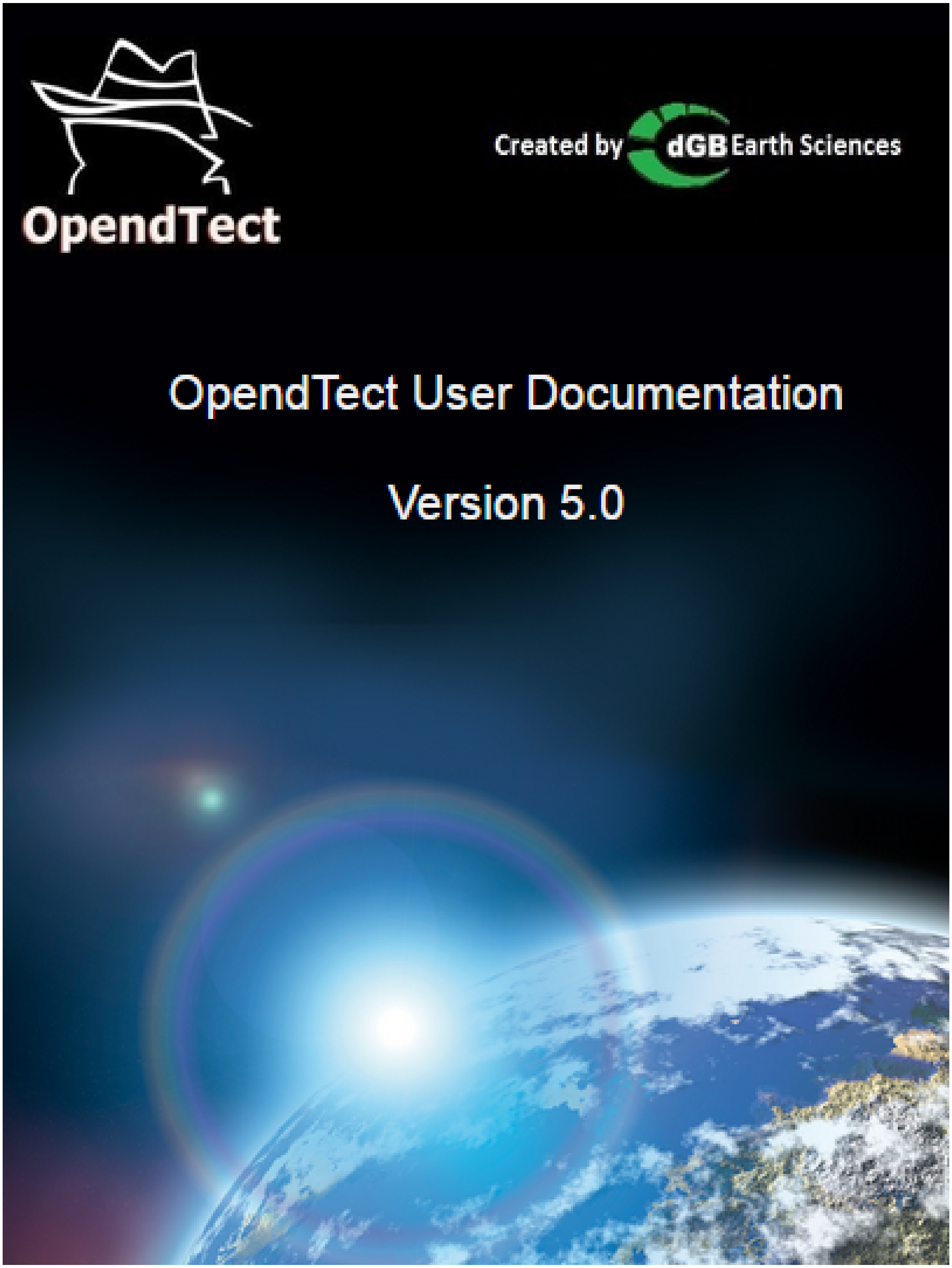

OpendTect
Created by dGB Earth Sciences
OpendTect User Documentation
Version 5.0

Table of Contents

OpendTect User Documentation - Version 5.0

dGB Earth Sciences

Preface

OpendTect

About this Manual

This manual combines primer, reference guide, and theory into one document. New users are advised to read the Getting Started section, which acts as a primer to the system. Detailed information on windows and parameter settings are described. Advanced users may also look at some of the Appendices. Appendix A describes attributes and filters, what they do, and what parameters can be set.

This document was written using the online Documentation Tool system. Two versions are published: an html manual for online use and a pdf version for printing. The html manual comes with the OpendTect software when it is downloaded from the Download Center. Both the html and pdf manual can be separately downloaded from the OpendTect website: http://www.opendtect.org/index.php/support.html

While every precaution has been taken in the preparation of this manual, it is possible that last minute changes to the user interface are not reflected in the manual, or are not described accurately. Please help us improve future editions by reporting any errors, inaccuracies, bugs, misleading or confusing statements you encounter.

Please note that apart from this user manual, there is also an Application manager manual. These can be found in the doc/SysAdm/base directory or the software installation directory. Commercial plugins are not described in this manual but in separate Help files.

Release Notes

This is the user documentation for release OpendTect v.4.6 - an open source post-processing, and seismic interpretation system launched by dGB.

OpendTect is an open source software system that is released via the internet. Users can download the software from the OpendTect website.

OpendTect system is a complete seismic interpretation system that can compete with any commercial system available on the market to date.

OpendTect V4.6 is released under a triple licensing strategy:

1) under the GNU GPL license.
2) under a Commercial license.
3) under an Academic license.

Under the GNU / GPL license OpendTect is completely free-of-charge, including for commercial use.

The Commercial license enables the user to extend the system with (closed source) commercial plugins that can either be purchased or leased.

Under the Academic license agreement universities can get free licenses for OpendTect and commercial plugins for R&D and educational purposes.

OpendTect is currently supported on the following platforms:

- PC-Linux 32/64bit
- Windows 8, 7, Vista, XP (32/64bit)
- Mac OS X 10.5

About OpendTect

OpendTect is a seismic post-processing and interpretation system, as well as an R&D platform, for developing innovative seismic interpretation tools.

The system is released under a triple licensing strategy:

- under the GNU GPL license.
- under a Commercial license.
- under an Academic license.

For more information go to dGB's OpendTect download center.

This manual describes the functionality in the OpendTect system. Manuals for commercial plugins have their own help files.
The manual may contain figures with dGB plugins like HorizonCube, SteeringCube, Neural Networks etc ... However, these commercial plugins are described individually in different sections in the dGB-plugin documentation.

OpendTect system features:

- Visualization and analysis of 2D and 3D seismic data in a single survey.
- Sophisticated 2D and 3D horizon tracking including auto-tracking, plane-by-plane, line and manual tracking.
- On-the-fly calculation and visualization of various attributes and filters.
- Multi-platform distributed computing
- Plug-in architecture.

Other features in OpendTect:

OpendTect is built with MSVC 2005 starting from version 4.0, allowing for better integration with the third party libraries and stability on Windows Operating Systems. New features have now been made available, previously unavailable for versions built with Cygwin. OpendTect is available for both 32 bit and 64 bit Systems for Windows.

Installation and directory structure: The installation and directory structure are entirely different from the previous releases. Unlike the previous versions, OpendTect is a complete Windows application and does not require any Unix shell scripts for its operation. The console window is no longer necessary for OpendTect on Windows. OpendTect can automatically find its current installation folder and load the data, plugins and icons from respective folders. Use of DTECT_WINAPPL environment variable no longer needed.

Progress Windows: The launching of batch programs and view progress windows is now implemented in similar manners to that of Linux, which means the application has almost the same features on both Unix and Windows (Previously, Cygwin Installation could not launch view progress window).

User Messages: User messages have been standardized, containing relevant text on the buttons rather than only yes/no options.

Implementation of a new Axis Drawer: An axis drawer has been implemented with better looking arrows and annotations. The new Axes will not interfere with the other 3D objects in the scene. It is now possible to take the snapshot of the scene along with axes, allowing the user to understand the orientation of the survey in the snapshot image.

Implementation of Top Bottom Image: Users can add rectangular Images on top or bottom of the survey box with transparency. This feature is useful if users want to add any reference map of the survey so that they can understand the geographical position and corresponding seismic profile in a better way.

View Orientation: A View North, and View North-Z options are available to the Graphical Tools toolbar along with View Inline and View Crossline buttons.

Wacom digitizing tables

OpendTect is the first seismic interpretation system to support Wacom digitizing tablets. Workflows for horizon tracking, fault interpretation, drawing of polygons & bodies etc... have been adapted to benefit from the superior hand-eye co-ordination offered by the pen device.

For a video demo, click here

Copyright

Acknowledgements

The OpendTect system is developed around concepts and ideas originating from a long-term collaboration between dGB and Statoil. Most of the system was and is developed through sponsored projects. We are indebted to all past, present and future sponsors. To name a few:

- Addax
- ARKCLS
- BG Group
- Chevron
- ConocoPhillips
- Detnor
- DNO
- ENI
- GDF Suez
- Geokinetics
- JGI
- Marathon Oil
- MOL
- OMV
- RocOil
- Saudi Aramco
- Shell
- Statoil
- Talisman
- Tetrale
- The Dutch Government
- Thrust Belt Imaging
- Wintershall
- Woodside

Getting Started

Table of Contents

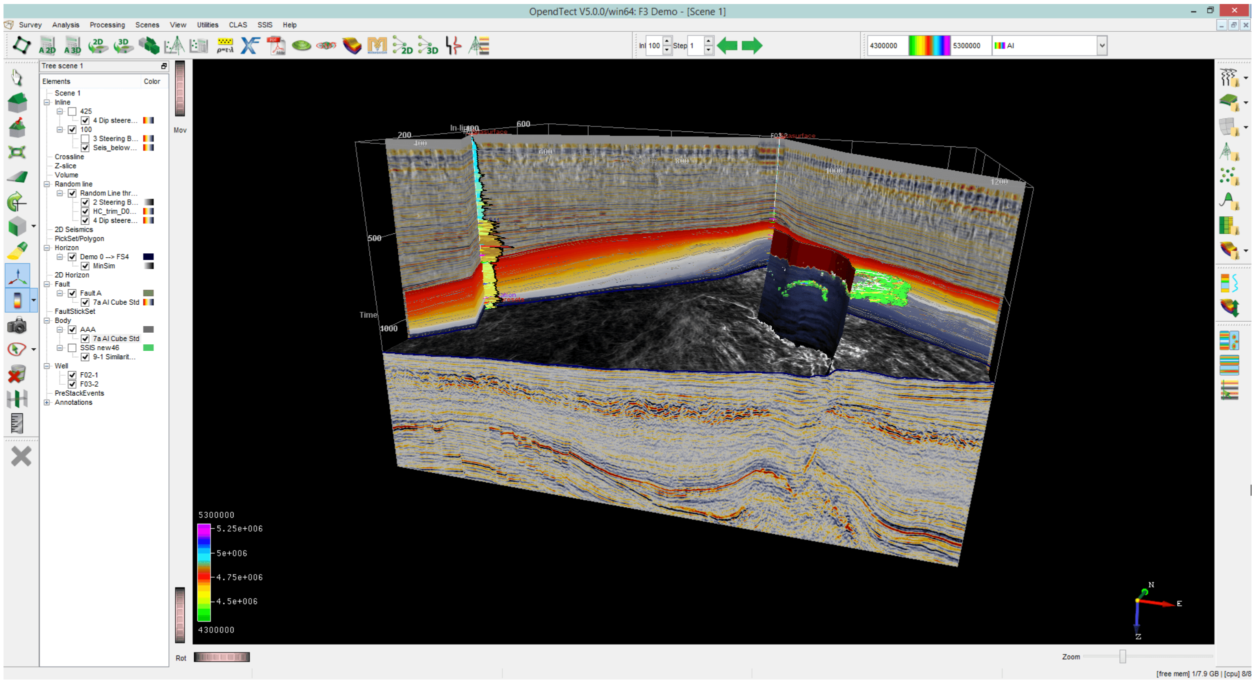
OpendTect

System Overview

OpendTect contains several commercial and non-commercial plugins that have their own menus/icons/toolbars.

Commercial plugin tools appear on the main window if the licenses are available.

The main user interface is shown below. The interface is comprised of main menu bar, several toolbars (OpendTect, Graphical, Tracking, Color Table etc), a tree scene control, and a 3D scene. Within the user interface, a scene contains the displayed data and data is loaded as an element of the tree. The important toolbox is an attribute set window, which is used to define seismic attributes as an attribute set. Such set can then be used to display attributes on-the-fly, or to create an attribute output. The attribute set can also be combined to generate a Meta-attribute. Meta-attributes can be predicted by using Neural Networks, mathematical manipulations and/or logical operations. ChimneyCube, SaltCube, FaultCube are our *Patent Meta-attributes*.



OpendTect main user interface including commercial plugins:

Just some of the functions for which OpendTect can be used:

- To perform seismic data interpretation.
- To create advanced single-attribute cubes.
- To filter the seismic data.
- To track and filter horizons
- To combine attribute and filter responses by maths and logic (IF..THEN..ELSE, ..AND.., ..OR.. etc.)
- As a platform to develop new applications or plug-ins.

Projects in OpendTect are associated with a seismic survey, which must be defined at the start (File - Survey). The survey specifications define the geographical boundaries of the study area and the relationship between inline/cross-line and X/Y co-ordinate system. Seismic data is loaded from external sources (e.g. SEGY) into OpendTect's CBVS (Common Binary Volume Storage) format.

An important thing to remember is that OpendTect works with an *active attribute set* (You can auto-load an attribute-set which will be active the next time you open the survey. Do the following: open an attribute set window, then select *File > Autoload Attribute set*). Only attributes in the current active set can be used to make displays. You must select an existing attribute set or create a new one before you can apply attributes. If you wish to test a different attribute or, in case you simply wish to change the parameters of an attribute, you can do so by modifying the current attribute set.

Interactive testing of attributes is a key benefit of OpendTect. Attributes and input data can be visualized in multiple graphic windows ("Scenes") using multiple plane viewers (inline, crossline, randomline, Z-slice). Each scene has its own graphic tree that controls the elements that can be switched on/off. The plane viewer dimensions can be re-sized to reduce calculation times. The attribute specification window supports a direct application mode for fast testing of attribute parameters. Press the "Swift" icon to apply the current attribute to the currently selected active element. Up to eight different attributes can be displayed in one element (inline, crossline, 2D line, Z-slice, randomline).

Multi-Threaded Programming

A thread of execution is the smallest unit of processing that can be scheduled by an operating system. It generally results from a fork of a computer program into two or more concurrently running tasks. The implementation of threads and processes differs from one operating system to another, but in most cases, a thread is contained inside a process.

Multithreading allows multiple threads to exist within the context of a single or multiple process. These threads share the process' resources (such as memory) but are able to execute independently.
OpendTect uses the power of multithreading to run different applications in data loading, visualization, processing etc ..

Multithreaded:	**Not multi-threaded:**
BG Steering*	Constant Steering
Central Steering	Convolve (Wavelet option)
Convolve (all except Wavelet option)	DeltaResample
Curvature	Energy (Gradient option)
Dip Angle	Event Steering
Velocity Fan Filter (=DipFilter)	FaultDip
Energy (all except Gradient option)	FingerPrint
Event	GapDecon
FreqFilter*	HorizonCube Curvature
Frequency*	HorizonCube Dip
Full Steering	HorizonCube Spacing
HorizonCube Data	Horizon
HorizonCube Density	Match Delta
HorizonCube Layer	Maths (recursive expression)

Hilbert	Perpendicular Dip Extractor
Instantaneous**	SampleValue
Local Fluid Contact Finder	Scaling (scaling type AGC and stats type = detrend)
Maths (except when expression is recursive)	Shift
Polar Dip	
Position	
PreStack	
Reference	
Scaling (all except scaling type AGC and stats type = detrend)	
Semblance	
Similarity	
Spectral Decomposition*	
SpectrogramDip (create steering cube, FFT steering)	
Texture	
Tutorial	
Volume Statistics	

*multi-threading implemented in March 2014
**multi-threading improved in March 2014

Toolbars

Please refer to the relevant sections below for details:

OpendTect Toolbar
Manage Toolbar
Graphics Toolbar
Take Snapshots
Directional Lighting
Tracking Toolbar
Slice Position Tool

OpendTect Toolbar

The OpendTect toolbar contains icons to launch OpendTect specific modules:

starts the Survey definition module

starts the Attribute module 3D

starts the Attribute module 2D

starts the 3D output module

starts the 2D output module

starts the Volume builder

starts Attribute vs. Well crossplot

starts the Attribute vs. Attribute crossplot

leads you to the RockPhysics library

starts the XField Plugin*

starts the PDF 3D Plugin*

starts the Fluid Contact Finder (CCB) Plugin*

starts the Generic Mapping Tools Plugin

starts the HorizonCube Plugin*

starts the Madagascar link

starts the 3D Neural Network Plugin*

starts the 2D Neural Network Plugin*

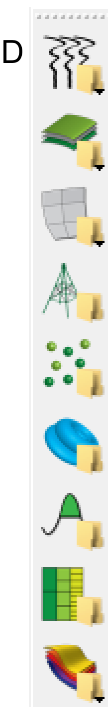
starts the SynthRock Plugin*

starts the Well Correlation Panel Plugin*

* These are commercial plugins, that are available under license. See: http://opendtect.org/index.php/support/licenses

Manage Toolbar

The Manage Toolbar allows fast access to the various data management windows via the following icons:



Opens the Seismics Manager (options for 3D, 3D Prestack, 2D and 2D Prestack)

Opens the Horizons Manager (options for 3D and 2D)

Opens the Faults & FaultStickSets Manager

Opens the Well Manager

Opens the Pickset Manager

Opens the Bodies Manager

Opens the Wavelet Manager

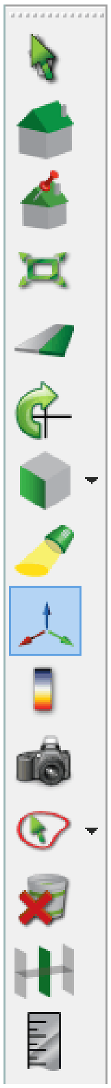
Opens the Stratigraphy Manager

Opens the HorizonCube Manager* (options for 3D and 2D)

* HorizonCube is a commercial plugin, available under license. For more details, see: http://opendtect.org/index.php/products/commercial-products/horizoncube

Graphics Toolbar

The graphics toolbar contains icons to manipulate the scenes (graphics window) using the icons listed below:



 When this icon is visible, you are in *interact mode* on. Click on the element that you wish to move or edit. If the element is an inline, crossline or Z-slice, a frame with handles (green squares) appears around the clicked element. The handles are used to re-size the frame. Clicking and dragging inside the element is the way to move the entire element in the in-plane direction. Tip: For accurate positioning of a data element, use the Position option from the right-hand mouse button pop-up menu in the tree.

 When this icon is visible, you are in *view mode*. In this mode, you can rotate the view and zoom in and out. Tip: in order to evaluate data values, zoom in and switch to interact mode by clicking on the appropriate icon. Move the cursor over the zoomed-in data and watch the returned position (inline, crossline and X,Y,T plus value) in the status line at the bottom of the main window.

 Resets the view to the position that was saved when you pressed the "save home position" icon, one below.

 Saves the current view as the home position that can be recalled with the previous icon

 Allows to reset the view such that all data are visible.

 Toggles the view between orthographic & perspective view.

Clicking this icon then clicking at a particular point in the scene will centre the scene on that point. (Ctrl+S serves the same purpose)

Use this icon to set the scene orientation to a particular 'standard' direction. Options include 'View Inline', 'View Crossline', 'View Z', 'View North' and 'View North Z'

Will open the directional lightening dialog box

The display rotation axis is used to show/hide the N-E-Z (North-East-Z) orientation arrows. This button can only be activated in View mode, but the arrows remain active in both view mode and interact mode.

Toggles the colorbar on and off.

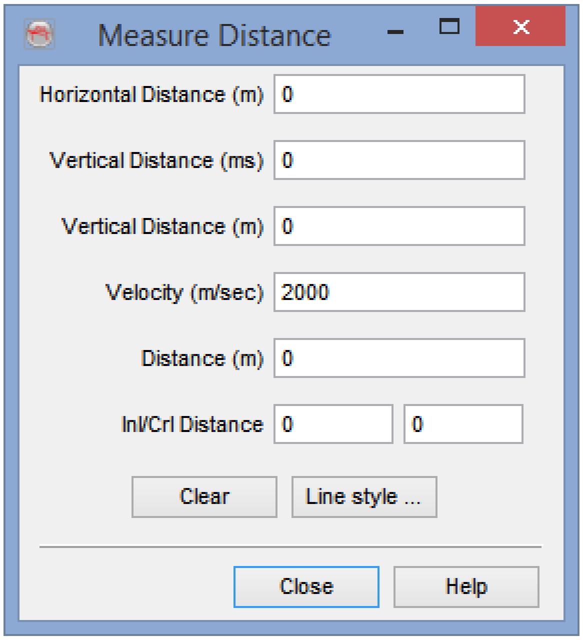
Opens the snapshot window, so that the user can grab pictures of the scene, window, and/or the desktop.

Sets the selection to polygon or rectangular modes and allows the user to select an area (or elements within an area, i.e faults sticks..)

For removal/deletion of selected elements.

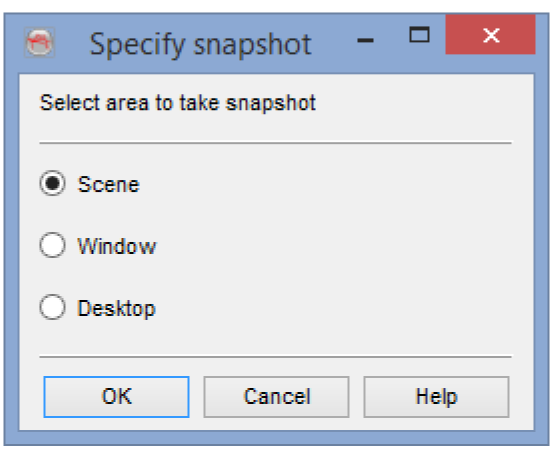
Displays selected element only. When more than one element is displayed in the tree, one can quickly view a single element and toggle between elements by clicking the different elements in the tree.

Tool to measure distances by drawing a polygon on an inline/crossline or on a Z-slice. The tool returns horizontal distance (meters), vertical distance (ms in time survey or meter/feet in depth survey) and the inline or crossline distance respectively. The distance is computed by using input constant velocity.



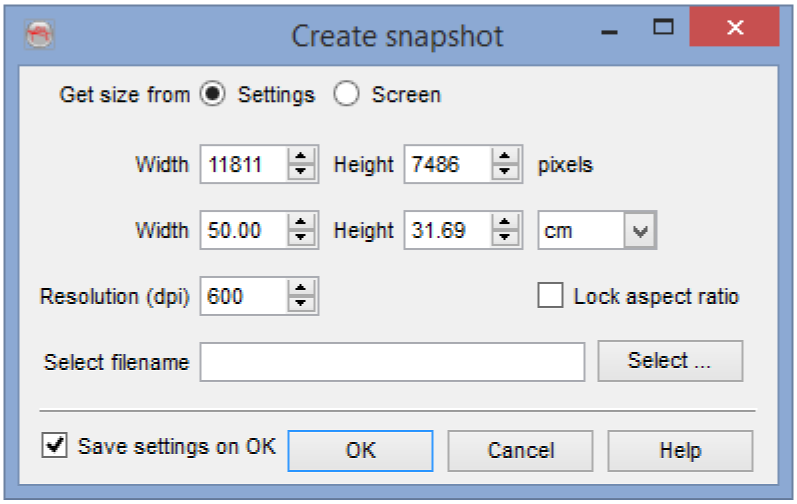
Take Snapshots

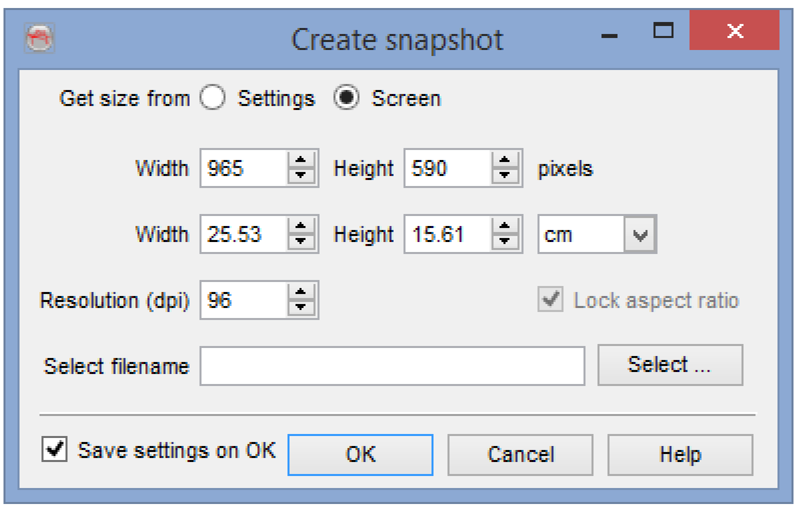
It is possible to take different kinds of snapshots in OpendTect. Three options are available: *Scene*, *Window*, and *Desktop*.



The *Scene* option allows the user to grab the displayed (selected) scene within OpendTect.The OSG (OpenSceneGraph) 3D library is used, allowing the output picture to have a better resolution than the screen resolution. Every element displayed in the scene will be in the output picture, including the annotations (color bar, orientation etc) if displayed. Any overlapping windows will be neglected.

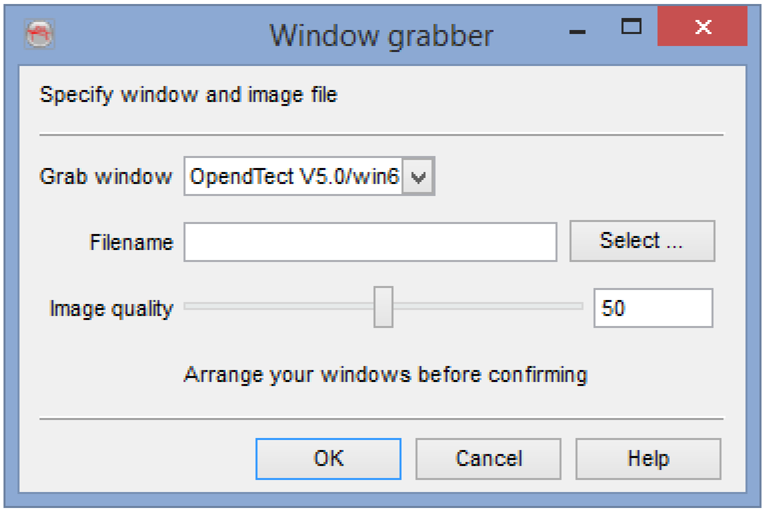
You can change the image properties e.g. height, width, resolution etc. If you save the settings, they will appear by default in all sessions next time you grab new snapshots. The '*Screen*' parameters corresponds to the parameters of the picture as displayed on your screen.

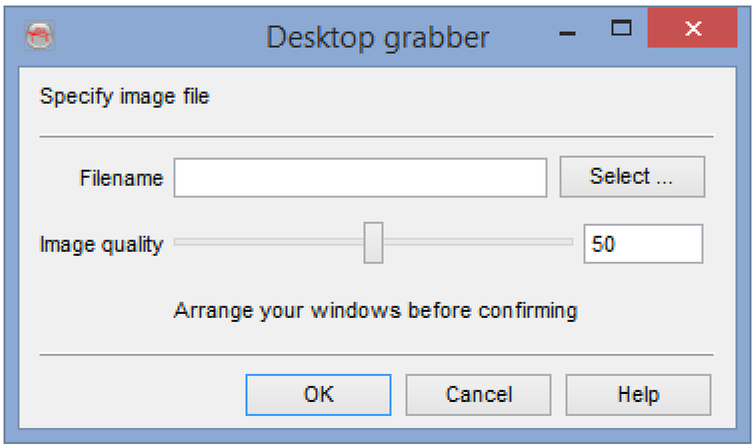




The *Window* option grabs the whole window, including the sidebars. It can be either the main window or any 2D viewer opened when the snapshot button was pressed.

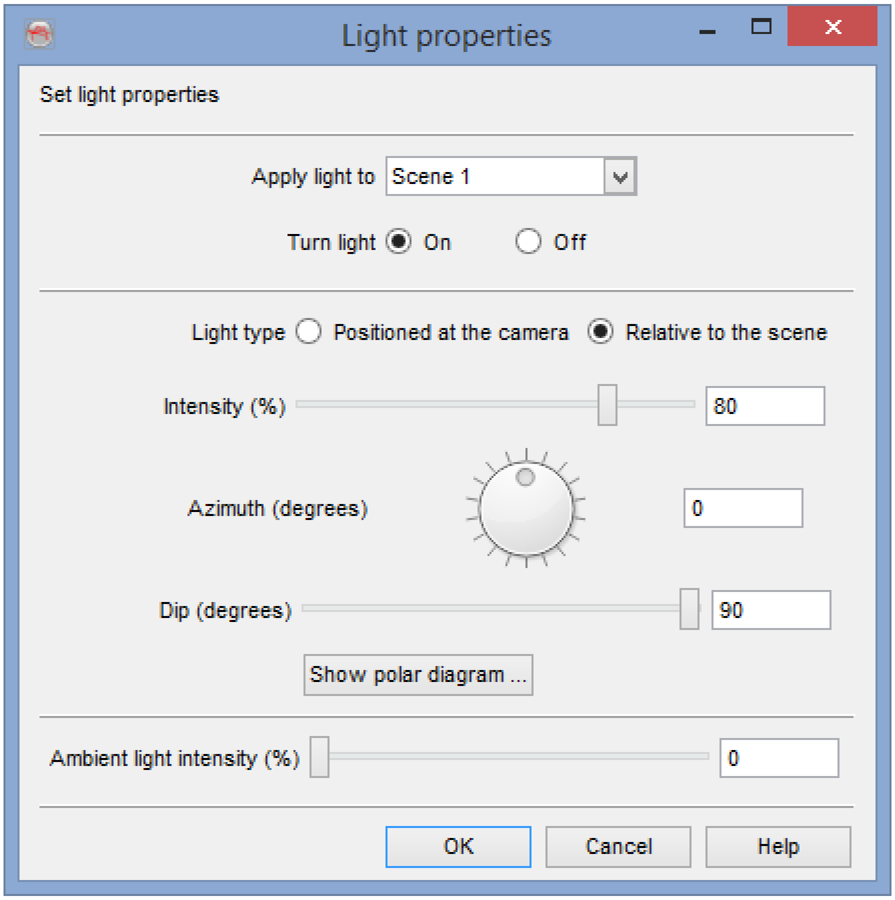
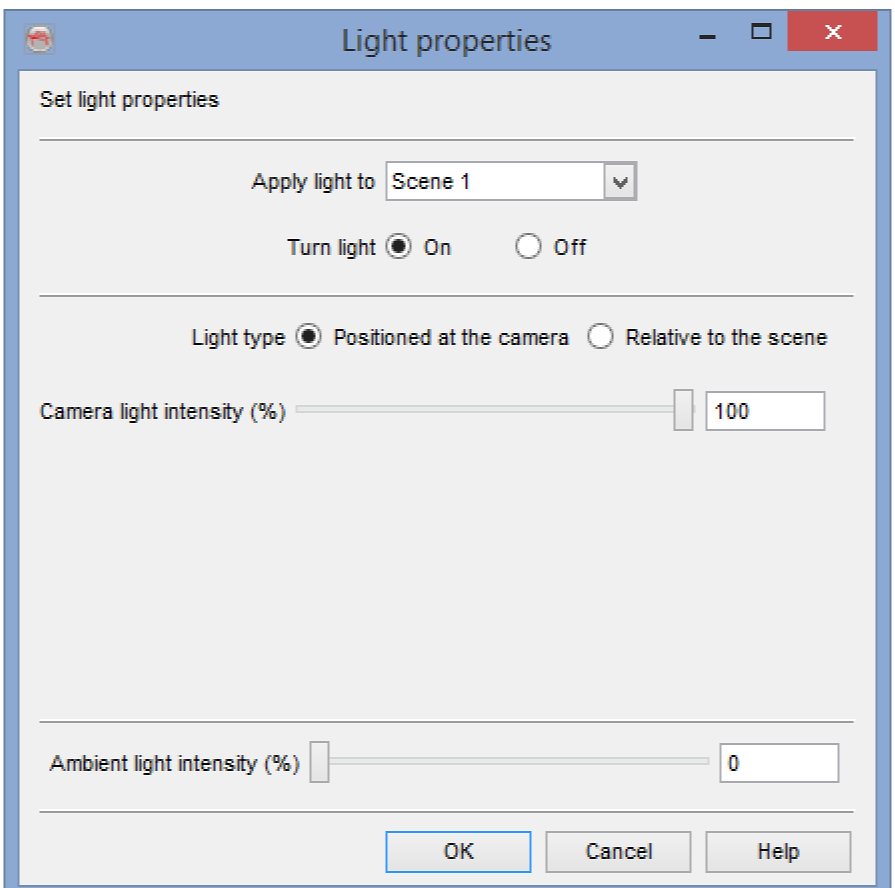
The *Desktop* option is similar and will snap the entire user desktop. Both options use the Qt library for grabbing the picture. As a result the output is limited to the actual screen resolution, and overlapping windows will appear on the snapshot.





Directional Lighting

The directional lighting feature is used to illuminate the objects (displayed data) at a specific inclination (or dip angle) and azimuth. The feature controls the main headlight i.e. the intensity of the camera light and the intensity of the directional light. The dialog is launched by clicking the icon shown above.


Light properties
Set light properties
Apply light to
Scene 1
Turn light
On
Off
Light type
Positioned at the camera
Relative to the scene
Intensity (%)
80
Azimuth (degrees)
0
Dip (degrees)
90
Show polar diagram ...
Ambient light intensity (%)
0
OK
Cancel
Help

Light properties
Set light properties
Apply light to
Scene 1
Turn light
On
Off
Light type
Positioned at the camera
Relative to the scene
Camera light intensity (%)
100
Ambient light intensity (%)
0
OK
Cancel
Help

Directional light Dialog

The directional light dialog updates the scene instantly to reflect the changes made to the properties. If the OK button is clicked, the changes are retained, whereas, the Cancel button rules out all changes.

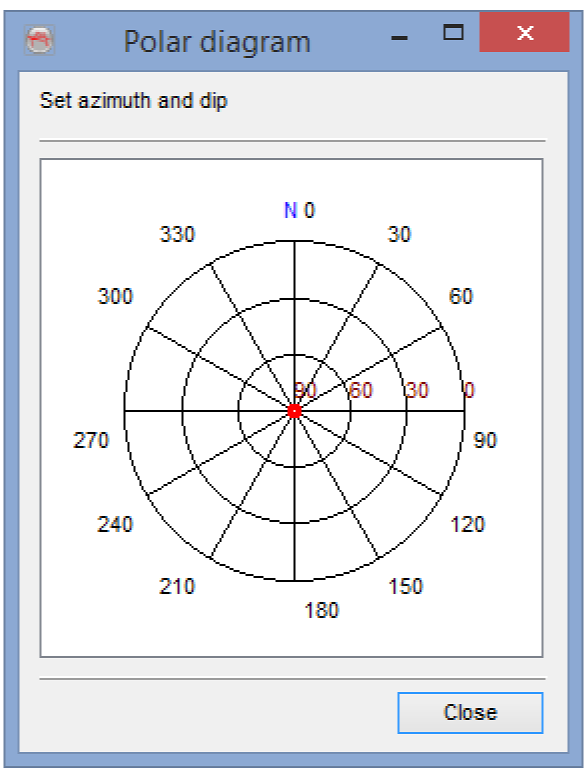
Apply light to: The directional lighting is independent for each scene, i.e the selected scene will be illuminated. However, selecting the option All in the drop down list will illuminate all scenes that are currently open.

Azimuth: This slider is used to set the azimuth (in degrees) value of the directional light. It can be any value from 0 to 360 degrees.

Dip: This slider is used to set the dip value (in degrees) of the directional light. The directional dip is limited from 0 to 90 degrees.

Intensity: Sets the percentage of the intensity of the additional directional light. 0% corresponds to total darkness while 100% corresponds to full intensity.

Show polar diagram: The azimuth and dip can be visualized using this diagram. This diagram can be used in combination with the sliders of the main dialog in order to position the directional light around the scene.



Polar diagram dialog

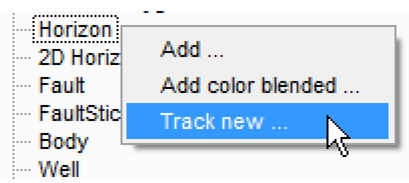
This is a dialog that displays the polar diagram for setting the azimuth and the dip values of the additional directional light. The location of the pointer (the red dot) determines the properties of the directional light. The pointer can be moved around by using the mouse within the polar diagram. The azimuth value can be read off the circumference of the outermost circle while the dip value is given by the location of the pointer along the radius of the circle.

Camera light intensity: Use this slider to change the percentage of intensity of the camera light or the head light. 0% corresponds to total darkness while 100% corresponds to full intensity.

> **Note:** Similar to the directional light dialog, the changes made to the azimuth and dip are instantly reflected in the selected scene(s).

Tracking Toolbar

This toolbar will remain hidden until you choose to track a new surface. Eg:



Once visible, the tracking toolbar will remain in position, just below the scene. Inactive icons are visible, but greyed-out.

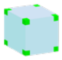
The Tracking toolbar contains icons to control tracking of surfaces:

 Opens [Tracking Settings]()

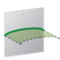
 Activates seed pick mode

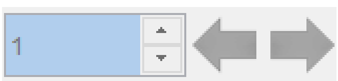
 Starts horizon tracking from seeds

 Removes all previously auto-tracked horizons, after which they are re-tracked from seeds only (within the tracking box).

 This icon toggles the tracking cube on/off. Only data within this volume is stored in memory. When the tracking cube is on, it can be re-sized using the green anchors.

 This icon displays the QC plane in the direction of the inline. The user can also QC the tracking on a crossline  or timeslice 

 The tracked horizon will be displayed at current section only



The tracking plane can be moved N positions back / N positions forward i.e. towards a lower/higher inline, crossline or time (depth) number. N is the specified step size (in this case, 1).

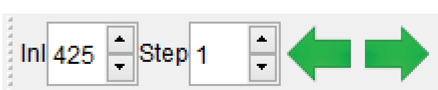
 Polygon selection tool

 Remove / delete the selected elements (within the selection area).

 Undo /redo

 Save

Slice Position Tool



The slice position toolbar is used to move the 3D slices (inline, crossline or timeslice) according to the user-defined steps. The slice position and number of steps are manually entered in the fields. The forward and backward arrows of this toolbar are used to move the selected slice in increasing or decreasing directions. The workflow is very simple. Add an inline in a scene. By default, a blank attribute (inline, crossline or timeslice) is added in the middle of the survey box. Right click on the attribute and add display the seismic data. Select the inline in the tree. Note that the toolbar items will be updated and the arrows will be activated. Update the inline position and the steps in the toolbar. Use the arrows to move the inline position according to the given steps.

TIP: While moving the planes, the progress bar appears by default every time when a user moves the plane to display the data. This can be avoided by changing the Personal Settings. Additionally, if the computer memory is sufficient, the seismic volume can be preloaded for displaying it quickly. Use Preload settings to keep a seismic volume in the memory.

TIP: Keyboard shortcuts can be used to ease the sliding: *Utilities* > *Settings* > *Keyboard shortcuts*. The step used will be the one specified in the slice position tool.

Mouse Controls - Scenes and Graphical Interaction

The workspace contains one or more graphics windows. There are two basic manipulation modes: view mode and interact mode. The latter has two sub-modes: moving and picking. One can toggle between the view and interact mode with the *Esc* key or by directly clicking on the icon.

In view mode (active when the "hand" icon is displayed), the cursor is displayed with two circular arrows ( ). In this mode, each scene can be rotated and zoomed independently using mouse button operations. Left-click and drag is for rotating the image. Simultaneous left+middle button click and drag is for moving the camera closer or further away. Shifting the image is done by middle mouse button click and drag movements.

In interact mode (active when the "arrow" icon is displayed), the cursor is an arrow. In interact mode, you can move the elements in 3D space (move mode) or you can pick example position to train neural networks (pick mode).

In pick mode, the current pickset is highlighted in the tree and the status bar displays a message; for example: "Nr of picks: 25". A new position is added to the current pickset by clicking with the left-hand mouse button. The status bar message is updated: "Nr of picks: 26". Picks from the current pickset can be removed by holding the Ctrl-key (the Apple key on MacOS) and pressing the left-hand mouse button.

Move mode is used to move elements to other positions in the data cube (ie: to move/reposition an inline, crossline, Z-slice or volume). An element is selected by clicking with the left mouse button in the tree, or on the element in the graphics window (only if you are not in pick mode). Arrows and a frame with handles appear around the active element. Moving the element to another parallel plane is done by left-clicking and dragging on one of the arrows. The new position is also indicated in the status bar at the bottom of the OpendTect main window. The element can be re-sized by left-click and drag on one of the green handles. Left-click and drag inside the element moves the element in the in-plane direction. Data will be loaded into the new position after left-clicking outside the element frame. To reset to the initial position, right-click and chose reset manipulation. Left-click outside the element validates the move.

Direct accurate positioning of data elements is possible using the Position option from the pop-up menu in the tree. In interact mode, as well as move the cursor position (inline, crossline, Z-slice), the data value at the cursor position are returned. These are displayed in a status line at the bottom of the OpendTect main window. Scenes can be rotated and zoomed in and out simultaneously using the wheel controls and the zoom slide-bar in the OpendTect main window. These controls operate irrespective of the view mode (view, interact, pick).

With Mouse

View Mode		**Interact Mode**			
		Move Mode		Pick Mode	
's' + left-click	zoom to clicked position	left-click	activate element	left-click	pick position
left-click + drag	rotate	left click outside active element, or left click in empty space of tree	de-activate element	CTRL-left click	delete pick
middle-click	shift	left click+drag arrow	move element to parallel plane		
left- and middle-click + drag	move camera closer/away	left click+drag inside active element	move element in-plane direction		
left-click + drag + quick release	spin	left click+drag handles	resize		
		move cursor	return position and	move cursor	return position and

			value		value

Using Touchpad

View Mode	
Hold down left-click and move	Rotate
Hold down Ctrl+left-click and move	Pan
Hold down Ctrl+shift+left-click and move	Zoom

Note: De-activate elements is also possible by clicking (left-hand) in an empty space of the tree.

Notes for MAC users: The Apple key usually has the same function as the CTRL key on a PC. However, it is recommended to use a three-button mouse. All manipulations mentioned in this table (and elsewhere in the documentation) are also possible with a single button apple mouse, in combination with SHIFT and/or the Apple key and/or the CTRL key, but it is easier to work with a three-button mouse.

Color Tables

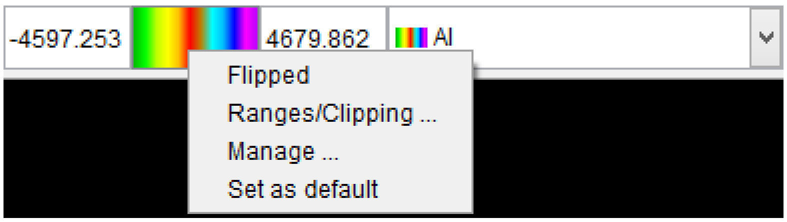
A colortable is a predefined group of color settings that can readily be applied to any attribute. This group includes items such as the primary colorbar, undefined color settings, color segmentation, and opacity. Changes made to the colortable are applied universally to any item that uses that colorbar.

A colorbar is the specific colortable for a particular element's (line/horizon) attribute that is displayed in a scene. Updating the colorbar can update the selected element's attribute. The image below is an example colorbar:

The colorbar is composed of four elements: the color display itself, the minimum and the maximum value of the variation for the colortable (as it is defined for the currently selected item), and a set of colortables.

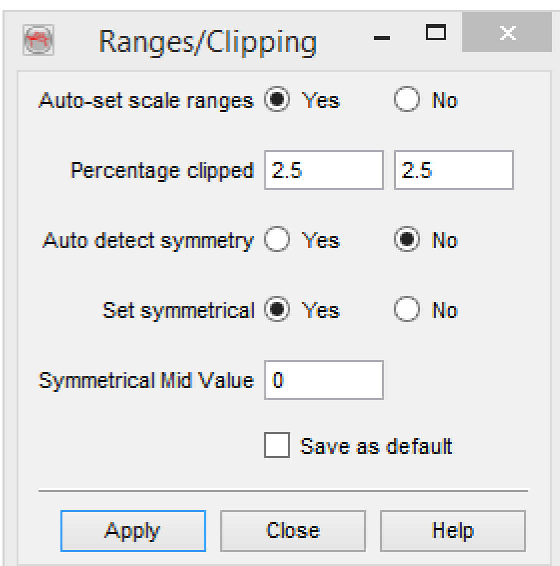
This drop down list of predefined colortables appears when the user clicks on the name of the colortable being used in the colorbar (ex: Channels). If no item is selected in the tree, the colorbar will not show any value/range although it can be manipulated. The colortable is manipulated by right-clicking on the colorbar.

The pop-up sub-menu contains several manipulation functionality. These are described in the following:



Flip causes the scale to be flipped. (The color assigned to the high value, now becomes the color assigned to the low value, etc.)

Ranges/Clipping allows the user to change the range of the color scale, or clip a certain percent of the scale. Please be aware that because of display time consuming, only 2000 random sampling points are by default used to clip data. The clip values thus change from one data set to another.(An alternate method for clipping is described in the Inline, Crossline & Z-slice sub-chapter).

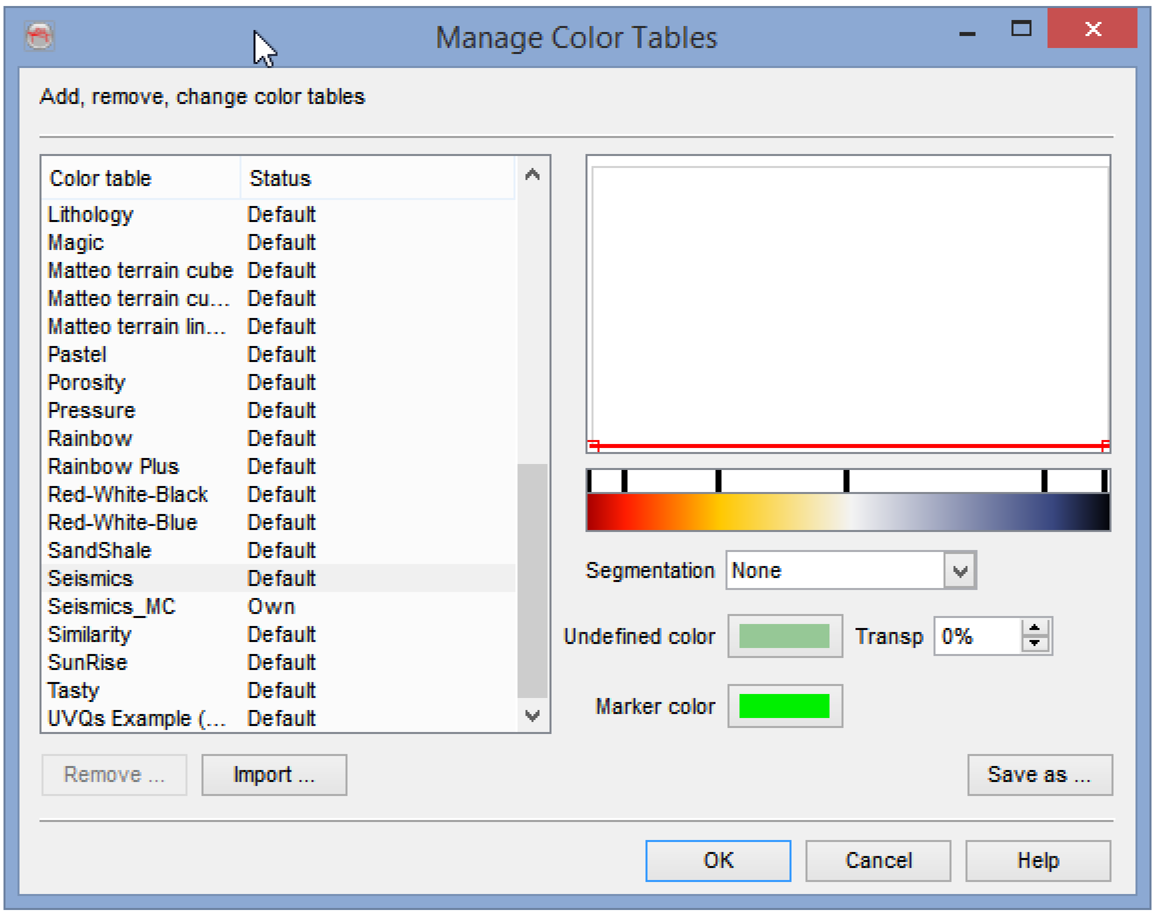


Set as default sets the current color settings as the default color scheme for all elements in the tree.

Manage is used to modify the current colortable and to create new colortables with the current one as a starting point. Colortables are modified by adding, removing, changing colors, varying opacity, and defining the colorbar to be gradational or segmentized. The effect of the changes on your displayed element can be seen directly. Colortables can be removed from the list by pressing the Remove button. (OpendTect Default colortables can not be removed). Moreover, the user can import user-defined colortables by pressing the *Import* button.

Colortable Manager window

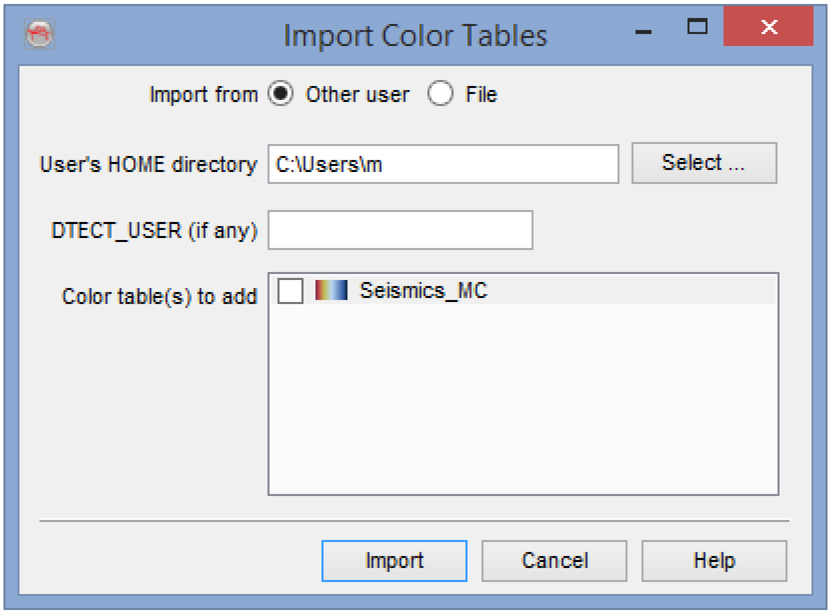
The *Colortable Manager* window opens when the user selects the *Manage* option described above.



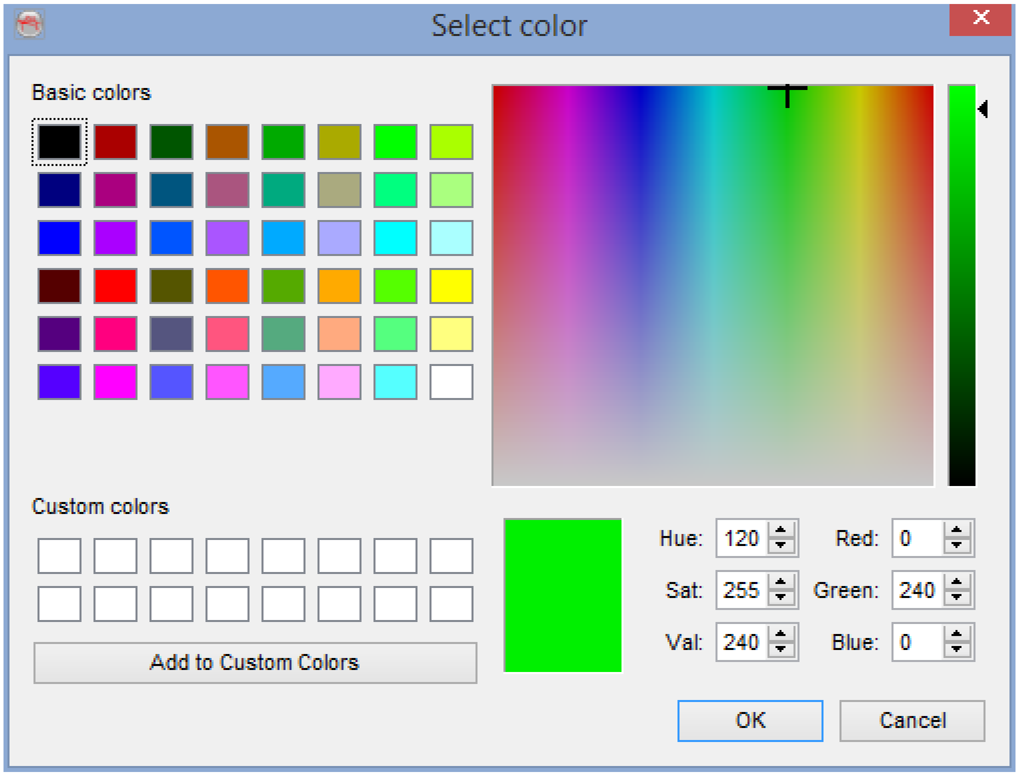
A marker is the color you see in the colorbar. The black lines, in the white field above the colorbar, are the marker boundaries. The marker boundaries are where the settings for the markers are defined. Right-clicking on a marker boundary shows the following options: *Remove color, Change color,* and *Edit Markers*.

Remove color deletes the marker boundary from the colortable.

Import colortable file: The colortables can also be imported by pressing the From other user... button. The default colortables are stored in a file (ColoTabs) that is located in the OpendTect installation directory (/root/opendtect-4.6/data/). Moreover, the colortables saved by a user are stored in a file (settings_coltabs.user) that is located in the user home directory ($HOME/.od/), here user is the OpendTect username. These files can be modified or imported by using import color table window (see below).



Marker color brings up a standard color definition window, where this defined color can be changed.



Edit Markers opens the *Manage Marker* window that displays all markers: marker ID, position, and current color. The marker's position, in relation to low and high values, can be specified by number. The standard color definition window can be opened from here too, by double-clicking the marker color.

Opacity: A thin red line, capped on each end by small red nodes, is visible at the bottom of the histogram located in the top panel. By moving these nodes, or adding additional nodes, the user can vary the opacity of the colors below. One can add opacity nodes by double-clicking in this area. These opacity nodes can be dragged up or down to increase or decrease, respectively, the transparency of the color directly below it in the colorbar. A hatched area (visible in the color toolbar in the main window of OpendTect) indicates the part of the color bar that will display with some level of transparency. The darker the color of the hatch marks, the higher the level of transparency.

> **Note:** Transparency performance depends on the graphics card. When displaying two elements in exactly the same position, transparency may not work as you expect. It may help to set transparency values to the maximum to get the sort of display you desire. In addition, it may help to change the transparency of the element as a whole by right-clicking the element in the tree, and selecting Properties.

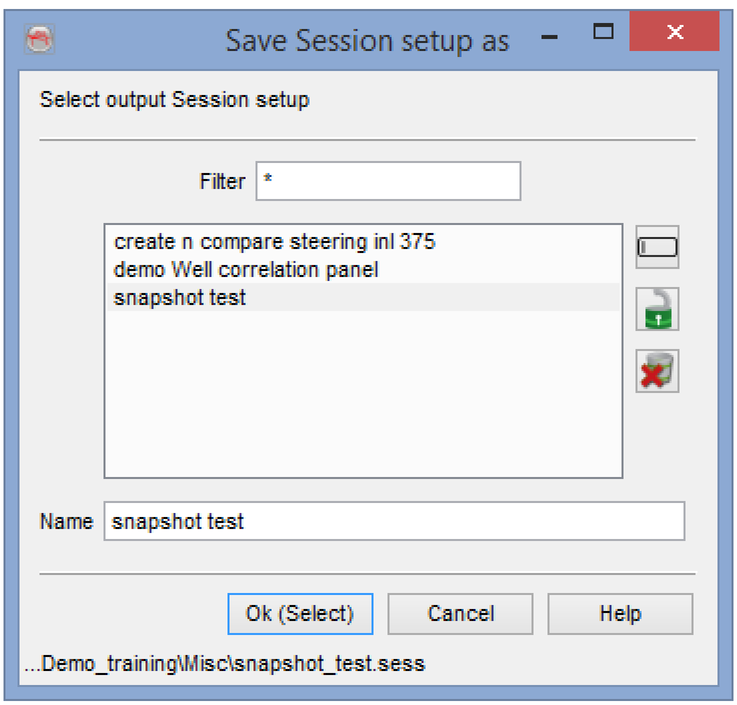
In the background of the opacity panel, a histogram is shown in light grey. This histogram shows the distribution of attribute values in the selected element. This helps you to tune the colorbar to the value range you may want to highlight. To alter the histogram see *Show Histogram* in the Inline, Crossline & Zslice sub-chapter.

Segmentation allows the user to segment the colorbar into a user-defined discrete number of colors. This can be done in a *Fixed* or *Variable* manner. *Fixed* allows the user to define the number of segments they would like to have, but does not allow the marker boundaries to be moved. *Variable* allows the user to both define the number of segments, and move the marker boundaries to suit specific needs. Fixed is good for purposes such as velocity and contour lines, while *Variable* is good for use with waveform segmentation.

Undefined color specifies the color that will be used to display undefined values in the data.

General Selection Window

The general selection window is used for various purposes:



- Selection of an attribute set (*2D/3D Attributeset* > *Open attribute set...*)
- Restoring/Saving a session. (*Survey* > *Session* > *Save/Restore...*)

The window contains following standard buttons:

 renames the object.

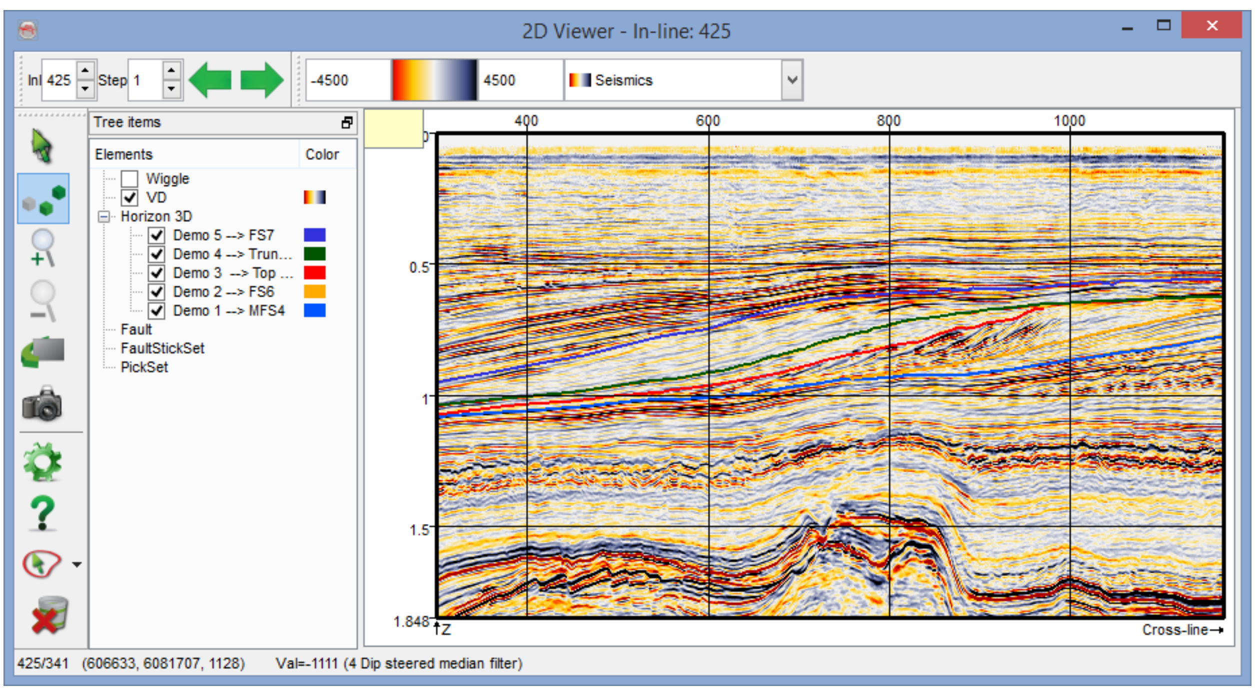
 toggles the object to *Read only*. Use this option to protect any object from overwriting.

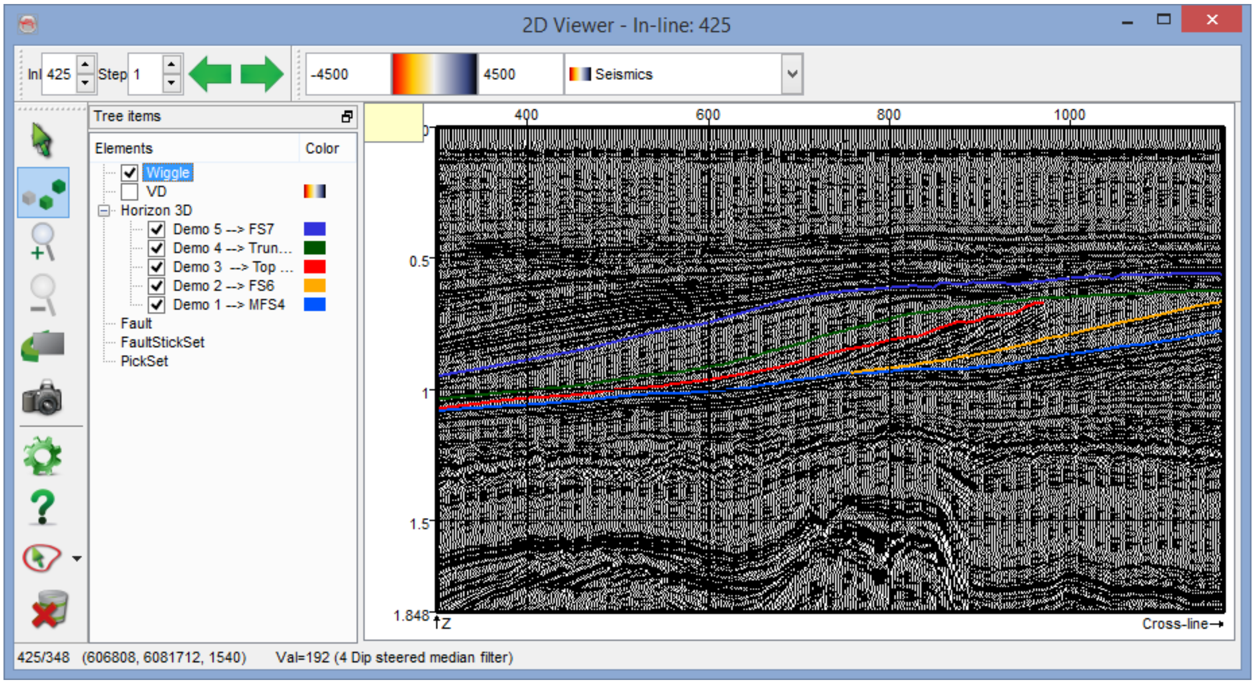
 removes the object.

When the user clicks Survey > Session, there are three options: *Save*.., *Restore*.. and *Auto*.., which enables the user to select a session which will restore, by default, each time the user opens OpendTect.

2D Viewer

The 2D Viewer is used for examining the seismic data of an inline, crossline, or a 2D lines in a 2D-screen. To open the main 2D viewer, display an attribute on e.g an inline. Right-click on that attribute, display in a 2D viewer as .. *Wiggles* or *Variable Density* (VD).





The 2D Viewer shows several options:

 Enables the user to select a zone of interest and to zoom in on this region: In this mode the mouse wheel will zoom in and out, and the left click will drag the data on the sides. The view is refreshed only once the left click button is released.

 This option can be used to zoom in a specific area within the viewer.

This creates seeds on the 2D viewer, this is especially possible when tracking a horizon. To track an horizon on 2D viewer, make sure that the picking mode is active in both the main scene and the 2D viewer.

Zooms in when clicking

Zooms out when clicking

Flip the 2D-section

enables the user to grab pictures

Set display parameters

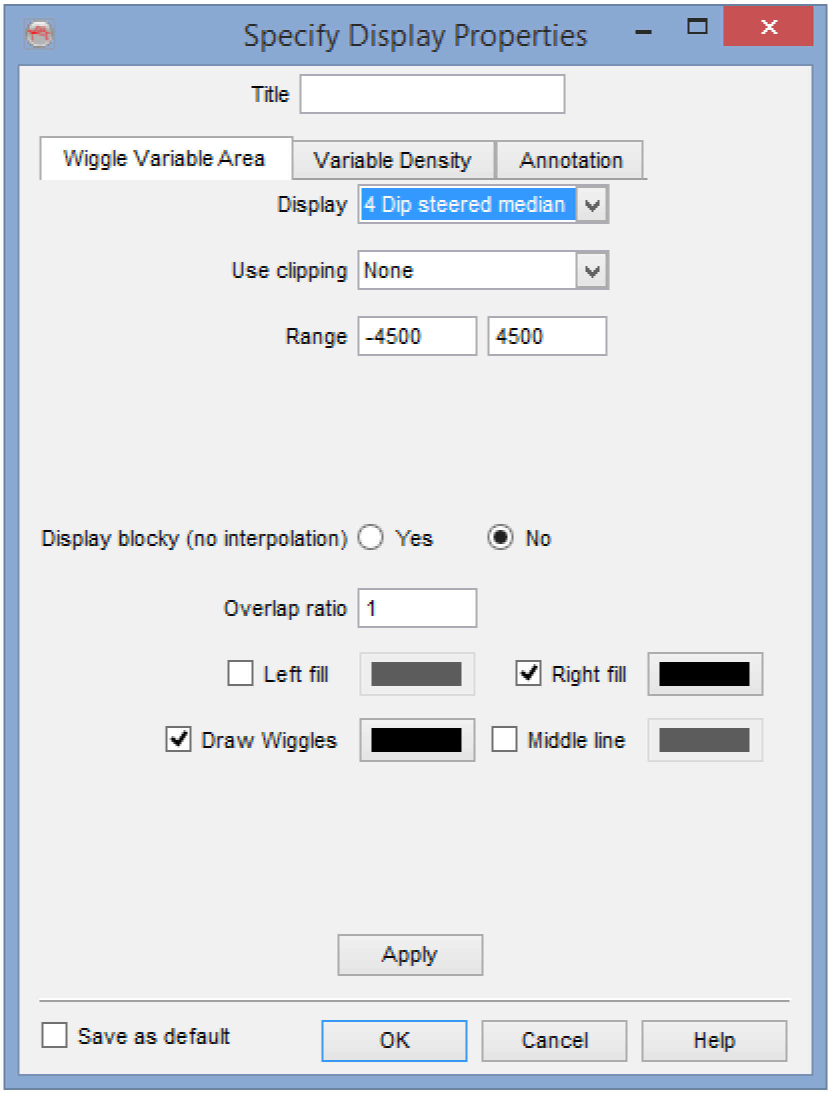
 2D-Viewer's Help

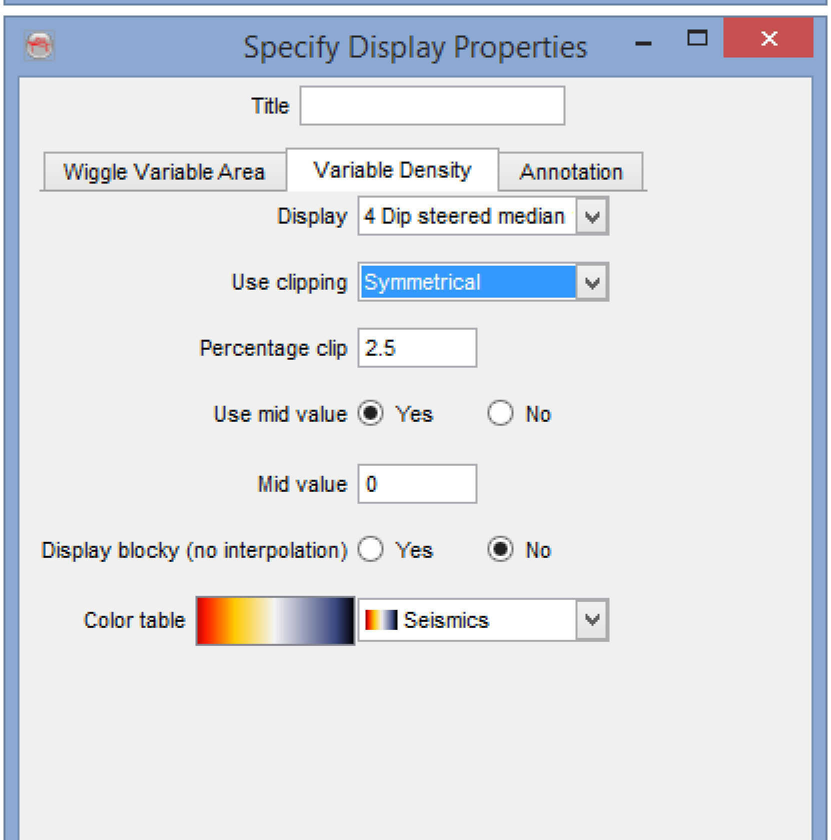
Manage the polygon selection

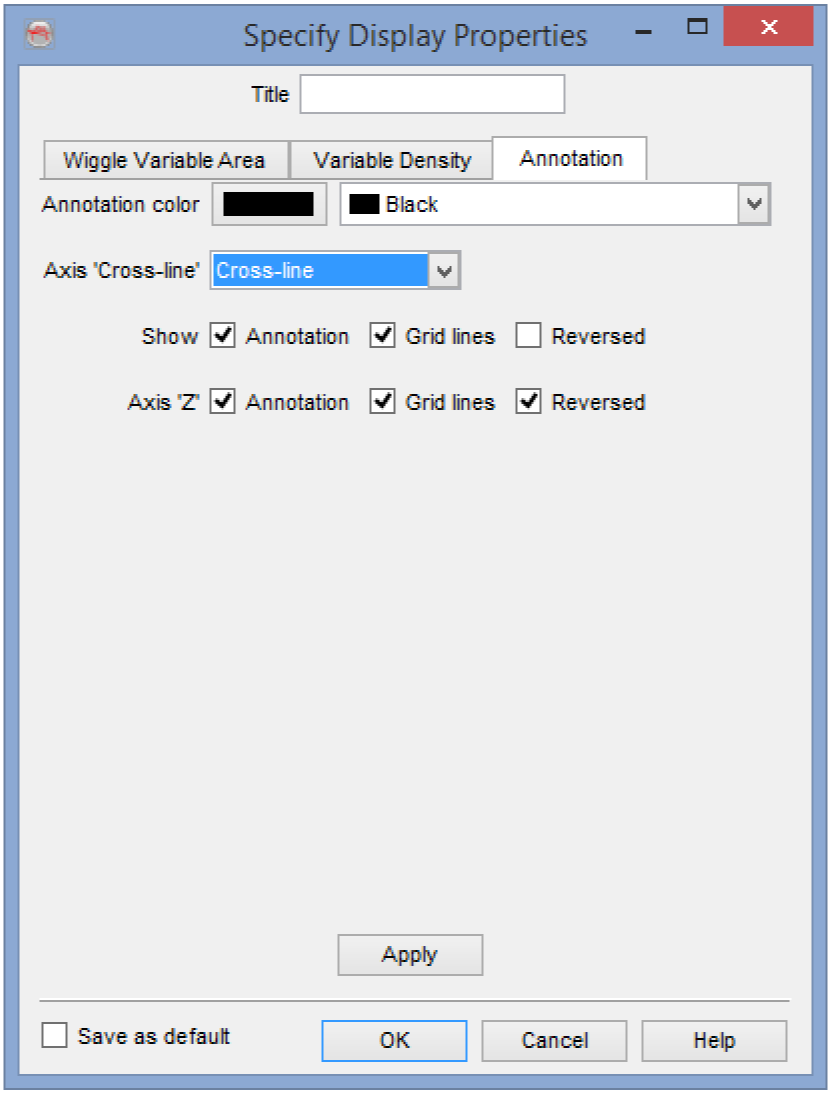
Remove the selected data within the polygon

Clicking this icon will start the HorizonCube slider

When clicking the right mouse-button on the 2D Viewer, and clicking *Properties*, the *Specify Display Properties* Window appears in which the display parameters can be set for both the *Wiggle Variable area* and the *Variable Density*.

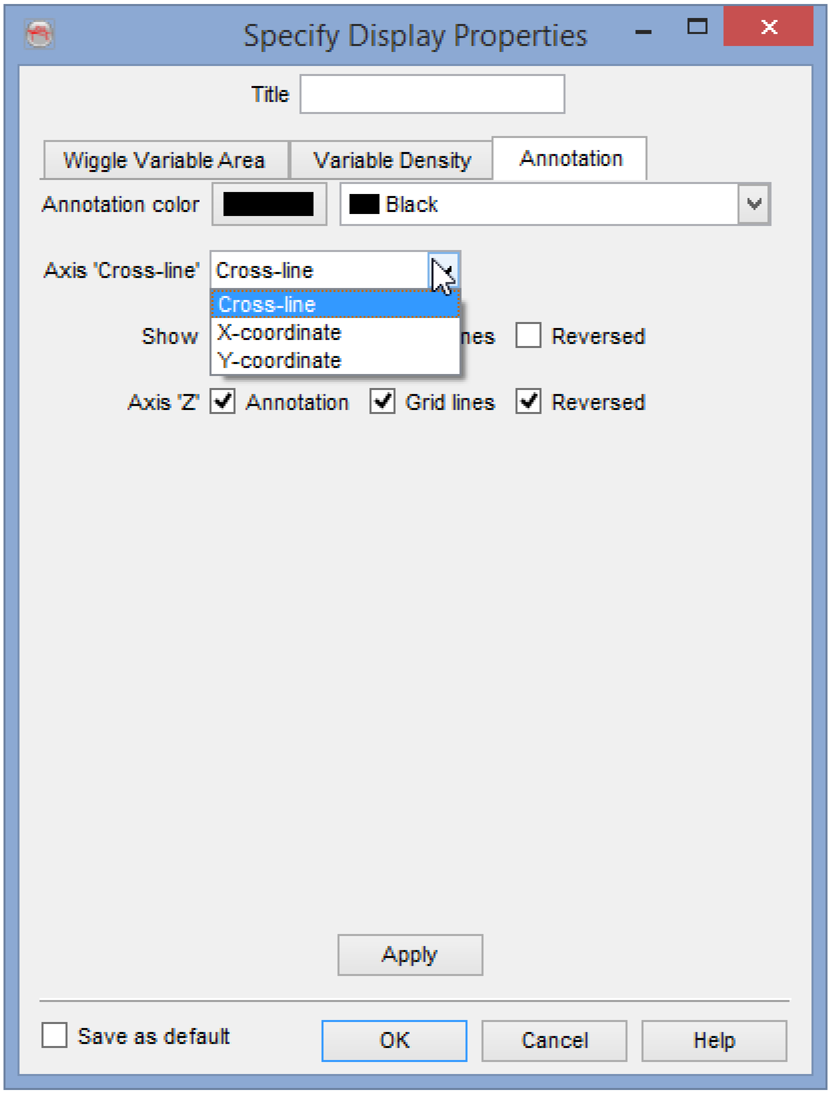

Specify Display Properties
Title
Wiggle Variable Area
Variable Density
Annotation
Display
4 Dip steered median
Use clipping
None
Range
-4500
4500
Display blocky (no interpolation)
Yes
No
Overlap ratio
1
Left fill
Right fill
Draw Wiggles
Middle line
Apply
Save as default
OK
Cancel
Help


Specify Display Properties
Title
Wiggle Variable Area
Variable Density
Annotation
Display
4 Dip steered median
Use clipping
Symmetrical
Percentage clip
2.5
Use mid value
Yes
No
Mid value
0
Display blocky (no interpolation)
Yes
No
Color table
Seismics



2D Viewer Wiggles Display Properties; 2D Viewer Variable Density (VD) Display Properties; 2D Viewer Annotation Settings

Please note that under the 'Annotation' tab, the user may choose between different distance values to be displayed on the Axis-distances:



- Trace number*: This will display the traces number on the X axis
- Reference position*: This displays the SP (shot points) number that is available in the shot point header field of the original SEG-Y
- X-Coordinate: This shows the X coordinate in the axis
- Y-Coordinate: This displays the Y coordinate in the axis

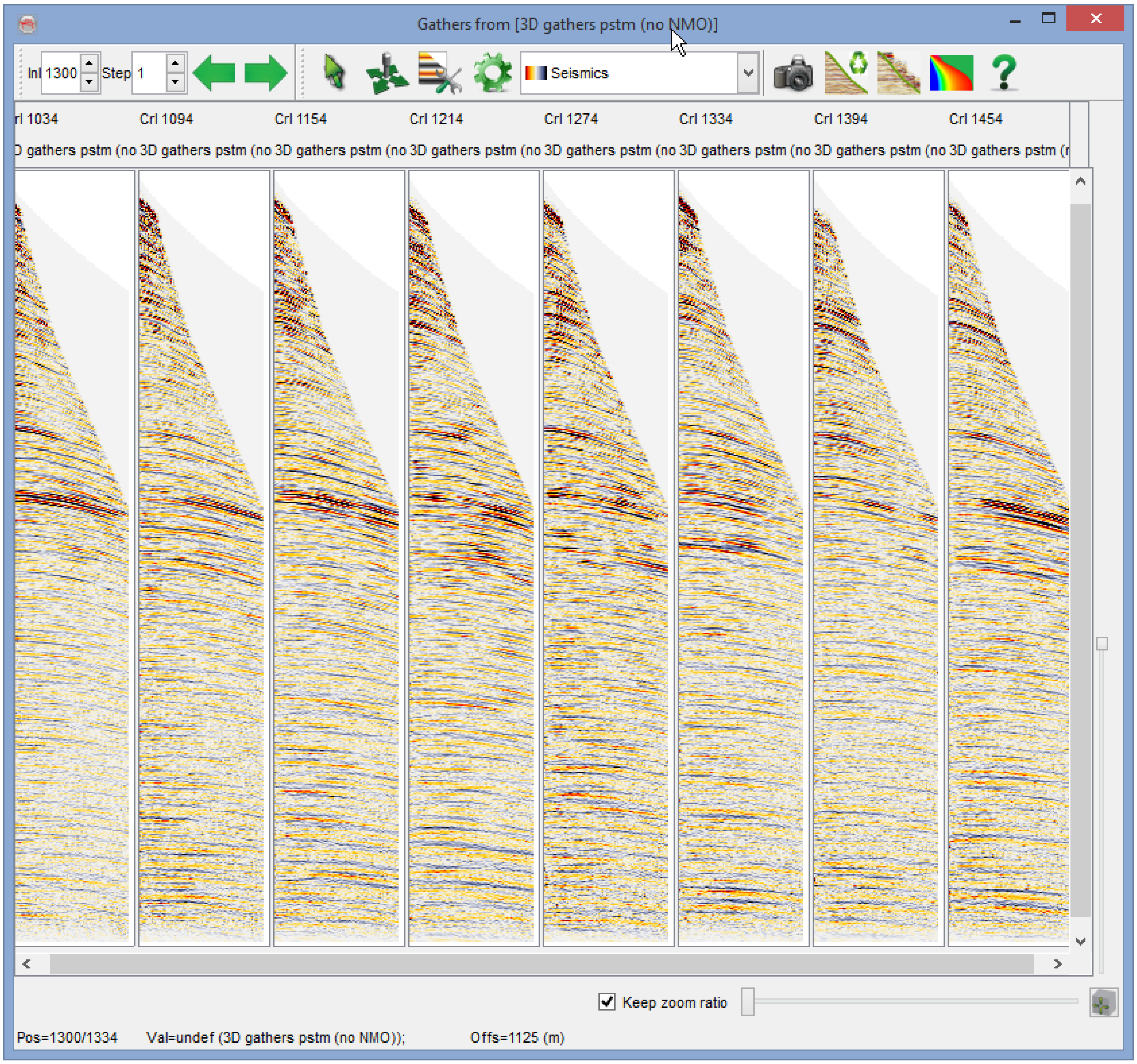
  * for 2D only

Pre-Stack 2D Viewer

In this window one or several pre-stack dataset can be viewed simultaneously. In this viewer you can:

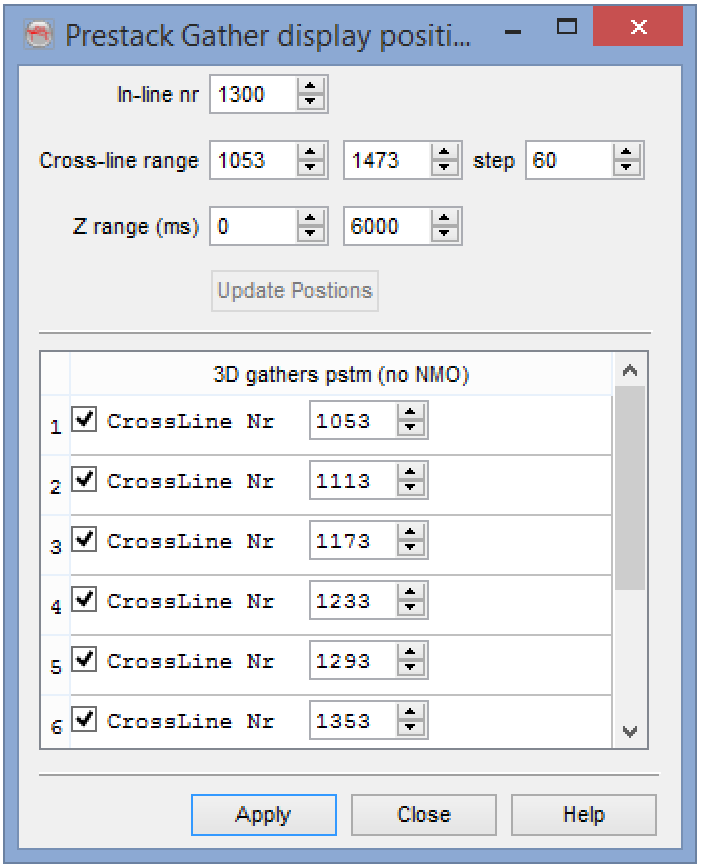
- Display gathers from one datastore at different locations.
- Display gathers from different datastores side by side.

- Use the two modes above in combination.
- Display mute definitions that were either imported or computed before starting the viewer.
- Apply a pre-processing on the gathers: AGC, application of the mute functions (stored or computed on the fly), ...
- Display an angle gather by providing the corresponding velocity model using a stored cube.



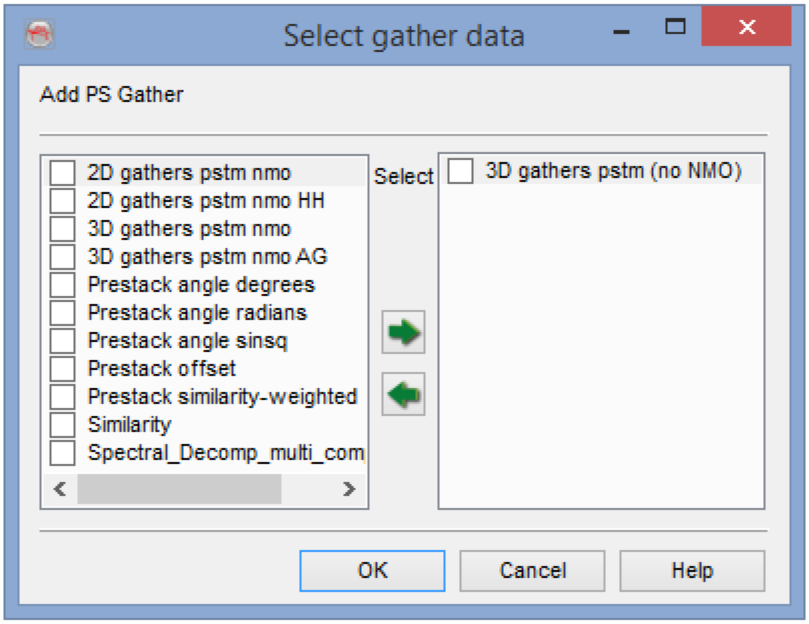
Gathers position selection

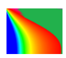
You can set the gathers to be displayed from this window. The top part is used to set a grid of regular positions from provided ranges. Keep in mind that you can get another 2D pre-stack viewer if you wish to have data from several inlines. From this regular grid, you set the position where gathers should be displayed. You can also manually change a crossline number. Press *Apply* to reload the view.



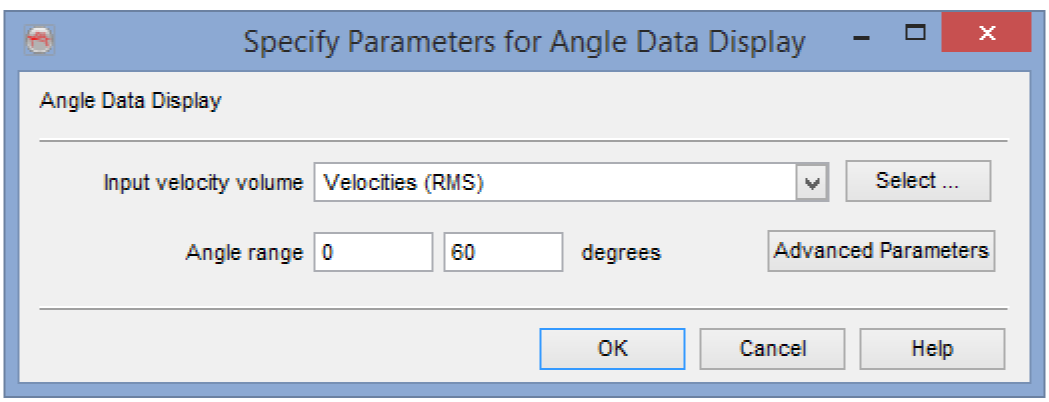
Selection of several datasets

Multiple gathers can also be added together in the 2D panels by pressing this  icon.

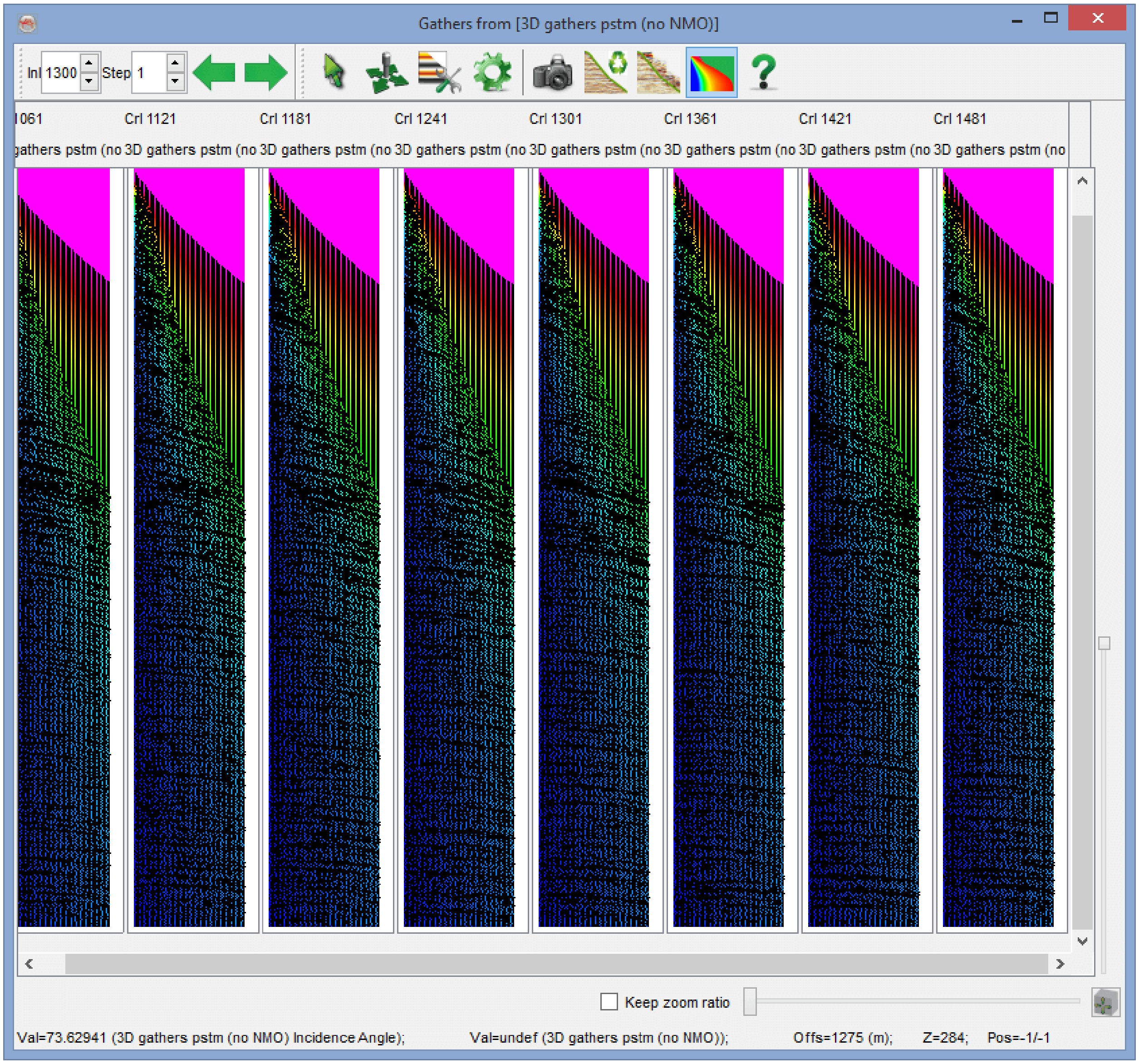


Display of angle gathers 

In this window you can set the parameters for creating the angle gather that corresponds to the prestack datastore. For each sample of the input seismic prestack gather, the incidence angle in degrees will be computed and color-coded with the rainbow colorbar. The seismic data will then switch automatically to wiggle display.



Example of pre-stack gathers (wiggle) with a mute functions, and the corresponding angle gather on the background:



Gathers from [3D gathers pstm (no NMO)]
Inl 1300
Step 1
l 061
Crl 1121
Crl 1181
Crl 1241
Crl 1301
Crl 1361
Crl 1421
Crl 1481
gathers pstm (no 3D gathers pstm (no 3D gathers pstm (no 3D gathers pstm (no 3D gathers pstm (no 3D gathers pstm (no 3D gathers pstm (no 3D gathers pstm (no
Keep zoom ratio
Val=73.62941 (3D gathers pstm (no NMO) Incidence Angle);
Val=undef (3D gathers pstm (no NMO));
Offs=1275 (m);
Z=284;
Pos=-1/-1

How To

Table of Contents

QuickStart a Project

OpendTect is a seismic attribute processing, visualization and interpretation system.

The primary input is:
A 3D seismic data volume or 2D seismic data set (most commonly as SEG-Y files)

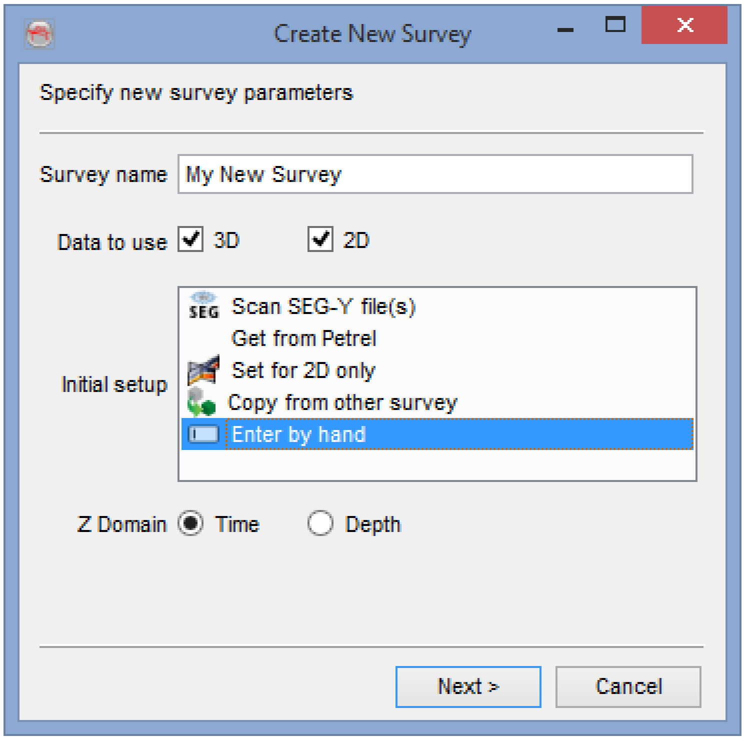
The primary outputs are:

- A 3D volume/2D data sets, such as a filtered cube/data sets or any other type of cube/data set produced by the system. Seismic data are exported via SEG-Y.
- Interpretations horizon/faults.

OpendTect Projects are associated with a *seismic survey*. The first time OpendTect is started, the user must define a new survey and import seismic data. At subsequent start-ups, OpendTect will start-up in the survey that was active when the program was closed. The user may select another, previously defined survey using *Survey* > *Select/Setup* and browsing to its location using *Set data root*.

Define a new survey. Select the Select/Setup option under the Survey menu or click on the survey setup icon . In the *Survey Setup and Selection* window, now press the

Create New Survey icon . The *Create New Survey* window appears, giving you several options on how to setup your new survey:



Choosing to *Enter by hand* brings you to the Edit Survey Parameters window where you can specify the name of the new survey. The ranges and the relationship between inline/crossline and X/Y coordinates can either be set manually, copied from another survey in the same data root, or obtained by scanning a segy file. After scanning, you can still adjust all parameters.

Optionally, a data access module can be purchased at dGB that gives access to Seisworks/OpenWorks (Landmark) or GeoFrame (GeoQuest IESX). The core of this module is developed by UK-based software providers ARK CLS. The license gives you access to the data store of your choice. It currently supports I/O of survey parameter, seismic data, horizons, and well data.

1. Import seismic data. Select Import-SEGY via *Survey > Import > Seismics > SEG-Y*and specify the input parameters. In case you do not know the parameters, use the *Examine* utility to dump the EBCIDIC header, the binary header and the first 100 traces header. From these it should be possible to determine the few input parameters needed by the program. The internal OpendTect format is 8,16 or 32-bit depending on whether the input Cube was

8, 16 or 32 bit.

2. Generating seismic attributes. Now, you are almost ready to go! You have created a survey, which contains at least one seismic volume. To do some real work, you must also create one or more attributes in the attribute engine that reveal relevant, useful seismic information. Several default attribute sets are available to get you started quickly. Open the attribute engine *Processing>Attributes* (or press the or icon) and press the button to open a default attribute set. You are asked to select the input data to complete the attribute definitions. Press "OK" to save the attribute set, this will bring you back to the main window.

3. Display your data or attributes. Now you are ready to display any of the defined attributes, anywhere in the volume. Display in the scene, for example, an inline element by right-clicking on *Inline* in the tree and pressing *Add*. The empty element is by default in the middle of the cube. You can move it in the scene using the arrows. Once the element has been added to the scene, right-click on the element in the scene and *Add > Attribute...* select in the data selection window that automatically pops up the data you wish to view on this element. You can either select from a stored volume or you select one of the defined attributes. If you choose an attribute, then this attribute will be calculated on-the-fly at all sample positions of the displayed element.

Note: The time taken to display an attribute on-the-fly can be significantly longer than the time taken to display a stored volume. That is why it is recommended to store your attribute(s) as stored volume or stored surface data. (See Create Seismic Output - 3D, Create Seismic Output - 2D and Create Horizon Output)

You can also find tutorial videos on the OpendTect's website, which will guide in steps to start a new project.

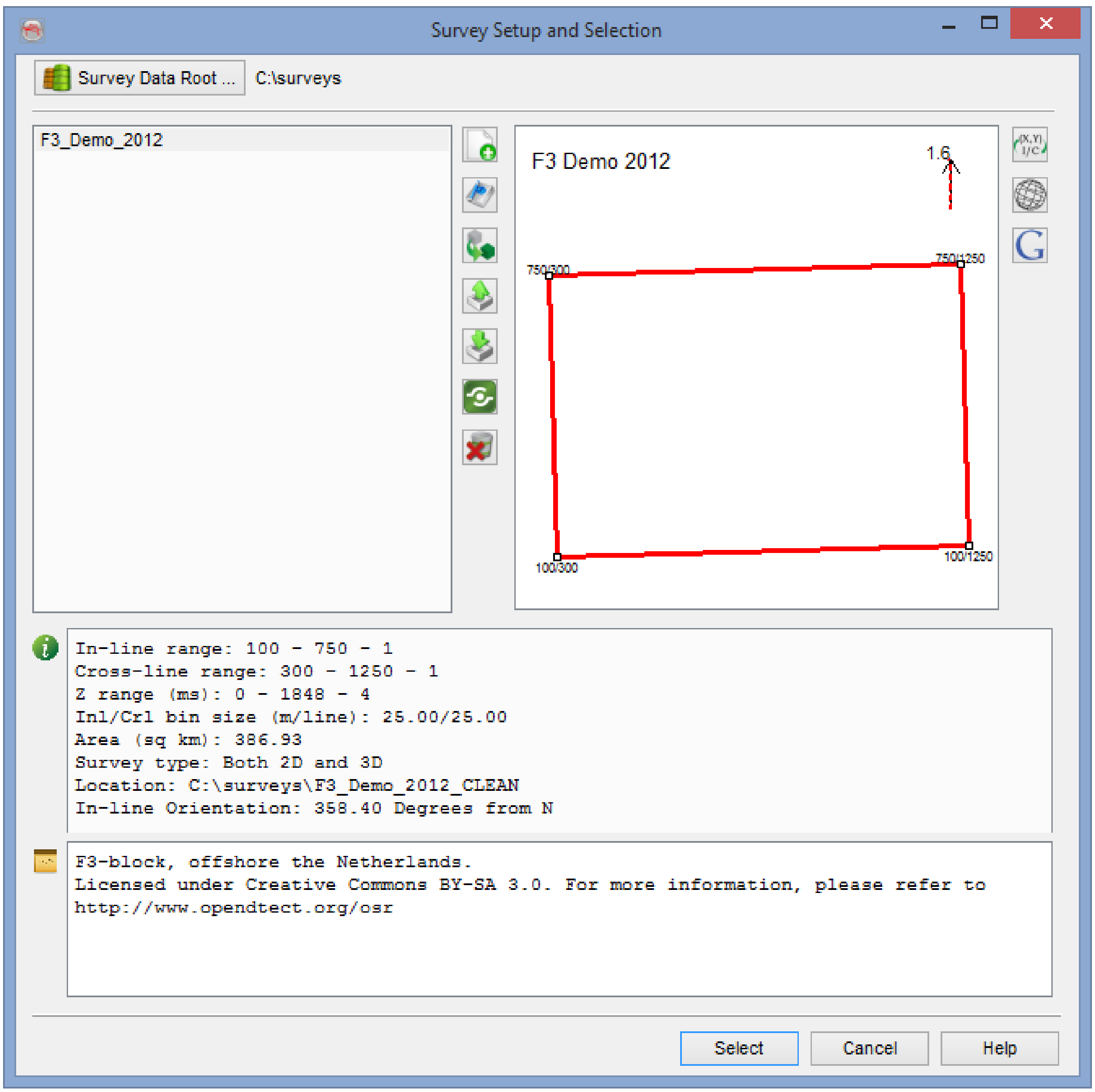
OpendTect Survey

The OpendTect Surveys are organized under *Survey selection* window which is used to change, create, edit or remove the survey(s). It is launched from *Survey > Select/Setup* or select survey icon .

The Survey selection window is divided into four panels with some key buttons. The left panel shows the already existing surveys in the present root directory. The root (location) directory of OpendTect survey can be changed by pressing the *Set Data Root*... icon . The right panel shows the map view of selected/loaded survey. It is helpful to QC the survey orientation (true north display).

The information panel gives the basic survey information (in-line, cross-line, Z-ranges, bin-size and 2/3D...etc)

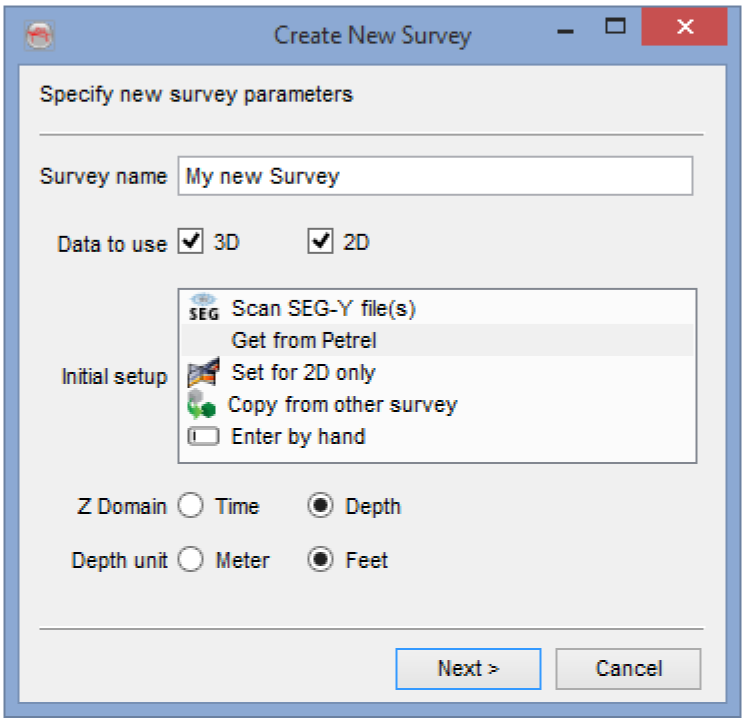
The bottom notes panel is the place to put important notes about the project (Type of Project, Client Company, Contact Person etc ...).



For more further information about this window, please go to Survey selection window.

Setup 2D & 3D Survey

To setup a new Survey (2D/3D) press *New* button  in the survey selection window. The *Create New Survey* window will pop up (see below) which will lead you through the setup process.



The survey setup defines the survey box. The key parameters are the survey type and its respective coordinates. These can be entered either manually, by scanning a SEG-Y file, or copied from another OpendTect survey if one already exists in the data root folder. Optionally, in case you have workstation access, you can get them from GeoFrame as well. Or if the OpendTect survey already exists, copy the coordinates by pointing the other survey.

For full details of this setup process, please refer to New Survey Window (and subsequent entries).

> **Note:** In OpendTect, everything is tied to a 3D (X,Y,Z) with inline/crossline coordinate system. This is regardless of whether you have 3D data at all.

Thus, for a survey which contains only 2D lines, a 'fake' 3D survey box of inline/crossline system is needed. If you want to use both 3D and 2D data, set up the survey according to the 3D cube.

Survey Name and Type

The top most panel of the survey setup window defines two important things along with its disk location. The *Survey directory* name is entered in the first field. A directory with this name would be created in the root where you will save the OpendTect survey. The field for *Full Survey name* is filled with proper name of the survey that will be seen later on in the OpendTect survey selection window. Any name can be given to the survey and its directory but we advise you to gives similar names. The name can be edited later. In third field select the location on disk where you want to save this survey.

The most important thing is the *Survey type* which can be *Only 3D, Both 2D and 3D* or *Only 2D*. The type is set accordingly:

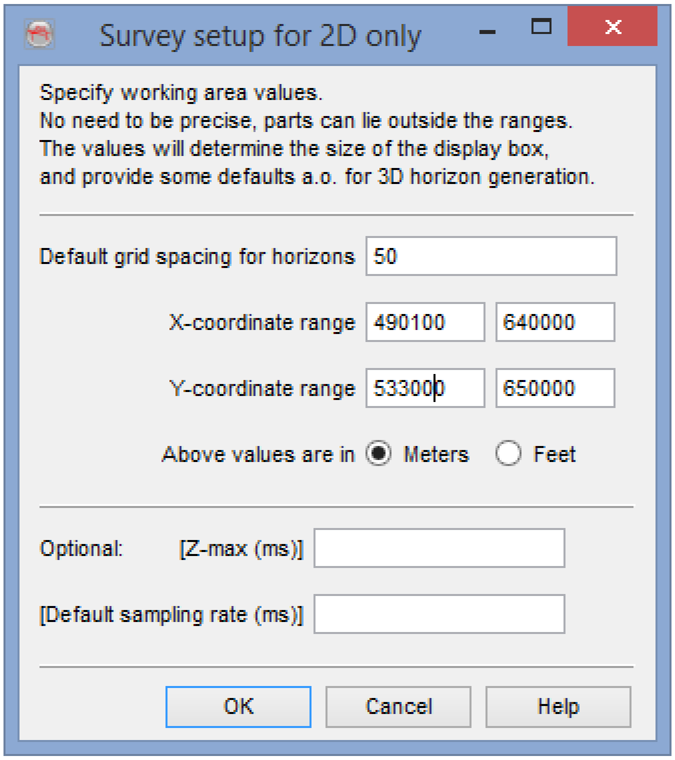
- Only 3D is used when the survey contains only 3D cube(s).
- Both 2D and 3D is set when survey contains both 3D cube(s) and 2D line(s).
- Only 2D is set when survey contains only 2D line(s). Remember: A fake inline/crossline system is defined to setup such a survey regardless of the fact that the 3D cube is not actually present.

Survey Ranges

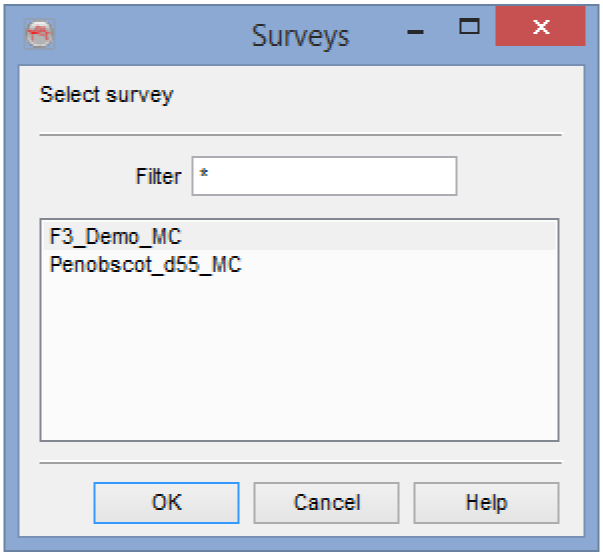
In the second panel of survey setup window the fields for Inline, Crossline and Z-ranges are filled with following possible ways:

- **Filling the ranges by scanning a SEG-Y file:** Set the Survey ranges in Scan SEG-Y File(s). The SEG-Y import window will pop up. In this window, select the input file/volume and press OK to start scanning the input file. After the scan is completed, the survey ranges will be filled automatically.
- **Entering Manually:** If you already know the extents of survey ranges (Inline/Crossline/Z-range), just fill the maximum and minimum ranges and corresponding steps.
- **For 2D only:** Set the Survey ranges into Set for 2D only and press GO... The window will pop up (as shown below) in which the average trace distance (which determines the grid spacing of the 3D horizons) and the ranges for XY-coordinates are entered; Fill the approximate values and press OK afterward. These parameters are the survey characteristics used to define the survey box. In the survey ranges the fake ranges for inline, crossline are automatically set. Then fill in the Z

range (no precision needed; 2D lines can stick out of the display box in all directions). The limit of the survey box will not affect the loading of 2D lines. The lines outside the survey area would be loaded normally. And that should be it. Press OK and select the new survey.



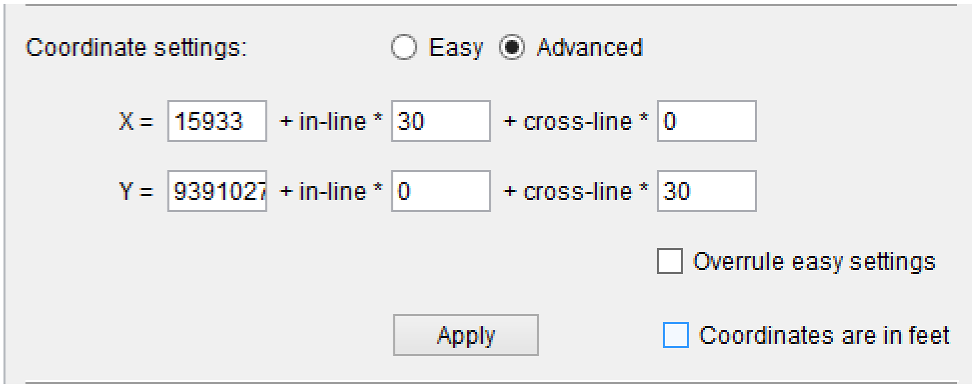
- **Get from GeoFrame:** This option is used if you have workstation access to get the survey ranges from a GeoFrame project.
- **Copy from other Survey:** Additionally, if the survey already exists, you can copy the ranges from that OpendTect survey.



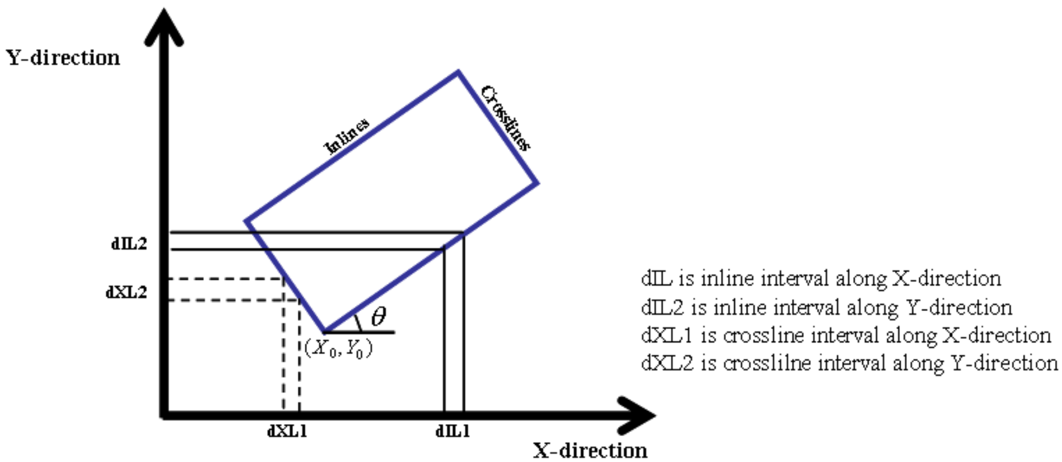
Coordinate Settings

The relationship between inline/crossline and X/Y is setup generally by using this panel. The coordinates are automatically setup after scanning the SEG-Y file. You will only fill them if you are inserting survey ranges manually. Nevertheless, the three point (easy) method is not accurate because of rounding off errors which may not be 100% accurate. Both methods have been described here.

- In **Easy mode**, the coordinates are transformed by using three known points. Under Coordinate settings panel of Survey setup window, fill the inline and cross-line and corresponding known XY-transformation as shown in figure of previous section.
- In **Advanced mode**, the XY coordinates are calculated by using an equation shown below. The inputs required from user are origin (X,Y) and inline/crossline intervals along X/Y directions respectively. By checking the *Overrule easy settings*, the current easy settings would be overwritten.



The basic idea behind this has been elaborated in following figure:



The equation can be written as:

$$\begin{bmatrix} X \\ Y \end{bmatrix} = \begin{bmatrix} X_0 \\ Y_0 \end{bmatrix} + \begin{bmatrix} dIL & 0 \\ 0 & dXL \end{bmatrix} * \begin{bmatrix} \cos\theta & -\sin\theta \\ \sin\theta & \cos\theta \end{bmatrix} \begin{bmatrix} Inline\# \\ crossline\# \end{bmatrix}$$

** The required inputs include origin X,Y pair, inline and crossline interval in both X & Y directions.

Note: If the provided coordinates are in feet, the Coordinates are in feet check box must be checked. If the survey is a time survey and the Z-range units are set to milli-seconds (ms), you might want to display the depth in feet in the scene/annotations. For the later purpose the check box Display depths in feet should be checked.

Data Loading

The data loading is briefly addressed in two sections. The first section will guide in loading the 2D/3D SEG-Y data, while the second will guide in loading well data.

Load Seismic (2D & 3D)

The loading of SEG-Y data is done via *Import* sub menu under survey main menu (*Survey > Import > Seismic > SEG-Y..*)

To start loading of **3D SEG-Y** Volume, follow the steps:

1. Go to *Survey >Import > Seismic >SEG-Y...* In SEG-Y Tool window choose File type: 3D Volume to import the SEG-Y data.
2. In the Input SEG-Y field, select and locate the input SEG-Y file.
3. Set check to the Multiple files to import more SEG-Y files. Write the number of files in the first text field and number of steps (e.g. Vol-1, Vol-2, Vol-3, ... Vol-n). Where 'n' is number.
4. Select the input file type: 2D/3D-Pre or Poststack data.
5. Select the number of traces that will be examined to display the trace header information.
6. Optionally, if you want to overrule the number of samples, set check to this field and fill the correct number of samples.
7. Select an appropriate SEG-Y file format.
8. If the bytes in a header have to be swapped, it can be set here.
9. Press OK to proceed further
10. The SEG-Y Scan and SEG-Y Examiner window will launch in next step. Fill the appropriate byte locations in the SEG-Y scan window. Write the output cube name and press OK to start loading the SEG-Y file.

Load Well Data

In OpendTect, the well data has been organized into four sub-categories: Well tracks, Logs, TD Models and Markers. Each category is either loaded via Survey > Import menu or via the Well manager. In this section, we will cover how to load/setup a complete well in OpendTect.

Load Well track and Time to depth Model (via *Import*): To load a well track, go to the main menu bar and select *Survey > Import > Well > Ascii > Track*. The Import Well Track window will be launched. In the Well Track file field select the input well track file. The

Well track file should contain three main columns i.e. X, Y, TVDSS (True Vertical Depth Sub Surface) and optional MD (Measure Depth) column. The file header can be defined in this window. Click the *Examine* button to view the input file before loading, and to determine the header type. See Import - Well section to learn how to set the file headers (fixed size/variable). The format definition is used to specify the columns (XYZ) definition and unit system of the input ASCII file. Optionally, a vertical well can also be defined if the file doesn't exist. This is done by setting uncheck to *Well Track File* field. In this case, specify the XY coordinates and depth range (default meters).

In the depth to time model field, locate and select the appropriate time-depth file. Specify the file header and format definitions. If the time-depth model/checkshot does not exist, set uncheck to this fill to set a constant velocity model to load the well. Later on the time-depth model/checkshot can be imported via the Well Manager. Additionally, specify whether the file is a checkshot data or not. The additional information (Well ID, Operator, State, Country) can be included in the well track by selecting the Advanced/Optional button.

Name the *Output well* and press *Go*, to complete the loading of well track. It will return a message, "Well track imported successfully". Press *Ok* to close information window. Press *Dismiss* to exit.

> **Note:** The surface coordinates and elevation above surface are defined from the well track file. In normal cases, these definitions will be in the first line of the well track file. If the well track file starts below the datum (KB, Kelly Bushing), you will need to manually adjust these parameters in the well track file before importing it as a well track. Elevation above the surface is normally considered positive if the reference datum is above sea level. Conversely, if the elevation reference is below sea level, you can define the negative values in the elevation as those above the surface elevation reference.

Load Logs & Markers (via *Well manager*): Launch the *Well manager* . Here you will find the loaded well track(s) from the previous step in the left panel.

To import *Logs*, select the recently loaded well and press the *Import* button. In pop-up window, select the input LAS log file, and then examine the file. The remainder of the window will be automatically filled in from input file. Importantly, if you only want to load certain logs, highlight only the wanted logs. (By default all available logs will automatically be highlighted.) Press *Ok*. The logs will be loaded and will appear in the right panel.

To import *Markers*, select the recently loaded well and press the Markers button. In pop-up window, select the *Read file* button and follow the same steps as before when loading Ascii files. Alternately, instead of reading a file, you can manually type in the depth, name and assign marker colors.

Updating well tracks, checkshot data, and depth to time models can also be done in this *Well file management* window.

Display Loaded Data

To display the loaded Seismic data, use a mouse click on inline/crossline in the tree and select the *Add* sub menu from the pop-up tree. A seismic line will be displayed in the middle of a survey that will contain a blank attribute. Right click on the attribute to select and display the loaded data. In a similar way, a 2D seismic line can also be loaded from the 2D Seismic element.

In order to load a horizon, click on the Horizon element in the tree to select the interpreted/imported horizons from a list. Select *Add* option from the pop-up menu tree. In the pop-up window, select one or more horizons to be displayed in the scene. The horizon with its Z-values will be displayed in the scene. You can edit/select the desired color-spectrum (Color Tables). And, just as for inline/crossline/z-slice, you can also add a defined attribute from the active attribute set that will be calculated on the fly along this horizon.

To display the loaded Wells in the scene, click on the tree element '*Well*' and click '*Add*'...Select the desired Wells. The Wells would be displayed in the scene. To manipulate a Well and set its properties, right click on the Well name in the tree. The pop-up list will show the options available. From here you can edit Well track, Show/Hide Markers/Logs, change the properties, select logs, etc.

Troubleshooting:

- Often, users do not find a good trace number in the SEG-Y headers. Without that, weird things can happen. Use the *Examine* button in the SEG-Y setup dialog to figure out what header value may be usable as trace number. Or better: find the number that you recognize the data by.
- If X and Y in the SEG-Y file are not correct, the data may be a long way outside the Survey box. Then, nothing may be there after loading a line, or even the entire Survey box disappears.
- If you have trouble with your graphics card, the lines may remain white after loading the Seismic data. But if your mouse goes over the line, you can see

Coordinates, trace numbers and amplitudes. Then try to:

- install the latest driver
- make sure you have a good card (hopefully NVIDIA-based)
- disable OpenGL shading in *Utilities >Settings >Look and feel... Visualization tab...*

If this does not help, please check through the Appendix-C (SEG-Y Check list)

Interpret Horizons

Please see the sub-sections below for details:

3D Horizons
Edit 3D Horizons
2D Horizons

3D Horizons

Horizons can be tracked in various modes:

- Autotracked
- Tracked on lines (in the 3D scene or 2D viewer)
- Manually picked on lines (in the 3D scene or 2D viewer)

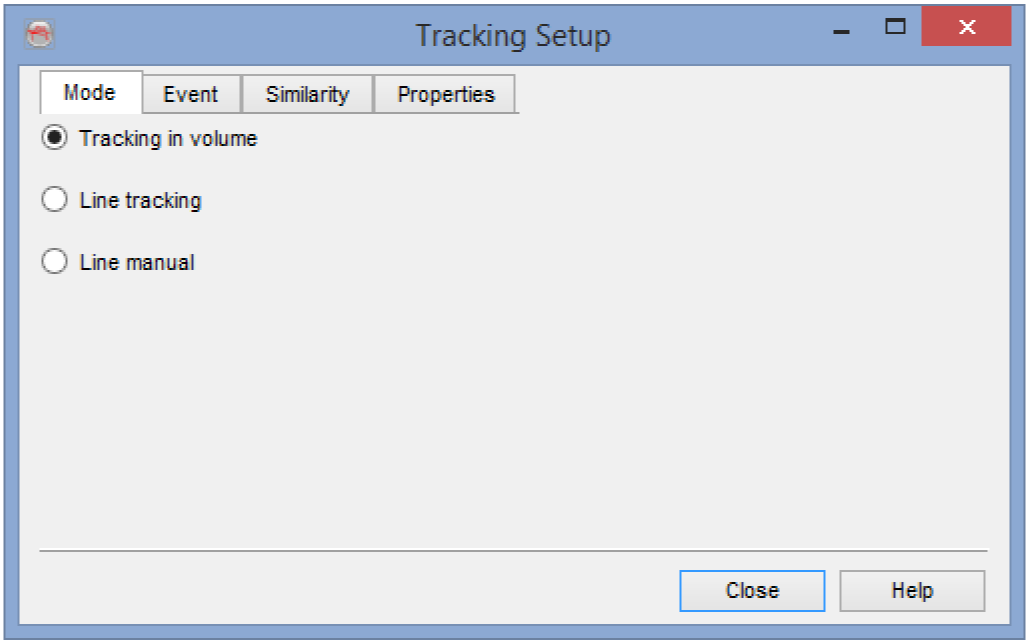
Basic Workflow:

1. Add an inline/crossline and display the seismic data you want to use for the interpretation
2. Right-click on *Horizon* in the tree and select *Track New ....*
3. Define mode and tracker settings.
4. Pick seeds, Autotrack (or interpret the horizon on lines).
5. QC and Edit the horizon.

These steps are explained more fully below:

1. Define mode and tracker settings
Right-click "Horizon" in the tree and select *New* ....This will launch a tracking setup which contains several tabs:

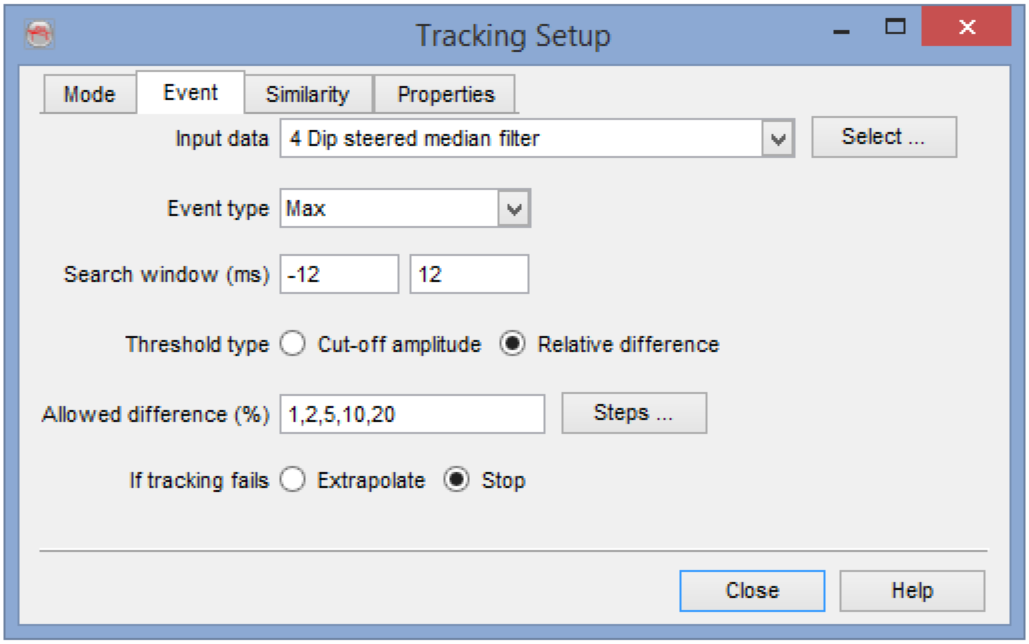
- **Mode:**



Choose the tracking mode:

- *Tracking in volume* is used to auto-track a horizon inside a user defined tracking area (3D (sub)volume). The tracker area can be moved and adjusted inside the survey box. This mode is preferred for most horizon tracking.
- *Line tracking* is used to track a horizon on a line (inline or crossline). This mode gives the interpreter more control and is the one mainly used when creating the initial horizon grid. In between the seeds the horizon is tracked along the line. The result is a grid that needs to be filled, either by autotracking or interpolation.
- *Line manual* mode will manually pick a horizon (interpolated line) between two seeds on lines. This workflow differs from the line tracking as in this case seeds are linked by a straight line: there is no automated tracking. This mode is used in difficult areas, for example to cross faults, push through noise zones or interpret unconformities.

- **Event:**



Volume tracking and *line tracking* are automated tracking workflows. Thus the user needs to define several tracking criteria:

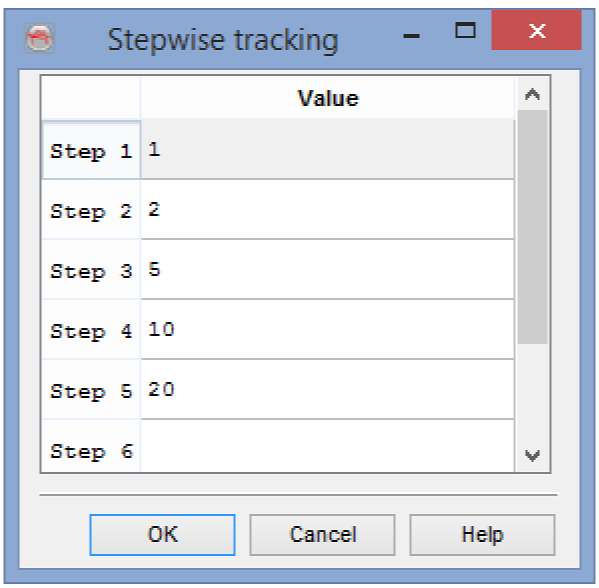
- Input data: The input data is automatically selected as being any seismic attribute active in the tree. This can be the original seismic volume, or a filtered seismic volume (preferred) or any other attribute. The horizon is linked to this input seismic. you can change the input seismic at any time: it won't change your saved interpretation. It can be useful if for example you receive an improve seismic and you want to re-track in volume your horizon according to this new seismic.
- Event type: Specify the event type you want to track. The tracker can track negative reflectors (Min), positive reflectors (Max), a Z-type zero-crossing (0+-), or a S-type zero-crossing (0-+). If the tracking does not seem to work, check that the event type corresponds to the event you actually interpreted the seed(s).
- Search Window: Search window: Automated tracking is based on amplitude: the he tracker searches for the chosen event type based on amplitude in a time window relative to the last tracked sample (see drawing A, below).
- Threshold type:
  - *Cut-off amplitude*: Here, an absolute amplitude is used as the stopping criteria for the tracker. When the tracker encounters a value below this threshold value it stops tracking. (For a max-event the tracker stops if the value is below this threshold value,

and for a min-event when it is above this threshold value). Tip: point your mouse at the event and the amplitude value is displayed at the bottom of your screen.

- *Relative difference*: The tracker will compare the amplitude of the last tracked point to the amplitude of the point that is candidate for tracking. If the difference exceeds the chosen percentage, the tracker stops tracking.

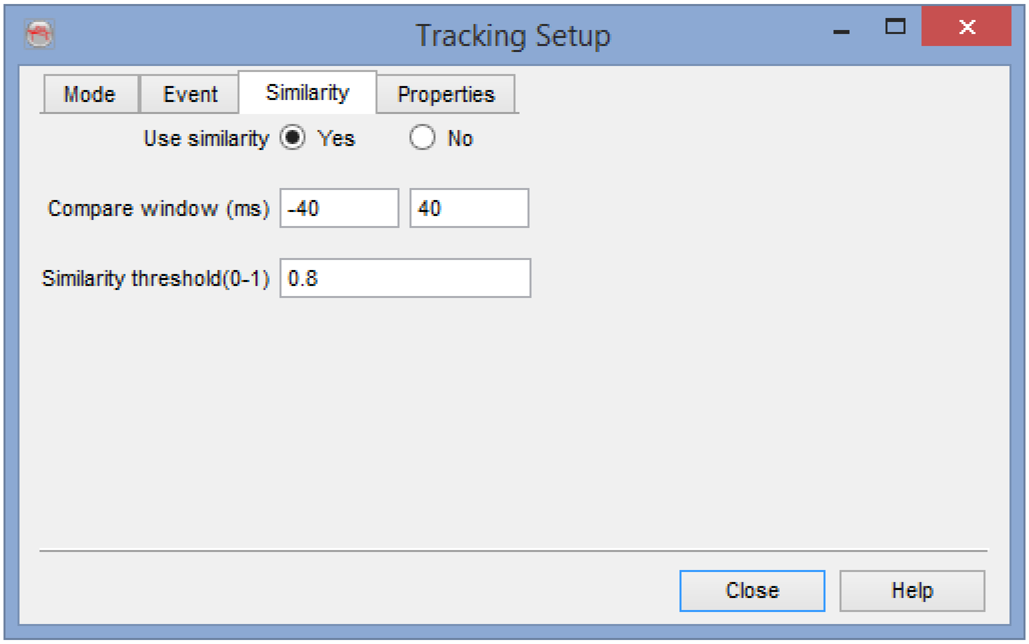
- Steps...: Step-wise tracking results in a better tracked horizons. Good parts of the horizon are tracked first, followed by the more difficult areas. The tracker will first track those part of the horizon that have a low difference to the seeds or parts that have a high amplitude. In subsequent steps the tracker settings become less strict. Therefore, the horizon will be of better quality and needs less editing.

  The user can define the subsequent percentage values (incremental: eg 1,2,5,10,20) , or subsequent amplitude values (decremental e.g. 2500,2000,1500,1000,500)

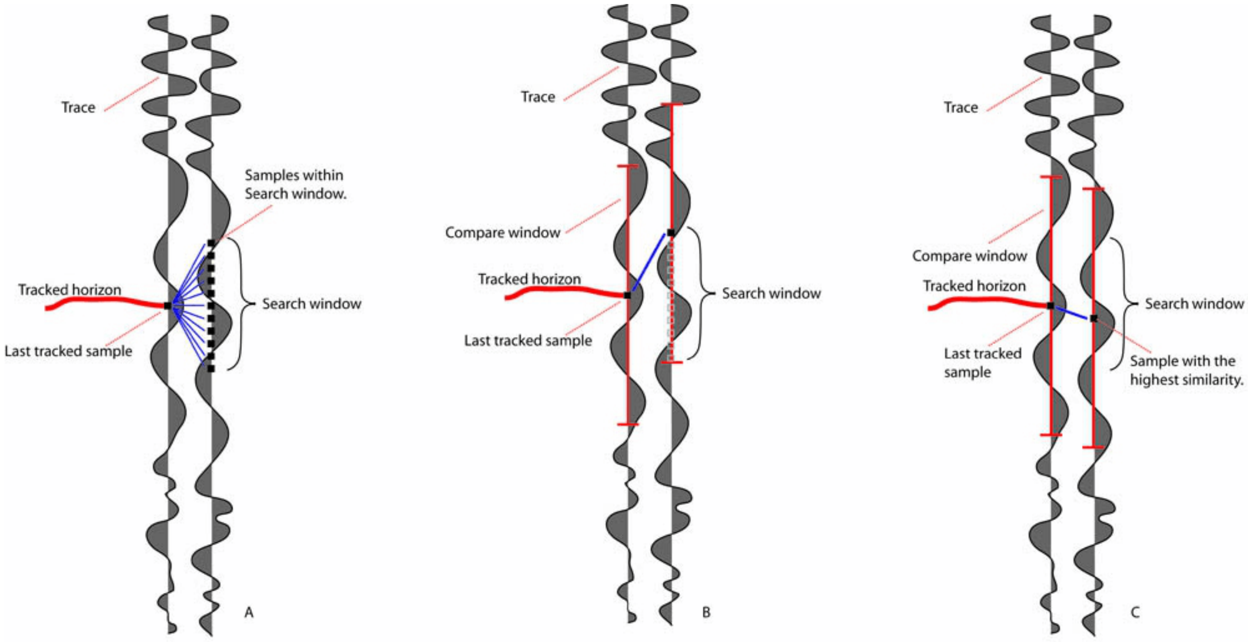


If tracking fails..: If the tracker can not find a neighbouring point (that complies with the specified relative difference or cut-off amplitude) it can either stop tracking or extrapolate the horizon. (Tip: When the tracker stops tracking before you want it to, adjust the Threshold value and/or *Similarity threshold* before choosing the *extrapolate* option.)

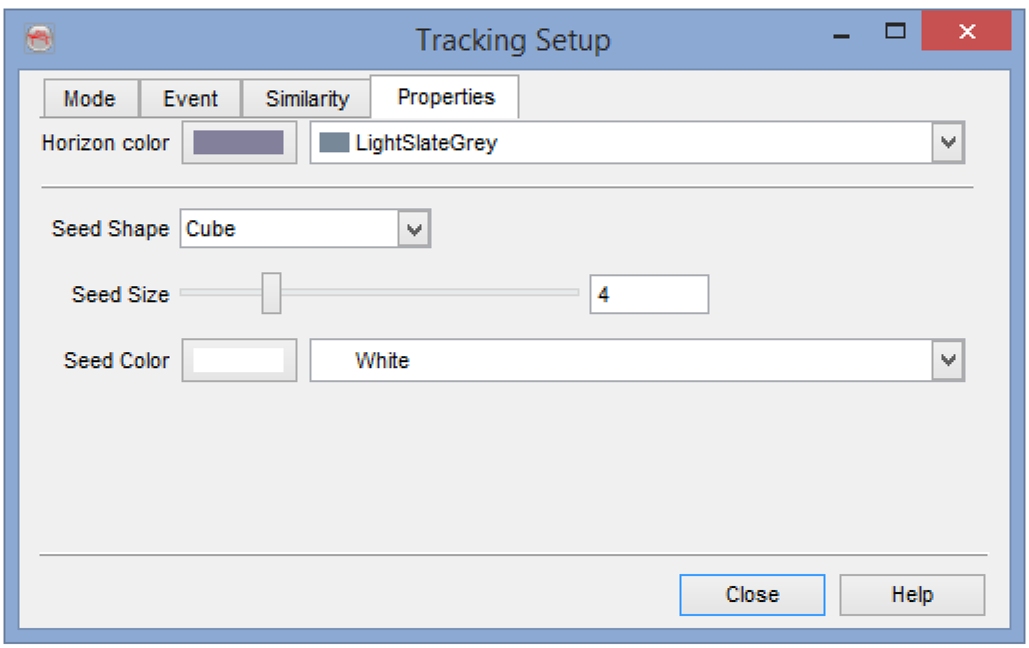
- **Similarity:**



Similarity is a kind of cross-correlation. A part of a trace (trace segment) around the last tracked point is compared to all the trace segments on the neighboring traces around the points that lie within the Search window (See figure below). The distance between the points is typically 1 ms. The number of comparisons is thus controlled by the search window, while the compare window controls the length of the trace segments. The measure of Similarity between the trace segments lies between 0 and 1. The tracker will choose the point that has the highest similarity drawing C, below). When the point with the highest similarity has a value below the defined threshold the tracker stops tracking. Tracking with "Similarity" is optional. It is more accurate, but increases the computation time.

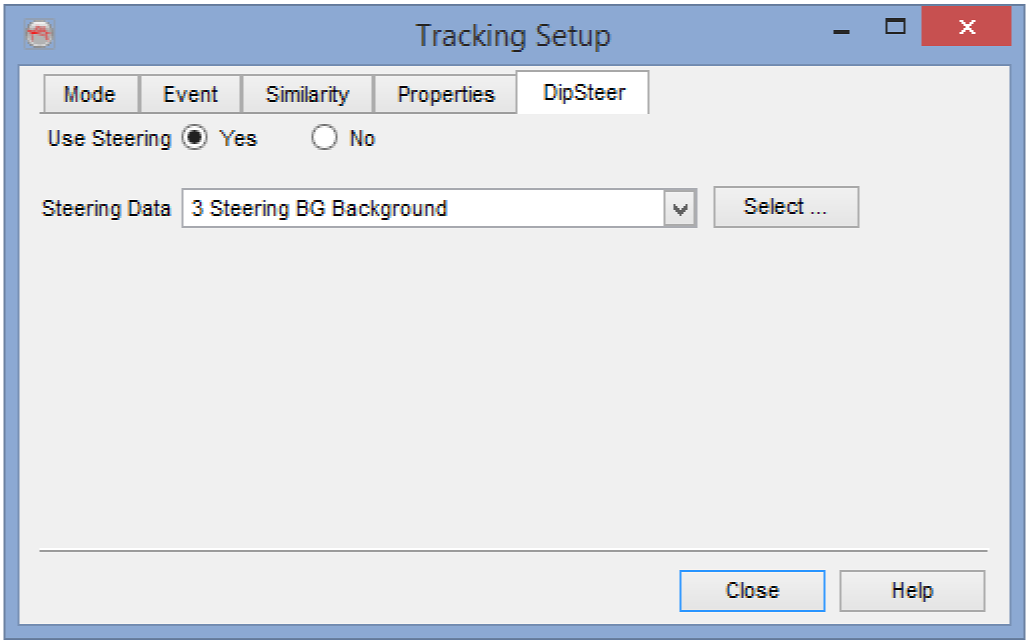


- **Properties:**



This tab is used to adjust the display properties of the horizon.

- **Dip Steering:**



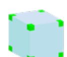
The dipsteering cube can be now added as constraint for horizon interpretation. This will improve the horizon tracking especially in the areas of dipping reflectors. Dip steering gives structural information.

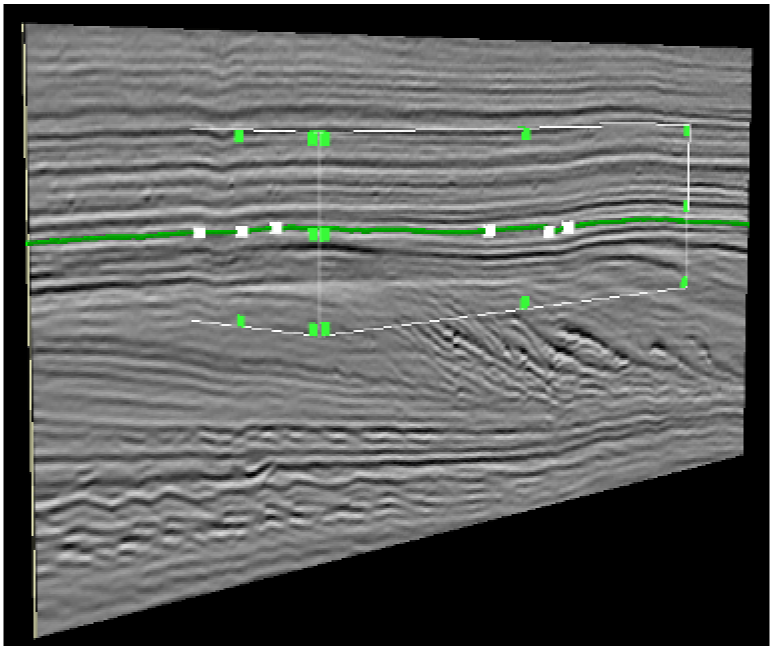
For loading a color blended horizon in a scene, please take a look at [dGB's How to...-manuals]

2. Tracking

After adjusting the parameters in the tracker setup (which can remain open during tracking), start picking seeds on a displayed inline/crossline. The corresponding tracking controls are described in the tracking toolbar section.

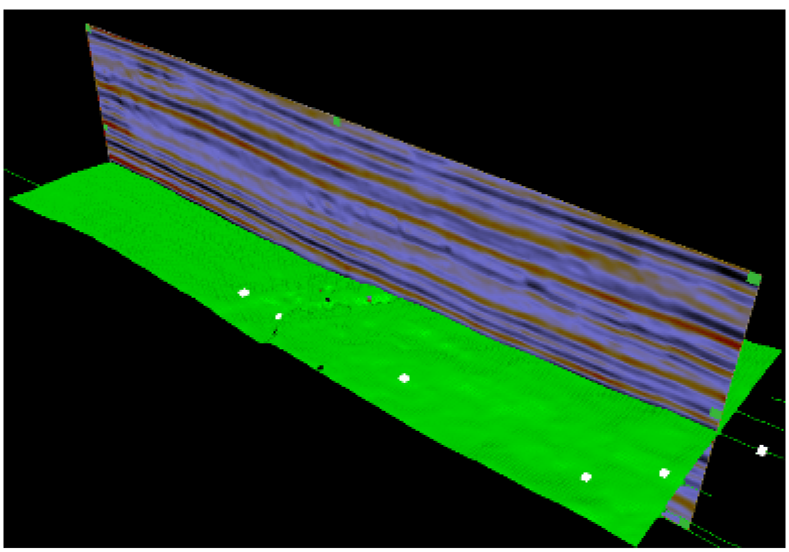
Tracking in volume:

Pick one or more seeds and display the tracking area . Resize the tracking area by dragging the green anchors on the edges of the cube, but do not exceed your computer's memory size or the size of your graphics card memory. If you run out of computer memory, OpendTect will crash.



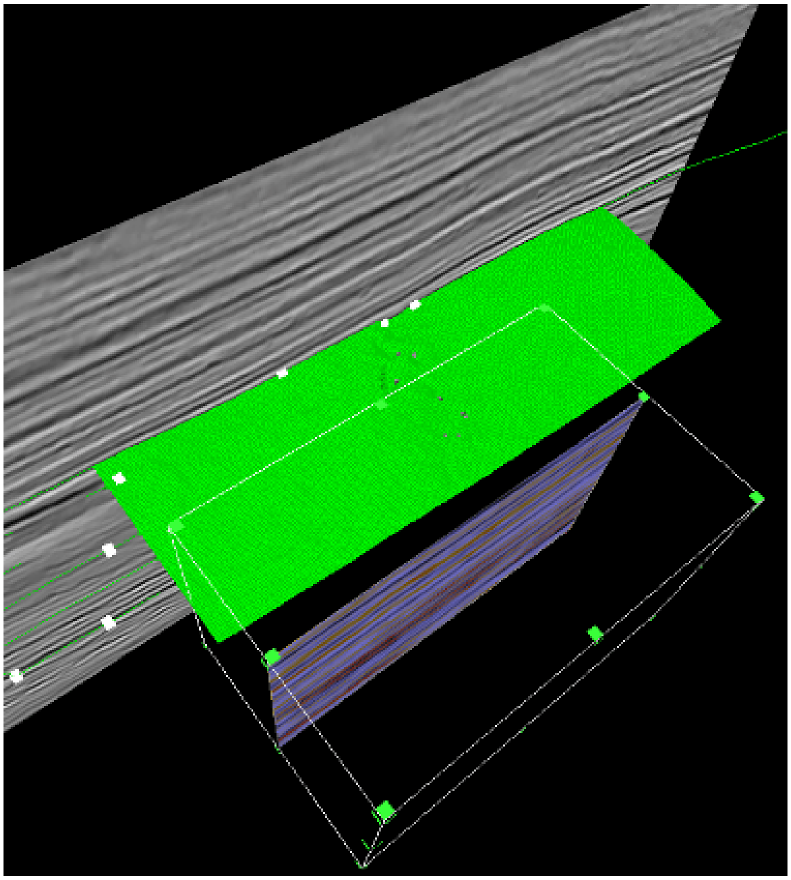
Seeds picked and tracking area resized

Now, click the auto-track icon . After the input data is loaded inside the tracking area the horizon is tracked

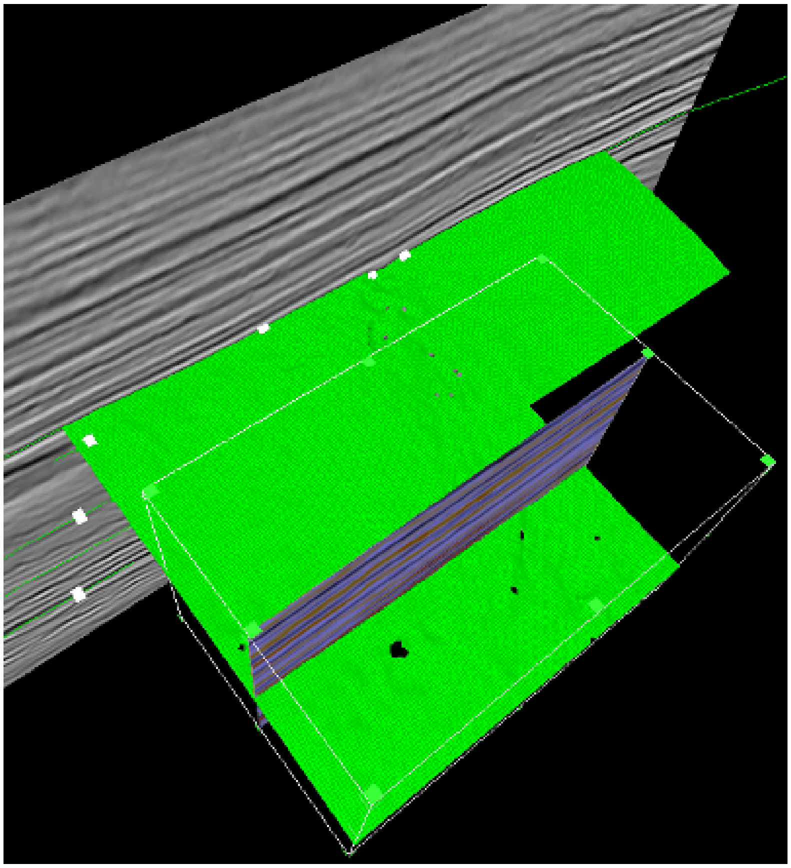


Auto-track the horizon within the tracking area

After tracking a part of the 3D-horizon, move the tracking cube to next place (click the top of the tracking cube and drag). A small part of the horizon should be inside the new position of the tracking area.


Move the tracking cube to next location

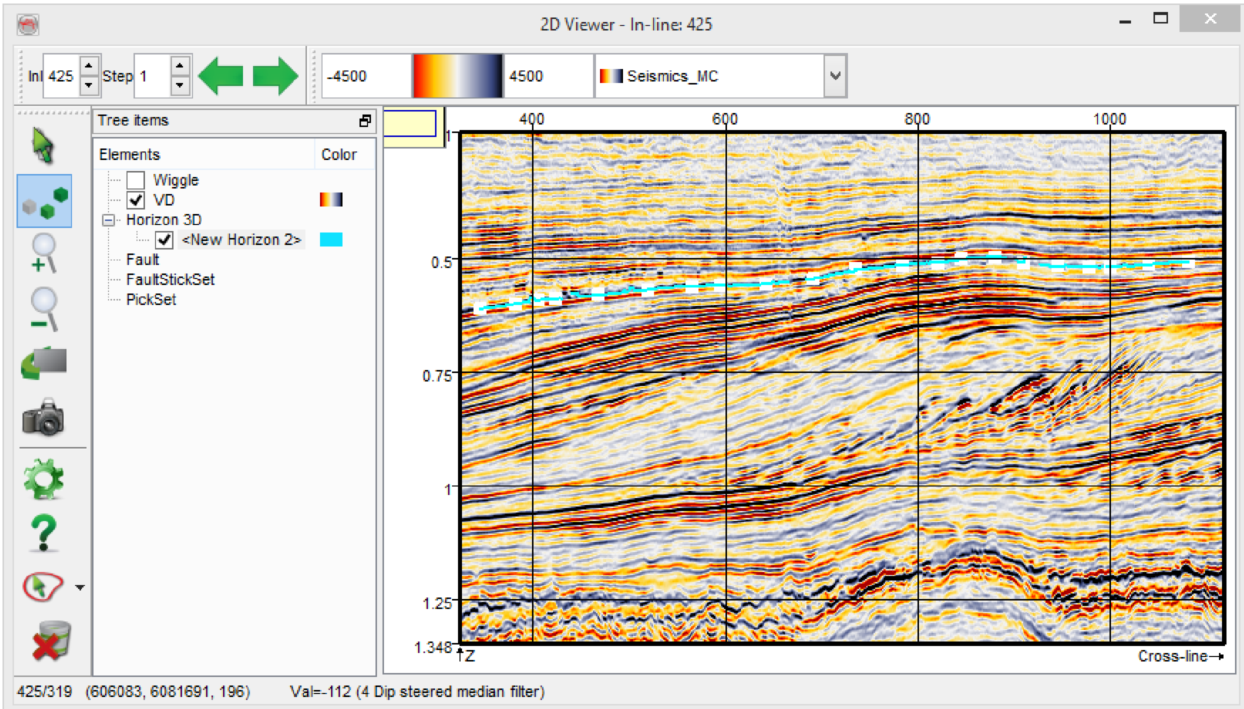
When the Cube is at its desired position, click  again.



Next location is auto-tracked

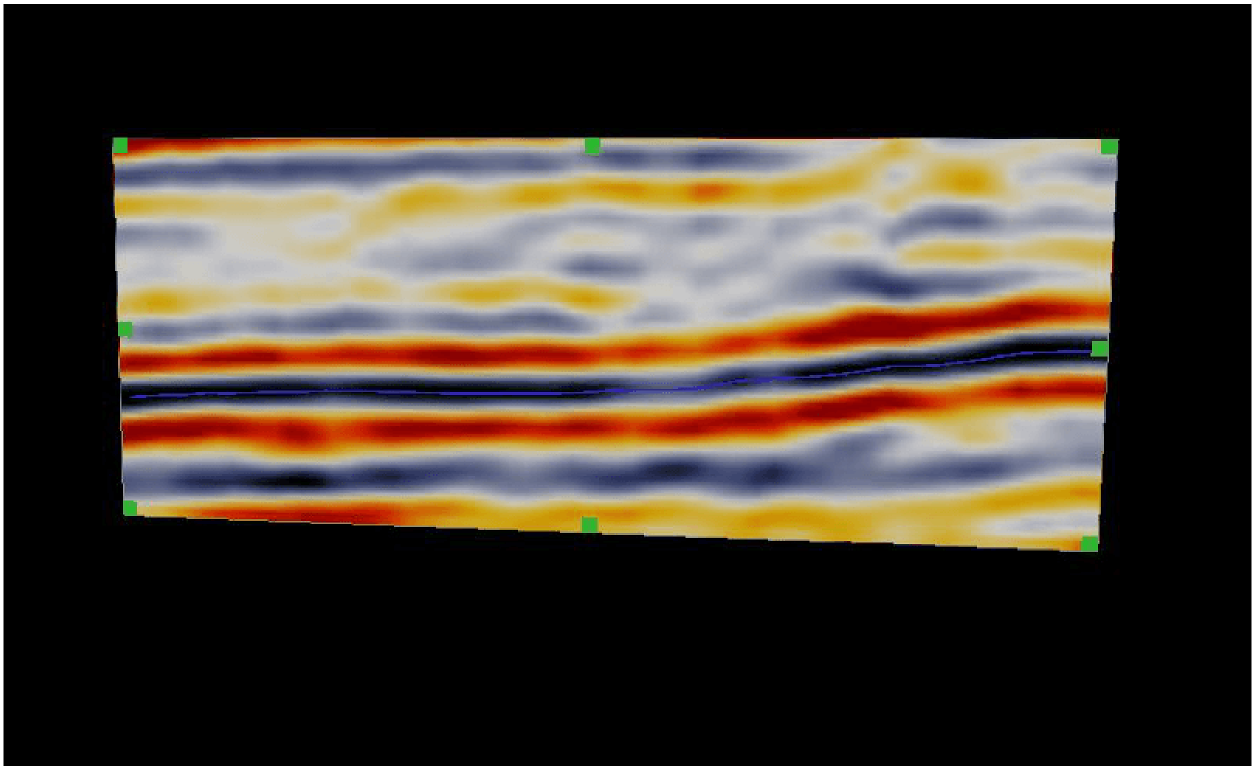
Optionally, you can chose to track from the seeds only (only the seeds are taken into account for the tracking).

Line tracking and line manual:
Interpreting on lines creates a grid, which allows for greater user control. Line tracking and manual interpretation can be done directly in the 3D scene and optionally in the 2D viewer for better accuracy. To launch a 2D viewer, the tree is used. Right click on the seismic data (at an inline/crossline) in the tree... and select *Display > 2D Viewer - VD*. Select the track/edit mode button and pick seeds, then move the line and repeat the process.



Edit 3D Horizons

When a horizon (or a part of horizon) is tracked, the quality can be assessed by using the QC tool . The horizon is displayed at section as a line on inline and/or crossline elements and on the *QC plane*. Drag the QC plane over the tracked horizon to QC the tracking. SHIFT-click on the tracker plane to QC in another direction or select another orientation in the tracker toolbar. If the horizon needs editing, there are several options:



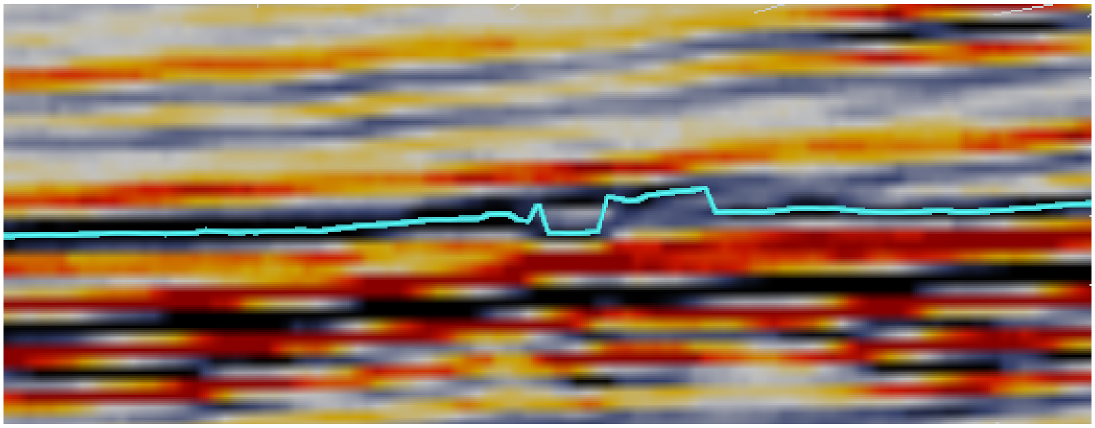
- *Retrack All.* Retrack all: If the tracking parameters need to be updated, change the settings and click the icon . This will first remove all the autotracked parts of the horizon while keeping the seeds. The horizon will then be re-tracked with the new tracker settings in the tracking area.
- *Polygonal / Rectangular tool.* Remove a part of a horizon with either a polygon or a rectangle and re-track it. First select the polygon / rectangular selection tool of the tracker toolbar and select an area to delete. Remove the part of the horizon selected in the tree inside the polygon by clicking the icon . Please note that your polygon is not self intersecting, otherwise the remove icon will not work.

  To fill the hole again:

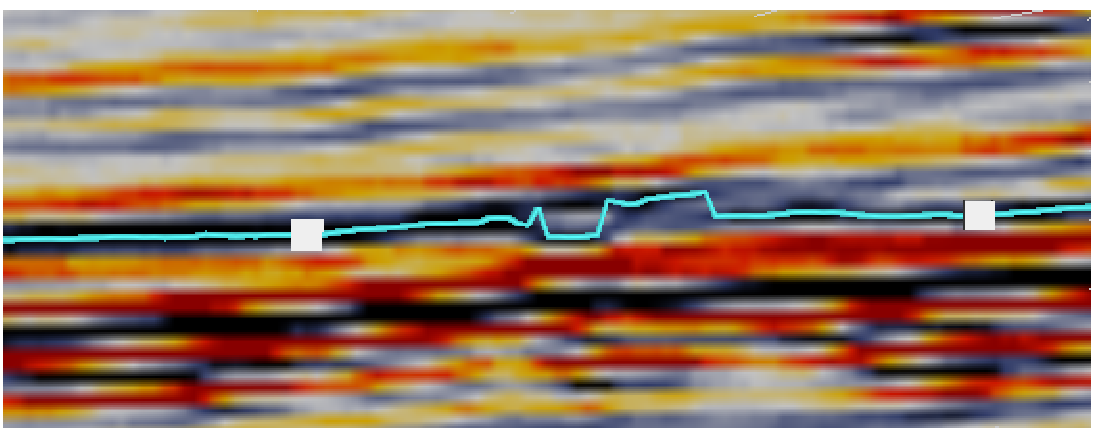
  - Autotrack will track the hole from the edges
  - Pick a seed on the QC-plain and track from seeds only
  - Interpolate

Tip: When QC-ing horizons, it may be necessary to manually correct auto-tracker errors over a limited region. This can be done by switching to '*Line Manual*' mode and placing two control points as boundaries to the area you wish to edit/re-draw. These control points can be set down using *Ctrl+Shft+left-mouse-button*. Changes made to the horizon are then limited to the area between these points.

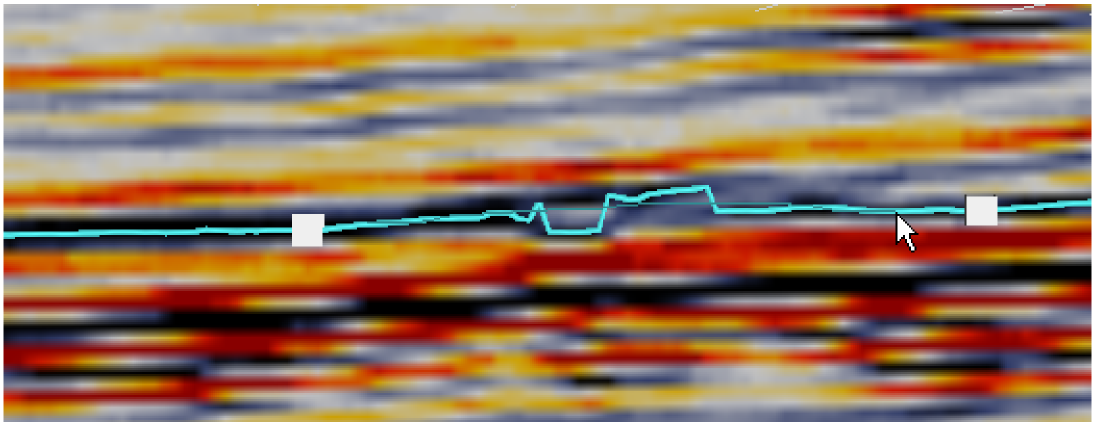
1. Whilst QC-ing in '*Display--> Only at sections*', locate a section that requires editing:



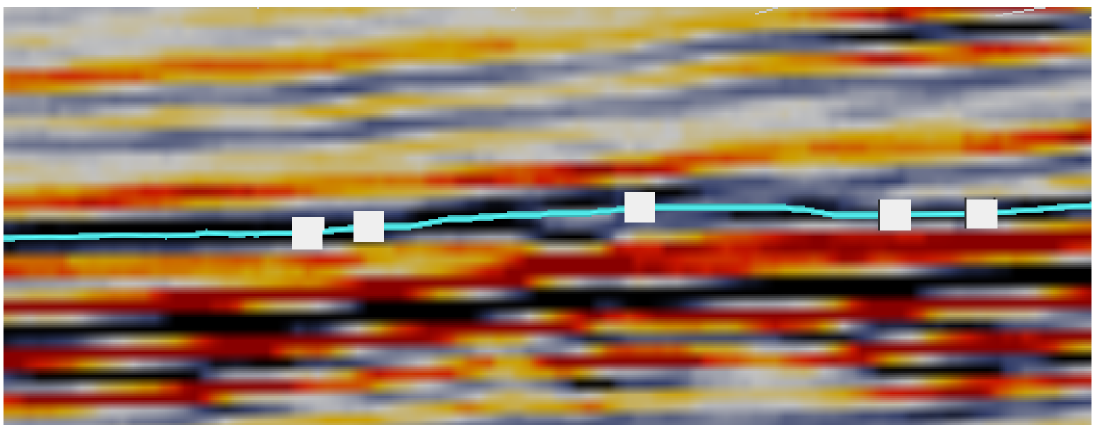
2. Switch to '*Line Manual*' mode and use *Ctrl+Shft+left-mouse-button* to insert two control points, one either side of the to-be-edited section:



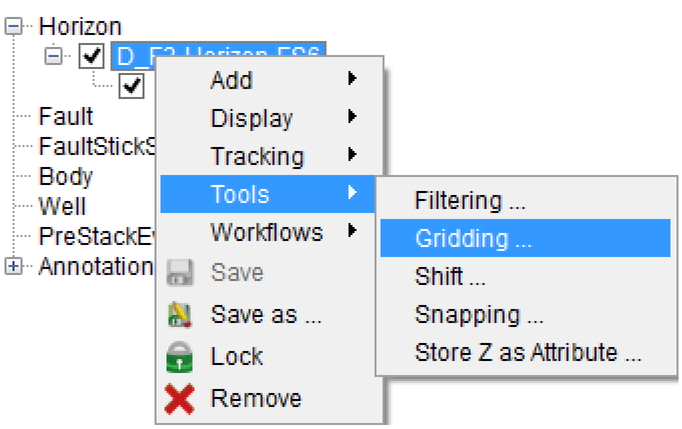
3. Holding the left mouse button down, draw the new line where needed inside the two control points:



4. On releasing the left mouse button, the old horizon is replaced with your edited version:



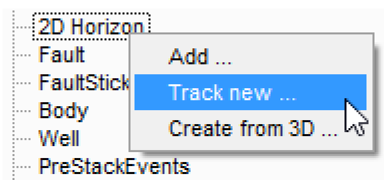
Tools:
There are several tools available to manipulate a horizon: '*Filter*', '*Gridding*', '*Shift*', '*Snapping*' and '*Store Z as attribute*'



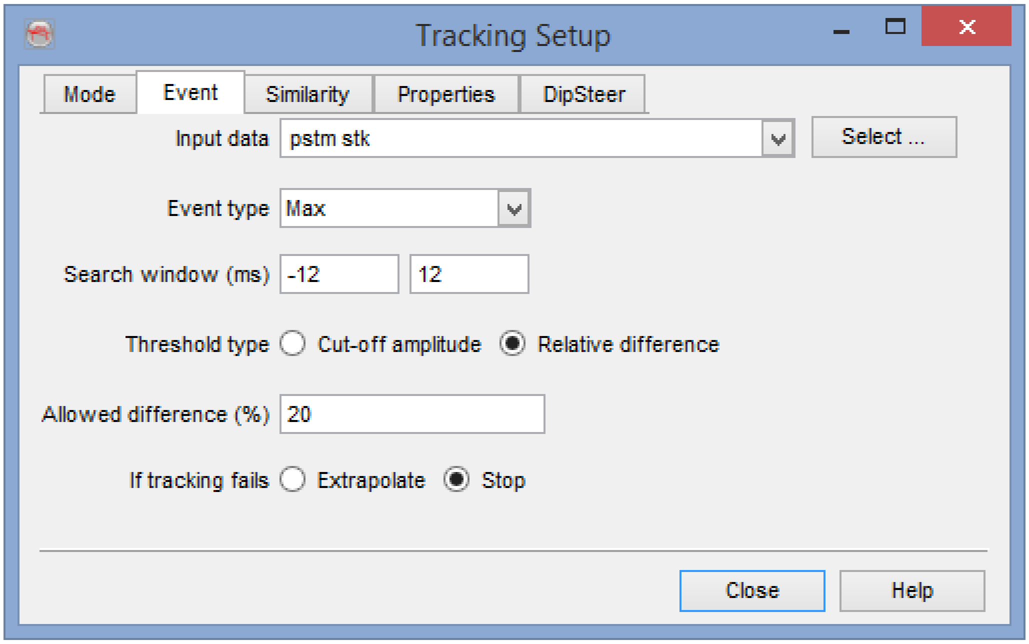
Note: Once you are satisfied with the results, do not forget to save your horizon using  .

2D Horizons

For a 2D horizon tracking, the workflow is slightly different. The tracking wizard is similar to the 3D tracking wizard shown in the previous section. To start tracking a 2D horizon, click on 2D Horizon element of the tree and select New option from the sub-menu. This will launch the tracking Setup window. The tracking setup contains the tabs similar to that of a 3D horizon. See description of the tracking setup in the chapter: Interpret 3D horizon.



First, define the tracking mode. Go to the *Event* tab to define the tracking parameters (time gate, event, thresholds etc). In this tab, the data on which to track the horizon must be selected.



Note The same horizon can be picked across several 2D lines, providing the horizon is interpreted on lines showing the same 'Input data' (see *Event* tab)

After defining the event parameters, pick a seed on a displayed 2D line in a scene. If the corresponding settings for event are not appropriate, edit and change the parameters (e.g. time gate and threshold type) and press *Retrack* icon in the tracking toolbar. This will re-track the same horizon with new settings. After adjusting the appropriate settings for the horizon, the tracking parameter window can be closed by pressing *Dismiss* button. When closing this window, the tracking settings are saved.

Note: In order to change the settings (for tracking), the tracking parameter window can be launched later on by right clicking on a horizon (in the tree) selecting Tracking > Change setup (click on the tracking setup icon ). Also at the bottom of the window, the tracking toolbar appears. This is routinely used to update the modes and for switching between seeds (TAB-key).

For seed picking the options are:

* left mouse-click on plane = pick seed & local track
+ Ctrl-Shift = drop seed
* left mouse-click on seed = local retrack
+ Ctrl = remove seed & local retrack

+ Shift = remove seed & local erase
+ Ctrl-Shift = undrop seed

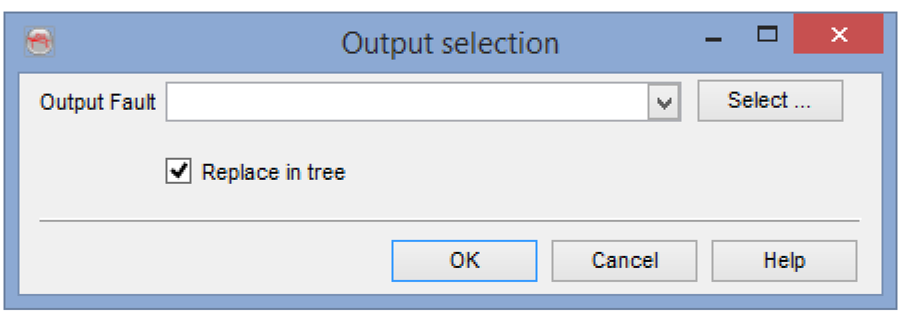
After finishing the horizon tracking on one 2D line, start tracking on next 2D line. The horizon can be displayed only at corresponding lines by right clicking on the horizon name in the tree, and selecting "Display only at sections". There are also several others options in the pop-up tree.

Tip: Always save the horizon routinely.

Interpret Faults

Click on the element **Fault** in the tree scene and select *Add* to select a fault or *New* to map a new one.

New will insert a new fault {New Fault x} with a blank attribute. Insert the inline/crossline in the scene where you want to interpret the faults. Highlight the newly inserted fault in the tree and start interpreting a fault by clicking on inline/crossline. In OpendTect, the fault model is arranged in an ordered sequence of faultsticks that are picked on vertical planes (inlines/crosslines) and horizontal planes (Z slices). Thus faults originally picked on inline/crossline cannot be picked on a Z-slice. When you are done, right-click on the fault and select *Save*. Write an appropriate name of the fault and leave the *Replace in tree* checked. Press OK. This will save and update the name of the interpreted fault. To continue fault picking on a next inline/crossline or Z-slice, a user can take benefit of the scrolling options available for inline/crossline/Z-slices.

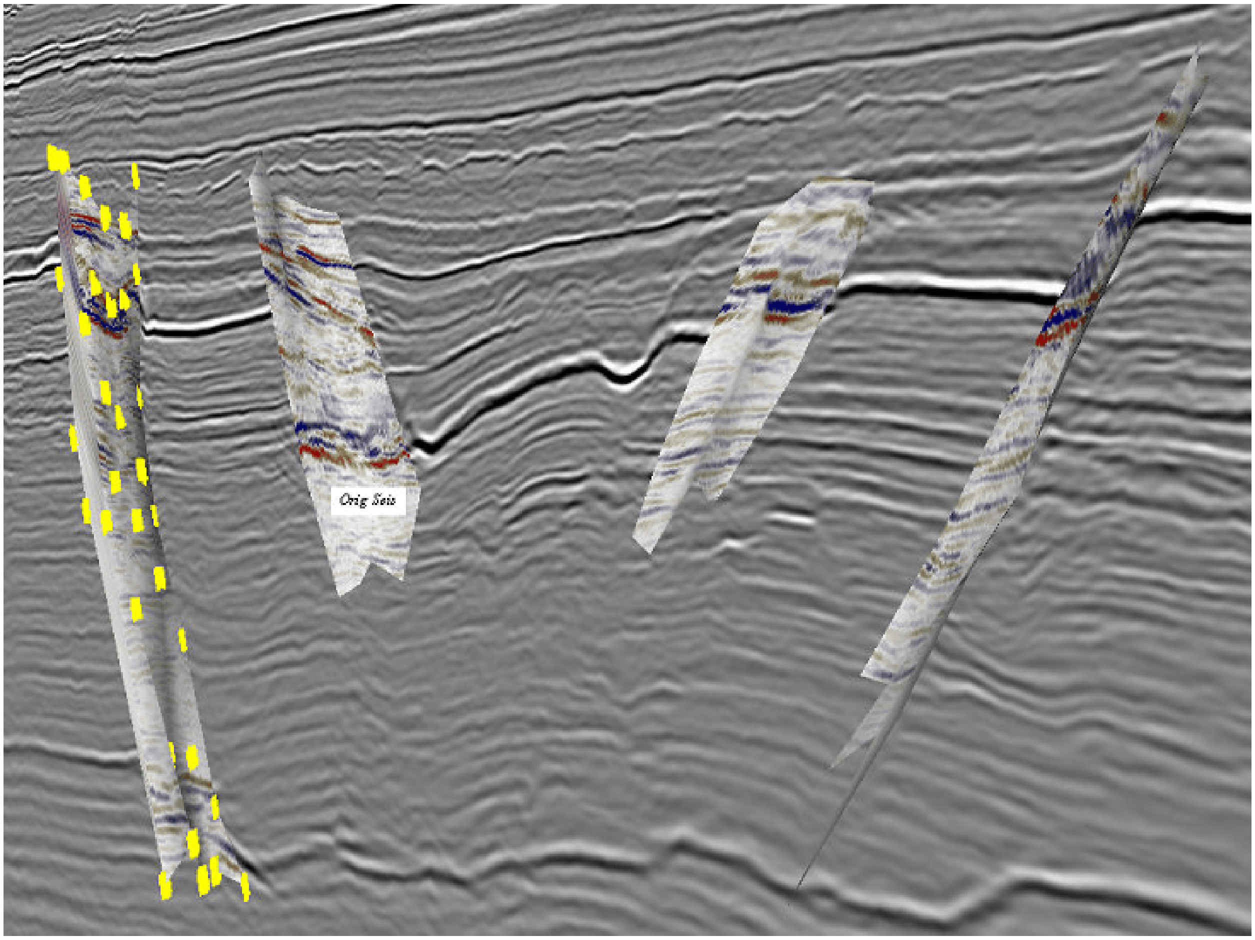


To remove a node, press Ctrl and click on the node to delete. To move the node position, use mouse to select and drag the seed. After editing/interpreting, please do not forget to save the faults.

Fault tree pop-up menu items:

- ***Add Attributes:*** The fault attribute in OpendTect works similar as other attributes. The selected attribute would be displayed on fault plane. It can be the original seismic, calculated attributes, or others. Up to eight (8) attributes can be inserted for one fault. In the tree, right click on the fault attribute and select the corresponding attribute from the list.
- ***Show histogram:*** The histogram of the faults attribute can be displayed using this option. For example, this allows the user to clip the range of a displayed attribute.
- ***Properties:*** The option allows to change the general display properties for faults, i.e. color, reflectivity, and transparency.
- ***Use single color:*** Use this option to display a fault plane in single color.
- ***Display:*** Three display modes for faults are categorized in the tree. To display only the fault planes in full, select the *Fault planes* option. The *Fault sticks* option will display the fault sticks in full. To display the faults only on the displayed intersecting inline/crossline, chose *At sections only*.
- ***Save:*** This option will save the newly inserted/edited fault.
- ***Save as:*** To save the fault with a new name.
- ***Lock:*** It will lock the current fault. The fault cannot then be edited.
- ***Remove:*** This will only remove the loaded fault from the tree. Remember, it will not delete the fault from the project. To **delete** the fault, launch the Fault Management Window by pressing  in the manage toolbar.

The figure below is an example section with interpreted faults. The displayed attribute along faults is the original seismic. Notice that how the events would have been characterized along the fault plane. You can edit and adjust the fault planes by using the yellow nodes, CTRL/SHIFT keys and mouse click as described earlier.

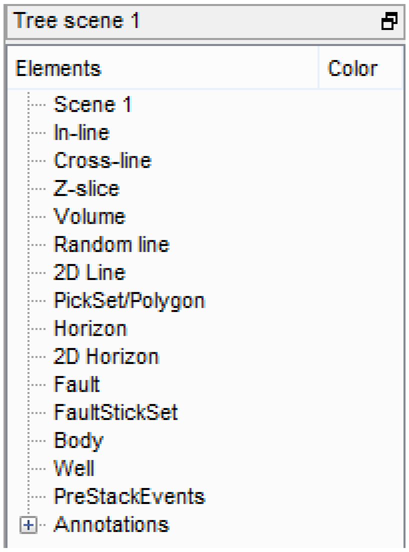


Note: The fault interpretation has only been developed for 3D surveys. For 2D, only fault stick sets are interpreted.

OpendTect Trees and Elements

Table of Contents

Each scene has a corresponding detachable tree to control the elements to be displayed. A tree consists of several elements of different type, which are described in detail below. Tree elements containing elements one level down can be expanded by clicking on the markers (+ or -) left of each main element. The order in which items appear in the tree can be changed by selecting the item that you want to move, then press and hold **shift**, and press the arrow up or down keys on your keyboard.

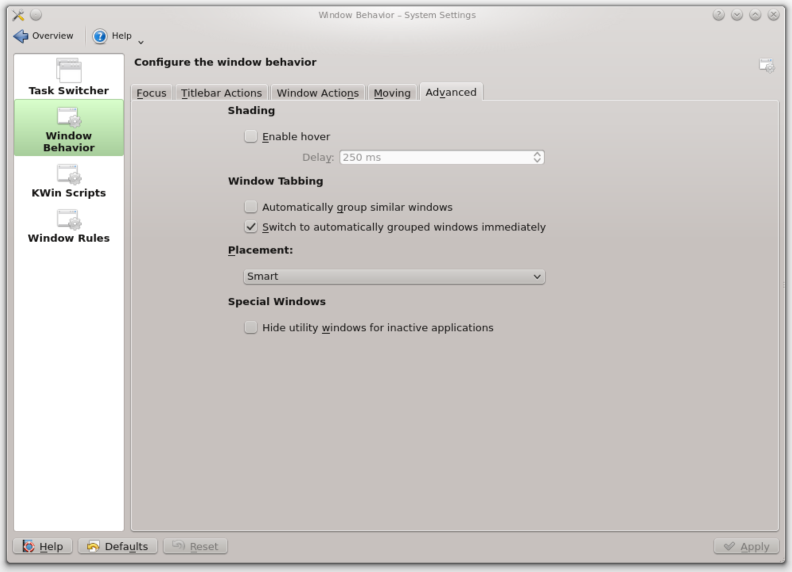


The tree is a utility window that can be moved outside of the main OpendTect window. To do so, first click once on the small squares at the top-right corner of the tree scene, then drag by clicking (left-click) on its title bar to place it anywhere.

Note: Some operating system hide utility windows when the main window is inactive.

A common issue with certain Linux distributions is this: decoupling the tree causes tree to disappear when using, for example, the attribute engine or when the progress bar is showing. To resolve this, please do the following:

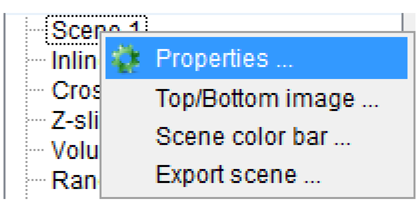
1. Go to the KDE Start button
2. Applications
3. Configure desktop
4. Under 'Workspace Appearance and Behavior' click on 'Window Behavior'. You should see this window:



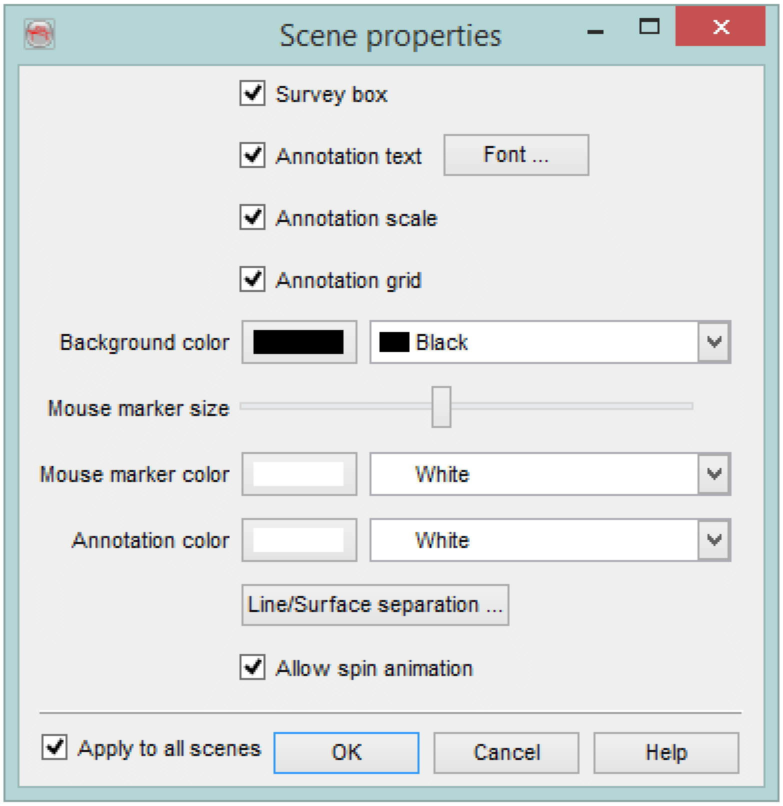
5. Make sure the option 'Hide utility windows for inactive applications' is not checked.
6. Click on Apply button.

Scene

In OpendTect, a *Scene* is a main working window associated with a tree. In the scene, it is possible to work in three separate domains i.e. Time-domain, Depth-domain, Flattened and Wheeler-domain. A time/depth-domain scene can be inserted via Scenes menu. Each scene has its own tree elements so that the elements and scene settings can be modified accordingly.

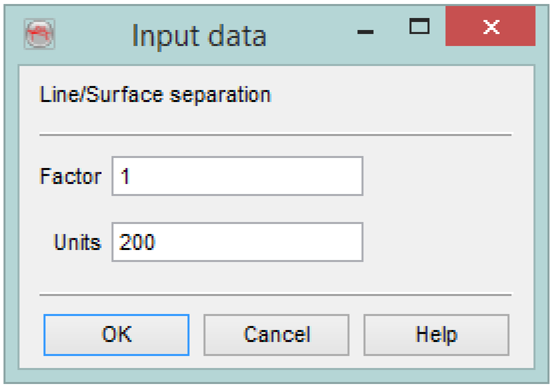


The scene **Properties** can be changed by right-clicking on a scene name in the Tree e.g. *Scene 1 > Properties*. This will pop up the following window:



In this window, the following options can be set for the specific scene:

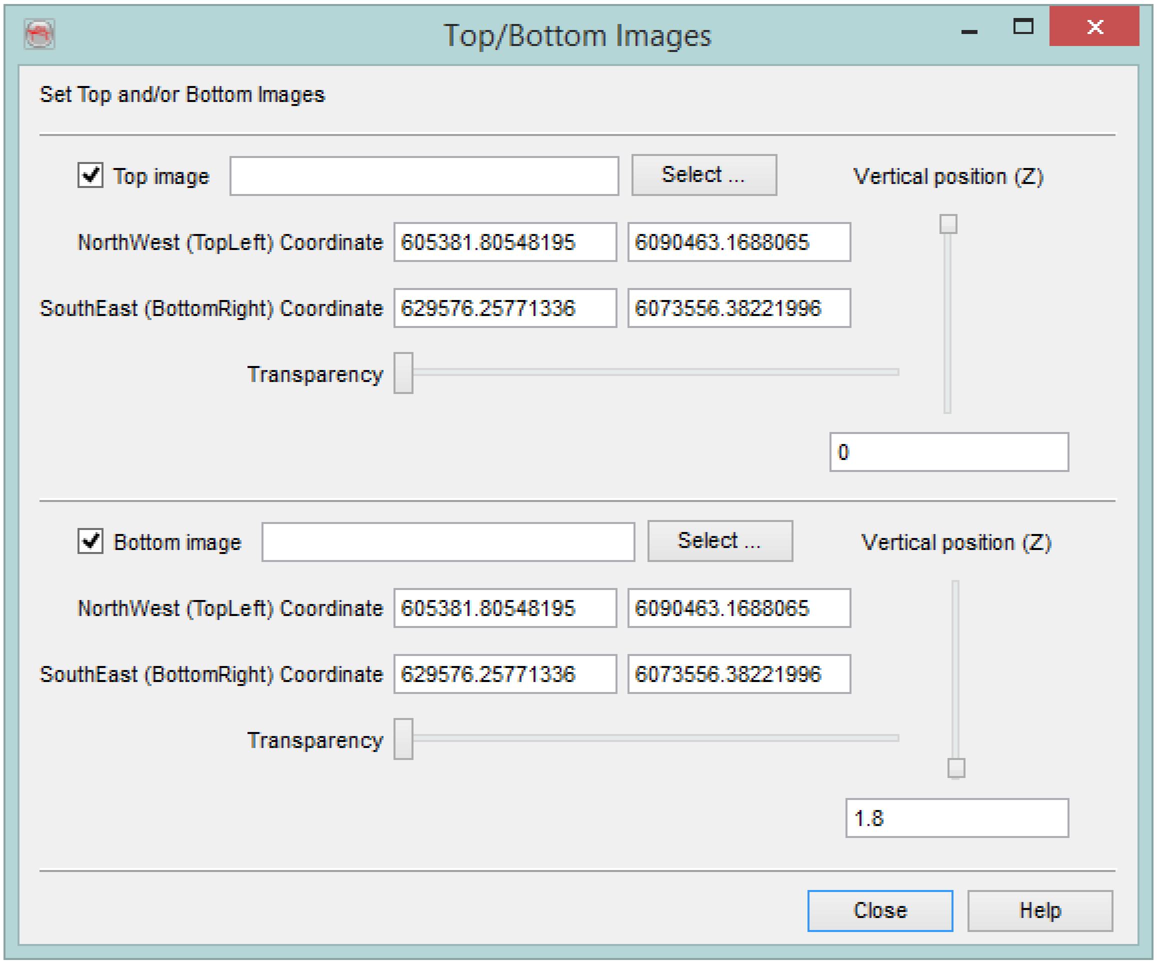
- *Survey box:* If checked, a 3D survey box will be shown in this specific scene.
- *Annotation text:* If checked, the survey box annotations (inline, crossline, TWT) will be displayed in the scene.
- *Annotation scale:* If checked, the numeric values of inlines, crosslines and Z-values will be displayed.
- *Background Color:* The user can specify his/her own background color for particular scene. By default it is black.
- *Mouse Marker Size:* This option is used to increase the Mouse Marker (a marker pointing mouse location on multiwindows) size on various windows in multiscene view.
- *Mouse Marker Color:* The color of Mouse Marker can be changed from here. The default color is white.
- *Annotation color:* The color of Annotation can be changed from here. The default color is white.
- *Line/Surface separation:* The user can change the setting between Line and Surface separation as shown below

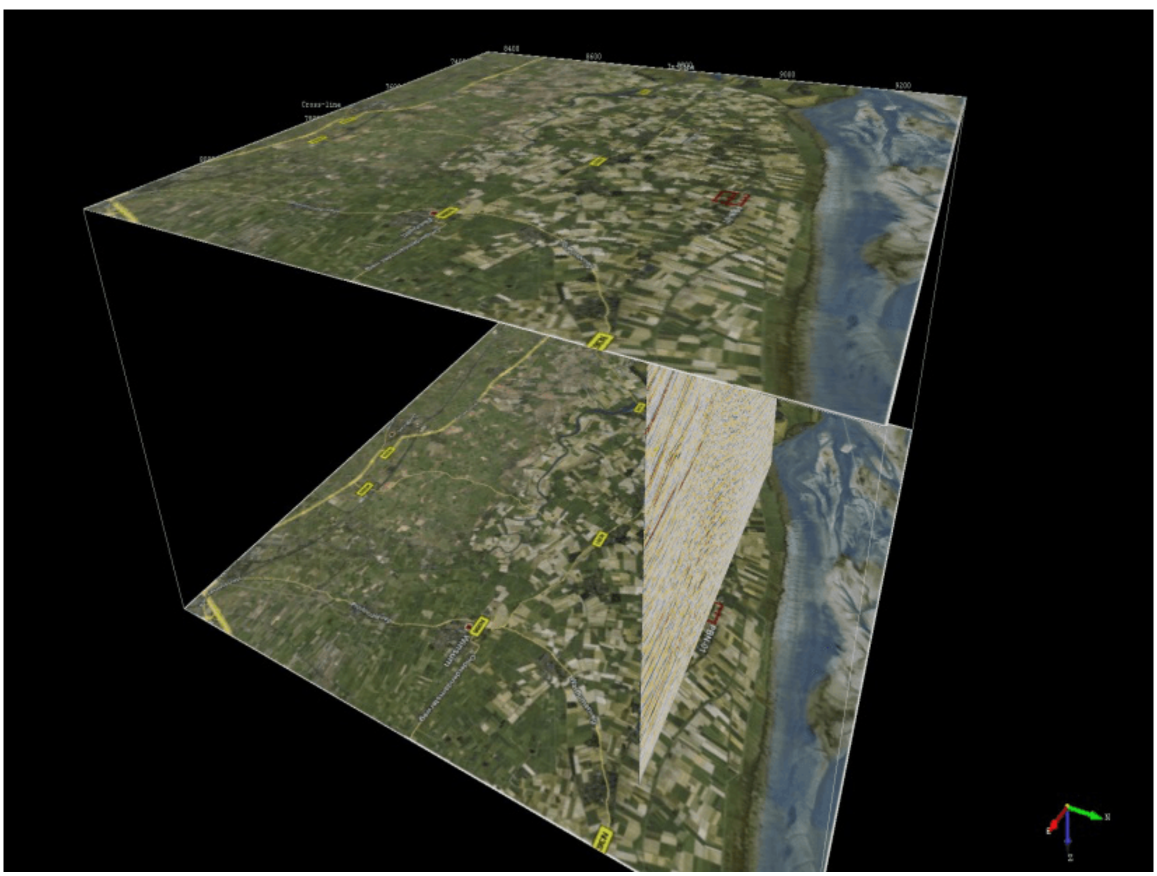


The **Top/Bottom image...**

This option allows to display images at the top and base of the survey area, with or without transparency. The picture will be stretched to fit into the TopLeft and BottomRight corners of the survey (in a north view). The proposed coordinates are the survey ranges.

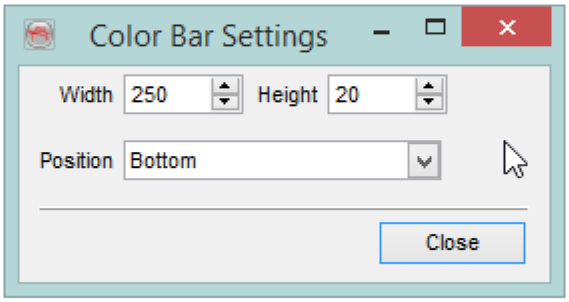
This feature is specially useful to add any reference map of the survey to understand the geographical position and corresponding seismic profile in a better way.


Top/Bottom Images
Set Top and/or Bottom Images
Top image
Select ...
Vertical position (Z)
NorthWest (TopLeft) Coordinate
605381.80548195
6090463.1688065
SouthEast (BottomRight) Coordinate
629576.25771336
6073556.38221996
Transparency
0
Bottom image
Select ...
Vertical position (Z)
NorthWest (TopLeft) Coordinate
605381.80548195
6090463.1688065
SouthEast (BottomRight) Coordinate
629576.25771336
6073556.38221996
Transparency
1.8
Close
Help

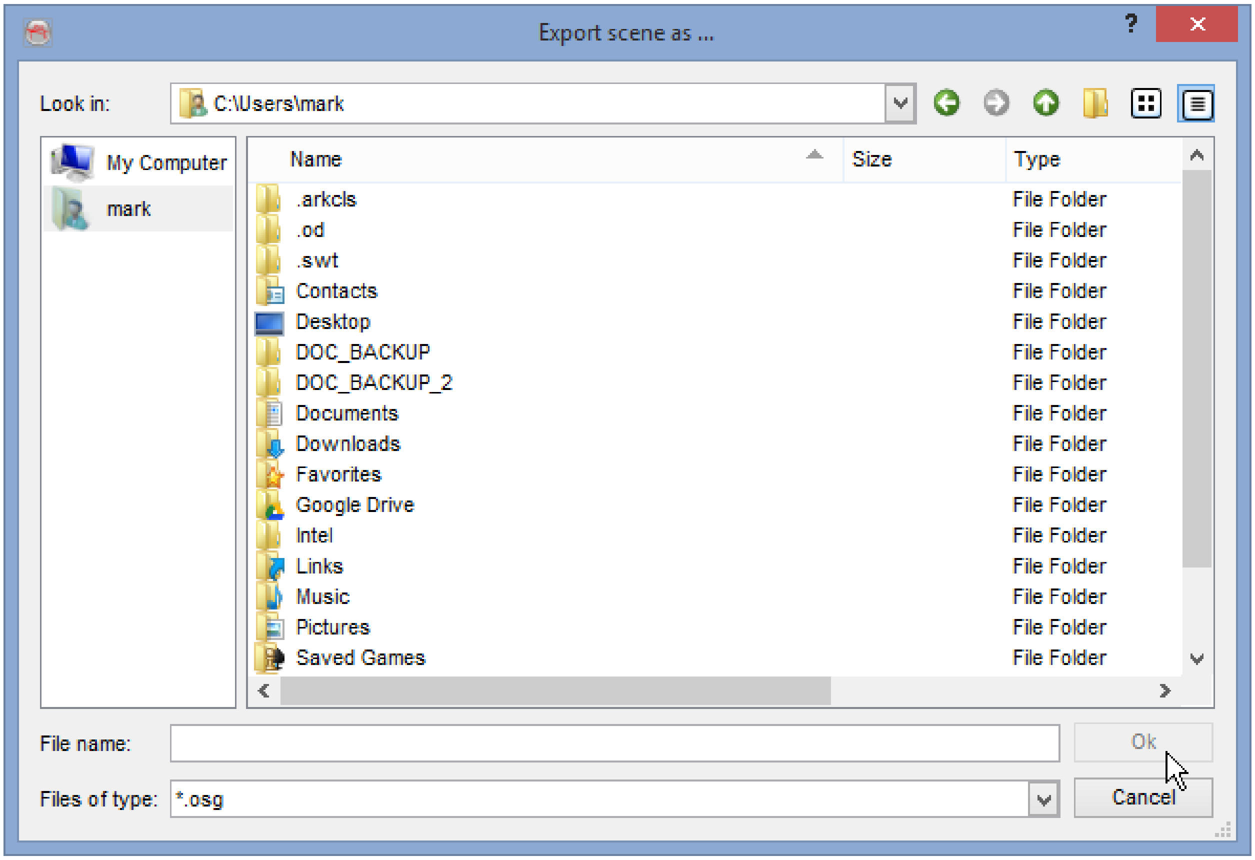


Note: You can use the Google export tool to convert the survey boundaries to latitude and longitudes. They can be then imported into a mapping program (like Google Earth) in order to take the appropriate screenshot.

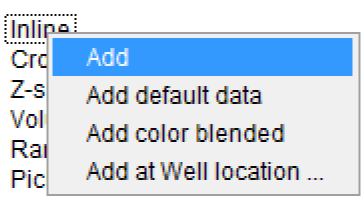
The **Scene Color Bar Settings** can be used to set the position and size of the colorbar in the scene:



The **Export scene** option is only made for the debugging purposes. The export option dumps the scene information into a *.osg* file (OpenSceneGraph), which can be sent to OpendTect and used for bug analysis.



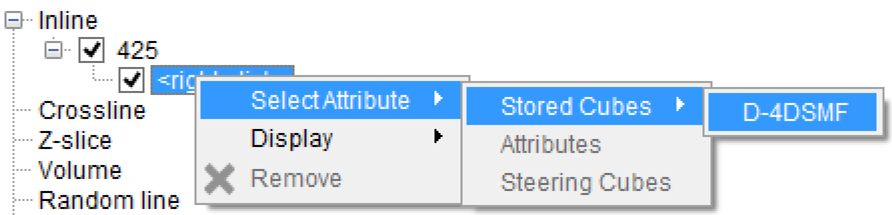
Inline, Crossline & Z-Slice



Inline, *Crossline* and *Z-slice* elements can be added into the tree by clicking on the element name and selecting *Add/Add default data/Add color blended* from the pop-up menu. The Add option will insert an inline/crossline/Z-slice with a blank line in the middle of the survey (ready for an attribute or stored cube to be loaded). For inline and crossline, it is also possible to *Add at Well location*. It will add an inline/crossline going through a selected well (at least one well has to be available in the survey)with a blank attribute. This blank attribute can be displayed in the scene by reading stored seismic data. When the mouse pointer is placed on a *Stored Cubes* sub-menu, a list of stored seismic volumes will be displayed.

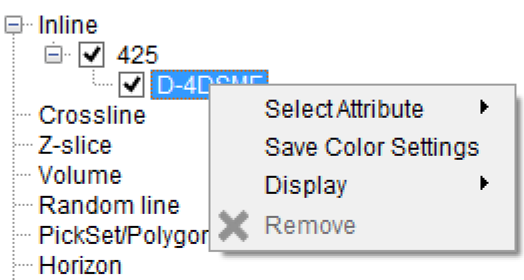
Pre-stack stored cubes will appear in the list surrounded by curly brackets {}. They can be displayed on the slices of the 3D scene as common offset volumes, similarly to the components of (see multi-components volumes). The prefix "O=" is then presented with the offset value in XY units (meter or feet). Attributes may be computed on common offset volumes like for components or multi-component volumes.

Any desired cube can be selected from the list. The selected cube will be added as an attribute for the displayed inline number. The added attribute can also be replaced at any time by right-clicking on it (see the figure below).



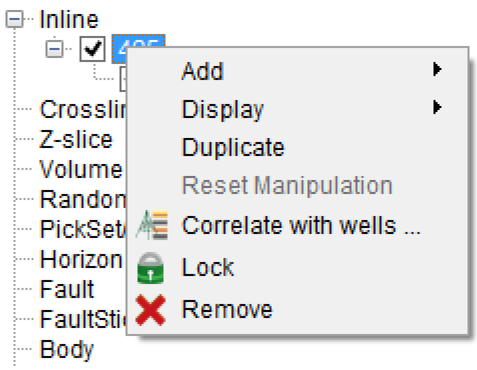
> **Note:** The check-boxes are used to show/hide the corresponding sub-element of inline/crossline/Z-slice.

After an attribute is displayed in the scene, it can be manipulated further by right-clicking over it (see figure below). The explanation for this list is described at the end of this section.



Pop-up List Menus

Each individual element (Inline/Crossline/Z-slice) has a similar pop-up list menu (see below). There are two different ways to open this list. Either display it from the Tree (as shown below) or display it in the scene. In the latter case, place the mouse over the inline/crossline/Z-slice and right-click to launch the tree. This menu list has several functions that are described in following text.

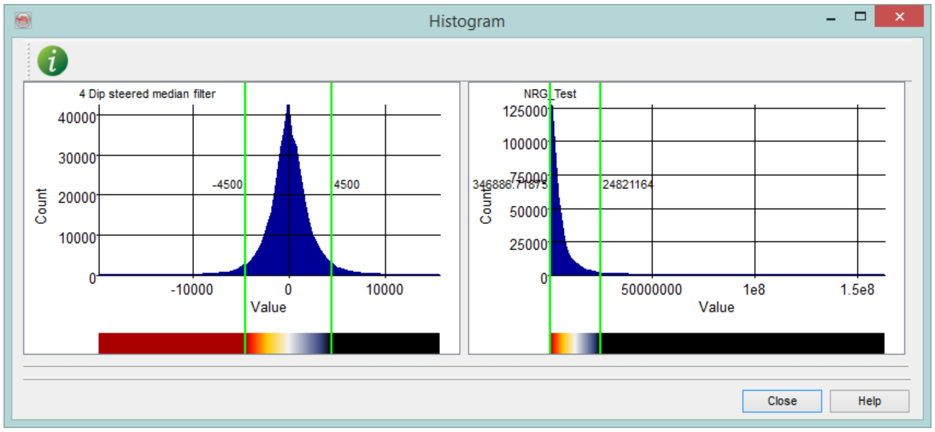


Add:

- *Attribute:* Add an additional blank attribute for the corresponding inline/crossline/Z-slice number. The attribute data (stored cubes or the attribute definition) can be displayed by right-clicking on it and selecting the desired attribute. Up to eight different attributes can be displayed within one element (inline, crossline, Z-slice).
- *Volume Processing attribute:* This option will add the volume processing attribute.
- *HorizonCube Display:* This will display the HorizonCube. It requires the HorizonCube plugin.
- *Systems Tracts Display:* This adds system tracts interpretation. It requires the SSIS plugin.

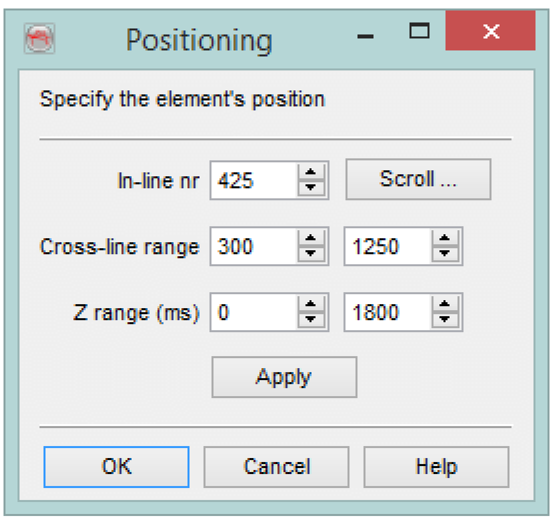
Display:

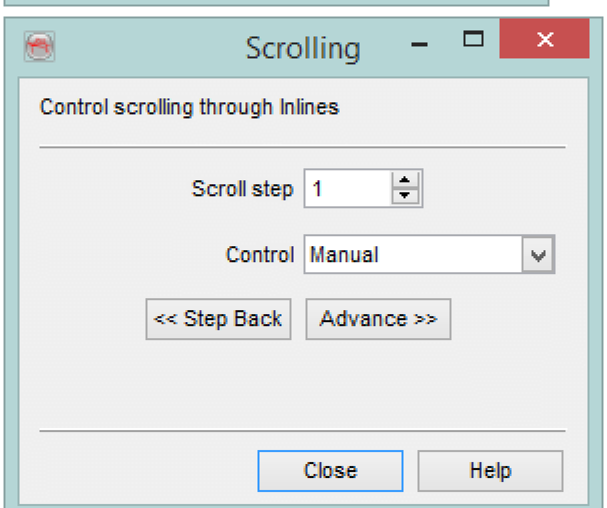
- *Histograms:* The histograms of all added attributes of an element can be displayed using the right-click option of the parent element (inline number, surface name...). It is a useful tool to clip the ranges of an attribute by using vertical green lines in the histogram: The vertical green lines show the current amplitude range and can be moved left or right using the left mouse-click. The display is updated when the mouse click is released. This is performed independently for each attribute. Please note that this will toggle off the automatic clipping.

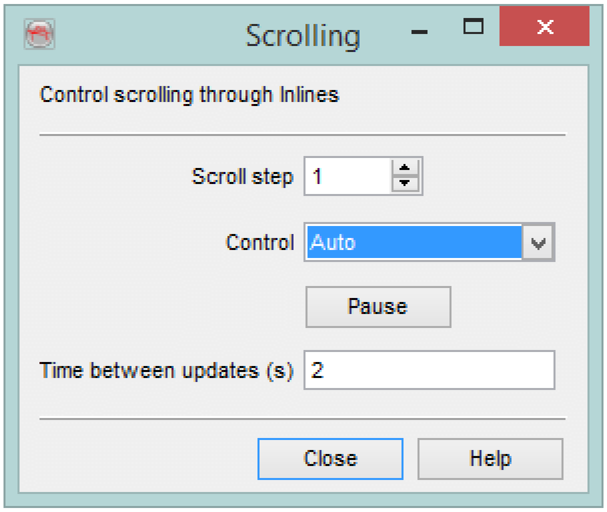


A multi-histogram display for an inline containing two attributes (DSMF Seismic and Energy).

- *Positions:* Change an inline/crossline/Z-slice number. This option is used to manipulate (sub-select a range of traces/time) a line or to quickly scroll through the data for visualization.



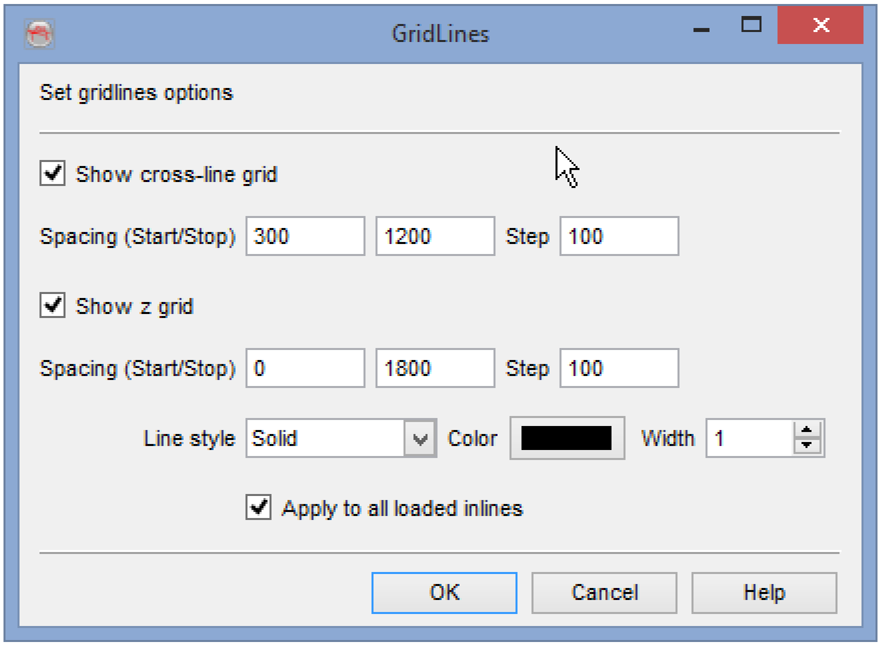




(L to R) Manipulate or scroll the inline; Manual Scroll and Auto Scroll

By pressing the Scroll button, elements are moved either manually (select *Control Manual*), or automatically (select *Control Auto*). Scroll in the inline/crossline direction by specifying a fixed *Scroll step*. In the manual mode, the line/Z-slice is stepped to the new position after each subsequent click on the *Advance* button. In the automatic mode, the line/Z-slice is updated in a movie-style with a fixed time interval (in seconds) - *Time between updates*. The auto-scrolling can be paused by pressing the *Pause* button. To resume the auto-scrolling again press *Go* button.

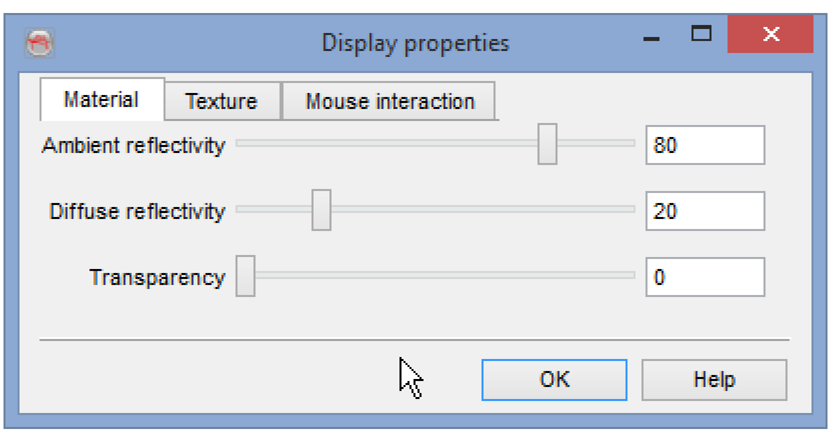
Gridlines: Enables displaying grid lines on the particular element. A new menu appears where the grid line spacing, style, color and width can be set.



Resolution: Edit the graphical resolution of the element. The Default does not involve any rescaling before the data is sent to the graphic card. The options Moderate and High do some pre-interpolation before the data is sent to the graphic card and generally results in a cleaner picture. If the memory of your graphic card does not allow high resolution, the element becomes black.

> **Note:** If Shading is on, the resolution option is not available anymore(except for the horizon element).

Properties: Access display parameters; Transparency, Ambience/diffuse relflection, texture and mouse movement (scroll and pan settings).



Duplicate: Add a new duplicate/copy of a selected element in the tree.

Reset Manipulation: Reset changes made in the position of the line/Z-slice. Restore the original configuration.

Display PS Gather: Pop-up a pre-stack viewer perpendicular to the inline or crossline.

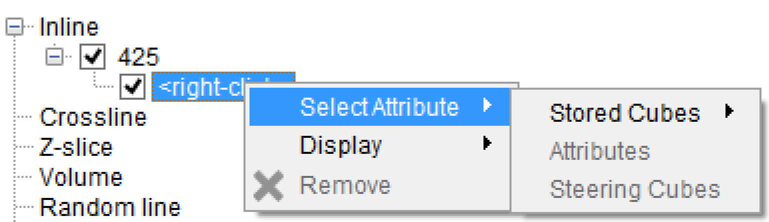
> **Note:** It is also possible to display the offset of each CDP gather similarly to any post-stack data. The pre-stack data is available in the list of stored cubes and is marked with quotes { }, at the end of the list.

Correlate with wells: This will correlate the line with 2D well

Lock: Lock the selected object. Prevents accidental removing, moving or displaying data on the object. After clicking Unlock, all editing is again enabled.

Remove: This removes the element from the OpendTect tree and the graphics area.

The options available for attribute pop-up menu list are briefly described here:



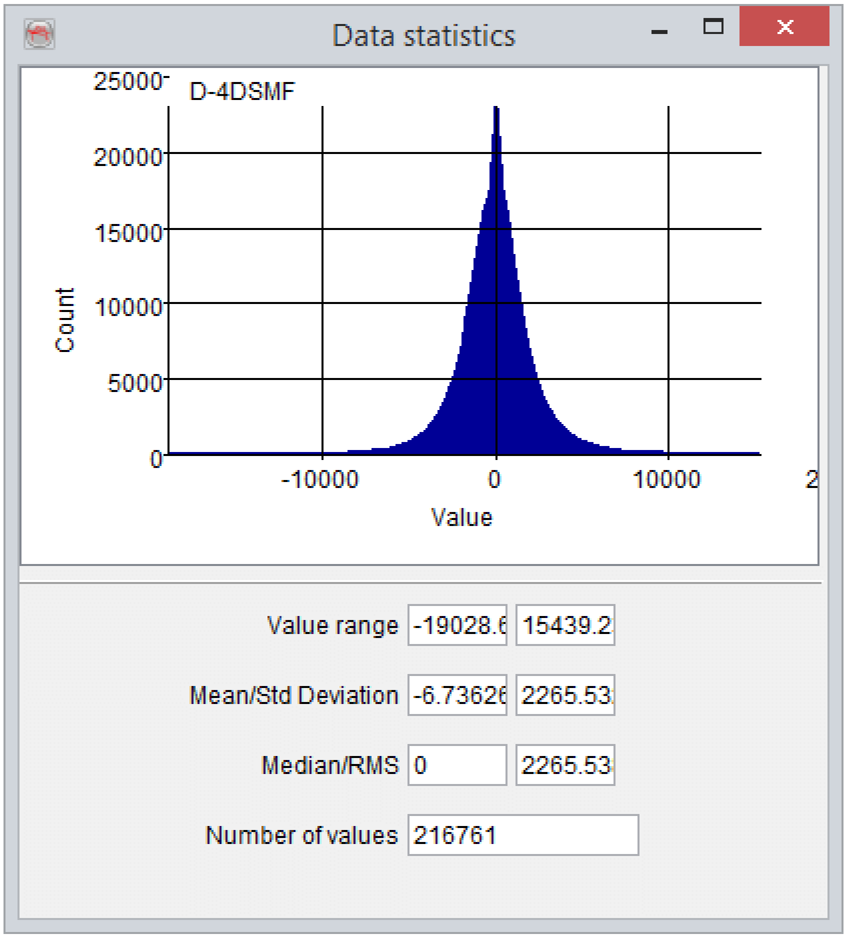
Select Attribute: When selected, data can be displayed from stored cubes or an attribute from the current attribute set (if available). To display an attribute, select or create an attribute set first.

Save Colour Settings: Save color settings for a specific stored volume and make them available for future use.

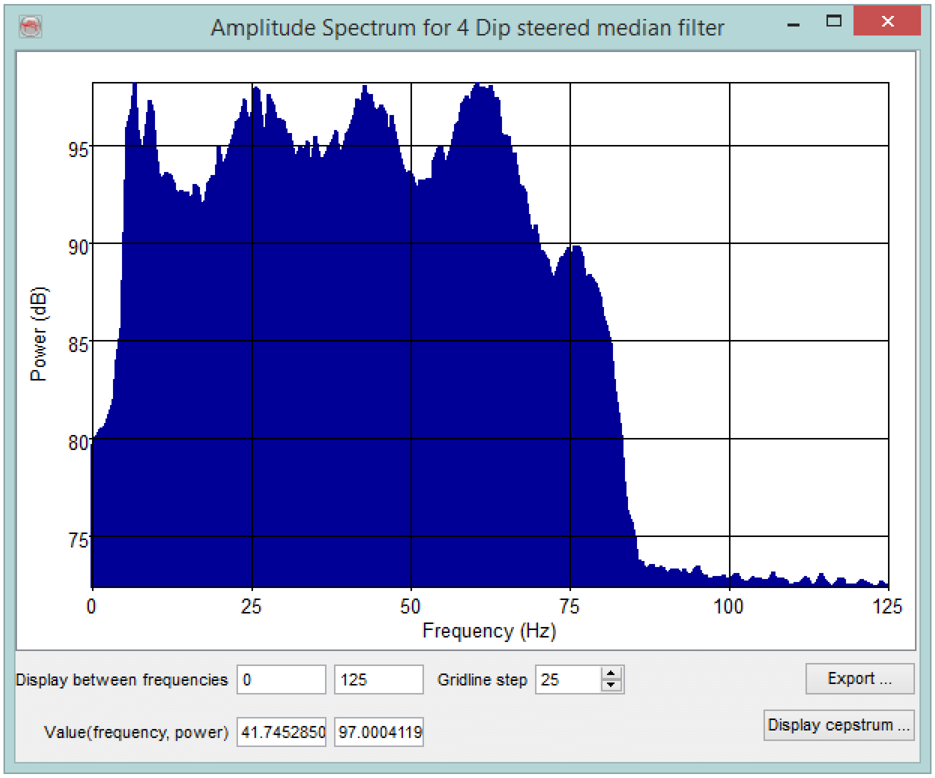
Move: Move the attribute up, down, to top of the list, or to bottom of the list.

Display:

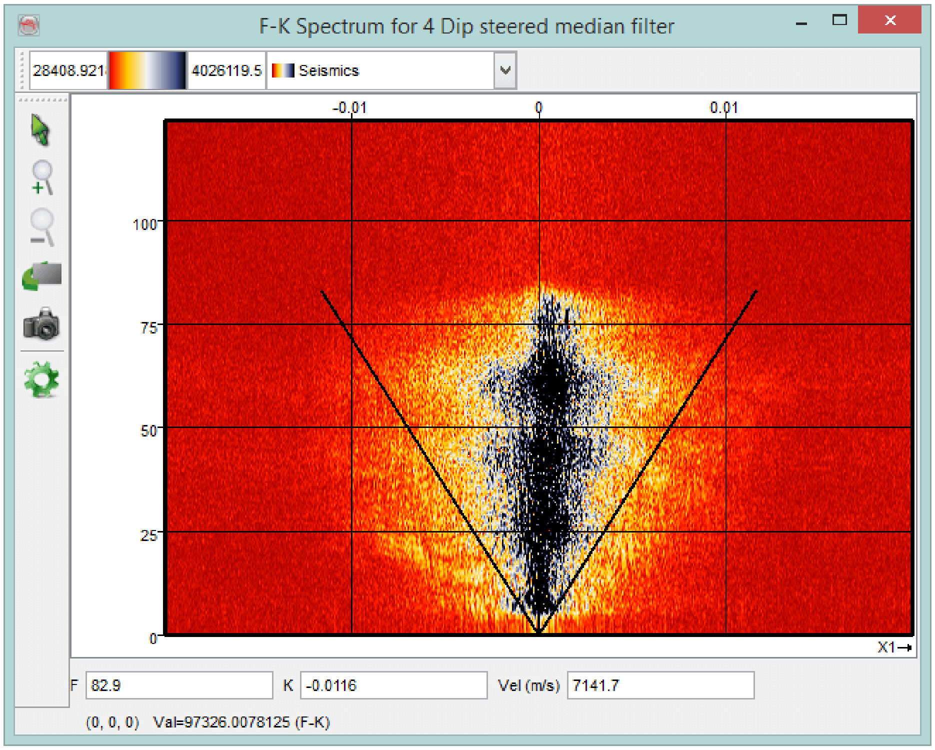
- *Show Histogram:* Display data statistics (selected attribute) of the defined volume as a histogram in a pop up window.



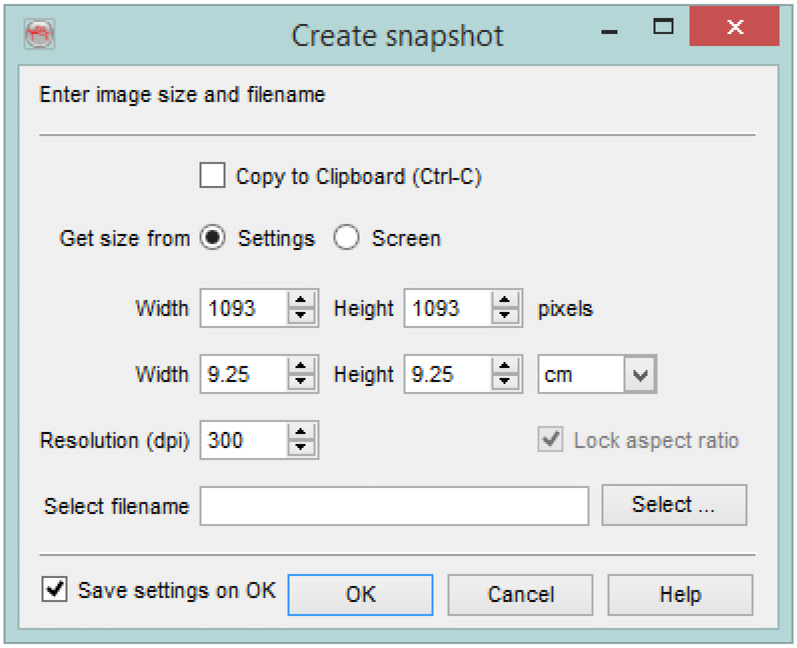
- *Show Amplitude Spectrum:* Amplitude vs frequency plot will be shown in pop up window. Moving the mouse over the spectrum displays the Values.



- Show F-K Spectrum: A two-dimensional Fourier transform over time and space where F is the frequency (Fourier transform over time) and K refers to wave-number (Fourier transform over space).



Pressing *Ctrl+P* in either the Histogram, Amplitude- or F-K Spectrum windows pops up a settings window where you may define parameters for a snapshot:



- *Change transparency:* Change the transparency of the attribute item to view one or more overlaying attributes simultaneously.
- *2D Viewer - VD / Wiggles:* Display an attribute in the 2D viewer as "Wiggle" or "VD" (Variable Density). For more details, please refer to: 2D viewer

Remove: Removes the attribute item from the tree.

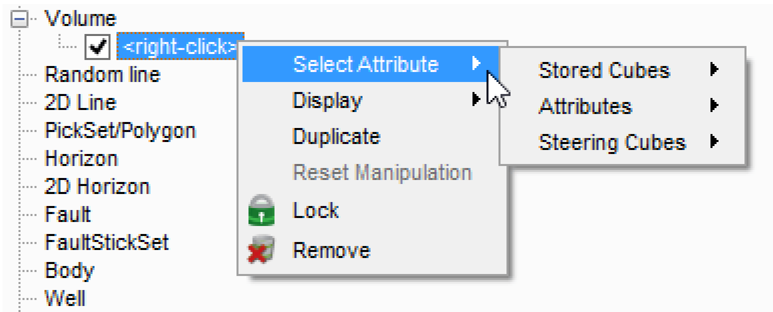
Volume

A volume can be added by clicking on *Volume* element in the tree and selecting *Add* option. A small volume box with blank attribute is added to the scene. An attribute in the newly inserted volume can be displayed by right clicking on the volume and selecting the *Select Attribute* option. This works similar to inline/crossline/Zslice.

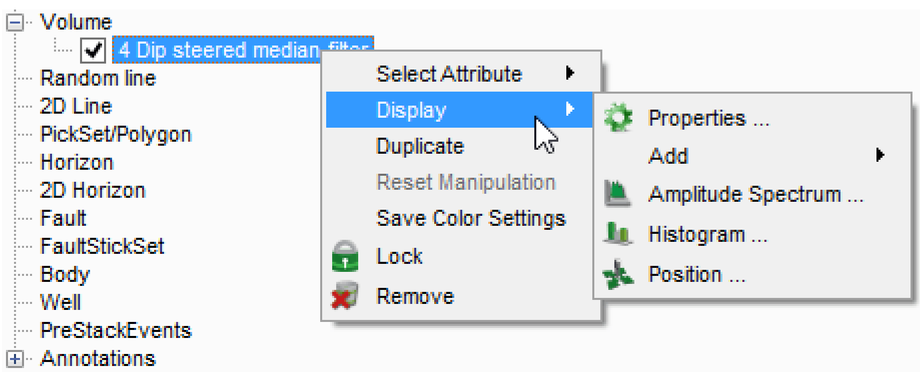
Note: You can display either the stored volumes or calculate the attribute within the sub-volume.

Tip: For faster response times, pre-load the data you wish to visualize using this tree element.

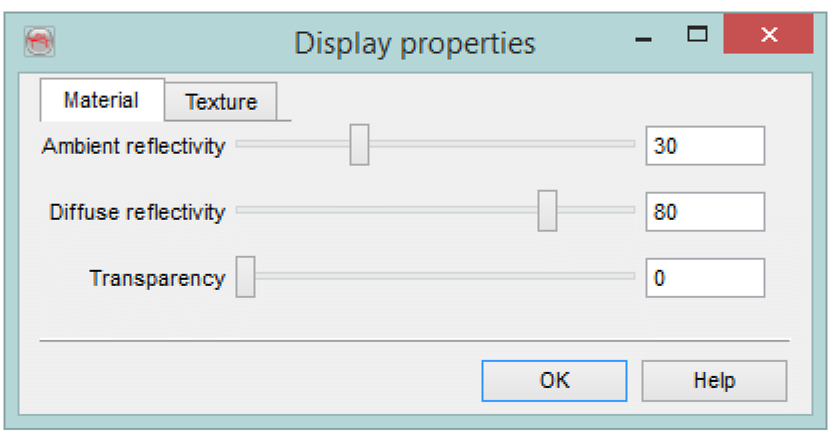
The pop-up menu for the Volume element resembles that described in the previous section for inline/crossline/Zslice:

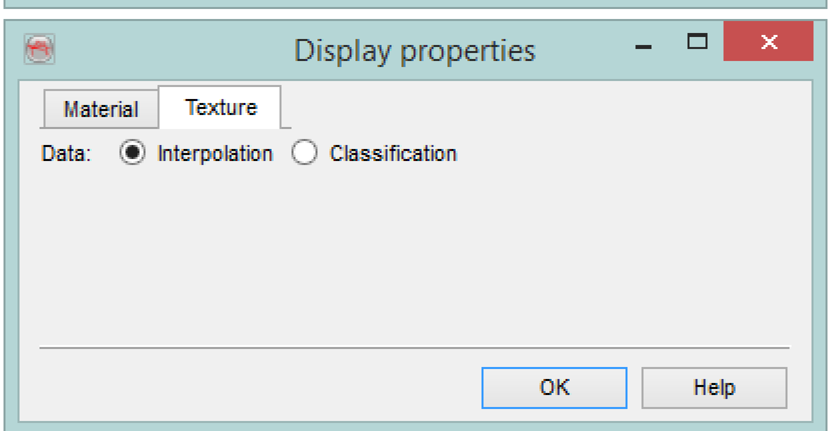


- **Select Attribute:** Select/change the data in a volume.
- **Display:**

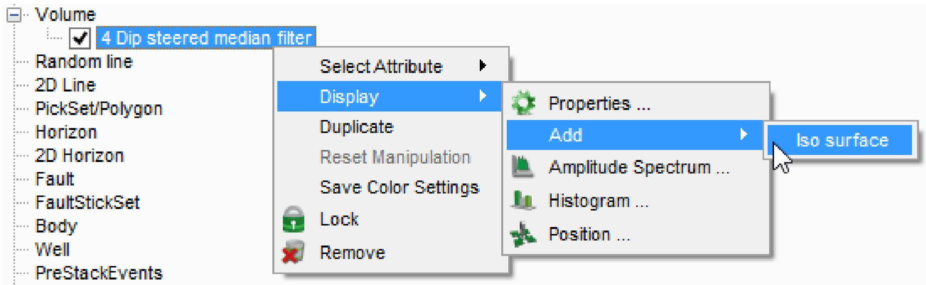


  - *Properties:* Change display parameters such as transparency and ambiance reflectivity:

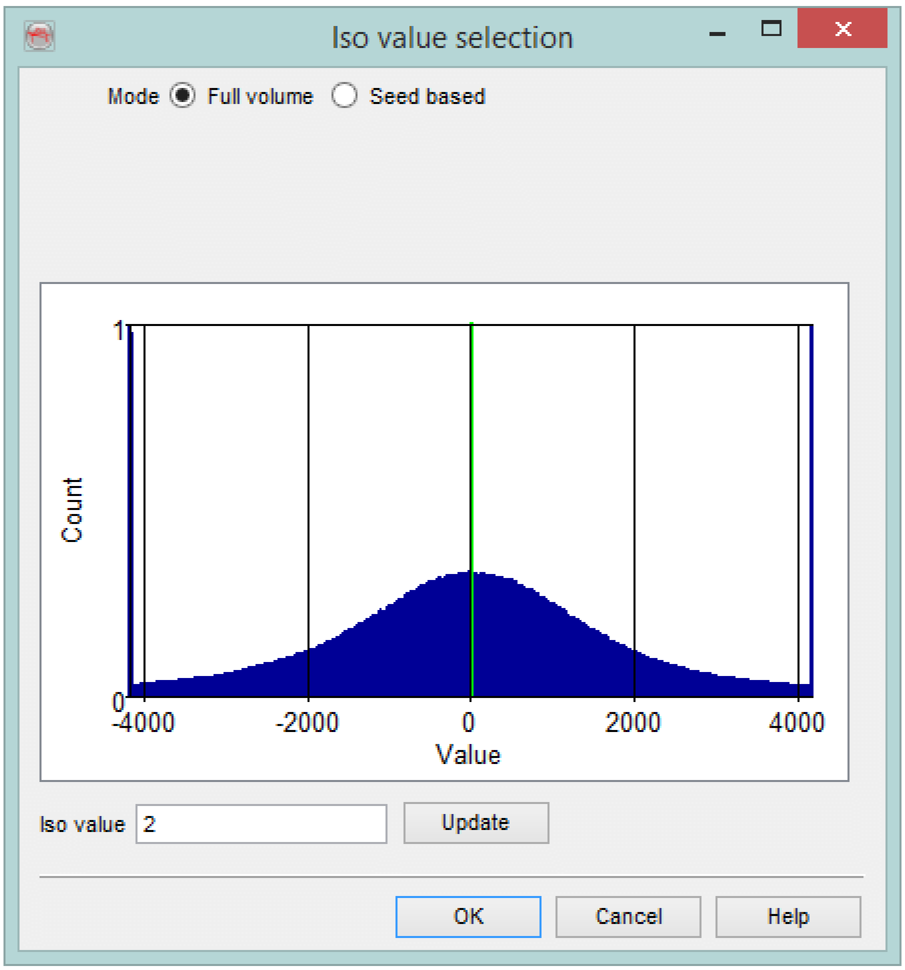


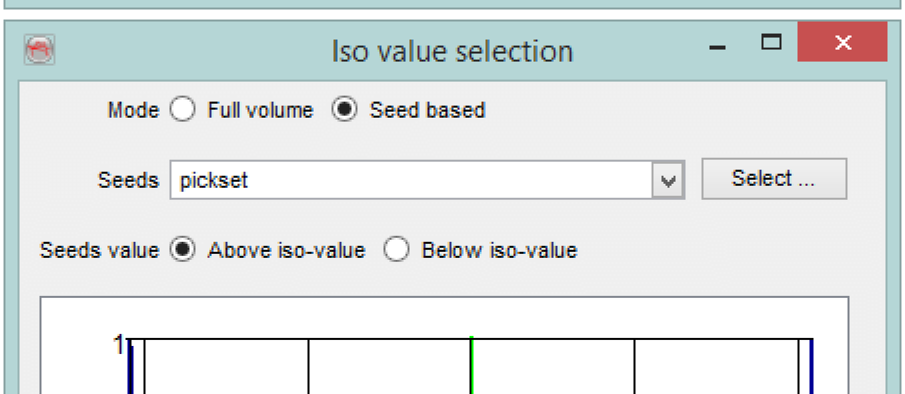


- *Add isosurface:* Compute arbitrary iso value surfaces and convert them into bodies.



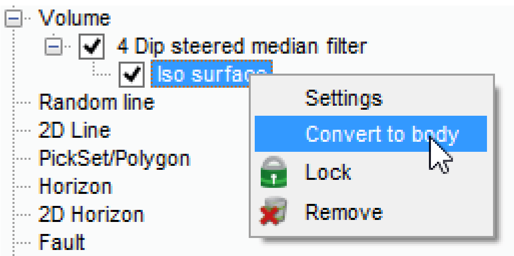
If the option *Add* > *Iso surface* is selected, the following window pops up with choice of modes:





The window displays the histogram of the data collected within the loaded volume (left), or from seeds only that are stored in a pickset (right). "Update" will update the display in the 3D scene (requires some computation time) while leaving the window open. OK will accept the currently selected (or updated) value and dismiss the selection window.

Right-click to display the iso surface menu and convert the iso surface into a stored body that in turn can be retrieved:



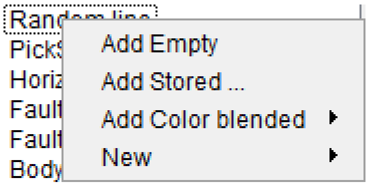
- *Amplitude Spectrum:* Pop-up amplitude vs frequency plot.
- *Histogram:* Display data statistics (selected attribute) of defined volume as histogram in pop up window.
- *Position:* Change the inl/xline/Z position within the survey boundaries.

- **Duplicate:** Create a duplicate/copy of the sub-volume.
- **Reset Manipulation:** Reset the changes made in the position of the sub-volume. This option is only available if the user has made any changes.
- **Save color settings:** Save the color settings.
- **Lock:** "Lock" option with lock the selected object, this will prevent accidental removing, moving or displaying of data on the object. After clicking "unlock" editing is enabled again.
- **Remove:** This will removes the element from the display.

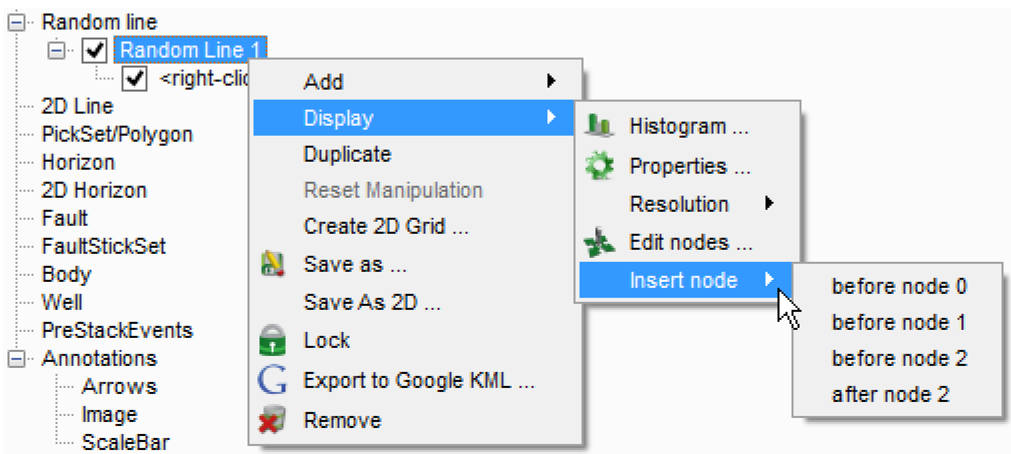
Note: In Interact mode, (see Mouse Controls) the cursor will return the position (inline, cross-line and X,Y,Z) and the data value at that position in the horizontal status bar of the OpendTect window.

Random Line

If you click on the *Random line* in the tree, four options will be available: *Add Empty, Add Stored, Add Color blended* and *New*.

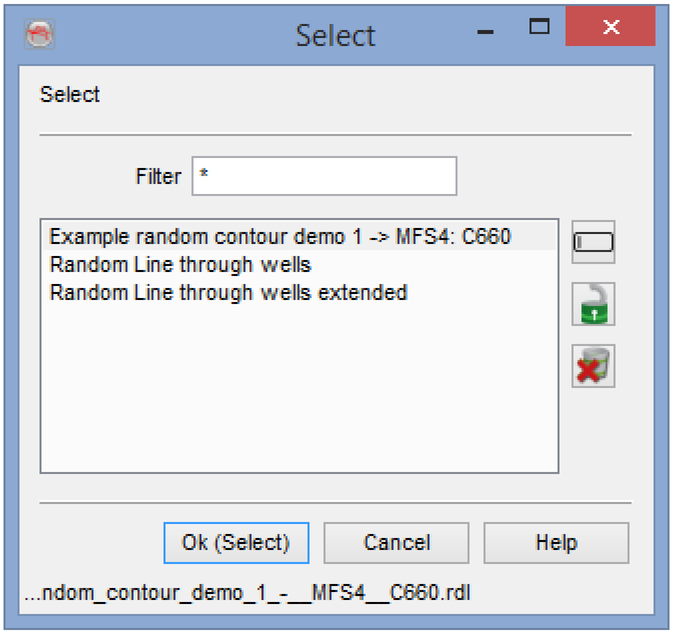


Add Empty: Right-click on Random line and select 'Empty'. The new line will be added as a sub-element of the random line. By default, this is the centre inline of the cube. To create the new arbitrary direction of random line, the user can modify nodes by editing or inserting nodes:

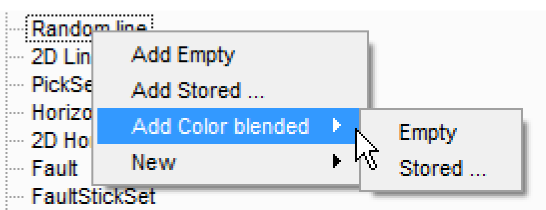


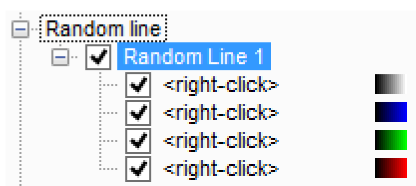
With multiple nodes, the random line can also consist of multiple flat sections. The sections of one single random line may intersect one another. In interact mode, the little plane of a node can be used to drag the node laterally, and the vertical tube can be used to shift the edge of the random line vertically. Nodes can be added from the pop menu by right clicking on the random line in the Interact mode.

Add Stored: If there are previously stored random lines available in the survey, this option should be used to display them. It will launch a random line section dialog that is used to select and display the line in the scene.



Add Color blended: A color blended (RGBA*) Random Line may be added. This may be either a color blended version of a previously-stored random line, or an 'Empty' color blended random line:

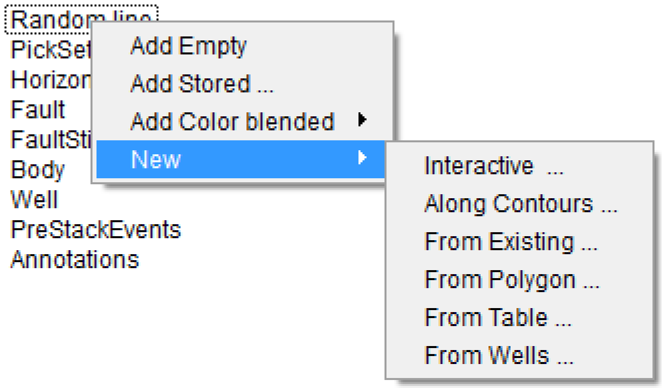




RGBA (color) blended attribute display is used to create a normalized color-blended display that often show features with greater clarity and enhances a detail map view. Traditionally, it is used to blend the iso-frequency responses (Spectral Decomposition), but a user can blend three/four different attributes that define a spectrum that is comparable. For instance, spectral decomposition outputs the amplitude at discrete frequencies. So, it renders the same output (unit=amplitude). Depending upon a geological condition or the objective, FFT short window or CWT (continuous wavelet transform) can be chosen.

Once you have your inputs selected for the appropriate color attributes, it is also possible to add the fourth attribute (the 'Alpha channel') to highlight structural features such as faults/fractures (ie: similarity).

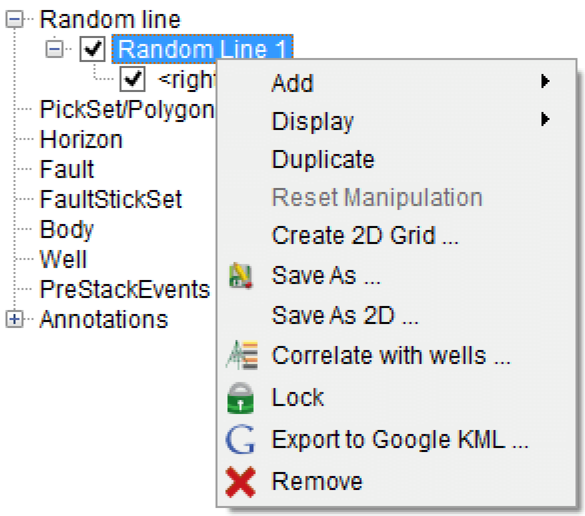
New: There are several ways to create a new random line:



- *Interactive:* When creating a random line from interactive mode, a horizon or Z slice must be loaded in the scene first, a random line can then be created by picking nodes on the displayed horizon/Z-slice.
- *Along Contours:* Create random lines between specified contour ranges. Note that a interpreted horizon grid will be required to provide the contours.
- *From Existing:* Generate random line(s) from existing random line(s). There is an option available to generate random line at some distance away from existing random geometry and store it in new random line geometry.
- *From Polygon:* Create random line from a saved polygon.
- *From Table:* Create random line from table. The input will be X/Y coordinates, Inline/Crossline and Z ranges.
- *Create From Wells:* Connect several wells by a random line. The line follows the deviated well paths (optional). By right clicking on the random line tree, and selecting Create from wells, a dialogue box appears with a list of wells that can be selected in order to set up the random line path.

  Tip: Use in HorizonCube creation, when it is created in a 2D line-section that follows the well paths.

When right-clicking on the newly created random line, the following options are available in a pop-up menu:



- **Add:**
  - *Add attribute:* When selected, choose to display data from stored cubes, from an attribute from the current attribute set or from an output node of the current neural network. To display an attribute or neural network, select or create an attribute set or neural network first.
  - *Add Volume processing attribute:* Display volume created from the volume builder
  - *Add HorizonCube display:* Display the stored HorizonCube
  - *Add System tracts display:* This option will add systems tract interpretation.

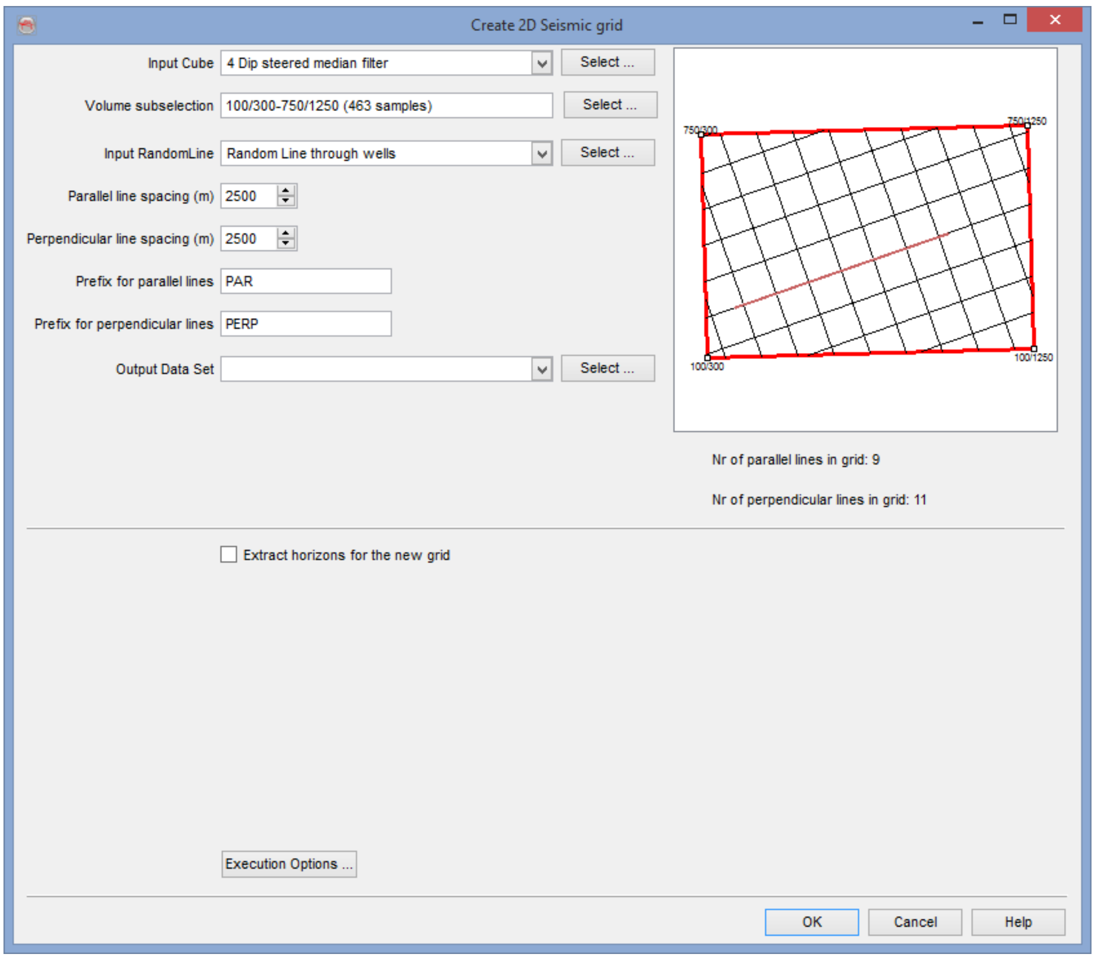
- **Display:**

  - *Histogram:* Displays multiple histograms for the randomline. If there are more than one attributes displayed, it will show the histograms of each in a pop-up view.
  - *Resolution:* Choose the resolution between standard/higher/highest
  - *Position:* It is used to manipulate the nodes / position of a random line. To read more, please go to the Manual mode sub-section of this chapter.
  - *Insert node:* Insert a new node before the selected node.
  - *Properties:* This option refers to display parameters such as *Ambient reflectivity, Diffuse reflectivity, Transparency.*

- **Duplicate:** Duplicate the line as an empty element in the tree. This option displays different attributes on the duplicated line whilst keeping the original data.

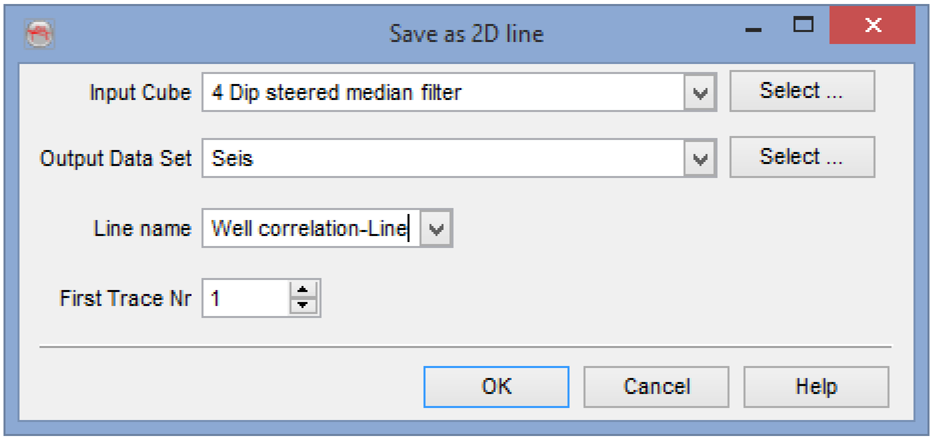
- **Reset Manipulation:** This will reset any change in the position of the random line (or its nodes) that you have applied and it will set line to its original position. This option is only available if changes have been made to the position of the element.

- **Create 2D Grid:** The random lines (with two nodes only) can be used to create a 2D grid with a fixed grid spacing. When selected, the *Create 2D Grid* window is launched (see below). Here, specify the input 3D seismic volume and the output data set name. The output grid is generated according to the dip (parallel) and strike (perpendicular) direction of the selected random line. The prefix labels are used as prefixes to the output line names, stored to the specified new data set name. The grid spacing is the constant spacing between the two lines. At the bottom, the total number of parallel and perpendicular lines will be updated according to the grid spacing. By pressing *OK*, a batch process will start to generate the 2D grid. When the batch program is finished, the data can be displayed in the scene (see 2D Seismic section for details).



- **Save As:** Save the random line as a new name or overwrite the existing.

- **Save As 2D:** Creates a 2D line from a Random line. Right-click on the random line in the tree and select *Save As 2D*. A window will pop up, as shown below. Select the *Input cube*, the *output line* and the *line name*. The *first trace nr* number of line is also necessary.



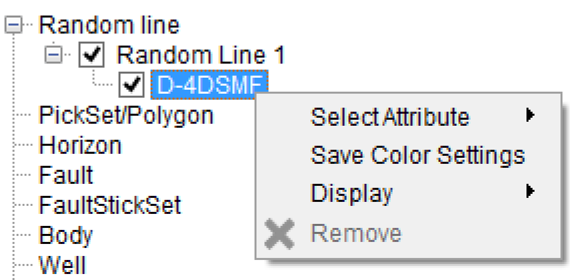
Note: The survey type should be 2D as well if you want to view the 2D line created from a random line.

- **Correlate with wells:** This option is used to correlate a random line with wells. Well - seismic correlation is normally done in the Well Correlation Plugin (WCP), which requires a commercial license.
- **Lock:** Locks the selected object. This will prevent accidental removing, moving or displaying data on the object. After clicking *lock* again, editing is again enabled.
- **Export to Google KLM:** Export selected random line to a Google KML file. Specify the KML file parameters in the pop-up dialog.

Note: annotate the start and end of the random line with a user defined *line annotation* in the output file settings.

- **Remove:** Remove the random line from the tree and the scene.

The options available for attribute pop-up menu list are briefly described here:



- **Select Attribute:** When selected, data can be displayed from stored cubes or an attribute from the current attribute set (if available). To display an attribute, select or create an attribute set first.

- **Save Colour Settings:** Save color settings for a specific stored volume and make them available for later use.

- **Move:** Move the attribute up, down, to top of the list, or to bottom of the list.

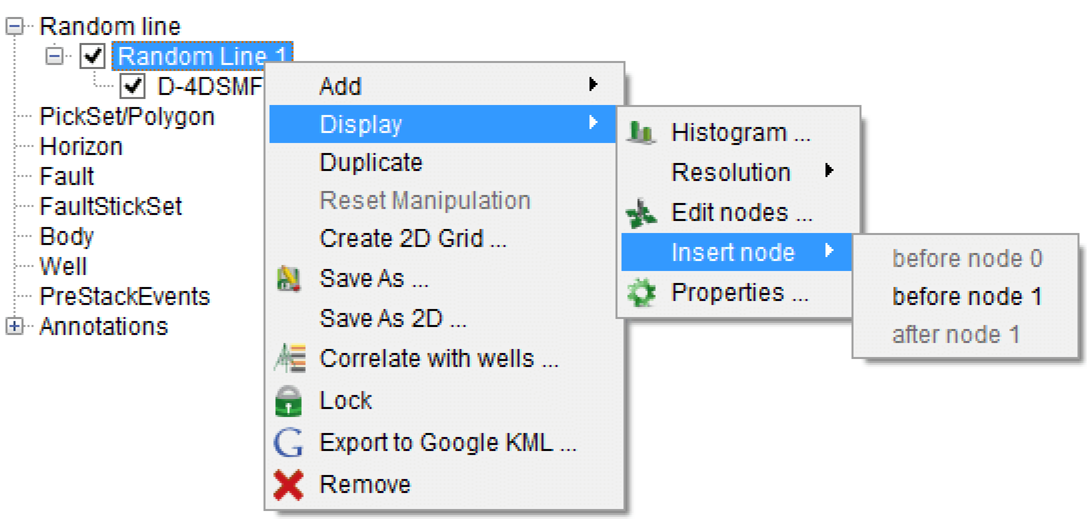
- **Display:** There are several display settings / features that are briefly explained below:

- *Show Histogram:* Display data statistics (selected attribute) of the randomline as a histogram in a pop up window.
- *Show Amplitude Spectrum:* Amplitude vs frequency plot will be shown in pop up window.
- *Change transparency:* Change the transparency of the attribute item to view one or more overlaying attributes simultaneously.
- *2D Viewer - VD / Wiggles:* Display the selected attribute in the 2D viewer as "Wiggle" or "VD" (Variable Density). For more details, please refer to: 2D viewer

- **Remove:** Removes the attribute item from the tree.

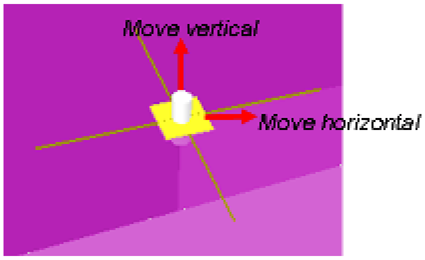
Manual Mode (Empty)

Manual Mode. In manual mode, the random line will first be displayed in the 3D scene. Nodes may be added and their position changed interactively, in a second step. This starting random line will have two nodes, one at each end of the central inline. More nodes can also be inserted in the right click menu of the random line in the tree (see figure below). Please note that the same menu is available with the right-click on the random line in the scene.



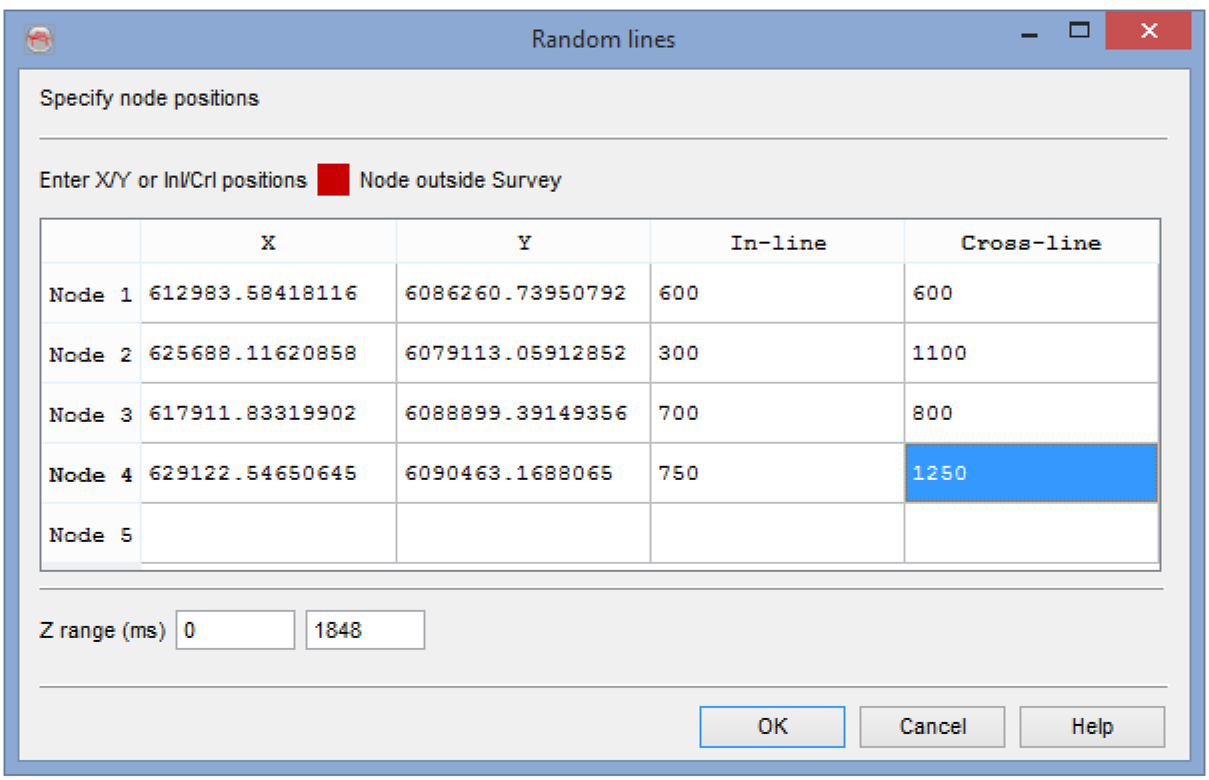
Adding/inserting new nodes between the existing nodes.

The node on the left-hand side of the newly created random line is designated node 0, and the one in the right hand side node 1. It is possible to insert a node before node 0, before node 1, and after node 1. The node will be created half-way between the two surrounding nodes. In order to move a node to a desired position, click on the random line to make the nodes visible/editable. In the interact mode, click at the node plane (horizontal/vertical) to move the node location. A purple surface appears around the node and the node can be moved in any direction inside the survey area.



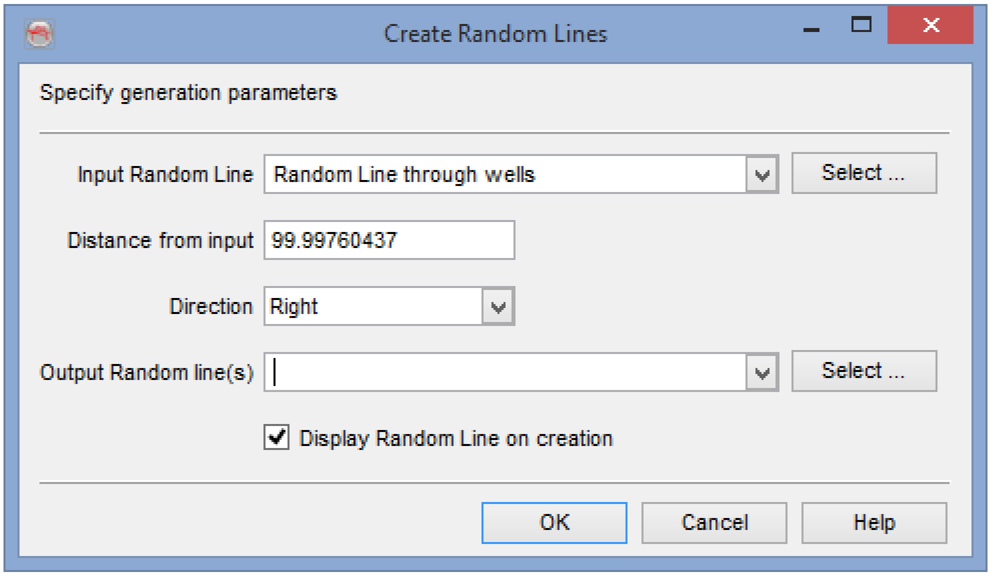
*The node can be moved in two directions (horizontal and vertical). The node's orientation can be changed by placing the mouse pointer over the node and pressing the **Ctrl** key.*

Editing or modifying the position of the nodes is also possible through clicking the option *Edit nodes...*. The following windows will pop-up and the nodes are editable. Modifying or inserting new nodes is also enabled. In this table, each node is defined by its inline/cross-line or X/Y position. The nodes can also be removed by right clicking over the desired cell and selecting the 'remove node' option. Similarly, for the pop-up menu, more nodes can be inserted before/after the selected cell (node).



Create from Existing

This option allows the user to generate random line offset from an existing random line. There is an option available to generate a random line at some distance away from existing random geometry and store it in new random line geometry.



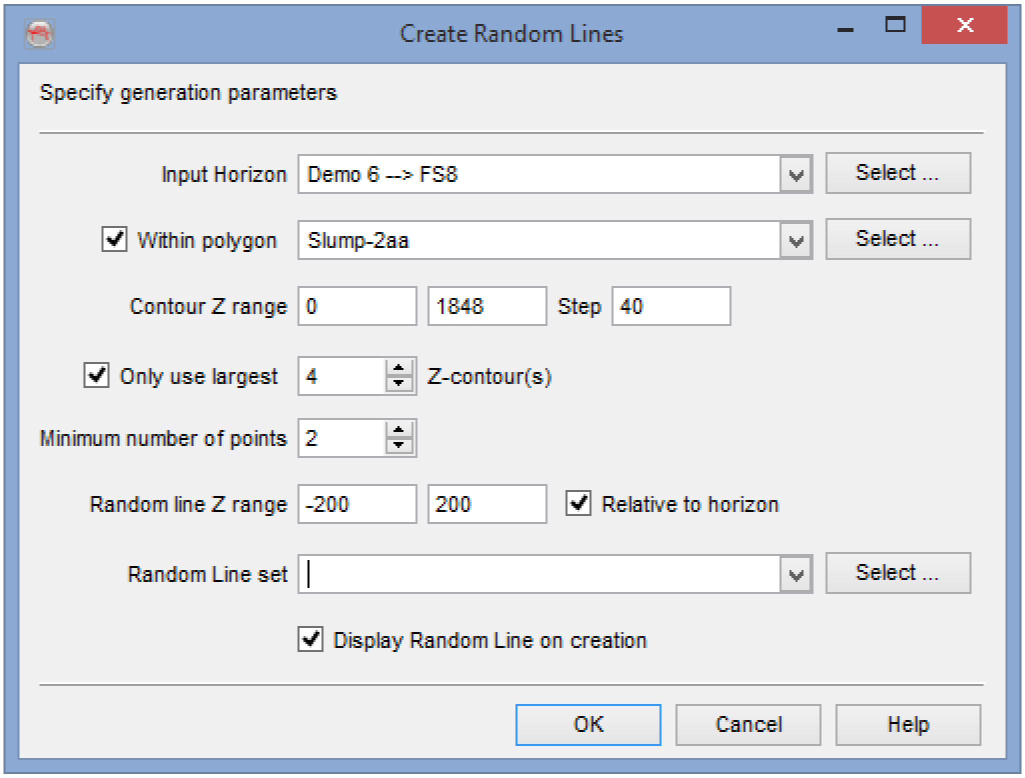
Create Random line from existing line geometry in left/right or both directions. The direction is defined by the path described by the nodes, in the order seen in the table.

The first generation parameter is the *input random line*, which has to be chosen between the already existing random lines. Then, define the *distance from input* in meters and the direction in which the node will be added. There are three directions: left, right, and both. The final step is to name the output random line.

Click on the *Display Random line on creation* box to immediately visualize the random line.

Create Along Contour

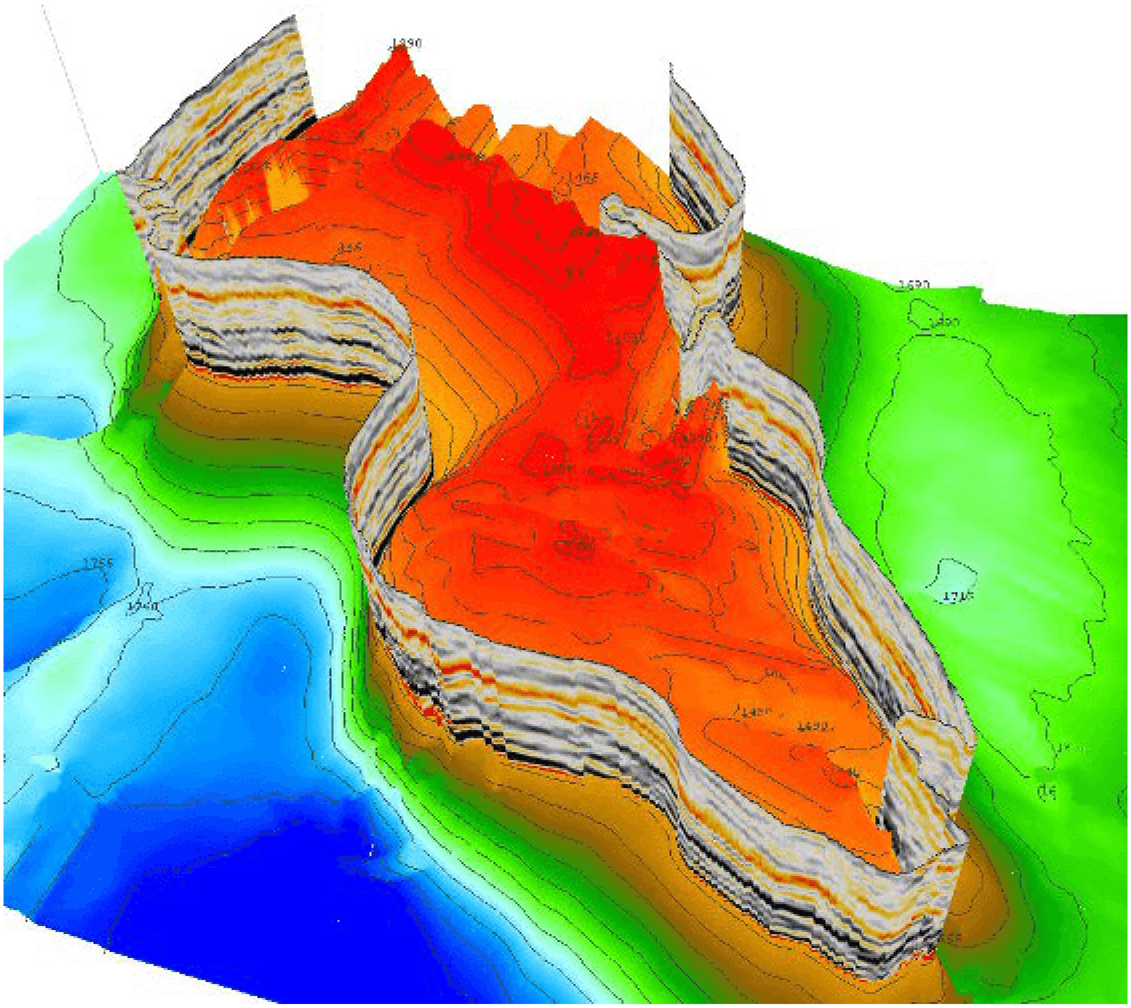
Random lines can be create along the contour geometry. For this, an interpreted horizon grid will be required. The horizon grid is use to generate the random lines at a selected index contours range. All contour values that are present within the given Z-range, are used to construct the random lines. When the process is completed, you can edit the random lines also. This functionality is important to see the seismic section along the contours (e.g. AVO responses, DHIs or hydrocarbon contacts).



First, specify the *input horizon*, and optionally select if it is *within a polygon*. Then define the *contour Z range* and the *step*. Then the *Random line Z range*, there is an option to chose whether the random line Z range will be made *relative to the horizon* or not. Last step is to give a name to the newly created *Random Line Set*. If you want to display the random line on creation, check the box *Display Random Line on creation*.

The option *Display Random Line on creation* can be selected and the new random line will be displayed after clicking OK.

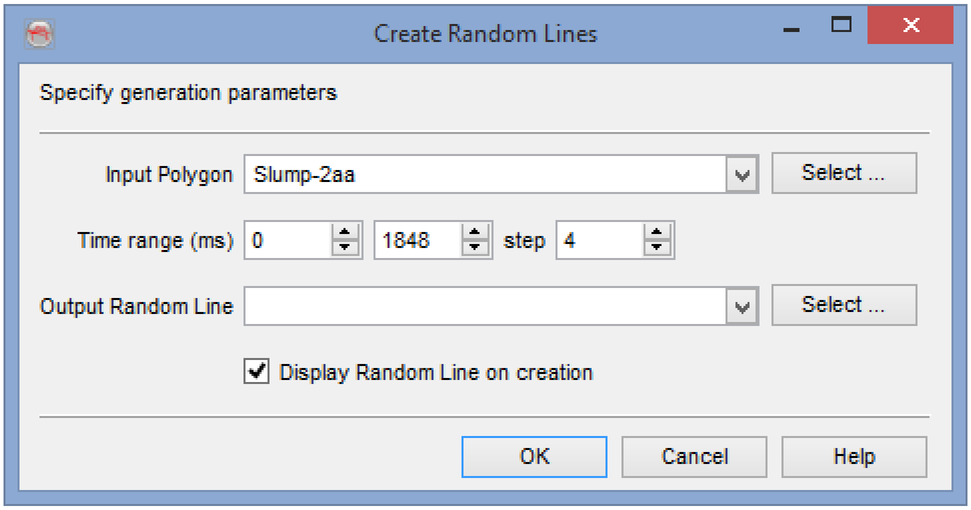
The following picture shows an example of result for a random line created along contour, it is composed of a set of random lines that satisfy the contour z range and therefore is composed of more than one element.

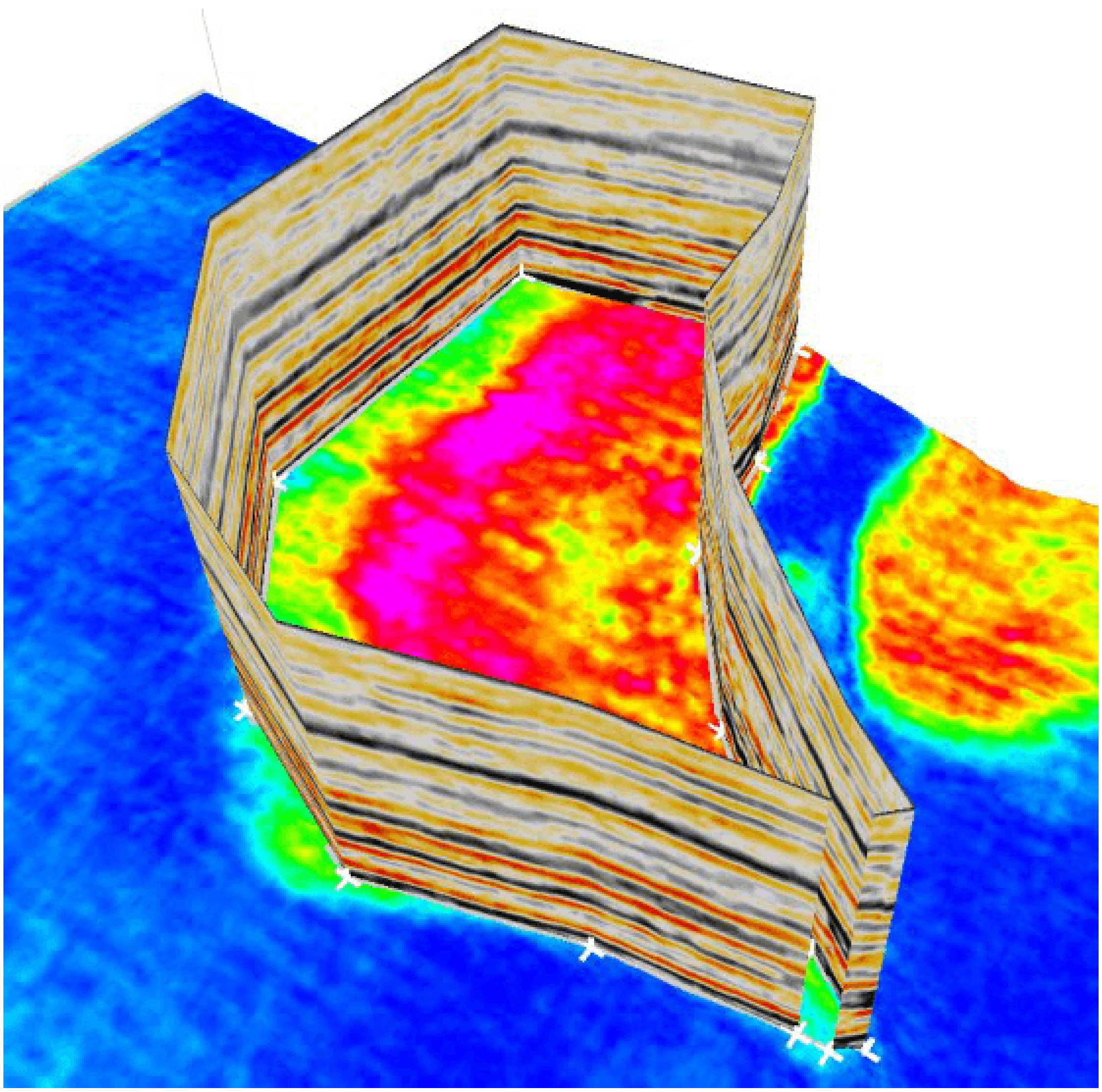


An example random line generated along a fixed contour line.

Create from Polygons

This option allows the user to create random line definition from already created polygon. In the parameters, select the existing polygon and sub-select the Z-range for the random line, which will be generated. Write an output name for this random line and optionally, set check to display random line on creation so that after creation it will be displayed in the scene/tree. Press OK to proceed.


Create Random Lines
Specify generation parameters
Input Polygon
Slump-2aa
Select ...
Time range (ms)
0
1848
step
4
Output Random Line
Select ...
Display Random Line on creation
OK
Cancel
Help



An example random-line generated along the white colored polygon. The polygon approximates the closure of a gas anomaly.

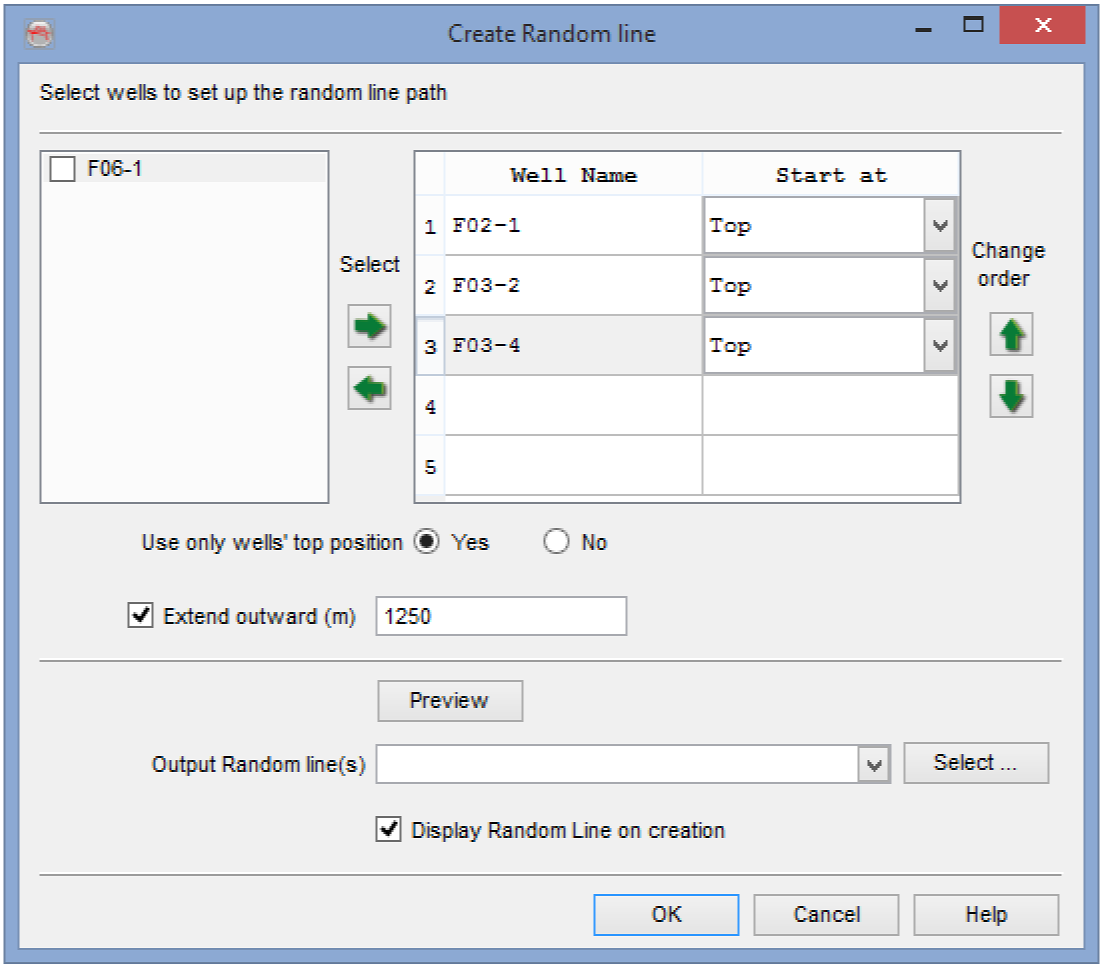
Create From Wells

A random line can be created in such a way that it follows wells path. By right-clicking on *Random line* in the tree, and selecting *Generate > From Wells ...,* a dialog box appears with a list of wells that can be selected in order to set up the random line path.

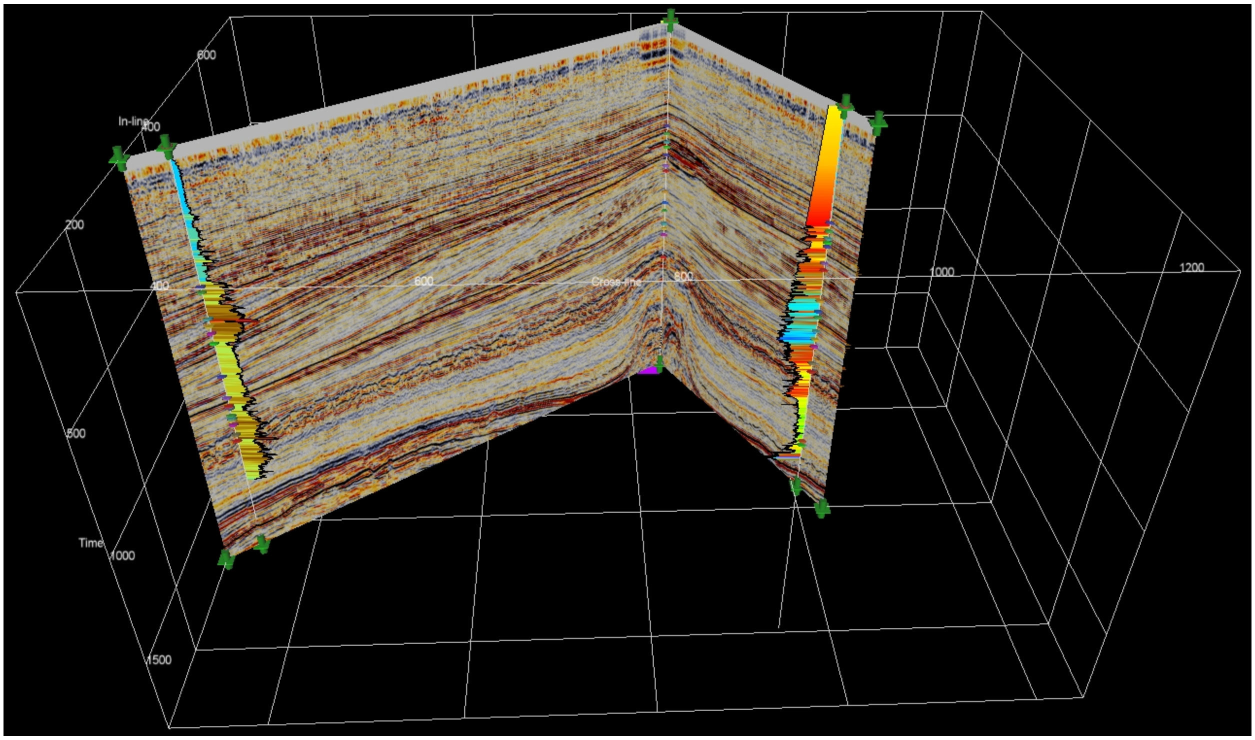
Use the arrows to add and/or remove wells. Use the second set of arrows to setup a well sequence. Specify whether you want to use only the well top position or not. When you use all well points, you can specify the order by clicking the *Change Order* arrows.

The *Extend outward* allows the extension of the random lines in both sides away from wells.

Press the preview button to see a top view of the random line that will be created. If the preview does not show exactly the desired random line, then change the parameters (the wells involved or the order in which they are listed). You can save the newly created random line by specifying the name in Output Random line(s) field. If you want to display the random line on creation, check the box *Display Random Line on creation*.



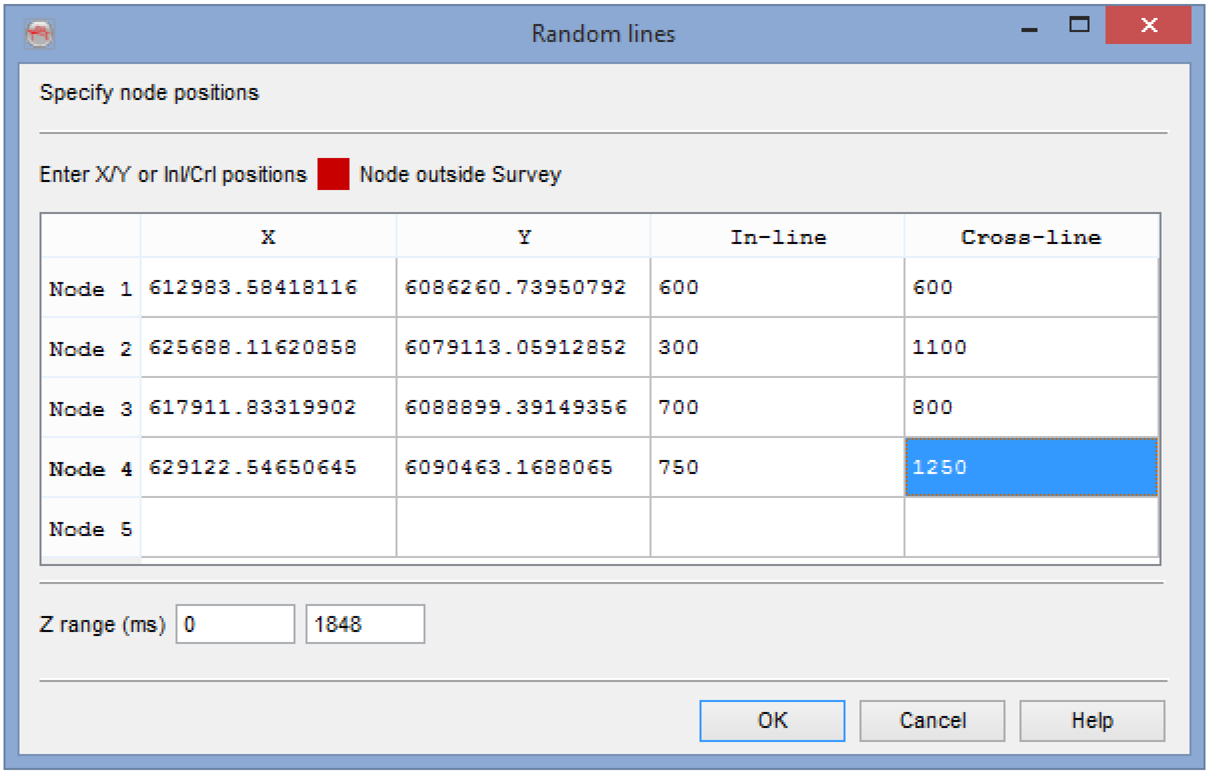
The following picture is an example of random line created from wells.



In this picture, a random line goes through four wells following a random path between these wells (which are used as constraints).

Create From Table

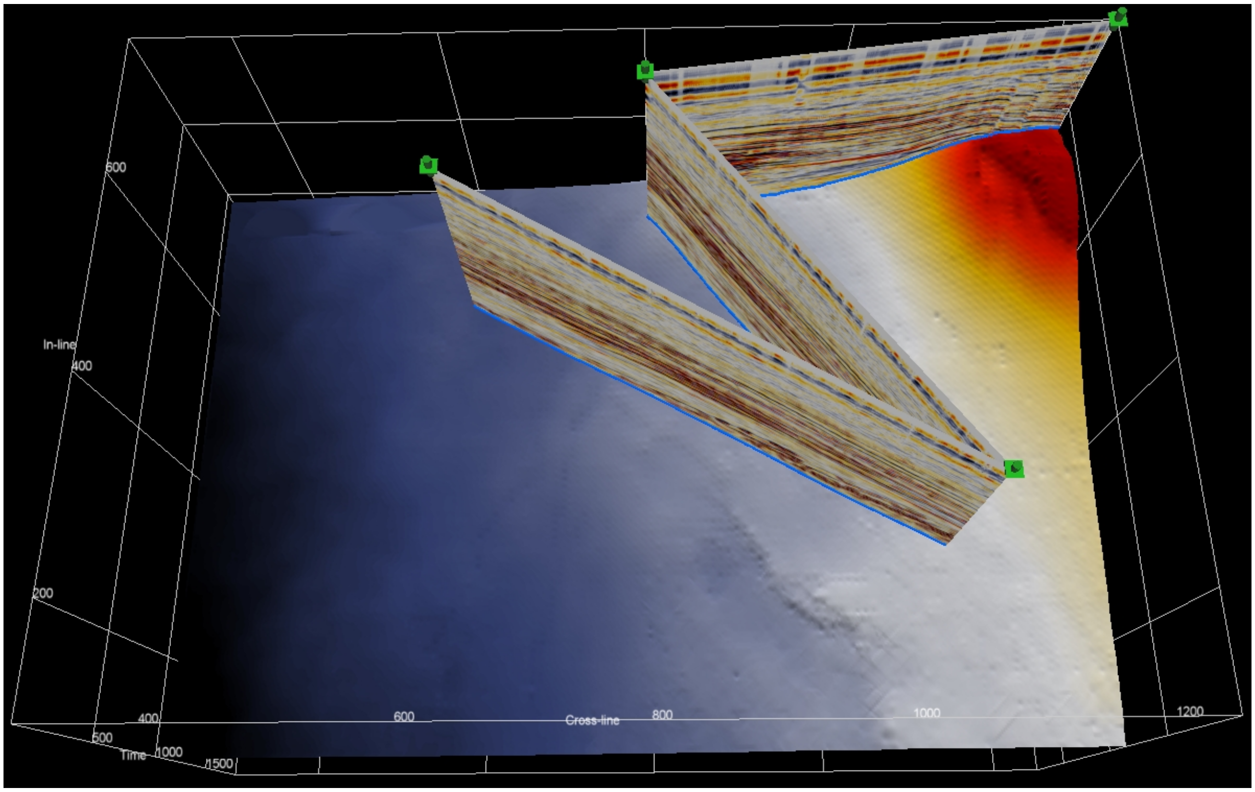
This is launched from: *Random line > right click > New > From Table*



This allows the user to create a random line from table. The input here are whether X/Y coordinates or Inlines/Crosslines and Z range

.

The random line resulting from this table is shown below

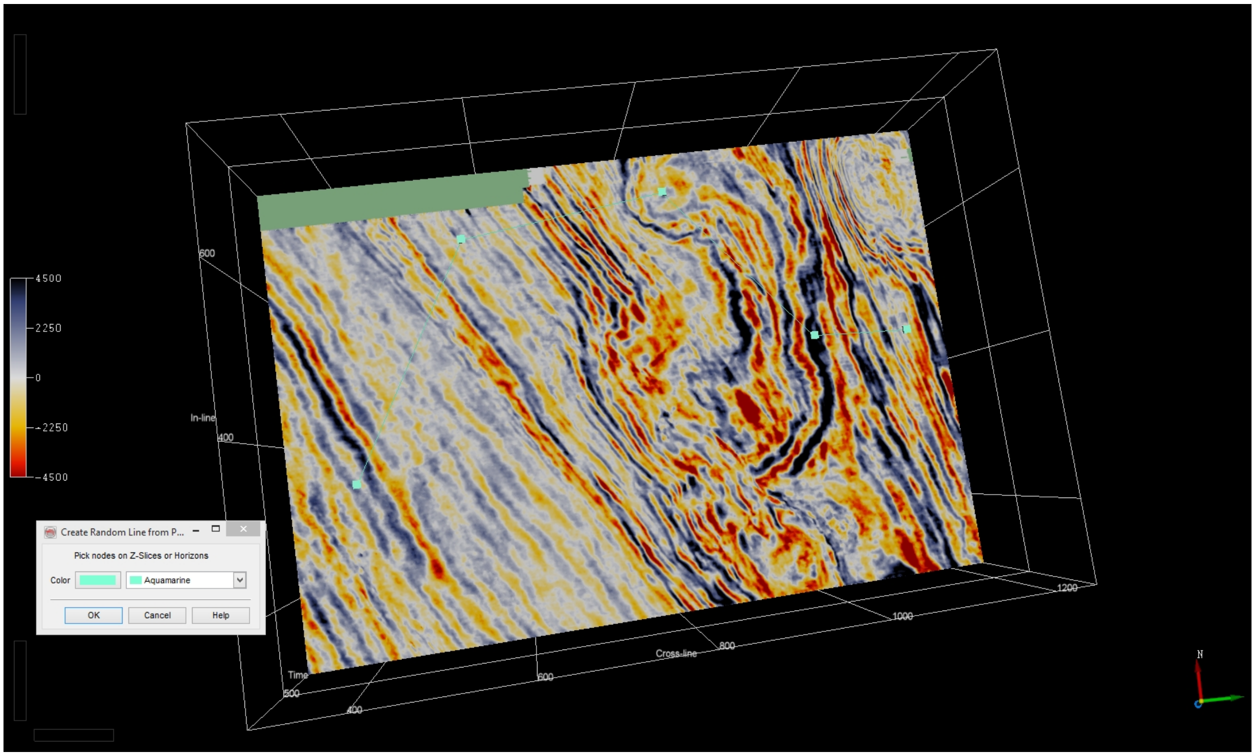


Interactive Mode

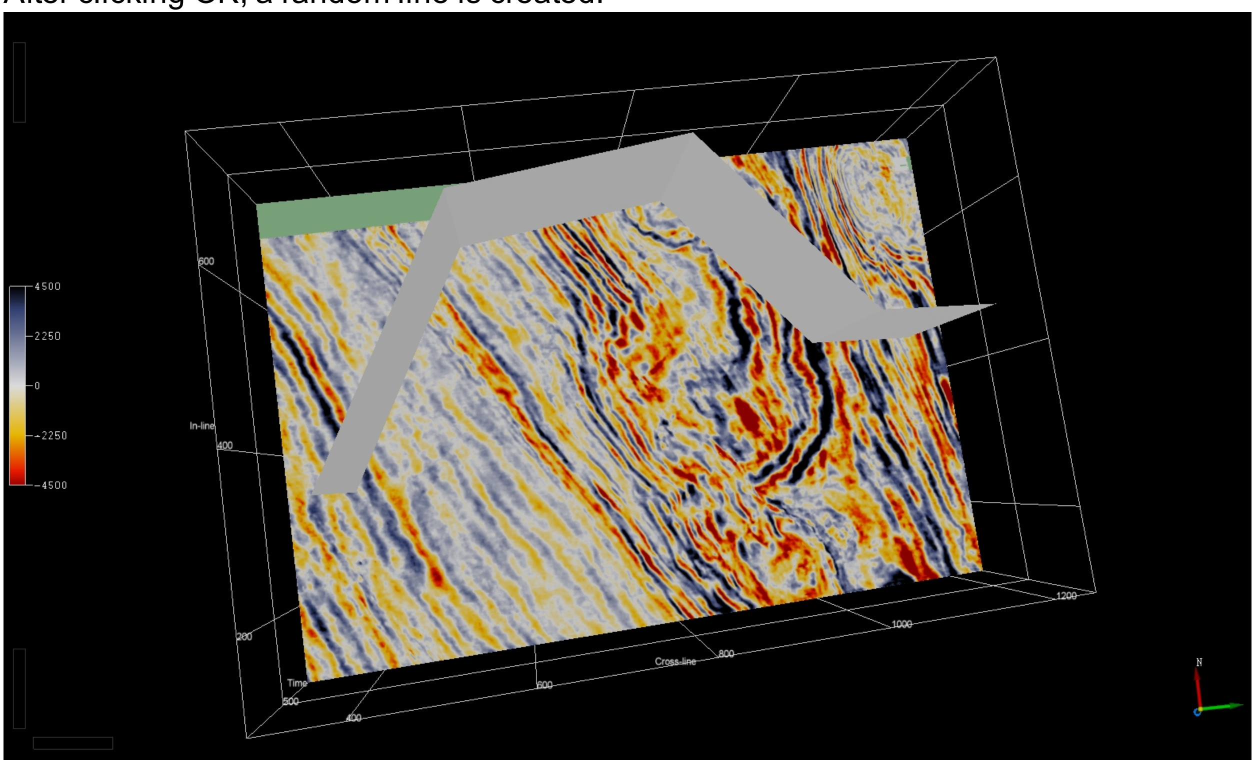
This option is launched via right-clicking on *Random line > New > Interactive...*

This allows the user to create random line from interactive mode. A horizon or Z slice is first loaded in the scene, then a random line can be created by picking nodes.

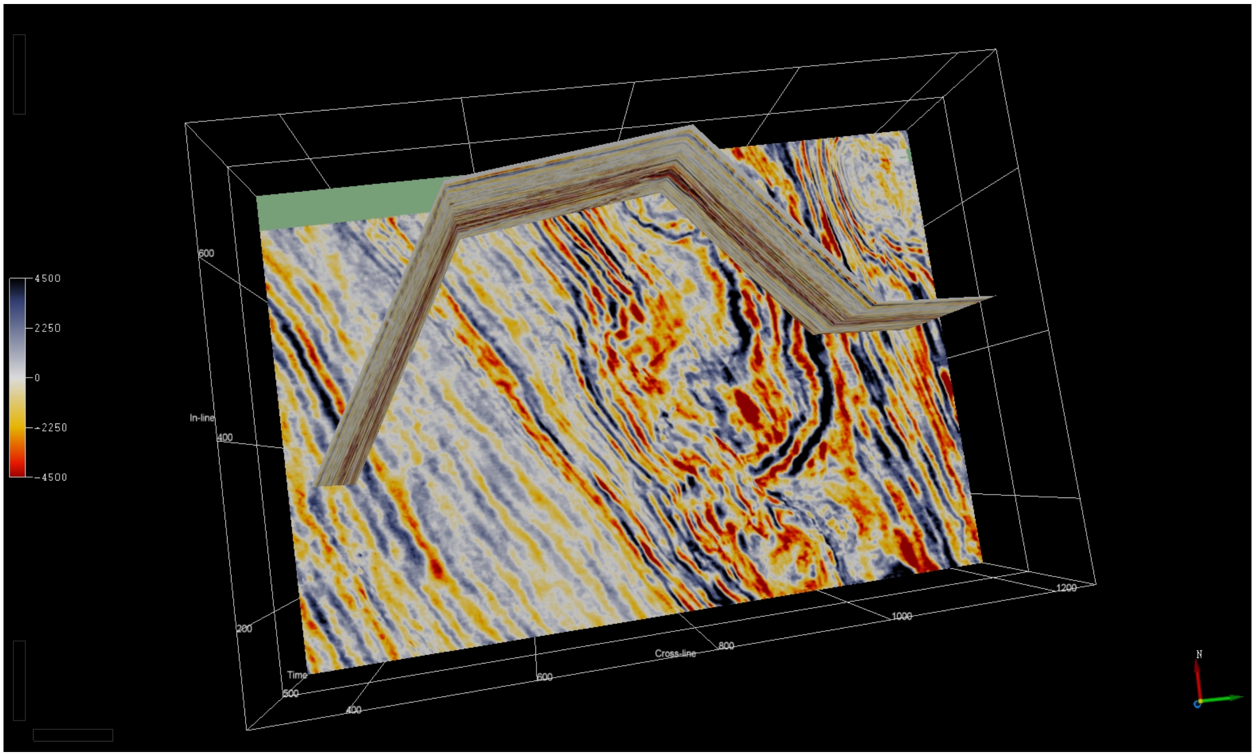
A window pops up asking the user to create a randomline from Polyline.
The user can now pick nodes on Z -slices or Horizons, as shown below:



After clicking OK, a random line is created:

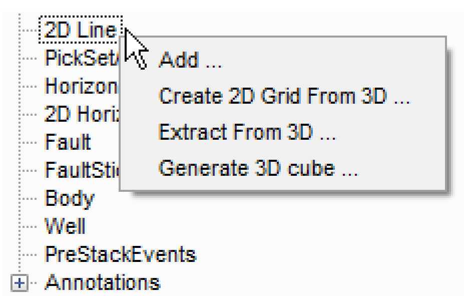


An attribute can then be displayed:

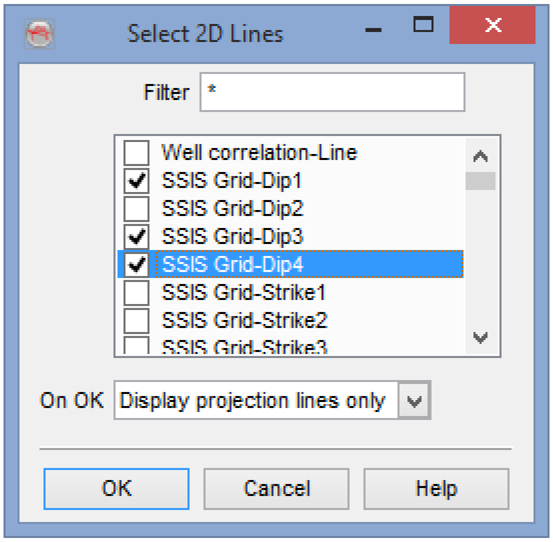


2D Seismic

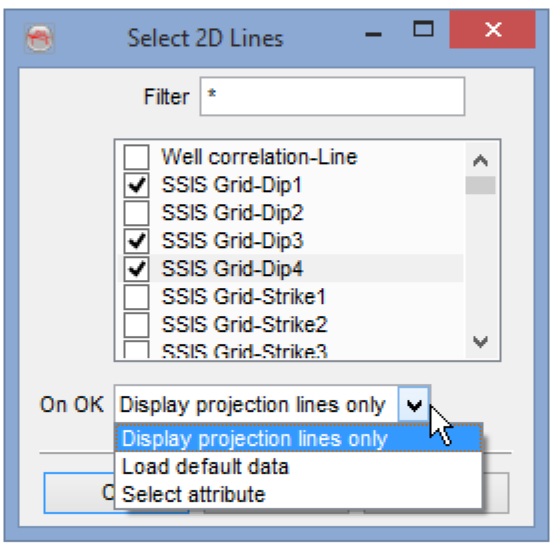
The user can either add 2D seismic lines, create a 2D Grid form 3D data, create new lines from 3D data or generate 3D cube from 2D data set:



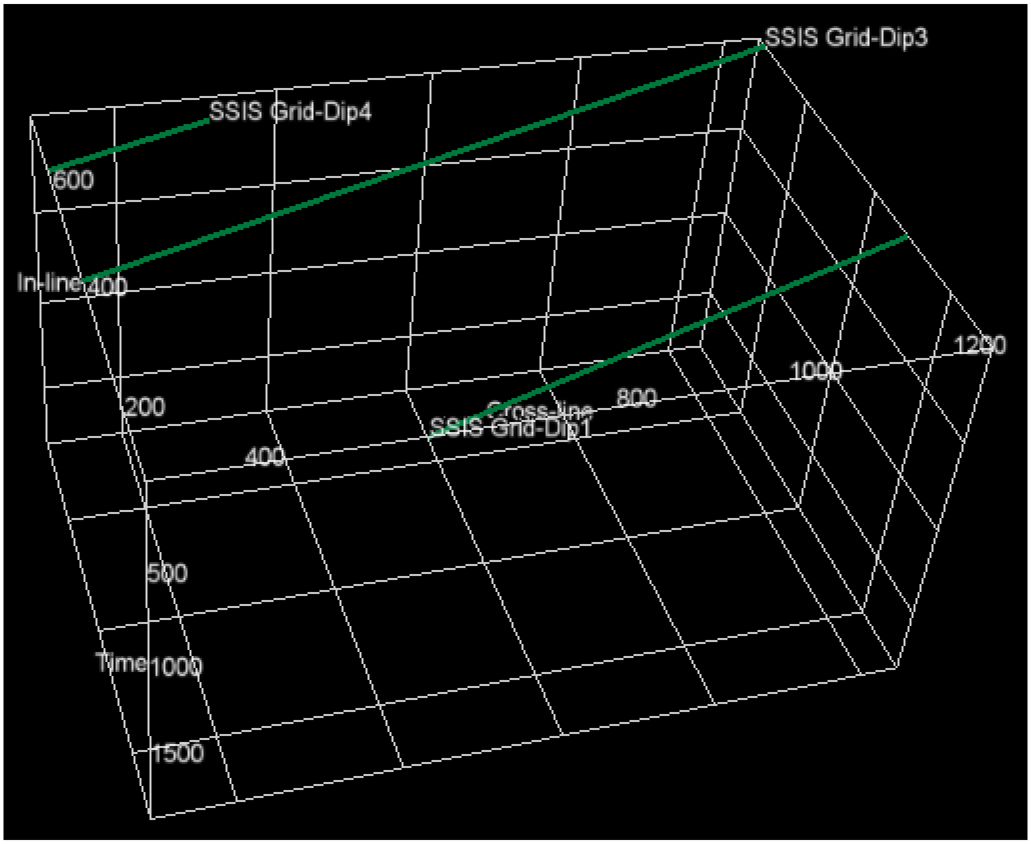
Right click on *2D* and select *Add* to add one or more 2D lines:



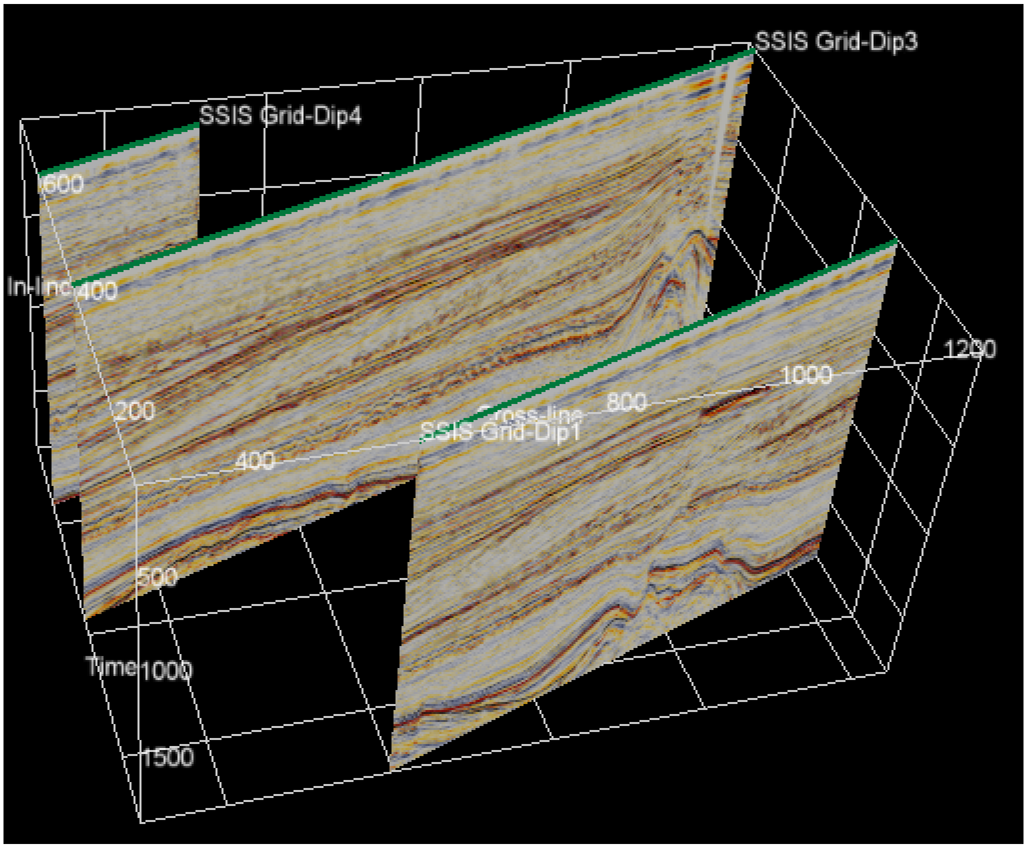
Before pressing '*Ok*', choose how you would like the lines to appear in the scene from the three options shown below:



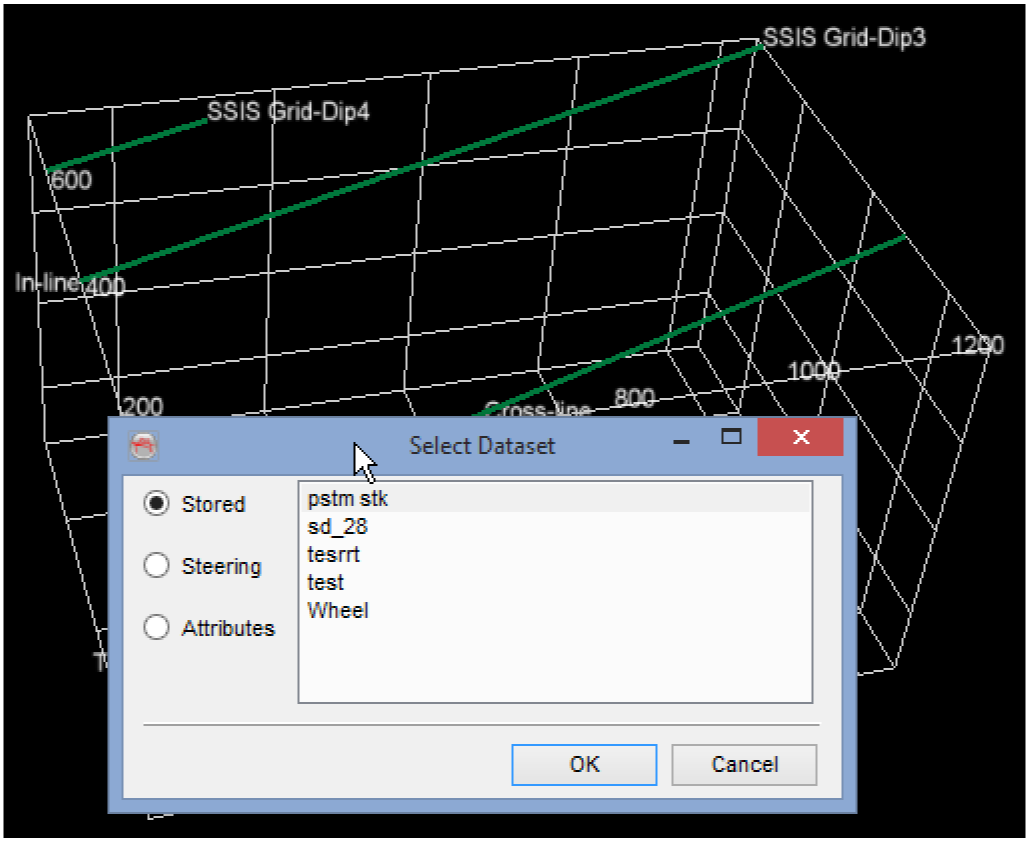
Display projection lines only: shows only the position of the 2D line(s) at the top of the survey:



Load default data: loads the line(s) into the scene, fully displaying them with the data selected as default in the 'Manage 2D Seismics' window:

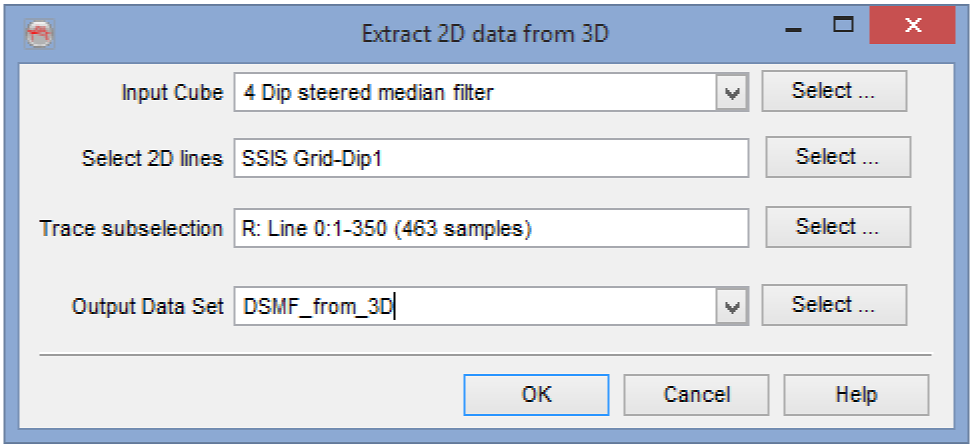


Select Attribute: loads the projection lines into the scene and brings up the '*Select Dataset*' window to choose what to display:



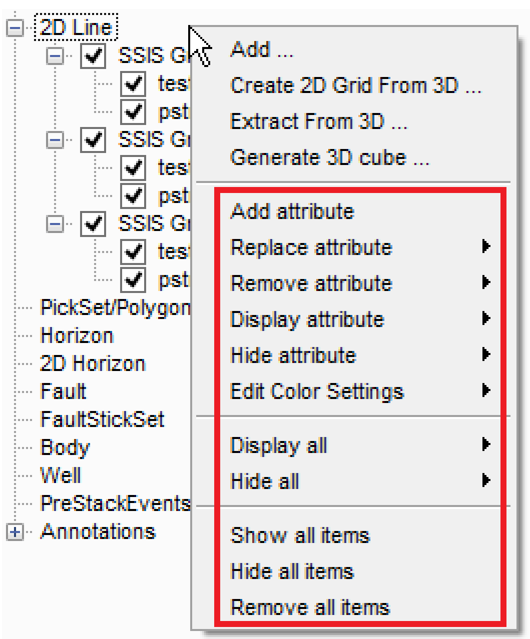
- **Create 2D Grid from 3D...** This option can be used to create a 2D grid with a fixed grid spacing. When selected, the *Create 2D Grid* window is launched. Here, specify the input 3D seismic volume and the output data set name. The output grid is generated according to the dip (parallel) and strike (perpendicular) direction of the selected volume. For more detail, click here

- **Extract from 3D...** Extract 3D data onto selected 2D lines. Input data is required in the form of a stored 3D volume. One or more 2D lines can be selected for the 3D data to be extracted onto. (Note: If just one line is selected, you may also sub-select a trace range.) The output data set requires naming:



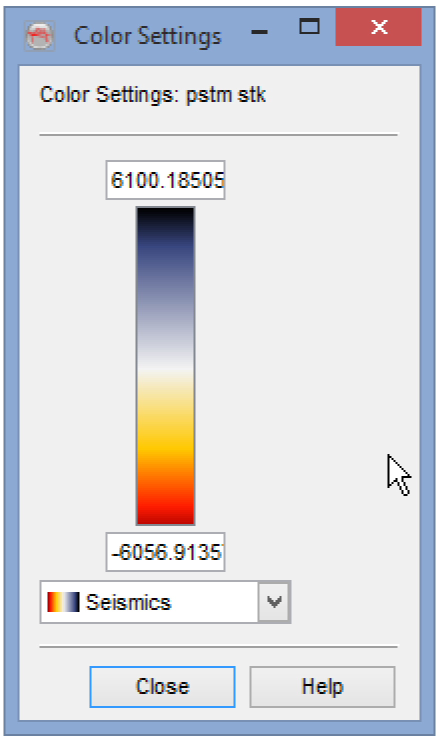
- **Generate 3D cube...** Allows for the creation of a 3D cube from a 2D data set. Interpolation can be based on 'Nearest Trace' or cab be 'FFT Based'. For more detail, click here

Once several lines displaying data are loaded, right-clicking on '2D Line' in the tree gives the following pop-out menu. In addition to the options explained above, selections can be made for all displayed lines:



- **Add attribute** Select an (additional) attribute to be added to the lines. (See above for details.)

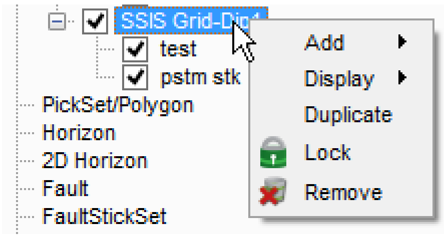
- **Replace attribute** Select an attribute from those displayed on the line(s). Once selected, this will launch the 'Select Dataset' window and a replacement can be chosen.
- **Display attribute** Choose which of the available attributes to display.
- **Remove attribute** Remove one of the loaded attributes (Only available if lines display more than one attribute)
- **Display Attribute** Checks on the selected attributes to display them on the lines showing in the scene
- **Hide attribute** Checks off the selected attributes to stop them displaying on the lines showing in the scene
- **Edit Color Settings** Select an attribute and set the color bar and ranges:



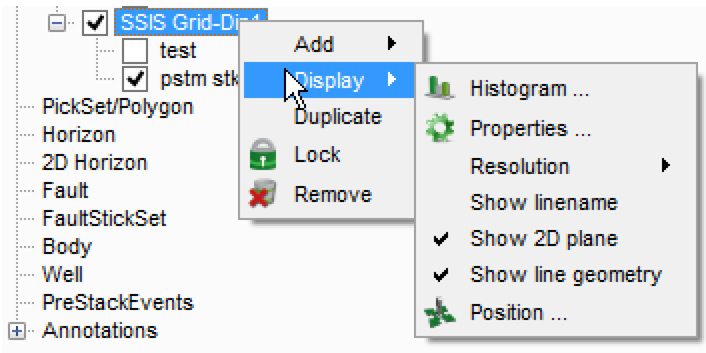
- **Display all/Hide All** Display or hide the line names, 2D planes and line geometry (projection lines).

- **Show/Hide all items:** Shows or hides all lines in the scene and checks/unchecks the line names in the tree
- **Remove all items:** Removes all lines from the scene and the tree

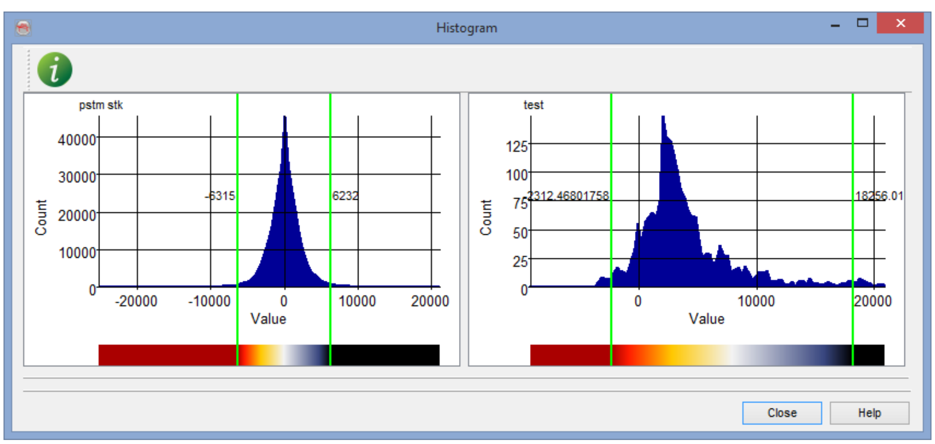
Right-clicking on an individual line in the tree gives the following options:



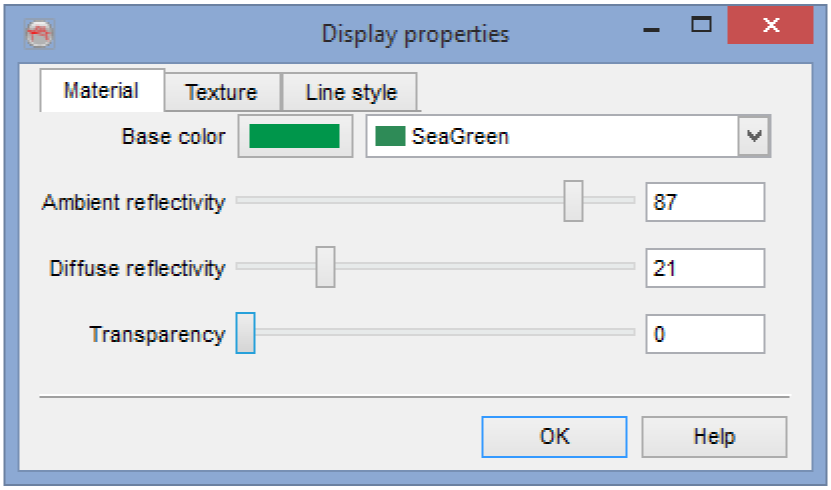
- **Add** Add either an attribute or a Volume processing attribute
- **Display** Allows the following:



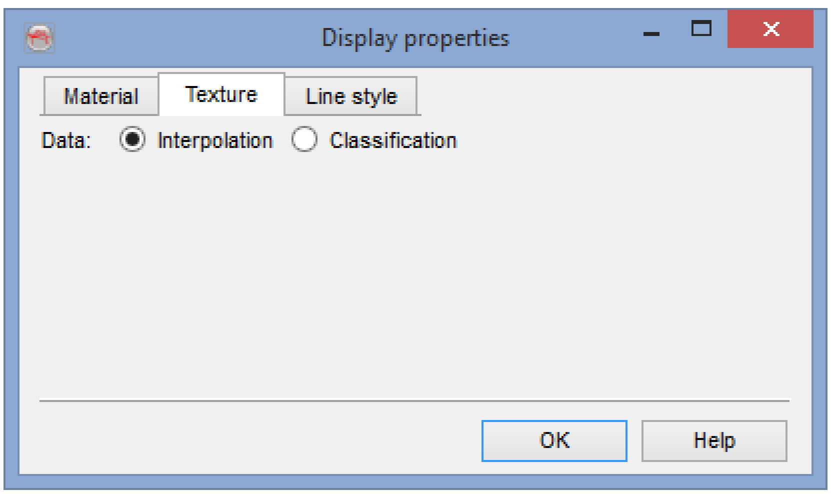
  - **Histogram:** The histograms of all added attributes of an element can be displayed using the right-click option of the parent element (inline number, surface name...). It is a useful tool to clip the ranges of an attribute by using vertical green lines in the histogram: The vertical green lines show the current amplitude range and can be moved left or right using the left mouse-click and drag. The display is updated when the mouse click is released. This is performed independently for each attribute. Please note that this will toggle off the automatic clipping.



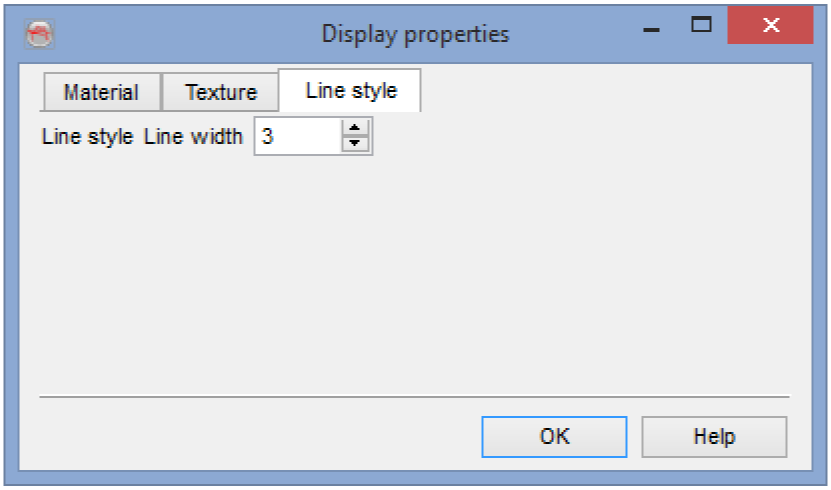
  - **Properties:** Use this option to set the *Material*, *Texture* and *Line Style* properties:



Material: Set the base color for the projection line (2D geometry) and set the reflectivity and transparency properties for the displayed attribute.

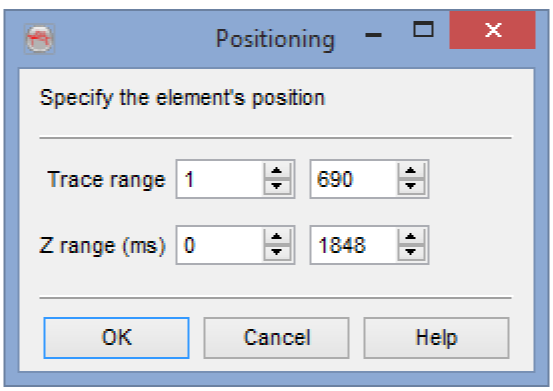


Texture: Set the texture type.



Line Style: Set line thickness.

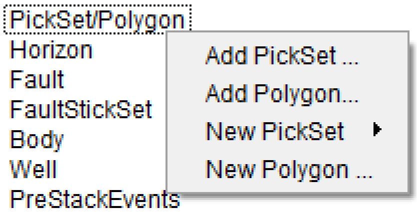
- **Resolution:** Set the resolution of the displayed data.
- **Show linename/2D plane/line geometry:** Toggle on or off each of the 2D line components per line.
- **Position:** Set the number of traces and Z-range of the displayed line:



- **Duplicate:** Duplicates the 2D line as displayed in the scene, including its displayed attributes. Can be a useful option to compare colorbar settings, or be used to 'extend' the eight-per-line attribute limit by replacing existing attributes with others on the duplicate.

Pickset & Polygon

A *Pickset* is a point data that can be used in OpendTect for several purposes, as simple point, vector quantizer, as polygons, etc. The *Pickset* pop-up menu has four options as shown below:

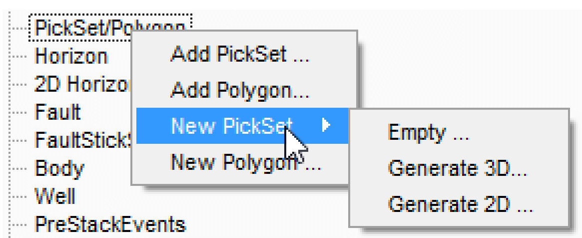


Add Pickset, Add Polygon will load existing picksets/polygons while *New Pickset, New Polygon* will create new ones.

Add Pickset... Load a previously created and saved pickset.

Add Polygon... Load a previously created and saved polygon.

New Pickset: A pickset can be filled using the following three options :



Empty: Start picking from scratch.

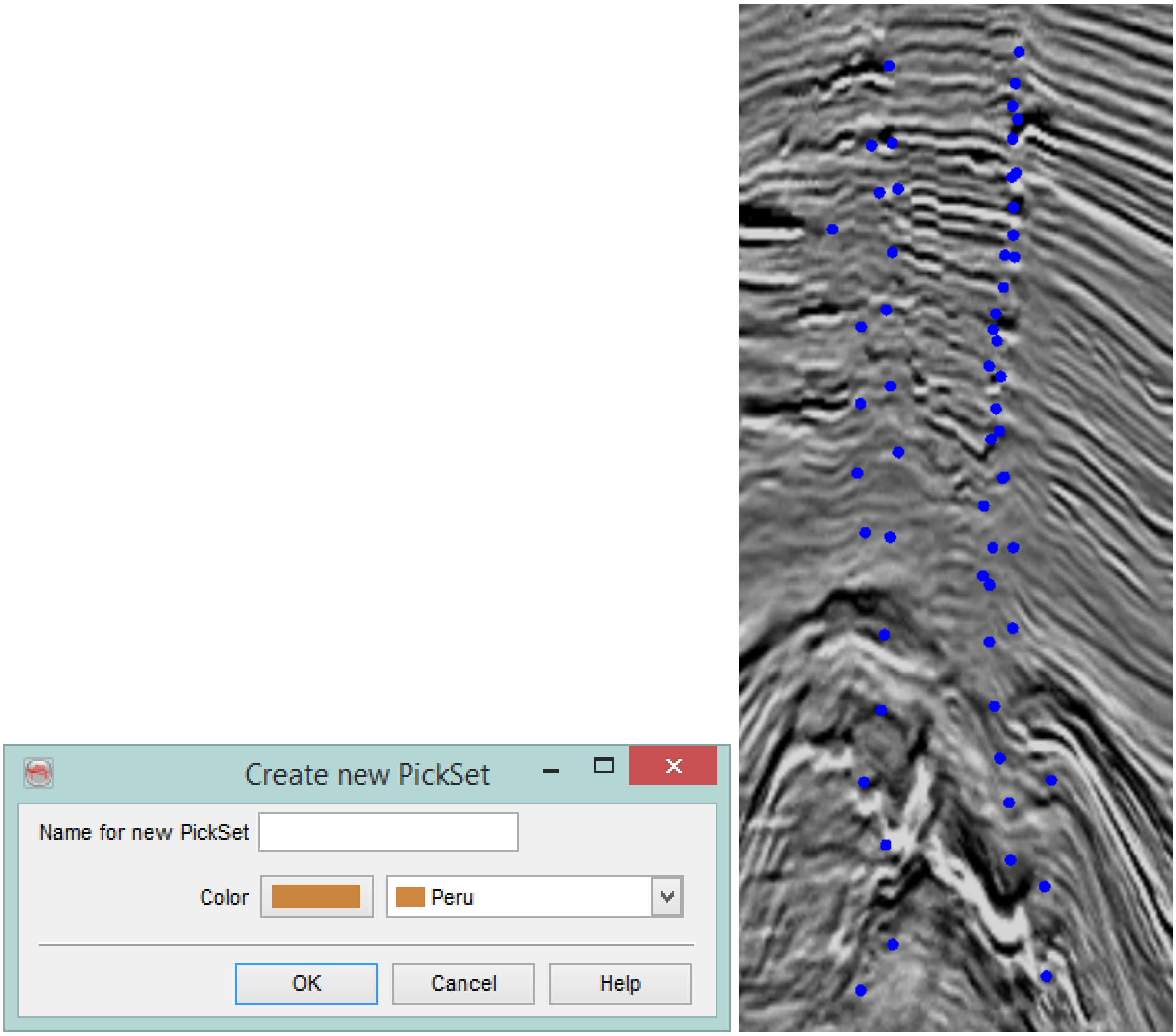
Generate 3D... Generate a pickset using *range, polygon, table*, or *surface*.

Generate 2D... Generate random picks for a 2D data.

New Polygon... Use picksets to draw, e.g a polygon around a contour (especially useful in Common Contour Binning). You can also store your polygon as fault.

Manual & Empty Picksets

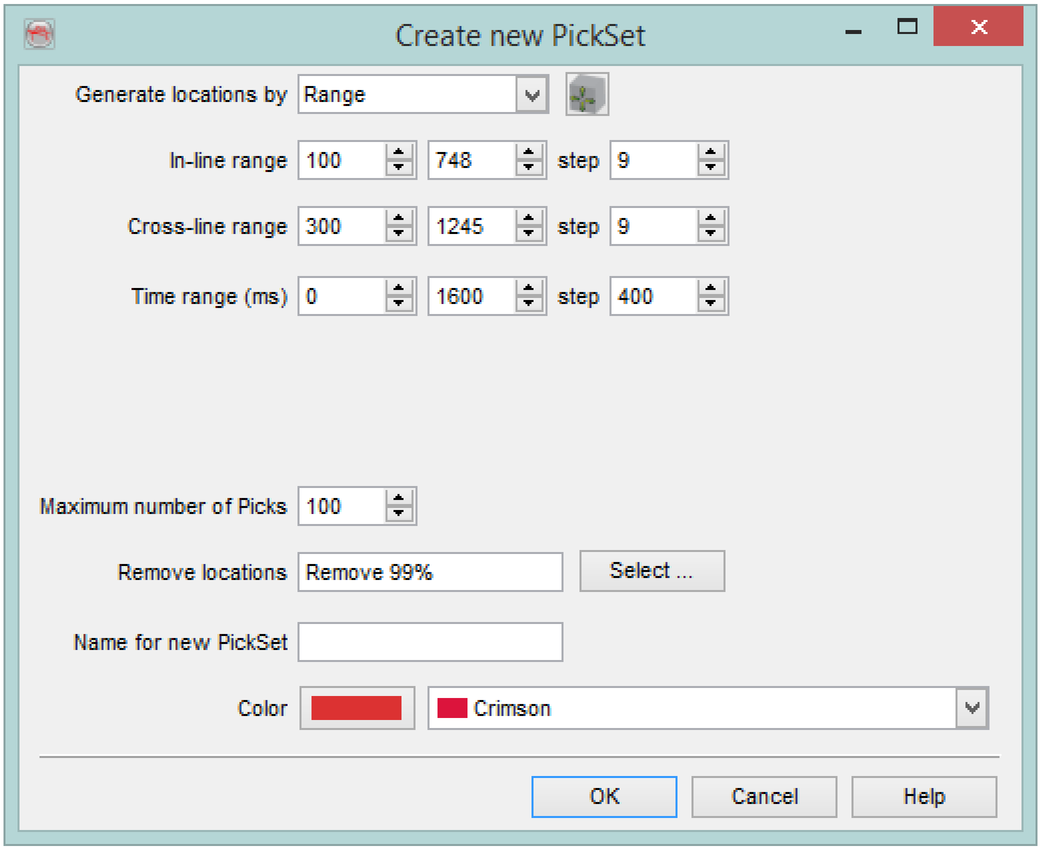
An empty pickset is added so that the locations (of object) are picked manually. The empty picksets are generally used for supervised neural network training (see the dGB plugins help documentation). An empty pickset is defined by right-clicking on the *Pickset* element and selecting *New* > *Empty* from the pop-up sub-menu. In the Pickset Creation window, write an appropriate name and press OK to insert an empty pickset in the tree. For instance, lets define an object by using manual picks that are representative of salt. Display the corresponding data (line/slice/horizon) in the scene. To start picking, please make it active by clicking on it (If active, it will be highlighted). Within the scene, start clicking on the displayed data (line/horizon). Each click will add a pick. If the pick is wrong, it can be removed by using **Ctrl** key and left mouse-button click.



Pickset creation window plus example of picks from the manual creation option. These picks will be used in Neural Network training.

Generate Random Picks (3D)

Random picks are very important especially for prediction purposes or object detection. These are defined specifically for unsupervised neural network training (see dGB Plugin documentation for more details). Unsupervised waveform segmentation (UVQ*) is one of them. The window is launched from the tree element *Pickset/Polygon* > *(right click)* > *New Pickset* > *Generate 3D....* The following window pops up.



Random (3D) pickset creation window

In this window write an appropriate name for the picks and select a distinct color. The generate location by contains several options: Range, Polygon, Table, Surface. It generates random picks within a selected range of inlines/crosslines/time, polygon, table, at a surface, or between two surfaces. Select and fill rest fields that are according to the selection in the *generate locations by...* option.

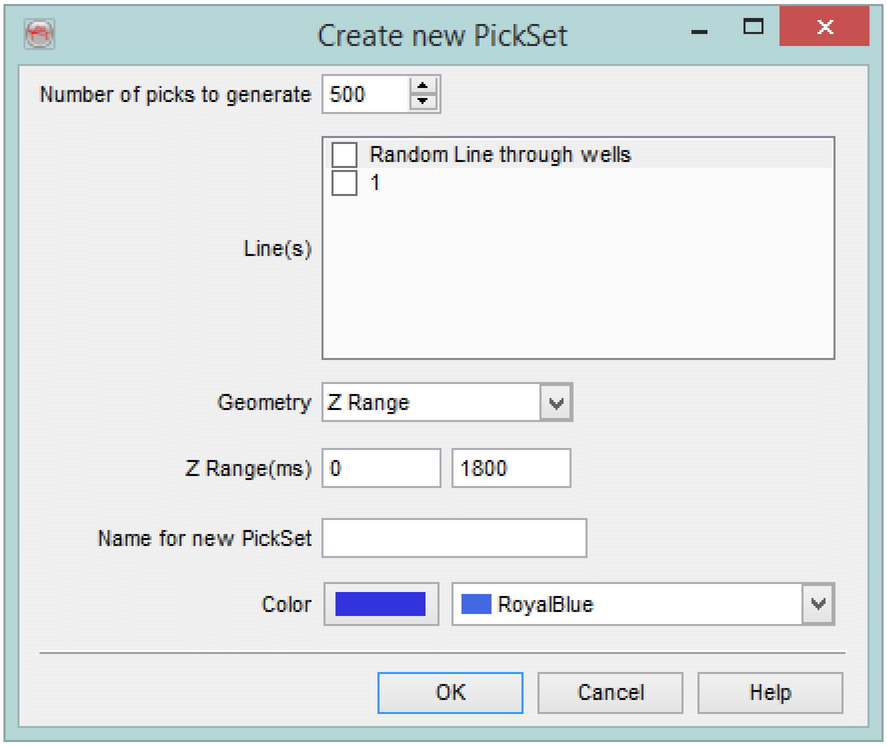
Optionally, a rejection filter can also be applied by selecting *Remove locations*. It passes each random position according to a selected filter (random, polygon, subsample, table, surface). It is useful to avoid random picks in unwanted regions e.g. by providing a polygon.

Provide the maximum number of locations to be inserted in the bottom text field. Fill an appropriate random number. Press OK to insert random picks.

> ***Note:** UVQs are explained in Neural Network section of the plugin documentation.

Generate Random Picks (2D)

Random picks for the 2D data can also be used for the same purposes as that for random picks for 3D (read previous section). The pickset creation window for 2D random picks can be launched from the tree element *Pickset/Polygon > (right click) > New Pickset> Generate 2D...* sub menu. It will launch the following pickset creation window.



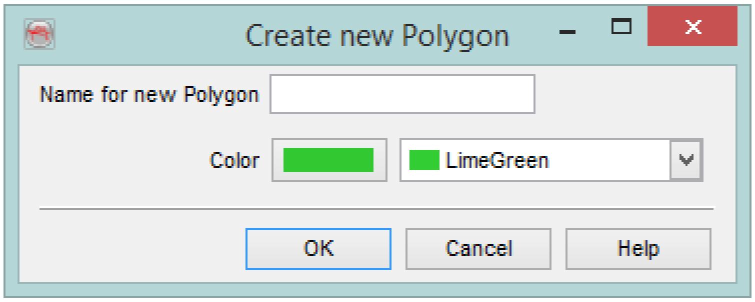
Random (2D) pickset creation window

In this window fill/select the fields for name, a color, the number of (random) picks to be generated, on a specific data set (or line(s) in a data set), geometry and Z-range. It may be noted that the specified random picks are generated according to the given geometry. The geometry is defined by a Z-range/ horizon (s). It depends upon the purpose/objective. For instance, if the objective is to detect facies by using random vectors (picks) on a surface, then horizon geometry shall be provided. Similarly, random picks can also be defined between two horizons on the selected lines.

Polygon

The new Polygon can be created by doing the following: *Pickset/Polygon > right-click > New Polygon...*

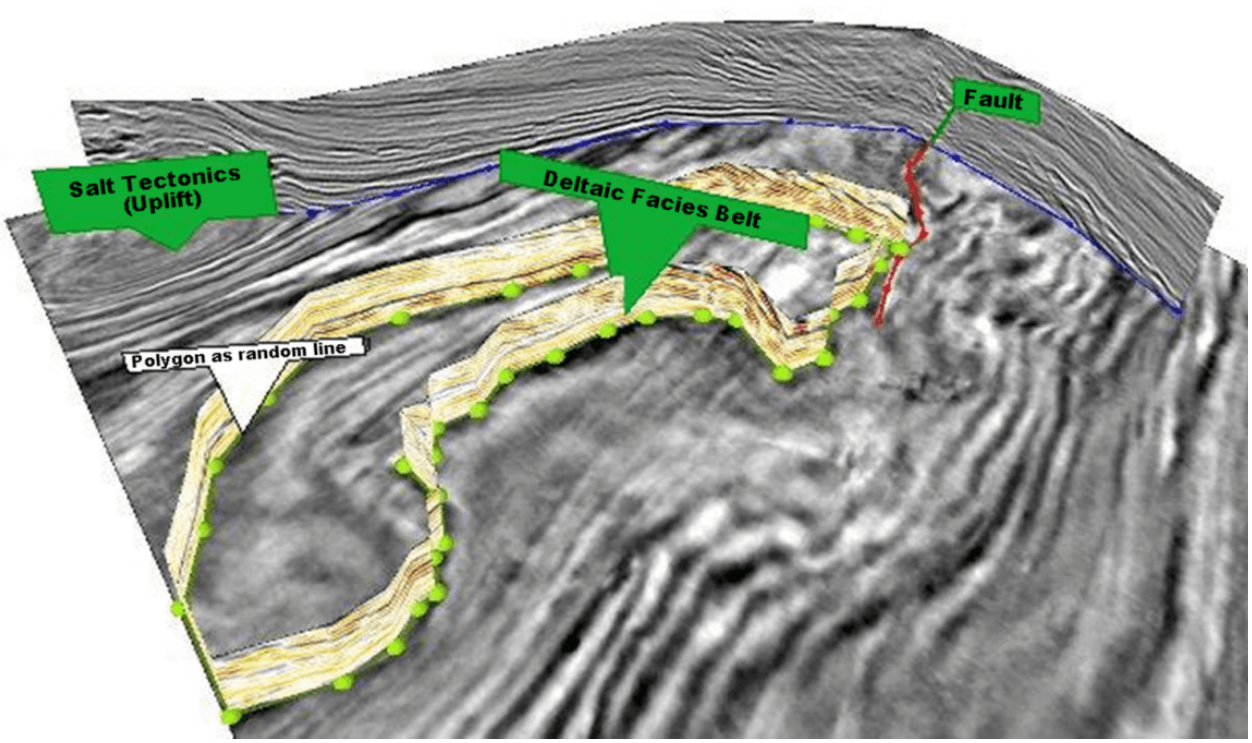
The window pops up as shown below:



Choose a name and color for the new polygon. An empty polygon folder is created and the user needs to pick a polygon on the chosen horizon or Z-slice.

Making polygons. Polygon picksets are activated by clicking on the element in the tree. The active polygon pickset is shown in reverse video and pick mode is automatically enabled. Left clicks in the graphics window will result in adding picks to the active polygon. The picks will be connected by a line. The last pick of the polygon can be removed from the active polygon with **Ctrl**-left clicks (use the Apple key on MacOS).

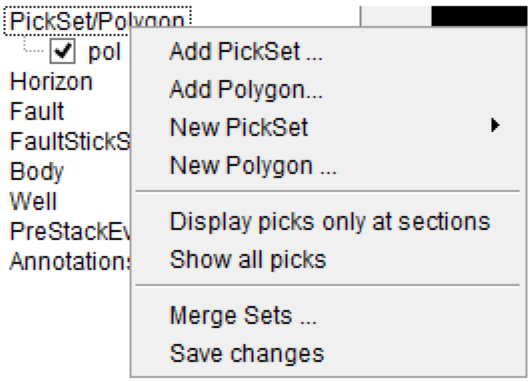
In the following picture we can see two examples of polygon picksets, closed polygon (deltaic facies belt), and non closed polygon (fault pickset).



Pop-Up Menus

Pickset/Polygon Element Pop-up Menu

This is the menu that pops-up when a user right-clicks on the main Pickset/Polygon element when at least one pickset/polygon is already loaded. It contains the following menu items.

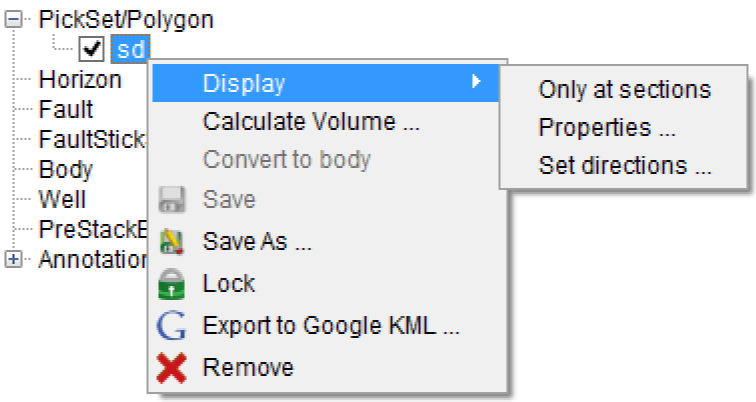


- **Load Pickset...** Display stored pickset(s).
- **Load Polygon...** Display stored polygon(s).
- **New Pickset** Adds a new pickset
- **New Polygon** Adds a new polygon
- **Save changes.** The pickset/polygon changes can be saved and reloaded at any time during the building process.
- **Display only at sections.** Display picks on the displayed elements in the graphics area only. This mode enables picking in a new location without being distracted by previously picked points throughout the survey volume.
- **Show all picks.** (Re)display all picks.
- **Merge Sets.** When at least one pickset is opened in the tree, it is possible to merge picksets by right-clicking "pickset". A window pops up and the sets can be selected. A name for the output set should be defined. Picksets can also be merged from the Pickset Manager.
- **Show all items.** Show the picksets from the tree-scene.
- **Hide all items.** Hide the picksets from the tree-scene.

- **Remove all items.** Remove the picksets from the tree-scene.

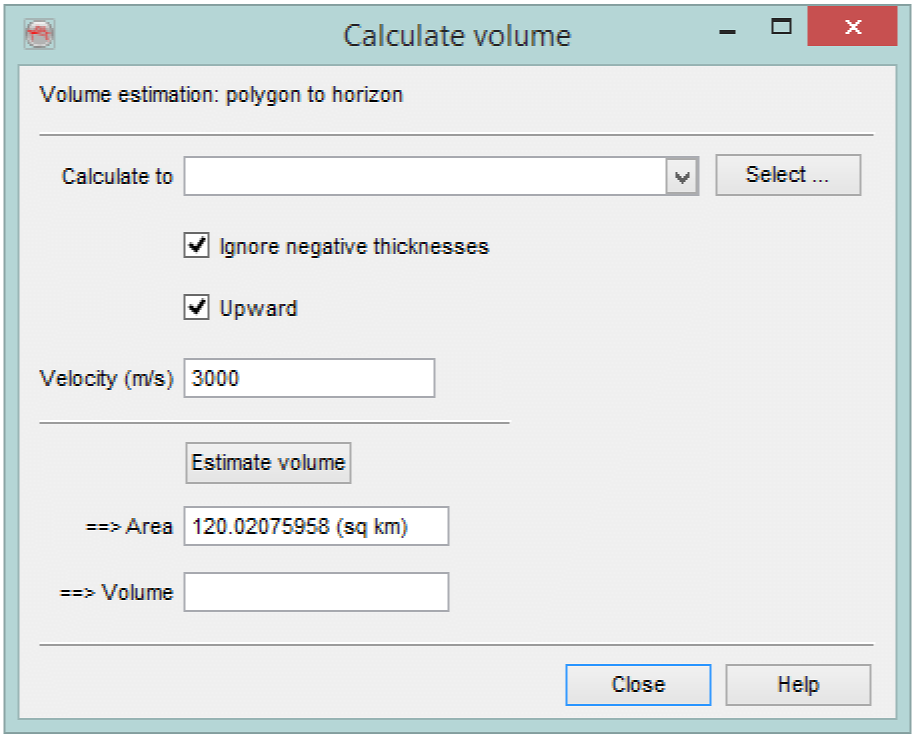
Pickset Sub-elements Pop-up Menu

If more than one pickset is loaded, the following options will be available in the tree:

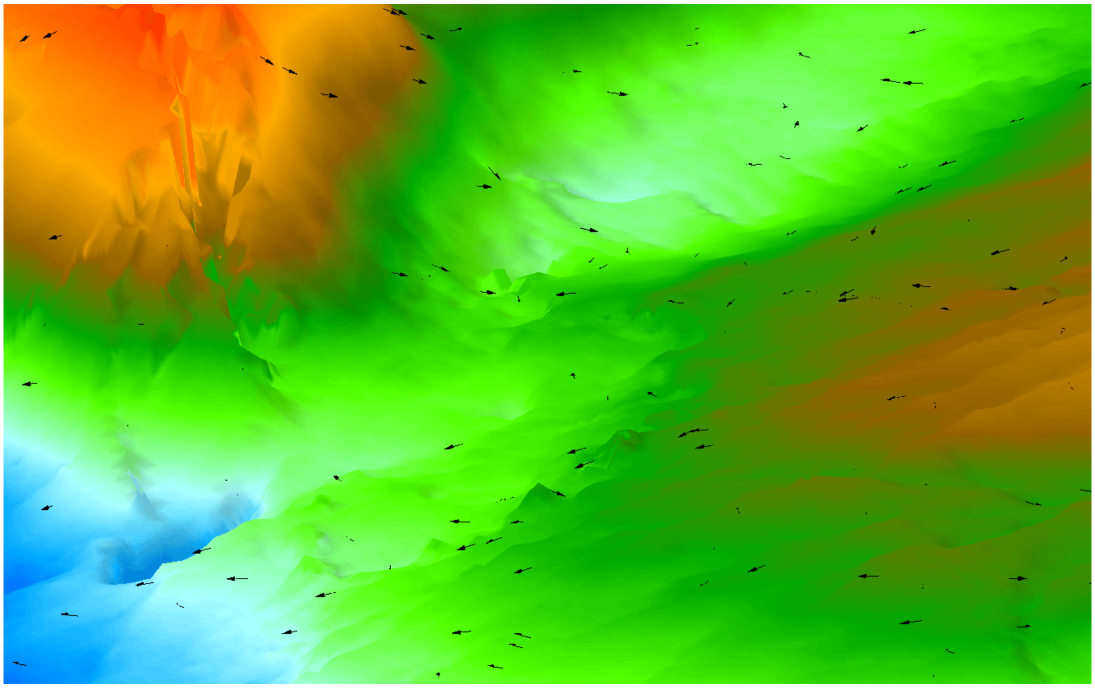


- **Calculate Volume**

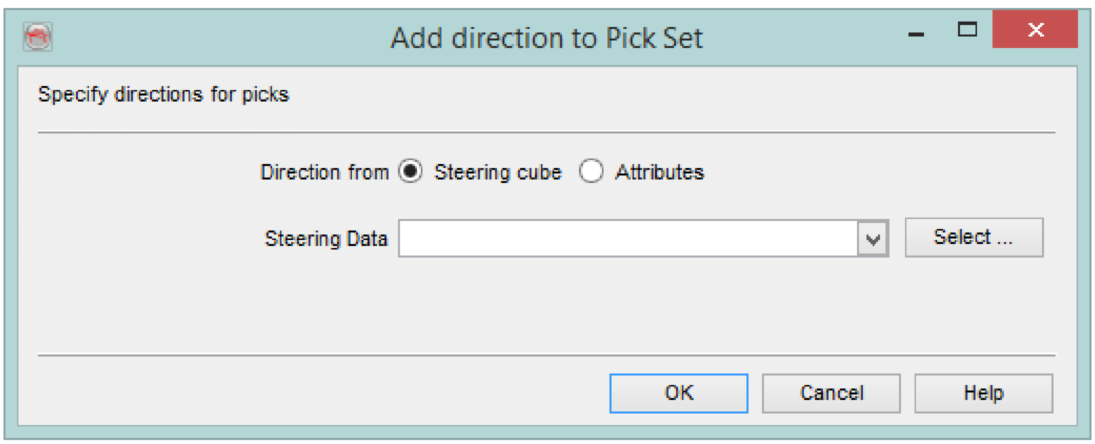
  In OpendTect, an estimated volume can be computed from a polygon to a given surface.
  The velocity default is set to 3000 m/s. Negative thicknesses can either be discarded or taken into account.

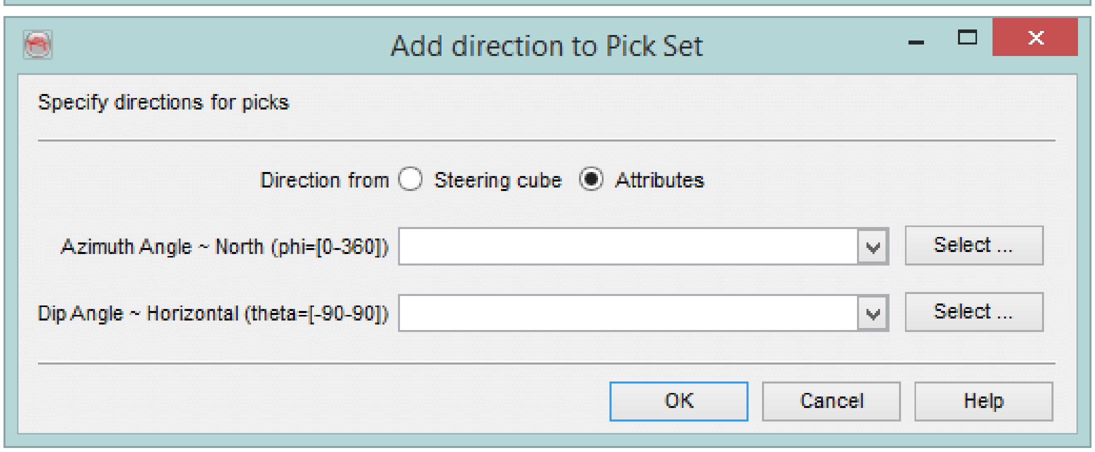


- **Close polygon.** During, and at the end of a picking session, picksets should be stored.
- **Properties:** In this window the *Type, Size* and *Color* of the pick markers on the graphics area can be set. The type *Arrow* is also automatically used when the pick is given directional information in the Set directions option under the pickset pop-up menu.
- **Display only at sections:** Shows picks on the displayed elements in the graphics area only. This mode allows to pick new locations without being distracted by previously picked points throughout the survey volume.
- **Convert to body:** Convert the polygon into a body.
- **Set direction:** Display direction, guided by the steering cube/attribute. This helps to understand the geological dips and fluid flow. It is assigned by setting a direction to each pick based on dip and azimuth information (attributes). In the pop-up window (see below), specify either a steering cube or two attributes providing the polar dip and azimuth in degrees. A velocity of 2000 m/s will be used in time survey to convert the dip from degrees to µs/m if the dip angle data is read from a stored cube instead of the dip angle attribute. Do not forget after setting the directions to save you pickset and change the display type to "Arrow".



An example of setting direction (black arrows) to a pickset.





Steering cube direction and Attributes input

- **Save/Save As:** Either overwrite the stored input by using option *Save* or store as new pickset by using *Save As* option.

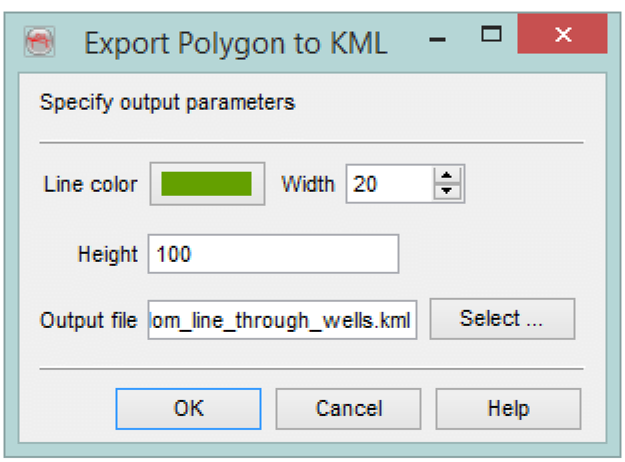
- **Create Body:** This option is available only for a polygon. It is used to create a body using the polygon as a constraining area. It requires a top and bottom horizon between which the body would be created. This feature is only enabled if SSIS plugin is loaded (or licensed). For details, please go to the Manual SSIS chapter of the plugin documentation.

- **Lock / Unlock:** Lock the selected object. Prevents accidental removing, moving or displaying data on the object. Clicking "Unlock" enables editing again.

- **Remove:** Remove pickset from tree.

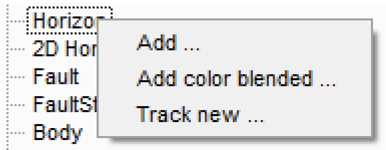
  Note: removing any unsaved pickset will result in complete loss of data.

- **Export to Google KML:** Export selected polygon to a Google KML file. When selected, the following export window is launched. Fill in the output KML parameters and write/select the output file location. Press the 'Ok' button to export the polygon in the selected location. The feature will prompt an additional conversion dialog if the conversion settings for the survey are not defined. For further information, please refer to the Survey Selection section.

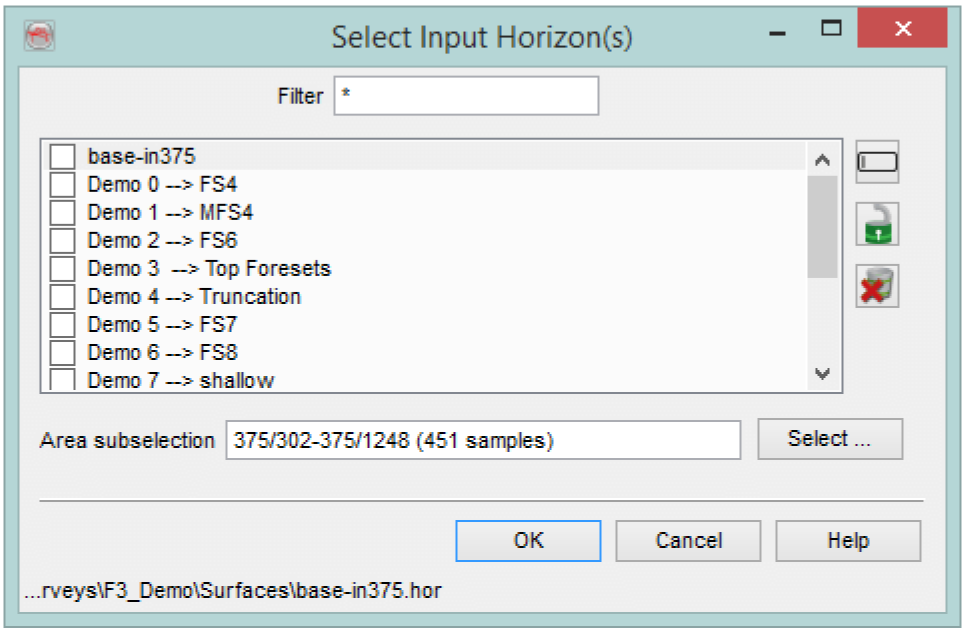


Export the polygon to a KML file

Horizon



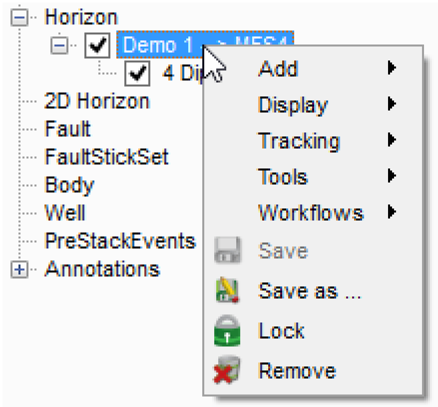
An existing horizon can be displayed in the scene by selecting *Add* option from the pop-up menu (see above). It will launch a horizon selector window from which multiple horizons can be selected. See also [Add color blended](#) .



> **Note:** Once at least one horizon is displayed, there is an addition to the pop-out menu, *Display All*, which contains several options: only at sections, in full or both at sections and in full. *Only at sections* results in a horizon display (as a line) on the inline/crossline/timeslice. *Full* displays the complete horizon in 3D space.

Track new sub menu is used to start a new [horizon interpretation](#).

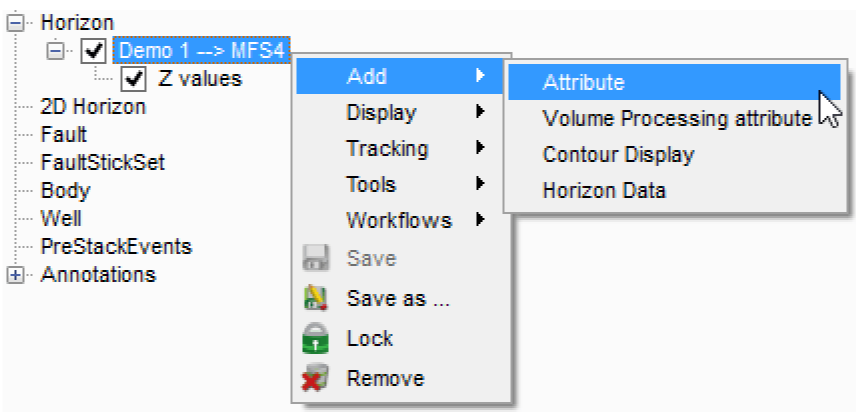
The popup menu from a displayed horizon has several options, which are covered in the following sections:



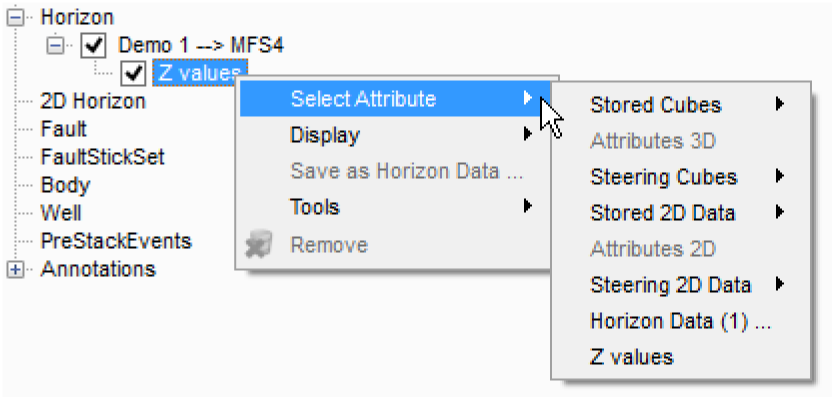
Add Attribute

This allows choosing the data to display on the horizon from stored cubes, a calculated attribute from the current attribute set, or horizon data that were included with the horizon already. For Horizon data a dialog will popup where you can select multiple data files. After loading you can browse through the data by pressing the 'Page Up' and 'Page Down' buttons on your keyboard.

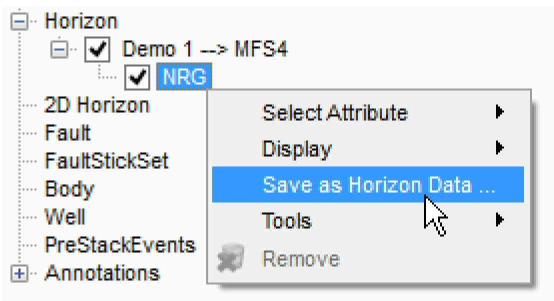
Note: for PgUP & PgDn to work, the mouse pointer must be in the scene.



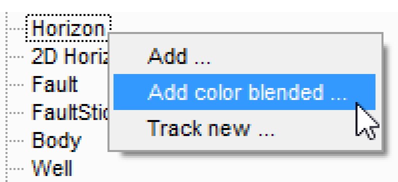
Once a horizon is added (with its Z-values displayed in the scene), it is possible to also right-click on '*Z-values*' in the tree to give you other options:



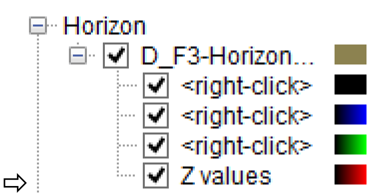
Furthermore, once a horizon has an attribute displayed, it is also possible to '*Save as Horizon Data*'... and will be visible in the '*Manage 3D Horizons*' window:



Color-Blended Display



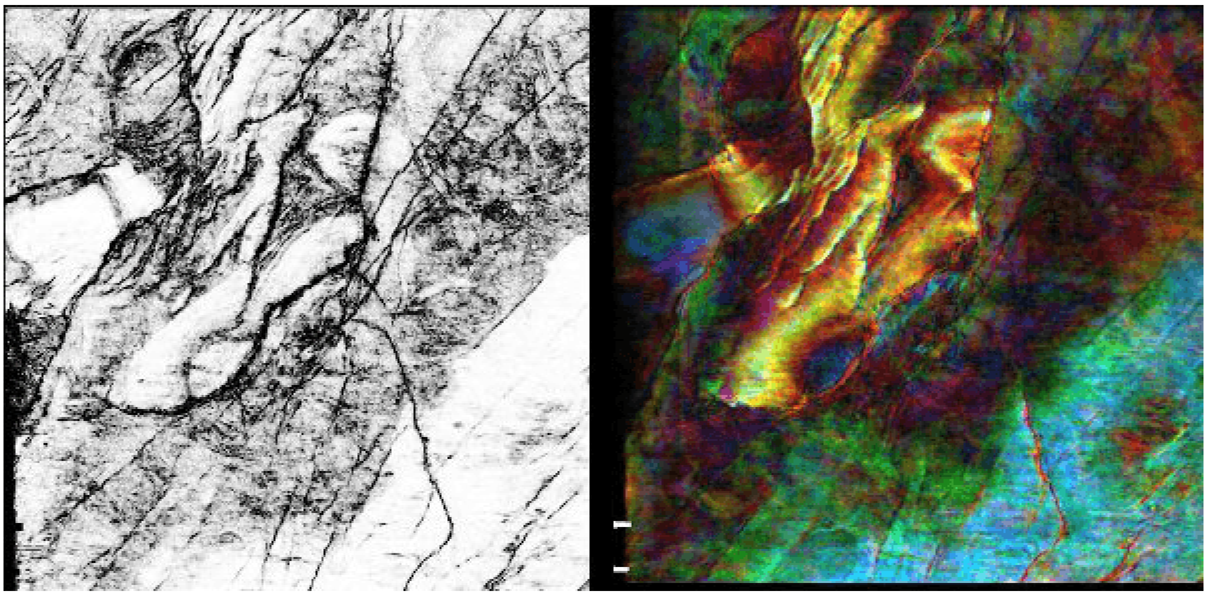
⇨ becomes ⇨



The load color blended sub-menu displays an RGBA (red-green-blue and alpha) blended horizon(s) in the scene. This is used to blend multi-attributes with similar spectral outputs. This is an interactive tool especially to color blend the iso-frequency grids (or attributes).

Color blended display:
RGBA blending attribute display* is used to create a normalized color-blended display that often show features with greater clarity and enhances a detail map view. Traditionally, it is used to blend the iso-frequency responses (Spectral Decomposition), but a user can blend three/four different attributes that define a spectrum that is comparable.

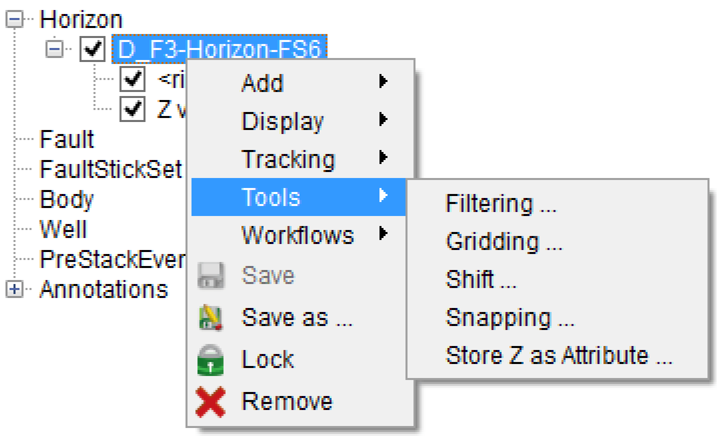
For instance, spectral decomposition outputs the amplitude at discrete frequencies. So, it renders the same output (unit=amplitude). Depending upon a geological condition or the objective, FFT short window or CWT (continuous wavelet transform) can be chosen. Results are best displayed on time/horizon slices, volume.



A color blended map view (image on right) of the spectral decomposition (red-10hz, green-20Hz, blue-40hz). Compare the results with the coherency map (image on left). Note that the yellowish colored fault bounder region is thicker as compared to the surrounding regions. The faults throw (red- color) are also clearly observable. Semblance/similarity together with color blended spectral images can reveal better geological information.

Tools

Several processing algorithms may be applied to horizon and will be described here:



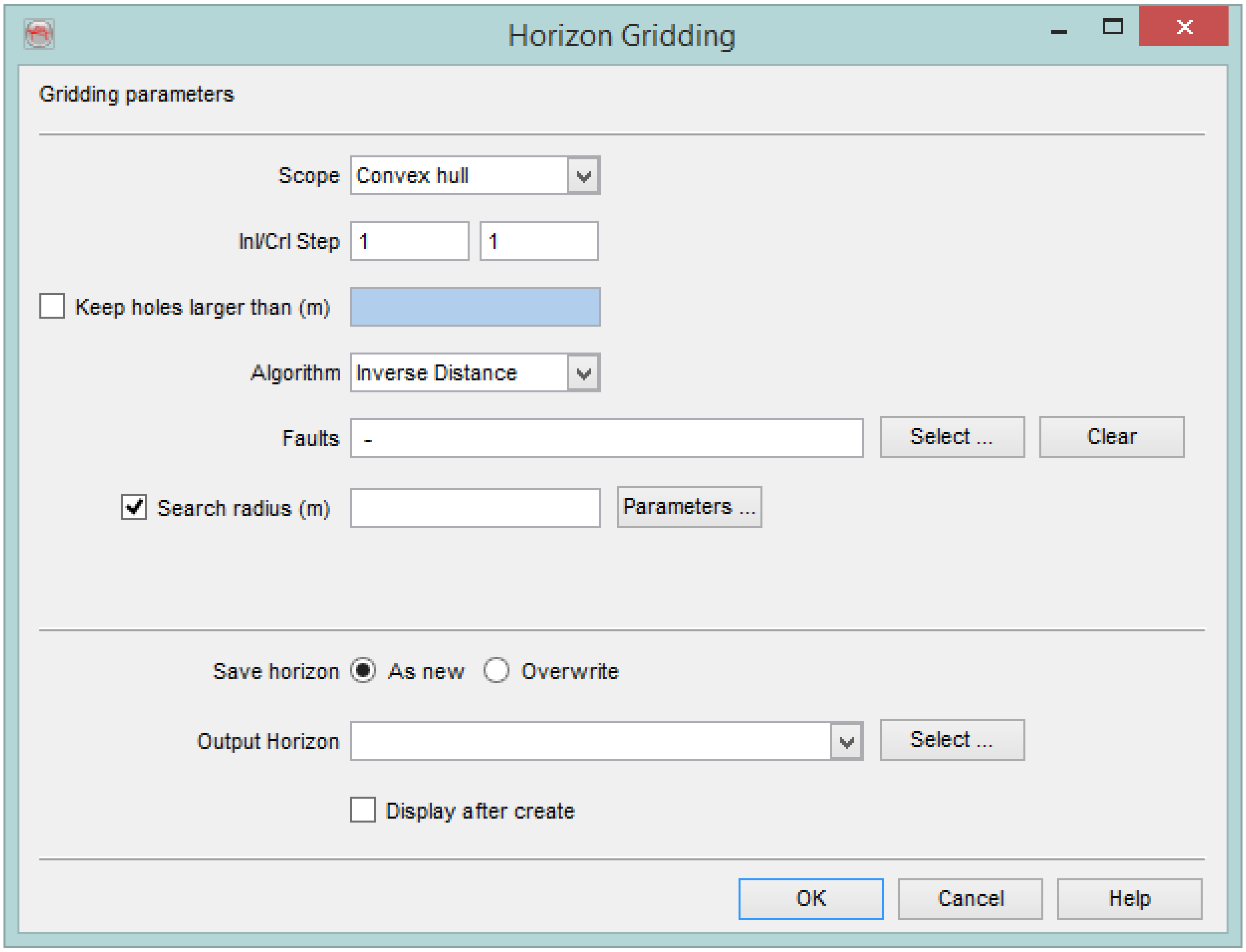
Adding picks to existing horizons by interpolation (grid)
Filtering interpretations
Snapping an existing interpretation to a given amplitude event.
Storing the Reference Z as an attribute

The recommended workflow order after horizon tracking is:

1. Snap the interpretation to a given event (min/max/zero crossing).
2. Grid the snapped grid since snapping can generate holes.
3. Filter the output grid.

Grid

This utility is used to grid/interpolate a horizon having gaps/holes or to filter (average/median) a horizon grid. There are several gridding algorithms supported in OpendTect.



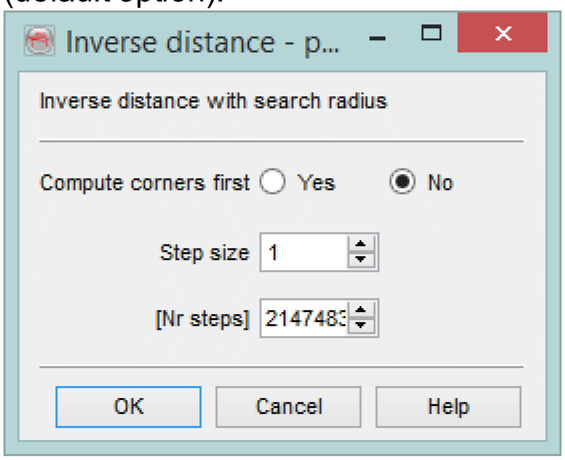
Gridding Parameters:

- **Geometry:** There are different types of geometries that are used to do interpolation. The Full survey is used to interpolate (in-/out-wards) the Horizon-Z values within the entire survey box. The Bounding Box defines the rectangle fitting the horizon geometry, which is generally smaller than the survey box. The Convex hull type of area fitting also restricts the gridding geometry within the horizon boundaries. To grid the gaps or holes in a horizons, the Only holes type of gridding geometry is used.

- **Inl/Crl step:** The default steps correspond to the sampling rate of the input horizon. The step can be decreased up to the survey sampling rate to get a higher resolution horizon.

- **Algorithm(s):** Inverse distance algorithm uses an inverse distance method of interpolation. Inverse distance requires the search radius with optional parameters (step-size and number of steps). The step size of '1' means that one bin would be used in all directions to interpolate the horizon Z-values. Whereas the number of steps define the number of concentric circles for inverse distance computation. For these steps, the grid computation can be set to the corner points for the defined radius or not (default option).

- *Inverse distance* algorithm uses an inverse distance method of interpolation. Inverse distance requires the search radius with optional parameters (step-size and number of steps). The step size of '1' means that one bin would be used in all directions to interpolate the horizon Z-values. Whereas the number of steps define the number of concentric circles for inverse distance computation. For these steps, the grid computation can be set to the corner points for the defined radius or not (default option).

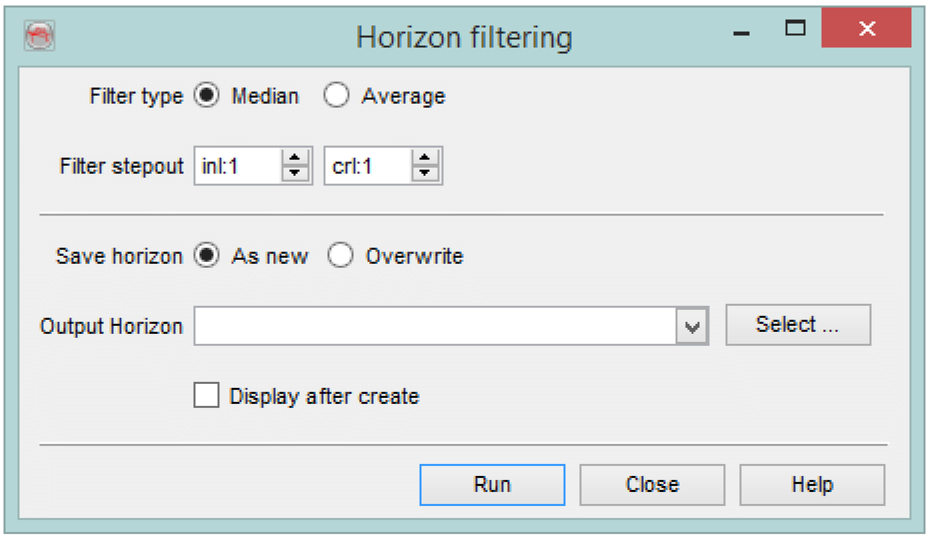


- *Triangulation* is a fast gridding algorithm, which uses triangulation method of interpolation. The interpolation can also be defined by providing an optional maximum distance (radius) by setting interpolate option checked.
- *Extension* uses a simple linear interpolation algorithm to extend the horizon Z-values outward using the number of steps (bins), which need to be defined in the following parameter field (Number of steps).
- *Continuous Curvature (GMT)* is a continuous curvature algorithm of interpolation, which is a part of the GMT Plugin of OpendTect. Please check the GMT website for further details. This algorithm only requires the tension parameter (ranges from 0-1), which controls the smoothing. The tension 0 gives minimum curvature type of surface interpolation, while the tension of 1 gives a harmonic surface.
- *Nearest Neighbour (GMT)* is also another interpolation algorithm coming from the GMT Plugin of OpendTect. This algorithm requires the

search radius to be defined. It is mostly useful for a regularly spaced grid data. Please check the GMT website for further details.

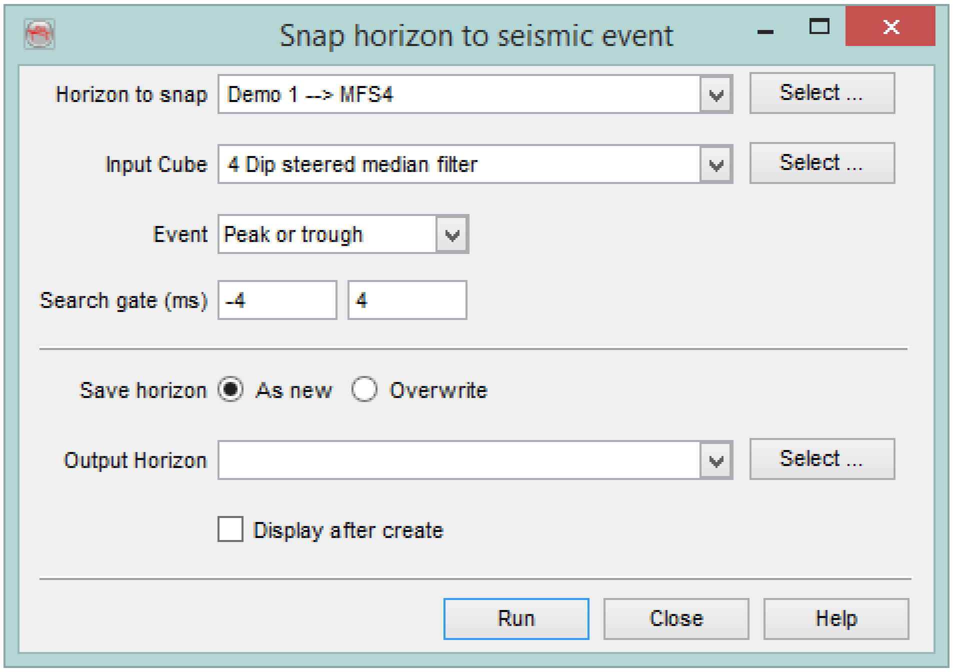
Filter

The "filter" utility enables filtering of the horizon using either median or average filter. The inline and crossline step-out should be defined. The larger the step-out, the smoother the result of the filter.



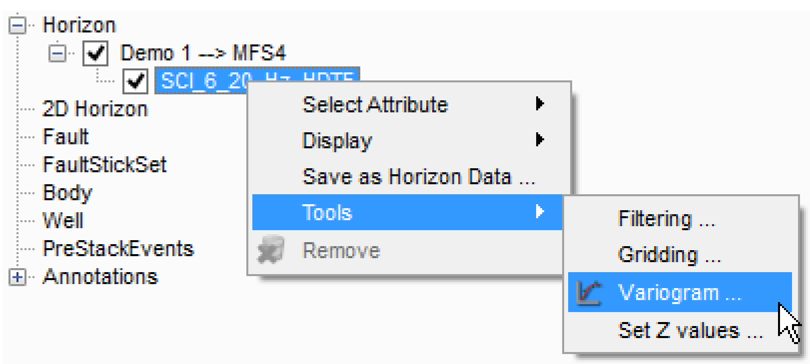
Snap-To Event

In case the horizon is not correctly snapped to a seismic event, this option can be used. The user should define the input data, the event type (peak or trough, zero-crossing etc.), the search gate relative to the original horizon, and whether the snapped horizon should be saved as new or overwrite the original horizon.

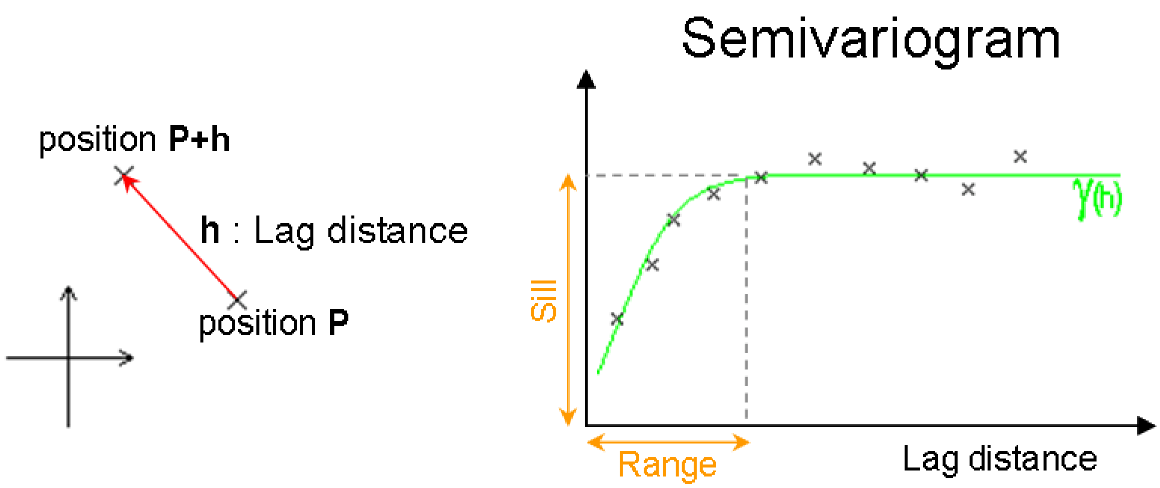


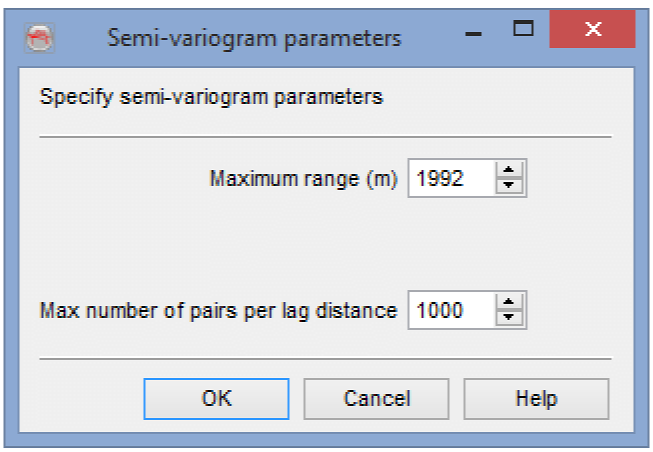
Variogram

For any horizon data displayed on a horizon, a horizontal variogram can be computed:

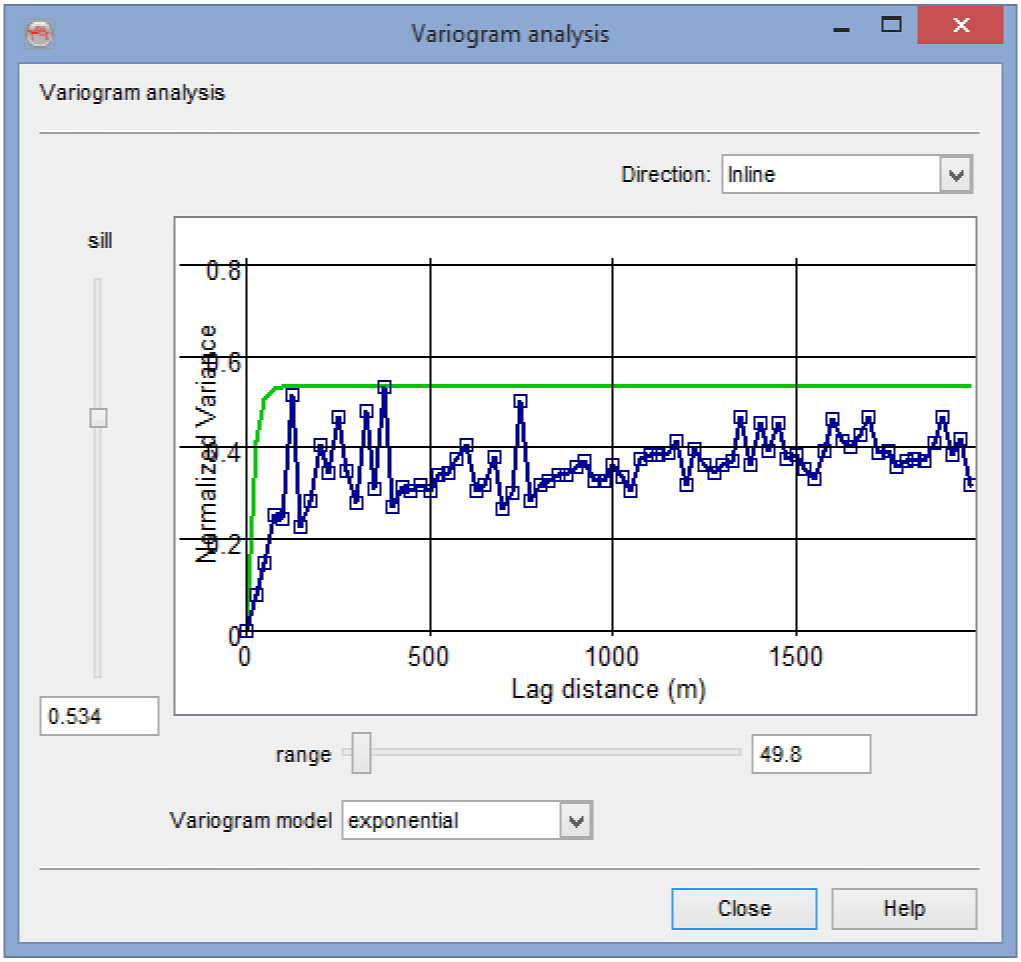


The variogram describes the spatial continuity, here in the horizontal direction but it can also be computed vertically from the crossplot tool. It is commonly represented as a graph that shows the variance in measure with distance between all pairs of sampled locations. Modeling of relationship among sample locations to indicate the variability of the measure with distance of separation is called Semivariogram or Variogram modeling. Variograms are important when doing inversion as it allows to predict a value at a location where it has not been measured.
To compute the variogram, parameters need to be provided: the maximum range (maximum distance allowed between the pairs for the analysis), the step and the minimum pairs per log distance:



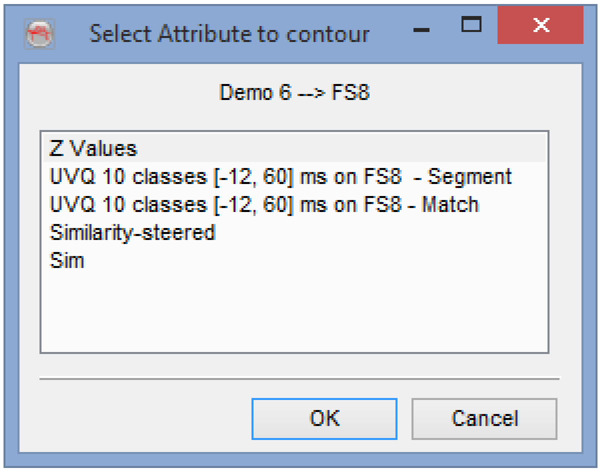


Once the variogram has been created, the analysis consists in finding the model that best fits the measured data in changing the variogram type and changing the sill and range:

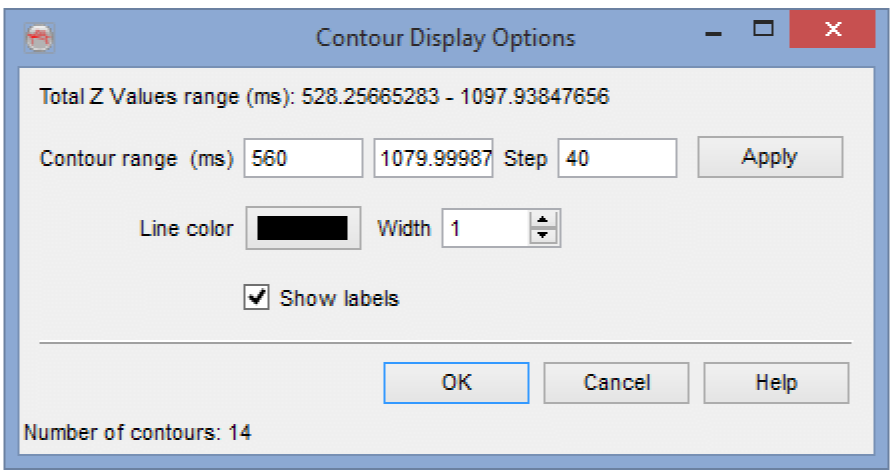


Display Contours

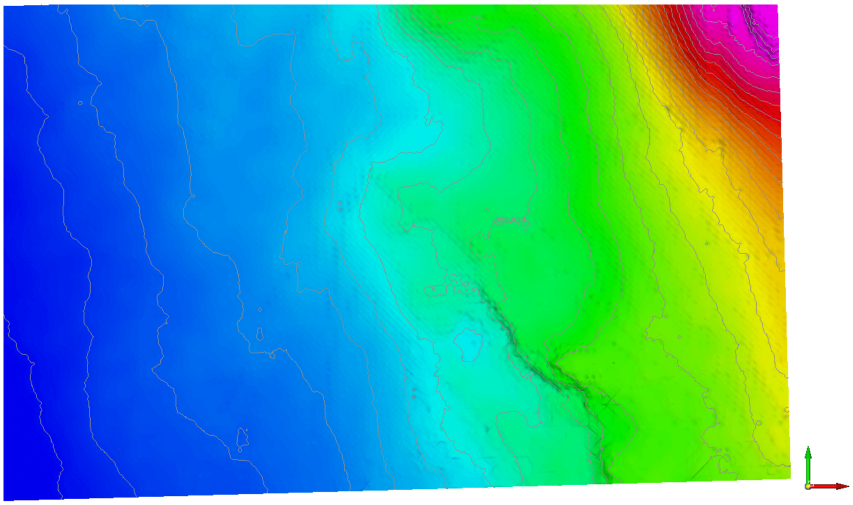
Add Contour Display: This option displays the contour on the horizon. That the contour step (interval) is automatically calculated but can be edited at any time. The input for the contour display can be either a reference Z or any surface attribute like Similarity, Energy, Dip etc ...

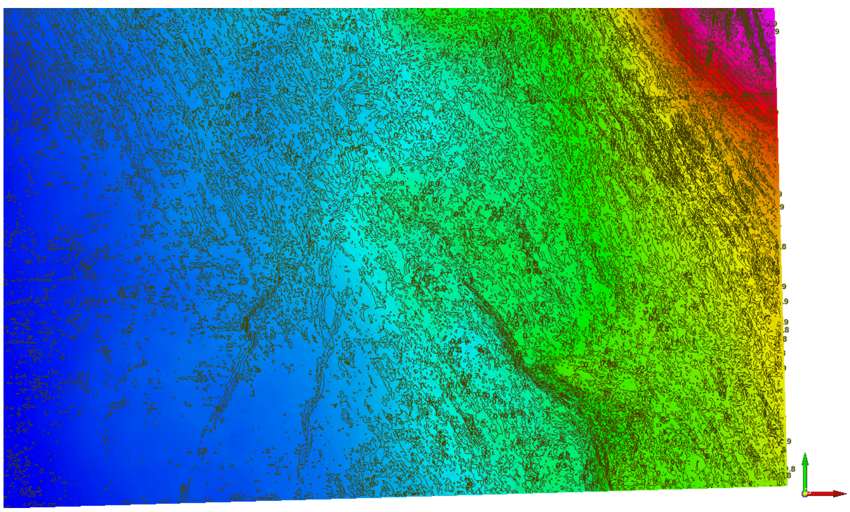


The contours properties can be manipulated by right-clicking on the Contour attribute in the horizon element.
A user can adjust contour range/index, color and line thickness.



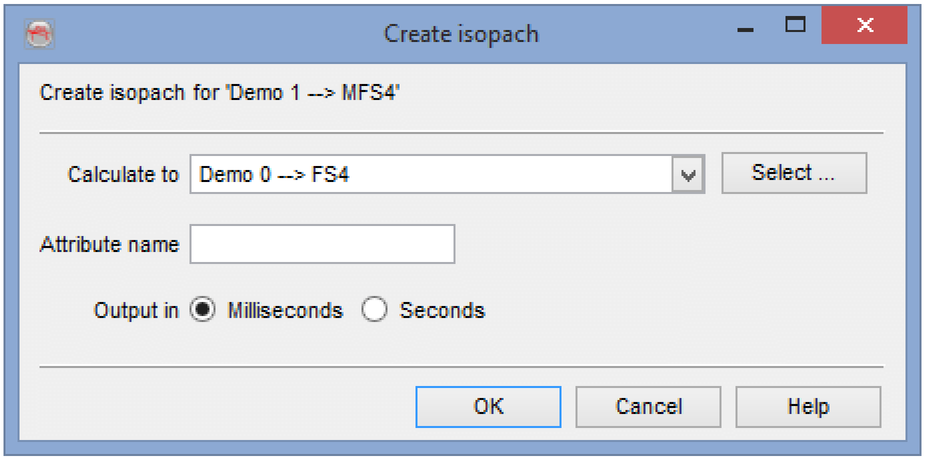
The images below show a horizon with both reference Z and Similarity contours, respect-ively:





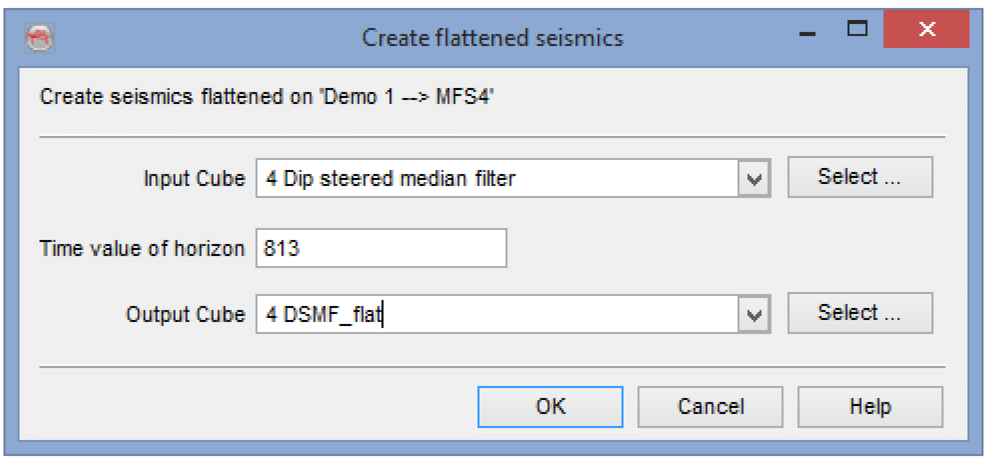
Calculate Isopach

Calculate isopach: This option will compute the time or depth difference between two horizons. The computed grid will be displayed as a new layer on this horizon and may be stored as a surface data. The output will always be in seconds, meters, or feet.

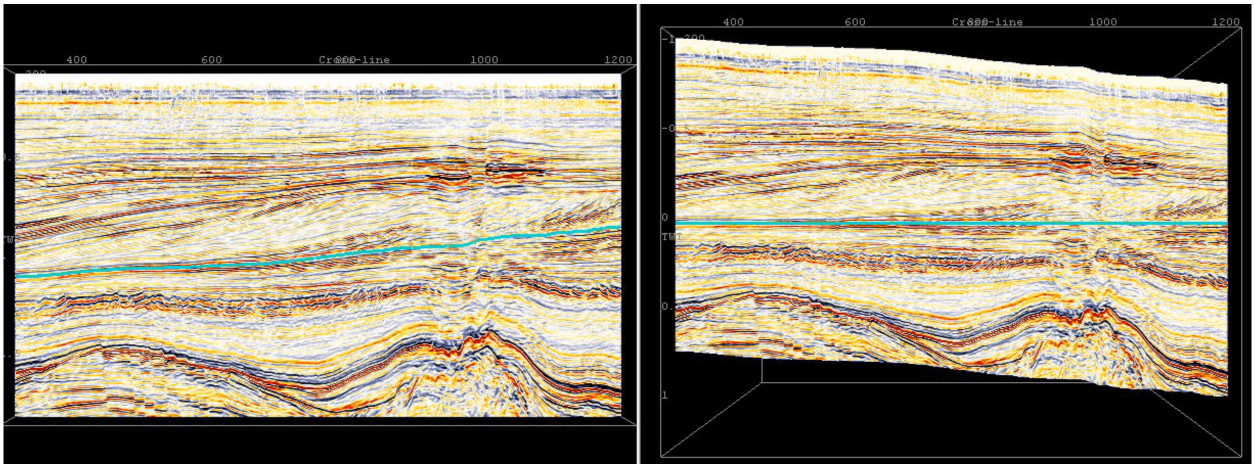


Flattening

Write Flattened Cube: It creates the flattened seismic at specified time value of horizon. The output is stored as a new flattened cube. The user can choose the benefit of this option by flattening the cube at the horizon.



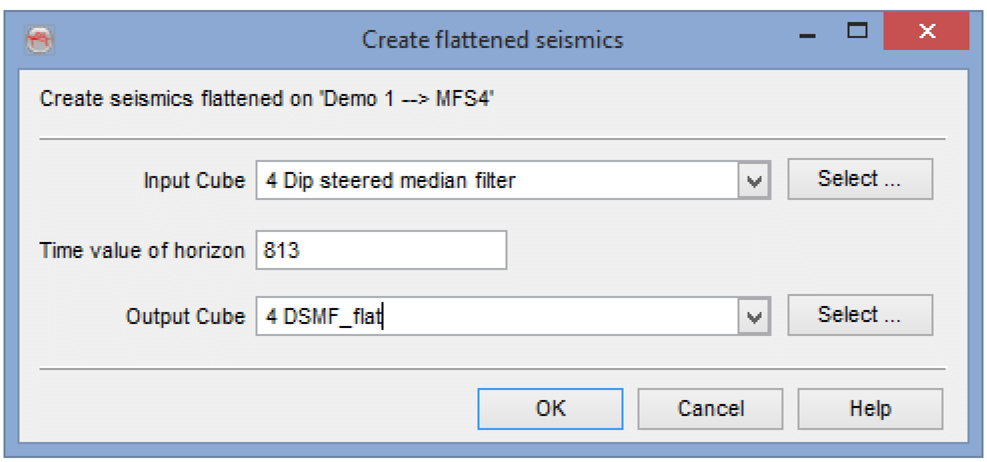
Create flattened scene: This option enables the user to create a second scene in which the data is displayed relative to the flattened horizon. This can be a very useful tool in specific situations. By flattening a horizon, the user gets an idea of the approximate section at the time of the deposition of this horizon. The tectonic history can be derived from the difference between the original section and the "restored" section. Another advantage of flattening the horizon is that it becomes easier to evaluate the depositional environments.



Unflattening the cube: Should you need to unflatten the cube then please refer to the following: Delta Resample Attribute

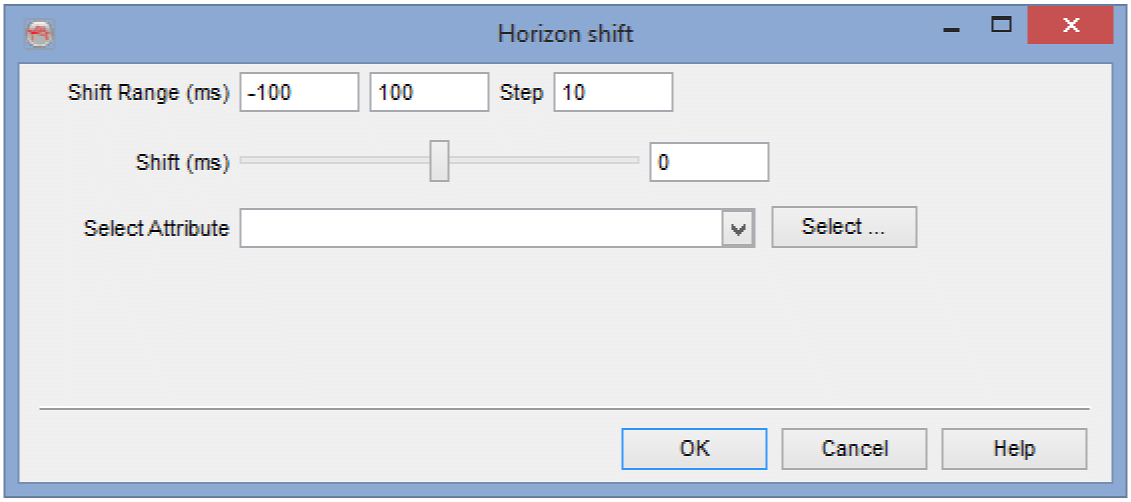
Tracking

The Tracking pop-up menu is used to enable horizon tracking for the selected horizon. To enable horizon tracking for a horizon, a user may select *Start Tracking* option, which will pop-up the *Tracking Setup* dialog and will also activate the tracking toolbar. After this, the user may start horizon tracking for the selected horizon. Please read How to ... Interpret Horizons for further details.



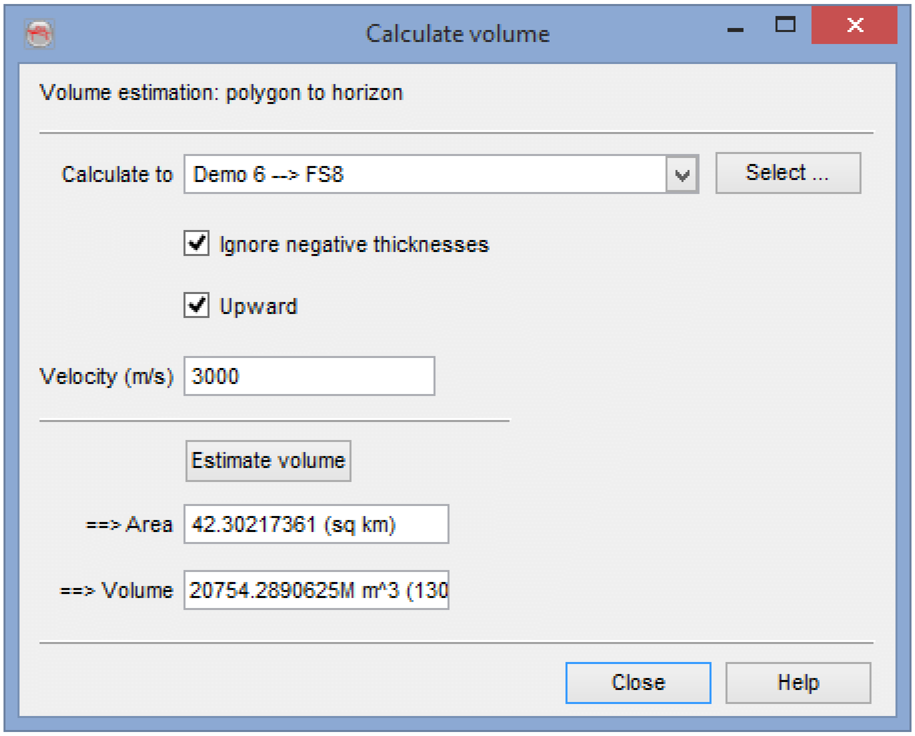
Shift

Shift: The scrollbar allows the user to scroll the 3D horizon vertically. The shift range allows the user to define the upper and lower boundaries of the scrollbar range. The step size defines the distance between each possible horizon position. (e.g. A range of -100 to +100 with a step of 10 allows for the user to scroll through 20 possible horizon positions, centered about the original position.) Different attributes can be calculated for the horizon in this user defined shift range. The user can then use the scrollbar to move up and down and view the attribute as it would appear on that horizon at the various shift positions. This shifted horizon can be saved as surface data to be viewed later.



Calculate Volume

Calculate volume: It is used to calculate the volume between the two horizons. The volume is calculated within an existing polygon. Select the polygon and press *Estimate Volume* button to calculate the volume within the polygon. To read more about this, please go to the chapter Pickset: Pop-up Menus



Other Options

Properties: The *Material* window allows changing of the graphical settings like transparency, line style, and thickness.

Resolution: The resolution of a horizon can be changed for performance reasons. When using a low-end graphics card, performance during rotating and moving the scene may suffer. By showing the horizons in a lower resolution, scrolling and rotating becomes smoother. By default, the resolution is set to *Automatic*. This setting uses higher resolution in areas where a horizon has a complicated shape, and low resolution in relatively flat areas. Also, when rotating, the resolution will be set to low in order to enhance responsiveness of the rotation action. When released again, the resolution is set higher again. Attributes displayed will always have full color resolution, only the shape of the horizon surface is affected by this setting.

> **Tip:** Horizon default resolution and colortable settings can now be defined under the '[Horizons]' tab via *Utilities--> Settings--> Look and Feel....*

Quick UVQ: This option is related the Neural Network plugin license, if it is available. It is used to create a quick unsupervised facies map. For further information please refer to the plugin documentation.

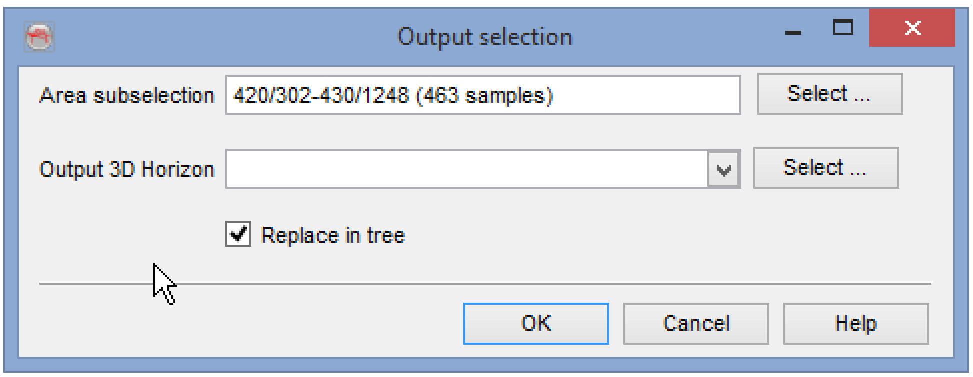
Use single color: When this option is selected, the horizon is displayed in a single color, which can be chosen from a standard color selection window.

Display: The horizon can be displayed on the sections (inline/crossline/2Dline/timeslice) as a line, as a 3D surface or both.

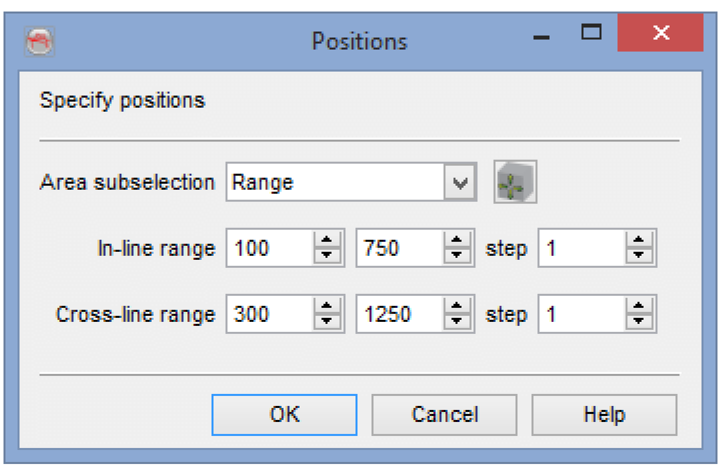
Tracking: Horizons can be edited and tracked through the survey. The various tracking options are described in here.

Save: The save option gets highlighted when changes are made to the surface geometry. *Save* saves the new geometry of the horizon. If a horizon consists of patches, you can save a sub-selection of these patches.

Save as: Save a sub-area or the complete horizon using an other name.



Position: It is used to re-position (selected inline/crossline range) the displayed horizon. In the position dialog, set the ranges of the inline or crossline to sub-select the horizon display.

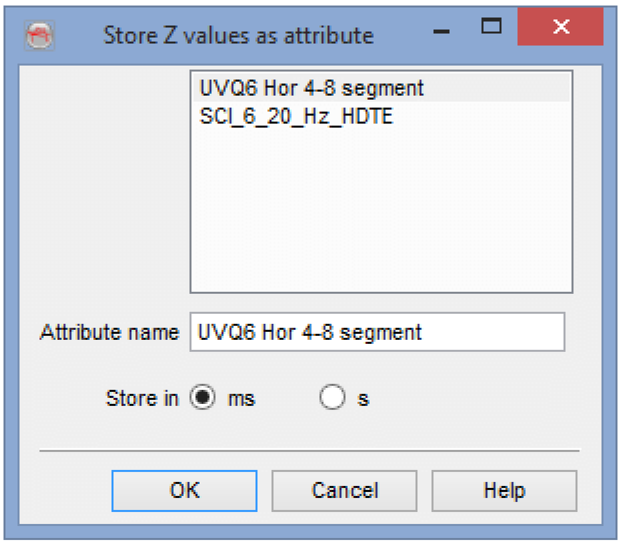


Lock: This will lock the selected object. It prevents accidental removing, moving, or displaying data on the object. After clicking unlock, all manipulations are possible again.

Remove: This option removes the horizon from the tree and the graphics area.

Store Z as Attribute

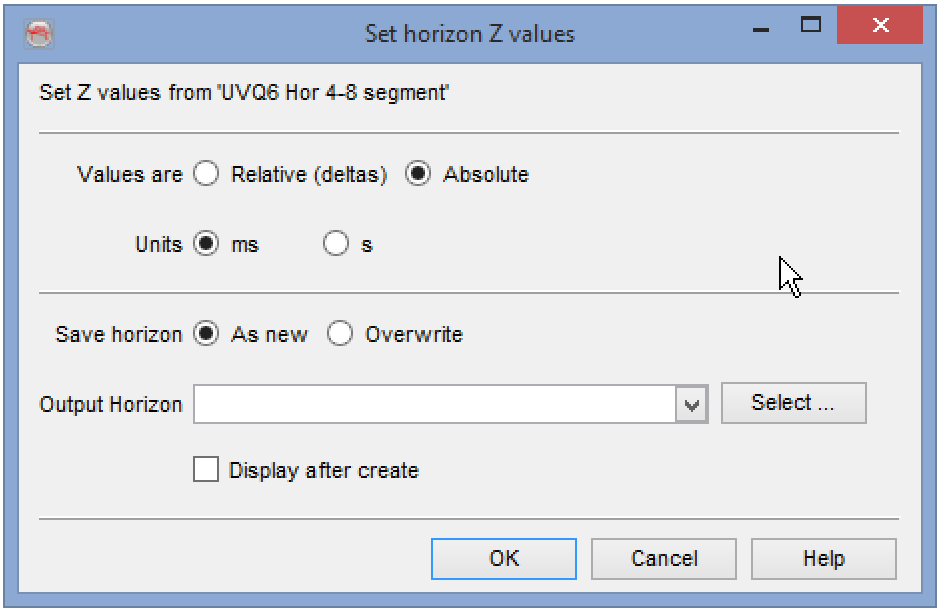
This option gives the possibility to store 'Z' values as a Horizon data for an horizon. Subsequently, this newly created attribute can be used to change 'Z' values of another horizon by means of Set Z values.



The name of the new 'Z' attribute and in which units it will be saved need to be specified.

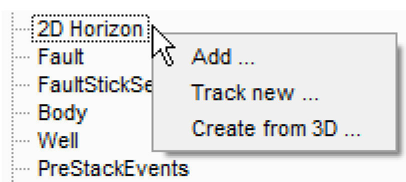
Set Z Values

A 'Z' value surface attribute (see Store Z as attribute) can be used to shift a horizon or completely change its 'Z' positions using the *Set Z values* option.



Specifying values as *Relative (deltas)* will shift the horizon; infact the software adds the attributes 'Z' values to the 'Z' values of horizon to achieve this shift. *Absolute* is used while completely changing the 'Z' values of the horizon to the 'Z' values of the surface attribute. Specification of units of 'Z' values (i.e. in 'milliseconds' or 'seconds') is also required.

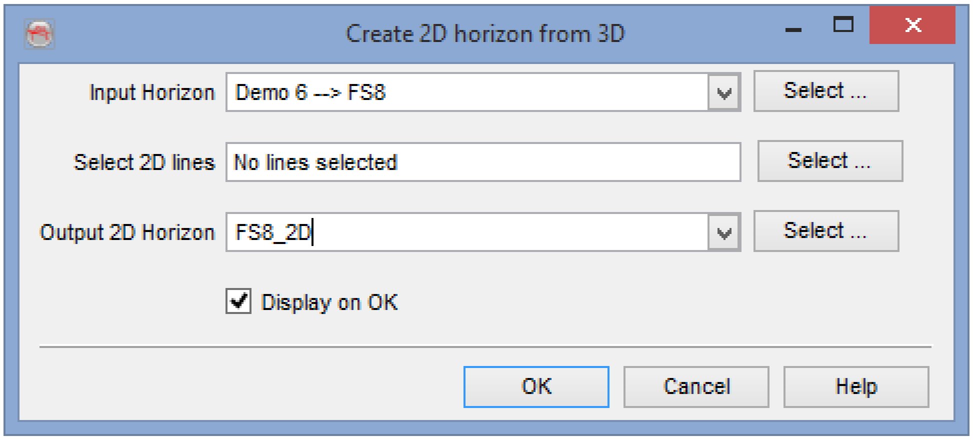
2D Horizon



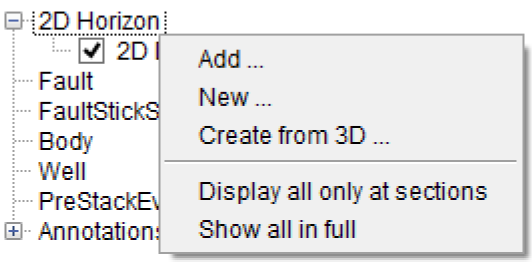
Add... an existing 2D horizon into the scene.

Track new... Allows a new 2D horizon to be created.

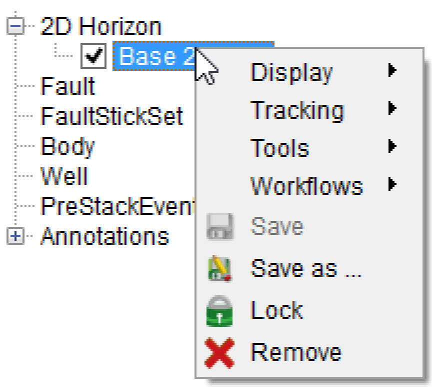
Create from 3D... Generate 2D horizons from existing 3D horizons by right-clicking on *2D Horizon* in the tree and selecting Create from 3D. A window pops up where you can designate the 3D horizon, the 2D data set and 2D line(s) where you want to create the new 2D horizon. This function allows for the 2D horizon to be created on one line, or on as many lines as a data set contains, all at once.



Once you have selected a 2D horizon, two other options become available:

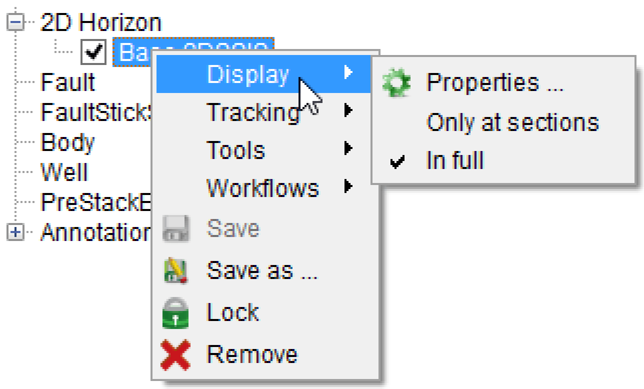


The selected horizon(s) will be displayed in the scene. To start a new 2D horizon interpretation, read the chapter How to interpret Horizons.
The displayed 2D horizon pop-up menu contains the following items:



These options are described in the following sub-sections.

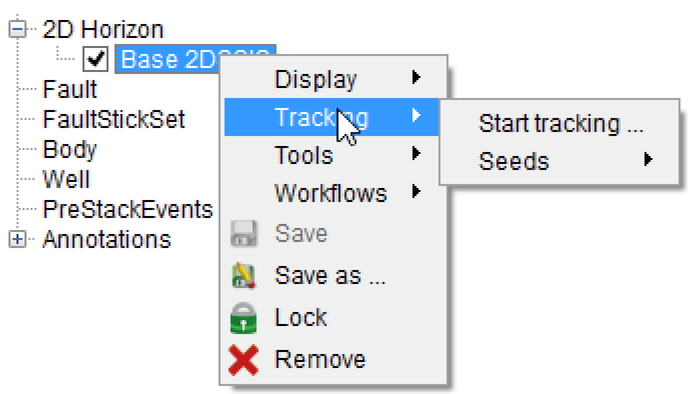
Display



Properties: Change the display settings for a horizon (color, reflectivity, line style).

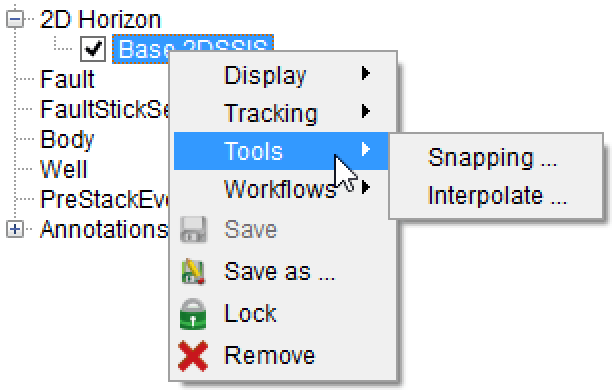
Only at sections: Display the tracking horizon at section. This is especially usefull for QC purposes to check if the tracked horizon lies on the expected reflector. Can be toggled back to **In full**

Tracking

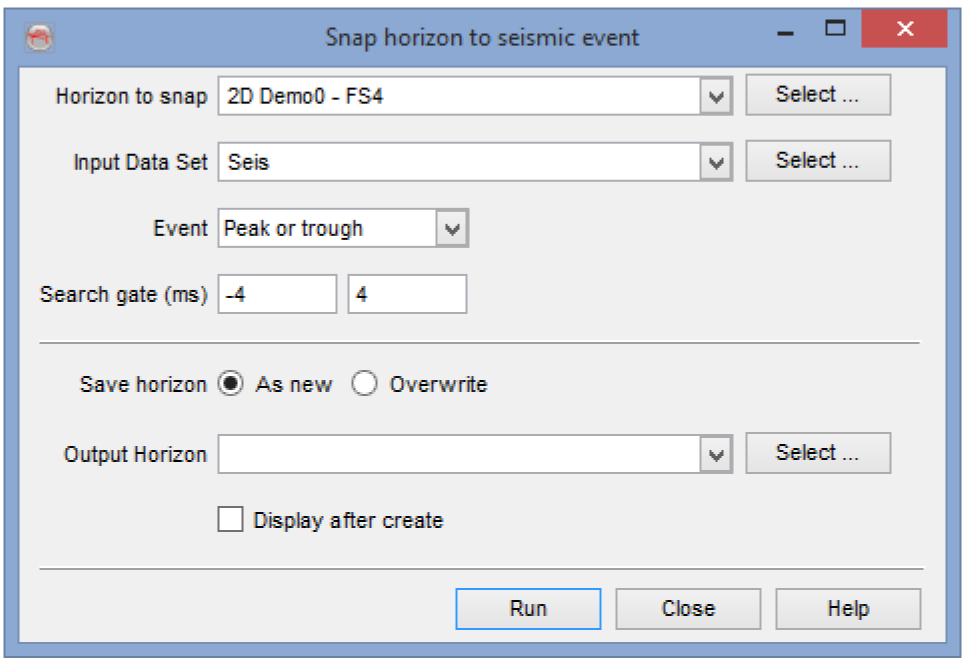


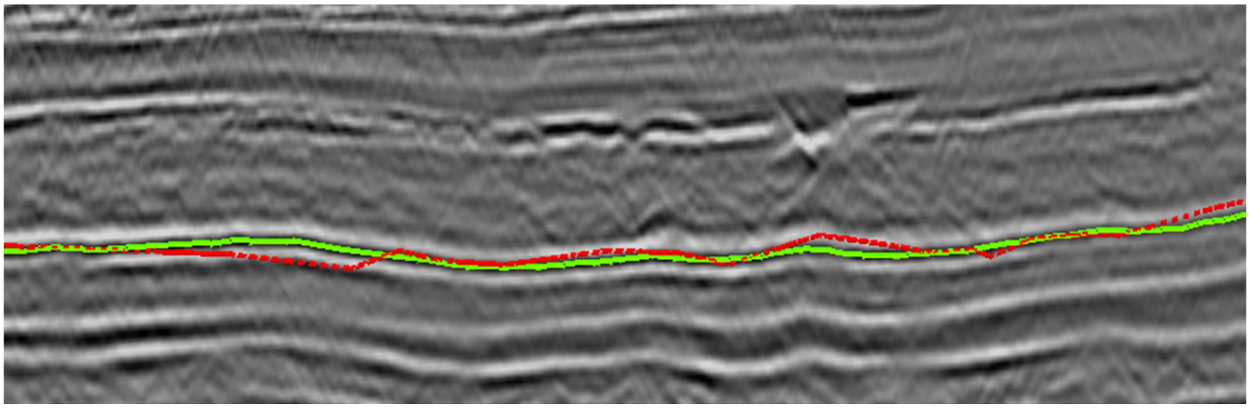
A 2D horizon displayed in the Scene can be re-interpreted by enabling the tracking option. Once it is enabled, it will launch the Tracking Setup window to start 2D horizon interpretation on 2D lines. It will also enable the tracking toolbar. Furthermore, the seed properties (e.g. color/size) can also be changed using this option.

Tools



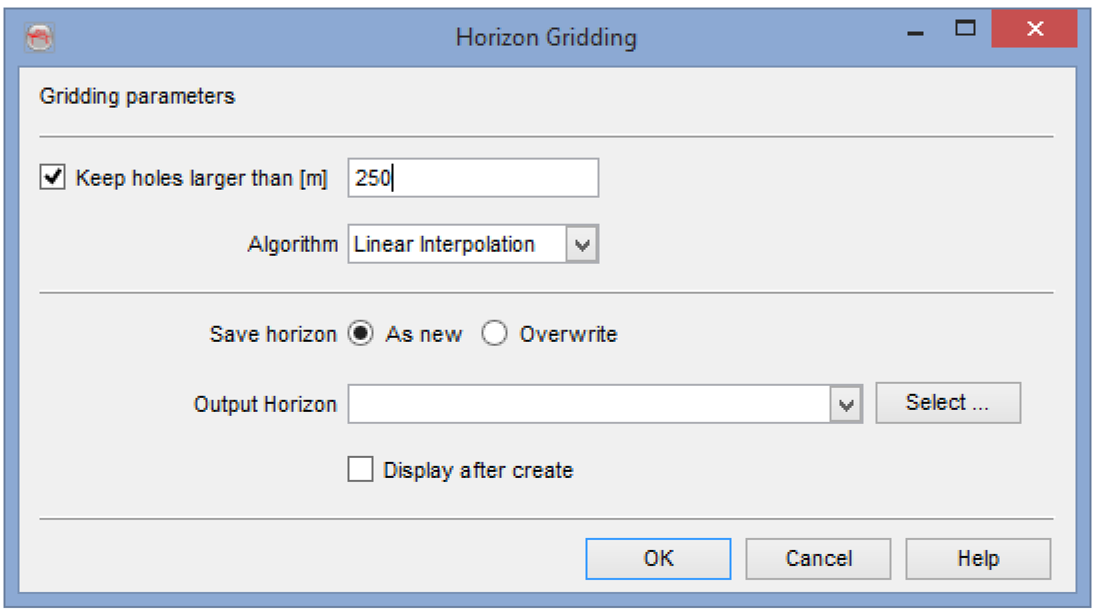
Snapping: This option allows for the selected 2D horizon to be 'snapped' to the nearest event defined in the Event option (see below).





Horizon before snapping(red), after snapping (green)

Interpolate: This is a 2D gridding option for horizon interpolation by filling the gaps/holes in interpretation. If this option is selected for a horizon, it will pop-up the following *Horizon Gridding* dialog box.

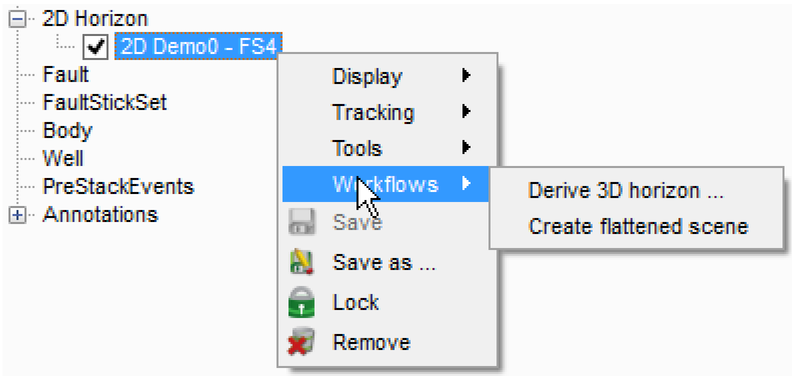


Keep holes larger than: By checking this box, the gridding interpolation area can be defined i.e. by defining a threshold e.g. 2500m. By setting this value, the gridding is restricted and gaps/holes up to a radius of 2500 meters will be filled.

Algorithm: Interpolation algorithms for 2D horizon(s). Currently, OpendTect supports linear and polynomial types of interpolation.

Output horizon: Overwrite or create a new horizon from the selected horizon
.

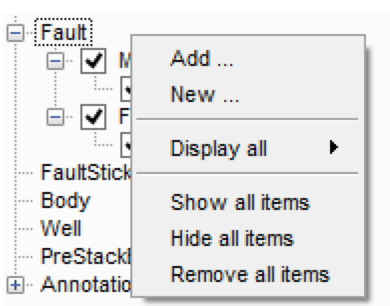
Workflows



Derive 3D horizon: Create a 3D-horizon from a 2D horizon. As soon as a 2D horizon is tracked, a 3D horizon can be derived by right-clicking the 2D horizon in the tree and choose *Derive 3D horizon*. A window pops up in which you can select the algorithm (inverse distance interpolation or triangulation), shown in images below. The results can be displayed immediately by selecting the *Display after generation* option.

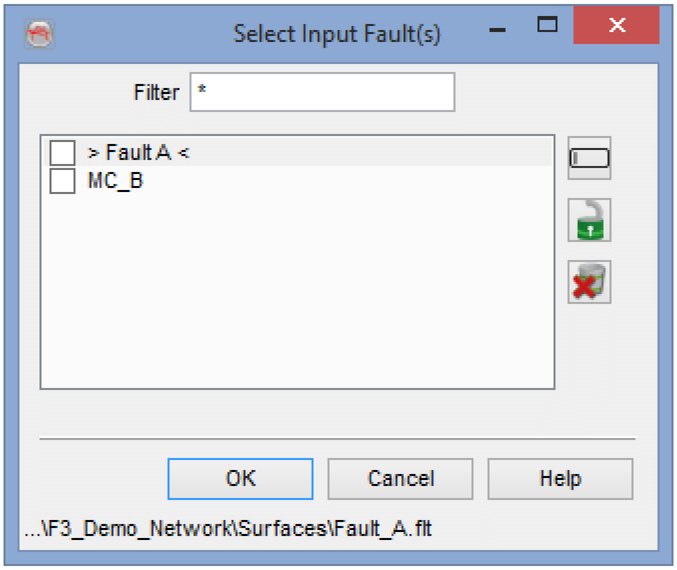
Create Flattened Scene: The 2D line(s) can also be flattened using this option. Once clicked, it will create a new flattened scene based on the selected 2D Horizon. (see also: Flattened Horizon Scenes)

Fault



The fault option enables interpretation of either a new fault or loading an existing one.

Add: Adds selected faults into the tree and displays them in the scene:



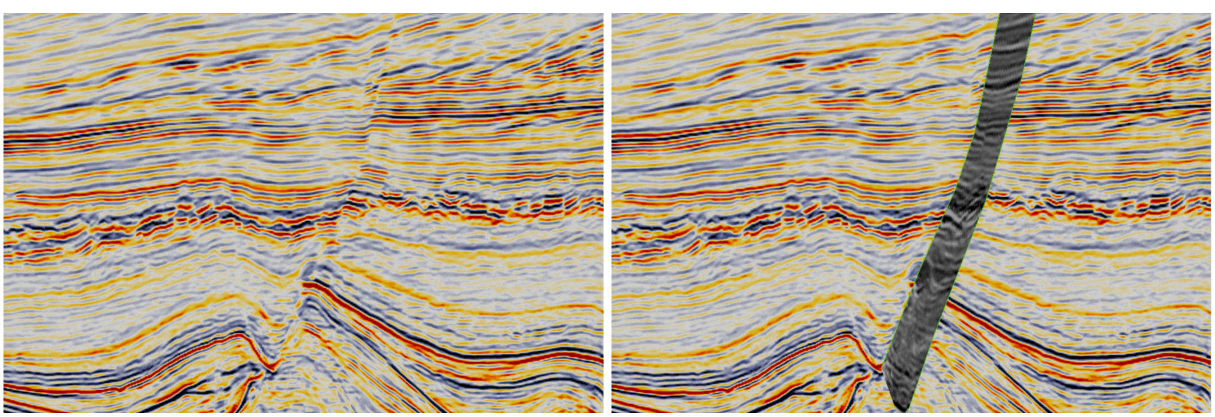
New: Adds an empty fault in the scene (New fault 1) that needs to be named and saved once the interpretation is completed.

Display all: If more than one faults have already been displayed or added in the tree, this option will be available. It is used to display all faults in full, only at sections, or at horizons, or both. It is also used to toggle On/Off the fault plane, sticks and both displays.

Show all items: It is used to check all items, which means that all items would be displayed in the scene.

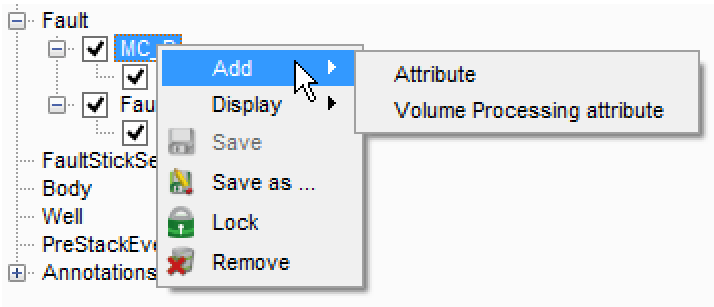
Hide all items: It is used to hide all (check out) the displayed faults.

Remove all items: To remove all faults that are added in the tree, this option is used.



An example of a picked fault line on a seismic section:

Once a fault has been added, right-clicking will pop-out the following menus :



Add:

Attribute: Add a new attribute for fault element. Right click and choose 'Select attribute' to select the desired seismic volume. The attribute will be displayed along fault planes. The example line with interpreted faults in a 3D volume has been shown below. Note that the faults have seismic data displayed as an attribute along their planes.
Volume Processing Attribute: It is used to add a special sub layer to the fault that belongs to volume processing attribute. To read more about this, please go to the Volume Builder Setup chapter.

Display:

Histograms: It shows multiple histograms of the displayed data along the selected fault plane.
Only at sections: It is a toggle that is used to display a fault plane on a section as a stick.
Only at horizons: To display a fault plane on a horizon as a fault trace, this option could be toggled On/Off.
Fault planes: If a fault has been displayed either on a section or a horizon, it can be back into a 3D fault plane. This option toggles On a fault plane display.
Fault sticks: To see the fault sticks only in 3D, this option should be toggled on.
Use single color: It sets a single color to a fault plane display. Any displayed attribute along the fault plane will become hidden an only the fault color would be displayed.
Properties: Set the Type, Size, and Color of the pick markers on the graphics area.

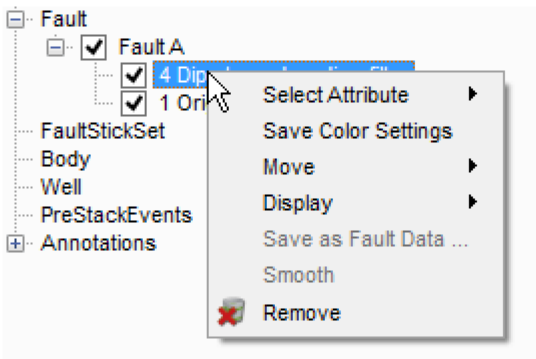
Save: It saves the selected fault.

Save As: To save the selected fault with a new name, this option is used.

Lock: Lock the selected object. Prevents accidental removing, moving, or displaying data on object. After clicking on *Lock* again (i.e unlocking), editing is enabled.

Remove: It removes the selected fault from the scene.

The sub-menus for the displayed attributes are explained below:



Select Attribute: It is used to select and display various types of data(see below).

> **Stored Cubes:** Any stored volume could be displayed along the fault plane in 3D.
> **Attributes:** Any attribute defined in the Attribute set window could be displayed. This requires a pre-defined attribute set in the Attribute set window. It will be inactive if no attribute is defined in that window.
> **SteeringCubes:** If a SteeringCube has already been pre-processed it can be selected and displayed along the fault plane.

Save Color Settings: The active color table could be stored permanently or updated for the displayed stored attribute. For instance, if you do not like the color bar for a particular seismic data (say PSTM) that is Red-white-blue (color table) and you want to change it into Magic, you could set it here. It will save the colour settings for this specific stored volume (PSTM).

Move: To change the display level of an attribute, it can either be moved up / down or placed to top / bottom.

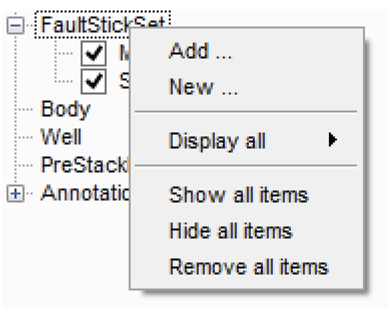
Display: To make a fault semi-transparent, the transparency is used. One can also visualize the histogram of the attribute.

Remove: It remove the selected attribute from the tree.

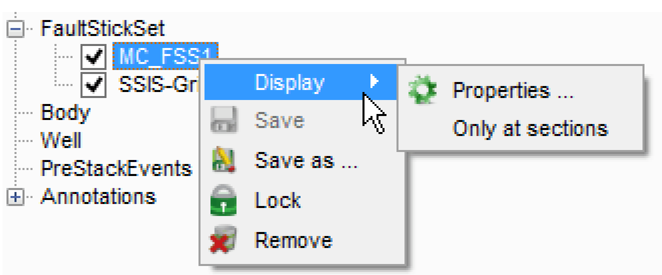
Note: For Fault interpretation, please see the interpret faults chapter

FaultStickSet

A *FaultStickSet* is a set of sticks for faults interpretation. Sticks are segments that are created by connecting two or more nodes.
The *FaultStickSet* tree item allows the user to create a new *FaultStickSet* or to load an existing one.



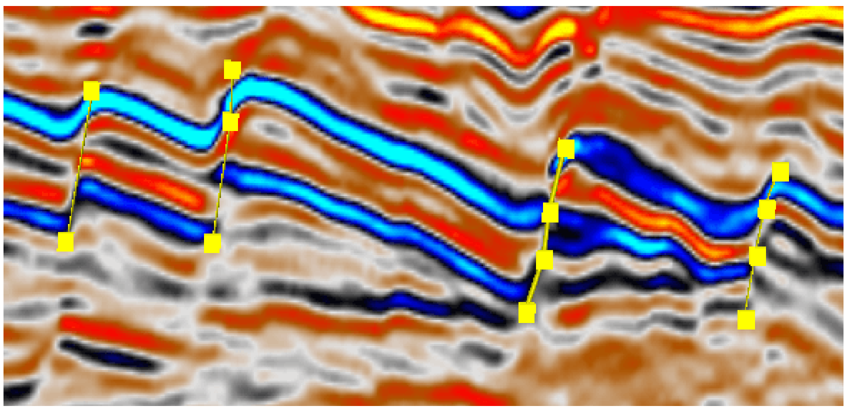
The new FaultStickSet is inserted by selecting the *New* option in the tree. The blank fault *New sticks 1* will be inserted as sub-element of FaultStickSet.
The user can then interpret the fault sticks on inline/crossline/Zslice and/or on 2D lines as well.



In order to create and edit a faultstickset, check first that the faultstisckset is active in the tree, then do the following:

1. Click along the fault to create your first fault stick for one specific section .
2. The second fault stick in the same section is created by *shift + leftclick* for the first pick then just *leftclick* for the next faultpick(s)
3. To remove a fault stick node, *Ctrl+leftclick* on the already picked nodes.
4. Once you are done with one section, move to another inline/crossline/timeslice/ or 2D line to create new fault sticks. A simple click will start the fault stick creation.
5. If you want to edit one stick while being busy with another, just click on one of it nodes to make it active. While editing, you can click and drag a node to another position.

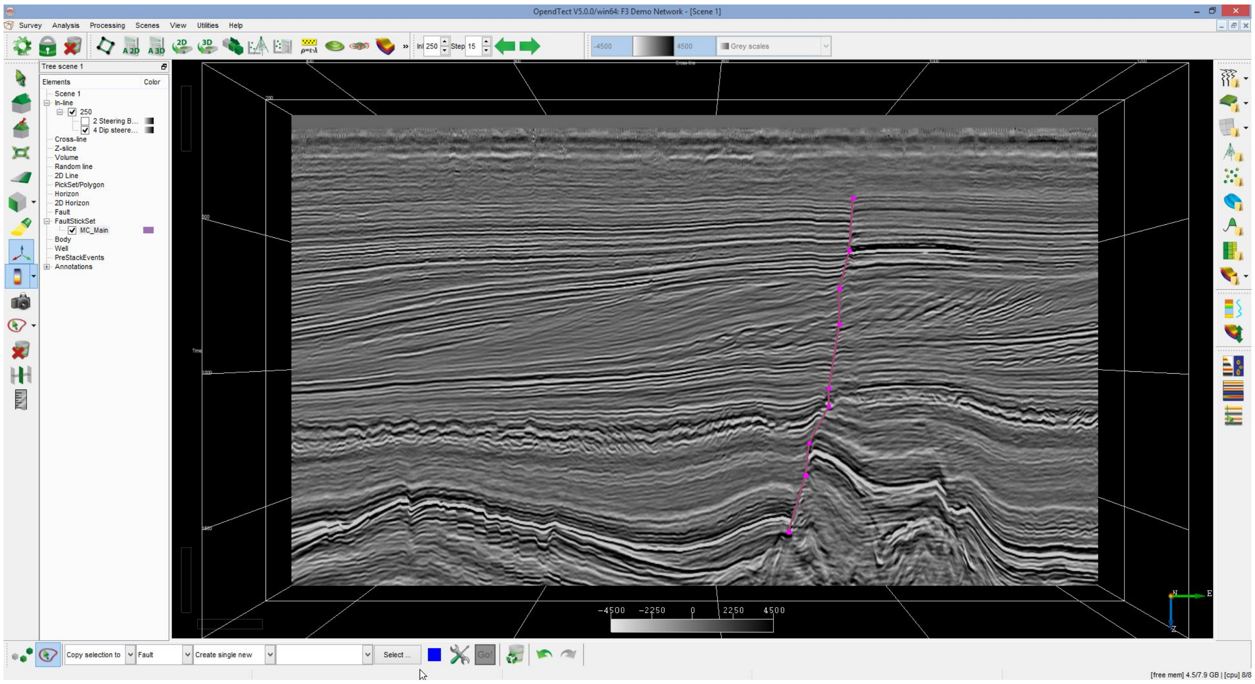
After interpreting the FaultStickSet, use the option *Save* to save your set with an appropriate name.



An example of a picked FaultStickSet with nodes, the active stick is the second from right (the node connecting line is thicker)

FaultStickSet to Fault

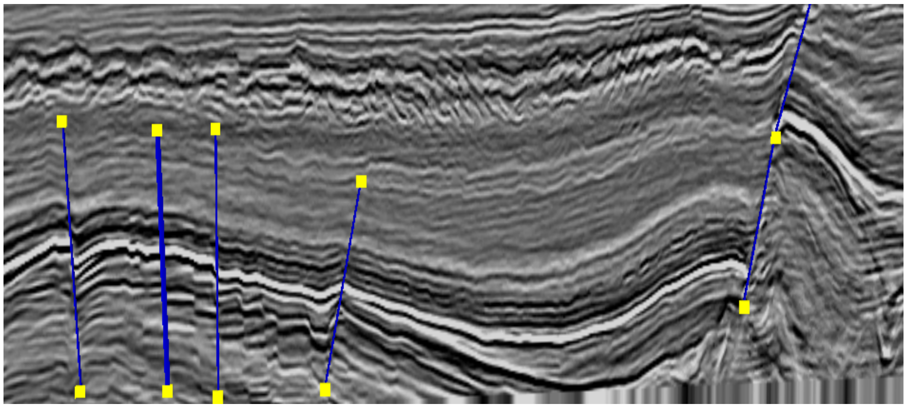
In OpendTect, newly interpreted faultsticksets (or a selection) can be transformed into 3D faults and vice versa, from 3D faults the user can output faultsticks.



The conversion between faults and faultsticks is done in a special toolbar as shown below:

In the toolbar, they are two modes: The **Edit mode** and the **Selection mode**:

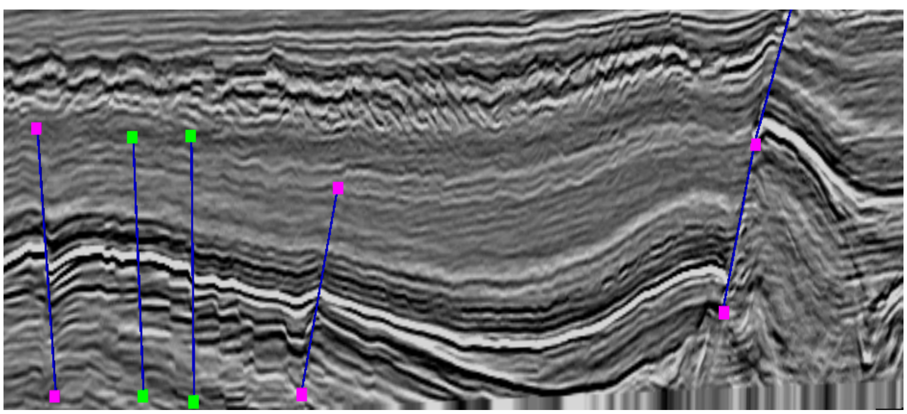
> **Edit Mode:** In this mode, nodes are yellow, the user can add nodes (click), remove nodes (Ctrl+click). Nodes can be dragged from one location to another. New sticks are created by Shift+click for the first node then just click for other sticks.
> When using Space+click, this will duplicate the node(s) and new sticks can be added to the user-defined direction.



Nodes are yellow in *Edit Mode*

Selection mode: When this mode is active, Faults/Faultsticks are selected, copied (or moved) to new or already existed faults/sticks group. The outputs are: New group, merge to existing one, replace (overwrite) the already existed group.

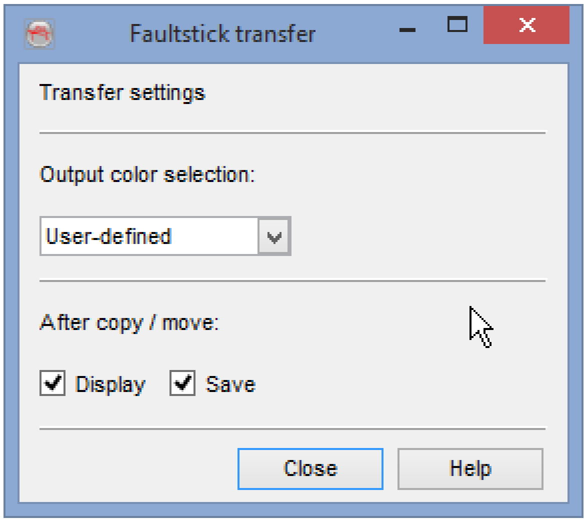
Creating new group of faults/sticks in series will automatically rename group by adding a suffix to the first new group name provided by the user. If the name is e.g Fault-Area then automatically generated names will be like, Fault-Area_1, Fault-Area_2, Fault-Area_3 etc ....



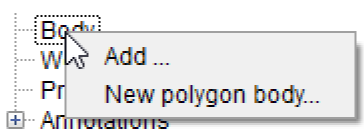
In *Selection Mode*, Nodes are green (selected) and pink (unselected)

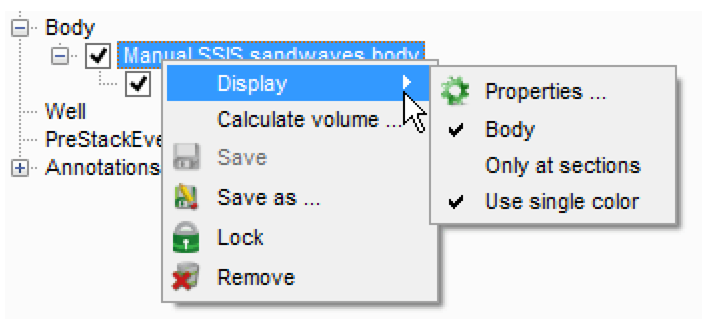
Note: When converting faultsticks into faults, please keep in mind that OpendTect doesn't support the files that contain (1) Crossing fault sticks, (2) Fault sticks interpreted on vertical (e.g. inline) as well as horizontal (e.g. Z slice) planes. If the input file contains such type of stick sorting, you might encounter problem in OpendTect to get a regular fault plane.

Clicking the  icon allows you to set the transfer (or conversion) settings which will be applied after the copy or move is put into action:



Body





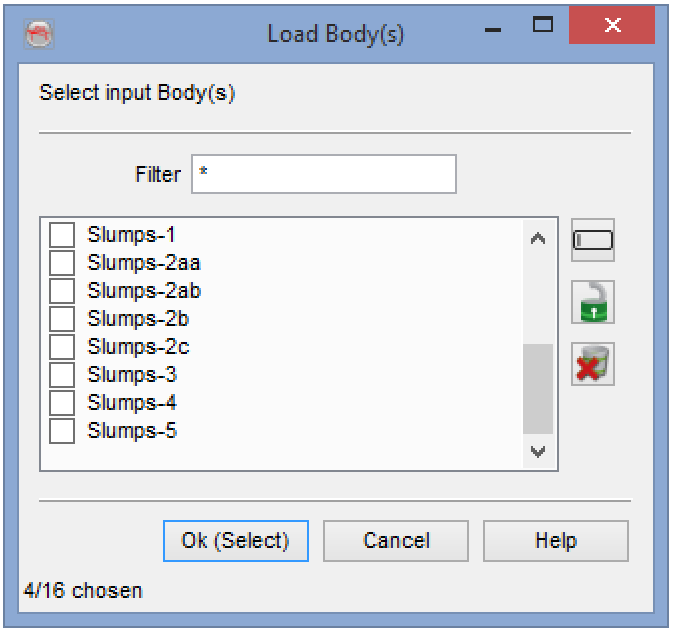
Bodies are displayed and created from this tree item. Using the option "New polygon body" the bodies can be drawn by picking on vertical and horizontal slices. The body will always be the convex envelope around the picked locations.

It is also possible to create bodies from:

- An isovalue surface (implicit representation): The body is extracted from a volume based on the amplitude distribution.
- A polygon projected between two horizons.

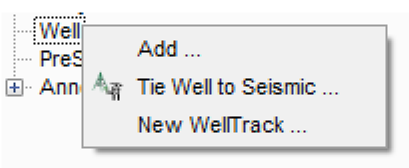
Bodies may be used for display but also the creation of volumes using the volume builder: The inner and/or outer parts of the body are filled with constant value(s).

Stored bodies may be displayed in the scene by using *Add...*



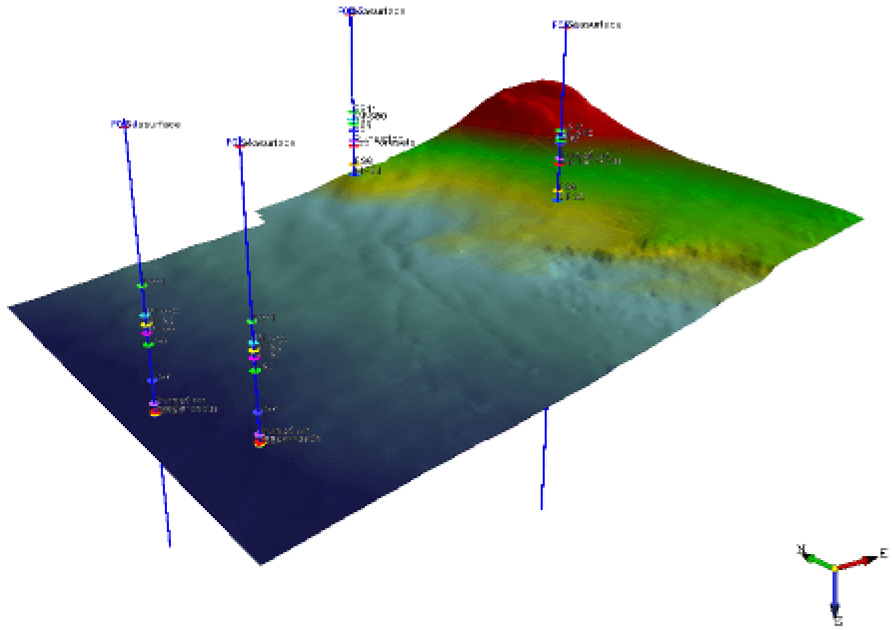
Well

Clicking the well-element in the pops up a menu with 3 options: *Add*, *Tie Well to Seismic* and *New WellTrack*



Basic Well Pop-up Menus

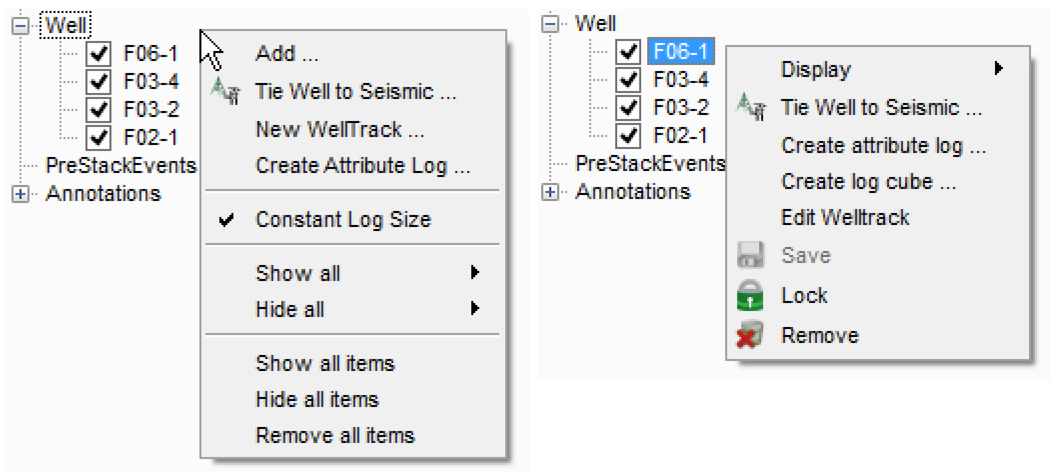
Add: Wells are added and displayed in the scene using Add option.



Tie Well to Seismic: Access the seismic to well tie module. Generally, three parameters are needed for a successful well-seismic tie: sonic/velocity log, density log and a reference wavelet. The wavelet can be either imported or extracted in OpendTect. Logs can also be created in the Well Manager.

New Well Track: Create new well tracks interactively in the 3D scene. After selecting this option the system will prompt for a well track name. After specifying the well track name, display an element (inline/crossline/2D line) in the scene. Drawing the well track on the selected element is enabled. After drawing the well track, right click on the well track name and select the *Save* option. Note that drawing a new well track works similarly to editing a existing well track. Well track-nodes can be picked on the active elements displayed in the scene. Also note that a display with a Z-scale (View - Z-scale) other than 1 distorts the appearance of distance in the 3D view.

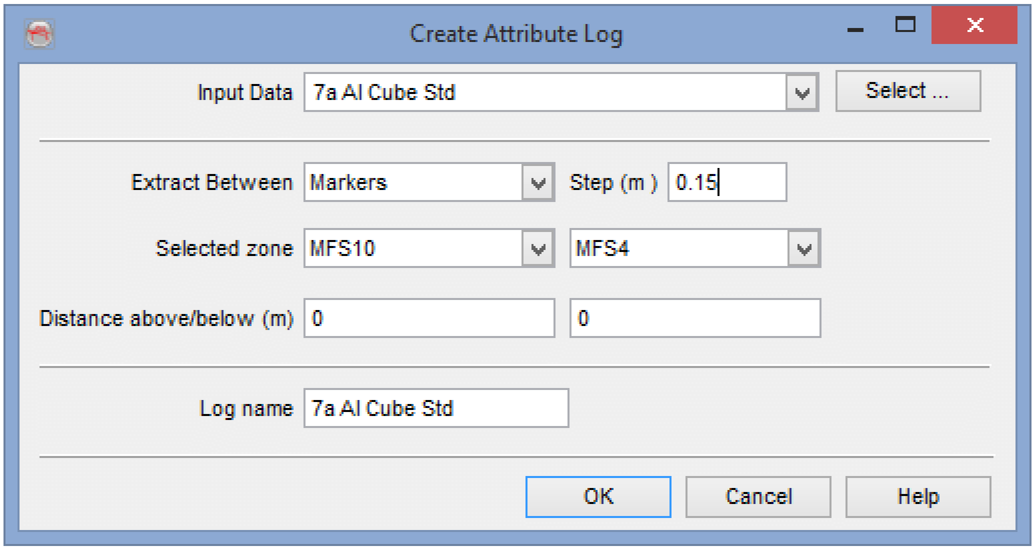
After loading new wells, items are added to the right-click menu as follows:



Well popup menu (multiple) and menu for individual wells.

Multiple Well Options
These options are available only when more than one well is loaded in the tree, and can be accessed by right-clicking Well in the tree. The new options available when multiple wells are loaded. Items described in the previous sections above will not be described again here.

Create Attribute Log: creates selected seismic data as a log for multi-wells.



Constant Log Size: keeps a well log display width relative to a scene zoom ratio i.e. a log display width increases with the zoom in and vice versa. However, this option can be toggled off by clicking on the sub menu item (Basic Well Pop-up Menus). In the later case, a log display width is adjusted opposite to the zoom i.e. if a scene is zoomed in, a log display width is reduced relative to the scene zoomed in ratio and vice versa.

Show all: allows the user to toggle on all well names (top),well names (bottom), markers, marker names, and logs.

Hide all: allows the user to toggle off all well names (top),well names (bottom), markers, marker names, and logs.

Show all items: allows the user to toggle on all wells currently loaded and visible in the tree.

Hide all items: allows the user to toggle off all wells currently loaded and visible in the tree.

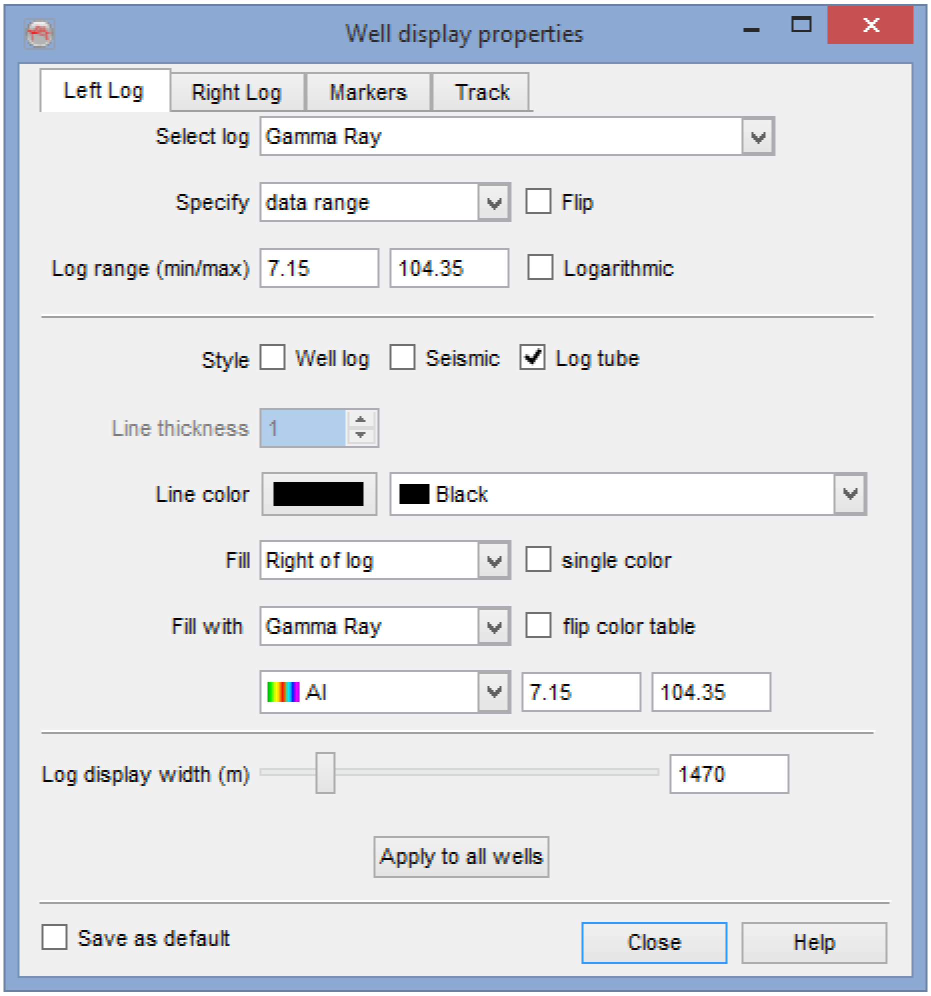
Remove all items: allows the user to remove all wells currently loaded and visible in the tree. This only removes the wells from the scene, it does not delete them from the disk.

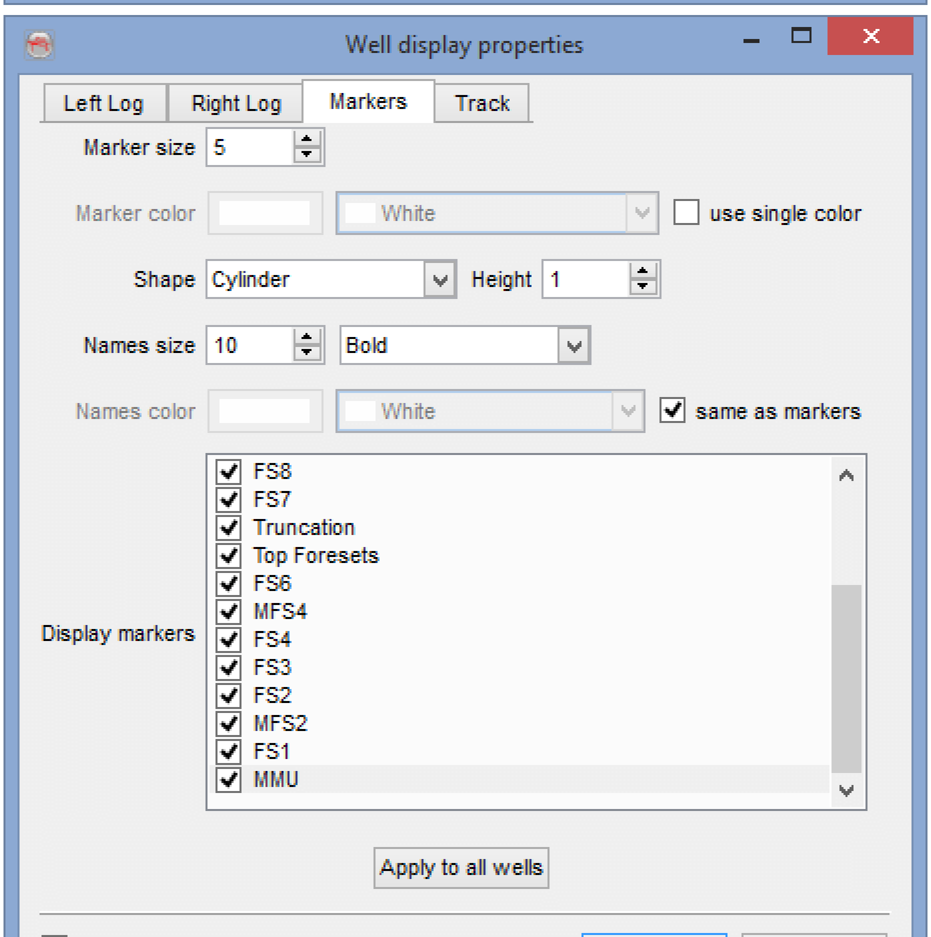
Individual Well Options
Once a well has been loaded into the scene and is visible in the tree, right-clicking an individual well pops-up a window with the following options:

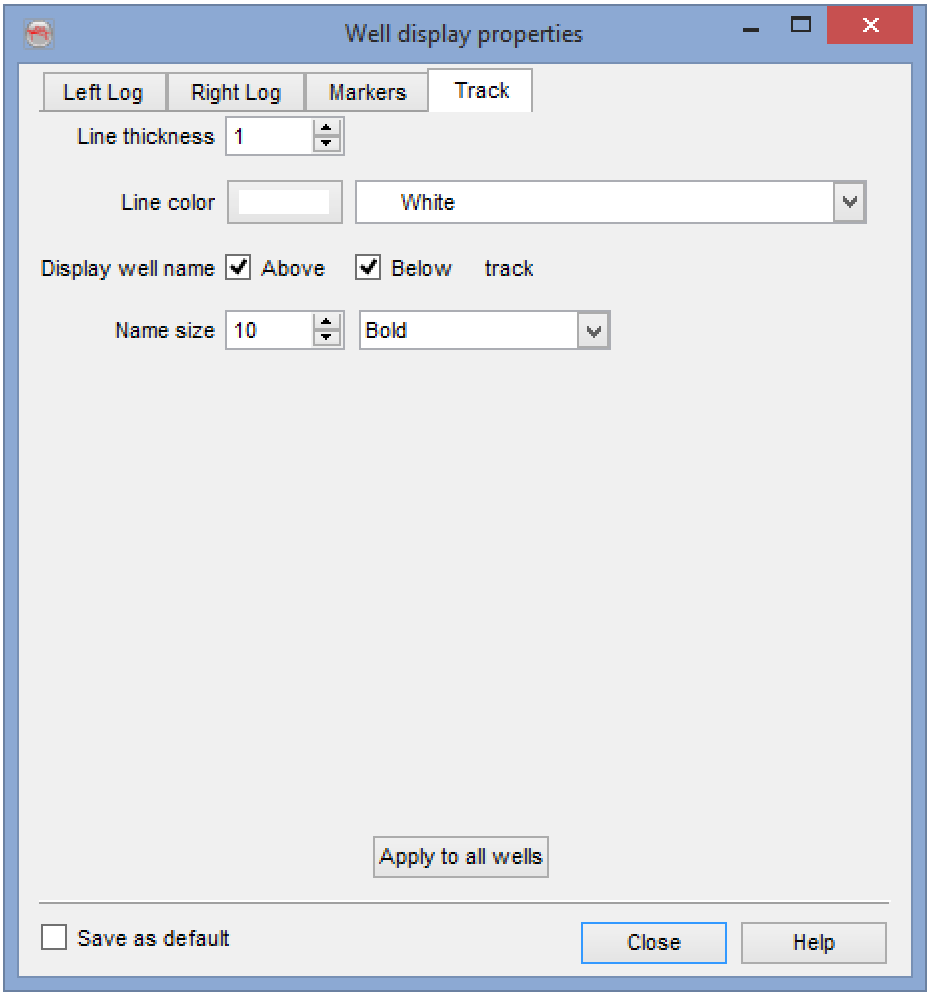
Create attribute log: allows to create a new log by calculating an attribute along the well track. A new window pops up where the attribute, log name, and the depth range should be provided. The Depth range is defined as start depth, stop depth, and sample distance.

Create log cube: enables to create a volume of a selected log. The log is duplicated on a user-defined number of traces around the well location. More than one log can be selected at once and one volume for each log will be generated. This allows easier comparison between well logs and seismic data.

Properties: sets various display settings of a well track, the logs, and the markers. The properties can be set for each well and can also be updated for all wells displayed in a scene. The later can be done using the button Apply to all wells available in the Well Display Properties window.


Well display properties
Left Log
Right Log
Markers
Track
Select log
Gamma Ray
Specify
data range
Flip
Log range (min/max)
7.15
104.35
Logarithmic
Style
Well log
Seismic
Log tube
Line thickness
1
Line color
Black
Fill
Right of log
single color
Fill with
Gamma Ray
flip color table
AI
7.15
104.35
Log display width (m)
1470
Apply to all wells
Save as default
Close
Help


Well display properties
Left Log
Right Log
Markers
Track
Marker size
5
Marker color
White
use single color
Shape
Cylinder
Height
1
Names size
10
Bold
Names color
White
same as markers
FS8
FS7
Truncation
Top Foresets
FS6
MFS4
FS4
FS3
FS2
MFS2
FS1
MMU
Display markers
Apply to all wells



Log display, Markers display and Track display properties

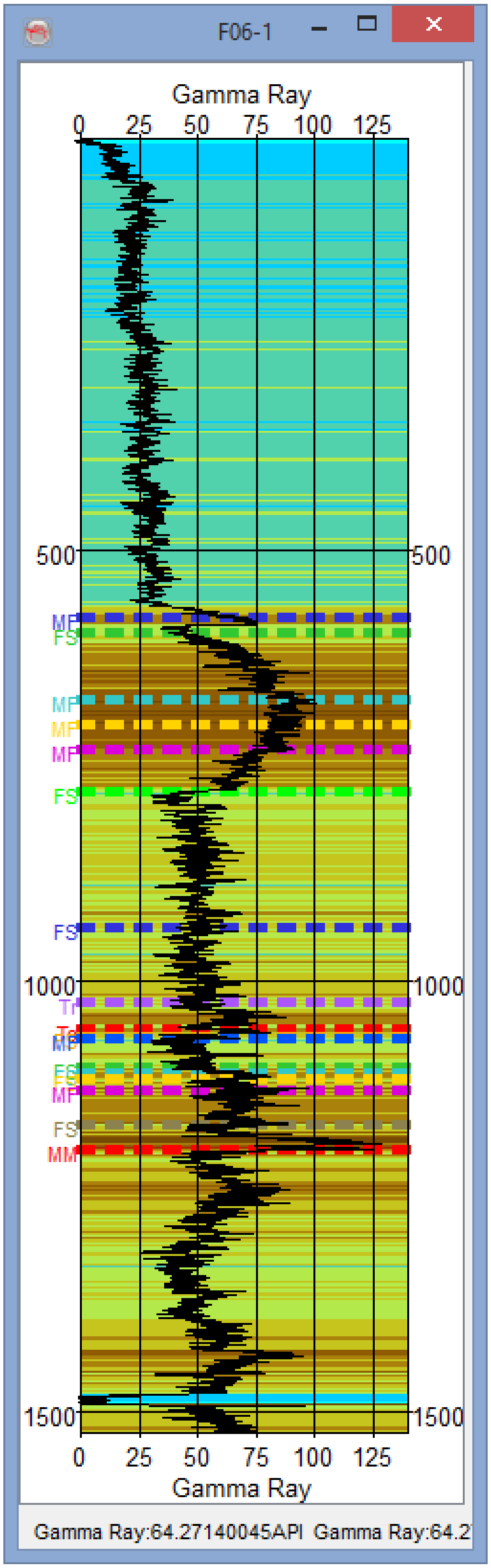
Well Log Properties: In a scene, the log are displayed using the Left Log and Right Log tabs. The logs are displayed on the left and/or on the right of a well track according to a current view. The log properties include the log selection, log range, fill color and the the thickness of the log line. None refers to no log selection/display. If the logs are already imported, the Select log should contain the name of the logs in the drop down list. The data ranges and the color ranges are updated automatically from the selected log. However, both fields are editable. Two types of log displays styles are supported. For a standard log trace display style Well log radio button is selected. For a wiggle display, the Seismic radio box is selected. The well logs can be filled with any selected color table. The color ranges can also be manually set/clipped. However, the seismic style contains different settings. The synthetic seismic traces can be displayed by toggling the Seismic radio box ON. The seismic traces can be repeated by specifying the repetition numbers in the spin box adjacent to the Repeat text. The Overlap field refers to the overlap percentage of the repeated traces. Optionally, dual logs spectra can be displayed together on the same side by displaying one log as a trace and filling the color with another log (Fill with Log).

Well Track Properties: The track properties are modified in this tab. The track line thickness is changed by scrolling the Line Thickness spin box. The well track/name color is updated by pressing the colored button. The well name can be displayed above and below the track. The name size can also be increased or decreased. It may be noted that the name size is adjusted relative to the 3D zoom.

Well Marker Properties: Well marker properties tab include the settings for marker's name size, color, shape (3D), etc. The marker size is adjusted using the spin box (up/down). The limits for the size are set from 1 to 100. The color of all markers of a well track can be changed in to a one unique color. This is supported by the 'use single color' option. If the same color is to be assigned to all available well markers, set check to this field and select the color. Additionally, three different 3D shapes are supported (Cylinder, Sphere, Square). The cylindrical shape is added for orthographic camera displays, which is better for the visualization purposes. The height of a cylinder is supported.

Edit Welltrack: Allows you to add or delete nodes to the well track. Deleting nodes is done by holding **CTRL** and clicking a node. Adding nodes is done by making "node picks" on any of the active elements on your screen. Remember that the Z-scale caused a vertical stretch, distorting the appearance of real distance in the 3D view.

2D Log viewer: This property allows you to display a well log in a 2D scene. The display in log viewer is driven by the 3D scene. To interactively display different logs, go to the well properties and make the desired changes.

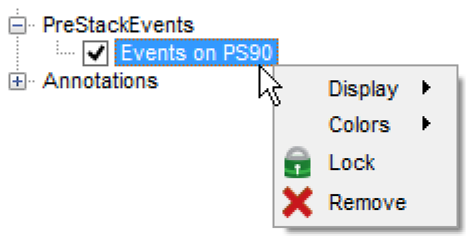

F06-1
Gamma Ray
0 25 50 75 100 125
500
1000
1500
MF
FS
MF
MF
MF
FS
FS
MF
FS
FS
MF
FS
MM
Gamma Ray:64.27140045API Gamma Ray:64.2

Save: Stores a new well or saves the changes that were made to an existing one. Provide a name for a new well, and if a depth to time model is available, select the file. The file should have the same format as when importing a well-track. Optionally, you can examine the file using the corresponding button. Specify if the model uses TVDSS or MS, also the measurement units.

Lock / Unlock: Locks the selected object. This prevents accidental removing, moving, or displaying data on the object. After clicking unlock all manipulations are possible again.

Remove: Removes the well from the tree (not from disc).

Pre-Stack Events



This tree item allows you to display picked or imported pre-stack events in the 3D scene. Note that this tree item is only display when there are pre-stack events in the current project. Otherwise it is hidden.

There are five display modes:

- **None:** nothing is displayed.
- **Zero offset:** A pick shows the location of each pre-stack events, and the Z value of the zero offset.
- **Sticks from sections:** Picks show not only the location but also the moveout curve described by the picked events.
- **Zero offset on sections:** Same as zero offset, but only the events that exists on displayed inlines/crosslines will be shown.
- **Sticks to gathers:** Same of sticks from sections, but only the events that exists on displayed 3D pre-stack planes will be shown.

The displayed picks are always linked with a thin line. Regardless of the display mode the picks are colour-coded with respect to the following color settings:

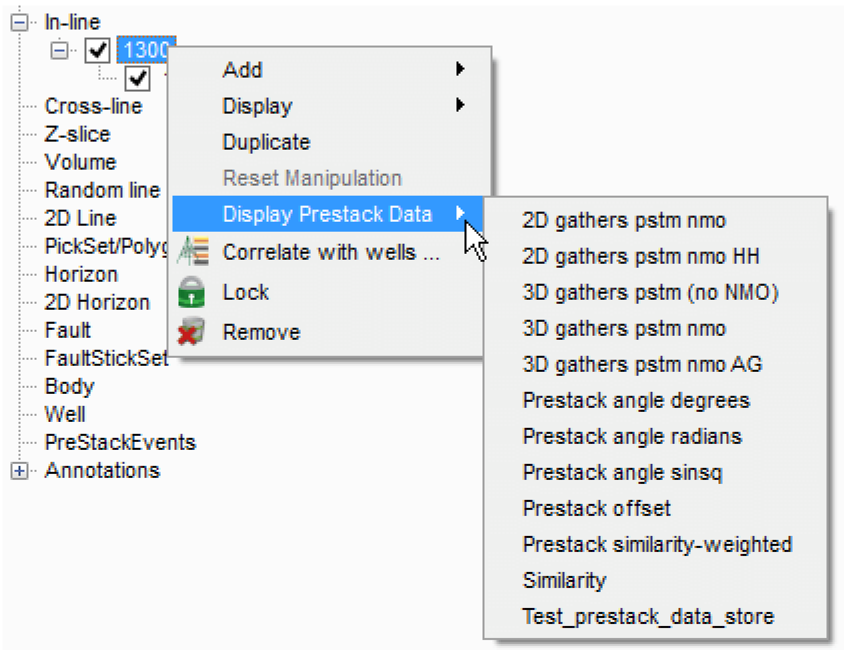
- **Single:** Default mode, one single color for all picks of the pre-stack events.
- **Quality:** The color of the picks is related to its quality. This attribute is either imported with the pre-stack event or set when picked in the Velocity Model Building plugin.
- **Velocity:** The color of the picks is related to the corresponding interval velocity. Note that for this to work the input pre-stack datastore and corresponding migration velocity must be specified in the velocity model building plugin.
- **Velocity fit:** The color of the picks is related to the deviation between the picked event and velocity of the best fitting normal/residual moveout curve. Note that for this to work the input pre-stack datastore and corresponding migration velocity must be specified in the velocity model building plugin.

The color of the picks, except in single mode, should be adjusted using the colorbar like with any attribute by ajusting the colorbar and amplitude ranges.

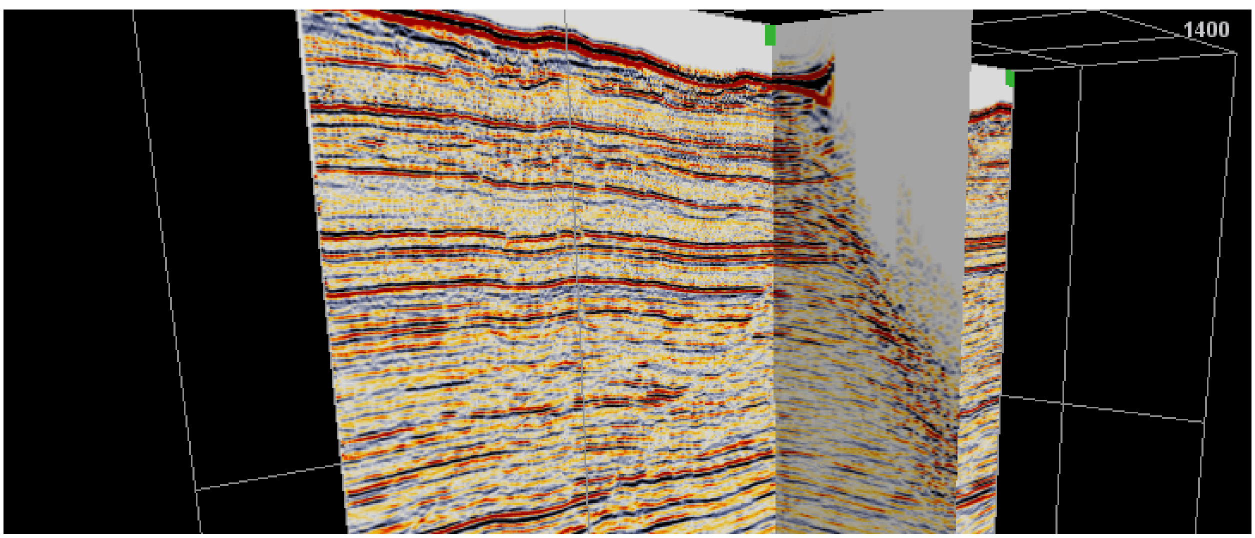
Pre-Stack 3D Viewer

Prestack gather selection from the tree

Pre-stack (PS) gathers can be displayed in the 3D scene, perpendicular to an inline or crossline. Post stack data must first be displayed on an inline/crossline. The post-stack data does not need to be linked to the pre-stack data. Once the line is loaded, go to interact mode (graphical toolbar, second icon from the left), right-click in the scene your inline and use the option *Display PS gather*. The menu will list any pre-stack data available (loaded) in the survey. Please note that multiple PS gathers can be displayed on the same inline, and moving the inline to another position will keep the position of the PS gather and update its content.



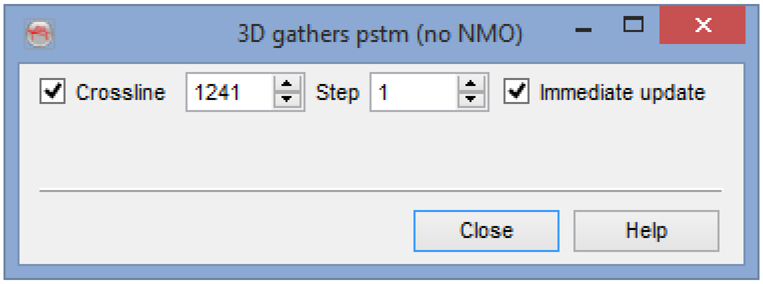
Right-click menu of a vertical slice tree item when pre-stack datasets are available.



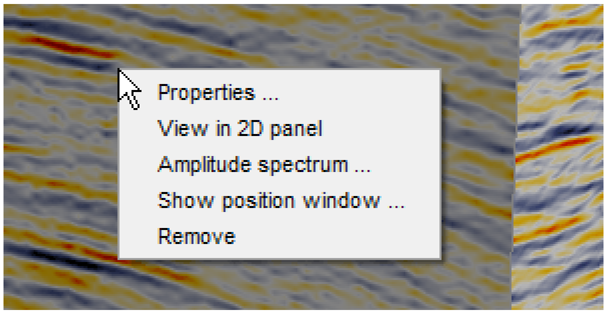
Post-stack data on an inline (left) with a prestack gather displayed perpendicular to it.

Positioning the prestack gather

When a pre-stack data is displayed in a scene, the pre-stack positioning window also pop-ups (look at the corners of the screen). By using this window, the pre-stack data can be scrolled interactively. The crossline value locates the currently-displayed prestack panel. That can be scrolled at given steps (increment). You can use either the arrows or the mouse wheel over the inline/crossline value.



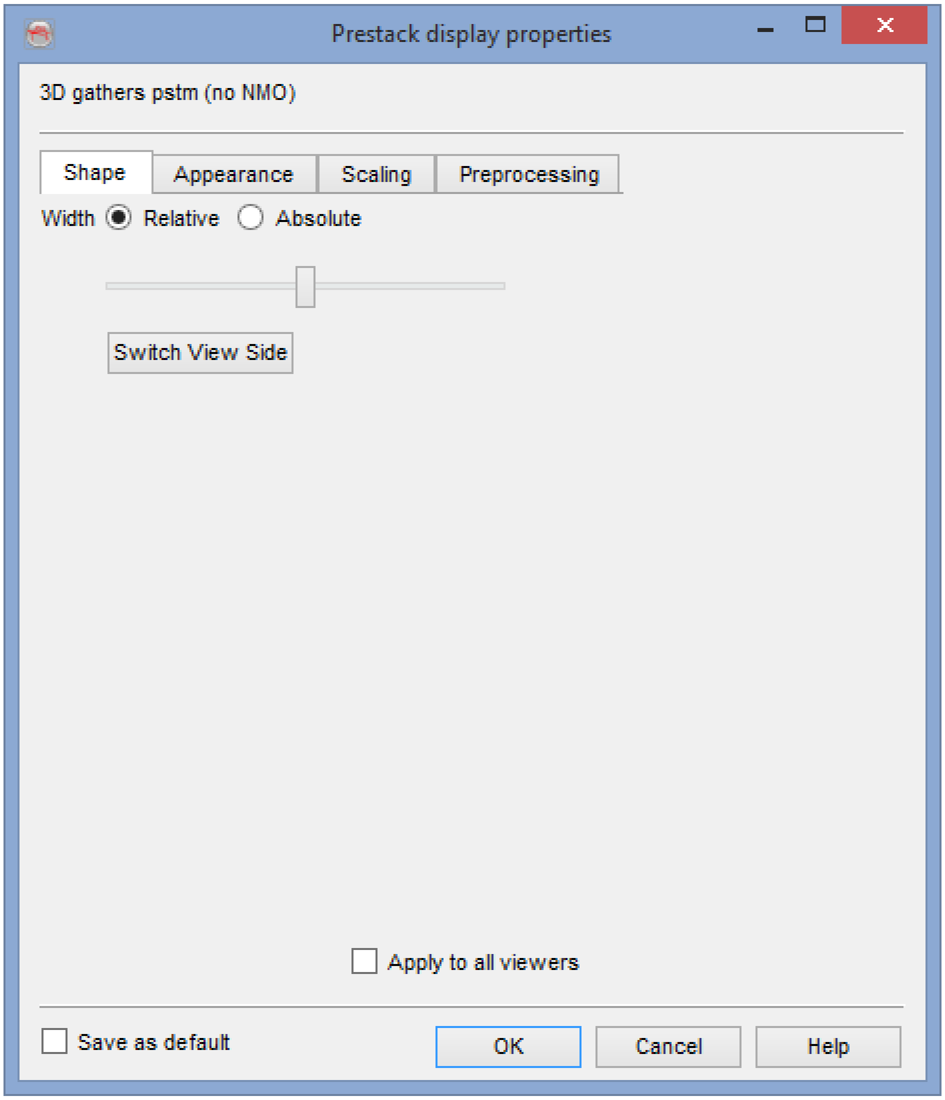
Right-click options for a prestack dataset

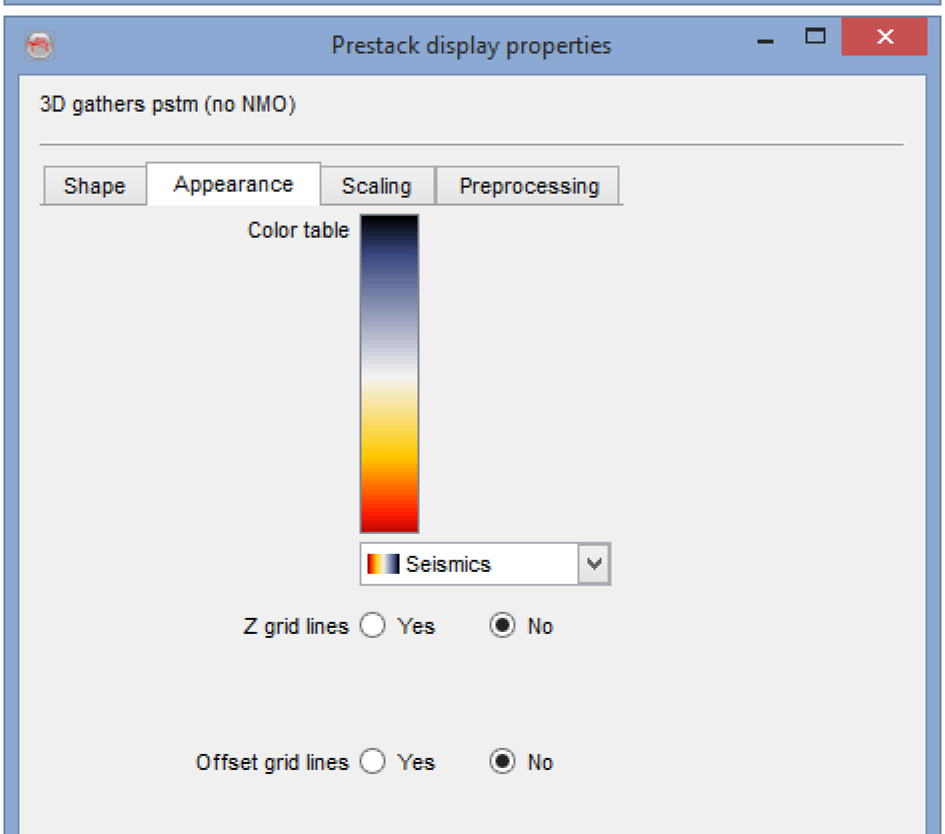


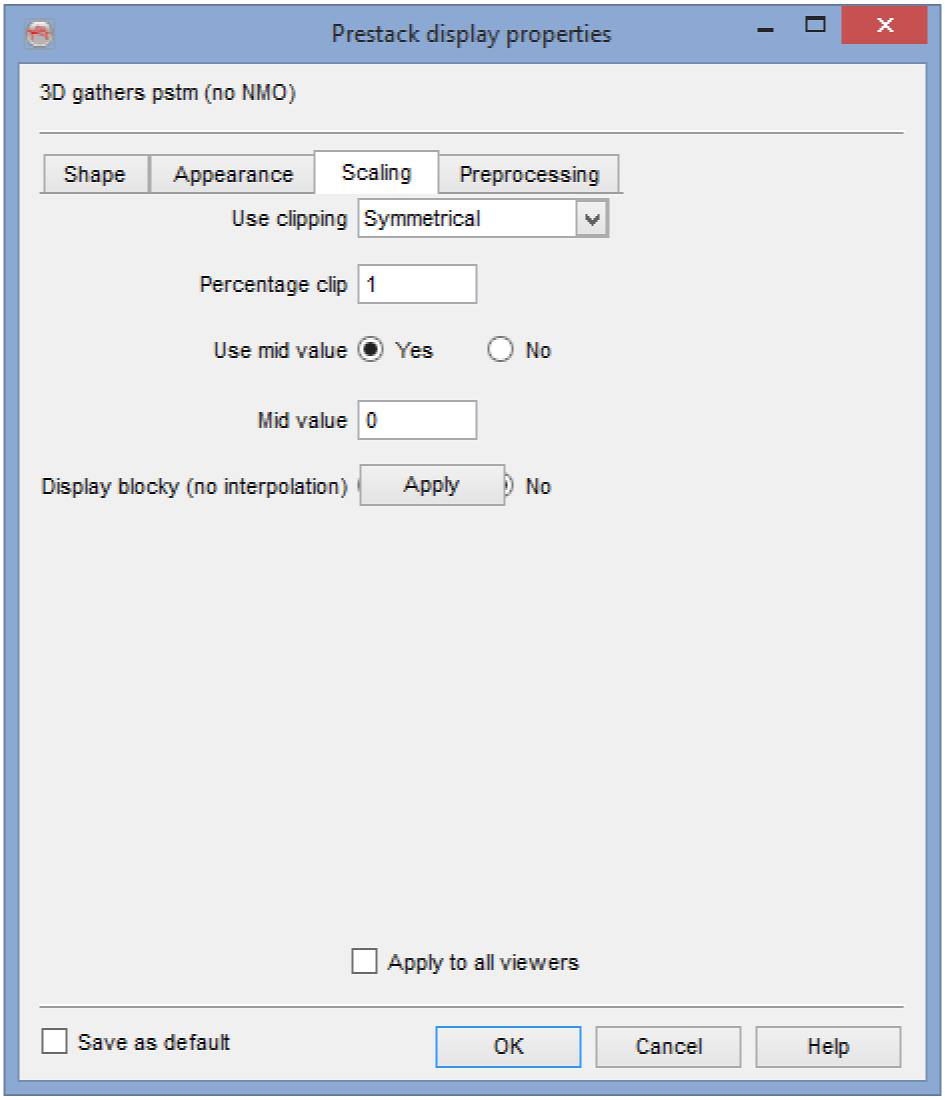
Right-click menu on the pre-stack gather display to open it in the prestack 2D viewer.

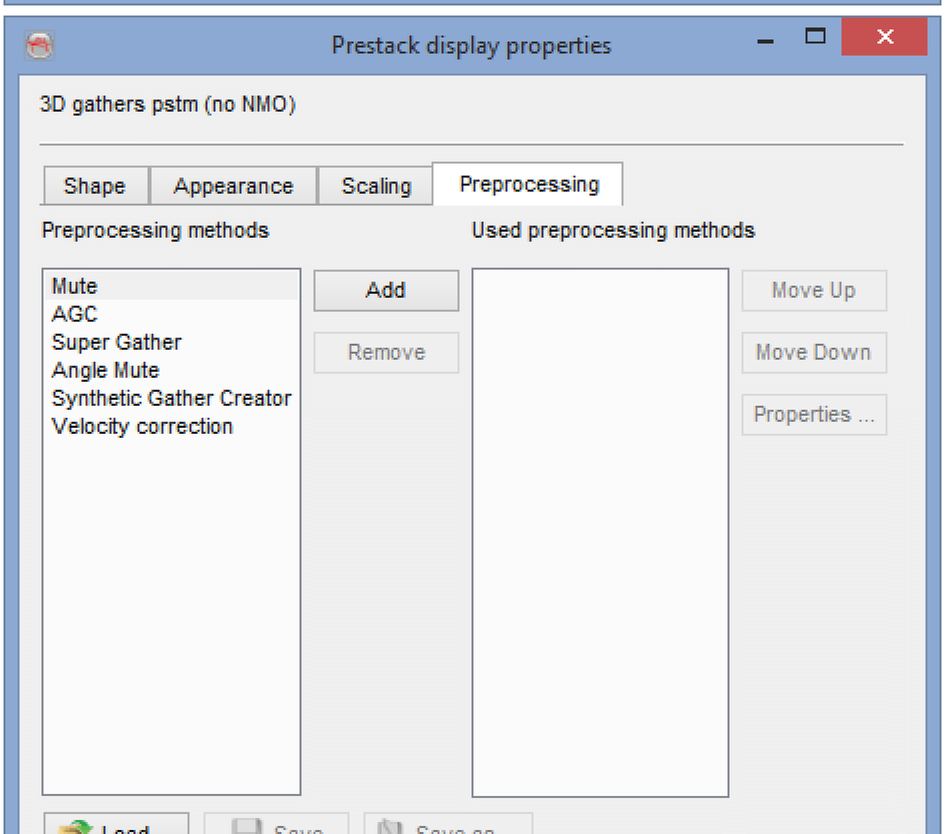
Properties:
There are several display properties available that are described below. Please note that the gathers are first displayed without any processing. This can be set together with other properties in the PS gather display properties:

> **Shape:** The shape tab will set the size of the gather with respect to other 3D elements and its relative position
> **Appearance:** Color bar, amplitude ranges and grid lines can be set in this tab.
> **Scaling:** This tab is used to scale (clip) the amplitude range of the displayed data.
> **Preprocessing:** Pre-processing may be applied to enhance the display. The available algorithms are presented in the prestack processing chapter.


Prestack display properties
3D gathers pstm (no NMO)
Shape
Appearance
Scaling
Preprocessing
Width Relative Absolute
Switch View Side
Apply to all viewers
Save as default
OK
Cancel
Help


Prestack display properties
3D gathers pstm (no NMO)
Shape
Appearance
Scaling
Preprocessing
Color table
Seismics
Z grid lines Yes No
Offset grid lines Yes No


Prestack display properties
3D gathers pstm (no NMO)
Shape
Appearance
Scaling
Preprocessing
Use clipping
Symmetrical
Percentage clip
1
Use mid value
Yes
No
Mid value
0
Display blocky (no interpolation)
Apply
No
Apply to all viewers
Save as default
OK
Cancel
Help


Prestack display properties
3D gathers pstm (no NMO)
Shape
Appearance
Scaling
Preprocessing
Preprocessing methods
Used preprocessing methods
Mute
AGC
Super Gather
Angle Mute
Synthetic Gather Creator
Velocity correction
Add
Remove
Move Up
Move Down
Properties ...

Resolution: Interpolates the data to get a better display (consumes more memory).

View in 2D panel: Transfer this dataset to view it in the prestack 2D viewer.

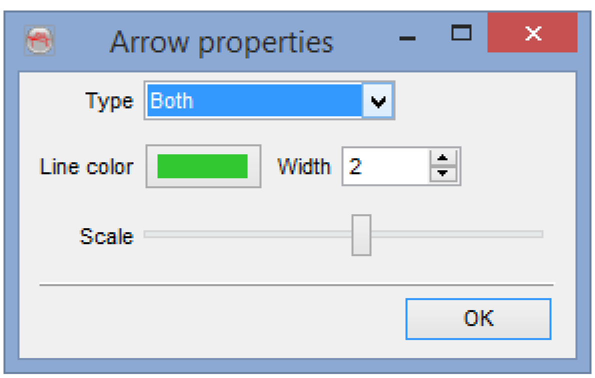
Amplitude spectrum: Display the average frequency spectrum of the trace of that gather, for the displayed Z range.

Annotations

With this option, you can draw arrows, load images, and write text on the display window by right clicking in one of the items in this tree.

Arrow: You can *add* new arrow groups, change the properties, lock-unlock, and remove them by right clicking on this element.

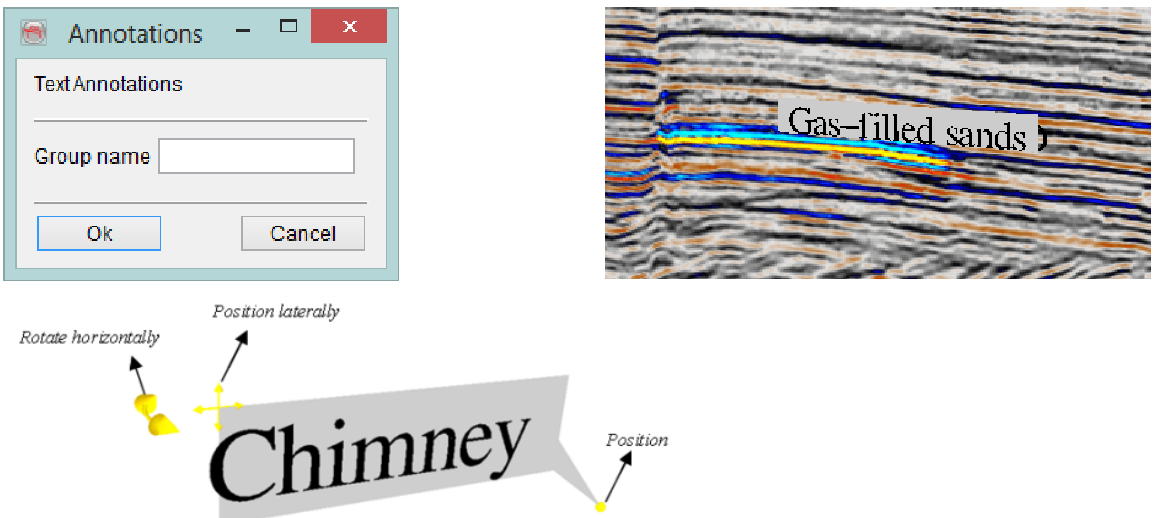
Once you have added a new arrow group, named it and saved it, you can now click in the scene to add arrows (CTRL+left-click to remove an arrow). The arrow properties can then be changed by right-clicking on the newly inserted arrow group and selecting the properties from the fold-out menu. In the arrow properties, arrow type (top, bottom or both heads), color, width and size are adjusted.



Arrow style properties

Image: Once you have *added* a new image group, you can click in the scene to add an image (CTRL+left-click to remove an image). It is then possible to store, resize, change image, lock-unlock, and remove it by right clicking on the relevant image group in the tree.

Text: You can *add* text by creating a text group. When a group is created, you can add multiple text annotations by clicking on an element (inline/crossline/timeslice/horizon) that is displayed in a scene. The text will be added at the clicked position. After clicking, write the text in the pop-up text box. The text can be modified from the scene by right clicking on the inserted text annotation and selecting *Change text* option from the pop-up list menu. Moreover, the text position can be adjusted in a 3D view. This is done by using the horizontal and vertical anchors (two sided arrow head) that are available in the interact mode. Use left mouse button to click on the anchors to move and relocate the text position. Text can be moved vertically, horizontally, or it can be rotated. Repositioning is also possible by clicking and changing the position pick. For details see the figure below. If one wants to remove the annotation, use CTRL+left mouse click on the annotation's pick (yellow circle).



Add a new annotation group and changing the position of the annotation

Text contains two pop-up menus:

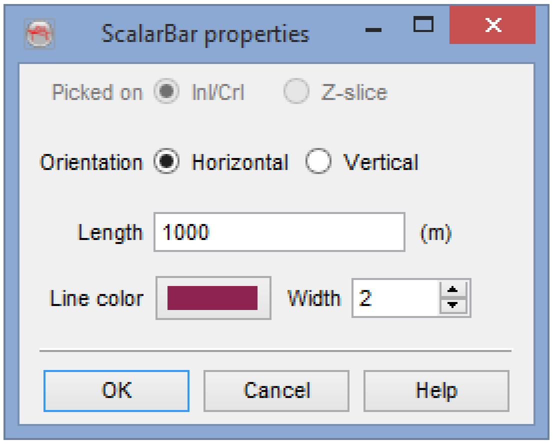
- **Add Text Group:** Adds a blank text group
- **Load:** It is used to load the stored annotation group(s).

The text group pop-up menu can be launched by right clicking on the text group name.

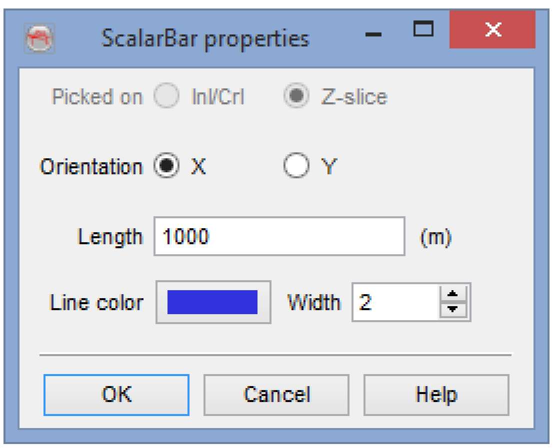
- **Size:** Resize the text group. It may be noted that it will resize the all inserted text sub-elements according to the new size.
- **Save/Save as:** Saves the text group in to an existing name/new name.
- **Change Text:** it is used to replace/change the text of the selected annotation. It may be noted that in the tree pop-up menu it si inactive. A user can only change the text of a selected annotation. It is done by right-clicking over the annotation in a working scene.
- **Background Color:** Modifies the background color of the annotation.

- **Lock:** If lock is selected, it will prevent further modification of the group.
- **Remove:** It removes the group from the tree/scene.

Scale Bar: Use this option to *add* a scale bar to an inline, crossline or Z-slice. Once added and saved, right-clicking on this element will also give you the option to change the properties.



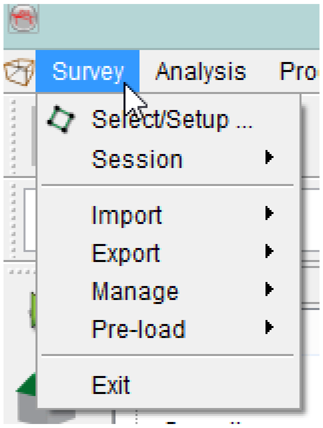
For Inlines and Crosslines, this includes the Horizontal or Vertical direction options



For Z-slices, this includes the X or Y direction option

Survey

Table of Contents

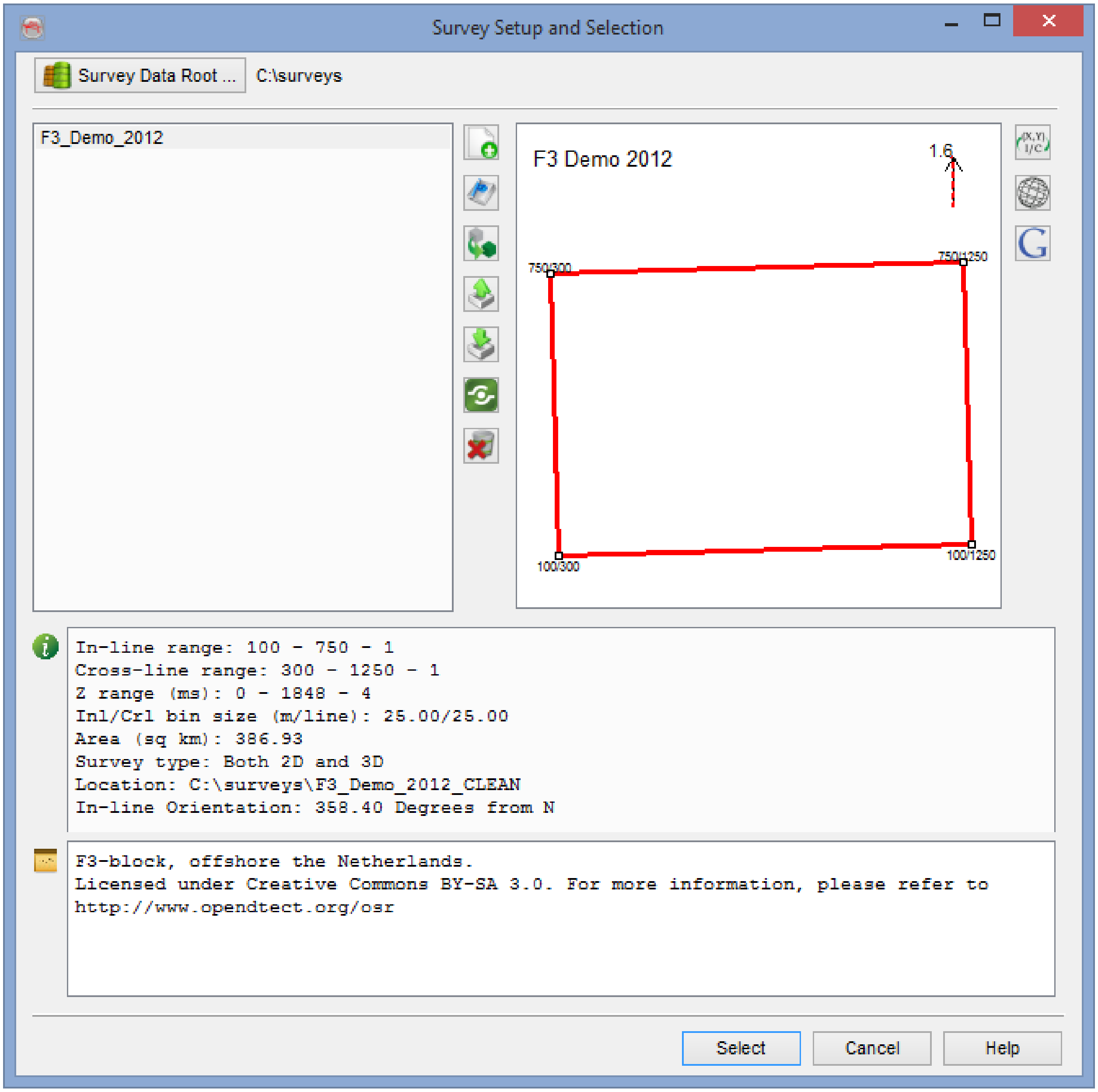


OpendTect

Select & Setup Survey

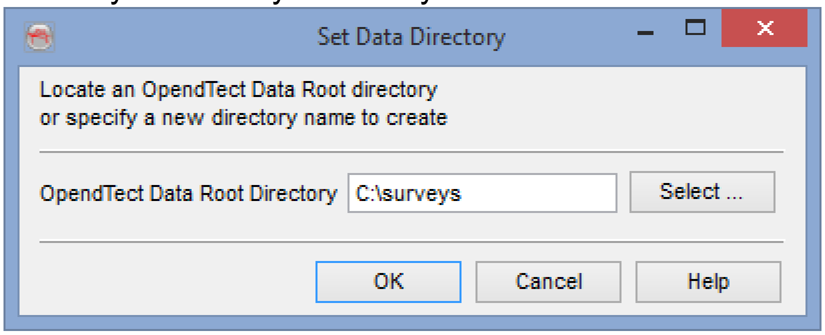
The Survey module is used to select, create, modify, delete or copy surveys. A survey defines the geographical boundaries of an OpendTect project and relevant positioning information such as the relationship between inline/crossline and X/Y co-ordinate systems. Each survey (project) stores its data in a separate directory that needs to be specified along with the survey reference name. Moreover, the survey rotation angle is displayed; this angle shows the deviation from geographic North.

Survey Selection Window



Select an existing survey from the list of surveys on the left or create a new one with *New*  ... (see below). The boundaries of the survey are depicted in the field to the right and detailed in the information field . The *Notes*  field is a free-format text field to store relevant survey notes.

When you install OpendTect, selected an OpendTect data directory where all your surveys are stored:



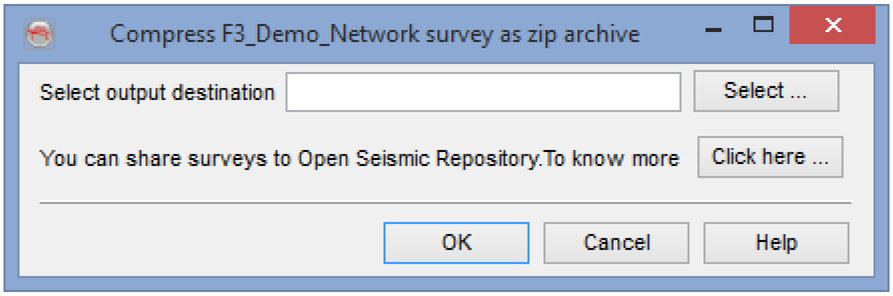
Any folder can be turned into an OpendTect folder, the only change being the addition of a parameter file (.omf).
Only surveys stored in the selected OpendTect folder are displayed and can be accessed. Later you can open another OpendTect folder clicking on *Survey Data root*. The current data root is always displayed on the top of the window.

 Use for creating new surveys

 Use for editing survey box ranges or update coordinate information (see Edit Survey Window)

 Copy whole surveys from your data root to a designated location.

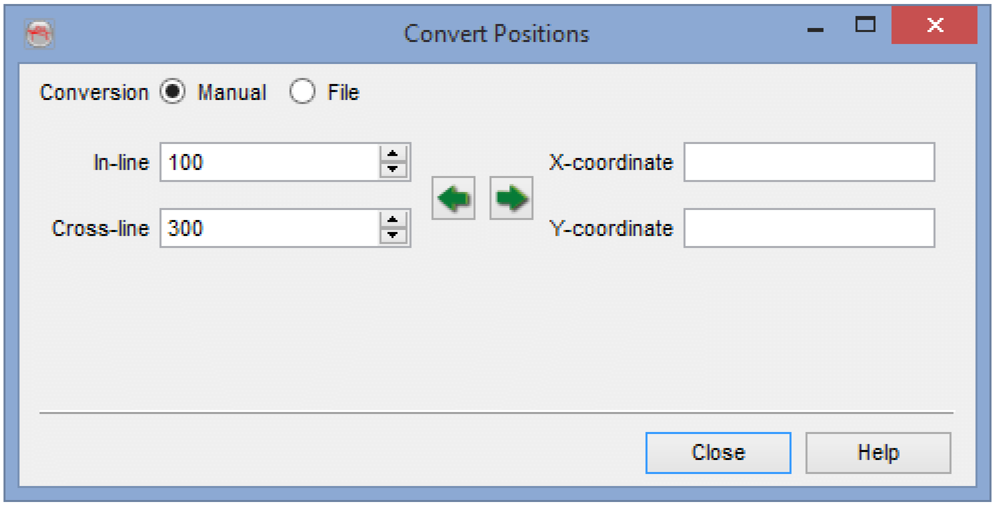
 Allows you to compress/pack your entire survey into a zip file. This is highly recommended when transferring your survey from a computer to another computer, especially if they do not use the same platform. All data from this survey will be contained in the zip file, with the exception of the SEG-Y and/or CBVS files that were used 'in-place' from another location (ie: those SEG-Y or CBVS files that were used but not actually put inside the survey folder)

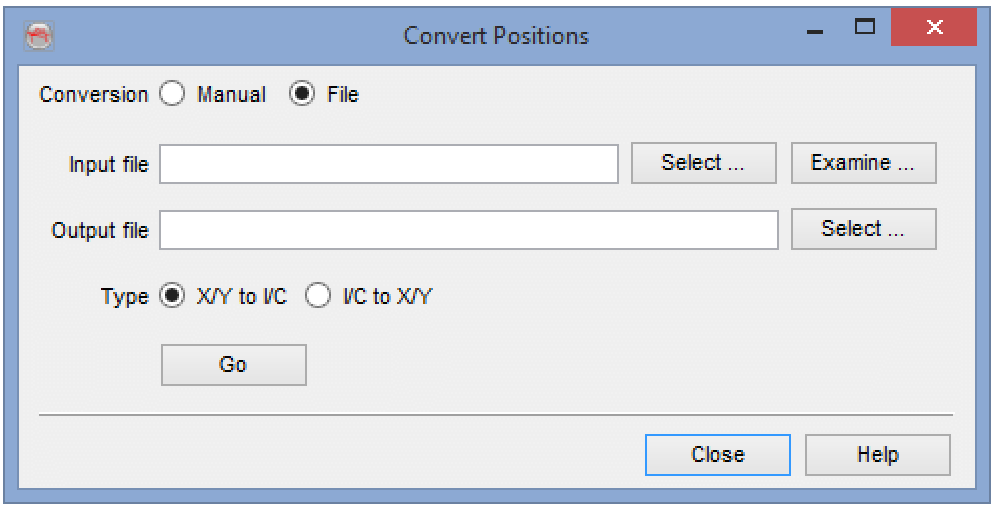


Unpacks a previously packed survey (see above) into your Data root folder. Most zip files could potentially be unpacked, but we support only the unpacking of survey packed using the OpendTect packing tool. If you wish to share your survey with the community, visit our [Open Seismic Repository](https://opendtect.org/osr/).

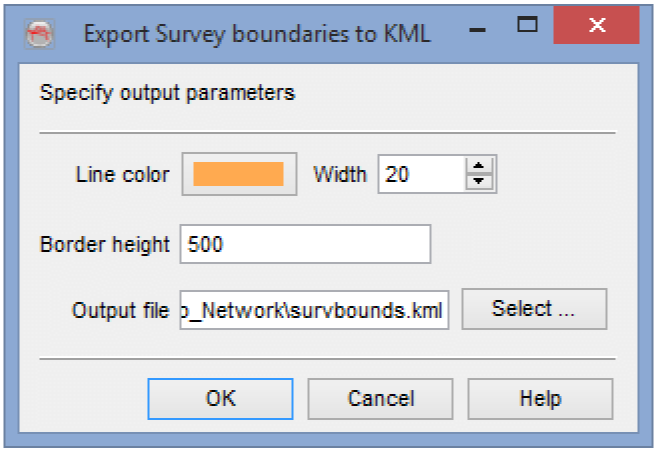
Takes the user to the Open Seismic Repository (OSR) page on the OpendTect website (https://opendtect.org/osr/). Here, one can find information on how to share surveys with the wider community.

In the position conversion window there are two modes available for coordinate conversion: Manual / File. In *Manual mode*, specify a inline/crossline pair, or a X/Y pair, and press the corresponding arrow key to obtain the position in the other domain. In *File mode*, browse the input file and create a new output file. By specifying the corresponding type conversion (XY to IC or IC to XY) and pressing the GO button, the desired conversion is written to the output file. There is no specific file type necessary for this input - even files without extension may be used. Simply *Select* them and, if desired, *Examine*, too.

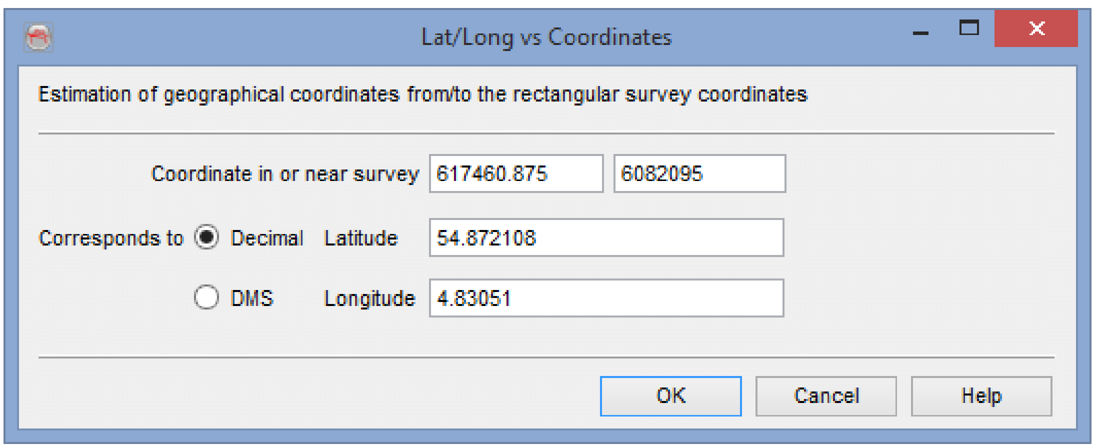


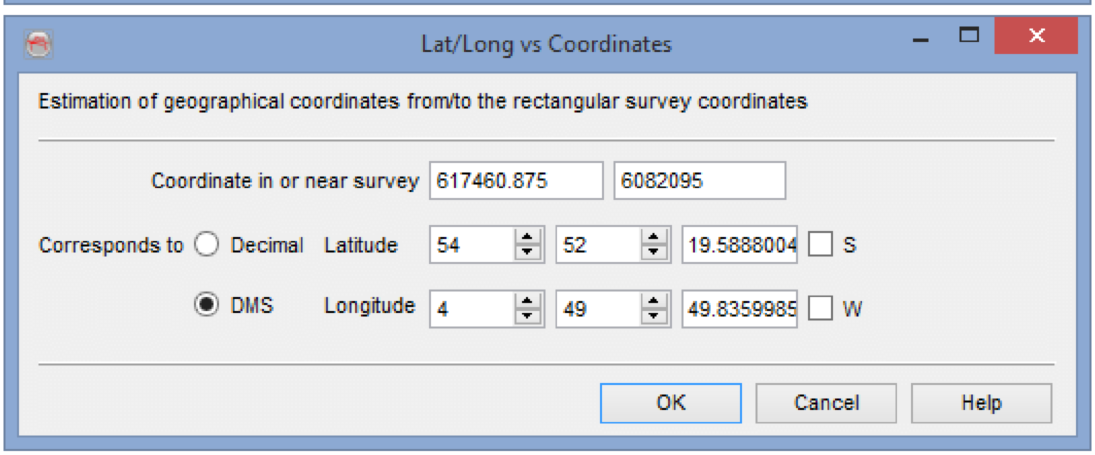


 is used to export the selected survey boundary in a *.kml file, which is accessible via Google Earth. The dialog box contains the editable fields for the survey box. The area of the survey box is filled with the selected color. The width is the horizontal thickness of the survey outline. The border height is the altitude of the line with respect to the ground. The Output file field is an output location of the *.kml file. On 'Ok' the file (*.kml) is written at the specified path, which can be opened directly in Google Earth.



Before exporting the *.kml file, specify the correspondence between X-Y coordinates and latitude/longitude at any location in the surveybox  :

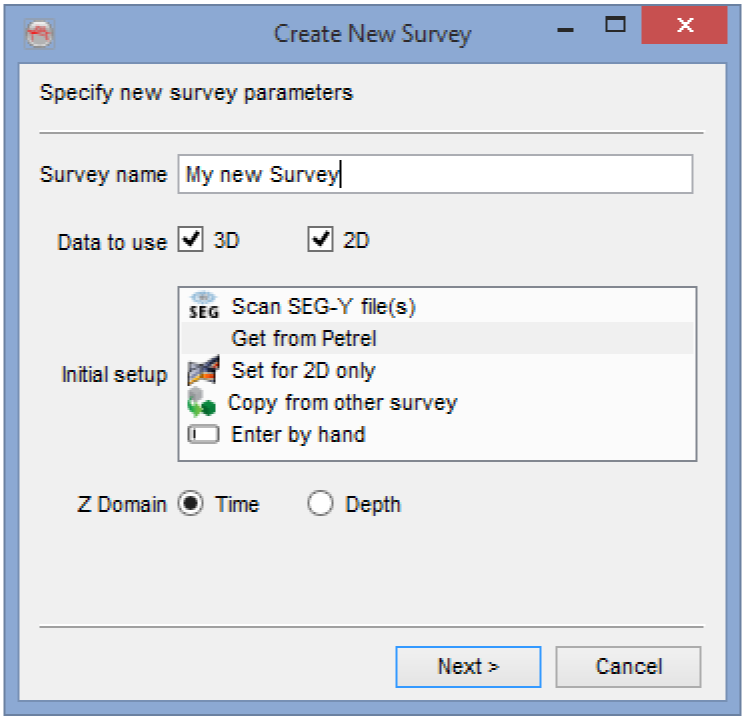




After pressing *Ok* provide an output filename.

New Survey Window

To launch the survey setup window select *New* in survey selection window. The following window will appear on your screen:



Survey name: In the text area specify the OpendTect survey name.

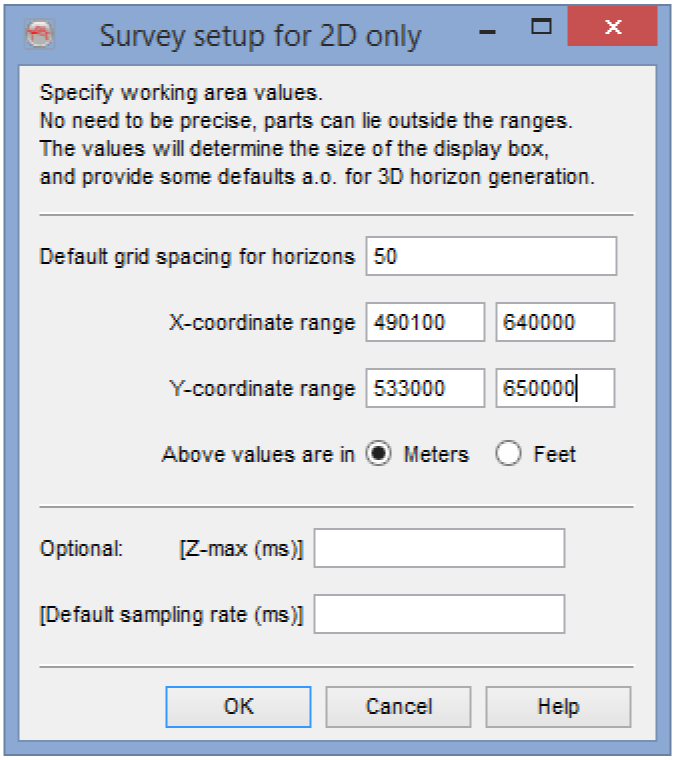
Data to use: Toggle on the data type(s) to be included in the survey (2D only, 3D only or both 2D and 3D)

> **Note:** Select only 3D, if the survey contains only 3D type data set. Select both 2D and 3D, if the survey contains both 2D and 3D type data set. If the survey contains only 2D type data set, select only 2D. Selection type here affects the tree structure and what functions are available to you in the survey.

Initial setup: Determines how you set up the survey ranges and coordinates:

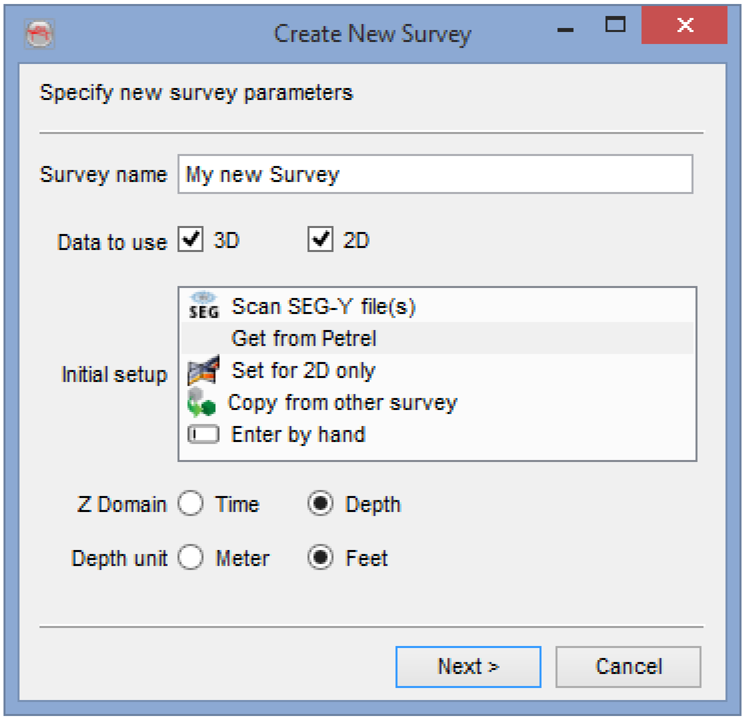
> **Scan SEG-Y file(s):** takes you to the SEG-Y tool to scan the file(s) for survey setup.
> **Get from Petrel:** allows you to copy the survey ranges/coordinates from another software. (Available only with the Petrel Connector plugin.)

Set for 2D only: takes you to the following window where you can enter the working area values:



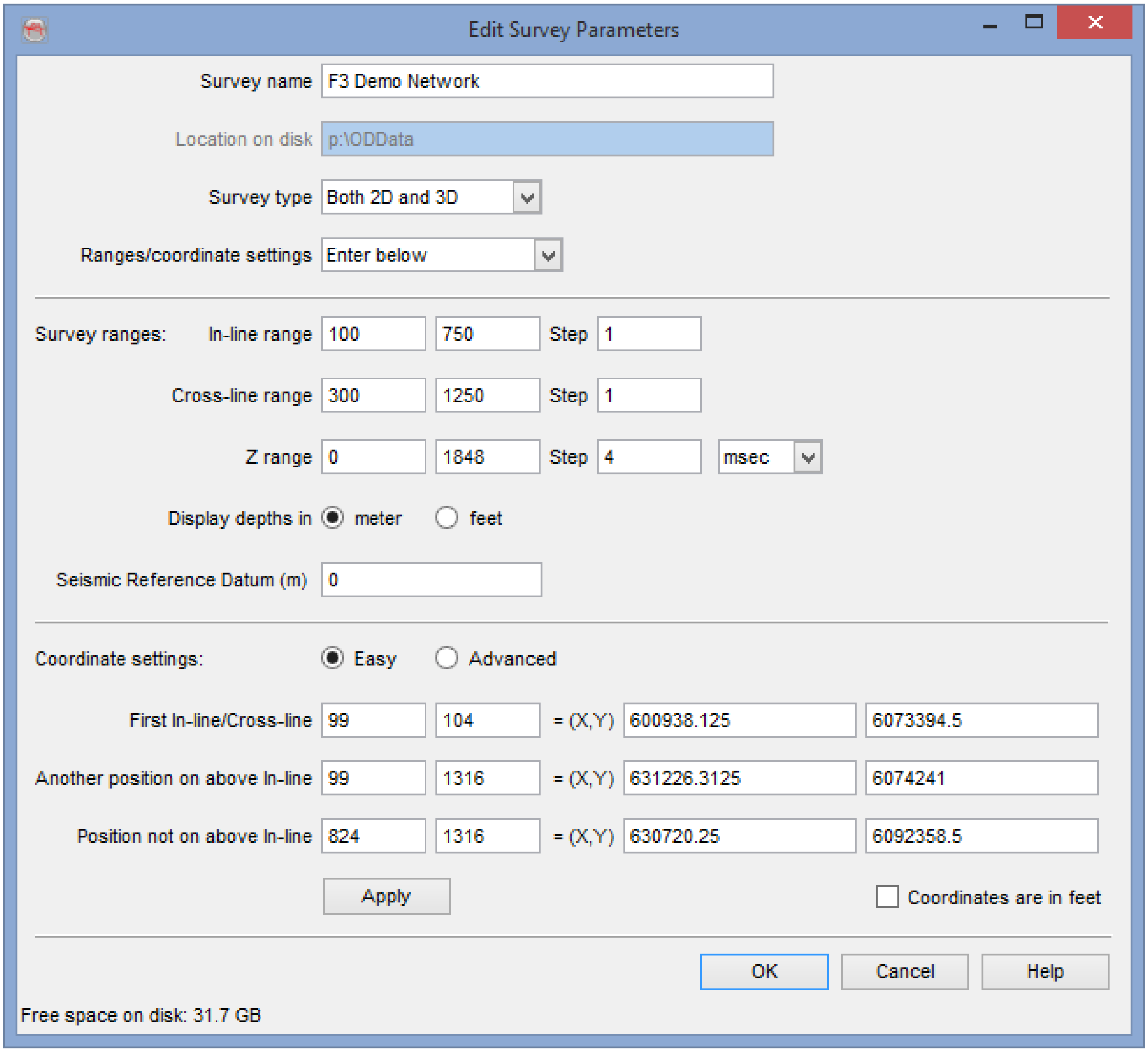
Copy from other survey: allows you to copy the survey setup from another survey on your drive/network.
Manual selection: enter the values manually (see Edit Survey Window)

Domain: Can be in time or depth (for depth, define here the unit):



Edit Survey Window

To launch the survey setup window select *Edit* in the survey selection window. The following window will appear on your screen:

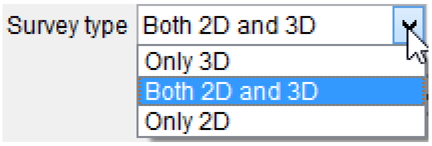


Note: All fields must be completed.

Survey name: In the text area specify the OpendTect survey name.

Location on disk: Specify a directory on disk where the OpendTect survey would be stored. The directory would be turned in to the OpendTect survey location.

Survey type: For the survey type, there are three options:



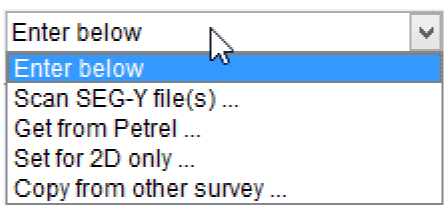
Note:

1- Select only 3D, if the survey contains only 3D data. Select both 2D and 3D, if the survey contains both 2D and 3D data. If the survey contains only 2D type data set, select only 2D. Selection type here affects the tree structure and what functions are available to you in the survey.

2- Ranges/Coordinate settings are described in the following sections.

Survey Ranges

The survey ranges are the inline, crossline and Z-range values. The ranges define a 3D survey area for 3D seismic surveys and 2D grid area for 2D seismic surveys. These fields can be filled manually, by scanning a SEG-Y file (2D/3D), using *set for 2D only* option in Ranges/coordinate settings, or by copying the ranges from another survey. If the *Workstation Access plugin* is available, one will see the *Get from GeoFrame* or *Get from Seisworks* option in the drop-down menu.



The set for *2D only* option is especially used to create a 2D seismic survey. Set the average trace distance and the x and y coordinate ranges, and these will automatically be translated into suitable survey settings.

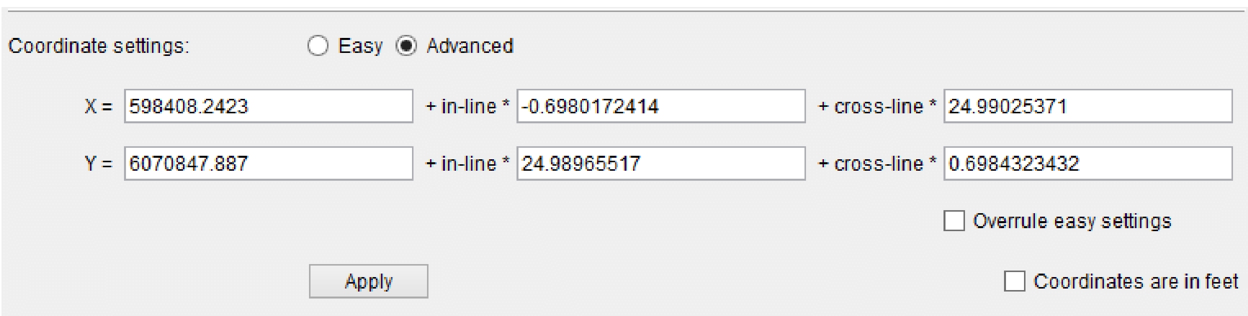
Click on the *Scan SEG-Y file(s)* button to select a SEG-Y file. In the new window, you set the SEG-Y settings, see also SEG-Y scan section. Pressing OK will start scanning the file(s). After scanning, you'll get a file report containing sampling info, data statistics, and the survey setup. The *Survey ranges* and *Coordinate settings* will be filled in automatically.

The *Z range* is specified in milliseconds, meter, or feet. The steps are incremental Z-steps of the survey i.e. the seismic sampling rate.

Coordinate Ranges

The relationship between inline/crossline and X/Y can be specified in two ways. The easy way is to specify three points, two of which must be on the same inline. Due to rounding off errors, this method may not be 100% accurate.

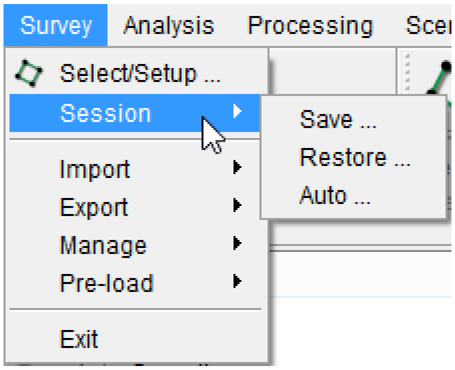
In the *Advanced* option, the exact transformation from one co-ordinate system to an other can be specified. The *Apply* button can be used to verify results graphically and to check the co-ordinate transformation formula.



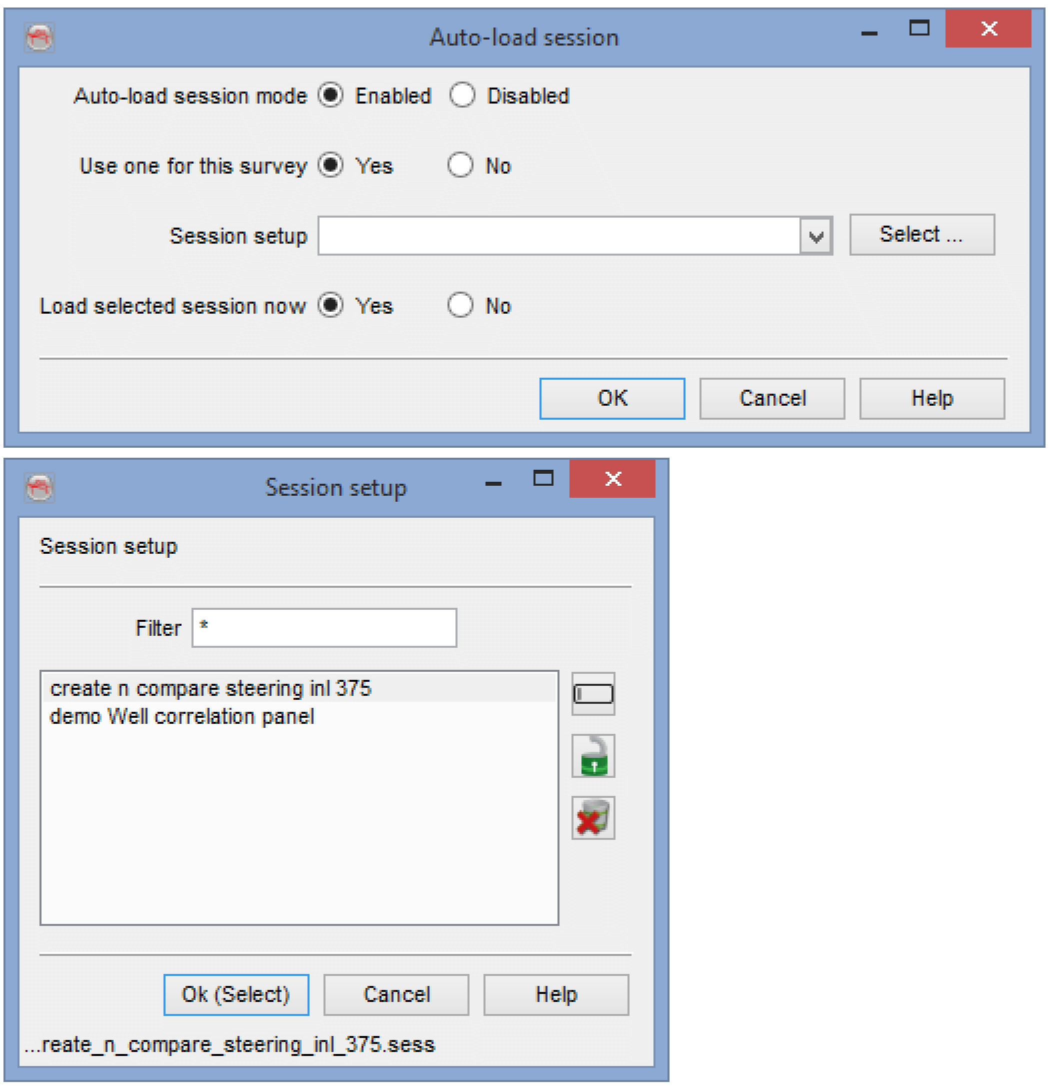
Note: Read more about *Coordinate Settings* in Setup 2D/3D Survey and following.

Session

The OpendTect session is generally used to save and to retrieve the specific settings of a scene. This helps to resume work from previous settings. The session will save all settings of the displayed elements, and can be restored at any later time. When clicking the Survey option in the tool bar and then click *Session*, three options appear. It is possible to save the session or restore a previously saved session. When clicking *Auto*, the session will restore itself automatically the next time you start OpendTect.



When a session is saved, the system stores all element positions and relevant information to recreate the images. The content of the elements is not saved but is re-created when the session is restored.



The auto-load window (left) and the 'Select' option (right)

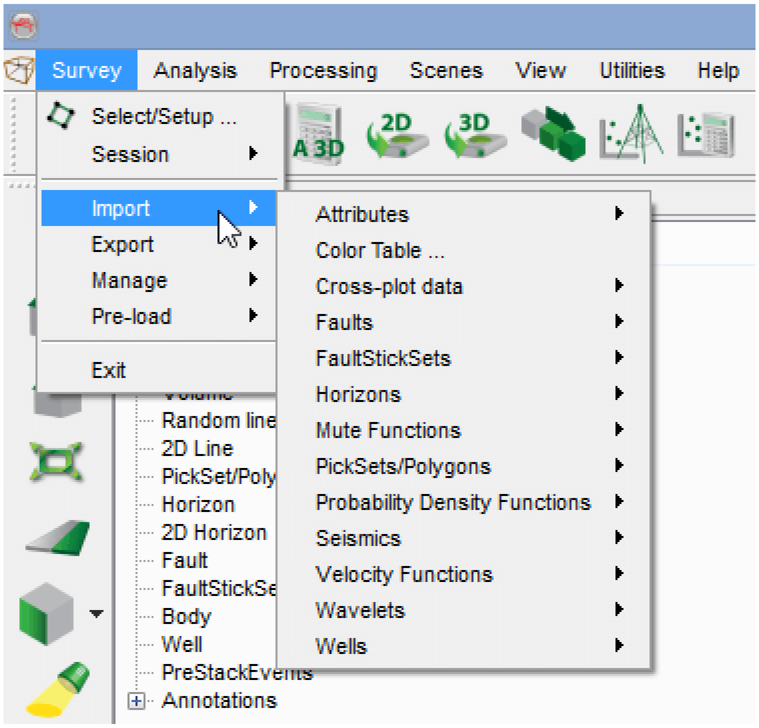
The user can *enable* or *disable* the *auto-load session* option. It is also possible to choose if one of the save sessions will be used in this session. Finally the user has the choice on whether or not to *load the selected session now*.

> **Note:** A similar function exists to auto-load one of the attribute sets.
>
> **TIP:** As mentioned earlier that the contents of the elements are not saved but are recreated. It is a common practice of the OpendTect user(s) to save and restore a session. The mistake a user(s) can normally make is to save a session with the contents of an element(s) (e.g. attributes) that takes a long time to compute. In this way, when such session is restored, it will take a way too long time to restore, because the session can only store the settings (or relevant information) but not the on-the-fly attributes. Thus, it re-calculates the contents. This can be avoided by creating the attribute outputs of such attributes. If an attribute already reside in a disk (a session is saved), the session will be restored very quickly. Similarly, the same thing can happen in a session that contains contents of surface data (the attributes calculated along horizon). The attributes applied along a horizon can be saved as a surface data. It is recommended, to save the surface data before saving a session.

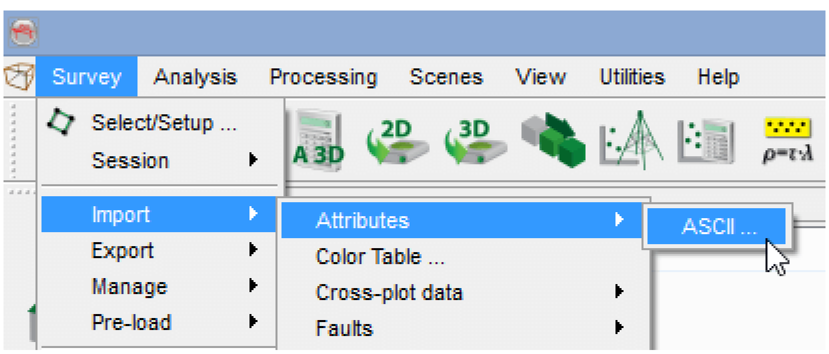
Import

The Import menu is used to import seismic data, horizons, faults, well data, picksets, wavelets, mute/velocity functions etc... Seismic data can be imported as SEG-Y file, Workstation (SeisWorks, GeoFrame-IESX), or from dGB's CBVS (Common Binary Volume Storage) formatted files. The SeisWorks and GeoFrame options are only available if a user have a workstation license*. Whilst OpendTect can handle data volumes with irregular boundaries and gaps, strong irregularity (e.g. frequent non-constant steps between inlines and/or crosslines) throughout the volume may cause problems.

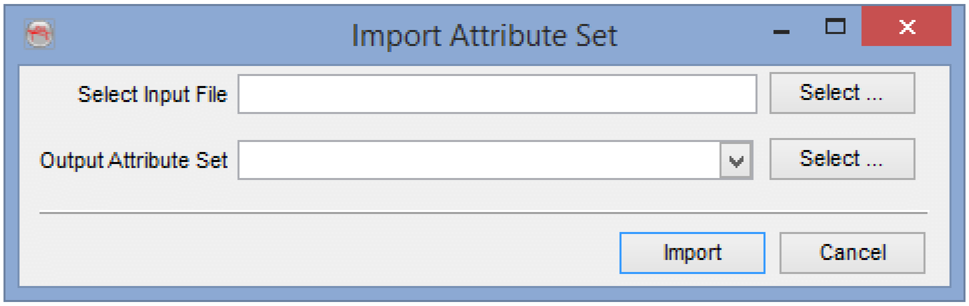
The default location for input data is the survey root directory, with the exception of cbvs volumes where the DTECT_DATA root location is the default folder.



Import Attributes



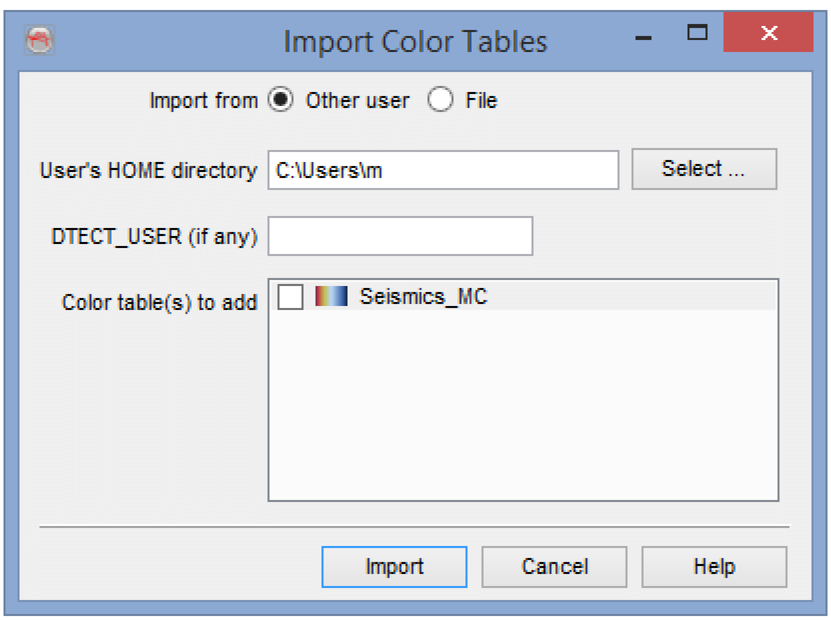
Clicking on *ASCII*, above, opens the simple '*Import Attribute Set*' window, allowing a user to import OpendTect attribute sets form any other (network) location. The file may be renamed on import in the '*Output Attribute Set*' field.



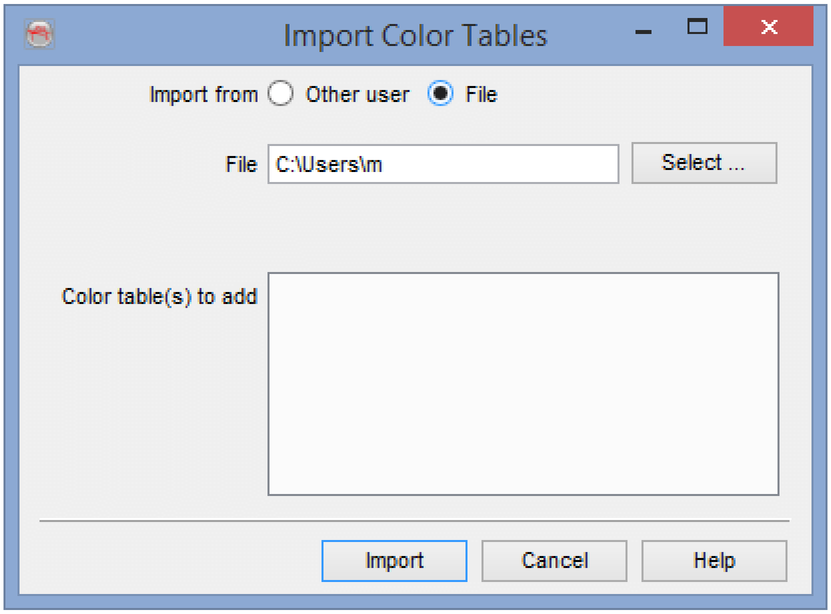
The imported attribute set can then be opened from the normal location within the Attribute Set window.

Import Color Table

Colortables can either be imported from 'Other user' or from 'File'. Default colortables are stored in a file (ColoTabs) that is located in the OpendTect installation directory (/root/OpendTect/5.0.0/data/). The colortables saved by a user are stored in a file (settings_coltabs.user) that is located in the user home directory ($HOME/.od/), here user is the OpendTect username. These files can be modified or imported by using import color table window (see below).



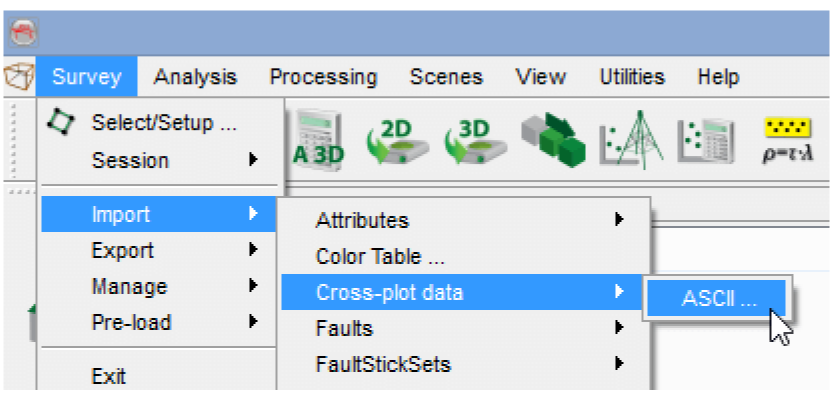
Import color table from other user



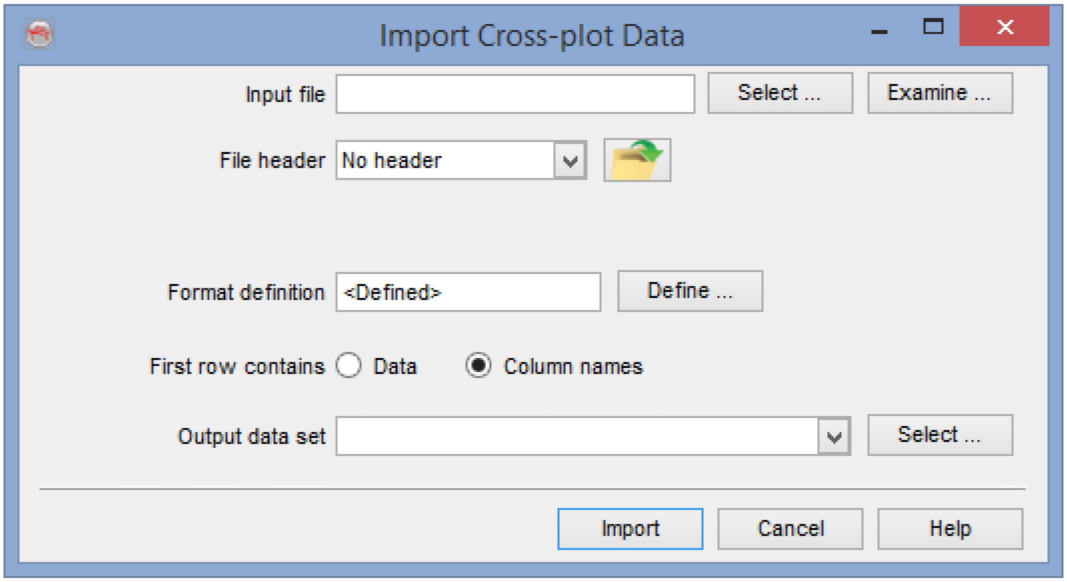
Import color table from file

Import Cross-Plot Data

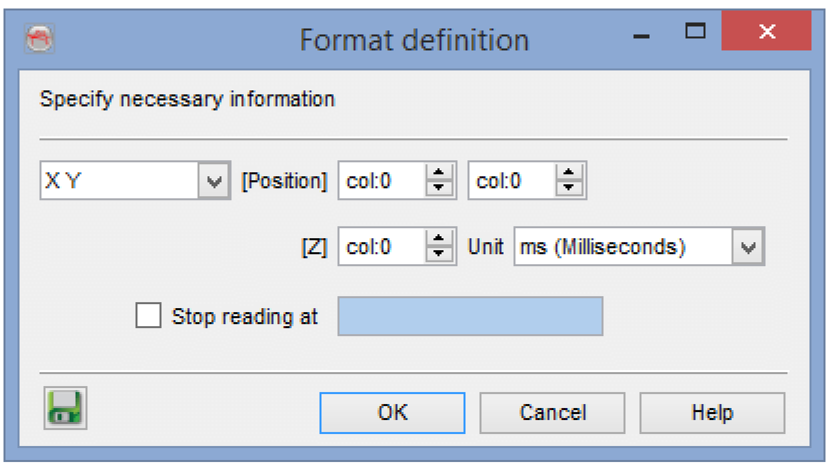
Select the input Ascii file. You may display the input file by pressing the *Examine* button. The input file should be column sorted with one point per record (line).



The main work is to specify the presence of a *file header* and the file *format definition*. The header, if present, can be of fixed length (number of lines), or delimited on its last line by a keyword.

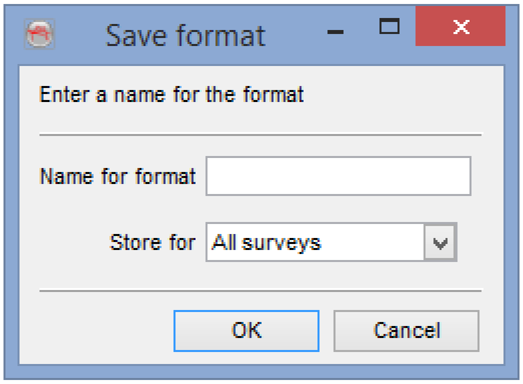


Predefined and saved file formats are available by pressing the icon. Otherwise the format must be manually specified. The *Define* button gives access to the format definition window.



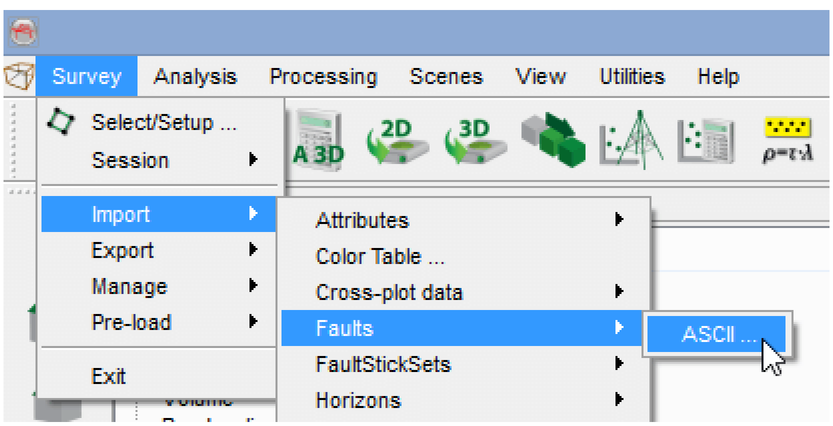
You must specify in the format definition window the column numbers for the position, in terms of an X-Y-Z or an inline-crossline-Z. The Z units can be seconds, milliseconds or microseconds (meters of feet in depth surveys). All other columns with be treated as amplitude data referenced with respect to the given position. The first row may contain either the first vector with its position and the corresponding amplitudes ("Data"), or the name of the attributes in each column ("Column names"). Reading may be stopped at a specific line by providing the adequate keyword.

It is recommended to save the format definition for a later use and QC, by clicking on the icon . In pop-up window, write the name of the format and store it. The format can be stored at different levels (All surveys, Current survey, Current OpendTect user level) depending on the usage. Press OK when done.



Import Faults

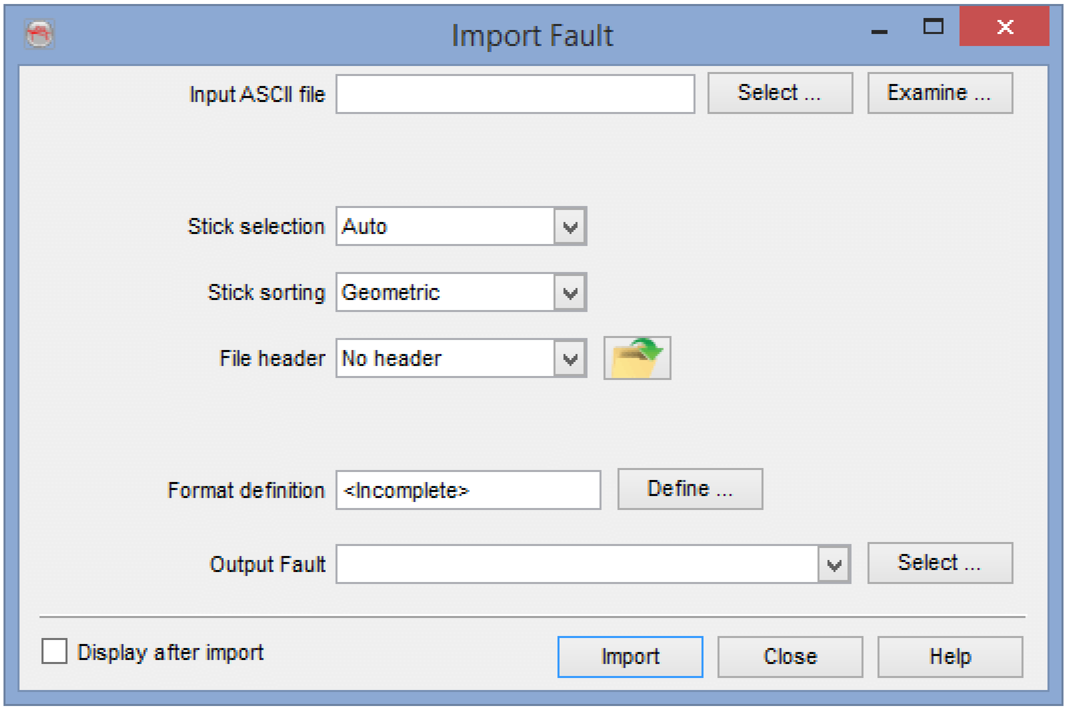
3D Faults (planes) can be imported in OpendTect via *Survey -> Import -> Faults*, from Ascii files or from GeoFrame Workstation (plugin).



Faults are non-editable objects that may be used as display element in the 3D scene, displayed in full or a section. Attributes can also be applied along faults. If you are looking for an editable object that can be converted at a later stage into a fault plane, please load your data as fault stick sets.

Import Fault Ascii 3D

Select the input Ascii file. You may display the input file by pressing the *Examine* button. The input file should be column sorted with one point per record (line).

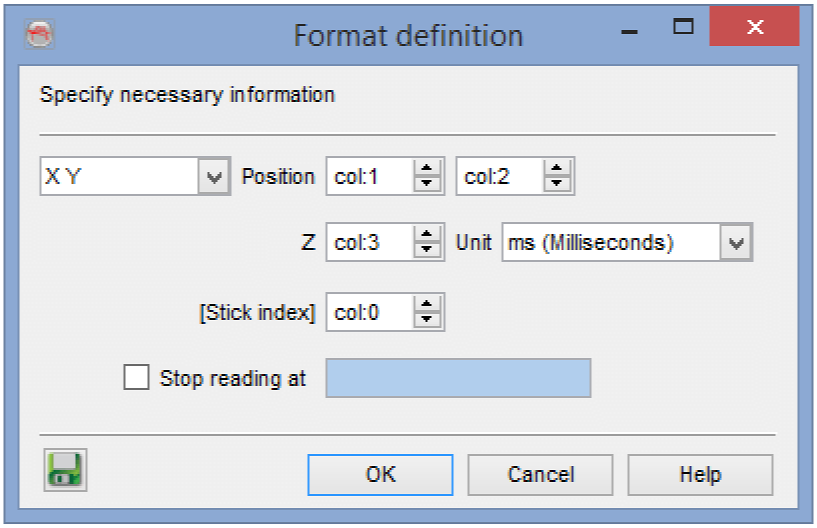


The main work is to specify the *type* of data, the presence of a *file header*, and the *file format definition*.

The sticks composing the planes can be either gathered automatically, either from picked slices (inlines or crosslines), and/or based on their slope. The sorting can be done based on the geometry of the fault sticks, on an index written in the input file, or in the order found in the file. The header, if present, can be of fixed length (number of lines), or delimited on its last line by a keyword.

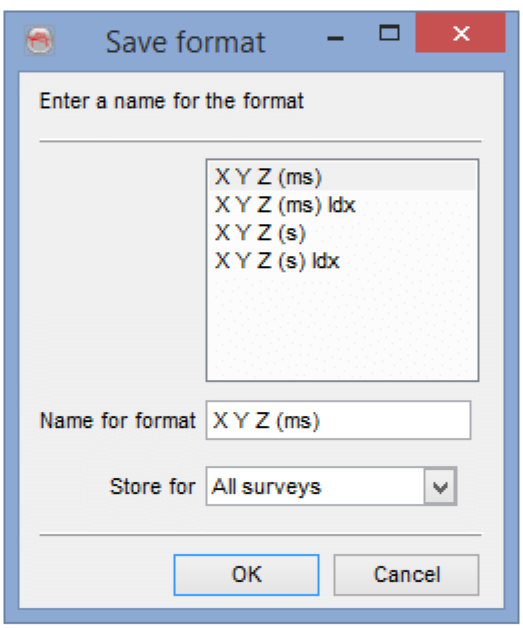
> **Note:** that OpendTect does not support crossing fault sticks (a fault plane cannot cross itself). If faults were picked on inlines, crosslines and horizontal slices, only the largest subset of the three will be used to import the faults.

Predefined and saved formats are available by pressing the  icon. Otherwise the format must be manually specified. The Define button gives access to the format definition window.



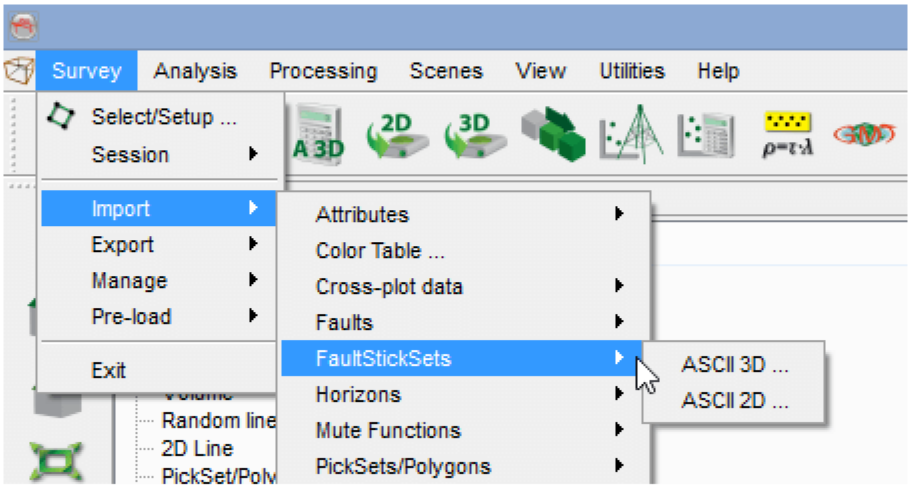
You must specify in the format definition the column numbers for the position; in terms of an X-Y pair, pick column, and optionally stick index (0 = no stick index). The Z units can be seconds, milliseconds or microseconds. Reading may be stopped at a specific line by providing the adequate keyword.

It is recommended to save the format definition for a later use and QC, by clicking on the  icon . In pop-up window, write the name of the format and store it. The format can be stored at different levels (All surveys, Current survey, Current OpendTect user level) depending on the usage. Press OK after having provided the name of the fault to be imported.



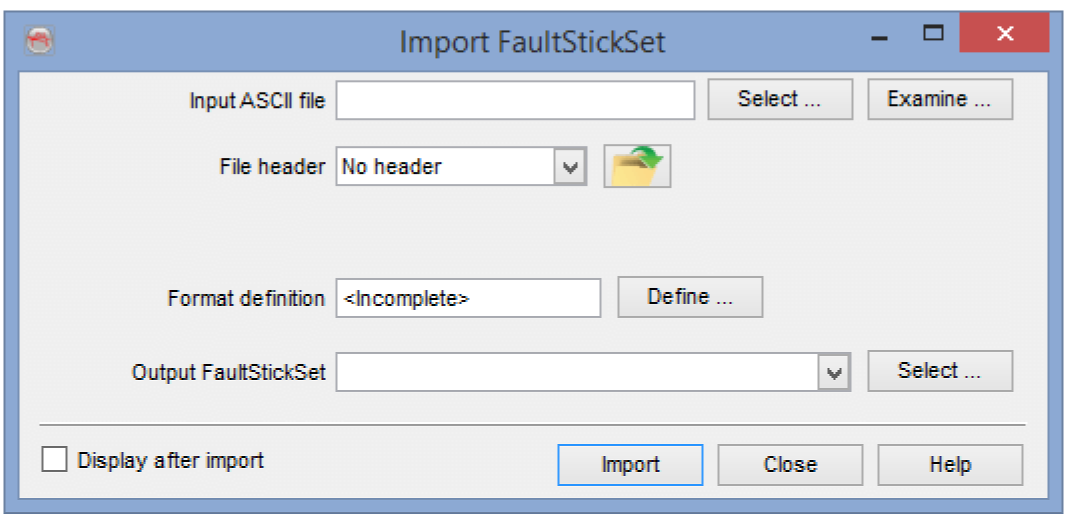
Import FaultStickSets

Fault stick sets are the editable version of the fault planes. Fault stick sets are fully editable objects either for faults interpretation , or later as fault input to correct for the fault throw. They can be imported in OpendTect via *Survey* > *Import* > *FaultStickSets*, from Ascii files of from GeoFrame Workstation (plugin).



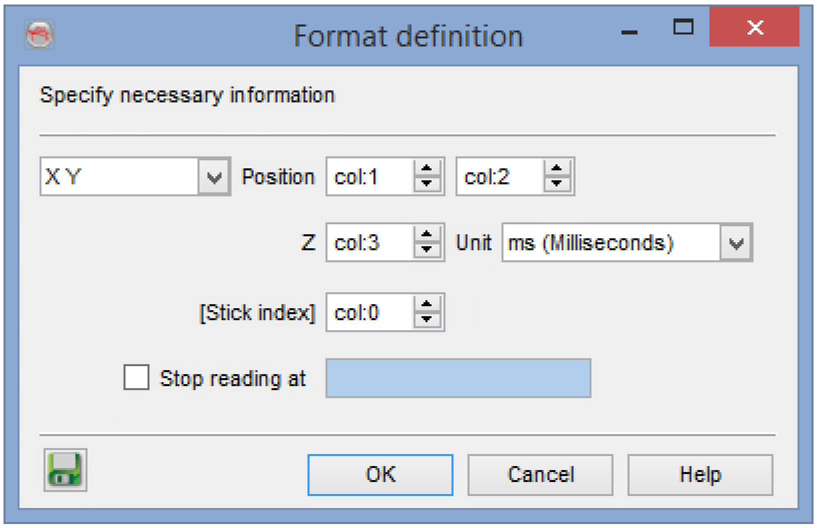
Import FaultStickSets Ascii 3D

Select the input ASCII file. You can display the input file by pressing the *Examine* button. The input file should be column sorted with one point per record (line).



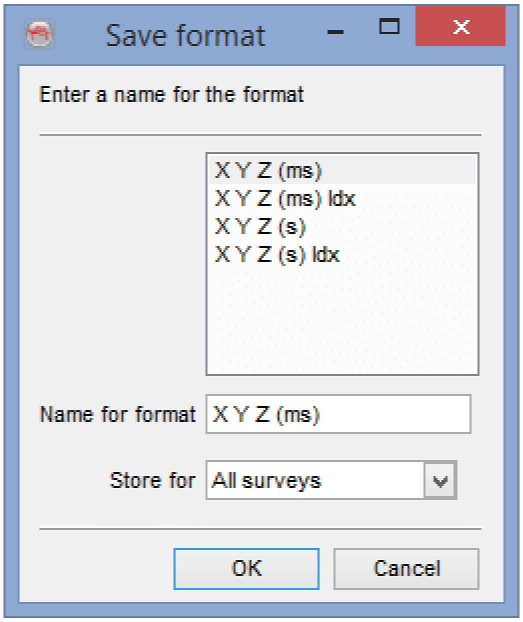
The important point is to specify the presence of a *file header* and the file *format definition*. The header, if present, can be of fixed length (number of lines), or delimited on its last line by a keyword.

Predefined and saved formats are available by pressing the  icon. Otherwise the format must be manually specified. The *Define* button gives access to the format definition window.



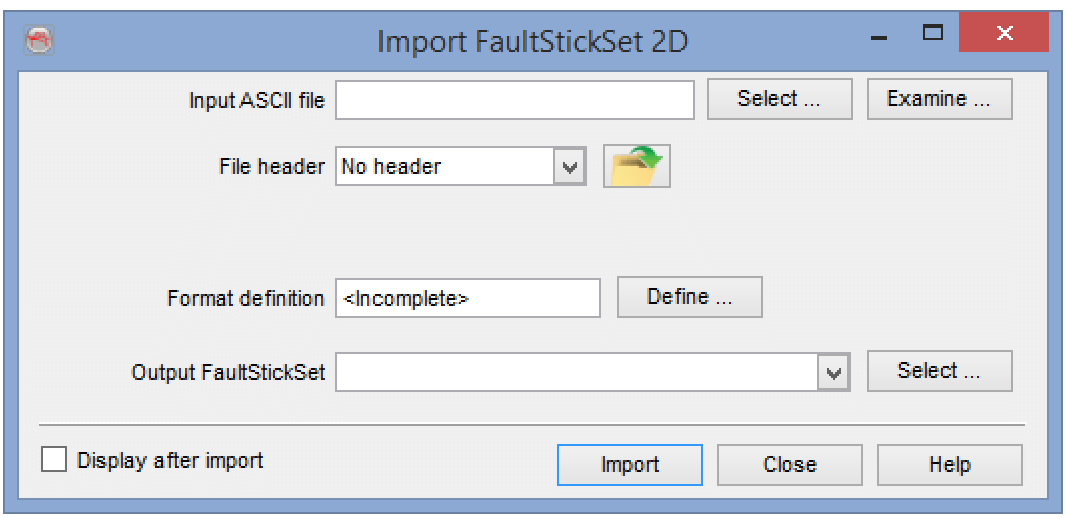
You must specify in the format definition the column numbers for the position, in terms of an X/Y pair, pick column, and optionally stick index (0 = no stick index). The Z units can be seconds, milliseconds or microseconds. Reading may be stopped at a specific line by providing the adequate keyword.

It is recommended to save the format definition for a later use and QC, by clicking on the  icon . In pop-up window, write the name of the format and store it. The format can be stored at different levels (All surveys, Current survey, Current OpendTect user level) depending on the usage. Press OK after having provided the name of the name of the faultstickset to be imported.



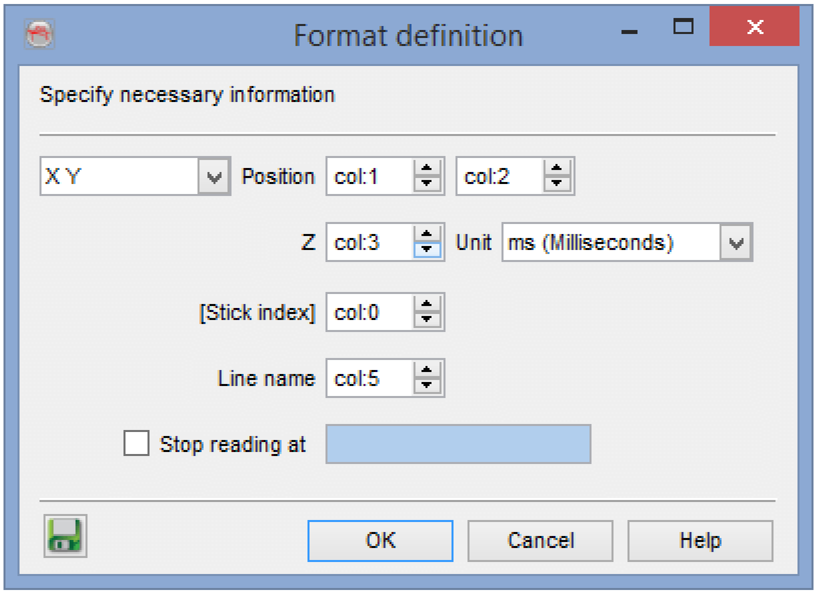
Import FaultStickSets Ascii 2D

Select the input Ascii file. You may display the input file by pressing the *Examine* button. The input file should be column sorted with one point per record (line).



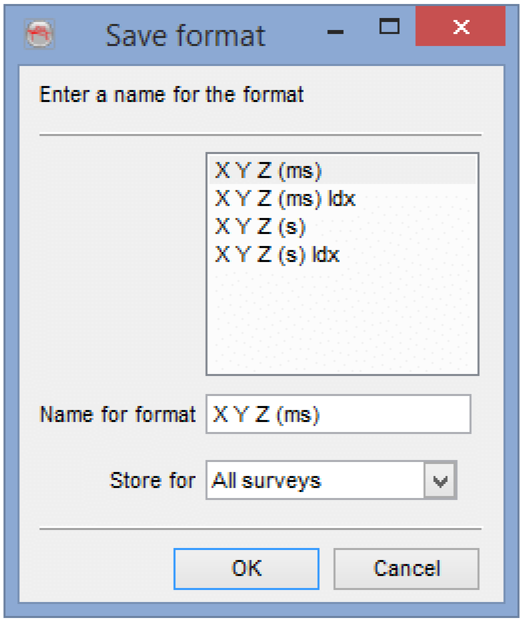
The main work is to specify the presence of a *file header*, and the file *format definition*. The header, if present, can be of fixed length (number of lines), or delimited on its last line by a keyword.

Predefined and saved formats are available by pressing the  icon. Otherwise the format must be manually specified. The *Define* button gives access to the format definition window.



You must specify in the format definition the column numbers for the position, in terms of an X-Y pair, pick column, and optionally stick index (0 = no stick index). The Z units can be seconds, milliseconds or microseconds. The name of the 2D line(s) must also be provided. Reading may be stopped at a specific line by providing the adequate keyword.

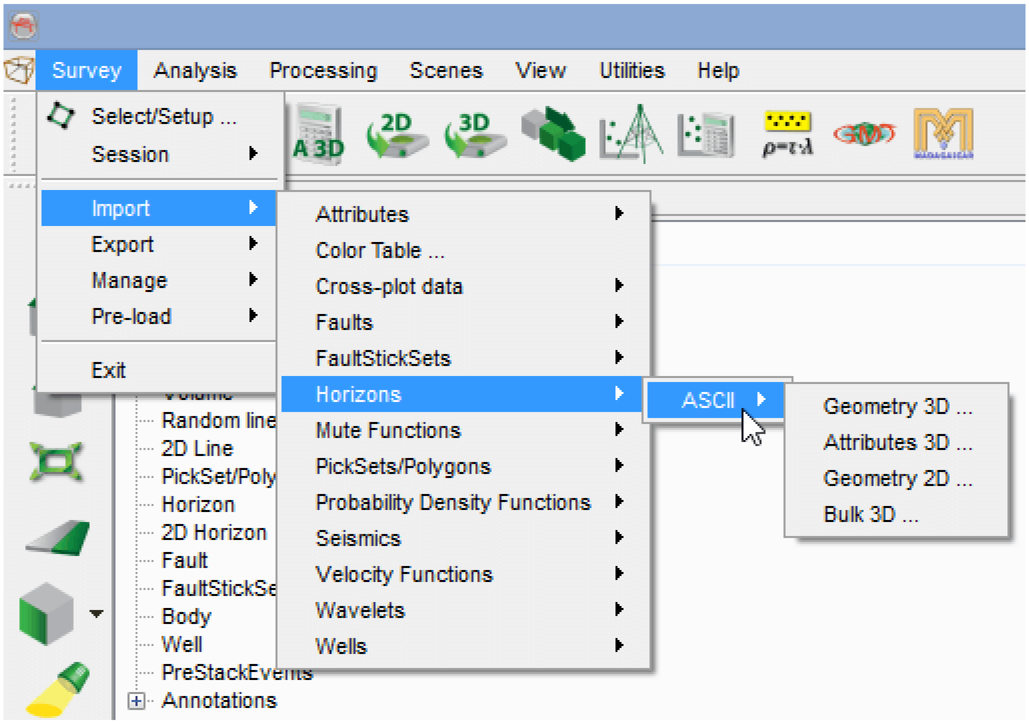
It is recommended to save the format definition for a later use and QC, by clicking on the  icon . In pop-up window, write the name of the format and store it. The format can be stored at different levels (All surveys, Current survey, Current OpendTect user level) depending on the usage. Press OK after having provided the name of the faultstickset to be imported.



Import Horizons

Horizons interpreted on 3D and 2D seismic data and (attribute) grids can be imported in a OpendTect survey via *Survey* > *Import* > *Horizons*. The grids are called "Surface data" in Opendtect and are attached to 3D horizons.

The horizon import supports the followings:



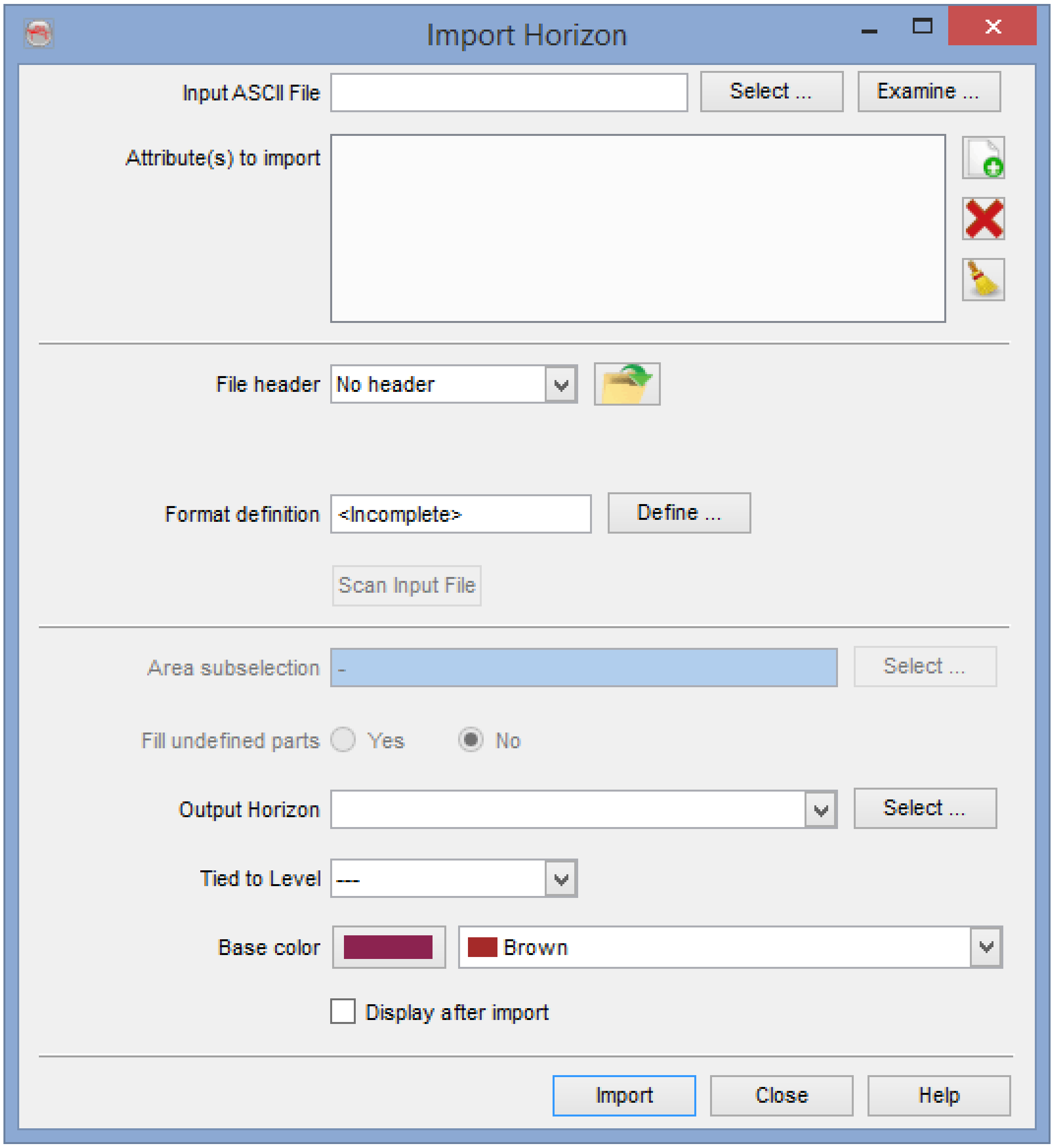
The standard input data is *Ascii* files. Three options are available (explained in the following subsections):

- Geometry 3D: Import horizons interpreted on a 3D grid (e.g. XYZ)
- Attributes 3D: Import an attribute grid as a surface data, for a given 3D horizon.
- Geometry 2D: Import horizons interpreted along 2D lines.
- Bulk 3D: Import multiple 3D horizons from a single file

Import Horizon from Geoframe/Petrel: Horizons can also be imported into OpendTect from Geoframe 2D/3D (*Survey > Import > Horizon > GeoFrame > 2D or 3D*). The following dialog will pop up. Select the GeoFrame project and the survey name that contains the horizon to be imported.

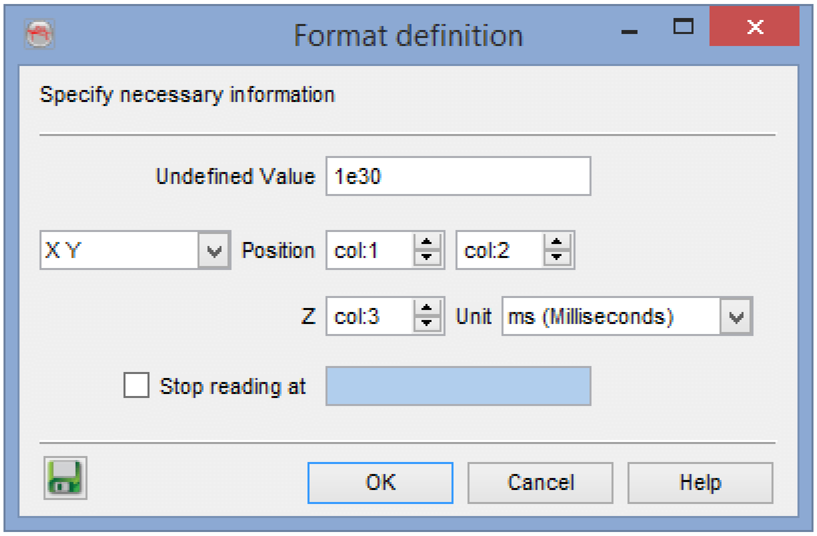
Geometry 3D

Select the input ASCII file. You may display the input file by pressing the *Examine* button. Available grids (attributes) present in the input file may also be imported simultaneously. The input file should be column sorted with one point per record (line).



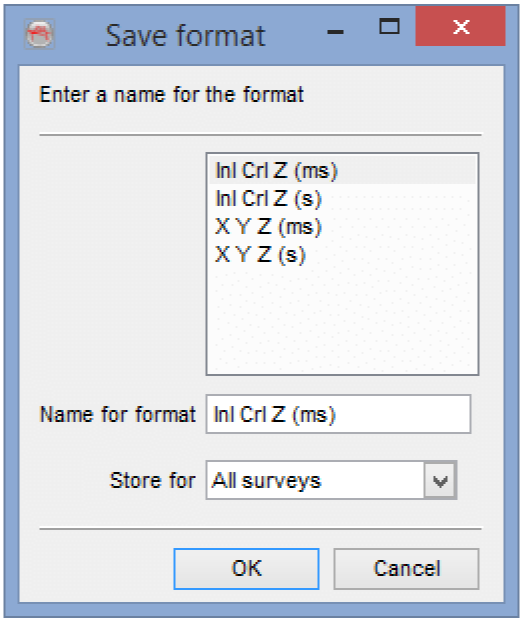
The main work is to specify the presence of a *file header* and the file *format definition*. The header, if present, can be of fixed length (number of lines), or delimited on its last line by a keyword.

Predefined and saved file formats are available by pressing the  icon. Otherwise the format must be manually specified. The *Define* button gives access to the format definition window.



You must specify in the format definition window the column numbers for the position, in terms of an X-Y pair or an inline-crossline pair, and the pick column. Picks that should not be read must all have the same numerical value, which is to be filled in as the "Undefined value". The Z units can be seconds, milliseconds or microseconds. Optionally, if attributes were added in the Import Horizon window, additional columns with given attribute(s) name(s) will also appear in this format definition window. Reading may be stopped at a specific line by providing the adequate keyword.

It is recommended to save the format definition for a later use and QC, by clicking on the  icon . In pop-up window, write the name of the format and store it. The format can be stored at different levels (All surveys, Current survey, Current OpendTect user level) depending on the usage. Press *Ok* when done.



It is highly recommended to scan the input file after providing its format, and prior to the actual import. The scanned information will pop-up and error(s) or warning(s) may suggest a change of the format definition.

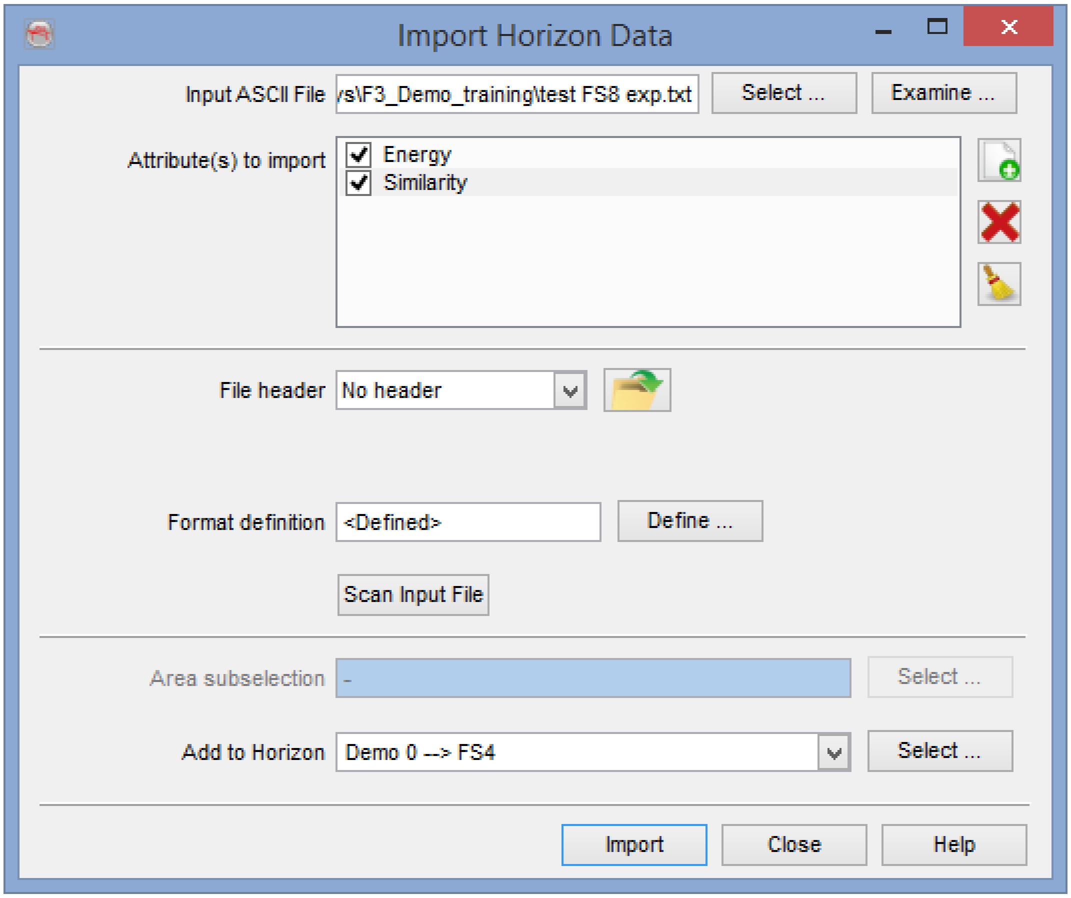
The area sub-selection can be used in two ways:
- Reduce the amount of data to be loaded: By reducing the inline/crossline ranges
- Specify a new grid size if a coarse input grid should be gridded during import: By decreasing the inline/crosslines steps to the survey steps (minimum).

The option *Fill undefined parts* will be toggle on if gaps were found during scan. A triangulation to the convex hull with an interpolation where the maximum size is the input grid step (in XY units, thus meters or feet), and *Keep holes larger than* toggled off, should be the optimal settings in most cases.

> **Note:** *Tied to level* is additional option specifically designed to tie horizons to well markers, for correlation purposes. In order to define the stratigraphic information of the survey, please read about Manage Stratigraphy.

Attributes 3D

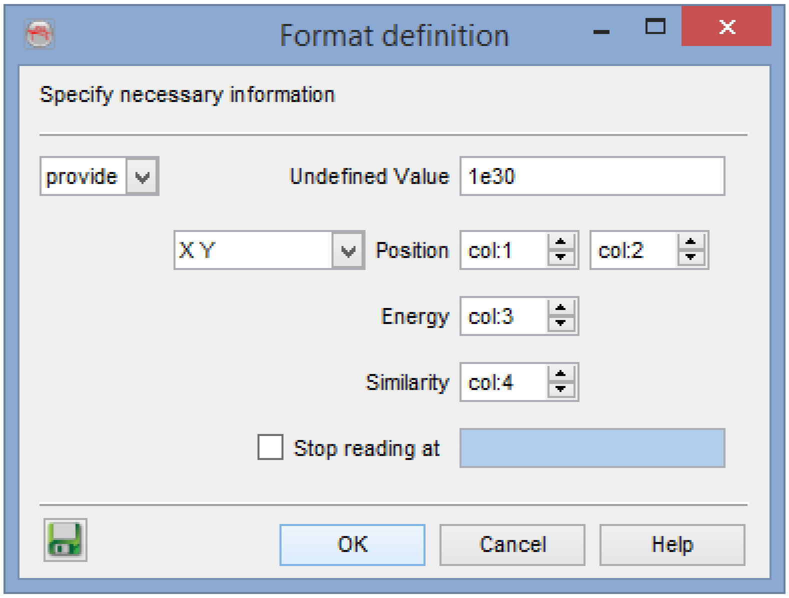
This window is used to import grids from ascii files and attached them to *Existing* 3D horizons. *Select* the input Ascii file. You may display the input file by pressing the Examine button. The input file should be column sorted with one point per record (line).



Grid names must first be provided in front of *Select Attribute(s) to import*. This can be done by pressing *Add new* right of it, and providing each time a new grid name. This will populate the list of importable grids. Only the highlighted grids will be imported, which is why each new grid is highlighted after providing its name.

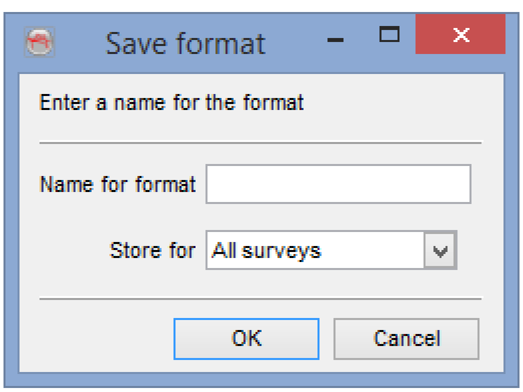
Next, the presence of a file header must be specified and the file format definition must be provided. The header, if present, can be of fixed length (number of lines), or delimited on its last line by a keyword.

Predefined and saved file formats are available by pressing the  icon. Otherwise the format must be manually specified. The Define button gives access to the format definition window.



You must specify in the format definition window the column numbers for the position, in terms of an X-Y pair or an inline-crossline pair, and the grid(s) column(s). Grid values that should not be read must all have the same numerical value, which is to be filled in as the *Undefined value*. Reading may be stopped at a specific line by providing the adequate keyword.

It is recommended to save the format definition for a later use and QC, by clicking on the  icon . In pop-up window, write the name of the format and store it. The format can be stored at different levels (All surveys, Current survey, Current OpendTect user level) depending on the usage. Please note that the full grid names will be saved as provided in the format definition. Press OK when done.

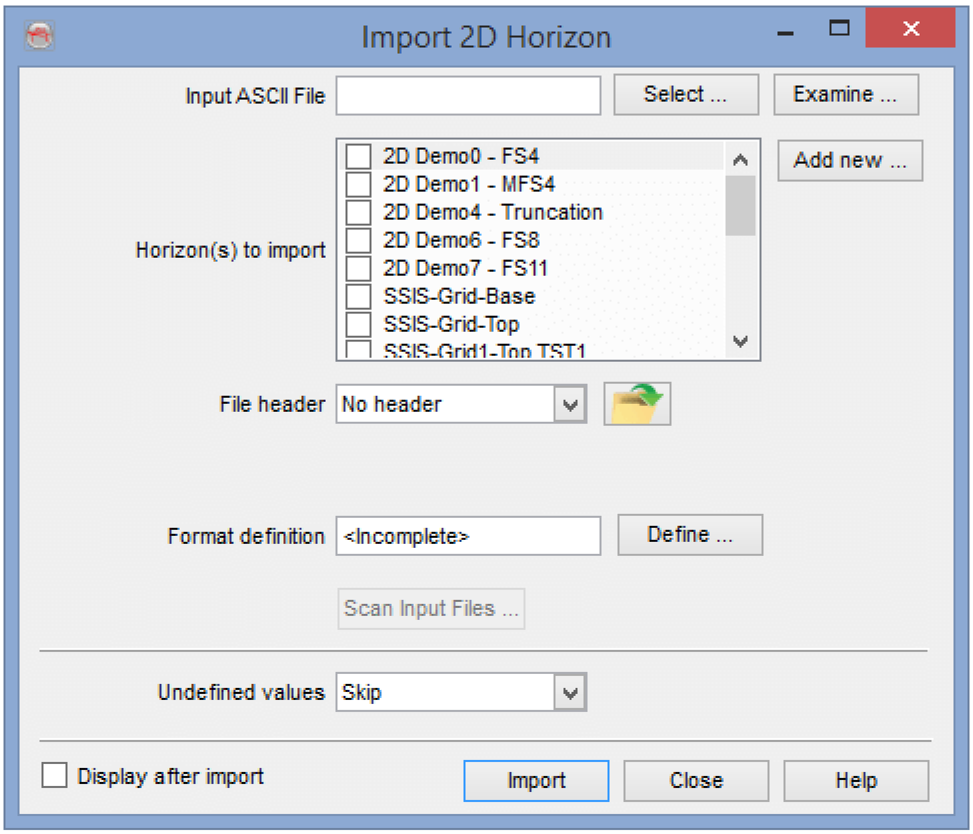


It is highly recommended to scan the input file after providing its format, and prior to the actual import. The scanned information will pop-up and error(s) or warning(s) may suggest a change of the format definition. The area subselection is essentially present to optionally reduce the amount of data to be imported, by reducing the inline/crossline range(s).

Finally an horizon must be provided, to attach the grid(s) to it. Grids will be accessible only after having loaded this horizon in the tree. Press *Go* to launch the import.

Geometry 2D

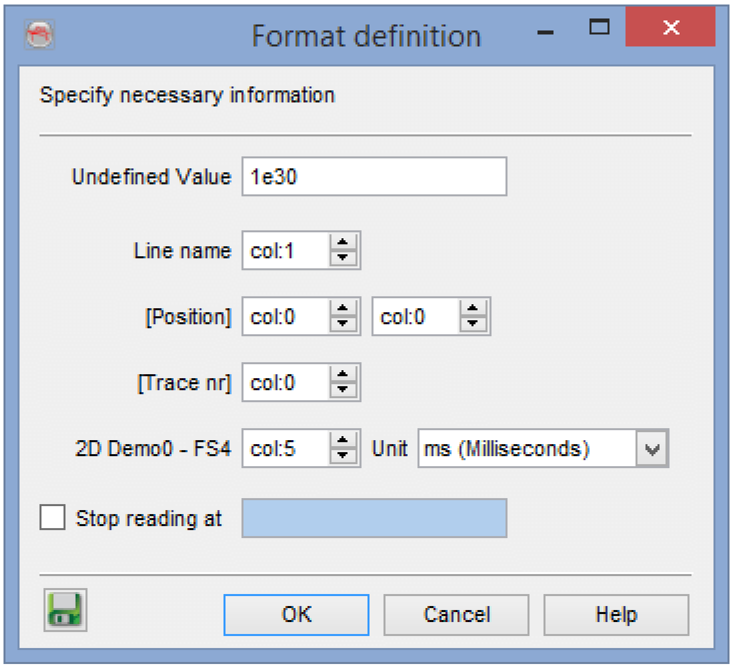
This window is used to import **2D interpretations** form ascii files. Select the input ascii file. You may display the input file by pressing the *Examine* button. The input file should be column sorted with one point per record (line).



Several 2D horizons can be imported at once. Their name should be provided in front of *Select Horizons to import*. This can be done by pressing *Add new* right of it, and providing each time a new horizon name. This will populate the list of importable horizons. Only the highlighted horizons will be imported, which is why each new horizon is highlighted after providing its name. Ctrl-left click may be used to highlight or deselect an horizon.

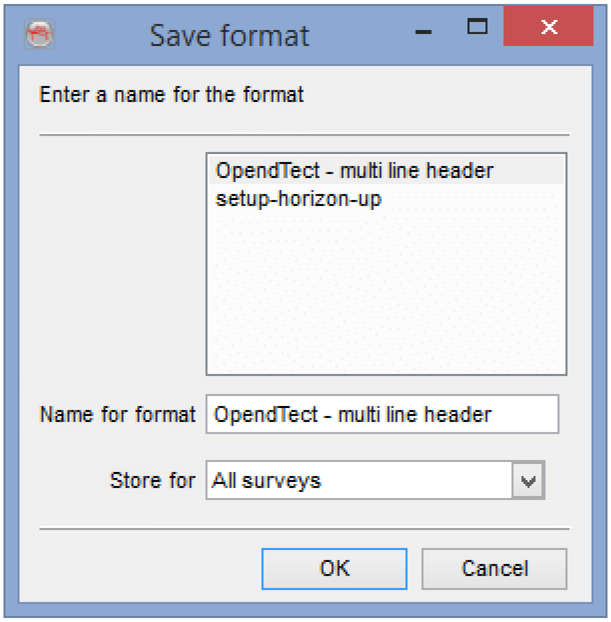
Next, the presence of a file header must be specified and the file format definition must be provided. The header, if present, can be of fixed length (number of lines), or delimited on its last line by a keyword.

Predefined and saved file formats are available by pressing the icon. Otherwise the format must be manually specified. The Define button gives access to the format definition window.



You must specify in the format definition window the line name, column numbers for the position, in terms of an X-Y pair or a unique trace number, and the horizon(s) column(s). Horizon Z values that should not be read must all have the same numerical value, which is to be filled in as the *Undefined value*. Reading may be stopped at a specific line by providing the adequate keyword.

It is recommended to save the format definition for a later use and QC, by clicking the icon . In pop-up window, write the name of the format and store it. The format can be stored at different levels (All surveys, Current survey, Current OpendTect user level) depending on the usage. Press OK when done.



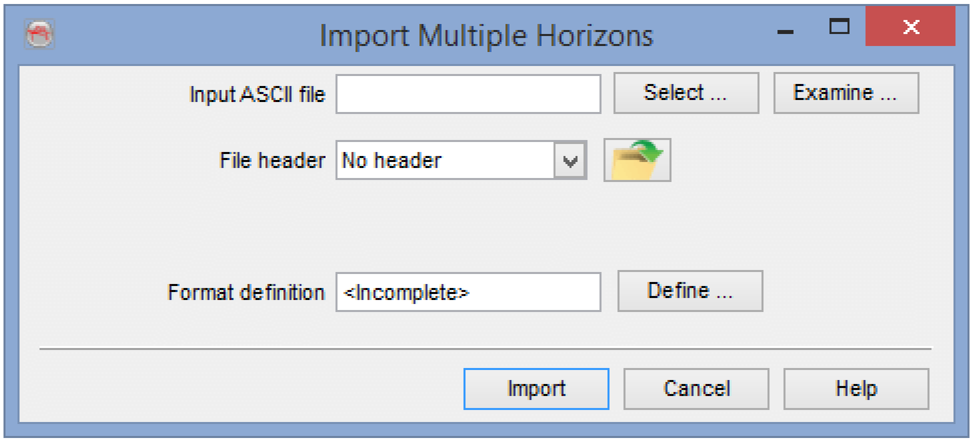
It is highly recommended to scan the input file after providing its format, and prior to the actual import. The scanned information will pop-up and error(s) or warning(s) may suggest a change of the format definition. Press *Go* to launch the import.

Bulk 3D

The bulk import tool allows for the import of multiple 3D horizons from one single file. The data is matched by name. This has the following implications:

The horizon name must appear on each line of the input file. The horizon name should not contain spaces, otherwise the matching with a given column number will not work as expected.

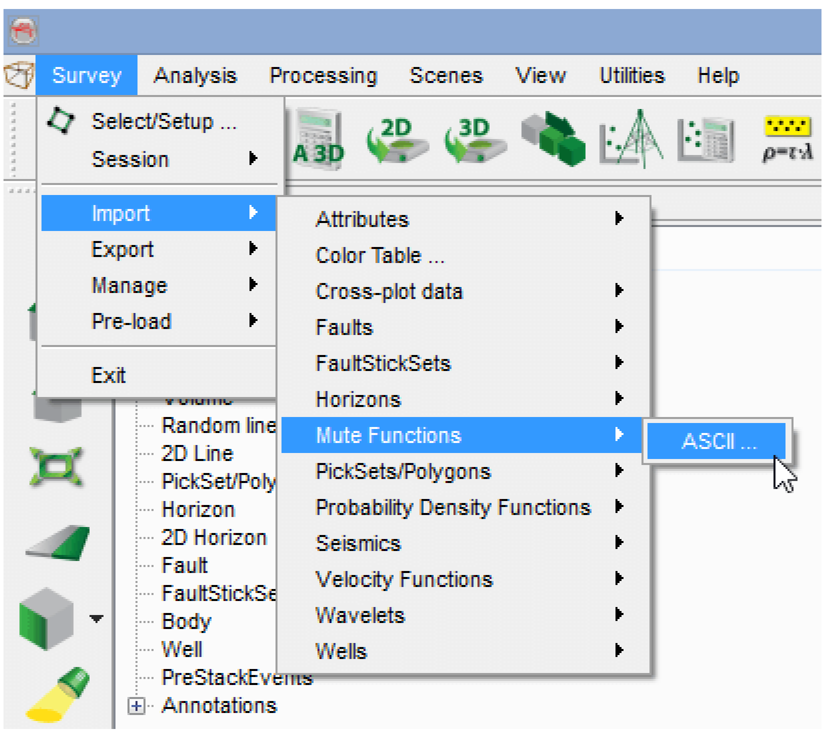
Apart from being a multiple horizon import tool, it behaves following the rules of the standard horizon import.



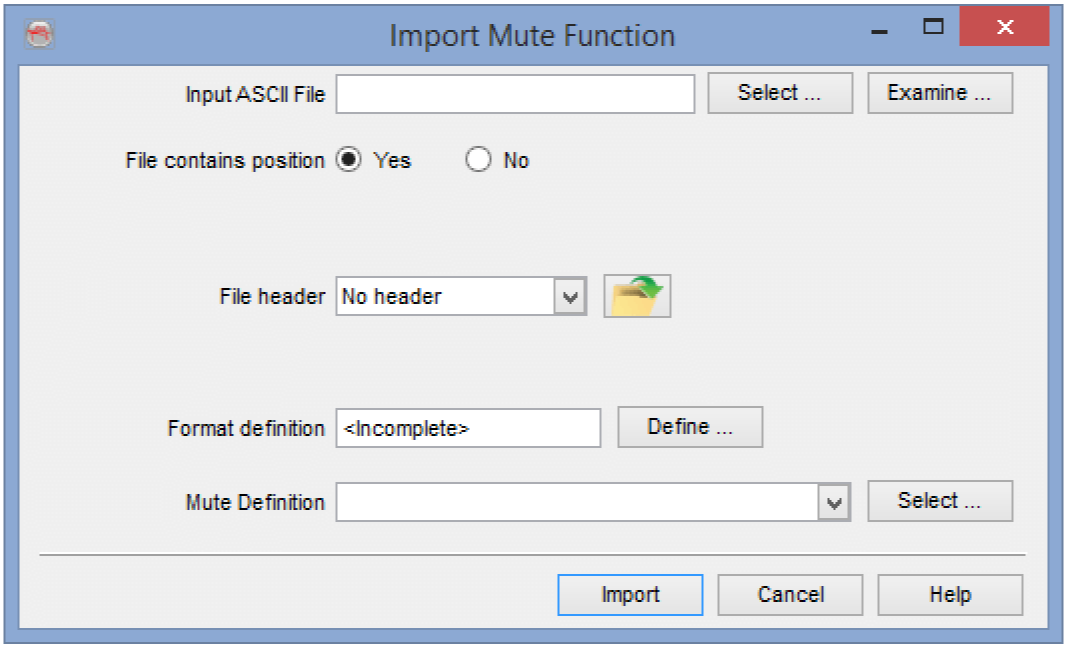
This option is not yet available for 2D horizons.

Import Mute Functions

Mute definitions can be used for pre-processing pre-stack seismic data.

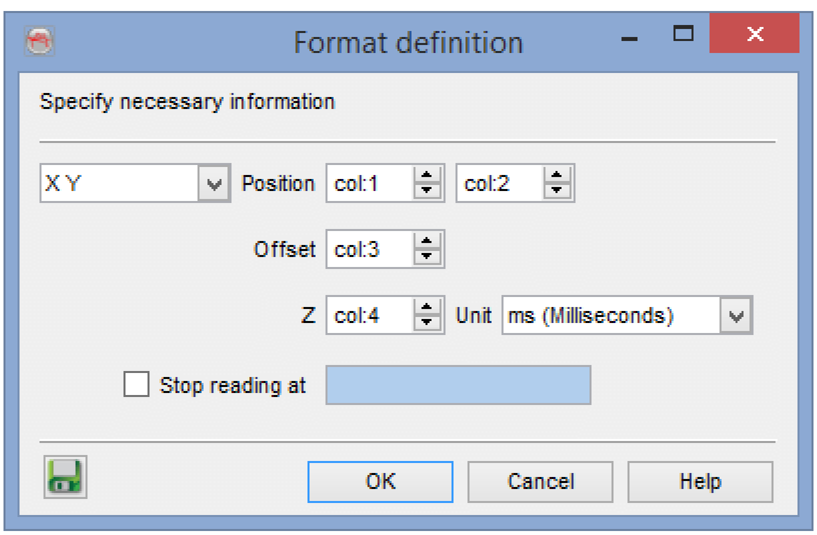


Mute definitions can be imported in OpendTect using Ascii files. The import window is launched from the OpendTect main menu (*Survey* > *Import* > *Mute definitions* > *Ascii*). Select the input Ascii file. You can display the input file by pressing the *Examine* button. The input file should be column sorted with one point per record (line).



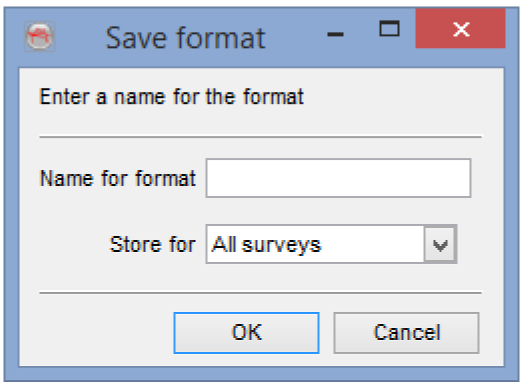
The main work is to specify the presence of a *file header* and the file *format definition*. The header, if present, can be of fixed length (number of lines), or delimited on its last line by a keyword. The mute definition can be either variable throughout the survey, in which case a position must be provided in the input file for all data points, or fixed. In this latter case, toggle *File contains position* to *No* and provide any location for the mute definition.

Predefined and saved file formats are available by pressing the icon. Otherwise the format must be manually specified. The *Define* button gives access to the format definition window.



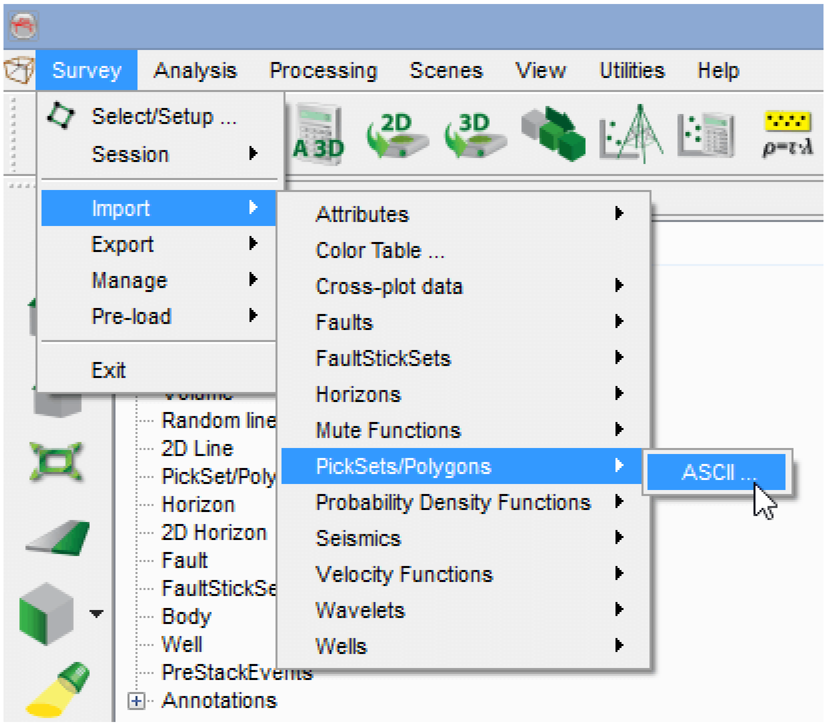
You must specify in the format definition window the column numbers for the position, in terms of an X-Y pair or an inline-crossline pair, and the pick column, in terms of an Offset-Z value pair. Picks that should not be read must all have the same numerical value, which is to be filled in as the "Undefined value". The Z units can be seconds, milliseconds or microseconds (meters of feet in depth surveys). Reading may be stopped at a specific line by providing the adequate keyword.

It is recommended to save the format definition for a later use and QC, by clicking on the  icon . In pop-up window, write the name of the format and store it. The format can be stored at different levels (All surveys, Current survey, Current OpendTect user level) depending on the usage. Press OK when done.

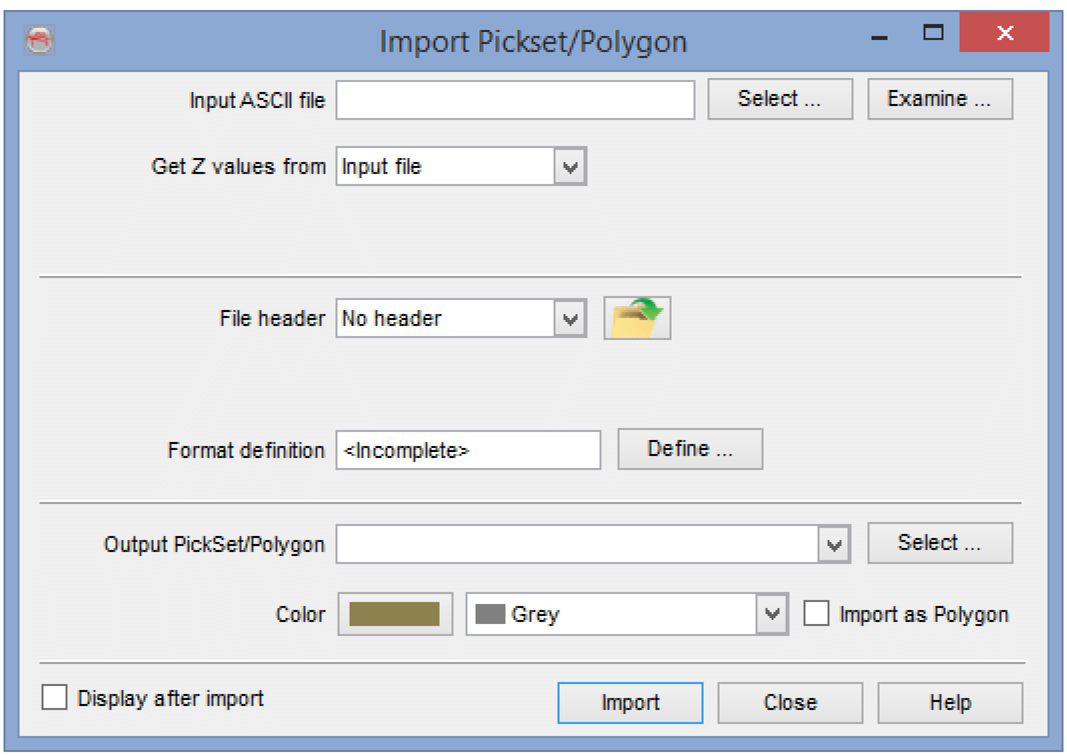


Import PickSets & Polygons

Point/vector data can be loaded in OpendTect from *Survey -> Import -> PickSet/Polygon*.

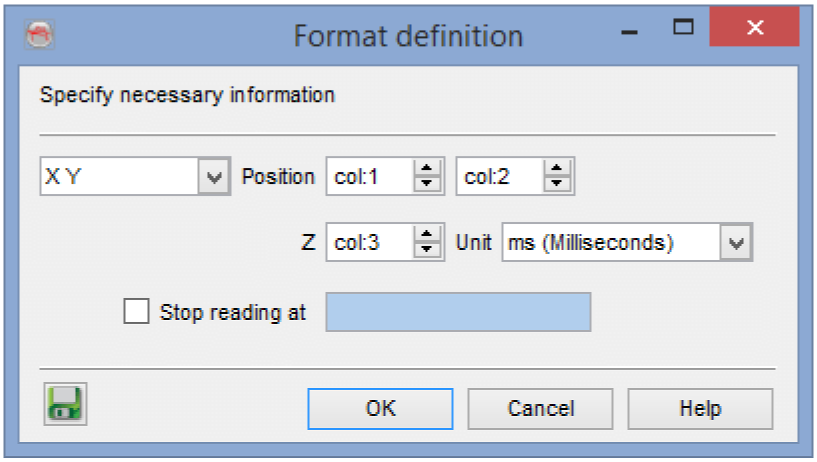


Select the input Ascii file. You can display the input file by pressing the *Examine* button. The input file should be column sorted with one position per record (line).



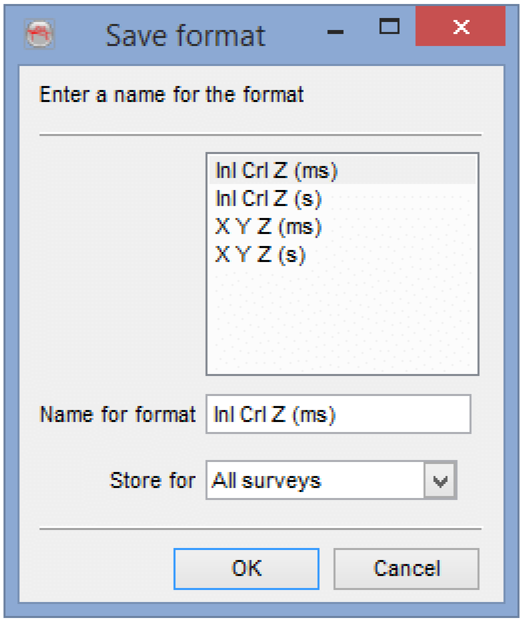
The main work is to specify the presence of a *file header* and the file *format definition*. The header, if present, can be of fixed length (number of lines), or delimited on its last line by a keyword.

Predefined and saved file formats are available by pressing the icon. Otherwise the format must be manually specified. The *Define* button gives access to the format definition window.



You must specify in the format definition window the column numbers for the position, in terms of an X/Y pair or an inline-crossline pair, and the pick column. Picks that should not be read must all have the same numerical value, which is to be filled in as the "Undefined value". The Z units can be seconds, milliseconds or microseconds. Reading may be stopped at a specific line by providing the adequate keyword.

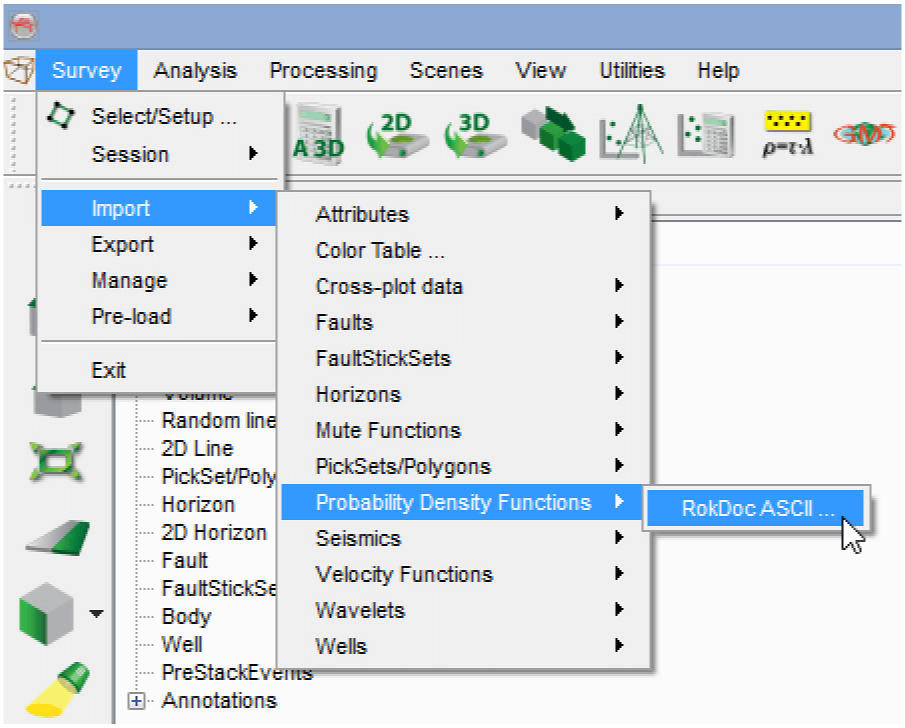
It is recommended to save the format definition for a later use and QC, by clicking on the icon . In pop-up window, write the name of the format and store it. The format can be stored at different levels (All surveys, Current survey, Current OpendTect user level) depending on the usage.



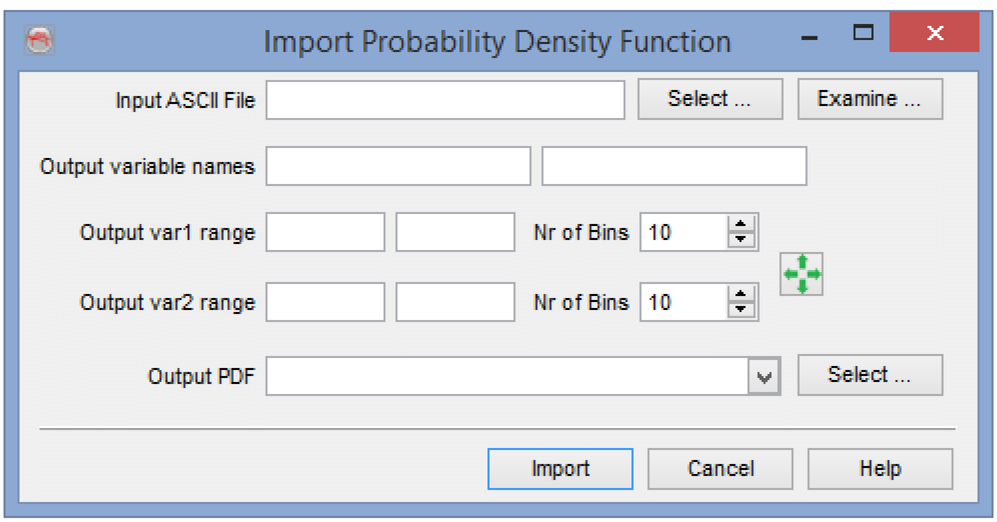
The option *Import as polygon* will flag this specific datatype to the loaded data. It also adds as constraint during loading that the points are ordered in the expected way. The import tool will not apply any sorting.
Press OK when done.

Import Probability Density Functions

Probability density functions can be imported in order to run Bayesian classifications. The manage tool can later be used to edit the PDF before running the Bayesian classification.



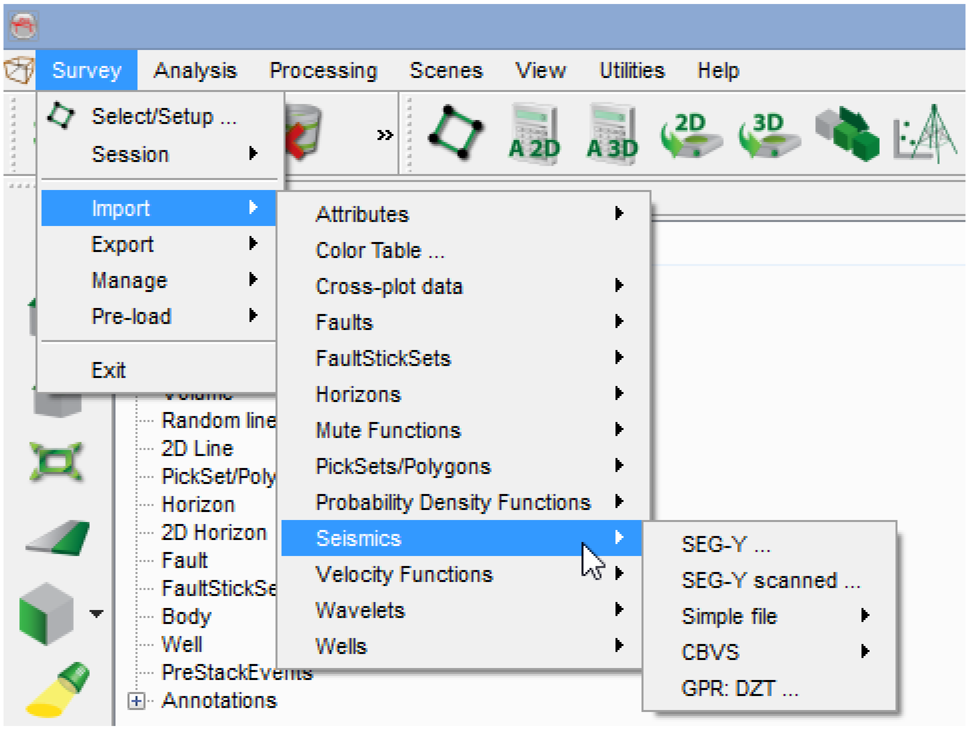
RokDoc formatted data is required for importing PDF in OpendTect. After having selected the input file, the two contained variables will be shown in the import window, together with their amplitude ranges and bin size.



The variable names and parameters may be modified before pressing the *Go* button that will launch the import. The  icon to the right can be used to quickly extend both variable ranges by one bin size outwards.

Import Seismics

Volumes and 2D Lines can be imported in the Survey menu from files in different modes:



SEG-Y: By converting a SEG-Y file to an OpendTect file.
SEG-Y scanned: By referencing (linking) a SEG-Y file to the survey without creating a new file.
Simple file: From a regular ascii or binary file, with or without header.
CBVS: By copying or linking an existing OpendTect (CBVS) volume.
Petrel: Import seismic data from other software (depending on system setup)
GPR-DZT: Import the files made by GSSI Ground Penetrating Radar (GPR) systems in the 'DZT' format.

SEG-Y

SEG-Y is the standard way to share volumes/lines of data. In OpendTect the files are loaded with a rigorous respect to the SEG standards using a loading wizard. Most SEG-Y files will be imported in a few clicks, and a number of exceptions can be set to load the most problematic data. Nevertheless there are a few guidelines that must be honoured:

- The traces must be sorted either by inlines and then crosslines or by crosslines and then inlines.

- The gathers of pre-stack data must be consecutive and ordered by increasing offset (i.e. no common offset sorting).
- Inline/crosslines or coordinates (and offset) must be written in every single trace header. Separate navigation data is not supported.
- The traces must have a fixed length.
- There is no support of extended textual headers.

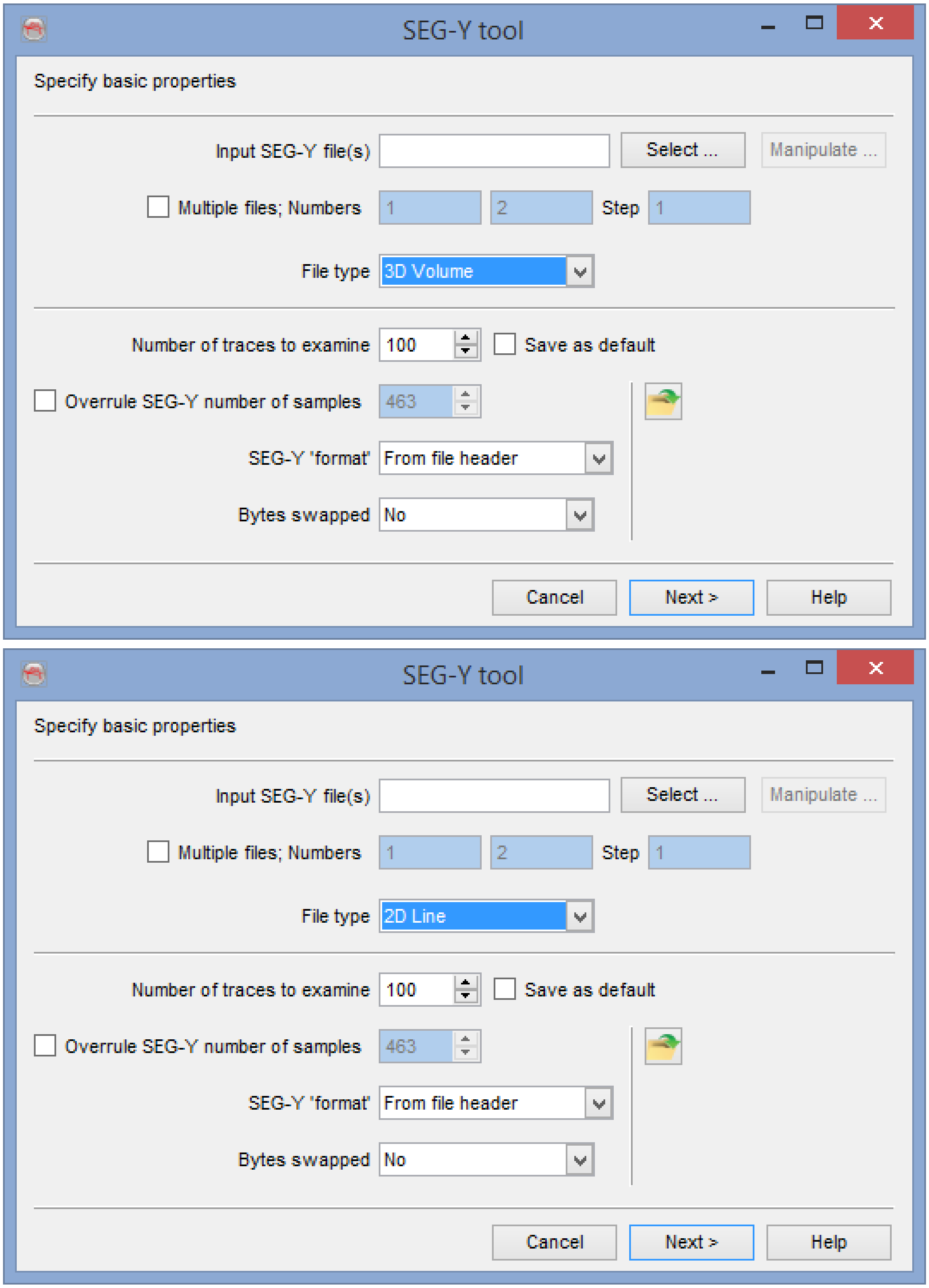
Once the above criteria are respected, you will then enjoy a large freedom:

- Gaps can be present.
- The traces can start at any time/depth, even negative.
- The files can be merged during import, re-scaled using linear equations,and the storage format can be changed.

> **Note:** Please, read the entire chapter before asking for support. If you need support please send us a screenshot of the first step, examine window, import window and scan report together with a description of your problem.

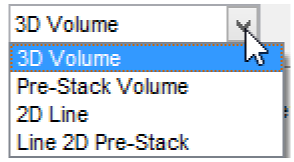
SEG-Y Import Preparation

The generic wizard called the "SEG-Y tool" can load pre-stack and post-stack seismic data, from one or multiple SEG-Y files (2D and 3D). This wizard must be launched from the survey menu. It consists of two mandatory steps: The data selection (first step) and the loading of the new file by providing the key parameters.

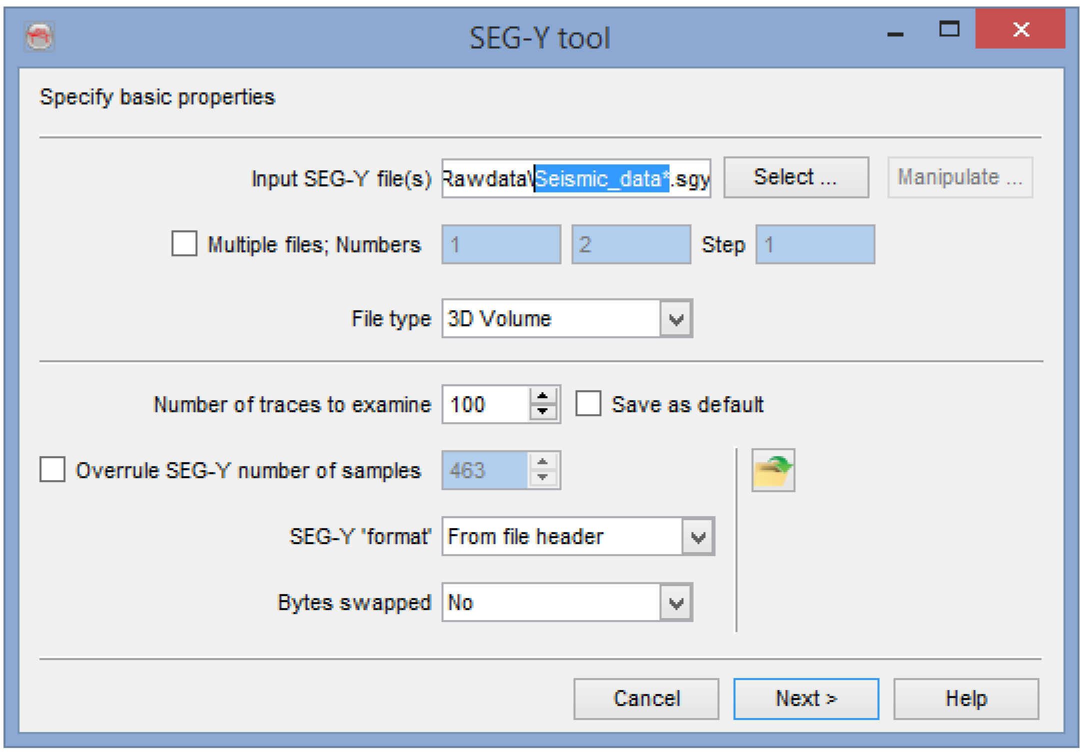


SEG-Y Tool Window (3D and 2D)

<u>Mandatory</u> workflow in the first step:

1. Select your (first) file using the file browser.
2. Choose the correct file type between: 

3. If multiple SEG-Y files representing one 3D volume (pre-stack or post-stack) have to be loaded you must specify the number of files to be merged during loading. The input SEG-Y files must contain blocks or inlines. File names must be indexed like file1.sgy, file2.sgy..., any file can be selected and the index must be replaced in the input field by a wildcard (*):

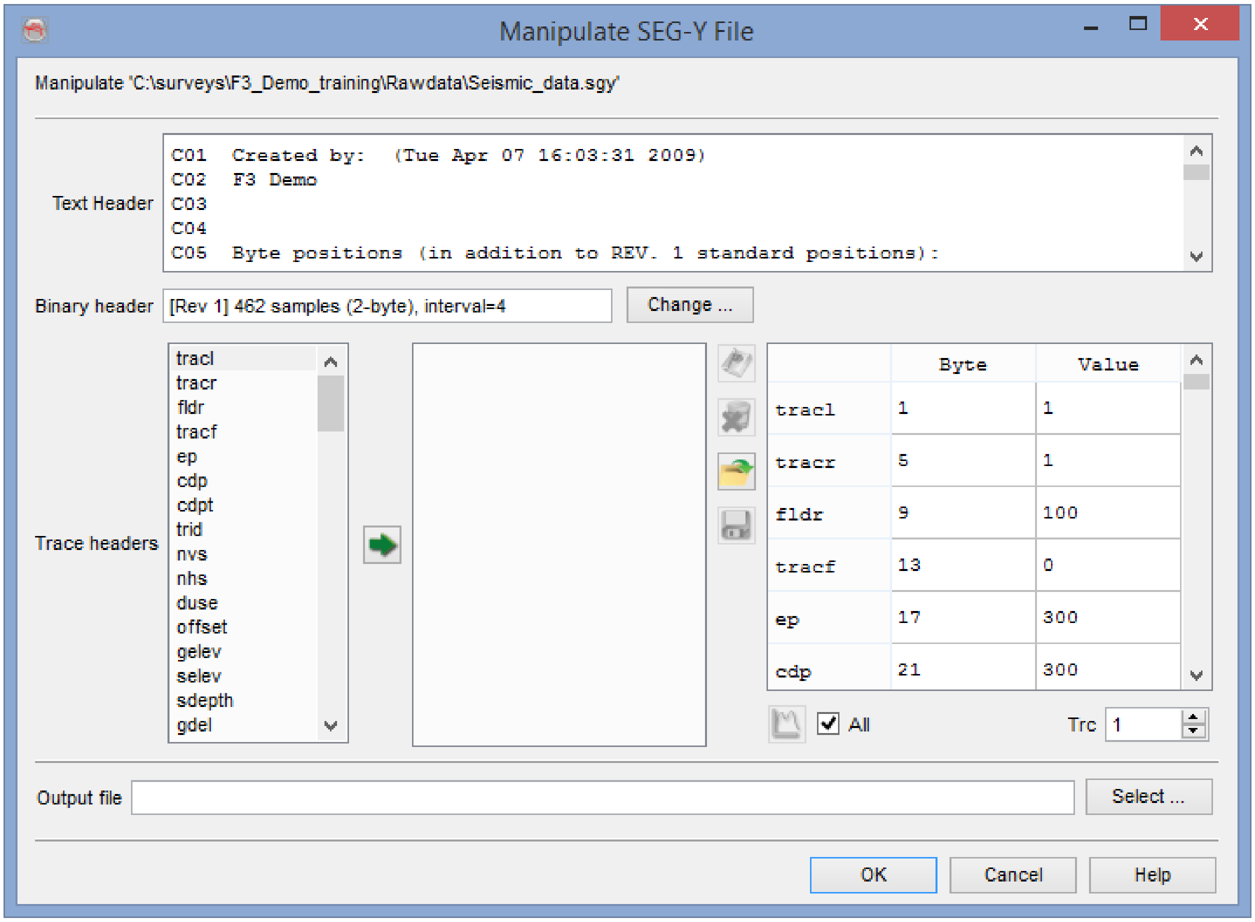


In most cases you can press the *Next* button after you have done the two actions listed above. Only if you have a proiri knowledge that the file might be having some problems you would like to perform one of the following actions before pressing *Next*.

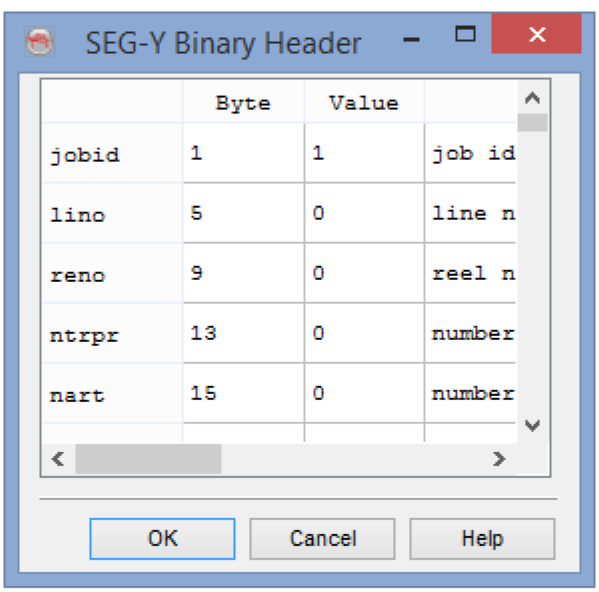
<u>Optional</u> actions in the first step:

1. Manipulating a SEG-Y file:The text, binary and the trace headers may be edited by using this option. This is accessed by clicking on the Manipulate button.

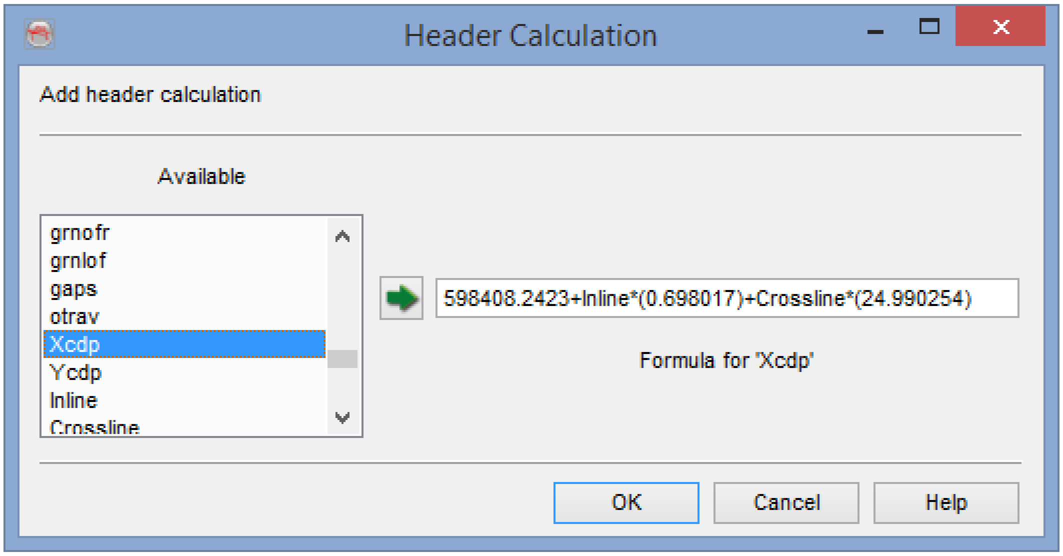
The text header can be directly edited.

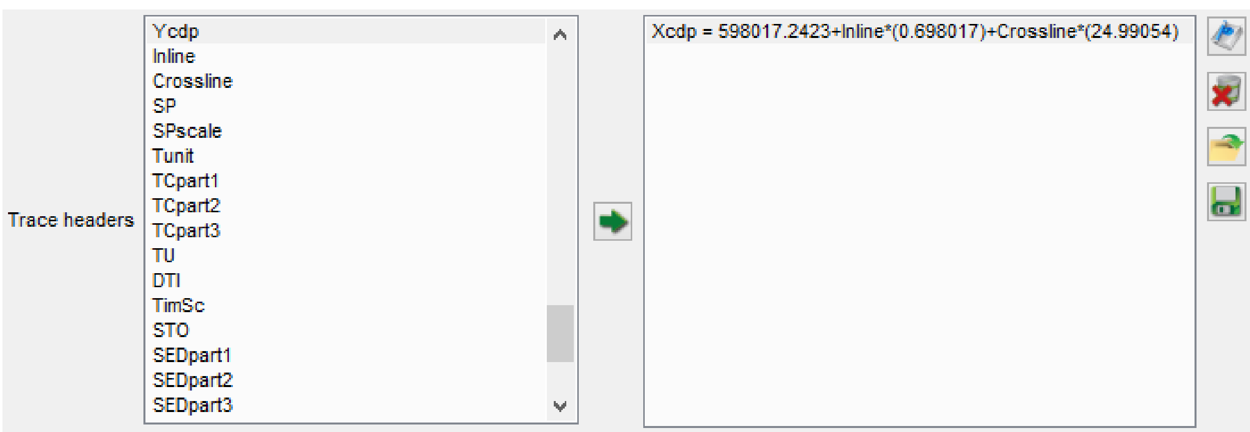


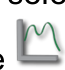
The binary header can be edited by clicking on *Change* button and correcting the *Byte* and *Value* fields (where required) in the pop-up window shown below.



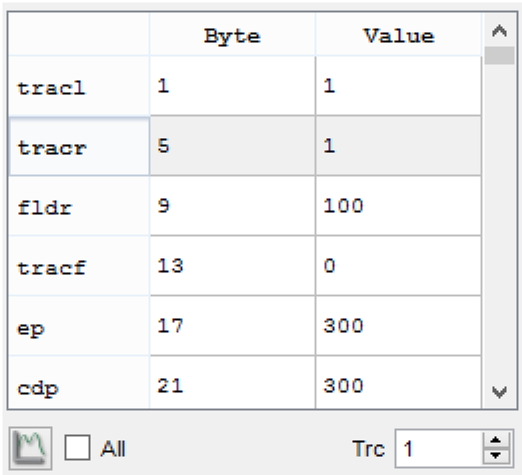
On the trace headers a *Header Calculation* can be performed to change their *Value*. This can be done by selecting one of the trace headers from the list and pressing the  icon.



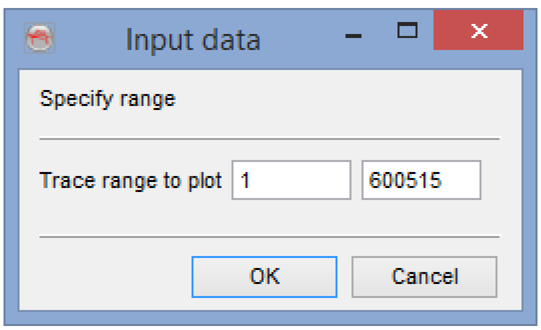


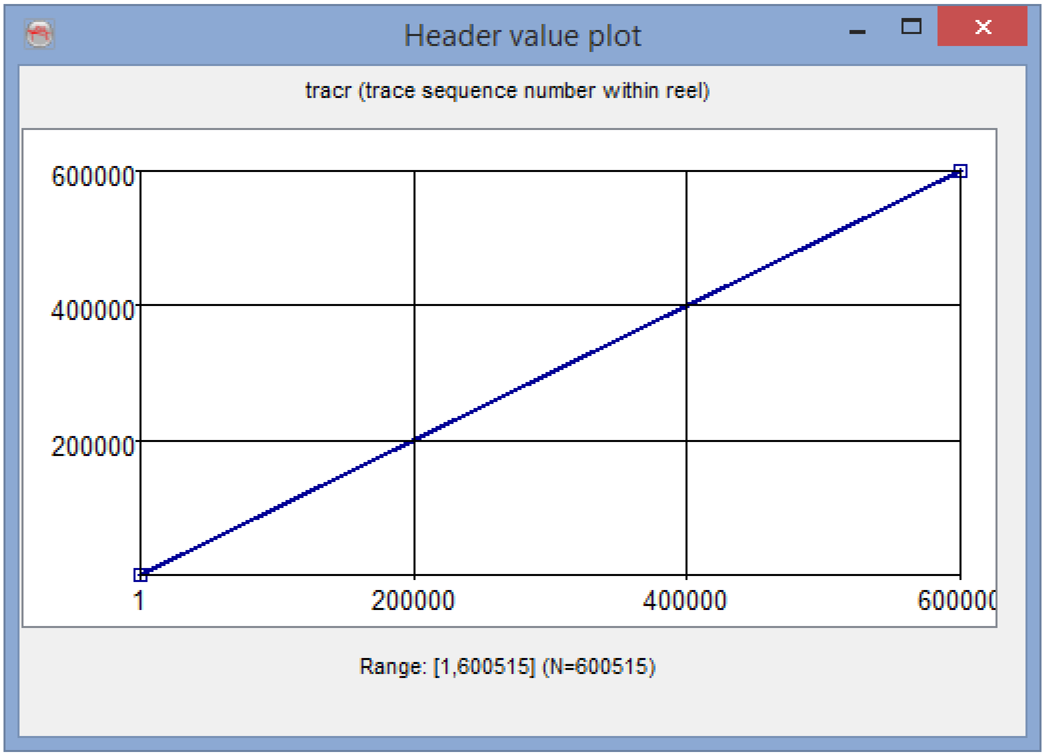
These calculation sets may be edited, deleted and saved for later use by using ,  and  icons respectively. Any previously saved sets can be restored by pressing . For visualization purposes one or more trace headers can be selected from the following table and plotted against a range of trace numbers using the  icon.

For information about the supported operators, functions and constants, please refer to the Mathematics attribute.



If the option *All* is not ticked, the following window will pop-up for choosing the required range of traces.





Inline header value plot

All the trace headers for any particular trace number can be viewed by changing the value in front of the *Trc* field. Finally, the modified SEG-Y file is outputted by specifying a name and location to save.

2. Number of traces to examine: A set of traces may be examined at the beginning of the file. Looking at the output is strongly recommended but not mandatory for a successful import. This can be disabled by entering "0". Pressing *Save as default* will save the number of traces to examine in your user settings.

3. Overrule SEG-Y number of sample: If the number of samples per trace is not or incorrectly written in the line and/or trace headers you can overrule it here.

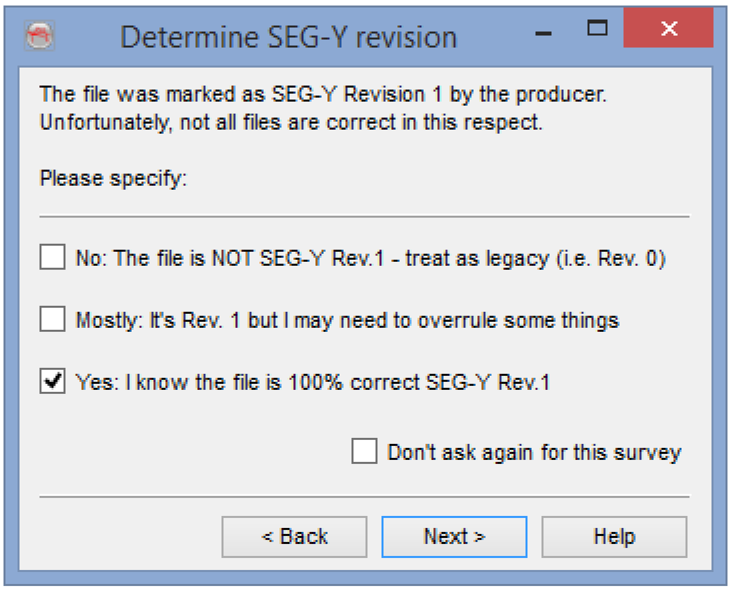
4. SEG-Y format: If the format of the data samples is not or incorrectly written in the line header you can overrule it here.

5. Bytes swapped: Most header values and/or data samples are written using several bytes for each word/sample. Therefore a correct byte order is a necessity. Thus, if the trace headers don't seem correct or if the sample values are unexpectedly large numbers, byte swapping might be required.

After pressing *Next* a popup question (SEG-Y revision) and the *SEG-Y examiner* window will appear. It is recommended to answer the questions and examine the traces before moving on to the second wizard window.

SEG-Y Revision

After preparing the SEG-Y import you will be asked to revise the type of your SEG-Y file (s). The time consumption and the smoothness of the import phase will depend on the answer.



1. **[No]:** the file is NOT SEG-Y Rev.1 - treat as legacy (i.e. Rev.0) - For un-lucky users the revision 1 flag might wrongly be set to "Yes" in the line header while obviously the file does not comply with the SEG-Y revision 1 norm. It may happen when the software blindly copies entire headers without refreshing all the necessary characters. "Rev.0" is an older way of loading SEG-Y files: The inline and crossline offset (bytes) must be present in the trace headers and their offsets must be provided to the software since it may vary from file to file.

2. **[Mostly]:** It's Rev.1 but I may need to overrule some things - The file is 100% SEG-Y Rev.1 but you would like to overrule some particular information e.g. Coordinates, Sampling rate and Start time etc.

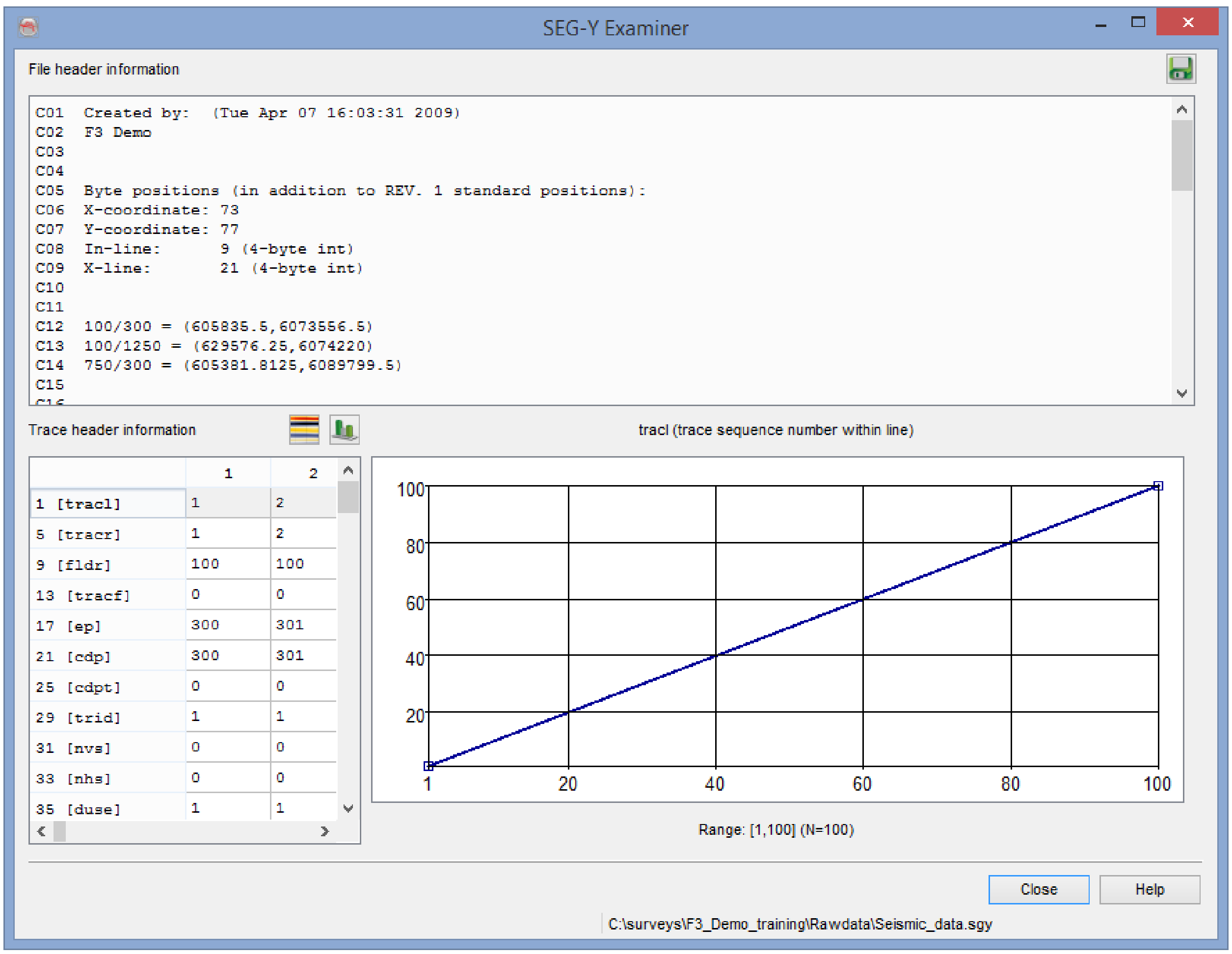
3. **[Yes]:** I know the file is 100% correct SEG-Y Rev.1 - For lucky people, this is by far the most easy and quick way to import your SEG-Y file. No additional settings are required except the final output volume/line name in the OpendTect database.

After selecting appropriate option, press OK to reach the import window.
You can save the answer to this question in your survey settings by activating the option "Don't ask again for this survey". If you set this flag by mistake and wish to go back you need to edit the ".defs" file (in the OpendTect survey directory) with a text editor and remove the line "SEG-Y Rev. 1 policy:".

SEG-Y Examiner

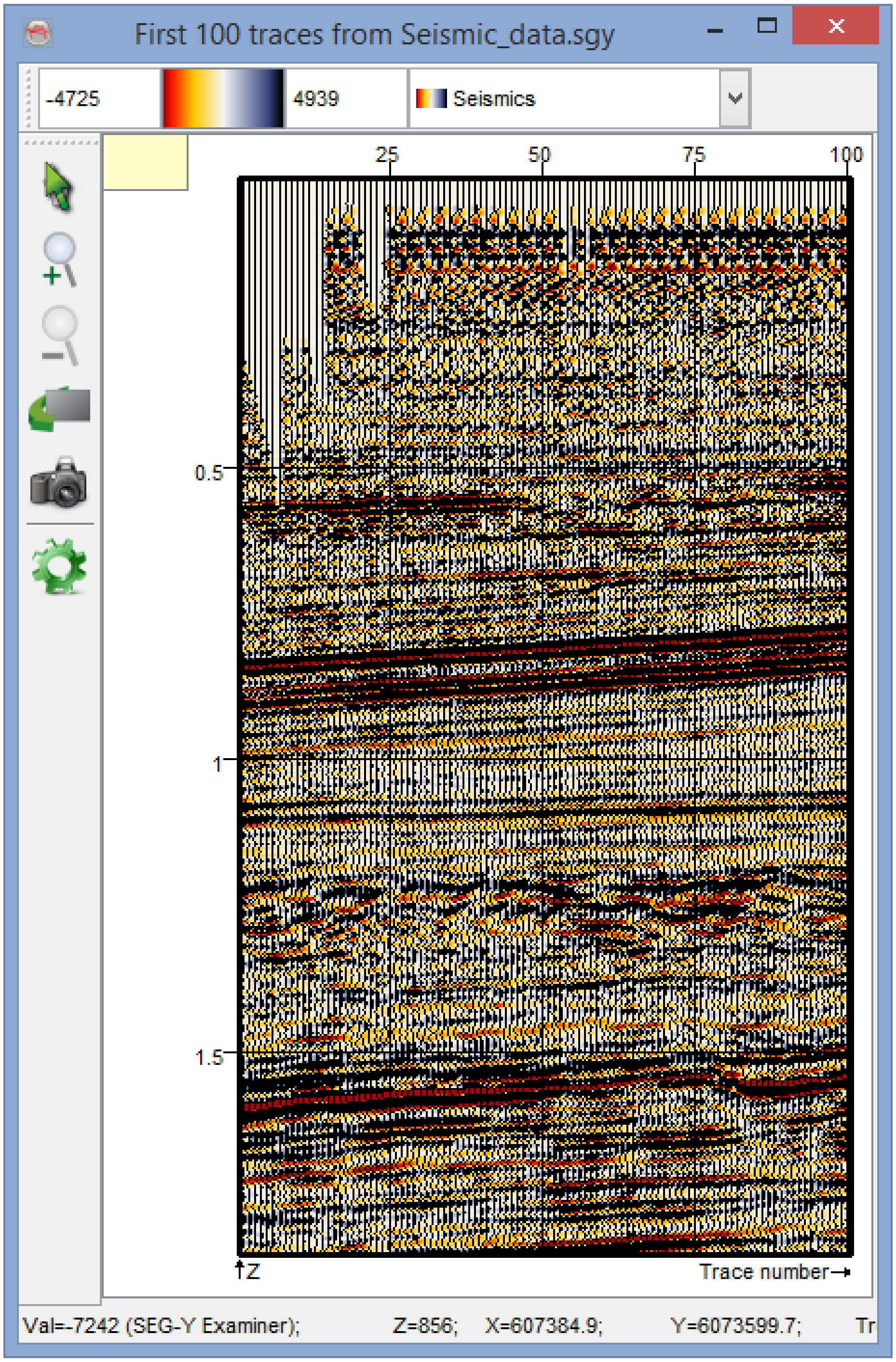
SEG-Y examiner window is used to examine the textual, line and first trace headers. It is a useful tool that serves two purposes:

1- Examination of the headers so that, if necessary, SEG-Y settings can be found and adjusted.



The plot between any individual trace header and the number of random traces examined is a very good tool to visualize its range and inspect for any discrepancy.

2- Viewing the first traces in a 2D viewer.



It is strongly recommended to always display the traces in the 2D viewer. This is done from the examine window by using the  icon . The examine window will show the entire textual header, the words in the line header that are different from zero, and the entire trace headers of the first traces. You can use the vertical scroll bar to browse through the file headers and locate the information to import.

Optionally, the save button  can be used to store the textual header.

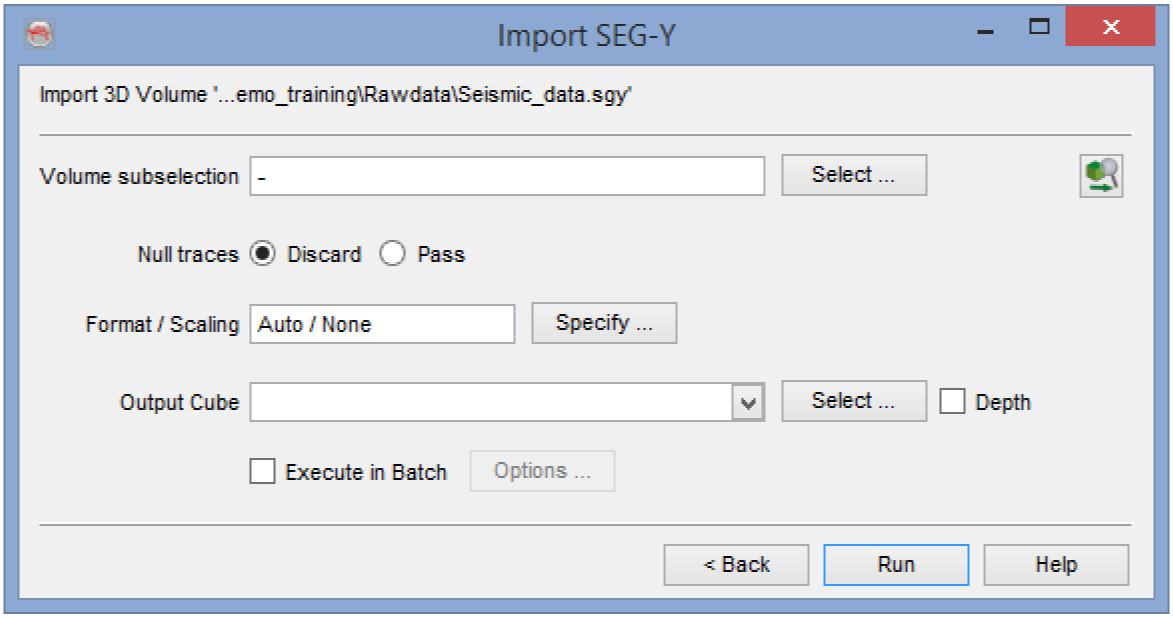
Use the lateral scroll bar at the bottom to locate the header words that change with the traces. They indicate traces, crosslines, offsets, coordinates etc. It is unlikely to see the inline number vary in the first traces. However,it is often at the bytes preceding the cross-line number. Increasing the number of traces to examine will help finding the inline number.

Please note that the examine window can stay open during import.

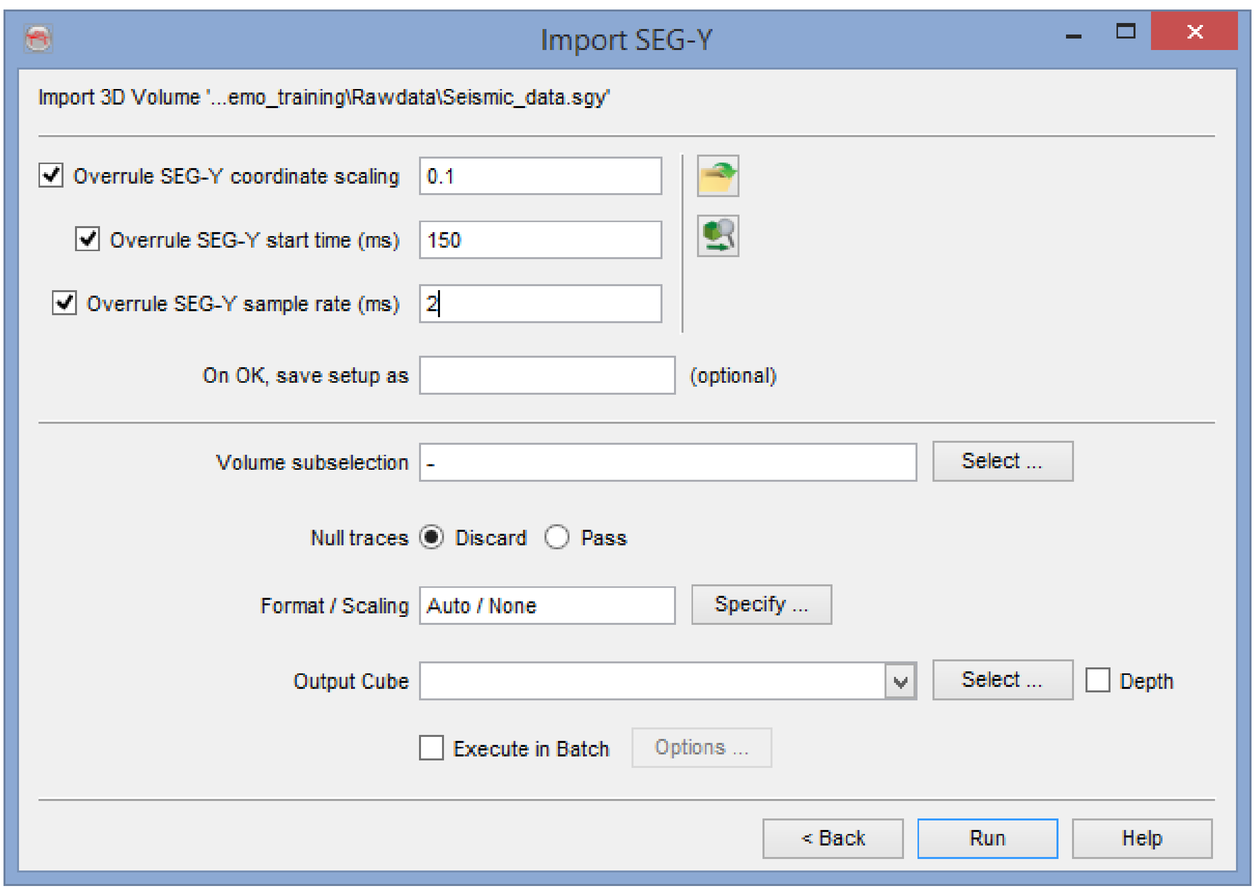
SEG-Y Import

The main import window will look different based on the type of SEG-Y file. At any moment a partial or full scan of the input(s) can be launched using the  icon on the right-hand side.

If the file is **Rev 1** standard then the import is almost complete: You must provide an output name and can optionally sub-select a range of the volume to be loaded and/or change the output format and/or re-scale the amplitudes. Pressing *Ok* will launch the import. Please note that **depth volumes** can be imported in time surveys and vice versa by using the depth/time toggle. They can be visualized using transformed scenes, providing that velocities are available.



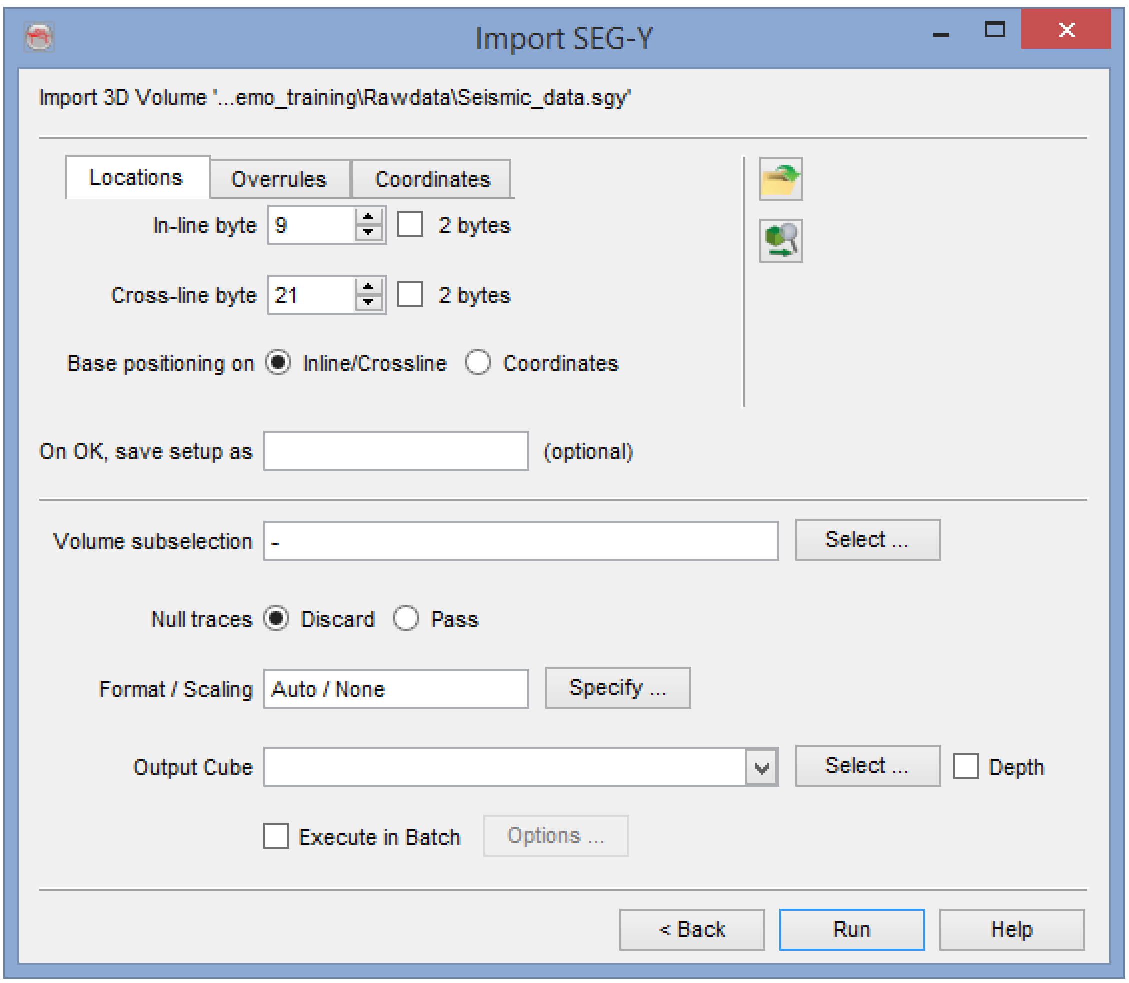
If the file is mostly Rev 1 but with **changed parameters** you will receive three additional fields that can overrule the values to be found in the headers:

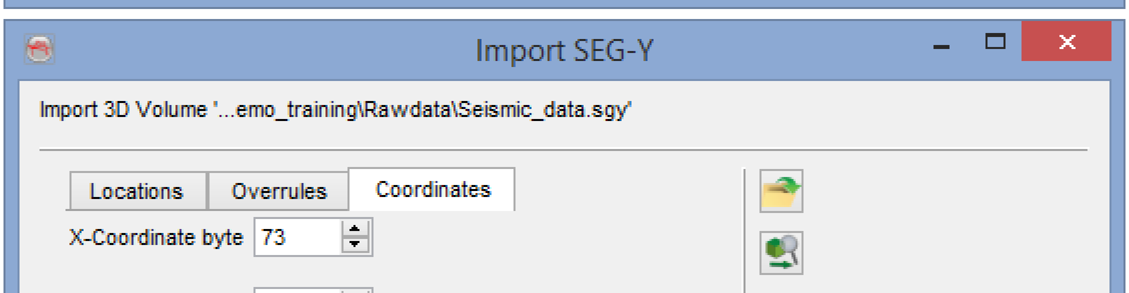


- SEG-Y coordinate scaling: Please provide the scaler that must be multiplied to the trace coordinates in order to get the loaded coordinates.
- Please provide the time assigned to the first sample of the traces, in milliseconds or meters. Negative times are allowed.
- Sample rate: Please provide the sampling rate of your data in milliseconds or meters/feet.

Those parameters constitute a SEG-Y setup that can be saved and retrieved using the yellow folder icon on the right. This setup will not only contain the parameters but also the path of the input files and settings of the preparation step. The setups are data dependent therefore they are stored in your survey.

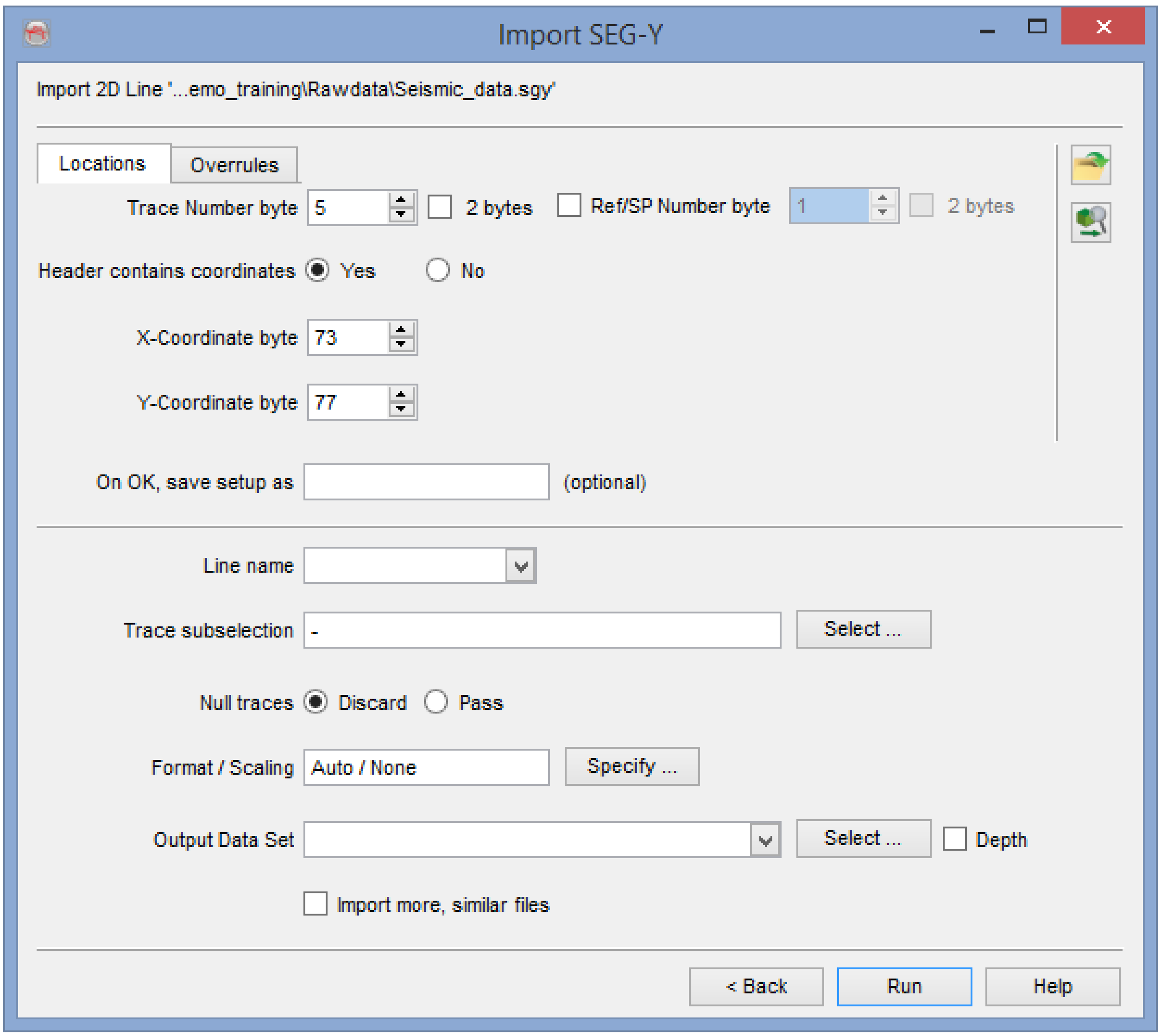
If the file is not Rev 1 you will get the overrule fields described above in a tab and two additional tabs to provide the byte locations of either the pair inline/crossline or the pair of X and Y coordinates. In the case of pre-stack data an additional tab will be present to provide the offsets/azimuth byte locations.





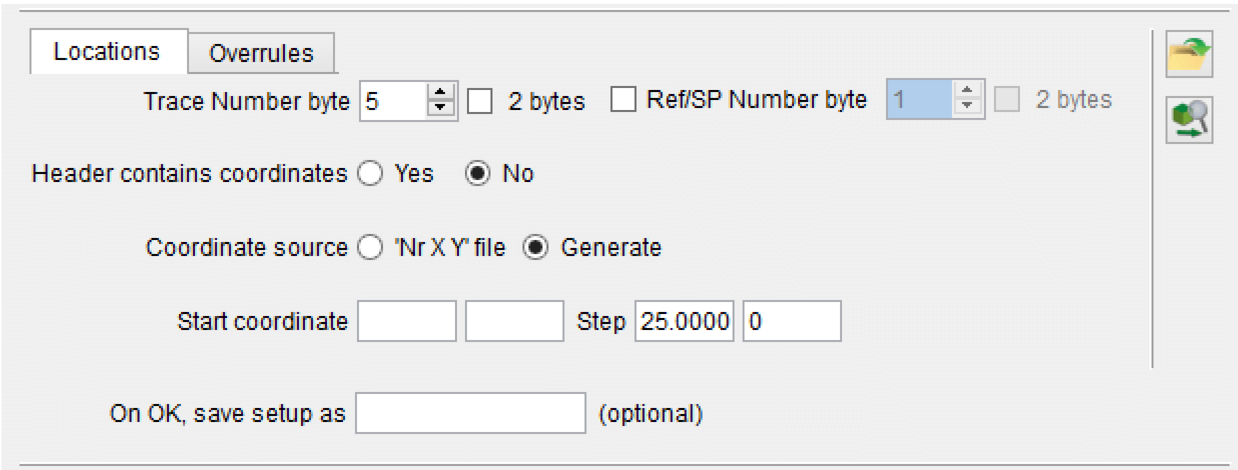
You need to look at the trace headers in the examine window and assign the right settings in this import window. Once again the entire import setup may be saved or retrieved. Once this is done you must provide an output name and can optionally subselect a range of the volume to be loaded and/or change the output format and/or rescale the amplitudes. Pressing *Ok* will launch the import.

The **import of 2D lines** is somewhat different: Inlines and Crosslines are replaced by trace numbers, that must be unique for each trace (therefore it can be the CDP but not the Shot Point).



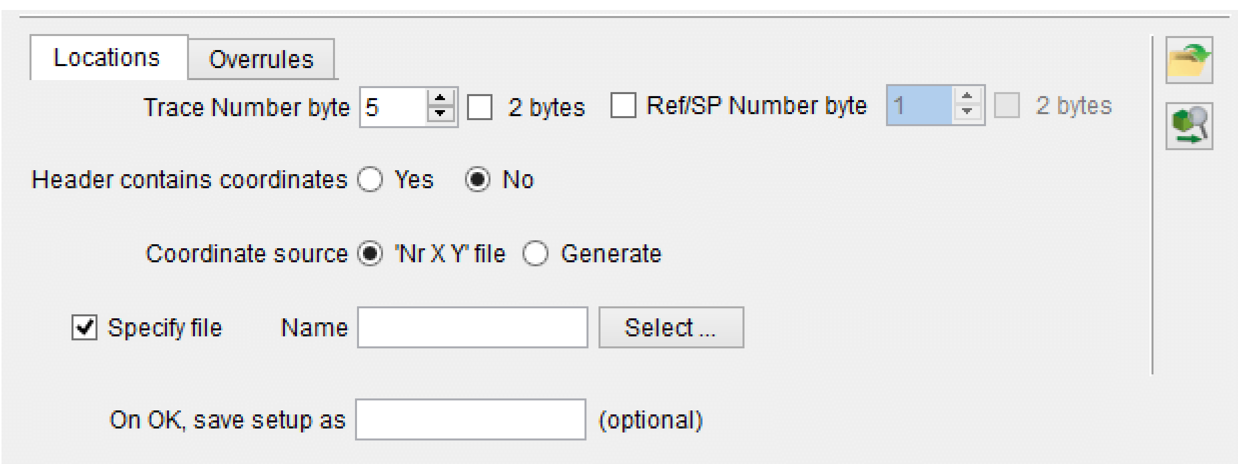
Coordinates can be imported from one auxiliary file or specified manually and generated during import coordinates if missing or wrong in the trace headers (see below on the right hand-side). This can be done by toggling off the X-coord byte field.

Generate XYs



The coordinates are generated for each trace position by providing the X and Y coordinates of the first trace, and a regular step in both directions. Units to be used are the same as specified in the survey definition.

Input Auxiliary file

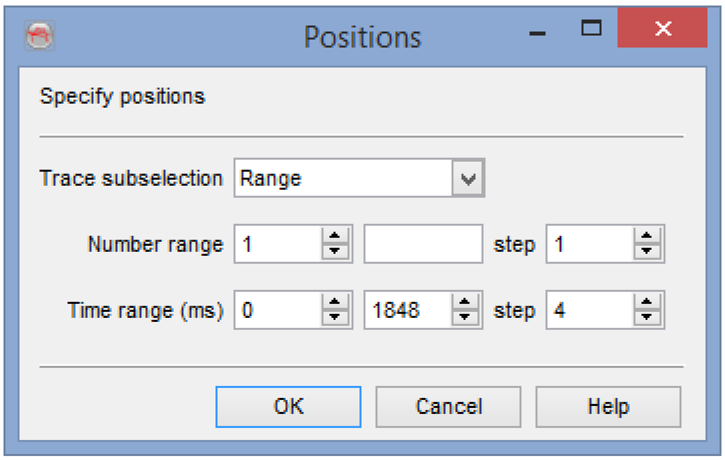


Optionally the coordinates can be specified using an auxiliary file. The format should be an input ascii file with one position per line in a fixed column format: File column should have the trace number, second column the X coordinate, third column the Y coordinate. Units to be used are the same as specified in the survey definition.

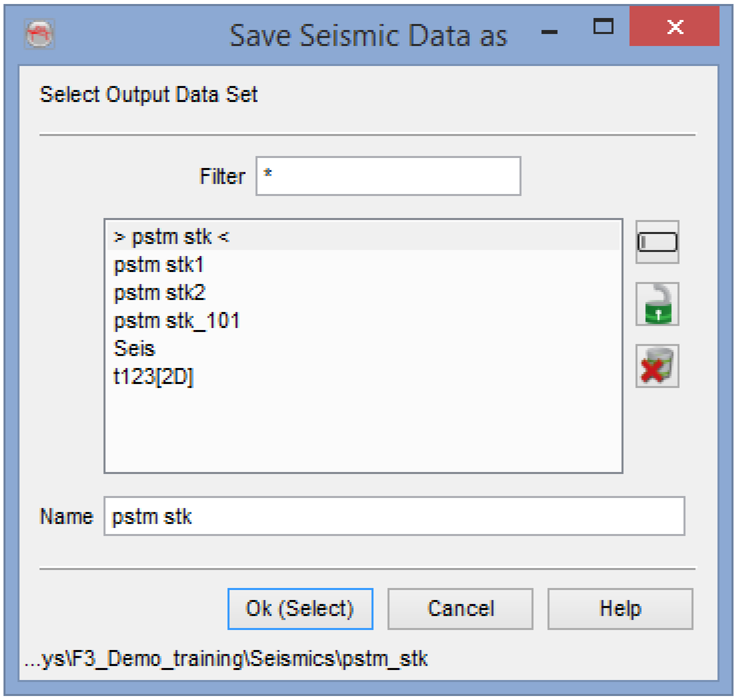
The **line name** is most often part of the input file name. It will be used only if a single line is loaded. Otherwise the line name is extracted as a part of the filename (see further below).

The **Output data set** name represents a 2D survey that comprises one or more lines. An OpendTect survey can have many 2D surveys (data sets), that are group of 2D lines that can be selected together for processing and interpretation.

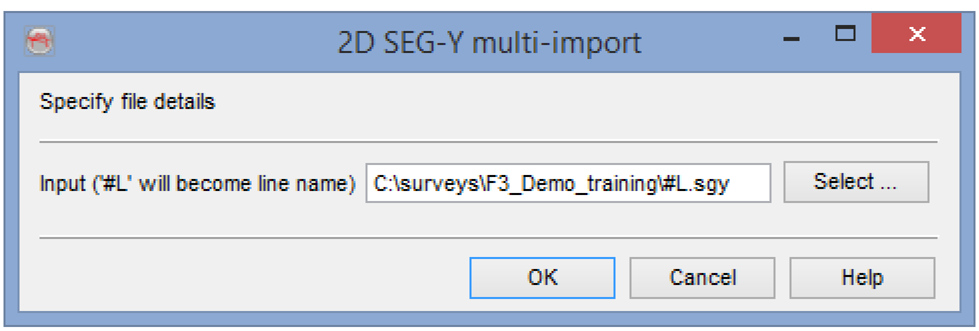
There is no format/scaling for 2D lines. The SEG-Y data format defines the OpendTect format. However, there is a trace sub-selection option to select either a trace "Range" or "All".



A **default attribute name** "seis" will be given to each line of the loaded data set. This can be changed by pressing "Select" and filling the empty "Attribute" field, like in the example below:



Multiple 2D lines loading must be enabled using the button "Import more, similar file" on the last line before pressing "OK". Any line can be used to go through the wizard, and the settings must be the same for all lines. If that is not the case then it is best to run the wizard several times per group of lines of similar SEG-Y settings.



This additional window is used to specify the generic line name out of the SEG-Y file-names. The line name must be replaced by "#L", while everything else (including the path and the extension) is shown as text, like in the above example:
$DATAPATH/Line_#L.sgy

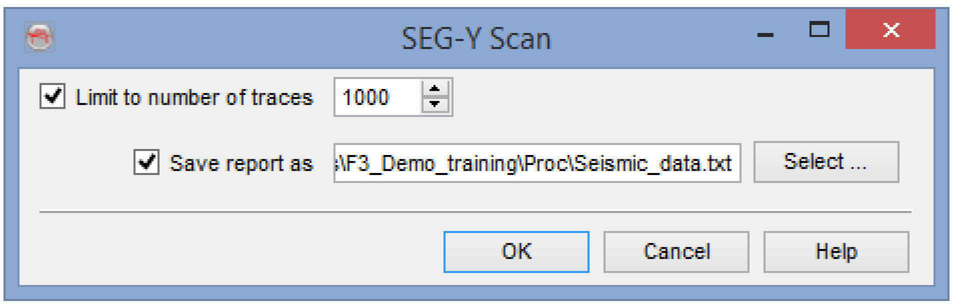
There will be one progress bar per input file during loading.

Tip: It is a good practice to display the 3D seismic data on a z-slice to check for any gaps in inline/cross-lines or null traces, after importing it.

SEG-Y Scan

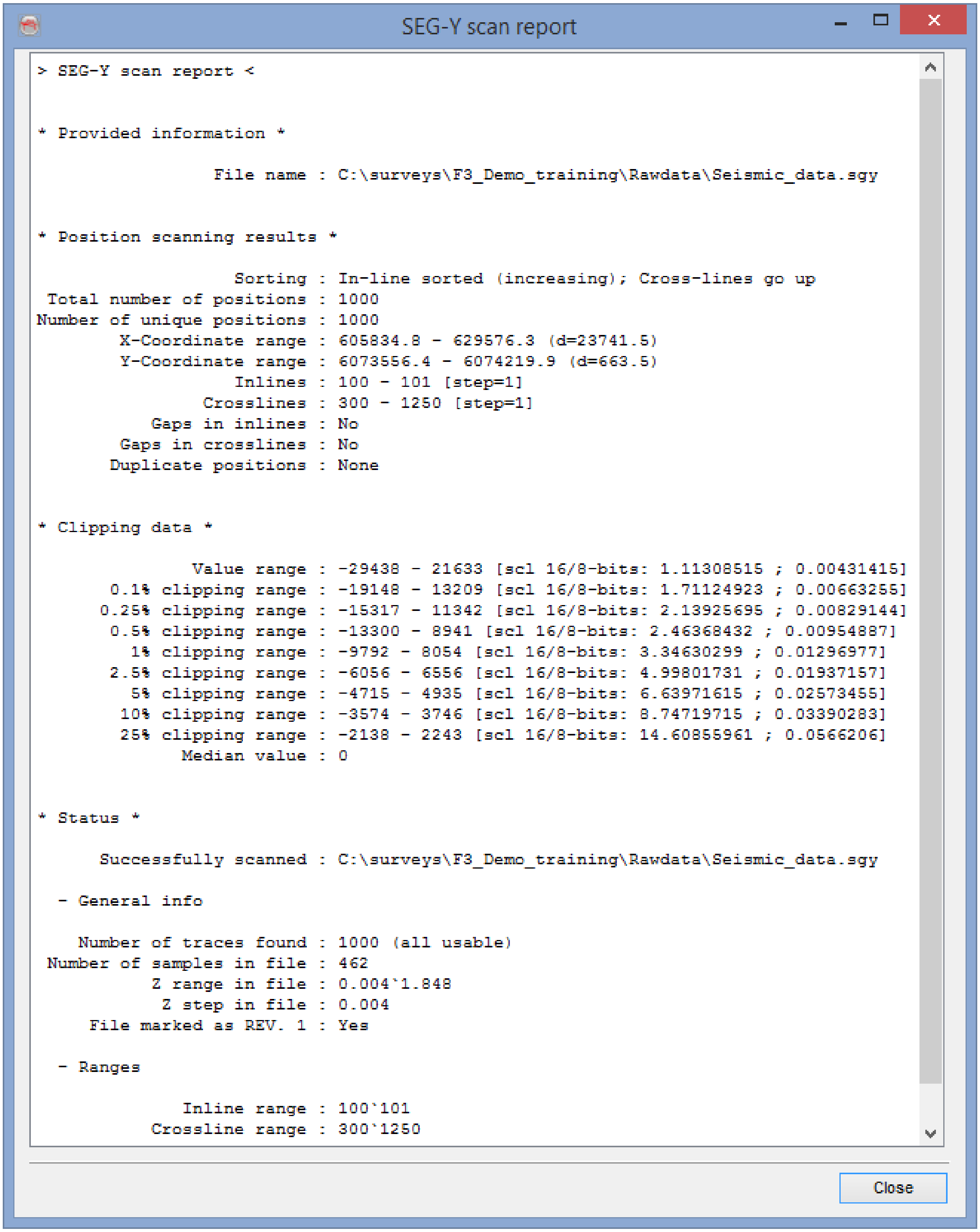
SEG-Y Scan is a useful tool to get an idea about the content of a SEG-Y file, and to check the loading settings. It is best performed on a limited number of traces (default is 100) when checking the *loading parameters*, and on the entire file when extracting geometry and ranges.

At any moment a partial or full can be launched using the  icon on the right-hand side in the *Import SEG-Y* window.



SEG-Y scanning is used to derive the survey geometry for a SEG-Y file. A successful scan will result in a successful loading, if the numerical values returned are in line with the correct parameters (inline range, coordinate scaling ...).

Here is an example of output of a SEG-Y scan report:


SEG-Y scan report

```
> SEG-Y scan report <


* Provided information *

                 File name : C:\surveys\F3_Demo_training\Rawdata\Seismic_data.sgy


* Position scanning results *

                   Sorting : In-line sorted (increasing); Cross-lines go up
 Total number of positions : 1000
Number of unique positions : 1000
        X-Coordinate range : 605834.8 - 629576.3 (d=23741.5)
        Y-Coordinate range : 6073556.4 - 6074219.9 (d=663.5)
                   Inlines : 100 - 101 [step=1]
                Crosslines : 300 - 1250 [step=1]
           Gaps in inlines : No
        Gaps in crosslines : No
       Duplicate positions : None


* Clipping data *

               Value range : -29438 - 21633 [scl 16/8-bits: 1.11308515 ; 0.00431415]
       0.1% clipping range : -19148 - 13209 [scl 16/8-bits: 1.71124923 ; 0.00663255]
      0.25% clipping range : -15317 - 11342 [scl 16/8-bits: 2.13925695 ; 0.00829144]
       0.5% clipping range : -13300 - 8941 [scl 16/8-bits: 2.46368432 ; 0.00954887]
         1% clipping range : -9792 - 8054 [scl 16/8-bits: 3.34630299 ; 0.01296977]
       2.5% clipping range : -6056 - 6556 [scl 16/8-bits: 4.99801731 ; 0.01937157]
         5% clipping range : -4715 - 4935 [scl 16/8-bits: 6.63971615 ; 0.02573455]
        10% clipping range : -3574 - 3746 [scl 16/8-bits: 8.74719715 ; 0.03390283]
        25% clipping range : -2138 - 2243 [scl 16/8-bits: 14.60855961 ; 0.0566206]
              Median value : 0


* Status *

      Successfully scanned : C:\surveys\F3_Demo_training\Rawdata\Seismic_data.sgy

  - General info

    Number of traces found : 1000 (all usable)
 Number of samples in file : 462
           Z range in file : 0.004`1.848
            Z step in file : 0.004
     File marked as REV. 1 : Yes

  - Ranges

              Inline range : 100`101
           Crossline range : 300`1250
```

Close

SEG-Y Scanned

Data duplication is a large problem when working with large datasets. All other import tool generate new OpendTect files from ascii or binary files. This new type of *SEGY import* works differently since it will not create any file but will link an existing SEG-Y file to an OpendTect entry, selectable as any other OpendTect data.

This special *import tool* is only available for both stacked and pre-stack data. However, it should be noted that for pre-stack data the performances for reading and processing may be lower than with OpendTect pre-stack datastores that are optimized by importing the usual way. The importation itself is 100% similar to the normal SEG-Y import. The only difference is that there will not be any loading after completing the wizard.

> **Note:** Please note that since this tool links to an existing file, moving or renaming the file outside OpendTect will break the link and make the dataset unavailable.

Simple File

The user can import simple ASCII or Binary file by using plain file Seismic I/O Plugin. This can be reached via *Survey > Import > Seismic > Simple File > 3D or 2D (Pre/Post-stack) etc*.

The input file must first be selected and its data format type specified, between ascii and binary (4-bytes floats). All data must be in the 'local' format, because a blunt binary read/write is performed.

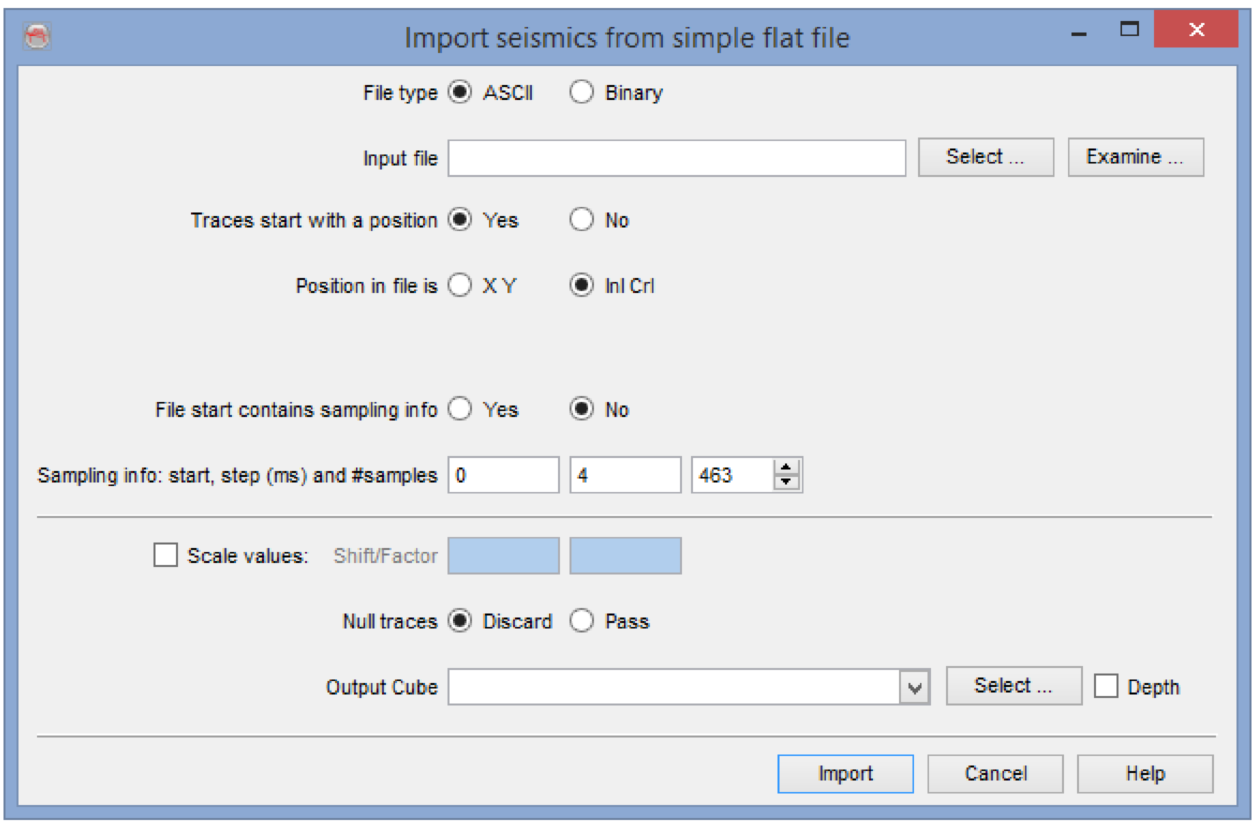
(Part of) the input file can be visualized by pressing the examine button. The data must consists of **one trace per record** (line). The samples are thus in columns, from shallowest to deepest, with a regular step. The trace position and time/depth index can be read from the input file, left of the trace, or can be provided. If provided, start, step and number of samples are requested in the corresponding directions, assuming the input file if regular and does not contain holes. Post-stack volume must be sorted by inlines, crossline, (offset), Z (time or depth).

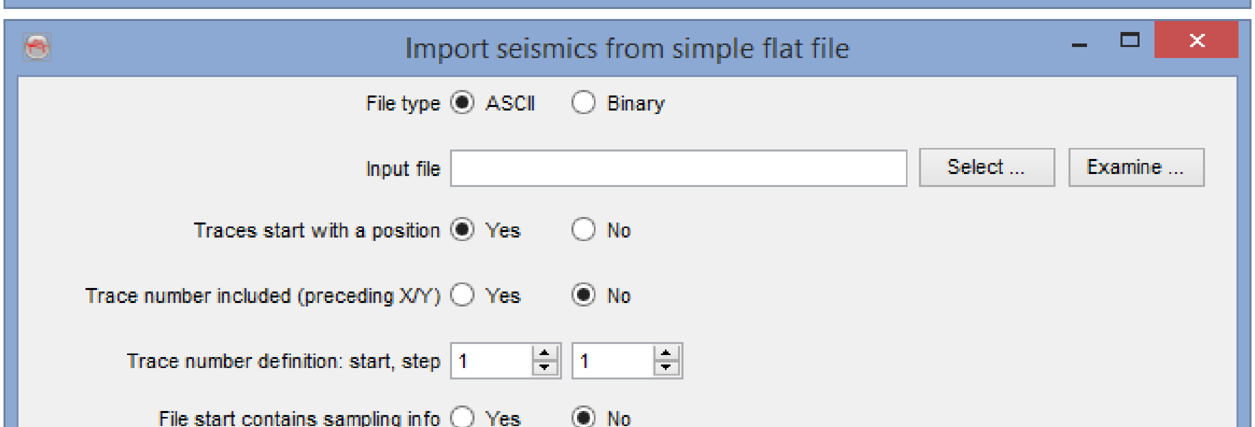
Optionally, the user can scale the cube before loading as well by mentioning the amount of shift and the corresponding factor. Either *pass* or *discard* the null traces before loading.

The easiest way to see what the format looks like is by producing a little export file from a bit of seismics. In the example below we exported inlines 500-501 and crosslines 600-603, Z range 1000-1020 step 4 (which is 6 samples):

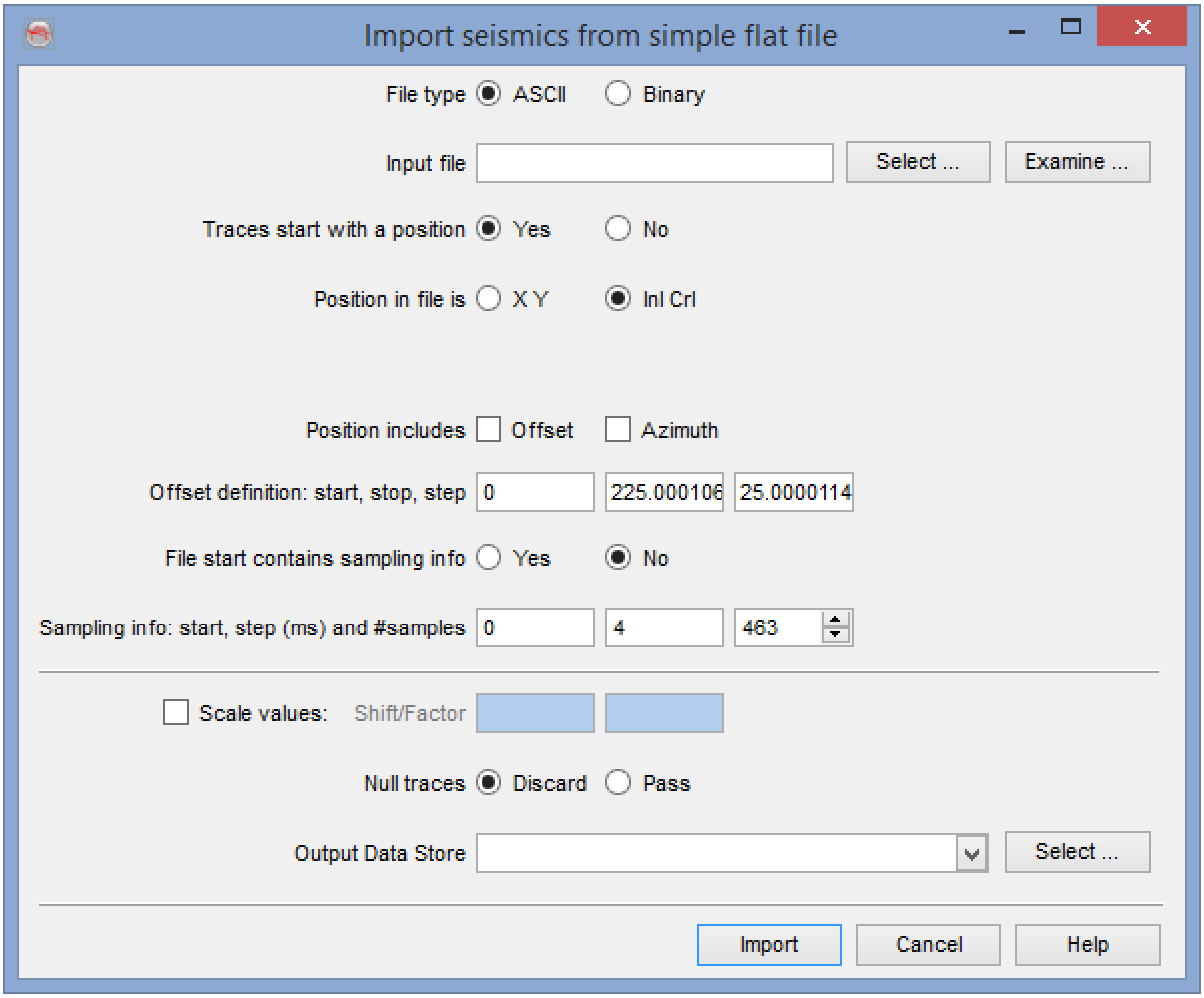
```
1000 4 6
500 600 1456 -688 -1502 4955 8935 1209
500 601 1429 -640 -967 5248 8362 527
500 602 1353 -424 -1040 5071 8059 -64
500 603 1428 -587 -1244 5139 8447 13
501 600 1450 -411 -1414 4792 8449 1117
501 601 1619 -456 -1243 4695 8271 702
501 602 1617 -213 -1272 4675 7903 393
501 603 1552 -248 -1088 4875 8004 204
```

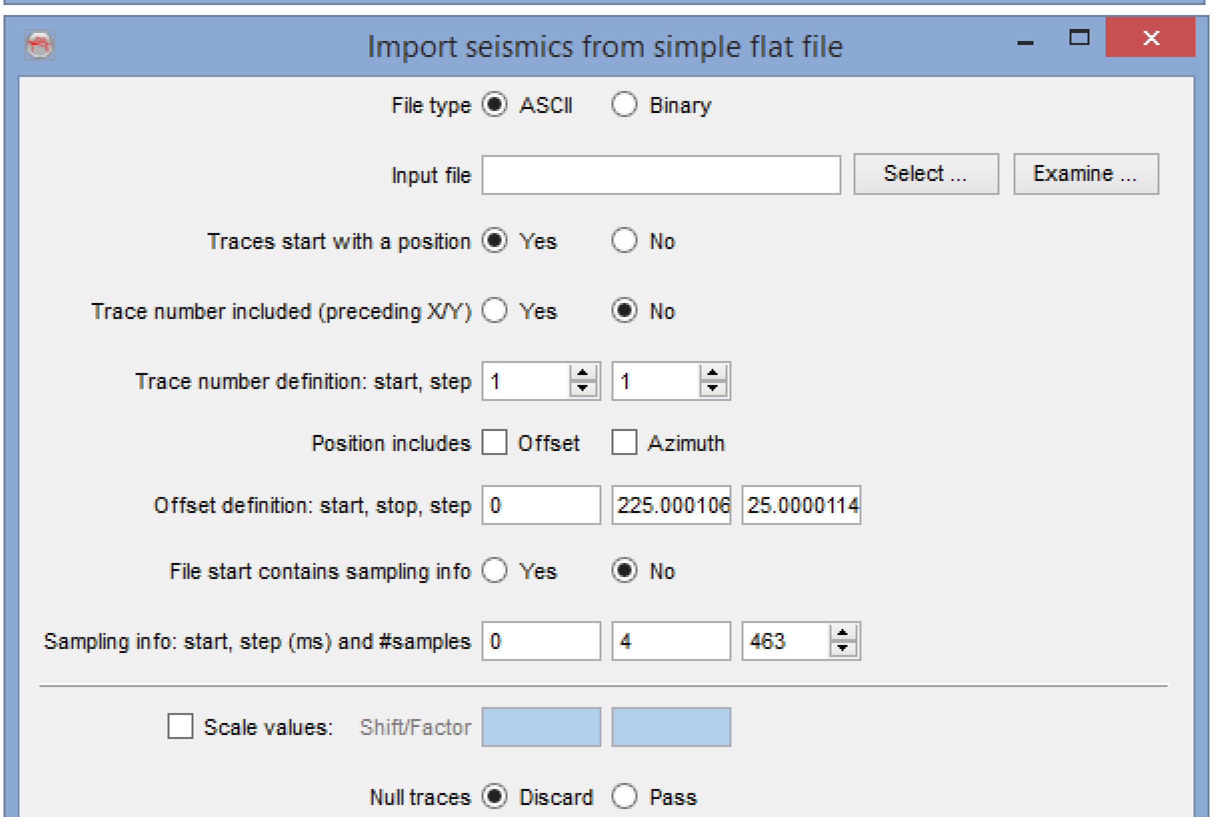
Below are respective examples of the import for 3D post-stack, 2D post-stack, 3D pre-stack, 2D pre-stack:





Simple post-stack 3D and 2D Seismic File Import Window

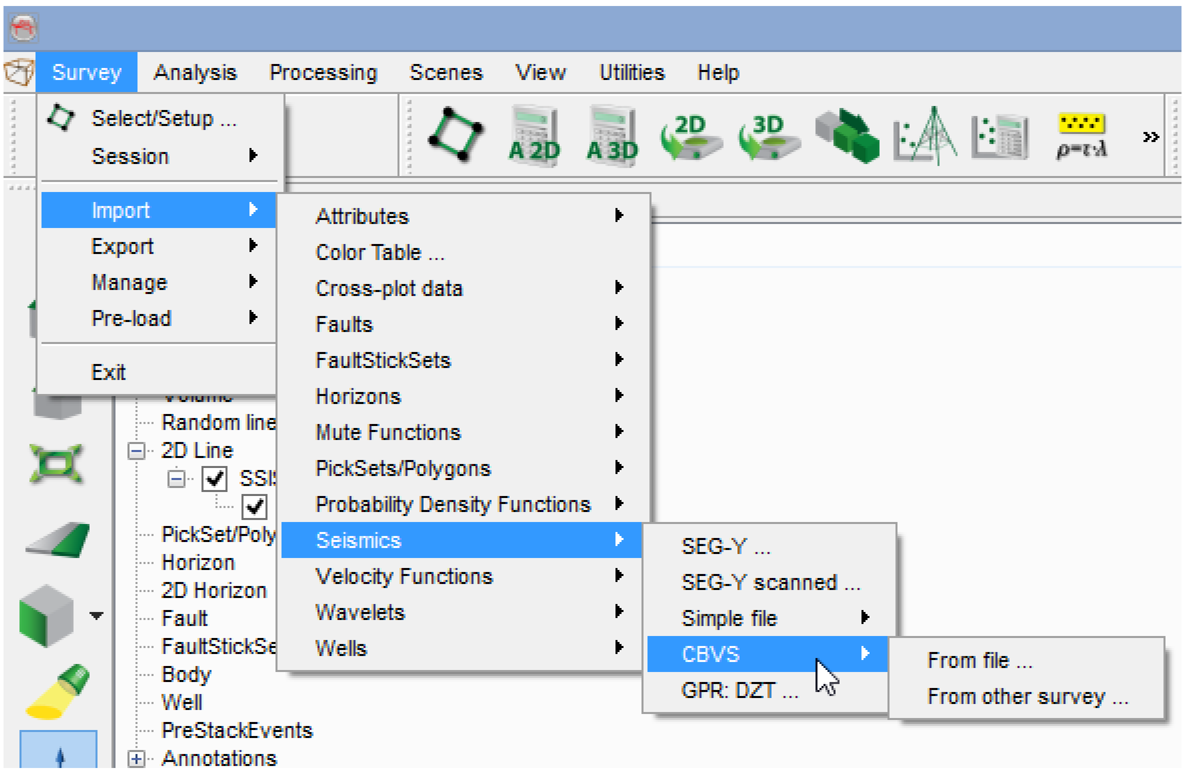




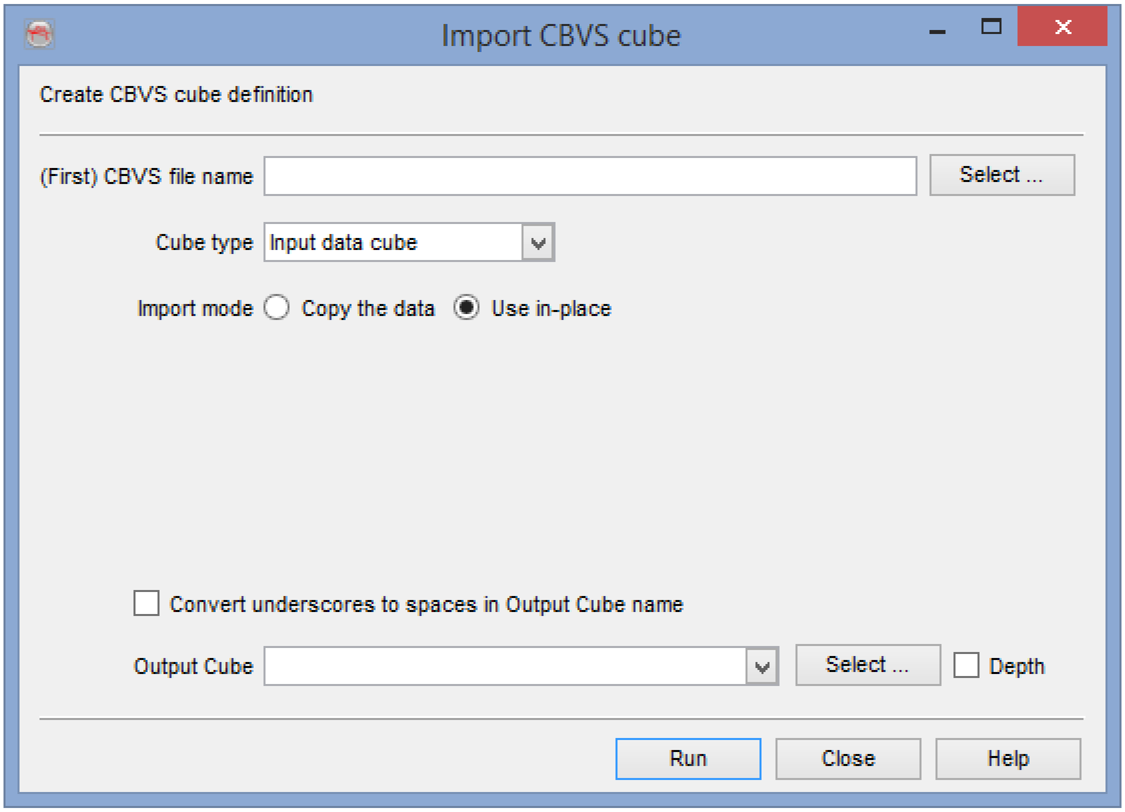
Simple Pre-stack 3D and 2D Seismic File Import Window

Import CBVS Cube

Seismic in CBVS format can be import either directly from cbvs file or from other survey (s) as shown below



From file:

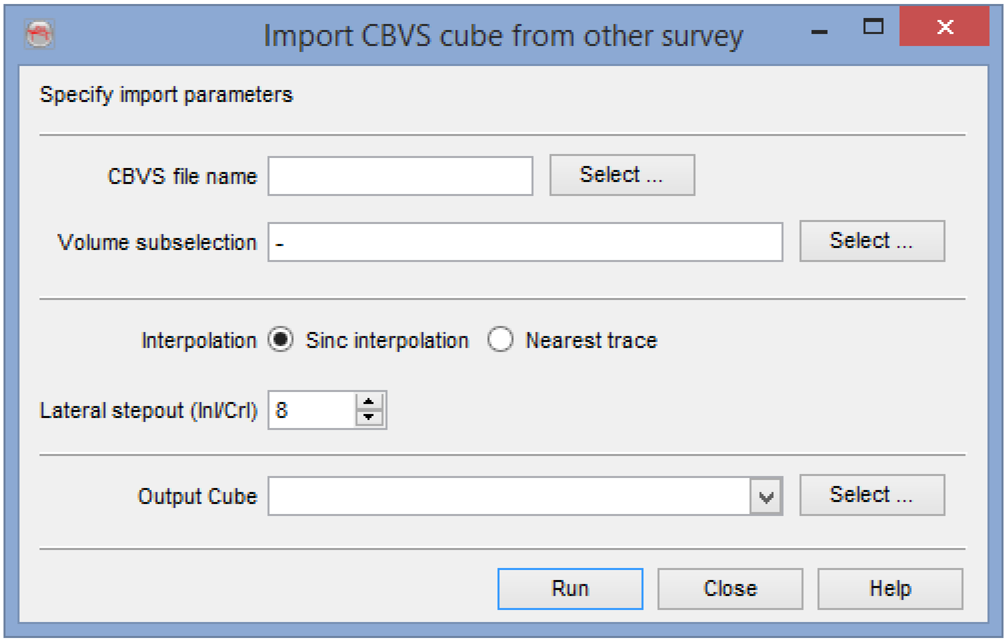


This module enables exchange of data between the OpendTect projects. The original CBVS (Common Binary Volume Storage) file can be located with a standard file browser. Some CBVS volumes are stored in several sub files. These can be recognized by the ^01 or ^02 (etc.) in the filename. To import the complete volume, select the base file without any ^xx marks.

The *Cube type* needs to be specified in order to give it the correct label for the software.

The *Import mode* indicates if the file should only be left at its original place and just be linked to the current survey (*Use in-place*), or if the volume should be copied entirely into the current survey directory (Copy the data). Moreover, while importing, the volume can be sub-selected (selected inlines/crosslines/time ranges) by pressing *Select* button in front of the *Volume subselection* field. If the data contains the Null traces, either discard or pass the traces by selecting the respective radio button. Before, importing the CBVS volume, the scaling (16bit, 32 bit etc) can be applied to the volume. The *Output Cube* field corresponds to the output volume name (that will be available in Manage Seismic window) for the input file.

From other survey(s):
Seismic cubes in cbvs format can also be imported from other survey(s)

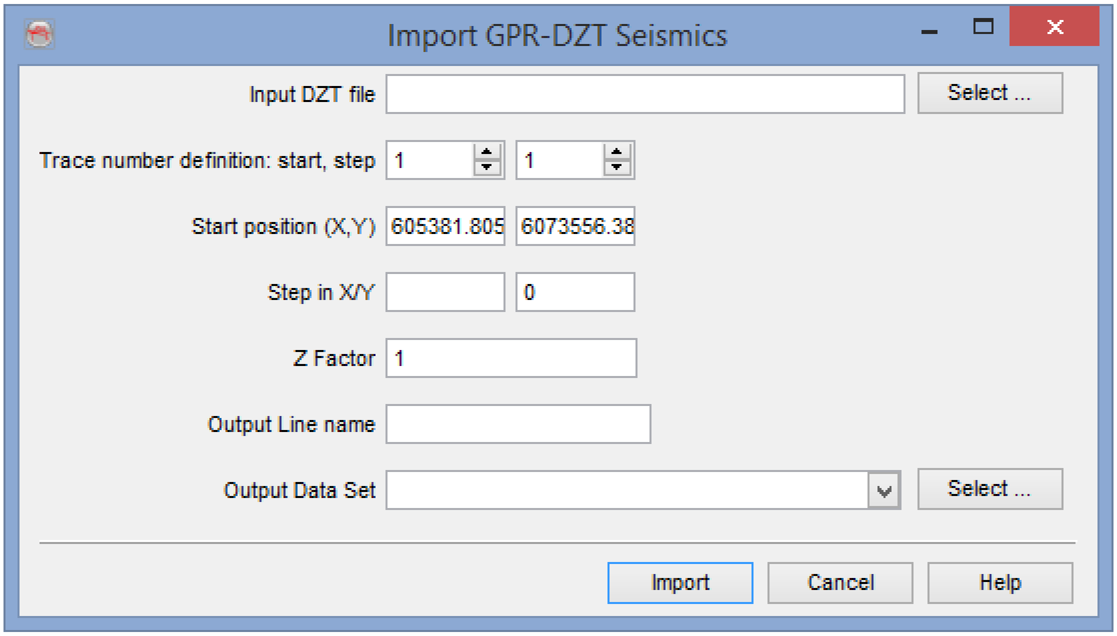


Few changes can be made prior to import the file: The volume subselection, the type of interpolation (sinc interpolation or nearest trace) and the stepout in inline and crossline.

GPR - DTZ

Ground penetrating radar offers an accurate solution to mapping the subsurface of the earth. It locates features of interest and subsurface layers in real time. The GPR data visualization and interpretation can be made in OpendTect, which enables the user to import the files made by GSSI Ground Penetrating Radar (GPR) systems in the 'DZT' format. The result is a 2D line in OpendTect.

Prior to loading a GPR data, the 2D survey should be setup according to the GPR acquisition setup. The data files are then imported as 2D geometries. The following *Import GPR Seismics* window allows the user to select one line and import the line according to the given setup. The time stamps or sampling rate in OpendTect is defined in milli-seconds. However, the DZT files are often sampled with nanoseconds sampling rate. To adjusted this, there is an input field available i.e. 'Z factor' that allows re-scaling of Z-axis or time. In order to visualize the data in OpendTect, this factor should be large enough. The remaining parameters i.e. Start X,Y position or X/Y steps could be filled according to the profile location.



Tagged Seismic Data

The imported volumes may contain any data. However several types can be specified during import and/or after:

- Depth poststack volumes/lines loaded in time surveys. A check box must be toggled on during SEG-Y or simple file import.

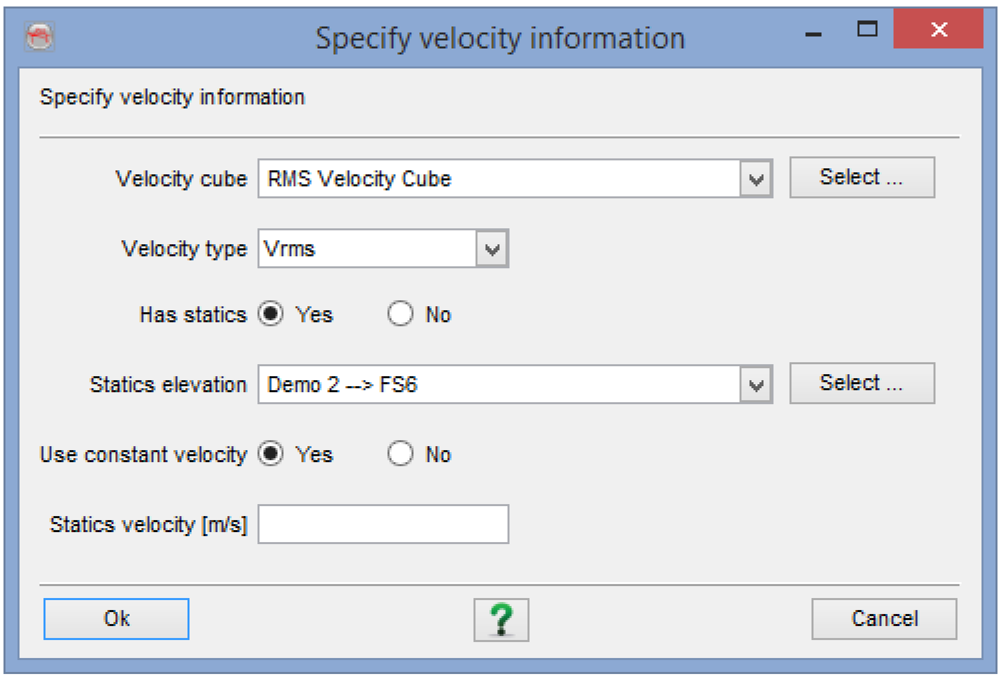
- Time poststack volumes/lines loaded in depth surveys. A check box must be toggled on during SEG-Y or simple file import.

- Velocity/anisotropy volumes must be tagged with respect to their type, when imported from an external file. The available types are:

  - Vint: Interval velocity
  - Vrms: RMS velocities (time domain only). A provided surface may provide elevation statics in meters. For time surveys a statics velocity must be provided in m/s, either from a velocity grid or using a constant velocity.
  - Vavg: Defined as the ratio between the depth and the travel time: Vavg (TWT)=2*Z/TWT=Z/OWT.
  - Delta: Thomsen anisotropy parameter of the same name.
  - Epsilon: Thomsen anisotropy parameter of the same name.

- Eta: Effective anisotropy parameter, combine from delta and epsilon. This tag can also be used to grid another quantity (the software does not actually check that eta values are input).

The assignment of velocity types (and properties) to a volume is called velocity edition. This window can be opened in most windows wherever velocity volumes are used (an exception is the attribute set window):
- Volume gridding
- Time-to-depth scenes
- Time-to-depth conversions
- Velocity conversions
- Application of velocity corrections on pre-stack gathers

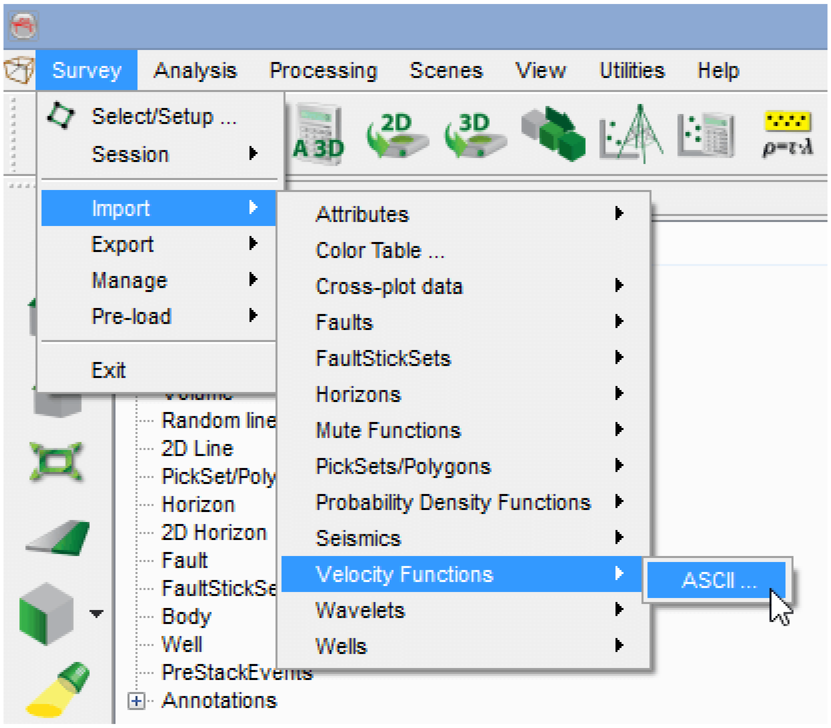


Note: that the velocity volumes might be scanned to get the Vavg range at their first and last sample. This allows the software to deduct and propose appropriate time/depth ranges during conversions (on-the-fly and batch).

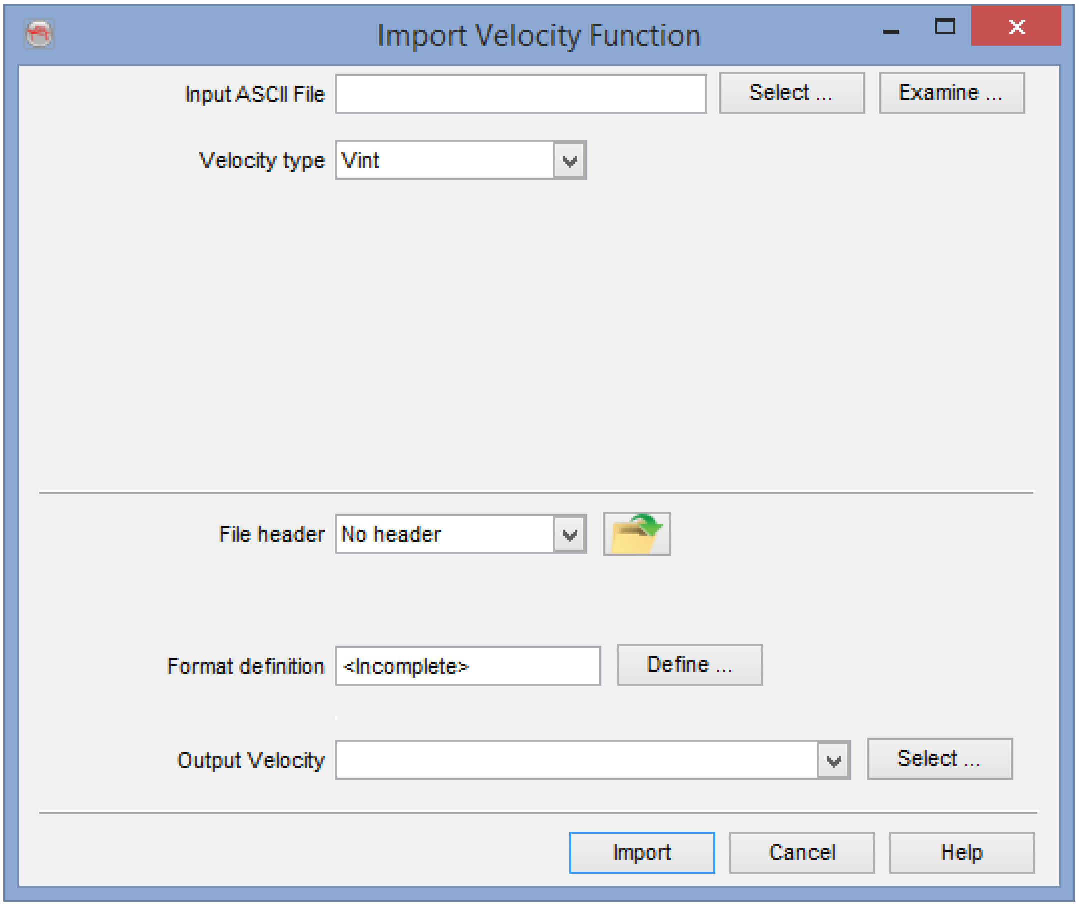
Import Velocity Functions

Velocity functions can be used to create volumes from irregularly sampled data, using the volume builder (that is the only way after import to display the data). Although designed to create velocity volume, it can be used for any other quantities that velocities:

All usual log types (density, sonic, impedance, ...), or quantities with a strong low frequency component. Velocity functions are not appropriate for regularization of band-passed data (seismic, relative impedance, ...).



Velocity functions can be imported in OpendTect using Ascii files. The import window is launched from the OpendTect main menu (*Survey > Import > Velocity functions > Ascii*). Select the input Ascii file. You may display the input file by pressing the *Examine* button. The input file should be column sorted with one point per record (line).



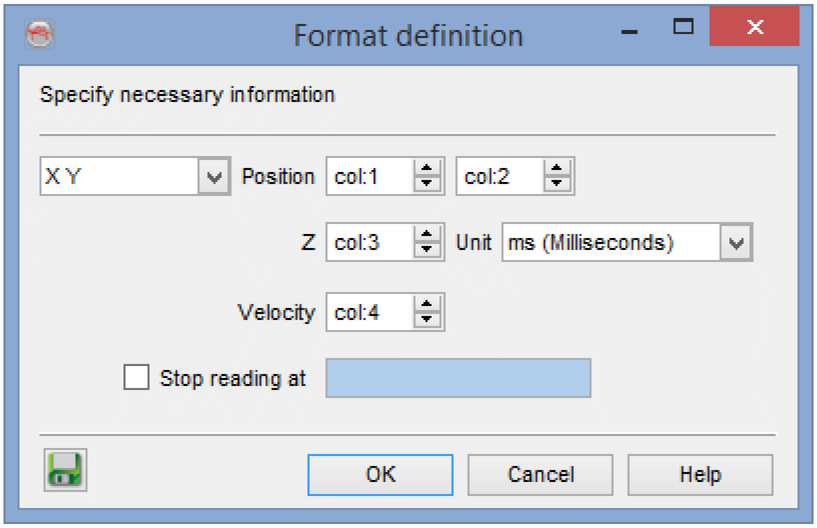
The amplitude type of the input data must be specified. This will be required for the interpolation of the values. Indeed the vertical interpolation will not be done directly on Vint and Vrms amplitudes, but on the corresponding time-depth function.

- For the Interval velocity quantity (Vint), the amplitude of a point accounts for the layer above it.
- Effective velocities (Vrms) can only be used in time surveys. They will be treated with a simple Dix for the extraction of the time-depth relation.If this type is selcted, you need to specify if it has statics or not.
- Vavg is the ratio at a given depth between depth and travel-time: Vavg = 2*Z/TWT = Z/OWT. As a result a Vavg quantity holds entirely the data from time-depth pairs.
- Delta, epsilon and eta functions will be vertically interpolated with linear interpolation.

Any other quantity type could be tagged as delta, epsilon or eta and gridded using the standard velocity gridding step of the volume builder.

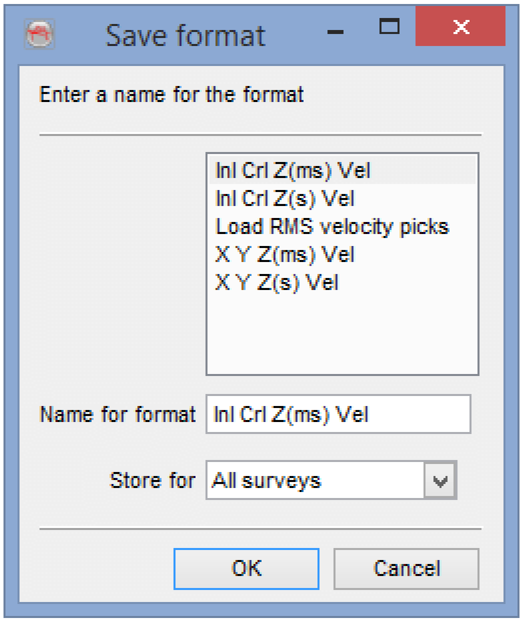
The main work is to specify the presence of a *file header* and the file *format definition*. The header, if present, can be of fixed length (number of lines), or delimited on its last line by a keyword.

Predefined and saved file formats are available by pressing the  icon. Otherwise the format must be manually specified. The Define button gives access to the format definition window.



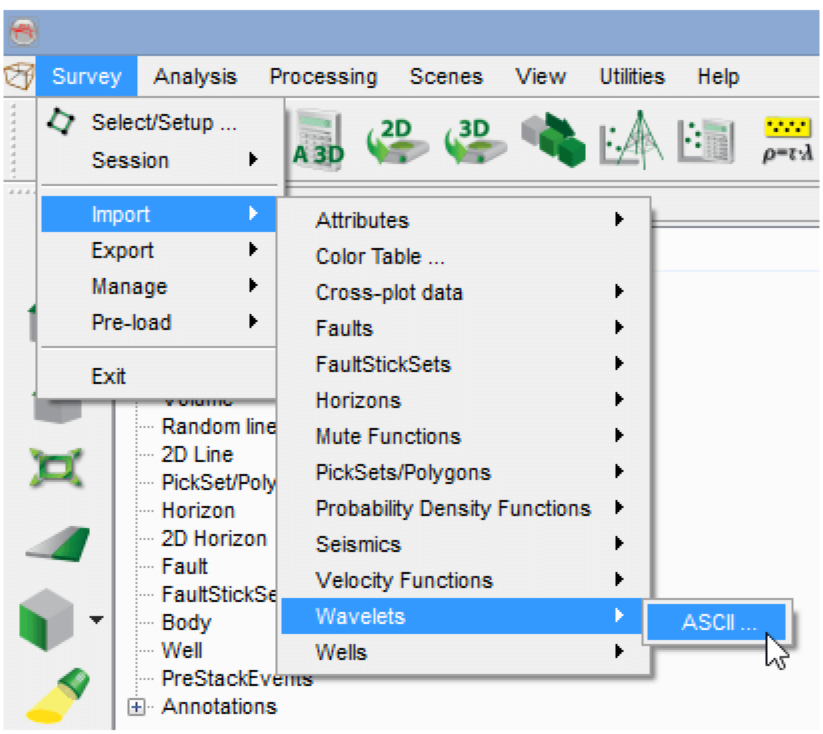
You must specify in the format definition window the column numbers for the position, in terms of an X/Y pair or an inline-crossline pair, the time/depth column, and the amplitude value. If velocities are read, linked anisotropies may also be imported in the same velocity functions object. This is used during velocity analysis only. Amplitudes that should not be read must all have the same numerical value, which is to be filled in as the "Undefined value". The Z units can be seconds, milliseconds or microseconds (meters of feet in depth surveys). Please note that all records must have an X-Y-Z position. Reading may be stopped at a specific line by providing the adequate keyword.

It is recommended to save the format definition for a later use and QC, by clicking on the  icon . In pop-up window, write the name of the format and store it. The format can be stored at different levels (All surveys, Current survey, Current OpendTect user level) depending on the usage. Press OK when done.

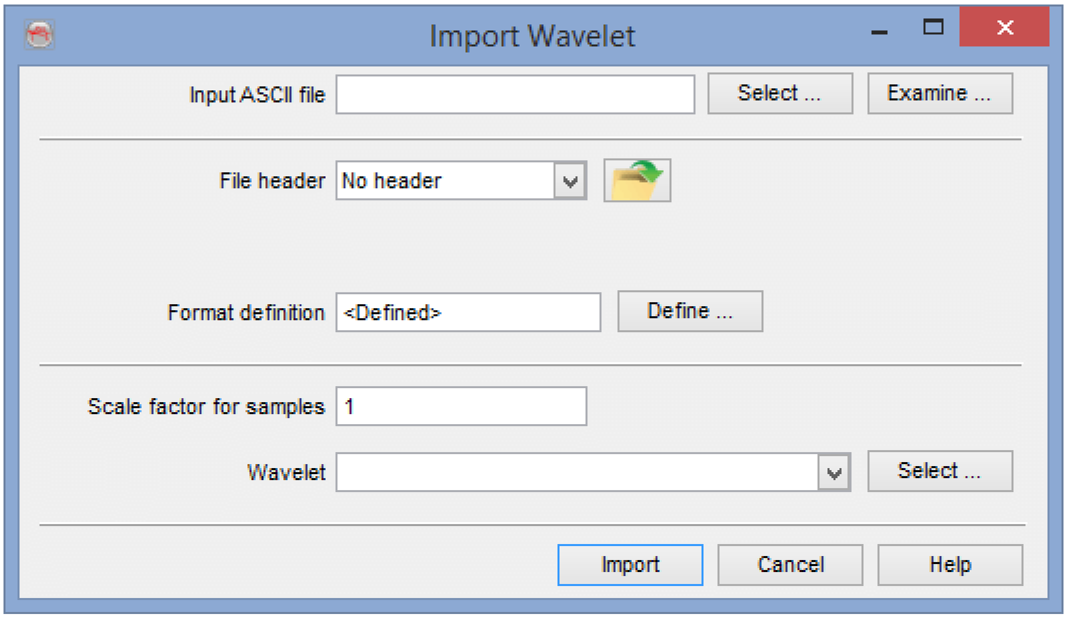


Import Wavelets

Wavelets can be used for synthetic-to-seismic tie, or convolution via the convolve attribute. Wavelets are also used to store all kinds of frequency-derived operators, like seismic spectral blueing and seismic coloured inversion operators.

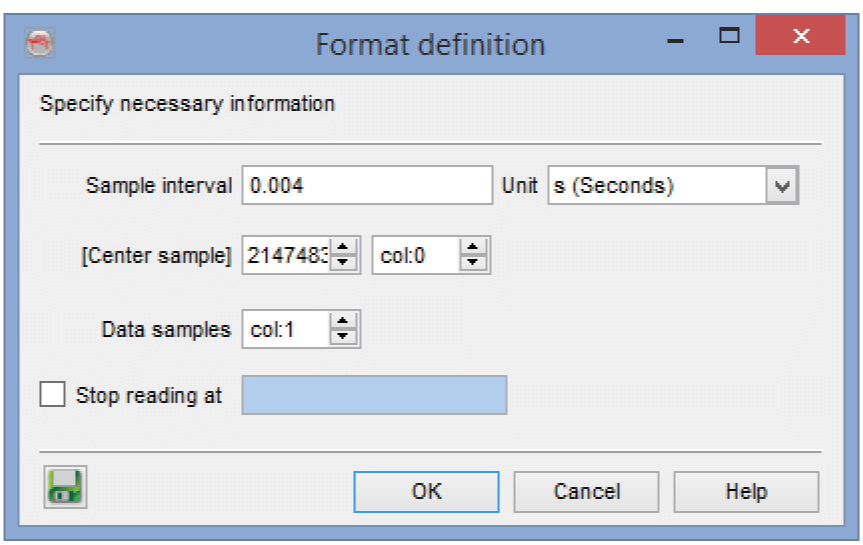


Wavelets can be imported into OpendTect as Ascii files. The import window is launched from the OpendTect main menu (*Survey* -> *Import* -> *Wavelets* -> *Ascii*). This will launch the *Wavelet Import* window. Select the input Ascii file. You may display the input file by pressing the *Examine* button. The input file should be column sorted with one time/depth sample position per record (line).



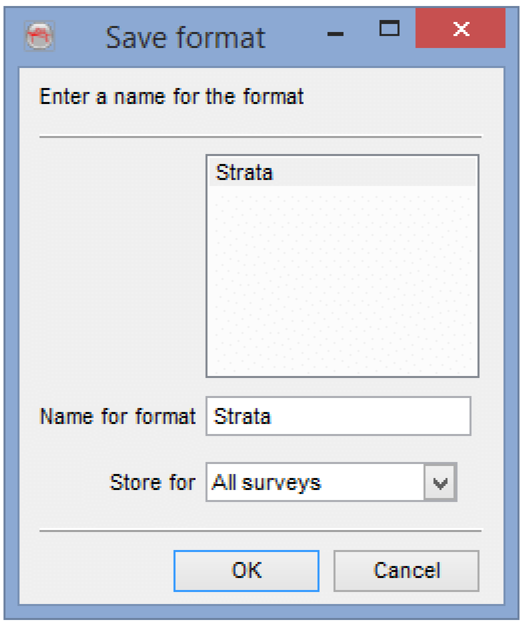
The main work is to specify the presence of a file header and the file format definition. The header, if present, can be of fixed length (number of lines), or delimited on its last line by a keyword.

Predefined and saved file formats are available by pressing the  icon. Otherwise the format must be manually specified. The Define button gives access to the format definition window.



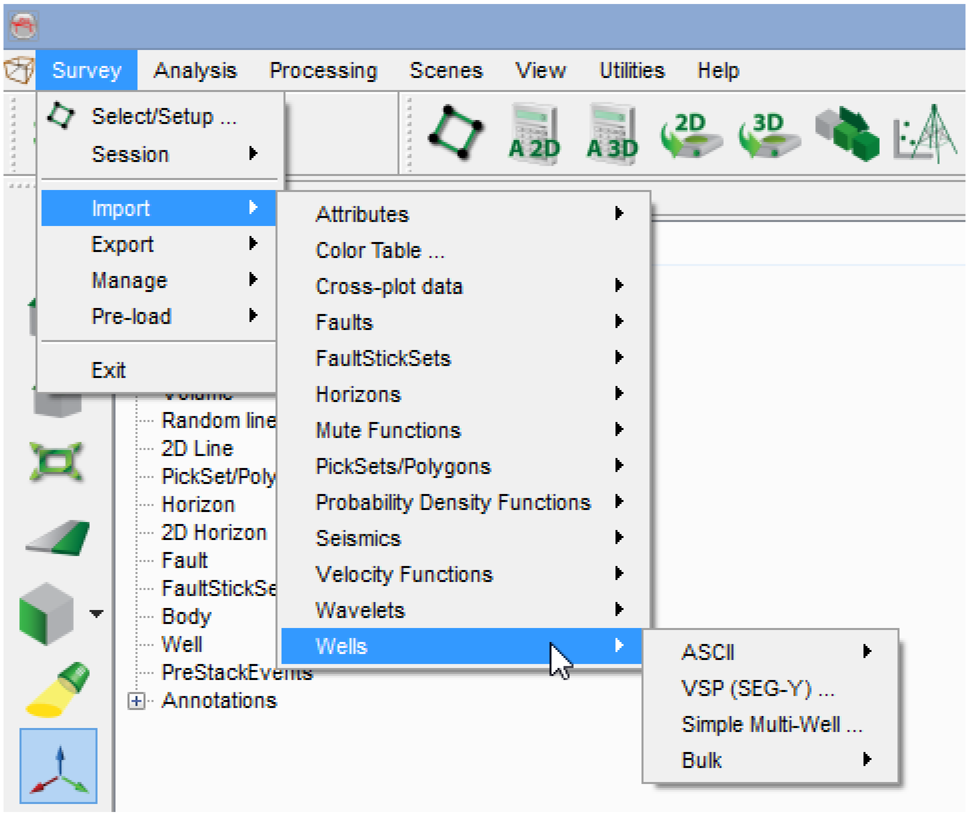
You must specify in the format definition window the column number where to read the amplitudes, the sample rate, and the location of the center sample. The sampling interval can be seconds, milliseconds or microseconds. The position of the center sample must be given as the number of lines between the first sample and the center sample. Both values can be dynamically extracted from the header by providing keywords for each, indexed using columns positions. Reading may be stopped at a specific line by providing the adequate keyword.

It is recommended to save the format definition for a later use and QC, by clicking on the  icon . In pop-up window, write the name of the format and store it. The format can be stored at different levels (All surveys, Current survey, Current OpendTect user level) depending on the usage. Press OK when done.

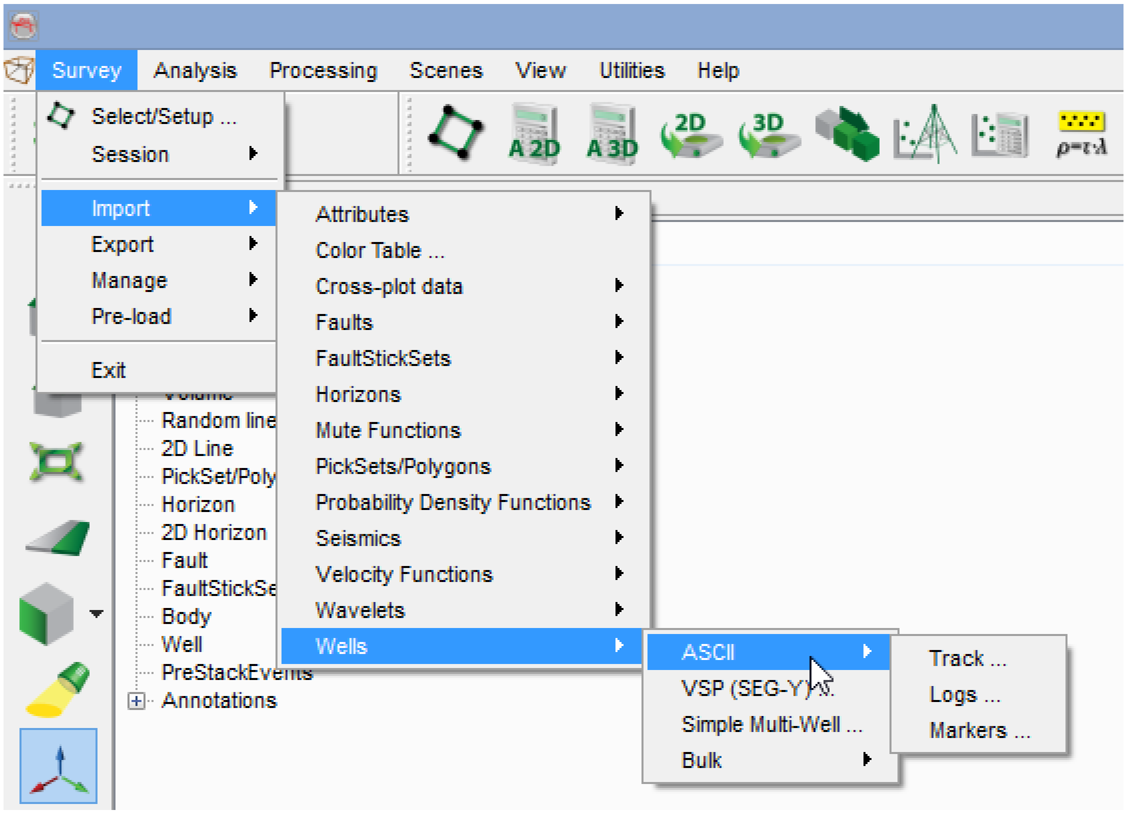


Import Wells

Wells can be imported in OpendTect via Survey > Import > Wells, from Ascii files. They can be used for visualization only, to be tied with the seismic data, for performing lateral correlation and/or train neural networks...

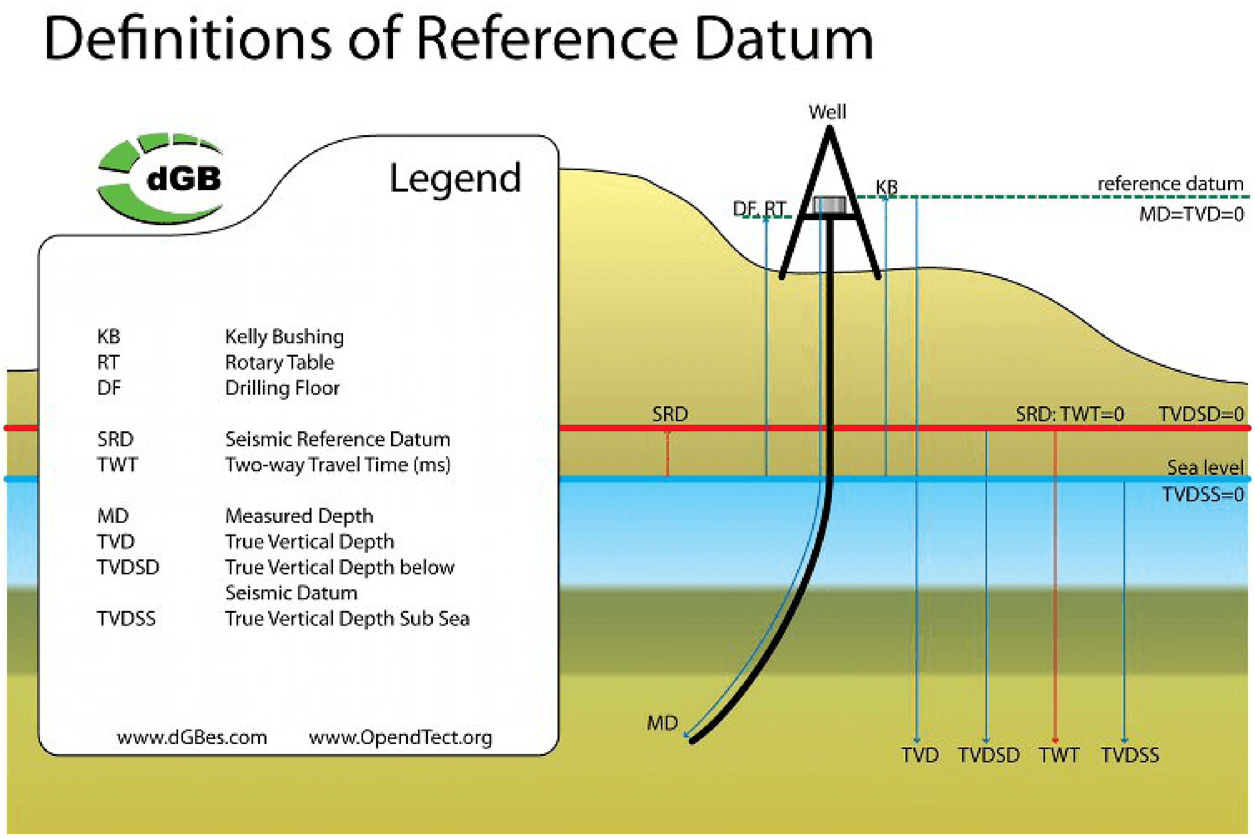

Survey
Analysis
Processing
Scenes
View
Utilities
Help
Select/Setup ...
Session
Import
Export
Manage
Pre-load
Exit
Random line
2D Line
PickSet/Poly
Horizon
2D Horizon
Fault
FaultStickSe
Body
Well
PreStackEvents
Annotations
Attributes
Color Table ...
Cross-plot data
Faults
FaultStickSets
Horizons
Mute Functions
PickSets/Polygons
Probability Density Functions
Seismics
Velocity Functions
Wavelets
Wells
ASCII
VSP (SEG-Y) ...
Simple Multi-Well ...
Bulk

From Ascii Files



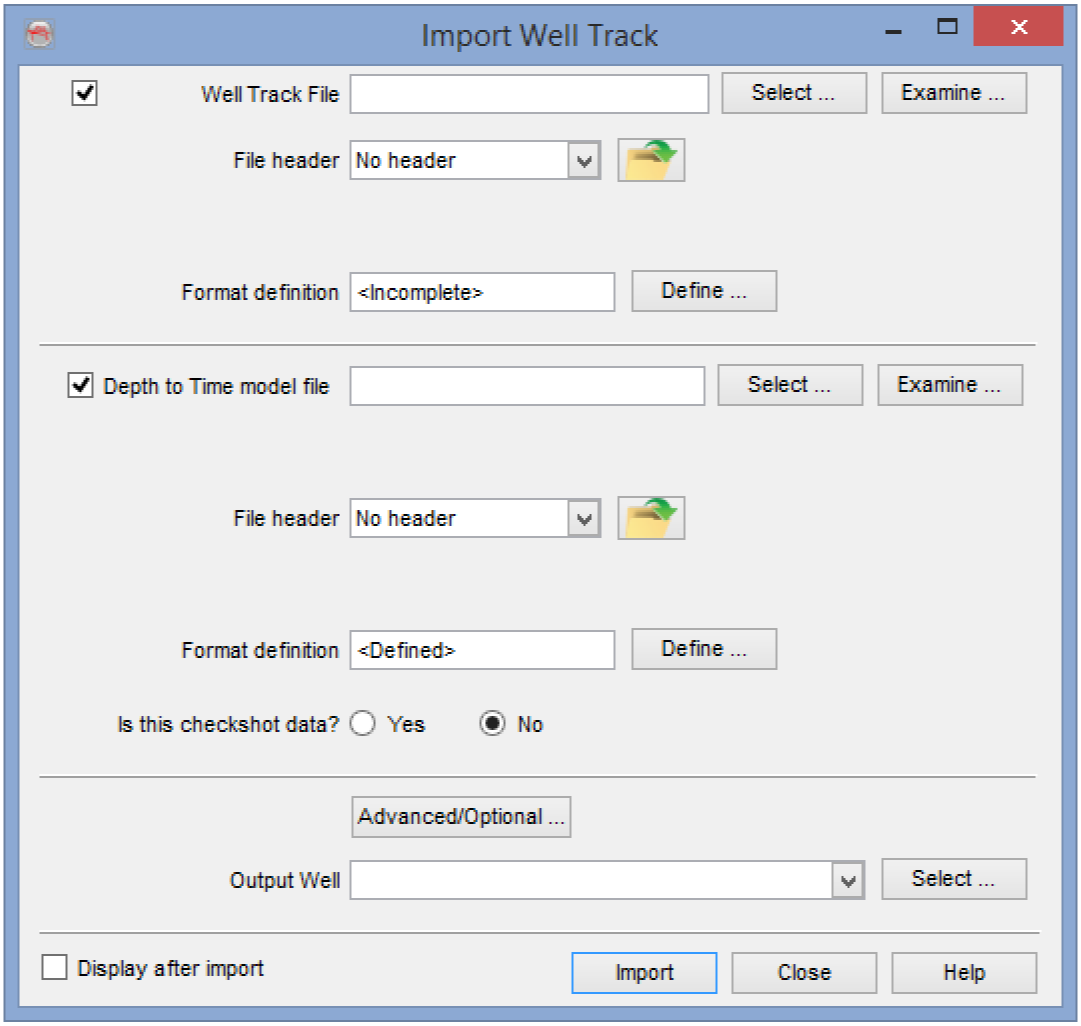
Well depths in OpendTect are always referenced using their Measured Depth. The alignment with the seismic data in done using the track (deviation survey) data and time-depth and/or checkshot data. The track data provides the relation between lateral coordinates, TVDSS depths and Measured Depths. The checkshot data and time-depth data provides the relation between Measured depths and travel times (TWT). Other formats can be used during import, but will be converted.

Please see the figure below for a list of datum definitions:



Track

The track import is loading the core part of the well, required for the visualization of the well track, and further loading of the markers and logs. It requires the loading of the deviation survey, and a depth-to-time model for seismic surveys in time. The import is done from the menu *Survey -> Import -> Wells -> Ascii -> Track*. The input ASCII file(s) must be column sorted with one point per record (line).



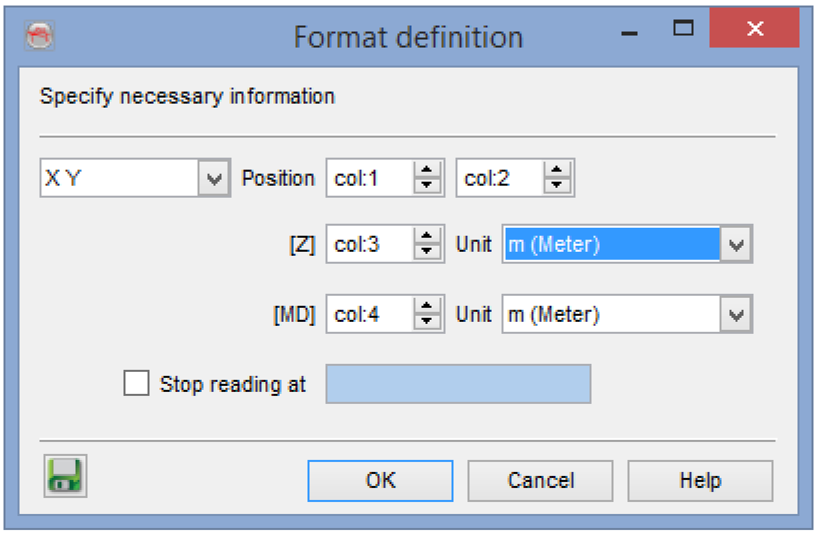
The file containing the track information must contain the true vertical depth sub sea (Z), neither TVD nor TVD-SRD. The MD quantity (measured depth) is always referenced with respect to the reference datum elevation (thus equal to zero at KB, RT or DF).

The file containing the time-to-depth model can either contain depth as TVDSS, TVD-SRD or MD. If the Seismic Reference Datum is not equal to zero and the time-depth model is using TVD-SRD then you must specify the SRD value in the advanced field.

Select the input Ascii file of the deviation survey first. Optionally, a vertical well can be created by deselecting the check box at the left of this field, and entering the surface coordinates, top and bottom depth (measured depth).

The main work is to specify the presence of a *file header* and the file *format definition* for the deviation survey. The header, if present, can be of fixed length (number of lines), or delimited on its last line by a keyword.

Predefined and saved file formats are available by pressing the  icon. Otherwise the format must be manually specified. The Define button gives access to the format definition window.



You must specify in the format definition window the column numbers of the X and Y coordinates (absolute values, not relative to the surface coordinates), in the same unit as used when defining the OpendTect survey. The Z depths that must be provided are TVDSS depths, in meters or feet, increasing downwards and equal to zero at a height of zero (sea level, geoid). This is regardless of the reference datum elevation of the wells or any seismic reference datum of the seismic data. Optionally the fourth column can contain measured depths values to provide the link between MD and TVDSS. Data loading can be stopped at a specific line by providing the adequate keyword.

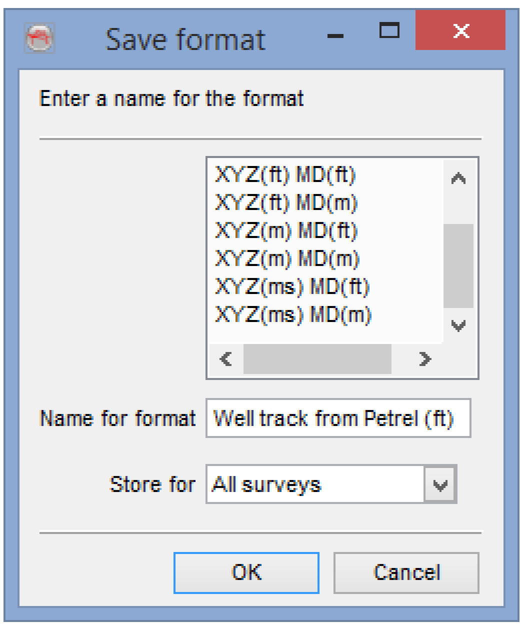
Notes:
The best way to ensure that the reference datum elevation is properly set is to have the deviation survey file start at MD=0. The first lines of the file would then look like:

X	Y	Z (TVDSS)	MD
623255.98	6082586.87	-34.10	0.00
623255.98	6082586.87	0.00	34.10
623255.98	6082586.87	65.90	100
623255.84	6082591.69	440.86	475

In this example the surface coordinates are X=623255.98,Y=6082586.87 and the reference datum elevation is equal to 34.10 m. Please note that the first point of this track is above the sea level, likely to happen in onshore situation, which is not a problem. The first line corresponds to the point with MD=0, while the second point corresponds to the point where TVDSS=TWT=0. (but both points are not mandatory to be present in the track file)

The track file determines the size of the usable and displayed well data. Any log or marker outside of the track Z range will neither be usable nor be displayed. On the other hand the well track is not limited to the survey Z range, especially it can be loaded outside the survey box.

It is recommended to save the format definition for a later use and QC, by clicking on the  icon . In pop-up window, write the name of the format and store it. The format can be stored at different levels (All surveys, Current survey, Current OpendTect user level) depending on the usage.



Next the depth-time model must be specified. A constant interval velocity can be used instead of a file. The selection of a file will lead to the same need to provide the *header* settings and file *format definition*.

Predefined and saved formats are again available by pressing the  icon. Otherwise the format must be manually specified. The *Define* button gives access to the *format definition* window.

You must specify in the format definition window the column where depths and times are located, and the type of data to be expected. Three types of depths are supported for loading a check-shot/Time-depth curve from a file. The supported depths are: Measured depth (MD), TVDSS, TVD rel SRD. Time values can be either one-way or two-way travel-times. Times (lines) that should not be read must all have the same numerical value, which is to be filled in as the *Undefined value*".

DT models are always stored using measured depths and two-way travel times in seconds. Therefore any other input format will cause a conversion of the input data. Data loading can be stopped at a specific line by providing the adequate keyword.

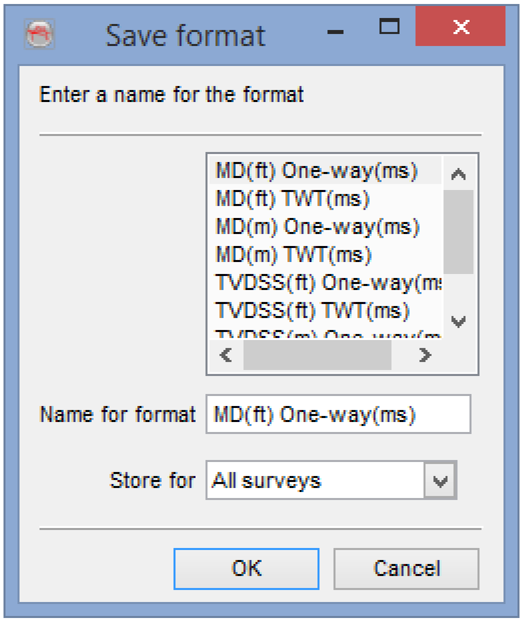
If you want to import a well track with TVDSS as Z input, please be aware that the measured depth of the track will always be ascending. This means that displaying horizontal wells using TVDSS is not supported yet.

Notes:
When loading a depth-time model please check that TWT (TVDSS=0) =0 for MD=KB. This is by definition mandatory. Many package break this relation while matching the well to the seismic data.
Make sure that the time-depth model does not start like the following example, in format TVDSS(m)-TWT(ms):
0.00 0
0.15 60
0.30 60.5
0.45 61

This would cause serious extrapolation problems of the time-depth model above TWT=0. It is highly recommended that the 2nd sample of the time-depth model corresponds to the start depth of your sonic log, unless the input is a measured checkshot survey.

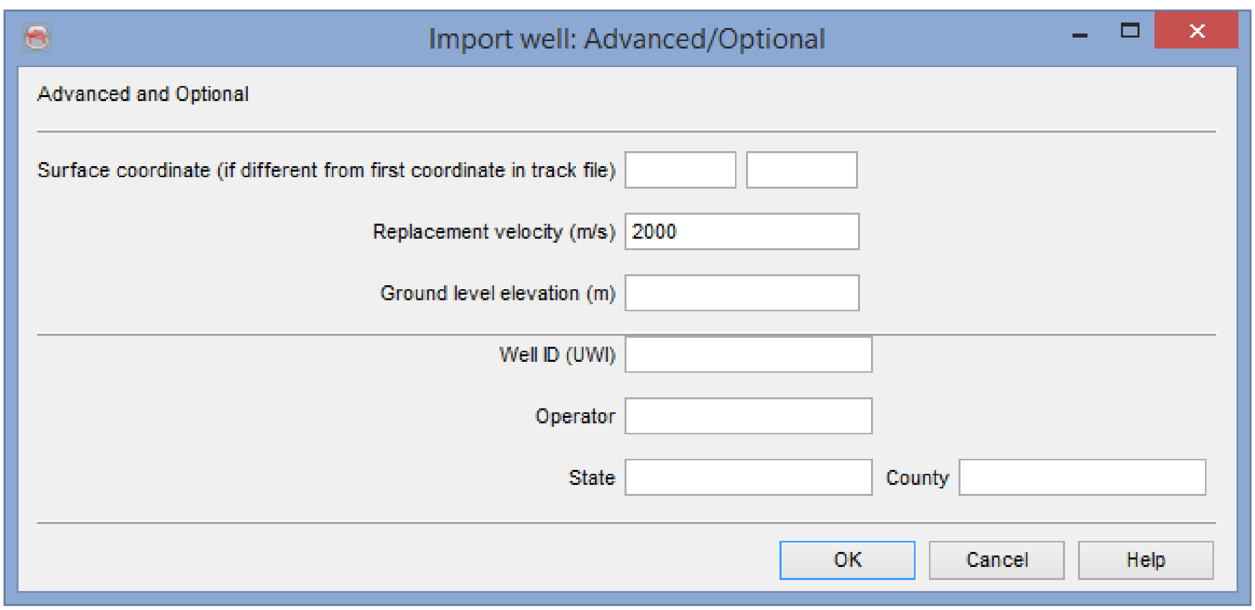
The Time/Depth model used during import can be either a checkshot model or a "normal" time depth curve. More information can be found in the well management chapter.

Similarly the depth-time model reading settings can be saved:



The *Advanced/Optional* button allows the user to provide optional additional parameters.

First surface coordinates can be provided, alternatively to the coordinates written in the first line of the track file.
Then the seismic reference datum (SRD) can be provided. Please note that SRD is not the Kelly Bushing (KB), Rotary table (RT) or drill floor (DF) elevations. Instead and despite being written in the wells header, it is the elevation between the seafloor and TWT=0 in the seismic volume.



Logs

The import of well logs requires the deviation survey to be imported first. Then logs can be imported or computed from the Well manager.

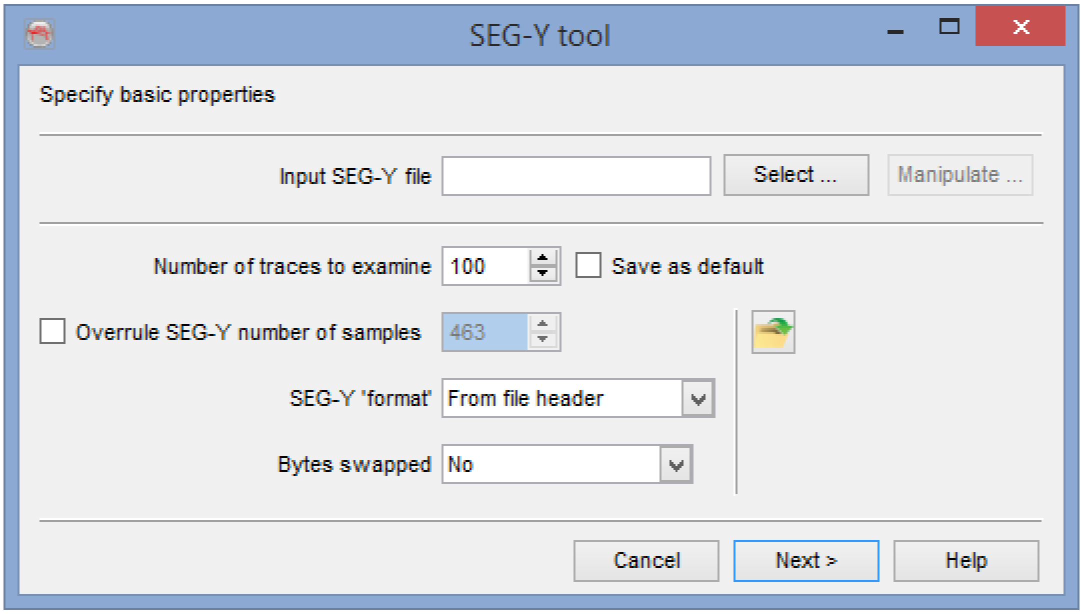
Markers

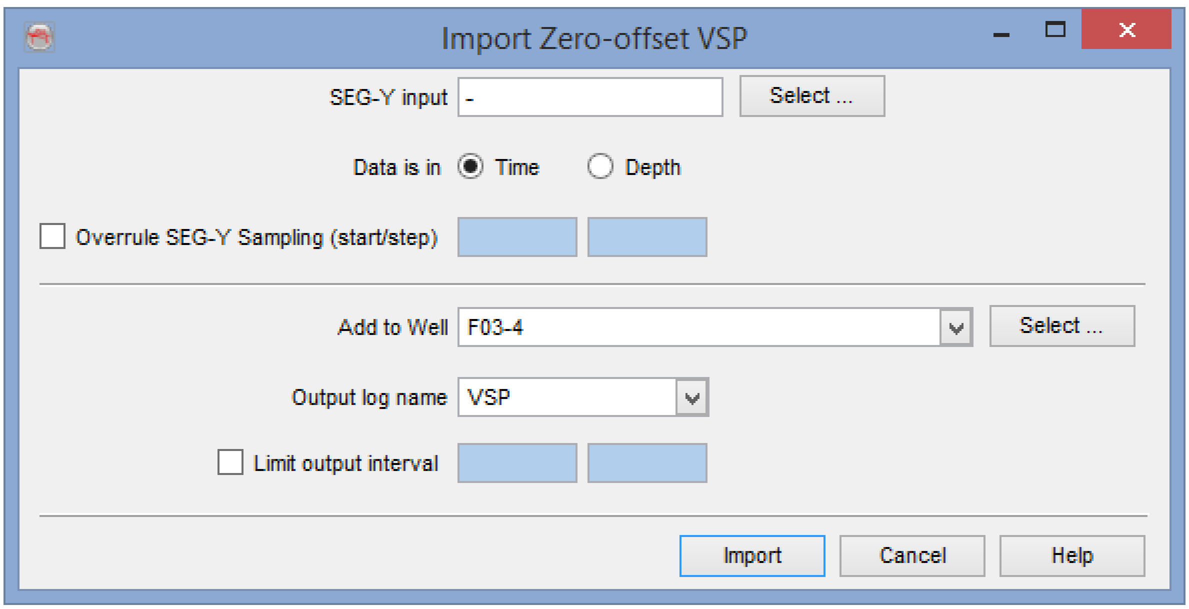
The import of well markers requires the deviation survey to be imported first. Then markers can be imported from the well manager.

Import Zero-Offset VSP

A zero-offset VSP data can be imported for a selected well via *Survey* > *Import* > *Wells* > *VSP (SEGY)*. The system will launch the SEG-Y Tool window. Browse and locate the input file and fill the respective parameters that are described in the Section-Import > SEG-Y. When done, press OK in the SEG-Y tool window to select the input file. In the pop-up window, input VSP file is first selected by pressing the Select button for the SEG-Y-input field. The system will return to *Import Zero-offset VSP* window. In this window, fill the remaining parameters. Select the vertical sampling information (TIme or Depth, MD or TVDSS). The input file can be re-sampled by overruling the sampling information. This is done in the overrule SEG-Y sampling field.

Next step is to attach the VSP as a log on an already loaded well. Select the *Add to the well* field accordingly and write a proper log name that will be the log name inside the OpendTect project. Optionally, the input log can be sub-selected by providing a proper range (Depth) in the Log-output interval. Press Go to import the VSP log.





Simple Multi-Well

Well depths in OpendTect are always referenced using their Measured Depth. The alignment with the seismic data in done using the track (deviation survey) data and time-depth and/or checkshot data. The track data provides the relation between lateral coordinates, TVDSS depths and Measured Depths. The checkshot data and time-depth data provides the relation between Measured depths and travel times (TWT). Other formats can be used during import, but will be converted.

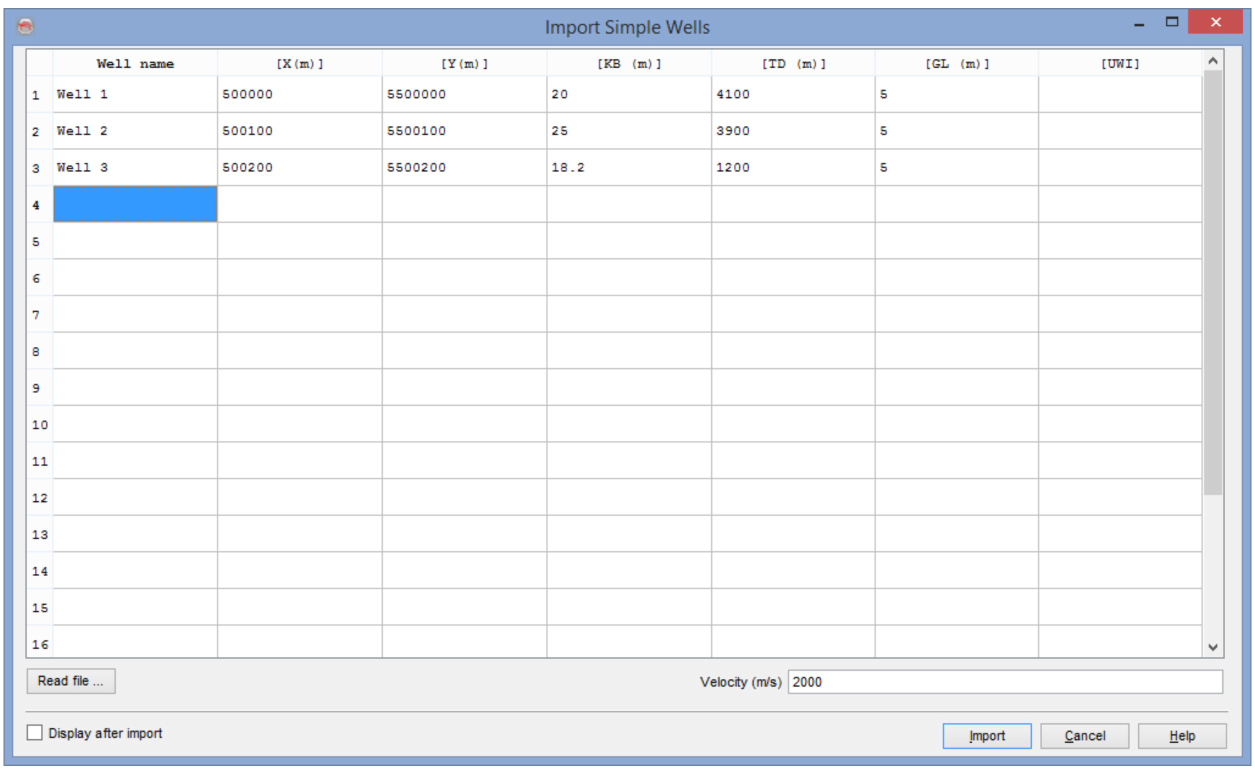
This utility window allows the quick creation of multiple vertical wells with a constant velocity as depth-time model provider. The table window below can either be filled manually or by reading a file.

The following parameters are mandatory:

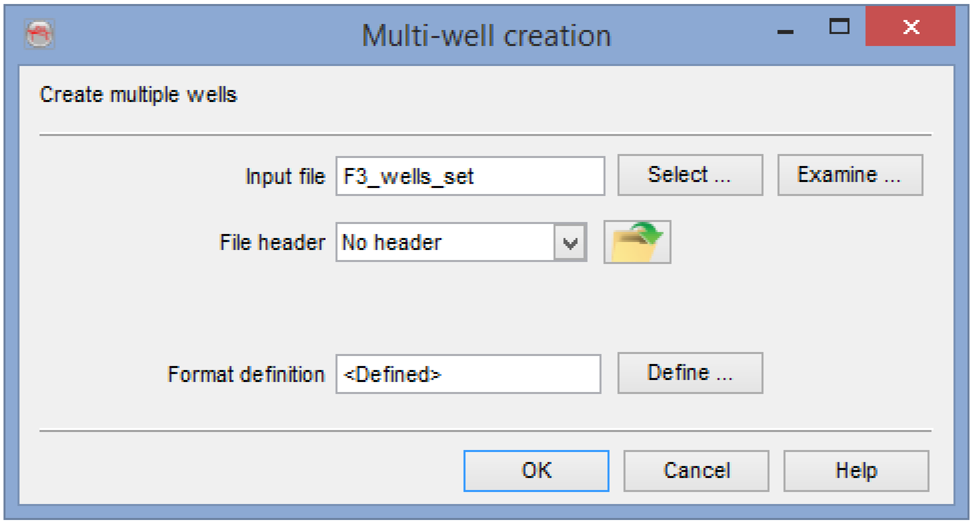
- Well name
- (Vertical) position along the X axis, in the same unit as the survey geometry.
- (Vertical) Position along the Y axis, in the same unit as the survey geometry.
- Reference datum elevation (KB or other): Altitude measured from sea level of the point MD=0., positive upwards. Can be left to 0 if unknown.
- Total depth (TD): Largest measured depth in the well. This parameter is half optional; If not provided the well track is created such that it will reach the survey base.

The following parameters are optional:

- Seismic reference datum (SRD): Altitude measured from sea level of the point TWT=0ms, positive upwards. SHOULD NEVER be larger than KB!
- UWI (Unique well identifier): You can input any number, string or combination.

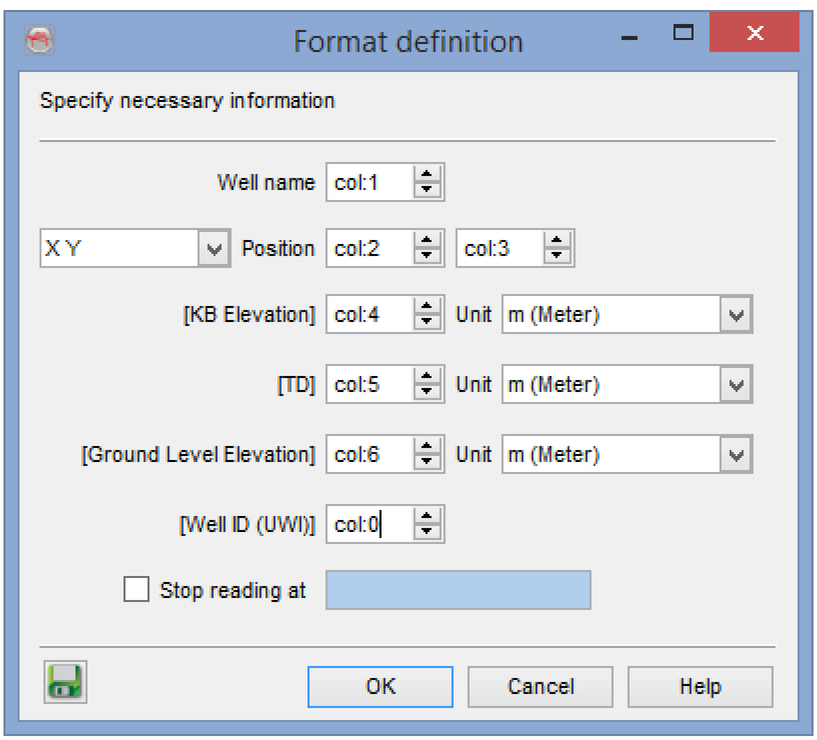


To read a file containing that information, press *Read* file and select the input Ascii file. One line in this file should correspond to one line in the output table.



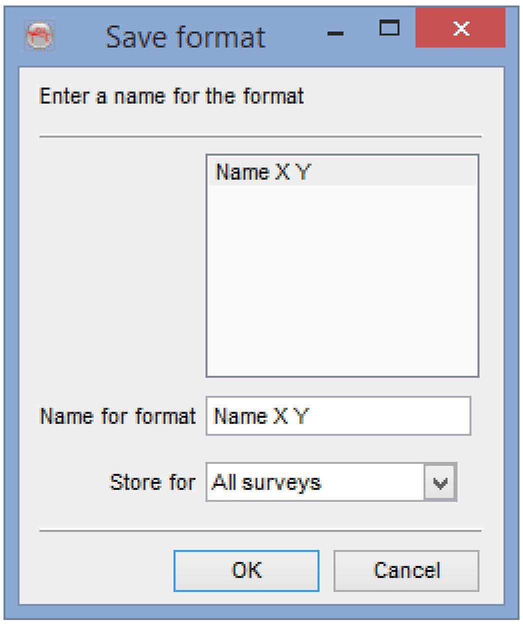
The main work is to specify the presence of a *file header* and the file *format definition*. The header, if present, can be of fixed length (number of lines), or delimited on its last line by a keyword.

Predefined and saved file formats are available by pressing the icon. Otherwise the format must be manually specified. The *Define* button gives access to the format definition window.



You must specify in the format definition window the column numbers of the X and Y coordinates (absolute values, not relative to the surface coordinates), in the same unit as used when defining the OpendTect survey. Reference datum elevation and TD should also be provided, while the SRD and UWI are less frequently used. Please note that KB and SRD both increase upwards and are positive above sea level, whereas MD is a depth and increases downwards (MD is never negative).

It is recommended to save the format definition for a later use and QC, by clicking on the  icon. In pop-up window, write the name of the format and store it. The format can be stored at different levels (All surveys, Current survey, Current OpendTect user level) depending on the usage.



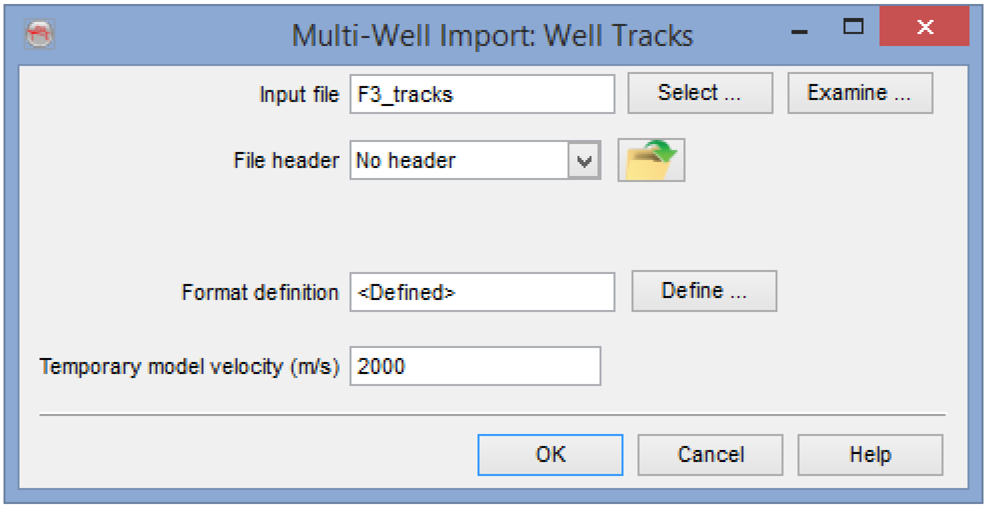
Bulk

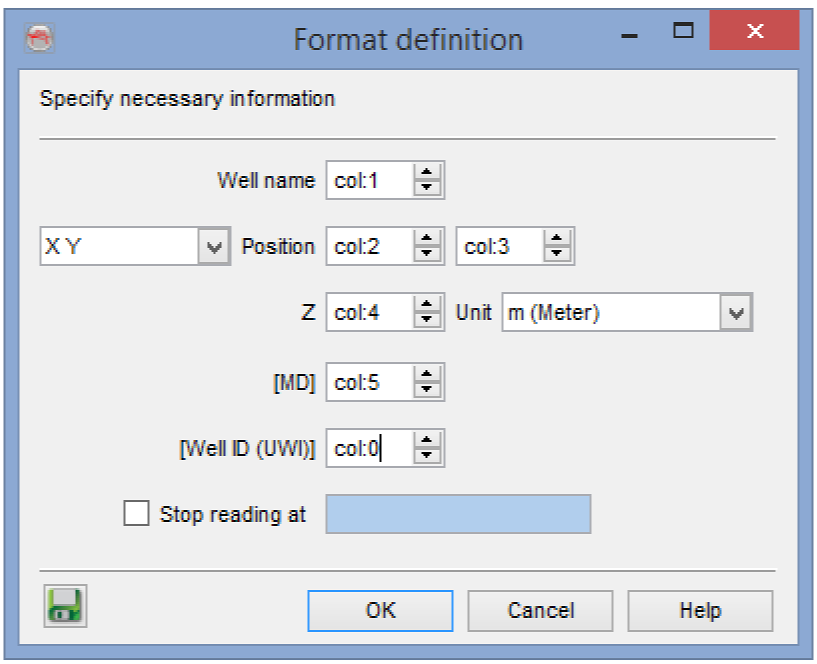
The bulk import tool allows for the import of well-data for different wells from one or several files. The data is matched against primarily the well name and, if available, against the Unique Well Identifier (UWI). This has the following implications:

- The well name must appear on each line of the input file. If the well already exists, then the UWI must match the database. The same applies for the UWI if it is used in combination with the well name.
- The well name should not contain spaces, otherwise the matching with a given column number will not work as expected.

Bulk Well Track Import

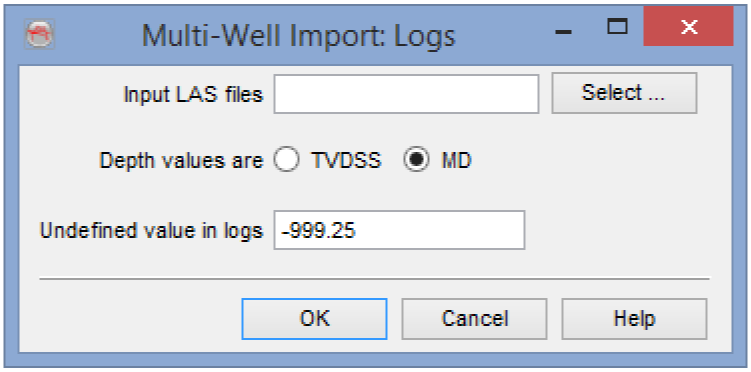
This dialog allows you to import tracks of several wells from one single file. Please read the note on the bulk well import tool. Apart from that, the specification for the input data is identical to the simple track ascii import.





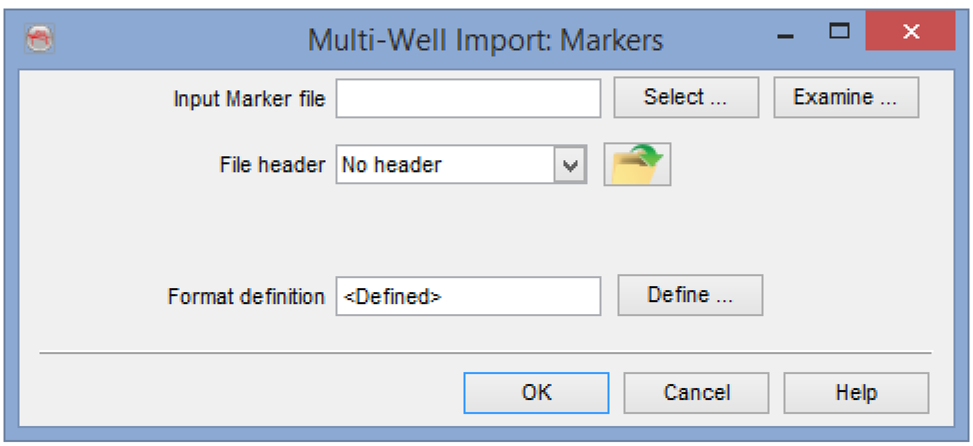
Bulk Well Log Import

This dialog allows you to import several las files in OpendTect. It may be used to create a track and dummy time-depth model if necessary, if the name of the well in the file does not match the current well database. Survey geometry and time-depth models can be later imported from the well manager.

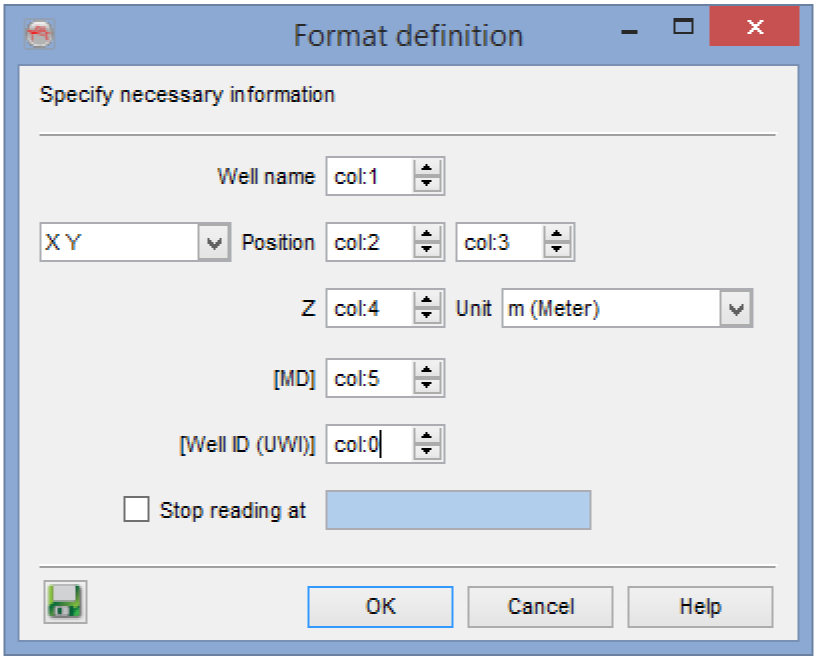


Bulk Well Marker Import

This bulk import tool allows the import of markers from several wells from a single file. Please read the note on the *bulk well import tool*.



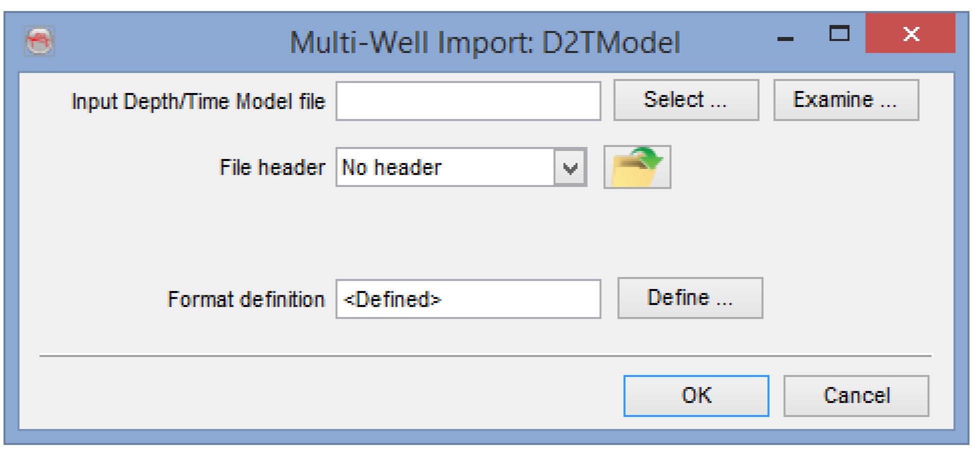
In the *Format Definition* dialog (Click *Define*...):



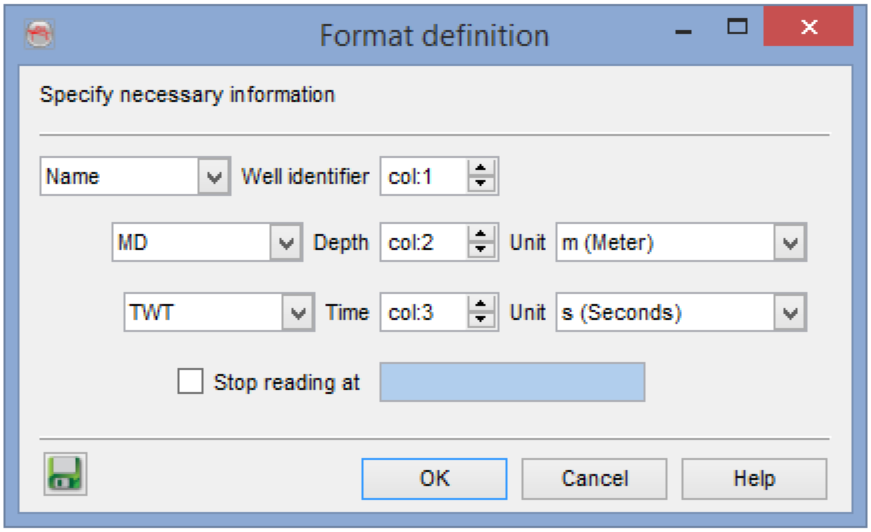
You will have the option to select either the well *Name* or *UWI* (Unique Well Identifier). And also to set depth as either *MD* or *TVDSS*. You may also toggle on the 'Stop reading at' choice and set a value here.

Bulk Well Time-Depth Model Import

Utilize this option for importing one or several time-depth models into OpendTect:



In the format definition, you can set the columns for Well Identifier, MD and TWT.

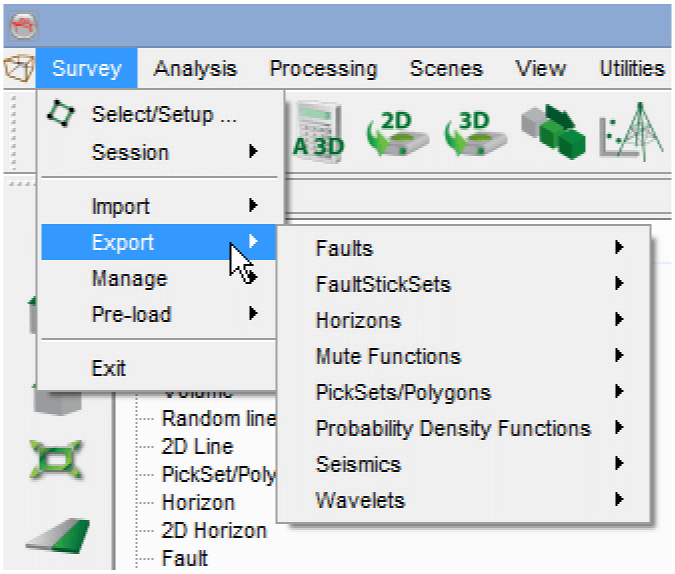


This format definition can be saved for later use via .

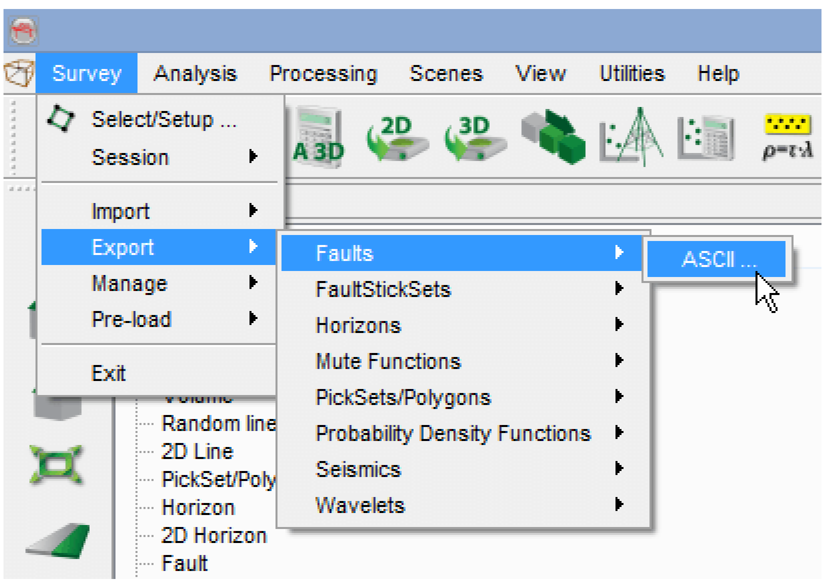
Export

The following data can be exported from OpendTect:

- Faults
- FaultStickSets
- Horizons
- Mute Functions
- Picksets/Polygons
- Probability Density Functions
- Seismics
- Wavelets

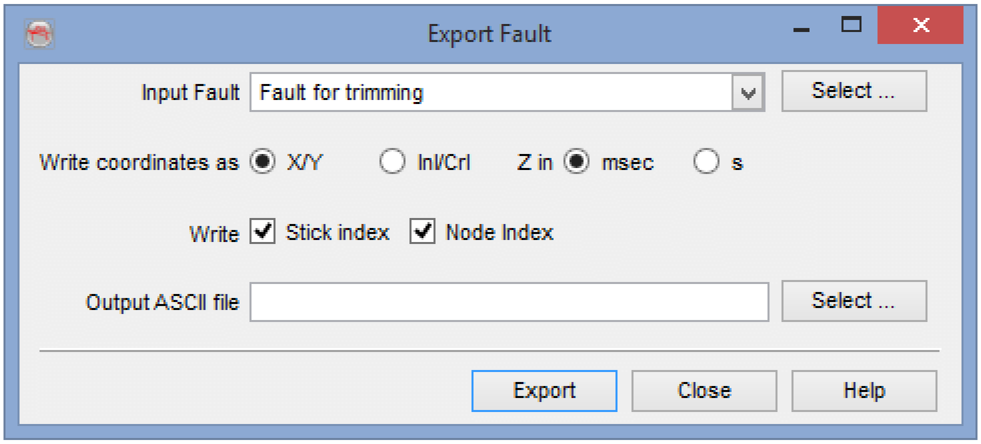


Export Faults



Export Faults:

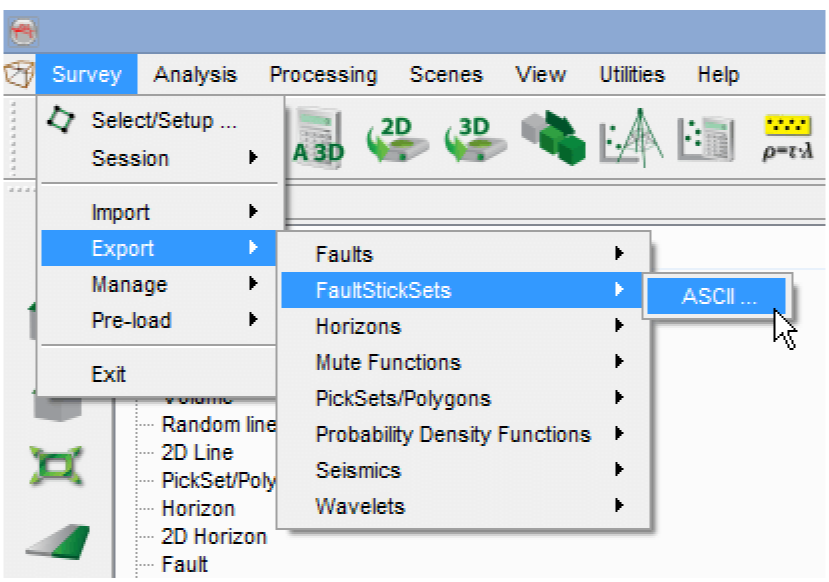
Faults can be exported as Ascii file from window that can be launched via *Survey* > *Export* > *Fault* > *Ascii...*



Faults Export Window

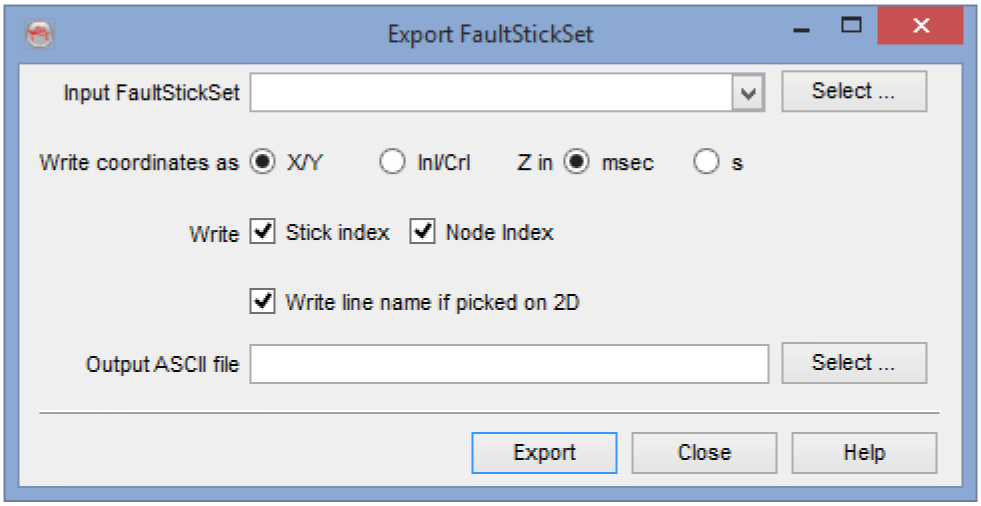
In this window, it is possible to specify the output format. First, select the input fault to export. Then, select the way the coordinates will be written: X/Y or Inl/Crl. By default Z values are stored in msec, unselect the option to store Z in second. Optionally it is possible to write a stick index and/or a node index. Finally, choose a name for the *Output Ascii file*. Press *Ok* to start exporting the fault.

Export FaultStickSets

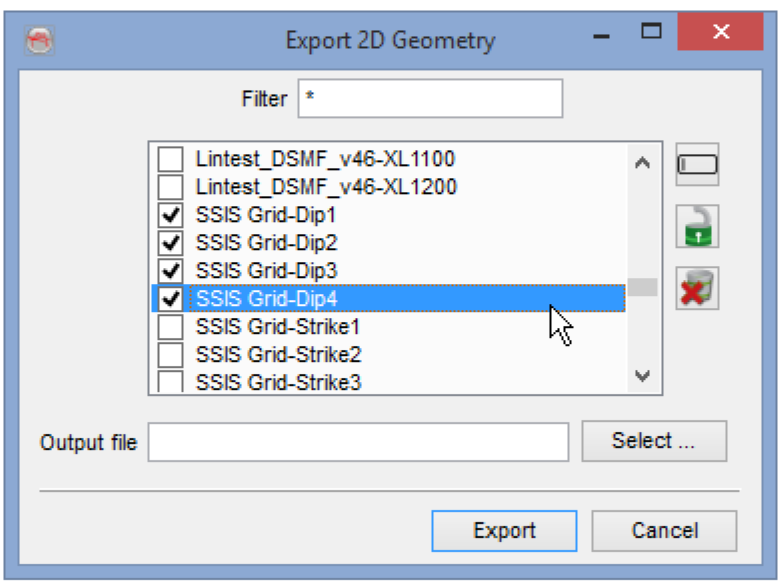


Export FaultSticksets:

The FaultSticksSets are exported by selecting *Survey* > *Export* > *FaultSticksSets* > *Ascii...* sub menu. It will launch export FaultStickSets window. In that window select the input faultstickset and mention the output coordinates format (X/Y or Inline/crossline). By default the option to export Z in msec is checked and it will add a z-column in the output file. In a depth survey, the Z values are exported by default in meter and you need to specify if they are in feet. Optionally the sticks can be stored with a node/stick index or a stick index only. Depending upon other packages requirements, an appropriate selection can be made for adding index information in the file. Locate the output file location with an output name. If you want to add a line name in the output file name, set check to the Write line name if picked on 2D field. Press OK to start exporting the fault sticks.



Export Geometry 2D

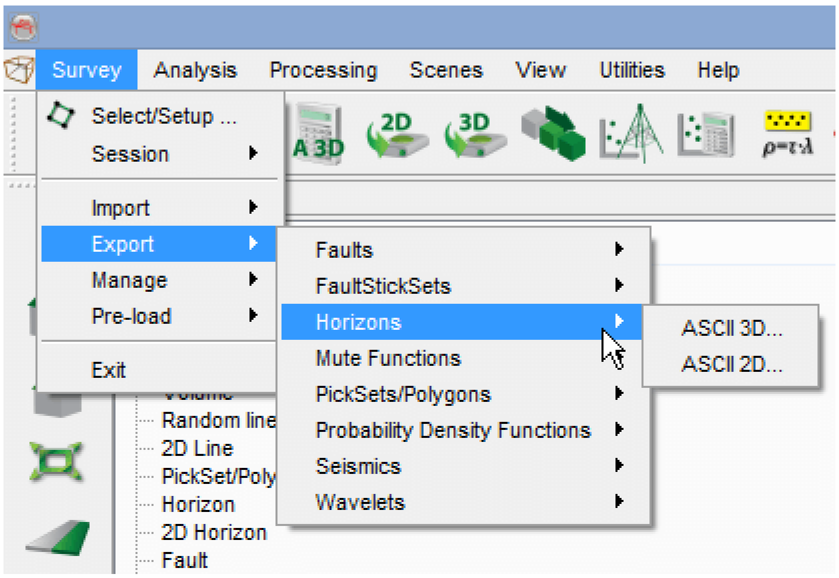


Allows users to export an ASCII file with the geometry of one or several 2D lines can be generated. By default, the output file contains trace numbers and X/Y locations.

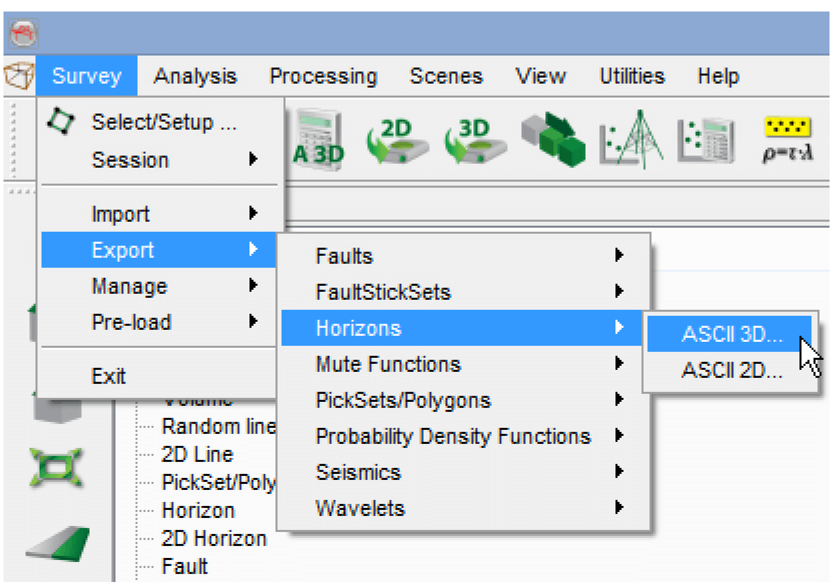
This export facility may be very practical if you want to generate a base map from 2D lines for a different software package.

Export Horizons

Horizons in OpendTect can be exported in 2D/3D Ascii format. (Or, if you have the relevant Workstation Access, directly to other software.)



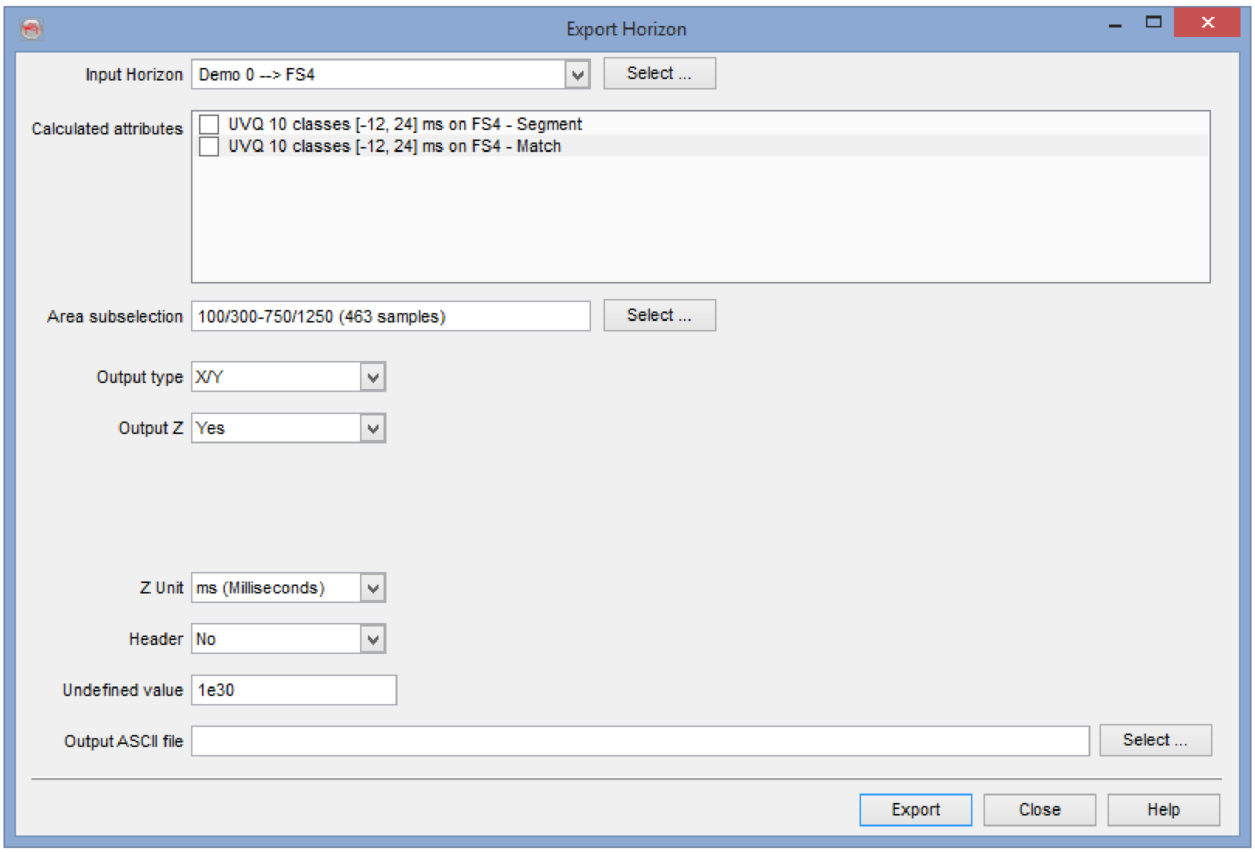
Export Ascii 3D Horizons

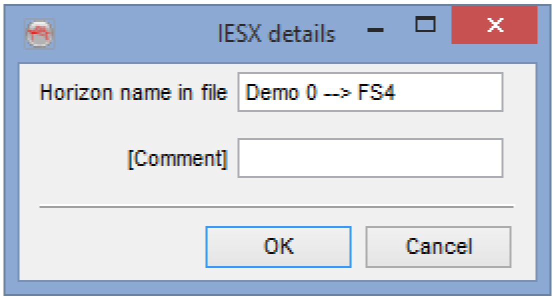


In the pop-up window, select the input surface (horizon). The stored surface data are listed in the *Calculated attributes* list. You can select one/more calculated attributes to be written in the output file. To de-select an attribute, use CTRL-key and left mouse click or click in the empty space. You can make a sub selection of the *Available patches*. Each patch will be stored in a different file with extensions ^1, ^2, etc.

Three types of ascii files are supported:

- X/Y saves in x/y information.
- Inl/Crl saves inline crossline.
- IESX (3d_ci7m) saves in GeoFrame IESX format. In this case you need to choose if the survey co-ordinate system as it is set up in OpendTect is in *feet* or in *meter*. Optionally, the horizon name in the file can be edited by clicking on the *Settings* button which appears on selection of the IESX format as output type.

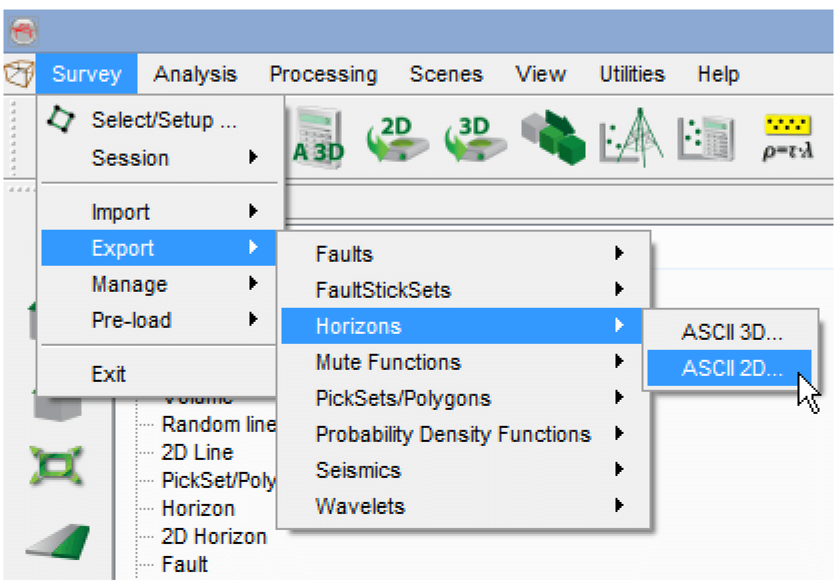


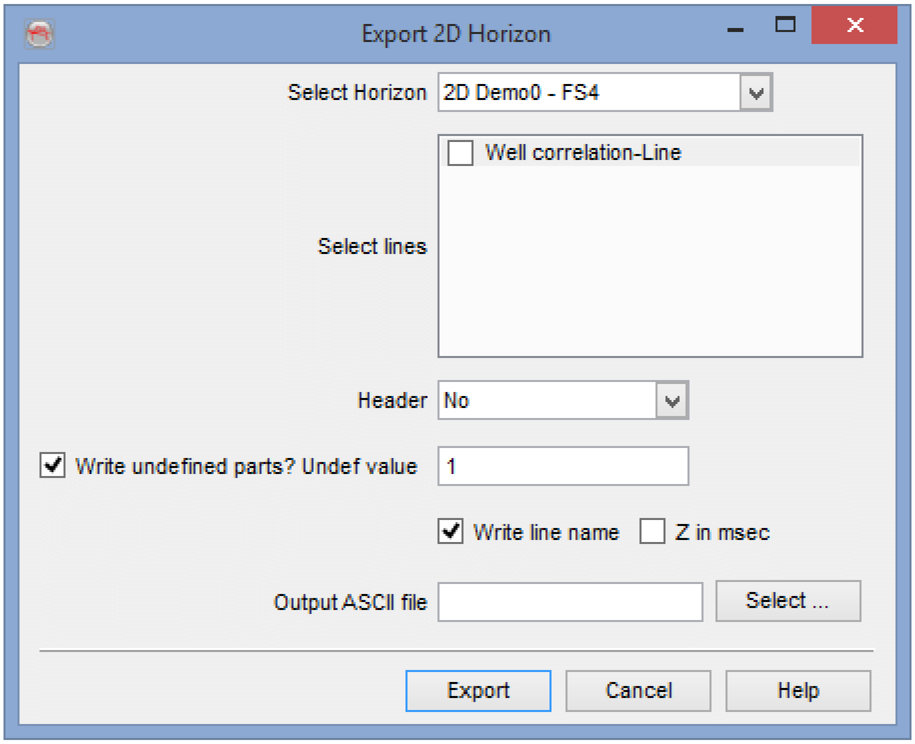


If the output type selected is either X/Y or Inl/Crl, optionally Z can also be outputted. Z can be in time (If time survey, units: in seconds, milliseconds or microseconds ; if depth survey, units: Meter, Feet, Inches, -Meter or -Feet) or Z can also be *transformed* and converted to depth using either a linear relationship (Vo and dV/dt to be specified), a velocity cube or a well (Z outputted in Meter, Feet, Inches, -Meter or -Feet if time survey, in seconds, milliseconds or microseconds if depth survey). Z is written as the last column of the output file. Additionally, a single line or multi line eader can be included in the output file. The standard value of undefined data can also be specified.

Finally, select the output file location with an appropriate file name. Press *Go* to start exporting the horizon.

Export Ascii 2D Horizons

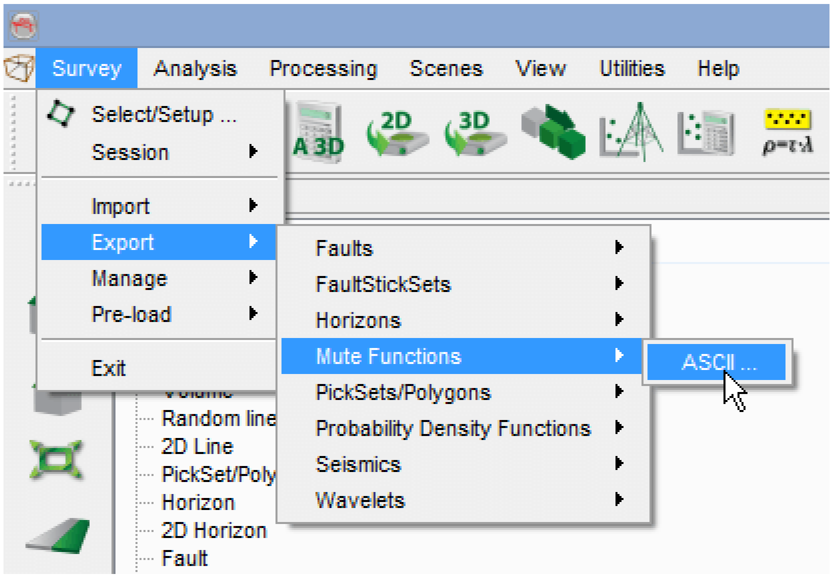




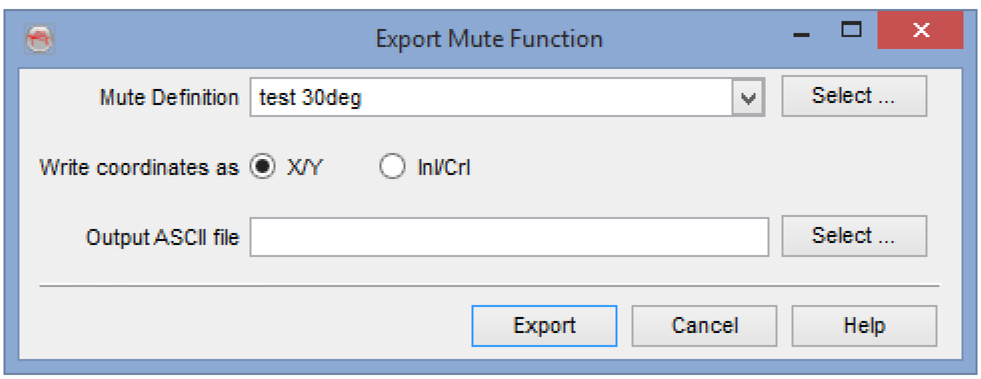
2D Horizon Export Window

For exporting a 2D horizon, you begin by selecting the horizon and the required 2D lines. If a header is needed it can be specified as either single line or multi line. Further, if the undefined parts in the data have to be written a standard value needs to be specified. Similarly, the Z value can be explicitly output in milliseconds irrespective of the original format used in the survey, it will be stored in seconds otherwise.

Export Mute Functions

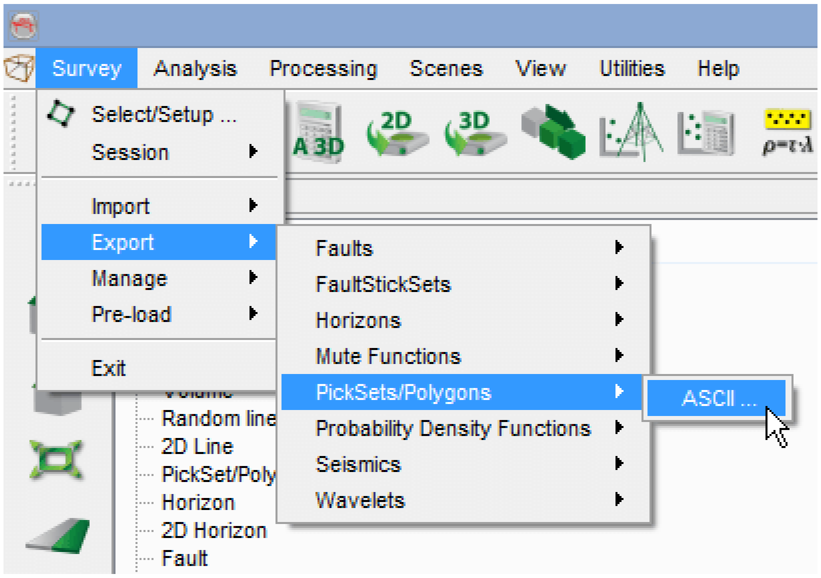


The format definition will be a column sorted text file with time (or depth if depth survey) and offset values, and with coordinates either in X/Y or Inline/Crossline.

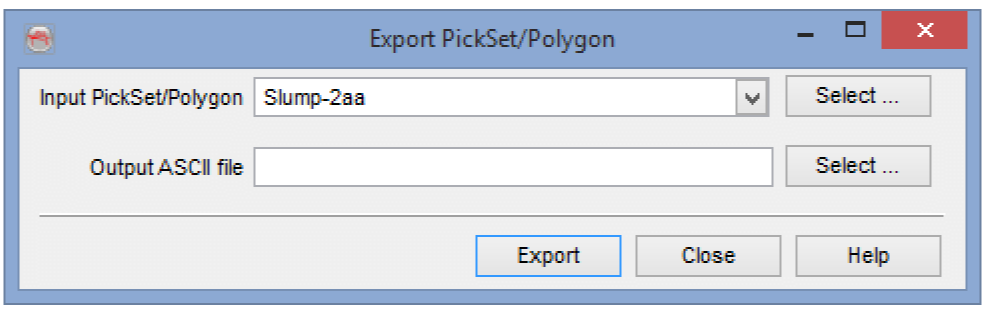


Mute Function Export Window

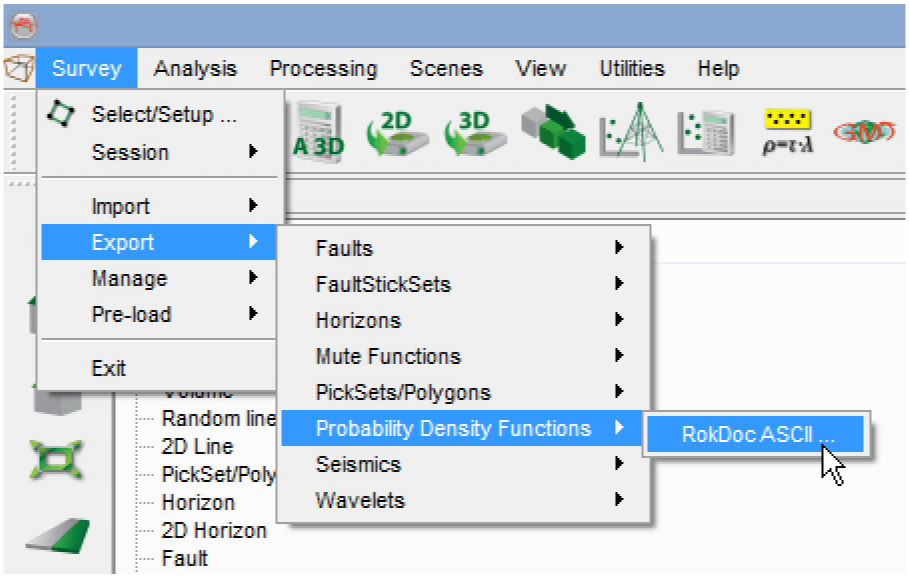
Export PickSets & Polygons



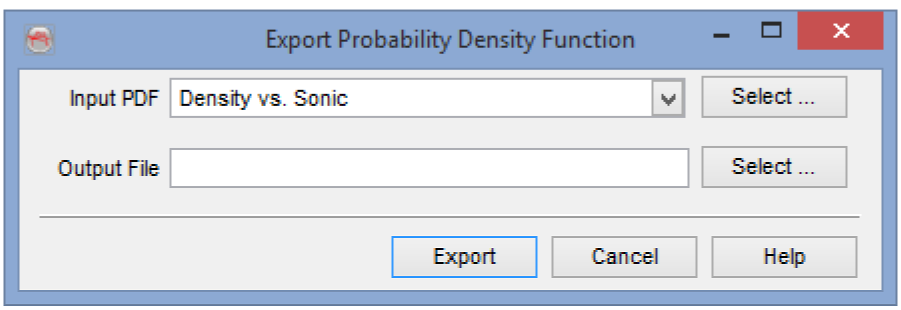
The export pickset module is used to export picksets or polygons. One can think of outlines of structures, picked faults, etc.



Export Probability Density Functions



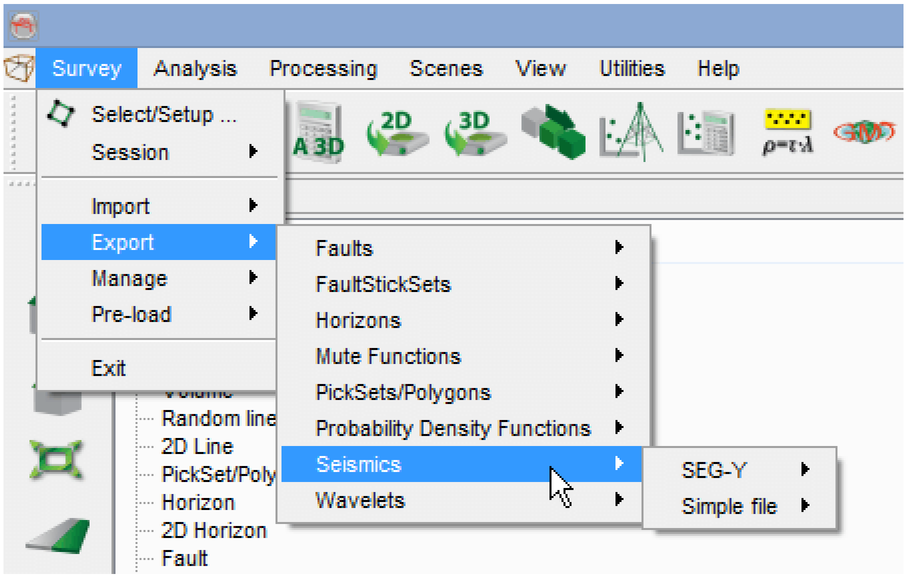
The selected OpendTect PDF data will be exported to an Ascii file, in RokDoc format.



Probability Density Function (PDF) Export Window

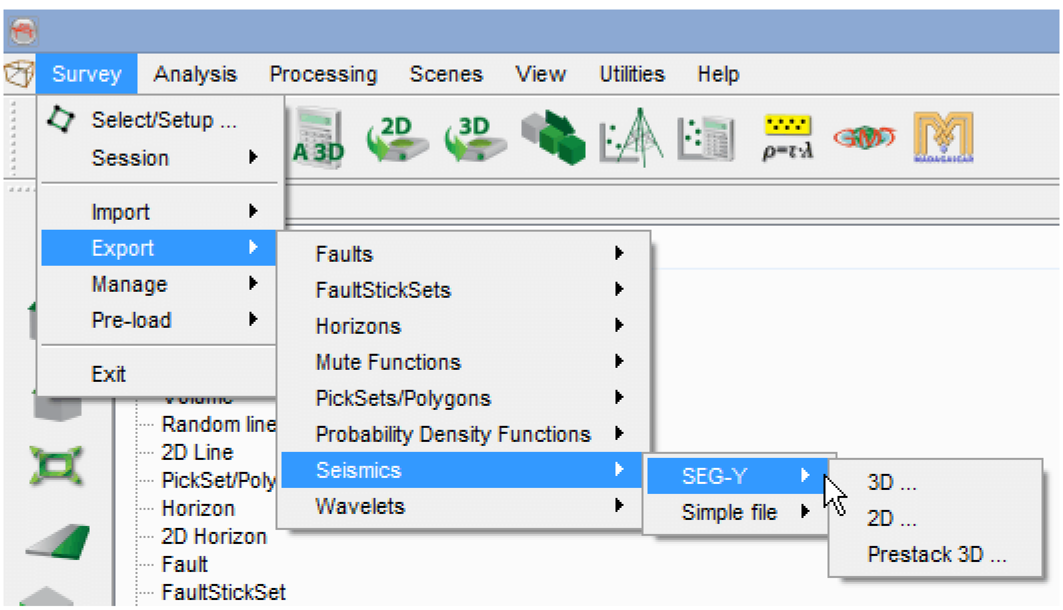
Export Seismics

The *export seismic* module is used to export 2D/3D seismic data as SEGY and/or as simple File.
2D/3D Seismic data from OpendTect can also be exported to GeoFrame Workstation (the option is present if the relevant Workstation Access plugin is installed.)



Export SEG-Y

2D/3D Poststack and 3D Prestack data can be exported from OpendTect in SEG-Y format:



The SEG-Y revision 1 default bytes locations will be used during export, but additional positions can be used with the personal setting keywords listed on the right-hand side.

The pick is a trace attribute stored in the OpendTect seismic files. It is most often not used.
The reference number is most often the Shot Point for 2D data but could be used for anything else. Please note that a SEG-Y scalar at bytes 201-202 apply for values stored in bytes 197-200. The SP scalar will always be -10, thus the value written on bytes 197-200 is 10 times the SP value.

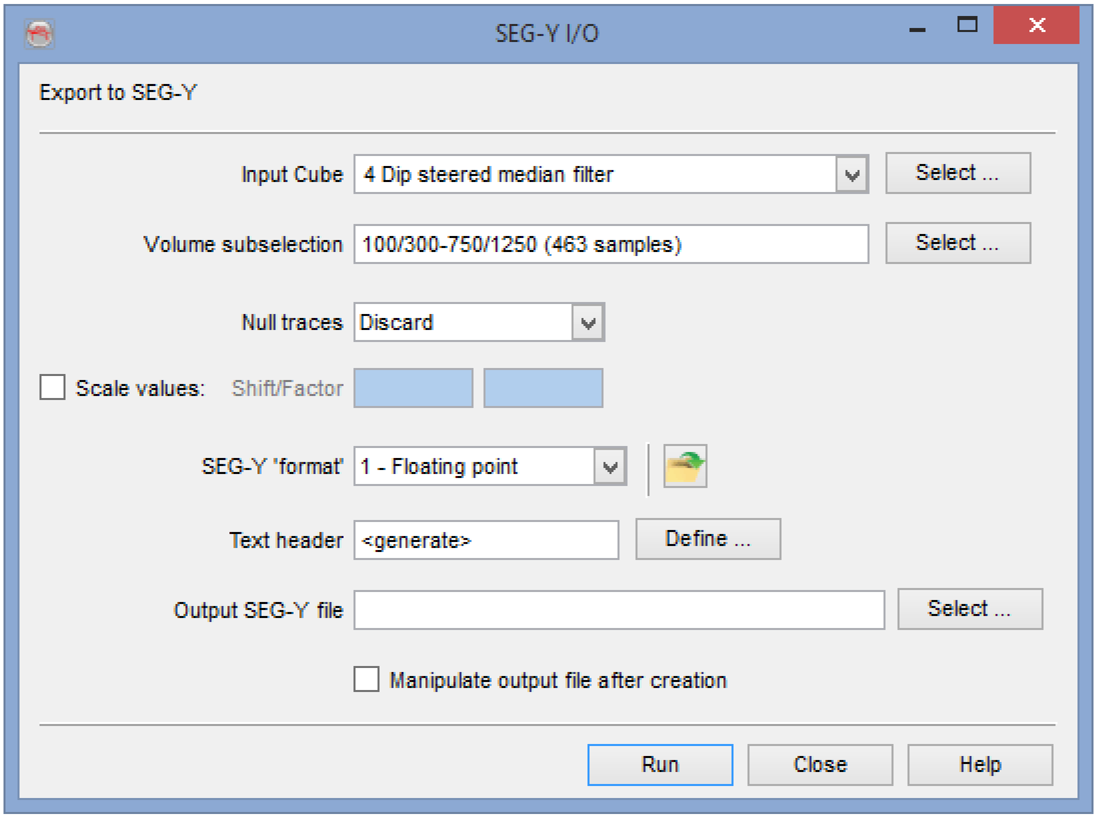
All values listed above are encoded on 4 bytes by default. The byte length can be overridden using the following personal key words, except for the coordinates:

- SEG-Y.Nr bytes for In-line
- SEG-Y.Nr bytes for Cross-line
- SEG-Y.Nr bytes for Offset
- SEG-Y.Nr bytes for Azimuth
- SEG-Y.Nr bytes for trace number
- SEG-Y.Nr bytes for Pick
- SEG-Y.Nr bytes for RefNr

The layout of the SEG-Y export window changes slightly based on the data type.

Export SEG-Y 3D

Stored 3D volumes can be exported from OpendTect in SEG-Y format.



All fields are optional, except the output filename that must be provided. The export will be launched when pressing Ok.

Volume subselection: Can be defined here in various ways.

Null traces: They can be either discarded or written in the SEG-Y file. For 3D SEG-Y export there is a third option in addition to the two described previously, Add. It basically adds null traces where there is NO data present in the seismic cube, such that the output SEG-Y cube will be without any gaps for one individual inline or crossline.

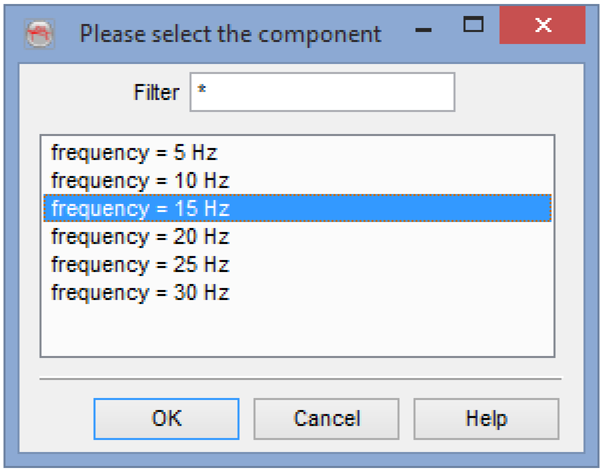
Scale values: A linear scaling can be applied while exporting the data.

SEG-Y format: Please note that this option may clip your data if the output format has less bytes than the input OpendTect format.

Text header: The SEG-Y textual header is automatically created, but may be provided by the user, either from a text file or directly from a SEG-Y file.

Multi-Component Export:

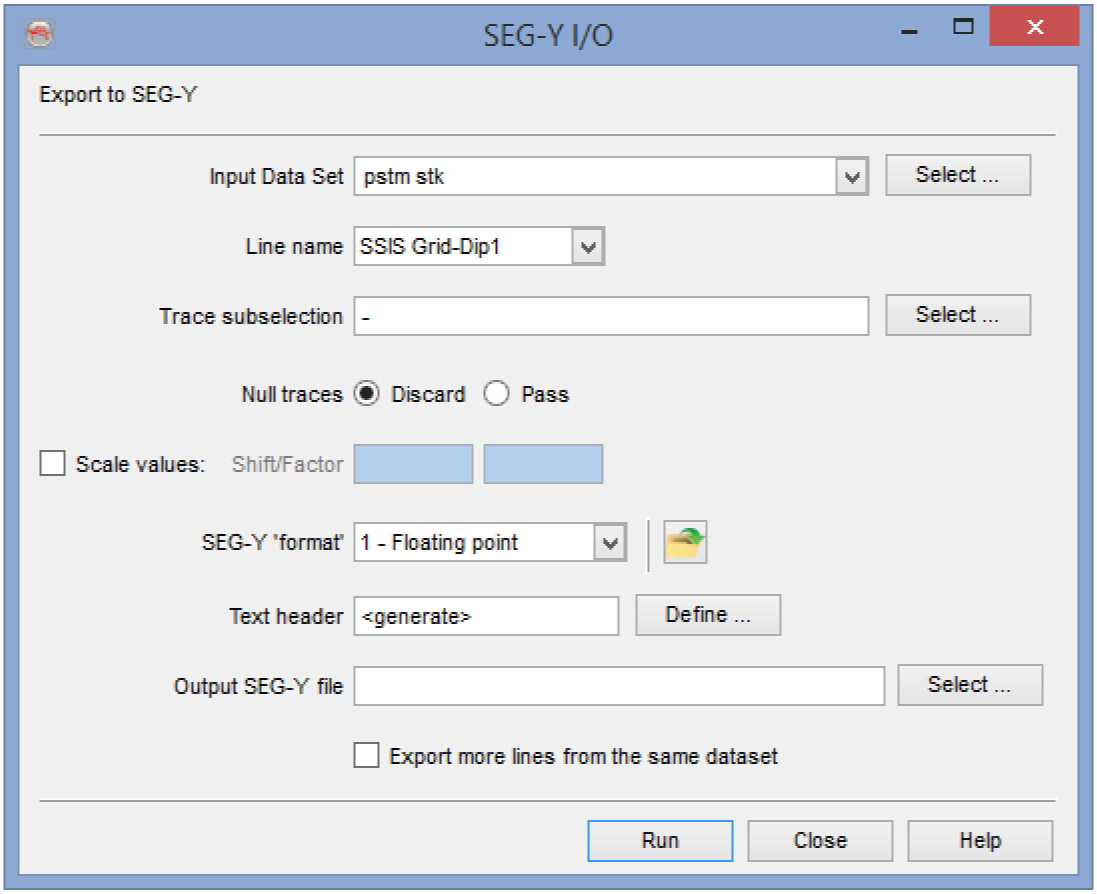
If the input cube contains multiple components, an additional window will pop-up on pressing *'Run'* and ask for which component to output (see below), since SEG-Y files can only contain one component per file.



Tip: It is a good practice to display the 3D seismic data on a z-slice to check for any gaps in inline/cross-lines or the presence of null traces, before exporting it.

Export SEG-Y 2D

Stored 2D data can be exported from OpendTect in SEG-Y format.



All fields are optional, except the output filename that must be provided. For 2D you also need to select a specific line. More lines can be exported if the option '*Export more from same dataset*' on the last line is selected.The export will be launched when pressing Ok.

Input Data Set: Select the data set to be exported.

Line name: The lines available in the data set are listed here. Only one line can be selected.

Trace subselection: Can be defined here in various ways.

Null traces: They can be either discarded or written in the SEG-Y file.

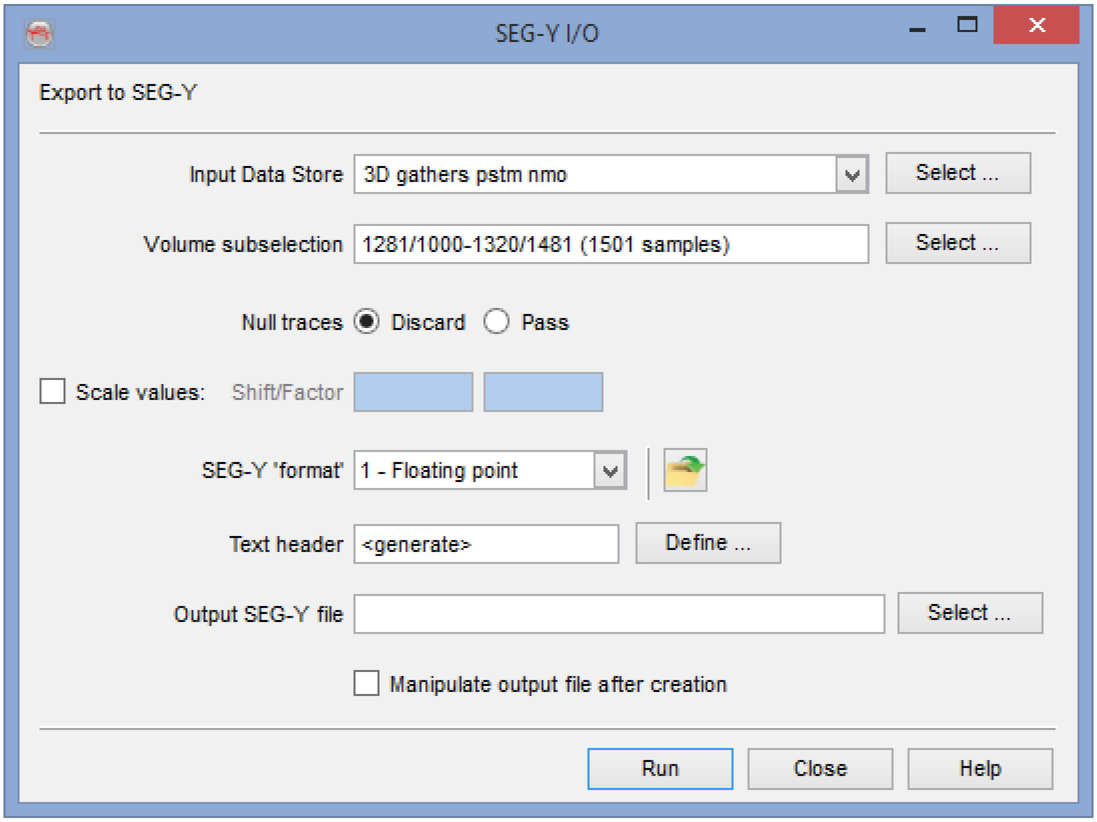
Scale values: A linear scaling can be applied while exporting the data.

SEG-Y format: Please note that this option may clip your data if the output format has less bytes than the input OpendTect format.

Text header: The SEG-Y textual header is automatically created, but may be provided by the user, either from a text file or directly from a SEG-Y file.

Export SEG-Y Prestack 3D

Prestack 3D data may be exported from OpendTect in SEG-Y format.



All fields are optional, except the output filename that must be provided.

Trace subselection: Can be defined here in various ways.

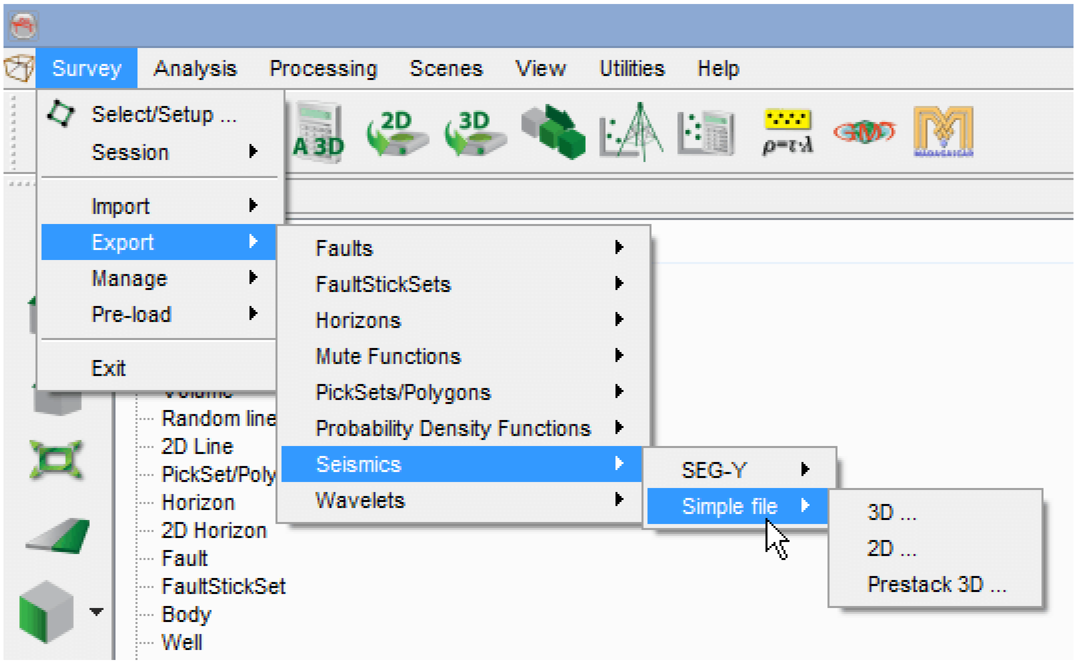
Null traces: They can be either discarded or written in the SEG-Y file.

Scale values: A linear scaling can be applied while exporting the data.

SEG-Y format: Please note that this option may clip your data if the output format has less bytes than the input OpendTect format.

Text header: The SEG-Y textual header is automatically created, but may be provided by the user, either from a text file or directly from a SEG-Y file.

Export Simple File

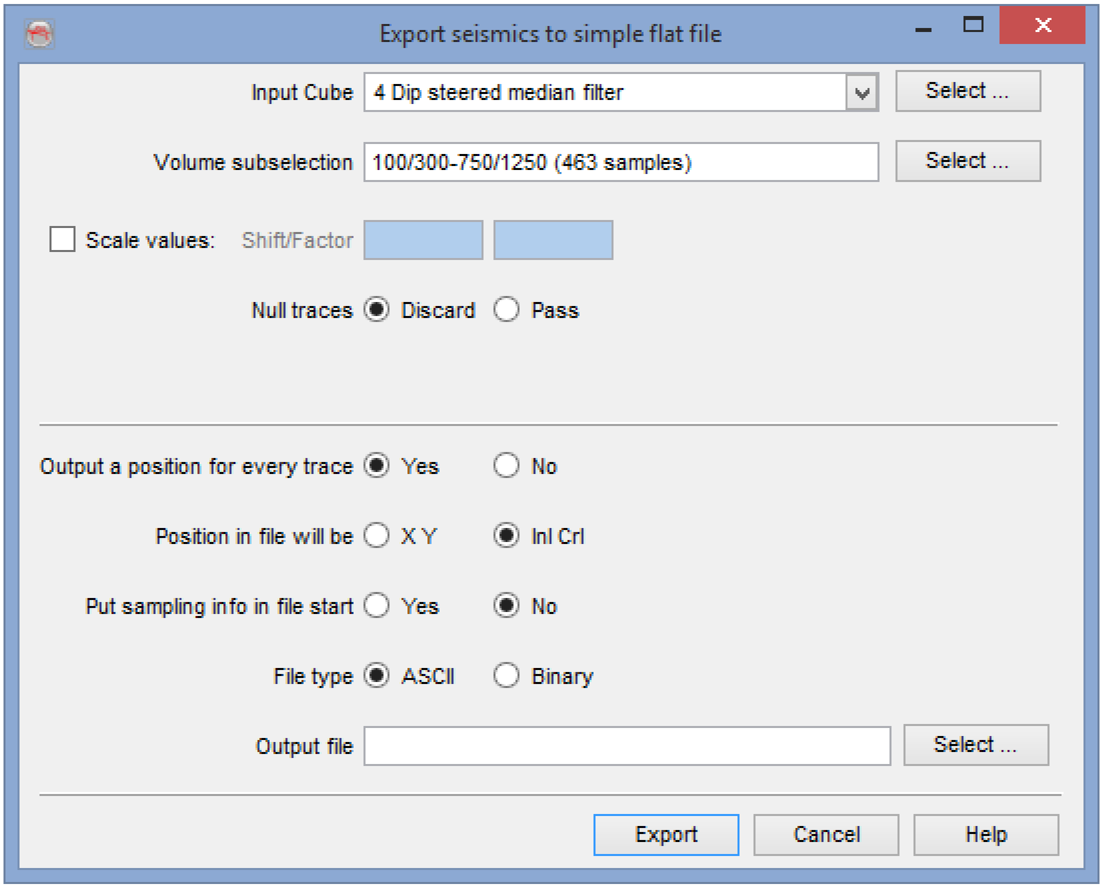


Export options for Simple File format seismic include:

- 3D
- 2D
- Prestack 3D

Export 3D Simple File Format

The stored volumes in the OpendTect survey can also be exported as a simple Ascii or binary file.



Simple 3D Seismic File Export Window

Input Cube: Select the volume to export.

Volume Subselection: A part of the line can also be sub-selected. If the entire line is meant to be exported: after clicking on Select..., define the number and time ranges. Leave the values default or select All in the trace subselection window.

Scale values: The data can be scaled in the output. The output will be calculated with: `Output = Factor * Input + Shift`

Null traces: Null traces can be discarded or left inside the line (Pass).

Output a position for every trace: If *Yes* then the output will contain a short header information. If *No*, no header will be added.

(If *Yes* selected in the previous option) Position in file will be: In the output file, the position information can either be XY locations or Inline/cross-line numbers

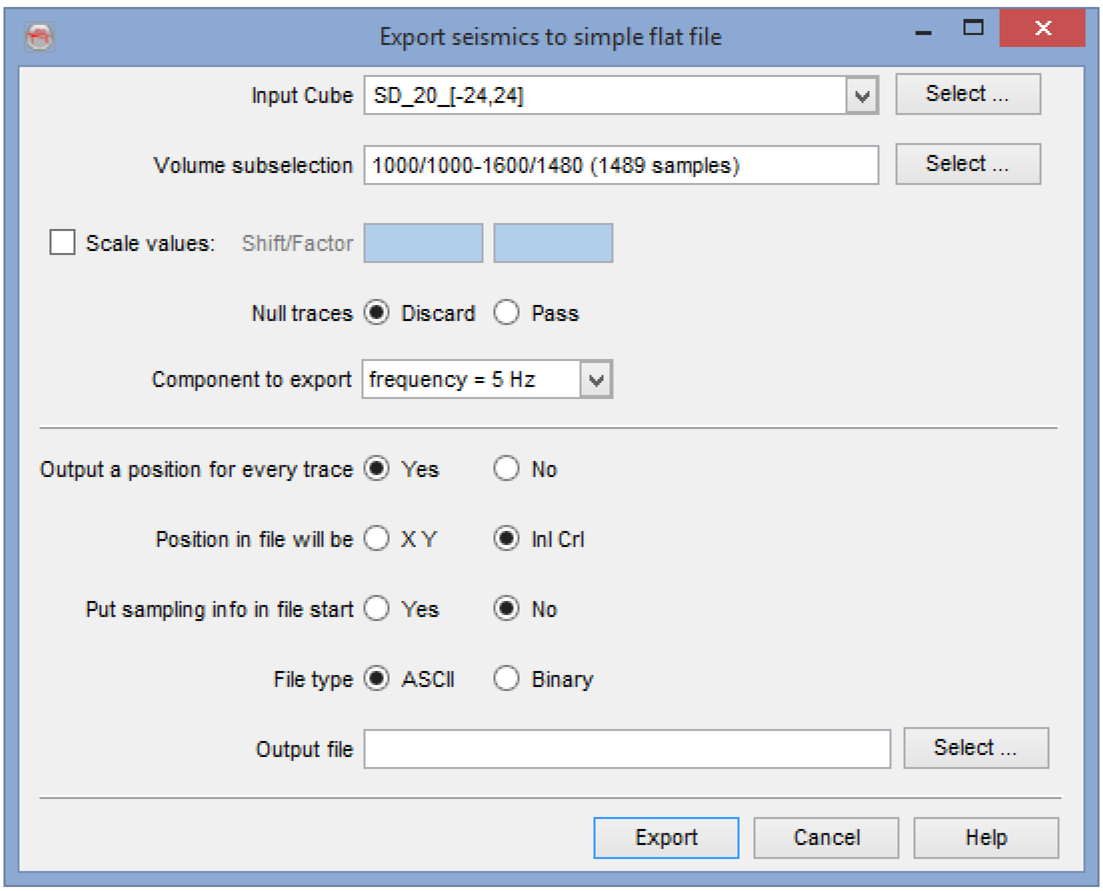
Put sampling info in file start: Select *Yes* to allow the sampling information to appear at the beginning of the file.

File type: Select the appropriate output file type: Ascii or Binary.

Output file: Select/write the output file location.

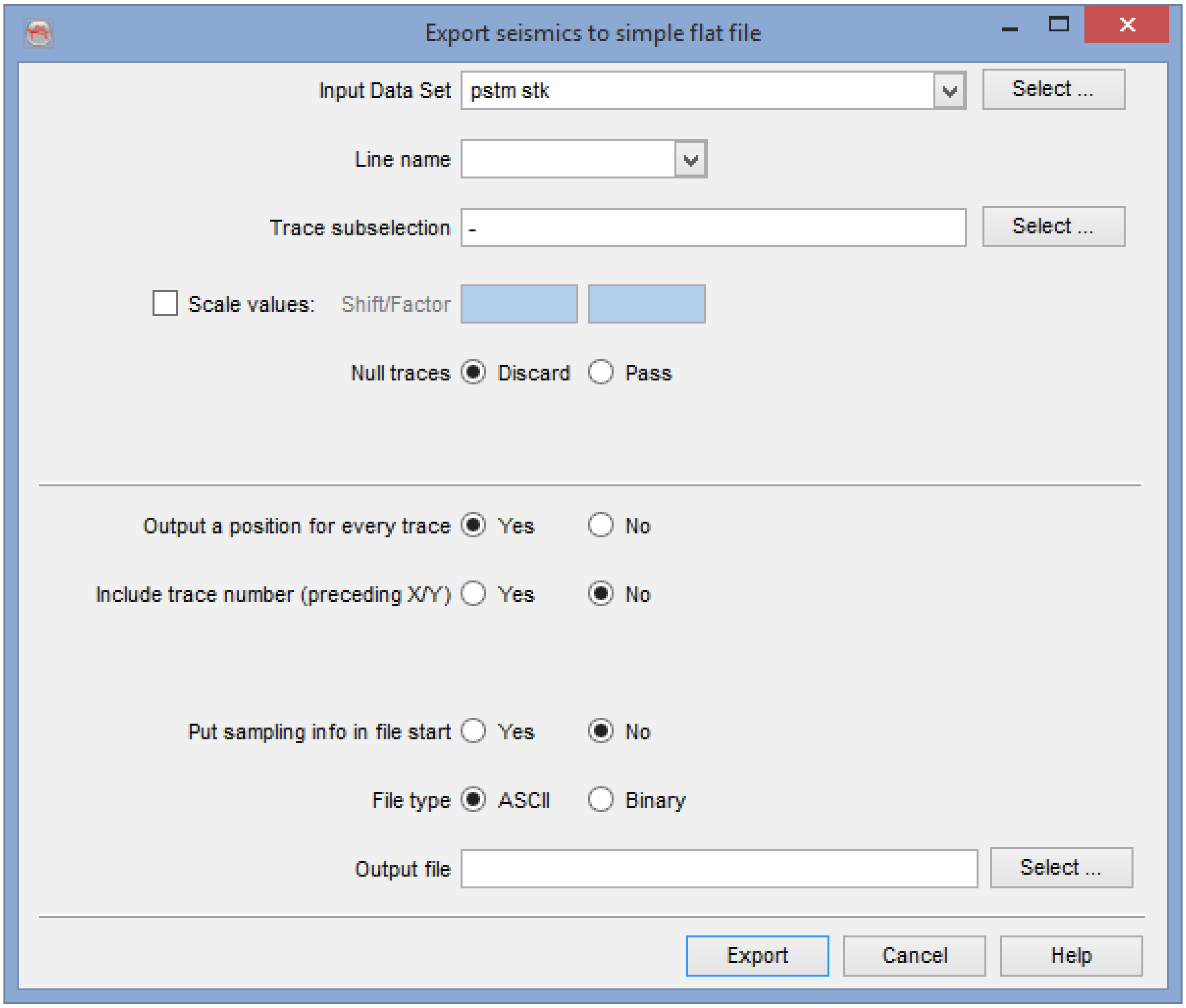
Multi-Component Export:

If the input cube is a stored multi-component data, an additional selection box will appear in the window. Select the desired component to be exported as an Ascii or Binary file.



Export 2D Simple File Format

A stored data set in the OpendTect survey can also be exported as a simple Ascii or binary file.



Simple 2D Seismic File Export Window

Input Data Set: Select the data set to be exported

Line name: The lines available in the data set are listed here. Only one line can be selected.

Trace Subselection: A part of the line can also be sub-selected. If the entire line is meant to be exported: after clicking on Select..., define the number and time ranges. Leave the values default or select All in the trace subselection window.

Scale values: The data can be scaled in the output file. The output will be calculated with: Output = Factor * Input + Shift

Null traces: Null traces can be discarded or left inside the line (Pass).

Output a position for every trace: If *Yes* then the output will contain a short header information. If *No*, no header will be added

(If *Yes* selected in the previous option) Include trace number (preceding X/Y): In the output file, it includes the trace numbers before X and Y locations.

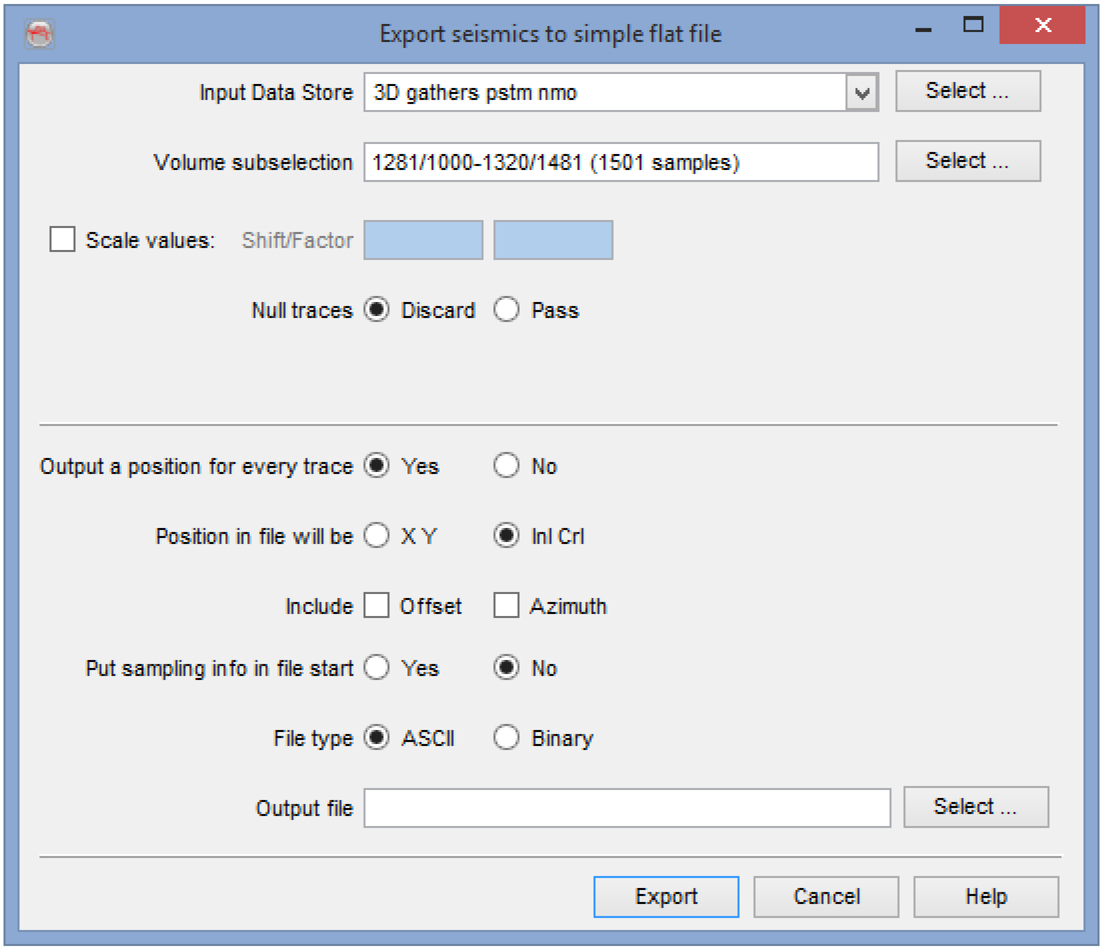
Put sampling info in file start: Select *Yes* to allow the sampling information to appear at the beginning of the file.

File type: Select the appropriate output file type: ASCII or Binary.

Output file: Select/write the output file location.

Export PreStack Simple File

In OpendTect, you can also export a simple seismic Ascii or Binary file from Pre-Stack 3D seismic data via *Survey > Export > Seismic > Simple File > Prestack 3D*.



Input Data Store: Select the Pre-Stack seismic to export.

Volume Subselection: A part of the line can also be sub-selected. If the entire line is meant to be exported: after clicking on *Select*..., define the number and time ranges. Leave the values default or select *All* in the trace subselection window.

Scale values: The data can be scaled in the output. The output will be calculated with: `Output = Factor * Input + Shift`

Null traces: Null traces can be discarded or left inside the line (Pass).

Output a position for every trace: If *Yes* then the output will contain a short header information. If *No*, no header will be added.

(If *Yes* selected in the previous option) Position in file will be: In the output file, the position information can either be XY locations or Inline/crossline numbers

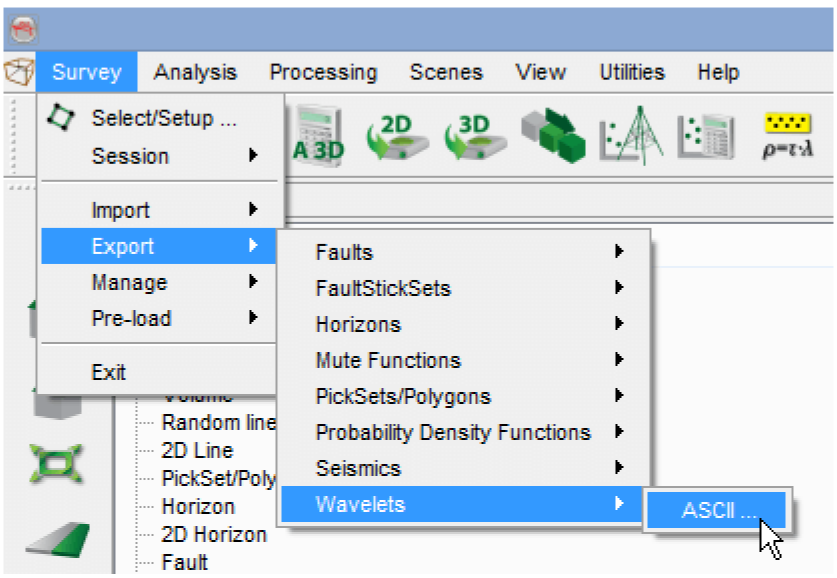
Include: Optionally you can select to include Offset and/or Azimuth information.

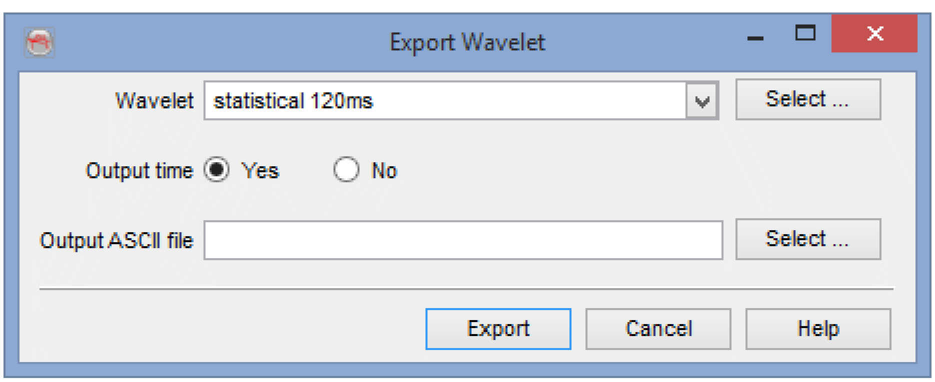
Put sampling info in file start: Select *Yes* to allow the sampling information to appear at the beginning of the file.

File type: Select the appropriate output file type: ASCII or Binary.

Output file: Select/write the output file location.

Export Wavelets





Wavelet Export Window

After selecting the wavelet to export, you can optionally also output the time (if time survey) or depth (if depth survey). Give an appropriate name and storage location, the wavelet will be exported when clicking on *Ok*.

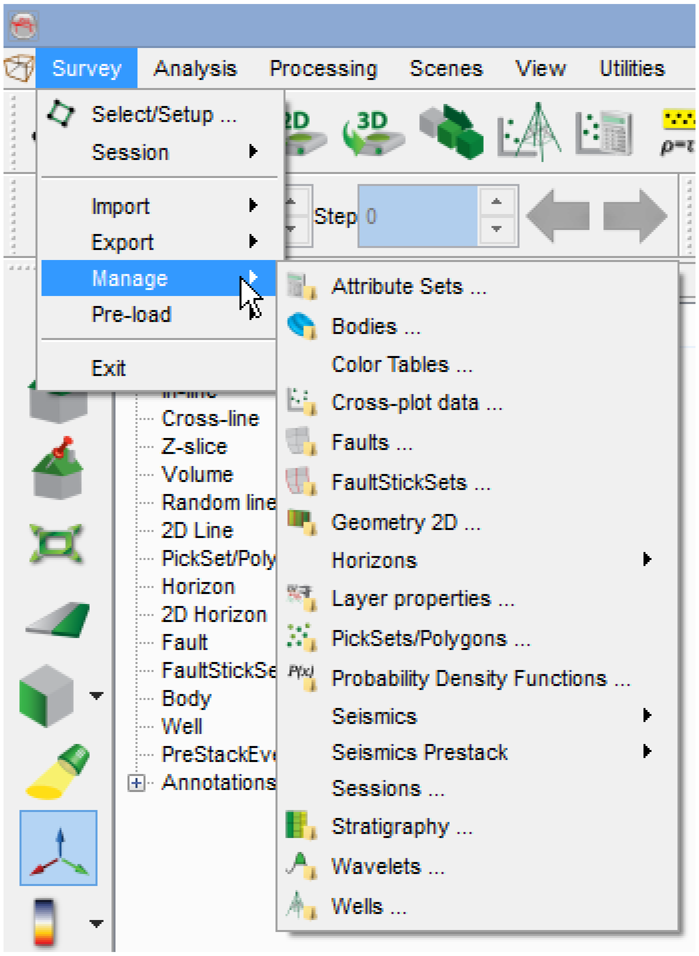
Manage

OpendTect keeps track of different files imported into or created by the system. Deleting, renaming and, in the case of seismic files, also merging of files are controlled from the *Survey--> Manage* menu. Seismic file, horizon, and well management is called from this menu.

The "Manage" option can be reached from two ways:

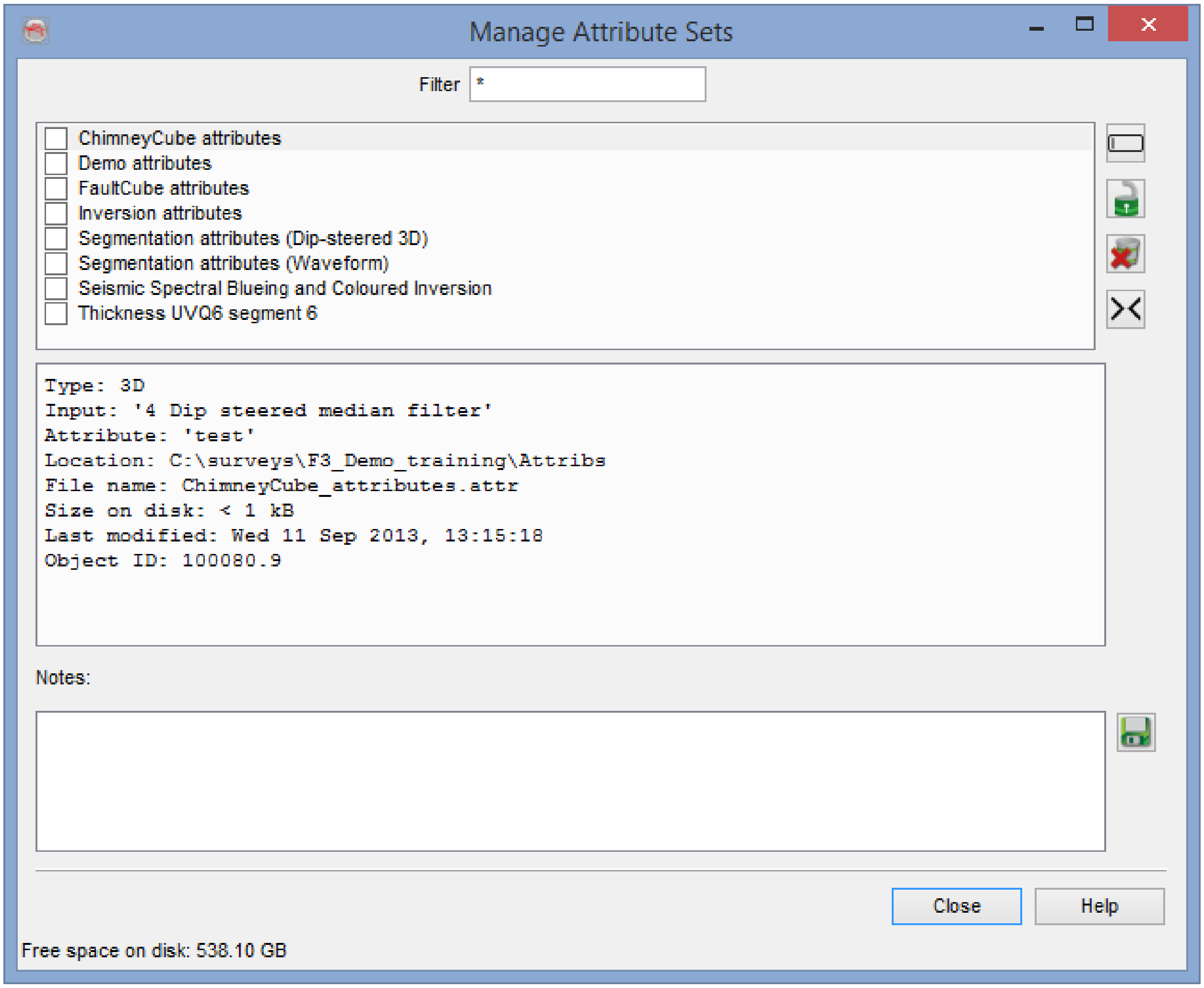
Directly from the main user interface:

or from *Survey > Manage...*

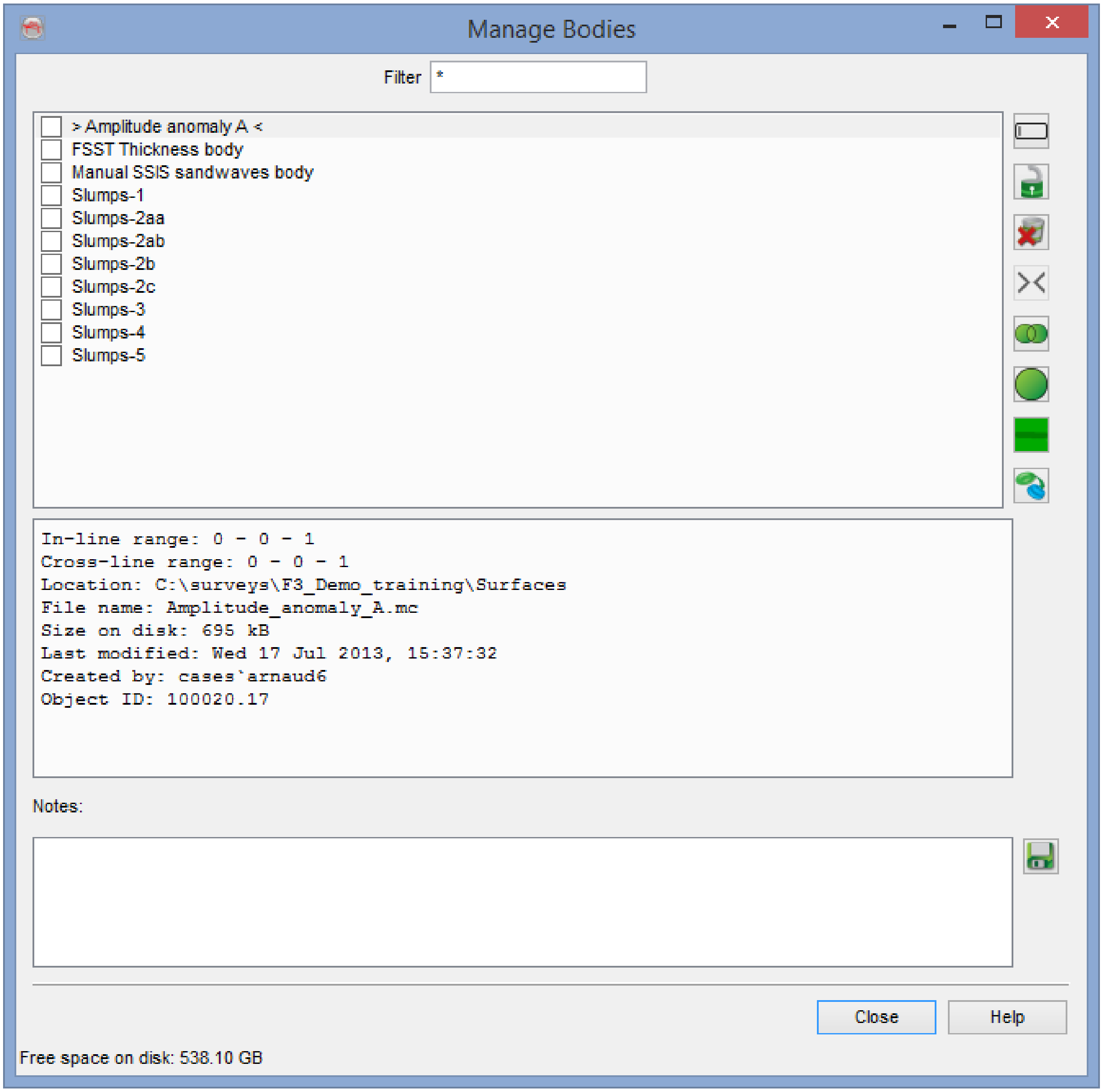


Manage Attribute Sets

The attribute set files can be managed from this window (See below). It is launched from *Survey* > *Manage* > *AttributeSets...* In this section, you can modify the attribute set name, set as default, remove etc. The window contains the buttons similar to the one from the [general selection window](#).


Manage Attribute Sets
Filter *
ChimneyCube attributes
Demo attributes
FaultCube attributes
Inversion attributes
Segmentation attributes (Dip-steered 3D)
Segmentation attributes (Waveform)
Seismic Spectral Blueing and Coloured Inversion
Thickness UVQ6 segment 6
Type: 3D
Input: '4 Dip steered median filter'
Attribute: 'test'
Location: C:\surveys\F3_Demo_training\Attribs
File name: ChimneyCube_attributes.attr
Size on disk: < 1 kB
Last modified: Wed 11 Sep 2013, 13:15:18
Object ID: 100080.9
Notes:
Close
Help
Free space on disk: 538.10 GB

Manage Bodies



Here you can rename, lock for editing and delete any *Body* by clicking on , and respectively. Plus, you could also make any *Body* a default body by clicking on . The remaining four icons are explained in the following separate sections:

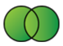
[Body Operator]

[Body Region Constructor]

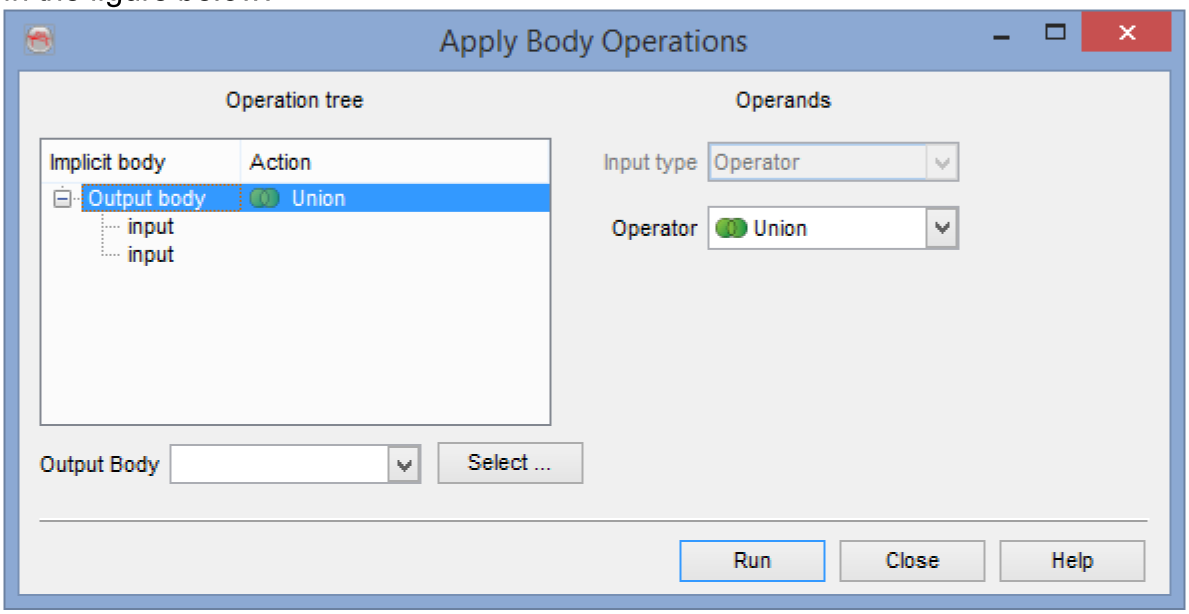
 [Estimate Body Volume]

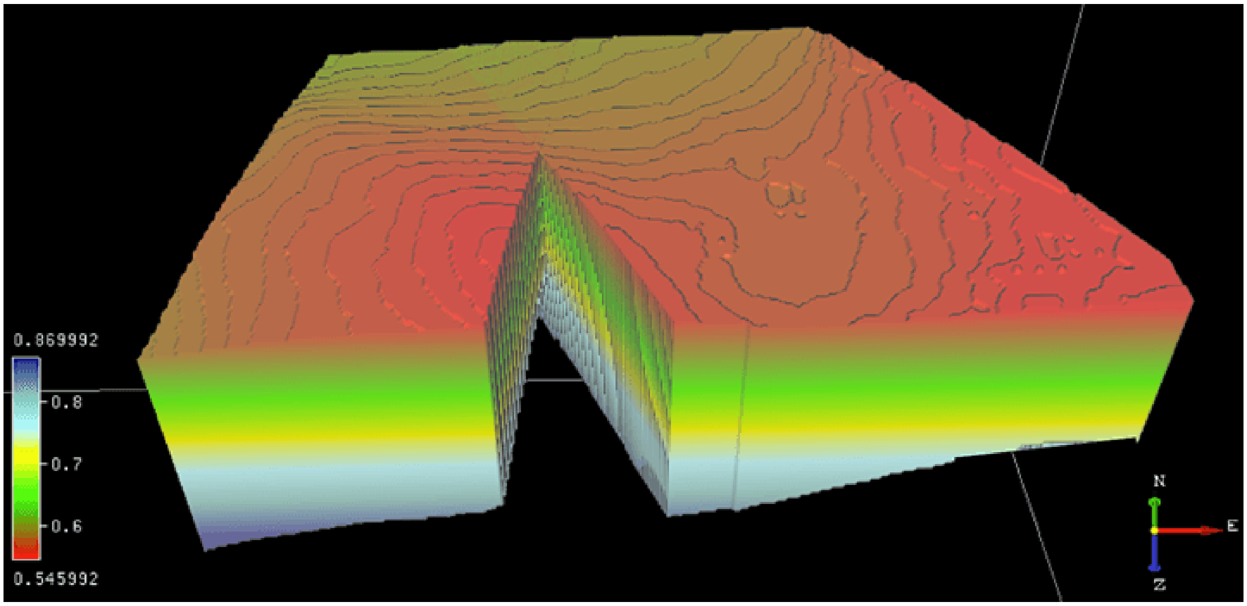
 [Switch Body Values]

Body Operator

The  icon enables the users to perform various operations on a geological body (or combinations thereof), as listed below.

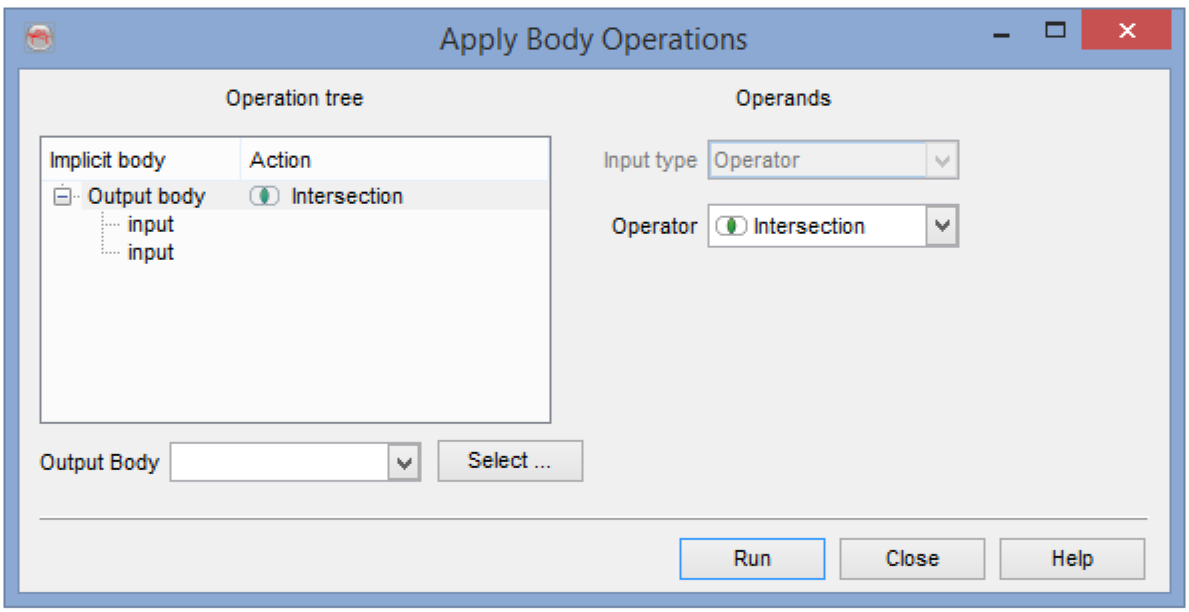
If you need to add to bodies, union operator enables you to merge two or more bodies as in the figure below:

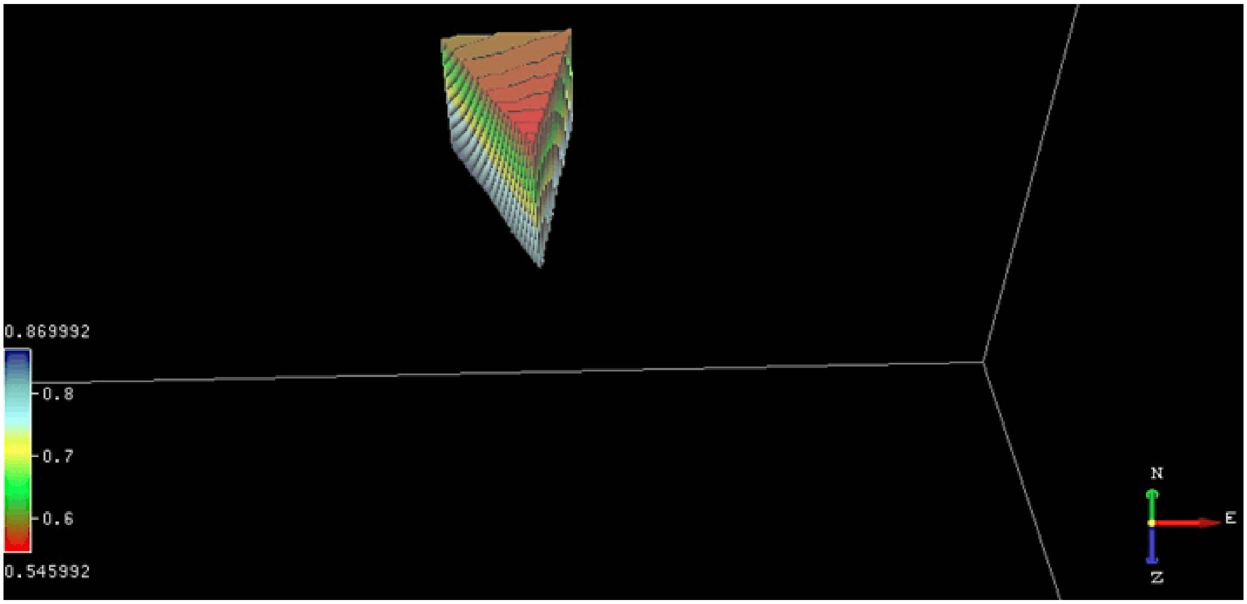




The result of the 'Union' operation (the two bodies are merged).

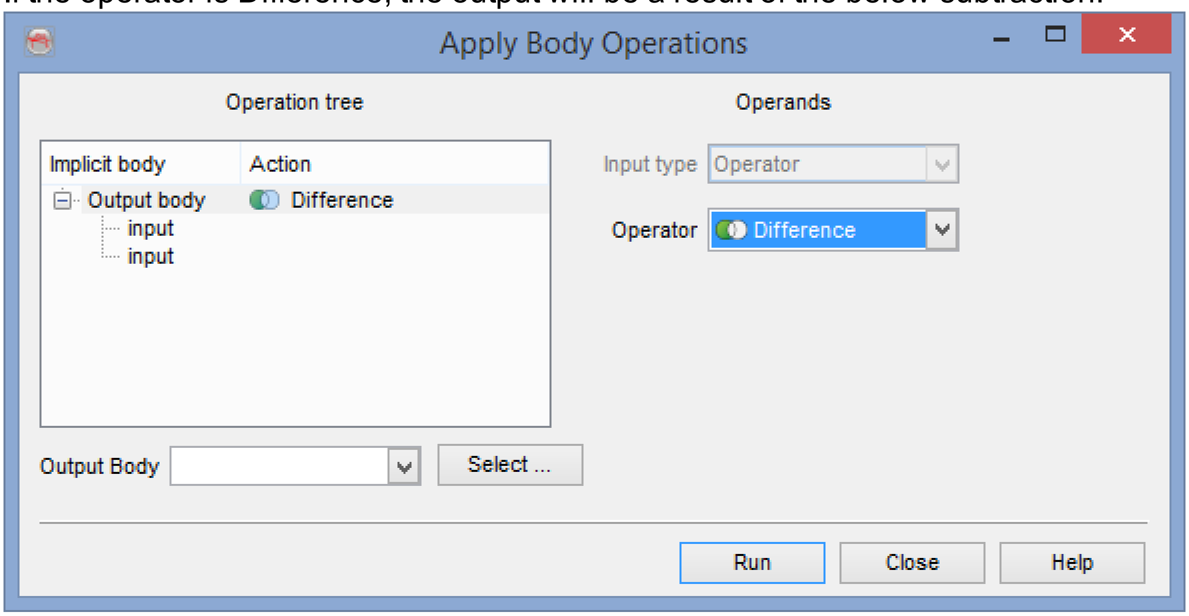
If you want to know the shared portion of two bodies, you can use intersection as an operator:

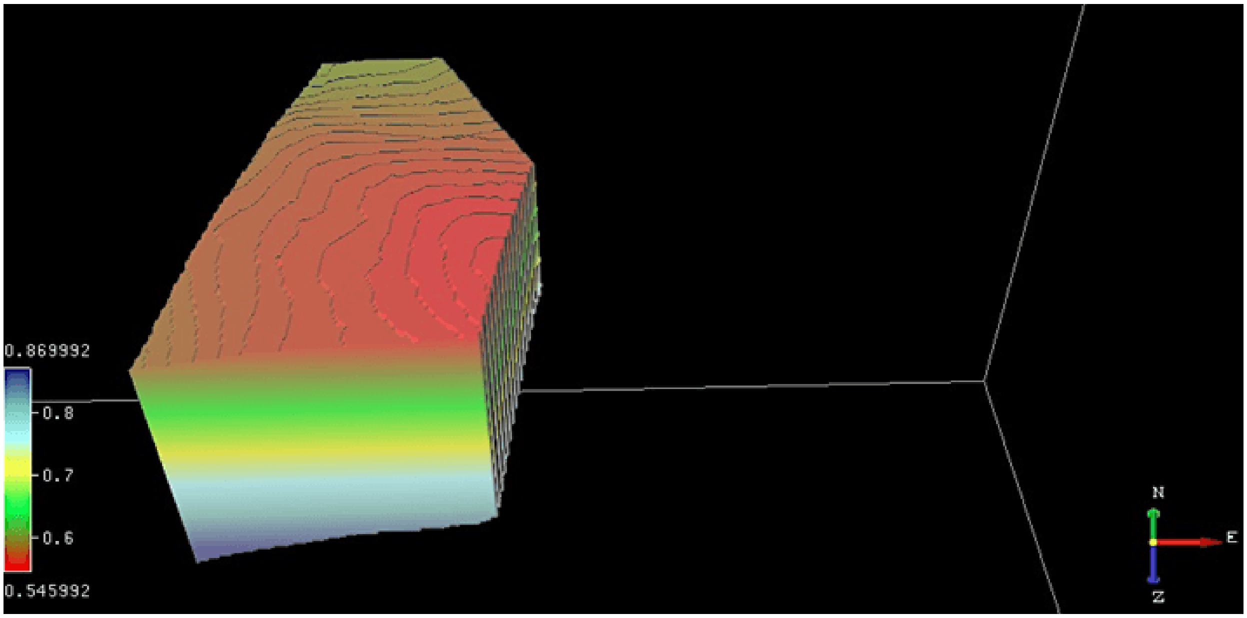




The result of the 'Intersection' operation.

If the operator is Difference, the output will be a result of the below subtraction:

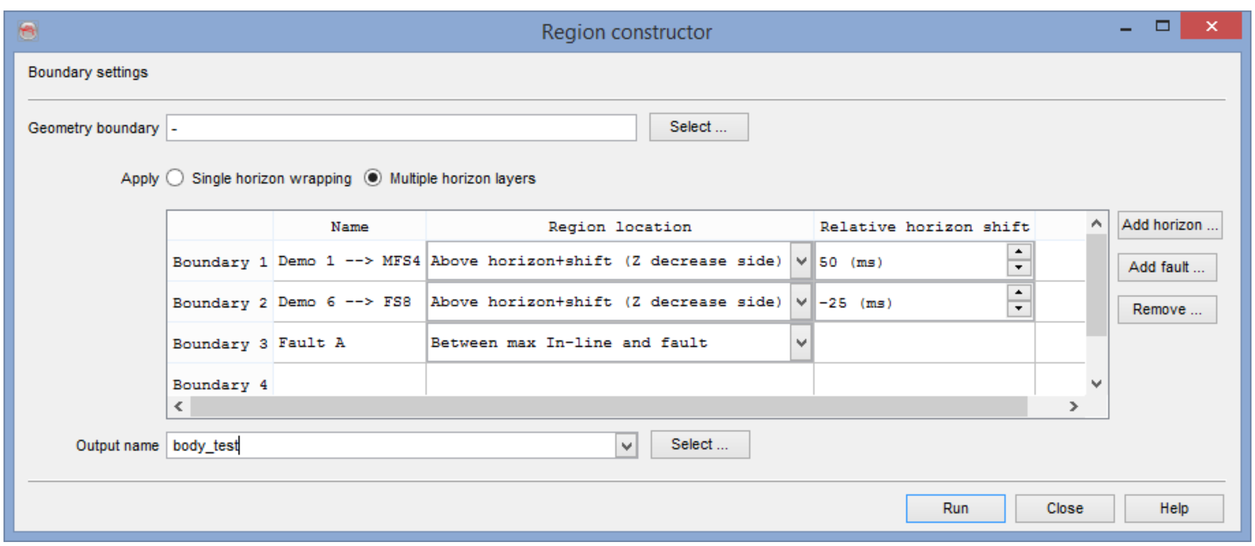




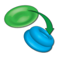
The result of the 'Difference' operation (the final body is the volume resulting from Body 2 minus Body 1).

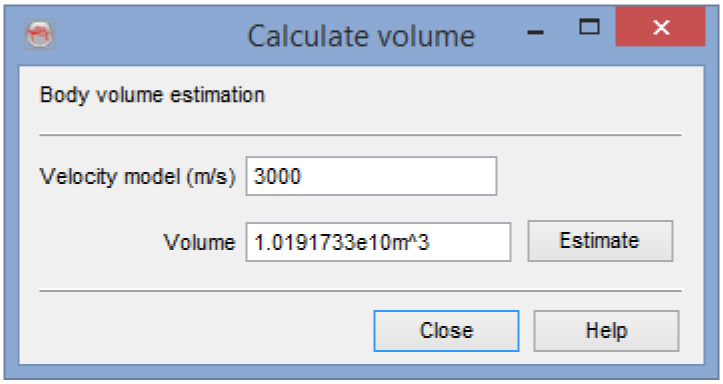
Body Region Constructor

This  tool allows you to create a regional body from your dataset using boundaries defined horizons and faults (or just a single wrapping horizon).



Estimate Body Volume

The  icon allows the user to calculate the volume of any body in the scene. The input data is an (estimated) velocity appropriate to the position of the body.

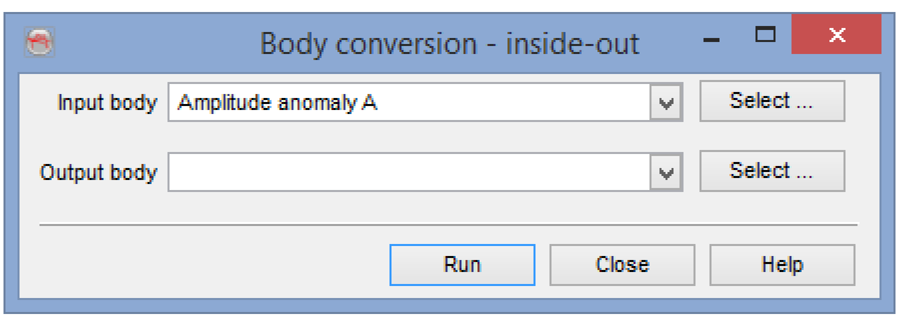


Switch Body Values

The  icon activates a window with dual functionality.

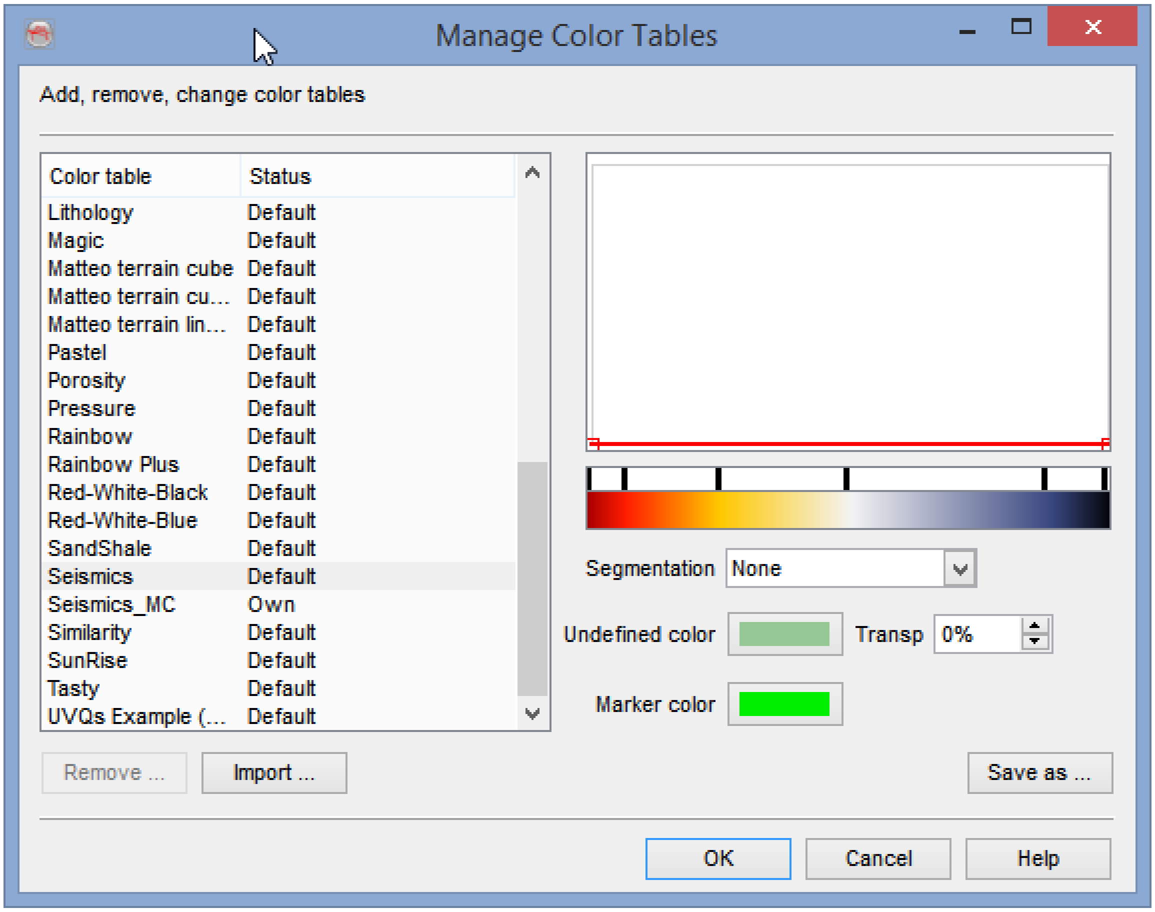
Firstly, if you have constructed a regional body using two horizons and a fault, creating a kind of 'compartment', this tool allows you to 'flip' the body to the 'negative' of the one constructed.

Secondly, previous versions of OpendTect had bodies with many different formats. This tools can also be used to 'convert' these various formats into the standard format for 4.6.



Manage Color Tables

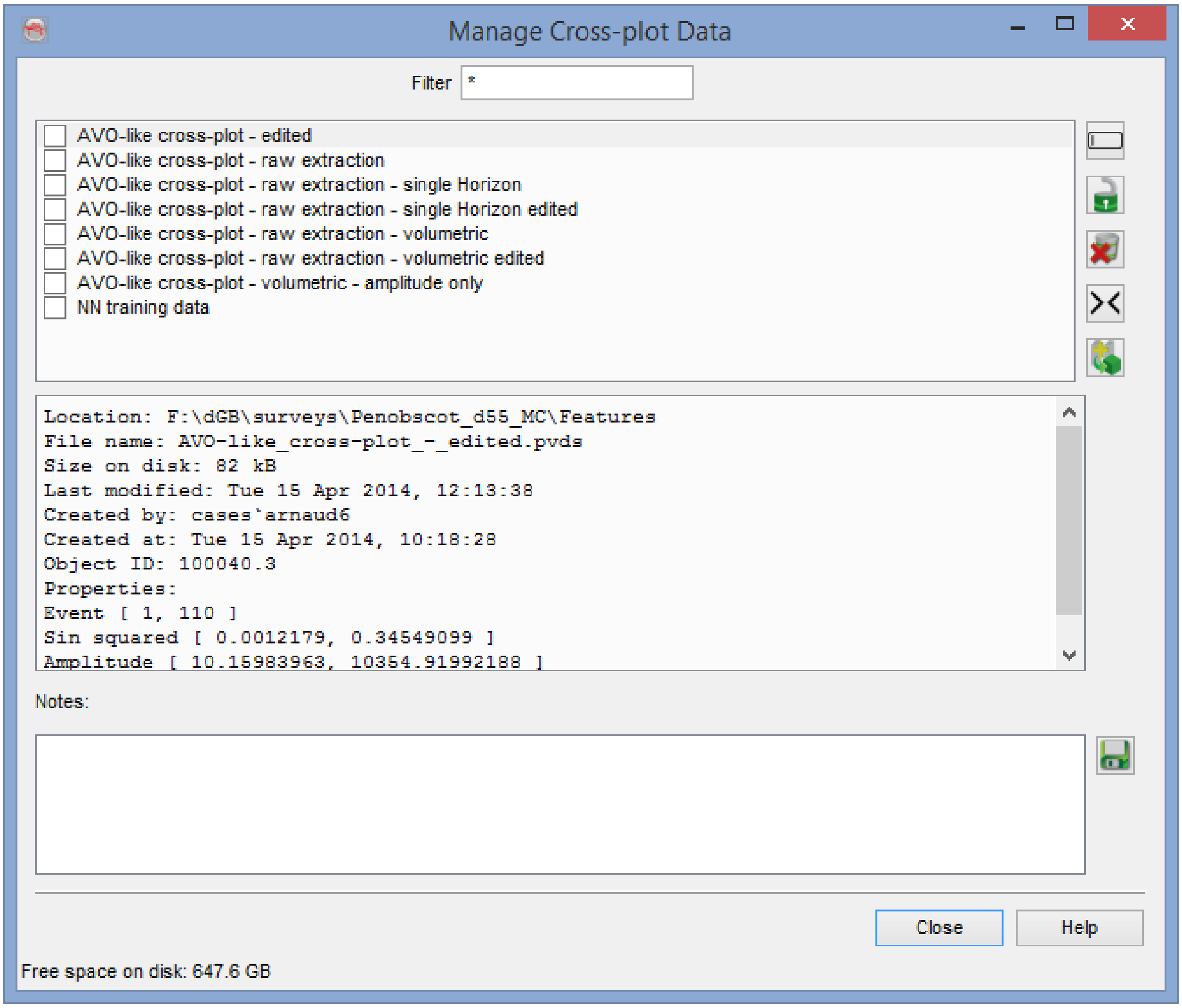
The Color Table Manager allows the user to edit various settings that are used within the color tables when visualizing seismic data, logs, etc.

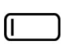


For a full description of the various options and possibilities, please see the following section: Color Tables

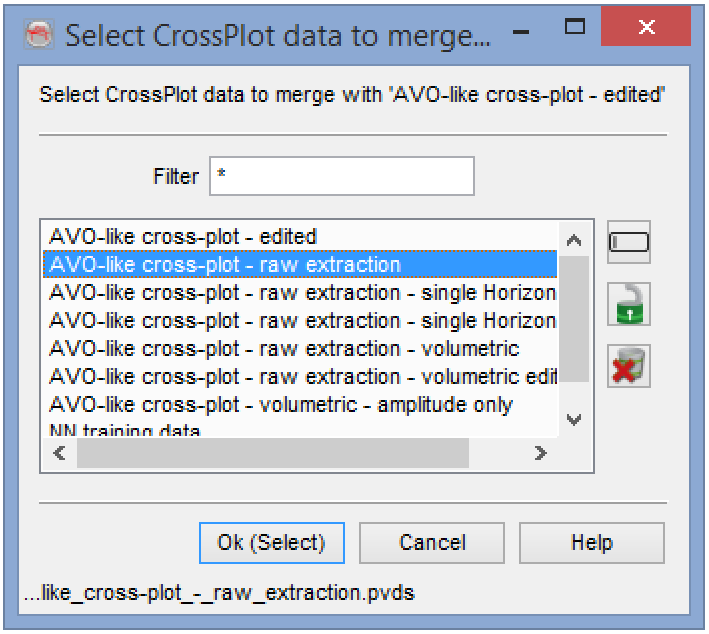
Manage Cross-Plot Data

The crossplot file management is used to rename, remove, merge etc the stored cross-plot data files.

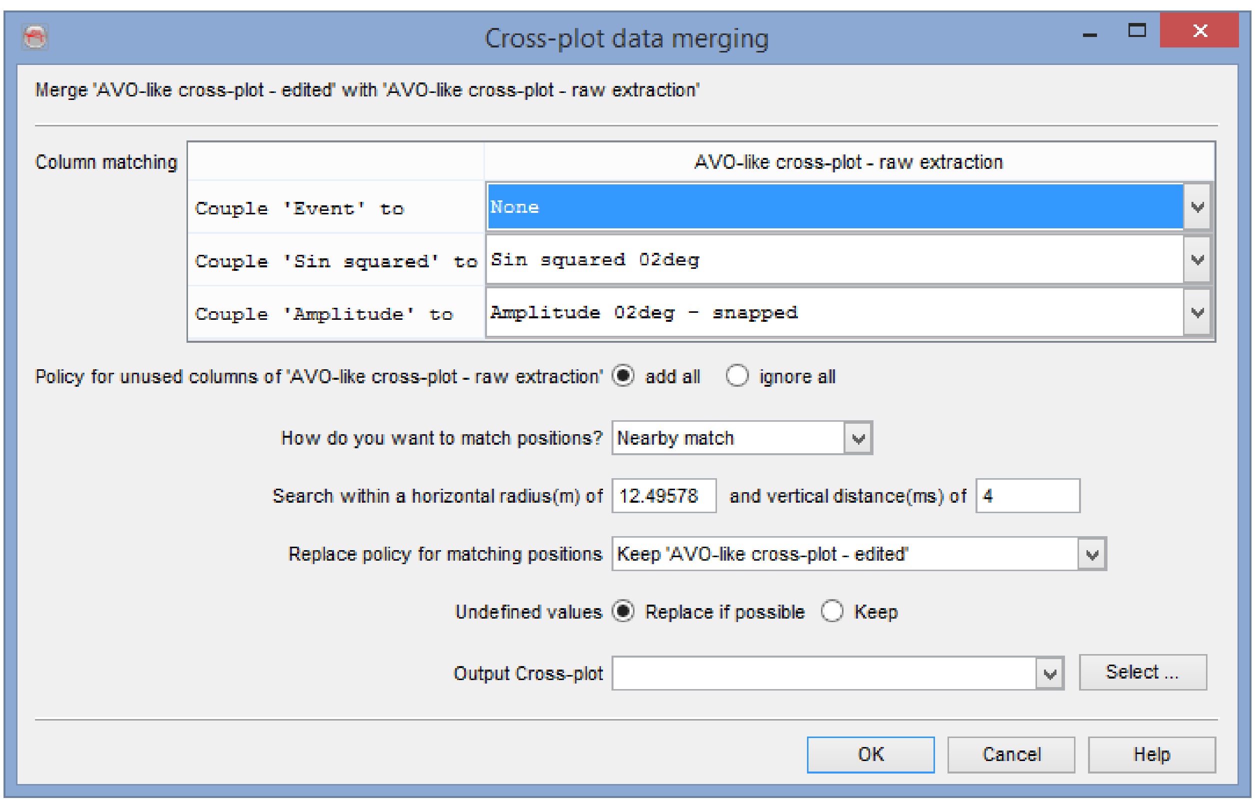


-  is used to rename the selected crossplot data file.
-  locks (read only) or unlocks the selected item.
-  remove the selected crossplot.
-  make the selected crossplot as a default data for crossplotting. If pressed, the selected item will be bounded by the signs (> Name <).
-  merge another crossplot with the selected one.

To merge two crossplots with different or similar attributes, you need to select a first crossplot and then after clicking on the  icon, choose the second crossplot in the following window:



. When pressing *Ok*, opens the window where you have to provide different information. If some columns have similar quantity in the two crossplots, it is possible to specify for each column, which column from the crossplot2 matches a column from crossplot1, even if they do not have the same name. If there is no match, just select None, then you can decide to either add the unmatched columns or to ignore them.



Then select the matching method:

- Exact match: same X,Y,Z
- Nearby match: almost same position or same depth.A horizontal and a vertical search radius have to be provided
- Never match, add all new : add all positions, even with same or almost same (X,Y,Z).

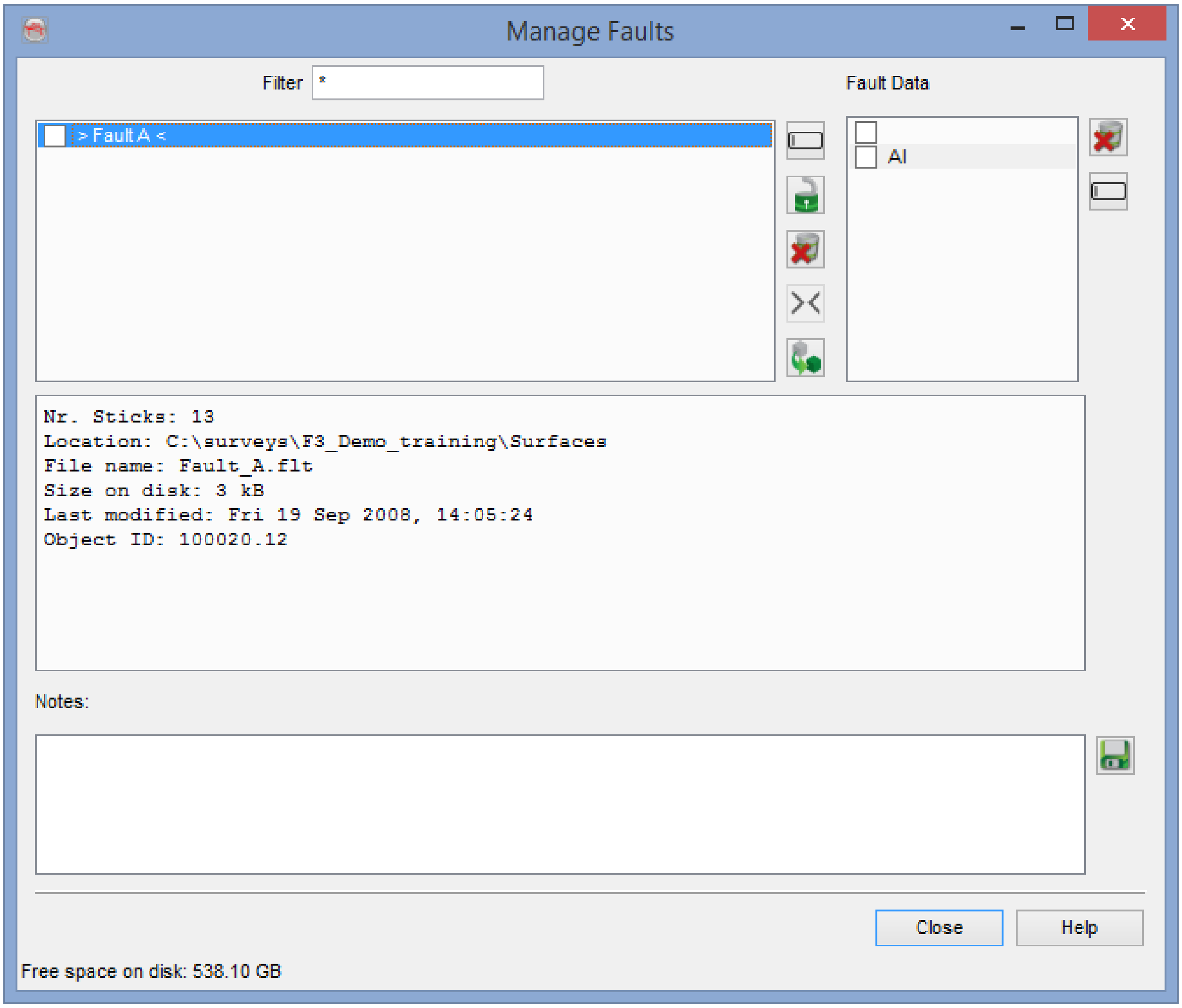
For all of them, the *replacement policy* for matching positions has to be specified : take value from crossplot1, take value from crossplot2 or take the average of the two. Undefined value can be kept or replace if possible.

Once the merging parameters have been defined, you can give an appropriate name and save it in clicking on Ok.

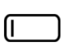
Manage Faults

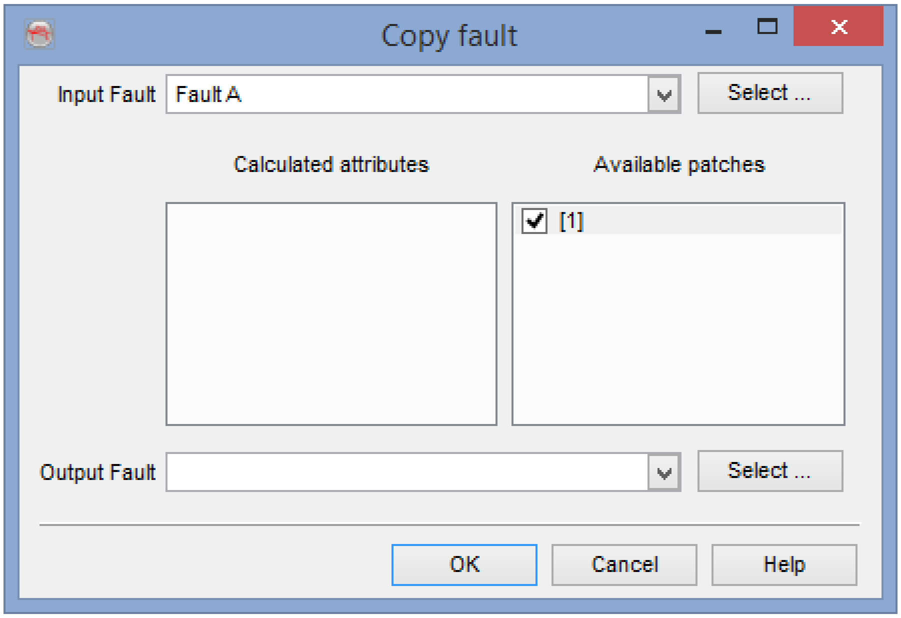
Fault management allows to manage the faults, different options are available: change disk location, rename, remove, copy etc. The fault management window can be launched either by pressing the icon from manage toolbar, or from menu bar i.e. *Survey > Manage> Faults...*

This window will contain the list of faults interpreted and saved in the OpendTect project as shown below.



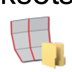
Select the desired fault, then use the various tools:

-  is used to rename the selected fault.
-  locks the selected fault from interpretation.
-  is used to trash (delete) the selected fault.
-  is used to set a fault as default.
-  copies the faults.

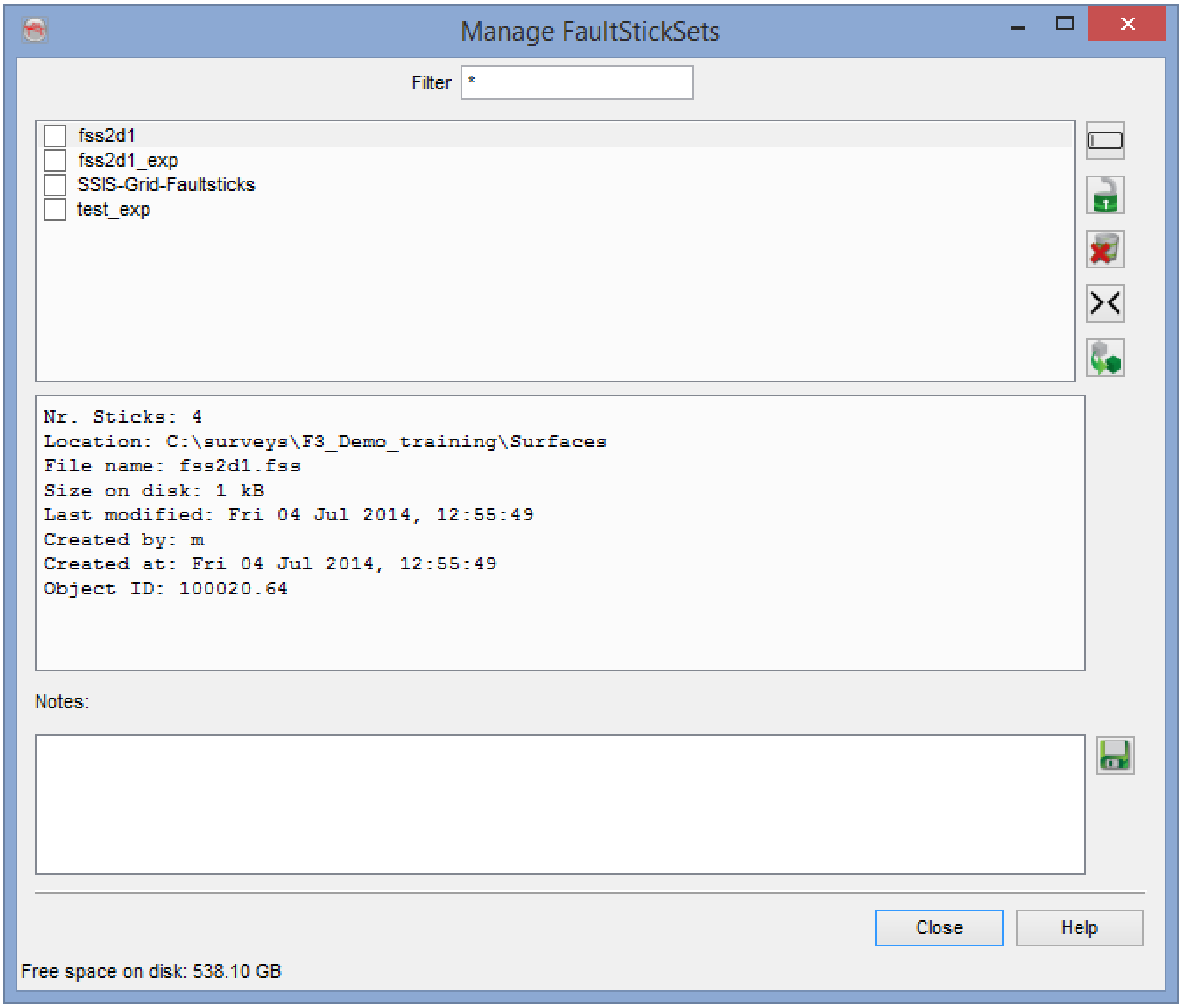


Note: The top filter is used to filter-out the objects with selected names. For instance, to display all faults that start with letter F use "F*".

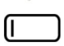
Manage FaultStickSets

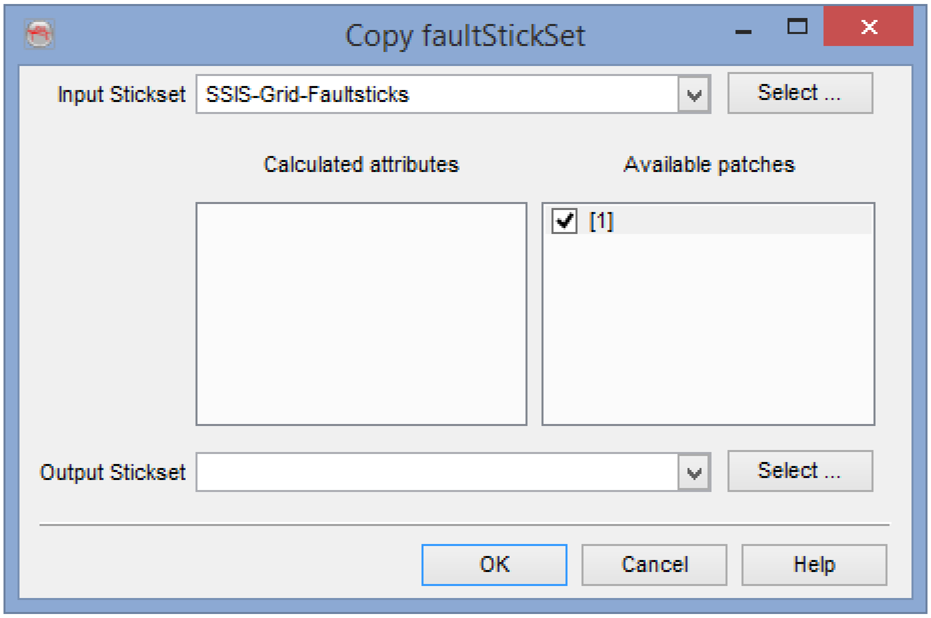
This option allows the management of faultsticksets. The faultstickset management window can be launched either by pressing the  icon from manage toolbar, or from menu bar i.e. *Survey > Manage> FaultStickSet...*

This window will contain the list of faultstickset interpreted and saved in the OpendTect project as shown below.



Select the desired fault, then use the various tools:

-  is used to change the current disk location of selected faultstickset .
-  is used to rename the selected faultstickset.
-  locks the selected faultstickset from interpretation.
-  is used to trash (delete) the selected faultstickset.
-  is used to set a faultstickset as default.
-  copies the faultsticksets.



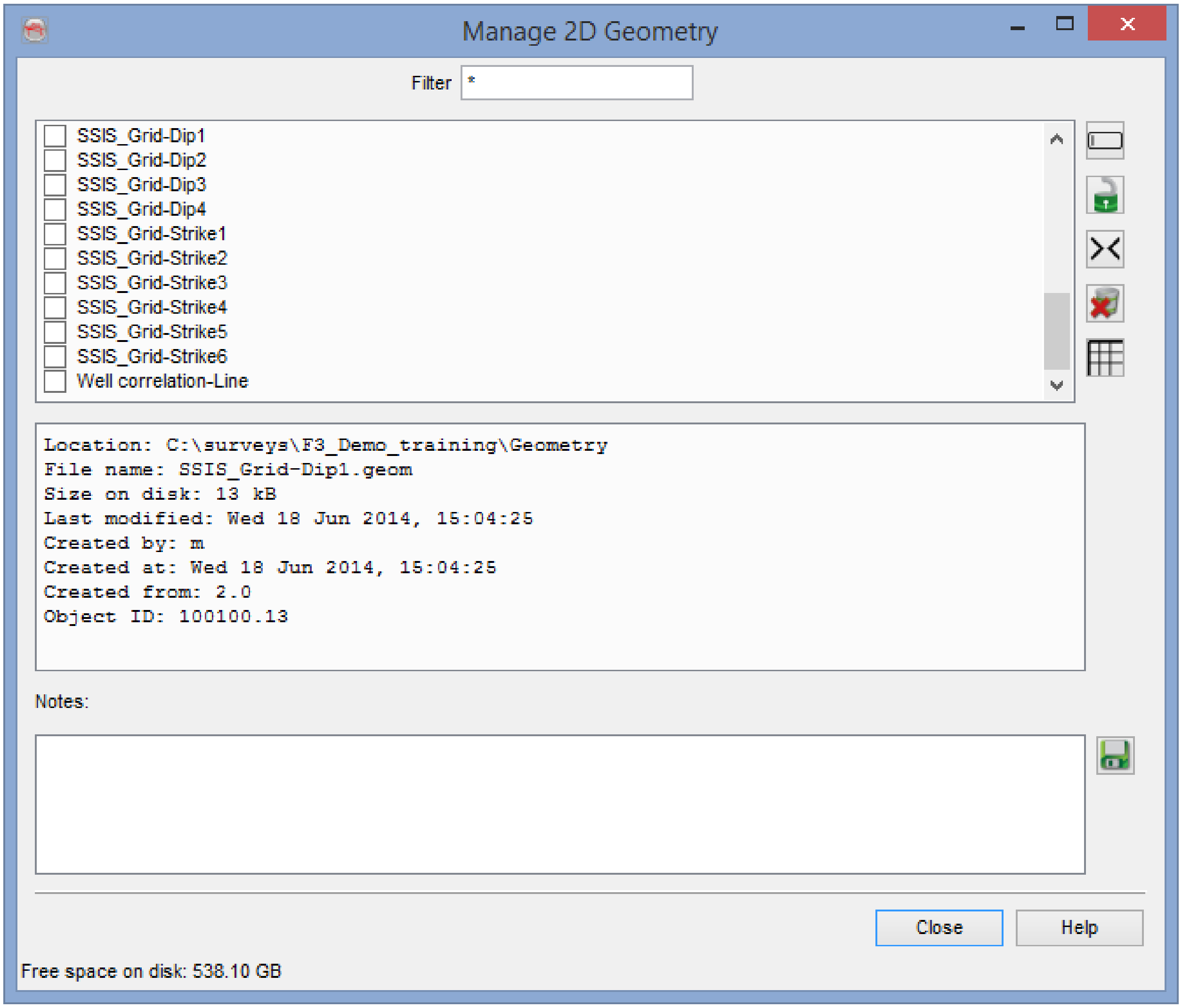
Note: The top filter is used to filter-out the objects with selected names. For instance, to display all faultsticksets that start with letter F use "F*".

Manage Geometry 2D

2D Geometry is launched from *Survey* > *Manage* > *2D Geometry...*

This window is used to manipulate the geometry of 2D seismic lines. The geometry consists of X-Y coordinate pairs for each trace of the 2D seismic, identified with a unique trace number (CDP most often). They are generally extracted from the SEG-Y trace headers or from an auxiliary file during import.

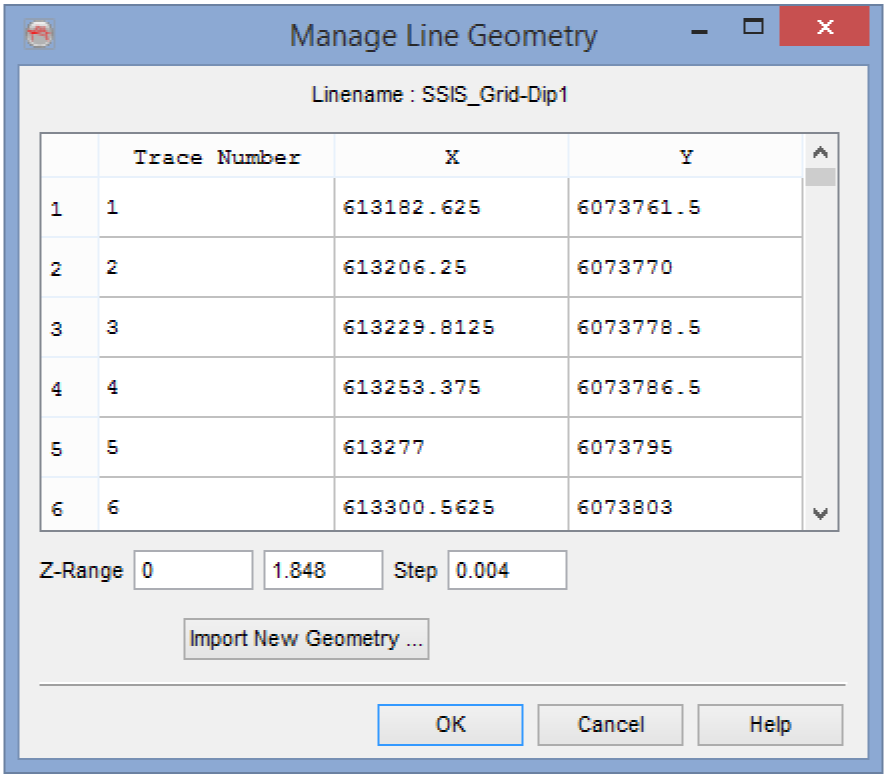
This window can be used to alter the coordinates of already imported 2D data. The geometry is separated from the actual 2D seismic data and 2D horizon that are solely referenced with respect to the trace number (CDP number). As a result the coordinates of the geometry can safely be edited without having to re-import the 2D seismic data and corresponding horizons.



Select a line and click on the  icon to start the line geometry editor.

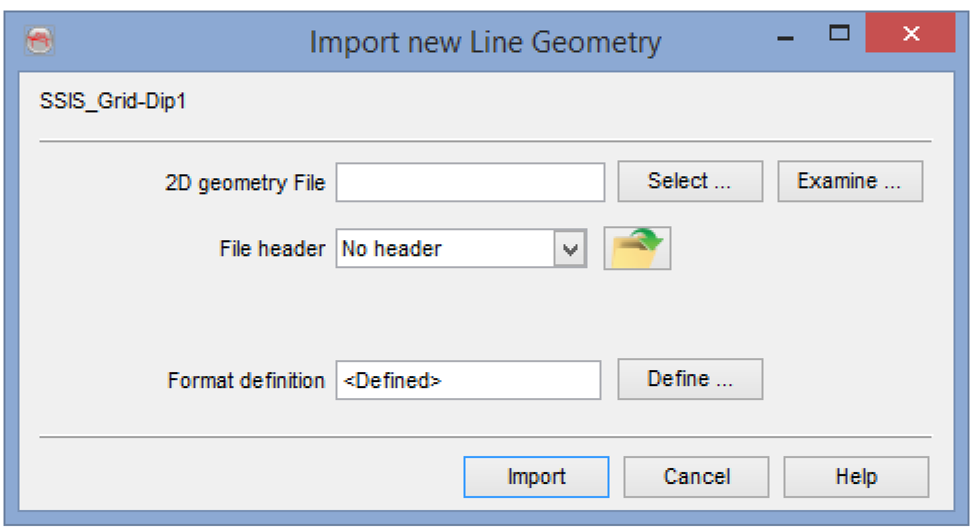
Line geometry editor

This editing window can be used to alter any trace number, X, Y values. Click on a field and type the new value. Changes will be saved on disk only after pressing *Ok*. Optionally one might want to update the geometry of the entire line by reading an text file (see below).



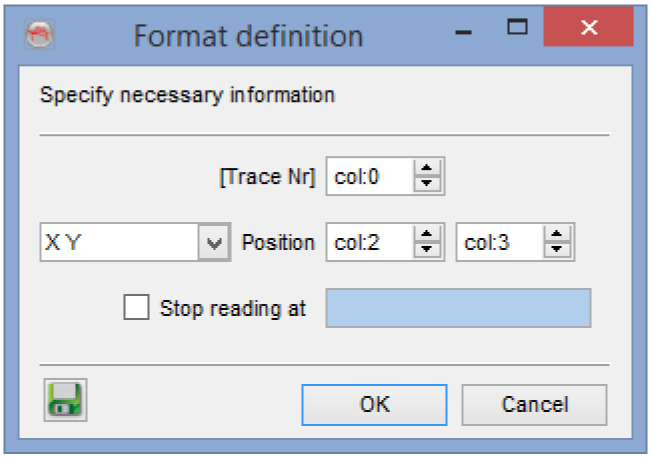
Line geometry import

Select the input Ascii file. To display the input file, press the *Examine* button. The input file should be column sorted with one point per record (line).

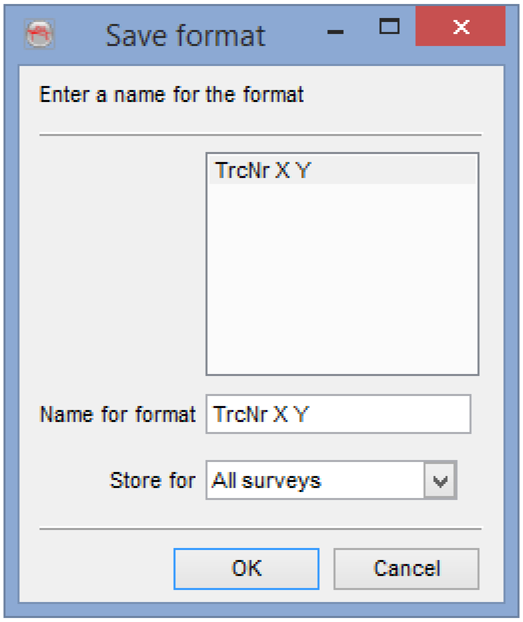


The presence of a *file header* and the file *format definition* have to be specified. The header, if present, can be of fixed length (number of lines), or delimited on its last line by a keyword.

Predefined and saved file formats are available by pressing on the  icon. Otherwise the format must be manually specified. The *Define* button gives access to the format definition window.

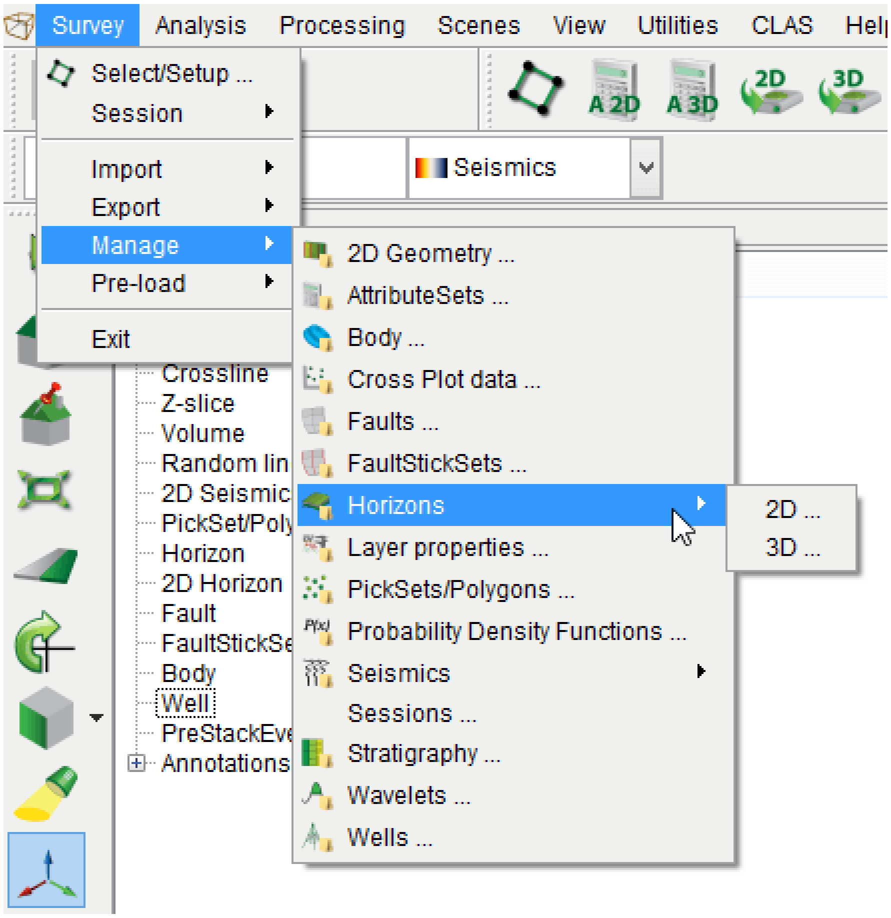


You must specify in the format definition window the column numbers for the position, in terms of an X/Y pair, and optionally the trace number column. The coordinate units must be in the same units as the coordinates of the survey corner points. Inline/crossline value may be used instead of X/Y but it is not recommended because of the grid spacing. Also it is not recommended to alter (re-specify) the trace numbers since it may corrupt the already loaded data. Reading may be stopped at a specific line by providing the adequate keyword.
It is recommended to save the format definition for a later use and QC, by clicking on the  icon . In pop-up window, write the name of the format and store it. The format can be stored at different levels (all surveys, this survey, Current OpendTect user level, levels in increasing priority order) depending on the usage. Press OK when done.



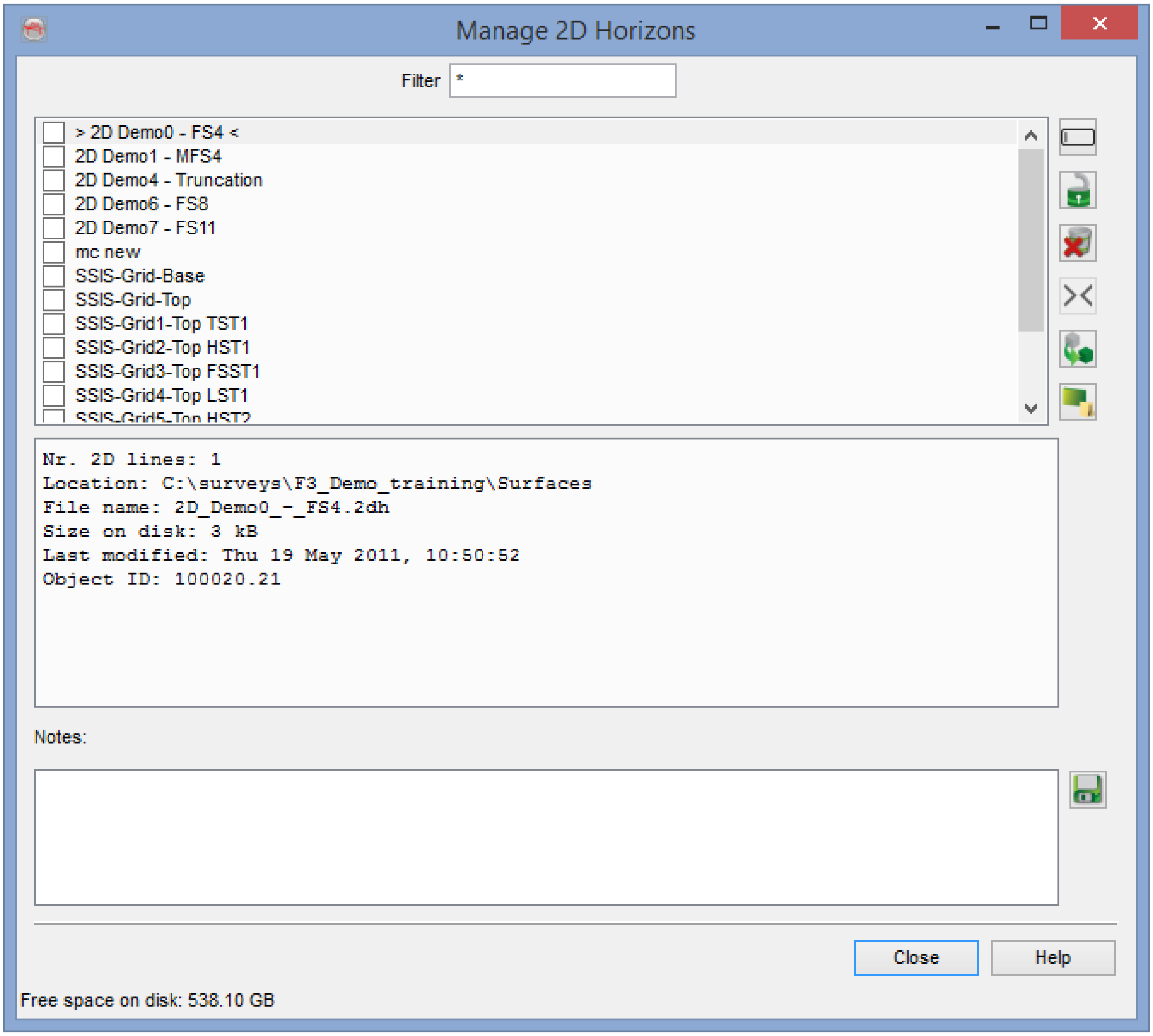
Manage Horizons

Manage either 2D or 3D horizons either via *Survey* > *Manage* > *Horizons*... or via the  icon.



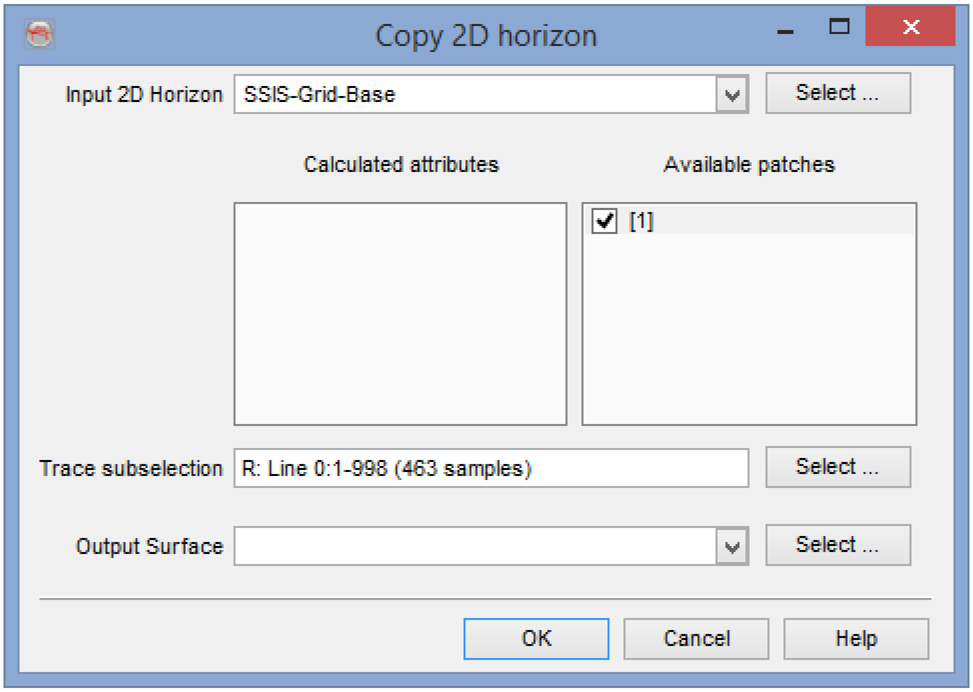
Horizon Manager 2D

To open the *Manage 2D Horizons* window, navigate through *Survey--> Manage--> Horizons--> 2D...* or use the  icon from the Manage toolbar. In the left panel of the window, the available horizons are displayed. It the bottom panel, information on the selected horizon is displayed (eg. location on disk, date last modified). At the base of the window the available disk space is noted.



Horizons can be *renamed*  , *locked* , *removed* , *copied* , *set as default*  or *viewed as a dataset group* .

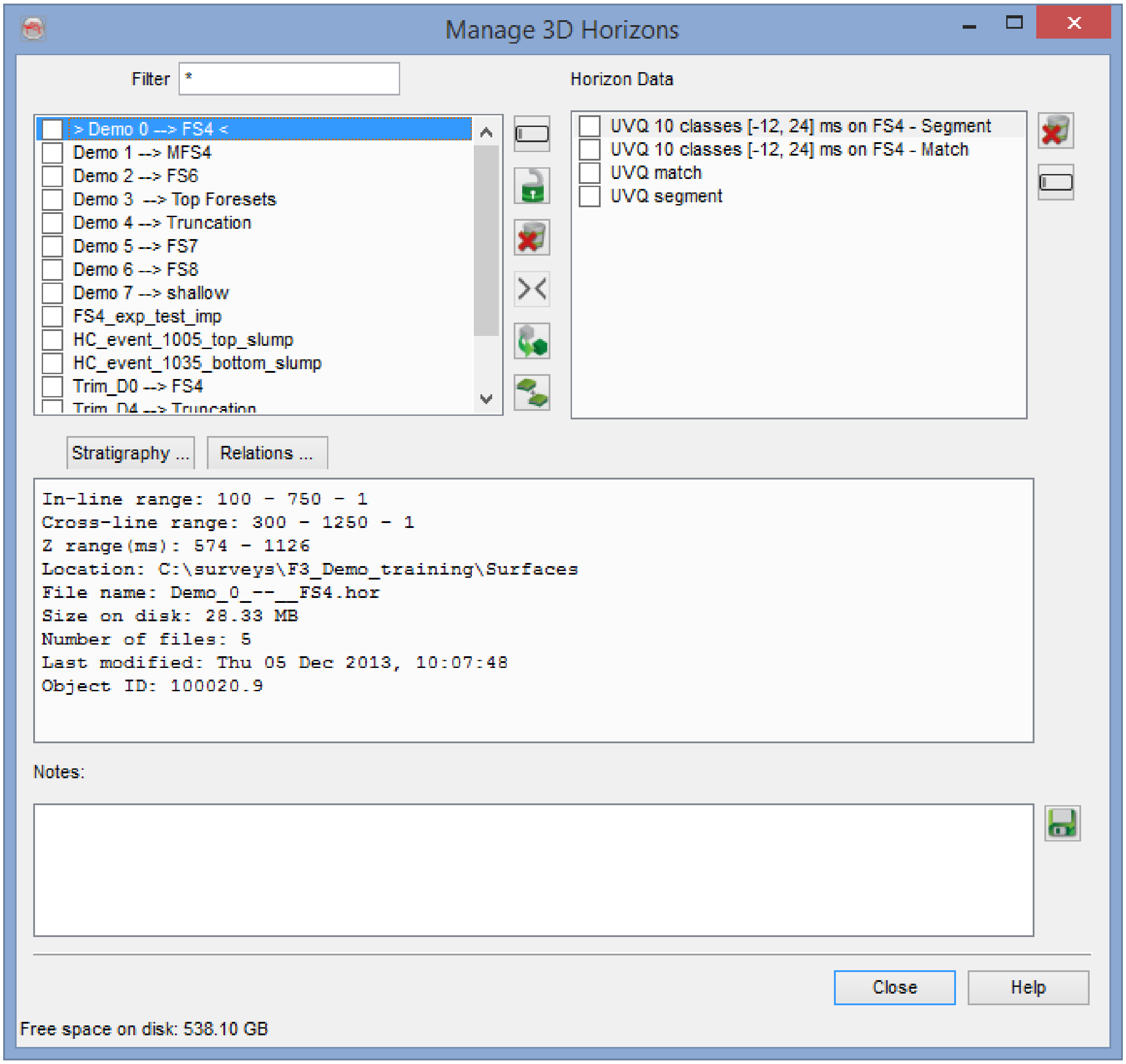
The following window is used to copy horizon surfaces and grids:



Note: The top filter is used to filter-out the objects with selected names. For instance, to display all horizons that start with letter D use "D*".

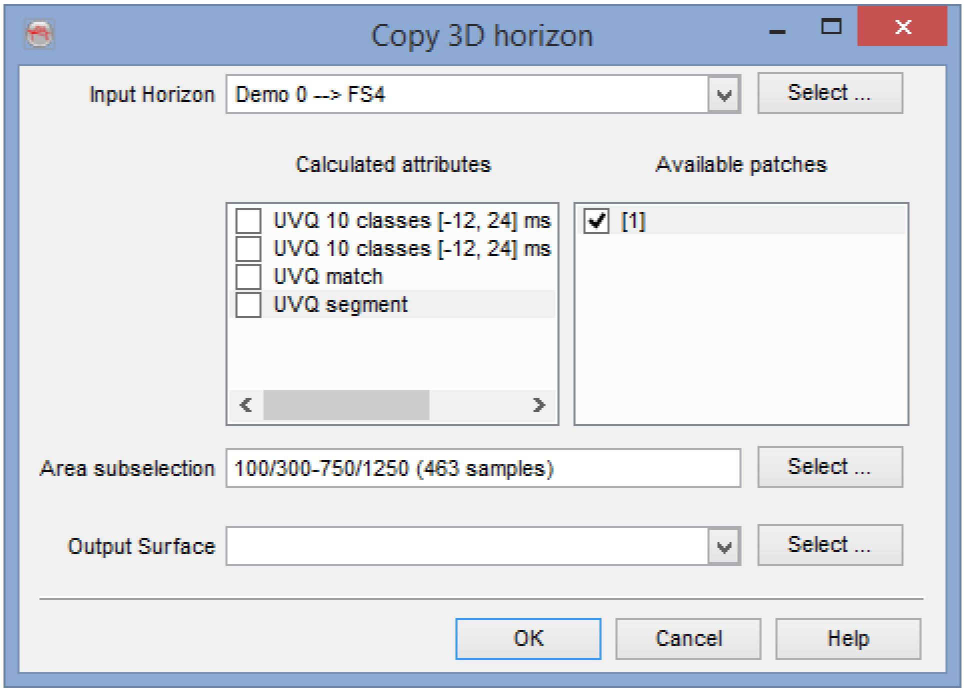
Horizon Manager 3D

To open the *Manage 3D Horizons* window, choose the  icon from the *Manage* toolbar. In the left panel of the window, the available horizons are displayed. In the right panel, the available attributes linked to the selected horizon will be displayed. In the bottom panel, information on the selected horizon is displayed (eg. location on disk, date last modified). At the base of the window the available disk space is mentioned.

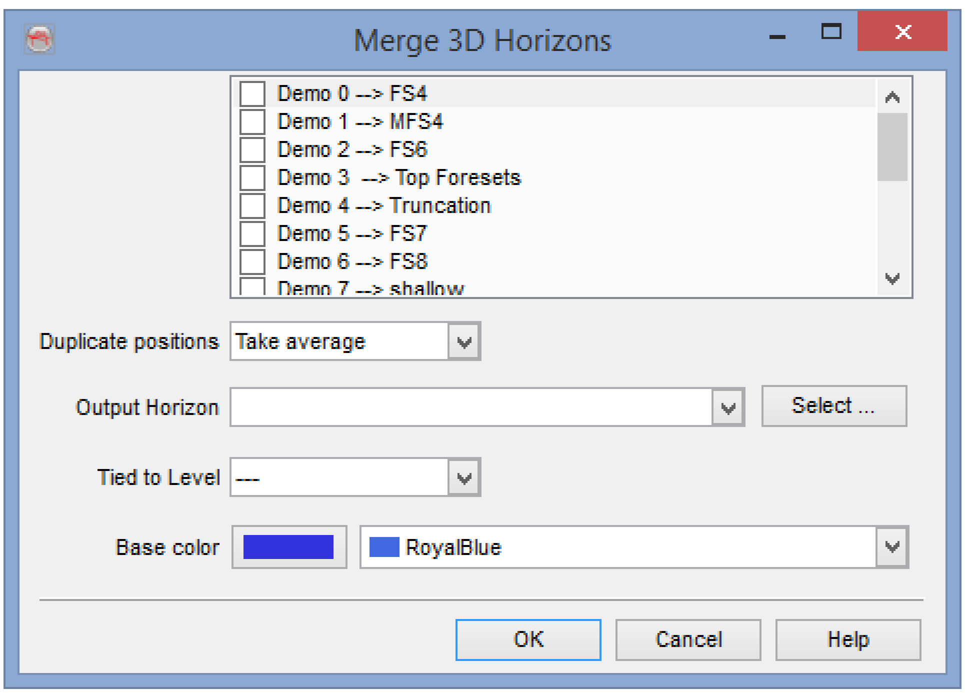


Horizons can be *renamed*  , *locked* , *removed* , *copied* , *set as default*  or *merged* . The top filter is used to filter-out the objects with selected names. For example, to display all horizons that start with letter D use "D*".

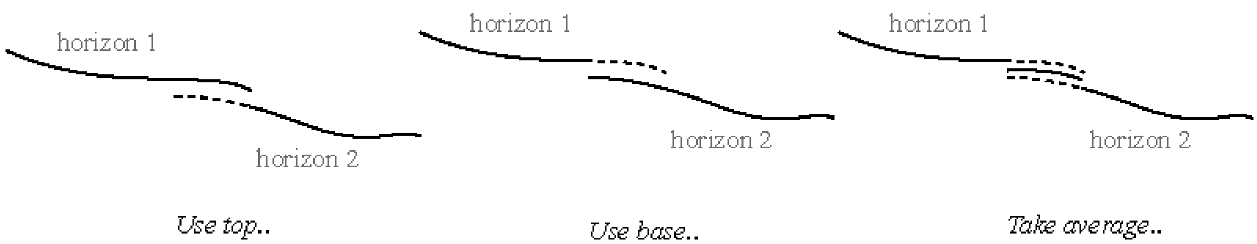
The following window is used to copy surface data and grids:



The following window is used to merge surface parts:



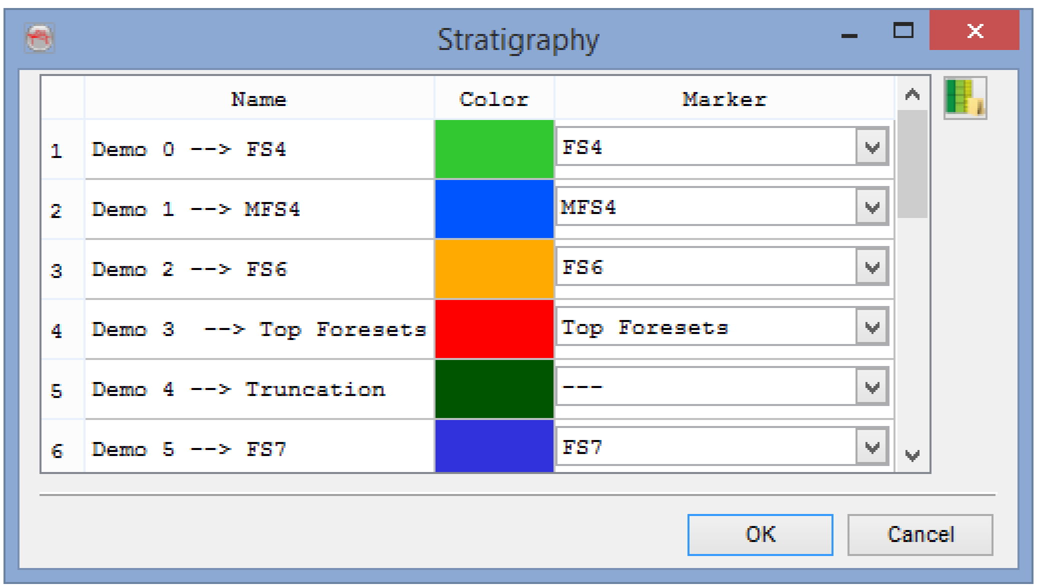
Select the horizon parts to be merged and select a mode for duplicate positions. Options are take average, use top or use base. The duplicate positions will then be handled in the following manner (dashed line portion represents removed data after merge):



The new, merged horizon can optionally be tied to a level, i.e. a regional marker (see below)

Stratigraphy

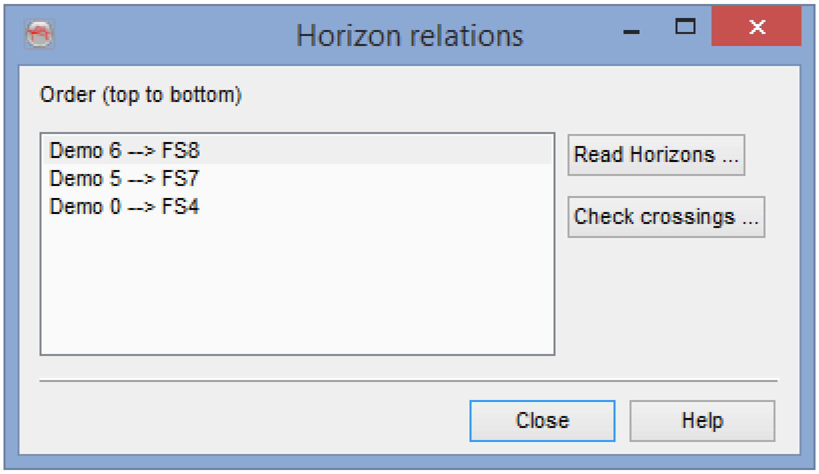
Assign stratigraphic markers to horizons with the Stratigraphy button.



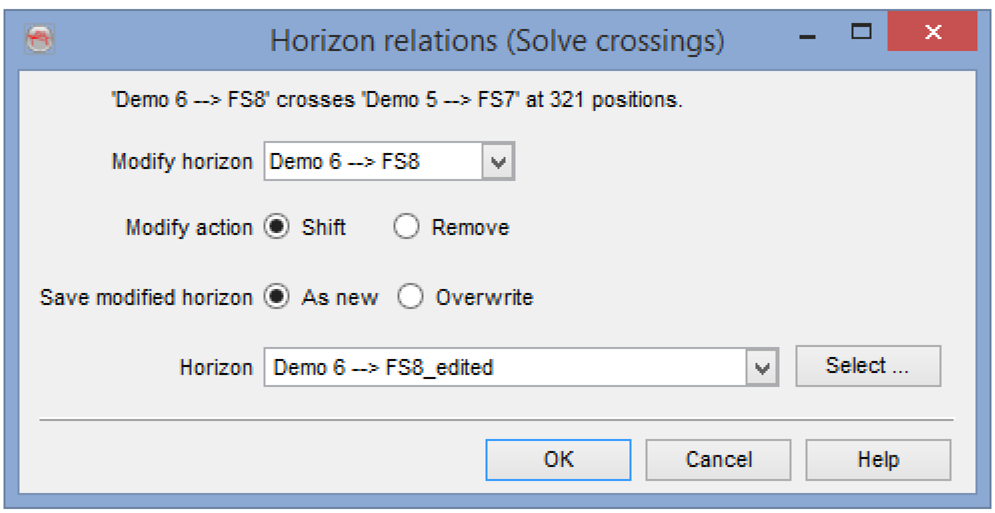
Stratigraphic marker can be assigned to one or more horizons. The horizons will get the marker color, this will facilitate for example the well to seismic tie.
For more details on how to define stratigraphic markers and the subsequent units go to Manage Stratigraphy.

Relations

The Horizon relation window is used to resolve conflicts between horizons crossing each other. Read Horizons .... is used to select all horizons that need checking. The horizons are then sorted automatically from top to bottom. The Check crossings... button is used to check the crossings between the listed horizons automatically and resolve them.

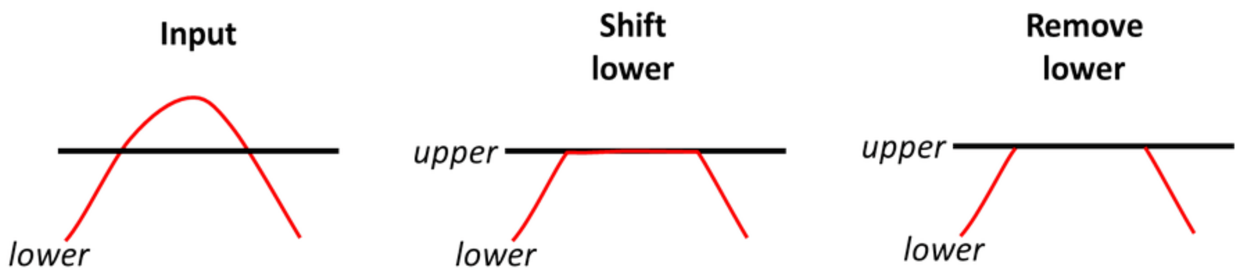


Solving crossing conflicts
To solve crossing conflicts select the horizon that will be modified. The software will check the number of positions where a conflict exits and modify the horizon by removing the conflict points or by changing the values to be equal to the overlying/underlying horizon.



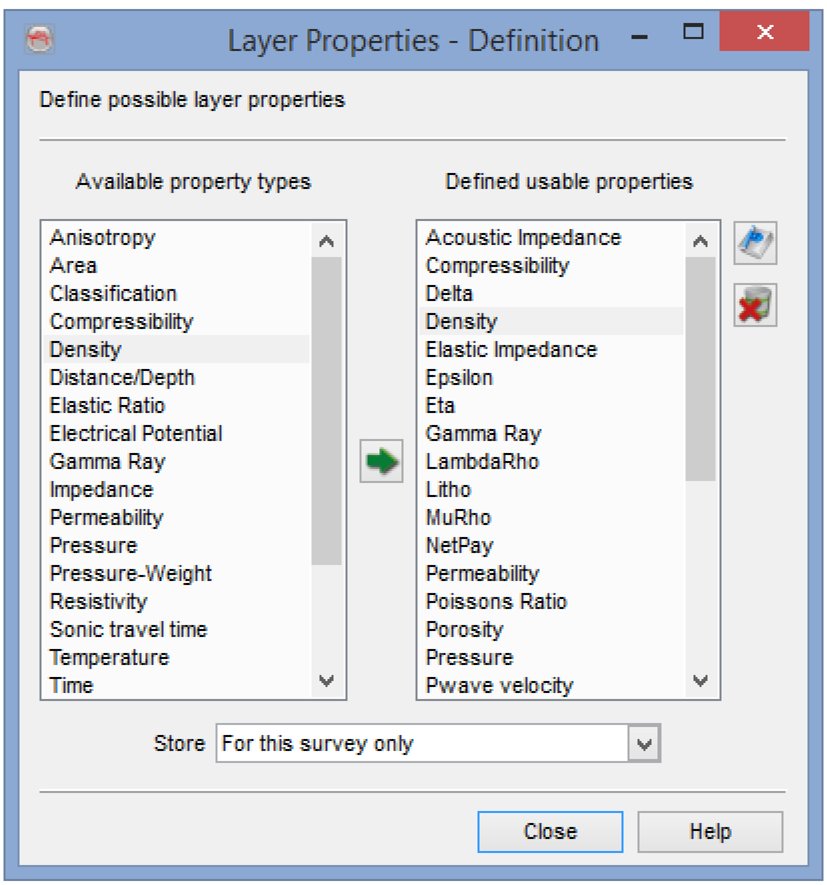
> **Note:** To honor the requirement that horizons cannot coincide, the actual values are not exactly equal, but they are within one sample position accuracy.

The figure below sketches what will happen to the lower (red) horizon if you select shift or remove. The software verifies that removing and shifting operations are executed properly and the correct HorizonCube calculation results are reached.



Manage Layer Properties

This window is also accessible from the Layer property selection window in the Layer Modeling module. In this manager, different usable layer properties are listed with their corresponding type. For example Delta and Epsilon properties belong to the Anisotropy type. Click on the usable property in the right column to see on the left column what is the type associated (highlighted).

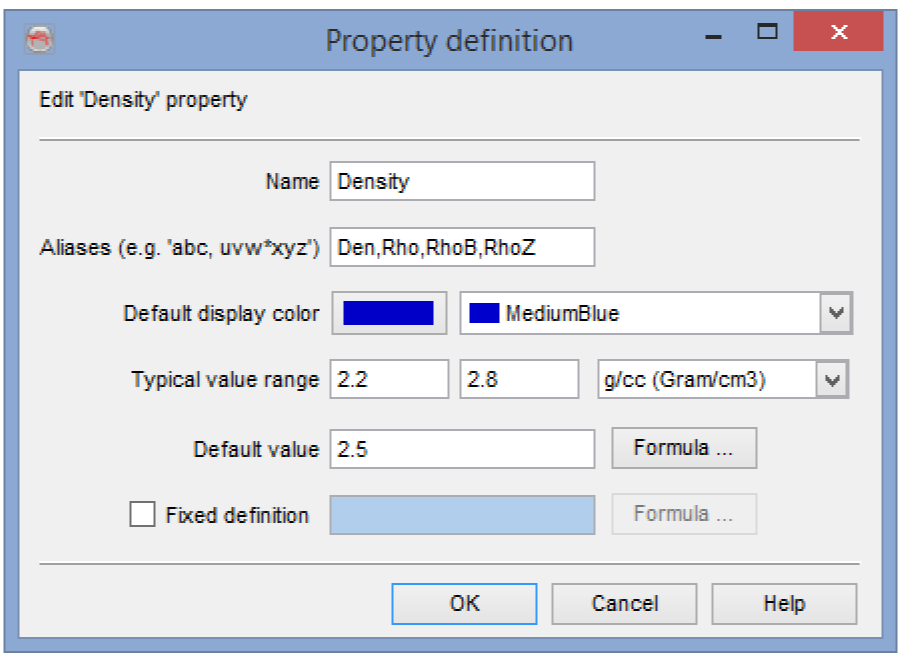


The two lists are hardcoded. However the user can ask for adding an extra property or type in sending an email to support. Usable properties listed can be removed of the list if they have no use in the current project.

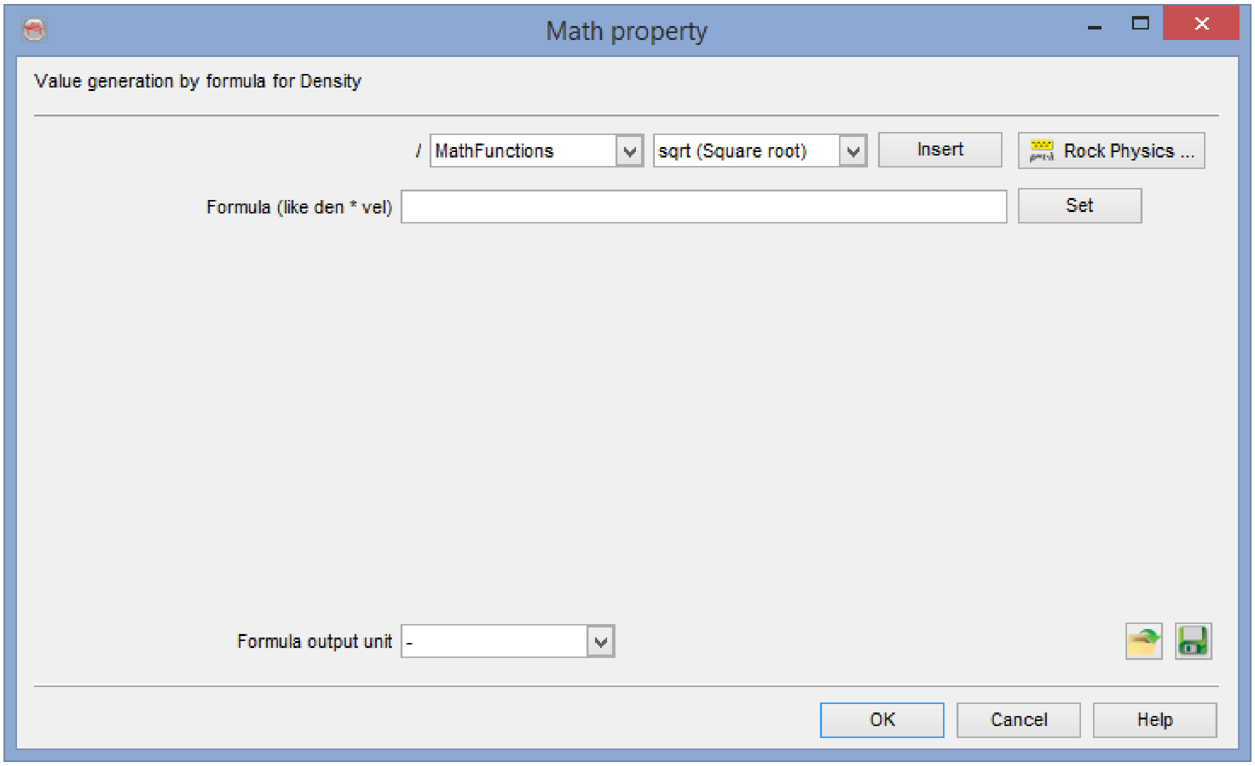
Define/Edit usable properties

One usable property can be correspond only to one property type. For changing the property type of a usable property, the usable property needs to be removed and added again in association to the selected property type. To add a usable property, select a property type and click on the in between the two list : it opens the property definition window.For each property needs a name. Possible aliases can be specified. It is useful to associate the correct log to a property: logs with different names can thus be related to the same property. Default display parameters are set up : the colour and the typical range value with the associated units.

Already defined usable properties can also be edited on clicking on the icon.



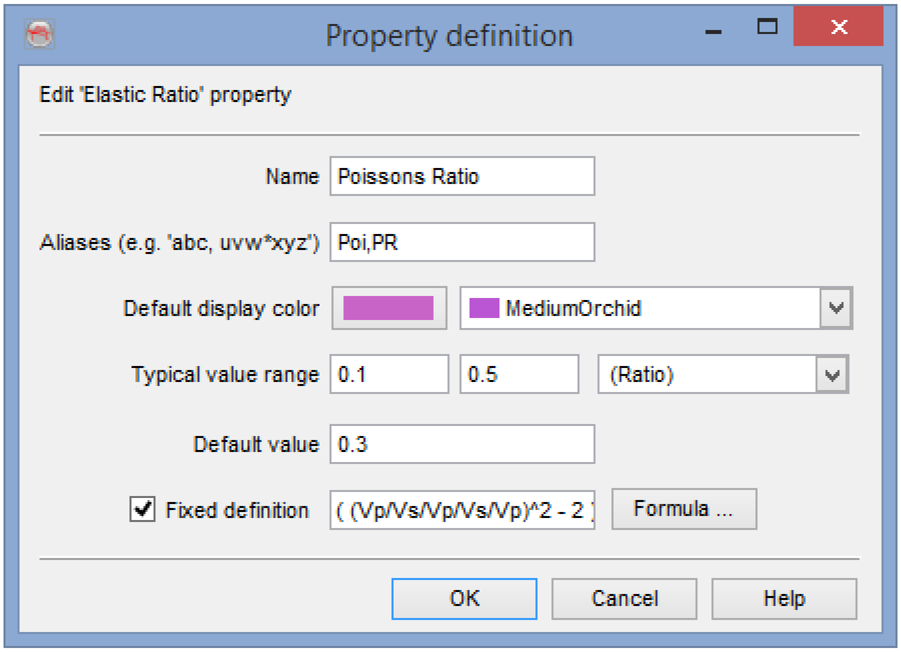
Optionally, the default value of any property can be defined by a specific equation that can either be typed in or retrieved from a list of saved formulas on clicking on *Formula*... Further formulas and equations may be utilized from the RockPhysics library, reached via the icon.

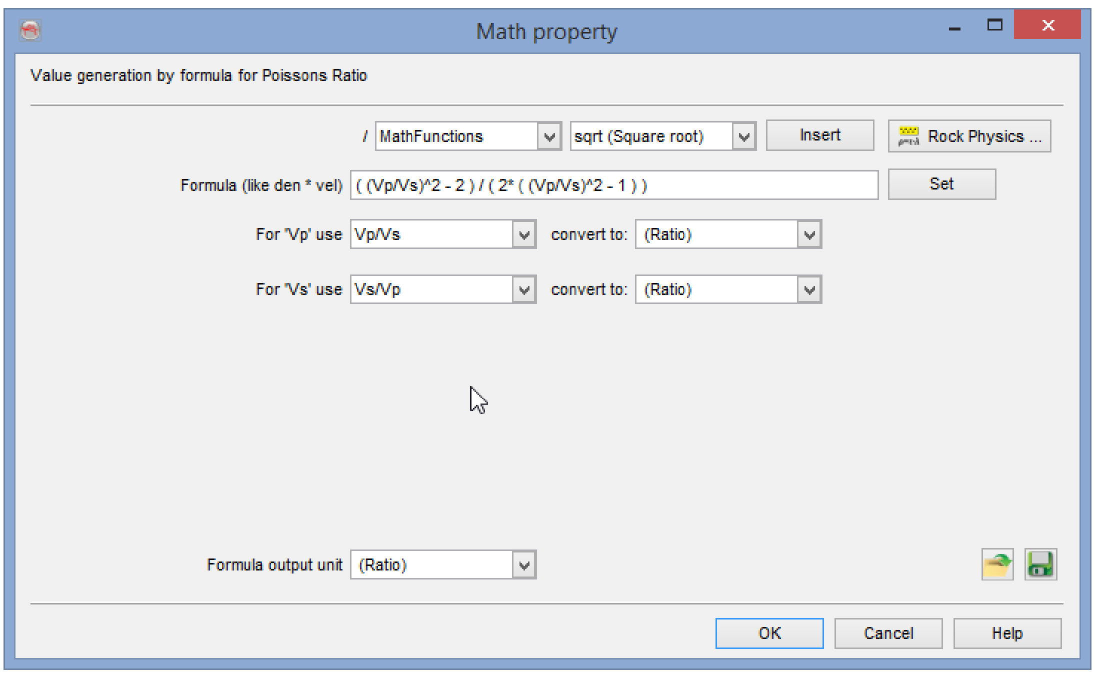


The edited Layer Properties are saved when clicking on *Ok*. There are also options here to either *Load*  or *Save*  Maths Formulas.

Fixed Definition: A property that will never be modeled directly but will be auto-computed in the background from other modeled properties. Formula for such properties always remain consistent, irrespective of any geological setting: Acoustic Impedance, Shear Impedance, Vp/Vs Ratio, Poisson's Ratio, Lambda-Rho, Mu-Rho etc.

The 'fixed definition' for these properties can be specified by ticking the 'Fixed definition' box and clicking on the Formula button, bringing up the RockPhysics library:





Storing levels

The level where it is saved is specified in the main window:

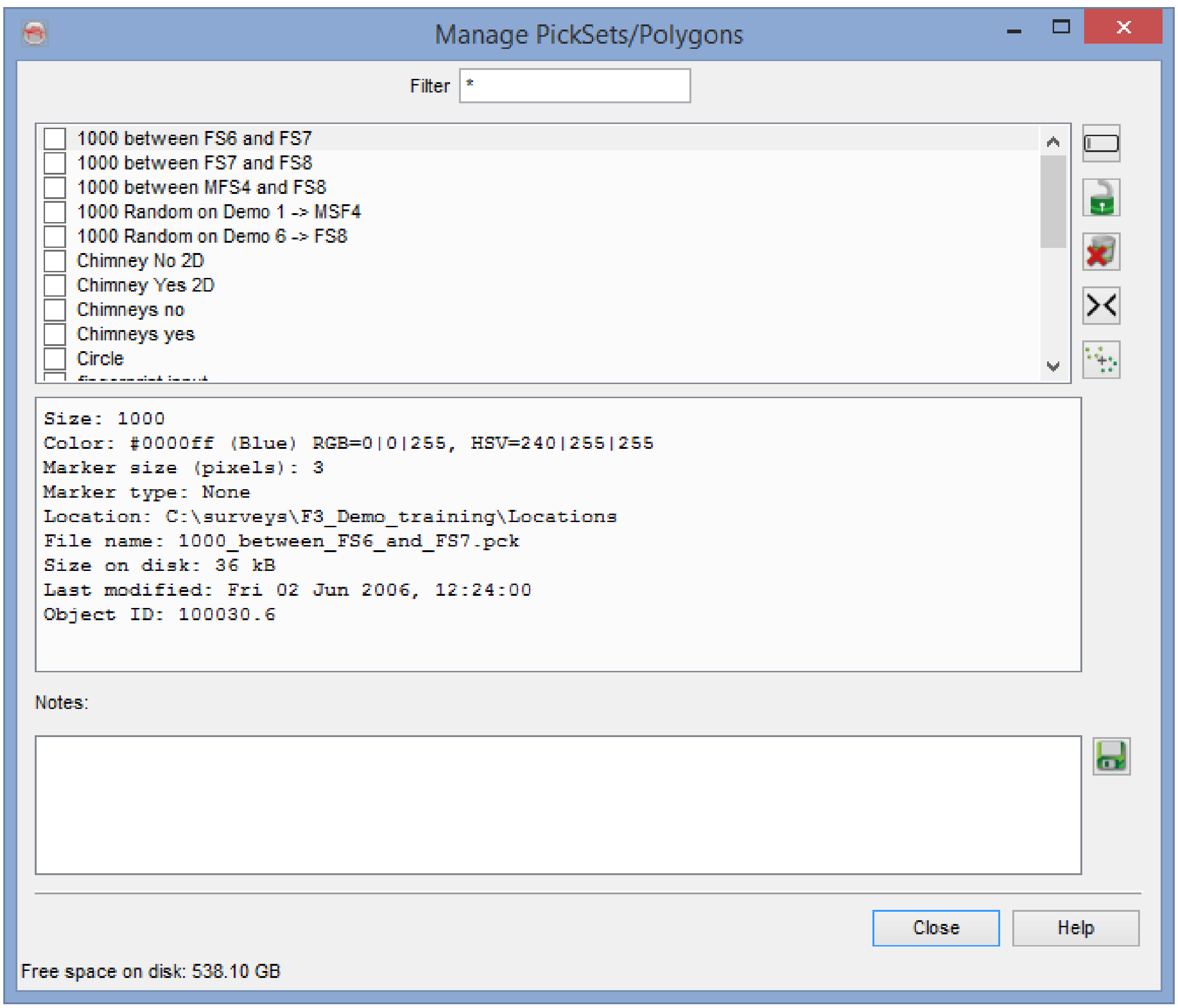
- For this survey only (default): save at the root of the survey, applicable for all users only for this survey (if no parameters saved for the user).

- As default for all surveys: save in dTect/data (where are all the surveys), applicable for all users and all surveys (if no parameters saved for the user or this survey)
- As default for my user ID only: save in home/.od. If it exists, it has the priority on the two others.

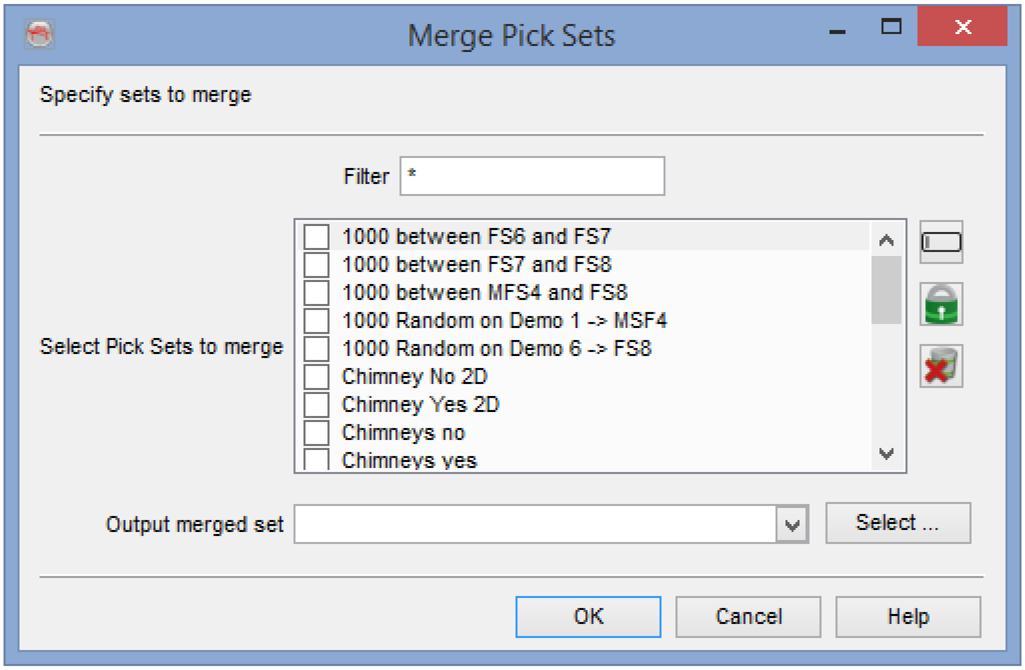
Once the parameters saved, the only way to access the default parameters provided by OpendTect is to delete the Properties file where it is located. OpendTect will then access its default properties file.

Manage PickSets & Polygons

In this window, all available picksets are shown. These picksets can be moved to a different location, they can be renamed, locked / unlocked, removed or set as default using the icons on the right.



Several pickset can be merge together using *Merge Pickset Button*.



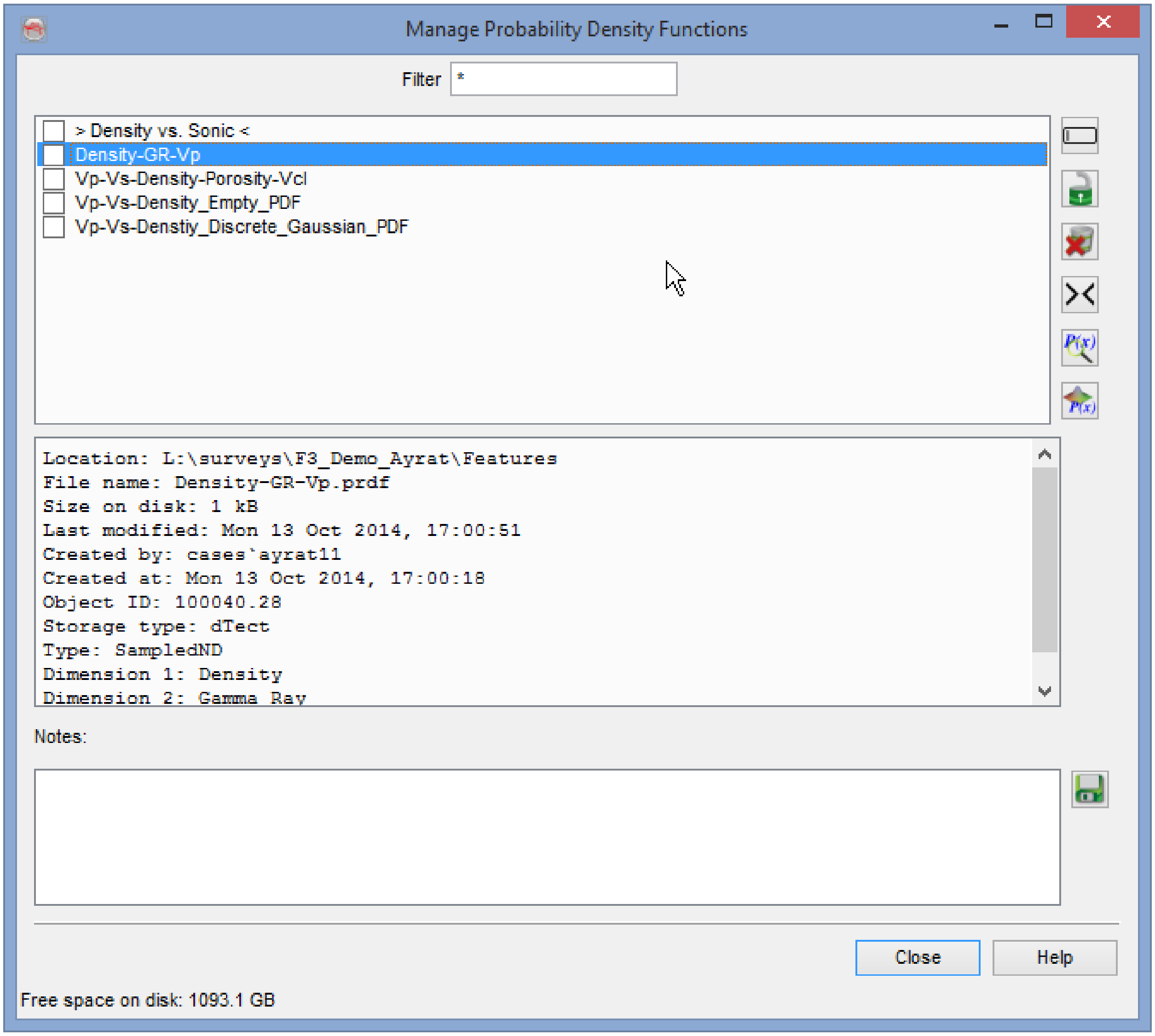
Note: The top filter is used to filter-out the objects with selected names. For instance, to display all picksets that start with letter S use "S*".

Manage Probability Density Functions

The window allows to manage Probability Density Functions. PDF may be imported, extracted from crossplots or generated with user-defined specifications. Main uses of PDF in OpendTect are Bayesian Classification and stochastic pseudowell modeling in SynthRock.

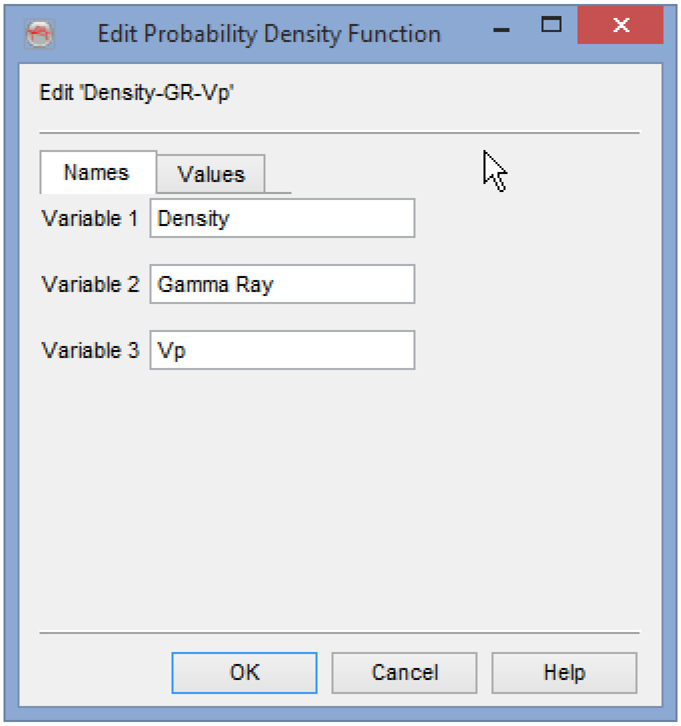
PDFs can also be exported for external use.

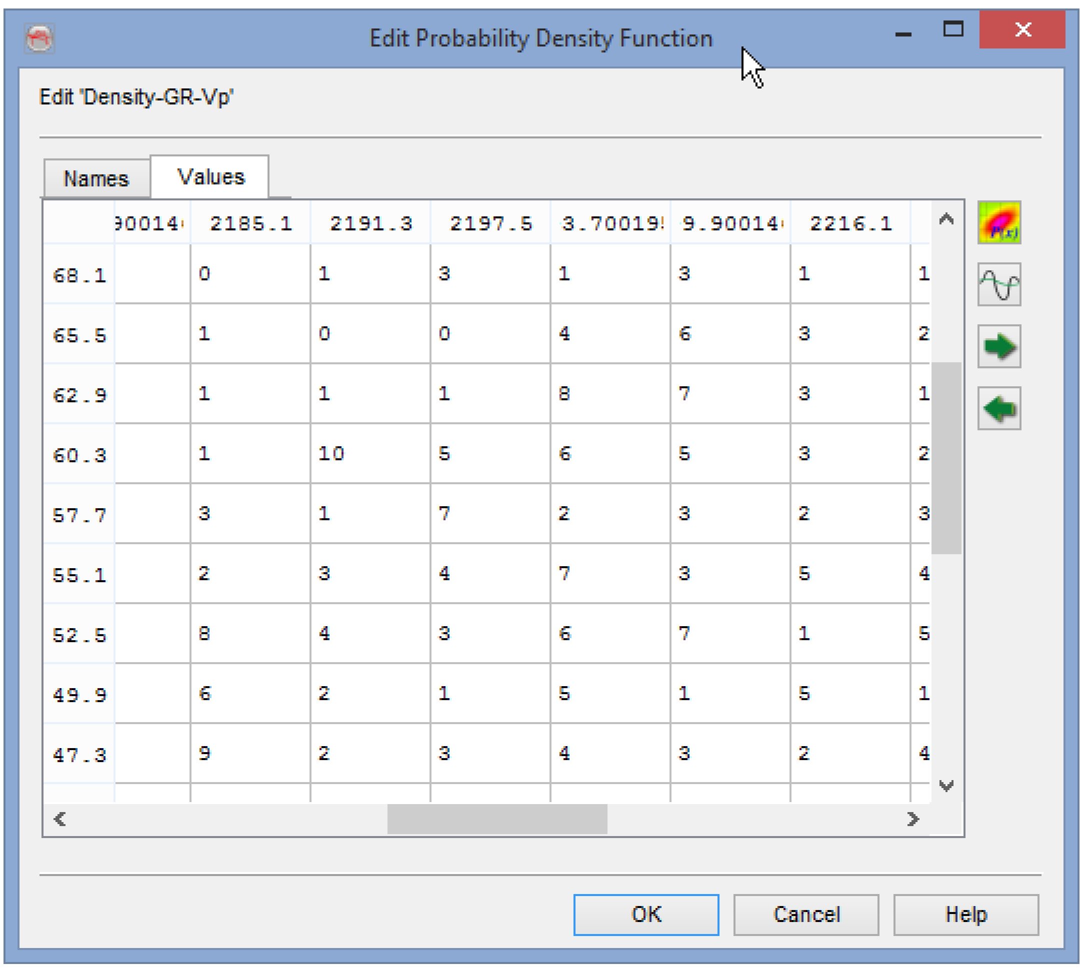
PDFs can be discrete or continuous. A discrete PDF can have up to 3 dimensions, while continuous PDF is not limited by number of dimensions. The example above shows a discrete PDF with 3 dimensions, each taken from a well log. The icons on the right hand side allow to perform the following actions, respectively from top to bottom: Rename, Lock/Unlock, Remove, Set as default, Browse/Edit and Generate PDF.



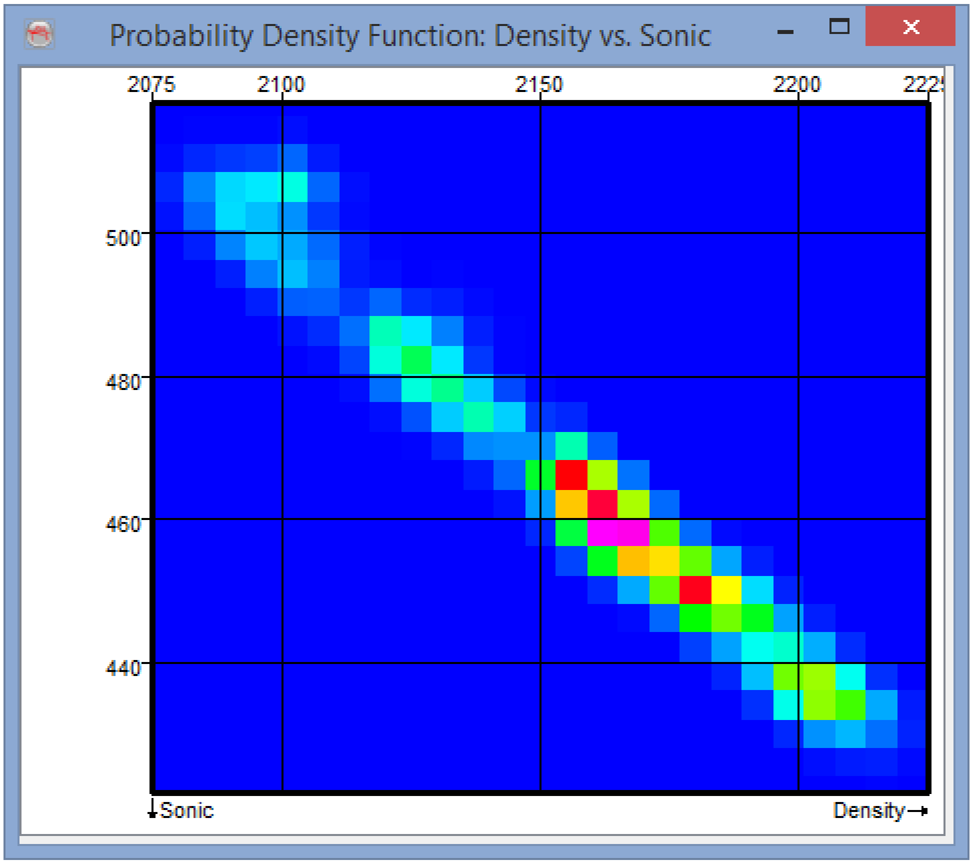
Browse/Edit Probability Density Function

By clicking on  in the Manage PDFs window PDF may be edited with respect to the names of the variables (*Names* tab, left), or in the table showing the stored values (*Values* tab, right). Changes applied in any of the two tabs will be saved only when pressing OK button. A popup window will give the choice to overwrite, save as new or cancel changes.


Edit Probability Density Function
Edit 'Density-GR-Vp'
Names
Values
Variable 1
Density
Variable 2
Gamma Ray
Variable 3
Vp
OK
Cancel
Help


Edit Probability Density Function
Edit 'Density-GR-Vp'
Names
Values
90014 2185.1 2191.3 2197.5 3.70019 9.90014 2216.1
68.1 0 1 3 1 3 1 1
65.5 1 0 0 4 6 3 2
62.9 1 1 1 8 7 3 1
60.3 1 10 5 6 5 3 2
57.7 3 1 7 2 3 2 3
55.1 2 3 4 7 3 5 4
52.5 8 4 3 6 7 1 5
49.9 6 2 1 5 1 5 1
47.3 9 2 3 4 3 2 4
OK
Cancel
Help

The first icon right of the table ( ) launches a 2D viewer that displays the values seen in the table. If the PDF has 3 dimensions, the left and right arrows may be used to navigate through the bins of the third variable with increasing and decreasing values respectively.



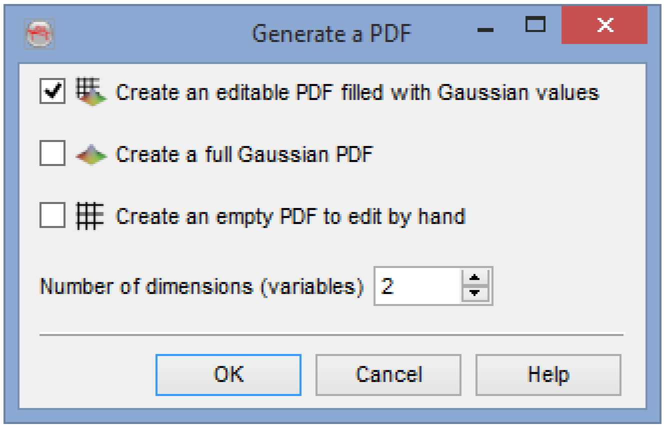
The second icon right of the table ( ) performs smoothing of the PDF data. Weighted average of a central sample with 1/2 weight and N neighbouring samples (excluding diagonal neighbours) each with 1/2N weight is calculated at every bin, where N=2, 4 and 6 for 1D, 2D and 3D PDF. This smoothing is rather gentle, and can be repeated multiple times for a more pronounced effect.

Generate Probability Density Functions

User-defined

A user defined PDF can be generated by clicking on the bottom icon ( ) in the Manage Probability Density Functions window.

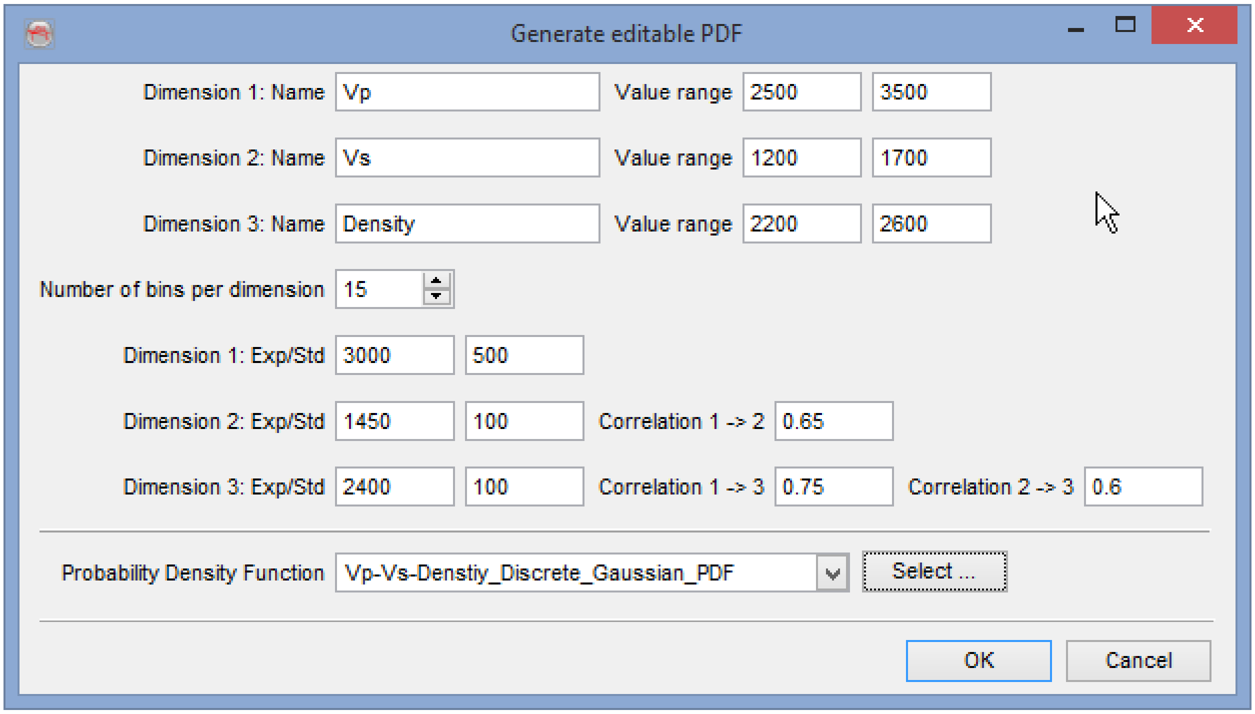
Three types of PDFs can be generated: discrete Gaussian (*Create an editable PDF filled with Gaussian values*), continuous Gaussian (*Create a full Gaussian PDF*), and discrete empty (*Create an empty PDF to edit by hand*).



Discrete Gaussian and discrete empty PDF can have up to 3 dimensions, while continuous Gaussian can virtually contain any number of dimensions. Values of discrete PDFs can be browsed, edited and smoothed after creation since they are stored in tables. Continuous Gaussian PDF exists only in the description form, the corresponding probabilities are computed on-the-fly.

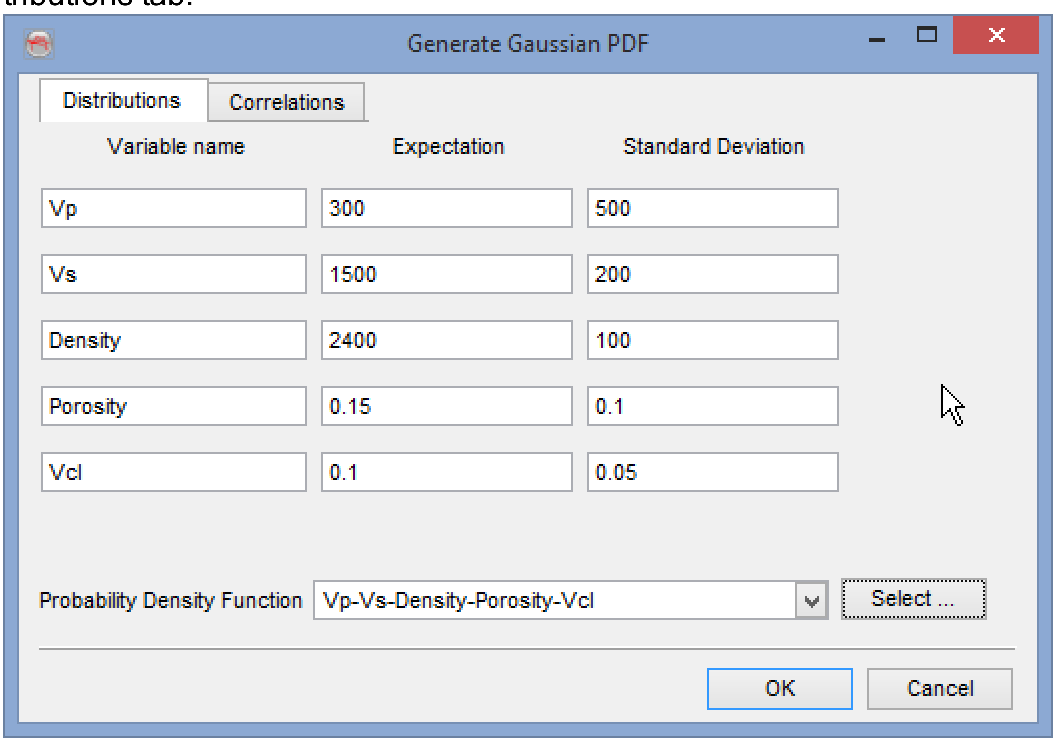
Create an editable PDF filled with Gaussian values

The example below shows generation of a discrete Gaussian PDF with 3 dimensions. Required parameters include dimension *Names, Value ranges, Number of bins per dimension, Expectations, Standard deviations* as well as *Correlation* coefficients between all dimensions (except for 1D). PDF is saved by specifying its name and clicking OK. It can be browsed, edited and smoothed through the Manage PDF window.

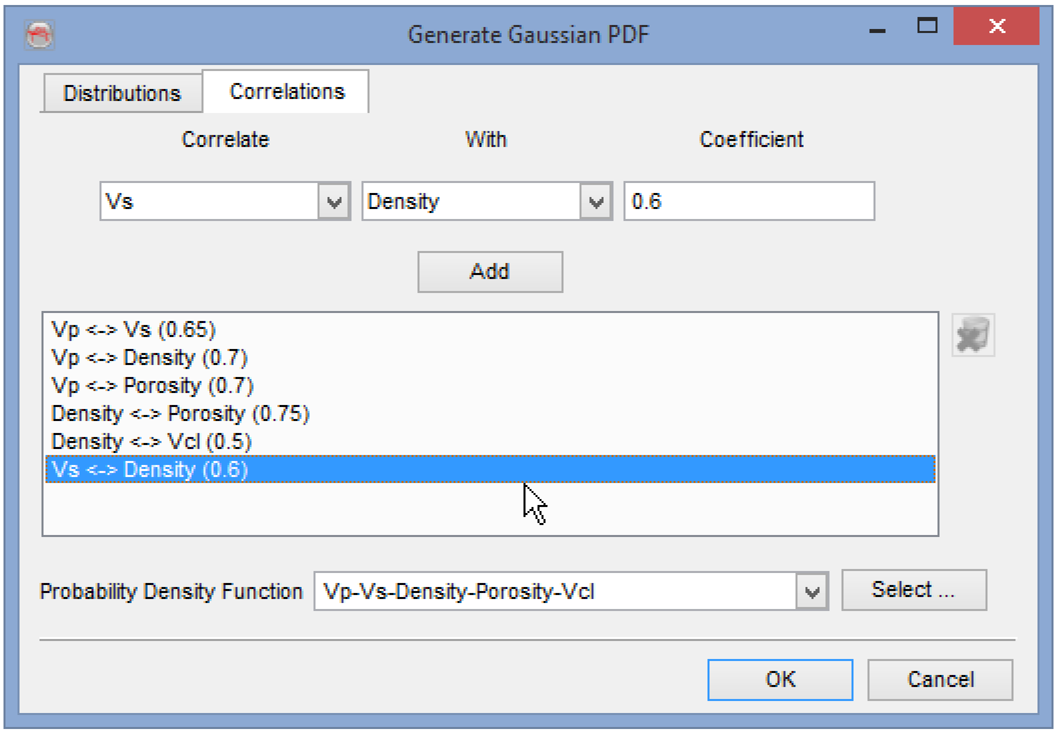


Create a full Gaussian PDF

The next example shows generation of a continuous Gaussian PDF with 5 dimensions. Dimension Names, Expectations and Standard Deviations are specified in the Distributions tab:



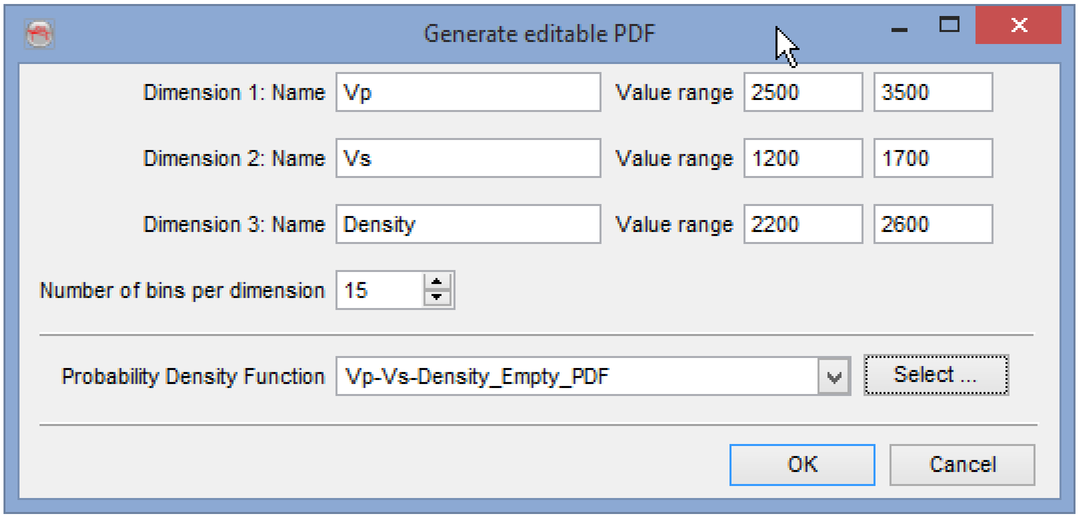
Correlations tab allows to define *Correlations* by selecting dimensions, setting their correlation coefficient and clicking *Add* button. Existing correlation can be selected from the list and edited by updating its correlation coefficient and clicking *Set* (Set button will appear instead of *Add*), or deleted by clicking the*Remove selected correlation* icon (). PDF is saved by specifying its name and clicking OK. Continuous Gaussian PDF is stored only in the description form which can be edited through Manage PDF window.



Note: Windows for generation of 1D and 2D continuous Gaussian PDFs shown below do not have Correlations tab

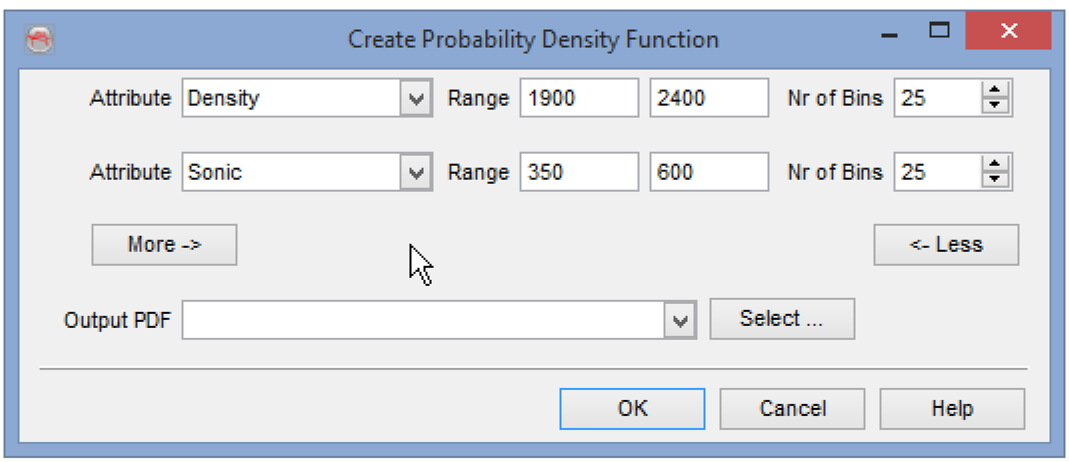
Create an empty PDF to edit by hand

This next example shows generation of an empty discrete 3D PDF. Dimension *Names, Value ranges* and *Number of bins per dimension* are required. PDF is saved by specifying its name and clicking OK. After creating empty PDF, probabilities must be filled in manually by clicking on the icon in the Manage PDF window and editing the table.



From Crossplots

Alternatively, a PDF can be created using the Cross-plot tool by clicking on the *P(x* icon in Cross-plot window . This icon launches a pop-up dialog that can be used for selecting attributes in order to create PDFs.



The number of PDF dimensions can be set to 1, 2 or 3 by clicking *More* and *Less* buttons. Note that all attributes from the Cross-plot table can be selected. Attribute ranges are generated automatically to fit the extracted data distribution. These can be edited before creating the PDF.

Manage Seismics

Post-stack Seismic data should be managed from these windows. There are separate managers for post-stack 3D and post-stack 2D. Access these via *Survey* > *Manage* > *Seismics...* or via the icon.

They all use common management icons on their right hand side:

Change the location of the file on disk

Rename

Toggle read-only on/off

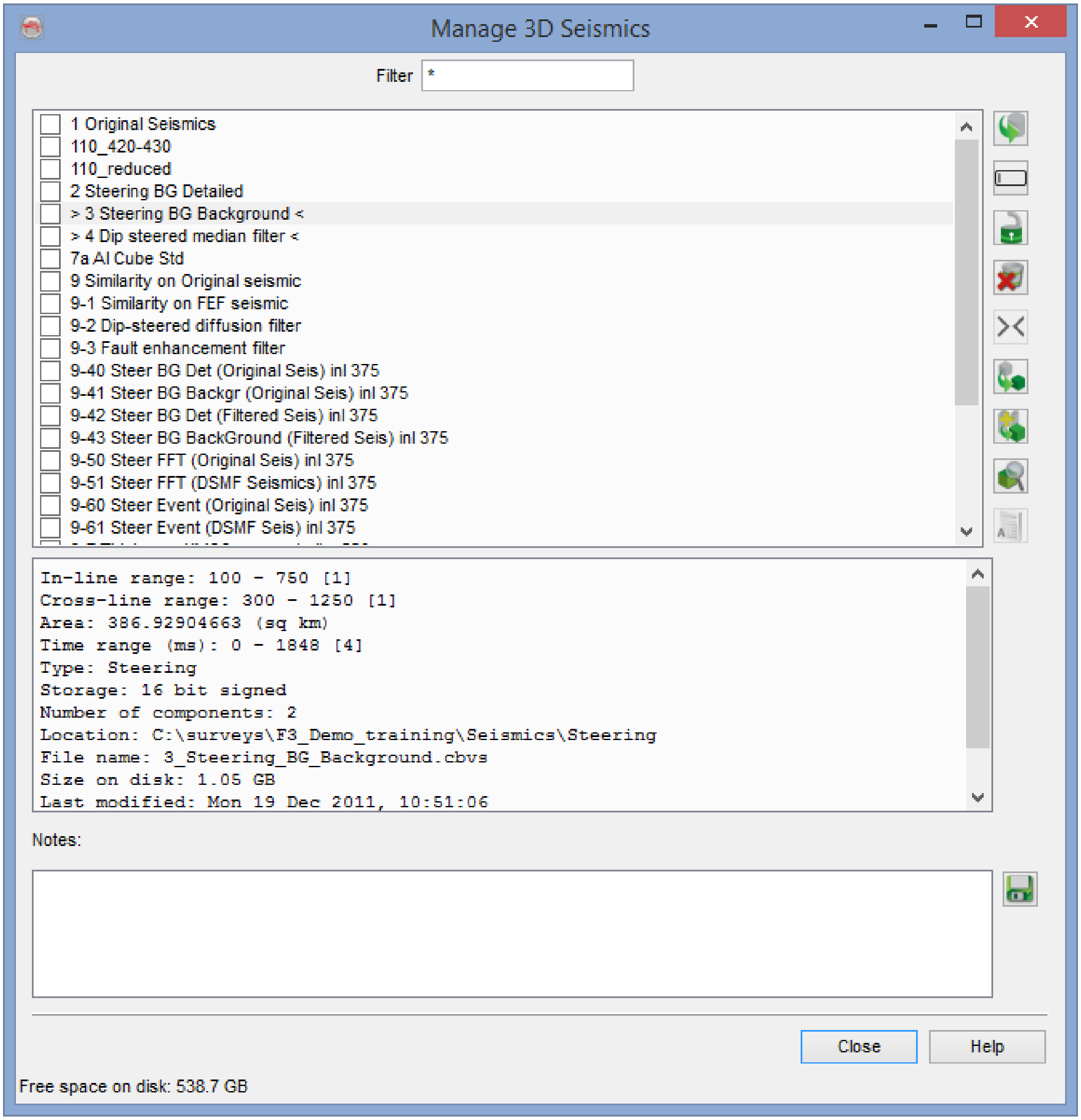
Delete

Set as the default object for its kind.

Note: The top filter is used to filter-out the objects with selected names. For instance, to display all volumes that start with letter S use "S*".

Manage 3D Seismics

The 3D Seismic file management window lists post-stack volumes loaded in the survey. Information related to the selected volume is displayed in the central field and personal/survey-related notes can be added and saved in the bottom field.



Alongside the standard actions (chnage disk location, rename, remove etc), the following volume-specific actions can be undertaken:

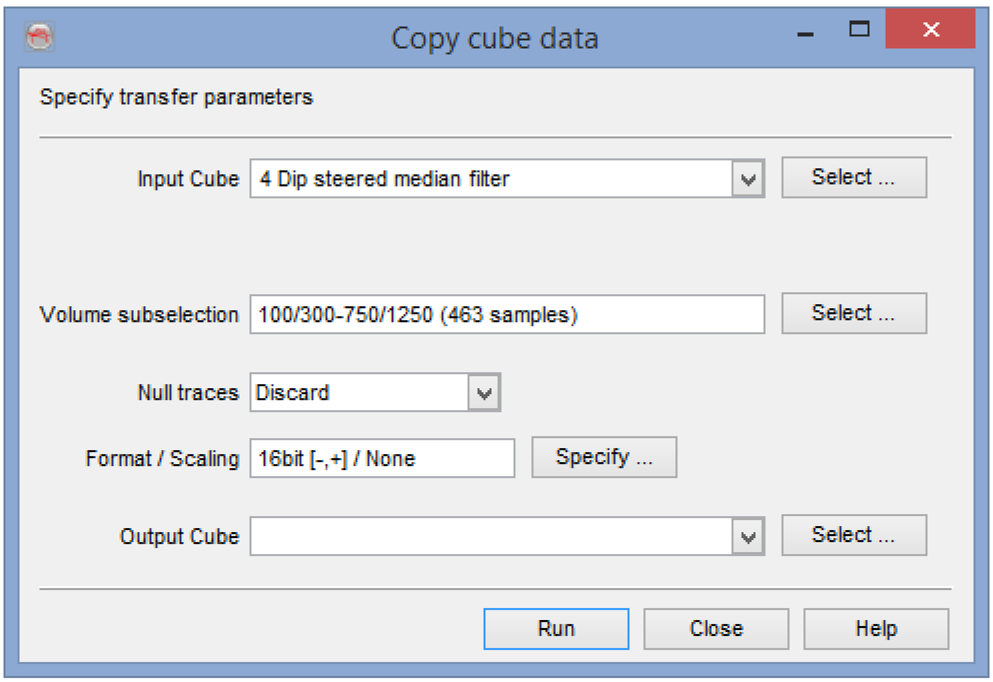
 Copy the volume to another volume (different size, format, sampling rate, ...)

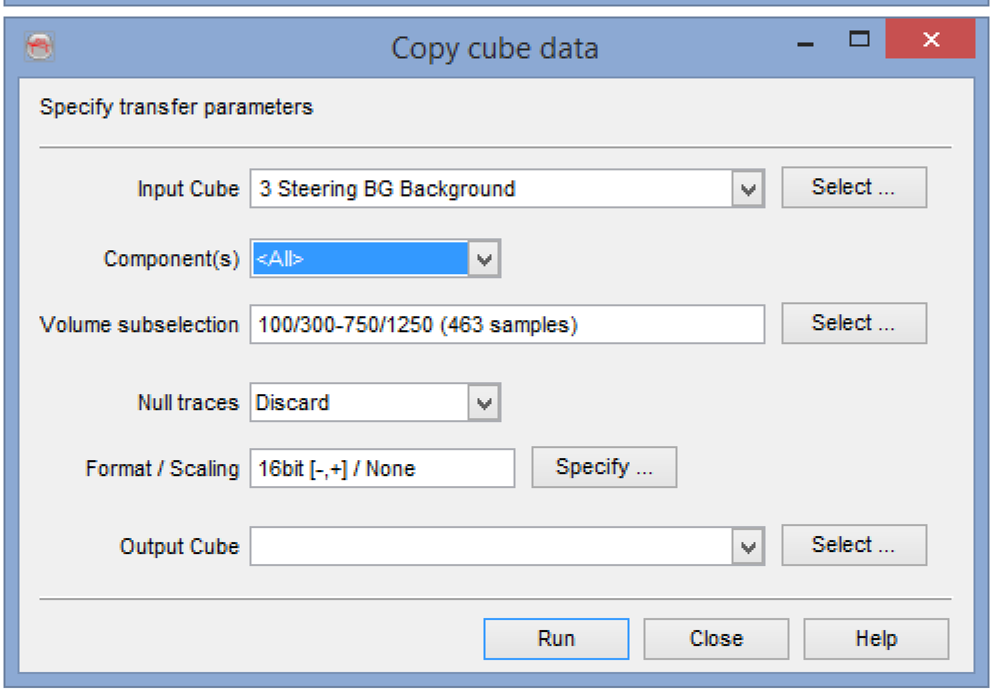
 Merge several overlapping or consecutive volumes

 Browse in the file

Copy Cube

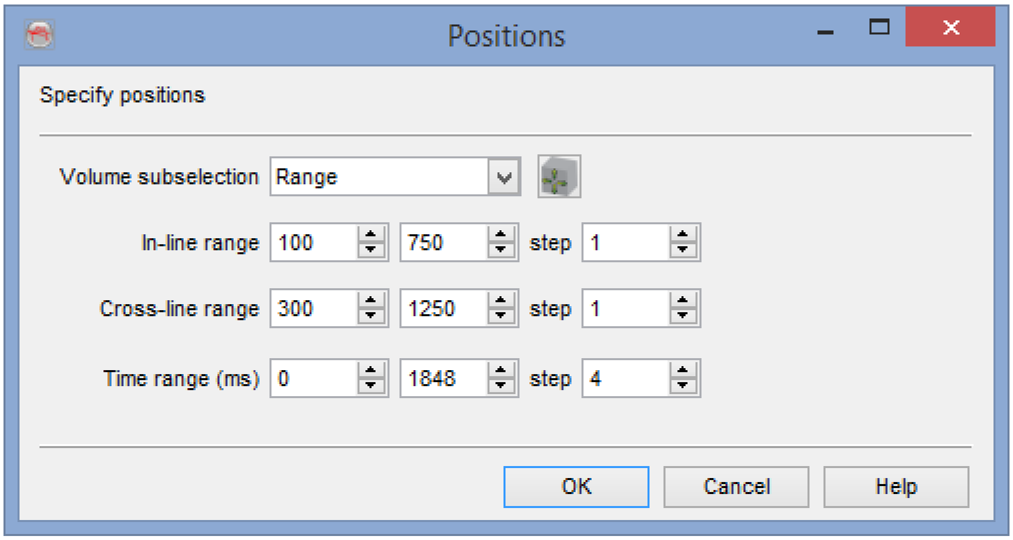
Any volume can be copied  into a new volume. The Volume subselection defines the selection of the input cube to be copied and the Format / Scaling sub menu allows to specify how to store the new cube. Rectangular volumes are not required by OpendTect. Therefore null traces are dismissed by default. They can be added back with the *Null traces > Add*, within the inline/crossline range of the input volume. Larger volumes can be obtained while using the *Null traces > Add* option and the volume subselection menu.





If the input cube is multi-component (e.g spectral decomposition cube with different components, right), an option will be available allowing the user to choose between all available components. *All* components is the default setting.

Copy - Volume Sub-Selection



This standard menu is available when *importing/exporting* a volume, line, or horizon, copying a cube or horizon, or processing an attribute.
In all those processes, the output might be limited with respect to the available input data. The limitation may be:

- A rectangular part of the survey, possibly with a larger horizontal and vertical stepout
- An area limited by an OpendTect polygon. The area within the polygon can be as well by decimated horizontal by using larger stepouts
- A table of positions from an OpendTect pickset or from a text file. The text file should contain inline and crossline values without header
- All: This last option will output the maximum number of trace with respect to the available data and possible stepouts

Notes:

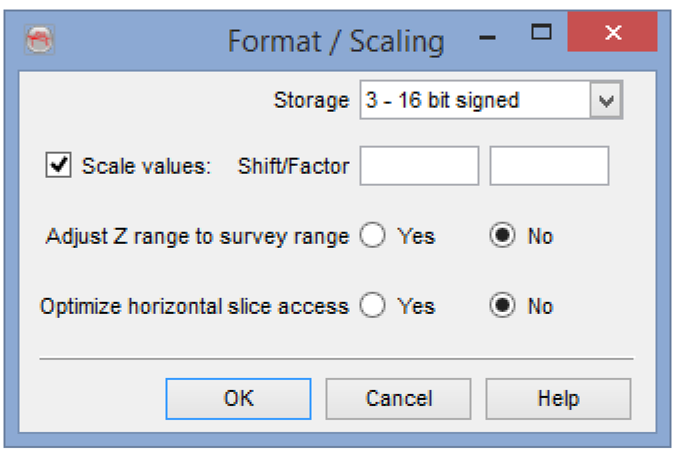
1. The use of larger *vertical* stepouts will cause the data to be decimated in the given direction. Please note that an anti-alias filter (using the frequency filter attribute) should be applied before decimating data. The copy-cube does not do it.

2. The use of smaller *vertical* stepouts will cause the data to be interpolated with a polynomial interpolation. This is mostly appropriate for seismic data.

3. Volumes tagged as Vint, Vrms or Vavg are not using a polynomial interpolation of the input amplitudes, as soon as Z start, Z stop and/or Z step are changed. Instead they are converted to the corresponding time-depth relation that is linearly interpolated (vertically), before back converting the interpolated TD function to the input type.

4. The copy-cube option does not do lateral interpolation of the data (but it can decimate). Use the Velocity gridder step of the volume builder to laterally grid a coarse volume.

Copy - Format & Scaling



This standard menu allows the change of the following elements:

- Change between storage type. Please note that this might clip your data.
- Scale the values given a linear equation.
- Adjust the Z range to the survey range by repeating the bounding samples up and down.

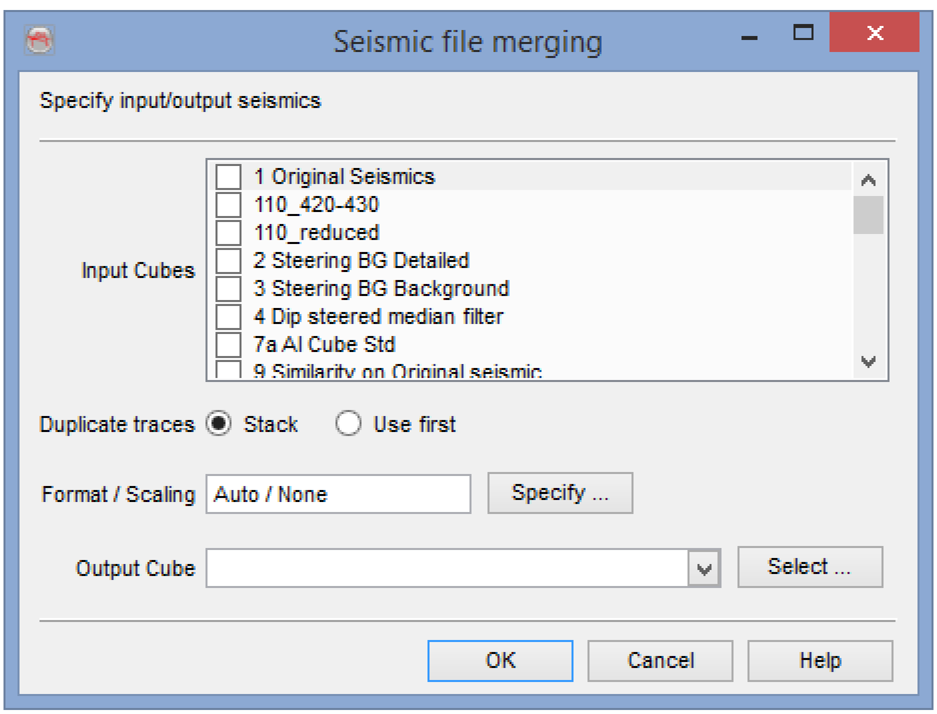
- Optimize the horizontal slice access. This will change the sorting mode in the volume on disk, and will cause inline/crossline accesses to be significantly slower based on the volume size.

Merge Files

The  icon is used to merge sub-volumes into one single volume. OpendTect processing time can be reduced by distributing automatically or manually batch jobs over multiple computers.

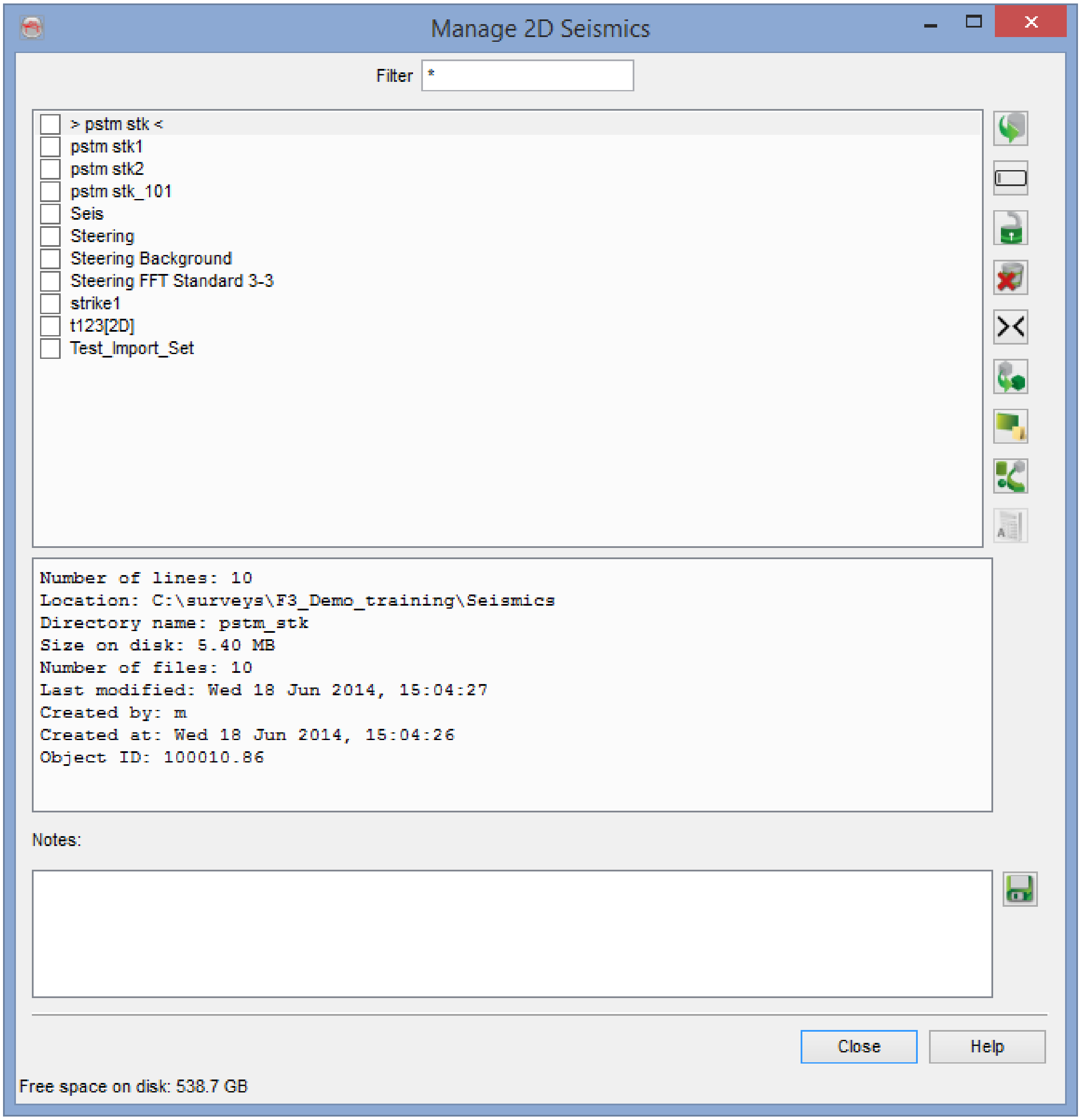
When merging two cubes, the duplicate traces can be stacked when merging e.g. two seismic cubes (the merging cube will reduce noise) or the traces of the first cube can be used. Priorities are set in alphanumerical order, as the volumes appear in the manager from top to bottom.

Select the input files from the multiple entry list and specify the *Output* file name. The user can remove the original files at a latter stage (use the *Remove* button  in the seismic file manager).



Manage 2D Seismics

2D surveys in OpendTect are grouped in datasets. These datasets have their own manager (shown below), separate from the Manage 2D Seismic Lines window.



The following actions can be applied on datasets, in addition to standard rename/delete options:

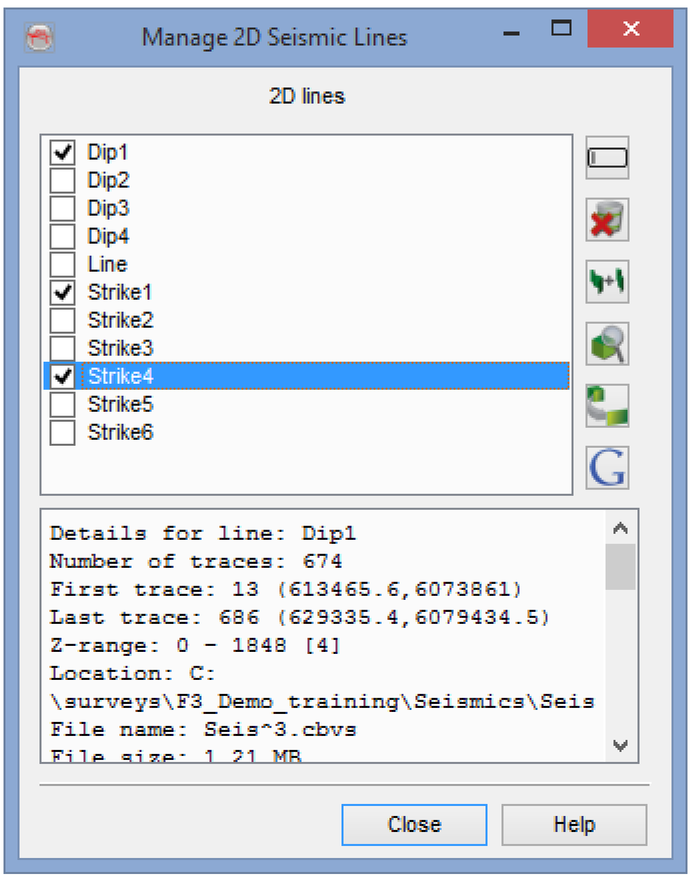
Copy all or part of the dataset to a new dataset

Access the 2D Lines Manager. Alternatively, double-click on the dataset name.

Dump the geometry (positions) to a text file.

Manage 2D Seismic Lines

Accessed via the 'Manage 2D Seismics' window, either by double-clicking on a dataset name or via the icon.



Lines can be renamed ( ) and deleted ( ). The following actions can also be made on the lines:

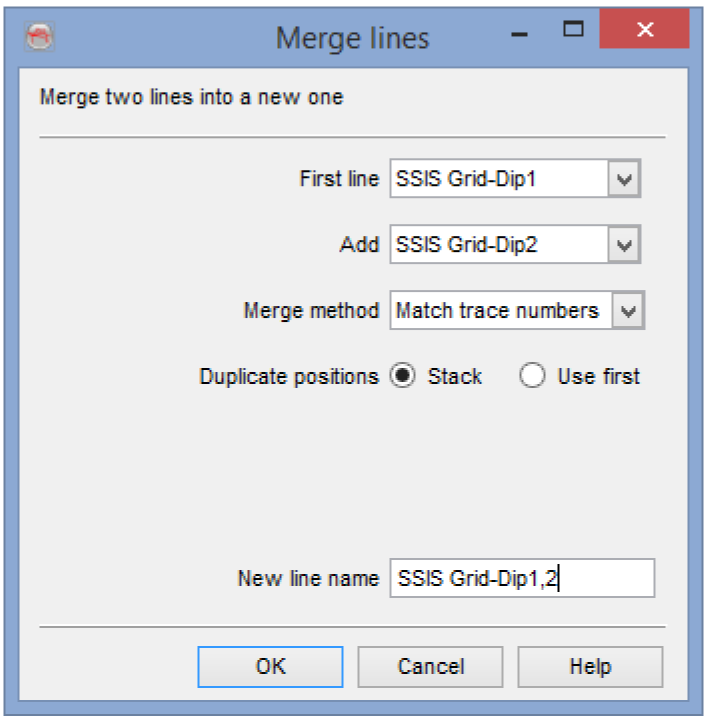
Merge of several lines to a new line.

Extraction (projection) from 3D volumes along the 2D lines.

Export of the geometry to GoogleEarth.

Merge 2D Lines

Two 2D lines can be merged together to create a single 2D line. The icon opens the merging window. The merge can be either of 2 lines with about the same geometry, or to append two (consecutive) lines to each other.



There are three alternative merging methods:

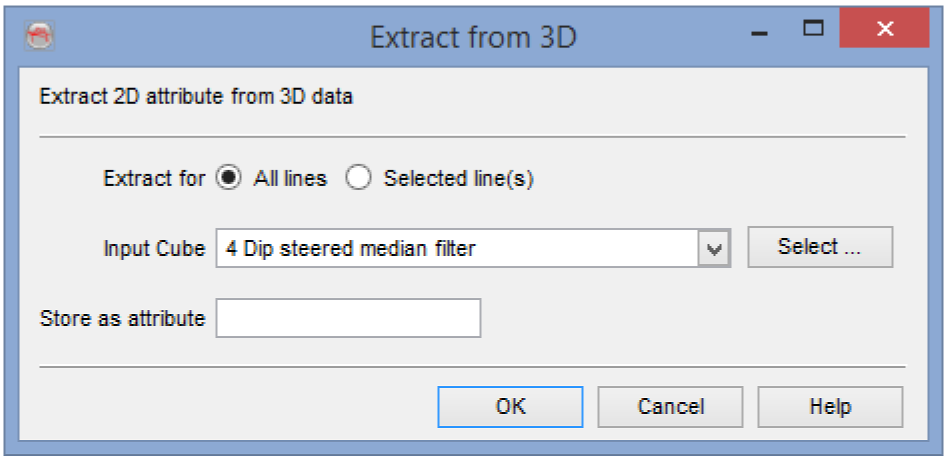
- Match trace numbers: Assumes the lines are at the same location, containing different attributes to be stacked and referenced using the same trace number array.
- Match coordinates: Same as above, but with different trace number arrays. Then the match will be based on coordinates, with a search

radius to match the traces. Please note that traces will be renumbered in this mode.

- Bluntly append. Append the line specified at the "Add" field line after the "First line". Please note that traces will be renumbered in this mode.

Extract 2D Attributes from 3D Volumes

This extraction tool, started from the icon, can be used to project a 3D volume onto 2D lines. This allows then to display the 3D volume along the lines, and to use the data from the 3D volume with 2D lines in the 2D attribute set.

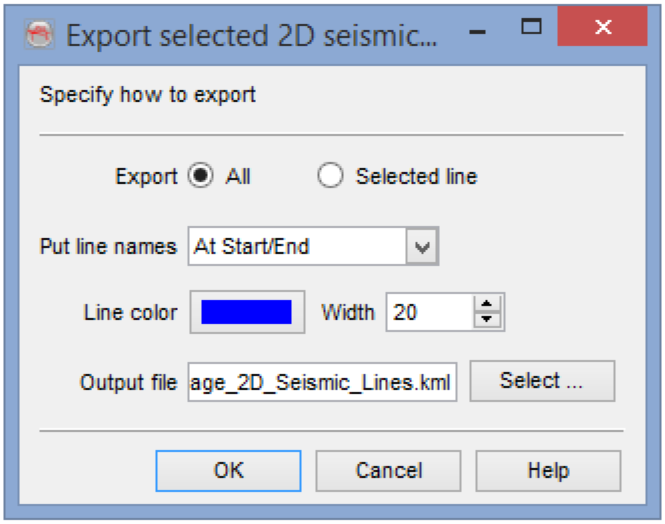


All lines may be processed, or a selection of lines made in the lines manager before going to this window. The settings are trivial: the 3D volume must be selected and an attribute name must be provided.

Please note that the polynomial interpolation does not fit an application of this tool to 3D seismic data.

Export 2D Geometry to Google Earth

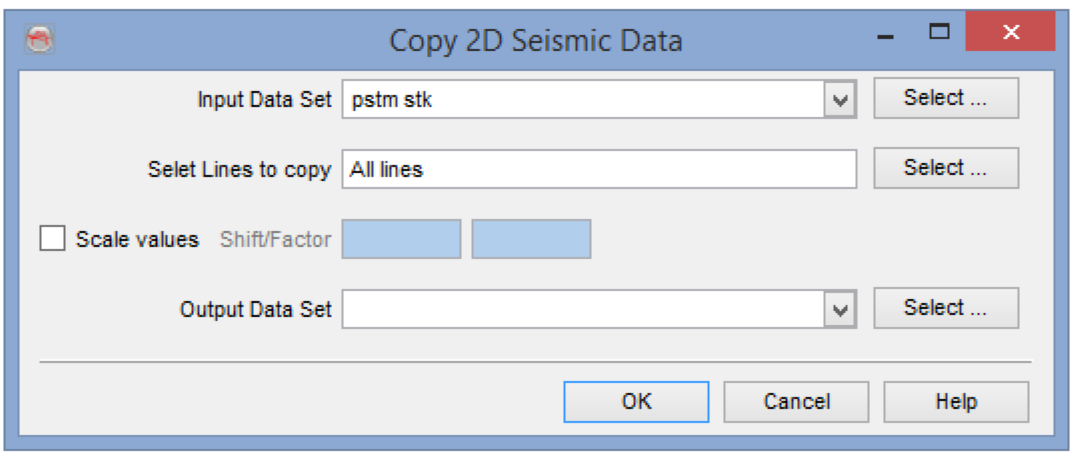
This window (see below),launched with the icon, is to export the 2D lines geometry in to a *.kml file. Different methods are supported (Start/End or both etc.) for labeling the line-names in the Google Earth file. The line color field is also editable. The width represents the thickness of the lines. The *Output file* field specifies the output location and name of the exported file (Format - kml).



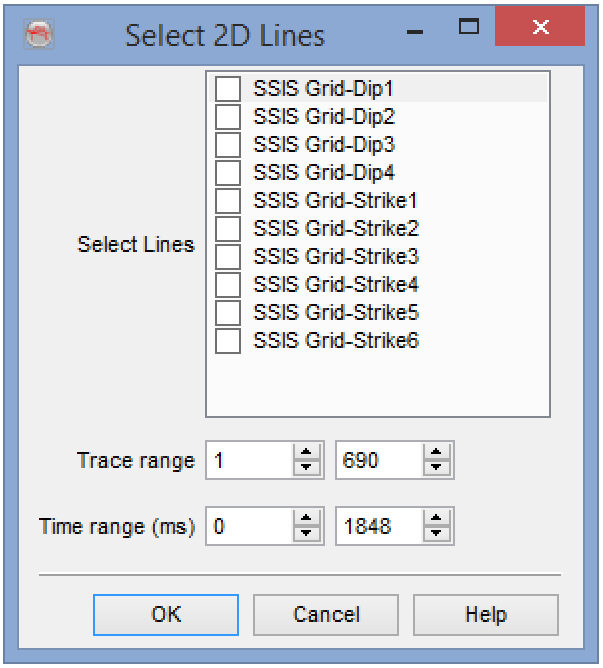
Export the 2D lines in a Google KML file.

Copy Data Set

This utility window can be used to copy (backup) a data set.

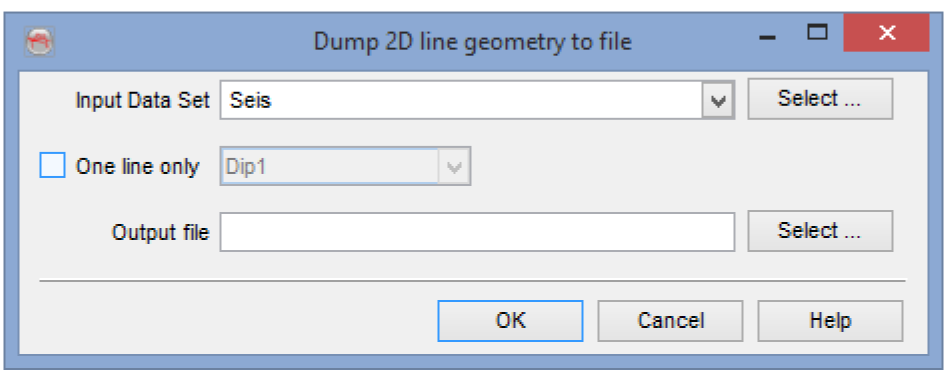


Pressing on the select button of the first line will allow the selection of which line(s) should be copied to the new data set:



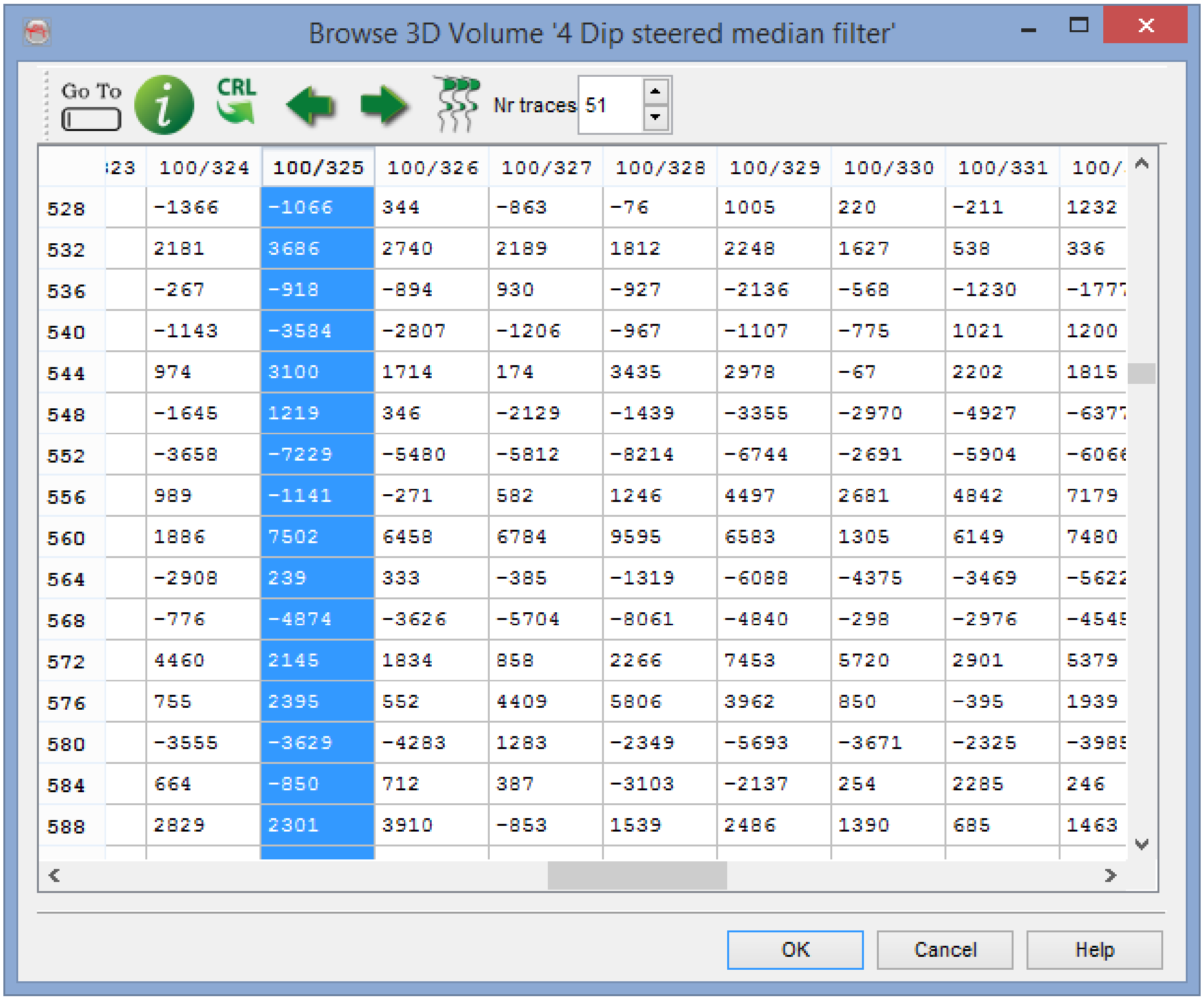
Dump 2D Geometry

With this module, accessed via , an ASCII file with the geometry of one or several 2D lines can be generated. By default, the output file contains trace numbers and X/Y locations.
This export facility may be very practical if you want to generate a base map from 2D lines for a different software package.



Browse & Edit Cube Locations

Cbvs files can be *browsed/edited* (edit the cube locations, positions, trace samples, etc) by pressing the icon. In the window that pops up (see below), sample values can be changed by editing any cell (similarly to an MS Excel sheet). Editing is disabled if the cube is write protected.



	23	100/324	100/325	100/326	100/327	100/328	100/329	100/330	100/331	100/
528		-1366	-1066	344	-863	-76	1005	220	-211	1232
532		2181	3686	2740	2189	1812	2248	1627	538	336
536		-267	-918	-894	930	-927	-2136	-568	-1230	-177
540		-1143	-3584	-2807	-1206	-967	-1107	-775	1021	1200
544		974	3100	1714	174	3435	2978	-67	2202	1815
548		-1645	1219	346	-2129	-1439	-3355	-2970	-4927	-637
552		-3658	-7229	-5480	-5812	-8214	-6744	-2691	-5904	-606
556		989	-1141	-271	582	1246	4497	2681	4842	7179
560		1886	7502	6458	6784	9595	6583	1305	6149	7480
564		-2908	239	333	-385	-1319	-6088	-4375	-3469	-562
568		-776	-4874	-3626	-5704	-8061	-4840	-298	-2976	-454
572		4460	2145	1834	858	2266	7453	5720	2901	5379
576		755	2395	552	4409	5806	3962	850	-395	1939
580		-3555	-3629	-4283	1283	-2349	-5693	-3671	-2325	-398
584		664	-850	712	387	-3103	-2137	254	2285	246
588		2829	2301	3910	-853	1539	2486	1390	685	1463

Several options are available:

Go To Brings you directly to a new position (inline/crossline).

Check selected trace information like: x/y coordinates, inline/crossline, vertical z ranges, number of samples.

CRL Switch to browse through Crosslines (and back to Inlines)

Step a preset number of inline/crossline positions to the left.

Step a preset number of inline/crossline positions to the right.

View the currently highlighted trace(s).

Manage Seismics Prestack

Prestack seismic data should be managed from these windows. There are separate managers for pre-stack 3D and pre-stack 2D. Access these via *Survey--> Manage--> Seismics Prestack...* or via the icon.

They all use common management icons on their right hand side:

Change the location of the file on disk

Rename

Toggle read-only on/off

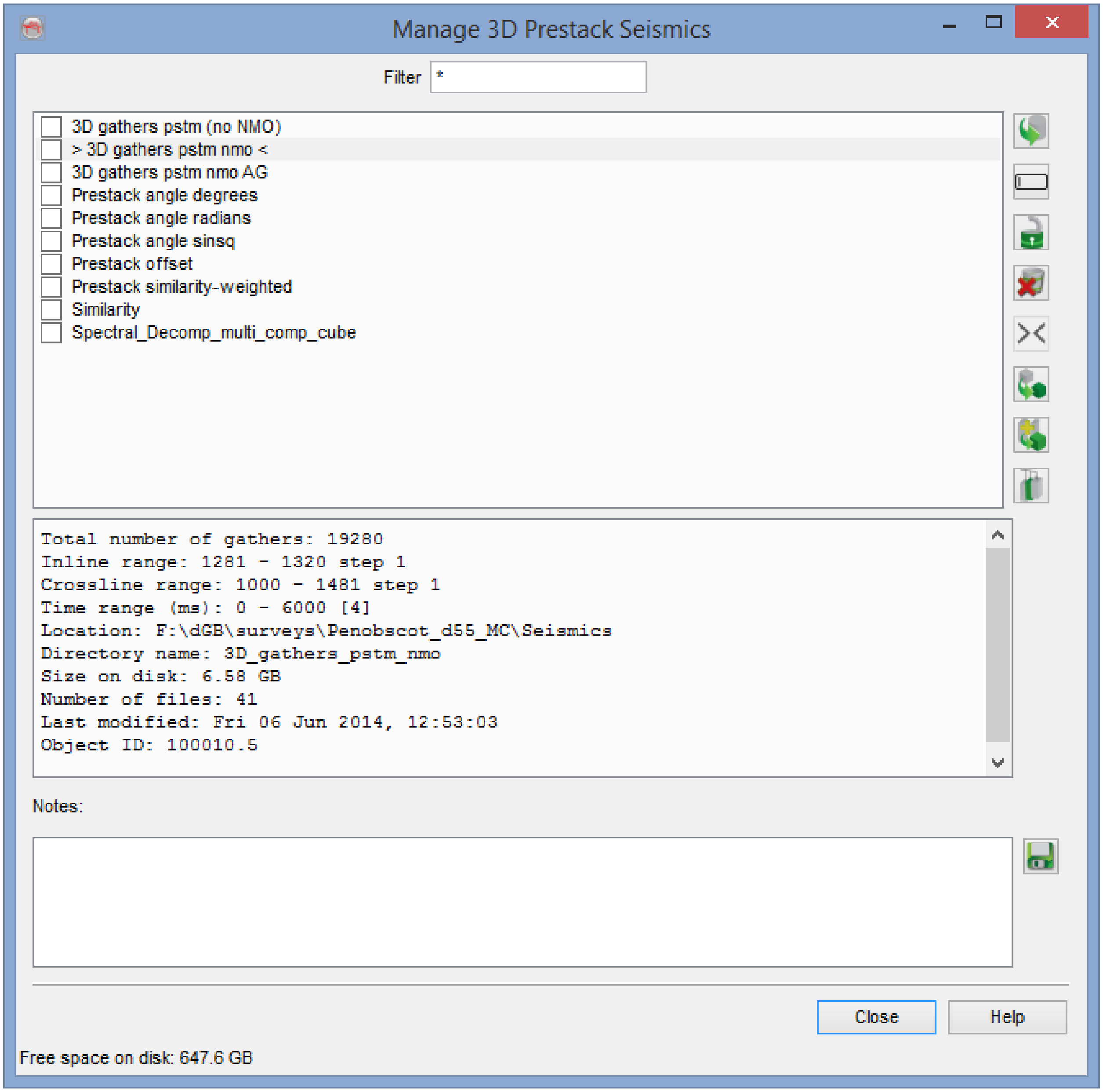
Delete

Set as the default object for its kind.

Note: The top filter is used to filter-out the objects with selected names. For instance, to display all volumes that start with letter S use "S*".

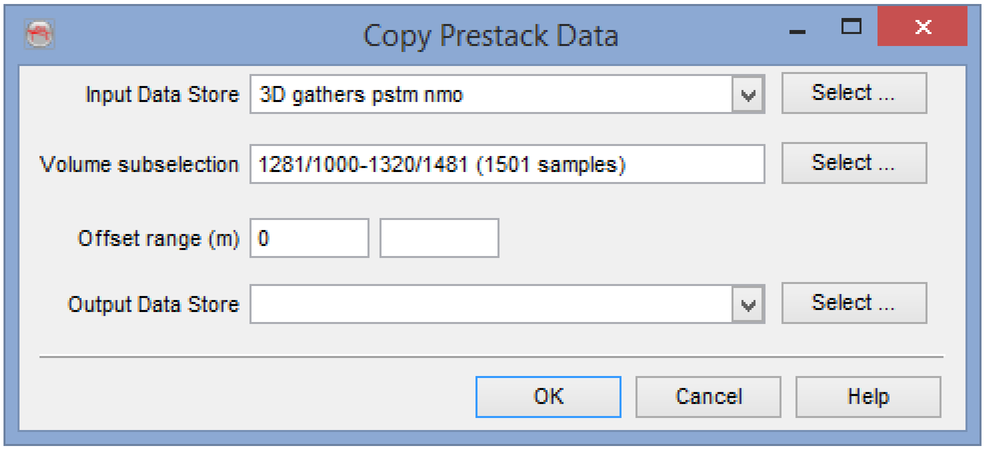
Manage 3D Prestack

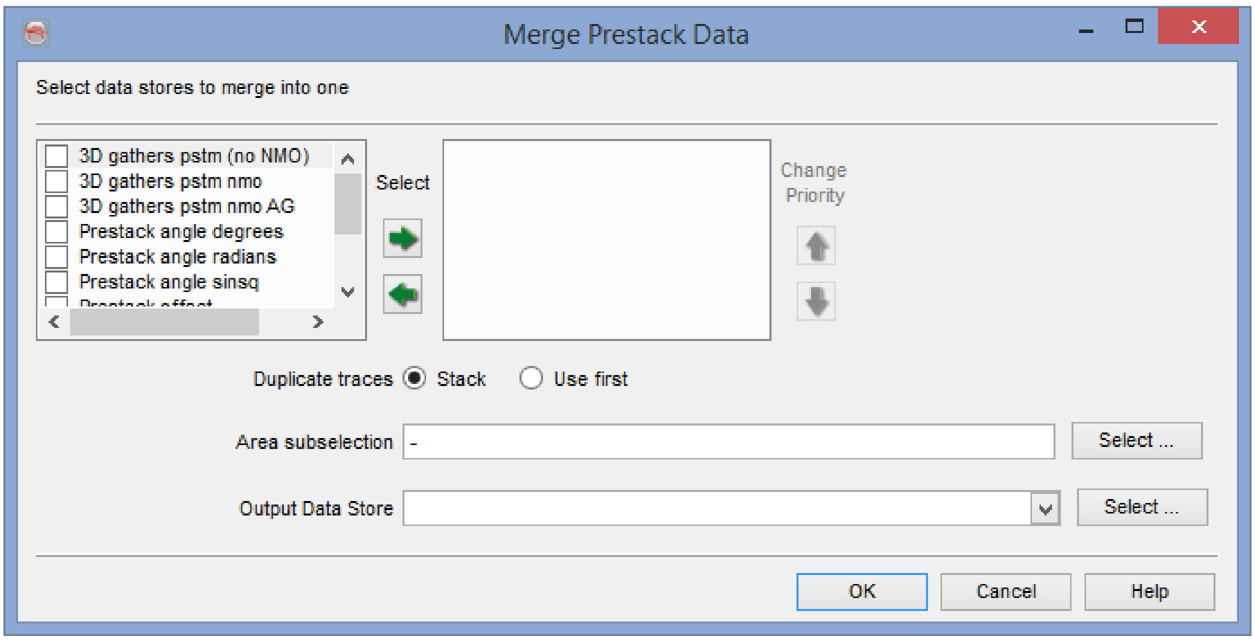
This window is opened via *Survey--> Manage--> Seismics Prestack--> 3D...*



Most options are common to the other managers: change file location, rename, lock, delete. '*Notes*' may be anything of interest to the survey and may be added to, edited and saved multiple times.

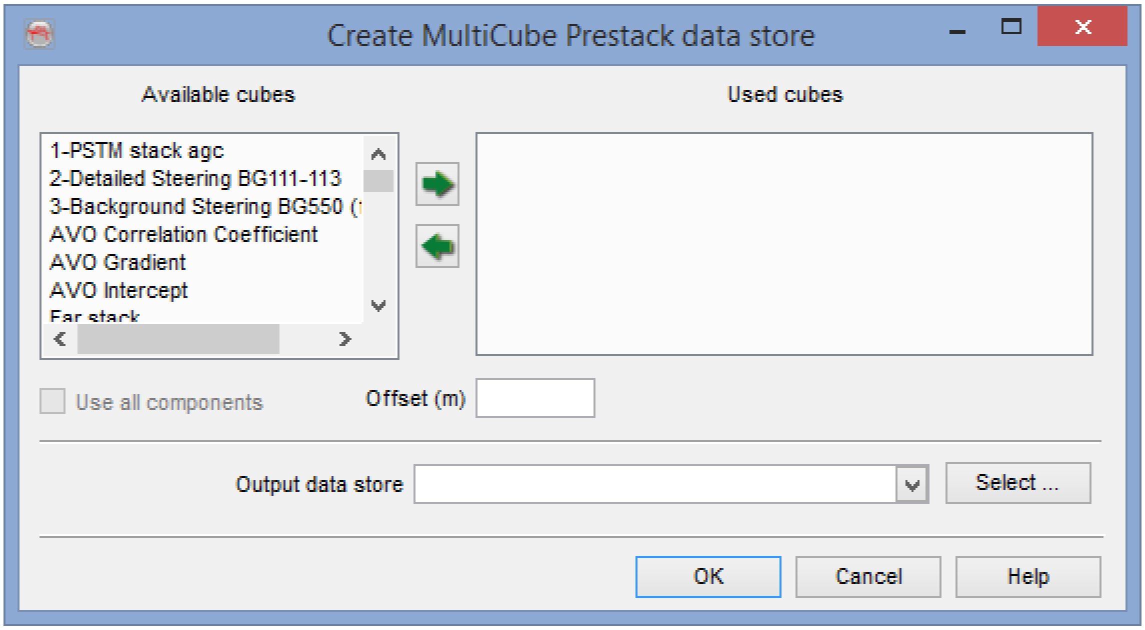
The options copy cube and merge blocks of lines work similarly to the 3D post-stack seismic manager.



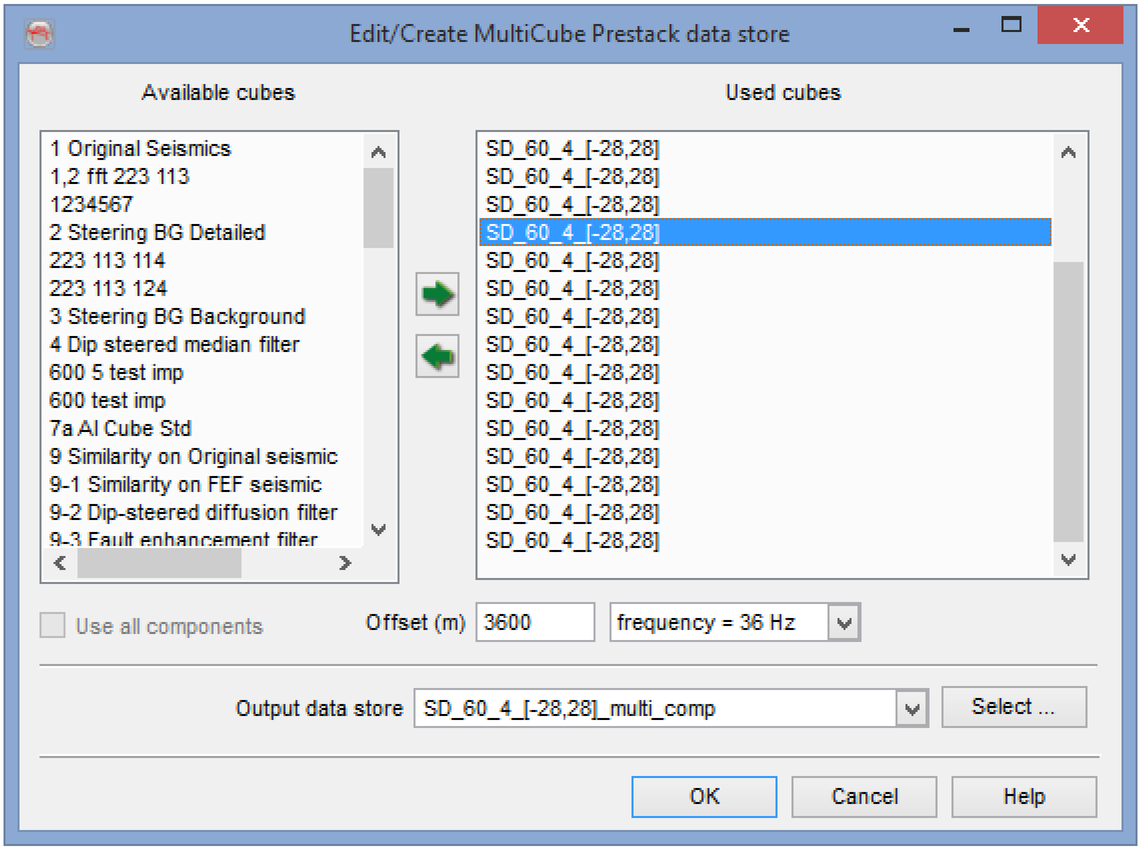


Pre-stack data stores are present on the disk in a folder of the same name within the survey ("Seismics" sub-folder). This folder contains one file per inline for quicker access, with auxiliary files. The manager will display information about the entire pre-stack data store: Folder name, number of files etc.

A pre-stack specific option, accessible via , allows creation of a pre-stack data store from two or more post-stack volumes. This can be used to create pre-stack data from a partial stack volume for AVO attributes extraction. A specific offset must be set in front of each volume.



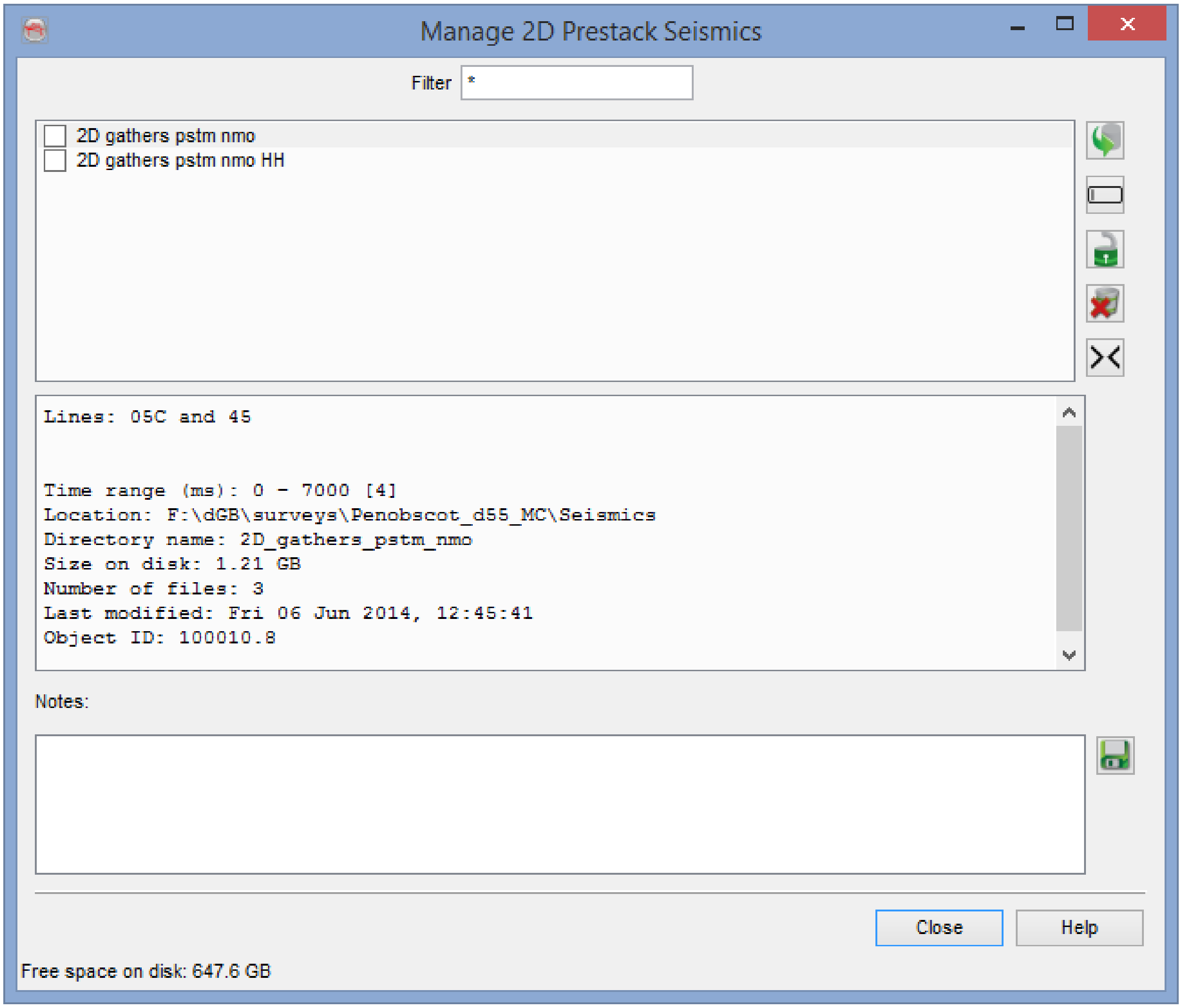
This option may also be employed in creating a multi-component cube for attributes with more than one component. For example, using this option, a user may create a multi-component Spectral Decomposition cube with each of the included frequencies given a pseudo-offset value. In the example (below), a multi-component Spectral Decomposition cube has been created, and for simplicity, the pseudo-offset used is a multiple of the frequency component. [The actual value used in these pseudo is irrelevant in this case, affecting only the width of the pre-stack display (which can be altered by right-clicking on the pre-stack displayed in the scene and choosing '*Properties...*']



Note: No new file is written to the disk. Therefore deleting a post-stack volume used in the pre-stack data store will cause problems. Please use same option to remove or modify the previously set multiple volume selection.

Manage 2D Prestack

This window is opened via *Survey--> Manage--> Seismics Prestack--> 2D...*

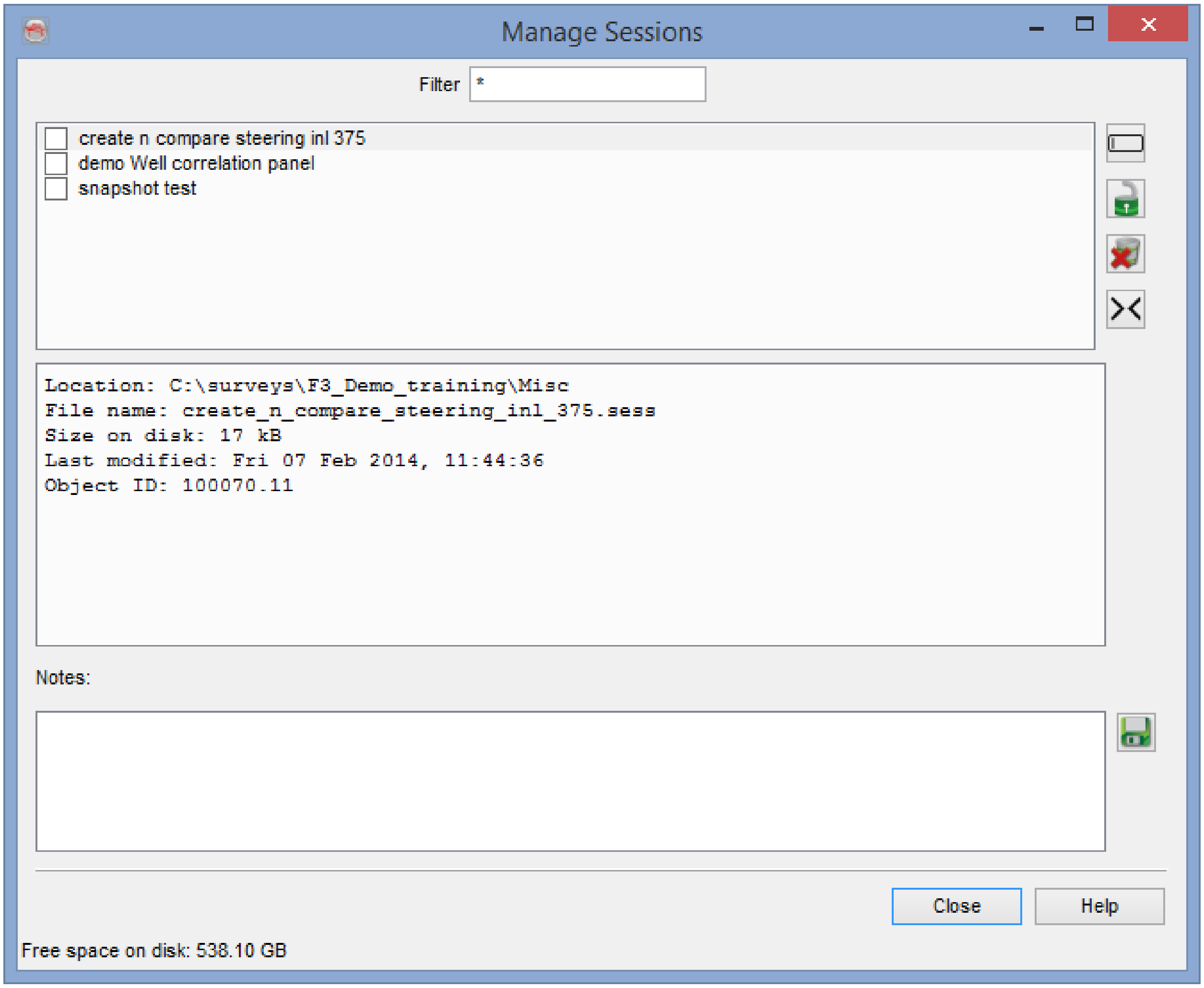


Options found here are common to the other managers: change file location, rename, lock, delete and set as default. '*Notes*' may be anything of interest to the survey and may be added to, edited and saved multiple times.

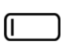
Manage Sessions

Sessions in OpendTect are generally used to save and to retrieve the specific settings of a scene. This can help the user to resume work from previous settings.
These sessions can be managed via: *Survey* > *Manage* > *Sessions...*

Sessions will save all settings of the displayed elements, and they can be saved/restored at any time from *Survey* > *Session*.



The following options are available:

 Rename

 Toggle read-only on/off

 Delete

 Set as the default object for its kind.

The '*Notes*' box is a free-text field where you may add notes related to the session, if desired, and save them.

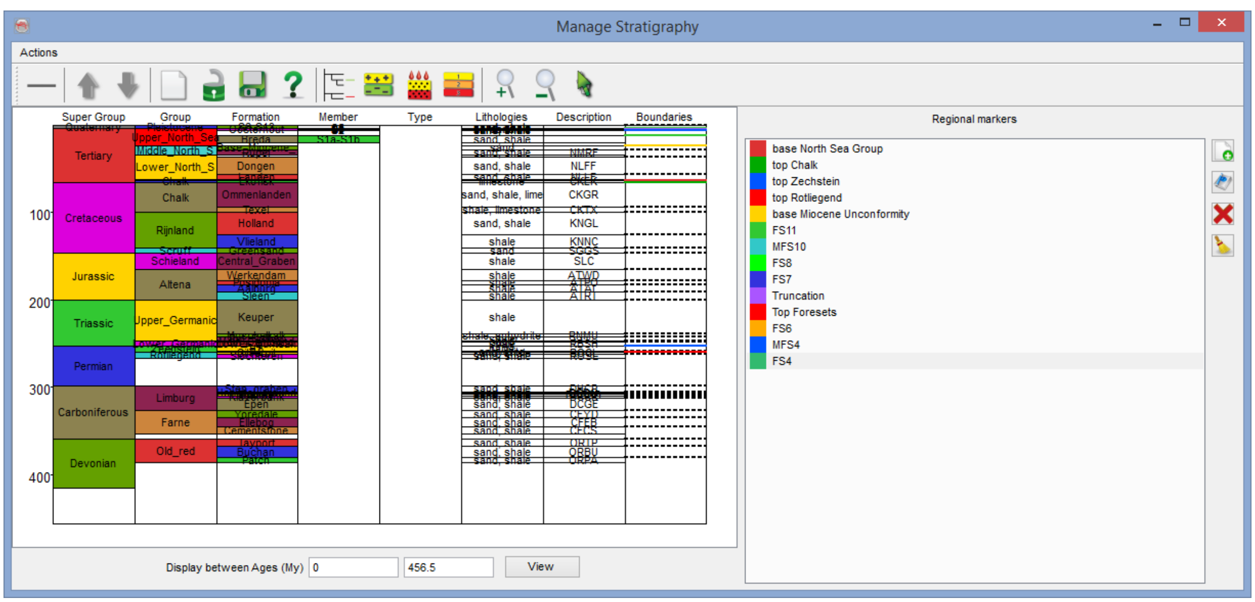
Manage Stratigraphy

The *Manage Stratigraphy* window can be launched with the icon from OpendTect Manage toolbar or via *Survey > Manage > Stratigraphy...* This window is designed to arrange the stratigraphic markers and the geological sub-units. It is used as base for the Layer Modeling.

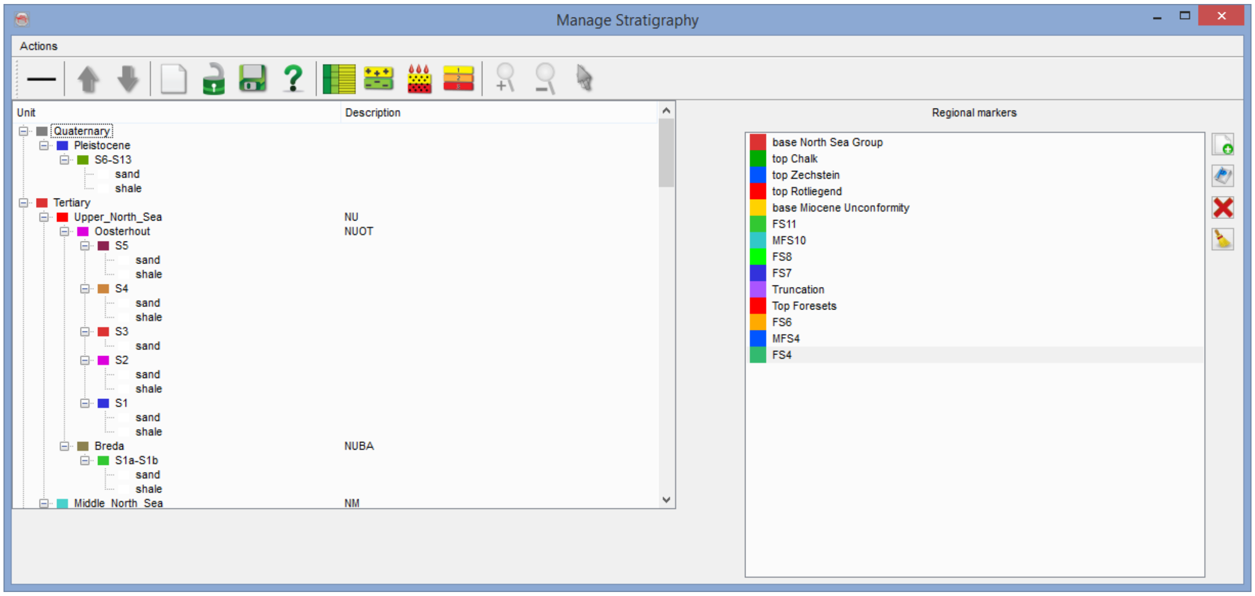
The first time you open the manager, a pop up window gives the options to either: 1) build a new stratigraphy from scratch or 2) to open an existing one (North Sea or Simple Reservoir). These two saved stratigraphy description are saved by default in another type of format. If edited, the edited version will be saved as classical stratigraphy description. Once the selection has been done, it is set as default. To re-access the selection window click on the icon to create/open a new description.

The user can create a specific information about the project and the different regional markers of his/her interpretation. This window is organized as units/sub-units bounded by different stratigraphic markers. Markers are assigned to the category the most on the right of the stratigraphic column. Depending upon user's description, markers can have the same name as seismic horizons or well markers and the units the names of epochs/eras.

To start, the user has two ways to display the stratigraphy tree: the time view and the tree view. The time view is chosen to display the absolute geological time while the tree view shows an overview of unit/sub-unit as leaves.



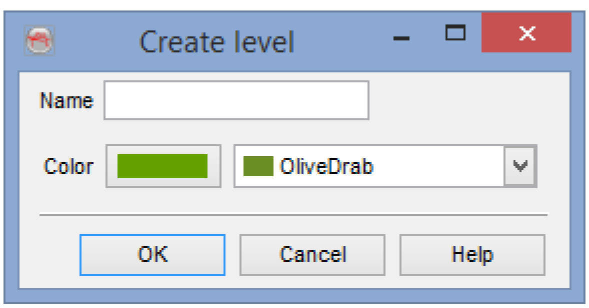
Stratigraphy window: The time view



Stratigraphy window: The tree view

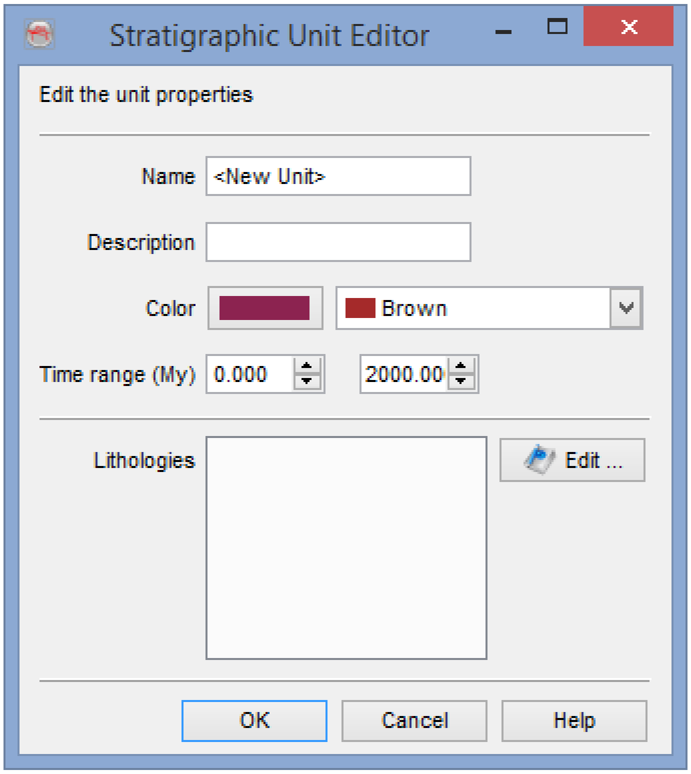
1- Regional Markers:

The regional markers can be associated to boundaries of stratigraphic units. These markers are added on the right-hand panel. Right click on ---*None*--- in the regional markers panel. And in the pop-up menu select the *Create New*... option. In the *CreateLevel* window, write an appropriate name for the stratigraphic marker and optionally provide the color. Press *Ok* to add the marker. They should have a coherent name. In the Well Marker Manager, markers can be linked to a regional marker and will be then renamed after it.. The inserted marker can then be assigned/linked as a top and base of the stratigraphic unit.



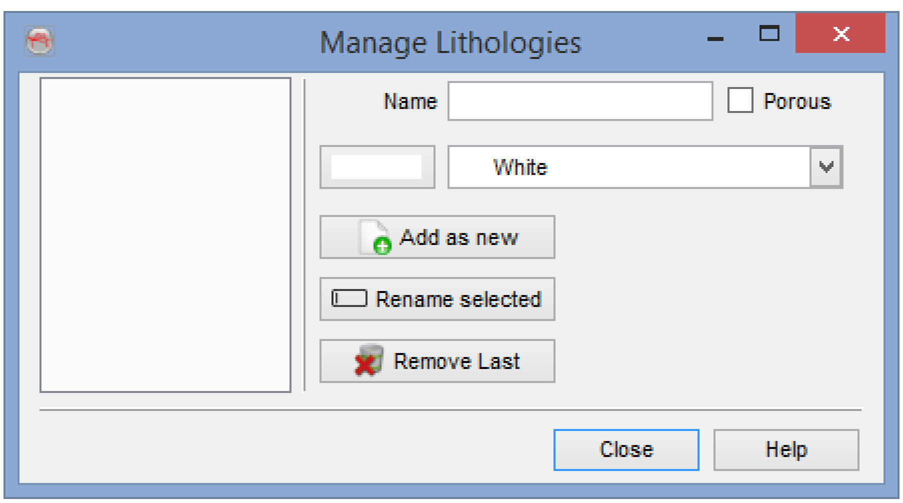
2- Stratigraphic Units:

On the left hand side of this window, the units are classified in a way that the top and base of each unit belong to certain marker. For the initial unit, right-click on *<Click to Add>*, the stratigraphic unit editor will pop up:

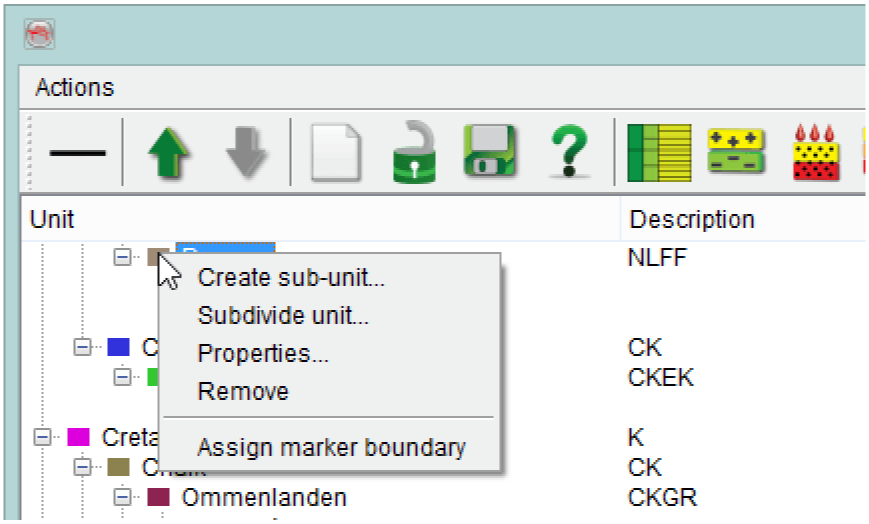


In this window, give a name of the unit area, the description, color, the age and lithology. The minimum requirement for creating a new unit is simply to define the name.

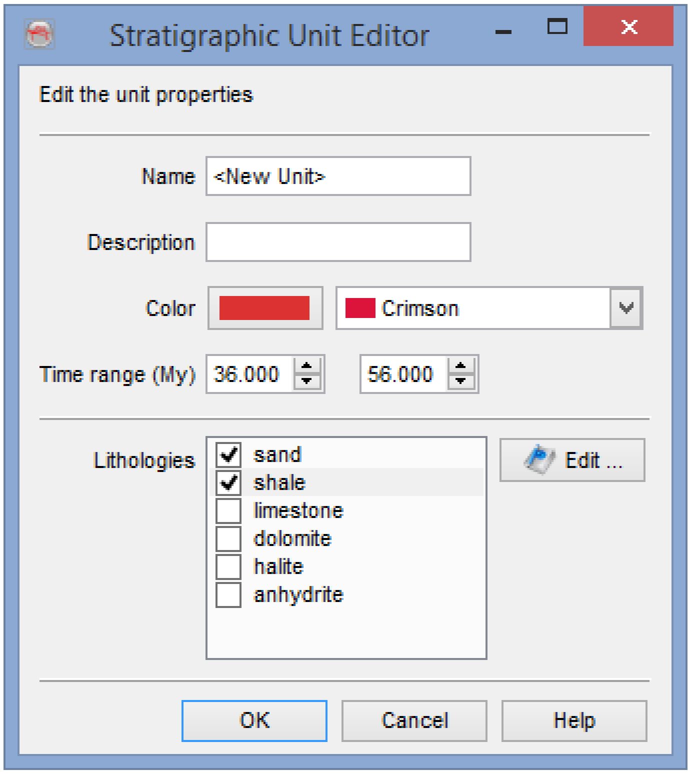
To add a lithology: Click on "*Edit*" then give the name, and optionally specify porosity then *Add as new*, click on *Ok*.



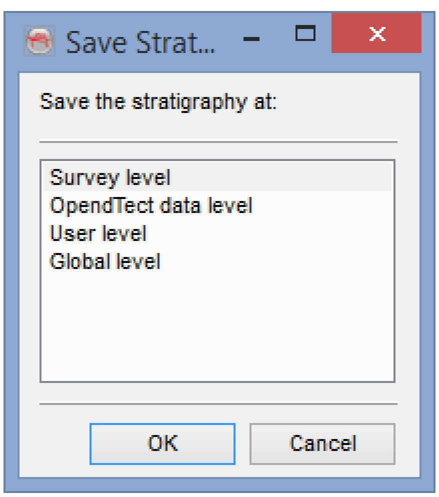
To add a sub-unit, right-click the unit name and select *Create sub-unit*, and define it in the same manner as a unit. Description and lithology of the unit can be added now or edited later.



Stratigraphic unit properties: Properties such as unit/sub-unit description and lithology can be defined or edited by right-clicking on the unit/sub-unit name and selecting Properties. A unit/sub-unit specific lithologic name can be entered directly into the Lithology field. For lithologies that may occur in multiple units/sub-units, a lithology can be defined and made universally available by clicking the Select button next to the Lithology field. In this Select Lithology window, the lithology type can be named, and added to a list that will be made available for all units/sub-units in this session. (Depending on your Save settings, these lithologies can be available outside of this session.) These options can also be defined when the unit/sub-unit is first added.

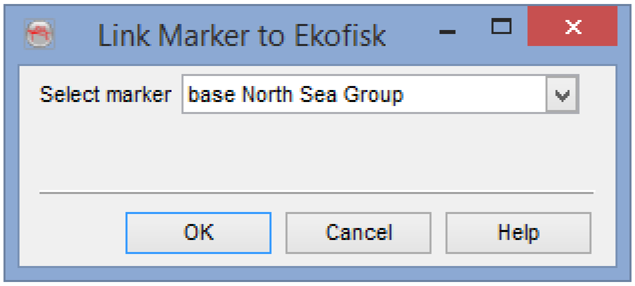


Save as: The defined stratigraphy can also be saved at different levels, e.g. *Survey levels*, *OpendTect data level, User level*, or *Global level*. For instance, if it is saved at *Survey level*, the stratigraphy will only be available for this survey. Alternately, if it is saved at a higher level, it will not be limited to only the survey in which it was defined.



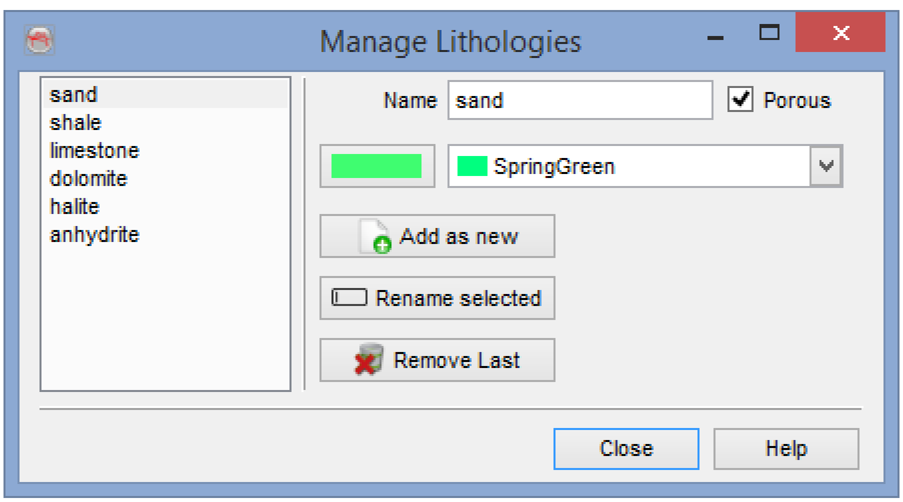
Assign Marker boundary:

This option links the regional markers with stratigraphic units. Right-click on boundary or unit/sub-unit then click on *Assign marker boundary* select regional markers top and bottom that are the appropriate boundaries for the unit/sub-unit.



Manage Lithologies

The *Manage Lithologies* window can be launched by clicking on the  icon in the main *Manage Stratigraphy* window. It allows to define the list of lithologies possibly present in the stratigraphic column. This list is then available when defining the different units of the stratigraphy. For Layer Modeling, the lithologies listed for each units are used in the layer description.



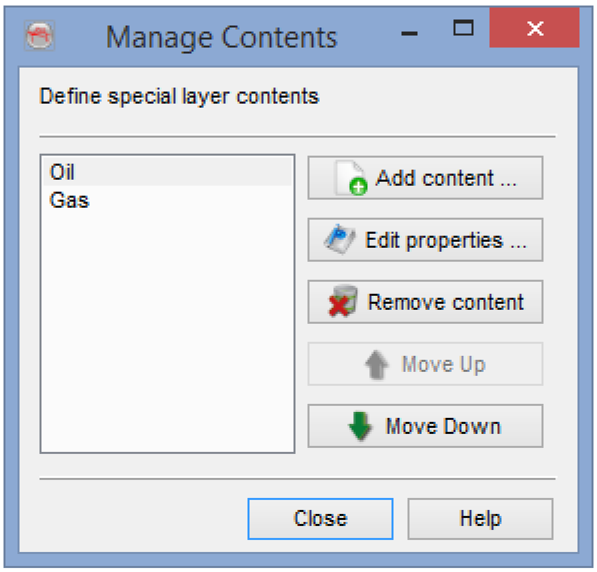
Following tasks can be performed:

- Lithologies can be added or removed
- Lithologies can be renamed

- Lithologies can be ascribed various colors
- Lithologies can be specified as Porous/Non-Porous by toggling on/off Porous (this is used if fluid substitution is carried out in further analysis with SynthRock plugin)

Manage Contents

Manage Contents can be accessed by clicking on the icon in the *Manage Stratigraphy* window.



This option is used to define a set of fluid contents. Afterwards, fluid(s) from the list can be assigned to lithologies for each layer when defining *Layer properties* for Layer modeling.

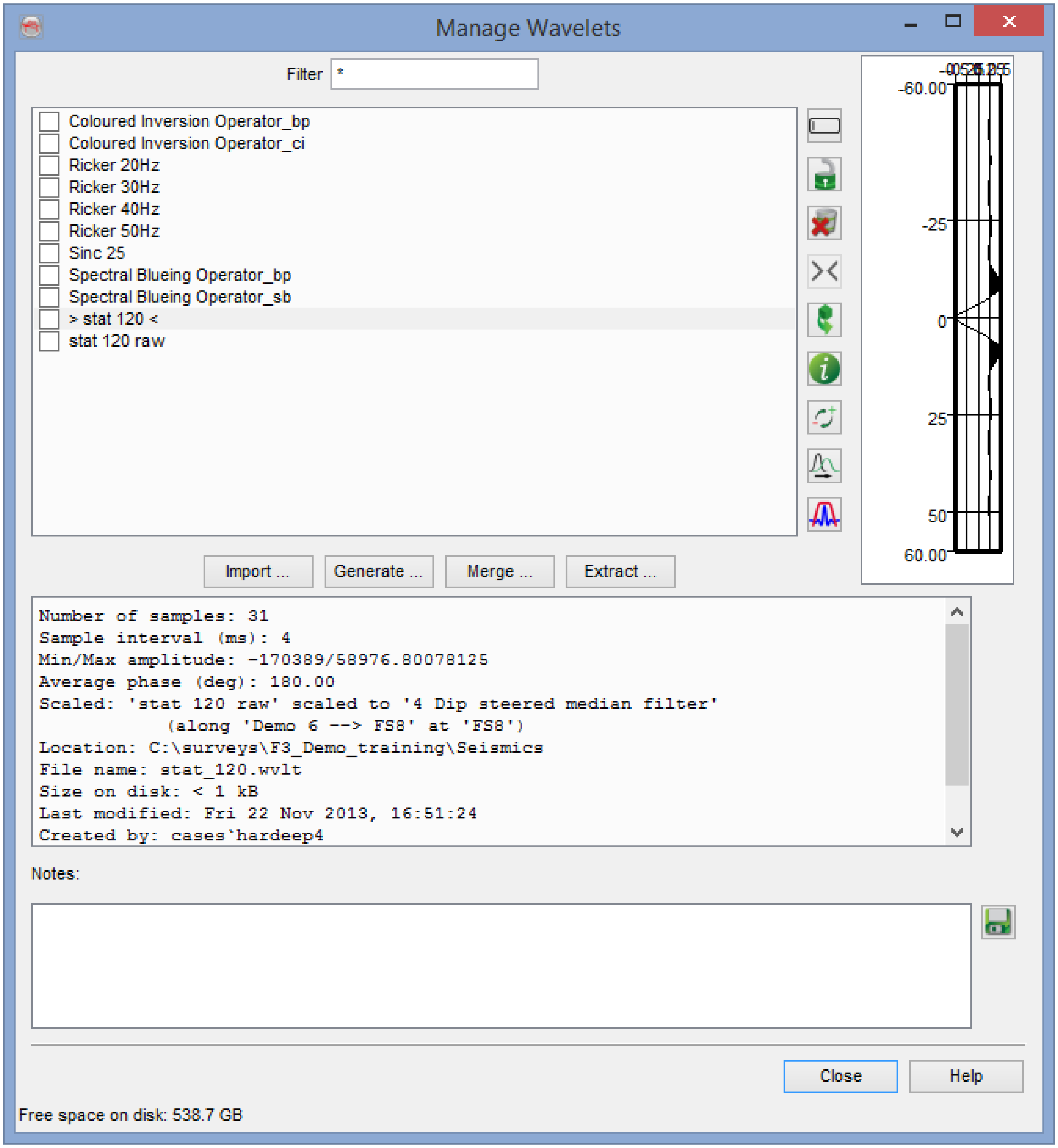
Layers & Synthetic Modelling

The icon starts the Layer/Synthetics modeling feature.

Manage Wavelets

This window is available from the *Survey* > *Manage* > *Wavelets...* menu and from the  icon. It provides management tools for wavelets. The left panel shows the available wavelets. The selected wavelet is visualized on the right panel. The storage information of the active wavelet is shown in the lower panel. The following actions can be performed:

- Import a wavelet
- Generate a synthetic wavelet
- Extract from seismic data (2D/3D)
- Merge several wavelets



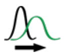
Options:

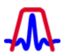
Alongside the standard '*Manage*' options (*Rename*, *Lock*, *Remove* and *Set as Default*), you may also, via this window:

 Get a wavelet from another survey

 Display a wavelet's properties dialog

 Change polarity

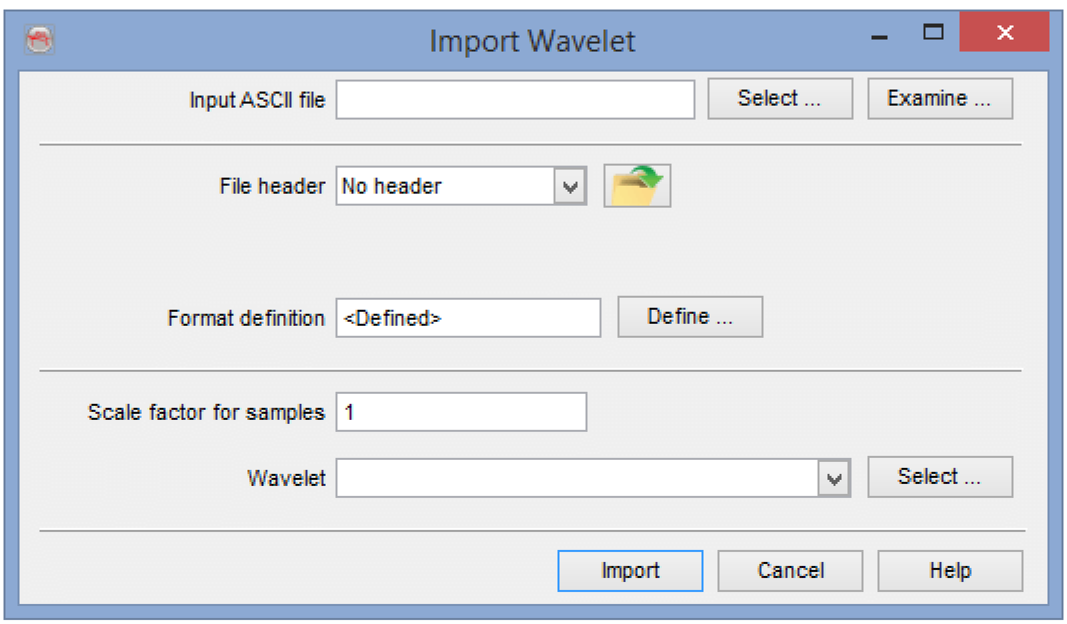
 Manually rotate a wavelet

 Taper a wavelet

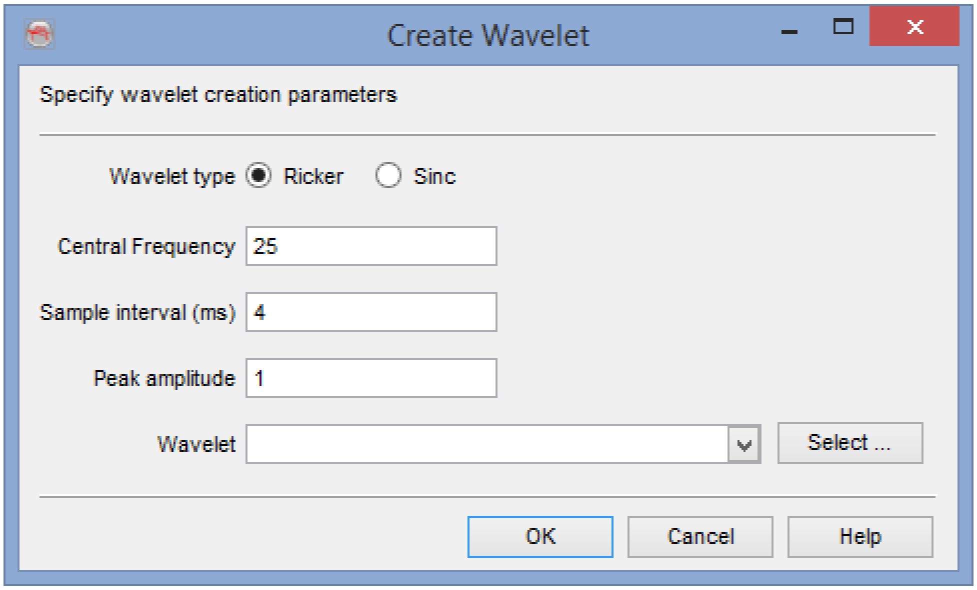
Note: The Filter is used to filter-out the objects with selected names. For instance, to display all wavelets that start with letter W use "W*".

Import Wavelet

When clicking on the *Import* button, the import wavelet dialog box pops up. Please follow the instructions in Import Wavelet
section.



Generate Synthetic Wavelets



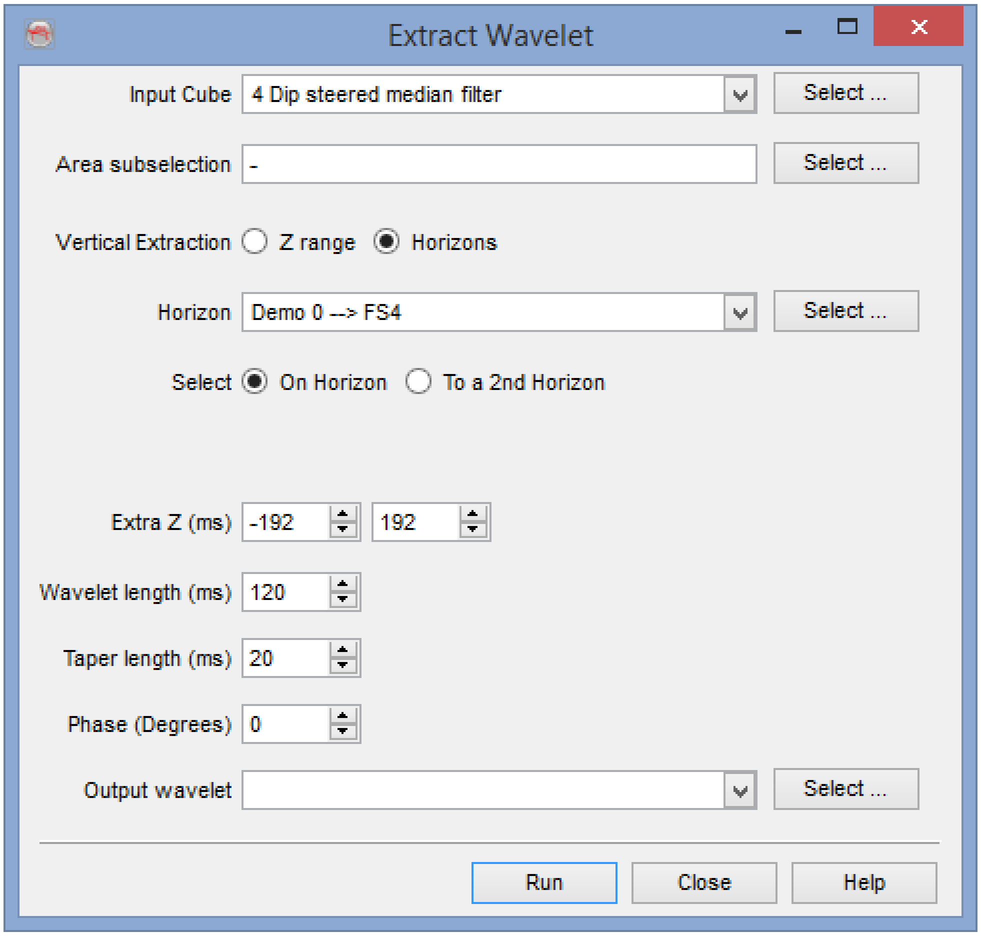
Generate a wavelet

Two types of synthetic wavelet are available - "Ricker" and "Sinc".

Statistical Wavelet Extraction

Statistical wavelets can be extracted from the seismic data.
The User first needs to choose the input seismic, i.e 3D volume or 2D line.

If 3D seismic is selected, the following window pops up:



It is recommended to use a sub-selection of the seismic data, e.g. every 10th inline/cross-line, and to use horizons to guide the extraction. The extract length of the seismic data should be at least 1 second TWT.

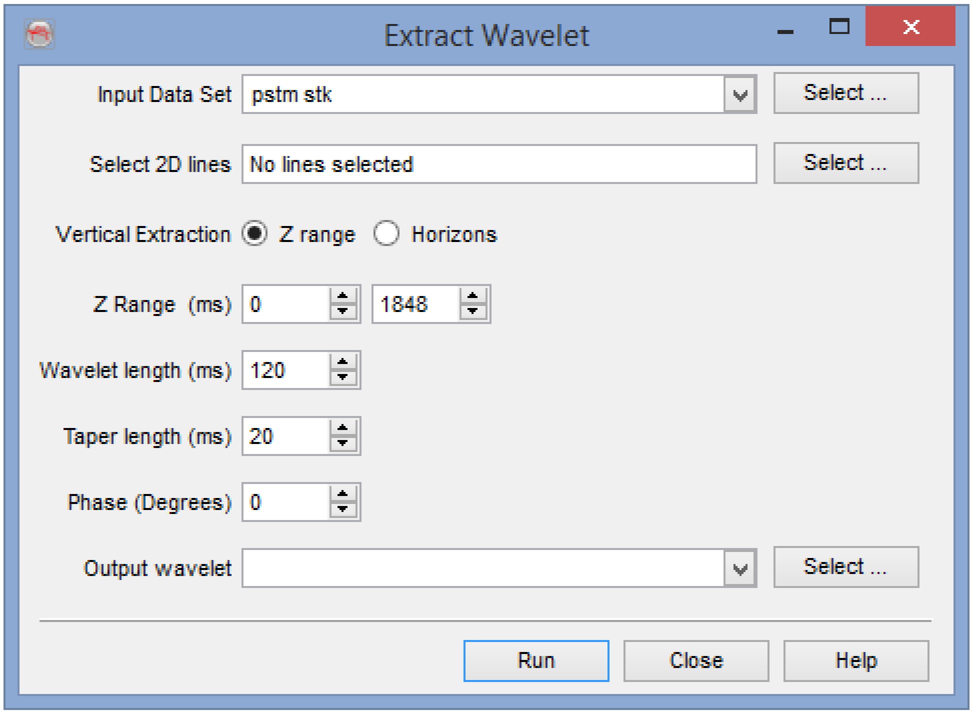
The wavelet length should never be too small (min 50ms), or too large (200ms max). A rule of thumb is that the first side lobe should be fully contained in the wavelet.

The extraction is performed using the following workflow:

1. Seismic traces are extracted and tapered
2. The auto-correlation of the seismic traces is computed, using the length of the desired wavelet
3. The frequency spectrum of the auto-correlation is computed.
4. The square root of the modulus of the frequency spectrum is taken, the zero Hertz component is muted to zero.
5. The inverse FFT is computed.
6. The zero phase wavelet is the real part of the inverse FFT output

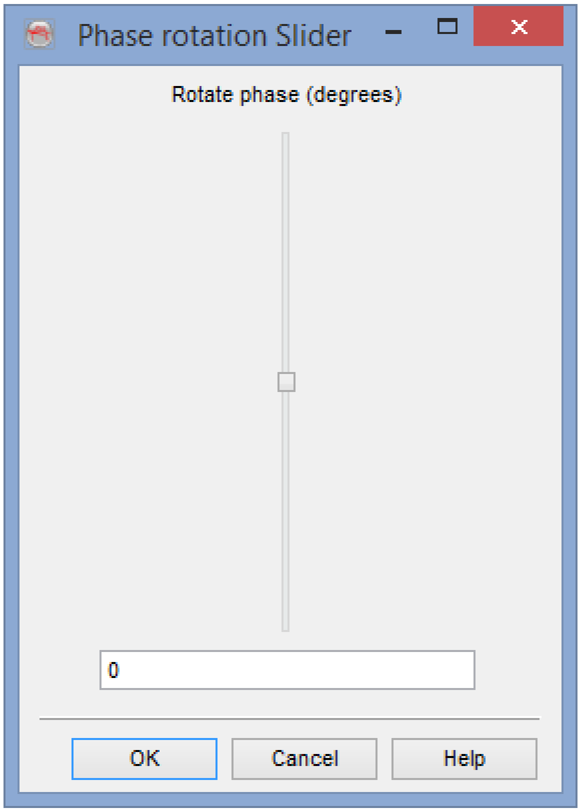
The output phase rotation cannot be set in the current version. It is being implemented.

The Wavelet extraction in 2D line is shown below:



Rotate Phase

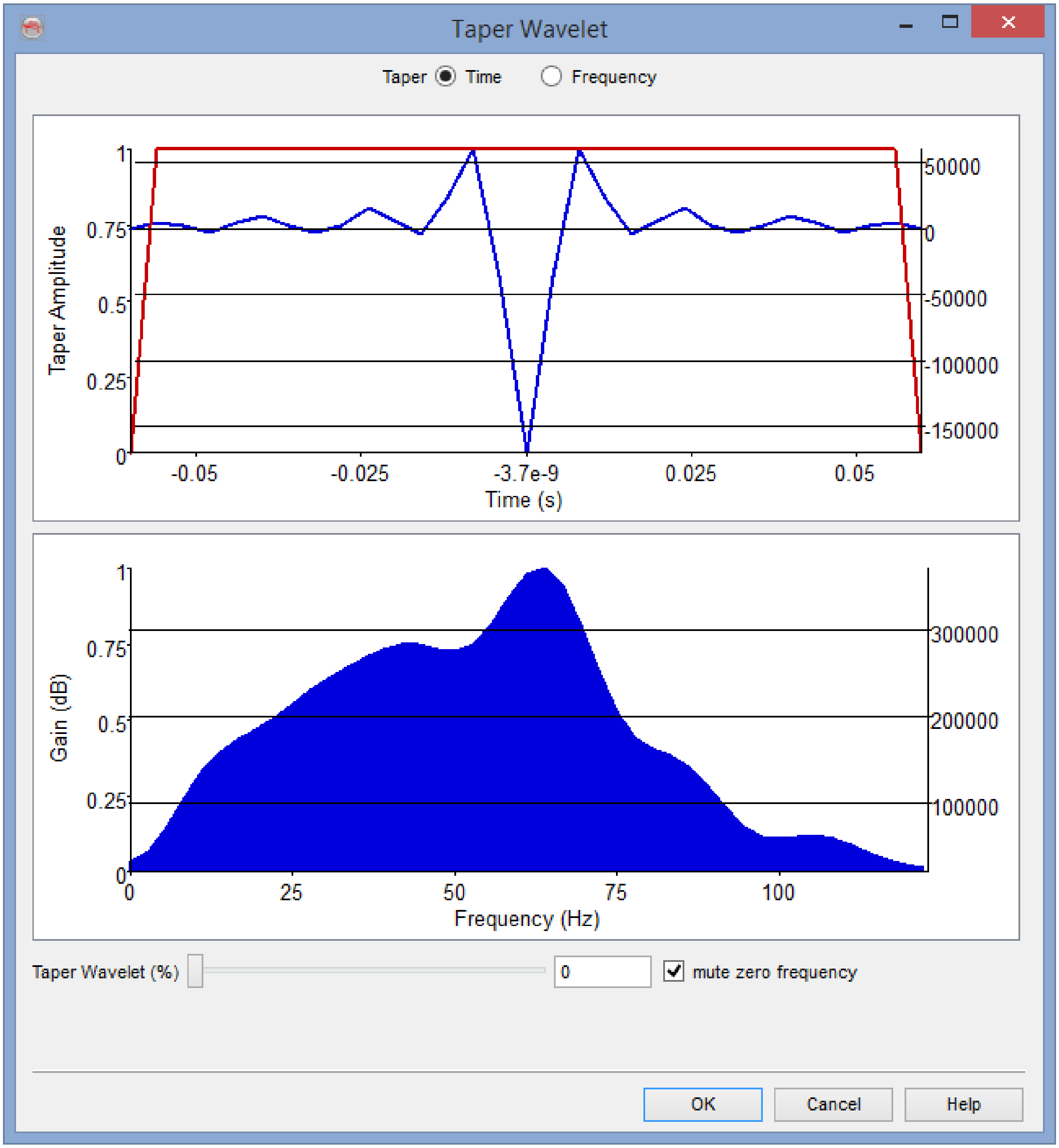
The phase of wavelets can be altered and saved using the following slider:



The new phase will be set when pressing "Ok".

Taper a Wavelet in Time or Frequency Domain

A wavelet tapering window is launched by pressing the icon from the wavelet management window. A wavelet is tapered in time or a frequency domain, depending what is selected from the top of the panel (see below).

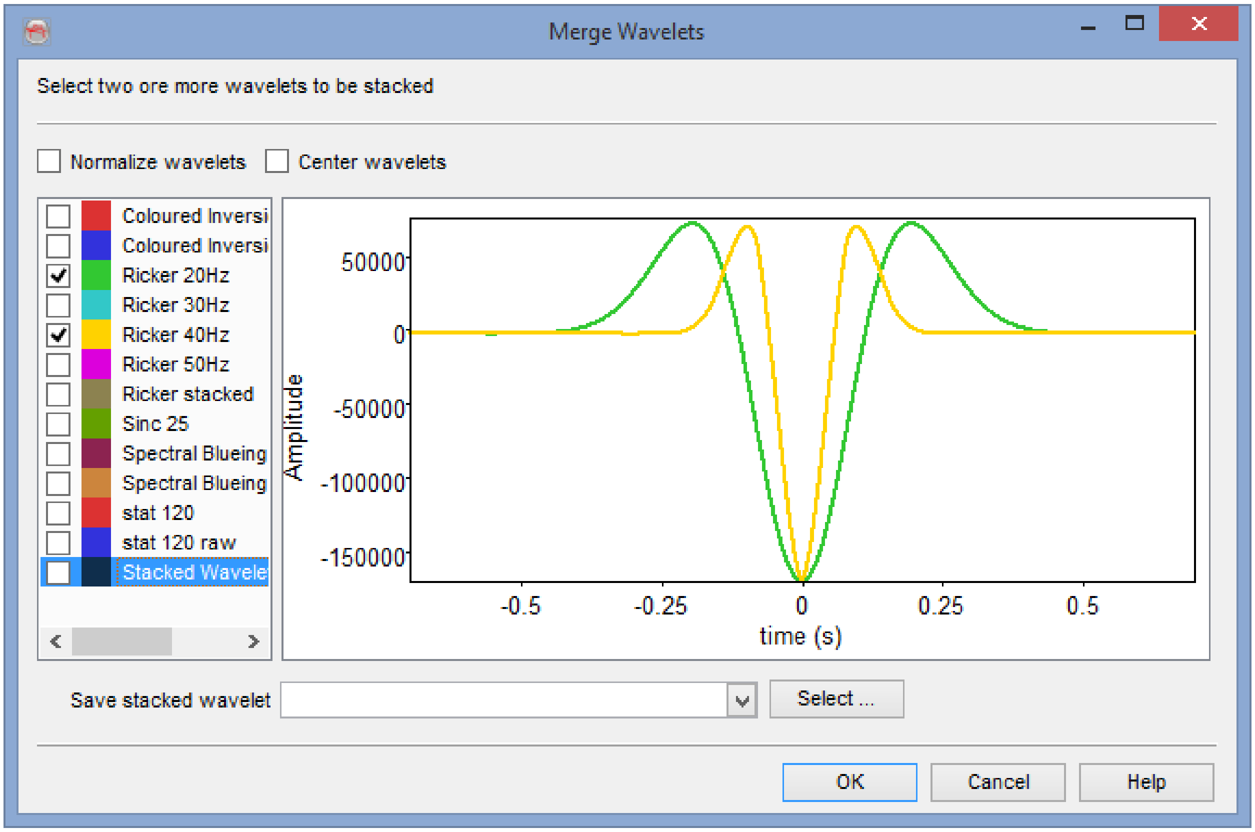


In time domain, the selected wavelet is tapered by selecting a tapering percentage (%), which is set from the slider available at the bottom of the window. This is done by moving the slider left or right. Additionally, the amplitudes at zero frequency can also be muted by setting check to *mute zero frequency* check box.

In frequency domain, the tapering can be applied to both ends of an amplitude spectrum, i.e. high and low frequencies. This is applied with a given slope (dB/Octave) value and placing the slider to an appropriate min/max position (Hz). The red line in the amplitude spectrum shows the resultant tapering pass, which is updated according to the given settings.

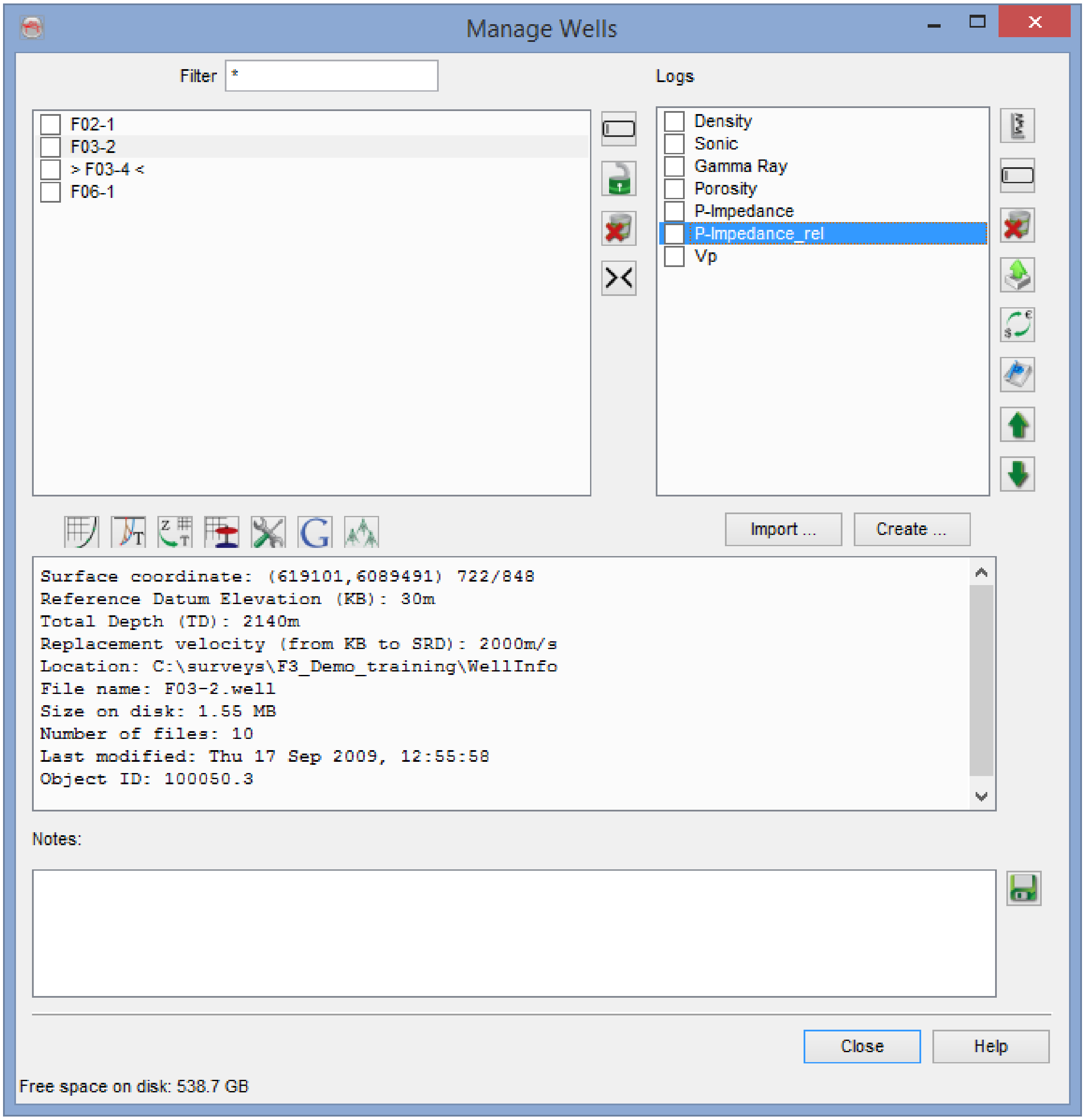
Merge Synthetic Wavelets

Two or more wavelets can be stacked using this option. The wavelets can be 'Normalized' and/or 'Center' at maximum amplitude/energy.

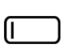


Manage Wells

The Well Management window can be open from *Survey* > *Manage* > *Wells...* or from the  icon.



The available wells imported in the project are listed on the left. Wells can be renamed, locked, removed or set as default with the buttons in the centre of the window:

 Renames the selected well

 Toggles the well to be read only(locked)/editable(unlocked)

 Removes the selected well

Sets as a default

The well track, checkshot and time/depth model can be loaded and/or edited using the lower left row of icons:

Well Track Editor

Checkshot Editor

Depth/Time Model editor

The well log tools can be used to remove spikes, to smooth and to clip the logs for the loaded wells.

Well log tools

Markers can be imported and managed from the following icon:

Markers editor

Wells can also be exported to be seen on Google Earth using the icon. It is also possible to either import or create Multiple single wells from the icon .

When selecting a well on the left list, the loaded logs for this given well are listed on the right hand side. They can be renamed, removed and exported with the buttons on the right of the log list. The unit of measure of the logs can be checked or changed if needed (this does not affect the log values). The logs themselves can be moved up/down within the list. For removal and export, multiple logs can be selected.

Renames the selected log(s)

Removes the selected log(s)

Exports the selected log(s)

View/Edit unit of measure for the selected log(s)

Edit Well log

Moves up the selected log in the list

Moves down the selected log in the list

Logs can be imported from Ascii files, click on *Import*..., or created using mathematical expressions from the well management window, click on *Create*....

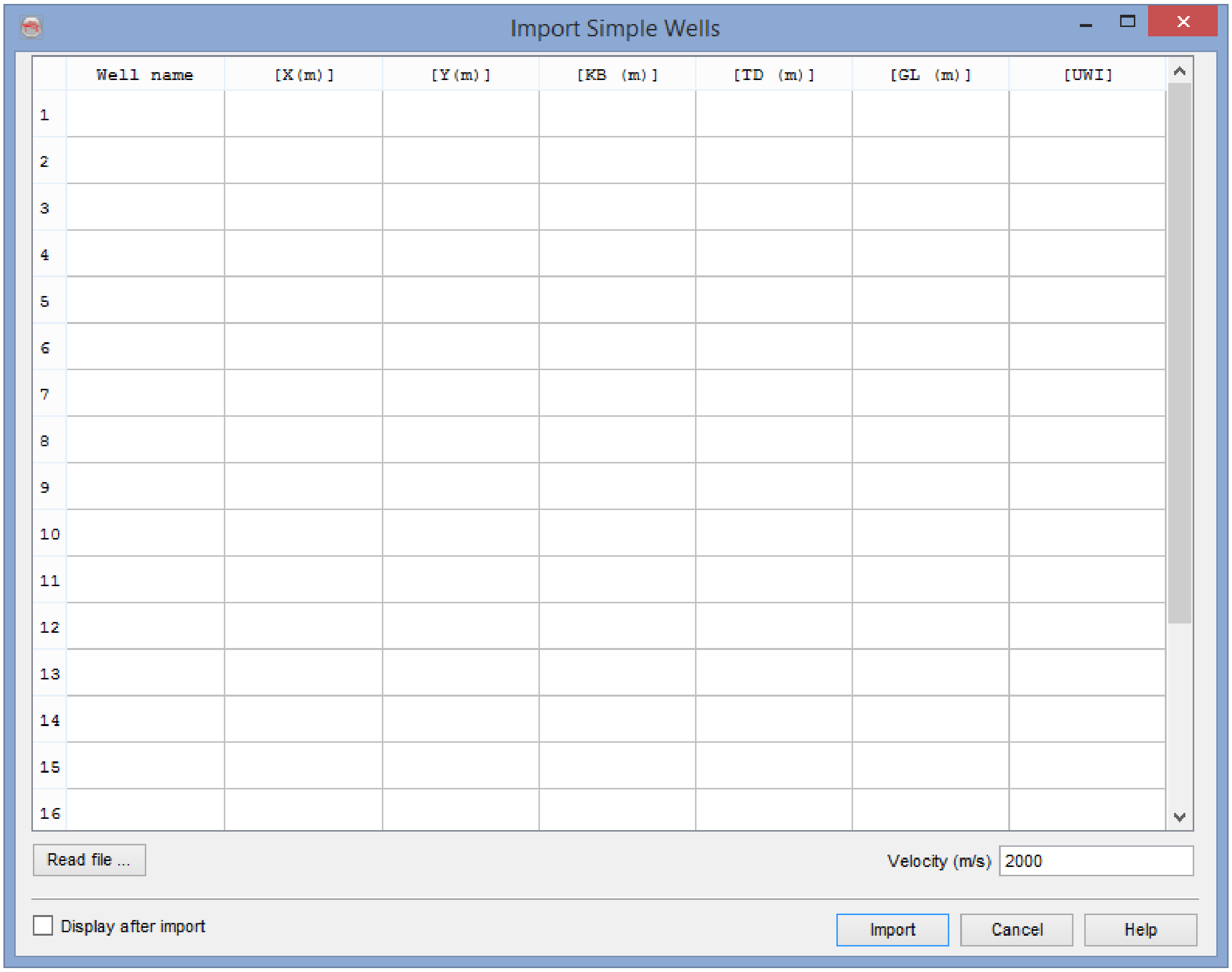
Other relevant information is indicated at the bottom of the window.

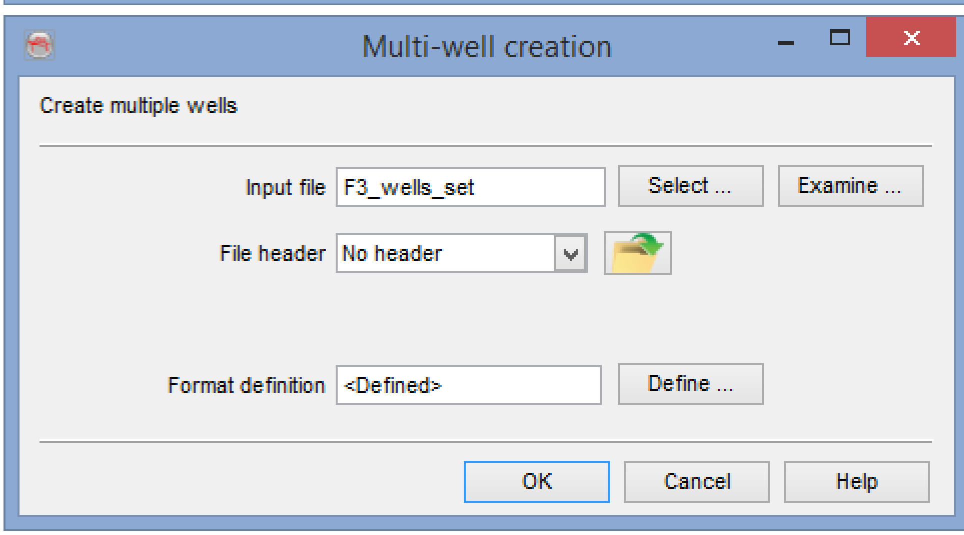
> **Note 1:** It is recommended to give logs the same name in all the wells. For example, the master density log should be called RHOB in every well. This enables the selection of one set of logs in all wells, ie: for use in the cross-plot tool. Please note that logs names and marker names are case sensitive during multiple selections.
>
> **Note 2:** The top Filter is used to filter-out the objects with selected names. For instance, to display all wells that start with letter W use "W*". This works only with text, not numbers or symbols.

Simple Multi-Well Creation

Multi-wells can be imported or edited through the *Import > Well* menu. This window contains editable fields. The new wells can be created by either importing them or entering directly the values and names. The *Read file* button can be used to import an ASCII file containing all well information.

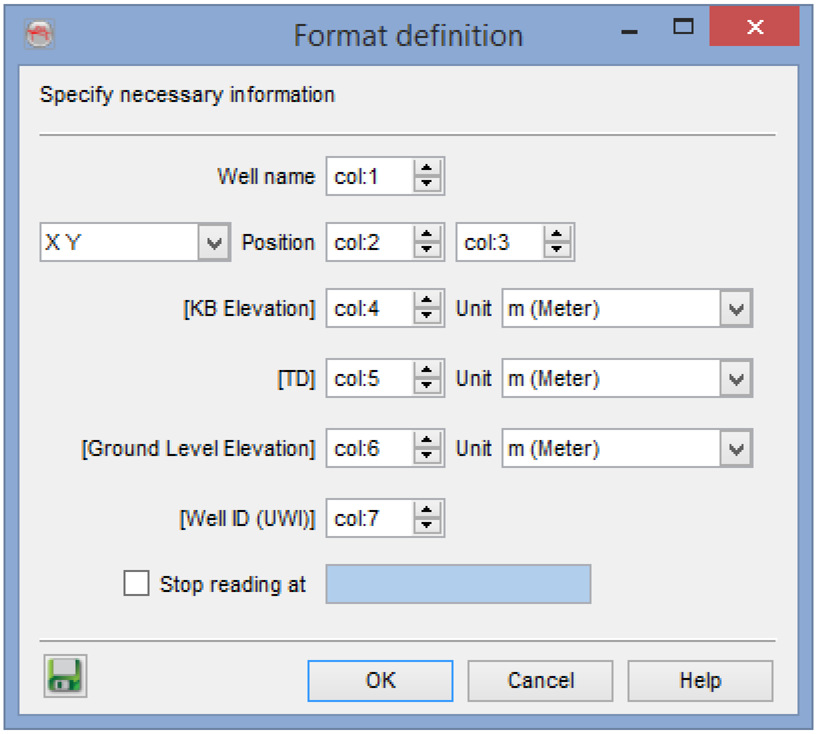
Select the input file (as shown below) and provide the appropriate format definition settings. To provide the format definition, the selected input file can be examined by pressing the *Examine* button. If the file contains the header lines, those lines can be eliminated by providing the file header information.



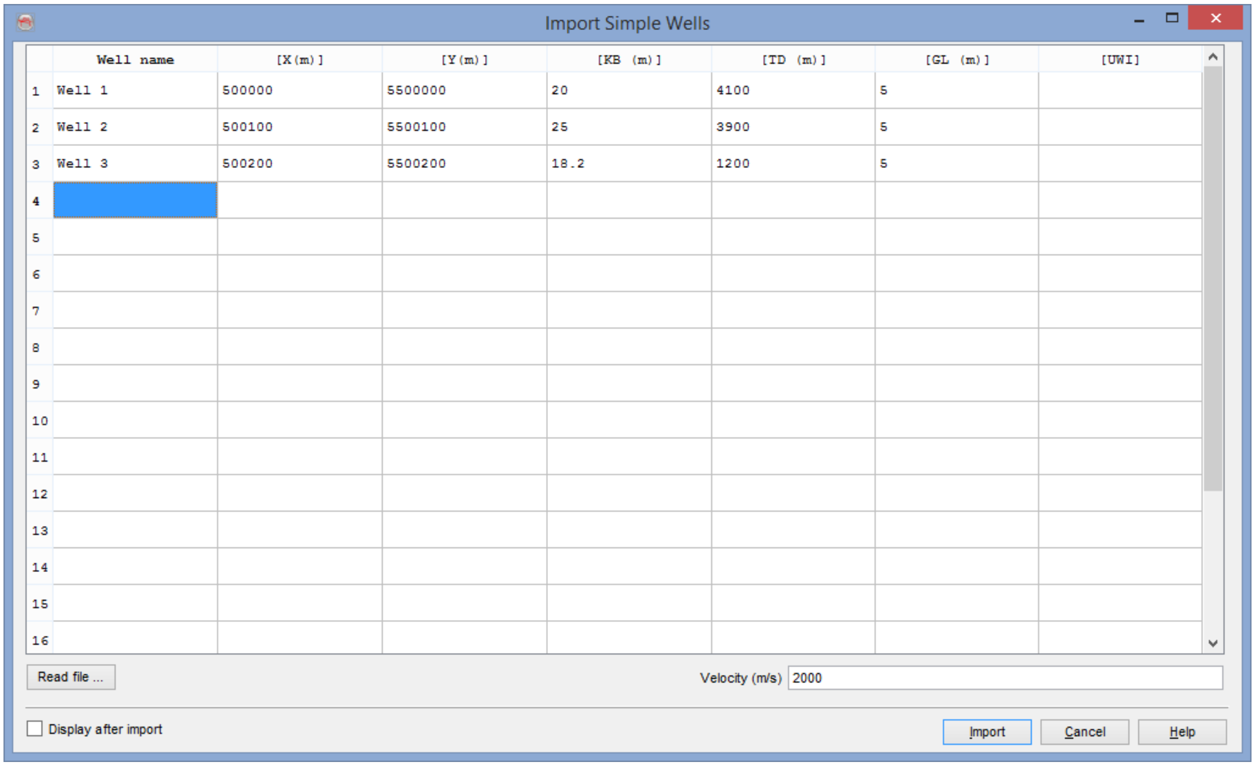


The file format definition is provided by pressing the '*Define*' button. In the format definition window, the default 'col:0' values can be modified according the the input file. When the correct file format is defined, the wells can be imported by pressing 'OK' button in the multi-wells creation window. By default the wells are loaded with a constant velocity.

The velocity data or the time-depth model can be provided while importing the [Time-depth model](#).



The simple multi-well file can now be imported and displayed after creation:

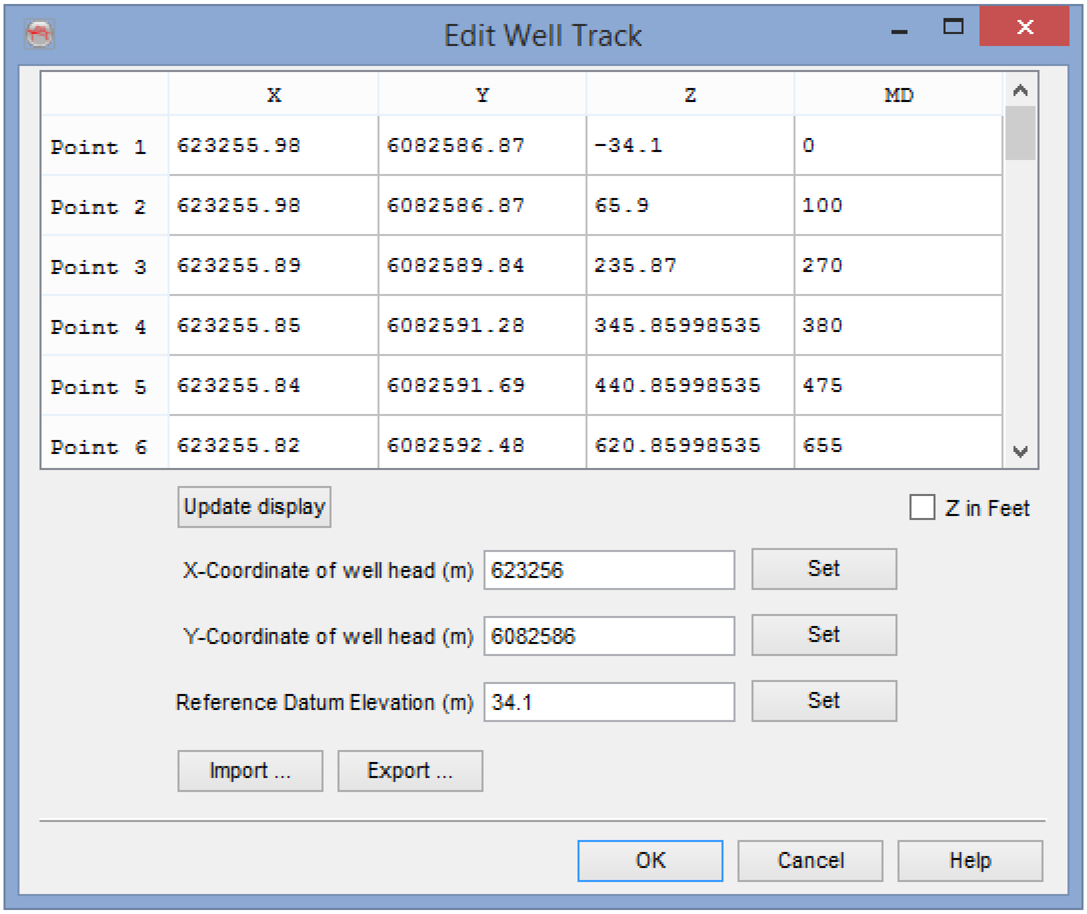


Well Track Editor

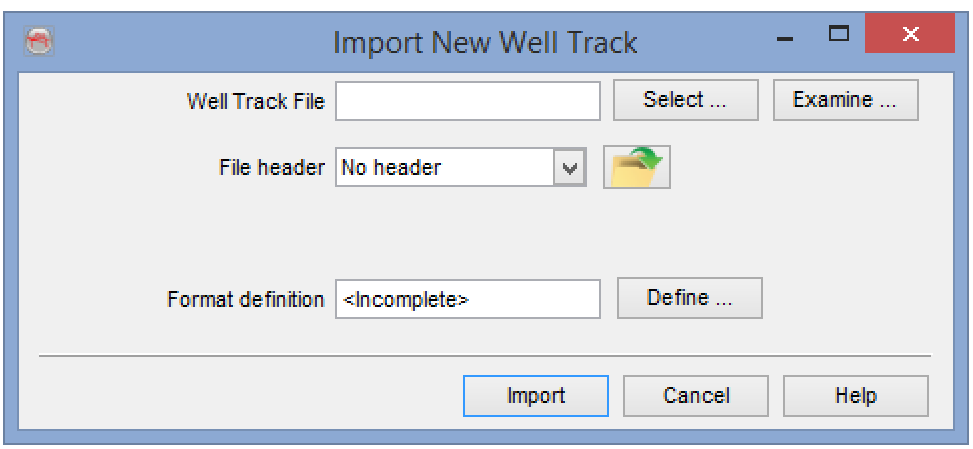
This table shows the import relation between the X and Y coordinates (first two columns), the TVDSS depths (Z, third column) and measured depths in the fourth column. This table is fully editable: Double-click on a cell to edit it, type a new number and press enter or select another cell. Please note that other values will be recomputed to reflect the changes.

The "*Update display*" button allows to update the displayed well track in the scene based on the modified table content. Optionally a whole new track file can be read from a file to replace the existing data, like during track import.

During edition the depths can be displayed in feet. Please note that this flag will be kept in the survey defaults and will apply in other windows. However it is only a display setting and the data on disk will not be affected.



The following window appears after having clicking on "*Read new*". The import settings are fully similar to that of the import step.



Checkshot and Time-Depth Models

Wells in OpendTect can have two different types of Time/Depth models:

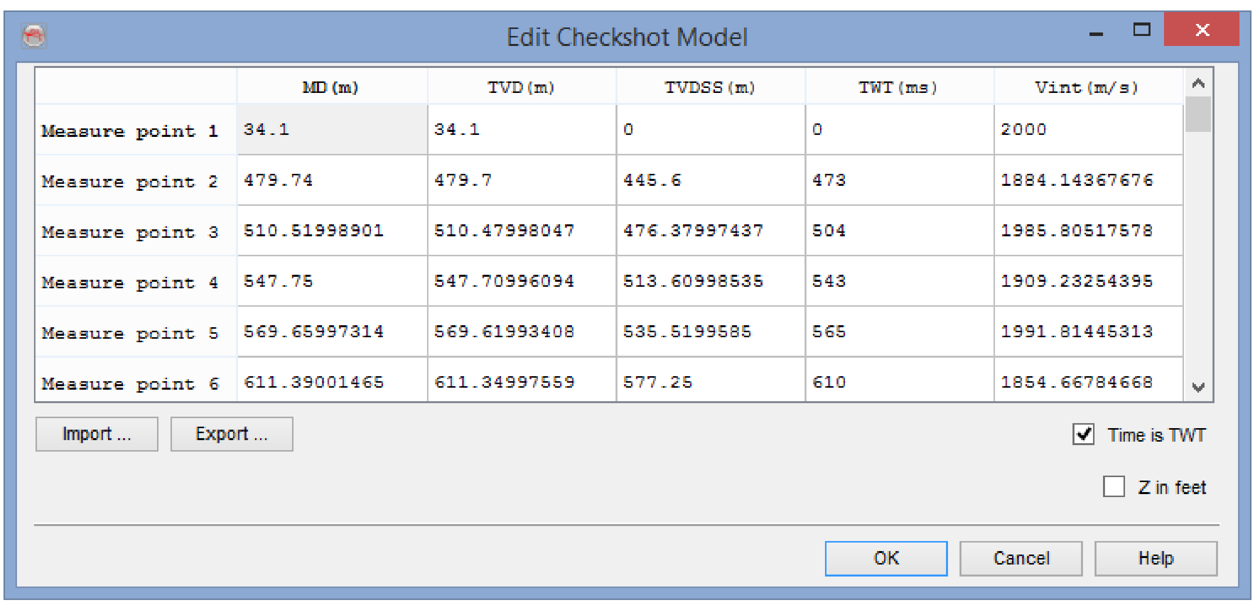
1. An optional checkshot model, often the first available time/depth model or a measured checkshot survey.

2. The Time/Depth model, that is always the active time/depth model for the well, used for data extraction and visualization.

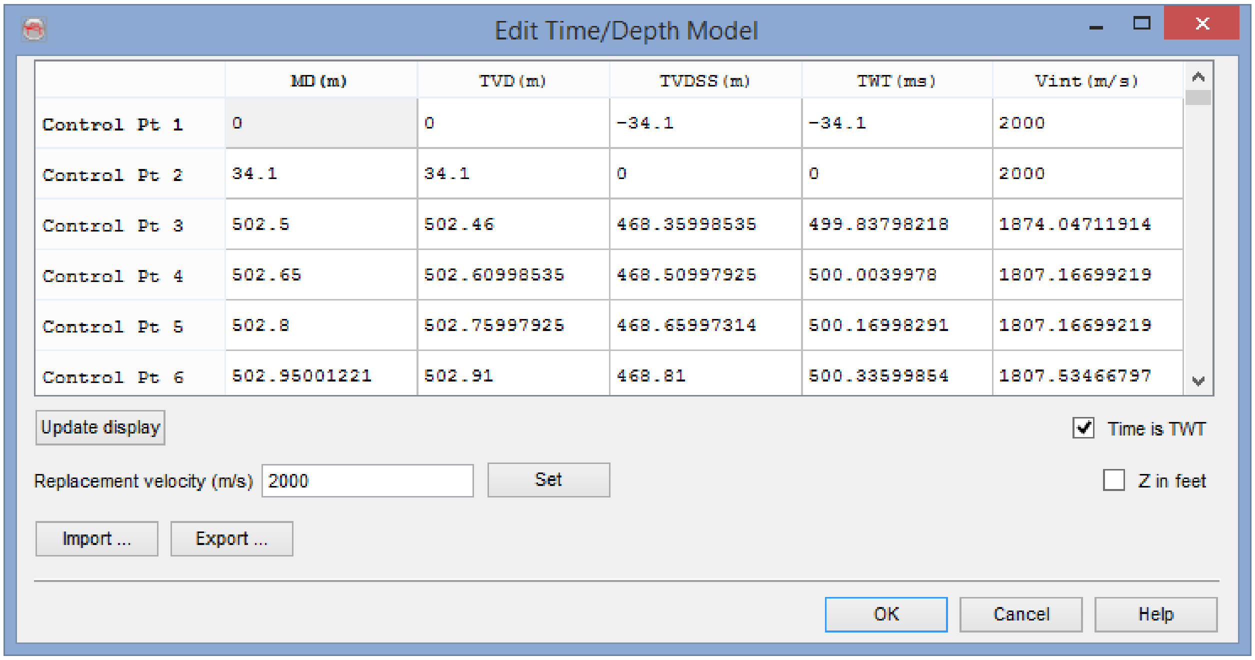
The main differences in usage are:

- If a Time/Depth model is flagged as being a checkshot model during import, the Time/Depth model will be a copy of the Checkshot model.
- If a Time/Depth model is not flagged as being a checkshot model during import there will be no checkshot model for this well.
- Checkshot models may be used during synthetic-to-seismic ties to constraint the output Time/Depth model. On the contrary the time/depth model provides the actual input mapping when starting the well tie.

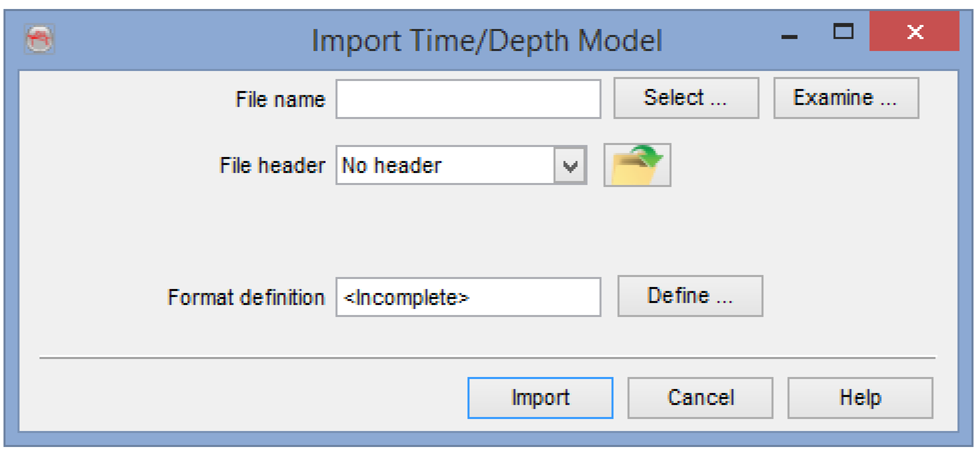
Both types have a similar editing window. It shows the mapping between measured depths and two-way traveltime, respectively in the first and second columns. Depths are displayed either in meters or in feet (toggle at the bottom of the window), and times are always displayed in milliseconds. These tables are fully editable: Double-click on a cell to edit it, type and new number and press enter or select another cell. The "*Update display*" button allows the user to update the displayed well data (track, markers and logs) in the scene to be updated based on the actual table content.



	MD (m)	TVD (m)	TVDSS (m)	TWT (ms)	Vint (m/s)
Measure point 1	34.1	34.1	0	0	2000
Measure point 2	479.74	479.7	445.6	473	1884.14367676
Measure point 3	510.51998901	510.47998047	476.37997437	504	1985.80517578
Measure point 4	547.75	547.70996094	513.60998535	543	1909.23254395
Measure point 5	569.65997314	569.61993408	535.5199585	565	1991.81445313
Measure point 6	611.39001465	611.34997559	577.25	610	1854.66784668

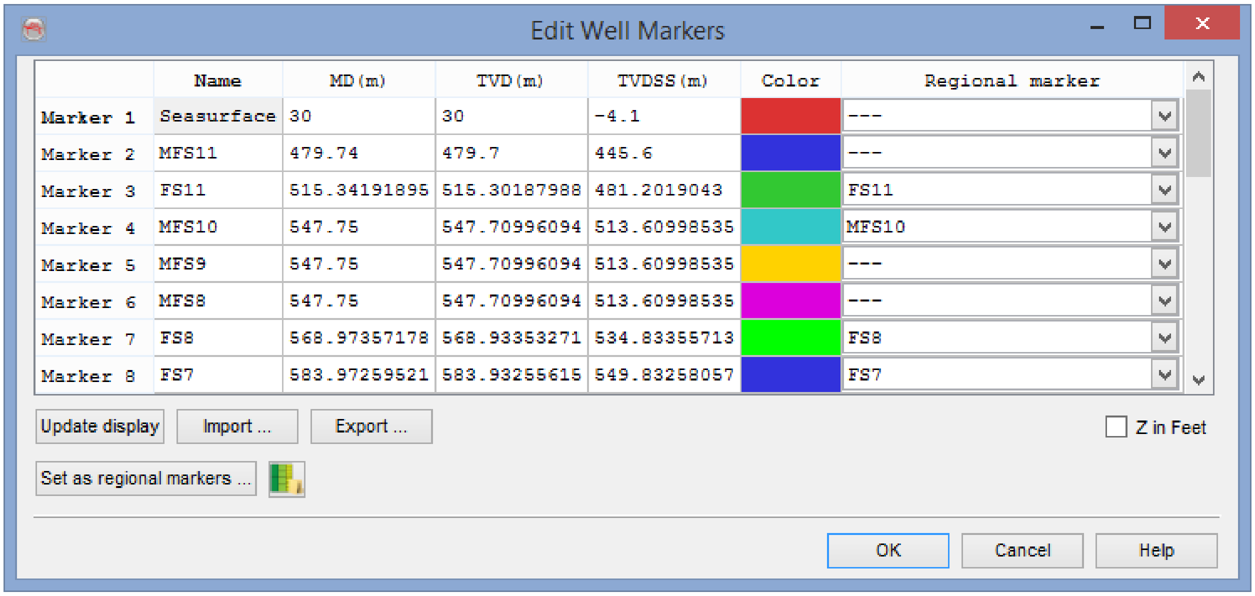


The following window appears after having clicking on "*Import*" in the edit Checkshot or Time-Depth model window. The import settings are fully similar to that the import step. The *Export* button allows to export the table in the same format to an output ASCII file.



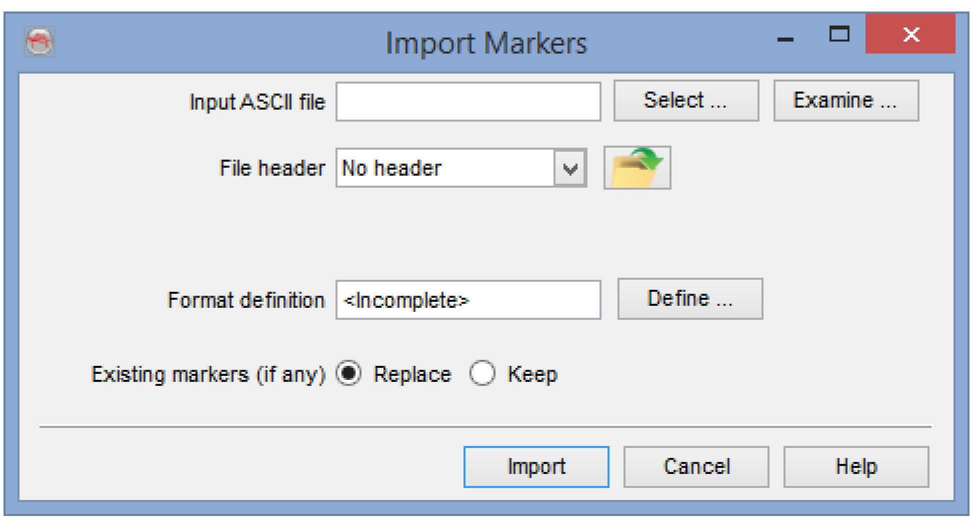
Manage Markers

Well markers can be manually provided or imported. They can be exported.



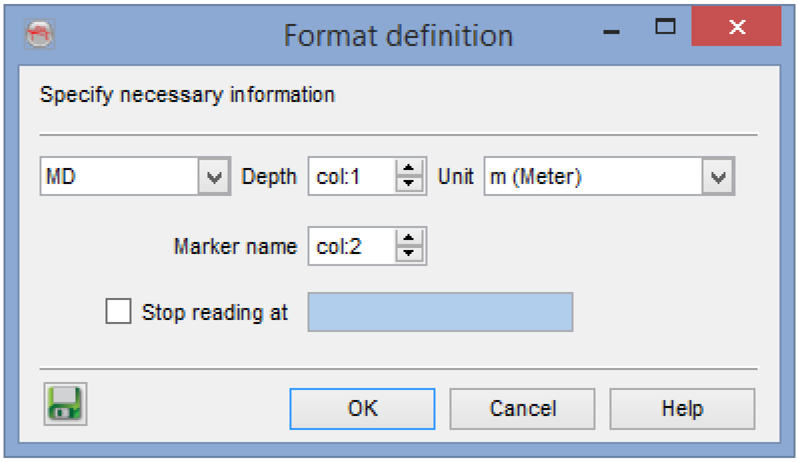
Edit markers: Individual markers can to be added or deleted right-clicking the mouse on an existing well marker in the marker table and choosing the appropriate command. To edit the name or value, double click in the appropriate cell. Levels can be set according to the stratigraphic framework, but please note that marker names and color will be updated according to the framework when setting a level to a well marker. The [Stratipgraphy manager](#) can be access from the  icon.

Add markers: Markers can be loaded from a file by clicking the *Import* button. The following window will then be displayed:



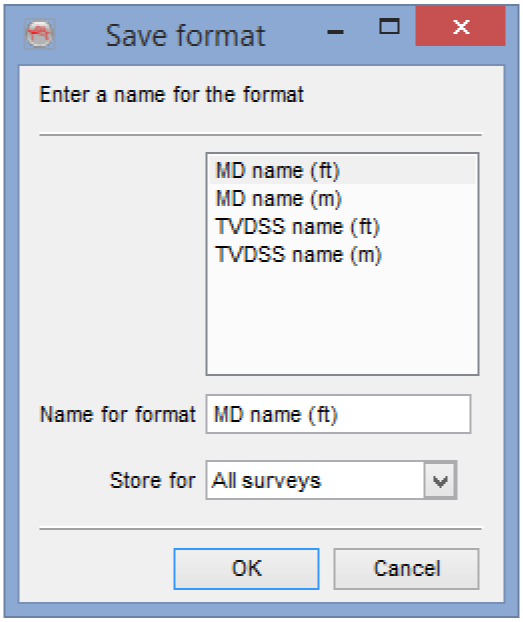
Select the input Ascii file. The main work is to specify the presence of a *file header* and the file *format definition*. The header, if present, can be of fixed length (number of lines), or delimited on its last line by a keyword.

Predefined and saved file formats are available by pressing the Open icon . Other-wise the format must be manually specified. The *Define* button gives access to the format definition window.



You must specify in the format definition window the column numbers of the marker name and depth. Please mind the spaces in the marker names that can break the fixed column format. For that reason it is recommended to have the depth in the first column, and to specify column 2 as the position of marker names. Then all strings found in column 2 and up will be used to form the marker names. Depths can be either measured depths or TVDSS depths. Data loading can be stopped at a specific line by providing the adequate keyword.

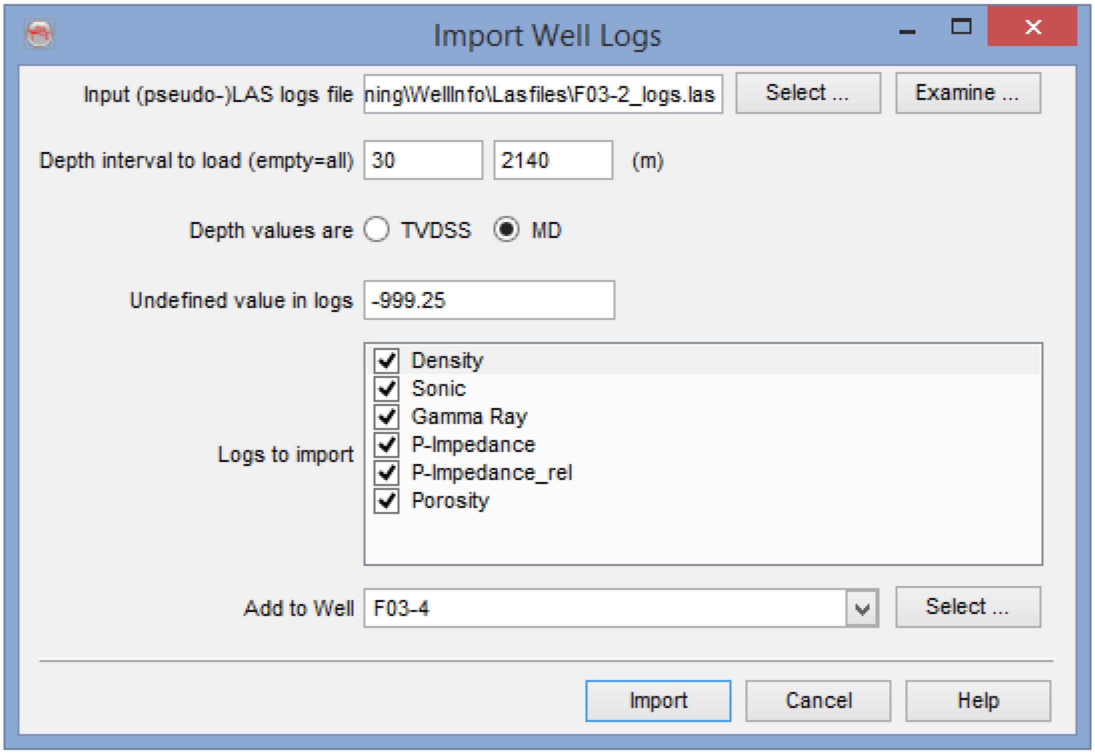
It is recommended to save the format definition for a later use and QC, by clicking on the Save icon . In pop-up window, write the name of the format and store it. The format can be stored at different levels (All surveys, Current survey, Current OpendTect user level) depending on the usage. Press *Ok* when done.



Update display: After adding and/or editing the markers, they can be refreshed in the main display by clicking on the*Update display* button.
Export markers: Finally, the edited markers may be saved to a new file/location by clicking on the *Export* button.

Logs Import

Logs can be imported from the Well Manger window.



Import Logs: The file should be in LAS format, with either MD, or TVDSS. Alternatively, the log files can be pseudo-LAS, meaning LAS (with one line of data per depth value) with the header replaced by a one-line definition: "Depth Gamma Sonic" etc (without quotes). Log names should be separated by blank characters (space or tab). For both LAS and pseudo LAS, the following units can be recognized. The recognition process is case insensitive.

Once the file has been selected all recognized logs will be listed in the *Select logs* section. Only the highlighted logs will be imported. Be careful that two logs do not have the same name. The depth interval can be limited to a subrange. The start depth, stop depth and step written in the LAS files are not used; instead the depths found on the same line as the amplitudes will be used.

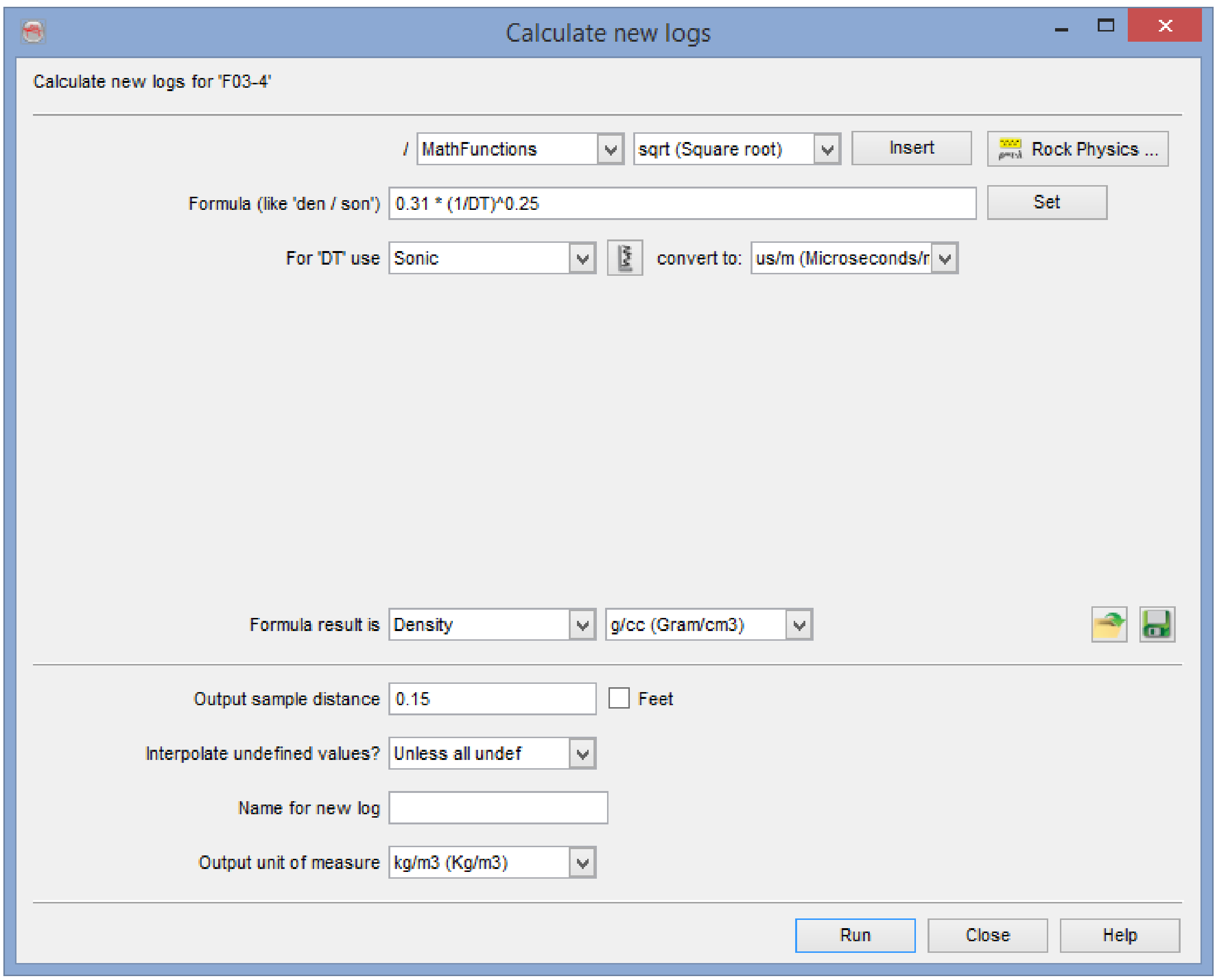
In pseudo LAS, units should follow directly behind the log name in parentheses, e.g. Depth(ft) Density(g/cc). Below are examples of text string that will match units:

- Time: *s, msec, μsec*
- Distance: *m, feet, f, ft, in*
- Density: *kg/m3, g/cc, g/c*
- Velocity: *m/s, ft/s, f/s, feet/s, km/s*
- Sonic: *s/m, us/ft, μsec/f, us/m, usec/m*
- Acoustic Impedance: *kg/m2s, kg/m2us, g/ft2s*
- Fraction (porosity, water saturation): *%, PU, or blank for unitless*

- Permeability: *k*
- Gamma Ray: *API*
- Electric Potential: *V*
- Resistance: *ohm*
- Compressibility: *1/Pa*
- Temperature: *K, deg.C, deg.F*
- Pressure: *Pa, bar*

Logs Creation

Logs can be created from log-log computations. Select one or more wells and click on 'Create' in the well management to open the log creation window as shown below:



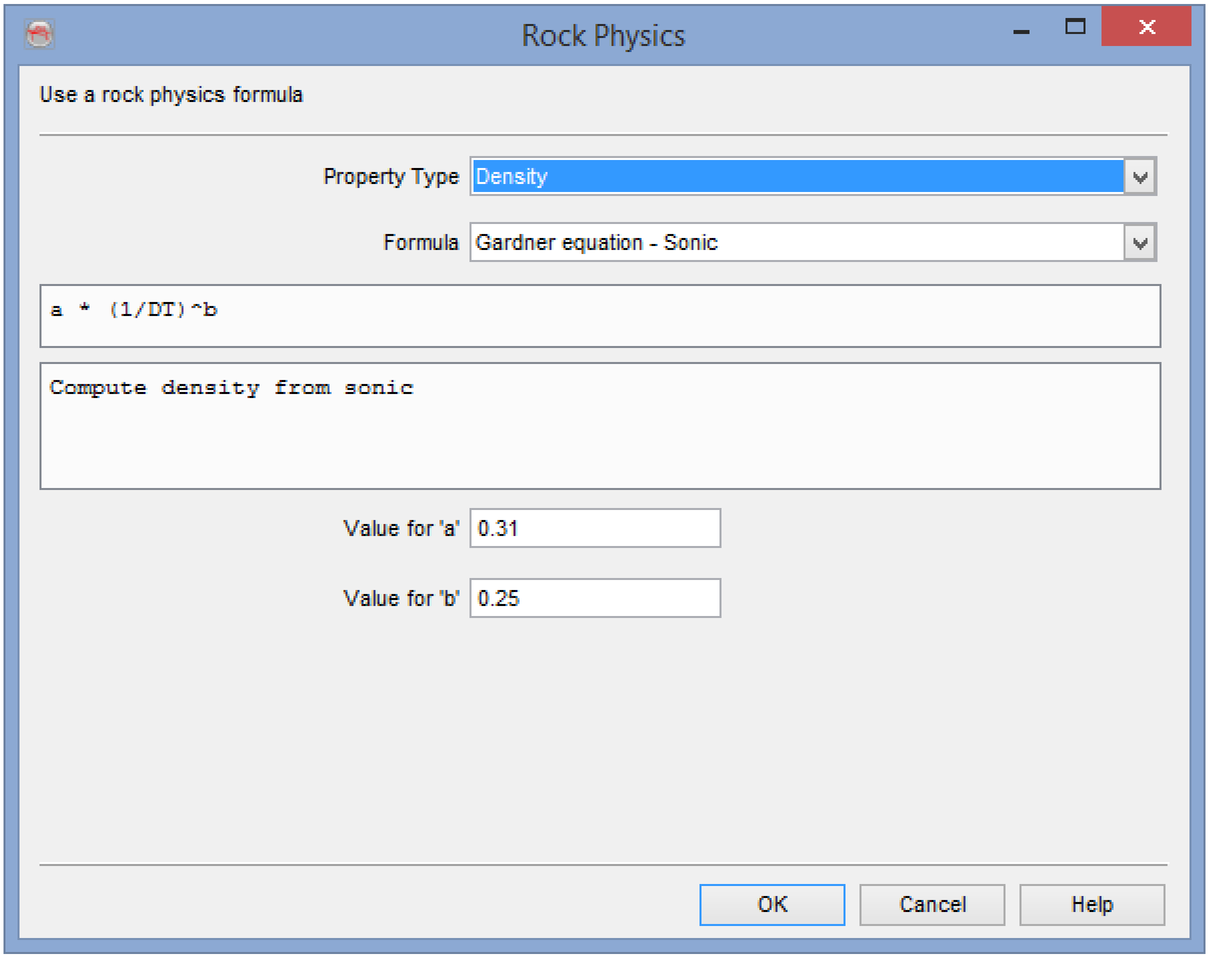
There is an inbuilt list of functions which can be used for creating new logs.This is supplemented by a Rock physics library containing more advanced resources.

The same syntax as the mathematics attributes should be used, with the following changes:

- The computation is done in the depth domain. No upscaling is performed.
- The various quantities (e.g. density) in the formula are matched with the input logs on which the calculation is being performed. The units of these quantities can also be changed, if needed.
- If "Fill empty sections" is not set for any log then the output will be defined where all logs do exist. Otherwise the input log that must be filled will be interpolated. In the shallow and deep parts the input log will be extrapolated by copying the data from the first and last sample respectively. This interpolation/extrapolation is done prior to computation.
- An automatic assignment of well logs based on the expression names is attempted when pressing "Set".
- The inputs logs have probably different Z ranges. The output Z range will be a regular array defined by the '*Output sample distance*'.

Rock Physics Library

A number of in-built advanced rock physics formulas can be selected to create various types of logs from available well log data.



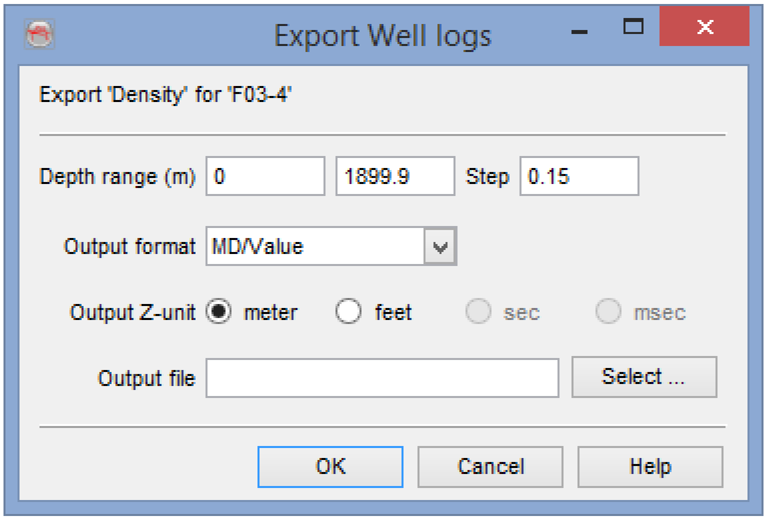
The type of output log quantity is chosen from *Property Type* list (e.g. Density, Velocity and Pressure). Afterwards, a specific formula out of a number of possible alternatives can be chosen to compute the required log quantity. The choice of equation depends on several factors such as the type of available log quantity and the region (e.g. a rock physics equations might work well in Gulf of Mexico but not in North Sea). After this selection has been made, the formula will be displayed along-with a short description below it.

The standard values for variables of the selected formula is also displayed.

Note: It is extremely important to keep in mind that the input log quantities (e.g. Sonic) MUST be converted in to particular units for the formula to work. These units are already selected by default and should NOT be changed. Same applies on the output unit of measure as well.

Logs Export

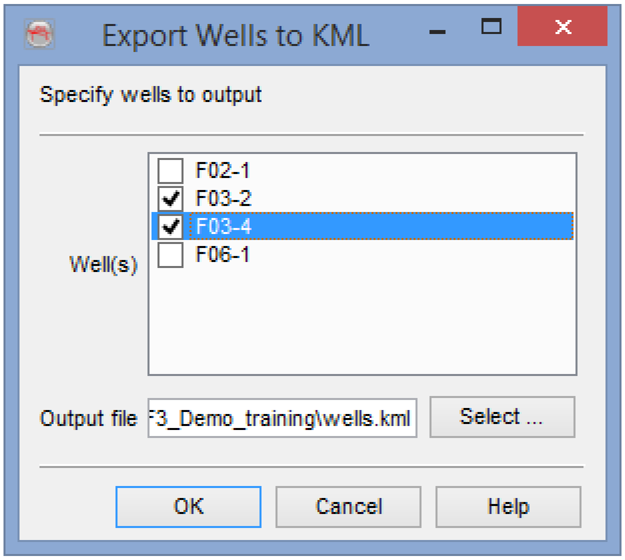
All or a selection of logs can be exported to an output text file. The input well must be selected, then select the logs to be exported, and click on the export icon .



The logs can be exported with respect to MD and TVDSS depths, optionally also with X/Y or Inline/Crossline positions. The depth range and step will specify the regular array on which the input logs will be interpolated prior to the export. The output file will be a column sorted Ascii file.

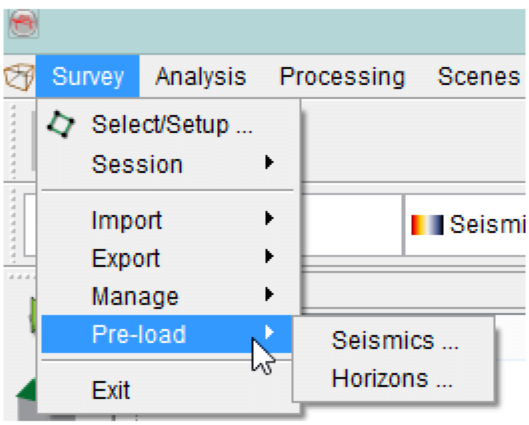
Export Well Surface Positions to Google Earth

The well locations (surface coordinates) can be exported in Google earth using the icon. Select a selection of wells (CTRL-left click to select several wells) to be exported in the popup window, and specify a filename for the kml file to be created. Press Ok, and open this file in GoogleEarth.



Pre-load

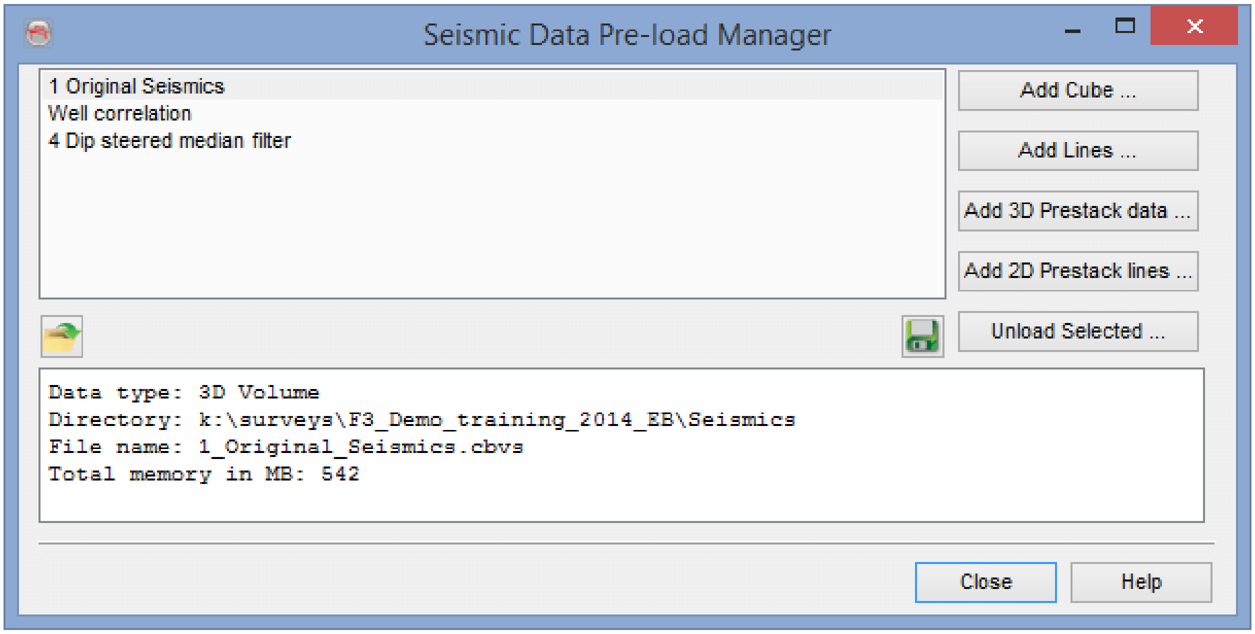
In OpendTect, the user can pre-load seismics or horizons. The advantage is to allow for faster display times in the scene. Your system must possess sufficient memory to store the pre-loaded data.



Preload Seismics

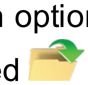
The pre-load seismic data functionality is introduced inside the OpendTect to speed-up the working efficiency with large volumes. A common practice is to add an inline/cross-line/2D line in the tree and display it in the scene. Each time the line is displayed, the stored volume is read. Therefore, the seismic volumes that are routinely used, can be preloaded. Also if the same seismic data like attribute cube is pre-loaded, the efficiency of displaying the data in the scene is improved.

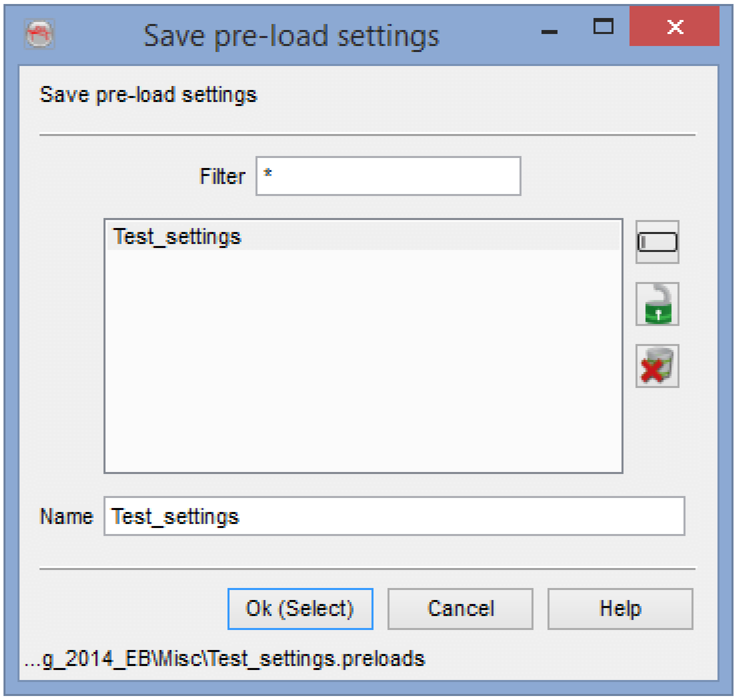
Seismic data can be pre-loaded in OpendTect by going to *Survey* > *Pre-load* > *Seismics...*
Clicking on *Seismics*.., the pre-load manager pops-up to allow the user to *Add*, i.e select the data to pre-load.

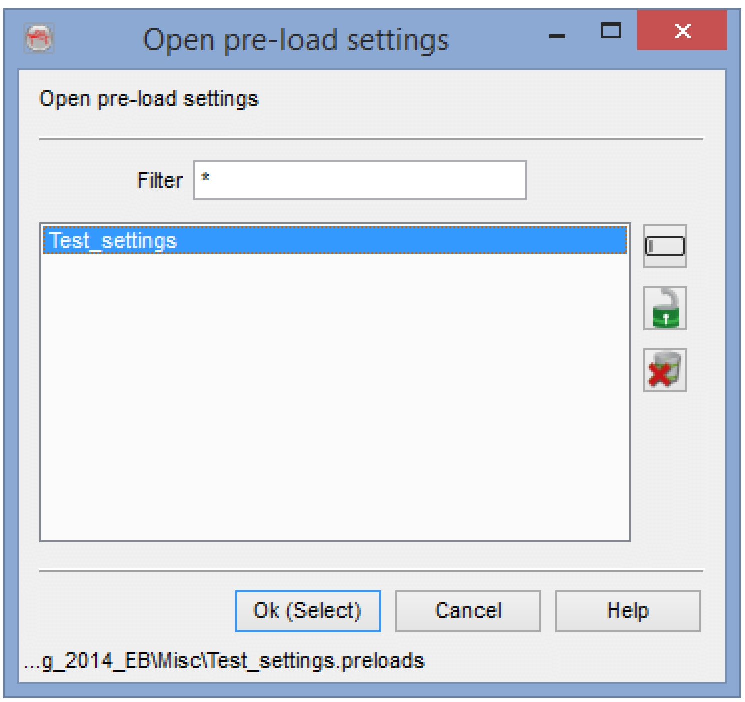


Pre-load seismic data manager

Preload manager is used to load a 3D volume, 2D lines, 3D Pre-stack volumes or 2D Pre-stack lines in the memory. Each data can be unloaded separately in selecting it and clicking on *Unload Selected*.

Additionally, after selected a pre-load data, the user can optionally save  his/her settings for the later use. These settings can then be opened  for use later.

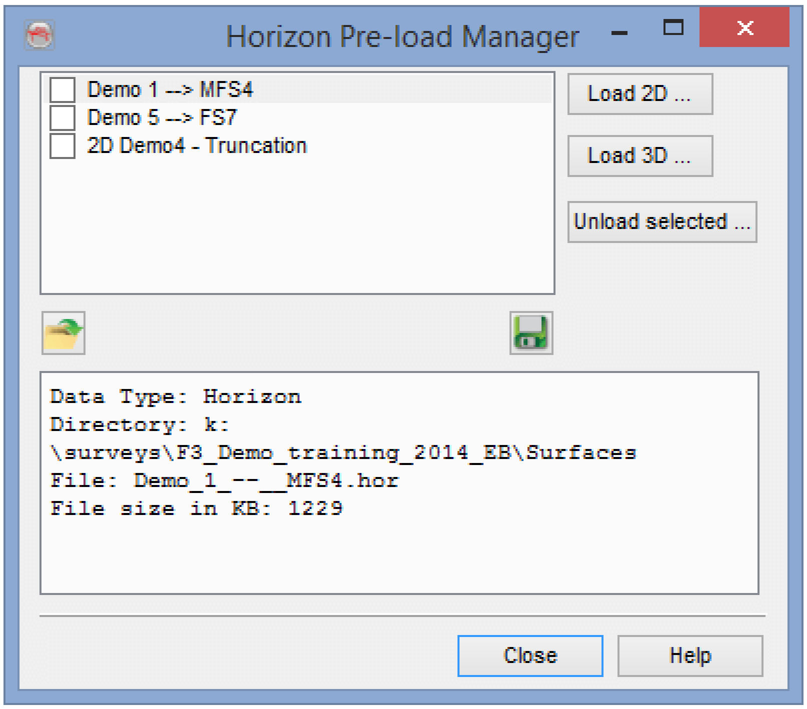




Note: The functionality is available for all stored seismic (2D/3D-Pre/Post) data in the relevant seismic data manager interface.

Preload Horizons

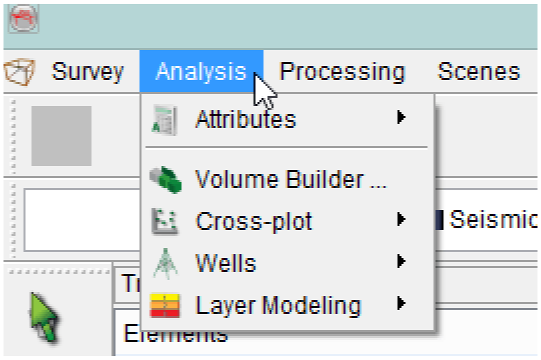
For fast visualizations, multiple horizons (2D/3D) can also be loaded in to the memory. Thus whenever the preloaded horizons would be displayed in the scene, it will take less time in reading the file from the disk. Therefore, the functionality improves the visualization speed. Press the *'Add ..'* button to select the desired horizons to be loaded in the memory. If you want to unload some horizons, select the horizons first and press *'Unload selected'* button.



The *Save*  and *Open*  buttons are used to store/open the pre-loaded horizons setup for a later use.

Analysis

Table of Contents



Attributes

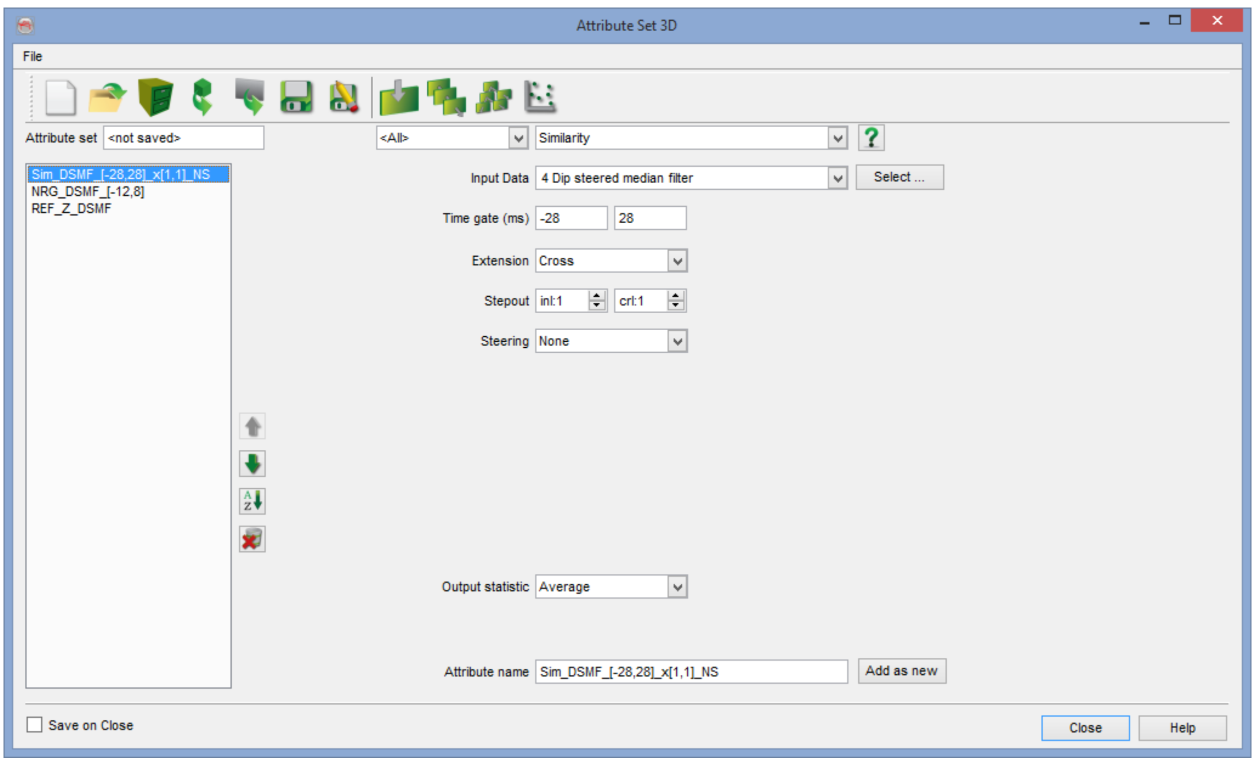
In OpendTect, seismic attributes are calculated/evaluated by using *Attribute-set Window*. In this window, many single/multi trace, pre/post-stack, dip-steered/non dip-steered attributes are available. Moreover, it also contains special filters (e.g. Gap decon, Frequency filters, dGB-special filters etc). The attributes are explained individually in Appendix A.

Attribute Set Window

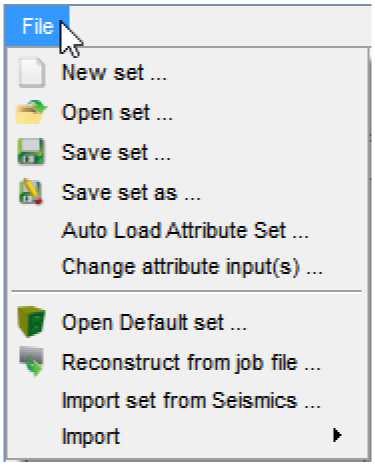
The attribute set window contains a set of seismic attributes definitions to be evaluated/calculated. While defining the attributes it is possible to work in the active scene. Attributes can also be calculated after saving the attribute set. In broad sense following workflows are applicable in OpendTect attribute calculation process (on sections and horizons):

- *Evaluate attribute*
- *On-the-fly attribute calculation*
- *Creating an attribute output (2D/3D)*

In following figure, different attributes are defined as an attribute set. It is considered as a routine practice in OpendTect environment (especially to define a meta-attribute). These attributes can be: evaluated on sections and horizons, applied on-the-fly and created as an output attribute. Importantly, some attributes show *steering selection* in the input parameter settings : the dip-steering plugin created by dGB provides various advanced steered attributes.



OpendTect works with the concept of an "active" attribute set. At start-up, there is no active attribute set. To create a new one (*New set...*), or to select an existing set (*Open set ...*) , select the corresponding option from the *File* menu (See below). OpendTect is also delivered with a collection of default attribute sets for some general testing (Fault, Chimney,Salt default attribute sets). This set can be selected from the Default set option under the *File* menu. To use a default set, the input seismic data and a steering cube (if steered attributes are available in the default set) have to be selected.



Clicking any attribute in the list will show its parameter settings. Notice that OpendTect uses SI units. For details on each of the attributes see Appendix A. Note that some of the parameter options depend on whether you are using 2D or 3D data as input. For example, the inline and crossline stepout field will be replaced by a single trace stepout field. Generally, an attribute set can only contain 2D attributes or only 3D attributes. Mixed attribute sets are not possible.

When parameters of an attribute are updated, the modified attribute can be added to the attribute set with a new *Attribute name* by clicking *Add as new*. Clicking on any other attribute in the list means that the updated parameters are accepted, while keeping the original attribute name. The *Revert changes* button only reverts changes to the original state before clicking on another attribute in the set. When *Ok* is pressed, the (updated) attribute set becomes the "active" attribute set. The attribute set is saved to disk when *Save on Ok* is ticked. To save an attribute set under a different name, use the corresponding option under the *File* menu.

Allows for the creation of a new attribute set

Open an existing attribute set

Open one of the default attribute sets (provided within the OpendTect package)

Import an attribute set from another survey

Reconstruct an attribute set from file

Save the attribute set

Save as...

Re-display the element with the current attribute

Evaluate the attribute (parameters)

Cross-evaluate attribute (parameters)

Cross-plot the attribute.

File - Change input... can be used to change the input data of all attributes in the "active" set simultaneously, which is useful in case, for example, a new seismic volume has become available.

File - Auto Load Attribute Set ...
It is now possible to have an attribute set already open at start up using the " *Auto load Attribute Set* " option in the *File* menu. This enables to choose the attribute set which will be active the next time the survey is opened.

Attribute Set Toolbar

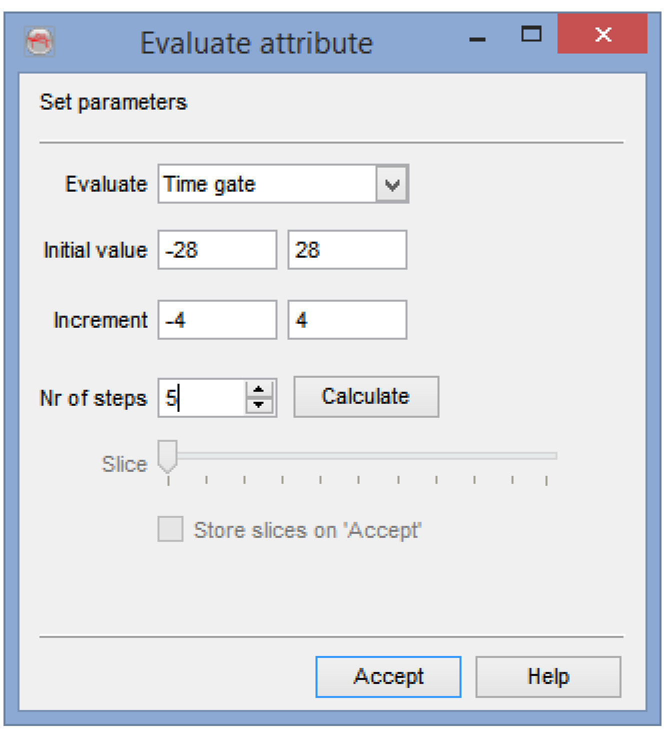
The attribute set toolbar comprises the following icons.

Also accessible via the File menu, these are, from left to right:

- *New set* clears the window to create a new attribute set. The attribute set name can be specified when saving it (press OK, or select File - Save set menu option).

- *Open set* opens a previously saved set in the current project (from the directory Attribs/).

- *Open default attribute set*. Filenames for input data must be re-specified.
- *Import attribute set from another survey*. Filenames for input data must be re-specified.
- *Reconstruct set from job*
- *Save set* saves the "active" attribute set in the Attribs/ directory of the current project.
- *Save as set* saves the "active" attribute set in the chosen folder
- *Redisplay element with current attribute* is used for direct display of the selected attribute on the active display element (shown in reverse video in the tree). The main graphic interaction buttons and options remain active while the attribute set window is open, so the active element can be changed. However no new element can be added to the tree
- *Evaluate Attribute* allows the automatic variation and evaluation of attributes and attribute parameters. If you have an "active" attribute and the current display element is a slice (inline, crossline or Z slice) or a horizon, a new window will pop up where it can be specified how to vary the parameters of the displayed attribute. For example, Spectral Decomposition:



Here six slices are created, with time gates of [-4,4], [-8,8] etc.... Use the slider to move through all the slices. When an attribute has been evaluated on a surface, the parameter can be updated in clicking on *Accept*. Enable this by checking *Store slices on Accept*.

As shown above, the "*Evaluate Attributes*" set contains a general selection of various attributes. It is intended as a guide or starting point for a scan through the wide range of different attributes and may be a starting point for an effective attribute analysis.

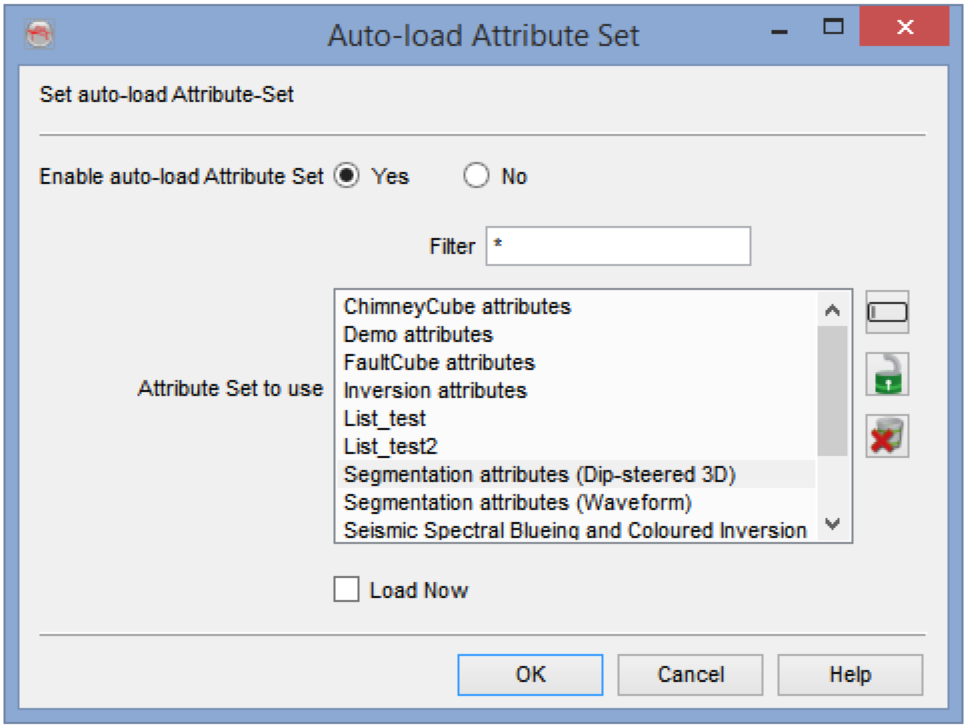
> **Note:** For more information on *Evaluate Attribute*, watch the tutorial on the OpendTect YouTube Channel.

- *Cross-evaluate attributes* Allows for the cross-evaluation of parameters within the attribute.

- *Crossplot attributes* allows to crossplot attributes from the current attribute set and saved volumes. Multiple attributes can be selected. The attribute values are extracted at picked locations (see how to create a pickset). Once attribute values are calculated, a crossplotting table is generated and crossplot(s) can be achieved.

Auto-Load Attribute Set

By default, no attribute set is loaded at startup. These settings can be over-ruled by selecting a specific attribute set to be auto-loaded in the list each time the OpendTect window is started. This can be set from the attribute set window under *File > Auto Load Attribute Set* sub-menu. If selected, it will launch the auto-load attribute set window. Selecting Yes will show the list of attribute set that can be be auto-loaded. Select one attribute set and press *Ok* button. This will save the settings and next time, whenever the OpendTect is started, the selected attribute set will be auto loaded. Such practice becomes useful when working with attributes evaluation at different stages of a project and that the same attribute set need to be updated.



Load Now will directly load the selected attribute set. If not selected, the attribute set will be loaded the next time the survey is opened.

Please note that a similar function exists for sessions.

Default Attribute Sets

OpendTect is provided with "*Default attribute sets*" to get you started. By selecting a default attribute set, a window appears to select the correct input volume(s) and the correct SteeringCube (see images below). These attributes (except "*Evaluate Attributes*") require the following dGB plug-ins:

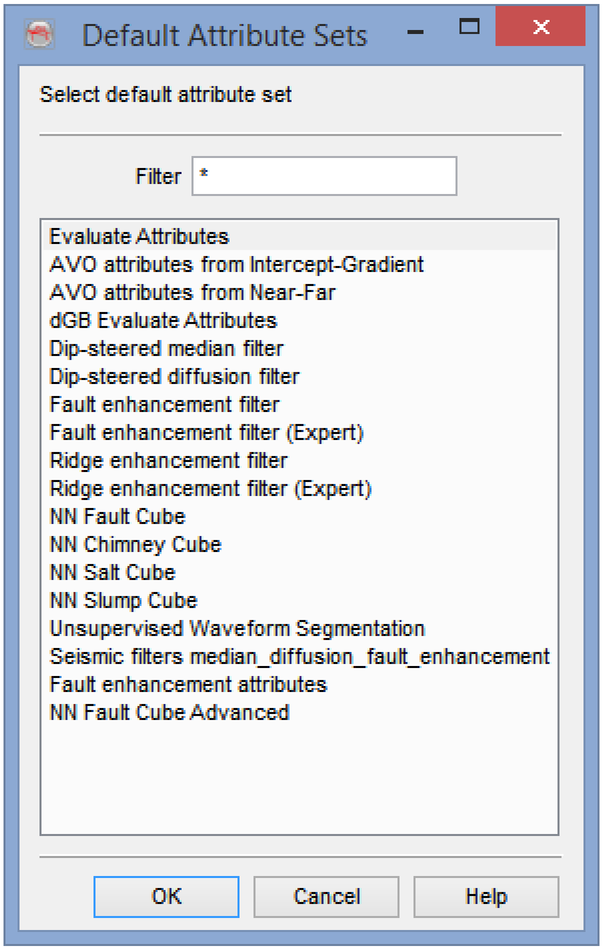
- *SteeringCube*: attributes and filters are calculated along user-driven, or data-driven directions
- *Neural Network*: Both supervised and unsupervised neural networks allow generation of meta attribute volumes that highlight any object of interest (e.g: Chimney, Faults, Salt,...).

The OpendTect version comes out with new "default attribute-sets" in addition to the already existing attribute sets like NN ChimneyCube, NN SaltCube, Unsupervised Waveform Segmentation, dGB Evaluate Attribute, etc .

- **Evaluate Attributes:** This default attribute set contains the default definitions of several basic attributes grouped together to give an idea of attributes evaluation in OpendTect. This default attribute set can be selected to start with OpendTect. After selection, only input seismic data is required.
- **dGB Evaluate Attributes:** This default attribute set is similar to above attribute set with additional dGB attributes (using dGB plug-ins). For this set, both seismic and steering data are required as input.
- **AVO attributes from Intercept-Gradient:** This attribute set requires two inputs; Intercept and Gradient (from AVO analysis) as first and second inputs respectively. It computes attributes like Envelope, Fluid Factor and Rp-Rs.
- **AVO attributes from Near-Far:** The input for this attribute set are the Near and Far stacked data sets in the same order. This includes attributes like Envelopes and Enhanced pseudo gradients.
- **Dip-steered median filter:** This default attribute set contains the definition of dip-steered median filter. It cleans up the seismic data by removing random noise. Both seismic and steering data are required as input.
- **Dip-steered Diffusion Filter:** This filter is mainly used to sharpen faults. Both seismic and steering data are required as input.
- **Fault Enhancement Filter:** This type of filter is used in the Fault/Fracture analysis, it dramatically sharpens the faults by suppressing random noise. It is a combination of the diffusion filter and the dip-steered filtered. Both seismic and steering data are required as inputs.
- **Fault Enhancement Filter (Expert):** This is a more sophisticated version of the basic Fault Enhancement Filter and uses similarity and dip-steered filtering. It also requires both seismic and steering data as input.
- **Ridge Enhancement Filter:** This filter detects lateral lineaments using different steered similarities (in inline, crossline and diagonal directions)
- **Ridge Enhancement Filter (Expert):** This filter is an advanced version of the above described filter and uses steered similarities in addition with their second derivatives (in inline, crossline and diagonal directions).
- **NN Fault Cube:** dGB standard default attribute set containing the

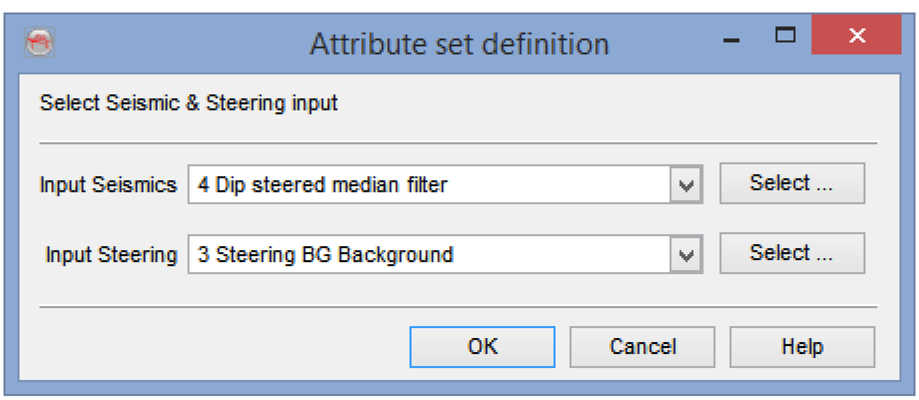
definitions of all attributes that are used in neural network (NN) training to create meta-attribute i.e. NN Fault Cube.

- **NN Chimney Cube:** dGB standard default attribute set containing the definitions of all attributes that are used in neural network (NN) training to create ChimneyCube (meta-attribute).
- **NN Salt Cube:** dGB standard SaltCube meta-attribute.
- **NN Slump Cube:** dGB standard SlumpCube meta-attribute.
- **Unsupervised Waveform Segmentation:** attribute set containing the definition of attributes that are used in unsupervised waveform segmentation (a.k.a UVQs).
- **Seismic Filters Median-Diffusion-Fault-Enhancement:** This is an advanced version of the "Fault Enhancement Filter". It enables the user to have much control on the input parameters by modifying the parameters of the dip-steered median filter, dip-steered diffusion filter and fault enhancement filter.
- **Fault Enhancement Attributes:** expandable attribute set containing the list of the attributes that are useful for fault visualization and fault interpretation.
- **NN Fault Cube Advanced:** most superior FaultCube (meta-attribute) attribute set that is used as input for neural network training to create fault probability cube.



Default attribute sets window containing the list of all available default attributes

When one of these default attribute-sets has been selected, a window pop-ups (see image below) to select the input seismic and optionally a steering (the attribute sets based on AVO analysis, the Fault enhancement filter and the Ridge enhancement filter require inputs as outlined in their respective descriptions).



Input selection Window

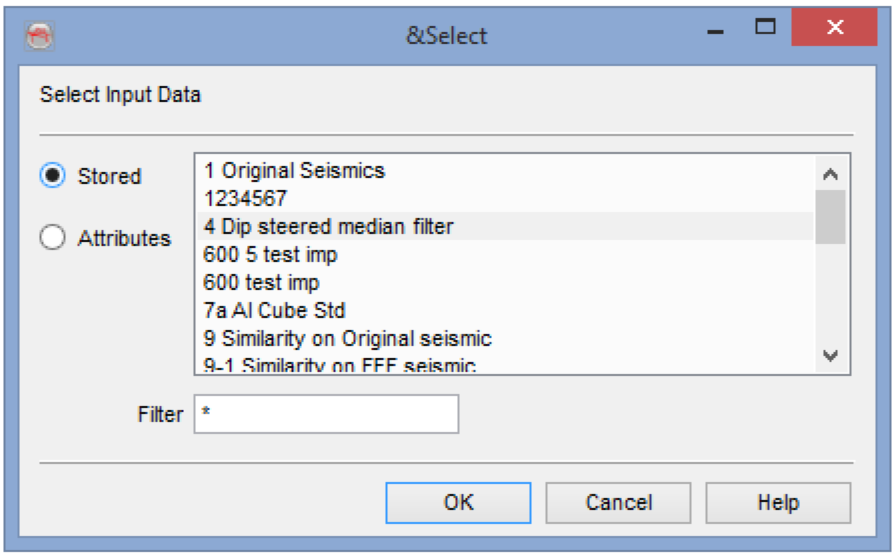
Input Selection

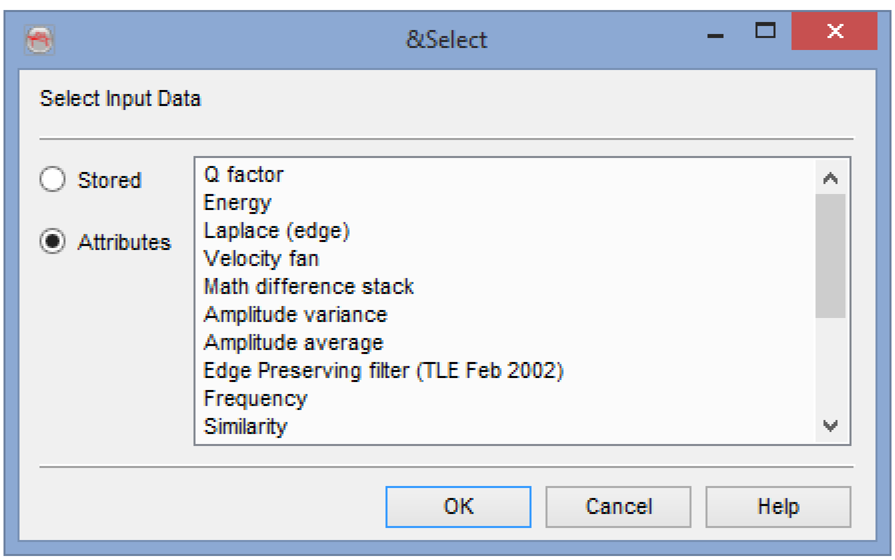
Every attribute requires input data. Both stored data and already defined attributes can be used as input to a new attribute. In other words, attributes can be embedded. However, circular references are not possible.

[Input Selection for 3D Attribute Sets](#)
[Input Selection for 2D Attribute Sets](#)

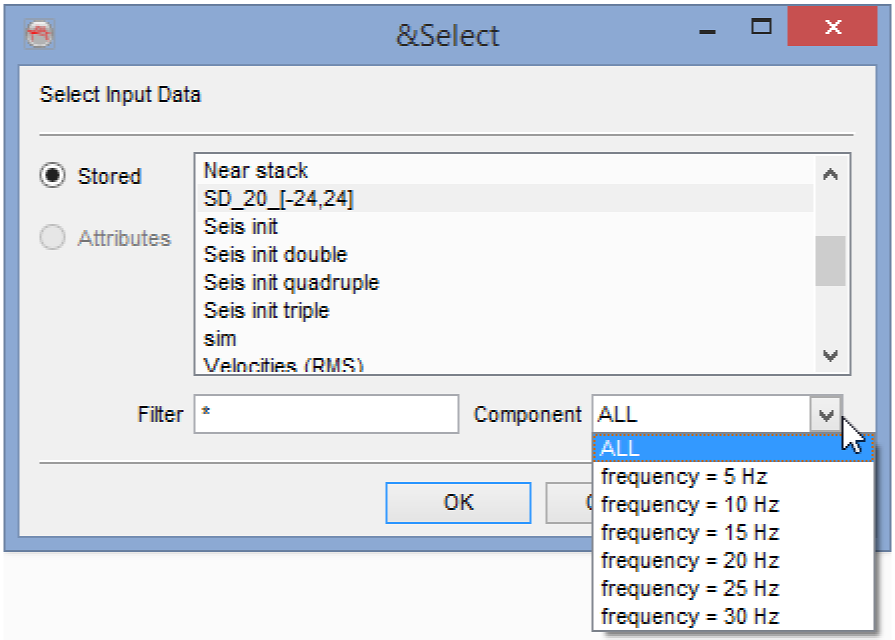
Input Selection for 3D Attribute Sets

Select from the stored data or from the list of defined attributes in the "active" attribute set.





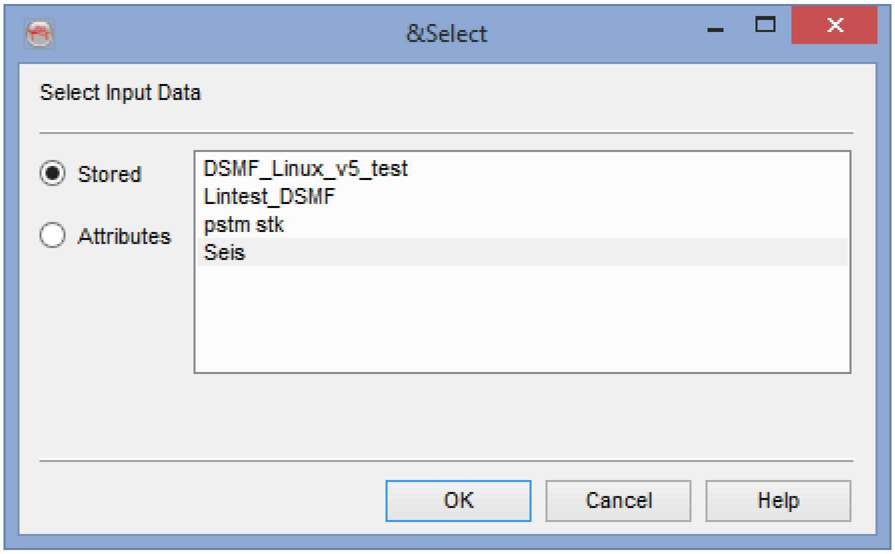
In case the input data is multi-component, it is possible to choose from the available components as shown below (or include *ALL*).

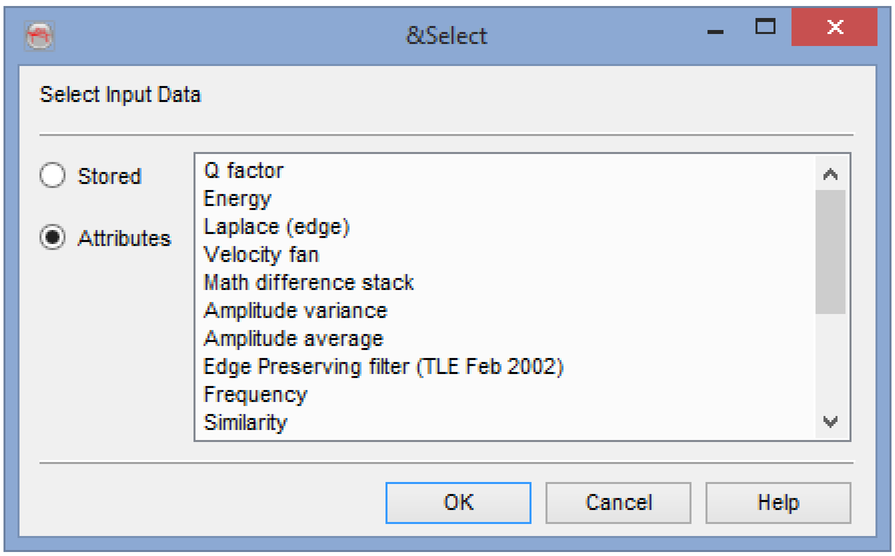


> **Note:** The Filter section allows to quickly find the right input. e.g type *S will look for all attributes/cubes started with S like Similarity.

Input Selection for 2D Attribute Sets

Select from the stored data or from the list of already defined attributes in the current attribute set.



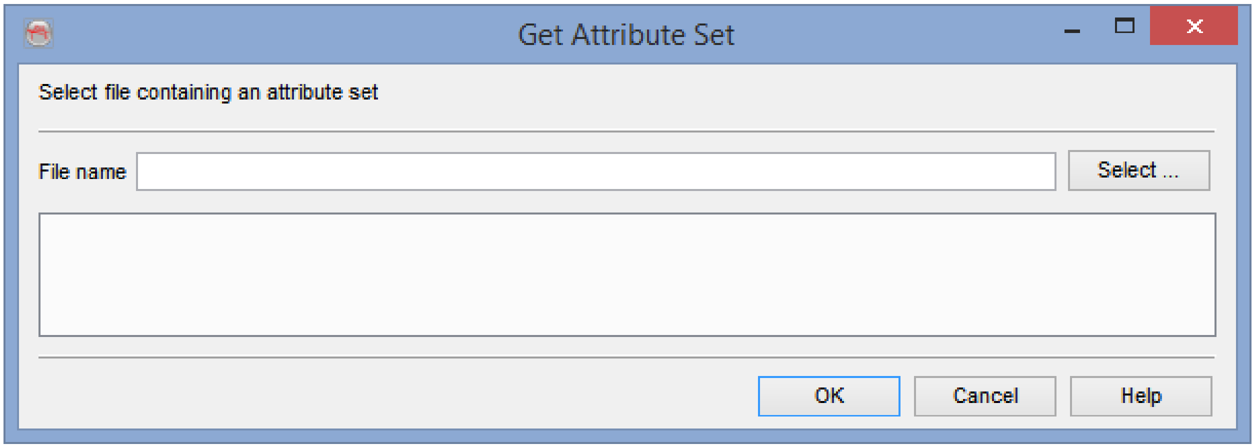


Note: If the selected stored data set is multi-component, the user will get an option to choose which component to select as input data:

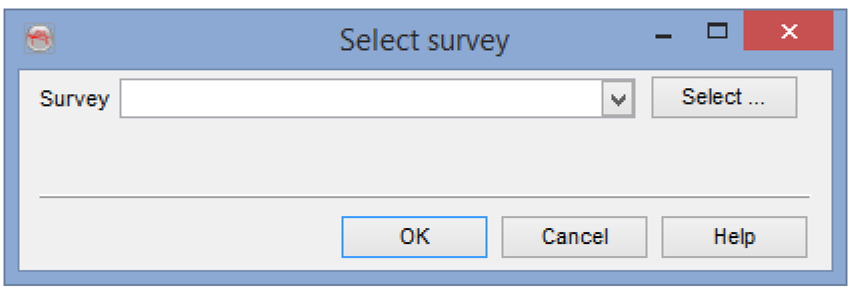
Import an Attribute Set from

Attributes are primarily stored in attribute set files of extension *.attr* Attribute definitions can also be found in the parameter files of a processing job when an attribute was used to process a volume or data set.

It is possible to import the attribute set of an attribute file from the menu: *File > Import set from file*. Existing attributes are stored in the *Attribs* folder of each survey. Optionally, attributes from another survey may also be imported: *File > Import set.* or by using 

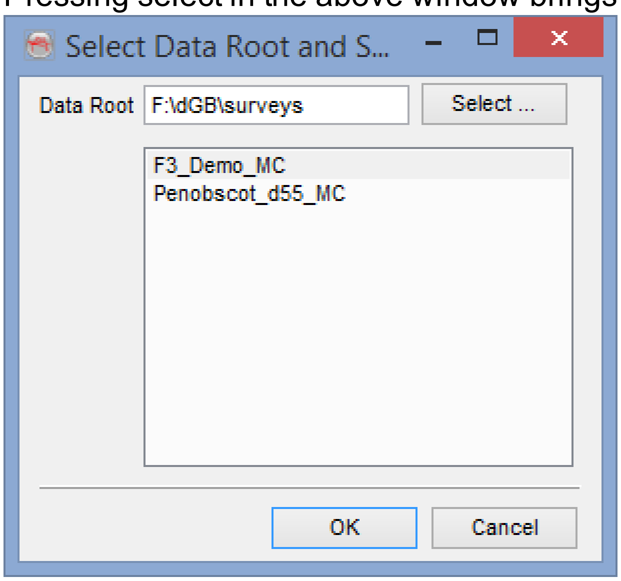


Import set from file

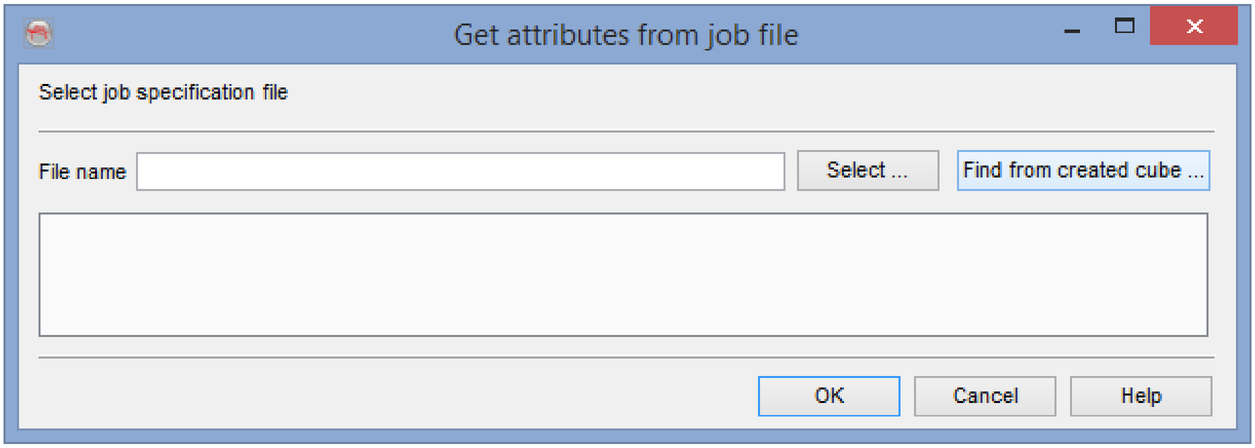


Import set from another survey

Pressing select in the above window brings the user here:

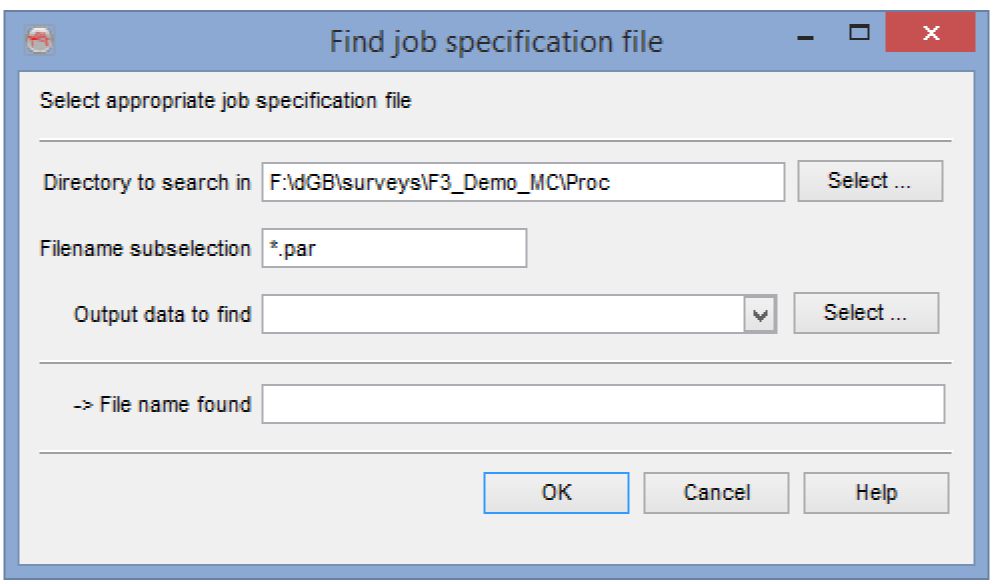


It is also possible to re-create the attribute set of existing processing file in the menu: *File > Reconstruct set from job file...* (or ) brings the user here:



Existing jobs are stored in the Proc folder of each survey, with the extension par. There are two options available to reconstruct the attributes definition: from an existing par-file or from a created cube file. In first case (from par-file) select the input parameter (*.par) file. In second case (find from created cube), another window pops-up in which the input volume and the corresponding parameter file are selected. The file name is found automatically.

Pressing *'Find from created cube...'* will open up this search/select window:



Finds the attribute set from an existing (created) cube, which was calculated inside OpendTect.

Note: When importing, new input volumes must be selected to replace the references stored in the input files.

Calculate Attributes

The attribute evaluation process has been considered critically and thus several key options are available for the user. For instance, *Evaluate Attribute* is considered as intermediate (but not necessary) step to quickly analyze the different parameters of any attribute within the working environment (View tutorial-Evaluate attributes). Similarly, the user can create a list of seismic attribute definitions as a working set that later on can be updated. The attribute set is then used to calculate the seismic attributes along lines/surfaces.

There are two possible ways of calculating seismic attributes in OpendTect: First, in order to calculate the results of any attribute in the foreground, user(s) can do it on-the-fly. Second possibility is to calculate attributes after evaluation by running a secondary process in the background. In OpendTect, seismic attributes are applied on several elements (inlines, crosslines, Z slices, random lines, 2D lines, volumes, horizons etc).

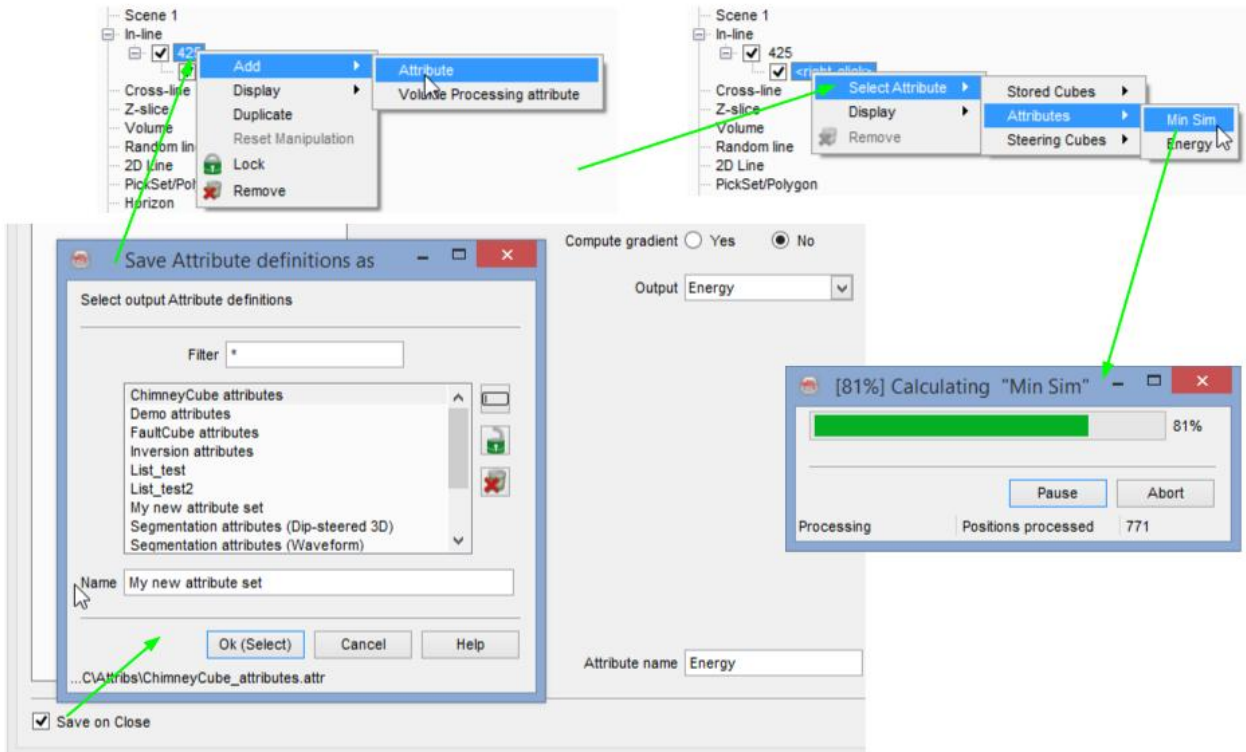
The workflow to calculate the selected attribute is quite simple:

1. Define (or use existing) attribute Set and save. For details see earlier sections of this chapter.

2. Calculate on-the-fly or Create Seismic Output or Create horizon attribute output.

3. If attribute is not calculated on-the-fly, retrieve results by displaying attribute in tree.

The example of first step is given in following figure. It highlights the sequential process (notice green arrows form left to right) of on-the-fly attribute calculation. Firstly, several attributes are defined. Secondly, by default, when a user presses *Ok* button in *Attribute Set* window, the *Save Attribute definition* window will appear to save the attributes definition as an Attribute set. The attribute can then be applied on an inline (for instance) by adding a blank attribute (right-click on inline number). Right-click on the blank attribute and select the attribute (Select Attribute > Attributes > "*User Attribute*"). The listed attributes are those that are defined in the attributes set window. Selection of any one, would start a process of on-the-fly attribute calculation. By following same workflow (as elaborated in figure) the same attribute can be calculated along other elements (e.g. cross-lines, Z-slices, volumes etc).

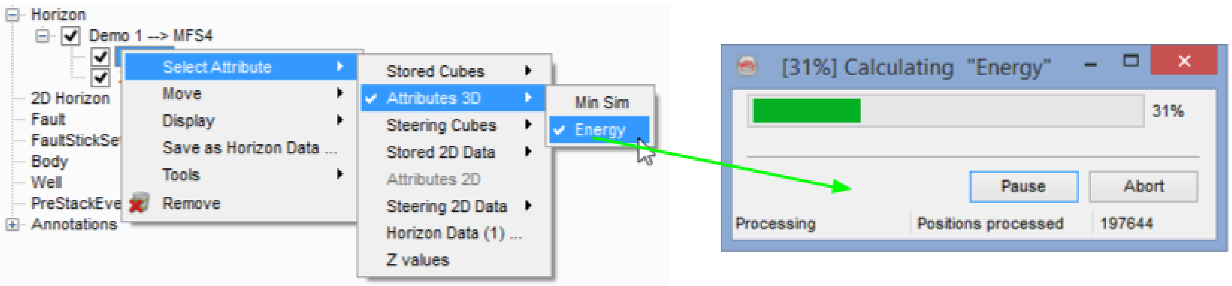


Schematic flow of on-the-fly seismic attributes evaluation on an inline.

Notes:
1. There are some attributes that can take too much time during the on-the-fly calculation process. This depends upon the type of attribute that how much calculation steps it considers e.g. multi-trace (e.g. Similarity) attributes normally take more calculation time than the single trace (instantaneous) attributes. Similarly, the attributes with steering normally takes more time in calculation. So, each time the attribute is displayed in the scene (as shown above), it is calculated in the fore-ground. If the user is committed with the attributes results, this can be resolved by creating seismic outputs (See Create Seismic Output and Create Horizon Output sections) in the background. This will also help to restore the saved sessions quickly.

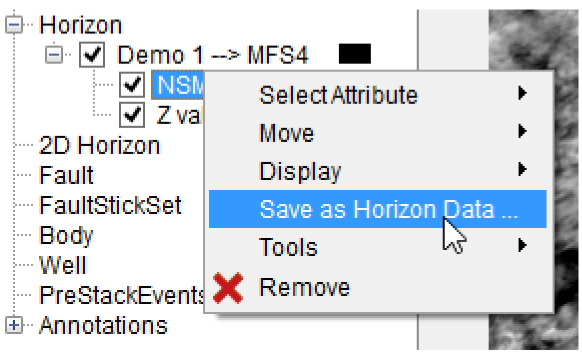
2. Additionally, **multi-component** output seismics-2D/3D (like spectral decomposition) are created by using create seismic output.

Another example of second step, is shown in following figures. The attribute can be calculated **along the horizon** by following the same steps described above, for inserting and displaying the attribute (as shown below). In this example, Similarity is calculated on-the-fly along a horizon. This attribute normally takes time (depending upon amount of traces involved). So, user can take benefit of saving the on-the-fly results that later on can be retrieved.



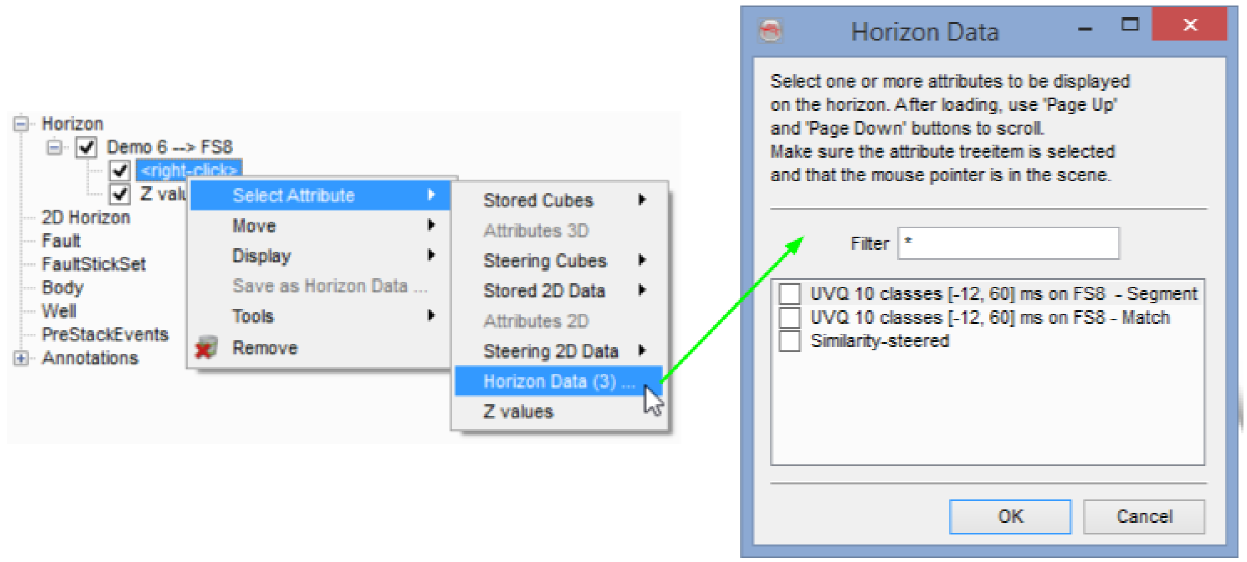
Schematic flow of on-the-fly seismic attributes evaluation at a horizon.

In order to save the calculated attribute as horizon data, right-click on the attribute and select *Save as Horizon Data...*. In pop-up window edit the name accordingly and press *Ok*. This will save the horizon attribute as its horizon data. That can be managed later on by using horizon management window. (see Management horizon)



Result of calculated similarity attribute. Saving the horizon attribute as horizon data

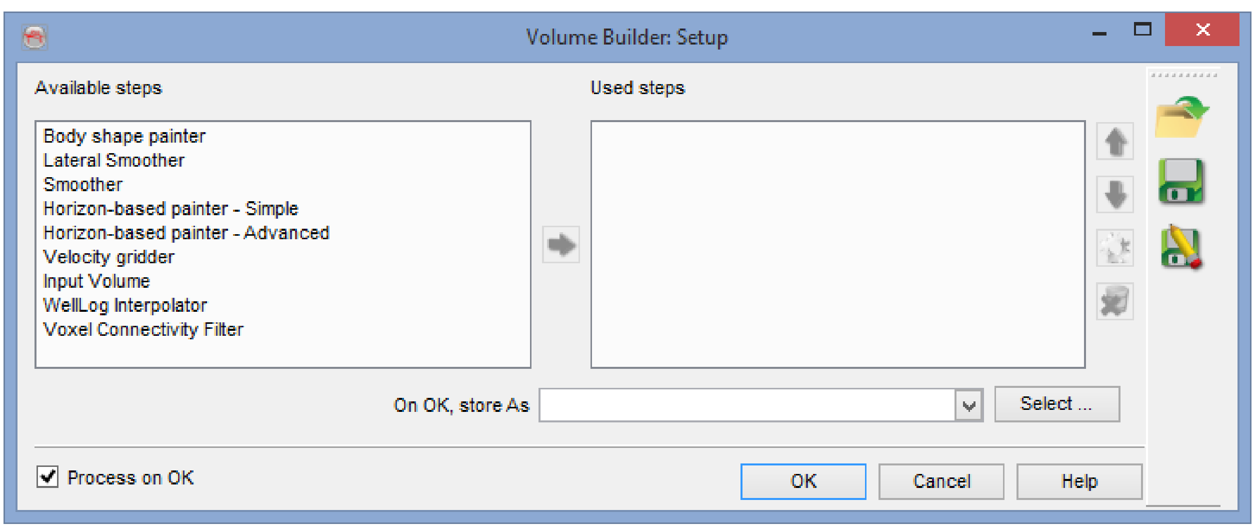
The stored attributes along horizon can be retrieved as horizon data. Right-click on horizon and add blank attribute. Right-click on the newly inserted blank attribute and locate Horizon data item in the sub-list of attribute (as shown below). In the horizon data selection window, select the desired attribute. This will display the selected attribute in the scene.



Retrieving the stored horizon data (attribute) of a horizon.

Volume Builder Setup

The volume builder setup is used to apply volume-based operations, unlike the attributes that work trace-by-trace. The setup is launched via the *Analysis* > *Volume Builder* menu. The setup is a very useful tool for gridding velocities or other rock properties.



The volume builder setup window contains several available steps that are applied sequentially to generate a volume. Any particular step can be selected from the Available steps by double-clicking on it. The later steps may replace the earlier ones, therefore care must be taken when ordering and setting up the workflow.

Once your workflow is defined, you will need to save this setup. This can be done by writing the name of your setup.

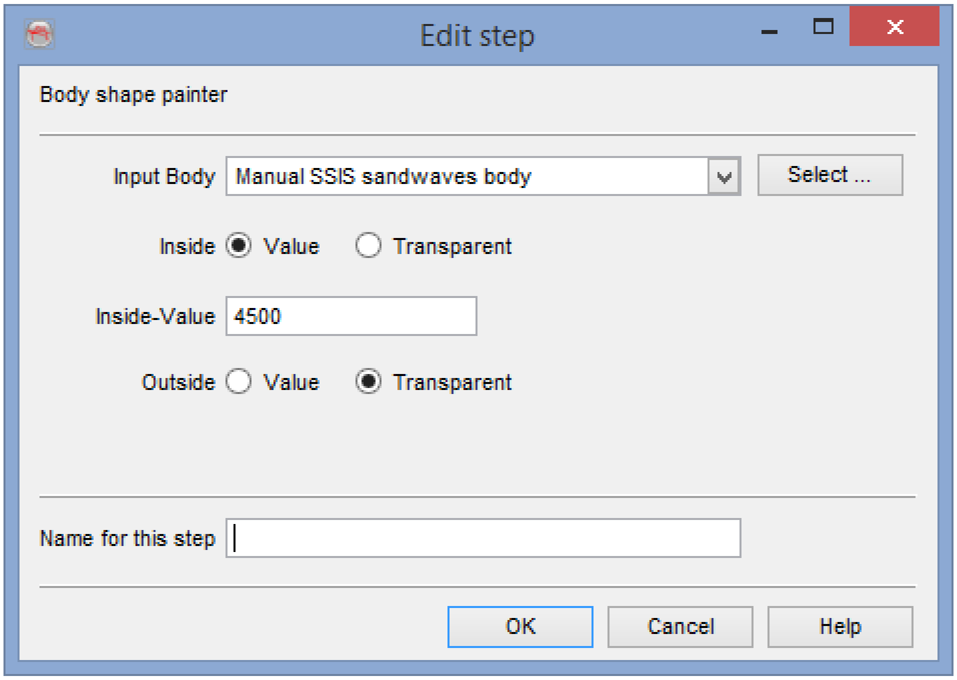
The computation and storage of a volume processing setup can be found under OpendTect's Processing menu: *Processing > Create Seismic Output > Volume Builder...*

The available steps:

- Body shape Painter
- Lateral Smoother
- Smoother
- Horizon-based painter (simple and advanced)
- Velocity Gridder
- Input Volume
- WellLog Interpolator
- Voxel Connectivity Filter

Body Shape Painter

The body shape painter is used to fill an OpendTect body with any constant value. It can also be transparent (i.e. undefined). For instance, if one wants to create a salt velocity cube, the values inside can be filled with a salt velocity and outside can be left as transparent. In that case the outer values will originate from the other steps. If no other step exists the undefined value is written.

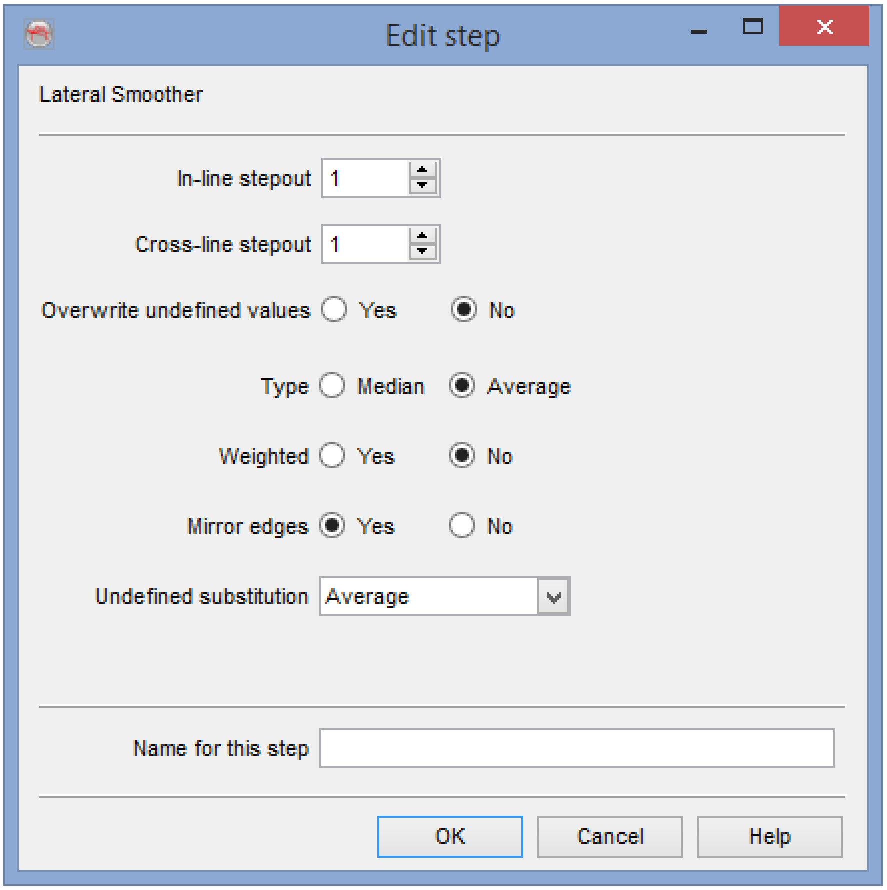


Lateral Smoother

The lateral smoothing is a rectangular two-dimension smoothing filtering method of the volume.

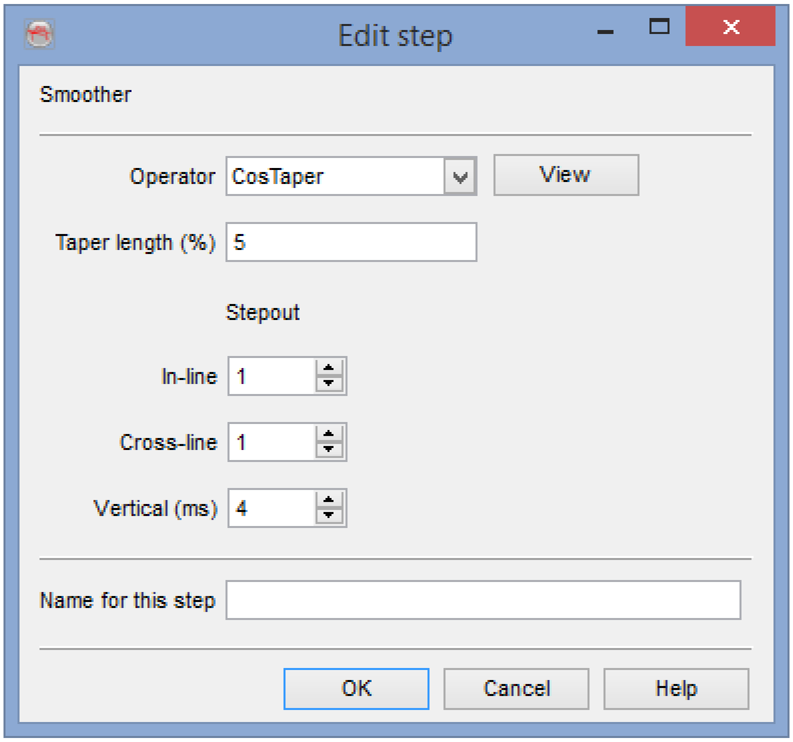
The average filtering will be done in the frequency domain by applying a 2D FFT to the Z slices. This requires a rectangular dataset, while the input can be irregular. The filtering type can be chosen between "Median" or "Average" which can optionally be "Weighted". Positions without data or with undefined values are first replaced by the option "Mirror edges" and "Undefined substitution". The "Undefined substitution" can be done by taking *Average* values between the defined points, *Fixed value* (one value needs to be specified which will be used everywhere) or using *Interpolate* between the defined points.

Finally, this step can be saved by giving a name.

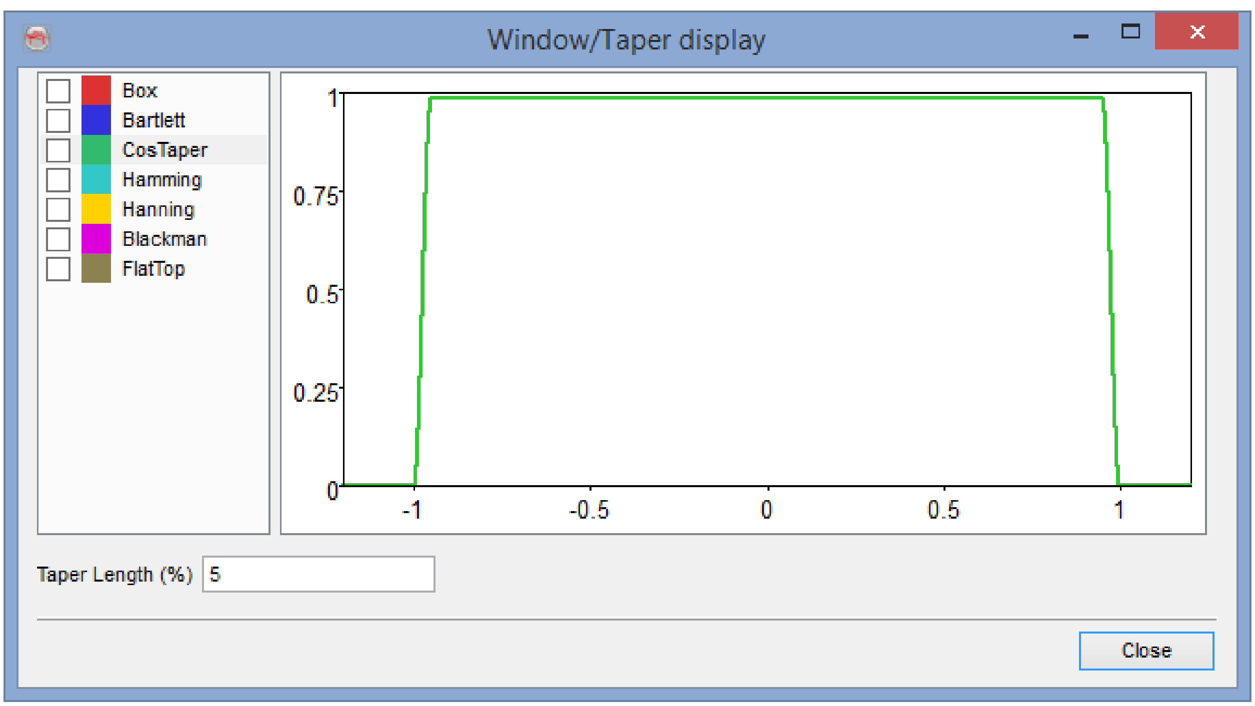


Smoother

The smoother step is used to apply a three dimensional smoothing operator by specifying In-line, Cross-line and Vertical (ms) stepouts.

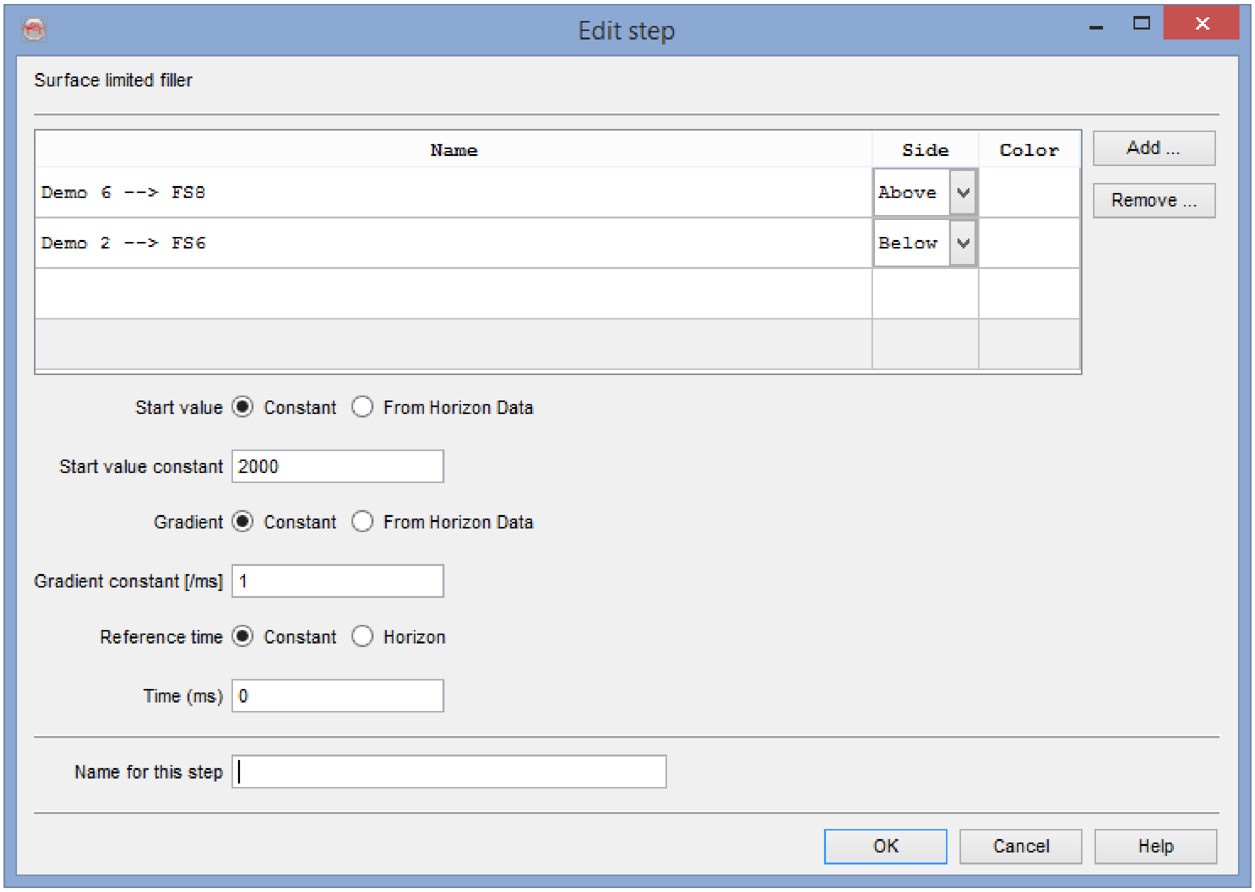


Various operator shapes can be chosen (e.g. Hamming) and can be visualized by pressing on the View button. The CosTaper also requires specification of a "Taper Length (%) ".



Horizon-based Painter - Advanced

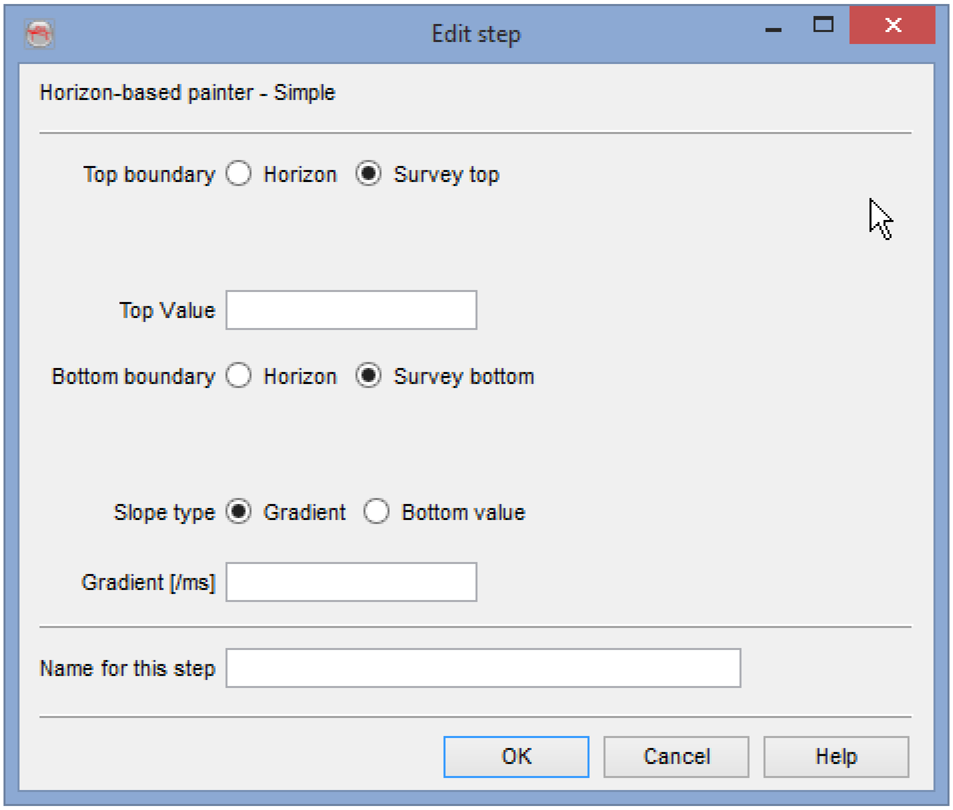
The horizon-based painter is used to create a model between two surfaces. The initial top and bottom values are necessary to be filled in the input. The intermediate values are interpolated to a survey or a horizon. In this window, horizons have to be selected as top/base values. The slope type is used in interpolation to define a slope.



The horizon-based painter paints velocities in a 3D area whose geometry is defined by one or more 3D horizons.

The painted velocities are referenced to a specific time. This time can be either constant (user-defined), or retrieved from a 3D horizon and not necessarily from one of the horizons defining the limit of the body. For example, an intermediary horizon could be used. Then velocities are painted from that reference time. The velocity must be provided as a velocity/gradient pair. The values are once again either user-defined or extracted from a surface data (grid) attached to a 3D horizon.

There is also a much simplified Horizon-based painter also available:

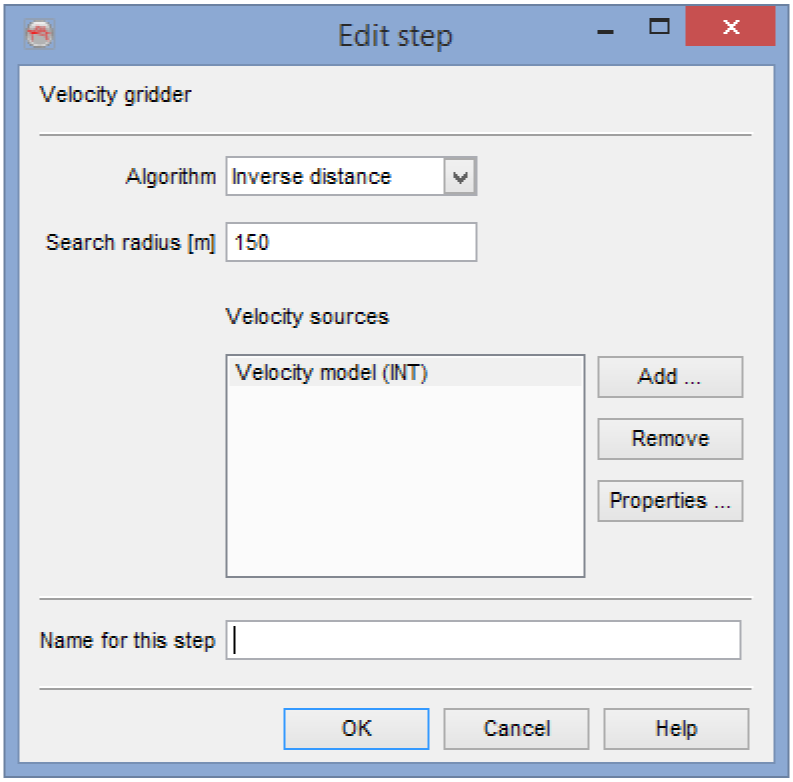


Velocity Gridder

The gridding will create a volume out of a sparsely sampled dataset. **The input source MUST be tagged with a velocity type.** Indeed the gridding is applied to the time-depth relation hold by the velocity source and not on the amplitudes of the velocity source. This preserves the time-depth relation and blocky-ness of interval velocity models. The gridded time-depth relation is converted back to the input velocity type in the output volume.

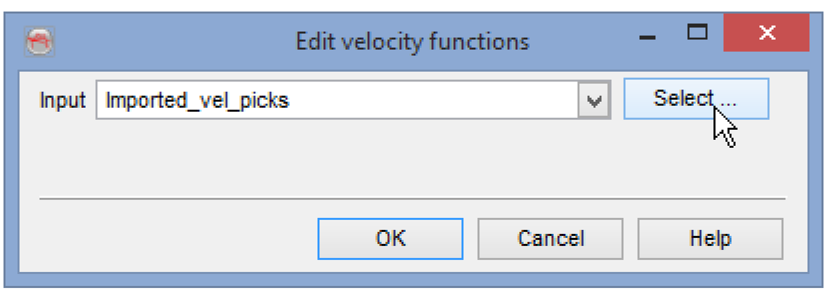
However any other data type could be gridded by this module (Thomsen parameters, temperatures, ...). These other types must be tagged as delta, epsilon or eta before being used for gridding. The functions will be vertically interpolated using a linear 1D interpolation before the lateral gridding.

Two interpolation methods are available for gridding: inverse distance interpolation or triangulation. The first method is designed for the interpolation of sparse dataset, while the second algorithm should be preferred if the input exists on a regular (but coarse) grid. In general gridding is always followed by some filtering.



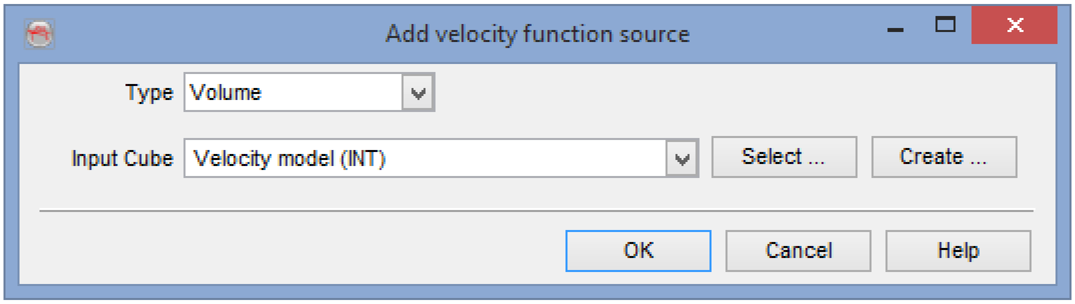
The input data can be either a (coarse SEG-Y imported) volume, stored (ascii) functions or velocity picks (requires the Velocity Model Building plugin)

Clicking on the '*Properties*' button will allow you to change the selected input for this step:

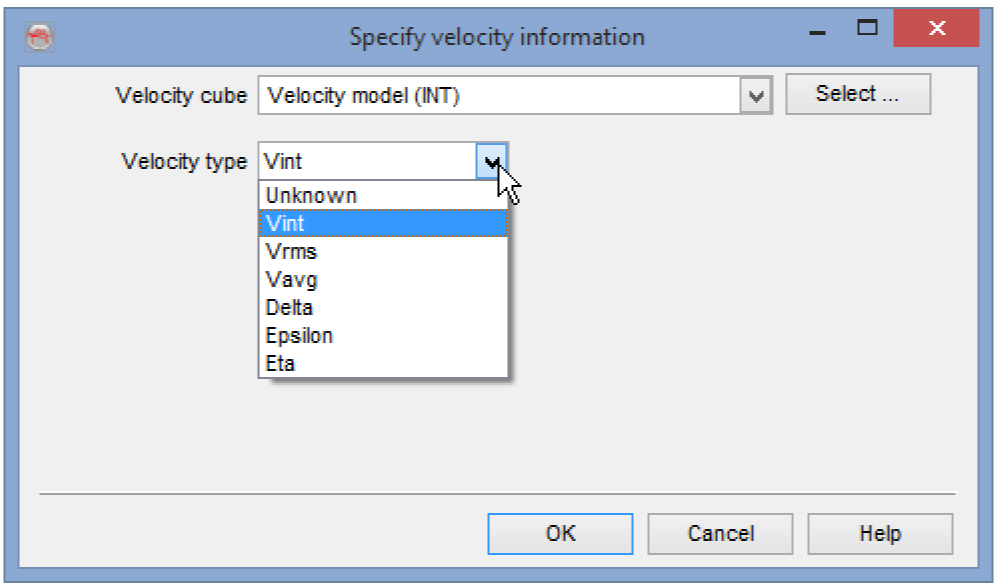


Note: Velocity volumes have to be tagged to recognize their type.

Tip: If you have no velocity volume available, press '*Add*' in the '*Edit Step*' window and then, in the window that pops up (shown below), press '*Create*':

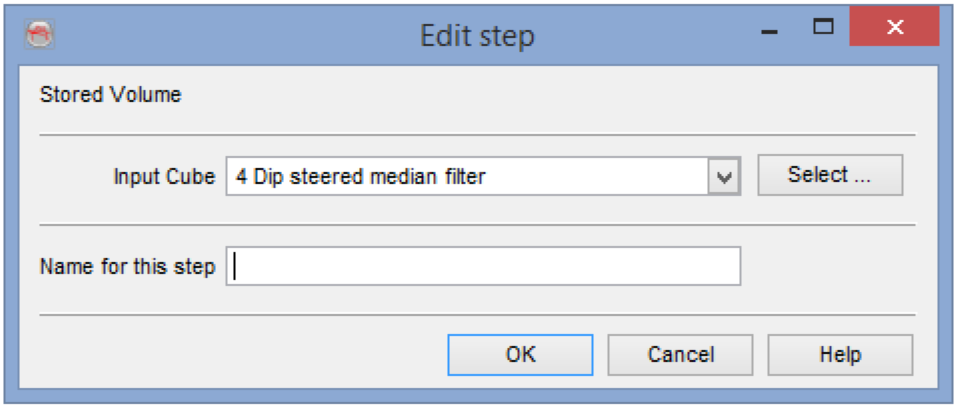


Here you may tag a volume with a velocity type, so that it can be used as input for the gridding step. Or change the tag that the volume has. This can be useful for interpolation of velocity cubes (for example, we strongly advise against trying to interpolate Vint or Vrms, but Eta-tagged volumes can be interpolated):



Input Volume

This step is in general used to provide a background volume before using spatially constrained steps, for instance before the body shape painter or horizon-based painter.

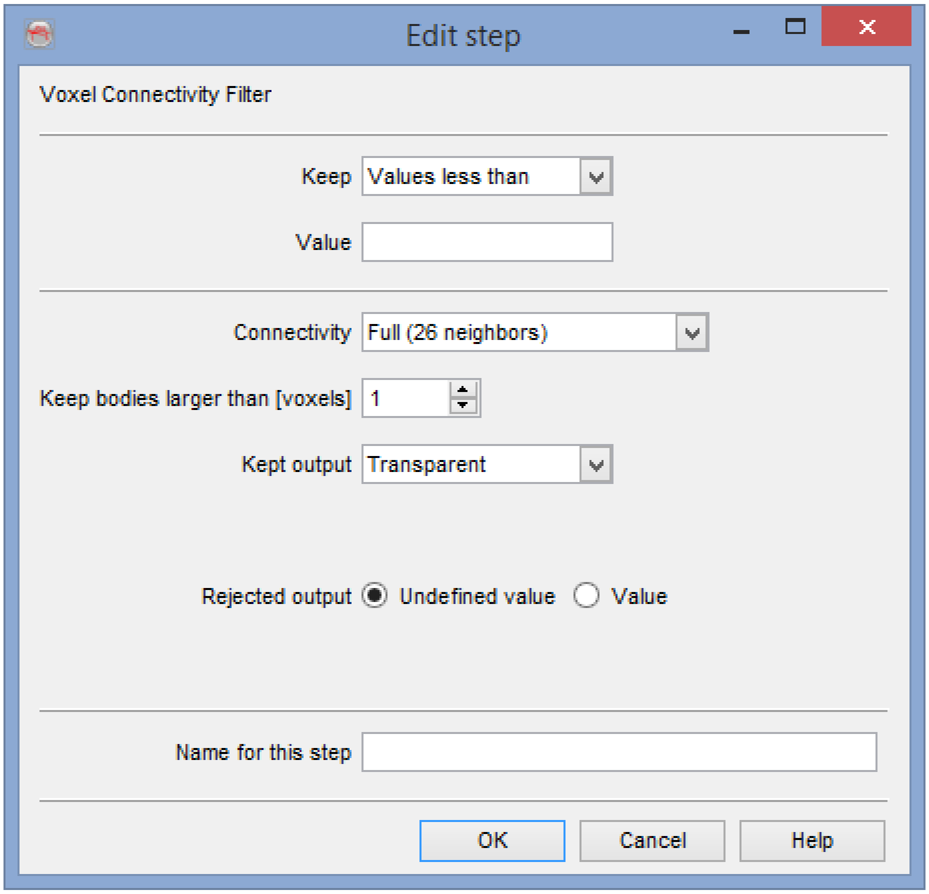


Voxel Connectivity Filter

Voxel Connextivity Filter is a special tool to create continuous bodies based on the amplitudes in a stored volume. A 'voxel' is defined as the volume around one sample. It is thus linked to the survey bin size and sampling rate.

> **Warning 1:** This volume builder step must be preceded by a step providing the necessary input data, like "Stored volume".
>
> **Warning 2:** This volume builder step implies a volumetric calculation. The result of the application on a single inline will differ from the result of the application to the whole volume.

The filter is based on a user-defined amplitude selection to compute the bodies. The samples interconnection is computed based on an amplitude criteria and geometrical spreading settings. It is a very useful tool to visualize seismic attributes in 3D. Other benefits of this tool are to get a volume of several bodies and visualize them in 3D or use it as an input to supervised Neural Network. A general and most popular use of this tool can be the DHIs detection. For instance, if creating a volume that represents DHIs only, it may be interesting to clip the amplitudes to visualize the DHIs present in the seismic data. For such case, when having the seismic amplitude attribute as a volume, this filter can be used to create new DHI volume.



The voxel connectivity filter has a number of parameters to set:

Keep: Specifies the part of the input dataset used to compute the bodies, based on their amplitudes.

- Values more than: The envelope of the amplitudes higher than the given value define the bodies to be computed. Example: 0 will select all positive amplitudes.
- Values less than: The envelope of the amplitudes lower than the given value define the bodies to be computed. Example: 0 will select all negative amplitudes.
- Values between: The envelope of the amplitudes between inside the given range define the bodies to be computed. Example: 9000, 14000 will select all values in between, like 12000.
- Values outside: The envelope of the amplitudes between outside the given range define the bodies to be computed. Example: -10000, 10000 will select all values lower than -10000 or larger than +10000 (the extremes).

Connectivity: Selects the method used to connect different voxels when computing the bodies. Each sample in the input volume acts like a seed.

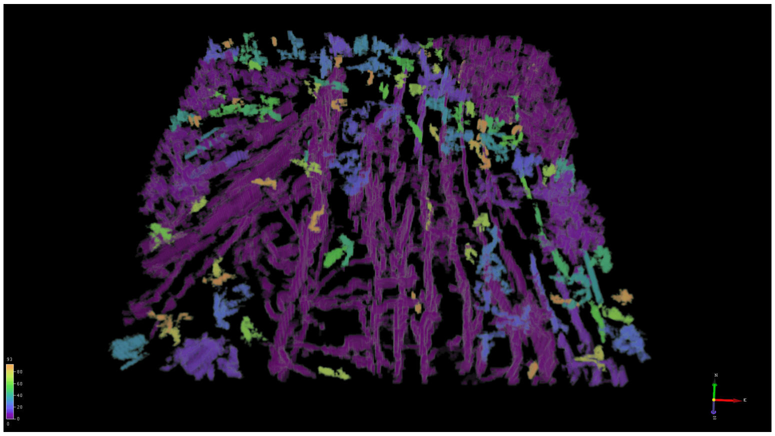
- Common Faces (6 neighbours): The propagation is done by strictly using the 6 faces adjacent to the current seed.
- Common Edges (18 neighbours): The propagation is done by using the 6 faces and the 12 edges adjacent to the current seed.
- Full (26 neighbours): The propagation is done is all directions, using the 6 faces, 12 edges and 8 corners adjacent to the current seed. This is the default mode.

Keep bodies larger than [voxels]: It defines the minimum number of voxels required to output a body. Actually all bodies are computed in the first pass. The smallest bodies are then dismissed. Minimum allowed is one.

Keep output: The following value(s) will be output on the samples inside the computed bodies:

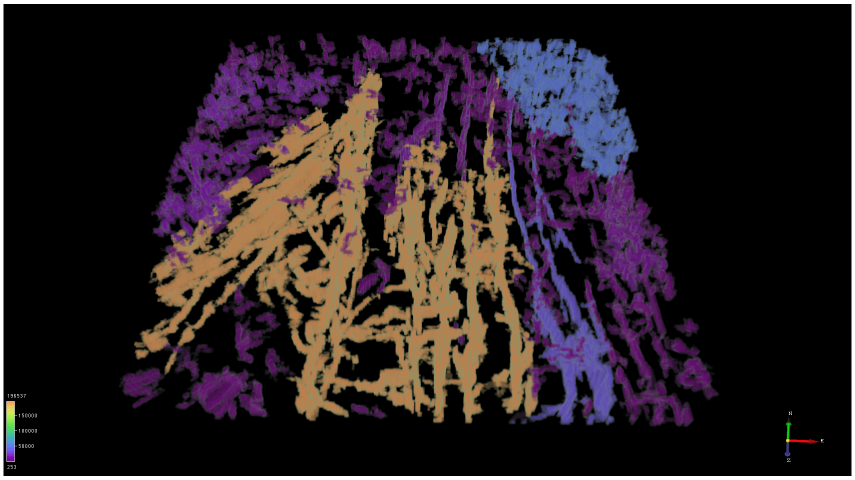
- Body-size rank:The output value is an integer with a constant, different value for each body. The values are sorted by decreasing body size, starting at zero: 0 is the largest body, 1 the second largest...

The example below is created using a similarity attribute to locate faults and fractures in a volume. It is set-up to create bodies connecting low similarity values (threshold of 0.5). All values that are above this threshold are ignored. Furthermore, it is also ignores the very small bodies (size < 10 voxels).
It shows number of connected bodies (purple being the largest ones) in a volume. Such a result can directly show which faults are connected and those that are not. Visualizing such a VCF result can be a valuable method in performing direct interpretation.



- Body-size: The output value is the size in number of voxels of each body. This gives an approximation of the real-world volume, when multiplying by the bin size. For example, a body of 2500 voxels (10 inlines, 50 crosslines, 5 samples), with a bin size 25m x 12.5m, at 4ms sampling with a constant velocity of 2000 m/s: Vol = 2500 * 25 * 12.5 * 2000*0.004/2 = 10000 m3...

In this second example, the same volume is being processed for Body-size. It shows the same patterns suggesting that the prediction is identical to the earlier result. However, the predicted voxels are being filled differently. Here the same bodies are defined by largest volume in cubic meters (m3).
Generally speaking, areas of higher faults/fractures density allow greater connectivity between bodies.This example below shows this case.



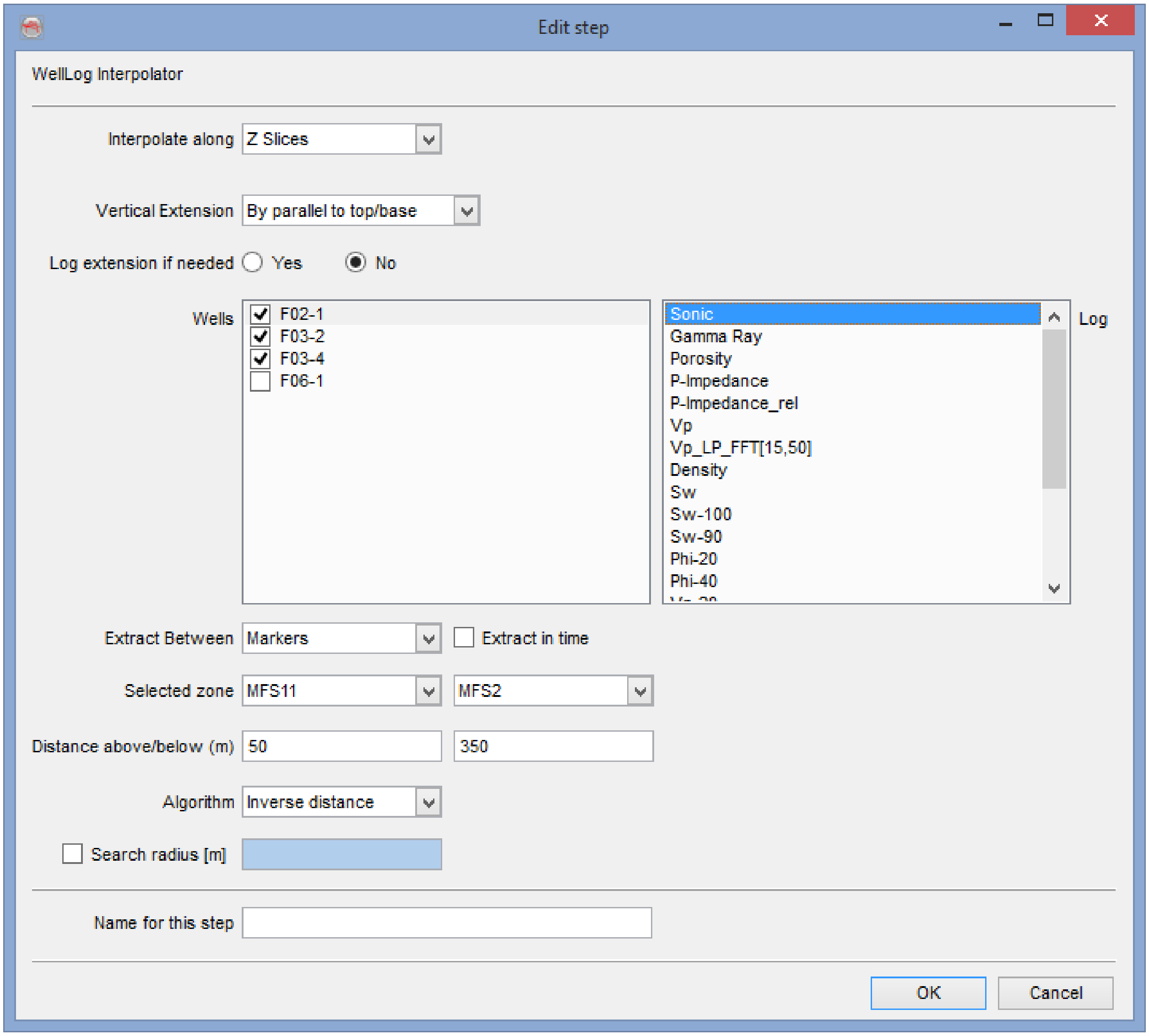
- Value: The output value is a user-defined value specified in the "Kept value" field underneath.
- Transparent: The output value is taken from the amplitude in the input volume.

Rejected output: The value outside the computed bodies can be either the undefined value or a user-defined value specified by the field "Rejected value" underneath.

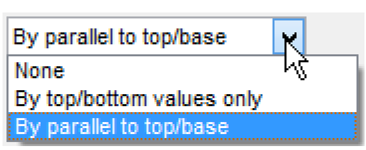
Name for this step: Provide a user-defined name for this volume builder step that will appear in the Used-steps list of the Volume builder.

Well Log Interpolator

This gridding step is used to populate a 3D volume using well logs by interpolating along Z-slices.



Vertical Extension: Select the method of vertical extension from the following options:



Log extension if needed: Extend the logs (if required) to match the *Selected zone*

Extract Between: Extract data from a marker-defined, depth-defined or time-defined range. You may also toggle on the option to extract the data in time.

Selected zone: Set the extraction zone suing either markers or start and end of data (or combination thereof).

Distance above/below: Extend, if desired, the extraction zone above and below the selected zone.

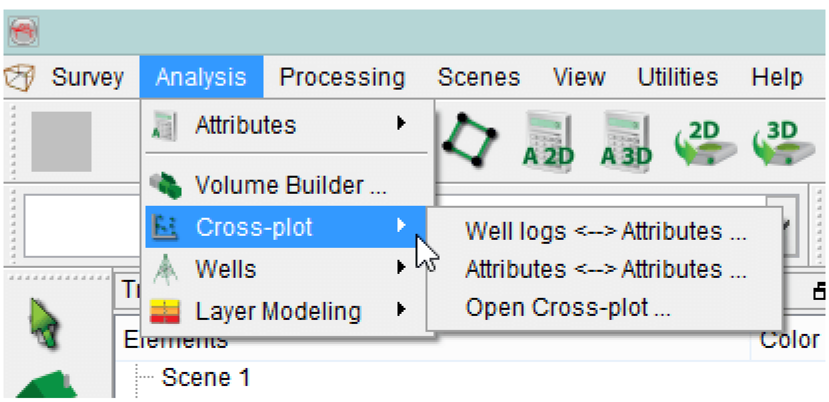
Algorithm: Choose between inverse distance or triangulation.

Search radius: For inverse distance only - set an optional maximum search radius for the algorithm.

After the selection of well(s) and log, parameters and algorithm, provide a name for this step at the bottom and proceed to the Volume Builder by pressing *'OK'*.

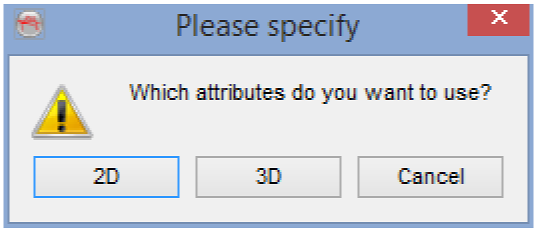
Cross-Plot

The *Cross-plot* tool is designed to create two dimensional cross plots between 2D/3D seismic data (attributes) and either other attributes or well data. The data can be analysed in multiple maners, using different kinds of colour coding and data selection tools. It may be launched from the menu *Analysis --> Cross-plot menu*.



Cross-Plot Data Extraction

The crossplot data must first be extracted, either on (a subset of) the horizon or along (deviated) well paths. 2D or 3D attributes can be used, and well logs if the extraction is done along the well paths. The extracted data will first be presented in a table before actually selecting the features to cross-plot.



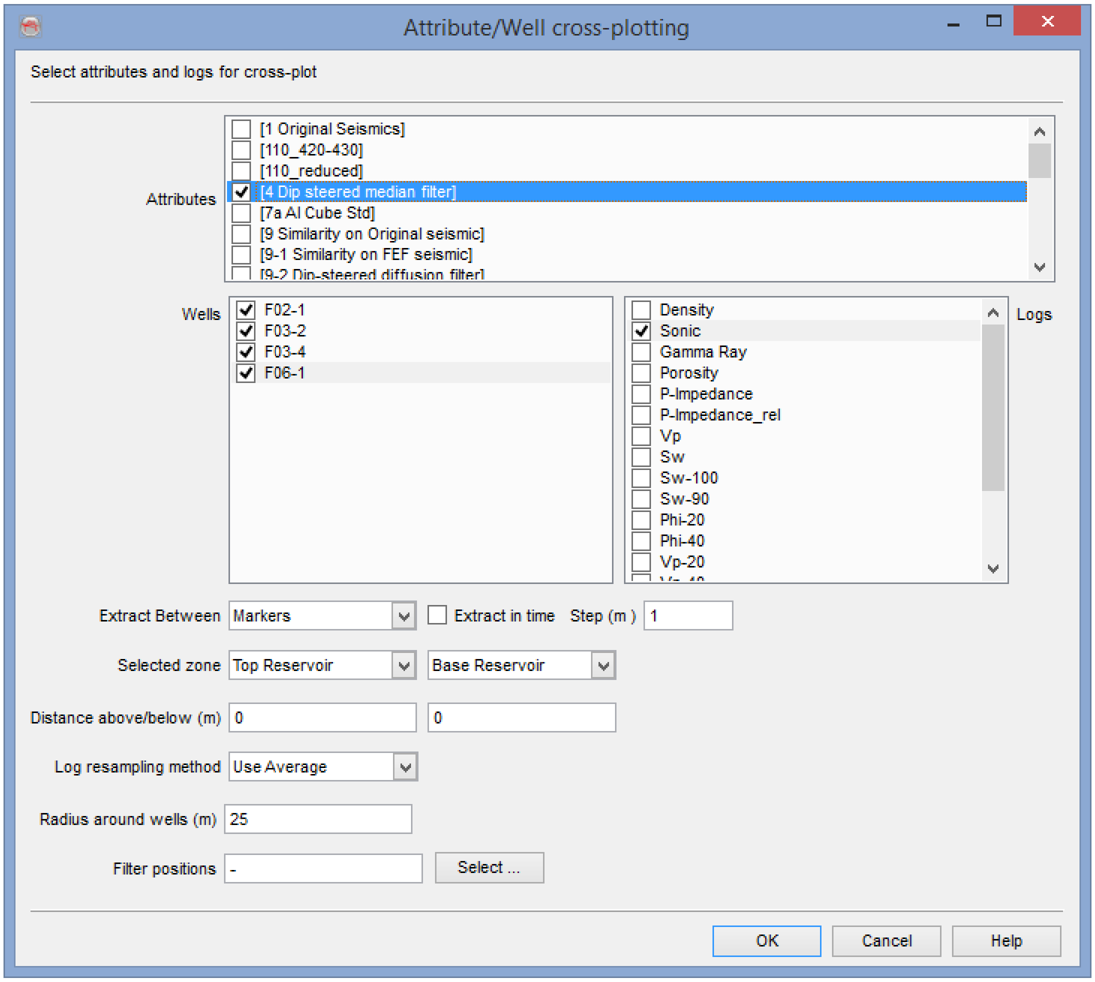
The extracted data can be saved in the cross-plot table window and reopened without repeating the data extraction, from the menu *Analysis --> Cross-plot --> Open*.

Well-based Data Extraction

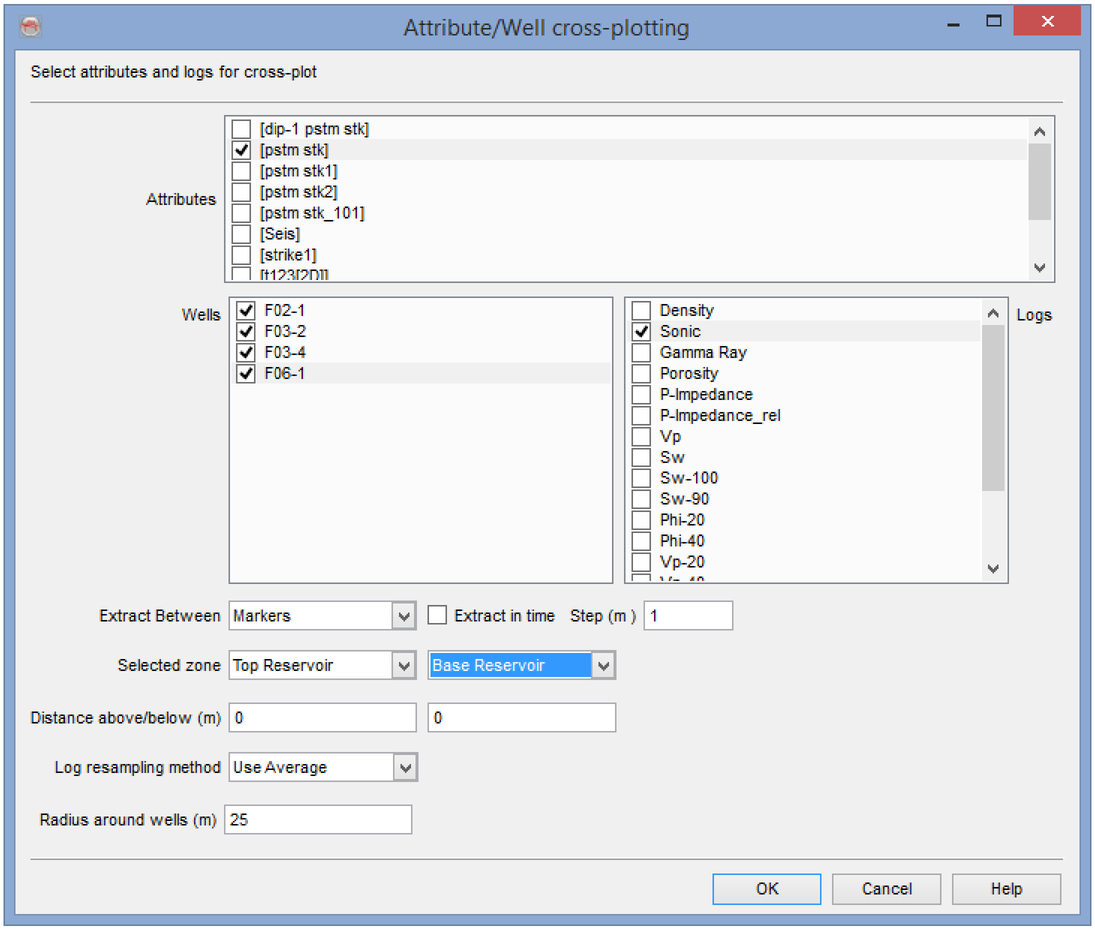
This window presents the attributes and/or logs that can be extracted along well path. The output will be presented in table before being used for cross-plot. At least one well must be selected, and one attribute or one log. It is also possible to select only attributes, or only logs.

The well track and time-depth model provide the locations where to extract the data. Values will be vertically extracted along a specially built measured depth axis. This axis is such that the step between two consecutive depth samples is constant but with few jumps, such that the Z difference (time or depth depending on the survey type) between consecutive depths is around the survey default sampling rate. Therefore at shallow level 1 seismic sample can correspond to 4 meters, then 8 meters at intermediate depths, 12, 16 and so on.

- Attribute values are vertically interpolated along that created MD axis, since they are unlikely to be along the Z axis defined by the survey geometry. A polynomial interpolation is performed.
- Log values are extracted in the depth domain around the depth to be computed, plus or minus half of the distance to the previous and next depths. All collected values are then processed (up-scaled) using a provided "Log resampling method" (see below).



3D Data extraction for Well vs. Attributes Cross-Plot



2D Data extraction for Well vs. Attributes Cross-Plot

The following specific extraction parameters are available:

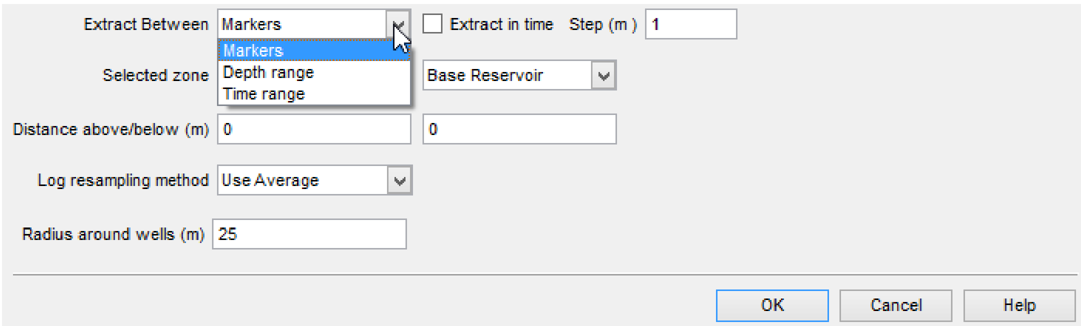
- **Extract between:** It is used to limit the z-range (depth or time) of the data to extract. There are three options supported: *Markers*, *Depth* and *Time*.

  If *Markers* is selected (which is default), the Start/Stop markers should also be selected from the combo boxes that are available below the extract between field.

  If *Depth* is selected in the extract between field, the start/stop (m) field will be toggled on. In the later fields, starting and stopping depth range is typed in to restrict the data extraction into an interval.

  Similarly, if *Time* is selected in the extract between field, the start/stop (ms) field will be toggled on.

Finally, the step-out for extracting both *Attributes* and *Well data* samples has to be defined. This can be defined in meters (default), feet (if the survey is in feet) or milliseconds. The *Extract in time* check box is usually toggled on if you want to define the data extraction step-out in TWT. It is advisable to check this box when extracting data for crossplotting against seismic volumes.

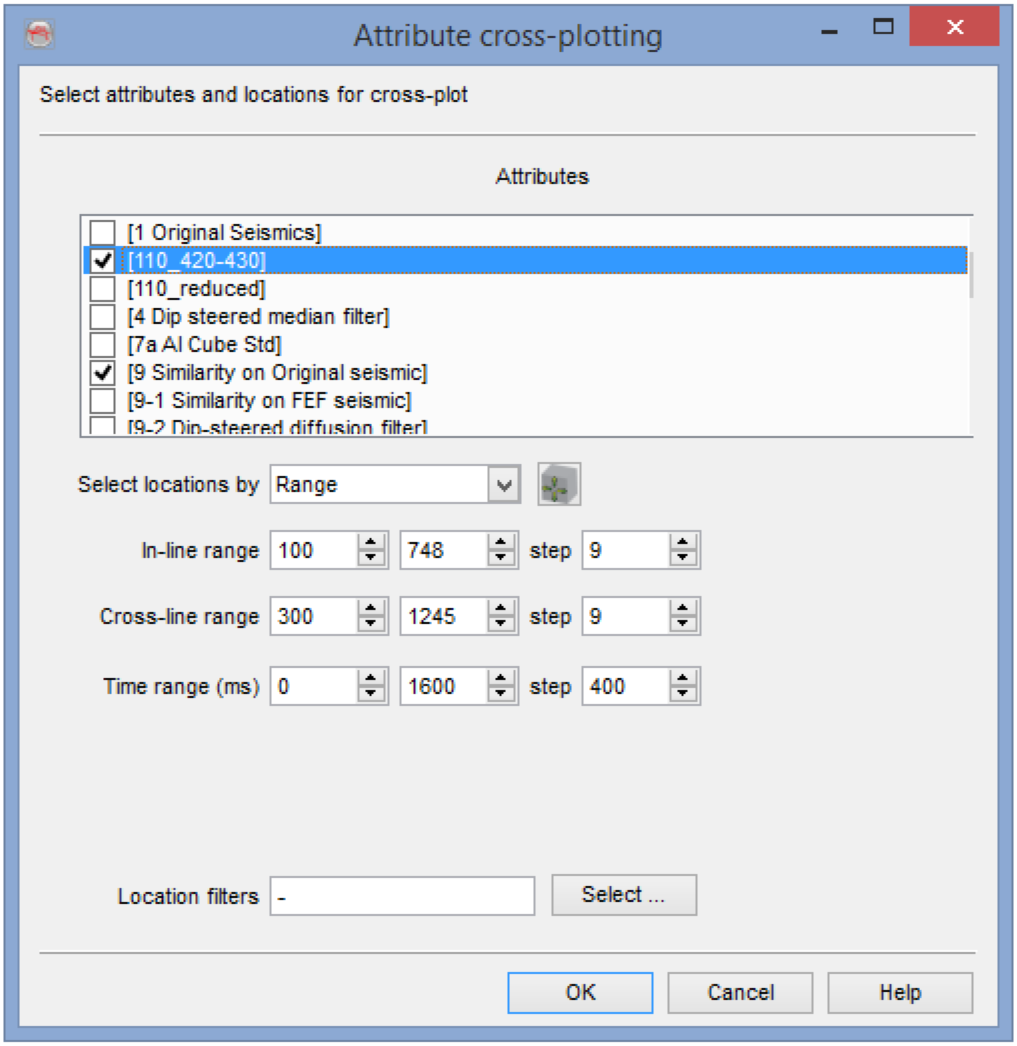


- **Distance above/below:** It is used to modify the vertical range of the extraction window using a relative distance from the provided well markers, in depth. A negative number will decrease the extraction window, a positive number will increase it.
- **Log resampling method:** Logs will be up-scaled using this method. *'Average'* should be used for most of the logs. *Median*, most frequent and nearest sample are more fit for discrete logs like lithology, but can also occasionally be used for other types.
- **Radius around wells:** All traces that can be reached within the search radius will be extracted. If several traces around a well are found, the same extracted log value is posted in front of the collected attributes values. This option will only duplicate all data if no attributes are extracted. The default value is the survey bin size, *use value "0" to extract only the nearest trace*, i.e. one value per well per depth.
- **Filter positions:** See the location filters section in the same chapter.

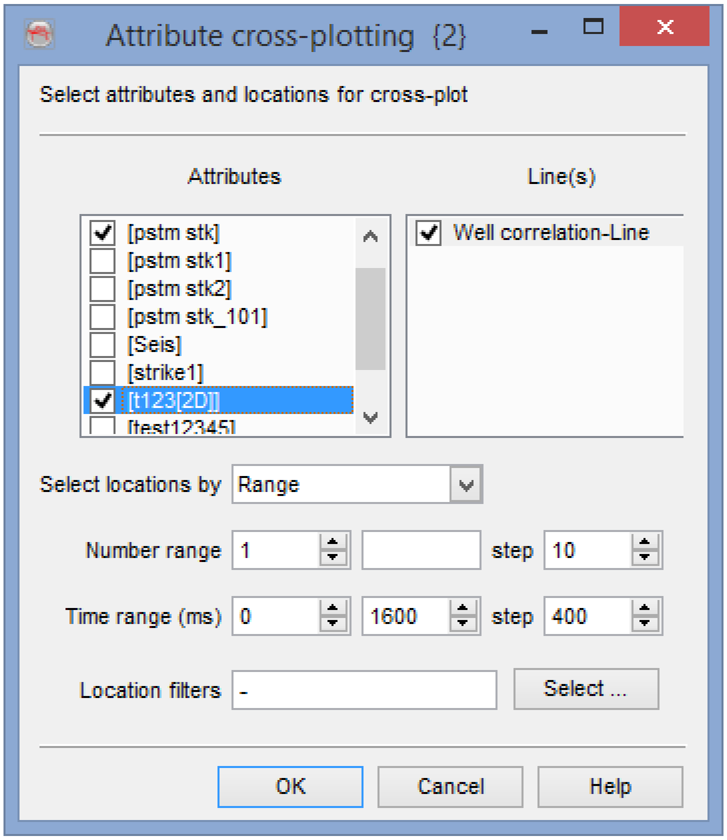
Attribute-based Extraction Window

Attributes based data extraction window is used to extract an attribute data (stored volumes or a defined attribute) within a volume defined by a range, polygon, surfaces, body or a well path with lateral extension. The same window (shown below) could also be used to extract an attribute data along a time slice or along a surface.

Note that at least one attribute must be selected prior to data extraction. "Attributes" list shows all attribute currently loaded in the window, and the stored volume under brackets. For multiple attributes selection, use the left mouse button by holding and dragging it up/down-ward. For 2D data extraction one or more attributes along-with their corresponding "Line names" (at least one) should be selected.



3D data extraction for Attributes vs. Attributes Cross-plot



2D data extraction for Attributes vs. Attributes Cross-plot

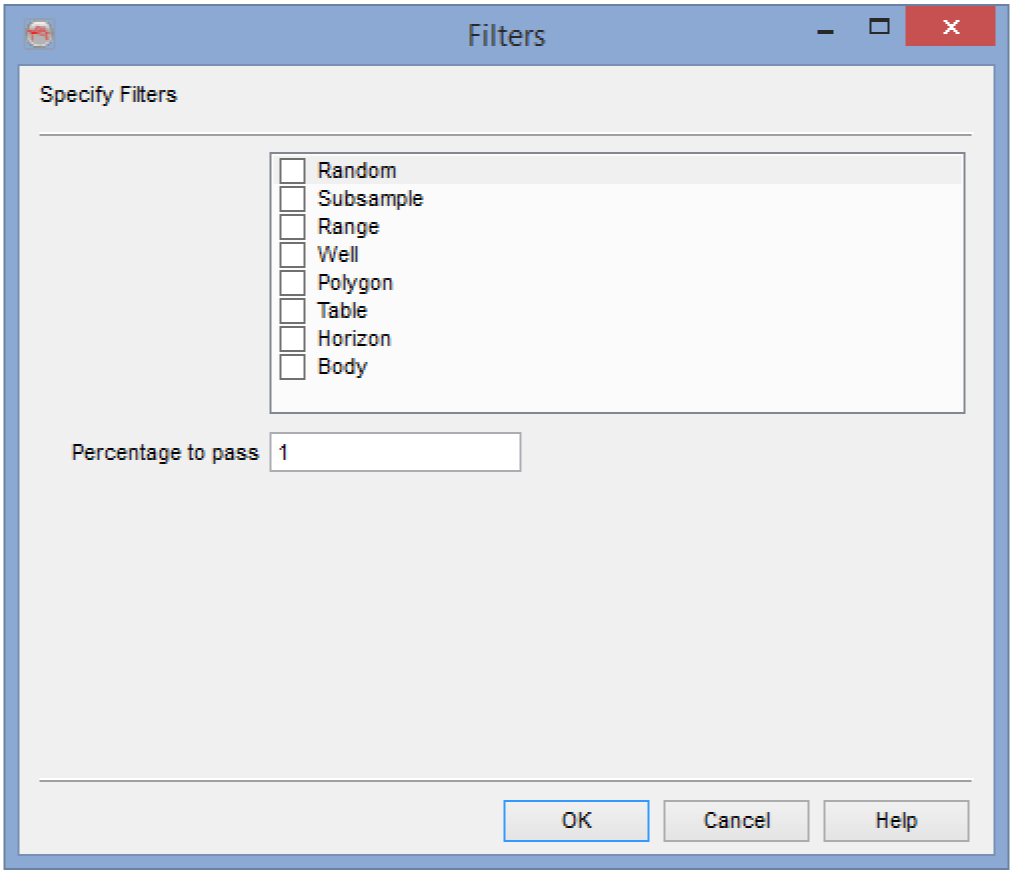
The volume/horizon based extraction is performed by selecting locations according to one of the following criteria:

- **Range:** It is selected to extract the data on a regular 3D volume or a grid (if it is a time slice). The steps are the increments in the corresponding range. To extract dense data points for a crossplot, smaller steps should be used. Note that the larger steps will decrease the amount of extracted data. Cubes coarser than the requested grid will not be interpolated, undefined values will be posted to the cross-plot table instead. *For 2D data extraction this is the only possible option and only the time range can be specified*.
- **Polygon:** The lateral extent for a crossplot data could also be defined by a polygon. Once the *Polygon* option is chosen, the desired polygon is chosen by pressing the *Select* button. The inline/crossline steps are the increments in the inlines/crosslines range within the polygon. The time range is also an additional constrain in data extraction that defines a vertical restriction window for a polygonal type of volumetric crossplot.
- **Table:** Only positions listed in a table will be used for the extraction. The table might be an OpendTect pickset, or a column-sorted ASCII file with inline, crossline and Z values in the first three columns respectively.

- **Surface:** Used for data extraction along a 2D/3D horizon, or between horizons. Please note that the attributes will be interpolated if extracted along an horizon. If the extraction is done between two horizons (volume based extraction using a user-defined Z step), it will not interpolate the attributes. The "Extra Z" values increase or decrease the extraction window size, and work similar as the attribute set time gates (relatively). The left value applies to the top horizon and the right value applies to the base horizon.
- **Body:** It is used to restrict the data extraction within a selected 3D body. The radio boxes inside/outside are used to extract the data either inside or outside the selected body. If it is outside, the further ranges are sub-selected in the *Within bounding box* field.
- **Well:** It is used to extract attribute data along the selected well paths. The data would be extracted vertically and according to the (TWT/depth dependent) survey setup. The time gate is defined by providing a time range with time steps (vertical sampling rate).
- **Location filters** can be added in order to add one or several restrictions to the area of extraction.

Location Filters

The filters should be used to further limit the amount of data to be extracted for making cross-plots. Several filters can be used simultaneously.



Filters are of two types:

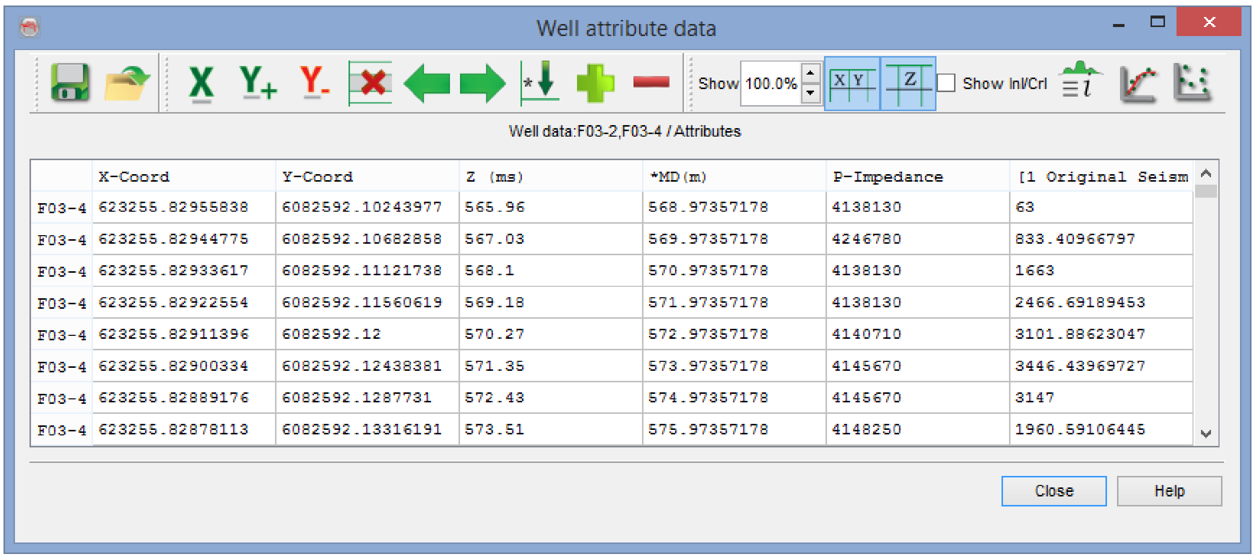
The first two filters, *Random* and *Subsample*, are not position related. The *Random* filter passes a certain percentage of random samples selected in the main extraction window, while the *Subsample* filter will pass a finite (user-defined) number of samples. For instance, for the *Random* option, if the value is 1, only 1% of all extracted data would be selected for cross-plotting.

The last four - *Range*, *Polygon*, *Table* and *Surface* (see previous section for definitions) are position based filters. These are used to define sub-areas that will complement the extraction settings provided in the main extraction window. Multiple filters can be chosen out of these. Thus, the points satisfying the main extraction settings and all defined filters will be used for the extraction of attributes.

Cross-Plot Table

The crossplot table displays the extracted data. It is used to edit and plot the data for a crossplot. A row in the table corresponds to one extracted data point, annotated by its position (X, Y and Z) and followed by the collected attributes values (forming at vector, from left to right: logs, attributes, stored data). The star adjacent to an attribute name indicates sorted column. Empty cells represent attributes than could not be extracted with the provided the settings for data extraction.

The table enables the manipulation and edition of the collected data, prior to making cross-plots. For instance, it is possible to sort the data from an attribute, and to delete the first or last rows, before plotting the data. Please note that the table window is interactively linked with the cross-plot window. Any editing done in the cross-plot window will reflect in the table window that remains open and active while working in the cross-plot window.

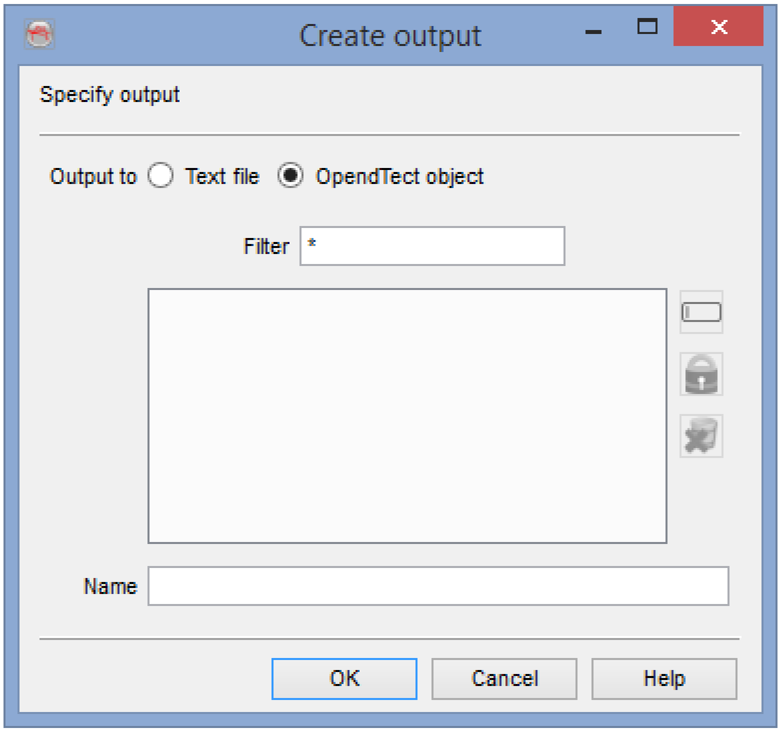


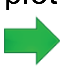
Standard workflow: Save, edit, click in a column and then X, click in another column and then Y+, launch the cross-plot window.

Note: You can select a column by either clicking on its title cell or by clicking on any single cell.

The toolbar contains the following editing tools:

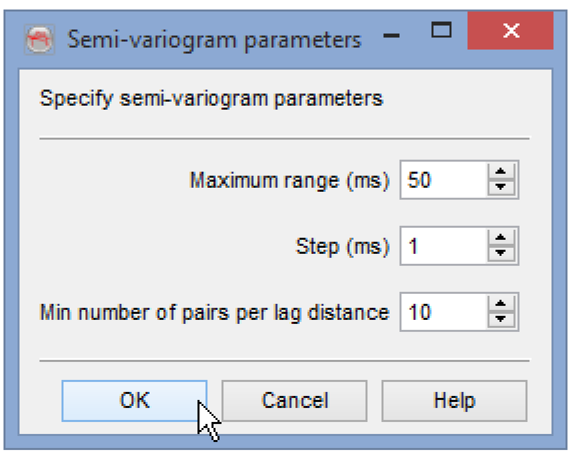
- Saves the data shown in the table to a file (simple text file or OpendTect object). OpendTect object is a special format to retrieve (open) the cross-plot. The format is called position vector data and the data is saved in the survey sub-directory (/Features/*.pvds). The Text file selection outputs the data to an ASCII (column sorted) file that later on can be used in 3rd party softwares e.g.Excel.

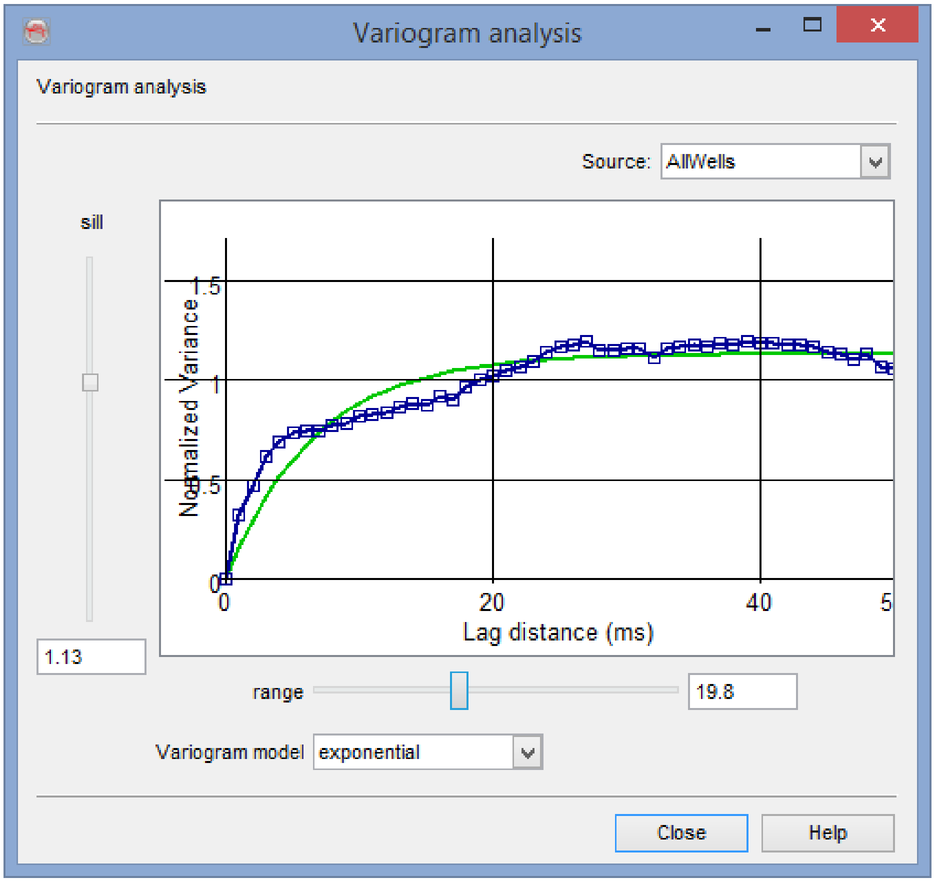


-  Open/Retrieve the stored crossplot data. An alternative short-cut is present in the menu *Analysis -> Cross-plot -> Open Crossplot*.
-  Assigns the X-axis of the cross-plot to a selected/highlighted column.
-  Assigns the Y-axis of the cross-plot to selected/highlighted column.
-  Removes the selected Y-axis for cross-plotting.
-  Removes the selected rows in the table.
-  Moves the selection of the primary Y-axis one column to the left. The cross-plot gets updated accordingly.
-  Moves the selection of the primary Y-axis one column to the right. The cross-plot gets updated accordingly.
-  Sorts the selected column to an increasing order, from top to bottom.
-  Add an empty column in the table. In the pop-up window, the column name is provided. The mathematical operation is done to compute the data for the new column. An example is the Acoustic impedance data computed from the velocity and density logs available in the crossplot table. For further information on the

mathematical operators, please find the description on the Mathematics attribute in the Appendix.

- Removes the selected column from the crossplot.
- Show 100.0% Displays selected percentage of the data. If a lot a data was extracted not all will be displayed in the table. Nevertheless the sort and remove lines tools may still apply to all the extracted data, upon request of the user.
- X Y Toggles on/off the display of the coordinates.
- Z Toggles on/off the display of Z values.
- Shows histogram and statistics on the selected column.
- Allows to achieve *vertical variogram analysis*. A variogram describes the spatial continuity. The vertical variogram can be computed for any of the attribute or log from the extracted data. To achieve a variogram analysis from well log, the log data is resampled at the variogram processing step and de-trended prior to the variogram computation itself. In the pop-up windows, the different parameters can be defined: the maximum range (maximum distance allowed between the pairs for the analysis), the step and the minimum pairs per log distance. Once these parameters provided, the variogram is computed and the analysis can be performed.





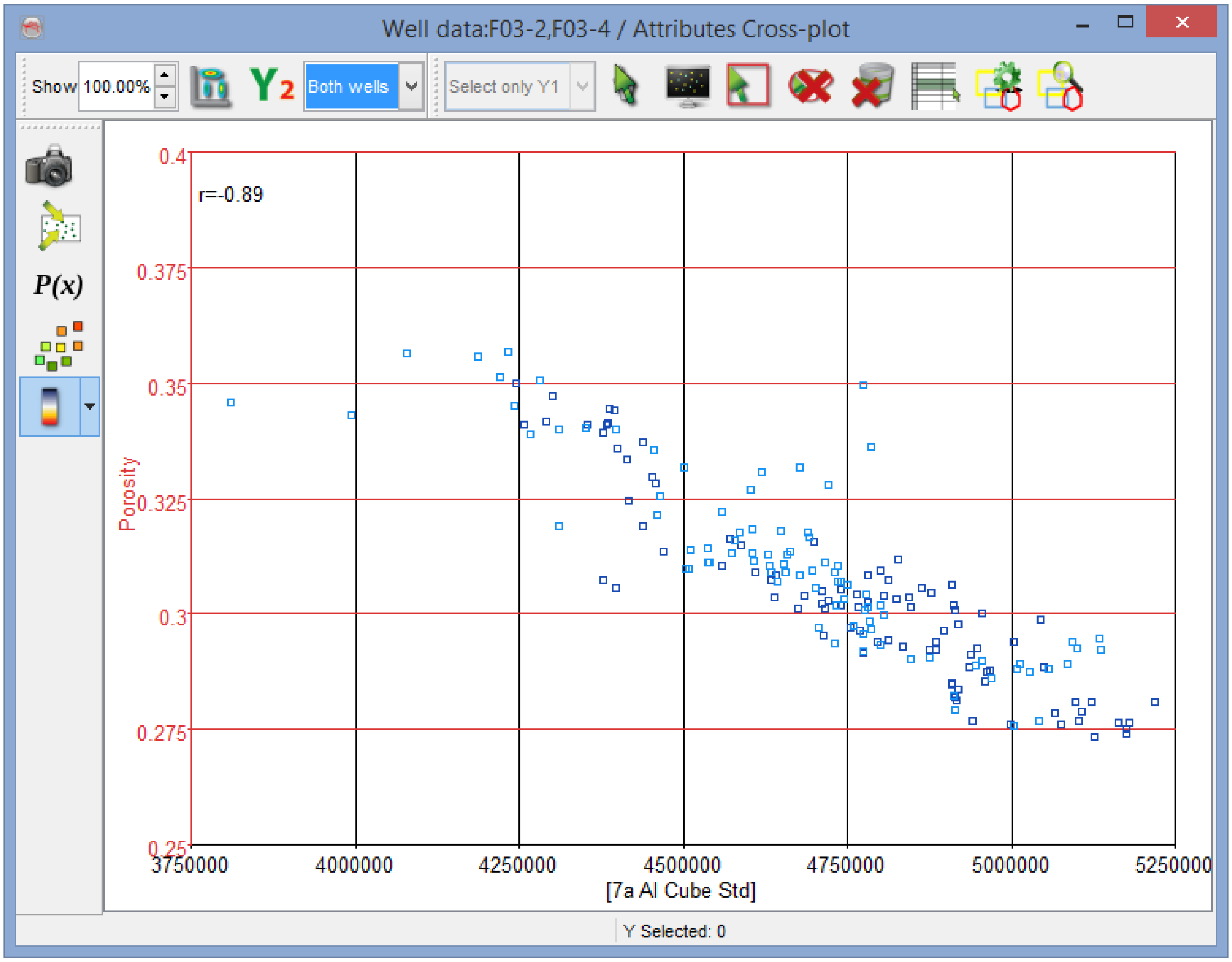
Variogram analysis: Set parameters (left), main window (right) : blue = real data, green = model

A synthetic variogram can be set in changing the sill, the range and also the variogram model (exponential, spherical, gaussian). The objective is to get a synthetic variogram that best describes the real variogram. The data can be analyzed for each well or for all of them. The analysis results can be used when achieving inversion.

-  Launches the crossplot window.

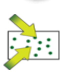
Cross-Plot Window

The cross-plot window shows the data previously extracted and shown the cross-plot . The window may start empty if insufficient data was selected in the table window.



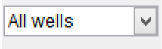
The cross-plot window contain three toolbars to perform various tasks:

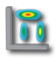
Crossplot manipulation toolbar

-  Take the snapshot of the cross-plot.
-  Gives access to the main crossplot window properties: scaling, statistics, regression line, density plot parameters.
- *P(x* Creates multivariate from the cross-plot data.
-  Colour code the points with respect to a .
-  Colour code the points with respect to the wells they were collected.

Crossplot display toolbar

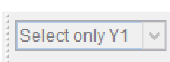
- 

Is used to update the amount of displayed scattered points in a cross-plot. It increases the efficiency to display the data quickly. Normally, a huge data slows down the machine performance, when displayed as scattered points. To avoid this, this option is used to display a selected percentage of the points. Moreover, the percentage is only set for the data that has values; undefined values are ignored in the percentage.
- 

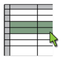
The scattered points of individual/all wells are displayed.
-  Used to show a coloured density cross-plot. Normally, if the displayed scattered data points are too large, it consume huge memory to display the data. In that case, the density plots are useful. In a density plot the data will not be filtered out in the display (*% point displayed*) as in the case of a scattered point display. In the cross plot properties window, there is a tab available for the . In that tab, a user can set a number (currently displayed data) to display the data. When this plot is ON, there will be a colour table displayed that can be changed. The colour spectrum designates the distribution of the points per pixel.
- 

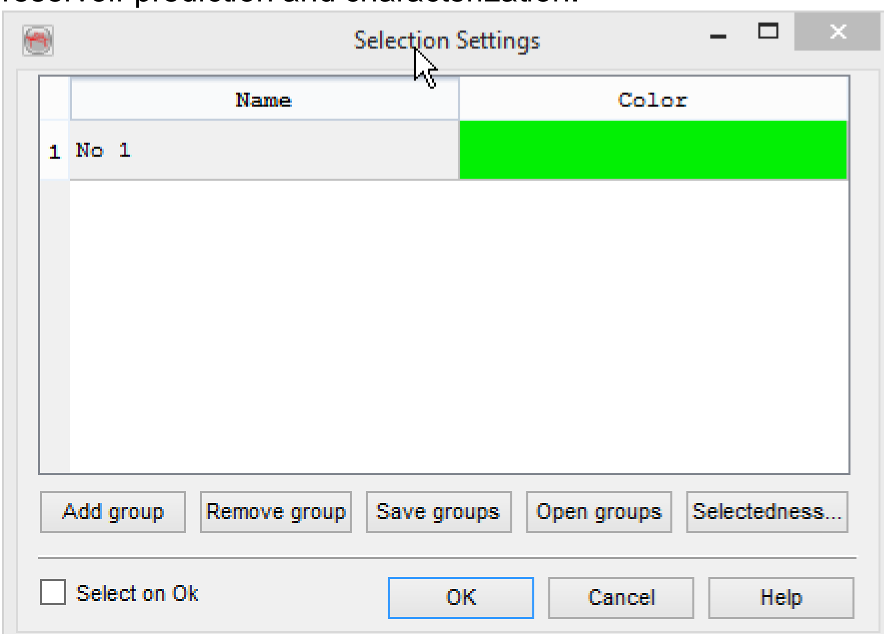
Used to Toggle on/off the second Y-axis (Y2) scattered points. It may be noted that when the second Y-axis (Y+) is selected to be cross plotted against Y1 and X, the data points may become too large to be displayed. Therefore, in this case, the system will prompt a warning to display a given percentage of the data *(% points displayed*).

Crossplot selection toolbar

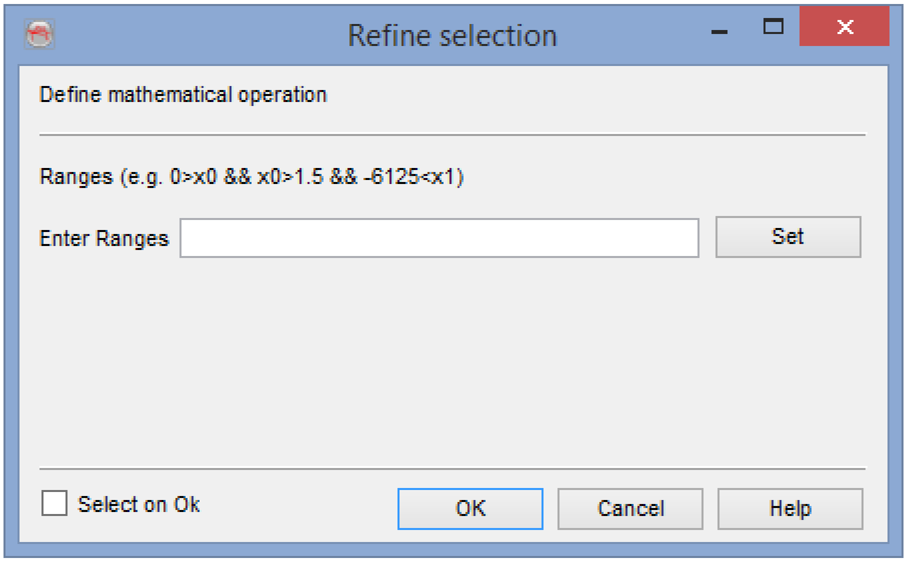
- 

Is inactive, if a cross-plot is created with one Y-axis. It allows to make section of the scattered points. The selection settings (*Select only Y1/Y2*, both) are important to remove unwanted points from the extracted data. When dual Y-axis are cross-plotted, a user can select individual or both Y-axis points by changing this option.
-  Toggles a pan/selection mode. Selection mode is used to select the scattered points. When pan is toggled on, rest selection options (see below) become inactive, which avoids editing/manipulation of the cross-plotted data.
-  Used to display the selected scattered point in an active scene. The selection of the data points is done by using selection mode. By using this option, the selected scattered data can be saved as a pickset/body. Right-click on an pick in a scene and from the pop-up menu select 'save as a pickset' or 'create body' option.
-  Selection mode (rectangle/polygon). The selection is made by holding the left-mouse button down and selecting the desired area within the cross-plot.

-  Used to de-select the selected data points (using selection mode tools).
-  The unwanted data points can be removed by using selection mode and this trash button. In order to remove the data points of Y1, Y2 or both, use selection mode tools to select an area within which the data is to be removed. After that press this button to remove the data.
-  Used to select the data from a cross-plot and display the corresponding selected data in the spread-sheet (Well/Attribute data window).
-  You can do multiple selections by adding new groups (see below). It is launched using this manage selection button. The multiple group selection allows you to select different clusters/trends on a crossplot in the form of groups. Second and subsequent selections are made by clicking on a group name and holding the CTRL key down from a keyboard prior. Then the corresponding polygon (with a given colour) is drawn over the crossplot display area. It is a very useful tool for reservoir prediction and characterization.



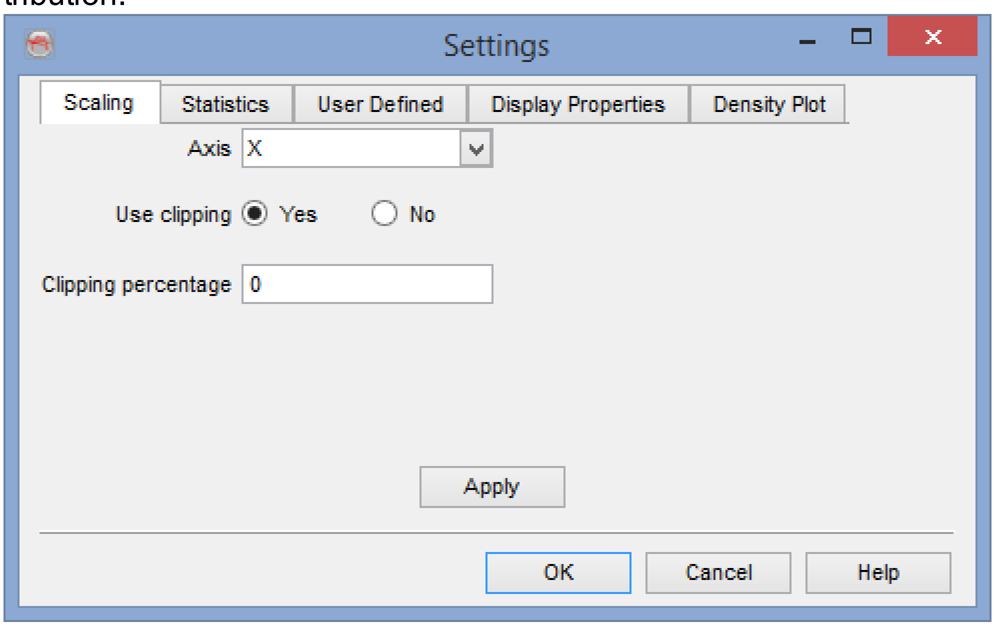
-  This option is used to by using a mathematical logic over a range. It restricts the selection according to the range set in the *Refine Selection* window (a pop-up window invokes when this button is pressed). For instance, a user may want to remove a data (x0) within a range of 3-4 from a cross-plot in which 1-2 values are overlapping. To do that, press this button to launch *Refine Selection* window. In the *Enter Ranges* field, an equation can be set i.e. X0 > 3 and X0 < 4. Where X0 is desired data in a cross-plot. After this a section can be made within a cross-plot to remove values within the polygon according to the equation.

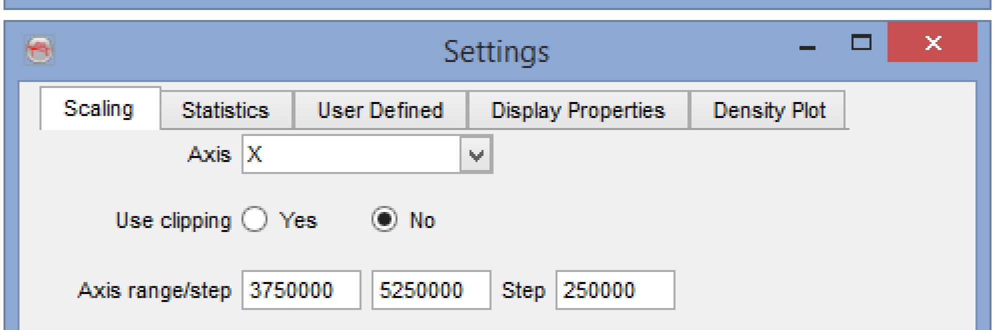


Cross-Plot Properties

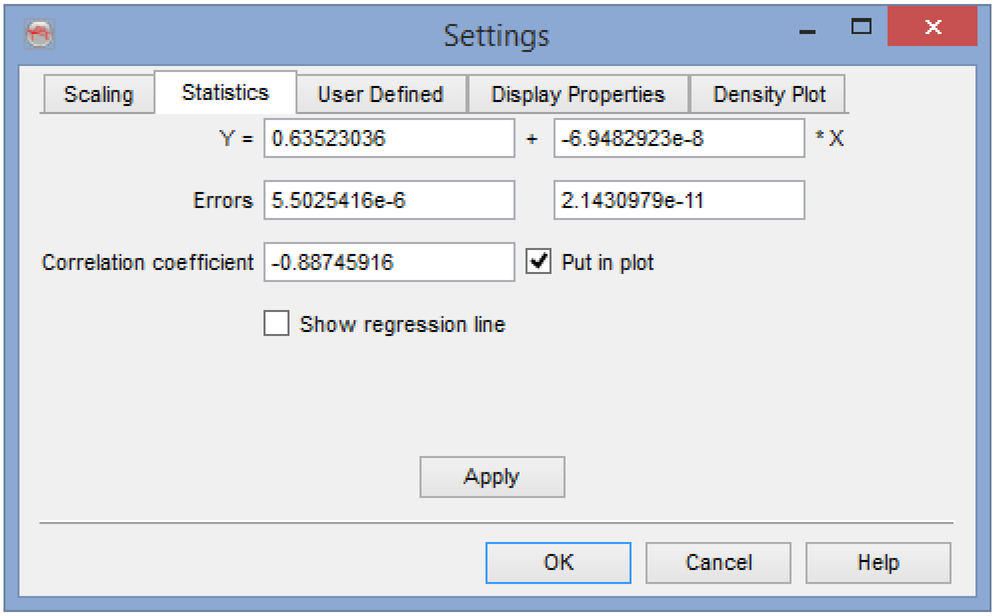
The properties window (accessed via )can be used to adjust the scale, view statistics, add regressing lines etc.

Scaling Tab: Sets the clipping state for each axis, or the amplitude range for the chart. Default: 0, which means that the window is adjusted to fit the entire amplitude distribution.

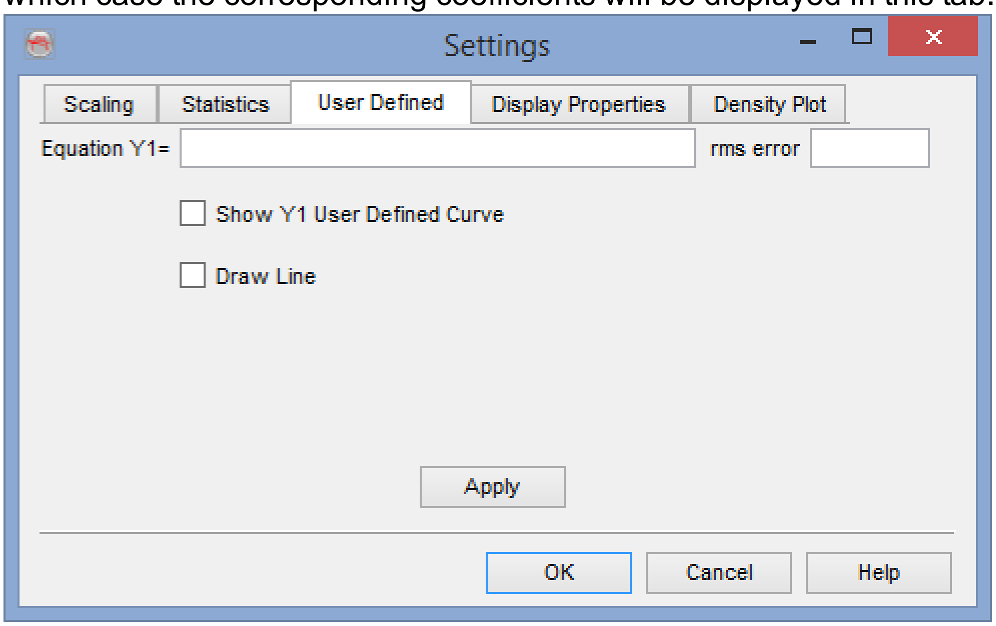




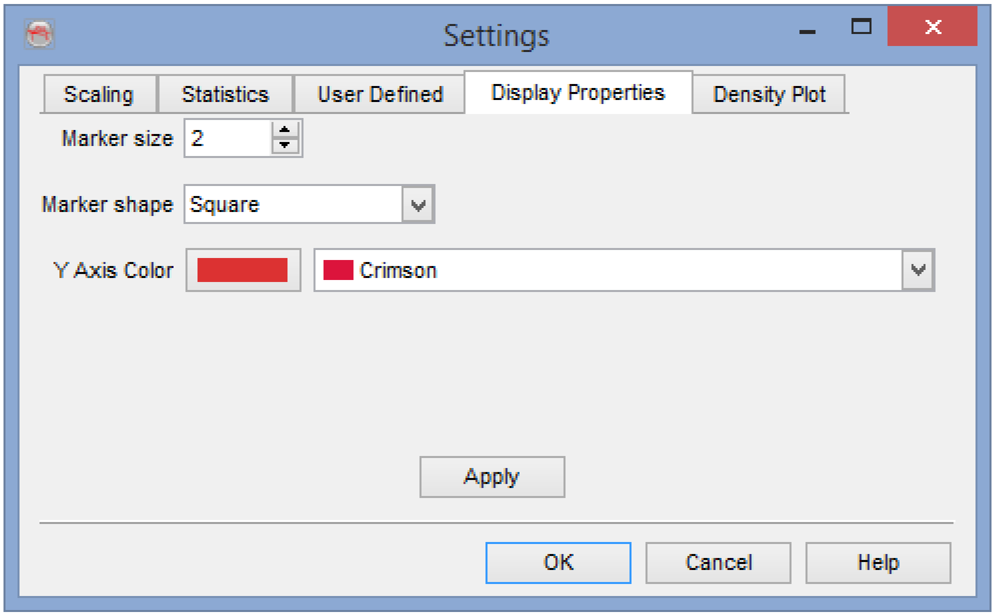
Statistics Tab: Shows the parameters of a least square fit between the attributes used as X and Y1 (values and errors). The regression line can be displayed in the cross-plot window, as well as the correlation coefficient.



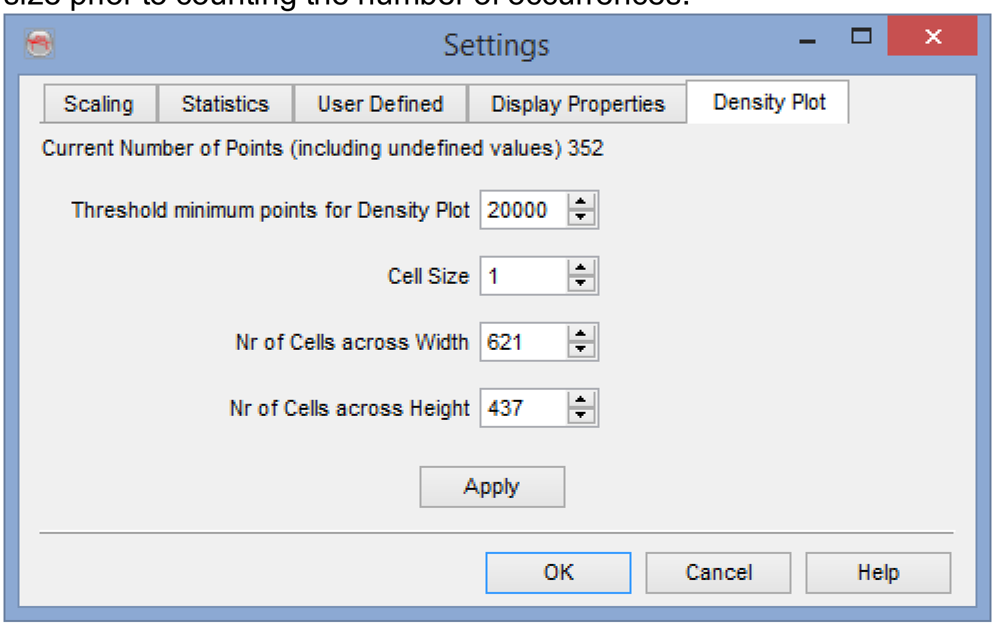
User Defined Tab: Sets a user-defined regression line, that can be displayed as well in the cross-plot window. A simple line can also be drawn in the cross-plot window, in which case the corresponding coefficients will be displayed in this tab.



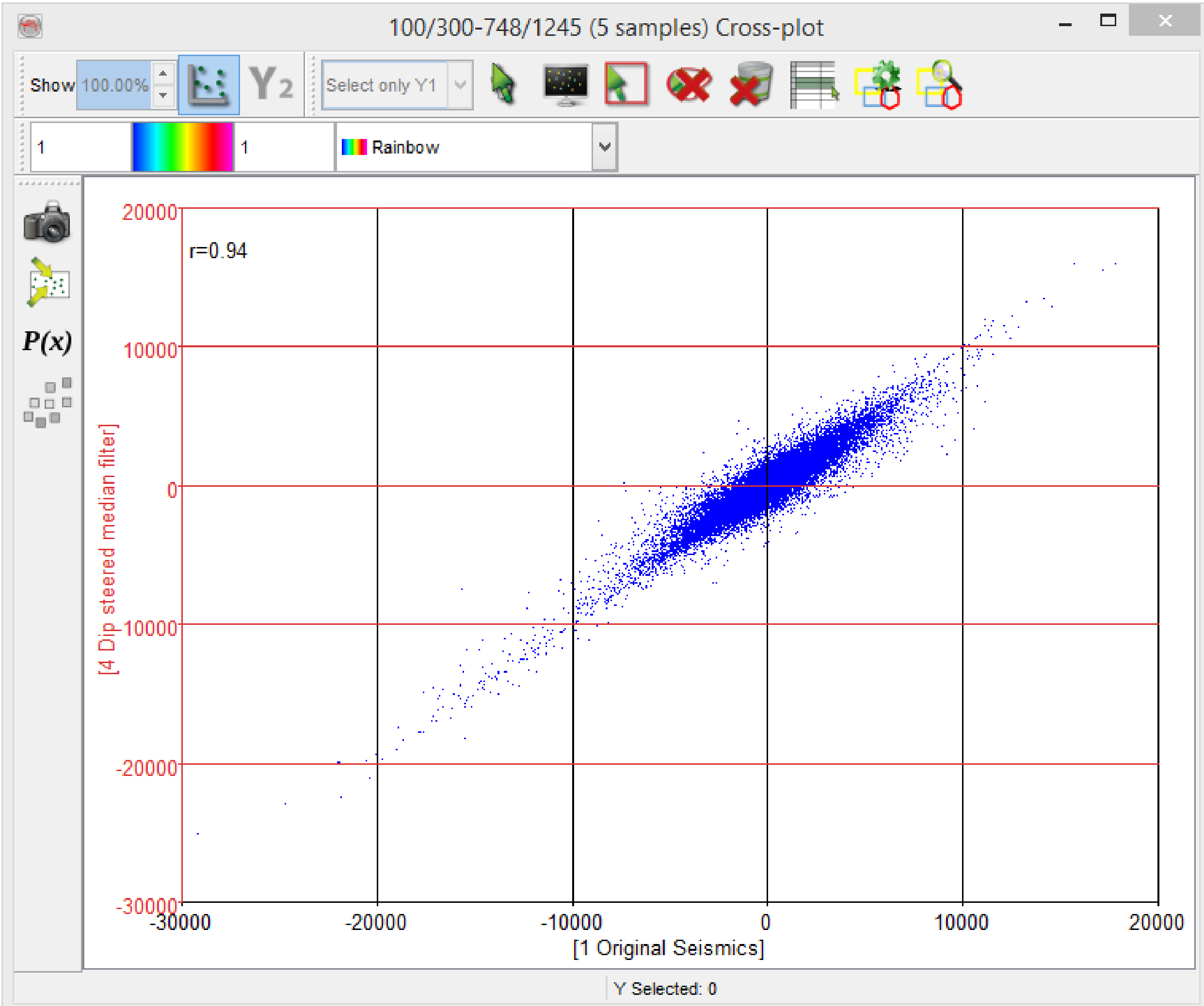
Display Properties Tab: Sets user-defined marker size, marker shape and the Y-axis color.



Density Plot Tab: This tab is used to set the minimum points for the automated density plot. Scatter plots will not be allowed below that number. The tab settings define the bin size prior to counting the number of occurrences.

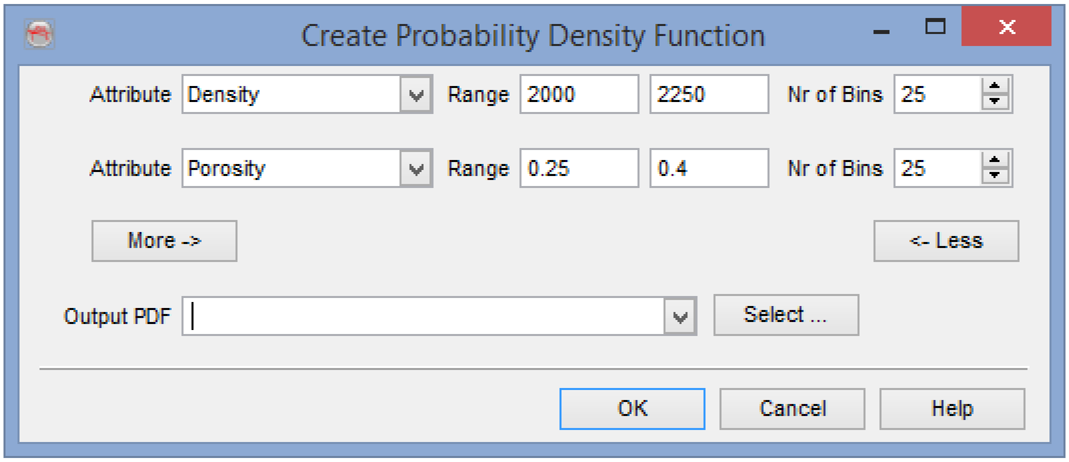


The following pictures shows an example of density plot. Please note that an additional colorbar has appeared. Units are the number of points that correspond to the color.



Probability Density Functions

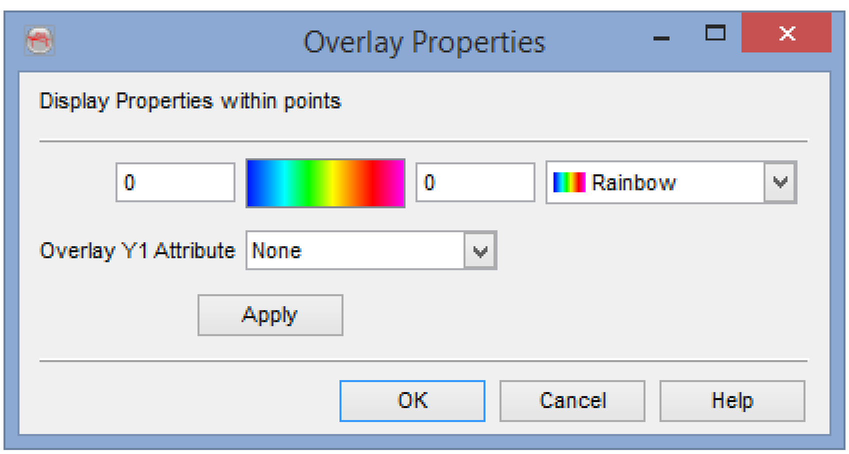
Probability Density Functions (PDFs) can be created from the cross-plot toolbar icon *P(x*. This icon launches a pop-up dialog that can be used for selecting attributes in order to create PDFs. The PDFs are stored in OpendTect Format, that can later be used for running Bayesian classifications.



Please note that all attributes from the table can be selected. Attribute ranges are generated automatically to fit the extracted data distribution. These can be edited before creating the PDF.

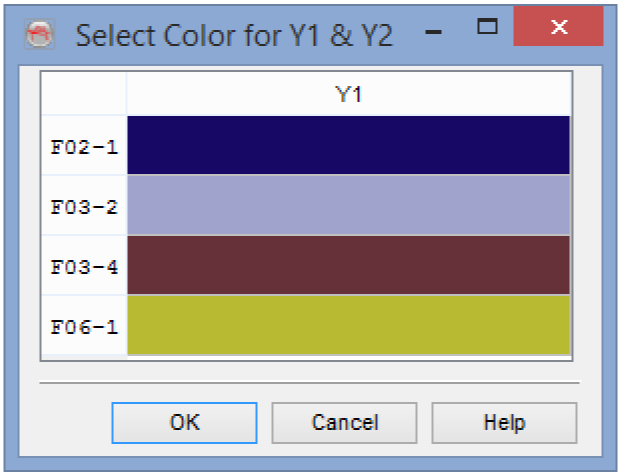
Overlay from a Third Attribute

Scattered points can be coded with respect to the amplitudes of an attribute using that option. The popup window requires the selection of that third attribute, and colorbar specifications (type and amplitude range)

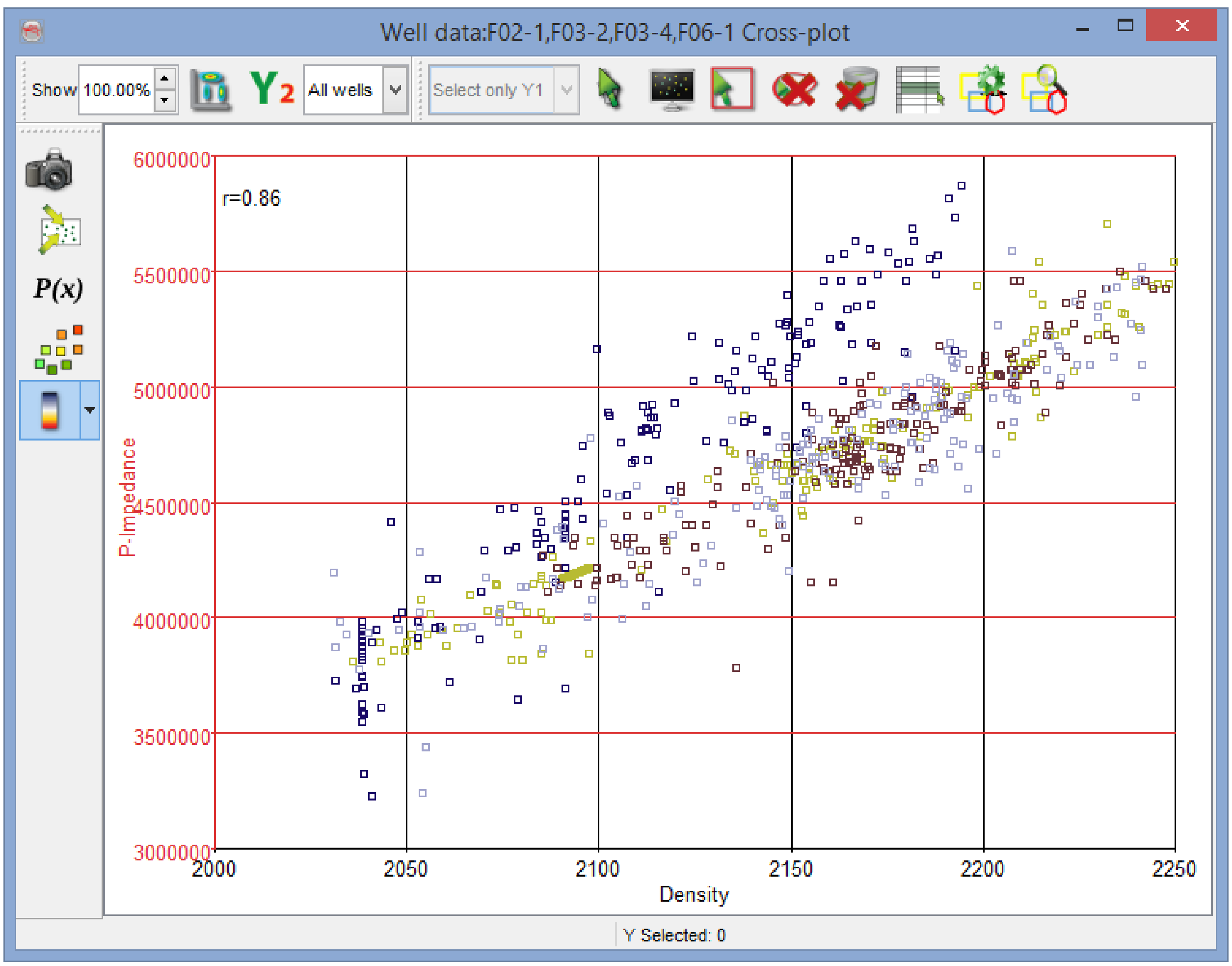


Well-based Color Coding

The scattered points in the cross-plot window can be coded with respect to the wells along which the data points were originally extracted. The following utility window can be used to control the colour associated to each well:

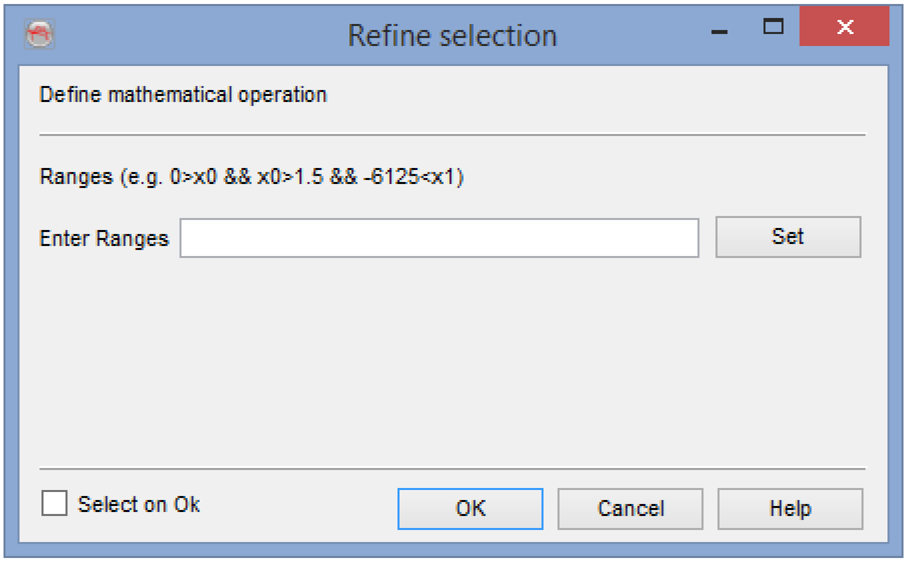


The configuration above gives the following result:

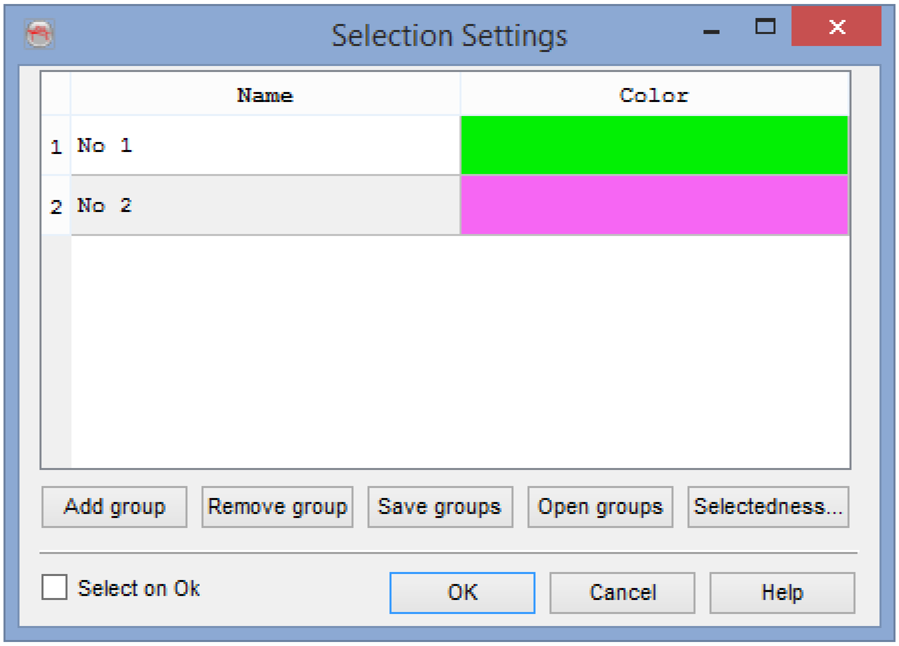


Selection Settings

The selections made interactively in the cross-plot window can be further refined and managed in this window.

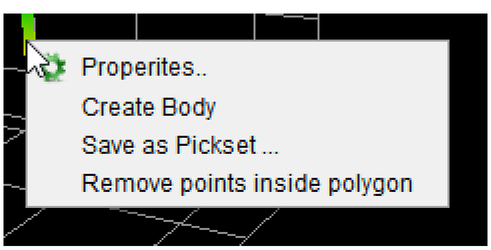


The 'Refine' option ()utilizes mathematical logic to restrict the selection according to the range set in the Refine Selection window (a pop-up window invoked when this button is pressed). For instance, a user may want to remove a data (x0) within a range of 3-4 from a cross-plot in which 1-2 values are overlapping. To do that, press this button to launch Refine Selection window. In the *Enter Ranges* field, an equation can be set i.e. X0 > 3 and X0 < 4. Where X0 is desired data in a cross-plot. After this a section can be made within a cross-plot to remove values within the polygon according to the equation.



The *'Manage Selection'* option () can be used to do multiple selections by adding new groups (see below). It is launched using this manage selection button. The multiple group selection allows you to select different clusters/trends on a crossplot in the form of groups. Second and subsequent selections are made by clicking on a group name and holding the CTRL key down from a keyboard prior. Then the corresponding polygon (with a given colour) is drawn over the crossplot display area. It is a very useful tool for reservoir prediction and characterization.

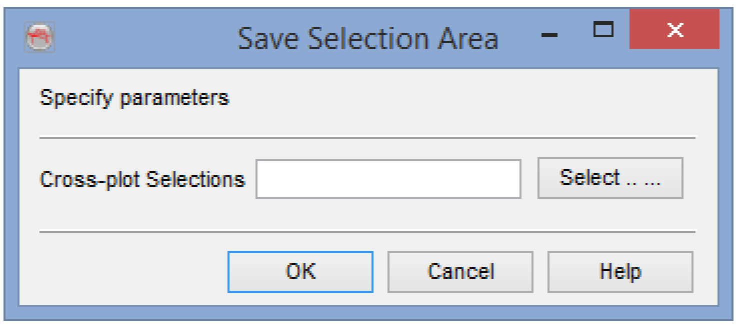
The selected scattered points can then be displayed in the active scene by clicking on OK. This allows an interactive display of the cross-plot in a scene. The displayed points (i.e. picks) can be saved in the OpendTect survey either as a Pickset or as a Body. Right-click on any point in the display, it will launch a pop-up menu (see below).



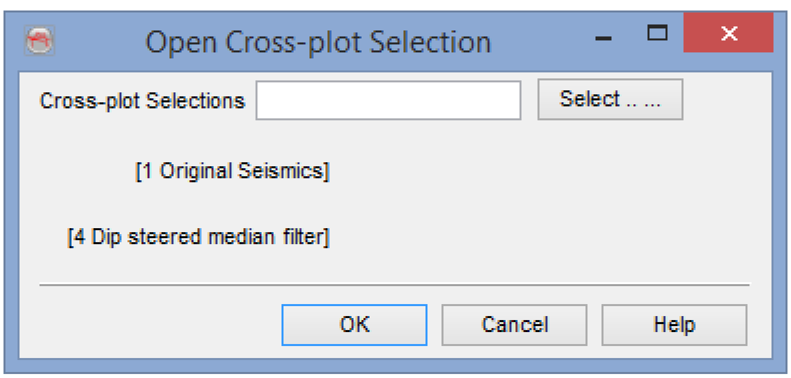
- **Properties:** It can be used to edit the size of the points.
- **Create Body:** Create a Body from the selected picks.
- **Save as Picksets:** To store the selected points as a pickset.
- **Remove selected points:** Removes the selected data.

You can do multiple selections by adding new groups (see below). It is launched using this manage selection button. The multiple group selection allows you to select different clusters/trends on a crossplot in the form of groups. Second and subsequent selections are made by clicking on a group name and holding the CTRL key down from a keyboard prior. Then the corresponding polygon (with a given colour) is drawn over the crossplot display area. It is a very useful tool for reservoir prediction and characterization.

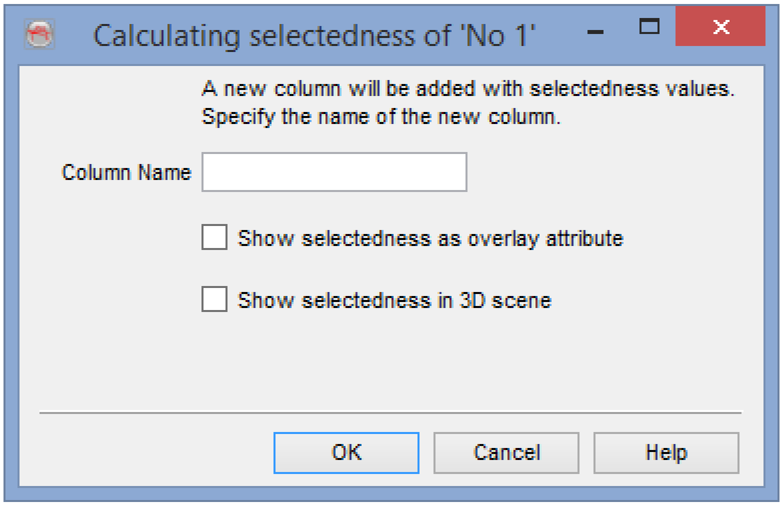
In the column name, a new name should be given that would be added in the crossplot data table. Show selectedness as an overlay, if checked, would display the colour coded selection ranging (between 0-1) as an overlay in the crossplot area. The colours represent the chosen colortable. Show selectedness in 3D scene would display the points within the selectedness range in the 3D scene.



The user can manage the groups (or selections) by saving them (Save groups)

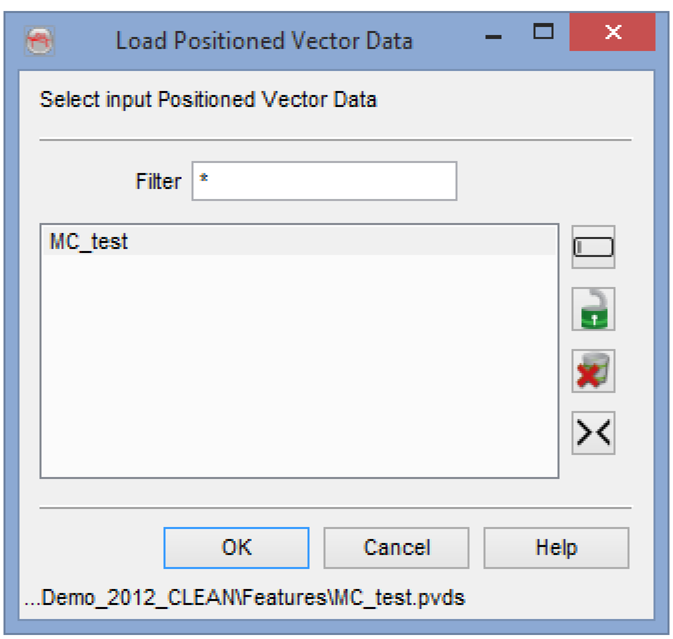


Furthermore, the user can also manage the groups (or selections) by opening (Open groups) the saved groups.



The Selectedness is a special data selection. It is a measure of how likely a point is to be selected in a particular selection. If a point is present in and around the center of a particular selection that has higher selectedness values. Whereas those belonging in the border regions are less likely to belong to that selection and thus it will have lower selectedness values. It is a measure of which points are better representative of a particular selection. The value of Selectedness ranges from 0-1. The points outside the selection has undefined values. It is added as a separate attribute in the table and can be seen in the form of an overlay attribute. To mark selectedness, one group is needed to be selected by using the selection mode.

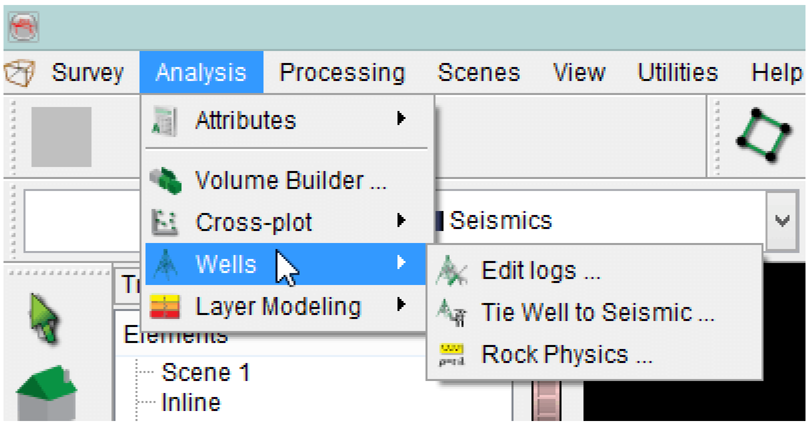
Open Crossplot



This option allows you to open previously saved or imported crossplot data and will directly open the data in the crossplot table.

Wells

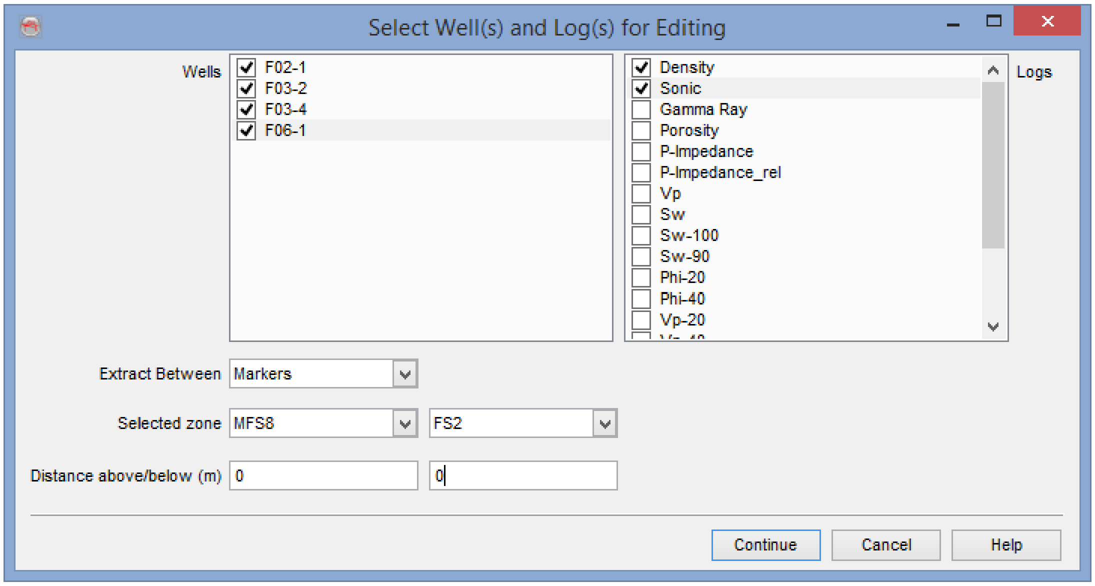
The *Wells* element in the Analysis menu gives you access to three features: *Edit logs...*, *Tie Well to Seismic...* and *Rock Physics...*



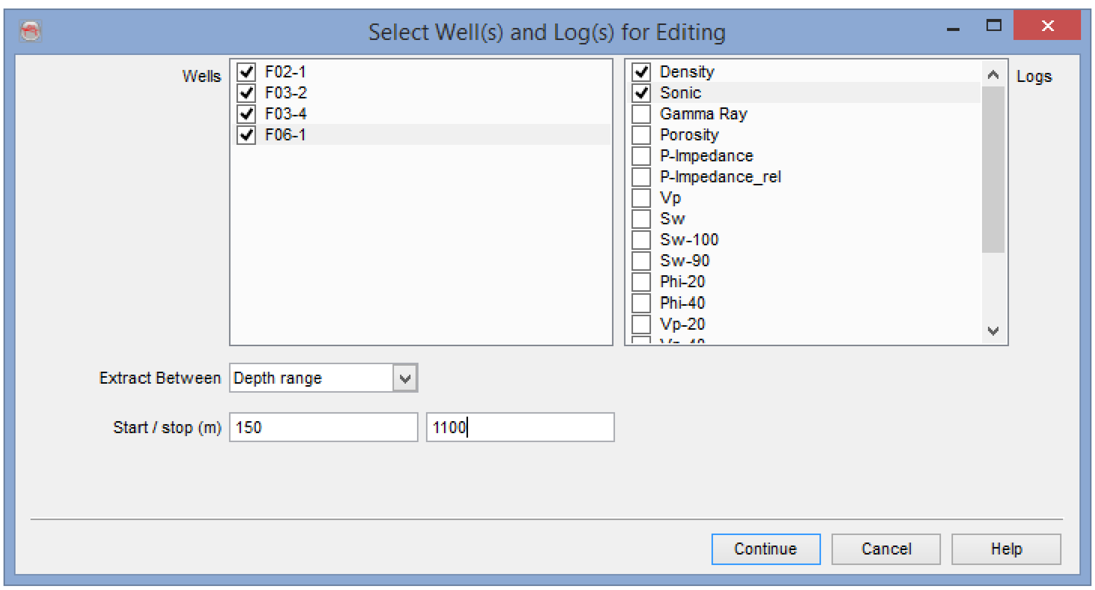
Well Log Tools

The well log tools can be used to remove spikes, smooth and clip the logs. it can also accessed by using the  icon in the *Manage Wells* window. Multiple wells can be selected at once along-with the various logs. The logs can be extracted between:

> **Markers:** The bounding markers have to be specified. If necessary, the distance above/below these bounding markers (from where the actual extraction of the logs is going to start/stop) can also be specified:

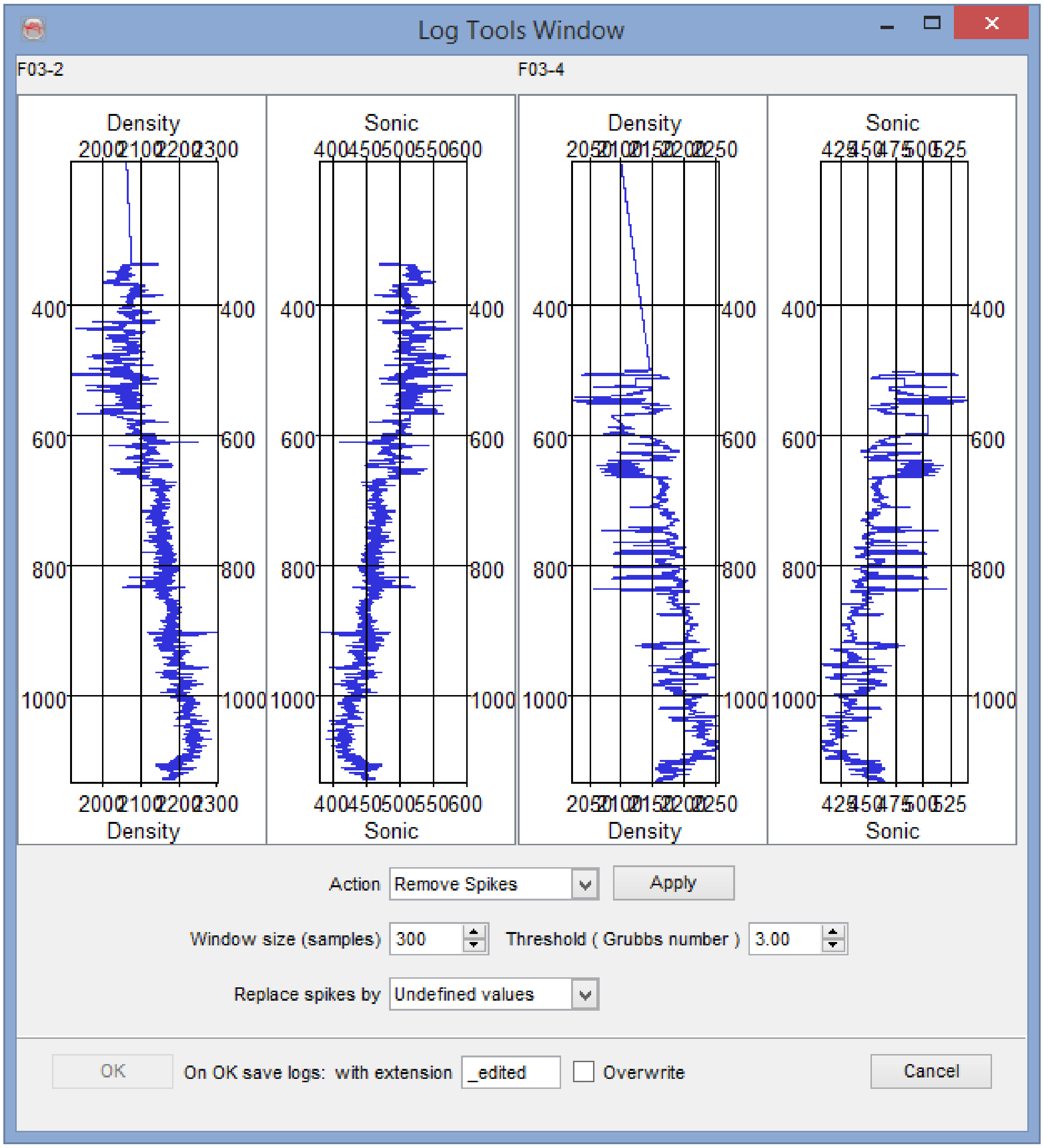


Depth range: The logs can be directly extracted between a particular depth range:



Time range: The logs can be directly extracted between a time window. The extraction may be done in time domain by toggling on 'Extract in time'.

On pressing *Go* the extracted logs are displayed and 'smoothing', 'clipping' and 'spike removal' can be performed on these well logs:



Smooothing: A window size (samples) should be defined in which the smoothing of the well log data will be performed.

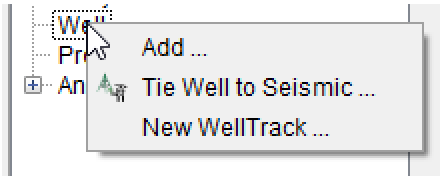
Clip: Percentage *Clip rate* has to be defined.

Remove spikes: De-spiking of the logs can be done by specifying a window size (samples) and the *Threshold* for the Grubbs algorithm. Further, the removed spike values can be replaced by 'Undefined values', 'Interpolated values' or can be manually specified.

Finally, the edited logs can be saved with an extension or can be overwritten.

Tie Well to Seismic

Launch the Well-Seismic Tie window from the main menu or, optionally, the Well-Seismic Tie window can be launched from the tree.



Well to the seismic tie is a major task for interpretation projects. It is used to correlate the well information (logs) to the 3D seismic volume or 2D seismic lines. This enables the comparison (crossplots, ...) of well-based and the 2D/3D seismic data.

Well-to-Seismic tie workflow:

1. Data preparation:

- Import the seismic volume or 2D line.
- Extract a wavelet.
- Import the wells: Each well requires a track, checkshot or time-depth curve and sonic log.
- Import the additional data: 3D/2D Horizons, well markers, additional time-depth curve if a checkshot was loaded.
- Edit the log database: Fill the missing sonic parts, create a density log from the sonic (constant value or Gardner's equation) if not available.

2. Synthetic-to-seismic tie

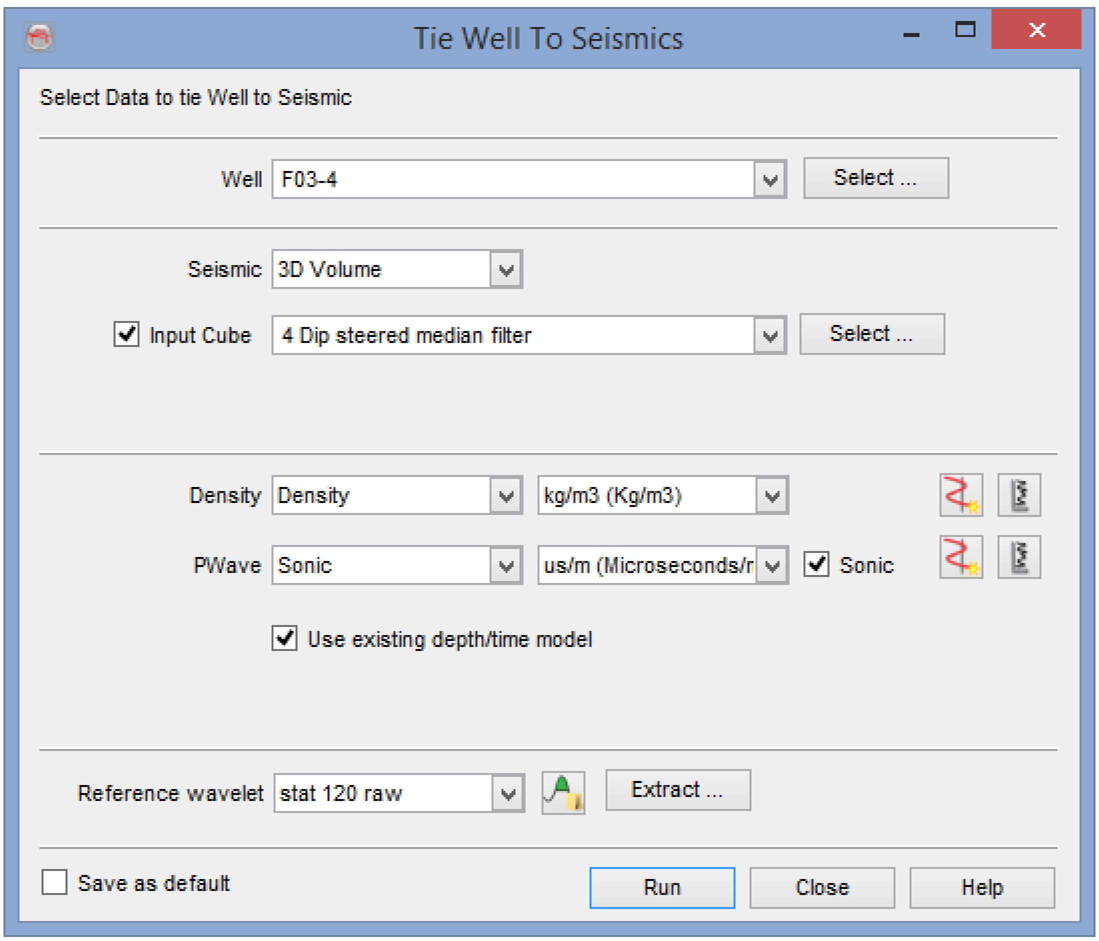
- The module is launched from the Analysis menu or via the right-click menu of each well.
- The input fields must be selected.
- Based on the available data the density and sonic logs will be combined into impedance and reflectivity, depth-time converted (includes an upscaling) and convolved with the wavelet. The result is a synthetic seismic traces for the well. This trace will be compared with a composite seismic trace extracted in the volume along the (deviated) well path, on the nearest trace. Both synthetic and composite seismic traces are cross-correlated, and the output value is an indication of the alignment and matching quality.
- The alignment will be carried out either by shifting the synthetic trace up or down, or by selecting several locations in both seismic traces, specifying and applying a shift function that varies with the travel time.The applied changes must be

validated, before being converted into a new time-depth function that replaces the previous one. No changes are being applied to the well logs.

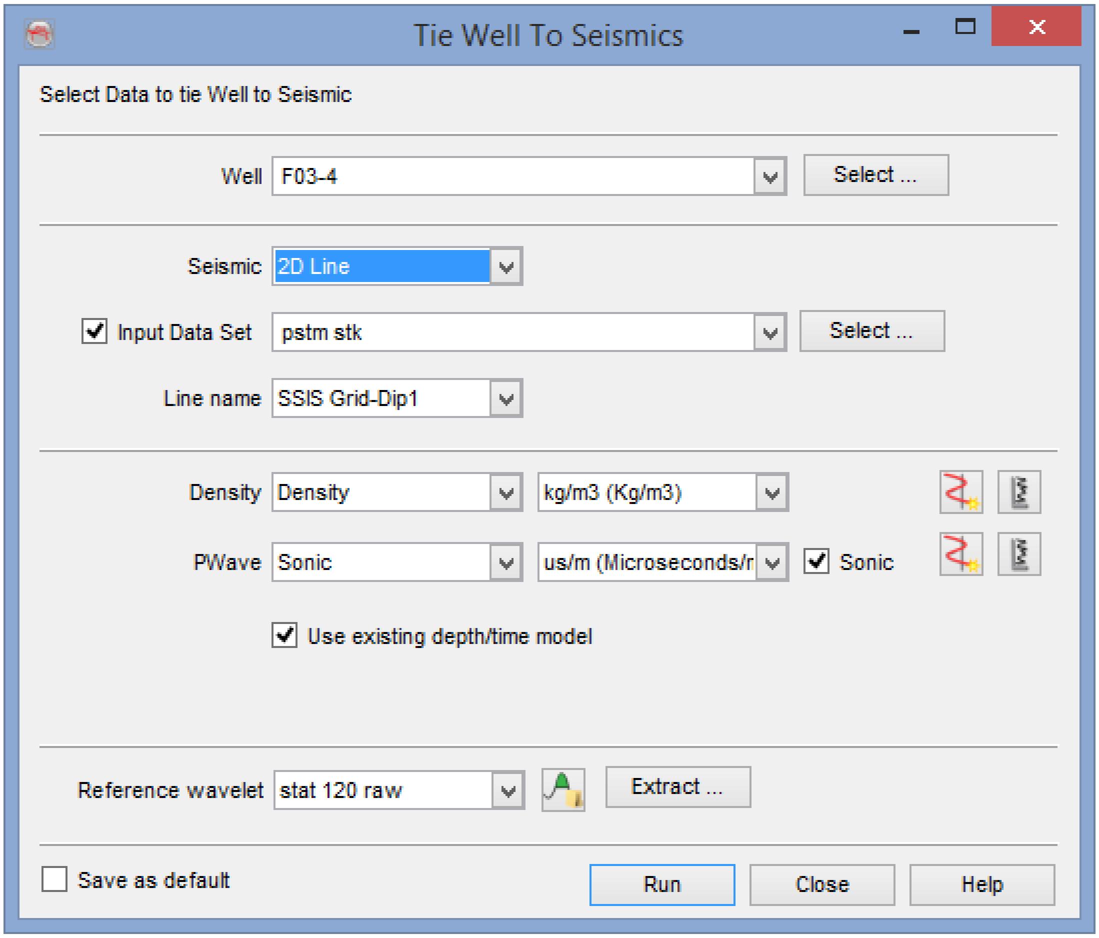
- At each step of the tie a deterministic wavelet can be estimated using the time-converted reflectivity log and the composite seismic trace. This deterministic wavelet should vary per well, and is known to link the well data to the seismic data more reliably.

Well-Tie Selection Window

The tie well to seismic window is used to select the necessary data for the Well-Seismic tie workflow. Please have a look at the introduction to see how to prepare the necessary data.
Well tie can be used to tie the well with the 3D seismic volume or 2D seismic line.



Well to 3D seismic tie window



Well to 2D seismic tie window

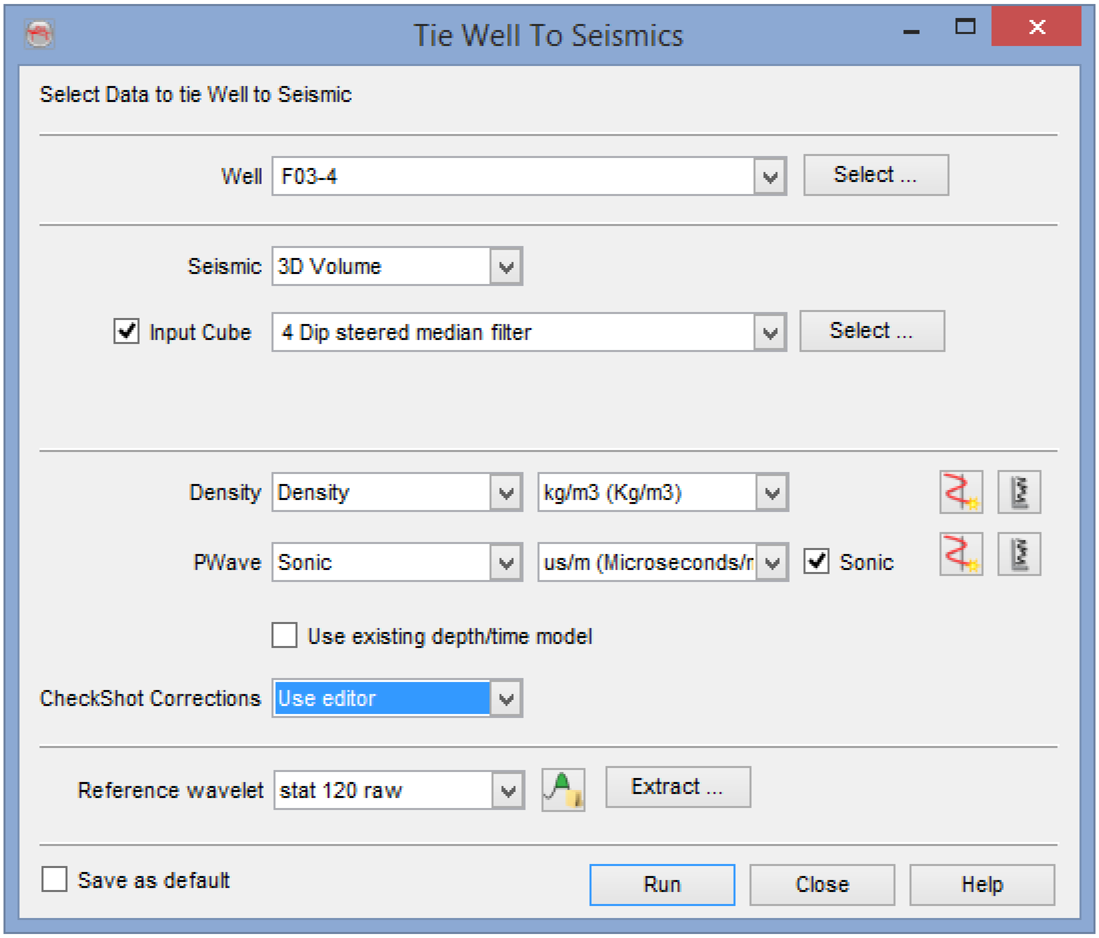
In both the 2D and 3D windows, there are additional features accessed via the following icons:

-  Opens the *Calculate New Log* window
-  Displays the particular log in the log viewer
-  Opens the *Wavelet Manager* window

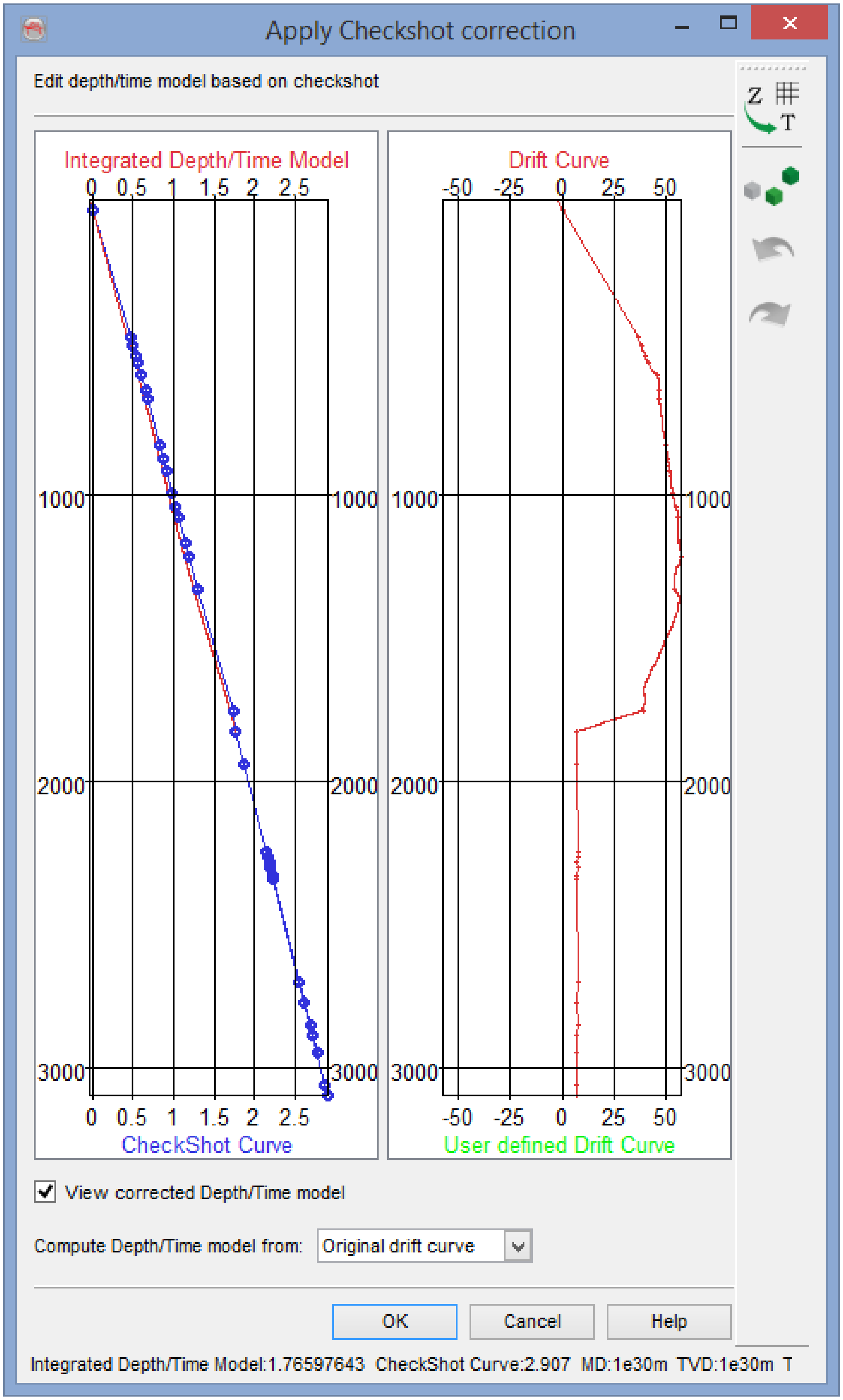
Well-Tie Checkshot Editor

In OpendTect, CheckShot corrections are applied before launching the Well to Seismic Tie window. If you have no depth/time model or have not selected any existing one, you will be proposed to correct the sonic integrated depth/time model, provided you imported a CheckShot model for your well.

The choices given are "None", "Automatic" or "Use editor". In the first case, the time depth curve will be computed directly from the sonic log without any correction (note, this is also the default mode if you do not have any CheckShot). In the automatic mode, the time/depth curve will be calibrated to the CheckShot without any user interaction. In the last case, you will be allowed to edit the calibration yourself using an editor window.



When clicking on 'Run', the '*Apply Checkshots correction*' window pops up.


Apply Checkshot correction
Edit depth/time model based on checkshot
Integrated Depth/Time Model
0 0.5 1 1.5 2 2.5
1000
2000
3000
CheckShot Curve
Drift Curve
-50 -25 0 25 50
User defined Drift Curve
View corrected Depth/Time model
Compute Depth/Time model from:
Original drift curve
OK
Cancel
Help
Integrated Depth/Time Model:1.76597643 CheckShot Curve:2.907 MD:1e30m TVD:1e30m T

The above window is divided in two panels. On the left one, the sonic 'Integrated Depth/Time Model' (red) and the 'Checkshot Curve' (blue) are plotted. The right panel displays the drift curves. The original 'Drift Curve' (red) shows the variations between the CheckShot and the sonic integrated model.

By adding points to the right display you can additionally generate a new 'User defined Drift Curve' (green). This is done by clicking the icon. Once this is done, select the correction to apply, either from the Original or from the User Drift Curve and push the Apply button. The newly computed depth/time model will appear in green on the left panel. You can modify the drift curve and re-apply the corrections until you are satisfied with the depth/time model. Push the OK button and the main well tie window will appear using the new calibrated depth/time model.

Well-Tie Display Panel

The display panel is the main window where the wells are tied to the seismic data. This module is primarly used to update the current (loaded) time-depth curve. Previous, intermediate and final TD curves can be exported to ascii at any moment using the following icon .

A secondary output of the synthetic-to-seismic tie is the estimated wavelet, that is estimated at any time and must be saved explicitly.

The view is separated in three frames, from left to right:

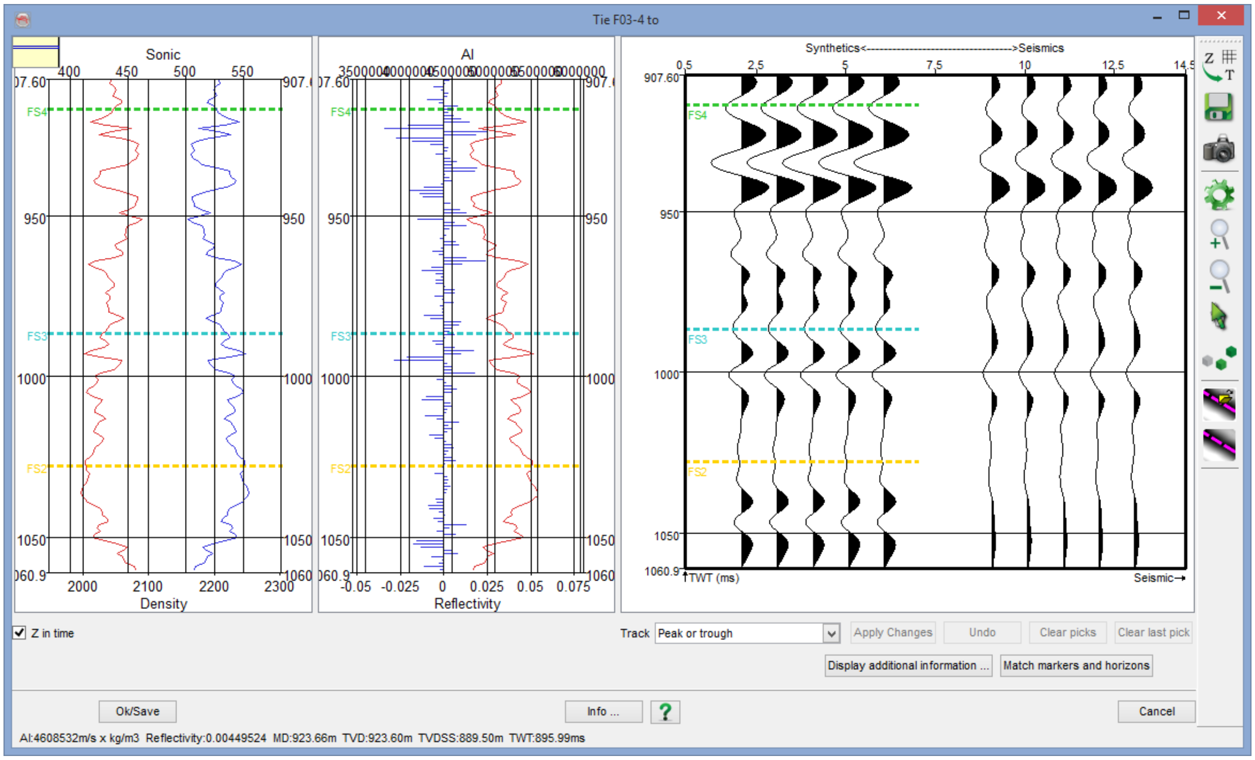
- The sonic (red) and density (blue) logs.
- The computed impedance (red) and reflectivity (blue) logs.
- The computed/extracted synthetic and composite seismic traces.

  Note: The raw logs are shown before upscaling. The vertical axis of all 3 frames is in travel time.

Key points:

- The time-depth conversion and synthetic seismic traces computation is done using the current time-depth curve and checkshot (if available).
- The checkshot data acts as a strong constraint, i.e. any input and output time-depth curve will be forced to honour the checkshot.
- The time of the depth TVDSD=0 will always remain at TWT=0 even when applying a bulk shift: The difference is absorbed between the point TVDSS=0 and the first

sonic log sample. The reference datum elevations definitions are summarized in the well track import chapter.



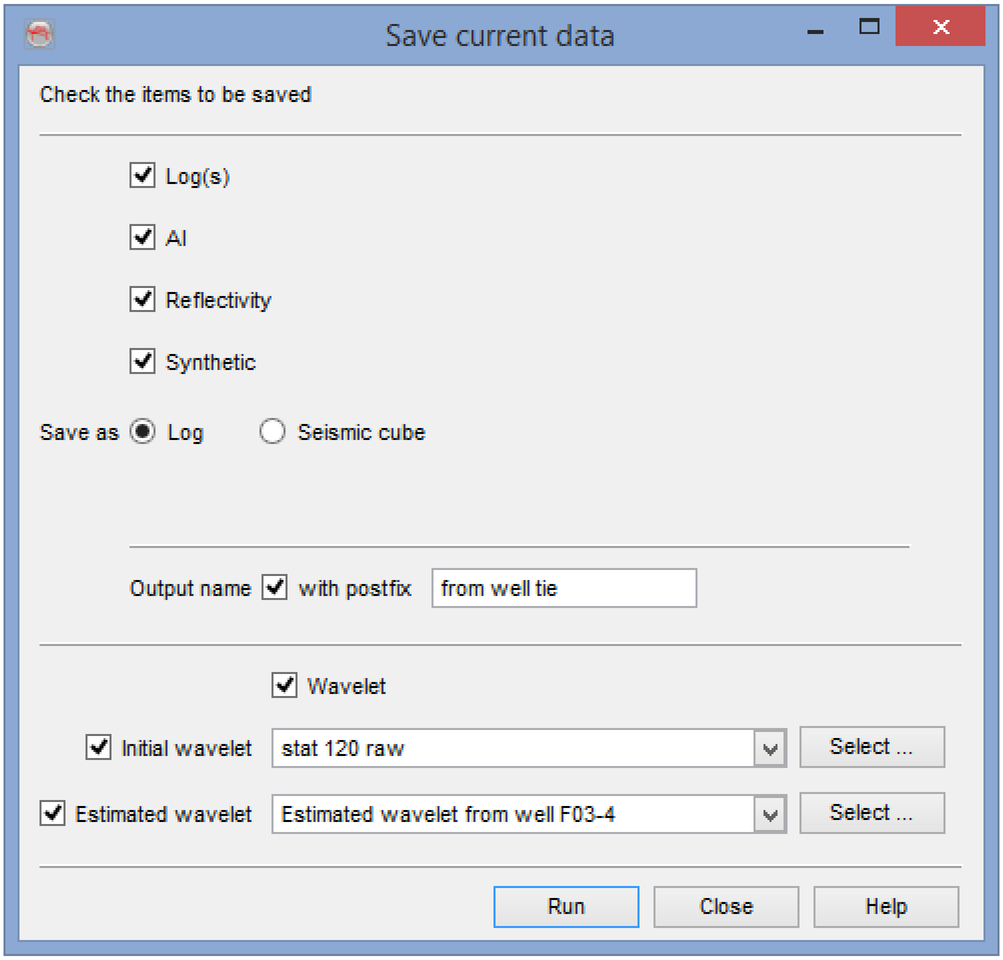
Display panel for the well-seismic tie.

At the bottom right corner of this display panel, there are several tracking controls. The options are used to pick an event to match the seismic and synthetic traces. After picking the event, press *Apply Changes* to reflect the changes and update the time to depth model.

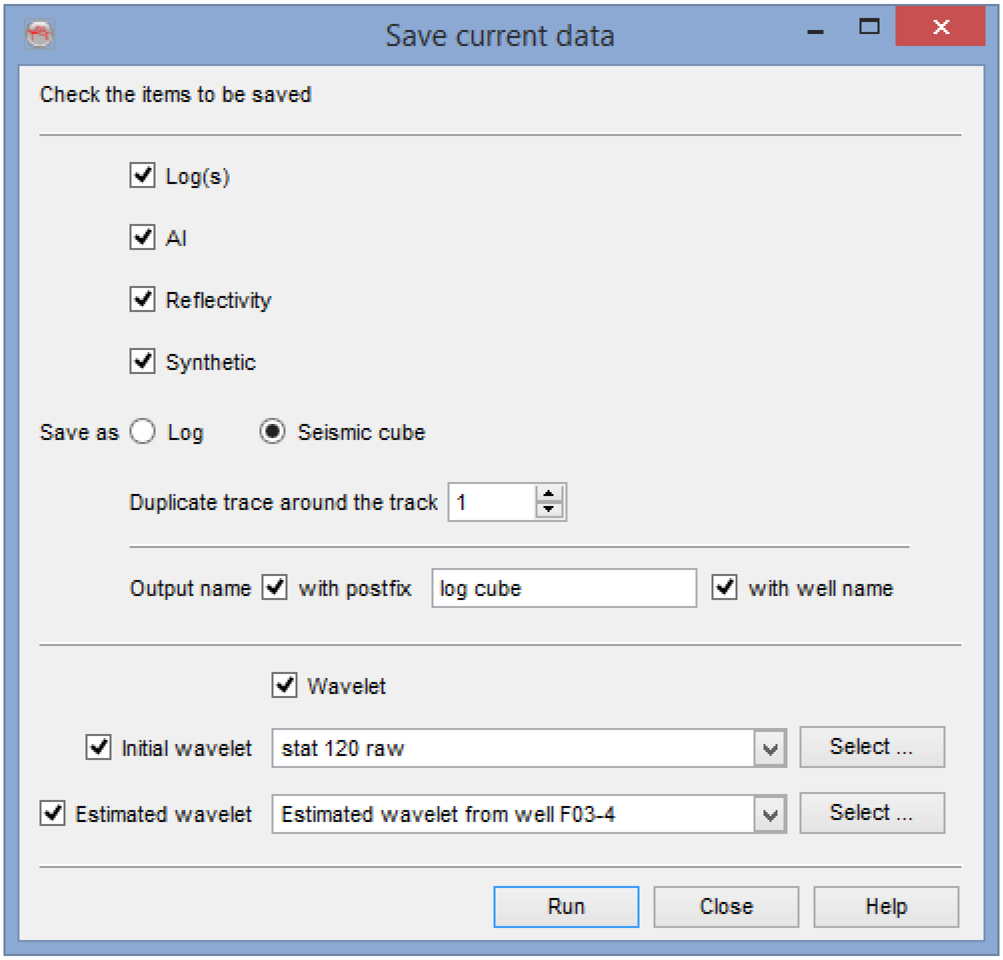
 Launches the Edit Time/Depth model window. In the pop-up window, press Export button to export the time-depth model as an ASCII file.

 Launches the save option.

You can either save the created logs:



Or save the synthetic trace as a seismic volume.



Takes the snapshot of the display panel

Display settings/properties for the panel. The settings are similar to that of the normal 2D-viewer model

Zoom-in

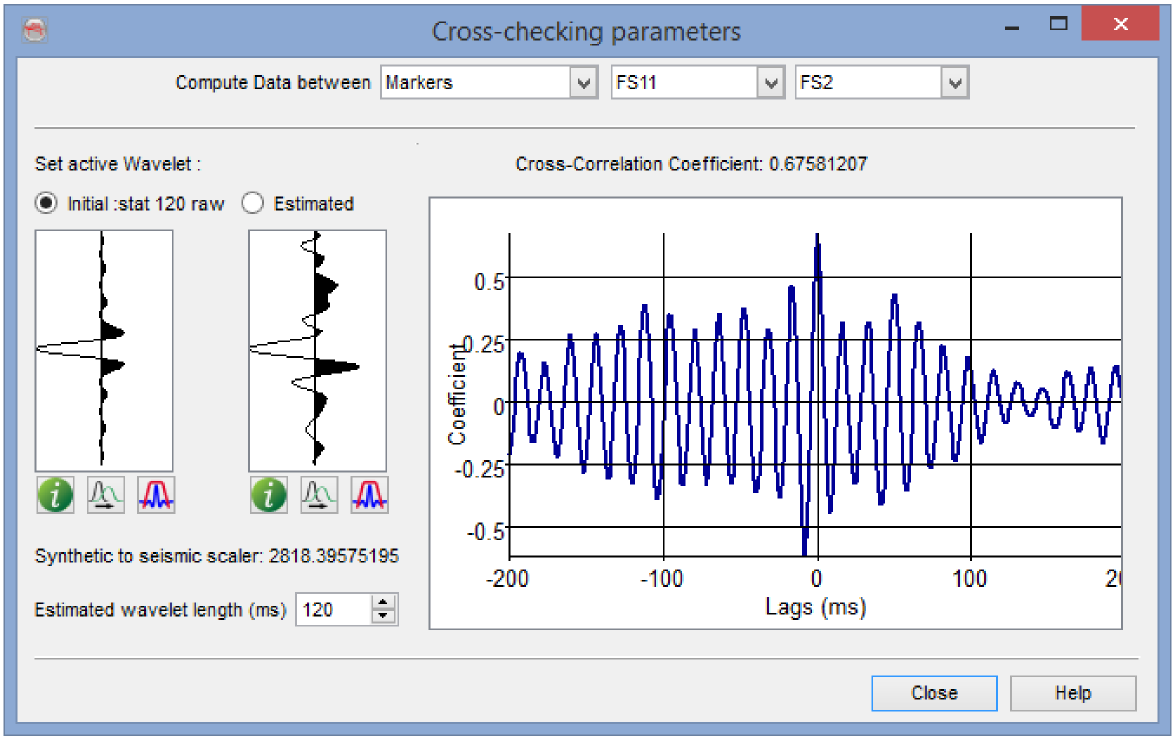
Zoom-out

Toggles interact mode ON/OFF

Pick seeds on the seismic or synthetic to update the time to depth model.

Change in depth/time model

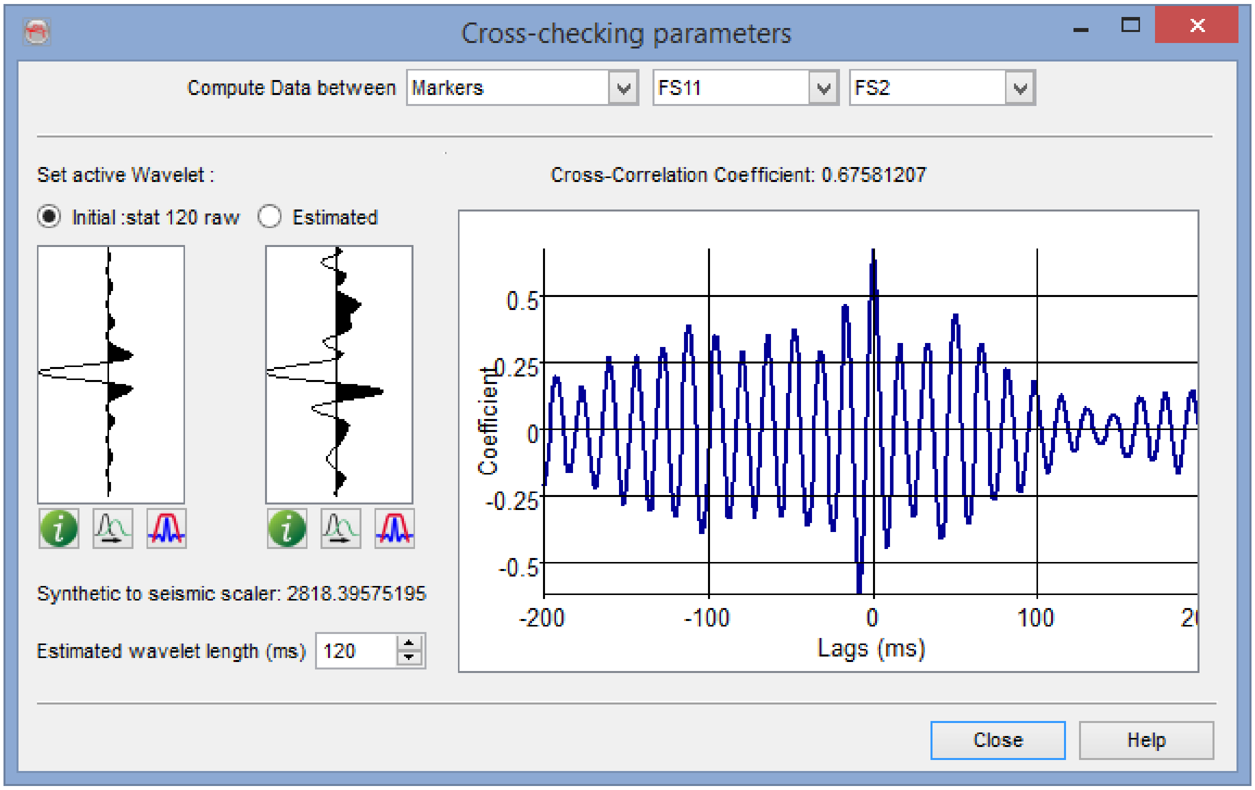
- Choose a *tracking mode* (e.g. maxima, minima, zero-crossings etc.) and select events in the synthetics/seismic displays by first selecting the *Pick mode* button. Events can not be picked separately. Each event in the synthetics must be linked with an other event in the seismic.

- Once all the events are selected on both synthetics/seismic displays, press *Apply changes* button. The depth/time model and the whole computational workflow will be recomputed. If needed, repeat the operation.
- The *Display additional information* button will open the *Cross-checking parameters* window, and provide useful cross-checking tools, such as correlation coefficient and estimated wavelet in a specific depth range. The estimated wavelet displayed here can also be saved:
- 

- The depth/time table can be saved between each state by pushing the *Save* button, in the toolbar to the right of the synthetics/seismic displays, and saving with an appropriate name. The *View/Edit Model* button allows the user to import a depth/-time table.
- Once a good correlation has been established, click on *Ok/Save* and save the depth/time model.

Well-Tie Crosscheck Parameters

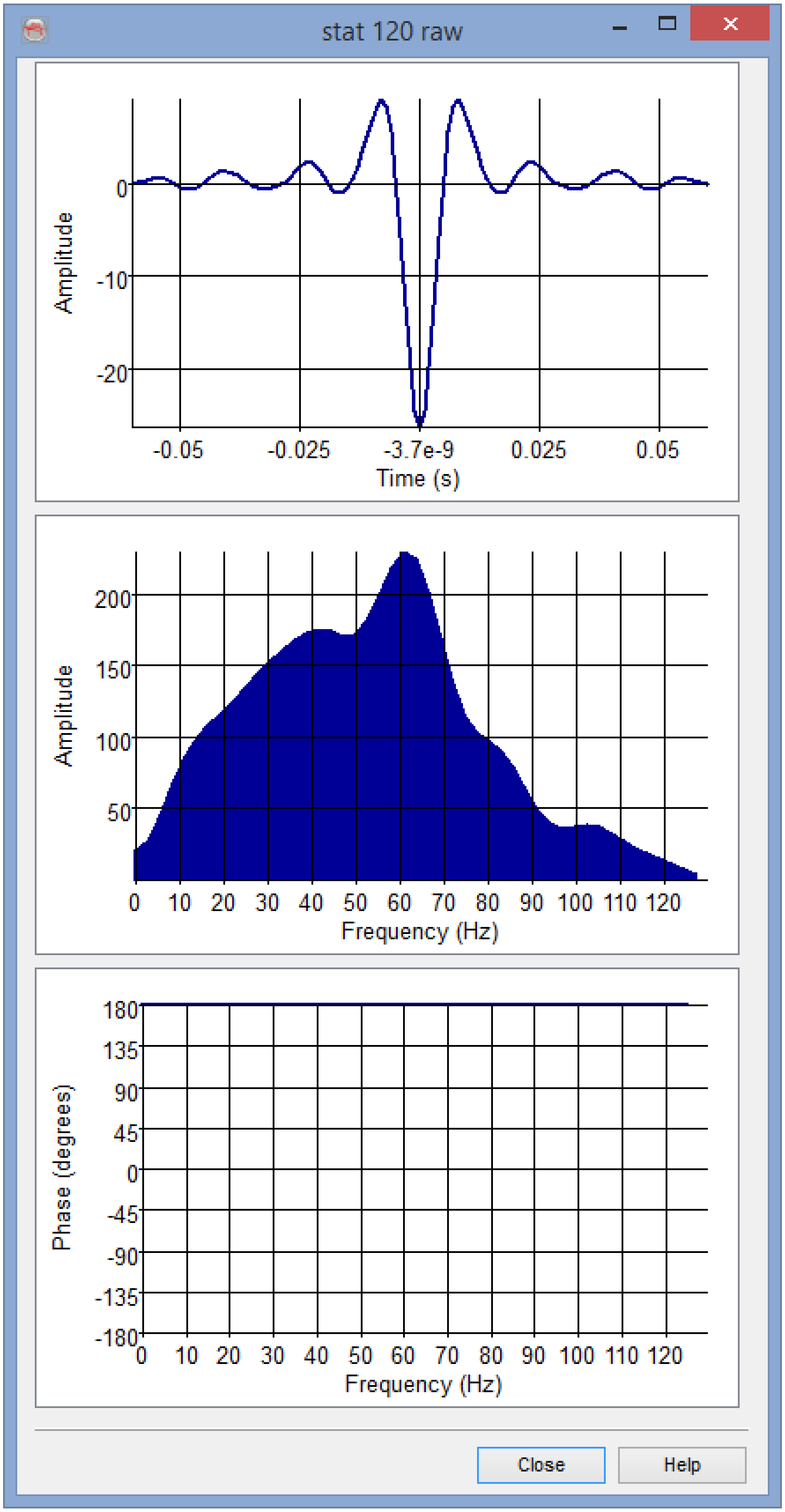
The cross-checking window is launched from the well ties display panel (Section- Well Tie: Display Panel) by clicking on the *Display additional information* button. The window contains the initial and estimated wavelet information. The wavelet can be computed between the two levels (start-end of data in time/depth or markers) that are provided at the top of this window. The window contains further key information: wavelets plot and the correlation coefficient. By changing the *compute data between* option, the correlation coefficient is auto-updated, this is done by using either Markers (Default) or Time/Depth. After achieving a high and positive correlation coefficient, the estimated wavelet can be saved. Importantly, the negative correlation coefficient shows that the polarity of the estimated wavelet is reversed. To avoid that the reference/initial wavelet's polarity has to be reversed.



Check-crossings window and correlation panel

To save the estimated wavelet, press the icon in the well tie main window .

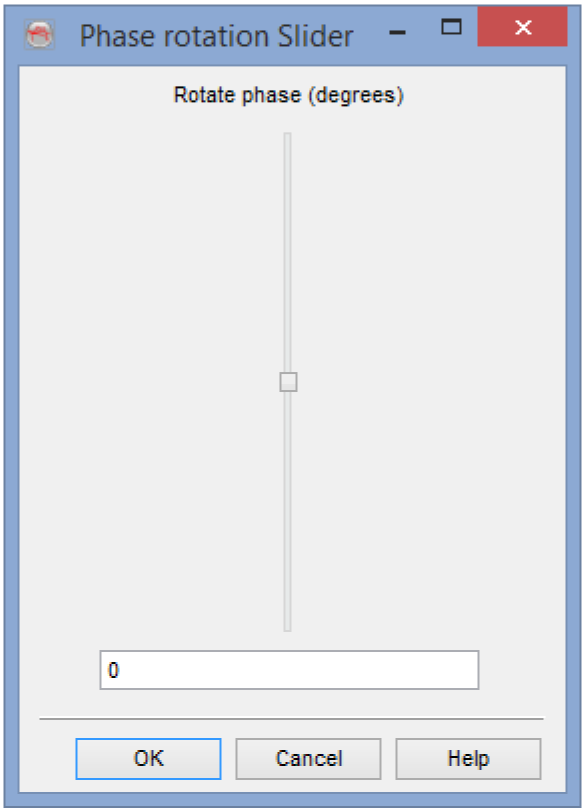
The wavelet *properties* ( ) can beshown as a graphical display of the wavelet, its amplitude spectrum and phase spectrum.


stat 120 raw
Amplitude
0
-10
-20
-0.05
-0.025
-3.7e-9
0.025
0.05
Time (s)
Amplitude
200
150
100
50
0
10
20
30
40
50
60
70
80
90
100
110
120
Frequency (Hz)
Phase (degrees)
180
135
90
45
0
-45
-90
-135
-180
0
10
20
30
40
50
60
70
80
90
100
110
120
Frequency (Hz)
Close
Help

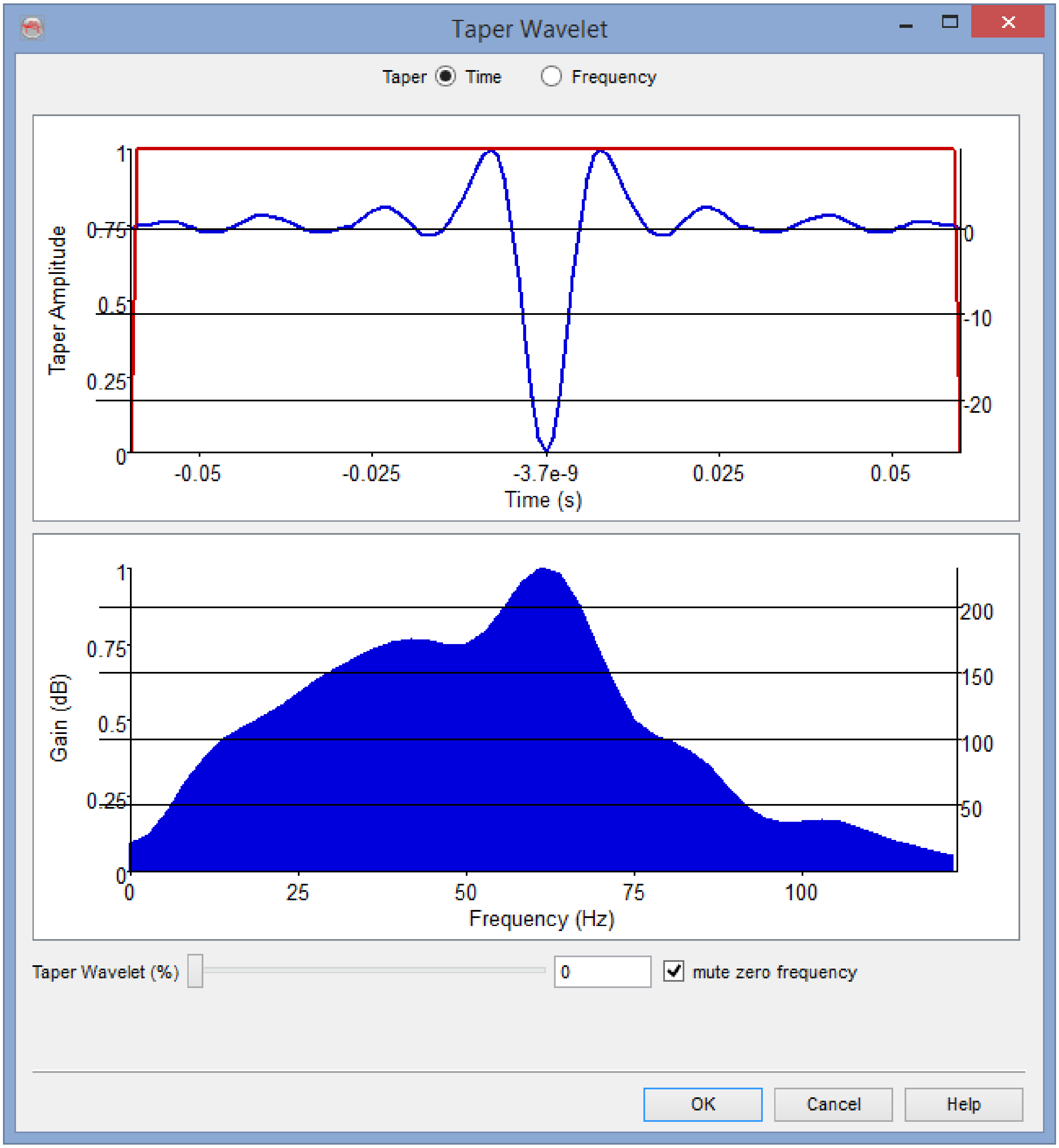
Save the estimated wavelet

The user can also click on to open a slider interface for shifting phase of the wavelet:

Rotate Wavelet: Using the slider, the user can edit the phase of a wavelet.



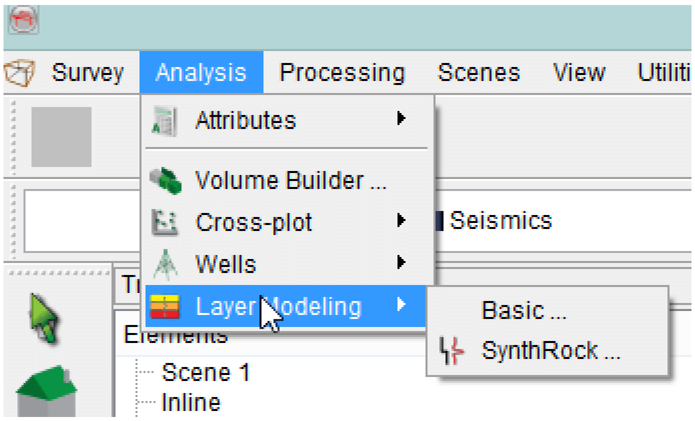
Taper Wavelet: This option enables the user to taper a wavelet by clicking on . The User will see the real-time changes in the amplitude spectrum.



Layer Modelling

Pseudo-wells are stratigraphic columns with attached well logs, but without geo-graphical location. Any pseudo-well can be seen as a possible realization of a newly drilled well in the area. The pseudo-wells generation is achieved following a model that has to be defined. To achieve *Layer Modeling*, preliminary an extended well data analysis has to be carried out. The stratigraphy must be defined and then the well logs behavior have to be known in order to be used in the modeling.During the modeling process, the stratigraphy description is fixed and cannot be edited.

The Layer Modeling is accessible from the *Analysis* menu but also from the Manage Stratigraphy window.



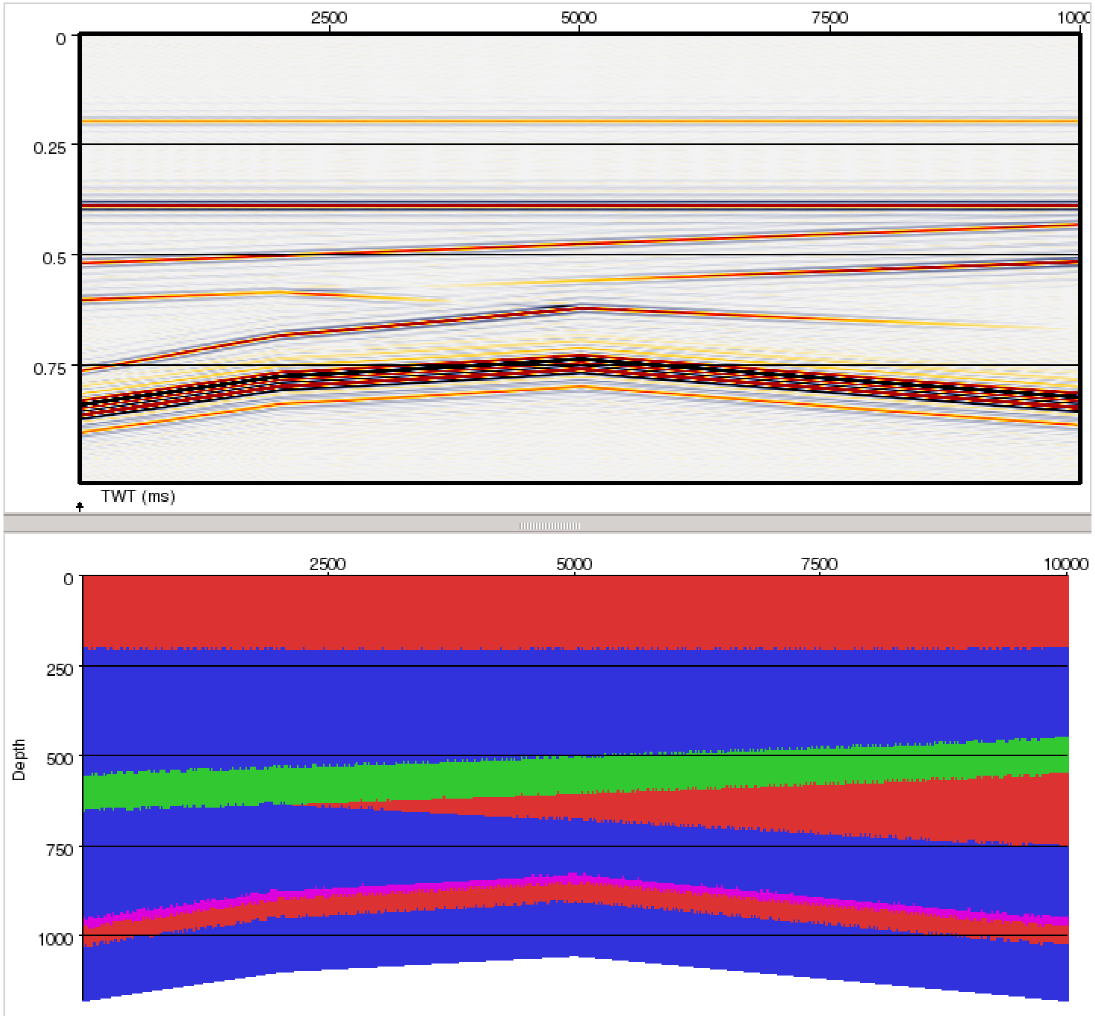
Basic modeling can be achieved in OpendTect. More advanced modeling are available in SynthRock plugin.

The Layer modeling workflow is divided into three main steps:

1. Model definition: using the stratigraphy description, properties are assigned to the different lithologies within each units. These properties are fixed or can vary. The model definition is used to generate the pseudo-wells.

2. Synthetic and Log generation: the pseudo-wells are generated and their associated properties can be displayed. With a wavelet extracted from the real seismic, zero-offset synthetics are generated. Using a ray tracer synthetics can be computed for different offsets and restricted angle stacks can be created. Thus their behavior with varying offset can be analyzed.

3. Pseudo-well data analysis: the properties from modeled logs and synthetic seismic can be compared and analyzed layer by layer, lithology by lithology.



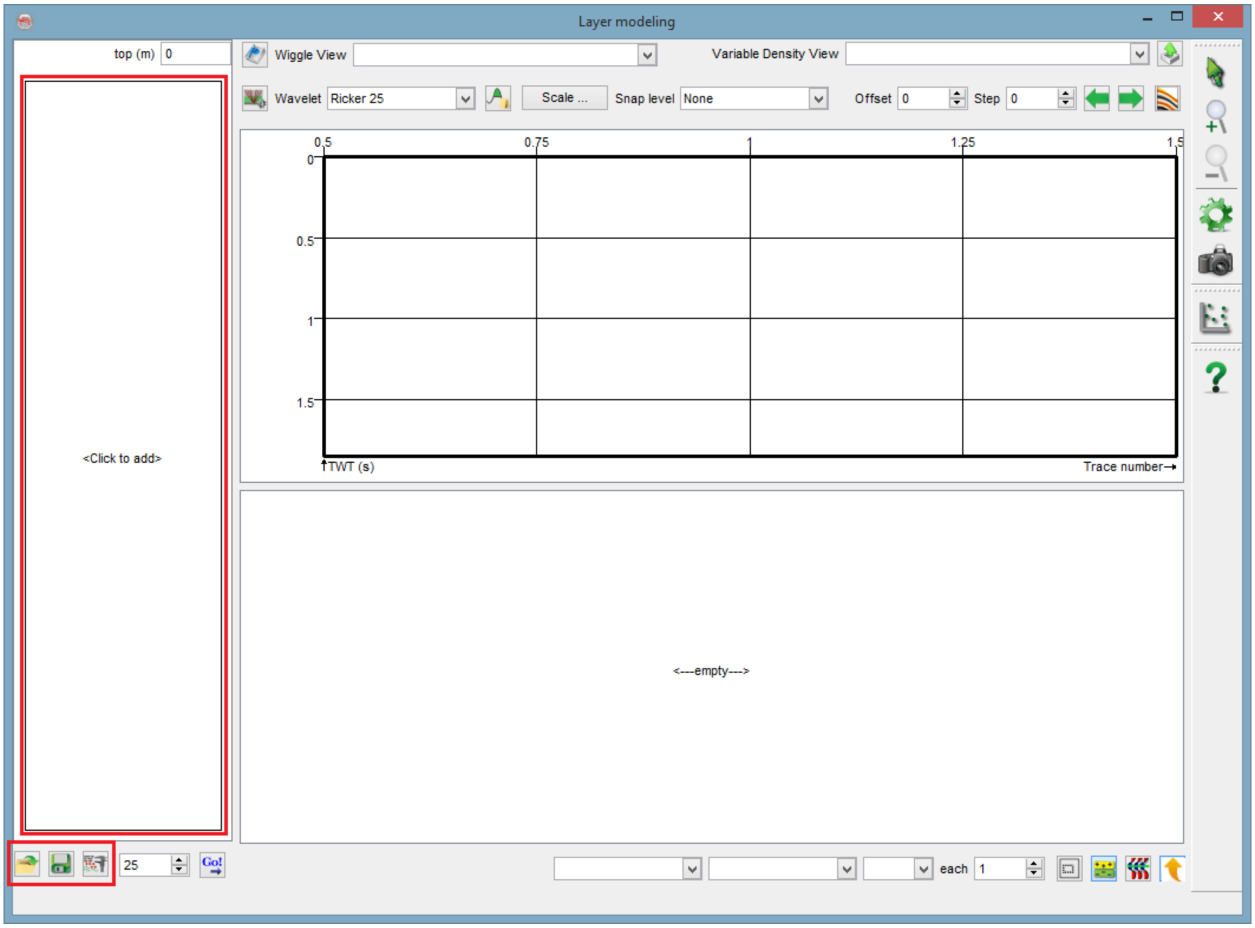
Example of a basic blocky model

Basic Modelling - Introduction

Each layer in the stratigraphy column is characterized by different rock properties. The model, based on the stratigraphy, is assigning properties to each lithology, layer by layer. The model is built using a blocky approach. The different properties are selected within a list. Their value can either be constant or vary within a given range.

Layer Description

First of all, the Layer Succession has to be defined and will be used to create the pseudo-wells.

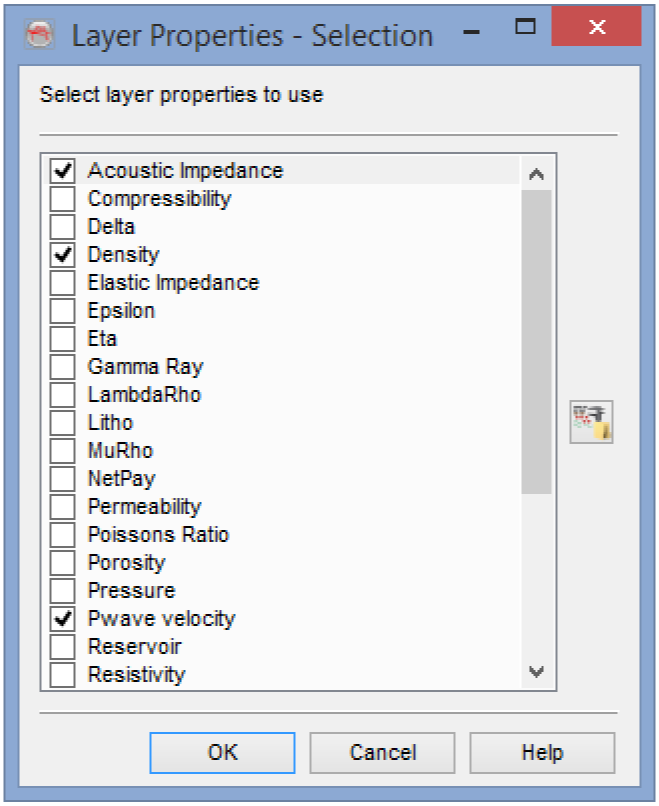


Layer Modeling Window (in red : Layer Description elements)

To start defining layers properties, the user has to click on "click to add" on the left rectangle. Once one layer has been defined, click on Ok and the layer appears in the left rectangle. To add a layer, click-right on the rectangle and select "Add above/below": you can then define a new layer. "Edit Layer" is also accessible from this right-click menu.

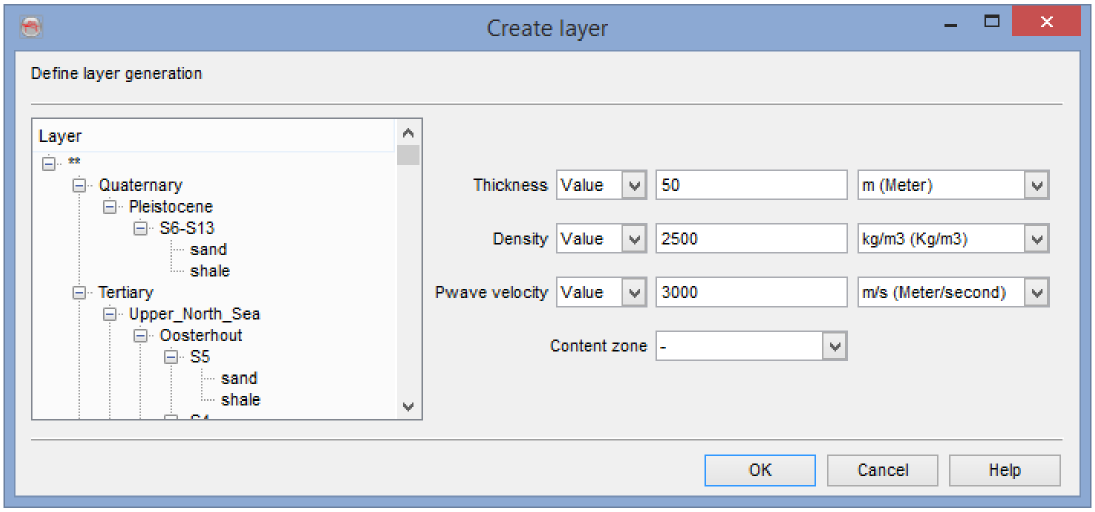
The description can be saved in clicking on the  icon and later be accessed in clicking on the  icon.

1. First of all the properties to be defined for the modeling have to be selected within a list. The properties in the list have been defined in the [Layer properties Manager] which is accessible from the  icon and can be edited.



The selection can always be edited in clicking on the  icon . To be able to generate synthetics, Density and Pwave velocity are selected by default. For the moment, it is not possible to combine properties together. So for example to get the Acoustic Impedance, you have to model the Acoustic Impedance log.

2. The Layers have been defined in the Stratigraphy Manager. To each lithology of each layer are assigned properties and if within the survey this property is expected to remain constant or to vary within a given range. The thickness of each layer can also stay constant or be varied. The variation is linear.



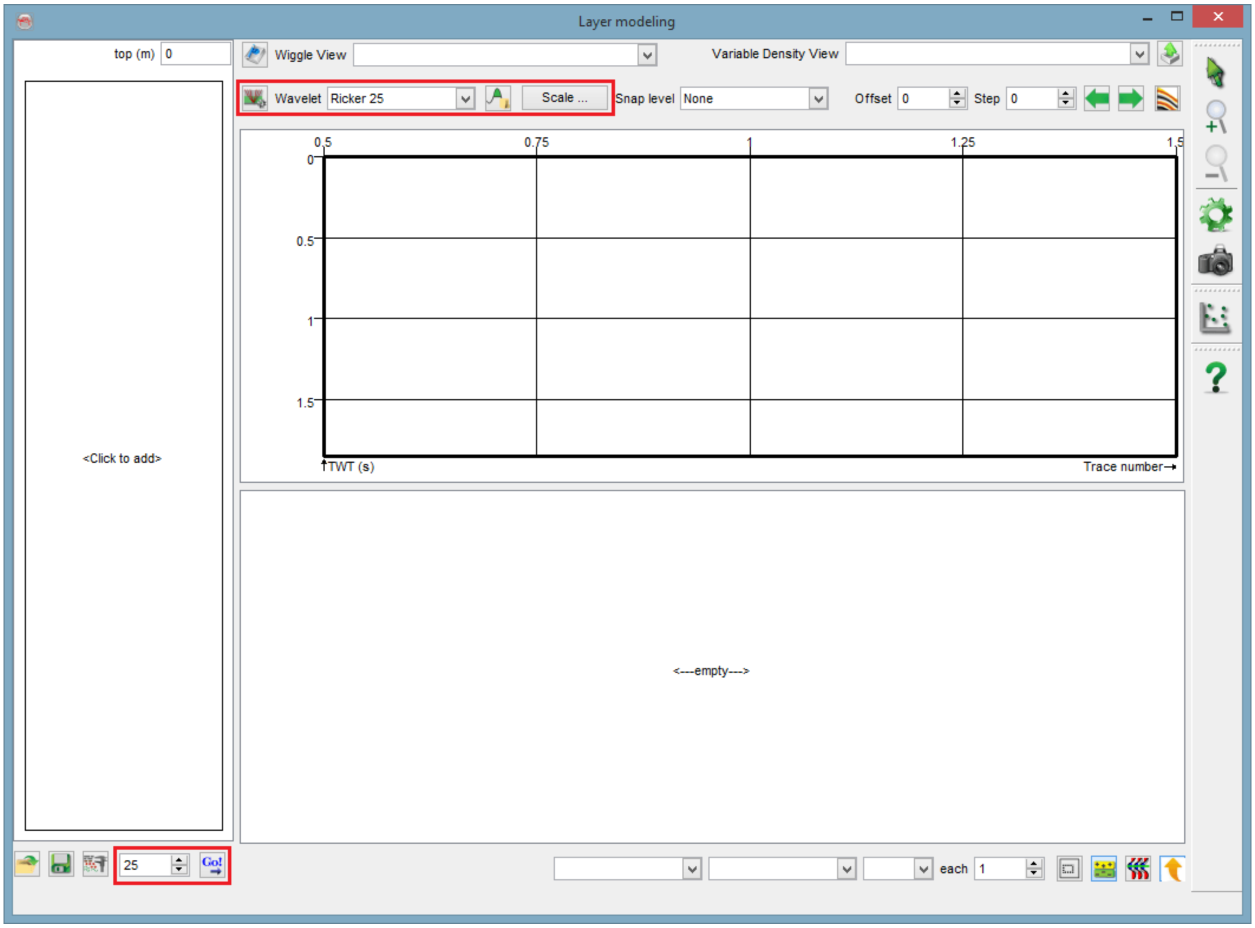
The thickness is a default property. When defining a thickness range, the starting thickness can be set to a negative value: it will appear as a truncation in the pseudo-wells.
The fluid content can also be specified.It had to be specified previously in the Content manager.

When clicking on OK, the Layer Description will appear on the left rectangle. To edit the properties of one or more layer, just click on it.

Synthetic- and Property-Log Generation

The property logs of the pseudo-wells only need the Layer Description to be created. To generate synthetic traces for each pseudo-well, three elements have to be provided:

1. To generate the synthetics: Wavelet
2. To generate synthetics for a range of offset :
   - Ray tracer parameters
   - Layer properties: The density, P-wave and S-wave have to be given, computed or derived from other quantities that have been modeled.

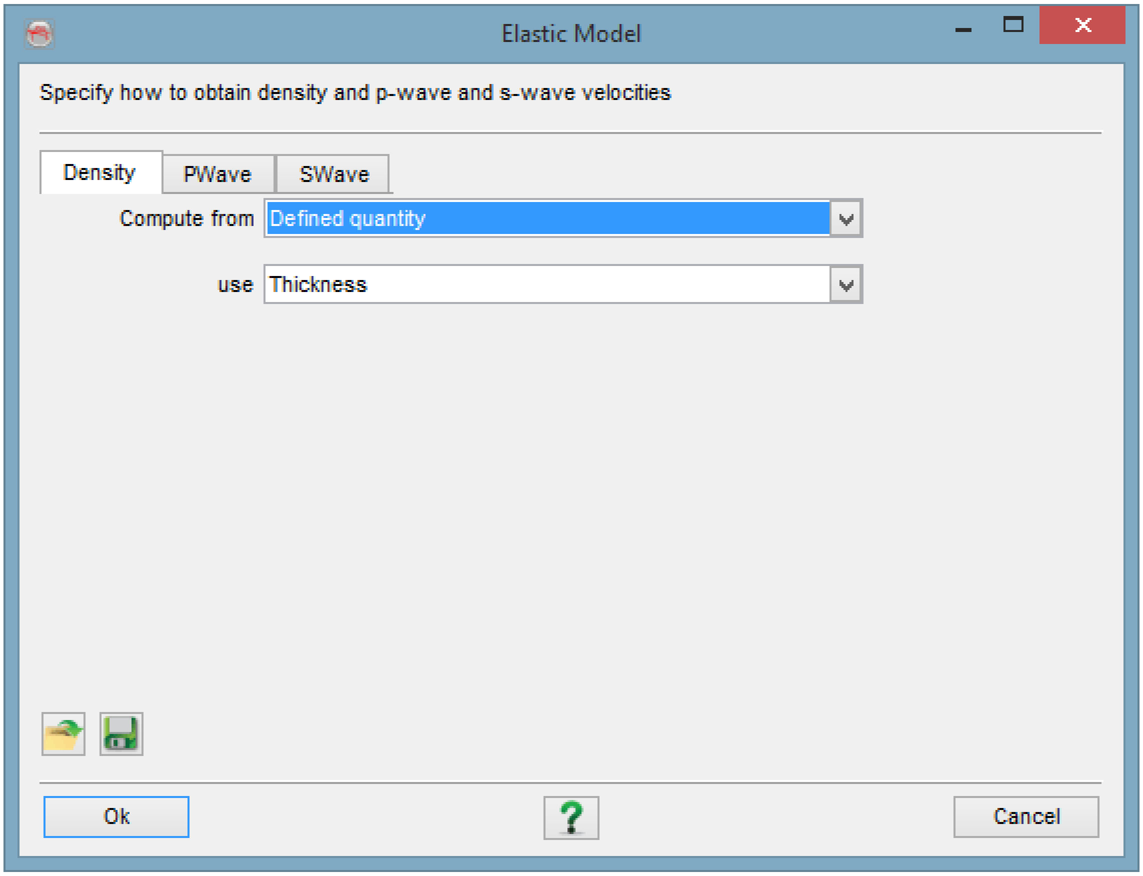


Once the Layers are defined, the parameters to generate the pseudo-wells must be defined (red rectangle)

The pseudo-wells are generated when clicking on Go, in the lower left side of the window. The number of pseudo-wells to be generated is user-defined.

Synthetic Layer Properties

The synthetic seismic generation requires different quantities : Density, P wave velocity and S wave velocity. These quantities can be specified in clicking on the icon  : they can be computed using formulas and the appropriate modeled quantities. If the quantity has been modeled, it can be used as Defined quantity .

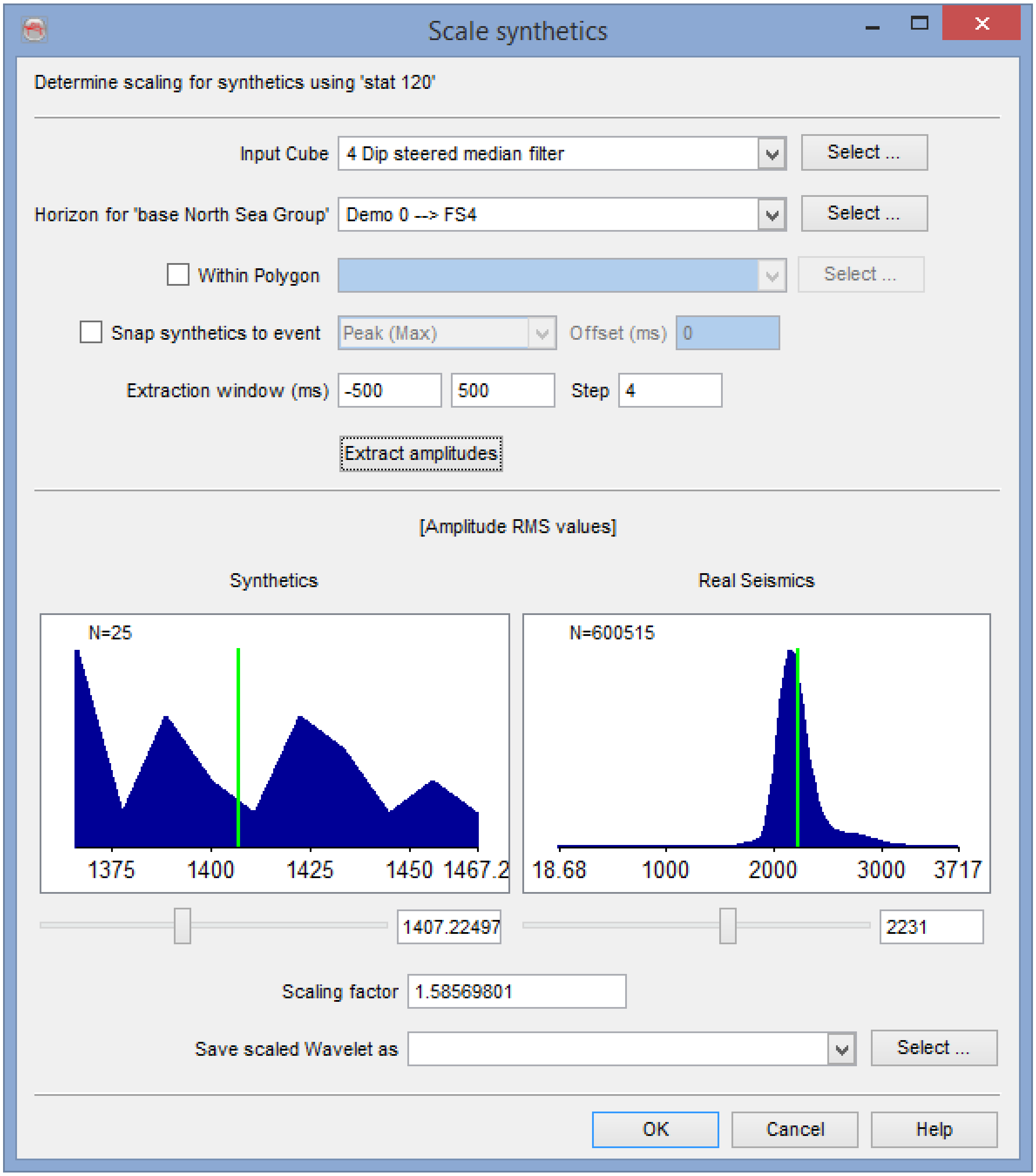


Possibilities to compute the layer properties

Wavelet

The wavelet can be selected from the one already available in the project and listed or a new one can be created in the Wavelet manager accessible from the icon .

Some workflows need to have the synthetic with the same amplitude that the real seismic. The purpose to the scaler is to scale the wavelet by comparison between the synthetic seismic computed at a given horizon and the real seismic extracted in a defined time window regarding this same horizon. To do so click on *Scale*.

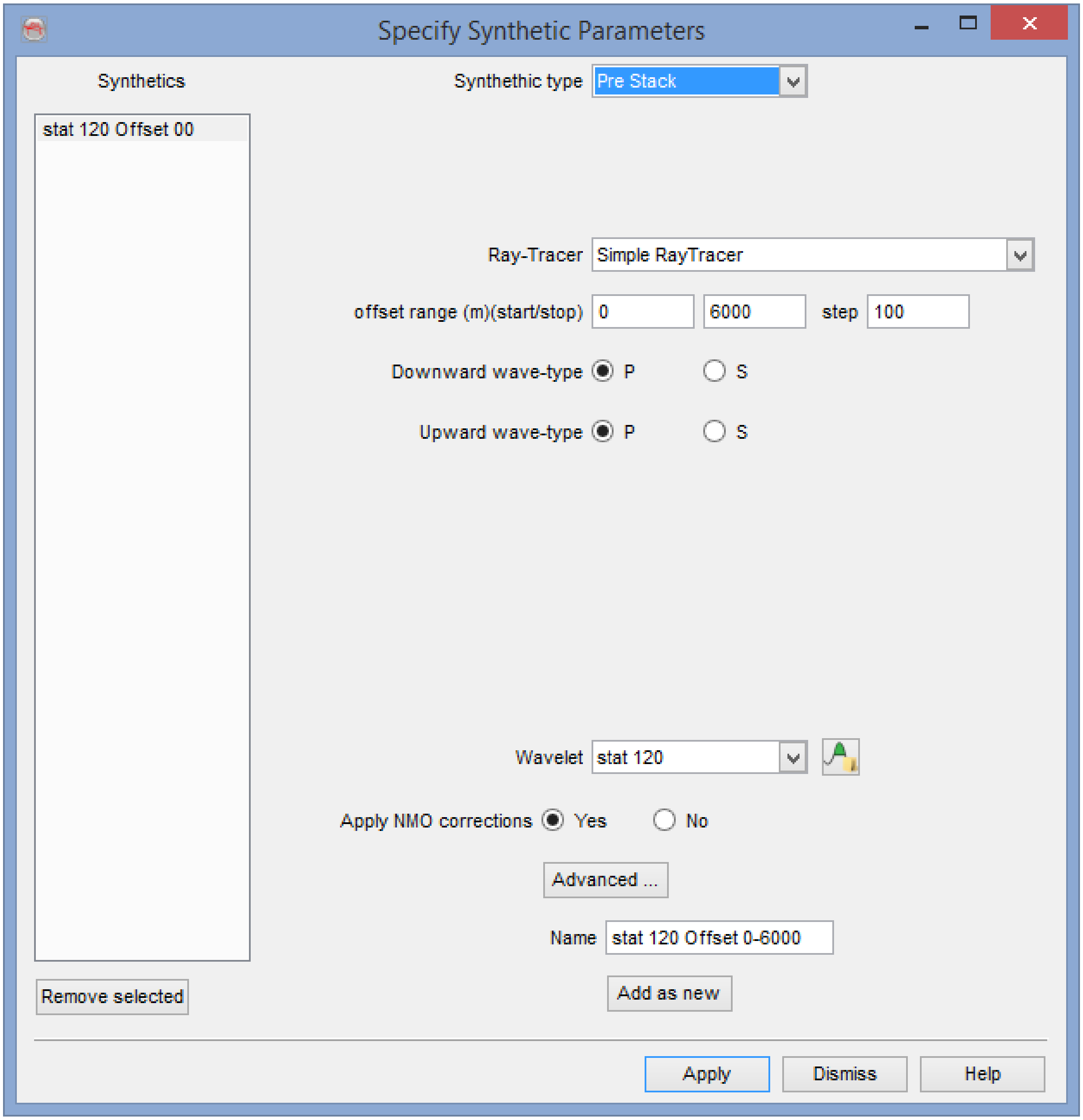


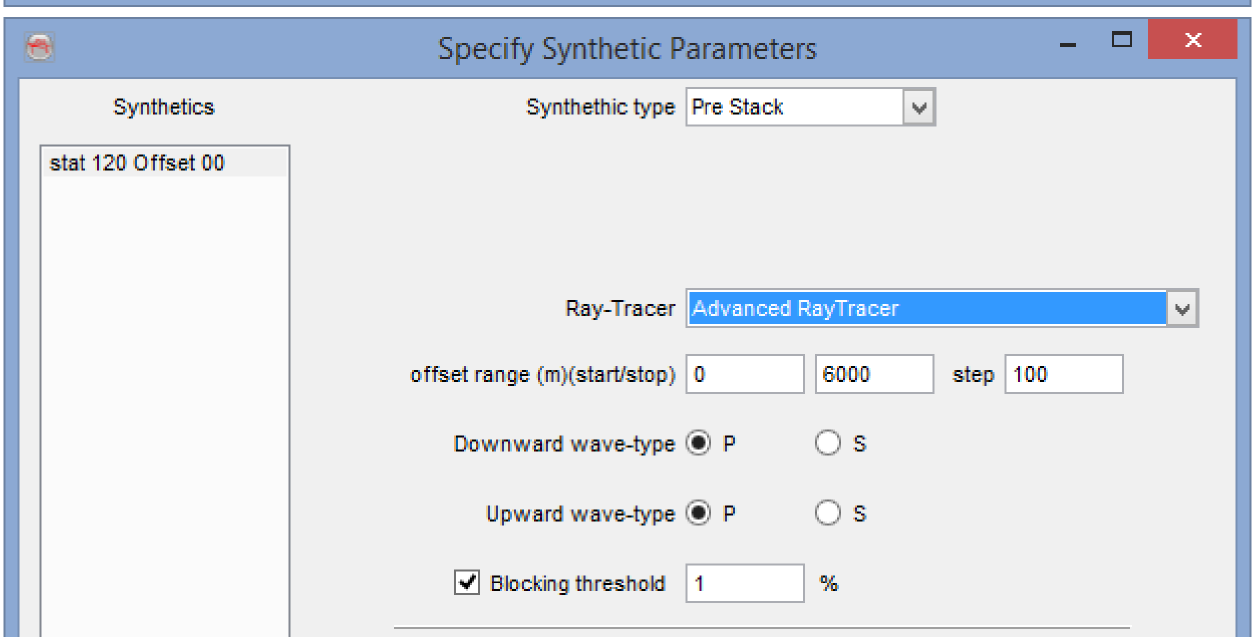
First of all you need to select your reference seismic as *Input Cube*, then the reference horizon for the extraction of the real seismic data. The reference for the extraction in the synthetics is the reference stratigraphic level selected in the main window. The extraction must be done at a level interpreted in the pseudo-wells and in the real seismic. It is possible to restrict the extraction to an area defined by a polygon. Also the reference level in the pseudo-wells does not necessary correspond to a specific event in all the wells, on the contrary horizons are most often interpreted following a same event. Thus it is possible to snap the synthetics to a specific event. Finally the extraction window around this reference level has to be specified. It will depends on the thickness of the interval of interest of your data. Once all these parameters have been given, you can *extract values*.

The histograms for the synthetics and the the real seismic are displayed side by side to be easily compared. A same pick is identified in the two cases and the difference between the two amplitude values is used to determine the *scaling factor*. The scaled wavelet can then be saved and used afterwards.

Ray Tracing

The ray tracer, available via the  icon, allows the creation of synthetics for different offsets and to perform different angle stacks. The source/receiver depths have to be provided. The offset range has also to be specified. The arrival times are calculated by ray-tracing through a horizontally layered isotropic earth model.


Specify Synthetic Parameters
Synthetics
stat 120 Offset 00
Remove selected
Synthethic type
Pre Stack
Ray-Tracer
Simple RayTracer
offset range (m)(start/stop)
0
6000
step
100
Downward wave-type
P
S
Upward wave-type
P
S
Wavelet
stat 120
Apply NMO corrections
Yes
No
Advanced ...
Name
stat 120 Offset 0-6000
Add as new
Apply
Dismiss
Help


Specify Synthetic Parameters
Synthetics
stat 120 Offset 00
Synthethic type
Pre Stack
Ray-Tracer
Advanced RayTracer
offset range (m)(start/stop)
0
6000
step
100
Downward wave-type
P
S
Upward wave-type
P
S
Blocking threshold
1
%

Ray Tracing parameters in the simple and advanced mode

In the advanced mode, the surface coefficient can also be defined if known as well as the spreading geometry.
When pressing *Go* , the synthetics for different offsets are computed. The view is set to be *Free* view by default, you are then able to display a single offset or a limited offset stack in ticking the *Stack* option.

When one offset is displayed, it is possible to make the offset vary from a given *Step* using the arrows .
From the icon , it is possible to display the gathers for the different models.

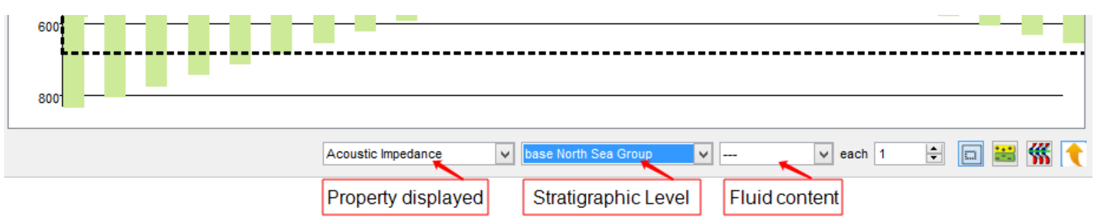
For further information, please refer to Appendix E - Synthetic Data Generation

Display

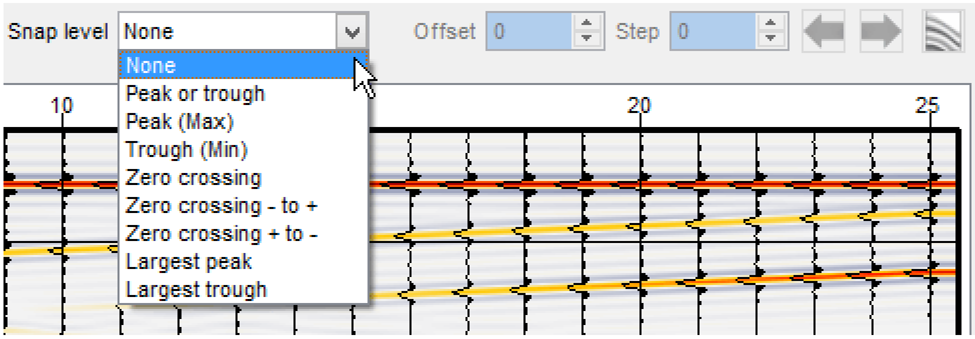
There are several display options within the Layer Modelling feature:

- By default the property logs are displayed regarding the block. When toggled on the icon, the representation is one color per lithology. The property displayed is selected in the selection menu in the lower part of the window.

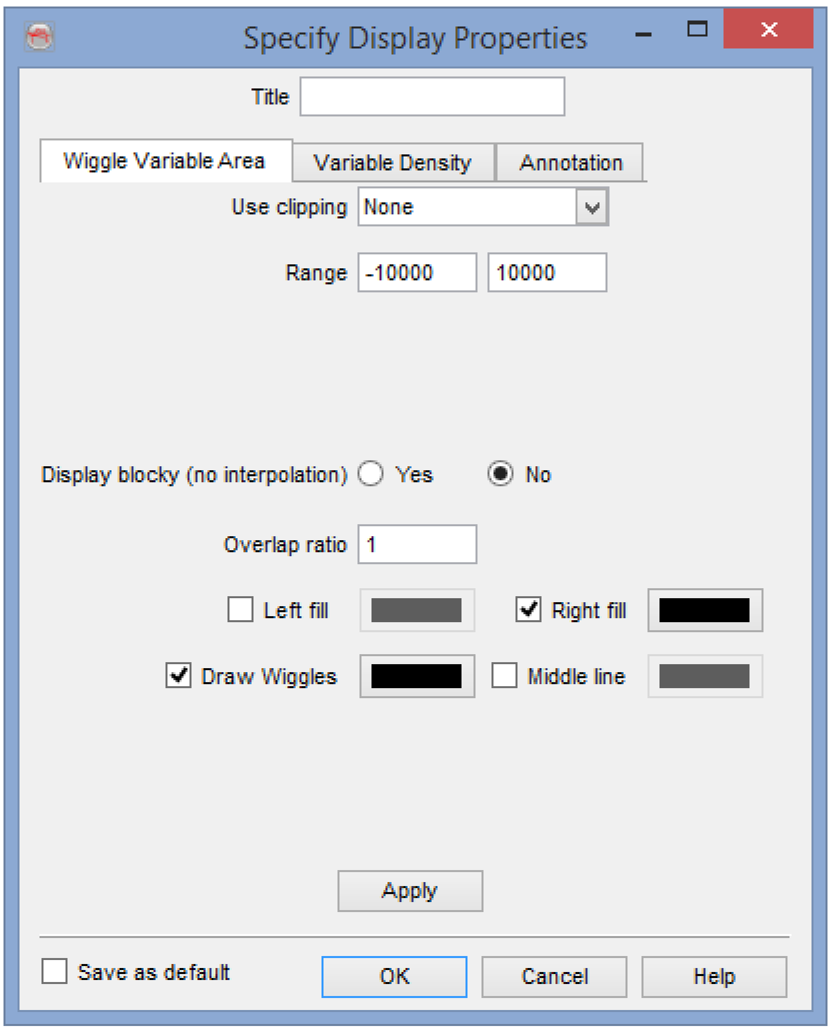
- When the icon is on, if the user zoom on the synthetics, it will not affect the property logs view. The icon is on by default.

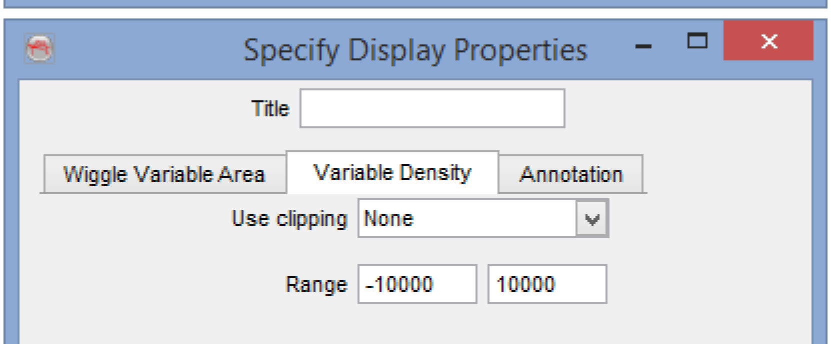


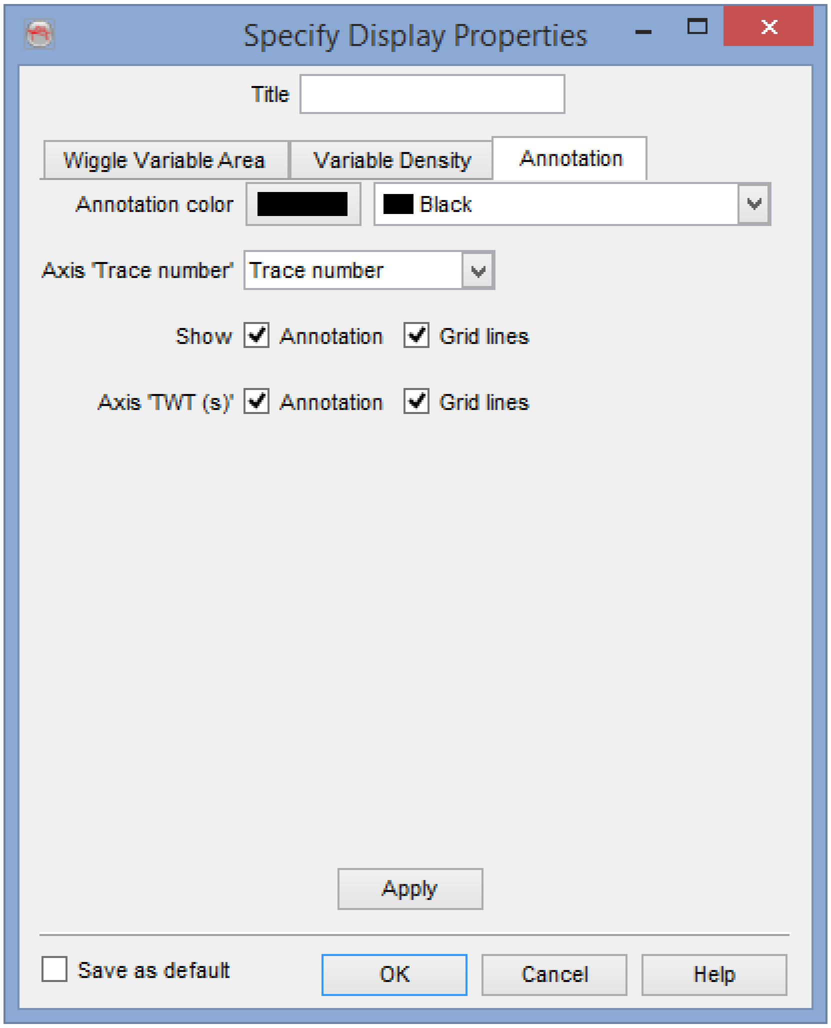
- The stratigraphic level is a marker. The marker position has been modeled and so its position within the pseudo-well can be displayed. In the real wells, markers come from the log data and geological information. It does not necessarily correspond to a given seismic event. On the synthetic from the pseudo wells, it is possible to snap a selected marker to a seismic event (pick, trough, zero-crossing...). This has to be done carefully as some information can be lost: there may be lateral variations of the rock properties that may impact on the phase of the seismic.



• Synthetics display parameters are accessible when clicking on



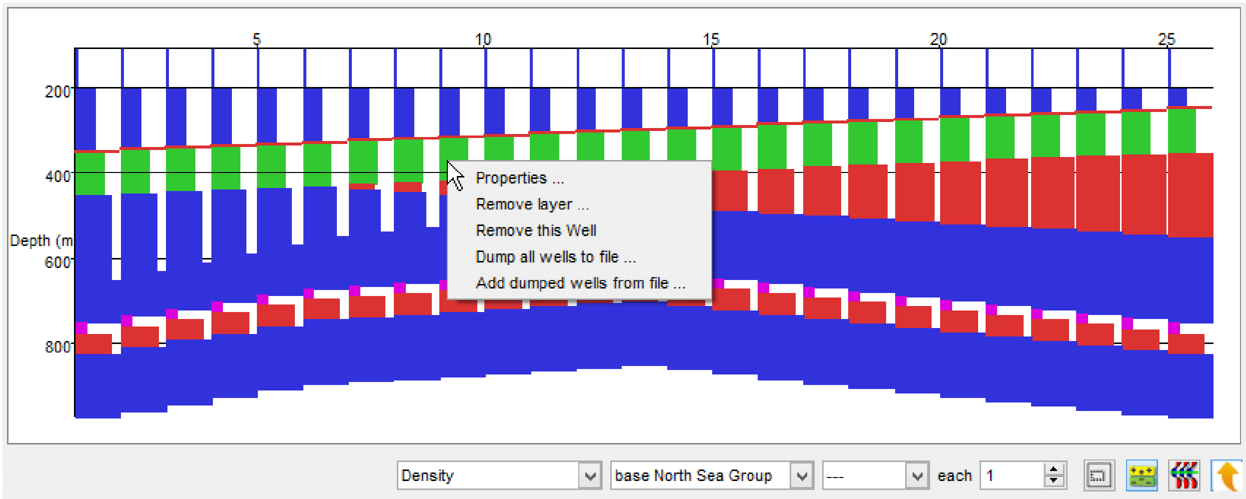




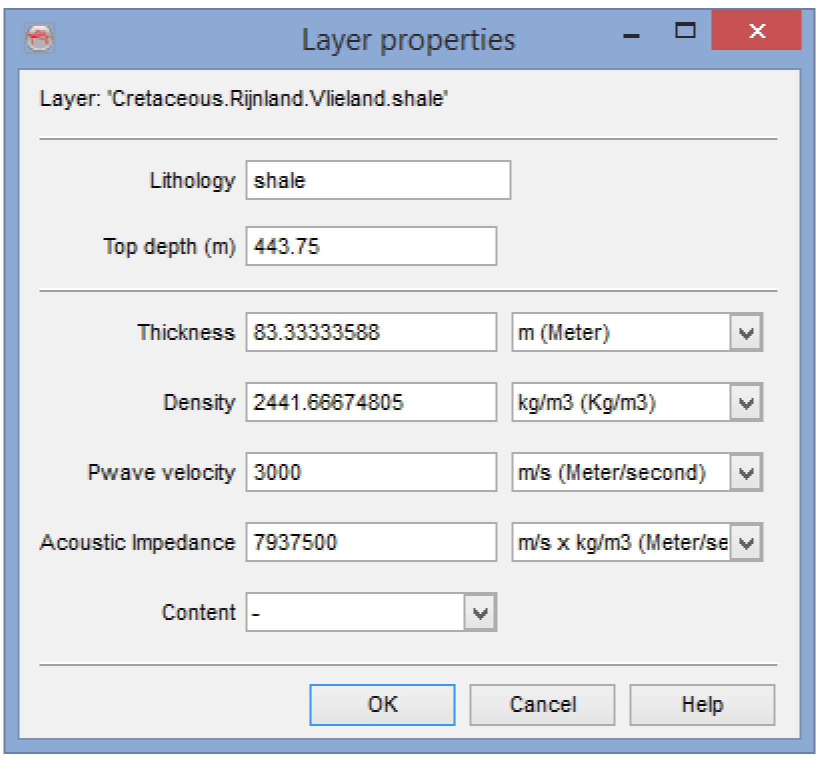
The "Wiggle Variable area" section concerns the display of the synthetic log itself. The "Variable density" section concerns the background, i.e the interpolation between the synthetic traces.

Layer Properties

Once the simulation has been run, the pseudo wells will have been generated and these well properties are then displayed in the lower section. The synthetics are also generated and display in the upper half of the window. When clicking on a given pseudo-well, a line appears to show the selected pseudo-well and right-clicking on a particular layer of this selected pseudo-well gives a menu to various options.



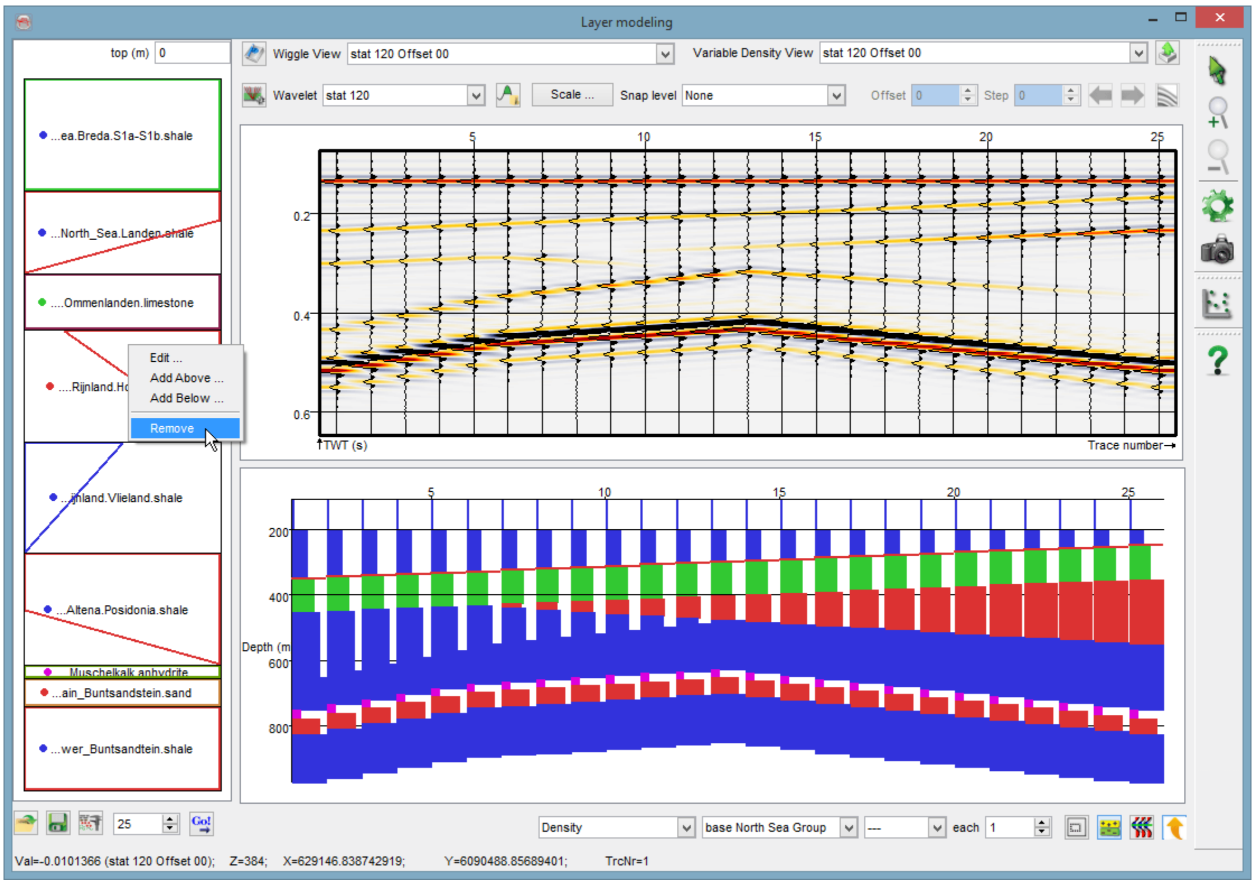
For this selected pseudo-well, the *Properties*... option gives access to the characteristics of this specific layer in term of thickness and modeled properties. In the layer-based modeling (basic or stochastic), these values can be manually modified. The fluid content can also be edited. Changes are saved when clicking *Ok* and the display is automatically updated.

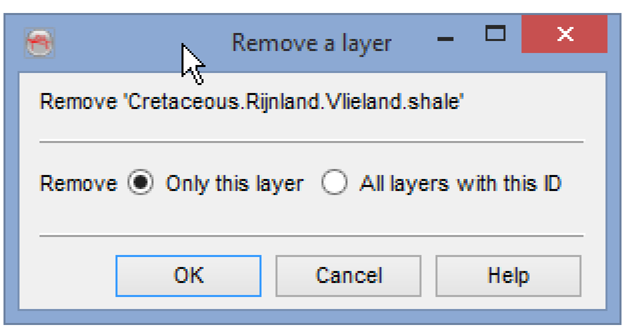


In the SynthRock plugin, in the Profile mode, a similar window is available on right-clicking on any trace on the lower rectangle where a selected property is displayed for the different pseudo-wells. In selecting *Inspect values*, the window opens. In this case it is an informative window: the different property values can not be changed. The fluid content however can be edited. The lithology in this case is unknown as it is based solely on well log(s).

Remove Layer

The existing layers of a model can be removed at any time, by right clicking on the left hand side pane, containing the simulation information.



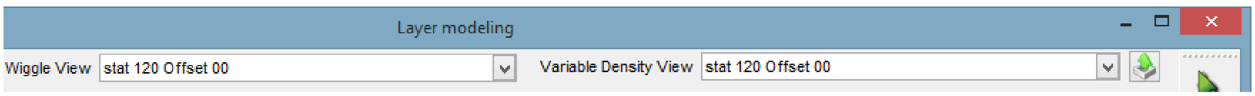


Thereafter, the regeneration of the pseudo-wells can be done by clicking on Go! .

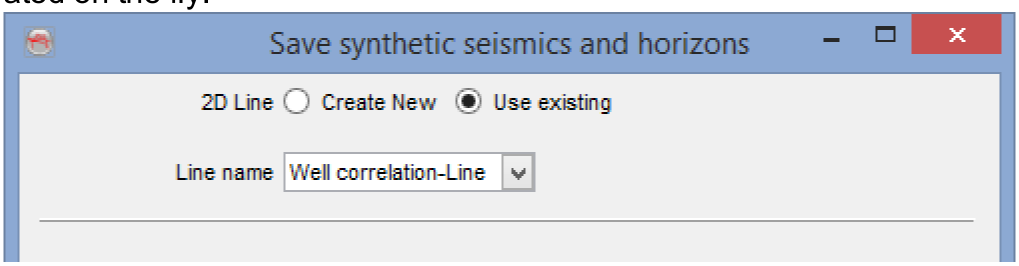
Export Synthetic Datasets

The synthetic seismic data (both post-stack and pre-stack), the layer property synthetics in Time (e.g. AI, Density etc.) and the stratigraphic levels/markers, from all modeling modules (i.e. Basic, Profile and Stochastic) can be exported along 2D lines. The stratigraphic levels/markers in the modeled pseudo-wells are essentially exported as 2D horizons.

This is achieved by clicking on the  icon at the top right of the modeling window (see below).

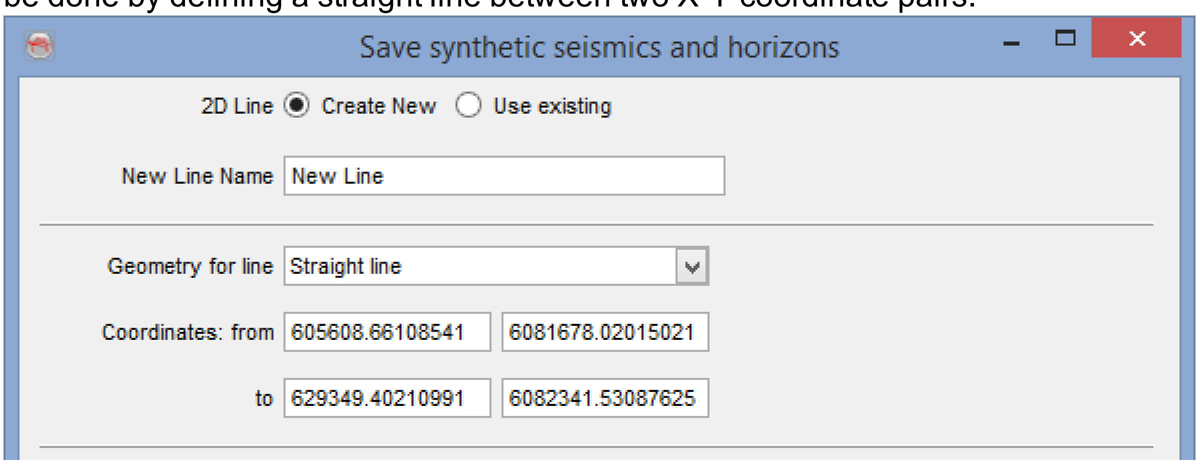


Export of the synthetics can be done onto an already existing 2D line or a new line created on the fly.



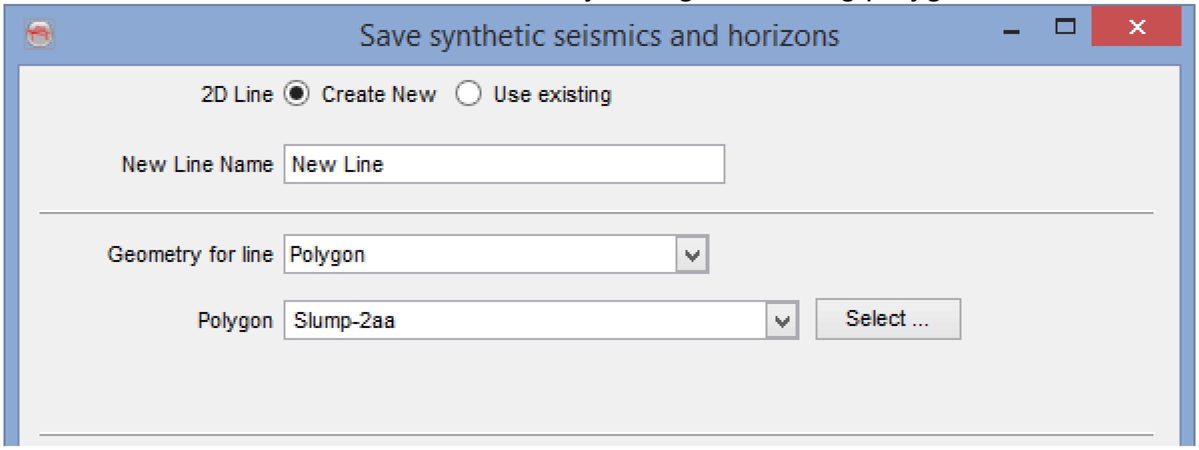
Selecting an existing 2D line

If the 2D line is created on the fly, the *Geometry for line* has to be defined as well. It can be done by defining a straight line between two X-Y coordinate pairs.



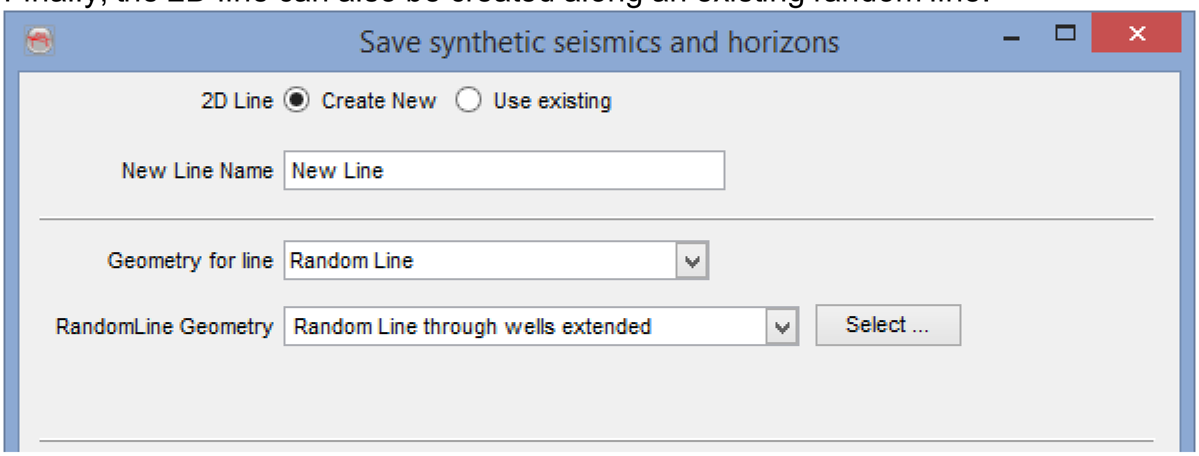
Creating a straight 2D line between two X-Y coordinate pairs

The 2D line can also be created, on the fly, along an existing polygon.



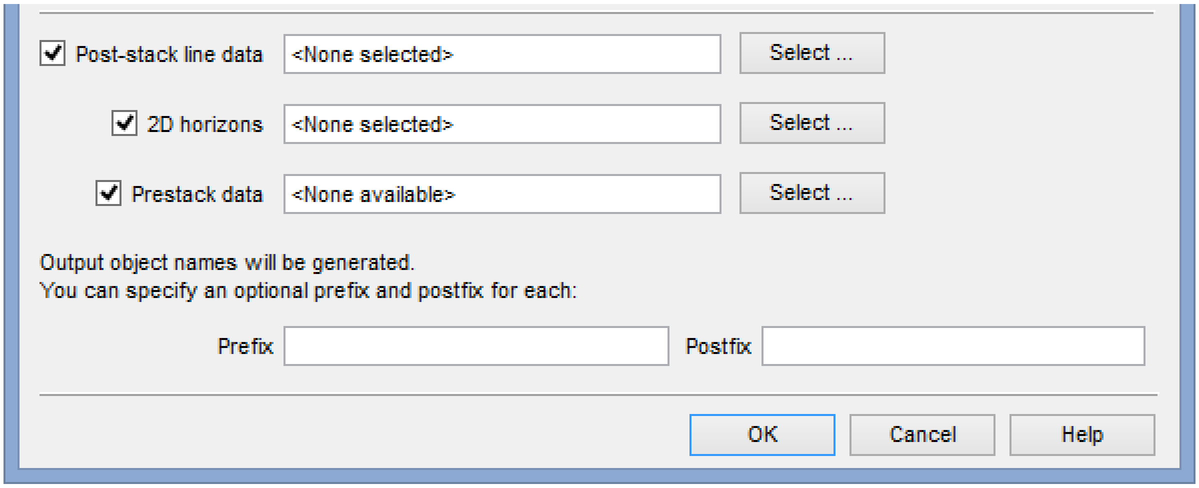
Creating a 2D line along a polygon

Finally, the 2D line can also be created along an existing random line.

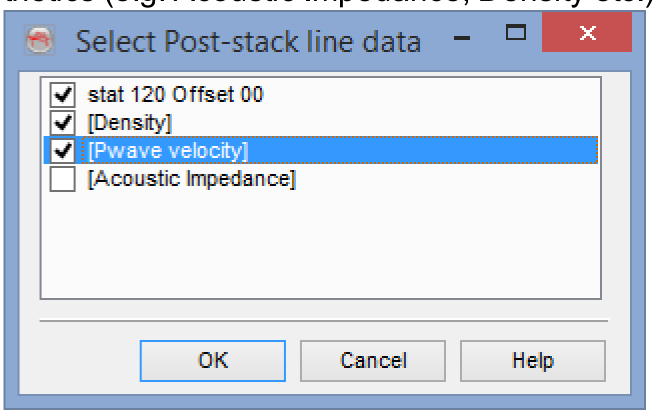


Creating a 2D line along a random line

Now, a selection on post-stack data, 2D horizons and pre-stack data can be made for exporting along the 2D line.

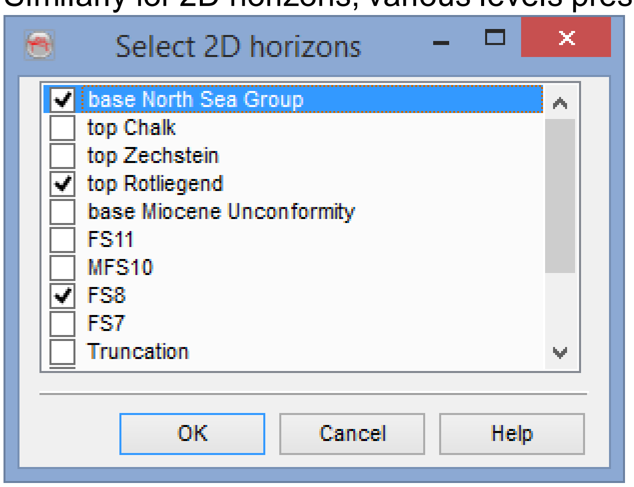


For post-stack data, user can select synthetic seismic and various layer property synthetics (e.g. Acoustic Impedance, Density etc.).



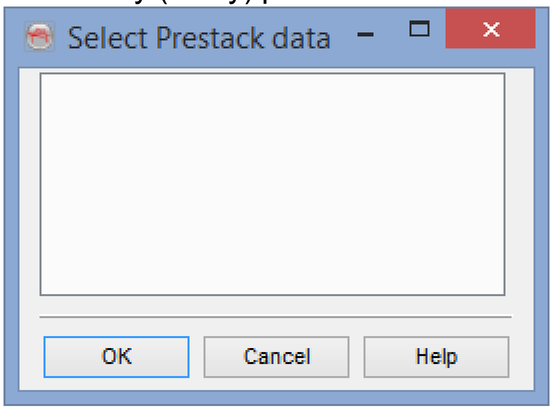
Post-stack data selection for export

Similarly for 2D horizons, various levels present in the pseudo-wells can be selected.



2D horizon data selection for export

and finally (if any) pre-stack data can be selected.

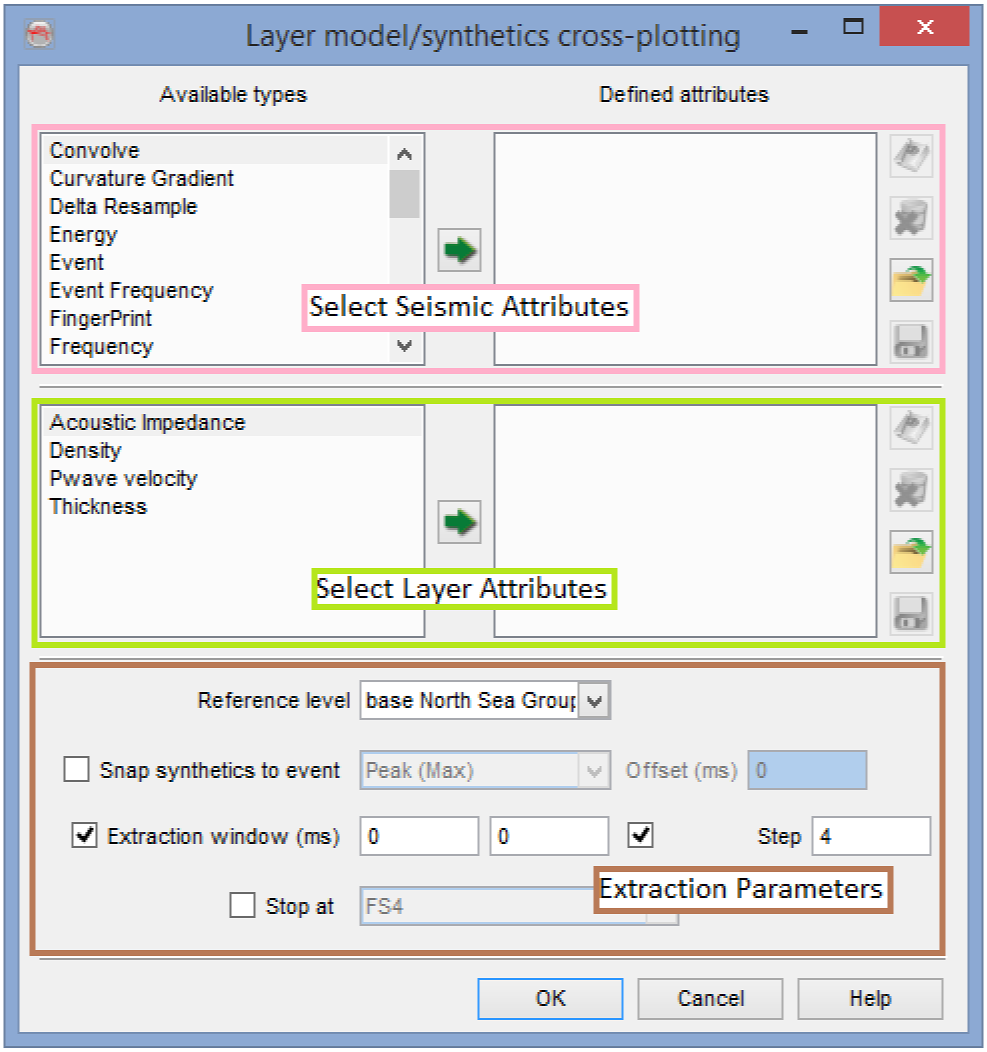


Pre-stack data selection for export

Optionally, a prefix and/or postfix can be specified for various data items. Pressing *Ok* will export the selected data items along the 2D line.

Cross-Plots

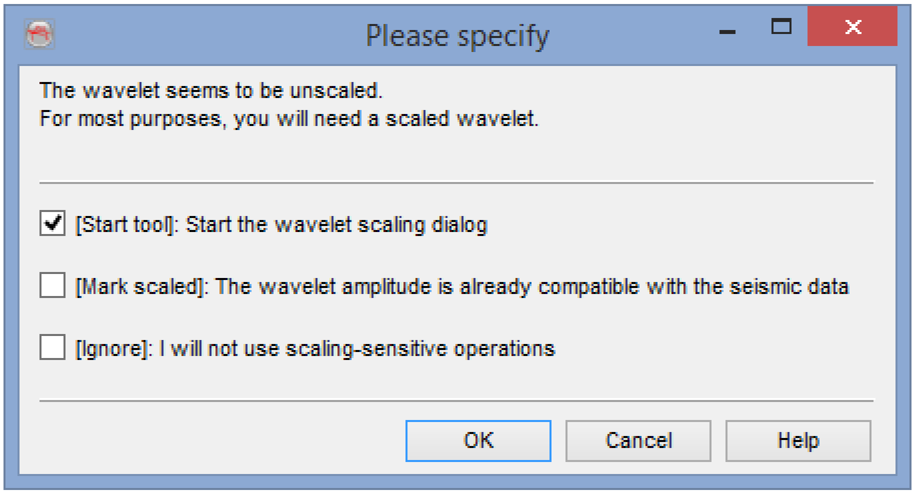
The cross-plot tool in the Basic layer modeling can be started from the icon  . It allows to analyze seismic and layer attributes from the modeled data. On the main window, the user select the attributes to be extracted and the extraction parameters.The extraction window is related to a reference level. Its length is user-defined. The appropriate extraction window size has to be defined regarding the interval of interest. The user has to provide a step that corresponds to the sample rate within the extraction window.



Once the attributes and the extraction parameters defined, the crossplot window opens and is similar to the one available for the classic seismic/well analysis.

Wavelet Scaling

If the wavelet has not been scaled to the real seismic, a pop up window will first appear prior to access to the attribute selection window.



The user has 3 choices :

- [Start tool] to start the Scale wavelet window
- [Mark scaled] if the wavelet is considered as scaled or does not need to be scaled
- [Ignore] if the attributes that are going to be extracted will not need a scaled wavelet

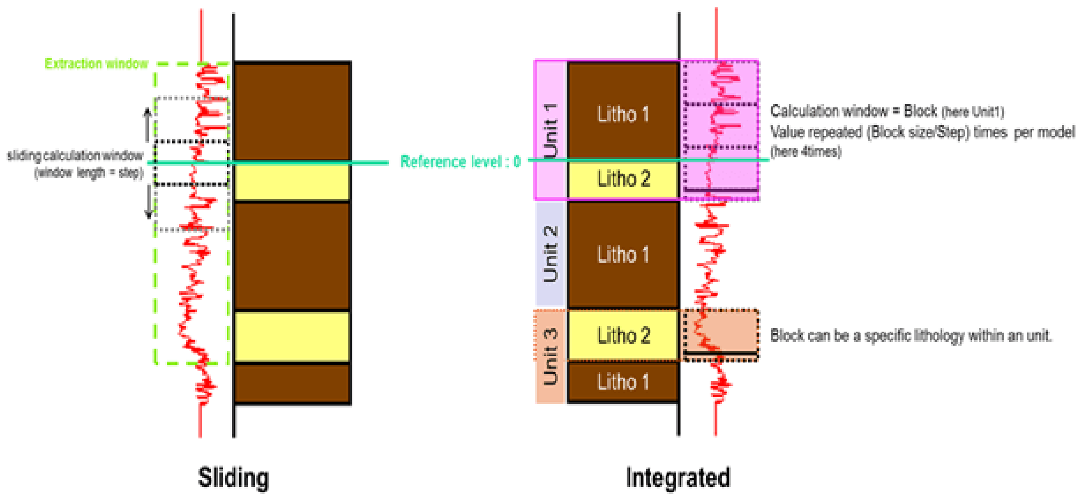
Once the wavelet is scaled or marked as scaled, it will be remembered and the window will not appear again.

Seismic Attributes

The seismic attribute selection/definition is comparable to the main attribute window: the same attributes are available. An attribute can be selected in the list of *Available* types and add to the *Defined attributes* using the  button. The parameters of the attribute have to be specified. Synthetic seismic generated from the models can be used as input data. All the listed attributes are not necessary using synthetic seismic.

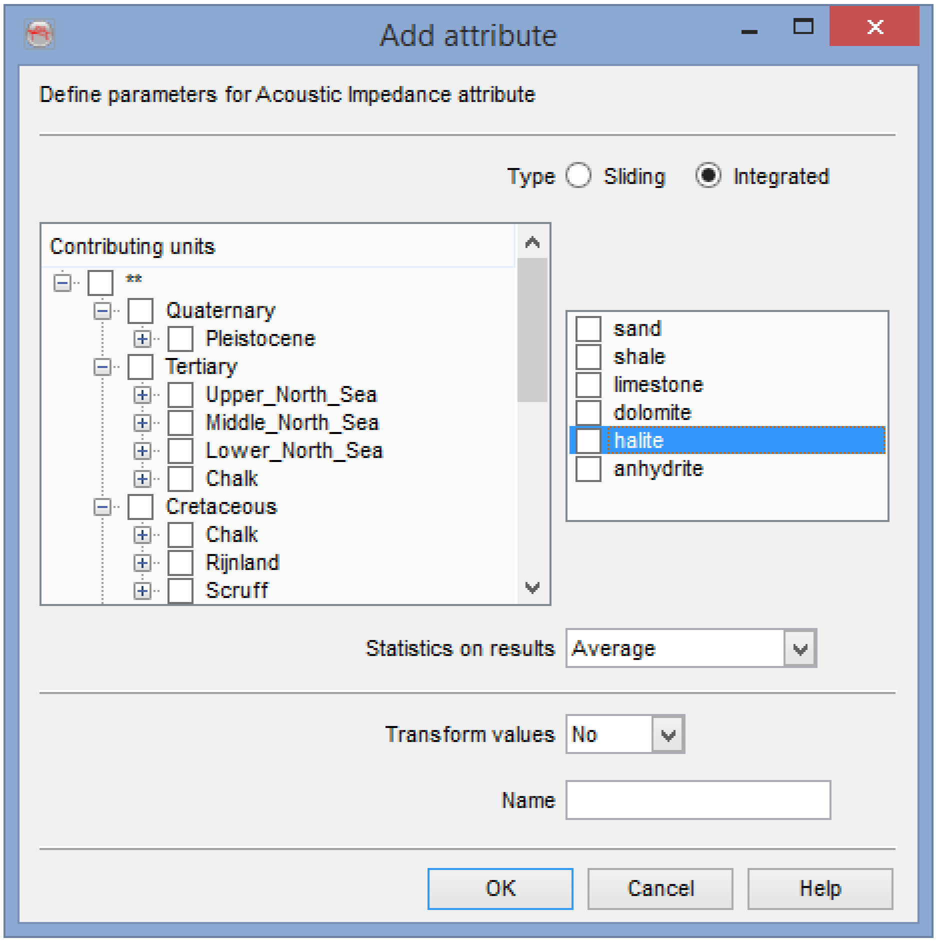
Layer Attributes

For each modeled property, data can be extracted either along the log using a defined extraction window or by layers:



• Sliding: the property value is extracted within the extraction window where a calculation window slides along the well. The size of the calculation window is defined by the step provided by the user on the first crossplot window. The output is the nearest sample, the average, the median, the RMS or the most frequent. At the end the attribute has *[number of pseudo-wells* round up (Extraction window size/Step)]* samples.

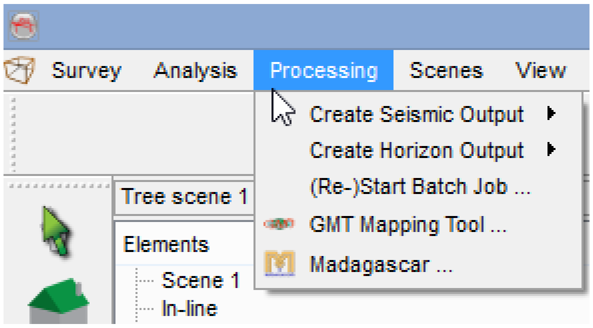
• Integrated: Different lithologies have different properties. In the Layer Description, the behavior of each lithology has been defined for the modeling. The modeled properties can then be extracted and easily compared with the crossplot tool. Each modeled property can be extracted at the different level of the framework, for one or more lithology of one or more layer. One value corresponds to one model. The output will be the average, the median, the standard deviation, the minimum or the maximum of the extracted data for each model. The number of sample depends on the step and the thickness of the extraction block. The thickness of the block may vary from one pseudo-well to another depending on the model definition. Thus the number of sample can vary from one pseudo-well to another. For one attribute, the number of sample is equal to the sum of the round up (Extraction block thickness/Step) of each pseudo-well.



In both case, it is possible to transform the attribute value in applying the function power, log or exponential. The attribute is extracted on each pseudo-well.

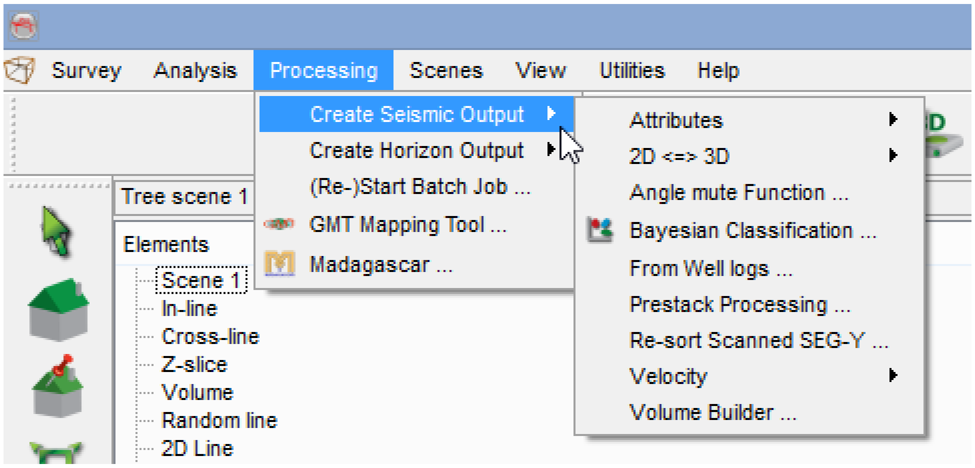
Processing

Table of Contents



Create Seismic Output

The Create Seismic Output option leads to a number of further choices, detailed in the following subsections:



[Attributes - 2D](#)
[Attributes - 3D](#)
[Multi-Attribute](#)
[MultiCube DataStore](#)
[Along Horizons](#)
[Between Horizons](#)

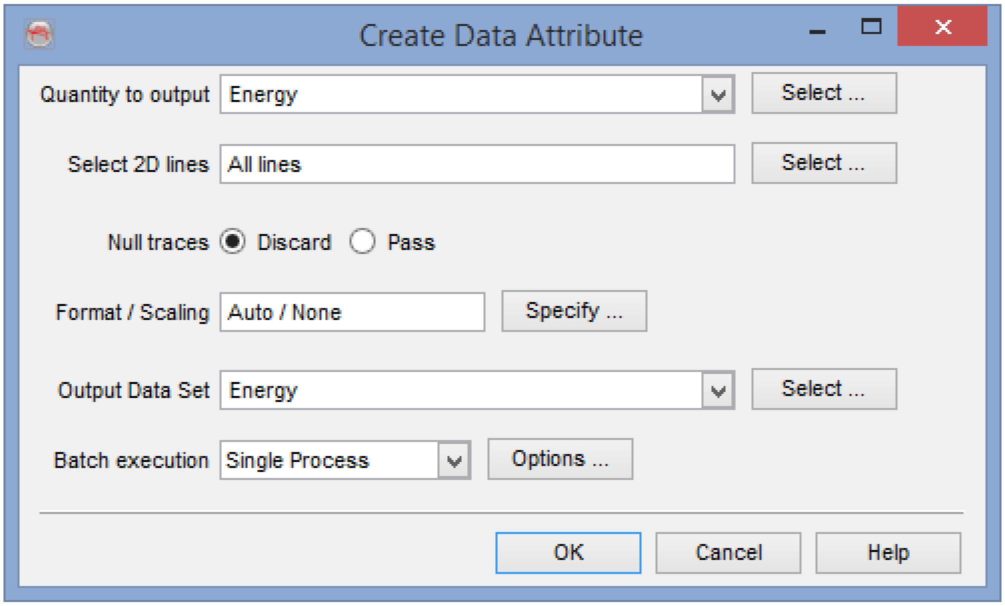
Attribute

In this module any (attribute) volume can be calculated and saved to disk. In case of *2D attributes*, the output is a new data set. The volume output module can be run in batch mode, allowing to continue working in the main window while the processing is running.

This module creates, for example, attribute cubes, neural network cubes, or filtered data cubes.

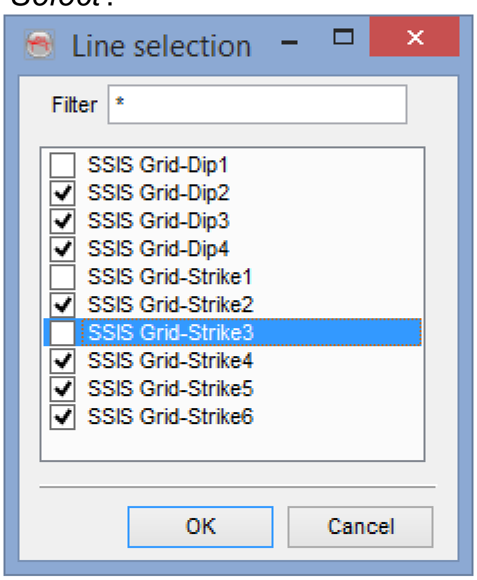
2D - Create Output

A data set attribute can be created through the following procedure: *Processing* > *Create Seismic output* > *Attribute* > *2D*



First, *Select* the output quantity : it can be either a stored 2D volume or an attribute from the active 2D attribute set.
Note that only attributes from the current attribute set can be selected in the "*Select quantity to output*" window. To output an attribute from another attribute set, you must select this attribute set in the attribute module.

Though the default is set to '*All Lines*', a selection of lines can be specified by clicking '*Select*':

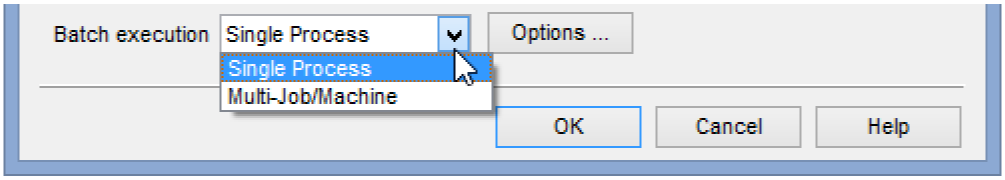


Optionally, the output can be scaled with a *Shift* and a *Factor*.

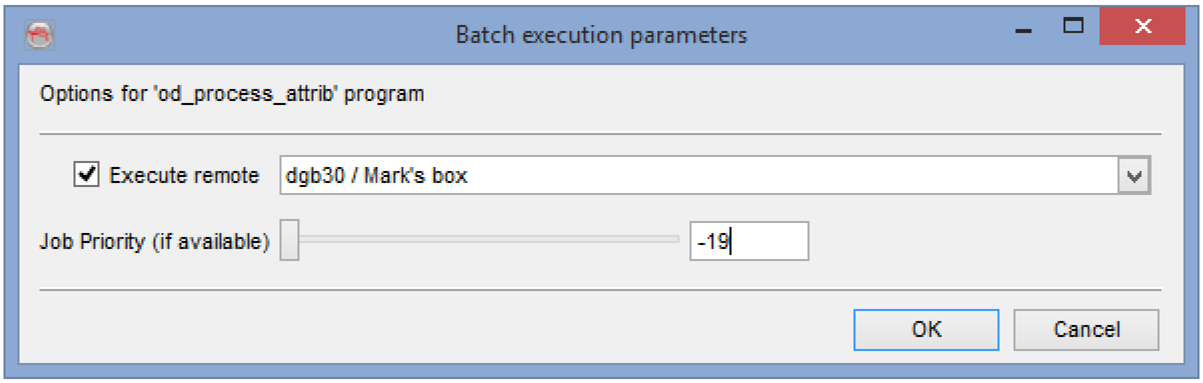
```
Output = Factor * Input + Shift
```

Null traces can be discarded.

It is also possible to choose between *Single Process* and *Multi-Job/Machine* processing:

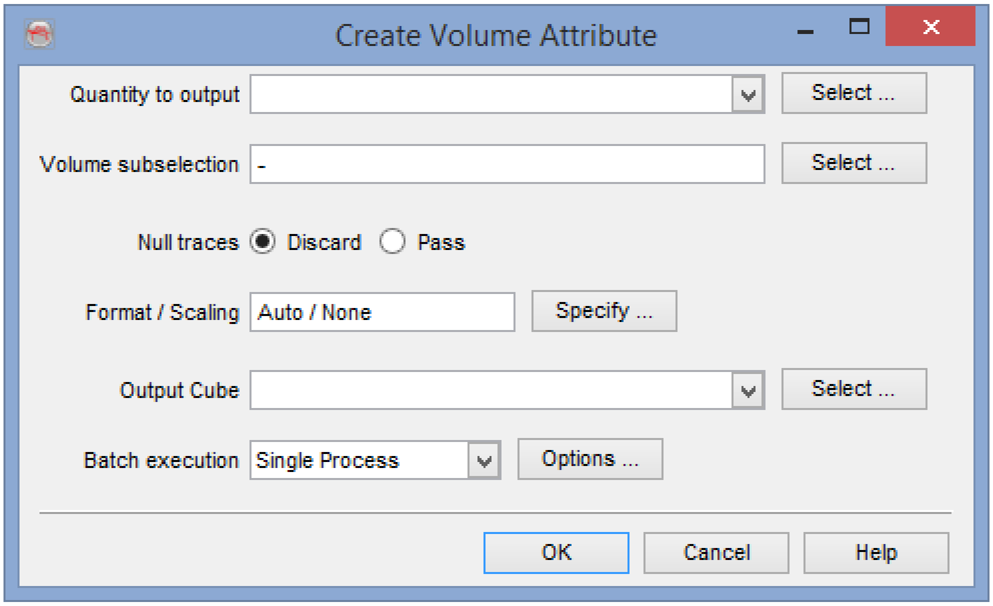


And to set the job priority (if deemed necessary):



3D - Create Output

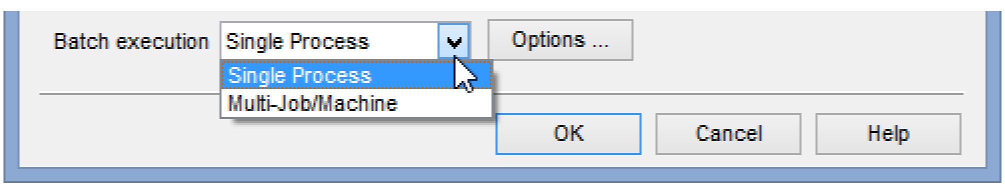
To create a 3D seismic output from an attribute, follow the path *Processing > Create Seismic Output > Attribute > 3D*.



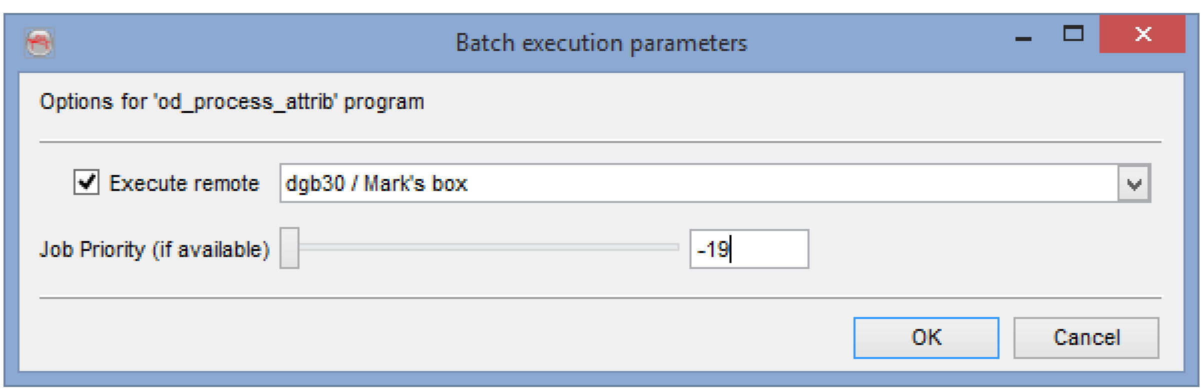
First, *Select* the output quantity. Optionally, a *sub-volume* can be specified.
Note that only attributes from the current attribute set can be selected in the '*Select output quantity*' window. To output an attribute from another attribute set, you must select this attribute set in the attribute module.

Null traces can be discarded.

It is also possible to choose between *Single Process* and *Multi-Job/Machine* processing:



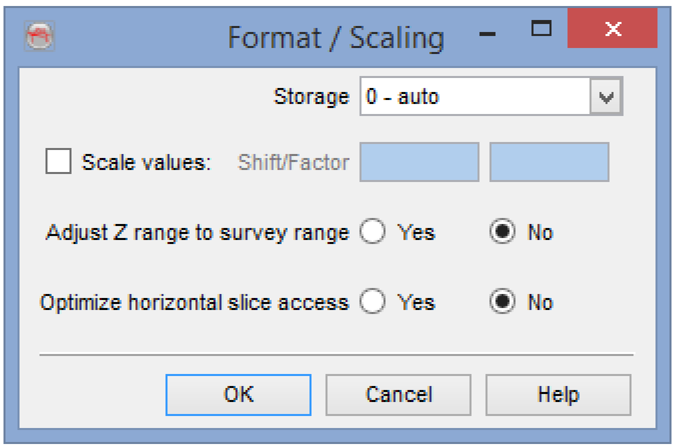
And to set the job priority (if deemed necessary). This is equivalent to the '*Nice level*' on Linux, determining how much priority the process will take on the remote machine:



Optionally, the output can be scaled with a Shift and a Factor:

```
Output = Factor * Input + Shift
```

Storage: OpendTect can store data internally in 8-, 16-, 32-, and 64-bit seismic data formats. 8-bit signed has a data range between -127 and +127. 8-bit unsigned ranges between 0 and 255. Similarly, 16-bit signed ranges between -32767 and +32767 (unsigned 0 - 65535). The data is stored in the same byte-format as the input by default (Storage is set to 0 - auto). This is chosen when specifying the *Format/Scaling.*



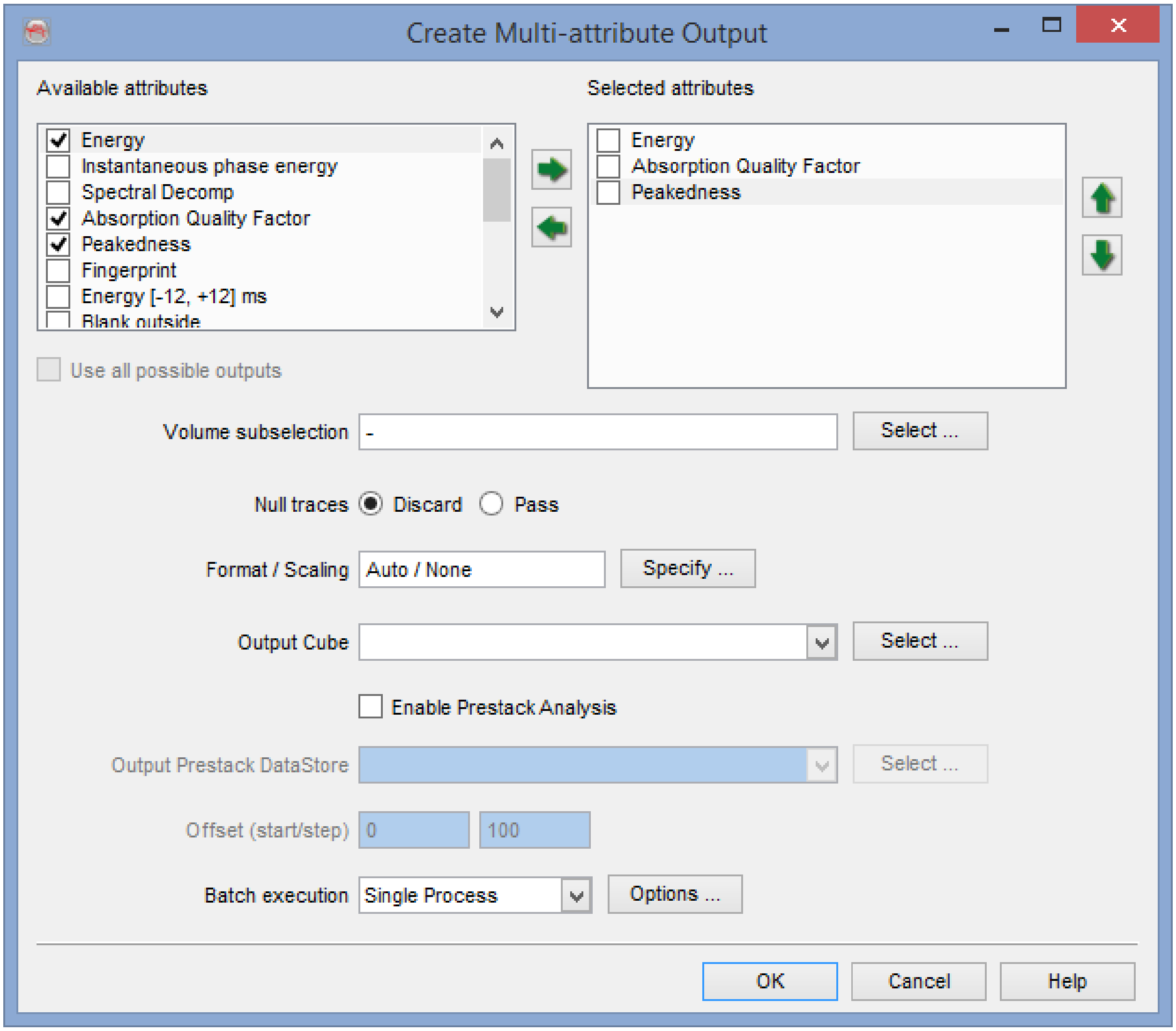
Adjust Z range to survey range: this option adjusts the new scaled seismic to the survey range.

Optimize horizontal slice access: For better performance when loading time slices, set this option to 'Yes'. This compromises some speed in loading crosslines, but it loads time slices significantly faster.

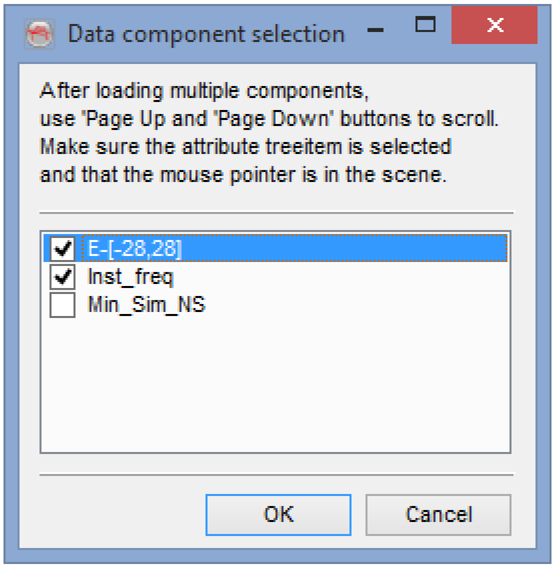
Processing: OpendTect batch jobs for 3D volumes can be processed on a *single-*, *multiple machines*, or Clustering. For more information look at batch processing.

Multi Attribute

Multi-attribute output enables the user to create a volume with several attributes.



On loading the volume onto an inline (for example), the user is prompted to select which of the available attributes he wishes to use:

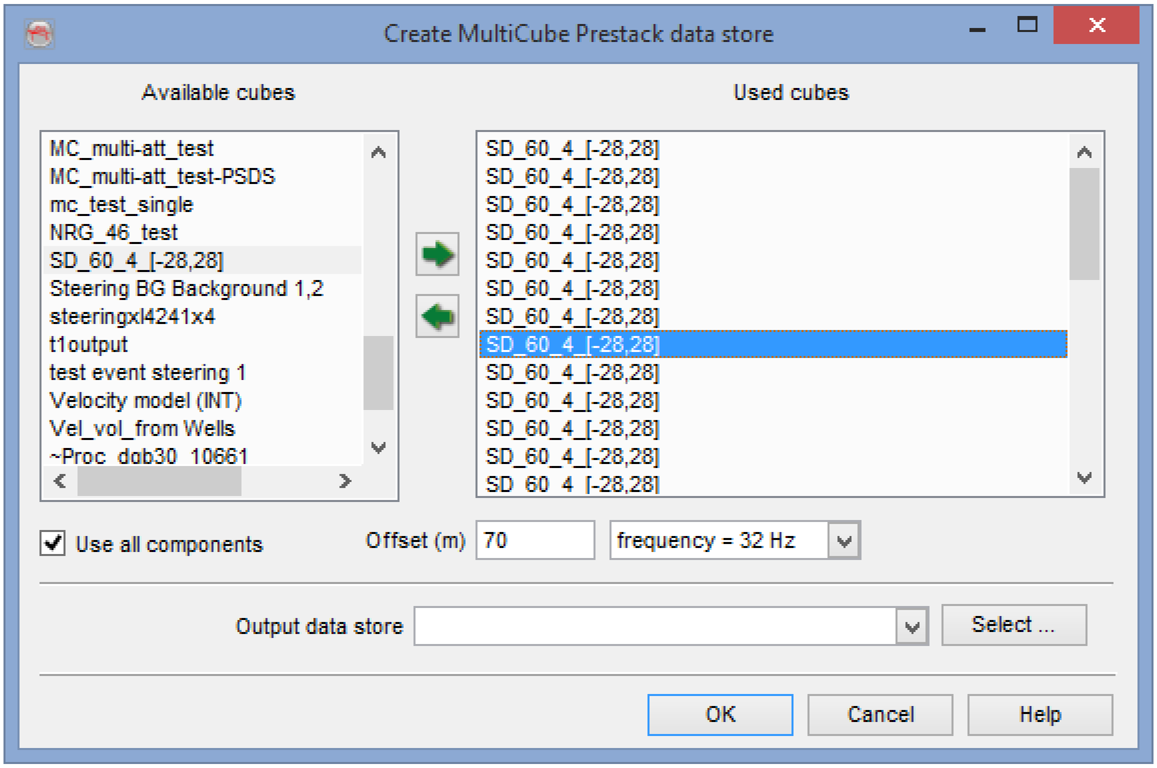


Multi-Cube Data Store

Multi-components cubes for some attributes (e.g Spectral Decomposition, Steering attributes) can be created.

First, create your attribute in the attribute engine.

Create the volume output: Go to *Processing* > *Create Seismic Output* > *Attribute* > *Multi-cube data store* and select a volume that contains multiple components (here, Spectral Decomposition as example):

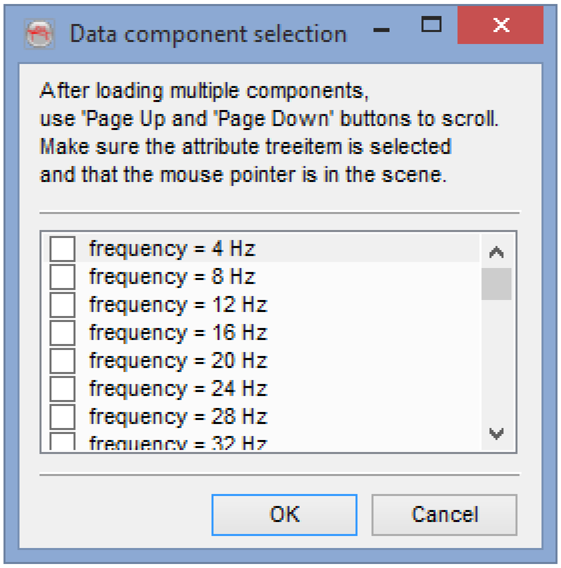


Select a component of the cube and assign it a 'pseudo-offset' value. Repeat this process of pseudo-offset assignment for all of the components that you wish to be present in the output, name it and press '*Go*'.

Multi-component cubes can be exported as SEG-Y or simple ASCII file but only one component per output. The choice of which component is given to the user during the export process.

How to display the Multi Component Volumes?

Whenever displaying a multi-component attribute volume on an inline/crossline, the component selection dialog box (below) will pop up. In this dialog one or more outputs to be displayed on the same section must be selected.



Once several components are selected and displayed for an element (e.g. an inline), place the mouse in the scene, and use the keyboard's PAGE-UP/PAGE-DOWN keys to view the next/previous slice in real-time.

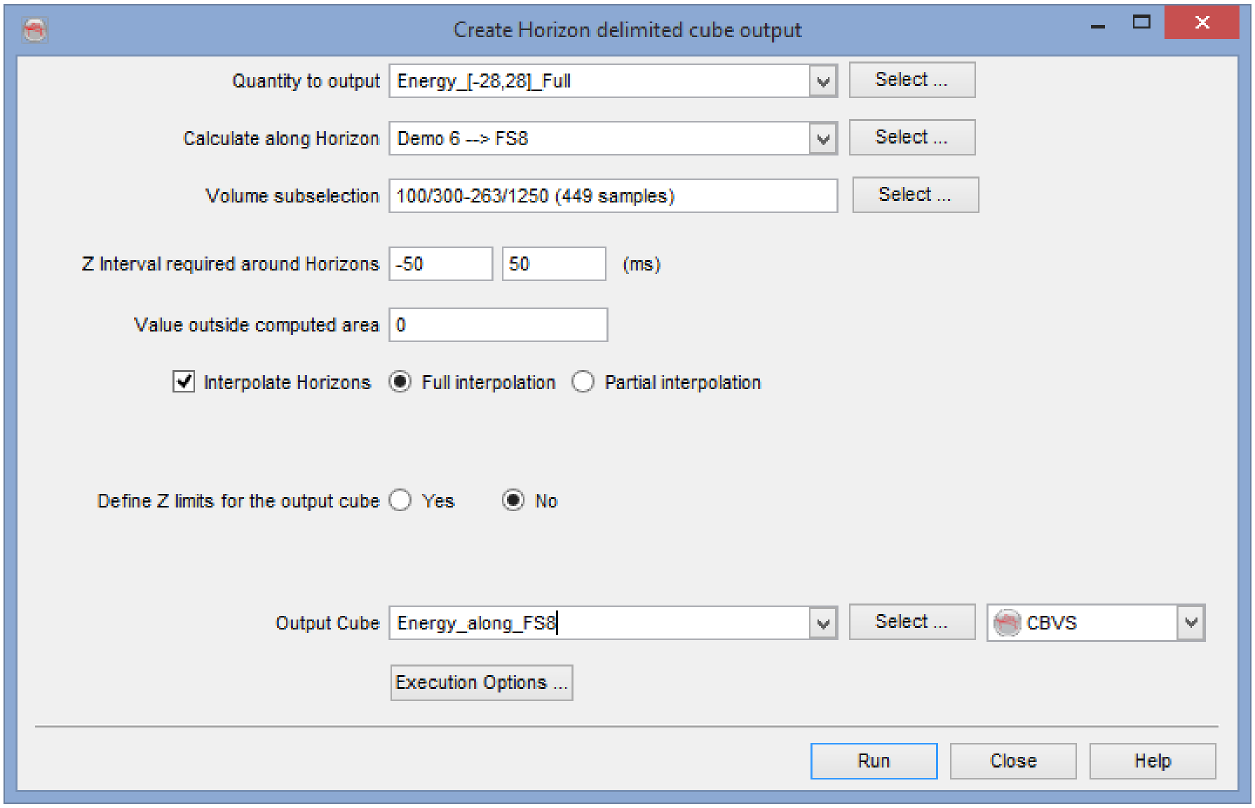
Along Horizon

To create a seismic output in a time interval relative to a single horizon, the quantity to output has first to be selected from the list of stored data or attributes from the current attribute set. Specify the horizon and the Z interval relative to this horizon. A sub-area can be specified.

The *Value outside the computed area* is the undefined value. The standard undefined value in OpendTect is `1e30`, but any other value can be specified.

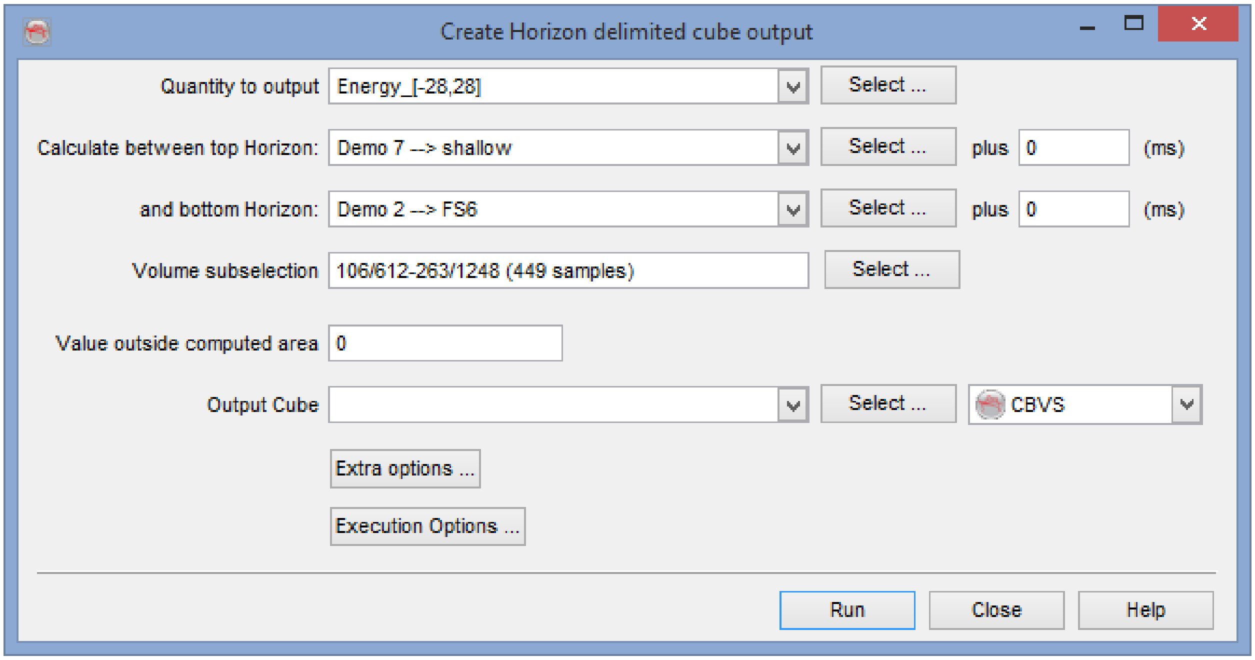
Optionally, the horizons can be interpolated. The interpolation can be full or partial.

A calculation parameter file is automatically created with a default name in the *Store Processing Specification* field. This file allows to re-start the calculation process easily if needed. The '*Execution Options*' can be set to use either single or multiple machines processing.



Between Horizons

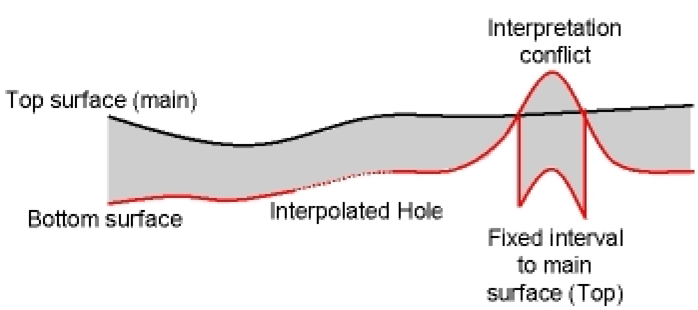
To create a seismic output between two horizons, first the quantity to output has to be selected from the list of stored data or attributes. Specify the horizons that form the upper and lower boundaries of the output volume. A *Z shift* to be specified can be applied to the upper boundary and/or to the lower boundary. A sub-area can be specified.



The *Value outside the computed area* is the undefined value. The standard undefined value in OpendTect is 1e30 or Undef, but any value can be specified.

In *Extra options*, horizons can be interpolated. The interpolation can be full or partial. When partial, only the gaps smaller than a user-defined number of traces are filled. A *fixed Z interval* length can be added to the leading horizon when the second surface is missing or in case of conflict during the interpolation. To constraint the interpolation, the *Z limits* for the output cube can also be specified.

For '*Execution Options*', please see: Batch Execution



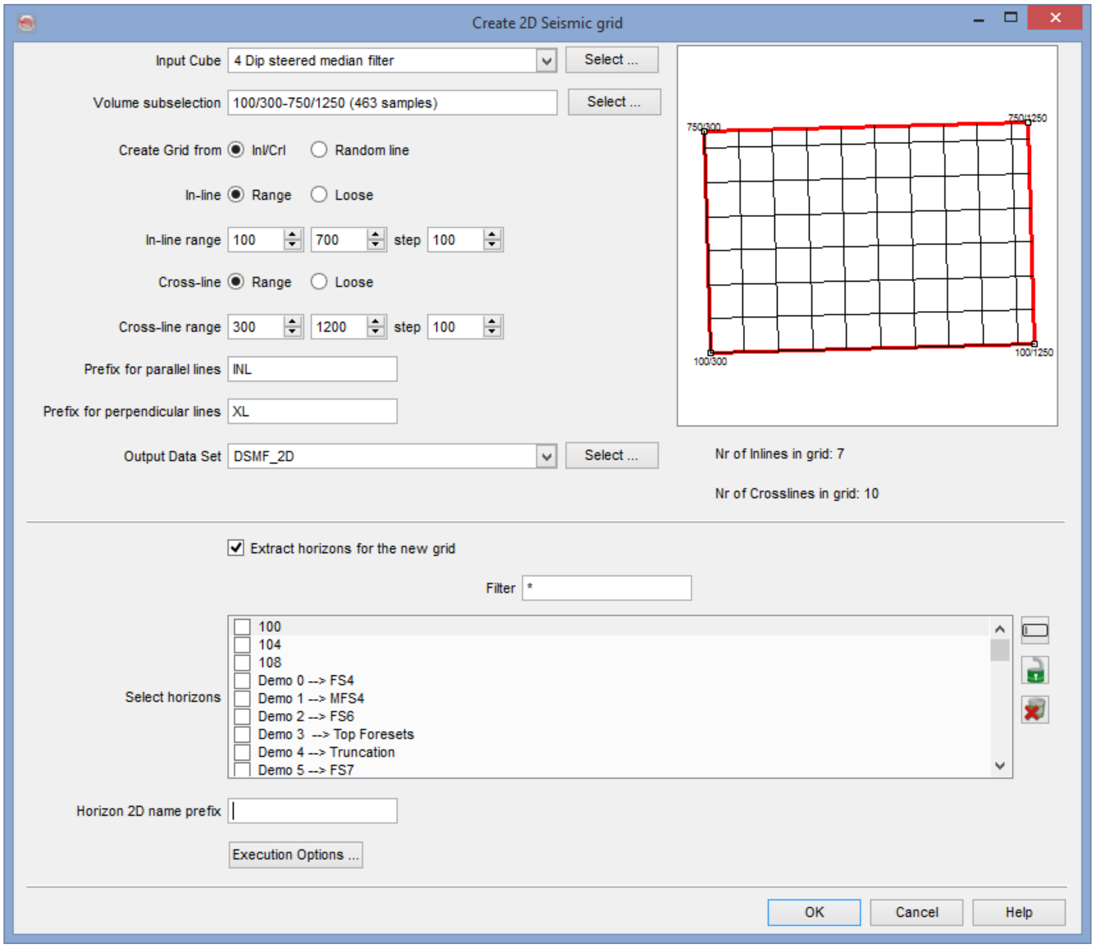
2D to 3D Conversions

There are several ways to convert data between 2D and 3D:

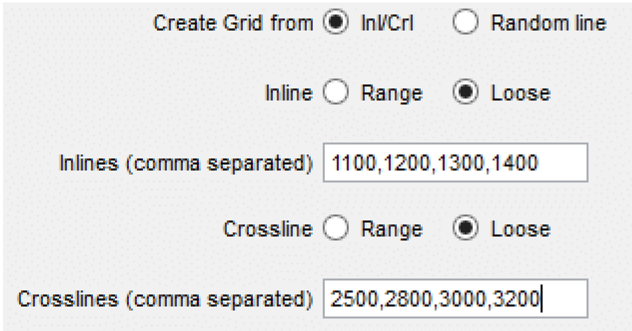
Create 2D Grid

The *Create 2D from 3D* option is an interactive tool for creating 2D-lattices from a 3D volumes.

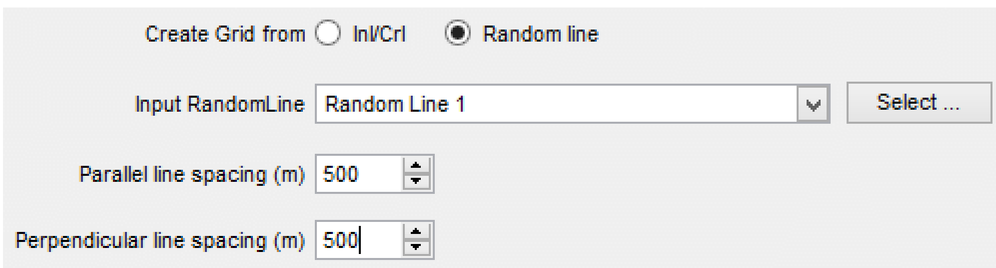
This option can be used to create a 2D grid with a fixed grid spacing. When selected, the *Create 2D Grid* window is launched (see below). Here, specify the input 3D seismic volume and the output data set name. The output grid is generated according to the dip (parallel) and strike (perpendicular) direction of the selected volume. The prefix labels are used as prefixes to the output line names, stored to the specified new data set name. The grid spacing is the constant spacing between the two lines. At the bottom, the total number of parallel and perpendicular lines will be updated according to the grid spacing. By pressing *Ok*, a batch process will start to generate the 2D grid. When the batch program is finished, the lines and data can be displayed in the scene



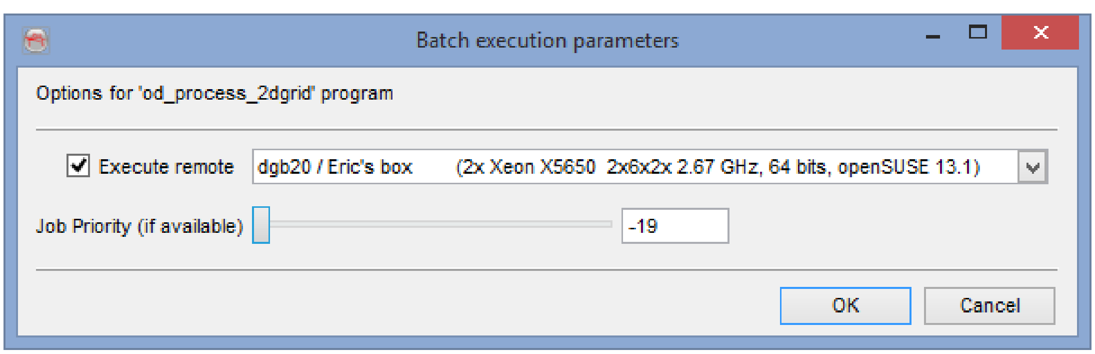
• **Input Cube:** 3D seismic data for conversion into 2D line-set.
• **Volume Subselection:** Restricts the extent of the 3D volume (optional). Choose inline/cross-line/time ranges for the selected volume.
• **Create Grid from:** Define the output grid geometry and orientation of the 2D lines. If *Inl/Crl* is selected, the inline/cross-line orientation (or geometry) of the selected 3D data will be used within the defined inline/cross-line/z-range and corresponding steps. Optionally, the inline/crossline range can be edited manually by using loosely spaced inline/cross-line numbers separated by commas.



Another option is to define the 2D grid geometry based on a *Random Line*. The grid can be created both parallel- and perpendicular to the direction of the random line. Set the fixed spacing in the *line spacing(m)* fields.

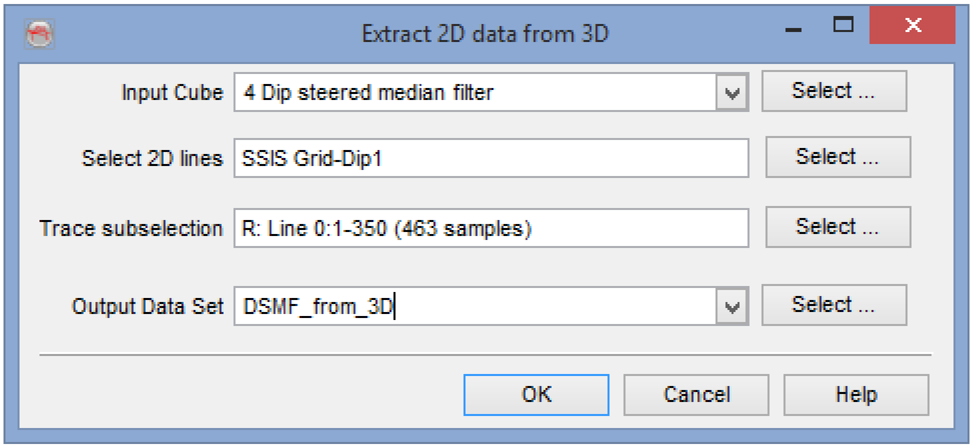


• **Prefix:** Label the 2D-line names.
• **Output Data Set:** Output for the 2D line-set. Provide a name for the line and the seismic data (attribute).
• **Extract Horizons:** Convert existing 3D horizons into 2D horizons by checking the box (optional). In the *Select horizons* list, select one or more horizons and, if desired, set a prefix for these selected horizons.
• **Execution Options:** Launches the 'Batch Execution Parameters' window and gives the option to execute the job remotely:



Extract 2D from 3D

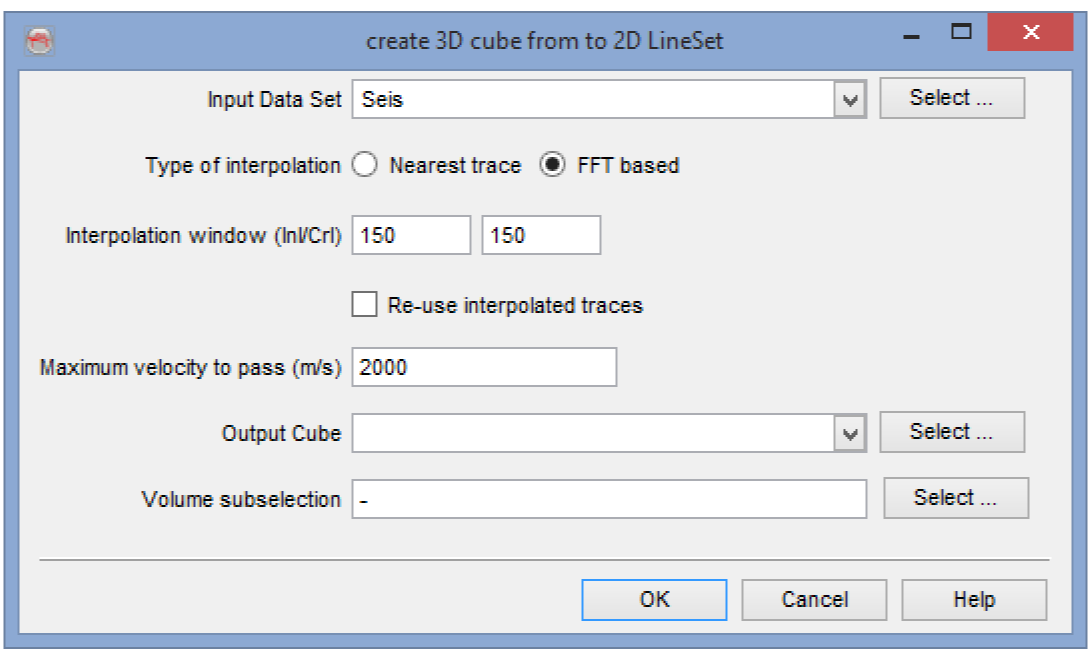
Extract 3D data onto selected 2D lines. Input data is required in the form of a stored 3D volume. One or more 2D lines can be selected for the 3D data to be extracted onto. The output data set requires naming:



Note: If just one line is selected, you may also sub-select a trace range.

Create 3D from 2D

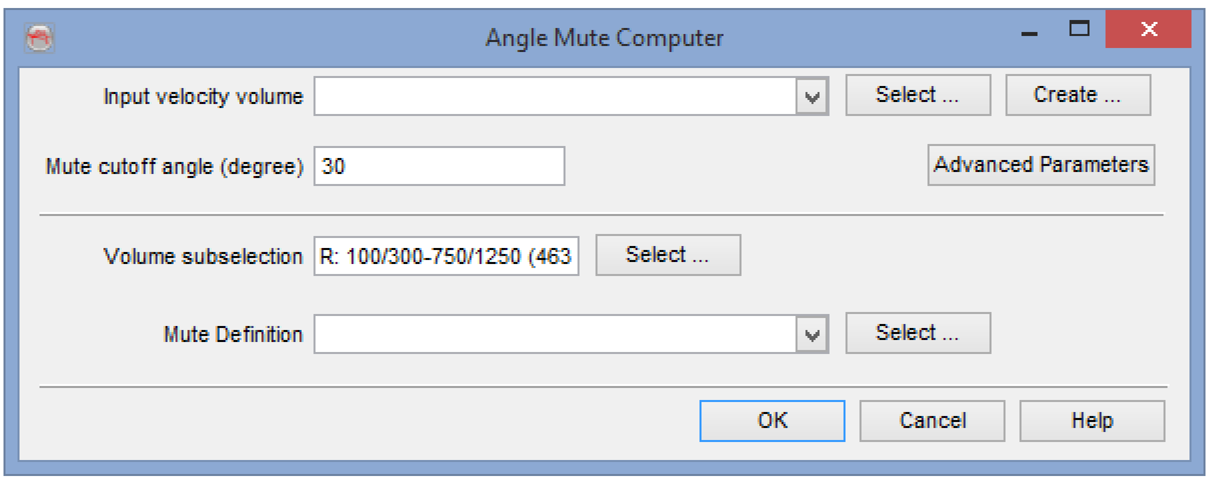
The *Create 3D from 2D* option creates 3D volumes from 2D line-sets.



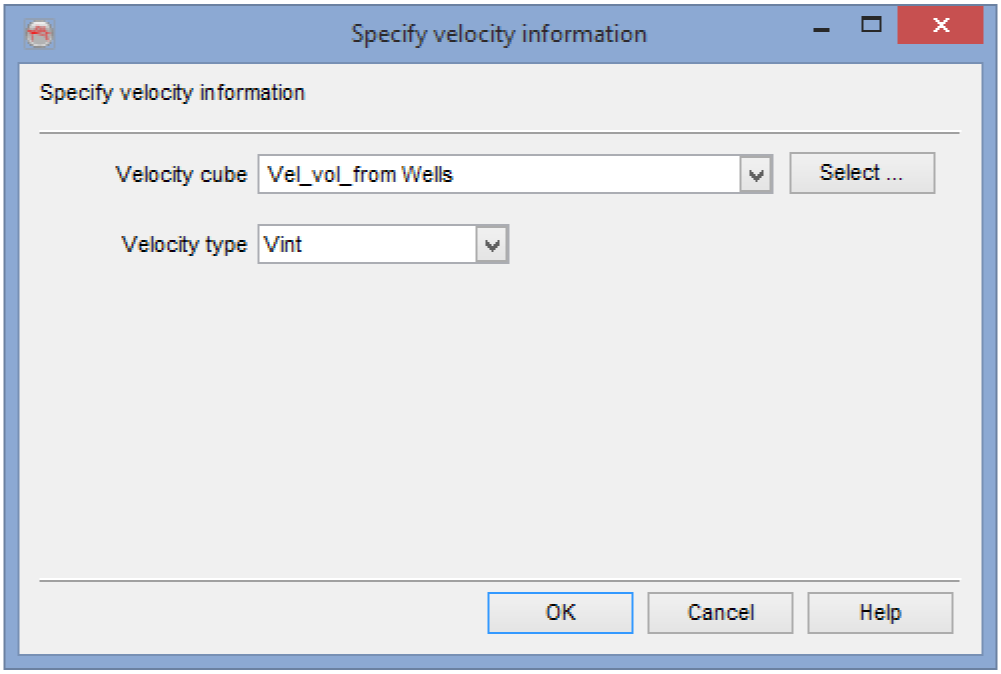
• **Input Data Set/Attribute:** Select 2D data set and data (attribute) for conversion into 3D.
• **Type of Interpolation:** Nearest trace, as the name implies uses the nearest trace for the interpolation. The FFT based interpolation is a 3D Fourier transform of Inlines and cross-lines set in the *Interpolation window*. Optionally, check the *Re-use interpolated traces*; this option will use interpolated values for the next iterations. Finally, set the *Maximum velocity to pass (m/s)*.
• **Output Cube:** Name for new 3D volume.
• **Volume Subselection:** Optionally, set the sub-selection using inlines/cross-lines and z-ranges.

Angle Mute Function

This module creates angle-based Z-Offset functions from velocity volumes. The primary input is a velocity model that provides the time-depth relation.



If you have no velocity volume available, click '*Create*'. This will bring up a window in which you can select a volume and tag it with a velocity type:



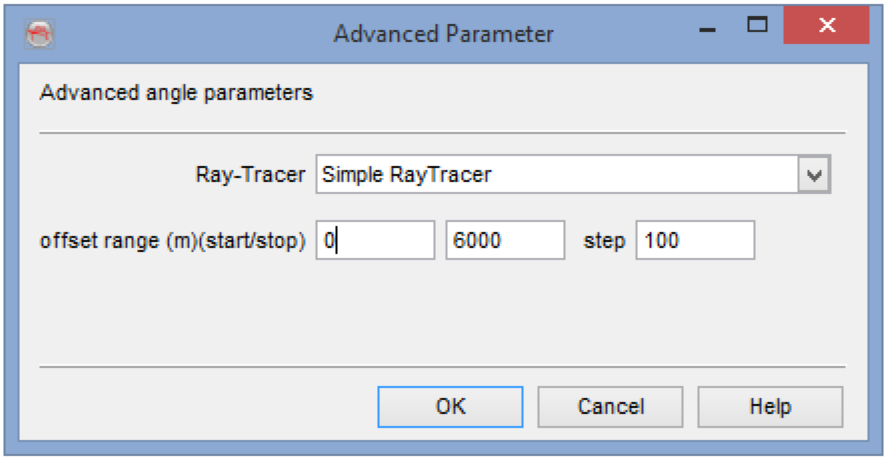
Vertical 1D ray-tracing is performed assuming a fully flat, isotropic earth model. The travel-time 0ms corresponds to the depth of the Seismic Reference Datum, defined in the survey definition window.
The offset range must be provided since the angle mute computer is not aware of any prestack datastore. It does not necessarily need to match the prestack data. The output function will have one point at the start and stop of the Z range, and one point at each offset specified by the offset range parameters.

The main output parameter is the incidence angle in degrees at which the mute function must be computed. By default the functions will be computed on a relaxed grid, every 20th time the inline/crossline stepout. This can be changed by selecting other "Volume subselection" parameters. In general it is not necessary to decrease that stepout.

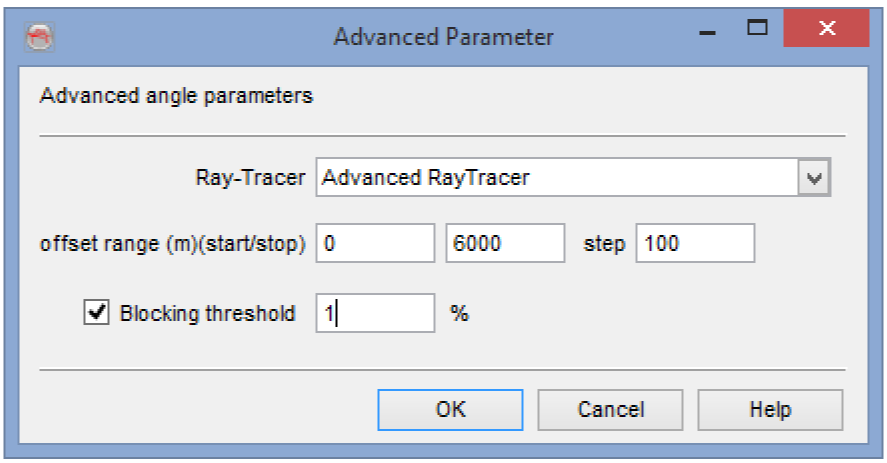
Advanced Angle Mute Paramaters

The ray-tracing can be performed in two ways:

• **Simple:** The ray is going directly from the source to the depth of the target layer, and up to the receiver in the same way. This does not account for ray bending, or velocity inversions.



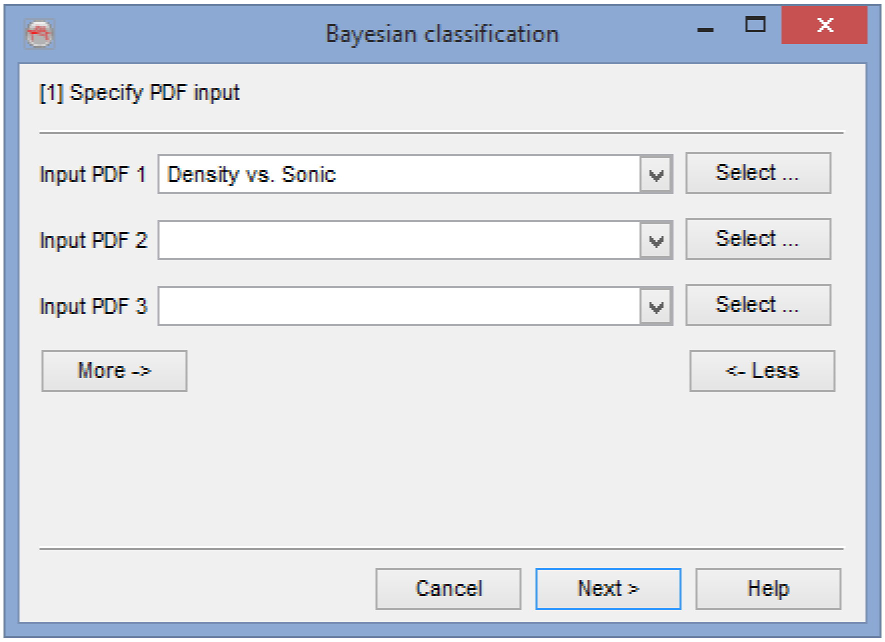
• **Advanced:** Will honour the ray bending according to Snell's law and thus velocity inversions as well. To reduce the processing time, the layers may be blocked: Consecutive layers with similar Vp, (and density, Vs if present) values are concatenated together. The ray is propagated in a straight line inside a concatenated layer.



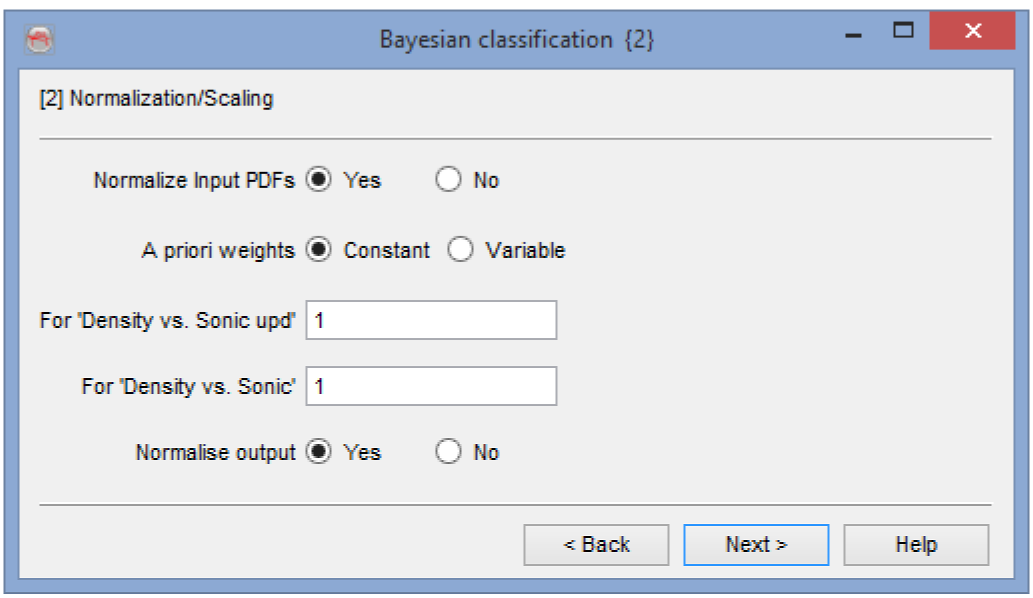
Bayesian Classification

Bayesian classifications are used to link several attributes based on one or several Probability Density Functions.

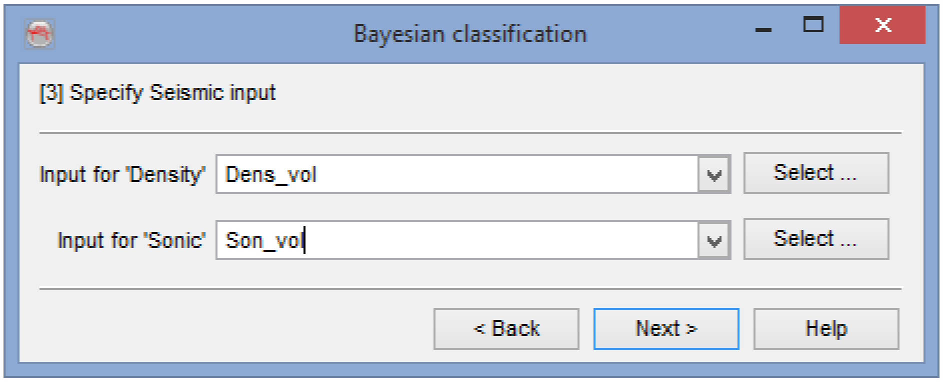
First, one or several PDF(s) need to be provided. *More* will allow to select more PDF.



After clicking on *Next* >>, the PDF(s) can then optionally be normalized based on a priori weights per PDF. The a priori weights can be provided by attribute volumes which will vary at every sample location.

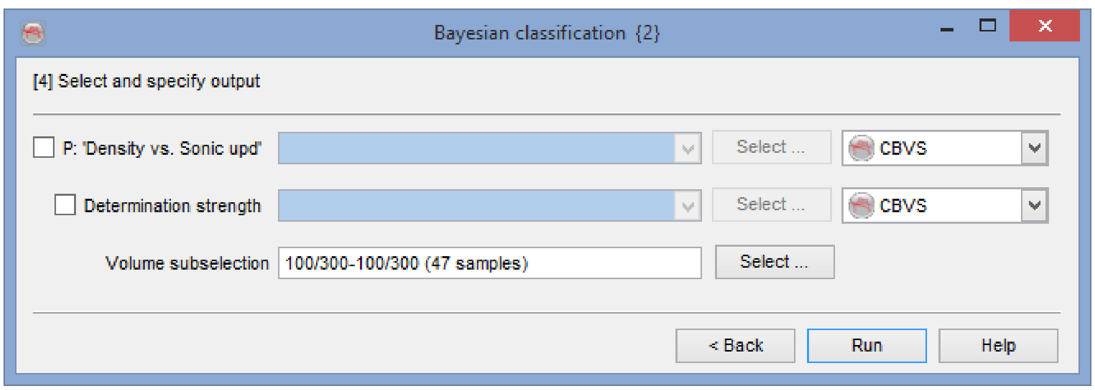


The processing will be based on (inverted) stored volumes that should correspond to the variables used for generating the PDF. Please note that OpendTect cannot make this check.



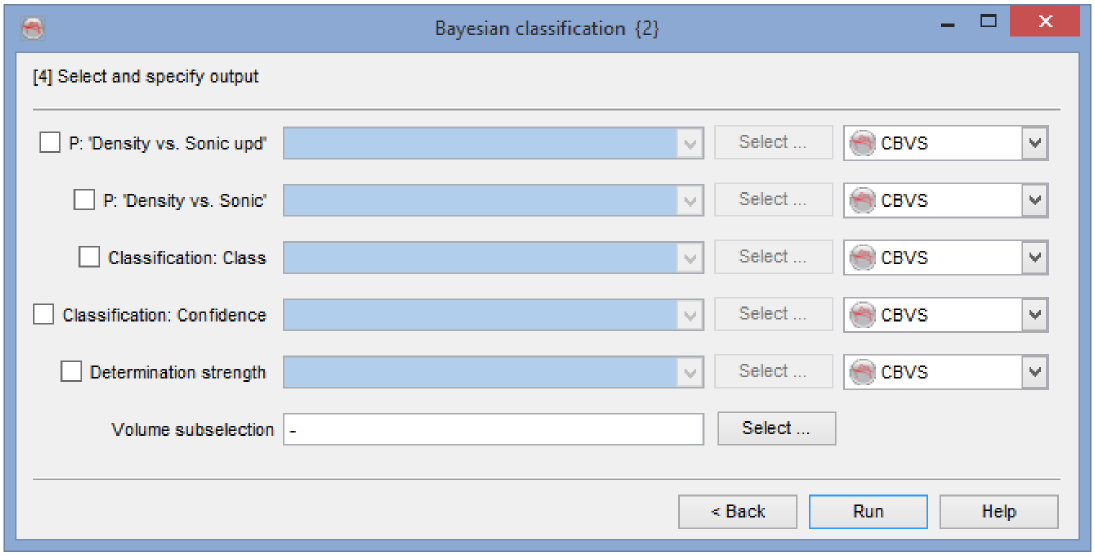
If only one PDF was used as input, the Bayesian classification volume will generate two volumes:

1. A probability volume "P:PDF name"
2. A determination strength volume, mainly for QC.



If more than one PDF were used as input, the Bayesian classification volume will generate:

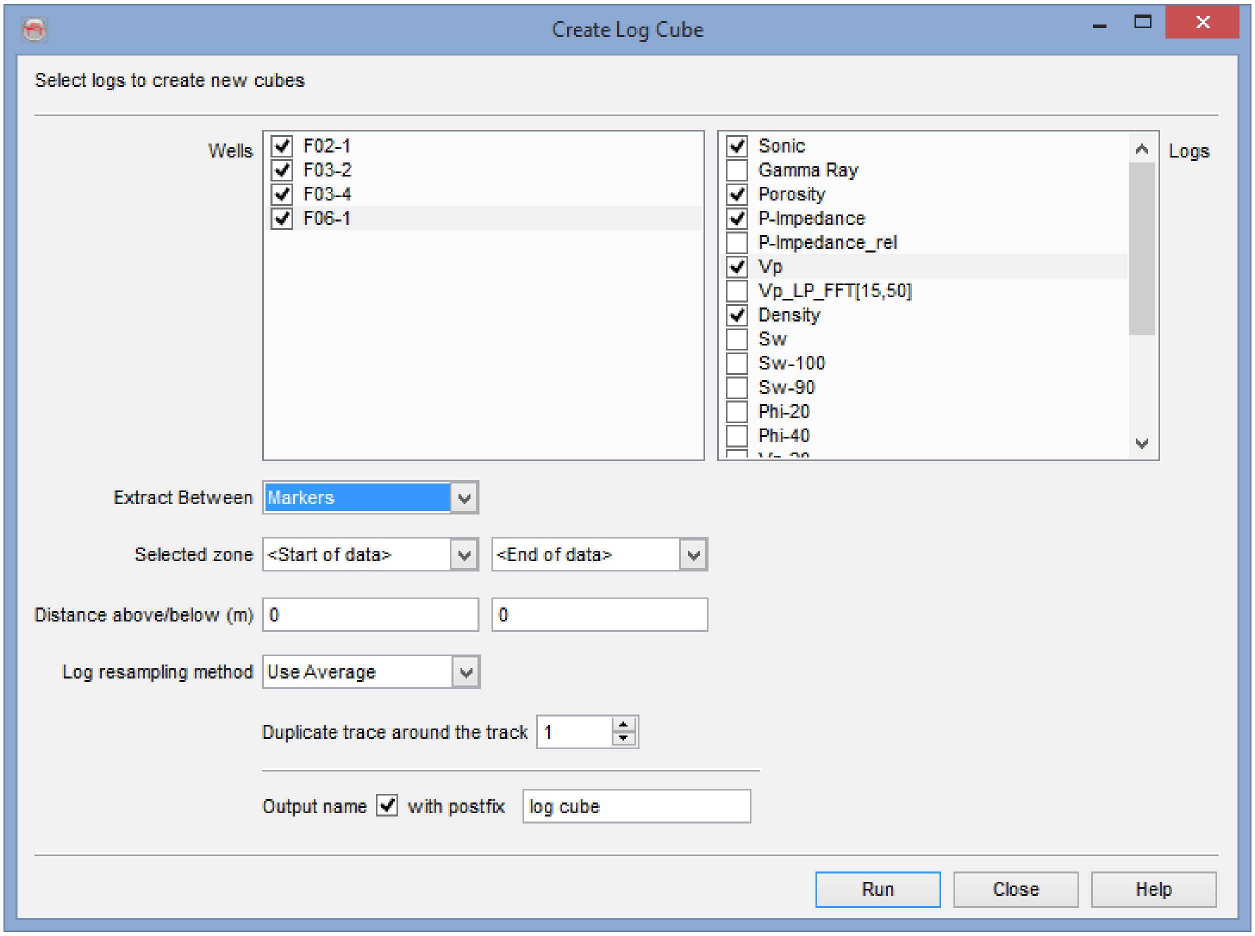
1. A probability volume "P:PDF1 name"
2. A probability volume "P:PDF2 name" (one for each input PDF)
3. Two classification volumes : Class (with the most likely class) and Confidence (the difference between the two most likely)
4. A determination strength volume, mainly for QC.



Create from Well Logs

The *Create seismic output from wells* option writes loaded logs as seismic volumes.

The dialog is accessed from the *Processing* menu in the main toolbar; choose the *Create from Wells*.. option.



Select the well(s) in the left and the log(s) in the right list, note that several wells and logs can be selected at once.

The log extraction can be done either between *markers* or by selecting a *depth range*. For the latter, select start/stop depths in meters. For extraction between markers select the wanted markers and add *distance above/below* (optional) for including intervals above and/or below the marker depths.

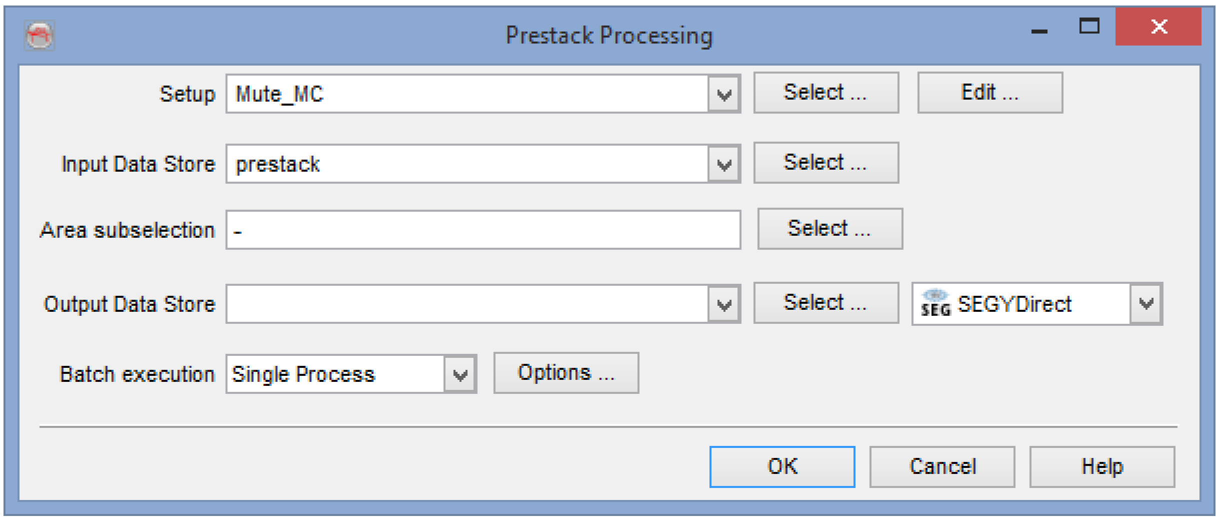
The logs need to be resampled in order to display correctly on seismic sections and volumes. Choose a resampling method from the dropdown list:

- Take Nearest Sample
- Use Average
- Use Median
- Use RMS
- Use Most Frequent

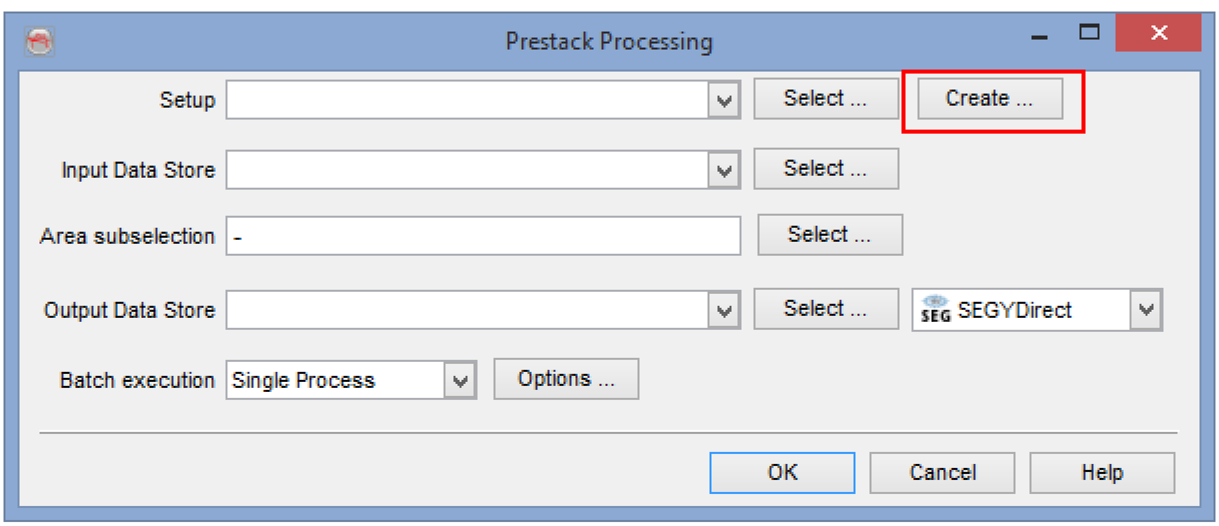
Next, choose how many traces should be duplicated around the well track. Essentially this will determine the dimension and geometry of the output cube. Finally give a name (suffix) to the CBVS volume (seismic volume), the volume itself will automatically be named according to logs selected in the list.

Pre-Stack Processing

Prestack processing can be applied using different methods. It is the only place where pre-stack data can be output in OpendTect based on another pre-stack datastore.
Open the Pre Stack processing window: *Processing > Create Seismic Output > Pre-Stack Processing*

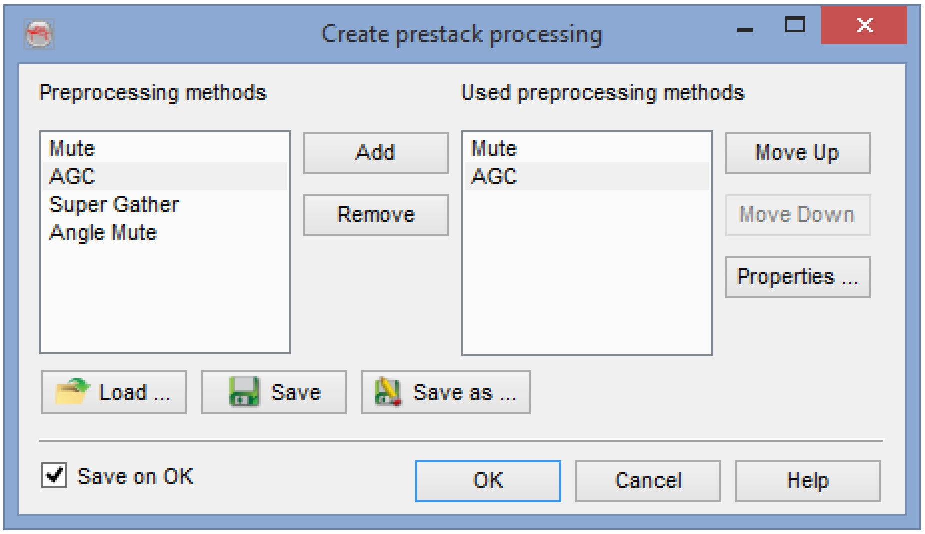


The processing can be done by a number of sequential steps. Either select a pre-defined set up (as above, which may be further '*Edit*-ed') or press '*Create*'...



Which brings you to the following window...

To use a preprocessing step, select it in the *preprocessing methods* and click on Add. It will then be listed in the used *preprocessing methods*, click on *Properties* to define the step parameters. Move up or Move down to change the preprocessing steps order for the processing.

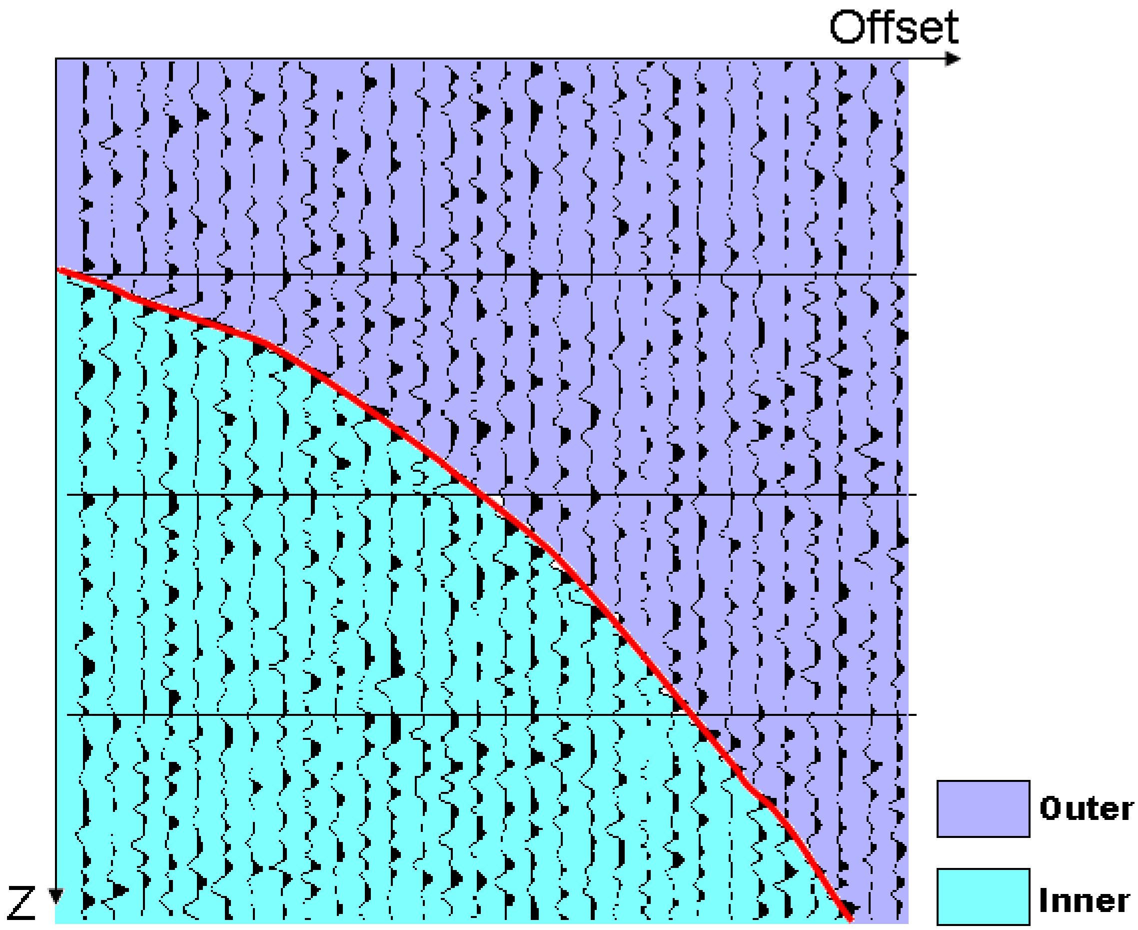


The following sections describe the available different steps.

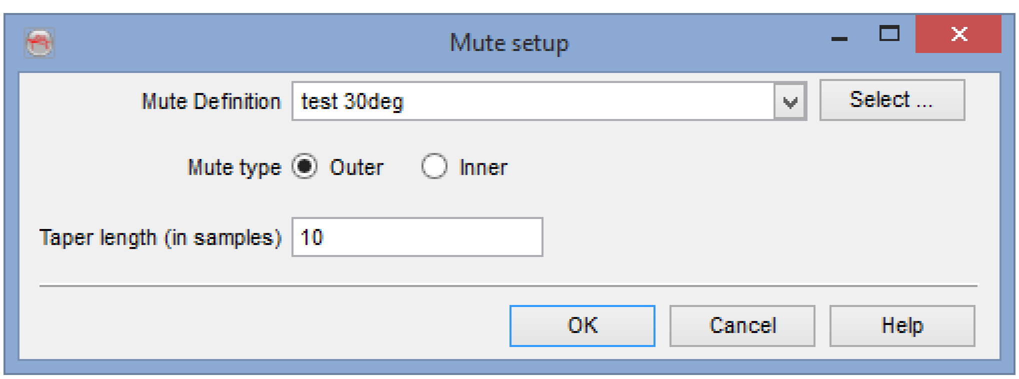
Mute

Mute functions may be applied to *Prestack gathers*. This window will allow you to choose the mute definition, as well as to specify settings such as:

- **Mute type:** Outter (top) or Inner (tail)

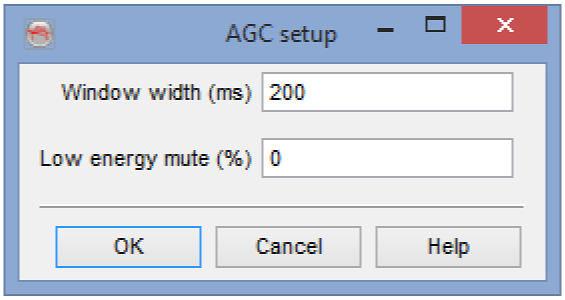


- **Taper length (in samples)**



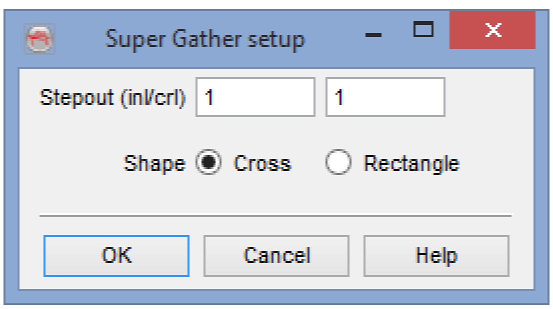
Automatic Gain Control

Automatic Gain Control (AGC) is one of the processing methods available for *Prestack gathers*. It will adjust the amplitude level using a sliding window of user-defined size (window width). Optionally, part of the lowest energy may be discarded from the amplitude level computation (in percent of the amplitude distribution).



Super Gather

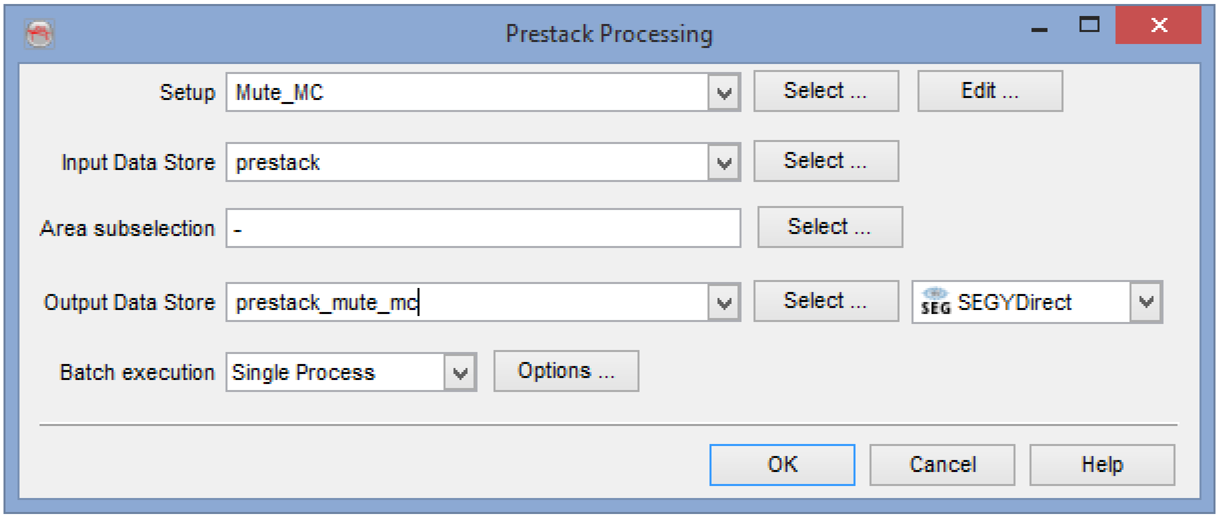
A *Super Gather* may be used to laterally stack the traces in order to increase signal-to-noise ratio of *Pre-stack gathers*. The stack is controlled by an inline/crossline stepout, and the Shape (Cross or Rectangle). The computation is similar to a (non-steered) volume statistics attribute with a zero time-gate.



Angle Mute

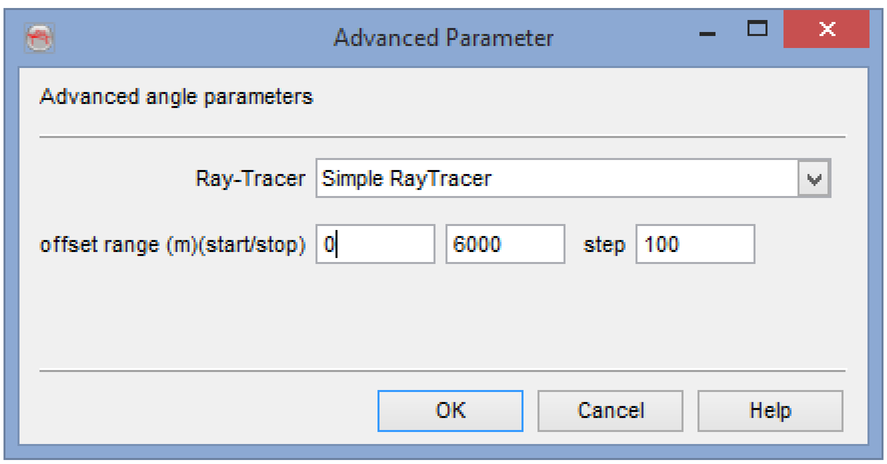
This processing method computes and applies a mute function. See the processing method documentation. The only difference is that this method reads the offset range from the input prestack datastore, such that there is need to specify it. See here where and when the input velocity source needs to be edited.

The application of the computed mute function is strictly identical to the application of a stored mute function.

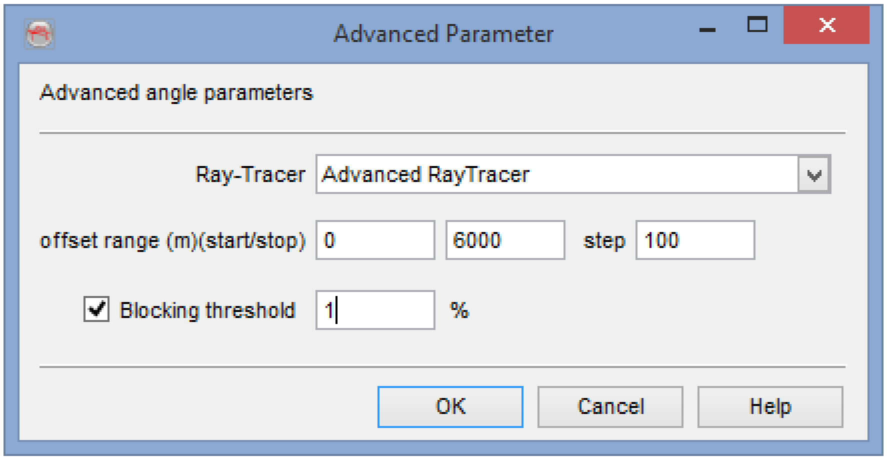


The ray-tracing can be performed in two ways:

• **Simple:** The ray is going directly from the source to the depth of the target layer, and up to the receiver in the same way. This does not account for ray bending, or velocity inversions.

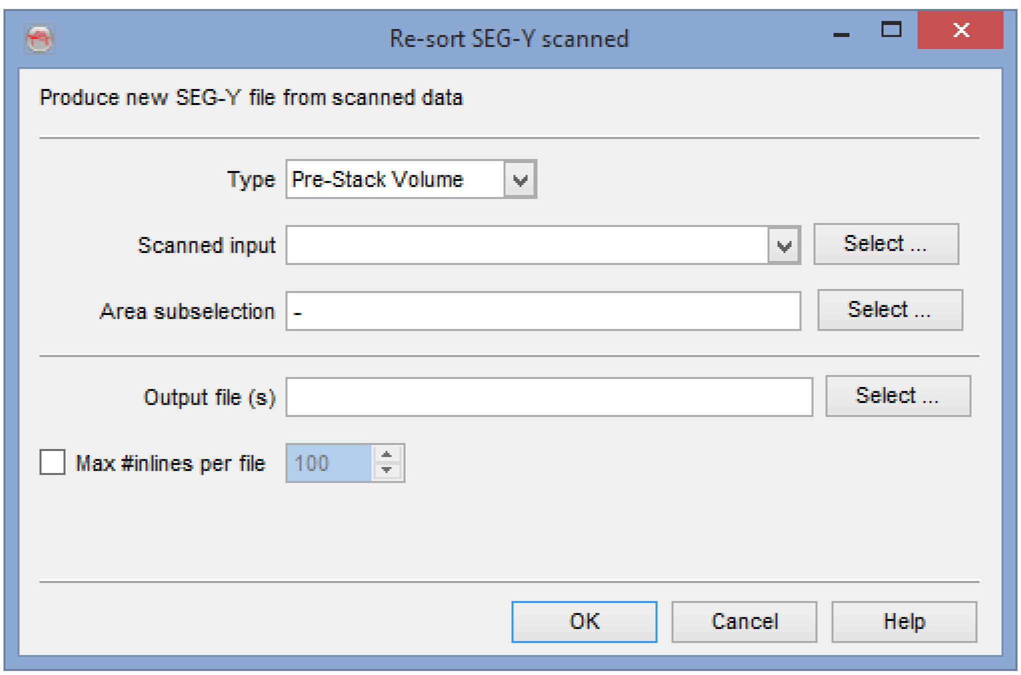


• **Advanced (not in the GPL version):** Will honour the ray bending according to Snell's law and thus velocity inversions as well. To reduce the processing time, the layers may be blocked: Consecutive layers with similar Vp, (and density, Vs if present) values are concatenated together. The ray is propagated in a straight line inside a concatenated layer.



SEG-Y Scanned Re-Sort

The *SEG-Y Scanned re-sort* uses a scanned SEG-Y file and outputs it as a new file and re-writes the file-header. This tool is useful in case information in the header is poor or poorly sorted.



In the *Type* field select the type of volume, either *Pre-Stack* or *3D volume*.

Next, select the *scanned input file* and optionally *area sub-selection*. Note that a SEG-Y file must be scanned prior to resorting. Choose a name for the *output file* and (optionally) restrict the number of inlines to be written per file. In case the latter option is used, multiple files will be written to disk, either using sequential numbers or the inline ranges included in the separate files.

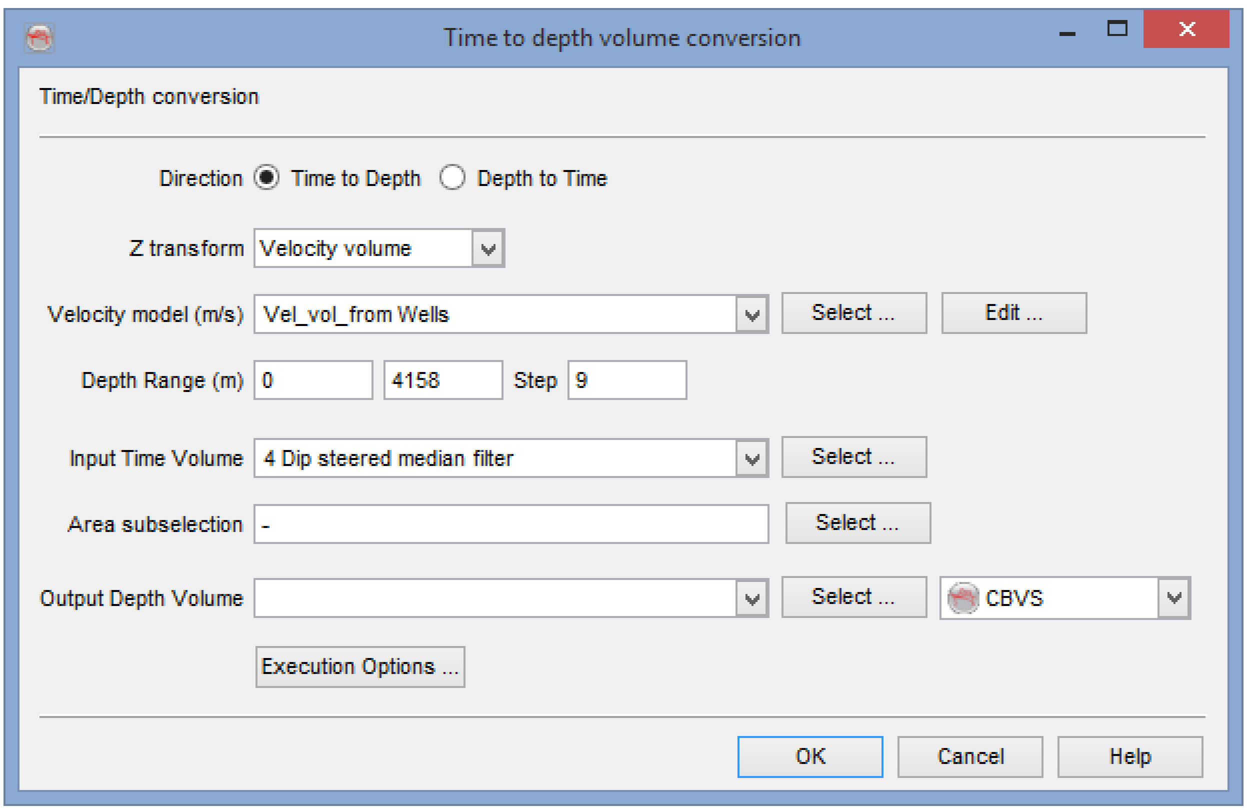
Velocity

Under 'velocity' sit two velocity-based conversion options, Velocity Conversion and Time-Depth Conversion.

Time-Depth Conversion

To create an time-depth converted output, follow: *Processing* > *Create Seismic Output* > *Time-Depth Conversion...*

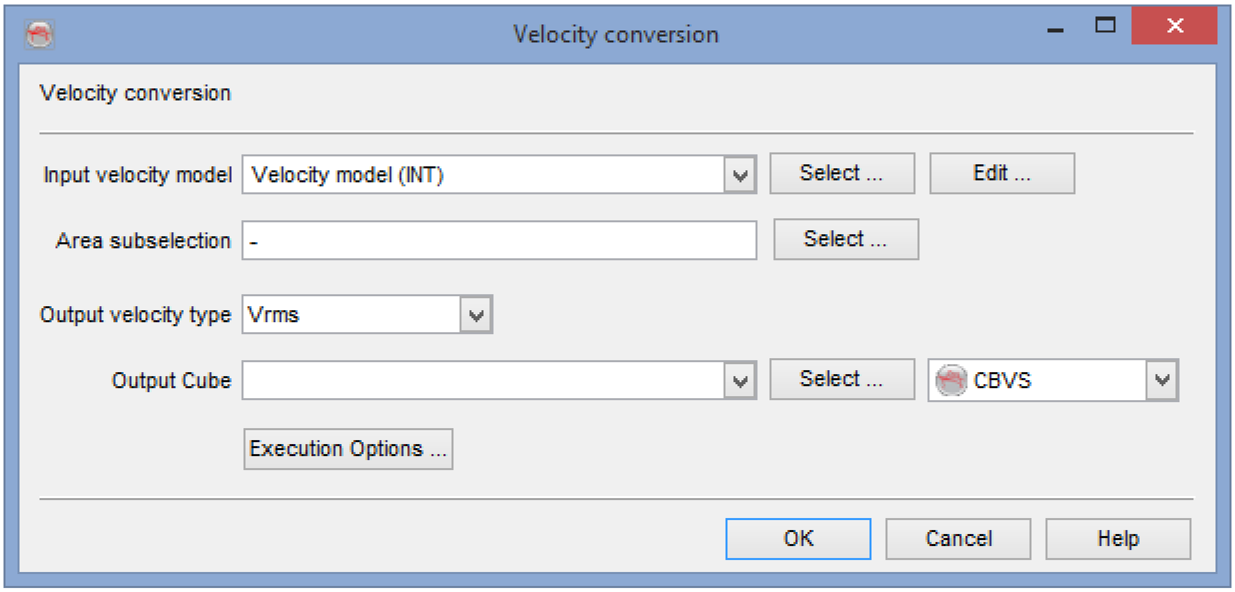
Time-depth conversion is done by applying Dix's equation, based on a interval- or RMS velocity volume. A velocity model and input time volume must be provided, and the direction of the conversion has to be set.



It is also possible to convert from Depth to Time. Instead of an input Time volume, a input Depth model has to be provided.

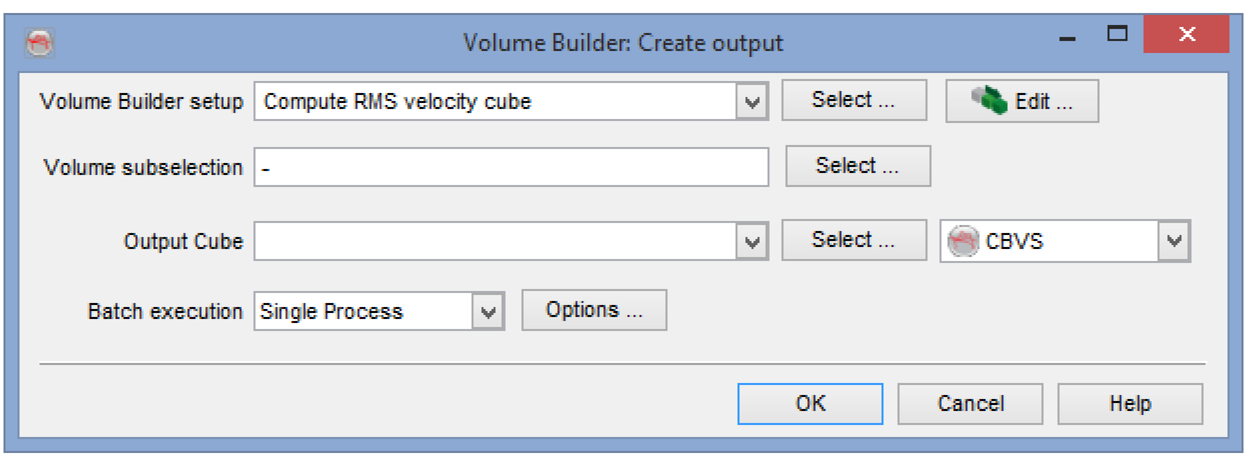
Velocity Conversion

This tool is started from *Processing* > *Create Seismic Output* > *Velocity conversion*. It can be used to convert interval velocity volumes to RMS velocity volumes and vice versa. The conversion is applied using Dix's formula. Please note that for this reason it can only be applied in the time domain.



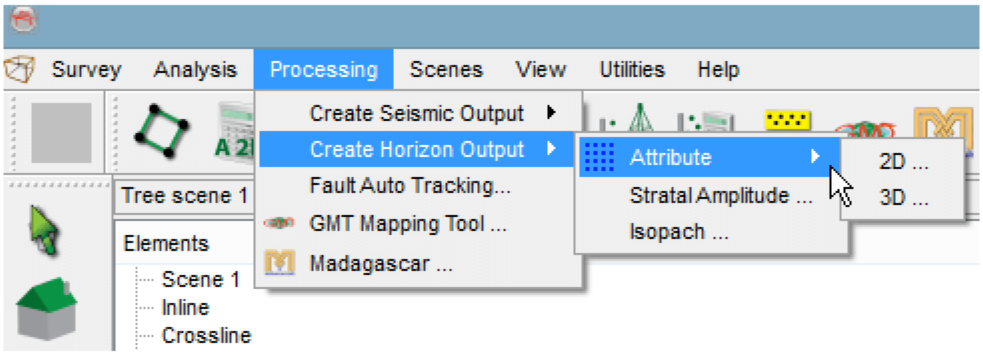
Volume Builder Output

Volume-builder creator window can be launched from *Processing* > *Create Seismic Output* > *Volume Builder.* It is used to create the output volume that has been defined in a volume builder setup. Optionally, if the initial volume builder setup is not defined, press the *Edit* button to define the setup. Press '*Ok*' to launch a batch processing window.



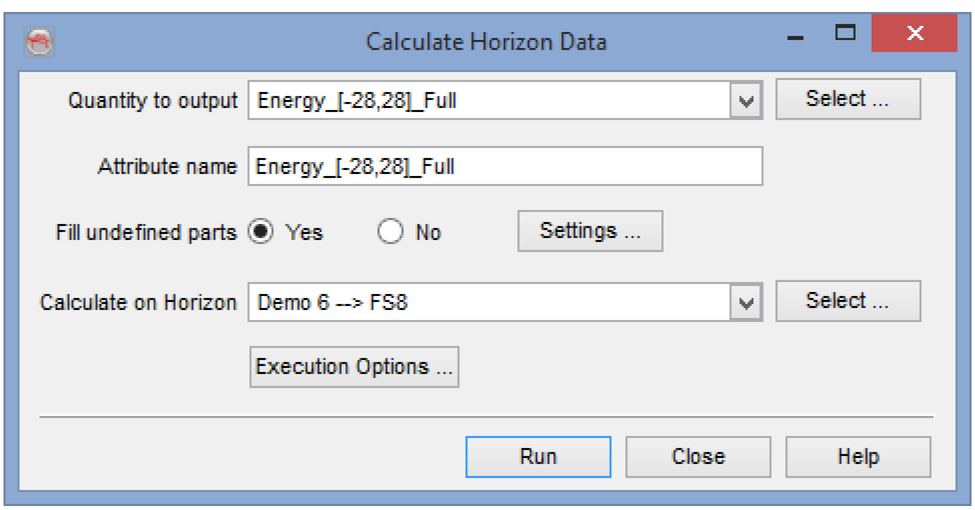
Create Horizon Output

This menu is used to create 2D/3D-grid based output. The data is stored as horizon data (or attribute) to the selected horizon.



Attribute

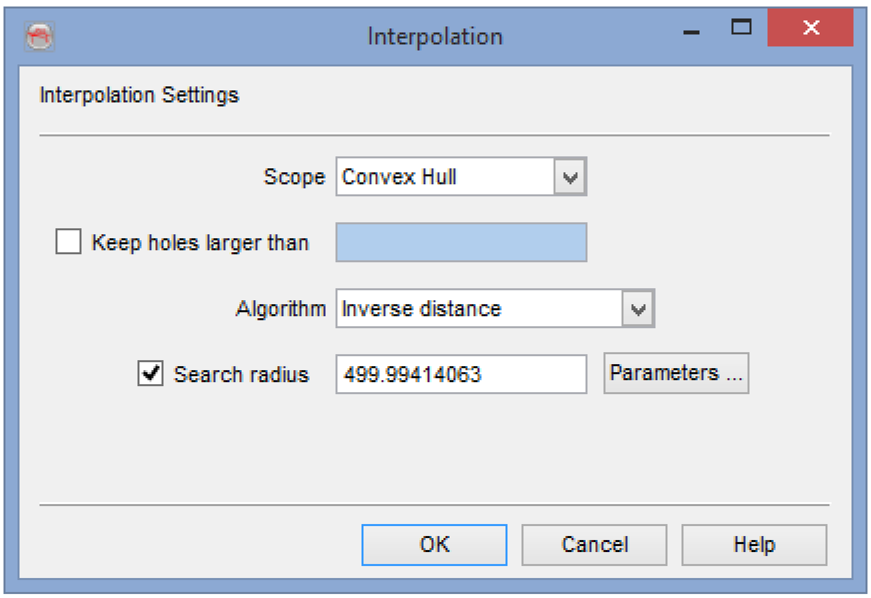
Some attributes consume significant calculation time e.g. curvature, spectral decomposition etc. It also depends on the size of the input seismic volume. Therefore, to create on-the-fly a horizon data in a scene may take significant time. Following *Processing > Create Horizon output > Attribute (2D or 3D)*, the attribute calculation is there processed in the background (batch processing) and thus other tasks can be done at the same time. By using a Horizon output, horizon attributes can be created at desired horizons independently.



Select the quantity to output (2D/3D) from the list of stored data or attributes. The attribute is (by default) saved with its own name, but it can be edited. Select the horizon on which the selected attribute will be calculated.

The parameter file, which is automatically created in the Store Processing Specification field, can have any name (a default name is provided) so that the calculation process can be easily re-started if needed. The batch processing can be achieved using a single or multiple machines. More information on this window is provided in single machine batch processing section. After the batch processing is finished, the result will be available as a horizon data: right-click on a blank horizon's attribute in the tree to select the horizon data (Select Attribute > Horizon Data).

If the option for 'Fill undefined parts' is toggled on, then the '*Settings*' button can be used to enter the interpolation settings:

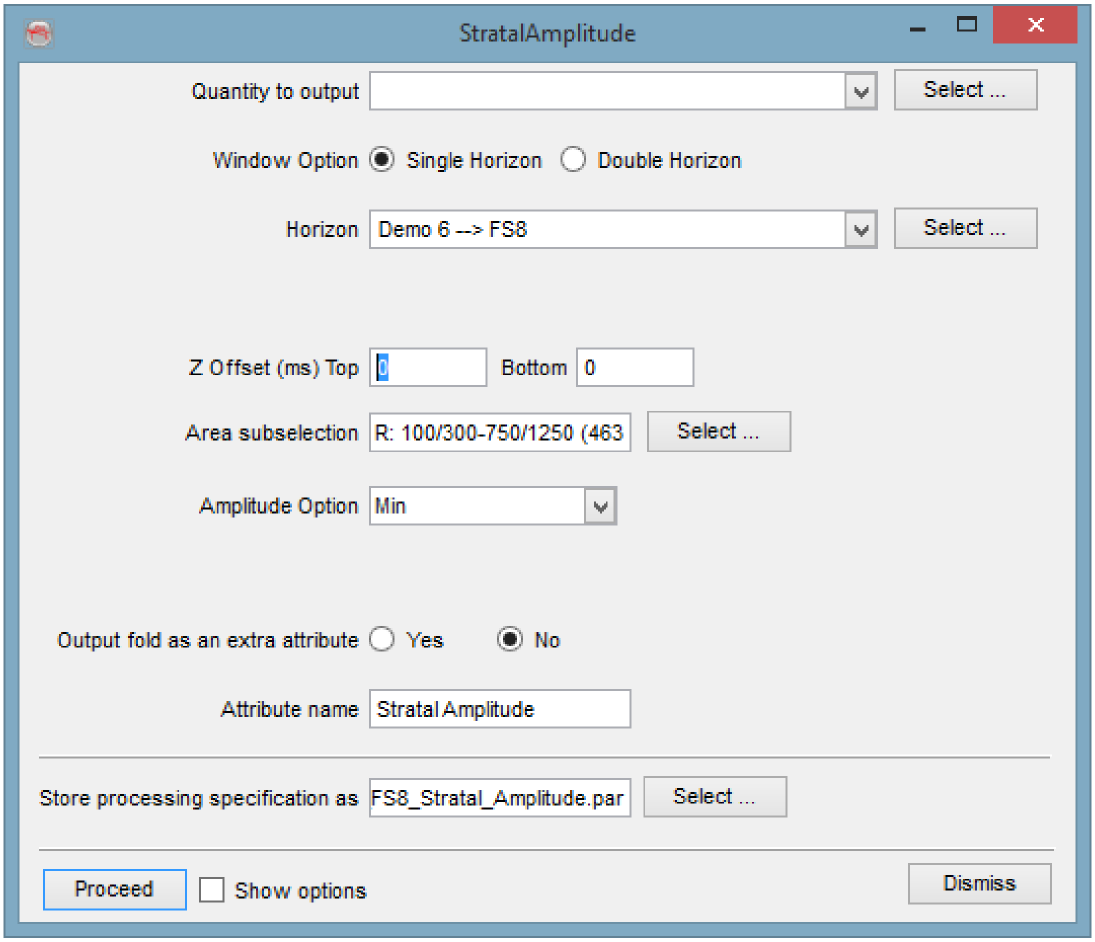


For the '*Execution Options*', please refer to the following topic: [Batch Execution Parameters]

Stratal Amplitude

Stratal Amplitude is a processing tool available to compute statistics (min, max, rms, etc.) from an attribute along a horizon or between two horizons. The window can be launched from *Processing* > *Create Horizon output* > *Stratal Amplitude*. The output will be stored as horizon data (grid) saved on the top or base horizon.

Note: This feature operates based on a single-trace calculation only (ie: no step-out). For multi-trace calculations (using step-out), you are advised to use the Volume Statistics attribute.

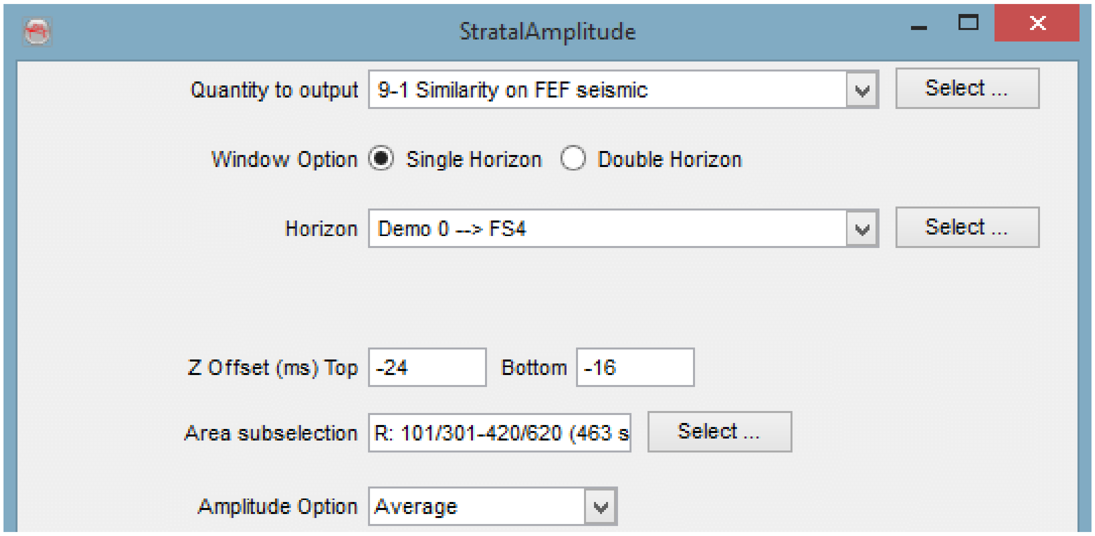


In this window, select the input attribute from which the values will be extracted. The extraction may be guided by a single horizon (*Single Horizon*) or between two horizons (*Double Horizon*).

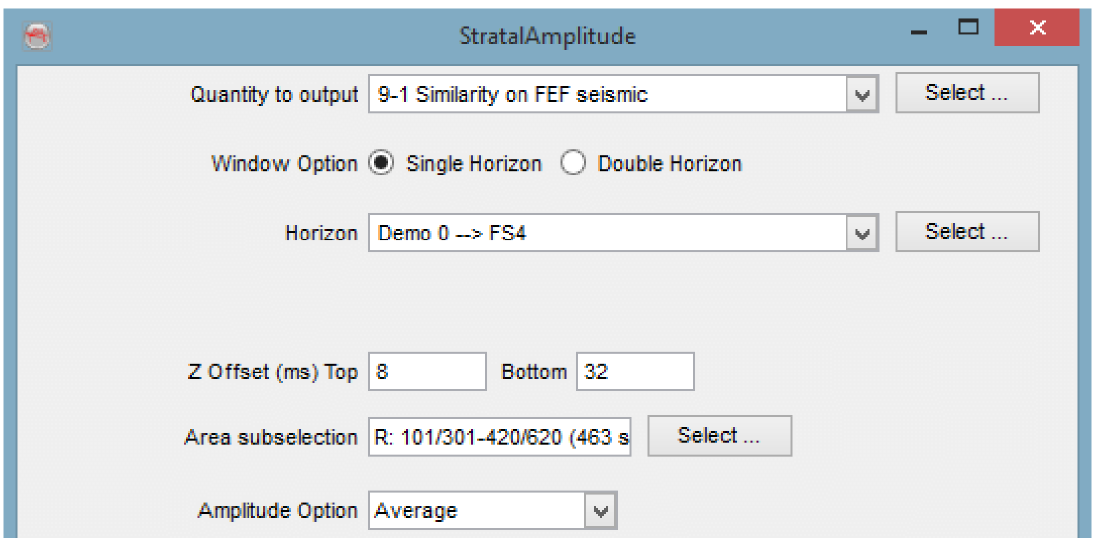
Single Horizon is used to extract amplitude along a horizon within fixed window relative to the pick.

Double Horizon is an option of amplitude extraction between two horizons. In this case, the Z-offset parameters (see below) may be defined to increase or decrease the area defined by the horizons.

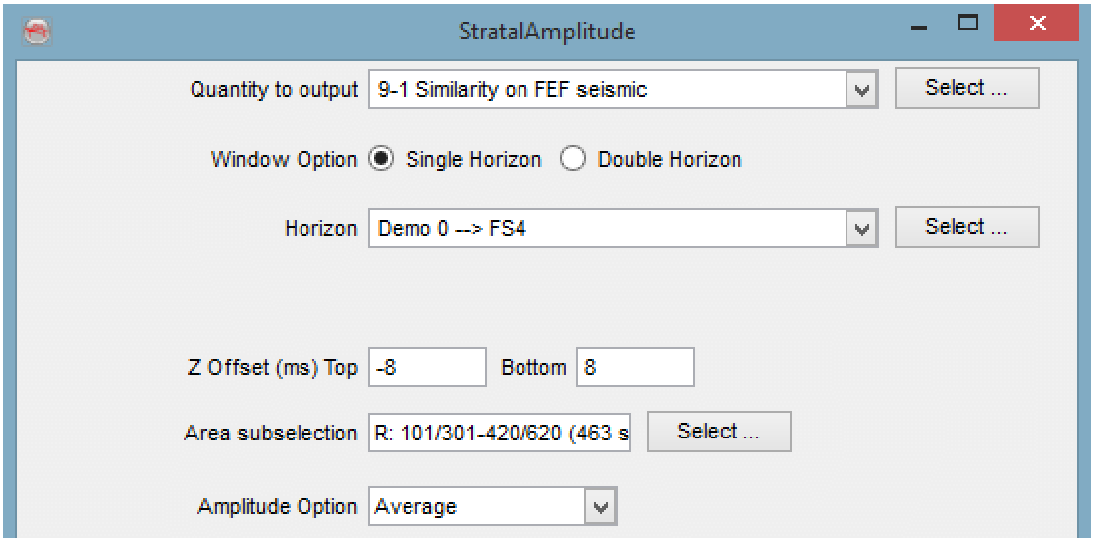
Z-offset is the offset window specification above (negative values) or below (positive values) a horizon to restrict or extend the calculation interval. For example:



*Settings to extract the average amplitude between 16 and 24 ms **above** ('-' values) the selected horizon.*



*Settings to extract the average amplitude between 8 and 32 ms **below** (for positive values there is no need to prefix with a '+' sign) the selected horizon.*



*Settings to extract the average amplitude 8ms equally **around** the selected horizon.*

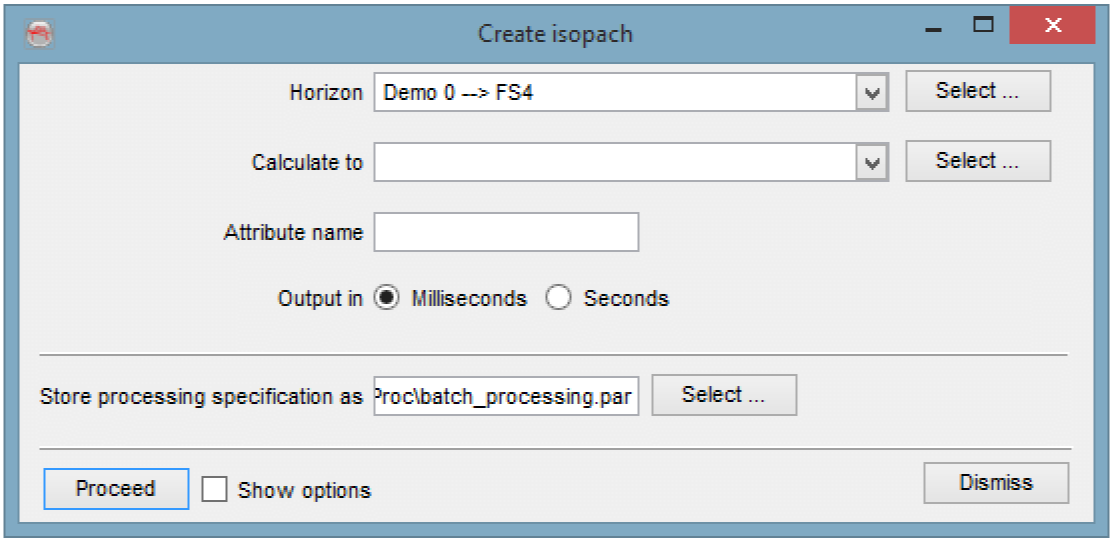
Area subselection is used to specify the area within which the attribute is output.

Amplitude options are the available statistics for amplitude extraction. Five amplitude statistics are available: Min, Max, Average or RMS and Sum.

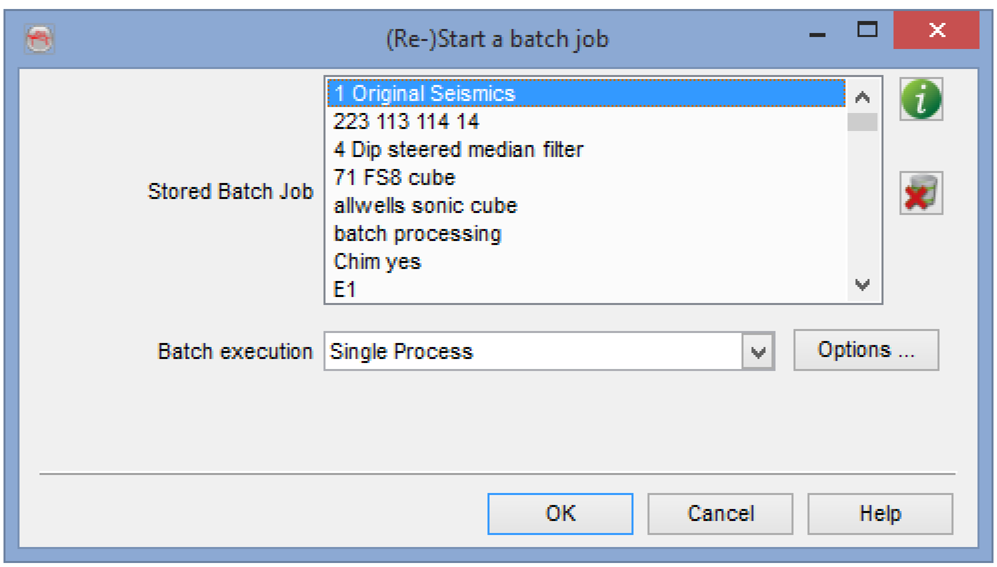
Output fold as an extra attribute optionally outputs data fold, i.e the number of point used for the processing, as separate horizon data.

Isopach

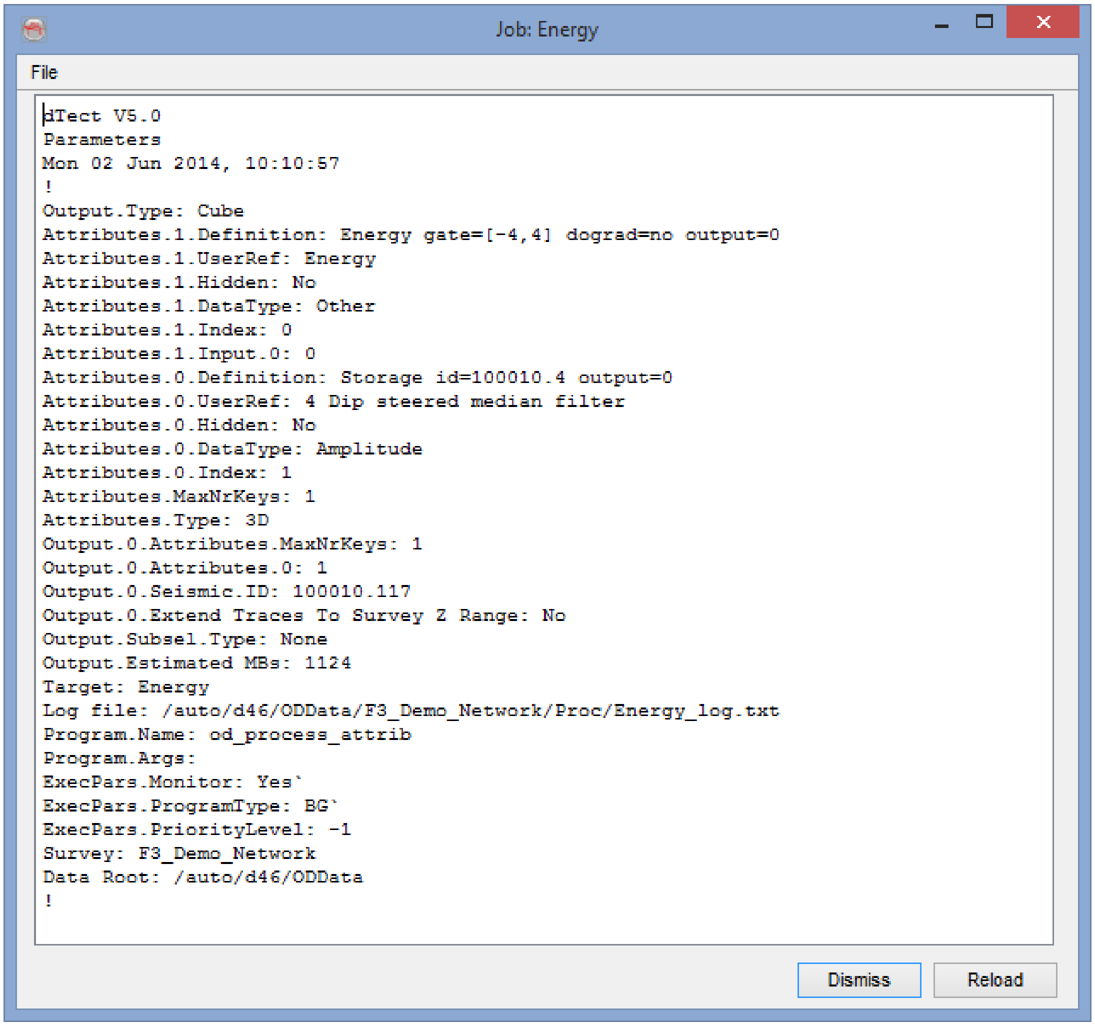
In OpendTect isopach maps can be quickly calculated. The *Create isopach* window is launched either from the *Processing* > *Create Horizon Output* > *Isopach* or from the right click menu of any horizon loaded in the tree: *Workflows* > *Create Isopach*. In this window, select two horizons between which the isopach has to be computed. The isopach map will be saved as a horizon data of the first selected horizon in the window or the horizon .



(Re-)Start Batch Job

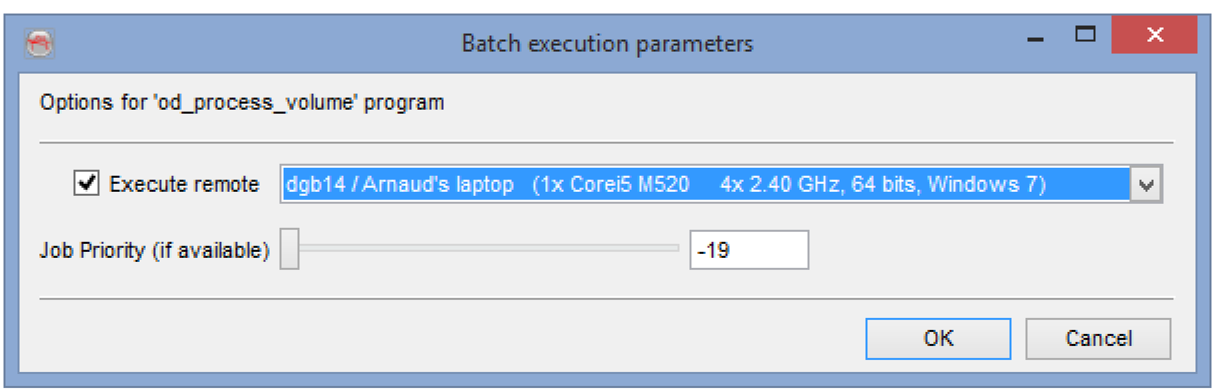


Batch jobs in OpendTect are stored under a job name in a file containing the inputs, parameters, log file and other relevant information. This information can be read by clicking on the '*Information*' icon, :

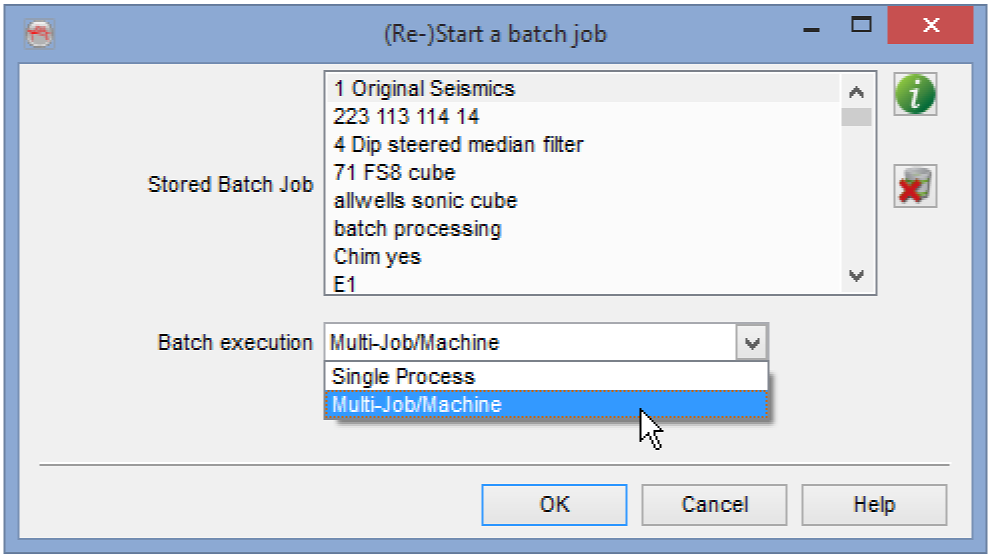


Jobs may also be removed using the  icon.

Batch Execution: The batch job may be executed as a Single Process or as a Multi-Job/Machine Process. When opting for '*Single Process*', the user may use his local machine or choose to send the whole job to a remote machine for processing:



'Execute remote' toggled on to send the job to a remote machine. The job priority can be changed (-19-lowest to +19 highest).



Multi-Job/Machine option. Selecting this options brings up the Multi-Machine Processing window.

> **Note:** If a job is selected that was created in OpendTect prior to the 5.0 upgrade, a warning will pop-up, stating "Pre 5.0 Job". These jobs can not be (re-) processed. Attempting to do so will bring up the Error message: "Can not run selected job".

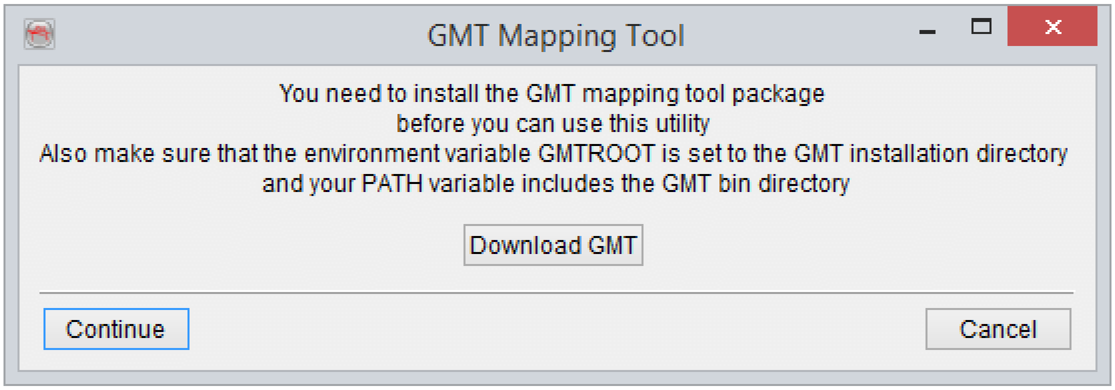
GMT

GMT (Generic Mapping Tools) is an open source collection of more than 60 tools for manipulating Geographic and Cartesian data sets. It can produce *Encapsulated Post-script File* illustrations ranging from simple x-y plots, via contour maps, to artificially illu-minated surfaces and 3D perspectives views. *OpendTect* supports an open source plugin that uses GMT tools to create scaled maps.

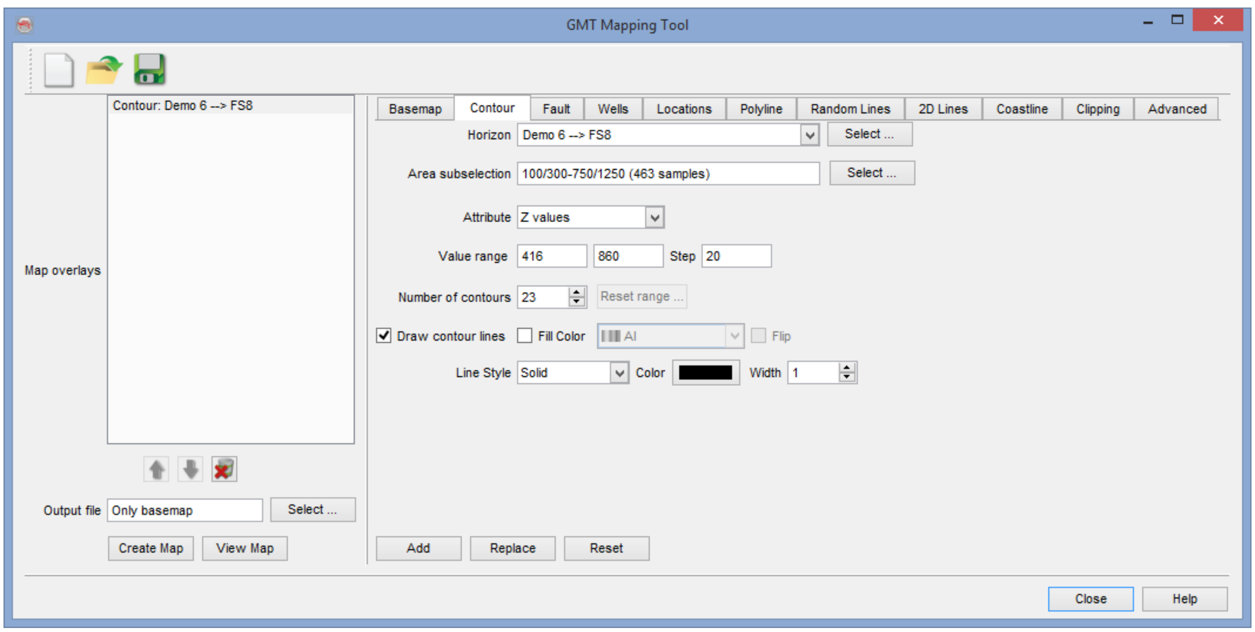
For more details please visit the official GMT website.

Initial Setup

To launch *GMT* tools, click on the icon in the OpendTect main toolbar. The first time you launch the *GMT* mapping tools, a warning message will pop up, if *GMT* is not already installed on your computer. This can be downloaded from the GMT website.



After successful installation of the package, the *GMT* user interface will be launched:



GMT User Interface

Create Postscript Maps

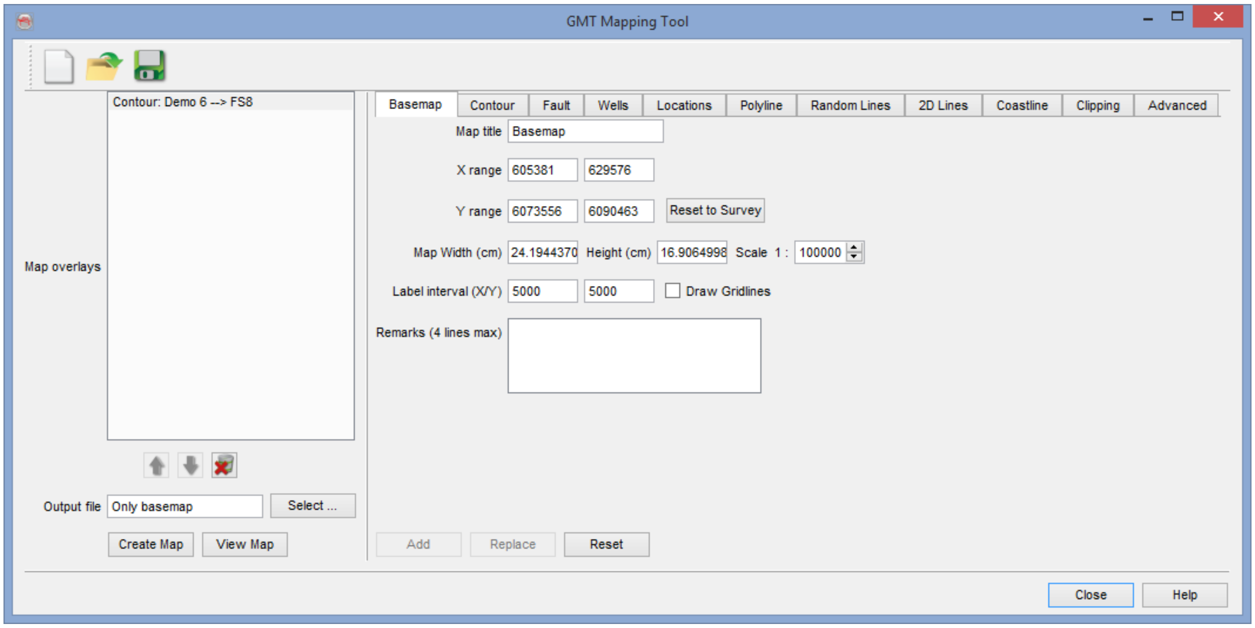
Several tabs have been arranged to specify the respective settings. The later part of this section shows a typical OpendTect example of a postscript map.

- Basemap: This tab is used to set the scale of the map and other map settings.
- Contours: It is used to create a horizon contour map.
- Faults: It is used to post the intersection of faults with constant times or the intersection with a surface.
- Wells: It is used to post wells in the map.
- Locations: It is used to post pickset data in the map overlay.
- Polyline: It is used to add polygons (e.g. lease boundaries) in the map overlay.
- Random Lines: It is used to post Random Line(s) in the map.
- 2D Lines: It is used to post 2D-Line(s) in the map.
- Coastline: It is generally used to draw coastal lines.
- Clipping: It is used to set up polygonal clip paths.
- Advanced: It is used to use customized GMT commands.

For all the sections it is possible to *Reset* the parameters and thus go back to the default ones. For all the section (except Basemap), *Add* will add the defined object to the map overlays and *Replace* will update it if the object has been previously defined.

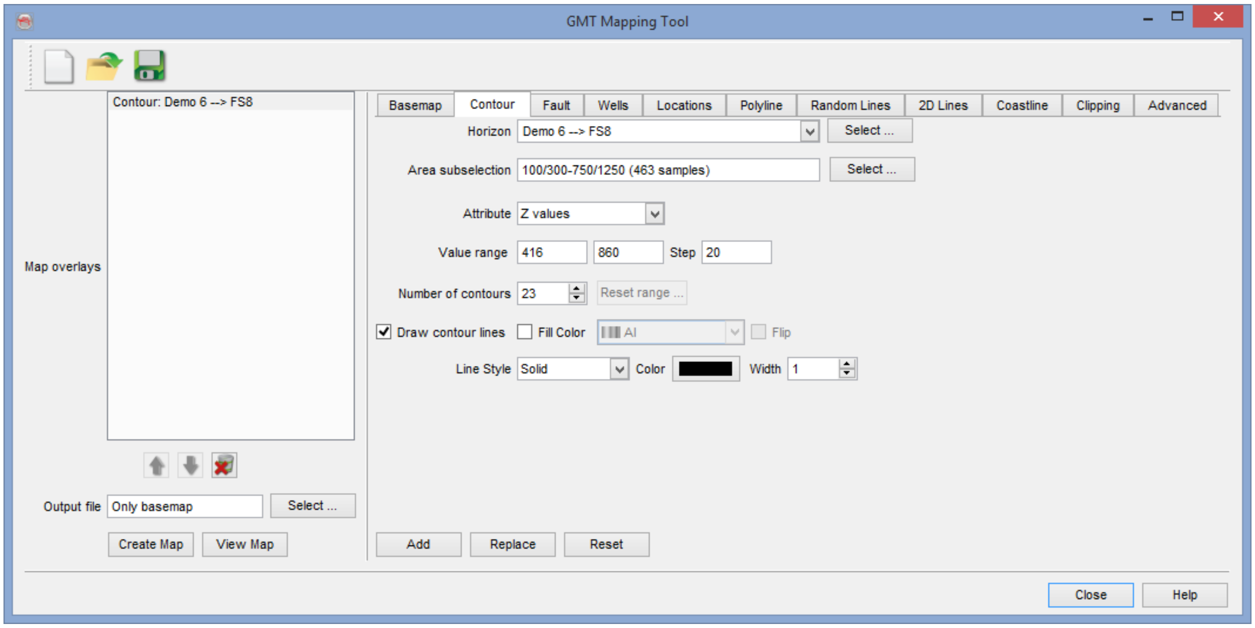
In the *Map overlays* are listed all the elements that have been defined to be displayed on the final Basemap. You can modify the order in using the icons or remove an object using the icon. The map will be created only when clicking on *Create Map*.

Basemap settings



The basemap tab is filled with default parameters including the X/Y range from the [Survey setup](). You can go back at any point to the default X/Y range in clicking on *Reset to Survey*.
The map can be renamed. The scale can be modified. Scale, map width and height are linked : any change of the scale, map width or height will affect the other two parameters. The label interval can be also be modified. The grid lines can be shown if you toggle on *Draw Gridlines*.Optionally you can also add *Remarks*.
Once the different parameters defined, give an appropriate name to *the output file* and specify the disk location and press *Create Map* button. *View Map* will display the map.

Create a Contour Map



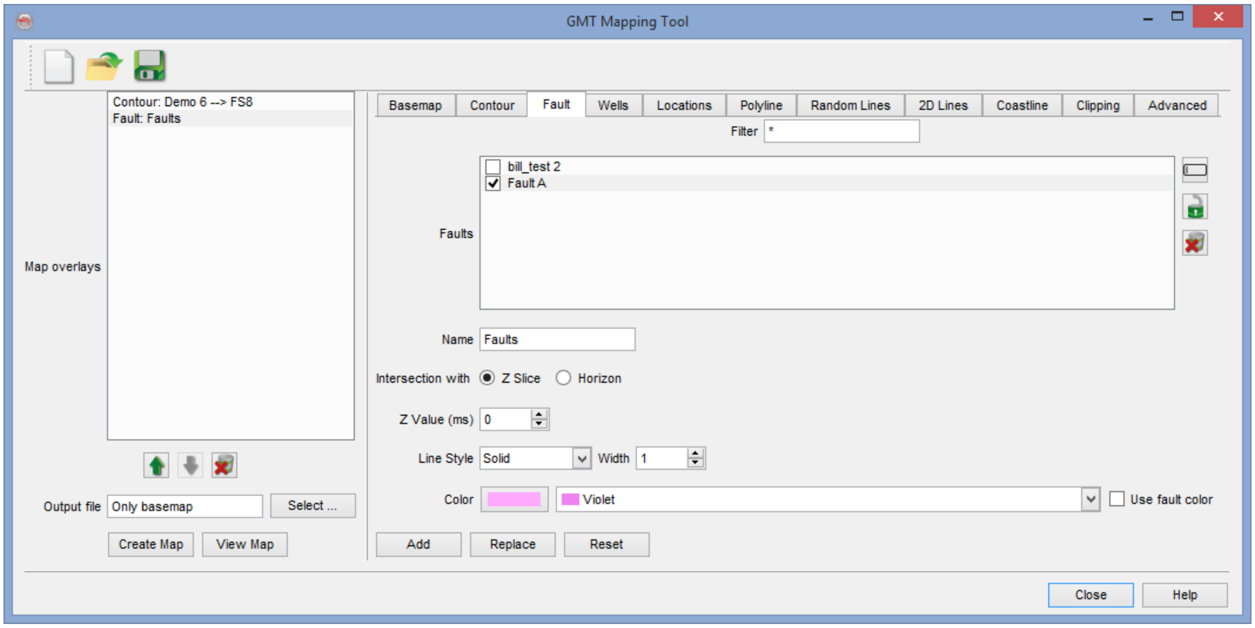
In Contour Map tab, first of all, select the horizon on which you want to create contours. The different parameters are then filled by default. It is possible to edit the value range and/or the number of contours. This will change the step. If you modify the step, it will automatically change the number of contours.

It is possible to change the display parameters. The contours can appear as simple contour lines or the space between the contours can be filled using a selected colourbar.

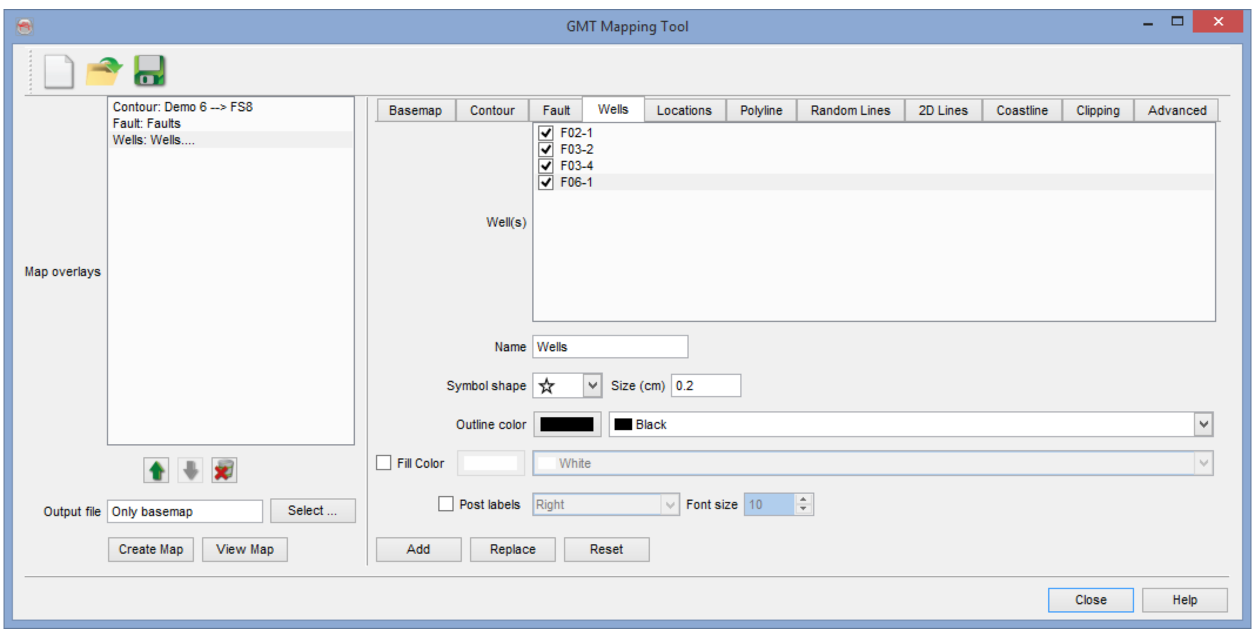
Once the parameters all defined, press *Add* button: the selected 2D data set(s) will appear on left *Map overlays* panel.

> **Note:** '*Attribute*' allows the user to select either Z-values (default option) or any of the *Horizon Data* saved to this horizon.

Insert faults



Insert Wells location



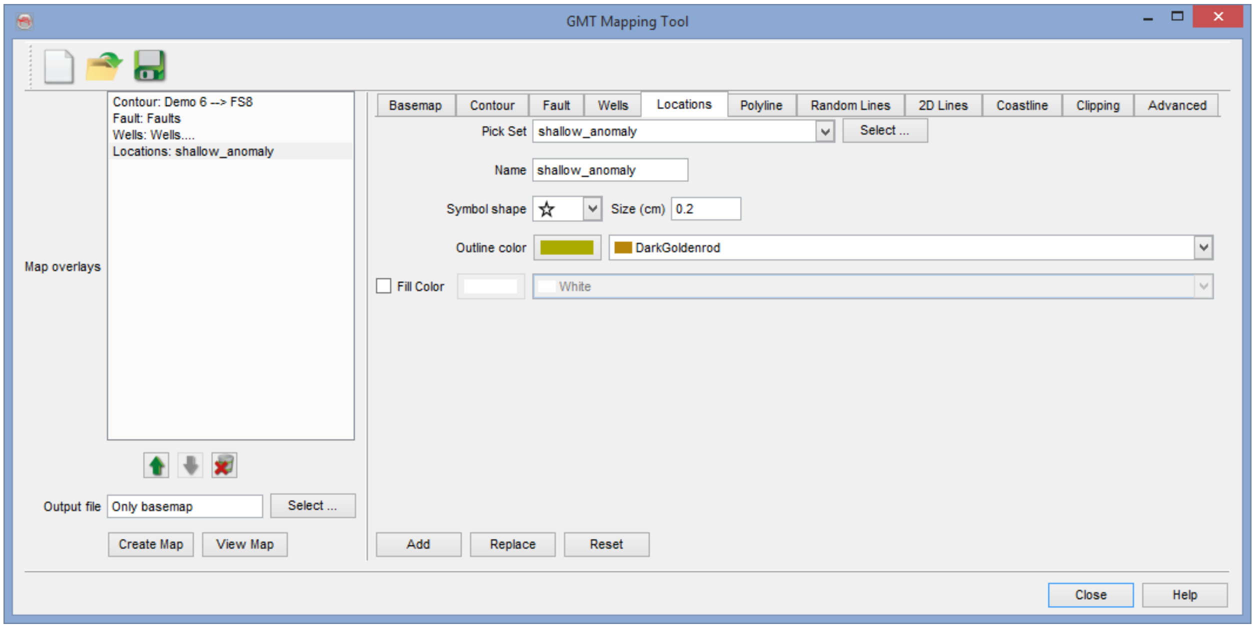
In Wells tab, specify:

1- Select OpendTect *Wells*

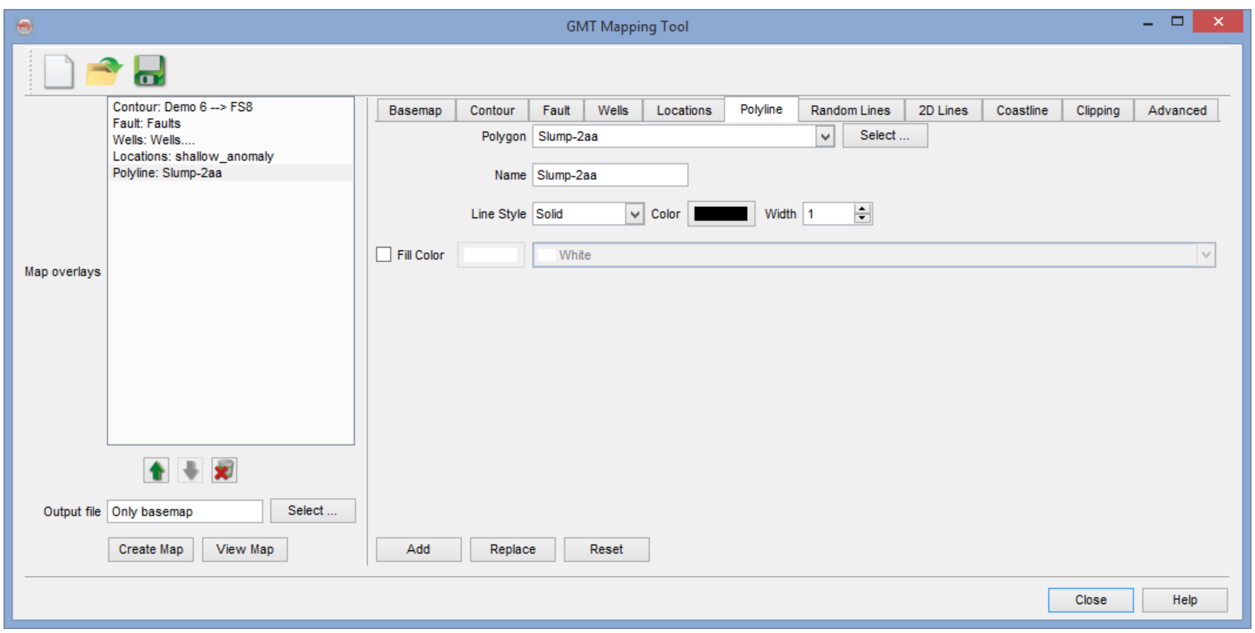
2- Optionally, edit the settings (symbols, size, color etc)

3- Press *Add* button... the selected Wells will appear on left *Map* *overlays* panel.

Insert locations



Create a Polyline



In Polyline tab, specify:

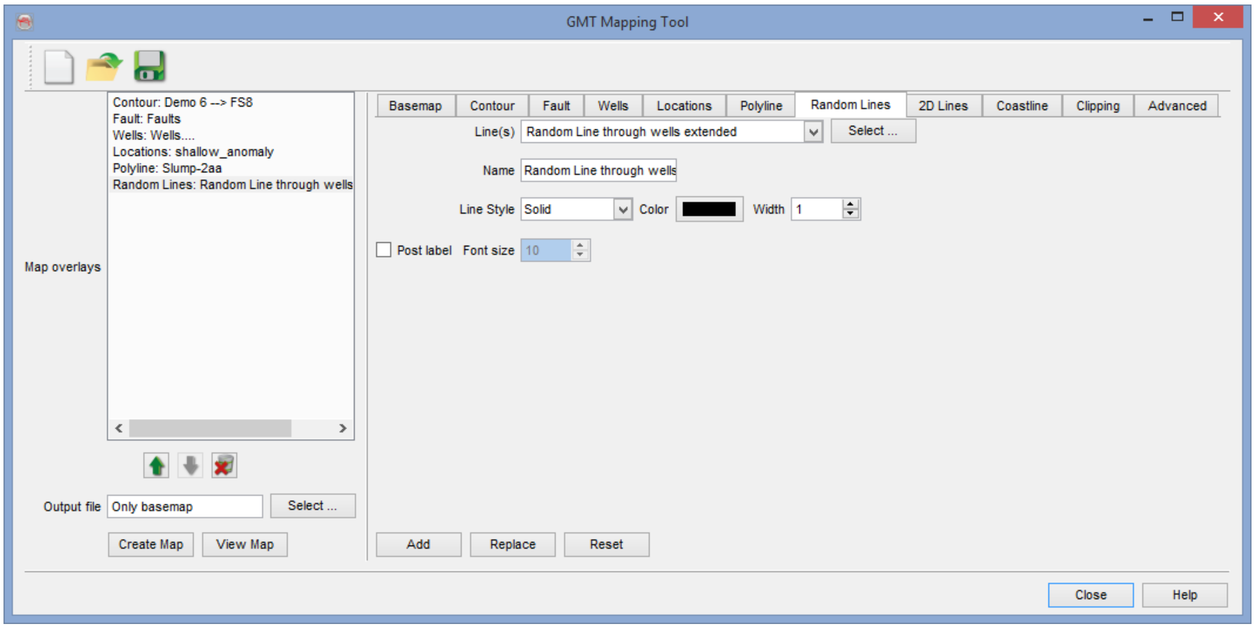
1- Select *Polygon*

2- Give a *Name* to the Polyline

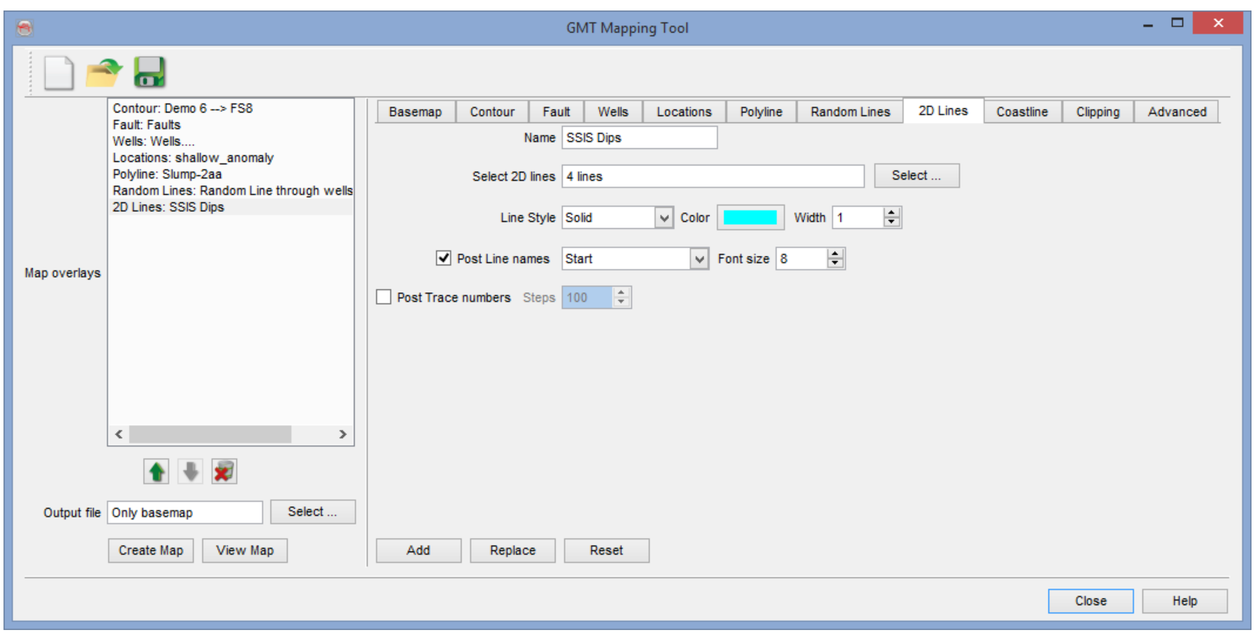
3- Optionally, edit the settings (symbols, size, color etc)

4- Press *Add* button... the selected Polygon will appear on left *Map overlays* panel.

Insert random lines



Insert 2D lines



In 2D Lines tab, specify:

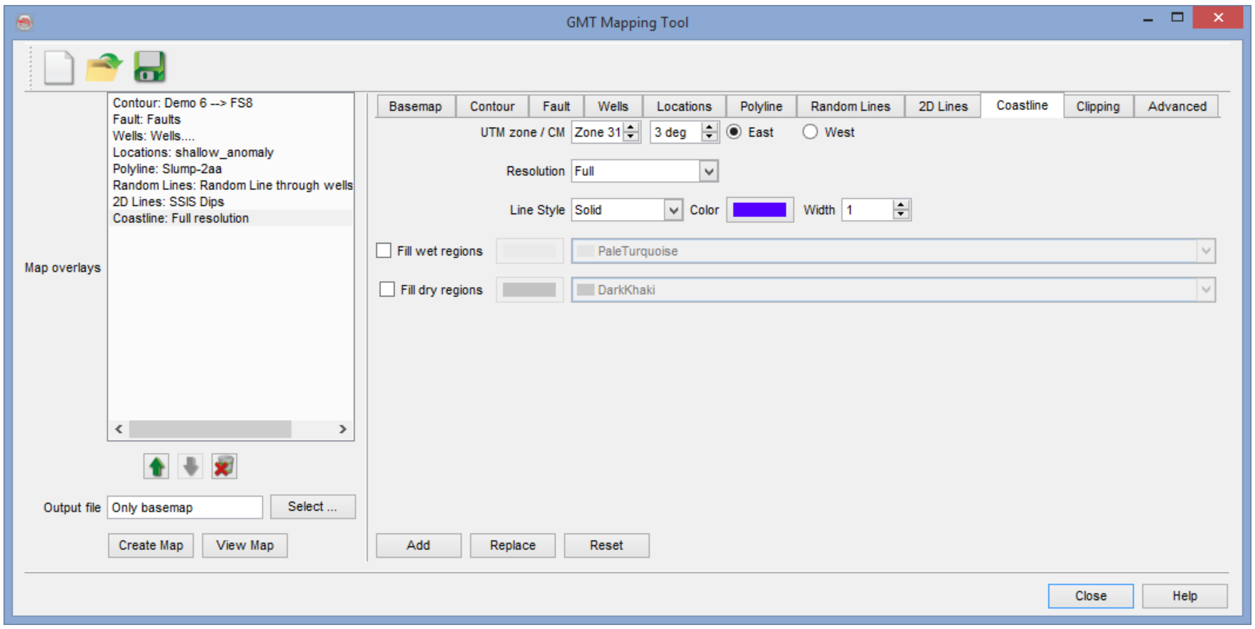
1- Select *2D line(s)*

2- *Name* the line(s) (group).

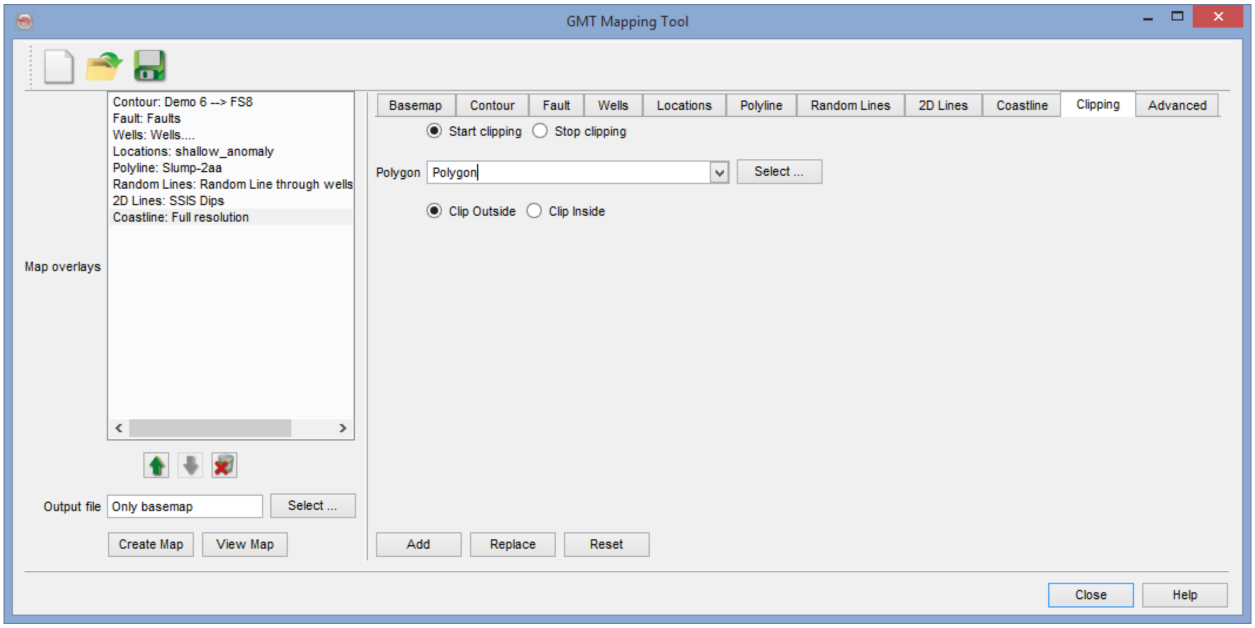
3- Edit the settings (symbols, size, color etc)

4- Press *Add* button... the selected 2D line(s) group name will appear on left *Map overlays* panel.

Insert coastline

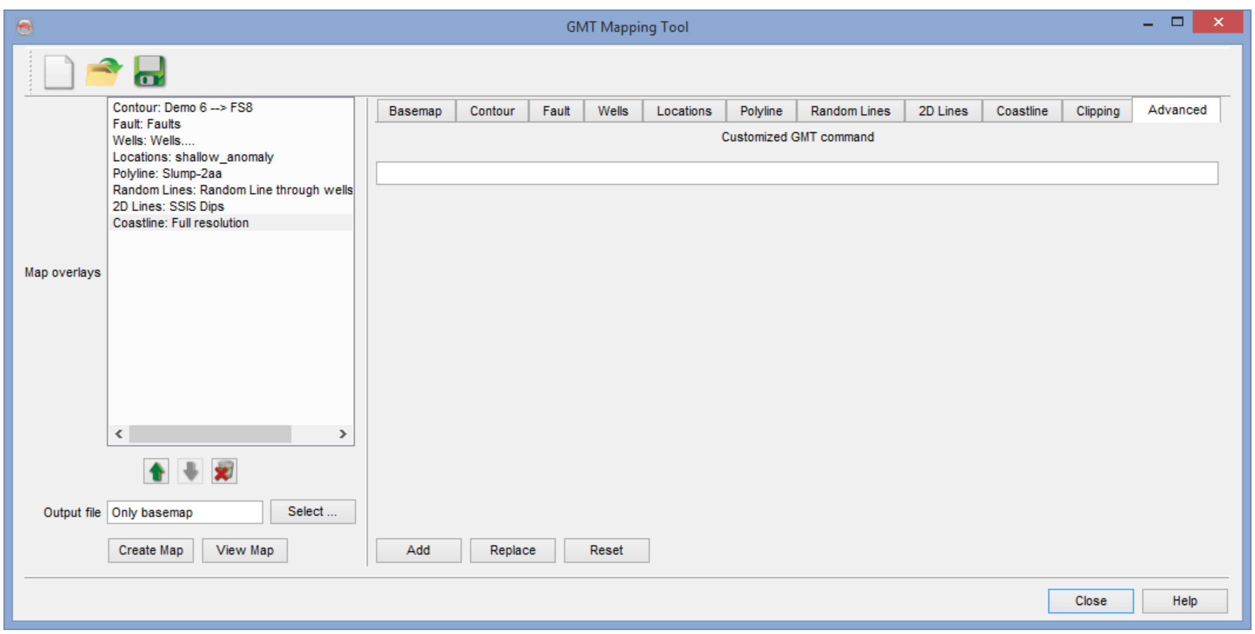


Clipping

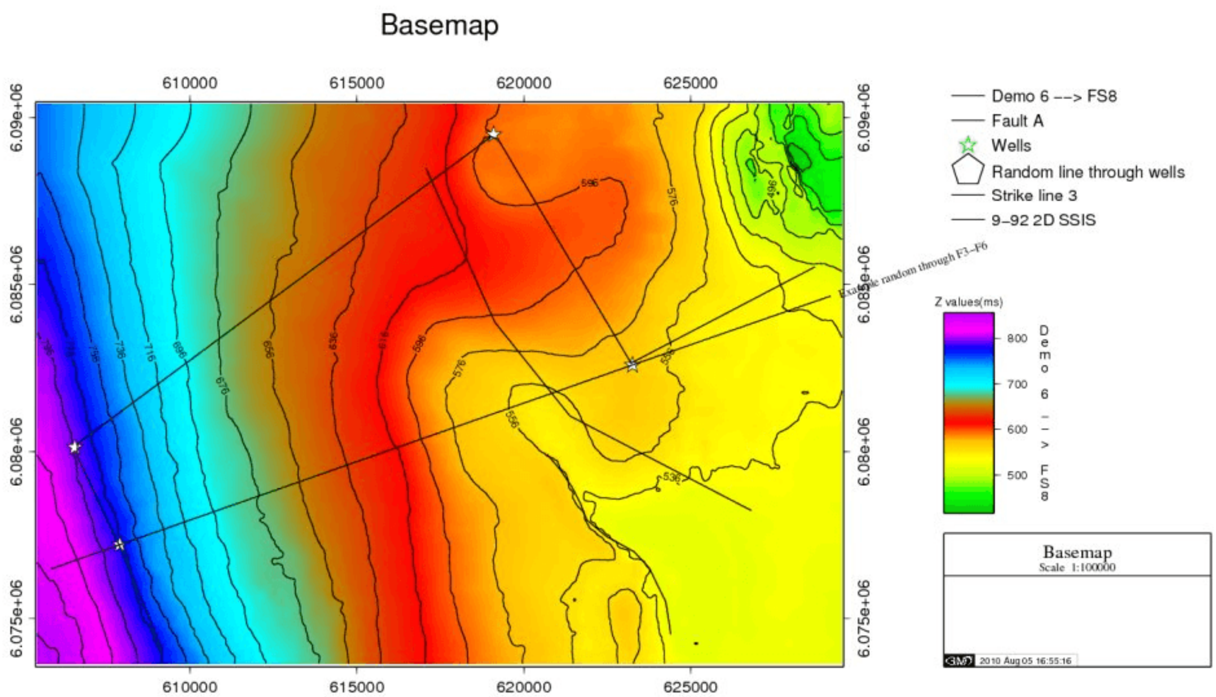


The final map will be restricted to the *inside* or *outside* of a given polygon.

Insert advanced commands



Typical output:



Madagascar

A generic user interface exists to Madagascar, an open source seismic processing package that is very popular in seismic R&D communities. In the builder, seismic pre- and post-stack input and output files are either OpendTect or Madagascar formatted. The processing flow is constructed as a sequence of Madagascar programs, using their parameters. These programs are selected from a list of available programs (presently over 300), with a search field included to guide the user.

Madagascar processing results can be further analysed in OpendTect.

Notes:

1- First Madagascar must be installed in order to use this interface between OpendTect and Madagascar.

2- It's not possible to view Madagascar plots directly from the OpendTect user interface on Windows. If the user wants to see the plot, she/he has to make her/his own arrangements like starting the xserver etc ...

Comment: Problems may occur occasionally when using Madagascar on a

Windows system.

Madagascar Installation

Madagascar is an open-source, standalone software. To be used with OpendTect, *Madagascar* must first be installed, otherwise, when starting Madagscar , the next window will display an error message and missing program boxes.

The *Madagascar* package needs to be installed (see install) and the RSFROOT variable has to be set to the installation directory. In order to get the full UI, ensure that the text doc is installed. This can be done with:

$RSFROOT/bin/sfdoc -t $RSFROOT/doc/txt

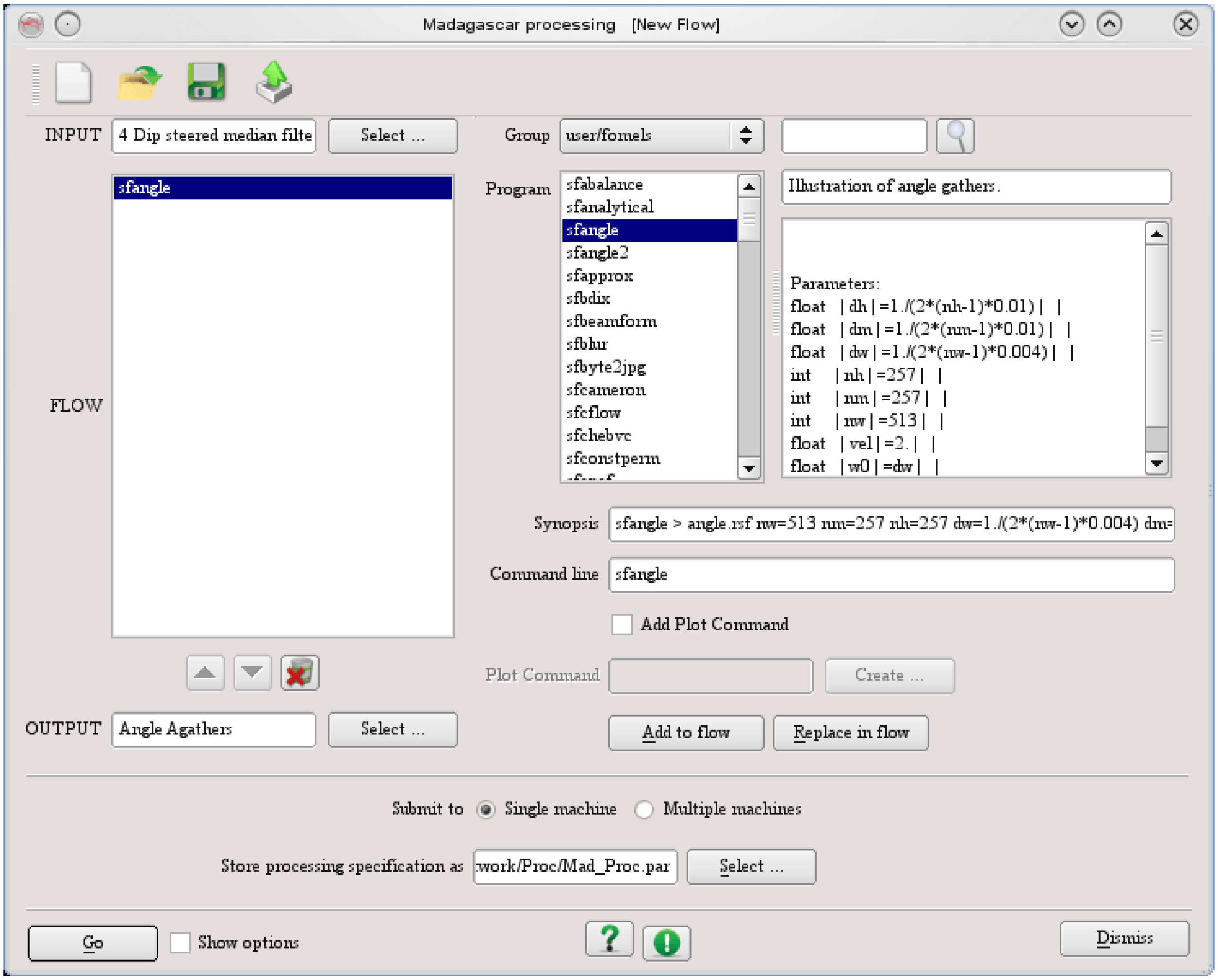
On **Windows**, Please ensure the following to be able to use the Madagascar link in OpendTect:

1. In Advanced System Settings -> Environment Variables, the variable RSFROOT must be set to the Madagascar installation folder. Setting this variable only in the Cygwin environment is not enough.

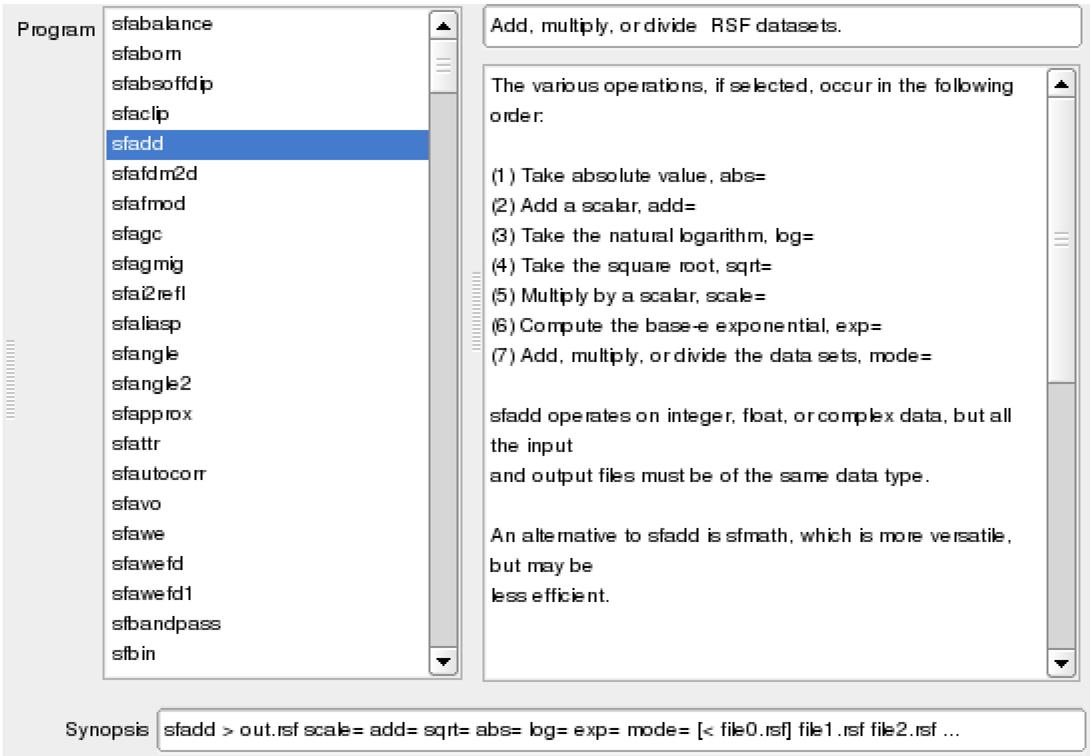
2. The variable PATH must include the Cygwin bin folder (e.g. C:\cygwin\bin).

Madagascar Processing Window

The *Madagascar* processing window can be launched from the OpendTect toolbar by pressing the *Madagascar* icon.



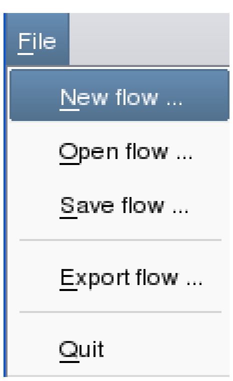
Select the input cube to be processed, and then choose a program or combination of programs. Programs are organized into groups of programs. Once one program is selected, a description of program's functions are shown in the neighboring frame.



The different steps, as well as a synopsis, of the computation are provided. The descriptions of each program are available on the Madagascar site.

Toolbar

The toolbar is composed of the *file* option and three shortcut items.
The *file* option is as follows:



The toolbar contains three shortcuts to *create*, *open*, and *save* the flow: 

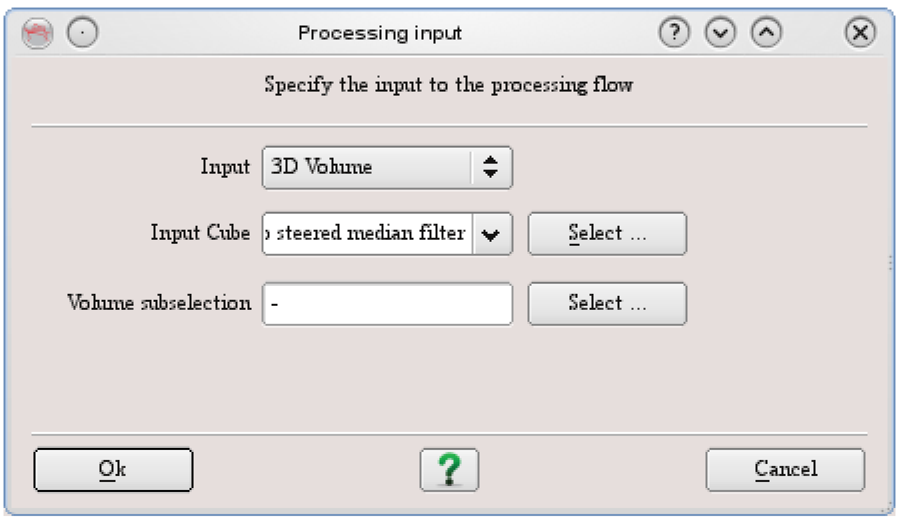
This creates a new processing flow.

This will open a saved flow.

This will save a newly created flow.

Processing Input

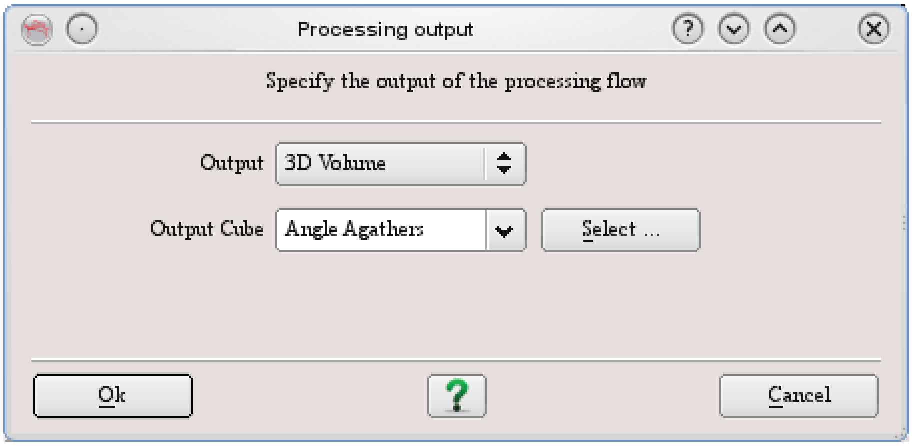
The first step is to select an input cube.



The input can be a 3D volume, a Prestack volume, a Madagascar volume, or None. It is possible to choose a volume sub-selection.

Madagascar Processing Output

The final step is to chose an output volume type.



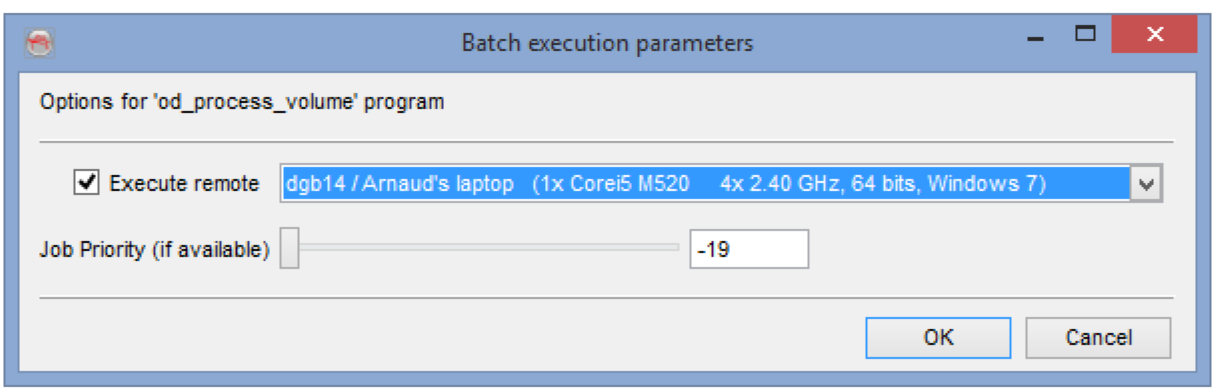
Similar to the input selection, it can be a 3D volume, a Prestack volume, a Madagascar volume, or None.

Batch Processing

Though *Batch Processing* does not actually appear in the pop-out menu under *Processing*, this is a convenient place to present the information on these processes. *Single-Machine*, *Multi-Machine* and *Cluster-Processing* are explained in following sub-chapters, along with the *Batch Log Files*.

Single Machine Batch Processing Window

In single mode processing, the data can be processed either on a *local* machine or on a *remote* host. All relevant information on the progress of the calculation will be stored, (see Job information file).



'Execute remote' toggled on to send the job to a remote machine. The job priority can be changed (-19-lowest to +19 highest).

Multi-Machine Batch Processing Window

The multi-machine batch processing window controls on which machines a volume output or steering cube batch job will be processed. Jobs are distributed over the *Used hosts* on a line-by-line basis (the number of inlines per job can be specified). Hosts can be *Added* and/or *Stopped* at all times. Processed results are stored in a *Temporary storage directory*.

At the end of the processing sequence, OpendTect will merge all processed lines and store the data in the output file that was specified in the *Volume output* or *Create steering cube* window, and it will delete the temporary disk files. If for any reason OpendTect fails to perform this data transfer, this can also be done manually in the *File - Manage* module. The temporary data store appears with a name starting with Proc_. Select this item and copy it to a new cube.

It is possible that at the end of a multi-machine batch job not all data was processed successfully. Some jobs may have failed (e.g. because one of the hosts was temporarily not available). OpendTect will come back with a warning message stating which jobs (i.e. which inlines) have not been processed yet. It is then advised to re-submit these jobs until all data are processed. The *Auto-fill* option automatically scans and fills gaps in the processed volume.

The *Nice* level sets the priority the process gets. With the nice level set on 19 for exemple the process has very low priority and other processes that run on the same computer get priority. If the nice level is set to 1 the process gets the highest priority.

The *Processes* menu allows to set the *Run*, *Pause*, or *Go - Only between* options. The *Go - Only between* option, pauses and runs the processes at user-defined times.

OpendTect calls the system utilities of the '*hostent*' (sethostent, gethostent, etc.) type to get a table of hosts that can be selected. How the Operating System builds the lists is dependent on the particular system setup; most likely /etc/hosts and/or the NIS tables are consulted. OpendTect supports multi-threading which means that all processors of multi-processor machines will be used.

> **Note:** Though we support multi-threading, not all calculations can be run this way due to some of the algorithms involved (ie: recursive calculations). See the following table:

Attributes that support multi-threading	Attributes that do not support multi-threading
Central Steering	BG Steering
Convolve (all except Wavelet option)	Constant Steering
Curvature	Convolve (Wavelet option)
Dip Angle	DeltaResample
Velocity Fan Filter (=DipFilter)	Energy (Gradient option)
Energy (all except Gradient option)	Event Steering
Event	FaultDip
Hilbert	FingerPrint
HC Data	FreqFilter
HC Density	Frequency
HC Layer	Full Steering
Instantaneous	GapDecon
Local CCB	HC Curvature
Math (except when expression is recursive)	HC Dip
Polar Dip	HC Spacing
Position	Horizon
PreStack	Match Delta
Reference	Matching Pursuit
Scaling (all except scaling type AGC and stats type = detrend)	Math (recursive expression)
Semblance	Perpendicular Dip Extractor
Similarity	SampleValue
SpectrogramDip (create steering cube, FFT steering)	Scaling (scaling type AGC and stats type = detrend)
Texture	Shift
Tutorial	Spectral Decomposition
Volume Statistics	

Multi-machine processing on Windows OS

The new system works with a Daemon Service running in background on every remote machine to be used for processing. The communication works with TCP/IP and requires some configurations to actually make things working.

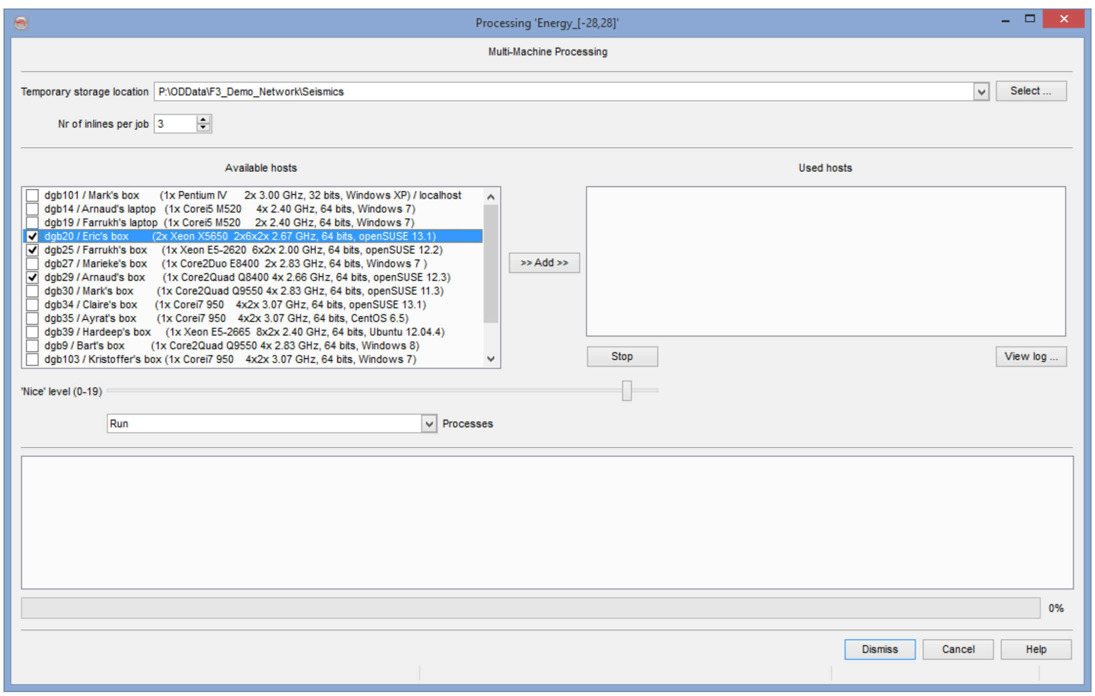
Mapping of Survey folder: We assume a DataRoot folder ($DTECT_DATA) to be in a centralised server. This folder has to be mapped on the same drive in every PC to be used as hosts. For example, we have 4 PCs (A,B,C,D). A is a server and the ODData is in drive D. Then map D:ODData on e.g. S: on B, C and D. PCs B, C, and D can then be used as your processing nodes.

OpendTect installation: You need to have OpendTect installed on all hosts, and make sure they all use the same survey at the same time (have access to). For example if B is using F3_Demo and want to process something in F3_Demo. then it has to be made sure that the rest of the two PCs also use the same folder as long as the processing is needed.

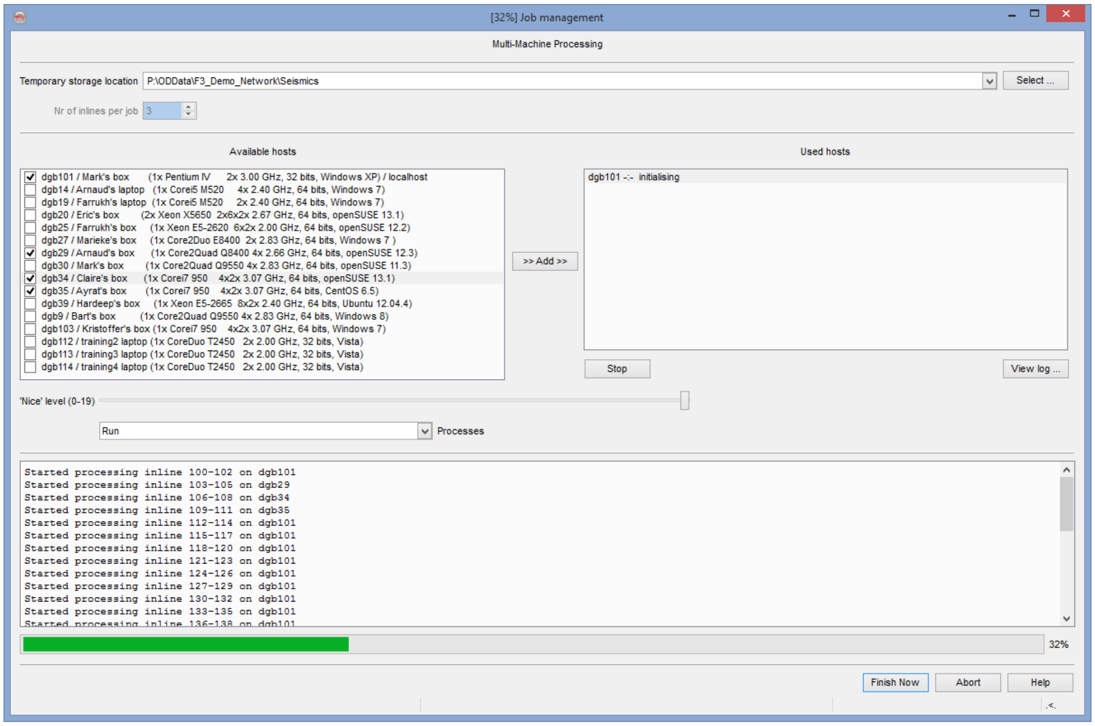
BatchHosts file: Add the IP addresses of C and D in the BatchHosts file inside the application data folder.

Start the daemon: If launching process from B to the other two, then B is his local machine, C and D are remote machine. In this case the Daemon service (odremoteservice) application has to be launched from binwin folder (win32/win64) only in the remote machines and not in the local machine (B). Please note the odremoteservice.exe not to be run directly instead a launching tool will be found in win32mmod_remote_service_manager. Use od_ remote_service_manager to launch the daemon which will also add anotification icon to the system tray. Once the service starts, the remote machines are ready.

Start processing: Select the PC's B, C and D from the list of machines in the multimachine launch window and start processing:



Select machines to use for processing from the list



Multi-machine batch processing progress window

For more information, please refer to the [OpendTect YouTube Channel] for the webinar on: [Multi-Machine Processing Set-Up]

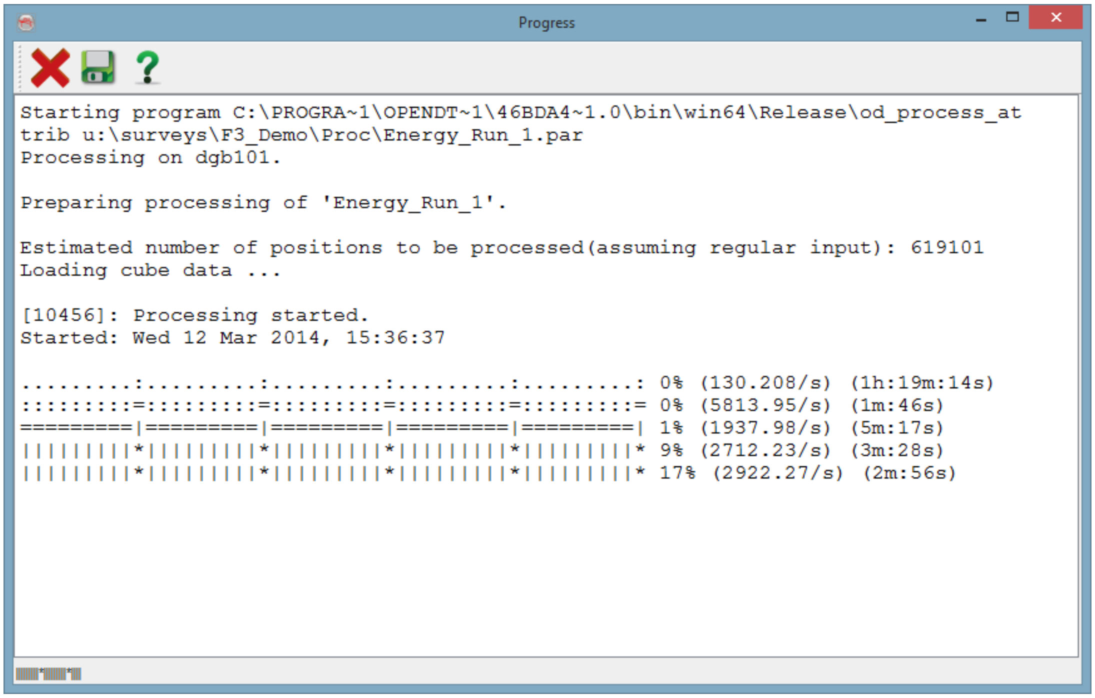
Batch Log File

A batch log file is produced for every volume output run. The information is streamed to a file if the batch job is executed on a remote computer. If the processing is done locally the log file is either streamed to a new, dedicated window, or to the standard output window. Every N traces the program will output a symbol to reflect the progress. There are five symbols the program can use. Which symbol it uses depends on the speed in number of traces times N per sec (given towards the end of a line in brackets after the percentage of traces processed) and the estimated remaining time until completion. The symbols indicate the following:

- a period (**.**) means 1 trace processed
- a colon (**:**) means 10 traces processed
- an equal sign (**=**) means 100 traces processed

• a pipe marker (|) means 1000 traces processed

• an asterisk (*) means 10000 traces processed



The following options are available:

 Stops the process.

 Saves the log file.

 Brings you to the help menu.

Cluster Processing

Batch jobs can be run from OpendTect to cluster management tools. So far dGB has successfully tested SLURM which is easy to install and even easier to use.

Cluster processing is enabled by following this prerequisites:

- The environment variable DTECT_CLUSTER_PROC is set to yes before starting OpendTect
- The cluster management tool must be up and running. In the case of SLURM it means that the daemon 'slurmctld' should be running on the master computer and that the daemon 'mlurmd' must be running on every node.
- This cluster management tool (SLURM) bin directory is included in your PATH variable

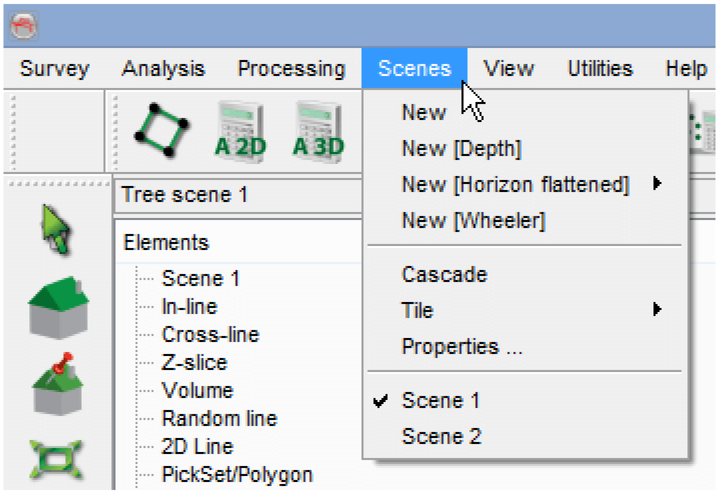
Cluster processing is started by choosing "Cluster" in the 'Submit' options menu of the volume processing window. Only 3D attributes can make use of cluster computing so far.

A new window will pop up that will list a number of directories use for the storage of temporary files. The jobs will be split using a user-defined number of inlines. The field named 'Cluster Processing command' represents the name of the binary from the cluster management tool used to run a process.

You can run the "Main script file" (default: ~/yoursurvey/Proc/clusterprocscript) from a command line which will run each job one-by-one using the above command and will also launch the UI to show progress and do post-processing merging of temporary data.

Scenes

Table of Contents



OpendTect

The OpendTect main window can have multiple scenes, most of them opened using this menu. The scenes behave like sub-windows within the main window: Each scene has its own tree and can be minimized, maximized, reduced or enlarged in size, without ever going out of the main window. The trees of different scenes can be move on top of each other and sorted as tabs, or completely separated from the main window (they are utility windows).

The *Cascade* option will restore a default size for each scene and sort them starting on the upper left corner of the main window.

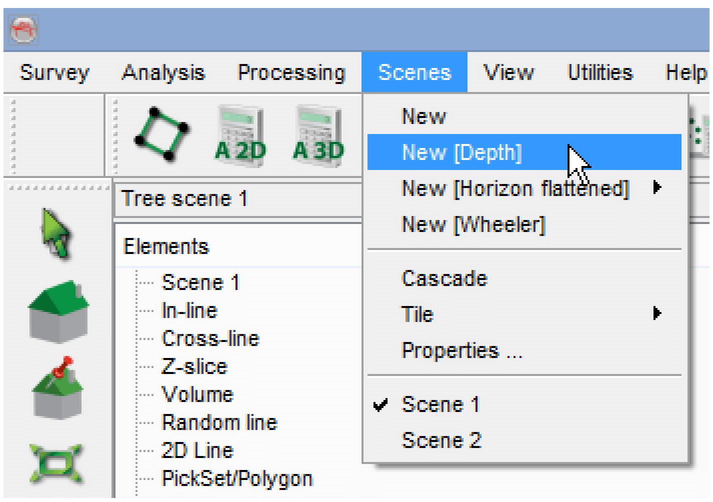
The *Tile* option is a shortcut to maximize each scene by sharing the space of the main window equally:

- **Auto:** The scenes are sorted automatically along the best fitting grid.
- **Horizontal:** The scenes are arranged along a single line.
- **Vertical:** The scenes are arranged along a single column.

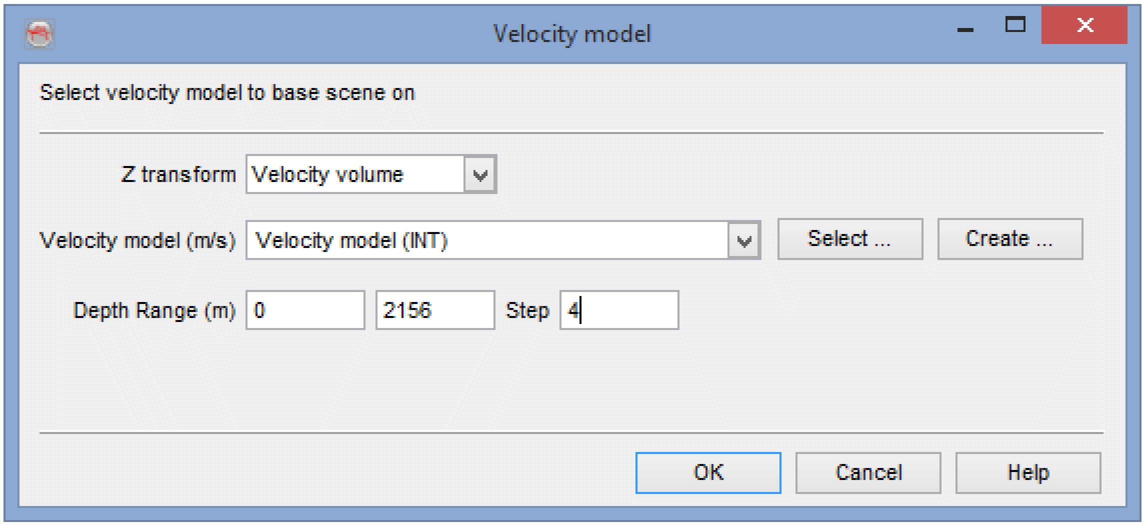
If all scenes are maximized the active scene will be annotated on the left in the *Scenes* menu. Clicking on another scene will make that one active.

Time- and Depth-Converted Scenes

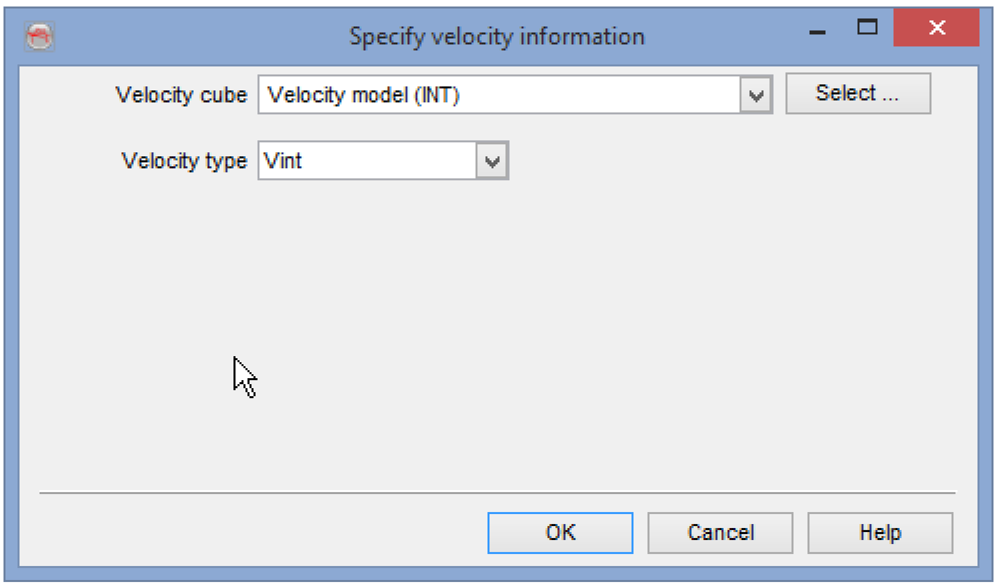
OpendTect can display time data in the depth domain and depth data in the time domain.



This is done using a user-selected velocity volume and computing the new Z range (depth or time) based on the original Z range (time or depth respectively). In all transformed scenes each and every display elements is re-positioned on-the-fly.

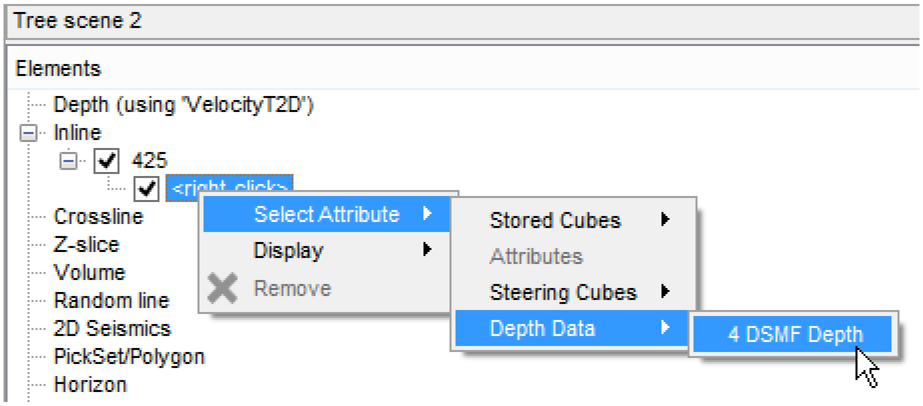


Pressing '*Create*' will pop up a dialogue that allows you to specify the velocity type for a given volume:



(See Tagged Seismic Data for more detail.)

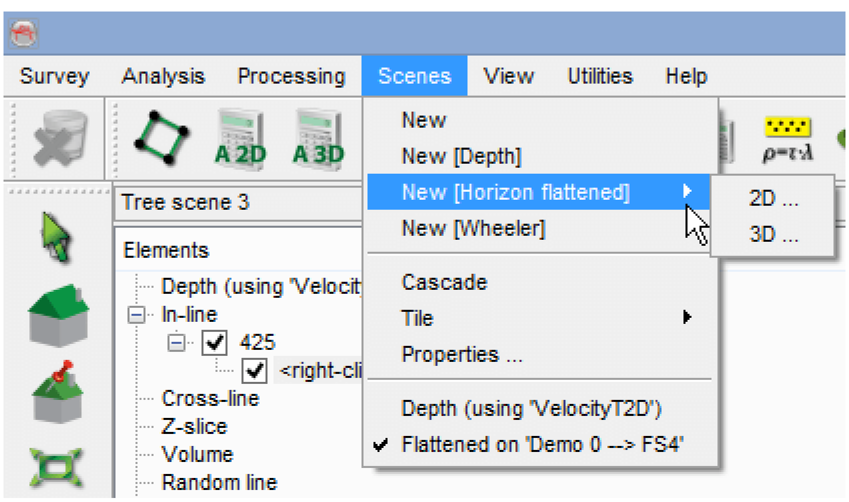
The only exception is 3D volume for which the on-the-fly transformation can be slow. Therefore time volumes can be depth converted (i.e. they become stored volumes) using an additional option in the right-click option of inlines and crossline, in the transformed scene:

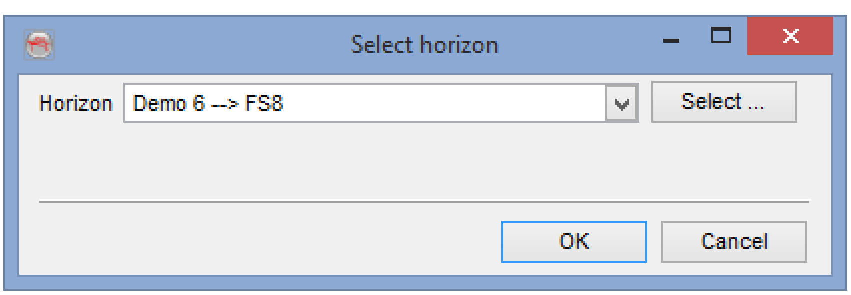


Please note that depth-stored volumes can also be imported via SEG-Y by settings the appropriate tag in the SEG-Y import wizard.

Flattened Horizon Scenes

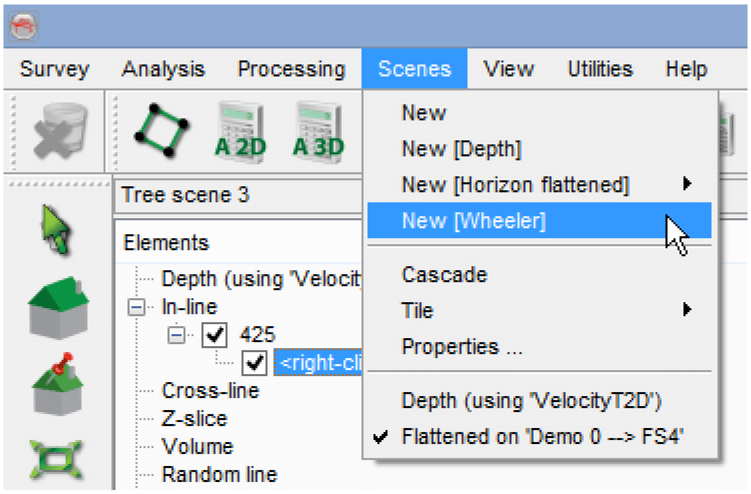
This option will generate a new scene flattened about the selected horizon. The Z range has the same unit as the original scene, but it is now relative to that horizon and no longer absolute.





Wheeler Scenes

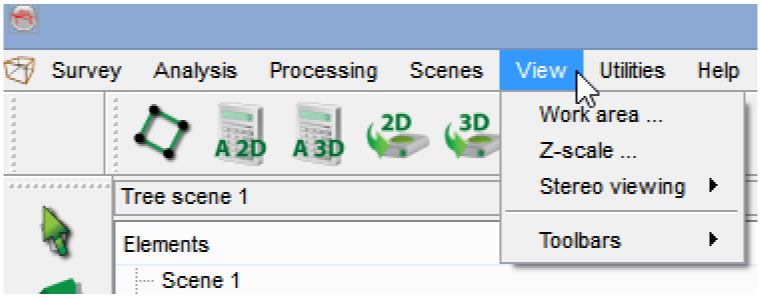
The Wheeler Scene is a transformation (flattening) of HorizonCube into relative geological time (RGT). Therefore, before adding a Wheeler Scene, the HorizonCube will need to be selected. You will be prompted for this if not already selected.



Note: This option is only available if the SSIS plugin is installed.

View

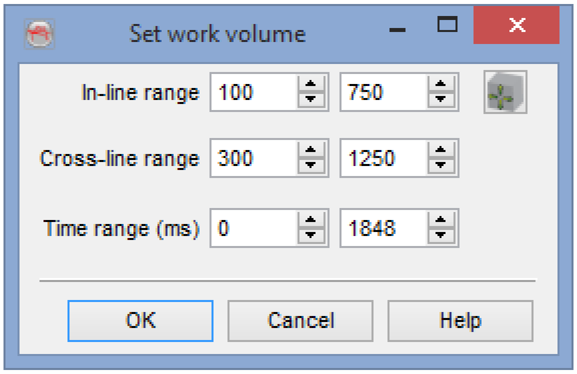
Table of Contents



OpendTect

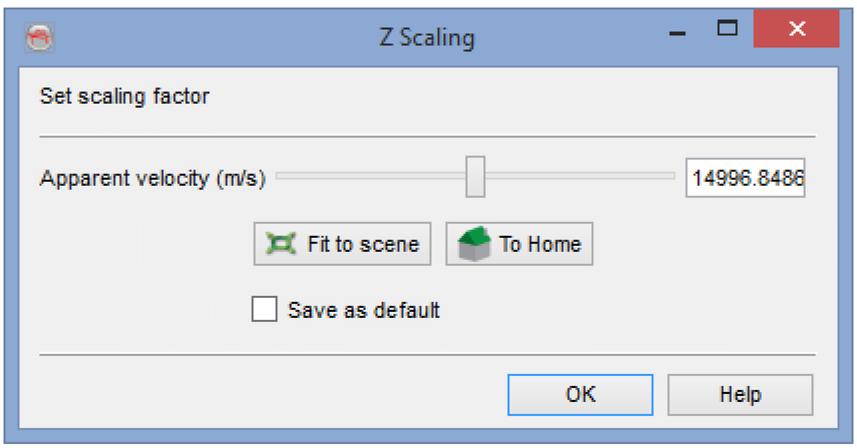
Work Area

The Work area dialogue is opened from the 'view' menu. The Work area sets the area bounded by the survey box. Displayed items will be cropped automatically to fit the set inline, cross-line and z-ranges.



Z-Scale

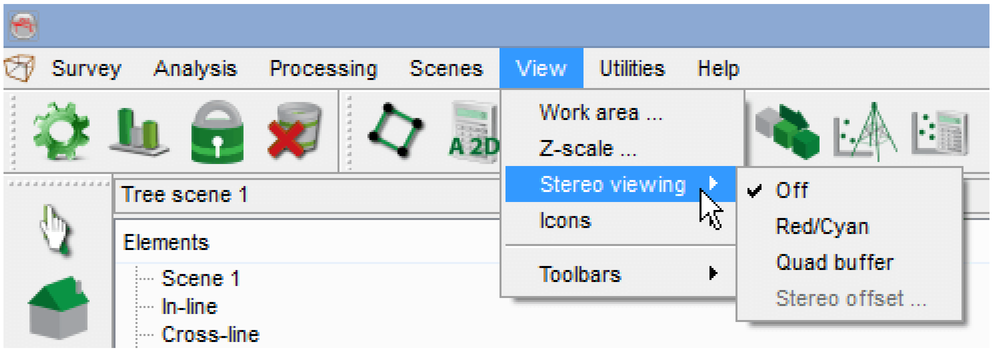
The *Z-scale* option allows scaling of the survey box vertically.



There are three options for *Z-scaling*: The slice bar, setting scaling according to current scene "*Fit to scene*" and resetting the scaling "*To Home*".

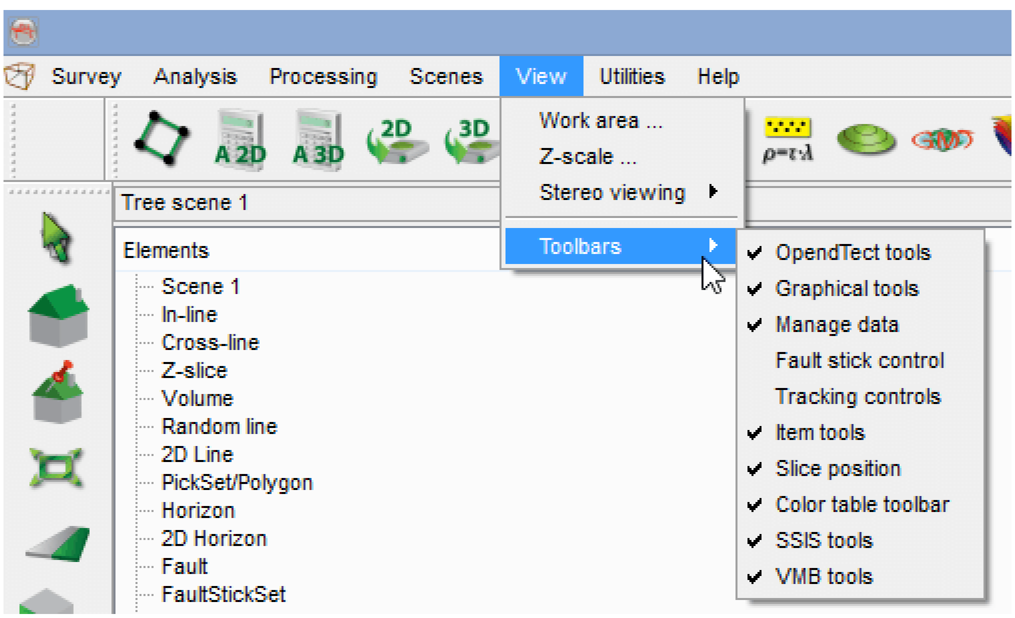
> **Tip:** you can set any position/orientation of the scene by clicking on as default Home.

Stereo Viewing



The Stereo viewing menu allows setting the stereo viewing on/off. Note that in order to use *Red/Cyan* stereo, appropriate glasses are needed. The offset between the red and cyan view can be manipulated with the *Stereo Offset* menu. The *Quad buffer* option has special hardware requirements in order to get passive stereo view on a screen with dual and polarized projection.

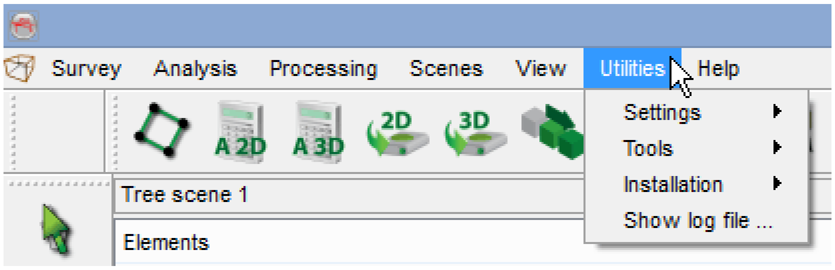
Toolbars



All elements available in the main OpendTect window can be switched on/off here. See Toolbars for the various actions of the buttons on the toolbars.

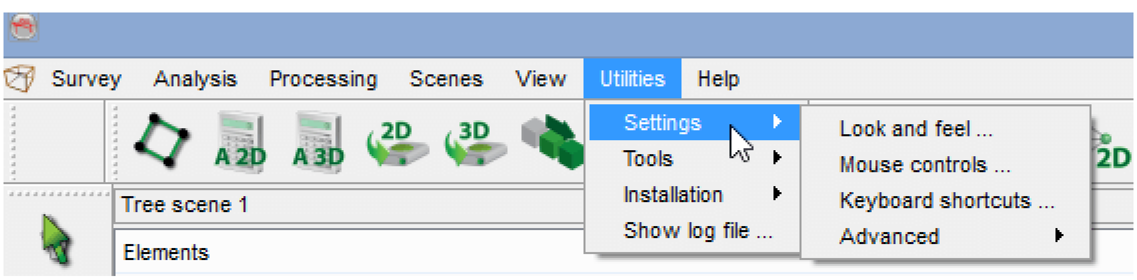
Utilities

Table of Contents

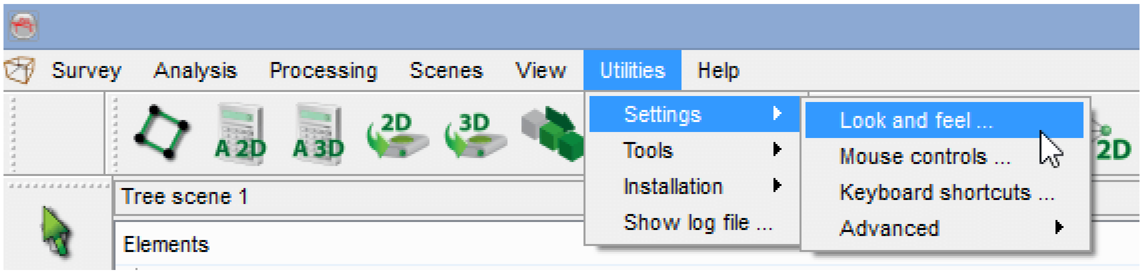


Settings

The settings for *Fonts*, *Mouse*, *Keyboard*, etc., can be changed from *Utilities--> Settings-->*



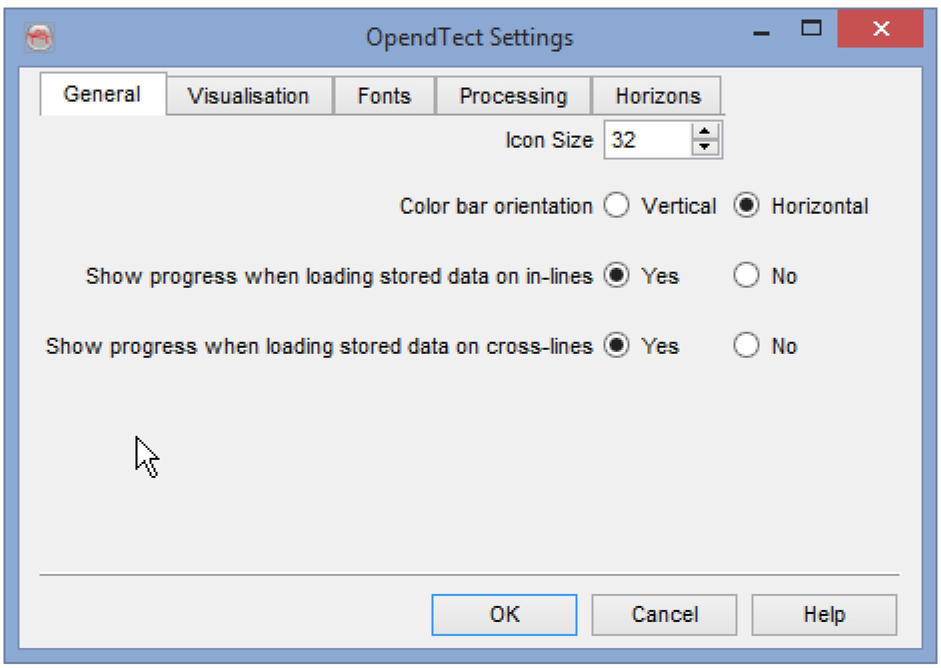
Look and feel



This option brings up an interface containing several tabs for defining various settings in OpendTect:

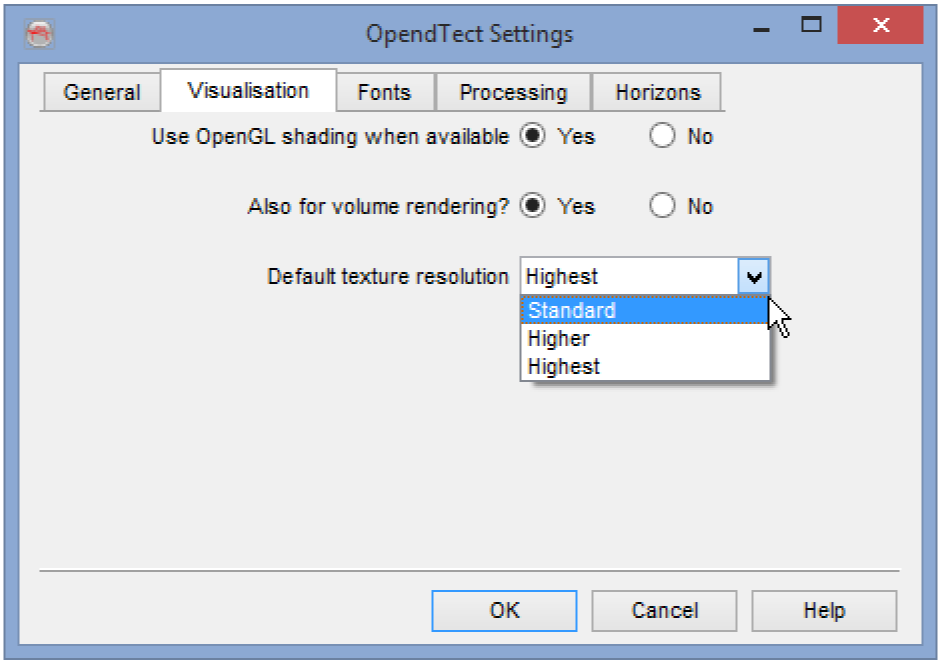
These are explained in detail below:

General:



Tip: Default Icon Size is 32. For systems with smaller screens (esp. laptops) it may be useful to reduce this value to 28 or even 24. In combination with reduction of Font size, this can prevent windows be 'oversized' for the screen.

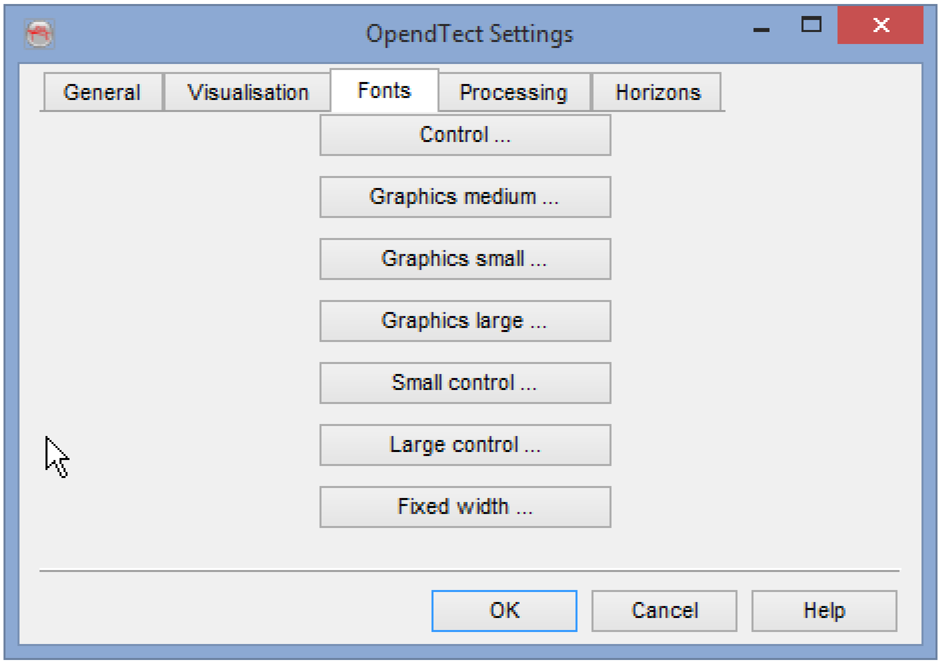
Visualization:



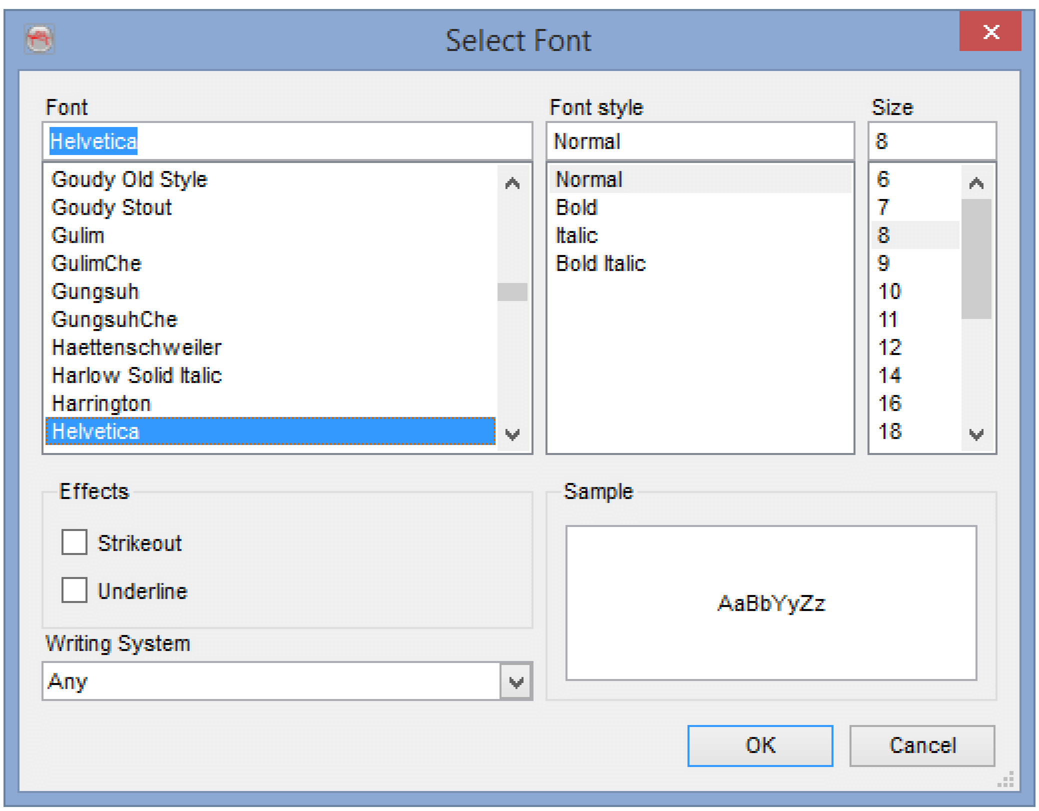
In the visualization tab , you may choose for OpenGL shading, also for volume rendering, or switch off this option.You may also define the texture resolution factor to one of these three settings:

> **Note:** Users facing data visualization issues may significantly improve their results by turning off the shading and setting the resolution to Standard.

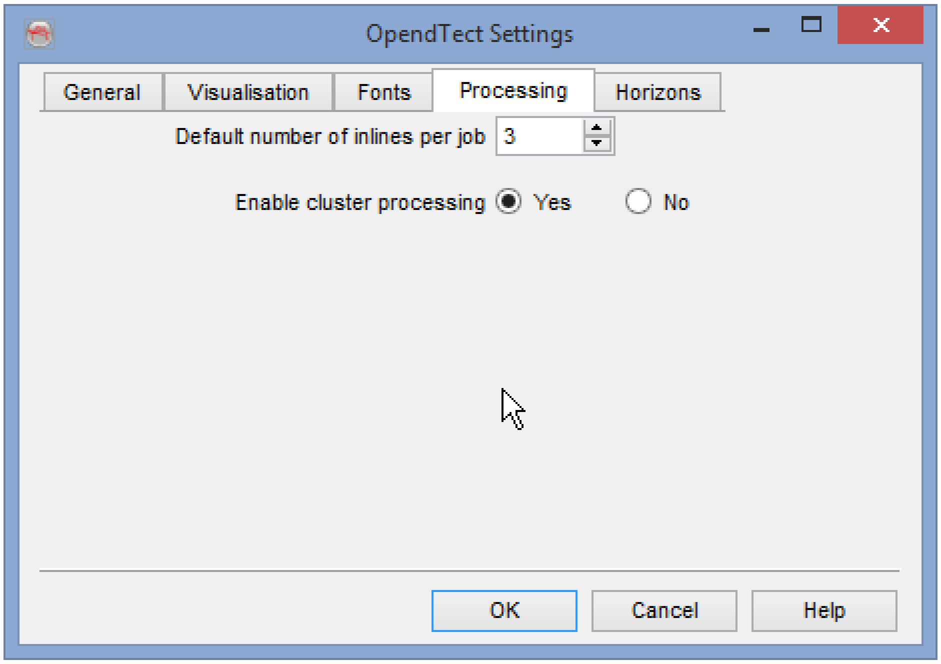
Fonts:



Clicking on any of the listed buttons brings up a standard font definition wion-dow:

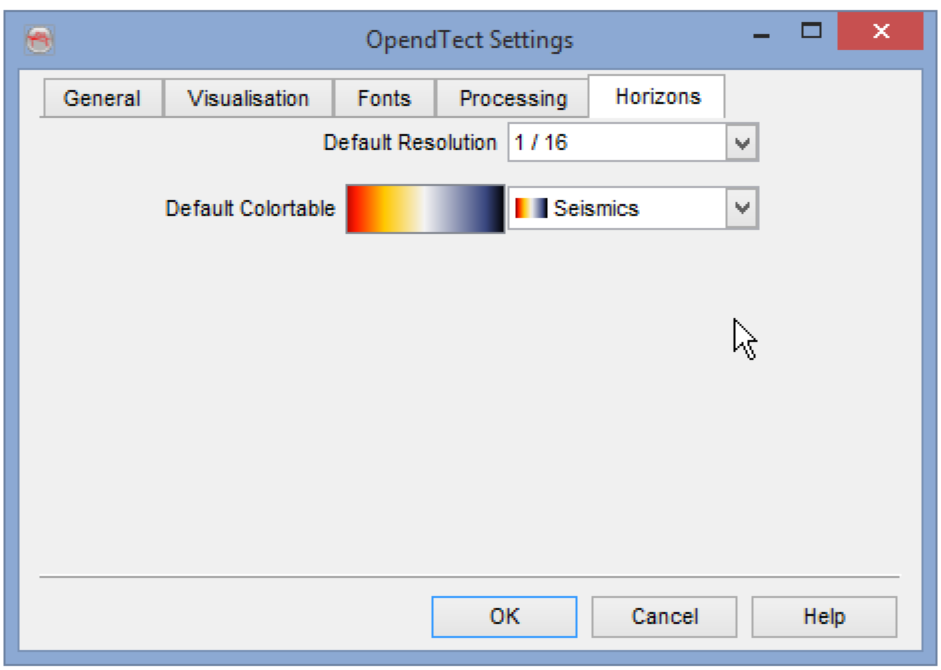


Processing:



Please see the following sections for full details of Batch Processing and Cluster Processing.

Horizons:

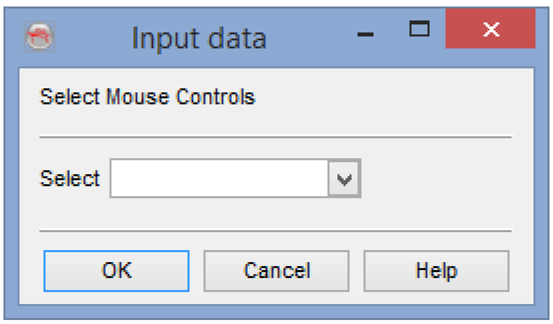


Using this option, one may set both the default resolution and default colortable for horizons. This is an especially helpful option for orientation in the early stages of a project when many horizons are loaded.

As with many of the other settings, a restart is required to apply these defaults.

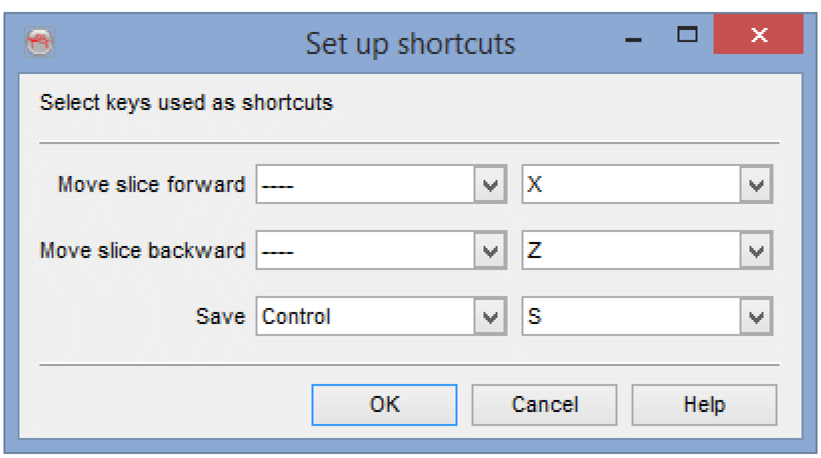
Mouse Controls

The mouse buttons can be set differently. System administrators can implement user-defined mouse button actions. See the Application Management Documentation for more details.



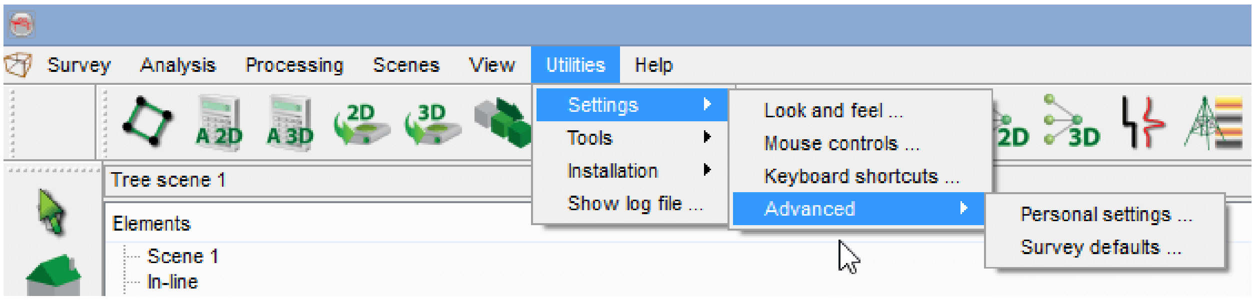
Keyboard Shortcuts

The user can define his/her own keyboard shortcuts to move a slice forward/backward. The user can use one key (set the first key to no-button) or a combination of control or shift key, plus another key which can be selected from a long list.



Advanced Settings

The *ADVANCED* user settings are used to change the default settings of specific keywords. The user can specify his/her own settings and/or setup or edit the Survey defaults settings.



These settings can be found in the .od/ directory from the user home directory. Depending on the platform used, this is located in:

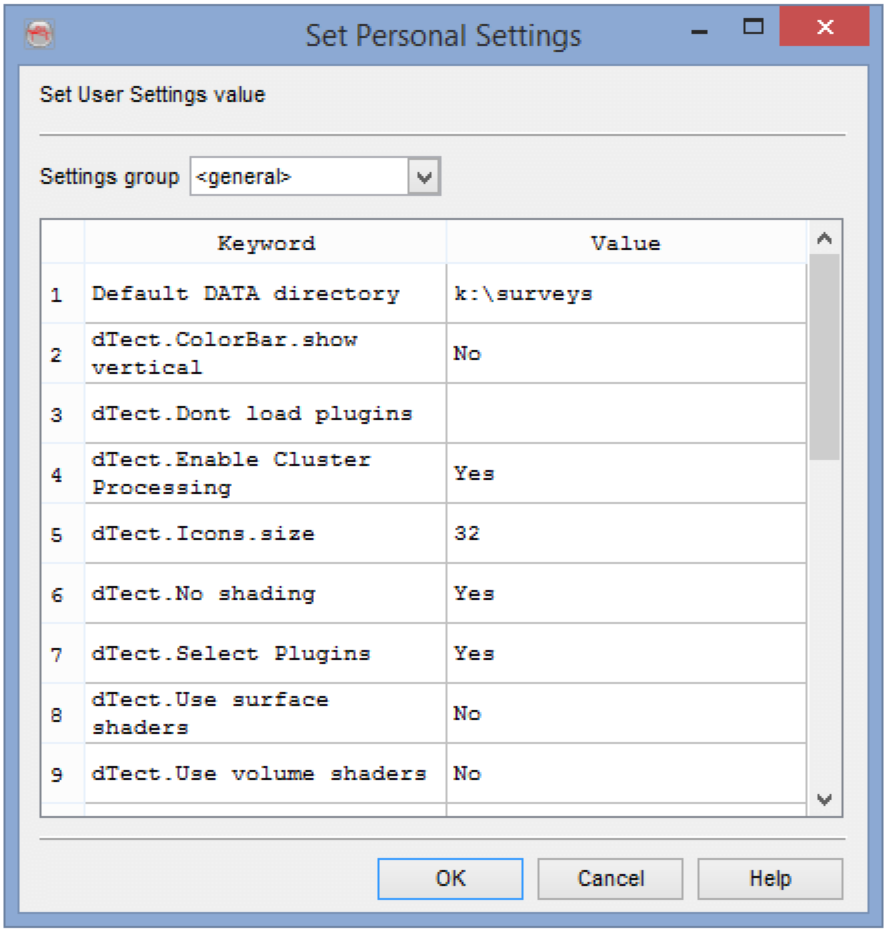
- /home/username/.od for Linux users
- root > Documents and Settings > username>.od in Windows 2000, XP and 2003
- root > Users > username >.od in Windows Vista and 7
- Users > username > .od for Mac users

To list the type of Personal settings and/or Survey defaults you have, please type in linux console l ~/.od/*DTECT_USER*

Please see the following sections for details: Personal Settings, Survey Defaults

Personal Settings

A user can update the *Personal Settings* by specifying a keyword (a variable name) and its value. The value column becomes editable if left mouse button is double clicked in a cell. A brief description on the available variables (and their values) is given below:



Note: Most of the changes are updated after restarting the OpendTect, other are active directly.

Pre-defined Keywords:
Press Select existing button to pop-up the pre-defined keywords. After selection of any variable, the values should be updated. The values are updated by pressing 'OK' button.

- **Company:** Write the company name. Keyword - Company, Value: Text (e.g. dGB Earth Sciences).
- **Default DATA directory:** It enables a user to setup a default OpendTect data directory. The value is a path pointing to a data directory e.g. /disk/ODData/.
- **Fonts:** The fonts for the OpendTect controls and graphics are set by selecting the Font.* or Graphic.* keywords. The wild card character (*) refers to the text followed by the Font/Graphic word. All of these key

words refer to a font dialog that may have a specific value. For instance, a value is written as Times New Roman`12`Normal`No [Font, Size, Style, Others]. The following keywords can have the similar values:

Font.def.Fixed width, Font.def.Graphics large, Font.def.Graphics medium, Font.def.Large control, Font.def.Small control, Font.for.Annotations, Font.for.Title, Graphics large font size, Graphics small font size.

- **Icons.size:** It is a size of OpendTect icons. Its value can be a numeric number e.g. 20, 24, 32 etc.
- **In-line byte:** This will enable the user to setup the inline byte location
- **Pre-stack Viewer Settings (3D PS Viewer.*)** With value=1,the Prestack viewer will change in autowidth mode
- **Bold font:** With Value set to 'Yes', the font is set to bold
- **Cross-line byte:** This will set up the default croosline byte location
- **Display UnitID:** This will enable the user to display the help window ID
- **MultiMachine.Nr inline per job:** A user may overrule the number of jobs (i.e. number of inlines) for each machine while running multi machine batch processing. By default the value is 3
- **Nr Processors:** It is a numeric number to specify the number of processors to be used for a processing. For instance, '0' means to use all processors
- **dTect.Measure LineStyle:** This is a standard settings for the distance measurement tool available in the Graphics Toolbar
- **Seg-Y headers:** Default file location to dump a SEG-Y headers. The value is the output path together with a file name (e.g. /disk/SGY_dump.txt).
- **SEG-Y.Examine.Number of traces:** Set a minimum numeric value (e.g. 100) as a default setting to examine any seg-y file
- **Ui.ToolTips.enable:** Refers to enable a tool tip guide. Set Yes/No in the value
- **dTect.Ask.*:** Set to prompt a warning message settings (Value: Yes/No) for several windows. This include a prompt while closing OpendTect, closing Attribute Set window, storing picksets, store session

- **dTect.Auto Attribute set:** Set an attribute name that will be auto loaded if the OpendTect is launched
- **dTect.Average even median:**
- **dTect.Color table.Name:** A default color table for the user. The default is Red-White-Blue. The value is a color table name
- **dTect.ColorBar.show on top:** Set a value to Yes/No to set the color table on top of OpendTect or not
- **dTect.ColorBar.show vertical:** Set a value to Yes/No to set the default orientation of the color table (vertical - Yes)
- **2D Viewer Settings (SeisView.*):** 2D Viewer Display Properties of Wiggle Variable Area

The following keywords can have the similar values:

SeisView.Display.WVA.Clipping, SeisView.Display.WVA.Display Null traces, SeisView.Display.WVA.Draw wiggles, SeisView.Display.WVA.Interpolate, SeisView.Display.WVA.Left color, SeisView.Display.WVA.Min Pix/Trace, SeisView.Display.WVA.Overlap, SeisView.Display.WVA.Reftime color, SeisView.Display.WVA.Reverse order, SeisView.Display.WVA.Reverse polarity, SeisView.Display.WVA.Right color, SeisView.Display.WVA.Show reftime, SeisView.Display.WVA.Var Area.

- **dTect.DTECT_SHOW_HELP:** This will allow the user to show ID on the help window icon
- **dTect.Disp.Default clip perc:** Set a default clipping value (1-100 %) to clip the seismic data
- **dTect.Default symmetry zero:**This will set the colorbar symmetry around zero
- **dTect.Dont load plugins:** This enables the User to load or not the plugins
- **dTect.Dump OI Menu:** This is a special setting to export a OpendTect scene. Set the value Yes/No to make this option available
- **dTect.Icons.size:** Set a default numeric value for OpendTect icon's size

- **dTect.ProcessVolumeBuilderOnOK:** This setting deals with Volume Builder setup. By default the value is Yes i.e the setup process the volume based on the available volume builder setup
- **dTect.Show inl progress:** Set Yes/No to show or hide the inline progress bar while displaying the data
- **dTect.Use VolRen shading:** Set Yes/No for volume rendering shading
- **dTect.Use shading:** Set Yes/No for shading
- **dTect.Zoom factor:** Set a value for wheel zooming factor. Default value is 1
- **dTect.KeyBindings.Default:**
- **dTect.Select Plugins:** This feature allows the user to select or not the plugins
- **dTect.VolRen shading:** Set Yes/No for shading
- **dTect.Use startup session:** This option enables the user to directly start a session when opening a specific survey in OpendTect
- **dTect.Maximum Visualization Texture Size:** Set the maximum size for the individual visualization data 'chunks'. Default is 4096 (though this may be decreased to 2048, 1024, 512...)

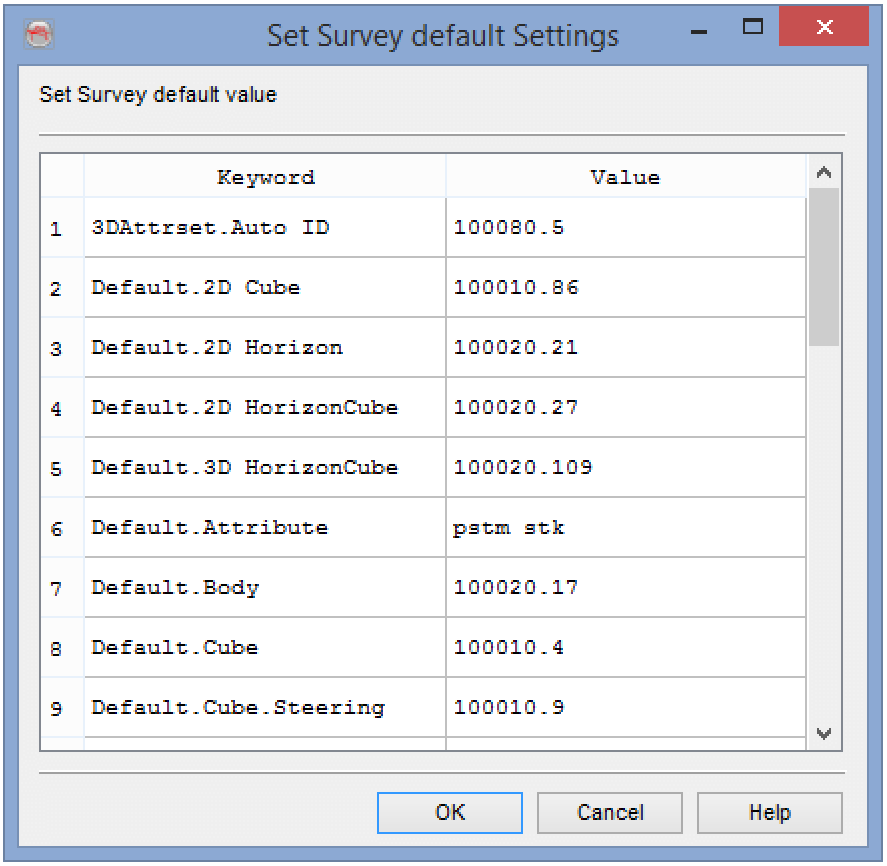
The following files are used to store the Personal settings:

- **settings.DTECT_USER:** This file is used to setup Personal settings
- **settings_coltabs.DTECT_USER:** This file is used to edit colorbar display properties
- **settings_snapshot.DTECT_USER:** This file allows the user to edit the snapshot properties like resolution, height, width etc ..
- **settings_flatview.DTECT_USER:** This file enables the user to edit the 2D viewer properties like Variable Density, Wiggle Variable Area.
- **settings_welldisp.DTECT_USER:** This file enables the user to edit the well display properties
- **settings_welltie.DTECT_USER:** This file is used to edit the Well to Seismic tie properties like Velocity/Density names, Wavelet ID etc..
- **qtsettings.DTECT_USER:** This file will allow the user to edit the QT properties
- **settings_3DPSViewer.DTECT_USER:** This file will allow the user to edit the 3D PreStack viewer properties
- **settings_GMT.DTECT_USER:** This file will allow the user to edit the GMT properties

- **survey.DTECT_USER:** This file is used to edit the Default Survey. Typing cds from a linux console will bring the user back to his/her default survey

Survey Defaults

This option enables the user to setup/edit the Survey default settings. The value can be changed by double clicking in a cell.



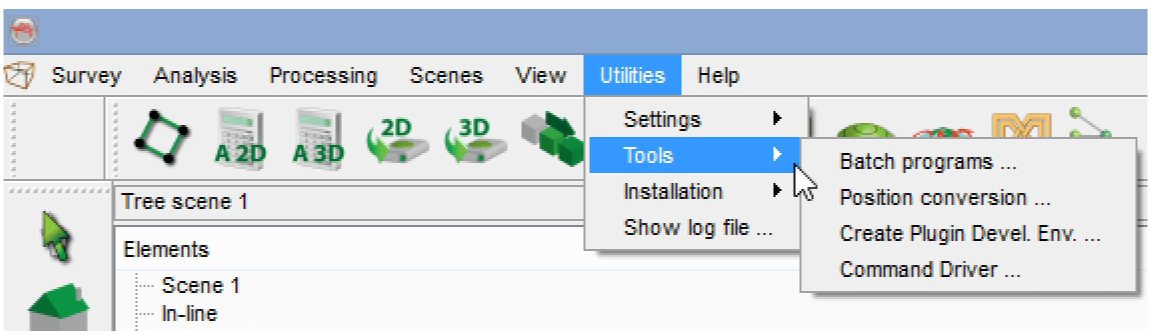
Depending on the user, several settings are available. Let us describe few:

- **3DAttrset.Auto ID:** This setting shows the ID value of the default auto-load attribute set
- **Default.Cube:** This will show you the default cube ID. If there is no value, this will mean that no Default Cube was set in that survey. To set a Cube as default, go to "Manage Seismic Data" in the main OD interface then click on the 'set as default'  icon after a cube has been

selected

- **Default.2D Cube:** This will show the ID value of the default 2D Data Set
- **Depth in feet:** Change the Value to 'No' if the Depth is in meter
- **SEG-Y Rev.1 policy:** When importing the SEG-Y,If the value is set to 1 the window will pop up asking a user to confirm the Rev1. With the value = -1 no confirmation pop-up window
- **Show depth in feet:** This option enables the user to show the depth in meter if the value is set to 'No'. Setting the value to "Yes' will display the depth in feet
- **Z Scale:** This shows the value of the survey Z-scale
- **Z Stretch:** This setting shows the ID value of the stretch

Tools



The four options located under '*Tools*' are explained in the following sub-sections.

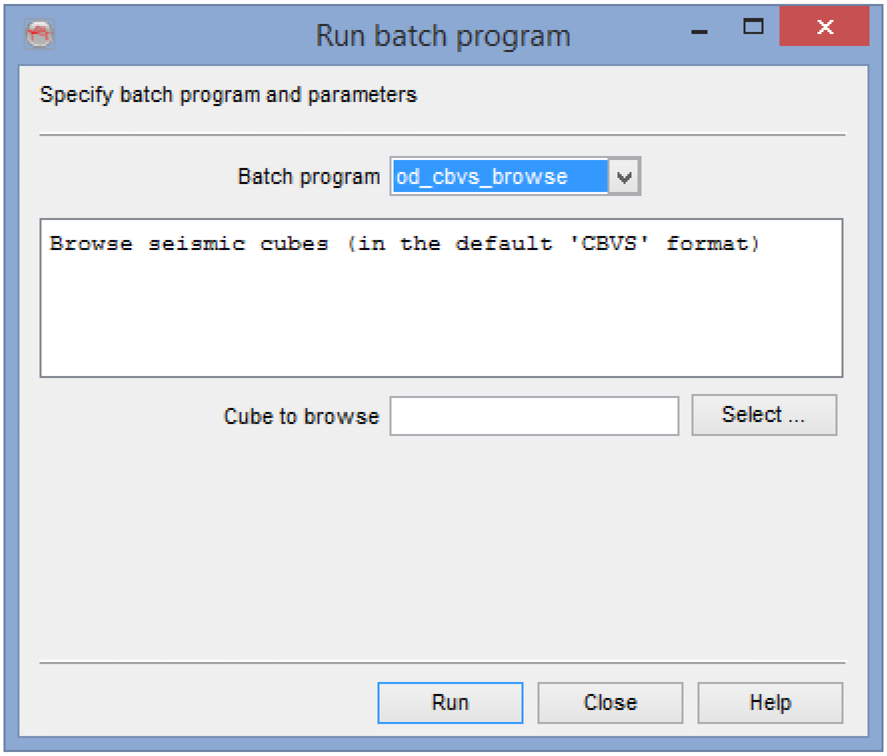
Batch Programs

To run the batch program go to: *Utilities* > *Tools* > *Batch programs*

Choose the batch program you need to run, the available are: cbvs_browse, cbvs_dump, lmhostid, glxinfo, ivfileviewer. The text box will show comments and details.

If *another OpendTect batch program is chosen*, fill in the required and (if needed) the optional parameters (indicated by the square brackets "[ ]").

The batch program will start in a new xterm window. For example, if a batch program is *cbvs_browse* like shown below, the cube to browse should be selected to run a batch program.

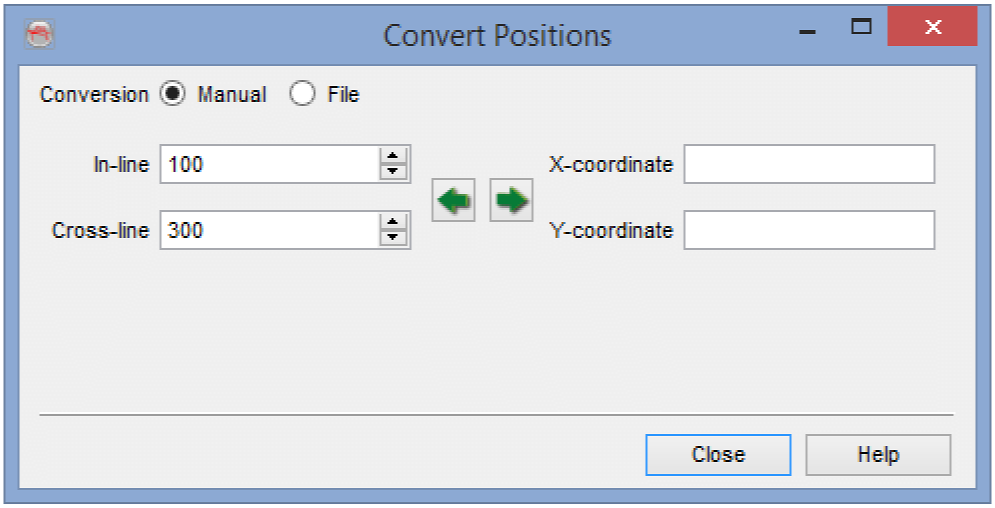


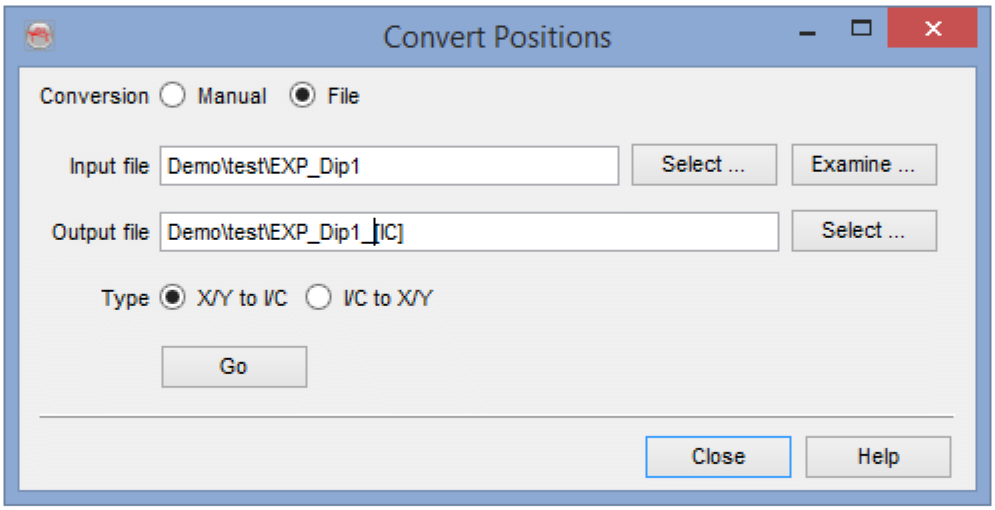
Position Conversion

Position Conversion is an utility that can be used to convert the position pairs from Inline/Crossline to X/Y, and vice versa. This utility can be launched either directly from *Utility > Position Conversion ...* or from Survey selection menu (*Survey > Select/Setup*).

In the position conversion window, there are two modes available for coordinate conversion: Manual/File. In *Manual mode*, the user specifies an inline/crossline pair (or X/Y pair), then press the corresponding arrow key to obtain the position in the other domain.

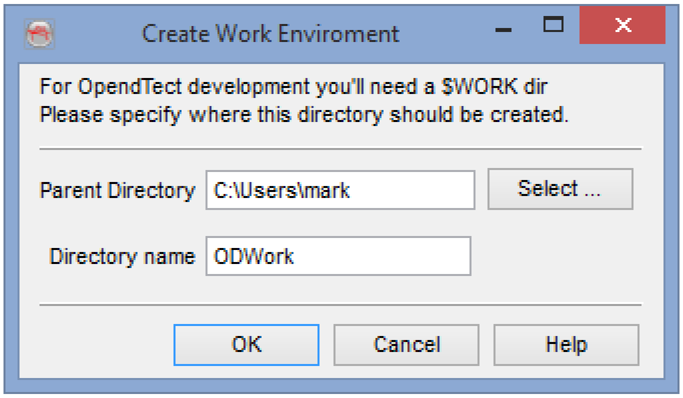
In *File mode*, the user browses the input file and create a new output file. By specifying the corresponding type conversion (XY to IC or IC to XY) and pressing the *GO* button, the desired conversion is written on output file.





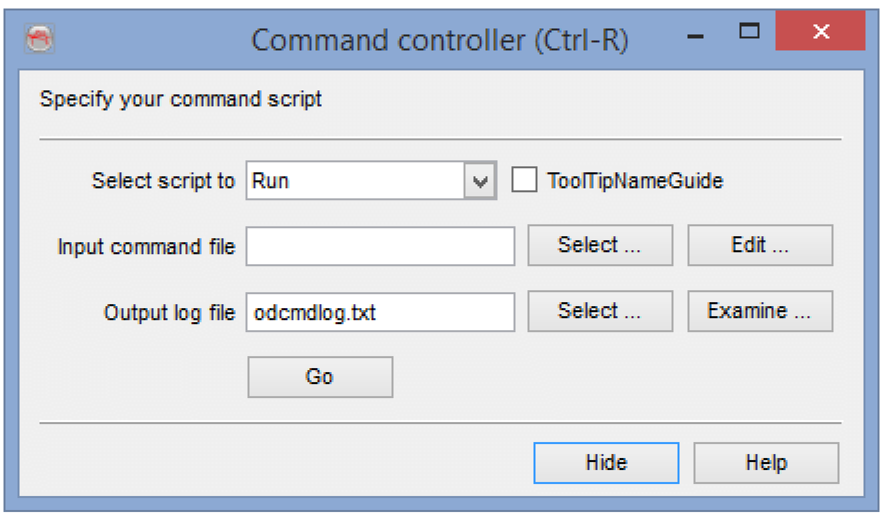
Create Plugin Development Environment

With this option the user can create the *OpendTect developer environment to develop a plugin to OpendTect*. The source code and all other relevant files are copied into a user specified directory (chosen from the Package Manager during the installation setup). More information can be found in the *Programmer manual* from the *Help* menu.



Command Driver

The *Command Driver* offers automated control of the current OpendTect application from a command script. The command script is a replacement for a series of keyboard and mouse interactions performed by the user. It can be used to automate parts of the work flow, and helps to speed up executing repetitive tasks or giving automated demonstrations in OpendTect.

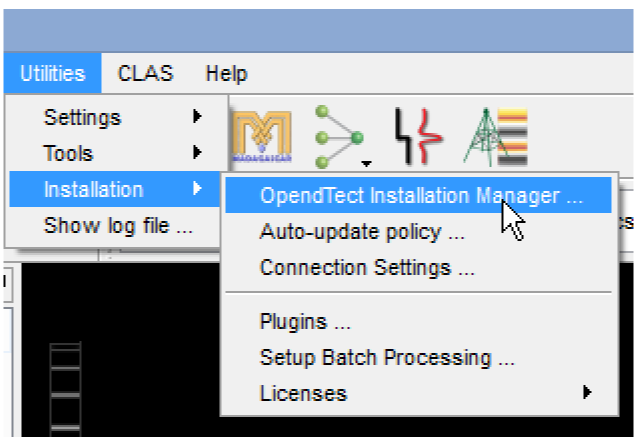


The Command Driver was created as a tool to make automated testing of the OpendTect releases possible. That means it is not optimized for usage as a scripting tool. It is clear, however, that power users have been starting to use the Command Driver tool for this purpose.

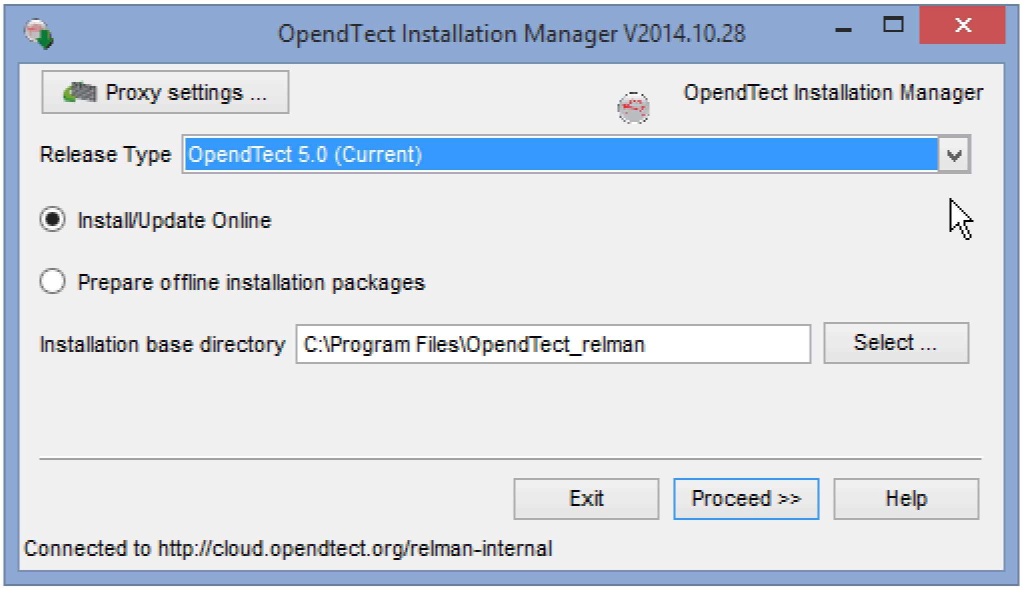
The list of available commands, their syntax, and semantics can be found in the Command Driver Manual.

Installation

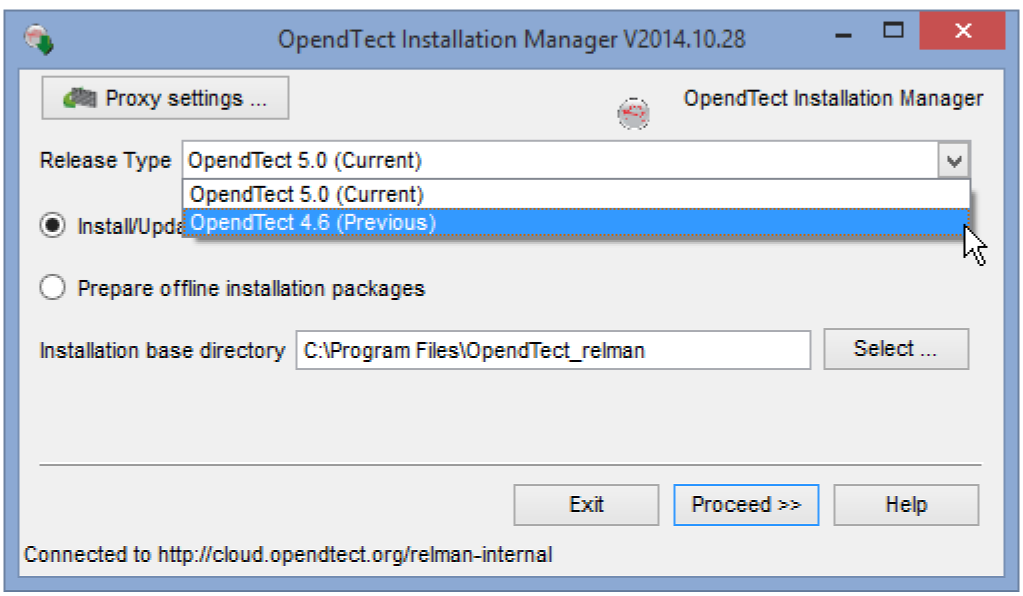
Table of Contents



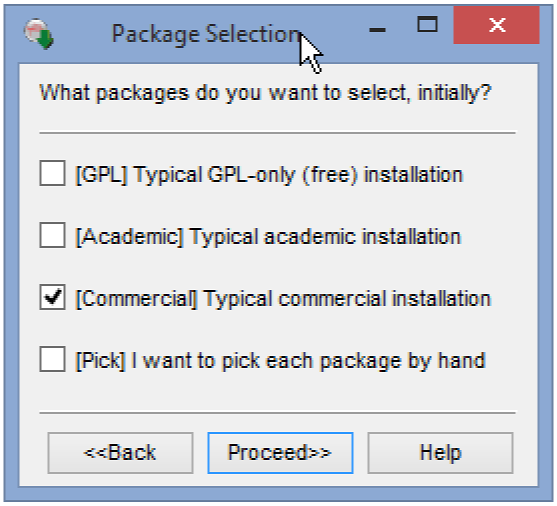
Installation Manager



The installation manager is a wizard to install/upgrade the existing OpendTect (Current / Previous) releases. The release type field is used to select the release that is needed to be installed/upgraded. In the new installer, you will have the choice as seen below:

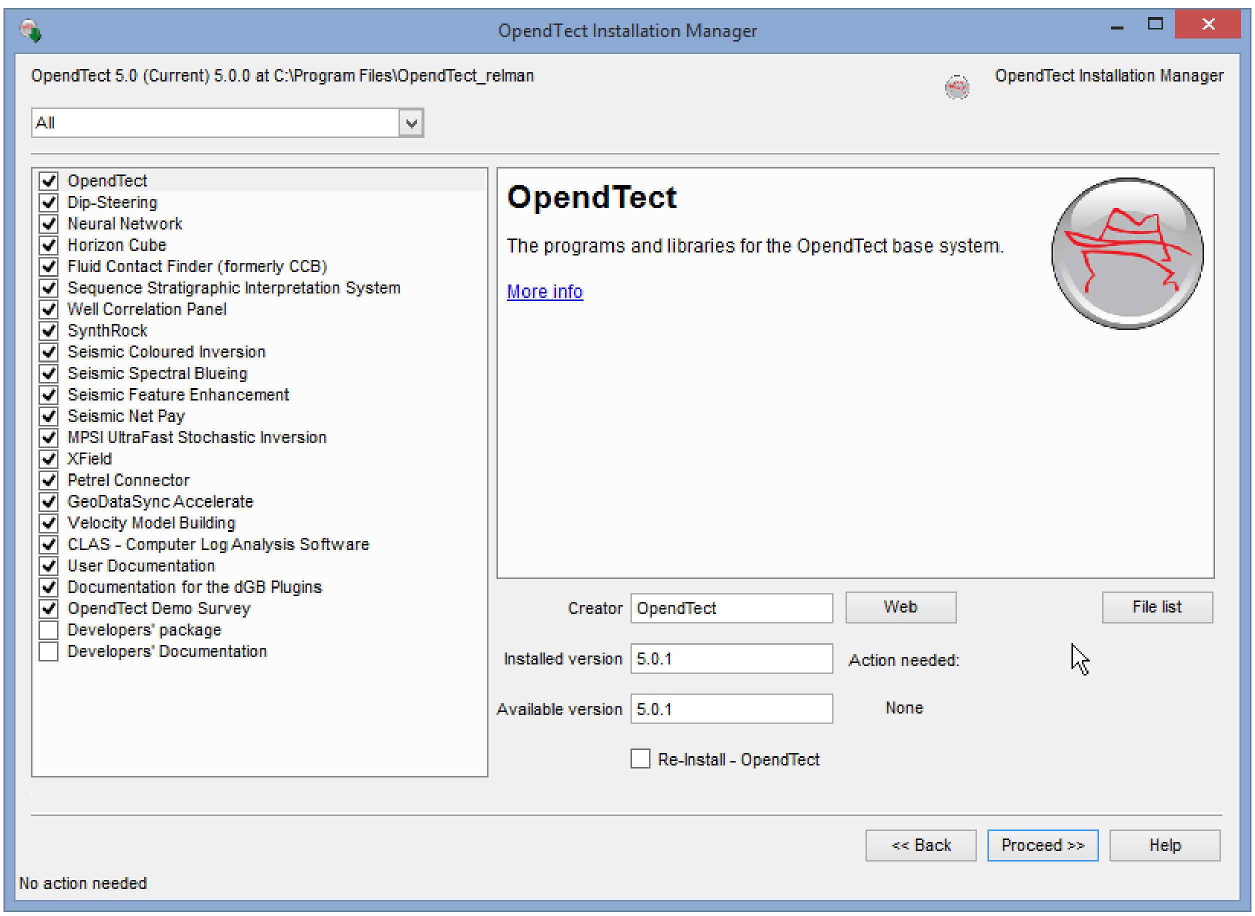


The information following in this section deals with online installation or upgrade. For creating offline installation packages, please see Offline Installation



The figure above suggests to select the package type of OpendTect. To read more about OpendTect packages type, please refer to our web-page of licensing types.

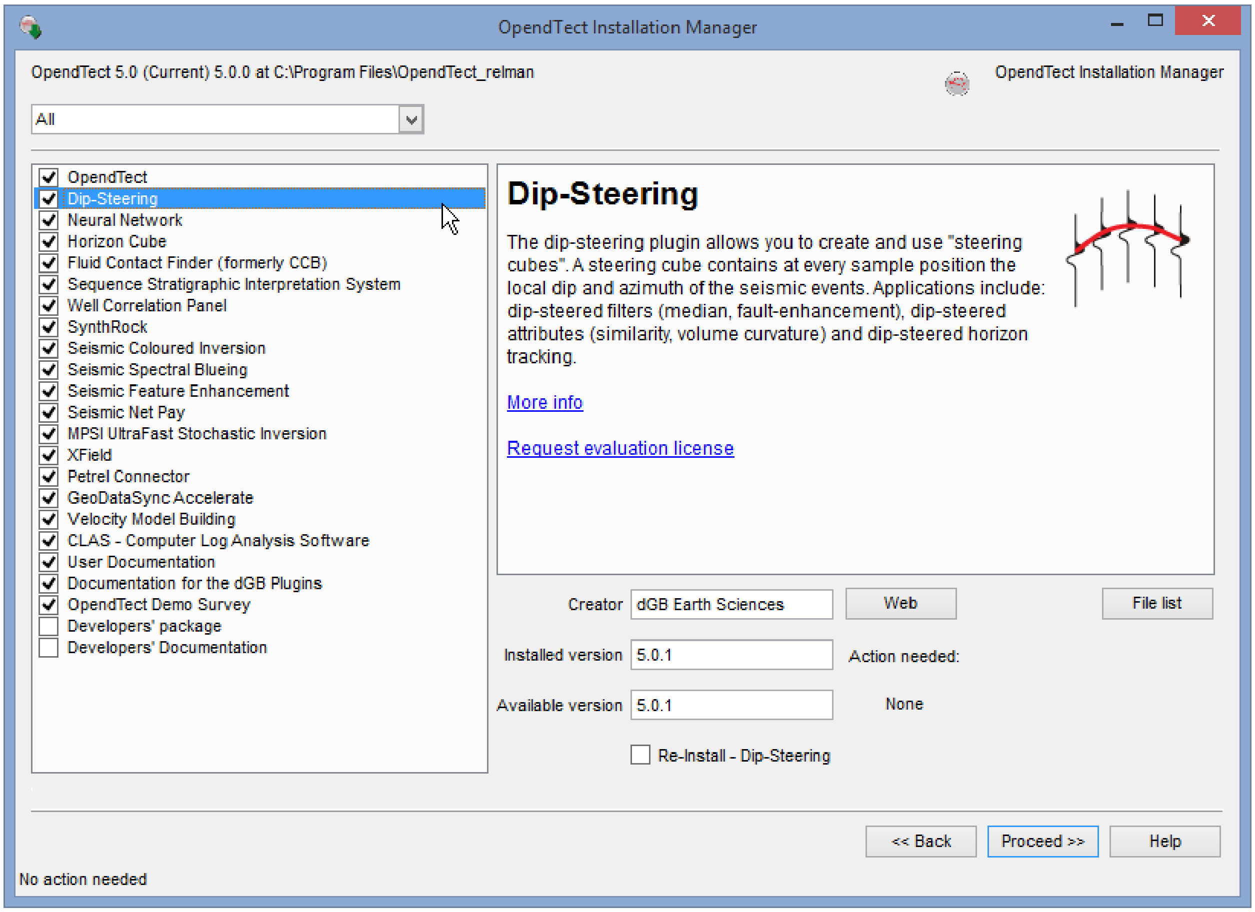
Package Manager



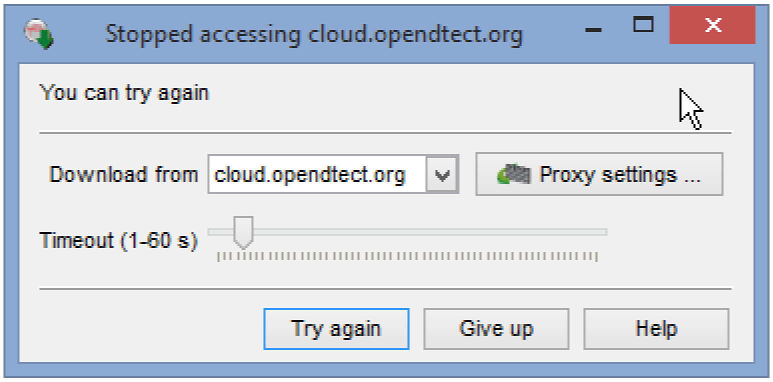
The last window of the wizard is the OpendTect Package Manager (see above figure). Multiple items can be selected from the list by checking the boxes (or not). Optionally, the relevant package combination could also be selected from the top list box.

The installation manager will automatically recognize the previously installed version at the selected path and will prompt it in the *Installed version* field.

To read more about a particular item in the list, select the item by clicking on it and read the description on to the right panel. For example, Dip-Steering:



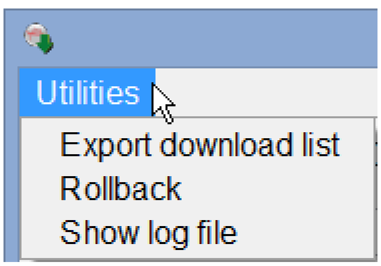
Note: If, for any reason, you should choose to abort the installation mid-download, you will see the following window appear:



This gives you various options, including increasing the time-out from its default setting, changing the download server or changing the Proxy settings.

Utilities Menu

On the top left corner of the package selection window there is a Utilities menu, which offers some useful functions for the installation manager:



The utilities menu has the following options:

1) Export download list
2) Rollback
3) Show log file

Export Download List

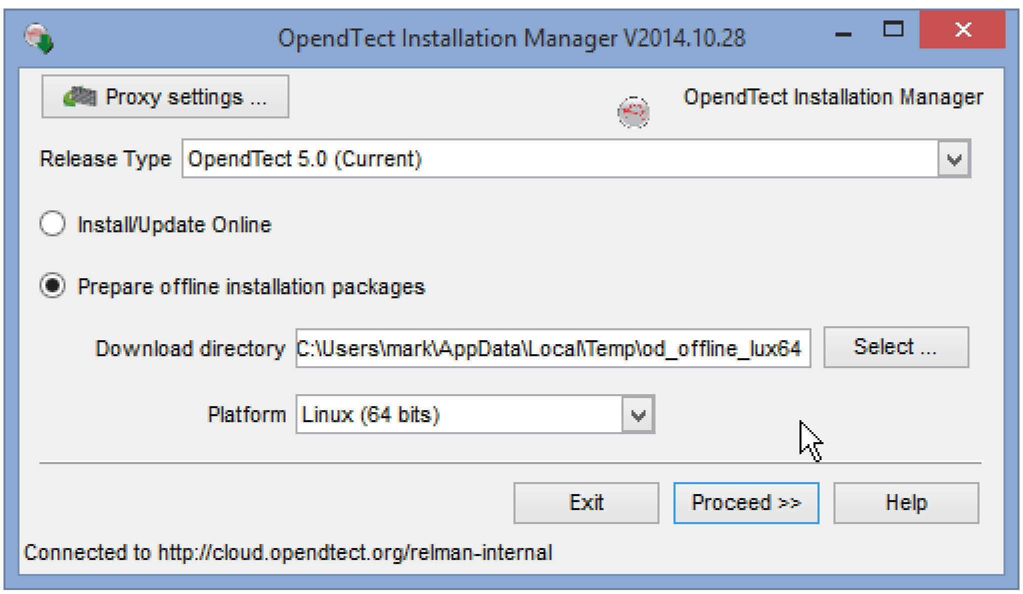
This option allows the user to download the list of URLs of the individual packages from the download site. This list is stored in a text file which can be used later to download these files directly without the help of the installer program. After downloading, user can run his/her own unzipping scripts to install the packages manually. This facility was only developed for the Linux users. Windows users can use this feature, provided they can prepare their own installation scripts for the installation.

Rollback

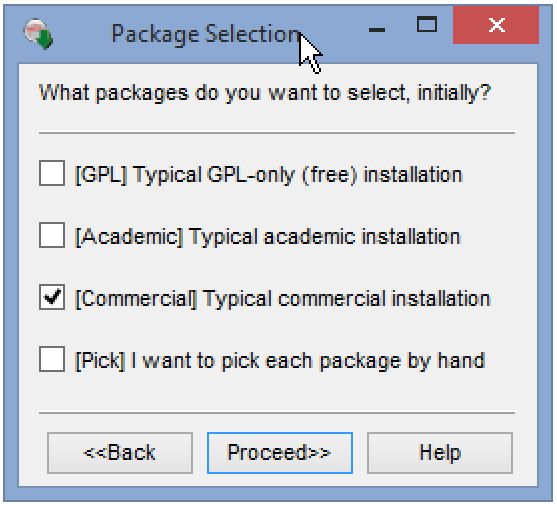
Rollback tool allows you to restore your previous version of the installation. If after updating the software you feel uncomfortable with some of the new features and want to go back to your previous installation, you have to use this tool. As this tool will change your entire installation so you have to use it cautiously.

Offline Installation

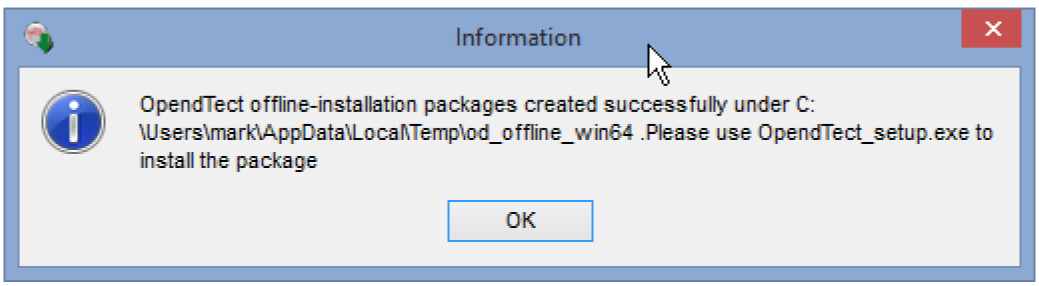
You may also choose to create packages for offline installation. These packages are created in such a way as to function cross-platform. For example, you may download the Linux 64bit package onto a Windows machine and then transfer and install it onto the Linux system or vice-versa.



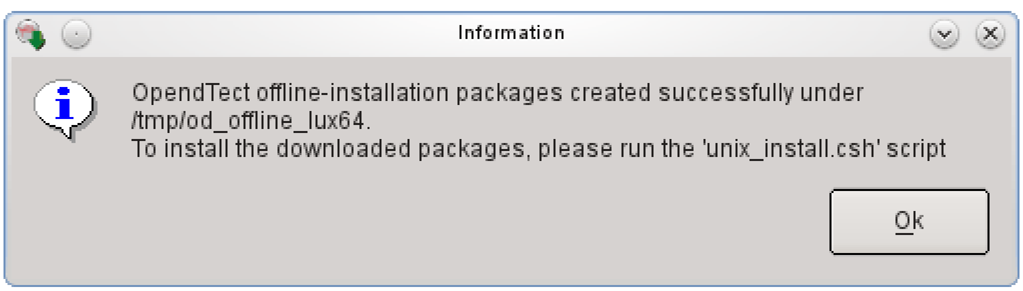
You will need to select the OpendTect version and toggle to 'Prepare offline installation packages'. You may either choose your download directory or leave the default.
You will first be prompted to select the package type of OpendTect. To read more about OpendTect packages type, please refer to our web-page describing licensing types:



On completion of the download, you will be reminded of the location in a pop-up window and informed of how to launch the installation package:



Windows offline pop-up info



Linux offline pop-up info

> **Note:** The 'Platform' option refers to the intended installation platform, and not the platform of the machine currently being used to download the packages (if different).

(For information on how to verify packages installed offline, please look here: Package Verification)

Package Verification

We generate signature files for all packages. Normally, a package is a zip file downloadable from our website:

For OpendTect version 4.6.0, please visit: http://www.opendtect.org/relman/4.6.0/
For OpendTect version 5.0, please visit: http://www.opendtect.org/relman/5.0.0/

For each zip-file, there is a zip.sig file containing a digital certificate that can be used to verify that the package has not been tampered with during transit.

To verify a package, download the corresponding zip.sig file and place it in the same directory as the package file. gpg (or pgp) must be used (an encryption program). These programs are normally installed in most linux installations, and can be found at http://www.gnupg.org

dGB's public key has to be downloaded to your keyring. This is only necessary for the initial verification. To obtain the key, use gpg itself:

```
gpg --keyserver pgp.mit.edu --search-keys ”support@dgbes.com”
```

You may chose any keyserver you want, as they all share data. Once you have located our key, import it to your keyring.
Secondly, to avoid warning messages, edit the key and tell gpg that you trust it:

```
gpg --edit-key support@dgbes.com
```

and then type "trust" as command. Once you have our key installed, you are ready to verify the packages. This is done by gpg:

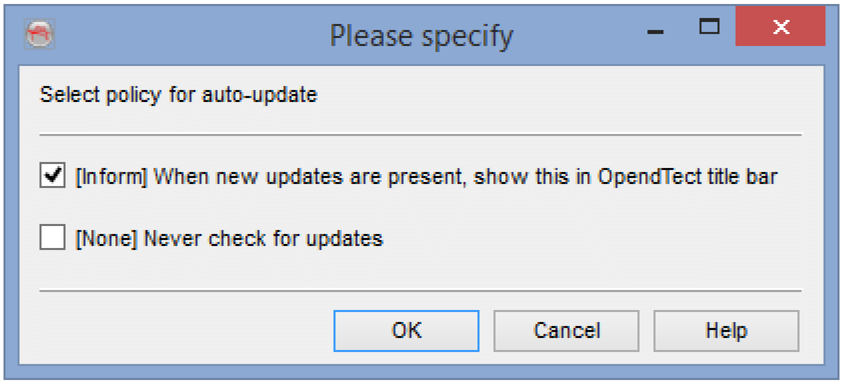
```
#bash-> gpg --verify demosurvey.zip.sig
```

This will check that demosurvey.zip has not changed since the file demosurvey.zip.sig was generated in our office. A positive output may look like this:

```
gpg: Signature made Thu Oct 4 08:46:01 2012 CEST using DSA key
ID A02F407E
gpg: checking the trustdb
gpg: 3 marginal(s) needed, 1 complete(s) needed, PGP trust
model
gpg: depth: 0 valid: 1 signed: 0 trust: 0-, 0q, 0n, 0m, 0f, 1u
gpg: next trustdb check due at 2022-03-13
gpg: Good signature from ”dGB Earth Sciences B. V. (Software
package signing key) ”
```

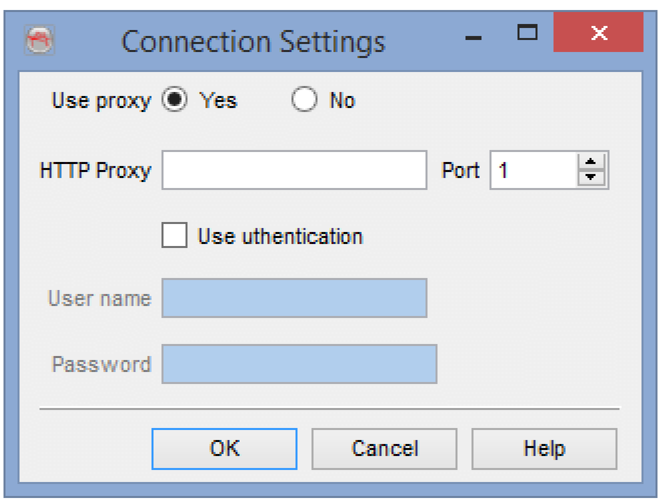
Auto-Update Policy

The auto-update policy can be defined and changed by a user. By default the option is set to Inform when the updates are available. This can be changed to Never check for updates if you prefer.



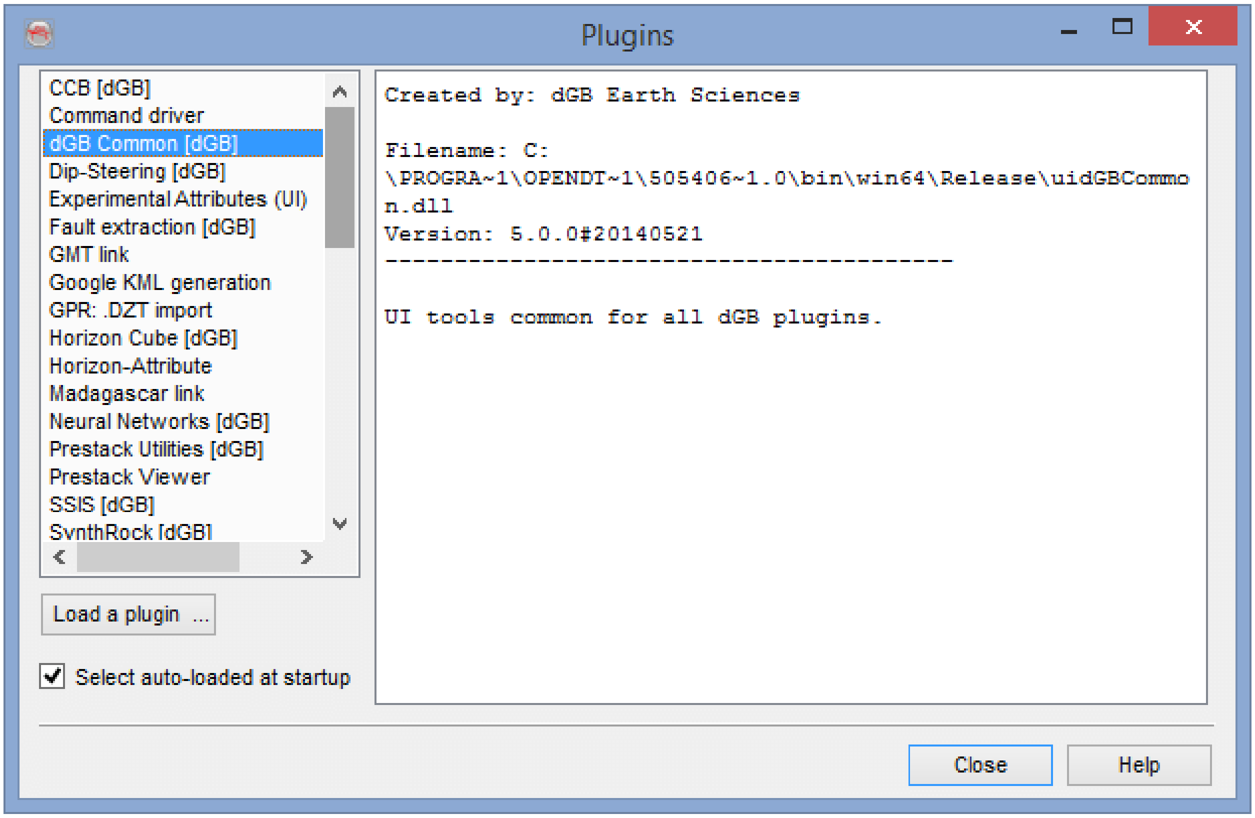
Connection Settings

To enter the proxy information, the correct proxy server information must be added in the *Connection Settings* before running the installation. This is done in the following dialog. This dialog is also available directly through the Installation Manager on clicking the *Proxy Settings* button.



Plugins

The plugins window lists the plugins that are currently loaded (or not) into OpendTect, and provides relevant license information.

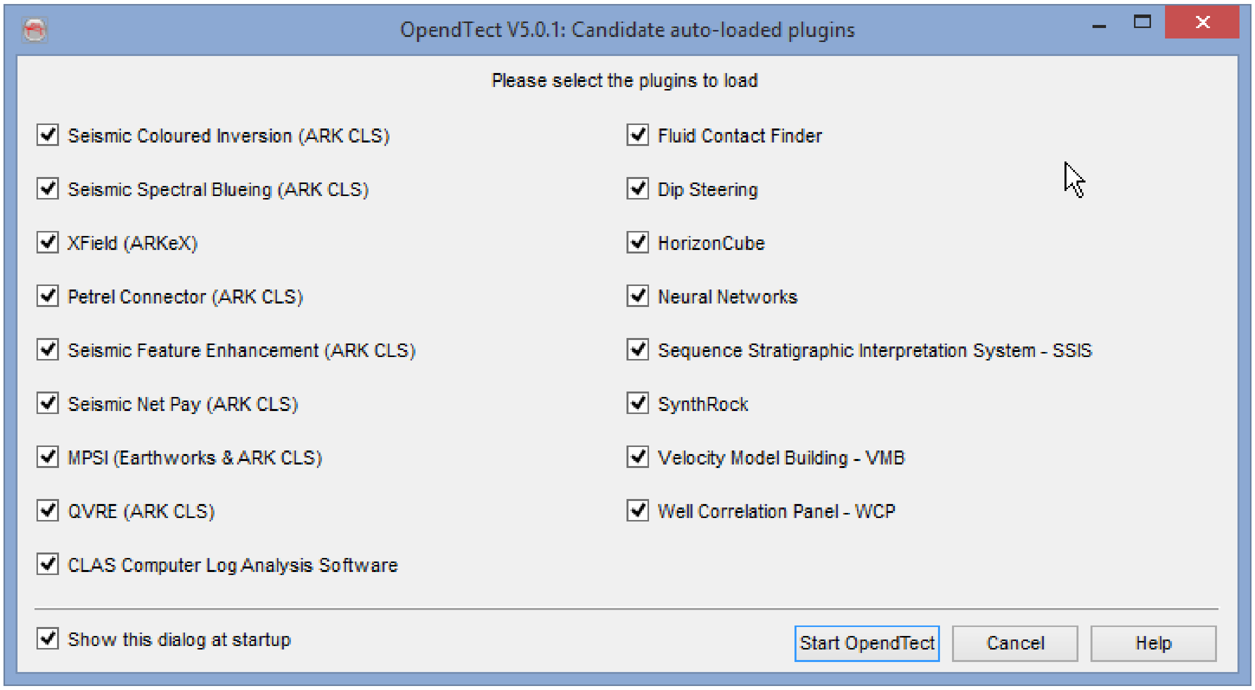


re[

Developers might want to use the option "Load a plugin" to manually load their plugin. The developers documentation describes how to add a plugin to the automatic loading procedure.

In OpendTect, there are several commercial plugins available. Each plugin adds extra functionality to OpendTect. To load a new plugin, browse to the appropriate file. More information on plugin design is available in the *Programmer manual*.

In general most plugins are loaded automatically at startup, based on the chosen options:

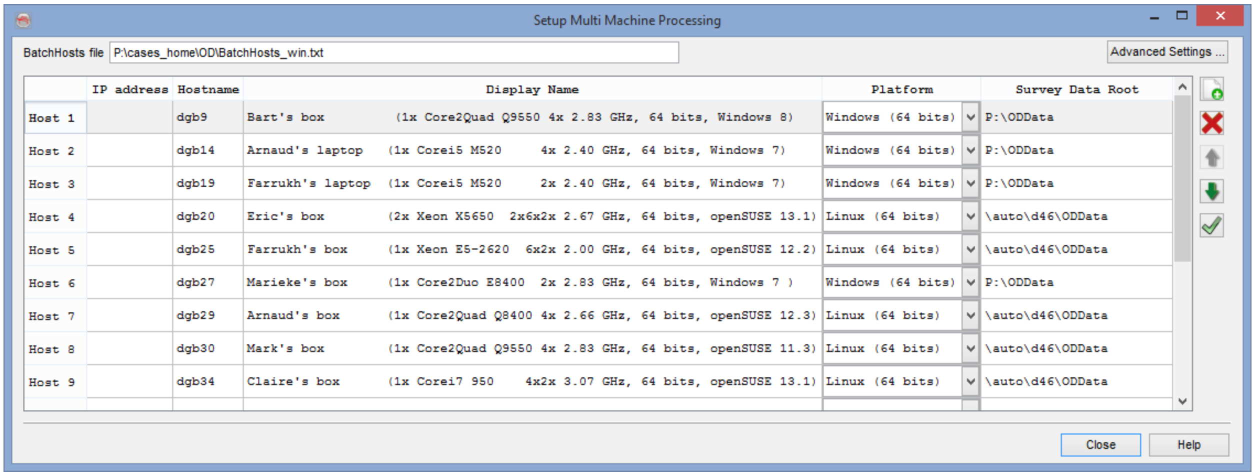


If you choose to toggle off the option "Show this dialog at startup" all plugins will be loaded at startup. It is recommend to install only the plugins for which you do have a license and to load them all automatically at startup.

Setup Batch Processing

In order to utilize OpendTect's capability for Multi-Machine Processing (MMP), a *BatchHosts* file must be used. This file must contain the list of remote machines (*host machines* or *nodes*) and some relevant details about these machines. Two BatchHosts_ example files come with the installation inside the OpendTect *data* folder, which may be amended for use. OpendTect will use this file to communicate to the remote hosts and launch processes remotely on them. Follow the example file format to add the list of remote machines and their details in the respective fields.

In order to minimize complications, the Setup Batch Processing tool can be used to create a tailor-made BatchHosts file (via *Utilities--> Installation--> Setup Batch Processing...*):



This tool allows the user to enter the details necessary for functioning MMP. On pressing 'Close', the file is saved. These include:

IP address: IP address of the node machine(s)

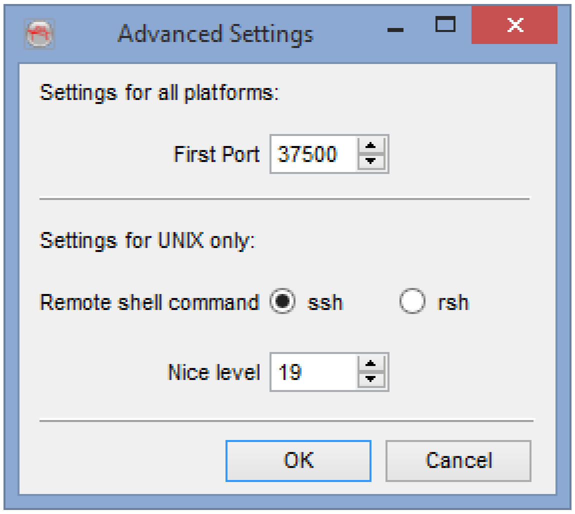
Hostname: Hostname of the node machine(s)

Display name: Free-text field. Text entered here appears in the Multi-Machine Processing window.

Platform: Select platform type.

Survey data root: Location of the survey (the path to the survey data root folder from the host machine)

Advanced Settings: Here you may change the first port value (in the case that it is blocked/in use). Linux users may decide to change the shell command from the default *ssh* to *rsh*. The *Nice level* sets the priority on the host machines, 19 being nicest and 1 being least nice).

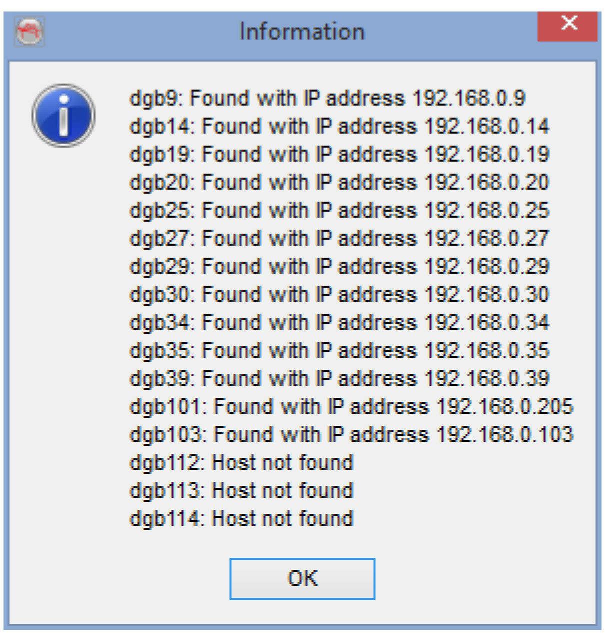


Add new host.

Remove selected host.

Move host up or down.

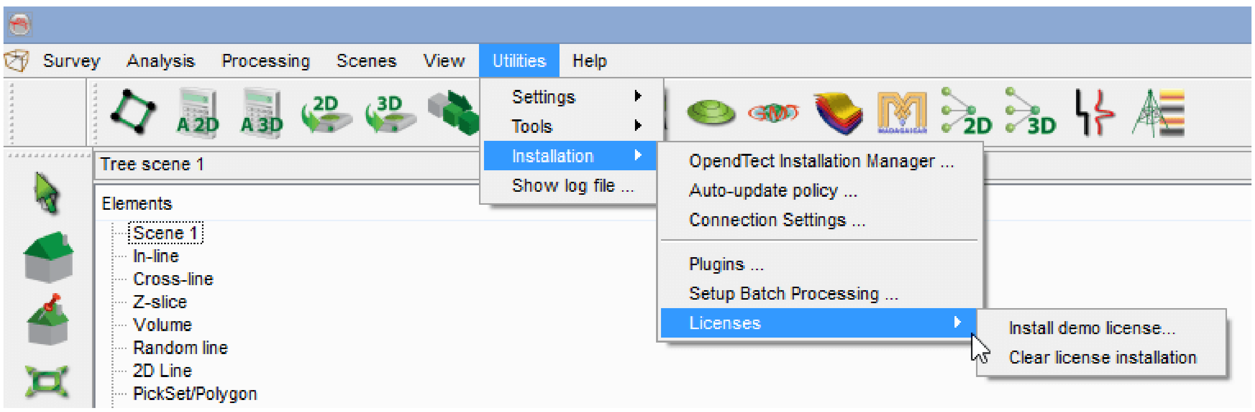
Test hosts. Will perform tests to ensure that the server and nodes can communicate to the necessary extent to perform the MMP. (ie: can the nodes find the data root folder and read/write into it)



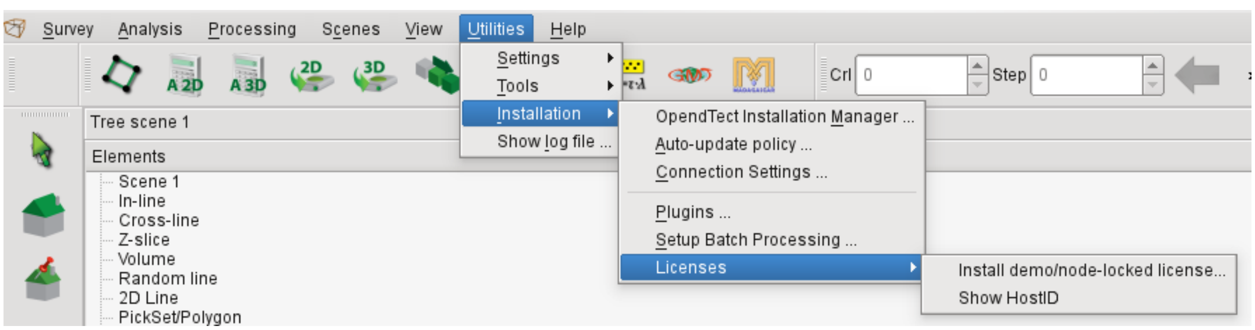
For more information on this topic, please refer to [OpendTect's YouTube Channel] where you may find the webinar: [Multi-Machine Processing Setup].
Alternatively, refer to the webinar on this page: http://opendtect.org/index.php/support/tutorials/tutorial-videos/webinars

Licenses

Under *Utilities--> Installation--> Licenses* you will see two options, differing per platform:



License options under Windows



License options under Linux

These three options are explained in the following sub-sections:

- Install demo License
- Clear license installation
- Show HostID

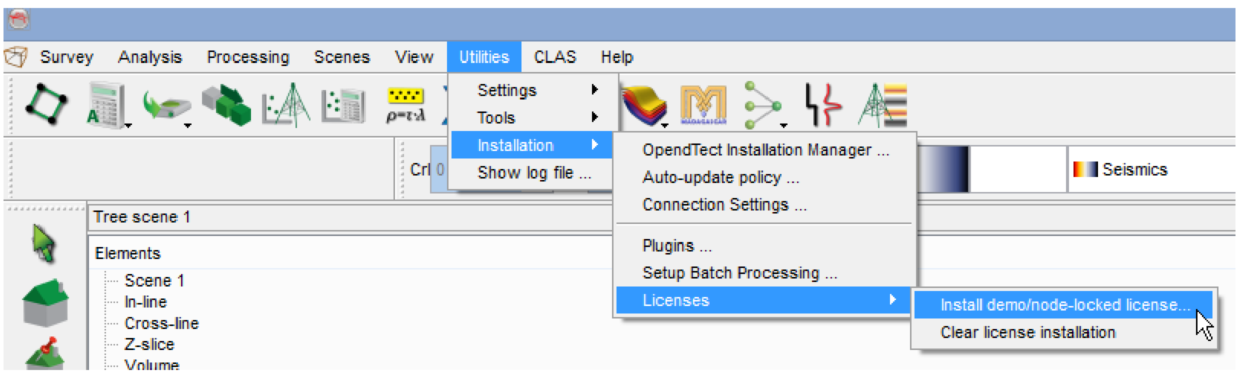
For information about floating or server-based licenses, please refer to: http://opendtect.org/lic/doc/flexnet_installation_guide.html

For more general information about OpendTect licensing options, please see: http://opendtect.org/index.php/support/licenses

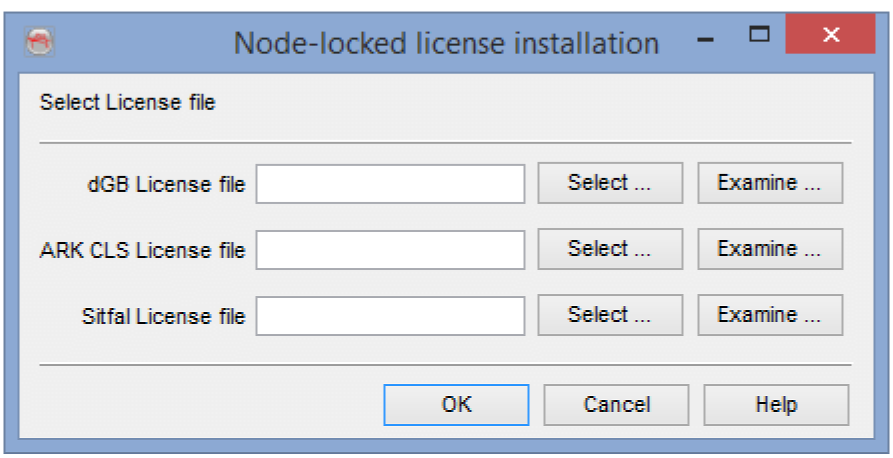
A more complete explanation of OpendTect license Installation can be found in the License Installation Webinar, available on OpendTect's YouTube Channel or via: http://opendtect.org/index.php/support/tutorials/tutorial-videos/webinars

Install demo/node-locked license

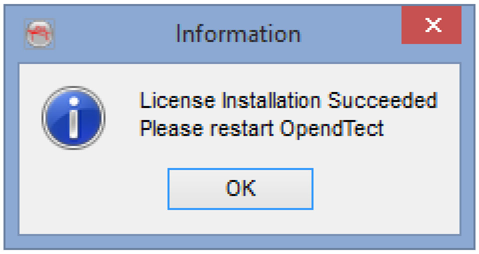
Plugins to OpendTect can be run either by using a license server or by using demo (evaluation) licenses. This second case is case called "node-locked license installation".



Use the following window to specify the path to the node-locked (demo/evaluation) license files that were given to you:



Here you can install each of the licenses by simply clicking ‘Select’, choosing the appropriate license file and clicking ‘Ok’ in the file selection window. Once you have selected all the licenses you are evaluating, click ‘*Ok*'. Your installation will be confirmed and you will be prompted to re-start OpendTect:

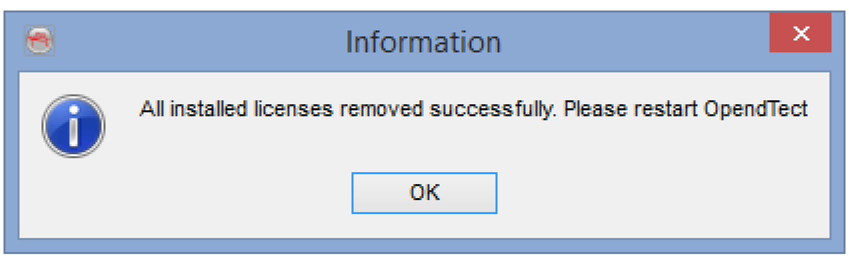


Clear License Installation

This option (Windows only) will clear:

- Demo or node-locked licenses installed via any route, including the 'Install demo license' option.
- Floating (or 'server') licenses that may have been installed (without stopping the license server).

Once cleared, you will be prompted to restart:



Users of Linux systems wishing to clear their license installation will need to do the following:

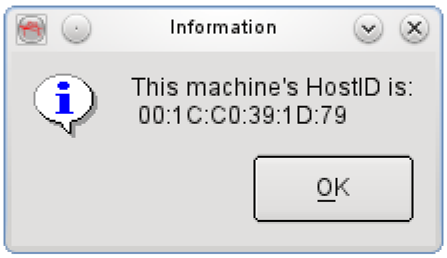
- Locate the .flexlmrc file in your HOME directory (eg: $HOME/.flexlmrc)
- Check in the file for specific lines referring to the OpendTect vendors (DGB, ARKCLS, SITFAL)
- If the file contains lines relevant to other software, then just delete the

individual lines. Otherwise, you may choose to delete the file.

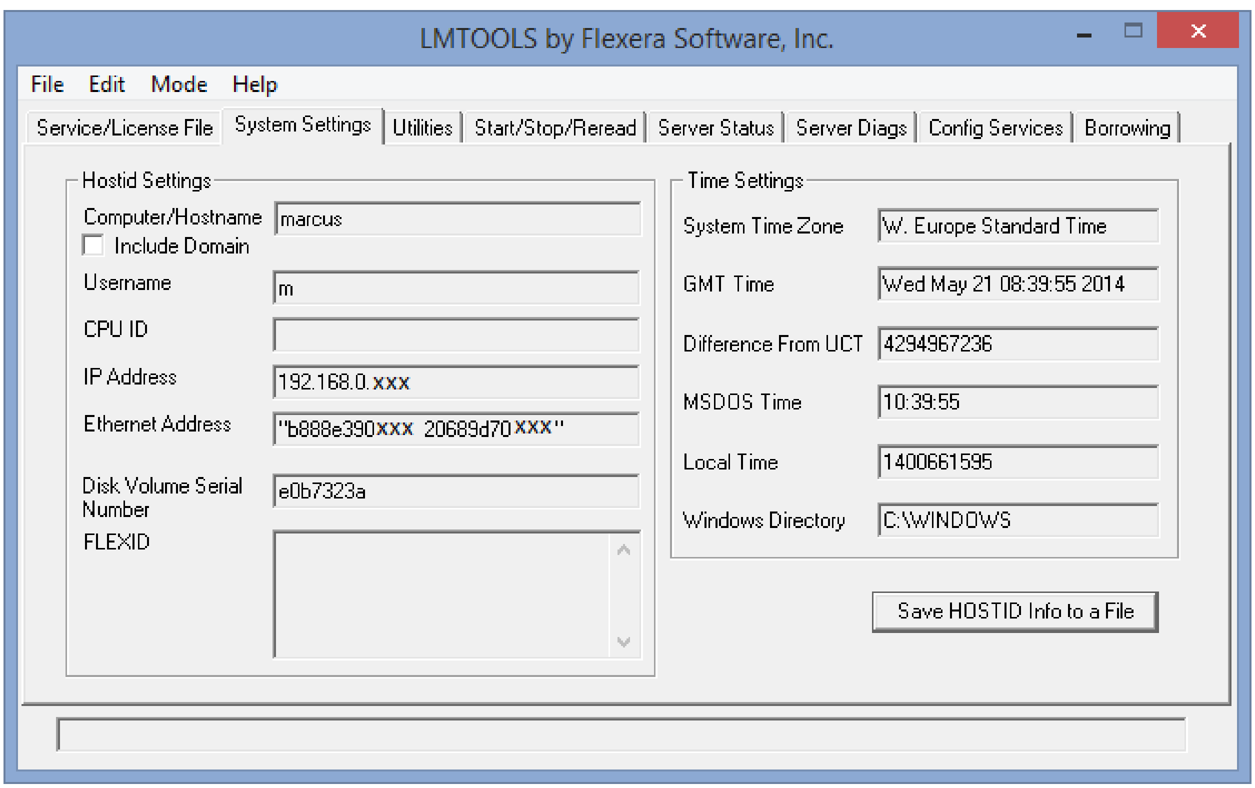
This method also applies to both demo/node-locked and floating licenses and will also not stop the server.

Show HostID

Clicking this option (only available on Linux systems) will pop up a simple dialogue showing the HostID of the machine:



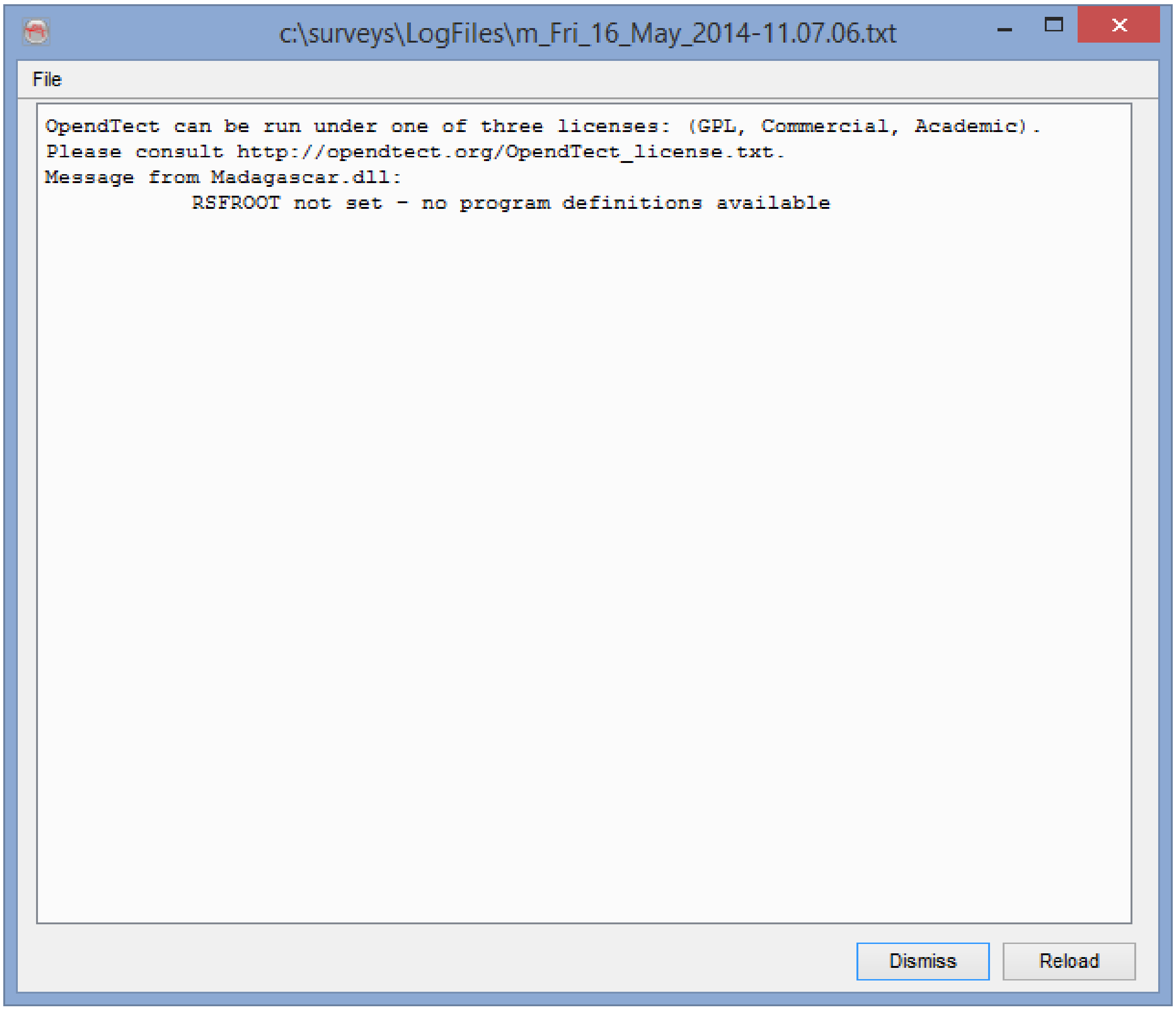
On Windows, accessing the HostID of the machine can be done via the LM Tools (available via the Start Menu or directly from ..\OpendTect\5.0.0\bin\win64\lm.dgb\lmtools.exe):



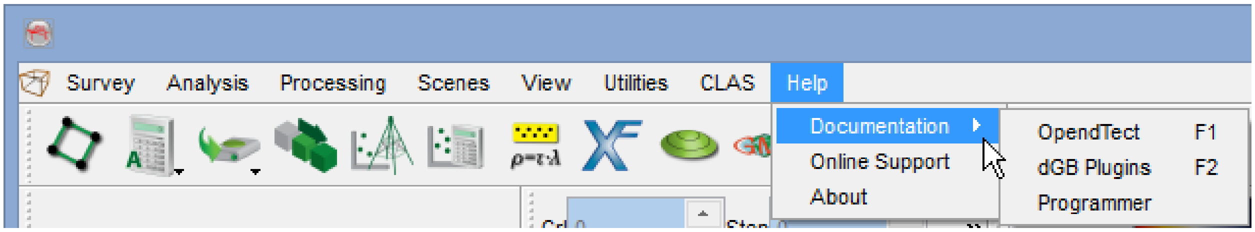
The option 'Save HOSTID Info to a file' will simply save the information displayed above into a .txt file for reference.

Show Log File

The user can check the log file from *Utilities* > *Show log file*. This will show the log of low traffic signals e.g. warning messages if a plugin (or license) is not properly loaded.

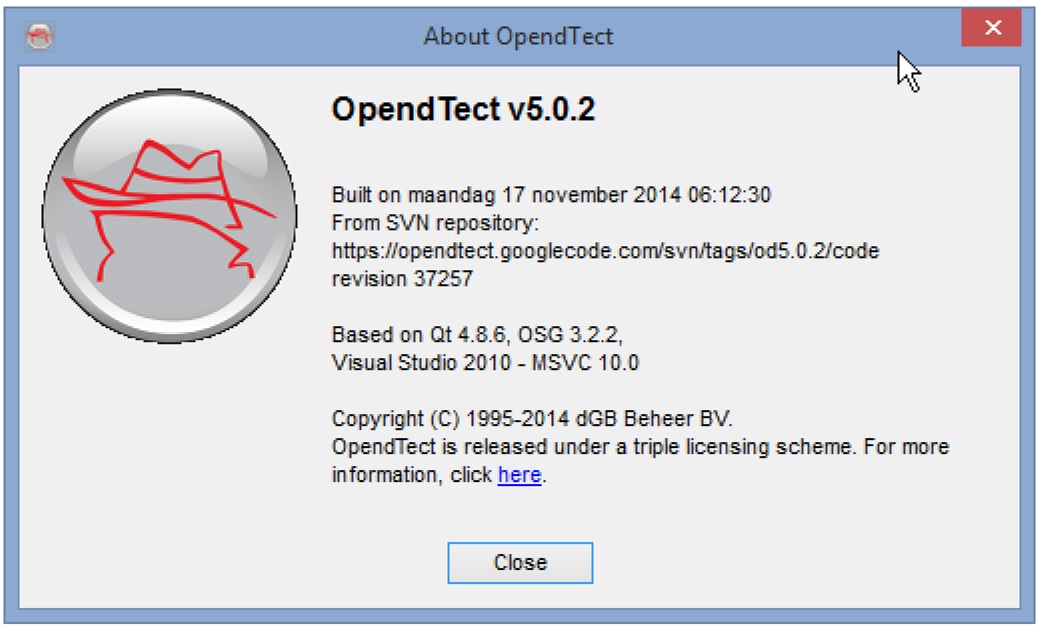


Help



From this drop-down menu, some of the various help aids can be accessed. This includes the *OpendTect User Documentation* (F1) and the *dGB Plugins User Documentation* (F2).

About gives you some basic information, as seen here:



Appendix A - Attributes and Filters

OpendTect attributes and filters are divided into 'dip-steering attributes' and 'no-steering attributes'.

A comprehensive list of all the attributes and filters available in the free, open source OpendTect core software is shown below (the so-called 'no-steering attributes'):

Note that other attributes and filters are only available if the corresponding plug-in has been installed. Information on commercial dGB plugins, please read the dGB Plugins Documentation.

Attributes related to Earthworks & ARK CLS plug-ins are documented in their corresponding UserDocs. These can be found by using following link:

> http://www.opendtect.org/index.php/support.html and clicking the *User-Documentation* icon.

Table of Contents

reflections (multiples)

- Grubbs Filter -- Attribute that removes outliers from normally distributed data
- Horizon -- Attribute that enables advanced calculations on horizons
- Instantaneous -- Attribute that returns a value at a single sample location
- Log -- Attribute that returns a well log value.
- Match Delta -- Attribute that extracts time shifts between similar events in different seismic volumes
- Mathematics -- Attribute that returns the result of a user-defined mathematical expression
- Position -- Attribute that returns any attribute calculated at the location where another attribute has its minimum, maximum or median within a small volume
- Prestack -- The pre-stack attribute can be used either to extract statistics on the gathers and their amplitudes, or to extract AVO attributes
- Reference -- Attribute that returns the definitions of the extraction position
- Reference shift -- Attribute that moves the extraction position in 3D space
- Sample Value -- Attribute that returns the input value at the sample location
- Scaling -- Attribute used for scaling of amplitude
- Similarity -- Multi-trace attribute that returns trace-to-trace similarity properties
- Spectral decomposition -- Frequency attribute that returns the amplitude spectrum (FFT) or waveletcoefficients (CWT)
- Texture -- Group of attributes that return statistical properties of a Grey-Level Co-occurrence Matrix (GLCM)
- Texture - Directional -- a multi-trace attribute that returns textural information based on a statistical texture classification.

- Velocity Fan Filter -- Attribute that returns energy with apparent velocities/dips inside a specified Min/Max range
- Volume statistics -- Attribute that returns statistical properties

CEEMD - Spectral Decomposition

Name

CEEMD (Complementary Ensemble EMD) – Group of Spectral Decomposition algorithms which are based on Empirical Mode Decomposition (Huang et al., 1998).

Description

The Empirical Mode Decomposition (EMD) algorithms implemented in OpendTect follow the work published by Jiajun Han and Mirko van der Baan (2013). Compared to Short Window FFT and Continuous Wavelet Transforms, the other two spectral decompositions methods supported in OpendTect, EMD methods reportedly yield significantly higher time-frequency resolutions.

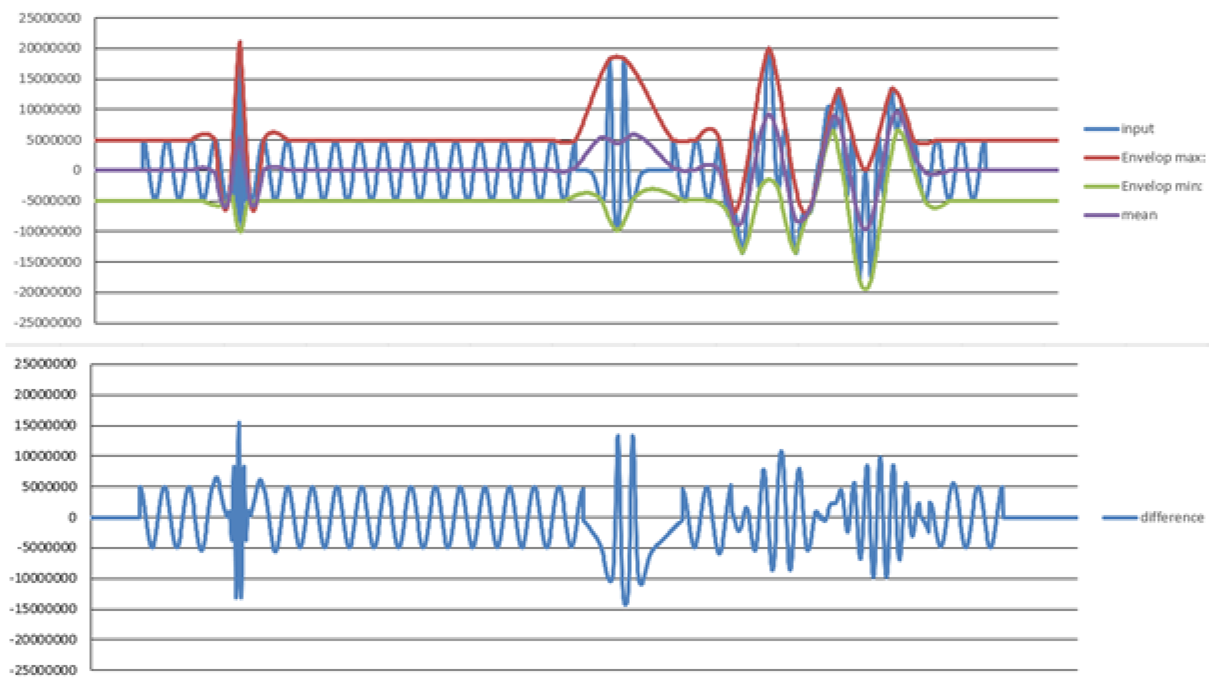
EMD is a data-driven spectral decomposition method developed by Huang et al. (1998). The method decomposes a time series (e.g. a seismic trace) into a set of intrinsic oscillatory components called Intrinsic Mode Functions (IMF's). An IMF is defined as a function that satisfies two conditions:

1) The number of extrema and the number of zero-crossings must either be equal or differ at most by one.

2) At any point the mean value of the envelope defined by the local maxima and the envelope defined by the local minima is zero.

These conditions ensure that IMF's contain only one mode of oscillation per cycle whereby a cycle is defined by the zero crossings. No riding waves are allowed. Riding waves can result in negative instantaneous frequencies, a major problem in any application that relies on instantaneous frequencies. An important characteristic of IMF's that is utilized in EMD is that instantaneous frequency can be defined everywhere.

To decompose the signal into the IMF components the algorithm performs a process called sifting. In sifting the local mean of the signal is subtracted from the signal. The local mean is computed from the envelopes (below). If the difference signal fulfils the IMF conditions defined above the first component is found. This will be the component with locally the highest frequencies. This component is subsequently subtracted from the original signal and the process is repeated until all components have been found. The last component contains the lowest frequencies, or represents the trend.



The sifting process. Envelopes of the signal are constructed by fitting a polynomial function through picked minima and maxima. From the envelopes the local mean of the signal is computed, which is subtracted from the input signal. It is then checked whether the difference signal (bottom) meets the IMF conditions. If not, the sifting process is repeated until the IMF conditions, or the sifting stopping criteria, are met.

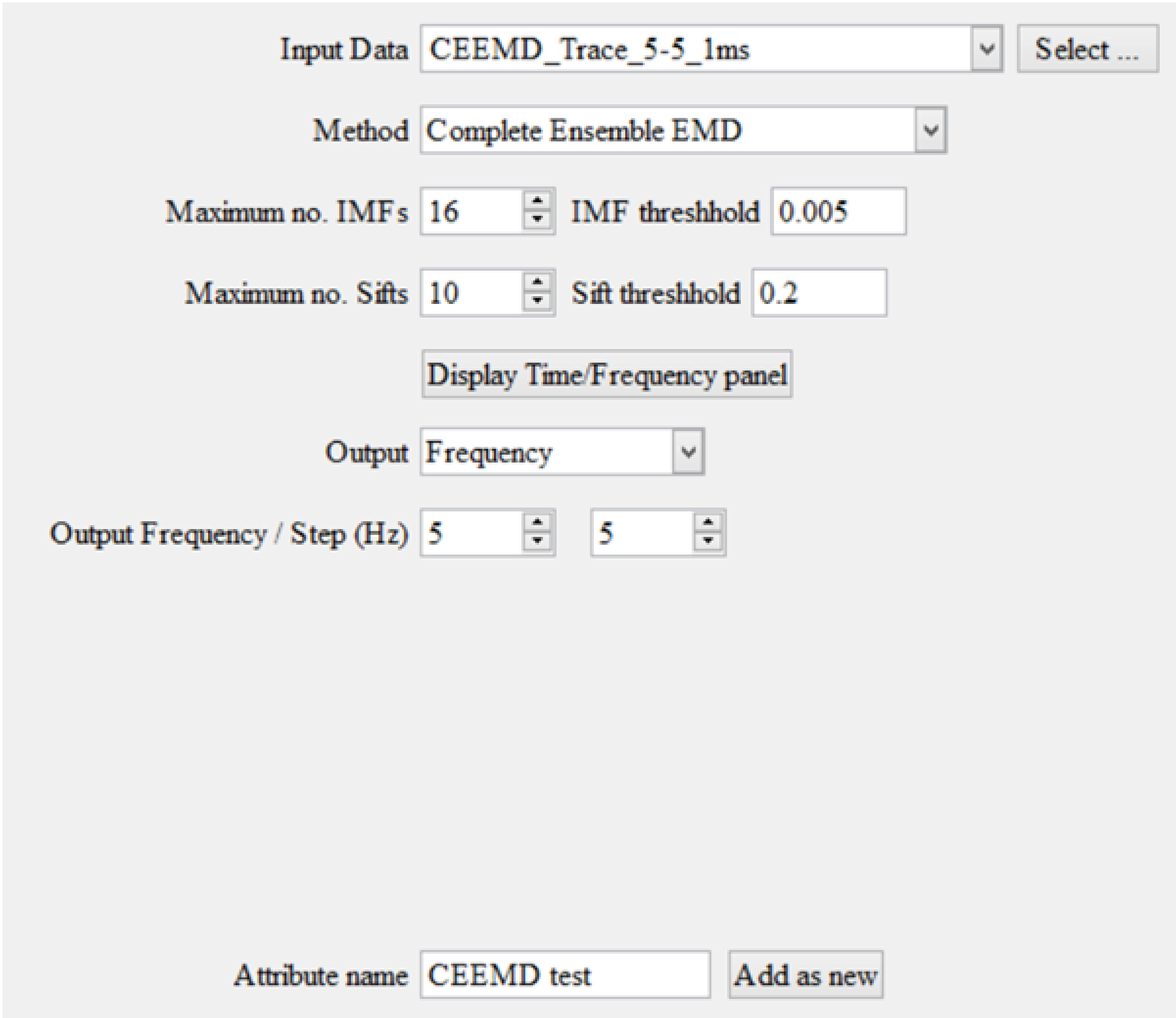
EMD is a relatively slow decomposition method and it has a problem called mode mixing. This is defined as either a single IMF consisting of widely disparate scales, or a signal of similar scale captured in different IMF's.

To overcome mode mixing two noise assisted methods have emerged, both of which are supported in the OpendTect attribute.

Ensemble Empirical Mode Decomposition (EEMD) adds a fixed percentage of white noise to the signal before decomposing it. This step is repeated N times after which all results are averaged. EEMD improves the mode-mixing problem but it cannot completely reconstruct the input signal from the resulting components.

Complete Ensemble Mode Decomposition (CEEMD) is also a noise-assisted method. Similarly the method decomposes the signal with N different noise realizations but here the results are averaged after each component is found. CEEMD solves the mode mixing problem and it provides an exact reconstruction of the input signal. CEEMD is however, a CPU intensive process and in the current implementation rather slow.

Input Parameters



Input Data is the seismic data to be decomposed. The current implementation decomposes the entire trace, hence decomposition along one horizon takes as much computing time as a decomposition in batch mode of a cube over the same area.

> **TIP:** To evaluate the results along one horizon, e.g. with RGB blending, it is faster to run a batch process on a time slice that encompasses the horizon-slice than to run the job interactively using the evaluate attribute option.

Method: Select the decomposition algorithm:

EMD – slow and possibly suffers from mode-mixing (see Description above).

EEMD – slower but partly solves mode-mixing, however signal cannot be reconstructed exactly.

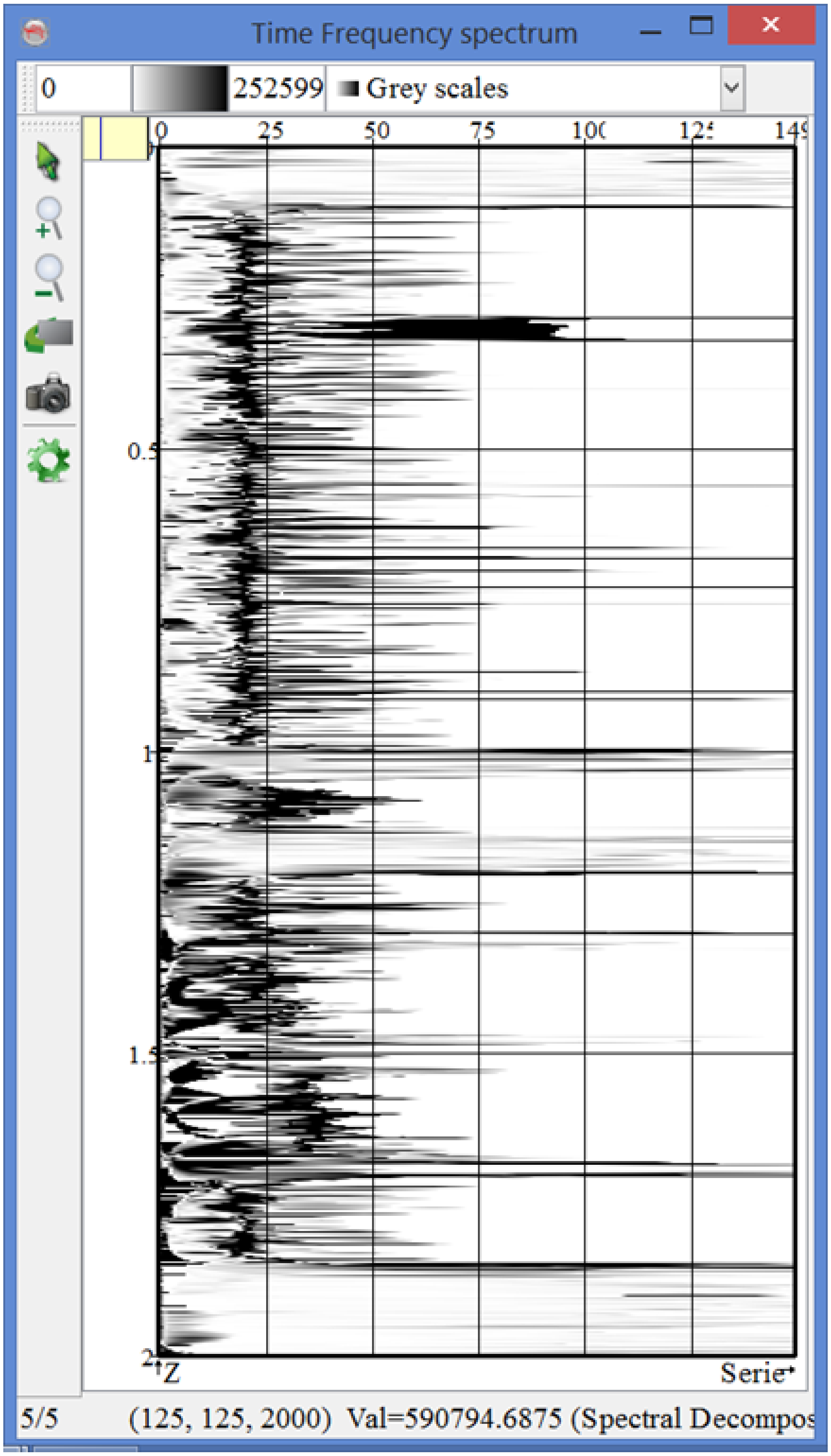
CEEMD – slowest but solves the mode-mixing problem and the signal can be reconstructed exactly from the components.

Maximum No. of IMFs is the maximum number of components into which a signal can be decomposed.

IMF threshold is a value below which the decomposition process is stopped. The value is computed as the standard deviation of the component divided by the standard deviation of the input signal.

Maximum no. of sifts and Sift threshold are stopping criteria for the sifting process. The Sift threshold is defined as the standard deviation of the signal after the sifting step divided by the signal before the sifting step. The typical range is between 0.2 and 0.3

Display Time Frequency Panel will decompose the selected trace and displays the result in a time-frequency plot with frequencies ranging between 0 and Nyquist Hz.



Time-Frequency plot of the synthetic test trace, below (zoomed in to show only frequencies from 0 to 150Hz.)

Output

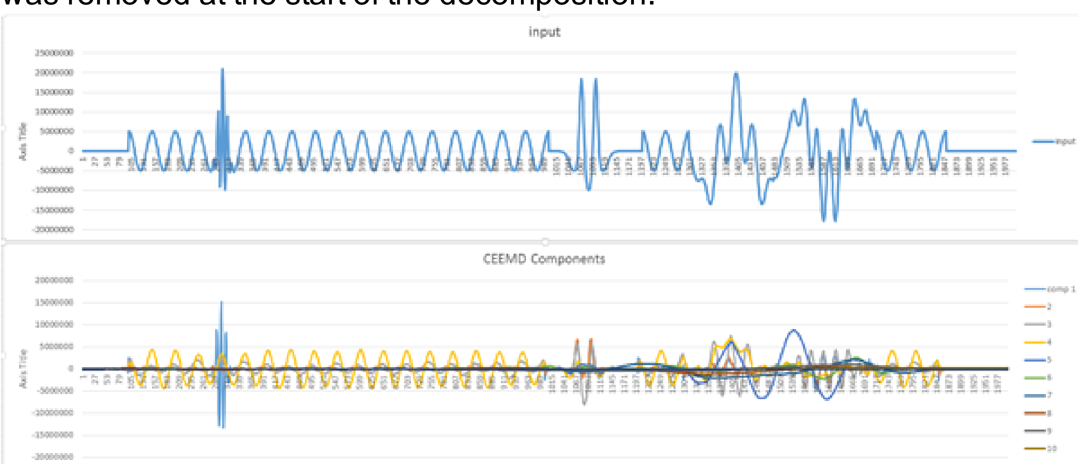
The current version supports four output Attributes:

Frequency This is computed from the instantaneous frequency of the IMF components and by re-gridding these irregularly sampled frequencies to the requested output frequencies. Output Frequency is the frequency that is output if the attribute is computed interactively. In Batch mode you can output all frequencies from Output Frequency to Nyquist with an increment of Step, e.g. 5, 10, 15, …, 120, 125 for an Output Frequency of 5, a Step of 5 and an input signal with 4ms sampling rate (Nyquist is 125 Hz). Please note that computing one output (interactively) takes nearly the same amount of computing time as computing all frequencies (in batch mode).

Peak Frequency This is the frequency with the largest amplitude in all IMF components. It captures information from the spectral decomposition into a single attribute that is related to tuning effects at varying thicknesses.

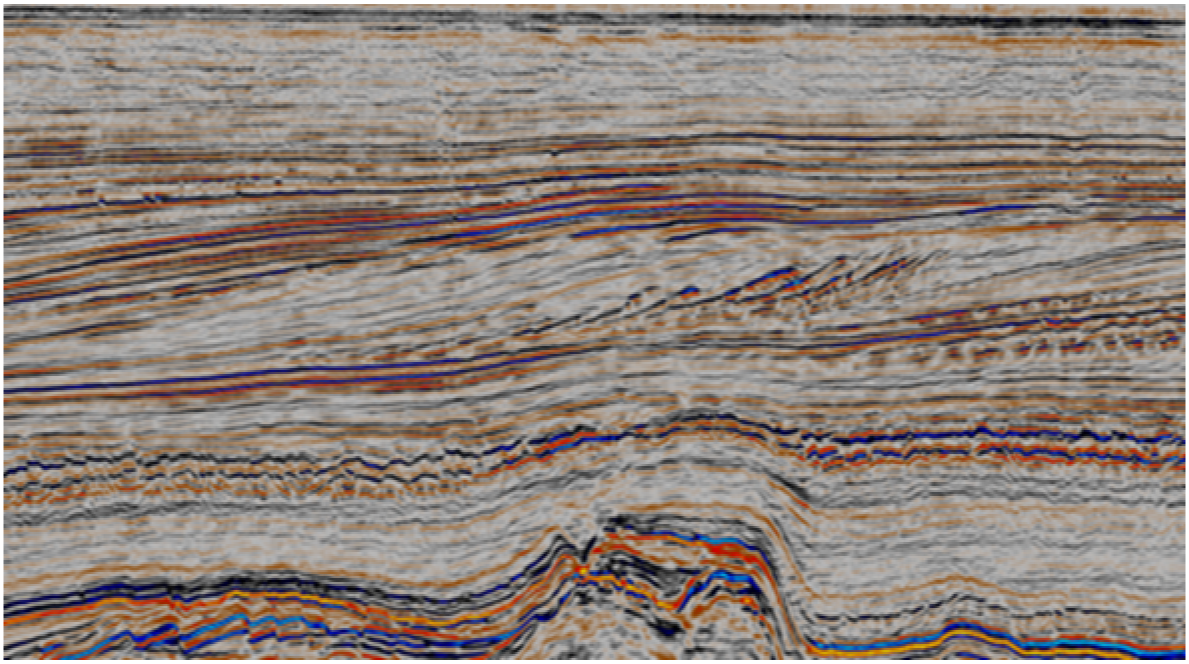
Peak Amplitude This is the highest amplitude in all components.

IMF Component Outputs the IMF components (below). This corresponds to the real parts of the decomposed signal. The first component corresponds to the highest frequency oscillations in the signal. A decomposition may result into an unknown number of components. When running a job in batch mode the output is stored in a multi-attribute file (cbvs format) with N+1 number of attributes where N is the maximum no. of IMF's. If a trace is decomposed into less than N components the remaining attributes are filled with zeroes. The N+1 attribute contains the average of the input trace (= DC component) that was removed at the start of the decomposition.

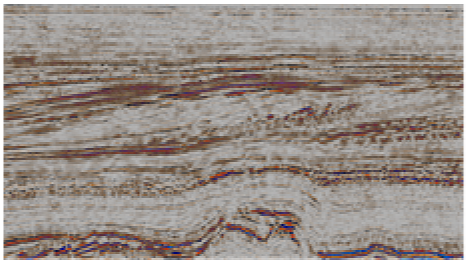


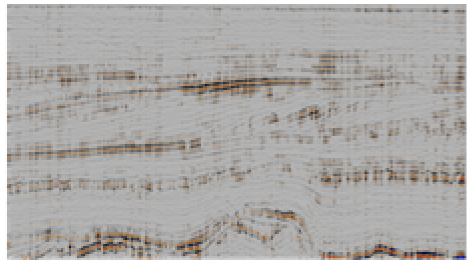
Synthetic trace (top) decomposed with CEEMD into its IMF components (bottom).

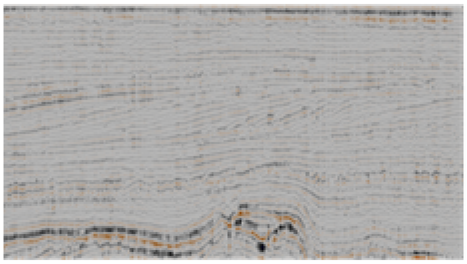
Examples

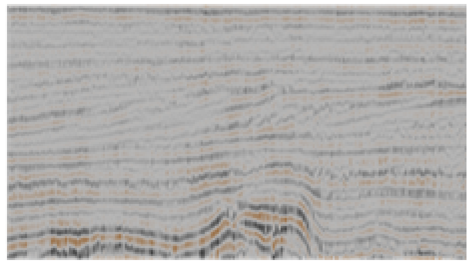


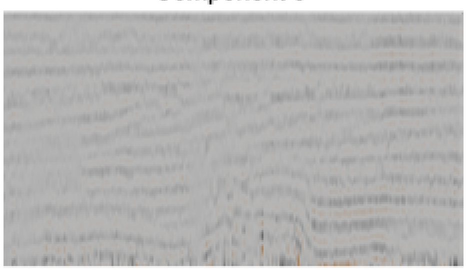
Line 425 of F3 Demo is the input for the decomposition with CEEMD. The IMF components; Peak Frequency components; and Peak Amplitude are shown below


Component 1

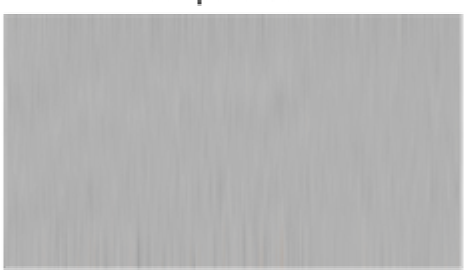

Component 2


Component 3

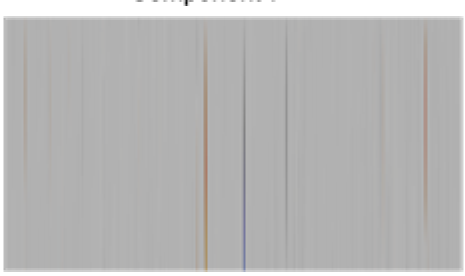

Component 4

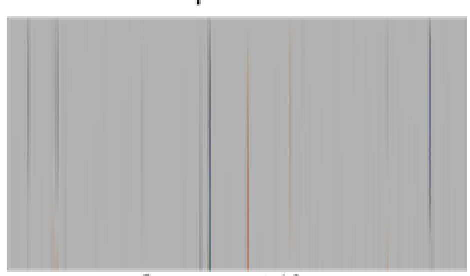

Component 5

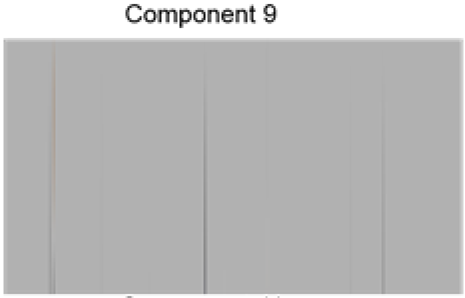

Component 6


Component 7


Component 8

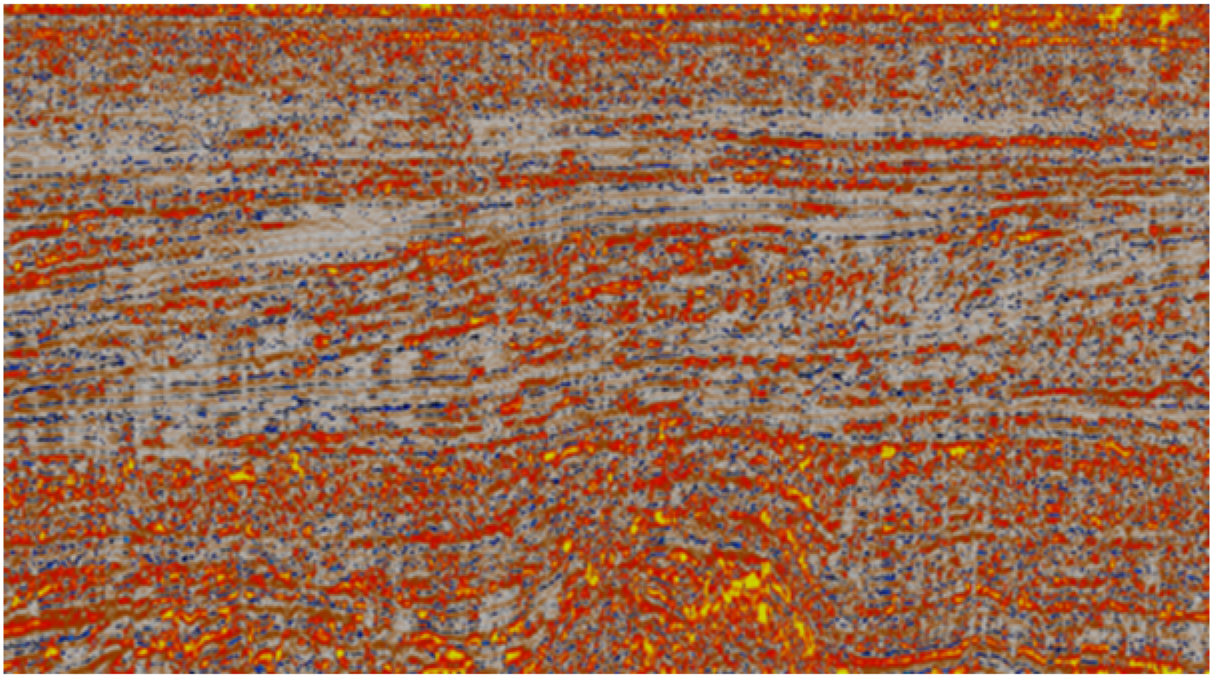

Component 9


Component 10

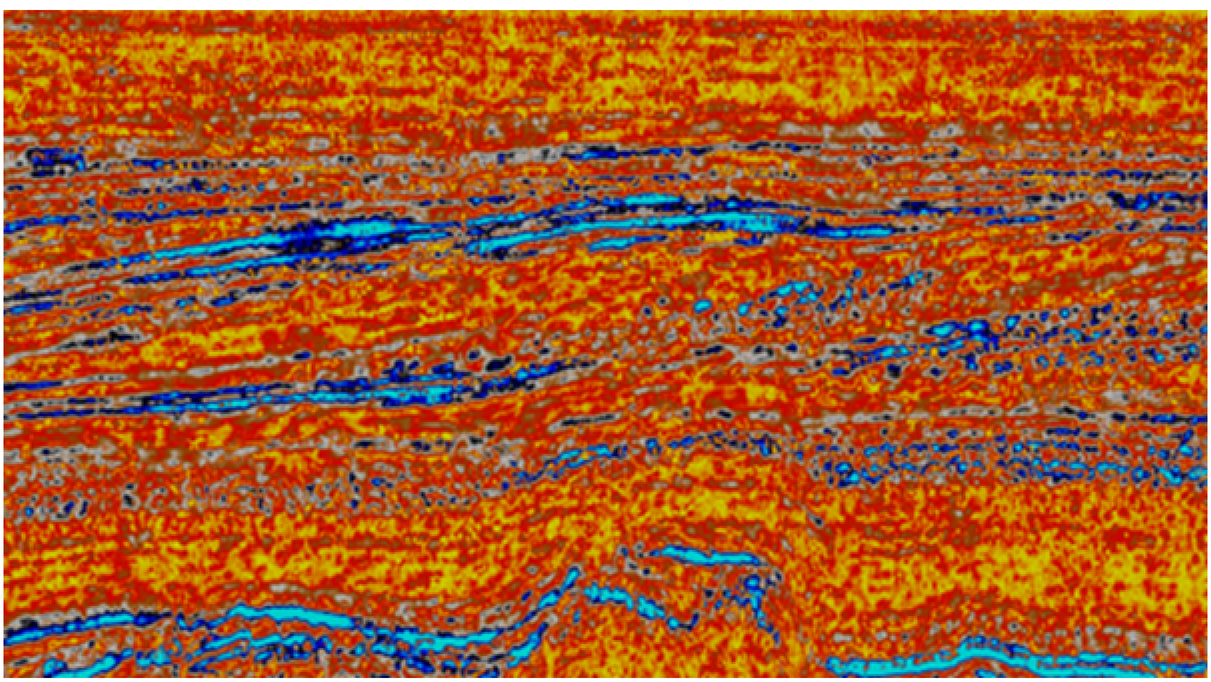

Component 11


DC Component

IMF components (CEEMD) of Line 425.



Peak Frequency (CEEMD) of Line 425.



Peak Amplitude (CEEMD) of Line 425.

References

Han J. and Van der Baan M., 2013. Empirical mode decomposition for seismic time-frequency analysis. Geophysics, 78 (2), O9-O19.

Huang, N.E., Shen, Z., Long, S.R., Wu, M.C., Shih, H.H., Zheng, Q., Yen, N.C., Tung, C.C. and Liu, H.H., 1998. The empirical mode decomposition and the Hilbert spectrum for nonlinear and non-stationary time series analysis: Proceedings of the Royal Society A: Mathematical, Physical and Engineering Sciences, 454, no. 1971, 903-995.

Convolve (2D & 3D)

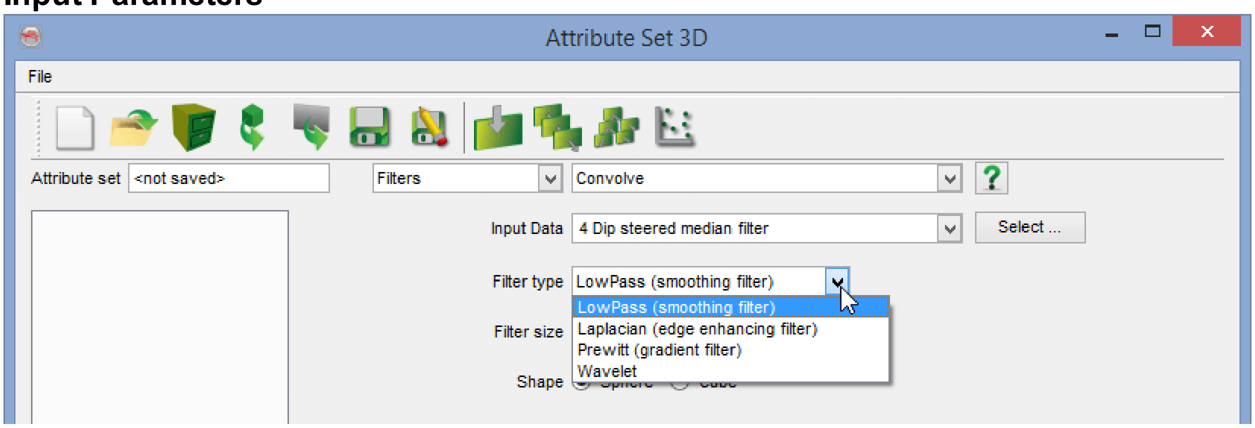
Name

Convolve (2D/3D) -- Attribute that returns a filtered response.

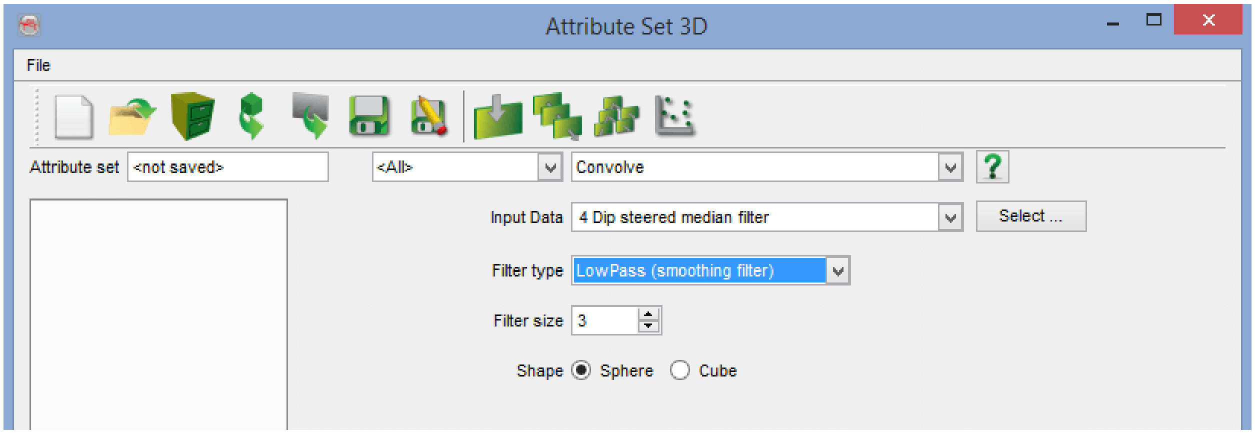
Description

The input data is convolved with a three-dimensional kernel specified by Filter type and associated parameters. Lowpass, Laplacian, and Prewitt are well known filters in image processing.

Input Parameters

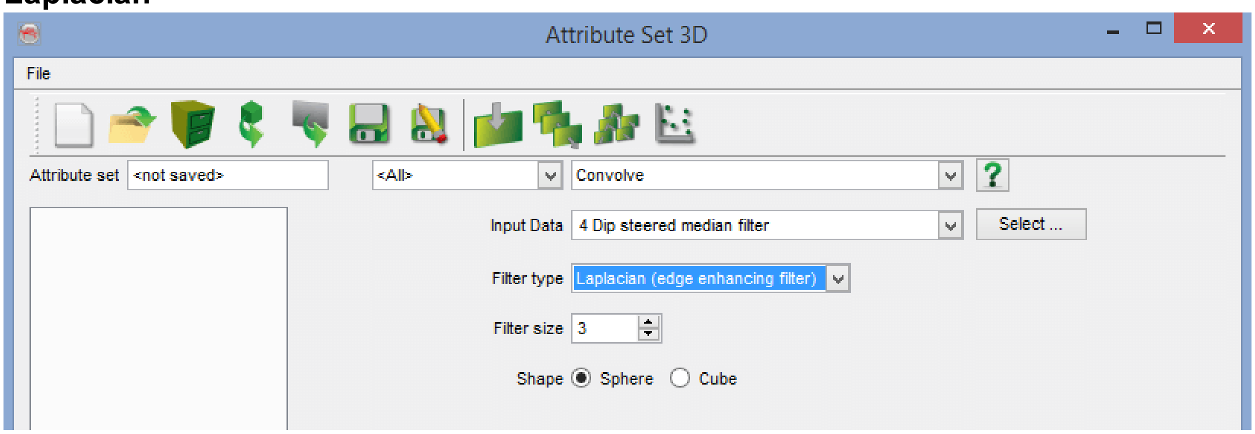


Lowpass



Lowpass is an arithmetic averaging filter which is used to smooth seismic data. The smoothness is determined by the *Filter size*. The *shape* parameter specifies whether the input samples are collected in a sphere or a cube centered around the evaluation point. The output is the sum of the sample values divided by the number of samples.

Laplacian

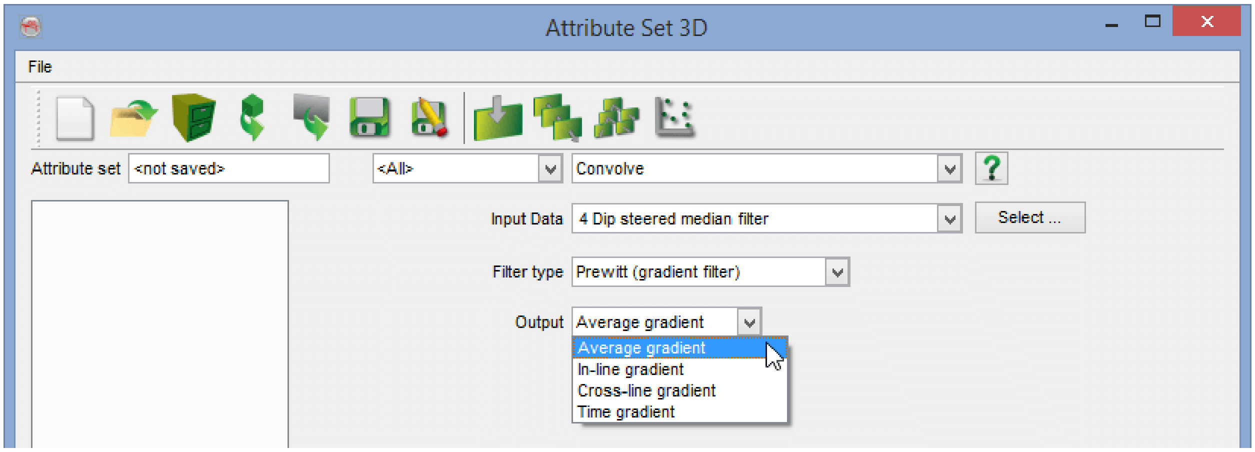


The Laplacian filter is an edge enhancement filter. The sharpness is determined by the *Filter size*. The *shape* parameter specifies the input samples to the filter. In a 3x3x3 Laplace filter, the output is calculated by multiplying the central sample value with 26 and subtracting all surrounding sample values. The convolution is characterized by the following kernel:

- (-1,-1,-1) (-1,-1 ,-1) (-1,-1,-1)
- (-1,-1,-1) (-1,+26,-1) (-1,-1,-1)
- (-1,-1,-1) (-1,-1 ,-1) (-1,-1,-1)

In case all sample values are equal and non-zero (either positive or negative), the effect of this operation is zero.

Prewitt



Prewitt is a contrast enhancement filter. This filter computes the gradient in different directions from a 3x3x3 input cube. The *Output* is returned in the specified direction (inline, crossline, or Z-plane). A 3x3x3 Prewitt kernel to calculate a horizontal gradient is given by:

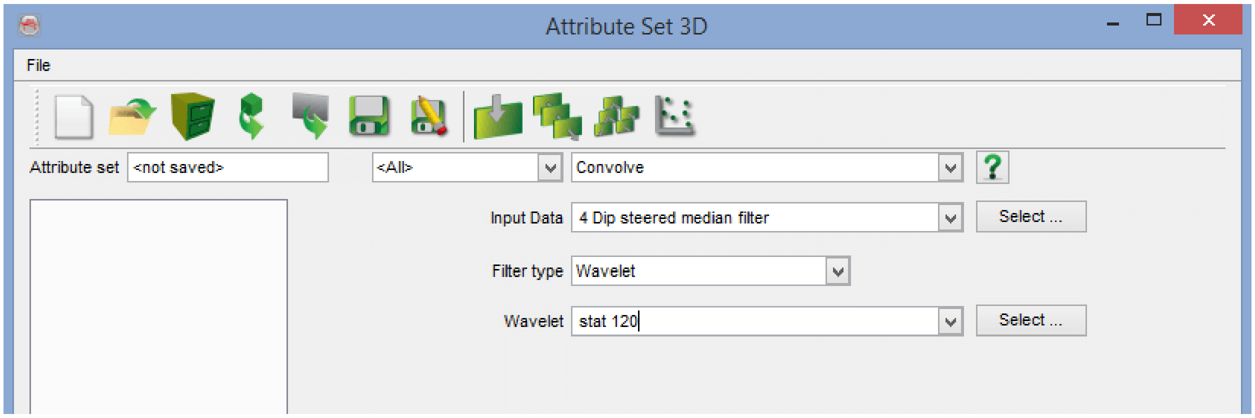
- (1,0,-1) (1,0,-1) (1,0,-1)
- (1,0,-1) (1,0,-1) (1,0,-1)
- (1,0,-1) (1,0,-1) (1,0,-1)

Note that the inline gradient returns the difference in amplitude in the inline direction. This is best visualized on a crossline. Similarly, a crossline gradient is visualized best on an inline.

A 3x3x3 Prewitt kernel that returns the vertical gradient is given by:

- (-1,-1,-1) (-1,-1,-1) (-1,-1,-1)
- (0, 0, 0) (0, 0, 0) (0, 0, 0)
- (1, 1, 1) (1, 1, 1) (1, 1, 1)

Wavelet



This option enables the user to convolve the data with a wavelet. In this context, a wavelet is the time series response of a filter. The wavelet should be imported into OpendTect first or it can also be created in OpendTect (From Wavelet management or ARKCLS Spectral blueing attribute).

Delta Resample

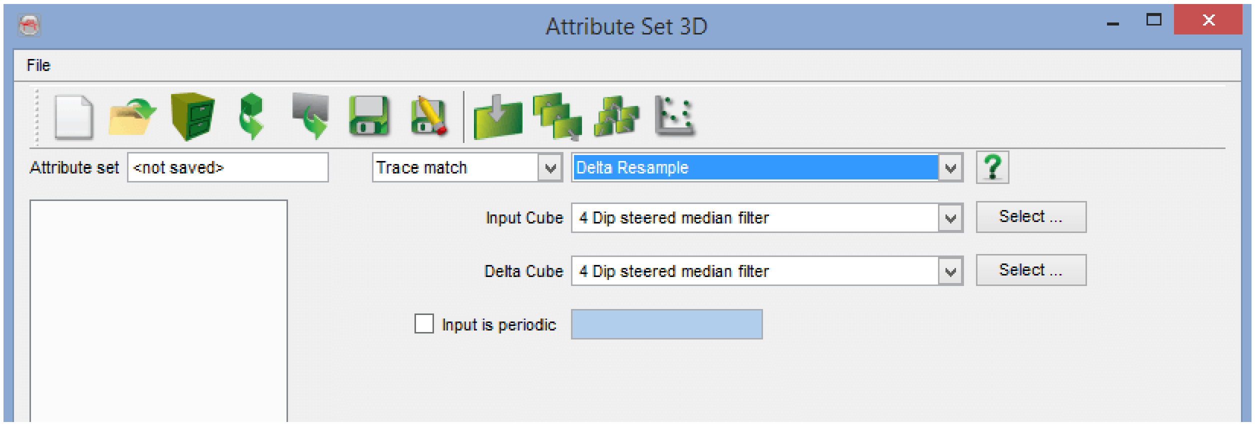
Name

Delta Resample -- Attribute that enables residual alignment of seismic volumes

Description

This attribute is used to make shifts inside seismic volumes. By defining an input cube and a delta cube, which represents the shifts that should be applied to the input data, a correctly aligned output volume can be generated. The delta cube can be generated by using the attribute "Match Delta". Note that you must use the Z-unit of the input cube. Also note that you must apply a negative shift to move the cube down and a positive shift to move the cube up. Using the "Delta Resample" attribute is very useful in case of working on, for example, multi-azimuth volumes which frequently show some degree of misalignment. This technique can also be very useful for time-lapse seismic data and NMO-corrected data. Only after aligning, a correct comparison can be made between two different volumes.

Input Parameters



The delta cube can be generated by using the Match Delta attribute. An advanced option in the attribute engine is that, when the input is periodic and contains phases, it is possible to define a (maximum) period. The box "Input is periodic" should be clicked and the period should be defined in the box. However, a situation like that is very exceptional and, in 99 out of 100 cases, the shift can be applied without using this option.

One of the side-effects of this residual alignment is that existing horizons are not consistent with the data any more. The horizons should be re-snapped to the aligned seismic data by using the option "Algorithms" when right-clicking a horizon in the tree. The third option "Snap to event" enables the user to make the horizon consistent again. Only after snapping the horizons can the user, for example, calculate horizon-steered attributes or performing waveform segmentation.

Flattening or Unflattening a Cube:
Another useful application for this attribute is to flatten or unflatten a cube. The example given below outlines how you would unflatten a cube that you have flattened (for example, using the Flattening option under the Horizon Workflows fold-out):

You would create a cube that would have, as a function of the position only, the Z difference between flattened and unflattened volume. For example, you may have set the TWT of the flattened cube to 1000 ms. Using the Horizon attribute you would read the TWT values of the horizon, let's say 1300ms for a particular trace. The shift is thus defined with a mathematics attribute as:

shift(ms) = (FlatTWT(s) - HorTWTAttrib) * 1000 = (1 - 1.3) * 1000 = -300

Applying this shift attribute as 'Delta Cube' for your flattened volume (your 'Input Cube') will undo the flattening (in this case, with a negative value, thus shifting the cube down).

Energy

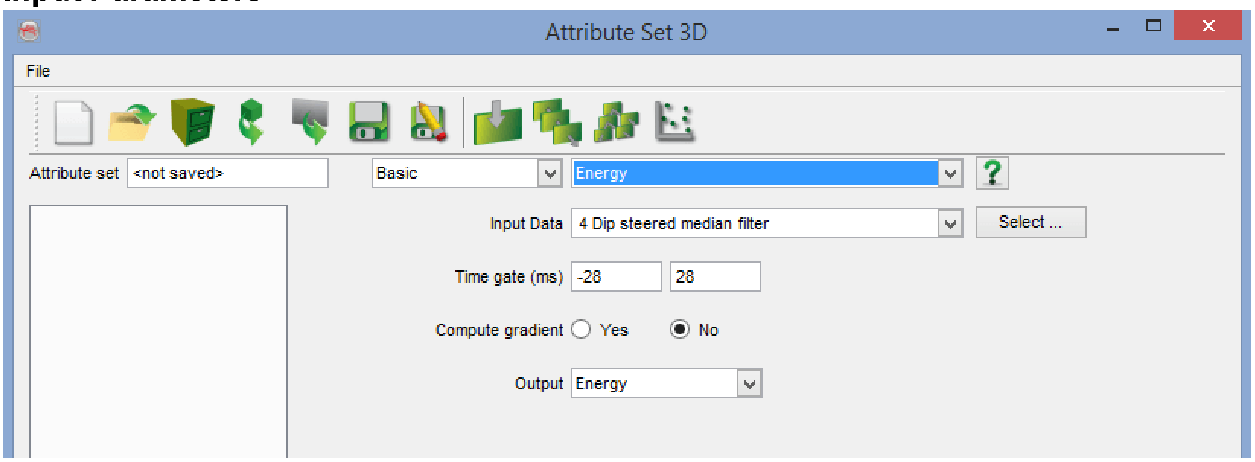
Name

Energy -- Response attribute that returns the energy of a trace segment

Description

This attribute calculates the squared sum of the sample values in the specified *time-gate* divided by the number of samples in the gate. The Energy is a measure of reflectivity in the specified *time-gate*. The higher the Energy, the higher the Amplitude. This attribute enhances, among others, lateral variations within seismic events and is, therefore, useful for seismic object detection (e.g. chimney detection). The response energy also characterizes acoustic rock properties and bed thickness.

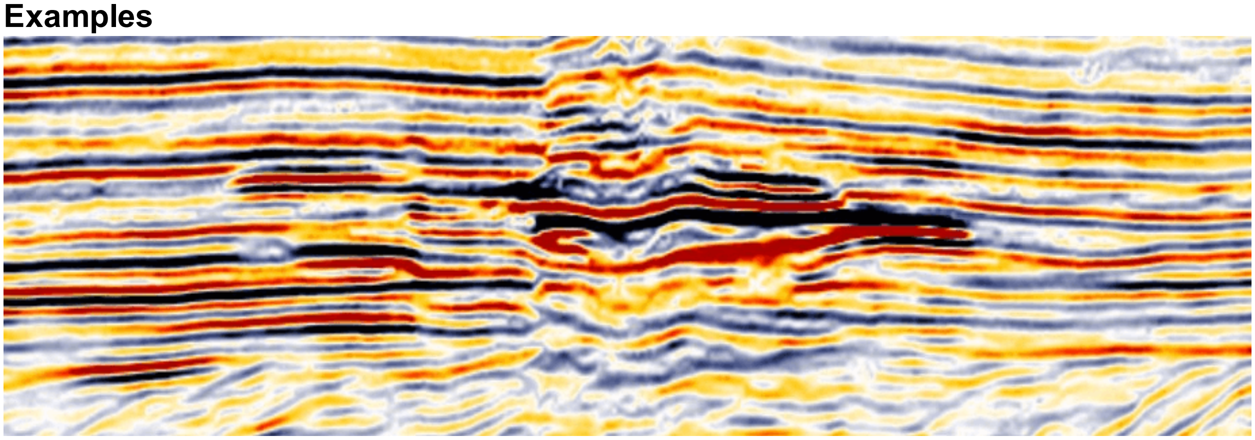
Input Parameters



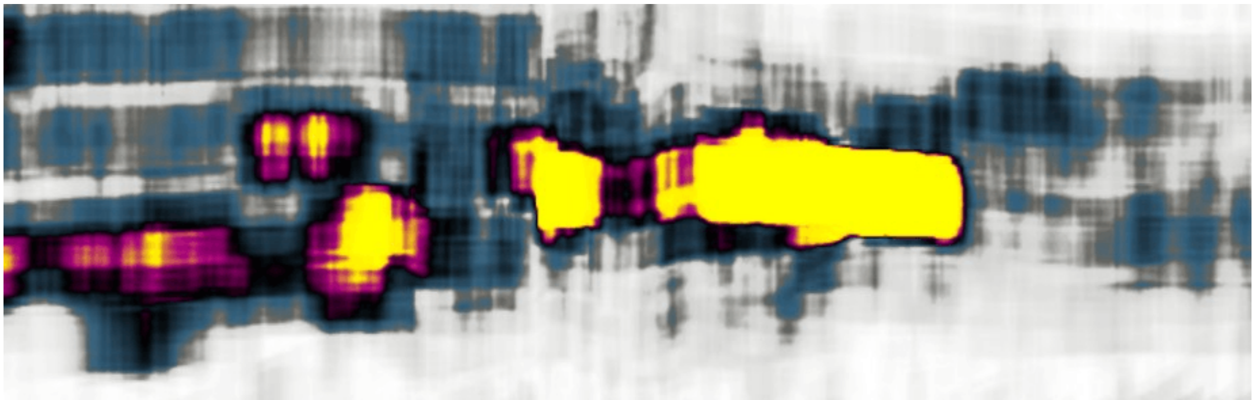
Output

Energy, Sqrt(Energy) and **Ln(Energy)**

Examples



Seismic data showing anomalously high values



Energy attribute (time-gate [-28,28]) enhances anomalously high values (yellow)

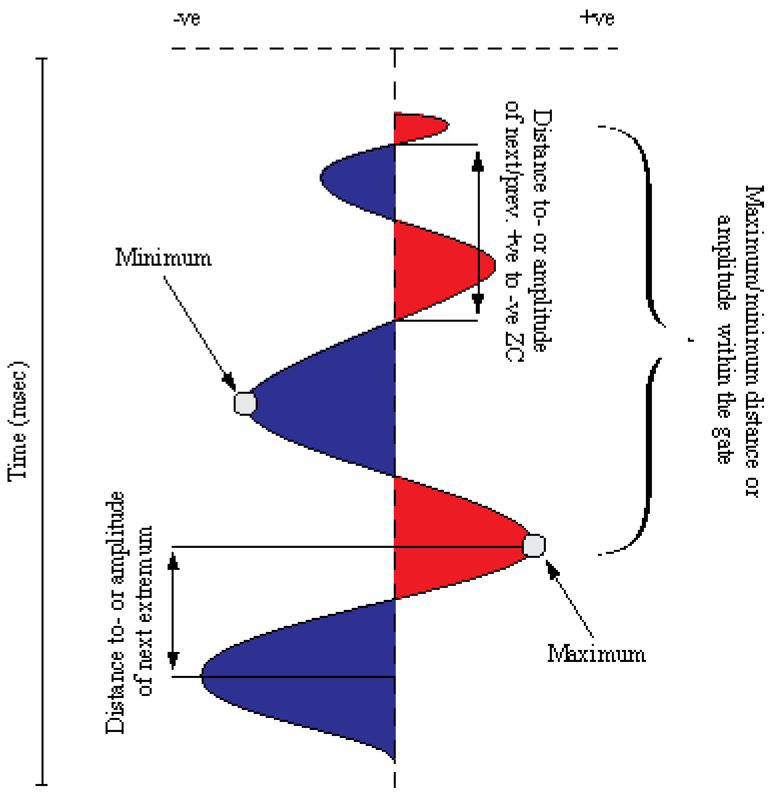
Event

Name

Event -- Attribute that quantifies an events shape or distance relative to a next event

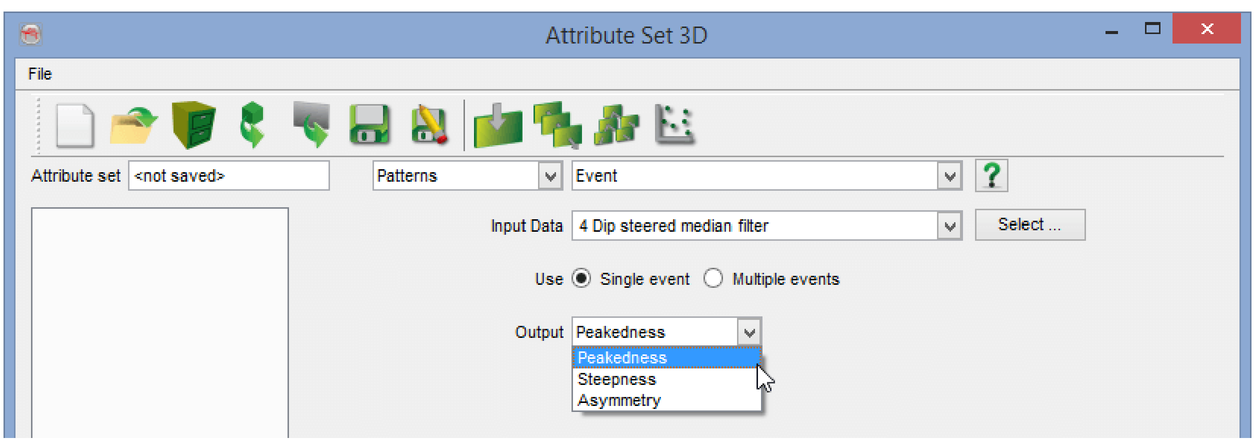
Description

The event attribute is a useful tool when determining the quality of horizons in seismic data that can also be applied to in-lines, cross-lines, or z-slice elements. A sketch of the several event attribute applications is shown below:



Input Parameters

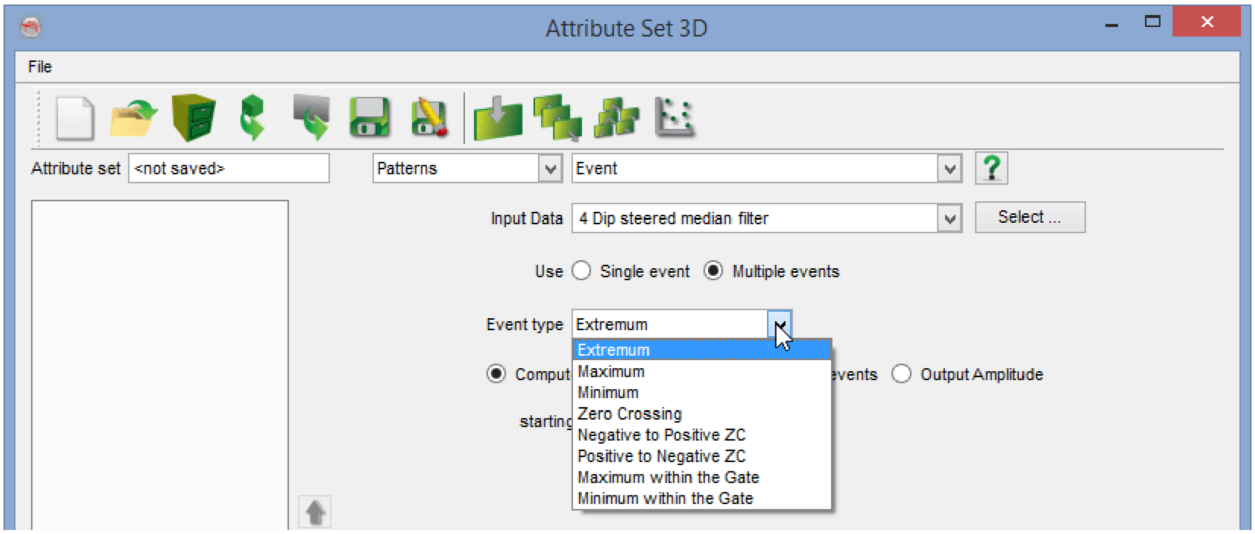
In the *single event mode*, the algorithm searches for the extremum and quantifies the shape around the event in terms of either *Peakedness, Steepness* or *Asymmetry*.



- **Peakedness:** The ratio between the Extremum value and distance between next and previous zero crossings (ZC)
- **Steepness:** The slope of tangent to the seismic trace at a zero crossing
- **Asymmetry:** The asymmetry of event. Mathematically it can be

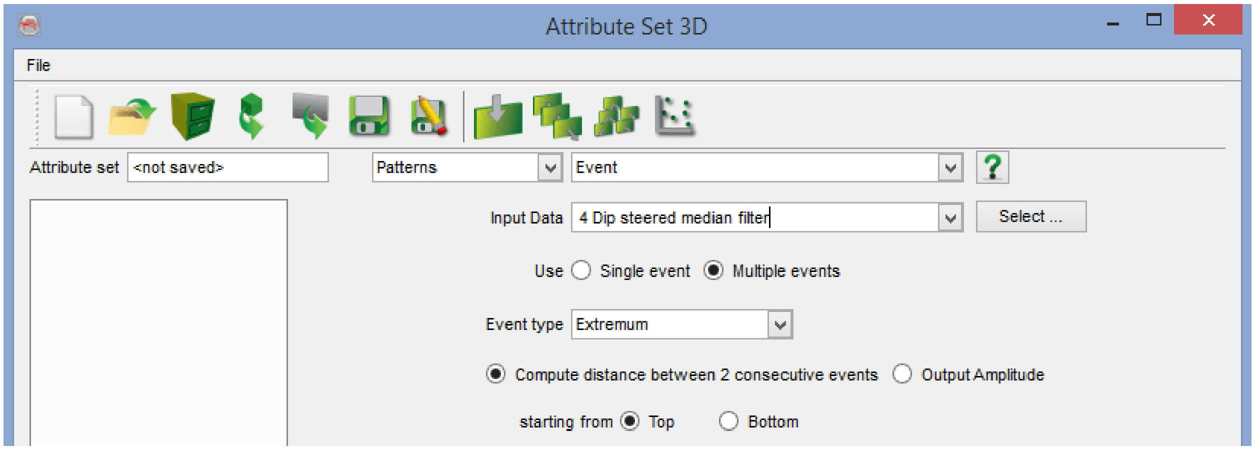
presented as: (L-R)/(L+R) where L is the distance between previous ZC and extremum and R is distance between next ZC and extremum

In the multiple event mode, the event type needs to be specified from the Event type drop-down list:

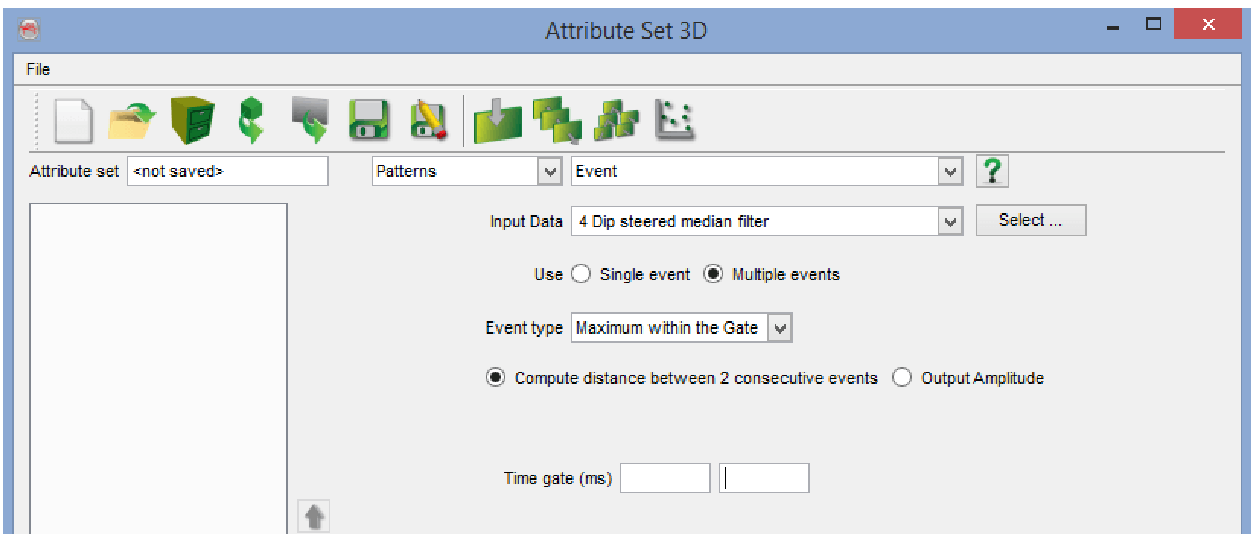


- *Extremum*
- *Maximum*
- *Minimum*
- *Zero Crossing*
- *Negative to Positive Zero Crossing*
- *Positive to Negative Zero Crossing*
- *Maximum within the gate*
- *Minimum within the gate*

Output
The output is either the *distance* between the chosen event and the next/previous similar event or the *amplitude* of the event. The output is determined by the checkbox below the event type drop-down list.



For the event types *Maximum within the gate and Minimum within the gate*, a *time-gate* (in milliseconds) needs to be specified. The algorithm computes the distance, within the specified time-gate from the current point, e.g. on a horizon, to the nearest maximum or minimum.



The Event attribute is for example useful while quality-checking horizon grids. The attribute can also aid in finding the distances between two events and using it as an estimate of relative thickness changes between them. The adaptive use of Volume Statistics along with the Event attribute can also bring relative changes to derive meaningful geological aspects.

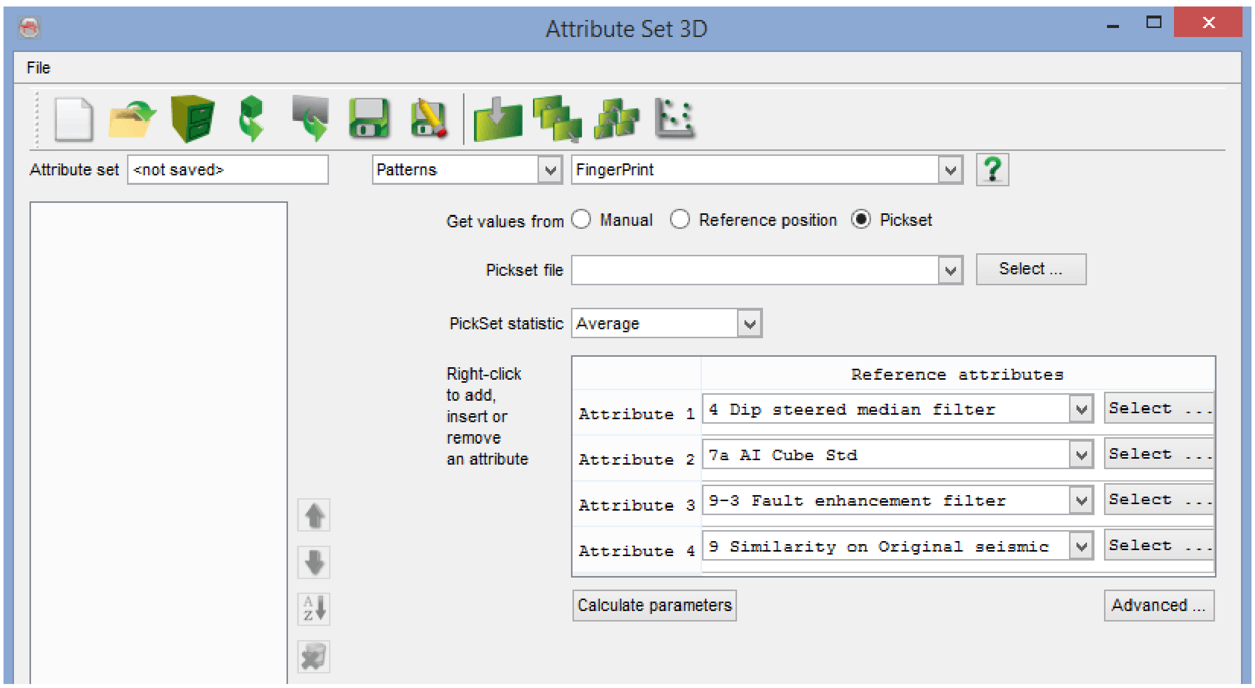
Fingerprint

Name

Fingerprint -- Attribute that computes the similarity between a user-defined vector of attributes and the equivalent vector taken at each position in a cube.

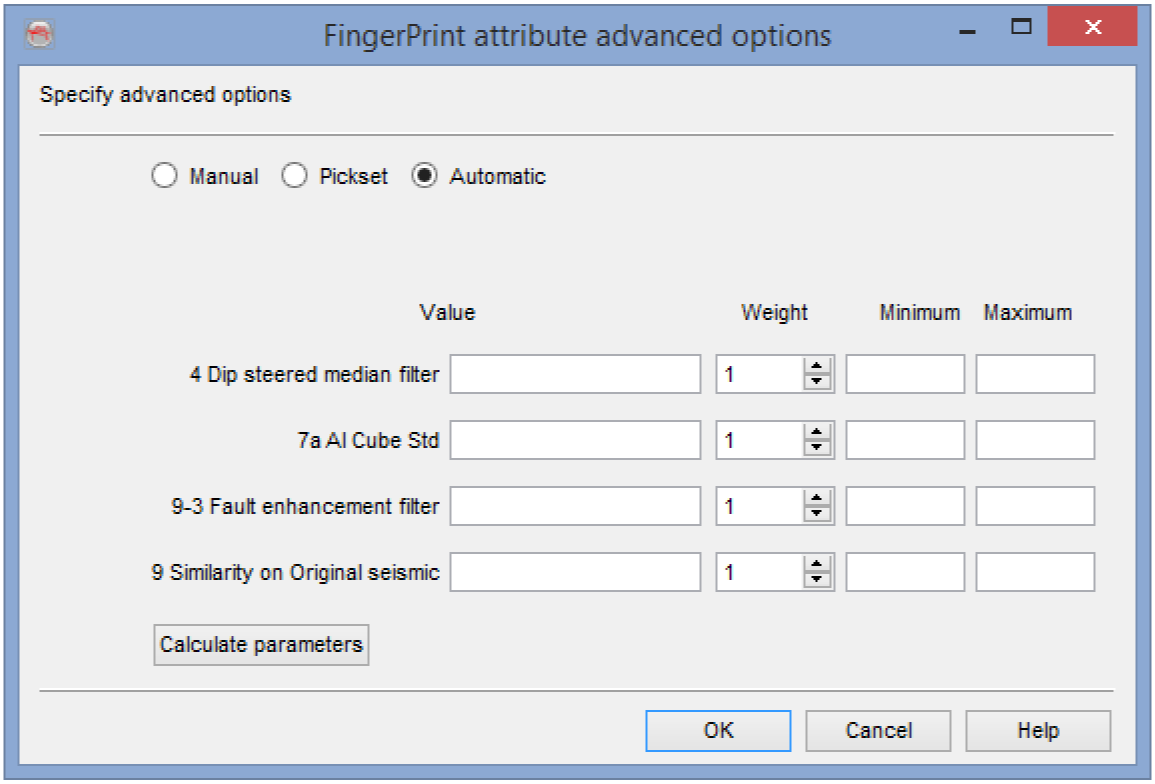
Description

This attribute computes the similarity between a user-defined vector of attribute values and the equivalent vector taken at each sample position inside the cube. The reference vector can be constructed from one or more positions. A statistical property (average, median, variance, minimum or maximum) is calculated after the construction of the vector. Also, it is possible to construct the vector manually by editing the attribute values of the fingerprint vector. The similarity between the fingerprint vector and the equivalent vector at the evaluation point is computed as the normalized Euclidean distance between the two vectors and ranges between 0 (vectors are not identical at all) and 1 (vectors are 100% identical).

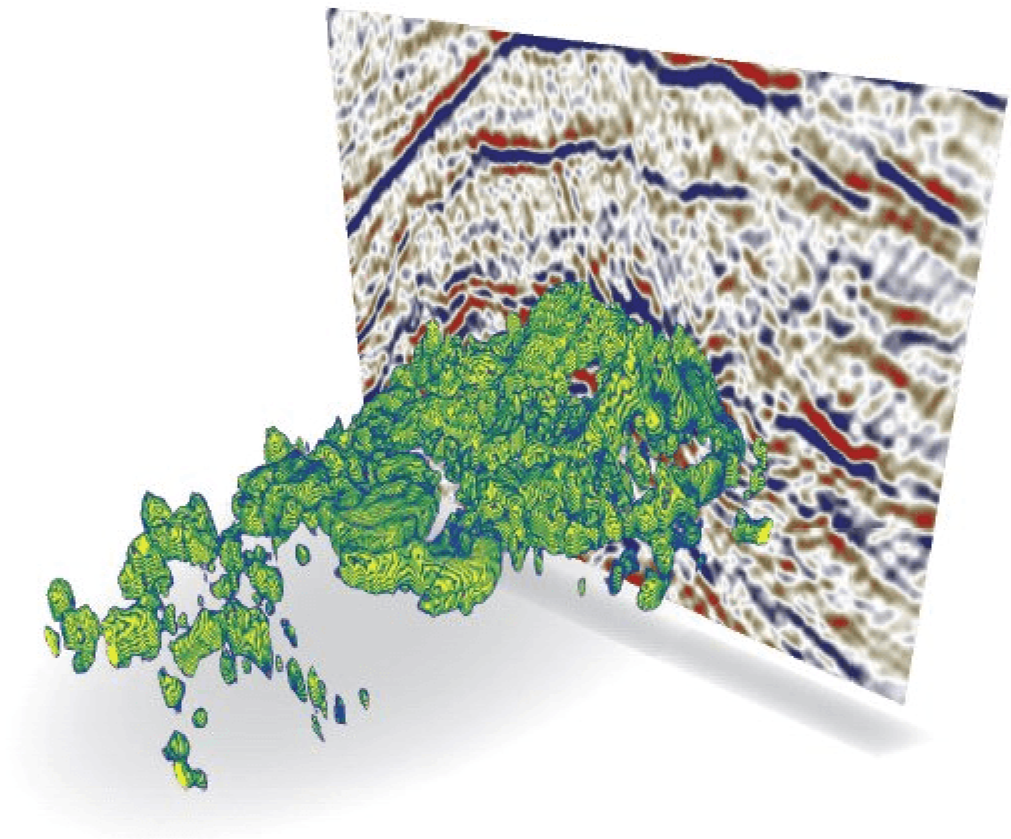


Note: If you want to insert/remove the reference attributes, right click in the empty area of Reference attribute window.

Advanced options



In order to compare vectors, you can use the "Advanced" option to obtain the ranges for the input values. The ranges are automatically calculated using random picks when you press the button "Calculate parameters". You can also use a pickset to find the ranges or even manually introduce them. In the "Advanced options" window, a weight can be assigned to each individual attribute. The default value of the weight is set to 1.



Example of result application fingerprint attribute

Frequency Filter

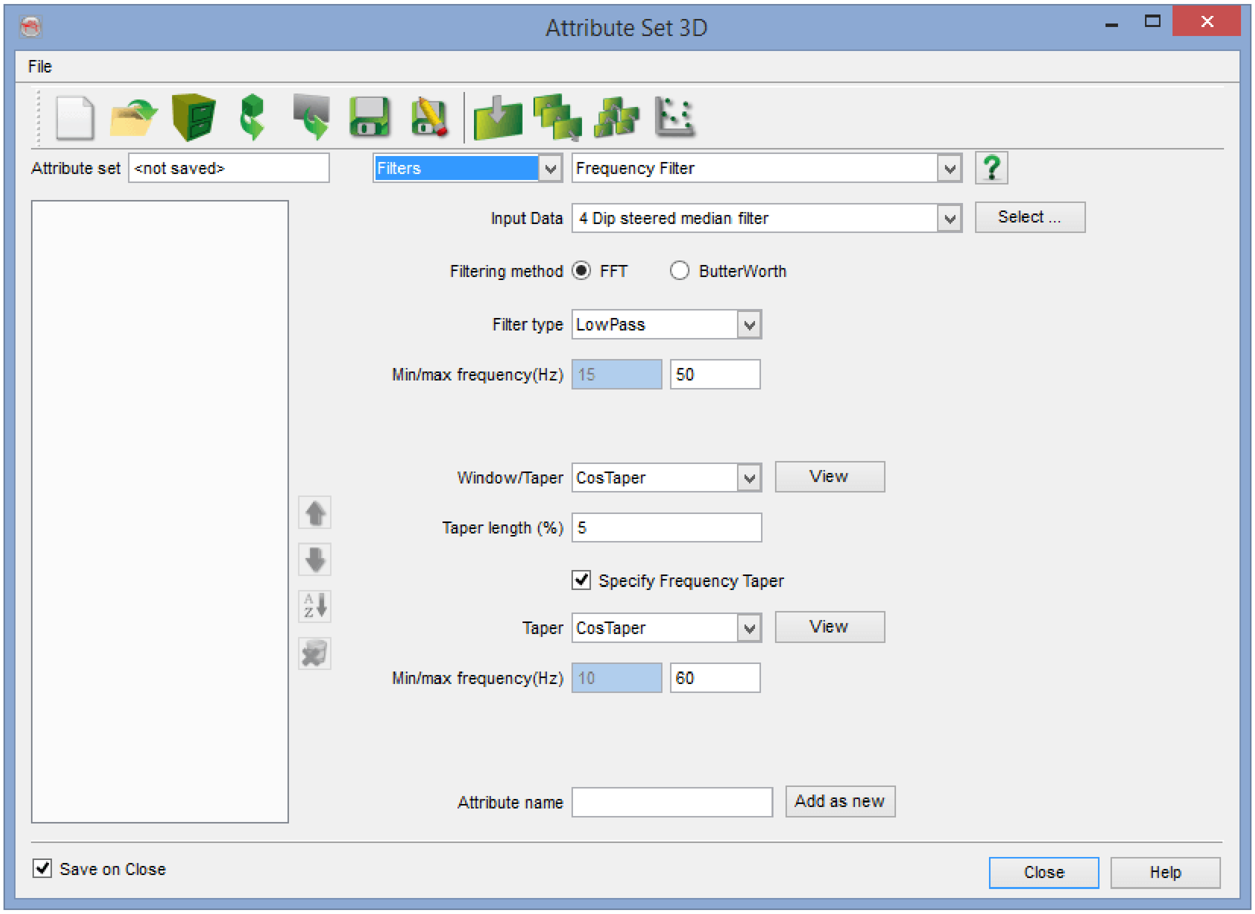
Name

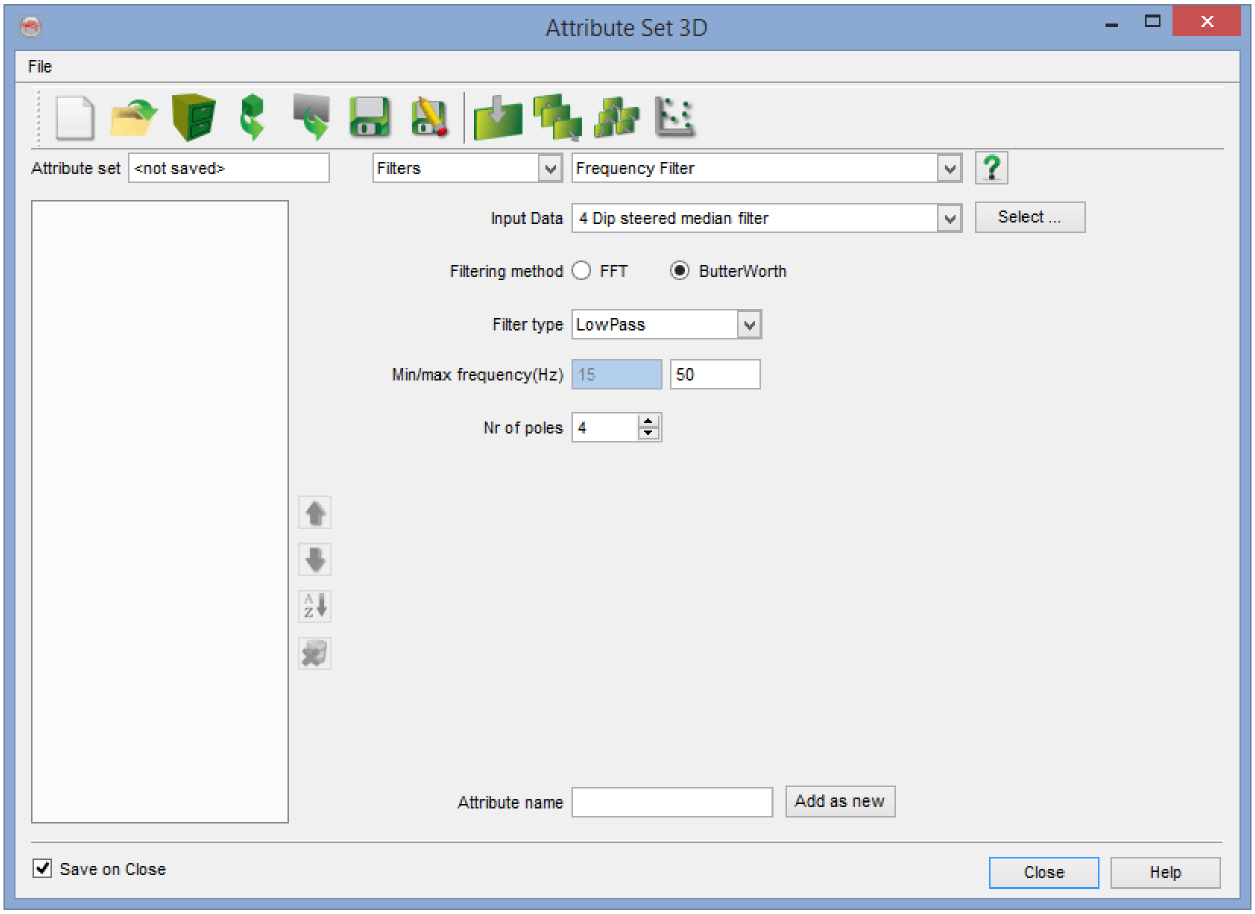
Frequency Filter -- Attribute that returns filtered data using FFT or Butterworth filter types

Description

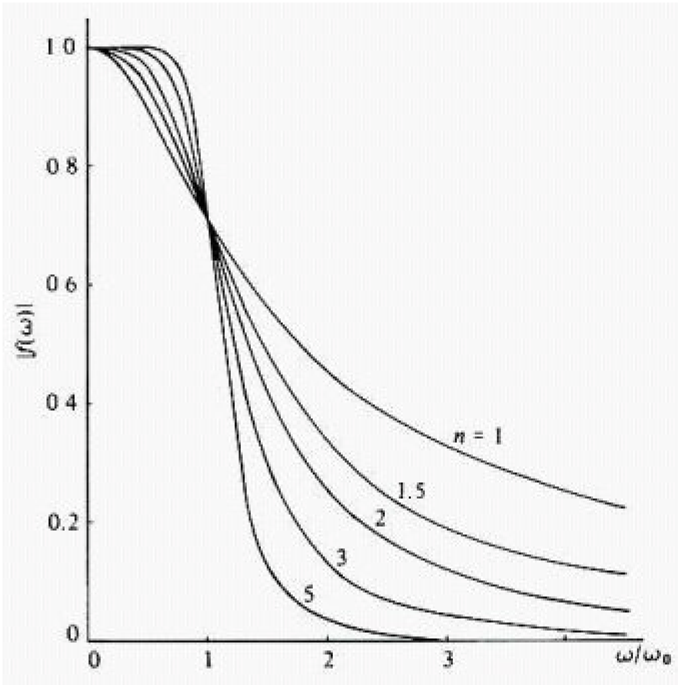
The specified **Input Data** is bandpass filtered with the commonly used Fast Fourier Transform or Butterworth filter.

Input Parameters


Attribute Set 3D
File
Attribute set
<not saved>
Filters
Frequency Filter
Input Data
4 Dip steered median filter
Select ...
Filtering method
FFT
ButterWorth
Filter type
LowPass
Min/max frequency(Hz)
15
50
Window/Taper
CosTaper
View
Taper length (%)
5
Specify Frequency Taper
Taper
CosTaper
View
Min/max frequency(Hz)
10
60
Attribute name
Add as new
Save on Close
Close
Help



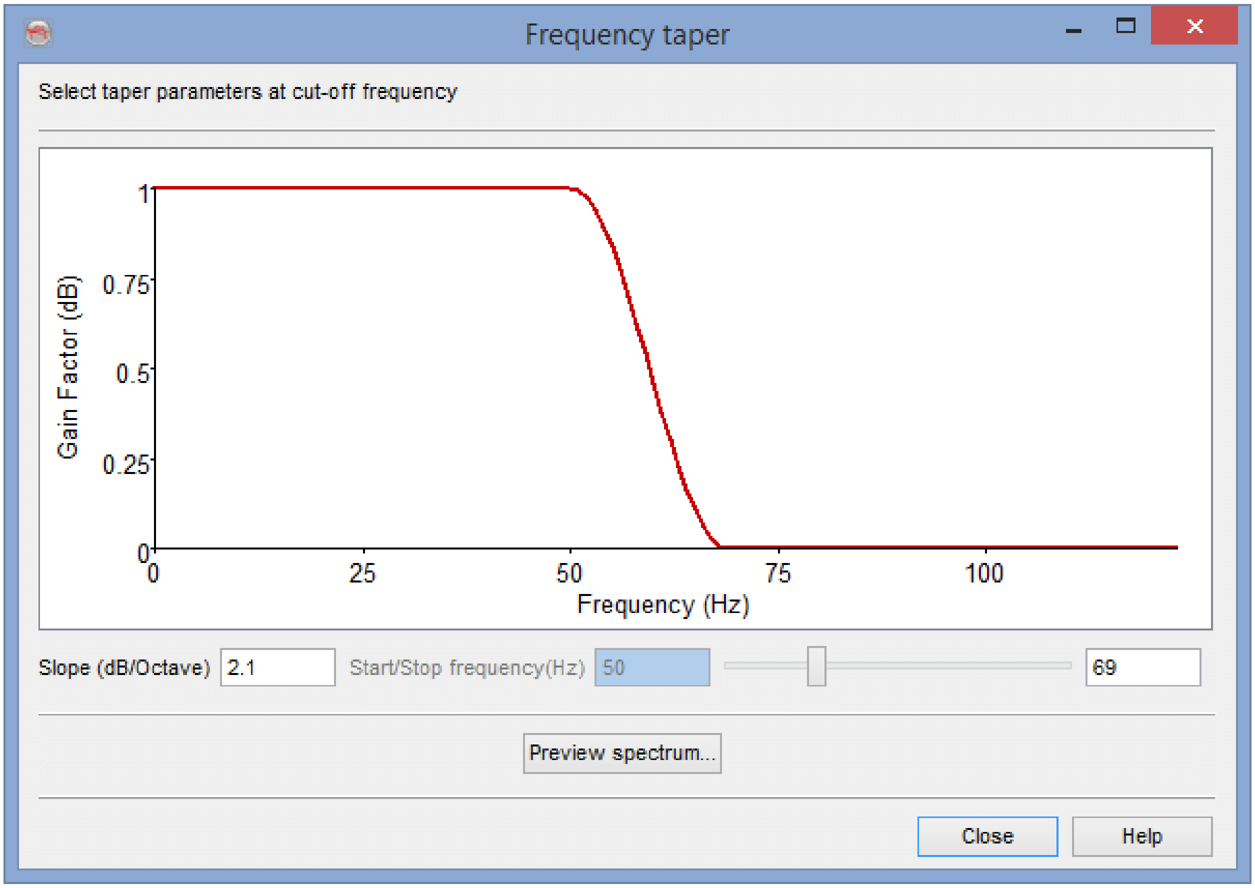
The difference between using the FFT or Butterworth filtering method is that, for the FFT, one considers the complete trace while for the Butterworth filter, only a "small" segment, depending on the selected number of poles, is taken into account. The user should keep in mind that using the Butterworth Filter results in a small shift in the seismic data.



The curves of the Butterworth filter for various numbers of poles

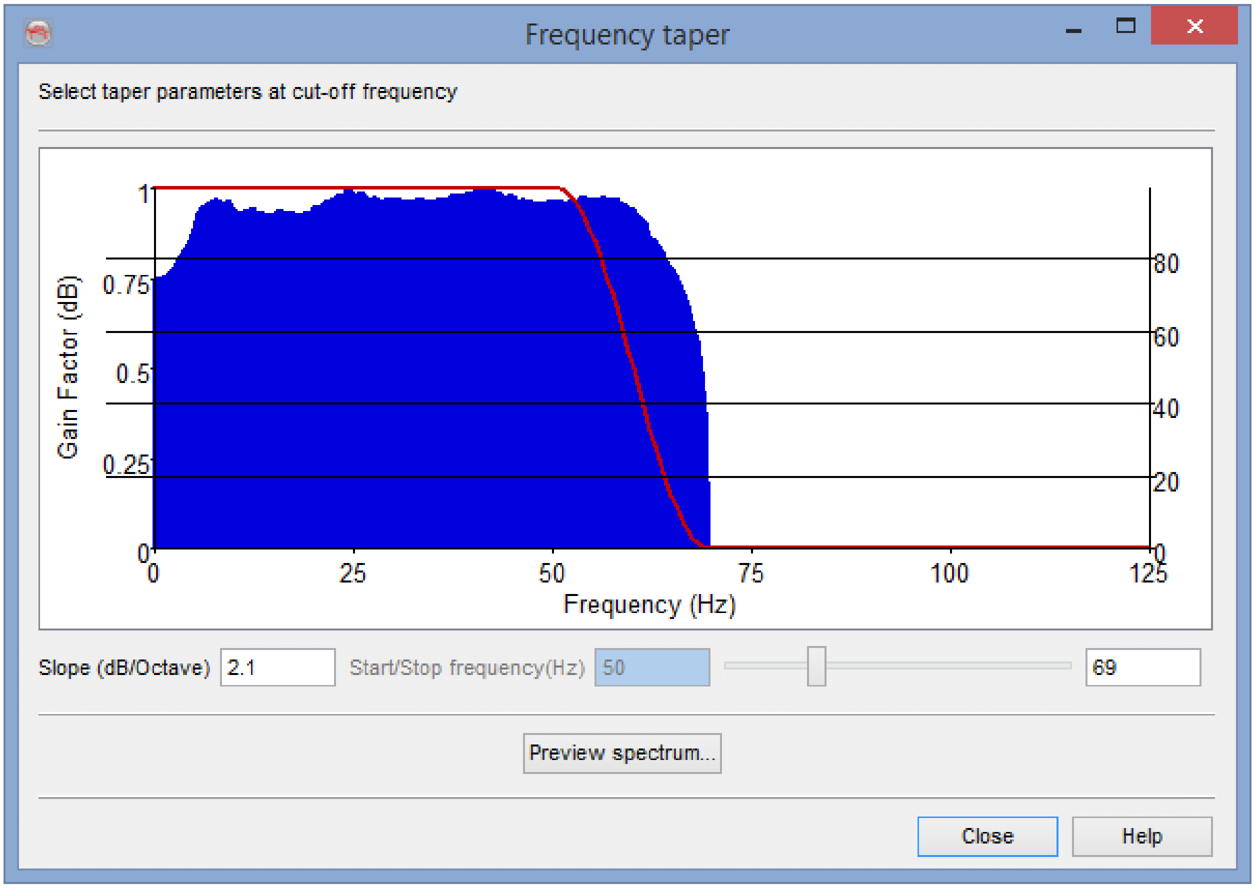
The *Filter type* is set to *LowPass, HighPass* or *BandPass* according as it is discriminating against frequencies above or below a certain limiting frequency or outside of a given band of frequencies. The top frequency tapering applied to the extracted data in the time domain.

Frequency Taper:
Additionally, a frequency taper can be applied by setting the min/max Hz tapering window. This could be done by toggling *Specify Frequency Taper* ON. Press the View button to specify an appropriate frequency taper.



Frequency taper settings in the attribute definition (for filter type: 'BandPass').

Once the Frequency Taper is displayed (red line in a pop-up window), a spectrum on any inline/crossline can be viewed by pressing the *Preview Spectrum* button. It will prompt to select an inline/crossline ("Select line from Data"). In that dialog select either inline or crossline radio button and press *Next*. Sub select the part of inline/crossline and proceed. A blue coloured amplitude spectrum will be displayed (as shown below). Now adjust the parameters (Slope or Start/Stop Frequency) and finalize the frequency taper settings.



Interactive display of the frequency taper parameters (for filter type: 'BandPass').

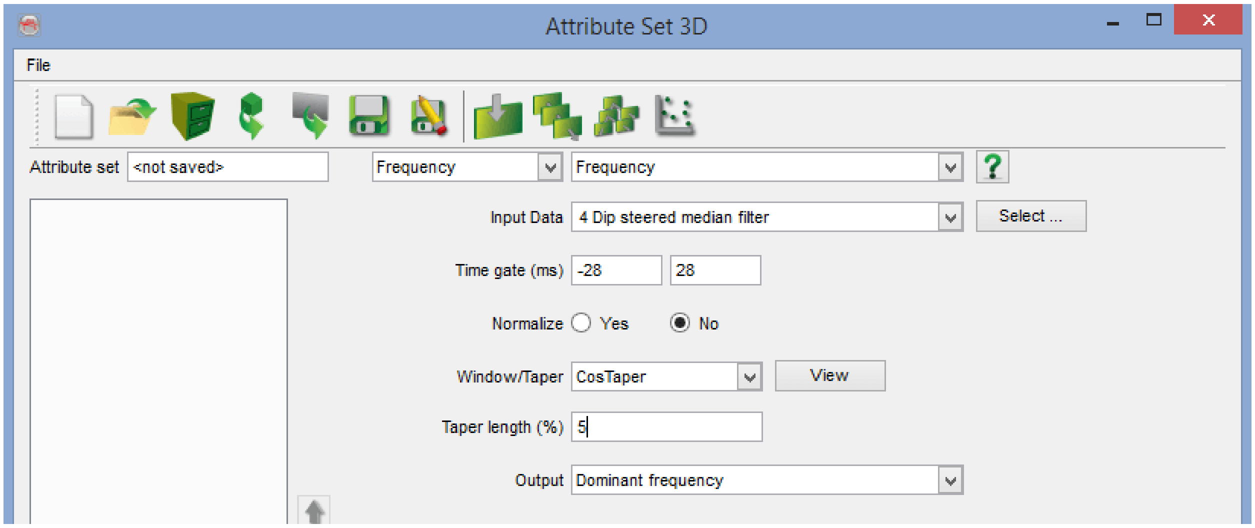
Frequency

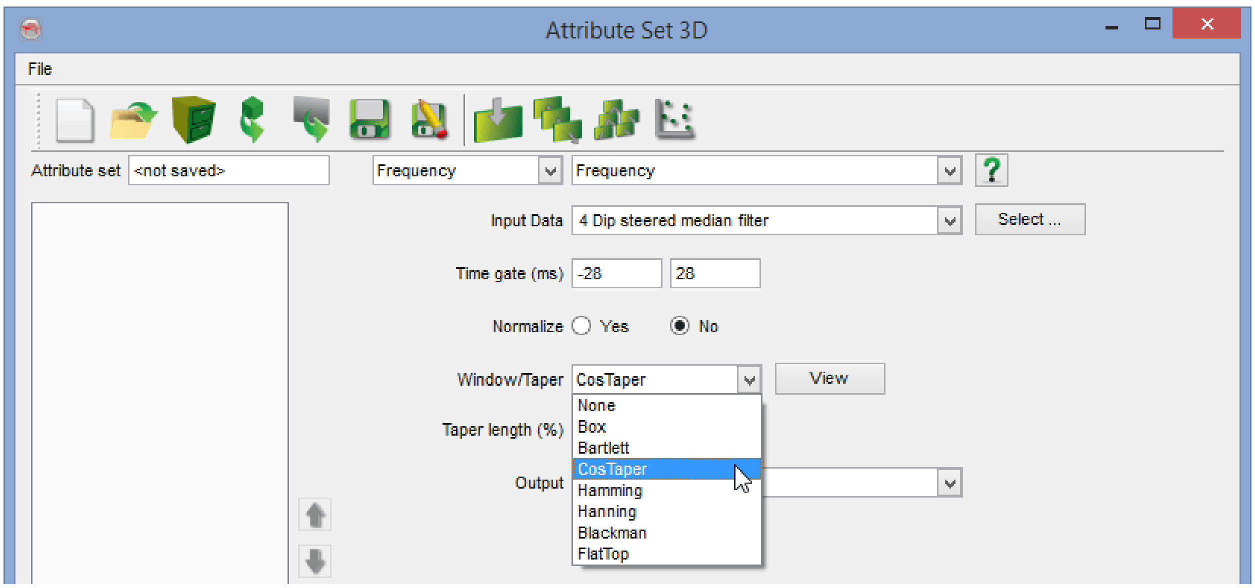
Name

Frequency -- Response attribute that returns frequency properties

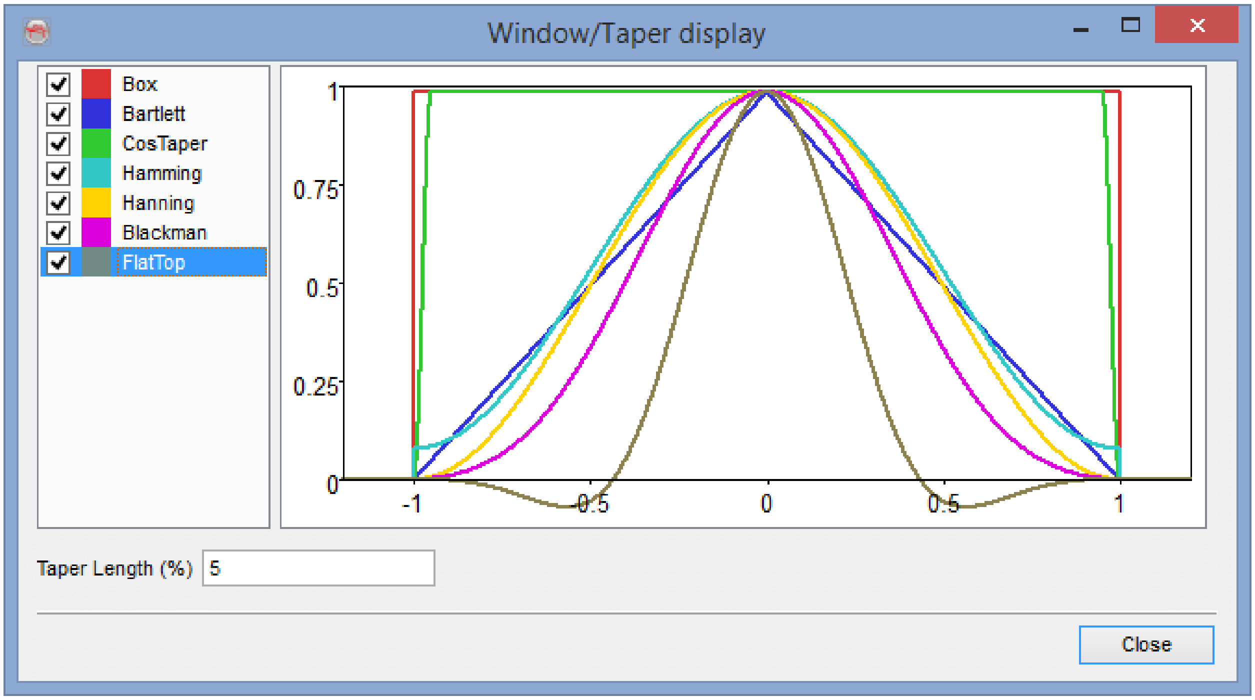
Description

Input Parameters



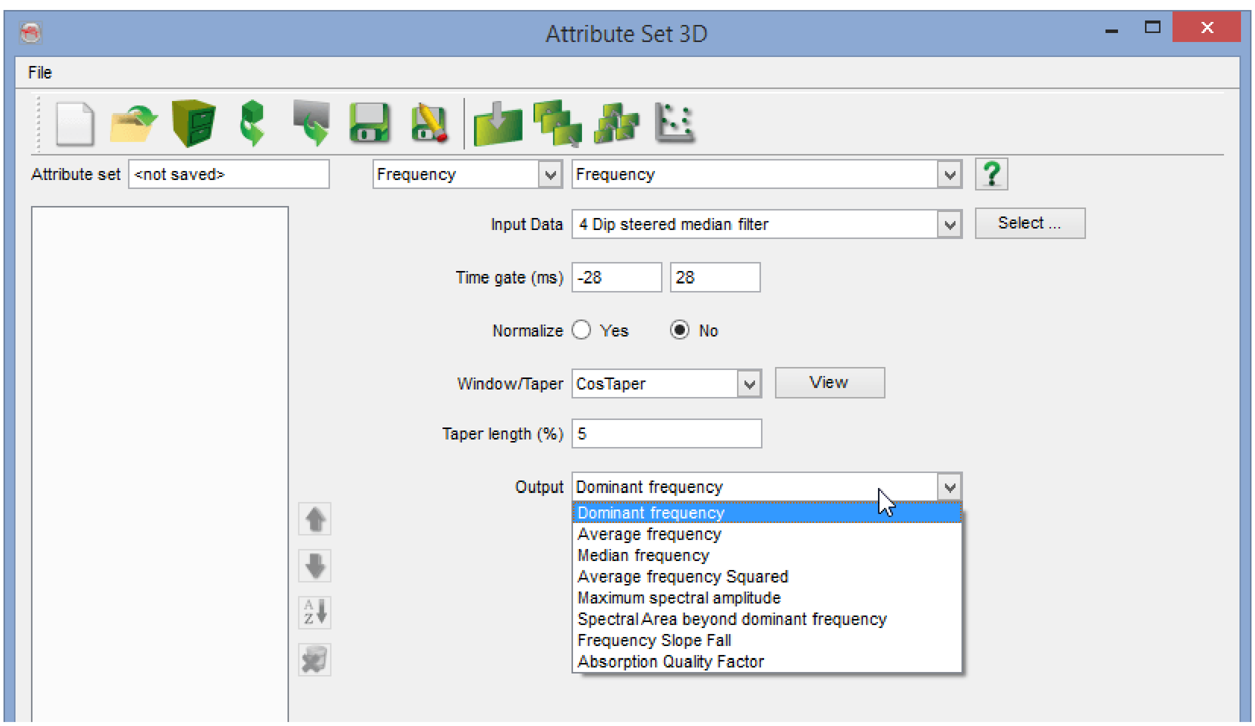


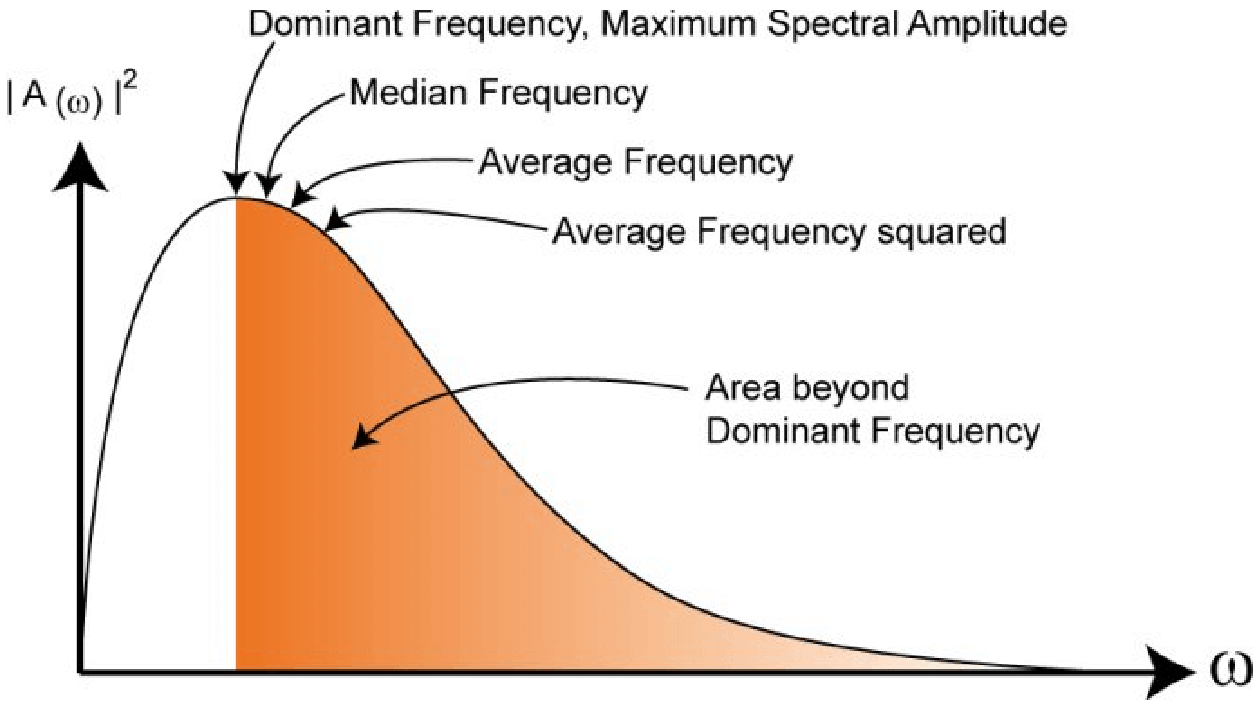
The specified *time-gate* is transformed to the Fourier domain and the requested *output* is calculated. The time-gate is tapered with the specified *Window/Taper* prior to Fourier Transform. The shape of the various tapers is shown in the figure below.



If *Normalize* is enabled, the frequency spectrum is normalized with regard to its area. This will make it possible to compare attributes from areas with high and low energy.

Output





All possible Frequency outputs are in X-axis, Y-axis = Amplitude values

Dominant frequency
Returns the dominant frequency from the frequency spectrum--the frequency with the highest amplitude

Average frequency
Returns the arithmetic mean of the frequency spectrum.

Median frequency
Returns the weighted median value of the frequency spectrum, which is the frequency at half the spectral area on each side. The median frequency might be somewhat more robust than average frequency, at the cost of lower precision.

Average frequency squared
Returns the average of the squared frequencies.

Maximum spectral amplitude
Returns the maximum amplitude of the frequency spectrum, i.e. the amplitude of the dominant frequency.

Spectral area beyond dominant frequency
Returns the spectral area beyond the dominant frequency (see figure below).

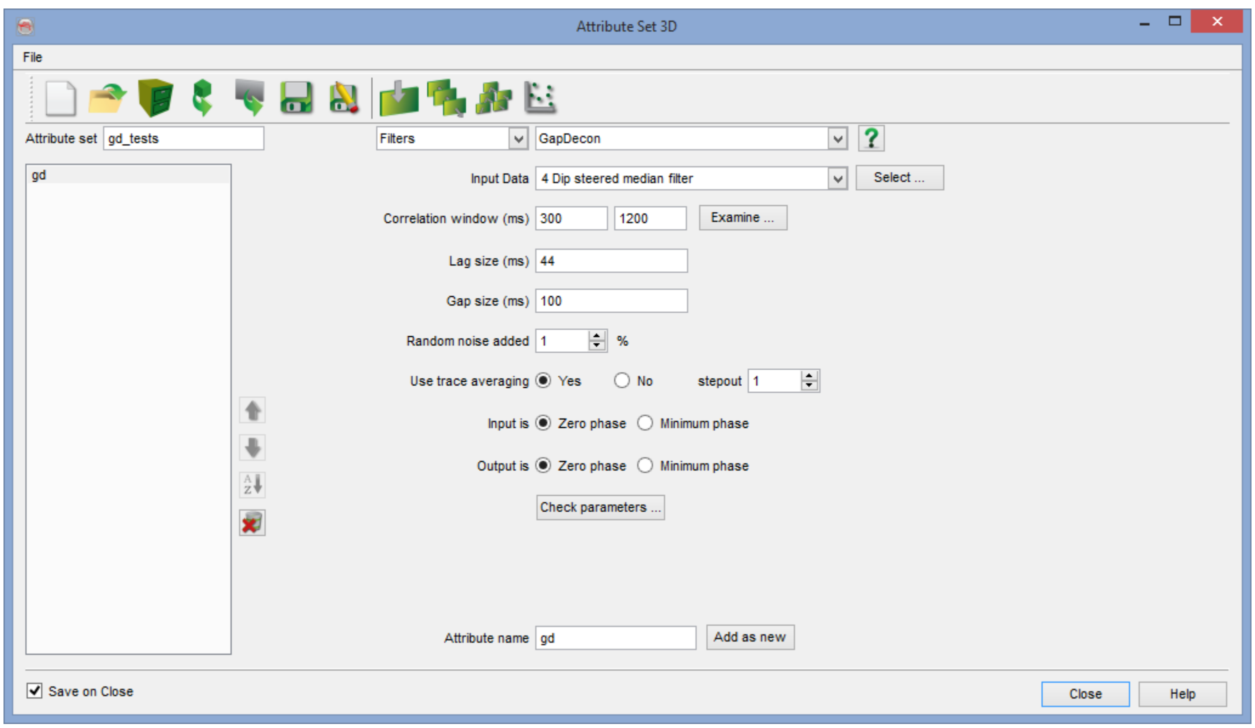
Absorption quality factor
Returns the area beyond the dominant frequency weighted by frequency.

Frequency slope fall
Returns the peakedness of the spectrum. It is defined as: 1+(MSA-Spectral Area)/(MSA+Spectral Area). A value approaching zero indicates a flat distribution while a value approaching one indicates a peaked distribution.

GapDecon

Name
GapDecon -- Attribute that aims to attenuate repetitions of primary reflections (multiples).

Description
The type of multiple removal algorithm chosen for this application is the well known inversed filtering method also known as Gap deconvolution. This filter aims to attenuate a user-defined part of an auto-correlation function. The underlying idea is that multiples in the data are secondary reflections, i.e repetitions of the primary reflections that show up in the auto-correlation function at a time that corresponds with the extra travel time. The filter can be applied on-the-fly or in batch-mode to produce a filtered output cube. The user defines the *GapDecon* attribute from the list and specifies a number of input parameters.

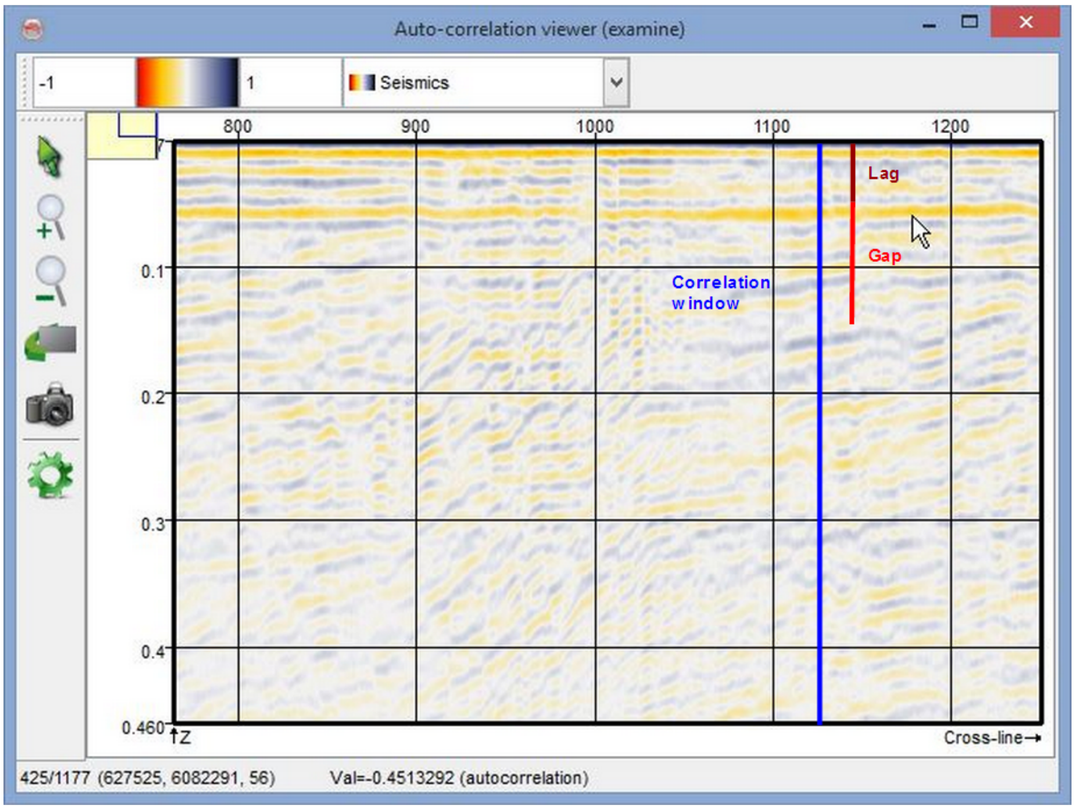


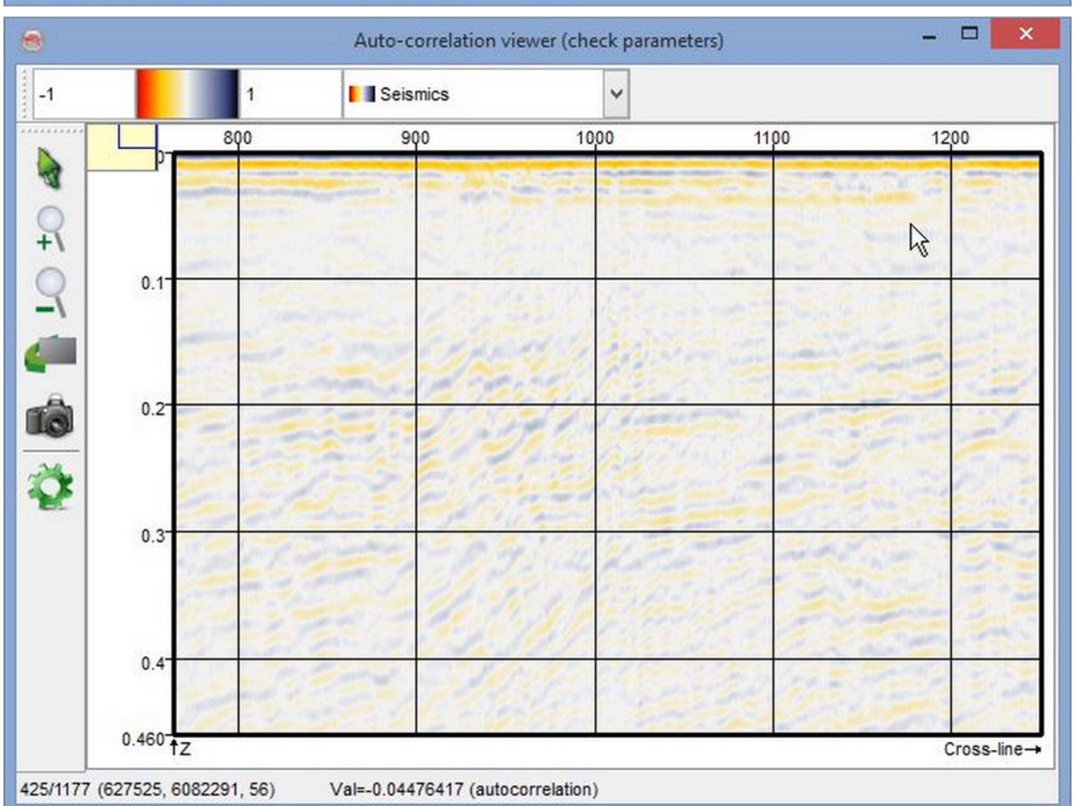
- **Correlation window** = the window for calculating the auto-correlation function.
- **Examine** = pressing this button will open a new window where the user selects a subset of the data (e.g one inline, or one cross-line) on which the result of the auto-correlation will be calculated and displayed for filter design purposes. This result can be used to determine the lag and gap size (see example below).
- **Lag-size** = the window length in the auto-correlation function that is unaffected by the filter. This window contains the wavelet-shape information.
- **Gap-size** = the window length in the auto-correlation function that the filter aims to blank. This window contains repetitive (multiple) information.
- **Noise level** = random noise that can be added to stabilize the filter derivation. The noise level is set to 1% by default.
- **Stepout** = a moving average across neighbouring traces that is applied prior to the calculation of the auto-correlation function. This trace averaging results in a smoother end-result. The higher the stepout, the smoother the result.
- **Input is** = The user can set the input to minimum or zero phase; if the input is zero phase, a phase rotation of 90 degrees is applied before applying the GapDecon filter.
- **Output is** = The user may set the output to zero phase, then the inverse 90 degrees rotation is applied after the filtering.
- **Check parameters** = check whether the (multiple) energy has indeed been removed. A QC of the parameters : the GapDecon-filter with the selected parameters, lag 44 and gap 100 (see example below), is applied on a user defined line and the auto-correlation of the filtered data is displayed in a 2d-viewer to check whether the parameters are correct. If they are not correct, the parameters can still be changed.

Spiking Deconvolution.
The Gap Decon attribute can also be used for Spiking Deconvolution or Whitening. The goal of Spiking Deconvolution is to flatten the output spectrum. This is achieved by shortening the embedded wavelet and attempting to make it as close as possible to a spike (zero-lag spike). One should keep in mind that the frequency bandwith of the data might limit the extent to which this whitening is possible. At higher frequencies, Spiking Deconvolution might cause an increase in noise.

Example

In the attribute defined above, we define an auto-correlation window between 300ms and 1200ms. After pressing '*Examine*' we can see that the following parameters could give us the result we desire: a lag of 44 ms and a gap of 100ms. Pressing 'Check Parameters' shows the effect this would have if we were to output this attribute with the current parameter settings:





Grubbs Filter

Name

Grubbs Filter -- Attribute that removes outliers from normally distributed data.

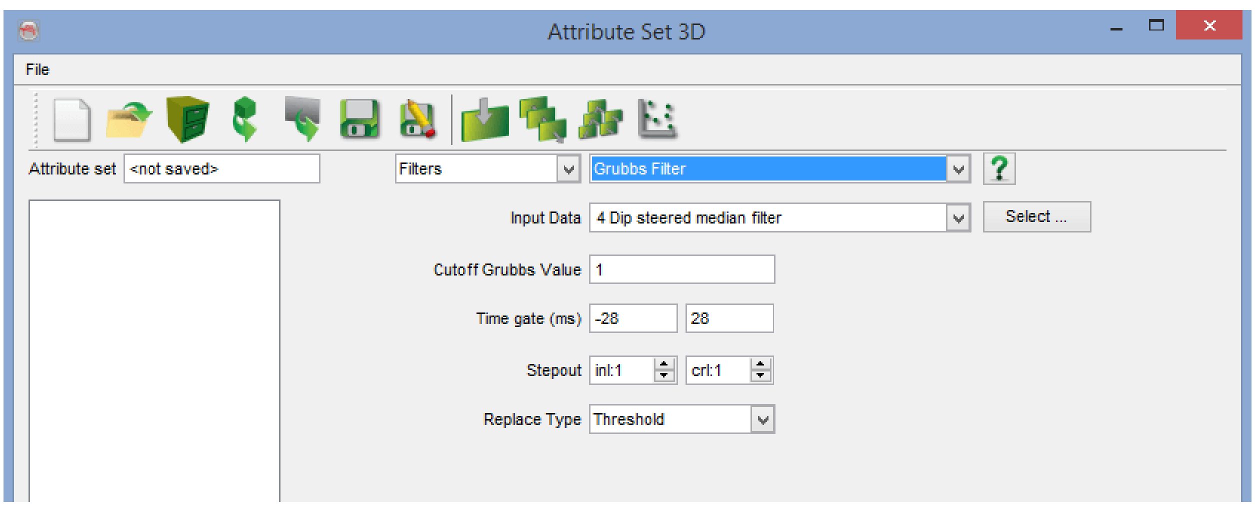
Description

Grubbs' test , (also known as the maximum normed residual test), is a statistical test used to detect outliers in a univariate data set assumed to come from a normally distributed population. It is based on the assumption of normality. That is, one should first verify that the data can be reasonably approximated by a normal distribution before applying. The test detects one outlier at a time. This outlier is expunged from the dataset and the test is iterated until no outliers are detected.

Please note: Multiple iterations change the probabilities of detection, and the test should not be used for sample sizes of six or less since it frequently tags most of the points as outliers.

For a full definition, including formulas, please see the Wikipedia entry.

Input Parameters



Horizon

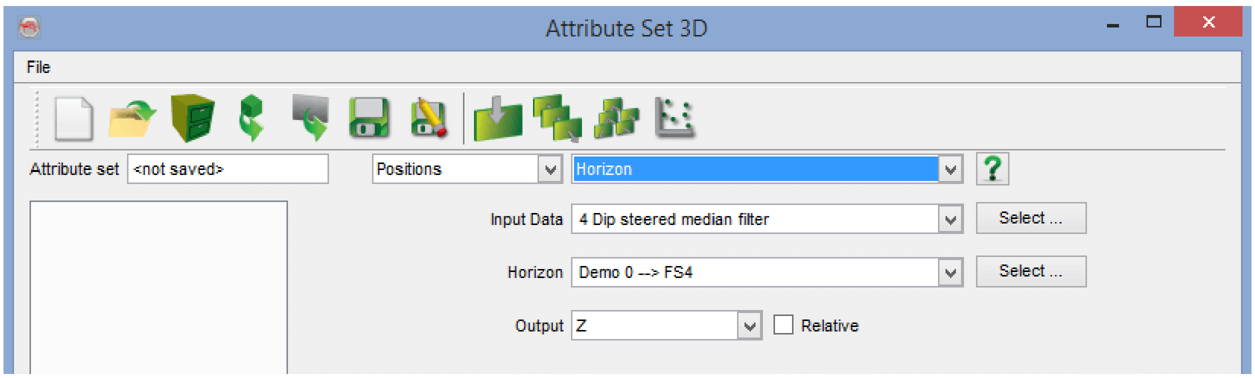
Name

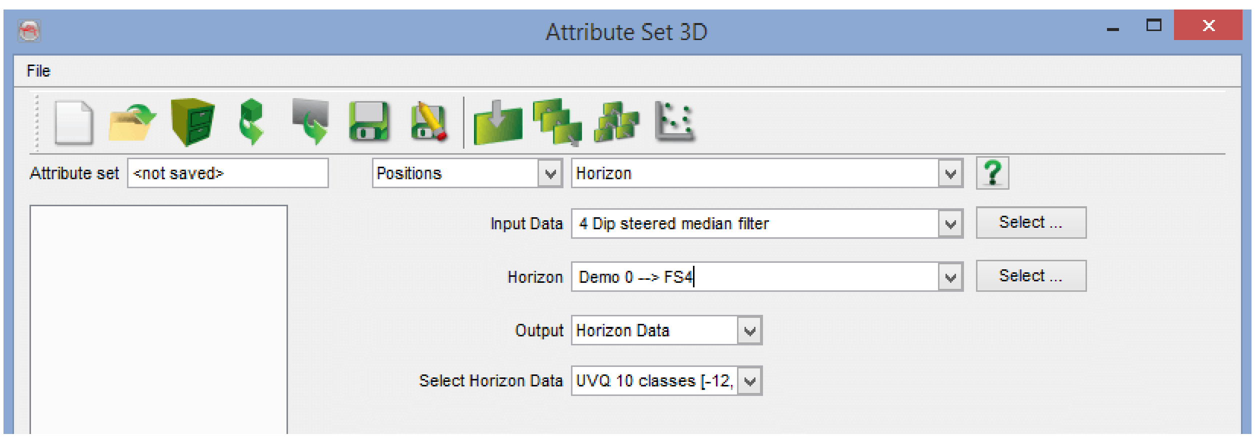
Horizon -- Attribute that enables advanced calculations on horizons

Description

This attribute is designed to extend the use of horizon data and can be used for several purposes. When applying it to a horizon only, it will give similar results as calculating attributes on the horizon.
However, when using the output of this attribute as input to a mathematical expression, the output can vary from just a simple thickness calculation to a highly advanced combination of several attributes. You can think of this attribute as different to the other attributes in OpendTect in the sense that it takes horizon data and converts it into seismic data which can be applied as input in volumes.

Input Parameters





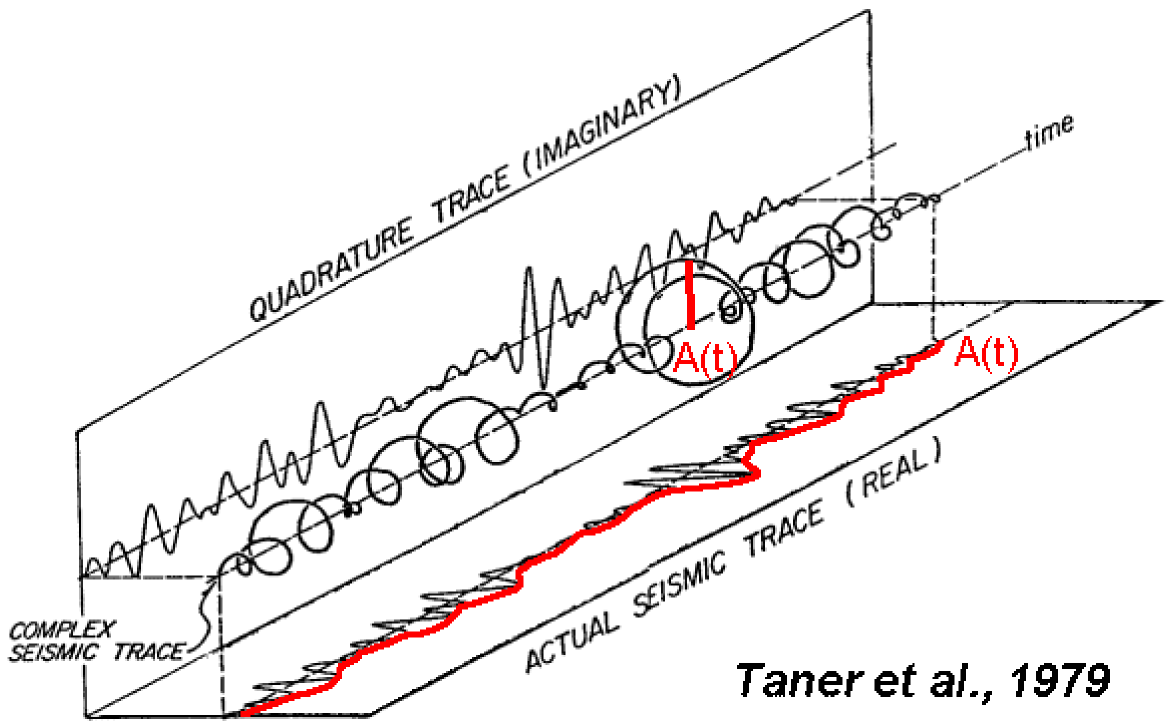
As shown above, there are two possible outputs, namely Z (depth) and horizon data. For each position on the horizon, the value (Z or horizon data) is used as output along the complete trace.

Instantaneous

Name

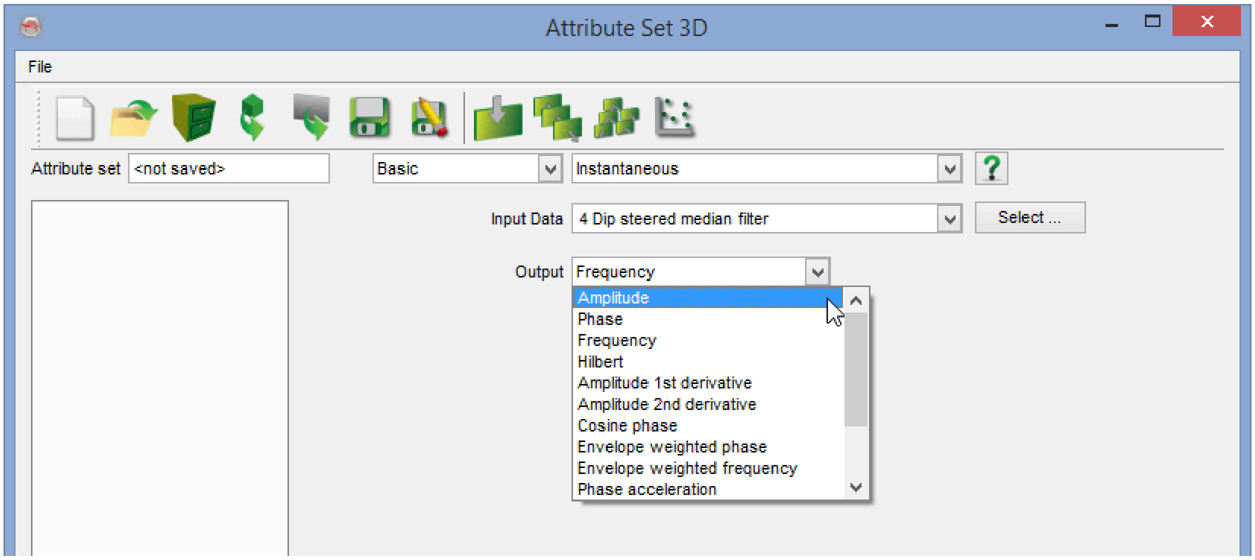
Instantaneous -- Attribute that returns a value at a single sample location.

Description
Instantaneous attributes are computed sample by sample, and represent instantaneous variations of various parameters. Instantaneous values of attributes such as trace envelope, its derivatives, frequency and phase may be determined from complex traces. (see figure below)



Complex Seismic Trace Attributes (From Taner et al.,)

The imaginary part of the complex trace is computed via the Hilbert transform. Possible outputs are:



Instantaneous Amplitude (Trace Envelope)
Outputs the instantaneous amplitude (or envelope) of the selected data volume at the sample location.
It can be used as an effective discriminator for the following characteristics:
Mainly represents the acoustic impedance contrast, hence reflectivity,
Bright spots, possible gas accumulation,
Sequence boundaries,
Thin-bed tunning effects,
Major changes in depositional environment,
Spatial correlation to porosity and other lithologic variations,
Indicates the group, rather than phase component of the seismic wave propagation.

- **Amplitude 1st derivative:** Time derivative of the instantaneous amplitude i.e time rate of change of the envelope. It shows the variation of the energy of the reflected events. It is used to detect sharp interfaces and discontinuities.

- **Amplitude 2nd derivative:** Second derivative of the envelope. It provides a measure of the sharpness of amplitude peak. It can be used to identify all reflecting interfaces within the seismic bandwidth.

Instantaneous Phase
Calculates the instantaneous phase at the sample location, it emphasizes spatial continuity/discontinuity of reflections by providing a way for weak and strong events to appear with equal strength.
This attribute is of central importance since it describes the location of events in the seismic trace and leads to the computation of other instantaneous quantities.
The instantaneous phase makes strong events clearer and is effective at highlighting discontinuities of reflectors, faults, pinch-outs, angularities and bed interfaces. Seismic

sequence boundaries, sedimentary layer patterns and regions of onlap/offlap patterns often exhibit extra clarity

The instantaneous phase relates to the phase component of wave-propagation, it is also used to compute the phase velocity.

- **Cosine phase:** Cosine of the instantaneous phase, also called normalized amplitude. It has the same uses as instantaneous phase with one additional benefit: It is continually smooth. By providing the +/-180 degree discontinuity that occurs with instantaneous phase, the cosine of instantaneous phase can be further processed (e.g, filtered and stacked) using conventional seismic processing tools. Amplitude peaks and troughs retain their position, but with strong and weak events now exhibiting equal strength
- **Envelope weighted phase:** Instantaneous phase, weighted by the envelope over the given time window
- **Rotate Phase:** Phase output is rotated through a user-specified angle

Instantaneous Frequency

Outputs the instantaneous frequency at the sample location.

The instantaneous frequency attribute responds to both wave propagation effects and depositional characteristics, hence it is a physical attribute and can be used as an effective discriminator.

Its uses include:

Hydrocarbon indicator by low frequency anomaly.

Fracture zone indicator, since fractures may appear as lower frequency zones.

Bed thickness indicator. Higher frequencies indicate sharp interfaces such as exhibited by thinly laminated shales, lower frequencies are indicative of more massive bedding geometries, e.g. sand-prone lithologies.

- **Envelope weighted frequency:** Instantaneous frequency, weighted by the envelope over the given time window
- **Phase acceleration:** Time derivative of the instantaneous frequency
- **Thin bed indicator:** Difference between instantaneous frequency and Envelope weighted frequency
- **Bandwidth:** The absolute value of the envelope time derivative
- **Q factor:** Instantaneous frequency divided by the bandwidth

Hilbert (Quadrature Amplitude)

The quadrature trace is the imaginary part of the complex seismic trace (see image above), and can be computed from the real trace via the Hilbert transfrom.

Both the real trace and its quadrature counterpart share the same amplitude spectrum; the quadrature however is phase rotated by 90 degrees. Zero-crossings on the real trace transform to peaks and troughs on the quadrature trace and peaks and troughs on the real trace transform to zero-crossings on the quadrature trace.
The quadrature is used in various mathematical combinations to compute other complex trace attributes such as instantaneous phase and instantaneous frequency. it is sensitive to energy, frequency, and phase.

Log

Name

Log -- an attribute that returns a well log value.

Description

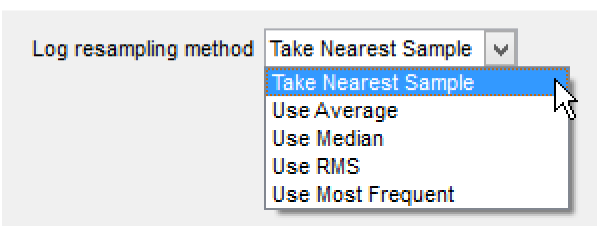
This attribute takes the value form an input well log and returns this value throughout the volume at the corresponding Z value.

Input Parameters

Firstly, the '*Input Well*' is chosen and, secondly, one of the associated logs from the auto-filled '*Select Log*' list.

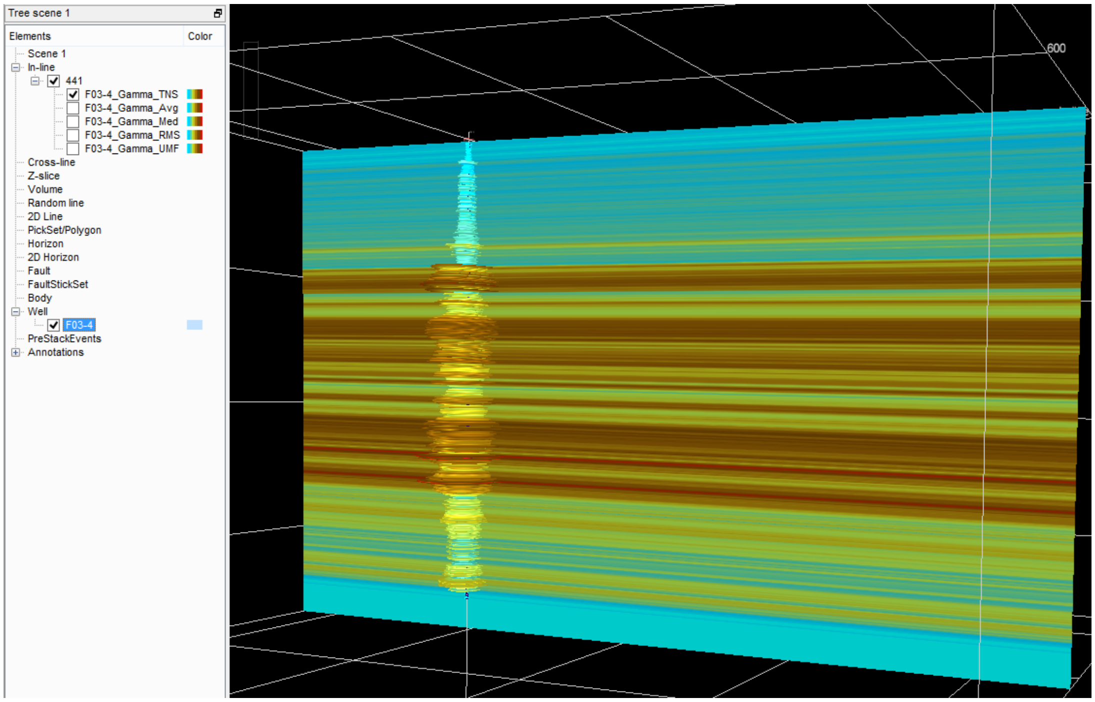
Output

There are five possible outputs:



Examples

Below is a shot of the Gamma Ray log from F03-4 displayed on inline 441 (F3 Demo dataset), as '*Take Nearest Sample*':



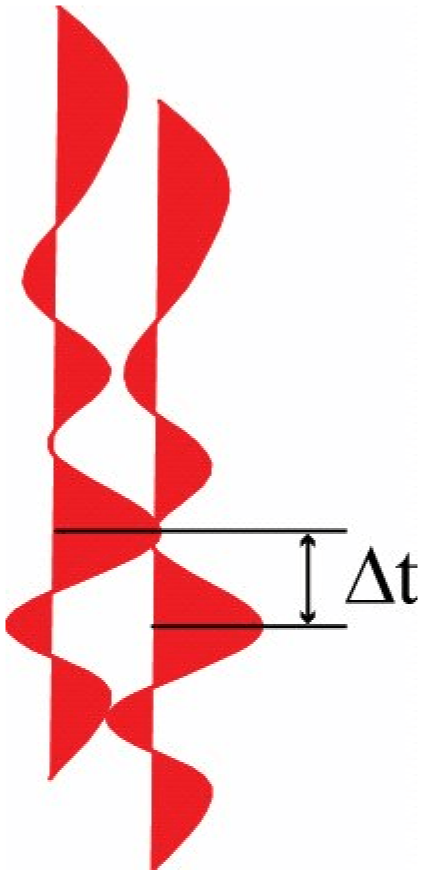
Match Delta

Name

Match Delta -- Attribute that extracts time shifts between similar events in different seismic volumes

Description

This algorithm extracts the time difference *delta t* between peaks in different seismic volumes. A search window is set to avoid loop-skips. After extracting all delta t values, they are interpolated. The resulting cube is the delta cube in ms or meter/feet which can be used for the Delta Resample attribute.



Extraction of delta t from two neighboring traces.

In a little more detail, the algorithm proceeds along these lines:

1. For each trace, all peaks are found.

   Result: a list of peaks and the time of occurrence.

2. Each peak is coupled with a counterpart if it is found to be within the specified search window. (If none are found, there is no addition to the list of differences.)

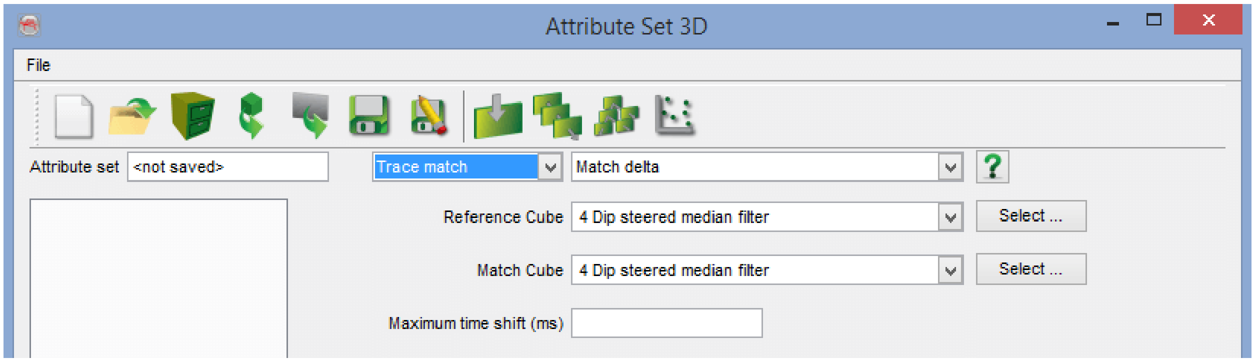
   Result: a list of time differences (deltas)

3. The delta list is interpolated (linear, constant extrapolation at the ends)

   Result: a delta value for each sample

After using this attribute, it will be clear that there are many 'jumps' and 'skips' in the displayed sections. Because we know that the deltas should not vary that quickly - not going from trace to trace, but also not going from sample to sample, these can be effectively removed by utilizing a double filter: first a rather large median filter over many traces and a 2 to 3 samples, then an average or FFT-based filtering in all directions. The resulting output should now be relatively 'noise-free'.

Input Parameters



Besides the reference and the match cube, a maximum time-window must be defined in order to avoid loop-skips.

Mathematics

Name

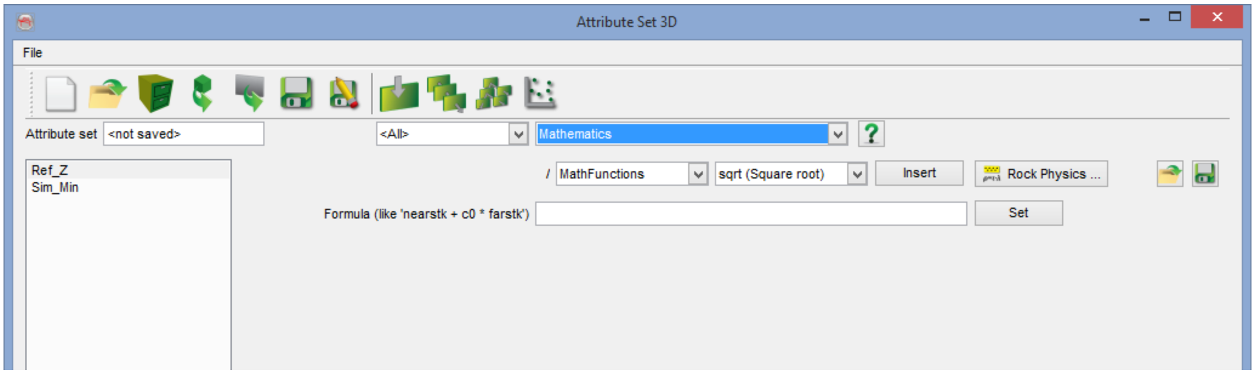
Mathematics -- Attribute that returns the result of a user-defined mathematical expression

Description

The Mathematics attribute is a way to combine data from stored data and attributes into a new data.

The input data can be volumes, 2D lines or well logs. The output data has always the same dimension (3D, 2D, 1D respectively) as the input, with the exception that 3D and 2D attributes can also be computed along surfaces (horizons and faults). The mathematical expression is calculated at each sample position.

Input Parameters



A mathematical expression is specified in the Formula text field with variables, constants, numerical values and/or recursive operators.

- Constants must start with the letter C and be following by a number: C1, C5, C9... The number does not necessarily start at 0 and does not have to be consecutive. Constants can be evaluated just like stepouts and time gates for the other attribute. They are not available for logs creation
- Variables expressions can be any string: seis, energy, sim, ... They can be called with a number between brackets like seis[-1] or seis[4], in which case the value represents a shift in number of samples. seis[-1] represents seis on the sample above, seis[4] represents seis four samples below
- Recursive expressions are a way to call back the result of the mathematical formula on a sample above for computation. This result is called with the expression OUT[-i] (case insensitive) where i is a number of samples

Note 1: There is no limit in number of variables/constants used in an expression.
Note 2: There are special constants for which it is not necessary to provide the numerical value: DZ is the sampling rate of the input data, Inl and Crl are respectively the inline and crossline numbers of the current trace. The special constant are case sensitive.
Note 3: Mathematics attributes containing a recursive equation should only be applied to the full volume/lines, not along a surface because of the integration length.

Parentheses ( ) are allowed and even welcome. Once the mathematical expression has be entered you must press *Set*. Then a table will appear where you will be able to assign data to the variables and values to the constants and recursive settings: Start time (or depth) of the recursive function and value at this time.

Supported operators are:

*+, -, *, /, ^, >, <, <=, >=, ==, !=, && (and), || (or), |xn* |(absolute value).

Supported mathematical functions are:

sin(), cos(), tan(), ln(), log(), exp(), sqrt(), min(), max(), avg(), sum(), med(), var(), rand(v) and *randg(std).*

Where avg is the average, med is the median, and var is the variance of the input parameters. The input parameters in parentheses should be separated by a comma.
The function *rand(v)* gives a random number from a uniform distribution between 0 and v. The function *randg(std)* generates a random number from a Gaussian (normal distribution) with standard deviation std and expectation 0.

Supported constants are:

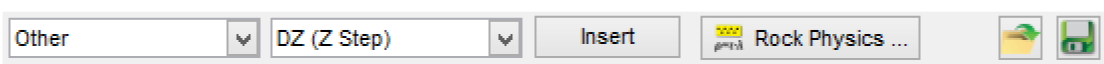
pi (3.1415927), *undef* (The OpendTect undefined value: 1e+30)

In addition it is possible to make logical operations using IF .. THEN .. ELSE statements. The following syntax must be used:

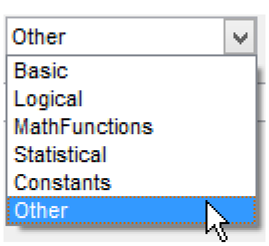
CONDITION ? OUTPUT IF TRUE : OUTPUT IF FALSE

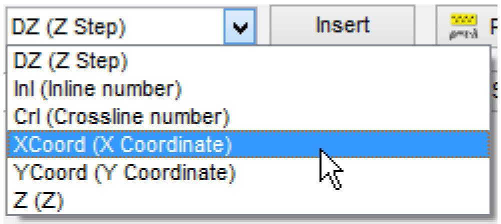
The three parts can be any set of variables and/or constants. IF..THEN..ELSE statements can be embedded like here:
CONDITION1 ? TRUE : (CONDITION2 ? TRUE : FALSE)

Other options



Predefined functions, constants, and operators can be inserted into the '*Formula*' field using a combination of the drop-downs, followed by the '*Insert*' button. These are grouped so:





The grouping '*Other*' contains the above.

Pressing this button, Rock Physics ..., will open the '[*Rock Physics*]' library of formulas. Clicking '*Ok*' in this additional window will directly insert the chosen formula into the '*Formula*' field.

Formulas in Mathematics can be saved ( ) and loaded from a stored location ( ).

Examples of expressions

- X0 == X1 returns 1 if X0 equals X1, 0 is returned if the statement is false, i.e. X0 is not equal to X1
- X0 != X1 returns 1 if X0 is not equal to X1 and 0 if X0 equals X1
- X0 && X1 returns 1 if X0 and X1 are not equal to 0. If one of these equals zero a 0 is returned
- X0 || X1 returns 1 if X0 or X1 is not equal to 0. If both are zero a 0 is returned
- |X0| returns the absolute value of X0
- X0 > 1 ? X1 : X2 means if X0 is larger than 1 return X1, else return X2
- sin(X0) returns the sine of X0

Let's say you have one input cube, say 'Cube1' and one attribute already defined, Energy40, which is the Energy per sample calculated in a [-20,20] ms window around the current sample. Then, you could define 'Damped amplitude' as;

- Select Mathematics attribute
- Enter formula: seis / energy
- Press Set

- Select 'Cube1' for seis and 'Energy40' for energy
- Provide a name for the attribute, and press Add as new

Additional examples

1 - *Centred differentiation example:* Centred differentiation can be coded using the formula (seis[+1]-seis[-1])/(2*DZ) where DZ is the sampling rate. Please note that for lateral shifts the reference shift attribute must still be used.

2 - Recursive filters can be created using the syntax "OUT[-i]". The most general form of recursive equation is the following:

y[n] = a0*x[n] + a1*x[n-1] + a2*x[n-2] + ...
+ b1*y[n-1] + b2*y[n-2] + ...

where x[] is the input volume, y[] is the output volume and the a's and b's the coefficients. n is the current sample number.

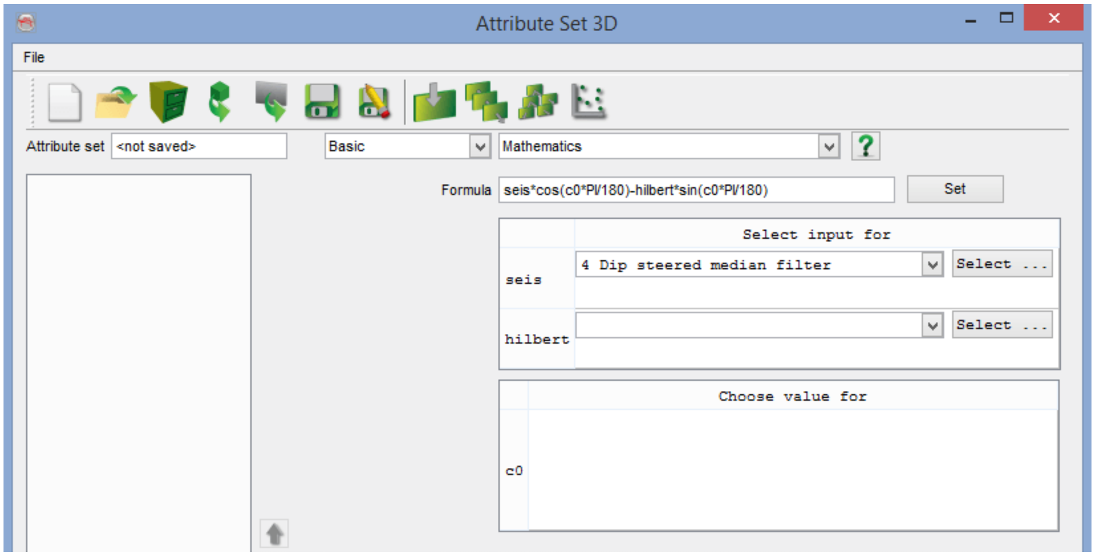
In the mathematics attribute the current sample index "n" does not need to be specified. Therefore the equation above can be entered as:
c0*x0 + c1*x0[-1] + c2*x0[-2] + c3*OUT[-1] + c4*OUT[-2] + ...
where OUT[-1] stands for y[n-1] and OUT[-2] stands for y[n-2]

For each instance of OUT[-i] a starting value and attached time/depth must be provided.

Two examples of low pass and high pass recursive filters are provided in the default attribute set "Evaluate attributes":
"Single pole low pass recursive filter" and "Single pole high pass recursive filter". Best results are achieved when providing an input of impedance or velocity type.

3 - The phase rotation is an attribute available in the Evaluate attribute set and in the dGB Evaluate attribute set.

This attribute allows the user to apply a phase rotation of any angle to the data.



It applies the formula: seis*cos(c0*PI/180)-hilbert*sin(c0*PI/180)
where seis is the seismic data
hilbert is the Hilbert transform of the seismic data
c0 is the applied angle for the rotation

c0 is in degrees.

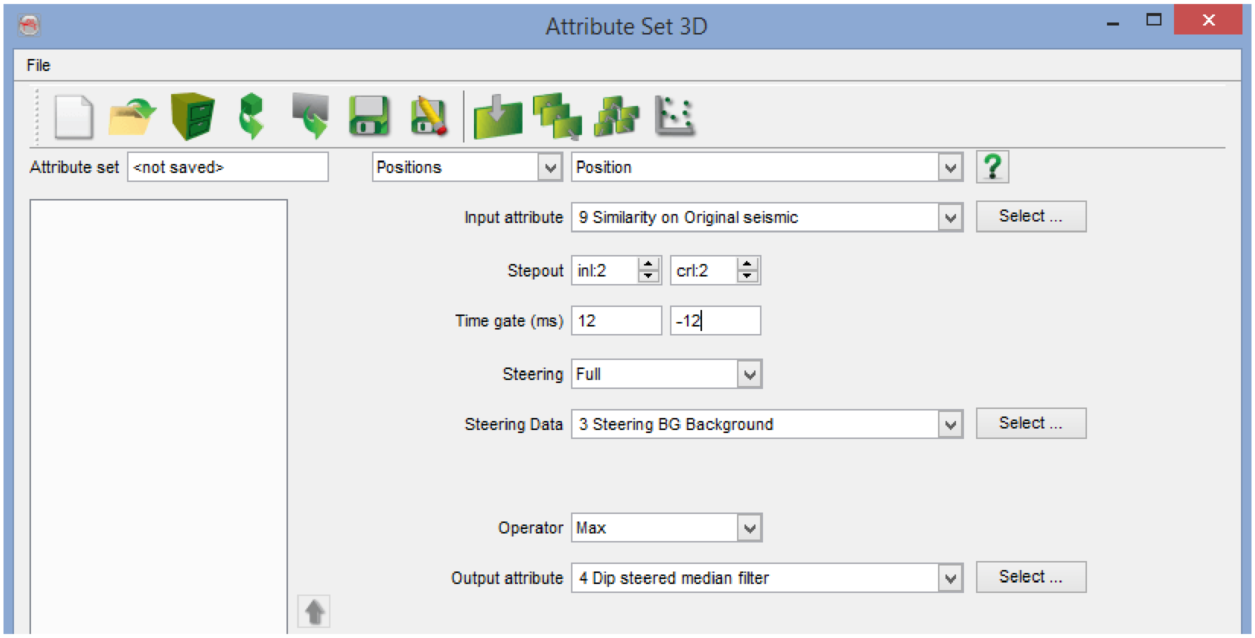
Position

Name

Position -- Attribute that returns any attribute calculated at the location where another attribute has its minimum, maximum or median within a small volume.

Description

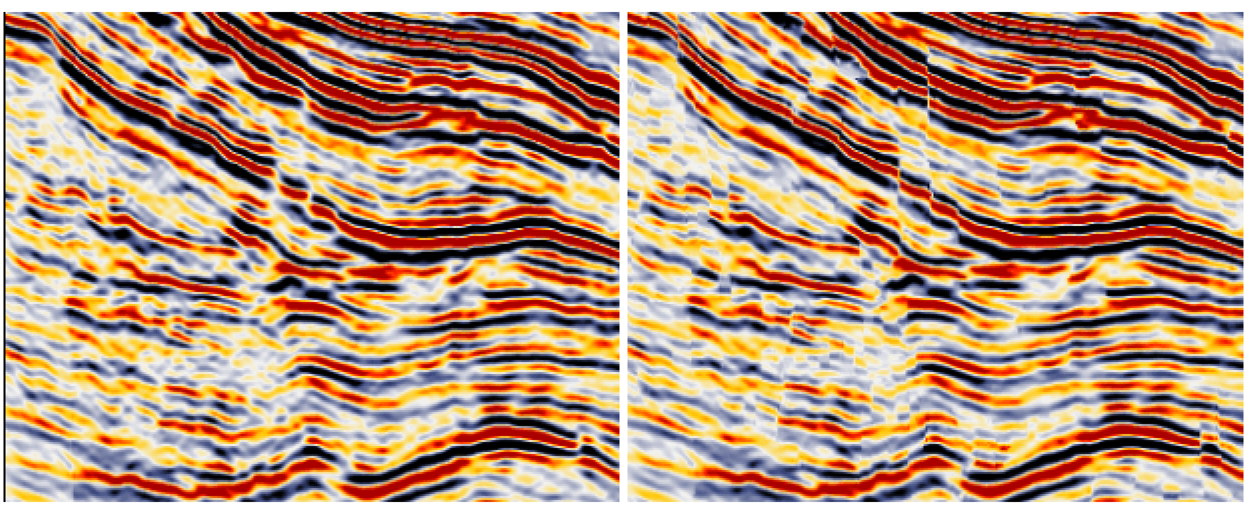
The input attribute is the criteria used to determine the position at which the output attribute has to be calculated. The stepouts and time-gate define the volume in which the input attribute is evaluated. In case of a 2D attribute as input, the inline/crossline stepout is replaced by a single trace stepout.



The Operator determines which position is returned from this analysis; the position of the minimum, maximum, or median of the input attribute. This position is the position at which the output attribute will be calculated.

Examples

The position attribute can be used for several purposes. For example, one can determine where in a small volume the energy is minimal and output the frequency at the location of this lowest energy. Also, application of the position attribute is an important step for Fault Enhancing Filtering. In this case, the user takes the Minimum Similarity as Input attribute and as Output, for example, filtered data (using Max as Operator). The stepout is set to, for example, 1 in both inline and crossline direction and the time-gate is defined [0,0]. The result below shows a sharply defined fault:

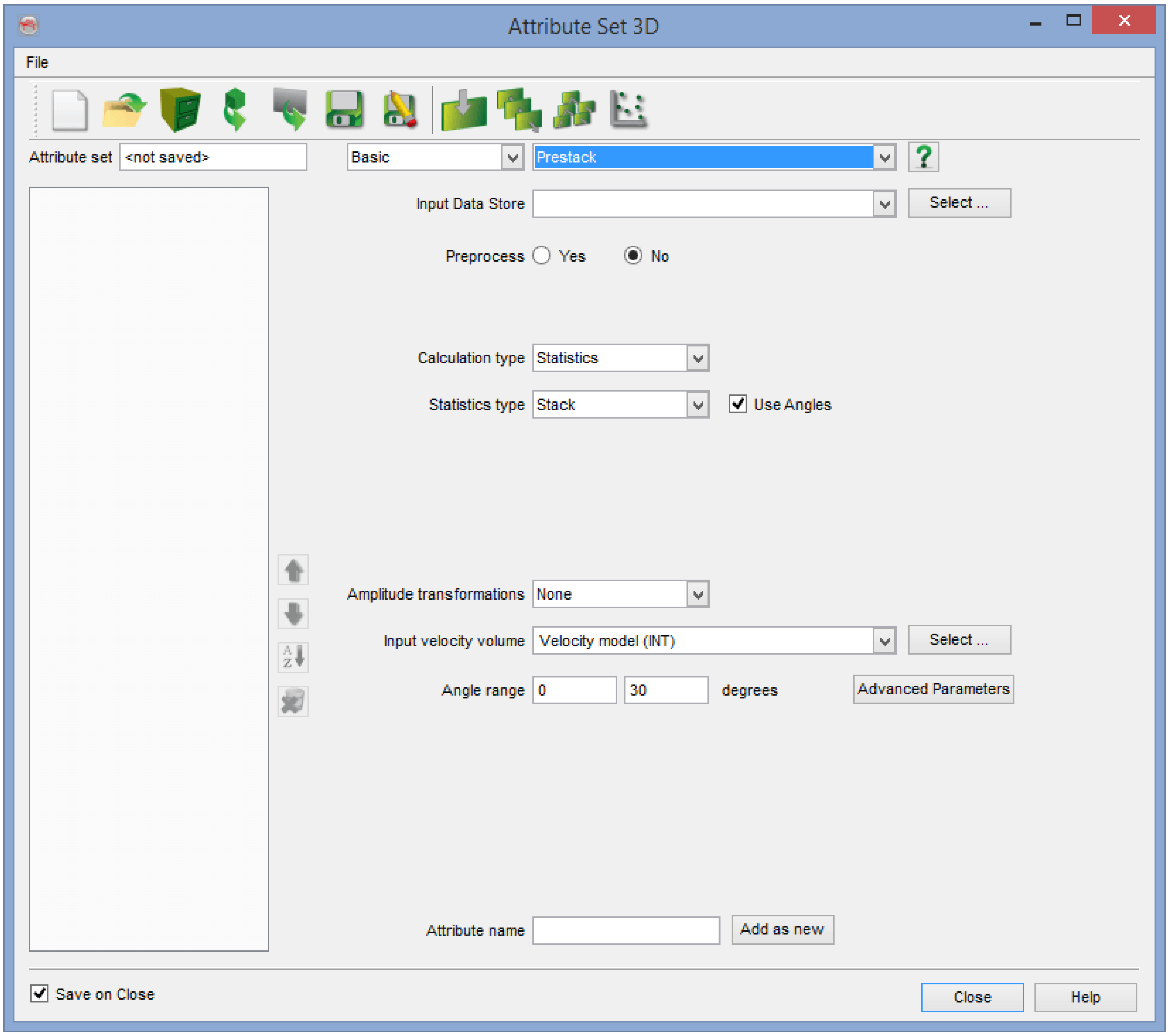


Before applying the position attribute After applying the position attribute

Prestack

Name

Prestack -- The pre-stack attribute can be used either to extract statistics on the gathers and their amplitudes, or to extract AVO attributes.



This attribute requires pre-stack data as input, but will output post-stack data. The following workflow is used in the above example:

- Pre-processing is applied to the gathers, like mute and AGC

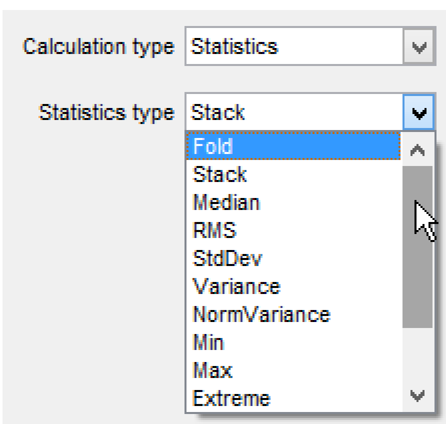
- The data is extracted in the offset range [500, 3000] m and sorted per CDP and depth sample
- For each CDP and depth a crossplot of amplitude vs. square root of the offset is made
- A least square regression is computed
- The intercept value is returned

The pre-processing of the gathers is optional. When used it must be setup in a separate window either by selecting an existing setup or creating a new one.

The offset range is used to define the extraction window. It requires absolute offset values, in meter or feet (depending on the survey unit definition). Optionally if angles were used instead of offsets while loading, then angle ranges can be used. Once again they have to be in the same unit as in the input trace (SEG-Y) headers.

After optional pre-processing and data extraction, a vector of amplitudes is available for each CMP/CIG, at each depth. From each vector either statistics or a least-square regression can be made.

Statistics:

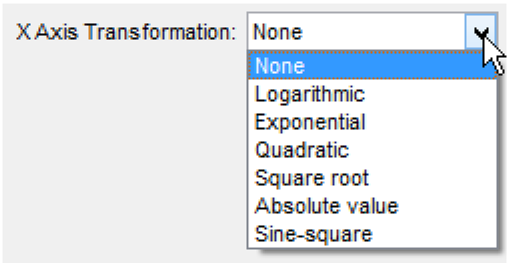


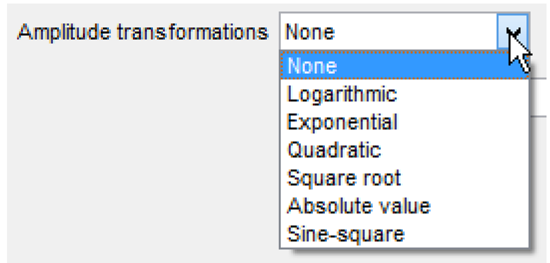
The above list of statistics can be returned. The highlighted output option provides the fold of each bin, while average or RMS may be used to generate full or partial stacks (partial stacking occurs if the offset range defined is not full). At the foot of this list are also: *Sum*, *SquareSum* and *MostFrequent*

Least Square:

The Least-Square calculation type is cross-plotting the extracted amplitudes, computing a least-square regression and returning one of its property.

Before the crossplot the amplitudes can be transformed using the following axis transformation (left hand-side):

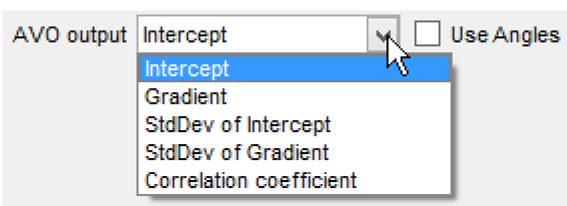




Similarly the offset values that are associated with those amplitudes can also be transformed (right-hand side). This is useful, e.g. to have *the sin*^{2} *of the angle* as x-axis.

AVO attributes will become AVA attributes if the X axis becomes Azimuth (i.e. angle). Please note that the azimuths (angles) must be provided in the trace headers when extracting AVA attributes.

The extractable AVO (Least-Square) attributes are Intercept, Gradient, their standard deviation and the Correlation Coefficient. It is also possible to have angle/offset constrained AVO attributes; by specifying the required angle/offset range instead of using the full range. Please note that further transformations can be achieved using the output as input to another attribute.



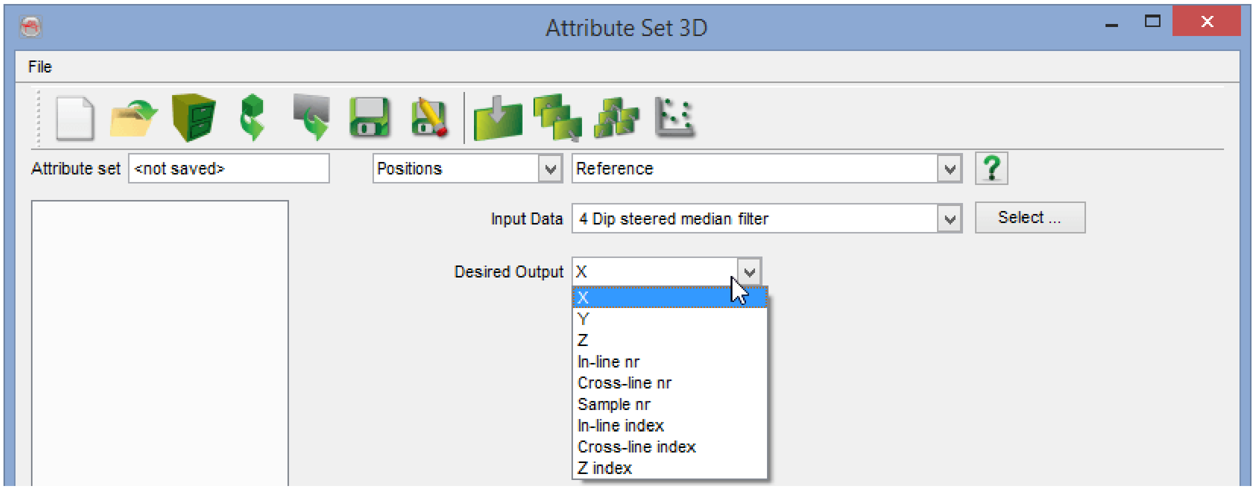
Reference

Name

Reference -- Attribute that returns the definitions of the extraction position

Description

The X, Y, Z, Inline number, Crossline number, Sample number, Inline index, Crossline index and Z index position of the reference (extraction) point is returned.



Output

- X: outputs the X-coordinate
- Y: outputs the Y-coordinate
- Z: outputs the depth according to the survey setup (e.g. miliseconds, meters or feet)
- Inline nr: outputs the inline number
- Crossline nr: outputs the crossline number
- Sample nr: outputs the sample number starting from 0 ms or the top of the survey when the survey has a negative start time. The first sample has sample number 1. If the survey has 4 ms sampling and starts at 0 ms, at 400 ms the sample number is 101. If the survey has 4 ms sampling and starts at 2500 ms, the first sample has sample number 626. If the survey has 2 ms sampling and starts at -200, the sample number at 0 ms is 101
- Inline index: outputs the number of inlines from the edge of the survey starting at 1 (e.g. If you have an inline range of 2000 to 3000, the inline index will range from 1 to 1001)
- Crossline index: outputs the number of crosslines from the edge of the survey starting at 1 (e.g. If you have a crossline range of 2000 to 3000, the crossline index will range from 1 to 1001)
- Z index: outputs the sample number from the top of the survey starting at 1. (Note: this attribute is identical to sample number if the survey has a negative starting time or starts at 0.)

Note: The Reference attribute replaces the old Reference time attribute. Attribute sets containing the old Reference time attribute will be automatically updated to the Reference attribute, with Z as output, which gives an identical output.

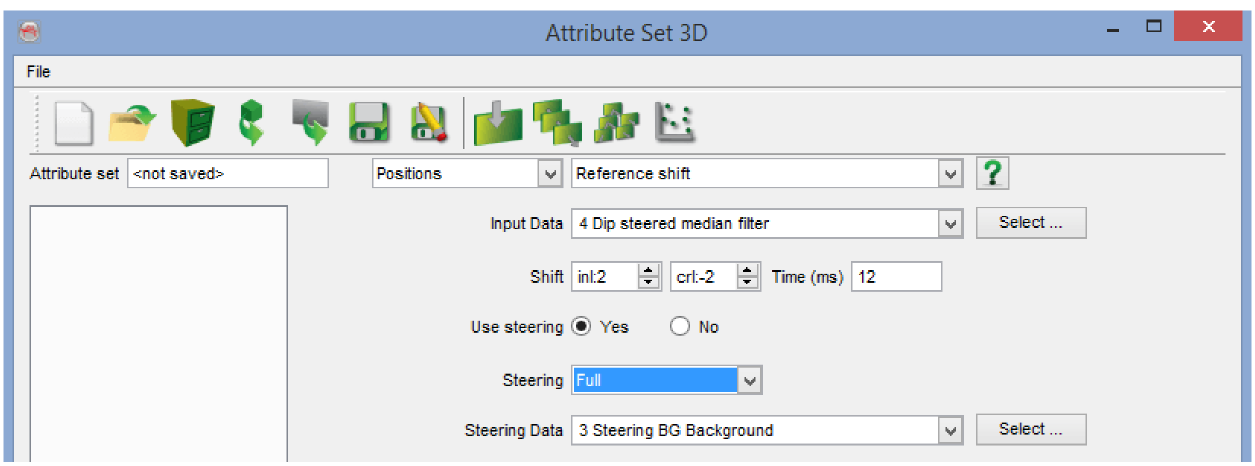
Reference Shift

Name

Reference shift -- Attribute that moves the extraction position in 3D space

Description

The *Input attribute* is extracted at the shifted position. The original reference (extraction) point has inline/crossline co-ordinates (0,0). Relative number 1 means the next inline or crossline, respectively. In case of input of a 2D attribute, the inline/crossline shift is replaced by a single trace shift. The vertical shift is specified in milliseconds using the *Time* option. There is also the option to use Steering while calculating the Reference shift.



The Input attribute is extracted at the shifted position. The original reference (extraction) point has inline/crossline co-ordinates (0,0) and Time 0.

The attribute will take the value from the shifted position and display it at the original reference point. Say that an original position (inl:0,crl:0,Time:0) has a shift of (25,25,100) applied to it, then this will take the output value at (25,25,100) and display it at (0,0,0). (In case of input of a 2D attribute, the inline/crossline shift is replaced by a single trace/time shift - ie: 25,100.)

It is important to remember that the vertical element of the shift is 'positive upwards' and 'negative downwards'.

> Think of a time-shift of +100:
> Original TWT = 500
> Shifted TWT = (500 + 100) = 600
> The value from TWT = 600 will be displayed at TWT = 500, giving the impression of a upward shift.
>
> Think of a time-shift of -100:
> Original TWT = 500
> Shifted TWT = (500 - 100) = 400
> The value from TWT = 400 will be displayed at TWT = 500, giving the impression of a downward shift.
>
> Think of an horizontal shift of (Inl:5,Crl:8):
> Original position = 100,150
> Shifted position = 105, 158
> The value from 105,158 will be displayed at 100,150.

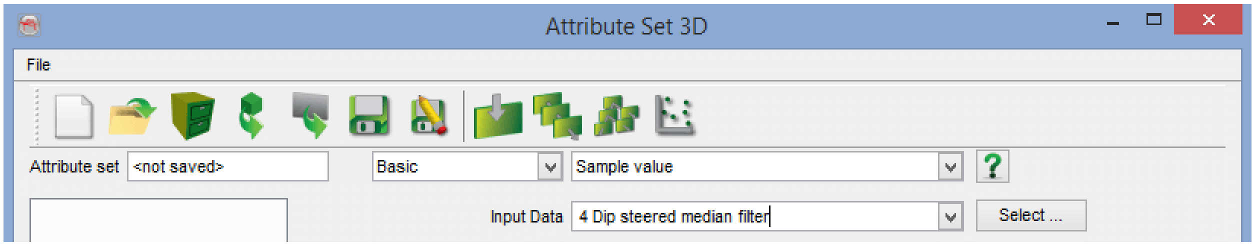
Shifting the reference position is a form of directivity that is useful in multi-attribute analysis. Examples of use:

- calculating reflectivity in an Acoustic Impedance volume. Values extracted at +4 and -4 ms would be as input to a Mathematics attribute to perform the actual calculation.
- highlighting flat spots - one may consider attributes that are extracted in a horizontally aligned window.
- measuring differences across a fault plane and comparison of such values.

Sample Value

Name
Sample Value -- Attribute that returns the input value at the sample location

Description
'Sample value' gives the value of the input volume at the sample location. It's only use is witihn the SynthRock Crossplot tool where it can be used to select a stored volume.



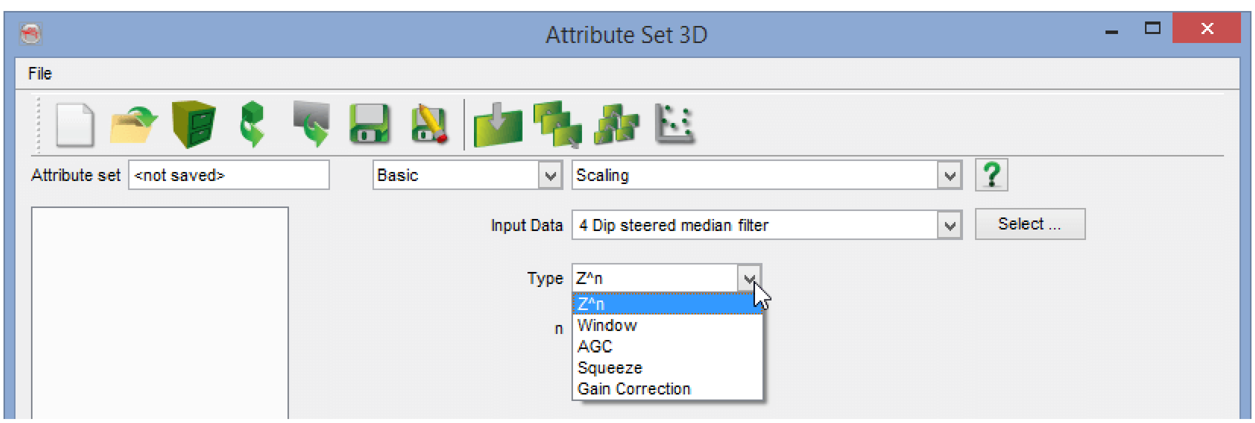
Scaling

Name
Scaling -- Attribute used for scaling of amplitude

Description

Input Parameters
The amplitude of the Input Data can be scaled in five modes:

- Using a time/depth variant weighting function
- Using weight(s) extracted in static time/depth window(s)
- Using Automatic Gain Control (single dynamic window)
- Using Squeeze ('non-clipping' limiter of range input)
- Using Gain Correction (correct/apply gain)



Output
The output amplitudes are always the ratio of the input amplitudes over a weighting function w(i), i being the sample index.

1. **Z^n scaling**
The weight function is defined by Z^n where Z is the time/depth of the current sample and n is a user-defined exponent:

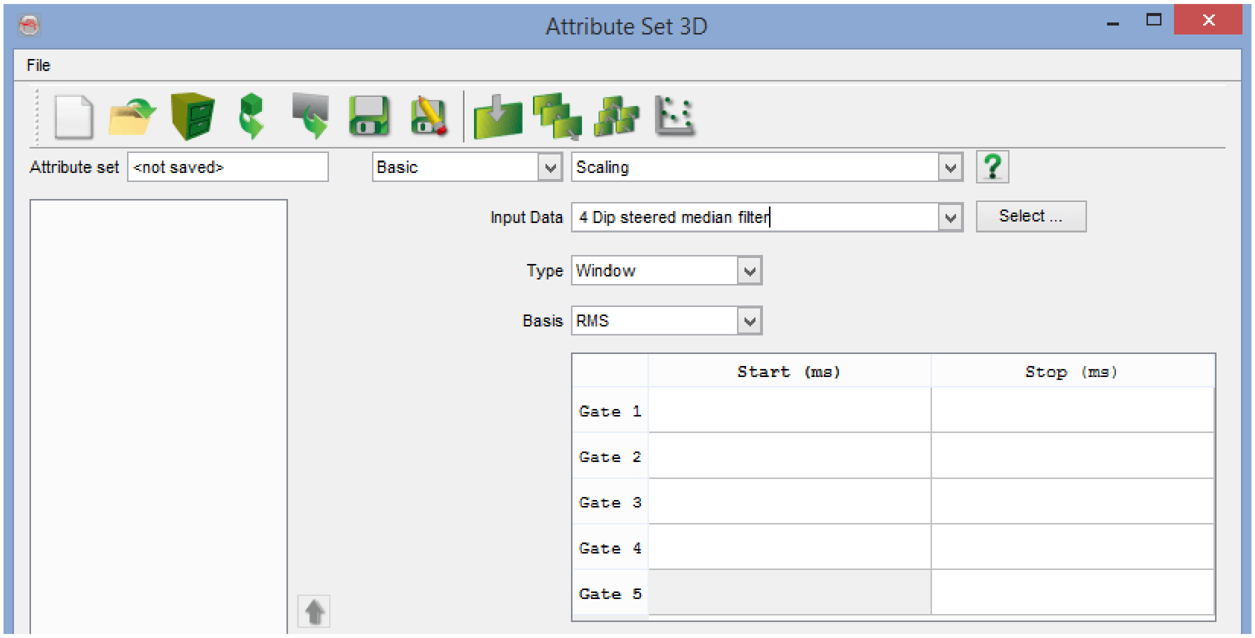
w(i) = Z(i)^-n where is Z is the time or depth of the ith sample.

The exponent is a float and thus it can be negative, positive and equal to zero (unity operator). An exponent larger than zero will apply a correction proportional to the depth, while an exponent smaller than zero will apply an inversely proportional correction with depth. The output amplitude is the normalized sum of the input amplitude using a weight equal to Z^n.

2. **Window(s) scaling**
The weight function is a step function: w(i) is constant over a static time/depth window, equal to the "basis" value than is computed from the input amplitudes using the following mathematical definitions:

- The Root Mean Square (RMS)
- The arithmetic mean
- The maximum
- A user-defined value (float)
- Detrend. This option removes the trend but rather than doing so following a constant α, it will detrend following αɣ + β. (See: http://en.wikipedia.org/wiki/Trend_estimation for further info.)

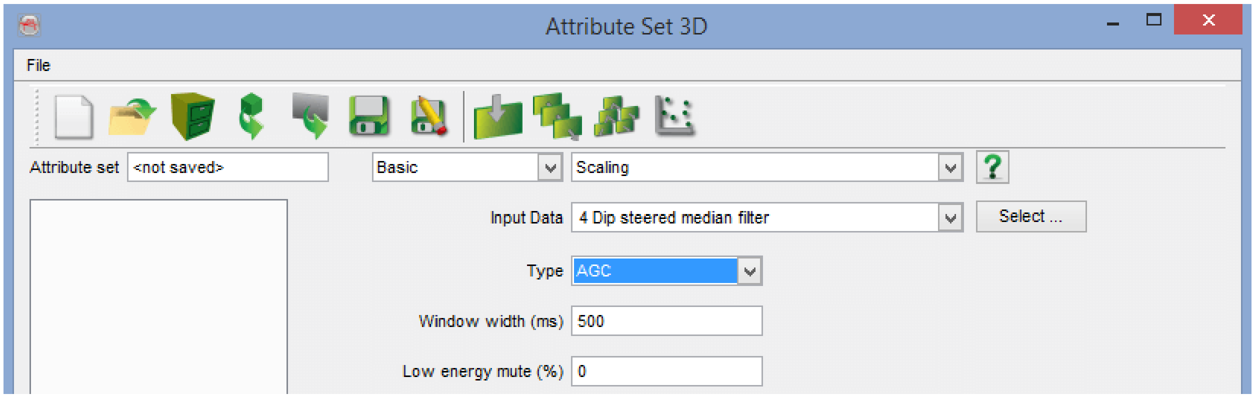
Please note that the window time/depths are float and do not need to be on a sample. The time/depths will be round to the nearest sample when defining the extraction window. Unlike most of the window definitions in the attribute engine you must provide in this scaling attribute absolute time/depths values and not values relative to the actual sample. A weight of 1 (no scaling) will be given to the samples not covered by a user-defined time gate. The weights are cumulative: If several windows overlap the output weight will be the sum of the "basis" output for the samples belonging to multiple windows.



3. **Automatic Gain Control scaling**

The AGC is a special case of the window scaling. In this case the window is defined relative to the actual sample and the "basis" is the energy value in that sliding window. The window width is a total size, i.e. the relative width corresponds to +/- half of the total window width. The low energy mute will mute the output samples that have an energy lower than a ratio of the trace energy distribution: The energy of the input trace is computed and the output values are sorted per increasing energy value.

Given 1000 samples, the energy of sample 250 (for a low energy mute at 25%) corresponds to the mute level: If the energy computed in the AGC window is lower than this level the value 0 will be output. Otherwise the sum of the squares over the number of (valid) samples will be output. Undefined values are not used for the computation and a zero is output if all values of a time window are undefined. In other words, the energy of all elements within the defined window are calculated and then ranked, then the (user-defined) percentage of the lowest energy levels are muted out.



4. **Squeeze**
The purpose is to put a limit to the value range of the input. Rather than clipping the value (which would be equivalent to a simple Mathematics formula like 'x0 > c0 ? c0 : x0'), the value can be squeezed into a range.

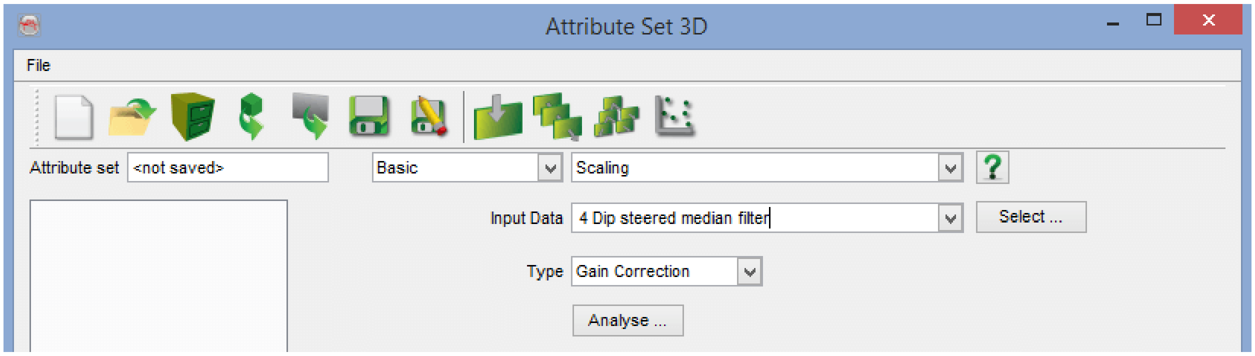
The first parameter is the 'Value range'. it defines the hard limits to the value range. One of these limits may be empty, signifying 'unlimited'.

The second parameter is the 'Untouched range'. If no limits are entered there, Squeeze will degrade to a simple clipping operation. If specified, it will squeeze rather than clip, constraining the squeezing to the ranges outside this range.

For example, Value range [0,10] and untouched range [2,8]. Values outside the [2,8] range will be modified to fit between [0,10]. This means the values in the range [-infinitiy,2] will be squeezed into the range [0,2] via a hyperbolic function. That function is continuous in value (and first derivative) at 2. Similarly, values higher than 8 will go somewhere between 8 and 10.

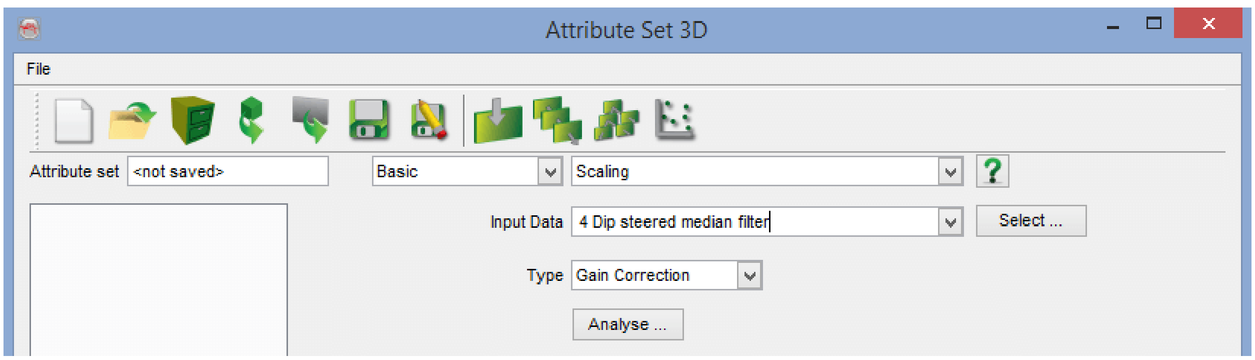
Example application: predicted porosities. Predictions of porosity tend to have values below 0. To counter this, you could squeeze all values below 1%. Use value range [0,] and untouched range [1,]. If you also want a more fuzzy upper limit, starting at 25% to absolute maximum 30, you may specify [0,30] and [1,25].

This is shown in the attribute set below:

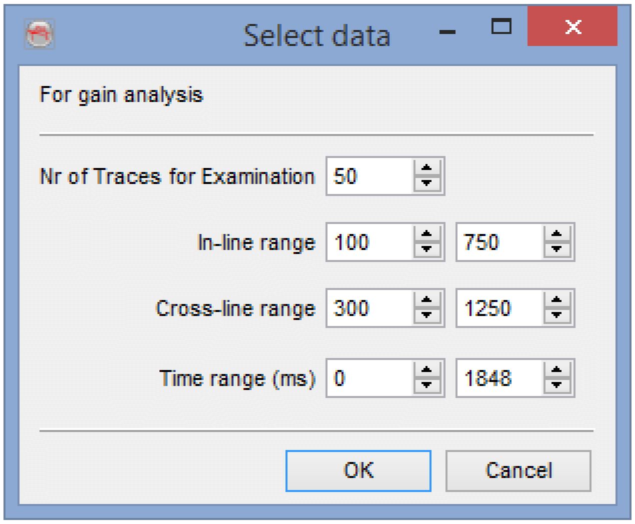


5. **Gain Correction**

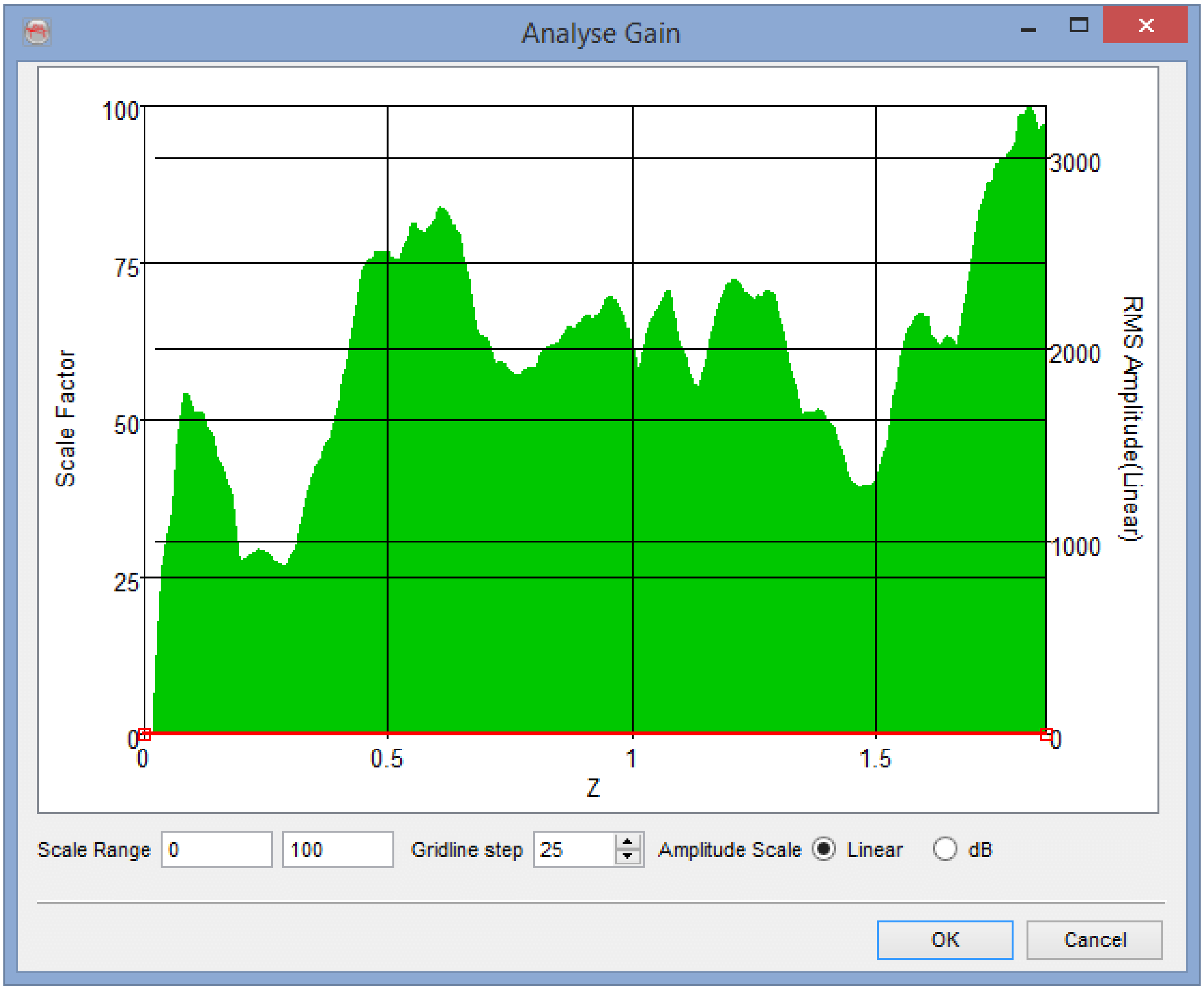
This attribute is used to correct for any undesirable gain applied previously or to apply a new gain function on the seismic data. This is applied by first selecting the input data for gain correction and clicking on *Analyse* button.

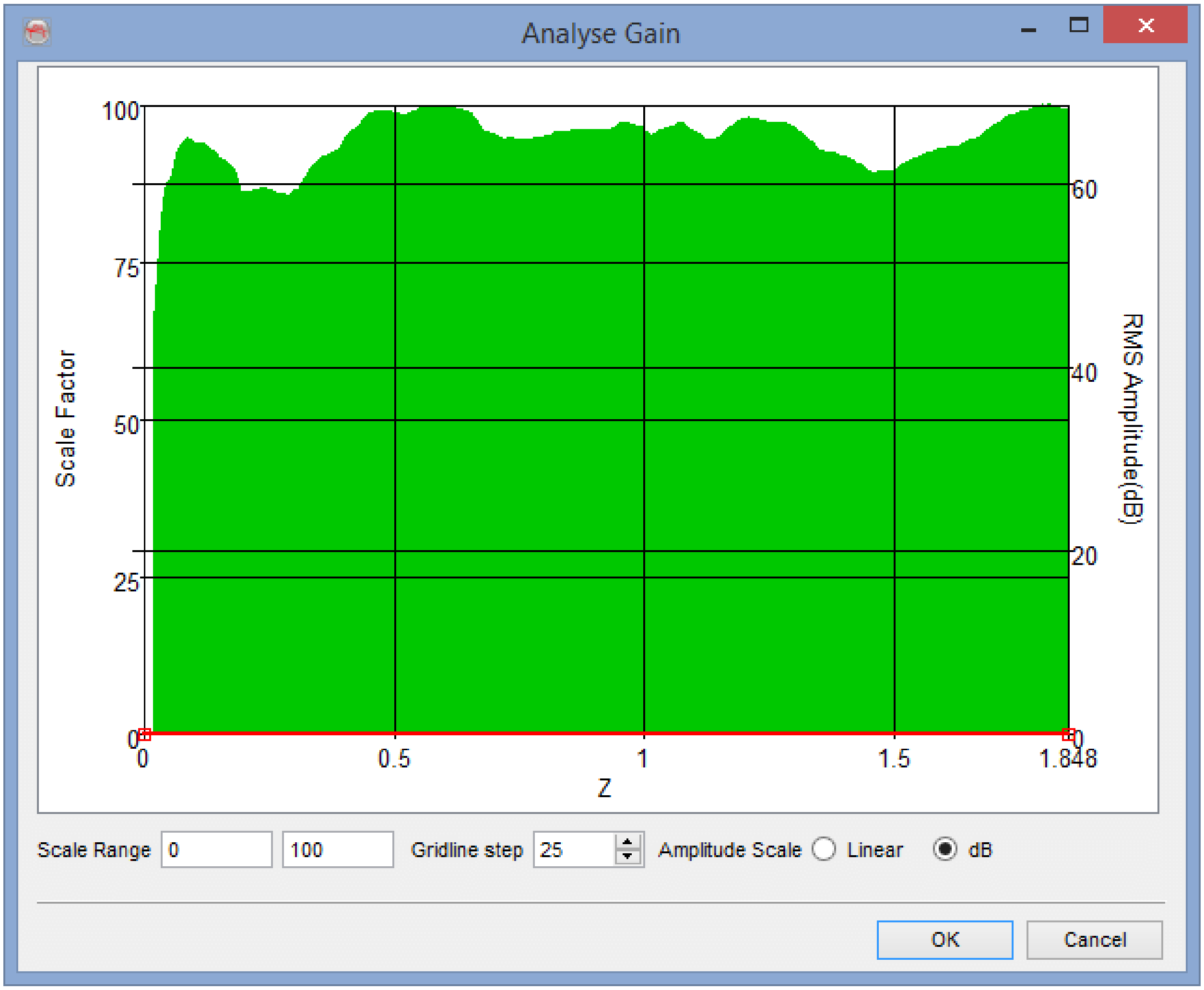


The newly popped *Select data* window requires specifying a number of random traces in the 'Nr of Traces for Examination' field for visually analyzing and defining the gain behavior in time/depth. The volume from which these random traces will be selected is outlined by modifying the inline, crossline and time ranges. Finally, *OK* is pressed to begin the examination of the random traces.



The *Analyse Gain* window has the 'Z' range (in seconds) of the seismic data as the horizontal axis, while the left vertical axis shows the 'Scale Factor' and the right vertical axis is the 'RMS Amplitude'. The amplitude scale can be set to 'Linear' or 'dB' (i.e. decibel) for visualization purposes. Further, the 'Scale Range' could be changed to use a different scale. The 'Gridline step' could be changed as the name says to modify the gridline steps.


Analyse Gain
Scale Factor
100
75
50
25
0
3000
2000
1000
0
RMS Amplitude(Linear)
0
0.5
1
1.5
Z
Scale Range
0
100
Gridline step
25
Amplitude Scale
Linear
dB
OK
Cancel



Finally, a 'Gain correction trend' can be defined by moving the red curve such that for any particular 'Z' interval a specific 'Scale Factor' range is used to scale the seismic amplitudes in that interval. For defining boundary points of these intervals user can double click on the red curve and move the curve as desired.

Pressing *OK* will save the 'Gain correction curve'.

Semblance

Name

Semblance -- Attribute that returns a value (semblance coefficient) which is a measure of multichannel coherence.

Description

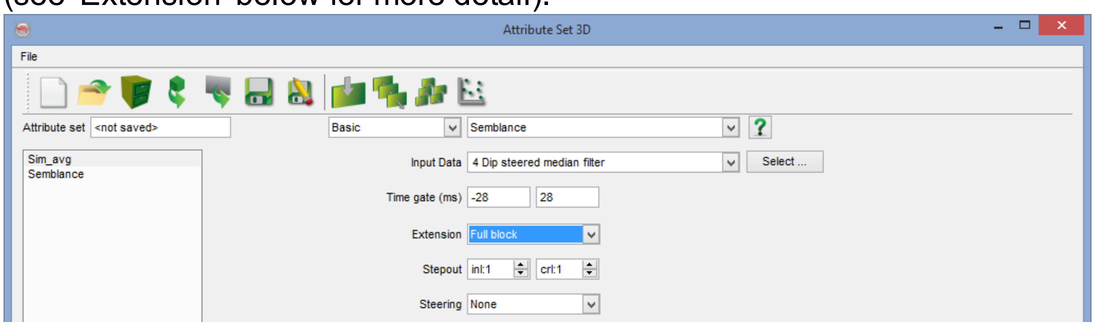
Semblance is, essentially, a measure of how similar a particular trace is to a group of traces within a user-defined correlation window. This attribute will calculate the semblance coefficient and return a value between 0 (completely dissimilar) and 1 (identical). This is done by summing the squares of the energies within the stack and dividing by the sum of those energies.

> **Note:** Whereas the Similarity attribute works by comparing pairs in given positions, Semblance uses all points within the window to create the score.

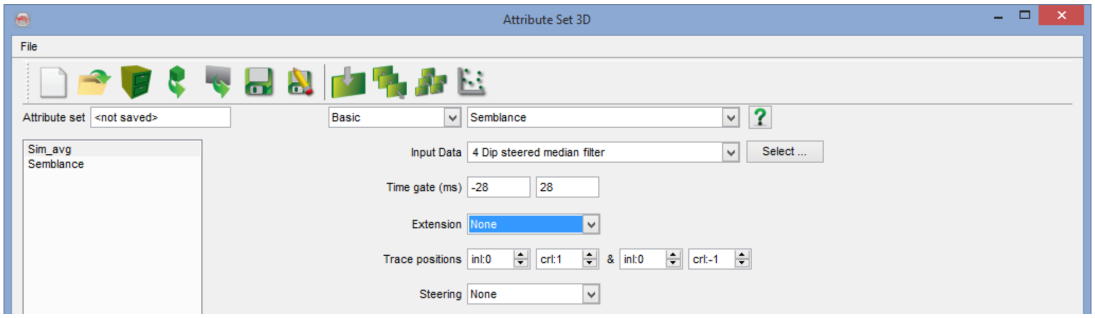
Input Parameters

The size of the correlation window is defined by the Time Gate and Stepout, with the 'Extension' defining its shape.
Steering may be used to enhance the results, if the DipSteering plugin is installed (with a valid license.)

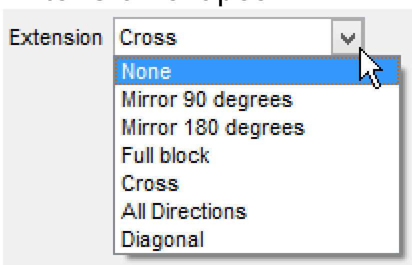
> **Time-Gate:**
> Defines the size of the vertical element in the correlation window (ie: the trace length to be used in the calculation).
>
> **Stepout:**
> Applies for *'Extensions'* of shape *Full block*, *Cross*, *All Directions* and *Diagonal*. Defines the extent of the horizontal element in the correlation window (see 'Extension' below for more detail).



> **Trace Position:**
> Applies for *'Extensions'* of shape *None*, *Mirror 90 degrees* and *Mirror 180 degrees*. Defines the positions of the traces to be used in the correlation window (see 'Extension' below for more detail).



Extension shapes:



- *None:* Semblance is calculated using only the traces defined in 'Trace positions'.
- *Mirror 90 degrees:* Semblance is calculated using two additional traces: the defined trace pair (as in 'None') and the trace obtained by 90° rotation. (Not available for 2D data)
- *Mirror 180 degrees:* Semblance is calculated using two additional traces: the defined trace pair (as in 'None') and the trace obtained by 180° degree rotation.
- *Full Block:* Semblance is calculated using all possible traces in the column defined by the time-gate/step-out. Beware of potentially long processing times with large time-gates/stepouts.
- *Cross:* Semblance is calculated using all possible traces in the '+' - shaped column bounded by the time-gate/step-out.
- *All Directions:* Semblance is calculated using all possible traces in the ' ✳ ' -shape bounded by time-gate/step-out. This is the extension found to be most useful: it gives a degree of accuracy almost equal to that of 'Full Block' but with significantly less processing time (depending on the step-out, up to a factor of 10).
- *Diagonal:* Semblance is calculated using all possible traces in the 'x' - shaped column bounded by the time-gate/step-out.

References

For a very concise explanation of semblance, please see: http://wiki.seg.org/wiki/Dictionary:Semblance

Similarity

Name
Similarity -- Multi-trace attribute that returns trace-to-trace similarity properties

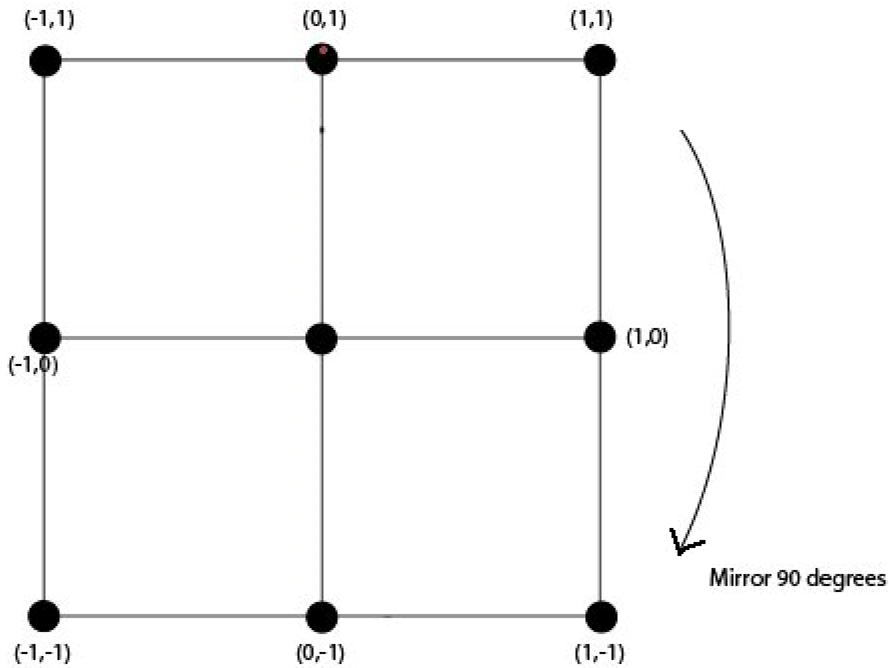
Description
Similarity is a form of "coherency" that expresses how much two or more trace segments look alike. A similarity of 1 means the trace segments are completely identical in waveform and amplitude. A similarity of 0 means they are completely dis-similar.

In OpendTect, we favor a different approach. We first try to find the direction of best match at every position, which is a result by itself: the dip. By using this dip we can then calculate the best Similarity between adjacent traces. Similarity is based on fundamental mathematics: the samples of the trace are seen as components of a vector, and the Similarity is defined in terms of distance in hyperspace.

The point about using the Similarity is that it's mathematically simple; it is very clear what is going on. Then, by combining different kinds of similarities and other attributes, you can always get much better results with lots less computing time.

Consider the trace segments to be vectors in hyperspace. Similarity is then defined as one minus the Euclidean distance between the vectors, normalized over the vector lengths.

The trace segments is defined by the *time-gate* in ms and the positions specified in relative co-ordinates. In case of using input from 2D data, the trace positions are defined by a trace step-out only, not by inline and crossline stepout. The *Extension* parameter determines how many trace pairs are used in the computation. This is visualized in the image below.



Definition of trace positions relative to the reference point at (0,0).

Input Parameters

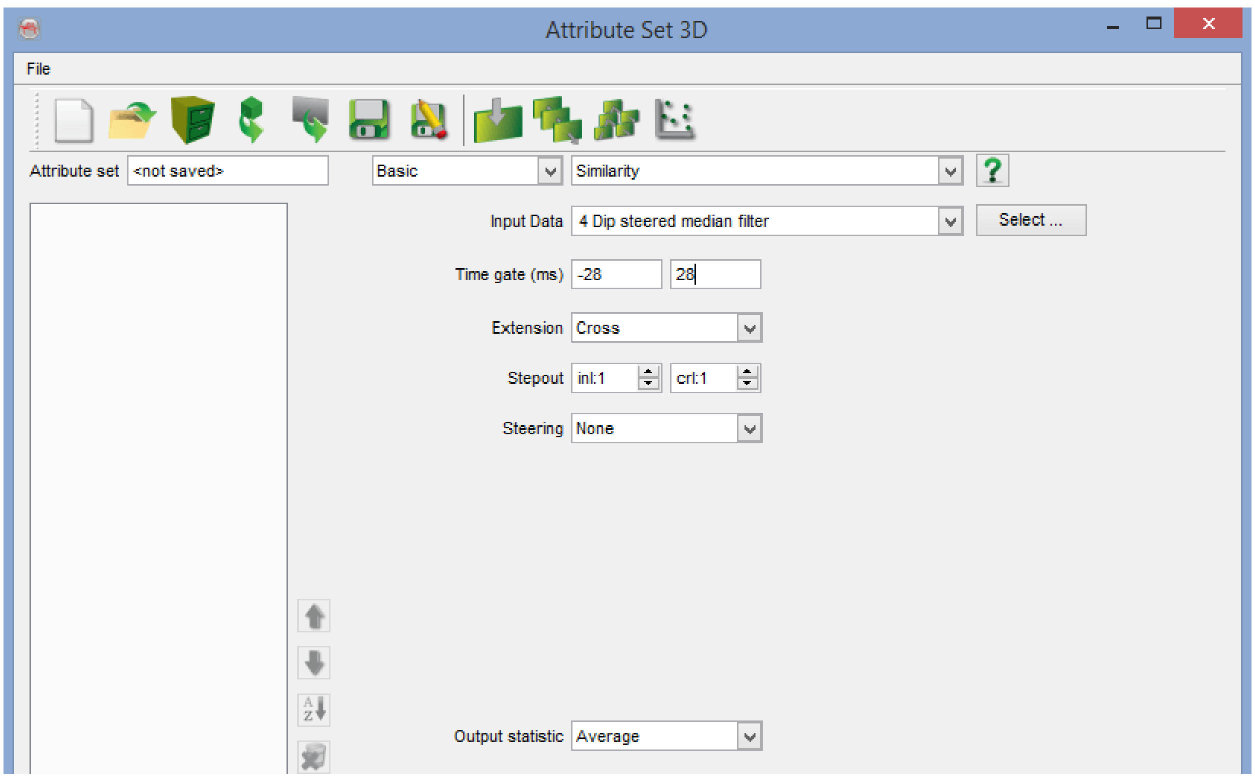
Extension Definitions

- None: Only the similarity between the pair of traces defined in 'Trace positions' is calculated.
- Mirror 90 degrees: Two similarities are computed: one for the defined trace pair (as in 'None') and one for the pair obtained by 90 degree rotation. (Not available for 2D data)
- Mirror 180 degrees: Two similarities are computed: one for the defined trace pair (as in 'None') and one for the pair obtained by 180 degree rotation.
- Full Block: Similarities between all possible trace pairs in the rectangle defined by the step-out are computed.
- Cross: Similarities between all possible trace pairs in the '+' -shape defined by the step-out are computed.
- Diagonal: Similarities between all possible trace pairs in the 'x' -shape defined by the step-out are computed.
- All Directions: Similarities between all possible trace pairs in the ' ' -

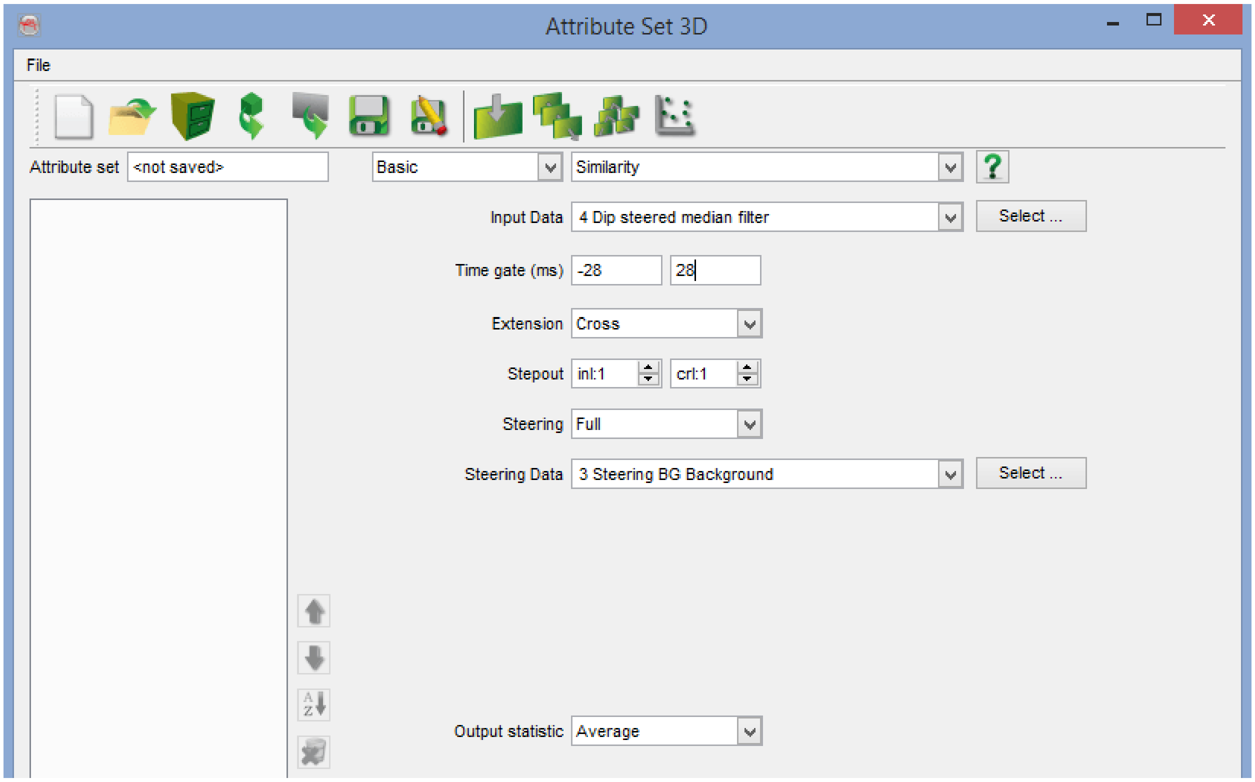
shape defined by step-out are computed. This is the extension found to be most useful: it gives a degree of accuracy almost equal to that of 'Full Block' but with significantly less processing time (depending on the step-out, up to a factor of 10).

The attribute returns the statistical property specified in *Output statistic*. The *Steering* option enables the user to follow the local dip to find trace segments that should be compared instead of comparing two horizontally extracted trace segments. The *Steering* option supports five different modes of data-driven steering: None, Central, Full, Constant direction Steering and Browse dip.]

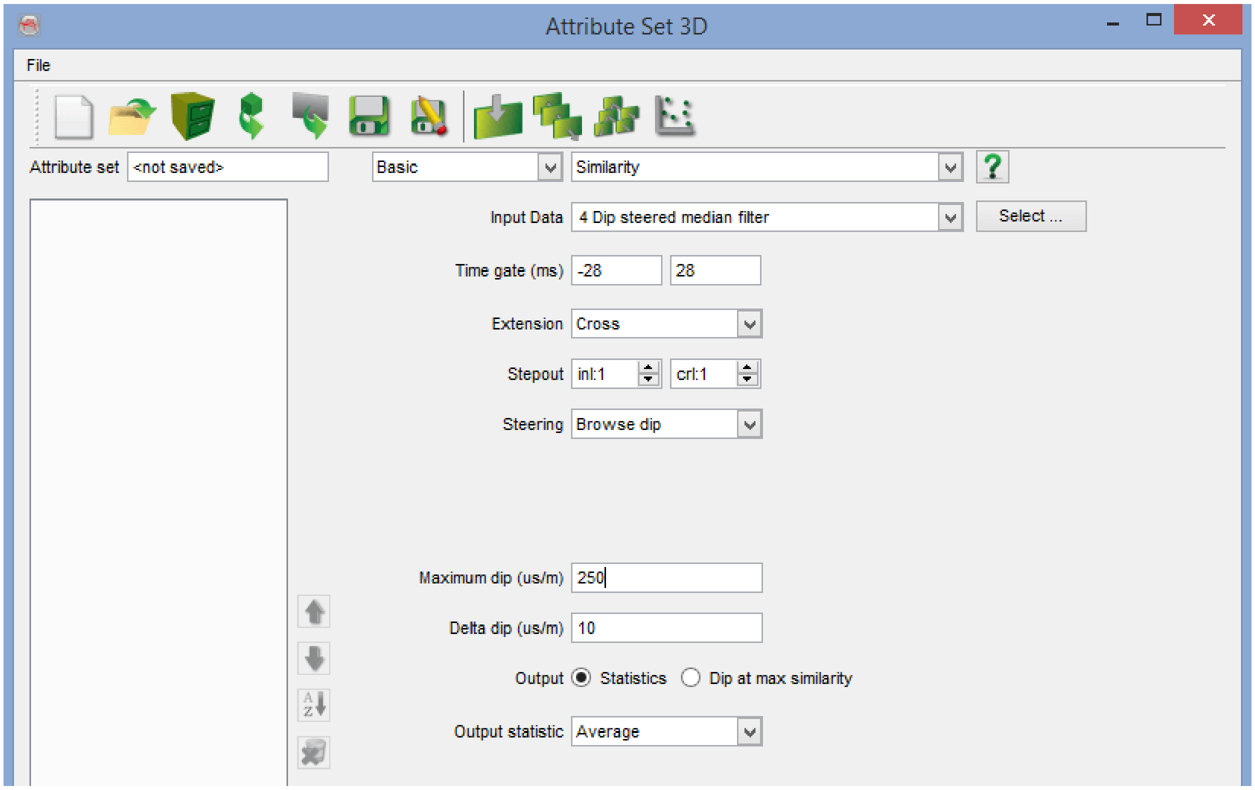
Similarity "None" Steering: This option is used when non steeringcube algorithm is used. This is ok in the case the layering is mainly horizontal (with less dip).



However, in very complex geology, the similarity result using "None" as steering option will deteriorate. Full steering should be used instead. The Dip-Steering plugin is required.



Another steering option to use is the "Browse dip". This is a similarity feature acting as a 'Coherency' attribute.



It enables the calculation of 'Similarity' by comparing one trace with the next trace. Then a value between 0 (not similar at all) to 1 (completely similar) is awarded. In order to compare traces, two variables should be specified:

The 'Maximum dip' represents the maximum dip in microseconds per metre (μs/m), relative to an event in one trace, in which the algorithm will look for similar events along the neighbouring trace. Default is 250.

The "Delta Dip" is a variable which represents the window in microseconds per metre (μs/m) which is shifted along the neighbouring trace to detect similar events whithin the earlier specified 'Maximum Dip'. The closer the value to 1 the more precise the results will be. The default value is 10. Using this value will result in a good balance between calculation time and quality of the results, this also depends on the quality of the data itself.

Mathematical description

Let us assume two vectors X, Y of length N=15 samples:

Xi, i=1,15

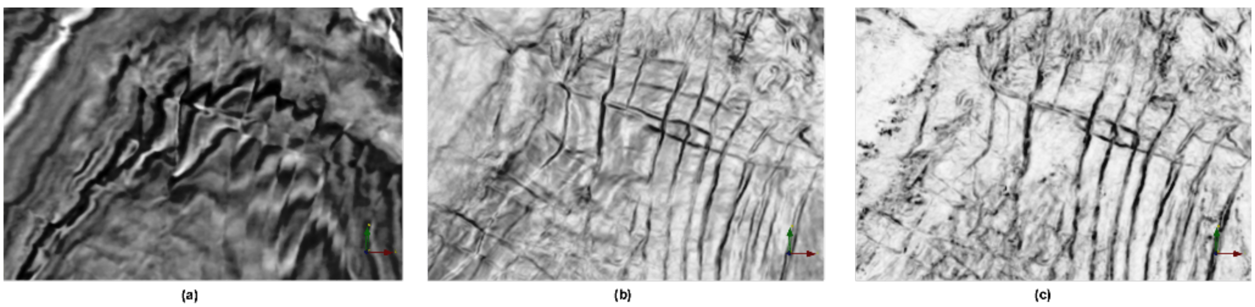
Yi, i=1,15

The similarity is 1 minus the Euclidean distance between the vectors divided by the sum of the length of each vector. Please note that the length of a vector is its L2 norm, also called RMS value:

$$\boxed{sim = 1 - \frac{\sqrt{\sum_{i=1}^{N}(X_i - Y_i)^2}}{\sqrt{\sum_{i=1}^{N} X_i^2} + \sqrt{\sum_{i=1}^{N} Y_i^2}}}$$

$$sim = 1 - \frac{\sqrt{\sum x}}{\sqrt{y}}$$

Examples



An example timeslice is highlighting fault structure: (a) Dip-steered Filtered Seismic, (b) Non-Steered minimum Similarity, (c) Steered minimum Similarity. Notice that the definition of faults has been improved with the Similarity attributes. The steered minimum Similarity (c) is highlighting precise fault definitions as compared with the result of non-steered minimum Similarity (b).

Spectral Decomposition

Name
Spectral decomposition -- Frequency attribute that returns the amplitude spectrum (FFT) or waveletcoefficients (CWT)

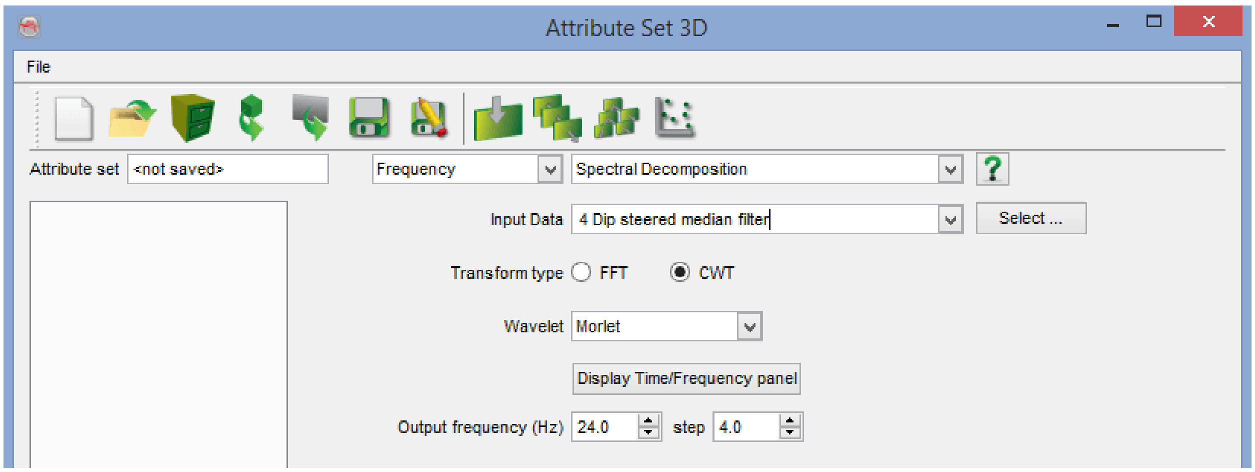
Description
Spectral Decomposition unravels the seismic signal into its constituent frequencies, which allows the user to see phase and amplitude tuned to specific wavelengths. The amplitude component excels at quantifying thickness variability and detecting lateral discontinuities while the phase component detects lateral discontinuities.

It is a useful tool for "below resolution" seismic interpretation, sand thickness estimation, and enhancing channel structures.

Input Parameters
The user can choose between two types of transform:

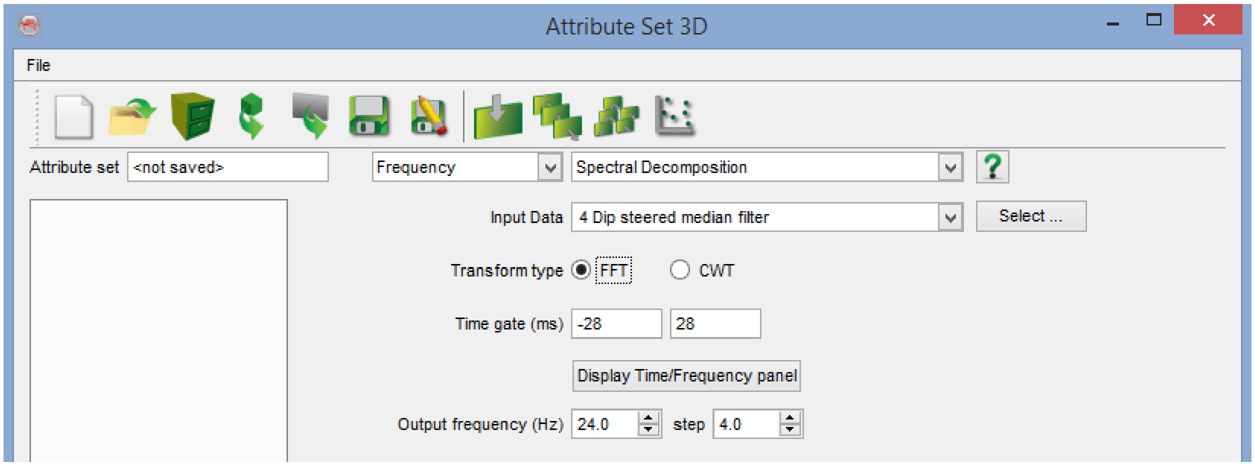
- *FFT* the Fast Fourier Transform. The FFT requires a short window (time-gate) and a step-size between the analyzed frequencies. This step can be interpreted as the frequency resolution.
- *CWT* the Continuous Wavelet Transform. The CWT requires a wavelet type.

When choosing the CWT, you can set the wavelet type:



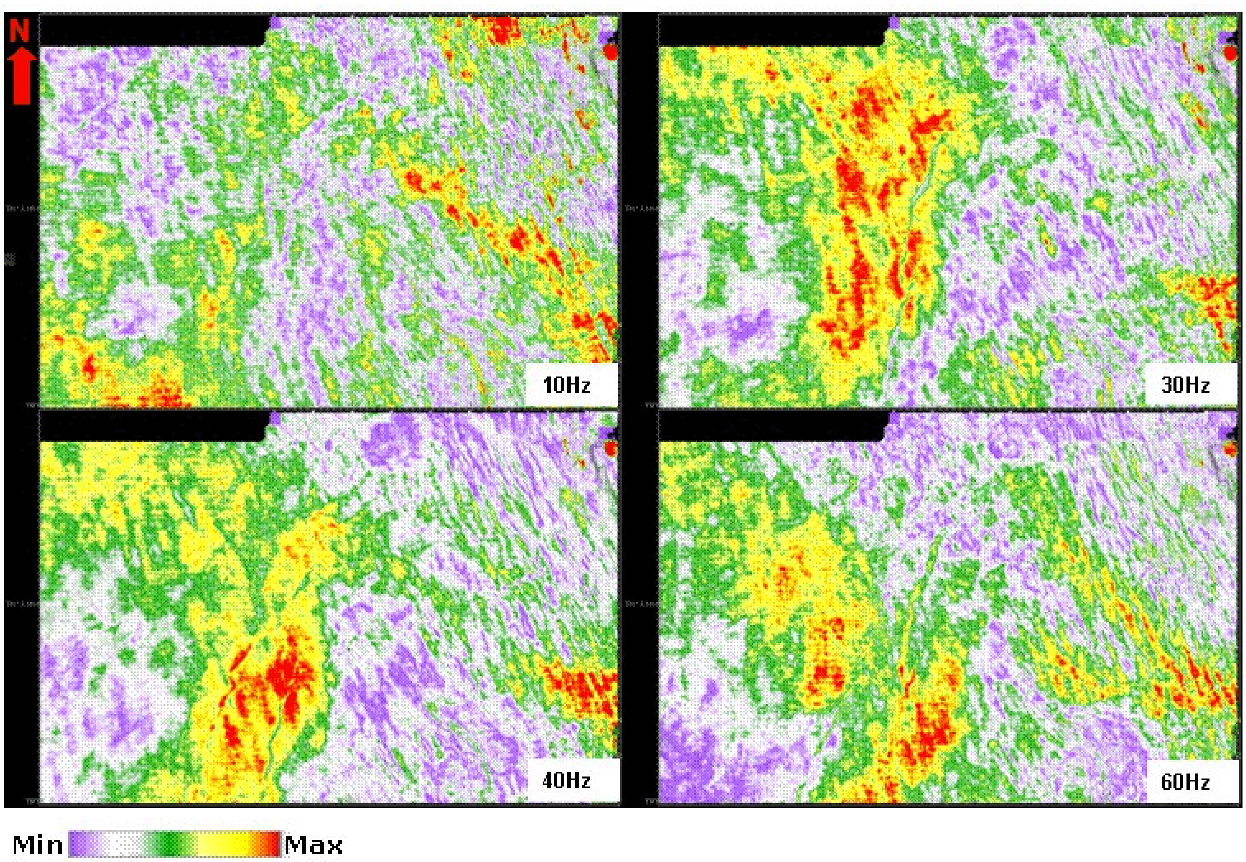
- *Morlet*
- *Gaussian*
- *Mexican Hat*

In FFT only, the signal within the time-window will be transformed into frequency domain. The given step determines the output resolution, if necessary zeros will be added to acquire this resolution. The amplitude spectrum is calculated for the requested frequency. The time-window slides from top to bottom to cover the complete signal. In an ideal situation, the time-window encompasses one seismic event, which may be a superposition of multiple geological events which interfere in the seismic trace.



Output and Examples

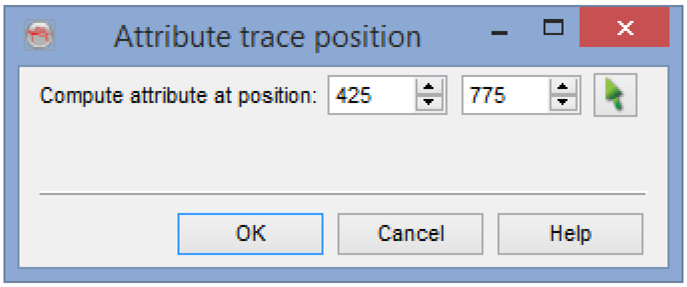
The CWT is defined as the sum over the signal multiplied by a scaled and shifted wavelet. The wavelet is shifted along the signal and at each position the correlation of the wavelet with the signal is calculated. The result is called a wavelet coefficient. The given frequency corresponds to the central wavelet frequency. The step determines the output resolution, which is especially interesting when evaluating this attribute.



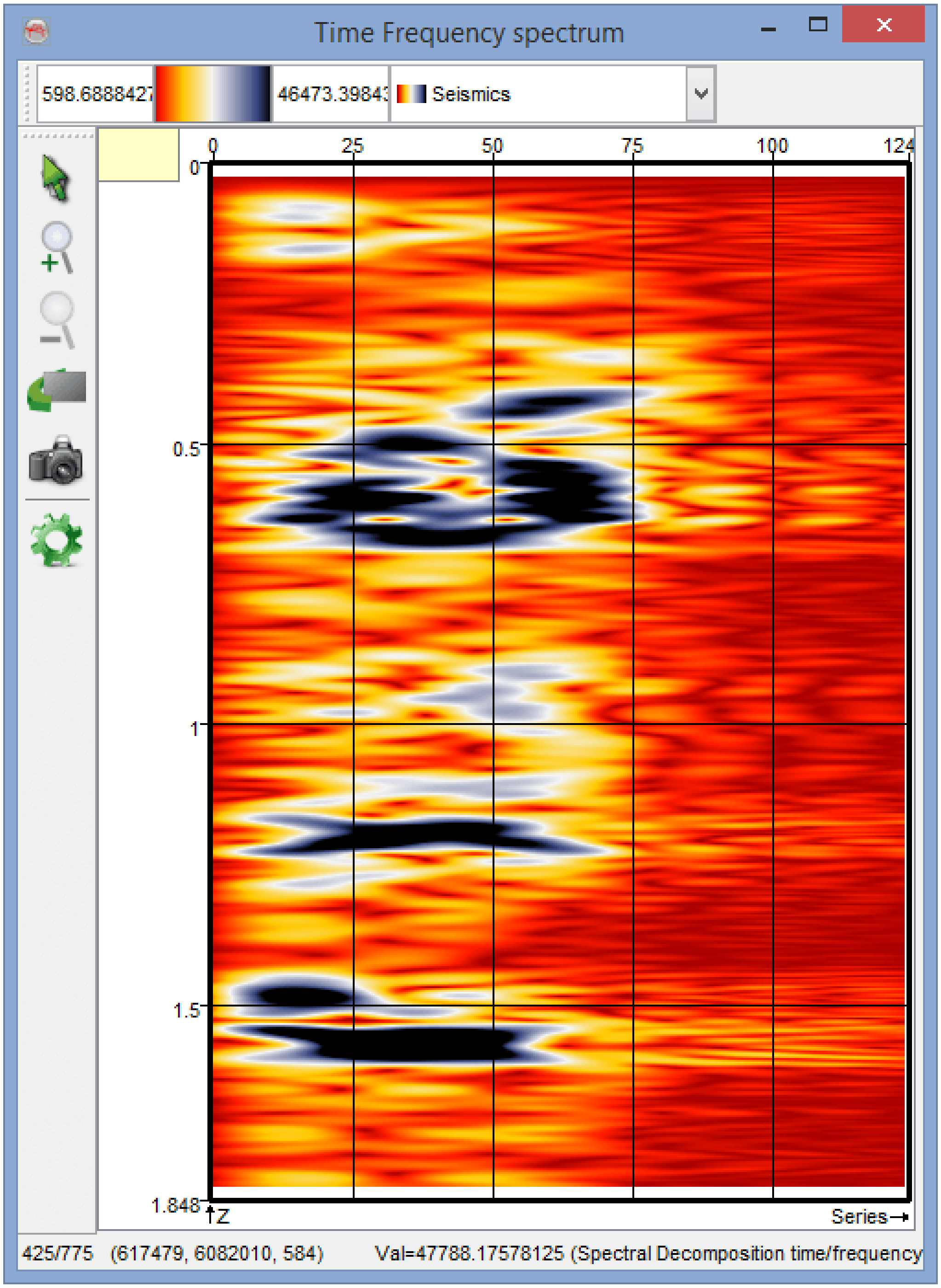
Spectral Decomposition (CWT) applied to a horizon. Notice that each index frequency is describing the specific parts of channels (NNE-SSW oriented). Also thinner and thicker parts of horizon along channels are highlighed clearly.

Time-frequency spectrum

The output frequency is best determined using the time-frequency spectrum panel. This panel displays the spectral decomposition output for all frequencies between 0 and the Nyquist frequency of the data, computed with a step of 1Hz. One must first select a position for this single trace analysis:

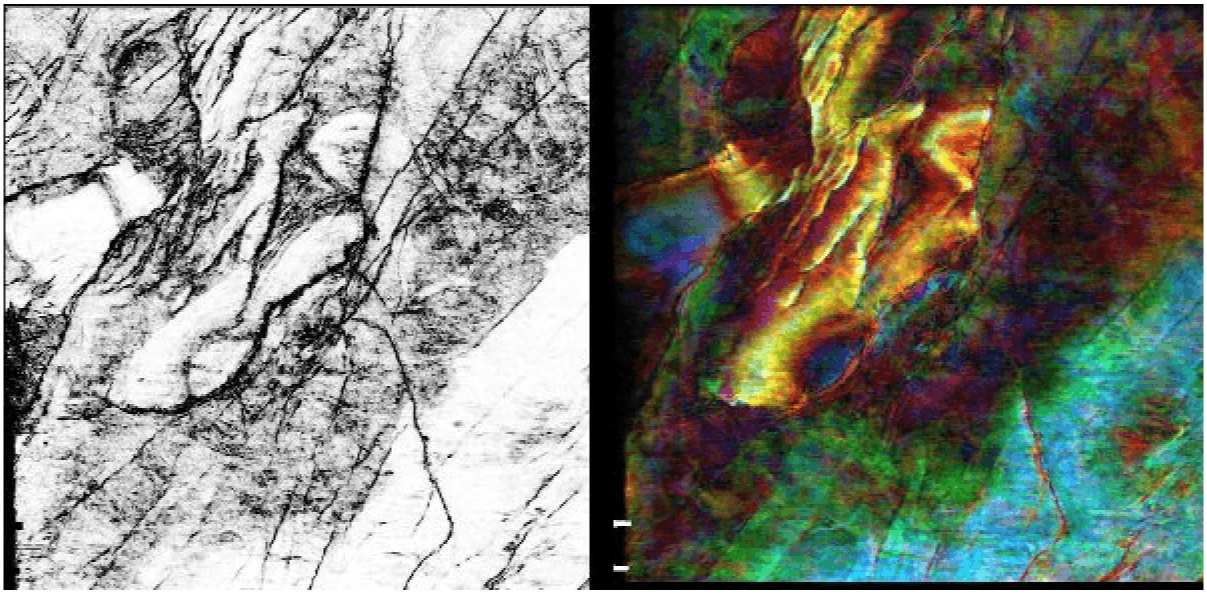


The time-frequency is then displayed in a 2D panel like this:



Color blended display:
RGB(A) blending attribute display* is used to create a normalized color-blended display that often show features with greater clarity and enhances a detail map view. Traditionally, it is used to blend the iso-frequency responses (Spectral Decomposition), but a user can blend three/four different attributes that define a spectrum that is comparable. For instance, spectral decomposition outputs the amplitude at discrete frequencies. So, it renders the same output (unit=amplitude). Depending upon a geological condition or the

objective, FFT short window or CWT (continuous wavelet transform) can be chosen. Results are best displayed on time/horizon slices, volume.
(* RGB(A)- Red, Green, Blue, (Alpha) -channel)



A color blended map view (image on right) of the spectral decomposition (red-10hz, green-20Hz, blue-40hz). Compare the results with the coherency map (image on left). Note that the yellowish colored fault bounder region is thicker as compared to the surrounding regions. The faults throw (red- color) are also clearly observable. Coherency/similarity together with color blended spectral images can reveal better geological information.

Texture

Name

Texture -- Group of attributes that return statistical properties of a Grey-Level Co-occurrence Matrix (GLCM)

Description

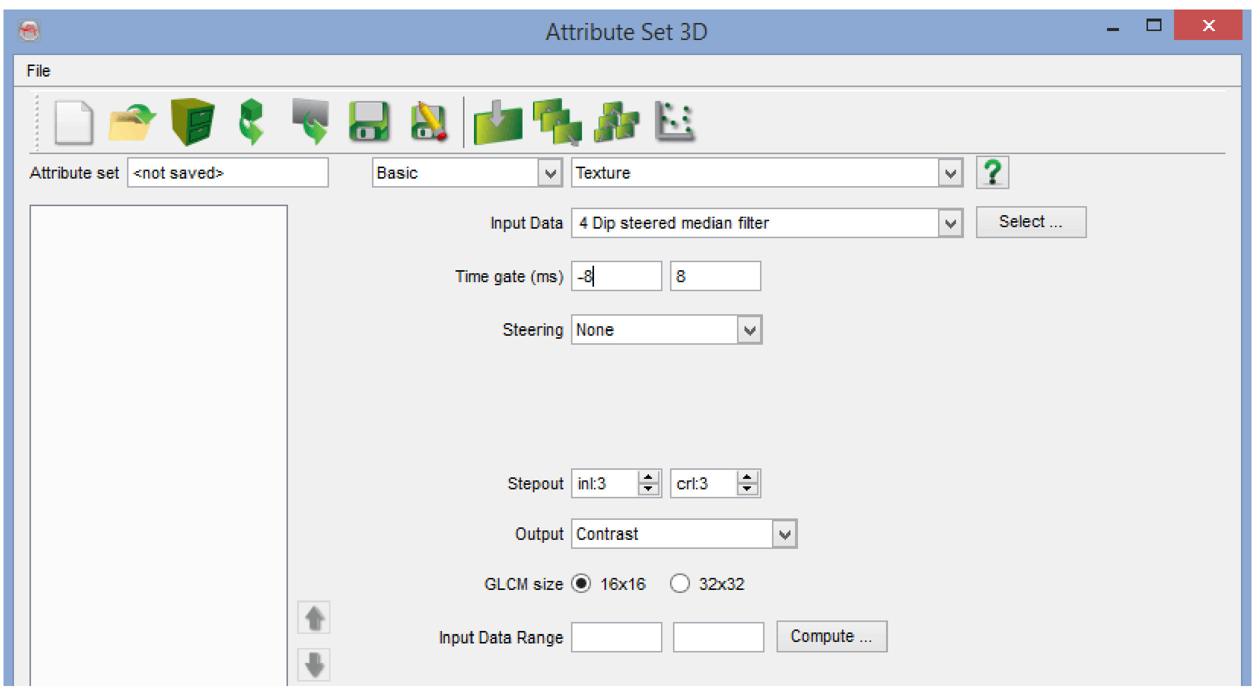
The texture attributes implemented in OpendTect are described in detail in the [GLCM Texture Tutorial] by Mryka Hall-Beyer.

GLCM texture attributes come from image processing and were developed to capture roughness / smoothness of an image. The attribute response is calculated in two steps: First the GLCM is computed for an area (volume) around the evaluation point. Secondly a statistical property from the GLCM is returned. The GLCM is a 2D matrix that captures how often the neighbor values A and B occur in an image. Look at the GLCM as a matrix of N x N dimensions that captures the amplitude response of the reference position in the columns and the amplitudes of the neighboring position in the rows. N is the range of all values the data can have. Let's say we have a data set in which amplitudes can have values 0,1,2,or 3 (GLCM matrix is 4 x 4). We then fill the matrix by comparing each amplitude in the input area (volume) with its direct neighbor and increase the occurrence of the corresponding matrix cell. The matrix is made symmetrical by comparing neighbors in both directions: reference vs neighbor and neighbor vs reference, and it is normalized by dividing through the total number of occurrences. The normalized GLCM matrix is a kind of a probability matrix that tells us how probable it is to find pairs of neighboring amplitudes in the area (volume) around the evaluation point.

In OpendTect the GLCM is computed on re-scaled data. The input data is re-scaled linearly to 4-bits (values ranging from 0 to 15; GLCM 16 x 16), or to 5-bits (values from 0 to 31; GLCM matrix 32 x 32). To re-scale the data the user must give the clipping range of the input data. Neighbors are compared in the inline and cross-line directions. The matrix is further filled by looping over the user-defined time-gate. Note that when dip-steering is used the input extraction area (volume) follows the local stratigraphy, which leads to better responses in dipping strata.

Texture attributes in seismic interpretation are typically used in facies analysis. They can be used in their own right, or (when you have access to the Neural Network plugin) as inputs to a neural network to create 3D seismic facies volumes. This can be done in a supervised approach (MLP network), or in an unsupervised approach (UVQ network).

Input Parameters



Time-gate, **step-out** and **Dip-steering** (optional; requires Dip-steering plugin) settings determine the input area (volume) for which the GLCM matrix is computed. Larger step-outs results in smoother outputs that will follow the stratigraphy better if dip-steering is used.

GLCM-size is the size of the GLCM matrix. 32 x 32 may give somewhat sharper outputs at the expense of more CPU time.

Input Data **Minimum** and **Maximum** define the clipping range of the data, which is needed to rescale the data to 4-bits (16 x 16), or 5-bits (32 x 32).Input data range is automatically calculated from the compute option.In the "analysis" window number of traces need to be selected which computes the scaling range from the selected input traces.

Output
OpendTect supports three groups of Texture Attributes:

1. Contrast group: Measures related to contrast use weights related to the distance from the GLCM diagonal along which neighboring values are equal. Attributes in this group: Contrast, Dissimilarity, Homogeneity

2. Measures related to orderliness. Attributes in this group: Angular Second Moment (ASM), Energy, Entropy

3. Group using descriptive statistics of the GLCM texture measures. Attributes in this group: GLCM Mean, GLCM Variance, GLCM Standard Deviation, GLCM Correlation

In all equations given below N denotes the size of the GLCM matrix; i refers to the column and j to the row. P is the GLCM Probability matrix.

Contrast

$$\text{Contrast} = \sum_{i,j=0}^{N} P_{i,j}(i-j)^2$$

When i and j are equal, the cell is on the diagonal and (i-j)=0. These values represent amplitudes entirely similar to their neighbor, so they are given a weight of 0.
If i and j differ by 1, there is a small contrast, and the weight is 1.
If i and j differ by 2, contrast is increasing and the weight is 4.
The weights continue to increase exponentially as (i-j) increases.

Dissimilarity

$$\text{Dissimilarity} = \sum_{i,j=0}^{N-1} P_{i,j} \, |i-j|$$

In Dissimilarity the weights with which GLCM probablities are multiplied increase linearly away from the diagonal (along which neighboring values are equal).

Homogeneity

$$\text{Homogeneity} = \sum_{i,j=0}^{N-1} \frac{P_{i,j}}{1+(i-j)^2}$$

Dissimilarity and Contrast result in larger numbers for more contrasting windows. If weights decrease away from the diagonal, the result will be larger for input areas (volumes) with little contrast. Homogeneity weights values by the inverse of the Contrast weight, with weights decreasing exponentially away from the diagonal.

Angular Second Moment

$$\text{ASM} = \sum_{i,j=0}^{N-1} P_{i,j}^2$$

ASM and Energy use the GLCM probability as a weight for itself. The name for ASM comes from Physics, and reflects the similar form of Physics equations used to calculate the angular second moment, a measure of rotational acceleration. High values of ASM or Energy occur when the input area (volume) is very orderly.

Energy

$$\text{Energy} = \sqrt{\text{ASM}}$$

See above.

Entropy

$$\text{Entropy} = \sum_{i,j=0}^{N-1} P_{i,j} \left(-ln P_{i,j}\right)$$

Entropy is the opposite of energy; it is a measure of chaos. In physics (thermodynamics) entropy refers to the quantity of energy that is permanently lost to heat ("chaos") every time a reaction or a physical transformation occurs. Entropy cannot be recovered to do useful work. Because of this, the term is used in non technical speech to mean irremediable chaos or disorder. Also, as with ASM, the equation used to calculate physical entropy is very similar to the one used for the texture measure.

GLCM Mean

$$\text{GLCM mean } \mu_i = \sum_{i,j=0}^{N-1} i\left(P_{i,j}\right) = \mu_j = \sum_{i,j=0}^{N-1} j\left(P_{i,j}\right)$$

The left hand equation calculates the mean based on the reference pixels, i. The right-hand equation calculates the mean over the the neighbor pixels, j. These two values are identical because OpendTect computes a symmetrical GLCM, where each amplitude is counted once as a reference and once as a neighbor.

GLCM Variance

$$\text{Variance } \sigma_i^2 = \sum_{i,j=0}^{N-1} P_{i,j}\,(i-\mu_i)^2 = \sigma_j^2 = \sum_{i,j=0}^{N-1} P_{i,j}\left(j-\mu_j\right)^2$$

Variance is a measure of the dispersion of the values around the mean. It is similar to entropy. It answers the question "What is the dispersion of the difference between the reference and the neighbour pixels in this input area (volume)?"
GLCM Variance in texture measures performs the same task as does the common descriptive statistic called variance. It relies on the mean, and the dispersion around the mean, of cell values within the GLCM. However, GLCM variance uses the GLCM, therefore it deals specifically with the dispersion around the mean of combinations of reference and neighbor amplitudes, so it is not the same as variance of input amplitudes that can be computed with the "Volume Statistics" attribute.

Variance calculated using i or j gives the same result, since the GLCM is symmetrical.

GLCM Standard Deviation

$$\text{Standard Deviation } \sigma_i = \sqrt{\sigma_i^2} = \sigma_j = \sqrt{\sigma_j^2}$$

There is no particular advantage to using Standard Deviation over Variance, other than a different range of values.

GLCM Correlation

$$\text{GLCM Correlation} = \sum_{i,j=0}^{N-1} \left[\frac{(i-\mu_i)(j-\mu_j)}{\sqrt{(\sigma_i^2)(\sigma_j^2)}} \right]$$

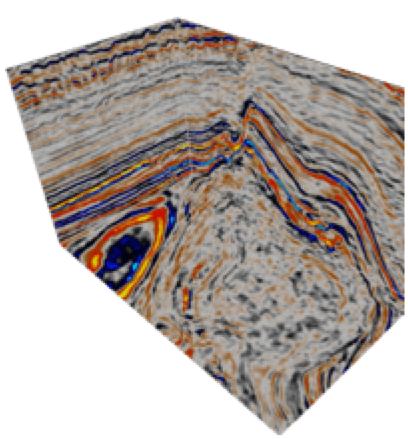
The Correlation texture measures the linear dependency of input amplitudes on those of neighboring amplitudes.

GLCM Correlation is quite a different calculation from the other texture measures described above. As a result, it is independent of them (gives different information) and can often be used profitably in combination with another texture measure. It also has a more intuitive meaning to the actual calculated values: 0 is uncorrelated, 1 is perfectly correlated.

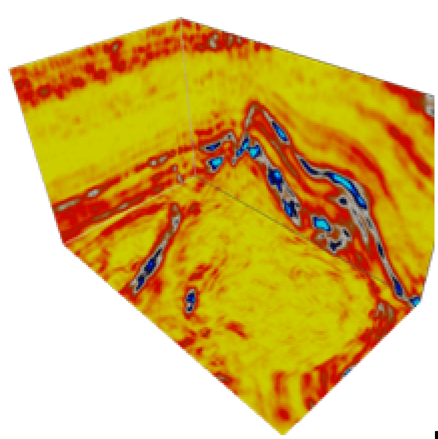
GLCM Correlation can be calculated for successively larger window sizes. The window size at which the GLCM Correlation value declines suddenly may be taken as one definition of the size of definable objects within an image.

If the input is completely uniform the GLCM variance is 0 and the correlation function is undefined. OpendTect will in that case return the value 1.

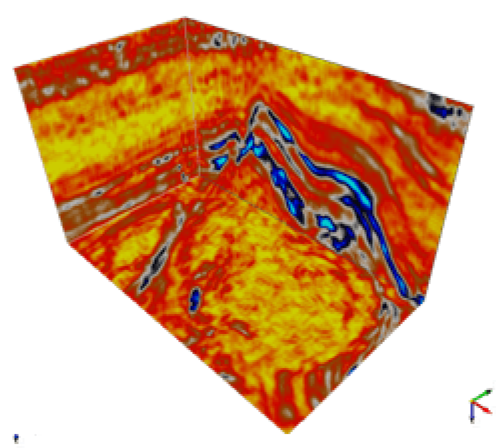
Examples
DS in the examples below stands for Dip-Steering.



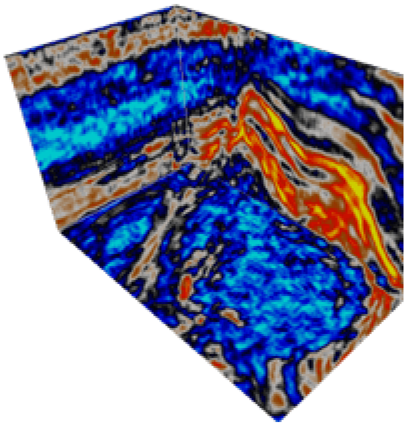
Seismic



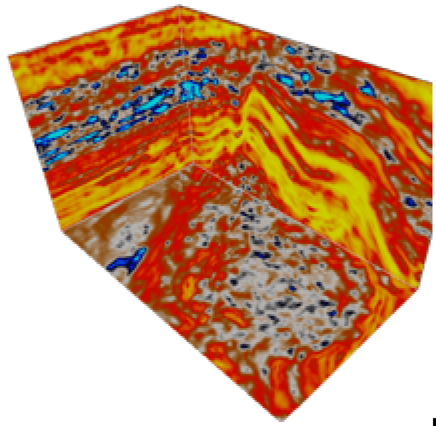
Contrast: 3 x 3 x [-8,8], DS



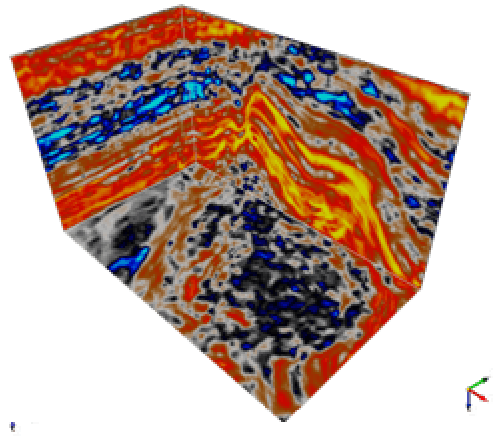
Dissimilarity: 3 x 3 x [-8,8], DS



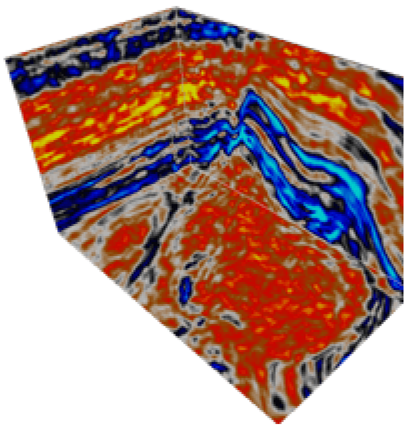
Homogeneity: 3 x 3 x [-8,8], DS



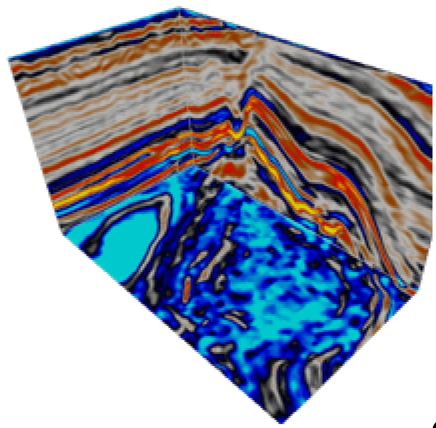
ASM: 3 x 3 x [-8,8], DS



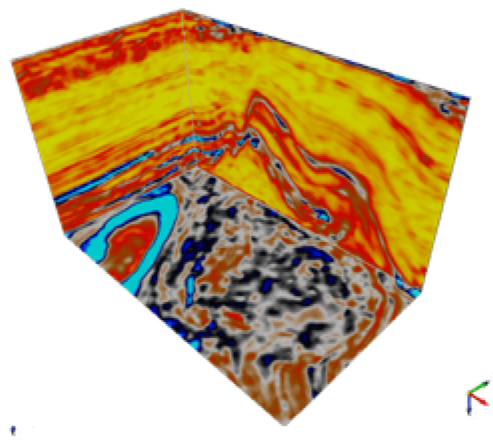
Energy: 3 x 3 x [-8,8], DS



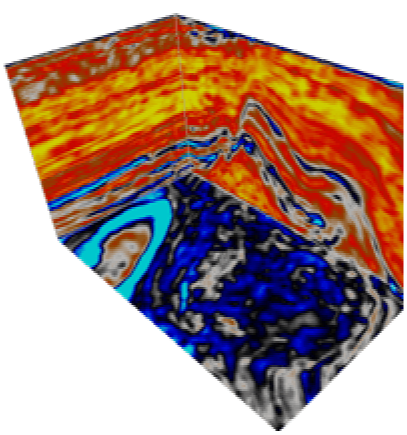
Entropy: 3 x 3 x [-8,8]



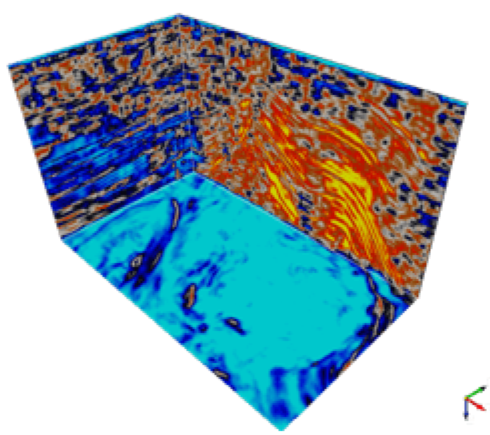
GLCM Mean: 3 x 3 x [-8,8], DS



GLCM Variance: 3 x 3 x [-8,8], DS



GLCM Standard Deviation: 3 x 3 x [-8,8]



GLCM Correlation: 3 x 3 x [-8,8], DS

References

- Chopra, S. and Alexeev, V., 2005. Application of texture attribute analysis to 3D seismic data. CSEG Recorder, Sep. 2005 pp 29-32.
- Hall-Beyer, M. GLCM Texture Tutorial. Available: Online [Accessed 9 Oct. 2012].

Texture - Directional

Name

Texture - Directional -- a multi-trace attribute that returns textural information based on a statistical texture classification.

Description

Texture- Directional uses the grey level co-occurrence matrix (GLCM) and its derived attributes are tools for image classification that were initially described by Haralick et al. (1973). Principally, the GLCM is a measure of how often different combinations of pixel brightness values occur in an image. It is a method widely used in image classification of satellite images (e.g. Franklin et al., 2001; Tsai et al., 2007), sea-ice images (e.g. Soh and Tsatsoulis, 1999; Maillard et al., 2005), magnetic resonance and computed tomography images (e.g. Kovalev et al., 2001; Zizzari et al., 2011), and many others. Most of these GLCM applications are for classification of 2D images solely. The application of GLCM for seismic data has been a minor topic in comparison to common seismic attributes such as coherence, curvature or spectral decomposition. Today, a high percentage of the available seismic data is 3D seismic. Therefore, it is important for the classification of seismic data to adapt the GLCM calculation to work in the three-dimensional space. Few authors have described the application of GLCM for 3D seismic data with various approaches to this topic (Vinther et al., 1996; Gao, 1999, 2003, 2007, 2008a, 2008b, 2009, 2011; West et al., 2002; Chopra and Alexeev, 2005, 2006a, 2006b; Yenugu et al., 2010; de Matos et al., 2011; Eichkitz et al. 2012, 2013, 2014).

The calculation of GLCM-based attributes can be done in separate space directions. For the 2D case 4 space directions exist. For the 3D case the number of possible space directions is extends to 13. The principal workflow of GLCM-based attribute calculation consists of transformation of the amplitude cube into a grey level cube, the counting of pixel co-occurrences within in a given analysis window, and the calculation of attributes based on the co-occurrence matrix. In Figure 1 the principal calculation of 2D GLCM in four space directions is shown for a sample image.

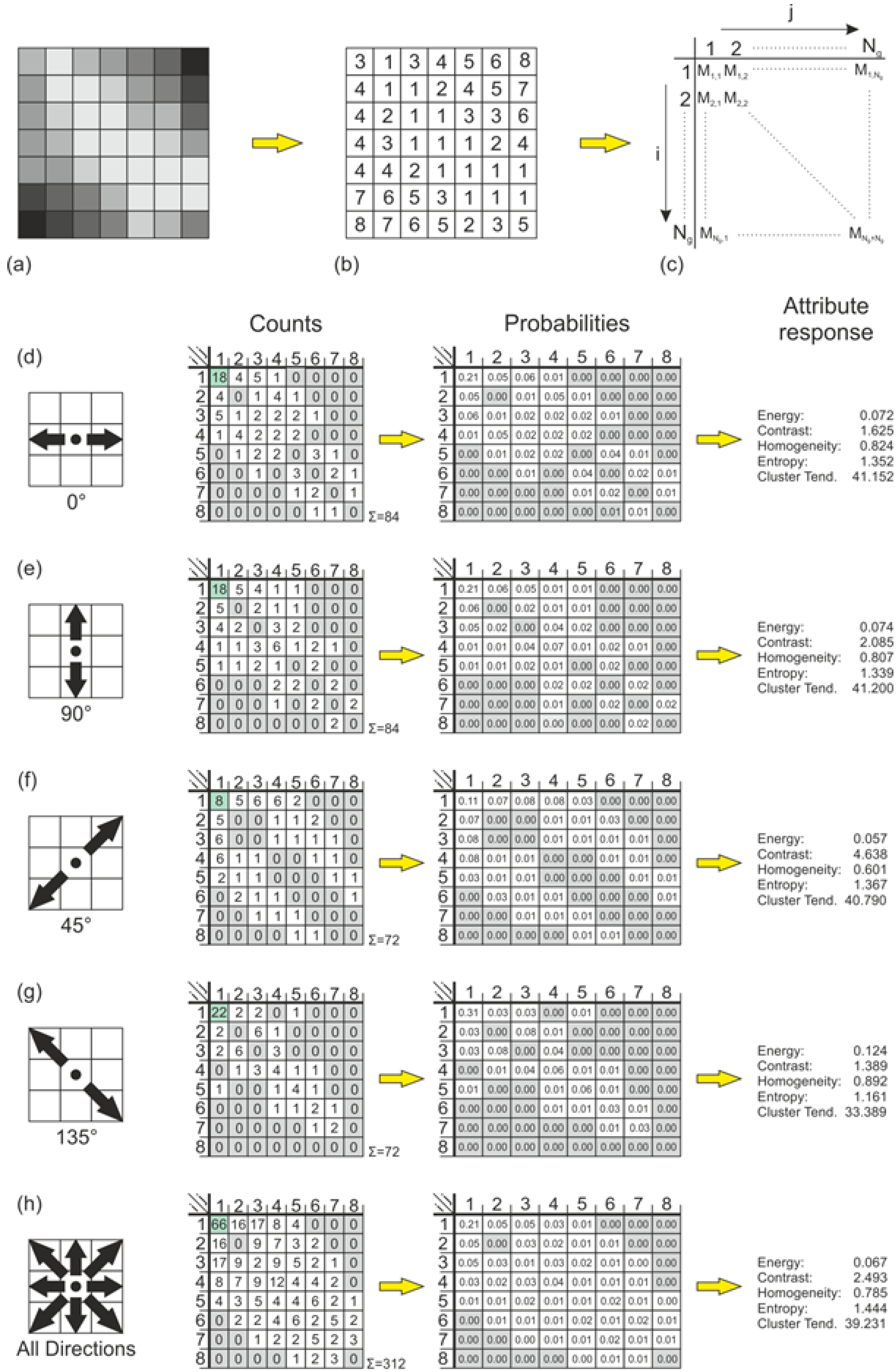

(a)
(b)
(c)
j
i
N_g
Counts
Probabilities
Attribute response
(d)
0°
Σ=84
Energy: 0.072
Contrast: 1.625
Homogeneity: 0.824
Entropy: 1.352
Cluster Tend. 41.152
(e)
90°
Σ=84
Energy: 0.074
Contrast: 2.085
Homogeneity: 0.807
Entropy: 1.339
Cluster Tend. 41.200
(f)
45°
Σ=72
Energy: 0.057
Contrast: 4.638
Homogeneity: 0.601
Entropy: 1.367
Cluster Tend. 40.790
(g)
135°
Σ=72
Energy: 0.124
Contrast: 1.389
Homogeneity: 0.892
Entropy: 1.161
Cluster Tend. 33.389
(h)
All Directions
Σ=312
Energy: 0.067
Contrast: 2.493
Homogeneity: 0.785
Entropy: 1.444
Cluster Tend. 39.231

Figure 1: Example for the calculation of grey level co-occurrence matrix-based attributes using eight grey levels for a randomly generated 2D grey-scale image (a). The grey-scales of the image can be represented by discrete values (b). The number of co-occurrences of pixel pairs for a given search window are counted and a grey level co-occurrence matrix (c) is produced. Based on this co-occurrence matrix, several attributes can be calculated. In this example, the grey level co-occurrence matrices are determined for the horizontal (d), the vertical (e), the 45° diagonal (f), the 135° diagonal (g), and for all directions at once (h). The first step in calculation is the determination of co-occurrences (column 2). Zero entries are marked in light grey and the highest value of each matrix is marked in dark grey. It is evident that calculations in single directions lead to sparse matrices. The GLCM is normalized by the sum of the elements to get a kind of probability matrix (column 3). Finally, the probabilities are used for the calculation of GLCM-based attributes. In column 4 the results for Entropy, Contrast, Homogeneity, Entropy, and Cluster Tendency are shown.

In the case of 3D data the number of possible directions increases to 13. In Figure 2 a simple Rubik's cube is taken to explain the 13 possible directions for a 3D dataset. This Rubik's cube is build-up of 27 small cubes. The small cube in the center (the turning point in a Rubik's cube) is the point of interest for which the calculations are performed. This center point is surrounded by 26 neighboring cubes. If we now take the center point and draw lines form it to all neighboring cubes, we get 13 directions on which the neighboring samples are placed.

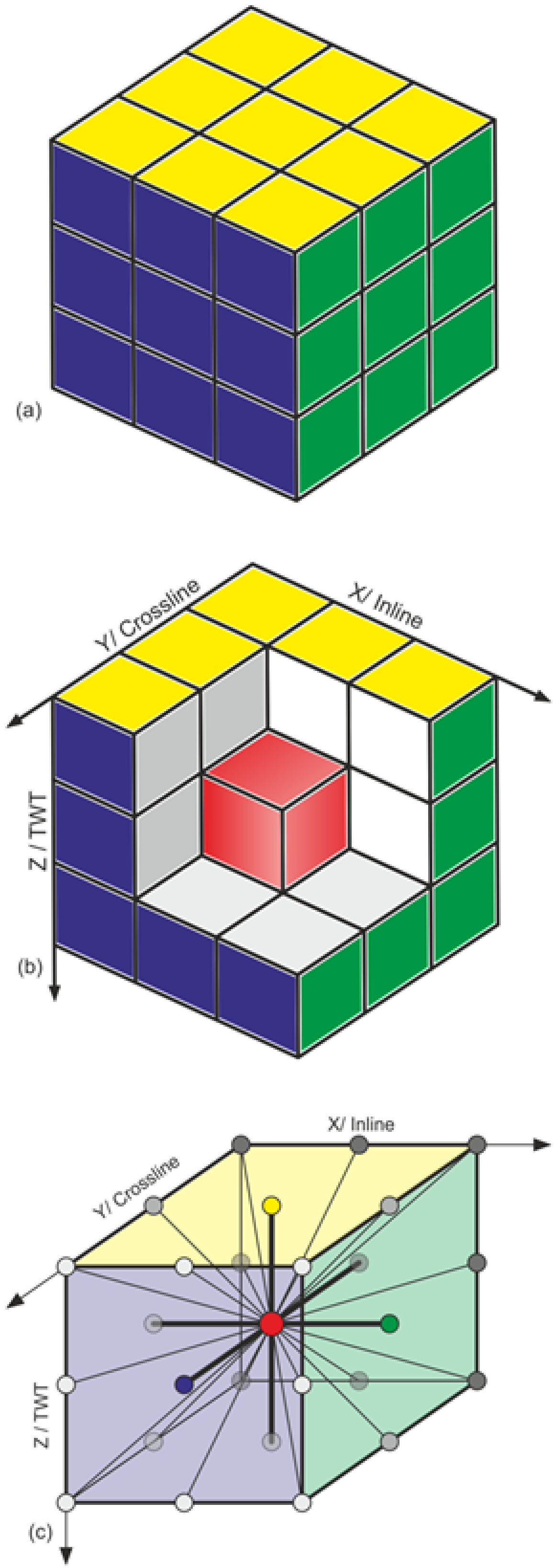

(a)
Y/ Crossline
X/ Inline
Z / TWT
(b)
X/ Inline
Y/ Crossline
Z / TWT
(c)

Figure 2: The number of principal neighbors for one sample point can be best explained by looking at a Rubik's cube (a). The center of the Rubik's cube (core mechanism for rotating the cube, red box in (b)) has in total 26 neighboring boxes (including diagonal neighbors). These boxes are aligned in 13 possible directions. Analogous to this, a sample point within a seismic sub-volume has 26 neighbors aligned in 13 directions (c). In the developed workflow it is possible to calculate the GLCM along single directions, along combinations of directions (e.g. inline direction, crossline direction, …), or all directions can be calculated at once (after Eichkitz et al., 2013).

Input Parameters

Input Data

The input for the GLCM-based attribute calculation can be any seismic amplitude 3D cube/2D section. In the process of GLCM calculation this amplitude cube is converted to a grey level cube.

Compute Amplitude Range

For the transformation of the amplitude cube to a grey level cube the range of the amplitude values is needed. This can either be inserted manually, or be computed. In the case of computed amplitude range, the amplitude range will be symmetrical around zero.

Number of Grey Levels

The number of grey levels used for the transformation of amplitude cube to grey level cube. Higher numbers generally improve the quality of the GLCM output. Common numbers for the grey levels are 16 to 256.

Number of Traces

The number of traces defines the horizontal analysis window. This horizontal analysis window is always symmetrical around the center trace. Number of traces equal 1 means 1 trace left and right of the center trace (thus 3 traces).

Vertical Search Window

The vertical search window defines the number of samples included in the search window. The vertical size of the analysis window should be according to the average wavelength (Gao, 2007). This is typically in the range of 15 samples (+/- 7 samples).

GLCM Attribute

In total 23 GLCM-based attributes can be calculated (see below in Mathematical description)

Direction of Calculation

The GLCM-based attribute calculation can in principal be done in 13 space directions for 3D input data (4 space directions for 2D data). The algorithm allows the calculation in single directions or the combined calculation of several directions (inline, crossline, time-/depthslice) or all 13 space directions can be calculated at once. Multiple directions give smoother results, but subtle features might be missed. Detailed analysis of single directions might give information about fracturing or facies distribution of the subsurface.

In this process azimuth of 0° is equal to inline direction; azimuth of 90° is equal to crossline direction. Dip of 0° is equal to horizontal direction, dip of 90° is equal to vertical direction.

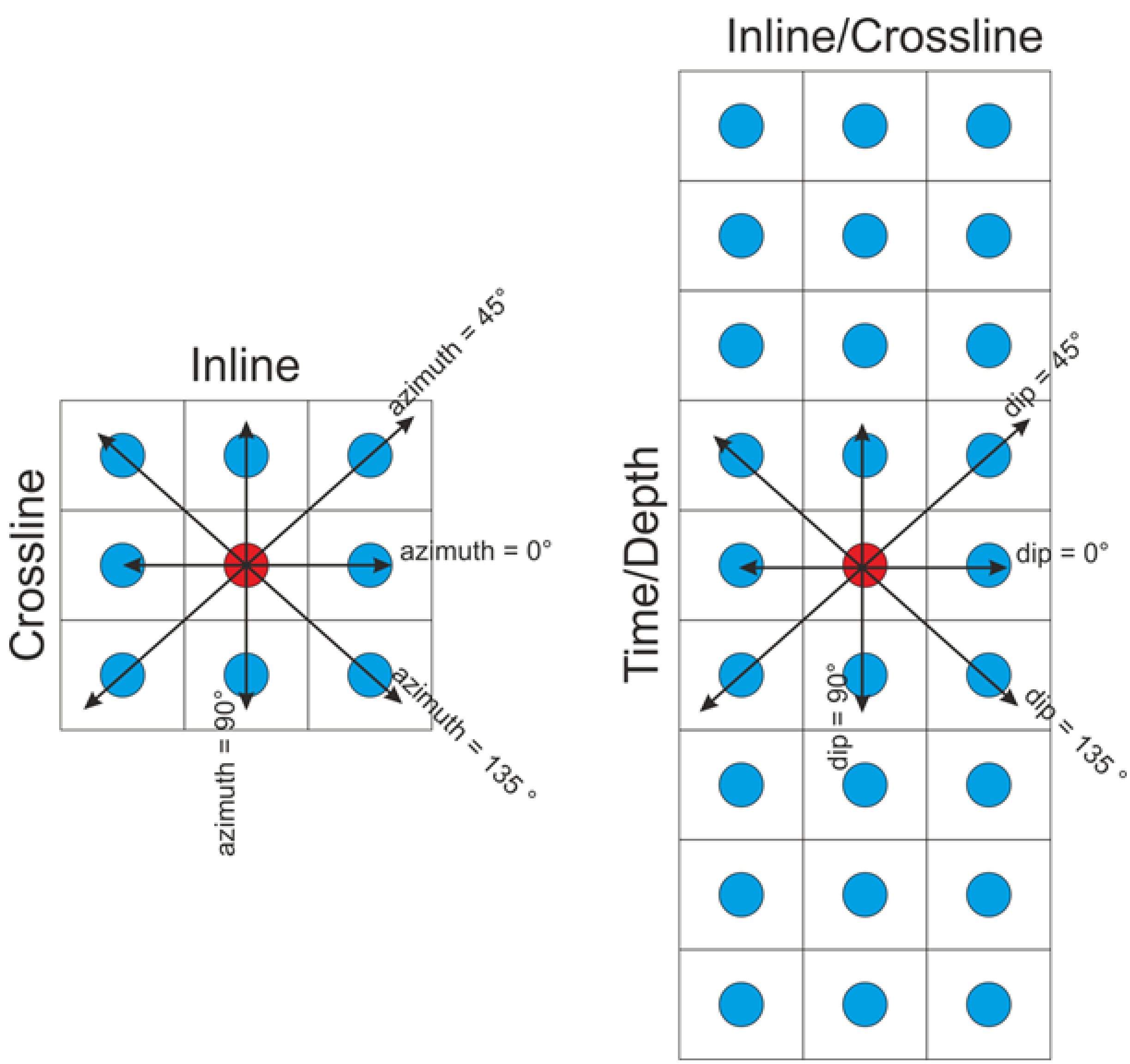


Figure 3: Definition of directions.

Steering

GLCM attribute calculation can be done with or without steering. The integration of dip steering generally improves the signal-to-noise ratio in calculated attributes.

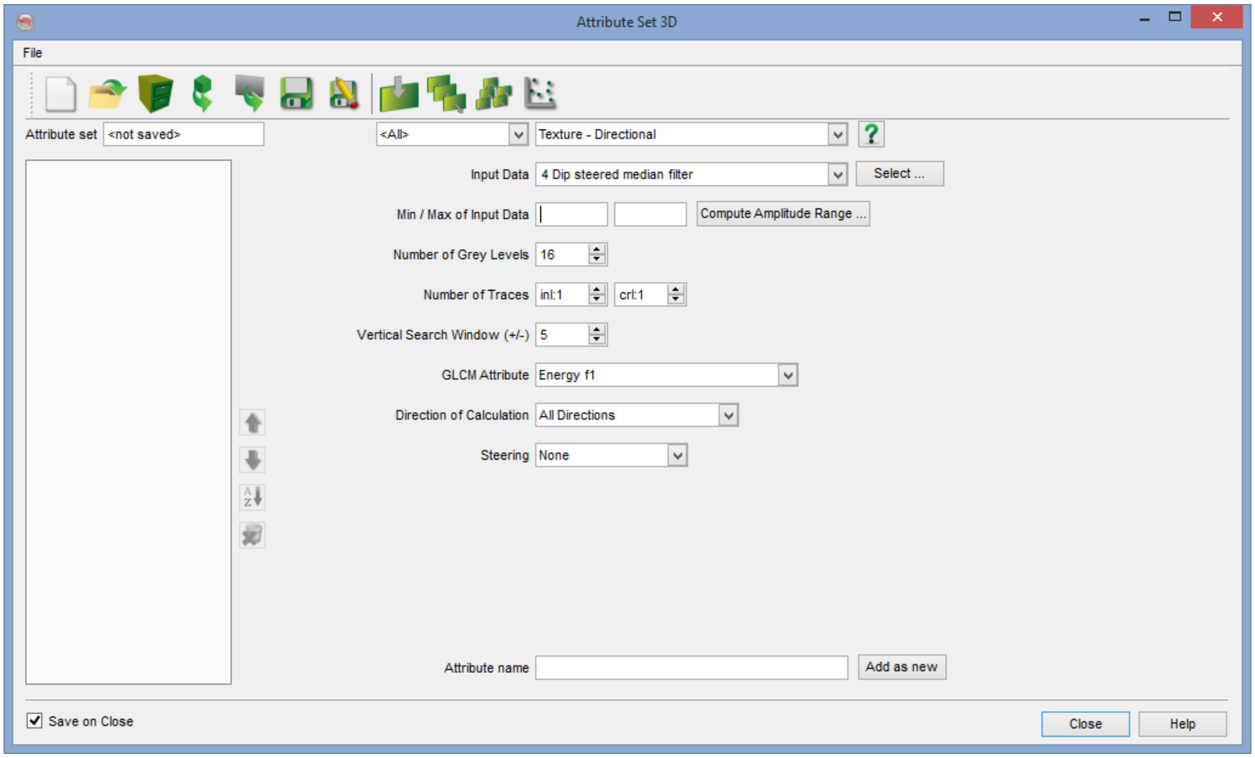


Figure 4: Texture attribute window within OpendTect.

Mathematical description

The GLCM is a measure of how often different combinations of neighboring pixel values occur within an analysis window. For a 2D image the immediate neighboring pixels can be in four different directions (0°, 45°, 90°, and 135°). For the calculation of 2D GLCM the following equation is used:

$$M(i,j) = \sum_{x=1}^{X}\sum_{y=1}^{Y}\begin{cases} 1 & ,G(x,y) = i \bigwedge G(x+dx,y+dy) = j \\ 0 & ,G(x,y) \neq i \bigvee G(x+dx,y+dy) \neq j \end{cases}$$

where i and j vary from 1 to Ng (number of grey levels).

In this equation G(x,y) are the center sample points and G(x+dx, y+dy) are the neighboring sample points. Usually, the distance between center and neighboring samples is one, but in general greater distances could also be taken for the calculation. It is, in principal also possible to combine the four principal directions to form an average GLCM. By this approach the spatial variations can be eliminated to a certain degree (Gao 2007). In the case of 3D data the number of possible directions increases to 13. The 3D case implies a modification of the above given equation:

$$M(i,j) = \sum_{x=1}^{X}\sum_{y=1}^{Y}\sum_{z=1}^{Z}\begin{cases}1 & ,G(x,y,z) = i \wedge G(x+dx, y+dy, z+dz) = j\\0 & ,G(x,y,z) \neq i \vee G(x+dx, y+dy, z+dz) \neq j\end{cases}$$

Similar to the 2D case, it is possible to calculate the GLCM in single directions, to combine several directions, or to calculate an average GLCM. Previous works on 3D GLCM calculation use 2D GLCM calculations in various directions and combine the results of these calculations to form a pseudo-3D GLCM attribute cube.

Based on the grey level co-occurrence matrix, it is possible to calculate several attributes. Haralick et al. (1973), in their work, describe 14 attributes that can be calculated from the GLCM. In literature a few more attributes based on the GLCM have been developed (e.g. Soh and Tsatsoulis, 1999; Wang et al., 2010). For the calculation of any of these GLCM-based attributes it is necessary to normalize the GLCM to generate a kind of probability matrix. This is done by dividing each matrix entry by the sum of all entries. The different GLCM-based attributes can be divided into three general groups. The first group is the contrast group and includes measurements such as contrast and homogeneity. All the attributes from this first group are basically a function of the probability of each matrix entry and the difference of the grey levels (i and j). Therefore, these contrast group attributes are related to the distance from the GLCM diagonal. Values on the diagonal (where i and j are the same) result in zero contrast, whereas the contrast increases by increase of distance from the diagonal.

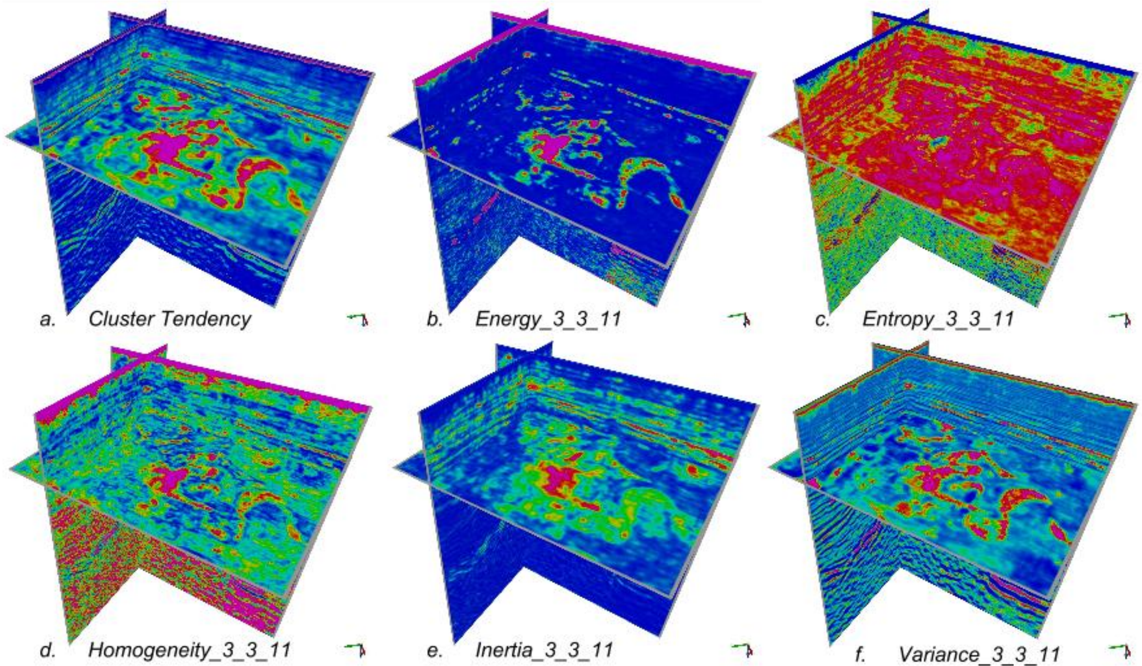
The second attribute group is the orderliness group, which includes attributes such as energy and entropy. Attributes in the orderliness group measure how regular grey level values are distributed within a given search window. In contrast to the first group all attributes from this group are solely a function of the GLCM probability entries.

The third attribute group is the statistics group, which includes attributes such as Haralick et al.'s (1973) measure of mean and variance. These are common mean and variance calculations applied onto the GLCM probabilities.
The following tables summarize all GLCM equations:

f_1	**Energy**	$f_1 = \sum_{i}^{N_g} \sum_{j}^{N_g} {M_{ij}}^2$
f_2	**Contrast**	$f_2 = \sum_{n=0}^{N_g-1} n^2 \left\{ \sum_{i}^{N_g} \sum_{j}^{N_g} M_{ij} \right\}$
f_3	**Correlation**	$f_3 = \sum_{i}^{N_g} \sum_{j}^{N_g} \frac{ijM_{ij} - \mu_x \mu_y}{\sigma_x \sigma_y}$
f_4	**Variance**	$f_4 = \sum_{i}^{N_g} \sum_{j}^{N_g} (i - \mu)^2 M_{ij}$
f_5	**Inverse Difference Moment**	$f_5 = \sum_{i}^{N_g} \sum_{j}^{N_g} \frac{M_{ij}}{1 + (i - j)^2}$
f_6	**Sum Average**	$f_6 = \sum_{n=2}^{2N_g} n M_{x+y,n}$
f_7	**Sum Variance**	$f_{7,1} = \sum_{n=2}^{2N_g} (n - f_8)^2 M_{x+y,n}$
f_8	**Sum Entropy**	$f_8 = -\sum_{n=2}^{2N_g} M_{x+y,n} \log(M_{x+y,n})$
f_9	**Entropy**	$f_9 = -\sum_{i}^{m} \sum_{j}^{n} M_{ij} \log(M_{ij})$
f_{10}	**Difference Variance**	$f_{10} = Var(M_{x-y})$
f_{11}	**Difference Entropy**	$f_{11} = -\sum_{n=0}^{N_g-1} M_{x-y,n} \log(M_{x-y,n})$
f_{12}	**Information Measures of Correlation**	$f_{12} = \frac{f_9 - HXY_1}{max\{HX, HY\}}$
f_{13}	**Information Measures of Correlation**	$f_{13} = \left[1 - e^{-2(HXY_2 - f_9)}\right]^{\frac{1}{2}}$
f_{14}	**Maximum Correlation Coefficient**	$f_{14} = \lambda(Q)$

g_1	**Homogeneity**	$g_1 = \sum_i^{N_g} \sum_j^{N_g} \frac{M_{ij}}{1 + \lvert i-j \rvert}$
g_2	**Sum Mean**	$g_2 = \frac{1}{2} \sum_i^{N_g} \sum_j^{N_g} M_{ij}(i+j)$
g_3	**Maximum Probability**	$g_3 = \max_{i,j} M_{ij}$
g_4	**Cluster Tendency**	$g_4 = \sum_i^{N_g} \sum_j^{N_g} (i + j - 2\mu)^k M_{ij}$
g_5	**Cluster Shade**	$g_5 = \sum_i^{N_g} \sum_j^{N_g} M_{ij}(i + j - \mu_x - \mu_y)^3$
g_6	**Cluster Prominence**	$g_6 = \sum_i^{N_g} \sum_j^{N_g} M_{ij}(i + j - \mu_i - \mu_j)^4$
g_7	**Dissimilarity**	$g_7 = \sum_i^{N_g} \sum_j^{N_g} M_{ij} \lvert i-j \rvert$
g_8	**Difference Mean**	$g_8 = \frac{1}{2} \sum_i^{N_g} \sum_j^{N_g} M_{ij}(i-j)$
g_9	**Autocorrelation**	$g_9 = \sum_i^{N_g} \sum_j^{N_g} ij M_{ij}$
g_{10}	**Inertia**	$g_{10} = \frac{1}{\left(N_g^2 - 1\right)^2} \sum_i^{N_g} \sum_j^{N_g} M_{ij}(i-j)^2$

Examples



References

Chopra, S., and V. Alexeev, 2005, Application of texture attribute analysis to 3D seismic data: 75th SEG meeting, Houston, Texas, USA, Expanded Abstracts, 767-770.

Chopra, S., and V. Alexeev, 2006a, Application of texture attribute analysis to 3D seismic data: The Leading Edge, 25, no. 8, 934-940.

Chopra S., and V. Alexeev, 2006b, Texture attribute application to 3D seismic data: 6th International Conference & Exposition on Petroleum Geophysics, Kolkata, India, Expanded Abstracts, 874-879.

de Matos, M.C., Yenugu, M., Angelo, S.M., and K.J. Marfurt, 2011, Integrated seismic texture segmentation and cluster analysis applied to channel delineation and chert reservoir characterization: Geophysics, 76, no. 5, P11-P21.

Eichkitz, C.G., Amtmann, J. and M.G. Schreilechner, 2013, Calculation of grey level co-occurrence matrix-based seismic attributes in three dimensions: Computers and Geosciences, 60, 176-183.

Eichkitz, C.G., de Groot, P. and Brouwer, F., 2014. Visualizing anisotropy in seismic facies using stratigraphically constrained, multi-directional texture attribute analysis. AAPG Hedberg Research Conference “Interpretation Visualization in the Petroleum Industry”, Houston, USA.

Franklin, S.E., Maudie, A.J., and M.B. Lavigne, 2001, Using spatial co-occurrence texture to increase forest structure and species composition classification accuracy: Photogrammetric Engineering & Remote Sensing, 67, no. 7, 849-855.

Gao, D., 1999, 3-D VCM seismic textures: A new technology to quantify seismic interpretation: 69th SEG meeting, Houston, Texas, USA, Expanded Abstracts, 1037-1039.

Gao, D., 2003, Volume texture extraction for 3D seismic visualization and interpretation: Geophysics, 68, no. 4, 1294-1302.

Gao, D., 2007, Application of three-dimensional seismic texture analysis with special reference to deep-marine facies discrimination and interpretation: Offshore Angola, West Africa: AAPG Bulletin, 91, no. 12, 1665-1683.

Gao, D., 2008a, Adaptive seismic texture model regression for subsurface characterization: Oil & Gas Review, 6, no. 11, 83-86.

Gao, D., 2008b, Application of seismic texture model regression to seismic facies characterization and interpretation: The Leading Edge, 27, no. 3, 394-397.

Gao, D., 2009, 3D seismic volume visualization and interpretation: An integrated workflow with case studies: Geophysics, 74, no. 1, W1-W24.

Gao, D., 2011, Latest developments in seismic texture analysis for subsurface structure, facies, and reservoir characterization: A review: Geophysics, 76, no. 2, W1-W13.

Haralick, R.M., Shanmugam, and K. Dinstein, I., 1973, Textural features for image classification: IEEE Transactions on systems, man, and cybernetics, 3, no. 6, 610-621.

Kovalev, V.A., Kruggel, F., Gertz, H.-J., and D.Y. von Cramon, 2001, Three-dimensional texture analysis of MRI brain datasets: IEEE Transactions on medical imaging, 20, no. 5, 424-433.

Maillard, P., Clausi, D.A., and H. Deng, 2005, Operational map-guided classification of SAR sea ice imagery: IEEE Transaction on Geoscience and Remote Sensing, 43, no. 12, 2940-2951.

Soh, L.-K., and C. Tsatsoulis, 1999, Texture analysis of SAR sea ice imagery using gray level co-occurrence matrices: IEEE Transactions on Geoscience and Remote Sensing, 37, no. 2, 780-795.

Tsai, F., Chang, C.-T., Rau, J.-Y., Lin, T.-H., and G.-R. Liu,, 2007, 3D computation of gray level co-occurrence in hyperspectral image cubes: LNCS 4679, 429-440.

Vinther, R., Mosegaar, K., Kierkegaard, K., Abatzi, I., Andersen, C., Vejbaek, O.V., If, F., and P.H. Nielsen, 1996, Seismic texture classification: A computer-aided approach to stratigraphic analysis: 65th SEG meeting, Houston, Texas, USA, 153-155.

Wang, H., Guo, X.-H., Jia, Z.-W., Li, H.-K., Liang, Z.-G., Li, K.-C., and Q. He, 2010, Multi-level binomial logistic prediction model for malignant pulmonary nodules based on texture features of CT image: European Journal of Radiology, 74, 124-129.

West, B.P., May, S.R., Eastwood, J.E., and C. Rossen, 2002, Interactive seismic facies classification using textural attributes and neural networks: The Leading Edge, 21, no. 10, 1042-1049.

Yenugu, M., Marfurt, K.J., and S. Matson, 2010, Seismic texture analysis for reservoir prediction and characterization: The Leading Edge, 29, no. 9, 1116-11.

Zizzari, A., Seiffert, U., Michaelis, B., Gademann, G., and S. Swiderski, 2001, Detection of tumor in digital images of the brain: Proc. of the IASTED international conference Signal Processing, Pattern Recognition & Applications, Rhodes, Greece, 132-137.

Velocity Fan Filter

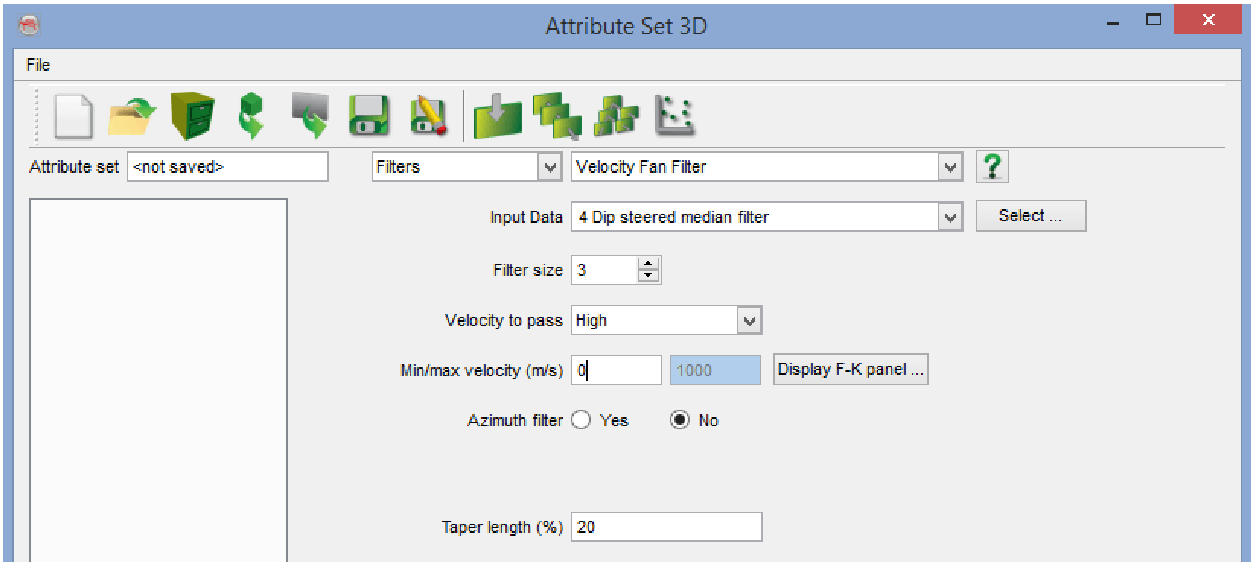
Name
Velocity Fan Filter -- Attribute that returns energy with apparent velocities/dips inside a specified Min/Max range.

Description
The velocity fan filter passes energy with apparent velocities (for Time surveys) or apparent dips (for Depth surveys) inside the specified Min/Max velocity/dip range. The filter supports three options:

- pass low velocities/dips (i.e. suppress high velocities/dips)
- pass high velocities/dips (i.e. suppress low velocities/dips)
- pass velocities/dips within a specified cone.

Therefore, this attribute can be used to filter out or enhance certain dip/azimuth events.

Input Parameters
The Filter size is the size of the 3D kernel. Filter size 3 means the data is convoluted with a 3x3x3 kernel. To reduce edge effects it is recommended to apply a cosine square taper. A Taper length of N means (100-2N)% of the specified velocity range will be flat. Azimuth filter is a special option that allows the dipping energy to be passed inside the specified Azimuth to pass direction only.



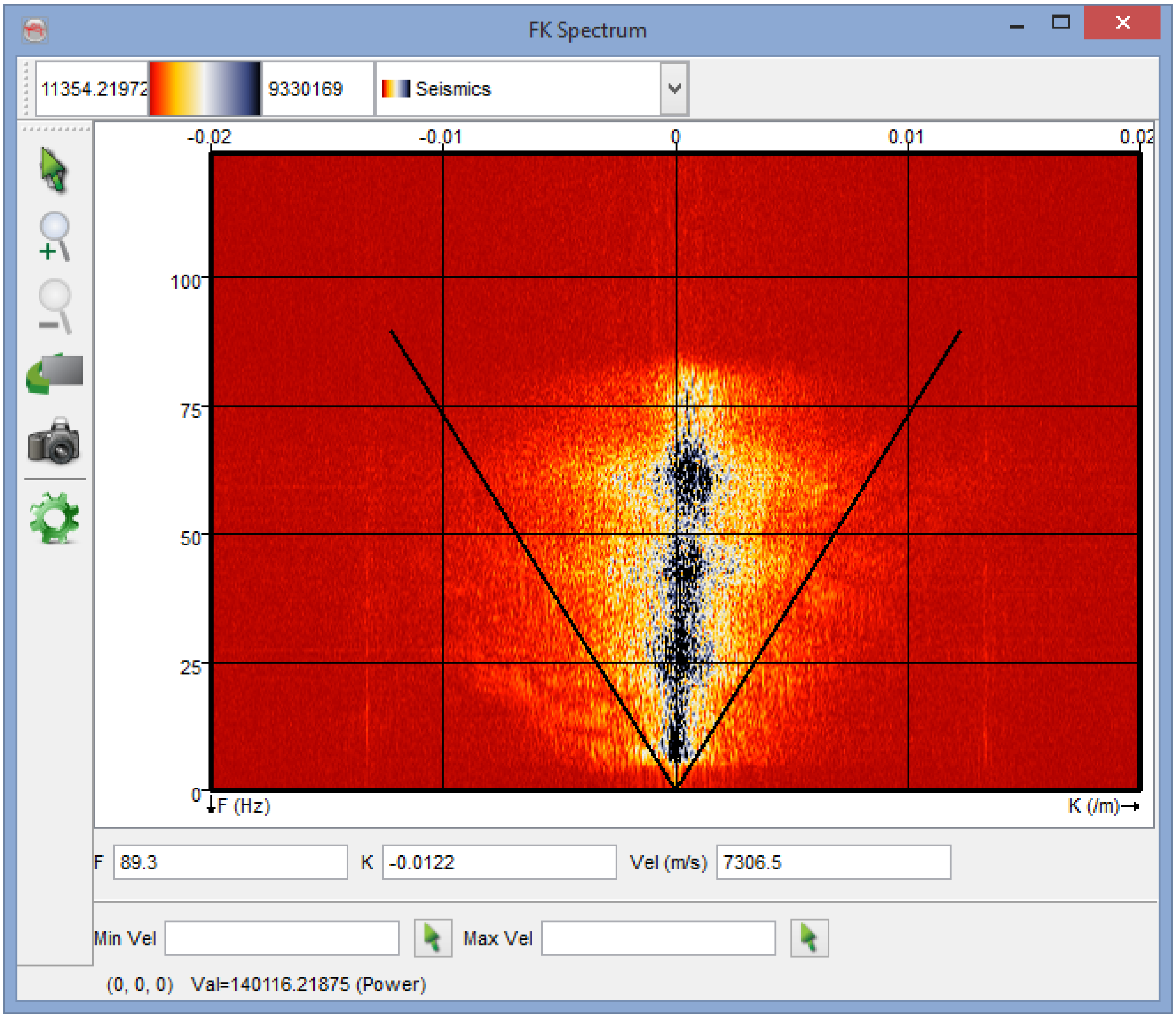
The different shapes of the filter (low pass, high pass, interval velocity/dip) are shown below:

```
type = HighPass
          *     * minvel > 0
          **   **
          *** ***
          *******
          *** ***
          **   **
          *     *

type = LowPass
          ******* maxvel > 0
           *****
            ***
             *
            ***
           *****
          *******

type = BandPass
             *       *  minvel
            ***     ***
            ****   **** maxvel > 0
              *** ***
                 *
              *** ***
            ****   ****
            ***     ***
             *       *
```

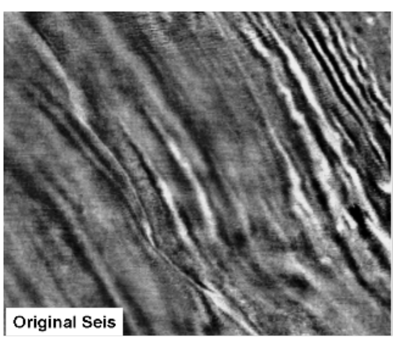
Note: Please be aware that in Time surveys, flat events have infinite velocity, and vertical events have a zero velocity. The opposite is observed in Depth surveys where not velocities but dips are used: Horizontal events have zero dip, while vertical events have 90 degrees dip.

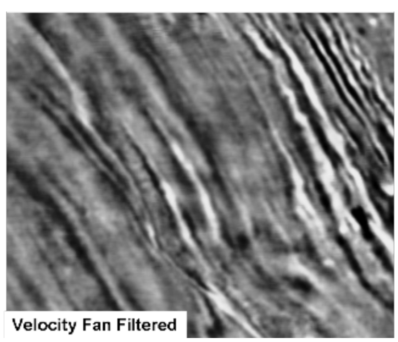
Display F-K Panel:

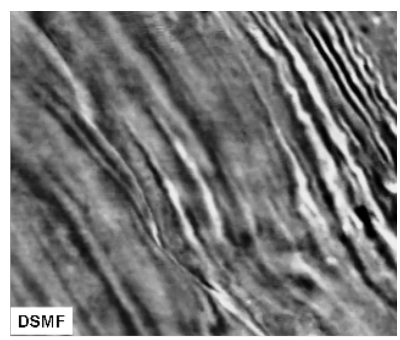
This option allows you to display a two-dimensional Fourier transform over time and space where F is the frequency (Fourier transform over time) and K refers to wave-number (Fourier transform over space).



Examples







An example of a velocity fan filter (high-pass) applied on a time slice. By applying the appropriate filtering parameters, the random noise has been suppressed thus enhancing the amplitudes visibility. Also notice the comparison of this filter with the DSMF (Dip-

Steered Median Filter) that is almost same with an assumption of high pass of velocities in the middle image.

Volume Statistics

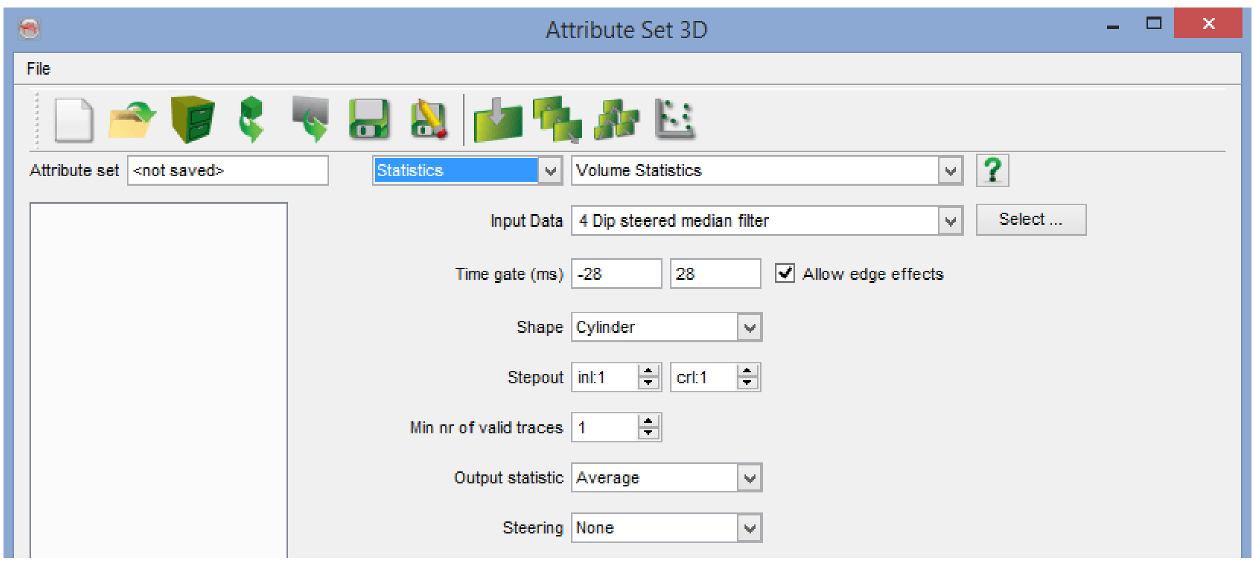
Name

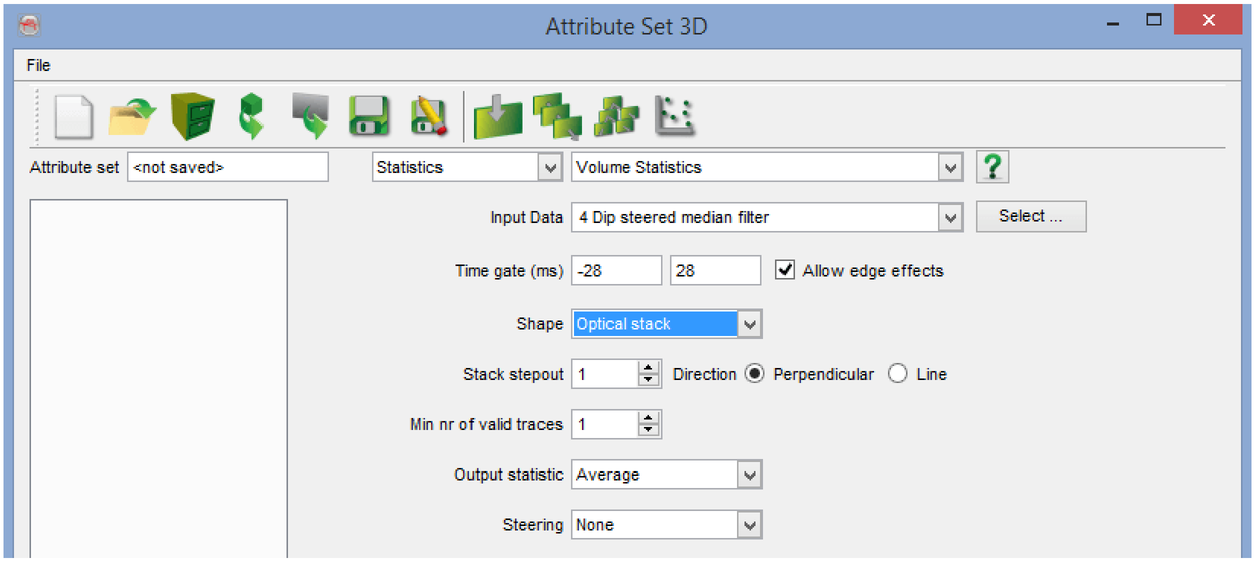
Volume statistics -- Attribute that returns statistical properties

Description

This attribute extracts data from a cube or line using a small 3D probe and returns a statistical property from the samples collected.

Input Parameters





The probe is defined using relative times/depths and trace stepouts with respect to the actual location. The probe can be:

- Cube: The three dimensions listed above define the 3D cube.
- Cylinder: The cylinder option requires the same settings as the cube, but traces outside of the ellipsoid will not be used.
- Optical stack: The probe is not 3D but a 2D vertical plane in the direction of the data, or perpendicular to it. This settings is useful only when applied on random lines where the azimuth is different from the inline and crossline direction.

Please note that the probe direction can be reduced by using:

- A [0,0] time gate: The probe becomes a flat rectangle or disk. Only one sample per trace is used at the actual time/depth.
- A 0 stepout: Only one sample is used in one or both directions.

Output
The following statistical properties can be output:

- Average
- Median
- Variance = Square root of the standard deviation
- Min, Max, Sum
- Norm Variance

- Most Frequency
- RMS: Root Mean Square

Please note that undefined values collected by the probe will not be used for the computation. In the case of a stack only the valid values will define the number of values.

Appendix B. Command Driver Manual

Table of contents

Introduction

Around 2006, it became clear that OpendTect was growing so fast that extensive testing of every part was becoming a real big task. This is a well known issue in the Agile development literature. The usual approach is to implement automated testing.

When designing the implementation, the main issues we had to cope with were:

- A dynamic user interface that is defined by relations between fields. Therefore, the exact layout of windows can change considerably by small changes
- Small changes or fixes in algorithms can produce slightly different outputs for the same input
- A test system should be programmable, not just be a replay of previously run tasks

Therefore, we couldn't use approaches based on replay of clicks, or comparison of outputs with known previous runs. Instead, we create a series of screen snapshots that can quickly be inspected by the test manager.

This resulted in a Command Driver based on the following principles:

- Fields are addressed by names, wildcards supported
- Operations are defined to select, fill, push, manipulate, ...
- Additional flow control operations can be used
- Scripts can call other scripts
- Variables can be set or used from the environment

Examples:

```
ListClick "Calculated attri*" "F03-2" Double
```

```
Include "$SCRIPTSDIR$/EvalutionEnergyAttrib.odcmd"
```

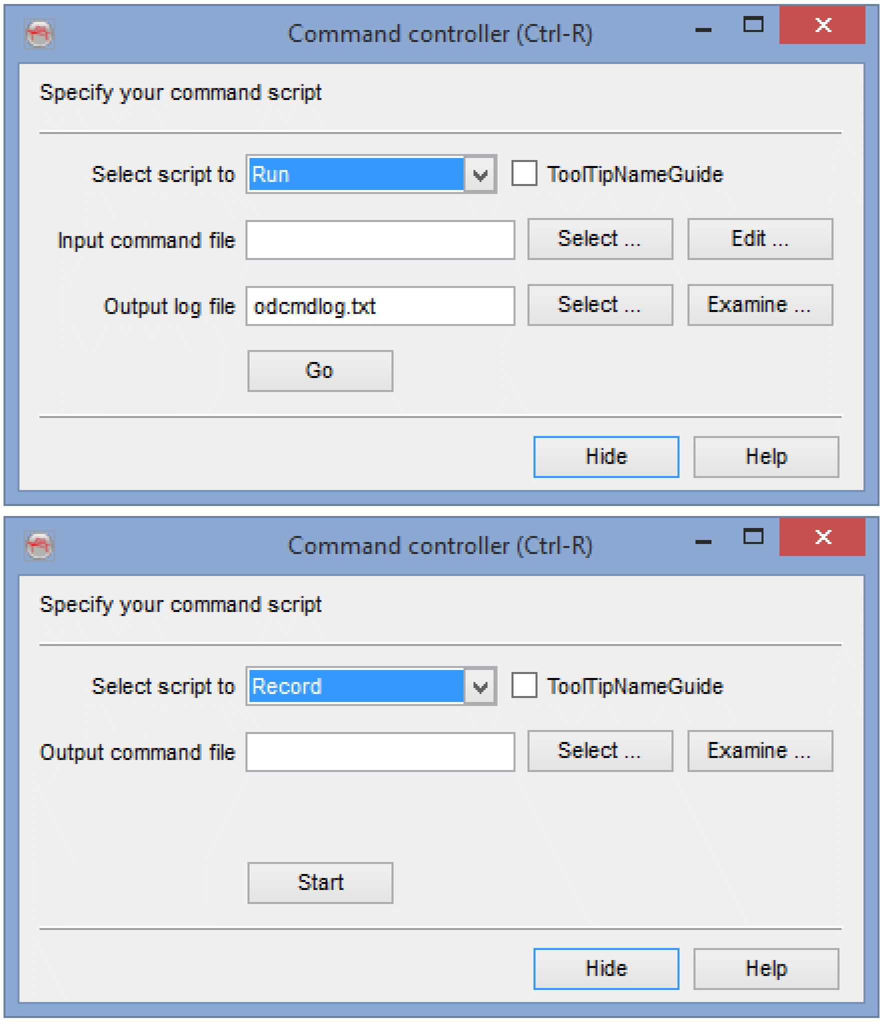
Almost immediately we noticed that finding out the names of the fields was very hard without some tool. Therefore, we made the 'Tooltip name guide', which will put OpendTect in a state where the name of a field is shown as a tool tip (overriding any existing tool tip). When it became clear that our automated test facility would be used as a scripting system for power users, we implemented a tool to record actions.

A webinar held in March 2010 gave an extensive overview on the use of OpendTect's Command Driver plug-in. A forty-minutes flash video is available here: Command Driver webinar

The Command Driver control window
Provided that the CmdDriver plug-in has been (auto)-loaded, one can access the Command Driver control window in two ways. Firstly, it can be launched from the menu bar of OpendTect Main Window:

Utilities->Tools->Command Driver ...

Secondly, this so-called Command Controller also pops up when pressing Ctrl-R with the mouse pointer inside any OpendTect window or dialog. This is particularly useful if the Command Driver menu bar is greyed out or menu access is blocked after a modal dialog has been popped up. Only a few Qt-borrowed QDialogs like the QMessageBox, QFileDialog and QColorDialog will nevertheless prevent any user input to the Command Controller, and have to be clicked away first.



"Command Controller (Ctrl-R)"-window in run-mode and record-mode respectively.

The Command Controller has a combobox to switch between two appearances. One for running and one for recording a command file. The TooltipNameGuide checkbox can be (un)checked at both appearances. It sets a tooltip mode that displays the hidden name of any uiObject pointed to. Tooltip names are displayed double-quoted on a cyan background. Go to the Survey->Select/Setup window for a small demo-example. Click the '(X,Y) <=> I/C' -button and unveil the hidden names of buttons and input fields with your mouse pointer. OpendTect (plugin) developers might have a look in the file $WORK/src/uiIo/uiconvpos.cc to see how these names are annotated in the code.

Running the Command Driver requires selecting an input command file with the extension '.odcmd', or the obsolete '.cmd'. Filenames without a full path are defined relative to the 'Proc'-directory of the current survey. The 'Edit'-button launches a simple text editor to view and edit the input command file directly from OpendTect. One can also specify an output log file to have the Command Driver write its progress and error messages. The

'Examine'-button launches a scroll window to view this log file even while running. The Command Driver starts running when pushing the 'Go'-button, after which 'Pause' and 'Abort'-buttons allow a temporary stop or premature termination.

The Command Recorder is able to record a sequence of user interface actions performed by the user. One has to specify an output command file with the extension '.odcmd'. Filenames without a full path are defined relative to the 'Proc'-directory of the current survey. The 'Examine'-button launches a scroll window to view which commands are written during recording. Recording starts after pushing the 'Start'-button. Next perform a sequence of user actions and push the 'Finish'-button. The Command Controller will automatically switch to run-mode afterwards, so that one can play back the recorded script in order to verify its correct operation. View the log file for eventual errors. Finally, the recorded script can be edited to avoid these errors, improve robustness and general applicability, and insert auxiliary commands.

Executing scripts from the command line

One does not necessarily have to run a command script from OpendTect's user interface. One or more scripts can also run straight from the command line, and with that they might be called from shell scripts as well. An example of the command line syntax on Linux:

```
start_dtect cmd=/full_path_1/my_script_1.odcmd ... cmd=/full_
path_N/my_script_N.odcmd
```

This line opens a new instance of OpendTect and starts running the given scripts in there. OpendTect will not automatically close when all scripts are finished. If necessary, this must be done from the last script using the 'Close All' command. Note that command line execution is only possible if your OpendTect auto-loads the Command Driver plug-in at startup.

The user can also create command scripts named 'autoexec.odcmd'. If available, these scripts will be run at start-up before any script specified on the command line. The first autoexec-script to be run is searched in the settings directory. This is the '.od'-subdirectory of your personal home directory, unless the DTECT_SETTINGS or DTECT_WINSETTINGS environment variable specifies otherwise. The second autoexec-script to be run is searched in the 'Proc'-directory of the start-up survey. Next, the command line scripts will be run in order. Moreover, every time a new survey is selected in OpendTect, the 'Proc'-directory of this new survey will also be searched for an autoexec-script to be run at that moment.

Apart from the availability of autoexec-scripts, their execution can always be disabled by adding 'cmd=noautoexec', 'cmd=nosettingsautoexec' or 'cmd=nosurveyautoexec' as an extra argument on the command line. Command line argument 'cmdlog=/full_path/my_

odcmdlog.txt' will override the location of the output log file. Its default name is 'odcm-dlog.txt' and it is located in the 'Proc'-directory of the start-up survey, if writable, or in the personal user directory otherwise.
To run a command file other than 'autoexec.odcmd' on Windows, one has to create a DOS batch file. That file must contain a command line similar to this:

```
X:\full_ path_ 0\bin\win64\od_ main   cmd="X:\full_ path_ 1\my_
script_1.odcmd" ...
```

Go to your OpendTect installation folder to ascertain the right name and location of the OpendTect executable (i.e. which disk, bin-path, win32 / win64, and '.exe'-file name?). Executing your DOS batch file will open a new instance of OpendTect and run the given script at startup.

Window management
Commands always apply to the current window. This is OpendTect Main Window when starting the driver. Any modal window popping up will automatically become the current window. Modeless windows that pop up can be appointed current window by using the Window-command. In case a modal current window closes, its parent will become the current window. If a modeless current window closes, the latest former current window still existing will be restored as current window.

Subwindows in the workspace of another window cannot become current window. No commands can be addressed to the progress window popped up by a uiTaskRunner, since the driver will await its decease before executing a next command. Neither addressible are the Command Controller window and its descendants.

QDialog windows popping up are a special case. OpendTect is borrowing the QMessageBox, QFileDialog, QFontDialog, and QColorDialog from Qt. It is impossible for the command driver to manipulate the contents of these windows. The only QDialog parts that can be manipulated by the driver are those buttons resulting in closing the QDialog. One may use the standard commands Button, Ok, Cancel and Close for this aim. The special commands ColorOk, FileOk and FontOk will be available to specify the desired color, file, or font from the command script directly, meanwhile closing the QDialog.

Neither can the command driver manipulate the contents of windows popped up by batch programs that were launched from OpendTect. An example of this is the multi-machines processing window. Except for manual intervention, there is no workaround yet.

A higher form of window management is provided by the so-called window assertions. These are square-bracketed window names in between the command lines. All commands succeeding a window assertion are expected to be executed in the asserted window. If the current window does not match the latest window assertion, this results into the detection of an error. A window assertion is cancelled either by the next window assertion or by an empty assertion: [].

The benefit of window assertions is in error recovery. If OpendTect does not pop up a window that was expected by the command file or does pop up a window that was not expected by the command file, the Command Driver is able to proceed under guidance of these window assertions. This already solves a lot of cases in which the flow of OpendTect happens to be mildly data or environment dependent. For example, if an error message like "File already exist! Overwrite?" appears but is not handled in the command file (or the other way around), the Command Driver can continue as it was most probably intended by the user: close the window and write the file. Another example is any command to switch on/off a button or menu item. If it happens to be switched on/off already, window assertions enable the Command Driver to skip all commands following from that switch as well.

Although not compulsory, it is advised to make systematical use of window assertions. It also improves the readability of your command file. See any file generated by the Command Recorder to get an idea.

When using window assertions in combination with control flow commands, the most easy-going and efficacious method is having each individual command (If, ElseIf, Then, Fi, For, Rof, DoWhile, Od, Do, OdUntil) both preceded and succeeded by the window assertion in force. Wrongly omitting a window assertion around a control flow command will not result in an assertion violation error, but it does severely reduce the possibilities of the Command Driver to recover in case of other errors.

Search keys

Many commands supported by the Command Driver use search keys to address window names, names of buttons, input fields and other GUI-elements, item names of menus, lists, trees and tables, etc. The matching of these search keys is strict where necessary and accommodating where practical. The matching process honors the following principles:

The asterisk '*' is used as a wildcard character. It may appear any number of times in the search key to match an arbitrary substring.

Any sequence of white space characters like spaces, tabs, newlines, etc. will be removed at the beginning and at the end of both the match name and search key, and compressed into one single space when found in between.
The ellipses found at the end of menu item names and button texts are ignored. Any dot after the last non-white space character of the match name and search key is removed.

Underscores denoting keyboard shortcuts in menu item names, tab item names and button texts are ignored. The ampersands denoting these shortcuts in Qt-interface programming code should not appear in your search key, unless you want to match an actual ampersand of course.

If the name to match contains a symbol interpreted by the Command Driver as something special, the symbol must be preceded by the escape character '@' in your search key. Symbols for which this may apply include '*', '#', '$', '`', '"', '[' and ']', depending on the context in which the search key occurs. Escaping of symbols that do not have a special meaning is unnecessary but harmless. In the rare case that the match name happens to contain '@' characters preceding a symbol, these themselves have to be escaped in the search key. Alpha-numerical and white space characters are not considered symbols, so a single '@' character in a search key like "jaap.glas@dgbes.com" does not have to be escaped.

One example to demonstrate the use of escape characters. The table column header "* [All lines]" will be matched exactly by the search key "@* [All lines]". The asterisk must be escaped to prevent it from being interpreted as a wildcard for postfix matching, while the brackets do only need escapes when occurring inside window assertions.

Any pair of bracket-like characters enclosing a name to match are ignored, unless the search key does explicitly specify them. Outer character pairs that may be stripped include "...", [...], <...>, and >...<. This matching protocol is recursive. If the match name and search key do not match, eventual outer brackets of the match name are removed, and the whole matching process is repeated.

The Case-command allows the user to determine whether upper-case and lower-case alphabetical characters will match each other. The default is case-insensitive.

Identifiers and expressions

Command scripts may contain identifiers to store data and (re)use its values. Every identifier consists of a letter followed by any sequence of letters, digits and underscores ('_'). Identifiers are treated case insensitive. Identifiers can represent constants, script variables, environment variables, built-in function names, or user-defined procedure names.

The Command Driver has only one internal data type. All identifier values are represented as character strings. The string can be a boolean value, an integer value (even octal or hexadecimal), a fixed or floating point value, one single character or indeed a whole character string. The operator or built-in function using the identifier will define into which data type the string has to be converted, and will produce an error if the conversion fails. User-defined procedures should do the same. Boolean values are actually mapped onto numbers, where 0 is false and 1 is true, and any other number is also interpreted as true.

Predefined identifier constants include FALSE==0, TRUE==1, PI==3.14159265..., and UNDEF==1e30. Command actions may assign either SUCCESS==1, FAILURE==0, or

WARNING==-1 as return value. Any 'Is...On'-question command returns ON==1, OFF-F==0, or UNSWITCHABLE==-1 as an answer.

Expressions are assembled from identifiers, numbers, string constants ("..."), parentheses, built-in functions, and about twenty mathematical and logical operators (see Expressional specifications). The assign command (<ident> = <expr>) stores the evaluated result of an expression into an identifier.

The value of an identifier can be substituted anywhere in the command script by putting the identifier between dollar-signs ($<ident>$). Identifier substitution is the only and most flexible method to give command actions a variable meaning. Multiple substitutions in one command are allowed, but nested substitutions will be misinterpreted. Assign the result of the inner substitution to an auxiliary identifier, and use that one in the outer substitution.

In order to keep the parsing of command actions tractable, few commands will directly accept expressions as argument(s). Apart from the assign command, the only ones are control flow commands: If, ElseIf, For, DoWhile, OdUntil, Return, and any user-defined procedure call with value parameters. If you want to use an expression in an arbitrary command action, assign the expression result to an identifier and substitute its value.

Identifier substitution can also be used to simulate array variables (e.g. array[index]). The Command Driver does not explicitly support them, but index substitution can be applied for that purpose: array_$index$.

The predefined directory identifiers BASEDIR, DATADIR, PROCDIR, APPLDIR, USERDIR, SCRIPTSDIR, SNAPSHOTDIR, IMPORTDIR and EXPORTDIR are environment variables that can be substituted in file-path specifications. Their purpose is defining a file-path relatively to either OpendTect, the current survey, user directories, etc. The Command Driver will automatically change Unix-style file-paths into Windows-style file-paths on Windows-platforms and vice versa. The special identifier FILEIDX represents an integer variable that is automatically incremented after every occurrence in the command stream. It may be substituted in file-paths to generate unique filenames, although meanwhile more sophisticated methods exist as well (see Repetitive task example).

The scope of any identifier used in the body of a command script is global. The scope of any identifier used inside a user-defined procedure is local by default, and may shadow a global identifier with the same name. Attach the scope operator '@' in front of an identifier to force access to its global namesake. That is all for (re)assigning an identifier or (re)defining a procedure. Two extra scope rules apply for reading an identifier or calling a procedure. If an unscoped identifier value is locally undefined, the search will continue at the global scope level directly. If an unscoped procedure call is locally undefined, the search will continue at the previous scope level, and recursively descend to the global level as long as no definition is found.

Command specifications

Any text editor can be used to produce or modify OpendTect scripts. Every command file has to start with the following four-lines header. The correctness of version number, date and time is not vital for a successful execution of the command file.

```
dTect V4.0

OpendTect commands

Mon Apr 20 09:20:09 2009

!
```

The Command Driver expects one command per line and one line per command by default. Multiple commands on one line must be separated by a semicolon (';'). Long commands exceeding the width of one line may be broken by adding a backslash ('\') as last non-white character on the line. Never put a break before white space, since preceding white space on the next line will be considered indentation. Instead of using a backslash break, one can also leave the Enter-key unpressed at all and simply have the line run through the right margin. Empty lines and (commentary) lines starting with a '#'-symbol are allowed and ignored.

One can make a quick start by having the first command file generated by the Command Recorder, and start editing from there.

The specification of available commands is currently written in a top-to-bottom sequential form. This means that command arguments are explained only at their first appearance. Placeholders between angular brackets are used to specify the command syntax. If a syntax placeholder is not defined at a particular command, it has already been defined at an earlier command or is considered atomic. The online user documentation will provide a hyper-link in the former case. Our tabular representation might cause lengthy command (argument) specifications to be formatted over multiple lines. Beware that the Command Driver itself will not accept this, unless the lines are explicitly broken by backslashes. All syntax definitions are using the following tags:

<abc>	syntax placeholder
|	exclusive-or between alternatives
?	zero or one occurrences
*	zero or more occurrences
+	one or more occurrences

<action>	**I. Basic commands (also generated by Command Recorder)**
Window "<winname>" <winname> = <searchstr><disambiguator>? <searchstr> = (<wildcard>?<textstr>)*<wildcard>? <wildcard> = '*' <disambiguator> = '#'<selnr> <selnr> = <posint> | <negint>	Switches the focus between different windows on screen. Any window not having a modal child displayed may be specified as the new current window. This command is needed to access modeless windows, since only modal windows that pop up will automatically become the current window. Window names may contain one or more wildcards to match arbitrary substrings. If a (wildcarded) window name matches more than once, a number can be attached to disambiguate the search. Windows are numbered in order of appearance. A negative number will count the windows in reverse.
'**[**' <winname>? '**]**'	This window assertion tells the Command Driver what is supposed to be the current window when the commands succeeding the assertion are processed. A mismatch will result into an error. The latest assertion is cancelled by either a next assertion or the empty assertion: [ ]. If a window assertion refers to a modeless window, it will be set as the new current window. This makes the Window-command above obsolete in case window assertions are used.
Menu "<menupath>" <onofftag>? <onofftag> = (**On** | **1**) | (**Off** | **0**) <menupath> = <pathstr> <pathstr> = <itemname> (<sep><itemname>)* <sep> = '`' <itemname> = <searchstr><disambiguator>?	Selects a menu (sub)item by providing the whole menu path. Checkable menu items are toggled, unless the optional On/Off-argument specifies their desired state. Any search name provided to the Command Driver, including menu (item) names, may contain one or more wildcards to match arbitrary substrings. If a (wildcarded) item name matches more than once in a (sub) menu or any other list, a number can be attached to disambiguate the search. A negative number will count the matching items in reverse. To give a few exotic examples: the menu path "Survey`*#-1"

	would select the last item of the Survey-menu, while the path "Survey`#1" would refer to its first empty-named item.
Button "<keystr>" <onofftag>? <keystr> = <objname>(<sep><objname>) *<disambiguator>? <objname> = <searchstr>	Press a (push/radio/check/tool)-button by providing one or more search keys. Multiple search keys are not allowed for QDialog windows. Checkable buttons are toggled, unless the optional On/Off-argument specifies their desired state. The best search key is of course the button name, but some objects do not have a (visible) name and lots of times names are not unique. Therefore, a key may just as well refer to another object in the neighbourhood of the button to click. The list of possible buttons will be narrowed down for every next key provided. If the referred object is closer to one button than another, the latter will be dropped. The vicinity criterion is based on the youngest common ancestor of referred object and button in the object tree. If the user is unable to specify a set of keys that leaves exactly one button, as a last resort one of the remaining buttons can be selected by attaching a disambiguation number after the last key. Beware that the order of those buttons will be fixed, but unlike menus not always in a left-right top-down fashion. The buttons will be counted in reverse in case the disambiguator is negative.
ButtonMenu "<keystr>" "<menupath>" <onofftag>?	Selects an item from the menu attached to a button. Some tool-buttons contain a left-clickable arrow part to have such a menu appeared. Checkable menu items are toggled, unless the optional On/Off-argument specifies their desired state. Selection of the button is analogous to the button selection described above. Selection of the menu (sub)item is analogous to the Menu-command.

Input "<keystr>" <inpdata>? <entertag>? <entertag> = **Enter** | **Hold** <inpdata> = "<inputstr>" | <number> | **FilePath** "<filepathstr>"	Inputs a string or number into one of the fields of the current window. The field selection is analogous to the button selection described above. The Hold-option only triggers actions defined on changing the input field, while the Enter-option also triggers actions defined on pressing the enter/return-key afterwards. The latter is the default. If no input data is provided, the current contents of the selected field will be entered. The FilePath-option forces the input string to be treated as a file-path concerning platform independence. This will happen automatically in case the input string contains any directory identifier substitute ($...DIR$), or in case the input field comes together with a Select-button and perhaps an Examine-button into a graphical element called uiFileInput.
Spin "<keystr>" <spinsteps> <entertag>? <spinsteps> = <posint> | <negint>	Clicks a spinbox any number of steps upward (positive) or downward (negative). The spinbox selection is analogous to the button selection described above. The Hold-option only triggers actions defined on changing the spinbox value (after each step), while the Enter-option also triggers actions defined on the spinbox losing its focus afterwards. The latter is the default. If no number of steps is provided, the spinbox will keep its current value while losing its focus. Multiple steps are only supported as long as no action triggered in between does pop up a modal window. One can also edit the input field of the spinbox directly by using the Input-command instead.
Slider "<keystr>" <percentage> <nrsteps>? <nrsteps> = <posint>	Shifts a slider towards the specified percentage of its range displayed on screen. Beware that the relationship between this percentage and the actual scale represented by the slider is not necessarily linear. It is optional to perform the shift in more than one step, but it may yield nice

	animations on screen. Multiple steps are only supported as long as no action triggered in between does pop up a modal window. Selecting the slider is analogous to the button selection described above.
Wheel "<keystr>" <degrees> <nrsteps>?	Rotates a thumbwheel a specified number of degrees upwards/downwards or leftwards/rightwards. The number of degrees will only have a direct meaning if the thumbwheel represents an angular scale. It is optional to perform the rotation in more than one step, but it may yield nice animations on screen. Multiple steps are only supported as long as no action triggered in between does pop up a modal window. Selecting the thumbwheel is analogous to the button selection described above.
Combo "<keystr>" <itemsel> <itemsel> = "<itemname>" | <selnr>	Sets the selected item of a combobox, either by its name or by its ordinal number in the list. The list will be traversed in reverse if the number is negative, so for instance -1 refers to the last item. Using a selection number is just a shortcut for composing an item name "*#<selnr>" from merely a wildcard and a disambiguator. The selection of the combobox itself is analogous to the button selection described above. The Input-command may be applied to edit the input field of the current item of an editable combobox.
ListClick "<keystr>" <itemsel> <mousetag>? <mousetag> = <ctrlclick>?<doubleclick>?<leftrightclick>? <ctrlclick> = **Ctrl** <doubleclick> = **Double** <leftrightclick> = **Left** | **Right**	Clicks and (de)selects precisely one item in a listbox. Selection possibilities for listbox and item are analogous to combobox selection. The optional mousetag is defining whether the item is (de)selected by means of optional control- or double-clicking of either the left or right mouse button. Left is default in all cases. Beware that the mousetag is one united word and order counts. Some listboxes pop up a menu when (right)-clicking on an item, in which

	case one has to apply the ListMenu-command instead. Single-selection listboxes will only select the clicked item and deselect all others. Multi-selection listboxes act different in the following cases. Control-clicking toggles the selected state of the clicked item, while any other item keeps its current state. Right-clicking an already selected item leaves the whole selection unchanged.
ListButton "<keystr>" <itemsel> <mousetag>? <onofftag>?	(Un)checks the button in front of a listbox item. The check-button is toggled unless the optional On/Off-argument specifies its desired state. (De)selection of item and listbox is fully analogous to the ListClick-command. Beware that mousetags with Double or Right are known not to have a (lasting) effect on the check-button.
ListMenu "<keystr>" <itemsel> <mousetag>? "<menupath>" <onofftag>?	Selects a (sub)item from the menu attached to a listbox item. Checkable menu items are toggled, unless the optional On/Off-argument specifies their desired state. The selection of both listbox and item is analogous to the ListClick-command. However, since OpendTect is normally hiding its popup menus under the right mouse button, the default for no mousetag at all is set Right over here. Selection of the menu (sub)item is analogous to the Menu-command.
ListSelect "<keystr>" <firstitemsel> <lastitemsel>? <onofftag>? <firstitemsel> = <lastitemsel> = <itemsel>	Selects any number of items in a multi-selection listbox. Selection possibilities are more comprehensive than those at the ListClick-command. Now all items matching a given (wildcarded) item name can be specified. One can also specify the whole range between a first and a last item at once. The list will be traversed cyclically in case the first item succeeds the last. Without the optional On/Off-argument, all specified items will be selected and all other items deselected. With the On/Off-

	argument set however, only specified items will be selected/deselected respectively, while unspecified items keep their current state.
TableClick "<keystr>" <tableitemsel> <mousetag>? <tableitemsel> = <headitemsel> | <cellsel> <headitemsel> = <headtag> <itemsel> <headtag> = **RowHead** | **ColHead** <cellsel> = <rowsel> <colsel> | **Cell** <itemsel> <rowsel> = <colsel> = <itemsel>	Clicks either a row-head, col-head or cell in a table, and select precisely the one row, column or cell attached to it. Selection possibilities for the RowHead-, ColHead- and Cell-options are analogous to the item selection for list- and comboboxes. The Cell-option puts all cells row-after-row in a virtual list for this purpose. Another way to address a single cell is by selecting both its row and its column. Also these row and column selections are analogous to the item selection just mentioned. If the row or column selection is not made by number but by name, then the search for a match will start in its header. In case the table shows no header or no row/col-head is matching, then the search will proceed in the next column/row until a match is found. The selection of the table itself is analogous to the button selection described above. The option specifying a left, right or double mouse-click will be Left by default. Some tables pop up a menu when right-clicking a cell, in which case one has to apply the TableMenu-command instead.
TableFill "<keystr>" <cellsel> <inpdata>	Fills one cell in a table with new text data. Alike the Input-command, this might be a text string, a file-path string or a number. The selection of both table and cell is analogous to the TableClick-command, except that table headers cannot be filled.
TableMenu "<keystr>" <cellsel> <mousetag>? "<menupath>" <onofftag>?	Selects a (sub)item from the menu attached to a table cell. Checkable menu items are toggled, unless the optional On/Off-argument specifies their desired state. The selection of both table and cell is analogous to the TableFill-command. Selection of the menu (sub)item is ana-

	logous to the Menu-command.
TableExec "<keystr>" <cellsel> <action>	Executes a local command driver action within one cell of a table (instead of within the current window). The selection of both table and cell is analogous to the TableFill-command. Only those commands accepting a keystring argument might be appropriate actions to execute within a cell. For example, if the top-left cell of a table contains a single combobox, its selection can be made as follows: **TableExec** "my table" 1 1 **Combo** "*" "my item" Tables with cells containing multiple user-interface objects of the same or different kind can be handled too.
TableSelect "<keystr>" <tableitem-rangesel> <onofftag>? <tableitemrangesel> = <head-itemrangesel> | <cellrangesel> <headitemrangesel> = <headtag> <firstitemsel> <lastitemsel>? <cellrangesel> = <firstcellsel> <last-cellsel>? <firstcellsel> = <lastcellsel> = <cellsel>	Selects any number of rows, columns or cells in a table. This command is the table-equivalent of the ListSelect-command. Selection possibilities are more comprehensive than those at the TableClick-command. Now all row-heads, col-heads or cells matching a given (wildcarded) item name can be specified. One can also specify the whole block between a first and a last row, column, or cell at once. The table will be traversed cyclically in case the first (cell) row or (cell) column succeeeds the last. Without the optional On/Off-argument, all specified cells will be selected and all other cells deselected. With the On/Off-argument set however, only specified cells will be selected/deselected respectively, while unspecified cells keep their current state.
Tab "<keystr>"? "<tabname>" <tabname> = <itemname>	Puts a tab on top of the stack by name. Since windows with more than one tab-stack will be rare, its selection is optional. The selection is analogous to the button selection described above.

TreeClick "<keystr>"? <treenodesel> <mousetag>? <treenodesel> = "<treepath>" | PathCol "<treepath>" <colsel> <treepath> = <pathstr>	Clicks and selects precisely one node in a tree. The selection of a tree node is analogous to the selection of a menu (sub) item. In which column to click is optional, but it will be the first one by default. This column selection is almost analogous to the column selection for tables. If the selection is not made by number but by name, then the search for a match will start in the column header, and next proceed at the selected tree node if not successful. Selection of the tree itself is analogous to the button selection described above. It is optional because the current window will often contain only one (data) tree. The data tree with the lowest scene number is guaranteed to be the default for OpendTect Main Window. The option to specify a left, right or double mouse-click will be Left by default. Any tree node might pop up a menu when right-clicking on one of its columns, in which case the TreeMenu-command has to be applied instead.
TreeExpand "<keystr>"? "<treepath>" <onofftag>?	(Un)expands the subtree of a node in a tree. The expander is toggled unless the optional On/Off-argument specifies its desired state. The selection of both tree and node is analogous to the TreeClick-command, except that column selection is not an issue here.
TreeButton "<keystr>"? "<treepath>" <mousetag>? <onofftag>?	Presses the button in front of a node in a tree. The button is toggled unless the optional On/Off-argument specifies its desired state. The selection of both tree and node is analogous to the TreeClick-command, except that column selection is not an issue here.
TreeMenu "<keystr>"? <treenodesel> <mousetag>? "<menupath>" <onofftag>?	Selects a (sub)item from the menu attached to a column of a tree node. Checkable menu items are toggled, unless the optional On/Off-argument specifies their desired state. The selection of tree, node

	and column is analogous to the TreeClick-command. Selection of the menu (sub)item is analogous to the Menu-command.
CanvasMenu "<keystr>" "<menupath>" <onofftag>?	Selects a (sub)item from the menu popping up at a canvas area. Checkable menu items are toggled, unless the optional On/Off-argument specifies their desired state. Selection of the canvas area is analogous to the button selection described above. Selection of the menu (sub)item is analogous to the Menu-command.
Ok **Cancel**	These are special commands that 'Ok' or 'Cancel' a dialog. Usually, this has the same effect as pressing the Ok- or Cancel-button.
Close <closeoption>? <closeoption> = **All** | <subwinsel> <subwinsel> = "<keystr>"? "<winname>"	Clicks on the Close-button in the title bar of the current window. The All-option will close all OpendTect windows at once. This option is compulsory in case OpendTect Main Window is the current window, so that OpendTect cannot be killed by accident. The optional subwindow selection is available to close a window in the workspace of the current window by name. Since windows with more than one workspace will be rare, selection of the workspace is optional. This selection is analogous to the button selection described above.
Show <subwinsel>? <showtag> <showtag> = **Minimized** | **Maximized** | **Normal**	Clicks on the Minimized-, Maximized- and Restore-buttons in the title bar of the current window. The optional subwindow selection is available to resize a window in the workspace of the current window by name. Since windows with more than one workspace will be rare, selection of the workspace is optional. This selection is analogous to the button selection described above.
ColorOk <colorsel> <colorsel> = "<rgbtcolorstr>" | <color>	Specifies the desired color while closing a QColorDialog window. One may specify a

<transparency>? <color> = "<rgbcolorstr>" | <R> <G> <B> | <colortag> <rgbtcolorstr> = <rgbstr><sep><transparency> <rgbcolorstr> = <R><sep><G><sep><B> <R> = <G> = <B> = <transparency> = <byte> <colortag> = **Black** | **Blue** | **Brown** | **Cyan** | **Green** | **Grey** | **Lilac** | **Lime** | **Magenta** | **Olive** | **Orange** | **Purple** | **Pink** | **Red** | **White** | **Yellow**	color either by its RGB-values (0-255) or a color tag. In case the QColorDialog offers the possibility to specify transparency, the value of the optional t-channel (0-255) is passed as well. Its default value is 0 (non-transparent). The RGB-values and optional t-channel can also be specified in one composite RGB(t) color string.
FileOk "<filepathset>" <filepathset> = <filepathstr> (<sep><filepathstr>)*	Specifies one, or more file-paths while closing a QFileDialog window. The command will yield an error message if their number, type (file/directory), extension, and/or writability is not in agreement with the current mode of the QFileDialog. File-paths will be interpreted platform independently. Both absolute and relative file-paths are accepted. The current directory of the QFileDialog will be taken as base directory in the latter case. Also a set of directory identifiers has been predefined for substitution ($...DIR$) in file-paths. The FileOk-command does not provide the functionality of the QFileDialog-button "Create new folder". If a file-path specifies a non-existing file or directory, its parent directory must exist.

<action>	**II. Auxiliary commands (to be inserted by the user)**
Include "<filepathstr>"	Inserts another command file into the command stream. The file-path will be interpreted platform independently. In order to specify the file-path relative to either OpendTect, current survey, or user dir-

<table>
<tr><td></td><td>ectories, one of the predefined directory identifiers can be substituted ($...DIR$).</td></tr>
<tr><td>GreyOuts <greyoutstag>

<greyoutstag> = Count | Skip</td><td>Specifies whether to count or skip disabled (greyed out) user interface objects, tabs, menu and tree items. 'Skip' is the default. 'Count' means that also grey-outs will be counted when using selection numbers. Furthermore, it allows (question) commands to retrieve information from grey-outs. Command actions trying to manipulate a grey-out will be penalized.</td></tr>
<tr><td>Case <casetag>

<casetag> = Sensitive | Insensitive</td><td>Switches between case-sensitive and case-insensitive searching in menus, windows, lists, etc. Insensitive is the default. The command parsing itself (command names and options, function names, identifier names) is case-insensitive by nature and will ignore this setting.</td></tr>
<tr><td>OnError <errortag>

<errortag> = Stop | Recover</td><td>Switches between different procedures to handle an error. The Stop-option will exit the driver immediately. The Recover-option will make use of window assertions to safely proceed execution in specific cases. The default option is Recover.</td></tr>
<tr><td>LogMode <logtag>

<logtag> = Basic | Normal | All</td><td>Regulates the amount of warning and error messages in the log file. The All-option will show any message generated. The Normal-option is the default. It shows all parsing messages, but action messages are only shown if the action result is not assigned to an identifier. The Basic-option will also omit all warning messages.</td></tr>
<tr><td>Snapshot "<imagefilepathstr>" <frametag>?

<frametag> = CurWin | ODMain | Desktop

<imagefilepathstr> = <filepathstr><imageext>
<imageext> = .bmp | .jpg | .jpeg | .png | .ppm | .xbm | .xpm</td><td>Writes a snapshot of the current window (and its environment) to file. The default grabbing area is bounded by the CurWin-frame, but can optionally be enlarged towards the ODMain-frame or the whole Desktop-frame. The snapshot filename must have one of the prescribed image extensions. The file-path will be interpreted platform independently. Both absolute and</td></tr>
</table>

	relative file-paths are accepted. Also one of the predefined directory identifiers may be substituted ($...DIR$) in the file-path.
Sleep <seconds> <sleeptag>? <sleeptag> = **Regular** | **Extra**	Sleeps a period of time so that spectators can distinguish the consecutive steps from a command file on screen. The Regular-option will sleep until further notice between every two commands with a visual effect. The Extra-option is the default and will take an (additional) nap only once.
Wait <seconds> <sleeptag>?	Tells the next command to wait a period of time only if it is uncertain whether it has finished processing. This can happen to any command closing a modal dialog that was already open when the Command Driver started. The Regular-option will allow this waiting time to every command until further notice. The Extra-option is the default and will allow an (additional) wait only to the next command.
Pause "<textlines>"? <textlines> = <textstr>(<sep><textstr>)*	Temporarily hold the execution of the command script and have the Command Controller pop up a message dialog with a 'Resume'-button so that the user can decide when to continue. Specifying lines of text is optional.
Guide "<textlines>" (<guidetag> "<winname>")? <guidetag> = **Existent** | **Inexistent** | **Accessible** | **Inaccessible**	Temporarily hold the execution of the command script and have the Command Controller pop up a dialog requesting the user to take action. The text lines describe which actions the user has to perform. The Command Driver will automatically resume if some window matching a given name is no longer (in)existent or (in)accessible. If this option is not specified, the user gets a 'Done'-button to have the Command Driver resume manually.
Comment "<textlines>"	Inserts comment lines into the log file. Command lines starting with a '#'-symbol are containing comments that are not shown in log files.

<table>
<tr><td><action></td><td>III. Control flow commands (to be inserted by the user)</td></tr>
<tr><td>Assign <ident> <expr>

<ident> '=' <expr>

<ident> = <scope>?<identstr>
<identstr> = <letter><identchars>*
<identchars> = <letter> | <digit> | '_'
<scope> = '@'

<expr> = <ident> | <number> | "<textstr>" | '(' <expr> ')' | <functioncall> | <operatorexpr>

<functioncall> = <funcname>'(' <expressions>? ')'
<funcname> = <ident>
<expressions> = <expr> (',' <expr>)*</td><td>Assigns the evaluated result of an expression to an identifier. The command syntax is available in both procedural style and operator style. Identifiers consist of a letter followed by any sequence of letters, digits and underscores. Expressions are assembled from identifiers, numbers, string constants, parentheses, built-in functions, and about twenty mathematical and logical operators. Notice that a function call allows no space between the function name and its opening parenthesis. All functions, operators and predefined identifiers are listed below in Expressional specifications.</td></tr>
<tr><td>If <expr> <actions>

(ElseIf <expr> <actions>)*

(Else <actions>)?

Fi</td><td>Executes a number of command actions if a boolean expression evaluates to true. The ElseIf- and Else-branches are optional, but note that the terminating Fi-command is compulsory.</td></tr>
<tr><td>For <ident> '=' <expr> (To <expr>)? (Step <expr>)?

<actions>

Rof</td><td>Initializes an identifier with the evaluated result of a numerical expression and repeats a number of command actions as long as the identifier value does not exceed the evaluated result of the optional To-expression. After every loop iteration, the identifier is incremented by the evaluated result of the optional Step-expression. The default step is 1, but even negative values are allowed. Only the Break-command can escape from a For-loop in absence of a To-expression. Note that the terminating Rof-command is compulsory.</td></tr>
<tr><td>DoWhile <expr>

<actions></td><td>Repeats the execution of a number of command actions as long as a boolean expression evaluates to true. Note that the</td></tr>
</table>

Od	terminating Od-command is compulsory.
Do <actions> **OdUntil** <expr>	Repeats the execution of a number of command actions until a boolean expression evaluates to true.
Break	Escapes immediately from the innermost For-, While-, or Until-loop.
Continue	Skips the remaining actions in the current iteration of the innermost For-, While-, or Until-loop.
Try <ident> <action> <ident>? '~' <action>	Tries to execute a command action and assigns its result to an identifier. The command syntax is available in both procedural style and operator style. The possible outcomes are success (1), failure (0) and warning (-1), for which the identifier constants SUCCESS, FAILURE and WARNING have been predefined. The operator style syntax is also usable without identifier for the side-effect, because error messages will be temporarily ignored.
Questioncmd < **questioncmd** > <ident> <questionargs> <ident> '?' <**questioncmd**><questionargs>	Stores the answer from a question command into an identifier. The command syntax is available in both procedural style and operator style. All question commands implemented so far are listed below in Table IV to VII.
Def (<returnpar> '?')? <procname>'(' <valpars>? ')' <varpar>* <actions> **Fed** <returnpar> = <varpar> <procname> = <ident> <valpars> = <identstr> (',' <identstr>)* <varpar> = <identstr>	Specifies a user-defined procedure in which a number of command actions are executed. Note that the terminating Fed-command is compulsory. Nested procedure definitions are allowed. A definition may occur anywhere, as long as it precedes the first call to it. The course of a procedure depends on an optional number of value parameters (between the parentheses) and variable parameters (behind the parentheses). Procedure names can only be overloaded if there is a difference in the

	number of value or variable parameters. Distinction on data type is not possible, having only one internal (string) type. The optional return parameter is not counted at overloading, but acts like a variable parameter apart from that. It can be applied to give the procedure a more function-like appearance. Unlike built-in functions however, this does not at all mean that procedure calls can be used as (sub) expressions.
Return <expr>?	Escapes immediately from the current procedure. In case its definition includes a return parameter, it will be assigned with the optional expression that is returned.
(<ident> '?')? **<procname>'('** <expressions>? **')'** <ident>*	Calls a user-defined procedure specified earlier by means of the Def-command above. Its optional value parameters will accept expressions. Its optional return and variable parameters only accept identifiers, and these might be modified. Notice that a procedure call allows no space between the procedure name and its opening parenthesis.
End	Finishes the command stream immediately.

<questioncmd> <questionargs>	**IV. Question commands (to be inserted by the user)**
IsMatch "<searchstr>" "<textstr>" <casetag>	True (1) if the text string matches the search key according to the rules described in the Search keys paragraph, false (0) otherwise. The identifier constants TRUE and FALSE have been predefined for convenience. If the optional case-sensitivity of the match is not specified, the global settings defined by the Case-command will apply. The options are 'Insensitive' or 'Sensitive'.
IsWindow "<winname>"? <winproptag>? <winproptag> = **Existent** | **Accessible** |	True (1) if a specified window has a particular boolean property, false (0) otherwise. The optional property tells whether

Modal | **QDialog** | <showtag>	the window is either existent (default), accessible, modal, maximized, minimized, normal or a QDialog. The optional selection of the window is analogous to the Window-command. By default, the current window will be selected.
IsMenuItemOn "<menupath>"	On (1) if the selected menu (sub)item is checked, off (0) if it is unchecked, and unswitchable (-1) if it is not checkable at all. The identifier constants ON, OFF and UNSWITCHABLE have been predefined for convenience. The selection of the menu (sub)item is analogous to the Menu-command.
NrMenuItems "<menupath>?"	Returns the number of (enabled) items in the selected (sub)menu. The GreyOuts-command defines whether disabled items are counted as well. Selection of a sub-menu is analogous to the selection of menu items in the Menu-command. The root menu is denoted by an empty menu-path (""). Zero is returned if the menu-path leads to a leaf menu item.
GetMenuItem "<menupath>" <formtag>? <formtag> = **Text** | **Number**	Returns the text or number of a selected menu (sub)item. In which form is optional and 'Text' by default. In the 'Number' case, the GreyOuts-command defines whether disabled items are counted as well. Selection of the menu (sub)item is analogous to the Menu-command.
IsButtonOn "<keystr>"	On (1) if the selected radio-, check-, or toolbar button is checked, off (0) if it is unchecked, and unswitchable (-1) in case of a push button or a non-checkable toolbar button. The identifier constants ON, OFF and UNSWITCHABLE have been predefined for convenience. The button selection is analogous to the Button-command.
GetButton "<keystr>" <buttonformtag>? <buttonformtag> = **Text** | **Color**	Returns the text or color of a (push/radio/check/tool)-button. In which form is optional and 'Text' by default. The RGBt

<table>
<tr><td></td><td>color string format returned in the 'Color' case is defined at the ColorOk-command. If the button has no text or color, an empty string or transparent white is returned respectively. The button selection is analogous to the Button-command.</td></tr>
<tr><td>IsButtonMenuItemOn "<keystr>" "<menu-path>"</td><td>On (1) if the selected menu (sub)item of a button is checked, off (0) if it is unchecked, and unswitchable (-1) if it is not checkable at all. The selection of button and menu (sub)item is analogous to the ButtonMenu-command.</td></tr>
<tr><td>NrButtonMenuItems "<keystr>" "<menu-path>?"</td><td>Returns the number of (enabled) items in the selected (sub)menu of a button. The GreyOuts-command defines whether disabled items are counted as well. The selection of button and sub-menu is analogous to the ButtonMenu-command. The root menu is denoted by an empty menu-path (""). Zero is returned if the menu-path leads to a leaf menu item.</td></tr>
<tr><td>GetButtonMenuItem "<keystr>" "<menu-path>" <formtag>?</td><td>Returns the text or number of the selected menu (sub)item of a button. In which form is optional and 'Text' by default. In the 'Number' case, the GreyOuts-command defines whether disabled items are counted as well. Selection of button and menu (sub) item is analogous to the ButtonMenu-command.</td></tr>
<tr><td>GetInput "<keystr>" <inputformtag>?

<inputformtag> = Text | FilePath</td><td>Returns the current content of a selected input field. In which form is optional and 'Text' by default. Selection of the input field is analogous to the Input-command. In case the input field comes together with a Select-button and perhaps an Examine-button into a graphical element called uiFileInput, the 'FilePath' option forces the current filename to be preceded by the absolute file path to the current selection directory. Otherwise an empty string will be returned.</td></tr>
<tr><td>GetSpin "<keystr>" <spinformtag>?</td><td>Returns the text, value, minimum, maximum</td></tr>
</table>

<spinformtag> = **Text** | **Value** | **Minimum** | **Maximum** | **Step**	or step of the selected spinbox. In which form is optional and textual by default. Selection of the spinbox is analogous to the Spin-command.
GetSlider "<keystr>" <sliderformtag>? <sliderformtag> = **Text** | **Value** | **Minimum** | **Maximum** | **Percentage**	Returns the text, value, minimum, maximum or percentage of the selected slider. In which form is optional and textual by default. Beware that the returned percentage of the range displayed on screen will not necessarily have a linear relationship with the returned value. Selection of the slider is analogous to the Slider-command.
GetWheel "<keystr>" <wheelformtag>? <wheelformtag> = **Text** | **Angle**	Returns the text or angle (in degrees) of the selected thumbwheel. In which form is optional and textual by default. Selection of the thumbwheel is analogous to the Wheel-command.
NrComboItems "<keystr>"	Returns the number of items in a selected combobox. The selection of the combobox is analogous to the Combo-command.
CurComboItem "<keystr>" <formtag>?	Returns the text or number of the current combobox item. In which form is optional and textual by default. The selection of the combobox is analogous to the Combo-command.
IsComboItemOn "<keystr>" <itemsel>	On (1) if the specified combobox item is currently selected, and off (0) if it is currently deselected. Specification of the combobox and its item is analogous to the Combo-command.
GetComboItem "<keystr>" <itemsel> <formtag>?	Returns the text or number of a selected combobox item. In which form is optional and textual by default. Selection of the combobox and its item is analogous to the Combo-command.
NrTabs "<keystr>"?	Returns the number of (enabled) tabs in a selected tab-stack. The GreyOuts-command defines whether disabled tabs are counted as well. The optional selection of the tab-stack is analogous to the Tab-com-

	mand.
CurTab "<keystr>"? <formtag>?	Returns the text or number of the current tab. In which form is optional and 'Text' by default. In the 'Number' case, the GreyOuts-command defines whether disabled tabs are counted as well. The optional selection of the tab-stack is analogous to the Tab-command.
IsTabOn "<keystr>"? "<tabname>"	On (1) if the selected tab is currently on top, and off (0) if it is currently underneath. Selection of tab-stack and tab-name is analogous to the Tab-command.
GetTab "<keystr>"? "<tabname>" <formtag>?	Returns the text or number of a selected tab. In which form is optional and 'Text' by default. In the 'Number' case, the GreyOuts-command defines whether disabled tabs are counted as well. Selection of tab-stack and tab-name is analogous to the Tab-command.
IsCanvasMenuItemOn "<keystr>" "<menu-path>"	On (1) if the selected menu (sub)item in the pop-up menu of a canvas area is checked, off (0) if it is unchecked, and unswitchable (-1) if it is not checkable at all. The selection of canvas area and menu (sub)item is analogous to the CanvasMenu-command.
NrCanvasMenuItems "<keystr>" "<menu-path>?"	Returns the number of (enabled) items in the selected (sub)menu popping up at a canvas area. The GreyOuts-command defines whether disabled items are counted as well. The selection of canvas area and sub-menu is analogous to the CanvasMenu-command. The root menu is denoted by an empty menu-path (""). Zero is returned if the menu-path leads to a leaf menu item.
GetCanvasMenuItem "<keystr>" "<menu-path>" <formtag>?	Returns the text or number of the selected menu (sub)item in the pop-up menu of a canvas area. In which form is optional and 'Text' by default. In the 'Number' case, the GreyOuts-command defines whether disabled items are counted as well. The selec-

	tion of canvas area and menu (sub)item is analogous to the CanvasMenu-command.
IsShown <subwinsel>? <showtag>	True (1) if a selected subwindow in the workspace of the current window is minimized, maximized, or normal size respectively, and false (0) otherwise. Specification of size property and optional subwindow is analogous to the Show-command. Not selecting a subwindow will yield the size properties of the current window itself.

<questioncmd> <questionargs>	**V. List question commands (to be inserted by the user)**
NrListItems "<keystr>"	Returns the number of items in a selected listbox. The selection of the listbox is analogous to the ListClick-command.
CurListItem "<keystr>" <curtag>? <listformtag>? <curtag> = **Framed** | **Selected** <listformtag> = **Text** | **Number** | **Color**	Returns the text, number or background color of the current listbox item. In which form is optional and 'Text' by default. The RGBt color string format returned in the 'Color' case is defined at the ColorOk-command. If there is no current listbox item, an empty string, zero or transparent white is returned respectively. By default, the current listbox item is the one that is 'Framed'. Optionally, if precisely one item is 'Selected' (i.e. highlighted), it can be requested as current listbox item instead. The selection of the listbox is analogous to the ListClick-command.
IsListItemOn "<keystr>" <itemsel>	On (1) if a specified item in a listbox has been selected (i.e. highlighted), off (0) if it has been deselected, and unswitchable (-1) if the listbox does not support item selection at all. The specification of listbox and item is analogous to the ListClick-command.
GetListItem "<keystr>" <itemsel> <listformtag>?	Returns the text, number or background color of a selected listbox item. In which form is optional and 'Text' by default. The RGBt color string format returned in the 'Color' case is defined at the ColorOk-com-

	mand. Selection of both listbox and item is analogous to the ListClick-command.
IsListButtonOn "<keystr>" <itemsel>	On (1) if the button in front of a listbox item is checked, and off (0) if it is unchecked. Selection of both listbox and item is analogous to the ListButton-command.
IsListMenuItemOn "<keystr>" <itemsel> "<menupath>"	On (1) if the selected menu (sub)item of a listbox item is checked, off (0) if it is unchecked, and unswitchable (-1) if it is not checkable at all. The selection of the listbox, its item and the menu (sub)item is analogous to the ListMenu-command.
NrListMenuItems "<keystr>" <itemsel> "<menupath>?"	Returns the number of (enabled) items in the selected (sub)menu of a listbox item. The GreyOuts-command defines whether disabled items are counted as well. Selection of the listbox, its item and a sub-menu is analogous to the ListMenu-command. The root menu is denoted by an empty menu-path (""). Zero is returned if the menu-path leads to a leaf menu item.
GetListMenuItem "<keystr>" <itemsel> "<menupath>" <formtag>?	Returns the text or number of the selected menu (sub)item of a listbox item. In which form is optional and 'Text' by default. In the 'Number' case, the GreyOuts-command defines whether disabled items are counted as well. The selection of the listbox, its item and the menu (sub)item is analogous to the ListMenu-command.

<questioncmd> <questionargs>	**VI. Table question commands (to be inserted by the user)**
NrTableRows "<keystr>"	Returns the number of rows in the selected table. The selection of the table is analogous to the TableClick-command.
NrTableCols "<keystr>"	Returns the number of columns in the selected table. The selection of the table is analogous to the TableClick-command.
CurTableItem "<keystr>" <curtag>? <table-formtag>?	Returns the text, number or background color of the current table item. In which form

<tableformtag> = **Text** | **Number** | **Color**	is optional and 'Text' by default. The RGBt color string format returned in the 'Color' case is defined at the ColorOk-command. If there is no current table item, an empty string, zero or transparent white is returned respectively. Like the CurListItem-command, the current table item is by default the one that is 'Framed'. Optionally, if precisely one item is 'Selected' (i.e. highlighted), it can be requested as current table item instead. The selection of the table is analogous to the TableClick-command.
CurTableRow "<keystr>" <curtag>? <tableformtag>?	Returns the row-header text, number or background color of the current table row. In which form is optional and 'Text' by default. The RGBt color string format returned in the 'Color' case is defined at the ColorOk-command. If the table has no row header or no current row, an empty string, zero or transparent white is returned respectively. Like the CurListItem-command, the current table row is by default defined by the item that is 'Framed'. Optionally, if precisely one (entire row of) item(s) is 'Selected' (i.e. highlighted), the row at issue can be requested as current table row instead. The selection of the table is analogous to the TableClick-command.
CurTableCol "<keystr>" <curtag>? <tableformtag>?	Returns the column-header text, number or background color of the current table column. In which form is optional and 'Text' by default. The RGBt color string format returned in the 'Color' case is defined at the ColorOk-command. If the table has no column header or no current column, an empty string, zero or transparent white is returned respectively. Like the CurListItem-command, the current table column is by default defined by the item that is 'Framed'. Optionally, if precisely one (entire column of) item(s) is 'Selected' (i.e. highlighted), the column at issue can be requested as cur-

	rent table column instead. The selection of the table is analogous to the TableClick-command.
IsTableItemOn "<keystr>" <tableitemsel>	On (1) if a specified item in a table has been selected (i.e. highlighted), off (0) if it has been deselected, and unswitchable (-1) if the table does not support item selection at all. The specification of table and item is analogous to the TableClick-command. A row-head or col-head item is considered selected only if all table cells in that row or column are selected.
GetTableItem "<keystr>" <cellsel> <tableformtag>?	Returns the text, number or background color of a selected table cell. In which form is optional and 'Text' by default. In the 'Number' case, the table cells are counted row-by-row. The RGBt color string format returned in the 'Color' case is defined at the ColorOk-command. Selection of both table and cell is analogous to the TableFill-command.
GetTableRow "<keystr>" <tableitemsel> <tableformtag>?	Returns the row-header text, number or background color of a selected table item. In which form is optional and 'Text' by default. The RGBt color string format returned in the 'Color' case is defined at the ColorOk-command. Selection of both table and item is analogous to the TableClick-command. If the selected item is not a row-head item itself, it refers to the row-head item straight above it.
GetTableCol "<keystr>" <tableitemsel> <tableformtag>?	Returns the column-header text, number or background color of a selected table item. In which form is optional and 'Text' by default. The RGBt color string format returned in the 'Color' case is defined at the ColorOk-command. Selection of both table and item is analogous to the TableClick-command. If the selected item is not a col-head item itself, it refers to the col-head item left next to it.

IsTableMenuItemOn "<keystr>" <cellsel> "<menupath>"	On (1) if the selected menu (sub)item of a table cell is checked, off (0) if it is unchecked, and unswitchable (-1) if it is not checkable at all. The selection of the table, its cell and the menu (sub)item is analogous to the TableMenu-command.
NrTableMenuItems "<keystr>" <cellsel> "<menupath>?"	Returns the number of (enabled) items in the selected (sub)menu of a table cell. The GreyOuts-command defines whether disabled items are counted as well. Selection of the table, its cell and a sub-menu is analogous to the TableMenu-command. The root menu is denoted by an empty menupath (""). Zero is returned if the menu-path leads to a leaf menu item.
GetTableMenuItem "<keystr>" <cellsel> "<menupath>" <formtag>?	Returns the text or number of the selected menu (sub)item of a table cell. In which form is optional and 'Text' by default. In the 'Number' case, the GreyOuts-command defines whether disabled items are counted as well. The selection of the table, its cell and the menu (sub)item is analogous to the TableMenu-command.

<questioncmd> <questionargs>	**VII. Tree question commands (to be inserted by the user)**
NrTreeItems "<keystr>"? "<treepath>?"	Returns the number of (enabled) items in a selected tree node. The GreyOuts-command defines whether disabled items are counted as well. The optional selection of the tree is analogous to the TreeClick-command. This also holds for its tree node, except that column selection is not an issue here. The root node is denoted by an empty tree path (""). Zero is returned if the tree path leads to a leaf tree item.
NrTreeCols "<keystr>"?	Returns the number of columns in a tree. The optional selection of the tree is analogous to the TreeClick-command.
CurTreeItem "<keystr>"? <curtag>? <formtag>?	Returns the text or number of the current tree item. In which form is optional and tex-

	tual by default. In the 'Number' case, the GreyOuts-command defines whether disabled items are counted as well. If there is no current tree item, an empty string or zero is returned respectively. Like the CurListItem-command, the current tree item is by default the one that is 'Framed'. Optionally, if precisely one item is 'Selected' (i.e. highlighted), it can be requested as current tree item instead. The optional selection of the tree is analogous to the TreeClick-command.
CurTreePath "<keystr>"? <curtag>? <formtag>?	Returns the path to the current tree item. In which form is optional and textual by default. The 'Number' case is especially useful if 'Text' would yield an ambiguous tree path. The GreyOuts-command defines whether disabled items are counted as well. If there is no current tree item, an empty path ("") is returned. Like the CurListItem-command, the current tree item is by default the one that is 'Framed'. Optionally, if precisely one item is 'Selected' (i.e. highlighted), it can be requested as current tree item instead. The optional selection of the tree is analogous to the TreeClick-command.
CurTreeCol "<keystr>"? <formtag>?	Returns the column text or number of the current tree item. In which form is optional and textual by default. Note that the current tree item is merely column-specific in case of the default 'Framed' setting. The optional selection of the tree is analogous to the TreeClick-command.
IsTreeItemOn "<keystr>"? "<treepath>"	On (1) if a specified item in a tree has been selected (i.e. highlighted), off (0) if it has been deselected, and unswitchable (-1) if the tree does not support item selection at all. Specification of the tree and its node is analogous to the TreeClick-command, except that column selection is not an issue here.

IsTreeItemExpanded "<keystr>"? "<treepath>"	True (1) if a specified item in a tree has been expanded, false (0) if it has been collapsed, and unexpandable (-1) if it is a leaf node. Beware that an expanded tree node can have a collapsed ancestor. Specification of the tree and its node is analogous to the TreeClick-command, except that column selection is not an issue here.
GetTreeItem "<keystr>"? <treenodesel> <formtag>?	Returns the item number of a tree node or the text in one of its columns. In which form is optional and textual by default. In the 'Number' case, the GreyOuts-command defines whether disabled items are counted as well. The selection of the tree, its node and column is analogous to the TreeClick-command.
GetTreePath "<keystr>"? "<treepath>" <formtag>?	Returns the path to a selected tree node. In which form is optional and textual by default. In the 'Number' case, the GreyOuts-command defines whether disabled items are counted as well. It can be used for converting a path from one form to another. Specification of the tree and its node is analogous to the TreeClick-command, except that column selection is not an issue here.
GetTreeCol "<keystr>"? <treecolsel> <formtag>? <treecolsel> = <colsel> | **PathCol** "<treepath>" <colsel>	Returns the text or number of a selected tree column. In which form is optional and textual by default. In the 'Number' case, the GreyOuts-command defines whether disabled items are counted as well. Selection of the tree, its column and an optional node is analogous to the TreeClick-command. If the column selection is made by name, the search for a match will start in the column header. If not successful, it proceeds at the specified tree node in the 'PathCol' case, or traverses all tree nodes breadth-first otherwise.
IsTreeButtonOn "<keystr>"? "<treepath>"	On (1) if the button in front of a tree node is checked, and off (0) if it is unchecked. Selection of both the tree and its node is ana-

	logous to the TreeButton-command.
IsTreeMenuItemOn "<keystr>"? <treen-odesel> "<menupath>"	On (1) if the selected menu (sub)item attached to a column of a tree node is checked, off (0) if it is unchecked, and unswitchable (-1) if it is not checkable at all. The selection of the tree, its node and column, and the menu (sub)item is analogous to the TreeMenu-command.
NrTreeMenuItems "<keystr>"? <treen-odesel> "<menupath>?"	Returns the number of (enabled) items in the selected (sub)menu attached to a column of a tree node. The GreyOuts-command defines whether disabled items are counted as well. Selection of the tree, its node and column, and a sub-menu is analogous to the TreeMenu-command. The root menu is denoted by an empty menu-path (""). Zero is returned if the menu-path leads to a leaf menu item.
GetTreeMenuItem "<keystr>"? <treen-odesel> "<menupath>" <formtag>?	Returns the text or number of the selected menu (sub)item attached to a column of a tree node. In which form is optional and 'Text' by default. In the 'Number' case, the GreyOuts-command defines whether disabled items are counted as well. The selection of the tree, its node and column, and the menu (sub)item is analogous to the TreeMenu-command.

Expressional specifications

The logical and mathematical operators to be applied in Command Driver expressions are almost equivalent to those used in OpendTect's mathematics attribute definitions. Only the |x| operator is not provided to keep the parsing of expressions straightforward. Use the abs()-function instead. The AND-operator, OR-operator and conditional operator only evaluate their second (or third) sub-expression if needed for the result. For example, the reciprocal expression (x==0 ? UNDEF : 1/x) will not generate a division-by-zero error. The list of operators is sorted in order of precedence.

Precedence	*Associativity*	*<operatorexpr>*	**VIII. Logical and mathematical operators (to be used in expressions)**
1	right-to-left	! <expr>	Logical not

<table>
<tr><td></td><td>right-to-left</td><td>+ <expr></td><td>Unary plus</td></tr>
<tr><td></td><td>right-to-left</td><td>- <expr></td><td>Unary minus</td></tr>
<tr><td>2</td><td>right-to-left</td><td><expr> ^ <expr></td><td>Raise to power</td></tr>
<tr><td>3</td><td>left-to-right</td><td><expr> * <expr></td><td>Multiplication</td></tr>
<tr><td></td><td>left-to-right</td><td><expr> / <expr></td><td>Real division</td></tr>
<tr><td></td><td>left-to-right</td><td><expr> | <expr></td><td>Integer division</td></tr>
<tr><td></td><td>left-to-right</td><td><expr> % <expr></td><td>Modulo</td></tr>
<tr><td>4</td><td>left-to-right</td><td><expr> + <expr></td><td>Addition</td></tr>
<tr><td></td><td>left-to-right</td><td><expr> - <expr></td><td>Subtraction</td></tr>
<tr><td>5</td><td>left-to-right</td><td><expr> < <expr></td><td>Less than</td></tr>
<tr><td></td><td>left-to-right</td><td><expr> <= <expr></td><td>Less than or equal</td></tr>
<tr><td></td><td>left-to-right</td><td><expr> > <expr></td><td>Greater than</td></tr>
<tr><td></td><td>left-to-right</td><td><expr> >= <expr></td><td>Greater than or equal</td></tr>
<tr><td>6</td><td>left-to-right</td><td><expr> == <expr></td><td>Equality (numerical if possible, string otherwise)</td></tr>
<tr><td></td><td>left-to-right</td><td><expr> != <expr></td><td>Inequality (numerical if possible, string otherwise)</td></tr>
<tr><td>7</td><td>left-to-right</td><td><expr> && <expr></td><td>Logical AND</td></tr>
<tr><td>8</td><td>left-to-right</td><td><expr> || <expr></td><td>Logical OR</td></tr>
<tr><td>9</td><td>right-to-left</td><td><expr> ? <expr> : <expr></td><td>Conditional operator (if-then-else)</td></tr>
</table>

The mathematical and statistical functions to be applied in Command Driver expressions are a superset of those used in OpendTect's mathematics attribute definitions. Many other (string) functions have known equivalents in the C-library of the C++ programming language.

<functioncall>	**IX. Mathematical functions (to be used in expressions)**
abs(<expr>)	Absolute value
acos(<expr>)	Arc cosine
asin(<expr>)	Arc sine
atan(<expr>)	Arc tangent between -PI/2 and PI/2
atan2(<y_expr>, <x_expr>)	Arc tangent of y/x between -PI and PI
ceil(<expr>)	Smallest integer not less than
cos(<expr>)	Cosine
exp(<expr>)	Exponent
floor(<expr>)	Largest integer not greater than

ln(<expr>)	Natural logarithm
log(<expr>)	Base-10 logarithm
rand(<max_expr>?)	Uniform random value between 0 and optional maximum, 1 by default
randG(<stddev_expr>?)	Gaussian random value with mean 0 and optional standard deviation, 1 by default
round(<expr>)	Round to nearest integer
sgn(<expr>)	Sign (1 if greater than zero, 0 if zero, -1 if less than zero)
sin(<expr>)	Sine
sqrt(<expr>)	Square root
tan(<expr>)	Tangent
trunc(<expr>)	Round to integer in direction of zero

<functioncall>	**X. Statistical functions (to be used in expressions)**
avg(<expressions>)	Average
max(<expressions>)	Maximum
min(<expressions>)	Minimum
sum(<expressions>)	Sum
var(<expressions>)	Variance
med(<expressions>)	Median

<functioncall>	**XI. String functions (to be used in expressions)**
curWindow()	Title of current window
isAlNum(<expr>)	True (1) if all characters are alpha-numerical, false (0) otherwise
isAlpha(<expr>)	True (1) if all characters are letters, false (0) otherwise
isDigit(<expr>)	True (1) if all characters are digits, false (0) otherwise
isInteger(<expr>)	True (1) if representing an integer, false (0) otherwise
isLower(<expr>)	True (1) if all characters are lower-case letters, false (0) otherwise
isNumber(<expr>)	True (1) if representing a number, false (0) otherwise

isSpace(<expr>)	True (1) if all characters are white space, false (0) otherwise
isUpper(<expr>)	True (1) if all characters are upper-case letters, false (0) otherwise
strCat(<expressions>)	String concatenation
strLen(<expr>)	String length in characters
strSel(<str_expr>, <firstpos_expr>, <lastpos_expr>?)	Character selection. Last position is optional. Negative positions count in reverse.
sepStrCat(<expressions>)	Concatenation of separation-strings (menu- and tree-paths, RGB(t) color strings)
sepStrLen(<expr>)	Number of separated substrings
sepStrSel(<str_expr>, <firstpos_expr>, <lastpos_expr>?)	Selection of separated substring(s). Last position is optional. Negative positions count in reverse.
toLower(<expr>)	Converts all letters to lower-case
toUpper(<expr>)	Converts all letters to upper-case
wildcard(<selnr_expr>?)	Matching substring for a wildcard in the latest successful wildcarded command action. Default selection number is 1. A negative number counts the wildcards in reverse.
wildcardStr(<selnr_expr>?)	The whole matching string around a wildcard in the latest successful wildcarded command action. Wildcard selection is analogous to the wildcard()-function above.

All predefined identifier constants and environment variables are listed below. The Command Driver will automatically change Unix-style file-paths into Windows-style file-paths on Windows-platforms and vice versa. Redefining any predefined identifier will result in a warning, but is not forbidden. It allows the command script to overrule the values of the directory identifiers ($...DIR$) as set by the system environment in which OpendTect is running. Or you can reset the "increment"-identifier FILEIDX with the start value you like. Beware that most of the predefined identifier constants are unfit for change. They are only there to make command scripts more readable. The Command Driver will keep using the original values internally! For example, you can exchange the values of TRUE and FALSE, but the function call isAlpha("a") still returns 1. Therefore, an expression like isAlpha("a")==TRUE would suddenly get an opposite meaning.

<ident>	*$<ident>$*	**XII. Predefined identifiers (to be**

		used in substitution and expressions)
TRUE FALSE	1 0	Logical constants
PI	3.14159265 ...	Trigonometric constant
UNDEF	1e30	OpendTect's undefined value. Treated as such by all logical, mathematical, and statistical operators and functions defined above.
SUCCESS FAILURE WARNING	1 0 -1	Possible results of 'Try'-command
ON OFF UNSWITCHABLE	1 0 -1	Possible results of any 'Is...On'-question command
BASEDIR		Base data directory (setenv DTECT_[WIN]DATA)
DATADIR	$BASEDIR$/<cur_survey>	Survey directory
PROCDIR	$DATADIR$/Proc	Processing directory
APPLDIR		Installed software directory (setenv DTECT_[WIN]APPL)
USERDIR		Personal home directory (setenv DTECT_PERSONAL_DIR)
SCRIPTSDIR	$PROCDIR$	Overruled by setenv DTECT_SCRIPTS_DIR
SNAPSHOTSDIR	$DATADIR$/Snapshots	Overruled by setenv DTECT_SNAPSHOTS_DIR
IMPORTDIR	$DATADIR$/Import	Overruled by setenv DTECT_IMPORT_DIR
EXPORTDIR	$DATADIR$/Export	Overruled by setenv DTECT_EXPORT_DIR
FILEIDX	1000++	Integer variable that is automatically incremented after every occurrence in the command stream. It may be substituted in file-paths to generate unique filenames.

Repetitive task example

One of the recurring user questions about OpendTect is whether there is a quick way to do some kind of repetitive task. Loading a huge amount of wells, importing multiple 2D line SEG-Y data, etc. Such a service can only be offered by OpendTect itself if the task is very simple, common and straightforward. For example, one does have the possibility to select multiple horizons with the mouse in order to load them in one go. However, if the repetitive task is more complex, unique, or variant, the workflow can be automatized by means of a command script.

The current Command Driver capabilities are demonstrated by automizing a case raised by Magnus Lidgren. His scene contained a big number of 2D lines, and a number of attributes on each line. He needed to generate a picture for every attribute of every 2D line, and wanted to avoid doing that manually.

Automizing a repetitive task consists of three stages. Firstly, the Command Recorder is applied to record the mouse and keyboard actions needed to perform the task once. Secondly, a text editor is used to modify the recorded script. Some of the recorded actions have to be generalized, and a few new commands have to be added to make the script iterative. These commands are listed in Table III. Thirdly, the Command Driver is applied to run the modified script, initially to debug it and finally to perform the repetitive task.

Listed below is the recorded command script that makes a snapshot of one attribute on one 2D line. The passages that need to be generalized have been printed in red.

```
[OpendTect Main Window]

TreeMenu "2D Seismics`LS 5k`i5007`Seis" "Display in a 2D
Viewer as`VD"


[2D Viewer - Line: i5007]

Button "Save image"


[Create snapshot]

Button "Screen" On

Input "Select filename" "/d43/-
jaap/surveys/Demo2D/Snapshots/dump.png" Hold

Button "Ok"


[2D Viewer - Line: i5007]
```

```
Close

[OpendTect Main Window]
```

Listed below is the modified command script after it has been generalized and made iterative. All changes with regard to the originally recorded script have been printed in green.

```
Def res ? dumpAttribute( setnr, namenr, attrnr )
[OpendTect Main Window]
res ~ TreeMenu "2D Seismics`*#$setnr$`*#$namenr$`*#$attrnr$" \
                "Display`2D Viewer - VD"
If res==FAILURE ; Return ; Fi

pic_name = strCat( wildcard(1), "_", wildcard(2), "_" )
pic_name = strCat( pic_name, wildcard(3), ".png" )

[2D Viewer - Line: *]
Button "Save image"

[Create snapshot]
Button "Screen" On
Input "Select filename" "$SNAPSHOTSDIR$/$pic_name$" Hold
Button "Ok"

[2D Viewer - Line: *]
Close
```

```
[OpendTect Main Window]
Fed


For setidx = 1
For nameidx = 1
For attridx = 1
res ? dumpAttribute( setidx, nameidx, attridx )
If !res ; Break ; Fi
Rof
If attridx==1 ; Break ; Fi
Rof
If nameidx==1 ; Break ; Fi
Rof
```

The command script example above demonstrates a number of more advanced Command Driver features:

- Def-inition of procedure(s) to modularize, reuse and/or hide command sequences.
- "Try"-command (res ~ ...) to check whether a command executes.
- Wildcards ('*') to match arbitrary substrings.
- Disambiguators ('#') to select the n-th match out of multiple matches.
- Substitution ($...$) of identifier values into command actions.
- Backslash ('\') to spread a long command over multiple lines.
- If-command to execute command actions conditionally.
- Use of predefined identifiers: constant FAILURE and environment variable SNAPSHOTSDIR.
- Semicolons (';') to separate multiple commands on one line.
- Return-command to terminate a procedure immediately.

- Use of assignment and built-in function calls (pic_name = strCat(...,...)).
- Wildcard(.)-function to get matching strings from the latest successful wildcarded command action. Note that these strings must be secured before the next use of a wildcard (i.e. in window assertion [2D Viewer - Line: *]).
- (For)-loops to make a script iterative.
- Call to a user-defined procedure (res ? dumpAttribute(...,...))
- Break-command to escape from loops immediately.

Note that different operators are used to store the result of an expression (=), a command action (~), or either a user-defined procedure (?) or a question command (?) into an identifier. This eases the parsing of commands by the Command Driver, and it should make the user aware that the allowed complexity of (sub)-expressions does not go beyond the built-in function calls. The results of command actions, user-defined procedure calls, and question commands have to be assigned to auxiliary identifiers first.

The introduction of question commands allows the body of the previous script to be written in a different style. An identifier can take the answer to one of the many questions (Nr..., Cur..., Is..., Get...) from Table IV to VII about some property of a user-interface element. Instead of applying the Break-command to escape from loops when the procedure call to dumpAttribute(...,...) fails, now the number of items to iterate over can be asked and set before entering a loop (nritems ? NrTreeItems ...). Note that a procedure defined with a return parameter can also be called without. Listed below is the restyled body of the command script above.

```
nrsets ? NrTreeItems "2D Seismics"

For setidx = 1 To nrsets


nrnames ? NrTreeItems "2D Seismics`*#$setidx$"

For nameidx = 1 To nrnames


nrattrs ? NrTreeItems "2D Seismics`*#$setidx$`*#$nameidx$"

For attridx = 1 To nrattrs
```

```
dumpAttribute( setidx, nameidx, attridx )
Rof
Rof
Rof
```

Standard test scripts

In the 'doc' directory of the release, you can find a 'Scripts' subdirectory. It contains the standard test scripts for OpendTect. These test scripts all work on a survey 'F3_Demo', the demo data set for OpendTect.

The directory contains several scripts, many of which can be run stand-alone, but certainly not all. There is also an execute-all script: 'AllScripts.cmd'. Another composite script is 'AllAttributes.cmd', which will make snapshots in the Snapshots directory. This Snapshots directory is created automatically; the location is your_ surveys/F3_ Demo/Snapshots. The snapshots are created as an index followed by the file name.

Another script is 'ExportData.cmd'. While running this script an 'Export' directory is automatically created as a subdirectory of 'F3_Demo'. All the exported data will be stored in the above directory.

Some scripts are dependent on PlugIns like SSIS, VMB etc. These scripts are located in dgb/doc/Scripts. To run these scripts you should make sure the related plugins are loaded.

User history recording

The CmdDriver plugin also offers the possibility to record the user action history in the background from the moment OpendTect is started. It will be stored in the file userhistory.odcmdin the Proc directory of the starting survey. It is not guaranteed that the Command Driver can offer a full reproduction of the past by running this file, since not all possible user actions are covered yet. Mouse actions performed in 3D scenes or 2D viewers are recorded nor executed for the time being. Actions performed in the Command Controller window are recorded, but not executed as long as the difficulty of calling the Command Driver recursively has not been solved. Nevertheless, the recorded history can be of great help in reproducing a bug or crash reported by the user.

In order to enable recording of the user history, the user setting dTect.User history buffer must be set with a value other than zero. The magnitude of this value defines the size (in characters) of the buffer in which the user actions are stored temporarily. A positive size value means flushing the content of the buffer to file every time it overflows. A negative size value means dropping the oldest content once the buffer starts overflowing, and flushing only the newest actions to file when OpendTect finally exits or crashes. The menu item Utilities->Settings->Advanced->Personal settings will pop-up a dialog in which this user setting can be added or adapted.

Appendix C. SEG-Y Checklist

This document contains examples of SEG-Y loading problems and proposes solutions for the most often encountered problems.

In all cases, the SEG-Y import tool must be launched from the survey menu (*Import* --> *Seismics* --> *SEG-Y*). Enter the settings needed for the first step and press OK. In the next window (main import window) press scan and perform the scan. An examine window will pop up. Display the first traces in the 2D viewer.

You should have on your screen:

- **The main import window**
- **The examine window**
- **The scan report**
- **The first traces displayed in a 2D viewer**

Once you have this on screen check the most appropriate situation in the list below:

1. The textual header is not readable
2. The line header in not readable
3. None of the trace headers are readable
4. The first trace header is readable but no other - zero sample rate - last trace incomplete
5. The examine/scan reports incorrect coordinates
6. The loaded volume contains wrong amplitudes
7. The loaded volume contains holes - traces were rejected
8. The loaded volume is shifted with respect to the others - does not start at zero

Excel utilities

1. SEG-Y file size computation
2. SEG-Y trace numbers computation

SEG-Y checklist

The textual header is not readable

The top of the examine window may look like any of those two windows, although there is an enormous variety in textual headers:

```
C 1 CLIENT             COMPANY                      CREW NO
C 2 LINE      AREA                     MAP ID
C 3 REEL NO      DAY-START OF REEL       YEAR     OBSERVER
C 4 INSTRUMENT: MFG         MODEL           SERIAL NO
C 5 DATA COPY108179RLENGT1081794XILIARY TRACES/RECORD       CDP FOLD
C 6 SAMPLE INTERNAL     SAMPLES/TRACE     BITS/IN     BYTES/SAMPLE
C 7 RECORDING FORMAT    FORMAT THIS REEL       MEASUREMENT SYSTEM
C 8 SAMPLE CODE: FLOATING PT   FIXED PT   FIXED PT-GAIN   CORRELATED
C 9 GAIN TYPE: FIXED  BINARY  FLOATING POINT  OTHER
C10 FILTERS: ALIAS  HZ NOTCH  HZ BAND  -  HZ SLOPE  -  DB/OCT
C11 SOURCE: TYPE         NUMBER/POINT      POINT INTERVAL
C12   PATTERN:                LENGTH     WIDTH
C13 SWEEP: START   HZ END   HZ LENGTH  MS CHANNEL NO    TYPE
C14 TAPER: START LENGTH   MS END LENGTH   MS TYPE
C15 SPREAD: OFFSET   MAX DISTANCE   GROUP INTERVAL
C16 GEOPHONES: PER GROUP  SPACING  FREQUENCY  MFG       MODEL
C17   PATTERN:                 LENGTH   WIDTH
C18 TRACES SORTED BY: RECORD  CDP  OTHER
C19 AMPLITUDE RECOVERY: NONE  SPHERICAL DIV   AGC   OTHER
C20 MAP PROJECTION             ZONE ID    COORDINATE UNITS
C21 PROCESSING:
C22 PROCESSING:
C23
C24
C25
C26
C27
C28
C29
C30
C31
C32
C33
C34
C35
C36
C37
C38
C39
C40 END EBCDIC
```

```
C01  Created by:  (Tue Apr 07 16:03:31 2009)
C02  F3 Demo
C03
C04
C05  Byte positions (in addition to REV. 1 standard positions):
C06  X-coordinate: 73
C07  Y-coordinate: 77
C08  In-line:       9 (4-byte int)
C09  X-line:        21 (4-byte int)
C10
C11
C12  100/300 = (605835.5,6073556.5)
C13  100/1250 = (629576.25,6074220)
C14  750/300 = (605381.8125,6089799.5)
C15
C16
```

The left picture is a very common empty (automatically filled) textual header. Sometimes an operator fills the empty parts, but you can expect mistakes to occur. The second picture is a textual header from an OpendTect exported SEG-Y file. All the fields were directly copied from the project database (additional edits are possible).

Every SEG-Y file begins with 40 times 80 characters encoded in ASCII, or EBCDIC for old files. The main difference between them is that your favorite text editor (wordpad, vi or text edit) will not be able to translate EBCDIC encoded characters if you open the SEG-Y file with it, while if you open an ASCII SEG-Y textual header you will see the same as in the OpendTect examine window, without the line breaks.

- Problem: If you do not see the above described structure you **might not be reading a SEG-Y file** at all, but maybe a seismic file written in a different format (binary, SEG-D, ...).

  o Solution: **Translate** your file if possible.

- Problem: Some characters in the textual header are **weird**.

  o Solution: They were **badly translated** from EBCDIC to ASCII (or the opposite), either when writing the file or when reading it. Unfortunately there is no 1 to 1 translation between EBCDIC and ASCII, therefore this issue cannot be solved. That is why EBCDIC coding has been banned from the SEG-Y revision 1 norm.

The line header is not readable

The line header overview is provided in the top part of the examine window below the textual header. Use the right scroll bar to reach it. Only the non-zero values are shown, on the contrary to the other headers where the entire content is reported. A correct line header will look like this in the OpendTect examine window:

File header information

```
C40  END TEXTUAL HEADER

......
Binary header info (non-zero values displayed only):

          jobid      1          1          (job identification number)
          hdt        17         4000       (sample interval in micro secs for this reel)
          hns        21         462        (number of samples per trace for this reel)
          format     25         3          (sample format (1=float, 3=16 bit, 8=8-bit))
          tsort      29         4          (trace sorting code)
          mfeet      55         1          (measurement system code (1=m 2=ft))
          isrev1     301        1          ([REV1 only] SEG-Y revision code)
          fixdsz     303        1          ([REV1 only] Fixed trace size?)
```

The title line, field names (first column), byte offsets (second column) and explanations between brackets (fourth column) are provided by OpendTect. Only the numbers in the third column originate from the file. The byte numbers indicate where those values were found.

Look at the field isrev1 (301): If the value is zero your file is revision 0 compliant. If the value is equal to 1 then your file in (normally) revision 1 compliant. This enables to answer to the revision 1 question.

- Problem: If none of the values are reasonable but contain **a lot of 256 and 16777216**, or in general if none of the numbers make sense, the file might be byte-swapped (i.e. written using "little-endian" byte ordering instead of "big-endian"). See example below:

File header information

```
C40  END TEXTUAL HEADER

------
Binary header info (non-zero values displayed only):

          jobid     1      16777216   (job identification number)
          hdt       17     -24561     (sample interval in micro secs for this reel)
          hns       21     -12799     (number of samples per trace for this reel)
          format    25     768        (sample format (1=float, 3=16 bit, 8=8-bit))
          tsort     29     1024       (trace sorting code)
          mfeet     55     256        (measurement system code (1=m 2=ft))
          isrev1    301    1          ([REV1 only] SEG-Y revision code)
          fixdsz    303    256        ([REV1 only] Fixed trace size?)
```

Solution 1: In that case cancel the import window (that will bring you back to the first step), and put the "**Byte swapped**" parameter to "all".

Solution 2:.If this is not enough then the textual and/or line headers might be **corrupted**. Try to re-create your SEG-Y file if possible.

None of the trace headers are readable

- Problem: The line header is **truncated**: The line header should be 400 bytes long. It will not be possible to deduct the start position of the first trace if any of the first 3600 (=3200 + 400) bytes are missing. If the line header is truncated then you should see a lot of non-standard entries in the examine window, like in the example below:

File header information

```
......
Binary header info (non-zero values displayed only):

        jobid       1       1       (job identification number)
        hdt         17      4000    (sample interval (micro secs or mm))
        hns         21      463     (number of samples per trace)
        format      25      1       (sample format (1=float, 3=16 bit, 8=8-bit))
        tsort       29      4       (trace sorting code)
        mfeet       55      1       (measurement system code (1=m 2=ft))
        isrev1      301     1       ([REV1 only] SEG-Y revision code)
        Extra       308     256     (Non-standard - unassigned)
        Extra       310     1       (Non-standard - unassigned)
        Extra       312     43264   (Non-standard - unassigned)
        Extra       322     1       (Non-standard - unassigned)
        Extra       324     12288   (Non-standard - unassigned)
        Extra       330     256     (Non-standard - unassigned)
        Extra       336     256     (Non-standard - unassigned)
        Extra       340     256     (Non-standard - unassigned)
        Extra       370     255     (Non-standard - unassigned)
        Extra       372     62976   (Non-standard - unassigned)
        Extra       374     23660   (Non-standard - unassigned)
        Extra       376     32259   (Non-standard - unassigned)
        Extra       378     40957   (Non-standard - unassigned)
        Extra       380     43008   (Non-standard - unassigned)
```

o Solution: Use the excel utilities to determine if your file may be missing some bytes. If that is the case you need to re-create your file unless you know exactly how many bytes are missing and how to lengthen your line header.

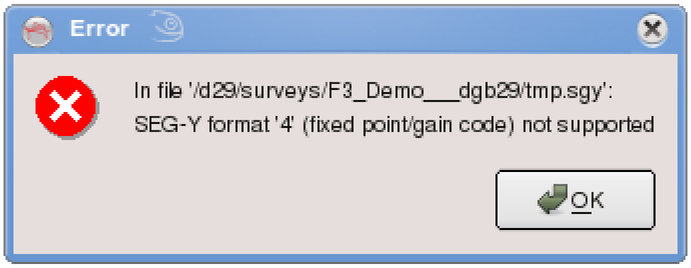
- Problem: The bytes are swapped.

  o Solution: See above.

The first trace header is readable but no other - zero sample rate - last trace incomplete

This happens when the trace size could not be computed successfully. The trace size is function of the sample size (format) and number of samples. The problem then occurs when either of those variables was not correctly written in the headers.

- Action 1: **Check** if the sample size (**format**) is correct in the line header: The sample format is reported in the examine window in the line header in front of the "format" field. It must report a value of 1, 2, 3, 5 or 8. If not the following warning will be shown when scanning the file or loading it:



A value of 1, 2, 5 represents a sample size of 4 bytes.
A value of 3 represents a sample size of 2 bytes.
A value of 8 represents a sample size of 1 byte.

Furthermore the warning "Err: Warning: replacing zero sample rate with survey default" is printed (but that is not the only cause) if the actual sample size is larger than the sample size deducted from the headers of overruled.

On the contrary the warning "last trace incomplete" is printed (but that is not the only cause) if the actual sample size is smaller than the sample size deducted from the headers or overruled.

  o Solution: **Overrule** the SEG-Y **format** to another format until the line header and all trace headers are readable in the examine window. Scan your file and check the output amplitudes since three different formats are available for a sample size of 4 bytes. A proper display of the data in the 2D viewer will indicate a success.

- Action 2: **Check** if the **number of samples** was correctly extracted from either the line or trace headers: The number of samples is reported at multiple positions in the SEG-Y file:
  In the examine window in the line header overview in front of the "hns" field.
  In each trace header at byte offset 115 (field "ns").
  OpendTect will only use the number of samples defined in the trace headers. Therefore the line header field "hns" might be missing or in contradiction with the trace headers, with no consequence upon the data loading as long as the trace headers "ns" field are correctly written.

  o Solution: **Overrule** the SEG-Y **number of samples** to another value until all trace headers are readable. The excel utilities might be helpful to check the correct value and to check if the file is not missing some bytes.

The examine/scan reports incorrect coordinates

The coordinates may be found in each trace headers on bytes 73 (CDP-X) and 77 (CDP-Y) or 181 and 185 respectively with SEG-Y revision 1 files. However those values will be scaled during scan and loading by the scaler coordinate that should always be found on bytes 71-72.

- Problem: The coordinates found in the trace headers do not make sense: Your file might contain coordinate trace header encoded as floats instead of integers. This is not allowed by any SEG-Y standard, although still being encountered sometimes.

  o Solution: Get a SEG-Y compliant file.

- Problem: The unscaled coordinates are reasonable but after scan/-loading the scaled coordinates are not correct.

  o Solution: Overrule the scaler coordinate to the right value in the import window. You need to specify a number that when multiplied by the trace header coordinate will return the actual (scaled) coordinate. Therefore the trace header scalco "-10" would have to be overruled by 0.1 in order to get the same scaling.

Please note the it is not possible to apply a static shift (easting/northing) to the coordinates, however this is not needed by OpendTect. Nevertheless you can load the SEG-Y file first and apply your shift to the survey coordinates afterwards in the survey definition window.

The loaded volume contains wrong amplitudes
This is linked to the **sample format** not being correctly set. The three formats 1, 2, 5 code the data on four bytes. You may need to switch and **overrule** between 1, 2 and 5.

The loaded volume contains holes - traces were rejected
Neither a SEG-Y file nor and OpendTect volume need to be rectangular, i.e. they do not need to contain all traces of rectangular survey. This is normal, except if you expect a rectangular volume. Please note that the default setting in OpendTect is to dismiss null traces, i.e. traces where all samples have a zero value.

- Problem: The warning "during import 123450 traces were rejected" appers when loading the file.

  o Solution 1: Display the loaded file. The warning sometimes pops up by mistakes such that your file may be already correctly loaded.

  o Solution 2: Make sure that the survey area is large enough to accommodate your new volume.

- In general: .

  o Solution: Use the excel utilities to compute the number of traces you can expect to have in your SEG-Y file, and compare it with the actual number of loaded traces, reported in the scan report. OpendTect will be able to load all full traces until the first missing byte is found in the input file or until the end of the file, even if the end of the file is in the middle of a trace. In that case only the last non-complete trace will not be acccessible.

The loaded volume is shifted with respect to the others - does not start at zero
Sometimes the first sample does not correspond to the time or depth 0. If that is the case the corresponding time or depth should be reported in each SEG-Y trace header at bytes 109-110 (delrt) and/or 105-106 (laga) with the opposite polarity like in this example:

Trace header information

	1	2	3	4	5	6
99 [sstat]	0	0	0	0	0	0
101 [gstat]	0	0	0	0	0	0
103 [tstat]	0	0	0	0	0	0
105 [laga]	-200	-200	-200	-200	-200	-200
107 [lagb]	0	0	0	0	0	0
109 [delrt]	200	200	200	200	200	200
111 [muts]	0	0	0	0	0	0
113 [mute]	0	0	0	0	0	0
115 [ns]	463	463	463	463	463	463
117 [dt]	4000	4000	4000	4000	4000	4000
119 [gain]	0	0	0	0	0	0

"laga" is equal to -200, delrt is equal to 200: both mean that the first sample corresponds to time 200ms.

- Problem : The start time is not specified in the trace headers but is expected to be different that zero.

  o Solution: Overrule the start time parameter in the import window.

- Problem: An incorrect start time was applied during loading.

o Solution: Either re-import the file by overruling the start time or use the reference shift attribute to apply a static shift to your traces.

Please note that in all cases you must have a priori knowledge of the start time.

Excel utilities

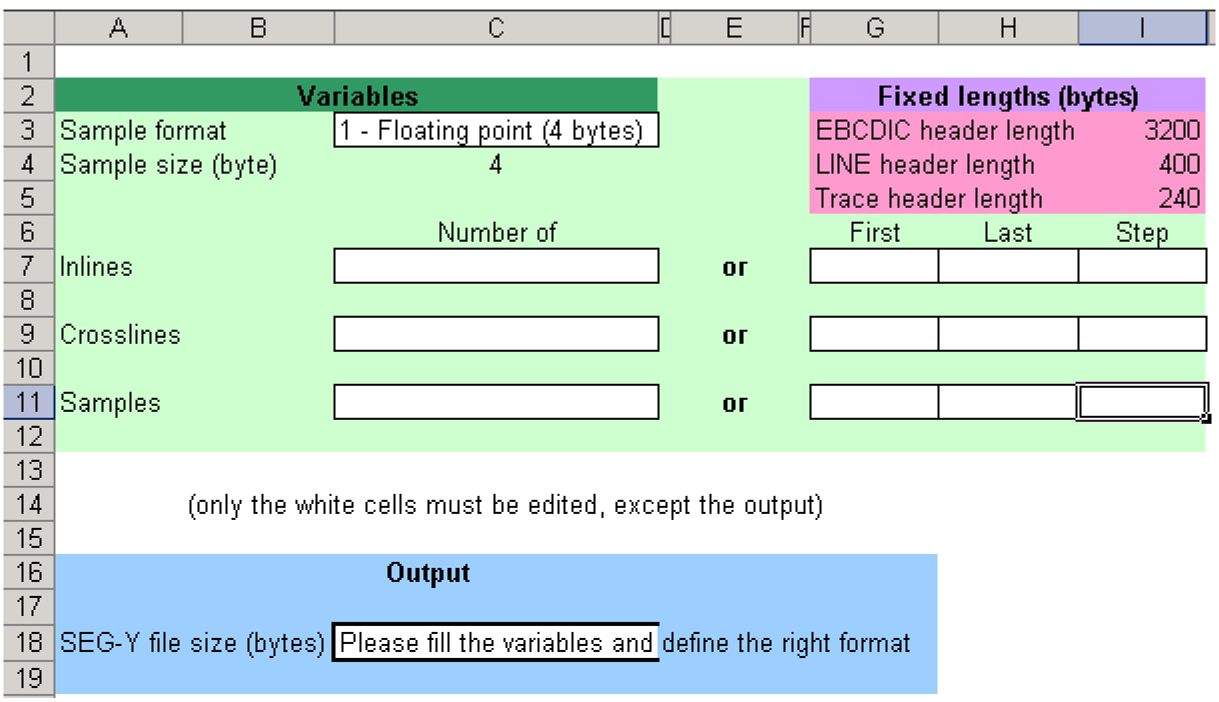
The following two utilities can be used to:

- Compute the SEG-Y file size based on the sample format, number of traces and number of samples. A successful application will be the indicator of a SEG-Y file without missing bytes.
- Compute the number of traces present in the SEG-Y file based on its size, the sample format and the number of samples per trace, assuming a constant trace length. If the returned number os an integer the SEG-Y file does not contain holes (except for a very unlikely occasion).

How-to use them: Double-click on the pictures below, open the file and fill the variables. The sample format must be specified using the list box.

Please note that extended textual headers are not supported.

SEG-Y file size computation



SEG-Y trace numbers computation

	A	B	C	D	E	F	G	H	I
1									
2	**Variables**						**Fixed lengths (bytes)**		
3	Sample format		1 - Floating point (4 bytes)				EBCDIC header length		3200
4	Sample size (byte)		4				LINE header length		400
5							Trace header length		240
6									
7	Filesize (bytes)								
8									
9			Number of				First	Last	Step
10	Samples				**or**				
11									
12	Inline / crossline (opt)				**or**				
13									
14									
15		(only the white cells must be edited, except the output)							
16									
17			**Output**						
18									
19	Number of traces		Please fill the variables and define the right format						
20	Number of crl/inl		Please fill the optional variables						
21									
22									
23	If you can provide the number of inlines the number of crosslines may be computed								
24	assuming a rectangular volume								

Appendix D. Wacom Digitizing Tablets

OpendTect is the first Seismic Interpretation System to support Wacom Digitizing Tablets. Workflows for horizon tracking, fault interpretation, drawing of polygons & bodies etc. have been adapted to benefit from the superior hand-eye co-ordination offered by the pen/tablet combination device.

Table of contents

- [Manually Edit Horizons]
- [Interpret Faults]
- [Supported Platforms]

Introduction
OpendTect combined with a Wacom Tablet has become a key application for hand-eye coordination. It has proven to be vital in manual interpretations with OpendTect. Many of the OpendTect interpretation- workflows were modified in version 4.2 for optional use of a tablet device. This documentation is thus written to provide a brief introduction on seismic interpretation on a tablet. The documentation assumes basic familiarity with the OpendTect environment.

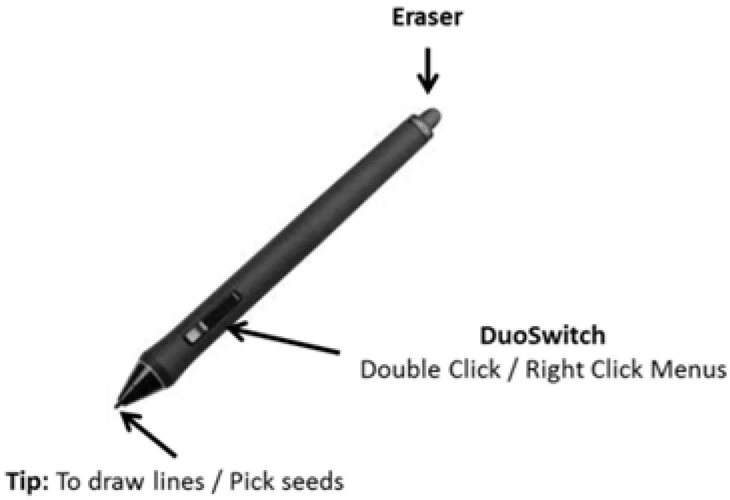
Several aspects of 3D interpretation will be covered:

- How to use the Wacom Pen device instead of a conventional mouse
- How to draw interpretations in OpendTect with a Wacom tablet
- Polygon/Picksets
- Horizon Interpretation
- Fault Interpretation

Before you start using the documentation, a general introduction will be given on the pen device and the features which are supported by OpendTect.

Pen Device
The following list describes how the pen device replaces traditional mouse features:

- *Mouse clicks* are replaced with the clicks being made by the tip of the pen device
- The mouse *drag* function is replicated by dragging the tip of the pen on the tablet device
- The right/left/double click buttons are supported via the *DuoSwitch*



The only thing required is to hold the pen device in your hand and begin interpreting the seismic data. If you want to draw a seismic object e.g. a horizon/polygon, you will simply need to drag the tip with a *light* pressure on the tablet. Light pressure here simply means that you touch the tablet screen using the tip of the pen. The tablet is pressure sensitive and will automatically detect that the user is intending to draw on screen. It will convert the screen coordinates back to the survey coordinates and will store the object.

Furthermore, the *Eraser* can be used to remove (parts of) the interpretation (specifically, the seeds). The eraser feature is accessed either by clicking on the node/seed to remove it, alternatively by rubbing it over the interpretation (drawn lines). The *DuoSwitch* is used to launch pop-up menus or for double clicking on an element. The single left mouse-button click is accessed through the tip of the pen (by 'tapping' on the screen).

Precautions:

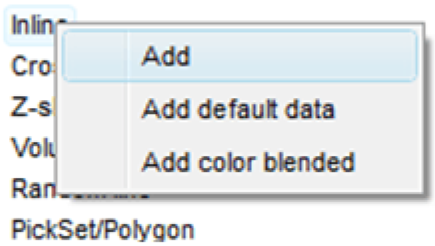
- When the pen is not in use, please place it back in the pen stand
- Please avoid using the mouse while the pen device is in your hand.
  You may either use a pen supported by the Wacom tablet or regular installed mouse

Basic Interaction

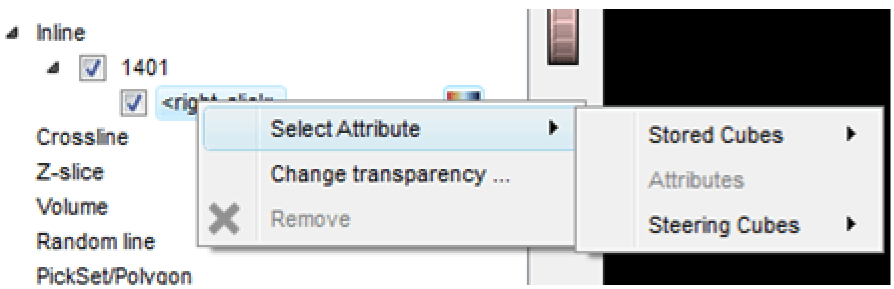
Menus/Icons:
The menus/icons are *clicked* using the tip of the pen device. The tip is tapped over the (sub) menu/icon to launch the corresponding dialog/application.

Display an Element:

To display an element in a scene, you will need to use the *Tree* of OpendTect. For instance, to display an *Inline*, simply place the tip of the pen at inline and select the *Add* option.



Pop-Up Menus:
Pop-up menus are launched using the lower button of the *DuoSwitch*. The pop-up menu options are *selected* by tapping the item with the tip of the pen.



Lines/Seeds/Points:
The lines, seeds or points (Picksets) in OpendTect are drawn by dragging the tip of the pen over the displayed element in the 3D scene, similarly to drawing with a pen on regular paper. The drawn line/point data can be removed interactively using the *Eraser* at the end of the pen.

Draw Polygons
The first example in this tutorial is creating a simple polygon on a horizon on the tablet device. Although the workflow is simple, several options and features are introduced so that you will familiarize yourself with other important features simultaneously. Please follow the steps set out below:

Note: In the following workflow, use the pen device instead of mouse. Tap/Press/Select in this workflow refers to the tip of the pen device.

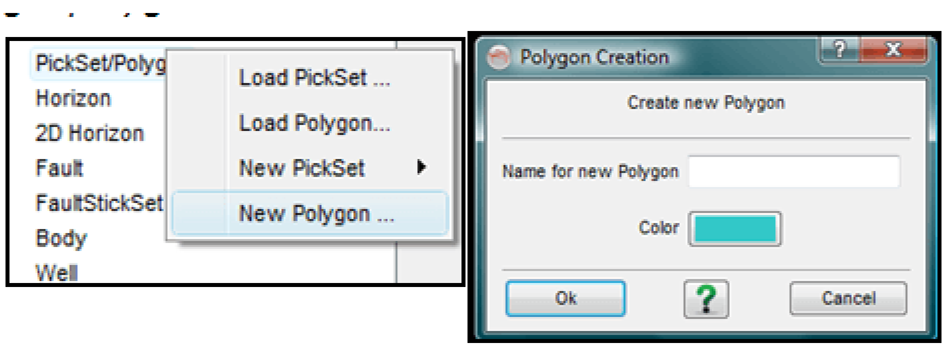
1. Hold the pen device in your hand

2. Drawing a polygon on a seismic horizon:

- o First, display a seismic horizon. Tap the tip of the pen on the Horizon element in the tree; a pop-up sub-menu appears
- o Select *Load*
- o The Horizon Selection dialog is then launched from where you can select one or more saved horizons
- o Select a horizon by tapping on it, or multiple horizons by tapping and dragging
- o Press Ok to display the selected horizon(s)
- o Once a seismic horizon on which you want to draw a polygon is displayed, you can continue to the next steps; drawing a polygon

3. From the tree, tap on the Pickset/Polygon element

4. From the pop-up menu select New Polygon

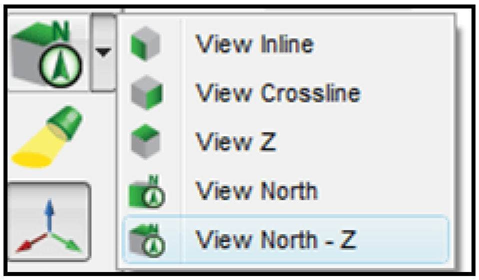


5. The Polygon Creation dialog box is opened, type a name for the new polygon. For typing, you may use either the conventional keyboard attached to your computer or the virtual keyboard supported in OpendTect. The virtual keyboard is launched using the lower DuoSwitch button (equivalent to the right-click button on a conventional mouse) whilst pointing the pen at the name field. Once done typing, simply close the virtual keyboard: the new polygon name is automatically inserted in the field

6. Hit Ok in the parent Polygon Creation dialog

7. A blank polygon is added in the tree, with the given name (step 5)

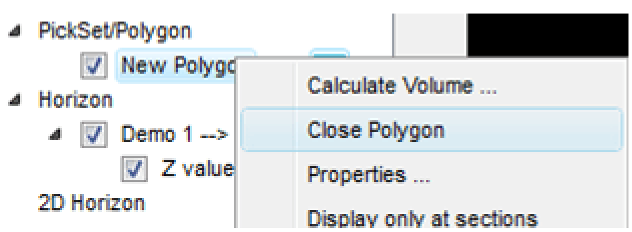
8. Optional: before drawing on the horizon, change to map-view by selecting View North-Z in the main toolbar to the left



9. Make sure the polygon element in the tree is active (tap it) and start drawing the polygon on the horizon. Two methods are available: (1) drag and release, in which you will have to drag the pen over the area where you want to draw a polygon or (2) tap pen, where you tap on the horizon to insert seeds, the seeds are then connected automatically

10. Whilst drawing the polygon, unwanted points can be removed with the eraser at the end of the pen

11. Finally, close the polygon: Right click (lower button on the DuoSwitch) on the polygon name in the tree and select Close Polygon option from the pop-up menu.



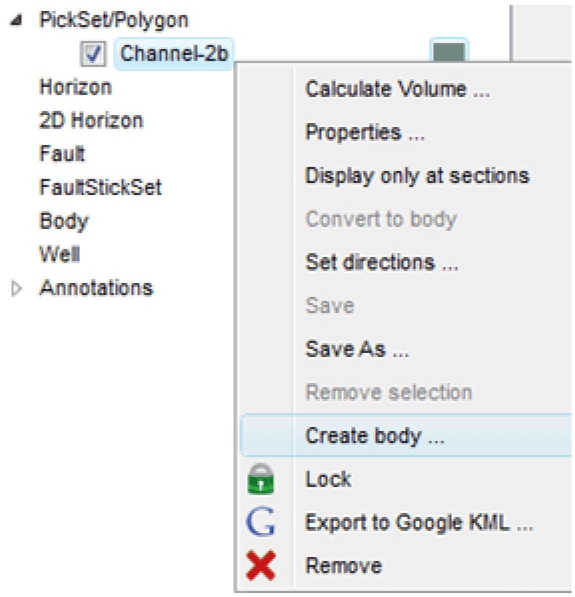
Create Bodies

Bodies are easily created after drawing a polygon. The workflow requires a combination of a saved polygon and top & bottom horizons between which the body will be created.

1. Display a stored (*Load*) or draw (*New*) a polygon by following

the workflow above

2. From the polygon pop-up menu (use the lower DuoSwitch), select *Create body*



3. In this dialog, select top and bottom horizons

4. Hit the *Ok* button

5. The above step (4) creates the body with an empty name and displays it in the scene. In the tree, the <New MCBody 1> sub element under the *Body* menu appears. Save it by launching the pop-up menu for the body by using the lower DuoSwitch

Interpret Horizons

Seismic horizon interpretation is fast and convenient on the Wacom tablet device. This is mainly due to the human adaptability with the pen. To help you get started with interpreting seismic horizons with OpendTect on the tablet device, simply grab your pen and follow the workflow set out below.

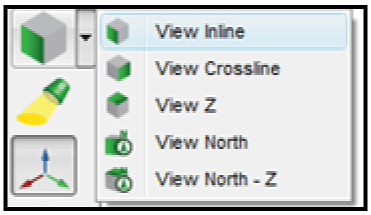
Note: Please use the pen device instead of mouse in the following workflow. Tap/Press/Select all refer to the tip of the pen device.

Setup an orthogonal Display:

1. A flat (orthogonal) display has proven useful when interpreting horizons on seismic data. This view enables you to view the inlines/crosslines/z-slices as '2D' planes. Simply switch the default perspective view to orthographic:

 Use the pen device to tap the button to toggle the *ortho-graphic* view on/off.

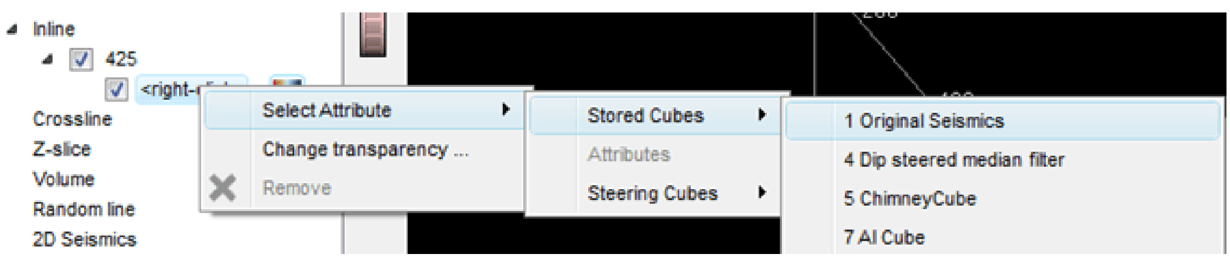
2. If you wish to interpret horizons on inlines, you may want to select the *View Inline* display option from the *Graphical Toolbar*.



3. *Optional: You may also adjust the zoom of the display using the touch strips available on the back side of the table.*

Display seismic data:

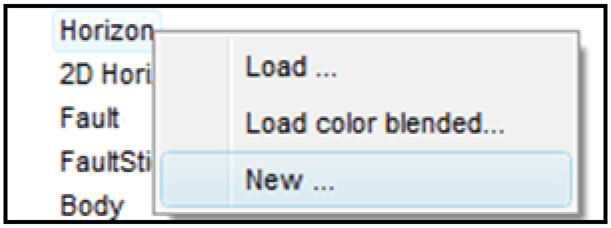
4. Display seismic data in the scene by tapping either *Inline* or *Crossline* in the tree. By default data is being displayed as a white colored 'empty' element in the centre of the survey. Select data by tapping the <right-click> sub-element. Use the lower DuoSwitch to access the sub-element and select seismic data to be displayed (as shown below).



o Once the seismic data is loaded, you can proceed to the next step

Add a New Horizon:

5. Use the pen tip to add a new horizon in the tree. Tap the *Hori-zon* element in the tree using the pen. A drop-down list appears; select *New*...



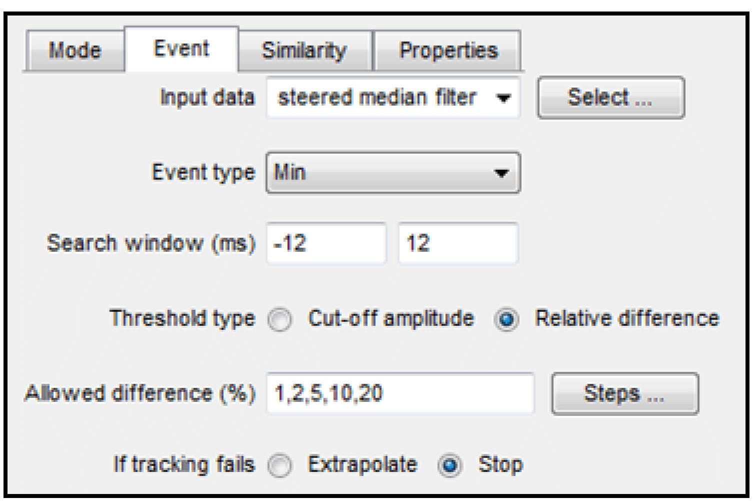
6. A new horizon is added in the tree, labeled <New Horizon 1> by default. A pop-up dialog also appears (i.e. the *Tracking Setup*).

Pick seeds and Auto-track:

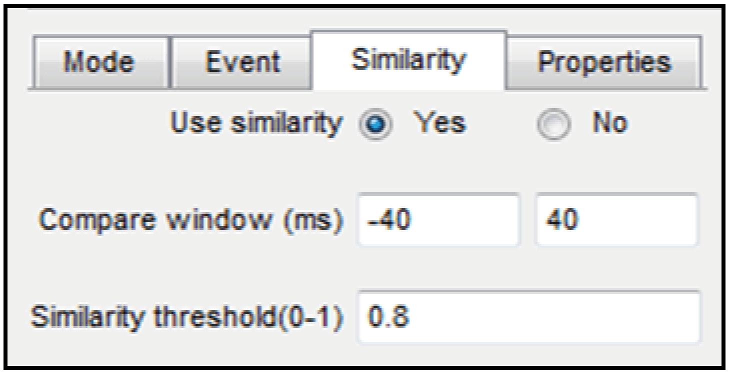
7. The *tracking setup* window features four tabs, presented briefly below. Please refer to the general help documentation of OpendTect for detailed descriptions.

o **Mode:** Select tracking mode i.e. *auto-tracking, tracking on a line,* or *tracking manually*.

o **Event:** Used if the mode is set to either *auto-tracking* or *line tracking*. Select seismic data in the *Input data* field and provide the *event type* (Peak/Trough/Zero-crossing). Please, note that Max refers to peak, *Min* refers to trough, 0+/- refers to positive to negative zero crossing, and 0-/+ refers to negative to positive zero crossing. Use the default search window as a starting point. These settings can be accessed and changed later. Also, use the default step-wise-tracking options i.e. *Relative* (amplitude) difference of 1, 2, 5, 10, 20 â€¦%. Leave the other settings to default.



o **Similarity:** Matches the seismic events based on the picked seeds and searches for the corresponding signal in the immediate vicinity. Please, leave this blank for now. [Tip: For fast tracking in a good quality area, reduce the threshold (e.g. 0.5), and the time gate (e.g. -16 and +16).



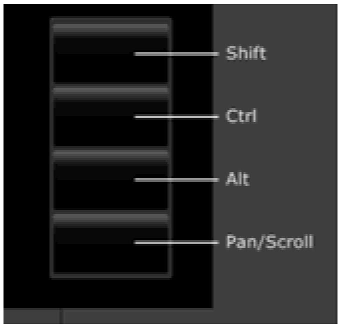
o **Properties:** Change horizon colour and the seed properties (e.g. size, colour and shape).

8. By default the seed mode is toggled on in the OpendTect tracking toolbar (shown below). Note that volume tracking is set on by default (i.e. tracking in a small sub volume)

9. Next, start interpreting by picking a seismic event using the pen device. [**Tip:** Drag the pen over a seismic event if the area is coherent and of good quality, otherwise use the pen clicks on the event to drop seeds]

10. Move the inline position and continue interpreting over the entire survey.

11. In order to remove a seed, simply use the *Eraser* on the end of the pen. Optionally, use the ExpressKeys on the tablet device (subject to availability).



o Pen tap on a plane = pick and local track

o Pen tap with Ctrl + Shift express keys pressed = drop/undrop a seed at a pen location

o Pen tap with Ctrl express key = remove the seed and track locally

o Pen tap with Shift express key = remove the seed and erase auto-tracking from that seed until the next seed(s)

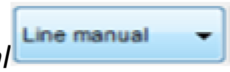
12. After interpreting a good part of the survey area, test the *auto-track* settings by specifying a small auto-tracking area. Select the *show tracking area*  button in the tracking toolbar.

13. A 3D boundary box appears around the interpreted horizon. Use the green anchors at the corners of the box to *re-size* or *move it*: place the pen tip at one of the anchors and drag in the respective directions (diagonal anchors resize the entire box in 3D with equal proportion, whilst the others stretch/squeeze the box horizontally/vertically). **[Tip:** *toggle view* mode to *perspective* to see the whole box.**]**

14. Once the tracking area is defined, tap anywhere in an empty area in the scene to read/load seismic data within the specified tracking volume. **[Tip:** Preferably, tap outside the survey area with a zoomed-out view.**]**

15. Press the *auto-track* button  to start auto-tracking the horizon within the tracking area. **[Tip:** It is recommended to save the raw interpretation as a separate file before attempting to auto-track.**]**

Manual Tracking:

16. Manual tracking is accessed by changing the *tracking* setup to *Line manual*  in the tracking toolbar drop-down list.

17. With *Line manual* selected, you will just need to draw horizons using the pen **[Tip:** Drag the pen over the event and release once done.**]**

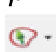
18. Move about 5 or 10 inlines (or cross-lines) forward/backward and repeat manual interpretation. **[Tip:** Instead of interpreting the data in 3D, you may interpret the data in a 2D viewer**]**

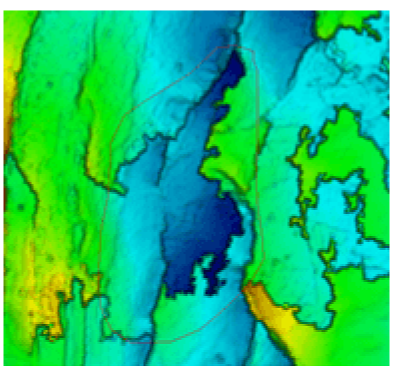
Save Horizon(s):

Click on the save button  that is present in the tracking toolbar to *Save* the <New Horizon 1>>, alternatively right click the horizon in the tree menu and choose either *Save* or *Save as...*

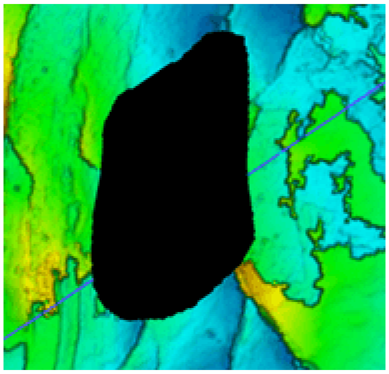
Save Session:
Save the session to continue interpreting the horizon at a later time if not completed

Manually Edit Horizons
This is an important exercise where you will learn how to *edit* a Horizon in OpendTect with your tablet device.

1. In *map view*, draw a polygonal area using the polygon selection tool  (from the Graphical Toolbar to the left).

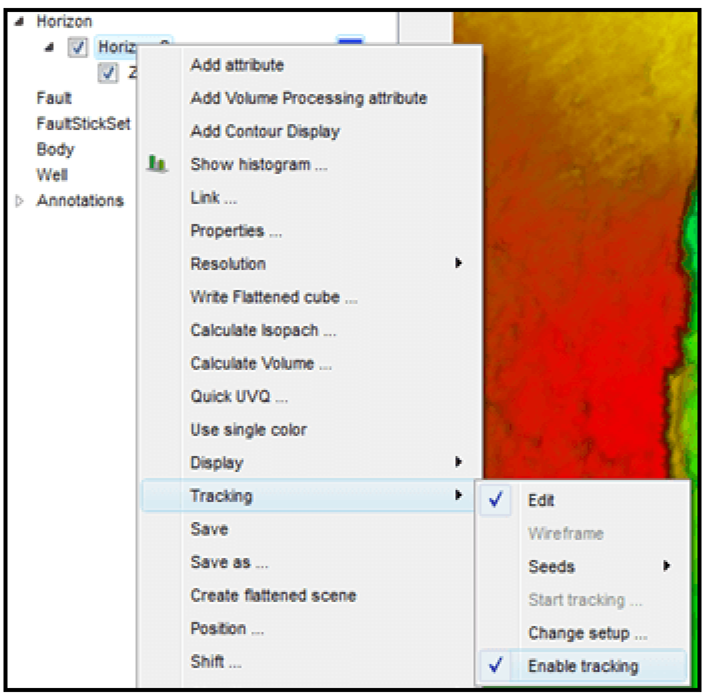


2. Click the horizon from the tree (to activate it) and press the trash icon  to remove the outlined portion of the horizon

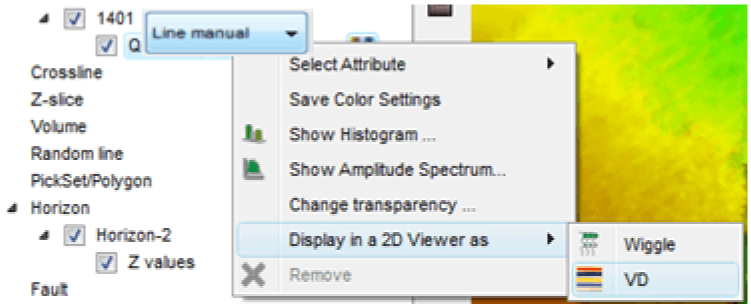


3. Now switch to inline/cross-line view in the area of the removed polygon

4. Enable the tracking mode; use the DuoSwitch to launch the use the pop-up menu of the horizon and select the *Enable tracking* option under the Tracking sub-menu. This will enable the tracking controls and now you can either edit the horizon in the 3D scene or in a 2D Viewer



5. From the tracking controls available at the bottom of OpendTect by default, set *setup* to *Line manual*

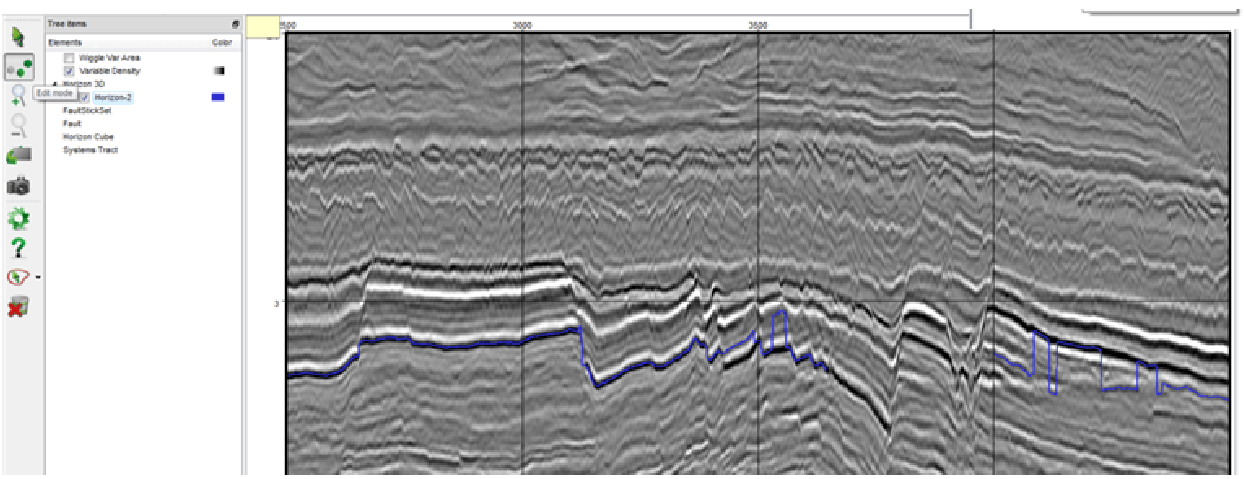


6. In this exercise, we will edit the horizon in a 2D viewer:

7. Display the inline in a 2D Viewer (use the lower DuoSwitch as shown above)

8. In the 2D viewer, you will have to display the same horizon

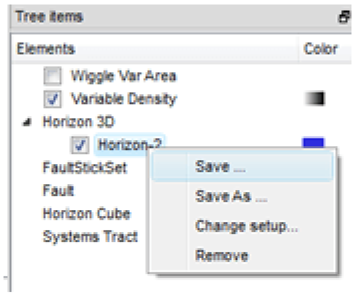
9. Toggle *Edit mode* ON (or as shown below)



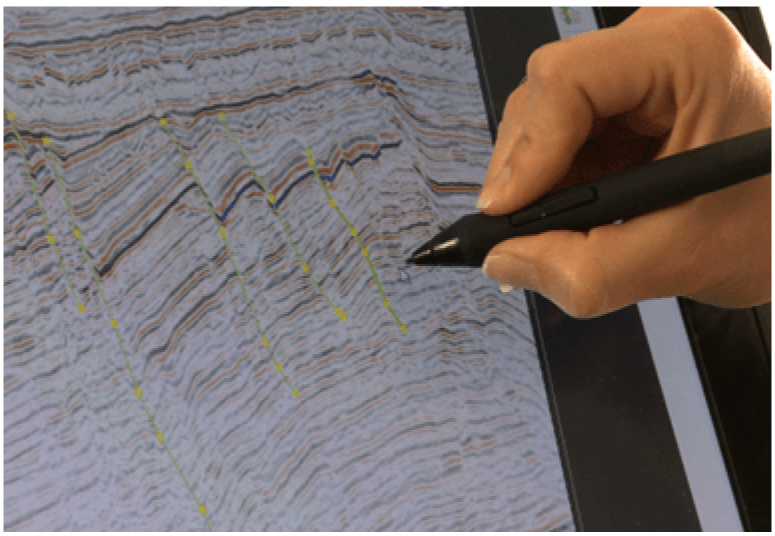
10. Now, start editing the horizon by drawing over an event using the pen

11. Move the inline/cross-line to next/previous 10 lines, and repeat the interpretation

12. Save the horizon after editing. This can be done directly from the tree available in the 2D Viewer



Interpret Faults



Faults in OpendTect can be interpreted as a FaultStickSets or directly as fault planes. The FaultStickSet (in the tree) is a set of sticks that can be converted to 3D fault planes. The FaultStickSet element can be used for a 2D line or for a 3D volume (inline/cross-line/timeslice). Contrary to the FaultStickSet, the Fault element in the tree is generally used to interpret a single 3D fault plane in a 3D survey, however FaultStickSets can be converted and merged to new or existing single or grouped faults at any time (and vice versa).

In this manual, you will learn the use of both mentioned methods of interpreting faults in a 3D survey.

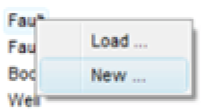
Note: In the following workflow, use the pen device instead of mouse. Tap/Press/Select in this workflow refers to the tip of the pen device.

Basic Fault Interpretation for 3D Seismic Data:

1. Display an inline (or a cross-line) in the scene. [Make sure that

the seismic data has already been displayed along the displayed inline]. **Optional:** Position the inline to a location at where you want to start the 3D fault interpretation. To position, you may use the slice position controls

2. Click on the *Fault* element to add a *New Fault* sub-element in the tree. Next, make sure that it is selected / active



3. Now in the scene, start drawing the fault stick on the inline. [Drag the pen over the inline at fault plane location]

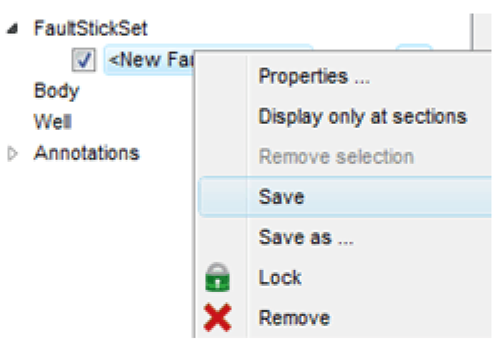
4. To remove a seed of a fault plane, you may use the *Eraser* of the pen

5. To position a seed to a new location, you may move the seed by clicking and dragging it in any direction

6. Move (step) the inline to the next position to interpret another stick of the fault in a new location (5 or 10 inlines forward or backward or smaller steps if continuation is unclear). **[Tip:** Display the fault plane on sections online. For this use may use the lower button of the DuoSwitch to launch the pop-up menu. In the pop-up menu please select the Display option**]**

7. Repeat the steps to interpret the fault on other inline/cross-lines

8. Use the lower button of DuoSwitch to *Save* the <New Fault 1>



Interactive Fault Interpretation using Auto-scroll method:

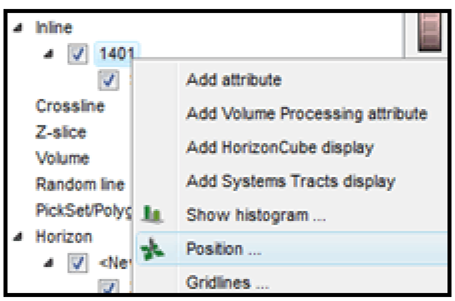
This is an alternative workflow for quick 3D fault interpretation.

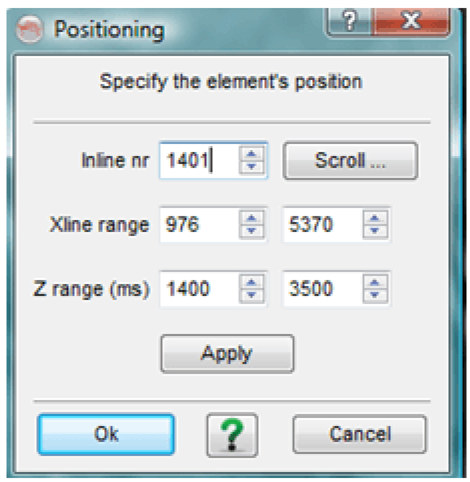
1. Preload the seismic data on which you are intended to interpret faults. **[Tip:** Use the Survey menu i.e. Survey > Preload > Seismics**]**

2. Display an inline (or a crossline) in the scene. [Make sure that the seismic data has already been displayed along the displayed inline]

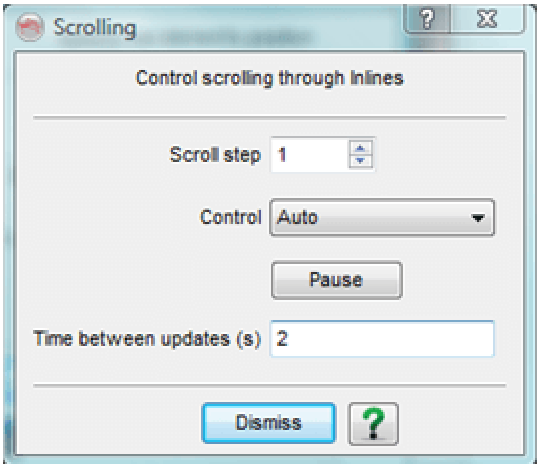
3. Click on the *Fault* element to add a New Fault sub-element in the tree. This will add a <New Fault 1> fault under *Fault* element. [Make sure that it is clicked / active]

4. Use the lower button of DuoSwitch on the Inline to launch the drop-down list and select *Position*... In the *Positioning* dialog, please click on the Scroll button





5. In the scrolling dialog, set the *scroll* step (i.e. number of inlines/cross-lines to move, use positive number for forward *scrolling* and negative number for backward scrolling) and *time* to scroll the inline to the next position (use for example 5 seconds)



6. Click on the to activate it

7. Now in the scene, start drawing the fault stick on the inline. Drag the pen over the inline at where you observe the fault. Once you have interpreted a fault stick on the current inline, wait for a few seconds for the inline to be moved forward/backward automatically

8. Continue the fault interpretation on all lines (step p-q)

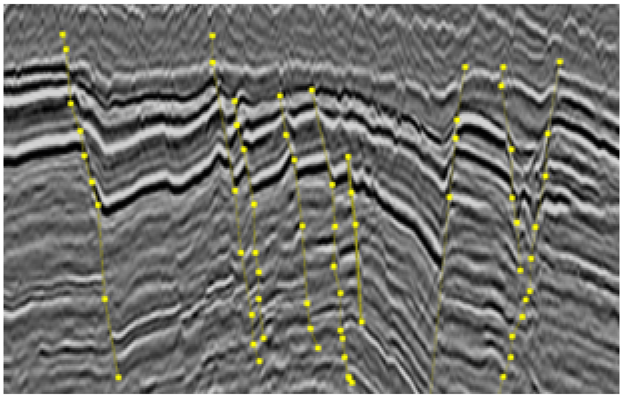
9. Use the lower button of DuoSwitch to *Save* the <New Fault 1>

Fault Sticks Interpretation for 3D Seismic Data:
This workflow allows for interpreting fault sticks only, which can later be converted to 3D fault planes. The benefit of this workflow is that you can interpret multiple sticks on an inline/crossline.

1. Display an inline/crossline in the scene. **[Tip:** Click on the Inline element to add a new inline**]**

2. Add a new FaultStickSet in the tree. [Click on the FaultStickSet, and select the *Add* option in the pop-up menu.]

3. Start drawing multiple fault sticks in the scene. To split sticks, use lower DuoSwitch



4. If you want to move a node of a fault stick, place the tip of the pen over the node to be modified. Click and drag the node in 3D and position it to a correct location

5. Save the FaultStickSet by launching the drop-down list **[Tip:** Use the lower DuoSwitch button.**]**

6. Move the inline/crossline to the next position and continue the interpretation

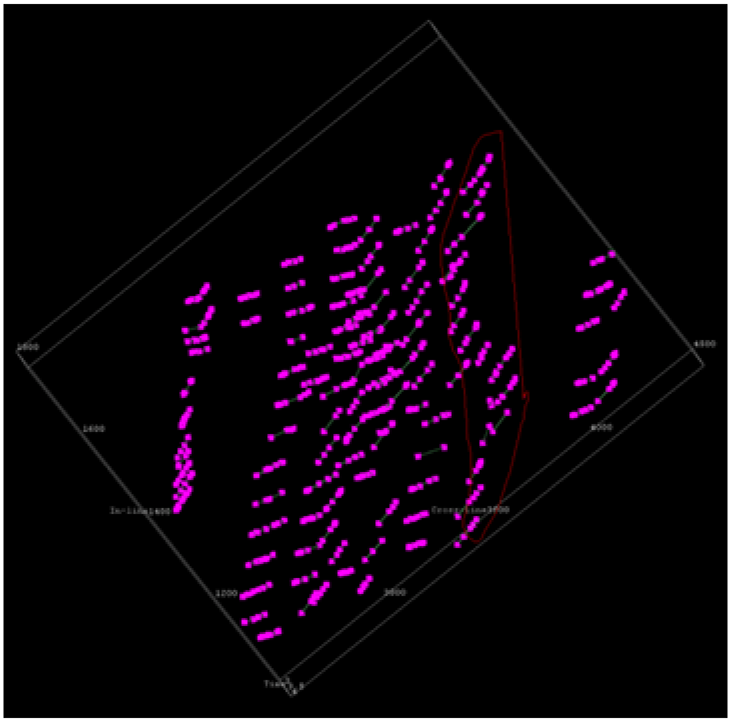
7. While moving the inline/crossline, you may still observe the sticks from previously interpreted sections. To hide them (and therefore avoid confusion), please display the fault sticks at sections only. **[Tip:** Use the pop-up menu for the fault stick set.**]**

Optional: Optionally combine the FaultStickSet interpretation workflow above with the *auto-scrolling* method.

Convert Fault Sticks to Fault Planes:
To convert fault sticks into fault planes, you will need to familiarize yourself with the Fault Sticks Toolbar (shown below). By default, this toolbar appears at the bottom of the OpendTect window.

1. Display a time slice of similarity attribute, so that you can identify the fault trends

2. Display the fault sticks in a 3D scene. **[Tip:** Use the pop-up menu for the fault stick set.**]**



3. Optional: Position the time slice (step-a) at where you can see the tops of the fault sticks

4. Activate the select sticks button 

5. Make sure that you are in interact mode

6. In the scene, draw a polygon to select the sticks that you want to convert into a fault plane. [Use the pen device and draw a red colored polygon.]

7. Once the tip of the pen is lifted away from the tablet, observe that the sticks within the polygon turn green. This means that the sticks have been selected (and can be converted to a fault plane)

8. Next, *copy* or *move* the selection to a single new fault plane. This option is illustrated in the above fault stick toolbar

9. Give a name in the text field of the toolbar

10. Hit the Go button to save and display the fault plane in the scene and tree

Useful Notes on Fault Sticks Toolbar:

- The 'Copy selection to' option is used to copy the selected fault sticks to a fault plane without remove the fault sticks from the original fault stick set. Contrary to this, the 'Move selection to' option removes the selected fault sticks from the original fault stick set and moves them to a fault plane.

- The Fault/FaultStickSet option is used to convert the selected fault sticks to a fault plane or to another fault stick set.
- There are different ways to name faults/FaultStickSets. This is done via the output operations list box i.e. Create single new (to create a new single fault plane or a fault stick set), Create new in series (automatically labels the faults with a numeric index), Merge with existing (to merge the fault plane to an existing fault plane or fault stick set), and Replace the existing (replace the selected fault plane with the newly selected sticks).
- The trash button in the toolbar is used to remove the selected fault sticks

Supported Platforms

Officially the Wacom Cintiq 21UX and Wacom Cintiq 24HD are only supported on Windows and Mac.

However it is possible to get it to work on Linux.

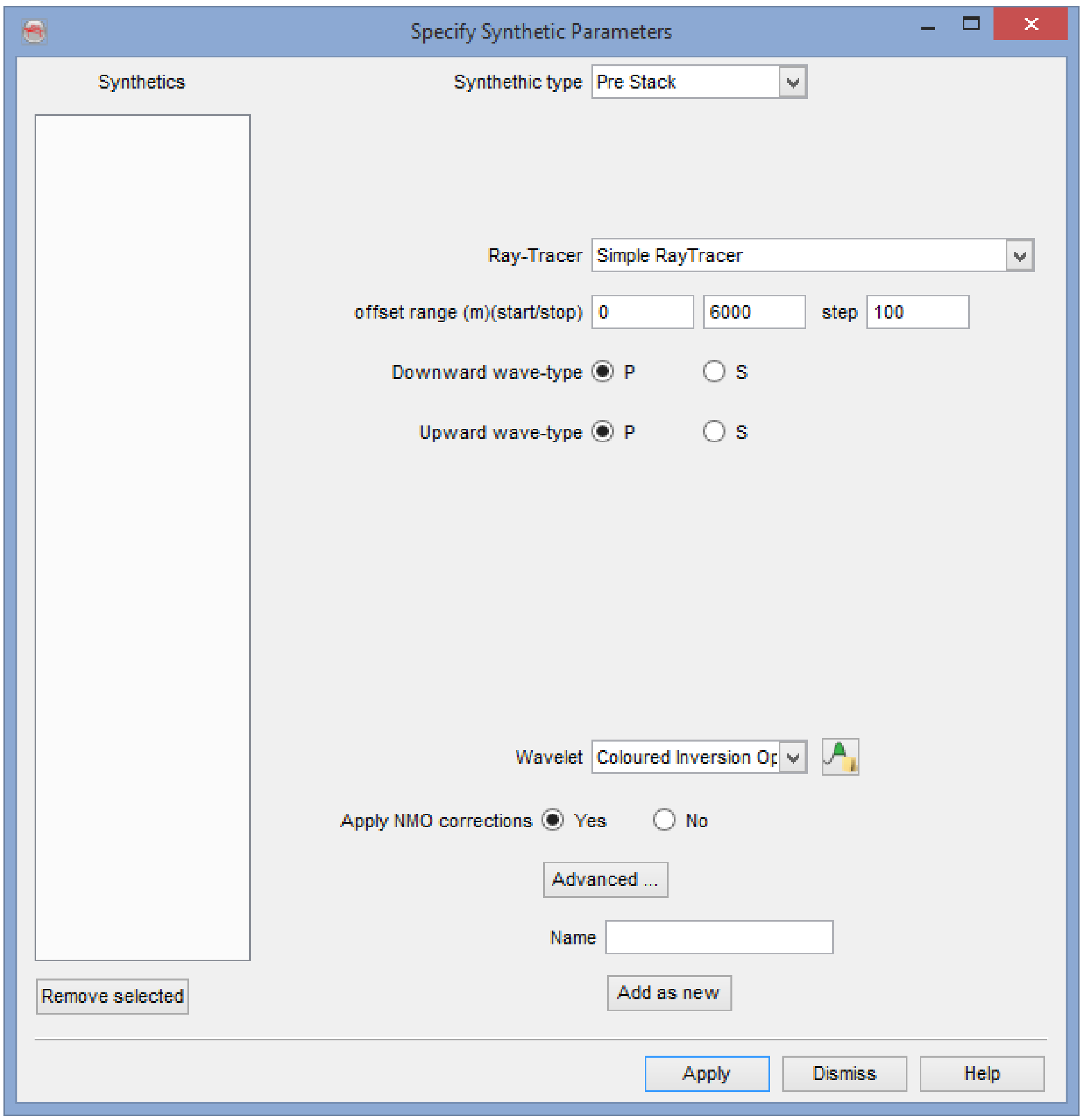
There is an Open Source group that writes and maintains drivers for the Wacom tablets.

For more information please go to this page.

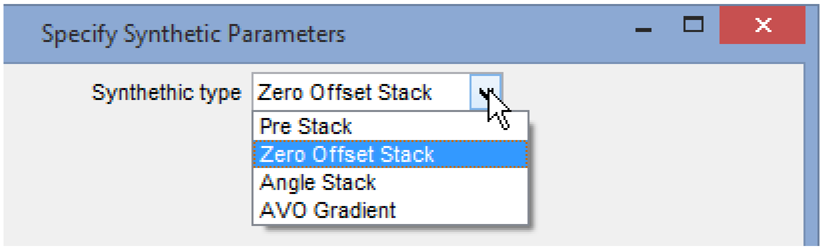
Appendix E - Synthetic Data Generation

Ray Tracing
Computation of the Zero Offset Reflection Coefficient
Computation of the Reflection Coefficient at any non-zero offset
Elastic Model

Synthetic seismic data is generated in *SynthRock* by clicking on the edit icon () in the top-left corner of the main Layer Modeling Interface. This will bring up the following window:



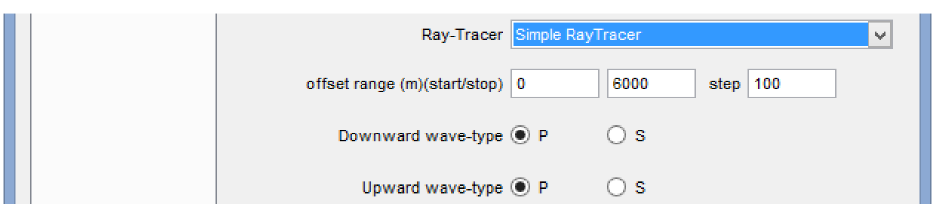
Here various types of synthetic data can be generated: Zero Offset Stack, Pre Stack gathers, Angle Stack and AVO Gradient:



Ray tracing

While Zoeppritz equations are used to compute angle-dependent reflectivity; ray tracing is required to compute the angle of incidence, at various interfaces of elastic model, of seismic rays recorded at various offsets. The 2D ray-tracing can be performed in two ways:

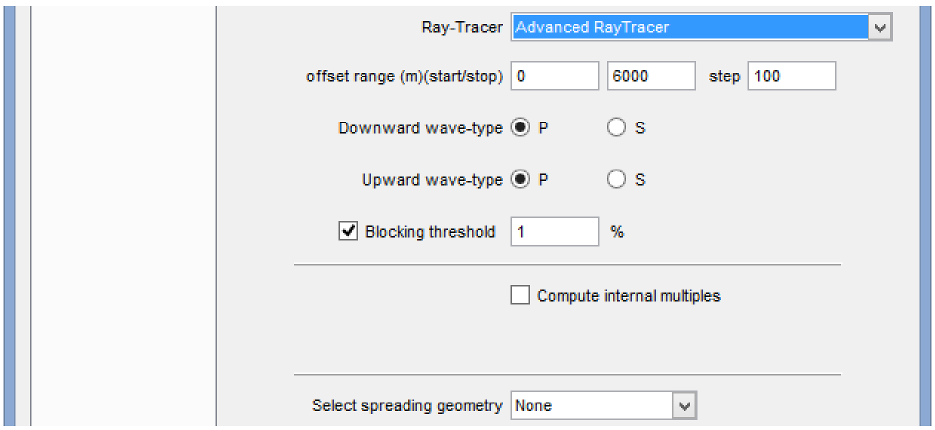
Simple Ray Tracer:
The ray is going directly from the source to the depth of the target layer, and up to the receiver in the same way. This does not account for ray bending, or velocity inversions. Here the user has to specify the offset range and the step for creating pre stack gathers; they could in theory be same as defined in acquisition/processing of the seismic data. It can model both Downgoing and Upgoing P-waves and S-waves. Now, ray tracer and Zoeppritz equations have produced angle-dependent or offset-dependent reflectivity traces, which can be convolved with user defined wavelet to produce pre stack gathers. It may be noted that in SynthRock, the conversion from offset domain to angle domain and vice-versa is done using the Vp of the Elastic Model [hyperlink with Elastic Model] (which is essentially the upscaled and time converted Vp log of pseudo-wells).



Simple RayTracer parameters

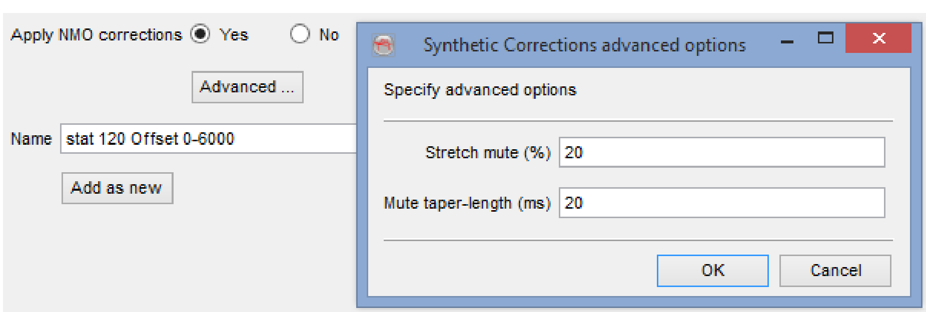
Advanced Ray Tracer (not in the GPL version)**:**

It works in a more sophisticated way than the simple ray tracer. It honours the ray bending according to Snell's law and thus velocity inversions as well. To reduce the processing time, the Elastic Model [hyperlink with Elastic Model] layers may be blocked: Consecutive layers with similar Vp, Vs and Density values are concatenated together, as defined by the threshold. For example the default threshold is 1%, which means if there is less than 1% difference in the elastic model values of two layers, they will be blocked. The ray is propagated in a straight line inside a concatenated layer.
It is also possible to compute internal multiples in the advanced ray tracer. Furthermore, incorporation of spherical divergence, is also possible, by defining the spreading geometry as either "Distance" or "Distance*Vint" .



Advanced RayTracer parameters

Afterward, NMO corrections can also be applied to create NMO corrected synthetic gathers. Here in the Advanced options, one can specify the % stretch mute typically applicable at far offsets. If the length of a full seismic waveform increases by more than the mute %, it will get muted. Moreover, the taper-length of the muting function, can be defined under this advanced options menu of NMO corrections:



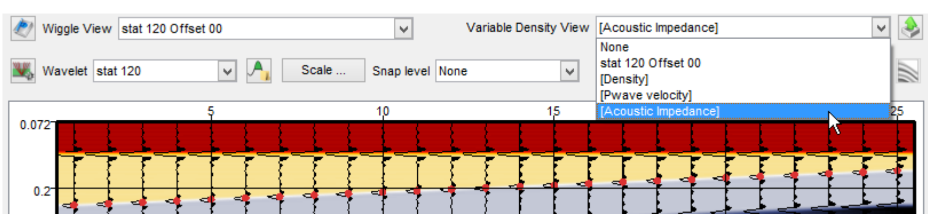
Advanced RayTracer: Advanced corrections options

Computation of the Zero Offset Reflection Coefficient For the simplest Zero Offset Stack, calculation of reflection coefficient at any interface is done using the simple formula:

$$R = \frac{(Z_1 - Z_0)^2}{(Z_1 + Z_0)^2}$$

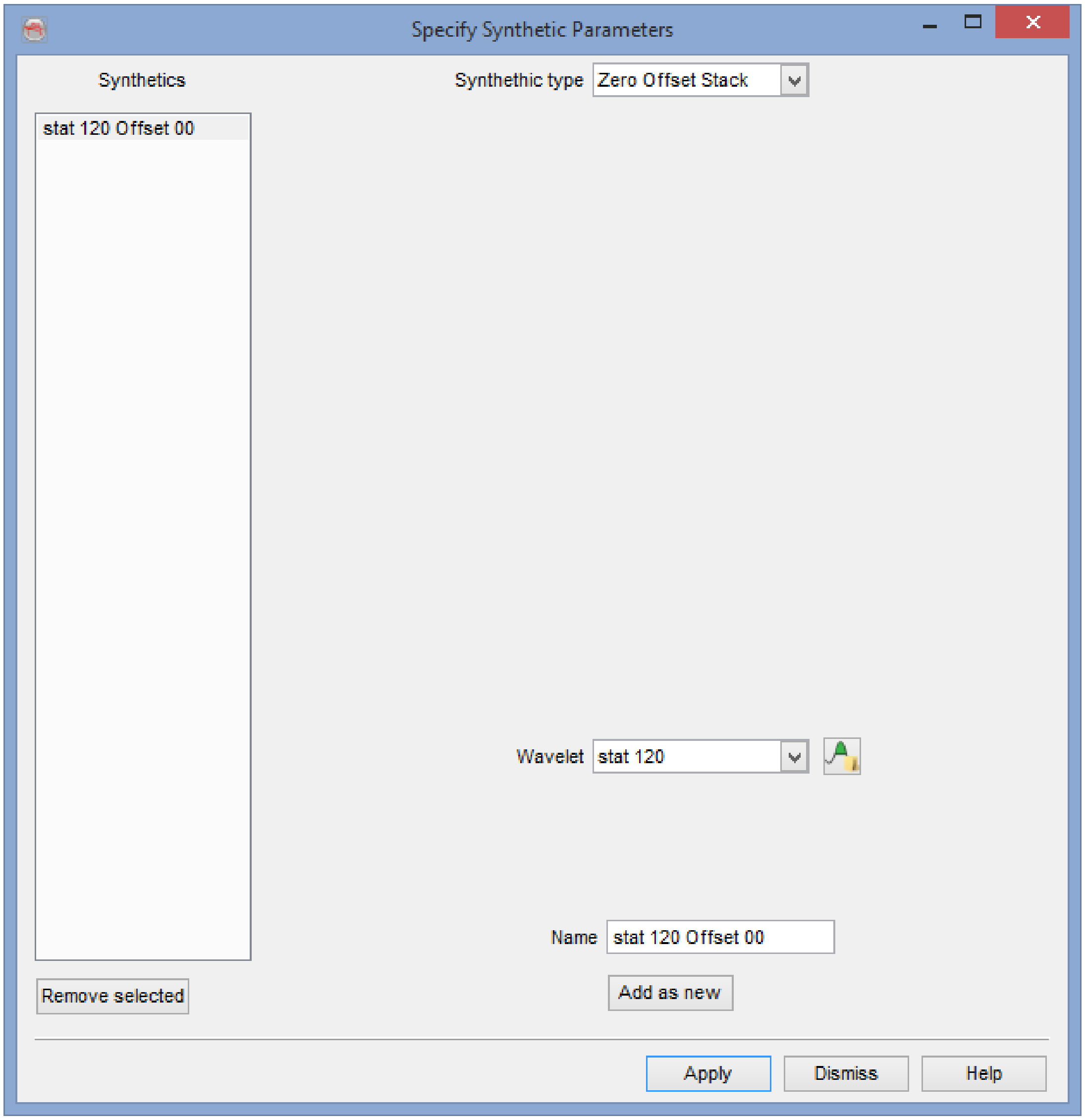
where Z_1 and Z_0 are the impedance of the top and bottom layers, respectively. These layers are basically upscaled and time converted version of various logs (Rho, Vp and Vs) in pseudo-well models, and as such comprise the Elastic Model [hyperlink with Elastic Model] for synthetic seismic generation. The upscaling is done using the Backus averaging algorithm in depth, but at a (variable) depth sampling rate which is equivalent to the seismic sample rate in time. Depth-to-time conversion of the pseudo-well logs, is done using the velocity model of the pseudo-wells itself.

> **Note:** Backus upscaling is done only for Vp, Vs and Density logs (and other logs based on them e.g. AI, LambdaRho, MuRho etc.). All other logs e.g. Phi, Sw etc. are upscaled using thickness weighted averaging (i.e. weights used for the averaging are the thicknesses of various pseudo-well layers) and are afterwards converted into time (using the velocity model of the pseudo-wells), at survey sample rate. A Nyquist filter, as defined by the survey sample rate is also applied on these time converted rock property traces; e.g. if seismic survey sampling is at 4 ms, Nyquist filter will allow a maximum frequency of 125Hz. These are accessible to user in real-time on the Variable Density View:



Now, computation of above described reflection coefficient, at all the possible acoustic impedance contrasts in upscaled pseudo-well layers, gives rise to a reflectivity trace in time.

This reflectivity trace is then convolved with a user defined wavelet, to create the Zero Offset Stack for all the pseudo-well models:

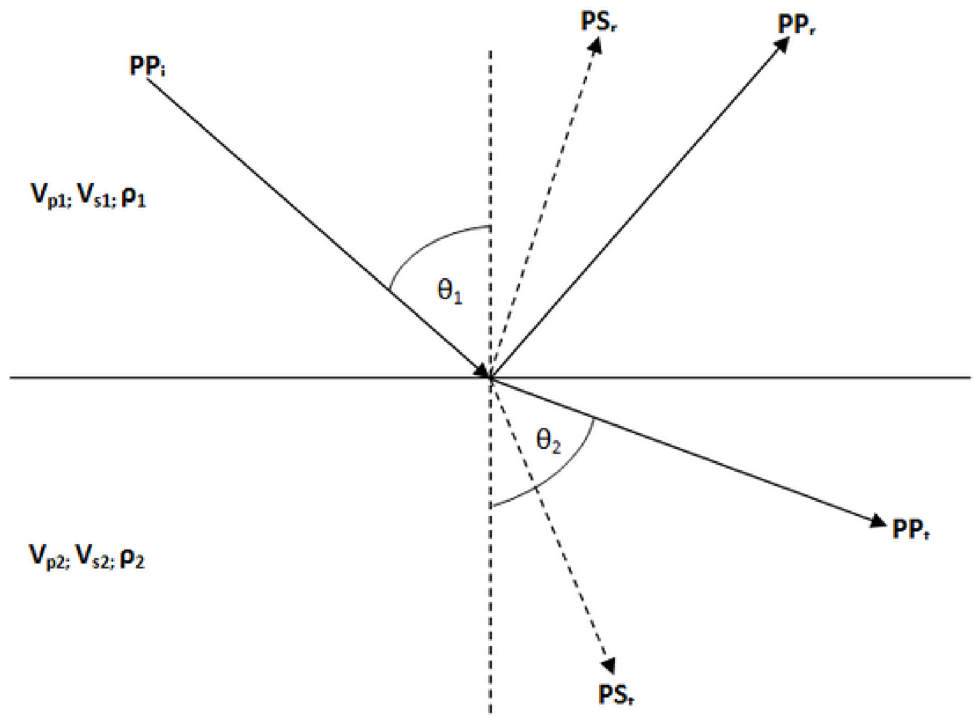


<u>Computation of the Reflection Coefficient at any non-zero offset</u> Pre Stack data (i.e. offset gathers) can be generated in OpendTect using full Zoeppritz equations and ray tracing (simple or advanced).

Full Zoeppritz equations are used to compute angle-dependent reflectivity from the elastic model (i.e. upscaled and time converted version of various logs (Rho, Vp and Vs) from pseudo-wells) at various interfaces as:

(images are from Wikipedia)

$$\begin{bmatrix} R_{\mathrm{P}} \\ R_{\mathrm{S}} \\ T_{\mathrm{P}} \\ T_{\mathrm{S}} \end{bmatrix} = \begin{bmatrix} -\sin\theta_1 & -\cos\phi_1 & \sin\theta_2 & \cos\phi_2 \\ \cos\theta_1 & -\sin\phi_1 & \cos\theta_2 & -\sin\phi_2 \\ \sin 2\theta_1 & \frac{V_{\mathrm{P}1}}{V_{\mathrm{S}1}}\cos 2\phi_1 & \frac{\rho_2 V_{\mathrm{S}2}^2 V_{\mathrm{P}1}}{\rho_1 V_{\mathrm{S}1}^2 V_{\mathrm{P}2}}\cos 2\phi_1 & \frac{\rho_2 V_{\mathrm{S}2} V_{\mathrm{P}1}}{\rho_1 V_{\mathrm{S}1}^2}\cos 2\phi_2 \\ -\cos 2\phi_1 & \frac{V_{\mathrm{S}1}}{V_{\mathrm{P}1}}\sin 2\phi_1 & \frac{\rho_2 V_{P2}}{\rho_1 V_{P1}}\cos 2\phi_2 & \frac{\rho_2 V_{S2}}{\rho_1 V_{P1}}\sin 2\phi_2 \end{bmatrix}^{-1} \begin{bmatrix} \sin\theta_1 \\ \cos\theta_1 \\ \sin 2\theta_1 \\ \cos 2\phi_1 \end{bmatrix}$$

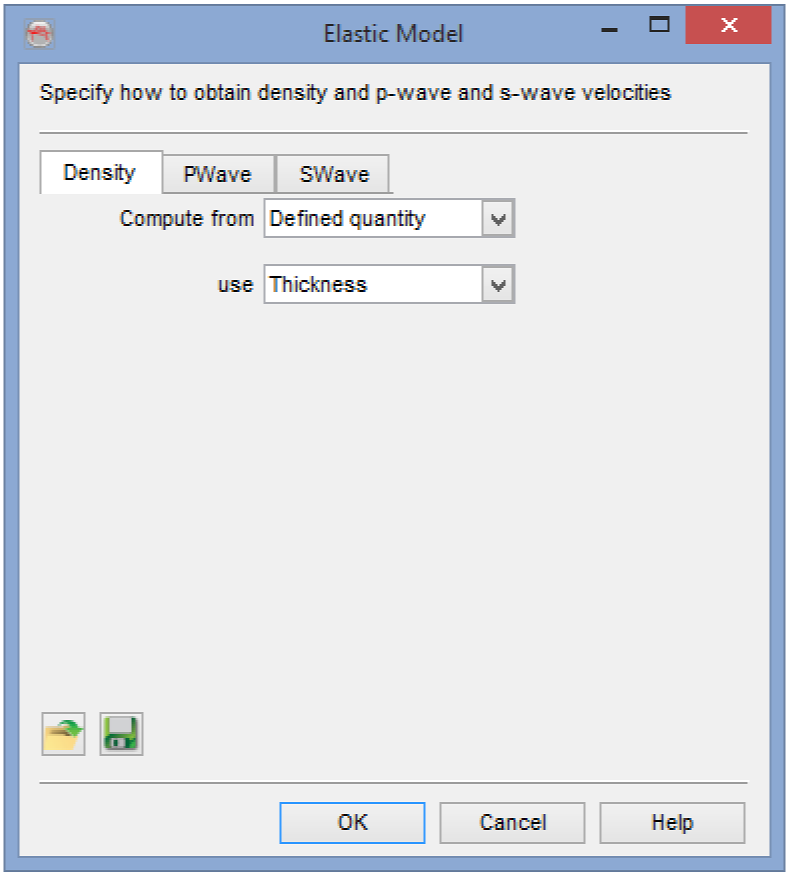


A P-wave (PP_i) propagating through a medium of density, ρ_1, P-wave velocity, V_{p1}, and S-wave velocity, V_{s1}, is incident upon an interface with a medium of density, ρ_2, P-wave velocity, V_{p2}, and S-wave velocity, V_{s2}, at an angle, θ_1. Mode conversions occur resulting in reflected P- and S- waves (PP_r and PS_r respectively) and transmitted (refracted) P- and S- waves (PP_t and PS_t respectively).

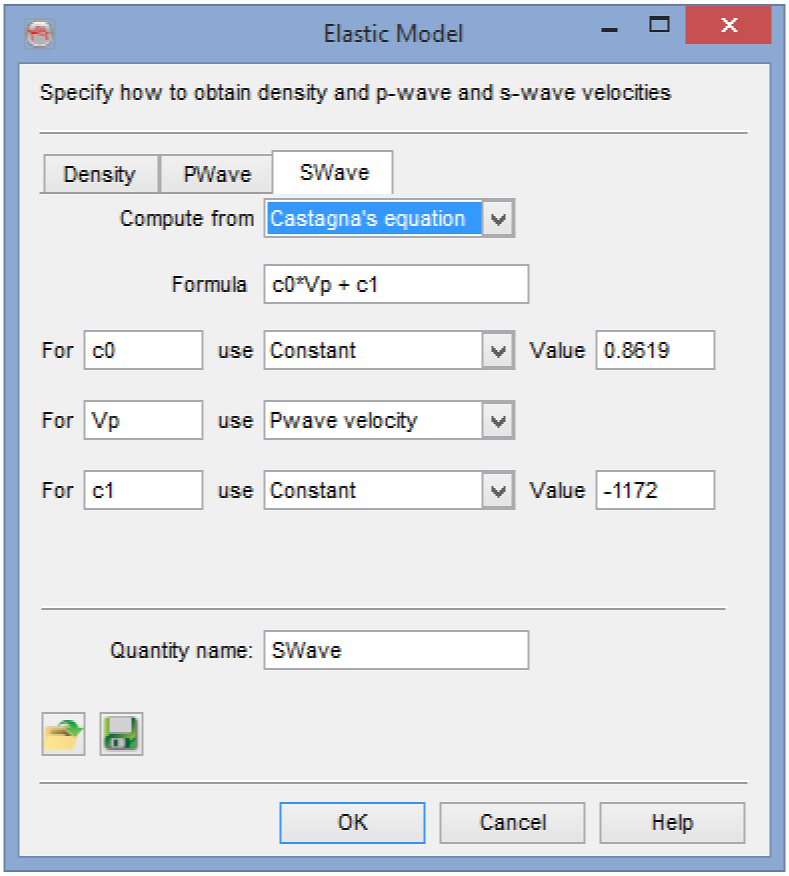
(above images are from Wikipedia)

Elastic Model

This Elastic Model can be accessed by clicking the  icon, just left of '*Wavelet*'. This model isrequired by OpendTect for generating synthetic seismic data (both zero offset stack and pre stack gathers). The elastic model essentially tells the software which quantities to use for the reflection coefficient computation and ray tracing, in terms of Density, Vp and Vs:



If "Compute from: Defined quantity" is chosen, OpendTect can use appropriate (upscaled and time converted) quantities from pseudo-wells. User can also chose to compute missing quantities (not modeled in pseudo-wells) using pre-filled rock-physics relations, e.g. Vs from Vp using Castagna's equation:


Elastic Model
Specify how to obtain density and p-wave and s-wave velocities
Density
PWave
SWave
Compute from
Castagna's equation
Formula
c0*Vp + c1
For
c0
use
Constant
Value
0.8619
For
Vp
use
Pwave velocity
For
c1
use
Constant
Value
-1172
Quantity name:
SWave
OK
Cancel
Help

Glossary

A

Absolute Impedance

Full-bandwith impedance inversion response in which the "missing" low-frequency part of the spectrum has been added by the inversion method. For example - in model-driven inversions the low-frequency model is typically created by interpolating impedance well logs guided by mapped seismic horizons.

Accomodation Space

The available space for sediments to fill (measured from seafloor to base-level).

AI

Acoustic Impedance: the product of seismic velocity and density.

Attribute

An attribute is a derived quantity from a seismic input set. Attributes in OpendTect are defined by a name, a value, and a position in 3D space (inline, cross-line and Z (2WT or depth)). Attributes can be calculated from single-trace, multi-trace, and multi-volume inputs. They can be steered and/or chained. Steered attributes are multi-trace attributes in which the trace segments are found by following a (pre-) calculated dip-azimuth. Chained attributes are attributes derived from other attributes. For example, Similarity and Energy are separate attributes that can be chained to calculate the Similarity of the Energy using the "Position" attribute.

Attribute Set

An attribute set is an entity consisting of a group of attributes. Usually attributes in a defined attribute set have something in common. For example, all attributes in a set have the potential to highlight an object type of interest, or a combined attribute, using all other attributes as intermediate results. This would be a desirable output.

B

Base level

The surface at which sediment supply, relative sea level changes and wave energy are in balance. This is the surface at which the accommodation space equals zero: there is neither deposition, nor erosion.

Body

A body is an element that defines an arbitrary three dimensional geological shape (or a geo-body). The body can also be created manually or by using polygons.

C

ChimneyCube

A volume that highlights vertical disturbances in seismic data. The cube is used in fluid migration path studies, in prospect ranking and for fault seal analysis. A ChimneyCube is generated by a neural network that was trained on picked examples (chimneys and non-chimneys). It gives at every sample location the "chimney probability" i.e. the likelihood of belonging to the class of identified seismic chimneys.

Chrono-stratigraphy

A set of relative geologic time lines as stored in a HorizonCube.

CLAS

A plugin for petrophysical analysis. CLAS stands for Computer Log Analysis System.

Closed Source

Software that is released in binary form only. The commercial plugins to OpendTect are released as closed source extensions. Such extensions are only permitted if OpendTect is run under a commercial (or academic) license agreement.

Color Blending

Combined display of three (four) attributes that are displayed in the Red Green and Blue color channels. Optionally the fourth channel (alpha) displays transparency. Color blending is aka as RGB (RGBA) blending.

Crossline Dip

Dip in the direction of the Crossline axis, or in the direction of increasing crosslines.

D

Dip-Steering

The process of auto-tracking seismic data by following the pre-calculated, local dip and azimuth of the seismic. Dip-steering is used for: a) extracting seismic trace segments along seismic reflectors as input to multi-trace attribute calculations, b) computing special attributes such as polar dip, azimuth, and volume curvature attributes, c) filtering seismic data (known as dip-steered filtering, aka structurally oriented filtering), and d) auto-tracking chrono-stratigraphic horizons in the creation of a HorizonCube.

Dip-Steering Cube

A volume computed from seismic data with at every sample position information about the local dip and azimuth of the seismic data. In a 3D Steering Cube this information is stored in two attributes per sample: inline dip and cross-line dip. On 2D seismic only one value is stored: the line-dip. Dips in a Steering Cube are measured in the line direction and expressed in us/m or mm/m, for time and depth data, respectively.

E

EEI

Extended Elastic Impedance. Scaled and rotated impedance response at a paarticular angle. Rotation is typically optimized to predict a certain well log property of interest.

EI

Elastic Impedance. Impedance response at a particular angle of incidence.

Element

An element is a sub-division of various items (of the tree) that are displayed in a 3D scene. Inline, crossline, timeslices, horizon, wells etc are some elements. Each element is sub-divided into a sub-element. For instance an inline element can have further sub-elements e.g. inline # 120 that can contain upto eight different attributes.

Eustatic sea-level

Sea-level relative to center of earth.

Explicit Representation

A representation of a 3D object in OpendTect in the form of a triangulated surface.

F

Fault Stickset

The faults are interpreted on a section as a stick, and all sticks that belong to one fault are grouped in one sticksets. Therefore, a fault stickset contains an unordered collection of the interpreted sticks.

Forced regression

Deposition characterized by progradation and incision. Base-level is falling decreasing accomodation space, forcing the system to prograde. Forced regression occurs during the Falling stage systems tract.

G

GMT

An open source mapping package developed and maintained by the University of Hawaii (http://gmt.soest.hawaii.edu/). GMT stands for Generic Mapping Tools.

GPL License

Gnu General Public License (http://www.gnu.org/licenses/gpl.html) is an open source license under which OpendTect can be run. The license allows redistribution of (modified) source code under the same licensing conditions (copy left principle). It is not allowed to combine the open source part with closed source plugins, which is why OpendTect is also licensed under a commercial license agreement and under an Academic license agreement.

H

Horizon Data

It refers to a stored attribute grid in a horizon. An attribute is calculated on-the-fly or in a batch process. On-the-fly, a user needs to store by right-clicking on it an selecting Save attribute... option. The saved attribute can also be managed in the Manage horizon. It may be noted that a horizon can contain unlimited stored attribute/horizon data.

HorizonCube

A dense set of auto-tracked (or modeled) seismic horizons that is indexed and sorted according to relative geologic time (= chrono-stratigraphy).

I

Implicit Representation

A representation of a 3D object in OpendTect in the form of an iso-surface through a cube of values.

Incision

Depositional feature caused by erosion.

Inline Dip

Dip in the direction of the Inline axis, or in the direction of increasing inline numbers.

M

Madagascar

An open source seismic processing package. See: http://en.wikipedia.org/wiki/Madagascar_(software)

Meta Attribute

A meta-attribute is an attribute created from multiple input attributes. In OpendTect, a meta attribute is created either through neural networks, or through mathematical manipulations and/or logical operations. For example, TheChimneyCube and TheFaultCube are meta-attributes. See the Ridge enhancement filter attribute set from the Default attribute sets for an example of a meta-attribute created through math and logic. The meta-attribute in this set is the last attribute in the list.

MLP Neural Network

Multi-Layer-Perceptron type of neural network. The network is used for seismic object detection (creating Chimney Cubes, Fault Cubes, Salt Cubes etc.) and for predicting rock properties from seismic data (Porosity, Vshale, Sw etc). An MLP network is trained on a data set with known examples (supervised learning). In the training phase the network aims to find the optimal, non-linear mapping between input attributes and target attributes. The network in OpendTect is a fully-connected, three-layer MLP (input layer, hidden layer, output layer). The non-linear transformation takes place in the hidden layer.

MPSI

A plugin for stochastic acoustic impedance inversion. MPSI stands for Multi-Point Stochastic Inversion.

N

Normal regression

Deposition characterized by aggradation and progradation. The base level is rising but the consumption of accommodation space by sedimentation exceeds the creation of accommodation space by the base level rise. Normal regression occurs during high stand and low stand systems tracts.

O

Open Source

Software that is released with its source code. OpendTect is released as open source product that can be extended with closed source plugins. Such extensions are only permitted if OpendTect is run under a commercial (or academic) license agreement.

P

PDF

PDF is Probability Density Functions. In OpendTect these are created in the cross-plot tool by selecting a desired area in the cross-plot domain. The density of the points in the selected area is a measure for the probability of the desired target variable that can then be predicted by applying the derived PDF function to (scaled) input volumes in a Bayesian classification scheme.

Pickset

A Pickset is a collection of picked locations, i.e. inline-crossline-Z information. Picksets are part of a Pickset Group. For example a Pickset Group containing picks at fault locations may consist of different fault Picksets to differentiate between large faults and small faults, or to reflect picks on different inlines.

Pickset Group

A Pickset group is a collection of different Picksets. Usually Picksets are grouped because they refer to the same object, e.g. Chimney_yes or Chimney_no.

R

Regression

Seaward shoreline & facies shift. Regression can be Normal (base level rises) or Forced (base level falls).

Relative Impedance

Band-limited impedance inversion response computed by methods such as colored inversion.

Relative sea-level

The net effect of eustatic sea level changes and local tectonic fluctuations.

Retrogradation

Depositional trend characterized by sediments building landwards aka back-stepping.

S

SEG-Y

A file format for exchanging seismic or seismic-like data. It is used for both 2D and 3D pre- or post-stack data. A file being SEG-Y compliant does not mean that it can be loaded into OpendTect. There are several possible problems. One of these is missing trace identification and/or positioning. Another issue is lack of true compliance ->SEG-Y Rev 0, -> SEG-Y Rev 1). The different types of SEGY are shown below: * SEG-Y Rev 0: The initial SEG-Y specification in 1975. It is very precise in some areas but totally unspecified in other, crucial areas. This has led to an almost uncountable number of variants. Some are sort-of SEG-Y standard, others blatantly non-compliant. * SEG-Y Rev 1: In 2002 the Revision 1 document made an end to the most obvious shortcomings of ->SEG-Y Rev 0, especially in the area of ->trace positioning and ->trace identification. Still many SEG-Y files or files claimed to be SEG-Y are Rev 0 or badly (i.e. not) compliant with Rev 1. This is why OpendTect has numerous options for the SEG-Y reading process. * SEG-Y Textual header: The first 3200 bytes of a SEG-Y file must be filled with textual comment on the contents of the SEG-Y file. Older textual headers are encoded in EBCDIC rather than ASCII, which makes them impossible to read in a standard text editor. * SEG-Y EBCDIC header: -> SEG-Y Textual header. * SEG-Y Tape Header: The part of a SEG-Y file that gives information about all traces in the file. This information is in the ->SEG-Y Textual header and ->SEG-Y Binary header. * SEG-Y Binary header: The second part of the SEG-Y Tape header contains binary information about, amongst others, values for number of samples per trace, byte encoding, sample interval, and SEG-Y Revision. * Trace identification: Every trace in OpendTect needs to have an identification in form of a trace number (2D data) or inline/crossline (3D data). For pre-stack data the offset forms and extra trace identification. * Trace positioning: In OpendTect, every seismic trace needs to be located in 3D space. For 3D data, the position can be derived from the->Trace identification (inline- and crossline numbers).

Traces in 2D lines have their own, separate X- and Y- coordinate. For pre-stack data there must also be an offset available.

SSIS

A plugin to perform a sequence stratigaphic analysis (systems tracts, Wheeler transforms) from seismic data using HorizonCube input. SSIS stands for Sequence Stratigraphic Interpretation System.

Stratal Slicing

The process of cutting through a seismic volume along surfaces that are computed proportionally between mapped top and bottom horizons, aka proportional slicing.

Systems Tracts

Subdivisions of sequences that consist of discrete depositional units that differ in geometry from other systems tracts and have distinct boundaries on seismic data. Different systems tracts are considered to represent different phases of baselevel changes.

T

Trace Identification

Every trace in OpendTect needs to have an identification in form of a trace number (2D data) or inline/crossline (3D data). For pre-stack data the offset forms and extra trace identification.

Trace Positioning

In OpendTect, every seismic trace needs to be located in 3D space. For 3D data, the position can be derived from the->Trace identification (inline- and crossline numbers). Traces in 2D lines have their own, separate X- and Y- coordinate. For pre-stack data there must also be an offset available.

Transgression

Landward shoreline & facies shift characterized by aggradation and retrogradation. The base-level is rising and more accommodation space is created than is consumed by sedimentation

Tree

The tree is the docking window, which is detachable and movable. This is used to display the data into a scene. The tree is attached to a scene and is labeled as Tree Scene 1. Where '1' is the scene number. Each tree has its own elements that are displayed in corresponding scene.

U

UVQ Neural Network

Unsupervised Vector Quantizer type of neural network. This network is used for clustering (segmenting) data into a user-defined number of clusters. Cluster centers are found in a training run on a subset of the data. In the application phase the network generates two outputs: 1) the index number of the winning cluster and 2) the match, a value between 0 and 1 indicating how close the input vector is to the vector representing the winning cluster. UVQ segmentation can be performed in 2D mode (waveform segmentation along mapped horizons) and in 3D mode (generates 3D bodies). A display of the cluster centers is a useful diagnostic in waveform segmentation (Neural Network module: Info button).

V

VMB

A plugin for picking velocities from semblance gathers, and in a surface-consistent-manner. VMB stands for Velocity Model Building.

W

Waveform Segmentation

Process of clustering seismic trace segments with a UVQ network along a mapped horizon into a user-defined number of clusters.

WCP

A plugin to pick and QC well log markers with the help of seismic data and (optionally) the HorizonCube. WCP stands for Well Correlation Panel.

Wheeler Transform

Process of flattening seismic data (or attributes) according to the chrono-stratigraphic horizons in a HorizonCube. In a Wheeler scene the vertical axis represent relative geologic time.

Index

D

E

F

G

H

I

K

L

M

N

O

P

R

S

T

U

V

W

Z