Data Matching feature provides an updated user interface, including new options for automatically retrieving log data and observed data, for matching pressure and temperature along the wellbore, and for manually entering data for pressure and temperature. The software also provides correlation mappings for data imported from PIPESIM.

The Flow Correlation Data Matching process optimizes the algorithm that is used for running VFP and Nodal Analysis simulations. Different sets of pressure and temperature data are used for matching purposes and automated retrieval and manually entered data types are supported. You can also perform quality control checks by graphically displaying the observed data in context with the selected optimized algorithms.

ECLIPSE Defined Well Productivity Index (PI) Models

The IPR Model Manager now allows you to automatically retrieve fluid and reservoir properties and IPR data from an ECLIPSE case.

You can retrieve fluid and reservoir properties and IPR data by importing them from an ECLIPSE Simulation Case. For each inserted time step, the software automatically populates perforation pressure, temperature, phase ratio properties, and liquid Well PI when available in the ECLIPSE case.

Import and Export PIPESIM well models

In addition to the current import functionality, you can now export a well model in a production model format (.pips) so that you can open it in PIPESIM v2013.

For each exported well, the following well data is exported: deviation profile, fluid data, fluid and reservoir association to each completion, surface equipment, and the associated flow correlation.

Production: Production Analytics

2014.1

Ocean API on completion level data

A plug in writer/creator has the ability to populate a completion level data object in Petrel through a plug-in. The types of data that are supported are:

- Pressures
- Temperatures
- Rates
- Ratios

Ocean API on Split Set Manager

A plug in writer/creator has the ability to programmatically create a split set, edit existing split sets, and delete split sets in a Petrel project.

The end user can create a split set by:

- Importing split set data from an Excel spreadsheet or an ASCII file.
- Copying/pasting information from an Excel spreadsheet or ASCII file directly into the Split Set Manager.
- Editing the well's perforation allocation directly in the Split Set Manager.

The Ocean API will allow the plug-in developer to:

- Create split sets with data from external sources.
- Edit/delete existing split sets.

Production: Production Interpretation

2014.3

Production Interpretation (PI) provides a streamlined workflow for interpretation of rate and pressure data to yield key reservoir and well parameters using quick semi-analytical simulation and industry standard interpretation tools. PI allows engineers to identify productivity problems and efficiently perform production management at early stages. The toolkit can be used to quickly evaluate production performance, interpret production data, and optimize production, stimulation and completion operations.

PI delivers the **Forecasting** and **Rate transient analysis** workflows.

The **Forecasting** workflow enables the quick setup and simulation of multiple models for screening, sensitivity analysis, and evaluating 'what-if' scenarios; aids you in performing pre-job design and sensitize/optimize on stimulation/completion parameters for unconventional wells.

The main features of the **Forecasting** workflow are:

- Creates analytical simulation model with Petrel wells/completions
- Creates analytical simulation case
- Runs analytical production simulation/forecasting

The **Rate transient analysis** workflow allows you to quickly estimate key reservoir/well parameters and run analytical simulation to match production. User performs flow regime recognition on diagnostic plots. Industry standard and Schlumberger-developed diagnostic plots (e. g., RNI diagnostic plot and Poe's plot) are provided. With user supplied analytical case, PI constructs diagnostic plots to aid you in identifying the flow regimes with fit-line technique. Interpretation results can be used to update the model. You can run analytical simulation with interpretation calibrated model. Alternatively, PI supports manual input of model parameters to run simulation.

The main features of the **Rate transient analysis** workflow are:

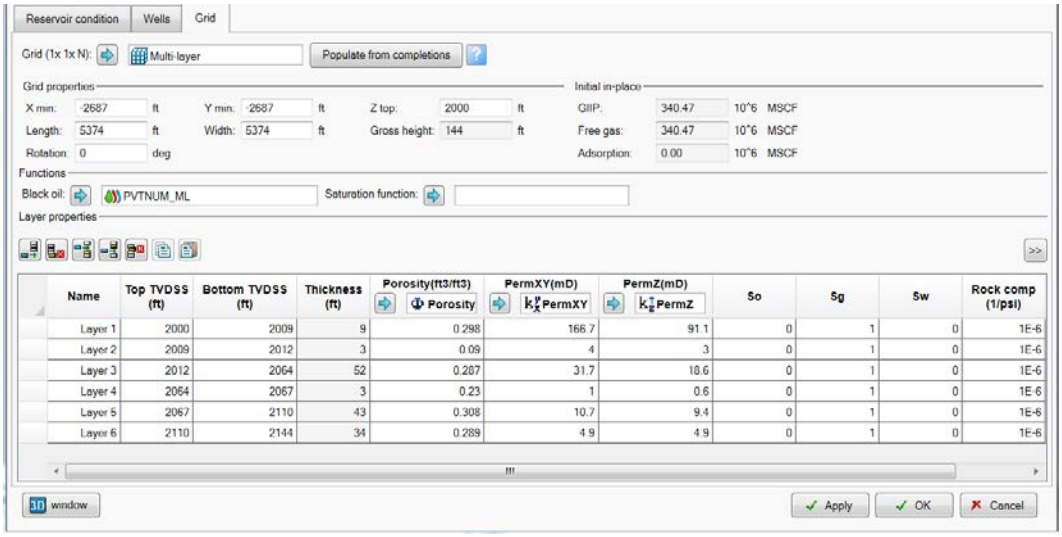
- Creates RTA diagnostic plots for production data
- Estimates key reservoir/well parameters from flow regimes on diagnostic plots
- Runs analytical simulation to match production
- Provides nonlinear regression

PI also provides these additional features:

- Data reduction to reduce data size through value and time threshold
- BHP computation to compute BHP from WHP and production rates using VFP table
- PTA pressure diagnostic plotting to create PTA log-log plots for observed data and ECLIPSE simulated data
- RTA diagnostic plotting to create RTA plots for observed data and ECLIPSE simulated data

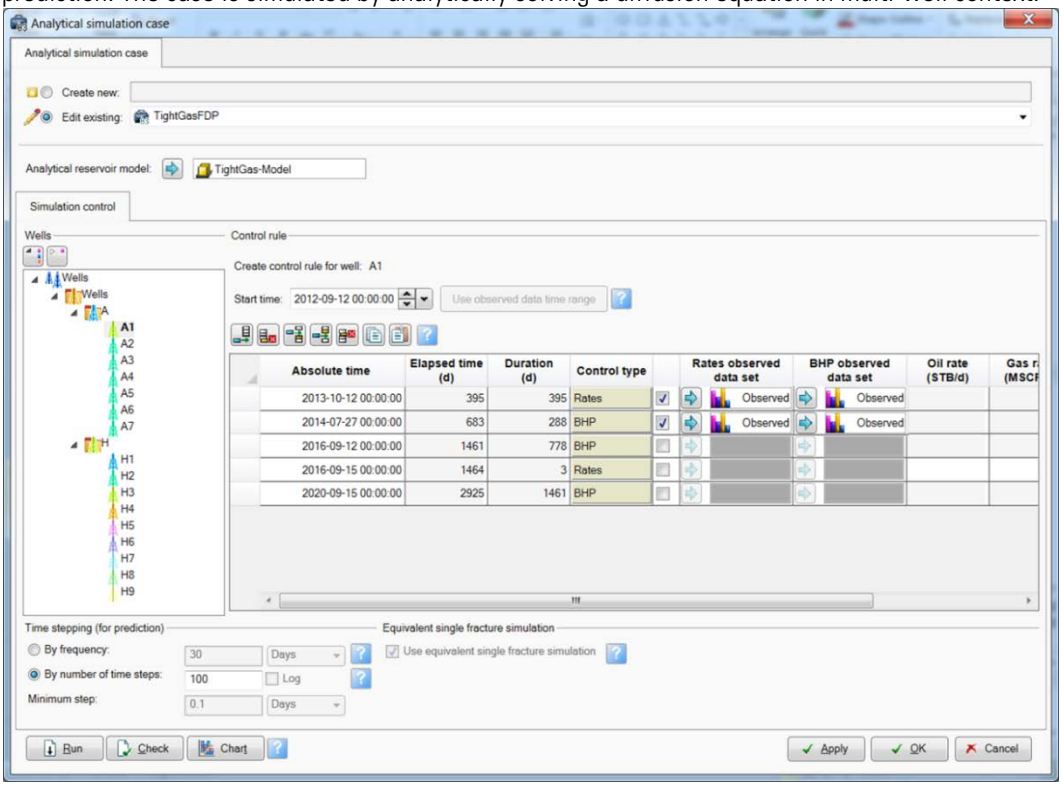
Create analytical simulation model

This model defines wells and completions, fluid and reservoir properties. An analytical reservoir model includes the static archive of wells and completions, fluid and reservoir properties. Petrel existing wells, completions, black oil fluid model and saturation function can be consumed. Alternatively, you can easily create wells, perforations, fractures from scratch inside PI. A simplified grid (1x1xN) is generated to represent a multilayer model geometry and properties.



Create analytical simulation case and run simulation

Select analytical model and define historical or predicted production scheme. Run analytical simulation for pressure and rate prediction. The case is simulated by analytically solving a diffusion equation in multi-well context.



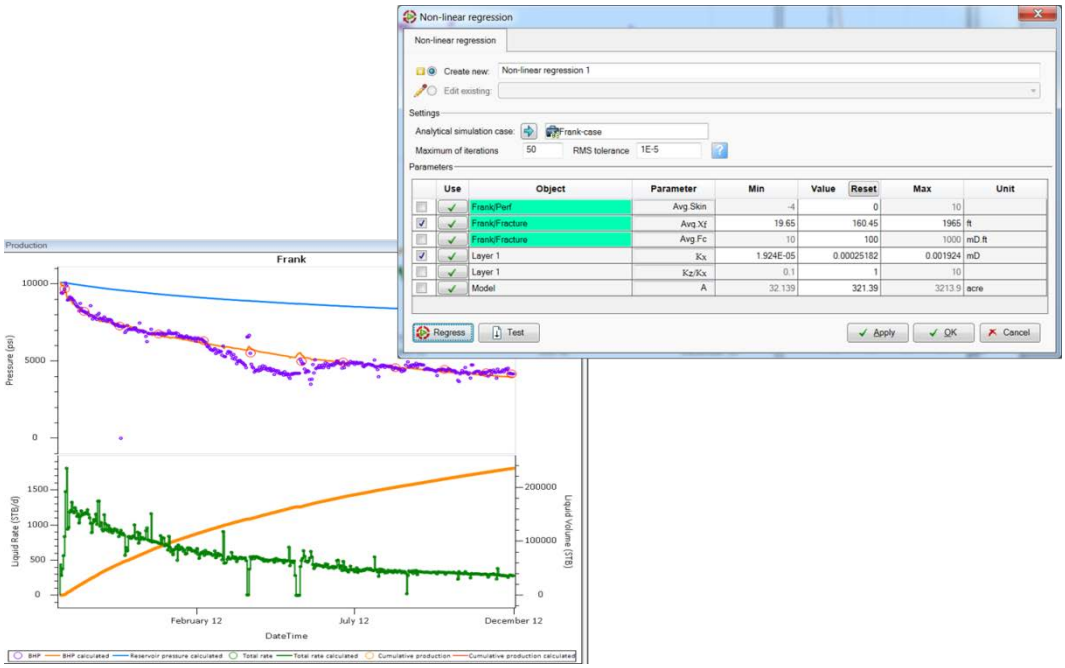
Rate transient analysis

The Rate transient analysis workflow allows you to quickly estimate key reservoir/well parameters and run an analytical simulation to match production. You can perform flow regime recognition on diagnostic plots. Industry standard and SLB developed diagnostic plots (for example, RNI diagnostic plot and Poe's plot) are provided. With a user-supplied analytical case, PI constructs diagnostic plots to aid in identifying the flow regimes with fit-line technique. Interpretation results can be used to update the model. You can run an analytical simulation with interpretation calibrated mode. Alternatively, PI supports user manual input for the parameters to run simulation.

Non-linear regression

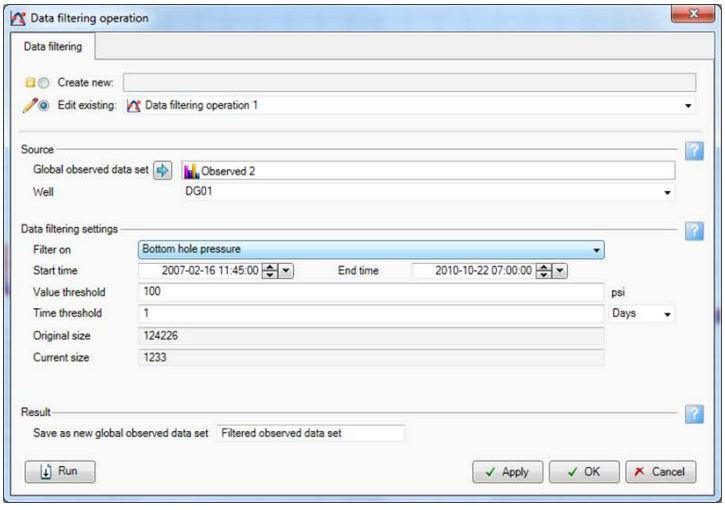
NLR (Non-linear regression) is a process employed to determine the most possible values of parameters that adequately model the observed data. The process consists of adjusting the analytical simulation model's parameters so that the simulation model reproduces the historic reservoir performance.

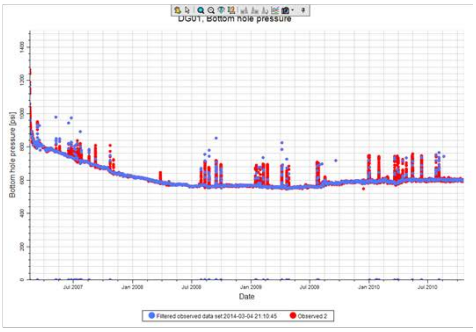
Regression points selection is the identification of the observed data where the observed-to-simulated matches need to be improved. A fundamental part of history matching is the selection of parameters to improve the match. It is essential, however, to ensure changes are within the model's geological uncertainty. A set of regression control parameters defines criteria for termination of regression.



Data reduction

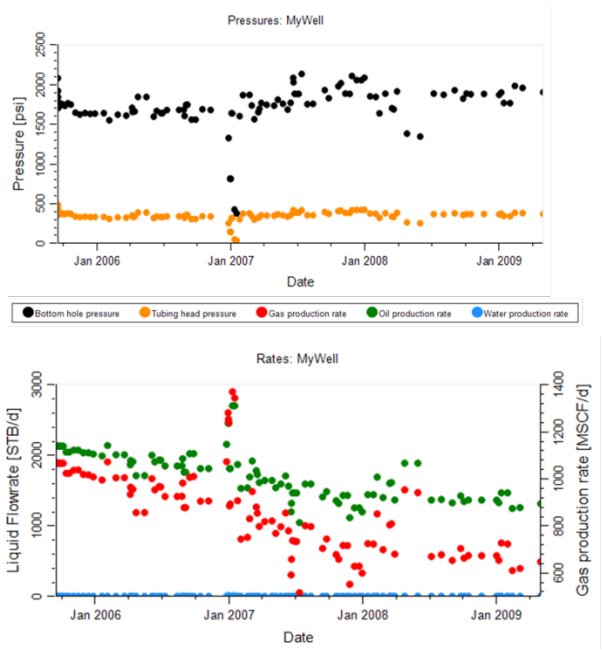
You can reduce data size through value and time threshold.

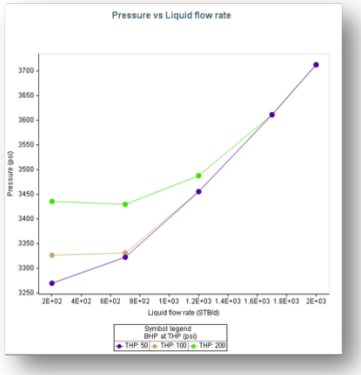




BHP computation

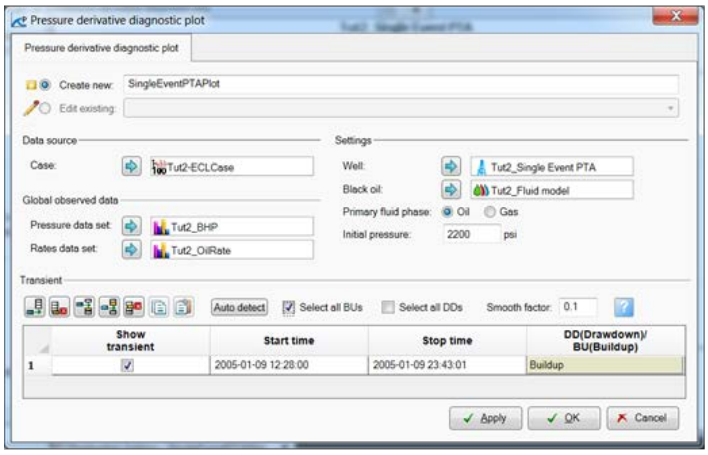
You can compute BHP from WHP and production rates using vertical flow performance (VFP) table. It applies to multi-well BHP computation. PI uses VFP table imported from ECLIPSE format keyword file or generated from the Petrel Well Deliverability module.

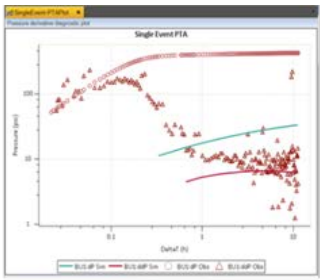


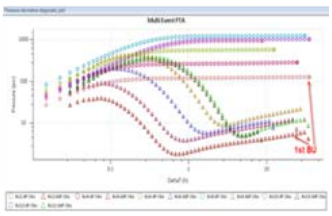


Pressure transient diagnostic plot

You can create pressure transient diagnostic plot from observed dataset, numerical simulation case or analytical simulation case. The pressure derivative diagnostic plot is used to spot trends in actual data that characterize well or reservoir system. Draw down and build up periods can be automatically detected. Alternatively, user can define specific transient periods to view the pressure transient plot overlay.

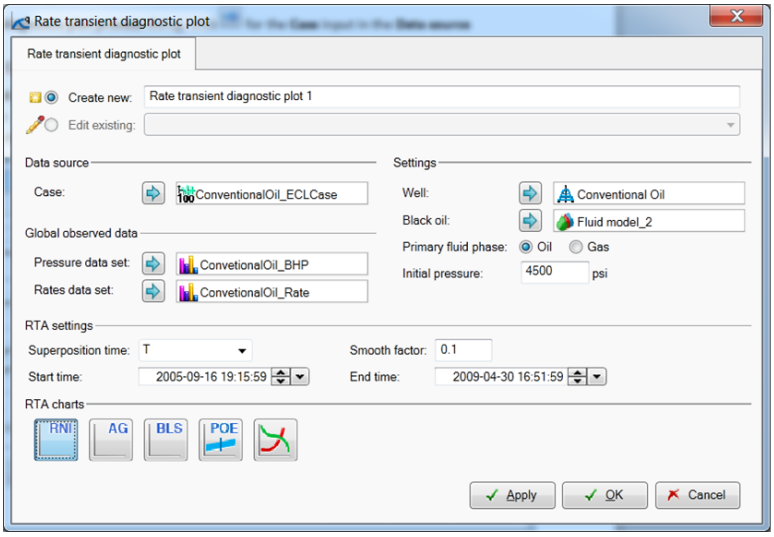


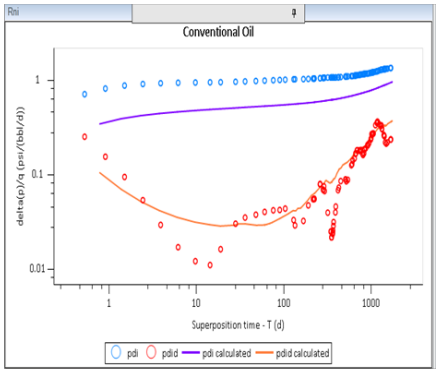




Rate transient diagnostic plot

You can create rate transient diagnostic plot from observed dataset, numerical simulation case or analytical simulation case.





Shale

2014.4

Raster Logs

Importing *.lic files with additional keywords

In Petrel 2014.4, *.lic files may contain additional keywords such as PAYPIX and DIGTRK, which are repeating keywords that refer to data, in the same way that DEPPIX does in the files. Importing *.lic files that include these keywords is now supported.

2014.3

Raster Logs

The performance for opening the "Advanced selection of files…" dialog while reconnecting multiple missing raster log files is improved.

2014.2

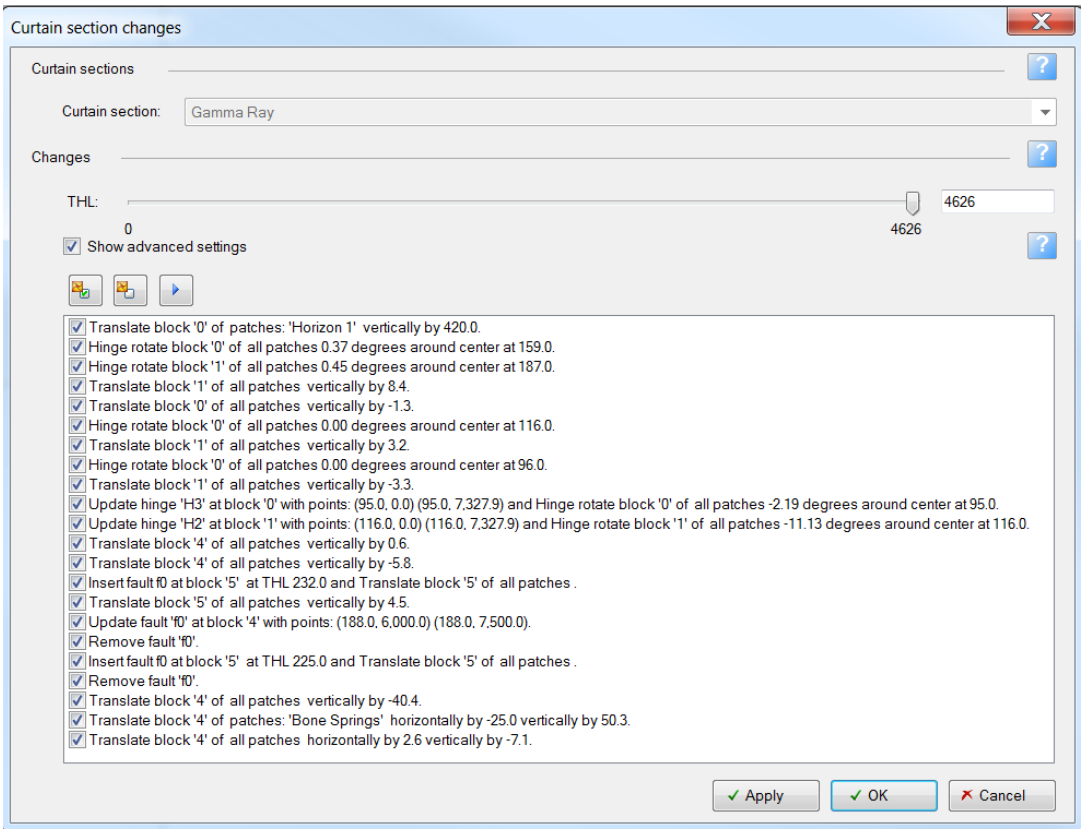
Geosteering

Creating a Curtain Section with Changes

You can create a cloned curtain section with your current changes at any point, while geosteering the well. The cloned curtain section can then be updated with one or all changes that you have made to your current curtain section.

During geosteering of the well, you may want to create a cloned curtain section of your current work to make changes to the current curtain section to compare two or more scenarios.

To create a cloned curtain section, go to the **Models** pane, navigate to the **Curtain sections** folder, and right-click the curtain section name with which you want to create a new curtain section with changes you have made. Select **Create curtain section with changes** from the right-click menu. A new curtain section is created in the models pane with the name "Clone of…" plus the name of the previous curtain section.



A **Curtain section changes** dialog box appears. The source curtain section is grayed out and set to the curtain section that you right-clicked. Set the THL limit for the curtain section changes to be applied, if needed, by dragging the slider or by entering a value in the box.

Check the **Show advanced settings** box to display the curtain section changes that you have made. Any **Automatic log matching** or **smoothing** done to the curtain section is not included in the changes and cannot be applied.

Click  to select all changes. Click  to unselect all changes. Click  to apply the next selected change. Changes can be undone by clicking on the **Undo** button in the **Geosteering window** toolbar or **Geosteering** tab (with the Ribbon activated). Click **Apply** once finished, or **OK** to apply the changes and exit. Click **Cancel** to close the dialog without making changes.

Applying Changes to Curtain Section

You can apply changes that you have made to the source curtain section to the cloned curtain section during geosteering of the well.

To apply changes to the cloned curtain section, go to the **Models** pane, under the **Curtain sections** folder and right-click the curtain section where you want to apply the changes that you have made to the source curtain section. Right-click **Apply curtain section changes** and the **Curtain section changes** dialog box appears. Select the source curtain section from the **Curtain section:** drop-down list.

Check the **Show advanced settings** box to display the curtain section changes. Changes made previously to curtain sections created with versions prior to 2014.2 are not included. Any **Automatic log matching** or **smoothing** done to the curtain section are not included in the changes and cannot be applied.

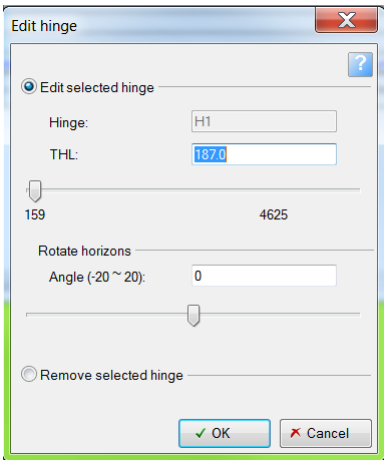
Click  to select all changes. Click  to unselect all changes. Click  to apply the next selected change. Changes can be undone by clicking the undo button in the **Geosteering window** toolbar or on the **Geosteering** tab. Click **Apply** once the selected changes are satisfactory, or **OK** to apply the changes and exit. Click **Cancel** to close the dialog without making changes.

Creating Blocks

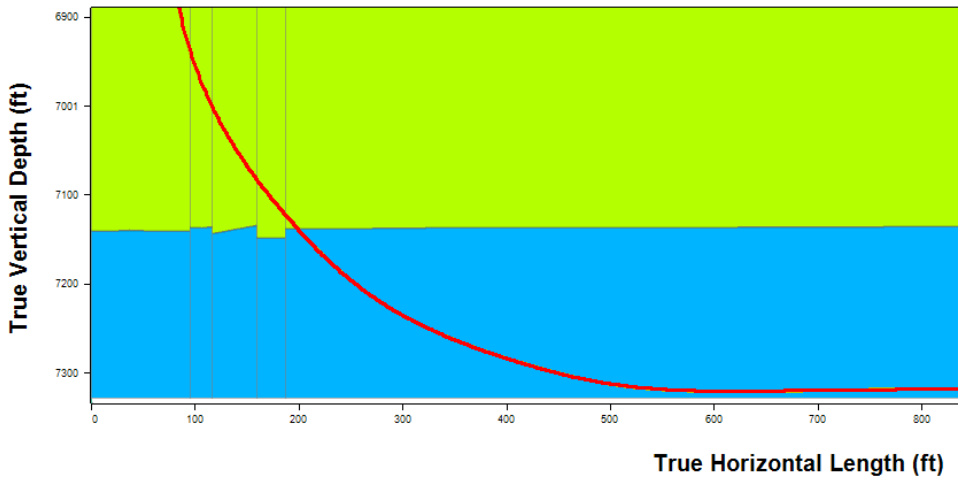
Hinges now appear as you select a point to rotate. Subsequent hinges create blocks that are rotated or translated individually. The hinges can be edited in the geosteering window. This is done by clicking a hinge and editing it in the **Edit hinge** dialog.

Blocks are now created as you rotate selected horizons. This allows you to better control the blocks dip and throw. Two hinges define a block, and as you click to rotate a point, a new block is created. A block can also be clicked on when you select **Translate Selected Horizons** to translate it.

To edit a hinge, in the **Geosteering window,** click the hinge that you want to edit or delete. The **Edit hinge** dialog box opens.



Rotate or **Translate selected horizons** or **Insert Slanted Faults** must be active for **Edit hinge** to work. The hinge line will only appear for hinge rotation.



Select the **Edit selected hinge** radio button**.** Use the trackbar to edit the THL of the hinge or enter a value in the **THL:** box. Use the **Rotate horizons** trackbar to rotate the horizons. Alternatively, you can enter a value in the **Offset Y (-100~100**) box.

Rotating horizons will rotate only the selected block.

To delete the hinge from the curtain section, select **Remove selected hinge** radio button. This does not delete the previous changes you have made with this hinge. Click **OK** to apply the changes or remove the hinge and close the dialog box.

Pad Placement

Deleting a Pad Configuration

You can delete a pad configuration in Pad Placement.

Created configurations can be removed from the pad configuration selection list. This action is irreversible whether or not you have saved the existing project, since the system removes it from the WellPadDesignPadConfigurations.xml file.

To delete a configuration from the **Drilling** tab in the **Pad well design** group, click **Pad placement**. The **Pad placement** process dialog box appears. Click the **Pad selection** tab. Click the **Create or add pad configurations** icon  .

Select a configuration to delete from the **Preview or edit existing:** drop-down list. Click the  icon to delete the selected pad configuration. A warning dialog appears, asking if you want to continue to delete the configuration from the configuration file. Click **OK** to continue the deletion, or **Cancel** to stop the operation.

Creating a New SAGD Configuration

You can create a new SAGD pad placement configuration or preview and edit the existing SAGD configuration. Creating a new SAGD pad configuration is similar to creating a new pad placement configuration.

To create a new SAGD pad placement configuration:

1. From the **Home** tab, select the **Shale Perspective**.
2. From the **Drilling** tab in the **Pad well design** group, click **Pad placement**. The **Pad placement** process dialog box appears.
3. Click the **Pad selection** tab.
4. Click the **Create or add pad configuration** icon  .
5. Click **Create new** to create a new configuration.
6. Enter the number of wells.
7. Check the **SAGD** check box if creating a SAGD pad configuration. If the SAGD option is checked, the preview display shows an injector/producer (I1/P1, for example) pair at the wellhead instead of well (W1, for example).
8. Click **Next**.
   Specify the following options:
   - Number of rows
   - Space between wellheads
   - Row space
9. Click **Next**. The **Wellhead** view is automatically displayed.
10. If you want to specify lateral directions for individual wells, click **Specify individual lateral direction**. Clicking this option displays **North** and **South** options that you can click for each well.
11. Customize or use the default values for the following options:
    - Horizontal spacing
    - Step out
    - Well length

    This is for preview display purposes only and is not used in the Pad placement **Pad Selection** dialog.
12. Click **Apply** to save the new pad configuration.

This configuration will appear in the **Preview or edit existing** menu. Clicking **Apply** automatically updates and saves the WellPadDesignPadConfigurations.xml file.

Buffer Zone Cost Function

You can create the cost function by entering in a buffer zone. Enter Min and Max values where the drillable area is between. Optionally, you can select the **Advanced Settings** check box to define a cost function following the previous workflow.

New Radial Pad Configuration

A new radial pad configuration is available for you to use. The radial pad configuration is in a radial pattern and based on a required angle.

Production Estimates

Use the **Production estimates** tab to estimate production and profit, based on typical P10, P50, and/or P90 production decline curves for your field or project, average cost of the well, and average price of oil or gas.

Drop in decline curves for P10, P50, and P90 by selecting the appropriate function and clicking  .These are functions created with a New Function object or imported. The template for x-axis must be set to time, and the y-axis template must be set to Oil production rate or Gas production rate. Set these templates in the **Function** tab of the Settings for the function.

The Scaling factor allows you to enter a reference length to use a linear scaling factor relative to this set reference length to estimate production for the new planned wells. Alternatively, a scaling function to define the scaling factor at a given lateral length can be defined.

The Production Period indicates how long you want to estimate production for each decline curve. You can estimate production for 1, 3, or 5 years, or output production estimates for all three time periods by checking the required boxes.

Profit estimates per well and per pad can be computed for each decline curve and production period. Indicate the average cost per well and average price of oil/gas. Profits are not estimated if the average price of oil/gas or the average cost per well is set to 0.

Once all the parameters are set properly, click **Estimate production**. An attribute is created for each decline curve and production period defined, as well as profit estimates on the Pad locations and Laterals that were generated previously. The attribute created is named Pxx-xYear for production estimates, and Pxx-xYearProfit for profit estimates. Right-click on the Pad locations and select Spreadsheet to view the results for each pad, or right-click on the Laterals and select Attribute Spreadsheet to view the results for each well.

Well Restriction Data Type

You can now use wells or all wells in a well folder as a restriction type. The restrictions are selected in the Geometric restriction fields in the **Regional geometry** tab of the **Pad placement** dialog box.

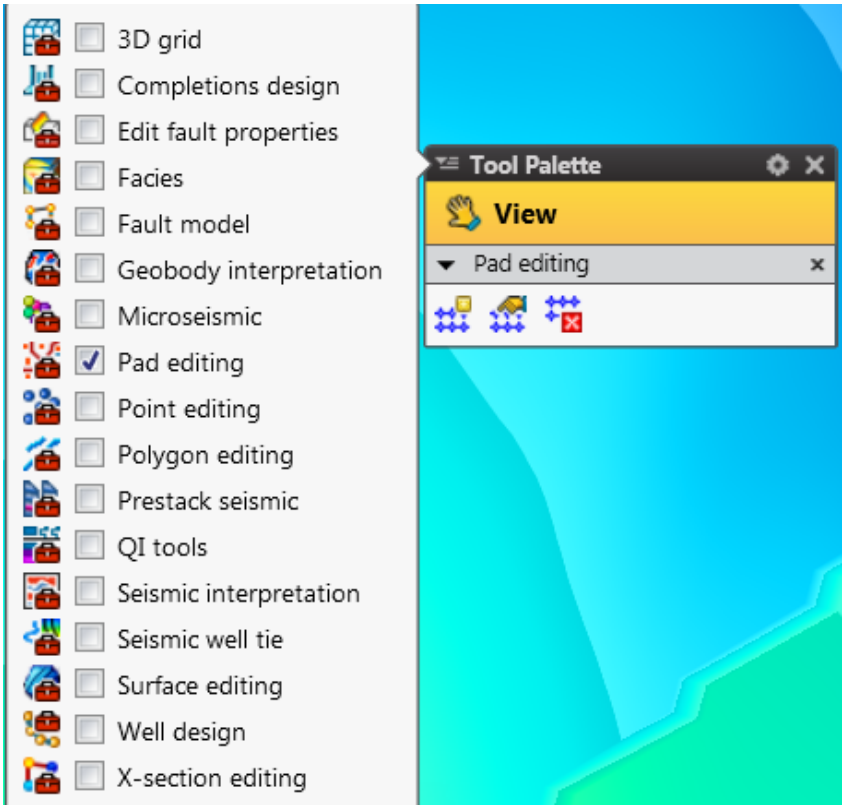
Restriction Data Type shows the type of restriction that has been dropped in the process. Any well or well folder can now be dropped and the restrictions will include all active plans or definitive surveys. If the surface applicable location is selected, the wellhead location is used as a constraint, and if the reservoir applicable location is selected, the full trajectory will be considered as a restriction.

Interactive Pad Functions in 2D windows

You can now create, edit, or delete new pads interactively in the 2D windows. This is achieved through using the new Tool Palette, which allows you to click a reservoir surface to create a new pad. Click an existing pad to edit the pad by moving interactively or updating the parameters or delete it.

To open the **Tool Palette:**

1. On the **Home** tab, select **Shale Perspective**.
2. From the **Drilling** tab, in the **Pad well design** group, click **Pad editing**. The **Tool Palette** appears for Pad editing.



To add a new pad:

1. From the **Tool Palette,** click **Add pad.** The **Pad specification** dialog box appears. Enter the required parameters, using the sliding bar or enter the value directly in the boxes. The pad will update interactively. These parameters are exactly the same as those used for the **Pad selection** tab of **Pad Placement.** See the online Help for this tab for more details on what each of the parameters is used for.
2. Click the 2D window to place the new pad where appropriate. Click and drag the new pad to the appropriate location. An active reservoir surface is required to place the pad.
3. Check the **Snap to ground location to grid** box to snap the new pad to the grid, or uncheck the box to place the pad wherever you want.
4. Click **OK** to apply the changes and close the dialog box.

To edit a pad:

1. From the **Tool Palette,** click **Edit pad.**
2. In the 2D window, select a pad to edit. The **Pad specification** dialog box appears.
3. Enter the required parameters, using the sliding bar or enter the value directly in the boxes. The pad will update interactively. These parameters are exactly the same as those used for the **Pad selection** tab of **Pad Placement.** See the online Help for this tab for more details on what each of the parameters is used for.
4. Click and drag the new pad to the appropriate location, if required.
5. Click **OK** to apply the changes and close the dialog box.

To delete a pad:

1. From the **Tool Palette,** click **Delete pad.**
2. In the 2D window, click a pad to delete. The pad is automatically deleted. There is no undo function for this, so be careful when using this tool.

Raster Logs

The Raster Log module is now licensed to either Well Correlation or Shale Suite. Twenty-four-bit depth images are now supported in the module.

Studio

Studio: Studio Client Petrel

2014.3

Annotate

You can now reorganize and move your annotate objects (e.g., notes, attachments, etc.) into several folders within the Annotate folder.

Find

Checkshot and Maximum inclination of a Well have been added to the repository index and can now be found and filtered on by using Studio Find in Petrel.

2014.2

Synchronization

To accommodate input that is less than an hour, the synchronization interval has been enhanced to accept a decimal value with the minimum of 0.05 hour.

Annotate

Annotate functionalities (such as, send to email, send to blog, etc.) are now compatible with Microsoft Office 2013.

Find

Studio search button

The Studio search button has been enhanced to be a state button. Click the button once to launch Find, click the button again to close Find.

Petrel frame of reference filter

You can use the “Petrel frame of reference” filter to narrow down the search results to only show data from Petrel data sources indexed in version 2014.1 and above with the same or a different frame of reference.

2014.1

Find

View Content in Search Results

A new view content display option has been added to Search Results.

With the view content display option, you will be able to preview the search results (for example: preview a surface, window contents, etc.) before loading them to Petrel project.

Indexing Petrel Windows

Petrel windows now can be searched and retrieved through Find.

Petrel windows will be indexed and added to default index during project indexing. They will be available in the search result and can be retrieved for you to reuse so that you don't have to set up new windows with the same display and settings. Windows that are open during indexing will have their thumbnails created and displayed under search results (with the content viewer display option or by opening the detailed information on the search result) so you can identify the contents easily.

Related Items Filter

A new related items filter has been added to allow users to search for data items related to an object through Find.

With the related items filter, you will be able to search all annotations attached to a specific object, all windows that contain a specific object, and all objects being used in a specific window that have been indexed.

Search Results Color Code Enhancements

The ability to use Petrel native and custom color tables to color code search results has been added.

You now have the ability to apply Petrel native color table or create a custom color table to be applied to discrete filters. Corresponding search results will be color coded accordingly, based on the specified color table. This feature is only available for the Find in Petrel interface.

Square Maps Display for Search Results

Search data is now displayed in square maps for visual analysis.

You have the ability to apply square charts to a filter category with numeric value. The search data will be displayed as square maps for visual analysis – where the size of the squares reflects the attribute values.

Filter the values of Filter Category

A text filter has been added to narrow down the list of values of a filter category.

You will be able to filter the values of a filter category based on a certain keyword. The list of available filter values will be updated and the values that match the keyword will be listed in bold at the top of the list.

Studio: Studio Server

2014.3

Data Transfer

Automatic re-parenting of seismic cubes and interpretations

Automatic re-parenting of seismic cubes and interpretations is now available during the transfer of superseding seismic survey from the repository to the Petrel project. The data administrator can supersede an existing small seismic survey to a new larger survey in the repository. This automatic re-parenting feature is optional - if enabled, when transferring the superseding survey from repository into the Petrel project, Studio will automatically re-parent the cubes and the 3D interpretations of the superseded survey to the superseding survey.

Automatic user attributes matching for well data

Automatic user attributes matching for well data is now available during data transfer to prevent proliferation of user attributes with different GUIDs. This feature is optional and can be enabled on your Studio repository by your Data Administrator. If enabled, well user attributes that have the same attribute name, attribute type, and template GUID, but a different attribute GUID between Petrel and the repository, will be matched during data transfer. The current automatic user attribute matching only covers a one-to-one scenario.

2014.1

Supported Data

Additional information is being preserved in Studio. The following additional information is being preserved in and can be retrieved from Studio:

1. Shapefiles with holes
   Petrel now supports display of holes for imported shapefiles. The hole information will be preserved when the shapefile is round tripped through Studio.

   Note: Displaying a shapefile with holes in a 3D window might cause the interface to flicker.
2. Pre-defined log attributes
   - Log alias
   - Log type
   - Log source
   - Log version
   - Run date
   - Sample rate
   - Tool type
3. Well uncertainty properties
   Uncertainty properties for well head position and ground offset:
   - Seabed or ground level elevation to MSL (Z unit)
   - Uncertainty of well head location – Radius (XY unit)
   - Uncertainty of well head location – Standard deviation(s) (unitless)

4. Well and Drilling
    - Support for multiple surveys per well
    - Definitive survey flag
    - Survey program
5. Multi-Z interpretation
   Multi-Z interpretation is now supported by Studio and available in all Studio workflows (transfer, find, filter subscriptions, data tables, etc.). Plane information, dip and azimuth information will be preserved in Studio. Normals are required to be assigned for the multi-Z to be transferable to Studio.

Technology

Technology: General

2014.3

Polygon Dragger

Petrel 2014.3 introduces significant improvements to polygon editing in the **3D** window by enhancing the usability of the dragger to move knee points in 3D space.

Technology: Licensing

2014.3

Profiles

Petrel 2014.3 introduces support for global license profiles by specifying a path to a global license profile file in the PetrelConfiguration.xml file.

Technology: Visualization

2014.3

Visualization performance

Petrel 2014.3 introduces significant performance improvements when displaying large number of well logs and further improves the performance of visualizing well-related data. Depending on your graphic hardware, you can now work with fully interactive 3D views that display thousands of well logs.

Technology: User Experience

2014.2

Pane behavior when in auto-hide mode

A new pane auto-hide and show behavior has been implemented in Petrel 2014.2. In order to bring out panes that are

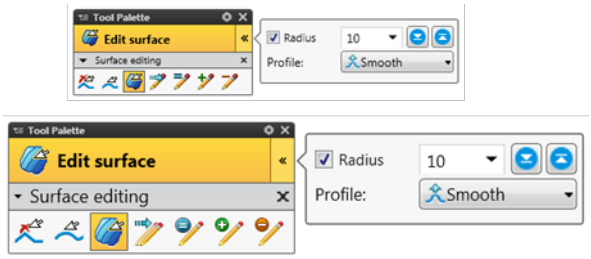
set in auto-hide mode and are collapsed in the left-hand side margin of Petrel you now need to click the tab of interest. This differs from previous Petrel versions where collapsed tabs auto-showed when hovered over by the mouse. As before, the tab will auto-hide again once you click in a non-pane location.

Tool palette

Support an option to switch between 16px Elements and 32px Elements

A large view option has been added to the tool palette for increased readability and usability on high-resolution monitors.
Available in the tool palette settings area the new 'Large view' toggle allows you to choose large icons and text to be used in the tool palette and its callouts.

Figure 126: Normal versus Large tool palette views



Support Styling of Contextual Tabs and Tab Groups

Contextual tabs have been implemented with a full vertical color band in order to better discriminate them from the domain tabs.

Save light tool configurations

Light tool configurations are now saved within the 3D Window folder in the Windows pane. They can be renamed or copied to other 3D windows.

Spot light

Petrel 2014.2 re-introduces the spot light functionality fully integrated into the Light Tool with controls to move and adjust the spot light. Spot light configurations can also be saved and shared through the 3D Window folder in the Windows pane.

Decoration tool in the 2D and 3D window

The rotate and zoom "thumbwheels" are reintroduced to allow single axis scene manipulation, specifically useful for optical stacking workflows on seismic volume rendering.

2014.1

Multi-Frame Display

Global cursor tracking and camera linking

You can now quickly enable cursor tracking for all your display windows, whether or not they are currently active or visible. This is useful when you use window layouts to view the same data from different perspectives or when you create a plot window with multiple viewports. To enable cursor tracking for all windows, click the **Track cursor** split button, then click the **Track all cursors** pull-down option. To disable from all windows, click the split button and select the **Don't track any cursors** pull-down option.

You can also synchronize the camera position in all your 3D windows, so that rotating and scanning your model in the active 3D window will have the same effect in other 3D windows. This is useful for plot windows as well as when your main display is tiled. To enable camera linking in all 3D windows, click the **Link camera** split button and select the **Link all 3D cameras** pull-down option. To disable from all 3D windows, click the split button and select the **Unlink all 3D cameras** pull-down option.

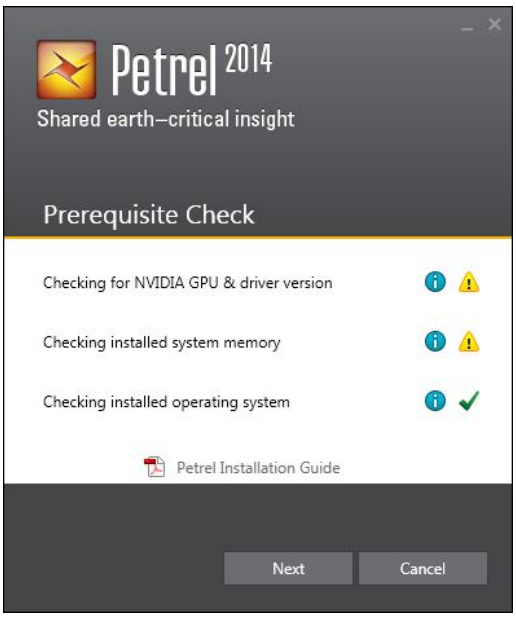
System enhancements

Petrel 2014.1 introduces a new installer that performs basic hardware and environment checks, including:

- Validation of the minimum required Graphic Card driver version
- Validation of the minimum required physical RAM
- Validation of the supported OS version

In addition to feedback on basic system requirement, the installer provides links to more detailed descriptions of the recommended hardware setup as described in the *Petrel Installation Guide*.

Figure 127: Sample results of the installer system checklist

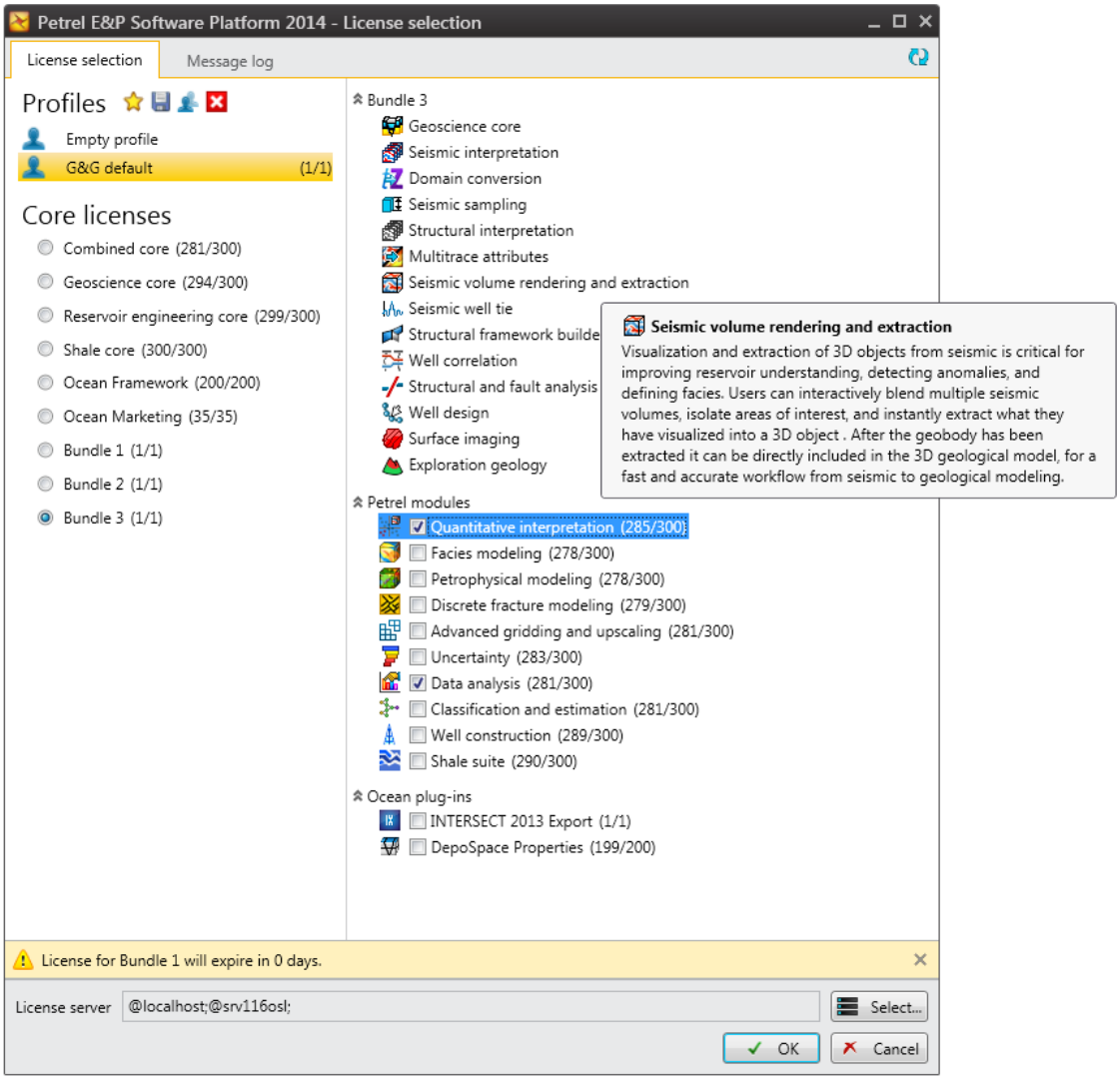


License selection user interface

Significant improvements in the license selection interface mean that you can now mix Stack-on-Start and License Packages (Bundles). This lets you top up License Packages with individual Stack-on-Start license features. You can also save selections as License profiles and hover over features to see descriptions. And we included a new message log tab, for improved error handling when selecting licenses.

Figure 128 illustrates the selection dialog when the core license is set to **Bundle 3**. This license choice extended the **Quantitative interpretation** and **Data analysis** features selected in the **Petrel modules** section. The selection is saved as a profile named **G&G default**. Also visible in Figure 128 is the mouse-over feature describing seismic volume rendering and extraction. The message log in this figure illustrates the kind of warnings you might receive while connecting to a license server.

Figure 128: The 2014 Licensing selection dialog, showing mouse-over descriptions and message log reminder



2D Scale bar

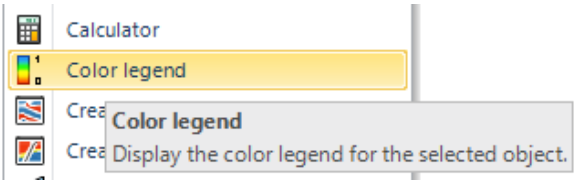
There is now an auto-adjusting, configurable scale bar object for the 2D window. As you zoom in and out in the display, the Scale bar automatically adjusts scale and unit. To change color, position, and font settings, double-click the Scale bar object in the **Windows** pane.

Figure 129: Scale bars appear by default in the lower left corner of a 2D window

Color legend

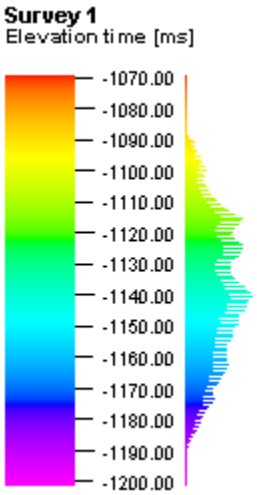
A new object-related **Color legend** feature complements the object-independent **Auto legend**. A color legend can be added by right-clicking an object in the tree and clicking **Color legend**.

Figure 130: Launching the Color legend from an object's context menu



In addition to showing the color distribution of the object, the color legend displays a histogram by default. The histogram is enabled and disabled from the color legend's Settings dialog or by right-clicking the color table in the display and switching the histogram on and off.

Figure 131: Color legend and histogram, associated with a surface



Color tables

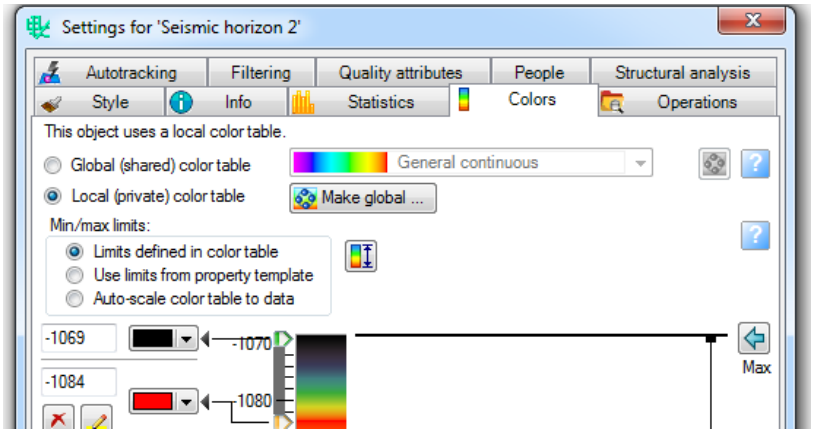
Petrel 2014.1 significantly improves color table handling and assignment by decoupling color tables from property templates. This allows you to change color tables for individual or multiple objects without creating a new template. Color tables can exist as local color tables applied to a specific object or as global color tables when applied to multiple objects. Options to create local color tables or promote them to global color tables are also available. All necessary color table operations, including scaling, can now be done directly from the object's Settings dialog, on the Colors tab or by using the Inspector.

There are three default scaling options:

- Limits defined in the color table
- Use limits from property template
- Auto-scale color table to data

Default min/max value ranges can also be defined in the property template, but they will be superseded by settings in the color table.

Figure 132: Colors tab in the Settings dialog of a seismic horizon



In addition to more flexible color table handling, Petrel 2014.1 introduces an easy way to select color tables from an extended list of more than 50 domain-specific color tables. Preset color tables are organized in a structured hierarchy in the **Templates** pane, as illustrated in Figure 133. Notice that the **Recently used** lists displays the last color tables selected. Click **More** to select from a searchable color table selector, as shown in Figure 134.

Figure 133: The color table selections available from an object's Settings dialog, on the Colors tab

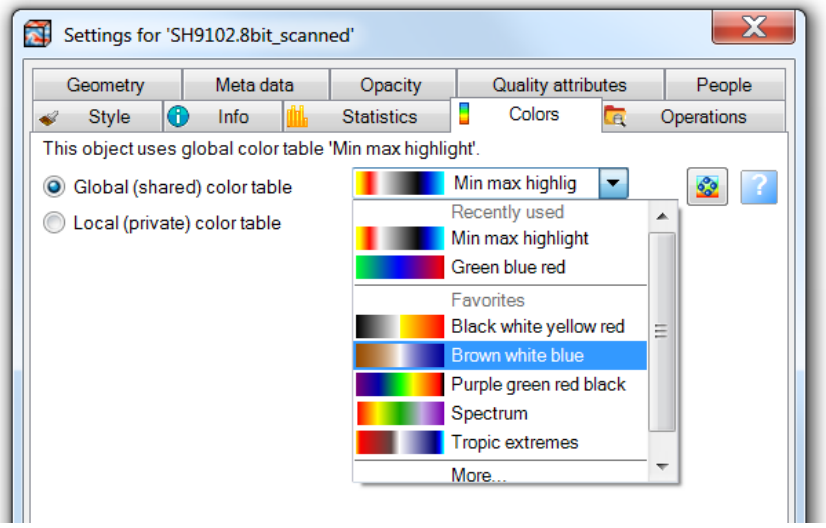
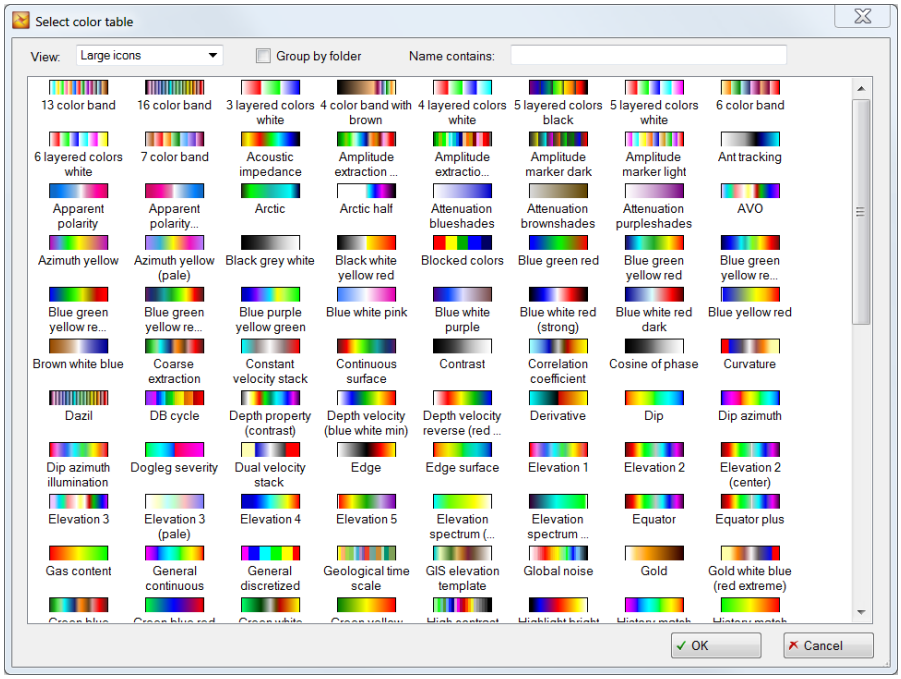


Figure 134: Searchable color table selector with different view options, from details to large icons



Visualization performance

Petrel 2014.1 introduces significant performance improvements when displaying large number of wells and well-related data, such as well trajectories, well tops, well symbols, and well labels. Depending on your graphic hardware, you can now work with fully interactive 3D views that display tens of thousands of well trajectories.

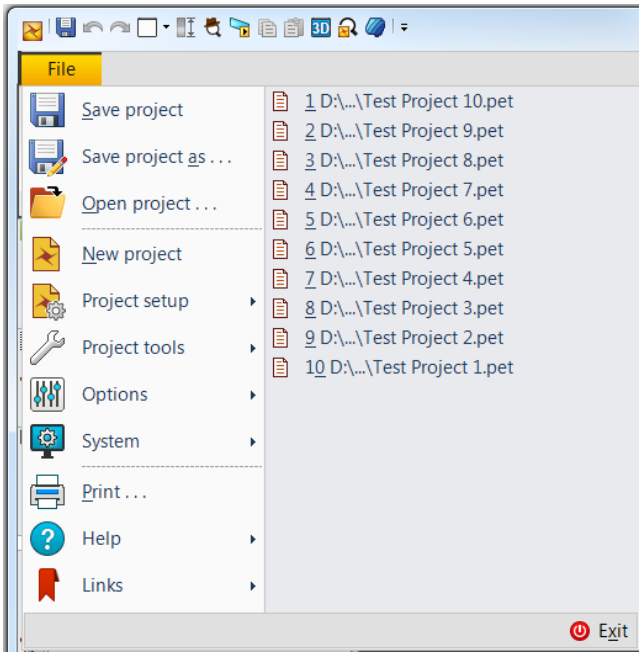
Polygon rendering and editing has been optimized, making it easy to select and edit large numbers of polygons that represent cultural data or interpretational boundaries, while maintaining full interactivity in the 3D or 2D window.

File Application Menu and Home Tab

As part of 2014 user experience improvements, common features and tools can be accessed independently of the selected perspective by clicking the **File** or **Home** tab.

The **File** application menu provides centralized access to project and system tools and information which was distributed over several menu items in previous versions of Petrel.

Figure 135: File tab menu gives quick access to recent projects as well as project and system settings



When you start Petrel, the **Home** tab is selected by default. It contains common tools and operations, including:

- View options for the Inspector, players, and panes
- Insert options for data, objects, and folders
- Petrel and Studio search features
- Data management tools for Studio or Reference Projects
- Notification and clipboard operations

Figure 136: Petrel ribbon, showing Home tab features

Some of the tools available on the **Home** tab are duplicated in the Quick Access toolbar. This means you can still have quick access to commonly used features when the ribbon is minimized. And the Quick Access toolbar can be customized.

Output sheet

In Petrel 2014.1, the Output sheet is saved with the project as a window, meaning that reports of results or residuals can be saved, and can be made part of a Layout.

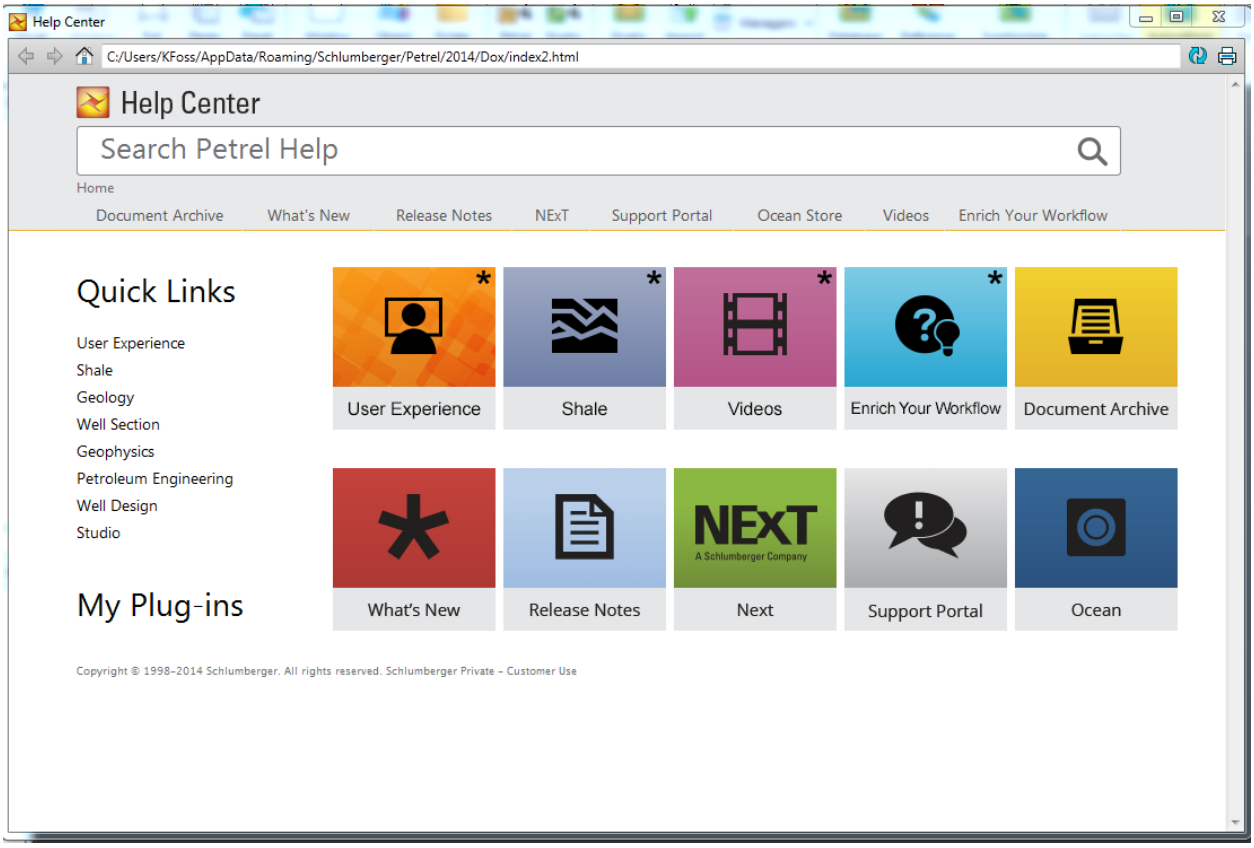
As a side effect of this change, Workflows that set and use visual filters may no longer work as they did previously. In order to recover them the window in which the filter is applied must be explicitly activated in the Workflow using the "Set active" command.

Technology: User Assistance

2014.1

Help Center

Petrel 2014 introduces many enhancements to the Petrel Help Center:



We continue to evolve our User Assistance offerings by moving towards a robust system with multiple entry points for every stage of use, giving you everything you need to fully understand your workflow.

In 2014 we introduce a new **Videos** tile containing videos detailing the new User Experience and Interface in Petrel. Five detailed videos are delivered with Petrel and several dozen more, covering all domains, and are available from the Software Download Center, accessible from the Help Center.

We have also added main tiles for the new User Experience and Shale. Clicking these takes you straight to the main landing pages where you can quickly get up to speed with new functionality and content.

Enrich Your Workflow

Enrich Your Workflow is a new feature in Petrel 2014 that gives you the information you need to get the most out of your Petrel experience. Petrel offers many workflow enhancements through plug-ins, but with so many offerings, it can be hard to know which plug-ins are right for you and how compatible licenses and plug-ins can enrich your day to day workflows. So we have taken the guess work out of it. For each domain, we tell you what other licenses would be beneficial for your work based on your current perspective and licenses in use. In addition, Petrel 2014 achieves the next level of seamless integration between Ocean plug-ins and the Petrel Platform by providing you with links to relevant Ocean plug-ins that will specifically enrich your workflow. From the Help Center, you can click through to the Ocean Store where you can utilize the new **Try Me** functionality that enables you to test drive plug-ins from the cloud, giving you a sense of how your experience can be enhanced. Finally, we recognize that in the fast-paced environments of oil and gas, there is always something to learn, so we provide links to relevant training courses from NExT that will add to your domain expertise.

The implementation of Enrich Your Workflow showcases the solution diversity and flexibility of our Platform, all while enhancing your experience as you get more out of Petrel than ever before.

Other Features

The Help Center also features a Document Archive, Release Notes, What's New Guide, Ocean Store, Support Portal, and NExT training.

Throughout the 2014 Petrel platform, we will be adding new and useful features to the Help Center that will enable you to get the Help you need, when you need it, for all aspects of your workflow.