CONTENTS

Introduction

This document describes the new features and enhancements that make up the T7.110 upgrade and differentiate it from the T7.100 release. T7.110 is distributed as a patch release (though a full download can be obtained from our download site) and can be installed over a T7.10x installation as a patch. For a more detailed description of the new features please refer to the relevant sections in the T7 Reference Manual. This document and the T7 Reference Manual use the term "T7" when referring to the current release version of the software.

Important Notes:

1) T7 license feature versions are “7.1”. T7.1 will not function with a T7.0 (or ealier) license. Please send T7.1 license requests to support@badleys.co.uk.

2) T7 uses FlexNet 11.12 for its licensing. The installation will include the 11.12 version of the FlexNet license manager daemon.

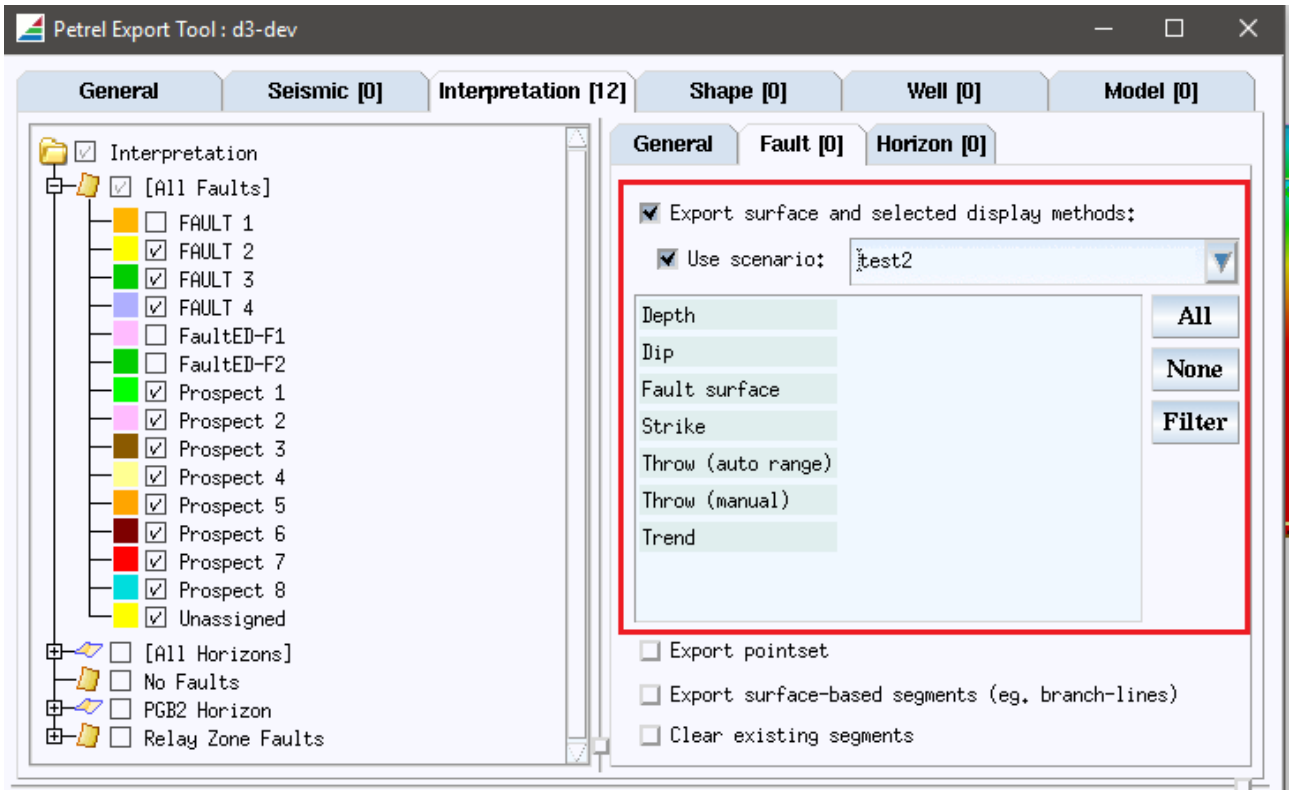
3) Once installed, T7.110 will run a database upgrade on existing T7 projects when they are opened. **Please backup you projects before using them with T7.110**. Once a project has been upgraded it will not be accessible using an older version of T7.

4) After a project has been upgraded, the volume editor default shortcut file (defining hotkeys) will be replaced with a more fully featured set of hotkeys and shortcuts. The original shortcut definition file will be renamed to "default.###" (where ### is the version of T7 last used on the project).

Export of Fault & Horizon Display Methods to Petrel

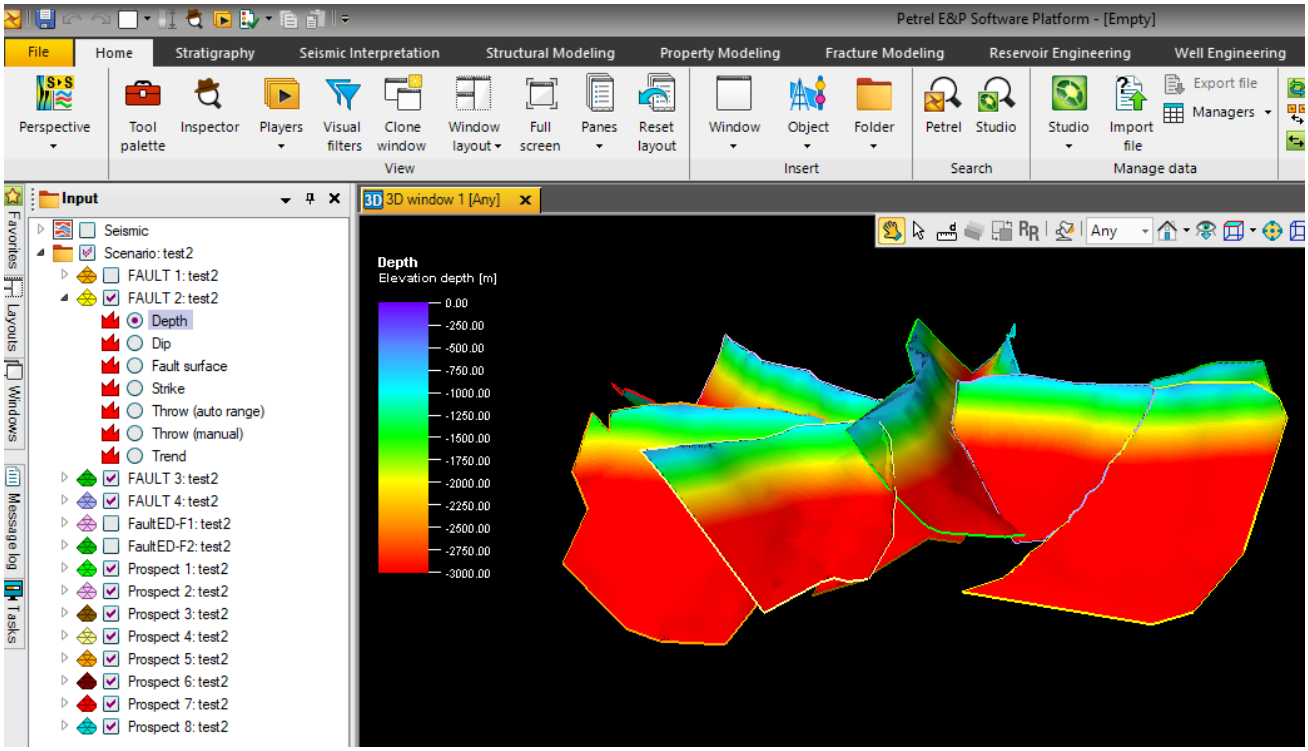
When exporting fault and horizon surfaces to Petrel using the Petrel Export Tool, it is now possible to export T7 Display Method output for visualisation.

Display Methods for surfaces are chosen in the “Fault” and “Horizon” subtab of the “Interpretation” tab in the Petrel Export Tool:



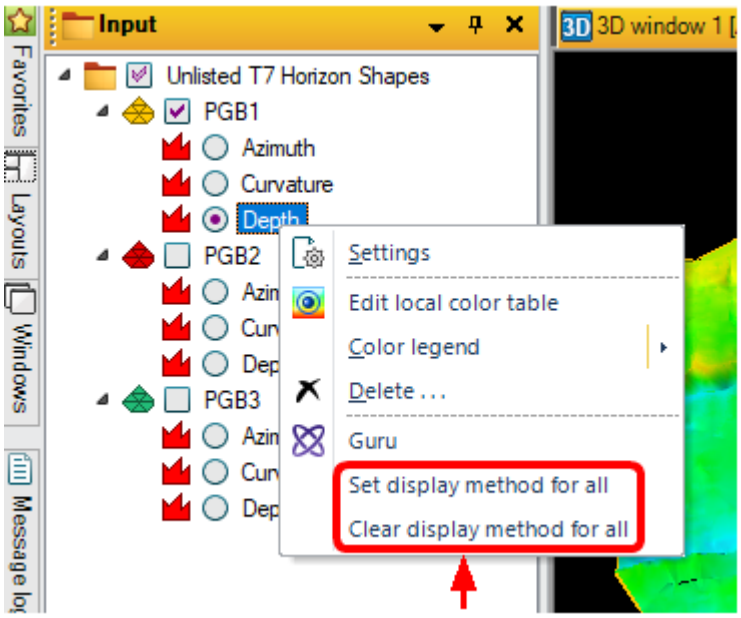
A scenario name may be specified for both fault and horizon Display Method export. When specified, fault and horizon surfaces will be created or updated in Petrel using a dedicated folder matching the scenario name. This allows visualisation of different scenarios for the same faults or horizons (eg. different Vshale models, column height calculations, stress scenarios, etc).

Upon import into Petrel using the Badley Petrel Plugin, fault and horizon surfaces are created as T7 triangle mesh objects. Display Methods are represented in the tree as radio buttons within each triangle mesh object:

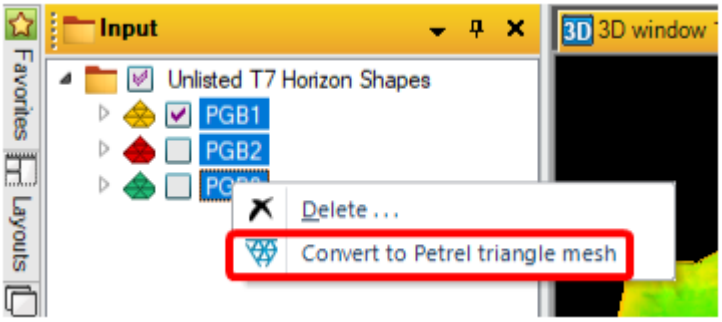


The active Display Method used to render the trimesh can be changed by clicking a child Display Method radio button. To show the trimesh using the native object color (no Display Method), the radio button can be unchecked.

A Display Method can be set for all trimesh objects under the current folder by right clicking the Display Method and choosing "Set Display Method for all" from the menu. A similar option "Clear Display Method for all" causes all trimeshes under the current folder to display using the native object color.



An option exists to convert (copy) a T7 trimesh to a new native Petrel trimesh. This can be useful if the trimesh is required as input for structural modelling, or if it requires transfer back to T7. Note that the Petrel trimesh does not support display methods

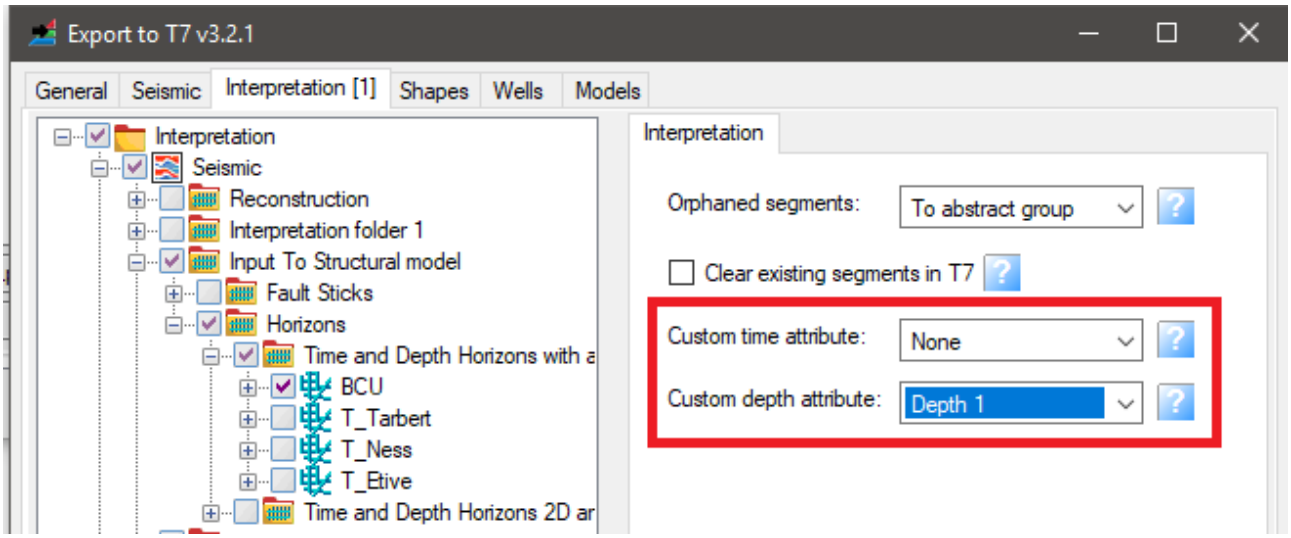


.

Petrel Plugin Custom Attribute Option

Schlumberger Petrel allows many objects to be extended by attributes, one use of which is to specify alternative Z data.

The Badley Petrel Plugin now allows a custom time or depth attribute to be specified when exporting fault and horizon interpretation, surfaces and pointset shapes:



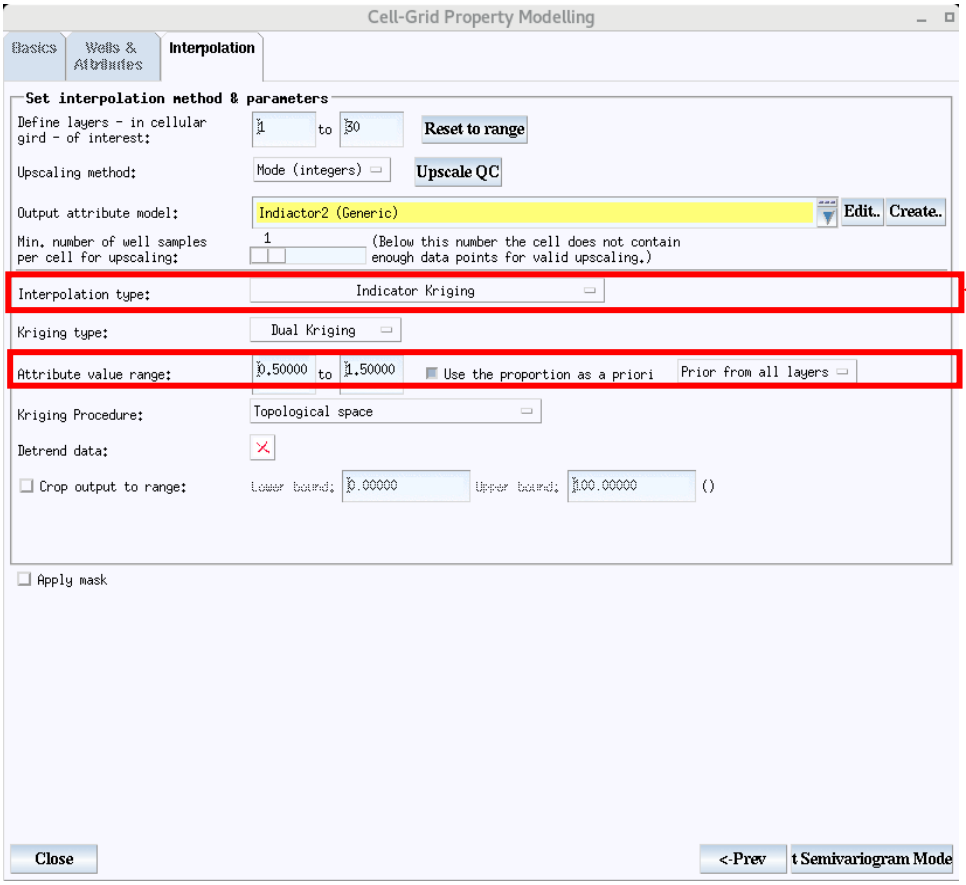
Note the image above shows the “Interpretation” tab, but the functionality is also available in the “Shapes” tab.

Both time and depth custom attributes may be specified. The list of available attributes is built based on the currently selected objects in the tree. Only attributes using the “Elevation depth” and “Elevation time” template are supported. Attributes that represent the domain object underlying Z data are also excluded.

Enhanced Cell-Grid Property Modelling QC Tools

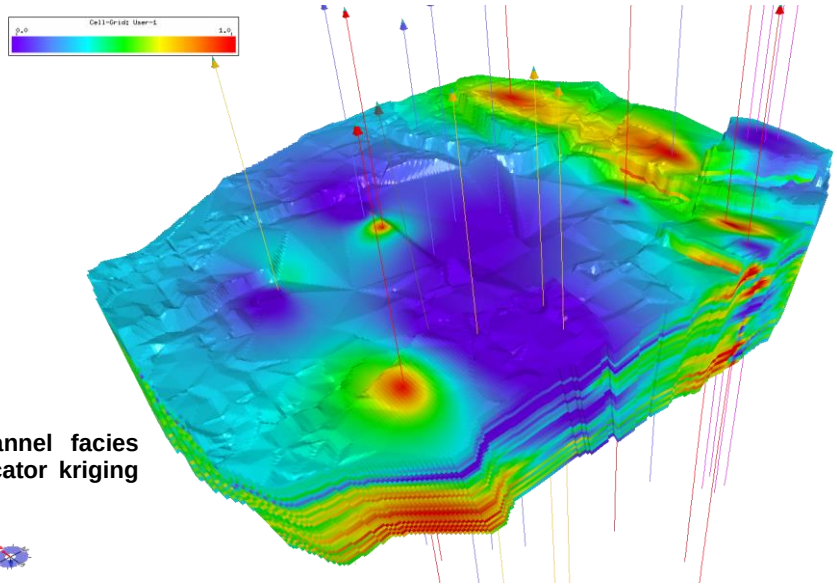
A significant number of additions have been made to the property modelling toolkit.

Indicator kriging functionality has been added to the deterministic suite of options. This enables the user to create probability fields for either a single category value (for facies integer codes) or real numbers (continuous variables such as V-Shale). In the case of attributes with continuous values, a suite of models can be created using different threshold ranges for the purposes of uncertainty analysis.



Select Indicatror Kriging option

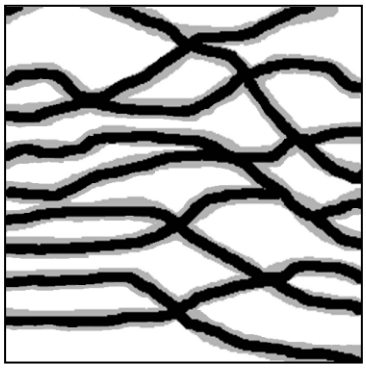
Settings to define the target range and how to populate cells whose estimates are not well constrained by the variogram models.

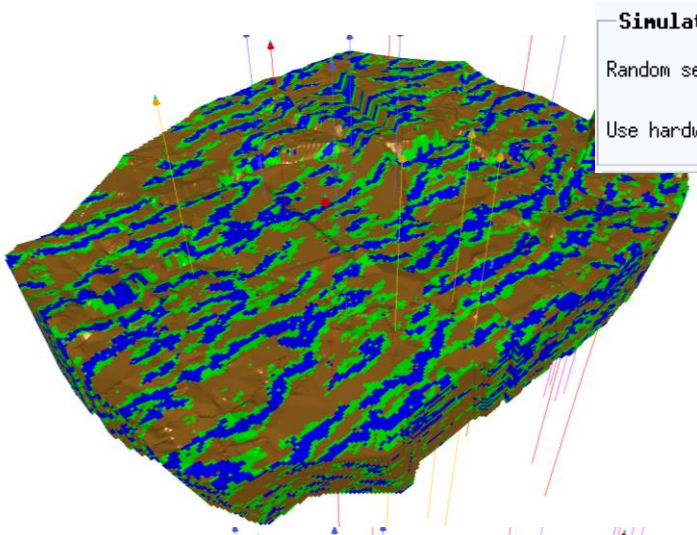


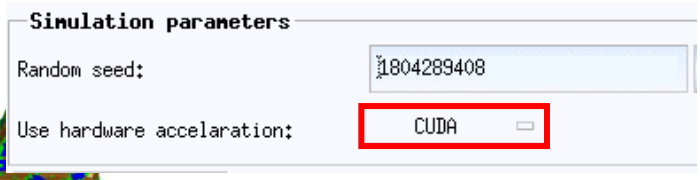
Probability field for channel facies produced using the indicator kriging method.

CUDA optimisation is now available for stochastic facies modelling. This is a hardware dependent feature (requires a Nvidia graphics card) that utilises hardware acceleration for the rapid computation of conditional probabilities during facies modelling. The principal advantage of this feature is that it produces higher quality models than the existing "fastest" option (using the Search List) but with comparable performance.

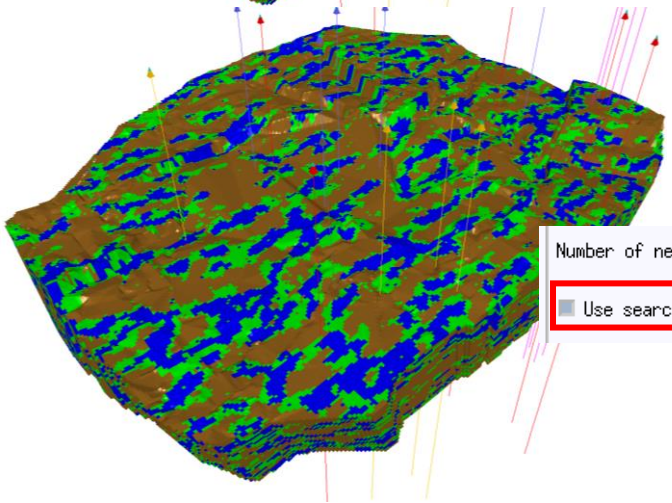
Training image of fluvial system



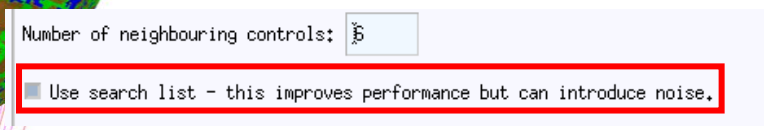




Select the CUDA option in the SFM tab of the main property modelling dialog. This enables production of high-quality models without compromising performance.



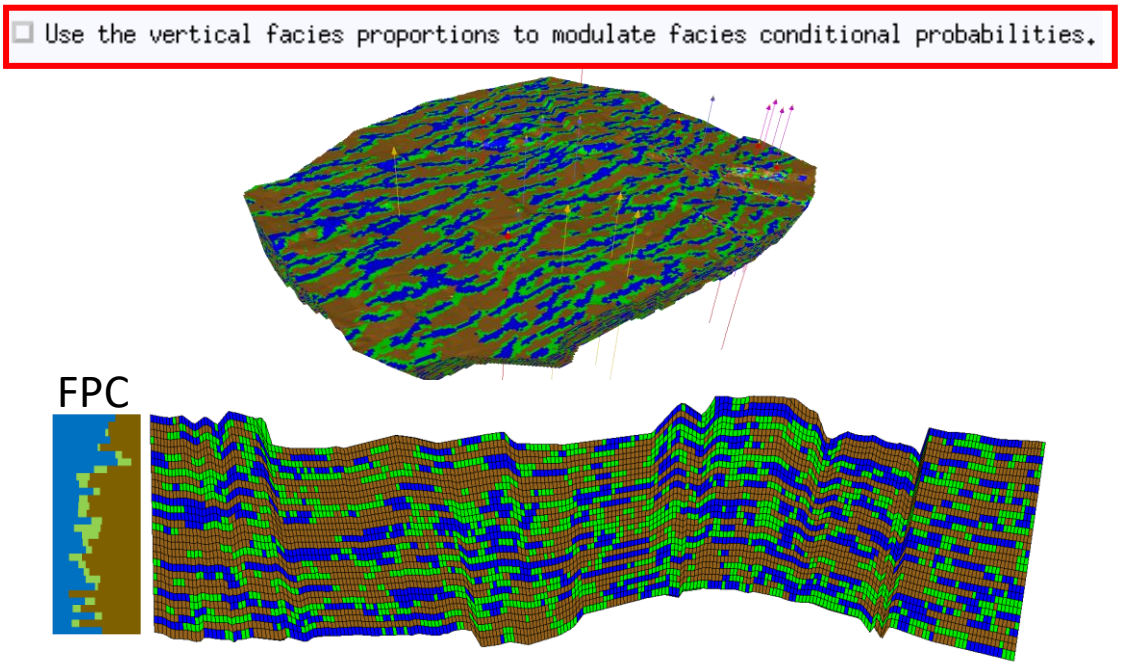
Without the hardware-enabled acceleration offered by CUDA, software-based optimisations need to be utilised. The existing system uses a Search List approach to speedup conditional probability computations. While this does produce good models, they can suffer from noise.



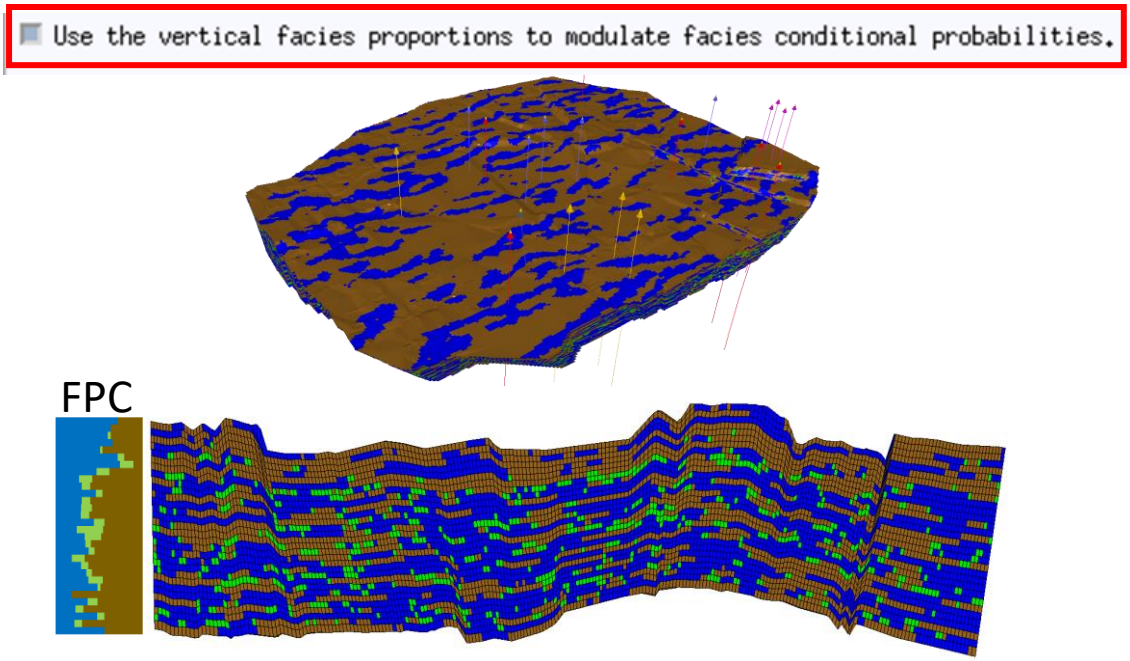
The search-list option is available for the CUDA-approach too but should be toggled off for the best results.

Facies proportion curves (FPC) can be used to constrain the proportions of the facies types in an output facies model. In more precise terms the posterior distribution ≈ prior distribution. If this is not chosen then the facies proportions tend toward those of the training image. These FPCs are computed from the upscaled well data (per layer).

The option is available in the Settings tab of the Geostatistical Wizard. When toggled off the output model tends toward the facies proportions in the training image.

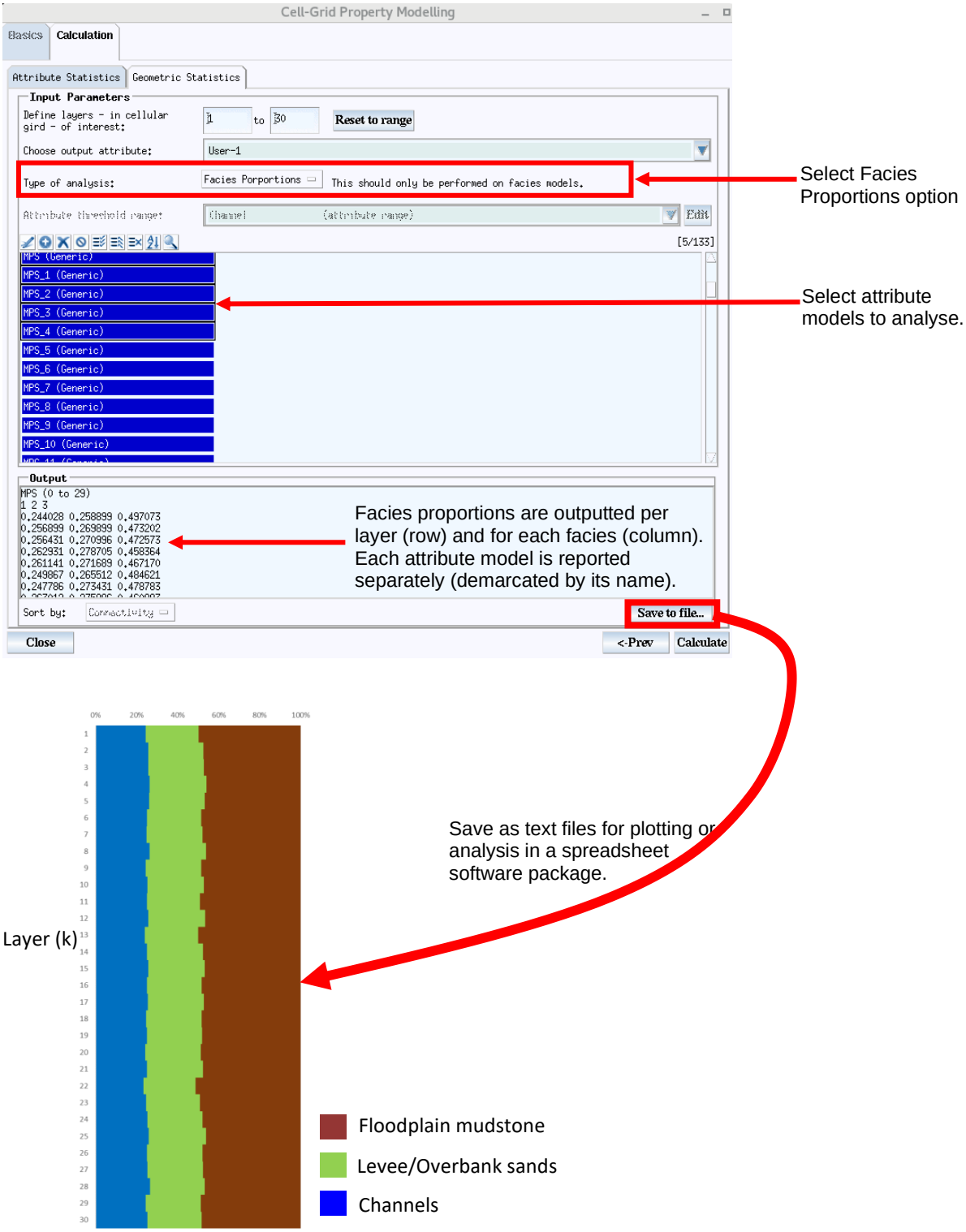


Section through model shows no variation in facies proportions through the model when compared with input FPC.

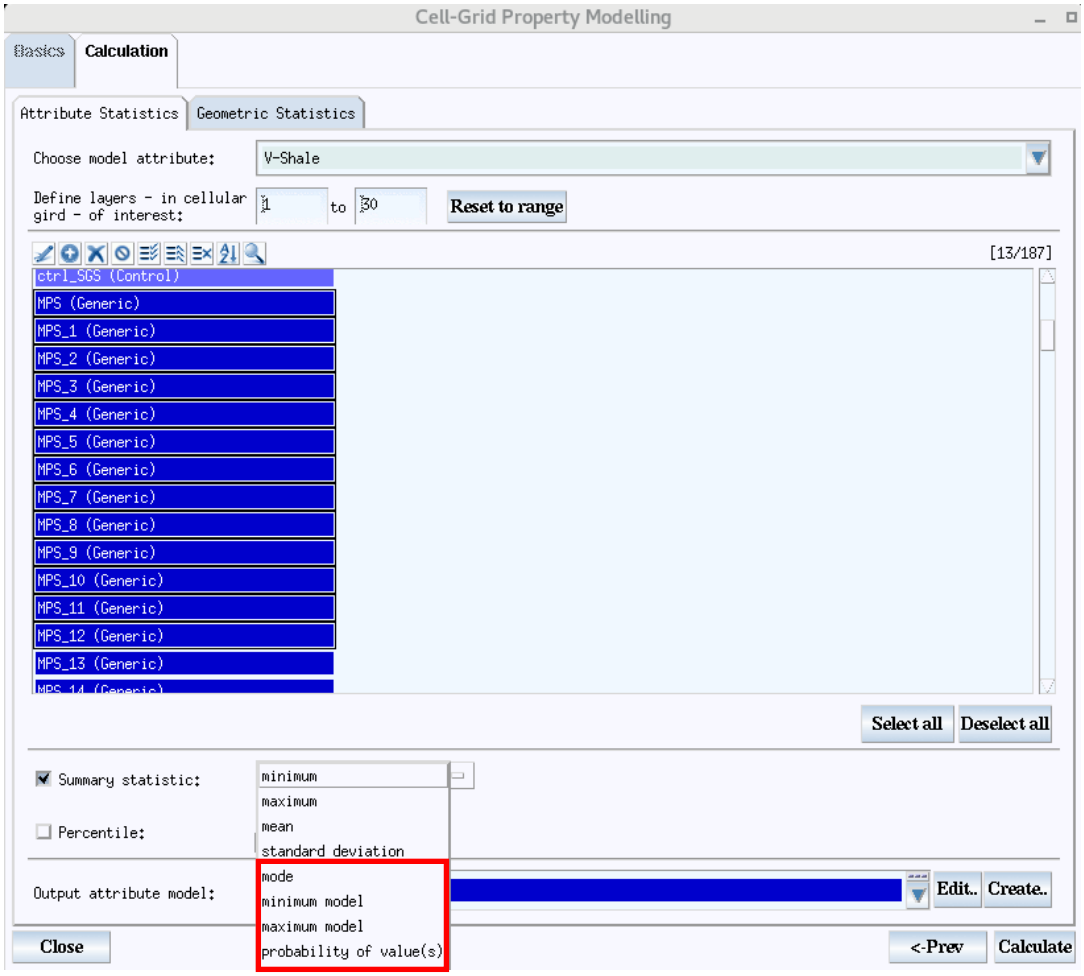


Section through model shows variation in facies proportions through the model in accordance with the input FPC.

Facies Proportion Curves (FPC) can now be computed from an output facies model using the Calculation property modelling tool. In the Geometric Statistics sub-tab, the user can now choose to either perform connectivity analysis or compute the facies proportion curves from a given selection of models. The results are reported in the Output textfield below and can be saved to file.



A number of new **Summary statistics** have been added to the Calculation tool. These greatly enhance the existing options enabling the creation of a wider range of summary models.



New options (should be added to current list):

- ***mode**: will output (per cell) in the the most common value encountered in the suite of selected models (useful for facies modelling).*
- ***minimum model**: populates the output model (per cell) with the model number (numerical suffix) of the model with that has the minimum value for the given selection of input models.*
- ***maximum model**: populates the output model (per cell) with the model number (numerical suffix) of the model with that has the maximum value for the given selection of input models.*
- ***probability of value(s)**: outputs the probability (per cell) of finding a particular value in the suite of selected input models.*
- ***equal to**: output the values (per cell) from a selected model.*

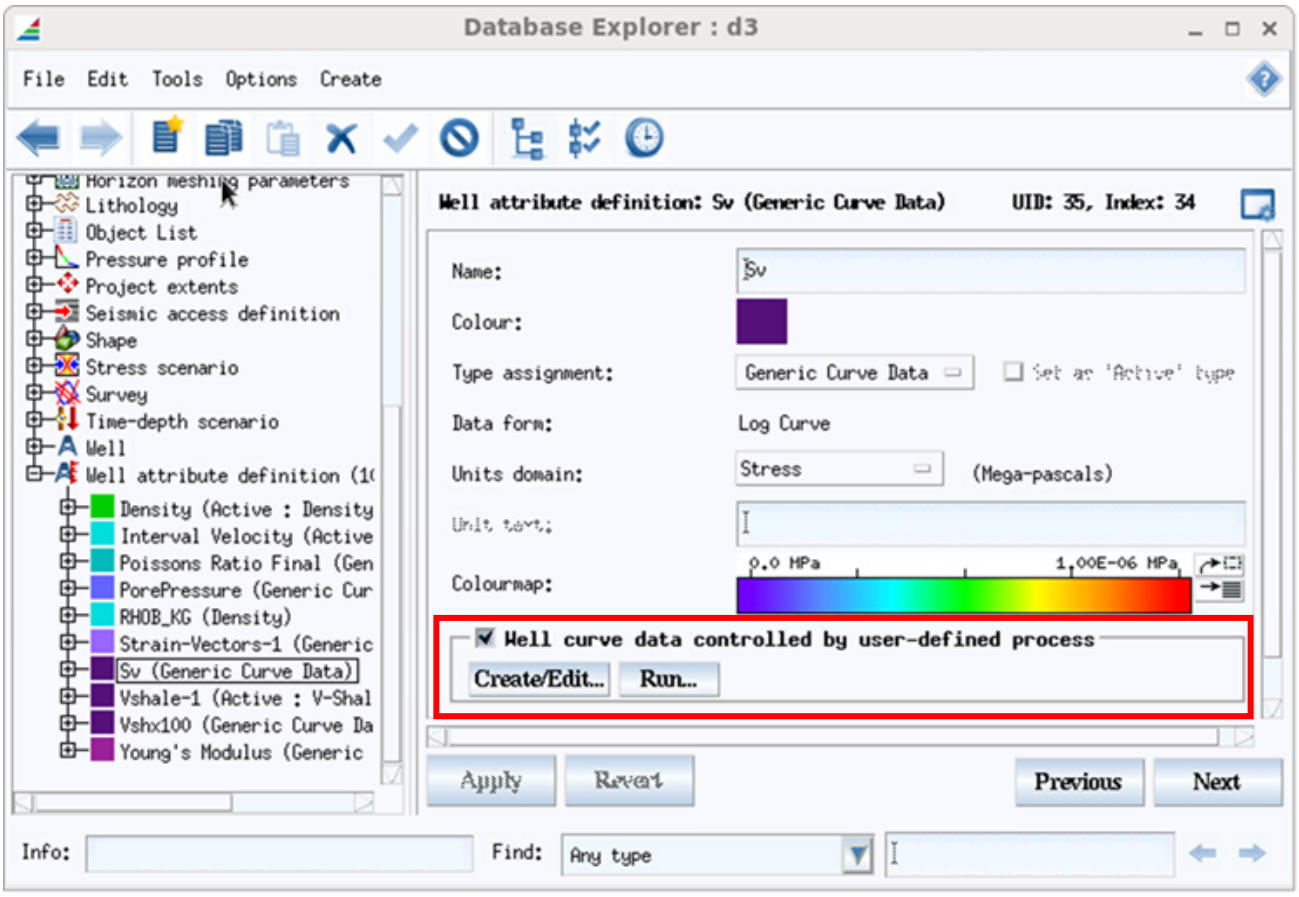
Other additions have been added which help with modelling of a more diverse range of geological systems and attributes along with more enriched QC options. Of these the most important are:

- New upscaling methods for Deterministic, SGS and Collocated cokriging. Added to the current options of arithmetic and geometric mean are the harmonic mean, mode and Voigt-Reuss-Hill method
- As part of the Collocated Kriging tool primary variables can be modelled using secondary trends derived directly from secondary attributes – this is more efficient than collocated kriging proper but provides similar results and maintains the petrophysical relationship between dependent attributes such as porosity and permeability.
- The Quality Control plots now have axes/plot options that can be modified using a simple dialog. This helps control how the data is computed and displayed.
- Multithreading has been introduced in a number of areas to make the overall workflow more streamlined and efficient. The computation of statistics in the Calculation tool are now multithreaded as are the derivation of well/cell intersections performed after completing the Well and Attribute tab (relevant for Deterministic, SGS, Collocated Cokriging and SFM).

Well-Attribute Calculator

A new system has been added to T7.110 to permit the user to define their own macros for computing well Log Curves. The Well-Attribute Calculator is much like the Attribute Calculator currently available for Faults, Horzons, Fracture Networks, Cell-Grids etc, but operates directly on the existing database constructs for Well Attribute Definitions and Well Attribute Log Curves. A macro can be enabled, either permanently or temporarily for any existing or newly created Well Attribute Definition. Once a macro has been enabled it may be executed to create new, or modify existing Log Curves for one or more chosen wells.

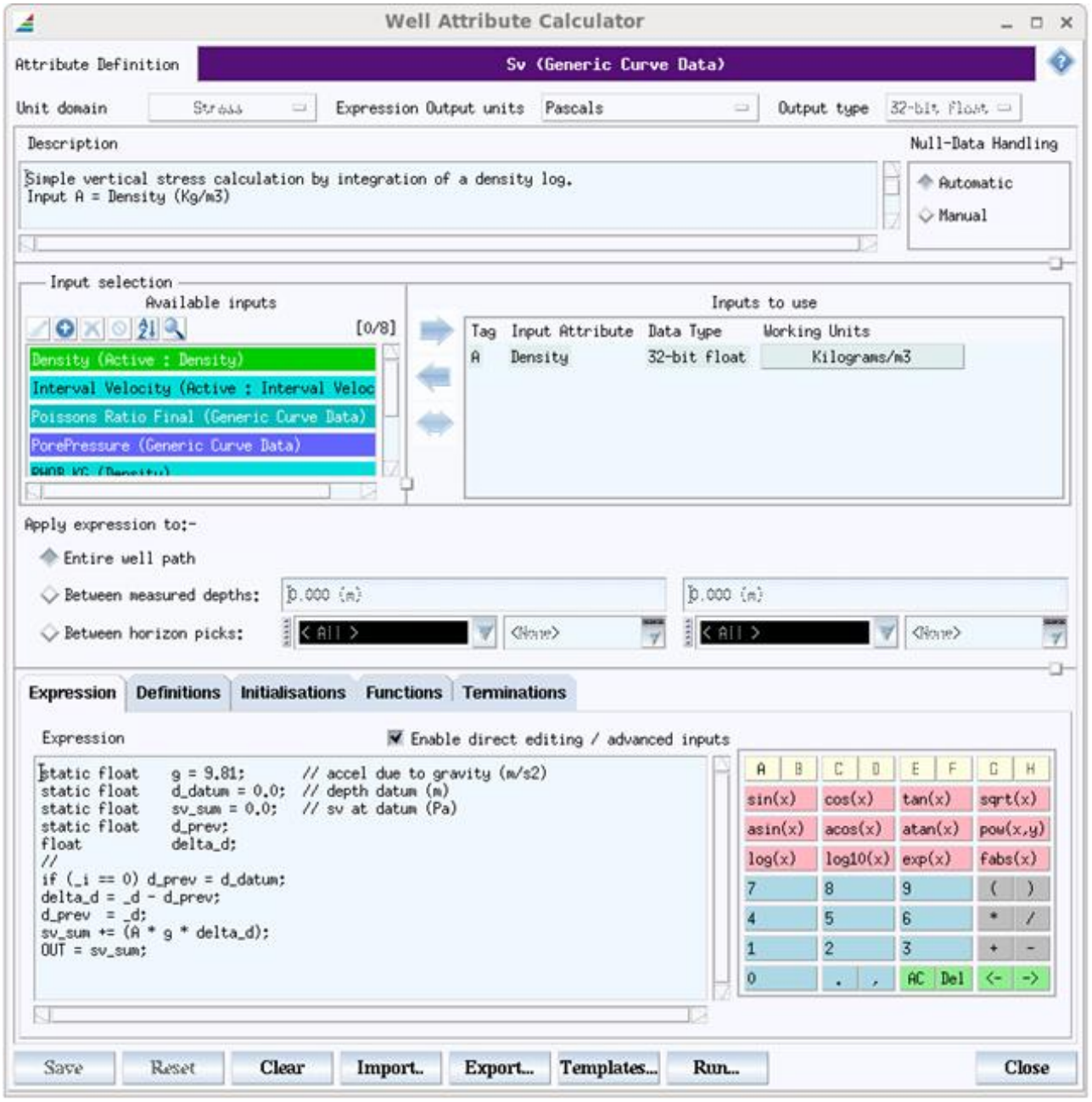
The first point of access is the Well Attribute Definition which is located in the Database Explorer or in any list of Well Attributes in other T7 applications.



A new option in the Well Attribute Definition editor, "**Well curve data controlled by user-defined process**", has been added. When this option is enabled, it provides access to the "**Create/Edit…**" and "**Run…**" buttons. The "**Create/Edit…**" button will invoke the Well Attribute Calculator which allows the creation and editing of a macro which will operate for that Well Attribute Definition. The "**Run…**" button opens a well-selection window where the user chooses the wells for which the macro will then be executed.

A further method of accessing the Well Attribute Calculator if from the **Edit** menu in the **Well Editor**. A new option "**Well Attribute Calculator…**" will open the Well Attribute Calculator window for the 'Active Log'.

The Well Attribute Calculator window is shown below.



The Well Attribute Calculator window is a tool for creating or editing the macro specific to a given Well Attribute Definition. The macro defines a process, written in standard C code, that can vary from a simple one line expression to a multitude of complex C functions.

The macro can take inputs that define Log Curve data for other Attributes and has access to basic down-hole properties such as depth, measured depth, X, Y etc. The macro can define a process that modifies or uses the Log Curve data in its own Attribute Definition.

As a very simple example, let's say that some V-Shale data was imported for several wells in the wrong units, with values in percent ranging from 0 to 100. We need to convert this to be a ratio in the range 0 to 1. We enable the Well Attribute Definition option "**Log-Curve data controlled by a user-defined process**", start the Well Attribute Calculator and write the simple macro expression:

```
OUT = OUT/100.0;
```

Having then saved the macro, we use the "**Run…**" option to choose all the wells for which we wish to process the V-Shale data. Having successfully executed the macro on the required wells we can go back to the Well Attribute Definition and disable the option "**Log-Curve data controlled by a user-defined process**" so that the process cannot be inadvertently executed again.

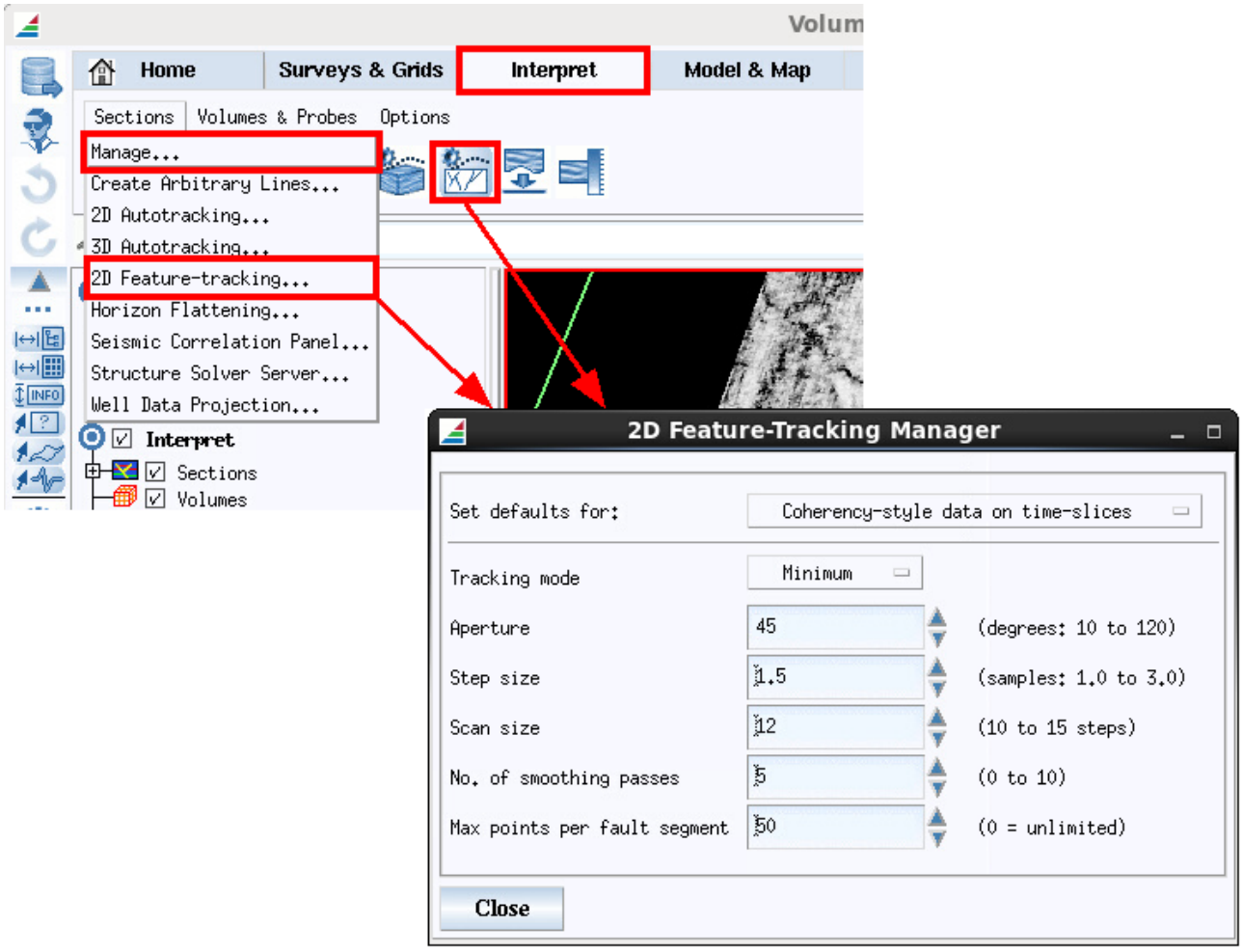
Other features supported by the Well Attribute Calculator include:

- Import and export of macros
- Loading and saving of macro templates

Feature-Tracking Interpretation Tool

The new 2D Feature-Tracking system in the Volume Editor is an assistive tool for picking fault segments on seismic sections. It is particularly effective when working with coherency-style displays on time-slices.

The 2D Feature-Tracking function is controlled by a set of parameters in the 2D Feature-Tracking Manager window. This is accessed from the Interpret tab's Section menu or Toolbar icon.



The Feature-Tracking system operates by tracking the "best path" through the displayed seismic data between pairs of manually picked points on the section being interpreted.

The control parameters are:

- **Set defaults for**: Select an option from the drop-down menu to set all parameters to data-specific defaults. Default parameters can be set for the following: Amplitude data on vertical sections, Coherency-style data on vertical sections and Coherency-style data on time-slices. Choose a set of defaults suitable to the required workflow and adjust the parameters to suit the data and the features to be tracked.

- **Tracking mode**: Set to Minimum to track a minimum amplitude, Maximum to track a maximum amplitude and Zero-crossing to track the minimum absolute amplitude. It is vital to set this mode to pick-out the required features. For standard amplitude

data on vertical sections it is best set to Zero-crossing; for coherency-style data it is normally set to Minimum.

•**Aperture**: This is the angle of deviation that a tracked point can take from its previous point relative to the direction to the target point. This angle is measured in unit sample space, so in vertical seismic data it should normally be reduced to account for the difference between the real horizontal and vertical sample dimensions. Smaller angles will result in less deviation along the path between the control points.

•**Step size**: This is the step size (relative to a single sample) between successively tracked points. Reducing the value will give finer fidelity but may prevent the tracking from stepping through areas of weak feature "signal".

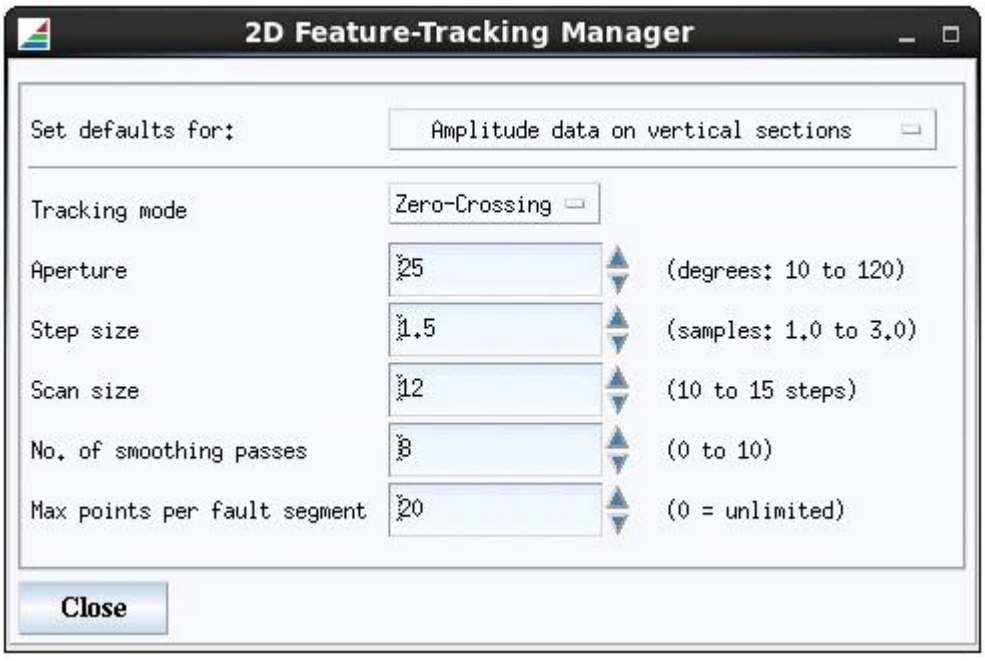
•**Scan size**: The tracking process is undertaken as a number of successive scans until the target point is reached. This is the number of steps (each of Step size samples) taken per scan. Increasing this value to is maximum of 15 will result in slower tracking but will more effective when tracking through areas weak feature "signal".

•**No. of smoothing passes**: this controls the degree of smoothing applied to the resultant fault segment points.

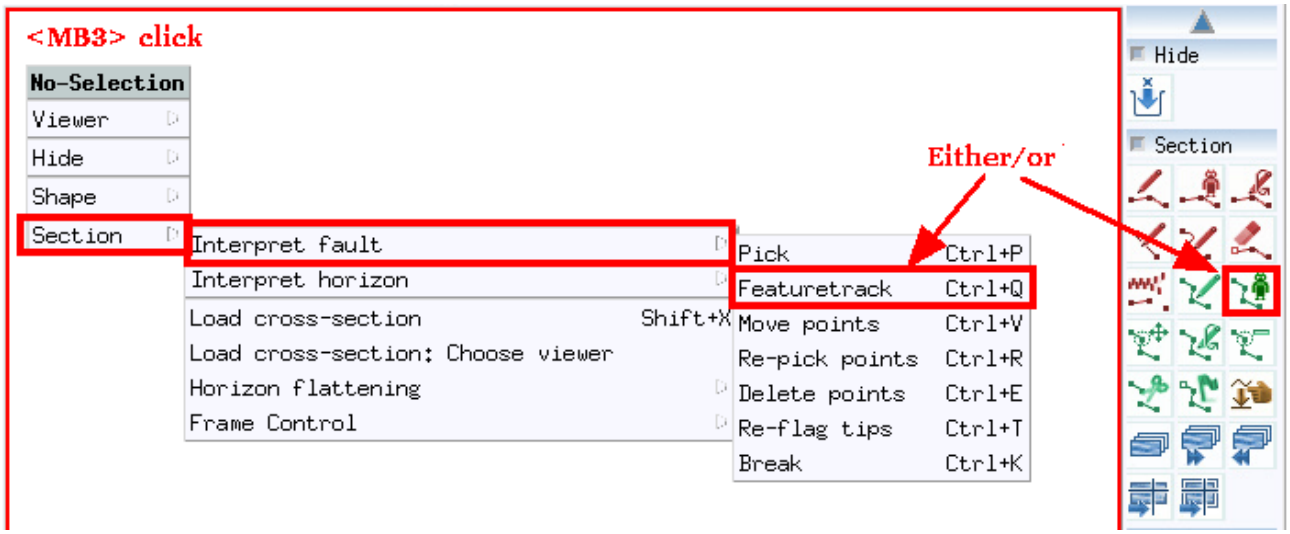
•**Maximum points per fault segment**: this defines the maximum number of points that will be used from the set of tracked and smoothed points to create the final fault segment. Set to zero for this limit not to be applied.

Example 1: Feature-tracking through amplitude data on vertical sections.

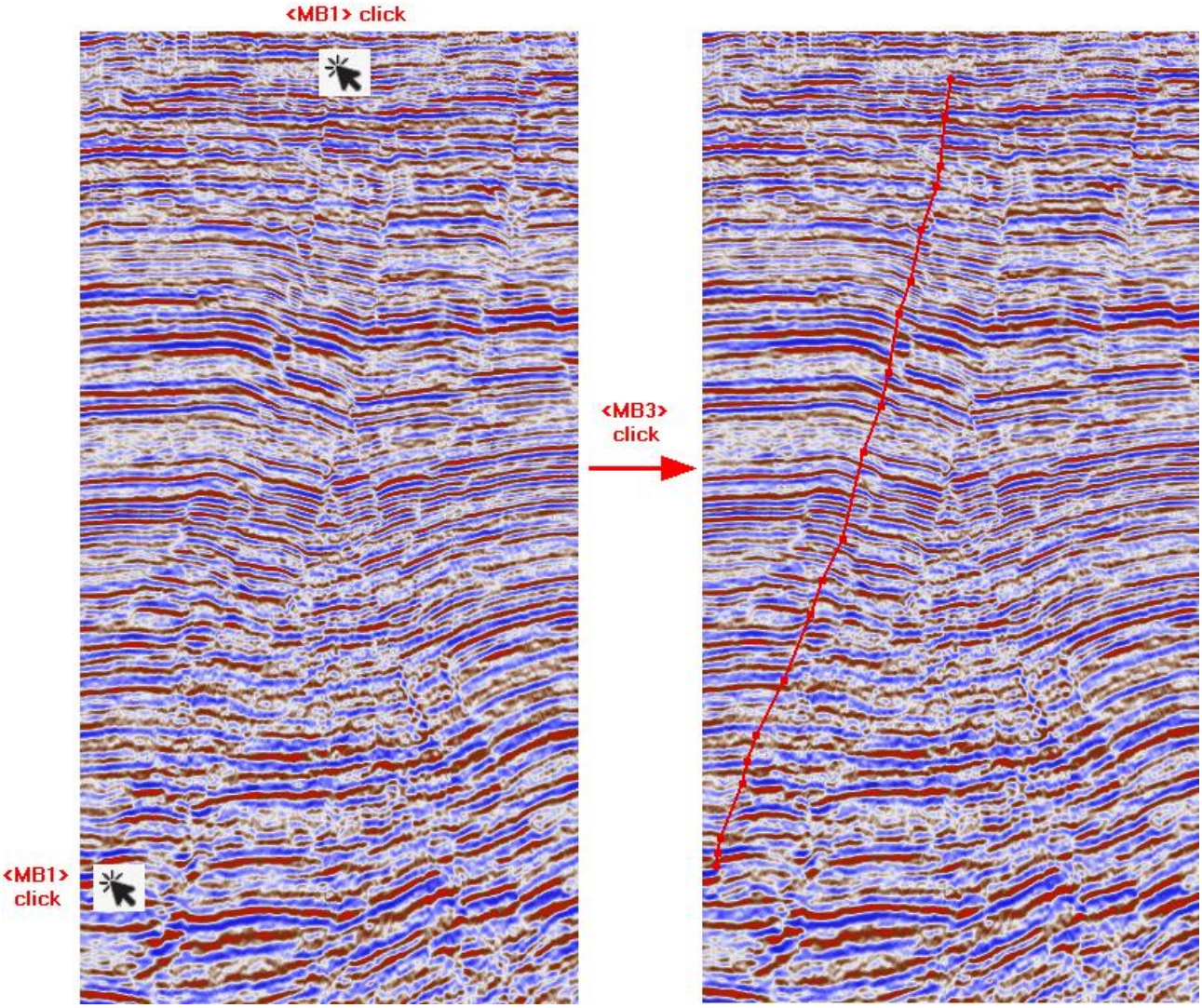
Step 1: Set appropriate control parameters:



Step 2: Enter Feature tracking mode:

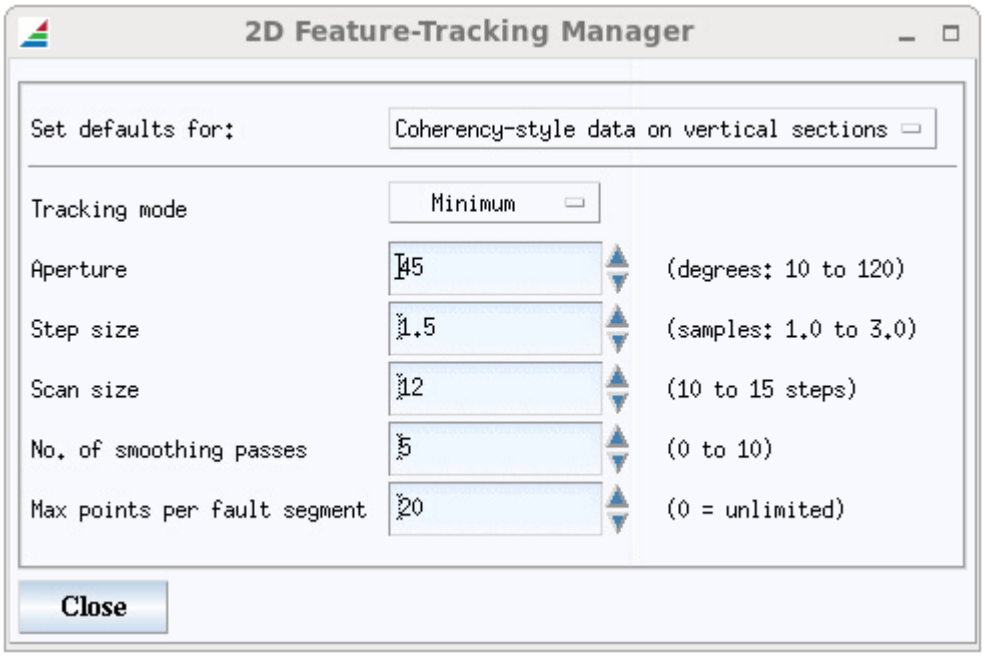


Step 3: Pick fault segments



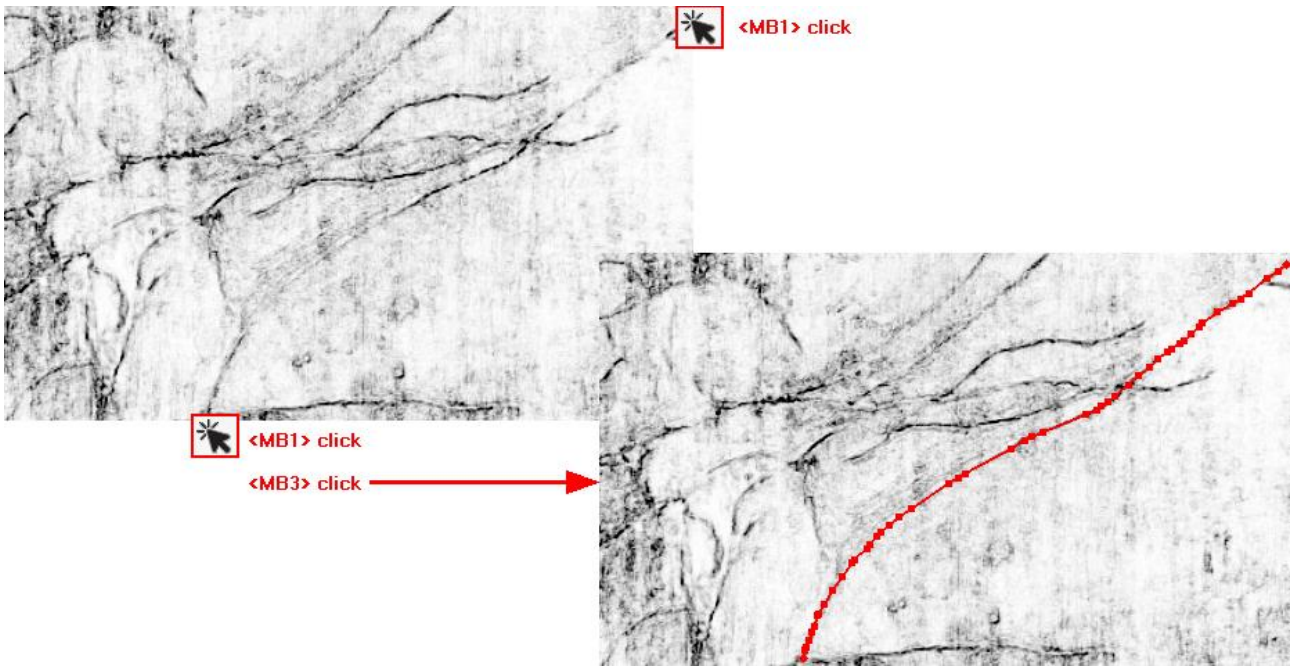
Example 2: Feature-tracking through coherency-style data on time-slices.

Step 1: Set appropriate control parameters:



Step 2: Enter Feature tracking mode: (as shown in Example 1)

Step 3: Pick fault segments

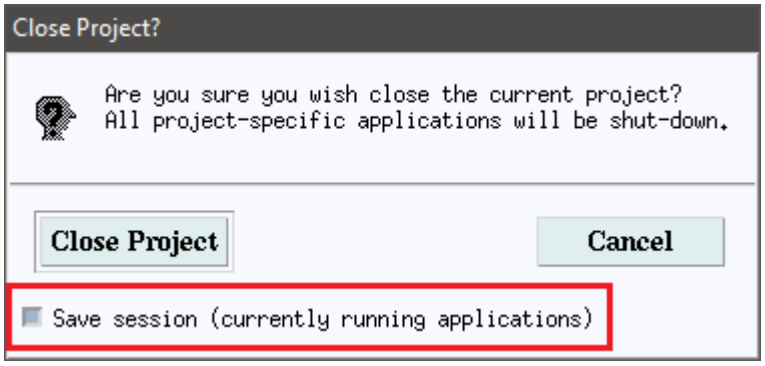


Project Sessions

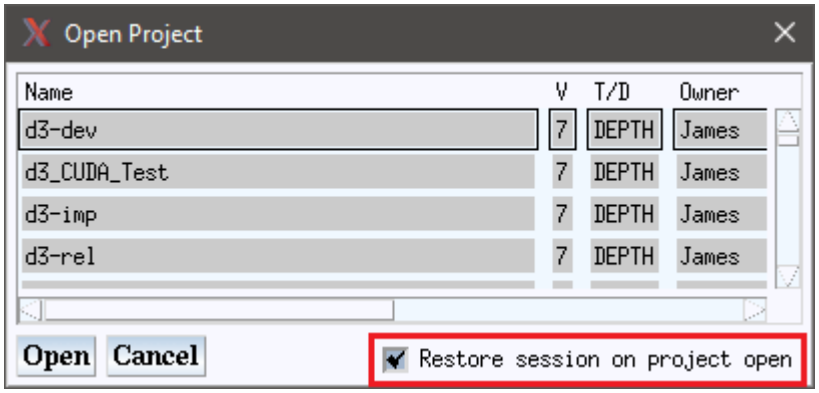
This feature allows the state of currently running applications to be saved during project close and restored during a subsequent project open.

All T7 applications will preserve their window size and location. In addition, most applications also preserve their internal state or session.

When closing a T7 project, a checkbox is available to specify if the currently running applications should be preserved:



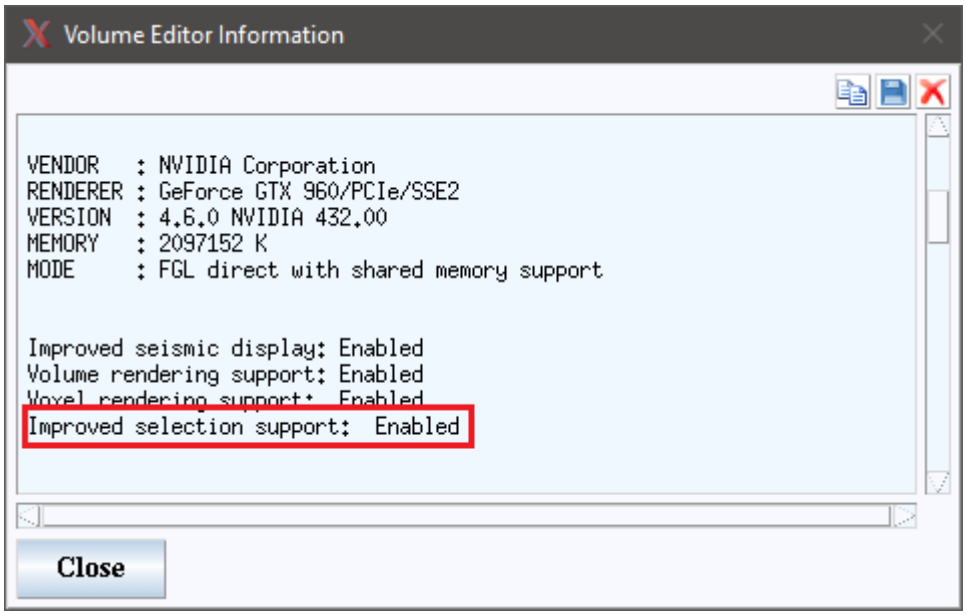
When opening a T7 project, a similar checkbox is available to specify if the saved session should be restored (*ie.* applications re-launched at their saved location and state):



Improved Volume Editor Selection System

Available for the Windows MKS and Linux version of T7, a replacement selection system has been implemented that can significantly reduce the delay when selecting complex objects (typically containing millions of points) in Volume Editor viewers.

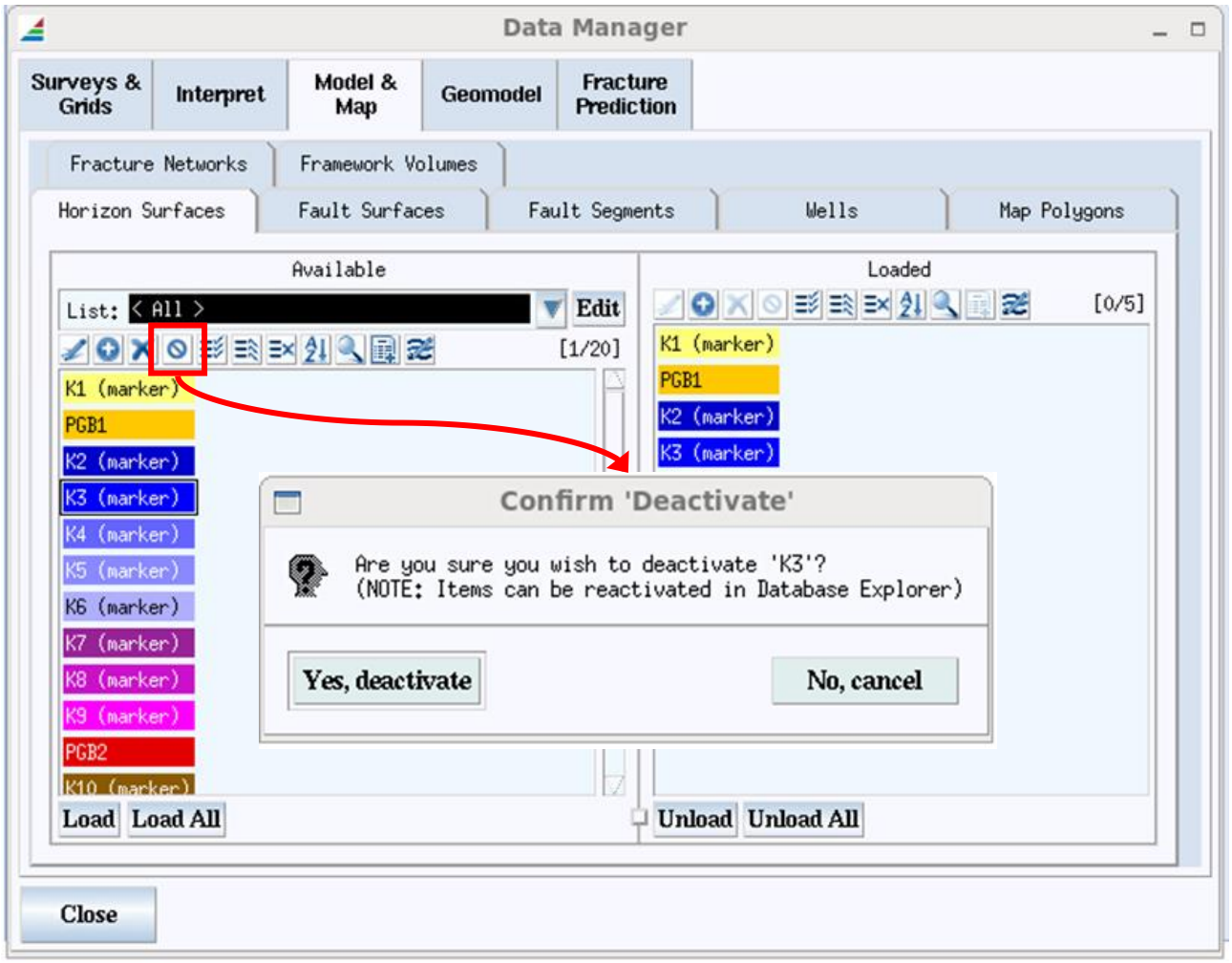
The improved selection system is enabled by default. This can be verified by checking the Information window upon starting Volume Editor:



Improved Access to Object Deactivation

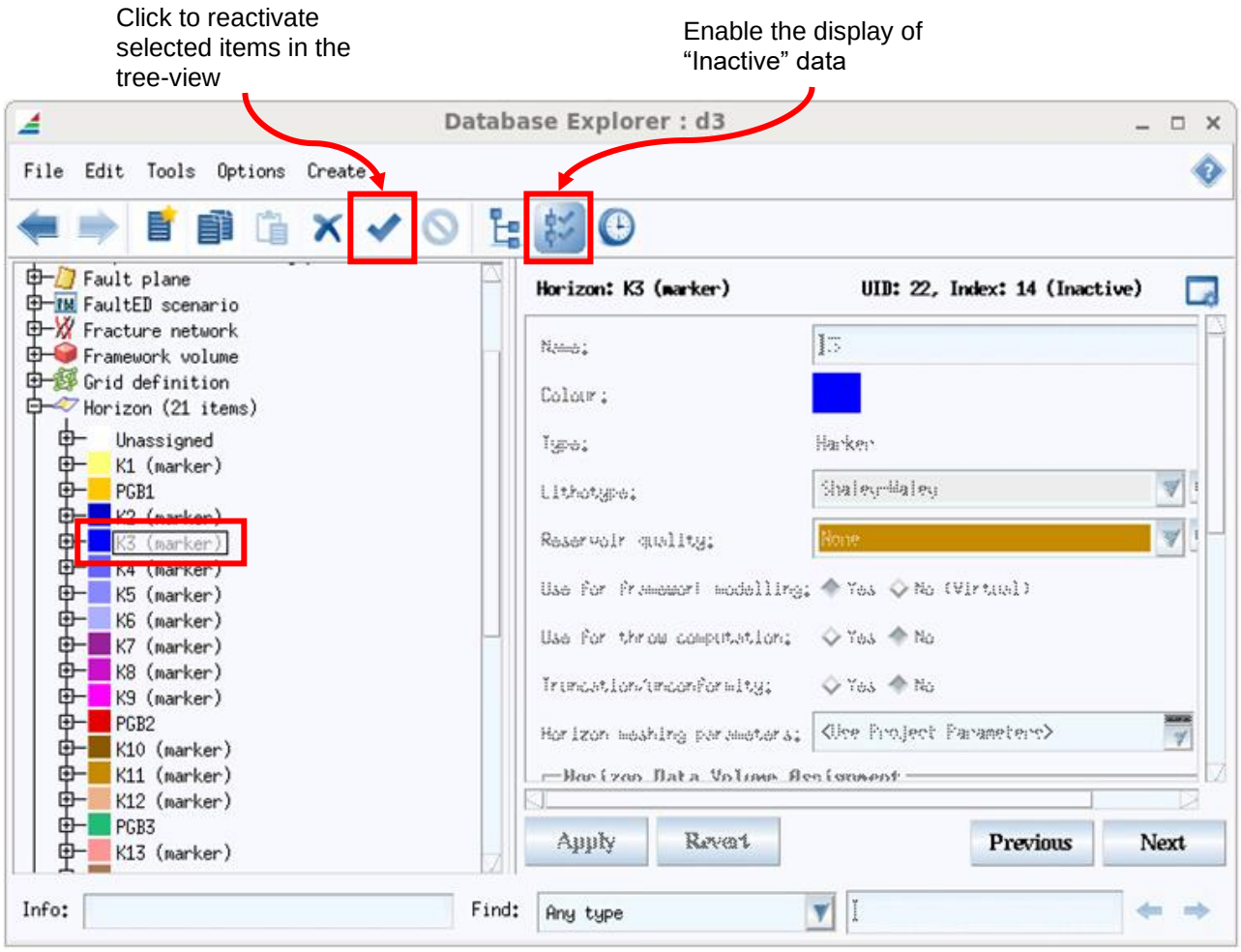
T7 supports the ability to deactivate and reactivate database items. Deactivating a database item is much like temporarily deleting it. Object deactivation and reactivation was, until now only possible in the Database Explorer. It is now possible to deactivate database items in any T7 application that manages database objects in list-views. Once an object is deactivated, it can only be reactivated in the Database Explorer (where it can still be seen).

The image below shows the horizon lists in the Volume Editor's Data Manager window.



The left hand list shows the horizon, "K3 (marker)" is selected. The list's toolbar has a new icon (also available in the MB3-popup menu) which can be used to deactivate the selected item(s). Clicking the option to deactivate will pop-up a confirmation window as shown above.

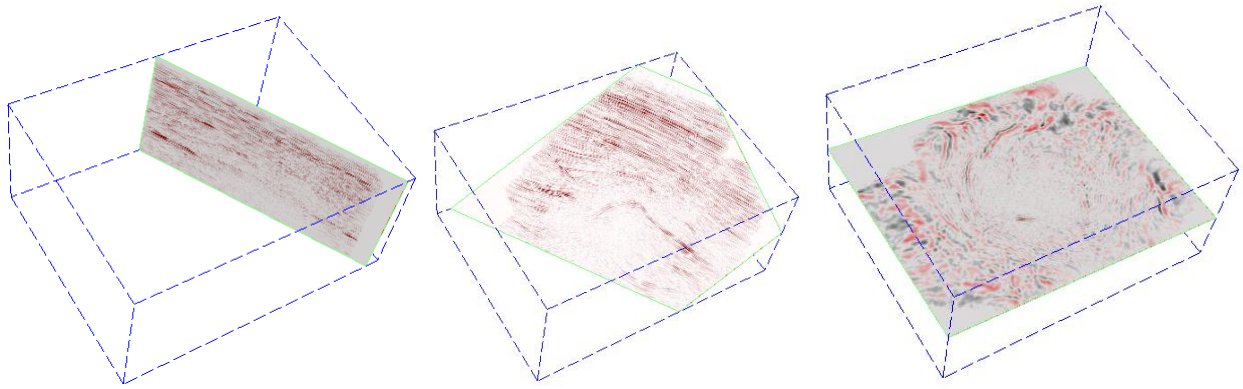
Once the selected items have been deactivated it will be removed from all T7 applications together with any database items that belong to it (eg Well Horizon Picks, Horizon Data Volumes, Horizon Surface etc in the above case). The only application where the deactivated items can be managed is in the Database Explorer (as below).



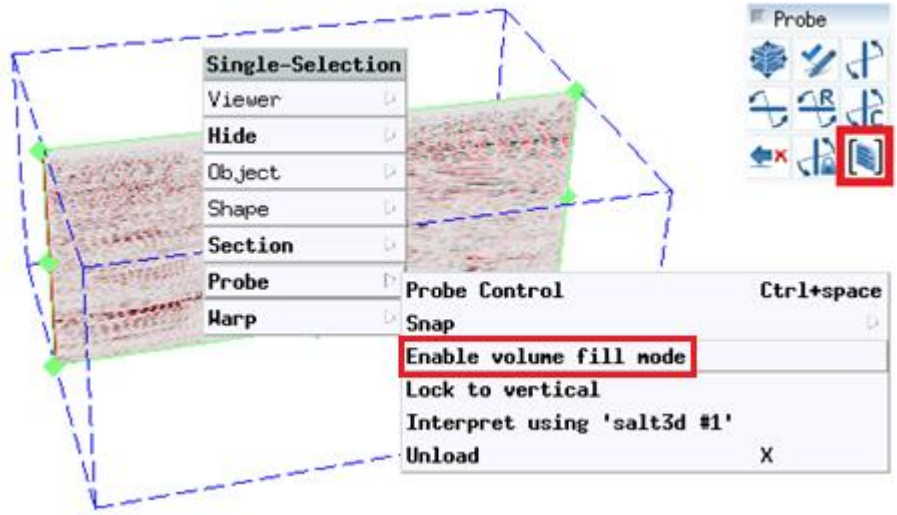
With the option set to "Display inactive objects" in the Database Explorer, the inactive items will appear greyed-out (but still selectable) in the tree view. Here they may be selected and reactivated using the tool-bar icon or the *MB3-pop-up-menu->Activate* option.

Planar probe "volume fill" mode

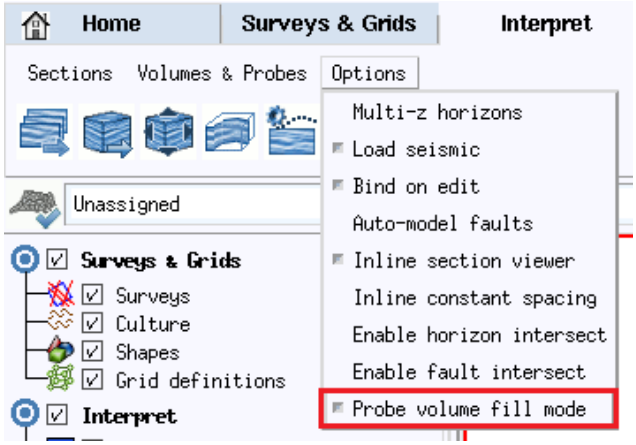
Volume Editor Planar probes now support a simplified "volume fill" mode of operation. Probes in this mode will fill to the extent of the volume, regardless of orientation:



Volume fill mode can be toggled for an existing planar probe via a new context menu item, and associated toolbar icon:

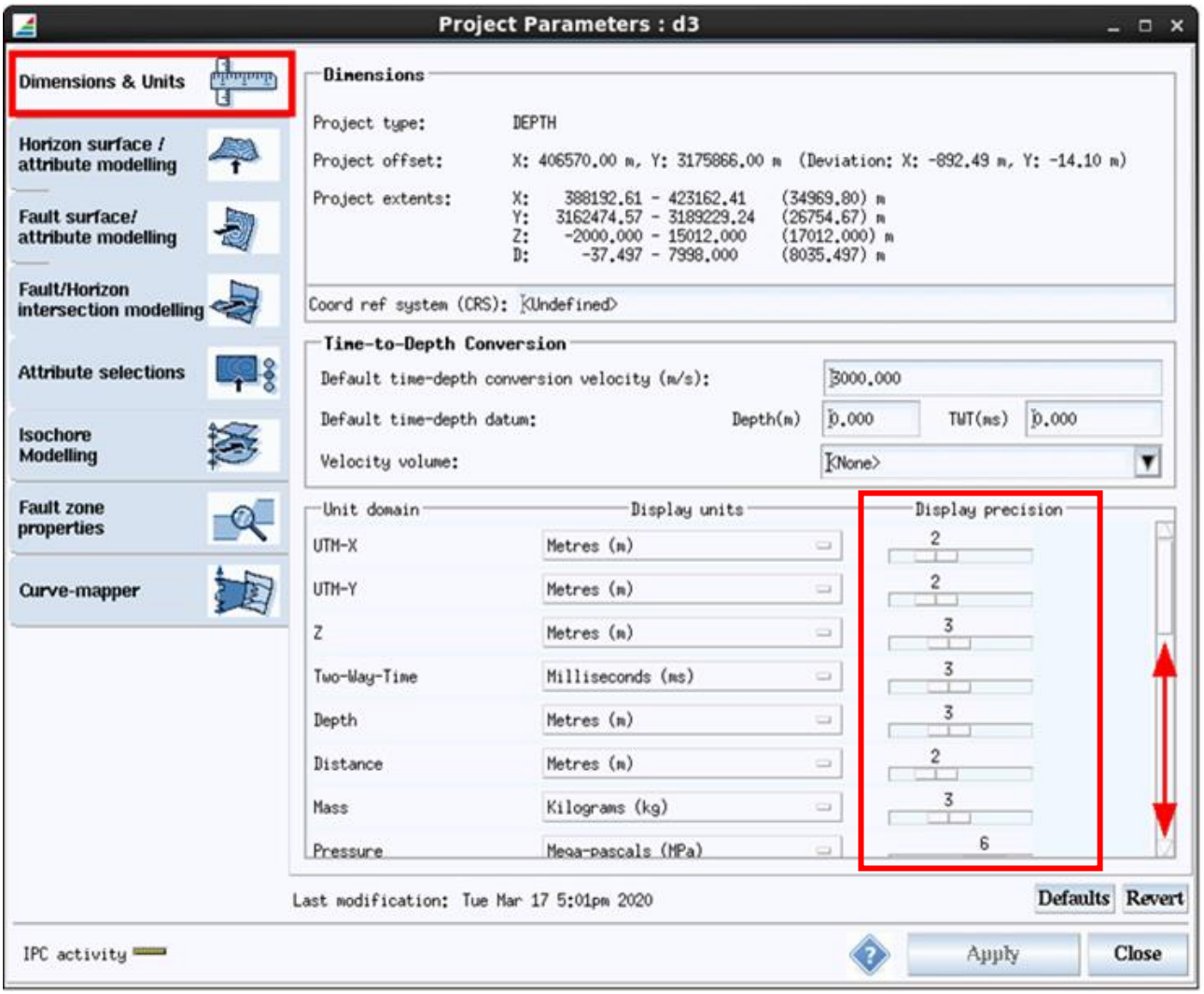


Volume fill mode can be specified as a project-level default for newly created planar probes, via a new Interpret module Options menu entry:



User-Defined Units Precision Settings

The Project Parameters Dimensions & Units settings now has additional options to control the precision level with which numerical data is displayed in T7 applications. Prior to the 7.110 update, the precision level was fixed at a set value for each supported unit domain.



The new “Display Precision” controls are found in the form of sliders to the right of the “Display Units” settings for each unit domain.

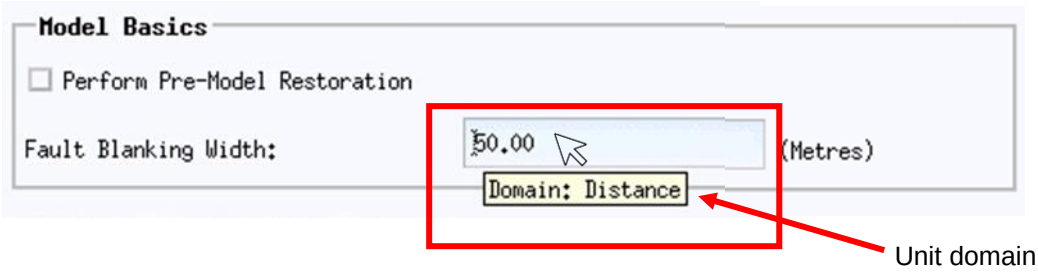
Each slider controls the level of precision used when data values in that domain are presented for display within T7 applications. The precision value will act as a "number of decimal places" for displayed values in the range 1.0 to 100,000,000. Beyond this range the precision will operate as a "number of significant figures" - see the examples below.

Example display values for a precision of 2:

Actual Value	Displayed Value	Precision mode used
0.0000123456789	1.2e-05	2 Significant Figures
0.000123456789	0.00012	2 Significant Figures
0.0123456789	0.012	2 Significant Figures
1.23456789	1.23	2 Decimal Places
123.456789	123.45	2 Decimal Places
123456789.1234	123456789.12	2 Decimal Places
1234567891.234	1.2e+09	2 Significant Figures

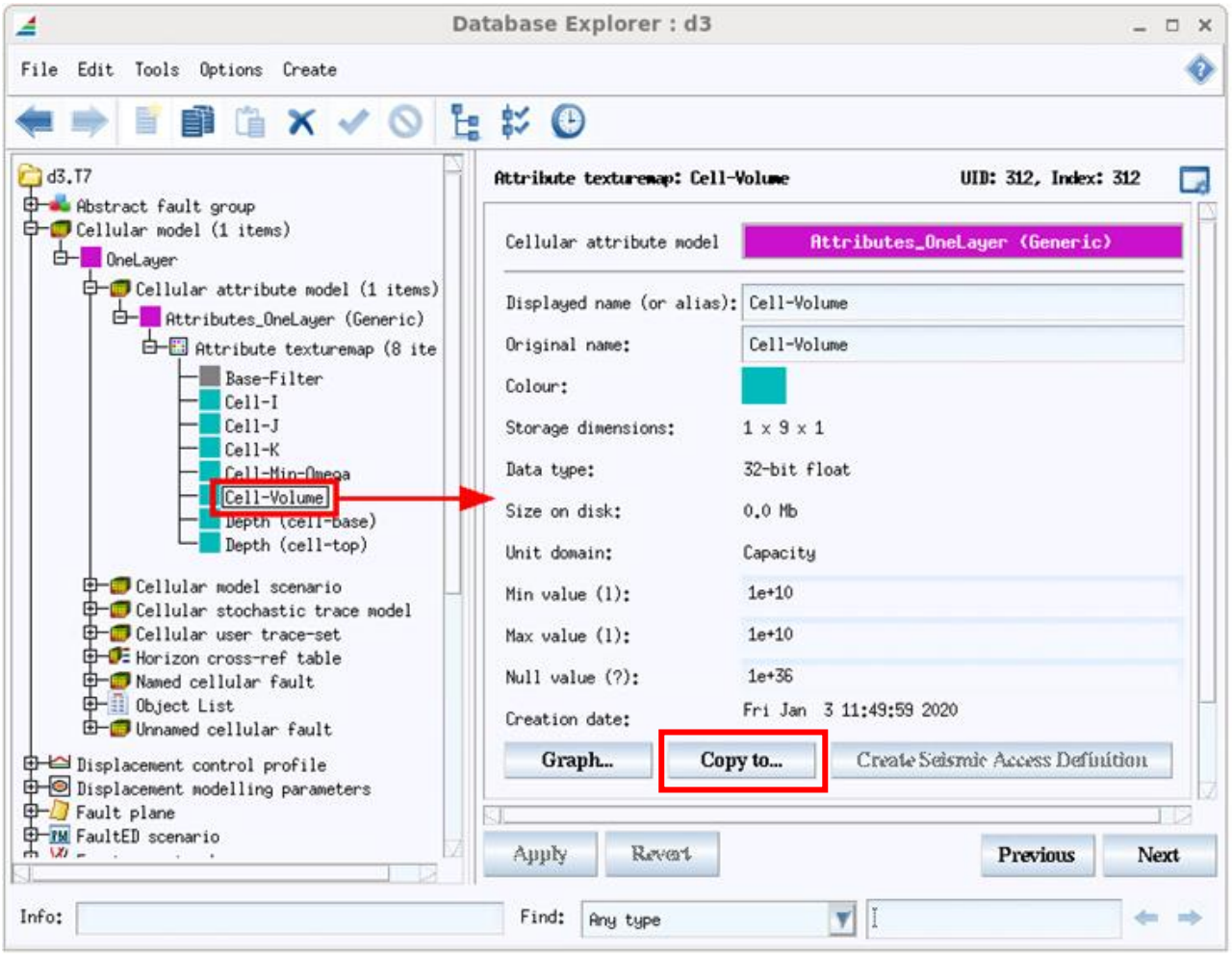
Unit Domain Fly-Outs

Many text-boxes in T7 user-interfaces that display domain data now provide a small fly-out, as the mouse-pointer is moved over, indicating the unit domain to which the value belongs. An example is shown below…

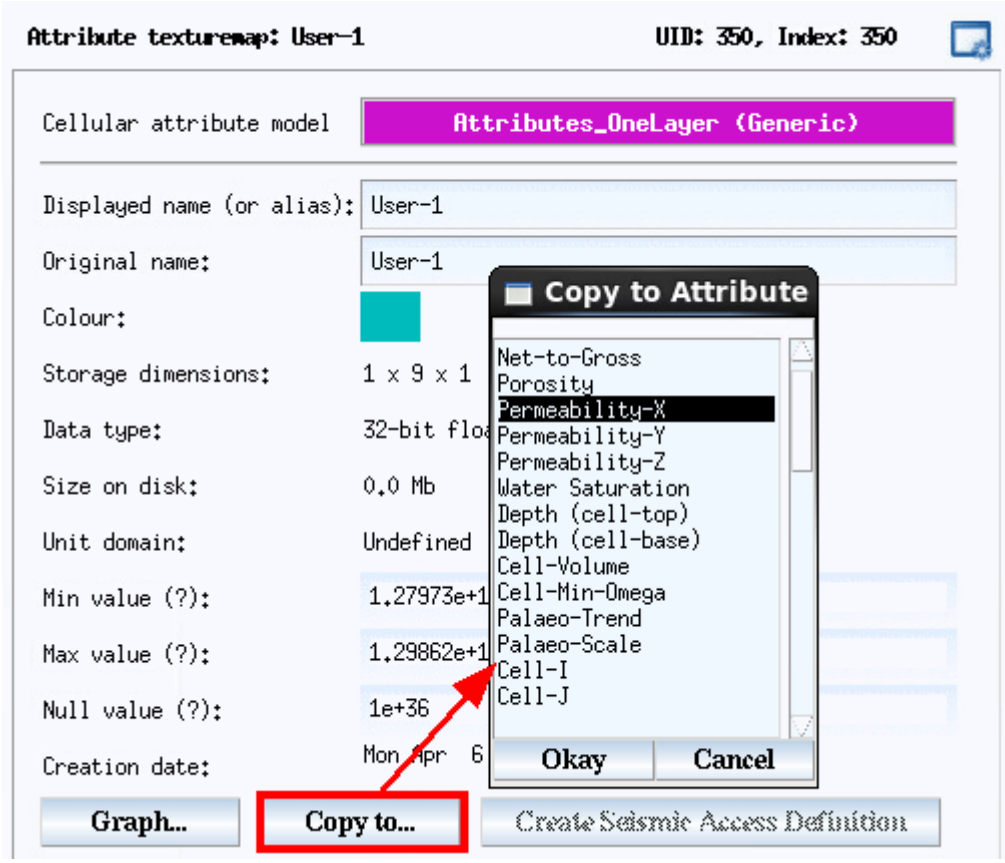


Attribute Texturemap Copy Function

The Attribute Texturemap data represents any of the various T7 "Task-based" attributes that may be created in or, in some cases, imported to a T7 project. These attributes form the core of many T7 data-analysis, modelling and visualisation work-flows (eg, Throw on a Fault Surface, or Cell Volume for a Cellular Grid. A description of the Attribute Texturemap data can be accessed from the Database Explorer's tree view as shown in the example image below.



A new "**Copy to…**" function has been added to the Attribute Texturemap editor window.



Click on the **Copy to...** button to open a list of potential target attributes. The list will only show attributes that have the same Data type as the attribute being copied (for example, it is not permitted to copy an integer attribute to a floating point attribute). Select an attribute from the list and *click* on **Okay** to complete the process or **Cancel** to abort.

While it is possible to use the Database Explorer's Copy and Paste functions to copy an **Attribute Texturemap** from one host to another - this does not permit the copying to a different attribute under the same host. The **Copy to..** option provides this latter functionality.

The chosen target attribute can be one that already exists or one that has not yet been created/imported.

Additional User-Attributes

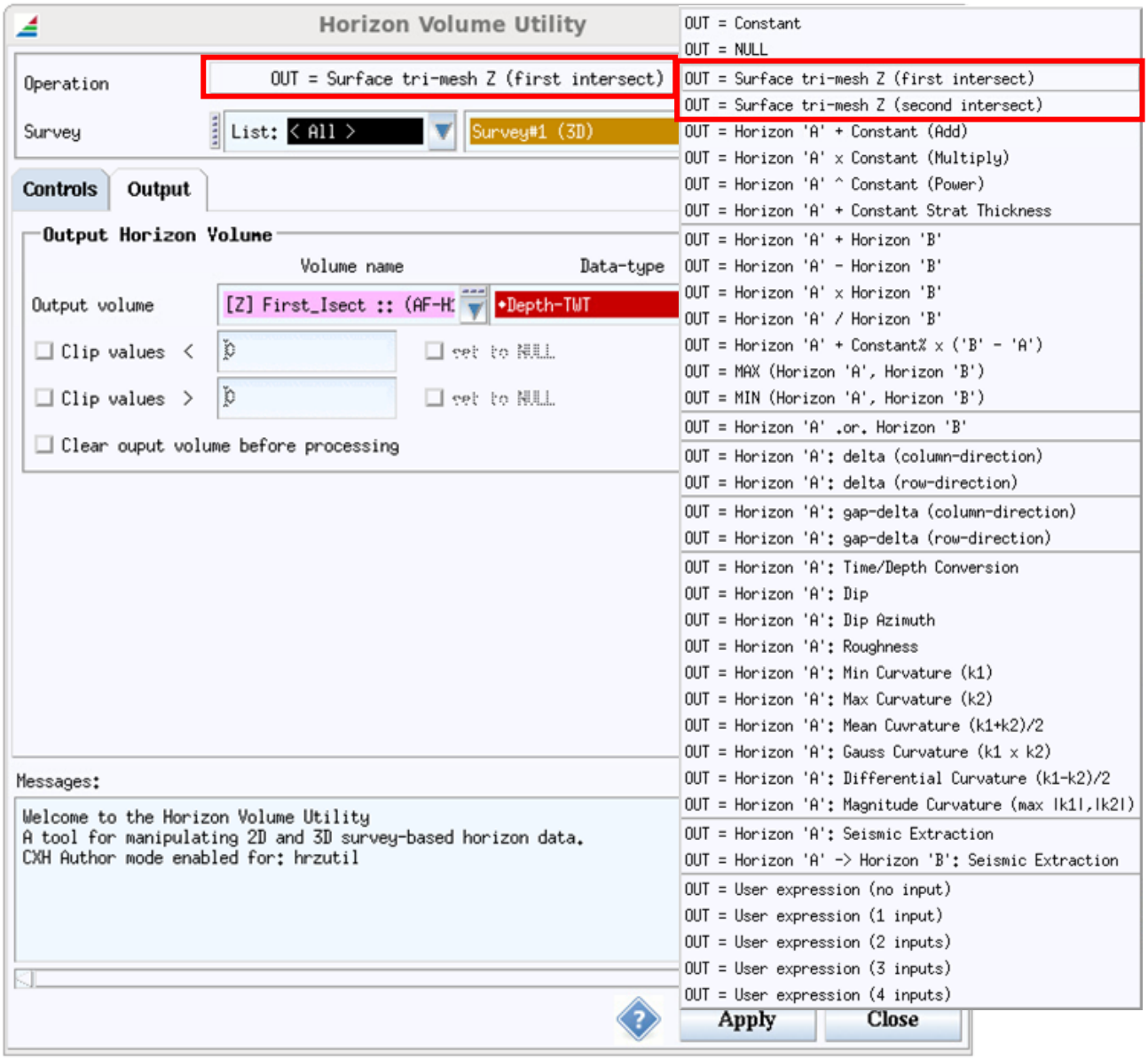
Provision has been made for additional user-attributes for several of the Task-based subsystems in T7 (ie Fault, Horizon, Cell-Grid etc). These user-attributes can be used in combination with the Attribute Calculator or the new Attribute Texturemap “Copy to” function (described above) to store multiple versions of a given attribute. The user-attributes can be given alternative names and unit domains using the Attribute Alias tool (accessed from *T7 Control Menu -> Utilities -> Attribute Aliases*) if required. The table below shows the new user-attribute slots that have been made available.

Task	**New User Attributes**
Fault	User-1 … User-16
Horizon	User-9 … User-16
Frac-Net	User-1 … User-16
Cell-Grid	User-9 … User-32

All of the user-attributes can be used in the general Task-based tools such as the Display Method system, Plot-Viewer, Attribute Export, Attribute Calculator etc.

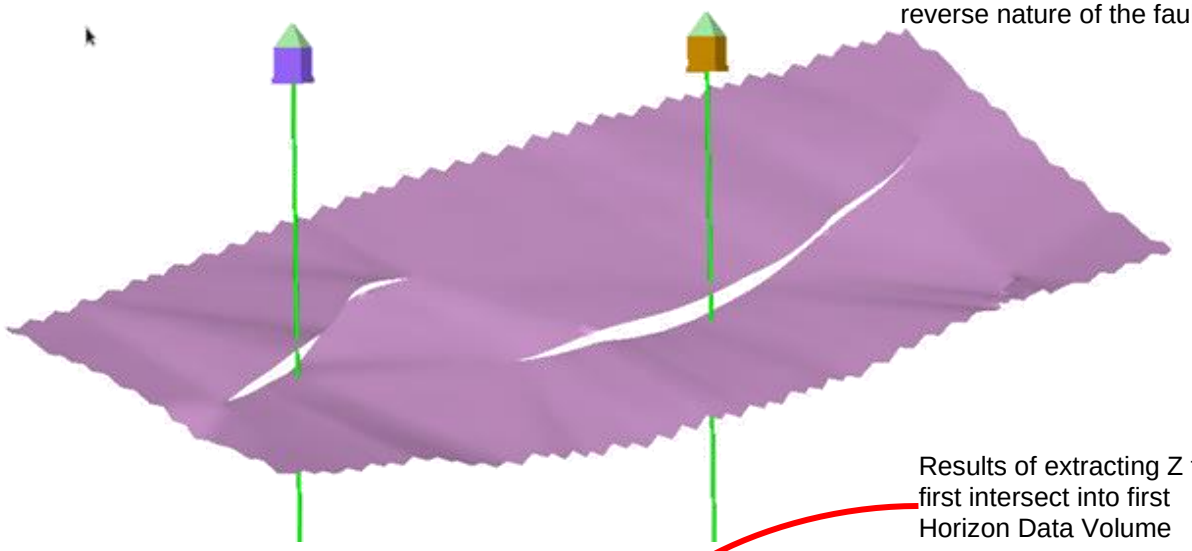
Horizon Volume Utility Tri-Mesh Reverse Structure Z-Extraction

The Horizon Volume Utility provides many ways to create and modify Horizon Data Volumes. One of the available processes allows the time or depth values (Z data) in a Horizon Data Volume to be extracted from a surface tri-mesh belonging to the same horizon. This process provides a means of generating horizon Z data than can be displayed and edited on sections (as interpretation data) when the original horizon data was only available in the form of a tri-mesh. If the tri-mesh contained reverse structure, the extracted Z-data would only define the upper-most surface at any given XY location. In order to improve the handling of reverse structure cases, there are now two tri-mesh Z-extraction processes available in the Horizon Volume Utility: (i) obtains the first (uppermost) Z value (first intersect), (ii) obtains the second Z value (second intersect). These two independent processes permit the user to extract the first intersection results to one Horizon Data Volume and the second intersection results to a second – thus preserving the reverse structure in a form that may be displayed and edited on sections.



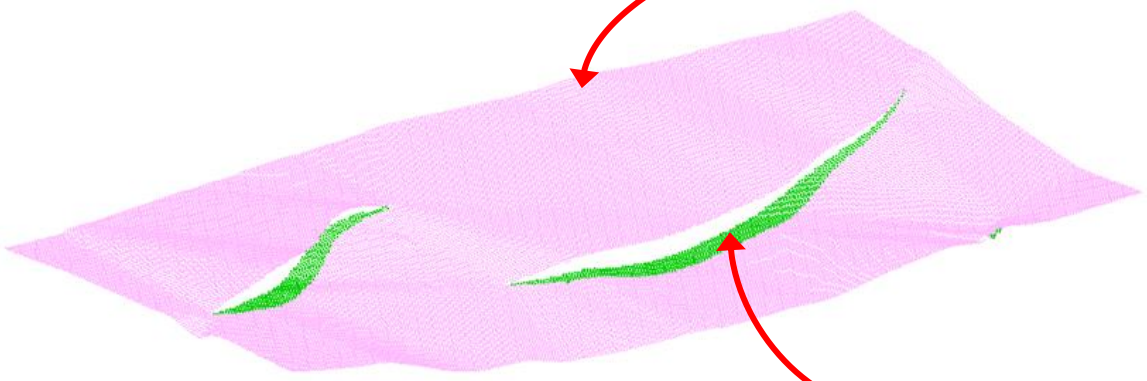
Horizon Volume Utility Window

The Images below illustrate the results of using both the first and second intersection processes into separate Horizon Data Volumes.



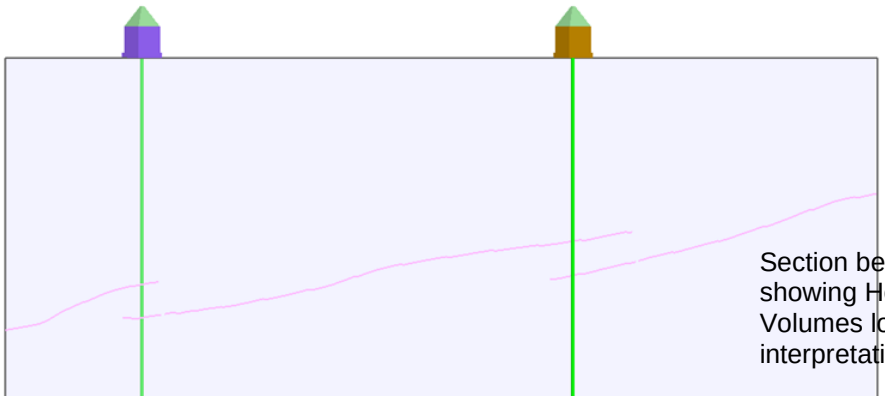
Tri-mesh with reverse-faulted structure. Two wells through the faults help illustrate the reverse nature of the faults

Results of extracting Z for first intersect into first Horizon Data Volume



or

second Horizon Data Volume (highlighted in green for clarity)

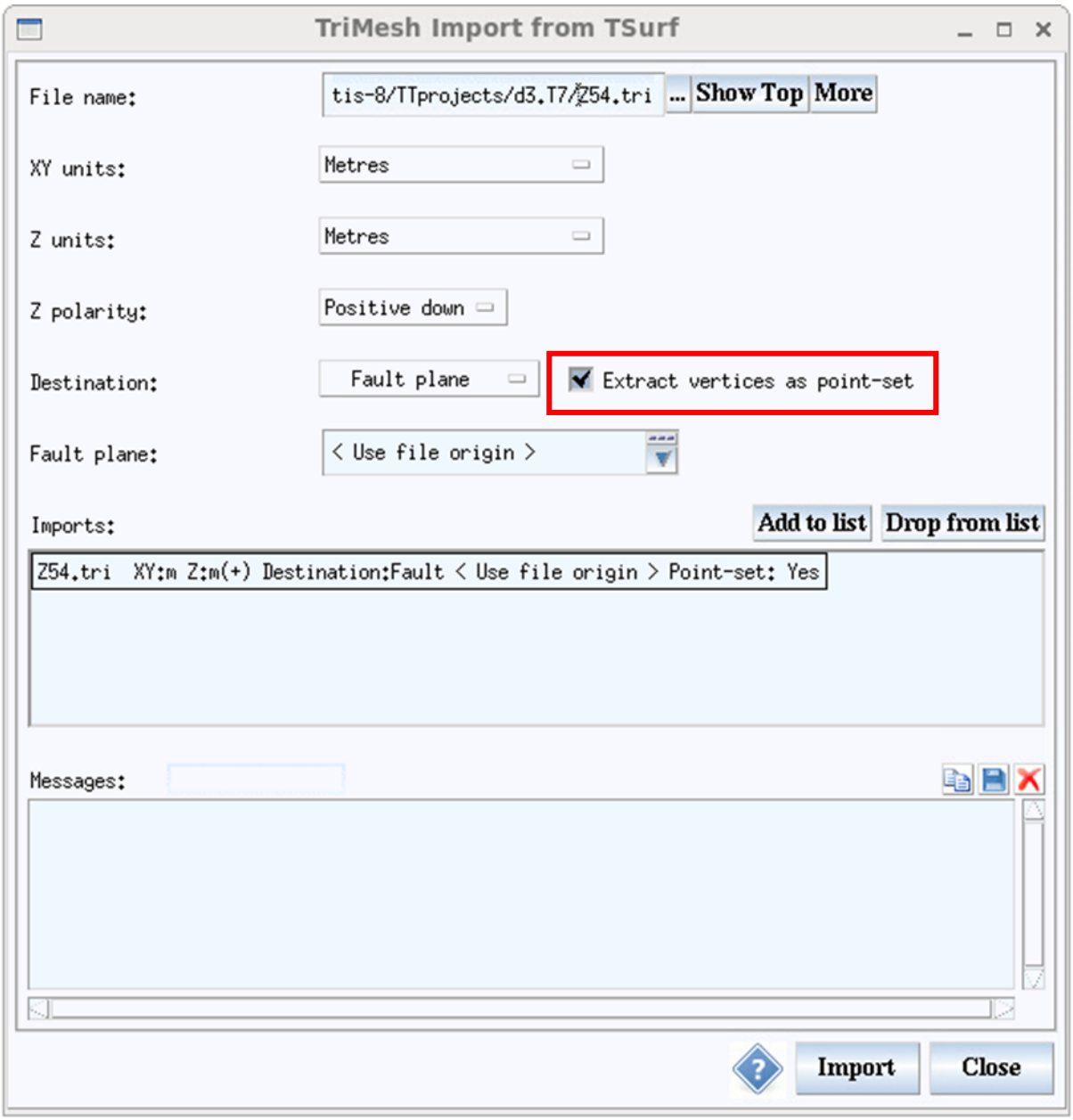


Section between both wells showing Horizon Data Volumes loaded as interpretation

GoCad Tri-Mesh Import Point-Set Option

The Gocad tri-mesh import tool (accessed from the ASCII import options under the T7 Control menu or the Database Explorer) has been provided with an option to enable or disable the creation of a point-set containing the vertices of the imported tri-mesh. This option is only available and relevant when importing a tri-mesh into a Horizon or Fault. Prior to this upgrade, the GoCad tri-mesh import (for Horizons or Faults) would operate such that the point-set would always be created, replacing what might already exist.

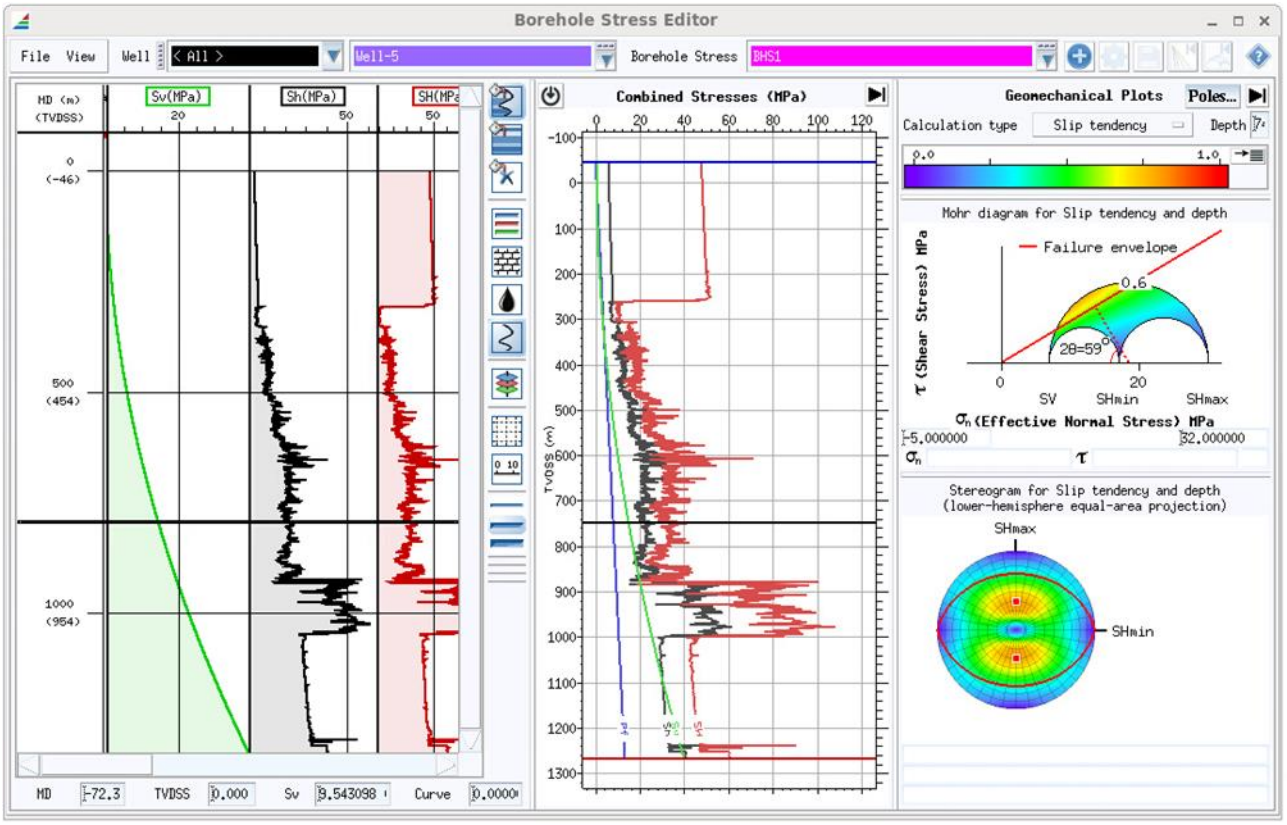
The default setting for the option is ON mirroring the earlier functionality. Very often, when importing tri-meshes for Faults and Horizons, it is not necessary to create the point-set, so this option can now be switched off if required. Point-set data can be imported independently using the ASCII point-set import tool.



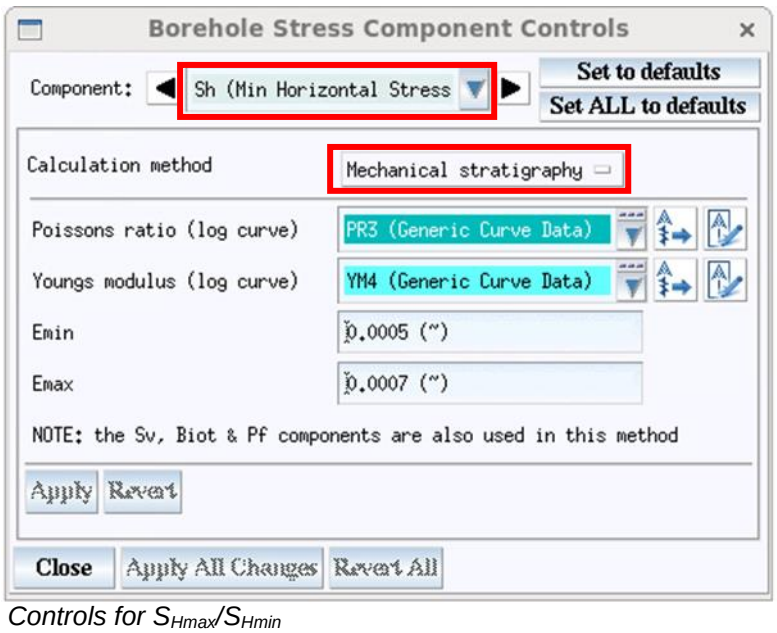
GoCad Tri-mesh Import Tool

Borehole Stress Editor New SH-min/SH-max Calculation Option

The Borehole Stress Editor, Introduced to T7 in the 7.1 release with the 3D in-situ stress analysis system, has been enhanced with new methods of calculating the minimum and maximum horizontal stress profiles (S_{Hmin} and S_{Hmax}) for a given well from mechanical stratigraphy properties. These new calculation methods provide a means of deriving the S_{Hmin} and S_{Hmax} profiles from standard equations that use existing pore pressure and Biot's constant profiles together with well curve data for Poisson's ratio and Young's modulus.



Borehole Stress Editor Window



Component set to modify the S_{Hmin} controls

Calculation method set to "Mechanical Stratigraphy"

Controls for S_{Hmax}/S_{Hmin}

The standard formulae for S_{Hmin} and S_{Hmax} (for example, Thiercelin & Plumb 1994) are given below.

$$S_{Hmin} = \frac{\nu_{stat}}{(1-\nu_{stat})}\left(S_v - \alpha P_p\right) + \frac{\mathrm{E}_{stat}}{(1-\nu_{stat}^2)}(\varepsilon_{min} + \nu_{stat}\varepsilon_{max}) + \alpha P_p$$

$$S_{Hmax} = \frac{\nu_{stat}}{(1-\nu_{stat})}\left(S_v - \alpha P_p\right) + \frac{\mathrm{E}_{stat}}{(1-\nu_{stat}^2)}(\varepsilon_{max} + \nu_{stat}\varepsilon_{min}) + \alpha P_p$$

Where:

S_v = Vertical Stress
ν_{stat} = static Poisson's Ratio
α = Biot's coefficient
P_p = formation pressure
E_{stat} = static Young's modulus
ε_{min} = strain in direction of min. horizontal stress
ε_{max} = strain in direction of max. horizontal stress

With the "Calculation Method" set to "Mechanical stratigraphy", the above expressions are used with the specified input curves to compute the S_{Hmin} and/or S_{Hmax} profiles.

(Thiercelin, M. J., & Plumb, R. A., 1994. A Core-Based Prediction of Lithologic Stress Contrasts in East Texas Formations. Society of Petroleum Engineers. doi:10.2118/21847-PA)

Picking Fault Polygons on Horizons

The ability to pick fault polygons (hanging wall and footwall cut-offs) in the Volume Editor has been extended to be available for horizons – that is, horizon surfaces (tri-mesh data), horizon data volumes (2D/3D interpretation data) and horizon point-sets. Previously, the availability of this functionality was restricted to fault surfaces – ie the polygon could only be picked on a fault surface.

Picking a hanging wall or footwall polygon on a horizon will associate the polygon with that horizon and the currently Active Fault in the Volume Editor – though this fault need not be loaded, and it need not possess a fault surface. In this way, it is possible to create fault polygons on a horizon for display and export purposes without the existence of a fault surface model. These polygons can be used to remove horizon data that fall within them.