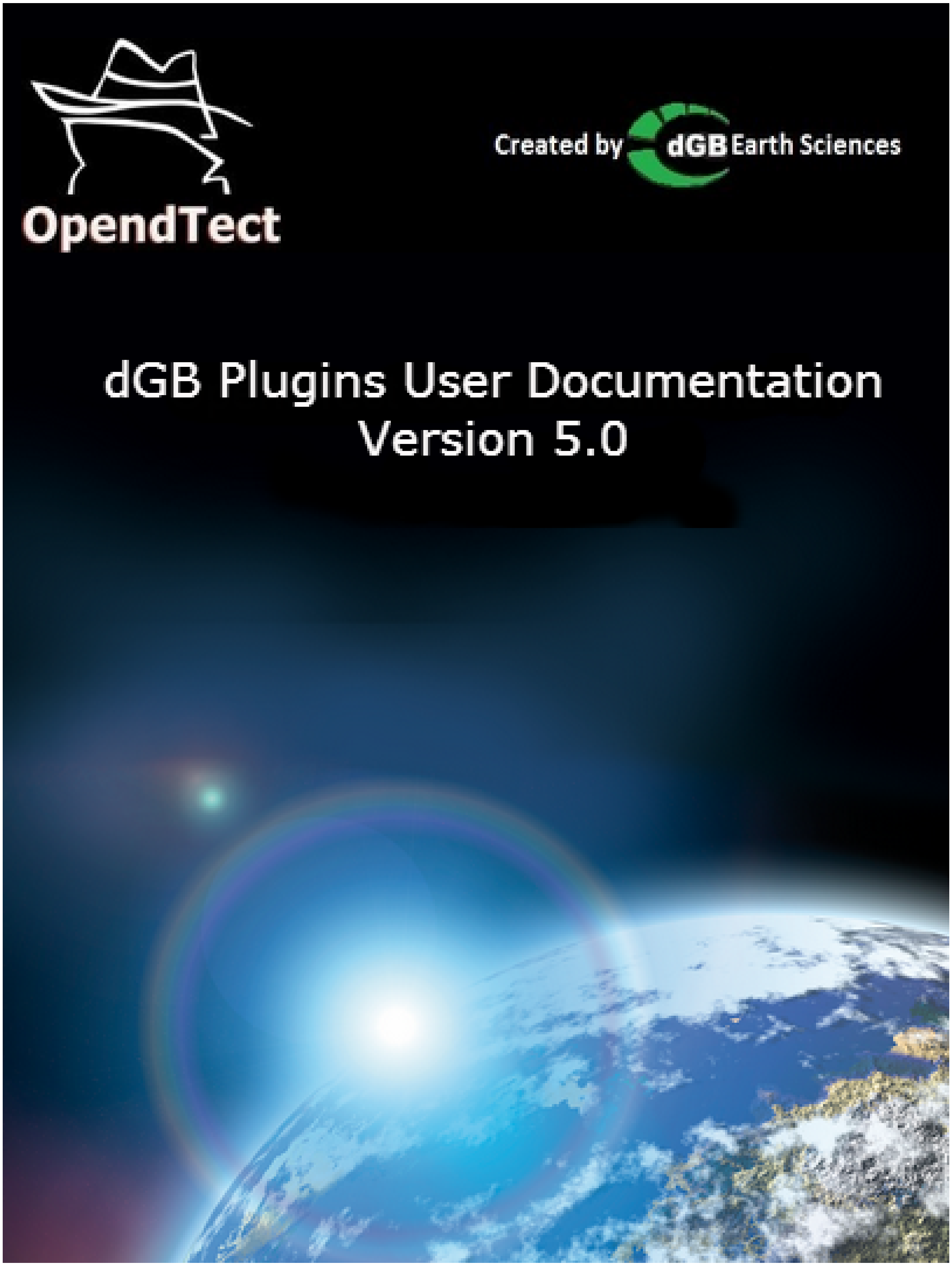


OpendTect

Created by dGB Earth Sciences

dGB Plugins User Documentation
Version 5.0

Table of Contents

Table of Contents 2

OpendTect DGB Plugins User Documentation - Version 5.0 8

Dip-Steering 12

- **Background** 12
- **Create Steering Data** 15
  - **Import Steering Data** 19
  - **Create SteeringCube** 20
- **Filter** 24
- **Display SteeringCube** 25
- **Attributes with Steering** 25
  - **Curvature Analysis** 26
    - **Volumetric Curvature** 28
    - **Horizon-Based Curvature** 29
  - **Dip** 30
  - **Dip Angle** 31
  - **Position** 32
  - **Reference Shift** 34
  - **Similarity** 35
  - **Volume Statistics** 38
  - **Perpendicular Dip Extractor** 39
- **Benchmark SteeringCube Creation** 39
  - **Speed vs Algorithm and Calculation Cube Size** 39
  - **Visual Quality Check** 40
  - **Crossline Dip Attribute** 41
  - **Filtering of the SteeringCubes** 42
  - **Steered Similarity Attribute** 44
  - **Choosing a Steering Algorithm** 45

HorizonCube 46

- **Introduction** 47
- **Processing** 49
  - **HorizonCube Control Center** 49
  - **Data Preparation** 50
    - **Horizons - Check Crossings** 50

- Horizons - Fill Holes, Gridding 52
- Horizon - Trim at Faults 53
- Create 2D Seismic Lattice 56
- Filtering the SteeringCube 58
- Create Horizons from a SteeringCube 59
- Edit Horizons with a SteeringCube 62
- Calculate HorizonCube Attributes 62

Create HorizonCube (2D, 3D) 64
- Model Driven Settings 68
- Data Driven Settings 68
  - Advanced Options 69
  - Continuous Events 71
  - Truncated Events 73

2D HorizonCube Processing 74
- Preparing the Framework 76
- Processing 2D HorizonCube 79
- Correlate 2D HorizonCube Regions 83

Tools 84
- HorizonCube Editor 84
- Add More Iterations 89
- Merge HorizonCubes 90
- Add Packages, Recalculate 3D Sequences 91
- Extract Horizons 94
- Convert HorizonCube to SteeringCube 95
- Truncate HorizonCube 96
- Get Continuous HorizonCube 97
- Grid HorizonCube 98

HorizonCube Well Log Interpolator 99

Analysis 101
- Display Properties for HorizonCube 2D, 3D 101
- HorizonCube Attributes 105
- 3D Slider 110

Import, Export 114
- Manage HorizonCube 114
- Preload HorizonCube 116

- ASCII Export (3D) 117
- Import HorizonCube Service Data 118

Sequence Stratigraphic Interpretation System 119

- Introduction 119
  - SSIS Toolbar 121
- Interpretation Window 122
  - Overview 122
  - Select, Define a Depositional Model 126
  - Interpretation Workflow 127
  - 2D Interpretation Window 133
  - Display System Tracts 135
  - SSIS Display Settings 137
  - Create Statistical Curves 138
- HorizonCube Sider 139
- Wheeler Transform, Wheeler Scene 140
  - Wheeler Scene 140
  - Create Wheeler Output (2D, 3D) 142
- Flatten Horizon, Seismics 144
  - Flatten 144
  - Unflatten HorizonCube 147
- System Tracts Attributes 148
- Manual SSIS 150

Well Correlation Panel 151

- Introduction 151
- WCP Main Window 152
- Correlation Displays and Settings 157
- Pick Markers and Correlate 163

SynthRock 165

- Introduction 165
- Volume Synthetics 167
- Stochastic Pseudowell Modeling 171
  - Add New Modeling Node 172
  - Model Definition 178
    - Random Distribution 181
    - PDF Distribution 182

        - Math-based Layer Properties 182
    - Analysis of the Existing Wells 185
    - Well-Log - based Generation 193
    - Manage layer Properties 197
    - Export Synthetic Dataset 202
  - Profile Modeling 206
    - Simple Wedge Model/Property Variations Using one Well 209
    - Model an existing line through well(s) 216
    - Put Together a Structural Situation from Scratch 227
    - Add, Edit Existing Well 248
      - Define Property Changes 251
    - Add, Edit Control Profile 254
    - Use Horizon 264
    - Set Process Parameters 269
    - Dump Model to File 272
  - Fuid Replacement 274
    - Define Fluid Contents 274
    - Fluid Replacement Parameters 275
  - HitCube Stochastic Inversion 281
    - Input Data and Inversion Type 284
    - HitCube Analysis 285
      - HitCube Parameters QC Evaluation 287
      - Define Output Layer Properties 288
    - Output Data - Inversion Batch Processing 289
- Neural Networks **291**
  - Introduction 291
    - Supervised Neural Networks 291
    - Unsupervised Neural Networks 293
  - Manage Neural Networks 294
    - Data Manager 294
    - Neural Network Management 295
    - Neural Network Information 297
      - Supervised Neural Network Information 297
      - Unsupervised Neural Network Information 298
    - Import GDI Networks Window 301

- Pattern Recognition (using Picksets) 302
- Property Prediction (using Well Logs) 307
  - Balance Data 310
  - NN Lithology Codes 311
- Training and Application 313
  - Unsupervised Training 313
    - Quick UVQ 314
    - Quality-based UVQ Stacking 316
  - Supervised Training from Picksets 318
  - Supervised Training from Well Data 319
  - Application 321

Velocity Model Building 324

- Introduction 324
- Vertical Velocity Analysis 325
- Horizon-based Velocity Update 339
- Input-Output 345
  - Pre-stack Event Import 346
  - Pre-stack Events Export 347
- Velocity Display 348
- Velocity Correction 349
- VMB Specific Gridding Step - Gridding of Velocity Picks 351
- VMB Specific Gridding Step - Surface-limited Filler 353

Fluid Contact Finder (formerly CCB) 355

- Introduction 355
- FCF Main Window 356
- FCF Analysis 357
- Local FCF Attribute 362

Applications 364

- How to Make the ChimneyCube 364
  - Workflow 364
  - Picking Example Locations 365
  - Neural Network Training 366
  - Evaluation and Application of the Trained Neural Network 367
- Dip-Steered Median Filter 368
  - Example Results 370

Create a Dip-Steered Median Filter 371
Note 372
Default Attribute Sets 373
Evaluate Attributes 374
dGB Evaluate Attributes 375
NN Chimney Cube 375
NN Fault Cube 378
NN Fault Cube Advanced 381
NN Salt Cube 382
NN Slump Cube 384
Unsupervised Waveform Segmentation 386
Ridge-enhancement Filter 388
Dip-Steered Median Filter 391
Dip-Steered Diffusion Filter 391
Fault Enhancement Filter 392
Fault Enhancement Attributes 393
Seismic Filters Median Diffusion Fault Enhancement 393
Licenses 395
Install demo/node-locked license 396
Clear License Installation 397
Show HostID 398
References 400
Appendix A - Synthetic Data Generation 402
Glossary 411
Index 424

- OpendTect is a complete Open Source seismic interpretation system, created by **dGB Earth Sciences B.V.**, that is released under a triple licensing scheme: Open Source, Commercial and Academic.
- Under the Commercial and Academic License Agreements OpendTect can be extended with closed source plugins for added functionality: HorizonCube, Well Correlation Panel, SSIS (Sequence Stratigraphic Interpretation System) plugin, Dip-Steering, Neural Network and other plugins, such as FCF (formerly CCB), MPSI, SSB, SCI, WA and others (see: **http://dgbes.com/index.php/products**).
- Directive attributes, filters, and Neural Networks, combined with modern visualization techniques, enabling the OpendTect user to illuminate information and features which would otherwise remain hidden using conventional methods. No real expertise is required to process and interpret TheChimneyCube, TheFaultCube, or any other Geologic Object Cube, such as Formations, Salt Sequences, Flat Spots, Bright Spots, 4D anomalies, etc.
- These plugins deliver enormous additional functionality to OpendTect:
- ***HorizonCube***: Game-changing plugin that auto-tracks a dense set of horizons (e.g. a seismic event every 4ms). HorizonCube impacts all aspects of seismic interpretation work and allows the interpreter to extract more geology from the data. HorizonCube is used for: detailed geologic model building, improving seismic inversion, sequence stratigraphic interpretation (SSIS) and correlating wells (Well Correlation Panel plugin).
- ***Well Correlation Panel*** *(WCP)*: an interactive viewer to pick well markers and correlate these from well-to-well using seismic data to guide the correlations. Provided

the user has access to the HorizonCube, the HorizonCube slider can be used for detailed seismic-steered correlations.

- ***Dip-Steering***: allows the user to create and use "steering cubes". A steering cube contains the local dip and azimuth of the seismic events at every sample position. The cube is used for:

  a) structurally oriented filtering (e.g. dip-steered median filter)

  b) improving multi-trace attributes by extracting attribute input along reflectors (e.g. dip-steered similarity)

  c) to calculate some unique attributes (e.g. Dip & Azimuth, 3D-Curvature, and Variance of the dip).

- ***Sequence Stratigraphy Interpretation System*** *(SSIS)* is an add-on to the HorizonCube. SSIS supports Wheeler transformations and sequence stratigraphic (systems tracts) interpretations based on chrono-stratigraphic horizons from the HorizonCube. The Wheeler transformation is one of the key features of SSIS Plugin. A Wheeler transformation is a flattening of seismic data according to the calculated chrono-stratigraphy. In the Wheeler domain we see when (in relative geologic time) and where (spatially) events were deposited, how the depositional center shifted over time (basin-wards or landwards) and how events are related in time. Gaps in the Wheeler domain are caused either by erosion or non-deposition

- ***SynthRock*** A powerful toolkit for creating and using forward models in qualitative and quantitative seismic interpretation studies by integrating Rock Physics, Geology and Seismic data SynthRock makes full use of the power of OpendTect to support a range of cutting edge modeling and inversion work flows. The following functions are supported:

  1. Rock-physics library (Castagna, Krief, Garder, Biot-Gassmann, etc.) and possibility to define any rock physics formula using math & logic.

  2. Pre-stack synthetics; PP, PS, optionally with multiples; Near, mid, far, full, angle, AVO attributes etc.

  3. Profile module: create cross-sections from existing wells with

manual updates of model parameters.

4. Stochastic module: stochastically varying pseudo-wells.

5. Inversion possibilities: HIT Cube (cross-matching procedure to create probability volumes), Probability Density Functions (derived from cross-plots), and Neural Networks (non-linear approach to predict rock property volumes)

- ***Fluid Contact Finder*** *(FCF)* is used to detect subtle hydrocarbon-related seismic anomalies and to pin-point gas-water, gas-oil, and oil-water contacts.FCF uses the power of stacking to enhance such anomalies. If we stack all traces along the same contour line, we can expect the hydrocarbon effect to stack up while stratigraphic variations and random noise will be canceled out.
- ***Neural Networks*** Both Supervised and Unsupervised NN allow generation of meta-attribute volumes that highlight any object of interest (Chimney, Salt, Faults, etc.).
- ***Volume Model Builder*** *(VMB)*: Uses the travel time of the acoustic waves to image the subsurface.
- ***Seismic Spectral Blueing*** *(SSB)*: A technique that shapes the seismic spectrum to optimize the vertical resolution without boosting noise to an unacceptable level. The spectrum is reshaped to 'match' observed behavior of reflectivity data obtained from wells. In a global sense, the well-reflectivity spectrum shows that the higher the frequency the higher the amplitude. We refer to this as the spectrum being blued.
- ***Seismic Coloured Inversion*** *(SCI*: enables rapid band-limited inversion of seismic data. A single convolution inversion operator is derived which optimally inverts the data. The spectrum of the inverted data honours the available well data spectra in a global sense. Generally, traditional inversion methods (e.g. sparse-spike) are time consuming, expensive, require specialists and are not performed routinely by the Interpretation Geophysicist, whereas SCI is rapid, easy to use, inexpensive, robust and does not require expert users. SCI and unconstrained sparse-spike appear to give broadly equivalent results.
- ***Workstation Access*** *(WA)*: Based on the Ideal toolkit from ARK CLS . It supports direct import and export of seismic volumes, horizons and well data to and from Landmark's SeisWorks and Schlumberger's GeoFrame-IESX workstations. The

Workstation Access plugin supports connection with GeoFrame v4.2 and v4.3 on Linux and connection with v4.0.4, v4.0.4.1, v4.0.4.2, v4.2 and v4.3 on Solaris.....

- ***MultiPoint Stochastic Inversion*** *(MPSI)*: Developed by Earthworks and ArkCls is considered the fastest stochastic inversion scheme currently on the market. MPSI is implemented as a series of five attributes in OpendTect's attribute engine. The attributes include: *Model builder, Error grid, Deterministic inversion, Stochastic inversion,* and *utilities*
- ***Seismic Net Pay*** *(SNP)* and ***Seismic Feature Enhancement*** *(SFE)*: Plugins by ARKCLS for the estimation of net pay thickness or net reservoir thickness and the enhancement of seismic feature, repectively.
- ***Computer Log Analysis Software*** *(CLAS)*: plug-in, developed by Geoinfo and through which well log petrophysics can be performed within OpendTect rather than having to be imported, will result in improved well-to-seismic ties, the enhanced calibration of seismic attributes to reservoir properties, more robust models and the more accurate interpretation of 3D seismic data. Key features of the CLAS plug-in within OpendTect include: Simple and advanced LAS format data import including well parameters and run information. The ability to carry out log calculations, such as temperature curve generation, porosity calculation from cross plots, sonic log editing, and shale volume calculations. Sophisticated water saturation analysis. And the ability to generate accurate petrophysics-based reports.

Dip-Steering

Table of Contents

- Background
- Create Steering Data
- Attributes with steering
- Benchmark Steering Cube Creation

Background

Dip-Steering is defined as the process of following seismic reflectors by auto-tracking the pre-calculated dip-field from a given starting position. The pre-calculated dip-field in OpendTect is stored in a Steering-Cube, which contains two values at each seismic sample position: the dip in the inline direction and the dip in the cross-line direction.

Dip-Steering is a key concept in OpendTect that is used to:

• Compute dip-steered multi-trace attributes. Examples are: Similarity, Texture, Volume Statistics, … Also Neural Network-based 'probability' volumes such as ChimneyCubes and FaultCubes include many dip-steered attributes.

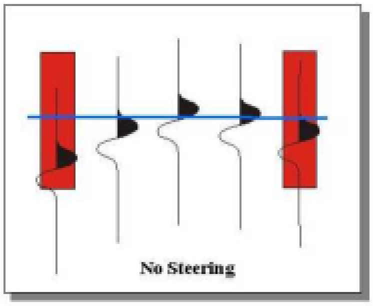
• Compute attributes directly from the Steering-Cube. Examples are: the set of Volume Curvature attributes and (geologic) dip and azimuth.

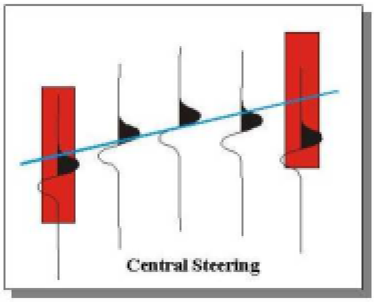
• Apply dip-steered filters. Examples are: dip-steered median filter and dip-steered fault-enhancement filter.

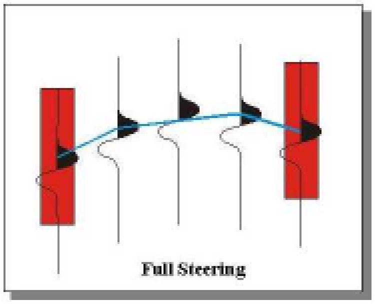
- Auto-track horizons. Examples are the HorizonCube auto-tracker which tracks thousands of horizons directly from the dip-field and OpendTect’s conventional amplitude & similarity tracker that can optionally include dip to constrain the tracker.
- Grid loosely mapped horizons. The dip-steered gridding algorithm fills in holes and unmapped parts to generate a horizon that exists at every seismic location.

Depending on the process it is often useful to work with more than one Steering-Cube. Typically, we use two Steering Cubes. The ‘Detailed Steering Cube’ contains dips as calculated by the dip-computation algorithm (OpendTect supports multiple algorithms, see …). The Detailed Steering Cube is used if we need to preserve details in the data, e.g. when computing Volume Curvature attributes in fracture detection studies. The ‘Background Steering Cube’ is a smoothed (usually median-filtered) version of the Detailed Steering Cube. The Background Steering Cube contains less noise and the dips follow larger structural trends. It is typically favoured for dip-steered filtering operations.

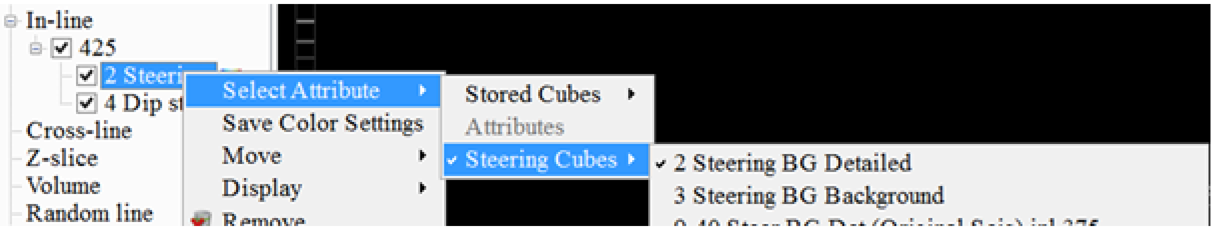
The Dip-Steering plugin for OpendTect supports two different modes of data-driven steering: Central Steering and Full Steering. In Central steering the dip / azimuth at the evaluation point is followed to find all trace segments needed to compute the attribute response / filter / track the event. In Full steering the dip/azimuth is updated at every trace position. The difference between 'no steering', 'central steering' and 'full steering' is shown in the following figures. Note that these figures show the 2D equivalent of steering, which in actual fact is a 3D operation.



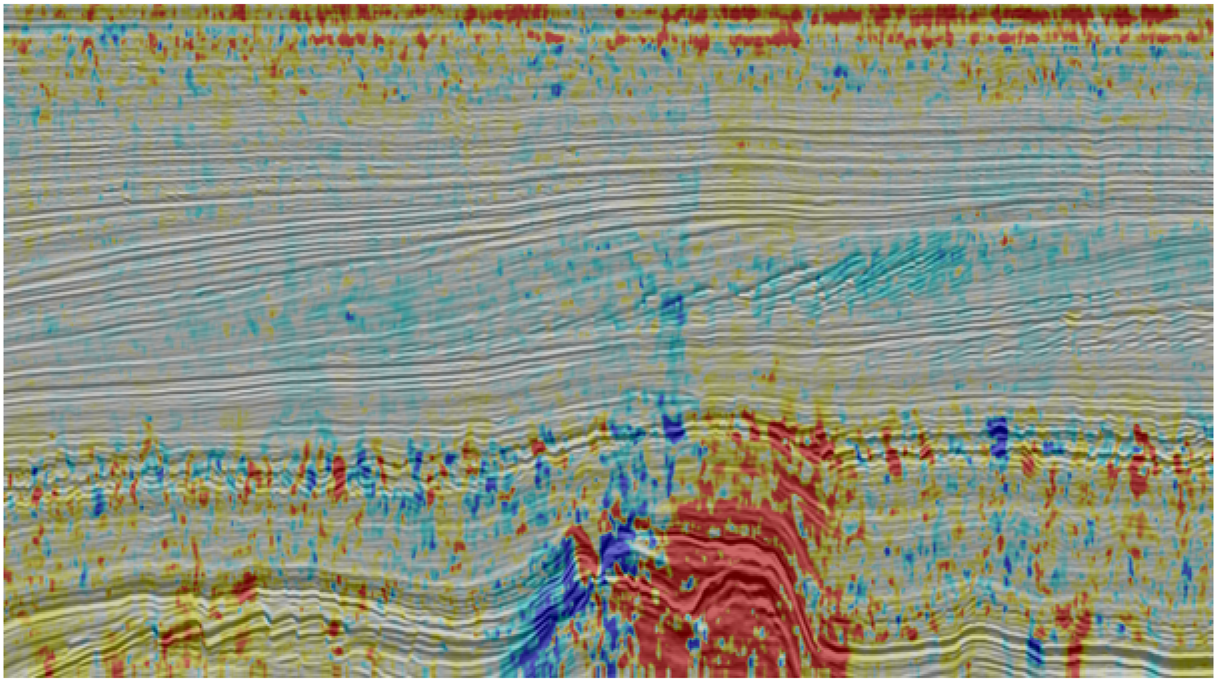




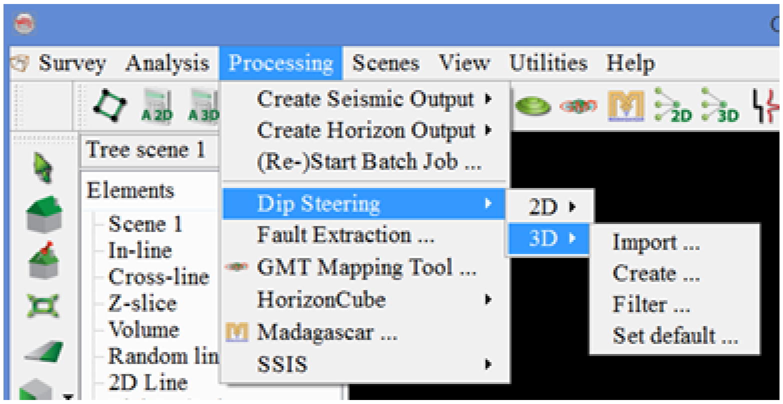
Steering Cube information can be displayed in a 3D scene directly from the OpendTect tree, as shown in the picture below. This option displays the cross-line dip along an inline, or the inline dip along a cross-line.



Detailed Steering Cube of inline 425 in F3-Demo. For display purposes the (cross-line) dip image was made (50%) transparent.



Dip-Steering options are launched from the Processing -> Dip Steering Menu. Dip-Steering is available for 2D seismic and 3D seismic. A Steering Cube for a 2D line stores at every sample position the dip in the line direction. A Steering Cube for a 3D volume stores two values per sample position: the dip in the inline direction and the dip in the cross-line direction.



Note: the windows pictured and described in this manual are the ones pertaining to 3D processing. 2D processing windows are similar and the same description can therefore be used for 2D dip-steering.

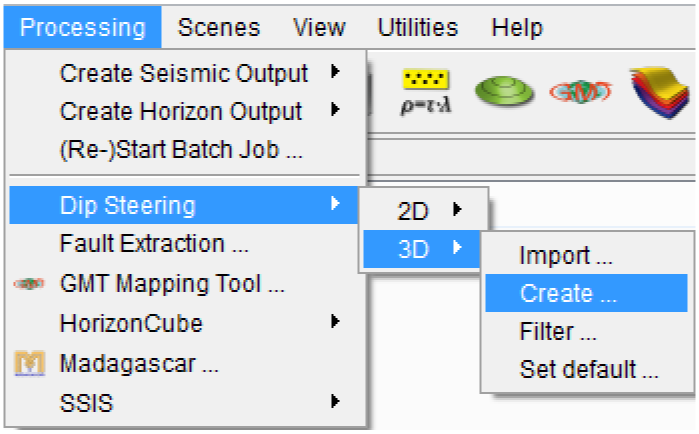
References:

The following references are available from the dGB website. Although slightly out of date, the concepts are still valid:

- Benchmark of HorizonCube algorithms: http://dgbes.com/images/PDF/od_steering_benchmark.pdf
- Introduction to Steering Cubes: http://dgbes.com/images/PDF/Introduction_to_SteeringCube.pdf
- Steering Cube work flows: http://dgbes.com/images/PDF/effectivedipsteeringworkflowusingbgsteering_primerodata.pdf

Create Steering Data

The processing main menu is use to start (2D/3D)-SteeringCube creation and filtering.

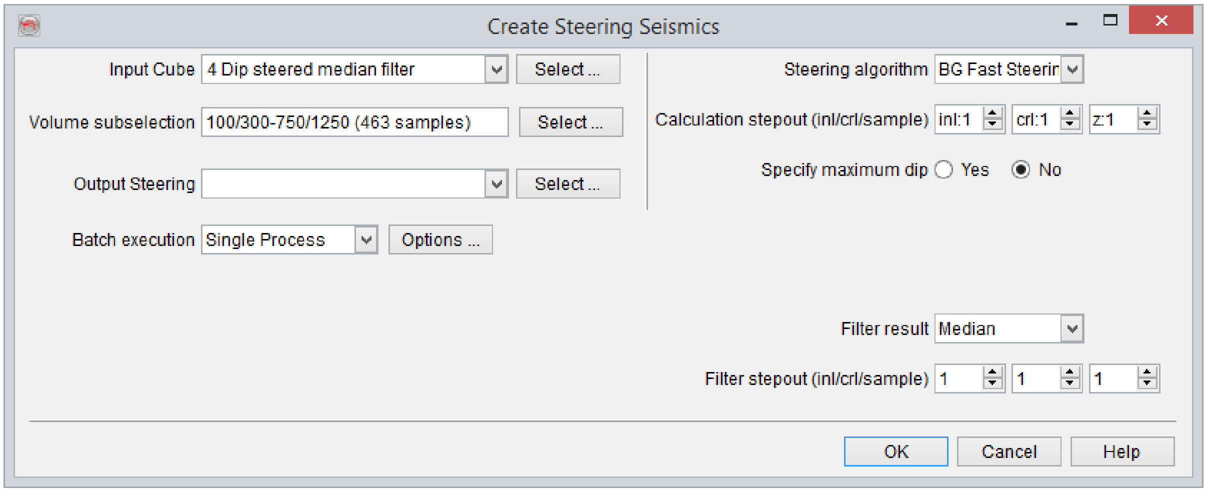


Select input data cube (usually a seismic volume) and optionally the sub-volume to process.

Five types of *Steering algorithms* are supported:

- *BG Fast Steering*
- *FFT Standard*
- *FFT Combined*
- *FFT Precise*
- *Event*

BG Fast Steering is a quick algorithm developed by BG. It is based on analysis of the gradient of the amplitude data, both vertically and horizontally. The BG fast algorithm is prone to noise, this can be overcome by adding a median or average filter.



Input Cube 4 Dip steered median filter

Volume subselection 100/300-750/1250 (463 samples)

Output Steering

Batch execution Single Process

Steering algorithm BG Fast Steerir

Calculation stepout (inl/crl/sample) inl:1 crl:1 z:1

Specify maximum dip ○ Yes ● No

Filter result Median

Filter stepout (inl/crl/sample) 1 1 1

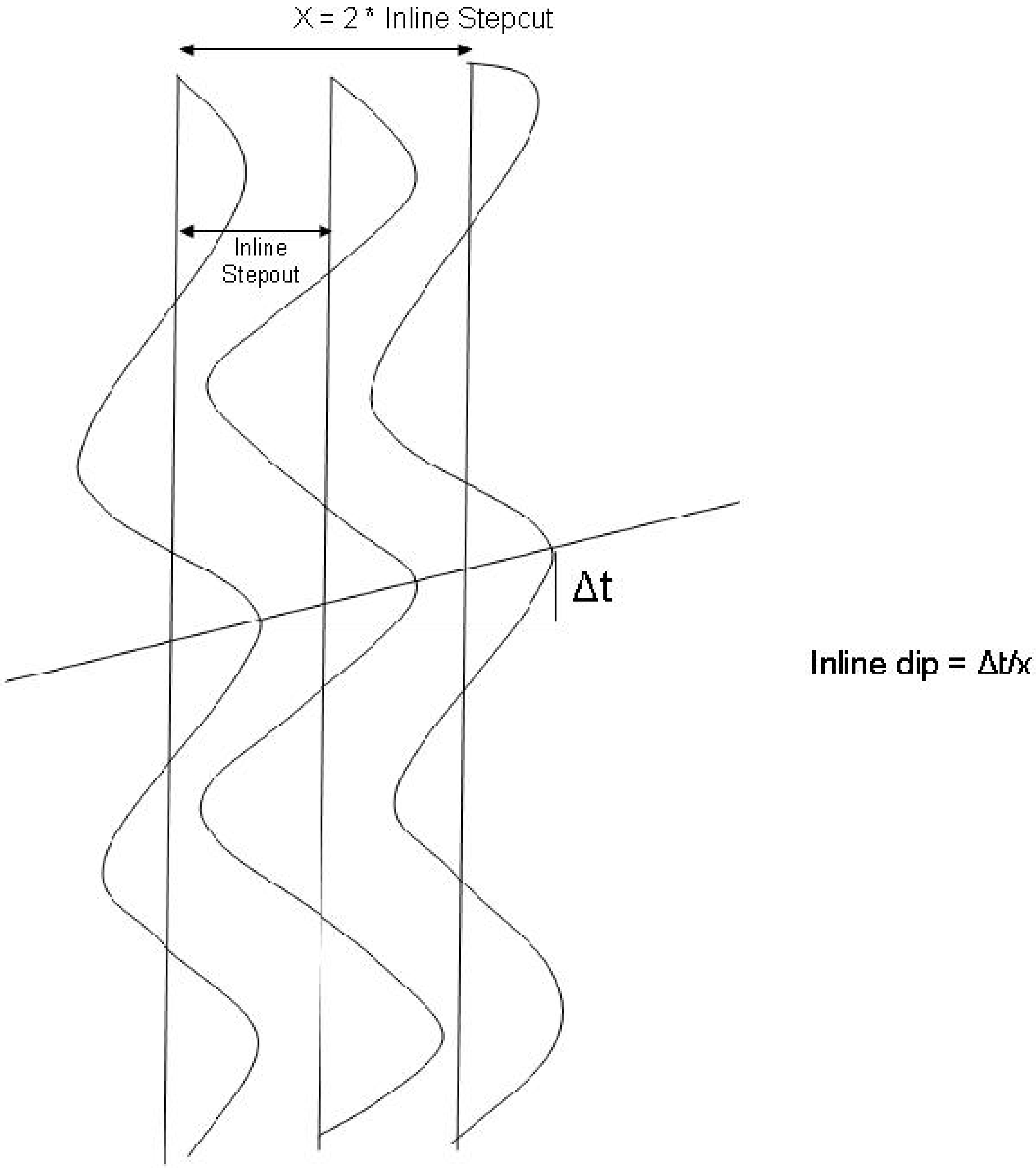
Standard, *Combined* and *Precise* are all Fast Fourier Transform based algorithms. They need a cube size specification. Standard is the recommended algorithm, and the *Precise* algorithm is most accurate but requires (considerably) more CPU time. The *Combined* algorithm uses the standard method, but applies the precise method when the standard method does not provide a stable solution.

The *Precise* algorithm uses zero padding of the signal before Fourier transformation. For example, if the input window is 7x7x7 samples, the algorithm adds zeros to all sides to obtain a 21x21x21 cube. A Fourier transform assumes that all inputs are periodic, and will give a high response at *1/(ns*dt)*. In case ns (the number of samples) is 7 and dt (the sample rate) is 4 *ms*, this will be at 36 *Hz*, interfering with the main frequencies of the signal. By zero padding, we shift this peak to 12 *Hz*. At the same time, the amplitude of the undesired peak is much lower. The *Precise* algorithm also uses another interpolation algorithm to find the maximum (hence dip) in the Fourier domain. The interpolation algorithm is 'true' 3D, i.e. a 3D signal is fitted at the maximum position. In the standard algorithm the maximum is found by three successive 1-D interpolations, which is much faster but less precise.

The Event Steering:

This is the latest dGB steering algorithm. To calculate the dip for a particular trace the algorithm looks for a maximum and minimum along the trace alternatively. i.e.

Max
Min
Max
Min
...
...
...



Finding dip in Inline direction

Each of these Max or Min events are matched with two neighboring traces, e.g. in the inline direction. The distances of the neighboring traces depends on the stepout. Now the difference in time values on these two neighboring traces is divided by the horizontal distance between the traces to get the inline dip. The same procedure is repeated for the crossline direction to get the crossline dip.

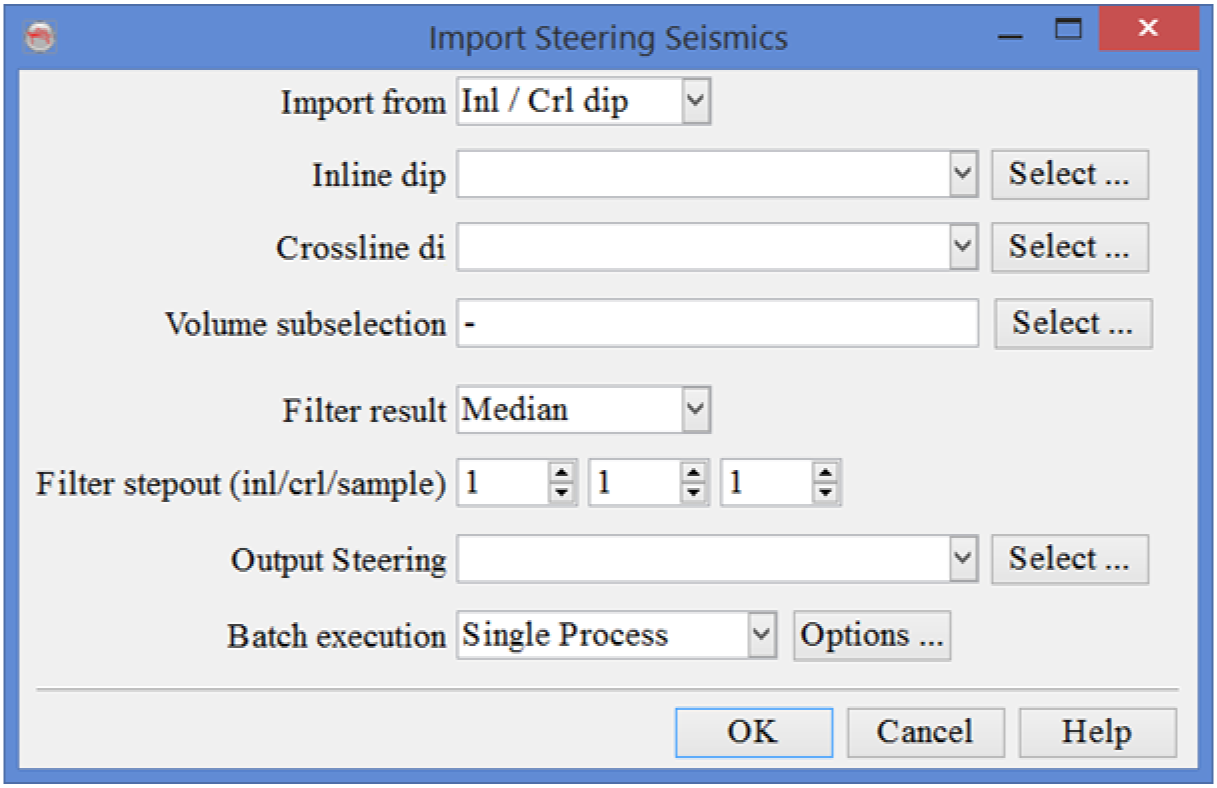
More background information, including a benchmark of the different algorithms and visual examples are presented in the **benchmark section**.

The optional *Specify maximum dip* limits dip values derived from the input data. Another option to avoid extreme dip values is to filter the steering data with a *Median filter*. This removes the outliers from the steering data. The stepouts are defined in samples, regardless sampling rate.

The processing specifications as defined in the window can be saved (optional). Provide a file name in the textbox to store the processing specification. If this space is left empty, the processing specification is not saved. If, for any reason, the processing is aborted, the process can be re-started with this parameter file using the *Re-start* option under the *Processing* menu.

The *Proceed* button starts the single machine or multi-machine processing mode. For more information on single and multi-machine processing, open the help menu from the Batch Processing window.

Import Steering Data



Import from: Inl / Crl dip

Volume subselection: -

Filter result: Median

Filter stepout (inl/crl/sample): 1 1 1

Batch execution: Single Process

Subject to Dip-Steering licensing permissions Steering data created in other software packages can be imported into OpendTect, Third-party steering data is supplied in the form of seismic volumes: two volumes for a 3D Steering Cube (inline dip and cross-line dip, or dip and azimuth), or one volume for 2D Steering Data (the dip in the line direction). The import functionality converts the 3rd-party data sets into OpendTect compatible Steering Cubes for 3D, or 2D, respectively.

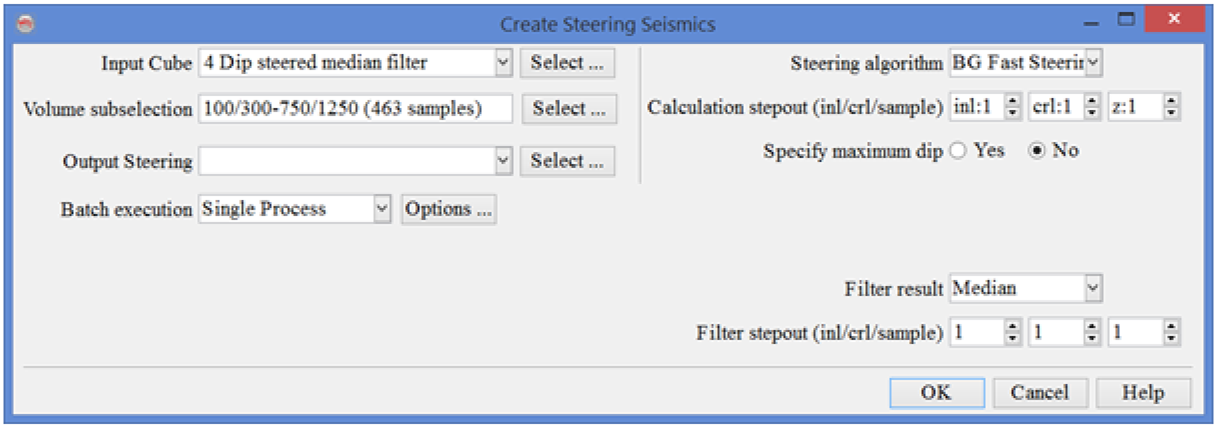
The input data must adhere to the following criteria:

- Dip values should not be negative and given in usec/m, or in m/m if the survey is in depth.
- Azimuth should be given in degrees from -180 to 180. Positive azimuth is derived from the inline in the direction of increasing cross-line numbers. Azimuth = 0 indicates that the dip is in the direction of increasing cross-line numbers. Azimuth = 90 indicates that the dip is in the direction of increasing in-line numbers.

Optionally the imported data can be smoothed by application of a median, or average *filter*. The default is not to smooth as smoothing filters can also be applied in a separate step afterwards (see 2.4 Filter Steering Cubes).

The import function can be executed in *batch*, on a single machine, or distributed over several computers

Create SteeringCube



Input Cube: 4 Dip steered median filter
Volume subselection: 100/300-750/1250 (463 samples)
Output Steering:
Batch execution: Single Process
Steering algorithm: BG Fast Steerir
Calculation stepout (inl/crl/sample): inl:1 crl:1 z:1
Specify maximum dip ○ Yes ● No
Filter result: Median
Filter stepout (inl/crl/sample): 1 1 1

Select the *Input Cube* (usually a seismic volume) and optionally specify the *Volume subselection* to process.

Give the Steering Cube to be computed a name under *Output Steering*.

Under *Batch execution* specify where the processing is to be done. *Single Process* means the Steering Cube is processed on the machine you are currently working on. Under *Options* … you can set the job priority (Linux nice level) from -19 (low) to 19 (top).

Multiple machines is the recommended mode as computing a Steering Cube is a relatively CPU intensive process. In this mode batch processing is distributed over all computers (workstations and clusters) that are available to the user. The batch processing window is launched after all parameters were set and OK is pressed.

> **Note:** that multi-machine batch processing is a unique selling point of OpendTect that is not automatically available after a default installation. This mode depends on the network environment and needs to be installed by the system administrator. For details, please see: Chapter 5.5 of the System Administrator's manual.

OpendTect supports three types of *Steering algorithms*:

- *BG Fast Steering*
- *FFT*
- *Event Steering*

BG Fast Steering is the default algorithm. It is the recommended algorithm for applying dip-steered filters and to compute dip-steered attributes. *BG Fast Steering* is a quick algorithm developed by the BG-Group. It computes dip in a small sub-volume defined by the step-outs from the gradient of the instantaneous phase. The *BG fast algorithm* is prone to noise, which can be overcome by adding a median or average filter. The default is to apply a mild *median filter* with step out 1,1,1.
Optionally, dip computation can be limited to a maximum dip (in us/m).

FFT Steering computes dip in a small sub-volume defined by the step-outs by searching for the highest energy in the 3D Fourier transformed domain. FFT Steering is the default for HorizonCube processing.
Standard is the recommended algorithm. It transforms the input cube to the Fourier Domain by applying a 3D FFT. The *Precise* algorithm pads zeros to the input volume before applying the 3D FFT. This results in a higher sampling in Fourier space, which allows for a more accurate positioning of the energy – peak and therefore, a more accurate estimation of dip. Although *Precise* is most accurate it also requires (considerably) more CPU time, which is why in practice it is seldom used. The *Combined* algorithm combines

the two methods: the stable solution is computed first. If a sensible result is obtained the output is accepted. Otherwise the precise method is used.

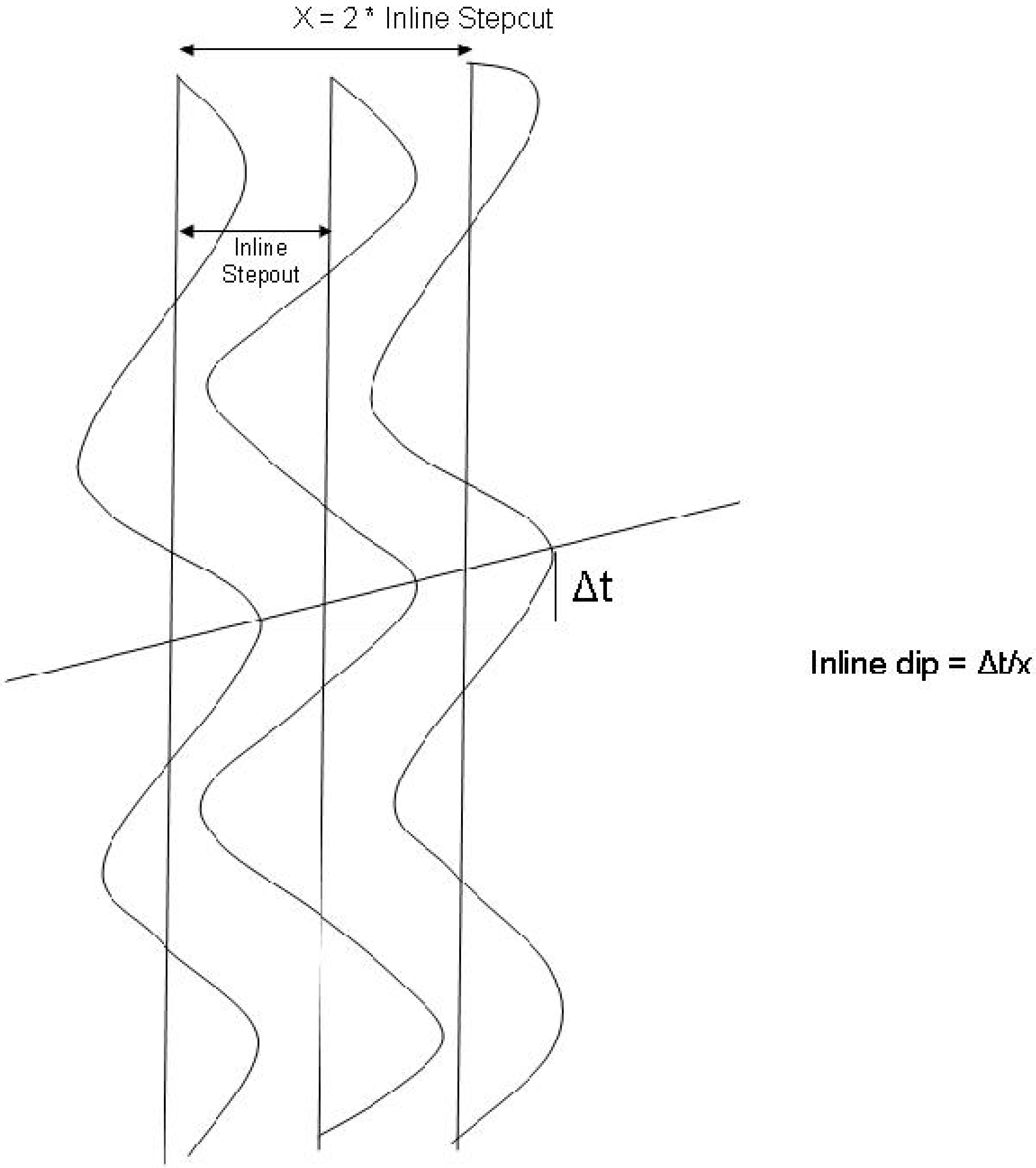
Also with *FFT Steering*, dip computation can optionally be limited to a *maximum dip* (in us/m) and the results can optionally be smoothed with a median or average *filter*. The default is not to smooth as smoothing filters can also be applied in a separate step afterwards (see **Filter Steering Cubes**).

Below some typical parameter settings are given for a HorizonCube computation in a workflow that first cleans-up the data by applying a dip-steered median filter:

1. Compute a Detailed Steering Cube with BG Fast Steering. Step-out 1,1,1.
2. Filter the Detailed Steering Cube to create a Background Steering Cube using a median filter with step-out range: 3,3,3 - 5,5,5.
3. Apply a dip-steered median filter to the input seismic data (typical radius range: 1 - 3).
4. Compute a new Steering Cube from the cleaned up seismic data set using FFT standard. Step-out 3,3,3 (or 3,3,5).
5. Optionally, apply a median filter to the Steering Cube generated above with step-out 1,1,3.
6. Compute the HorizonCube.

Note: that HorizonCube quality depends on the settings of these Steering Cube and Filter parameters. It is recommended to test various combinations by creating HorizonCubes in small test areas.

Event Steering calculates the dip in the inline direction and in the cross-line direction from extrema. All Maxima and Minima are determined on the central trace and two neighboring traces located at the *stepout* positions. Maxima and minima with the smallest Δt separation (see image) are considered to represent the same seismic event. This results in dip values at all extrema positions. Resampling these values to the seismic sampling rate yields the Steering data at the central trace position.



Finding dip in Inline direction

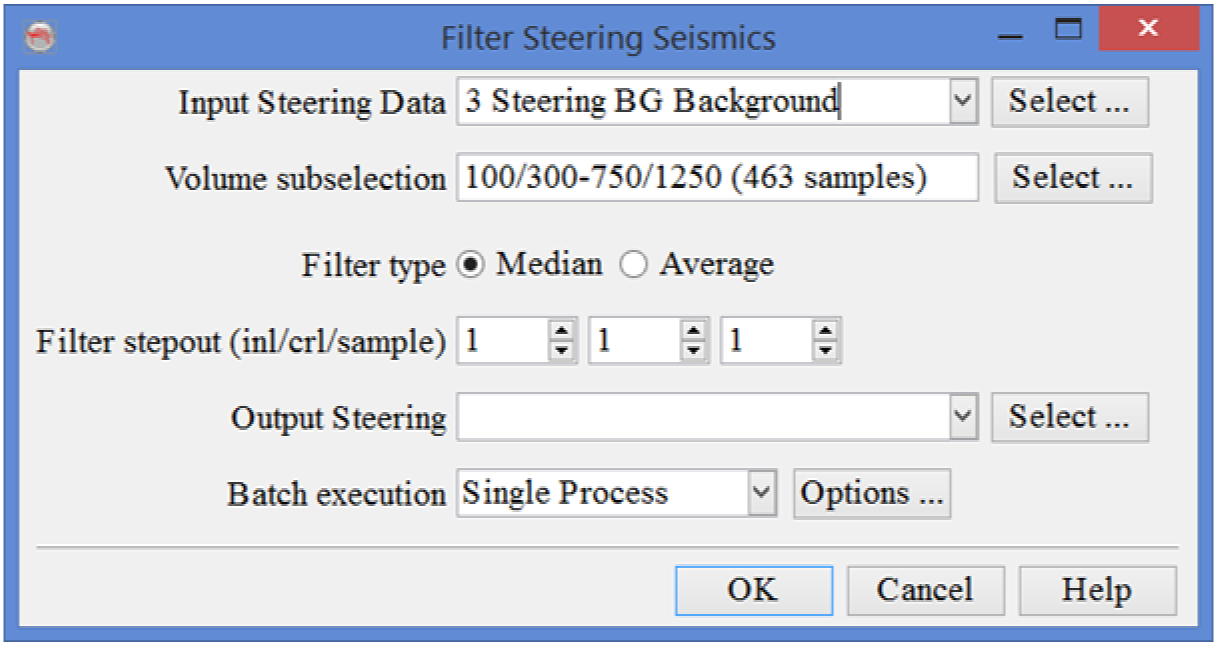
The *maximum dip (us/m)* parameter determines the search gate on the neighboring traces within which a maximum, or minimum must be found. The parameter ensures unrealistic dips are found, which may happen in noisy areas.

Also Event Steering supports optional application of smoothing *filters*. The default is not to smooth as smoothing filters can also be applied in a separate step afterwards (see **Filter Steering Cubes**).

The OK button starts the single machine, or multi-machine processing mode. For more information on single and multi-machine processing, open the help menu from the Batch Processing window.

Filter

SteeringCubes can be smoothed by a median or average filter during creation, or post-creation. Typical filter settings to create a Background Steering Cube from a Detailed Steering Cube are: *Median* filter with *stepouts*: 3,3,3, or 5,5,5, or 3,3,7 (to emphasize smoothing in the vertical direction).



Input Steering Data: 3 Steering BG Background

Volume subselection: 100/300-750/1250 (463 samples)

Filter type: ● Median ○ Average

Filter stepout (inl/crl/sample): 1 1 1

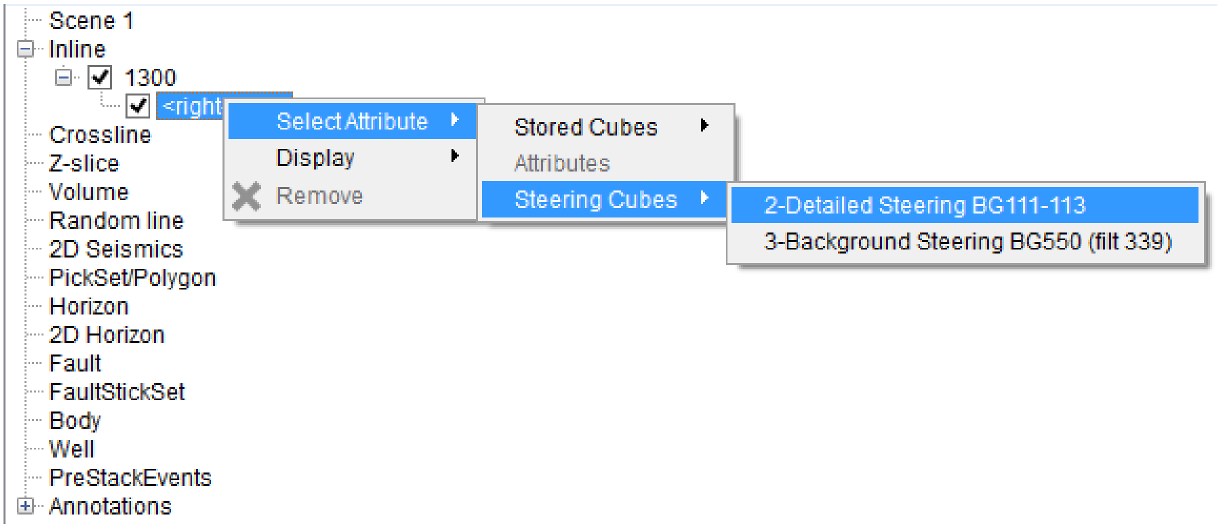
Output Steering:

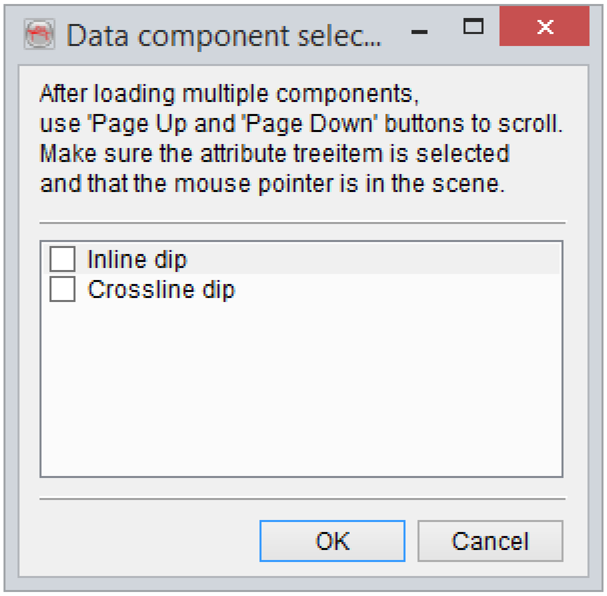
Batch execution: Single Process

Select the input SteeringCube and, optionally, the *sub-volume* to process. The *median* filter *stepout* settings are defined in samples, regardless of sampling rate. *Batch jobs* can be processed on a single machine or on multiple machines. For more information on single- and multi machine processing, open the help menu from the Batch Processing window.

Display SteeringCube

Once the SteeringCube has been calculated and filtered (optional), the results can be displayed in the OpendTect scene as a multi-component attribute, i.e on inline dip and/or cross-line dip.





After loading multiple components,
use 'Page Up and 'Page Down' buttons to scroll.
Make sure the attribute treeitem is selected
and that the mouse pointer is in the scene.

☐ Inline dip
☐ Crossline dip

Attributes with Steering

See the following chapters below.

Curvature Analysis

Curvature can be used as input to other attributes. The *Volume Statistics* attribute, in particular, proves to give very useful outputs. Just select the *Curvature* attribute as input and select the output statistic. For Fault Detection, *'Variance'* is a suitable output statistic.

A local surface is constructed at the evaluation point by following the dip information from the steering cube. The *Curvature* attribute specified in *Output* is then calculated according to Roberts (2001). In his Feb. 2001 *First Break* article, Roberts defines Curvature as a two-dimensional property of a curve that describes how bent a curve is at a particular point in the curve, i.e. how much the curve deviates from a straight line. The same concept is used to describe the Curvature of a surface. Curvature is measured on the curve which is the intersection between a plane and the surface. Since this can be done in numerous ways, there is an infinite number of Curvature attributes that can be calculated for any plane. The subset implemented here relates to the most useful subset of Curvatures that are defined by planes that are orthogonal to the surface and which are called normal Curvatures. A positive Curvature corresponds to an anticline and a negative Curvature indicates a syncline. A flat plane has zero Curvature. The application suggestions are from Roberts (First Break, Feb 2001).

Outputs:

Mean Curvature: The average of any two orthogonal Normal Curvatures through a point on the surface is constant and is defined as the Mean Curvature. Minimum and Maximum Curvature (see below) are orthogonal surfaces, therefore the Mean Curvature is also the sum of Minimum and Maximum Curvature divided by two. The Mean Curvature is not considered a very important attribute for visualization purposes, but it is used to derive some of the other attributes.

Gaussian Curvature: is defined as the product of the Minimum and Maximum Curvature. It is sometimes referred to as the Total Curvature. The Gaussian Curvature is not considered a very important attribute for visualization purposes, but it is used to derive some of the other attributes.

Maximum Curvature: From the infinite number of Normal Curvatures there exists one curve, which defines the largest absolute Curvature. This is called the *Maximum Curvature*. The plane in which Maximum Curvature is calculated is orthogonal to the plane of the Minimum Curvature. This attribute is very effective at delimiting faults and fault geometries.

Minimum Curvature: is the smallest absolute Curvature from the infinite number of Normal

Curvatures that exist. The plane in which Minimum Curvature is calculated is orthogonal to the plane of the Maximum Curvature. The Minimum Curvature is often quite noisy, but it can sometimes be a good diagnostic in identifying fractured areas. Also, it is used to compute other Curvature attributes.

Most Positive Curvature: returns the most positive curvature from the infinite number of Normal Curvatures that exist. The attribute reveals faulting and lineaments. The magnitude of the lineaments is preserved but the shape information is lost. This attribute can be compared to first derivative based attributes (dip, edge, and azimuth).

Most Negative Curvature: returns the most negative curvature from the infinite number of Normal Curvatures that exist. The attribute reveals faulting and lineaments. The magnitude of the lineaments is preserved but the shape information is lost. This attribute can be compared to first derivative based attributes (dip, edge, and azimuth).

Shape Index (Si): is a combination of Maximum and Minimum Curvature that describes the local morphology of the surface independent of scale. The attribute may reflect e.g. whether the surface corresponds to a bowl(Si=-1), a valley (Si=-1/2), ridge (Si=+1/2), a dome (Si=1) or it is flat (Si=0). Because the attribute is not affected by the absolute magnitude of Curvature, it is reported to be useful for picking up subtle fault and surface lineaments, as well as other patterns.

Dip Curvature: returns the Curvature of the intersection with the plane that defines the dip direction of the surface. This plane is orthogonal to the plane for the Strike Curvature. This Curvature method tends to exaggerate local relief contained within the surface and can be used to enhance differential compacted features such as channeled sand bodies and debris flows.

Strike Curvature: (also known as Tangential Curvature) returns the Curvature of the intersection with the plane that defines the strike direction of the surface. This plane is orthogonal to the plane for the Dip Curvature. The attribute describes the shape of the surface. It is extensively used in terrain analysis, e.g. to study soil erosion and drainage patterns. The attribute reveals how shapes are connected, e.g. how ridges are connected to the flanks of anticlines. It may be useful for fluid-flow studies.

Contour Curvature: (also known as Plan Curvature) is not a Normal Curvature. It is very similar to the Strike Curvature and effectively represents the Curvature of the map contours associated with the surface. Contour Curvature values are not very well constrained, and large values can occur at the culmination of anticlines, synclines, ridges, and valleys.

Curvedness: describes the magnitude of Curvature of a surface independent of its shape. The attribute gives a general measure of the amount of Total Curvature within the surface.

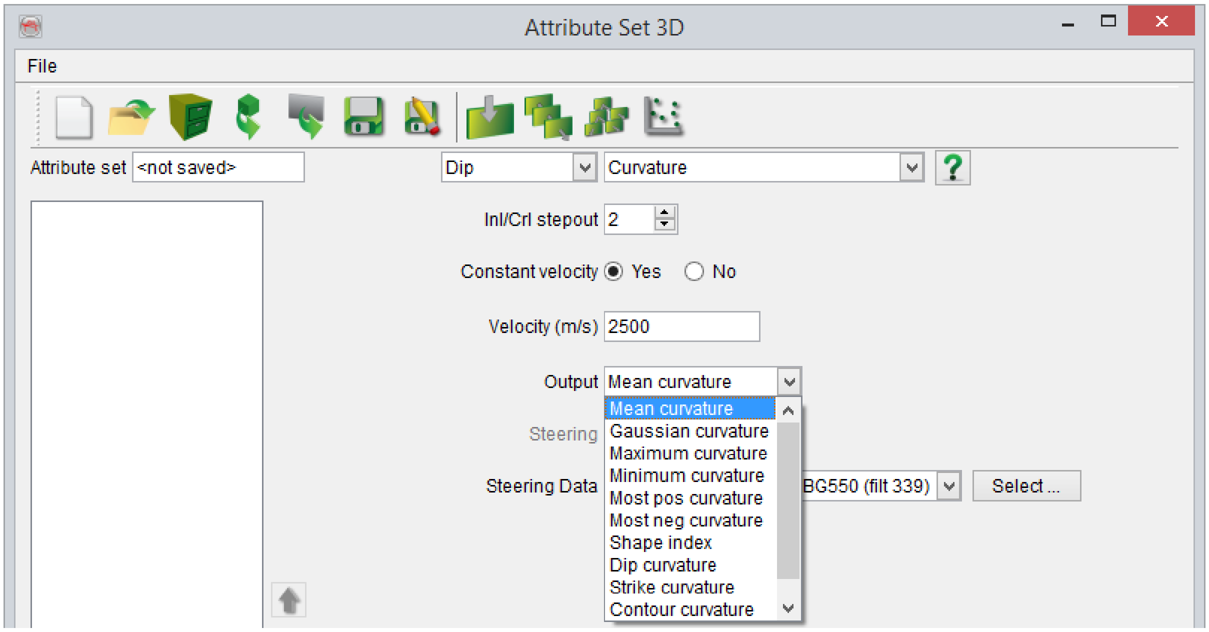
Volumetric Curvature

Returns *Curvature* properties from a steering cube

At every sample location, a 'virtual local horizon' is constructed on which the curvature is calculated (from the SteeringCube). This 'local horizon' is converted to depth in time surveys, which is why a velocity input is required. The output attribute is a 2D dataset or 3D volume that may be applied (projected) along horizons. Using a large stepout with a detailed steering cube in this attribute gives similar results as using a strongly filtered steering cube but with a small stepout. However the attribute runtime will be different.

For further detail, including descriptions of the output options, below, please refer to ***Curvature Analysis***

The *Curvature* options in OpendTect are shown below:



Attribute set <not saved>

Dip

Curvature

Inl/Crl stepout 2

Constant velocity ● Yes ○ No

Velocity (m/s) 2500

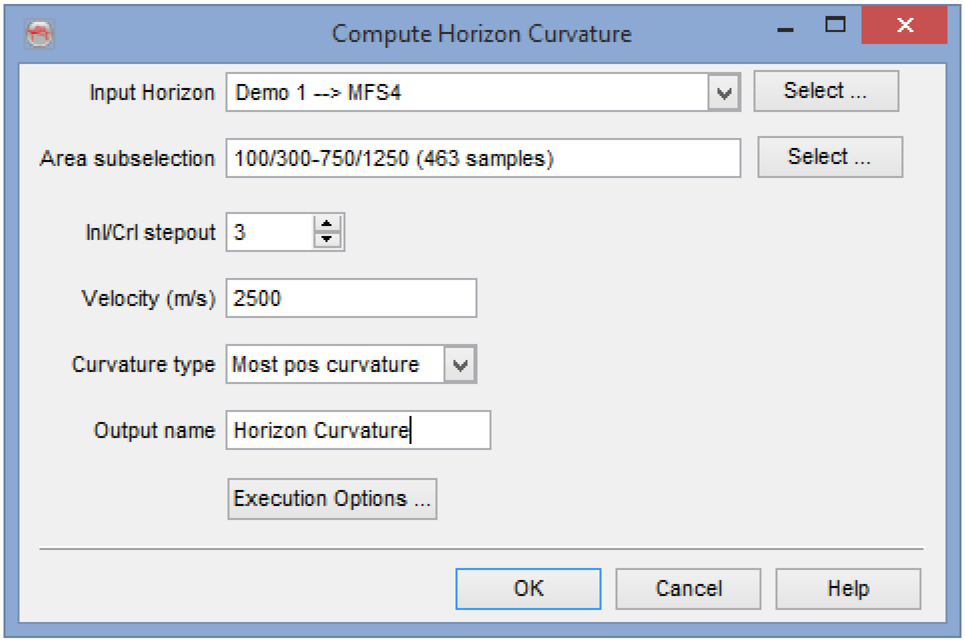
Output Mean curvature

Steering

Steering Data BG550 (filt 339)

Horizon-Based Curvature

Horizon Curvature can be calculated along horizons either via *Processing* > *Create Horizon Output* > *Compute Horizon Curvature...*:



Input Horizon: Demo 1 --> MFS4

Area subselection: 100/300-750/1250 (463 samples)

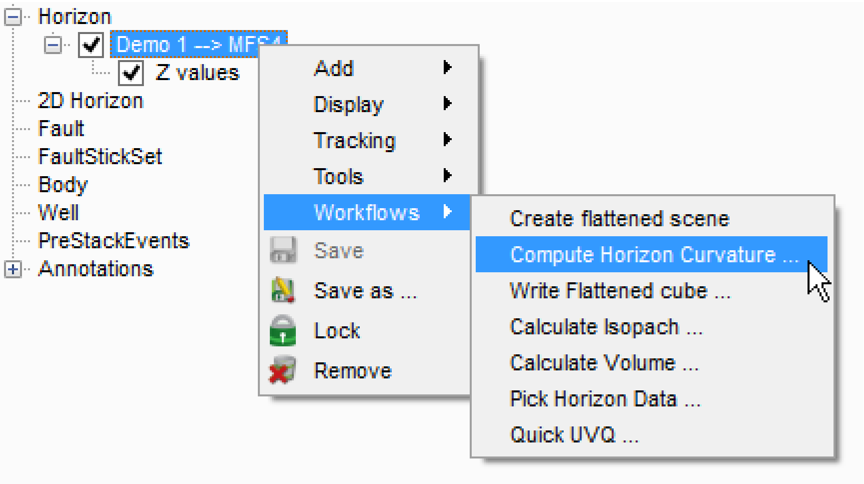
Inl/Crl stepout: 3

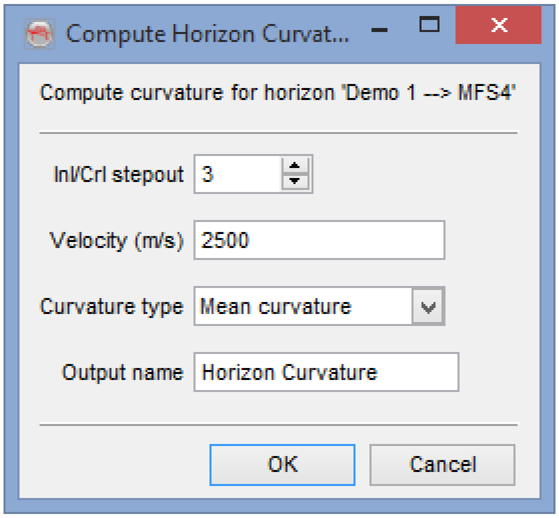
Velocity (m/s): 2500

Curvature type: Most pos curvature

Output name: Horizon Curvature

or via the tree, directly onto the selected horizon:





Compute curvature for horizon 'Demo 1 --> MFS4'

Inl/Crl stepout: 3

Velocity (m/s): 2500

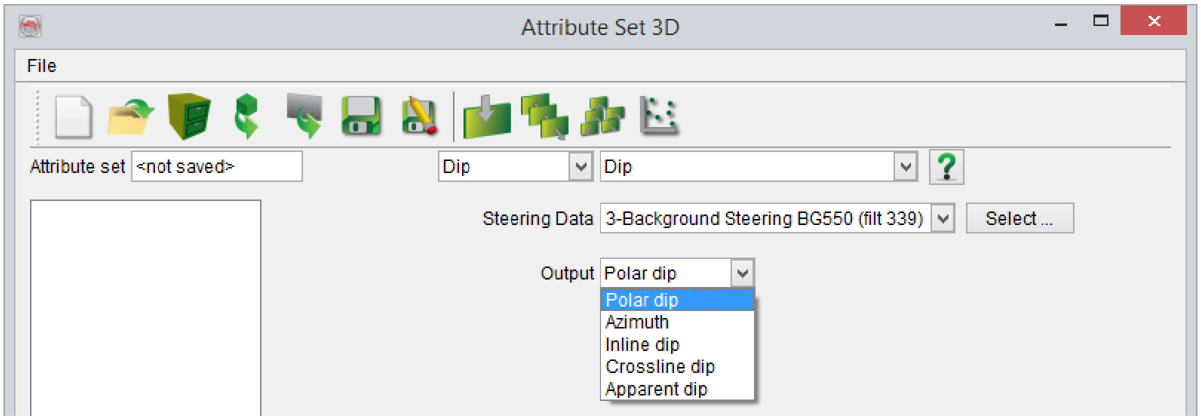
Curvature type: Mean curvature

Output name: Horizon Curvature

This input horizon is converted to depth in time surveys, which is why a velocity input is required. The output attribute is added on the horizon when using the tree menu, or as a grid in the database for that horizon when using the processing menu. Note that the first calculation is done within the current session, while it is done in batch from the processing menu (recommended for large horizons and/or large stepouts). For further detail, including descriptions of the output options, below, please refer to ***Curvature Analysis***

Dip

Returns dip/azimuth from a SteeringCube



Attribute set <not saved>
Dip
Dip
Steering Data 3-Background Steering BG550 (filt 339)
Output Polar dip

Description: The inline and crossline dips of a SteeringCube are transformed to the requested *Output* type (Polar dip, Azimuth, Inline dip, Crossline dip). When the SteeringCube was computed from seismic data sampled in time, the dips in a SteeringCube are apparent dips (slowness), and the returned attribute will also represent apparent dips. To compute real dips, please use the *dip angle* attribute.

Outputs:

> Polar dip: attribute converts extracted inline and crossline dips to polar dip, or true (geological) dip. The polar dip is the square root of the sum of (inline dip)2 and (crossline dip)2. The polar dip is thus larger or equal to zero.
> Dips are given in µseconds/meters in time surveys (millimeters/meters in depth survey), since they are a ratio between a vertical length and a horizontal distance. The **dip angle** attribute may be used to convert the polar dip output into degrees.

Note: along a 2D line the polar dip will return the absolute value of the line dip, the dip along the 2D line.

Azimuth: attribute returns the Azimuth of the dip direction in degrees ranging from -180 to +180. Positive azimuth is defined from the inline in the direction of increasing crossline numbers. Azimuth = 0 indicates that the dip is dipping in the direction of increasing cross-line numbers. Azimuth = 90 indicates that the dip is dipping in the direction of increasing in-line numbers.

Note: this output is not available in 2D.

Inline dip: returns the dip along the inline direction as extracted by the steering algorithm. It is the first stored component of the steering cube, in µs/m or mm/m.

Note: this output is not available in 2D.

Crossline dip: attribute returns the dip along the crossline direction as extracted by the steering algorithm. It is the second stored component of the steering cube, in µs/m or mm/m.

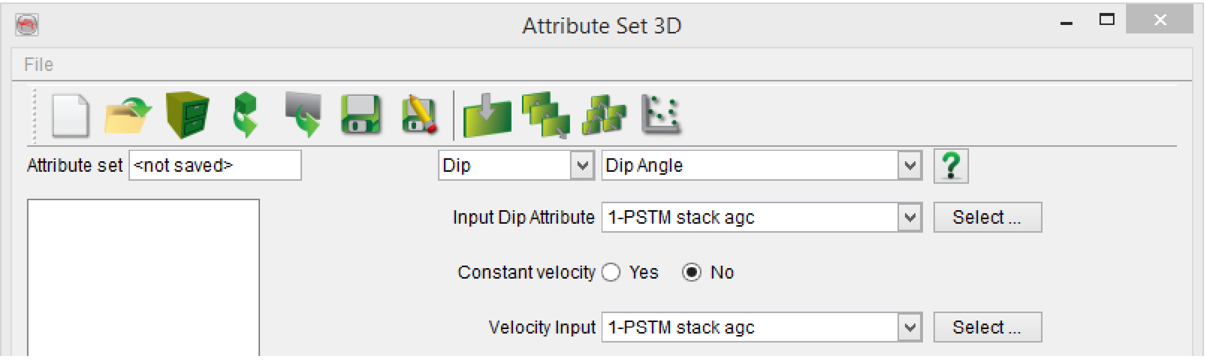
Note: this output is not available in 2D (use line dip for 2D survey instead).

Line dip: returns the dip along the 2D line. Computed steering lines are also two-component files with the first component equal to zero for all samples. The line dip is the second component.

Note: this output is not available in 3D.(use inline dip and/or crossline dip instead)

Dip Angle

The *dip angle* attribute converts the apparent dip (slowness-us/m) into degrees. It calculates the true dip from the apparent dip (slowness). If no velocity model is available, specify a constant *velocity* in the velocity field. The velocity should be given in meters per second (m/s).



Attribute set <not saved>

Dip

Dip Angle

Input Dip Attribute 1-PSTM stack agc

Constant velocity ○ Yes ● No

Velocity Input 1-PSTM stack agc

The dip angle is calculated as follow.

$$\text{Dip Angle } = \text{Tan}^{-1}\left(\frac{TWT-dip}{2E+06}xV\right) \text{ in time domain}$$

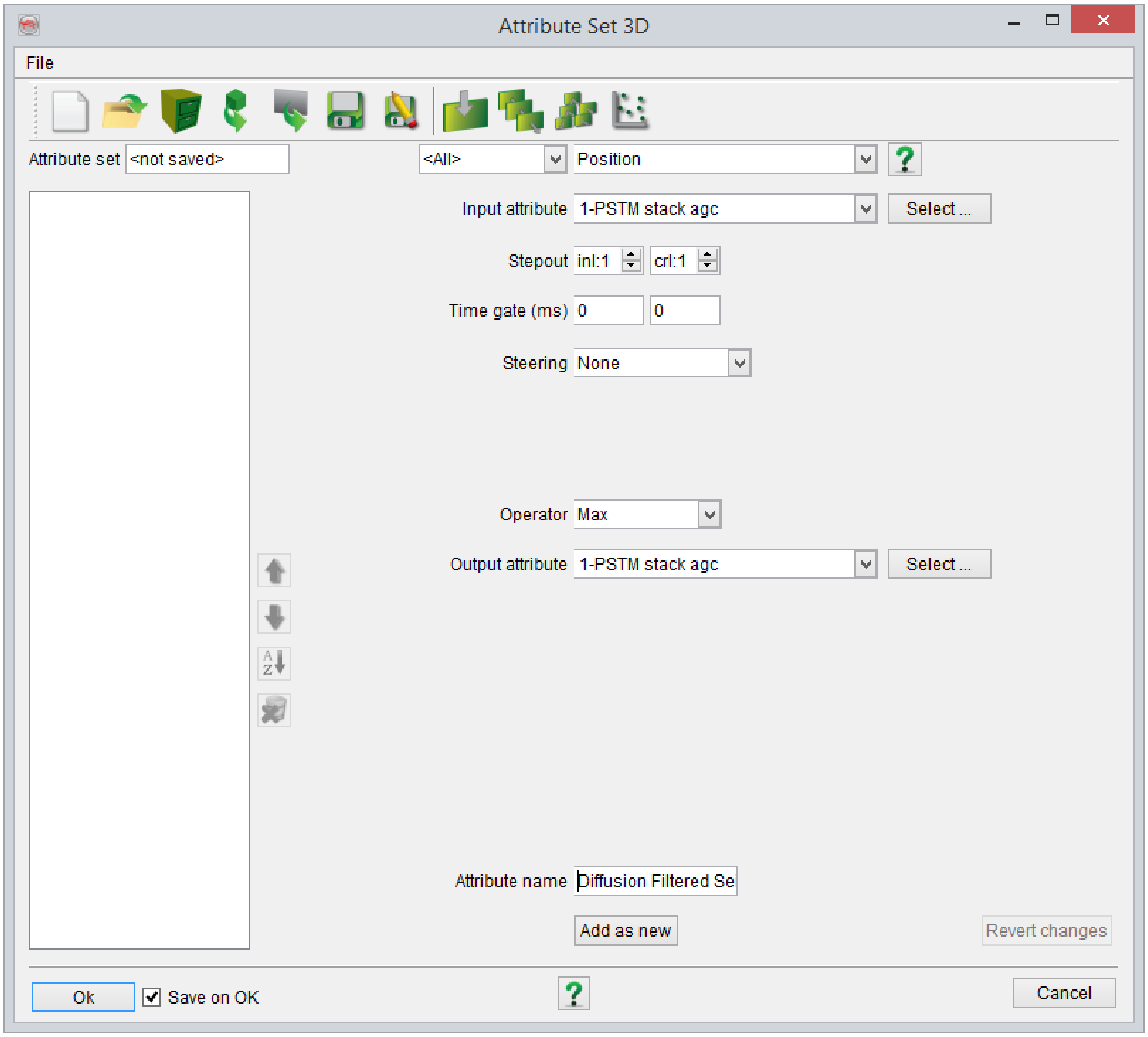
$$\text{Dip Angle } = \text{Tan}^{-1}\left(\frac{Z-dip}{1000}\right) \text{ in depth domain}$$

where TWT-dip is in micro seconds per meter and Z-dip is in millimeters per meter.

Note: OpendTect uses SI-unit systems.

Position

Position attribute returns any attribute calculated at the location where an other attribute has its minimum, maximum, or median within a small volume.



Attribute set <not saved>

<All> Position

Input attribute 1-PSTM stack agc

Stepout inl:1 crl:1

Time gate (ms) 0 0

Steering None

Operator Max

Output attribute 1-PSTM stack agc

Attribute name Diffusion Filtered Se

☑ Save on OK

Description: The input attribute defines the attribute that is used to determine the position at which the output attribute has to be calculated. The stepouts, time gate, and steering define the volume in which the input attribute is evaluated. 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.

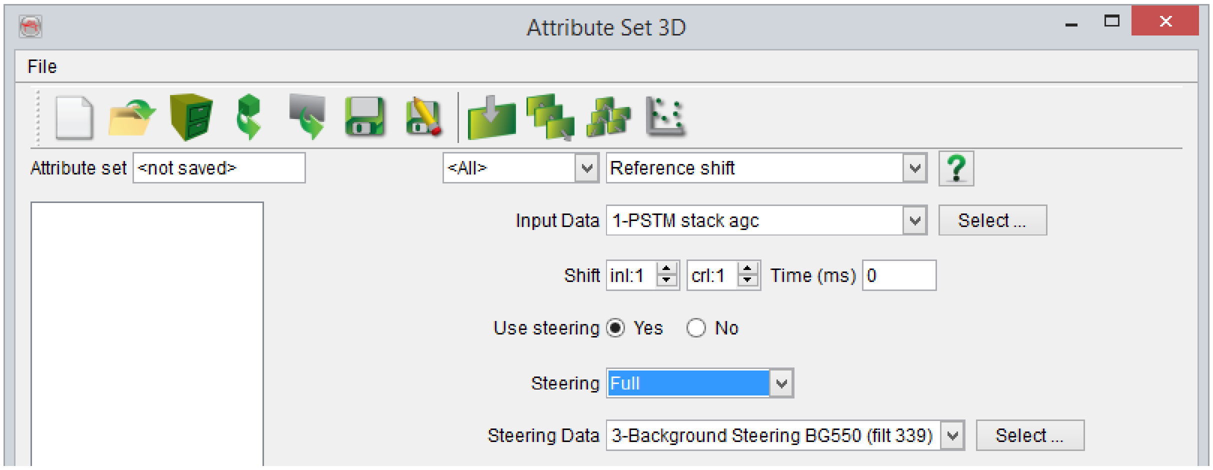
For example, one can determine where in a small volume the energy is minimal, and output the frequency at the location of this lowest energy. Another way of applying this attribute is to output Dip-steered filtered data at minimum values in areas where faults are present. In this way, the noise is reduced and the faults are sharpened.

Steering: In *Central Steering* the local dip information at the reference point is followed from trace to trace until all samples in the specified search radius are found. Central steering thus collects the input values along a dipping plane. In *Full steering* the dip information of the reference point is used only to find the position (and value) of the adjacent trace. The dip information at this new position is then used to find the position (and value) of the next trace and so on, until all samples in the specified search radius are found. Full steering thus corresponds to collecting the input values along a curved surface. In *Constant direction* the steering information is user-specified. The range of the *Apparent dip* is more than zero, and the *Azimuth* is defined from the inline, in the direction of increasing crossline-numbers. The azimuth ranges from -180 to 180 degrees.

In all forms of Steering the amplitude values at the intersection of trace and Steering surface are determined by interpolation.

Reference Shift

The *Reference shift* attribute moves the extraction position in 3D space.



Attribute set <not saved>

<All> Reference shift

Input Data 1-PSTM stack agc

Shift inl:1 crl:1 Time (ms) 0

Use steering ● Yes ○ No

Steering Full

Steering Data 3-Background Steering BG550 (filt 339)

Description: The *Input attribute* is extracted at the shifted position. Original reference (extraction) point has inline crossline co-ordinates (0,0). Relative number 1 means the next inline or crossline, respectively. The vertical shift is specified in *ms* using the *Time* option, or can be derived using *Steering*. Steering is specified in one of the following ways:

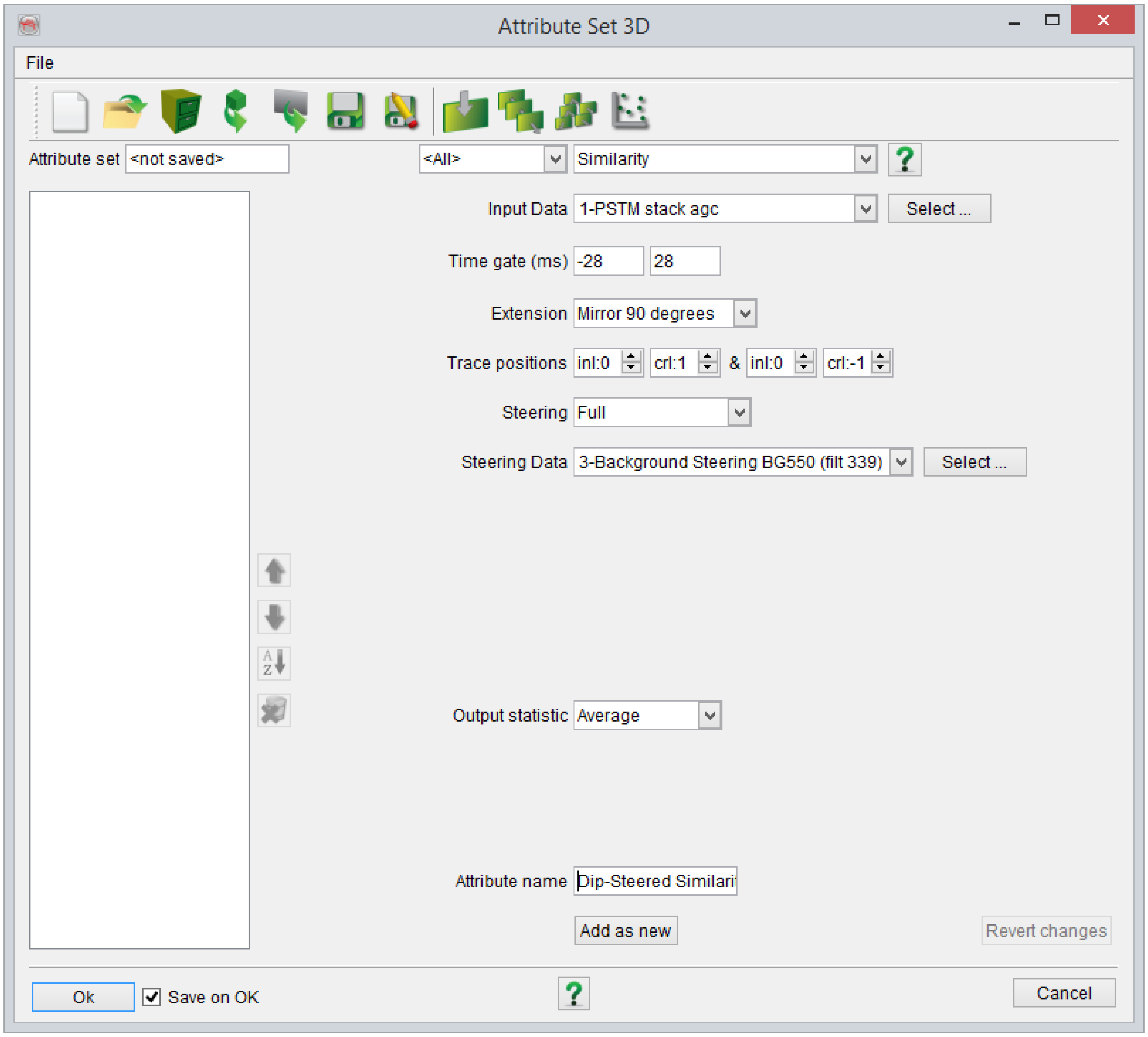
- *None:* The reference position is not shifted vertically.

- *Central:* The reference position is vertically shifted according to the dip and azimuth information at (0,0,0) from the SteeringCube.
- *Full:* The reference position is vertically shifted according to the dip and azimuth from the SteeringCube, from trace to trace from the starting position (0,0,0) to the position specified at *Inl/Crl shift*.
- *Constant direction:* The reference position is vertically shifted according to a user defined *Apparent dip* and *Azimuth*.

Shifting the reference position is a form of *directivity* that is useful in multi-attribute analysis. For example, to highlight flat spots, one may consider training a neural network on attributes extracted in three horizontally aligned windows.

Similarity

The *Similarity* attribute returns trace-to-trace similarity properties



Attribute set <not saved>

<All> Similarity

Input Data 1-PSTM stack agc

Time gate (ms) -28 28

Extension Mirror 90 degrees

Trace positions inl:0 crl:1 & inl:0 crl:-1

Steering Full

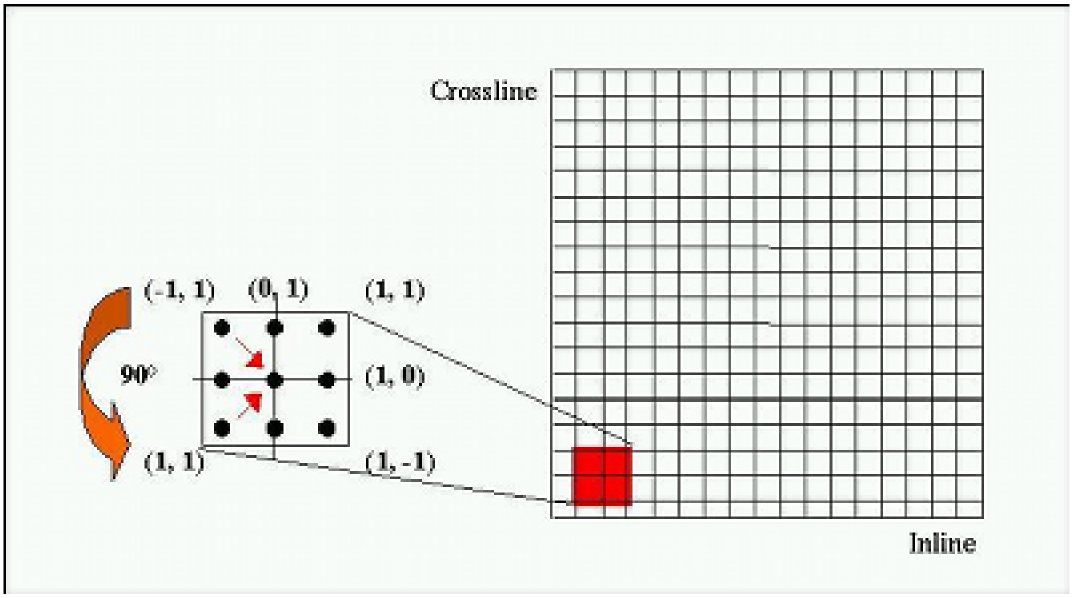
Steering Data 3-Background Steering BG550 (filt 339)

Output statistic Average

Attribute name Dip-Steered Similari

☑ Save on OK

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, while a similarity of 0 means they are completely dis-similar. Consider the trace segments to be vectors in hyperspace, similarity is then defined as the euclidean distance between the vectors, normalized over the vector lengths. The trace segments as defined by the *Time gate* in ms and are found by *Steering* from the reference point to the specified *trace positions*. Positions are specified in relative numbers (see figure). The *Extension* parameter determines how many trace pairs are used in the computation, see below.



Definition of trace positions relative to the reference point at (0,0)

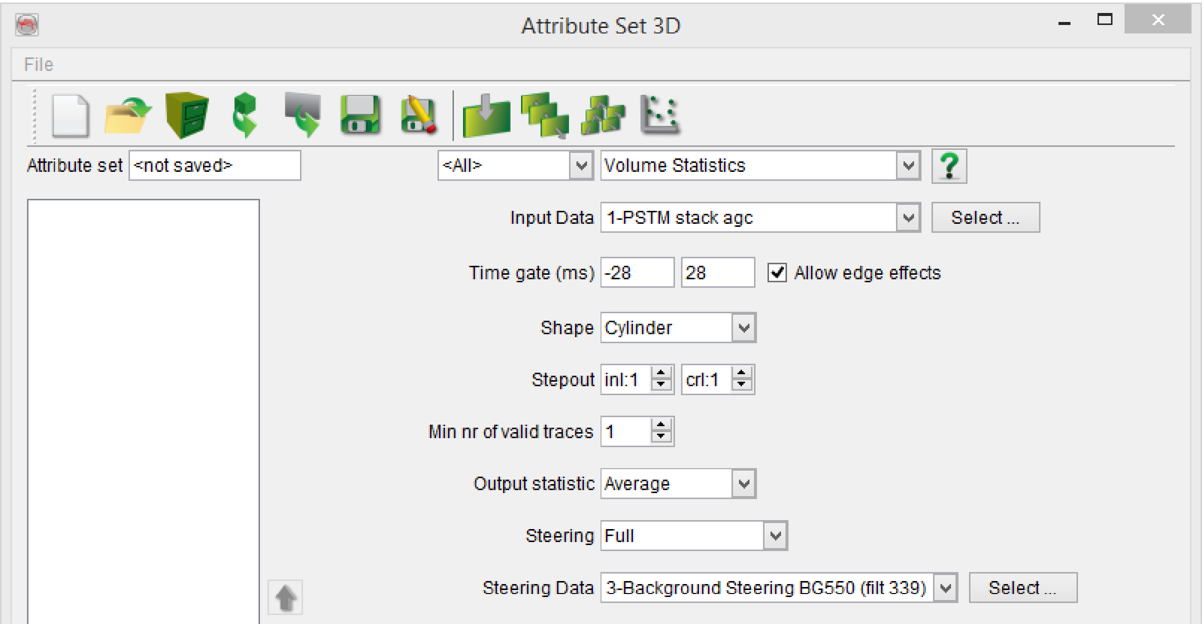
Extension: With *None* specified, only the trace pairs specified in *trace positions* are used to compute the output. *Mirror at 90 degrees* and *Mirror at 180 degrees* means that two similarities are computed: for the specified trace pair and for the pair that is obtained by 90 or 180 degrees rotation, respectively. The average, minimum or maximum of these pairs as specified in *Output statistic* is returned. In *Full block* all possible trace pairs in the rectangle defined by *Inl/Crl stepout* are computed. The statistical property specified in *Output statistic* is returned.

Steering: In *Central Steering* the local dip information at the reference point is followed from trace to trace until all samples in the specified search radius are found. Central Steering thus collects the input values along a dipping plane. In *Full steering* the dip information of the reference point is used only to find the position (and value) of the adjacent trace. The dip information at this new position is then used to find the position (and value) of the next trace and so on, until all samples in the specified search radius are found. Full steering thus corresponds to collecting the input values along a curved surface. In *Constant direction* the steering information is user-specified. The range of the *Apparent dip* is more than zero, and the *Azimuth* is defined from the inline, in the direction of increasing crossline-numbers. The azimuth ranges from -180 to 180 degrees.

In all forms of steering the amplitude values at the intersection of trace and steering surface are determined by interpolation.

Volume Statistics

The *Volume Statistics* attribute returns statistical properties from a small sub-volume



Attribute set <not saved>

Volume Statistics

Input Data 1-PSTM stack agc

Time gate (ms) -28 28 ☑ Allow edge effects

Shape Cylinder

Stepout inl:1 crl:1

Min nr of valid traces 1

Output statistic Average

Steering Full

Steering Data 3-Background Steering BG550 (filt 339)

Description: The statistical property specified in *Output statistic* is returned. The input values are collected from a cube (rectangle) or cylinder (ellipsoid) around the reference point defined by parameters: *Time gate*, *Shape* and *Inl/Crl stepout*. Optionally *Steering* is used to obtain the trace segments of the input sub-volume.

Steering: In *Central Steering*, the local dip information at the reference point is followed from trace to trace until all samples in the specified search radius are found. Central Steering thus collects the input values along a dipping plane. In *Full steering* the dip information of the reference point is used only to find the position (and value) of the adjacent trace. The dip information at this new position is then used to find the position (and value) of the next trace and so on, until all samples in the specified search radius are found. Full steering thus corresponds to collecting the input values along a curved surface. In *Constant direction* the steering information is user-specified. The range of the *Apparent dip* is more than zero, and the *Azimuth* is defined from the inline, in the direction of increasing crossline-numbers. The azimuth ranges from -180 to 180 degrees.

In all forms of steering, the amplitude values at the intersection of trace and steering surface are determined by interpolation.

Perpendicular Dip Extractor

This attributes works similarly to the **volume statistics** attribute, with one very important exception:

The time gate used for the extraction of the amplitudes is not vertical, but is perpendicular to the layer dip. This layer dip is read from a SteeringCube, which is why it is not possible to toggle off the full steering for this attribute.
Using this attribute instead of the standard volume statistics will most often return more "correct" amplitudes, because the extraction is done in the same geological layer with a real constant thickness, whereas a vertical time gate would represent an apparent thickness, varying through the survey.

Benchmark SteeringCube Creation

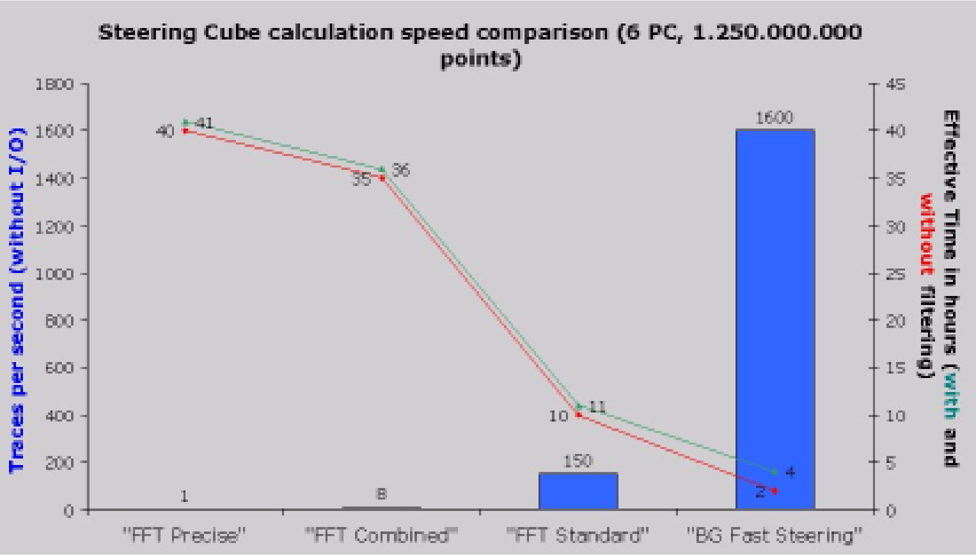
In this chapter the quality and speed of the available *steering algorithms* are compared in order to guide the choices for the right SteeringCube algorithm. First, the results of a test on a standard data set are presented to give an indication of the speed of SteeringCube calculation versus the chosen algorithm and *calculation cube size* parameter (where applicable). Then, a visual quality check on one of the inlines is presented. This is done by checking the crossline dip, which is directly extracted from the SteeringCube, and by reviewing the steered similarity, which uses information derived from the SteeringCube.

The final output quality is determined not only according to the steering algorithm. The size of the calculation cube also plays an important role. Next, the application of additional dip-steered median filter influences on the output quality are presented and finally, the use of dip limit.

Speed vs Algorithm and Calculation Cube Size

In *Figure 1* the relative calculation speed is displayed for the different algorithms. The test was done on a cluster of 6 computers using distributed computing. The input cube has 817811 traces. Each trace has 1551 samples with step of 4 ms, making a total of 6200 ms. Calculation speed is measured in traces per second.

Figure 1. Comparison of relative speed of the different steering algorithms.



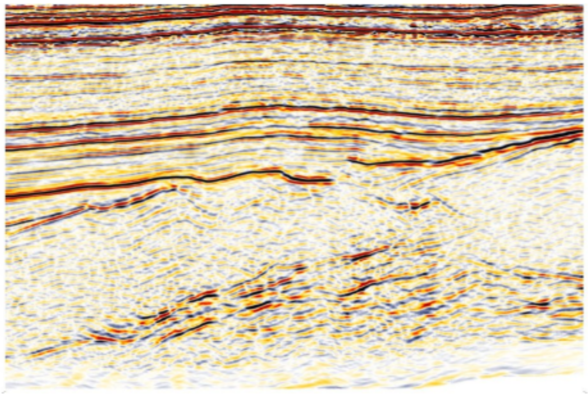
Visual Quality Check

In the following sections, the crossline dip component of a SteeringCube created with the different algorithms is presented. The figures contain the results from the *Precise FFT* algorithm, the *Combined FFT* algorithm, the *Standard FFT* algorithm and the *BG Fast Steering* algorithm (for technical details see **Create SteeringCube**) for a cube size of 3. Notice that the nomenclature convention used is the following:

- The FFT precise is called FFT+.
- The FFT combined is called FFT++.
- The standard steering algorithm is called FFT+++.
- The calculation cube size is specified after the algorithm (e.g. FFT7, BG5, etc.).
- The limit of dip is called maxdipXXX, where XXX is the limit in Âµs/m.
- Additional filtering are called medXYZ, where X is the inline stepout, Y the crossline stepout and Z the sample stepout.

The inline itself is displayed in *Figure 2* for reference. The inline was selected because many geological and seismic features are visible, which enable evaluation of the performance of the algorithm in different environments.

Figure 2. Seismic data of the test inline.



Crossline Dip Attribute

The crossline dip is one of the two SteeringCube components (together with inline dip) and is related to the dips projected in the crossline direction. In *Figure 3* to *Figure 5* the crossline dip is displayed for the algorithms mentioned in Figure 1 with cube sizes of 3, 5, and 7.

Figure 3. Crossline dip with (calculation cube size=3): precise steering (A), combined steering (B), standard steering (C) and BG steering (D).

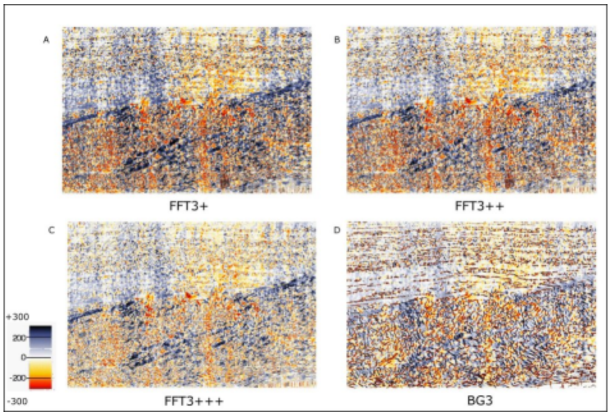


Figure 4. Crossline dip with (calculation cube size=5): precise steering (A), combined steering (B), standard steering (C) and BG steering (D).

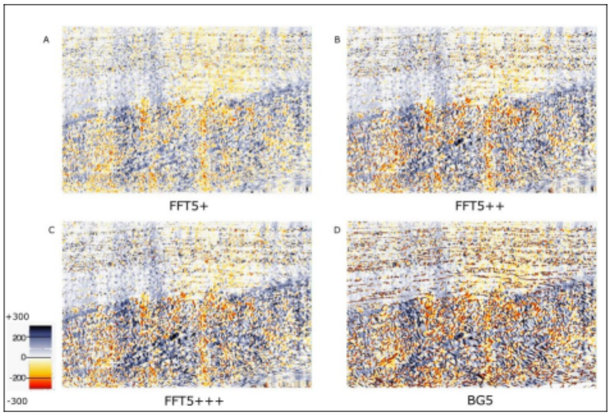
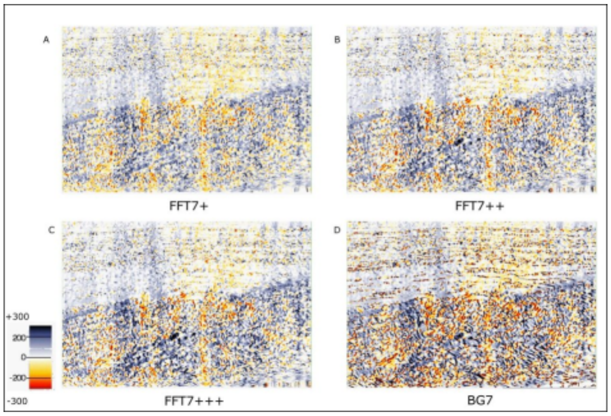


Figure 5. Crossline dip with (calculation cube size=7): precise steering (A), combined steering (B), standard steering (C) and BG steering (D).



From *Figure 3* to *Figure 5* it is notable that there is no significant difference between the combined steering algorithm- (FFT++) and precise steering algorithm (FFT+)speed and accuracy. The standard steering algorithm (FFT+++) is fast but apparently often produces erroneous results in high dip areas. In order to avoid getting a noisy steering cube the calculation cube-size of the FFT algorithms has to be at least 5 or 7. The latter is the default setting. The BG algorithm has a different behaviour: a cube size of 3 seems to be sufficient, but the raw steering cube is useless and a median filtering appears to be mandatory.

Filtering of the SteeringCubes

Figure 6 and *Figure 7* show the results after applying a median filter with different step-outs to the steering cubes. It is apparent that no lateral filtering and only a small vertical filtering, gives the best result with the FFT algorithm.

The BG steering needs to be filtered in the lateral and vertical direction. The best results were obtained with median filter with step-outs 1 1 3. After filtering, the outputs of the precise steering with cube size 7 and median filtered with step-out of 2 only in the vertical direction (FFT7+ med002) and Fast BG steering median filtered with the step-outs 1 1 and 3 (BG3 med113) are very similar in accuracy. However, the latter is produced 10 times faster.

Figure 6. Filtering of FFT+ using calculation cube size = 7: raw (A), median filter with step outs 0 0 2 (B), median filter with step-outs 0 0 4 (C), median filter with step outs 1 1 2 (D).

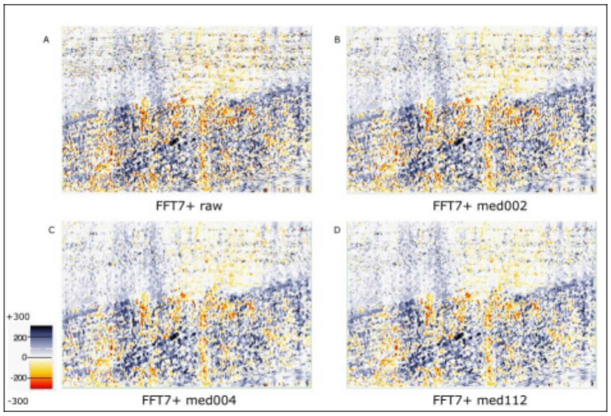


Figure 7. Filtering of BG steering using calculation cube size=3: raw (A), median filter with step-outs 0 0 2 (B), median filter with step outs 1 1 1 (C), median filter with step outs 1 1 3 (D).

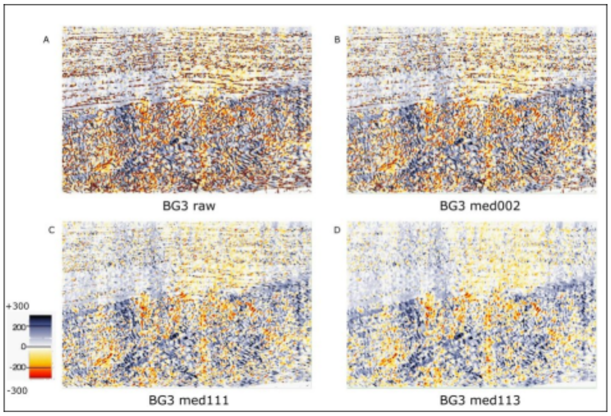
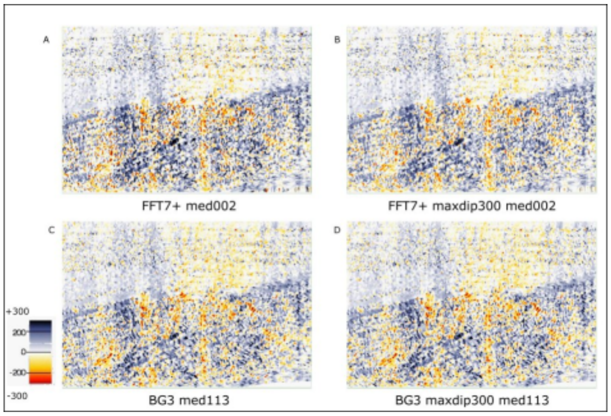


Figure 8 shows that adding a dip limit during the processing does not affect the speed of the algorithms. The final result is strictly identical when the dip is lower than the limit L, and the extreme values are rounded toward L. Using a limit requires a priori knowledge and is in the end a choice of the interpreter.

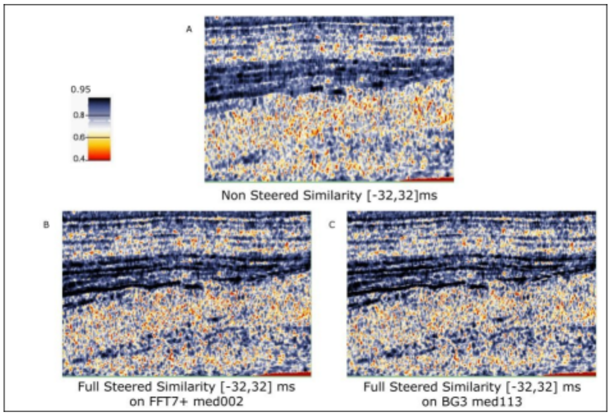
Figure 8. Filtering of FFT+ using calculation cube size=7: median filter with step-outs 0 0 2 (A) median filter with step-outs 0 0 2 and maximum dip of 300 (B), and filtering of BG steering using calculation cube size=3: median filter with step-outs 1 1 3 (C), median filter with step-outs 1 1 3 and maximum dip of 300 (D).



Steered Similarity Attribute

Figure 9 displays the Similarity attribute for the algorithms FFT+ and BG. As an extra reference, the non-steered similarity is added. All figures were calculated with the time gate [-32,32] ms.

Figure 9. Positive curvature using perfect FFT (time gate from[-32,32]ms): no steering (A), precise steering with cube size =7 (B), BG steering with cube size =3 (C).



Choosing a Steering Algorithm

Different Steering algorithms are available. The *precise* FFT algorithm yields an almost perfect SteeringCube at the costs of considerably longer calculation times. The *BG Fast Steering* algorithm seems to fit 95% of the situations and is very fast. dGB recommends the use of this algorithm, using its default calculation cube size of 3 and additional median filtering 113. Although depending on the geology, data quality, available computation time and purpose other choices can be made. A number of examples are presented:

- For a dataset of good quality with only small and low variance in dips, the *BG Fast Steering* method performs well enough. With a median filtering with step-out 1 1 3 applied the result is already very acceptable.
- For a poor quality dataset, one of the *FFT* algorithms should be used because the *BG Fast Steering* algorithm is sensitive to noise and will produce too much outliers. Minor vertical filtering might also help improve the results.
- For detailed studies at target level, the *precise FFT* algorithm can be considered for a sub-volume within the area of interest.
- When creating chrono-stratigraphy, the *Event Steering* is optimal because it looks for similar events (min, max) on neighboring traces in inline/crossline directions.

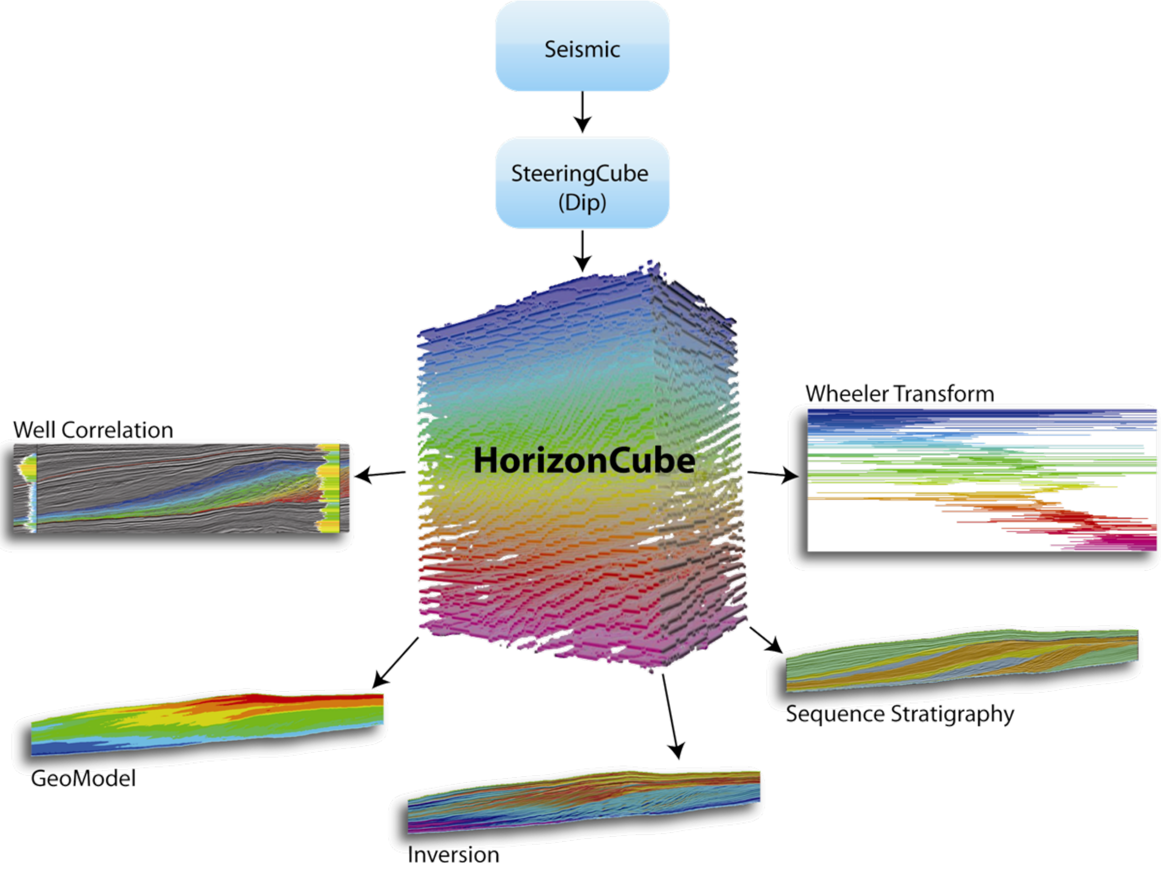
It is always possible to go back and spend much more time in producing a SteeringCube using the FFT precise algorithm.

HorizonCube

Table of Contents

- Introduction
- Processing
- Analysis
- Import/Export

Introduction



HorizonCube: A new plugin that auto-tracks a dense set of mapped 3D stratigraphic surfaces. The HorizonCube impacts all aspects of seismic interpretation work and allows the interpreter to extract more geology from the data. The HorizonCube is used for: detailed geologic model building, improving seismic inversion, sequence stratigraphic interpretation (SSIS) and correlating wells (Well Correlation Panel).

Purpose: The HorizonCube will impact the entire seismic interpretation workflow, leading to significant improvements, including:

- More accurate low frequency model building and robust geological models
- Superior quantitative rock property predictions
- Easy detection of stratigraphic traps (sequence stratigraphy)

Background: The HorizonCube plugin was first launched as the Chronostratigraphy that was part of the OpendTect SSIS (Sequence Stratigraphic Interpretation System) plugin. It consists of a dense set of correlated 3D stratigraphic surfaces that are assigned a relative geological age, with a corresponding colour. It was not long before we realized the potential of the HorizonCube and that the number of applications derived from HorizonCube exceeded just the sequence stratigraphy domain and had applications across the interpretation workflow. The stand-alone HorizonCube plugin was born with the 'HorizonCube' separated from SSIS.

Today, users can look forward to the following benefits:

Low Frequency Model Building & More Accurate, Robust Geological Models
In standard inversion workflows, the low-frequency model is considered the weakest link. Now, users can create highly accurate low frequency models by utilizing all the extracted seismic events from the HorizonCube, allowing a detailed initial model to be built.

In a similar fashion rock properties can be modeled. Instead of using only a few horizons, all horizons of the HorizonCube are used, resulting in greatly improved rock property models.
The HorizonCube & other Plugins .

Rock Property Predictions
The highly accurate low frequency models can be used to create geologically correct Acoustic Impedance (AI) and Elastic Impedance (EI) cubes using the Deterministic and Stochastic Inversion plugins. To complete the workflow, the Neural Networks plugin is used to predict rock properties from the Acoustic Impedance volume, avoiding the use of oversimplified linear models which can not accurately describe most rock property relations.

These advanced tools bring a high degree of precision to traditional seismic workflows, resulting in better seismic predictions and more accurate input into the reservoir management decision-making process.

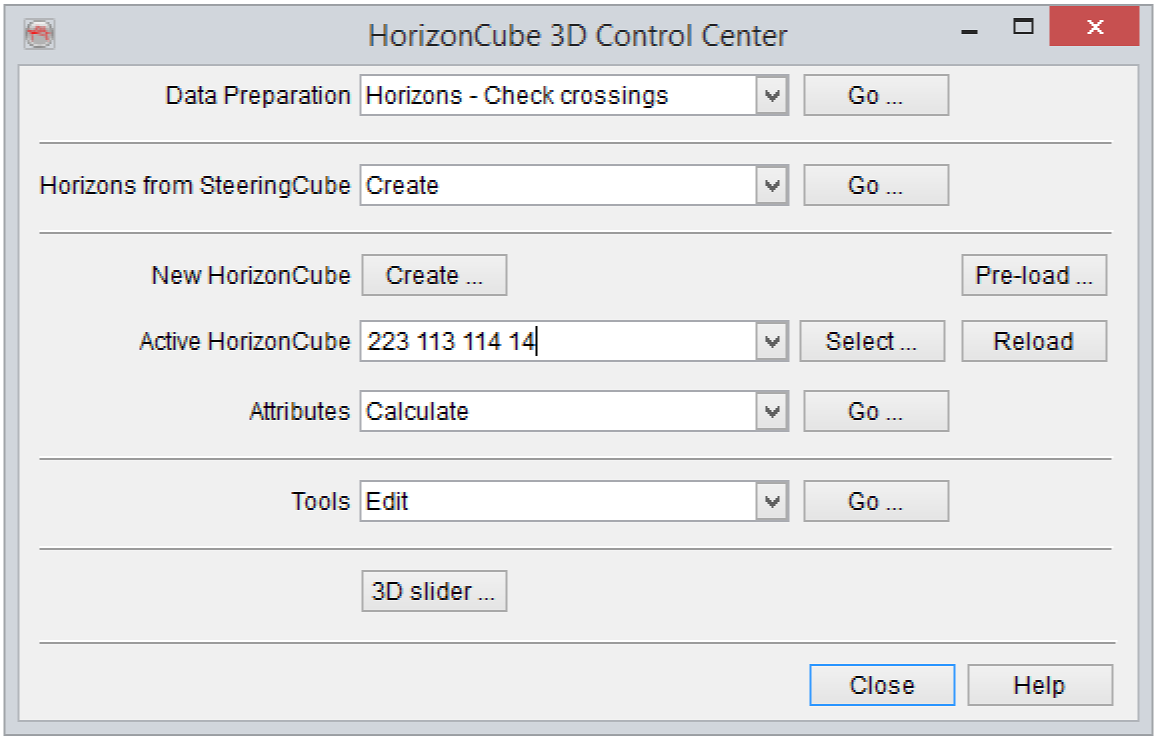
Sequence Stratigraphy (SSIS plugin)
The SSIS plugin works on top of the HorizonCube plugin. Users can interactively reconstruct the depositional history in geological time using the HorizonCube slider, flatten seismic data in the Wheeler domain, and make full system tracts interpretations with automatic stratigraphic surfaces identification and base-level reconstruction.

In the near future the SSIS plugin will be integrated with the Well Correlation Panel, enabling the HorizonCube slider and systems tracts interpretation to be integrated with the well correlation display and its interactive functionalities.

Processing

See the following chapters below.

HorizonCube Control Center



Data Preparation: Horizons - Check crossings

Horizons from SteeringCube: Create

New HorizonCube

Active HorizonCube: 223 113 114 14

Attributes: Calculate

Tools: Edit

It is a selection menu that can be placed anywhere on the desktop while working with OpendTect. It is designed as a control box that contains several sub-menus inside. The control box is used to run the followings:

1- **Data preparation**

2- **Horizons from SteeringCube**

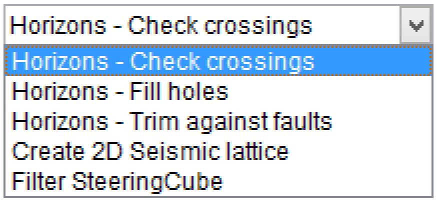
2- **Create a New HorizonCube**

3- **HorizonCube Attributes**

4- **HorizonCube Tools**

5- **3D Slider**

Data Preparation



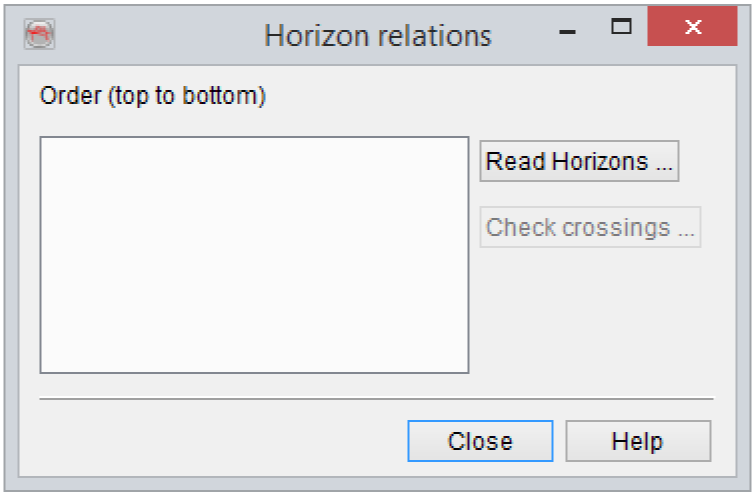
A list menu from the ***Control Center***

To create a HorizonCube, the following data is required: A framework horizons, a good steeringcube, optionally faults / faultsticksets can be added.

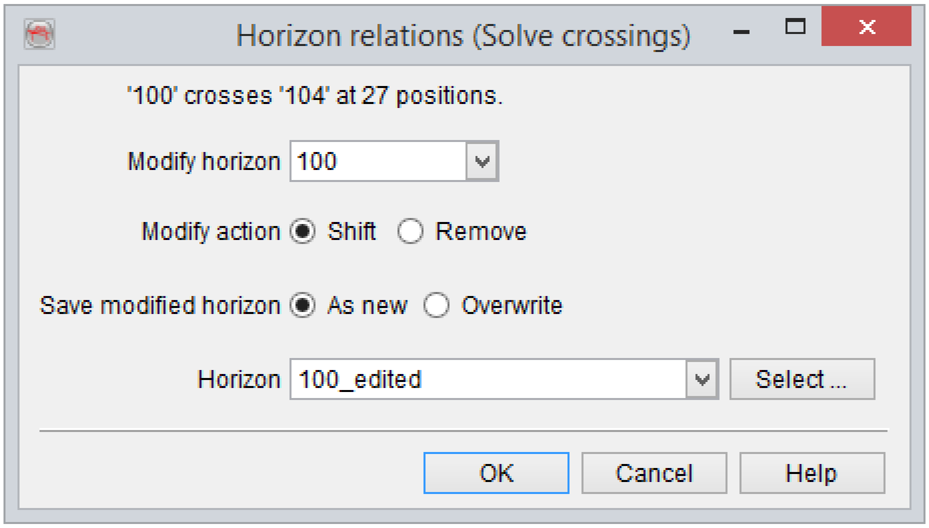
It should be noted that the quality of a HorizonCube is directly dependent on the quality of the above data. Most often the required data has some issues e.g. large gaps in horizons interpretation that requires gridding/filling gaps, the horizons are not tied with faults properly (trimming), horizons cross with each other, and so on. Therefore, several data preparation utilities are made available to fix such issues.

Horizons - Check Crossings

It is used to resolve conflicts between horizons crossing each other. Press the Go button for the Data preparation step to launch the Horizon relation window. The later window is used to select all horizons that need checking (*Read Horizons* ...). The horizons are 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.



'100' crosses '104' at 27 positions.

Modify horizon 100

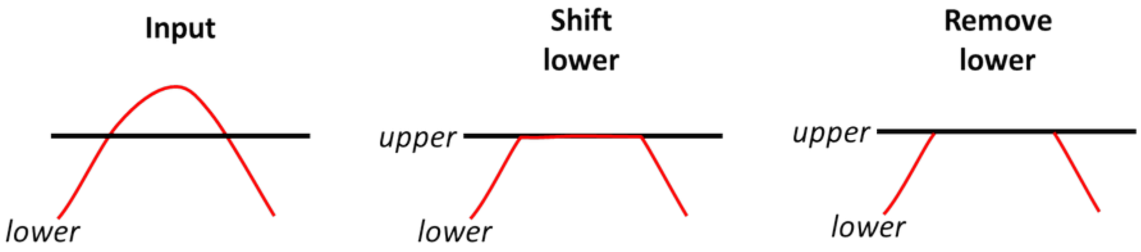
Modify action ● Shift ○ Remove

Save modified horizon ● As new ○ Overwrite

Horizon 100_edited

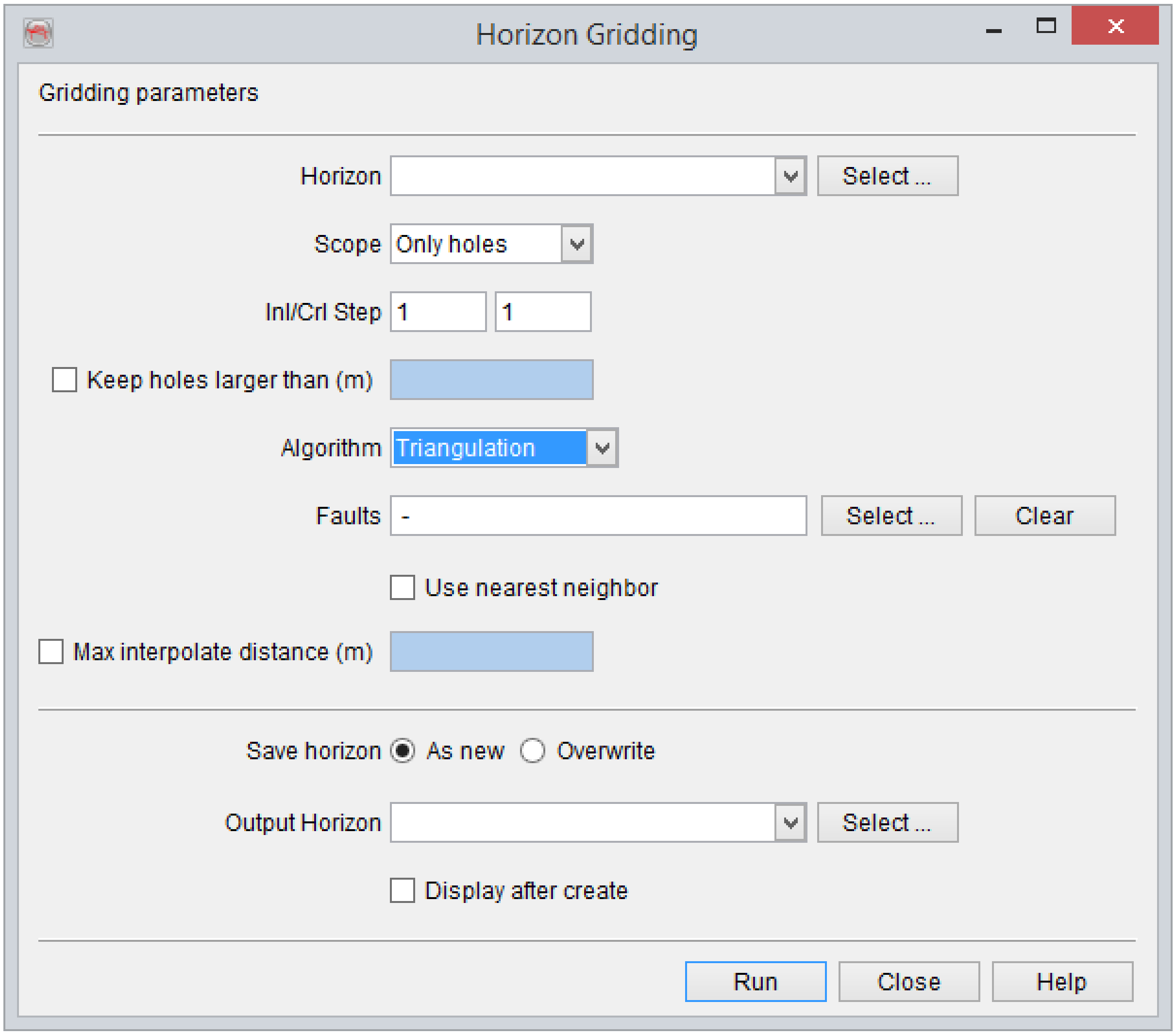
Note: To honor the requirement that horizons cannot coincide, the actual values are not exactly equal, but they are within one sample position accuracy.

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.



Horizons - Fill Holes, Gridding

Holes in horizons can lead to unexpected HorizonCube results. It is thus recommended to work with continuous horizons. The fill holes utility is based on either an iterative application of an inverse distance or triangulation algorithms. In successive steps, holes are interpolated which means that the algorithms stay local in each step.



Horizon Gridding

Gridding parameters

Horizon

Scope: Only holes

Inl/Crl Step: 1 1

☐ Keep holes larger than (m)

Algorithm: Triangulation

Faults: -

☐ Use nearest neighbor

☐ Max interpolate distance (m)

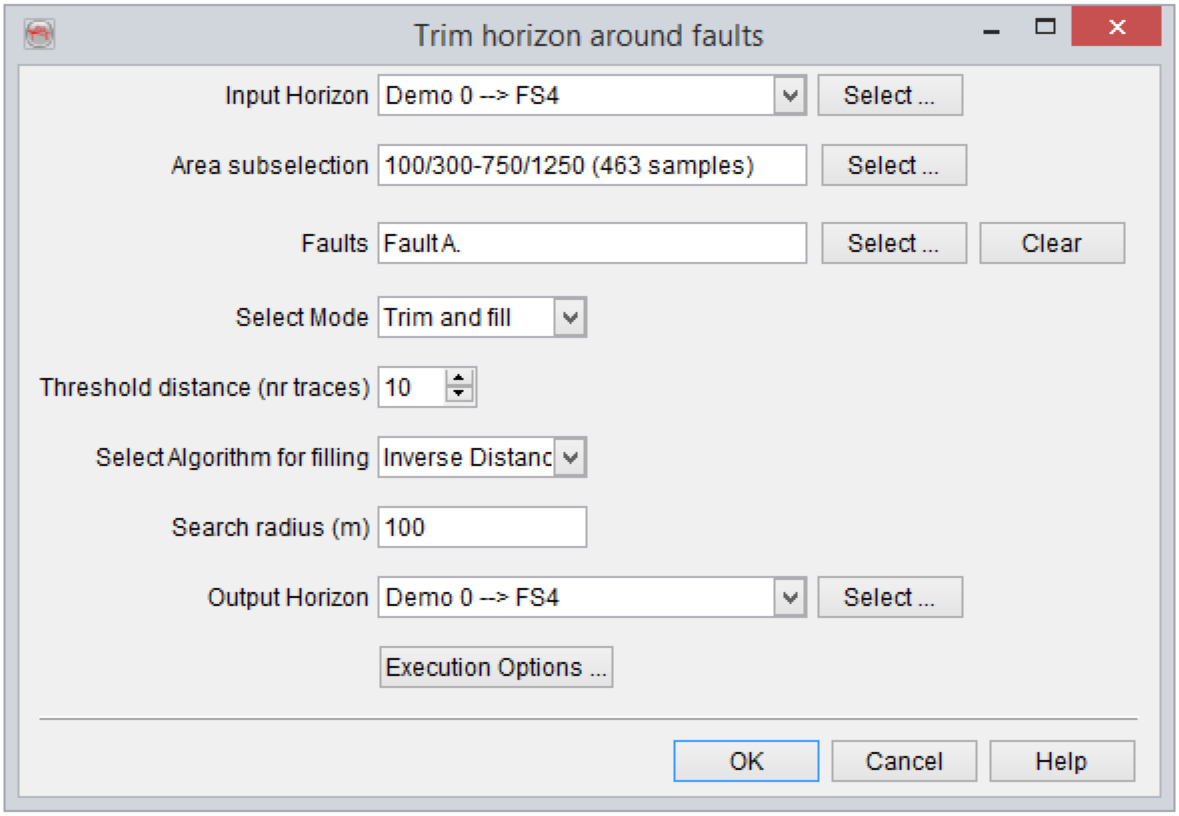
Save horizon ● As new ○ Overwrite

Output Horizon

☐ Display after create

Horizon - Trim at Faults

The trimming feature in OpendTect improves the fault/horizon relation.



Input Horizon: Demo 0 --> FS4

Area subselection: 100/300-750/1250 (463 samples)

Faults: Fault A.

Select Mode: Trim and fill

Threshold distance (nr traces): 10

Select Algorithm for filling: Inverse Distanc

Search radius (m): 100

Output Horizon: Demo 0 --> FS4

There are different modes to trim a horizon at given faults location.

Trim and Track

It is an option to remove an area defined by the trace threshold away from a fault plane. The removed part is then tracked back to the fault plane using a selected SteeringCube. Please note that on an average seismic data close to a fault plane, seismic data could be very noisy. Thus you can expect that a computed SteeringCube could also be very noisy. Therefore, it is suggested to use a background (heavily filtered) SteeringCube for this purpose.

Trim only

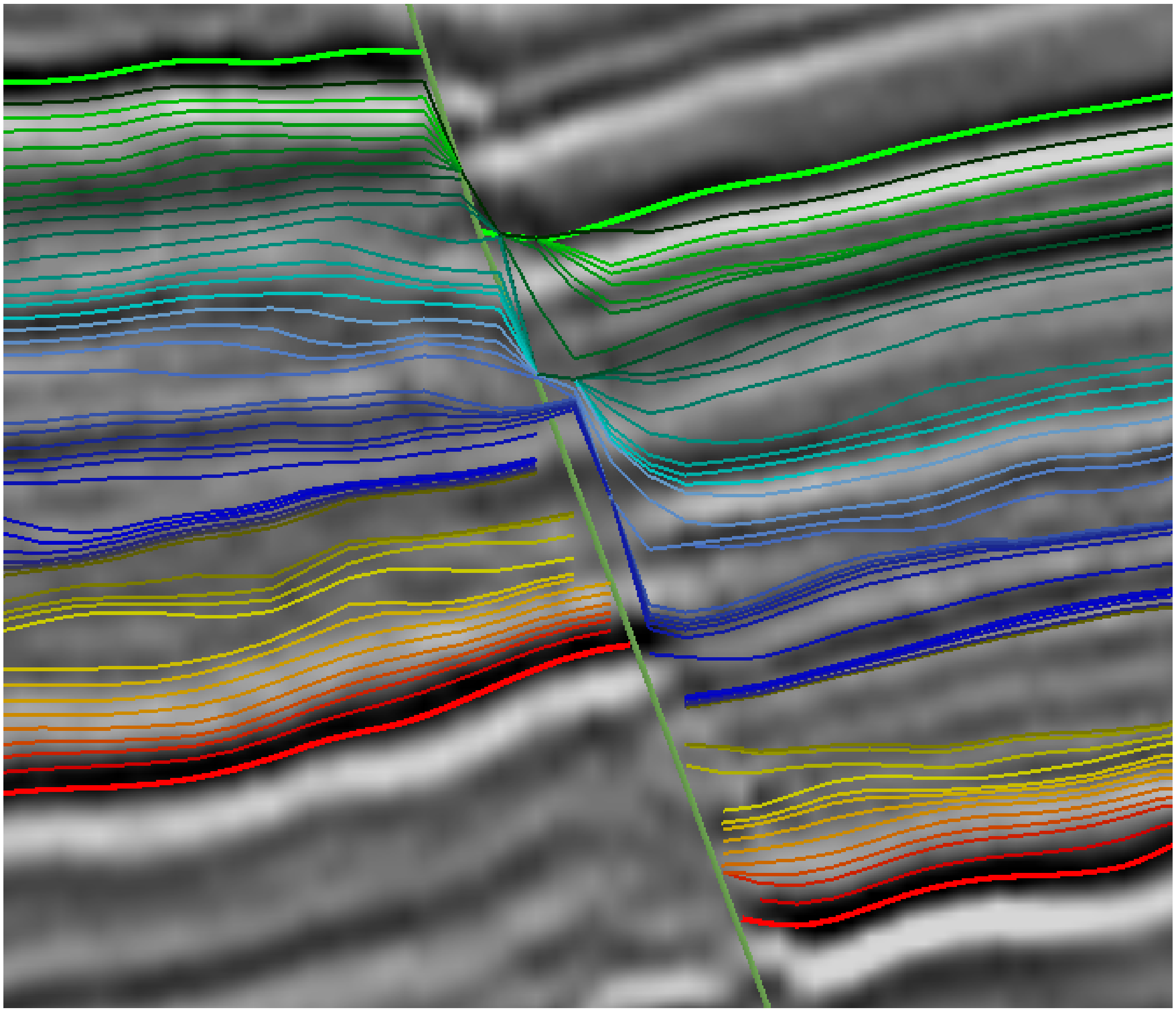
This will only remove the parts of a given horizon within a defined trace radius close to a fault plane.

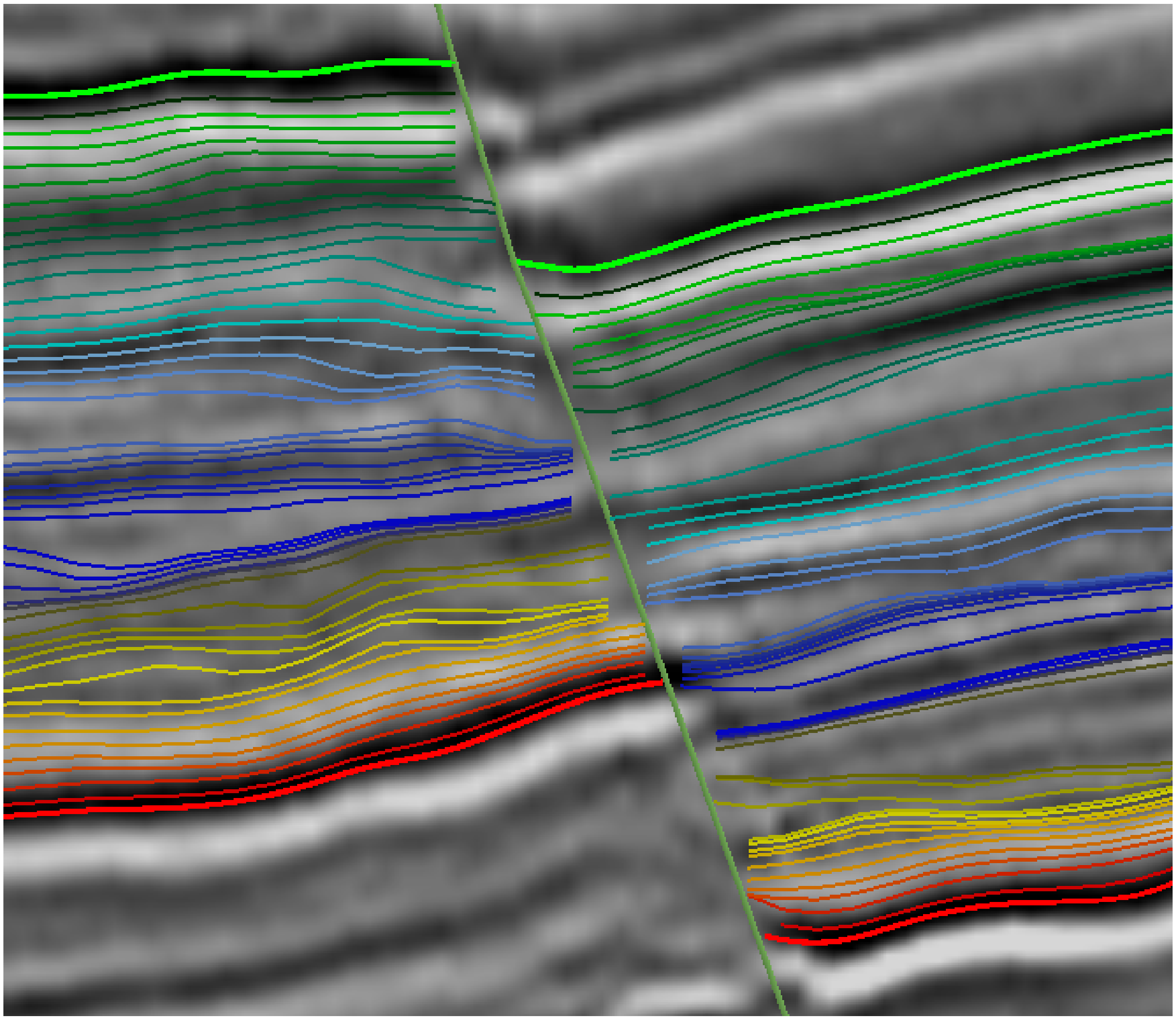
Re-track only

This option will track a horizon close to a fault.

The threshold should be chosen according to 'neatness' of the horizon around the fault. If a unnecessarily large threshold is chosen, a large part of the horizon will be deleted and re-tracked, thus making it prone to errors. Ideally, a threshold of 10 is sufficient (default value), whereas for an 'indisciplined' horizon a threshold of 30 or above may be needed. The same horizon may behave differently around different faults. Different thresholds for the same horizon but different faults or fault-sets may be chosen. In this case the trimming should be done in steps:

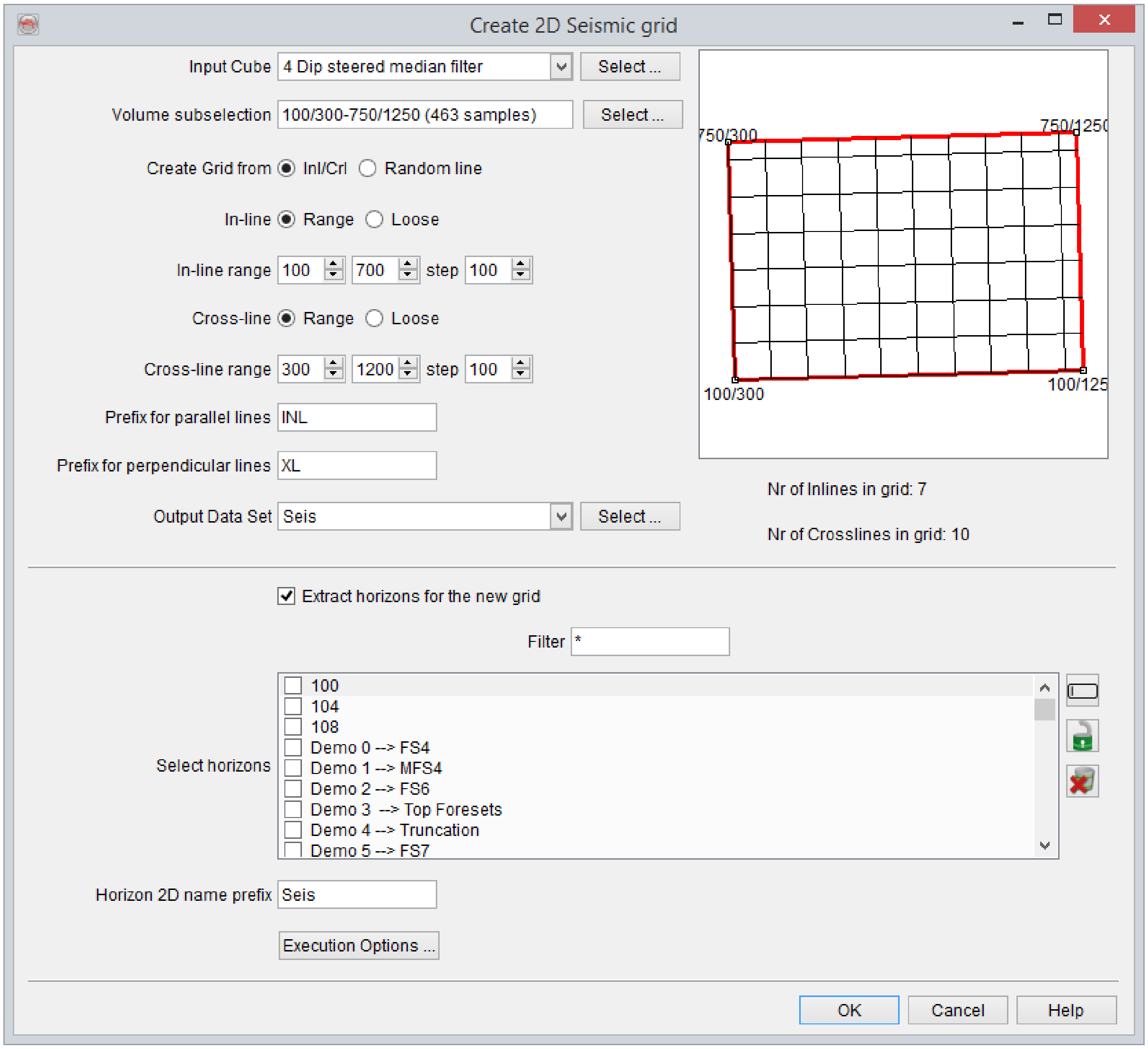
Original horizons ----> using 1st set of faults and a small threshold --> Horizon 1
Horizon 1 ----> using 2nd set of faults and slightly larger threshold --> Horizon 2
Horizon 2 ----> using 3rd set of faults and a large threshold --> Final output Horizon
An example of before- and after trimming is shown below:





Create 2D Seismic Lattice

This tool is used to create a 2D lattice/grid from 3D seismic data. This feature is released as a Multi2D workflow in the SSIS plugin (Chapter 4). The workflow is simple i.e. convert 3D seismic data into a coarse 2D grid with preferred orientation and geometry. After the conversion, the next step is to prepare a HorizonCube for all 2D lines. Using the HorizonCube for each 2D line, interpret sequence stratigraphic surfaces. The extracted sequence stratigraphic surfaces are then used to do detailed interpretations on the 3D seismic volume.



Create 2D Seismic grid

Input Cube: 4 Dip steered median filter

Volume subselection: 100/300-750/1250 (463 samples)

Create Grid from: ● Inl/Crl ○ Random line

In-line: ● Range ○ Loose

In-line range: 100 700 step 100

Cross-line: ● Range ○ Loose

Cross-line range: 300 1200 step 100

Prefix for parallel lines: INL

Prefix for perpendicular lines: XL

Output Data Set: Seis

Nr of Inlines in grid: 7

Nr of Crosslines in grid: 10

☑ Extract horizons for the new grid

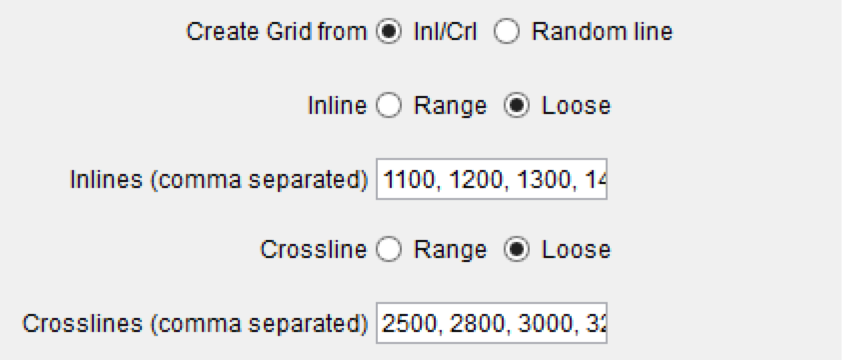
Filter: *

Select horizons:
- ☐ 100
- ☐ 104
- ☐ 108
- ☐ Demo 0 --> FS4
- ☐ Demo 1 --> MFS4
- ☐ Demo 2 --> FS6
- ☐ Demo 3 --> Top Foresets
- ☐ Demo 4 --> Truncation
- ☐ Demo 5 --> FS7

Horizon 2D name prefix: Seis

- **Input Cube:** Selected 3D seismic data to be used in this field.
- **Volume Subselection:** Optionally, the 3D volume can be restricted to an objective area. This subselection is made here by restricting inline/crossline/time ranges for the selected volume.
- **Create Grid from:** Define the output grid geometry and orientation of the 2D lines to be created. If Inl/Crl is selected for this option, the general inline/crossline orientation (or geometry) of the selected 3D data would be used within the defined inline/cross-line range and the corresponding steps. Optionally, the inline/crossline range can be edited manually by setting loosely spaced inline/crossline numbers separated by

commas.



Create Grid from ● Inl/Crl ○ Random line

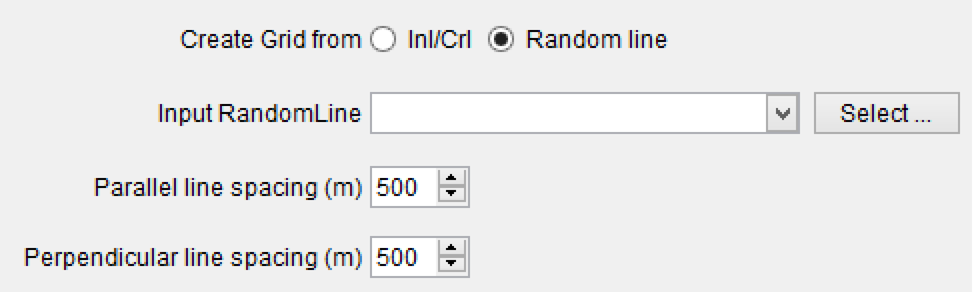
Inline ○ Range ● Loose

Inlines (comma separated) 1100, 1200, 1300, 14

Crossline ○ Range ● Loose

Crosslines (comma separated) 2500, 2800, 3000, 3

Another option is to define a 2D grid geometry using a *Random Line*. Using the selected random line, the grid can be created in both (parallel and perpendicular) directions of the randomline. This fixed spacing needs to be given in the *line spacing (m)* fields.



Create Grid from ○ Inl/Crl ● Random line

Input RandomLine

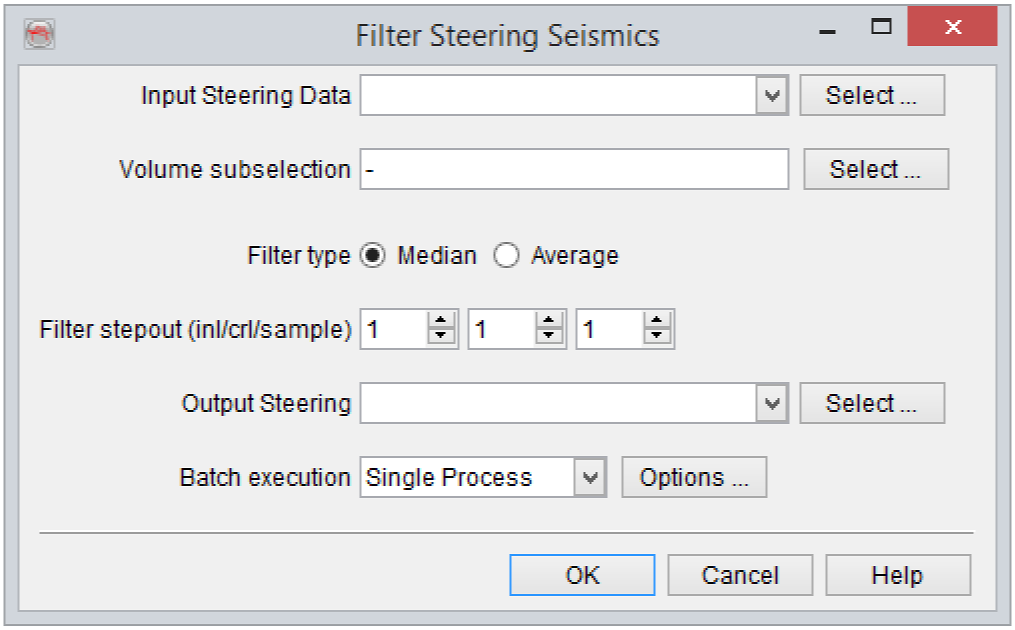
Parallel line spacing (m) 500

Perpendicular line spacing (m) 500

- **Prefix:** Label the 2D-line names in the fields.
- **Output Lineset:** Output for the 2D lineset. Please provide the lineset name and the name for the seismic data (attribute).
- **Extract Horizons:** Convert the 3D interpreted horizons into 2D horizons by checking this box (optional). In the *Select horizons list*, select one or more horizons.

Filtering the SteeringCube

The SteeringCube can be improved through smoothing. A good SteeringCube improves the HorizonCube calculations. Use the option *Filter SteeringCube*... to apply a median filter to the SteeringCube. Adjust the needed parameters: inline/crossline step-out, time gate size, and maximum dip according to your dataset. Filtering of the SteeringCube is described in **here**



Input Steering Data:

Volume subselection: -

Filter type: ● Median ○ Average

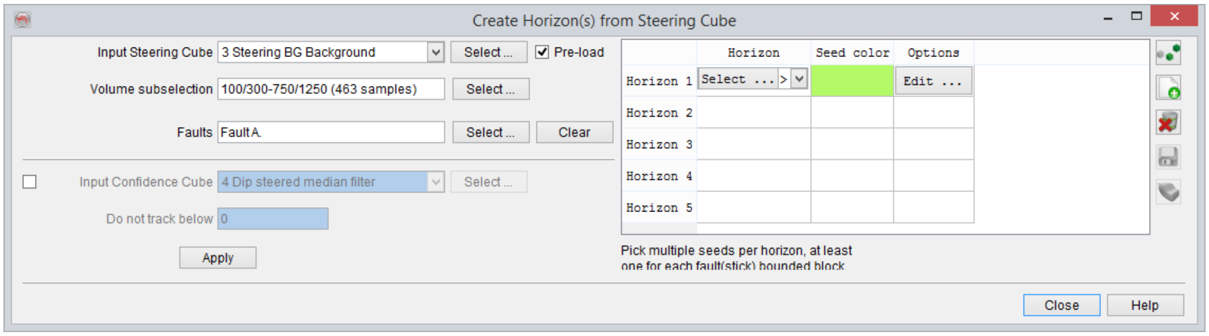
Filter stepout (inl/crl/sample): 1 1 1

Output Steering:

Batch execution: Single Process

Create Horizons from a SteeringCube

dGB provides utility features to create and to manipulate seismic horizons using SteeringCube. The utilities use an ultrafast algorithm to track horizons within a seismic volume. It requires a SteeringCube to produce full 3D horizons within a few seconds or minutes, without the need for griding or interpolation. These horizons can be used as an input to a HorizonCube. The utility to create a new horizon from a SteeringCube is launched from the HorizonCube sub-menu "*Create horizons from SteeringCube*".



Input Steering Cube: 3 Steering BG Background ☑ Pre-load

Volume subselection: 100/300-750/1250 (463 samples)

Faults: Fault A.

☐ Input Confidence Cube: 4 Dip steered median filter

Do not track below: 0

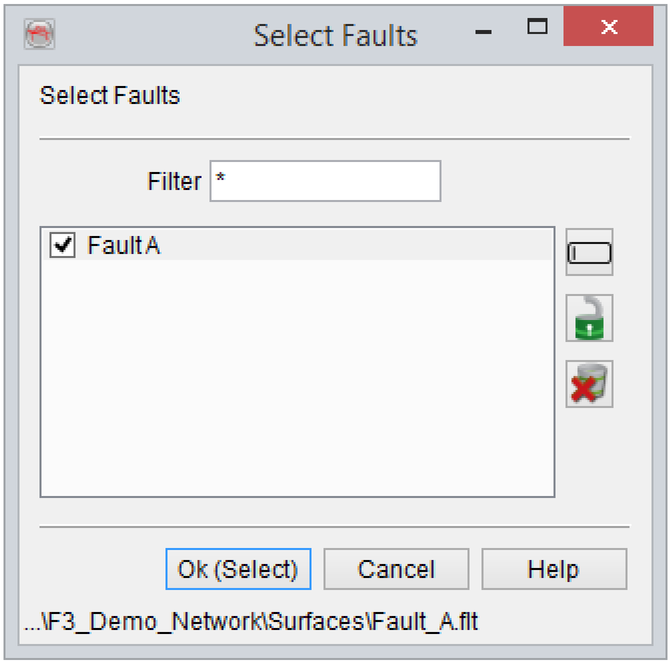
	Horizon	Seed color	Options
Horizon 1	Select ... >	🟩	Edit ...
Horizon 2			
Horizon 3			
Horizon 4			
Horizon 5			

Pick multiple seeds per horizon, at least one for each fault(stick) bounded block

Create horizons from SteeringCube.

To create a new horizon from a SteeringCube, several inputs are required: SteeringCube, faults and seeds. The SteeringCube is selected by clicking the *Select* button for the Input SteeringCube field. Optionally, the volume sub-selection for the SteeringCube can also be made if the intention is to create the output within a sub-volume.

The faults can be provided to adequately incorporate the fault throw while tracking the horizons from the selected SteeringCube. Press the *Select* button infront of the *Faults* field to select one or more faults from the pop-up fault selection dialog.



Filter *

☑ Fault A

...\F3_Demo_Network\Surfaces\Fault_A.flt

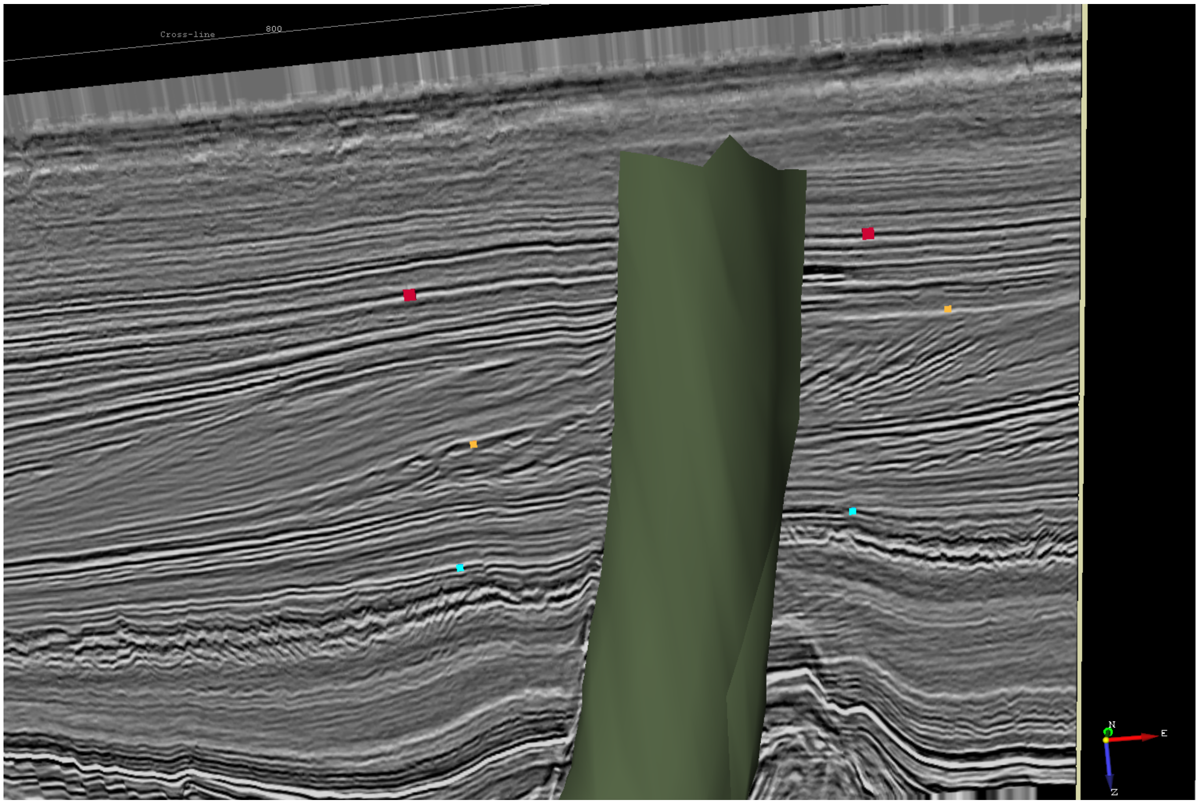
Furthermore, the Pre-load full volume is checked to load the SteeringCube into the memory before tracking horizons. This speeds up the tracking of multiple horizons. It is suggested that if the SteeringCube size is more than the size of the memory, please leave this option unchecked.

Apply button is used to start processing and create horizons. Once the horizon tracking is finished, the horizons are displayed in the scene.

In the Create horizon(s) from SteeringCube window, several icons are used to create and to save multiple horizons simultaneously.

- It is used to pick a seed to track a horizon. The seed is picked on a displayed seismic data (Inline/Crossline) in the scene. When this button is pressed and a horizon is selected from the table (click the corresponding row), the user is ready to pick one seed per horizon. Optionally, if the fault(s) are selected, the multiple seeds can be inserted for each fault block.

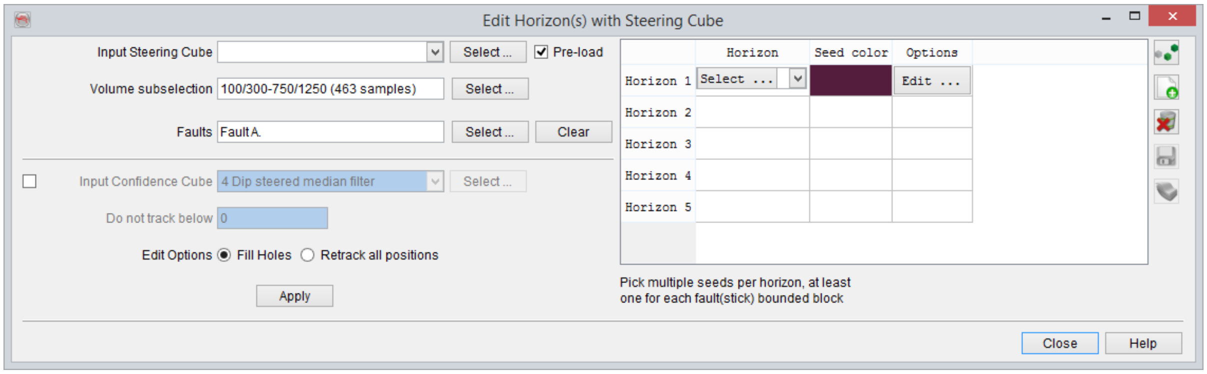
- This button adds a new horizon in the table.
- To remove the horizon from the table press this button.
- It saves the tracked horizons in the OpendTect survey.
- It launches the HorizonCube Creator setup (see **Create HorizonCube**)



An example inline on which the seeds (red, yellow, and blue) for three horizons are picked. Note that one seed is picked within a fault block. The current algorithm tracks a full horizon using one seed only and extends tracking outward within the range of the SteeringCube.

Edit Horizons with a SteeringCube

To fill holes or to re-track an existing horizon using SteeringCube, the similar utility can be launched from the HorizonCube Sub menu (Processing > HorizonCube > *Edit horizons with SteeringCube*. This will bring the following window. This window is almost identical to the Create Horizon(s) from SteeringCube window. The SteeringCube is selected for the *Input Steering Cube* field. Additionally, the faults can also be selected. The *Edit Options* are used to either fill the holes/gaps in an horizon using SteeringCube or to re-track a horizon using SteeringCube.



Input Steering Cube
☑ Pre-load
Volume subselection: 100/300-750/1250 (463 samples)
Faults: Fault A
☐ Input Confidence Cube: 4 Dip steered median filter
Do not track below: 0
Edit Options: ● Fill Holes ○ Retrack all positions

	Horizon	Seed color	Options
Horizon 1	Select ...	■	Edit ...
Horizon 2			
Horizon 3			
Horizon 4			
Horizon 5			

Pick multiple seeds per horizon, at least one for each fault(stick) bounded block

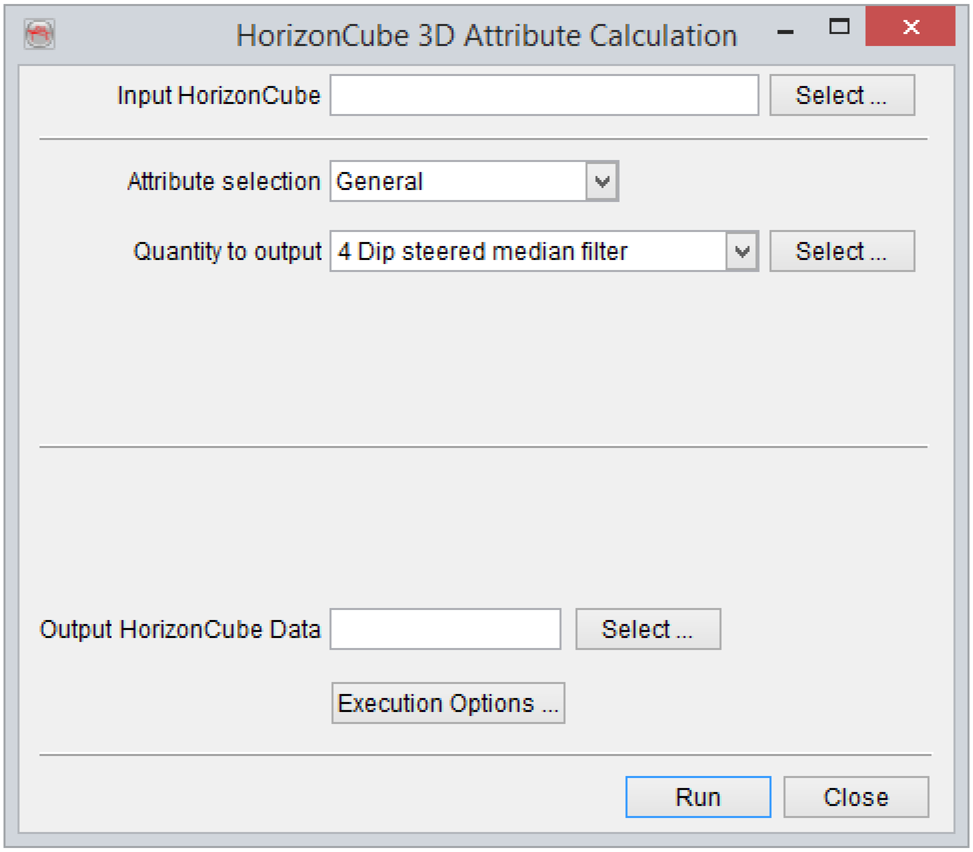
Edit existing horizon(s) using the selected SteeringCube.

Calculate HorizonCube Attributes

Calculate Attributes along 3D HorizonCube

These attributes are computed from the **HorizonCube Control Center**. The output of all of these attributes is a set of grids, which are collectively saved as **HorizonCube Data**. These attribute grids can be visualized using **3D Slider**.

In the *Select HorizonCube* option, you can select the entire HorizonCube, or a package or an event.



Input HorizonCube

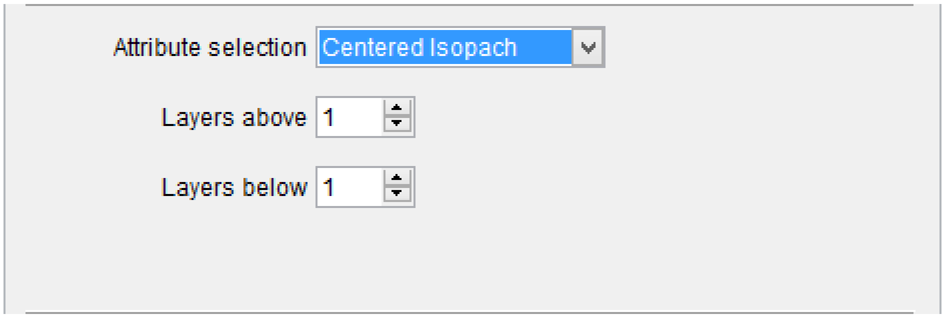
Attribute selection General

Quantity to output 4 Dip steered median filter

Output HorizonCube Data

General: This category is used to compute any existing attribute (stored or defined) along HorizonCube events. For instance, if you already have a stored AI or PHI volume and you would like to compute it along HorizonCube, you can use this option.

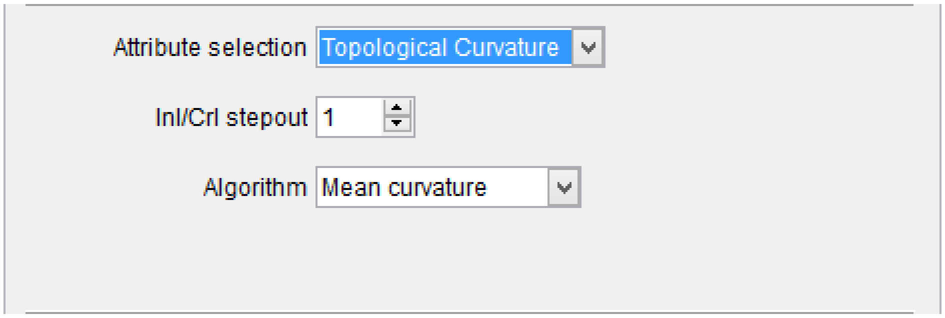
Centered Isopach: This attribute calculates the vertical thickness (TWT/depth) between two events such that they are centered by an event. In this manner, the thickness is computed along the centered event. Number of layers above/below define the relative number of events to compute the thickness. For instance, if the value for layers above/below is set to 1, for each central event, the thickness is computed by subtracting the TWT/Depth of an underlying event from the overlying event and storing the result for the central one.


Attribute selection: Centered Isopach

Layers above: 1

Layers below: 1

Topographical Curvature: This attribute defines geometrical curvature of an event, attribute or HorizonCube Data. The stepouts are number of inlines/crosslines to be used to compute the curvature. These attributes are mostly equivalent to the **conventional curvature attributes** that are directly computed from the SteeringCube.

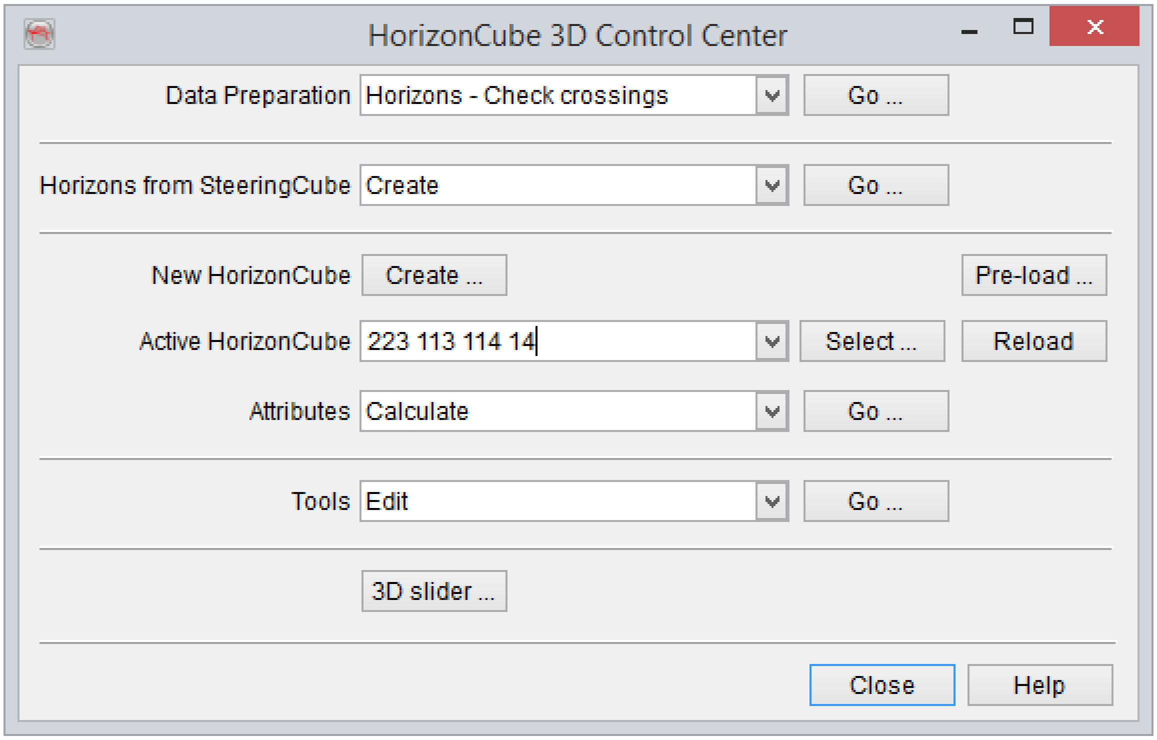

Attribute selection: Topological Curvature

Inl/Crl stepout: 1

Algorithm: Mean curvature

Create HorizonCube (2D, 3D)

A new 2D/3D HorizonCube is created from the **Control Center**.



Data Preparation: Horizons - Check crossings

Horizons from SteeringCube: Create

New HorizonCube

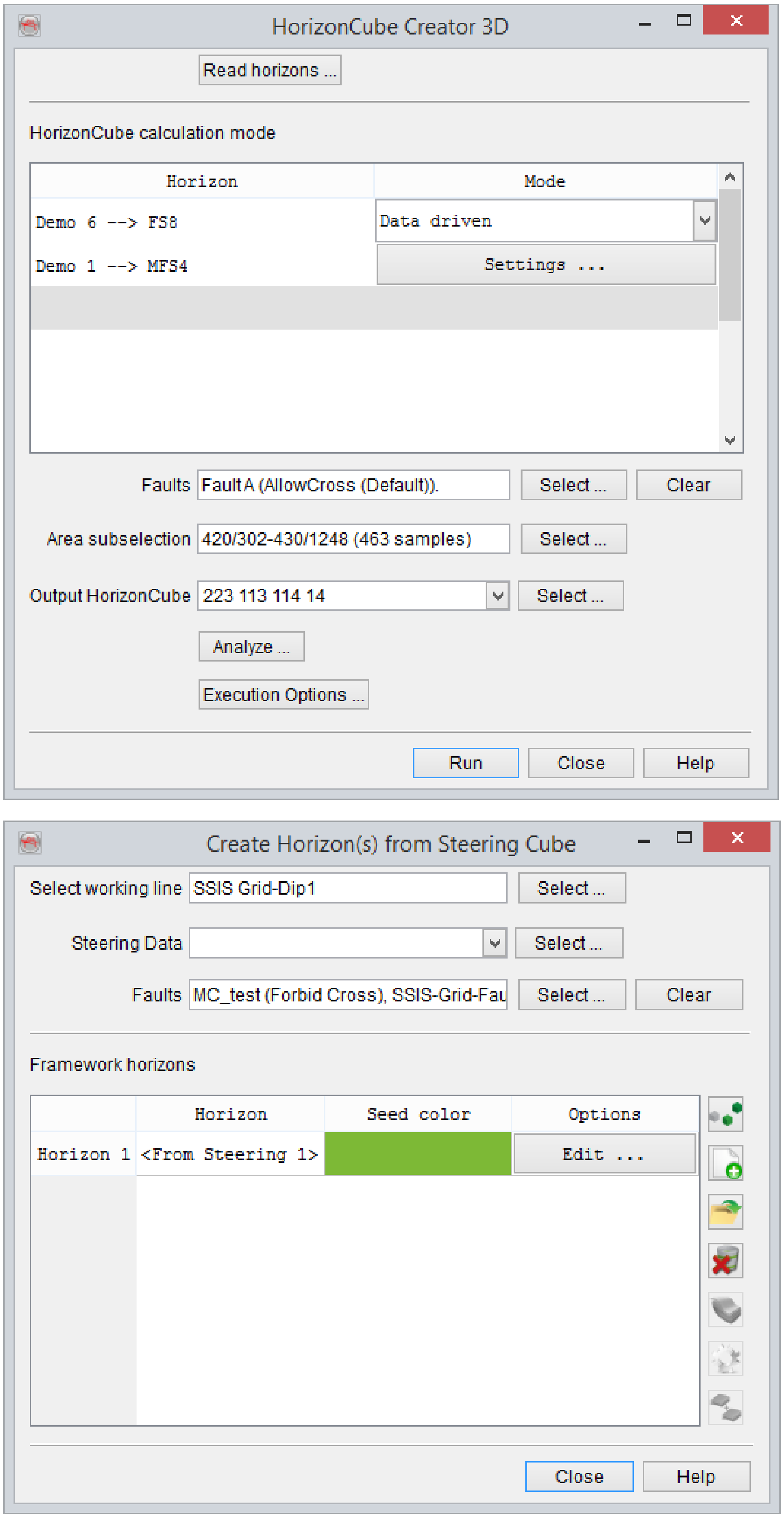
Active HorizonCube: 223 113 114 14

Attributes: Calculate

Tools: Edit

Use the New HorizonCube - Create button

The HorizonCube(2D/3D) is created between the given horizons (2D/3D). These horizons are selected (minimum of 2, top and bottom) by clicking the "*Read horizons*" button. When the horizons are read, they are automatically placed in stratigraphic order and the corresponding packages are defined. Each package is defined by two horizons (a top and base of the package). In case of 2D Horizon Cube, the input lineset/linename are selected to get the input geometrical information. This 2D lines selection is made by pressing the *Select* button next to the LineSet/LineName field in the HorizonCube Creator 2D window.



HorizonCube calculation mode

Horizon	Mode
Demo 6 --> FS8	Data driven
Demo 1 --> MFS4	Settings ...

Faults: Fault A (AllowCross (Default)).

Area subselection: 420/302-430/1248 (463 samples)

Output HorizonCube: 223 113 114 14

Select working line: SSIS Grid-Dip1

Steering Data:

Faults: MC_test (Forbid Cross), SSIS-Grid-Fau

Framework horizons

	Horizon	Seed color	Options
Horizon 1	<From Steering 1>	🟩	Edit ...

HorizonCube Creator-3D HorizonCube Creator-2D

HorizonCube Models

Three models are based on interpolation and one model uses a data driven approach:

Linear interpolation

Parallel to upper

Parallel to lower

Data driven

- **Proportional**: The proportional model can also be used for 3D Stratal Slicing. In settings, the user can specify the spacing between two HorizonCube horizons.
- **Parallel to upper**: This model best depicts the lapout patterns, such as onlaps. In settings, the user can specify vertical spacing between HorizonCube horizons.
- **Parallel to lower**: This model relates to upward truncation patterns. In settings, the user can specify vertical spacing between HorizonCube horizons.
- **Data driven**: This model is driven from data based upon steering information. The HorizonCube will follow the dip and azimuth information read from a **SteeringCube**.

Fault: 3D faults or 2D faultsticksets are selected from the main HorizonCube creator dialog. The user can select more than one fault / faultstickset.

Area Subselection: An area sub selection can be made to restrict the HorizonCube calculation. Please note that the HorizonCube can also be calculated within a polygonal area of interest. To know how to create a polygon in OpendTect, please refer to OpendTect Help documentation. However, this feature is not supported for a 2D HorizonCube.

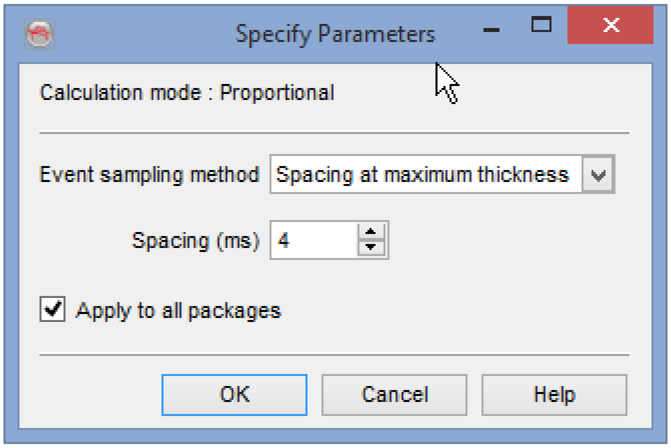
Output HorizonCube: This field is used to give an output name for the HorizonCube.

Analyze: There are several checks being designed to quality control the failure of a batch program running to create a HorizonCube. The cause of failure could be a bad start position e.g. edge of survey, a trace defining a fault location, issues with the framework horizons etc. Therefore, it is always suggested to Analyze the HorizonCube processing before starting the actual processing (Go button).

Go: It starts the HorizonCube processing in a pop-up window. The batch processing window will provide you the instantaneous progress of HorizonCube calculation. Once the batch program prompts "Finished batch processing", the output is ready to be visualized. Press "Show options" to get the possibility for remote processing (after having pressed "Go").

Model Driven Settings

The spacing (in ms / m /ft) is the only parameter required to define the settings for a model driven HorizonCube:



Calculation mode : Proportional

Event sampling method: Spacing at maximum thickness

Spacing (ms): 4

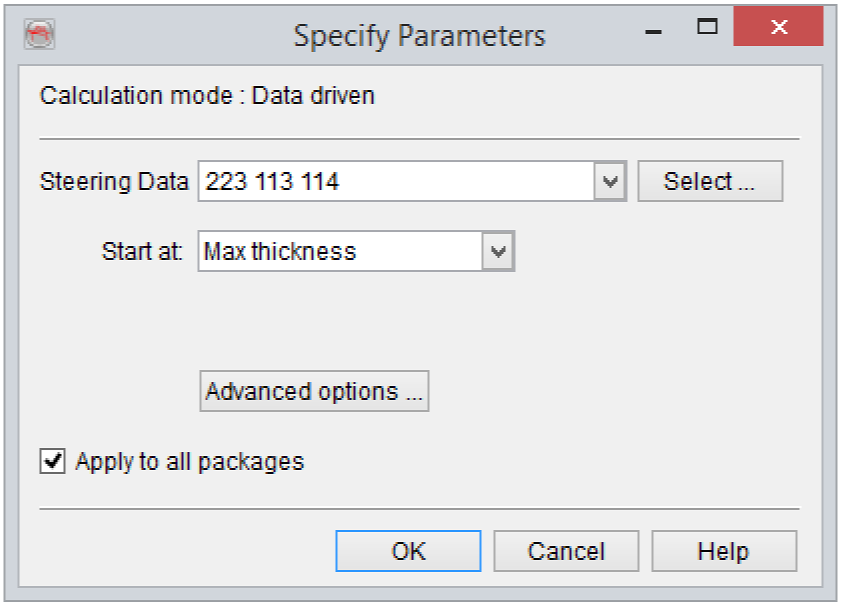
☑ Apply to all packages

Its only utility is to set the sampling rate of the data in the Wheeler domain, and the number of Chronostratic events that can be exported as horizons.

The option "Apply to all sequences" takes effect only when pressing "Ok" and cannot be undone.

Data Driven Settings

A data driven HorizonCube requires a SteeringCube to be selected, that will provide the sole input data for tracking. Thus, the quality of the HorizonCube depends on the quality of SteeringCube itself. Another key parameter to control the quality of HorizonCube is the start position that is defined in the Start at option in the settings dialog. The start position also defines indirectly the number of events to be initiated in the first iteration / pass, since the package thickness varies laterally.


Specify Parameters

Calculation mode : Data driven

Steering Data 223 113 114

Start at: Max thickness

☑ Apply to all packages

Start at maximum thickness: Ideally the start position should be at a maximum thickness defined by the given framework horizons. Sometimes the trace defining the maximum thickness is either at the edge of a survey or in a poor seismic quality area. In such cases, the start position should not be set to maximum thickness. [Tip] The best practices are to create the isochron (or isopach) maps for the given horizons and find out the thicker areas with good quality of seismic data.

Start at center: Center refers to a trace that lies in the middle of a survey.

Start at inline center / maximum thickness: It defines a trace position that lies at a location of maximum thickness but it uses the central trace of that particular inline.

Advanced options for **Continuous Events** and **Truncated Events** are available and explained in the followings sub-sections.

The option "Apply to all sequences" takes effect only when pressing "Ok" and cannot be undone.

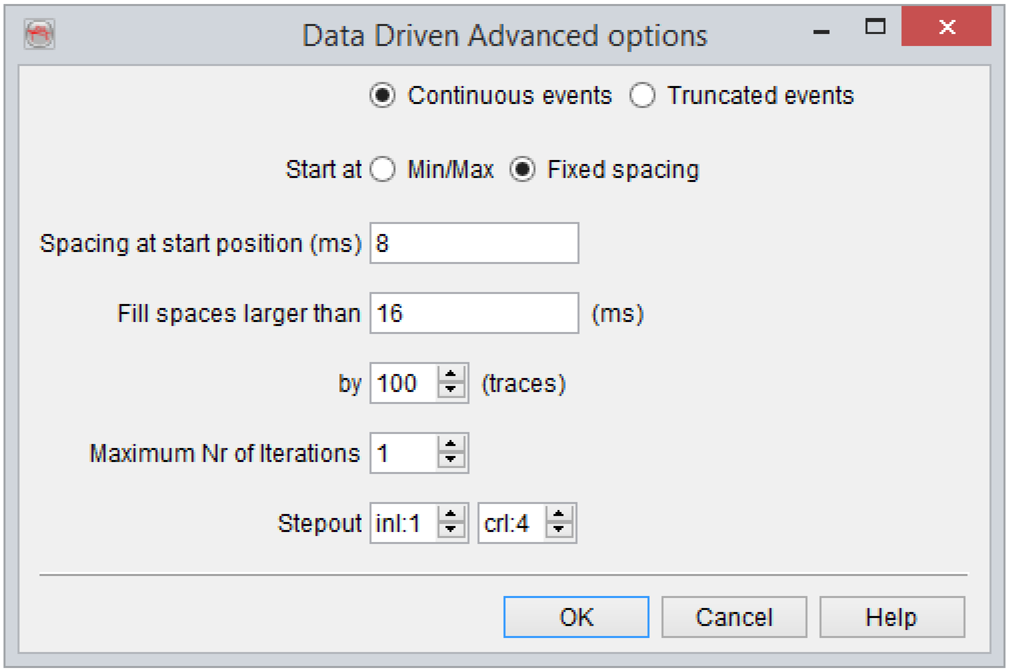
Advanced Options

Continuous / Truncated Events: The continuous events are fully mapped events in 2D/3D that converge / diverge with each other but are not allowed to cross each other. The truncated events are diachronous in 2D/3D i.e. when two horizons come close to each other, the tracking is stopped and a new horizon is initiated afterwards. The continuous HorizonCube is good for GeoModel building or for low frequency model building for seismic inversion. On the other hand, truncated HorizonCube is useful for Sequence Stratigraphic (SSIS) interpretation e.g. wheeler transformation.

The following most important parameters apply to both HorizonCube types:

Spacing at start position: Used at the start position only. Vertical spacing between the seeds from which the HorizonCube events will be initiated. Implies a regular sampling of the events at the start position. The continuous HorizonCube proposes an alternative mode (see corresponding section below).
Smallest spacing (e.g. 4) will result into a dense HorizonCube and a largest spacing (e.g. 16) will result into a coarse HorizonCube.

Stepouts: The stepout (inline : crossline) parameters control the spatial quality of horizon tracking in 3D. It defines the number of z-values (of an event) to be used to forecast the z-value at a new trace position. By default the inline steps are set to 1 (i.e. 3 z-values on a crossline plane) and the crossline steps are set to 4 (i.e. 9 z-values on inline plane). The smaller stepout mean faster and detailed dip field tracking and the largest stepout are preferable for a regionally continuous event. The best practices are to test it with asymmetric parameters (e.g. 1:4 or 4:8 or 1:12). The symmetric steps (e.g. 4:4, 8:8 or even higher 12:12) are useful to average-out very small details / noisy trails while tracking. This parameter is a key to be tested through this utility prior to creating any HorizonCube. It is recommended to find optimal stepout values by varying it for an individual horizon.



● Continuous events ○ Truncated events

Start at ○ Min/Max ● Fixed spacing

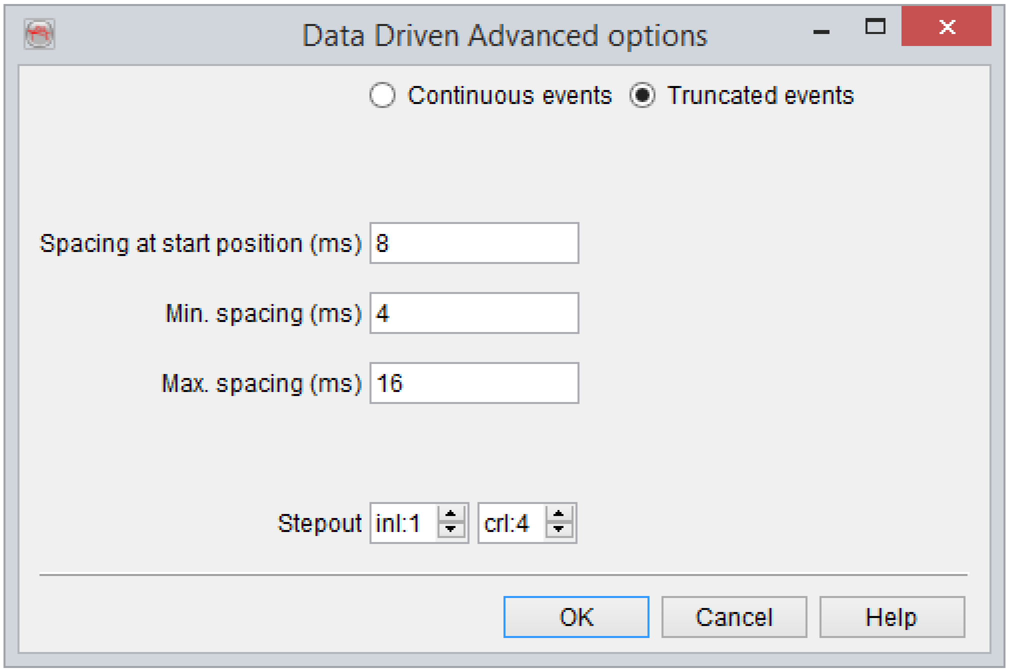
Spacing at start position (ms) 8

Fill spaces larger than 16 (ms)

by 100 (traces)

Maximum Nr of Iterations 1

Stepout inl:1 crl:4



○ Continuous events ● Truncated events

Spacing at start position (ms) 8

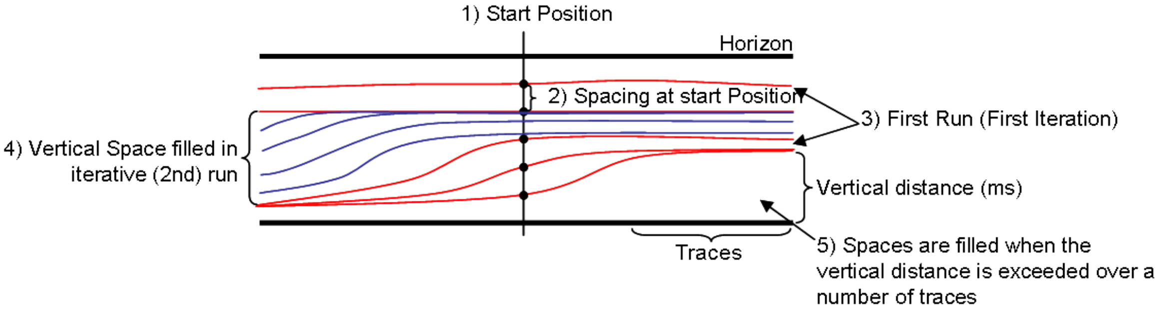
Min. spacing (ms) 4

Max. spacing (ms) 16

Stepout inl:1 crl:4

Continuous Events

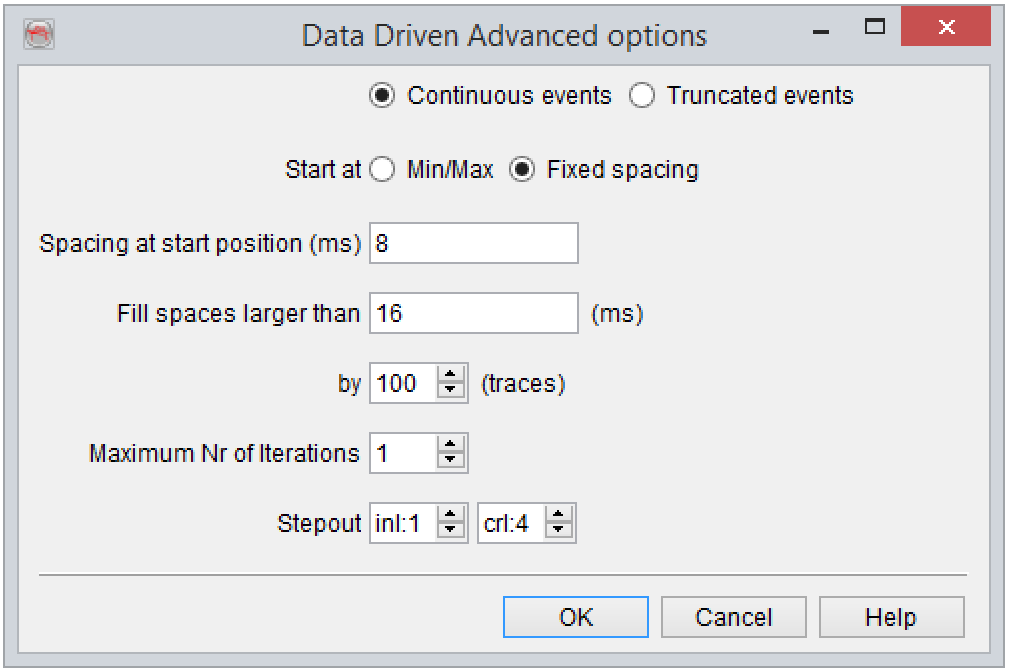
At the start position (1) numerous horizons are initiated at a user defined interval (2). Normally, the sample rate is used here in order to initiate a horizon at every sample. These horizons are tracked from the start position outward within the whole extend of the survey (3). When the horizons diverge large vertical spaces between the horizons are created, which are filled in iterative runs (4). To prevent very small vertical spaces to be filled (with horizons that are present in the whole survey) the spaces are defined by a vertical setting as well as a horizontal (5). A vertical space is filled when the vertical distance between the horizons exceeds a user specified amount over a lateral extend -the user defined number of traces-.



Start at: The (Start at) radio box (for Continuous HorizonCube) is used to define the trace position from which the horizons will be tracked. By default, *Fixed spacing* is used and the corresponding constant value is filled in the Spacing at start position field. The *Min/Max* relates to a given seismic cube (positive / negative amplitudes) i.e. the HorizonCube events will be initiated at a start position defined by the Min/Max amplitudes and this will not yield an evenly spaced HorizonCube at start position.

Fill spaces larger than (ms) or by (traces): This is used to specify the minimum allowed gap vertical (in ms / m / ft) or by distance (m / ft) to be filled in the subsequent defined iterations.

Max. Nr. of iterations Depending on the geologic thickness variations within a defined package, often the gaps are found after the first iteration of a HorizonCube. To fill the gaps in a HorizonCube the initial iteration value should be defined (either 1 or 2). The best practice is to create a HorizonCube with 1 iteration initially and then at later stages the gaps could be filled using the HorizonCube tools (Add more iterations). This is suggested as a quality control step because the HorizonCube calculation is slower for subsequent iterations. For instance, the HorizonCube with 1 iterations and smaller step outs could be generated in 1 or two hours. However, if the iterations are 2, the HorizonCube calculation time exponentially increases.



● Continuous events ○ Truncated events

Start at ○ Min/Max ● Fixed spacing

Spacing at start position (ms) 8

Fill spaces larger than 16 (ms)

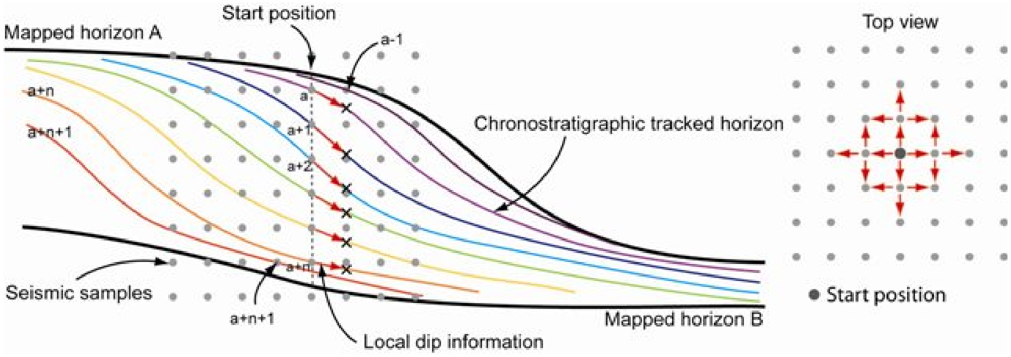
by 100 (traces)

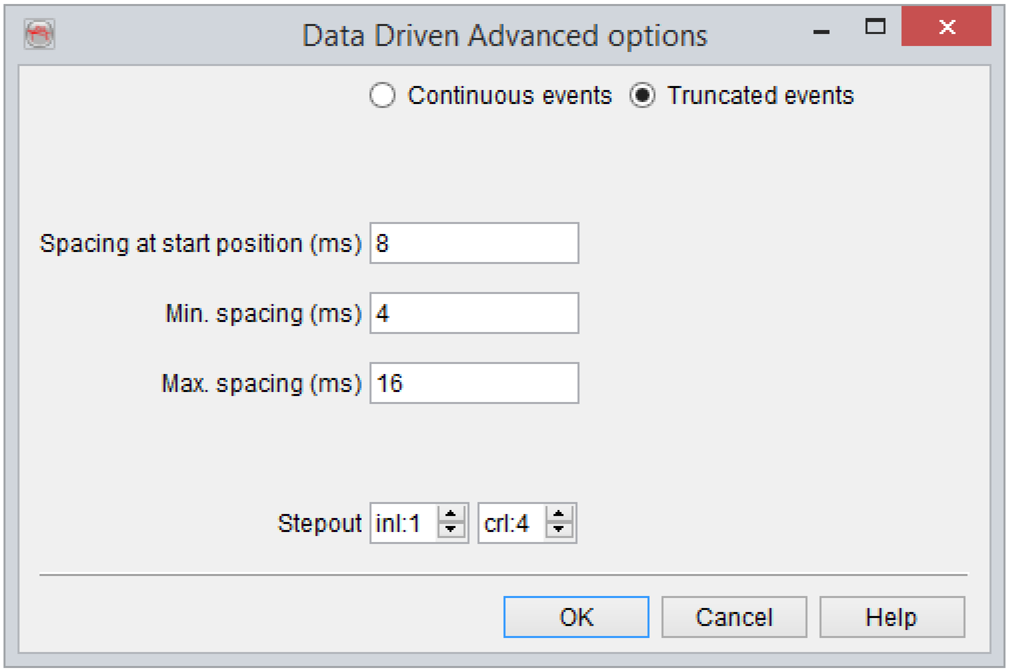
Maximum Nr of Iterations 1

Stepout inl:1 crl:4

Truncated Events

Min or Max. spacing (ms / m / ft): This parameter is defined for a HorizonCube with truncated events. Converging events cause one of the two events to be stopped if the vertical distance becomes smaller than the minimum thickness. Diverging events cause one additional event to be added between the two diverging events when their vertical distance becomes larger than the maximum thickness.




Data Driven Advanced options

○ Continuous events ● Truncated events

Spacing at start position (ms) 8

Min. spacing (ms) 4

Max. spacing (ms) 16

Stepout inl:1 crl:4

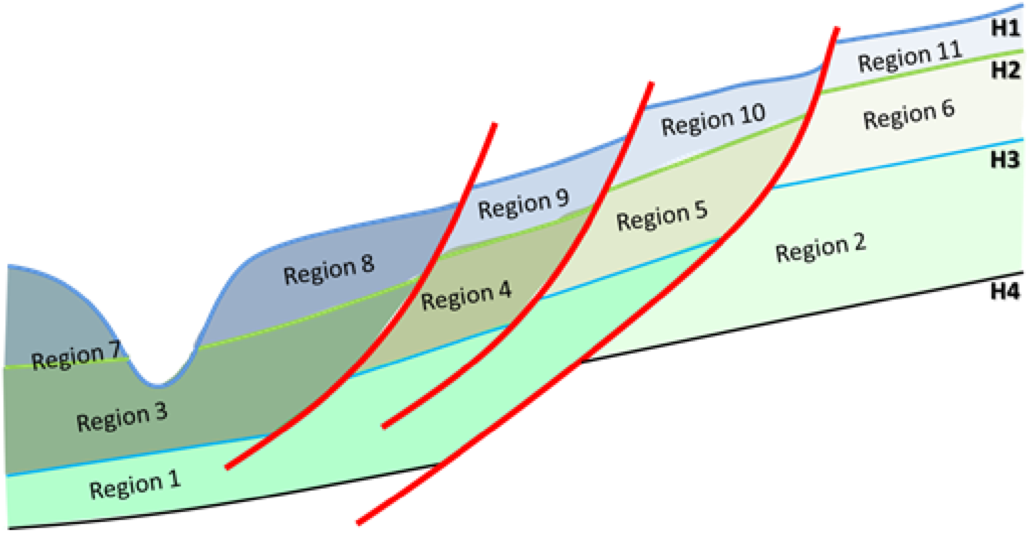
2D HorizonCube Processing

Introduction

OpendTect 5.0 supports a new way of processing a 2D HorizonCube. The workflow is interactive and operates in regions. A region is defined as a vertical segment bounded by horizons and faults. HorizonCube processing is performed on a line-by-line basis and per region. Next the user interactively correlates HorizonCube segments from region to region over the entire line. This results in a merged HorizonCube in which all segments are correlated and the number of horizons per segment is equal. To process grids of 2D lines the user correlates and merges separately processed HorizonCubes into a final HorizonCube for the grid.

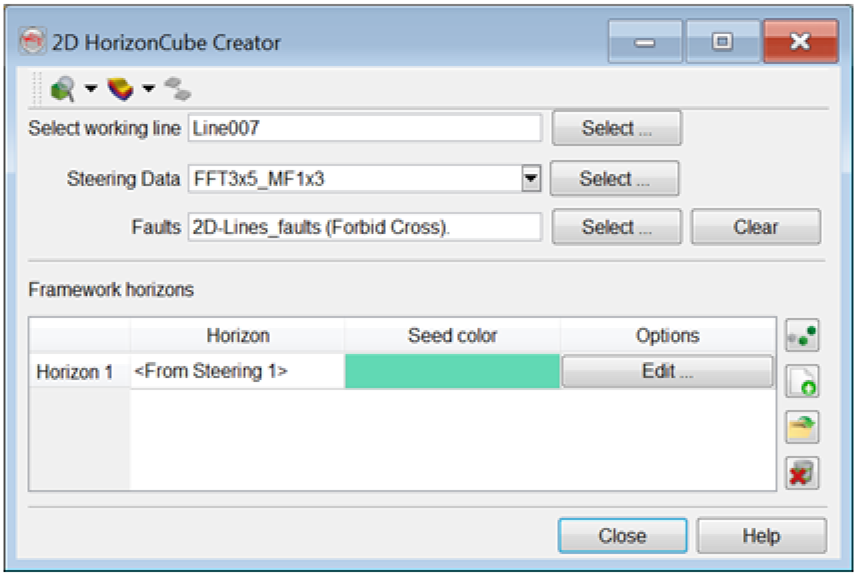
HorizonCube processing for one line is performed in three steps:

1. **Constructing the Framework**
    a. Select an existing set of horizons and faults.
    b. Optionally, add more framework horizons using the new multi-seed, dip steered tracking algorithm.
2. **Processing of HorizonCube segments per region**.
3. **Correlation of HorizonCubes between regions**.



HorizonCube regions are automatically constructed as closed polygons from the given set of horizons and faults. The illustration above shows a set of geological regions that are automatically constructed by the algorithm.

Preparing the Framework



Select working line: Line007

Steering Data: FFT3x5_MF1x3

Faults: 2D-Lines_faults (Forbid Cross).

Framework horizons

	Horizon	Seed color	Options
Horizon 1	<From Steering 1>	🟩	Edit ...

The user interface for the HorizonCube creator supports the following:

1. **Select working line**: Selection of a single 2D line on which a HorizonCube is processed.

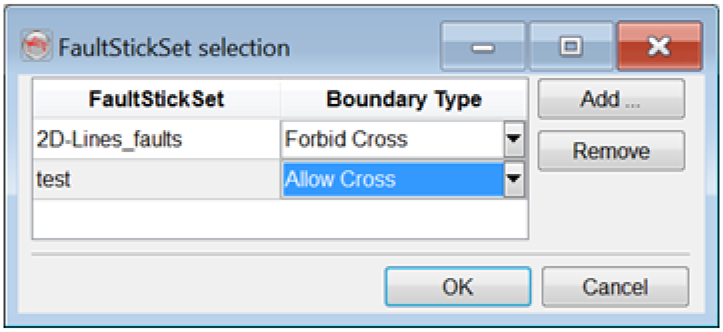
2. **Steering Data**: The steering data corresponding to selected 2D line should be provided.

3. **Faults**: This option is used to select multiple fault stick sets. The fault sticks will be automatically displayed in the scene. The tracking algorithm can treat faults in different ways:

    a. Forbid Crossing – Tracking of Framework horizons and HorizonCube events stops when a fault stick is encountered. This option generates HorizonCube segments that will require correlation to make a final merged output.

    b. Allow Crossing – Tracking of Framework horizons and HorizonCube events will continue across fault sticks. This option generates a HorizonCube that does not require merging. It approximates results produced in older versions of OpendTect.

Tip: A user may draw channels and other geological terminations as (fake) fault sticks and select these as constraints to construct regions. This combined with manual correlation can improve the results.



FaultStickSet	Boundary Type
2D-Lines_faults	Forbid Cross
test	Allow Cross

4. **Framework horizons**: Framework horizons are either imported, or they are created on-the-fly. To select existing seismic horizons press the open icon. To create a new dip steered horizon proceed as follows:

a. Press the + icon to add a new horizon.

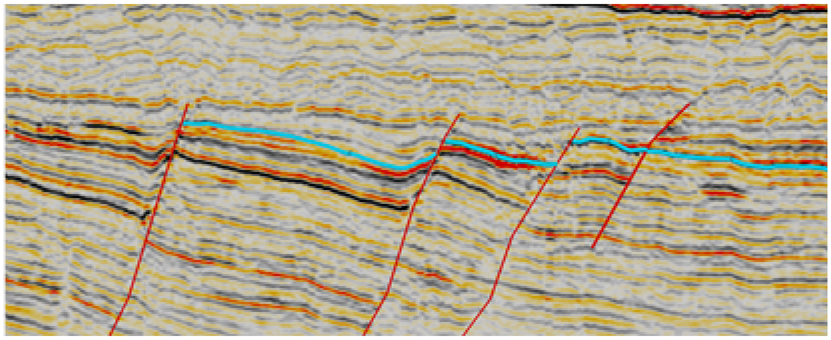
b. Press the seed icon and start picking in the scene. The auto-tracker follows the dip until it reaches the end of the line, or a fault stick it is not allowed to cross. Optionally tracking is controlled by adding more seeds. Options:

i. Insert seed: Left mouse click.
ii. Delete a seed: CTRL + Left mouse click.
iii. Rename a framework horizon by double clicking the horizon (column) cell.
iv. Edit button in the table: controls the range over which the tracking error of the tracked events is smoothed in the mid-section between two seeds.

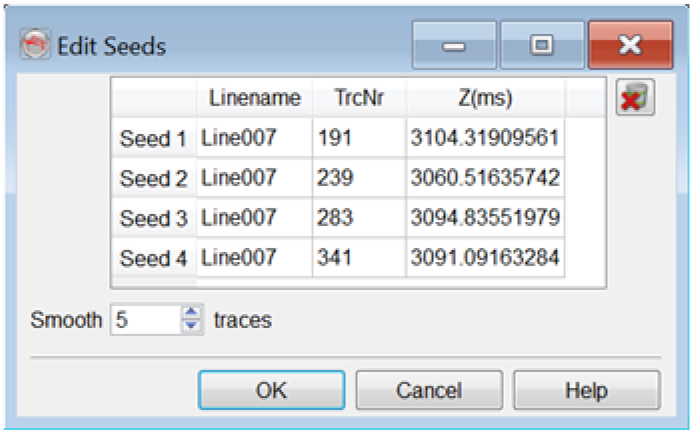
c. Framework horizons can be removed from the table using the trash button.

d. Pick mode is indicated by the mouse pointer and can be toggled ON/OFF with keyboard ESC. If pick mode is OFF the display can be changed:

i. Pan: click and drag with CTRL-Left, SHIFT-Left, or Middle mouse.
ii. Rotate: click and drag with Left mouse.
iii. Zoom: Rotate with middle mouse wheel.



Multi-seed picking of a horizon with the dip-steered auto-tracker.



	Linename	TrcNr	Z(ms)
Seed 1	Line007	191	3104.31909561
Seed 2	Line007	239	3060.51635742
Seed 3	Line007	283	3094.83551979
Seed 4	Line007	341	3091.09163284

Smooth 5 traces

A table (above) listing the picked seeds for a dip steered seismic horizon. It also allows changing the seed position (trace and z-value) and smoothing the tracking error between seeds. You can select multiple rows and delete all seeds (CTRL+A plus trash button, or select multiple entries while left mouse button is pressed followed by clicking the trash button).

Preview Regions:

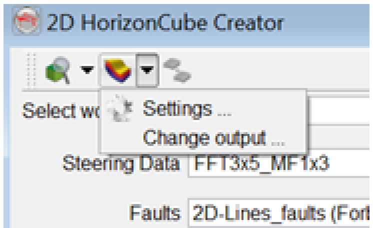
This feature is allows HorizonCube regions to be QC-ed. All regions are displayed as polygons overlying the mapped horizons and fault sticks. To update a region, either manually adjust fault sticks (using OpendTect's fault picking tools), or adjust framework horizons using the dip-steered auto-tracker in the 2D HorizonCube Creator window.

You can customize the display settings for this region previewer through the settings sub-menu or icon.

Please note that regions can also be previewed from the data-driven settings dialog, see **next Section**.

Processing 2D HorizonCube

Settings:



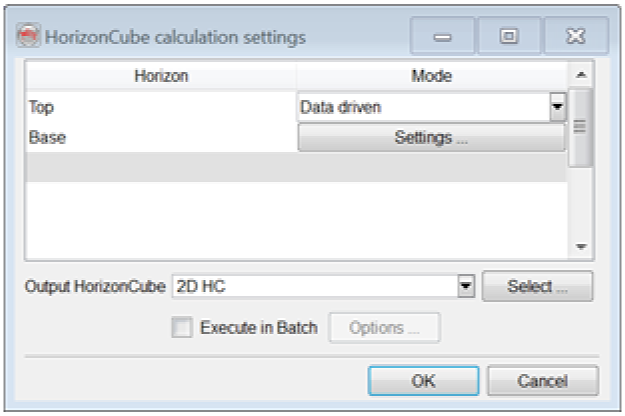
Select wo

Steering Data FFT3x5_MF1x3

Faults 2D-Lines_faults (Fort

To process a 2D HorizonCube, press the HorizonCube icon, or select *Settings* from the drop-down menu. The settings dialog window is launched.

Save As saves the current HorizonCube under a new name.



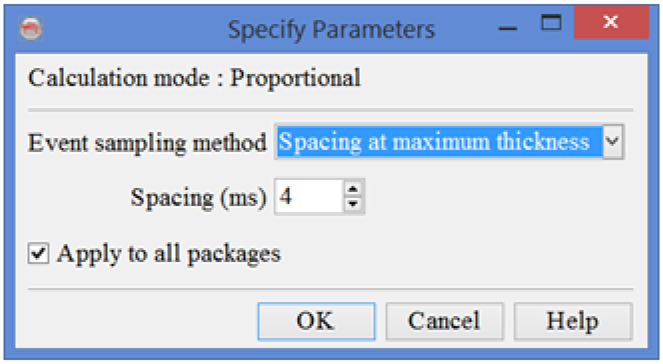
Horizon	Mode
Top	Data driven
Base	Settings ...

Output HorizonCube 2D HC

☐ Execute in Batch

Model-driven Modes: Two *Event sampling methods* are supported: *Spacing at maximum thickness* and the *number of events* for the package. The three model-driven modes are:

- Parallel to Upper
- Parallel to Lower
- Proportional



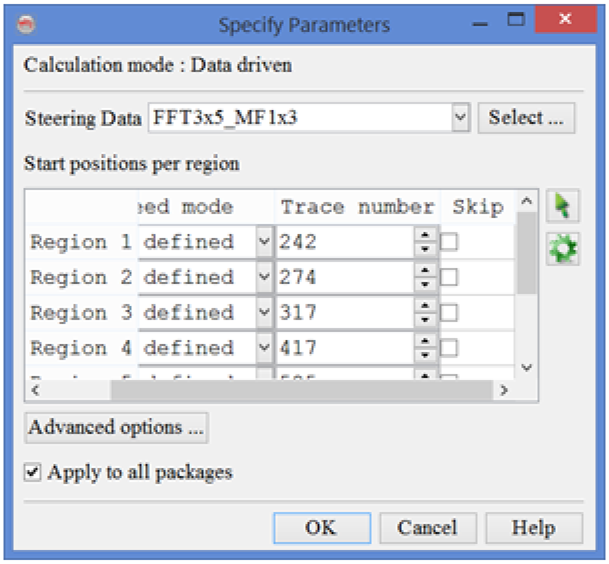
Calculation mode : Proportional

Event sampling method Spacing at maximum thickness

Spacing (ms) 4

☑ Apply to all packages

Data-driven Mode: In this mode dip steered horizons are auto-tracked to produce data-driven HorizonCube segments per region. The settings for this mode are defined per package. Pressing the *Settings* … button launches the settings dialog window and updates the 3D scene by showing regions and the proposed starting positions for the tracker. Hovering over the scene highlights the active region.



Calculation mode : Data driven

Steering Data FFT3x5_MF1x3

Start positions per region

	›ed mode	Trace number	Skip
Region 1	defined	242	☐
Region 2	defined	274	☐
Region 3	defined	317	☐
Region 4	defined	417	☐

☑ Apply to all packages

Options:

Steering – Select the steering input per package.

Start position – This is a crucial input parameter. The dip-steered auto-tracker simultaneously tracks all horizons from this position at the sampling interval specified in Advanced Settings. The starting position is displayed as a vertical line. This can be changed in the table but the recommended way to change the position is to interactively change this in the 3D scene. To do this press the green arrow icon. Now hover over the regions and QC and adjust the starting positions. Starting position should be selected in the thicker part of a region in good quality data areas and preferably not too close to a fault. The green wheel icon is used to change display settings of regions and start positions.

Skip a region – Regions can be skipped, e.g. in bad data areas. No HorizonCube is processed in skipped areas.

Advanced Options – See the section on **Advanced options**.

Processing (on-the-fly/batch mode):

HorizonCube processing can be done:

On-the-fly:

Recommended for smaller 2D lines (less than 20Km) with few regions only.

A Wheeler Scene is launched automatically once HorizonCube processing is finished.

Batch-mode:

Recommended for long 2D lines (greater than 20Km), or lines with many regions.

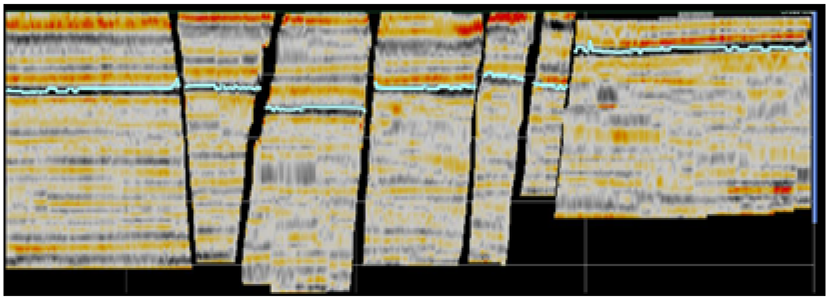
To inspect the data in the Wheeler domain a Wheeler scene must be added and filled manually.

Updating the Results after Processing:

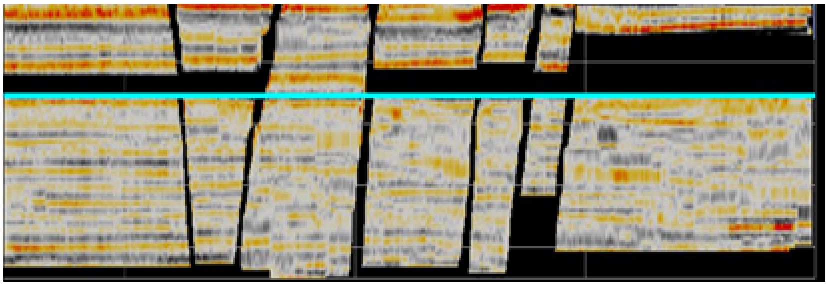
The Wheeler scene is an excellent tool to QC the HorizonCube processing results. Remember that segments are processed independently so that it cannot be expect that events are correlated over segments. Neither will the number of events be identical per segment. Correlation and merging will be done in a later step (see Correlating 2D HorizonCube Regions).

In this phase the Wheeler scene is used to study the flatness of seismic events. Dipping events in the Wheeler scene imply that the auto-tracker deviated from the actual seismic reflection patterns. HorizonCube processing can in that case be improved by adjusting the framework and rerunning the processing.

Framework horizons can be picked and update in the normal scene as well as in the Wheeler scene.



Picking a (dip-steered) seismic horizon in the Wheeler domain.



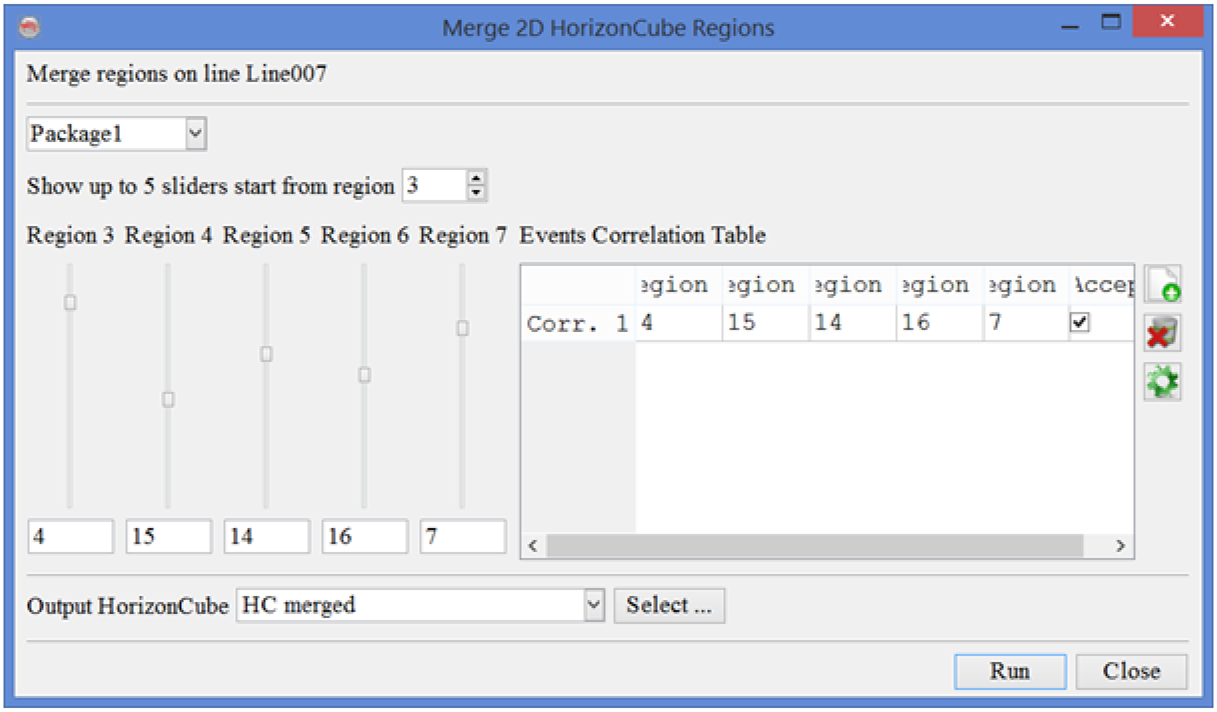
The updated Wheeler scene after reprocessing the HorizonCube.

Delete this text and replace it with your own content.

Correlate 2D HorizonCube Regions

HorizonCube segments must be correlated laterally. This interactive process is performed using a set of sliders that control HorizonCube displays per region. The user correlates one or more events over all regions after which a new, merged HorizonCube is computed. The merged HorizonCube has an equal number of horizons per package.

The correlation window is launched from the correlation window icon .



Merge regions on line Line007

Package1

Show up to 5 sliders start from region 3

Region 3 Region 4 Region 5 Region 6 Region 7 Events Correlation Table

	egion	egion	egion	egion	egion	Accep
Corr. 1	4	15	14	16	7	☑

Region sliders: 4, 15, 14, 16, 7

Output HorizonCube HC merged

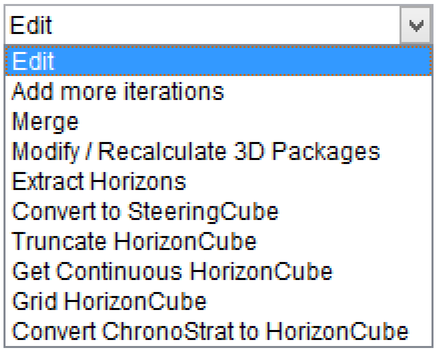
The sliders control the number of events per region to display. The sliders operate simultaneously in the normal scene and the Wheeler scene. Depending on the performance of the graphics card and the size of the

2D line to be manipulated simultaneously updating both scenes might be too slow for comfort. If so, it is possible to work in one scene only and to iconize, or close the other one.

Correlation is performed per *Package* on a Region by Region basis. Move the first slider and position it on a prominent seismic reflector. Now move the second slider and position it on the same reflector in the adjacent region. Repeat this for the first five regions. Next reposition the sliders laterally by adjusting the *start from region* position in the spinbox. Continue the process until the event is correlated across all regions. When satisfied check the *Accept* toggle such that the correlation cannot be changed accidentally.

To add further correlations press the *plus* icon and repeat the process. To change an already accepted correlation uncheck the *Accept* toggle and adjust the sliders. To remove a correlation uncheck the *Accept* toggle and press the *trash* icon.

Tools



HorizonCube Editor

HorizonCube editor is a tool to edit a 2D/3D **HorizonCube**. It is recommended to edit a continuous HorizonCube. The tool supports manual editing of one or more events in a loose grid (inlines/crosslines at a fixed interval e.g. 10 by 10). Once the events are edited, the user may proceed to correct the HorizonCube based on corrected events. There are two methods of editing supported: Error-based and Linear.

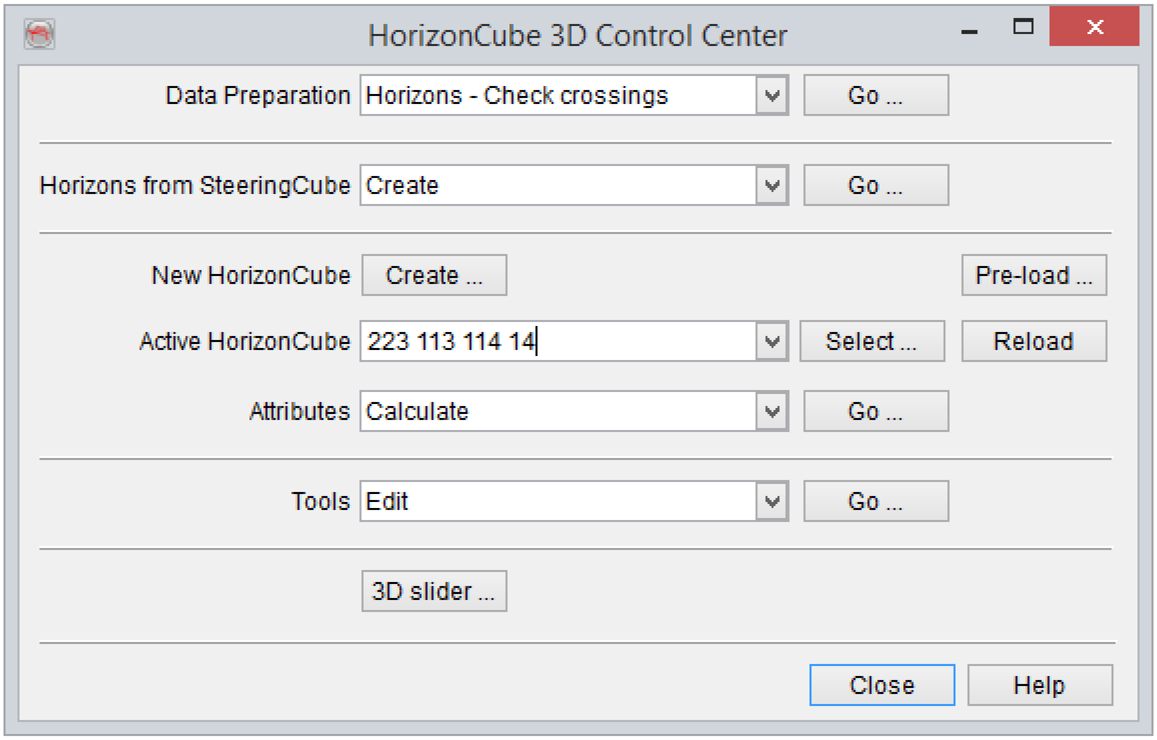
In the *error-based* interpolation method, the error between an edited and its non-edited event(s) is computed as delta-Z. This delta-Z is then linearly interpolated/extrapolated vertically as well as laterally. This method is generally not preferable as one sees the results as residuals, which may appear as wavy events.

In the *linear-based* interpolation method, the events are proportionally adjusted relative to the edited events. This is the default option and the preferable method for editing a HorizonCube if the Z-linear based interpolation type does not reveal satisfactory results.

In the *Z-based Linear* interpolation method, the events are adjusted by using a weighting factor on number of events prior to editing. It reduces the weight on spacing from the border to the center defined by the outermost seeds. This helps in smoothing the edges with less abrupt jumps and is, therefore, the preferred method.

The editor is launched from the HorizonCube control center (Tools--> Edit.. and Press *Go* button). When the editor is launched, a suffix '[Edit]' is added for an active HorizonCube that is present in the tree. This is done to avoid accidental editing of the parent HorizonCube.

The editor layout is shown below:



HorizonCube 3D Control Center

Data Preparation: Horizons - Check crossings

Horizons from SteeringCube: Create

Active HorizonCube: 223 113 114 14

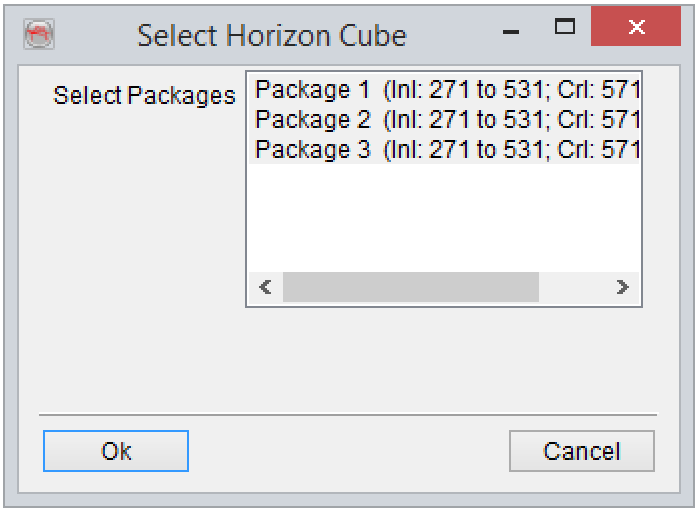
Attributes: Calculate

Tools: Edit

HorizonCube Editor Dialog

Package(s) Selection:

For HorizonCubes with more than one package, the user may utilize the package selection feature, selecting one package at a time for editing. This is particularly useful if the HorizonCube is very large. Note that the corrected HorizonCube will be a merged version (i.e. edited package and unedited packages).



Package 1 (Inl: 271 to 531; Crl: 571
Package 2 (Inl: 271 to 531; Crl: 571
Package 3 (Inl: 271 to 531; Crl: 571

Event Index:
This is a slider to *Add* events in the list for editing. Initially, the slider is positioned at 0-event level and all events of a selected HorizonCube are displayed in the scene on inlines/crosslines. When the slider is moved down/up the corresponding event is displayed in the scene. The event ID is always displayed in the text box below the the slider of the editor.

Add:
This button is used to add an event in the selected events list. It becomes active as soon as one moves the slider downward from the default position.

Selected Events:
It is a list of selected events as well as the framework events. The framework events are the events that are the original horizons used to create a HorizonCube. The framework events serve as a zero error boundaries per package i.e. neither the selected events nor the corrected events can cross the framework events. On the other hand, if one attempts to edit an event such that it crosses the framework, the edited event will merge together with the framework event (e.g. a pinch-out type event).

Please note that the color of an event is set from the colortable of the displayed HorizonCube.

For display purposes, you may right click in the selected events panel to uncheck/check all events. The All option on the top of panel does the same thing. If an event is unchecked, it will not be displayed in the scene. To avoid displaying too many events in the scene, you may display only selected events by unchecking the unwanted events in the display.

Toolbar

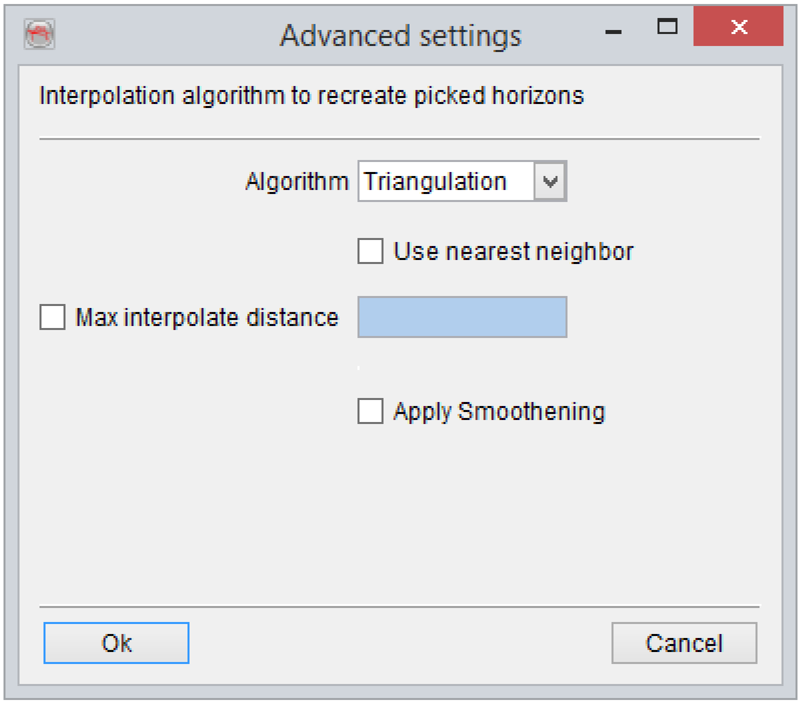
- **QC** *QC* the output of a corrected HorizonCube on an inline / crossline
- 🔓 *Lock* an event to make it a framework event. Another edited event cannot cross the locked event but it will merge with the locked one.
- ✖ *Remove* an event(s) from the selected list.
- 💾 *Save* seeds and the edited events for later use in editing.
- 📂 *Open* the saved events.

Interpolation Type:

User has to decide which type of interpolation s/he has to select. These types are explained at the beginning of this section. The linear based interpolation also supports smoothing of an event close to the edges. The edges are formed by using the seeds that defines a rectangular edited area. It is recommended to use a mild (5 by 5) smoothing filter.

Advanced Settings:

⚙ As the one edits the events on a loose grid, therefore, the events should be gridded based on a gridding alogrithm. The advanced settings are used to change the default gridding settings. At present, triangulation and inverse distance gridding algorithms are supported.



Algorithm Triangulation

☐ Use nearest neighbor

☐ Max interpolate distance

☐ Apply Smoothening

Apply:

It applies the interpolation between the edited events on the input HorizonCube and activates the other buttons.

Revert:

It reverts (UNDO) the changes made by the apply action. The scene goes back to actual editing of the HorizonCube events with seeds.

Save:

Save the corrected HorizonCube. For first correction, it will suggest you to provide a new name. For subsequent editing, it will overwrite the active HorizonCube.

Save As:

Save the corrected HorizonCube with a new name (recommended).

Saving a session:

Sessions save/retrieve are supported for the HorizonCube editor. It saves the pre-loaded HorizonCube, the editor layout, edited events with seeds, and the scene layout.

Tips

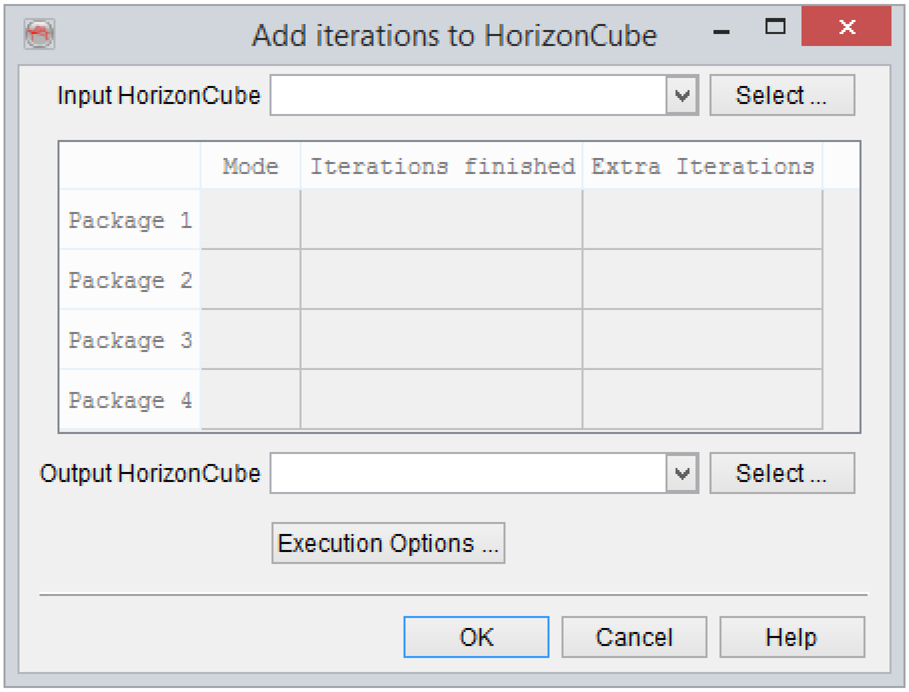
- Please avoid editing multiple areas. Best practice is to correct one area, and then continue editing the corrected HorizonCube in another area (if necessary).
- Save the edited HorizonCube with new name.
- While editing on sections, you may QC the output.
- If you edit an edited HorizonCube, you may over-write the same. This avoids too many copies of the same HorizonCube.
- Always use Revert option if you are not satisfied with the corrections.
- Save a session (Survey > Session > Save) with HorizonCube Editor Active.

Add More Iterations

If a 2D/3D HorizonCube is created and there are still large areas with unwanted holes/gaps, more iterations can be inserted in a package to fill these gaps. There are four modes supported to fill the gaps:

1. **Data Driven:** This option fills the gaps using the SteeringCube i.e. a data driven HorizonCube with the same parameters as that were defined in the previous iteration.
2. **Proportional:** A model driven option to fill the holes proportional to upper and lower events of existing HorizonCube.
3. **Parallel to upper:** A model driven option to fill the holes using the geometry of the upper event of a HorizonCube..
4. **Parallel to lower:** Another model driven approach to fill the holes parallel to the geometry of the lower event of a HorizonCube.

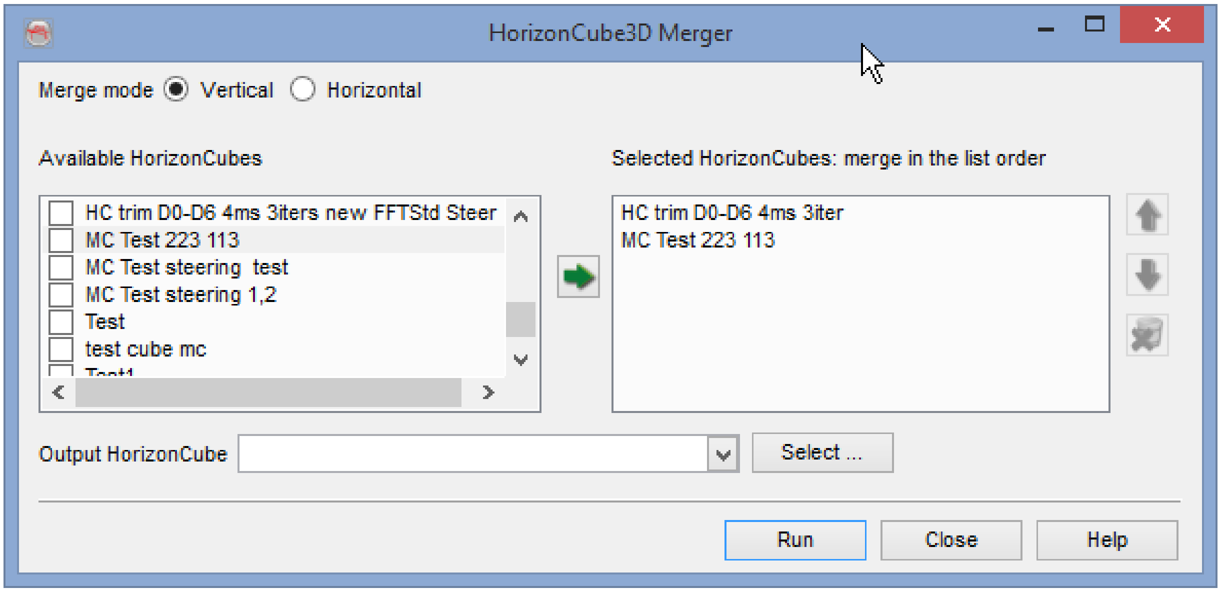
In the 'Input HorizonCube' field, the stored HorizonCube is selected whereafter the corresponding previously finished iterations are displayed in the table. The extra iterations in the next column are set to fill the gaps in a HorizonCube. The results of this processing can be stored into a new HorizonCube.



User interface of 3D HorizonCube to add more iterations.

Merge HorizonCubes

The merge tool allows for selection of multiple horizoncubes to be sorted into a list for merging:



Merge mode ● Vertical ○ Horizontal

Available HorizonCubes:

☐ HC trim D0-D6 4ms 3iters new FFTStd Steer
☐ MC Test 223 113
☐ MC Test steering test
☐ MC Test steering 1,2
☐ Test
☐ test cube mc

Selected HorizonCubes: merge in the list order

HC trim D0-D6 4ms 3iter
MC Test 223 113

The process may be performed vertically or horizontally. The named output horizoncube will be processed and be available for all future processing.

Add Packages, Recalculate 3D Sequences

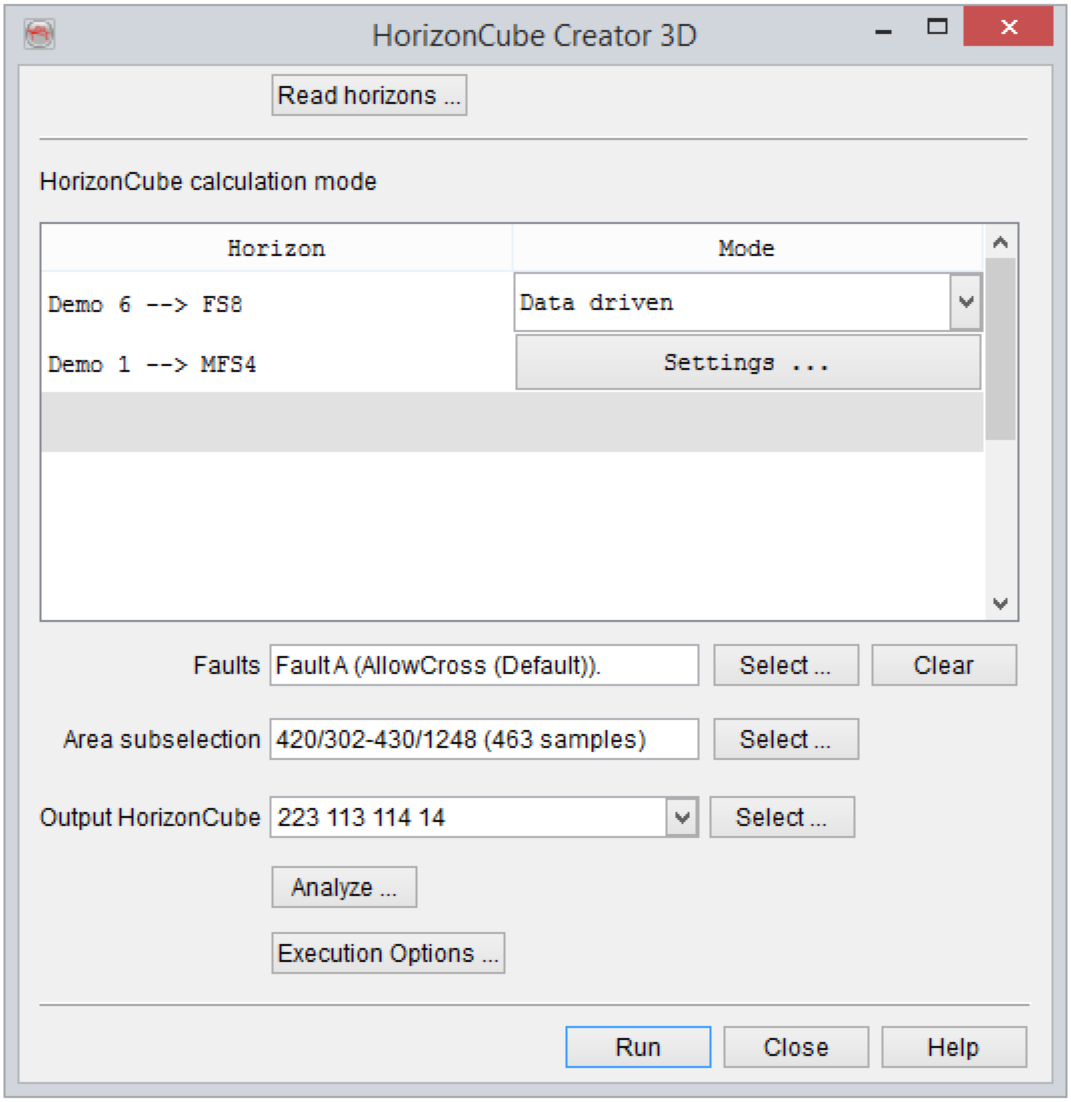
This tool is used to modify the HorizonCube (either 2D or 3D) with a new 'mode' or alternatively by using another SteeringCube. This tool is valuable in complex geologic intervals or when dealing with very noisy seismic data. In such cases changing parameters/settings for a package might be beneficial. Specifically, the calculation mode might be changed from 'data driven' to 'model driven' or vice versa. SteeringCubes can also be set individually for each package. Thus, this tool includes two options for any package of a HorizonCube: *Keep* or *Re-calculate*.

In the following window, the input 3D HorizonCube is provided. The table for HorizonCube calculation mode is populated according to the previous settings for the selected HorizonCube. The *Read Horizons* button can also be used to add more horizons (packages) to the selected HorizonCube. Note: If *Read Horizons* is clicked, even wanted existing horizons in the HorizonCube have to be selected.

In the table, set calculation *Mode* and set the *Action* to re-compute. The corresponding settings can also be changed. Additionally, more interpreted faults can also be added to the existing HorizonCube by click on the *Select* button for the *Faults* field. The *Analyze* button is used to check the inconsistencies and problems that might be encountered during the calculation of the HorizonCube.

Clicking the *Go* button starts the HorizonCube modify/re-calculate batch program for the selected HorizonCube with the new settings.

Note: The *Show Option* check-box is an optional tool for the batch program configuration. By default (unchecked) the batch program runs in a pop-up window. Further, a batch program can be run in a log file (without showing the pop-up window). Remote connections to other computers for multi-machine processing is also accessed here.



HorizonCube calculation mode

Horizon	Mode
Demo 6 --> FS8	Data driven
Demo 1 --> MFS4	Settings ...

Faults: Fault A (AllowCross (Default)).

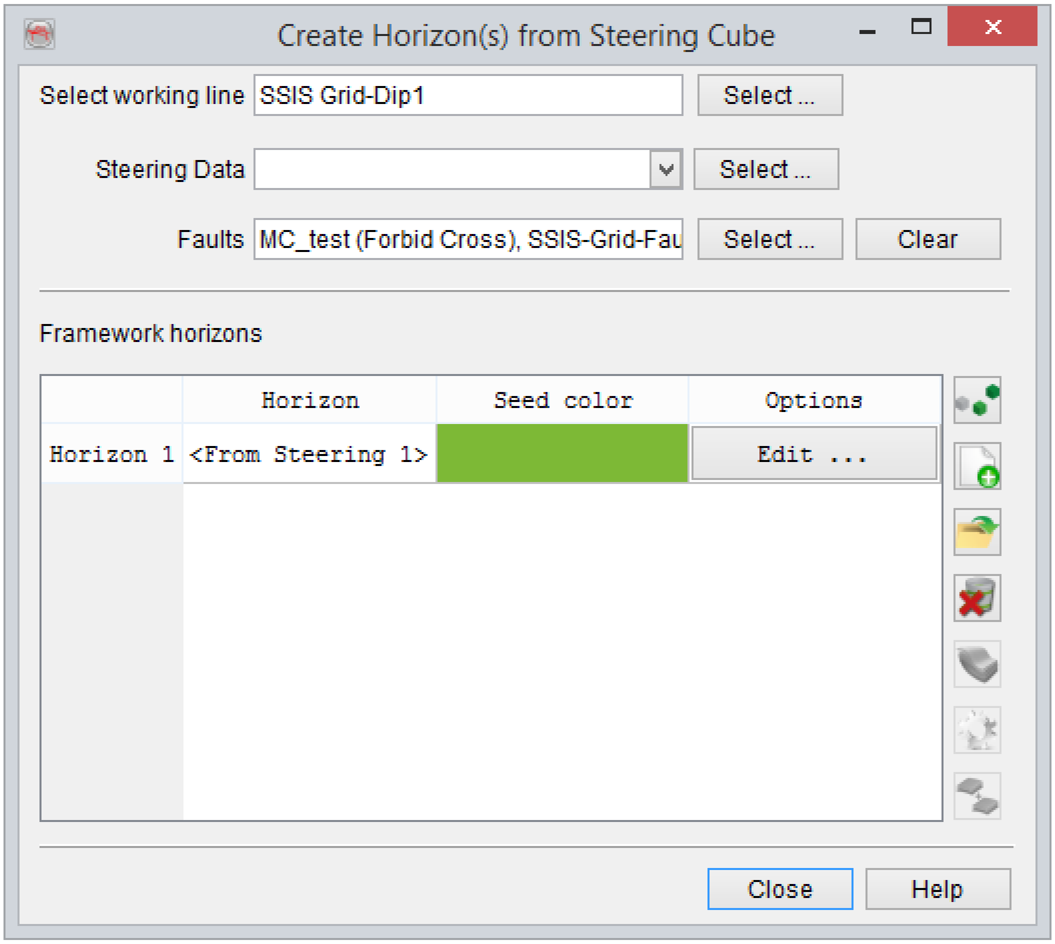
Area subselection: 420/302-430/1248 (463 samples)

Output HorizonCube: 223 113 114 14

Modify a previously calculated HorizonCube with another Mode/SteeringCube with different settings.

Add / Recalculate line

The workflow for modifying/re-calculating 2D HorizonCubes is in many ways similar to the 3D workflow. Set 'Input HorizonCube', choose which lines to add/modify ('Select lines to add/modify'). Packages and faults can then be added, modified or kept as-is.



Select working line SSIS Grid-Dip1

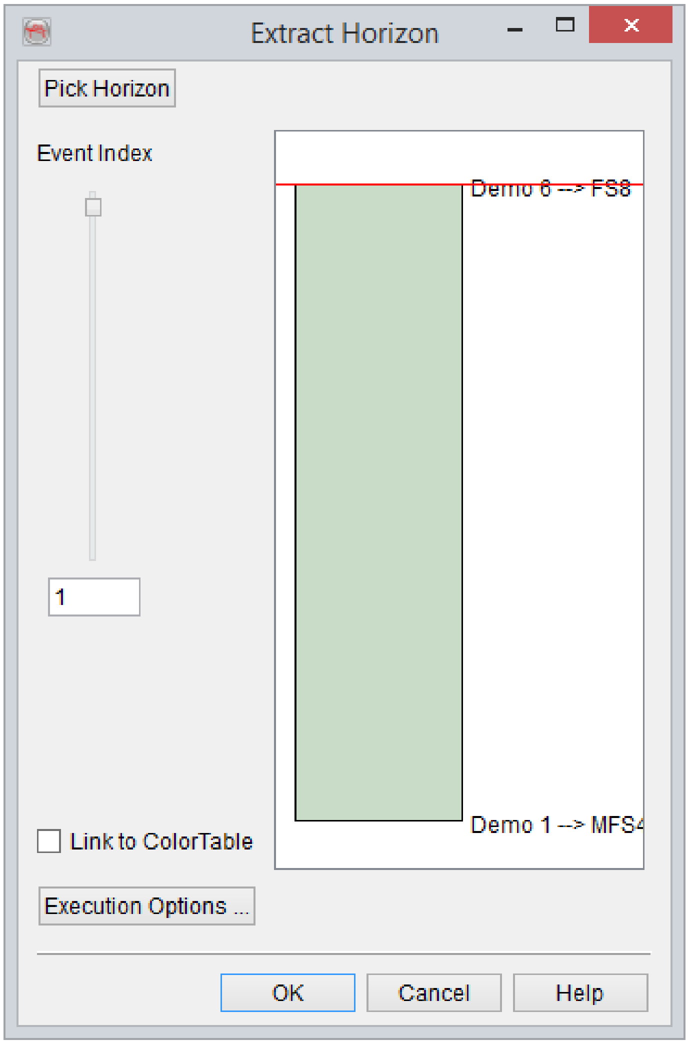
Steering Data

Faults MC_test (Forbid Cross), SSIS-Grid-Fau

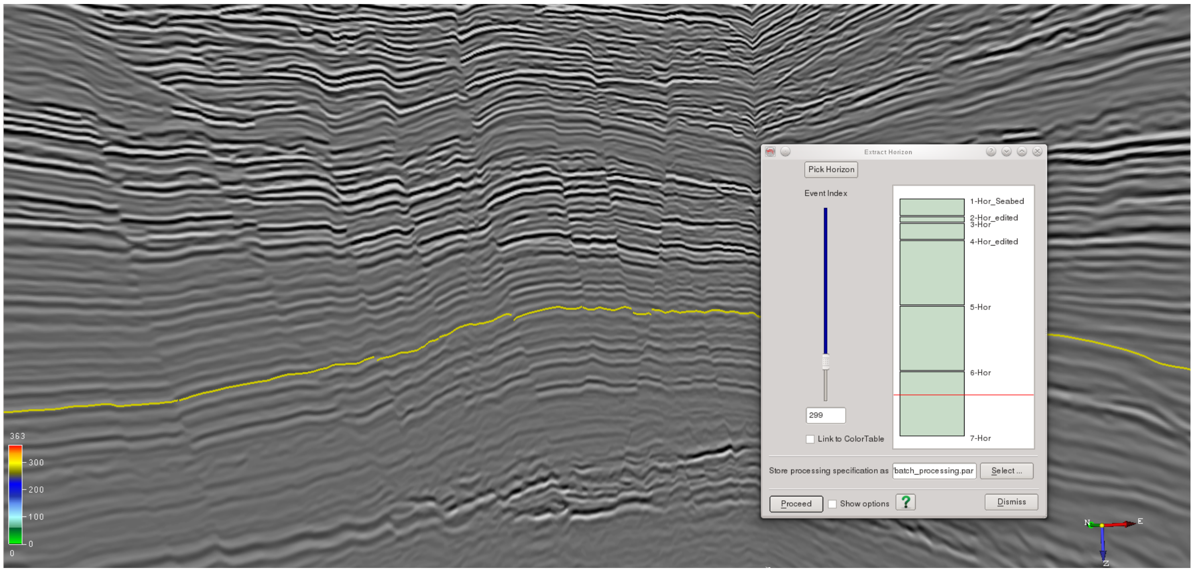
Framework horizons

	Horizon	Seed color	Options
Horizon 1	<From Steering 1>	🟩	Edit ...

Extract Horizons



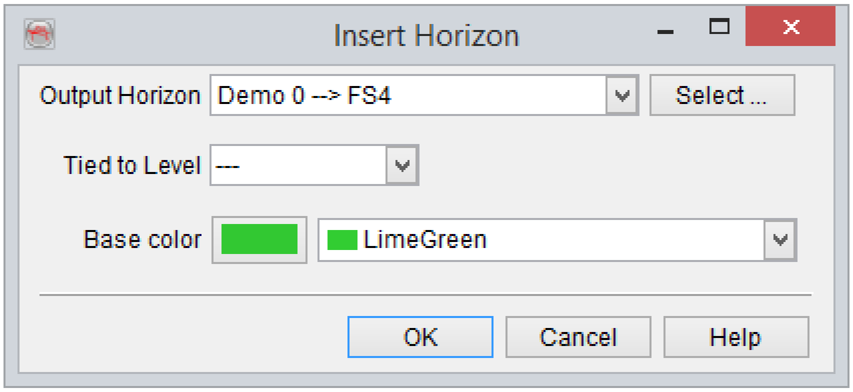
Event Index: 1



Event Index: 299

Store processing specification as 'batch_processing.par

This tool is used to extract seismic horizons from the **selected HorizonCube**. The slider is first positioned to a point where the user is interested in extracting a full 3D horizon. The relative position of the slider is always presented as a red-line in the graphical display of the packages in a HorizonCube.After setting the slider position, the Pick Horizon button is pressed to extract a horizon at the selected position. The output name, color (and optional stratigraphic level) for the new horizon is specified in the pop-up window. This process can be repeated for any number of horizons. Clicking on 'Proceed' in the main window extracts and saves all picked horizons in a batch process..



Output Horizon: Demo 0 --> FS4

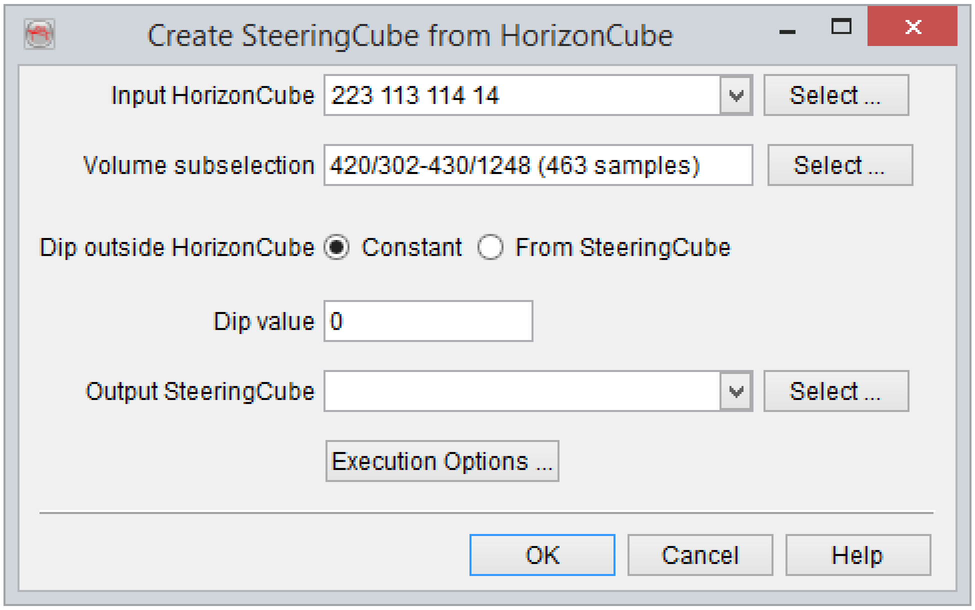
Tied to Level: ---

Base color: 🟩 LimeGreen

Pick Horizon: Creates an output horizon at a defined slider position.

Convert HorizonCube to SteeringCube

A HorizonCube can be transformed into a SteeringCube using this tool. The input HorizonCube is selected first. The Volume sub-selection is optional. The dip values outside the area of the selected HorizonCube can either be filled with a constant dip value or read from another SteeringCube. The output name for the SteeringCube is provided in the output field.



Input HorizonCube: 223 113 114 14

Volume subselection: 420/302-430/1248 (463 samples)

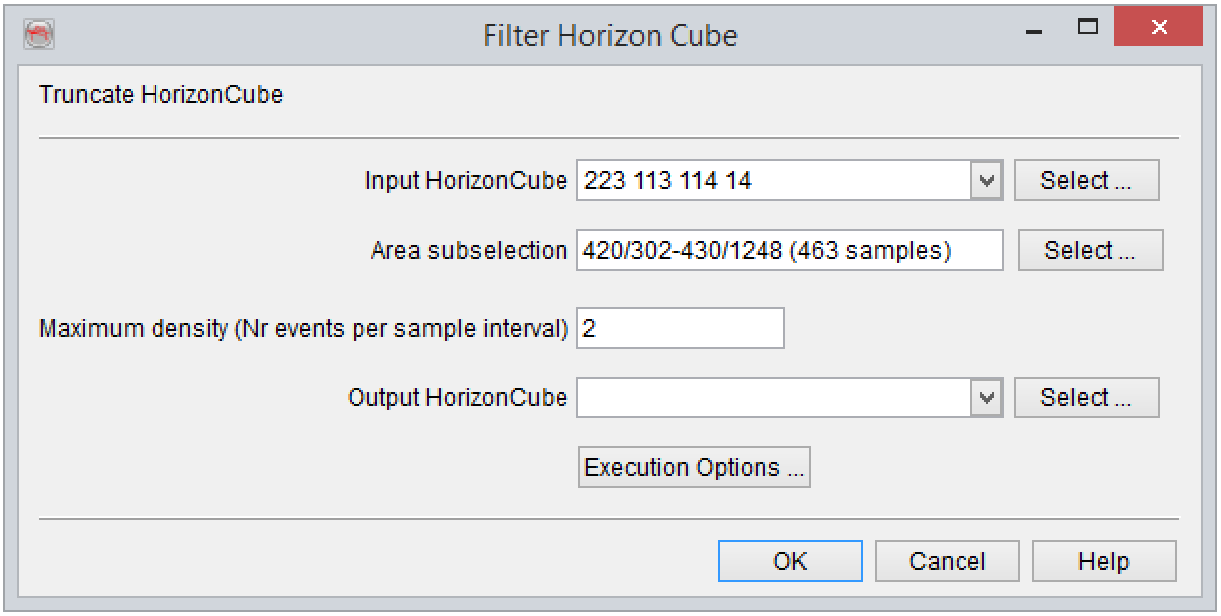
Dip outside HorizonCube: ● Constant ○ From SteeringCube

Dip value: 0

Output SteeringCube:

Truncate HorizonCube

An existing **continous HorizonCube** often contains very closely spaced events. The HorizonCube can later be filtered by defining a maximum density (threshold) as a number of closely spaced events. In the following window the data selection is very simple. Select the input HorizonCube (continuous) in the input field and type the output HorizonCube name in the output field. The *Maximum density* defines a threshold for two horizons in the HorizonCube. The value '1' defines a threshold of one event per sample of a seismic data. Optionally, the area sub-selection can also be made to do the conversion within a sub-volume only.



Input HorizonCube: 223 113 114 14

Area subselection: 420/302-430/1248 (463 samples)

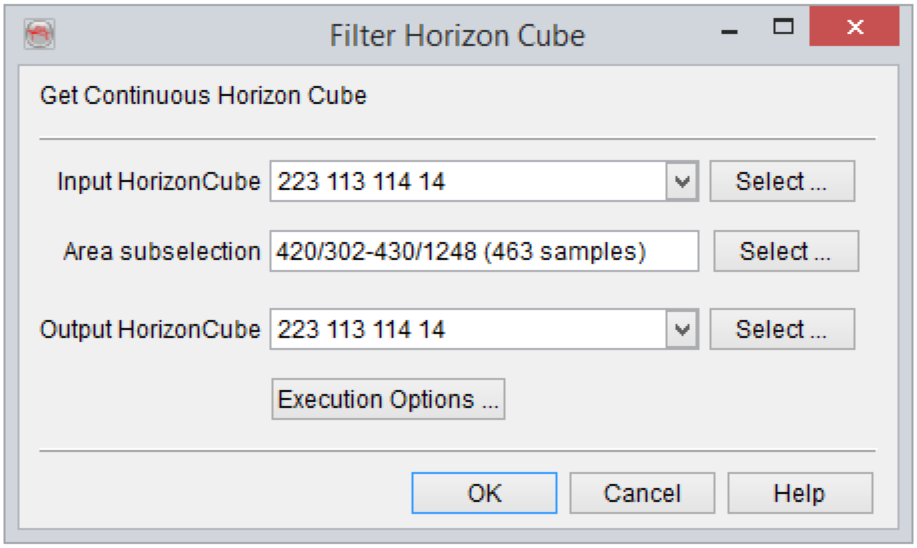
Maximum density (Nr events per sample interval): 2

Output HorizonCube:

To learn more about the benefit of applying a density filter to the continous HorizonCube, please read the **Wheeler Transformation**.

Get Continuous HorizonCube

An existing **truncated HorizonCube** can be converted to a HorizonCube with **Continuous Events** using this tool. In the following window, the data selection is very simple. Select the input HorizonCube (truncated) in the input field and type the output HorizonCube name in the output field. Optionally, the area subselection can also be made to do the conversion within a sub-volume only.



Get Continuous Horizon Cube

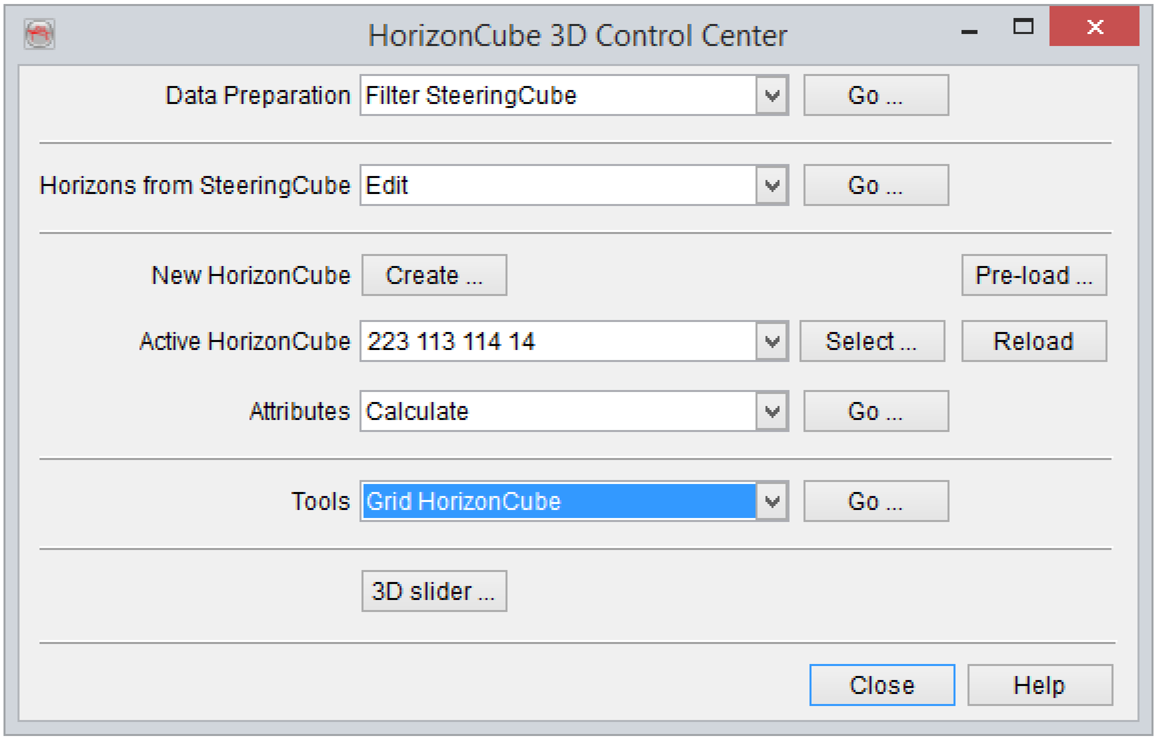
Input HorizonCube: 223 113 114 14

Area subselection: 420/302-430/1248 (463 samples)

Output HorizonCube: 223 113 114 14

Grid HorizonCube

After processing the HorizonCube, if the events have many holes, one can grid the output result. This is done through the HorizonCube control center.



Data Preparation: Filter SteeringCube

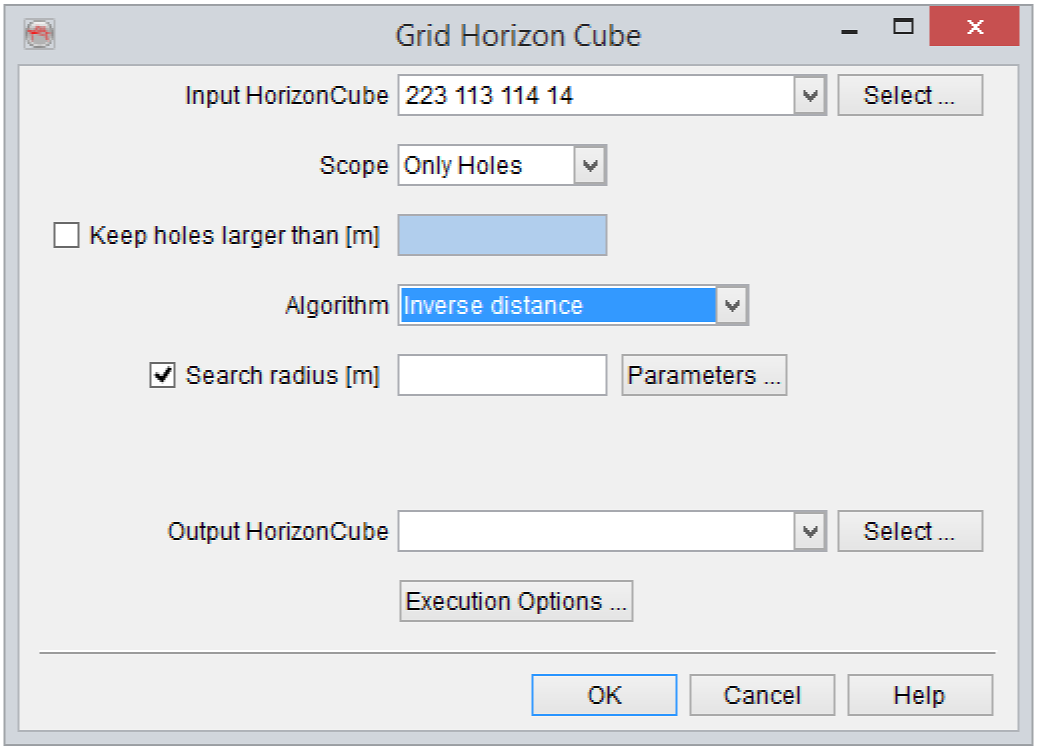
Horizons from SteeringCube: Edit

Active HorizonCube: 223 113 114 14

Attributes: Calculate

Tools: Grid HorizonCube

Different gridding algorithms can be used: Inverse distance, Triangulation, Extension, GMT, and the Dip-Steered gridding as shown below.



Input HorizonCube 223 113 114 14

Scope Only Holes

☐ Keep holes larger than [m]

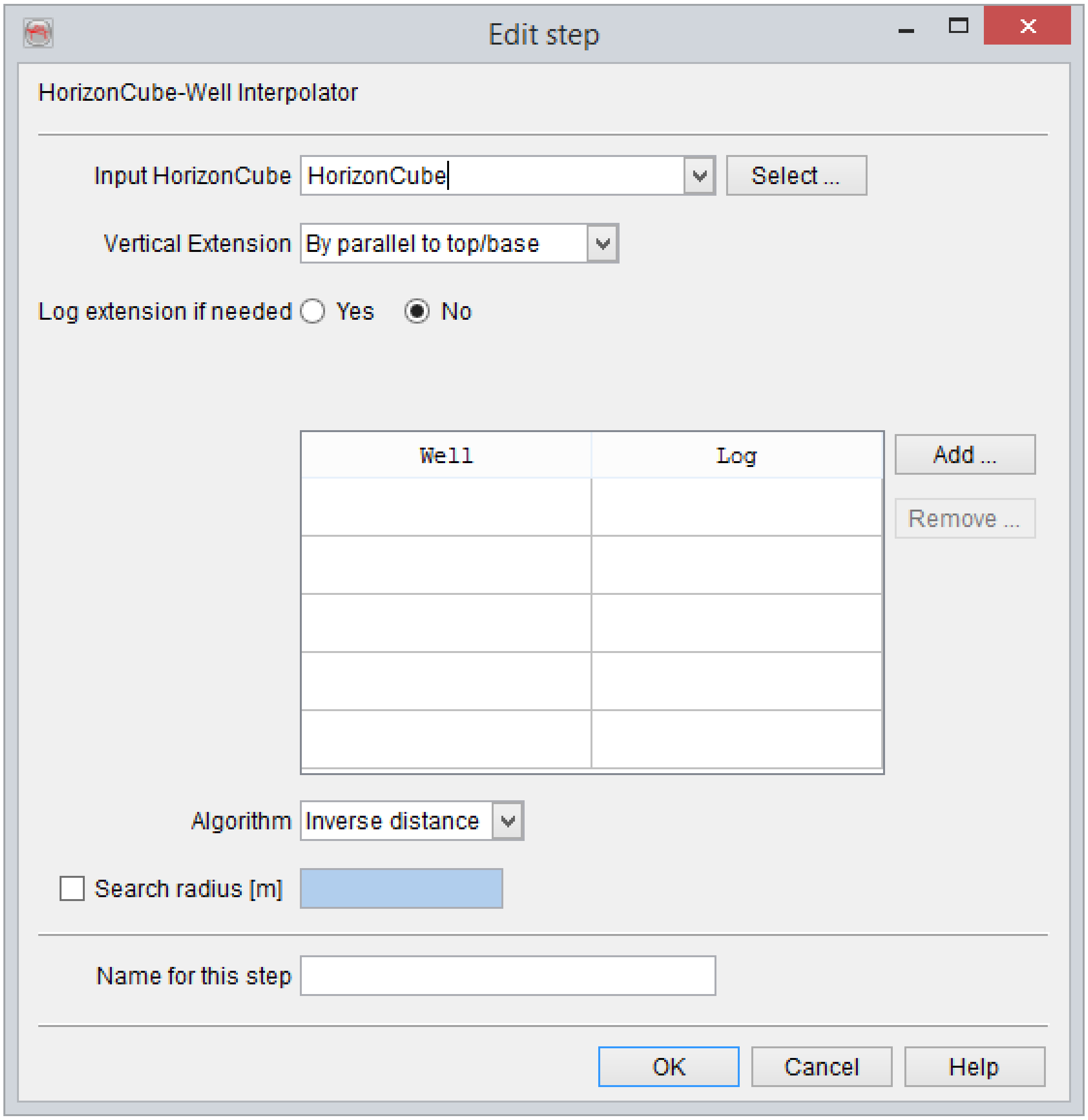
Algorithm Inverse distance

☑ Search radius [m]

Output HorizonCube

HorizonCube Well Log Interpolator

This gridding step is used to populate a 3D volume using well log(s). The HorizonCube provides the necessary steering to guide the interpolation of the well logs. This is equivalent to conventional gridding, but that would take place in the **Wheeler domain** where all seismics events are flat.



Edit step

HorizonCube-Well Interpolator

Input HorizonCube: HorizonCube

Vertical Extension: By parallel to top/base

Log extension if needed ○ Yes ● No

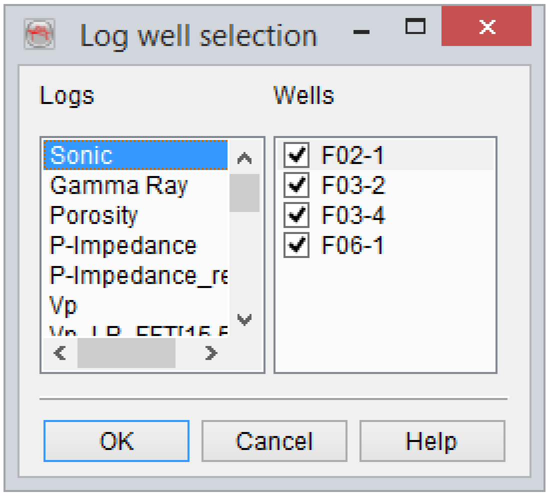
Algorithm: Inverse distance

☐ Search radius [m]

Name for this step

The HorizonCube well log interpolator includes two fields to be provided by the user: a HorizonCube and a table of well logs. Select a 3D HorizonCube by pressing the 'Select' button.

To select wells and corresponding logs, press 'Add' button. To remove a selected well/log, select it and press the 'Remove' button. To change the log, select and press the 'Change log' button. It is easier to first select the wells; The list of logs will then get updated and present only the logs that are common to all selected wells.



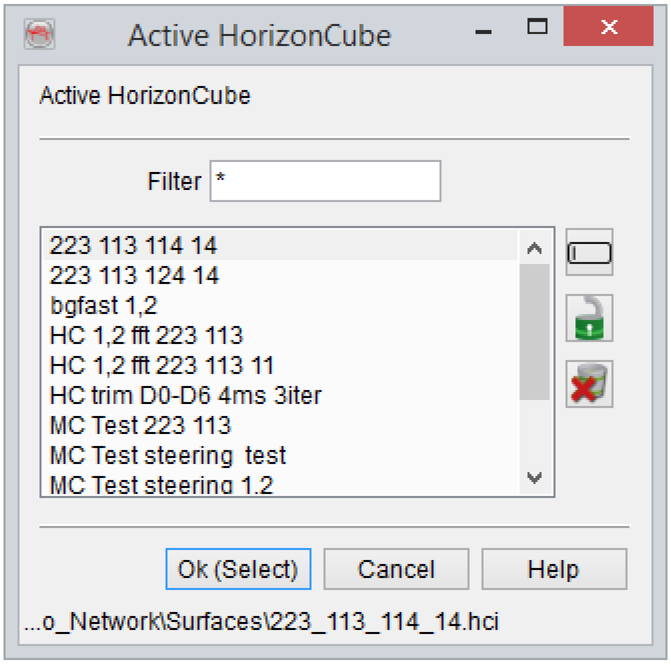
After the selection of both wells and the HorizonCube, provide a name for this step at the bottom and proceed to the Volume Builder by pressing OK.

Analysis

See the following chapters below.

Display Properties for HorizonCube 2D, 3D

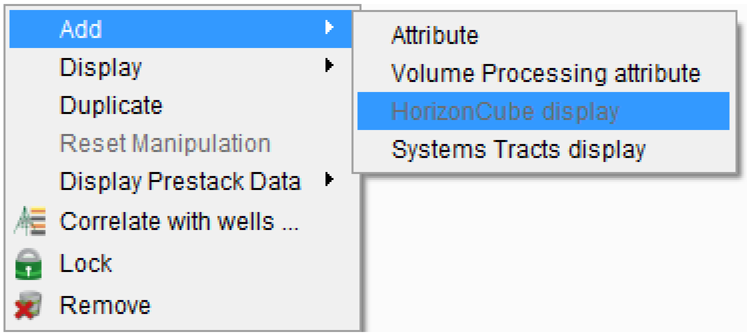
A HorizonCube is selected from the OpendTect Processing Menu (Processing > HorizonCube > 3D/2D..). The HorizonCube Control Center pops up. In this window, the active HorizonCube is selected via the select button. Once a HorizonCube is selected, it can be displayed on the element (e.g. Inline/Crossline/2D Line) on which it has been computed.



Filter *

223 113 114 14
223 113 124 14
bgfast 1,2
HC 1,2 fft 223 113
HC 1,2 fft 223 113 11
HC trim D0-D6 4ms 3iter
MC Test 223 113
MC Test steering test
MC Test steering 1.2

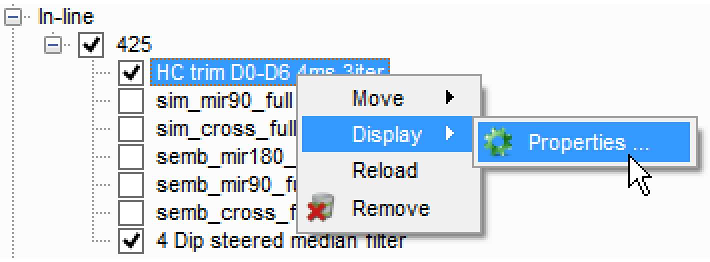
...o_Network\Surfaces\223_113_114_14.hci

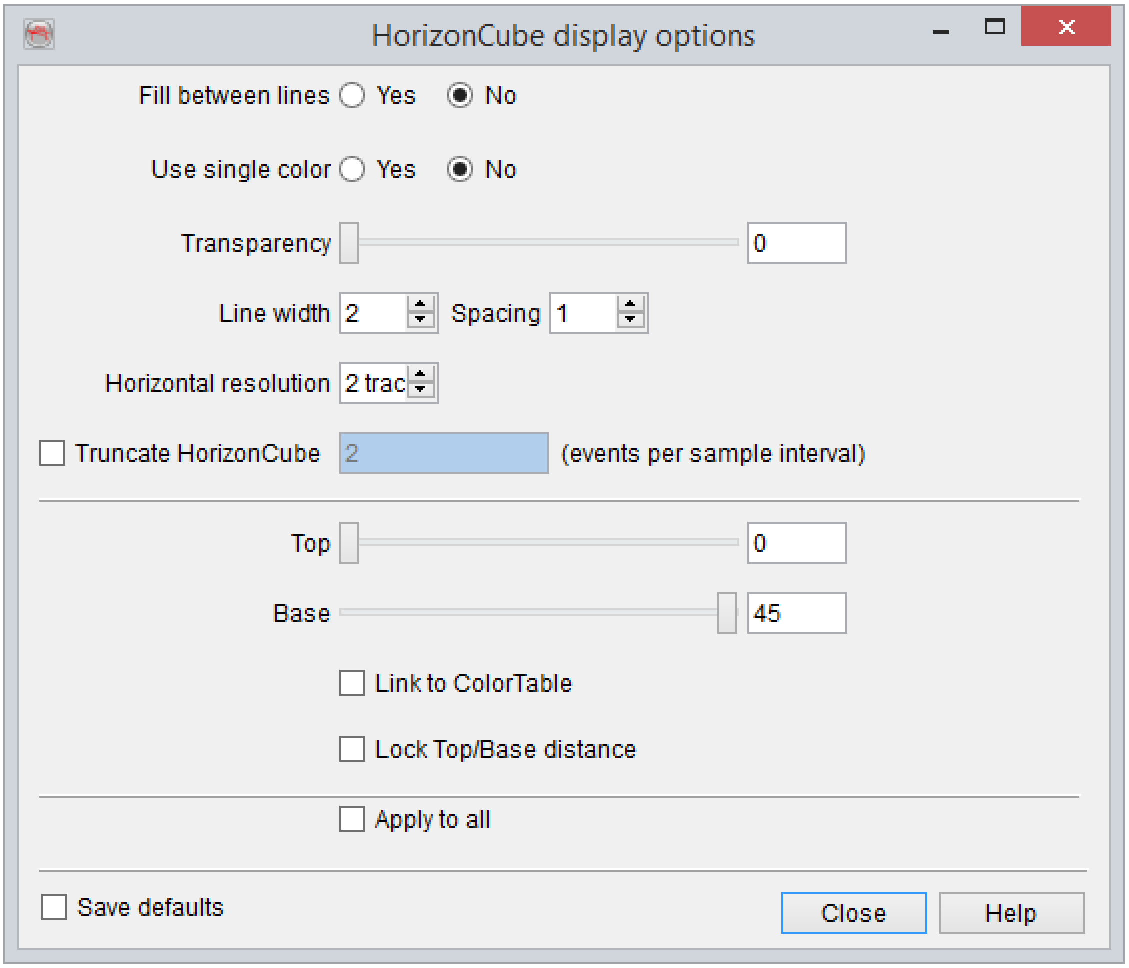


HorizonCube Selection *A HorizonCube display on a selected inline*

A HorizonCube can be displayed on-the fly on a section (inline, crossline or 2D Line). This is done by selecting a stored HorizonCube and displaying it via the OpendTect Tree. For instance, display an inline in the scene, and right click on the inline number to select the "Add HorizonCube display" option.

HorizonCube display options





Fill between lines ○ Yes ● No

Use single color ○ Yes ● No

Transparency 0

Line width 2 Spacing 1

Horizontal resolution 2 trac

☐ Truncate HorizonCube 2 (events per sample interval)

Top 0

Base 45

☐ Link to ColorTable

☐ Lock Top/Base distance

☐ Apply to all

☐ Save defaults

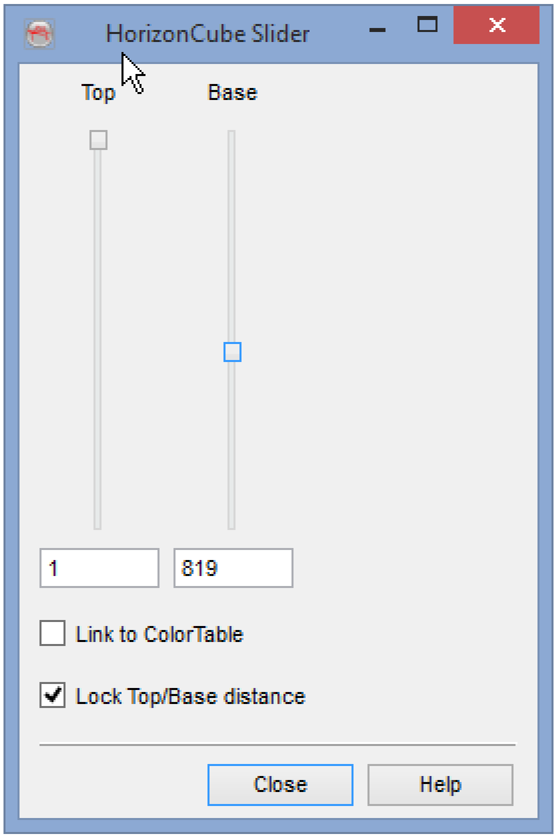
The display properties of an active HorizonCube in the scene.

From the tree, right-click on the displayed HorizonCube and select 'Display > Properties' menu item. It will launch a new dialog appears as shown above:

In this particular window, you can change the line styles (fill/ single color display), spacing between horizons, line width. Importantly, a slider (top of bottom) is also available to slide the horizons. For instance, if you have displayed thick horizon lines and you want to see that how the layers are developed, you can move the slider within same window. The horizontal resolution option will increase or decrease horizontal smoothing according to the selected traces.

Truncated HorizonCube is an option to apply on-the-fly truncation filter on a Continuous HorizonCube. The value of 2 means that only 2 events would be kept within a sampling interval and the remaining would be removed in the display. Note that the truncated HorizonCube is optimum display for interpreting depositional trends in a Wheeler scene. Therefore, this option is a valuable tool to make **SSIS Interpretation**.

Another useful visualization tool is the HorizonCube Slider:



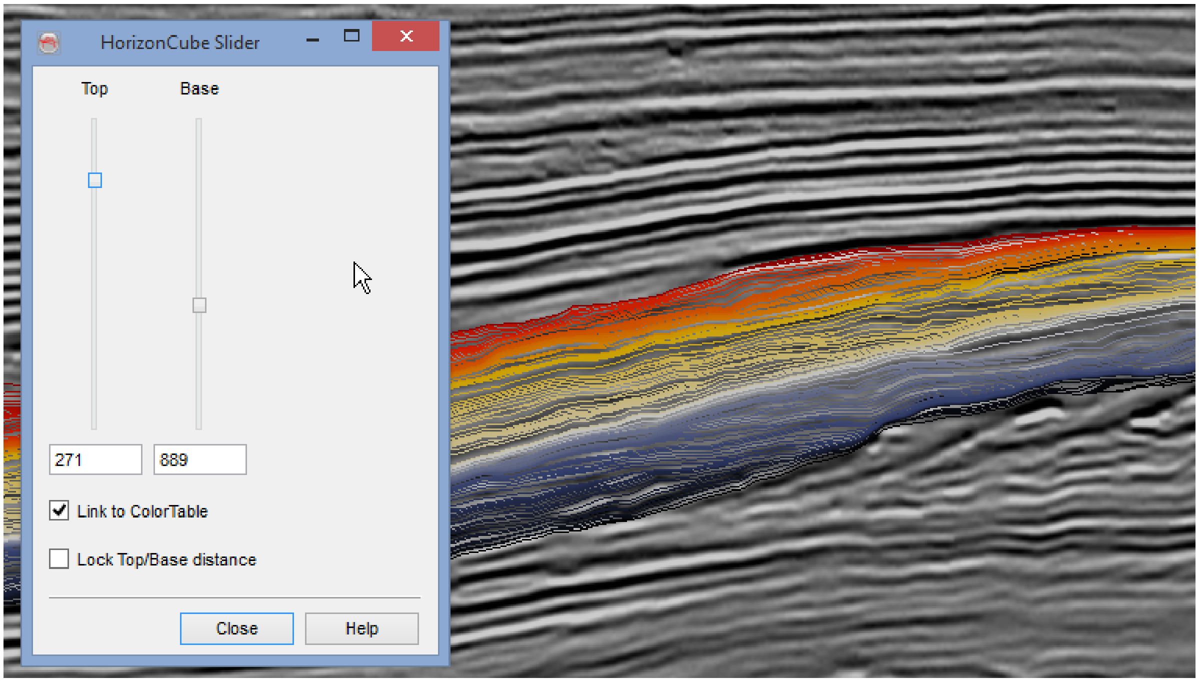
Top: 1

Base: 819

☐ Link to ColorTable

☑ Lock Top/Base distance

Once you have the HorizxonCube displayed on, for example, an inline, pressing the HorizonCube slider button brings up a simple dialogue which allows for control of the horizons displayed in the scene:



Top: 271

Base: 889

☑ Link to ColorTable

☐ Lock Top/Base distance

The top and base horizons can be manipulated via the slider or set by entering a Z value into either field. Additional options allow the user to '*Link to ColorTable*' and/or '*Lock Top/Base Distance*' to aid browsing.

HorizonCube Attributes

HorizonCube attributes are new geometrical attributes introduced by dGB Earth Sciences. These are purely stratigraphic attribute that help in identification and characterization of stratigraphic traps. These attributes can be computed directly from HorizonCube and stored as volume (as the Attribute Set Window does) or stored as a separate element such as HorizonCube Data, which contains grid based attributes. The advantage of computing HorizonCube attributes as a volume is that one could easily render it in 3D as one cannot easily render 100s of grids in 3D scene. Further advantage of this approach is that one could also integrate the HorizonCube attributes with conventional seismic attributes by using logical expressions or neural networks to characterize seismic data, to predict hydrocarbon traps etc.

Attribute Set Window

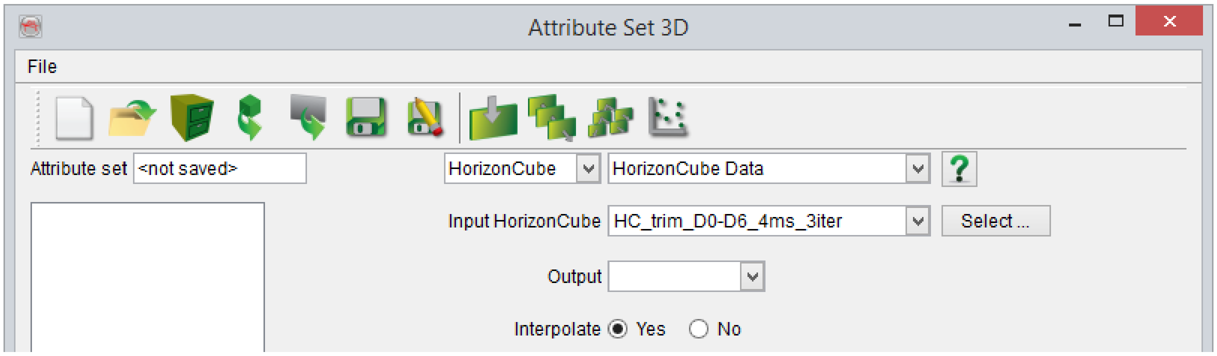
These attributes can be found in general OpendTect Attribute Set window, which can be launched via Analysis > Attributes > 3D/2D.

Using Attribute set window, one can define the following HorizonCube attributes:

HorizonCube Data

This is an attribute is used to convert the attributes that are stored under HorizonCube data into a volume. The input HorizonCube should be a 3D HorizonCube for this case as it is not supported for a 2D HorizonCube. The output field is populated if the selected 3D HorizonCube contains pre-computed **HorizonCube Data**. By default, the *Interpolate* option is set to yes as there could be gaps that you might want to fill. If you want to keep the gaps empty, set this option to No.

It is equivalent to OpendTect's basic attribute i.e. horizon. To read further, please follow the section below on *Calculate attributes along HorizonCube*.



Attribute set <not saved>

HorizonCube | HorizonCube Data

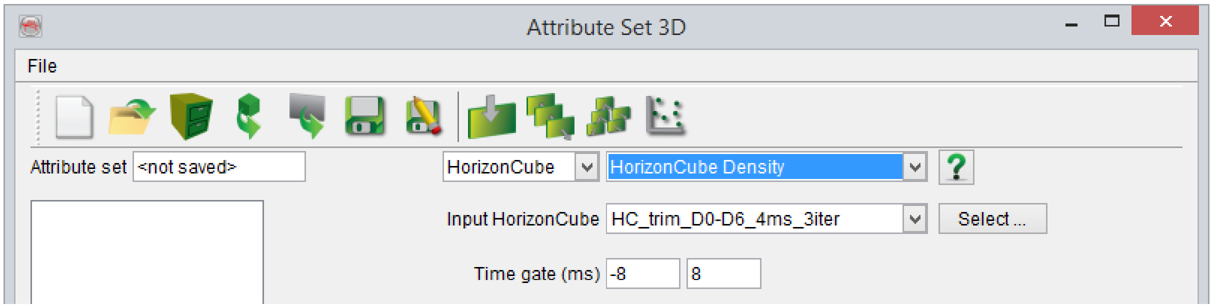
Input HorizonCube HC_trim_D0-D6_4ms_3iter

Output

Interpolate ● Yes ○ No

HorizonCube Density

It is an 2D/3D attribute that defines the density of HorizonCube events within a defined time gate. It returns an *event count*, which defines total number of events within a time gate. The output values are number of events.



Attribute set <not saved>

HorizonCube | HorizonCube Density

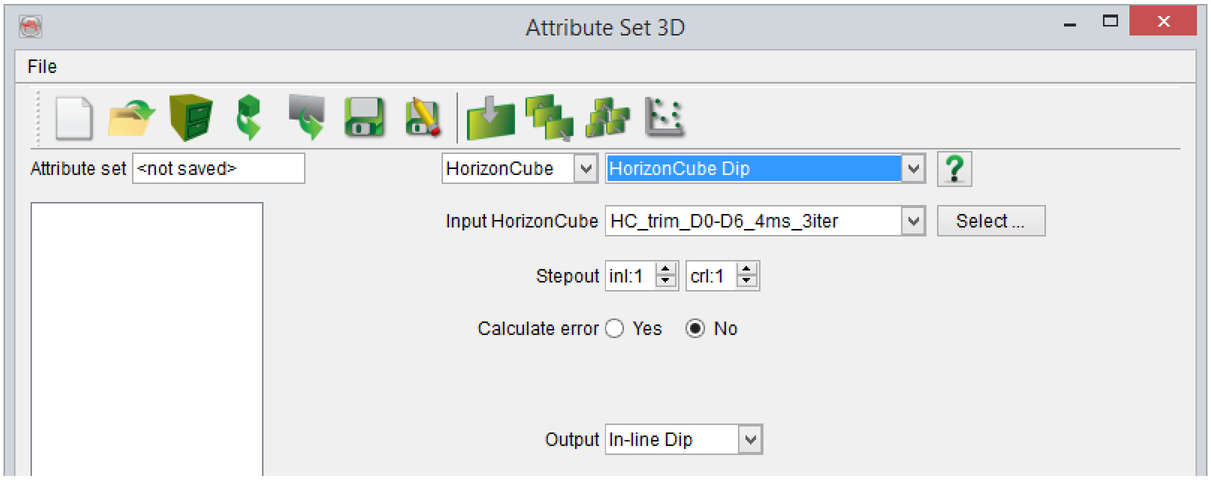
Input HorizonCube HC_trim_D0-D6_4ms_3iter

Time gate (ms) -8 8

The attributes can be use to map or visualize pinchouts, unconformities, condensed intervals, etc. These objects will appear as high density values.

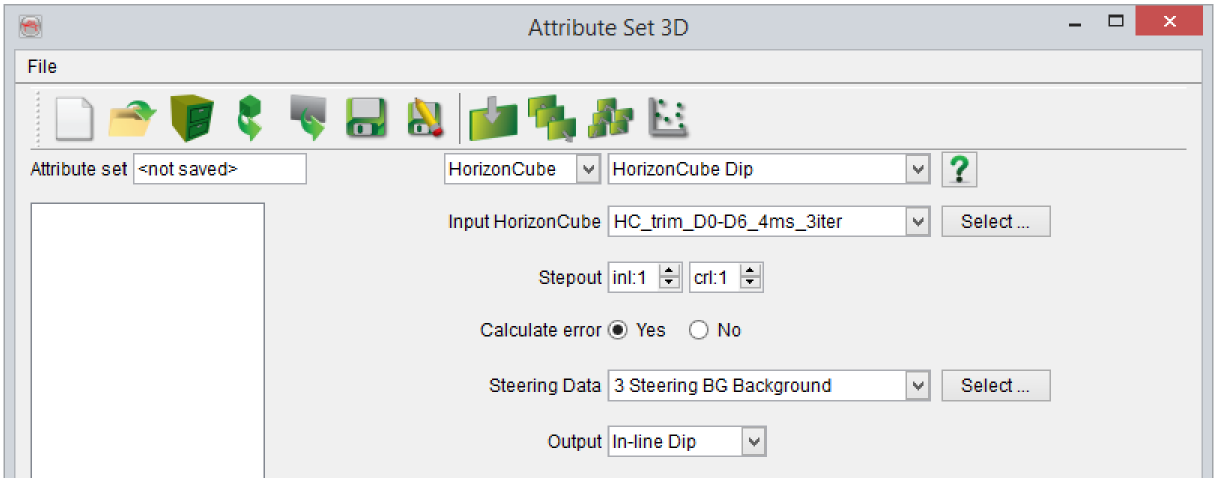
HorizonCube Dip

This attribute returns the dips of 2D/3D HorizonCube as a volume. The stepouts are the radii to compute the dips along each event. Larger stepouts define the regional dips along an event.



Attribute set <not saved>
HorizonCube
HorizonCube Dip
Input HorizonCube HC_trim_D0-D6_4ms_3iter
Stepout inl:1 crl:1
Calculate error ○ Yes ● No
Output In-line Dip

In order to QC the dips of HorizonCube vs. input SteeringCube, one may set the *Calculate error* field to Yes. This would allow you to select the parent SteeringCube to further evaluate the HorizonCube results. The errors are computed for each seismic sample by subtracting the dips of HorizonCube from the selected SteeringCube. The result is a dip difference. Lower dip errors mean that HorizonCube events do not deviate much from the selected SteeringCube and vice versa.

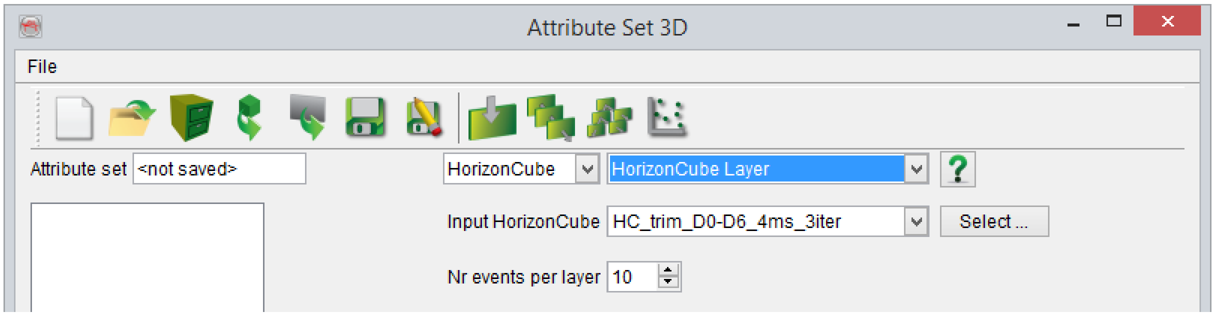


Attribute set <not saved>
HorizonCube
HorizonCube Dip
Input HorizonCube HC_trim_D0-D6_4ms_3iter
Stepout inl:1 crl:1
Calculate error ● Yes ○ No
Steering Data 3 Steering BG Background
Output In-line Dip

HorizonCube Layers

It is very convenient to visualize 100s of HorizonCube events in 3D scene. However, one could convert them into a volume and could easily render it in 3D. This attribute does such a job, which primarily governs to QC HorizonCube results.

The number of events per layer field allows to select how many events should be considered to defined a layer. The default value is set to 10, which means that each resultant layer contains at least 10 HorizonCube events. The output of this attribute is a range of values defining number of layers.



Attribute set <not saved>

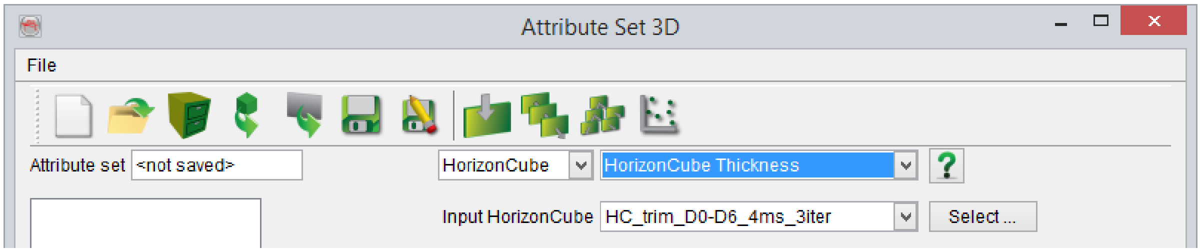
HorizonCube — HorizonCube Layer

Input HorizonCube HC_trim_D0-D6_4ms_3iter

Nr events per layer 10

HorizonCube Thickness

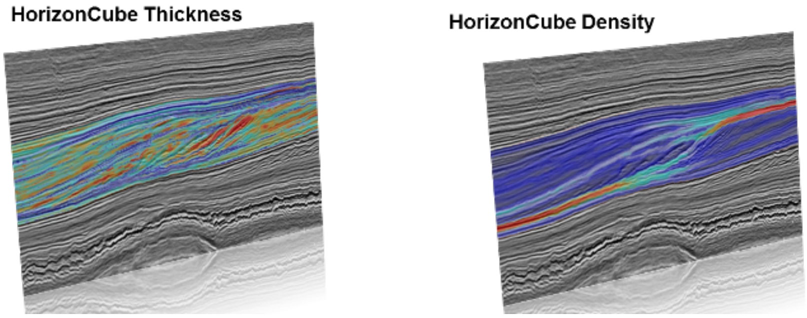
It is a vertical thickness (isochron) between two HorizonCube events. It is also a trace attribute along the HorizonCube events. Such an attribute can also be used to visualize pinchouts or unconformities (zones of near zero thickness).



Attribute set <not saved>

HorizonCube — HorizonCube Thickness

Input HorizonCube HC_trim_D0-D6_4ms_3iter



HorizonCube Thickness

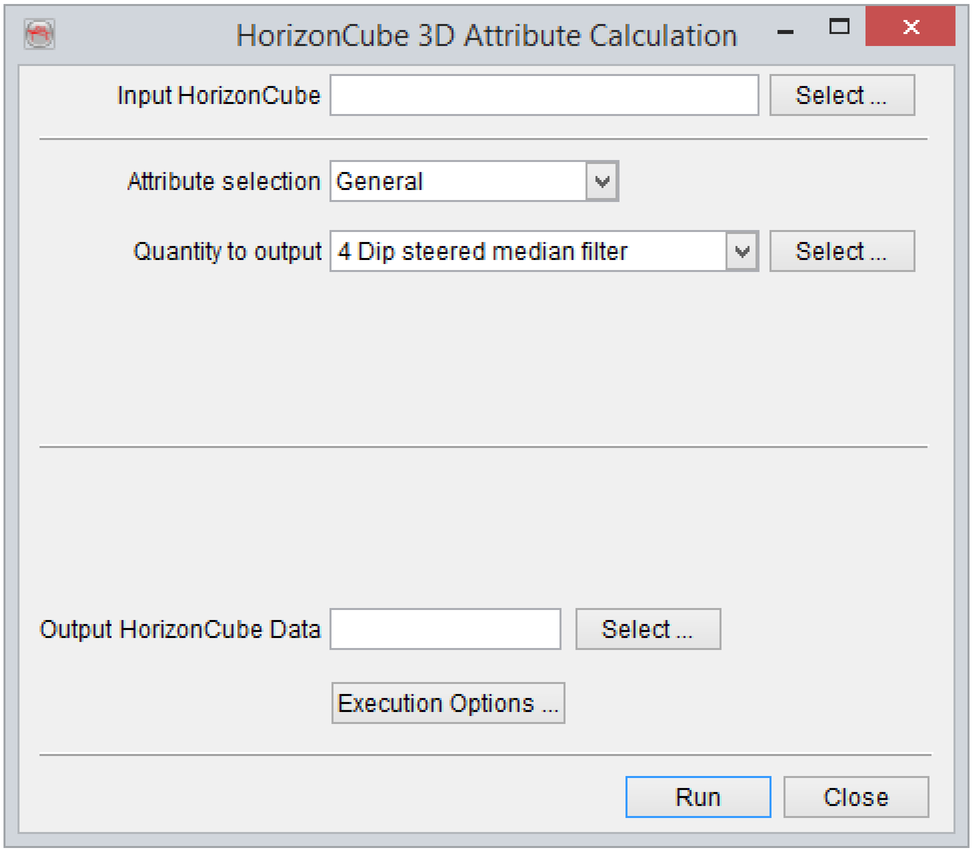
HorizonCube Density

Example of HorizonCube attributes overlain on seismic data.

Calculate Attributes along 3D HorizonCube

These attributes are computed from the **HorizonCube Control Center**. The output of all of these attributes is a set of grids, which are collectively saved as **HorizonCube Data**. These attribute grids can be visualized using **3D Slider**.

In the *Select HorizonCube* option, you can select the entire HorizonCube, or a package or an event.

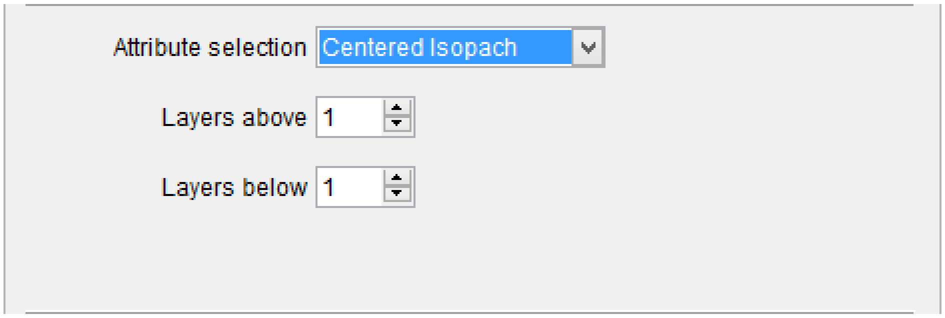


Attribute selection: General

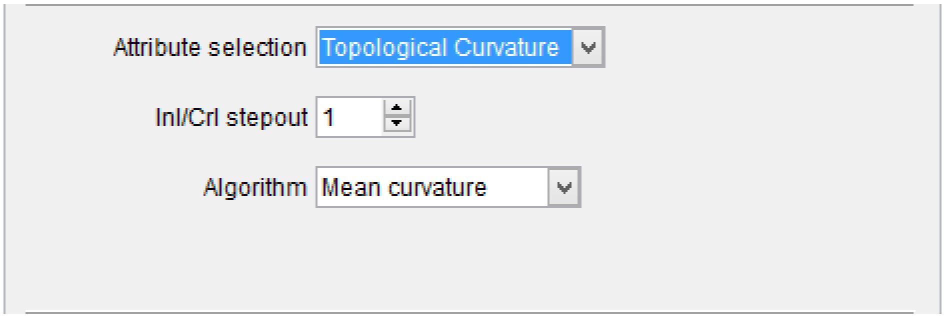
Quantity to output: 4 Dip steered median filter

General: This category is used to compute any existing attribute (stored or defined) along HorizonCube events. For instance, if you already have a stored AI or PHI volume and you would like to compute it along HorizonCube, you can use this option.

Centered Isopach: This attribute calculates the vertical thickness (TWT/depth) between two events such that they are centered by an event. In this manner, the thickness is computed along the centered event. Number of layers above/below define the relative number of events to compute the thickness. For instance, if the value for layers above/below is set to 1, for each central event, the thickness is computed by subtracting the TWT/Depth of an underlying event from the overlying event and storing the result for the central one.


Attribute selection: Centered Isopach
Layers above: 1
Layers below: 1

Topographical Curvature: This attribute defines geometrical curvature of an event, attribute or HorizonCube Data. The stepouts are number of inlines/crosslines to be used to compute the curvature. These attributes are mostly equivalent to the **conventional curvature attributes** that are directly computed from the SteeringCube.


Attribute selection: Topological Curvature
Inl/Crl stepout: 1
Algorithm: Mean curvature

References

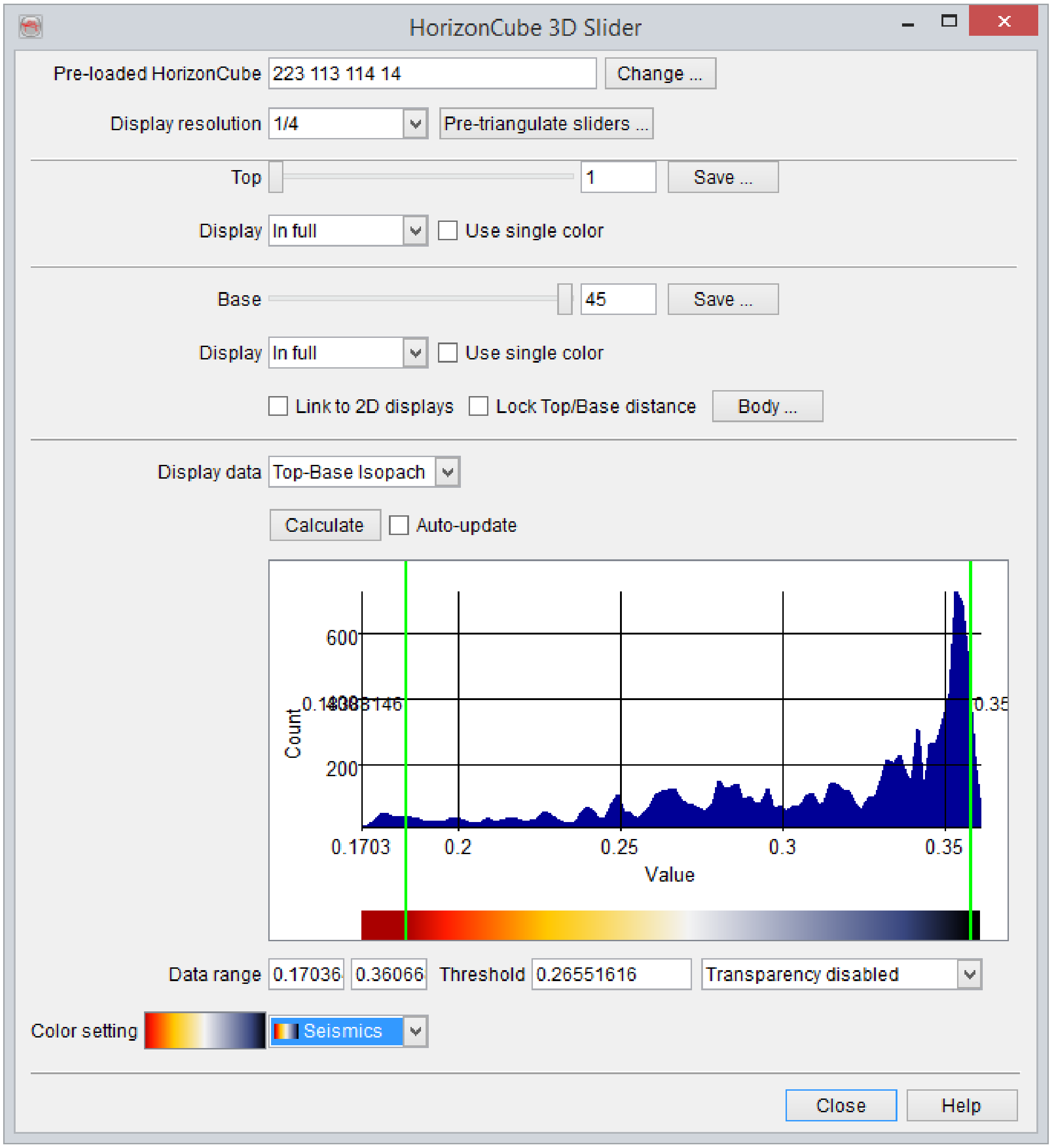
de Groot, P., Qayyum, F., On the Horizon, Oilfield Technology, p. 24-28, January 2013.

de Groot, P., Qayyum, F., Attributes play important role in seismic interpretation, Hart's E&P Magazine, p. 31-34, October 2012.

Weblink: **OpendTect Attribute Matrix**

3D Slider

A tool that is not only used to visualize the HorizonCube events in 3D but also to interpret depocenter, pinchouts, geologic bodies based on thickness /attribute map views along the events. The thicknesses are computed on the fly and are displayed as a grid on the horizons. From the isopach maps one may furthermore create *bodies*. Another benefit of the 3D slider is to save the key events as conventional OpendTect horizons.



Pre-loaded HorizonCube 223 113 114 14

Display resolution 1/4

Top 1

Display In full ☐ Use single color

Base 45

Display In full ☐ Use single color

☐ Link to 2D displays ☐ Lock Top/Base distance

Display data Top-Base Isopach

☐ Auto-update

Data range 0.17036 0.36066 Threshold 0.26551616 Transparency disabled

Color setting Seismics

Preload HorizonCube:

A user must preload a HorizonCube into the memory.

Tip: If the HorizonCube is big and there is insufficient RAM installed on the system, you may preload a package.

Top/Bottom Slider(s):

There are two sliders available to display the corresponding events in the scene and also to compute the isopach thicknesses between the events. The user has an option to hide either event.

Link to 2D displays:

It links the (top/bottom) sliders movement to the sections (inlines/crosslines) on which the preloaded HorizonCube is displayed. So, if the slider is moved, the corresponding HorizonCube overlay on sections will also be moved.

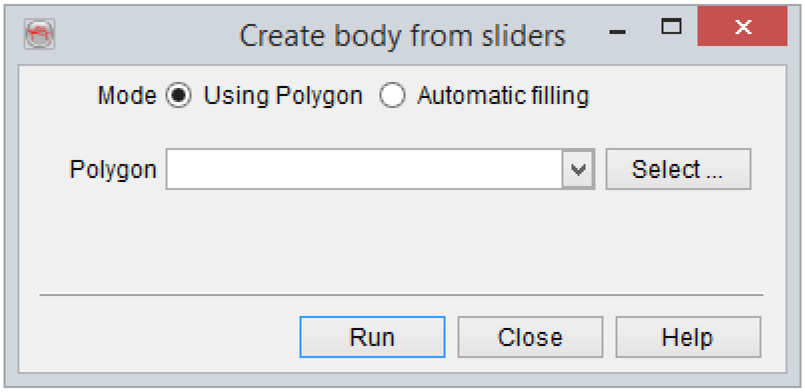
Lock top-bottom distance:

It is used to lock the number of events between the sliders positions. Once it the number of events are locked, one may use only one slider to compute the isopach between them.

Body:

It is used to create a body object within OpendTect. The body can be created either within a polygon or by using mentioning the threshold value. The threshold value is the position of the red line in the histogram of the 3D Slider. The polygon is selected if a pre-defined polygon exists. If not, you will have to create a polygon first (for details please read the pickset/polygon section of the User Documentation). The second mode i.e. *Automatic filling* requires a threshold value. For instance, if one wants to create the bodies of all isopach values that have 0.25sec thickness, 0.25 should be given as a threshold. Optionally, one could move the red line in the slider to provide that value. The body is filled either with values that are below the threshold or with the values that are above the threshold. These are set in the *Body value* radio boxes.

The body is created within a selected polygon and between the two HorizonCube events positioned using the slider.



Mode ● Using Polygon ○ Automatic filling

Polygon

Create bodies from the 3D Slider

Surface data:

None: Nothing will be computed on the displayed (Top/base) events in the scene.

Depth: Will display the Z-values (TWT / Depth) along the events.

Top-Base Isopach: It *computes* the isopach thickness between the two events selected by a user at particular sliders position. Once the user presses the *Calculate* button, the histogram of the thickness is displayed. The histogram display can be used to define the transparency on the thickness map. The green lines on the top of the histograms define the clipping of the histogram. The red line on the histogram is used to define the transparency.

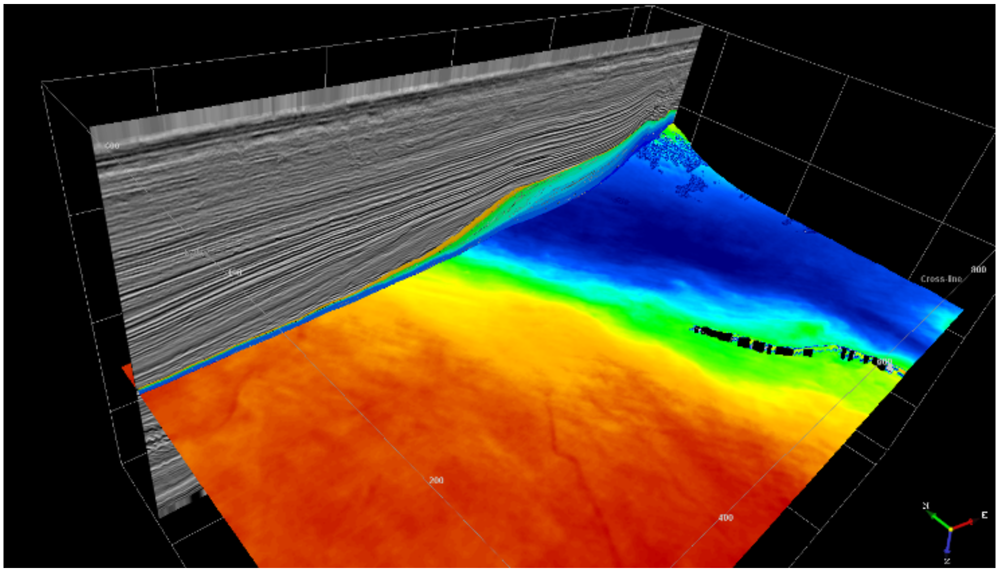
New Attribute: It is used to compute a stored seismic volume or a defined attribute along the horizons.

Transparency:

This field defines the transparency range on a colour bar. The first two fields adjacent to the transparency are the colour ranges. The third field is the transparency cut-off value. Disable option in the list box defines no transparency, above defines transparency above a given cut-off value and below defines transparency below the cut-off value.

Colour bar:

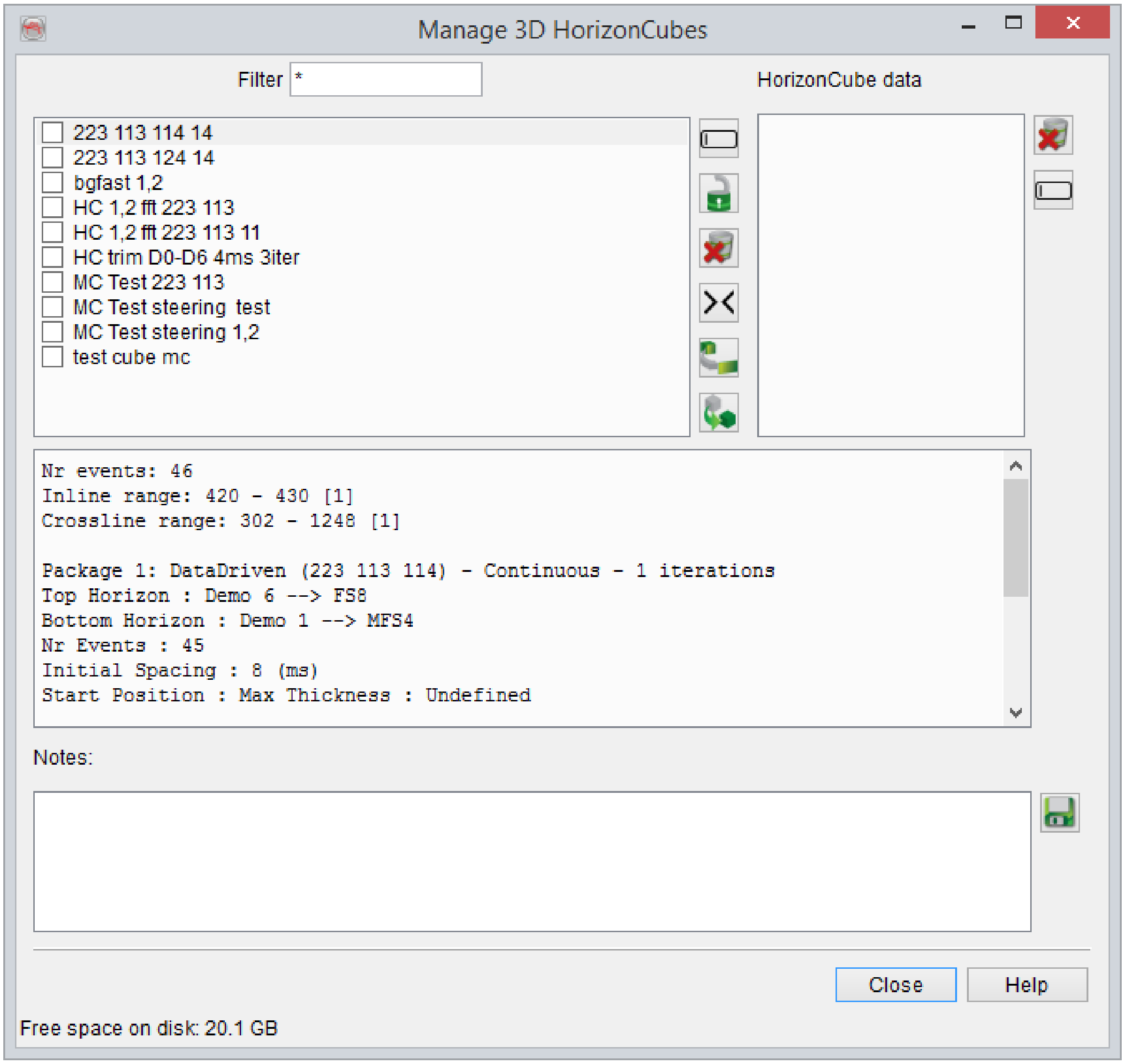
The colour ranges and the corresponding selected colour bars are available at the bottom. The active colour bar is used to display the thickness map on the displayed horizons. The colour bar can interactively be changed by scrolling the available list of pre-defined colour tables. Furthermore, one may right click on the colour bar and use the pop-up menu that work similar to the general OpendTect colour bar.



Import, Export

See following chapters below.

Manage HorizonCube



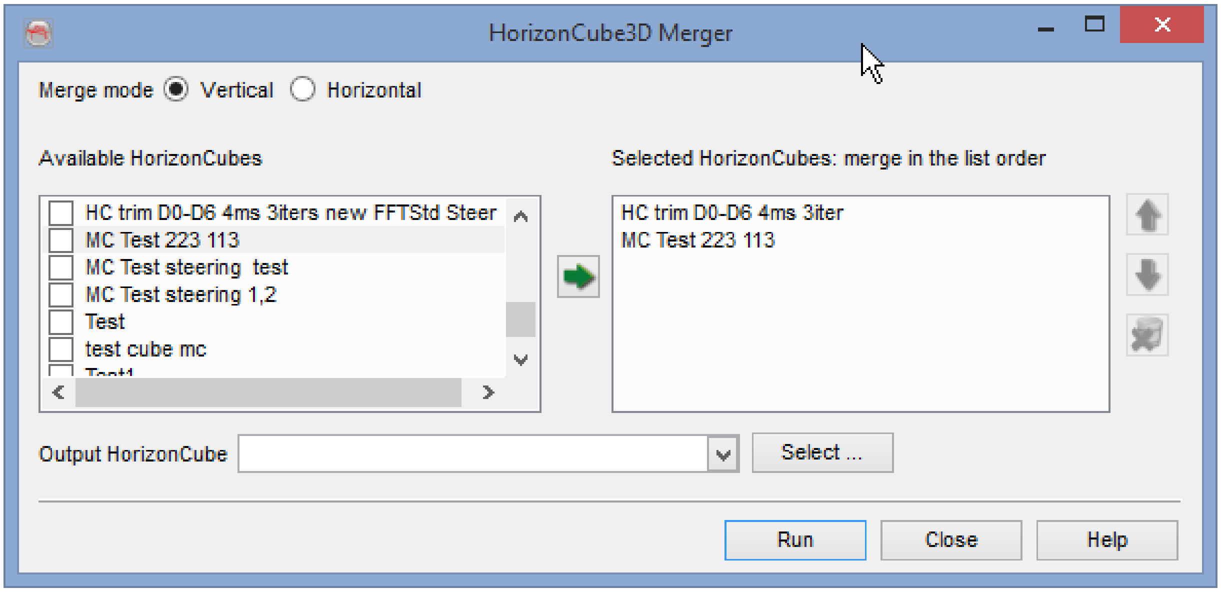
☐ 223 113 114 14
☐ 223 113 124 14
☐ bgfast 1,2
☐ HC 1,2 fft 223 113
☐ HC 1,2 fft 223 113 11
☐ HC trim D0-D6 4ms 3iter
☐ MC Test 223 113
☐ MC Test steering test
☐ MC Test steering 1,2
☐ test cube mc

```
Nr events: 46
Inline range: 420 - 430 [1]
Crossline range: 302 - 1248 [1]

Package 1: DataDriven (223 113 114) - Continuous - 1 iterations
Top Horizon : Demo 6 --> FS8
Bottom Horizon : Demo 1 --> MFS4
Nr Events : 45
Initial Spacing : 8 (ms)
Start Position : Max Thickness : Undefined
```

Free space on disk: 20.1 GB

- *Rename:* Rename the selected HorizonCube.

- *Lock:* Sets the selected HorizonCube to a read-only mode and disables editing.
- *Remove/Delete:* Removes the selected HorizonCube from the survey.
- *Set as Default:* sets the HorizonCube to default.
- *Copy Horizons:* Extracts horizons (or patches) from the selected HorizonCube. The 'copy to horizon' button extracts one or more seismic horizons from the HorizonCube. This launches the above window. Either a horizon geometry from one sequence (i.e. Package) or from individual automated horizons (Event)can be extracted. The area sub-selection can also be used to restrict the output horizon geometry either within a volume or within a polygon. The output horizon can then be used for further analysis.
- Is used to *merge HorizonCubes* vertically. It will launch the *HorizonCube Merger* dialog from where multiple HorizonCubes would be selected.



Merge mode ● Vertical ○ Horizontal

Available HorizonCubes

☐ HC trim D0-D6 4ms 3iters new FFTStd Steer
☐ MC Test 223 113
☐ MC Test steering test
☐ MC Test steering 1,2
☐ Test
☐ test cube mc

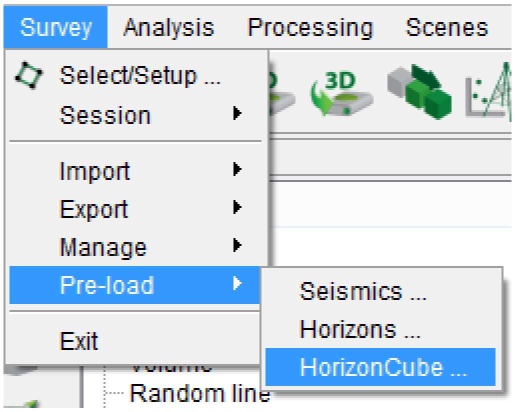
Selected HorizonCubes: merge in the list order

HC trim D0-D6 4ms 3iter
MC Test 223 113

HorizonCube Data: It contains the list of all attributes that are computed and stored along HorizonCube using **HorizonCube Control Center**. A HorizonCube data is equivalent to "horizon data". It contains set of grid based attributes that are computed along selected/entire events of HorizonCube. This data can also be converted to a volume using **HorizonCube Data attribute**.

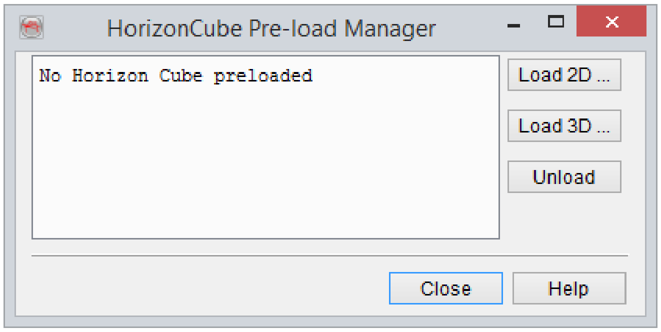
Preload HorizonCube

This feature is a useful tool in 2D/3D-HorizonCube visualization. If you intend to visualize a HorizonCube in the scene or on an inline/crossline, then you may also want to load it in the computer's memory. This will allow you to quickly display the HorizonCube on section without re-reading the data always. A HorizonCube can be preloaded from the top menu (see below).

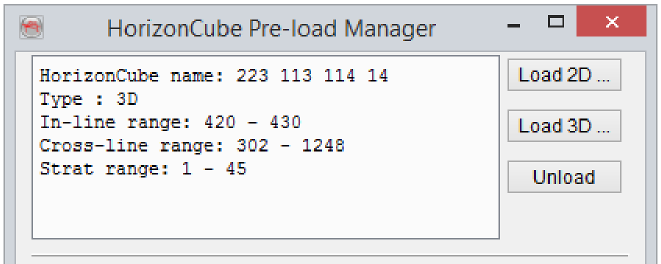


How to launch the HorizonCube pre-loader dialog.

In the Preload HorizonCube window, you can select one HorizonCube to be loaded into the memory. This is done by pressing the *Load HorizonCube* button. In the pop-up HorizonCube selection dialog, you can select either all packages or selected events by clicking the radio box. Further sub-selections can also be made by reducing the area of sub-selection i.e. restrict the HorizonCube within an inline/crossline ranges. The OK button for the HorizonCube selection would start reading and loading the selected HorizonCube.



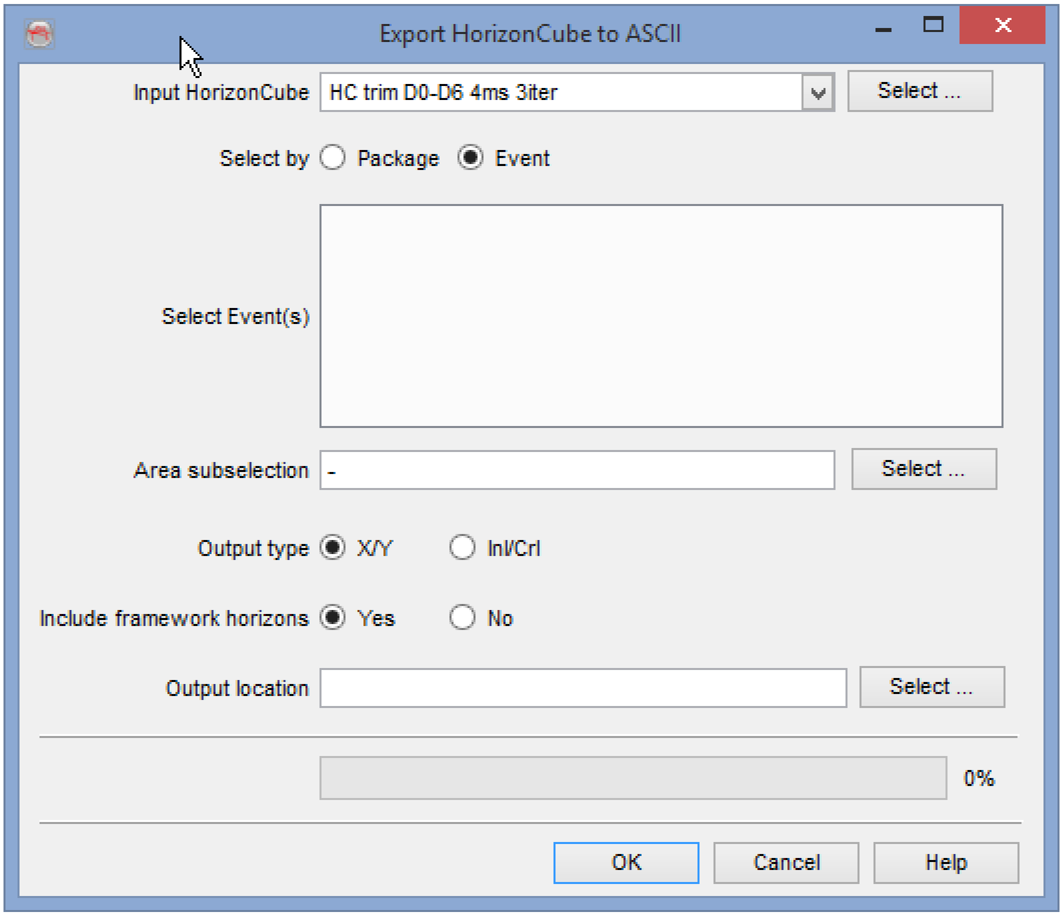
```
No Horizon Cube preloaded
```



```
HorizonCube name: 223 113 114 14
Type : 3D
In-line range: 420 - 430
Cross-line range: 302 - 1248
Strat range: 1 - 45
```

ASCII Export (3D)

The 3D HorizonCube can be exported as a single/multiple horizon, into an ASCII file. This can be done via export sub menu (Survey > Export > HorizonCube 3D). In the 'Export HorizonCube to ASCII' window, select the HorizonCube and export either the package or the events.



Export HorizonCube to ASCII

Input HorizonCube: HC trim D0-D6 4ms 3iter

Select by ○ Package ● Event

Select Event(s)

Area subselection: -

Output type ● X/Y ○ Inl/Crl

Include framework horizons ● Yes ○ No

Output location

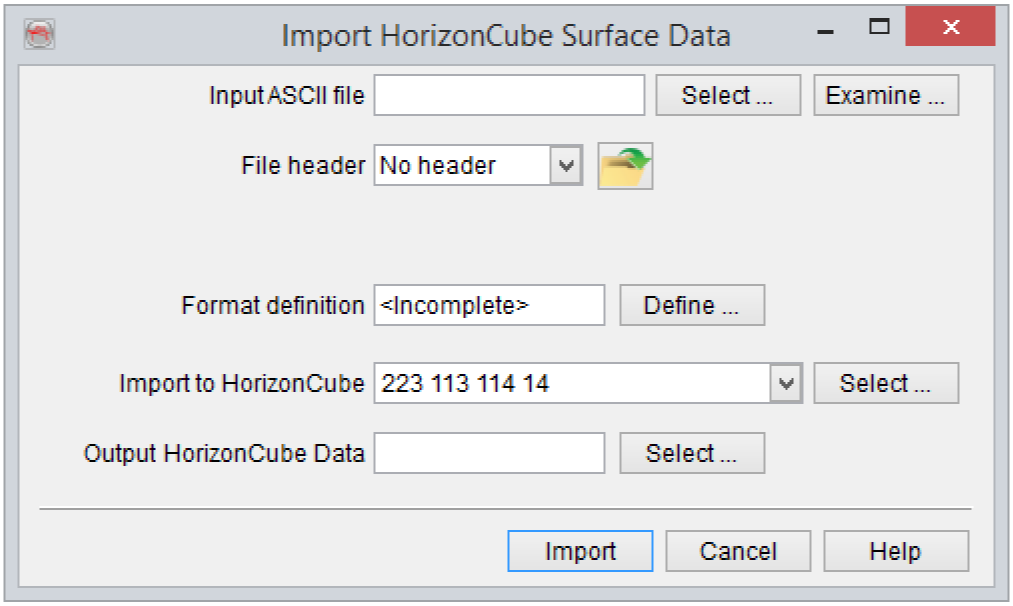
0%

- **Output Events:** It is used to export a selected event(s) to an ASCII file. Multiple selection could be made by dragging the mouse over a range of events.
- **Output Package:** It is used to export the events in a selected package(s) to ASCII files. Multiple selection of the packages could also be made by dragging the mouse over multiple packages. To export a pacakge, the output location must be a directory.

- **Area sub-selection:** It is an optional sub-selection to export the events. The user can restrict the output files within an area of interest defined by a range/polygon/table.
- **Output type:** To export the coordinates in a file, you can select X/Y radio box. To export the data in inline/crossline sorted formate, the Inl/Crl radio box is selected.

Import HorizonCube Service Data

The HorizonCube surface data can be imported in OpendTect as an ascii file. This is done via:



File header: No header

Format definition: <Incomplete>

Import to HorizonCube: 223 113 114 14

Output HorizonCube Data

Some few specifications are needed such as: File header, the format definition. The surface data is then imported as HorizonCube.

Sequence Stratigraphic Interpretation System

Table of Contents

- Introduction
- Interpretation Window
- HorizonCube Slider
- Wheeler Transform / Wheeler Scene
- Flatten Horizon/Seismics
- Systems Tracts Attributes
- Manual SSIS

Introduction

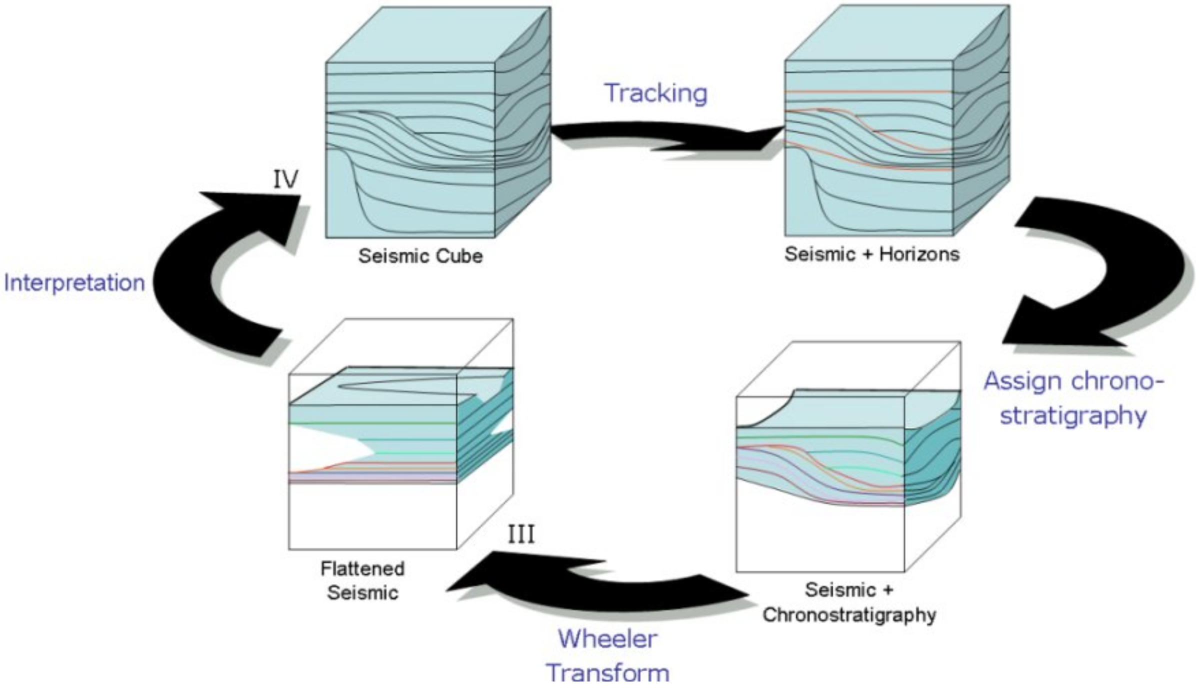
Automated horizon-tracking, Wheeler transforms, and systems tracts interpretations are unique 3D seismic interpretation capabilities that are supported in OpendTect SSIS; dGB's new Sequence Stratigraphic Interpretation System (SSIS for short).

In SSIS, seismic interpreters are offered new ways of visualizing and analyzing seismic data, which leads to better insights of sediment deposition, erosion, and timing. In combination with OpendTect's neural networks plugin, users can follow up with seismic facies clustering and study the resulting patterns and bodies, and their spatial distribution, in relation to geologic timing and systems tracts.

The OpendTect SSIS workflow is an *iterative process* that consists of four basic steps:
First, major sequence boundaries are interactively mapped with OpendTect's horizon trackers (or these are imported from other interpretation systems). Next, all possible intermediate horizons are auto-tracked with

sub-sample accuracy. Each intermediate horizon corresponds to a geologic time line, i.e. a chrono-stratigraphic event, and can be identified by a unique index indirectly to relative geologic age of the event. The index is automatically assigned to each horizon in stratigraphic order according to its relative thickness. Horizons that form the base of one sequence and the top of the sequence below it, are assigned the same index. This index can be manually overruled for both indexes separately. In this way the duration of deposition and duration of hiatuses can be changed. When a horizon is diachronous (time transgressive), the index is the youngest index for the top of a sequence and the oldest index for the base of a sequence.

Two auto-track modes are supported: *model driven* and *data driven*. In the model driven approach the HorizonCube is calculated by interpolation or by adding horizons parallel to upper or lower bounding surfaces. In the data-driven mode, seismic horizons are auto-tracked by following the local dip and azimuth of the seismic events. This mode requires OpendTect's dip-steering plugin to pre-calculate the SteeringCube containing the dip and azimuth information.



SSIS Workflow

The third step in the process is the actual *Wheeler Transform*. Basically, this flattens the seismic data (or derived attributes) along the auto-tracked horizons and honors truncations and erosional/depositional hiatuses. Studying the data in the Wheeler transformed domain increases our understanding of the *spatial distribution and timing of sediment deposition*. To quote sequence stratigrapher Peter Vail: *"You never fully* *appreciate the implications of a sequence stratigraphic interpretation until you've transformed it to a Wheeler diagram"*.

The fourth step in the SSIS workflow is *systems tract interpretation*. Inspecting the spatial distribution of the sequences and lap-out patterns of seismic events, in both the normal domain and the Wheeler transformed domain, enables the user to segment the seismic sequences into systems tract. Systems tracts are specified per HorizoncCube range. To accommodate different naming conventions, the software allows the default systems tract names (*High Stand, Falling Stage, Low Stand, Transgressive*) to be replaced by user-defined names.

With the HorizonCube calculated and optionally systems tracts interpreted, the user can continue the sequence stratigraphic analysis with seismic facies interpretation. Visualizing systems tract-interpretations together with the HorizonCube and overlaying the normal- and Wheeler transformed domains helps in identifying depositional features of interest. More advanced seismic facies analysis is possible with OpendTect's neural network plugin. Waveform segmentation along any HorizonCube event is a simple and straightforward approach for visualizing seismic patterns per stratigraphic event. Similarly, seismic attributes can be clustered by a neural network to reveal 3D bodies. Further, the user can train a neural network to recognize seismic bodies in 3D from user-defined examples (supervised mode). These approaches have already been supported in OpendTect's neural network plugin. The new version of SSIS allows the results to be visualized and analyzed in the stratigraphic domain, i.e. without distortion by the structure. To appreciate this, you could load (a subset of) a Wheeler-transformed seismic volume (e.g. an attribute, neural network clustered result, or AI cube) in the volume viewer of the Wheeler scene and use the time-slice display to scroll the data. Time slicing in the Wheeler domain corresponds to horizon-slicing in the normal (structural) domain, hence all patterns observed in each slice belong to the same geological event.

SSIS Toolbar

The SSIS toolbar contains the icons that are used by a user to manage the HorizonCube and corresponding stratigraphic interpretations:

- Launch **SSIS interpretation module** . In the SSIS interpreation window the user can interpret sequence stratigraphy by using the HorizonCube. Note that the correct HorizonCube needs to be selected first.

- The HorizonCube-slider is used to visualize the seismic horizons in a stratigraphic manner i.e. slide the horizons from top to bottom to visualize the depositional patterns.

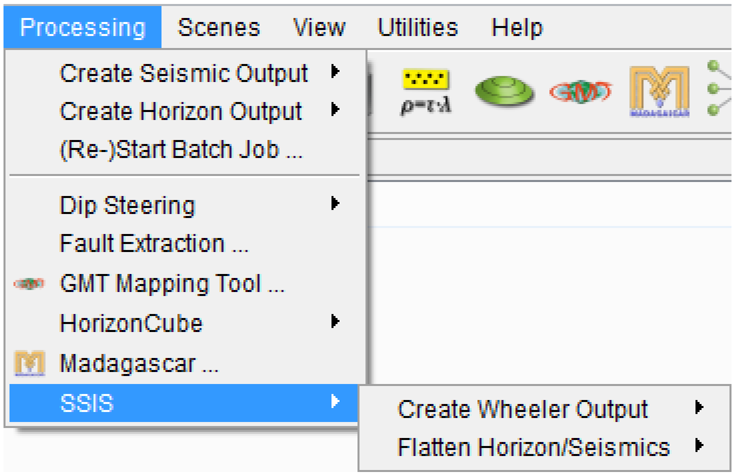
Interpretation Window

See following chapters below.

Overview

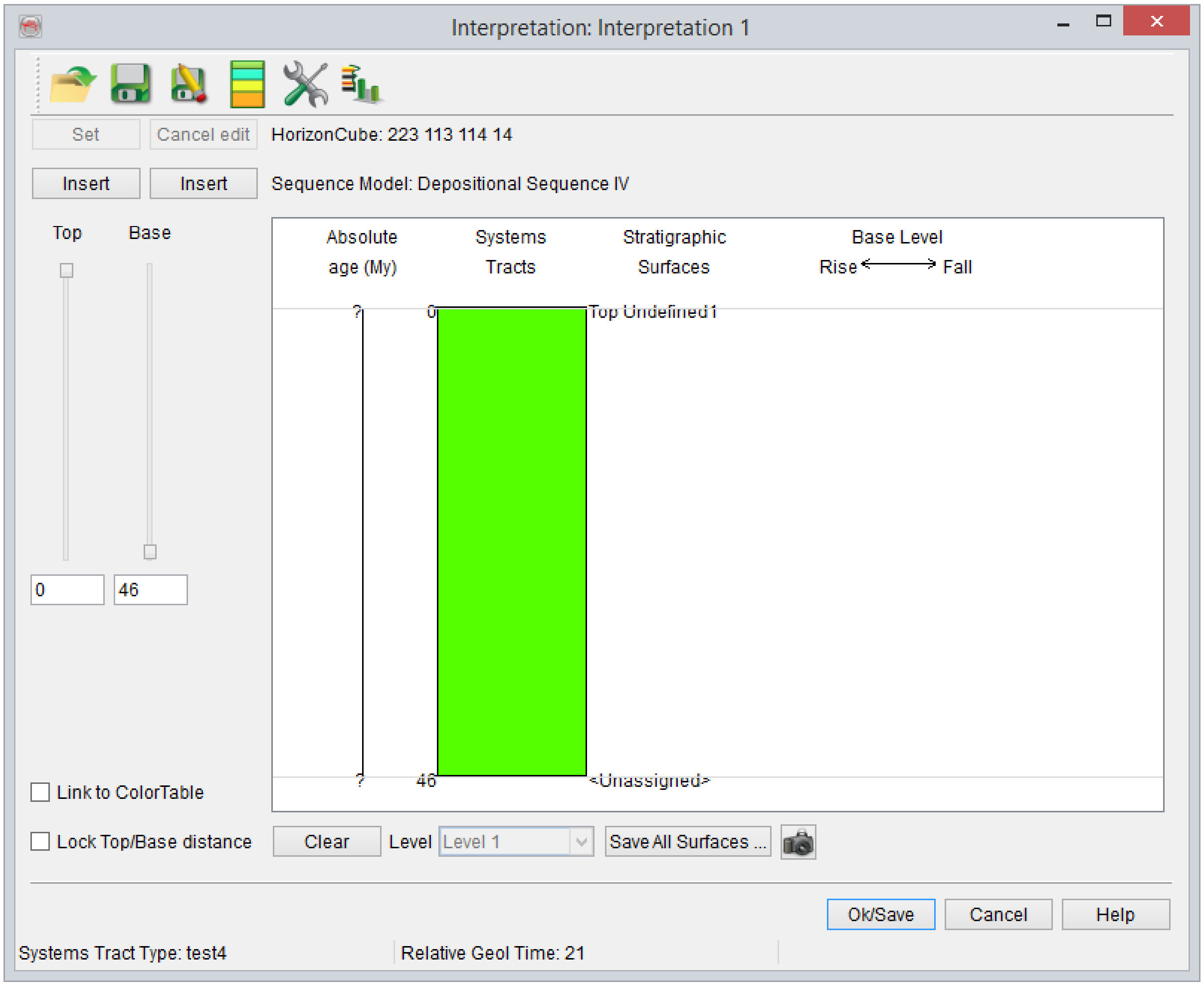
In OpendTect-SSIS, an interactive module is used to interpret a sequence in details. This is done by displaying the HorizonCube on an element (inline/crossline/2D line) and launching the interpretation setup:

- Select the HorizonCube.
- Add the HorizonCube display in the tree.
- From the SSIS menu select *SSIS* > *Interpretation* > *New/Edit* or use the short-cut button from the SSIS toolbar.



Launch the interpretation window

This window is used to perform a sequence stratigraphic interpretation from the HorizonCube (2D/3D). The interpretation is done in an interactive manner i.e. by sliding the **HorizonCube-slider** up and down (right panel) and visualizing the depositional changes in the OpendTect scene. When a shift in depositional pattern is apparent, a sequence stratigraphic surface can be inserted by pressing the *Insert* button. This is also interactive, i.e. the slider will update the display in the OpendTect scene and the surfaces will be added in the graphical area (left panel) of the Interpretation window. An example of this interpretation window with a sequence stratigraphic interpretation is shown in the figure below.



Interpretation: Interpretation 1

HorizonCube: 223 113 114 14

Sequence Model: Depositional Sequence IV

Top: 0

Base: 46

Level: Level 1

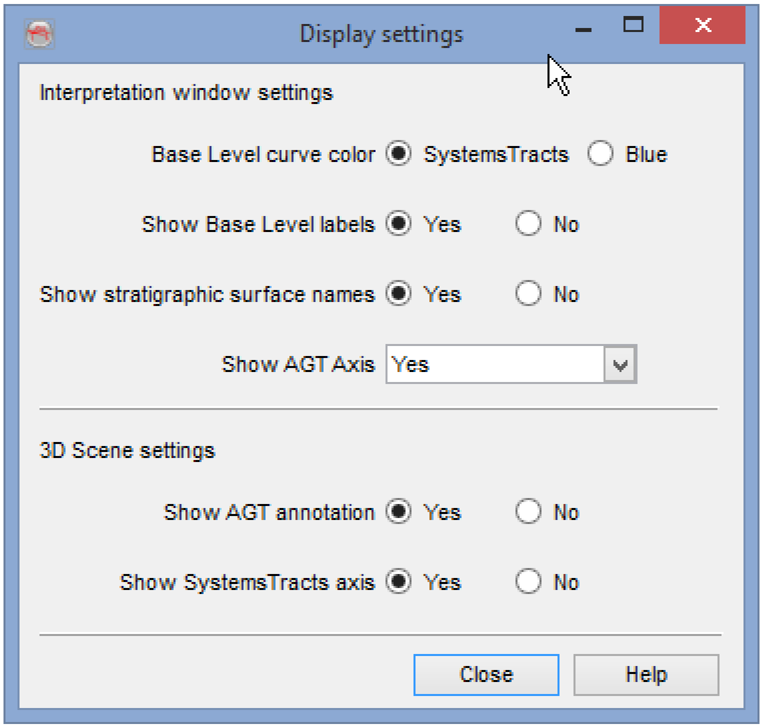
Systems Tract Type: test4

Relative Geol Time: 21

Interpretation window for SSIS.

- Open the stored interpretation. Note that more than one interpretation can be stored for any calculated HorizonCube.

- Save the interpretation.
- Save the interpretation with a new name i.e. Save As..
- [Manage the Sequence Model](#).
- Display settings:



Interpretation window settings

Base Level curve color ● SystemsTracts ○ Blue

Show Base Level labels ● Yes ○ No

Show stratigraphic surface names ● Yes ○ No

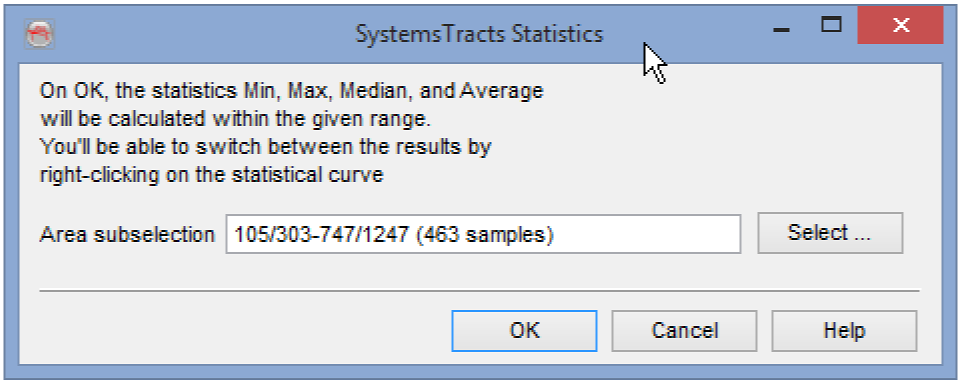
Show AGT Axis Yes

3D Scene settings

Show AGT annotation ● Yes ○ No

Show SystemsTracts axis ● Yes ○ No

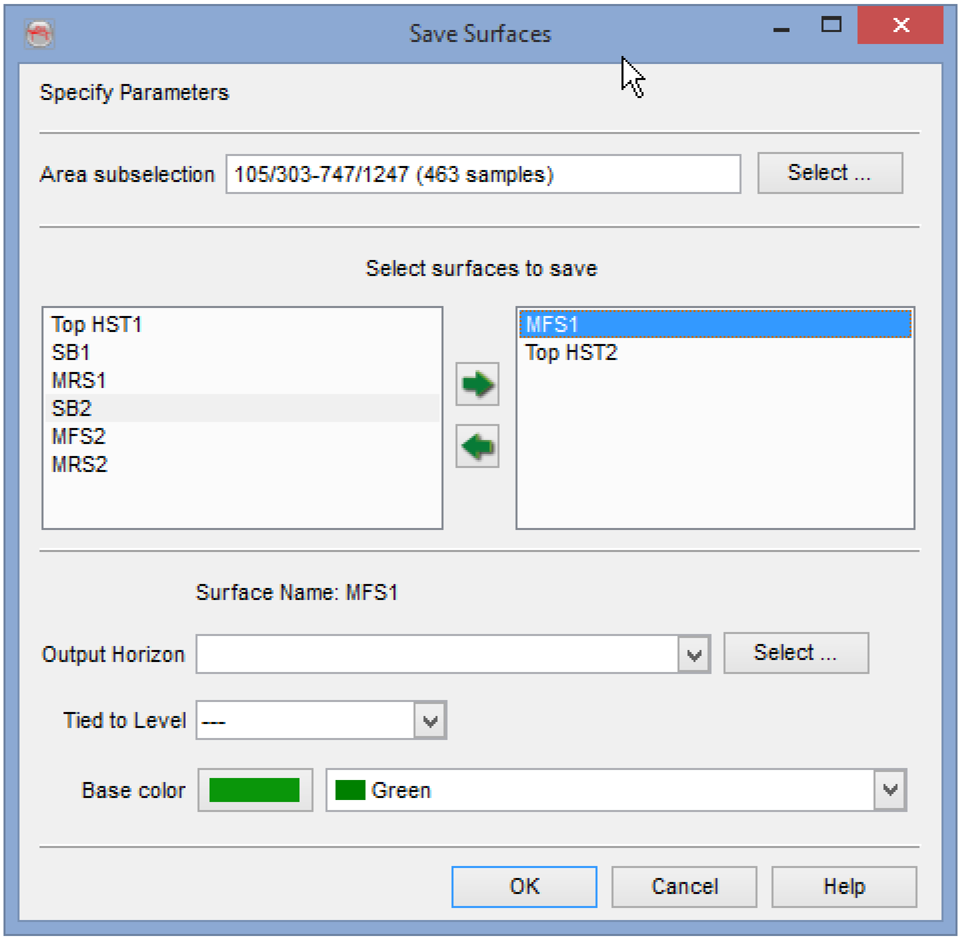
- Calculate the statistics for your model:



On OK, the statistics Min, Max, Median, and Average
will be calculated within the given range.
You'll be able to switch between the results by
right-clicking on the statistical curve

Area subselection 105/303-747/1247 (463 samples)

- Take a snapshot of the graphical area and save it.
- *Link to ColorTable:* This options is used to lock the color table while moving the slider position up and down.
- Save All Surfaces ... will present you with a dialogue where you may select the surfaces you wish to save and set basic options for these:



Save Surfaces

Specify Parameters

Area subselection: 105/303-747/1247 (463 samples)

Select surfaces to save

Top HST1
SB1
MRS1
SB2
MFS2
MRS2

MFS1
Top HST2

Surface Name: MFS1

Output Horizon:

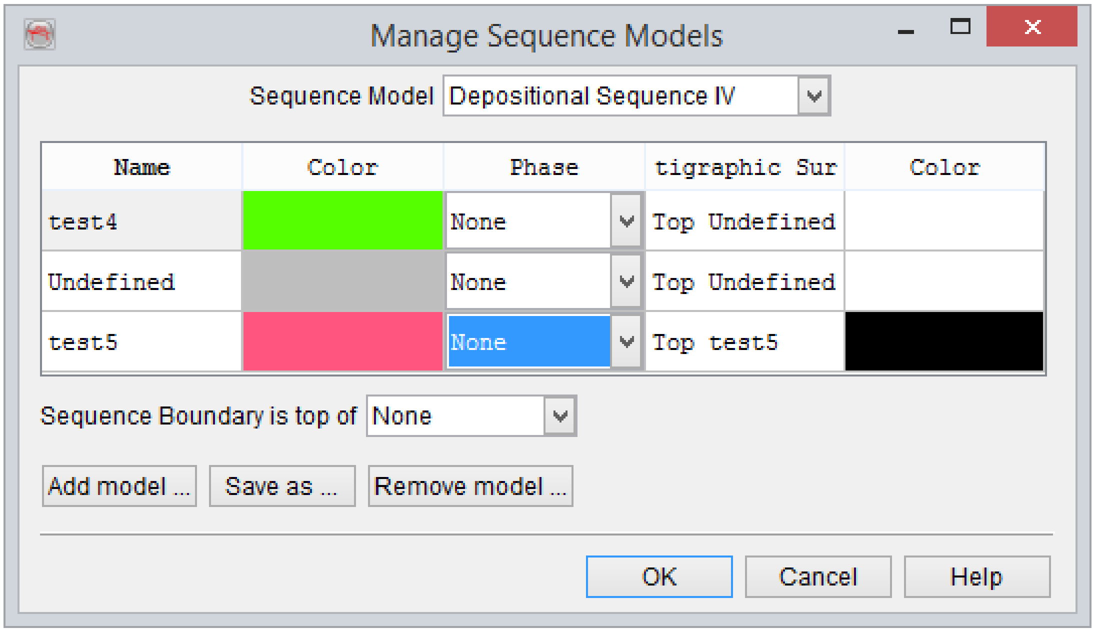
Tied to Level: ---

Base color: Green

Colour-coded Base level Curve

The base level curve colour could be changed in such a way that a segment bounded by two nodes/surfaces represents the corresponding systems tract colour. This is done by right-clicking on the curve and selecting the option colour coded.

Select, Define a Depositional Model



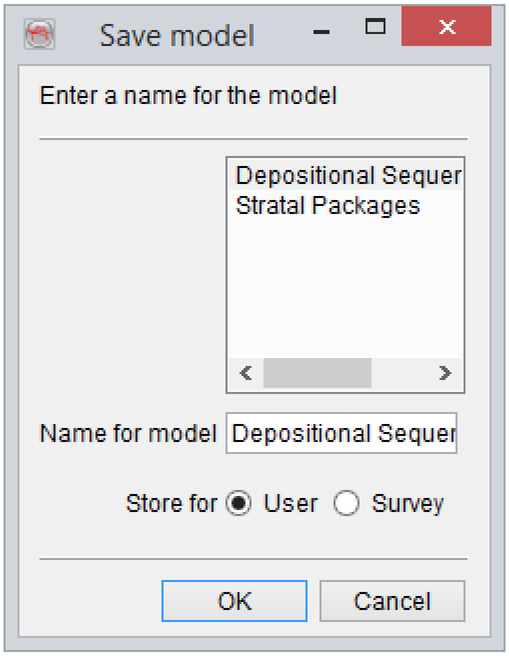
Sequence Model: Depositional Sequence IV

Name	Color	Phase	tigraphic Sur	Color
test4	🟩	None	Top Undefined	
Undefined	▨	None	Top Undefined	
test5	🟥	None	Top test5	⬛

Sequence Boundary is top of: None

Sequence Models Dialog

Seismic sequence stratigraphic interpretations in SSIS are started by selecting/creating a depositonal model. The models are color-coded representations of systems tracts in OpendTect and the corresonding base level curve is automatically drawn in the Interpretation system. To select/define one of such model, click the *Select Sequence Model* 🛠 icon. By default, there are five different standard depositional sequence models available in the setup. You can either select any one of them or press *Add Model* to add new model in the *Systems tract* setup window. The model can be saved at a user or at a survey level. The user level is used to save the model for the current OpendTect user name only and it will be accessible by that user name only in all surveys. However, you can also save it at a survey level, which means that the model would be available to all surveys in the current surveys directory.



Name for model: Depositional Sequer

Store for ● User ○ Survey

A user defined Sequence Model

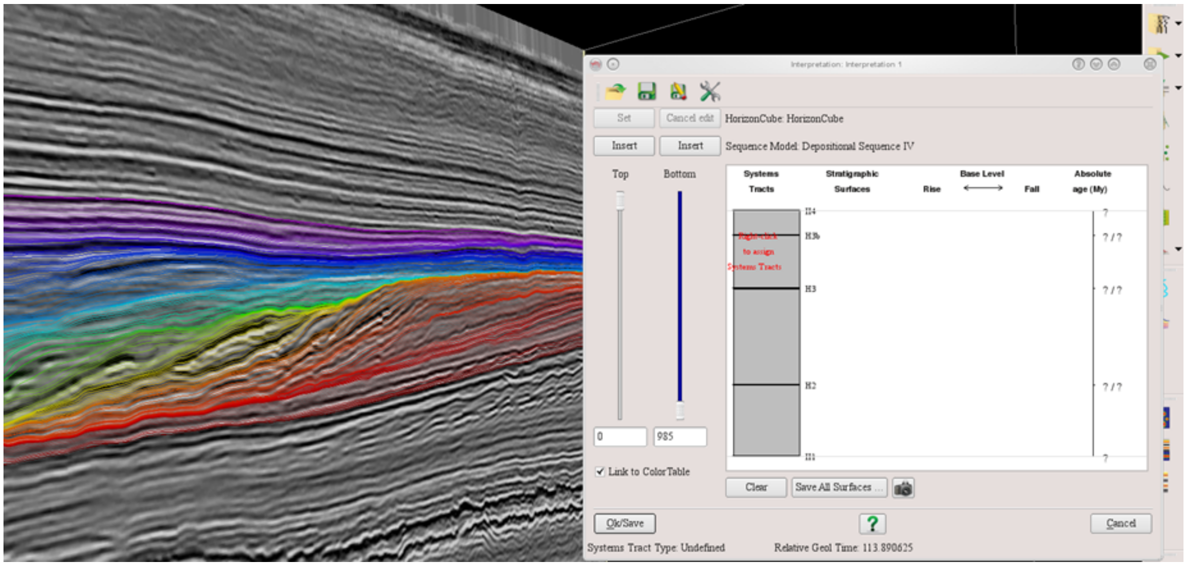
In order to define a new model, press the 'Add Model' button and type the name (*Name for Model*) in the pop-up window. Store the newly created model either for the current user or for the entire survey. On 'OK', a blank model will be added in the systems tracts Setup. Right-click on the *Undefined* text in the *Name Column* and select *Insert row after* sub menu. This will add a blank line below the undefined row. Type a name for the systems tract of the newly created model. To assign a color for the newly added systems tract, double click on the white cell of the same row. Select an appropriate color for the systems tract.

The sequence boundary could also be changed for a depositional model. You may put the sequence boundary at a selected systems tract name. This sequence boundary will appear in the base level curve.

Note: Most of the fields (name, colour) are editable by double clicking on the corresponding cell. You may also override the colour and name as you like. The standard colours and names of the surfaces are used while converting the surfaces into general OpendTect horizons.

Interpretation Workflow

After defining/selecting the right **depositional model**, you are ready to start interpreting sequence stratigraphic surfaces and the systems tracts. First, create an overlay of the HorizonCube on an inline/crossline/2D line, and position the scene in a way that you will notice main depositional changes (see below).



HorizonCube: HorizonCube

Sequence Model: Depositional Sequence IV

Top: 0

Bottom: 985

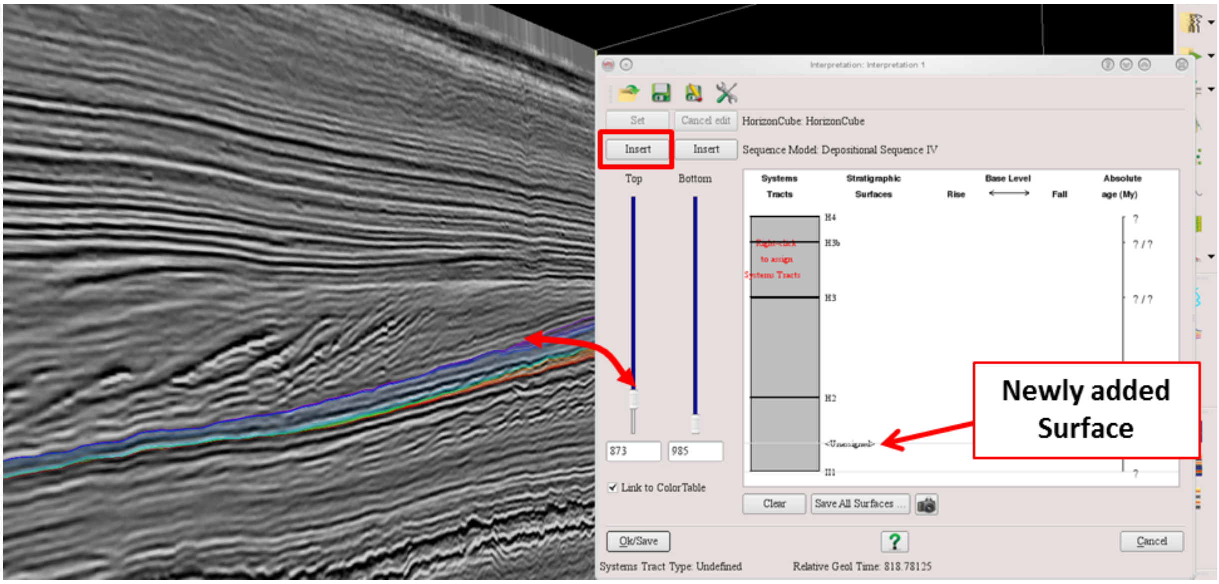
☑ Link to ColorTable

Systems Tract Type: Undefined

Relative Geol Time: 113.890625

The displayed seismic data is on an inline and the displayed HorizonCube is created for the entire survey range. The interpretation window is placed to the left so that the identified sequence stratigraphic surfaces can be inserted easily.

Check the *Link to ColorTable* option to optimize (stretch/squeeze) the colors for the HorizonCube relative to the slider position. Now, move the *Top* slider slowly down/up to observe the depositional changes. Start your interpretation from the bottom events of the HorizonCube. Move the *Top* slider up from the lowest position and position it where you observe a depositional shift or a sequence stratigraphic surface. At this point, you can add a surface to the current interpretation. Press *Insert* button to add a sequence stratigraphic surface in the display panel. This will add an surface in the Systems Tracts (grey) column (as shown below).



HorizonCube: HorizonCube

Sequence Model: Depositional Sequence IV

873

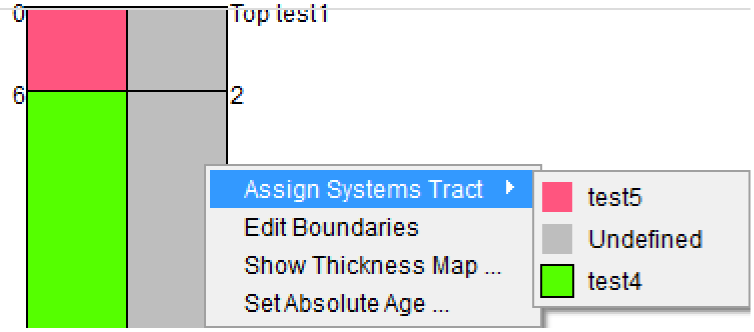
985

Systems Tract Type: Undefined

Relative Geol Time: 818.78125

The top slider position is adjusted to view at where the depositional chagnes are observed (e.g. in this case from transgression to normal regression).

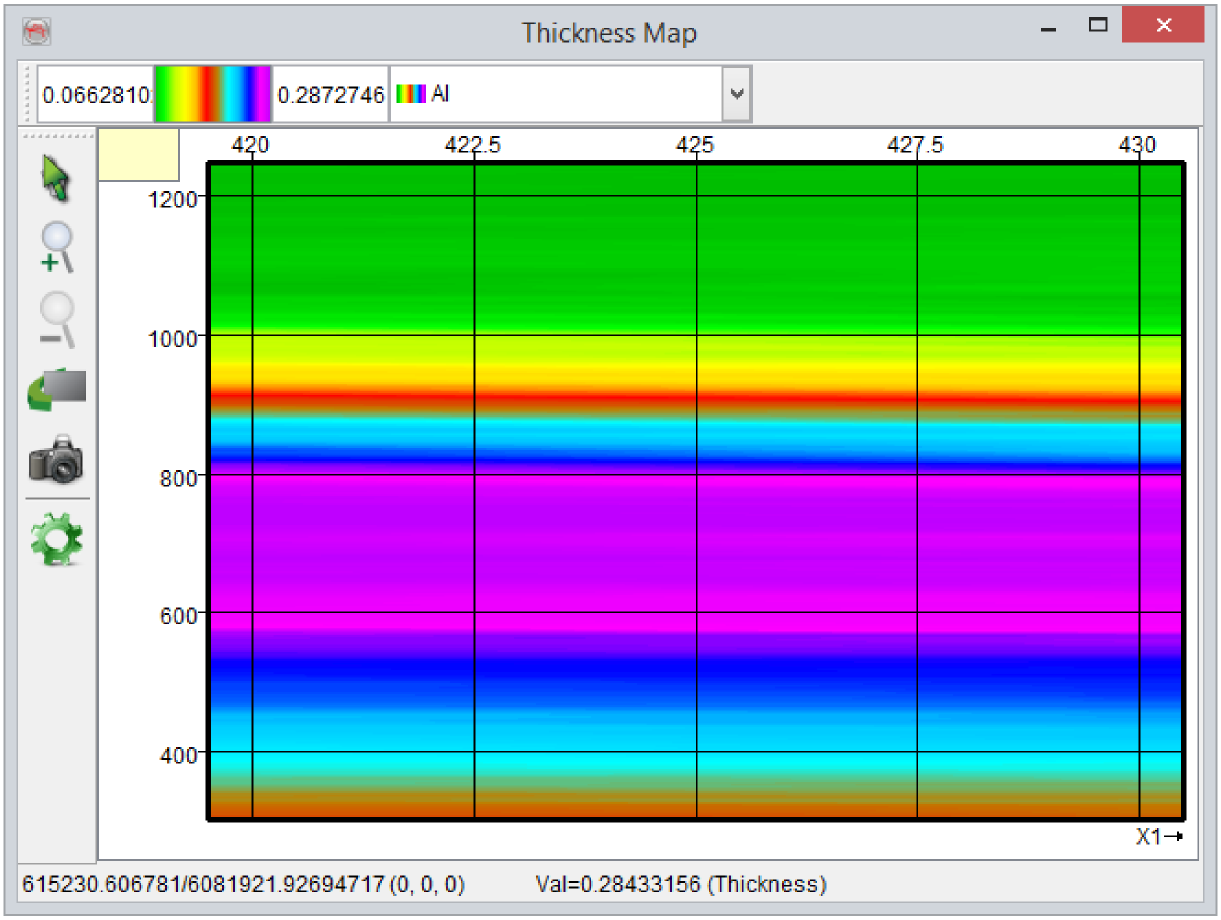
Now, the next step is to assign a name to the sequence stratigraphic surface. Right-click within the gray area i.e. just below the newly inserted stratigraphic surface. In the pop-up tree, you will see four options. From the 'Assign Systems Tract' option, select a sequence systems tract. Once you have selected this, the package will be filled with the corresponding color. Note also, that the relative change in base level is automatically plotted according to the depositional sequence model. Now you can continue interpreting the entire section by positioning the slider to next position(s) and following the same steps.



The pop-up menu is briefly explained below:

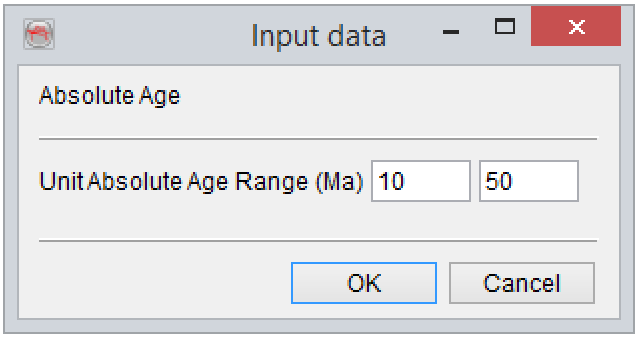
- *Assign Systems Tract:* Note that each depositional model has its own systems tracts-definition. Thus, if you have selected depositional model IV the corresponding systems tract that you can assign will be listed as above.

- *Edit Boundaries:* If you are not satisfied with the inserted stratigraphic surface for the systems tract, select the 'edit boundaries' option from the tree. On the right panel you can move the HorizonCube-slider to a new position. When done, press Set button or Cancel edit.
- *Show Thickness Map:* This will show the time-thickness variations.



615230.606781/6081921.92694717 (0, 0, 0) Val=0.28433156 (Thickness)

- *Set Absolute Age:* Set the geological absolute age (My) to any stratigraphic boundary. In the next pop-up window, give absolute age for top and bottom of the system tract.



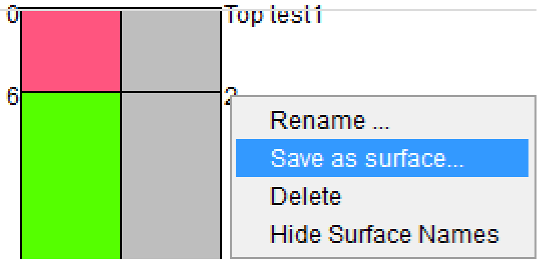
Absolute Age

Unit Absolute Age Range (Ma) 10 50

Save Interpretation

Once you complete your interpretation on the entire section, you are ready to save the interpretation. This can be done by pressing the save button 💾 . By default, the interpretation is saved with a default name (Interpretation 1). Any HorizonCube can have more than one Interpretations. Therefore, you can also save the interpretation with another name by pressing the Save As button 💾 .

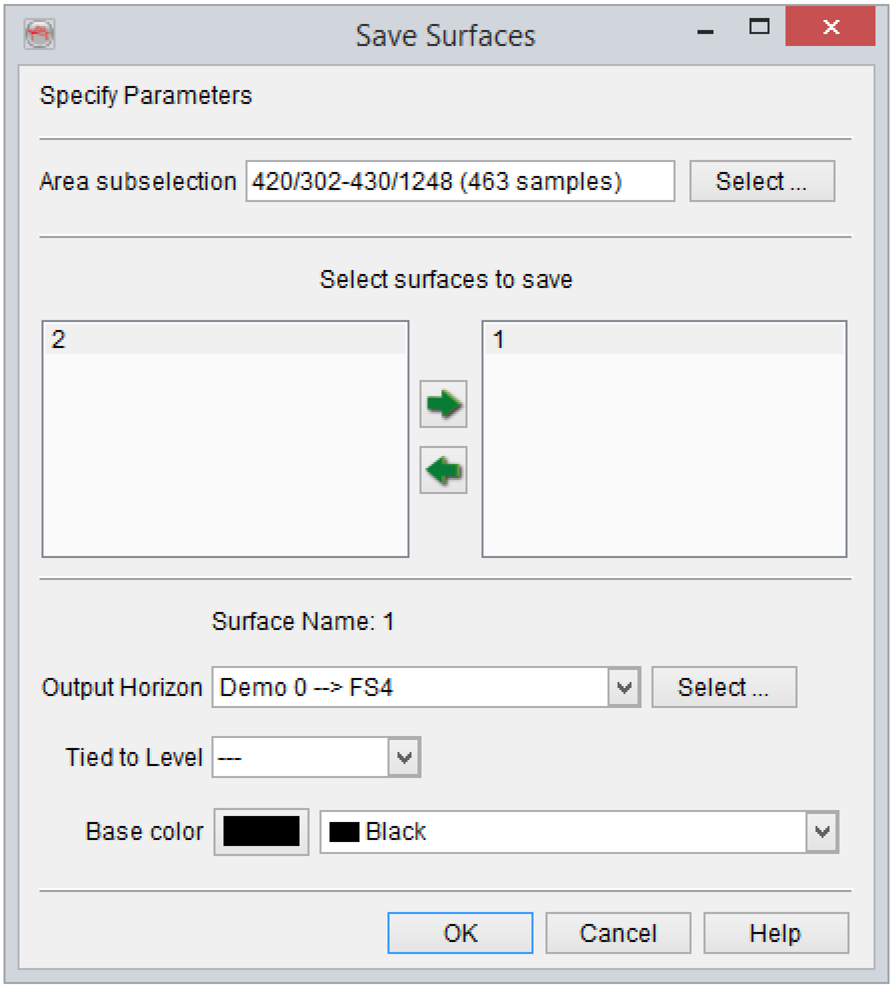
Safe Surfaces



A *stratigraphic surface* can be saved as an output surface. Either right click on the stratigraphic surface name, dots of base level curve or at numeric values of absolute age (if defined).

Once you are satisfied with the interpretation, press *save* or *save as* button to save the interpretation.

Select the area and write the name of output surface. This output surface can be displayed later from the Horizon tree.



Area subselection: 420/302-430/1248 (463 samples)

Surface Name: 1

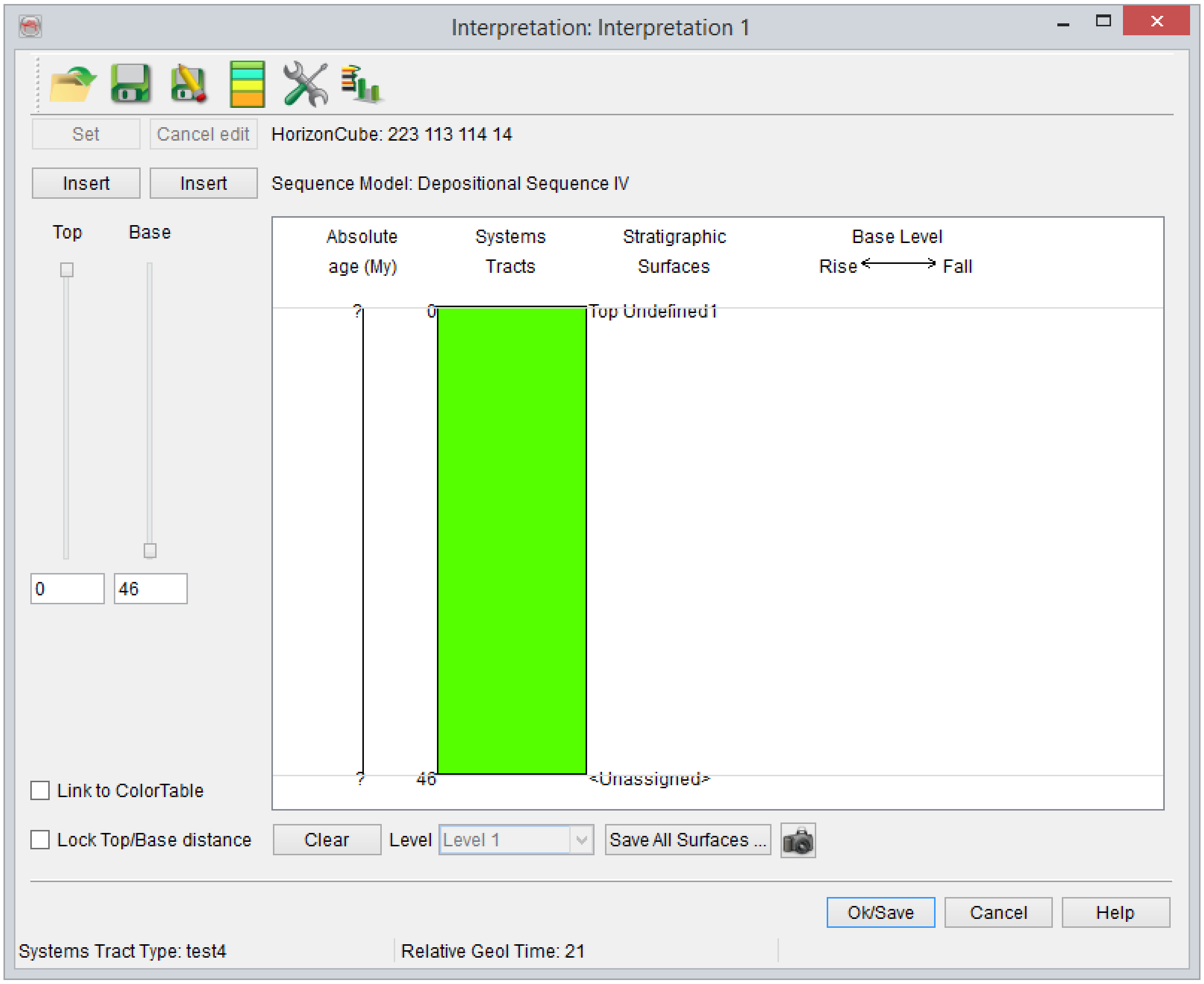
Output Horizon: Demo 0 --> FS4

Tied to Level: ---

Base color: Black

You can also save all surfaces by pressing *Save all* surfaces button.

2D Interpretation Window



Interpretation: Interpretation 1

HorizonCube: 223 113 114 14

Sequence Model: Depositional Sequence IV

Top: 0

Base: 46

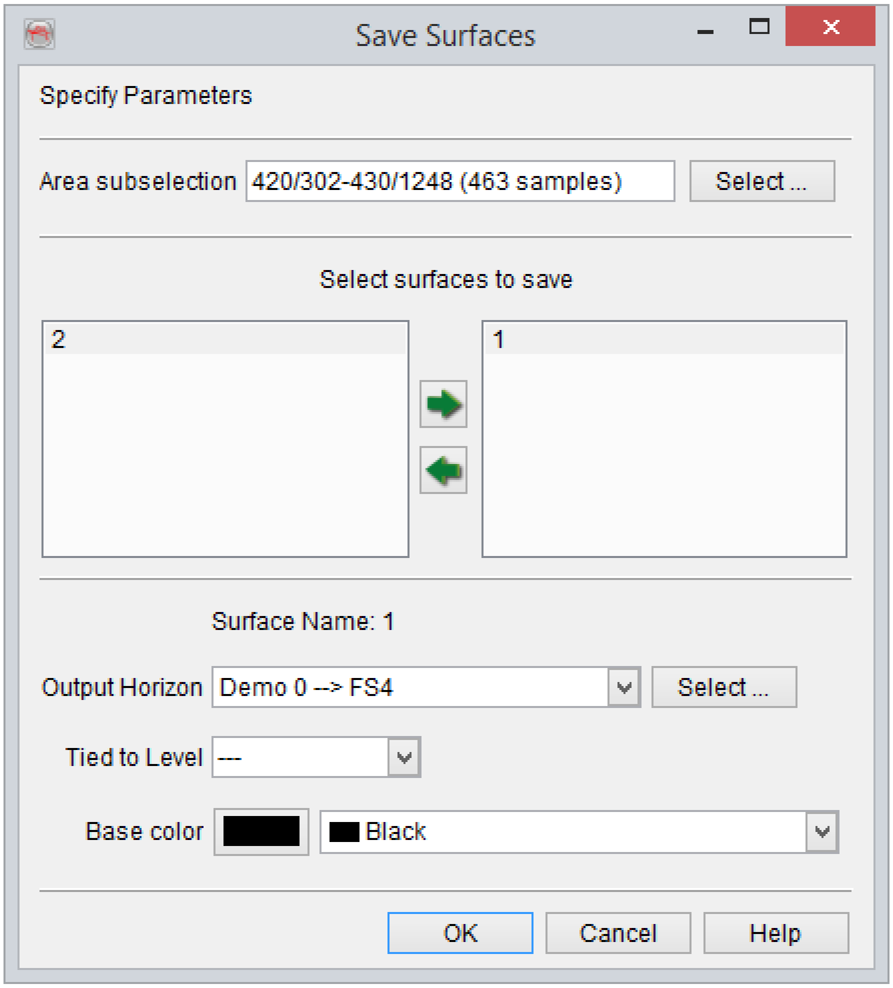
Systems Tract Type: test4

Relative Geol Time: 21

2D Interpretation windows is somewhat identical to the conventional 3D window. The only difference is that you need to select the relevant 2D line in this window on which the interpretations would be made. Note that the slider (Top / Bottom) works individually for a selected line only.

It is also supported to copy one line's interpretation to another line using the *Copy* button. If the interpretation has already been made on an intersecting line, you could copy that interpretation to another intersecting line. In this manner, you would keep the same number of systems tracts on the new intersection and the boundaries would be placed automatically. This will also avoid the mis-ties at the intersections.

Save *All Surfaces* button will allow you to save the interpreted sequence stratigraphic surfaces as 2D horizons. The surfaces that you want to save are added on to the right list box using the arrows. The corresponding lines 2D lines on which the interpretation was made for the selected lines must also be selected by the user. The output name / colour should be mentioned prior to pressing the OK button.



Area subselection: 420/302-430/1248 (463 samples)

Surface Name: 1

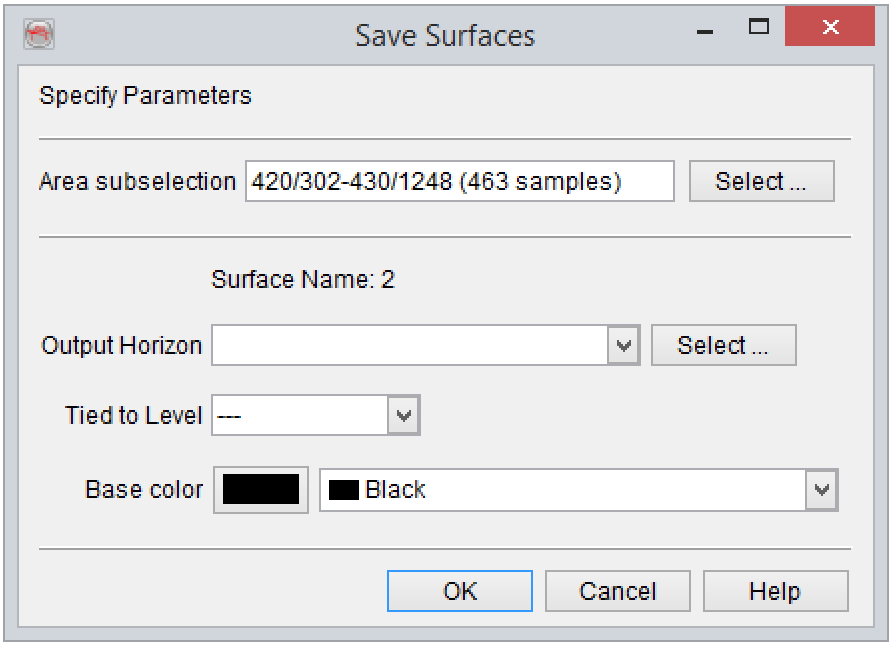
Output Horizon: Demo 0 --> FS4

Tied to Level: ---

Base color: Black

Note that you could also save/rename/delete each surface individually by right clicking on the surface name in the graphical area. It will launch the pop-up menu with the following options:

- **Rename:** It will rename the surface either at a user-defined manner or Auto-set. In the user-defined manner, the provide name would be used. If the name is set to Auto-set, it will automatically number the default surfaces according to the selected model.
- **Save as surfaces:** It is used to save the surface as a 2D horizon. In the pop-up window, the corresponding lines are listed on which the surface has been interpreted. The line selection must be made by the user in the *Select 2D lines* list. The output name / colour should also be provided.



Area subselection: 420/302-430/1248 (463 samples)

Surface Name: 2

Output Horizon:

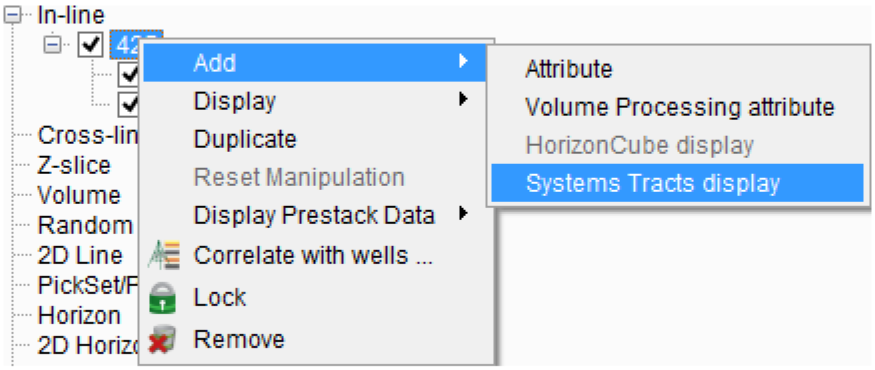
Tied to Level: ---

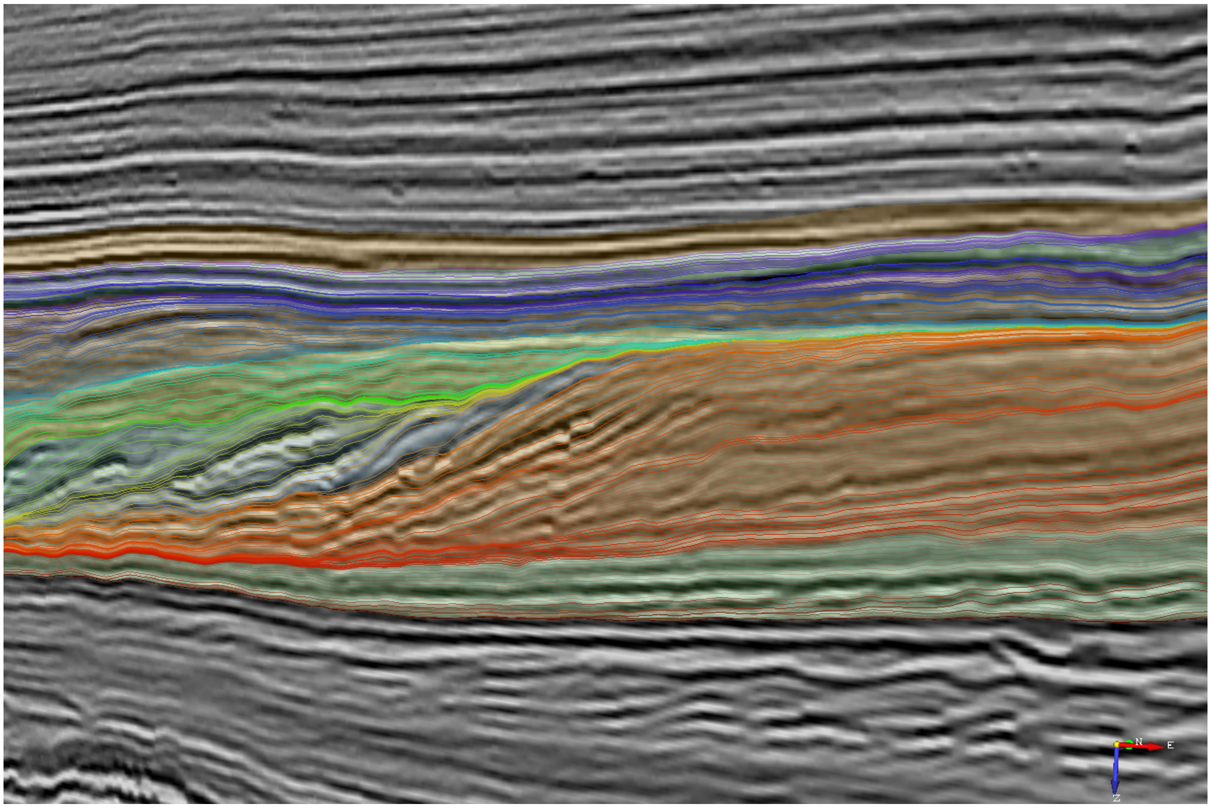
Base color: Black

- **Delete:** It will remove the selected boundary and would extend the lower interpreted systems tract up.
- **Hide surface names:** This is used to hide the surface label.

Display System Tracts

The interpreted systems tracts in the interpretation window can be displayed on the inline/crossline/2D line using a pop-up menu. Right-click on the elmenent on which you want to display the systems tracts and select the *Add Systems Tracts display option*.

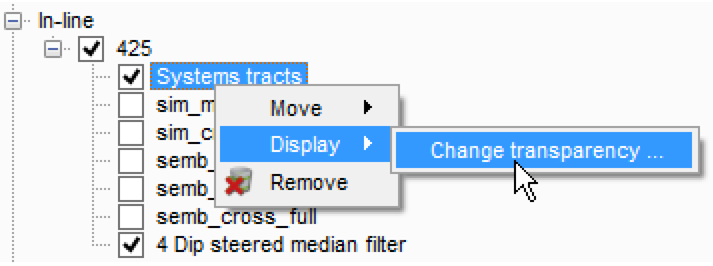


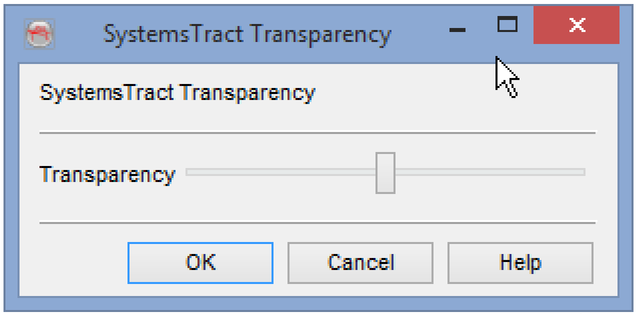


An example overlay of the interpreted systems tracts (semi-transparent) on an inline from a 3D HorizonCube.

Change Transparency

To change the transparency of the systems tracts overlay, right click on the systems tracts element from the tree (as shown below) and select the transparency-option from the pop-up menu. From the pop-up window (SystemTracts transparency), the transparency can be set by moving the slider towards right.

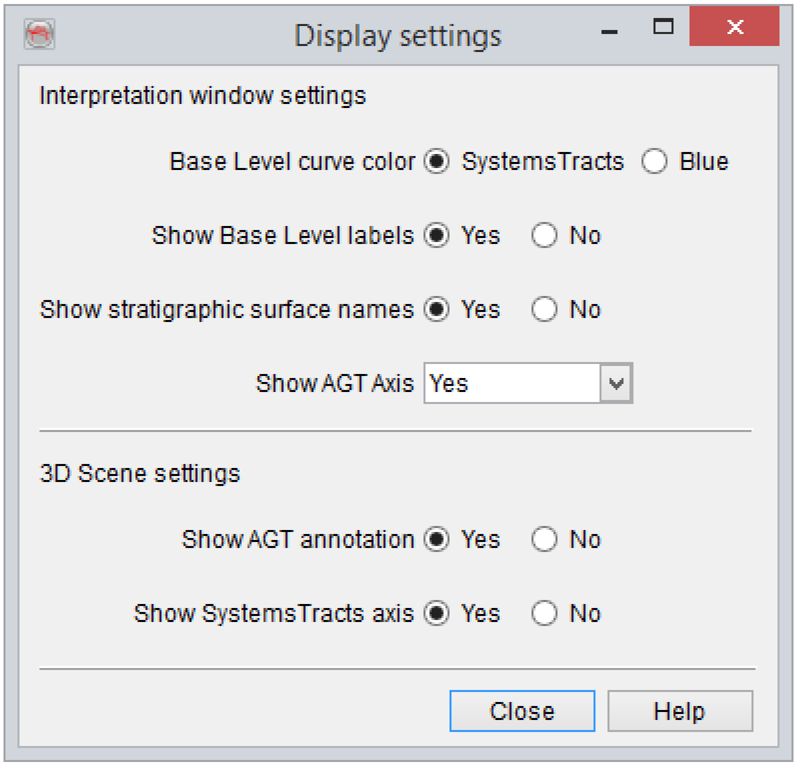




Moreover, the *Move* option in the pop-up menu for the System Tracts changes the order of the system tracts within the element. However, it doesn't change the display layout in the OpendTect scene. Select *Remove* item to remove the display of the System Tracts from the scene.

SSIS Display Settings

The display settings of the interpretation window are changed using this dialog.



Display settings

Interpretation window settings

Base Level curve color ● SystemsTracts ○ Blue

Show Base Level labels ● Yes ○ No

Show stratigraphic surface names ● Yes ○ No

Show AGT Axis Yes

3D Scene settings

Show AGT annotation ● Yes ○ No

Show SystemsTracts axis ● Yes ○ No

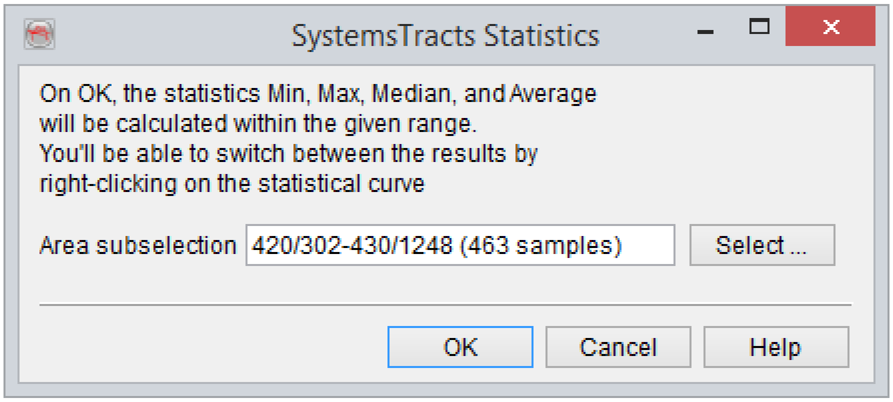
- *Base Level curve color* is by default set to the color of a systems tract. If a systems tract is blue, the color of the base level curve segment should be blue also. If the default is changed to Blue, the entire curve will appear as blue curve.
- *Show Base Level labels* allows to annotate the labels such as End of Regression, Onset of Base level fall, End ot Transgression etc.
- *Show stratigraphic surfaces names* allows to annotate the name of sequence stratigraphic surfaces as they appear in the selected **Depositional Model**.

- *Show AGT annotation* is an option that interacts with the 3D Scene and survey box. If this option is set to No, the AGT annotation will not appear in the scene.
- *Show Systems Tract axis* also interacts with the 3D Scene. If this option is set to No, the systems tracts' color codes will not appear in the scene.

Create Statistical Curves

Statistical curves are the thickness based curves that are defined based on various statistics (Min, Max, Median, average) for each systems tract . These curves are valuable to graphically illustrate the thickness variations adjacent to base level curve. Note that the base level curve is mostly a conceptual illustration that is not derived from the data. It is this thickness curve, which is an add-on to the base level curve.

The statistical curves are computed by launching the *Systems Tracts Statistics Dialog* using the icon .



On OK, the statistics Min, Max, Median, and Average
will be calculated within the given range.
You'll be able to switch between the results by
right-clicking on the statistical curve

Area subselection: 420/302-430/1248 (463 samples)

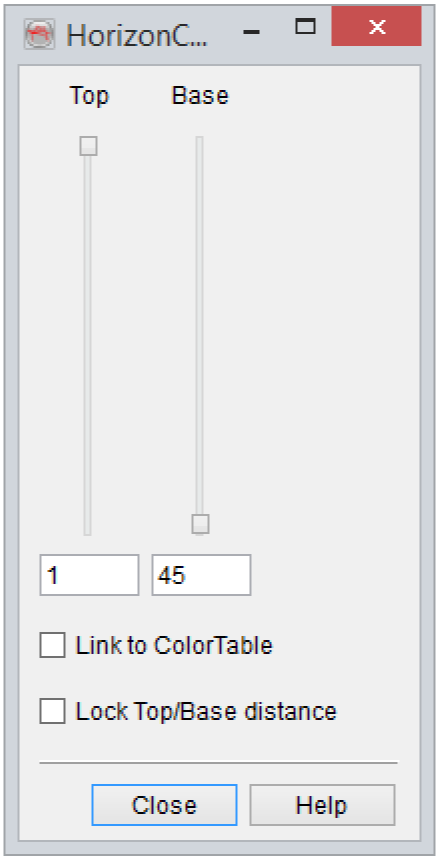
It is recommended to use more than one trace location, or an area defining the depocenters.

After proceeding, the curve is displayed on the right-most side of the base level curve. The default curve is average, which can be changed by right clicking on the newly displayed curve and selecting various statistics only the fly.

Note: At present, these curves are not stored in the interpretation window and are temporarily displayed while the dialog is open.

HorizonCube Sider

The HorizonCube slider is launched from the **SSIS toolbar**, which is used to visualize the displayed HorizonCube and to understand the depositional patterns/geometries. The slider in the SSIS interpretation is of vital importance as it is being used in the SSIS interpretation module.



Top: 1
Base: 45

☐ Link to ColorTable

☐ Lock Top/Base distance

Link to ColorTable: If this is checked, the active HorizonCube colour bar will be dynamically stretching/squeezing relative to the slider position.

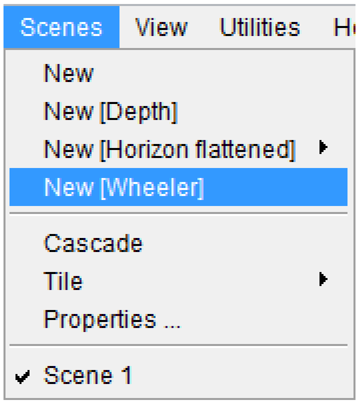
Lock Top/Bottom distance: It is used to display fixed number of events. For instance, if you want to visualize one event at a time, you may set the slider gap of zero i.e. the top and bottom slider positions are reflecting the same event. Thereafter check this option and move either the top or bottom slider.

Wheeler Transform, Wheeler Scene

See following chapters below.

Wheeler Scene

The Wheeler Scene in OpendTect is another scene, which is added from the SSIS menu. It may be noted that the Wheeler Scene is a transformation (flatenning) of HorizonCube into relative gelogical time (RGT). Therefore, before adding a Wheeler Scene, the HorizonCube should be selected.

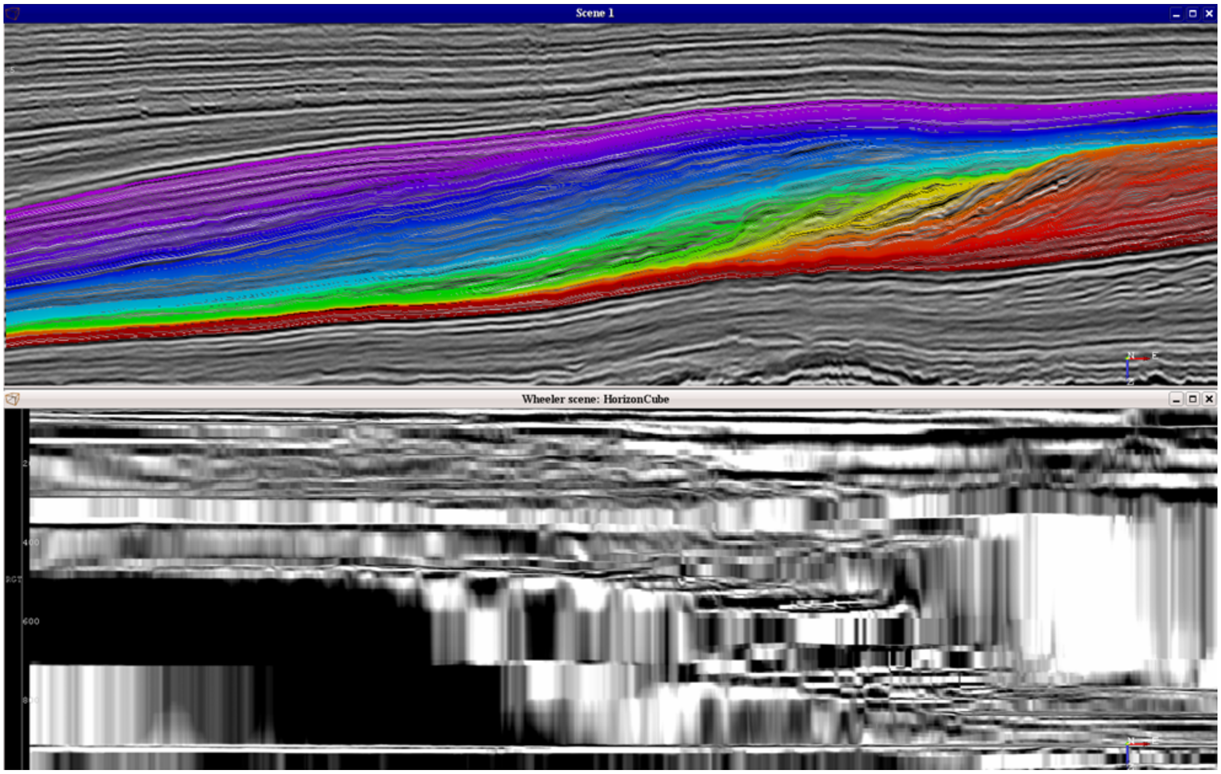


In the Wheeler scene, flattened seismic data can be displayed by adding elements (inline, crossline, and timeslice) in the tree. By displaying flattened seismic data, one can assess the quality of the selected HorizonCube. Making a good HorizonCube is an iterative process. It will probably take several runs before the result is satisfactory. In order to improve HorizonCube the user can try to:

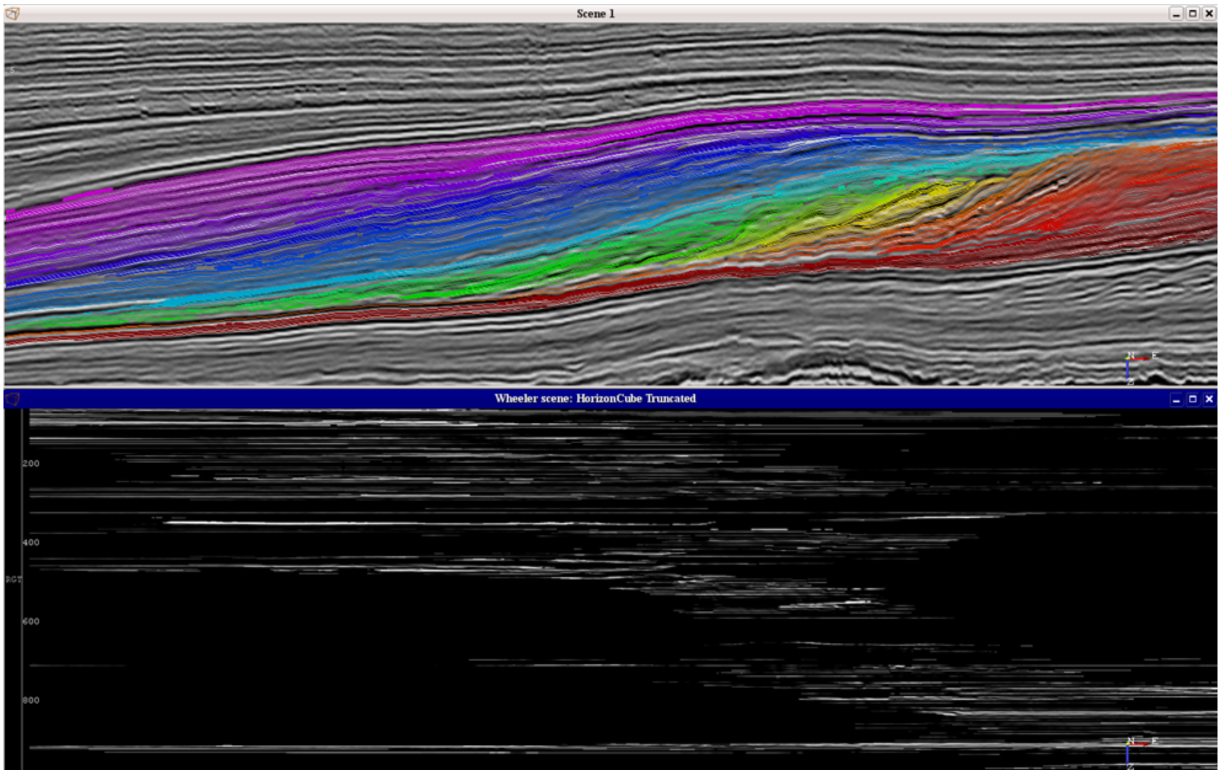
- *map additional horizons (sequence boundary)*
- *fill holes in mapped horizons*
- *make horizons continuous and watertight*
- *filter the SteeringCube (data-driven approach)*

Note: In the Wheeler scene, stored data is transformed on-the-fly to the flattened domain. All attributes and neural network outputs are calculated in the normal domain first and then transformed to the Wheeler domain. Because of this transformation, quickly scanning through your Wheeler is only possible after **creating a Wheeler cube**. On-the-fly calculations are not supported for the volume viewer, which only accepts input from pre-calculated Wheeler cubes.

Below, the two types of Wheeler transformation are displayed. One with **Continuous HorizonCube** and the other is the same HorizonCube with a **density filter**. Note that for Wheeler Transformation, the closely spaced events in a continous HorizonCube needs to be filtered out so that the hiatus, and stratal terminations become prominent and one can interpret base-level changes.

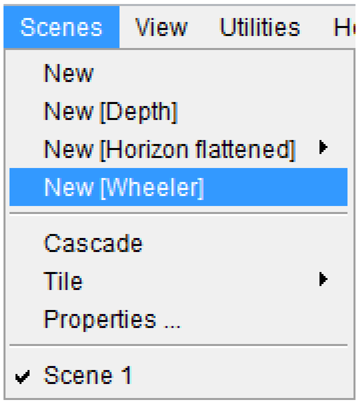


Top scene is a display of a seismic data and the HorizonCube with continuous events. Bottom display is the Wheeler Transformed Scene of the HorizonCube. The same seismic data is displayed in the wheeler domain.



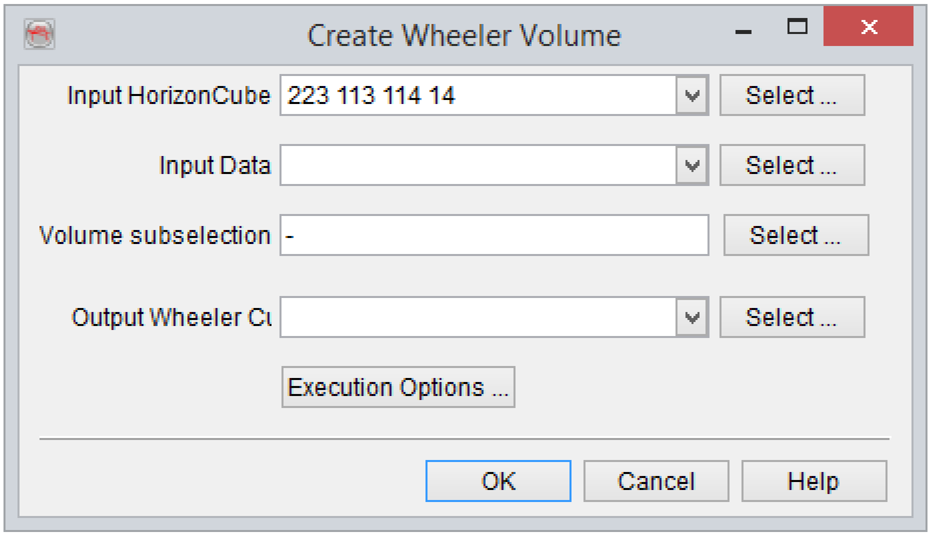
*Top scene is a display of a seismic data and the continous HorizonCube with a **density filter** applied. Bottom display is the Wheeler Transformed Scene of the displayed HorizonCube. The same seismic data is displayed in the wheeler domain.*

Create Wheeler Output (2D, 3D)



A WheelerCube is a seismic volume in the Wheeler domain. When a WheelerCube is displayed on an element in the Wheeler scene, no transformation is needed as the Wheeler volume is created on-the-fly.

Note: Before creating a WheelerCube a HorizonCube must be selected.


Create Wheeler Volume

Input HorizonCube 223 113 114 14

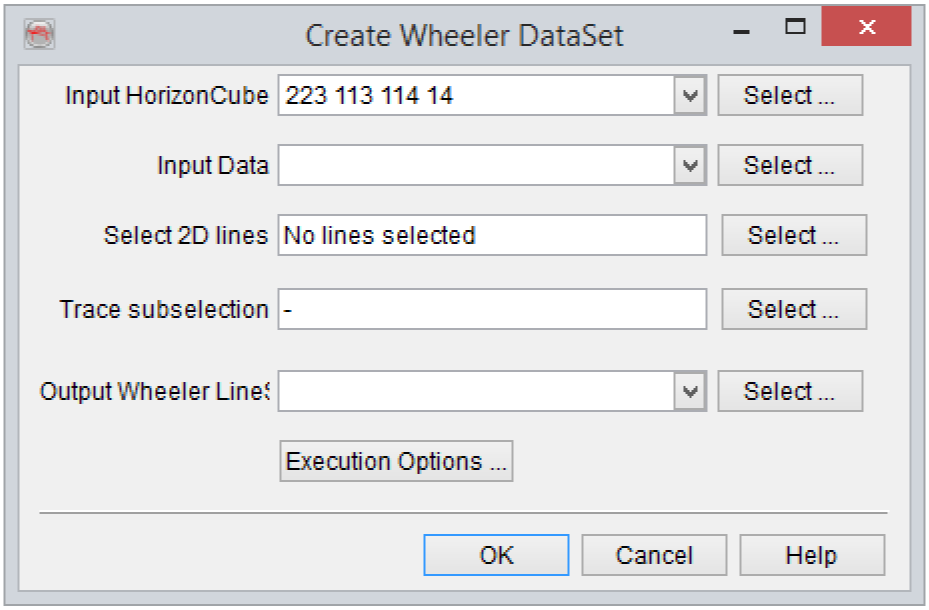
Input Data

Volume subselection -

Output Wheeler Cu

Create Wheeler output volume for a 3D volume

Any stored 3D-data (seismic, attribute, neural network output) can be converted into the Wheeler domain. Select the stored volume you want to transform and specify a name for the WheelerCube. Moreover, the wheeler volume can be reduced by specifying the **area-sub-selection**. The Wheeler Cube is created in batch-mode (see corresponding section in the OpendTect User Documentation). A stored WheelerCube can be displayed by right-clicking an element in the tree, then click select *attribute* > *Wheeler Cubes*.


Create Wheeler DataSet

Input HorizonCube 223 113 114 14

Input Data

Select 2D lines No lines selected

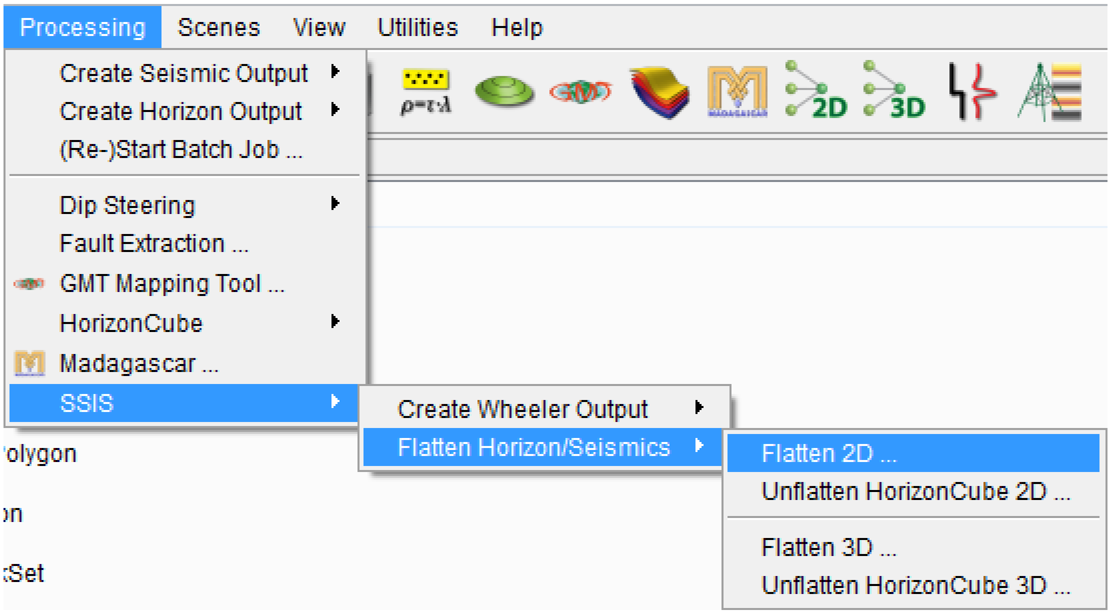
Trace subselection -

Output Wheeler Line

Create Wheeler output volume for a 2D Lineset/Line

For a 2D type of Wheeler transformation, 2D HorizonCube is required. Select the 2D HorizonCube in the input. This will be used as a transformation matrix for the Wheeler transformtion. Next, provide the input 2D data e.g. seismic/AI data. Optionally, you may create it for one line only and also restrict the output to a limited number of traces (*Traces sub-selection*). After this, the Wheeler output is stored to the written line-set and the attribute (format: Lineset_Name|Output_Wheeler_Attribute_Name).

Flatten Horizon, Seismics



The flattening and un-flattening workflow can be used for SSIS interpretations specifically in structurally distorted regions. The workflow starts with flattening a volume/line using a reference horizon. The selected seismic data and horizons are flattened into a new survey. In this flattened survey, the SteeringCube can be prepared, followed by processing of the HorizonCube. After the HorizonCube in the flattened survey is satisfactory, it can be un-flattened in the parent (un-flattened) survey for further SSIS interpretation.

Flatten

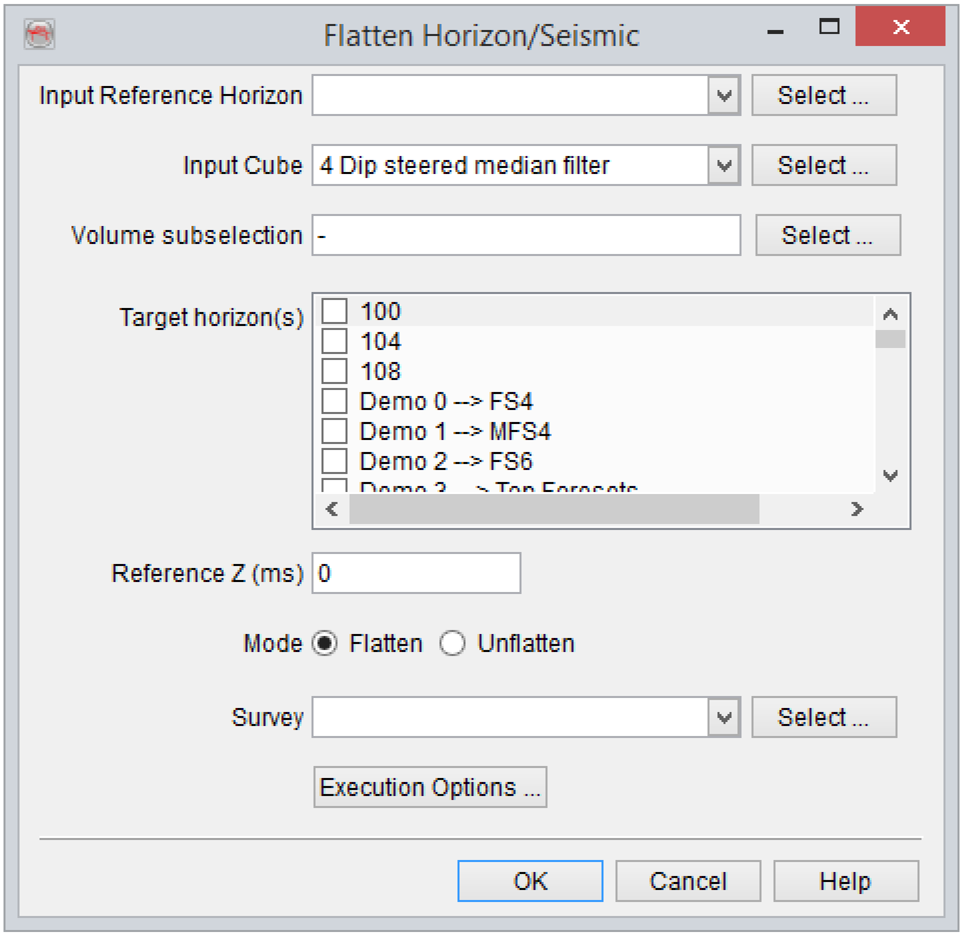
3D Survey

To flatten a 3D survey using a reference horizon, this window is used. It can be launched from the SSIS menu. The input *Reference Horizon* is selected from the active survey, which would be used as a datum for the flattened survey. This datum is a *Reference Z(ms)* value. The corresponding seismic volume is selected in the *Input Cube* field. This volume would be used to prepare SteeringCube (for HorizonCube) in a flattened survey. Optionally, volume sub-selection can also be made to do work in a sub-volume.

The *Target Horizon(s)* are selected as these horizons are used as input in the HorizonCube, for building the framework. In the list one or more horizons can be transformed in the selected output i.e. a flattened survey (give a name in the *Select Survey* field).

Mode: Flattening / un-flattening mode. If flattening is set here, the selected data from the active survey will be flattened in a new (flattened) survey. The un-flattening mode is used to un-flattened a flattened (active) survey.

After providing the right selection(s), the user may proceed further by clicking on the *Proceed* button. This will start a batch program. Optionally, the user may run the batch program on a remote host by checking the *Show Option(s)* box before pressing the *Proceed* button.



Input Reference Horizon

Input Cube: 4 Dip steered median filter

Volume subselection: -

Target horizon(s):
- ☐ 100
- ☐ 104
- ☐ 108
- ☐ Demo 0 --> FS4
- ☐ Demo 1 --> MFS4
- ☐ Demo 2 --> FS6
- ☐ Demo 3 --> Top Foresets

Reference Z (ms): 0

Mode ● Flatten ○ Unflatten

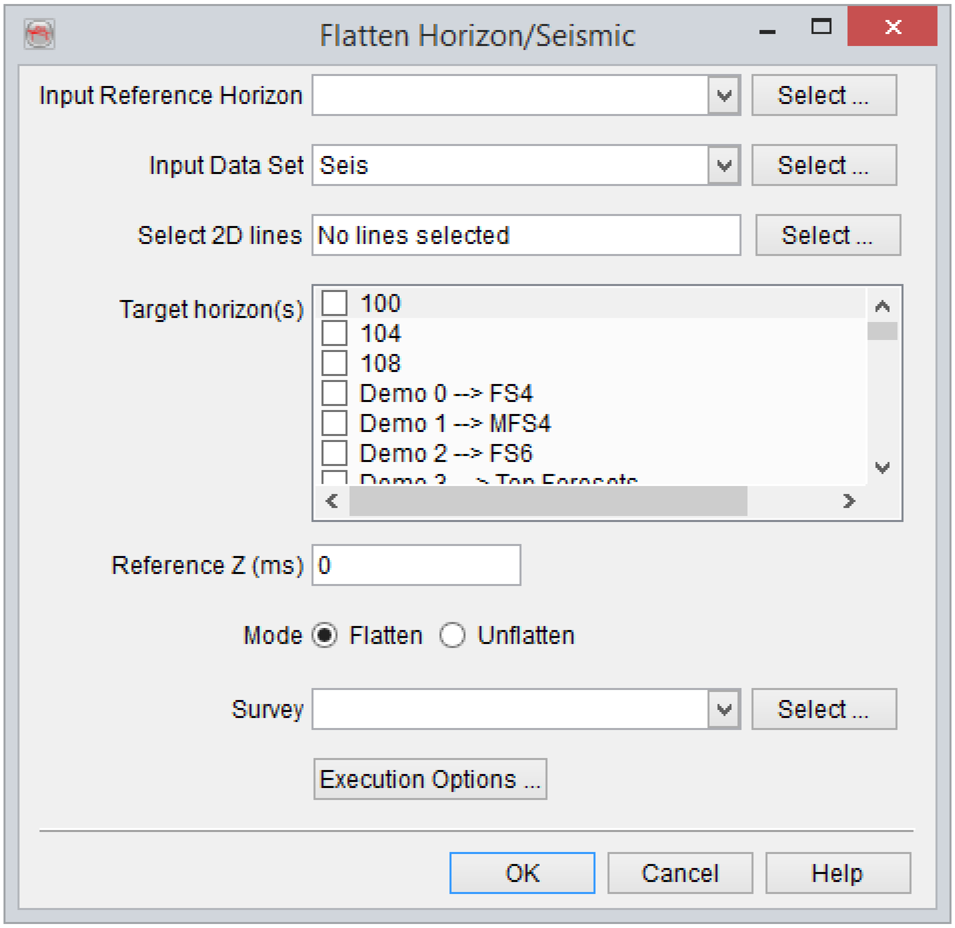
Survey

Flattening a 3D seismic data and the horizons to prepare a HorizonCube in a flattened survey.

2D Survey

The same workflow can be applied for a 2D survey. If the window is launched in an active 2D survey, the 2D lines and the corresponding 2D horizons can be flattened using a reference horizon. The reference horizon defines a flattening datum with a given *Reference Z value*. This is selected from the *Input Reference Horizon* field. The 2D lines available in a *Lineset* are selected for the *Lineset/Line Name* option by pressing the select button.

Together with the seismic data, the corresponding 2D horizons from the active survey may also require flattening. These horizons are selected from the *Target Horizon(s)* field. Finally, select the flattening mode, and give an appropriate output survey name and press the *Proceed* button.



Input Reference Horizon

Input Data Set Seis

Select 2D lines No lines selected

Target horizon(s)
☐ 100
☐ 104
☐ 108
☐ Demo 0 --> FS4
☐ Demo 1 --> MFS4
☐ Demo 2 --> FS6

Reference Z (ms) 0

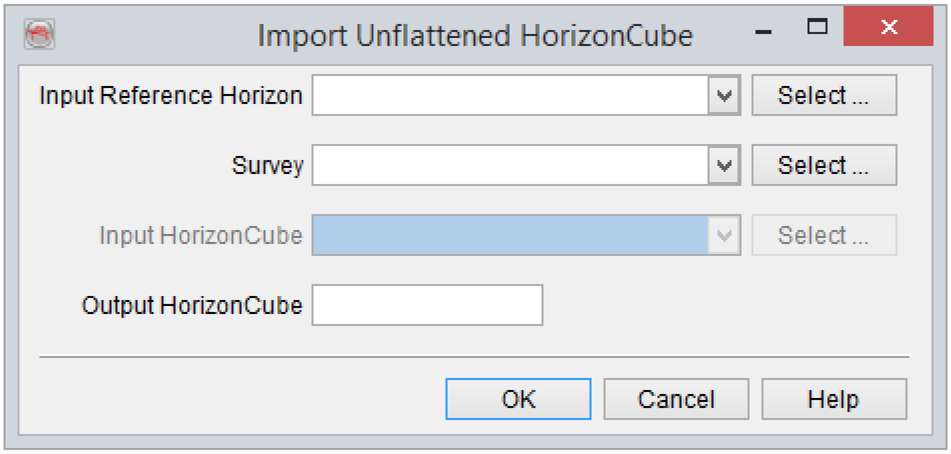
Mode ● Flatten ○ Unflatten

Survey

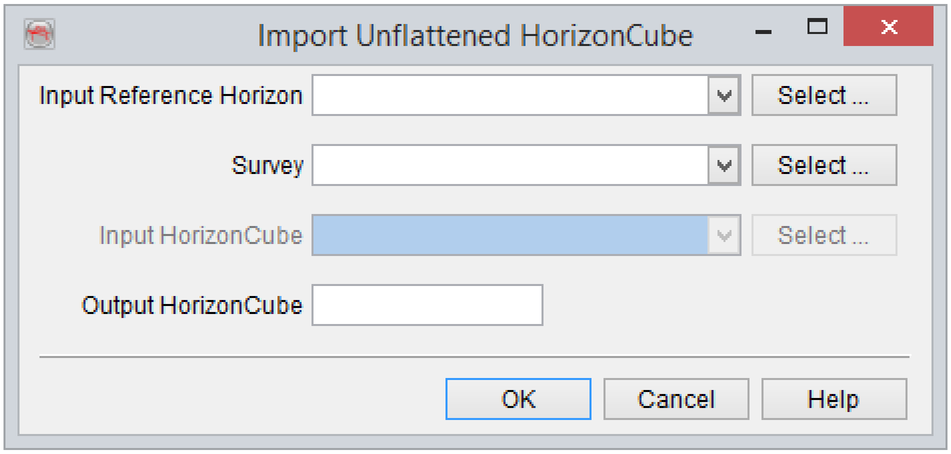
Flattening 2D seismic data and the horizons for preparing a HorizonCube in a flattened survey.

Unflatten HorizonCube

The HorizonCube calculated in a flattened survey can be transformed back to its parent (un-flattened) survey. A input reference-horizon for the 2D/3D case needs to be provided. The horizon should be the same as the one selected to create a flattened survey. The input flattened 2D/3D survey, in which the HorizonCube is prepared, is selected from the *Select Survey* field. The input flattened HorizonCube available in the selected survey is selected in the *Input HorizonCube* field. Finally the output name for the HorizonCube is given. If the 'OK' button is pressed, the batch program will start to transform the flattened HorizonCube (2D/3D) into a un-flattened HorizonCube in the active survey.



Un-flatten a HorizonCube from a 3D flattened survey.



Un-flatten a HorizonCube from a 2D flattened survey.

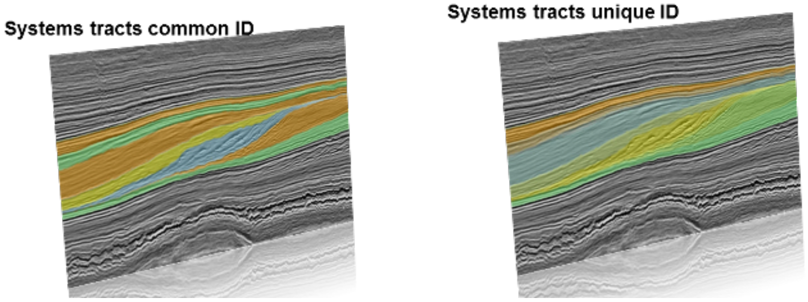
System Tracts Attributes

A systems tract attribute defines a volume that contain the sequence stratigraphic interpretation. Once an interpreter has completed 3D SSIS interpretation, the interpreted systems tracts can be stored as an attribute volume. This is normally done using this attribute. Once this attribute is defined using the Attribute set window, one may either store it as an output volume or can visualize it in 3D. This attribute can also be used for other purposes e.g. reserve estimation, prediction of seismic facies bounded by the systems tracts etc.

This attribute returns three outputs:

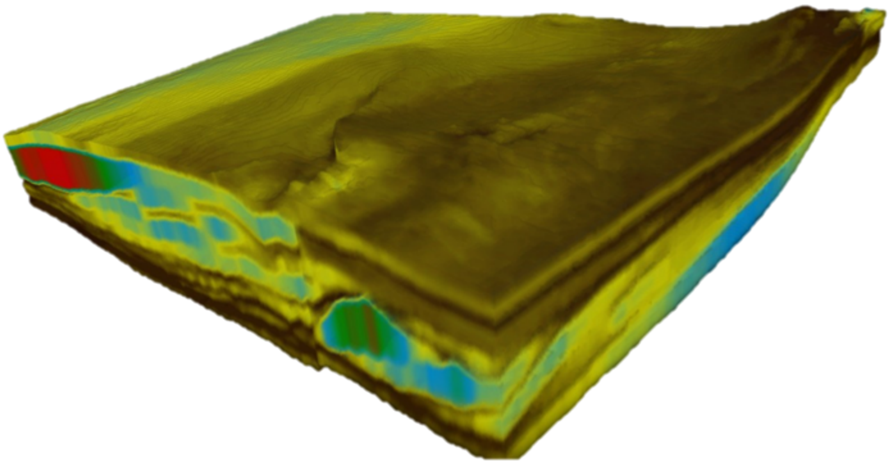
1. **Common ID:** It is a common ID for a systems tract within a volume. For instance, if there are four HSTs interpreted, all four HSTs will have same common ID (i.e. 0). Such an attribute is chiefly useful for 3D visualizations of a systems tract with a common colour. However, for seismic prediction, it will have little usage.

2. **Unique ID:** It returns a unique number to each systems tract. Such an attribute is valuable tool to do seismic prediction using **Neural Networks**.



An example of systems tracts sub-attribute.

3. **Isopach:** It is an isopach between the top and bottom of a systems tract. It can be used to interpret the preserved thickness within a systems tract. It can also be calibrated with absolute geologic time to interpret the actual sedimentation rate within a systems tract.



Manual SSIS

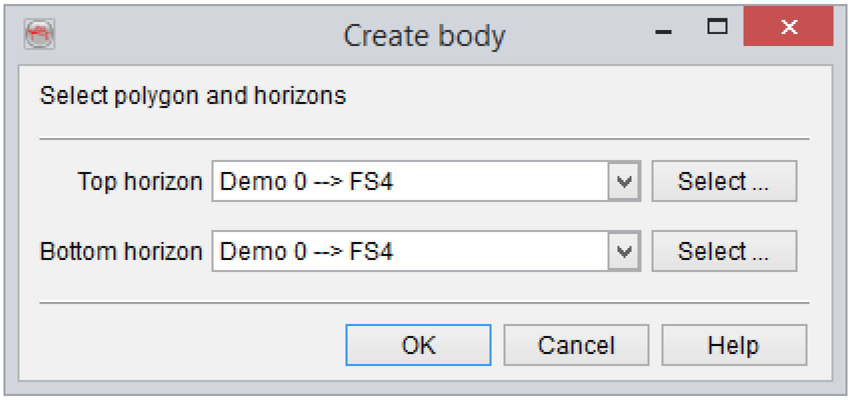
The manual SSIS workflow is designed for manual sequence stratigraphic interpretation. Currently, a body can be extracted from a drawn polygon. A polygon can outline a geobody e.g. channel, valley, or other geomorphological feature. Extraction of such bodies is important for in-depth stratigraphic interpretation. By doing this, the volume is limited through drawing a polygon where after different attributes can be displayed on-the-fly, which in turn can be interpreted further. The workflow is as follow:

1- Create a new polygon that defines a geological shape.

2- Right-click on the new polygon name and select Create body

3- In the pop-up window select the top and bottom horizons to define a vertical limit. Press OK to continue

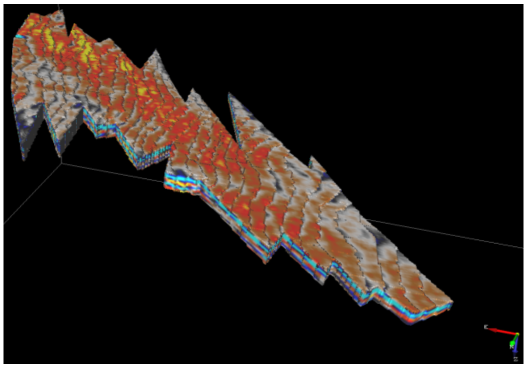
The setup will create and display the output body. An example body of sand-waves from offshore seismic data is shown below.



Select polygon and horizons

Top horizon: Demo 0 --> FS4

Bottom horizon: Demo 0 --> FS4



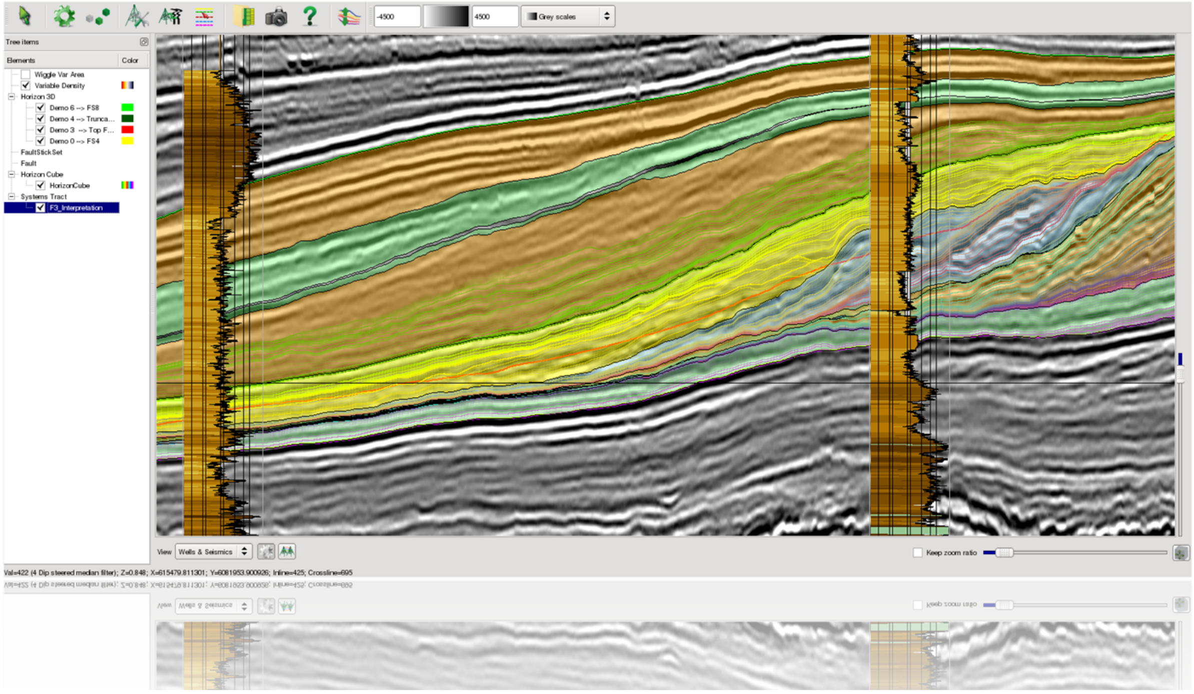
Well Correlation Panel

Table of Contents

- Introduction
- WCP Main Window
- Correlation Displays and Settings
- Pick Markers and Correlate

Introduction

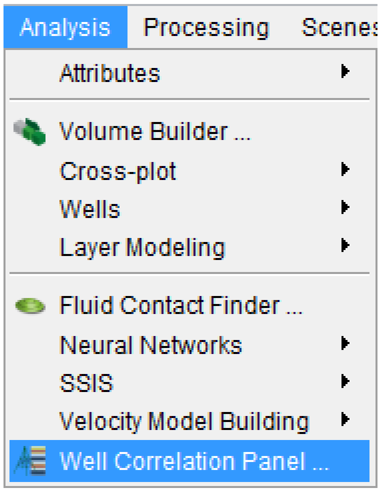
The Well Correlation Panel (WCP) is used to create multi-well correlation(s) sections using the interactive overlays of the HorizonCube, seismic data and SSIS Interpretation (systems tracts). The aim of using WCP in OpendTect is to build a sequence stratigraphic framework by connecting wells using the HorizonCube. This is done easily by picking new stratigraphic markers while visualizing densified Horizons. Additional options are available, such as editing the well marker database by moving the marker vertical position using the mouse. Moreover, the interpreter can also create a transect (2D Line) from a given 3D seismic data by connecting several wells. The following section explains these options in detail.



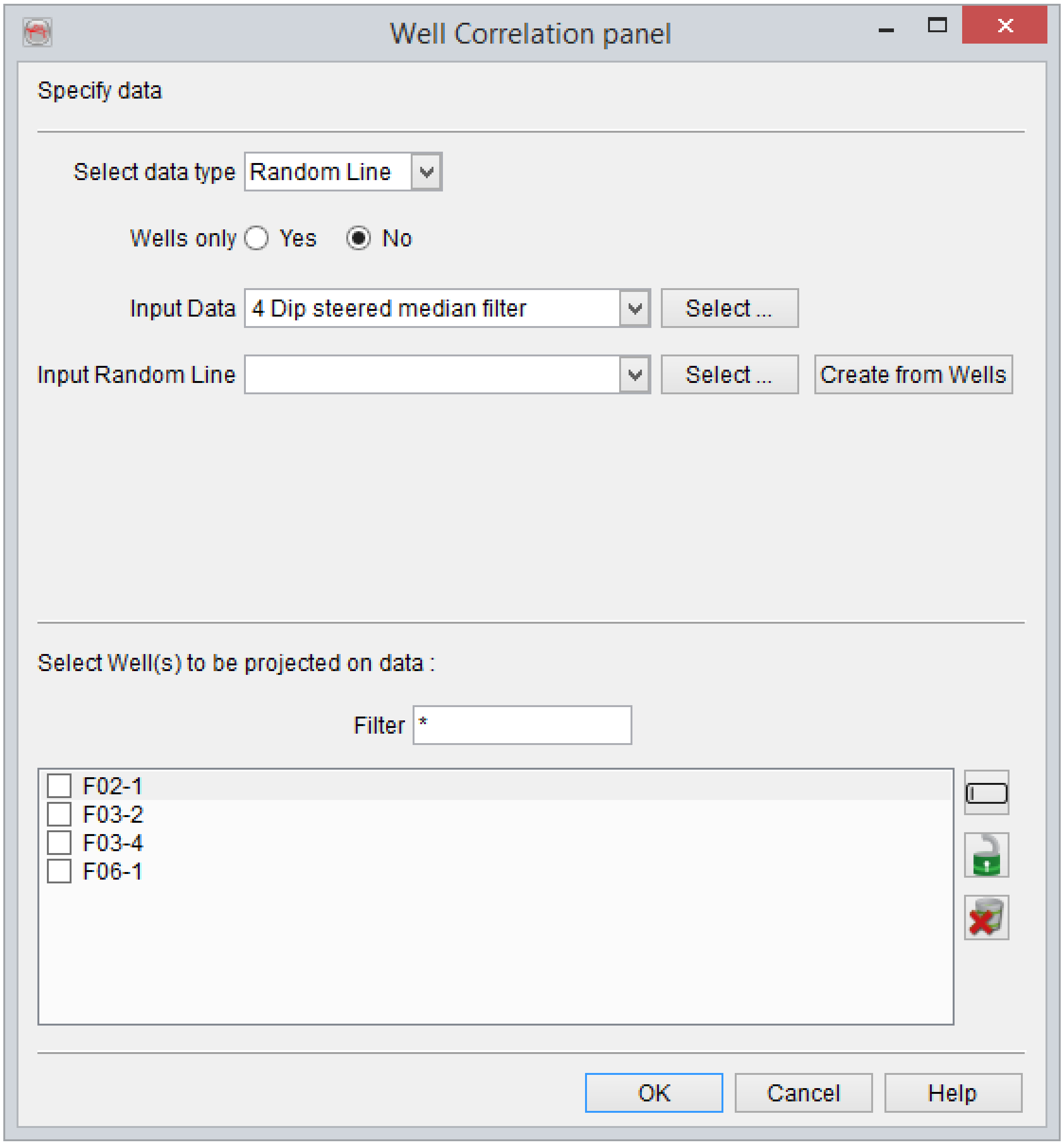
```
Val=422 (4 Dip steered median filter); Z=0.848; X=615479.811301; Y=6081953.900926; Inline=425; Crossline=695
```

WCP Main Window

The 'Well Correlation Panel' is launched from the Analysis menu:



The application can also be launched in the OpendTect main menu (Analysis > Well Correlation Panel) or by clicking the 'WCP' icon in the main toolbar. In the following window, input data-type is needed, e.g. Inline/Crossline or a 2D line.



Well Correlation panel

Specify data

Select data type: Random Line

Wells only ○ Yes ● No

Input Data: 4 Dip steered median filter

Input Random Line:

Select Well(s) to be projected on data :

Filter: *

☐ F02-1
☐ F03-2
☐ F03-4
☐ F06-1

Select Data Type:

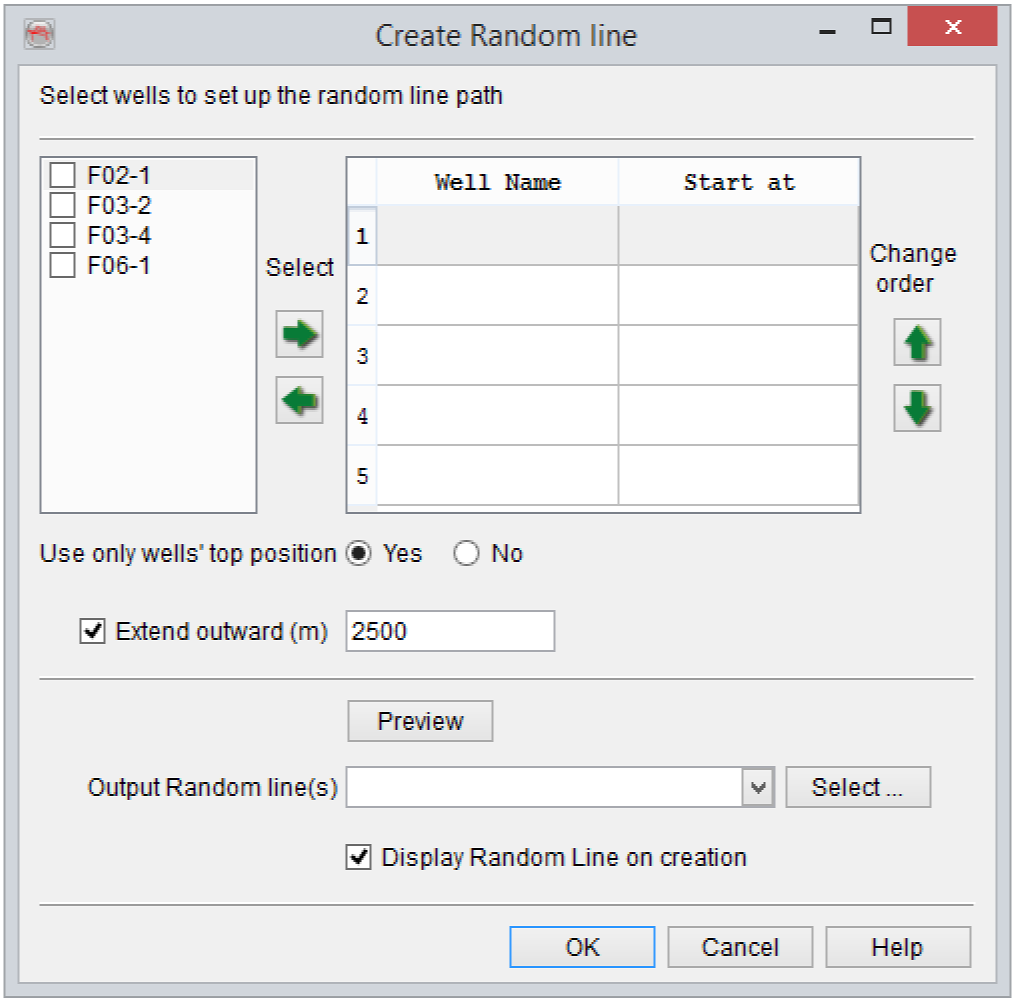
- 2D Line: Correlates a 2D line close to selected wells. A sub-selection can be made for the line-set and the corresponding line. Furthermore, a trace sub-selection can also be made if the line is too big and the wells lie within the defined trace range.

- Inline/Crossline Used to select a 3D seismic data (inline/cross-line) close to the selected well.

Selection of key wells:

- The selected wells are projected onto selected data (2D/3D line). One or several wells can be selected in the well list. To select all, please use left mouse button (select and drag down the list). Optionally, use CTRL key plus left mouse click(s).

Create a 2D line (from 3D) between wells



Select wells to set up the random line path

☐ F02-1
☐ F03-2
☐ F03-4
☐ F06-1

	Well Name	Start at
1		
2		
3		
4		
5		

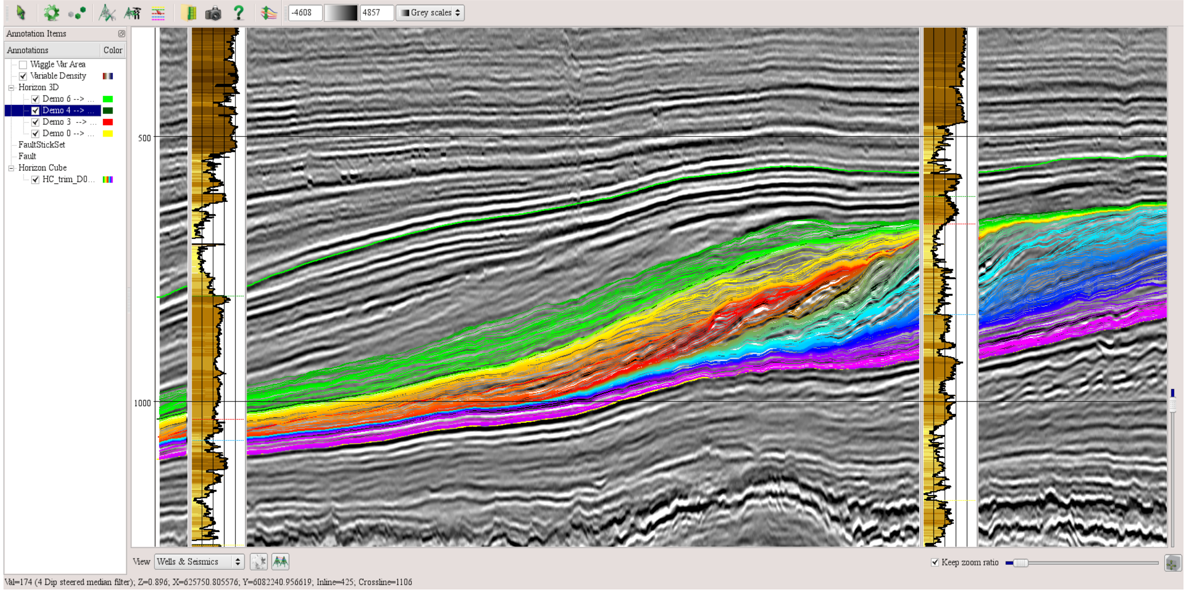
Use only wells' top position ● Yes ○ No

☑ Extend outward (m) 2500

☑ Display Random Line on creation

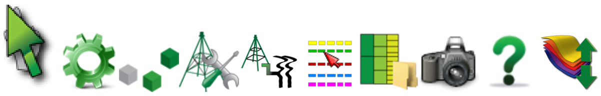
A 2D line created from 3D seismic data can be created directly in the WCP input data selection window. To define a random line geometry connecting the wells, selected the wells from the left panel using the arrow (s). The table is filled and ordered according to the selection made. However, the order of the wells can be changed using the top/bottom arrows available for the *Change Order* option. Optionally, the geometry can be visualized in a 3D Scene by pressing the *Preview* button. The geometry of the random line (*Start at*) can either be defined by the top position of a well or by a bottom position of the well (deviated wells). The *extend outward* field is by default set to 2500m such that the line is extend 2500m away from the first and the last well positions.

Once the geometry of the random line is defined, select the 3D seismic volume and provide a name to this line and corresponding line-set. This stores the geometry of random line as a 2D line, which can be loaded later. Now, this stored 2D line can be used as an input seismic data type for the WCP.



Val=174 (4 Dip steered median filter); Z=0.896; X=625750.805576; Y=6082240.956619; Inline=425; Crossline=1106

WCP Toolbar Icons



The top toolbar for the well correlation panel.

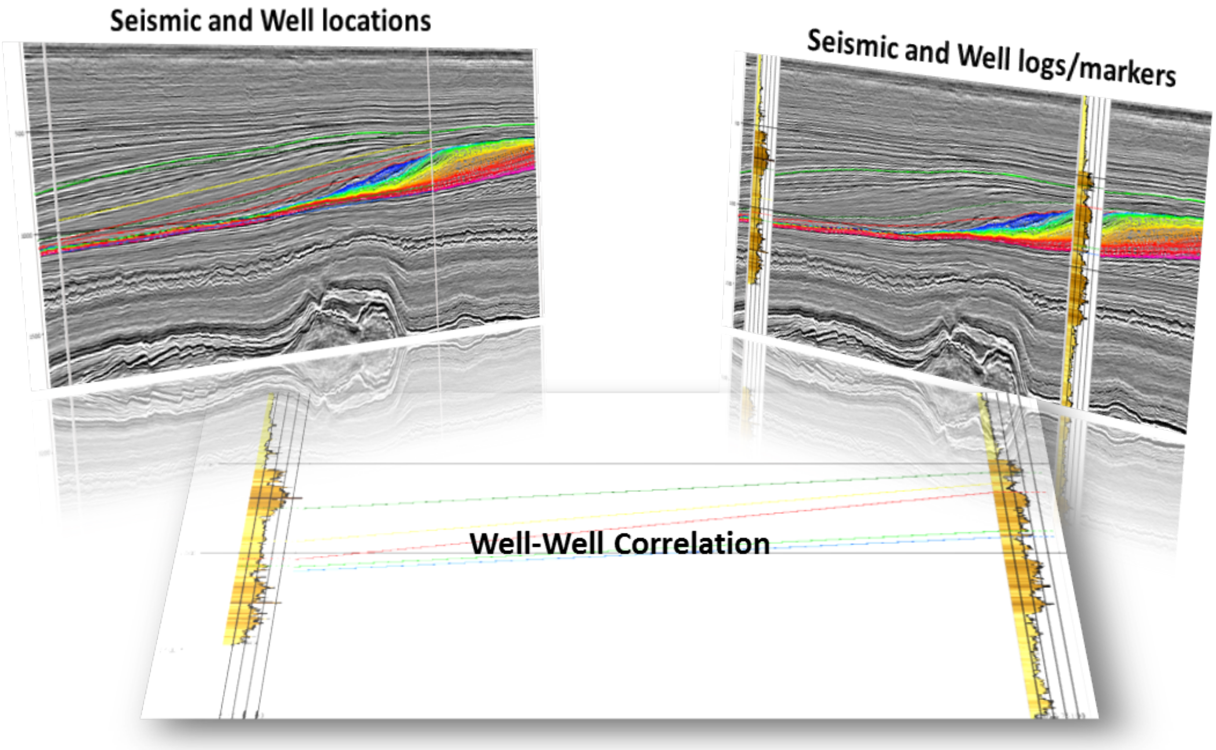
- Pan the display and pick horizons/markers on the seismics/well.
- Set the color properties (change/clip the colour and its range) of the seismic

data.

- *Seed Mode:* Start 3D/2D horizon picking on the displayed seismic data.
- Set the well properties, e.g. select logs for a well, fill logs with colours, display stratigraphic column, or add more log panels.
- Launch the seismic-well tie module of OpendTect. Please press F1 key and browse to section 6.4 for further details.
- *Pick Markers:* Pick new/modify well markers in the displayed wells.
- *Manage Stratigraphy* Launch the stratigraphy management window of OpendTect.
- *Snapshot tool:* Grab an image of the correlation panel.
- Help documentation on Well Correlation Panel.
- *HorizonCube Slider:* Active when the HorizonCube has already been displayed in the panel.
- View Wells & Seismics ⬍ Toggle WCP view e.g. Display wells only, seismic only seismic and well together.
- Create various displays: wells on top of seismic, wells as separate panel, equidistant wells correlation.
- Correlate markers by drawing a marker connection.
- Re-sets the zoom of the display to a default preview i.e. entire seismic transect is shown.
- ☑ Keep zoom ratio Associated with the zoom (in/out) sliders. It is checked by

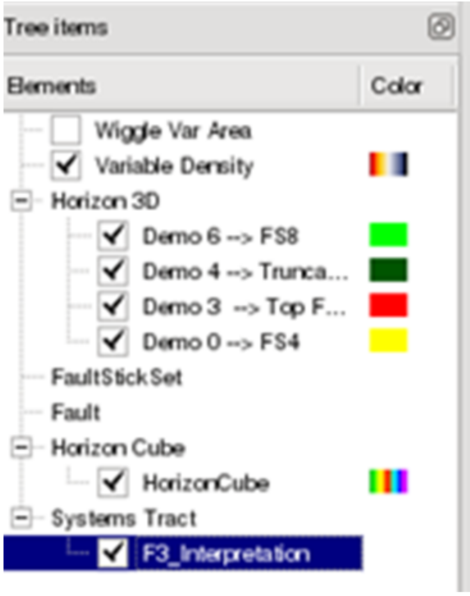
default to keep the vertical and the horizontal ratio equal while either slider (vertical/horizontal) is moved to zoom/unzoom the display.

Correlation Displays and Settings



Several displays for WCP

WCP supports several displays for correlations; Equidistant display, in which the distance between the selected wells is set to an equal distance or normal distance display, in which the real well-to-well distance is scaled and is displayed in the correlation panel. Moreover, the well data can also be overlain on top of seismic or with a gap between seismic. Any wells can also be displayed without using seismic. These display settings can be accessed from the <="" a=""> and from the tree.



Tree items

Elements | Color

- ☐ Wiggle Var Area
- ☑ Variable Density
- Horizon 3D
  - ☑ Demo 6 --> FS8 🟩
  - ☑ Demo 4 --> Trunca... 🟩
  - ☑ Demo 3 --> Top F... 🟥
  - ☑ Demo 0 --> FS4 🟨
- FaultStickSet
- Fault
- Horizon Cube
  - ☑ HorizonCube
- Systems Tract
  - ☑ F3_Interpretation

Variable Density: Right-click to display seismic data.

Horizon3D/2D: Display a 3D horizon if an inline is displayed in the panel or a 2D horizon if a 2D line is displayed in the panel. Additionally, start interpreting a new 2D/3D horizon.

FaultStickSet: Display a fault-stick set or interpret a new fault stick set.

Fault: Display/Interpret a 3D fault.

HorizonCube: Add a HorizonCube display.

Systems Tracts: Add an overlay of SSIS interpretation i.e. systems tracts.

Correlation Views

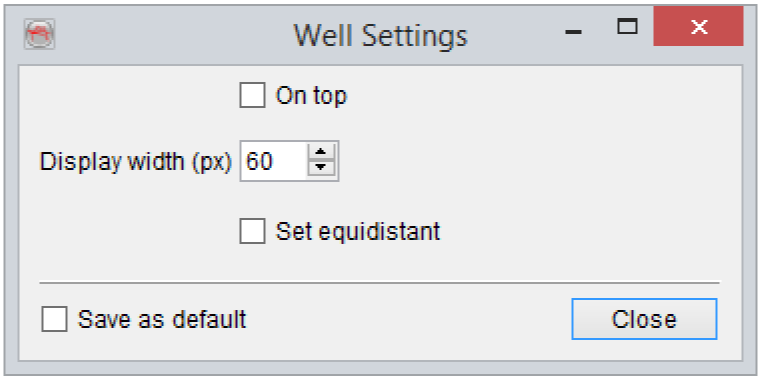
The toolbar, available at the bottom of a well correlation panel, is used to prepare different type of correlation views e.g. well-well only without seismic, well-seismic correlations without displaying logs and the well-seismic correlations using well logs and seismic. The combo-box (shown below) is used for setting these properties.

The well settings are used to display a well panel either as a gap between seismic transect or on top of the seismic transect. By default, the well panel is displayed as a separate panel connecting the seismic transect. Once the *On top* box is checked, the well logs panel is displayed on top of the seismic data. Additionally, the top log information header may be toggled off/on.

View Wells & Seismics

The width (pixels) is used to control the well panel width. The minimum range is 35pixels.

Set an *Equidistant Correlation View* check the *Set Equidistant* box.



Well Settings

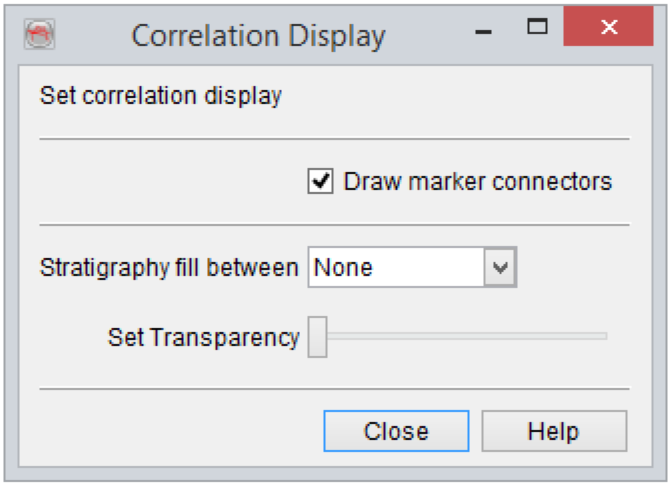
☐ On top

Display width (px) 60

☐ Set equidistant

☐ Save as default

Draw Marker Correlation(s) by pressing this icon . In the pop-up window i.e. 'Correlation Display', the *Draw marker connectors* box should be checked in order to draw a straight line marker connection between the wells. In addition, the section between the markers/horizons with the stratigraphy may be filled.



Correlation Display

Set correlation display

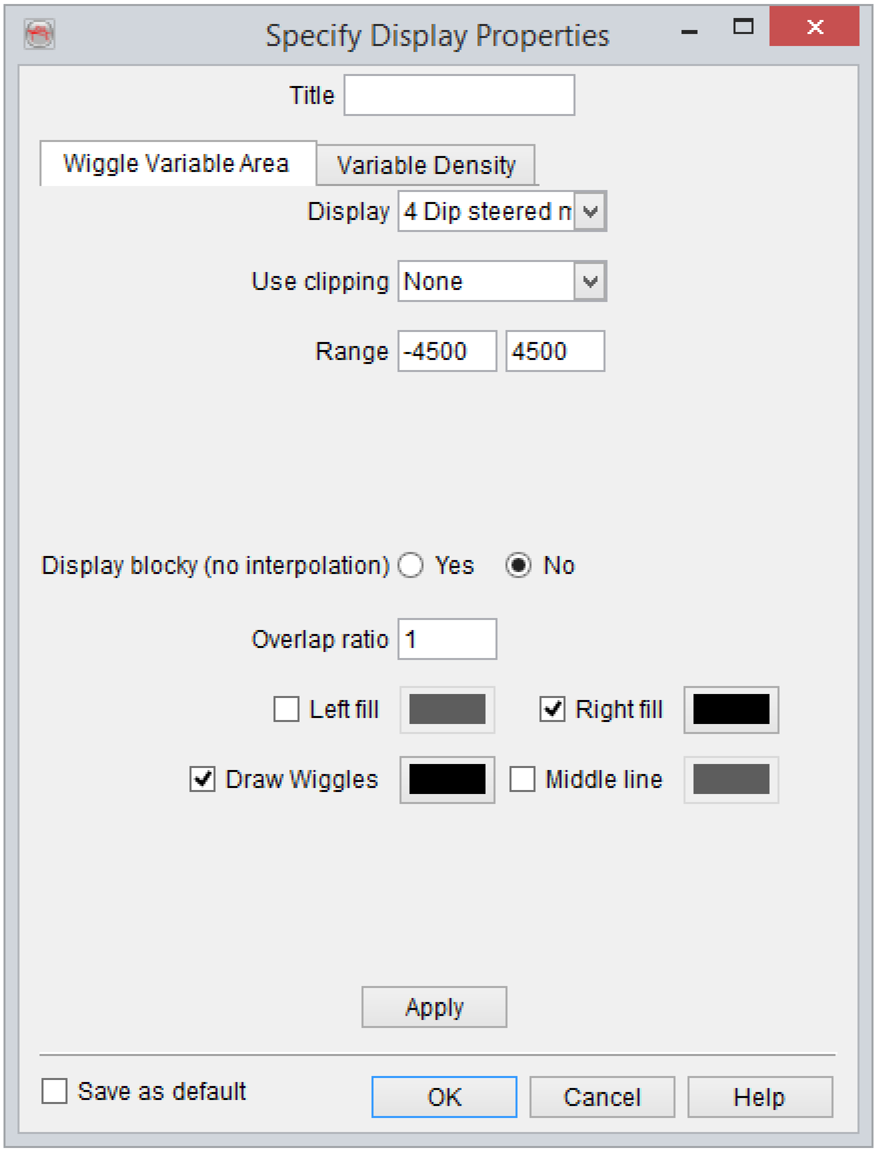
☑ Draw marker connectors

Stratigraphy fill between None

Set Transparency

Seismic Display Properties

The icon ⚙ launches the seismic display properties dialog. This window (below) is used to change the color spectrum of the seismic and to define amplitude clipping range. The clipping range can be set to 'none' for no clipping. Once the parameters are set, press the *Apply* button to see changes of the seismic display in the WCP.



Title

Display 4 Dip steered m

Use clipping None

Range -4500 4500

Display blocky (no interpolation) ○ Yes ● No

Overlap ratio 1

☐ Left fill ☑ Right fill

☑ Draw Wiggles ☐ Middle line

☐ Save as default

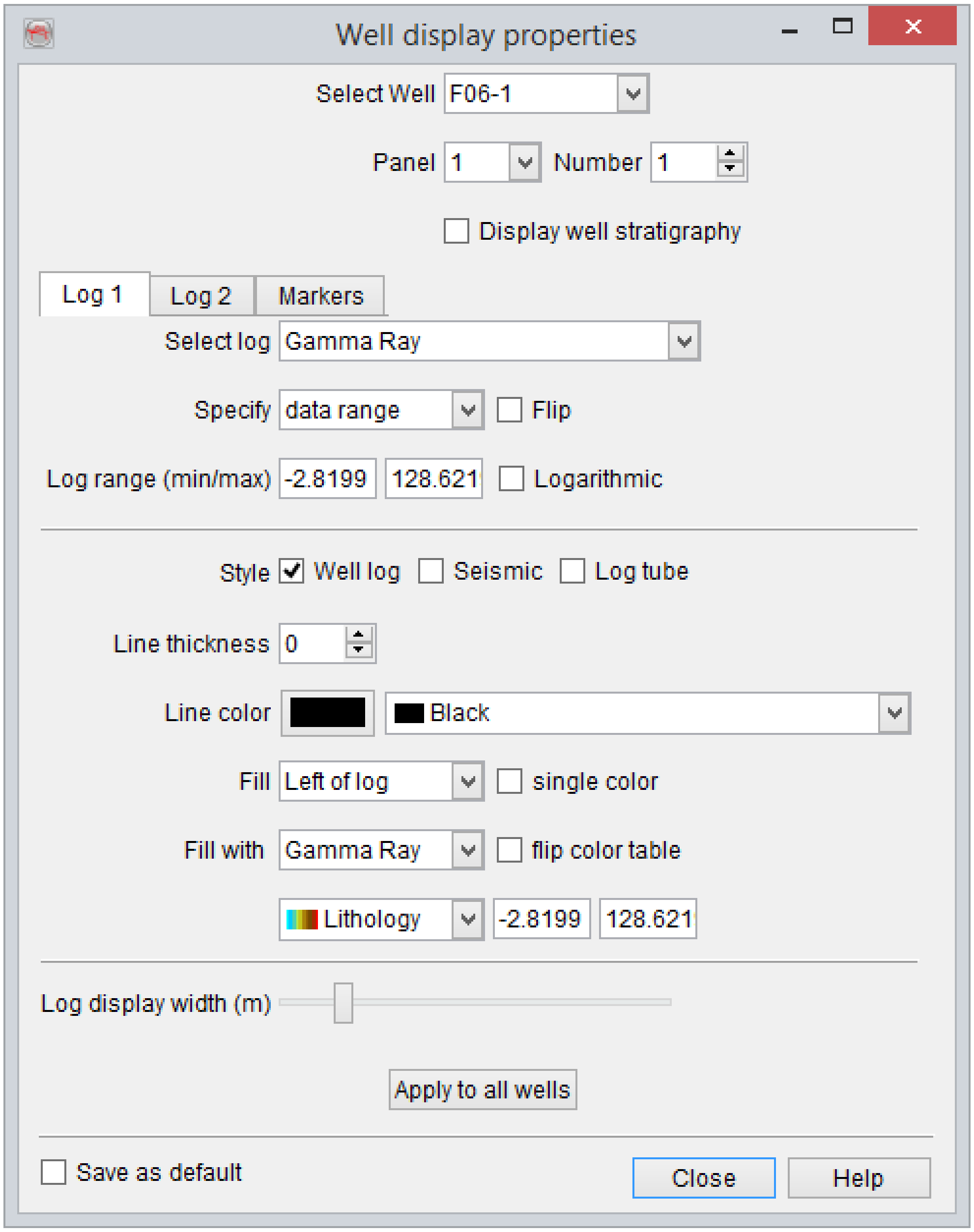
The color settings for the displayed seismic line are changed here.

Well Display Properties

The well display properties are launched by pressing this icon . The well display properties window opens the settings for individual well or for all wells. By default the settings are defined only for an individual well, as seen on the top of this window. To apply same settings to all wells displayed in the WCP, use the *Apply to all wells* button. Moreover, the tabs *Log 1* and *Log 2* are normally defined independently. However, both tabs have identical options.

By default, only one panels is displayed for the log display. The *Panel -- Number* field on the top of this window is used to add more panel and its corresponding settings. For instance, if *Number* is 2 and *Panel* is 1, and the user defines the settings in Log 1 tab, then the settings are only defined for the Panel 1. To apply new settings for another log panel, please change the *Panel* to 2.

In the *Log 1* tab, the log is selected from the drop down list of *Select* log field. The *Specify* option enables specification of a clipping/non-clipping range. If the *Specify* field is set to 'data range', the min/max log values are used to display the log curve (i.e. non-clipped). A user may over-rule this by setting *Log range (min/max)* manually. The selected log ranges can also be flipped by checking the *Mirror* option. The *logarithmic* option can be used to change the log curve display into a logarithmic display. The line *thickness* and *color* properties can also be modified. The log curve can be filled by setting the *Fill* properties. There are three types of filling properties currently supported: filling the curve on the left- or right side of the log or the full panel. Optionally, the log may be filled with a single user-defined color. Another option is to flip the color table, which reverses the color spectrum. Similar settings can be set for the other log from the tab *Log 2*.



Select Well F06-1

Panel 1 Number 1

☐ Display well stratigraphy

Select log Gamma Ray

Specify data range ☐ Flip

Log range (min/max) -2.8199 128.621 ☐ Logarithmic

Style ☑ Well log ☐ Seismic ☐ Log tube

Line thickness 0

Line color ■ Black

Fill Left of log ☐ single color

Fill with Gamma Ray ☐ flip color table

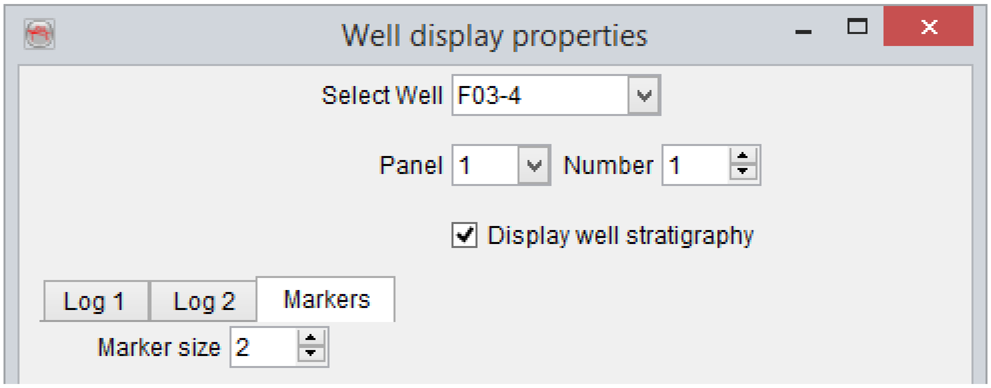
Lithology -2.8199 128.621

Log display width (m)

☐ Save as default

Well logs are selected and their display settings are changed in this window.

Another WCP display option is to overlay the WCP with Stratigraphy defined in OpendTect. This is added from the WCP well display properties (*Stratigraphy* tab)


Well display properties

Select Well F03-4

Panel 1 Number 1

☑ Display well stratigraphy

Marker size 2

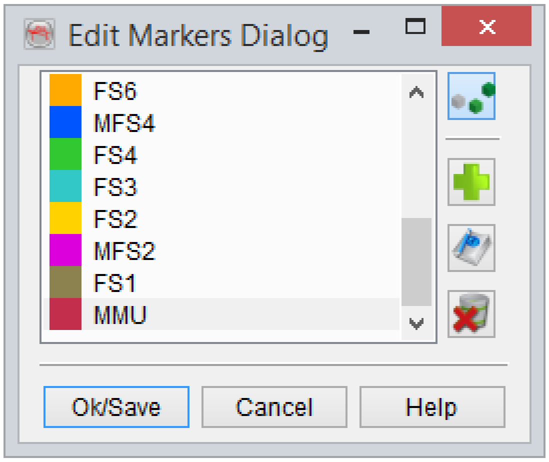
Display Stratigraphy in WCP

Pick Markers and Correlate

New well-markers are defined or alternatively, the existing markers can be modified using WCP. The purpose of this functionality is to build a new stratigraphic framework of the area. The *Edit Makers dialog* is launched by clicking on this icon.

How to pick markers?

By keeping this dialog box open, markers are added to the well using a mouse click on the well-panel displayed in the WCP. Before starting picking, the 'interact' (Pick) mode should be toggled on . Once markers are interpreted and added to all wells, press the *OK/Save* button to save the markers.


Edit Markers Dialog

- 🟧 FS6
- 🟦 MFS4
- 🟩 FS4
- 🟦 FS3
- 🟨 FS2
- 🟪 MFS2
- 🟫 FS1
- 🟥 MMU

This dialog box is used when picking new markers.

SynthRock

Table of Contents

- Introduction
- Stochastic pseudowell modeling
- Profile modeling
- Fluid replacement
- HitCube stochastic inversion

Introduction

The SynthRock plugin is a forward pseudo-well modeling and probabilistic inversion package to achieve qualitative and quantitative seismic interpretation. It supports wedge models, stochastic models, pre- and post-stack synthetic seismograms generation and cross-matching (HitCube) inversion. This commercial plugin extends the capabilities of the **Basic Layer modeling** module of OpendTect.

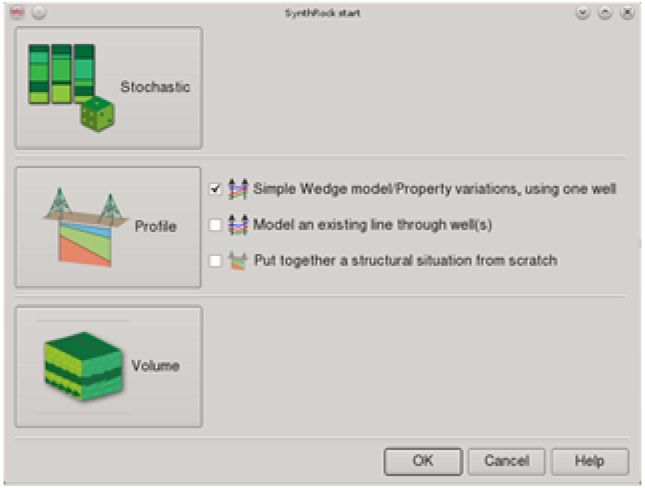
Forward modeling of synthetic seismic data plays a key role in understanding the seismic response of a target interval. A thorough understanding of the response enables interpreters to separate true hydrocarbon anomalies from false positives and to make more accurate reservoir predictions. In a forward model, geologic information and synthetic seismic data is integrated through a rock physics model.

Sophisticated workflows in which model parameters are varied stochastically can be run to create a data base of pseudo-wells, representative of the expected geologic and seismic variations at target level. Such models are then used to predict rock properties with uncertainties from pre- and post-stack seismic volumes.

The new “Volume” module of SynthRock allows the generation of post-stack and pre-stack synthetic seismic data from 3D rock property volumes (AI-SI, Vp-Vs-Rho), for comparing the synthetic seismic data with the recorded seismic data.

SynthRock can either be started in two ways:

- From the corresponding icon in the main OpendTect toolbar .
- From within the stratigraphy manager with the icon .



☑ Simple Wedge model/Property variations, using one well
☐ Model an existing line through well(s)
☐ Put together a structural situation from scratch

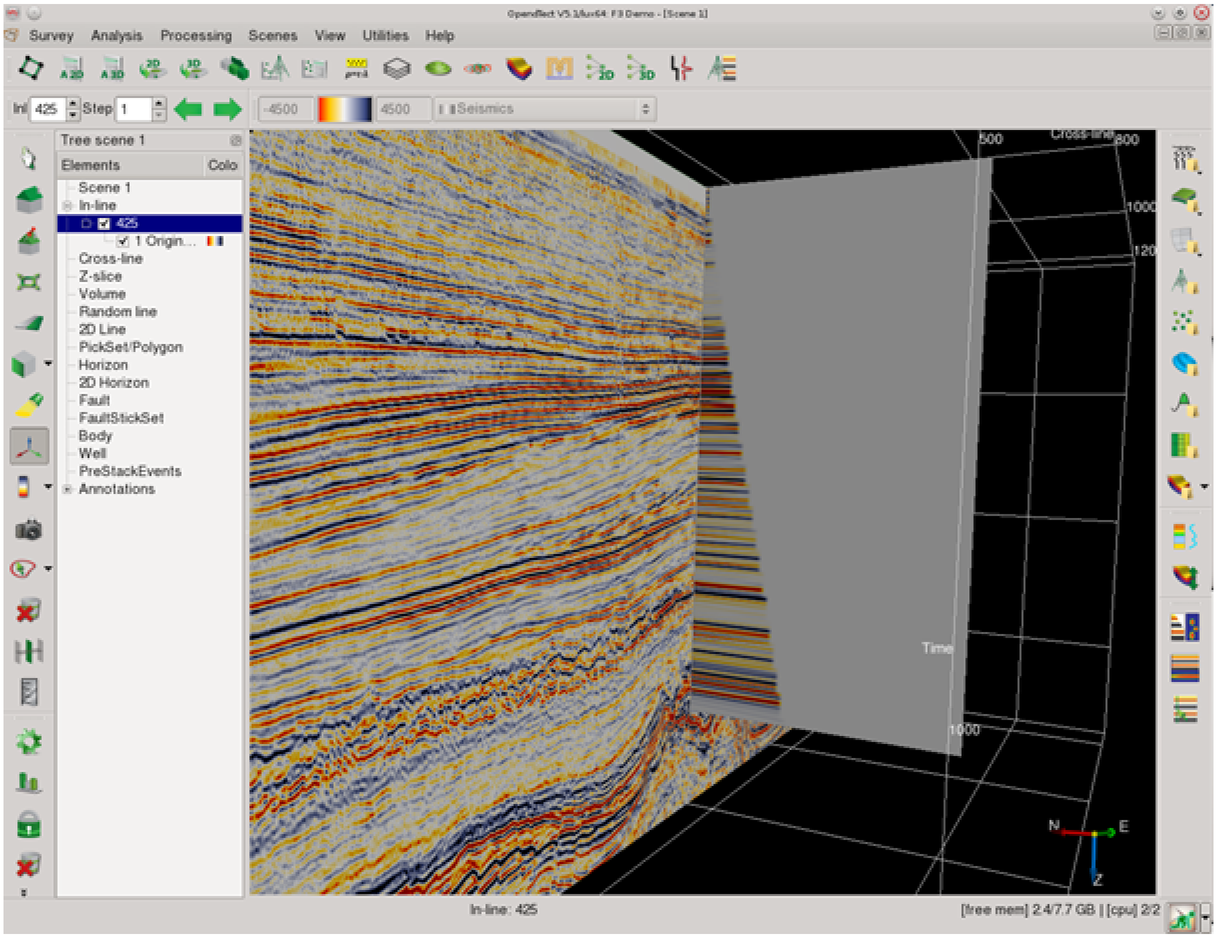
SynthRock can be used to do the following tasks:

1. Perform a stochastic pseudowell modeling.
2. Create pseudowells profiles by interpolating between/from existing wells.
3. Applying Gassmann fluid substitution on the modeled pseudowells.
4. Generate synthetic pre-stack seismic using full zoeppritz from either the pseudowells or 3D rock property volumes.
5. Extract synthetic seismic attributes and/or well layer attributes from the modeled pseudowells and optionally export them along a 2D line.
6. Run the HitCube stochastic inversion from the modeled wells.

Note: Once a layer modeling window is started, you cannot change its type between basic, stochastic and profile. To use another model type you need another window, to be launched similarly from either the analysis menu or from the main toolbar icon. Only one layer modeling window (thus SynthRock window) can be used at the time. That is because the stratigraphy object needs to be locked during the modeling. A full uncoupling will be implemented in later releases. For the time being you will need to launch additional OpendTect main windows if you want to work with several models simultaneously (at your own risk).

A stand-alone SynthRock application will be made available post 5.0.

Volume Synthetics



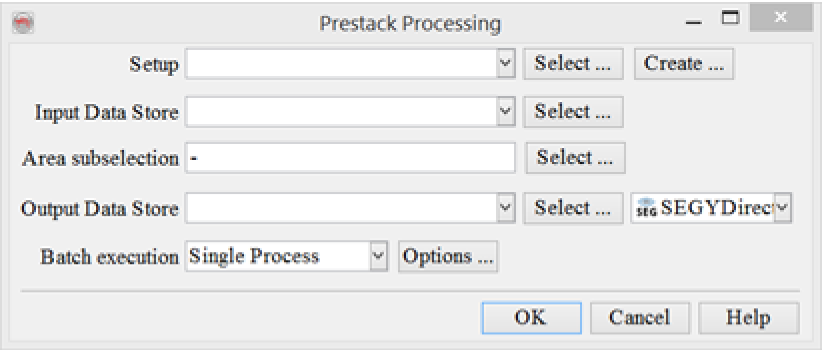
Introduction

Volume Synthetics is used to create Acoustic, or Elastic synthetic seismograms from user-supplied elastic parameters. For acoustic modelling the user either inputs volumes of P-wave and Density, or a volume of Acoustic Impedance values with Density as an optional input. For elastic modelling the inputs are: P-wave, S-wave and Density, or Acoustic Impedance and Shear Impedance with Density as optional input.

Synthetics are calculated by ray-tracing through a horizontally layered isotropic earth model. SynthRock uses a sophisticated, ray-trace based, synthetics generating algorithm that accurately solves Zoeppritz' equations. It supports generation of: PP, PS; near, mid, far, full & angle stacks, from which other properties such as Gradient Impedance, AvO attributes and Extended Elastic Impedance, can be derived. Optionally multiples can be generated.

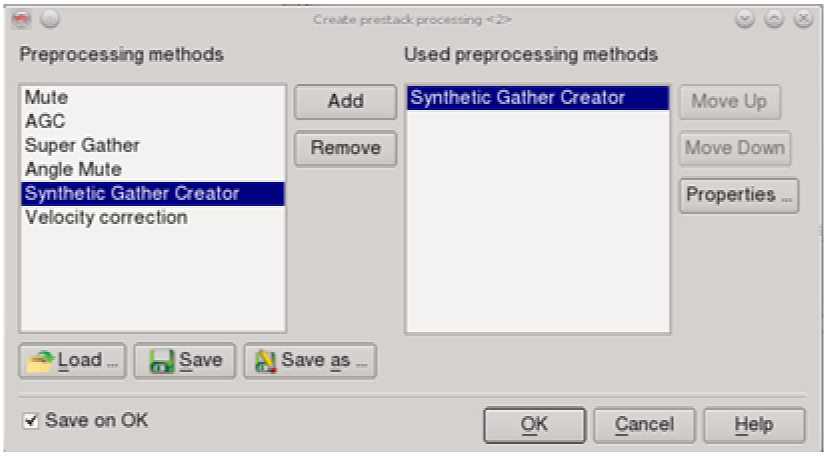
Volume Synthetics is integrated in the Pre-stack processing system under the option Synthetic Gather Creator. This option can only be included as the first step in a pre-processing workflow.

Volume synthetics is either launched from the SynthRock launch menu, or from the Processing -> Pre-stack Processing … menu. From the latter menu the Pre-stack Processing window is launched. Here the locations where synthetics are to be computed are specified under *Area sub-selection*. To define a pre-processing work flow press *Create*.

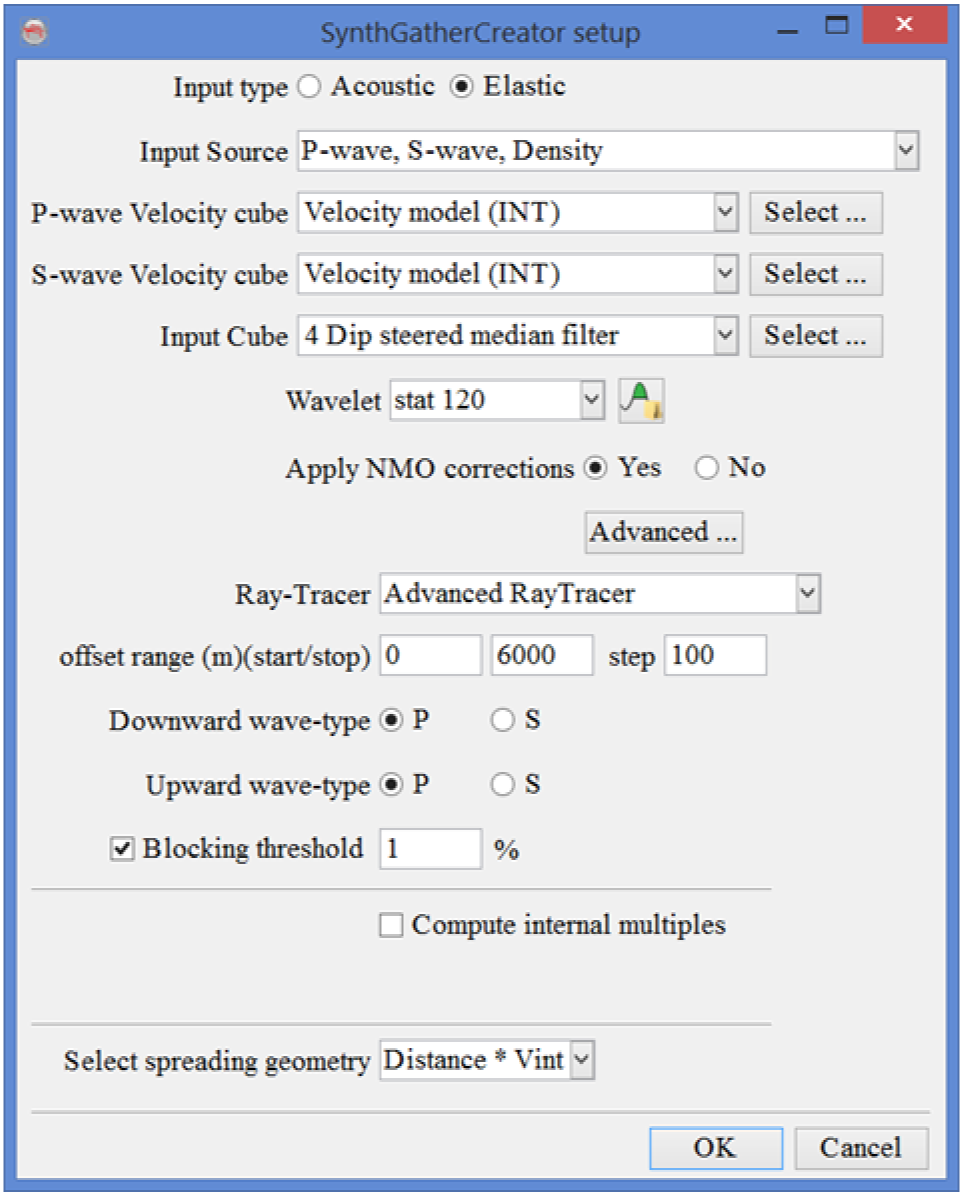


Area subselection -

Batch execution Single Process



Input parameters are specified in the SyntheticGatherCreator setup Window, which is launched from the Properties … button.



Input type ○ Acoustic ● Elastic

Input Source P-wave, S-wave, Density

P-wave Velocity cube Velocity model (INT)

S-wave Velocity cube Velocity model (INT)

Input Cube 4 Dip steered median filter

Wavelet stat 120

Apply NMO corrections ● Yes ○ No

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 %

☐ Compute internal multiples

Select spreading geometry Distance * Vint

In this window select the synthetics modelling type (Acoustic, or Elastic), the input volumes, the wavelet, and what output to generate. The various options are described in **Appendix A - Synthetics Generation**.

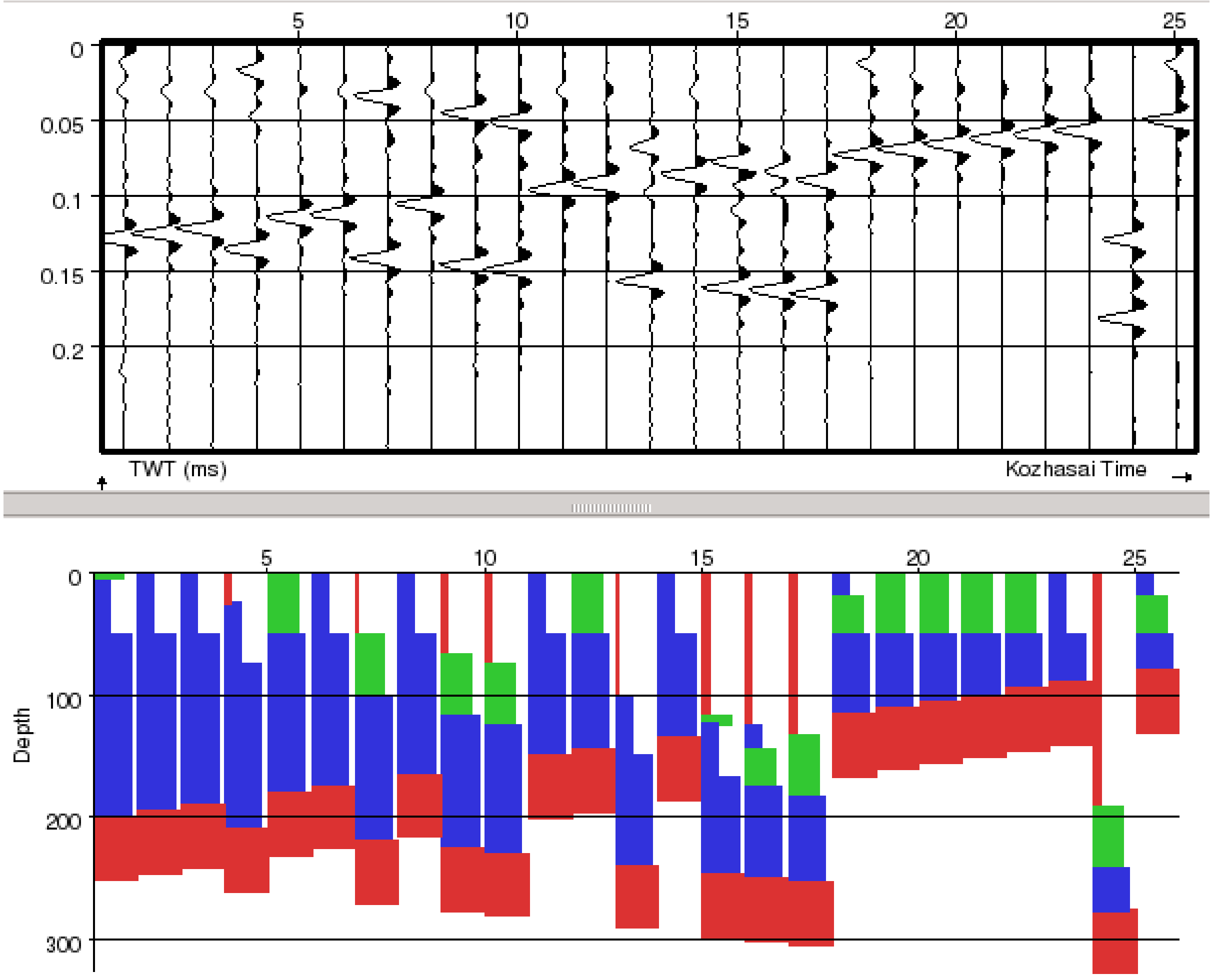
Stochastic Pseudowell Modeling

The stochastic modeling is performed using the following workflow:

1. First the **modeling nodes** must be selected. They can be single layers, formations, or meta-formations.
2. For each member of each node the **stochastic parameters** must be set. Default constant parameters are provided.
3. Pressing "Go" will draw a user defined number of pseudowells that honor the modeling description.
4. Then **fluid substitution** may be applied. This process duplicates the pseudowells into a brine set and a fluid-filled set.

The modeling itself will take place on the left side of the layer definition window. The modeling description can be saved, but not the stochastically derived pseudowells, except by extracting the corresponding data using the **crossplot extractor**.

The pseudowells are re-drawn by either pressing on the Go! ➡ button or resizing the layer modeling window (sic). It is advised to start with a relatively low number of pseudowells while building the model, and then to increase it for the final application (several hundreds for a **HitCube inversion** are needed). An example is given below.

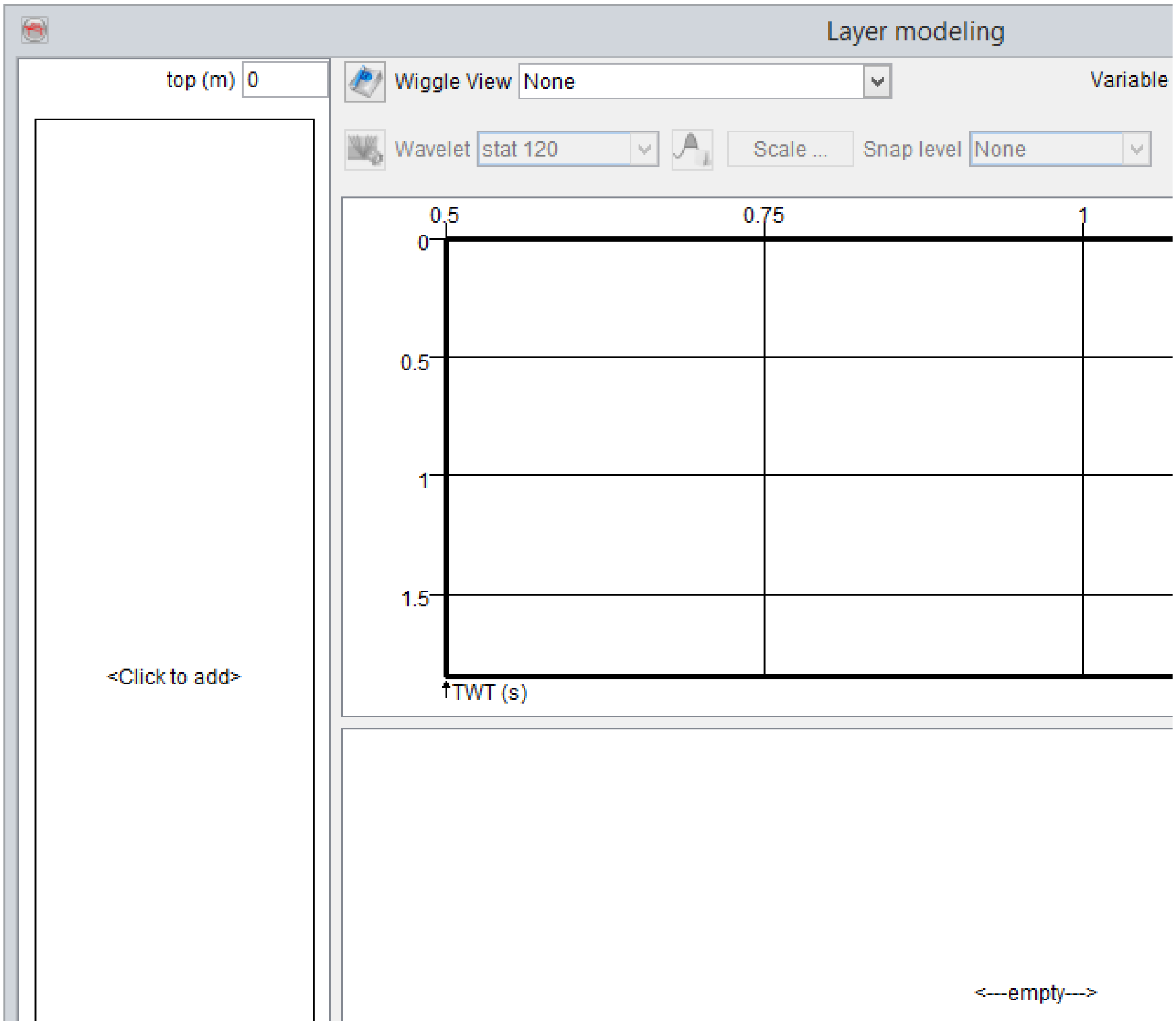


Add New Modeling Node

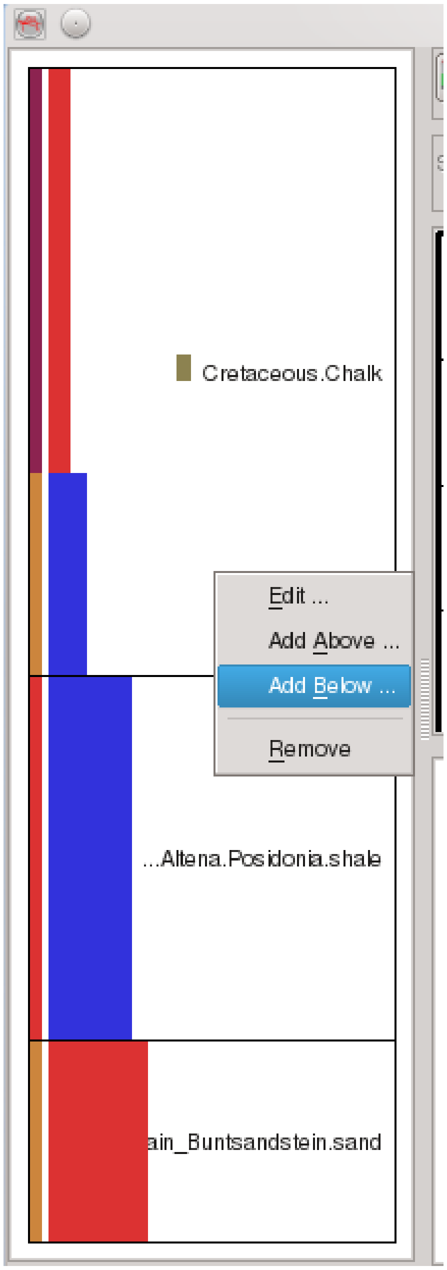
The modeling description is typically composed of a number of nodes that are added in a given order in the layer description. The nodes are taken from the current stratigraphic framework, thus this framework must first be filled with the formations and lithologies that will be used during the modeling, prior to the modeling.

How-to add a node:

Click on the left side of layer modeling window for the clean start, or right-click in this column when there is already at least one top node:



empty vs existing



For the first clean run of the simulation (i.e. before any node has been added), specification of layer properties to be used in the modeling is mandatory. Various rock properties can be selected from the available list. It is also possible to define a new property by pressing . For more details go to **Manage Layer Properties**.

Simulation units can be:

- Single layers, one lithology of a given formation. This is useful only if you want to insert a single, most often thick, blocky formation into the modeling.

  > **Tip:** This option may be chosen to model relatively thick caprock shale above a (sand) reservoir as well as for modelling the underlying shale.

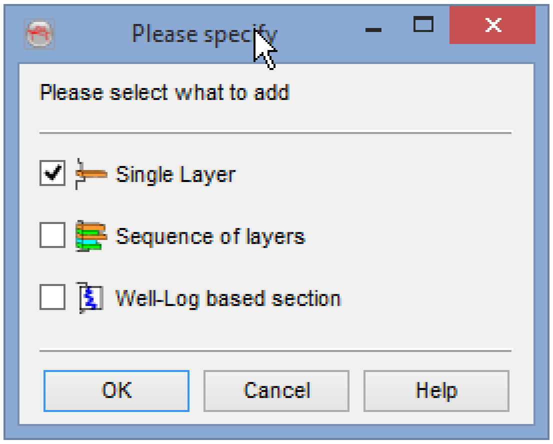
- Sequence of layers, it could either be a single formation composed of all the lithologies it can contain or a meta-formation. Meta-formations regroup all the formations (and their corresponding lithologies) underneath, under a single tree structure, as individual simulation units (example below).

  > **Tip:** The above option is useful while modeling multiple lithologies within a formation, e.g. an interbedded sand-shale sequence in the reservoir with a particular Net-to-Gross ratio or Sand-to-Shale ratio.

- Well-log based section, it can contain either full or part of well log data, which has been upscaled/blocked at a constant size.

  > **Tip:** This option is of particular use when a detailed overburden model is required, e.g. for generating pre stack synthetic gathers, where in the ray-tracing phase requires specification of the elastic model from ground-level down to the interval of interest. This could simply be achieved by choosing Vp, Vs and Density logs from an available well, where the logging is done from ground level down to the interval of interest.

Adding a Single Layer:

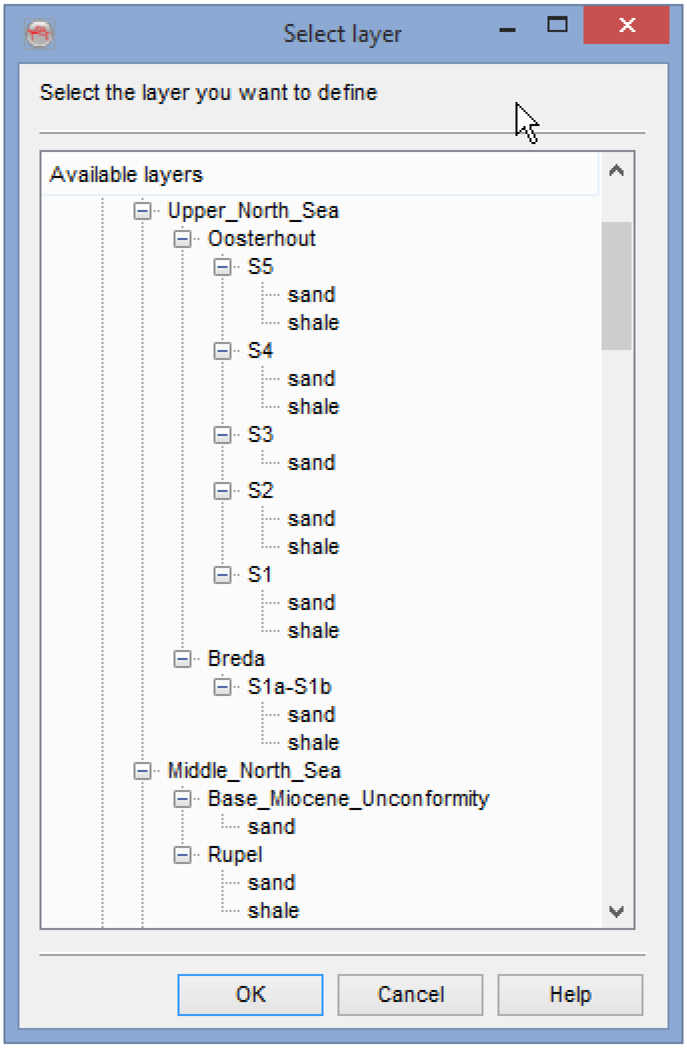


Please select what to add

☑ Single Layer

☐ Sequence of layers

☐ Well-Log based section



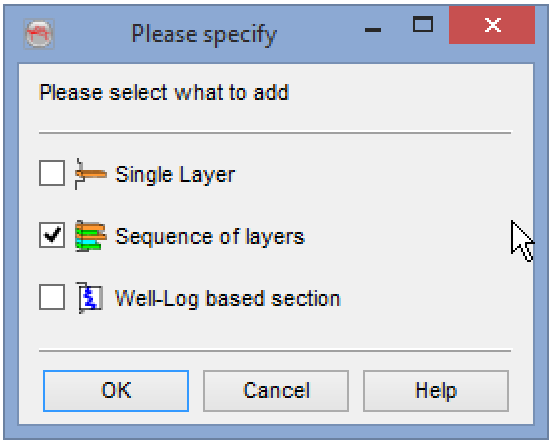
Select layer

Select the layer you want to define

Available layers

- Upper_North_Sea
  - Oosterhout
    - S5
      - sand
      - shale
    - S4
      - sand
      - shale
    - S3
      - sand
    - S2
      - sand
      - shale
    - S1
      - sand
      - shale
  - Breda
    - S1a-S1b
      - sand
      - shale
- Middle_North_Sea
  - Base_Miocene_Unconformity
    - sand
  - Rupel
    - sand
    - shale

Adding a Multiple Layers:



Please specify

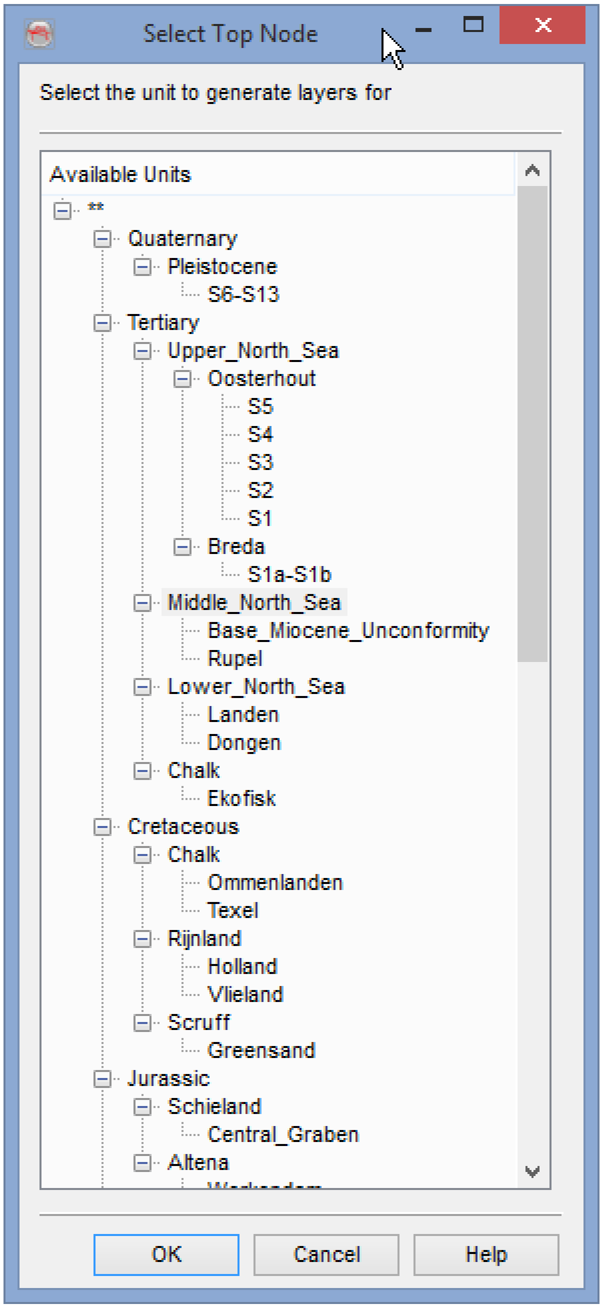
Please select what to add

☐ Single Layer

☑ Sequence of layers

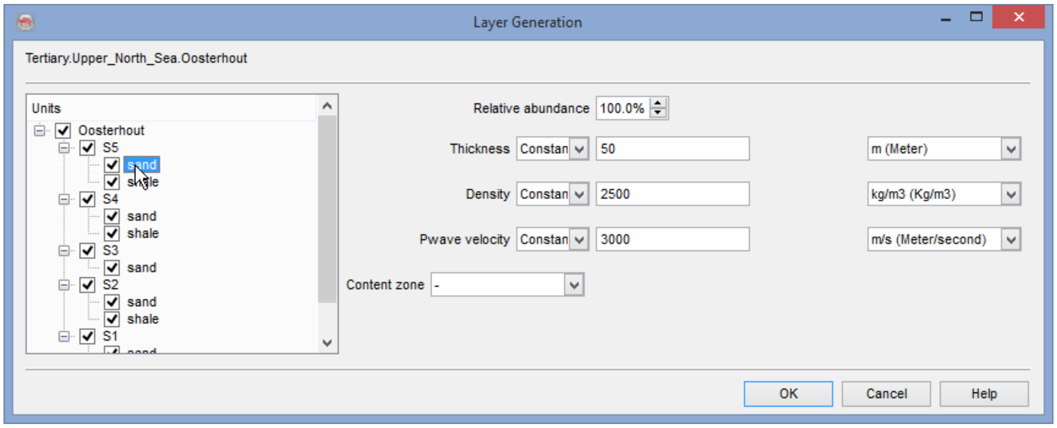
☐ Well-Log based section

Note: On first-time addition of mulitple layers, you will be prompted to select the top node from the following window:



Select the unit to generate layers for

Before being taken to the Layer Generation window:



Tertiary.Upper_North_Sea.Oosterhout

- ☑ Oosterhout
  - ☑ S5
    - ☑ sand
    - ☑ shale
  - ☑ S4
    - ☑ sand
    - ☑ shale
  - ☑ S3
    - ☑ sand
  - ☑ S2
    - ☑ sand
    - ☑ shale
  - ☑ S1

Relative abundance 100.0%

Thickness Constan 50 m (Meter)

Density Constan 2500 kg/m3 (Kg/m3)

Pwave velocity Constan 3000 m/s (Meter/second)

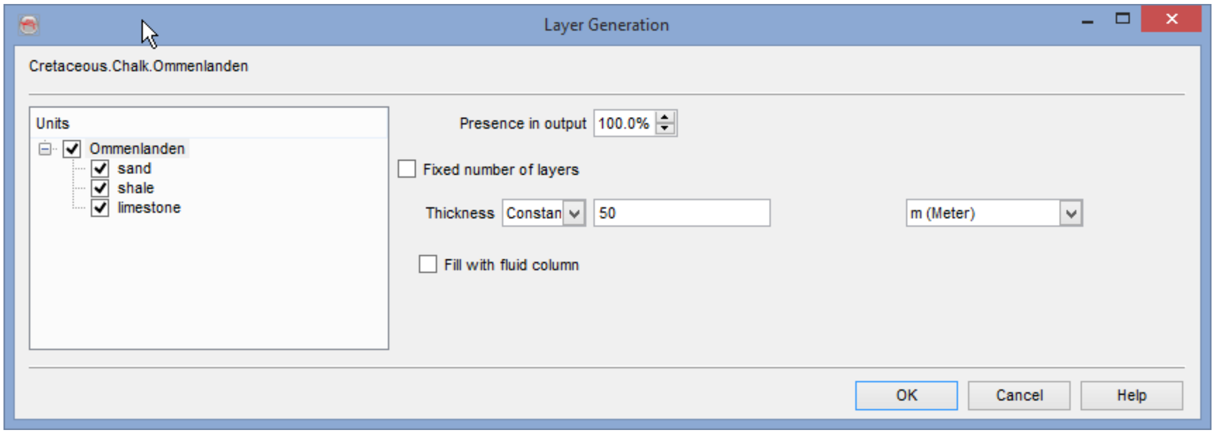
Content zone -

Model Definition

The modeling will distinguish three types of nodes:

1. **Meta-formations**

These nodes are specific by the fact that they do *not* have lithologies on the level below, but formations. They are used to **control the order of appearance** of the formations during the simulation. For instance in the example below the top node "Chalk" is used to make sure that the "Ommenlanden" formation is always modeled on top of the "Texel" formation. Optionally one can also set the probability of presence of the entire node.



Cretaceous.Chalk.Ommenlanden

- ☑ Ommenlanden
  - ☑ sand
  - ☑ shale
  - ☑ limestone

Presence in output 100.0%

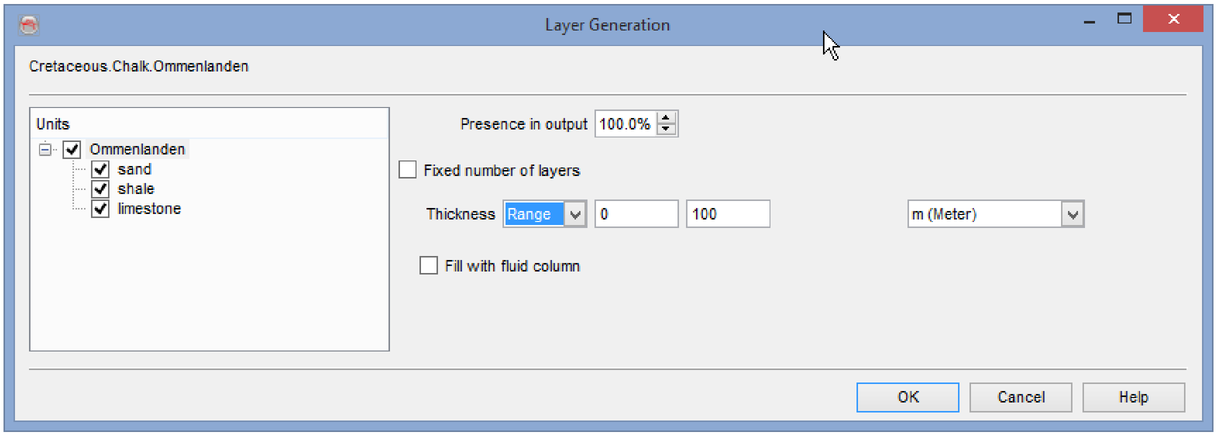
☐ Fixed number of layers

Thickness Constan 50 m (Meter)

☐ Fill with fluid column

2. **Formations**

These nodes are specific by the fact that they only have lithologies on the level below. The simulation will add blocks taken from the leaves from bottom upwards according to the thickness and probability of presence settings of the leaves, until the thickness of the formation is reached. See example below:



Cretaceous.Chalk.Ommenlanden

- ☑ Ommenlanden
  - ☑ sand
  - ☑ shale
  - ☑ limestone

Presence in output 100.0%

☐ Fixed number of layers

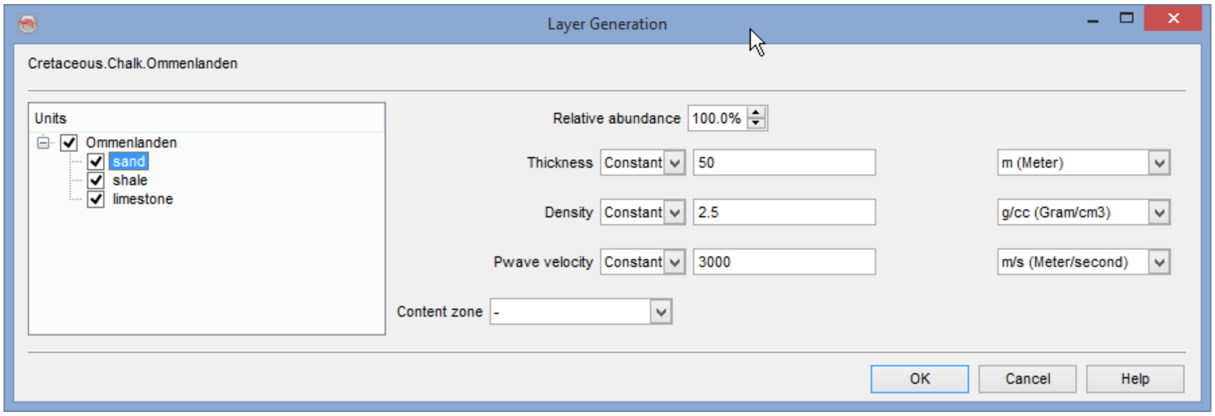
Thickness Range 0 100 m (Meter)

☐ Fill with fluid column

Optionally the thickness of the formation can be constrained by the number of blocks, instead of the sum of their thickness. See example below:

3. **Leaves**

The leaves are the key item of the pseudowell modeling. Each leave that is modeled is a small layer, stacked on top of the previously created layers of the pseudowell. It belongs to a given formation, has a top depth (Z), thickness, log values, and a given fluid content. See example below:



Cretaceous.Chalk.Ommenlanden

- ☑ Ommenlanden
  - ☑ sand
  - ☑ shale
  - ☑ limestone

Relative abundance 100.0%

Thickness Constant 50 m (Meter)

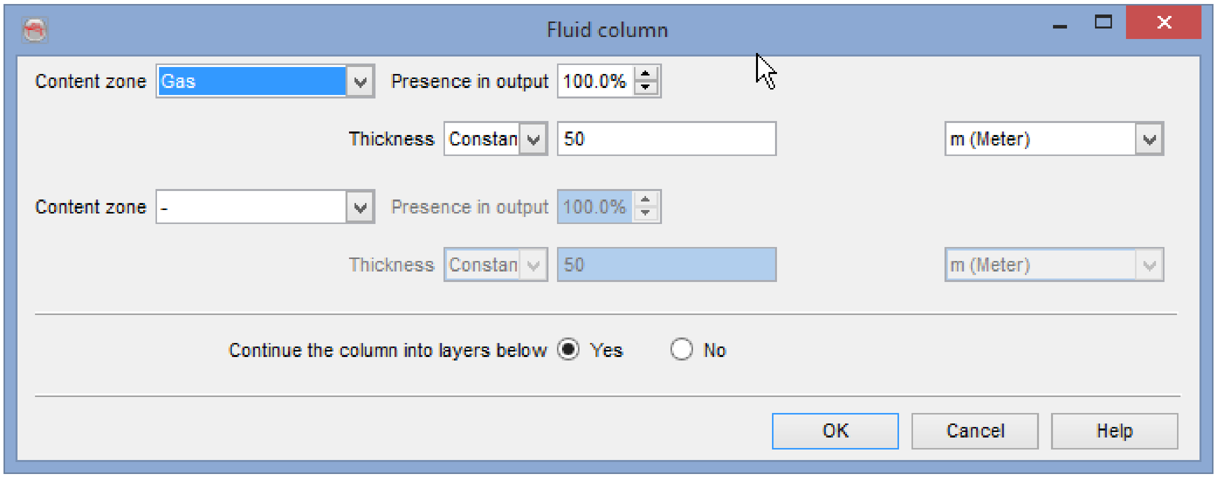
Density Constant 2.5 g/cc (Gram/cm3)

Pwave velocity Constant 3000 m/s (Meter/second)

Content zone -

The relative abundance defines the probability to find a given lithology in a formation. It is used to draw the lithology type to add when inserting a new block in a formation, after the previous. It can be used to model a net-to-gross ratio, but the user will need to make sure that the sum of the relative abundances of all leaves is equal to 100%.

It is possible to specify if some fluid is present or not by selecting the box *Fill wth fluid column* for the meta-formations and formations. If the option is selected, the characteristics of the fluid column have to be *Defined*. It will consist in at least one fluid.There can be as many fluids present as you defined in the Manage Contents window. This fluid column will be present in every generated pseudo-wells if the *presence* in output is 100%. In addition to the probability of occurrence, its thickness can be set to constant or varied randomly or within a user-defined range.The fluid column can be continued into the layers below if for example the layer below is also a reservoir for the case if the thickness of the fluid column exceeds the total thickness of the layer where it is defined in some pseudo-wells.


Fluid column

Content zone: Gas
Presence in output: 100.0%
Thickness: Constan 50 m (Meter)

Content zone: -
Presence in output: 100.0%
Thickness: Constan 50 m (Meter)

Continue the column into layers below ● Yes ○ No

At the lithology level, the fluid content can be simply one of the fluids available in the Content manager or a fluid column can also be defined.

All numerical values can be set in the following ways (select the desired method and press set if applicable):

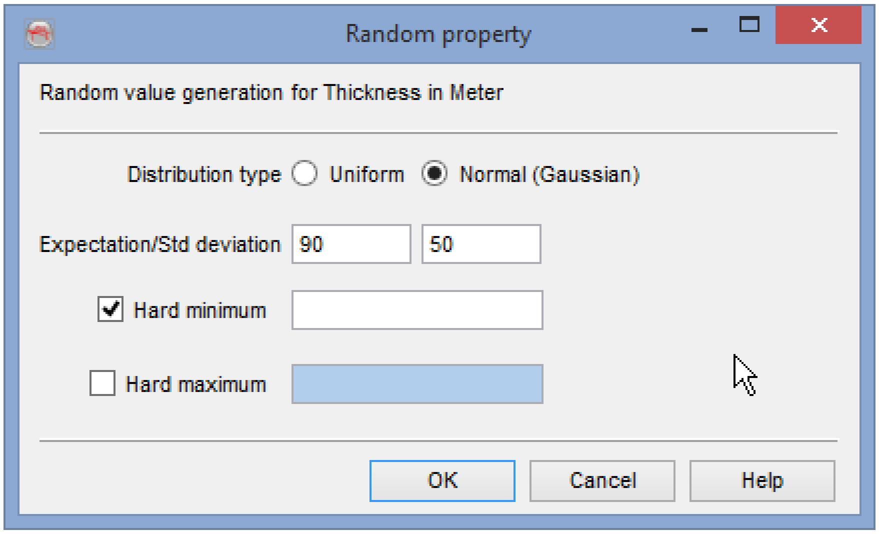
- Constant value (default when starting).
- Range: A linear variation with increasing pseudowell index
- **Random**: Gaussian or uniform distributions.
- **PDF**: A distribution drawn from a saved probability density function.
- **Math**: Value computed from other quantities.

Random Distribution

The modeled parameter value can be drawn from a uniform or normal (gaussian) distribution. This later is the most realistic for most logs, and formation thicknesses. It can nevertheless happen that the thickness distribution is uniform throughout the survey. A good way to estimate what distribution could be used is plot the histogram of the available data: Well logs for the logs, isopach maps for the thicknesses when the top and base have been interpreted. Displaying the histogram of the available data is more important than one could expect: If the log is spiky, the distribution will be skewed and the extracted average will be shifted from the real position.

It is possible to constraint the drawn value with hard extrema. There are two modes to do this:

- Either the value is re-drawn until it falls within the boundaries. You would typically do this for a sonic or density log.
- Or the value is clipped to the extrema. This is typical of a porosity log at its minimum (0%) or a water saturation log at its maximum (100%), since you can expect many of the layers to have exactly the extrema value.



Random value generation for Thickness in Meter

Distribution type ○ Uniform ● Normal (Gaussian)

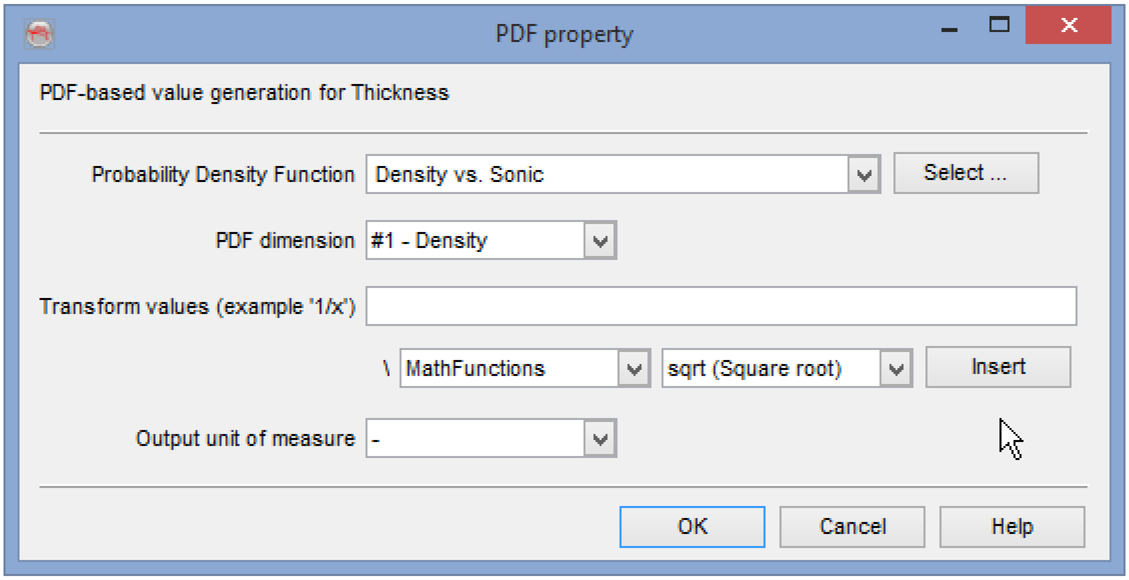
Expectation/Std deviation 90 50

☑ Hard minimum

☐ Hard maximum

PDF Distribution

The modeled parameter value can be drawn from the distribution read in a stored probability density function. Stored PDFs have multiple dimensions, thus you will need to select what dimension to use. A single PDF can thus be used to provide the input for multiple quantities. Also a transformation can be applied on the values stored in the PDF, to simulate sonic when the PDF contains P-wave, or to simulate log(K) when the distribution contains K (permeability).



PDF property

PDF-based value generation for Thickness

Probability Density Function: Density vs. Sonic

PDF dimension: #1 - Density

Transform values (example '1/x')

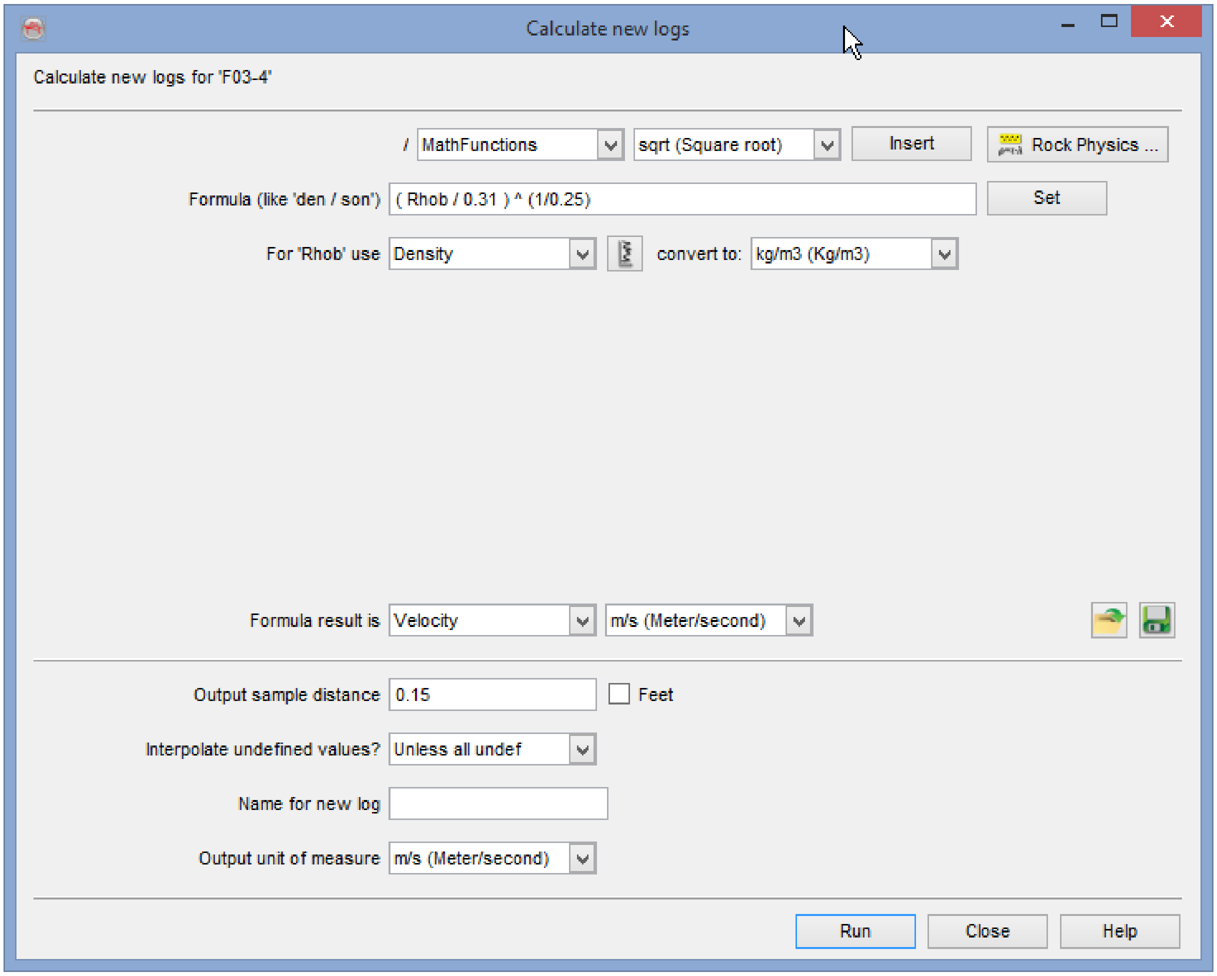
\ MathFunctions

sqrt (Square root)

Output unit of measure: -

Math-based Layer Properties

Mathematics equations can be set to derive pseudowell properties from other modeled properties, or using simple rock property models. For instance, this first example computes the P-wave velocity value in each layer from the modeled density.



Calculate new logs for 'F03-4'

/ MathFunctions sqrt (Square root)

Formula (like 'den / son') (Rhob / 0.31) ^ (1/0.25)

For 'Rhob' use Density convert to: kg/m3 (Kg/m3)

Formula result is Velocity m/s (Meter/second)

Output sample distance 0.15 ☐ Feet

Interpolate undefined values? Unless all undef

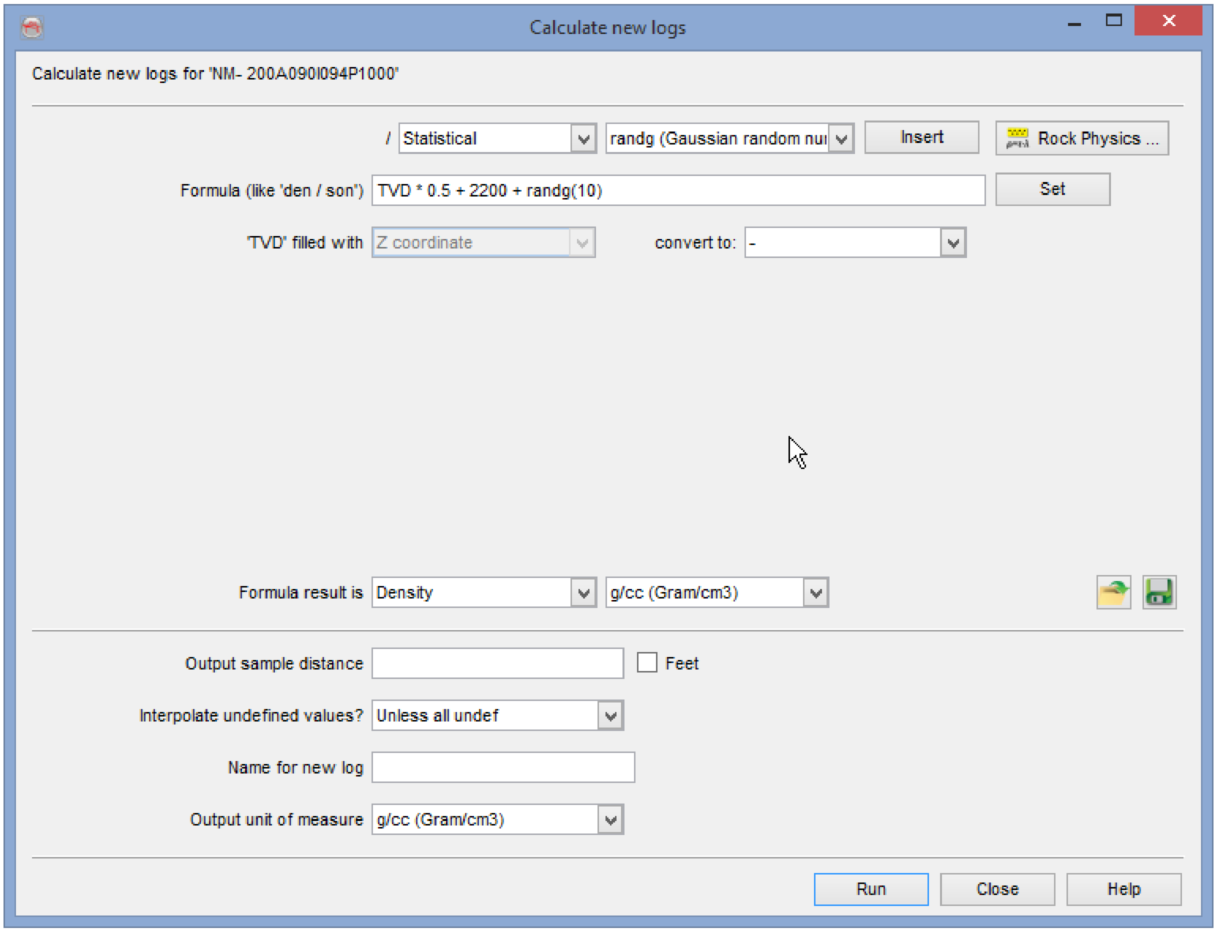
Name for new log

Output unit of measure m/s (Meter/second)

Input properties are converted to the selected unit of measure before the equation is computed. The Formula output unit have to be specified, since it might be different than the default property unit of measure. Example: The formula delivers a result in g/cm3, but the preferred display unit is set to kg/m3 (in the layer properties manager for the selected property).

Layer features like thickness and the depth at the top of the block can be used in the mathematical equation as input. Note that thickness is available from the drop-down property selection menu. While TVD (Depth) is available through the *Insert* option of type *'Other'*.

The second example creates a vertical velocity gradient starting from 2200 m/s at zero depth with a slope of 0.5 (m/s)/m and added Gaussian noise with a standard deviation of 10 m/s.



Calculate new logs

Calculate new logs for 'NM- 200A090I094P1000'

/ Statistical | randg (Gaussian random nu

Formula (like 'den / son') TVD * 0.5 + 2200 + randg(10)

'TVD' filled with Z coordinate convert to: -

Formula result is Density | g/cc (Gram/cm3)

Output sample distance ☐ Feet

Interpolate undefined values? Unless all undef

Name for new log

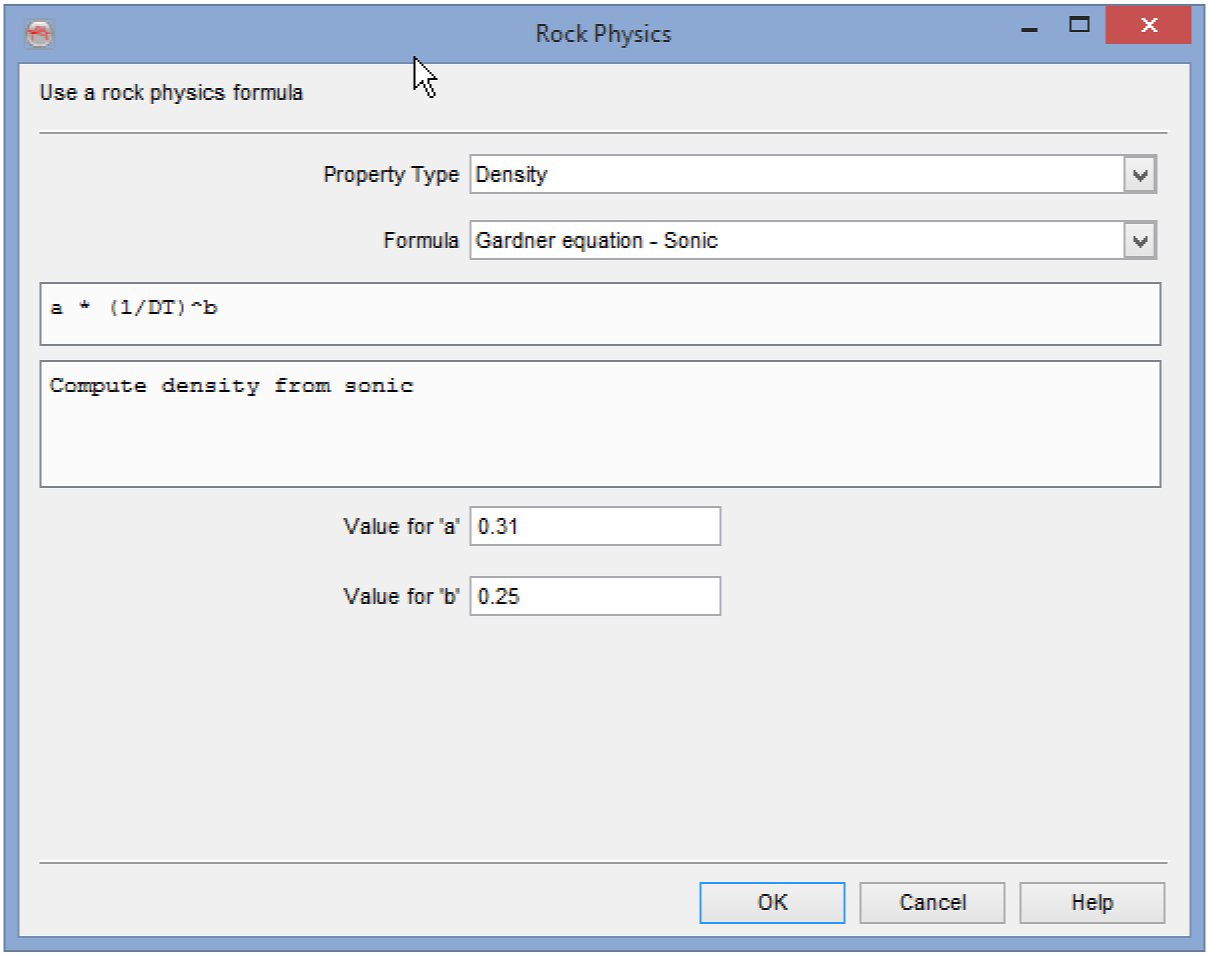
Output unit of measure g/cc (Gram/cm3)

User-defined formula can be saved (💾) and retrieved (📂).

The same syntax as in the mathematics attributes should be used.

Rock physics link

The rock physics library, via Rock Physics ..., provides a comprehensive set of equations that can be used to derive logs from others. Please refer to the OpendTect User Documentation.



Use a rock physics formula

Property Type: Density

Formula: Gardner equation - Sonic

```
a * (1/DT)^b
```

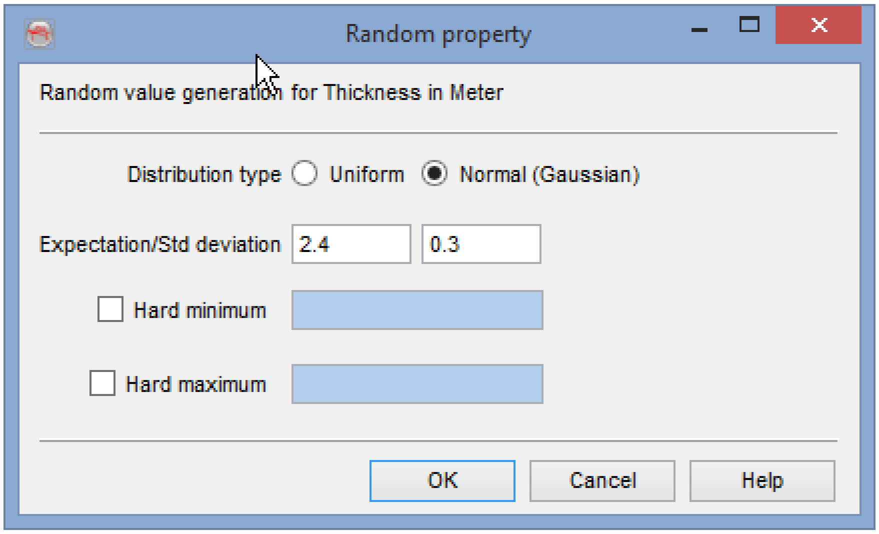
```
Compute density from sonic
```

Value for 'a': 0.31

Value for 'b': 0.25

Analysis of the Existing Wells

Well data analysis is a module that feeds the modeling description with stochastic parameters, either uniform/normal distributions or probability density functions, derived at existing wells. This module is accessed by clicking on the icon when defining a ***Random property*** or on the P(x) icon when defining a ***PDF property***.



Random property

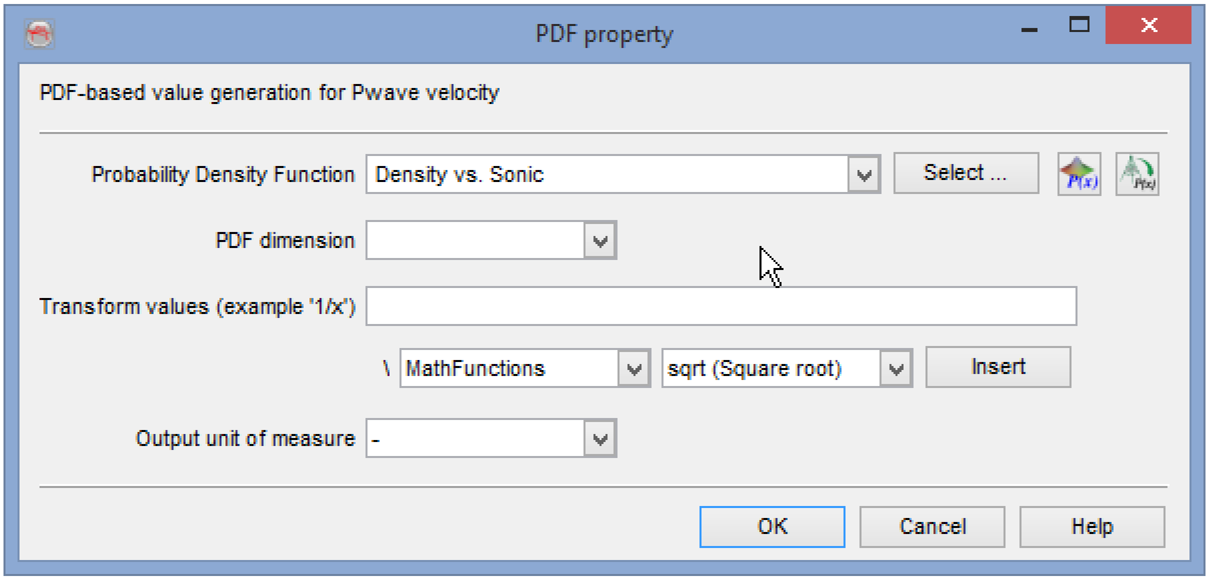
Random value generation for Thickness in Meter

Distribution type ○ Uniform ● Normal (Gaussian)

Expectation/Std deviation 2.4 0.3

☐ Hard minimum

☐ Hard maximum



PDF property

PDF-based value generation for Pwave velocity

Probability Density Function: Density vs. Sonic

PDF dimension:

Transform values (example '1/x'):

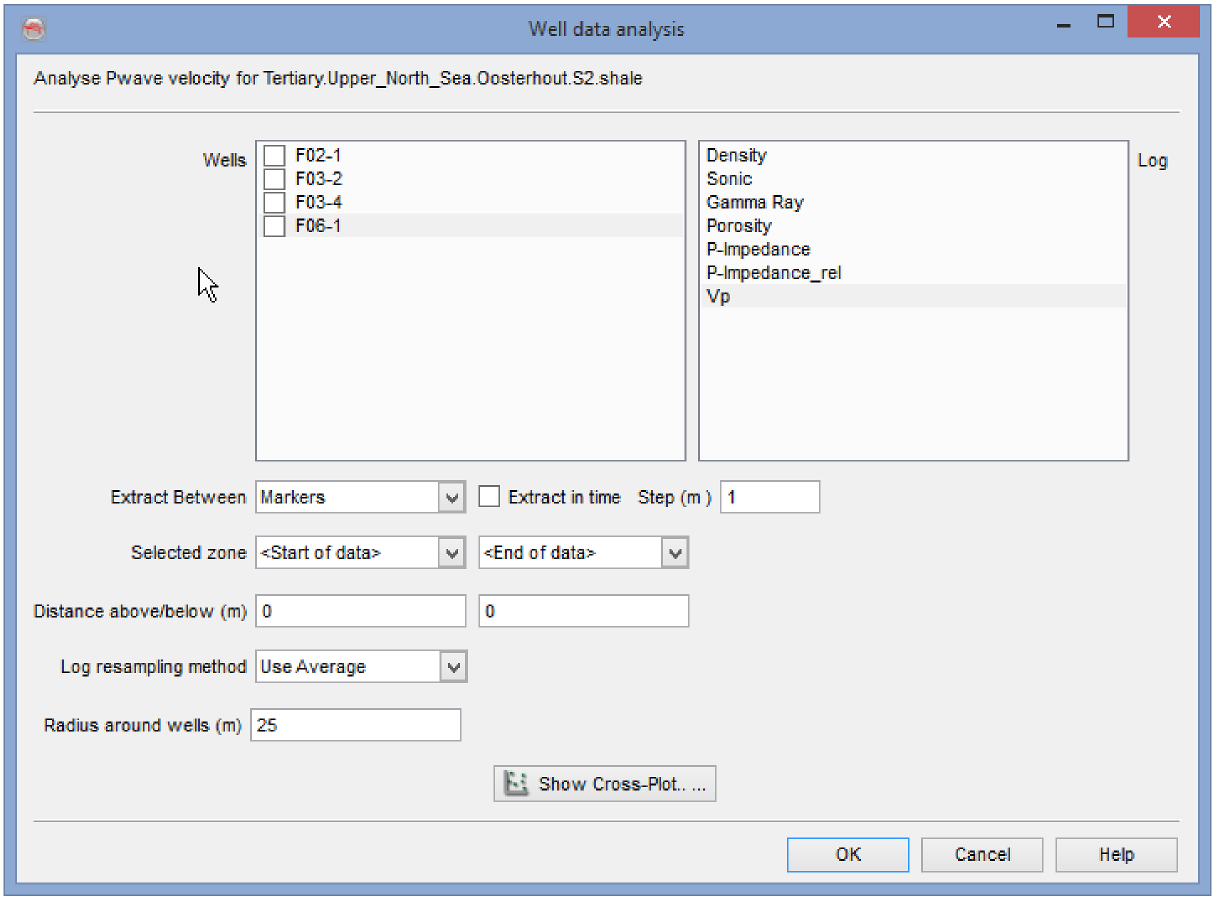
\ MathFunctions, sqrt (Square root)

Output unit of measure: -

Well data analysis window looks very similar to *Cross-plot tool* in both cases and have many similar options and features.

Random property

For a *Random property*, a single log (Density in the example below) is selected and analysed in the specified interval.



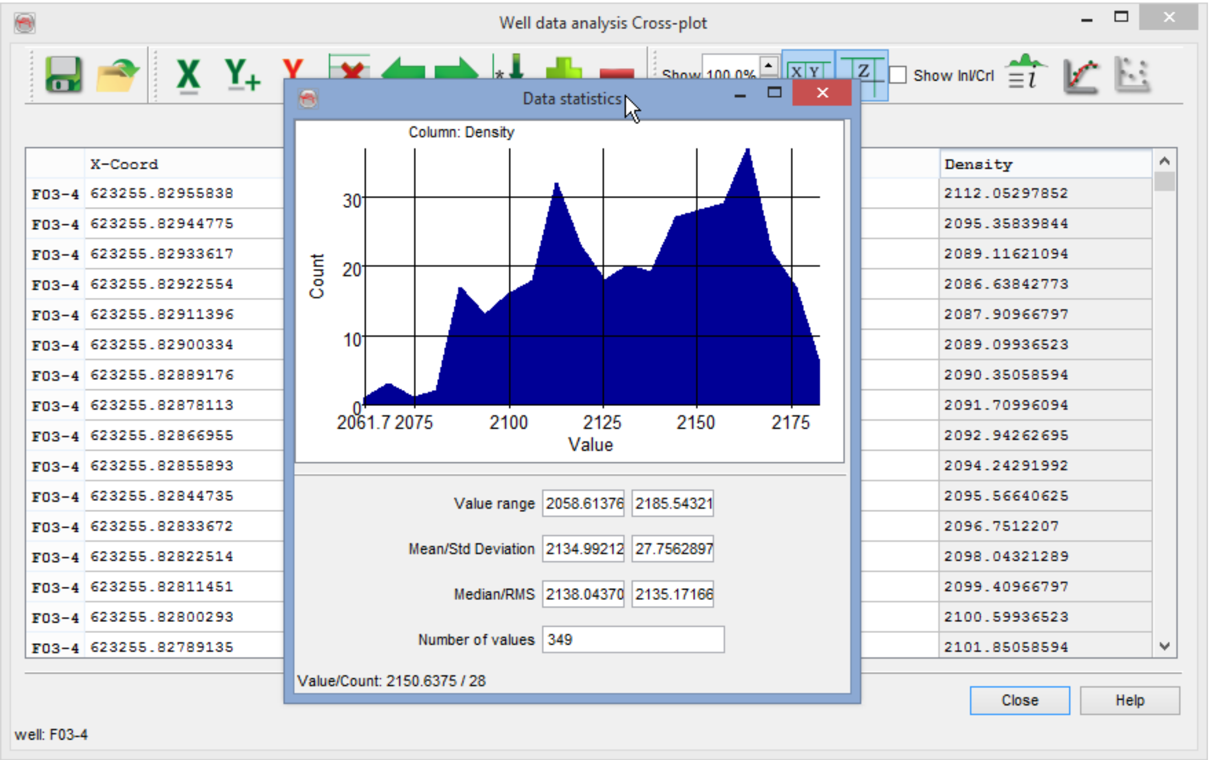
Analyse Pwave velocity for Tertiary.Upper_North_Sea.Oosterhout.S2.shale

Wells
☐ F02-1
☐ F03-2
☐ F03-4
☐ F06-1

Log
Density
Sonic
Gamma Ray
Porosity
P-Impedance
P-Impedance_rel
Vp

Extract Between: Markers
☐ Extract in time
Step (m): 1
Selected zone: <Start of data> <End of data>
Distance above/below (m): 0 0
Log resampling method: Use Average
Radius around wells (m): 25

It is highly recommended, even though not required, to click on Show Cross-Plot button in order to extract a table, display and edit the data if necessary. For example, viewing a log histogram by clicking on the icon in the cross-plot table window can help to determine if a given property follows Gaussian or Uniform distributions in a certain interval and refine the automatically derived parameters:



	X-Coord	Density
F03-4	623255.82955838	2112.05297852
F03-4	623255.82944775	2095.35839844
F03-4	623255.82933617	2089.11621094
F03-4	623255.82922554	2086.63842773
F03-4	623255.82911396	2087.90966797
F03-4	623255.82900334	2089.09936523
F03-4	623255.82889176	2090.35058594
F03-4	623255.82878113	2091.70996094
F03-4	623255.82866955	2092.94262695
F03-4	623255.82855893	2094.24291992
F03-4	623255.82844735	2095.56640625
F03-4	623255.82833672	2096.7512207
F03-4	623255.82822514	2098.04321289
F03-4	623255.82811451	2099.40966797
F03-4	623255.82800293	2100.59936523
F03-4	623255.82789135	2101.85058594

Column: Density

Value range 2058.61376 2185.54321

Mean/Std Deviation 2134.99212 27.7562897

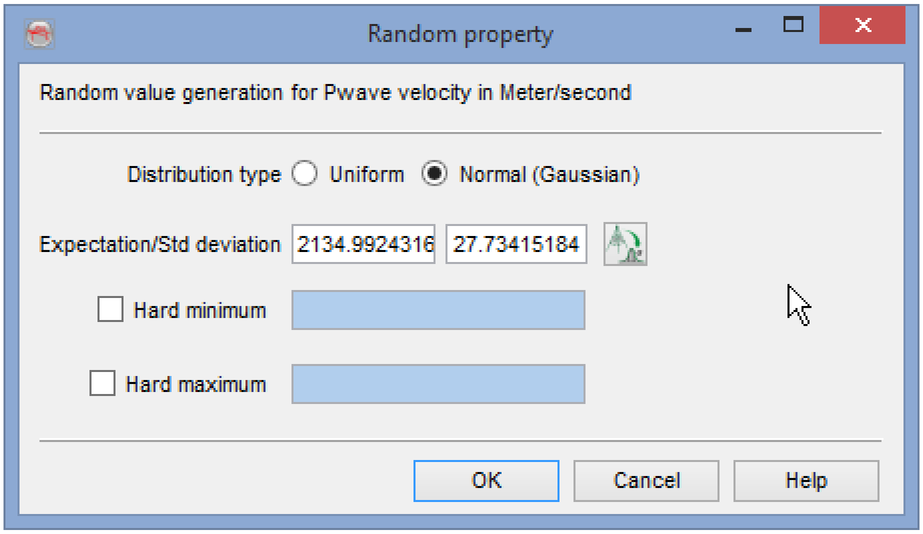
Median/RMS 2138.04370 2135.17166

Number of values 349

Value/Count: 2150.6375 / 28

well: F03-4

Upon OK click in the *Well data analysis* window, the derived distribution parameters are automatically fed to the*Random property* window:



Random value generation for Pwave velocity in Meter/second

Distribution type ○ Uniform ● Normal (Gaussian)

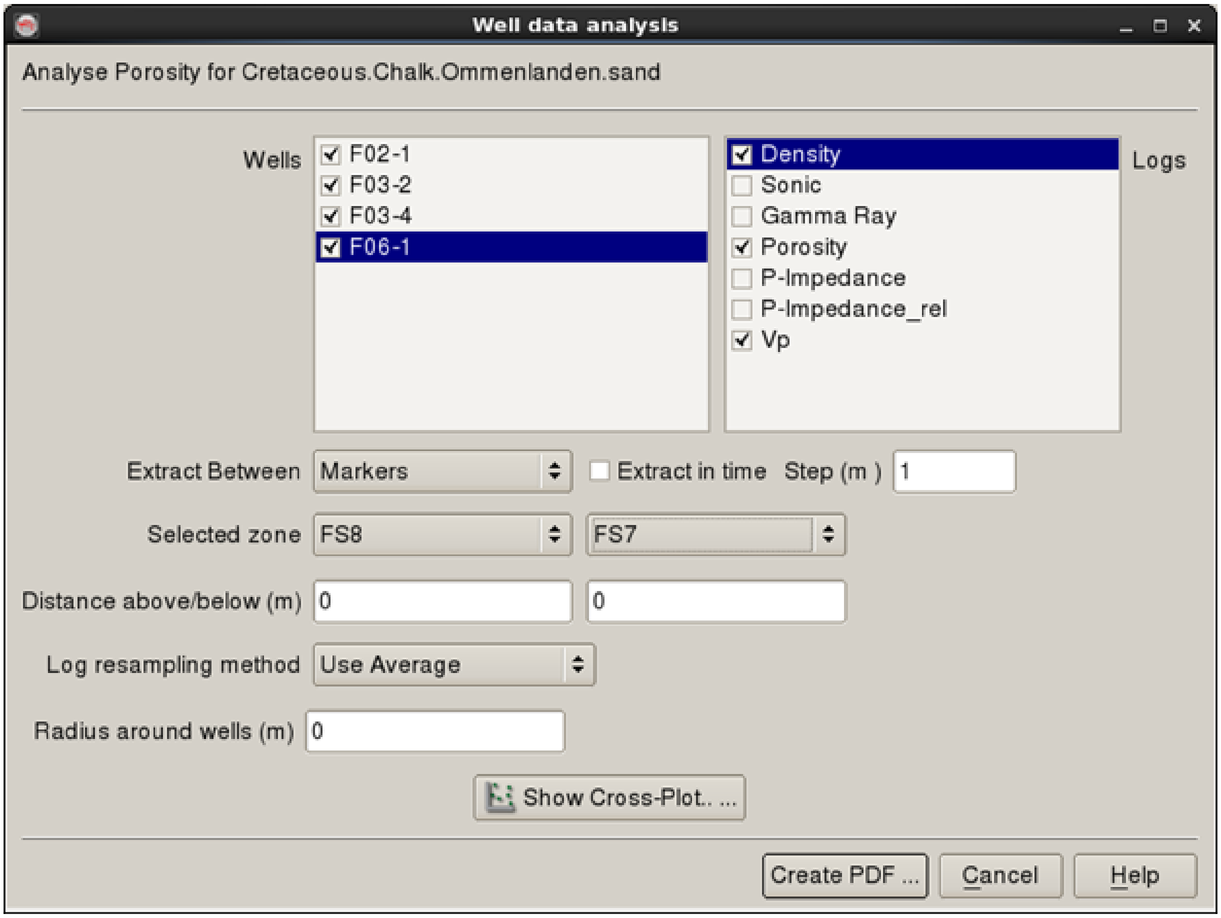
Expectation/Std deviation 2134.9924316 27.73415184

☐ Hard minimum

☐ Hard maximum

PDF property

For a PDF property, several logs can be selected and analysed in the specified interval.



Well data analysis

Analyse Porosity for Cretaceous.Chalk.Ommenlanden.sand

Wells
- ☑ F02-1
- ☑ F03-2
- ☑ F03-4
- ☑ F06-1

Logs
- ☑ Density
- ☐ Sonic
- ☐ Gamma Ray
- ☑ Porosity
- ☐ P-Impedance
- ☐ P-Impedance_rel
- ☑ Vp

Extract Between: Markers ☐ Extract in time Step (m): 1

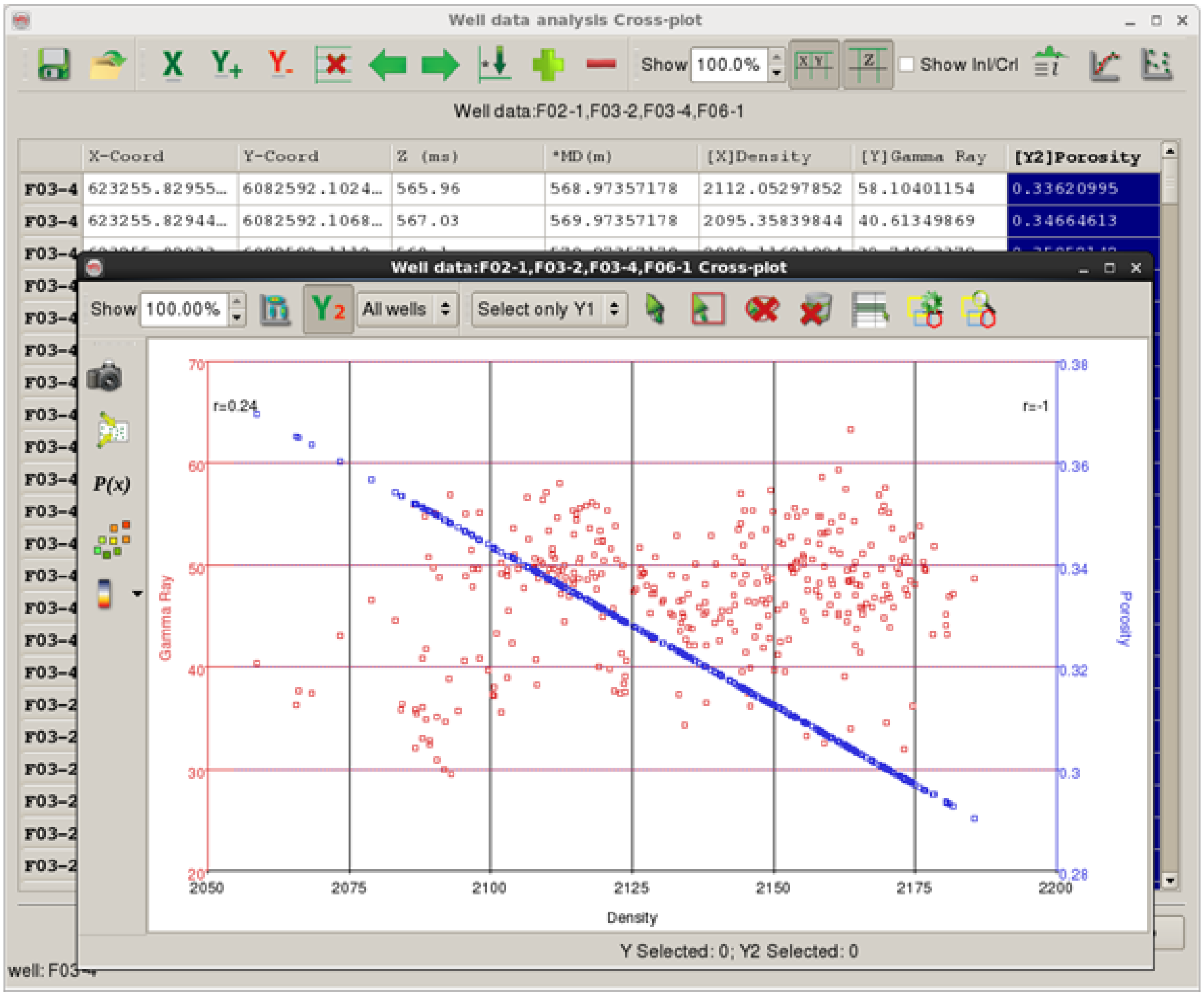
Selected zone: FS8 FS7

Distance above/below (m): 0 0

Log resampling method: Use Average

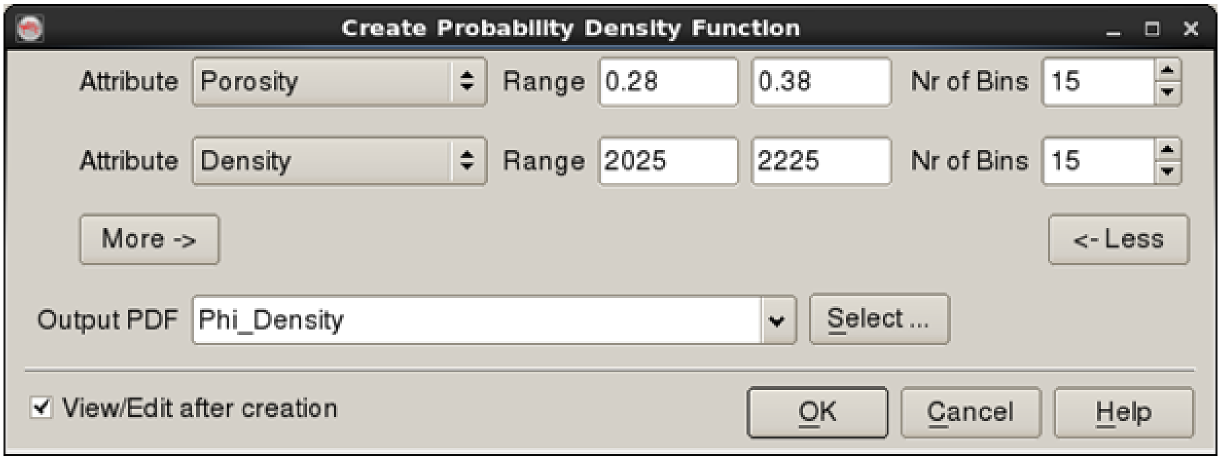
Radius around wells (m): 0

It is highly recommended, but not required, to click on Show Cross-Plot button in order to extract a table and graph properties of interest by clicking on the icon which can help to modify automatically determined PDF parameters:



	X-Coord	Y-Coord	Z (ms)	*MD (m)	[X]Density	[Y]Gamma Ray	[Y2]Porosity
F03-4	623255.82955…	6082592.1024…	565.96	568.97357178	2112.05297852	58.10401154	0.33620995
F03-4	623255.82944…	6082592.1068…	567.03	569.97357178	2095.35839844	40.61349869	0.34664613

After clicking on the *Create PDF* button in *Well data analysis* window, the Create PDF window allows to define the number of dimensions, a property for each dimension, their ranges and number of bins. Remember that only up to three properties can be stored in a discrete PDF. Wells commonly contain a small number of data points, it is therefore recommended to check *View/Edit* after creation option.


Create Probability Density Function

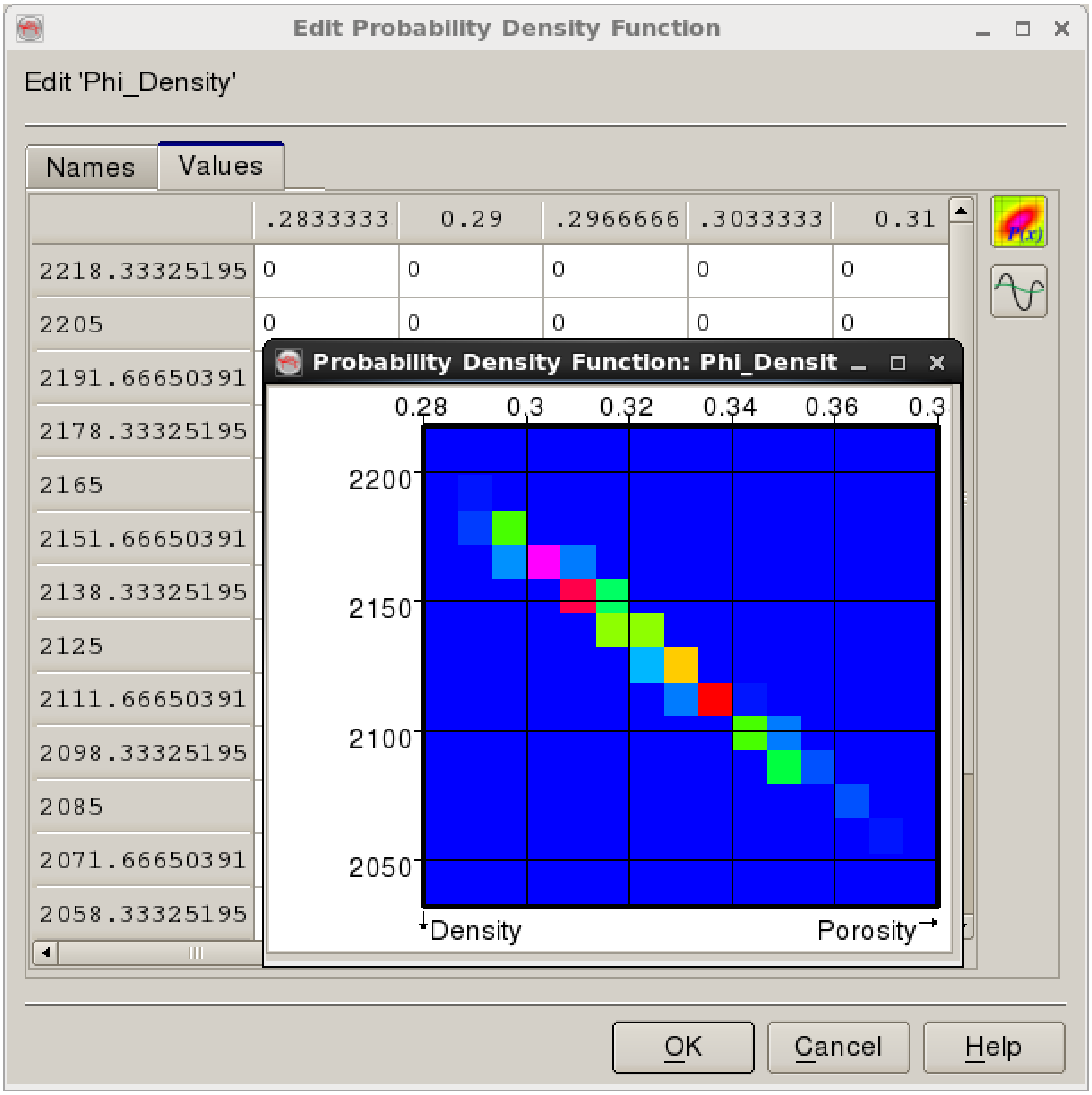
Attribute Porosity Range 0.28 0.38 Nr of Bins 15

Attribute Density Range 2025 2225 Nr of Bins 15

Output PDF Phi_Density

☑ View/Edit after creation

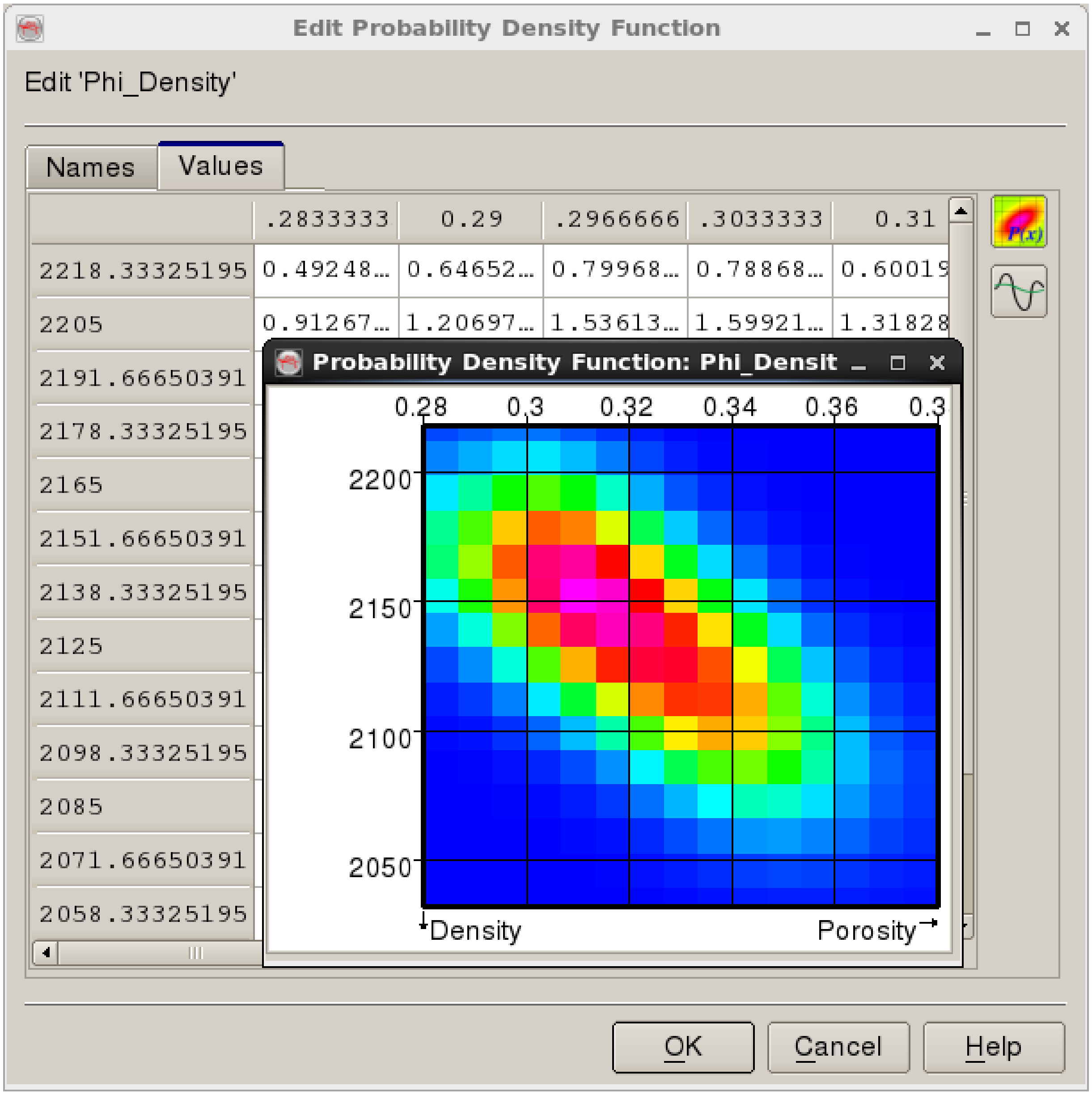
It allows to browse and edit a PDF in a table, as well as display it graphically by clicking on the P(x) icon.



	.2833333	0.29	.2966666	.3033333	0.31
2218.33325195	0	0	0	0	0
2205	0	0	0	0	0
2191.66650391					
2178.33325195					
2165					
2151.66650391					
2138.33325195					
2125					
2111.66650391					
2098.33325195					
2085					
2071.66650391					
2058.33325195					

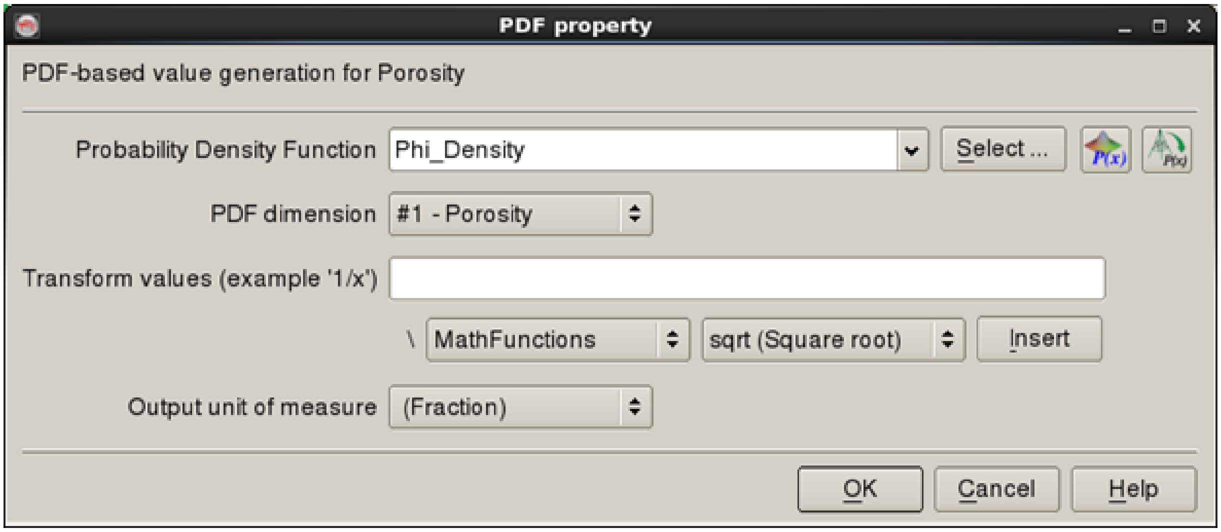
Editing of a PDF can be done manually in a table or smoothed by clicking on the ∿ icon. For other details check the Manage PDF section of the OpendTect User Documentation

.



	.2833333	0.29	.2966666	.3033333	0.31
2218.33325195	0.49248...	0.64652...	0.79968...	0.78868...	0.60019
2205	0.91267...	1.20697...	1.53613...	1.59921...	1.31828
2191.66650391					
2178.33325195					
2165					
2151.66650391					
2138.33325195					
2125					
2111.66650391					
2098.33325195					
2085					
2071.66650391					
2058.33325195					

Upon clicking OK button in either the*Create PDF* or *Edit PDF* window, the derived PDF is automatically fed to the PDF property window. Stored PDF can have multiple dimensions, thus you will need to select what dimension to use. Consequently, a single PDF can be used to provide the input for multiple quantities. Also, a transformation can be applied on the values stored in the PDF: for example, to simulate sonic when the PDF contains P-wave, or to simulate log(K) when the distribution contains K (permeability).



PDF property

PDF-based value generation for Porosity

Probability Density Function: Phi_Density

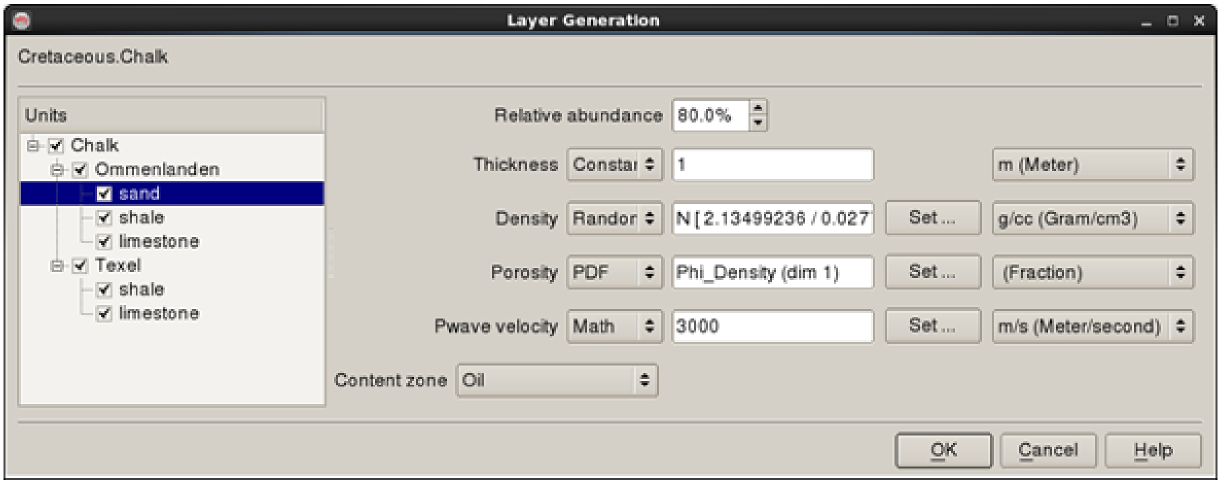
PDF dimension: #1 - Porosity

Transform values (example '1/x'):

\ MathFunctions — sqrt (Square root)

Output unit of measure: (Fraction)

Finally, confirmation by clicking OK button in either the *Random property* or *PDF property* windows updates the corresponding fields in *Layer Generation window* as shown below for Density and Porosity.



Layer Generation

Cretaceous.Chalk

Units
- ☑ Chalk
  - ☑ Ommenlanden
    - ☑ sand
    - ☑ shale
    - ☑ limestone
  - ☑ Texel
    - ☑ shale
    - ☑ limestone

Relative abundance: 80.0%

Thickness: Constai — 1 — m (Meter)

Density: Randor — N [2.13499236 / 0.027 — g/cc (Gram/cm3)

Porosity: PDF — Phi_Density (dim 1) — (Fraction)

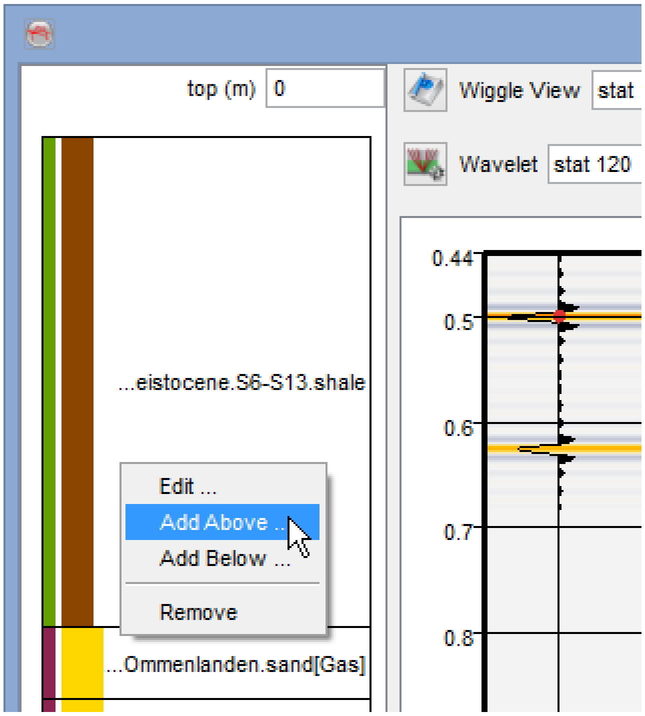
Pwave velocity: Math — 3000 — m/s (Meter/second)

Content zone: Oil

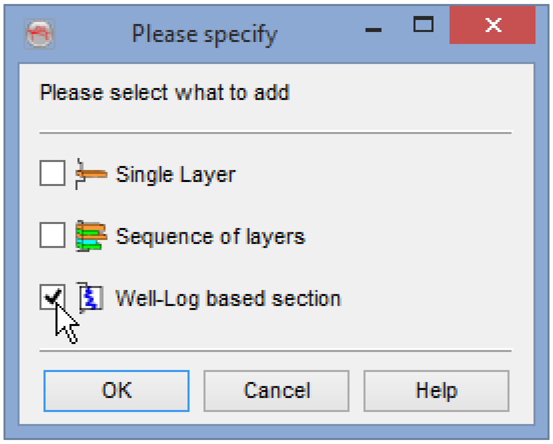
Well-Log - based Generation

Well-log based section can be either full or partly full of well log data, which has been upscaled/blocked at a constant size. This option is recommended for modeling a realistic overburden which is required for ray-tracing when working with pre-stack data.

Right-clicking inside the *Layer Description* panel of the *Layer Modeling* window, will bring up the following popup:



Clicking either '*Add Above...*' or '*Add Below...*' will bring up the following selection option:



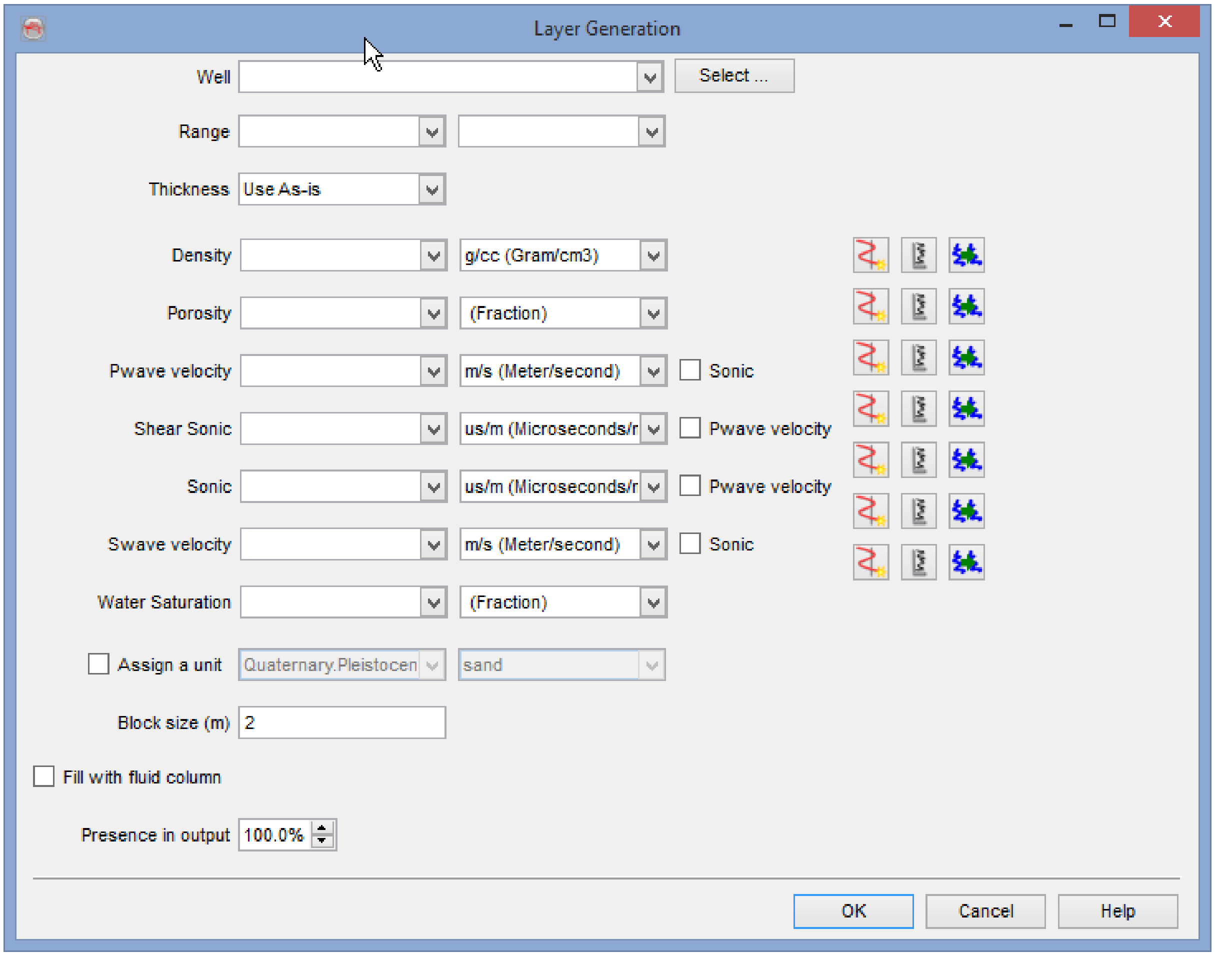
Please select what to add

☐ Single Layer

☐ Sequence of layers

☑ Well-Log based section

Choosing '*Well-Log based section*' will bring you here:



Well

Range

Thickness: Use As-is

Density: g/cc (Gram/cm3)

Porosity: (Fraction)

Pwave velocity: m/s (Meter/second) ☐ Sonic

Shear Sonic: us/m (Microseconds/r ☐ Pwave velocity

Sonic: us/m (Microseconds/r ☐ Pwave velocity

Swave velocity: m/s (Meter/second) ☐ Sonic

Water Saturation: (Fraction)

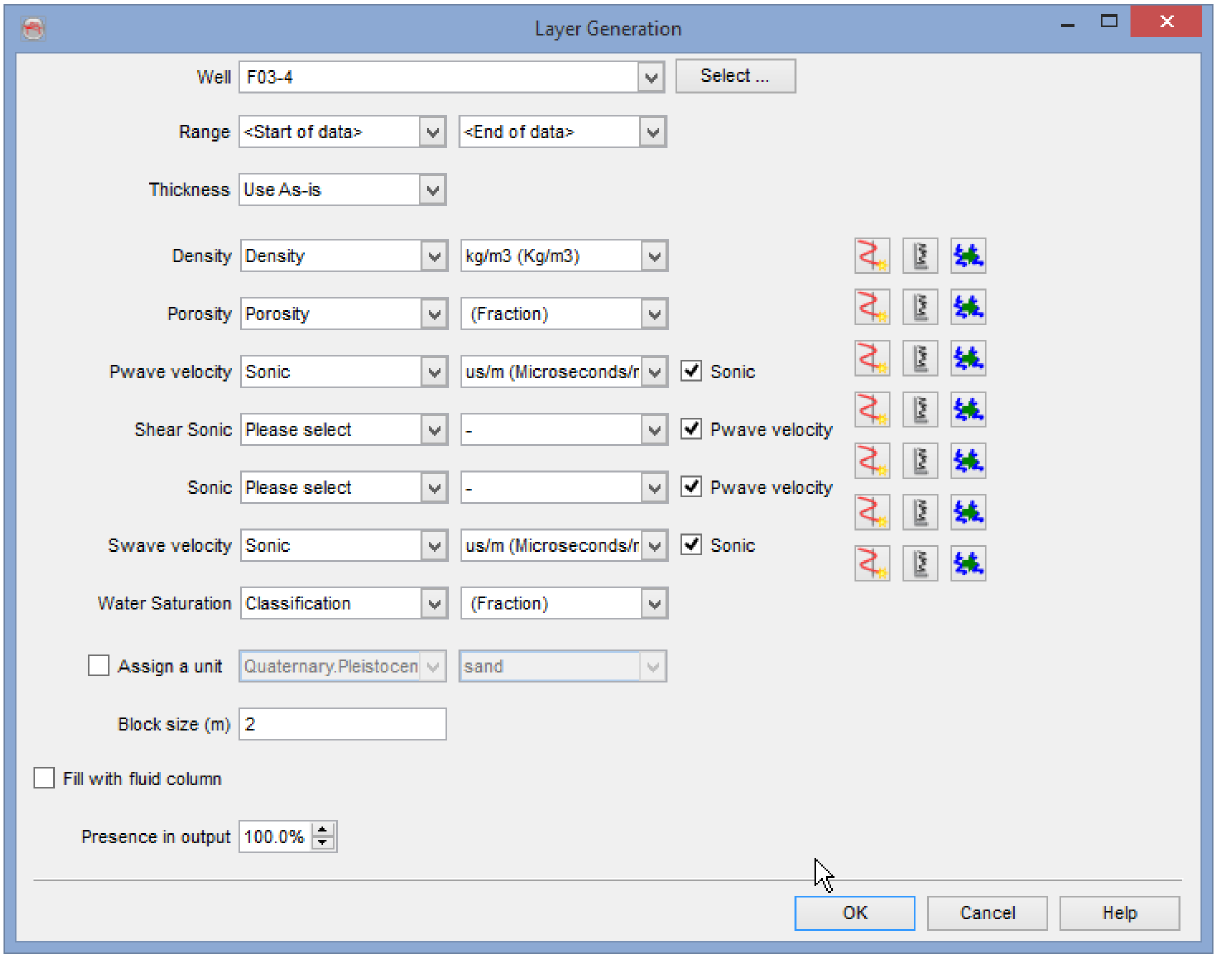
☐ Assign a unit: Quaternary.Pleistocen sand

Block size (m): 2

☐ Fill with fluid column

Presence in output: 100.0%

And selecting a well will allow auto-input of any available logs. Well-log based section can be either full or part of well log data, which has been upscaled/blocked at a constant size. This option is recommended for modeling a realistic overburden which is required for ray-tracing when working with pre-stack data. See following example for F03-4:



Layer Generation

Well: F03-4 | Select ...

Range: <Start of data> | <End of data>

Thickness: Use As-is

Density	Density	kg/m3 (Kg/m3)	
Porosity	Porosity	(Fraction)	
Pwave velocity	Sonic	us/m (Microseconds/r	☑ Sonic
Shear Sonic	Please select	-	☑ Pwave velocity
Sonic	Please select	-	☑ Pwave velocity
Swave velocity	Sonic	us/m (Microseconds/r	☑ Sonic
Water Saturation	Classification	(Fraction)	

☐ Assign a unit: Quaternary.Pleistocen | sand

Block size (m): 2

☐ Fill with fluid column

Presence in output: 100.0%

The example above shows Layer Generation window for adding a well-log based section to a model description. Once the well is selected, define the interval bound by formation tops to be used. Thickness of the modeled interval can be Used as-is, or modified using one of the three options Cut from Top, Cut from Bottom or Stretch/Squeeze. If modified it can be set to a Constant, Range or Random (Uniform or Gaussian distribution). For example, Cut from Top in conjunction with Range or Random can be used to model subcropping strata, and Stretch/Squeeze along with Range or Random - to model thickness variations.

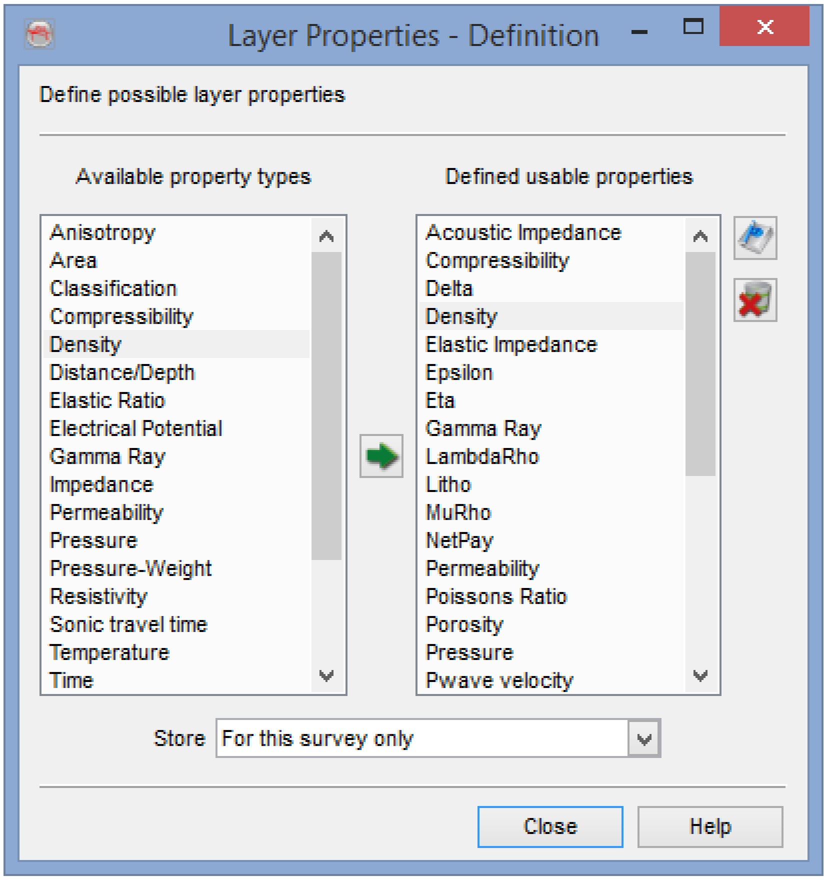
The next section of this window allows to choose well logs corresponding to the layer properties used in the modeling. Note that all used properties are required to have a log selected. Missing logs can be either created ( ) without having to exit SynthRock or imported using the Well Manager. Selected log can be viewed ( ), or changed ( ).

Optionally, a well-log section can be assigned to a specific unit by selecting both formation and lithology member. If no unit is assigned, a name derived from bounding formation tops is displayed in the modeling description part of the Layer modeling window, but an actual layer name as well as lithology are left undefined ('-'). Default colors for undefined formations and lithologies are white and grey respectively. Note that if no unit name is assigned, the subsequently created layers can not be used for plotting Attributes vs model properties or be a target quantity during the HitCube inversion.
All the logs are upscaled/blocked at the user-defined Block size. Note, that when modeling large intervals, Block size should be chosen with care as block sizes which are too small significantly increase time to generate pseudowells and memory consumption.

If Fill with fluid column is checked, the fluid column has to be ***Defined***. Presence in output controls the percentage of wells having the given well-log based section.

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).



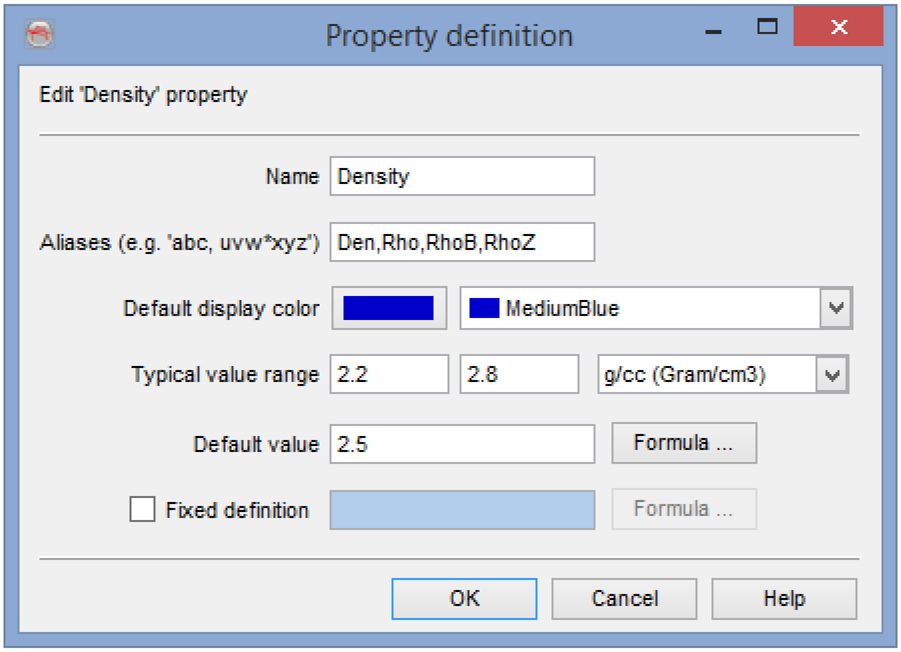
Store: For this survey only

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.



Property definition

Edit 'Density' property

Name: Density

Aliases (e.g. 'abc, uvw*xyz'): Den,Rho,RhoB,RhoZ

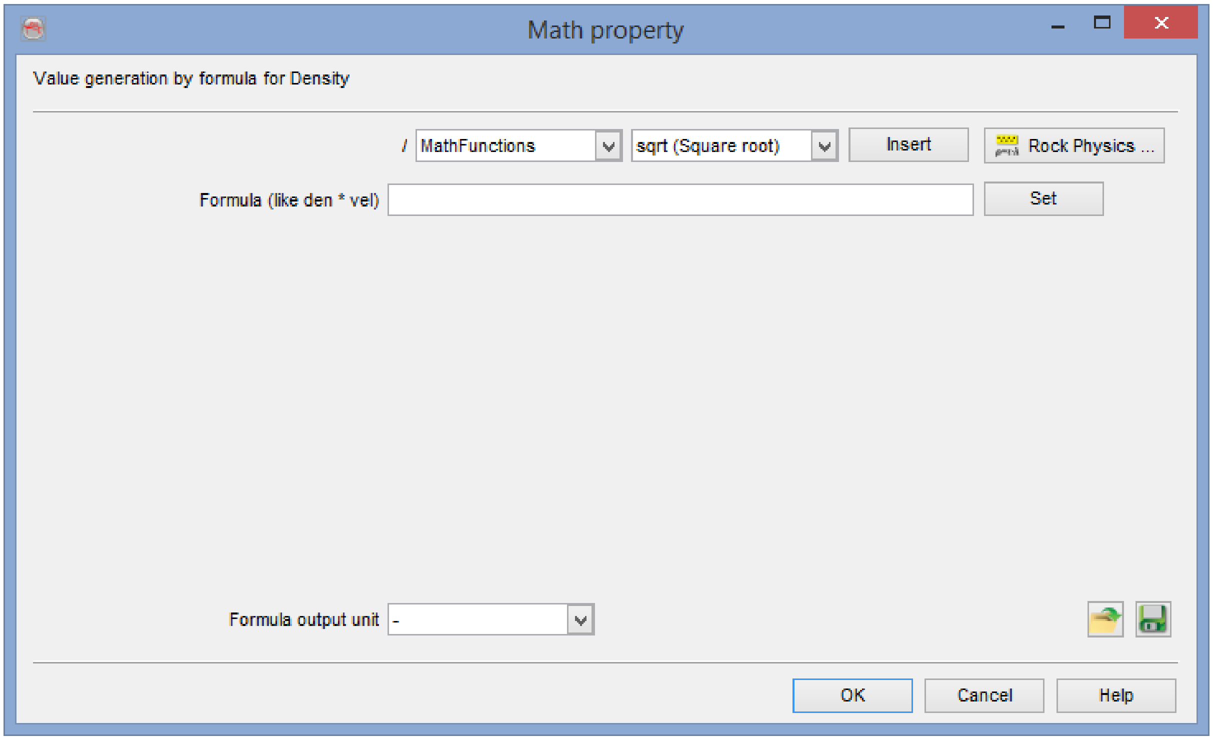
Default display color: MediumBlue

Typical value range: 2.2 2.8 g/cc (Gram/cm3)

Default value: 2.5

☐ Fixed definition

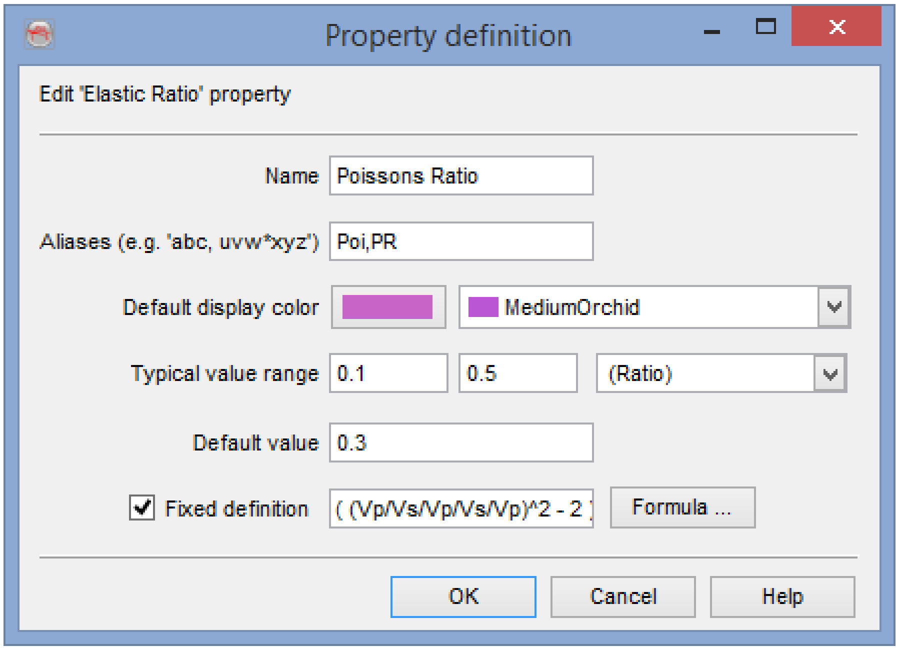
Optionally, the default value of any property can be defined by a specific equation that can either be typed in or retrieved from a list of saved formulas on clicking on *Formula*... Further formulas and equations may be utilized from the RockPhysics library, reached via the $\rho=\tau\cdot\lambda$ icon.



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:



Edit 'Elastic Ratio' property

Name: Poissons Ratio

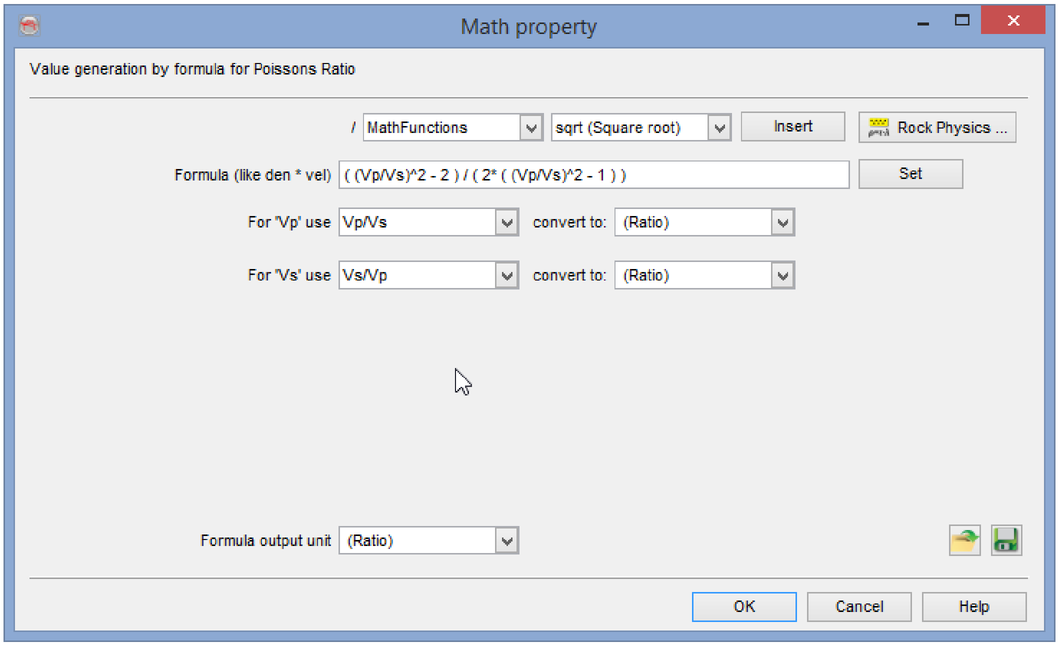
Aliases (e.g. 'abc, uvw*xyz'): Poi,PR

Default display color: MediumOrchid

Typical value range: 0.1 0.5 (Ratio)

Default value: 0.3

☑ Fixed definition: ((Vp/Vs/Vp/Vs/Vp)^2 - 2



Value generation by formula for Poissons Ratio

/ MathFunctions sqrt (Square root)

Formula (like den * vel): ((Vp/Vs)^2 - 2) / (2* ((Vp/Vs)^2 - 1))

For 'Vp' use: Vp/Vs convert to: (Ratio)

For 'Vs' use: Vs/Vp convert to: (Ratio)

Formula output unit: (Ratio)

Storing levels

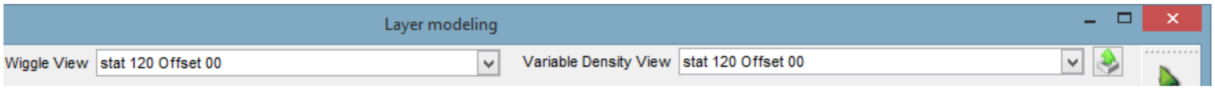
The level where it is saved is specified in the main window:

- 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.

Export Synthetic Dataset

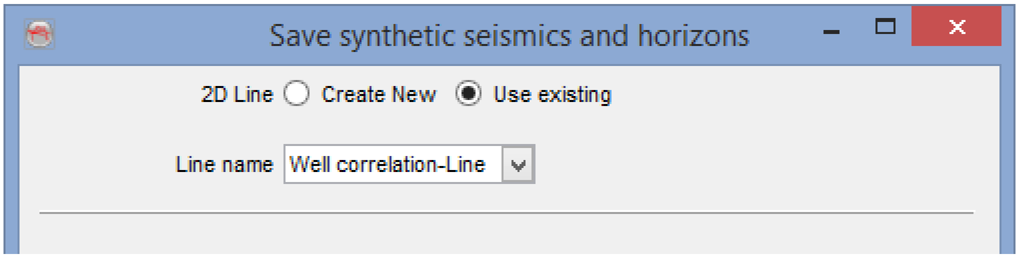
The synthetic seismic data (both post-stack and pre-stack), the layer property synthetics in Time (e.g. AI, Density etc.) and the stratigraphic levels/markers, from all modeling modules (i.e. Basic, Profile and Stochastic) can be exported along 2D lines. The stratigraphic levels/markers in the modeled 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).



Wiggle View: stat 120 Offset 00

Variable Density View: stat 120 Offset 00

Export of the synthetics can be done onto an already existing 2D line or a new line created on the fly.

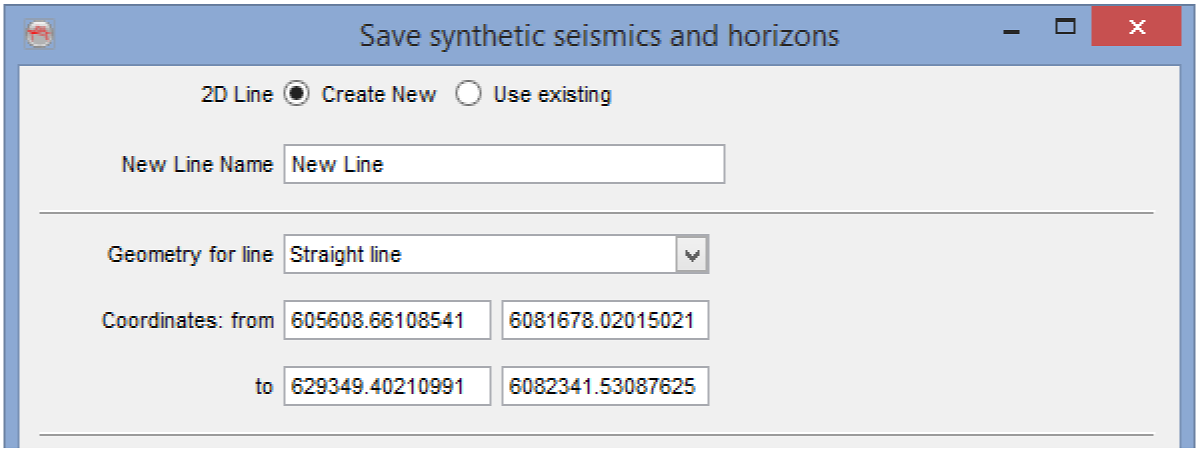


2D Line ○ Create New ● Use existing

Line name: Well correlation-Line

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.



Save synthetic seismics and horizons

2D Line ● Create New ○ Use existing

New Line Name: New Line

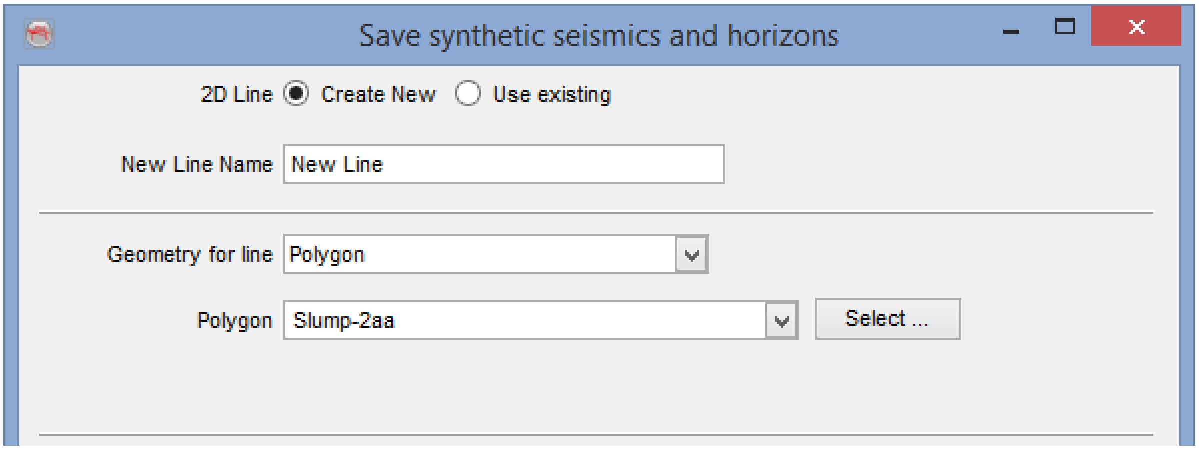
Geometry for line: Straight line

Coordinates: from 605608.66108541 6081678.02015021

to 629349.40210991 6082341.53087625

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.



Save synthetic seismics and horizons

2D Line ● Create New ○ Use existing

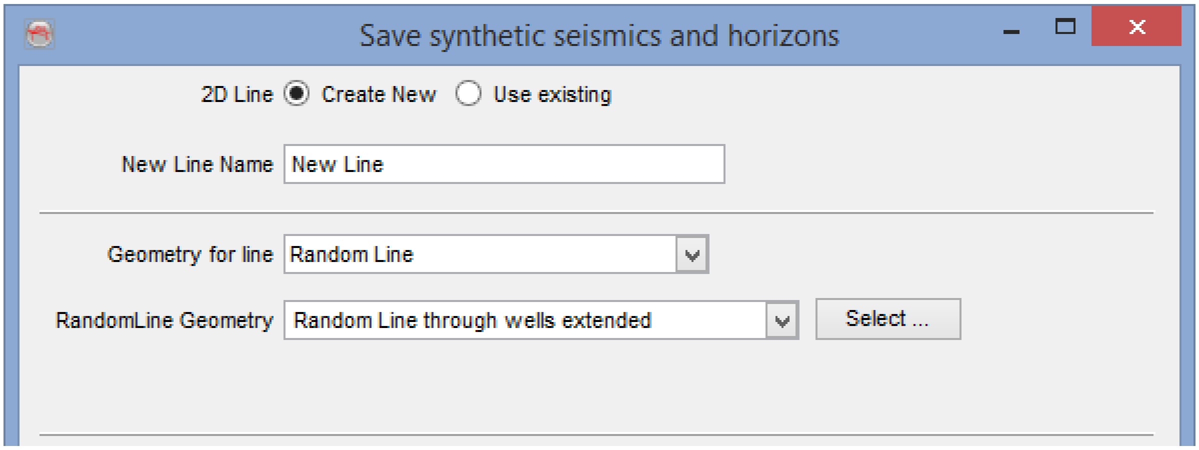
New Line Name: New Line

Geometry for line: Polygon

Polygon: Slump-2aa

Creating a 2D line along a polygon

Finally, the 2D line can also be created along an existing random line.



Save synthetic seismics and horizons

2D Line ● Create New ○ Use existing

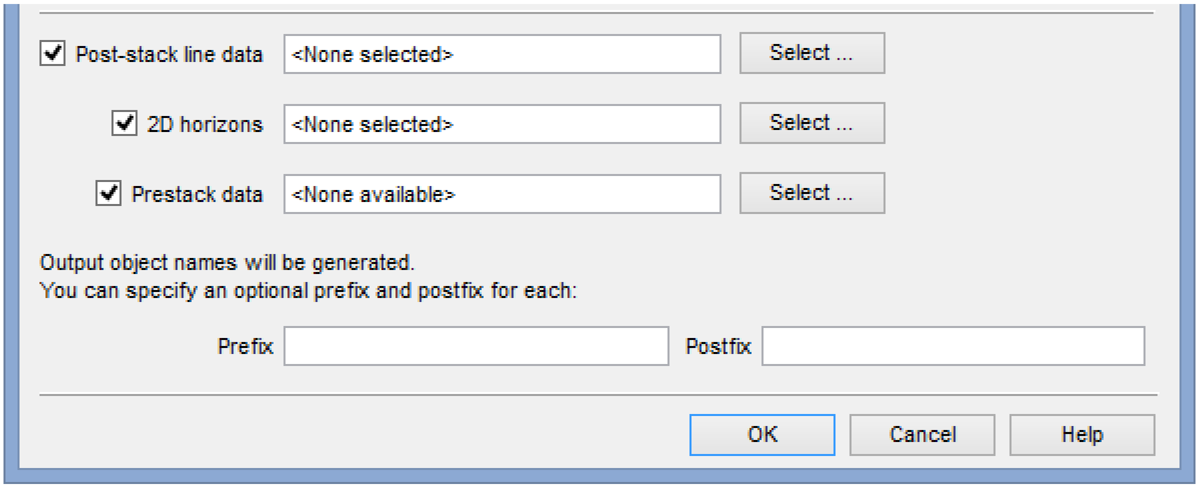
New Line Name: New Line

Geometry for line: Random Line

RandomLine Geometry: Random Line through wells extended

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.



☑ Post-stack line data: <None selected>

☑ 2D horizons: <None selected>

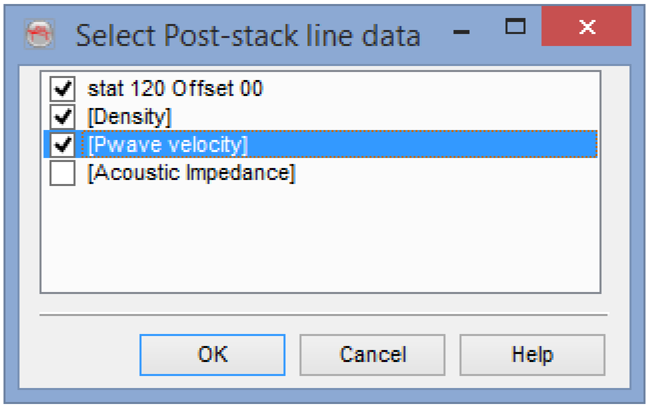
☑ Prestack data: <None available>

Output object names will be generated.
You can specify an optional prefix and postfix for each:

Prefix:

Postfix:

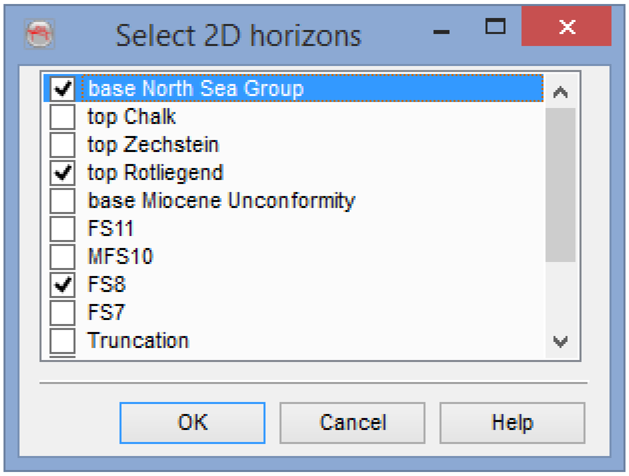
For post-stack data, user can select synthetic seismic and various layer property synthetics (e.g. Acoustic Impedance, Density etc.).



☑ stat 120 Offset 00
☑ [Density]
☑ [Pwave velocity]
☐ [Acoustic Impedance]

Post-stack data selection for export

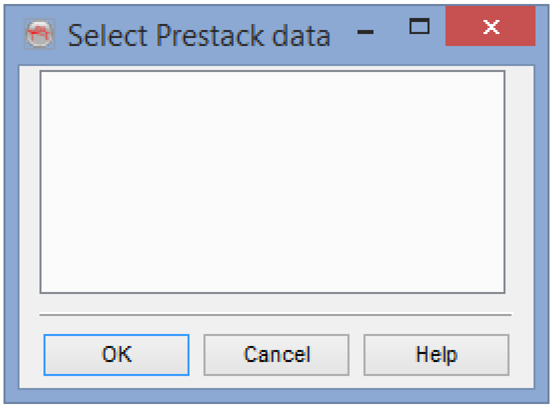
Similarly for 2D horizons, various levels present in the pseudo-wells can be selected.



☑ base North Sea Group
☐ top Chalk
☐ top Zechstein
☑ top Rotliegend
☐ base Miocene Unconformity
☐ FS11
☐ MFS10
☑ FS8
☐ FS7
☐ Truncation

2D horizon data selection for export

and finally (if any) pre-stack data can be selected.

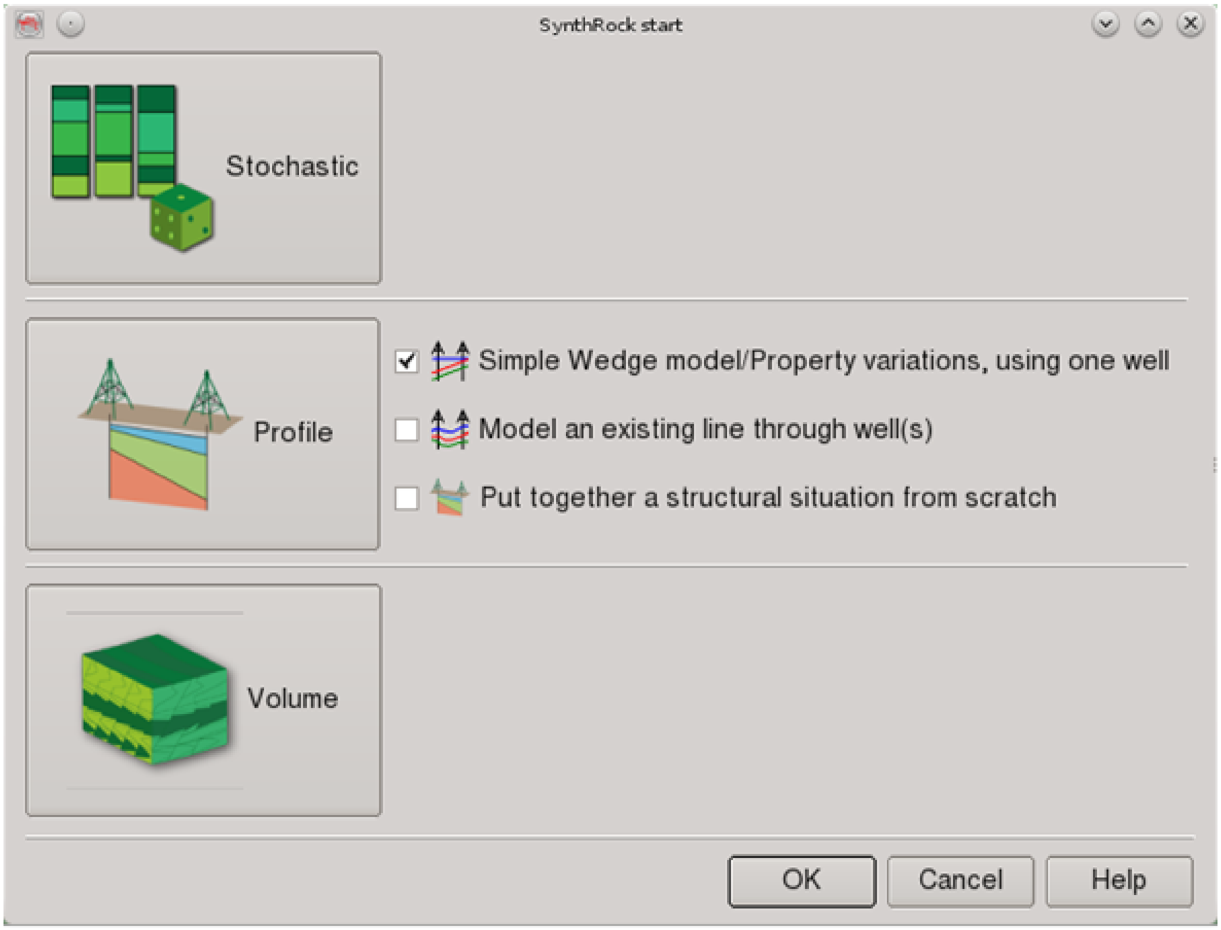


Pre-stack data selection for export

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.

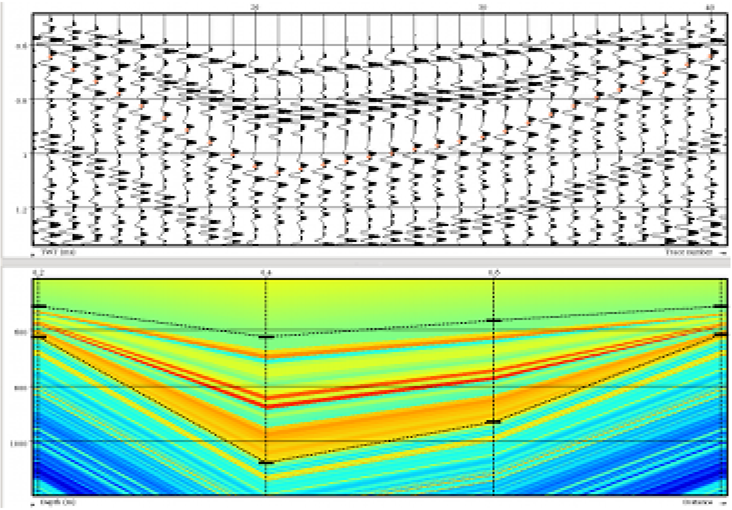
Profile Modeling

The profile modeling is used to create pseudowells by interpolating well logs between control points. The control points are inserted from real wells loaded in the project, with well-log data upscaled/blocked into layers of user-defined size, or added as control profiles during the modeling.



☑ Simple Wedge model/Property variations, using one well
☐ Model an existing line through well(s)
☐ Put together a structural situation from scratch

The modeling is performed in two steps by adding the control points and modifying the model parameters . An output example is given below. The modeling itself takes place in the bottom part of Layer modeling window. The modeling description can be saved, while interpolated pseudowells are not saved along with it. The pseudowells can be either dumped to a text file, or the corresponding data can be extracted using the crossplot extractor. The pseudowells are re-drawn by pressing on button. The number on the left of button defines the number of pseudowells to be created. It is advised to start with a relatively low number while building the model, and then increase it for the final application (several hundreds for a HitCube inversion are needed).



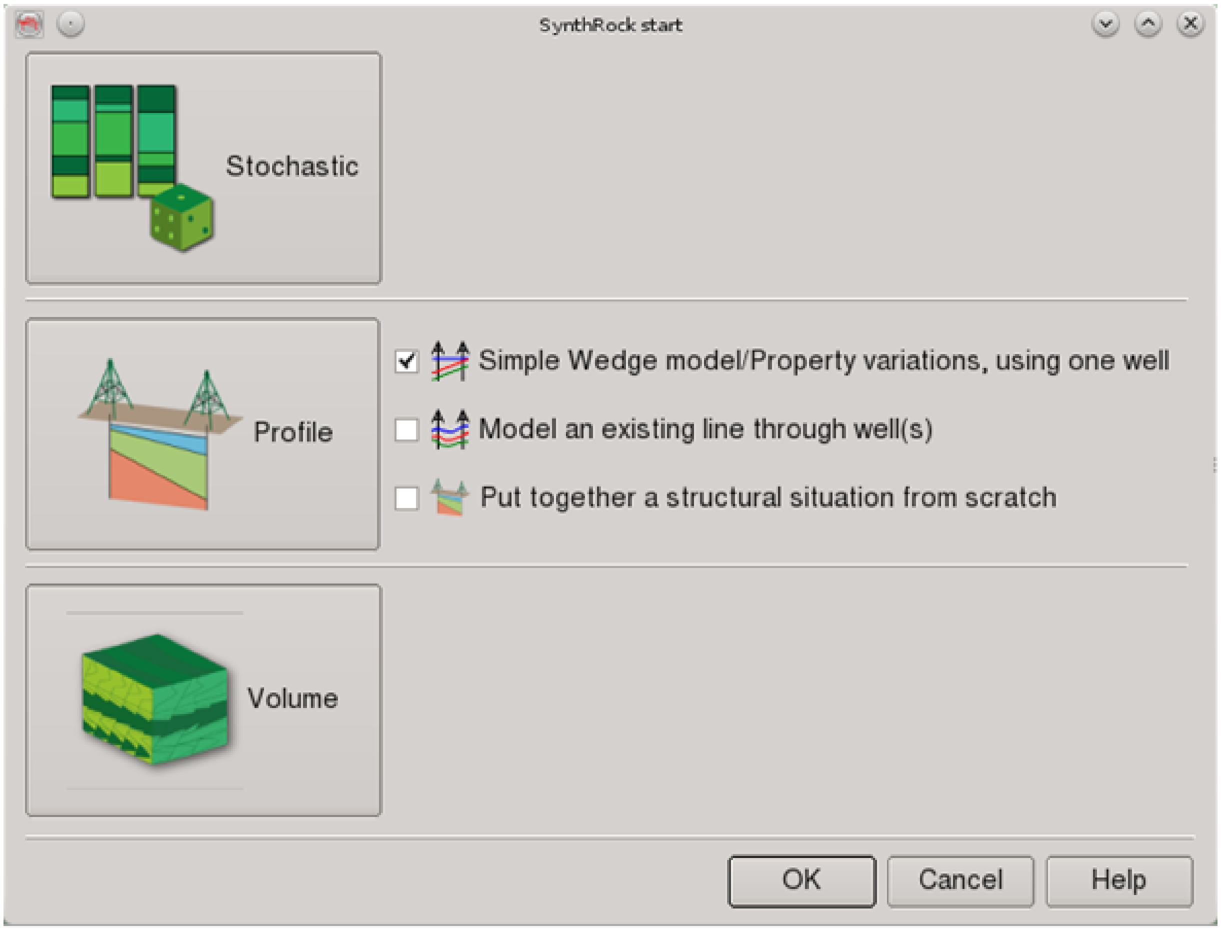
Profile modeling offers three options to start modeling.

Simple Wedge model/Property variations, using one well and Model an existing line through well(s) are two pre-defined workflows that allow user to quickly set up most commonly used models. These workflows are pre-defined in a sense that a user is guided through the required sequence of windows by the software. They allow to build simple models much faster than doing so from scratch. Once a simple model is built, further modeling can be continued using the classic Profile mode functionality in the Layer modeling window. Note, that some windows in the pre-defined workflows are unique and might not be accessible from Layer modeling window if accidentally closed by a user. In such case, a user either have to continue in the Layer modeling window with the classic Profile mode functionality, which allows to do the same things but requires more user input, or start the pre-defined workflow again.
The third option, the classic Profile mode, is accessable either by clicking on the large Profile icon or by selecting Put together a structural situation from scratch option and clicking OK.

- **Simple Wedge model/Property variations, using one well**
- **Model an existing line through well(s)**
- **Put together a structural situation from scratch**

Simple Wedge Model/Property Variations Using one Well

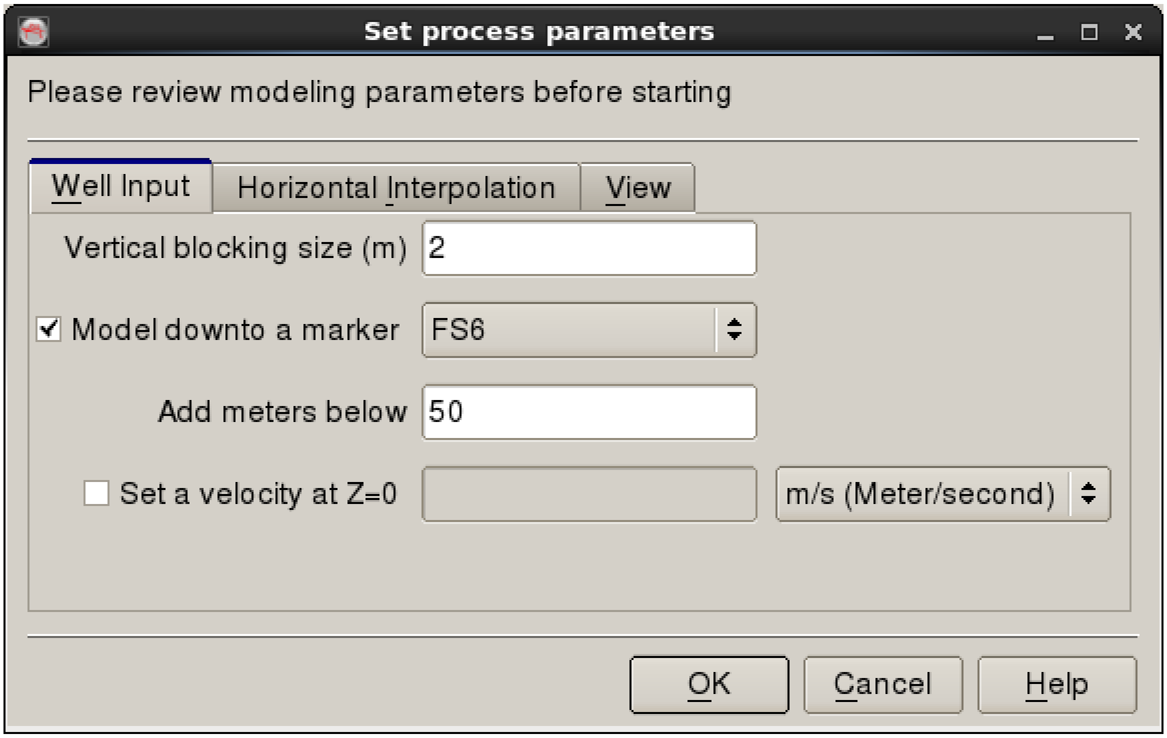
Simple Wedge model/Property variations, *using one well* is the first pre-defined workflow in SynthRock profile modeling which allows to quickly build a simple wedge model and/or model lateral property variations. It is launched by checking the corresponding box in *SynthRock* start window and clicking OK.



☑ Simple Wedge model/Property variations, using one well
☐ Model an existing line through well(s)
☐ Put together a structural situation from scratch

A user is expected to go through the pre-defined sequence of windows. The workflow starts, as any profile modeling in SynthRock, with setting modeling parameters in Set process parameters window and selecting properties to be modeled in Layer Properties - Selection window. Once the workflow is completed, both sets of parameters can be modified by clicking on and icons of the *Layer modeling window* respectively.

In the example below, few changes of the default parameters are made in the first window in order to limit the modeled interval.



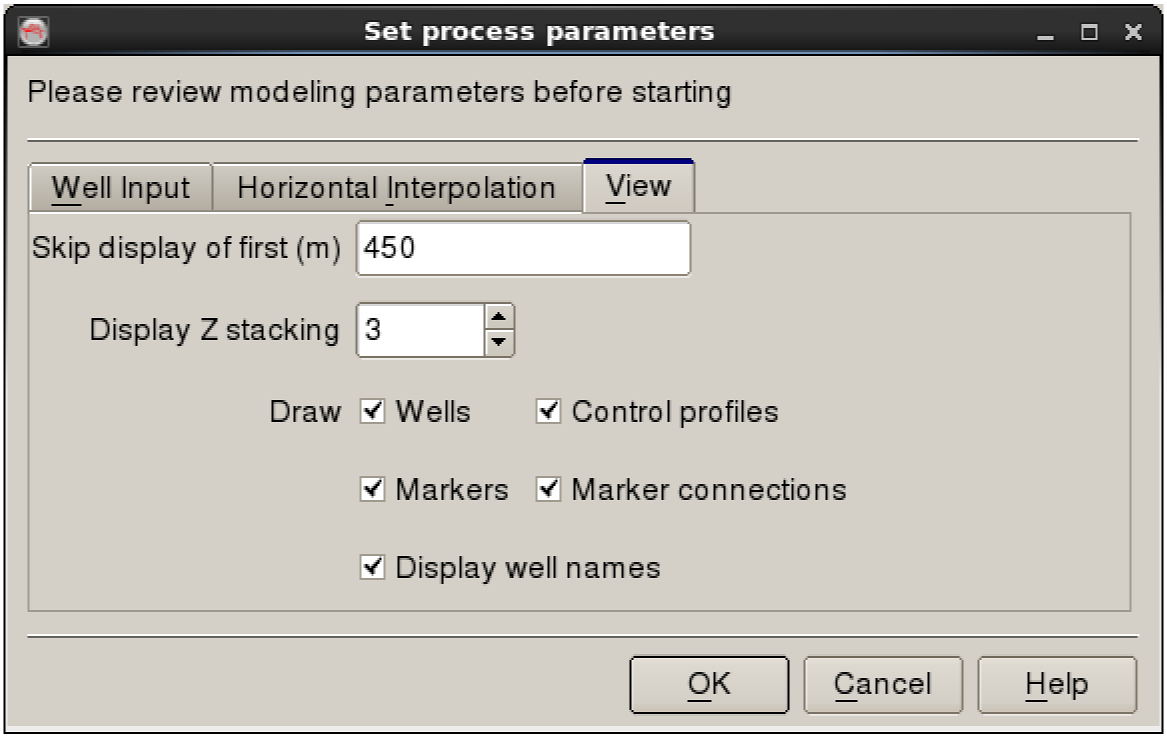
Please review modeling parameters before starting

Vertical blocking size (m) 2

☑ Model downto a marker FS6

Add meters below 50

☐ Set a velocity at Z=0 m/s (Meter/second)



Please review modeling parameters before starting

Skip display of first (m) 450

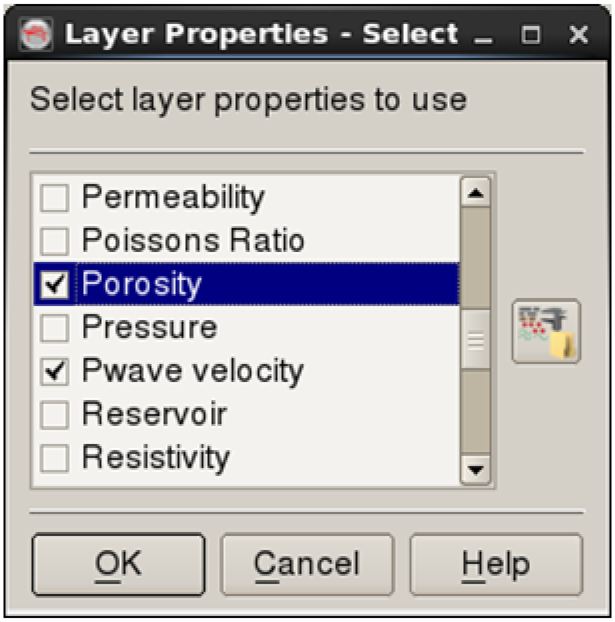
Display Z stacking 3

Draw ☑ Wells ☑ Control profiles

☑ Markers ☑ Marker connections

☑ Display well names

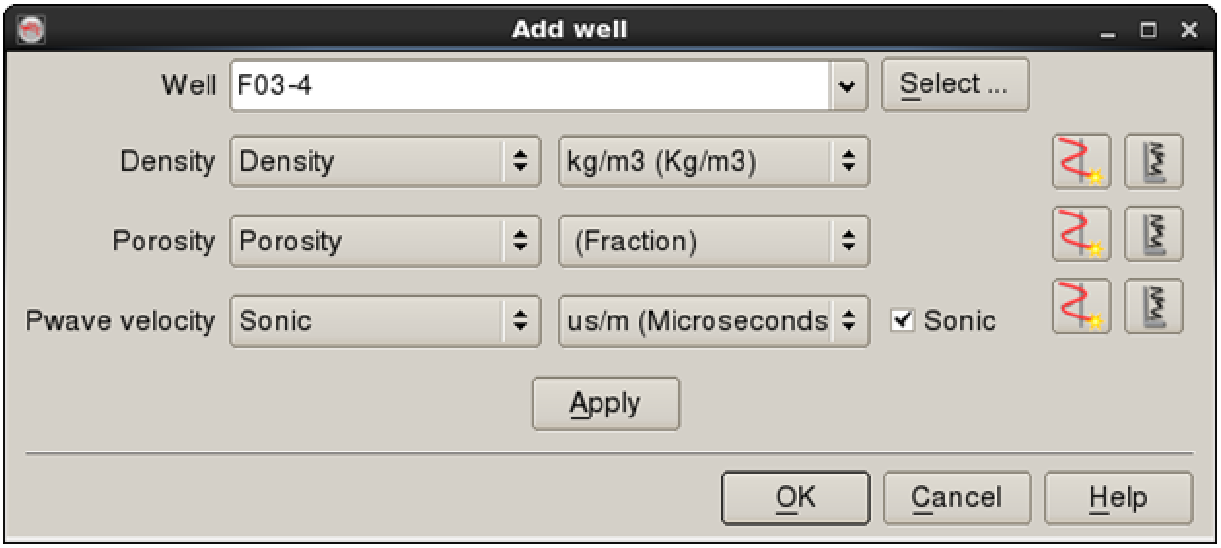
Porosity is added in addition to the default properties in the second window.



Select layer properties to use

- ☐ Permeability
- ☐ Poissons Ratio
- ☑ Porosity
- ☐ Pressure
- ☑ Pwave velocity
- ☐ Reservoir
- ☐ Resistivity

Once the modeling parameters and the property list are confirmed, Add well window allows to choose an existing well from a project database. In the example below *F03-4* is selected. An attempt to automatically determine well logs matching the modeled properties is made by the software, but it is recommended to always confirm the correct logs are selected. Note, that all properties chosen in *Layer Properties - Selection window* are required to have a log selected. Missing logs can be either created by clicking on the icon without having to exit SynthRock or imported using Well Manager. Selected log can be displayed by clicking on the icon.



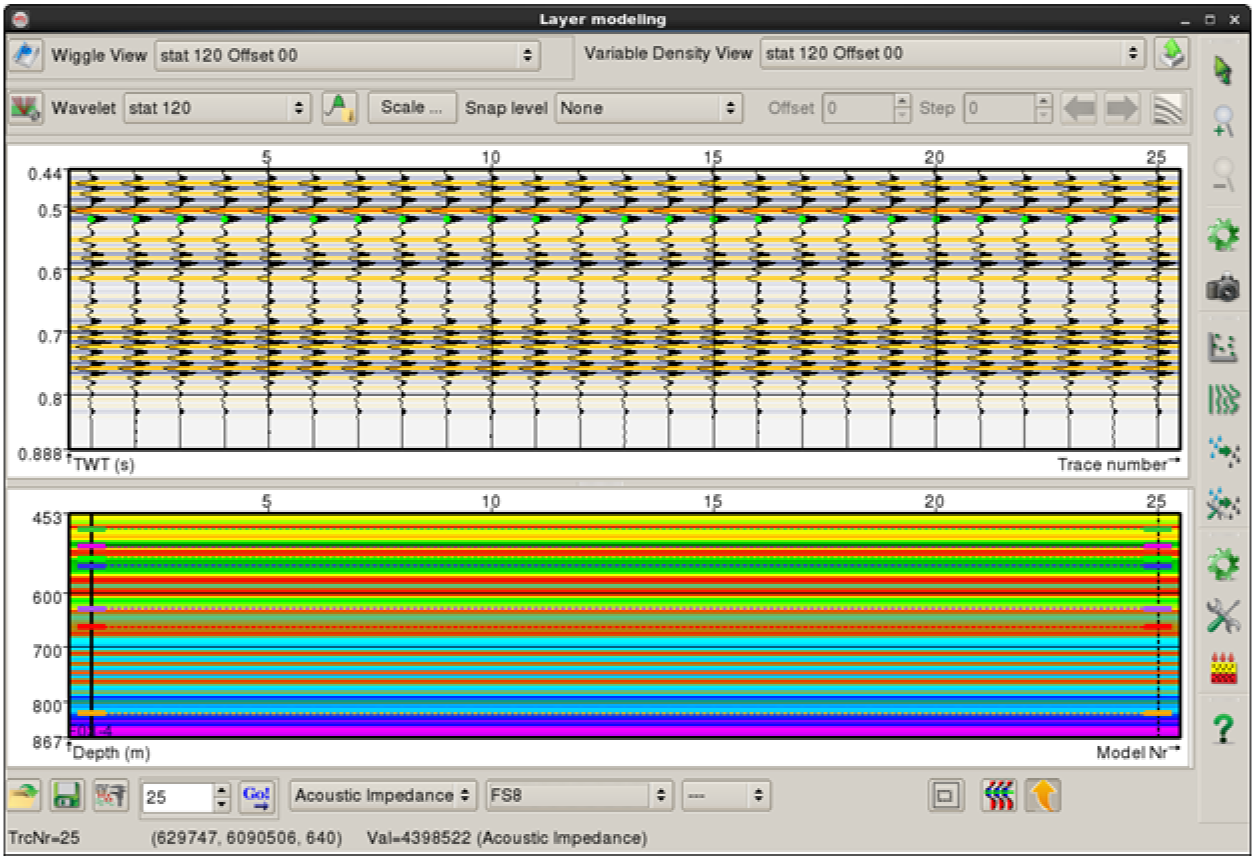
Well: F03-4

Density: Density, kg/m3 (Kg/m3)

Porosity: Porosity, (Fraction)

Pwave velocity: Sonic, us/m (Microseconds, ☑ Sonic

Before proceeding further observe changes in the *Layer modeling window* after clicking on the *Apply* button. A default number of pseudowells has been created, which are in fact simply duplicates of the selected well with logs upscaled according to Set process parameters window. The original well is positioned at the far-left side of the profile and displayed as a solid vertical line. The control profile is positioned at the far-right side of the profile and displayed as a dashed vertical line. Corresponding synthetic seismic section has been generated using the chosen wavelet and displayed in the top part of *Layer modeling* window.



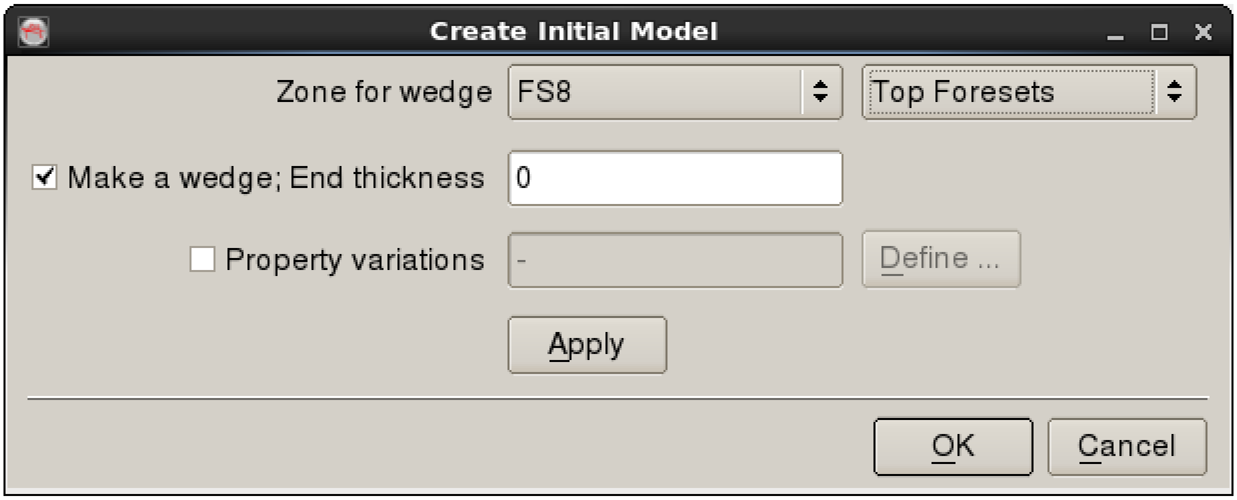
Wiggle View: stat 120 Offset 00
Variable Density View: stat 120 Offset 00
Wavelet: stat 120
Snap level: None
Offset: 0
Step: 0
25
Acoustic Impedance
FS8

TrcNr=25 (629747, 6090506, 640) Val=4398522 (Acoustic Impedance)

Clicking on OK button prompts to *Create Initial Model* window where a wedge thickness change and/or lateral property variations can be defined. First, a user must specify a zone by choosing existing well tops.

A wedge model

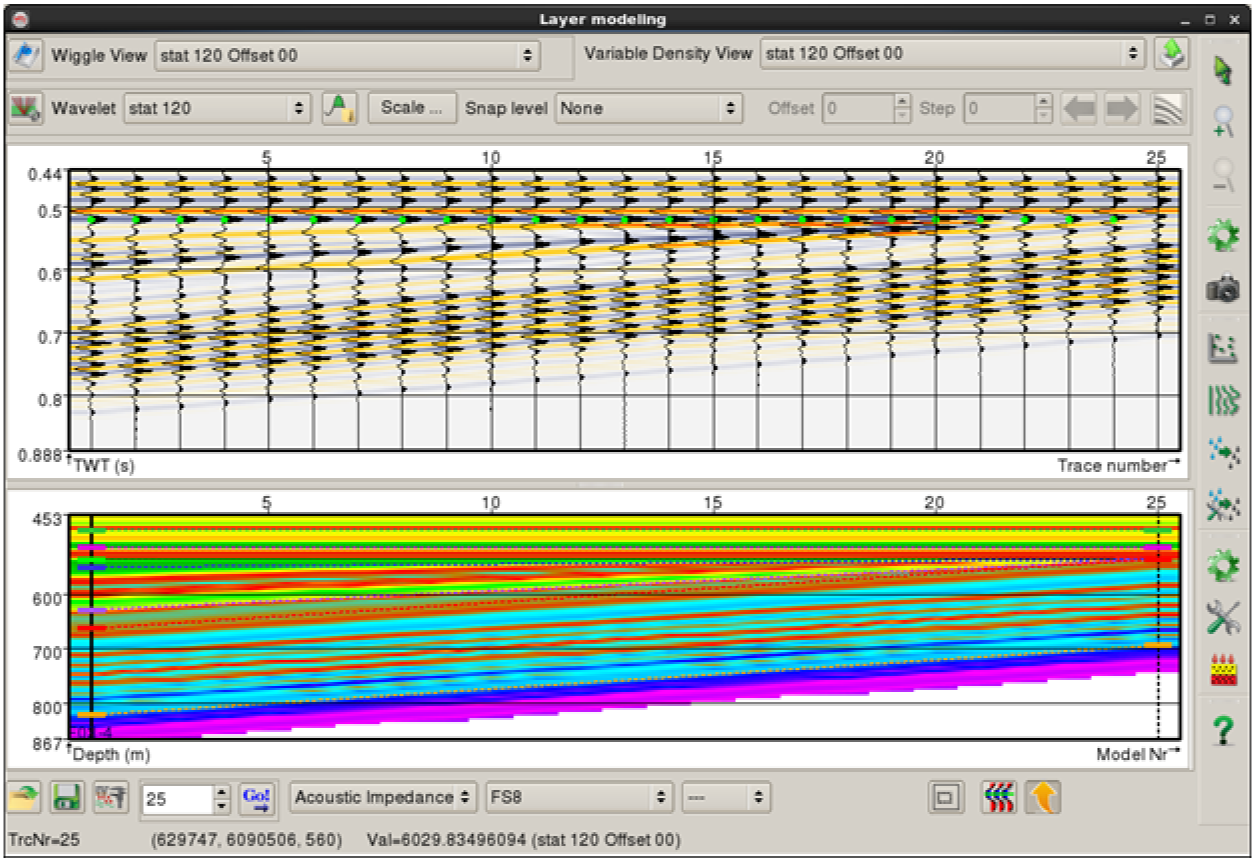
If thickness changes are modeled in the selected zone, *Make a wedge* option has to be checked, and *End thickness*, corresponding to the control profile at far right, has to be set. In the example below, wedge model of the *FS8 - Top Foresets* interval is defined by varying its thickness from the original value at the well (128 m) to 0 m. Clicking OK generates the defined model.


Create Initial Model

Zone for wedge: FS8 | Top Foresets

☑ Make a wedge; End thickness: 0

☐ Property variations: -


Layer modeling

Wiggle View: stat 120 Offset 00

Variable Density View: stat 120 Offset 00

Wavelet: stat 120

Snap level: None

Offset: 0

Step: 0

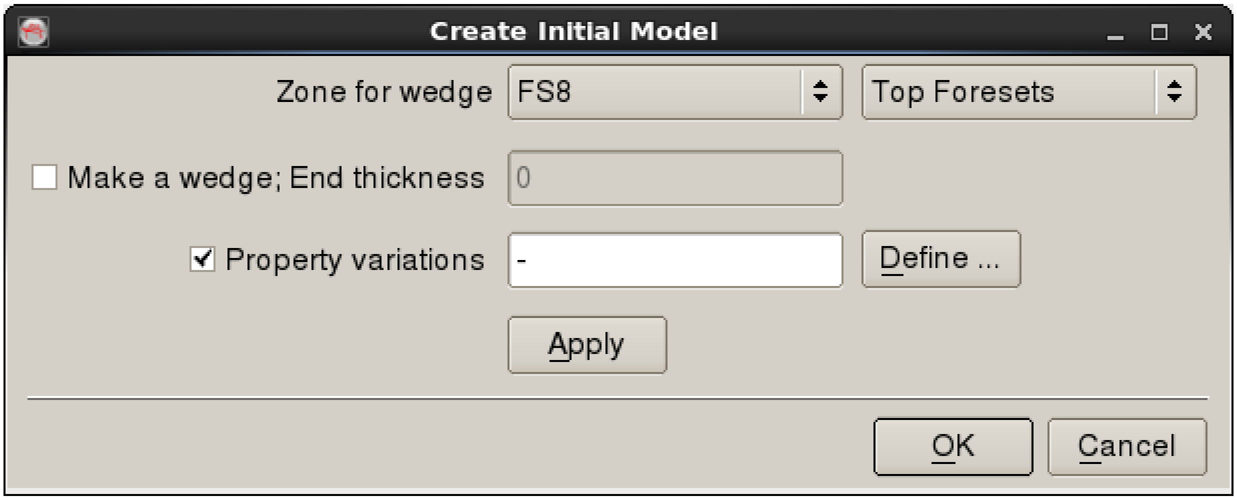
25

Acoustic Impedance | FS8 | ---

TrcNr=25 (629747, 6090506, 560) Val=6029.83496094 (stat 120 Offset 00)

Define property changes

If property changes are modeled alone, *Property variations* option must be checked and *Make a wedge* option must be unchecked. *Property variations* have to be specified by clicking on Define button.

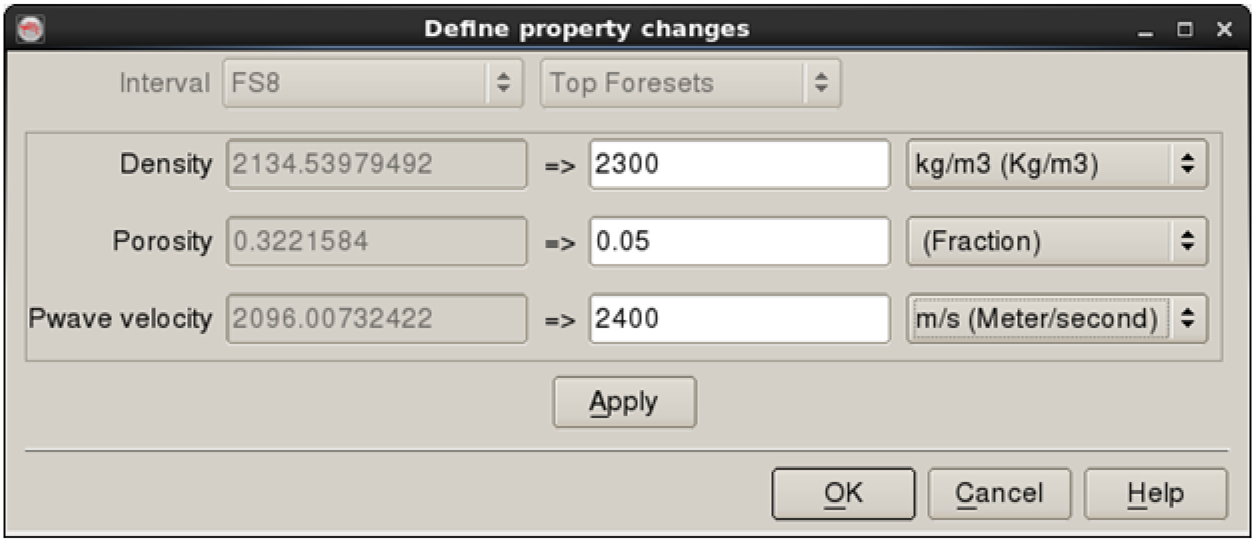


Zone for wedge: FS8 / Top Foresets

☐ Make a wedge; End thickness: 0

☑ Property variations: - (Define ...)

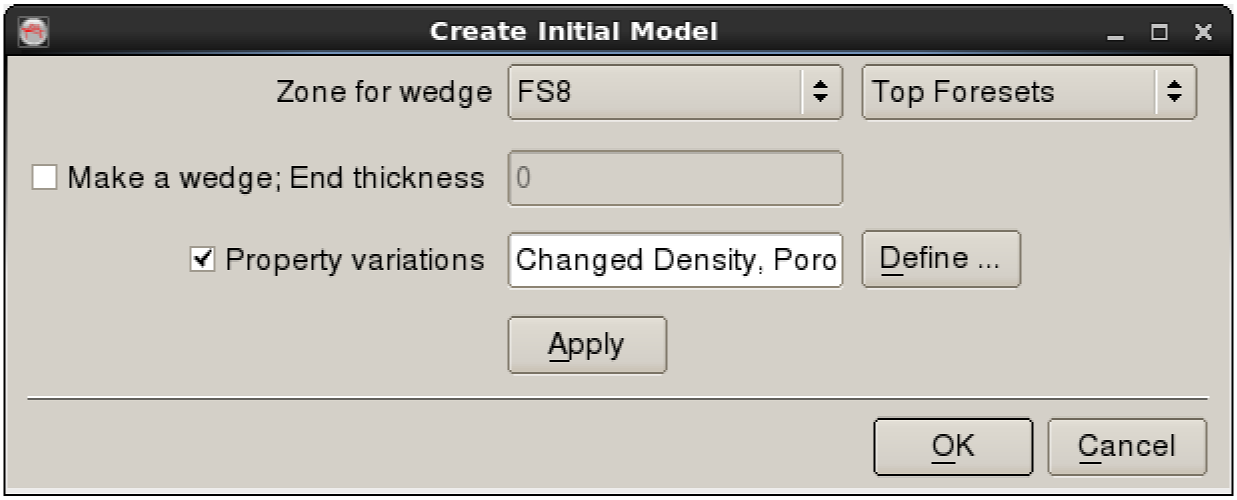
In *Define property changes* window, properties of a selected interval can be changed in such a manner that individual samples of a property are scaled to have the user-specified average in the given interval. The greyed-out left column contains initial average property values, and new average values must be specified in the right column. In the example below, *Porosity* is decreased to 5 percent with corresponding increases in *Density* and *Pwave velocity*. Click OK to confirm the changes.



Interval: FS8 / Top Foresets

Density	2134.53979492	=>	2300	kg/m3 (Kg/m3)
Porosity	0.3221584	=>	0.05	(Fraction)
Pwave velocity	2096.00732422	=>	2400	m/s (Meter/second)

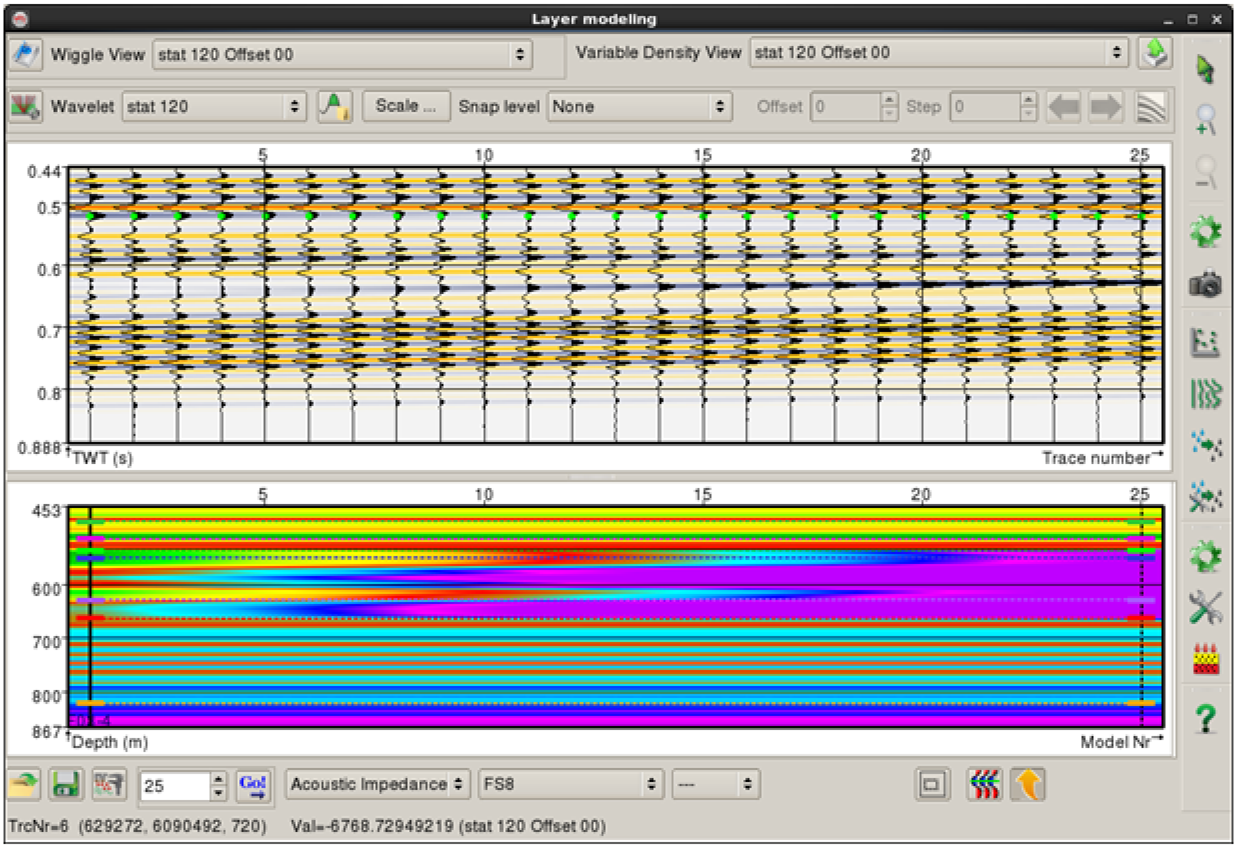
The *Create Initial Model* window is updated as shown below. Click *Apply* or OK to generate the model.



Zone for wedge: FS8 · Top Foresets

☐ Make a wedge; End thickness: 0

☑ Property variations: Changed Density, Poro



Wiggle View: stat 120 Offset 00

Variable Density View: stat 120 Offset 00

Wavelet: stat 120

Snap level: None

Offset: 0

Step: 0

25 · Acoustic Impedance · FS8 · ---

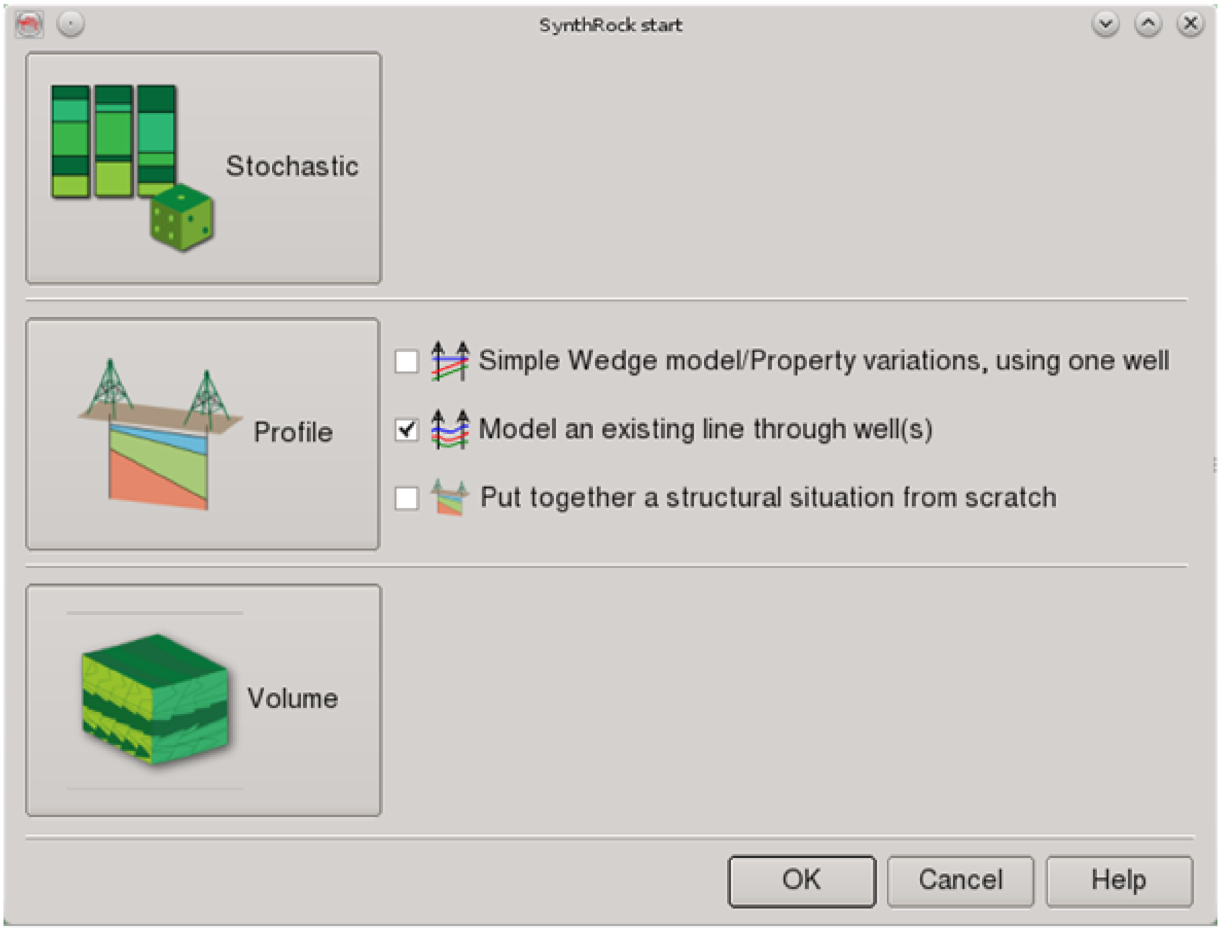
TrcNr=6 (629272, 6090492, 720) Val=-6768.72949219 (stat 120 Offset 00)

A wedge model with property variations

Finally, a wedge model can be combined with property variations by checking both options and defining the parameters as described above.

Model an existing line through well(s)

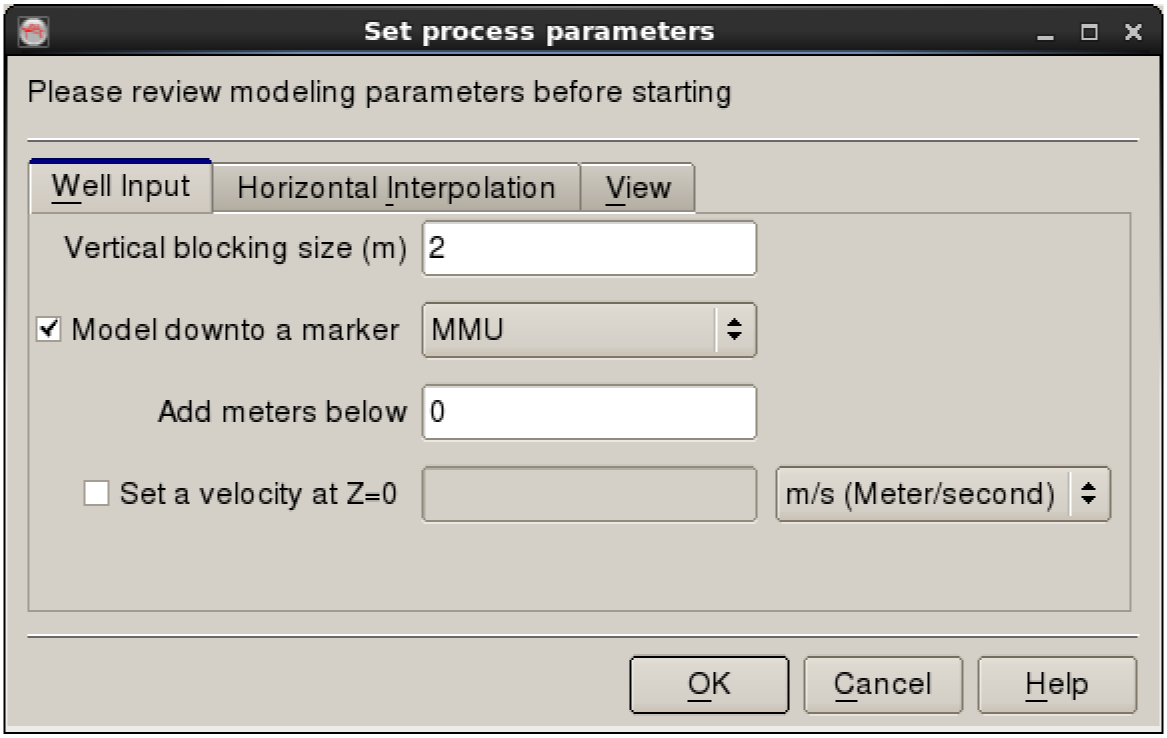
Model an existing line through well(s) is the second pre-defined workflow in SynthRock profile modeling which allows to quickly build a model through the existing wells in the project database. It is launched by checking the corresponding box in *SynthRock* start window and clicking OK.



☐ Simple Wedge model/Property variations, using one well
☑ Model an existing line through well(s)
☐ Put together a structural situation from scratch

A user is expected to go through the pre-defined sequence of windows. The workflow starts, as any profile modeling in SynthRock, with setting modeling parameters in Set process parameters window and selecting properties to be modeled in Layer Properties - Selection window. Once the workflow is completed, both sets of parameters can be modified by clicking on the and icons of *Layer modeling* window respectively.

In the example below, few changes of the default parameters are made in the first window in order to limit the modeled interval to [300 m; MMU].



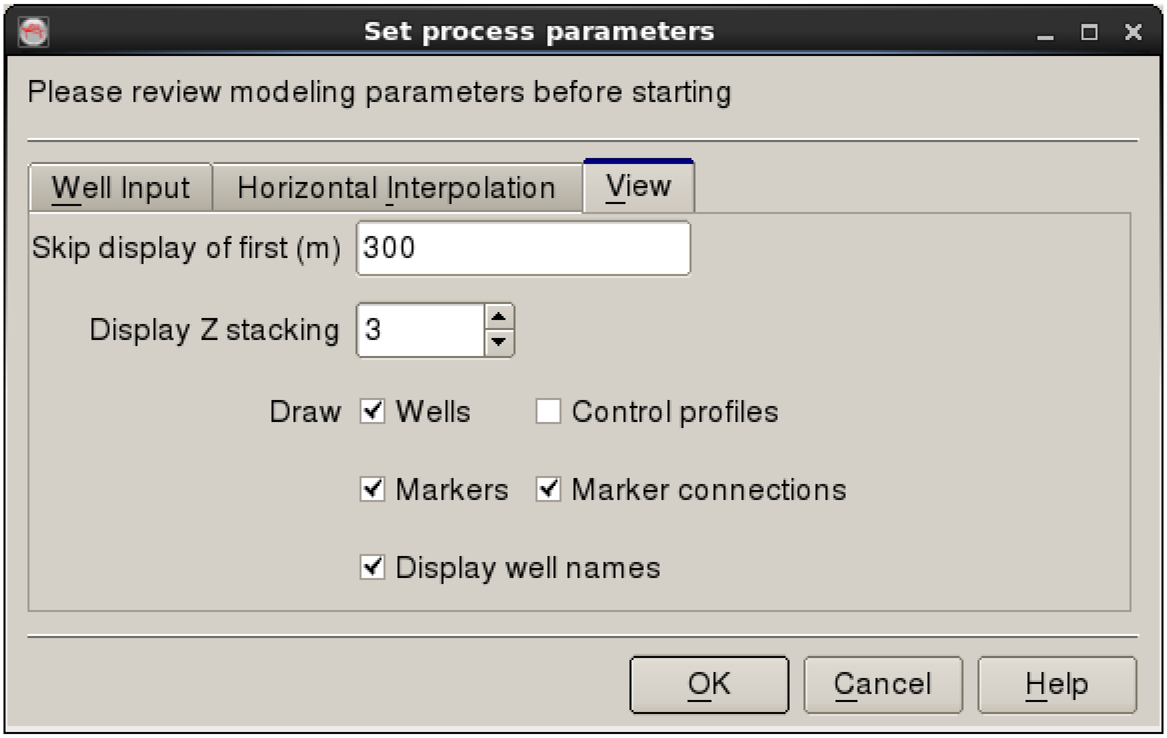
Please review modeling parameters before starting

Vertical blocking size (m): 2

☑ Model downto a marker: MMU

Add meters below: 0

☐ Set a velocity at Z=0: m/s (Meter/second)



Please review modeling parameters before starting

Skip display of first (m): 300

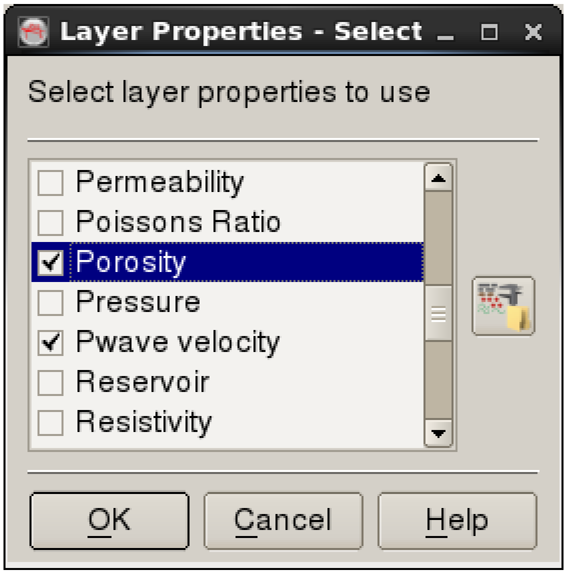
Display Z stacking: 3

Draw ☑ Wells ☐ Control profiles

☑ Markers ☑ Marker connections

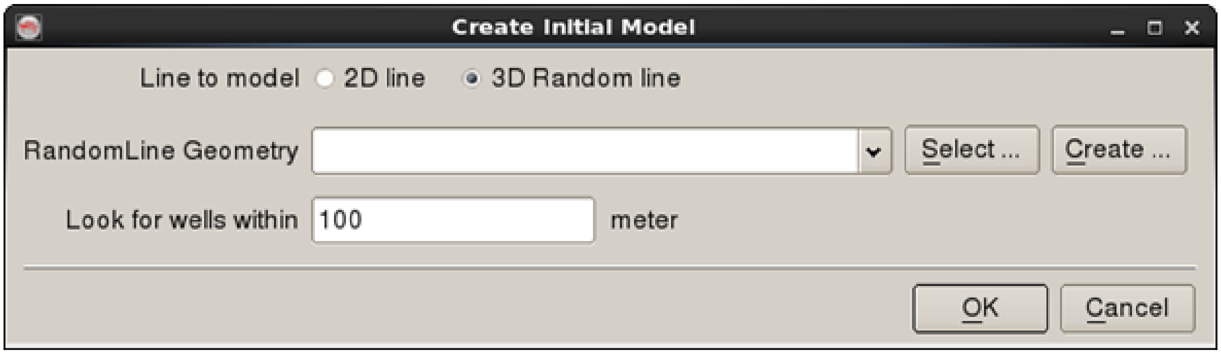
☑ Display well names

Porosity is added in addition to the default properties in the second window.



☐ Permeability
☐ Poissons Ratio
☑ Porosity
☐ Pressure
☑ Pwave velocity
☐ Reservoir
☐ Resistivity

Once the modeling parameters and the property list are confirmed, *Create Initial Model* window appears. First, the *Line to model* option must be selected as either *2D line* or *3D Random line*.

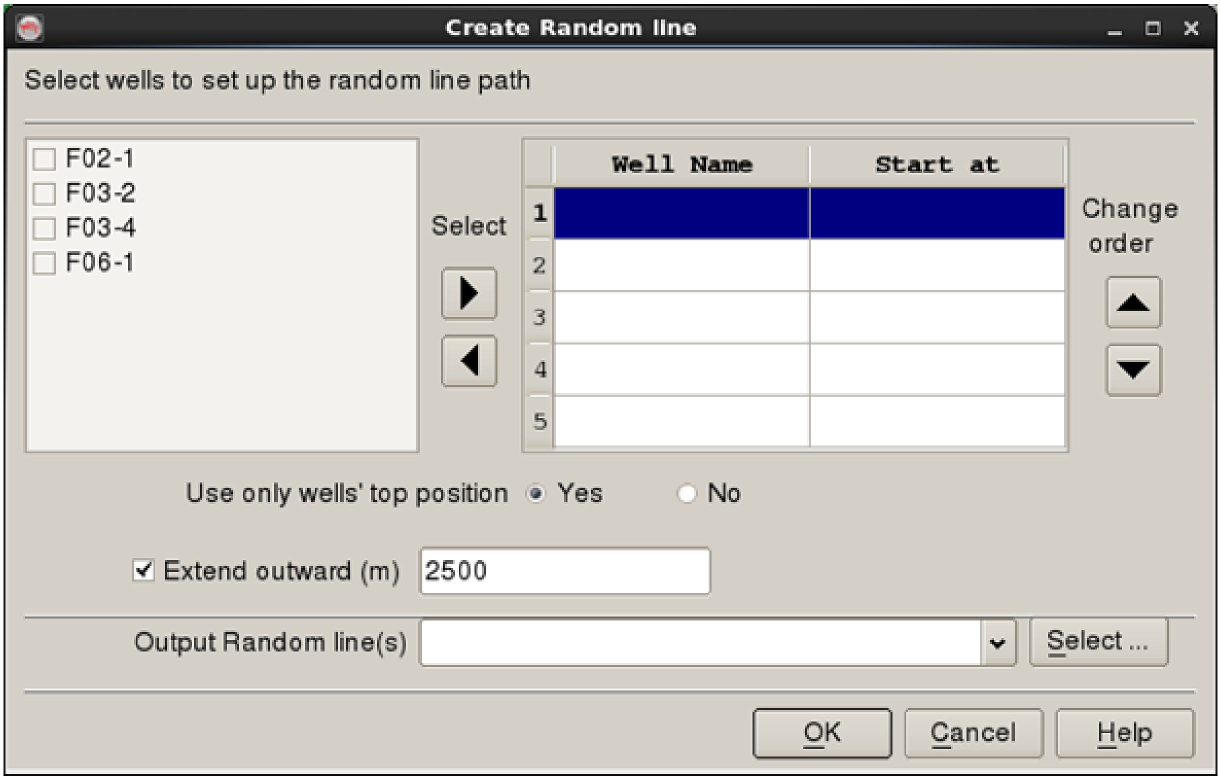


Line to model ○ 2D line ● 3D Random line

Look for wells within 100 meter

Create 3D Random Line

Geometry of a 3D random line through the existing wells can be created by clicking on *Create* button.

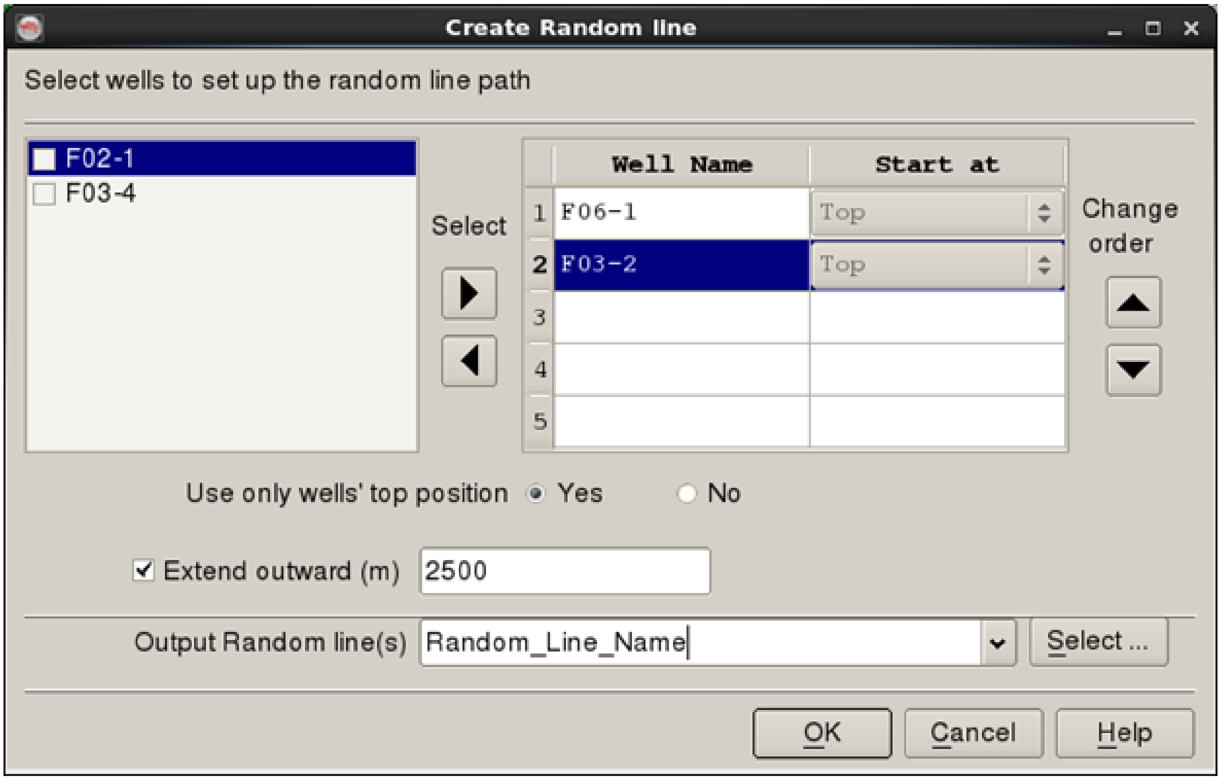


☐ F02-1
☐ F03-2
☐ F03-4
☐ F06-1

Use only wells' top position ● Yes ○ No

☑ Extend outward (m) 2500

Wells to be used must be first selected and then added/removed using the '*Select*' arrows. Order of the wells can be defined using the '*Change order*' arrows and this order defines the shape of a random line. If *Use only well's top position* is set to *Yes*, the random line is drawn through the well tops. Alternatively, No gives an option to choose *Top* or *Bottom* of each well. A random line can be optionally extended beyond the locations of the first and the last wells by checking the box and specifying the distance. Finally, after giving a name click OK to create the random line and return to *Create Initial* Model window.



Select wells to set up the random line path

☐ F02-1
☐ F03-4

	Well Name	Start at
1	F06-1	Top
2	F03-2	Top
3		
4		
5		

Use only wells' top position ● Yes ○ No

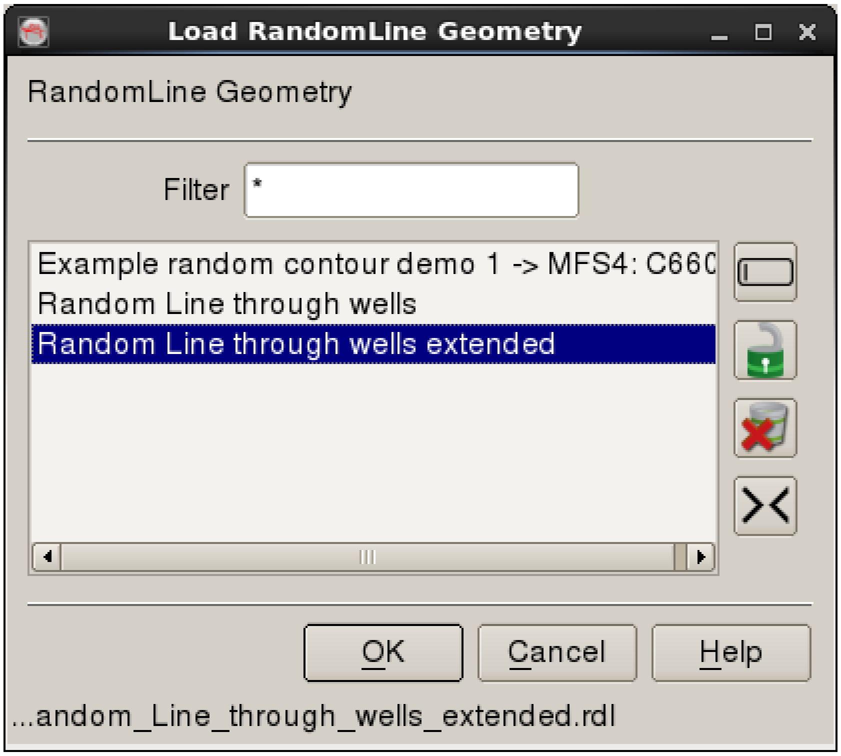
☑ Extend outward (m) 2500

Output Random line(s) Random_Line_Name

Existing 3D Random line

If a model is created along an existing 3D random line, click on *Select* button to choose from the list of existing random lines in the project database.

As an example, a model is created along a *Random Line through wells* extended which passes through all 4 wells, *F06-1*, *F02-1*, *F03-2* and *F03-4* and extends outwards on both sides.

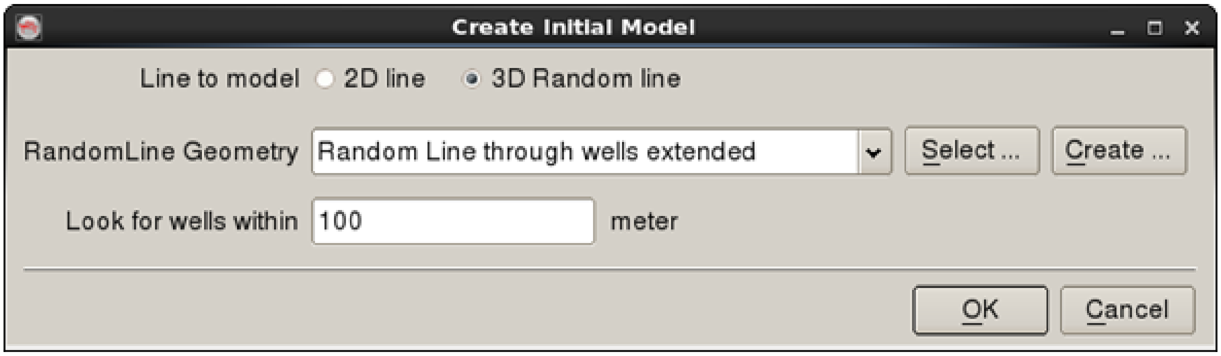


Filter *

Example random contour demo 1 -> MFS4: C660
Random Line through wells
Random Line through wells extended

...andom_Line_through_wells_extended.rdl

Once the line is selected, its name appears in the *Create Initial Model* window. *Look for wells within* allows to include wells that are not positioned exactly on the selected line.

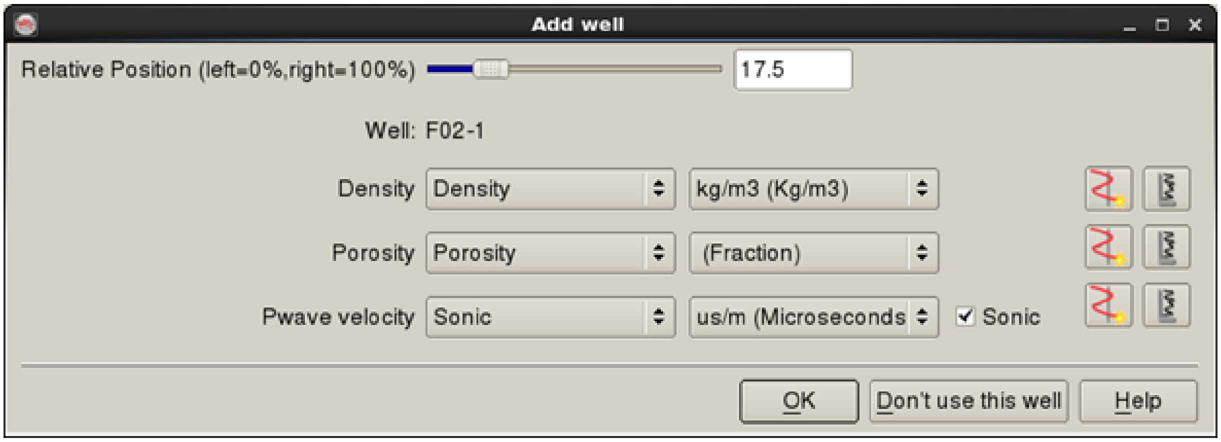


Line to model ○ 2D line ● 3D Random line

RandomLine Geometry Random Line through wells extended

Look for wells within 100 meter

Click OK to proceed, and then go through a number of *Add well* windows. Such window is shown for every well located within the specified distance from the selected line. Automatically determined *Relative Position* and logs can be modified if needed. A user can also select not to use a particular well by clicking on *Don't use this well* button.



Add well

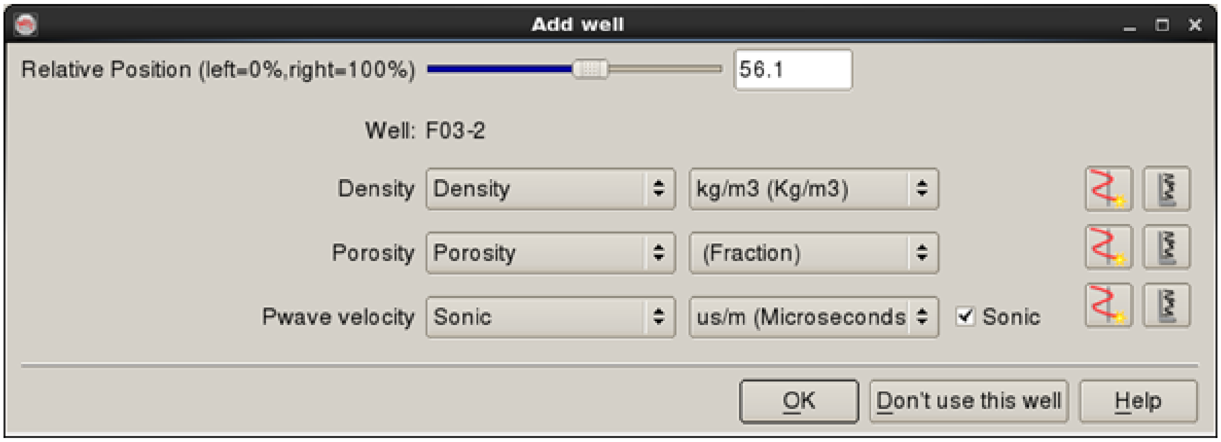
Relative Position (left=0%,right=100%) 17.5

Well: F02-1

Density: Density, kg/m3 (Kg/m3)

Porosity: Porosity, (Fraction)

Pwave velocity: Sonic, us/m (Microseconds ☑ Sonic



Add well

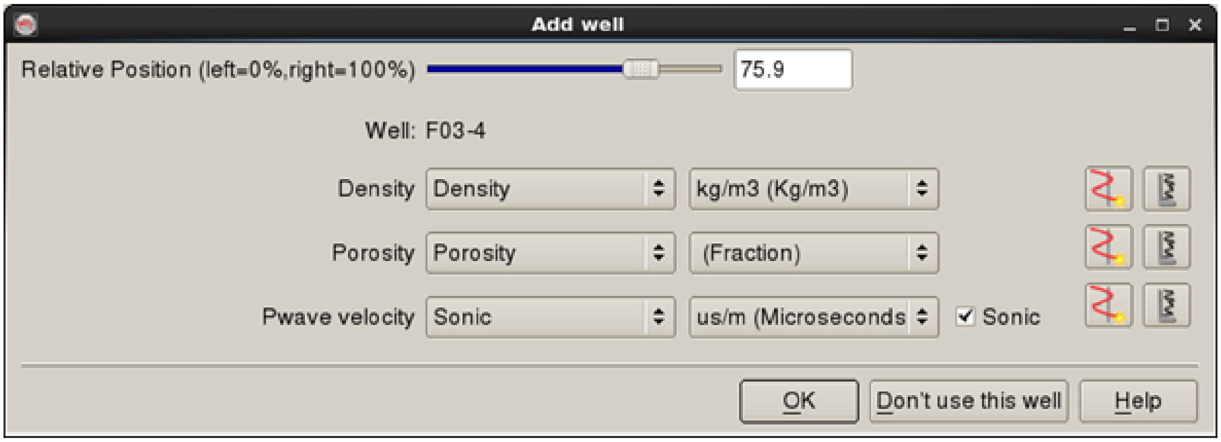
Relative Position (left=0%,right=100%) 56.1

Well: F03-2

Density: Density, kg/m3 (Kg/m3)

Porosity: Porosity, (Fraction)

Pwave velocity: Sonic, us/m (Microseconds ☑ Sonic



Add well

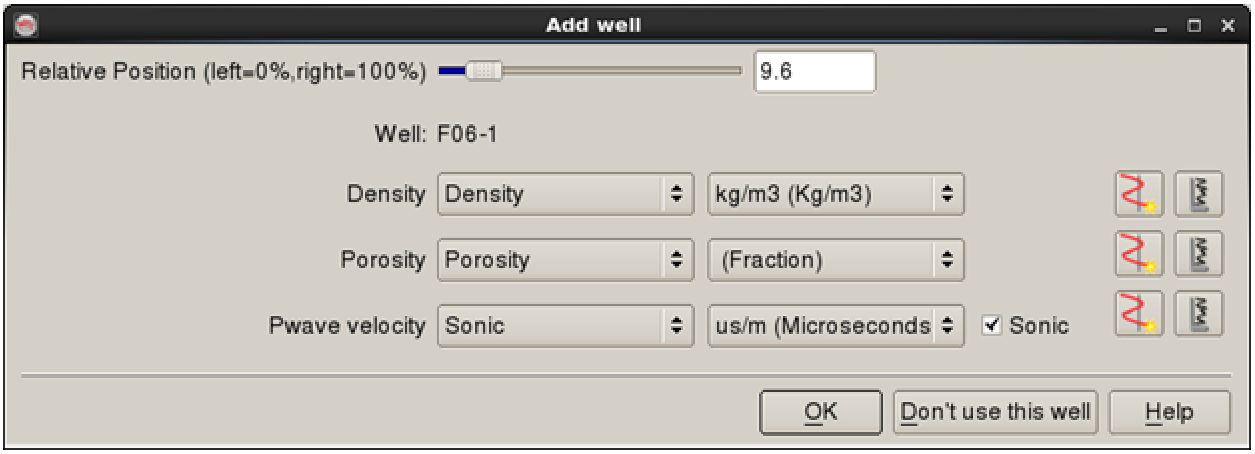
Relative Position (left=0%,right=100%) 75.9

Well: F03-4

Density: Density, kg/m3 (Kg/m3)

Porosity: Porosity, (Fraction)

Pwave velocity: Sonic, us/m (Microseconds ☑ Sonic



Relative Position (left=0%,right=100%) 9.6

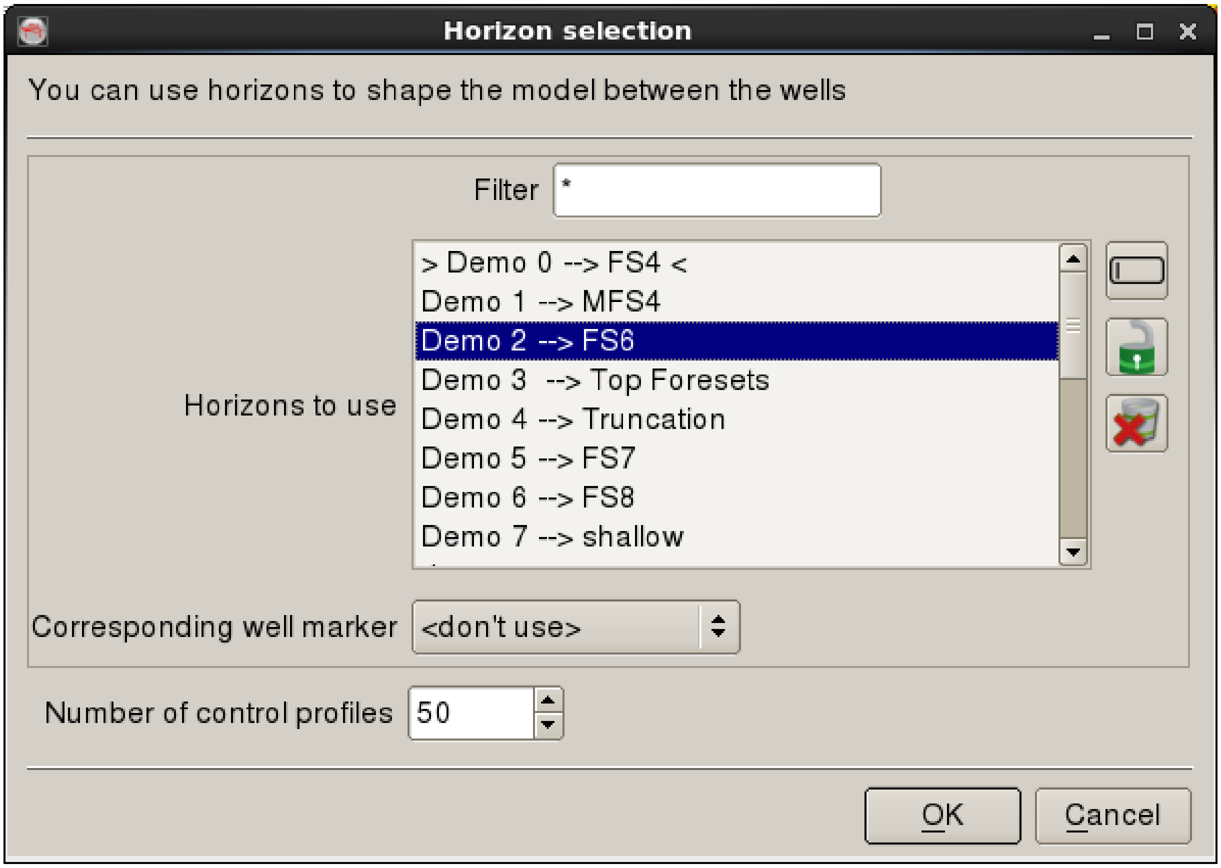
Well: F06-1

Density: Density, kg/m3 (Kg/m3)

Porosity: Porosity, (Fraction)

Pwave velocity: Sonic, us/m (Microseconds ☑ Sonic

Next, the *Horizon selection* window allows to choose horizons and corresponding formation tops to steer the horizontal interpolation. Select any horizon in *Horizons* to use list and note that the *Corresponding well marker* is set to <*don't use*> by default. This means that the selected horizon is not used in modeling.



You can use horizons to shape the model between the wells

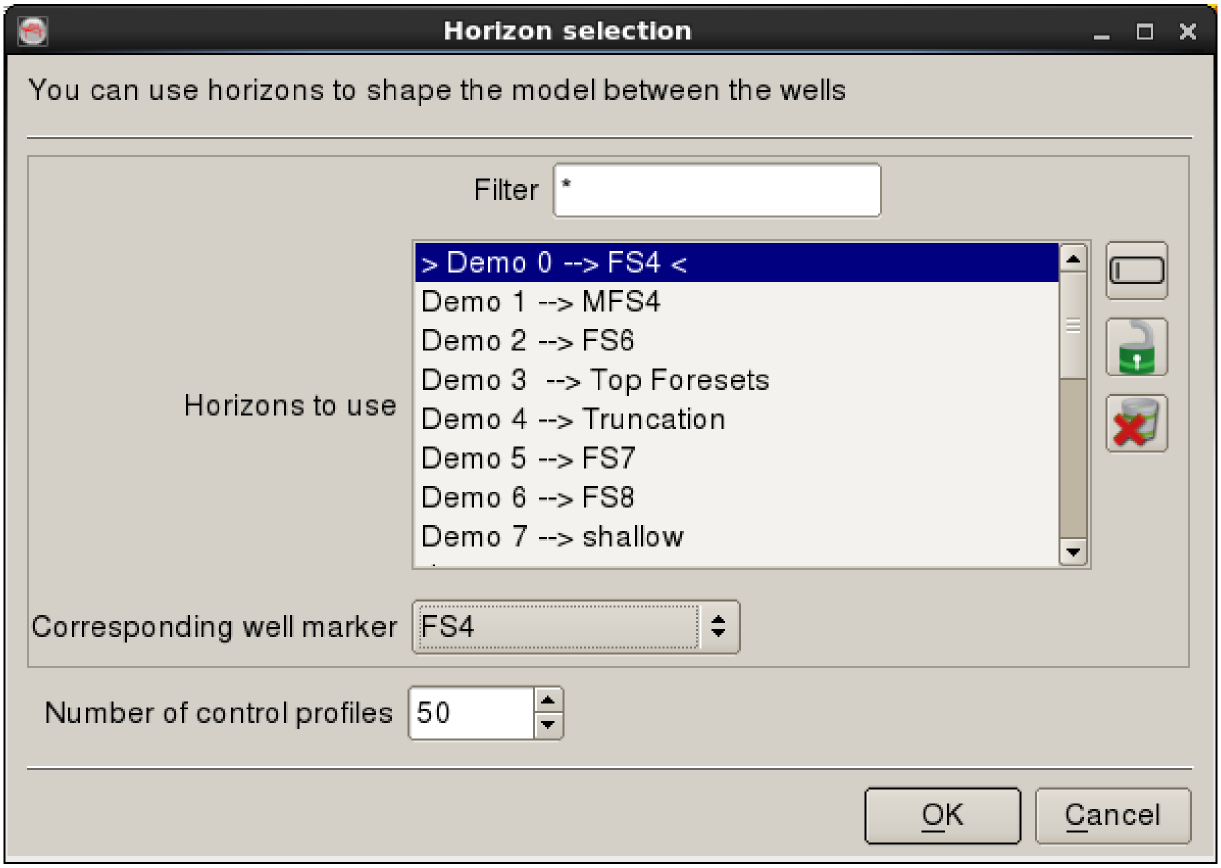
Filter *

Horizons to use:
- > Demo 0 --> FS4 <
- Demo 1 --> MFS4
- Demo 2 --> FS6
- Demo 3 --> Top Foresets
- Demo 4 --> Truncation
- Demo 5 --> FS7
- Demo 6 --> FS8
- Demo 7 --> shallow

Corresponding well marker <don't use>

Number of control profiles 50

Select the first horizon to be used in the modeling and set the corresponding marker as shown below for '*Demo 0 --> FS4*' horizon with a corresponding *FS4* well top. Similarly, well tops have to be selected for all the horizons to be used in the modeling. Finally, *Number of control profiles* can be specified. Clicking OK generates a model with default number of pseudowells.



You can use horizons to shape the model between the wells

Filter *

Horizons to use

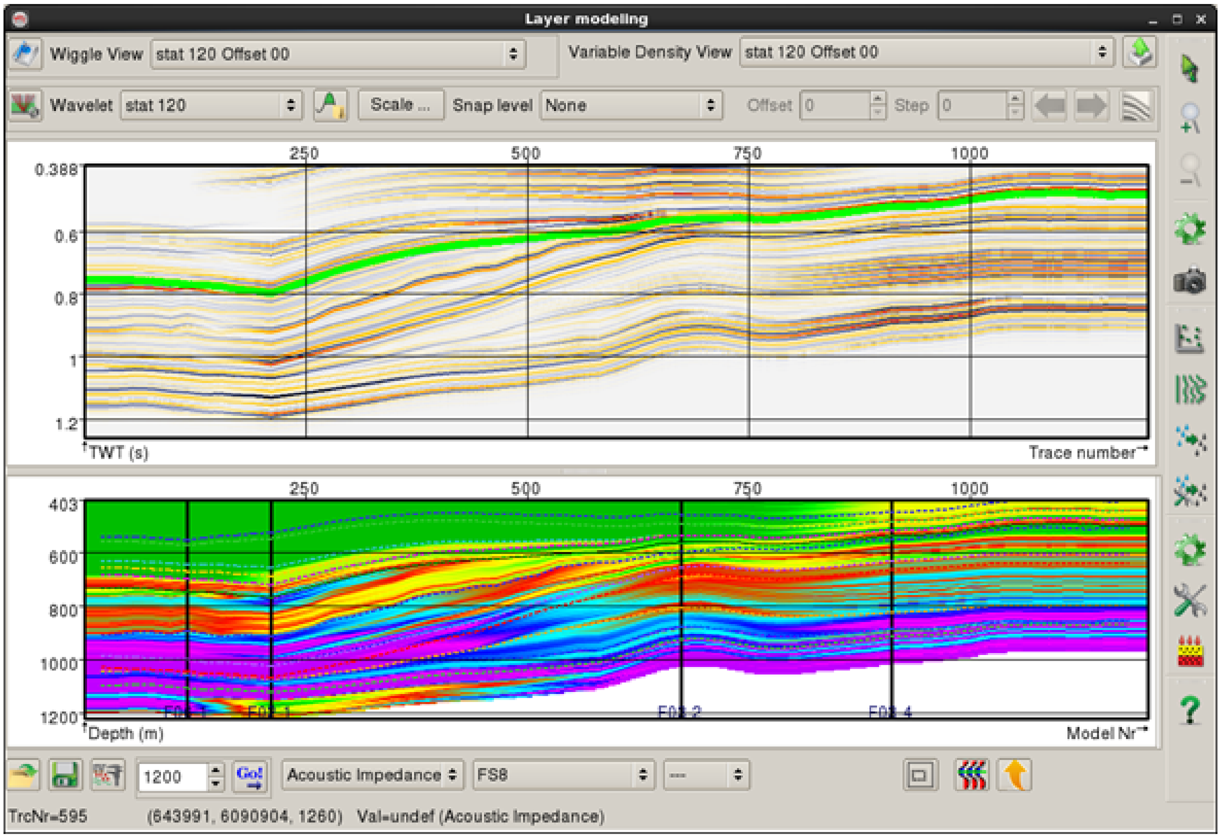
- > Demo 0 --> FS4 <
- Demo 1 --> MFS4
- Demo 2 --> FS6
- Demo 3 --> Top Foresets
- Demo 4 --> Truncation
- Demo 5 --> FS7
- Demo 6 --> FS8
- Demo 7 --> shallow

Corresponding well marker FS4

Number of control profiles 50

At this point the pre-defined workflow is finished, and further modeling can be continued using the classic Profile mode functionality in *Layer modeling* window.

An example below shows the final model along the *Random Line through wells extended*. Five horizons and corresponding well tops are set as following: *'Demo 0 --> FS4' - FS4; 'Demo 1 --> MFS4' - MFS4; 'Demo 4 --> Truncation' - Truncation; 'Demo 5 --> FS7' - FS7; and 'Demo 6 --> FS8' - FS*8. Note, that default number of pseudowells is changed to 1200, and some display parameters are modified to create this image. The green line overlying synthetic seismic corresponds to *FS8* well top.



Wiggle View: stat 120 Offset 00

Variable Density View: stat 120 Offset 00

Wavelet: stat 120

Snap level: None

Offset: 0

Step: 0

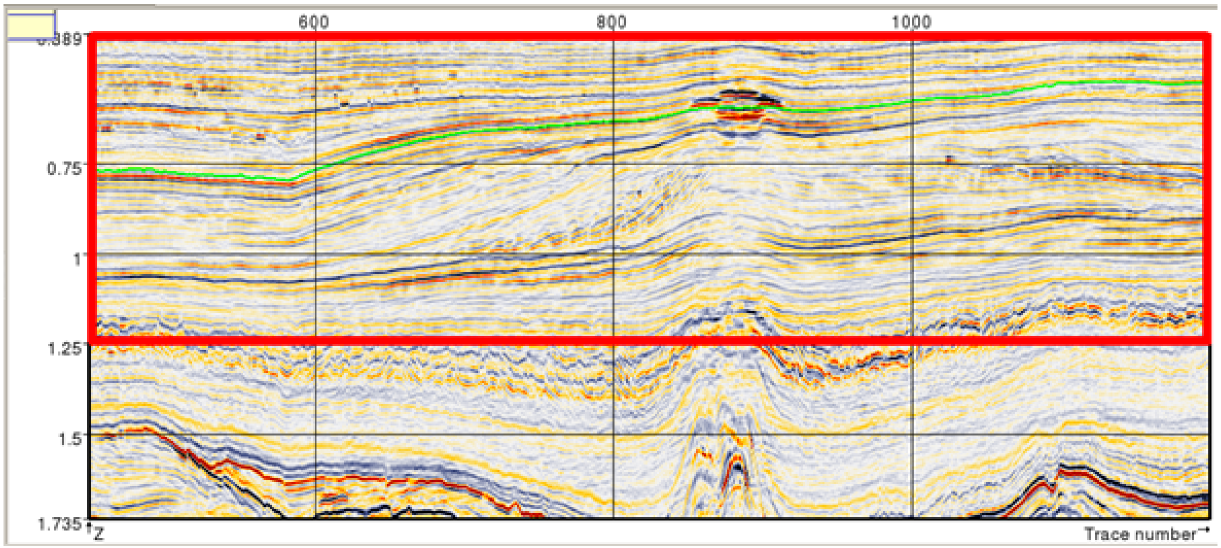
1200

Acoustic Impedance

FS8

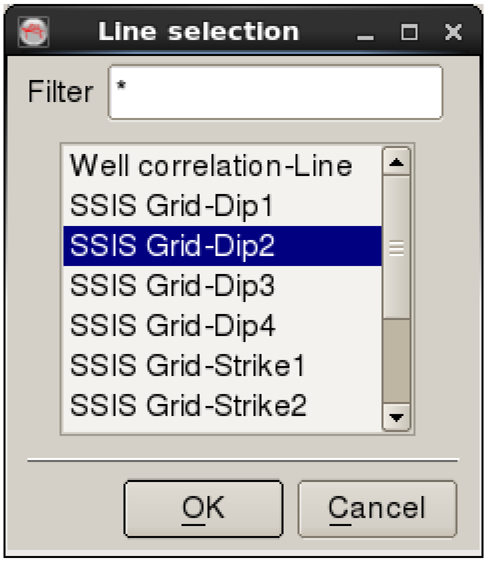
TrcNr=595 (643991, 6090904, 1260) Val=undef (Acoustic Impedance)

Below is the corresponding seismic section along the selected random line. The red box outlines the modeled interval, and the light green line corresponds to '*Demo 6 --> FS8*' horizon. The constructed model is a good start for further detailed modeling. The detailed modeling can be done to support existing interwell interpretation. For example, by modifying properties of the control profiles in the vicinity of the high-amplitude anomaly seen in the middle of the seismic section along the green horizon.



Existing 2D line

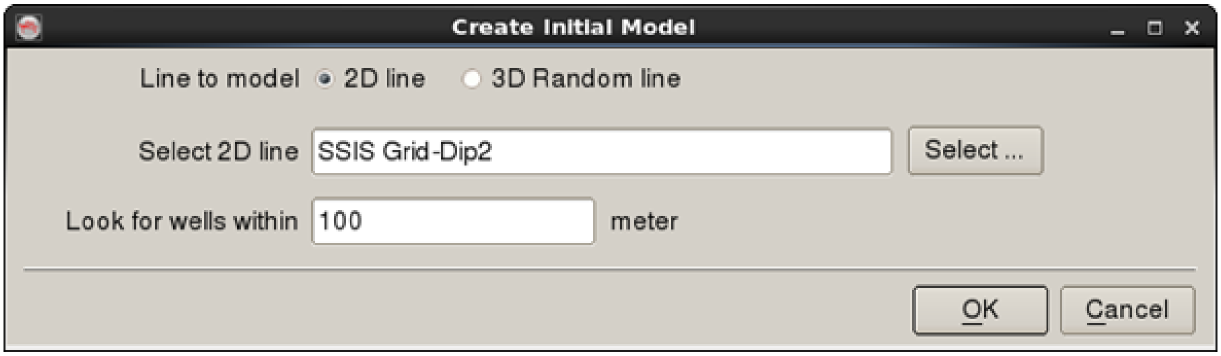
Switch Line to model to 2D line option and click *Select*. In the *Line selection* window choose the existing 2D line in the project database along which modeling is to be done.



Filter: *

Well correlation-Line
SSIS Grid-Dip1
SSIS Grid-Dip2
SSIS Grid-Dip3
SSIS Grid-Dip4
SSIS Grid-Strike1
SSIS Grid-Strike2

Once the 2D line is selected, specify the appropriate distance from the line to include all required wells for the modeling and click OK. Afterwards, follow the instructions described for Existing 3D Random line.



Line to model ● 2D line ○ 3D Random line

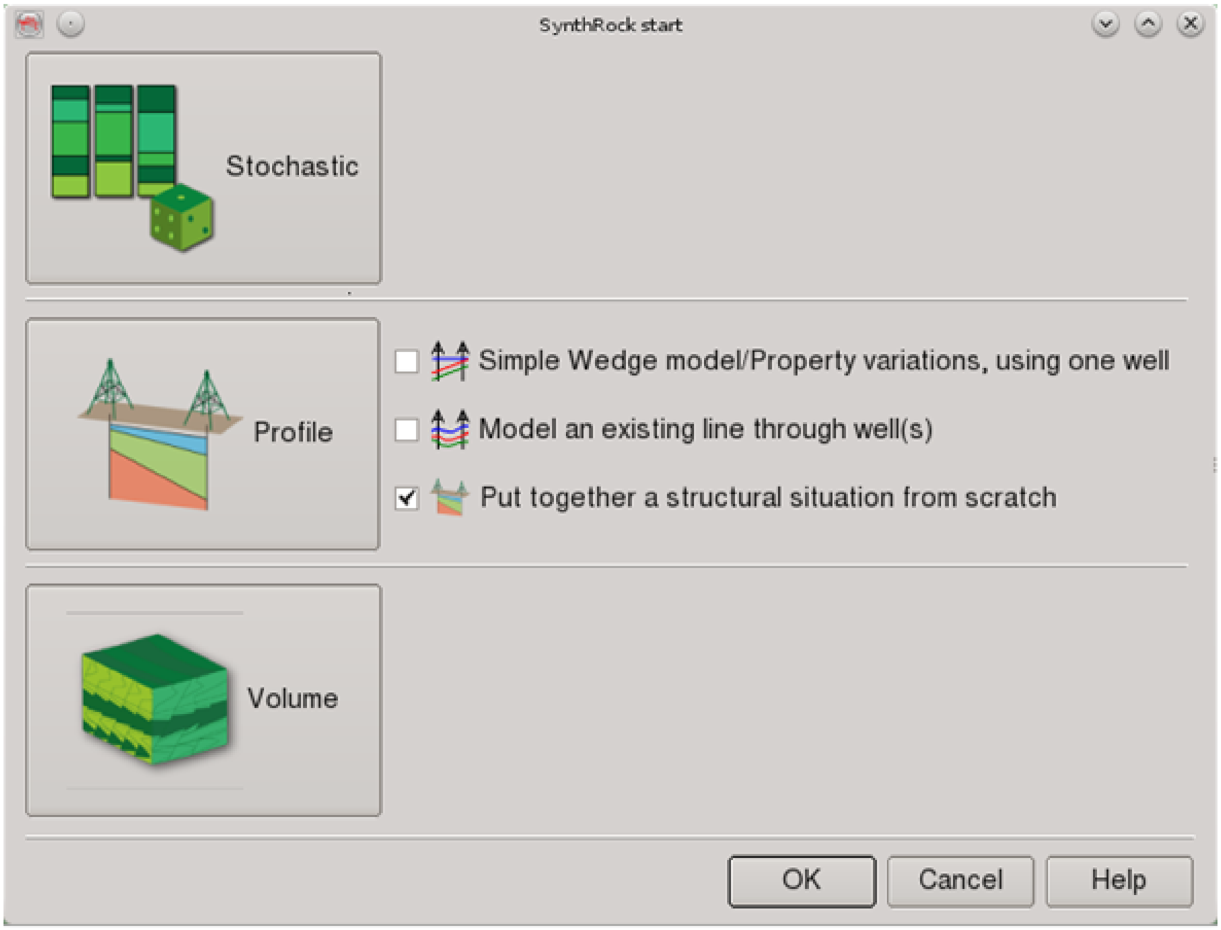
Select 2D line: SSIS Grid-Dip2

Look for wells within: 100 meter

Put Together a Structural Situation from Scratch

Put together a structural situation from scratch is a powerful modeling tool with all classic Profile mode functionality. In this mode a user starts building a model from scratch. It gives all the flexibility available, but also requires more time than the pre-defined workflows. Therefore, it is recommended to start with pre-defined workflows if possible and, once a simple model is built, continue with a classic Profile mode functionality as described below.

Put together a structural situation from scratch can be launched from *SynthRock* start window either by checking the corresponding box and clicking OK or simply by clicking on the large Profile icon.



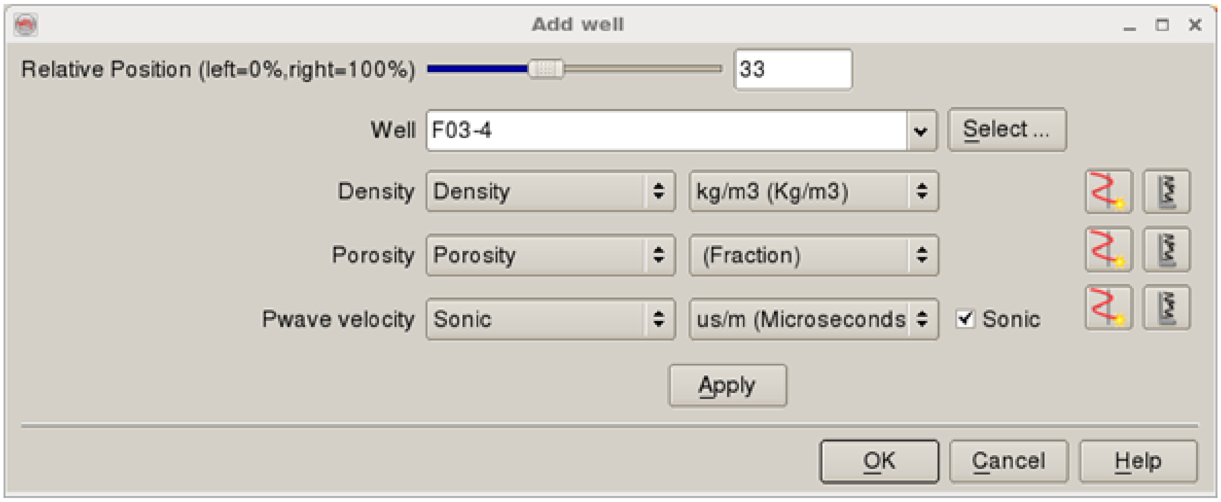
☐ Simple Wedge model/Property variations, using one well
☐ Model an existing line through well(s)
☑ Put together a structural situation from scratch

Add/edit well and control profile

Control points for the profile modeling are added with a right-click in the bottom part of Layer modeling window. Before the first control point is added, a pop-up window *Set process parameters* requires to confirm/specify modeling parameters and pop-up window *Layer Properties - Selection* allows to select properties to be used in modeling. Later during the modeling both sets of parameters can be modified by clicking on the and icons respectively. The first control point must be an existing well in the project. Additional control points can be created by adding more wells (Add well) or by setting existing pseudowells as control profiles (*Add Control Profile*).

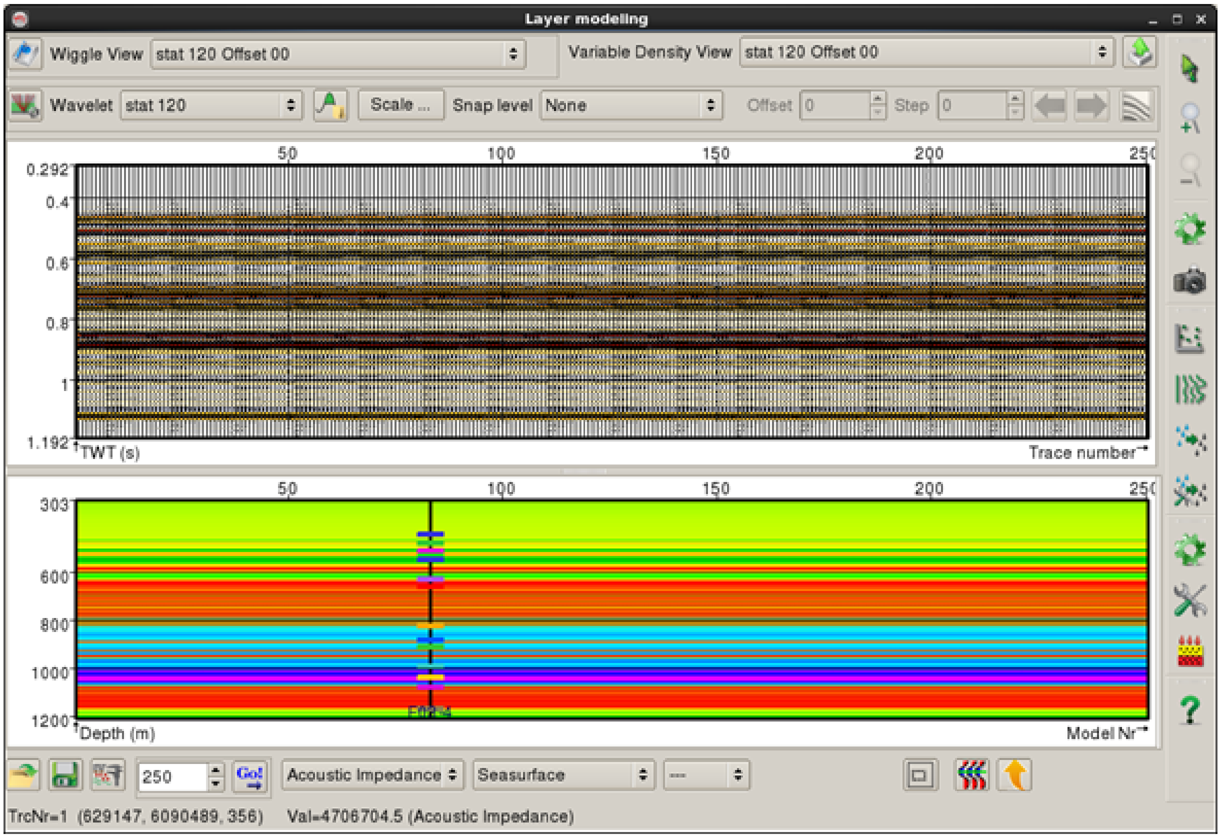
Add the first well

Click in the area where it says <*Click to add a well*>. Once the modeling parameters and the property list are confirmed, the first control point can be added. *Add well* window allows to choose the well to be added.



Relative Position (left=0%,right=100%) 33
Well F03-4
Density Density kg/m3 (Kg/m3)
Porosity Porosity (Fraction)
Pwave velocity Sonic us/m (Microseconds ☑ Sonic

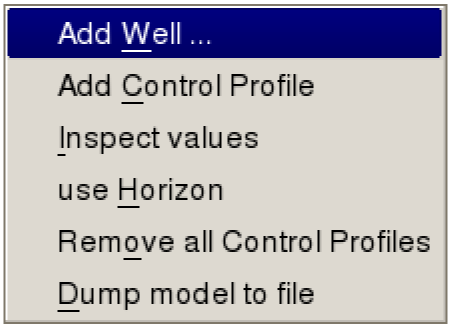
The *Relative Position* element specifies where the new well is added in the profile. Here, 0 and 100% correspond to the left and right ends of the profile respectively. Thus, the position of the added well is determined as (*N * Relative Position)/100* where *N* is the number of pseudowells to be generated (specified left of the Go! button). Note, that all properties chosen in *Layer Properties - Selection* window are required to have a log selected. Missing logs can be either created by clicking on the icon without having to exit SynthRock or imported using the *Well Manager*. The selected log can be displayed by clicking on the icon. Pressing Ok or *Apply* creates a specified number of pseudowells that have been interpolated/extrapolated from the selected well with logs upscaled according to *Set process parameters* window. The original well is displayed as a solid vertical line. When the first well is added, the generated pseudowells are simply duplicates of the original one. Corresponding synthetic seismic section is generated using the chosen wavelet and displayed in the top part of *Layer modeling window*.



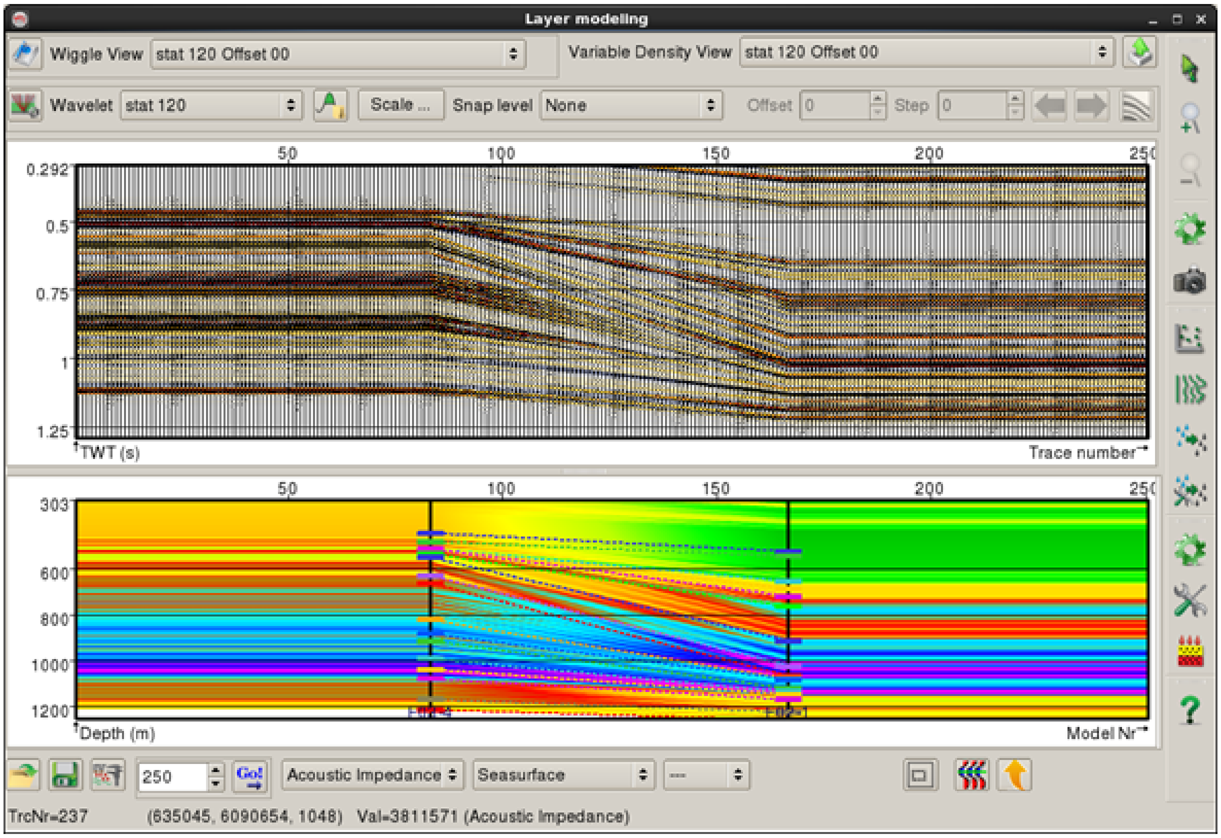
TrcNr=1 (629147, 6090489, 356) Val=4706704.5 (Acoustic Impedance)

Add more wells

Once the first well is added, right click in the lower part of the *Layer modeling* window brings up several options as shown below.



The *Add Well* option brings exactly the same window as the first time. After adding a new well, the pseudowells are re-generated by horizontal interpolation between the original wells in the profile.



Wiggle View: stat 120 Offset 00

Variable Density View: stat 120 Offset 00

Wavelet: stat 120

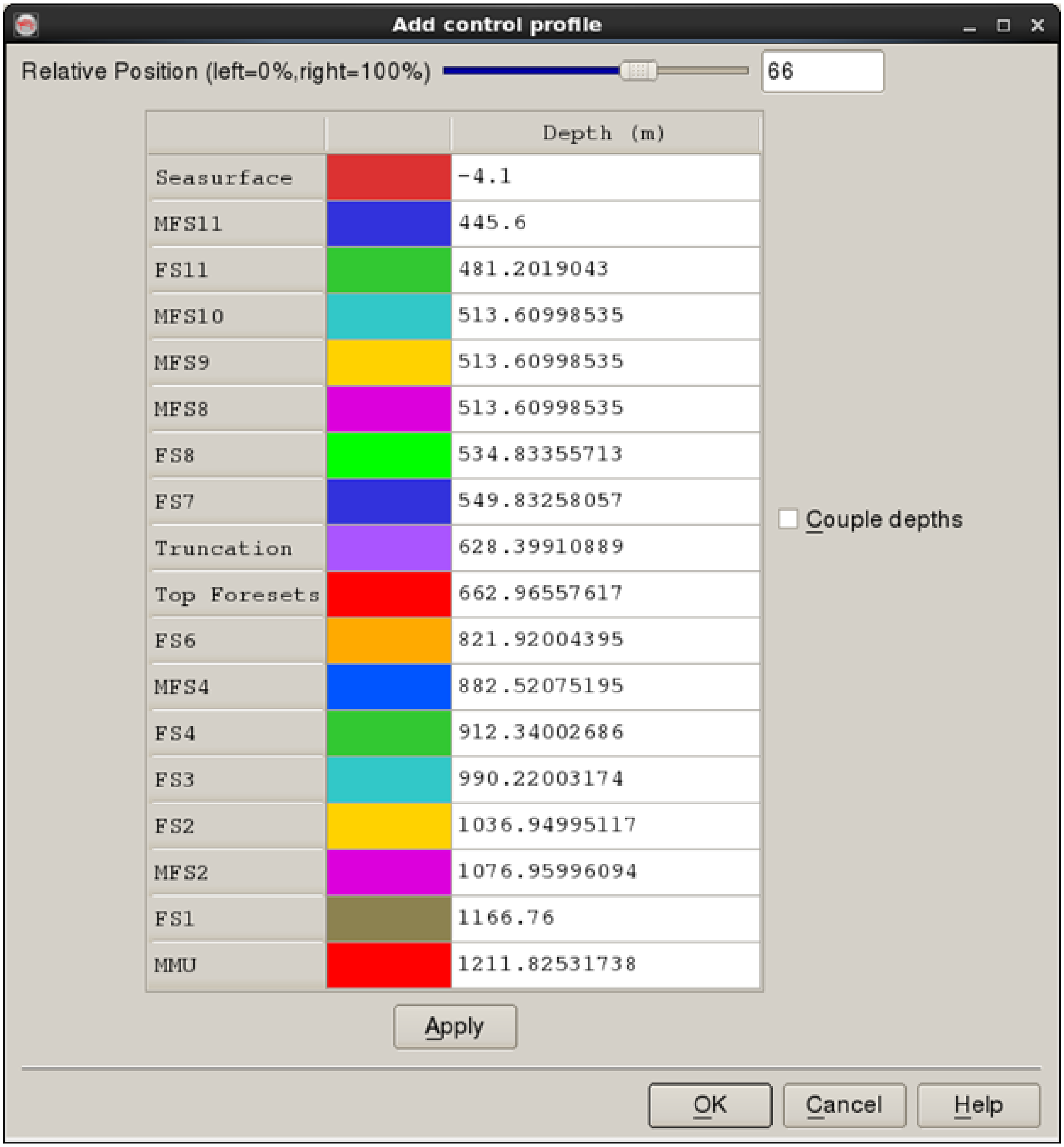
Snap level: None

250 | Acoustic Impedance | Seasurface | ---

TrcNr=237 (635045, 6090654, 1048) Val=3811571 (Acoustic Impedance)

Add control profile

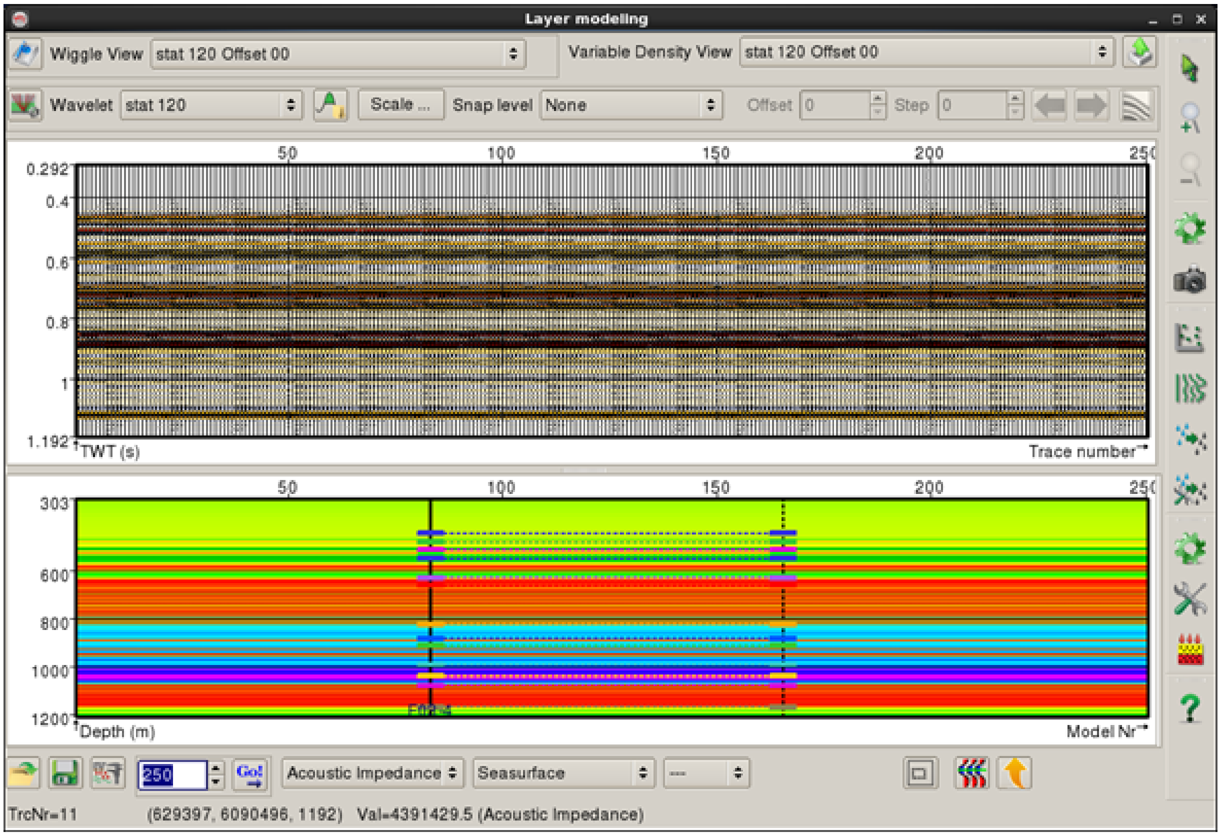
Right click on existing profile outside added wells and select to *Add Control Profile*. Specify *Relative Position* and click *Apply* to find out that a new control profile is added to the *Layer modeling* window without thickness or depth changes so far. In the example below, a control profile is added to the model containing only one original well *F03-4*. The control profile is displayed with a dashed line.



Relative Position (left=0%,right=100%) 66

		Depth (m)
Seasurface	🟥	-4.1
MFS11	🟦	445.6
FS11	🟩	481.2019043
MFS10	🟦	513.60998535
MFS9	🟨	513.60998535
MFS8	🟪	513.60998535
FS8	🟩	534.83355713
FS7	🟦	549.83258057
Truncation	🟪	628.39910889
Top Foresets	🟥	662.96557617
FS6	🟧	821.92004395
MFS4	🟦	882.52075195
FS4	🟩	912.34002686
FS3	🟦	990.22003174
FS2	🟨	1036.94995117
MFS2	🟪	1076.95996094
FS1	🟫	1166.76
MMU	🟥	1211.82531738

☐ Couple depths



Wiggle View: stat 120 Offset 00

Variable Density View: stat 120 Offset 00

Wavelet: stat 120

Snap level: None

Offset: 0

Step: 0

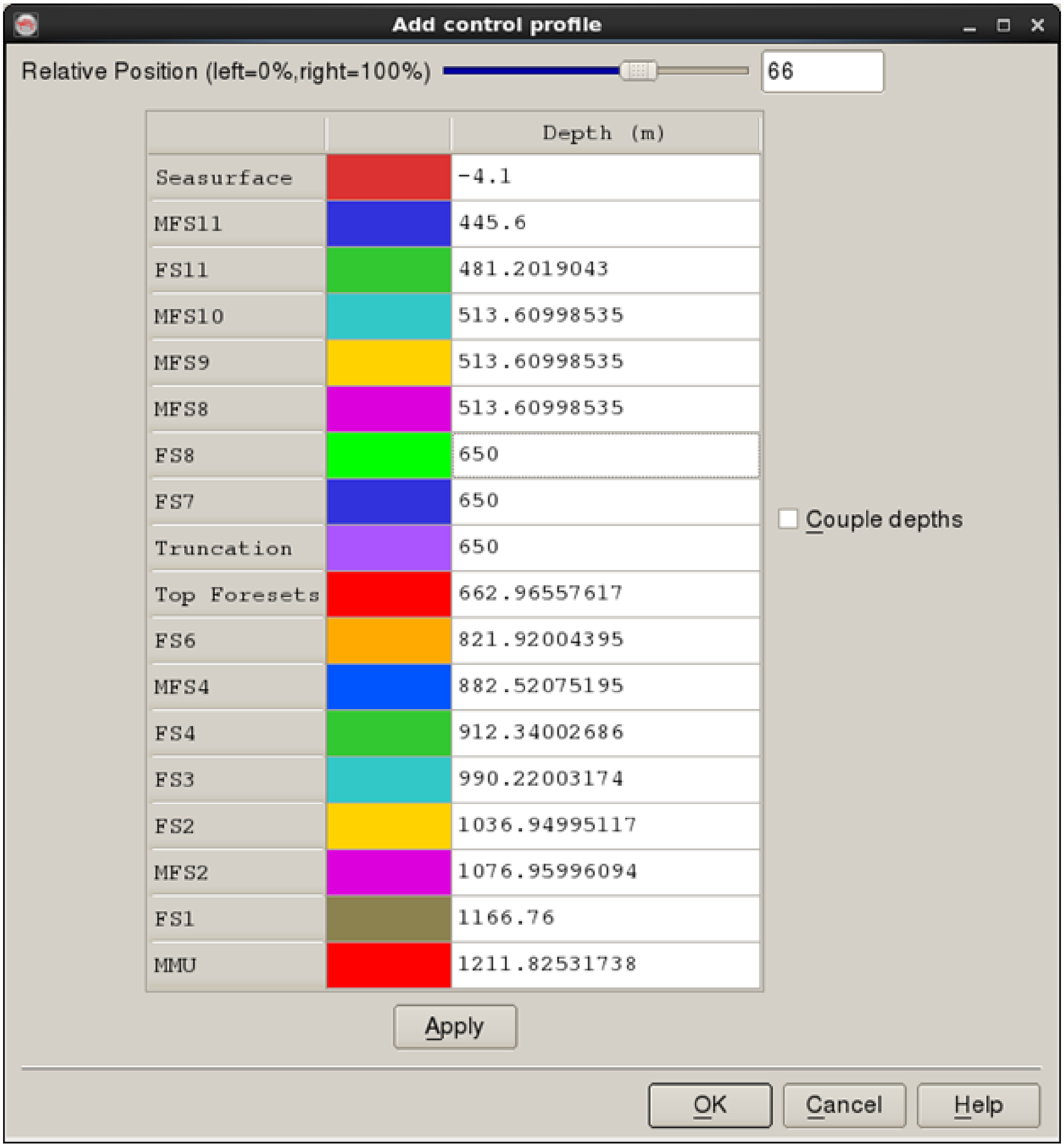
250

Acoustic Impedance

Seasurface

TrcNr=11 (629397, 6090496, 1192) Val=4391429.5 (Acoustic Impedance)

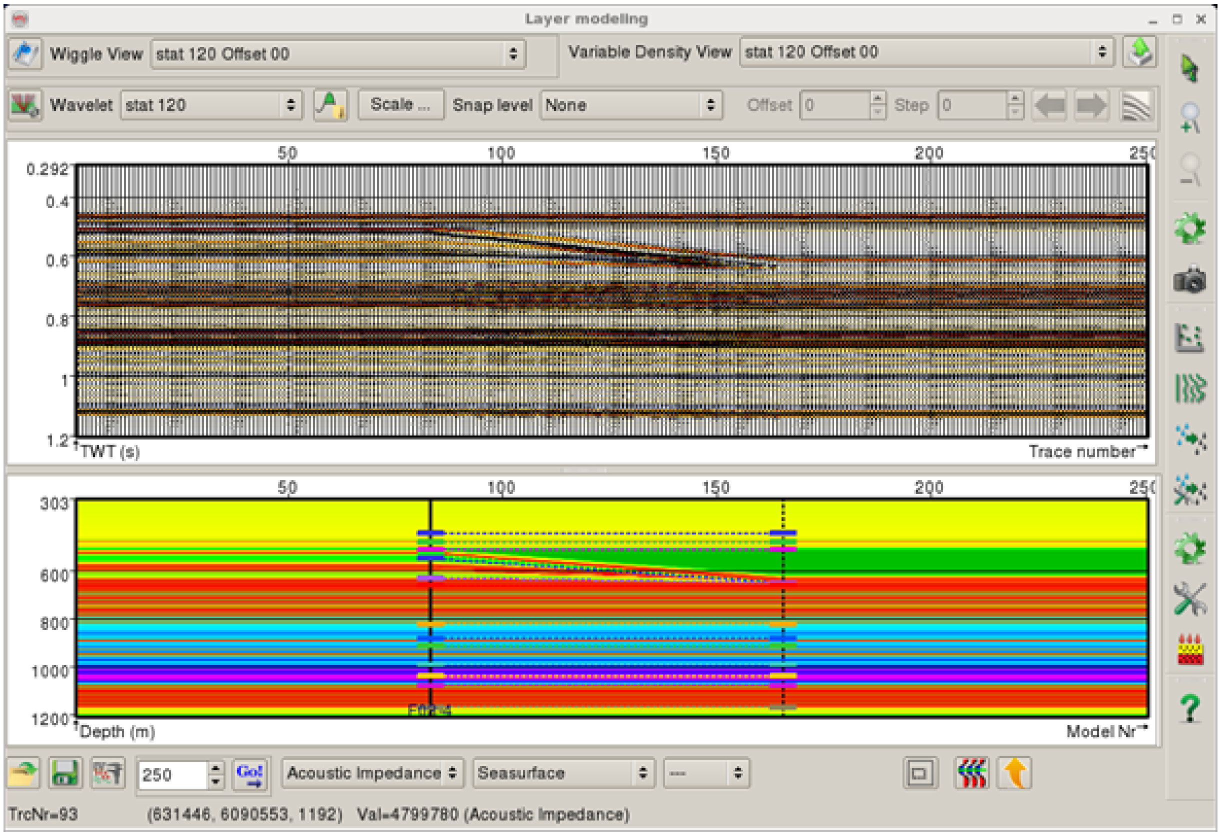
When adding a control profile, depths of a single top or of several tops simultaneously can be changed in *Add control profile* window. The depth of a single top can be modified when *Couple depths* option is unchecked. Note, that the same modified depth is assigned to those tops whose stratigraphic order is broken by the modification, while the depths of all other tops are kept unchanged. In the example below *FS8* depth is changed from 535 to 650 . Note, that *FS7* and *Truncation* tops are assigned the same depth in order to preserve the stratigraphic order of the tops. All the other depths are kept unchanged.



Relative Position (left=0%,right=100%) 66

		Depth (m)
Seasurface	🟥	-4.1
MFS11	🟦	445.6
FS11	🟩	481.2019043
MFS10	🟦	513.60998535
MFS9	🟨	513.60998535
MFS8	🟪	513.60998535
FS8	🟩	650
FS7	🟦	650
Truncation	🟪	650
Top Foresets	🟥	662.96557617
FS6	🟧	821.92004395
MFS4	🟦	882.52075195
FS4	🟩	912.34002686
FS3	🟦	990.22003174
FS2	🟨	1036.94995117
MFS2	🟪	1076.95996094
FS1	🟫	1166.76
MMU	🟥	1211.82531738

☐ Couple depths



Wiggle View stat 120 Offset 00

Variable Density View stat 120 Offset 00

Wavelet stat 120

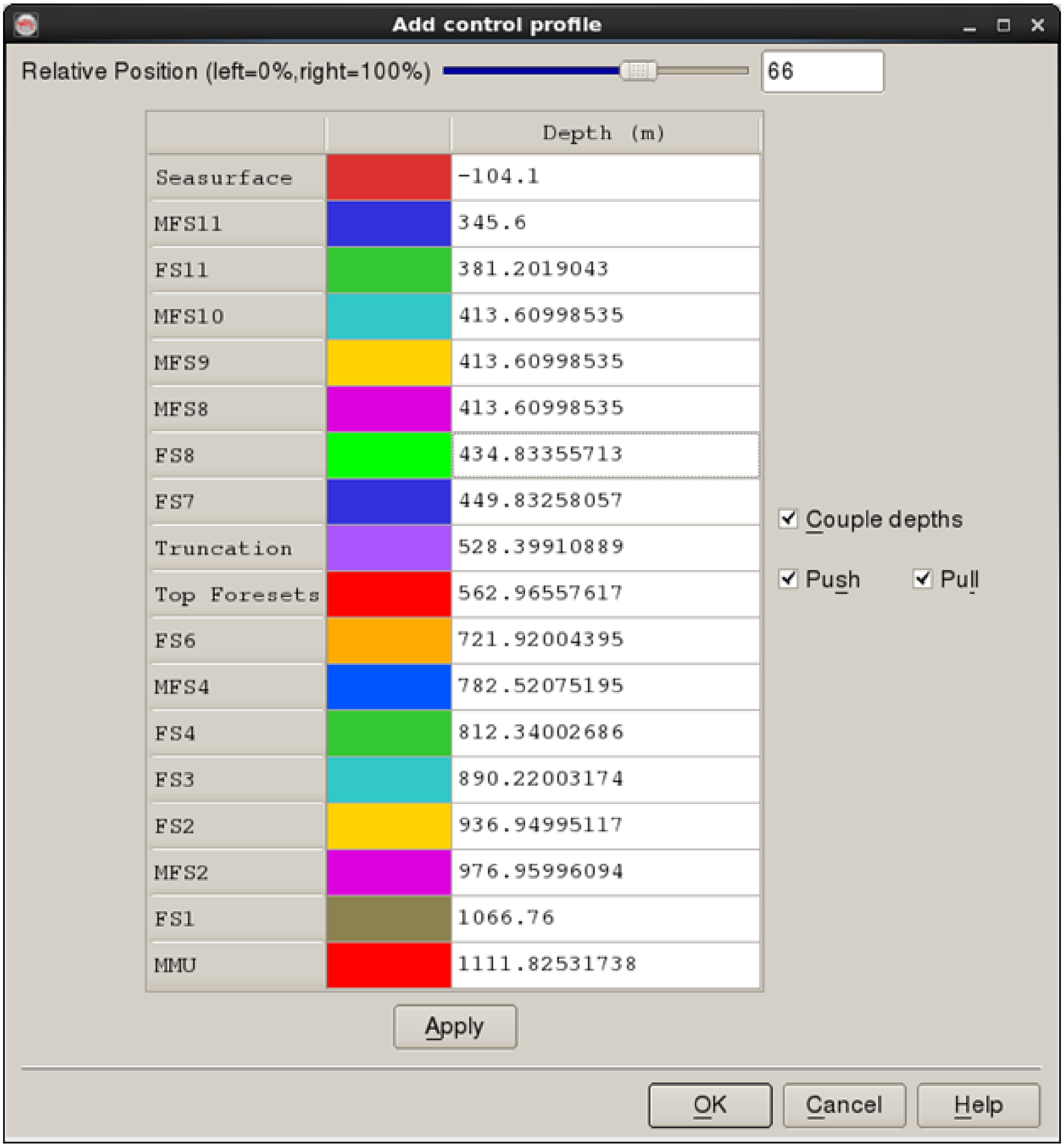
Snap level None

Offset 0 Step 0

250 Acoustic Impedance Seasurface ---

TrcNr=93 (631446, 6090553, 1192) Val=4799780 (Acoustic Impedance)

If the *Couple depths* option is checked, depths of several tops can be modified simultaneously. If both *Push* and *Pull* are checked, all tops can be shifted up or down by changing one depth by the desired shift. Note, that the stratigraphic order of tops is never broken in *Push* and *Pull* case. In the example below, the depth of *FS8* top is decreased by 100 m, thus shifting all formation tops 100 m up.

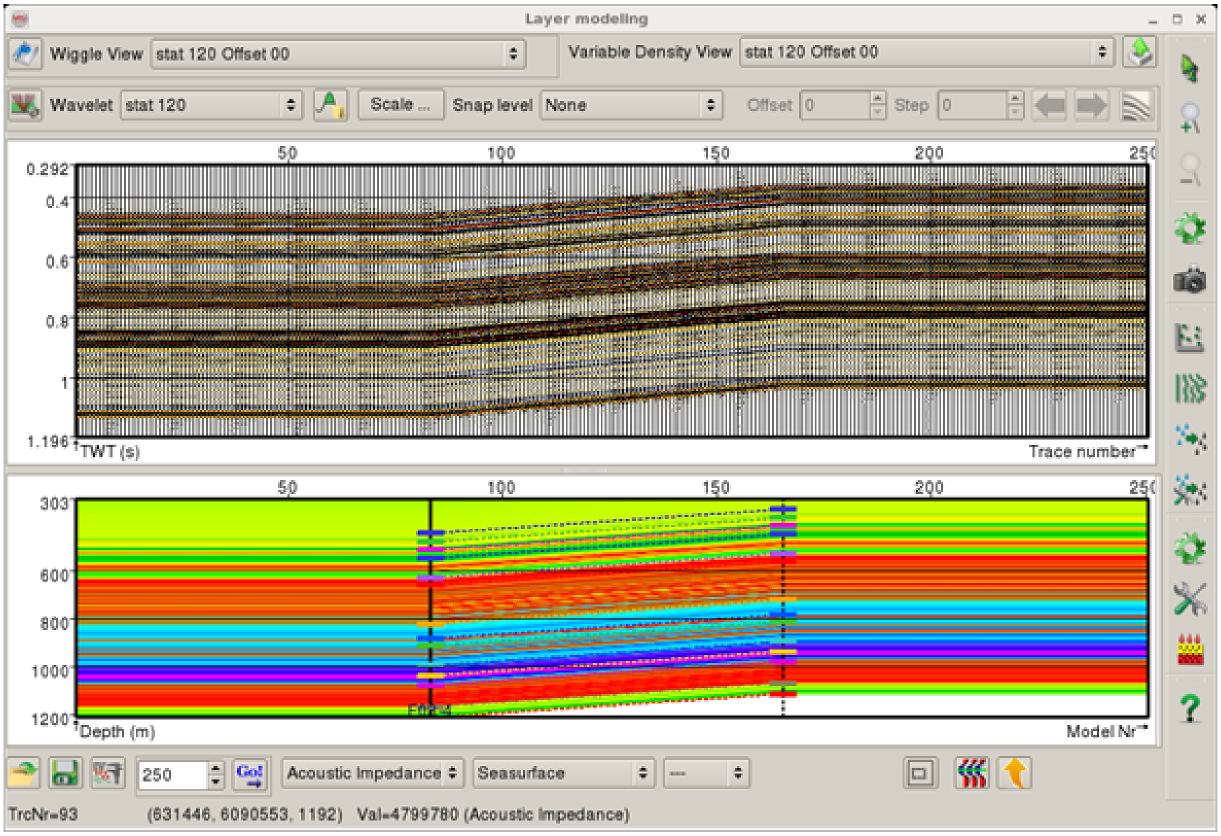


Relative Position (left=0%,right=100%) 66

		Depth (m)
Seasurface	🟥	-104.1
MFS11	🟦	345.6
FS11	🟩	381.2019043
MFS10	🟦	413.60998535
MFS9	🟨	413.60998535
MFS8	🟪	413.60998535
FS8	🟩	434.83355713
FS7	🟦	449.83258057
Truncation	🟪	528.39910889
Top Foresets	🟥	562.96557617
FS6	🟧	721.92004395
MFS4	🟦	782.52075195
FS4	🟩	812.34002686
FS3	🟦	890.22003174
FS2	🟨	936.94995117
MFS2	🟪	976.95996094
FS1	🟫	1066.76
MMU	🟥	1111.82531738

☑ Couple depths

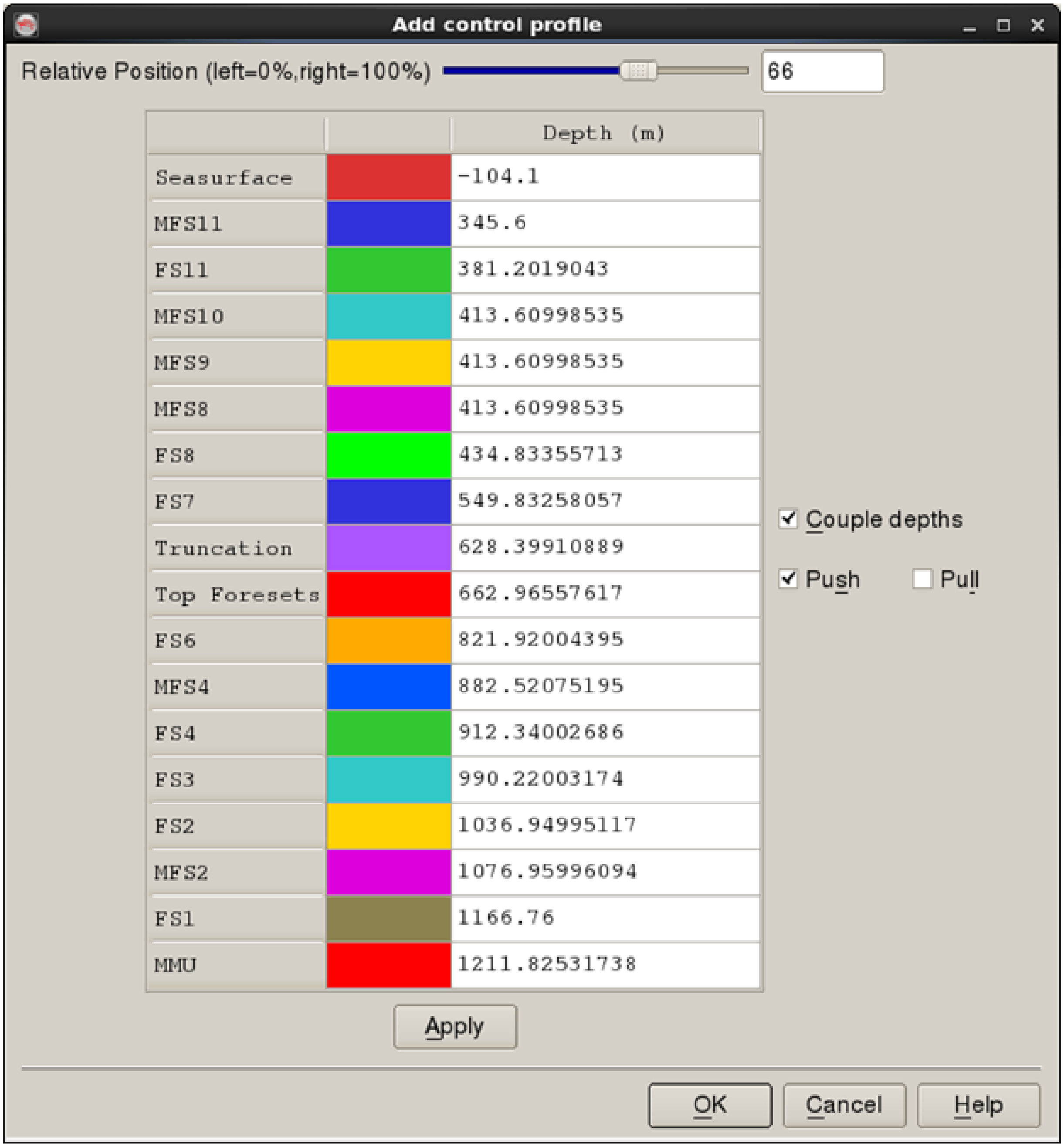
☑ Push ☑ Pull



Wiggle View: stat 120 Offset 00
Variable Density View: stat 120 Offset 00
Wavelet: stat 120
Snap level: None
250
Acoustic Impedance
Seasurface

TrcNr=93 (631446, 6090553, 1192) Val=4799780 (Acoustic Impedance)

If only *Push* is checked and the modified top is shifted up, then the tops above it are pushed up by the same amount. Similarly, if only *Push* is checked and the modified top is shifted down, then the tops below it are pushed down by the same amount. Note, that the stratigraphic order of tops is never broken in the *Push* only case. In the example below, the depth of *FS8* top was decreased by 100 m, thus shifting it and all the tops above it 100 m up, while the tops below it are kept unchanged.

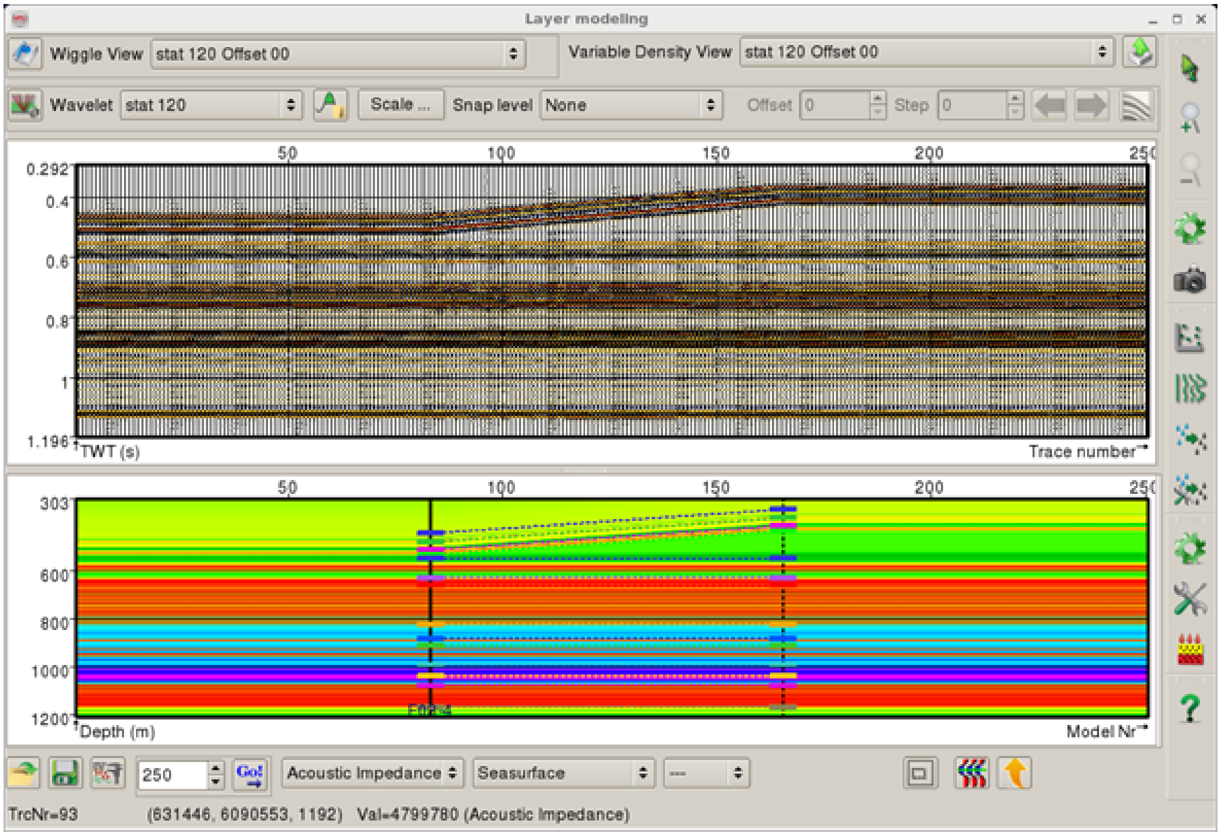


Relative Position (left=0%,right=100%) 66

		Depth (m)
Seasurface	🟥	-104.1
MFS11	🟦	345.6
FS11	🟩	381.2019043
MFS10	🟦	413.60998535
MFS9	🟨	413.60998535
MFS8	🟪	413.60998535
FS8	🟩	434.83355713
FS7	🟦	549.83258057
Truncation	🟪	628.39910889
Top Foresets	🟥	662.96557617
FS6	🟧	821.92004395
MFS4	🟦	882.52075195
FS4	🟩	912.34002686
FS3	🟦	990.22003174
FS2	🟨	1036.94995117
MFS2	🟪	1076.95996094
FS1	🟫	1166.76
MMU	🟥	1211.82531738

☑ Couple depths

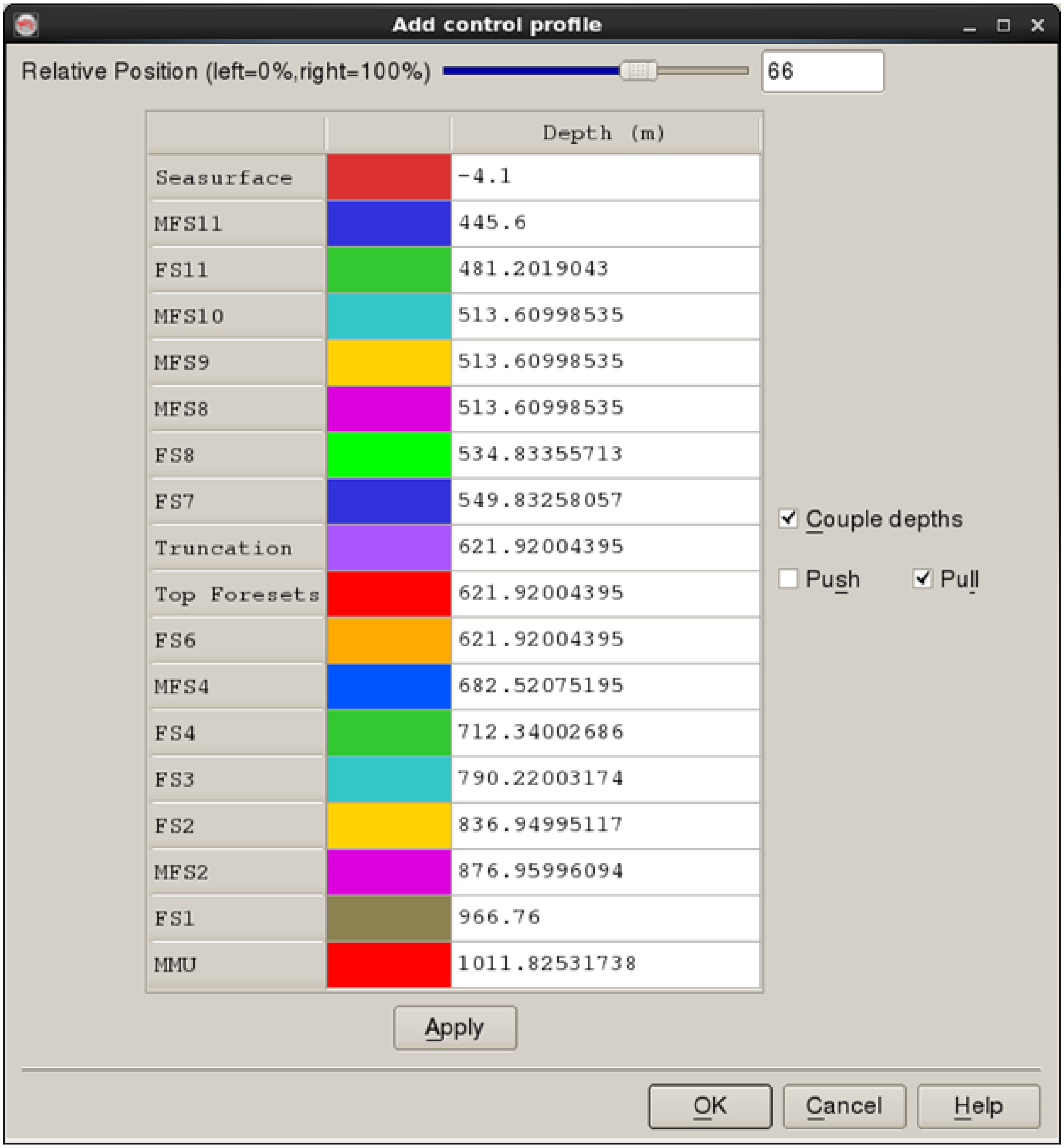
☑ Push ☐ Pull



Wiggle View stat 120 Offset 00
Variable Density View stat 120 Offset 00
Wavelet stat 120
Snap level None
Offset 0
Step 0
250
Acoustic Impedance
Seasurface

TrcNr=93 (631446, 6090553, 1192) Val=4799780 (Acoustic Impedance)

If only *Pull* is checked and the modified top is shifted up, then the tops below it are pulled up by the same amount. Similarly, if only *Pull* is checked and the modified top is shifted down, then the tops above it are pulled down by the same amount. Note, that the stratigraphic order of tops can be broken in the *Pull* only case. In the example below, the depth of *FS6* top was decreased by 200 m, thus shifting it and all the tops below it 200 m up. Note, that two tops above *FS6*, *Truncation* and *Top Foresets*, are assigned the same depth as *FS6* in order to preserve the stratigraphic order of the top. All the other tops above it are kept unchanged.

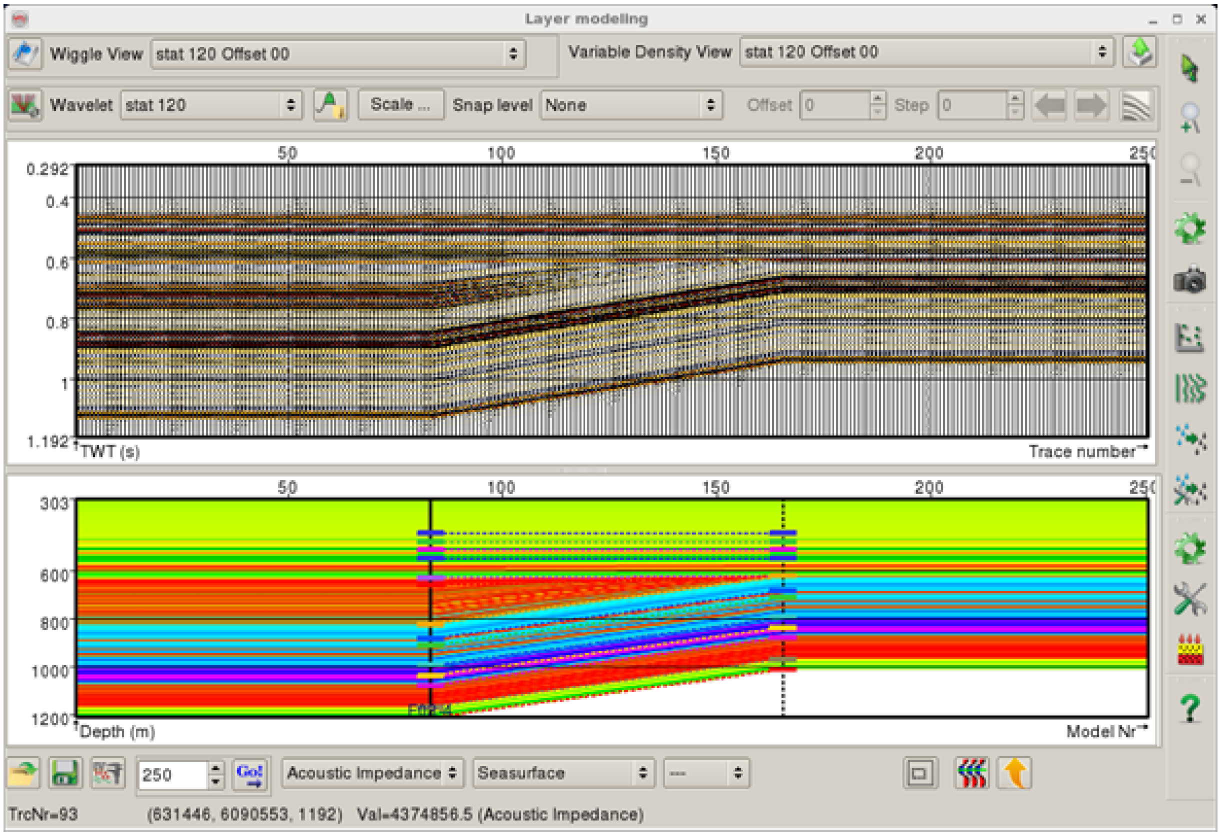


Relative Position (left=0%,right=100%) 66

		Depth (m)
Seasurface	🟥	-4.1
MFS11	🟦	445.6
FS11	🟩	481.2019043
MFS10	🟦	513.60998535
MFS9	🟨	513.60998535
MFS8	🟪	513.60998535
FS8	🟩	534.83355713
FS7	🟦	549.83258057
Truncation	🟪	621.92004395
Top Foresets	🟥	621.92004395
FS6	🟧	621.92004395
MFS4	🟦	682.52075195
FS4	🟩	712.34002686
FS3	🟦	790.22003174
FS2	🟨	836.94995117
MFS2	🟪	876.95996094
FS1	🟫	966.76
MMU	🟥	1011.82531738

☑ Couple depths

☐ Push ☑ Pull

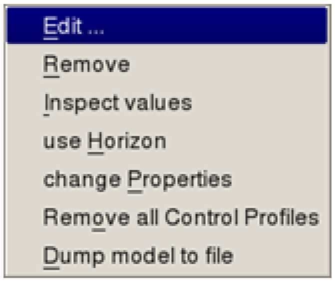


Wiggle View stat 120 Offset 00
Variable Density View stat 120 Offset 00
Wavelet stat 120
Snap level None
Offset 0
Step 0
250
Acoustic Impedance
Seasurface

TrcNr=93 (631446, 6090553, 1192) Val=4374856.5 (Acoustic Impedance)

Edit well or control profile

Right click on added wells or control profiles allows to choose the *Edit* option.

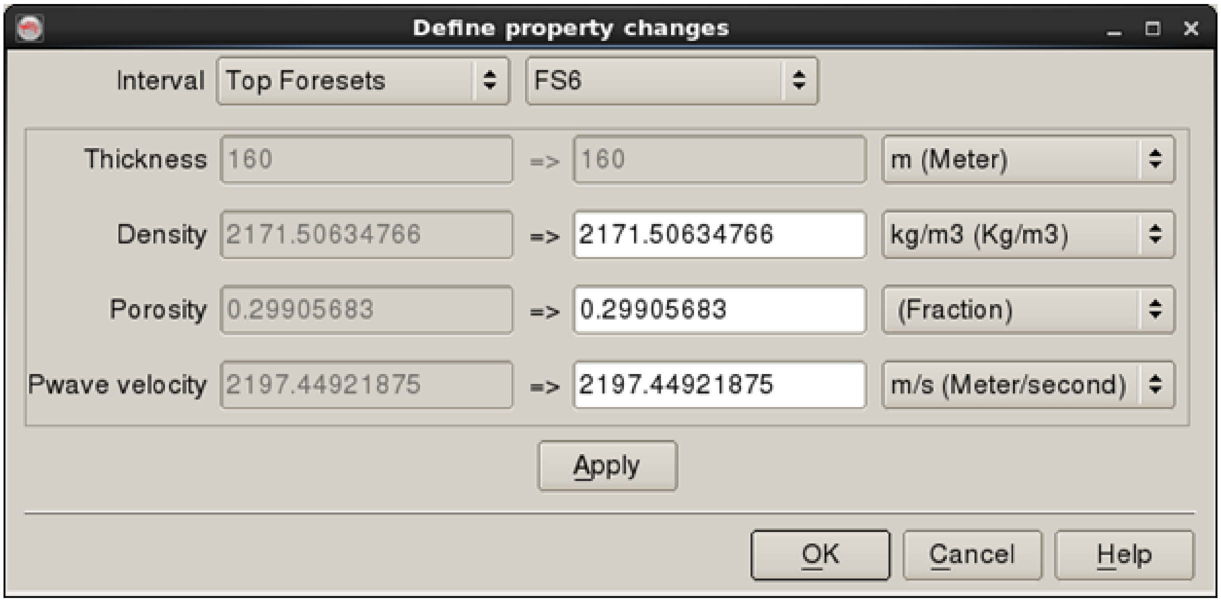


Edit ...
Remove
Inspect values
use Horizon
change Properties
Remove all Control Profiles
Dump model to file

Edit well parameters and *Edit control profile* windows are almost identical to the *Add well* and *Add control profile* windows respectively and allow changes of already added objects. Optionally, a single well or control profile can be removed by choosing *Remove* option.

Define Property Changes

Right click change Properties option brings up a Define property changes window. Properties of a selected interval can be changed in such a manner that individual samples of a property are scaled to have the user-specified average in the given interval. Note, that this window for added wells does not allow thickness to be changed, whereas the same window for a control profile does allow.


Define property changes

Interval Top Foresets FS6

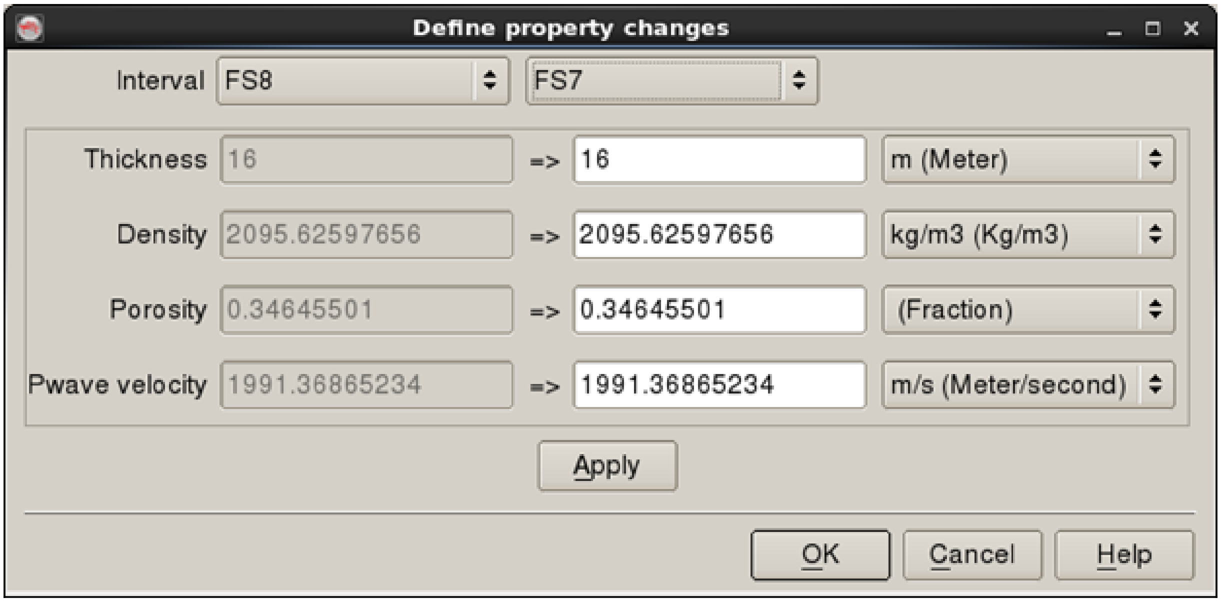
Thickness 160 => 160 m (Meter)

Density 2171.50634766 => 2171.50634766 kg/m3 (Kg/m3)

Porosity 0.29905683 => 0.29905683 (Fraction)

Pwave velocity 2197.44921875 => 2197.44921875 m/s (Meter/second)

Define property change for a well


Define property changes

Interval FS8 FS7

Thickness 16 => 16 m (Meter)

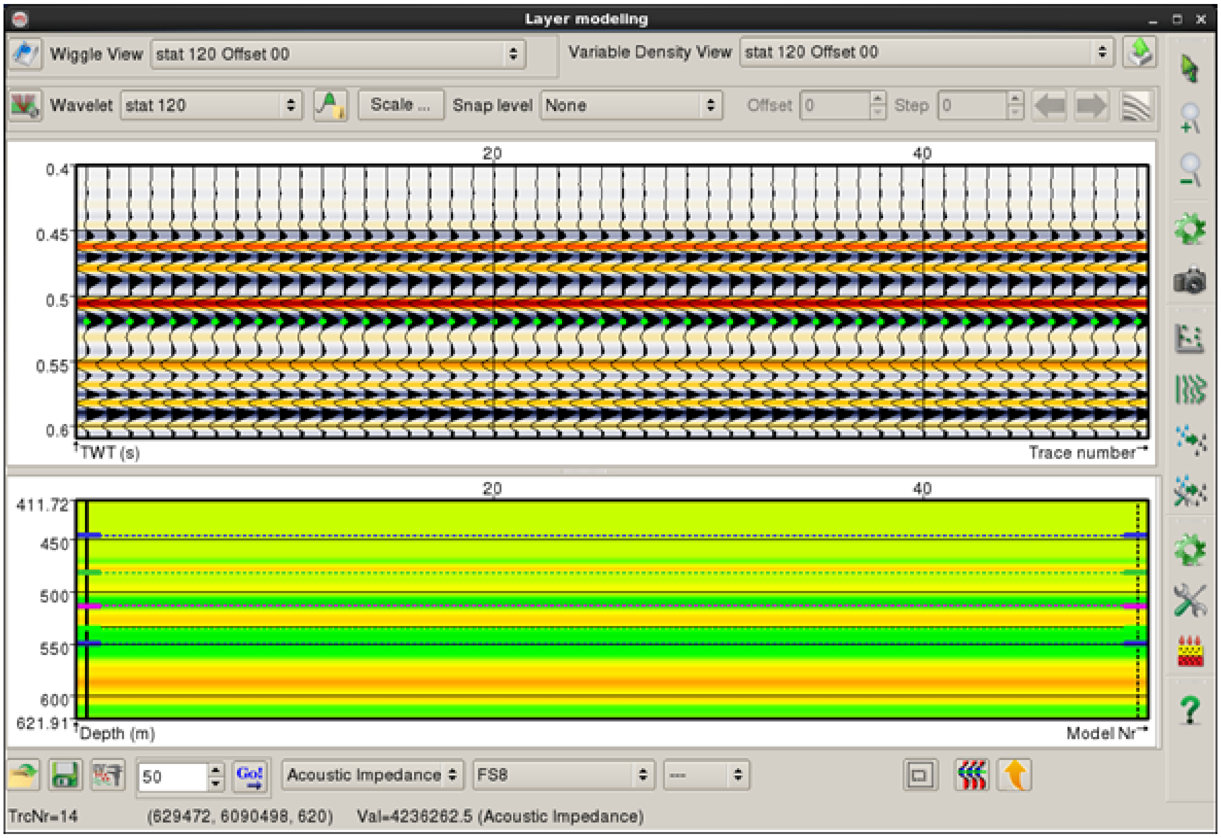
Density 2095.62597656 => 2095.62597656 kg/m3 (Kg/m3)

Porosity 0.34645501 => 0.34645501 (Fraction)

Pwave velocity 1991.36865234 => 1991.36865234 m/s (Meter/second)

Define property change for a control profile

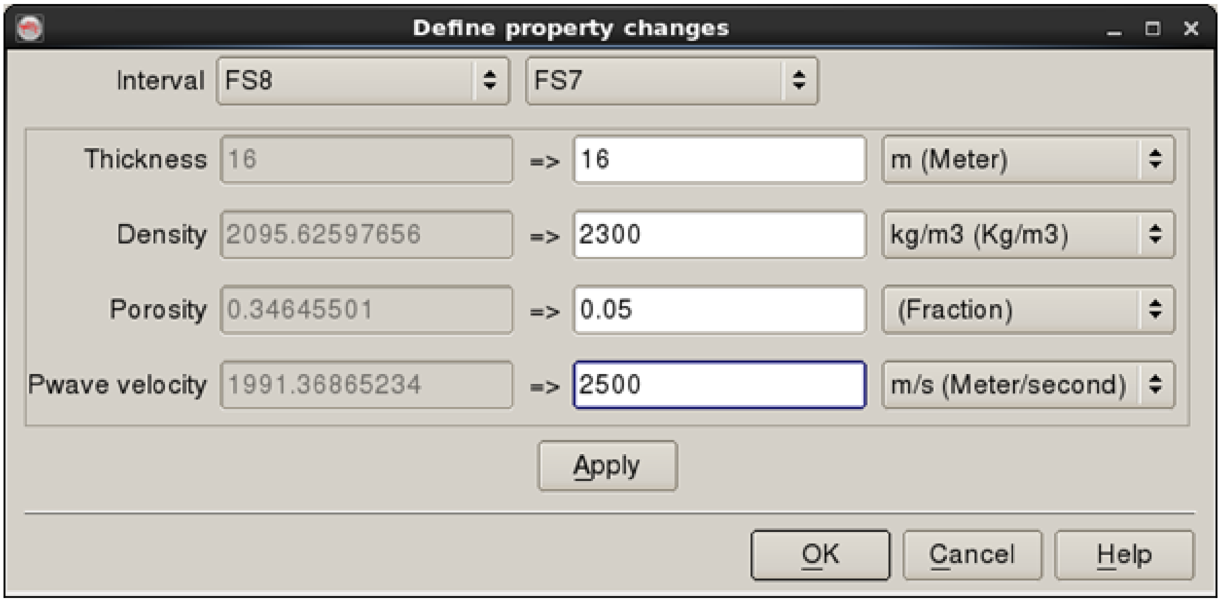
The first example shows construction of a simple model of a lateral change in porosity and acoustic impedance. The model consists of one well at the far-left side and one control profile at the far-right side of the profile. The two lower markers, light green and blue, correspond to *FS8* and *FS7* tops respectively. Green dots found on synthetic seismic panel correspond to *FS8* top. So far, the properties are not varied along the profile.



Wiggle View stat 120 Offset 00
Variable Density View stat 120 Offset 00
Wavelet stat 120
Snap level None
Offset 0
Step 0
50
Acoustic Impedance
FS8

TrcNr=14 (629472, 6090498, 620) Val=4236262.5 (Acoustic Impedance)

Right click on the control profile and choose the *change Properties* option. Once the interval *FS8-FS7* is selected in the *Define property change* window, its greyed-out current properties are shown in the left column. User-modified properties are typed in the right column. Clicking *Apply* or OK re-generates pseudowells and synthetic seismic to reflect the changes.



Define property changes

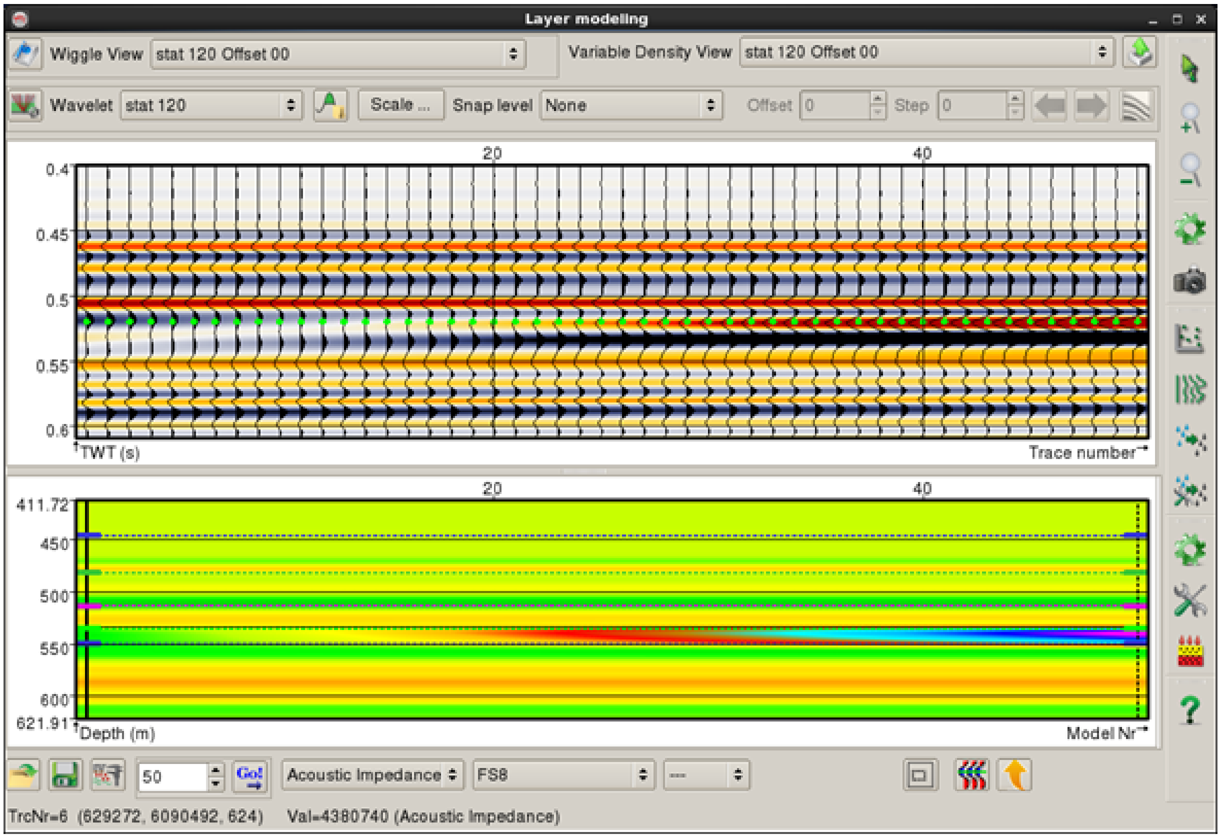
Interval: FS8 / FS7

Thickness: 16 => 16 m (Meter)

Density: 2095.62597656 => 2300 kg/m3 (Kg/m3)

Porosity: 0.34645501 => 0.05 (Fraction)

Pwave velocity: 1991.36865234 => 2500 m/s (Meter/second)



Layer modeling

Wiggle View: stat 120 Offset 00

Variable Density View: stat 120 Offset 00

Wavelet: stat 120

Snap level: None

Offset: 0

Step: 0

50

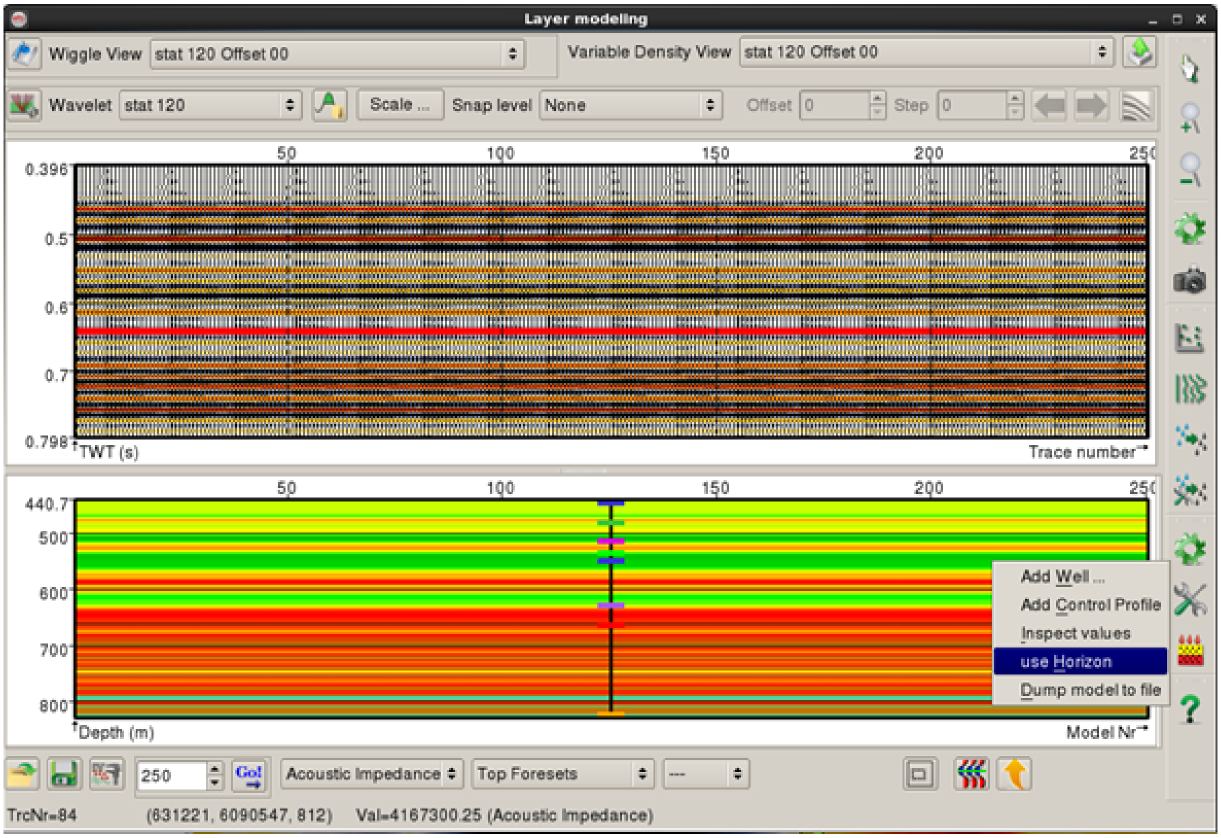
Acoustic Impedance

FS8

TrcNr=6 (629272, 6090492, 624) Val=4380740 (Acoustic Impedance)

Use Horizon

Existing horizons from the project database can be used to easily and quickly mimic an interpreted (3D or 2D) structure and align the pseudo-wells along it in the profile modeling. This workflow is based on an automated creation of a number of *Control Profiles* that follow the depth trend of the selected horizon. It can be started by right clicking in the bottom part of *Layer modeling* window and selecting use *Horizon* option. The layout of *Use horizon for profile* window depends on the number of wells used in the modeling.



Wiggle View: stat 120 Offset 00

Variable Density View: stat 120 Offset 00

Wavelet: stat 120

Snap level: None

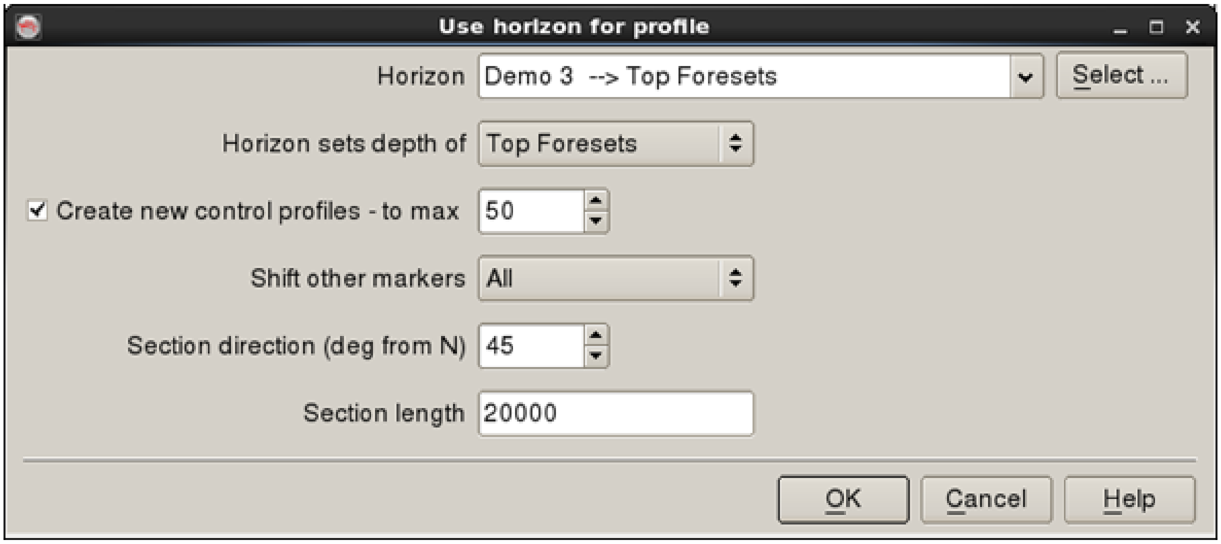
Offset: 0

Step: 0

250 | Acoustic Impedance | Top Foresets | ---

TrcNr=84 (631221, 6090547, 812) Val=4167300.25 (Acoustic Impedance)

If there is only one (real) well loaded in the layer, the following dialogue window will appear:



Horizon: Demo 3 --> Top Foresets

Horizon sets depth of: Top Foresets

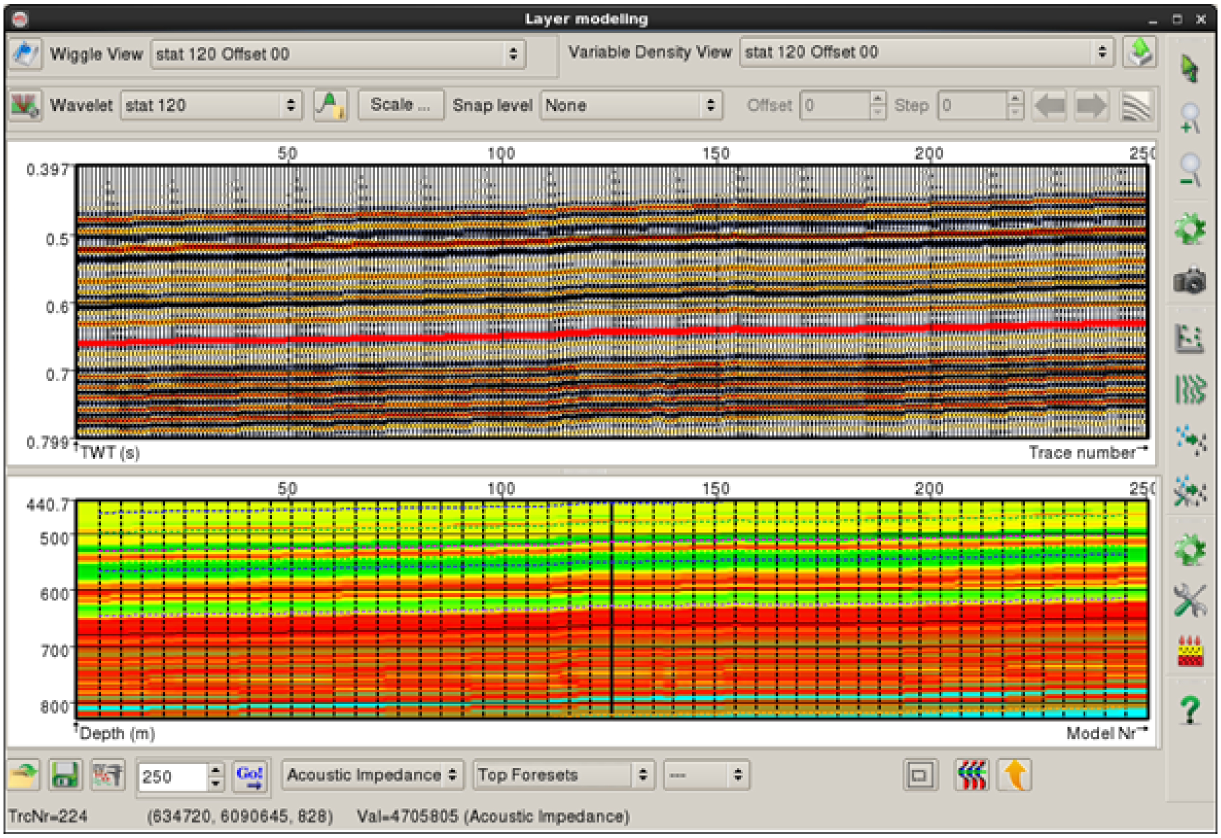
☑ Create new control profiles - to max: 50

Shift other markers: All

Section direction (deg from N): 45

Section length: 20000

In the *Use horizon for profile* window the user must select a horizon (2D or 3D) on which the simulation of the structure is based. Then, the corresponding formation top has to be chosen. The algorithm adds a user-defined or automatically computed (if unchecked) number of control profiles. At these control profiles, depths of the chosen top are changed using the selected horizon. Linear interpolation of the top depths is performed between them. Handling of other formation tops is set with Shift other markers option to one of the following: *All, No, Only Above* or *Only Below. Section direction* and *section length* should be thought of as an orientation and a length of a 2D profile through the well along which the specified horizon is extracted. The example below shows the case when a single *F03-4* well is used in the modeling. Note, that it is a zoom-in, red line on synthetic seismic corresponds to *Top Foresets*. Specifying parameters, as shown and described above, and clicking OK re-generates the pseudowells. Control profiles shown with dashed line guide the structural modeling.



Wiggle View: stat 120 Offset 00

Variable Density View: stat 120 Offset 00

Wavelet: stat 120

Snap level: None

Offset: 0

Step: 0

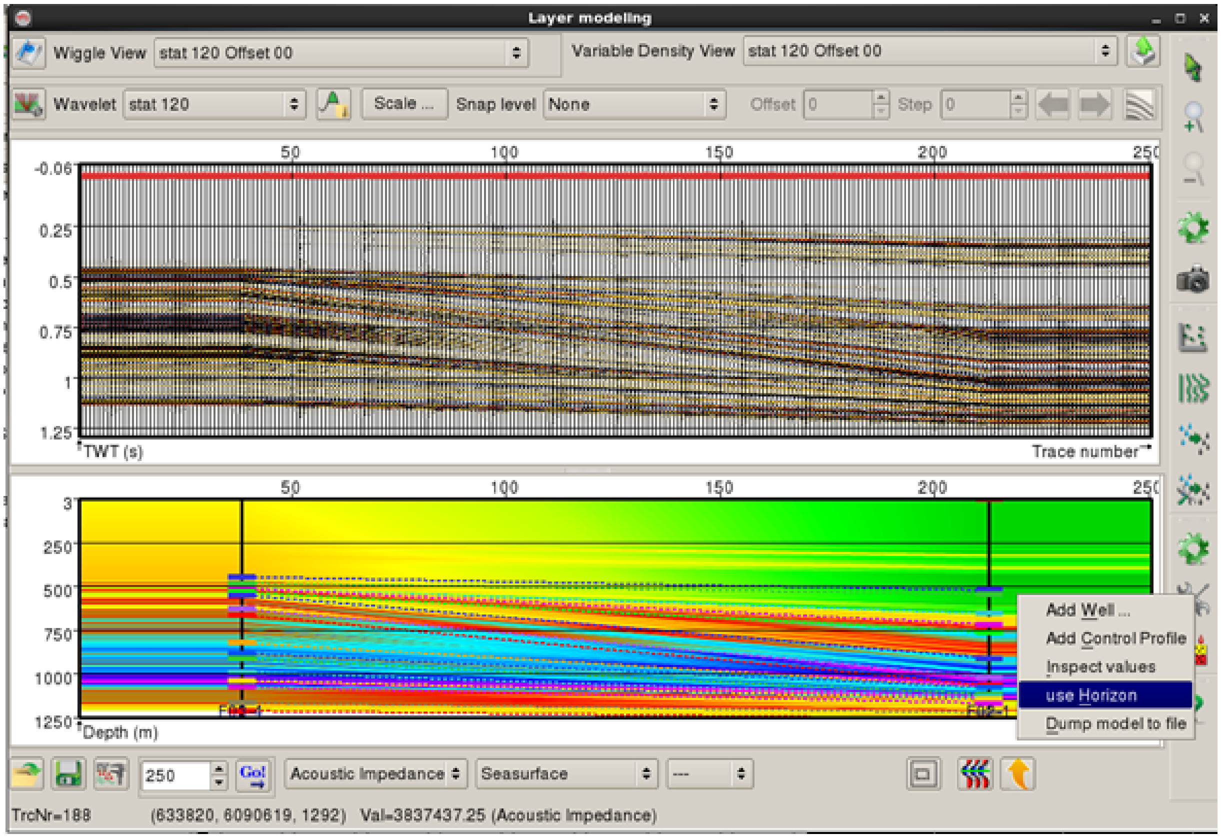
250

Acoustic Impedance

Top Foresets

TrcNr=224 (634720, 6090645, 828) Val=4705805 (Acoustic Impedance)

If there are more than one (real) wells loaded in the layer, the *use Horizon* option will bring-up a slightly different dialogue. Two last options from a single well case are eliminated, because well locations determine the direction of the profile along which the specified horizon is extracted.



Wiggle View: stat 120 Offset 00

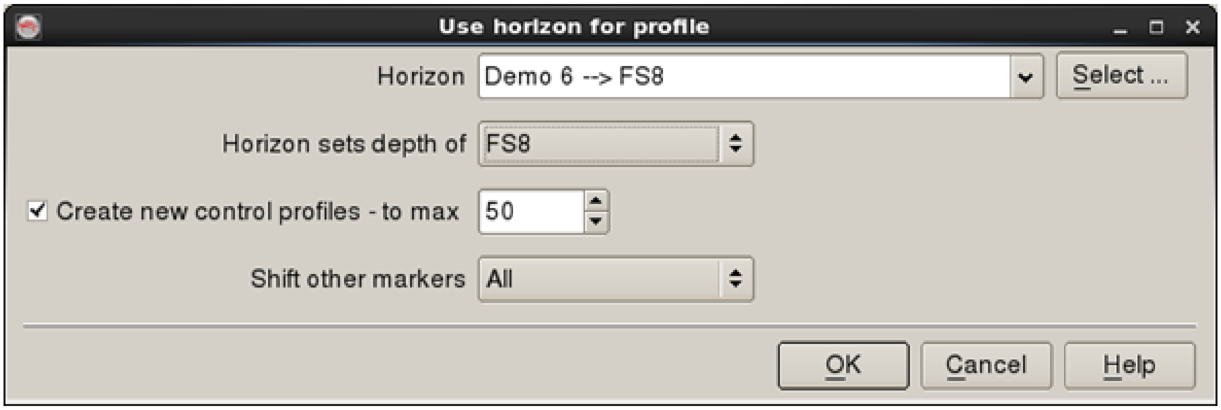
Variable Density View: stat 120 Offset 00

Wavelet: stat 120

Snap level: None

250 Acoustic Impedance Seasurface ---

TrcNr=188 (633820, 6090619, 1292) Val=3837437.25 (Acoustic Impedance)



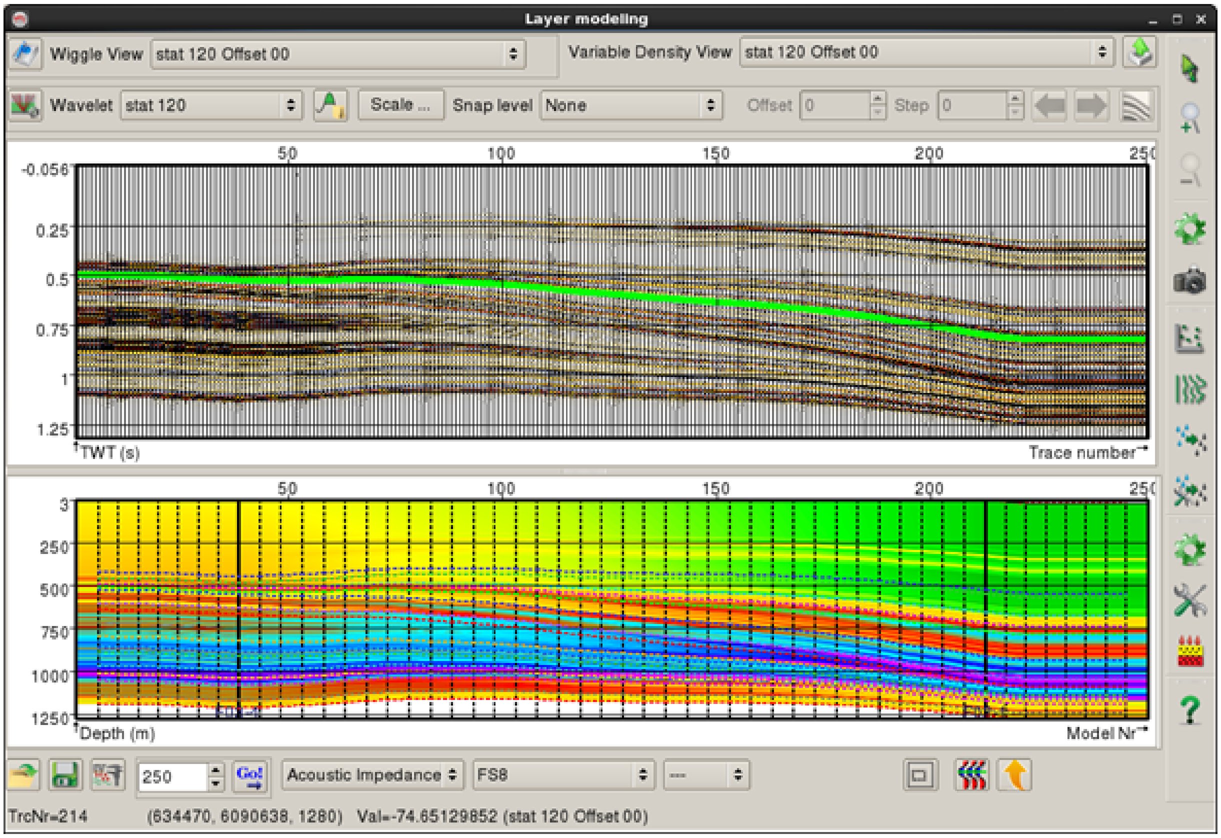
Horizon: Demo 6 --> FS8

Horizon sets depth of: FS8

☑ Create new control profiles - to max: 50

Shift other markers: All

The example below shows the case when two wells, *F03-4* and *F02-1*, are used in the modeling. Parameters are set as in the figure above. Upon clicking OK in the above window, control profiles are added as dashed lines, and pseudowells are re-generated as shown below. The green line on synthetic seismic display corresponds to *FS8* top.

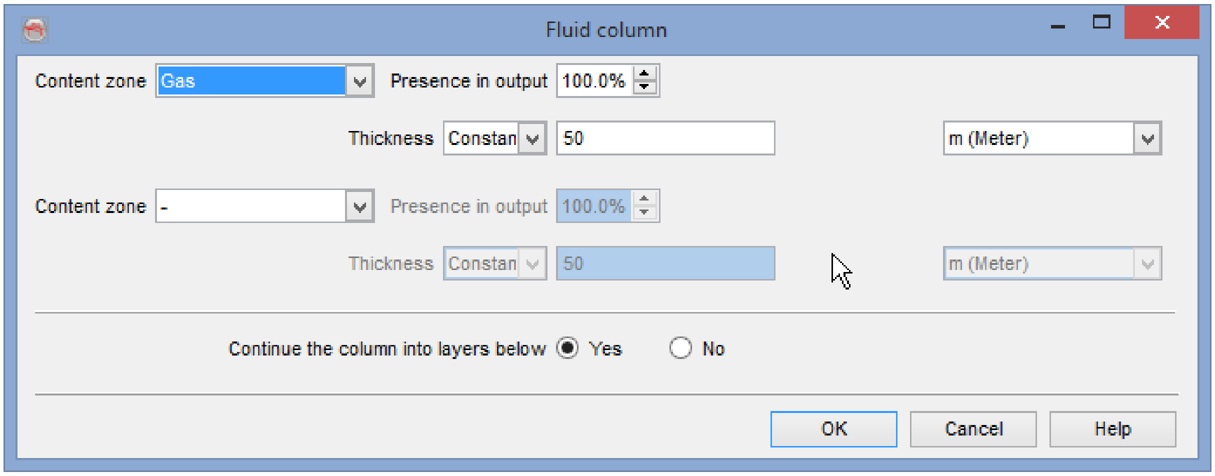


Finally, it is useful to note that depth trends of multiple horizons can be used to build a single profile model, by repeating the above workflow for different horizons and shifting only the corresponding marker each time. Perhaps, in such a case a much quicker option is to use one of the pre-defined workflows called ***Model an existing line through well(s)*** .

Add, Edit Existing Well

Control points for the profile modeling are added with a right-click where it says "Click to add a well". Before the first control point is added you will need to **specify the list of logs** that will be interpolated.

- **Add the first control point: Blocking a real well to generate the pseudo-well**
  Clicking in the bottom part of the main layer modeling window to add a well, will first pop-up the modeling **parameter selection** window. Once the process parameters are set, the program is ready to start loading the first real well for generating pseudo-well models. The following dialogue window is used for adding (and later editing) a well.


Fluid column

Content zone: Gas — Presence in output: 100.0%

Thickness: Constan — 50 — m (Meter)

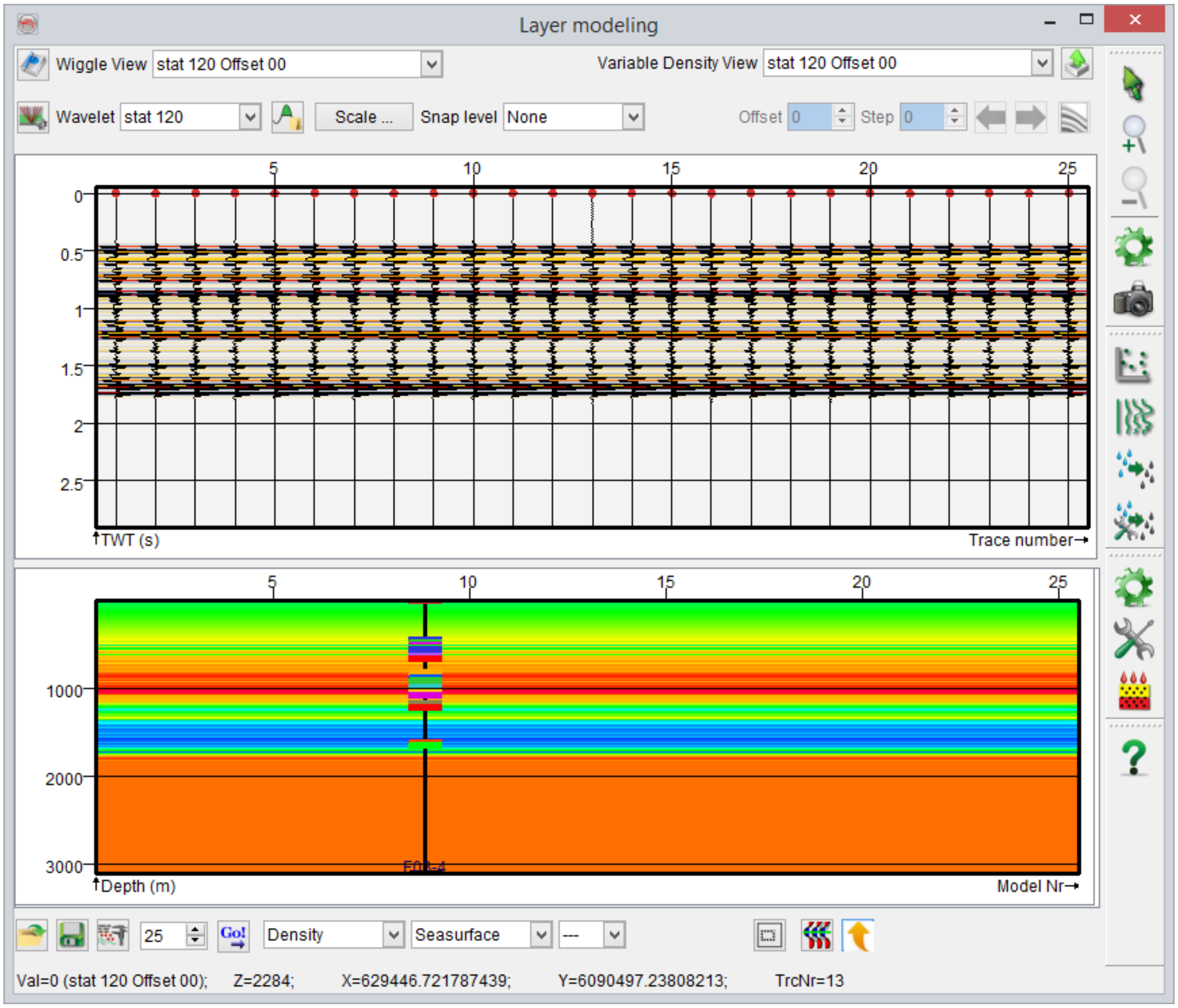
Content zone: - — Presence in output: 100.0%

Thickness: Constan — 50 — m (Meter)

Continue the column into layers below ● Yes ○ No

Definition of the first control point from well F03-4

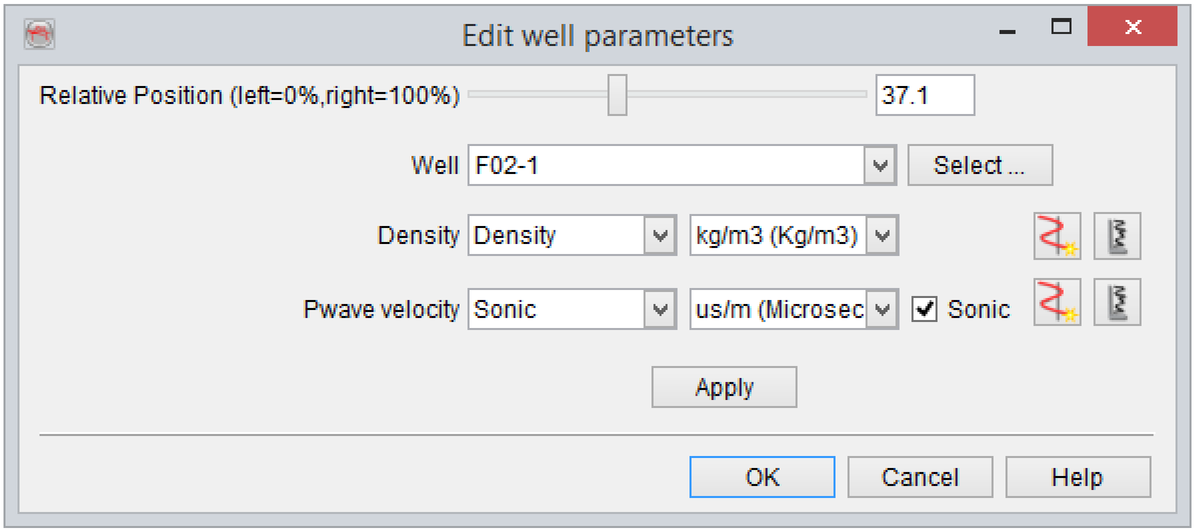
The *Relative Position* element specifies where the new well will be added in the profile. Here, 0% corresponds to the left-most end of the profile whereas 100% corresponds to the right-most end. For example, if N is the total number of pseudo-wells to be generated (displayed on the bottom left corner, left of the Go button), using '33%' as the relative position will add the new well at position number Nx33/100. Afterward, select an existing real well from the OpendTect database for the blocking, and a stored log corresponding to various properties, with the corresponding unit. The log units are read from the project database and should not be changed; SynthRock will not convert the units automatically. Here, there is also a functionality to create new input logs from the existing well logs (e.g. Density from Sonic), on the fly, by clicking on . Additionally, a log can be viewed for QC purposes by pressing this button. Pressing *Ok* or *Apply* will create a number of pseudo-wells by blocking the real well logs. An already added real well is displayed as a solid vertical line.



Val=0 (stat 120 Offset 00); Z=2284; X=629446.721787439; Y=6090497.23808213; TrcNr=13

25 pseudo-wells created from F03-4 well

- **Editing an existing well:** Using the above mentioned method more wells can be added. The edition of the added (real) well logs or of their relative position, can be done by right-clicking close to the solid vertical line (corresponding to the well) and then selecting *Edit*, to pop-up the dialogue window explained in the section above.



Relative Position (left=0%,right=100%) 37.1

Well F02-1

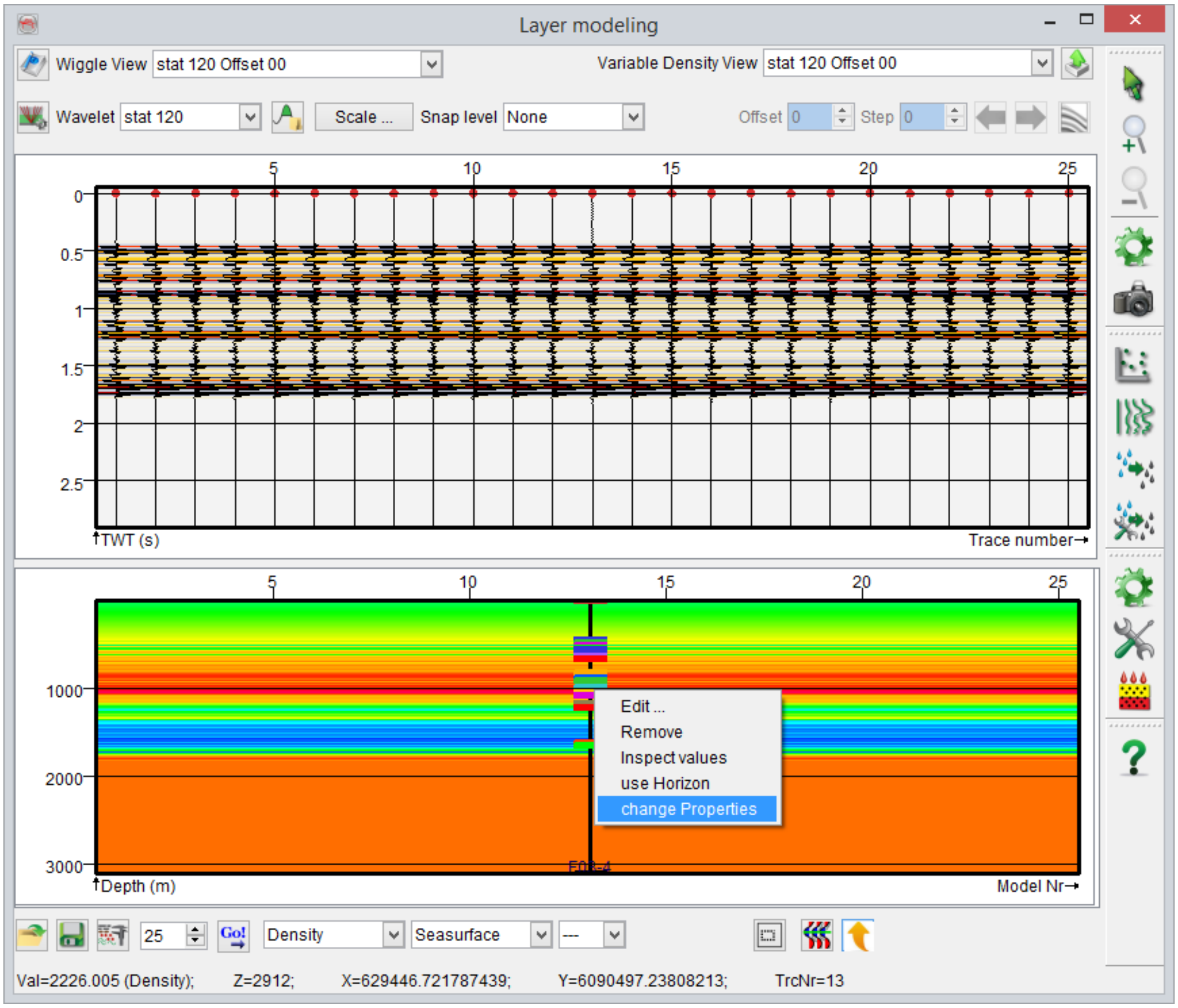
Density Density kg/m3 (Kg/m3)

Pwave velocity Sonic us/m (Microsec ☑ Sonic

Editing the relative position of well F03-4

Define Property Changes

Right clicking on an added well, in the lower part of *Profile* module window, can be used to *Edit Properties* of a part of that well. The image below shows a very simple model in the *Profile* module, having 25 identical pseudo-wells based on one real well. The lower window here displays a Density profile.



Wiggle View: stat 120 Offset 00

Variable Density View: stat 120 Offset 00

Wavelet: stat 120

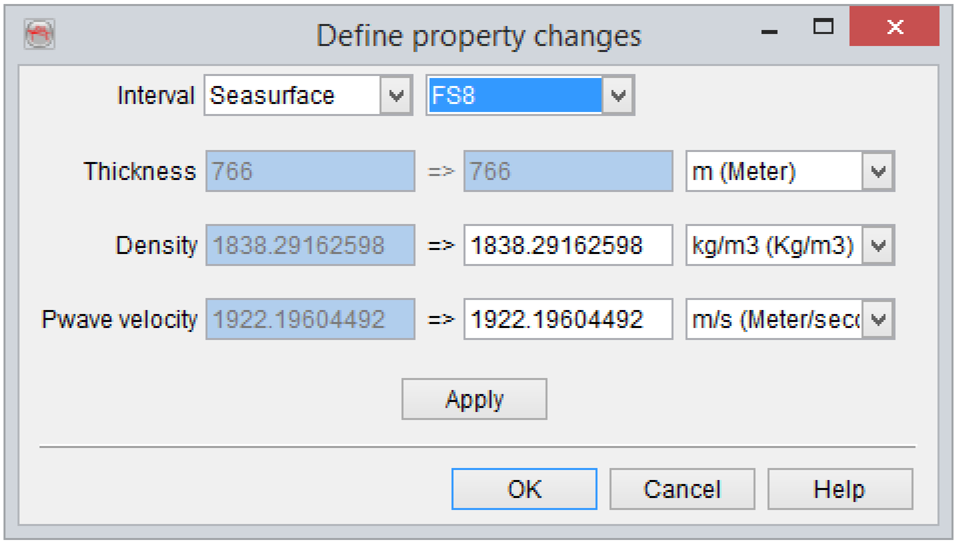
Snap level: None

Offset: 0 Step: 0

25 Density Seasurface ---

Val=2226.005 (Density); Z=2912; X=629446.721787439; Y=6090497.23808213; TrcNr=13

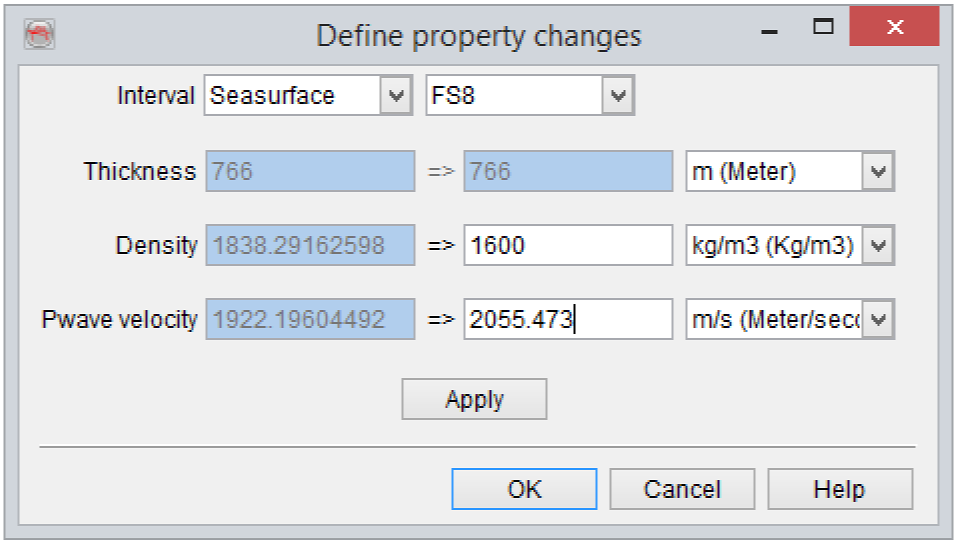
The *Define property changes* dialogue shows thickness and average values of various layer properties (e.g. Density, Vp, AI etc.) in a given (vertical) interval, for a real well.



Interval Seasurface FS8
Thickness 766 => 766 m (Meter)
Density 1838.29162598 => 1838.29162598 kg/m3 (Kg/m3)
Pwave velocity 1922.19604492 => 1922.19604492 m/s (Meter/sec

Original average property values in a particular interval

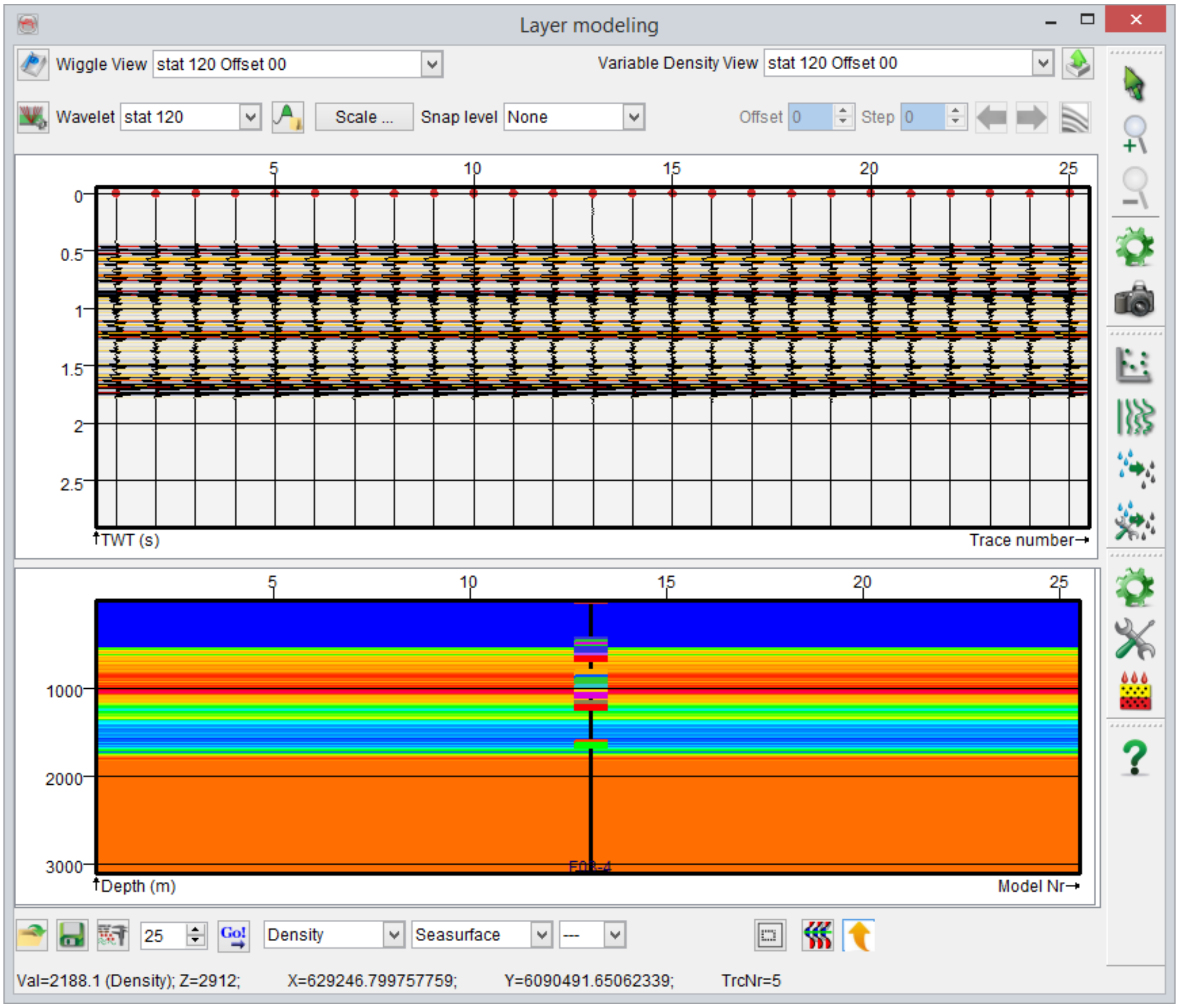
Here, the user can scale all the samples of one or more properties in that interval by specifying a different average value. This essentially means that the individual samples of a property will be scaled, so as to have the user-specified average in the given interval.



Interval Seasurface FS8
Thickness 766 => 766 m (Meter)
Density 1838.29162598 => 1600 kg/m3 (Kg/m3)
Pwave velocity 1922.19604492 => 2055.473 m/s (Meter/sec

Edited average property (Density) values in a particular interval

Pressing *Apply* or *Ok* will update the layer properties of the well, and consequently of the pseudo-wells and their corresponding synthetic seismic data.



Wiggle View stat 120 Offset 00

Variable Density View stat 120 Offset 00

Wavelet stat 120

Snap level None

Offset 0

Step 0

25

Density

Seasurface

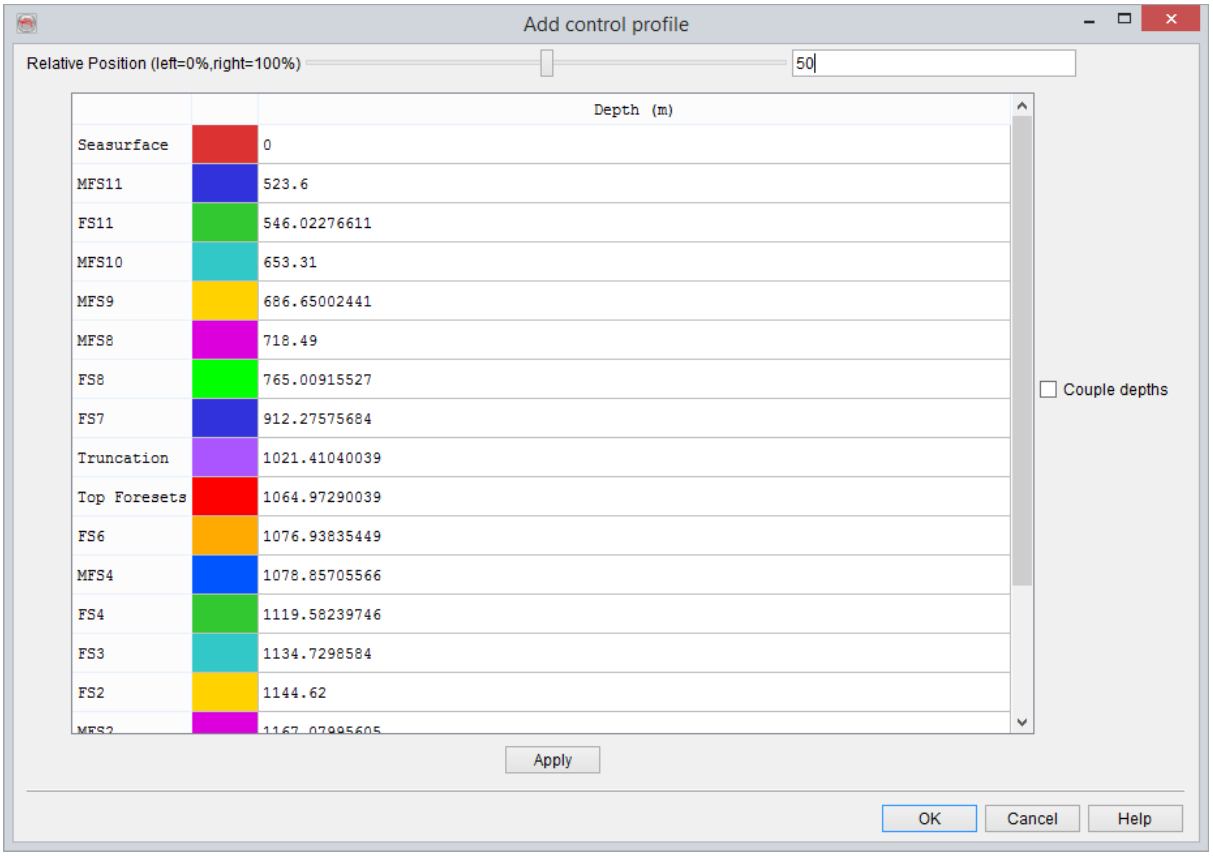
```
Val=2188.1 (Density); Z=2912;     X=629246.799757759;     Y=6090491.65062339;     TrcNr=5
```

In real world scenarios, where multiple wells are used in modeling, the layer properties for the pseudo-wells between the edited wells and the unedited wells, are recomputed using horizontal interpolation.

Add, Edit Control Profile

The second type of available control points in the profile modeling, other than the (real) wells are called *Control Profiles*. They are basically used to introduce a structural element in the modeling; to essentially steer the log data interpolation along a well defined structure, which is in fact defined by changing depths of a number of well markers at various positions in the profile.

- **Adding a new control profile:** Control profiles are added with a right-click where it says Add Control Profile. Just like for the well based control points, the Relative Position element specifies where the new well will be added in the profile. Here, 0% corresponds to the left-most end of the profile and 100% corresponds to the right-most end. After specifying the position, the depth of various markers can be edited at this position.



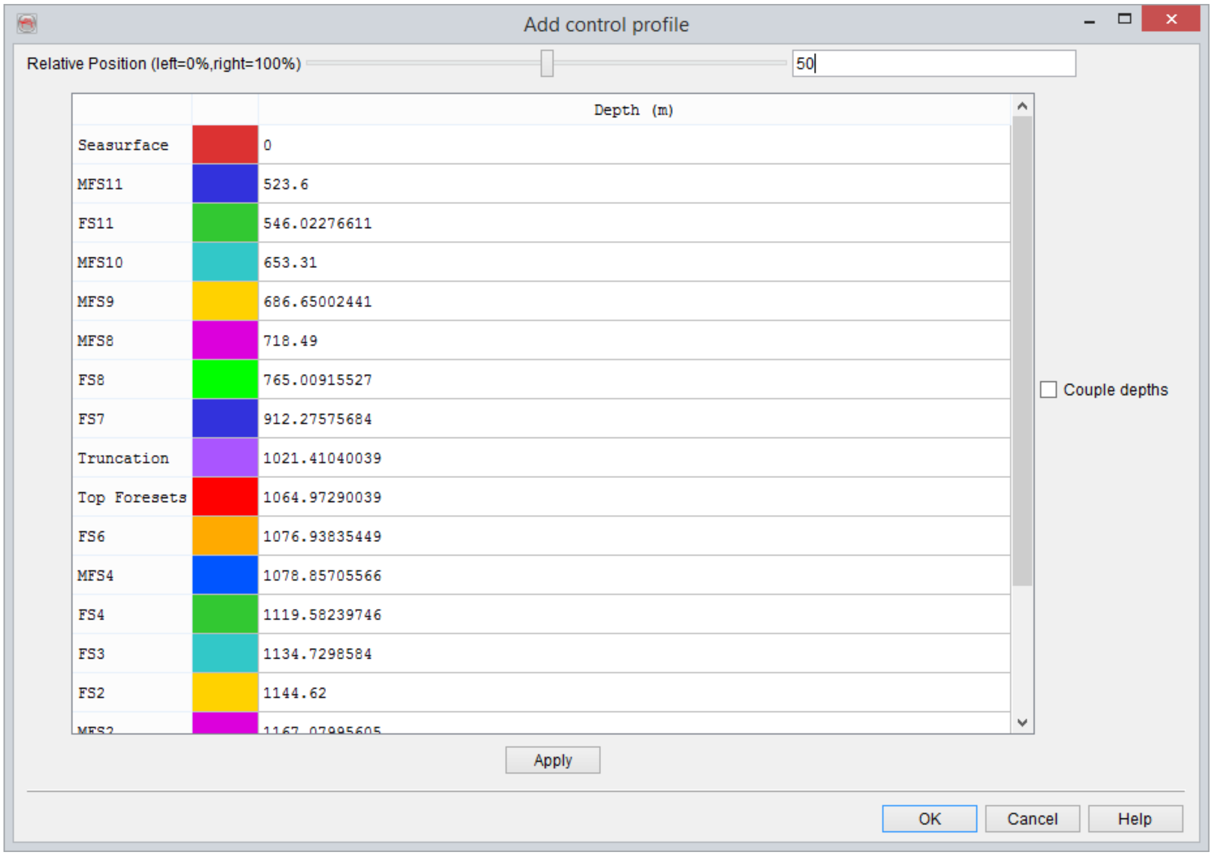
Relative Position (left=0%,right=100%) 50

☐ Couple depths

		Depth (m)
Seasurface	🟥	0
MFS11	🟦	523.6
FS11	🟩	546.02276611
MFS10	🟦	653.31
MFS9	🟨	686.65002441
MFS8	🟪	718.49
FS8	🟩	765.00915527
FS7	🟦	912.27575684
Truncation	🟪	1021.41040039
Top Foresets	🟥	1064.97290039
FS6	🟧	1076.93835449
MFS4	🟦	1078.85705566
FS4	🟩	1119.58239746
FS3	🟦	1134.7298584
FS2	🟨	1144.62
MFS2	🟪	1167.07995605

Defining a control profile

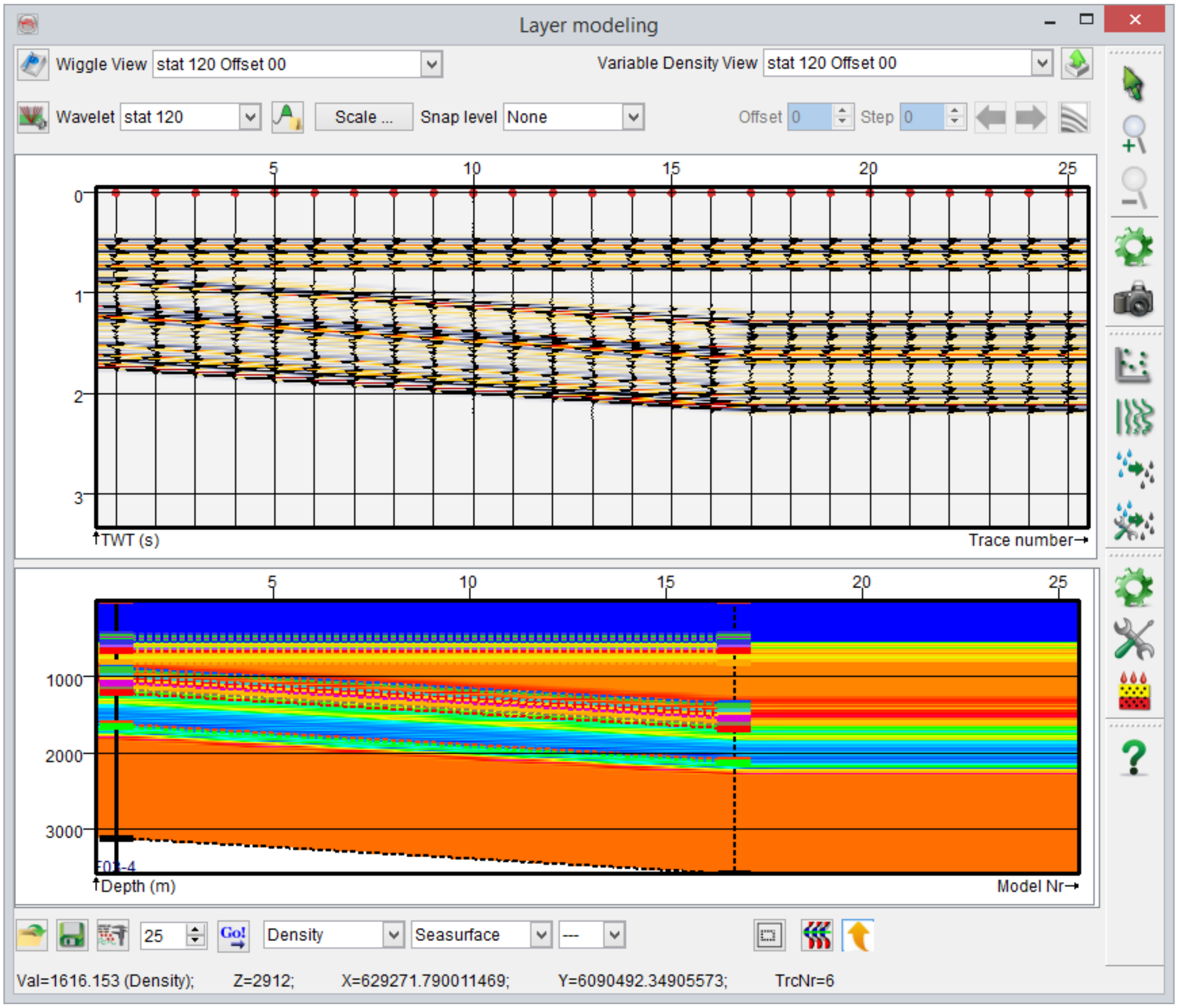
If *Couple depths* option is kept unchecked, then editing any particular marker's depth is not going to change any other marker's depth; except when post-edition depth of that marker becomes lower than the depth of markers above. If this is indeed the case, all those markers will also be given the (post-edition depth) of the marker in consideration, for example below.



Relative Position (left=0%,right=100%) 50

		Depth (m)
Seasurface	🟥	0
MFS11	🟦	523.6
FS11	🟩	546.02276611
MFS10	🟦	653.31
MFS9	🟨	686.65002441
MFS8	🟪	718.49
FS8	🟩	765.00915527
FS7	🟦	912.27575684
Truncation	🟪	1021.41040039
Top Foresets	🟥	1064.97290039
FS6	🟧	1076.93835449
MFS4	🟦	1078.85705566
FS4	🟩	1119.58239746
FS3	🟦	1134.7298584
FS2	🟨	1144.62
MFS2	🟪	1167.07995605

☐ Couple depths



Wiggle View stat 120 Offset 00

Variable Density View stat 120 Offset 00

Wavelet stat 120

Snap level None

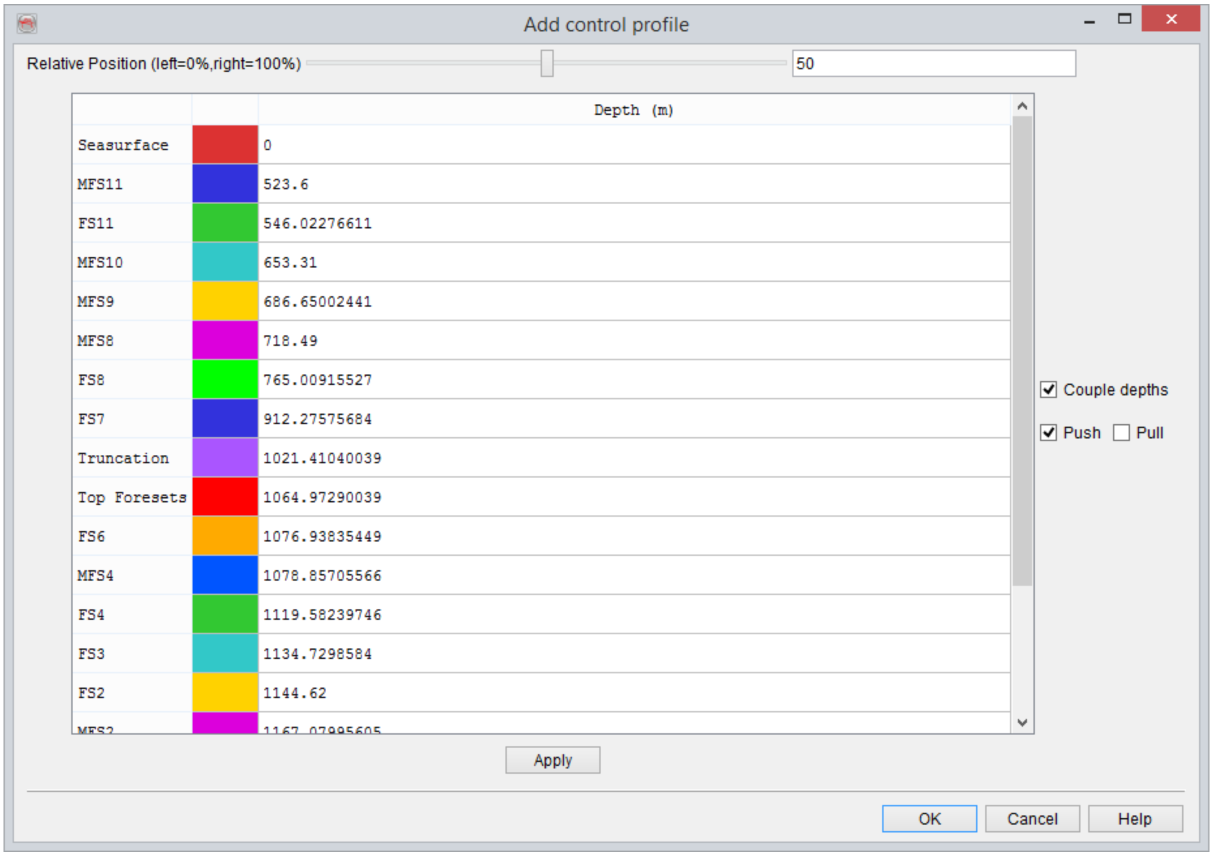
Offset 0 Step 0

25 Density Seasurface ---

Val=1616.153 (Density); Z=2912; X=629271.790011469; Y=6090492.34905573; TrcNr=6

Defining a control profile by editing depths of FS8 marker, without coupling depths of other markers

If *Couple depths* option is checked and only *Push* is selected - Editing any marker's depth is going to push all other markers in the direction of the change, starting from the point of change. For the given example, the direction of change is from deep (initial depth of FS8 marker 535m) to shallow (post-edition depth of FS8 marker 400m) and the point of change is FS8 marker. Thus, all the markers above FS8 have also been pushed upwards (i.e. from deep to shallow), by the same amount as the FS8 marker itself, i.e. 135 meters.

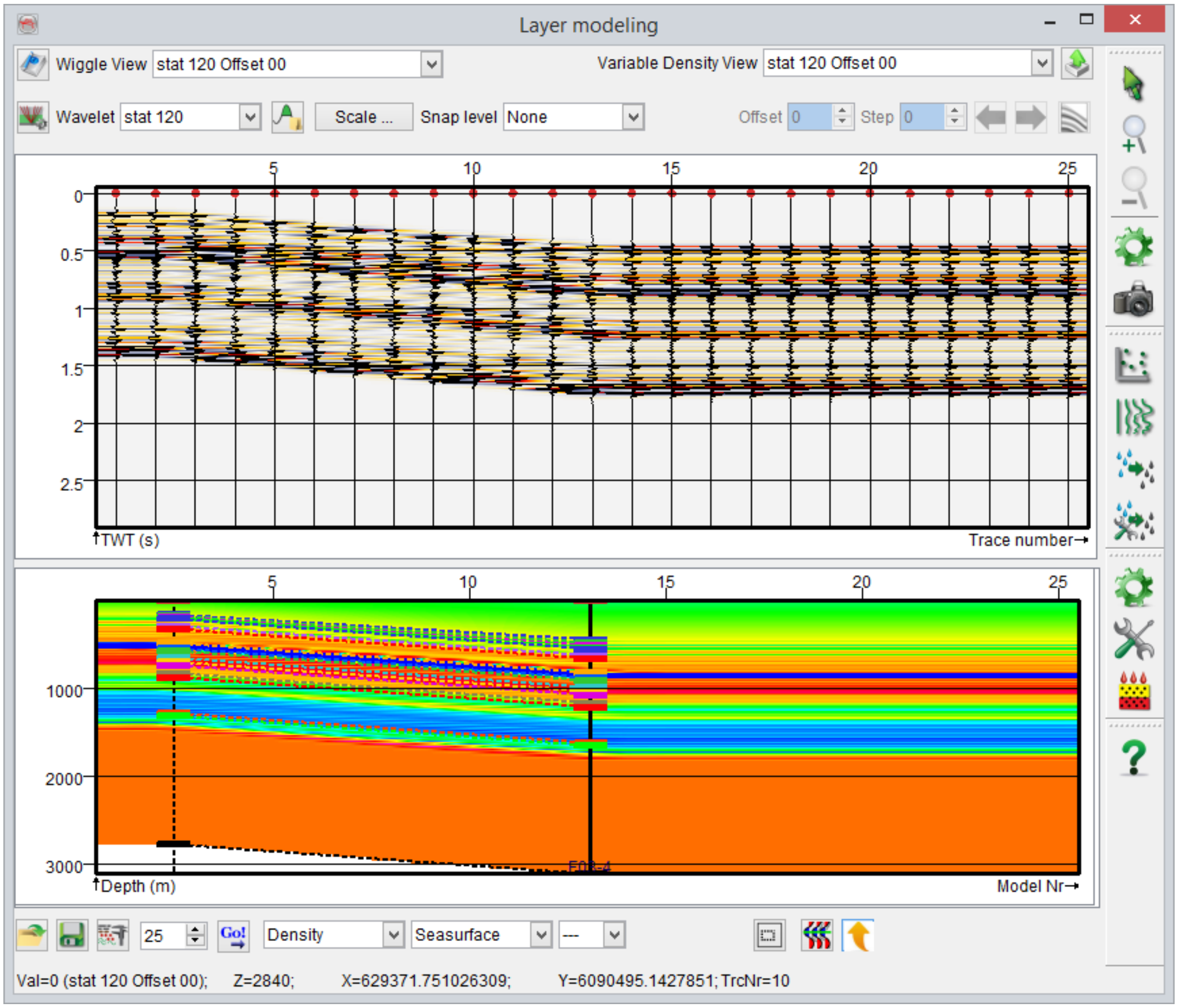


Relative Position (left=0%,right=100%) 50

		Depth (m)
Seasurface	🟥	0
MFS11	🟦	523.6
FS11	🟩	546.02276611
MFS10	🟦	653.31
MFS9	🟨	686.65002441
MFS8	🟪	718.49
FS8	🟩	765.00915527
FS7	🟦	912.27575684
Truncation	🟪	1021.41040039
Top Foresets	🟥	1064.97290039
FS6	🟧	1076.93835449
MFS4	🟦	1078.85705566
FS4	🟩	1119.58239746
FS3	🟦	1134.7298584
FS2	🟨	1144.62
MFS2	🟪	1167.07995605

☑ Couple depths

☑ Push ☐ Pull



Layer modeling

Wiggle View: stat 120 Offset 00

Variable Density View: stat 120 Offset 00

Wavelet: stat 120

Snap level: None

Offset: 0

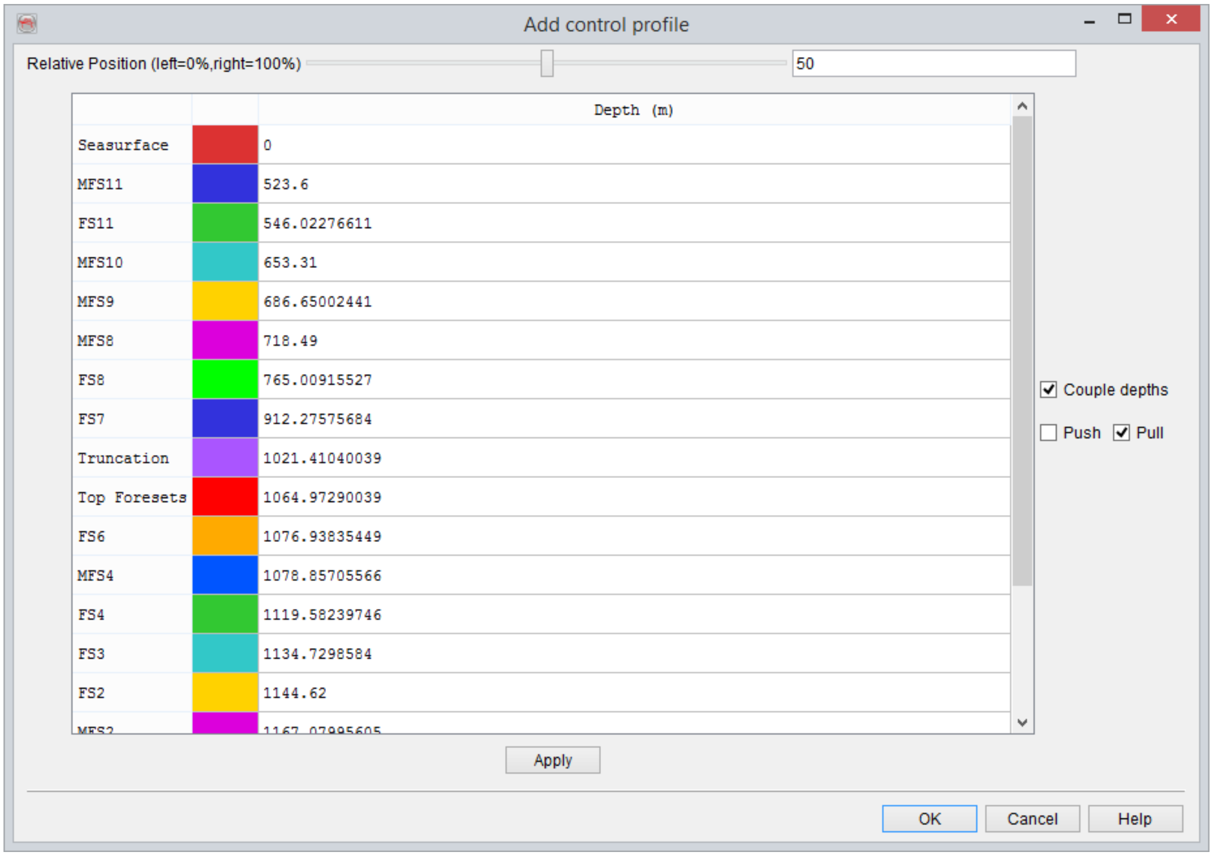
Step: 0

25 Density Seasurface ---

Val=0 (stat 120 Offset 00); Z=2840; X=629371.751026309; Y=6090495.1427851; TrcNr=10

Defining a control profile by editing depths of FS8 marker and pushing markers above upwards

Now if *Couple depths* option is checked but only *Pull* is selected - Editing any marker's depth is going to pull all other markers in the direction of the change, starting from the point of change. For the given example, the direction of change is from deep (initial depth of FS8 marker 535m) to shallow (post-edition depth of FS8 marker 400m) and the point of change is FS8 marker. Thus, all the markers below FS8 have been pushed upwards (i.e. from deep to shallow), by the same amount as the FS8 marker itself, i.e. 135 meters. It should be noted here that the change in the depths of markers above FS8 is because of the afore-mentioned exception.

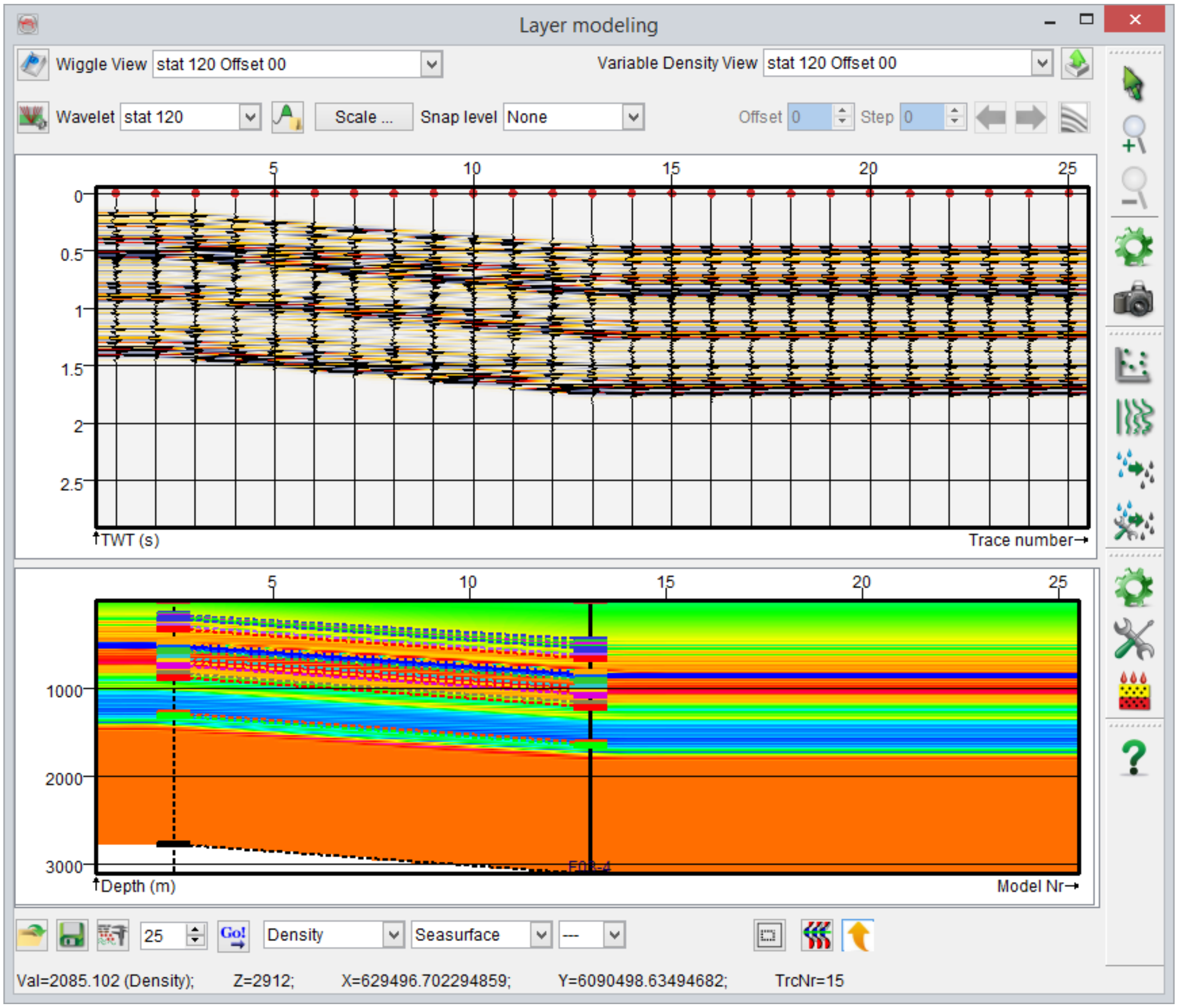


Relative Position (left=0%,right=100%) 50

		Depth (m)
Seasurface	🟥	0
MFS11	🟦	523.6
FS11	🟩	546.02276611
MFS10	🟦	653.31
MFS9	🟨	686.65002441
MFS8	🟪	718.49
FS8	🟩	765.00915527
FS7	🟦	912.27575684
Truncation	🟪	1021.41040039
Top Foresets	🟥	1064.97290039
FS6	🟧	1076.93835449
MFS4	🟦	1078.85705566
FS4	🟩	1119.58239746
FS3	🟦	1134.7298584
FS2	🟨	1144.62
MFS2	🟪	1167.07995605

☑ Couple depths

☐ Push ☑ Pull



Layer modeling

Wiggle View stat 120 Offset 00

Variable Density View stat 120 Offset 00

Wavelet stat 120

Snap level None

Offset 0

Step 0

25

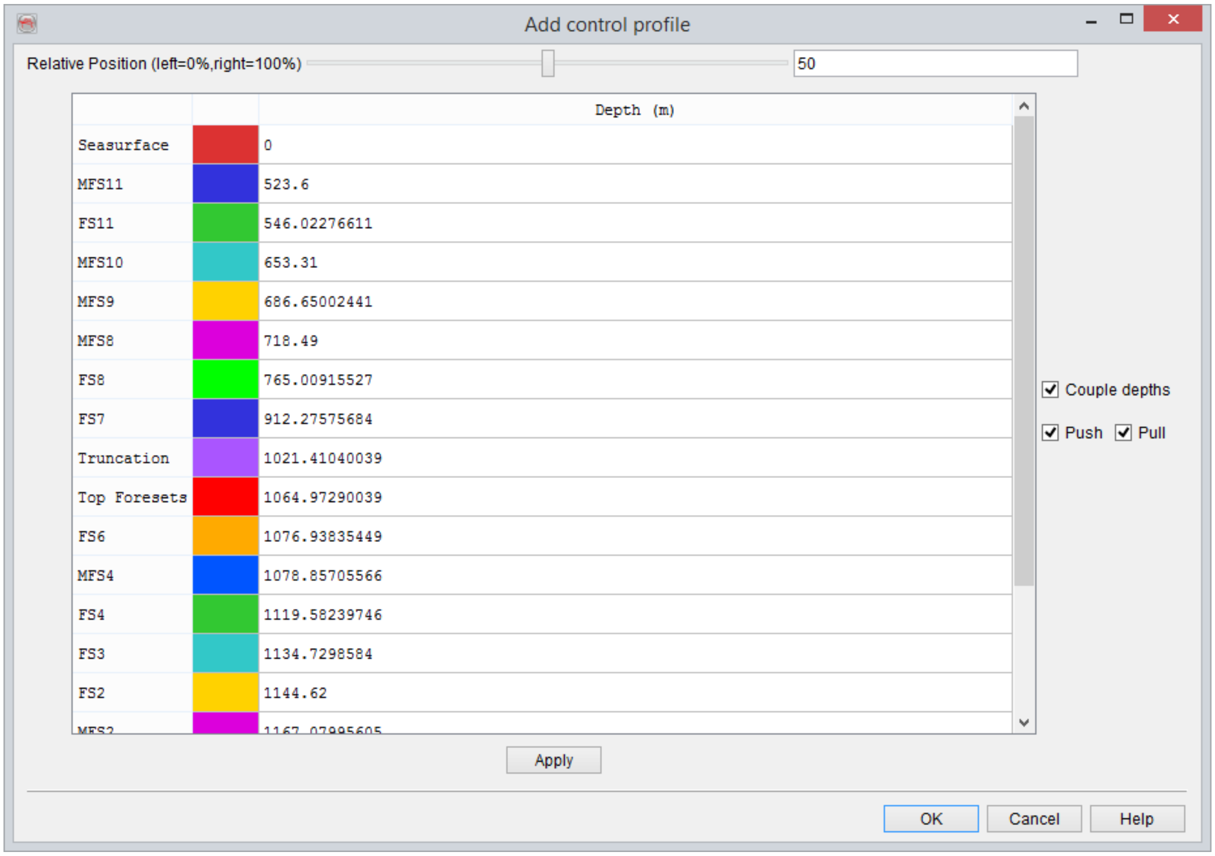
Density

Seasurface

Val=2085.102 (Density); Z=2912; X=629496.702294859; Y=6090498.63494682; TrcNr=15

Defining a control profile by editing depths of FS8 marker and pulling markers below upwards

Finally, if *Couple depths* option is checked and both *Push* and *Pull* are selected - Editing any marker's depth is going to move (both push and pull) all other markers in the direction of the change. For our example, the direction of change is from deep to shallow. Thus, all the markers have been moved upwards by the same amount as the FS8 marker itself, i.e. 135 meters.

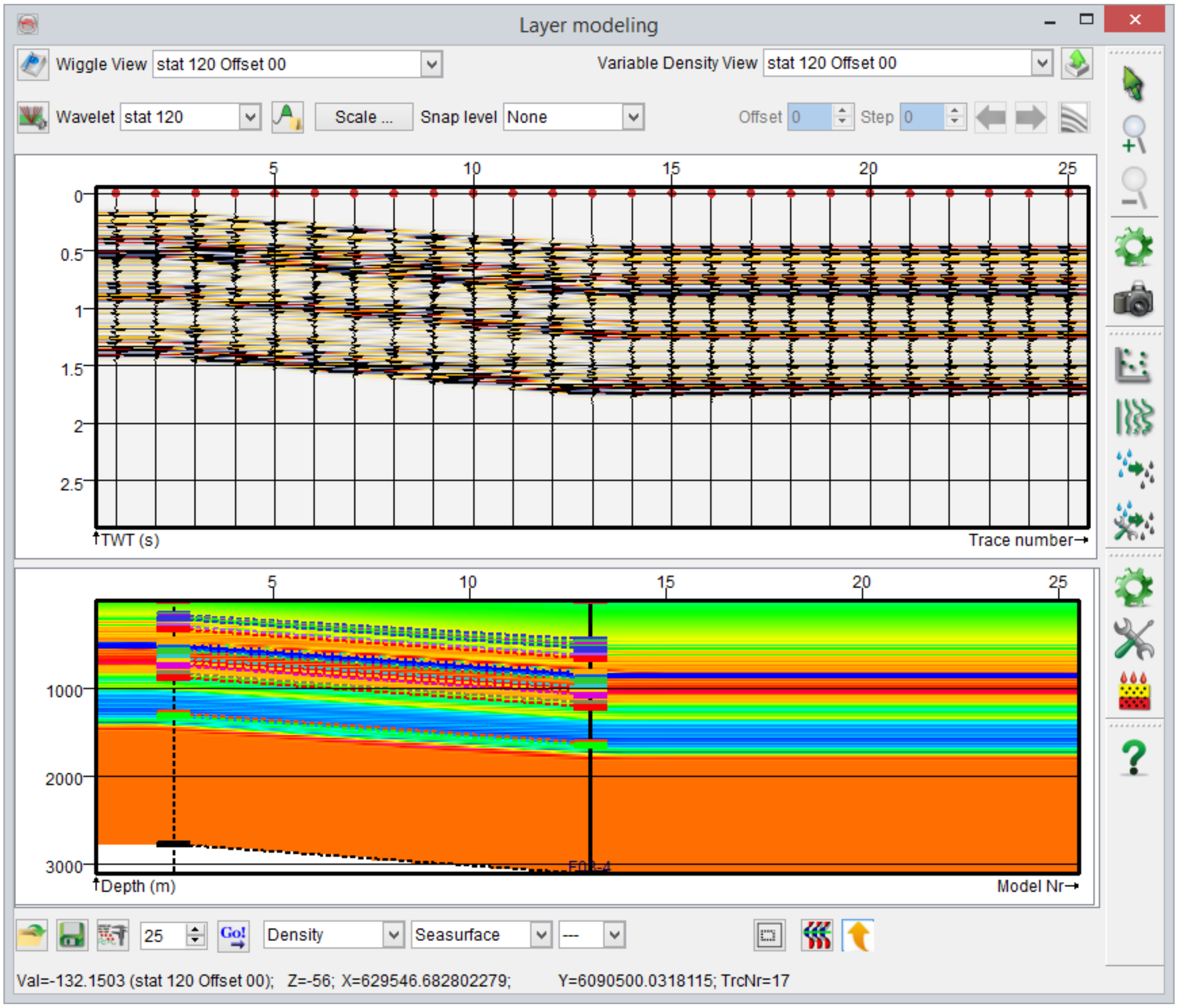


Relative Position (left=0%,right=100%) 50

		Depth (m)
Seasurface	🟥	0
MFS11	🟦	523.6
FS11	🟩	546.02276611
MFS10	🟦	653.31
MFS9	🟨	686.65002441
MFS8	🟪	718.49
FS8	🟩	765.00915527
FS7	🟦	912.27575684
Truncation	🟪	1021.41040039
Top Foresets	🟥	1064.97290039
FS6	🟧	1076.93835449
MFS4	🟦	1078.85705566
FS4	🟩	1119.58239746
FS3	🟦	1134.7298584
FS2	🟨	1144.62
MFS2	🟪	1167.07995605

☑ Couple depths

☑ Push ☑ Pull



Wiggle View stat 120 Offset 00

Variable Density View stat 120 Offset 00

Wavelet stat 120

Snap level None

Offset 0

Step 0

25

Density

Seasurface

Val=-132.1503 (stat 120 Offset 00); Z=-56; X=629546.682802279; Y=6090500.0318115; TrcNr=17

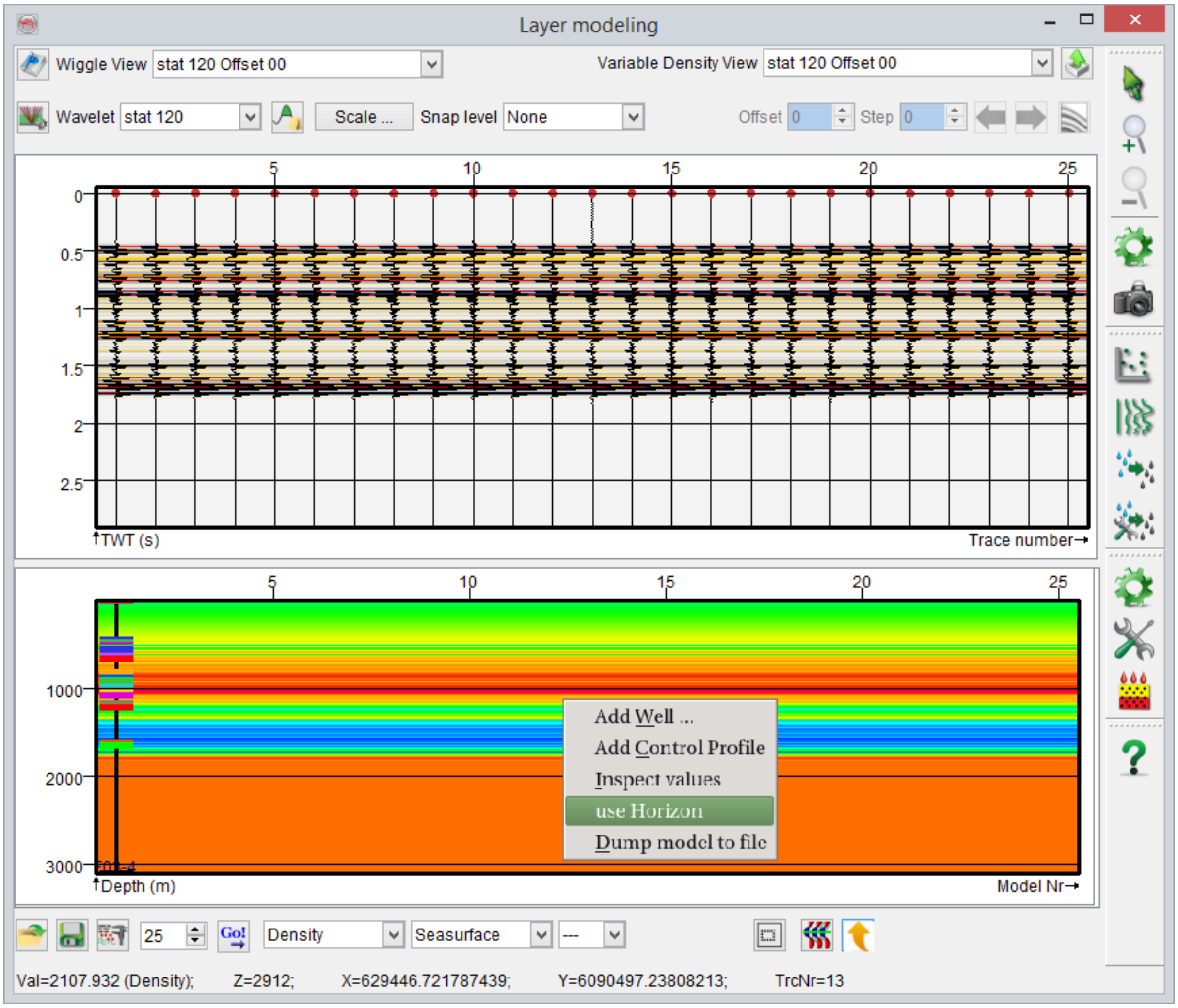
Defining a control profile by editing depths of FS8 marker and moving all other markers upwards

The added control profiles are displayed as dashed vertical lines.

- **Editing an existing control profile:** The edition of an existing control profile is done by right-clicking close to the dashed vertical line (corresponding to the control profile) and then selecting Edit, to pop-up a dialogue window similar to the one explained in the section above. Just like adding a new control profile, both relative position and well marker depths can be edited, of an existing control profile.

Use Horizon

Existing horizons from the project database, can be used to easily and quickly mimic an interpreted (3D or 2D) structure and align the pseudo-wells along it, in the profile modeling. This is essentially achieved by an automated creation of a number of *Control Profiles*, that follow the depth trend of the selected horizon. This can be achieved by right clicking on the bottom and selecting *Use Horizon*.


Wiggle View: stat 120 Offset 00

Variable Density View: stat 120 Offset 00

Wavelet: stat 120

Snap level: None

Offset: 0

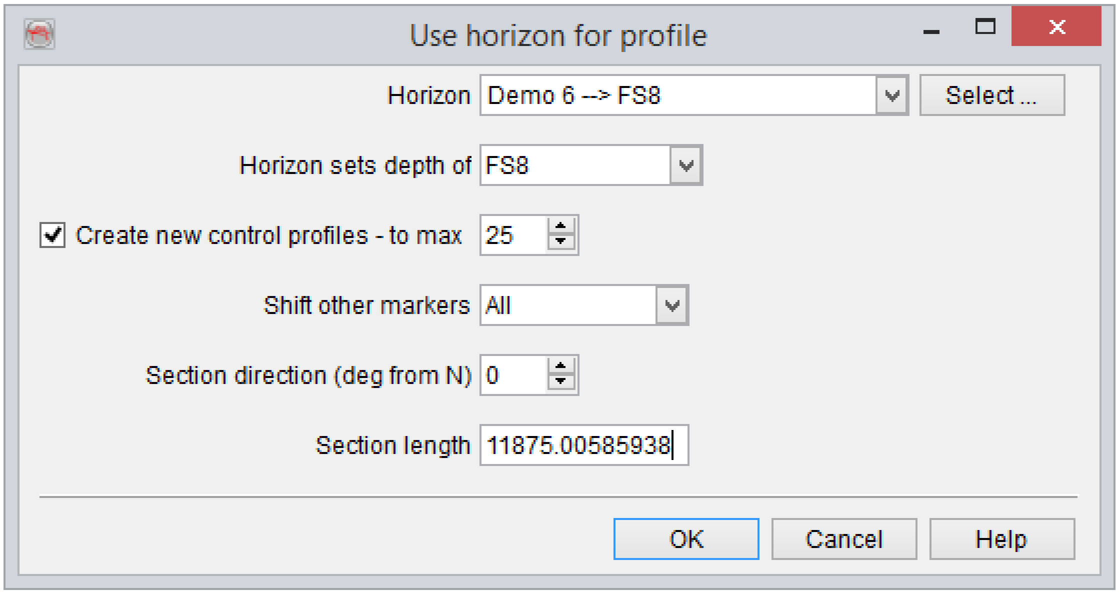
Step: 0

Density

Seasurface

Val=2107.932 (Density); Z=2912; X=629446.721787439; Y=6090497.23808213; TrcNr=13

If there is only one (real) well loaded in the layer, the following dialogue window will appear after clicking the link.



Horizon: Demo 6 --> FS8

Horizon sets depth of: FS8

☑ Create new control profiles - to max: 25

Shift other markers: All

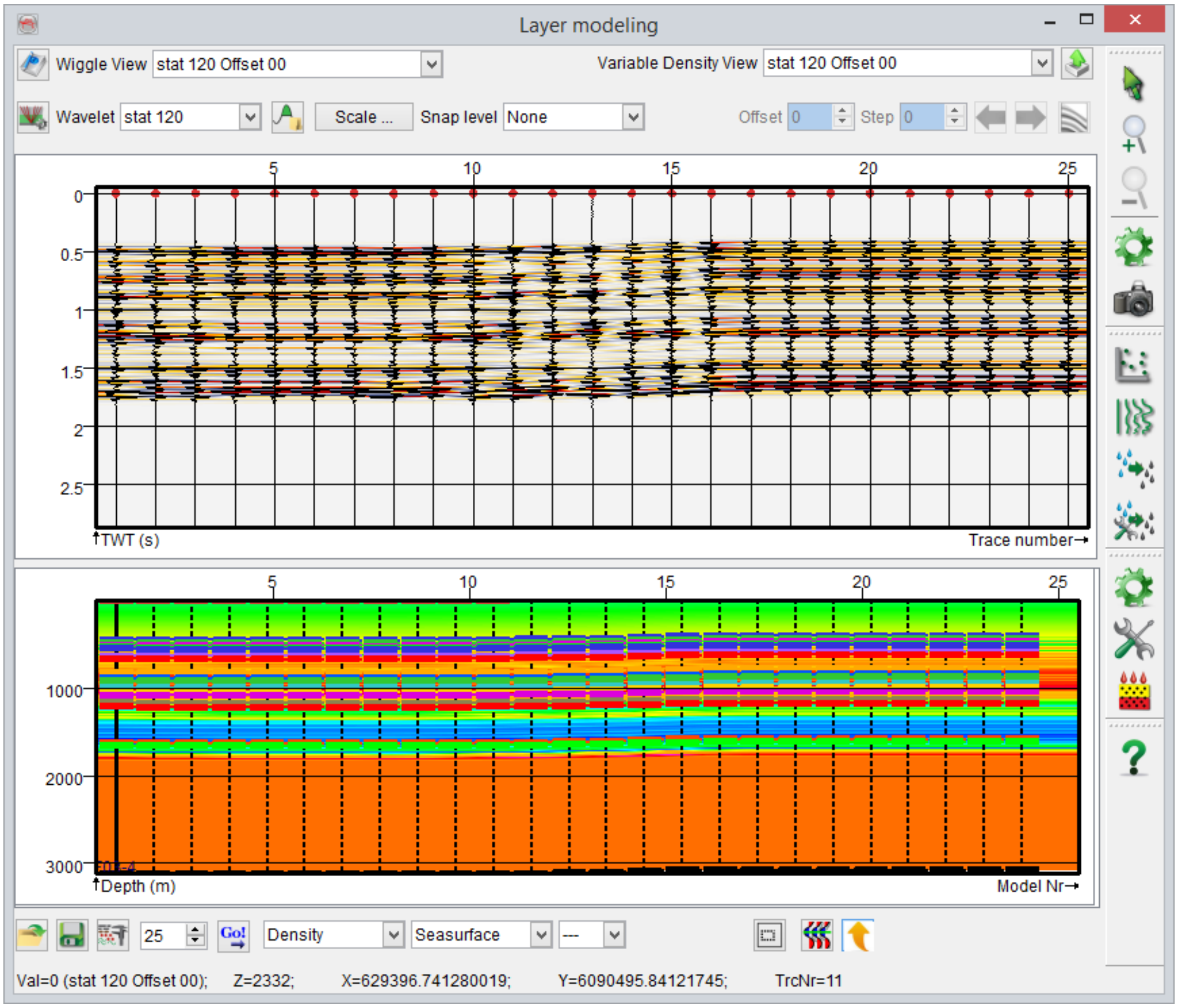
Section direction (deg from N): 0

Section length: 11875.00585938

Here, the user can select an appropriate horizon on which the simulation of the structure will be based. This selected time horizon is converted into depth using the velocity-depth trend of the pseudo-well models.

Next thing is to specify the corresponding marker, whose depths will be changed using the depth values of the (depth-converted) horizon. Afterward, there is an option to limit the maximum number of new control profiles to be added, in between the existing pseudo-wells. It is actually through the creation of these control profiles, the program effectively changes the depth of the marker, as to follow the depth trend of the selected horizon. Further, it is possible to also shift other markers relative to the marker in consideration. '*All markers*' or '*Markers only above*' or '*Markers only below*' or '*No marker*' are the options available here.

Finally, it is possible to use the depth trend of the horizon along a (pseudo) section, oriented in a particular direction (relative to North) and extending upto a certain distance from the well location. The *Section direction* is specified in Degrees relative to North and the *Section length* in X/Y units of the survey. Below is an example of a pseudo-well profile based on 'Demo0 FS4' horizon.



Layer modeling

Wiggle View stat 120 Offset 00

Variable Density View stat 120 Offset 00

Wavelet stat 120

Snap level None

Offset 0

Step 0

25

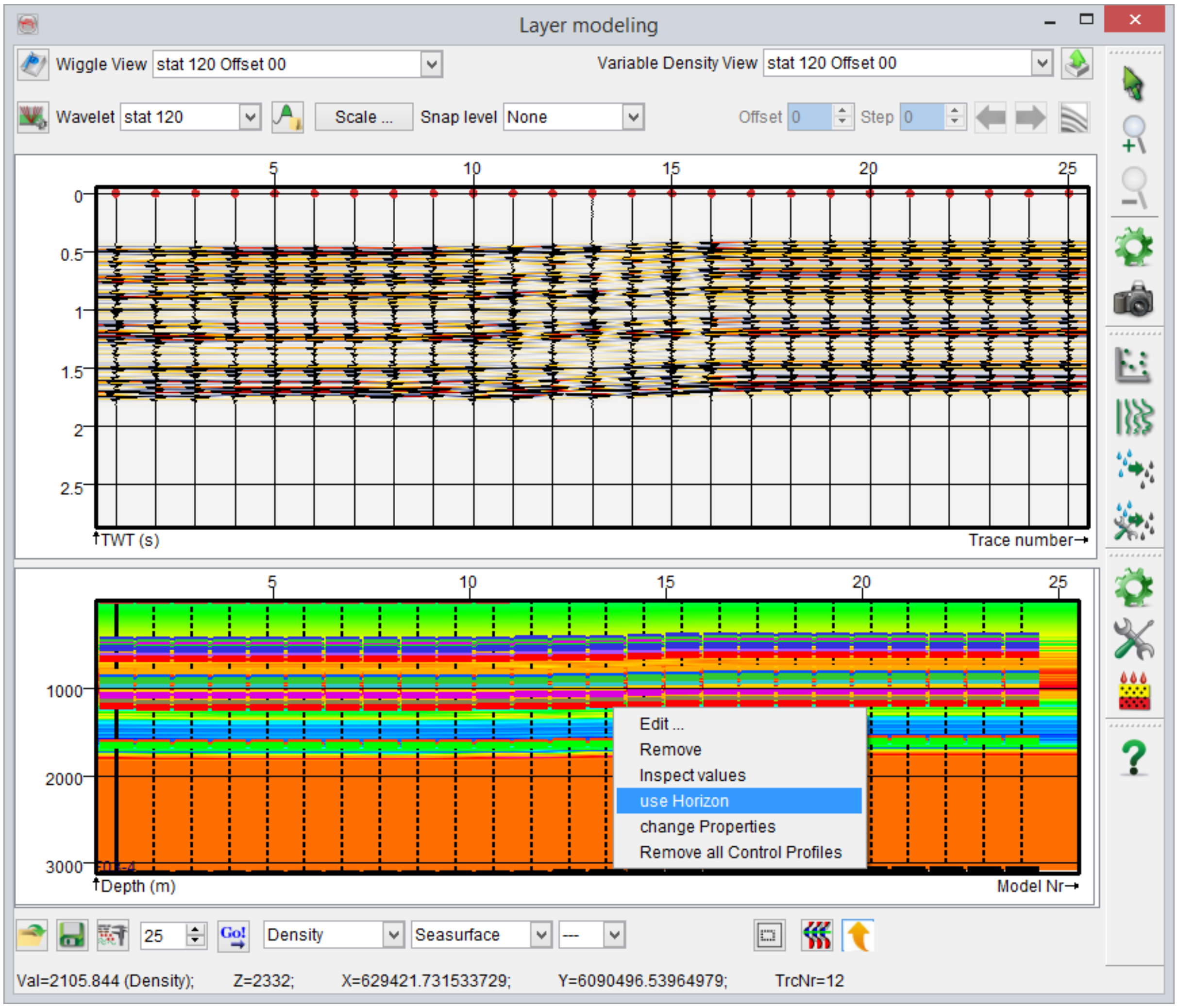
Density

Seasurface

Val=0 (stat 120 Offset 00); Z=2332; X=629396.741280019; Y=6090495.84121745; TrcNr=11

Using 'Demo0 FS4' horizon to create pseudo-well profile

If there are more than one (real) wells loaded in the layer, for example below, *Use Horizon* option will bring-up a slightly different dialogue.



Wiggle View stat 120 Offset 00

Variable Density View stat 120 Offset 00

Wavelet stat 120

Snap level None

Offset 0

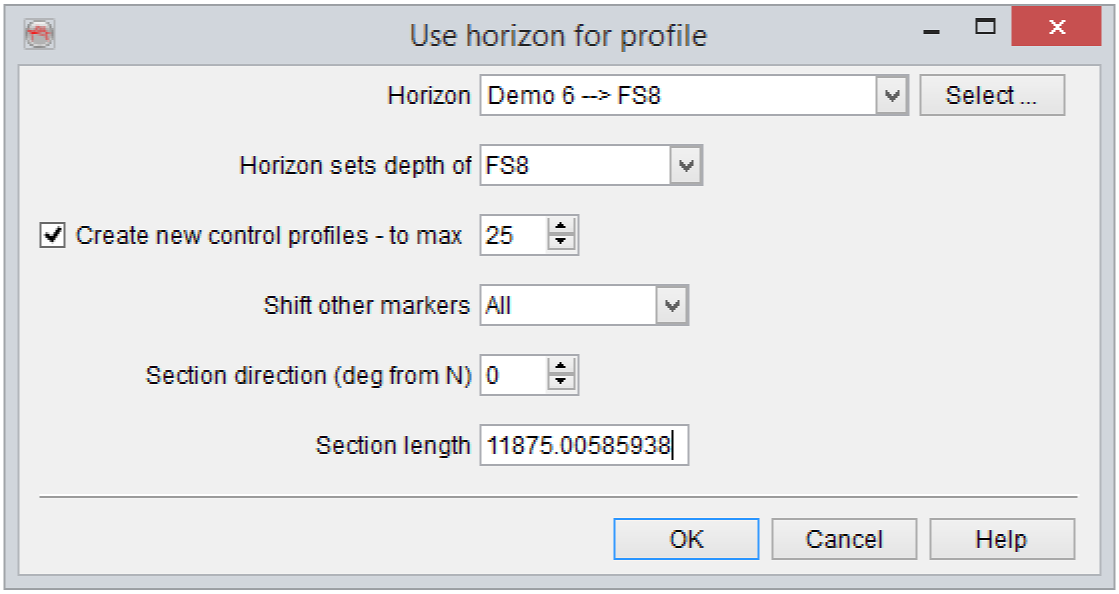
Step 0

25

Density

Seasurface

Val=2105.844 (Density); Z=2332; X=629421.731533729; Y=6090496.53964979; TrcNr=12



Horizon: Demo 6 --> FS8

Horizon sets depth of: FS8

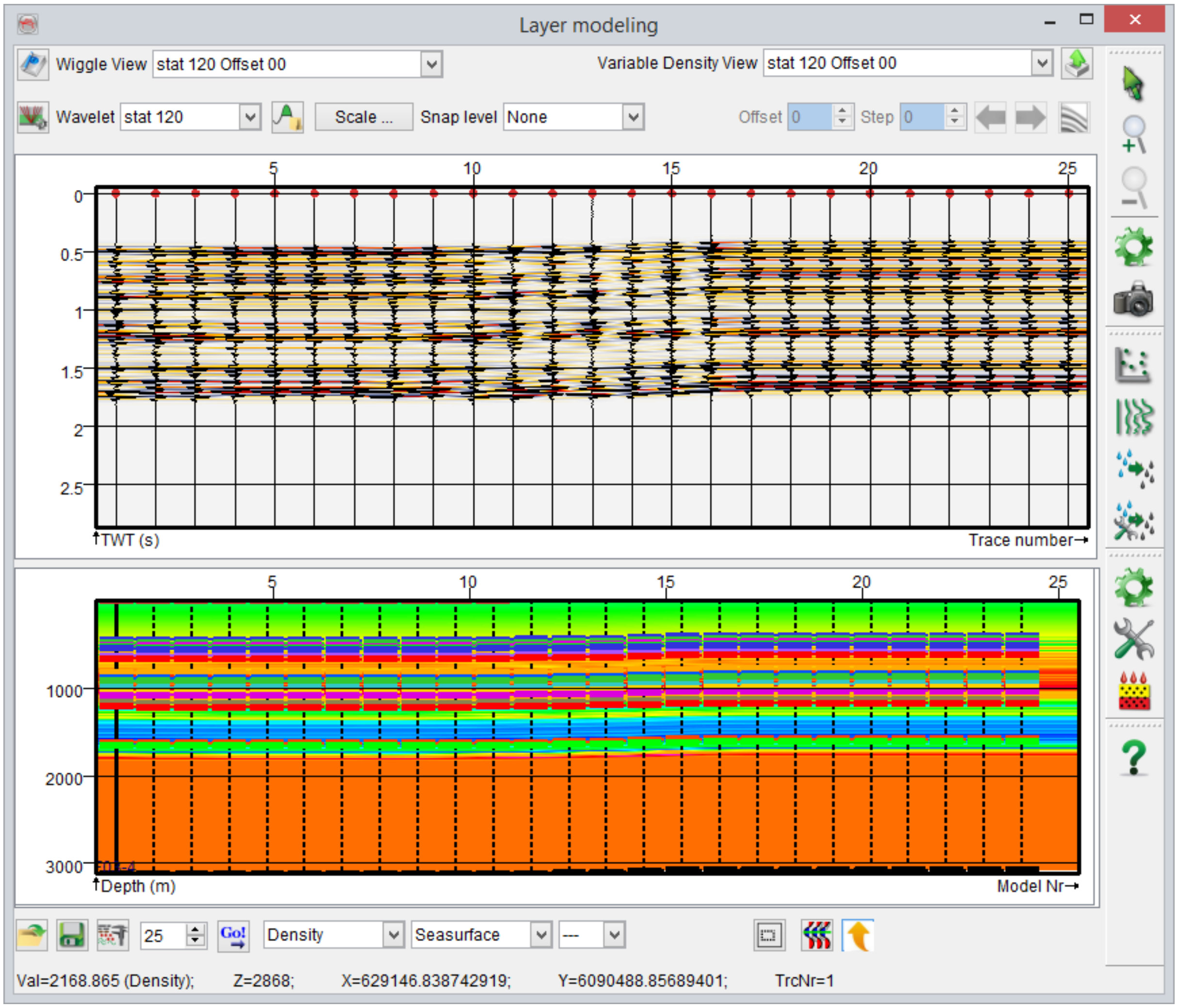
☑ Create new control profiles - to max: 25

Shift other markers: All

Section direction (deg from N): 0

Section length: 11875.00585938

Here, all the options are same as before, except that there is no need to specify the *Section direction* and the *Section length*. This is simply because the program will automatically compute the structural trend in depth of a particular horizon, between the wells (see below).



Layer modeling

Wiggle View stat 120 Offset 00

Variable Density View stat 120 Offset 00

Wavelet stat 120

Snap level None

Offset 0 Step 0

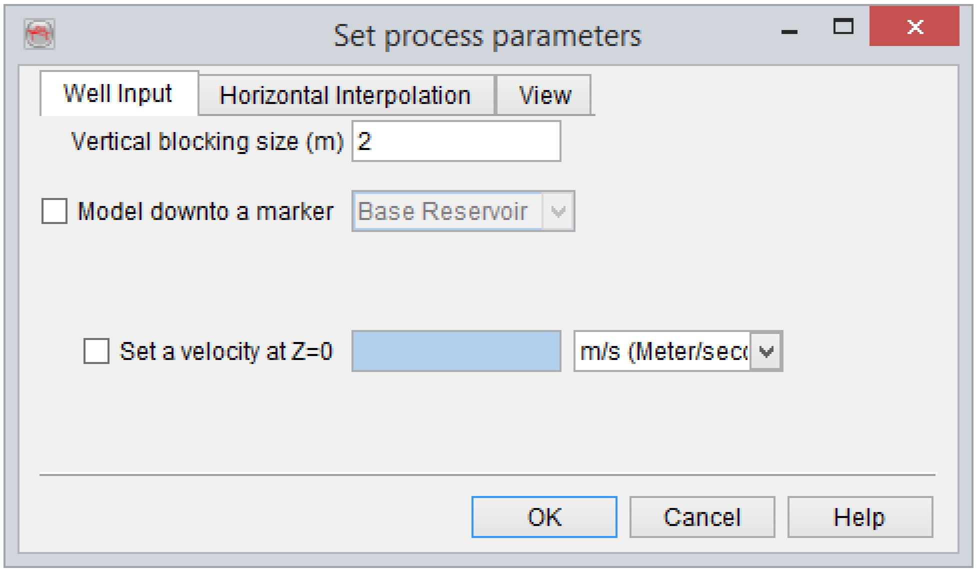
25 Density Seasurface ---

Val=2168.865 (Density); Z=2868; X=629146.838742919; Y=6090488.85689401; TrcNr=1

Finally, it is useful to note that depth trends of multiple horizons can be used to build a single profile model, by repeating the above work-flow for different horizons and perhaps shifting *only* the corresponding marker each time.

Set Process Parameters

The layer modeling settings and various annotations in the profile module can be edited by setting the process parameters. The Set process parameters window can be accessed by clicking on 🛠 , from the vertical list of icons on the right hand side of the layer modeling window. Also, clicking in the bottom part of the main layer modeling window to add the very first well, pops-up this modeling process parameter selection window.



Set process parameters

Vertical blocking size (m) 2

☐ Model downto a marker Base Reservoir

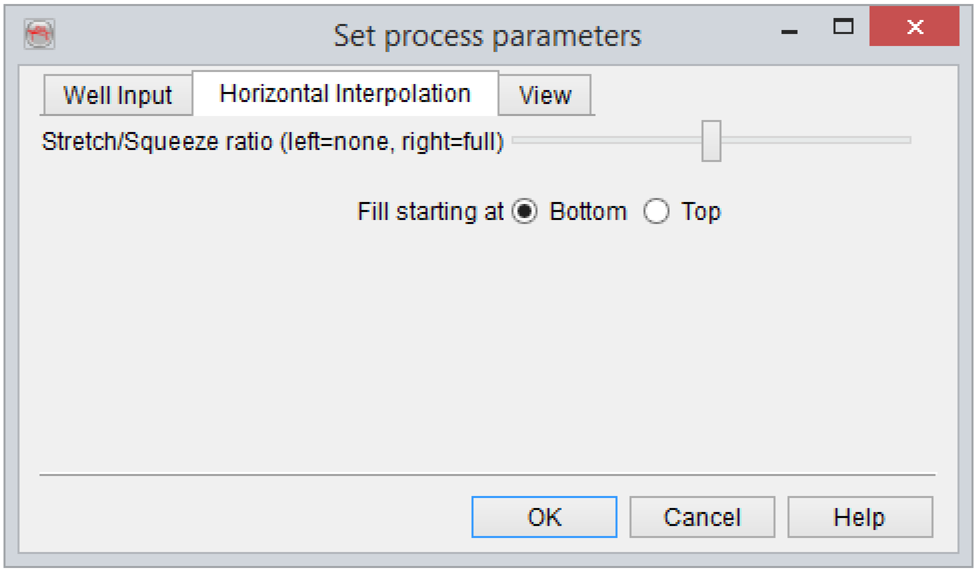
☐ Set a velocity at Z=0 m/s (Meter/sec

In the first tab (above) of the parameter selection window, a *Vertical blocking size* for various logs can be specified. Optionally, the model can be limited *down* to a particular well marker with some additional depth below it. Further, it is also possible to set a user defined surface velocity (i.e. velocity at Z=0). This options particularly comes in handy, when the input (real) Vp/Sonic logs do not start from the surface. Using this option, a reasonable overburden velocity model can be created, as the program will interpolate the velocity information between the surface (i.e. Z=0) and the first velocity sample in the well logs. If this option is not used, the top most velocity value from the well logs will be assigned to all the layers above, up to the surface.

The second tab (below) parametrizes how the horizontal interpolation is done between the wells (and/or control profiles) of various log data. At a certain point in time, the program needs to generate a new profile between two 'fixed' profiles - Two wells, or a well and a control profile, or two control profiles. In general the thickness, at the new profile location, will not be the same as any of the two controlling profiles. If a full stretch/squeeze ratio is used, then all layers from both 'fixed' profiles will be present. If no stretch/squeeze ratio is used, then two things are possible - Either cut the extra layers from top (*Fill starting at Bottom*) or cut layers from the bottom (*Fill starting at Top*). However, the using full stretch/squeeze may be rather non-geological and no stretch/squeeze is somewhat non-flexible. Therefore, the default option is a combination of the two and is the use of 50% stretch/squeeze ratio, for the horizontal interpolation.

For example, consider a scenario, where there are 10 layers on the left and 5 layers on the right (both sides blocked at 1m), and the program has to draw a profile in the middle which obviously is going to have a layer thickness of 7.5 meters. Now, the use of full stretch/squeeze ratio option is going to fill this 7.5m thickness with all the layers, by squeezing them in. If no squeezing/stretching is used, either the top 7.5 layers or bottom 7.5 layers can be outputted. If the default 50% stretch/squeeze ratio option is used, either the top 8.5 layers or bottom 8.5 can be squeezed in the total thickness of 7.5 meters, at the new profile location.

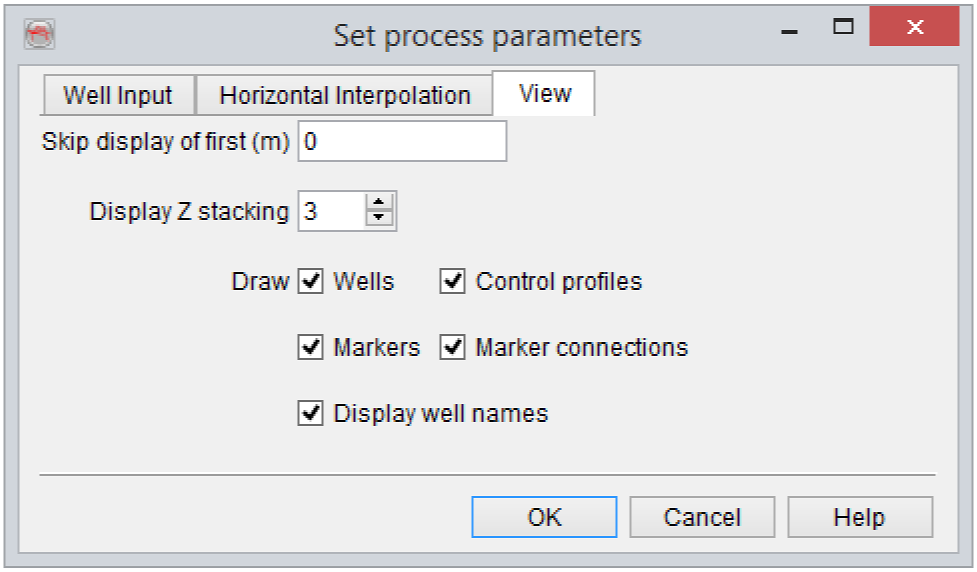
It may also be noted that geologically speaking, cutting the top layers out or *Fill starting at Bottom* is a more sensible option.



Stretch/Squeeze ratio (left=none, right=full)

Fill starting at ● Bottom ○ Top

Finally, the third tab of the parameter selection window, controls the display options for the layer model. Here, it is possible to *Not* display the layer model down to a user defined depth, starting from the surface (i.e. Z=0). Also, it is possible to *display* the layer model in a blocky manner, by stacking a number of vertical blocks (block size defined in the first tab), using the *Display Z stacking* option. For example, if vertical blocking size of the well log data is defined as '2' meters and the default Z stacking of '3' is used, three adjacent vertical blocks (of 2 m thickness each) will be stacked together, only for display. Next, it can be chosen, whether to draw or not, the annotations for various elements of the layer model - Wells, Control Profiles, Markers, Marker connections and Well names.



Skip display of first (m) 0

Display Z stacking 3

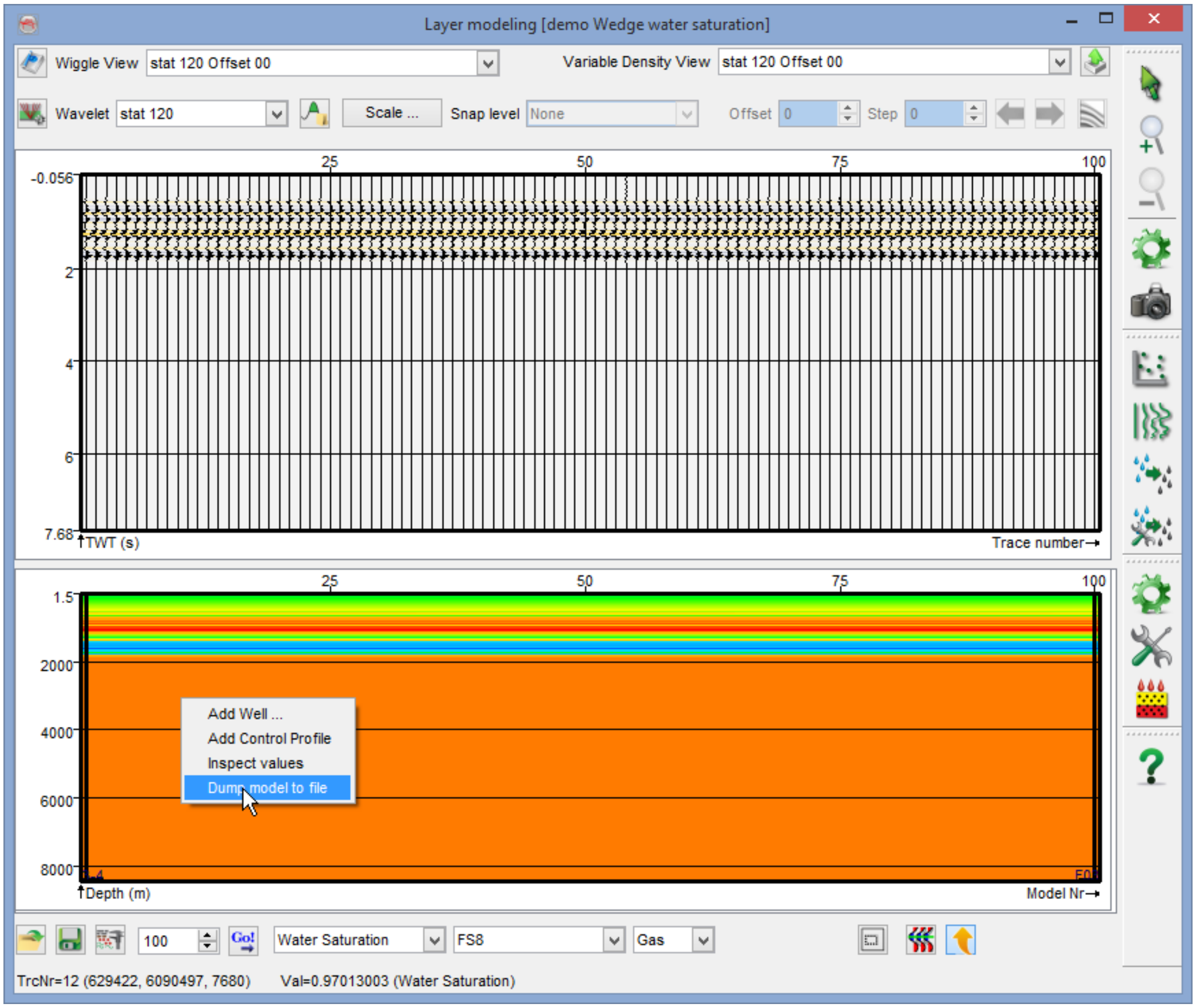
Draw ☑ Wells ☑ Control profiles

☑ Markers ☑ Marker connections

☑ Display well names

Dump Model to File

All the pseudo-well models forming a profile simulation can be saved to a *.txt* file using this option available in the right-click menu.

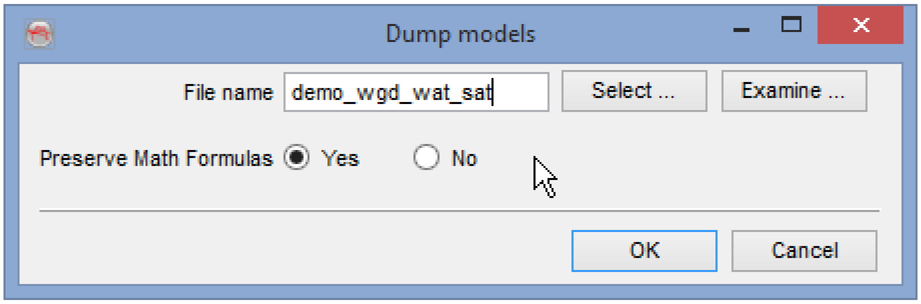


Wiggle View stat 120 Offset 00 Variable Density View stat 120 Offset 00

Wavelet stat 120 Snap level None Offset 0 Step 0

100 Water Saturation FS8 Gas

TrcNr=12 (629422, 6090497, 7680) Val=0.97013003 (Water Saturation)



File name demo_wgd_wat_sat

Preserve Math Formulas ● Yes ○ No

If there are some layer quantities present in the pseudo-well models that are computed from other quantities using Math formulas, it is possible to save those formulas as well in the output dump file.

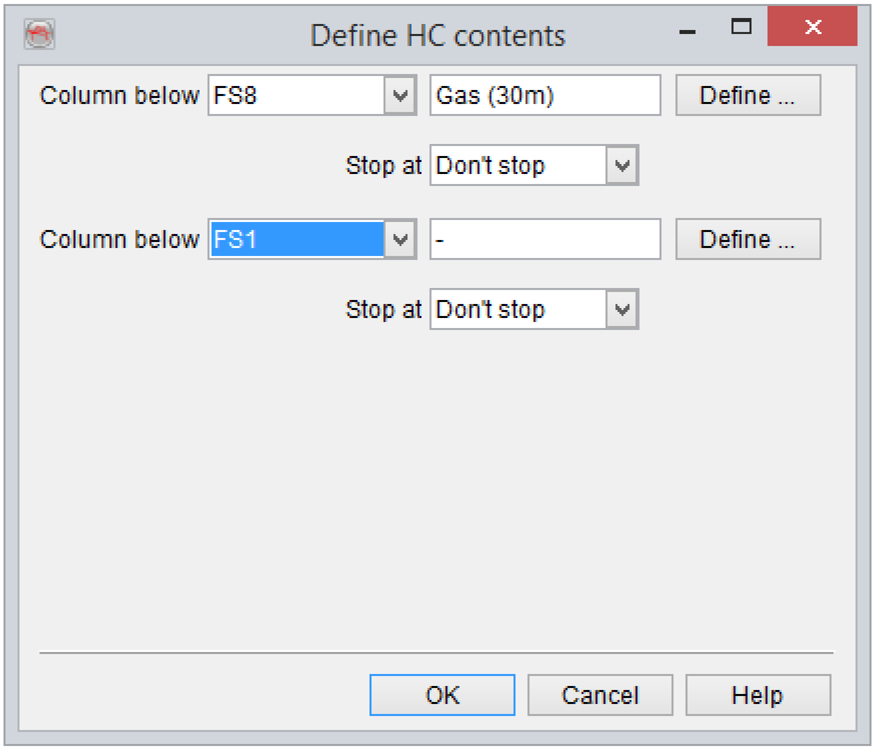
Fuid Replacement

This module is used to apply fluid substitution to the modeled pseudo-wells. It computes the changes in the elastic properties (e.g. P-wave velocity, S-wave velocity and Density) of the pseudo-wells when fluid A (e.g. brine) is replaced by fluid B (e.g. oil).

Define Fluid Contents

The fluid replacement algorithm in SynthRock is based on Gassmann's equation. Prior to any simulation, fluid possibly present can be specified in the *Contents Manager*.

In the *Profile* module, the hydrocarbon column has to be defined by particular thickness relative to a well marker(s). This is done by clicking on the bottom right icon, in the layer modeling window of the *Profile* module. Then a number of fluid contents, relative to various markers, can be defined in clicking on *Define:* below any individual marker, multiple fluid columns (e.g. oil column, gas column) of (possibly) different thickness can be specified.

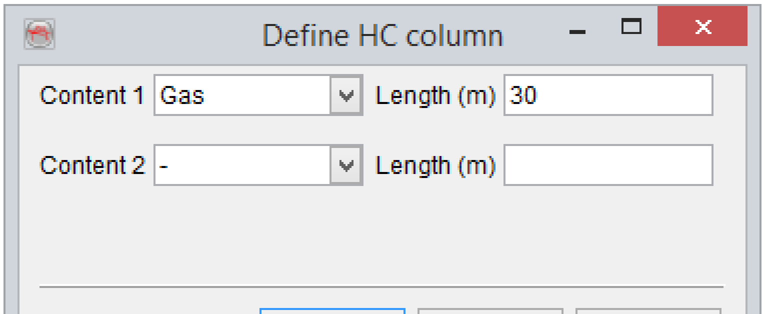


Column below FS8 Gas (30m)

Stop at Don't stop

Column below FS1 -

Stop at Don't stop

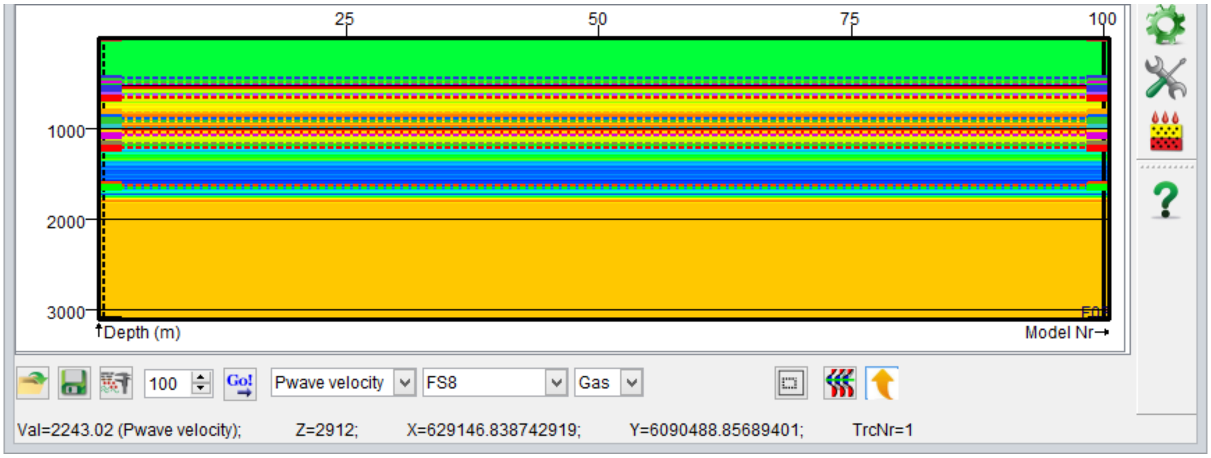


Content 1 Gas Length (m) 30

Content 2 - Length (m)

Optionally, the user can choose to limit the HC column down to another marker, below the original marker relative to which the HC column is defined. Pressing *Ok* will set-up the HC content.

The gas column of the above example can be visualized by zooming in at the appropriate level and specifying *Gas* as fluid content, in the bottom part of the main layer modeling window. It appears as a rectangle made of red colored lines (see below). The *Outline color* and/or the *Pattern* of the fluid visualization can be changed under Manage Contents by clicking on the in the Manage Stratigraphy window.



Pwave velocity FS8 Gas

Val=2243.02 (Pwave velocity); Z=2912; X=629146.838742919; Y=6090488.85689401; TrcNr=1

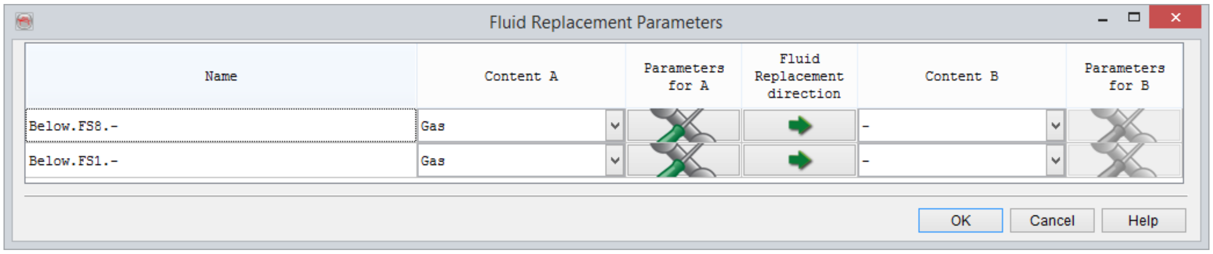
In the Stochastic or Basic Layer modeling, the fluid column has to be defined in the **model building** itself. The display is done in the same way as in the Profile mode.

Fluid Replacement Parameters

Once a fluid column has been defined in the pseudo-wells, the fluid replacement can be carried out by specifying the parameters for the Gassmann's equation.

Define Fluid Replacement Parameters

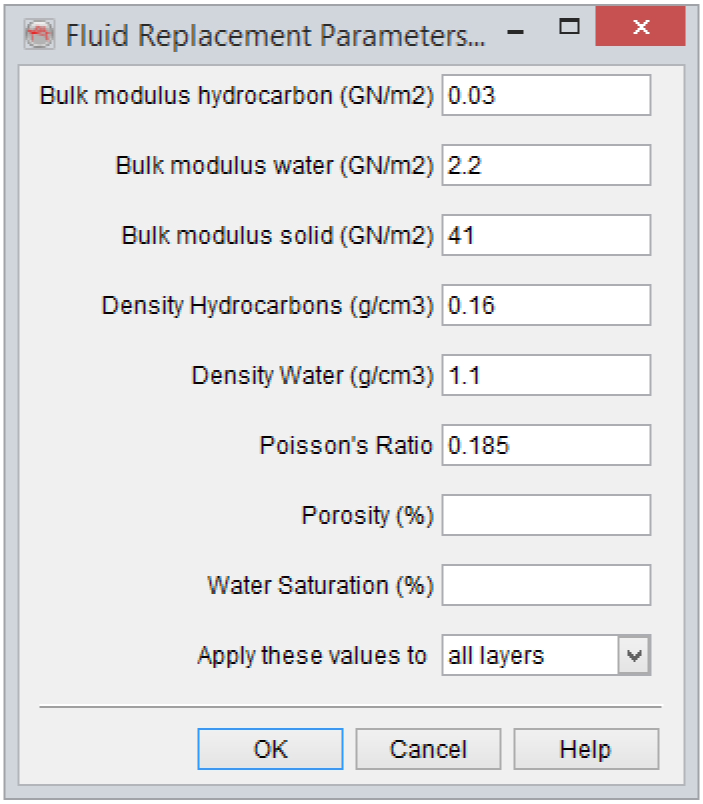
Click on from the vertical list of icons on the right side pane, in the layer modeling window, to specify the fluid replacement parameters for various layers with fluid content.



Name	Content A	Parameters for A	Fluid Replacement direction	Content B	Parameters for B
Below.FS8.-	Gas	🛠	➡	-	🛠
Below.FS1.-	Gas	🛠	➡	-	🛠

Here, the user can specify, for various layers, the fluid Content A (initial) and the fluid Content B (after replacement). The parameters for individual fluids before/after replacement are specified by clicking on the 🛠 icon. Now, in the above example, the initial fluid content in the two intervals (below the two well markers) is brine and is denoted by a '-' sign, by default. As the initial fluid content A is brine, the parameter specification icon has grayed out. The final fluid content, B, is gas and therefore clicking on the above mentioned icon, pops-up the following window. It should be noted that the direction of fluid replacement is from brine to gas, the direction can be changed by clicking on the ➡ : this will also change the direction of the arrow.

Fluid Replacement Parameters per unit



Bulk modulus hydrocarbon (GN/m2) 0.03

Bulk modulus water (GN/m2) 2.2

Bulk modulus solid (GN/m2) 41

Density Hydrocarbons (g/cm3) 0.16

Density Water (g/cm3) 1.1

Poisson's Ratio 0.185

Porosity (%)

Water Saturation (%)

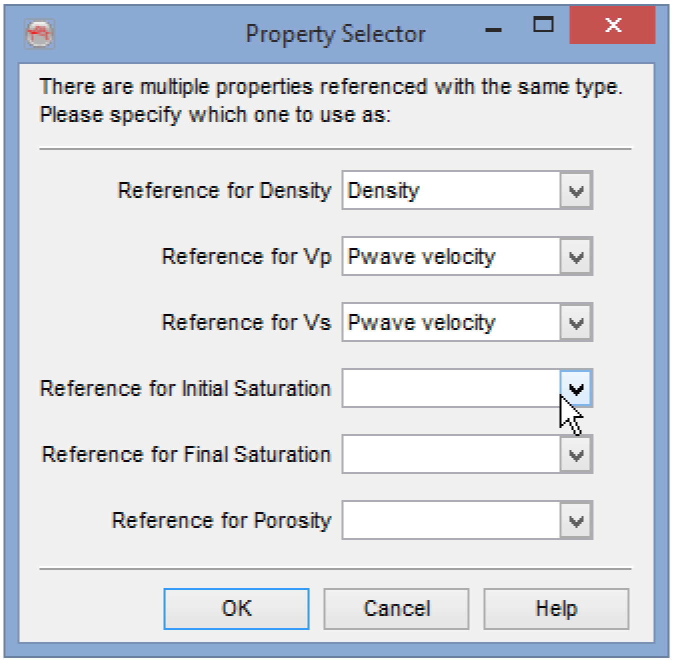
Apply these values to all layers

The top five parameters have to be filled in. If the Poisson's Ratio, Porosity and Water Saturation information is available in the layer model, they can be left blank. In absence of this information, a constant values can be filled in for these three parameters. Finally, these parameters can be defined for the layer in consideration, similar layers or all the layers having fluid content defined.

Apply Fluid Replacement

Now, once the fluid replacement parameters have been defined, the icon can be pressed to apply the fluid replacement. This will update the elastic properties (e.g. Vp, Vs, Density etc.) of the pseudo-well layers with fluid content, which will in turn impact the synthetic seismic response.

When the aforementioned icon is pressed the first time, the user has to specify various rock properties defined in the pseudo-wells, as required by the Gassmann's fluid replacement algorithm. They are Vp, Vs, Density, initial Water Saturation, final Water Saturation and Porosity.



There are multiple properties referenced with the same type.
Please specify which one to use as:

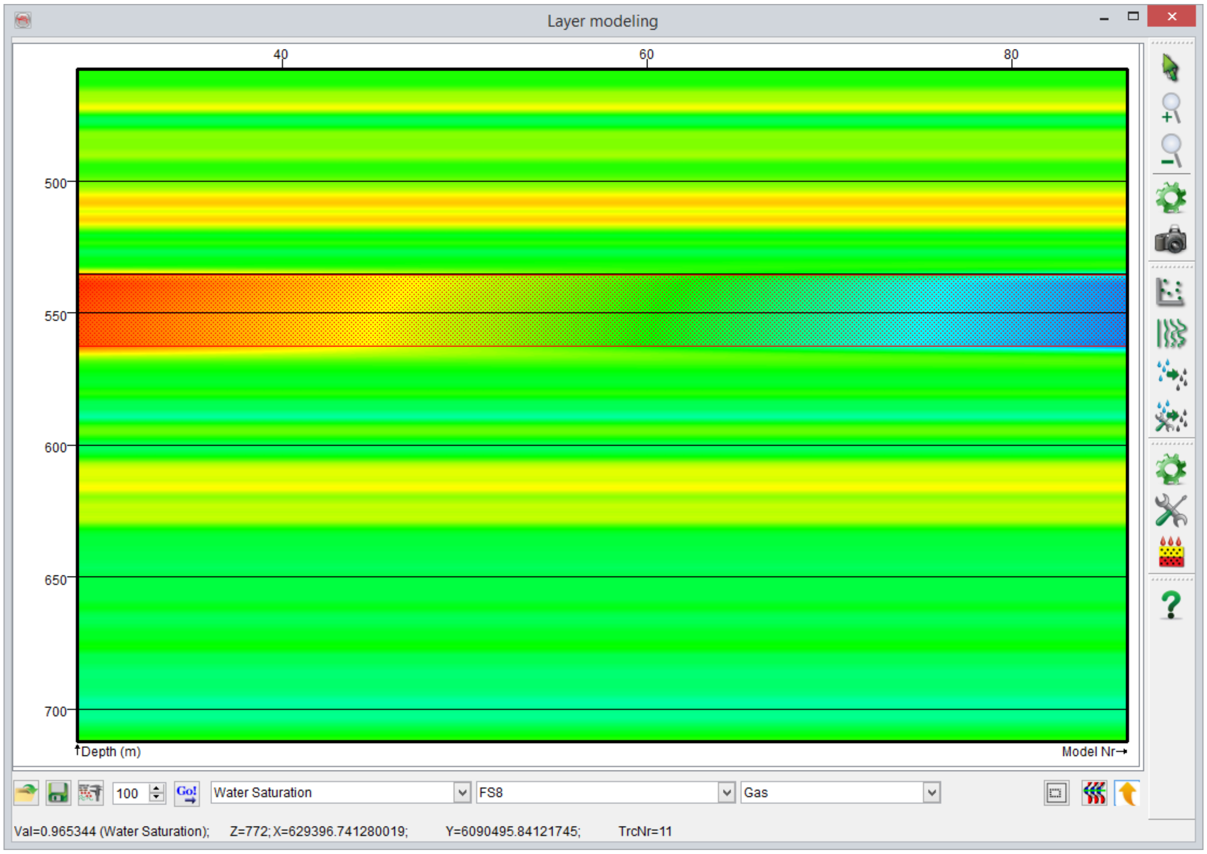
Reference for Density: Density
Reference for Vp: Pwave velocity
Reference for Vs: Pwave velocity
Reference for Initial Saturation:
Reference for Final Saturation:
Reference for Porosity:

If only fluid type is changed, while keeping a constant water saturation, e.g. oil to gas (at constant Sw = x) or from oil/gas (at Sw = x) to brine (i.e. Sw = 1) same water saturation property should be specified as reference for initial and final saturations. If the final saturation is kept empty, the software automatically uses the initial saturation as final saturation as well. It is also possible to specify constant values of initial and

final water saturations as well as porosity, in the Fluid Replacement Parameters dialogue. If that is done, any references defined for saturations and porosity in the above window will not be honoured.
However, if the target is to compute elastic properties at different water saturations with/without changing the fluid type, e.g. oil (at Sw = x) to oil (at Sw = y) or oil (at Sw = x) to gas (at Sw = y), one needs to specify separate initial and final saturations. Note that in order to do so, two layer properties of the same type (i.e. Volumetrics → Water Saturation) need to be predefined in the **Manage Layer Properties** and selected in the simulation in question.

> **Tip:** For shaly sands, use effective porosity and effective saturations, in the fluid replacement.

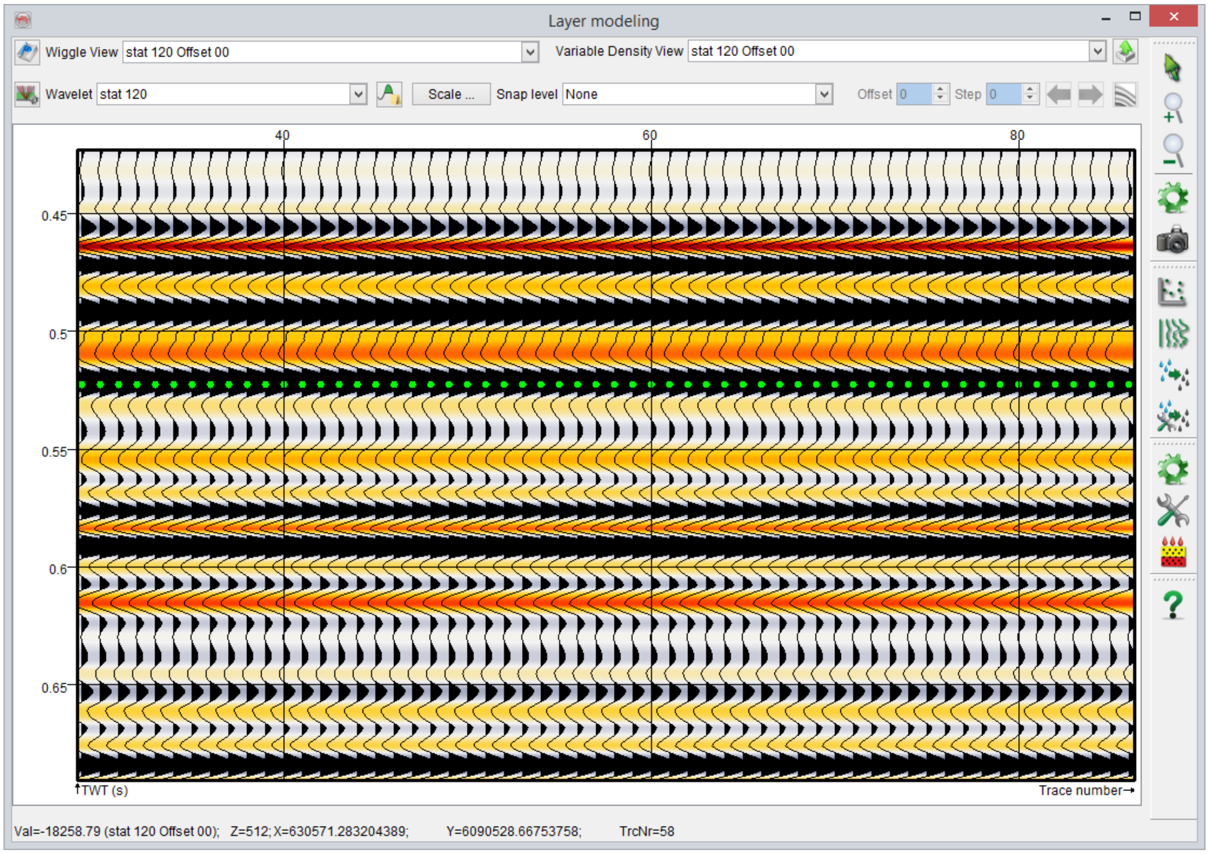
Below is an example of water saturation modeling, based on F03-4 well of F3 Demo data-set, wherein a gas column of thickness 30m is attached below the FS8 marker. Here modeling is done to decrease the logged water saturation from 100% to 90%, from left to right to replace with gas (i.e. gas saturation increases from 0% to 10%). Afterward, fluid replacement is applied to compute the change in seismic response corresponding to the change in fluid fill.



100 | Water Saturation | FS8 | Gas

Val=0.965344 (Water Saturation); Z=772; X=629396.741280019; Y=6090495.84121745; TrcNr=11

Example profile of water saturation variation from 90% to 100%, from left to right



Wiggle View: stat 120 Offset 00

Variable Density View: stat 120 Offset 00

Wavelet: stat 120

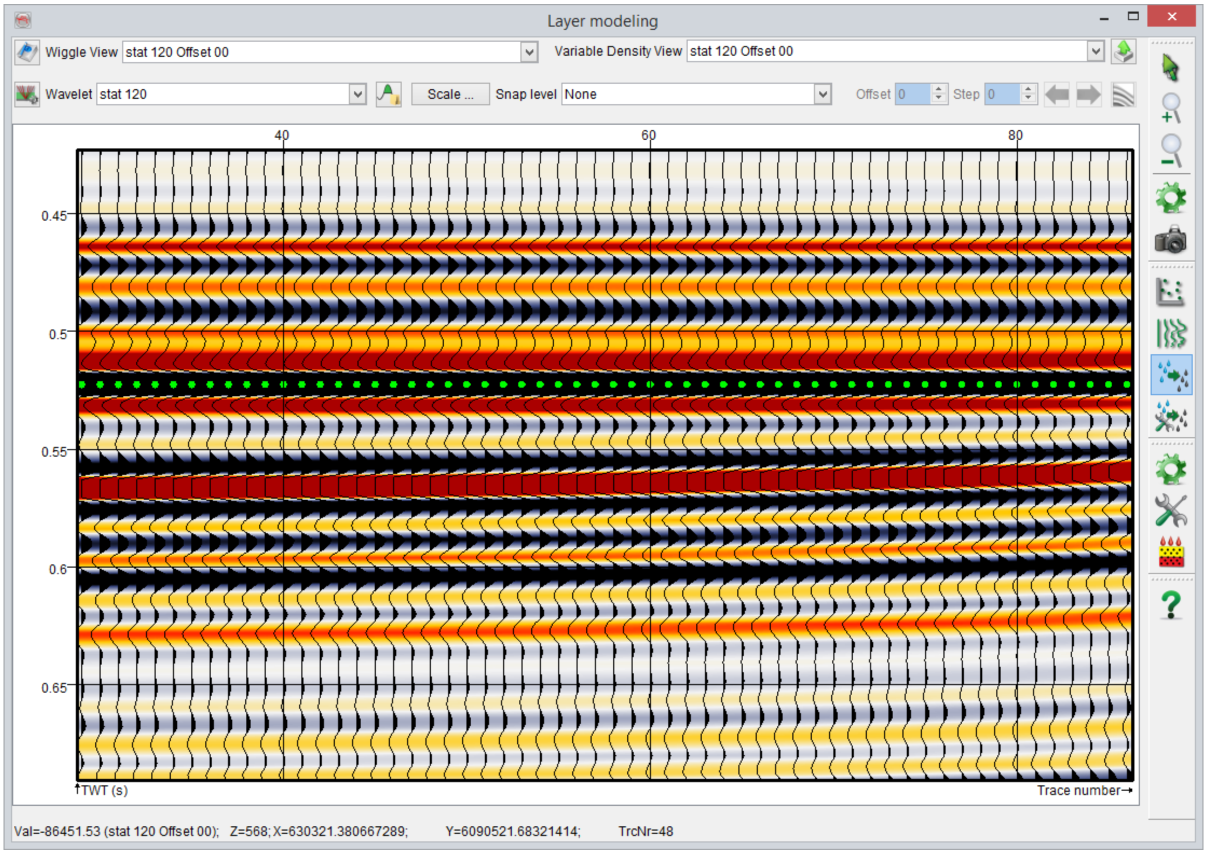
Snap level: None

Offset: 0

Step: 0

Val=-18258.79 (stat 120 Offset 00);   Z=512; X=630571.283204389;   Y=6090528.66753758;   TrcNr=58

Original synthetic seismic, corresponding to fully brine case (i.e. water saturation of 100%)



Wiggle View stat 120 Offset 00

Variable Density View stat 120 Offset 00

Wavelet stat 120

Snap level None

Offset 0

Step 0

Val=-86451.53 (stat 120 Offset 00); Z=568; X=630321.380667289; Y=6090521.68321414; TrcNr=48

Synthetic seismic after replacing brine with gas, corresponding to varying water saturation from 90% to 100%, left to right

HitCube Stochastic Inversion

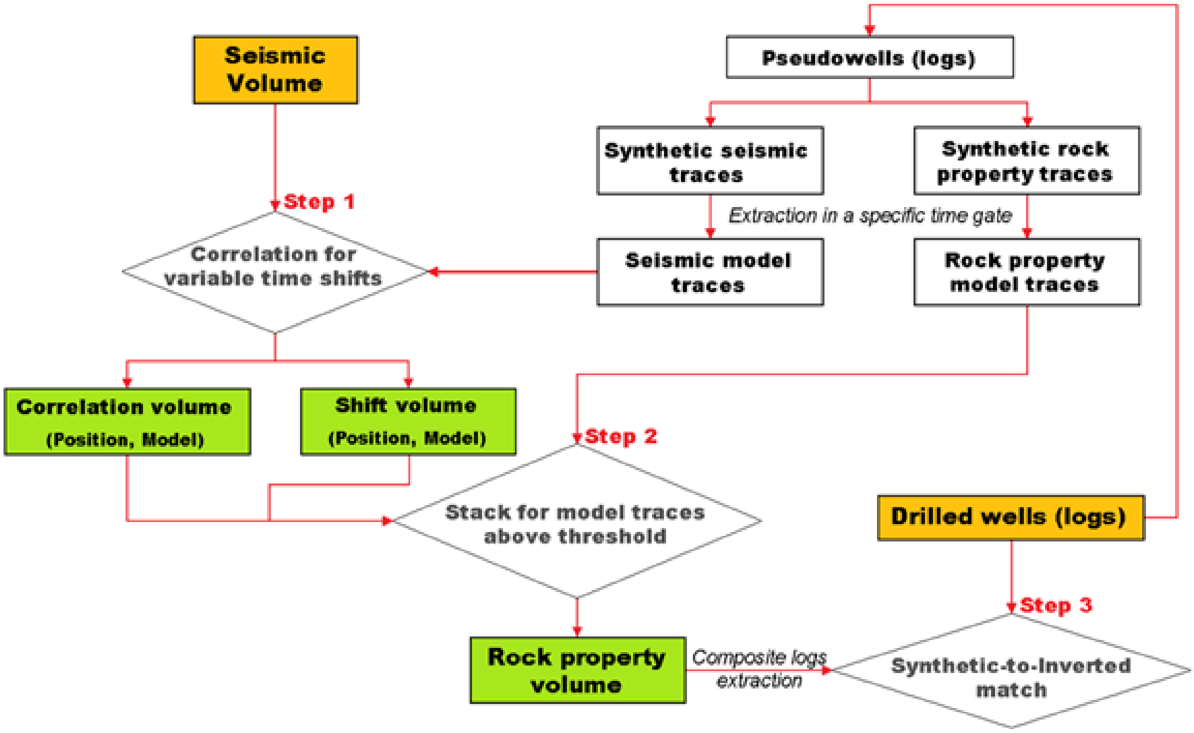
The HitCube is a stochastic inversion process. It assigns a spatial location to the simulated pseudo-wells such that rock properties can be computed from the tied models. The objective is indeed to predict reservoir properties with their relative uncertainties.

The HitCube workflow is divided in two steps:

1. The actual seismic traces in the volume are correlated with synthetic seismic from modeled wells.

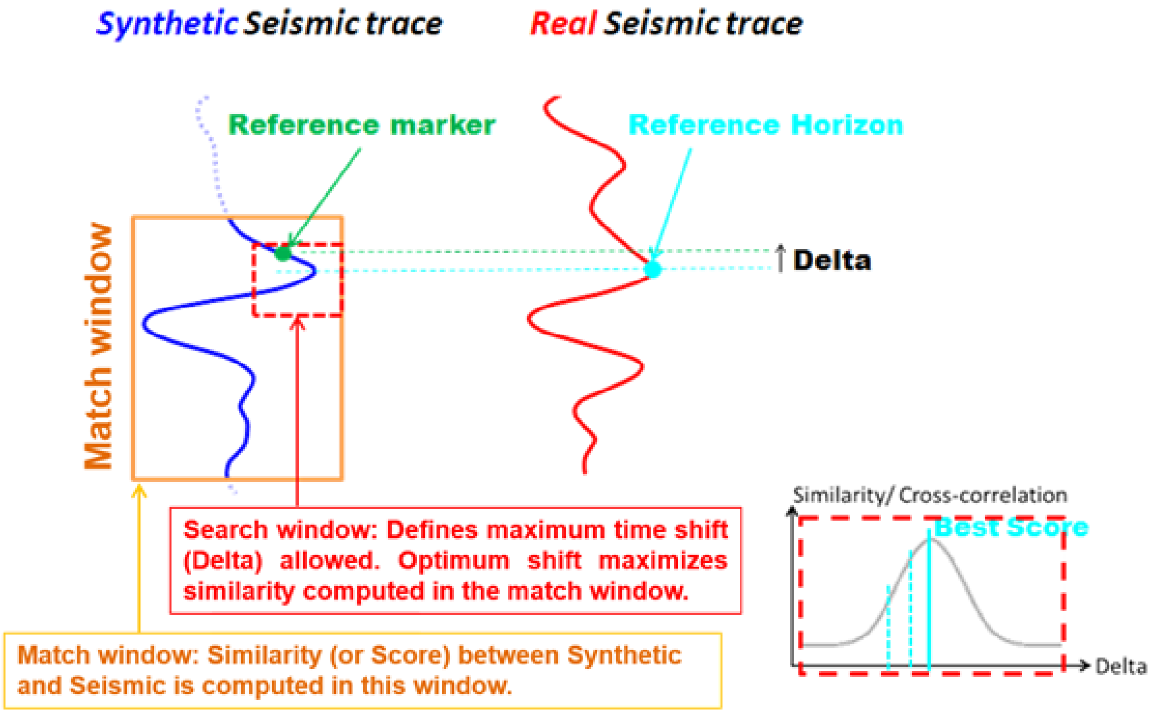
2. The property traces from corresponding models with a good correlation are stacked to build the output probability grids.

The inversion can be proceed regarding a selected horizon or within the entire volume. The workflow is very similar in the two cases. The inputs are the reference seismic and, for *along horizon*, a target/reference horizon.



Step 1 : Matching

The synthetic and real seismic are compared using a selected *matching method* (Similarity, Cross-correlation or Amplitude spectrum). The similarity is often preferred as it is scale sensitive and thus enables to distinguish between identical waveforms of different amplitude levels. The matching is achieved in a *matching gate* defined regarding the selected horizon if *along horizon* or if in *volume*, this time window is moved in a *Searching* time range that need to be specified. The matching window is determined in looking at the frequency spectrum of the data and QC in looking at the histogram of the output. The output of this first phase is a similarity and time shift pair for each model and each trace of the data, that is stored (internally, not visible to user) in two volumes called *ScoreCube* and *DeltaCube*. In these volumes, each sample corresponds to a model. The best matching window is the one where most of the traces correlate, i.e the less undefined values in the final outputs. It has to be selected carefully as a too large matching window will result in similarity between two different events. The matching gate can be QCed in looking at the histograms : the mean of the maximum Score (*Use best 1 models*) and the number of values used in the Delta cube at maximum Score must be optimized and the Delta cube at maximum Score must have a symmetrical histogram.



Step 2 : Stacking

The stacking step will generate the end products of the inversion, using the ScoreCube and DeltaCube as inputs, together with output features and/or logs from the pseudo-wells. For each model correspond a set of property logs that have been modeled. First the number of best models that sufficiently match the seismic data is selected. These models provide the dataset that will be stacked in order to compute the output target.The stacking can be done in taking the average or the median of the values. The final result is a predicted property volume based on probability. Additionally, various statistics (e.g. average) on score and delta values of the best matching pseudo-well models can be outputted.

Step 3: QC

The QC of HitCube inverted rock properties (e.g. AI, porosity) against available well log data can be done in OpendTect using cross-plots and visual comparisons.

Discrete property prediction

Usually property logs are continuous. But it exists logs like lithology logs which are discrete. In a litholog, the log will output a number for each lithology, for example : 1= sand, 2= shale, 3= limestone. To achieve a property prediction, each discrete value has to be considered on its own. In the case of the lithology prediction, a binary log will have to be computed for each lithology (1= one lithology, 0= the others). Then the HitCube process can be run for each of them. The result would be a probability of occurrence of the lithology at every location. The facies cube is computed in combining these probability volumes : the most likely lithology is output at each location. A confidence cube can also be created as the difference between the probability of the most likely lithology and the second most likely. The more this difference is important, the more trustable is the facies distinction at this location.

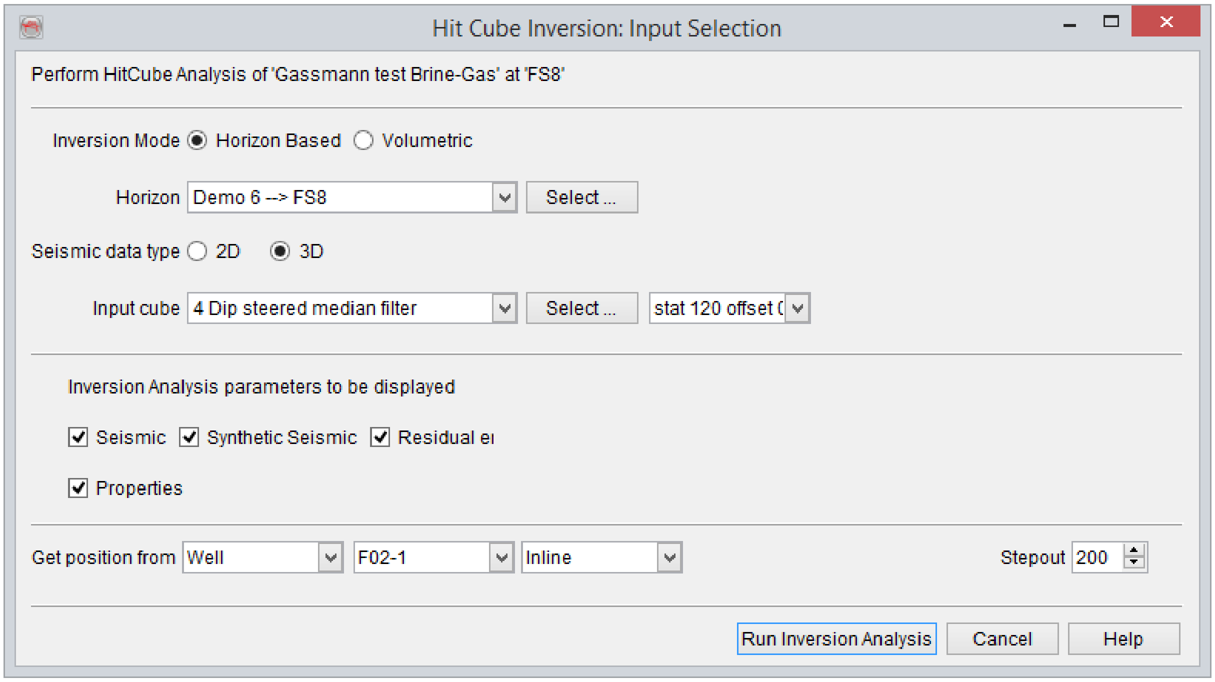
Discrete logs can be computed using any continuous log. For instance, using the Acoustic Impedance log, a new log can be computed that is equal to 1 when the AI value is higher than a given value and 0 otherwise. The HitCube will then give the probability that, at one location, AI is higher than this given value. The prediction of any log with more than two possible value follows the same workflow as the facies prediction.

Input Data and Inversion Type

The input window firstly allows you to chose the inversion type. The first one is the mode which proceeds the scoring regarding a selected horizon. The second one proceeds the scoring within the entire volume.

Then, before selecting any input data, you have to select the type of this one if it is a Lineset (2D) or a Cube (3D). And finally, the synthetic which will be used to match the data has to be selected, if your choice is not the default one.

When these fields are selected click on Ok to go to the Analysis window.


Hit Cube Inversion: Input Selection

Perform HitCube Analysis of 'Gassmann test Brine-Gas' at 'FS8'

Inversion Mode ● Horizon Based ○ Volumetric

Horizon Demo 6 --> FS8

Seismic data type ○ 2D ● 3D

Input cube 4 Dip steered median filter | stat 120 offset (

Inversion Analysis parameters to be displayed

☑ Seismic ☑ Synthetic Seismic ☑ Residual e

☑ Properties

Get position from Well | F02-1 | Inline | Stepout 200

HitCube Analysis

The Analysis window is maybe the most important window in the HitCube, because it allows you to set the parameters and to evaluate their QC.

The first part of this window allows you to select some parameters. Here, you can select the *Matching part of the models* by setting a time range. Notice the button on the right of this time range selection field. This button allows you to watch the matching traces defined with the previously selected time range. Then you can chose the matching method between four proposals :

- Similarity which is Amplitude and phase sensitive
- Cross-correlation which is only phase sensitive
- Similarity of amplitude spectra
- Cross-correlation of amplitude spectra

Then, you can select or not the *Search for the best match mode*. This mode gives you the access to a searching method you can apply before calculation and which will proceed the scoring at each tiny step in a range around an *initial shift*. This range is defined as plus and minus the aperture value you can set. The output score will be the highest found in this range. So this best match can be shifted in time. You will be able to select as output these time shifts.

Then, you are able to chose the outputs and to set the related parameters. So you can set the time gate of interest around the horizon you want to output if the *Along horizon* mode was selected in the input window. The step will be the sampling path of your output.

Just above, you can chose the *Matching type*. This option proposes two methods.

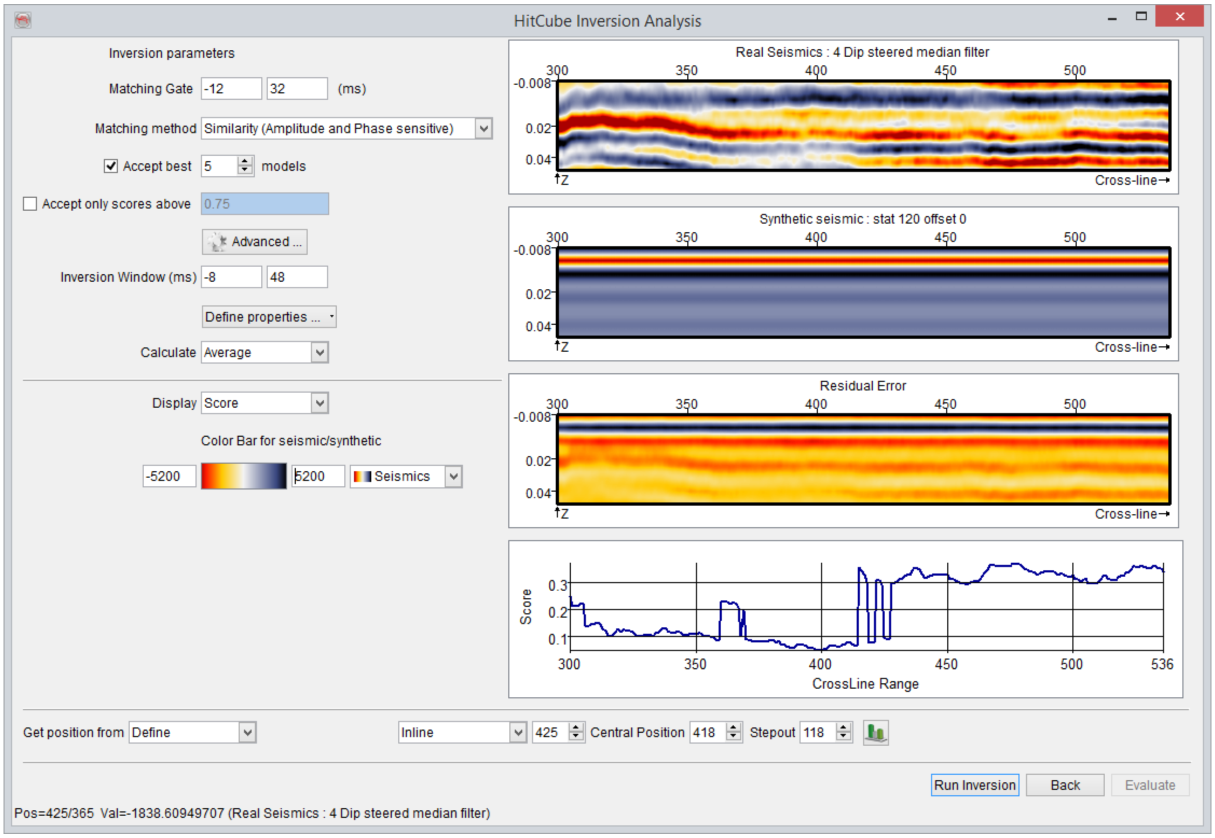
The first is the *Fixed window* one. This method will process only one score at each trace. This score is calculated around the horizon in the range set in the Matching part of the models field.

The second is the *Sliding window* method. This one will process a score at each step of the *Output time gate around the horizon*. The value is also calculated the time range defined in the *Matching part of the models field*.

Just below the output time get field, you chose the number of best models you want to use.

And finally, you have several output selection fields for which you can evaluate the QC of their related parameters with an evaluation window. The access to this evaluation window is the button at the end of each selected line. With these fields, you can select different outputs you will store :

- Score : stores the average, the median or the minimum value of the best pseudo-wells at each trace for the fixed window method and at each sample for the sliding window method, which number is set just above ;
- Snap delta Z : stores, for each trace or sample, the average or median of Z shifts, only available with the search mode ;
- Number of hits : stores the number of pseudo-wells which score value is higher than the one selected at each trace or sample ;
- Top model number ;
- Model properties : stores the average weighted by score values, the median, the most frequent value (adviced for discrete log) or the standard deviation of the selected property at each trace or sample. You can also normalize the score values by squeezing them in the range which can be set.



HitCube Inversion Analysis

Inversion parameters

Matching Gate -12 32 (ms)

Matching method Similarity (Amplitude and Phase sensitive)

☑ Accept best 5 models

☐ Accept only scores above 0.75

Inversion Window (ms) -8 48

Calculate Average

Display Score

Color Bar for seismic/synthetic

-5200 5200 Seismics

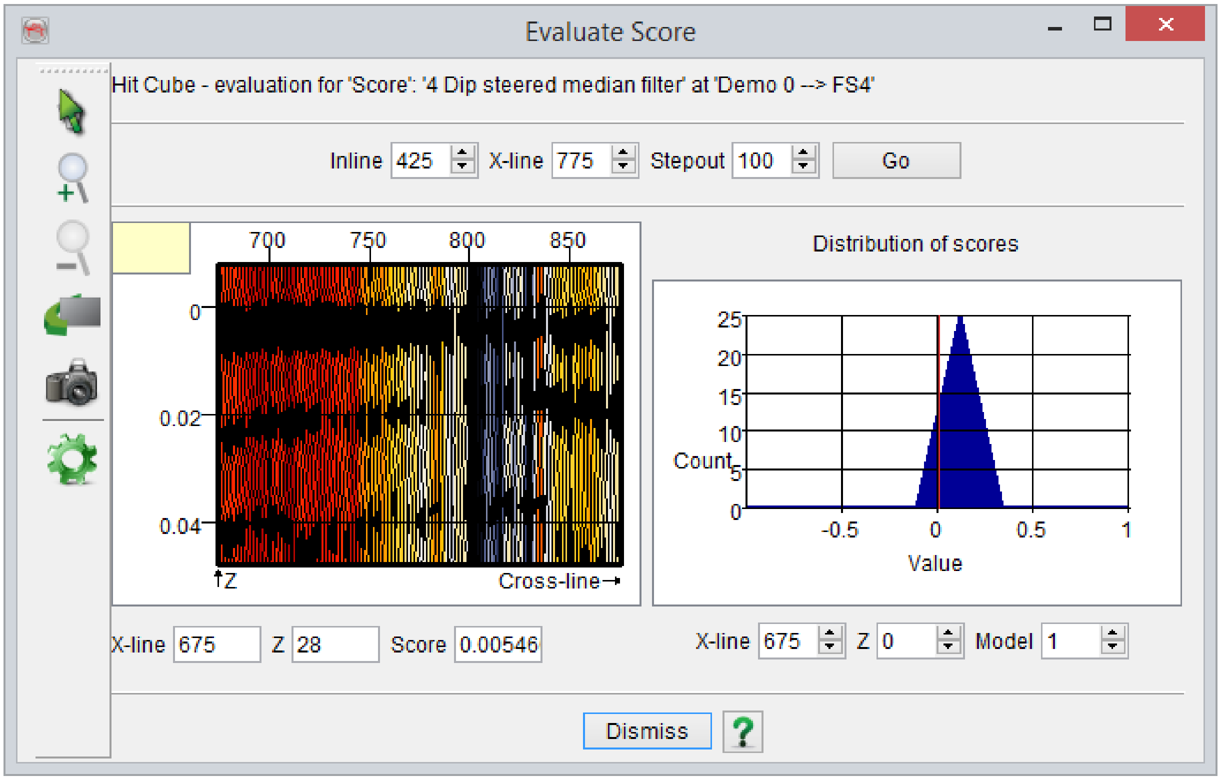
Get position from Define

Inline 425 Central Position 418 Stepout 118

Pos=425/365 Val=-1838.60949707 (Real Seismics : 4 Dip steered median filter)

HitCube Parameters QC Evaluation

The evaluation windows will allow you to test different sets of parameters and to watch a preview regarding these sets with graphs in different inlines and crosslines. You can also change the scale of the left preview window by changing the stepout.


Evaluate Score

Hit Cube - evaluation for 'Score': '4 Dip steered median filter' at 'Demo 0 --> FS4'

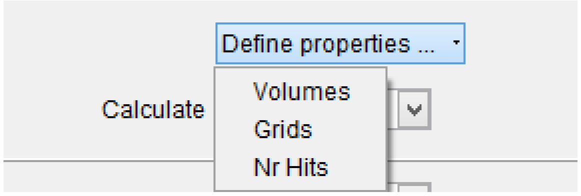
Inline 425 X-line 775 Stepout 100

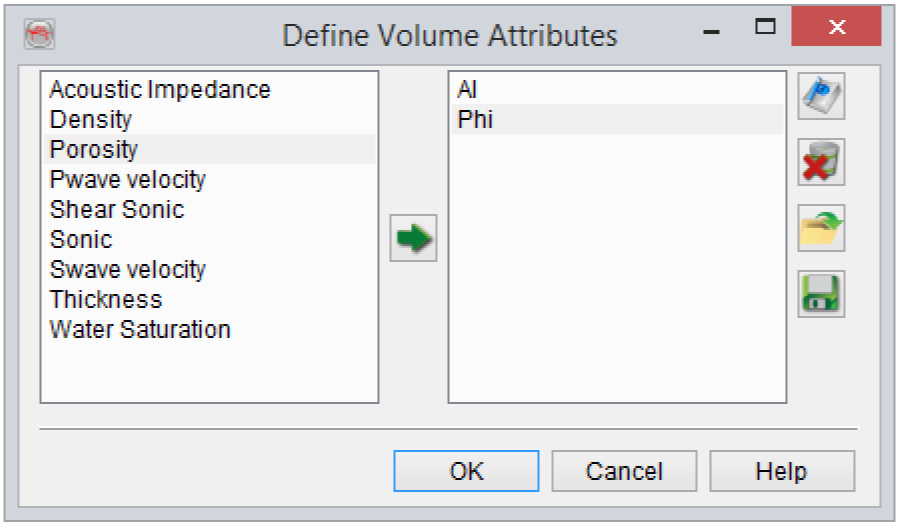
X-line 675 Z 28 Score 0.00546

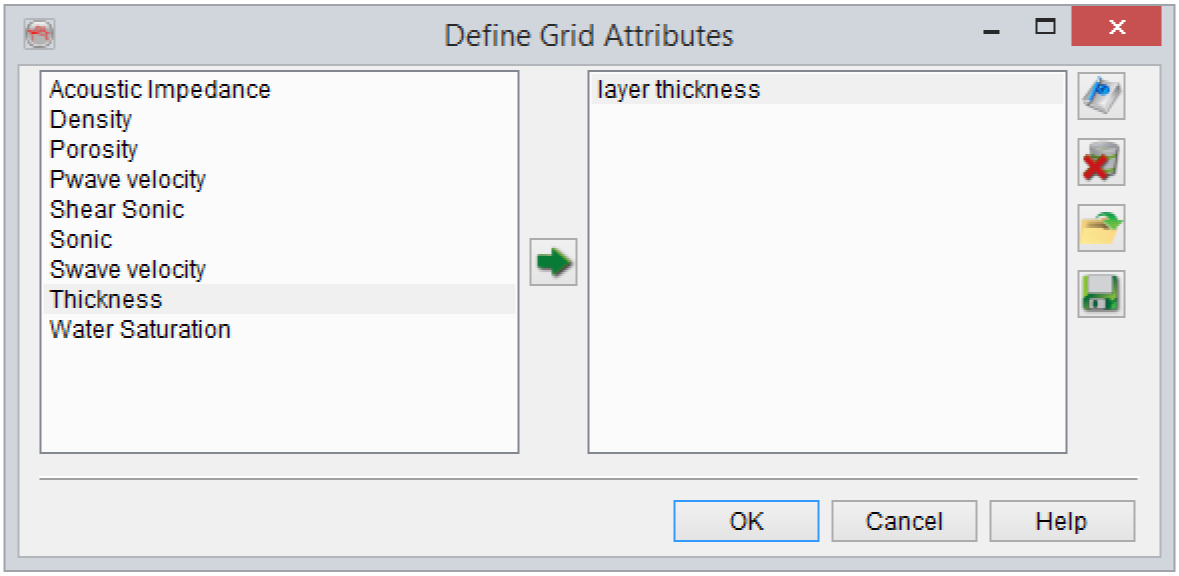
X-line 675 Z 0 Model 1

Define Output Layer Properties

For the last field ("Store model properties"), you need to select a property before QC it.

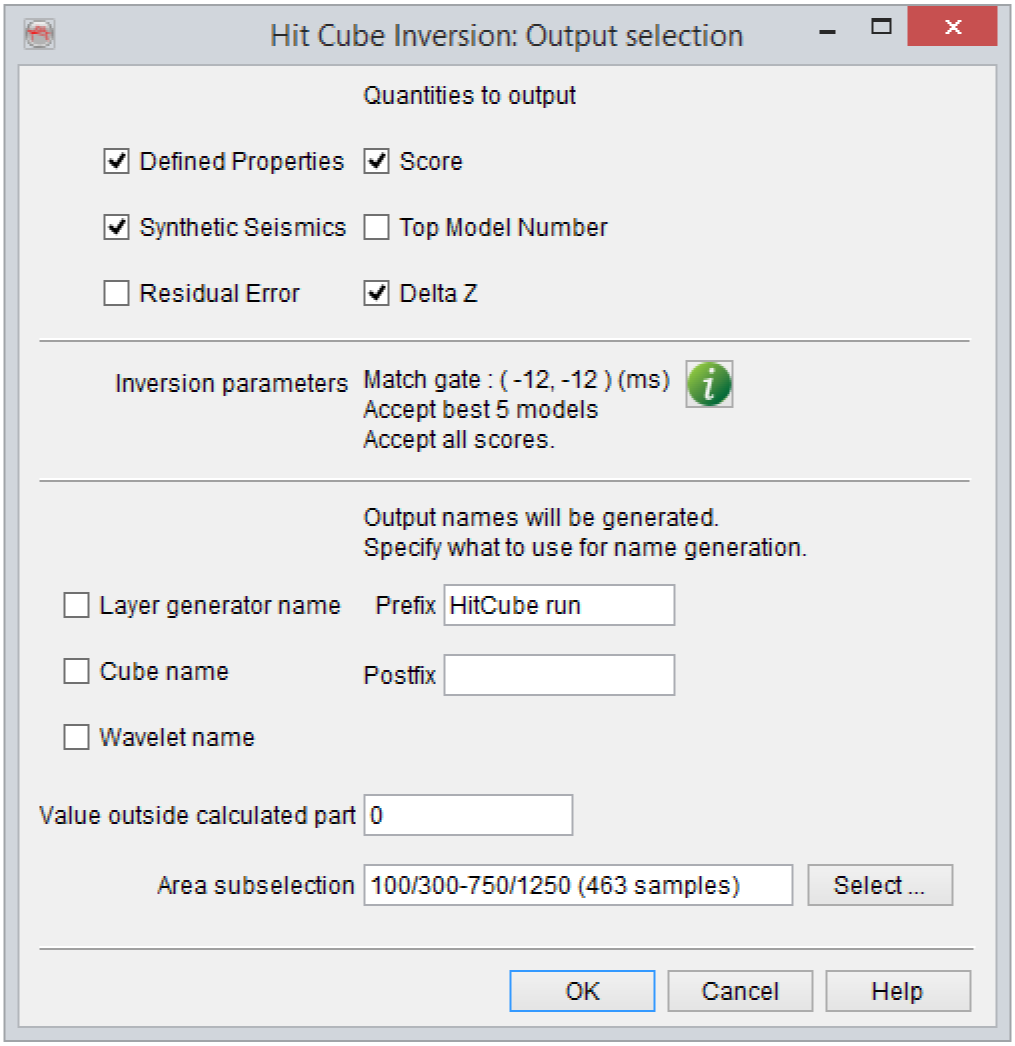






Output Data - Inversion Batch Processing

The last window, the output one, allows you to set the final parameters for the *Matching type, Output time gate around horizon* (if *Along horizon* mode was selected) and the *Area subselection*. Then you have the name generation options, like prefix, postfix or the wavelet name if it is available.



Quantities to output

☑ Defined Properties ☑ Score

☑ Synthetic Seismics ☐ Top Model Number

☐ Residual Error ☑ Delta Z

Inversion parameters Match gate : (-12, -12) (ms)
Accept best 5 models
Accept all scores.

Output names will be generated.
Specify what to use for name generation.

☐ Layer generator name Prefix HitCube run

☐ Cube name Postfix

☐ Wavelet name

Value outside calculated part 0

Area subselection 100/300-750/1250 (463 samples)

Neural Networks

Table of Contents

- Introduction
- Manage Neural Networks
- Pattern recognition (using PickSets)
- Property prediction (using well logs)
- Training and application

Introduction

The neural network module is used to manage, design, and train supervised and unsupervised neural networks.

Supervised Neural Networks

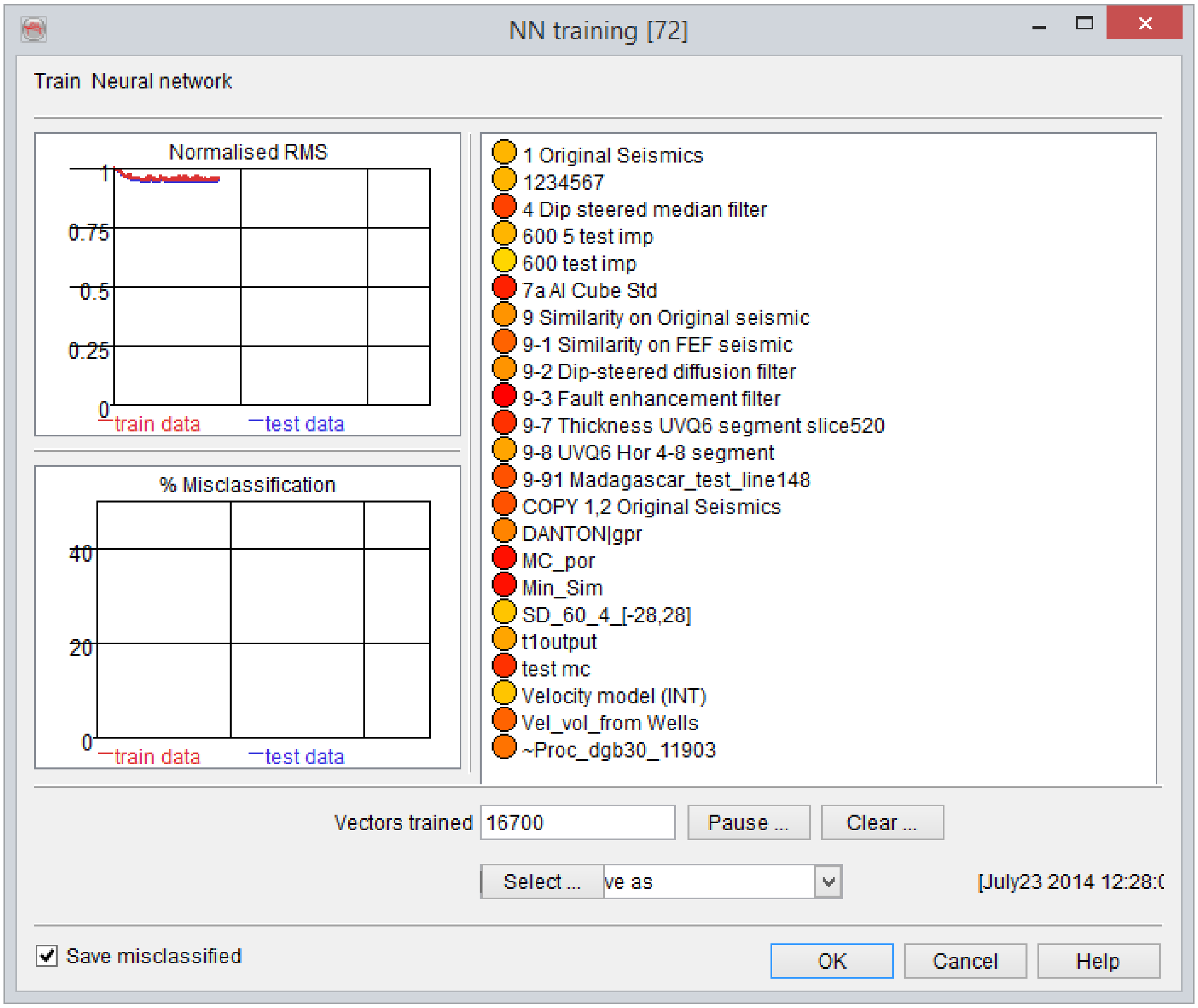
By supervised training, the user is teaching the network to distinguish between two or more pick sets. At each pick location, a number of attribute values is collected.

As an example, fault detection requires two picksets. The user needs to pick locations where a fault is present (pickset A), and locations where there aren't any faults (pickset B). This whole operation can serve two purposes:

- Efficiency/Automatization. You don't have the time to go through the entire set and pick each and every little fault.

- Find "more than meets the eye". Subtle faults can be present without noticing it but the network picks them up anyway.

It will be clear that the interpreter needs to define attributes that are sensitive to fault characteristics. Therefore, you need to make picksets of positions that contain different objects ("yes, there is a fault here") and counter-examples ('no, there aren't any faults here'). Other examples include *Channel vs. Non-channel* or maybe three choices, e.g. *Channel deposits, Overbank deposits and Shale*. All in all, the very first requirement is that you can pick examples from the data set.



1 Original Seismics
1234567
4 Dip steered median filter
600 5 test imp
600 test imp
7a AI Cube Std
9 Similarity on Original seismic
9-1 Similarity on FEF seismic
9-2 Dip-steered diffusion filter
9-3 Fault enhancement filter
9-7 Thickness UVQ6 segment slice520
9-8 UVQ6 Hor 4-8 segment
9-91 Madagascar_test_line148
COPY 1,2 Original Seismics
DANTON|gpr
MC_por
Min_Sim
SD_60_4_[-28,28]
t1output
test mc
Velocity model (INT)
Vel_vol_from Wells
~Proc_dgb30_11903

Vectors trained 16700

☑ Save misclassified

Neural Network Training for the ChimneyCube

During application, the software extracts the same attributes at volumes or horizons and uses the same neural network to predict whether we have a type A or type B situation there. When displaying, you can choose between two network outputs:

- The probability that a certain position is a "type A" location
- The type number (A=1, B=2 etc.).

At some locations, the network is more 'certain' than at other locations. You can get this ['confidence' by looking at the *probability* or at the *Confidence*.
During training of the neural network, you can monitor whether the network can figure out, given the attributes you have defined, whether a location is more like the ones in set A than the ones in set B. If the network would predict that a position would be type A, but in fact it actually is picked by you in pickset B, then that is a *misclassification* of the network. The lower the number of misclassification, the better.

Unsupervised Neural Networks

In the *unsupervised approach*, you want the network to come up with a "*natural*" division of the seismic data. This approach is very useful when you want to perform, for example, horizon-based or volume-based segmentation. After training the network and application of the neural network output to an element, the results should be interpreted.

The (single) pickset holds the example positions at which the software calculates the chosen attributes. Therefore, each position in the pickset will yield a vector of values. The result of the extraction of the attributes at each picked location is the *training set*.

The neural network tries to cluster this set of vectors. Similar vectors go into the same *Segment*. This operation can be seen as subdividing the hyperspace of the attribute vectors in compartments. Each compartment has a centre: *the cluster center*.

After the training, the network can be applied to a horizon, time-slice, or volume. That means that the vectors are extracted in a volume or along a horizon. The network can then classify all those vectors into a *Segment*. A vector can be close to the cluster centre or further away from it, which is indicated by the *Match*. The closer to the cluster center, the higher the match.

Manage Neural Networks

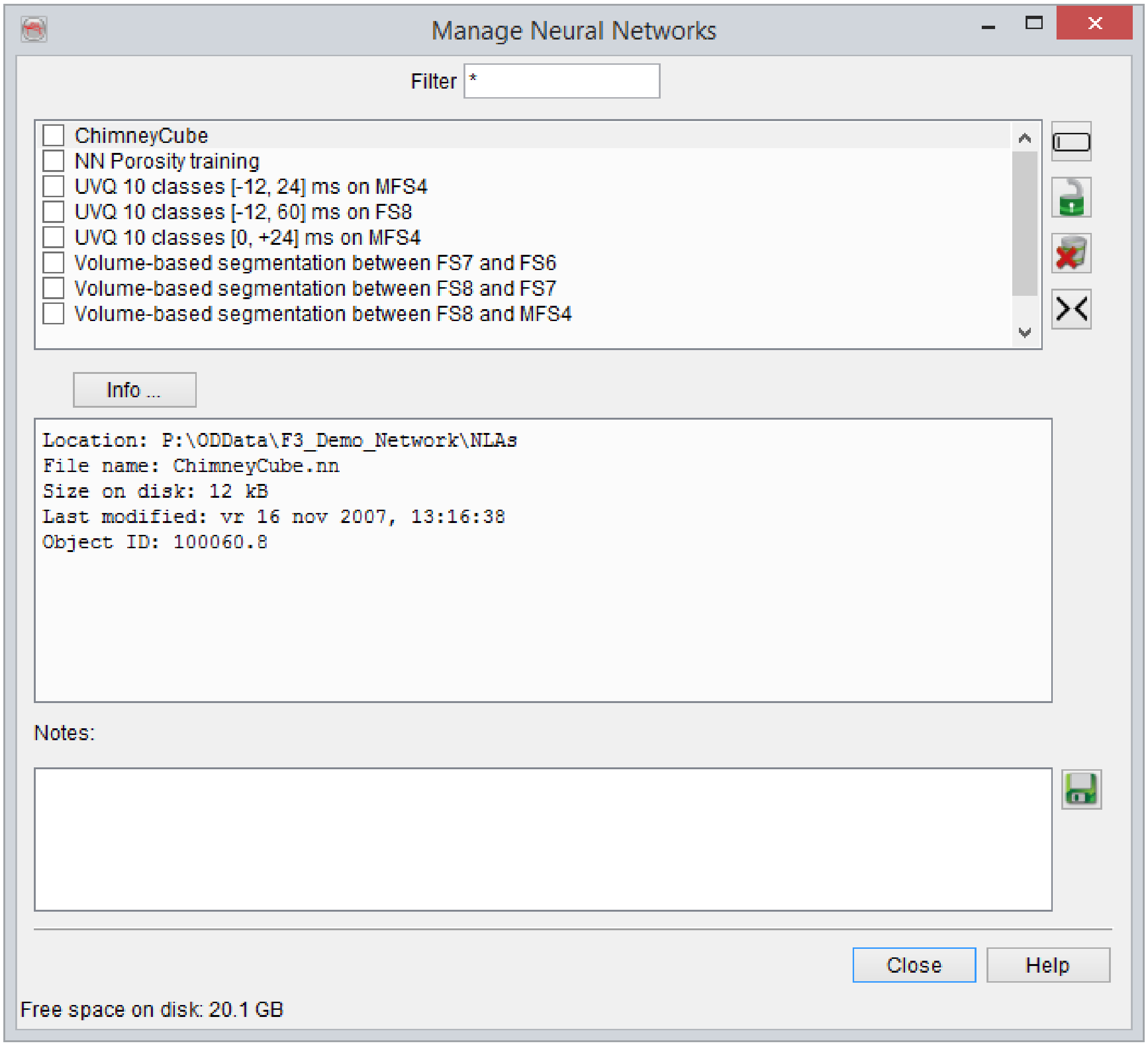
Neural networks can be managed from two locations:

- From **Survey-->Manage**: Delete, rename, set the default neural network.
- From **Analysis-->Neural Networks**: Create a new neural network, load an existing before applying it on tree elements.

Data Manager

The neural network management allows to manage them, i.e.: Change disk location, rename, remove, copy, etc.

This window will contain the list of stored neural networks as show below:



Manage Neural Networks

Filter *

☐ ChimneyCube
☐ NN Porosity training
☐ UVQ 10 classes [-12, 24] ms on MFS4
☐ UVQ 10 classes [-12, 60] ms on FS8
☐ UVQ 10 classes [0, +24] ms on MFS4
☐ Volume-based segmentation between FS7 and FS6
☐ Volume-based segmentation between FS8 and FS7
☐ Volume-based segmentation between FS8 and MFS4

```
Location: P:\ODData\F3_Demo_Network\NLAs
File name: ChimneyCube.nn
Size on disk: 12 kB
Last modified: vr 16 nov 2007, 13:16:38
Object ID: 100060.8
```

Notes:

Free space on disk: 20.1 GB

For each neural network, you can press the Info button to get information on how it was trained.

Neural Network Management

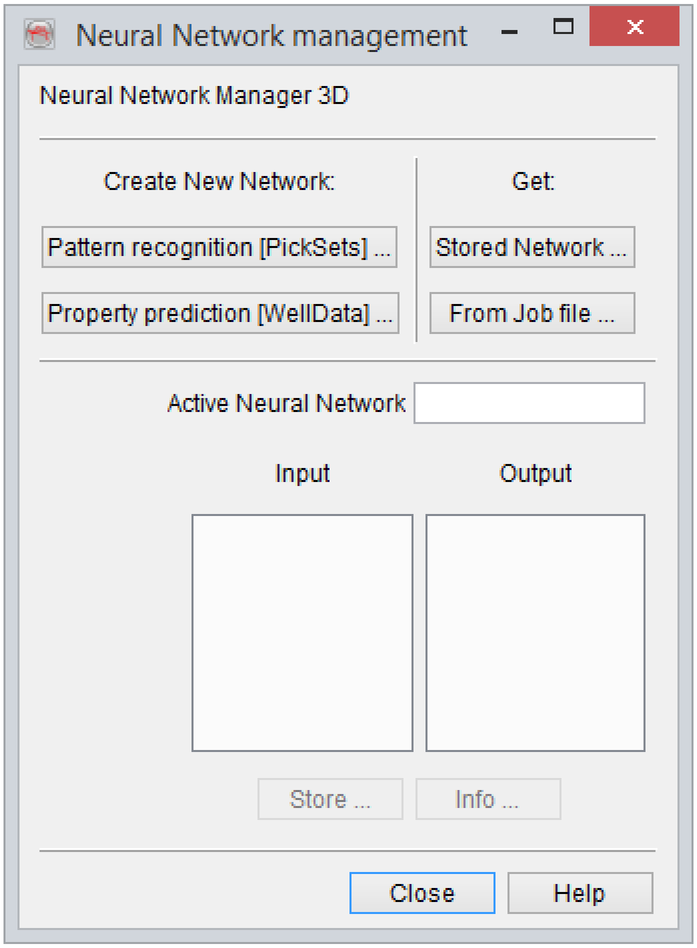
When clicking the icon, the *Neural Network management* window appears. This window allows to work with neural networks in the current session: Train a new network, save the current network, load a stored network before applying it to a tree item.

To create a new network, one must first choose the appropriate mode:

- **Pattern recognition**: To cluster your dataset(s) in several groups (classes), whose distribution is either unknown (unsupervised mode: 1 pickset), or already interpreted (supervised mode, at least 2 picksets)
- **Property prediction**: To generate a volume representing the prediction of a given well log, extracted by relating it to seismic attributes.

When a neural network is loaded (clicking Stored Network), the input and output nodes of this network are shown in the lists.

When the training of a neural network is finished, the user can store the trained neural network or look at the relevant statistics with the **Info** button.



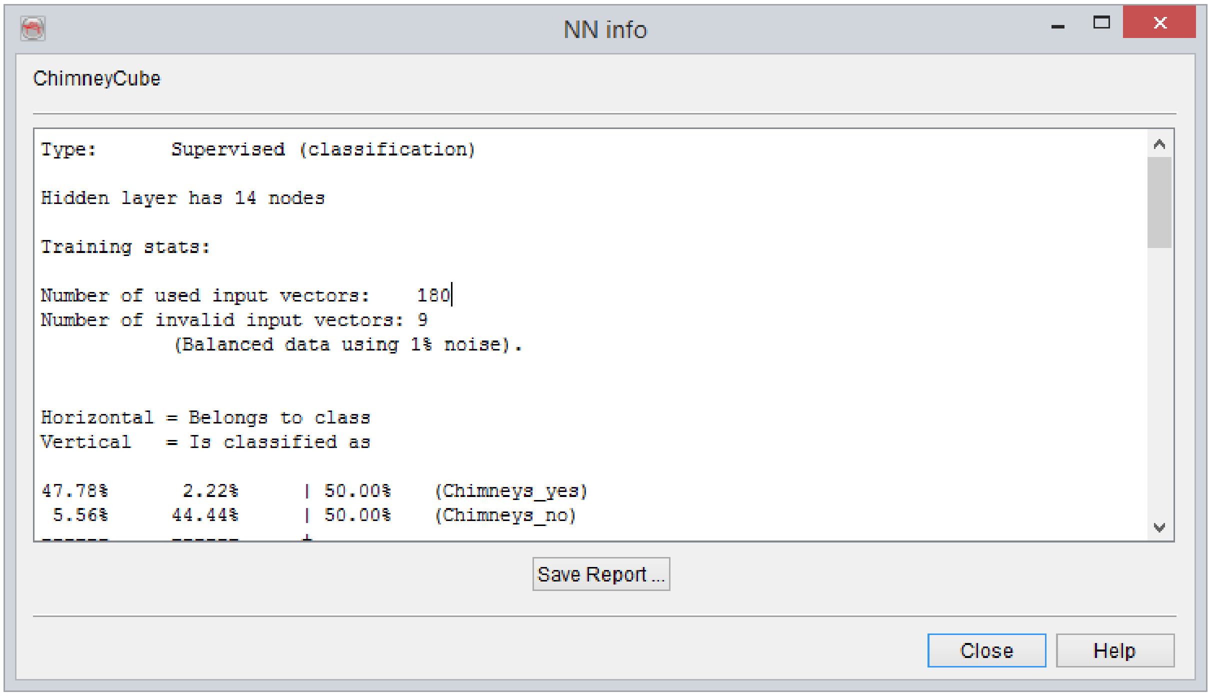
Neural Network Information

After training of a neural network, the information window shows some information on the network topology, the attributes that were used, and the relative importance (weight)--on a scale from 0 to 100--of the individual attributes. Be careful to judge these weights. This information tells a lot about individual attributes, but only a little about relationships between attributes. This information can be stored into a report (a simple text file) by pressing the *Save Report* button. Press *Dismiss* to go back to the Neural Network Manager.

Supervised Neural Network Information

The information window shows neural network information for the training set and the test set. For each set the numbers of usable and invalid vectors are shown.

Balancing is automatically performed between the data extraction and the training. This ensures that there is an equal representation of all inputs picksets. Misclassification information is displayed based on the membership to the input pickset.

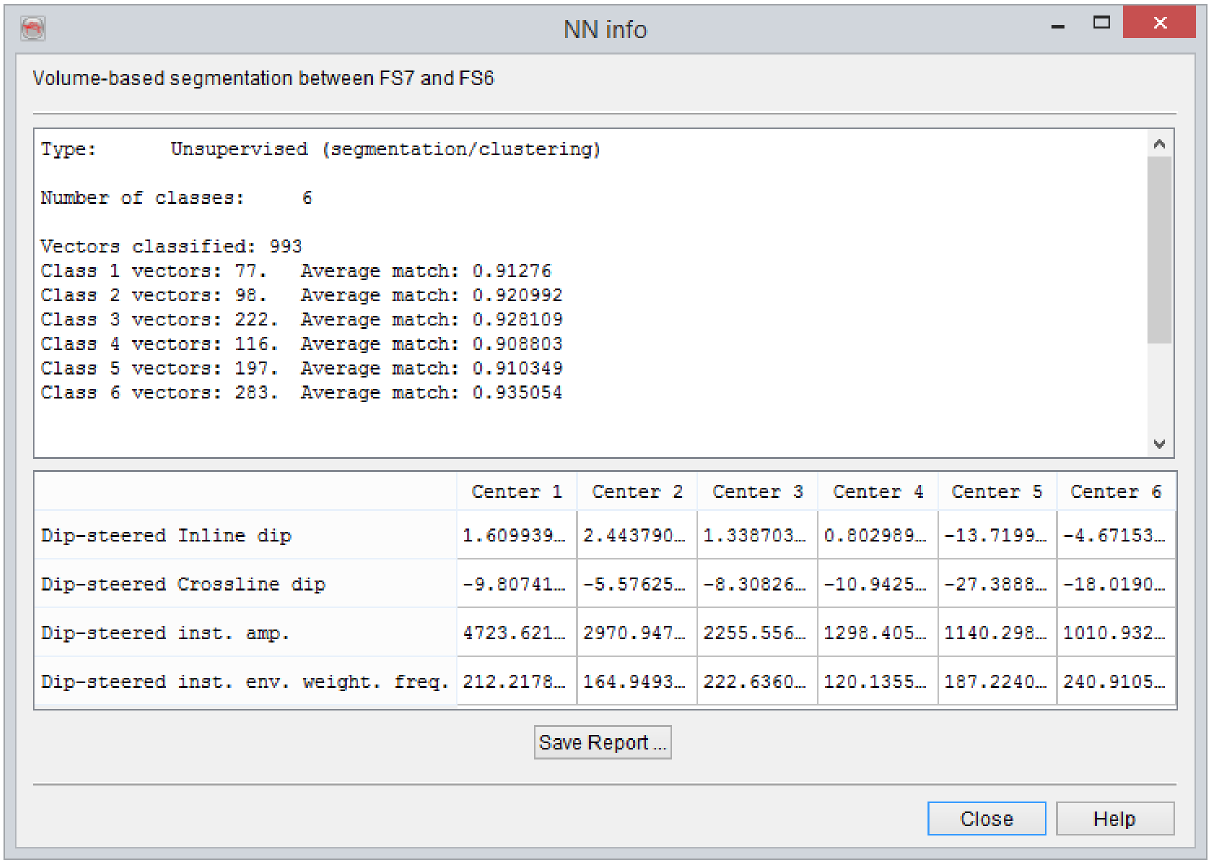


```
Type:        Supervised (classification)

Hidden layer has 14 nodes

Training stats:

Number of used input vectors:     180
Number of invalid input vectors: 9
            (Balanced data using 1% noise).


Horizontal = Belongs to class
Vertical   = Is classified as

47.78%      2.22%      | 50.00%    (Chimneys_yes)
 5.56%     44.44%      | 50.00%    (Chimneys_no)
```

Unsupervised Neural Network Information

The information window for an unsupervised neural network will show the number of vectors assigned to each class and their average match to the corresponding class center.

The last table will display the attribute values of each class center:

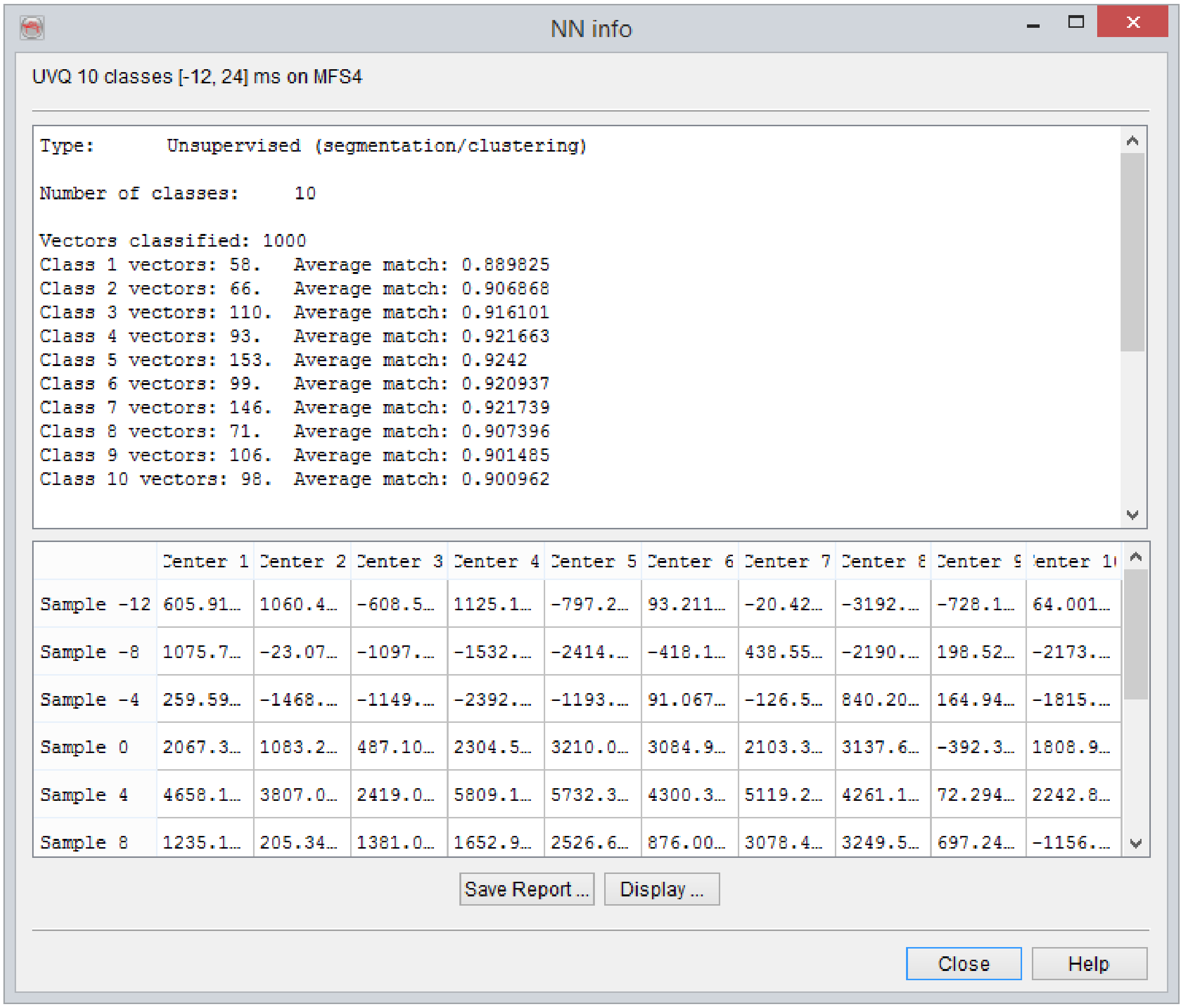


Volume-based segmentation between FS7 and FS6

```
Type:       Unsupervised (segmentation/clustering)

Number of classes:     6

Vectors classified: 993
Class 1 vectors: 77.   Average match: 0.91276
Class 2 vectors: 98.   Average match: 0.920992
Class 3 vectors: 222.  Average match: 0.928109
Class 4 vectors: 116.  Average match: 0.908803
Class 5 vectors: 197.  Average match: 0.910349
Class 6 vectors: 283.  Average match: 0.935054
```

	Center 1	Center 2	Center 3	Center 4	Center 5	Center 6
Dip-steered Inline dip	1.609939...	2.443790...	1.338703...	0.802989...	-13.7199...	-4.67153...
Dip-steered Crossline dip	-9.80741...	-5.57625...	-8.30826...	-10.9425...	-27.3888...	-18.0190...
Dip-steered inst. amp.	4723.621...	2970.947...	2255.556...	1298.405...	1140.298...	1010.932...
Dip-steered inst. env. weight. freq.	212.2178...	164.9493...	222.6360...	120.1355...	187.2240...	240.9105...

When the segmentation was performed on a waveform (i.e. all input attributes contain the string "Sample") an extra option *display* will enable the visualization of the data of this table in a 2D viewer in order to see the class centres:

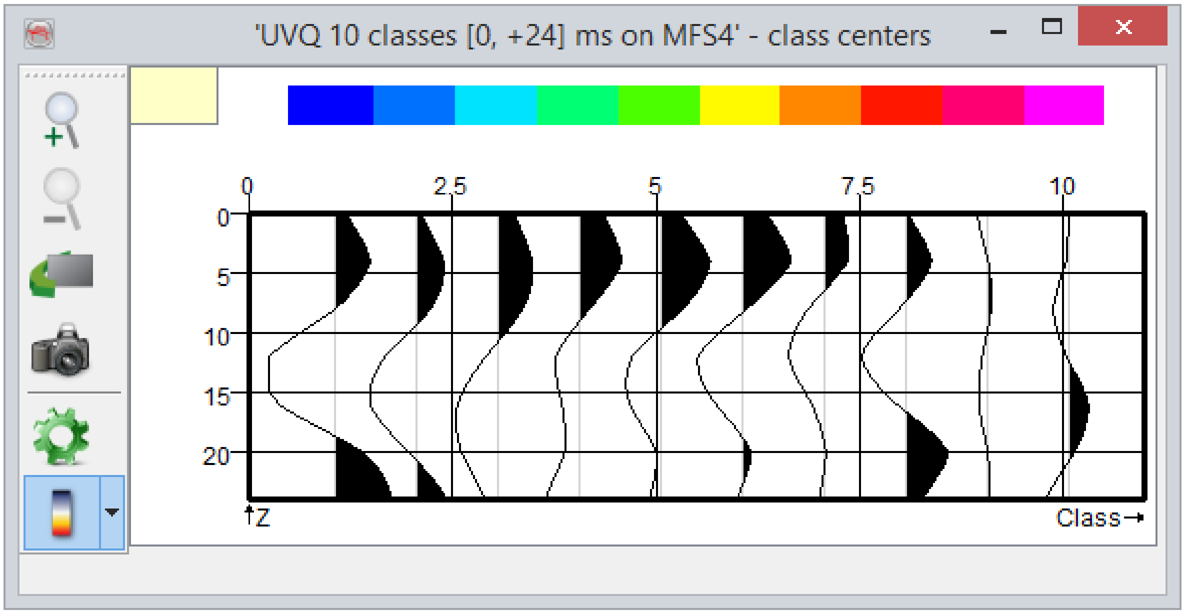


UVQ 10 classes [-12, 24] ms on MFS4

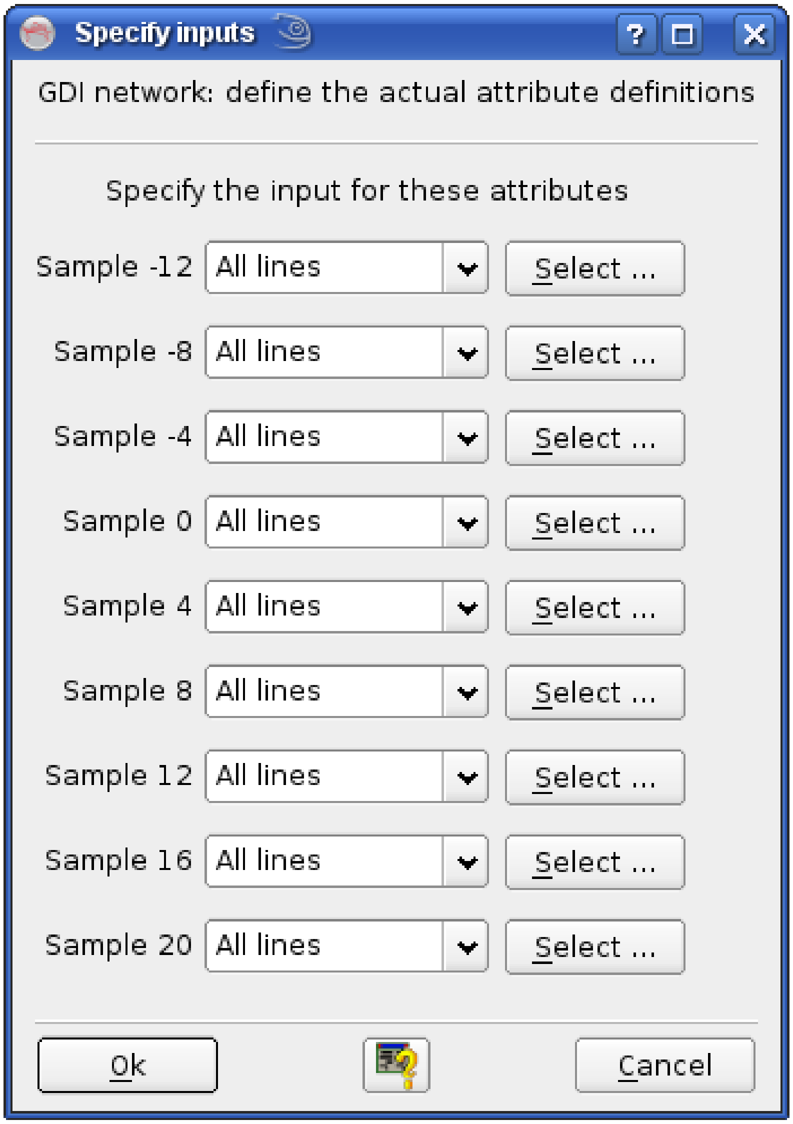
```
Type:       Unsupervised (segmentation/clustering)

Number of classes:     10

Vectors classified: 1000
Class 1 vectors: 58.   Average match: 0.889825
Class 2 vectors: 66.   Average match: 0.906868
Class 3 vectors: 110.  Average match: 0.916101
Class 4 vectors: 93.   Average match: 0.921663
Class 5 vectors: 153.  Average match: 0.9242
Class 6 vectors: 99.   Average match: 0.920937
Class 7 vectors: 146.  Average match: 0.921739
Class 8 vectors: 71.   Average match: 0.907396
Class 9 vectors: 106.  Average match: 0.901485
Class 10 vectors: 98.  Average match: 0.900962
```

	Center 1	Center 2	Center 3	Center 4	Center 5	Center 6	Center 7	Center 8	Center 9	enter 1
Sample -12	605.91...	1060.4...	-608.5...	1125.1...	-797.2...	93.211...	-20.42...	-3192....	-728.1...	64.001...
Sample -8	1075.7...	-23.07...	-1097....	-1532....	-2414....	-418.1...	438.55...	-2190....	198.52...	-2173....
Sample -4	259.59...	-1468....	-1149....	-2392....	-1193....	91.067...	-126.5...	840.20...	164.94...	-1815....
Sample 0	2067.3...	1083.2...	487.10...	2304.5...	3210.0...	3084.9...	2103.3...	3137.6...	-392.3...	1808.9...
Sample 4	4658.1...	3807.0...	2419.0...	5809.1...	5732.3...	4300.3...	5119.2...	4261.1...	72.294...	2242.8...
Sample 8	1235.1...	205.34...	1381.0...	1652.9...	2526.6...	876.00...	3078.4...	3249.5...	697.24...	-1156....

]


'UVQ 10 classes [0, +24] ms on MFS4' - class centers
0
2.5
5
7.5
10
0
5
10
15
20
Z
Class

Import GDI Networks Window



When a GDI neural network is used in OpendTect, the input to the network will be examined. For each input, there are two options:

1. OpendTect recognizes the attribute and has an equivalent available. In that case, you'll need to specify from which cube OpendTect should extract this attribute. This is usually (but not necessarily) the cube from which the training set is extracted. This has to be specified in the top section of the window.

2. OpendTect has no equivalent attribute available. You'll have to extract the

attribute into a "*Seismic Cube*" in GDI and provide this cube in the bottom section of the window. Note that "*Frequency*" is one of these unmatched attributes (GDI has event-based, OpendTect has FFT-based frequency).

Depending on the availability of the attributes in OpendTect, the applicable part of the window shown here appears.

Pattern Recognition (using Picksets)

The Neural Network plugin supports two types of neural networks based on picks: fully connected Multi-Layer-Perceptrons (MLPs) and Unsupervised Vector Quantizers (UVQs). MLPs are used in supervised (*with a priori, learn by example*) mode, while UVQs are used in unsupervised experiments (*segmentation = clustering*).

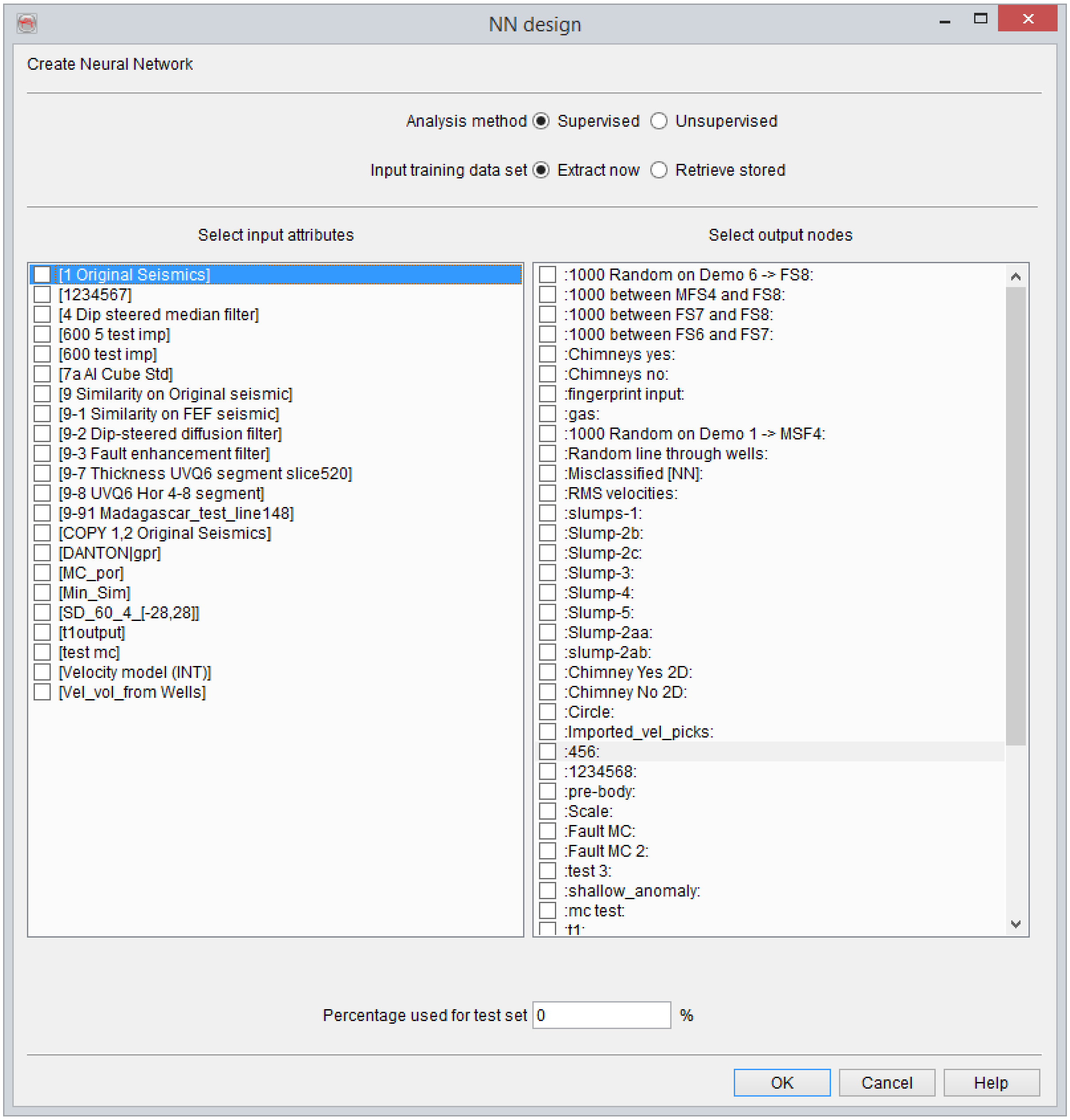
Analysis method. The *Supervised* method allows the choice of one or more output nodes. The groups of nodes (*PickSets*) indicate how the neural network should separate the character found in the input attributes. *Unsupervised* separates the nodes in the (single!) pickset based on a clustering in a user defined number of classes. A saved pickset can be used but a random pickset can also be created for this purpose.

Input Training data set. Specify whether the input data set must be extracted on the fly, or retrieved from a stored input set. An input training set consists of a range of attributes (names and values) at given example locations. To create a training set the user must specify which attributes to use and at which locations these attributes must be calculated.

Select input/output attributes. The *Select input attributes* lists all attributes defined in the current attribute set as well as all data that is stored on disk. Select any or all of these to serve as input to the neural network. The *Select output nodes* on the right contains all available pickset groups. Select the pickset group containing the locations at which attributes must be extracted to create a training set. Note that for an object probability cube such as TheChimneyCube the user needs two pickset groups: chimneys and non-chimneys.

Percentage used for test set. In the supervised mode, it is recommended to create a subset for testing the neural network's performance during training, specify a *Percentage* of the pickset to use for the test set. The test set is created by randomly drawing example locations from the selected picksets. Test set examples will not be used to update the neural network weight set during training, they are merely passed through the network to compare the network's classification with the actual classification.

Number of Classes. In unsupervised mode, attributes at locations in the specified pickset are clustered (segmented) into the specified *Number of classes*. During the training phase the UVQ network learns to find the cluster centers. At each iteration, when an vector of values has been assigned to a cluster, the cluster center is moved to minimize the (euclidean) distance with the different vectors of attributes values. In the application phase the input attributes are compared to each cluster center. The input is assigned to the winning segment, which is a number from 1 to N, where N is the number of clusters. In addition, the network calculates how close the input is to the cluster center of the winning cluster. This measure of confidence is called a *match*, which can range between 1 (perfect match, i.e. input and cluster center are the same) and 0 (input and cluster center are completely different).



Create Neural Network

Analysis method ● Supervised ○ Unsupervised

Input training data set ● Extract now ○ Retrieve stored

Select input attributes

- ☐ [1 Original Seismics]
- ☐ [1234567]
- ☐ [4 Dip steered median filter]
- ☐ [600 5 test imp]
- ☐ [600 test imp]
- ☐ [7a AI Cube Std]
- ☐ [9 Similarity on Original seismic]
- ☐ [9-1 Similarity on FEF seismic]
- ☐ [9-2 Dip-steered diffusion filter]
- ☐ [9-3 Fault enhancement filter]
- ☐ [9-7 Thickness UVQ6 segment slice520]
- ☐ [9-8 UVQ6 Hor 4-8 segment]
- ☐ [9-91 Madagascar_test_line148]
- ☐ [COPY 1,2 Original Seismics]
- ☐ [DANTON|gpr]
- ☐ [MC_por]
- ☐ [Min_Sim]
- ☐ [SD_60_4_[-28,28]]
- ☐ [t1output]
- ☐ [test mc]
- ☐ [Velocity model (INT)]
- ☐ [Vel_vol_from Wells]

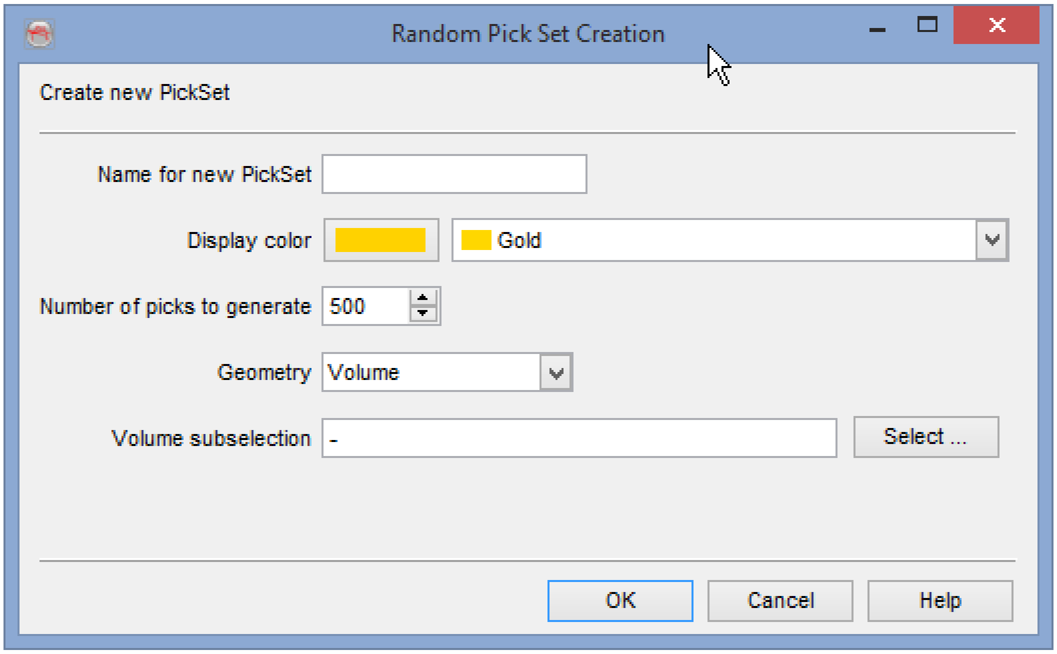
Select output nodes

- ☐ :1000 Random on Demo 6 -> FS8:
- ☐ :1000 between MFS4 and FS8:
- ☐ :1000 between FS7 and FS8:
- ☐ :1000 between FS6 and FS7:
- ☐ :Chimneys yes:
- ☐ :Chimneys no:
- ☐ :fingerprint input:
- ☐ :gas:
- ☐ :1000 Random on Demo 1 -> MSF4:
- ☐ :Random line through wells:
- ☐ :Misclassified [NN]:
- ☐ :RMS velocities:
- ☐ :slumps-1:
- ☐ :Slump-2b:
- ☐ :Slump-2c:
- ☐ :Slump-3:
- ☐ :Slump-4:
- ☐ :Slump-5:
- ☐ :Slump-2aa:
- ☐ :slump-2ab:
- ☐ :Chimney Yes 2D:
- ☐ :Chimney No 2D:
- ☐ :Circle:
- ☐ :Imported_vel_picks:
- ☐ :456:
- ☐ :1234568:
- ☐ :pre-body:
- ☐ :Scale:
- ☐ :Fault MC:
- ☐ :Fault MC 2:
- ☐ :test 3:
- ☐ :shallow_anomaly:
- ☐ :mc test:
- ☐ :t1:

Percentage used for test set 0 %

Fast pickset creation

The neural network extraction module can quickly create in unsupervised mode the required input set of training locations by pressing the '*Create*' button:



Name for new PickSet

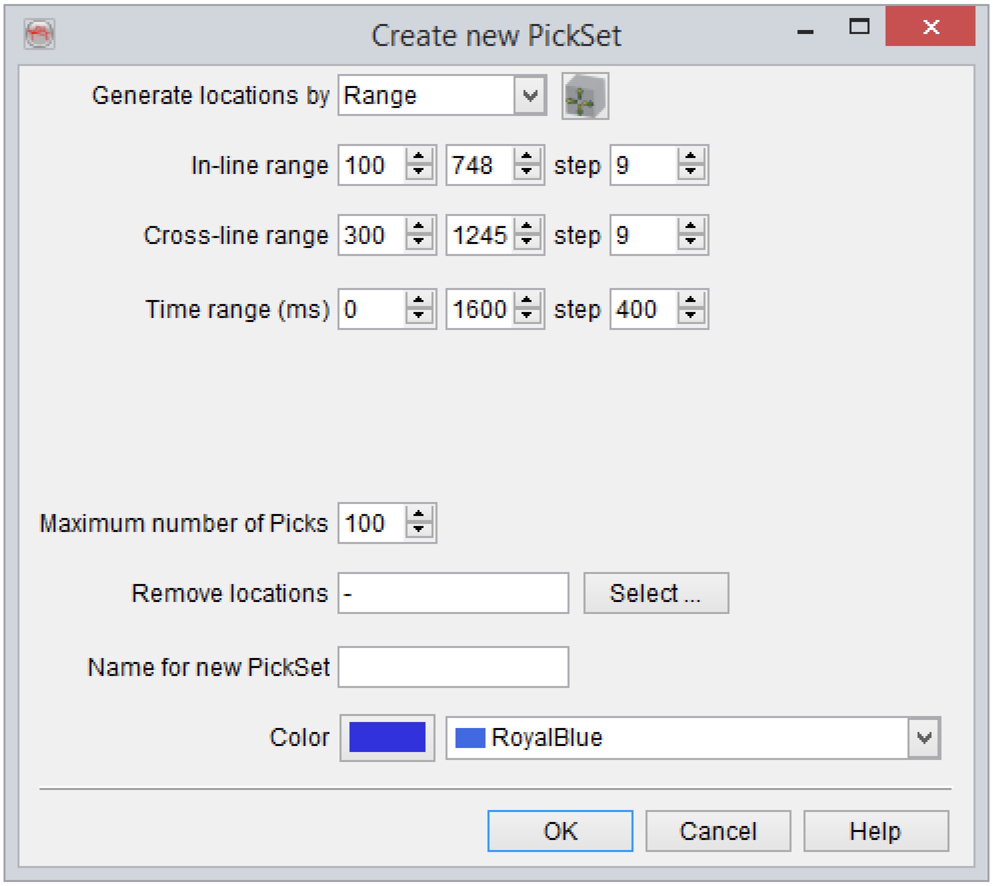
Display color 🟨 Gold

Number of picks to generate 500

Geometry Volume

Volume subselection -

Note: that more sophisticated 3D pickset creation tools are available from the tree by right-clicking on the element '*Pickset/Polygon*'.



Create new PickSet

Generate locations by: Range

In-line range: 100 748 step 9

Cross-line range: 300 1245 step 9

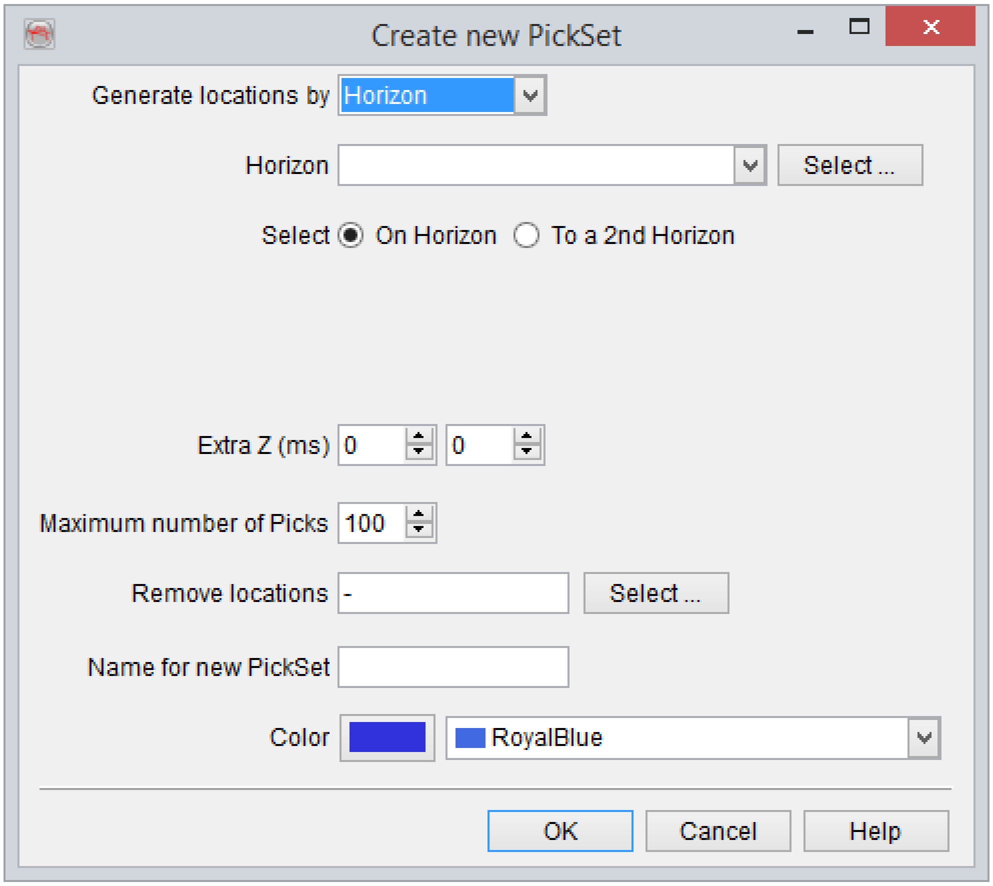
Time range (ms): 0 1600 step 400

Maximum number of Picks: 100

Remove locations: -

Name for new PickSet:

Color: RoyalBlue



Create new PickSet

Generate locations by: Horizon

Horizon:

Select ● On Horizon ○ To a 2nd Horizon

Extra Z (ms): 0 0

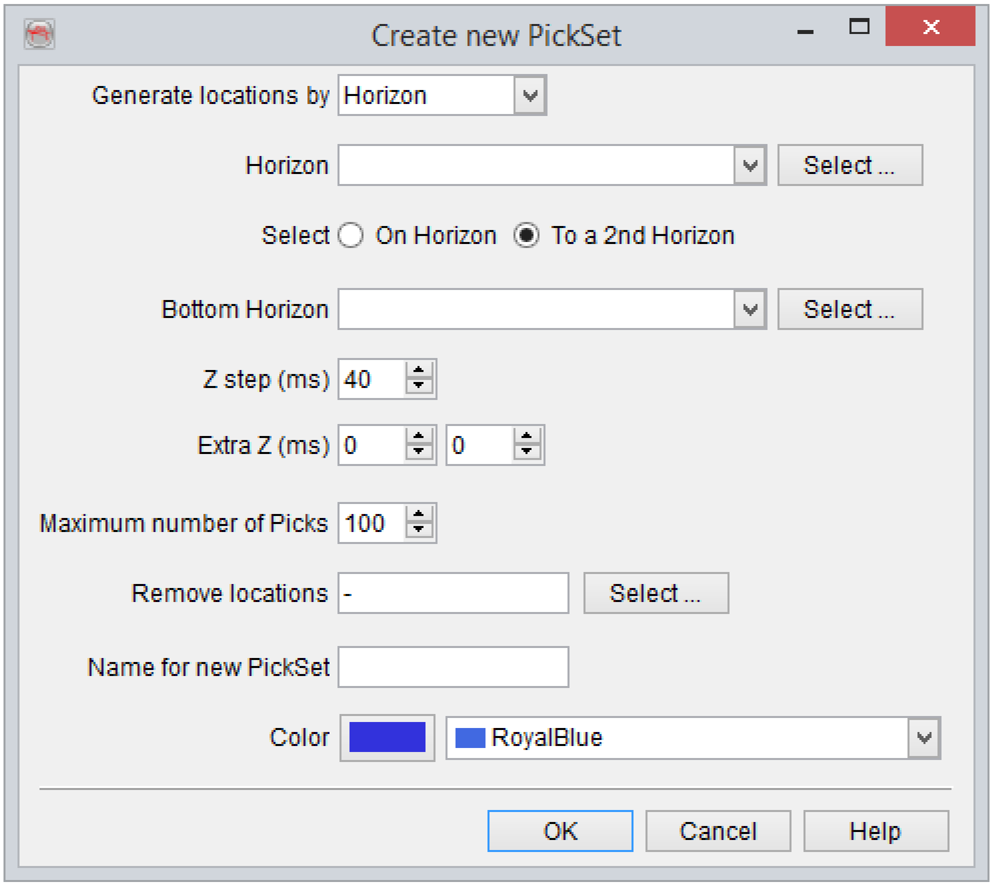
Maximum number of Picks: 100

Remove locations: -

Name for new PickSet:

Color: RoyalBlue

Extraction setting in volume for 3D UVQ *Extraction settings between horizons for 3D UVQ*



Generate locations by: Horizon

Horizon:

Select ○ On Horizon ● To a 2nd Horizon

Bottom Horizon:

Z step (ms): 40

Extra Z (ms): 0 0

Maximum number of Picks: 100

Remove locations: -

Name for new PickSet:

Color: RoyalBlue

Extraction setting along an horizon for 2D UVQ

Property Prediction (using Well Logs)

With this module, it is possible to relate any well log to seismic data. Select an input variable (from cubes and attributes) and, an output log variable (e.g. porosity or Vshale). The neural network will try to look for a relation between the two sets. Provided a working relationship is established, the user can then apply the trained neural network to a larger volume.

Please be aware that neural networks are good interpolators, but not good in extrapolating. If you train a neural network on a certain formation or interval, it is not recommended to apply it outside that formation or interval.

Since log data is being related directly to seismic data, it is essential that the well logs have a very good tie and are well aligned with the seismic data in the interval of interest. If this is not the case, results may easily become disappointing.

Input training data set: The training data set is the collection of input and target values that the neural network is trained on. Usually, the user will leave this option at *Extract now*. In case you stored an input training data set before, tick then *Retrieve stored* and select the input training data set with a standard file browser.

Select input/target attributes: The input attributes from the active attribute set or any of the stored cubes can be selected in the left screen. The stored cubes appear in square brackets [ ] at the bottom of the list. On the right, select the output log variable. All available logs from all wells are shown here. If the log of interest has different names for different wells, rename first these logs so that they all have exactly the same name. The log(s) can be renamed in *File - Manage - Wells*.

Target contains: Specify if the target contains ordinary well logs value or if a lithology log is available and can be used.

Wells to use: Select the wells on the right. From these wells, the *selected target* log is retrieved, if available.

Extract between: Select the markers to specify the interval the neural network should be trained on. Just as with the target logs, all available markers from the available wells are displayed. You can use the same marker twice, in combination with non-zero distance above/below. It is up to the user to make sure a marker with exactly the same name actually exists in the selected wells. To edit the markers and their names, see *File - Manage - Wells*.

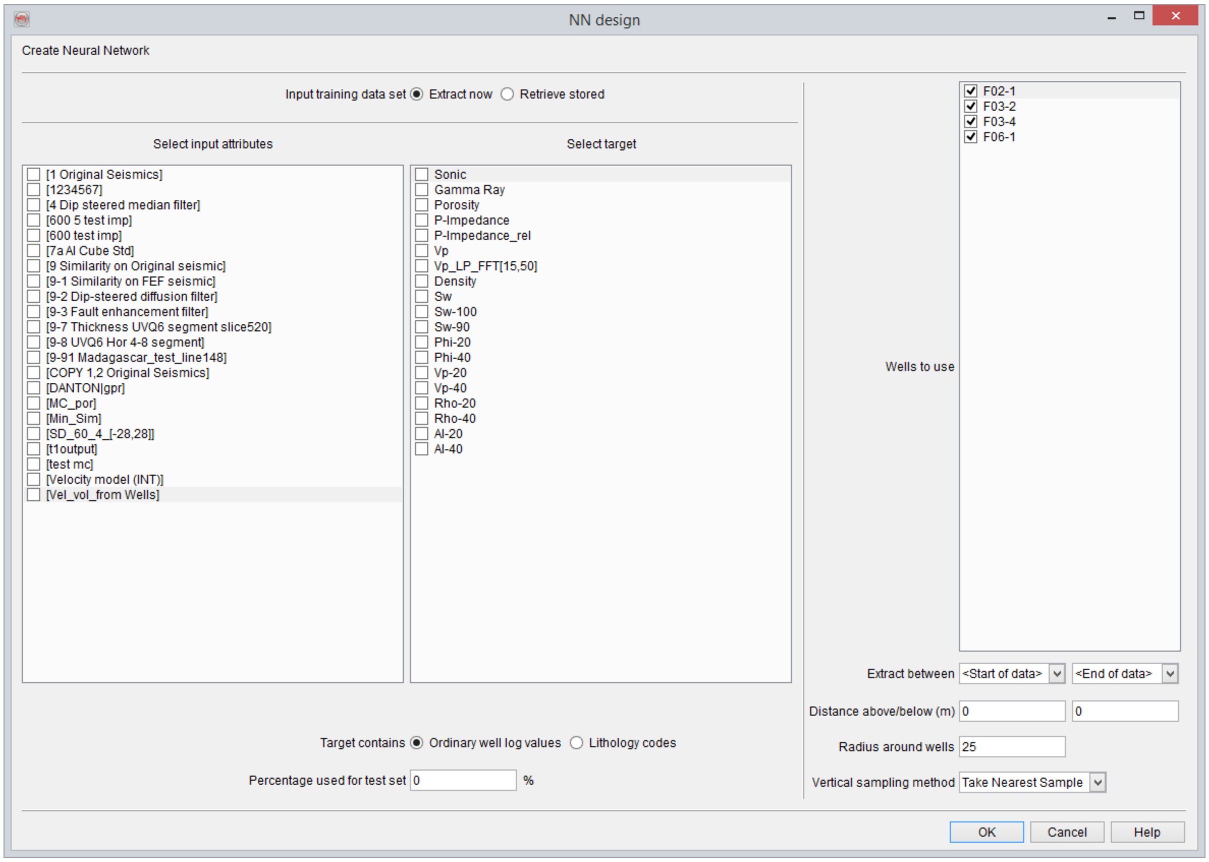
Distance above/below: Indicate the extra distance above and below the start and stop marker respectively that should be taken into account in the training. Negative values are possible; negative above the top marker means start below the top marker. Negative below the bottom marker means stop before the bottom marker.

Radius around wells: Indicates the radius around wells where the selected input attributes are calculated for each depth of extraction. All traces within the radius are selected. For example for nearest trace leave blank or put zero.

Vertical sampling method: Well data have a much higher (vertical) resolution than seismic cubes. This means several log values correspond to a single seismic sample. Averaging can prevent aliasing problems, although this is not necessarily a problem. Therefore, one can use the median or average of the well log values corresponding to the seismic sample location.

When predicting a binary variable (like sand/shale), or a lithology code, the most frequent filter will be necessary. *Nearest sample* selects only the nearest log value. This will be the best option in pre-filtered curves.

On pressing *OK*, the software starts collecting all necessary data. When all data is collected, the training starts and this can be monitored in the **NN training window**.



Input training data set ● Extract now ○ Retrieve stored

Select input attributes
☐ [1 Original Seismics]
☐ [1234567]
☐ [4 Dip steered median filter]
☐ [600 5 test imp]
☐ [600 test imp]
☐ [7a AI Cube Std]
☐ [9 Similarity on Original seismic]
☐ [9-1 Similarity on FEF seismic]
☐ [9-2 Dip-steered diffusion filter]
☐ [9-3 Fault enhancement filter]
☐ [9-7 Thickness UVQ6 segment slice520]
☐ [9-8 UVQ6 Hor 4-8 segment]
☐ [9-91 Madagascar_test_line148]
☐ [COPY 1,2 Original Seismics]
☐ [DANTON|gpr]
☐ [MC_por]
☐ [Min_Sim]
☐ [SD_60_4_[-28,28]]
☐ [t1output]
☐ [test mc]
☐ [Velocity model (INT)]
☐ [Vel_vol_from Wells]

Select target
☐ Sonic
☐ Gamma Ray
☐ Porosity
☐ P-Impedance
☐ P-Impedance_rel
☐ Vp
☐ Vp_LP_FFT[15,50]
☐ Density
☐ Sw
☐ Sw-100
☐ Sw-90
☐ Phi-20
☐ Phi-40
☐ Vp-20
☐ Vp-40
☐ Rho-20
☐ Rho-40
☐ AI-20
☐ AI-40

Wells to use
☑ F02-1
☑ F03-2
☑ F03-4
☑ F06-1

Target contains ● Ordinary well log values ○ Lithology codes

Percentage used for test set 0 %

Extract between <Start of data> <End of data>

Distance above/below (m) 0 0

Radius around wells 25

Vertical sampling method Take Nearest Sample

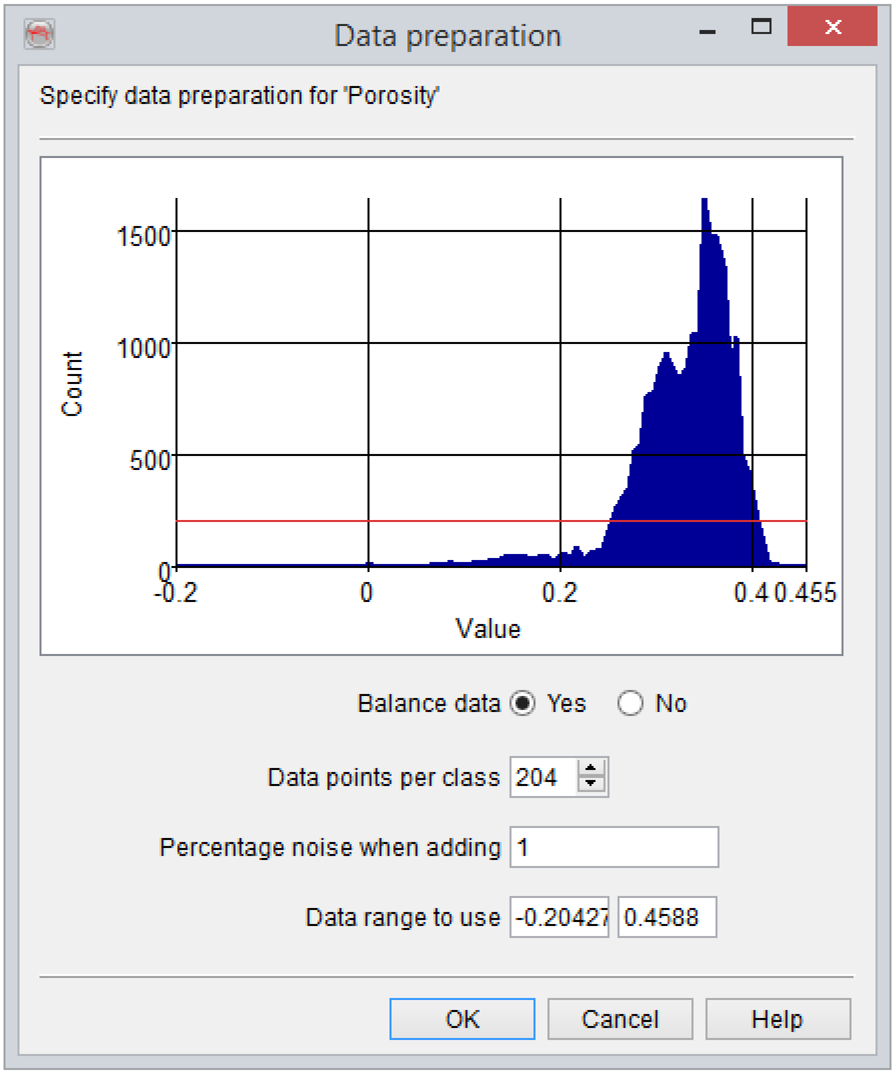
Balance Data

Balancing is a mandatory pre-processing step for neural network training. The distribution of input vectors is modified in order to get a flat distribution for the output quantity (target log), in the training data set. Balancing is done automatically when training from picksets or on a target log that is containing lithology codes. In both cases, the output values are discrete (integers), thus *Balancing* can be automated safely. An ordinary log will show continuous values between a minimum and maximum. Based on the histogram display the user may:

- Adjust the output level for the flat distribution using the *Data points per class* parameter.
- Train for a range of values smaller than the extracted minimum and maximum values by specifying another *Data range to use*.

The over-represented classes will be decimated to the *data point per class* parameter.
The under-represented classes will be duplicated to the *data point per class* parameter, with a small change of the target value for each duplicated vector. This change is controlled by the parameter *Percentage noise when adding*, but the default value will be appropriate for most situations.

The binning is automatically performed to compute the number of classes based on minimum, maximum and number of points. At least 10 classes will be used, with a maximum of 100 classes. 20 and 50 classes may be used to reach an optimal of 25 vectors per class. This optimal is considered as the best compromise between statistical representation and training speed.



Data preparation

Specify data preparation for 'Porosity'

Balance data ● Yes ○ No

Data points per class 204

Percentage noise when adding 1

Data range to use -0.20427 0.4588

NN Lithology Codes

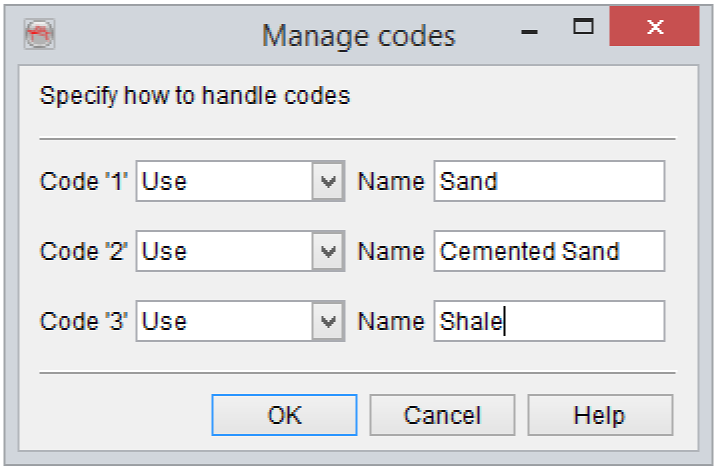
The neural network may be used to train for lithologies. For such a prediction the target log must contain lithology codes in form of integers. Multiple outputs will be available, on the contrary to training from ordinary log values:

- A *classification* output providing the integer corresponding to the most likely lithology at each sample.
- For each lithology code, the *probability* for a sample to belong to this lithology.
- A *confidence* output that is the difference between the probability of the most likely

lithology and the probability of the second most likely probability.

A maximum of 20 lithologies may be present in the input lithology logs. The upscaling (vertical sampling) method will be set to *Most Frequent* when using this option "Target contains: Lithology codes" such that the integers will remain integers after the up scaling. If non integer values are found in the input logs, they will be round-up to the nearest integer.

All lithology codes present in the logs may not be used for training. For instance the input log may contain four lithology codes for respectively sand, cemented sand, shale and salt. The first step is to assign a name to each code:



Manage codes

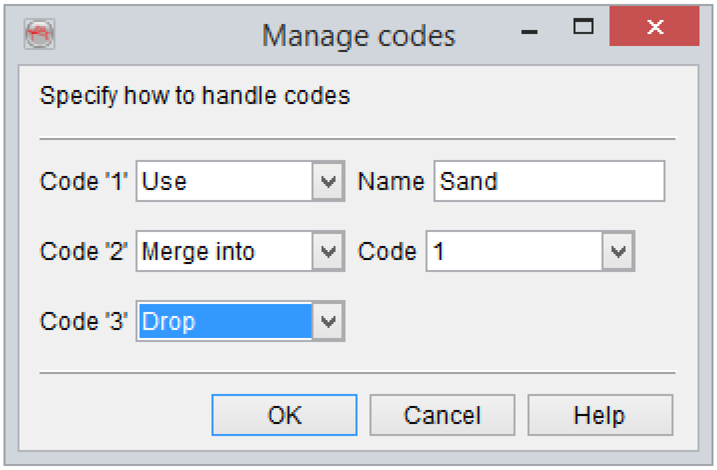
Specify how to handle codes

Code '1' Use Name Sand

Code '2' Use Name Cemented Sand

Code '3' Use Name Shale

The second step may be to group (*Merge into*) several similar lithologies and/or to discard lithologies that will not be used to training (*Drop*):



Manage codes

Specify how to handle codes

Code '1' Use Name Sand

Code '2' Merge into Code 1

Code '3' Drop

Training begins when pressing "OK" on this window. The training window is similar to the training window for pattern recognition (**Picksets**)

Training and Application

This window pops up as soon as OpendTect has created or retrieved the train and test sets. In order to create a training set, the software must compute all selected attributes at all picked locations. This may take some time. The user will be notified when the data is collected. Acknowledging the notification will automatically start the training phase.

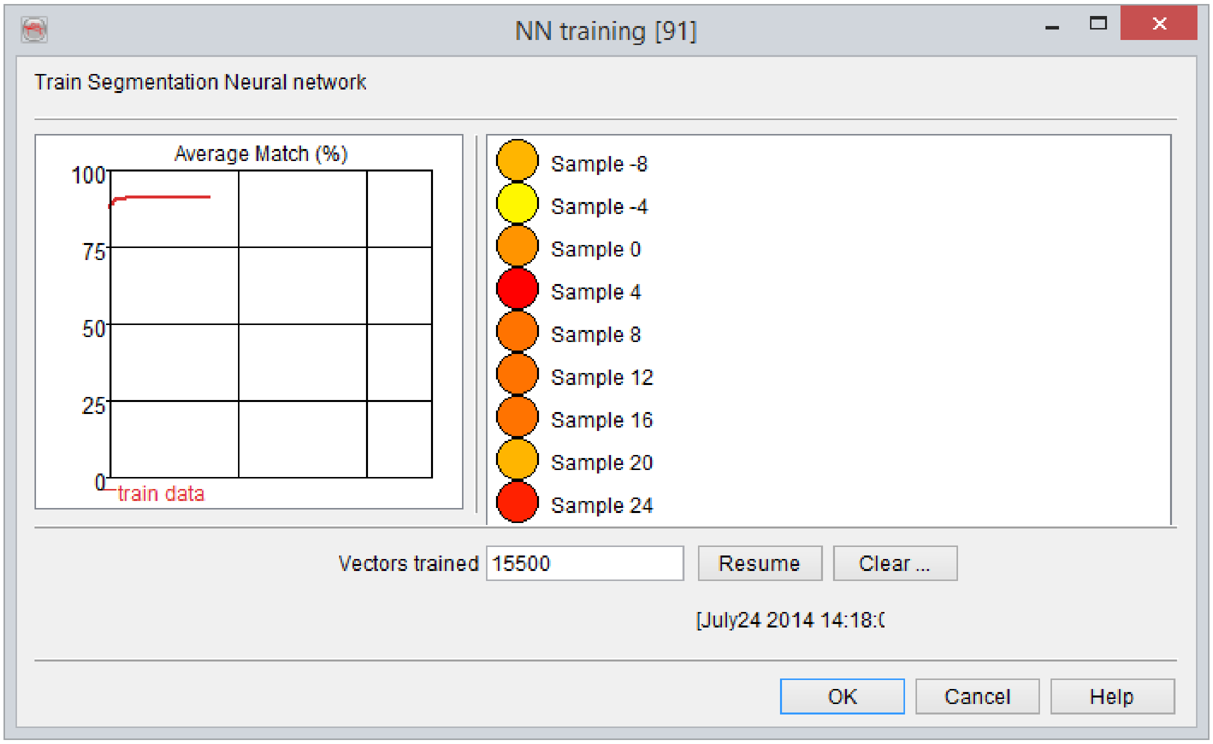
Training can be stopped and restarted with the *Pause (Resume)* button. *Clear* randomizes the weights of the current network. This implies that all training results are lost. The Clear option can be used when over fitting (see below) has occurred and the user wishes to restart training from scratch. The randomized network is then re-trained and the network is stopped before over fitting occurs. This can be done manually (pressing Pause) or by specifying a number in the *number of training vectors* field.

Unsupervised Training

In *Unsupervised training*, the network performance is tracked in a graph that shows the average match (confidence) of clustered input. Typically, the average match increases in a step-function. Each step indicates that the network has found a new cluster. Training can be stopped as soon as the average match has reached a stable situation. Usually this will be around 90%.

The colors of the input nodes in an unsupervised network will also change during training. In unsupervised mode, these colors do not indicate that one attribute is more important than another. All attributes in a clustering experiment are equally important.

Optionally, the neural network can be stored immediately on pressing OK. To do this, enter a neural network name in the appropriate field at the bottom of the NN training window.

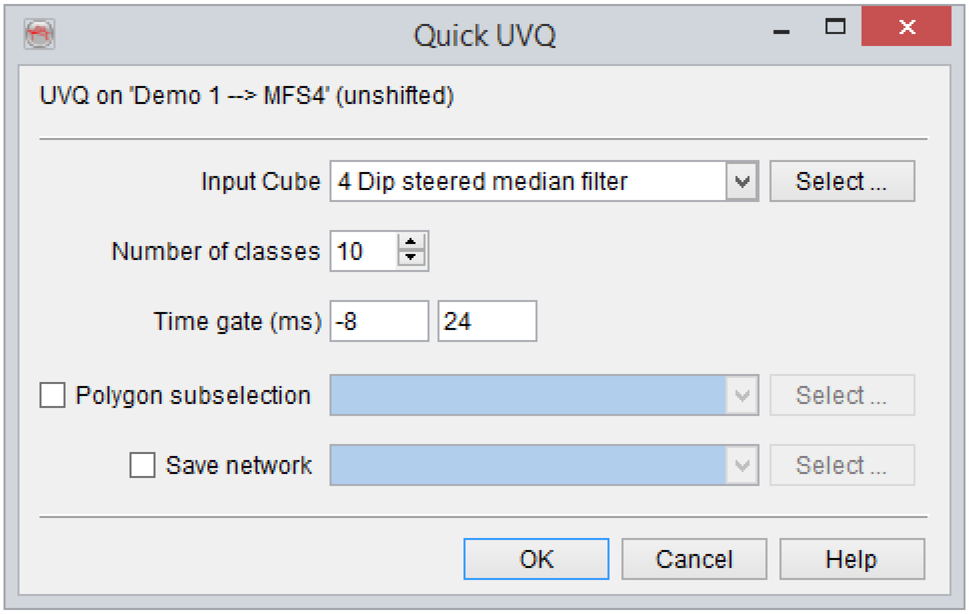


Vectors trained 15500

[July24 2014 14:18:(

Quick UVQ

The *Quick UVQ* horizon option can be used to quickly segment the seismic waveform along the interpreted horizon. The attributes and data selection are made automatically, contrarily to the standard NN training mode that request the user to define attribute and training locations (picks) before going into the **NN training module**.



UVQ on 'Demo 1 --> MFS4' (unshifted)

Input Cube: 4 Dip steered median filter

Number of classes: 10

Time gate (ms): -8 24

☐ Polygon subselection

☐ Save network

The sole window requests the user to select the input seismic data (2D or 3D according to the type of interpretation). Around 2000 input traces are randomly chosen to train the neural network. The waveform in a given time window is the main parameter to input, together with the output number of classes. It is highly recommended to save the output network.

The time window should match the length of the target of interest, and be extended on either sides by a few samples.
Example: You have a top reservoir horizon and the target is 50m = 32ms thick. An appropriate time window would be [-12, 44] ms along the horizon.
The number of class is hard to determine in advance. Too many classes will give redundant class centres, but too few classes will not match the seismic.

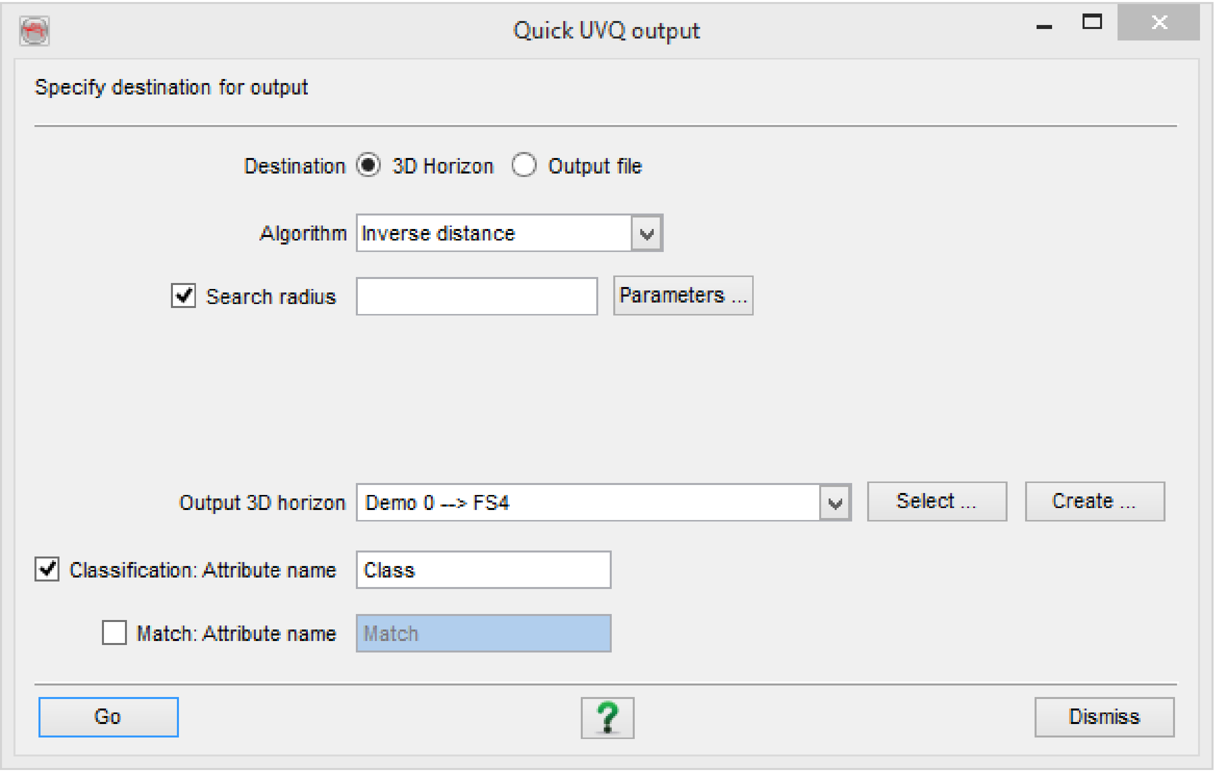
The 'OK' button will start the unsupervised Neural Network training. The training that results above 90% match could then be accepted to create output maps. The 'OK' button in the Neural Network Training window will automatically start processing the two outputs (Segmentation and match grids) on the 3D horizon. The processed grids are only saved to the database using the right mouse click on the processed attributes (Class, Match) of the horizon.

After training an additional Neural Network Info window will pop-up. (see above). This dialog gives a detailed report about the Neural Network training. The Display button in the Neural Network info dialog is used to pop-up a class center display in a 2D viewer (see below). The classes can be color-coded using the

Color Table, and the color table can be displayed in the 2D viewer by clicking on the 'show classification' button.

2D QUICK UVQs

The similar workflow is also available for a mapped 2D seismic horizon. Similar settings are required and the processing steps are also the same (as explained previously). After the Neural Network training is finished, the system will prompt to grid and save the classification and the match grids to a 3D horizon. Select or create a 3D horizon to process the classification and match maps.



Specify destination for output

Destination ● 3D Horizon ○ Output file

Algorithm Inverse distance

☑ Search radius

Output 3D horizon Demo 0 --> FS4

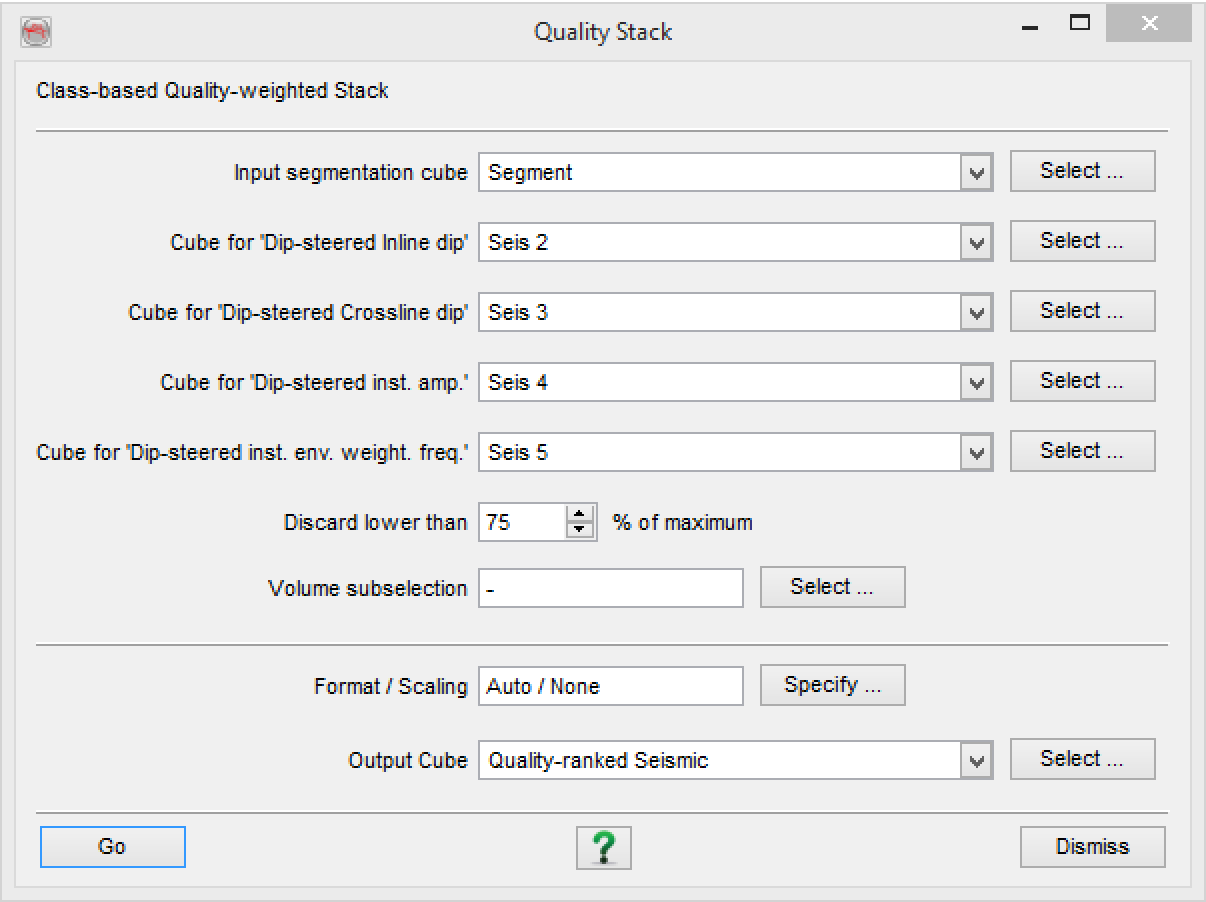
☑ Classification: Attribute name Class

☐ Match: Attribute name Match

Quality-based UVQ Stacking

It is possible to use the output of an unsupervised segmentation for the stacking of seismic data. This functionality allows stacking of multiple cubes based on class number. The input nodes must be a measure of quality for each of these cubes.

This function is only available if the environment variable OD_DGB_QUALITY_STACKING is set to "Yes". The quality stack button appears in the "NN info" window of the loaded neural network. It will not appear if a single volume is used as input to all attributes of the neural network, like for a UVQ classification.



Class-based Quality-weighted Stack

Input segmentation cube: Segment
Cube for 'Dip-steered Inline dip': Seis 2
Cube for 'Dip-steered Crossline dip': Seis 3
Cube for 'Dip-steered inst. amp.': Seis 4
Cube for 'Dip-steered inst. env. weight. freq.': Seis 5
Discard lower than: 75 % of maximum
Volume subselection: -
Format / Scaling: Auto / None
Output Cube: Quality-ranked Seismic

The segment cube must have been previously processed and saved on disk. It represents the first input. The other inputs are the volumes to be stacked, on for each attribute (so the input of the attributes used in the neural network). Once again only stored volumes can be selected.

The output will be a weighted stacking of the input volumes. The weights are represented by the class centre values (one for each attribute), and vary according to the best fitting class centre as represented in the segment volume. Optionaly low weight (relative to the maximum weight in a given class) may discard the corresponding volumes locally. Use a low value for the smoothest output, and a high value for the best discrimination.

Supervised Training from Picksets

In *Supervised mode*, the network's performance is tracked during training in two graphs: *Normalized RMS* and *% Misclassification*:

The *Normalized RMS* error curves (see network training picture below) indicate the overall error on the train and test sets, in red and blue respectively on a scale from 0 (no error) to 1 (maximum error). Both curves should go down during training. When the test curve goes up again the network is over fitting. Training should be stopped when (preferably before) this happens. Typically a RMS value in the 0.8 range is considered reasonable, between 0.8 and 0.6 is good, between 0.6 and 0.4 is excellent and below 0.4 is perfect. The normalized error is calculated as follows:

$$rms = \sqrt{\frac{1}{n}\sum_{i=1}^{n}(t_i - e_i)^2}$$

and

$$normrms = rms \Big/ \sqrt{\frac{1}{n}\sum_{i=1}^{n}(t_i - mean)^2}$$

where

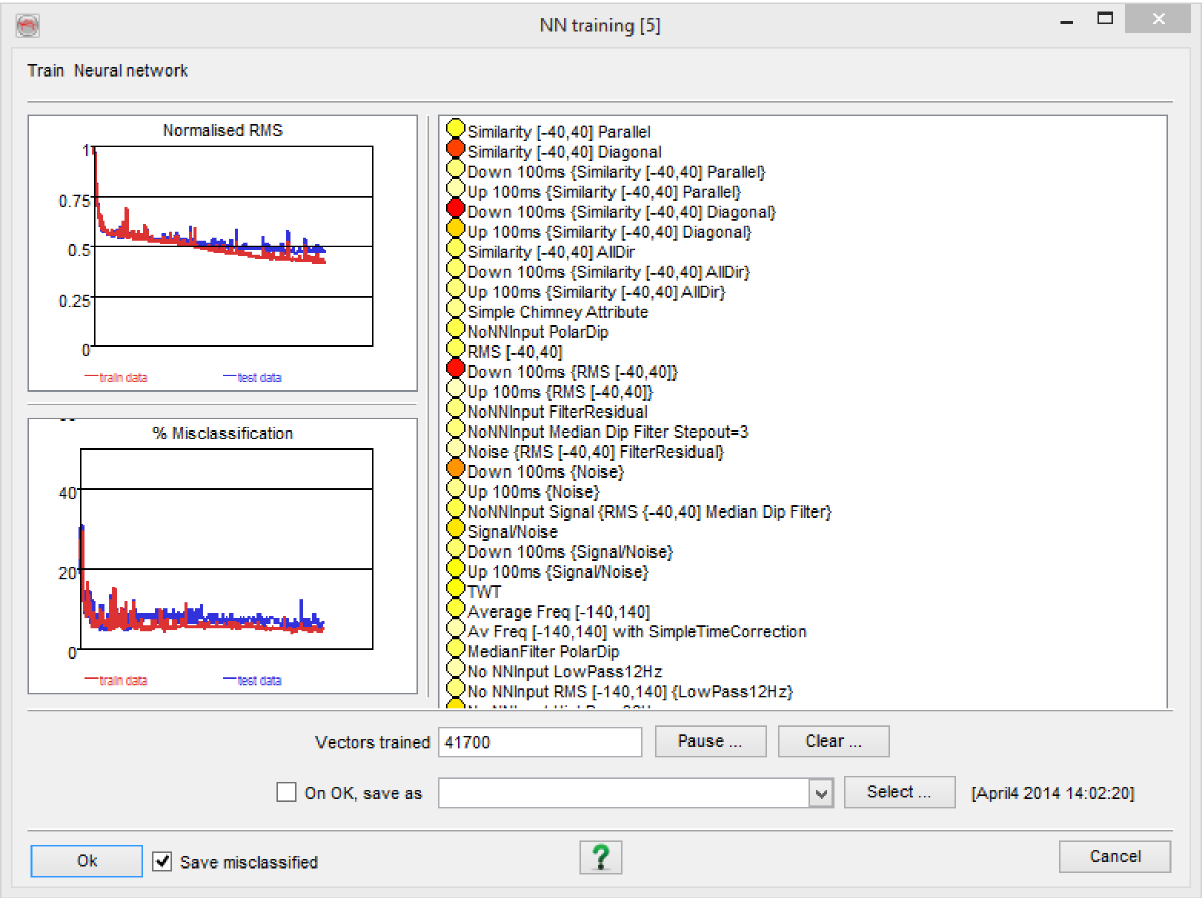
$$mean = \frac{1}{n}\sum_{i=1}^{n} t_i$$

The *percentage misclassification* shown in the lower left corner is a much easier quality control parameter to interpret. It simply shows how the percentage of the training and test set that is classified in the wrong class.

On the right-hand side of the window a graphical representation of the input attributes is shown. The circle in front of the attribute name changes color during training. The colors reflect the weights attached to each input node and are therefore indicative for the relevant importance of each attribute for the classification task at hand. Colors range from red (high weight means high importance) via yellow to red (relative small weights, less important). This feature is very useful when you wish to design small networks to increase processing speed.

Optionally, the neural network can be stored immediately by pressing the OK button. First, enter a neural network name in the appropriate field at the bottom of the NN training window.

The *Save misclassified* toggle allows saving the misclassified picks in a new Pickset. This Pickset is automatically loaded in OpendTect again. The Pickset can be indicative of picking errors. It is not recommended to bluntly remove the misclassified picks from a Pickset, since good picks, although misclassified during training, still help neural network training.



Vectors trained 41700

☐ On OK, save as

[April4 2014 14:02:20]

☑ Save misclassified

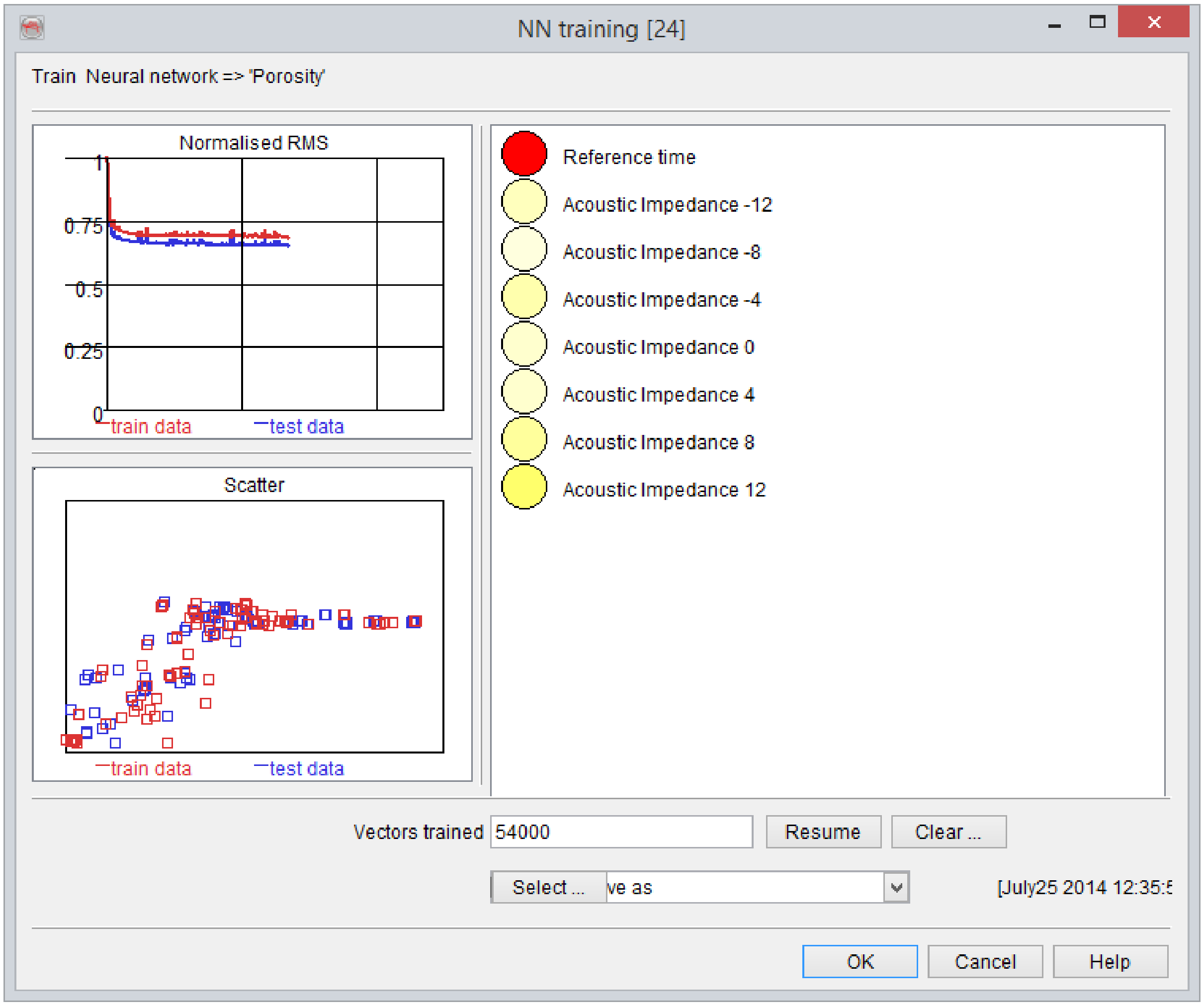
Supervised Training from Well Data

The Supervised training window from well data is very similar to the training window from a Pickset (see below). The only difference is the display of a *scatter plot* instead of a *% Misclassification plot*.

A scatter plot shows the actual target data on the horizontal axis and the predicted target data by the neural network, as it is at that moment, on the vertical axis. Not all nodes are plotted.

Only a random selection of the used train and test data is shown. Ideally, after sufficient training, all data points should be on the diagonal. That would mean that the trained neural network predicted all examples correctly. However, this will rarely be the case. In most cases, the data will cluster along the diagonal. The narrower this cloud, the better the neural network is trained.

Overtraining occurs when the Normalized RMS of the test data increases, while the Normalized RMS of the train set decreases. This usually also means that the cloud of train nodes becomes narrower, while the cloud of test nodes becomes wider again.



NN training [24]

Train Neural network => 'Porosity'

● Reference time
● Acoustic Impedance -12
● Acoustic Impedance -8
● Acoustic Impedance -4
● Acoustic Impedance 0
● Acoustic Impedance 4
● Acoustic Impedance 8
● Acoustic Impedance 12

Vectors trained 54000

[July25 2014 12:35:5

Application

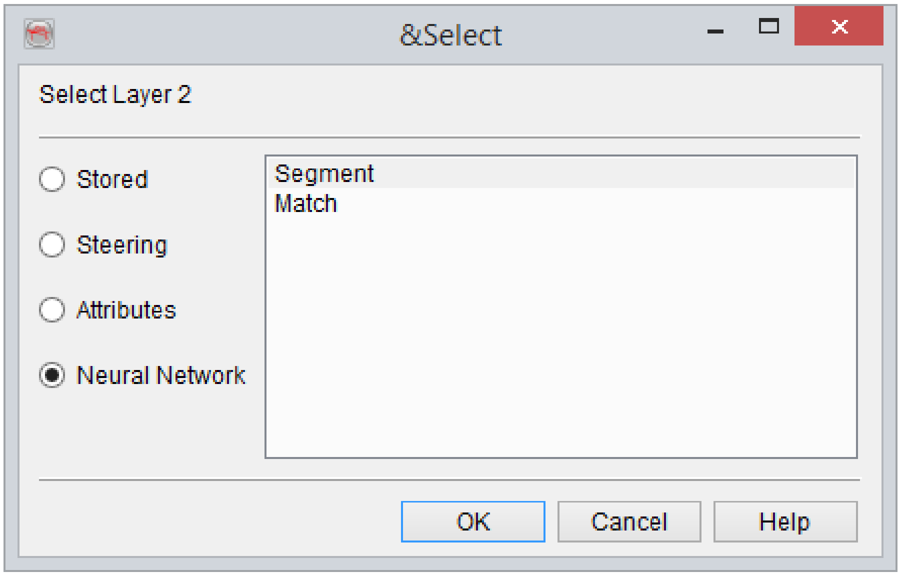
Neural networks should be applied like standard attributes:

On-the-fly application

Load a slice in the scene, use the option "Add --> attribute". Select the "Neural Network" list of attributes.

The type of output depends on the type of neural network:

1. Pattern recognition, unsupervised mode:



&Select

Select Layer 2

○ Stored

○ Steering

○ Attributes

● Neural Network

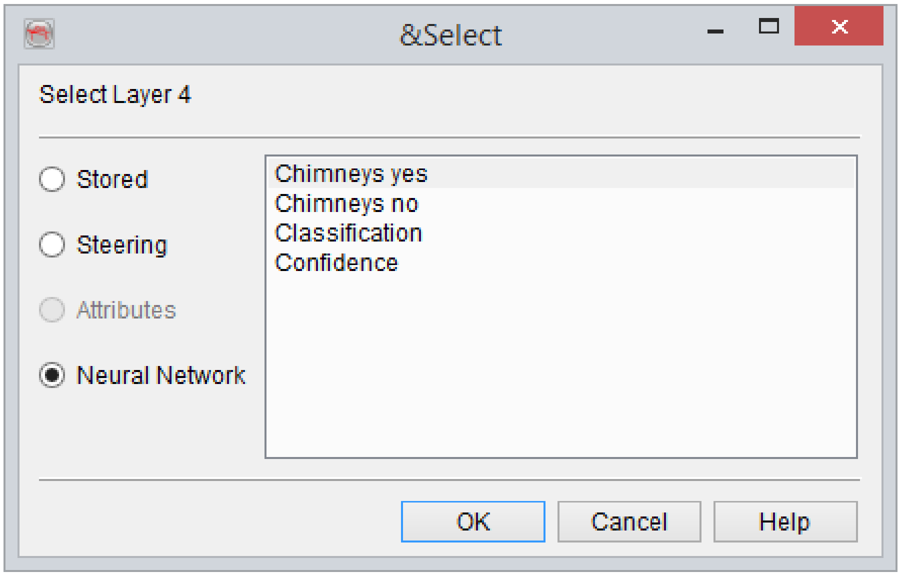
Segment

Match

o **Segment:** Index of the most likely class center (as seen in the **Neural Network Information**)

o **Match:** Measure of similitude between the most likely class center and the actual attributes. Between 0 and 1 (identical).

2. Pattern recognition, supervised mode:



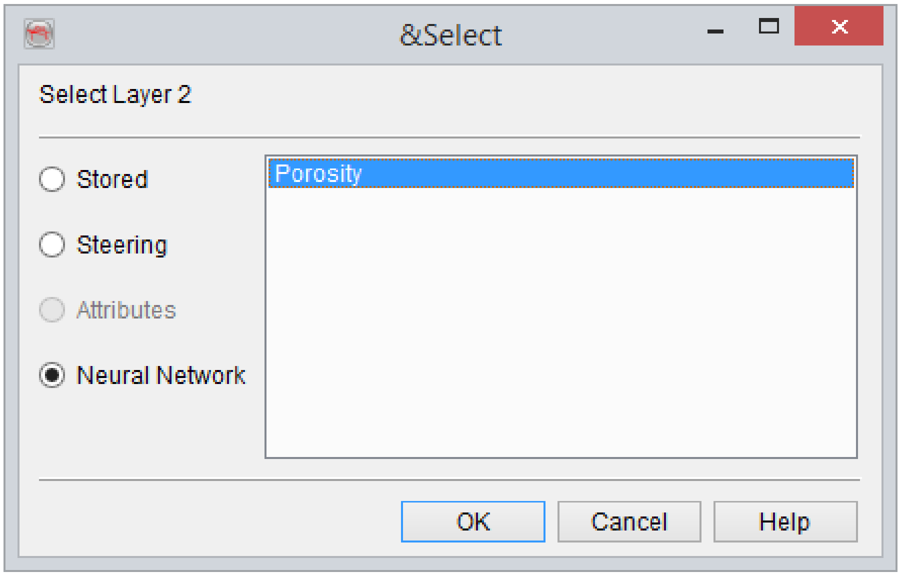
o **[Pattern 1]:** Neural network probability that a given sample matches the interpreted "Pattern 1": Returns amplitudes around 1 if the sample is similar to the picks of the corresponding first pickset. Returns values around 0 otherwise.

o **[Pattern 2]:** Neural network probability that a given sample matches the interpreted "Pattern 2": Returns amplitudes around 1 if the sample is similar to the picks of the corresponding second pickset. Returns values around 0 otherwise.

o **Classification:** Returns the index of the most likely pattern in the order seen above.

o **Confidence:** Returns the difference between the largest and second largest NN probabilities in the set (You can train with more than two picksets)

3. Property prediction



o **[Output log]:** Returns the predicted value for the target log. The uncertainty on the predicted value is given in the **training report**.

Batch processing

The batch processing on the elements described above should be started from the menu Processing --> Create Seismic Output --> Attribute (2D/3D).

Note that you need to launch one batch process for each element under the list "Neural networks".

Velocity Model Building

Table of Contents

- Introduction
- Vertical Velocity Analysis
- Horizon-based velocity update
- Input-Output
- Velocity display
- Velocity correction
- VMB specific gridding step: gridding of velocity picks
- VMB specific gridding step: Surface-limited filler

Introduction

This plugin will work only on a 3D survey

The basic concept of *velocity model building* is to use the travel time of the acoustic waves to image the subsurface. Unlike the amplitudes, which are linked to acoustic properties (density and Lame parameters), the travel time and ray geometry between source and receiver are a function of the velocity (and anisotropy) field(s). This is a consequence of the ray geometry obeying Fermat's principle of least time.

The velocity model building requires migrated *Common Image Gathers (CIG)*, as input. CIG are pre-stack seismic gathers migrated to depth, thus NMO corrected. The velocity field used for this migration will also be a necessary input for semblance calculation and thus RMO picking, and is linked to OpendTect CIG data store. The corresponding post-stack volume might also be appended to the pair CIG/Velocity model to create a link between the post stack horizons and the pre-stack events and to perform horizon-based velocity analysis.

Picking velocities is possible since the velocity model determines the travel times and ray geometry. A correct travel time and ray geometry will allow the image gather (migrated seismic in depth) to be stacked constructively. The constructiveness of the gathers is measured via a semblance function which outputs a semblance value for each possible RMS velocity. The picking of the high semblances enables, therefore, the interpreter to retrieve the correct velocity function. The process is made available on a semi-automated base in two modes:

(1) **Vertical velocity update**

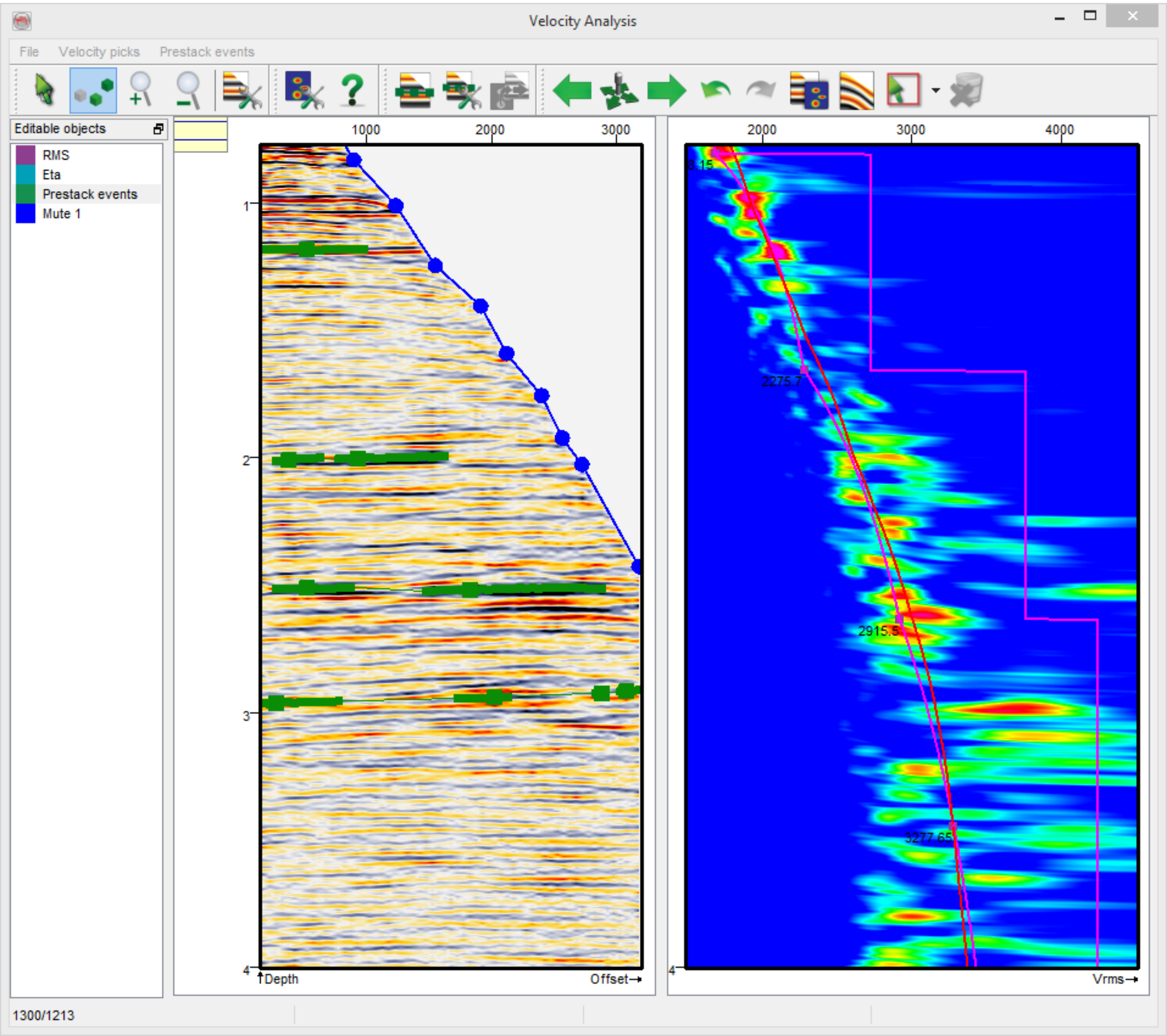
(2) **horizon-based velocity update**.

The vertical velocity update presents the semblance panel for a single trace. The picked RMO's generate a new velocity function with depth, which can be applied on-the-fly on the corresponding common image gather (CIG). Therefore, the flatness of the CIG can be very quickly appreciated during the picking phase. The maximum of semblance, therefore the change in the velocity function, is tracked between the picked positions and the next position. Therefore, the picking is called semi-automated. The picking may also be done along a post-stack horizon instead of on a trace-by-trace basis. A standard workflow would be to track horizons on a migrated volume, pick the RMO's along each horizon performing the horizon-based velocity update, and QC them in the vertical velocity update window.

Vertical Velocity Analysis

From the VMB toolbar, launch the vertical velocity update window by pressing the corresponding icon .

The window is composed by a Common Image Gather from a single trace on the left, and the corresponding semblance plot on the right:



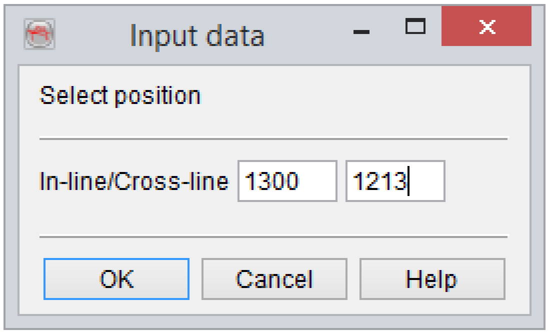
The display is annotated by separate display settings for both 2D viewers. The 2D viewer settings are similar to the standard 2D viewer, except that the semblance cannot be displayed using wiggles. Furthermore the semblance panel features one or two additional curves: The *migration velocity* (always shown) and the *picked velocity*, based on the velocity picks and migration velocity. Thus this second curve is shown once a velocity pick is made. If horizons are loaded in the main scene they will be shown as well in this vertical analysis window as horizontal coloured segments.

The velocity picks, mute definitions and tomography events are editable objects. As such they can be edited using the same tools as for the picksets and polygons: *left click to add a new, hold left click to drag and drop an existing pick, ctrl+left click to remove*. In all cases you must be in edit mode and make sure that the object you would like to edit is highlighted/selected in the list of editable objects to apply those actions. Selection and removal tools are available to quickly remove a large number of pick on either the CIG (for mute and pre-stack events) or the semblance gather (for RMOs). You will first need to make a square or polygonal selection before to press the trash icon , that will remove the selected picks. Before removing a large number of picks you might want to use the undo and/or redo buttons.

Back in the view mode you are able to zoom in and out on the panels, that always remain synchronized.

When moving the cursor over the semblance panel you will see an overview of the RMO correction that is linked to your cursor position.

In the two viewer you can jump to any position using this toolbar icon , that launches a CDP selection window:



Select position

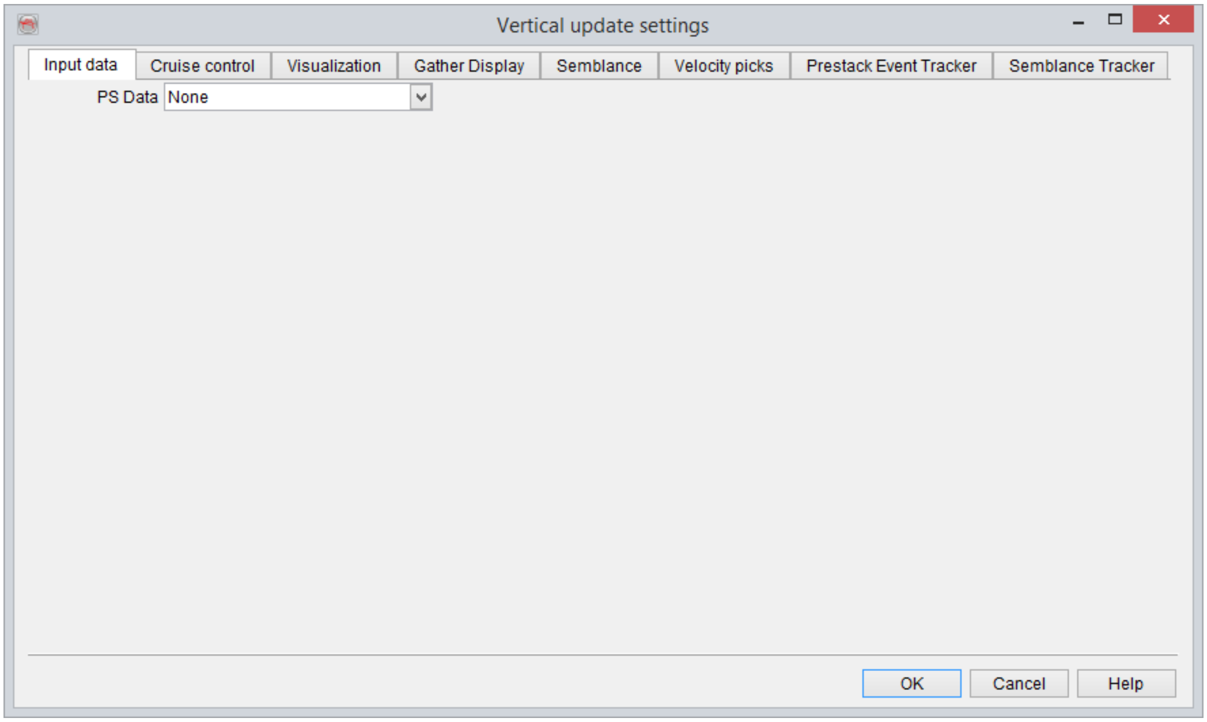
In-line/Cross-line 1300 1213

It is highly recommended to browse at start-up through the settings before picking a mute, RMO or tomography events. Those settings will be presented here below:

Settings: (available via *File--> Settings...*)

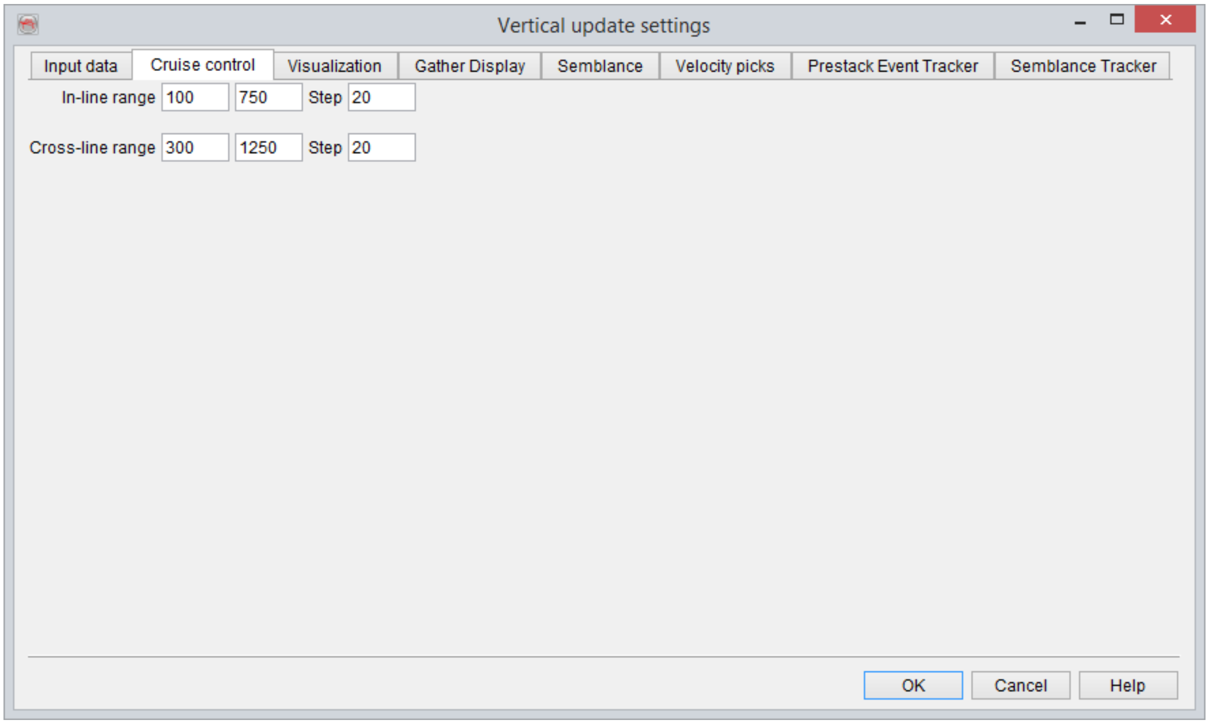
- **Input data:** You are requested to provide the pre-stack seismic data at the first launch, and specify the corresponding velocity model. The association will be stored and remembered for the next time.The prestack data, velocity, and post-stack seismic data can be in time or depth. As mentioned in the base documentation the velocity type must be set between Interval velocities (time or depth) and RMS velocity (time only). In the time domain the migrated gather can be NMO corrected or not. This velocity correction can be applied using the corresponding pre-processing step.

The CIG display settings are directly available using the icon .



PS Data None

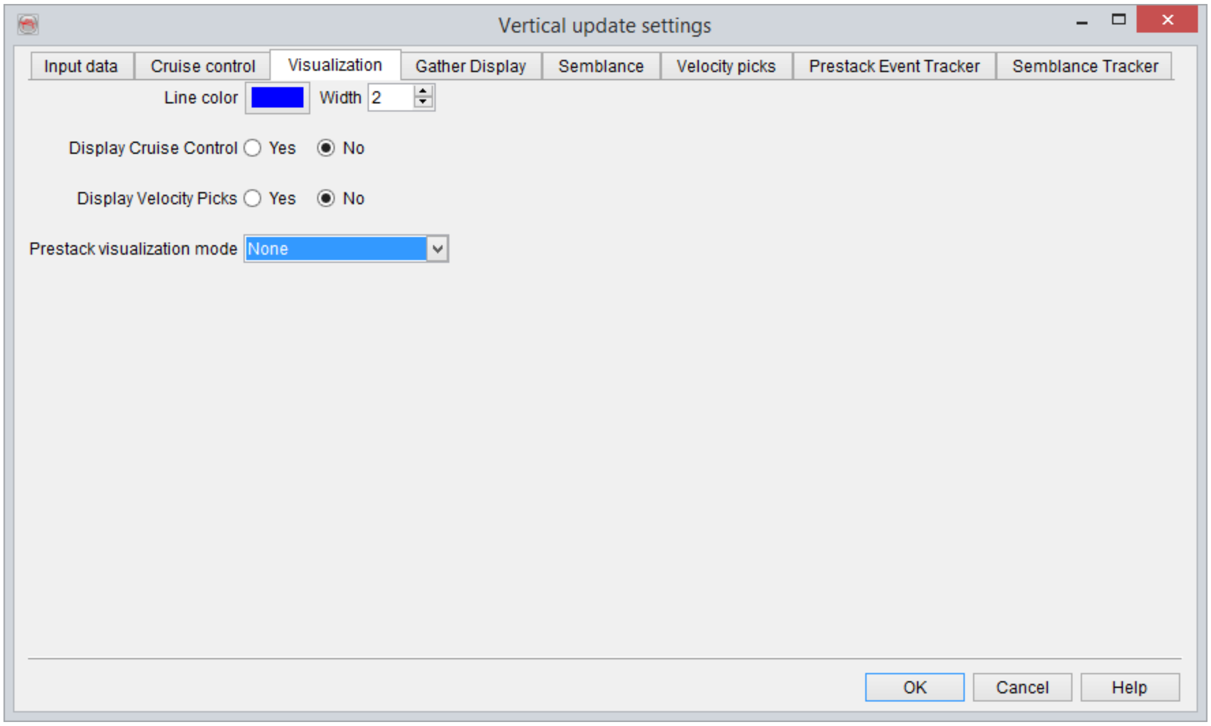
- **Cruise control:** Shortcuts are available during picking to go from one position to another. The cruise control settings define the positions that will be used for the quick browsing to the next ➡ or previous ⬅ CIG. Nevertheless the other positions remain accessible for the analysis via the pick icon in the main 3D scene and set position icons . When picking your location in the main OpendTect scene you need to be in edit mode in main scene and to pick on an inline or crossline.



In-line range 100 750 Step 20

Cross-line range 300 1250 Step 20

- **Visualization:** The visualization options enable you to display or hide the current analysis positions, all the positions defined from the cruise control settings, and the positions where RMO correction are picked. Those positions will be shown as vertical lines in the main OpendTect 3D scene. Optionally the cruise control positions and velocity picks may only be displayed at section, i.e. on the selected inlines and/or crosslines loaded in the scene. The current position will always remain visible. The last option allows the 3D visualization of the picked pre-stack events in the 3D scene



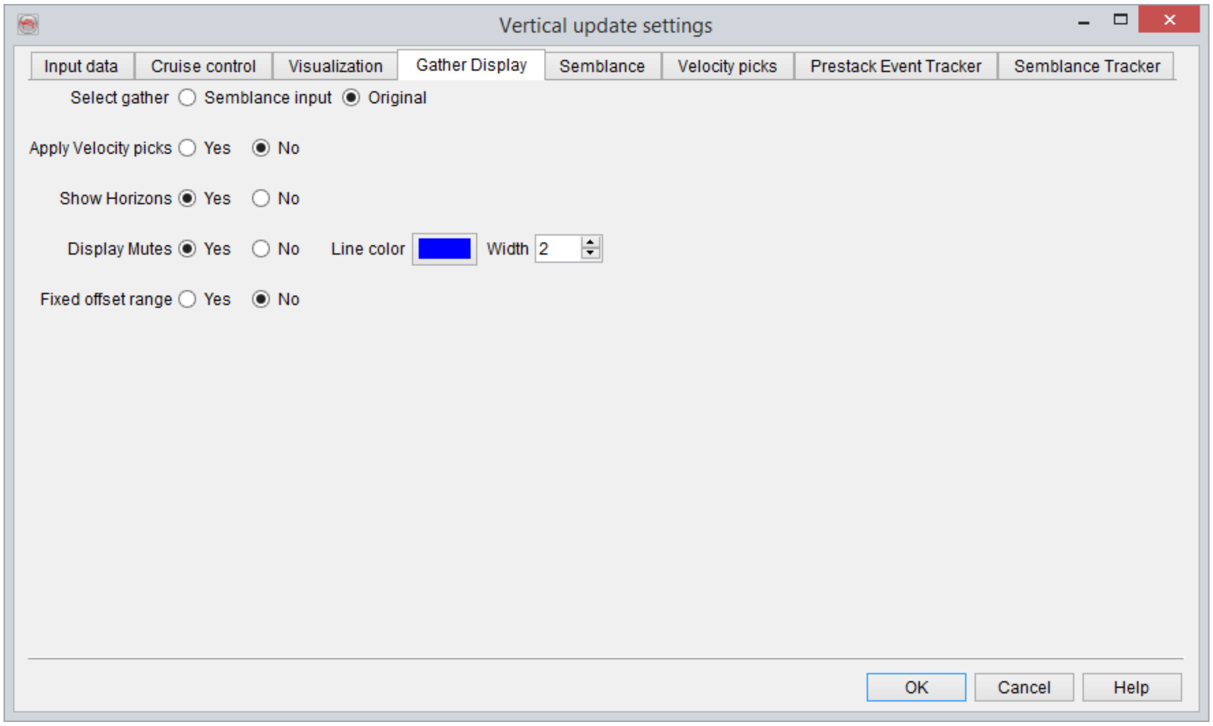
Line color 🟦 Width 2

Display Cruise Control ○ Yes ● No

Display Velocity Picks ○ Yes ● No

Prestack visualization mode None

- **Gather Display:** Those settings will affect the CIG display. Two toggles are available: The gather may be displayed as existing on disk (*original*) or after the semblance pre-processing (*semblance* input, see semblance settings tab). The gather may be *uncorrected* (no application of RMO picks) or *RMO corrected* if at least one pick is available in the survey. The mute and horizon annotations may be switched on/off. Please note that the mute display is toggled off if the RMO correction is applied, and that the horizons will only be shown in the nearest offsets.



Select gather ○ Semblance input ● Original

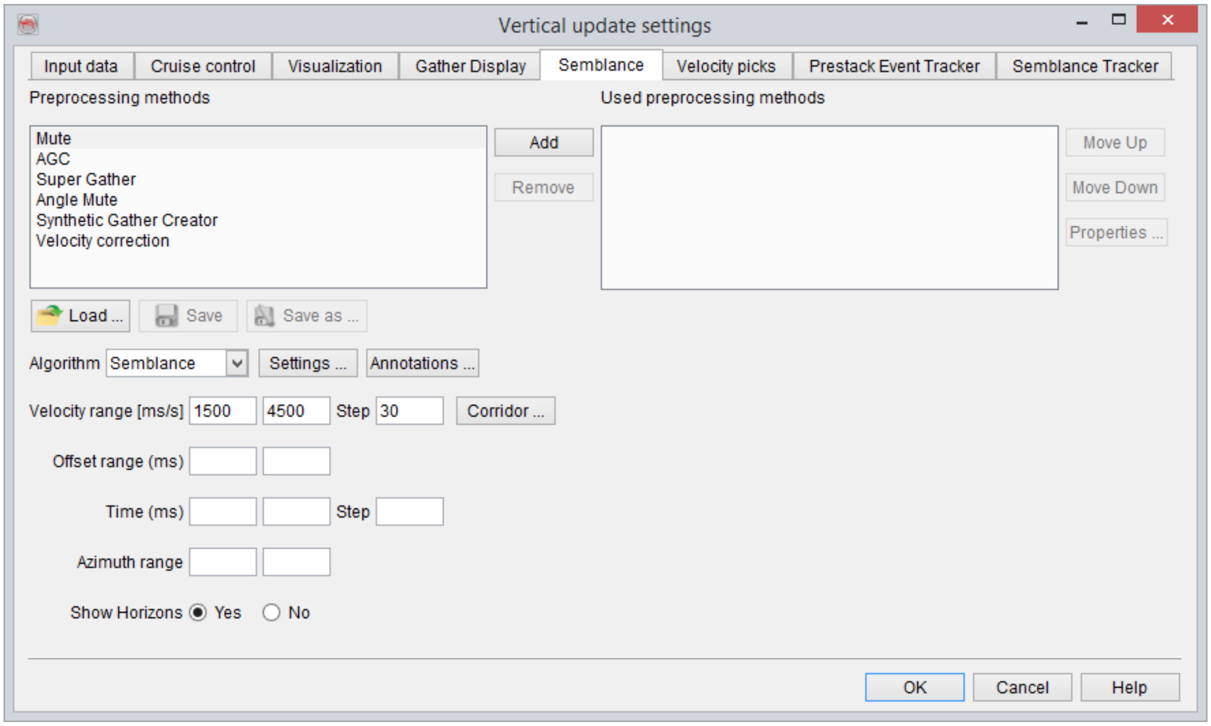
Apply Velocity picks ○ Yes ● No

Show Horizons ● Yes ○ No

Display Mutes ● Yes ○ No Line color 🟦 Width 2

Fixed offset range ○ Yes ● No

- **Semblance:** This setting tab is used to set the reference offset, RMO and velocity ranges, toggle on/off the horizon display, but more important it is meant to set and list the pre-processing methods applied to the gathers on disk before the semblance calculation: Mute, Automatic Gain Control and Vertical Stack are available. Please refer to the base documentation for a description of these pre-processing methods. Empty mutes might also be created from this tab by adding a mute step without name. When one or more pre-processing methods are used you can switch the seismic display between the pre-stack data as existing on disk or after pre-processing , i.e. the input to the semblance computation. The other semblance display settings are directly available using the icon .



Preprocessing methods

- Mute
- AGC
- Super Gather
- Angle Mute
- Synthetic Gather Creator
- Velocity correction

Used preprocessing methods

Algorithm: Semblance

Velocity range [ms/s]: 1500 4500 Step 30

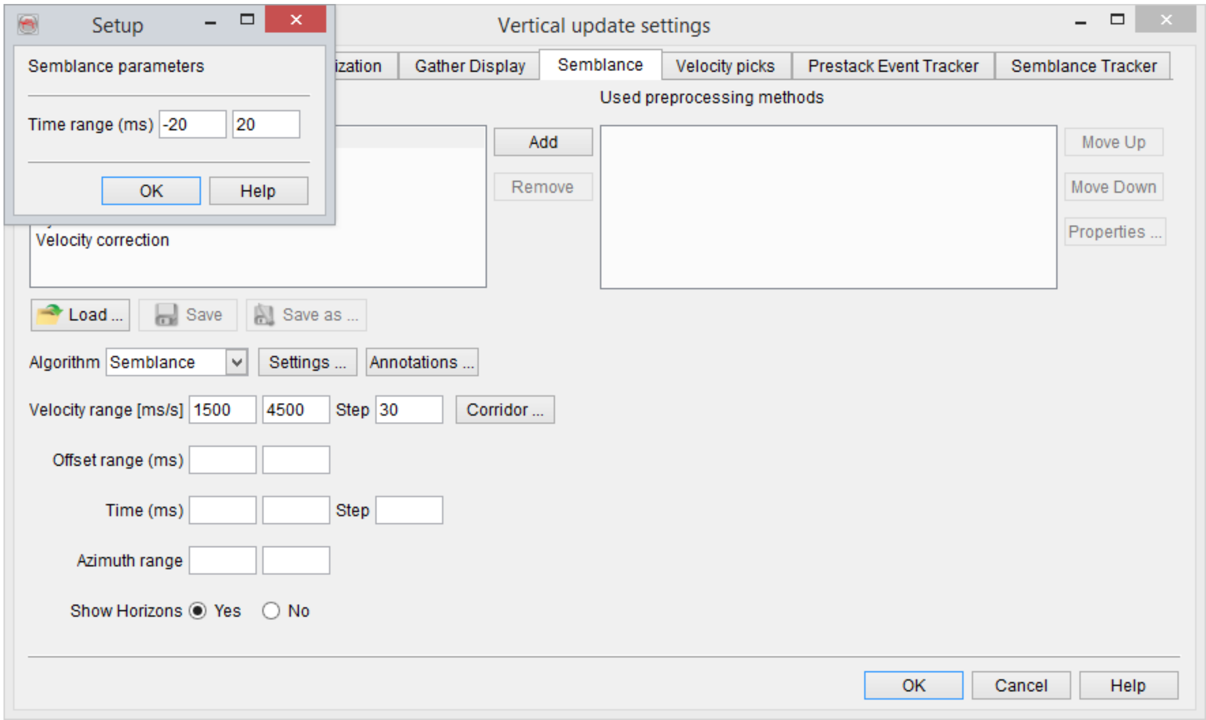
Offset range (ms):

Time (ms): Step

Azimuth range:

Show Horizons ● Yes ○ No

The following settings define the length of the time/depth gate used for the computation of the semblance:



Semblance parameters

Time range (ms): -20 20

Used preprocessing methods

Velocity correction

Algorithm: Semblance

Velocity range [ms/s]: 1500 4500 Step 30

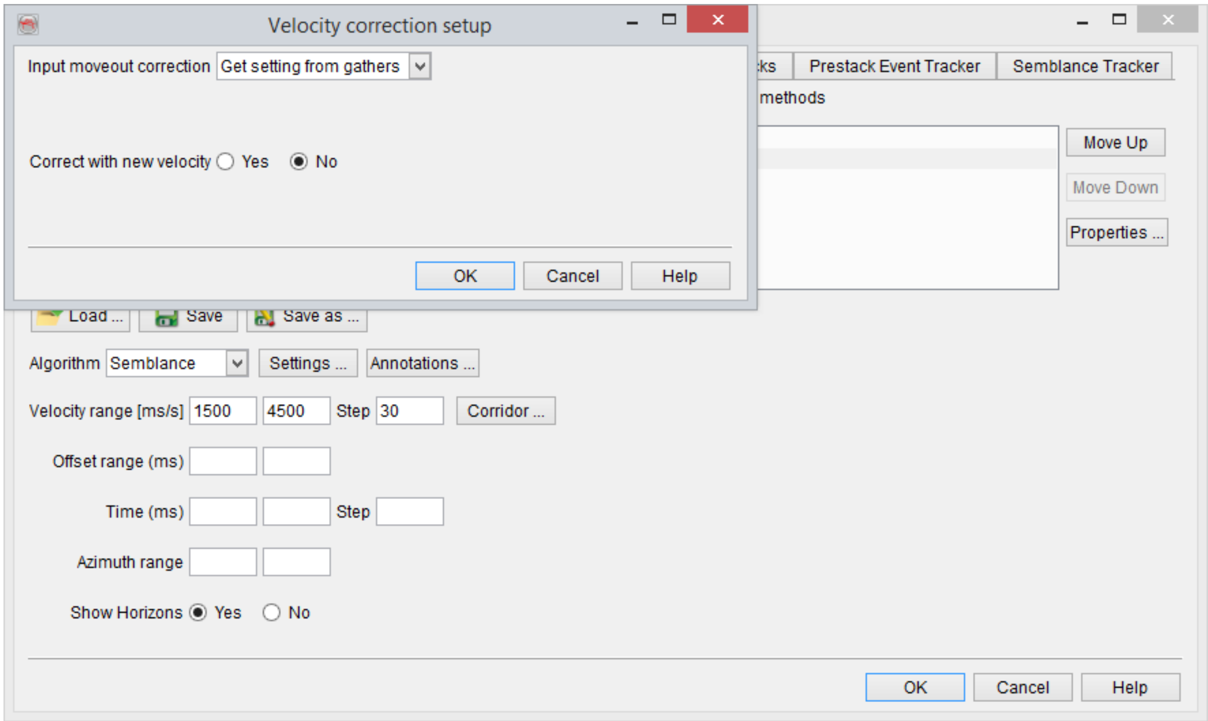
Offset range (ms):

Time (ms): Step

Azimuth range:

Show Horizons ● Yes ○ No

Additional VMB-specific pre-processing method are available, like Velocity correction:


Velocity correction setup

Input moveout correction: Get setting from gathers

Correct with new velocity ○ Yes ● No

Algorithm: Semblance

Velocity range [ms/s] 1500 4500 Step 30

Offset range (ms)

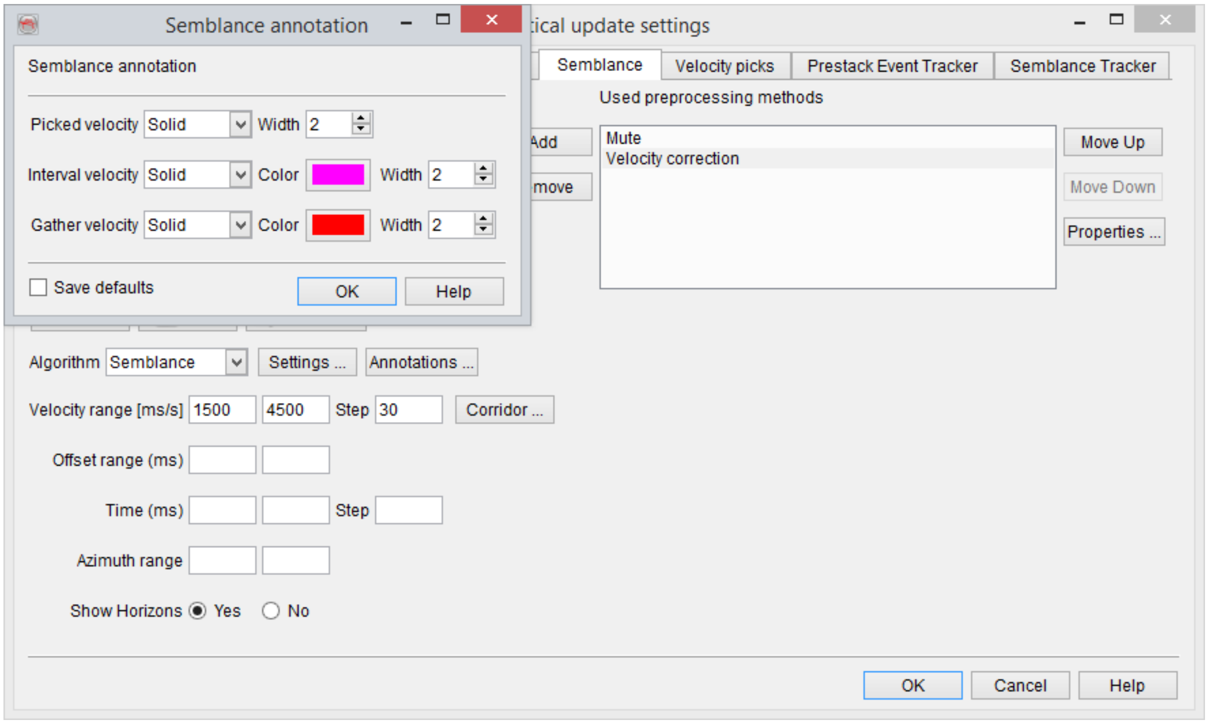
Time (ms) Step

Azimuth range

Show Horizons ● Yes ○ No

This pre-processing method will apply the velocity correction defined by the gridded velocity picks to the pre-stack data.

Semblance annotations Annotations for the semblance display can be set and saved as user default settings in the following window:



Semblance annotation

Picked velocity Solid Width 2

Interval velocity Solid Color 🟪 Width 2

Gather velocity Solid Color 🟥 Width 2

☐ Save defaults

Used preprocessing methods

Mute
Velocity correction

Algorithm Semblance

Velocity range [ms/s] 1500 4500 Step 30

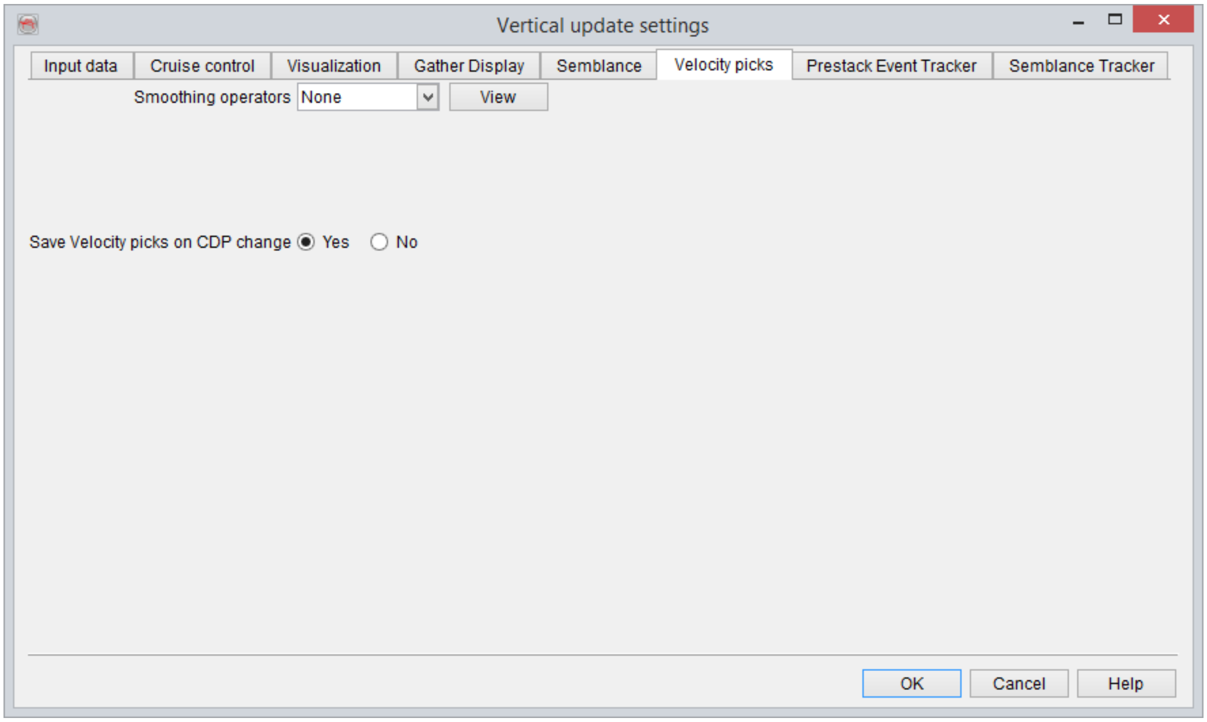
Offset range (ms)

Time (ms) Step

Azimuth range

Show Horizons ● Yes ○ No

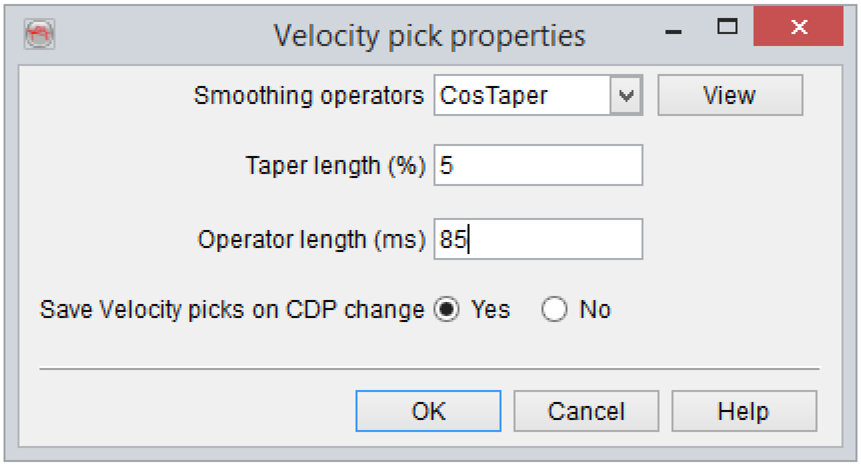
- **Velocity picks:** In this tab you can specify the tapering level and saving mechanism for the velocity picks. The main option is to set the tapering function and parameters used when deriving the new velocity curve from the velocity picks and migration velocity. The view option enables you to visualize the tapering functions.



Smoothing operators None

Save Velocity picks on CDP change ● Yes ○ No

Those settings can be retrieved from the *Velocity picks* menu with the option *Properties*:



Smoothing operators CosTaper

Taper length (%) 5

Operator length (ms) 85

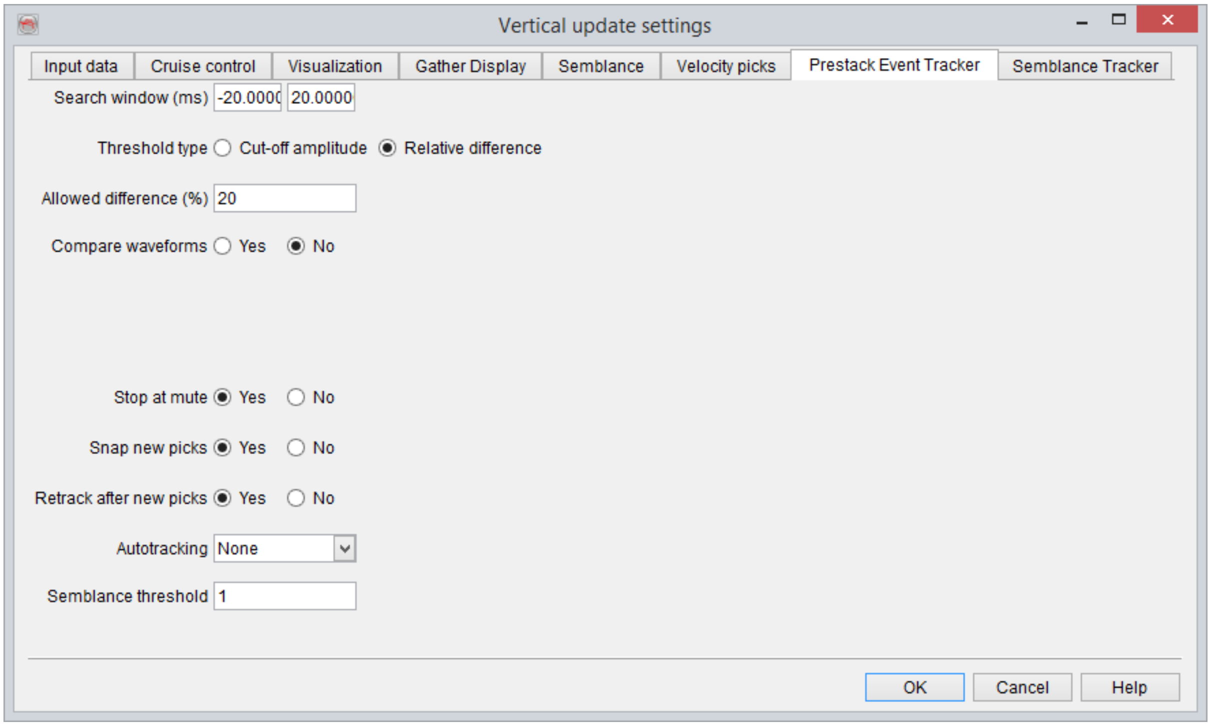
Save Velocity picks on CDP change ● Yes ○ No

- **Prestack Event Tracker:** The prestack event tracker presents similar options to the post-stack horizon tracker. The tracking is based on the absolute or relative amplitude changes from trace to trace given a search window, optionally using the trace-to-trace similarity. Nevertheless given a single prestack event tracking you will be able to track both peaks and troughs on the CIG. This decision will be made based on the position of the first pick of each prestack event.

The seeds appear larger than the tracked positions, although both types are editable (movable or removable).
Futhermore there is always one active prestack event displayed using a thicker bold line, except if the cursor is outside of the search window of all existing seeds of the CIG. Any new pick will be append to the active pre-stack event if the cursor is within the search window of an existing pick, except if the keys control+shift are pressed when picking. In that case a new pre-stack event will be created.

On a new pre-stack gather the pre-stack may be picked and tracked or autotracked . The autotracking is to be used at a new locations as it will become unavailable if one or more pre-stack events are picked. On the contrary at least one seed must exist before pressing the track option.

The autotracking option is meant to use either the calculated semblance or the post-stack horizons to create seeds for the autotracking of the pre-stack events. This option must be set in the tracking settings before using the corresponding icons in the velocity analysis windows. All autotracked events can be edited afterwards.



Search window (ms) -20.000 20.0000

Threshold type ○ Cut-off amplitude ● Relative difference

Allowed difference (%) 20

Compare waveforms ○ Yes ● No

Stop at mute ● Yes ○ No

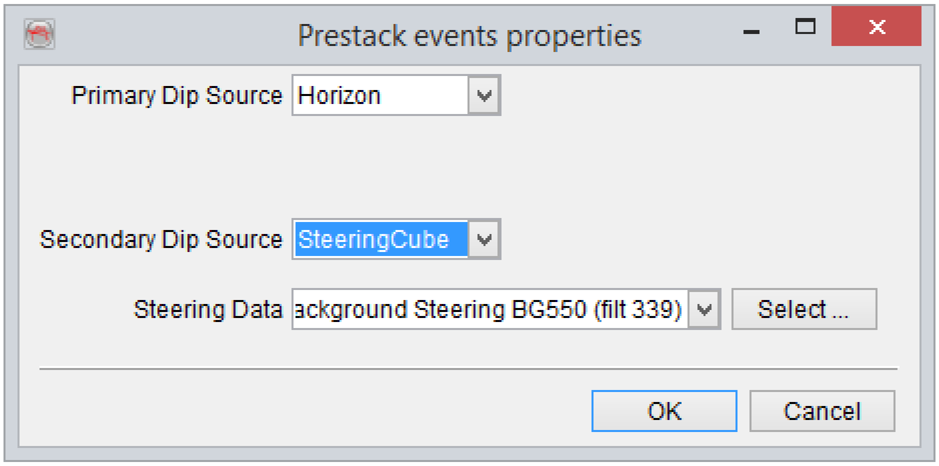
Snap new picks ● Yes ○ No

Retrack after new picks ● Yes ○ No

Autotracking None

Semblance threshold 1

Those settings can be retrieved from the *Prestack event* menu with the option *Properties*:

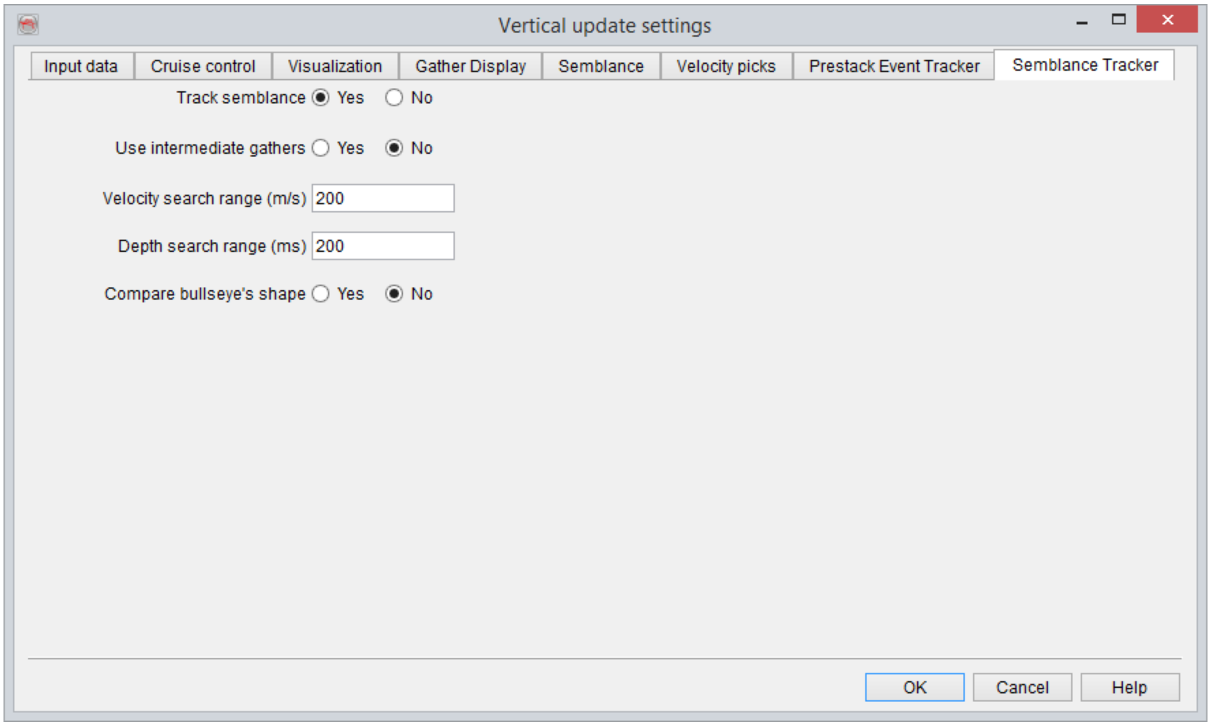


Primary Dip Source Horizon

Secondary Dip Source SteeringCube

Steering Data ackground Steering BG550 (filt 339)

- **Semblance Tracker:** The Velocity picks can be tracked by following the picked RMOs between the previous location and the current position using the semblances panels at both locations, or optionally using all intermediate semblance panels. Most of the settings are once again very similar to the horizon tracker and pre-stack events tracker.



Track semblance ● Yes ○ No

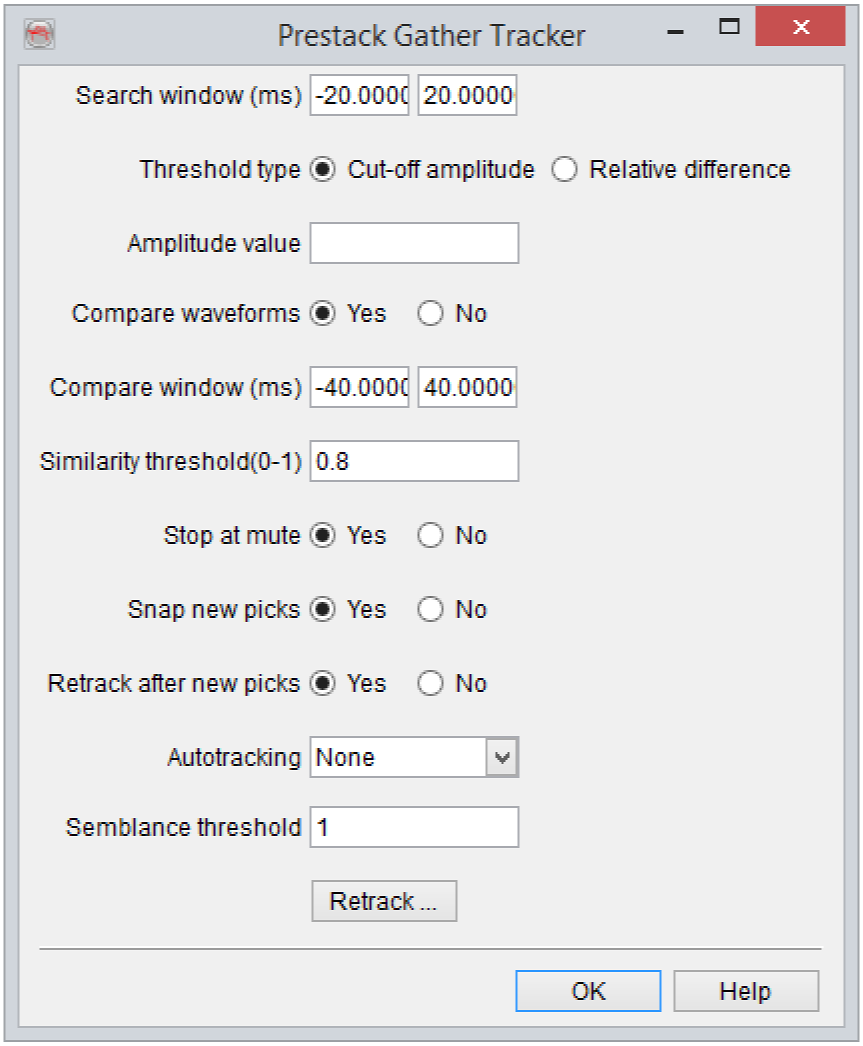
Use intermediate gathers ○ Yes ● No

Velocity search range (m/s) 200

Depth search range (ms) 200

Compare bullseye's shape ○ Yes ● No

Those settings can be retrieved from this icon in the toolbar:



Search window (ms) -20.0000 20.0000

Threshold type ● Cut-off amplitude ○ Relative difference

Amplitude value

Compare waveforms ● Yes ○ No

Compare window (ms) -40.0000 40.0000

Similarity threshold(0-1) 0.8

Stop at mute ● Yes ○ No

Snap new picks ● Yes ○ No

Retrack after new picks ● Yes ○ No

Autotracking None

Semblance threshold 1

Horizon-based Velocity Update

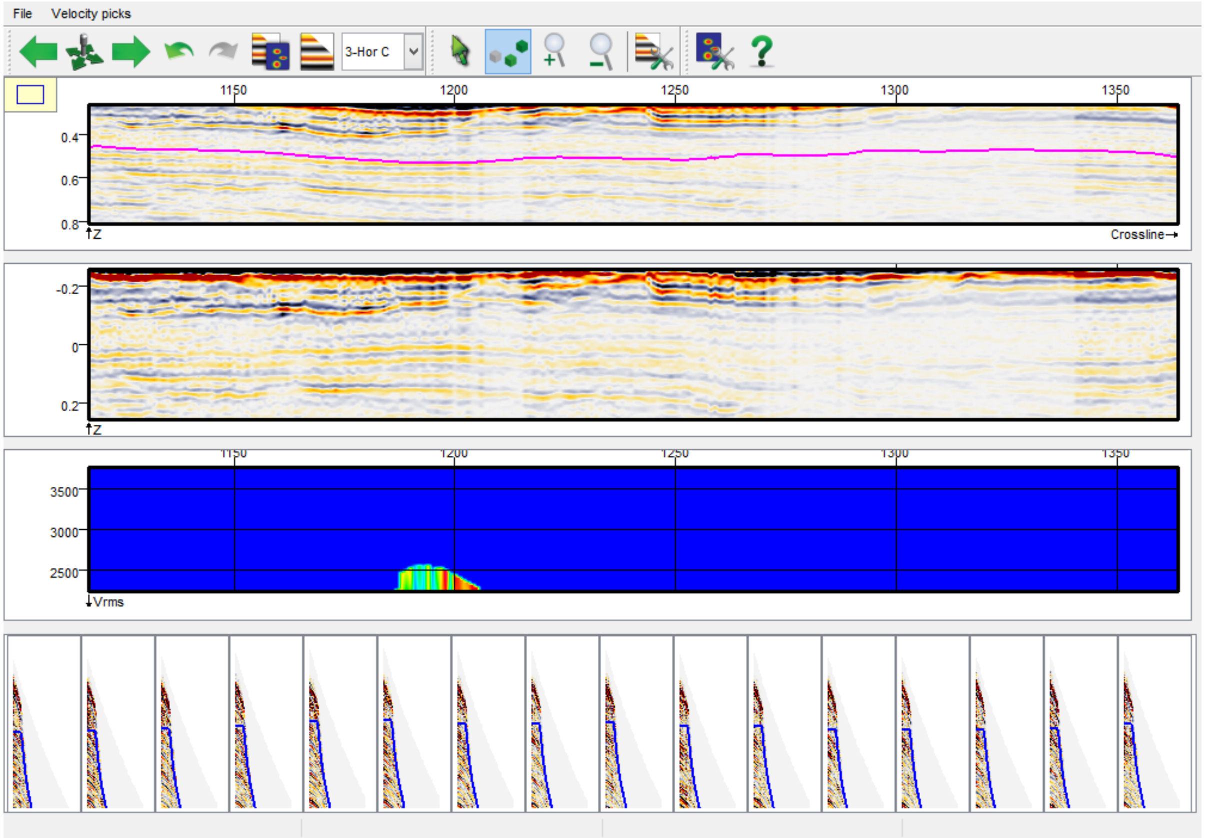
From the VMB toolbar, launch the horizon-based velocity analysis using the appropriate icon .

The window is setup using four 2D viewers, presented from top to bottom:

1. The migrated image on an inline or crossline
2. The migrated image on that same inline or crossline, flattened with respect to the active horizon

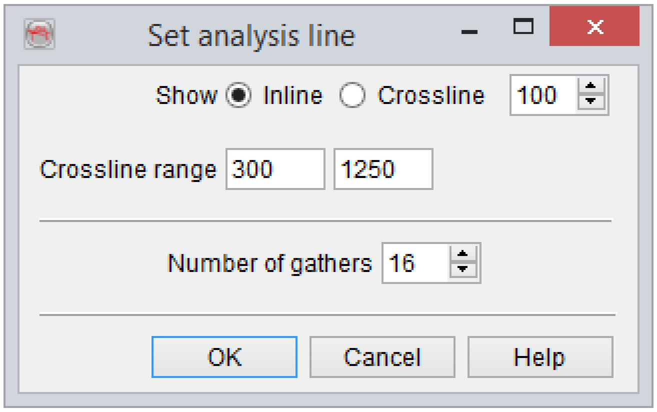
3. The semblance calculated at the intersection of the line and the active horizon

4. A set of CIG along the selected line, before or after semblance pre-processing, with or without RMO correction



The horizon-based analysis works similarly to the vertical velocity analysis, although only Velocity picks can be picked, and neither migration velocity nor picked velocity are displayed. The mutes and pre-stack events must be picked in the vertical velocity analysis window. The RMO function defined by the picks will be interpolated laterally over the entire section, providing RMO corrections at each CIG of the line. A combo box allows the quick toggle between the horizons loaded in the settings. Please note that the cursor as well as the velocity picks are synchronized between both velocity analysis windows.

In the two viewer you can jump to any inline or crossline any position using this toolbar icon , that launches a line selection window:



Show ● Inline ○ Crossline 100

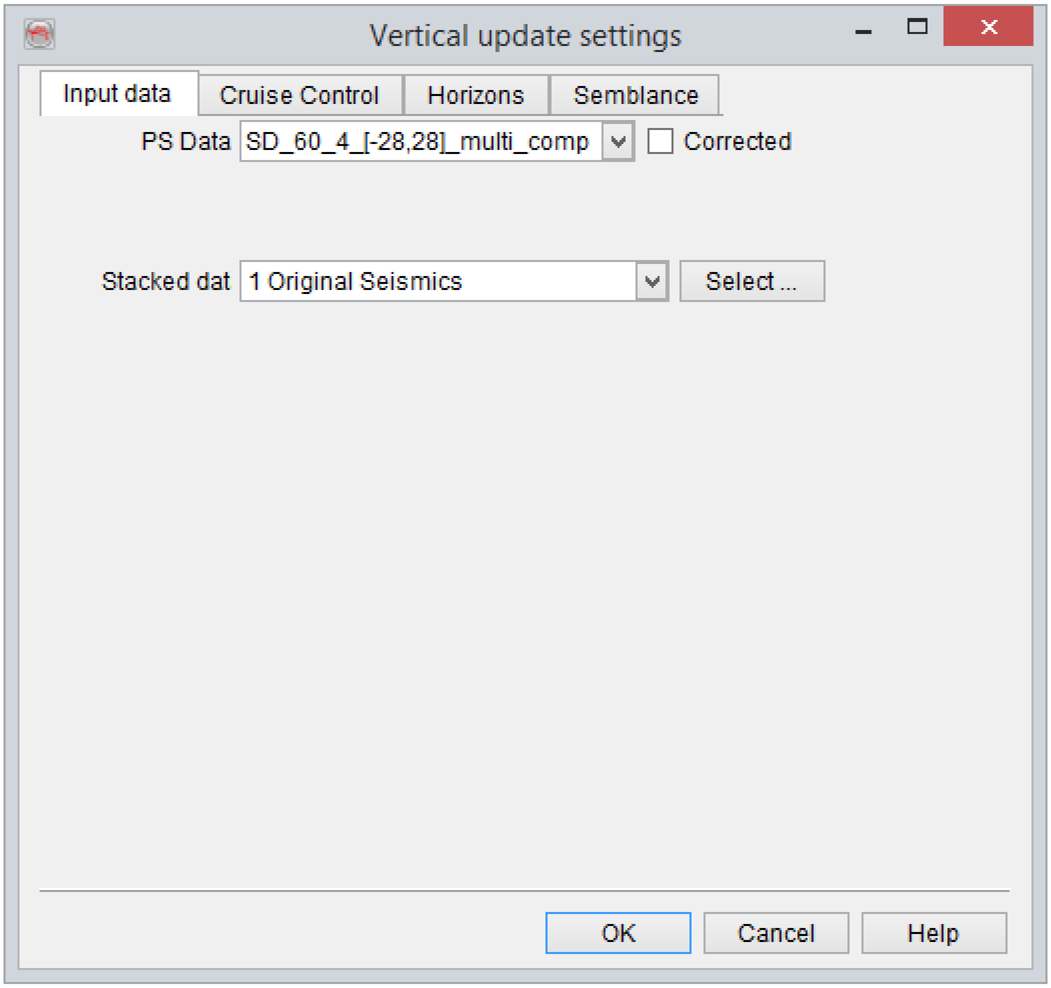
Crossline range 300 1250

Number of gathers 16

Settings: (available via *File--> Settings...*)

This section presents only the settings that are different from the vertical velocity analysis settings.

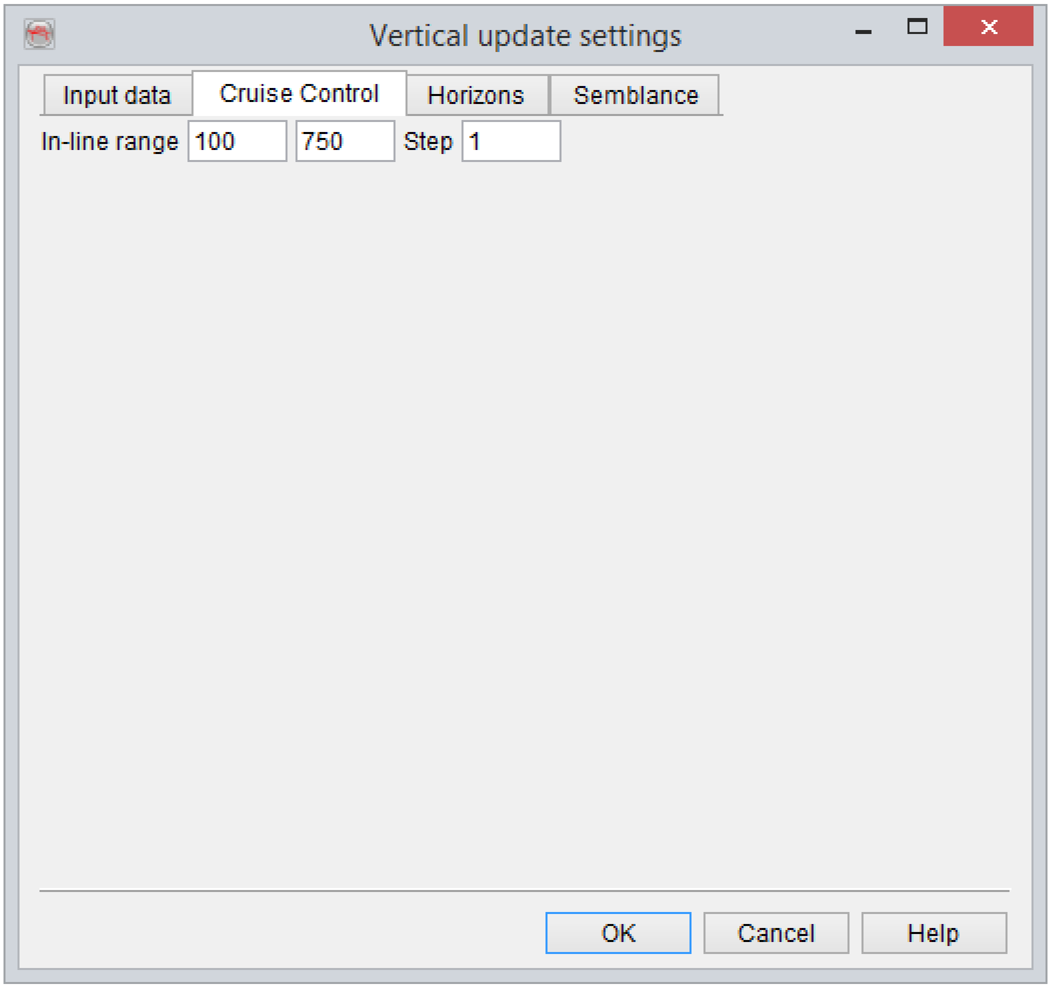
- **Input:** The input will link the prestack data and migration velocity to the corresponding stacked data visible in the two upper 2D viewers.



PS Data: SD_60_4_[-28,28]_multi_comp ☐ Corrected

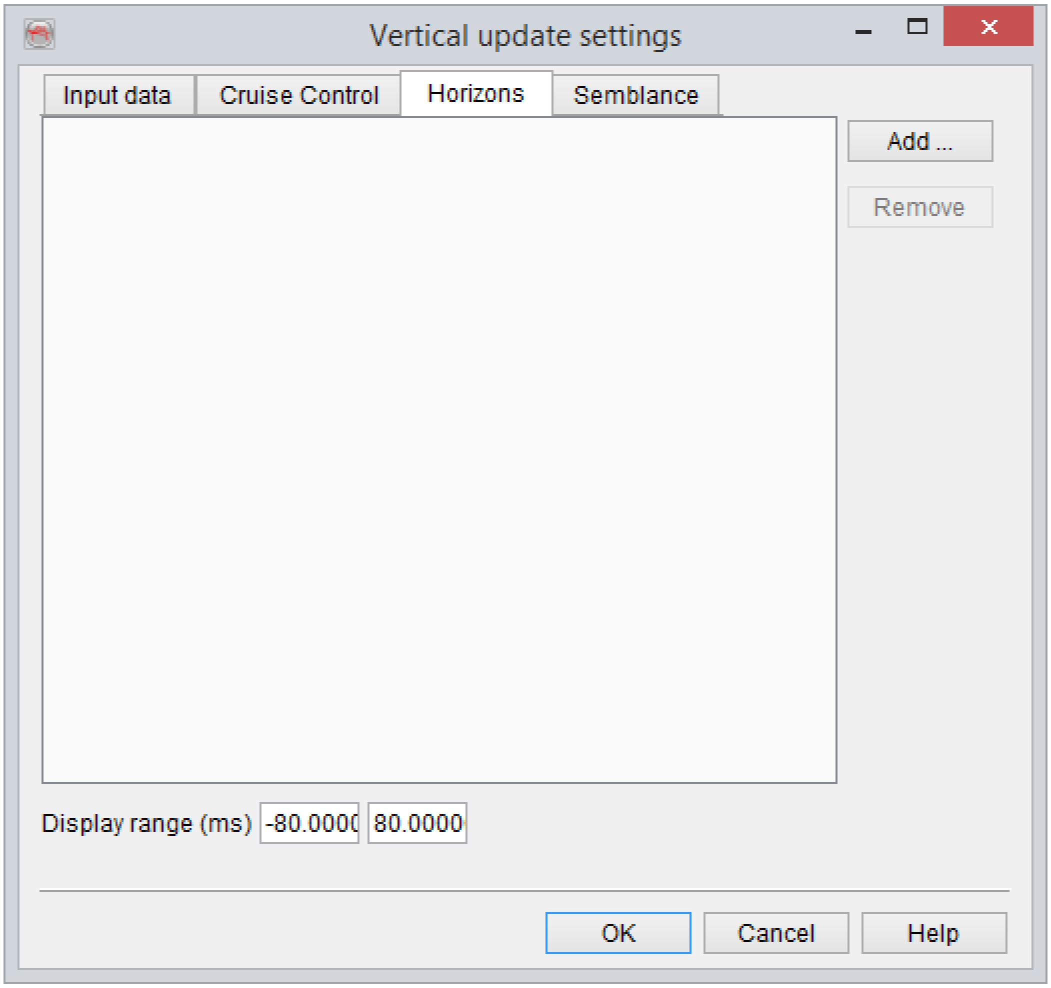
Stacked dat: 1 Original Seismics

- **Cruise control:** The available item is either an inline or crossline range, depending on the choice of active line. This can be changed using the "Go to any position" button where the analysis line (number and type) can be set.



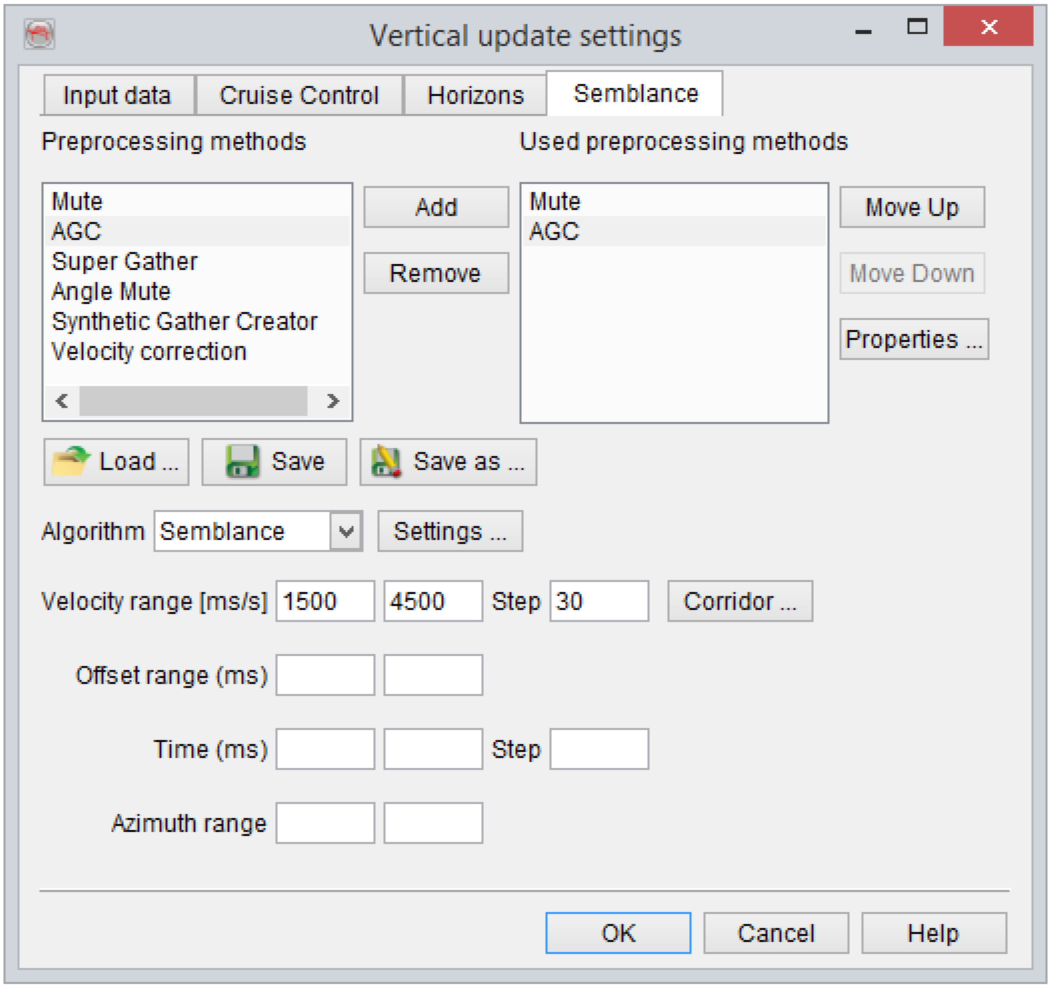
In-line range 100 750 Step 1

- **Horizons:** The horizons used for the analyzes must be defined here and are loaded by pressing the add button. You can select as many horizons as you want, and specify the display range of the flattened section (second 2D viewer from the top).



Display range (ms) -80.0000 80.0000

- **Semblance:** The semblance settings are fully similar as for the vertical velocity analysis window.



Preprocessing methods
Mute
AGC
Super Gather
Angle Mute
Synthetic Gather Creator
Velocity correction

Used preprocessing methods
Mute
AGC

Algorithm Semblance

Velocity range [ms/s] 1500 4500 Step 30

Offset range (ms)

Time (ms) Step

Azimuth range

Input-Output

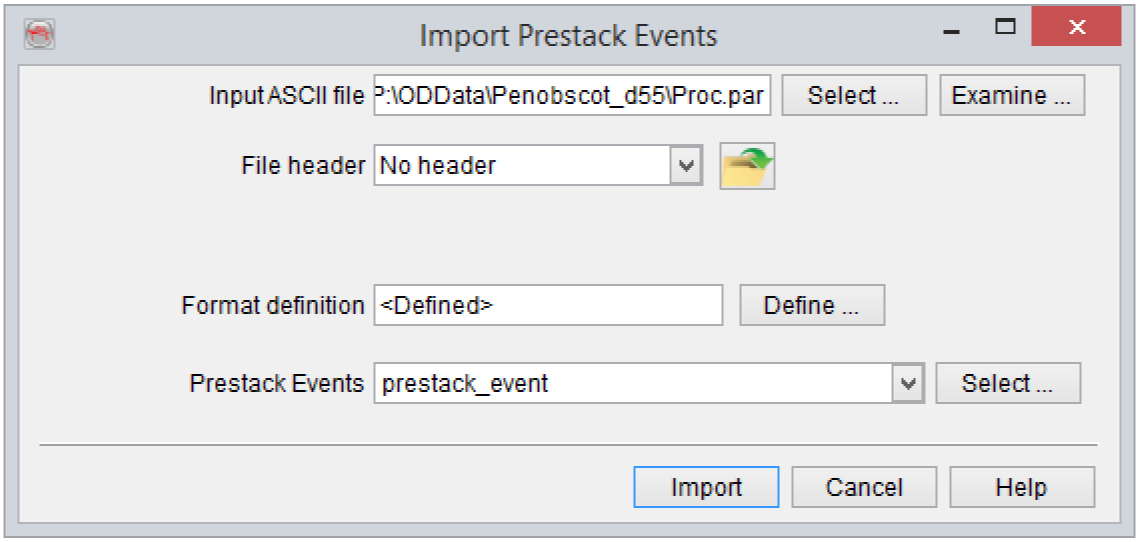
Most of the processing in the Velocity Model Building Plugin will not require data import and export:

The velocity volumes are built, and remain, in OpendTect, such as the picked mute definitions, RMO and tomography events.

Nevertheless, for convenience it is possible to export the volumes to SEG-Y files and the mute definition and pre-stack events to text files (ASCII). Please refer to the base documentation for the export of volumes and mute definitions.

Pre-stack Event Import

The import of pre-stack data is similar to the other import. After selecting the proper file, in ascii format, the presence/absence of header has to be specified. It can be of fixed or variable length. If variable, the "word" defining the end of the header has to be given. then the format needs to be define : the columns for each listed quantities have to be indicated.

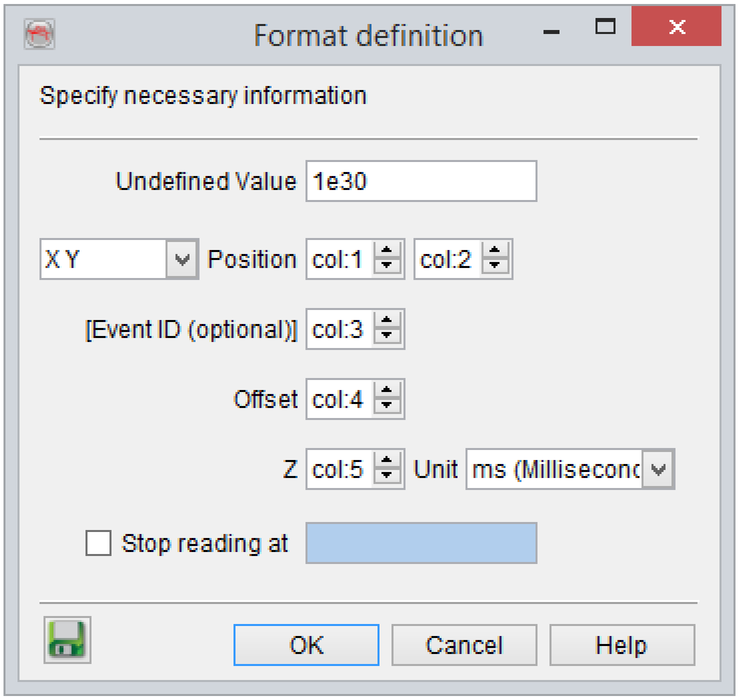


Input ASCII file P:\ODData\Penobscot_d55\Proc.par

File header No header

Format definition <Defined>

Prestack Events prestack_event



Undefined Value 1e30

X Y Position col:1 col:2

[Event ID (optional)] col:3

Offset col:4

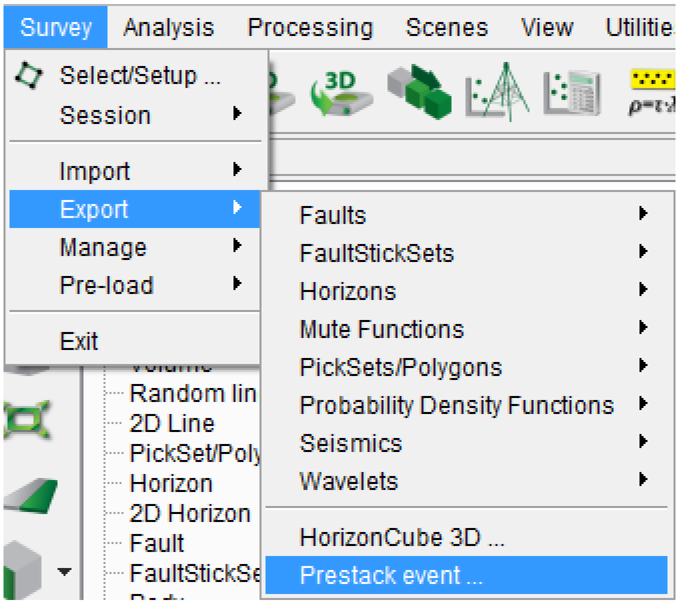
Z col:5 Unit ms (Millisecond

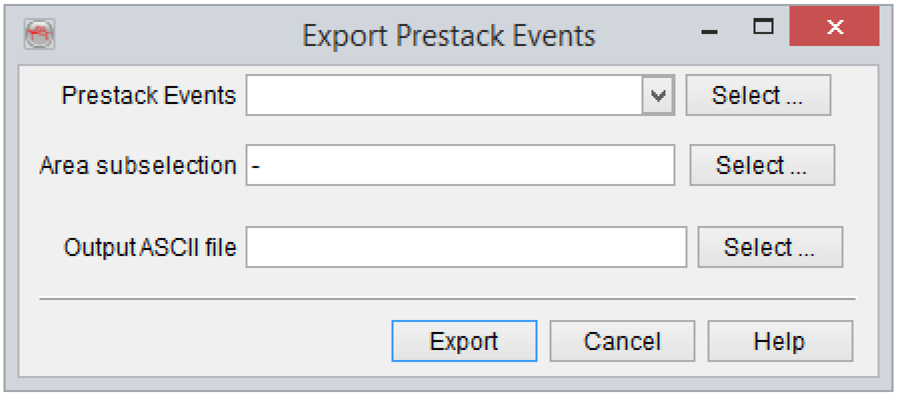
☐ Stop reading at

If the Event ID is not mentioned in the file, associate the *col:0* (first column in the file is *col:1*). The data import can be done until a given row, you just need to specify it in *Stop reading at*.

Pre-stack Events Export

The export menu enables exporting of picked pre-stack events to text (ASCII) files. The output file will be column-sorted with one row per pick. The following data will be found is each column:





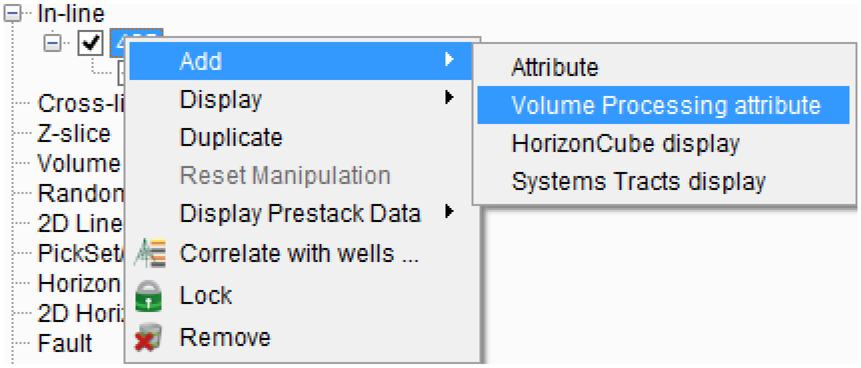
- Inline
- Crossline

- Event index (0-N)
- Dip, going to increasing inlines. 0 is written if dip is not available.
- Dip, going to increasing crosslines. 0 is written if dip is not available.
- Event quality, 0-255 (higher is better)
- Azimuth (0-2PI)
- Offset
- Depth
- Pick quality, 0-255 (higher is better).

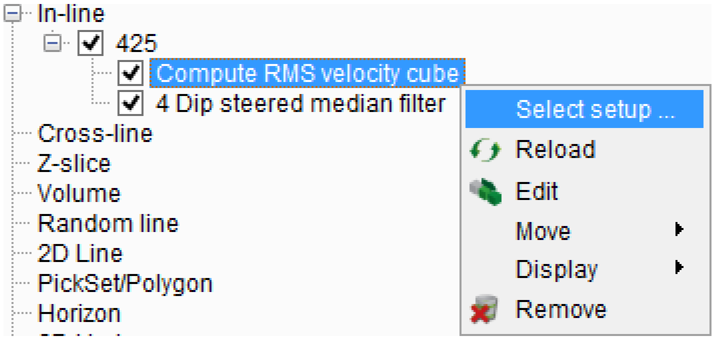
The dip comes either from a horizon (if the event is picked from an horizon) or from the SteeringCube. The pick quality is determined while tracking depending on the strengths of the event and how well the individual pick fits the rest.

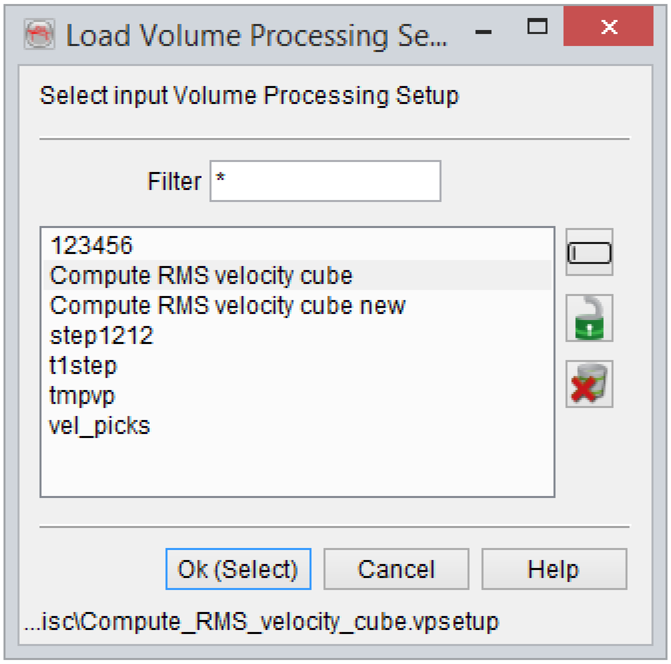
Velocity Display

Once velocity picks have been made they can be gridded using the volume processing tools. The VMB plugins contains a special tree item that enables a quick application of the volume processing workflow without the need to process the entire volume:



Pressing "Add velocity display" in the right-click menu of a slice (inline, crossline, time-slice) adds a special layer. Its right-click menu does not allow the selection of stored volumes or attributes, but the selection of a stored volume processing (gridding) flow.





Filter *

...isc\Compute_RMS_velocity_cube.vpsetup

The selection of a flow will launch its computation on the current slice only. If you already added a velocity display ,you can either reload it in case of modification or select a different processing setup.

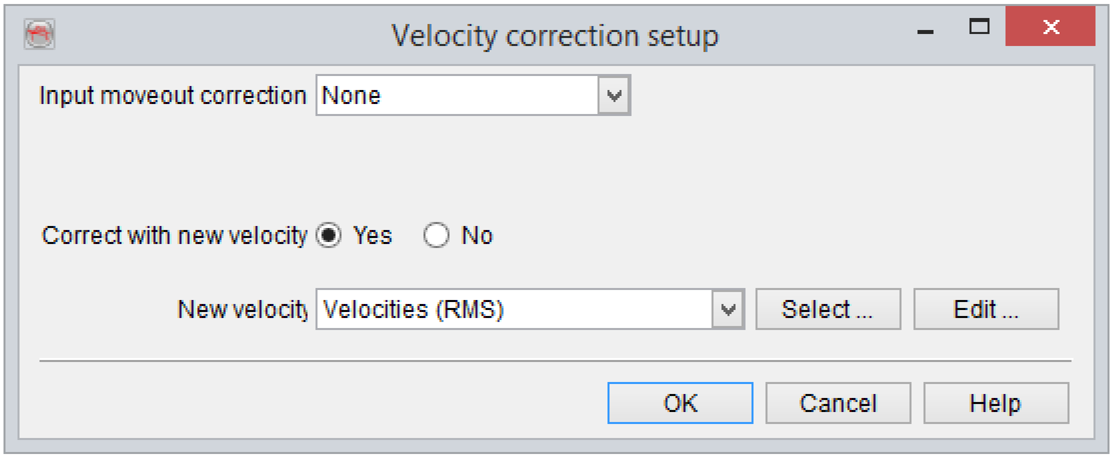
> **Note:** Although the display is for a single slice the computation is still volume-based. Therefore it can take some time to process and will make OpendTect unresponsive during that period of time. For large jobs batch processes may still be preferred.

Velocity Correction

In the Prestack processing steps available in *Processing > Create Seismic Output > Pre-Stack processing*, the Velocity correction method will (un-)apply a normal moveout correction based either the migration velocity and/or a new velocity model. An hyperbolic moveout is applied (non-hyperbolic moveout will be implemented later).

The following combinations can be performed:

1. The gathers **are not NMO corrected** and the objective is to **apply a NMO correction** with either the migration velocity volume or any other picked velocity model. In that case the following setup must be used:

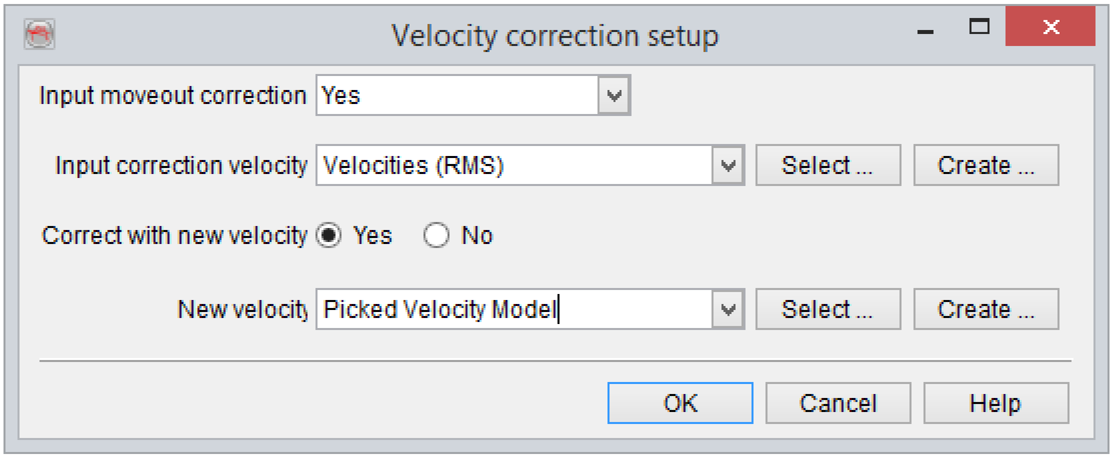


Input moveout correction: None

Correct with new velocity ● Yes ○ No

New velocity: Velocities (RMS)

2. The gathers **are NMO corrected** and the objective is to **undo and apply a a NMO correction with ANOTHER velocity model**. In that case the velocity model used for applying the NMO correction on the stored gathers must be specified (such that it can be undone), and the new velocity model must be selected:



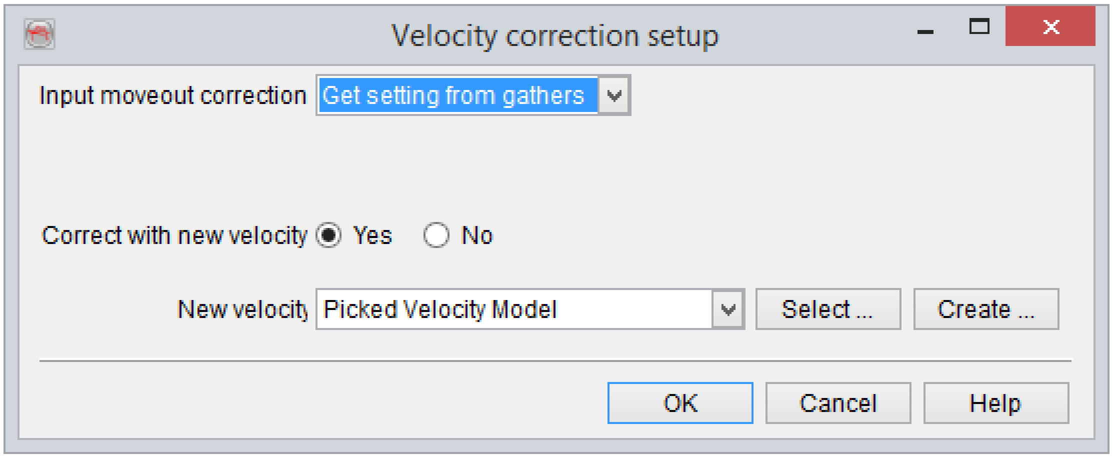
Input moveout correction: Yes

Input correction velocity: Velocities (RMS)

Correct with new velocity ● Yes ○ No

New velocity: Picked Velocity Model

Users of the VMB plugin may have already linked the gathers to a migration velocity volume, and set a flag saying that the gathers are NMO corrected or not. If that is that case the first part can be set to "Get setting from gathers" and one needs only to specify the correction velocity:



Input moveout correction: Get setting from gathers

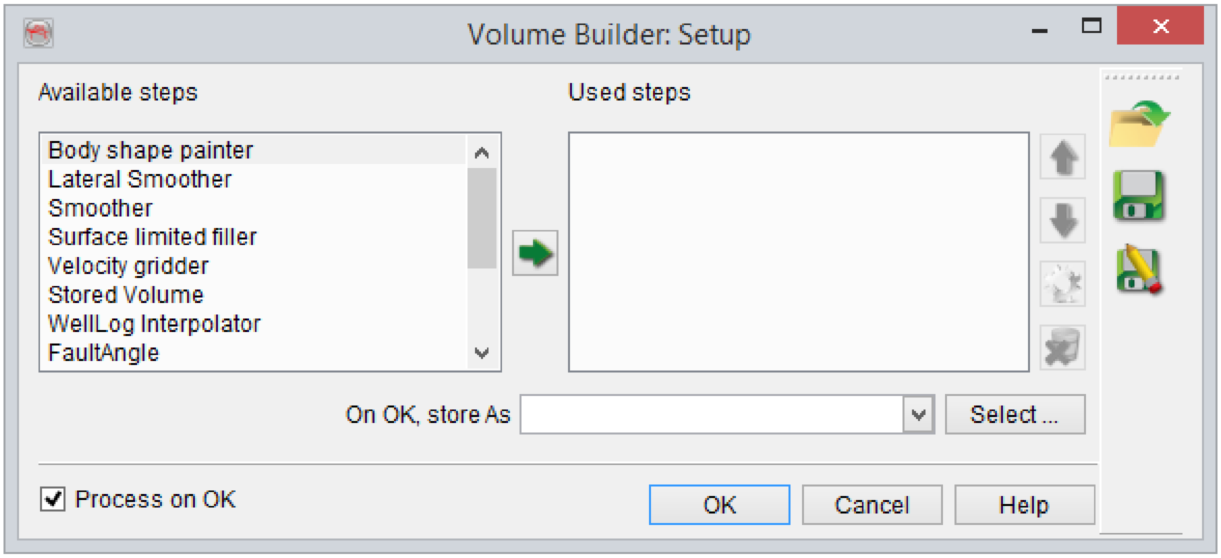
Correct with new velocity ● Yes ○ No

New velocity: Picked Velocity Model

VMB Specific Gridding Step - Gridding of Velocity Picks

The volume processing tool of OpendTect has a step called "Velocity gridder". It can grid either velocity functions or volumes.

The VMB plugin enables the selection of a third velocity source, the velocity picks made in the VMB analysis windows (vertical/horizontal). No parameter is required since the velocity picks are attached to the migration velocity during the **analysis**. The stored RMO will be converted to interval velocities and gridded in the volume. Please note that the gridding of velocities is done in order to maintain the time-depth relation hold by the velocity function.



Volume Builder: Setup

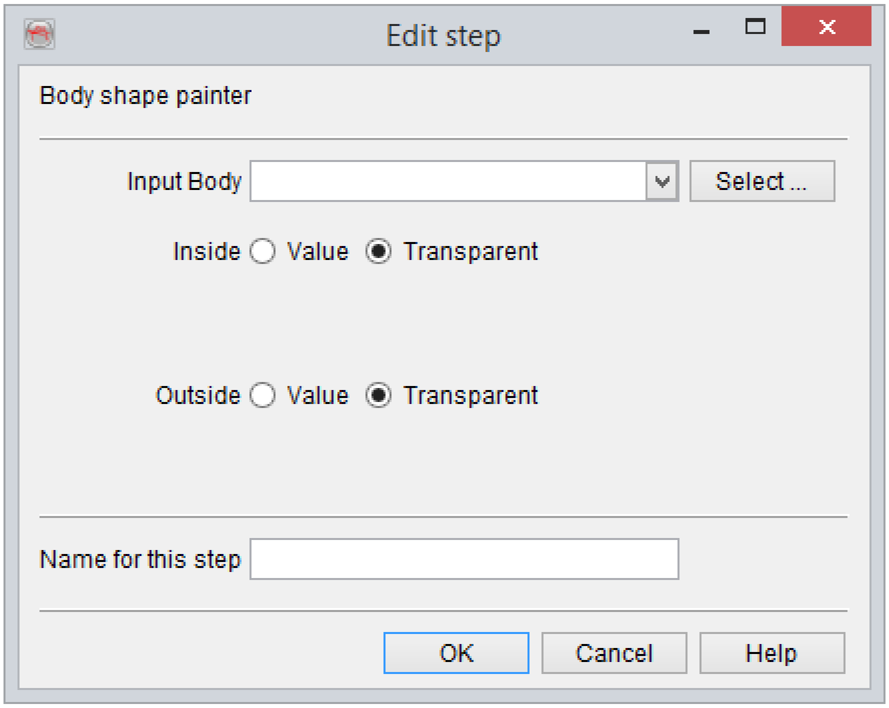
Available steps

- Body shape painter
- Lateral Smoother
- Smoother
- Surface limited filler
- Velocity gridder
- Stored Volume
- WellLog Interpolator
- FaultAngle

Used steps

On OK, store As

☑ Process on OK



Edit step

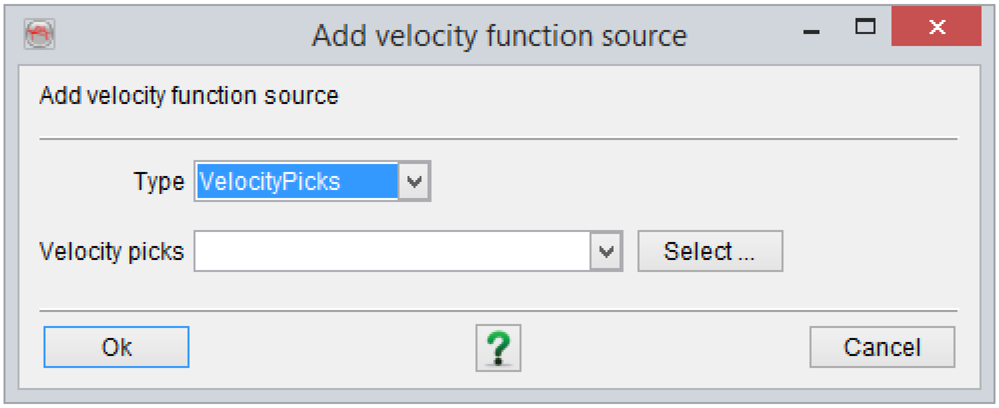
Body shape painter

Input Body

Inside ○ Value ● Transparent

Outside ○ Value ● Transparent

Name for this step



Add velocity function source

Add velocity function source

Type: VelocityPicks

Velocity picks

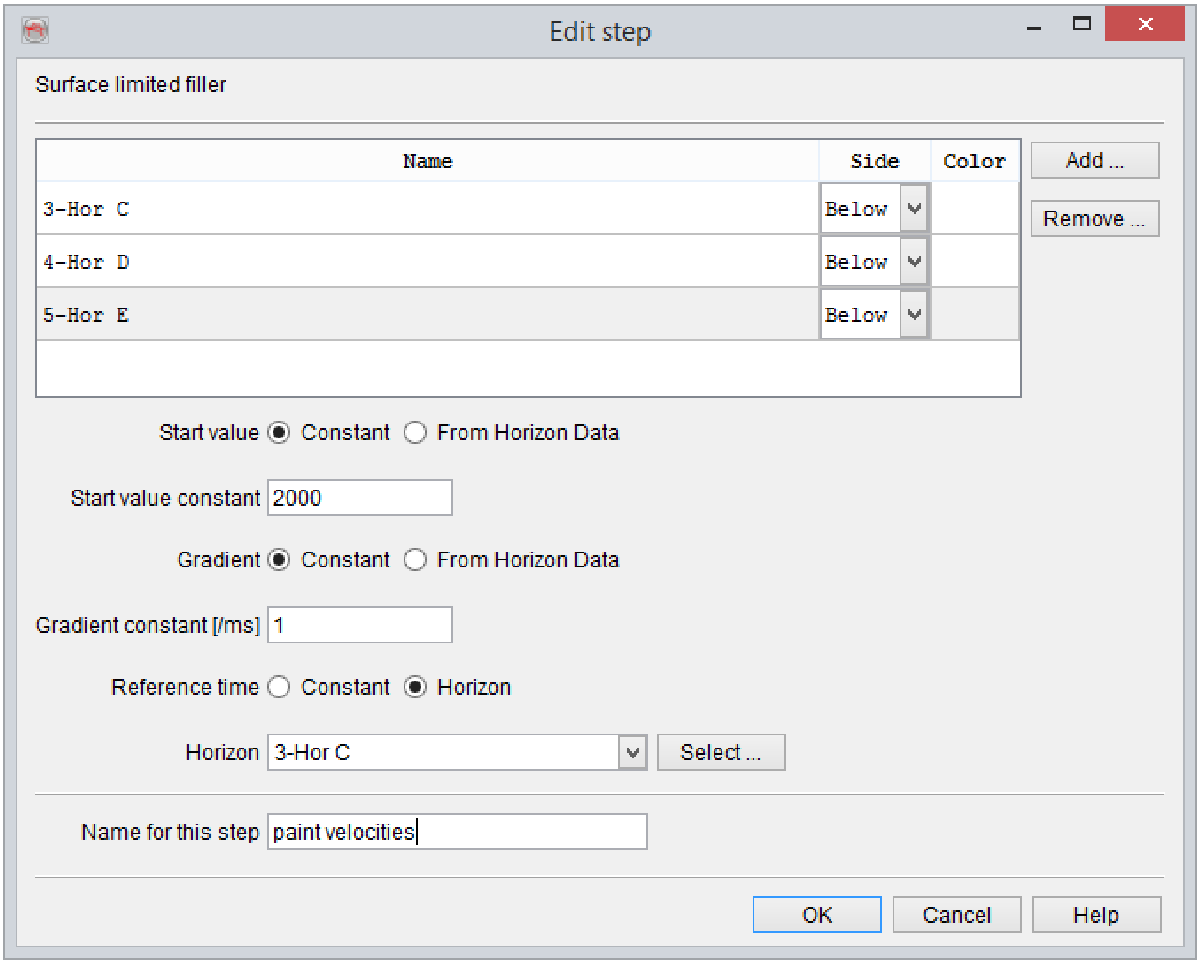
VMB Specific Gridding Step - Surface-limited Filler

The surface limited filler intends to paint velocities in a 3D area whose geometry is defined by one or more 3D horizons. This step may look like the inter-horizon filler but it is actually more powerful.

The Add and Remove buttons should be used to select the 3D horizons to be used in the actual step. The side defines the relative position of the horizon with respect to the area to be painted with velocities. For instance one of the horizon could be a salt flank loaded as a 3D horizon.

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. An horizon in between could for instance 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.



Surface limited filler

Name	Side	Color
3-Hor C	Below	
4-Hor D	Below	
5-Hor E	Below	

Start value ● Constant ○ From Horizon Data

Start value constant 2000

Gradient ● Constant ○ From Horizon Data

Gradient constant [/ms] 1

Reference time ○ Constant ● Horizon

Horizon 3-Hor C

Name for this step paint velocities

Fluid Contact Finder (formerly CCB)

Table of Contents

[Introduction]

[FCF Main window]

[FCF Analysis]

[Local FCF attribute]

Based on ideas of Jan-Gabe van der Weide and Andries Wever of *Wintershall Noordzee BV.*

Wintershall Noordzee BV has granted dGB the usage of the material on which they hold Intellectual Property claims.

Introduction

This plugin will work only in a 3D survey

Fluid Contact Finder (*FCF* for short) is a seismic filtering workflow that will stack the seismic traces with respect to the depth of a surface.

It is based on the principle that seismic traces that penetrate the reservoir at the same depth have identical hydrocarbon columns. Seismic traces with identical horizon depth (along a single contour line) may be display using random lines created along contours.

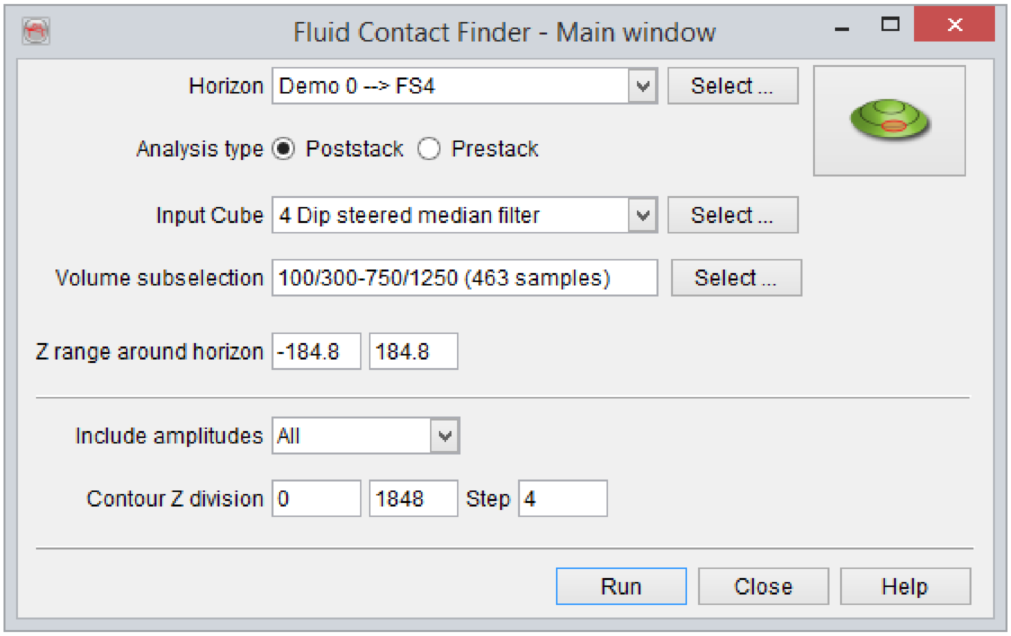
The Fluid Contact Finder may be started from the corresponding icon in the "OpendTect tools" toolbar , or via the "Processing" menu using the option "Fluid Contact Finder".

FCF Main Window

The FCF main window is used to specify the inputs and parameters required for the stacking.

Inputs are:

1. Surface defining the contour lines geometry
2. Seismic volume to stack, post- or pre-stack.



Fluid Contact Finder - Main window

Horizon Demo 0 --> FS4

Analysis type ● Poststack ○ Prestack

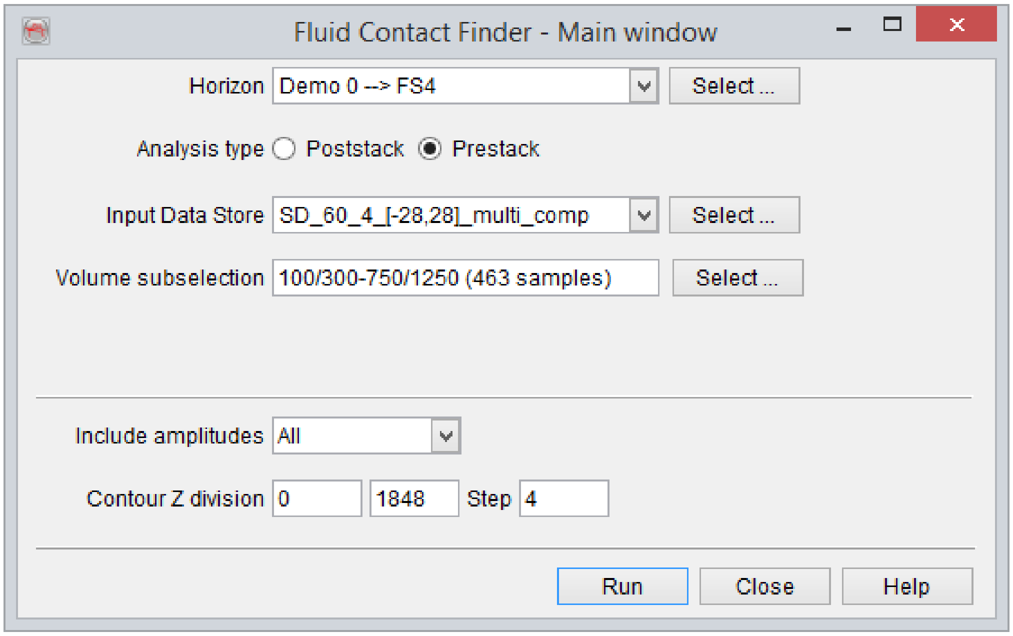
Input Cube 4 Dip steered median filter

Volume subselection 100/300-750/1250 (463 samples)

Z range around horizon -184.8 184.8

Include amplitudes All

Contour Z division 0 1848 Step 4



Fluid Contact Finder - Main window

Horizon Demo 0 --> FS4

Analysis type ○ Poststack ● Prestack

Input Data Store SD_60_4_[-28,28]_multi_comp

Volume subselection 100/300-750/1250 (463 samples)

Include amplitudes All

Contour Z division 0 1848 Step 4

Parameters are:

1. Volume subselection: Area defining the traces that will be used for stack. This can be an entire volume, a rectangular sub-selection (possibly decimated), a table sub-selection, or a polygonal sub-selection.

2. Contour Z division: start, stop and step value for the bin selection: A bin is a time/depth gate around a contour line [-step/2, +step/2]. It will be the X-axis of the 2D FCF display. Start and stop values are absolute time/depth values.

3. Z range around horizon: time/depth interval used the extraction of the seismic data prior to stacking. This is a relative gate with respect to the time/depth of the contour.

Numbers must be entered in Z unit of the survey: meters, seconds or milliseconds.

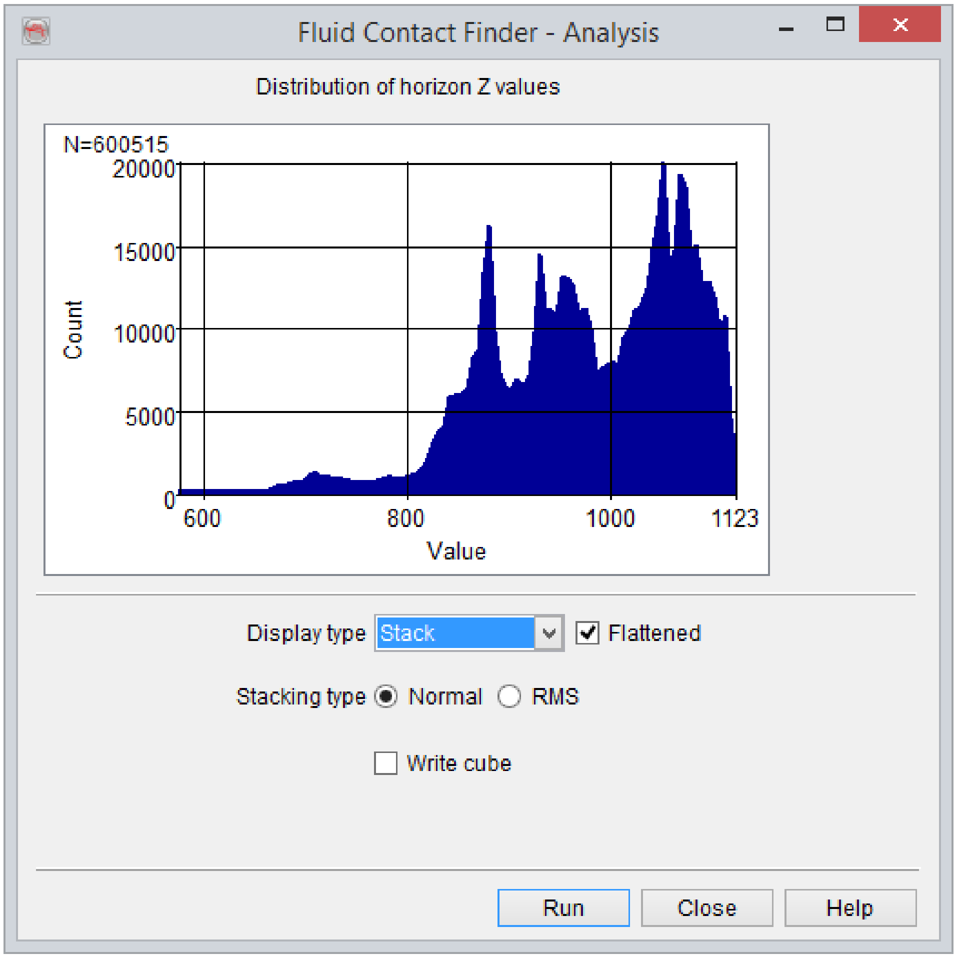
Based on the selected data the traces will be collected based on the time/depth (bin) of the horizon intersection with each trace.

FCF Analysis

The FCF analysis window has the two following functions:

1. It displays an histogram presenting the number of collected traces per bin (post-stack), or the histogram of amplitudes along the horizon (pre-stack).

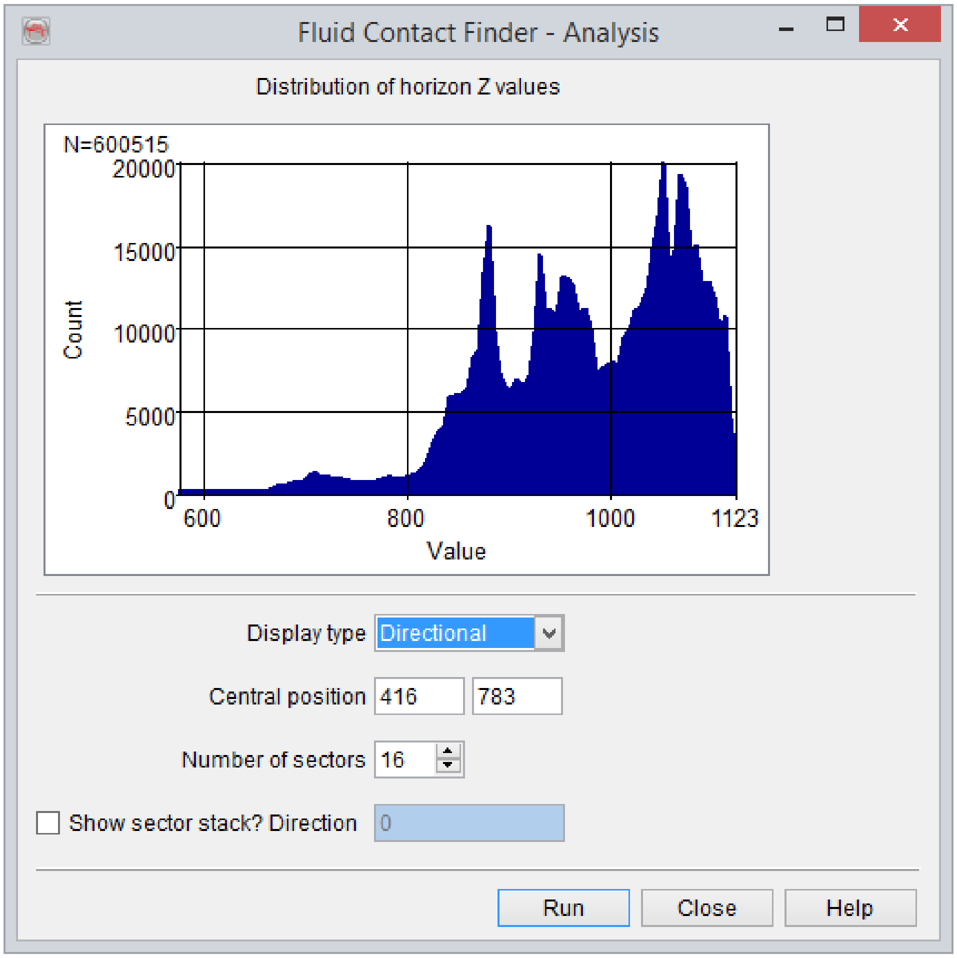
2. It allows choosing the type of display for the output stack, the stacking type used, and outputing the stacked traces to a 3D cube.



Display type Stack ☑ Flattened

Stacking type ● Normal ○ RMS

☐ Write cube



Display type: Directional

Central position: 416 783

Number of sectors: 16

☐ Show sector stack? Direction: 0

The X-axis of the histogram presents bin values in the Z unit of the survey. The Y-axis is the number of traces collected per bin (the maximum count value is shown in the top middle of the histogram).

The display (when pressing the "Go" button) will be in a 2D viewer.

- 'Stack' will display the stacked traces wrt the bin depth (post-stack), or the amplitude along the horizon in a crossplot bin depth vs. offset (pre-stack).
- 'Single Z' will display all traces of a single bin *before* stack.
- 'Directional' will display a FCF-stacked amplitude map along the horizon as a function of the distance to a central position and the azimuth

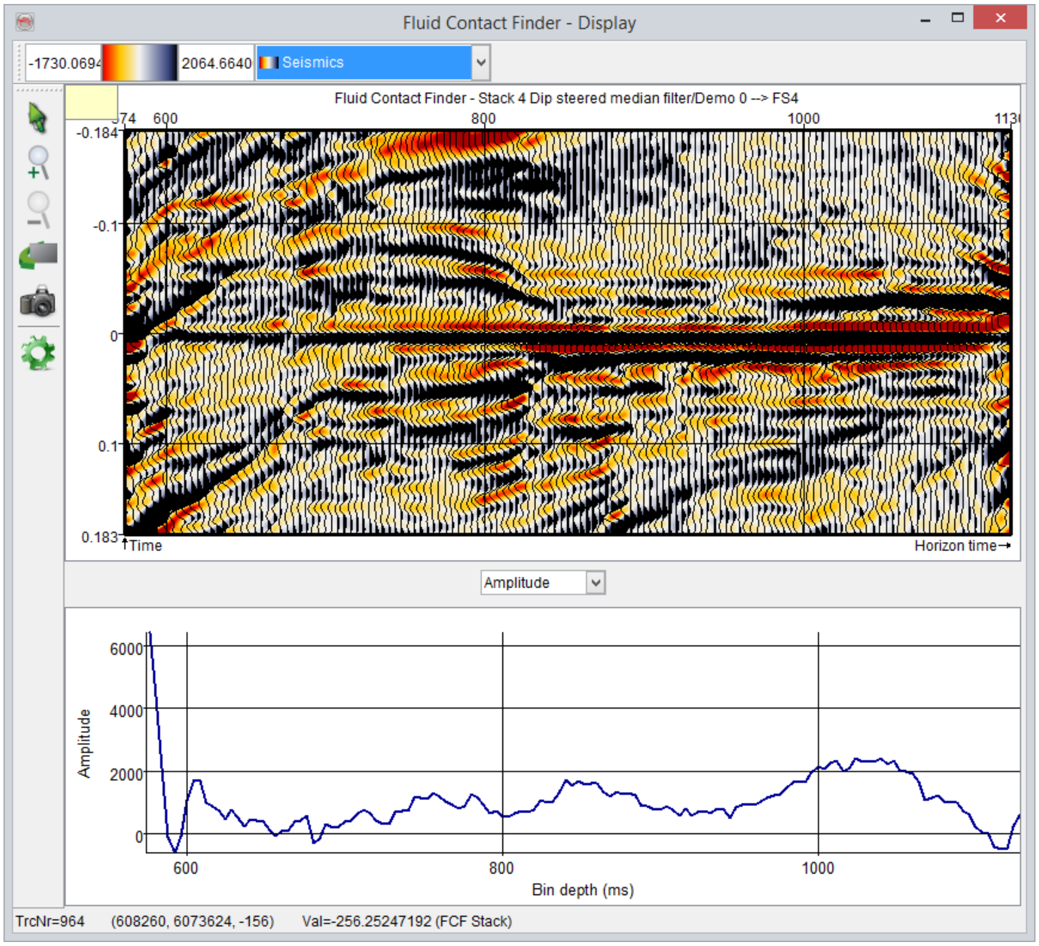
sector. The central position is pre-computed as the shallowest position with the selected area. It can be modified by the user, as well as the azimuthal sectors parameters.

Optionally, a 3D volume may be output. In this volume each input trace is replaced by the stacked trace of its corresponding bin. The output volume can then be used to make cross-plots and update amplitude maps for instance.

Please note that the FCF main window remains open when using and after closing this FCF analysis window. Multiple analysis windows can be created, with different volume subselections for instance. Nevertheless the main FCF window must remain opened to perform the stacking.

Example of 2D stack

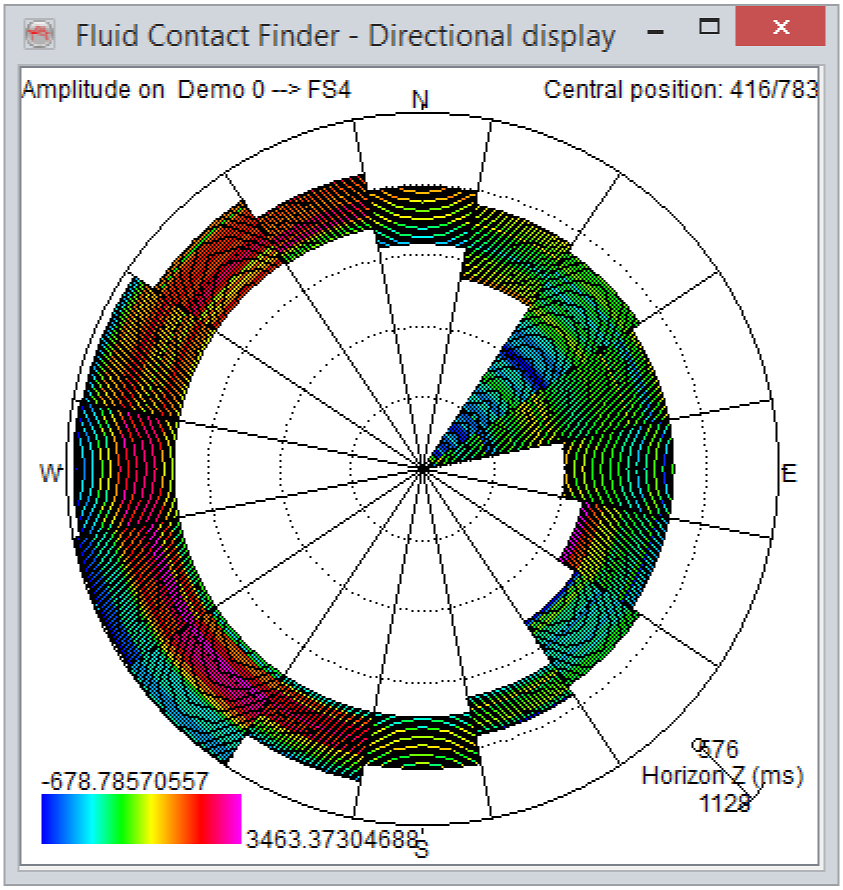
The following figure presents an example of 2D stack. Mind the increase of amplitude at 2130 ms. A crossplot of amplitude, frequency and phase is presented vs. bin depth is presented in the lower part. In the prestack FCF the crossplot shows AVO attributes



TrcNr=964 (608260, 6073624, -156) Val=-256.25247192 (FCF Stack)

Example of FCF directonal display

The following figure is an example of FCF directional plot. The colours represent the amplitude along the horizon. The grid represents the grid used for the binning of the seismic trace before stack.



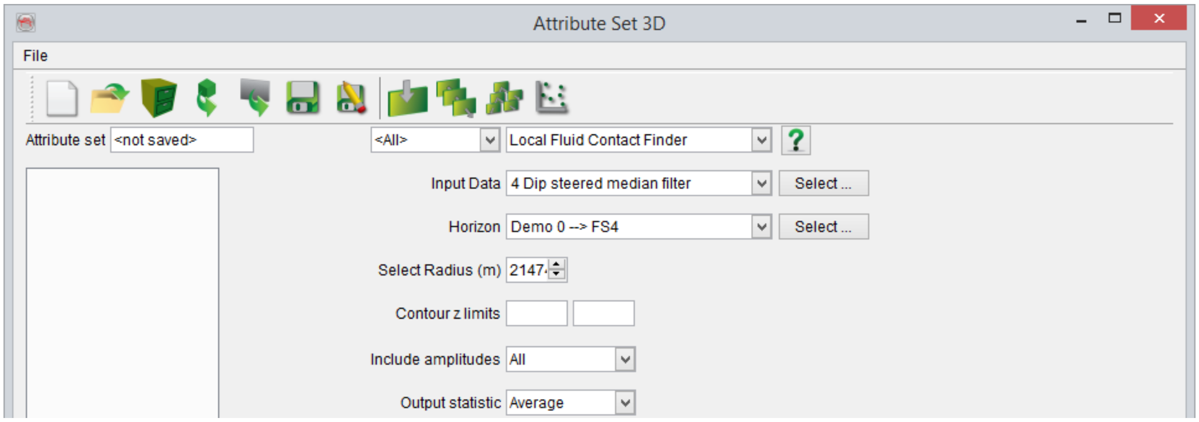
Local FCF Attribute

This statistical attribute makes local averages according to a given 3D horizon. Its primary application is the filtering of seismic data in the depth domain with the aim of removing the structural footprint and enhancing hydrocarbon-related flat spots. The output is a 3D volume with the same geometry as the input volume.

This attribute is comparable to the volume statistics attribute: The data is extracted in a user-defined radius within a time gate, and a statistic is output (average, median, RMS). The default time gate is +/- half the survey sampling rate.

The main difference is that only positions that have the same distance (+/- the time gate) between the actual sample and the pick of the 3D interpretation will be kept for the computation of the statistic. All other will be discarded. Therefore the samples extracted could be looked as a portion of contour lines computed from the 3D horizons, and shifted at the depth of the actual sample.

The radius of the data selection corresponds to the aperture of the migration: It limits the amount of data in a lateral sense collected and used by the stacking operator. Its unit should be a real world length in the same unit as the survey coordinates.



Attribute set <not saved>

<All> Local Fluid Contact Finder

Input Data 4 Dip steered median filter

Horizon Demo 0 --> FS4

Select Radius (m) 2147

Contour z limits

Include amplitudes All

Output statistic Average

Applications

Table of Contents

- How to Make The ChimneyCube
- The Dip-Steered Median Filter

How to Make the ChimneyCube

In the following, we will assume that you have selected a survey, imported a Seismic Cube, and calculated the SteeringCube. We also assume you have generated a default Chimney Cube attribute set.(For Default attribute-set see Chapter below)

Workflow

Creating a ChimneyCube is a simple process. The only input needed is picks from example locations, where chimneys have been identified along with counter examples, i.e. points that do not belong to a chimney. At these example locations, attributes will be extracted to train a neural network in order to classify the data into chimneys and non-chimneys. Typically, all attributes in the set are used but it is possible to select a sub-set only, e.g. to speed up the processing time. The ChimneyCube network has two output nodes representing chimneys and non-chimneys. Each node is given the value one (true) or zero (false). In other words, the network tries to predict a vector (1,0) when the example is a chimney and (0,1) when the example location is not a chimney. When the trained network is applied, it is sufficient to output only the chimney node. A value close to one describes a high chimney-probability at that specific location, while a value close to zero indicates low probability.

Please note that the procedure to create the ChimneyCube is not a true classification process. This, due to the output not being binary but rather continuous. However, the neural networks plugin for OpendTect does support a true classification output. This procedure is especially useful for classifying data into more than two classes, e.g. for supervised facies classification. If *Classify output* in the Create Neural Network window is set to "Yes", the network output consists of two nodes: *Classification* and *Confidence*. The Classification output returns the winning class, i.e. an integer number between 1 and N, where N is the number of classes. The Confidence indicates how close the output vector is to the optimal vector representing the winning class (e.g. 0,1,0,0 represents the second class in a 4 class problem). The match has a value between 1 (perfect) and 0 (very poor).

Picking Example Locations

Picking chimneys and non-chimneys is done in the graphics window using the plane viewers. Unless you know your data so well that you can immediately go to the inlines, crosslines, or timeslices (where you wish to pick chimneys), we recommend starting by making a few *similarity slices* at different times. Select the Time-plane viewer and add a new element (right-hand mouse button, select *Add*). Position the plane: click on the interact mode icon, click on the plane followed by a click and drag on one of the frame arrows. The position is displayed in the status bar below the scene. Click outside the frame when you have reached the desired position. The select view data menu pops up. Load one of the similarity attributes (option Attributes). If the Attributes option is not available and you can only select from the "Stored" data cubes, you have no active attribute set; create one in the attribute set editor from the *Processing* menu. Loading a similarity attribute from the set means that the attribute is calculated on the fly. This may take a few seconds depending on the data size and the hardware you use. To use similarity slices in context of processing a chimney cube check the display and look for circular patterns or other anomalous dots.

Display a seismic inline through one of the possible chimney locations (inline viewer). Let us assume that you have indeed identified a chimney on this line. You may want to display one or more attributes in a small window around the chimney (add another inline viewer, position it and resize it) to check how the chimney appears on single-attributes. (To compare different attributes you can add more viewers that can be toggled on/off in the tree, or you can add another scene (Windows menu, option New). You can also use the "Swift" icon to apply attributes from an open attribute set directly to an active element.

Having done this you are now ready to start picking chimneys and non-chimneys. The first action required is to create two new picksets: one for chimneys and one for non-chimneys. Use the right-hand pop-up menu from the picksets entry in the OpendTect tree to do this. Type the name of the pickset you wish to create, e.g. *chimneys_yes* and start picking locations (left-hand mouse button) inside the chimney. Picks can be removed by pressing *Ctrl* and left-hand mouse clicking the pick. Click on the data element in the OpendTect tree to be able to move the element to another position and repeat the process. Repeat this exercise until you have sampled all chimney points you need. Now select the *non-chimneys* pickset from the OpendTect tree and start picking non-chimney locations. When you are done save each pickset in a separate pickset Group (*Store* pop-up menu option from the tree).

Picking strategy: Picking example locations is the key step in this procedure. You should aim to create representative sets for both chimneys and non-chimneys. Try not to limit yourself to one chimney if there are more chimneys in your data set. Try to sample these consistently and over as wide a time-range as possible.

The default chimney attribute set is tuned to find chimneys that are characterized by energy variations, low trace-to-trace similarity and chaotic reflection patterns (variations in local dip). The default attribute set extracts attributes in 3 separate windows of 80ms that are aligned above, around and below the evaluation point. This arrangement will help the network to distinguish between a vertical disturbance (of approx. 240ms) and a local disturbance that is not vertical, hence should be classified as non-chimney.

So, if you use the default chimney set, you should select example chimney locations inside zones where the energy and the reflection pattern are disturbed over a vertical zone. non-chimneys should be picked at locations where you see good reflectors but also at locations of non-vertical disturbance (e.g. at faults). A typical set for training consists of several hundred to a few thousand points.

Neural Network Training

The next step is the training of a neural network. Click the corresponding icon or start the **network module** from the Processing menu. Create a New neural network, and select the attributes you wish to use (normally all) and the pickset groups containing the chimney and non-chimney locations. In general not all locations are used to train the network but a percentage (10 to 20) of the examples is used to avoid overfitting the network. The network will extract the attributes of your choice at the locations you specified, it will randomly split the data into train and test sets and it starts the training phase.

Training performance is tracked during training and presented in two figures. The normalized RMS error curves indicate the overall error on the train and test sets, respectively on a scale from 1 (maximum error) to 0 (no error). Both curves should go down during training. When the test curve goes up again the network is overfitting. Training should be stopped preferably before this happens. Typically an RMS value in the 0.8 range is considered reasonable, between 0.8 and 0.6 is good, between 0.6 and 0.4 is very good and below 0.4 is excellent.

A better feel for the network's performance is presented in the lower figure, which shows the percentage mis-classified for train and test sets as a function of training cycles.

Finally, you will notice that the nodes of the network change colors during training. The colors indicate how important each node (each input attribute) is for the classification. The colors run from red (most important) via yellow to white (least important).

Overtraining: Overfitting occurs when a network starts to recognize individual examples from the training set. The network performs better on the training set, but the performance on the test set decreases. For optimal results network training is stopped when the performance on the test set is maximum (minimum error). The point to stop can be seen from the performance graphics in the network training window.

Evaluation and Application of the Trained Neural Network

Now that you have a trained neural network, you are ready to create TheChimneyCube®. Before applying it to an entire volume, you may want to check the result on a few selected planes. Pop up the Select attribute data menu by clicking with the right-hand mouse button on a plane element in the tree. Note that the neural network option is now active. The network has two output nodes: chimney and non-chimney. Select the chimney node to generate the desired display.

When satisfied you can continue by applying the trained network to the entire data volume. This is done in the *Create Volume* module that is launched from the *Processing* menu. Instead of processing an entire cube it is also possible to limit the output range to a sub-volume. To increase speed, run the job on several machines simultaneously in multi machine mode. OpendTect will split the jobs over the specified machines and will combine the output at the end of the processing.

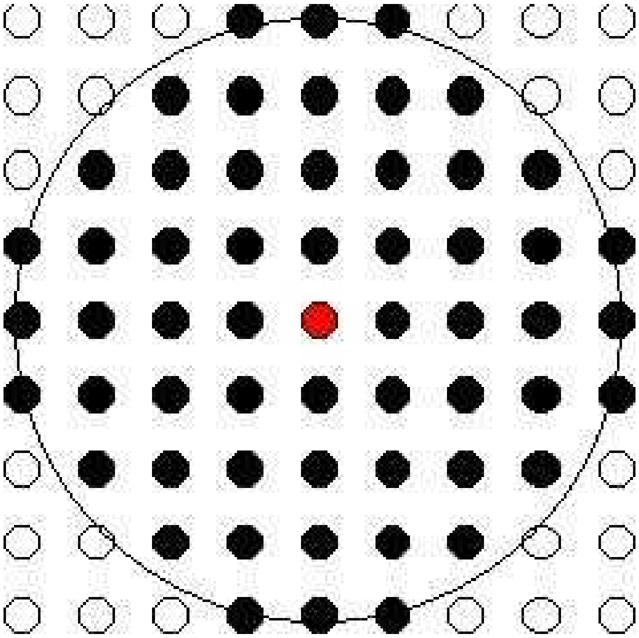
For more details on Chimney analysis workflow, look at : **dGB Tutorial Videos**

Dip-Steered Median Filter

The dip-steered median filter is a data-driven tool that yields a cleaned-up seismic data volume in which coherent events are enhanced and randomly distributed noise is reduced. The filter increases the general interpretability of the seismic data and improves the performance of automatic horizon trackers. Basically the filter collects all amplitudes inside a disc with user-specified search radius and replaces the value at the center by the median value of the amplitudes. The search disk (see graphic below) follows the local dips from the **SteeringCube**.

The filter, in combination with the SteeringCube works as follows:

1. A search radius is defined.
2. From a starting position (red dot) we extract the first amplitude.
3. The local dip and azimuth is followed to the next trace.
4. The interpolated amplitude at this point is extracted.
5. Step 3 and 4 are repeated for all traces inside the search radius (see Figure).
6. The amplitude at the starting position is replaced with the median value of all extracted amplitudes.
7. Points 2 to 6 are repeated for all samples in the cube.



Filter input for a 4-trace radius which corresponds to 57 points. Note that the disk is neither flat, nor horizontal but follows the seismic events from trace to trace.

A median value can be defined as the value associated with the central position of a ranked series. So, if we rank all N amplitudes from smallest to largest number than we find the median value by taking the value at position (N+1)/2, where N is an odd number. To understand the effect of a median filter, let us assume we are filtering a seismic event with a 3-points median filter. The event, e.g. the amplitudes along a horizon is given by the following series:

...0,0,1,0,0,1,1,3,1,0,1,1,1,.......

The 3-points median filtered response is given by:

....0,0,0,0,0,1,1,1,1,1,1,1,1,.......

To check this take 3 consecutive input numbers, rank them and output the value in the middle, then slide your input set one position and repeat the exercise.

Please observe that:

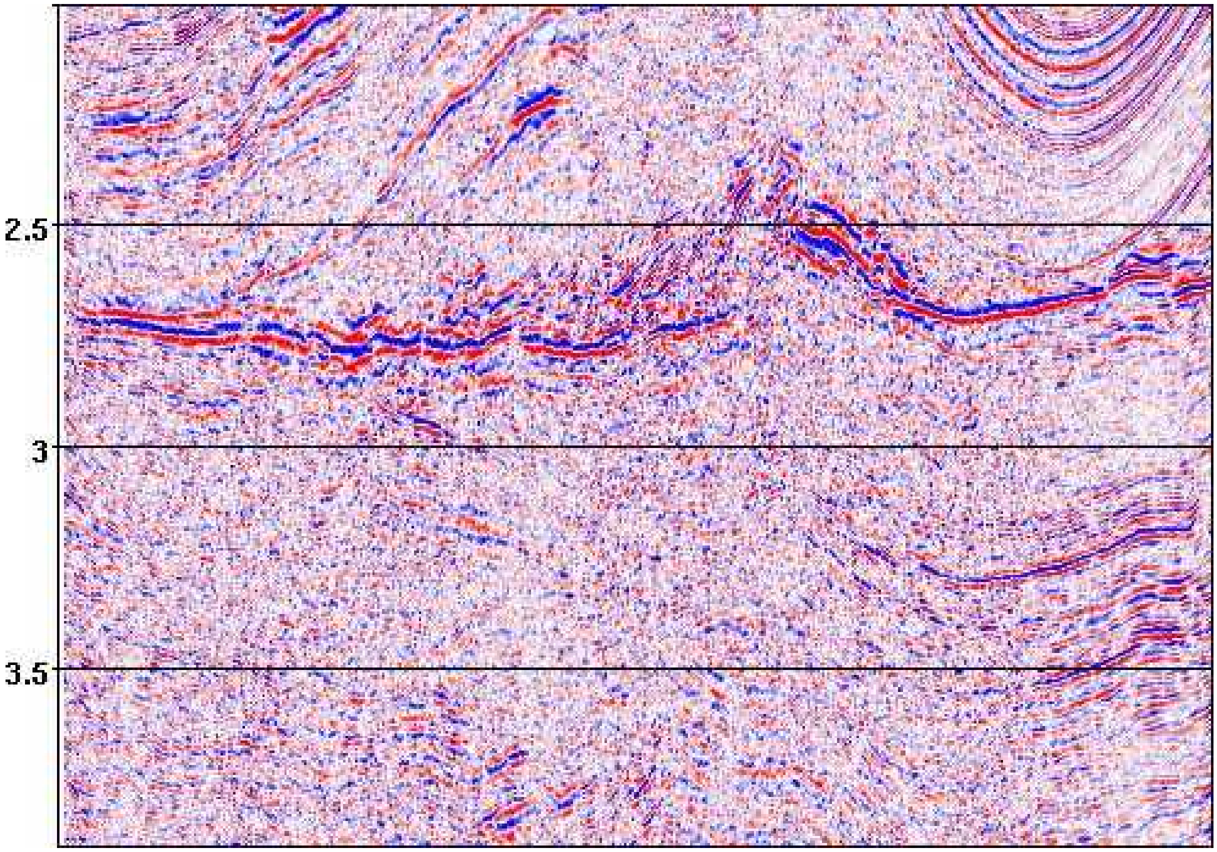
1. Events smaller than half the filter length are removed (e.g. the 1 on the left and 0 on the right)

2. Noise bursts are also removed (the value 3)

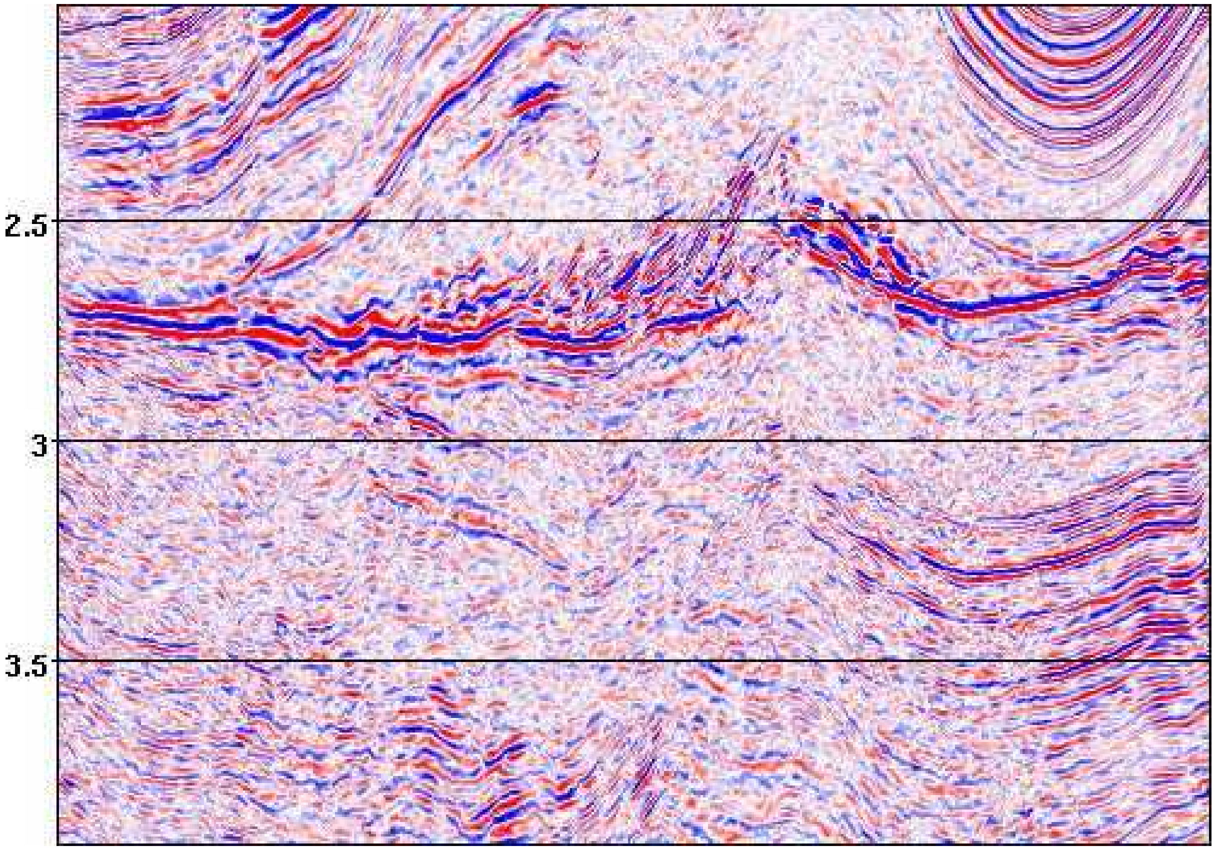
3. Edges are preserved (the break from mainly zeros to mainly ones stays exactly at the same position. In other words no filter tails are introduced).

Example Results

The following figures show an example of the effect of a 57-points (4 trace radius) filter.



Original seismic.



Seismic section from a 3D volume after 57-points dip-steering-median-filtering.

Create a Dip-Steered Median Filter

In OpendTect, filters are integrated with attribute extractions. The advantage of this approach is that all attributes can be filtered separately without creating intermediate results first. To apply a median filter to a seismic data set, you define the median filter as an attribute in the *active attribute set*. Start the attribute definition module (icon or Attributes option, Processing menu). Select an existing attribute set, or create a new one. Specify *Volume statistics* as attribute type, select the input seismic and specify *output* = *median*. Set the search radius e.g. 4x4 and specify the time gate as [0,0]. Select the *SteeringCube* and specify Full steering as the steering mode. A time-gate of [0,0] means that effectively the filter input is collected along a disk. Full steering means that the disk is curved according to the local dip information (see also Create Steering Data).

To apply the filter interactively, use the plane viewers in one or more scenes. For a good comparison between filtered and unfiltered displays, scale the data similarly (clipping option, right-hand mouse button on the color bar). To apply to a volume, select Volume output from the Processing menu, specify the dip steered median filter attribute as '*Quantity to output*'.

Note

As an alternative, one could also use the *Edge Preserving filter* described by Li You et al. in *The Leading Edge* of February 2002. Just like the dip-steered median filter, this Edge Preserving filter is included in the *Evaluate Attributes* default attribute set. It uses the **Position** attribute to locate the area where the variance in seismic amplitude is lowest, and outputs the average amplitude at that location to the current sample location.

Default Attribute Sets

Table of Contents

- Evaluate Attributes
- dGB Evaluate Attributes
- NN Chimney Cube
- NN Fault Cube
- NN FaultCube Advanced
- NN Salt Cube
- NN Slump Cube
- Unsupervised Waveform Segmentation
- Ridge-Enhancement Filter
- Dip-Steered Median Filter
- Dip-Steered Diffusion Filter
- Fault Enhancement Filter
- Fault Enhancement Attributes
- Seismic Filters Median-Diffusion-Fault-Enhancement

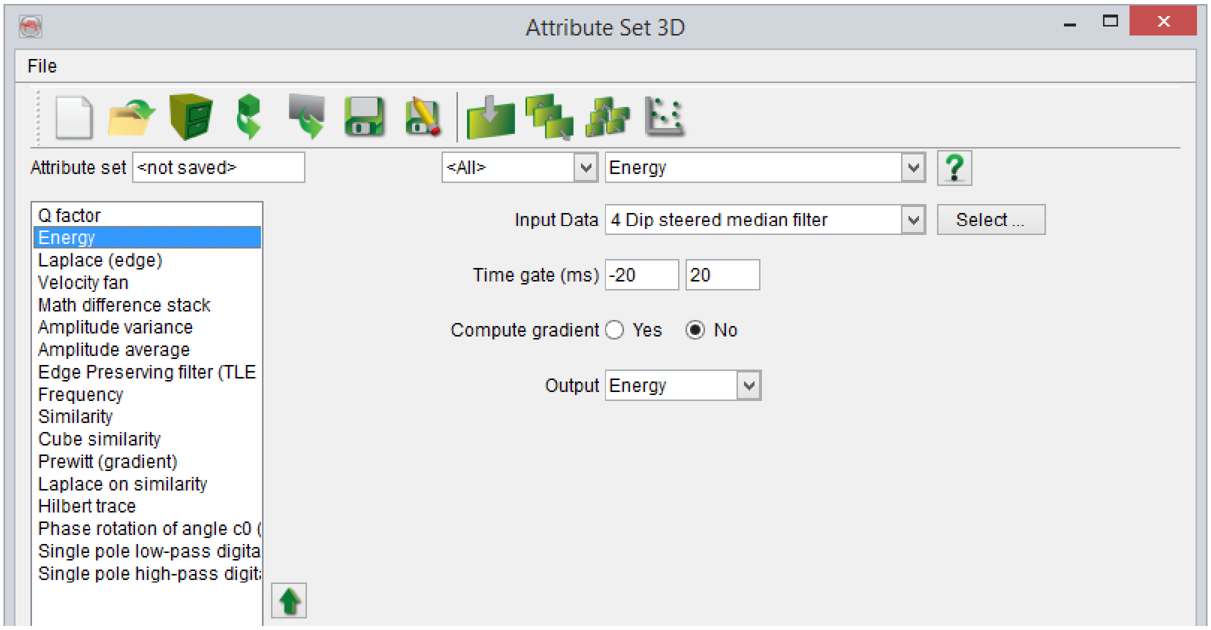
The steering and neural networks plugins for OpendTect are provided with *default attribute sets* to help get the user started. The sets have proven their value in several studies, and generally deliver good results in their respective applications. Attribute sets starting with NN are meant as input for a neural network and are optimized to detect certain geological features (ChimneyCube, SaltCube etc ...), The other attribute sets have different other purposes. Note that all default attribute sets need a SteeringCube.

In general, default sets give satisfactory results and therefore inexperienced user can use the sets without modifications. Experienced users can use the sets as starting point for attribute analysis. Fine-tuning is done by modifying attribute parameters, and/or adding or removing attributes. To give the user a good idea about the applicability of the default attribute set, typical examples are provided in the next sections of this appendix. All examples contain a short description of the attribute set and its characteristics, an example of seismic data and the result after applying the default attribute set to this data.

Evaluate Attributes

The "*Evaluate attributes*" is a default attribute set which gives the user the possibility to find the best parameter setting for a particular attribute. To evaluate attributes, use your visual inspection, common sense, seismic knowledge.

Only for this particular default attribute set, the use of dGB plugins (SteeringCube and/or Neural Network) is not needed.



Attribute set <not saved>

Input Data 4 Dip steered median filter

Time gate (ms) -20 20

Compute gradient ○ Yes ● No

Output Energy

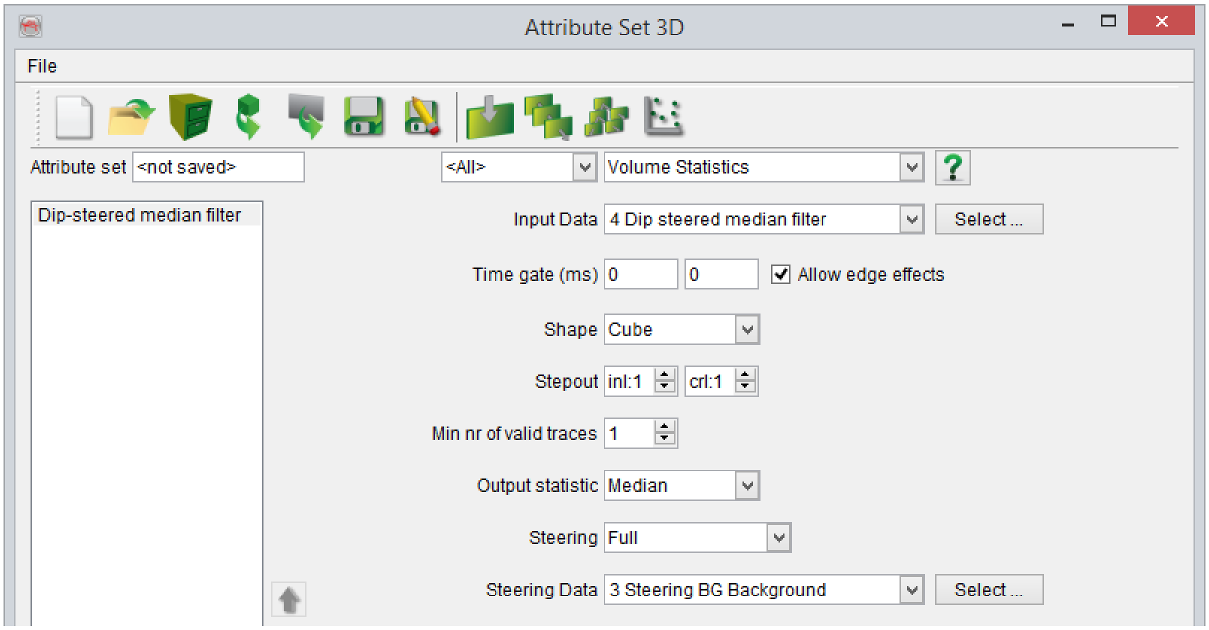
"Evaluate Attribute" default attribute-set

For more details on this topic, please look at **dGB Tutorial videos**

dGB Evaluate Attributes

The "*dGB Evaluate Attributes*" contains a general selection of various attributes, including steered attributes (polar dip, similarity, curvature etc.) and filters (dip-steered median filter and edge preserving filter). It is intended as a guide or starting point for a scan through the wide range of different attributes, and may function as a starting point for a custom made attribute set.

The difference with "Evaluate Attributes" is that dGB Evaluate Attributes use dGB plugins (with SteeringCube).



Attribute set <not saved>

<All> Volume Statistics

Dip-steered median filter

Input Data 4 Dip steered median filter

Time gate (ms) 0 0 ☑ Allow edge effects

Shape Cube

Stepout inl:1 crl:1

Min nr of valid traces 1

Output statistic Median

Steering Full

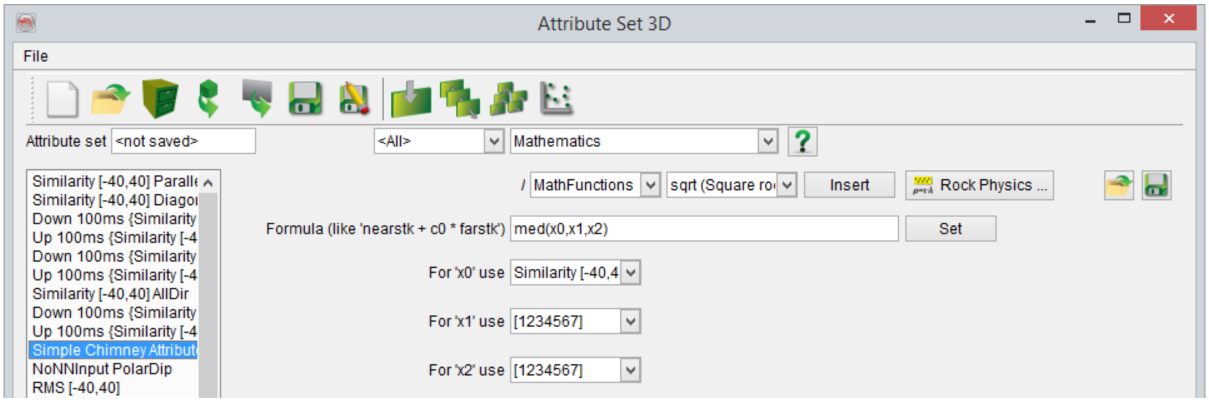
Steering Data 3 Steering BG Background

"dGB Evaluate Attribute" default attribute-set

NN Chimney Cube

This attribute set is meant for usage in a neural network. A key feature of this set is that most attributes are extracted in three separate time windows: one above, one centered around, and one below the point of investigation. In this way, we utilize the fact that chimneys are vertical bodies with a certain dimension. It is expected that similar seismic characteristics of Chimneys are present in all three windows and thus a correlation exists between these three windows. The neural network will recognize this and will thus be able to distinguish between real chimneys and other (more localized) features.

In this section, we compare the original seismic in Figure 1 with the results of chimney detecting neural network displayed in Figure 3. The main body of the chimney and its sidetracks are picked up, while other features (faults, low similarity/low energy layers) are rejected. In addition we compare a similarity attribute in Figure 2 with the neural network results in Figure 3. The similarity attribute highlights the chimney, but also other (unwanted) features are enhanced. The multi-attribute neural network is able to make a clear distinction between chimney and non-chimney. Neural networks with the NN ChimneyCube attribute set as input can (and in general will) also detect other fluid migration paths, e.g. dewatering structures.



Attribute set <not saved>

<All>

Mathematics

Similarity [-40,40] Paralle
Similarity [-40,40] Diago
Down 100ms {Similarity
Up 100ms {Similarity [-4
Down 100ms {Similarity
Up 100ms {Similarity [-4
Similarity [-40,40] AllDir
Down 100ms {Similarity
Up 100ms {Similarity [-4
Simple Chimney Attribut
NoNNInput PolarDip
RMS [-40,40]

/ MathFunctions

sqrt (Square ro

Formula (like 'nearstk + c0 * farstk') med(x0,x1,x2)

For 'x0' use Similarity [-40,4

For 'x1' use [1234567]

For 'x2' use [1234567]

"Chimney Cube" default attribute-set

Figure 1.Chimney in seismic data

Figure 2. The similarity attribute.

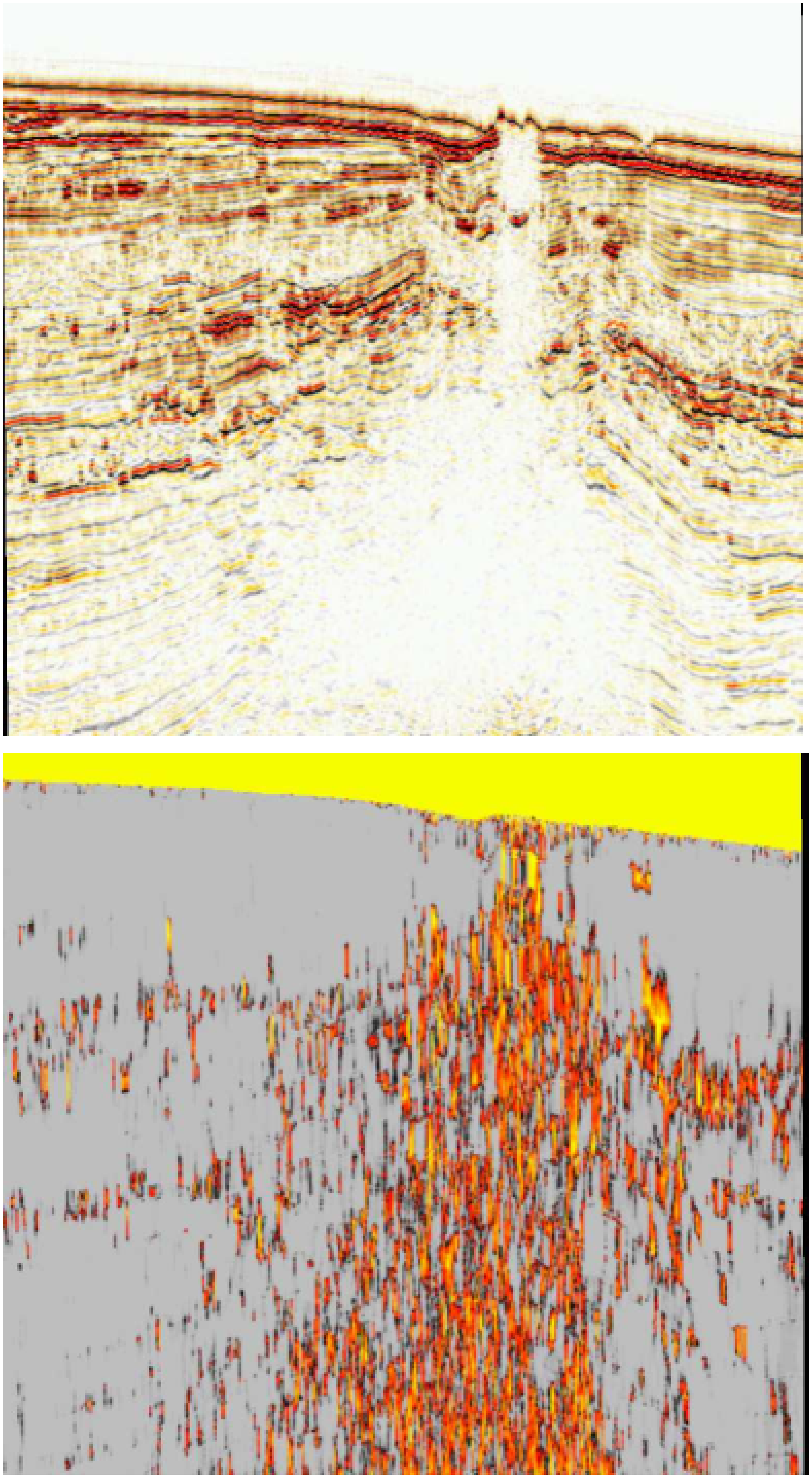
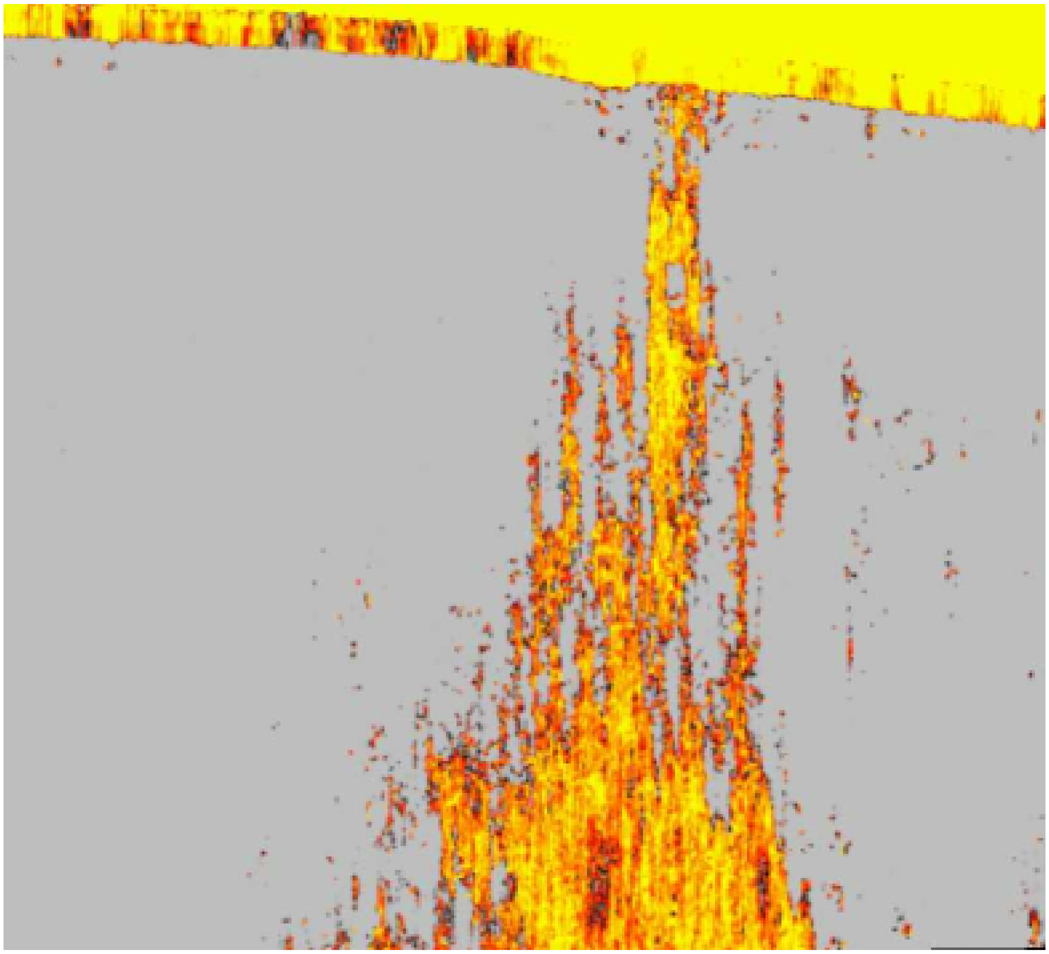


Figure 3. Result after applying a neural network with the NN Chimney Cube as input.

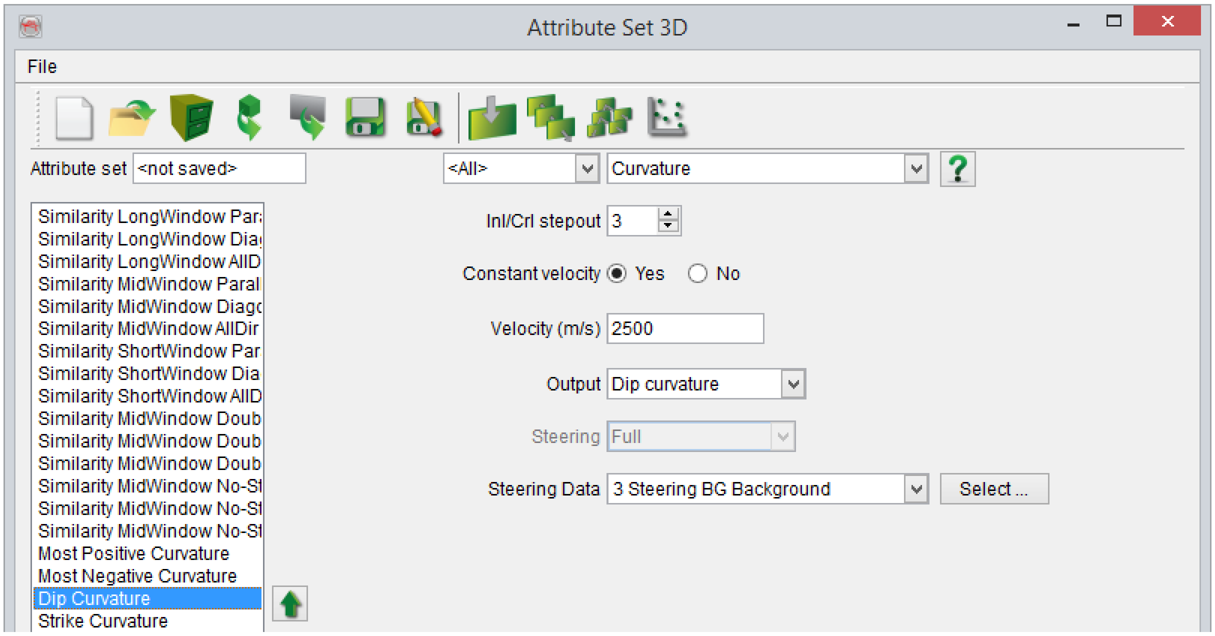


For more details on chimney analysis workflow, look at : **dGB Tutorial videos**

NN Fault Cube

This attribute set is meant for usage in a neural network. The attributes are tuned to pick up larger and smaller lateral discontinuities in the data. Depending on the character of faults on seismic, the parameters of the attributes can be modified. The defaults provide the best detection of steeply dipping faults of 1 to 3 traces wide. With wider faults or faulted zones longer windows and larger step-outs may improve the results. More flat lying faults are better detected using smaller (vertical) windows

Figure 4 shows seismic data with steeply dipping faults and Figure 5 shows the neural network generated fault cube result. Similarity is the most important attribute in fault detection, but the other attributes (energy, polar dip, dipvariance) enable the neural network to distinguish between faults and other low similarity features, e.g. chimneys or salt layers (provided enough counter-examples are picked for training). Also, they increase the fault continuity. For example, the more chaotic character of seismic data at a fault location is detected by the dipvariance attributes; at a fault location, local dip may vary much more from sample to sample even if the (steered) similarity remains high, and this is exactly what the dipvariance attributes detect. Sometimes the NN fault cube also tends to pick up the acquisition footprint of a survey. A good extension to the NN fault cube default set is adding one or two of the curvature attributes, see **Curvature** for further explanation on curvature.



Attribute set <not saved>

<All> Curvature

- Similarity LongWindow Par
- Similarity LongWindow Dia
- Similarity LongWindow AllD
- Similarity MidWindow Paral
- Similarity MidWindow Diag
- Similarity MidWindow AllDir
- Similarity ShortWindow Par
- Similarity ShortWindow Dia
- Similarity ShortWindow AllD
- Similarity MidWindow Doub
- Similarity MidWindow Doub
- Similarity MidWindow Doub
- Similarity MidWindow No-St
- Similarity MidWindow No-St
- Similarity MidWindow No-St
- Most Positive Curvature
- Most Negative Curvature
- Dip Curvature
- Strike Curvature

Inl/Crl stepout 3

Constant velocity ● Yes ○ No

Velocity (m/s) 2500

Output Dip curvature

Steering Full

Steering Data 3 Steering BG Background

"Fault Cube" default attribute-set

Figure 4. Faults in seismic data.

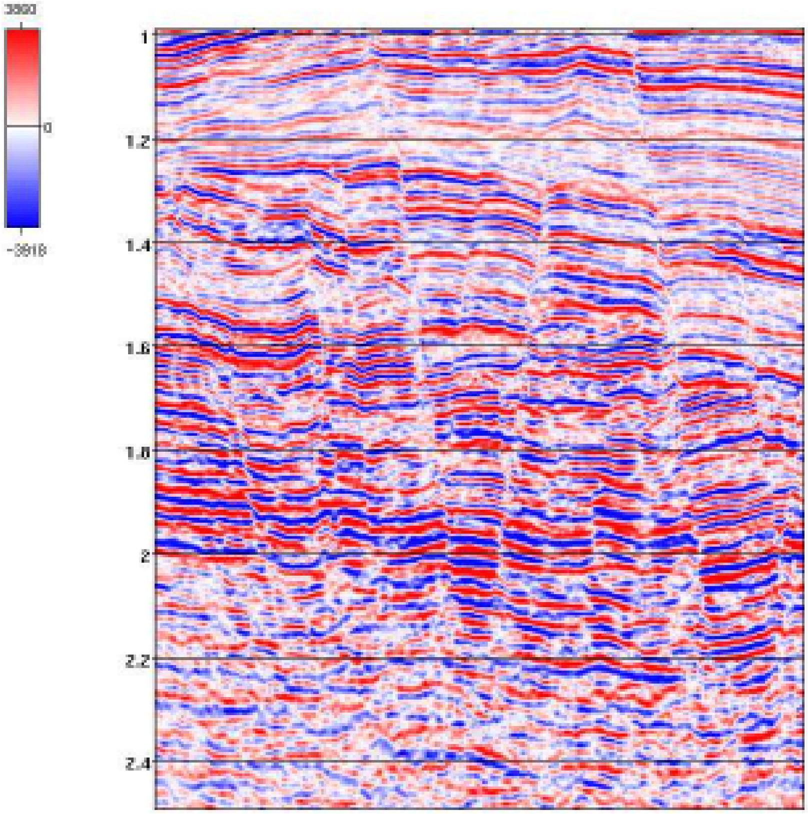
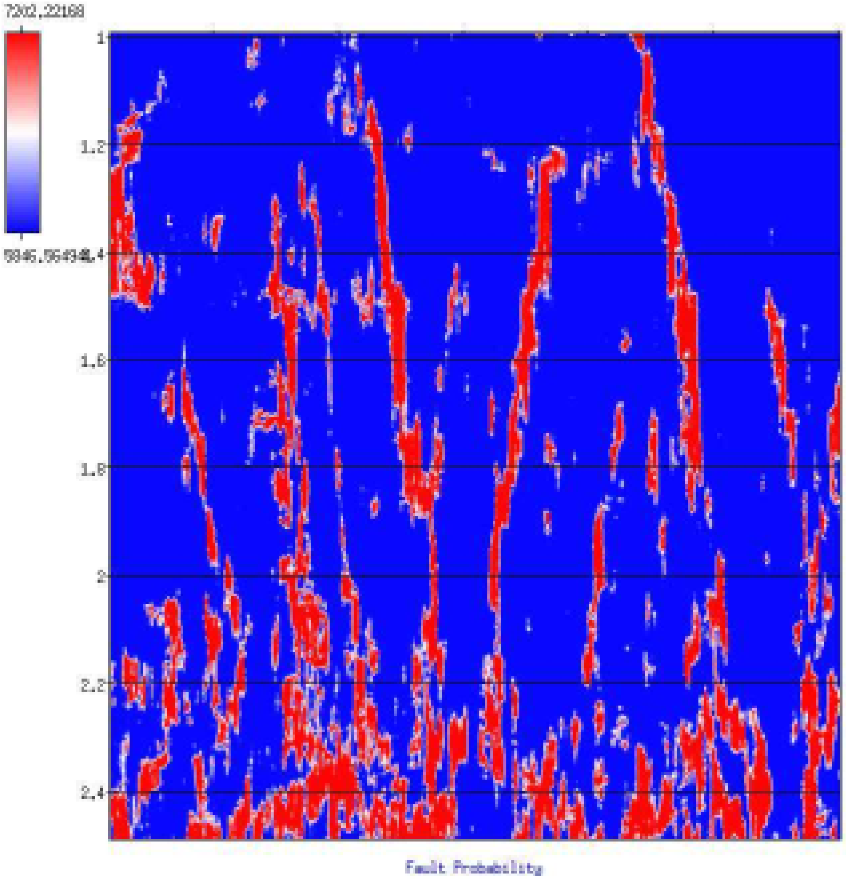
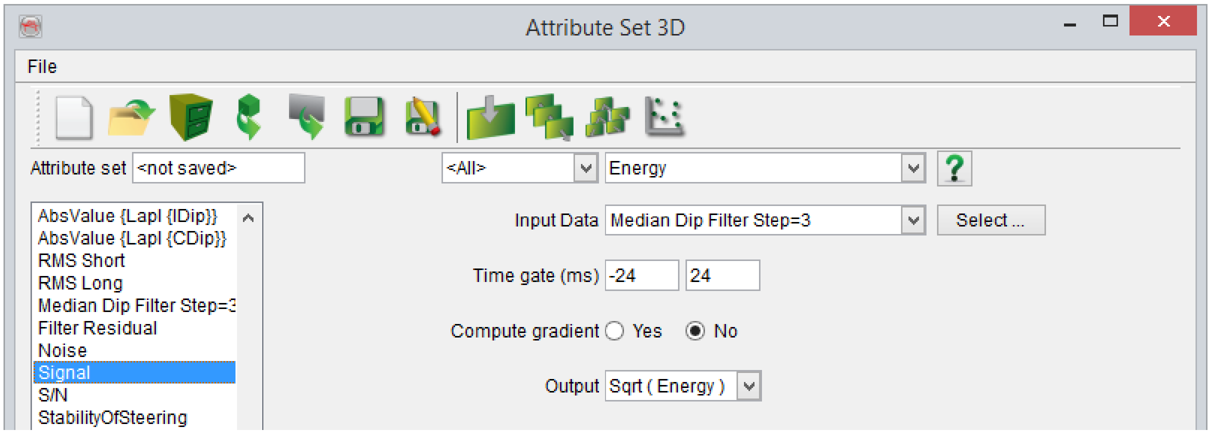


Figure 5. Result after applying a neural network with the NN Fault Cube as input. The output probability is re-scaled from 0-1 to 0-10000 and then clipped between 5850 and 7200.



NN Fault Cube Advanced

Using this default attribute-set, the user can create a Fault "probability" Cube for fault interpretation with more advanced parameters/attributes.



Attribute set <not saved>

<All> Energy

AbsValue {Lapl {IDip}}
AbsValue {Lapl {CDip}}
RMS Short
RMS Long
Median Dip Filter Step=3
Filter Residual
Noise
Signal
S/N
StabilityOfSteering

Input Data Median Dip Filter Step=3

Time gate (ms) -24 24

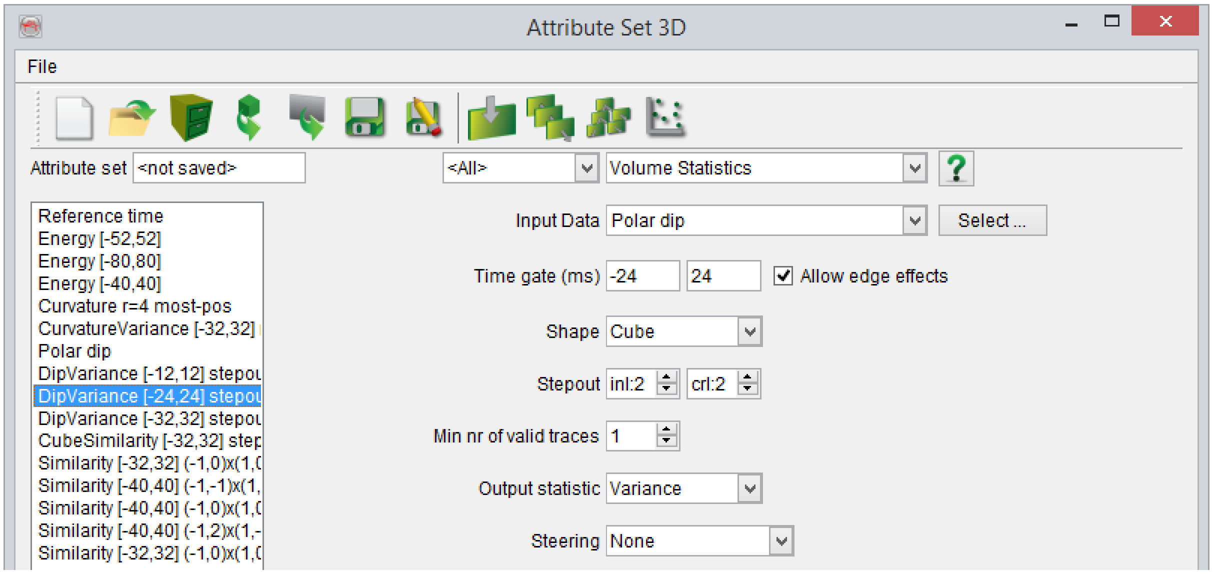
Compute gradient ○ Yes ● No

Output Sqrt (Energy)

"Fault Cube advanced" default attribute-set

NN Salt Cube

This attribute set is meant for usage in a neural network. It is focused on detecting the generally chaotic, low-energy character of salt on seismic data. Because salt has many appearances, there are several possibilities for optimization. For example, one can focus on salt layers (horizontally oriented) or salt domes (vertically oriented). For layers, you can consider to decrease vertical window lengths while for domes the vertical window lengths can be increased. Adding frequency attributes can also be considered since most salt bodies exhibit a different frequency content as compared to their surroundings.



Attribute set <not saved>

<All> Volume Statistics

Reference time
Energy [-52,52]
Energy [-80,80]
Energy [-40,40]
Curvature r=4 most-pos
CurvatureVariance [-32,32]
Polar dip
DipVariance [-12,12] stepou
DipVariance [-24,24] stepou
DipVariance [-32,32] stepou
CubeSimilarity [-32,32] step
Similarity [-32,32] (-1,0)x(1,0
Similarity [-40,40] (-1,-1)x(1,
Similarity [-40,40] (-1,0)x(1,0
Similarity [-40,40] (-1,2)x(1,-
Similarity [-32,32] (-1,0)x(1,0

Input Data Polar dip

Time gate (ms) -24 24 ☑ Allow edge effects

Shape Cube

Stepout inl:2 crl:2

Min nr of valid traces 1

Output statistic Variance

Steering None

"SaltCube" default attribute-set

In general, the NN SaltCube tends to pick up both salt layers and salt diapirs. In the example below, the bottom of the salt is very well resolved.

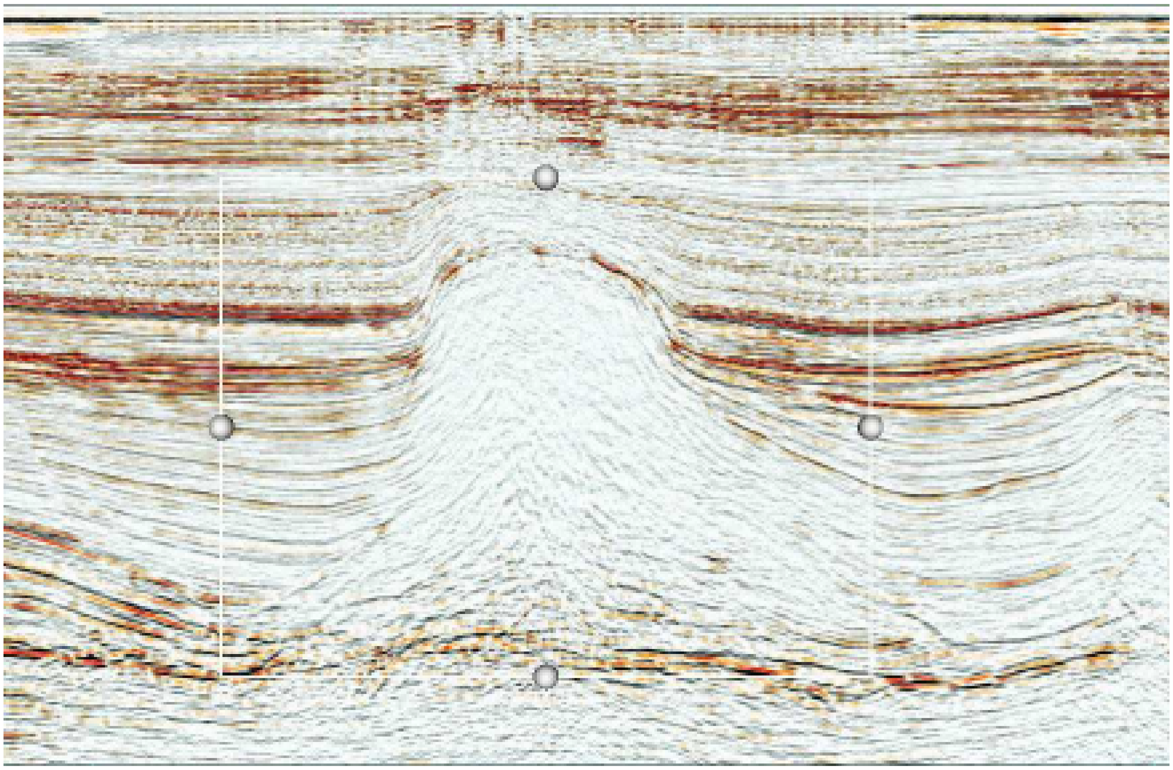


Figure 6. Salt in seismic data

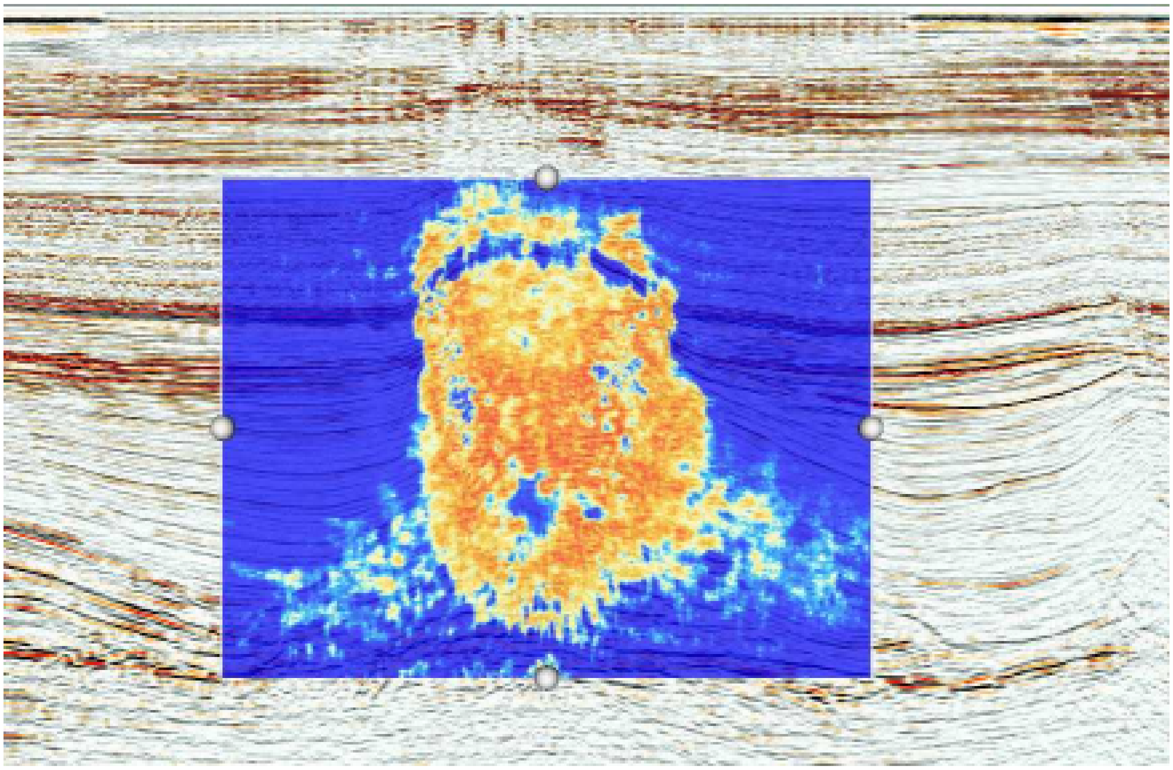
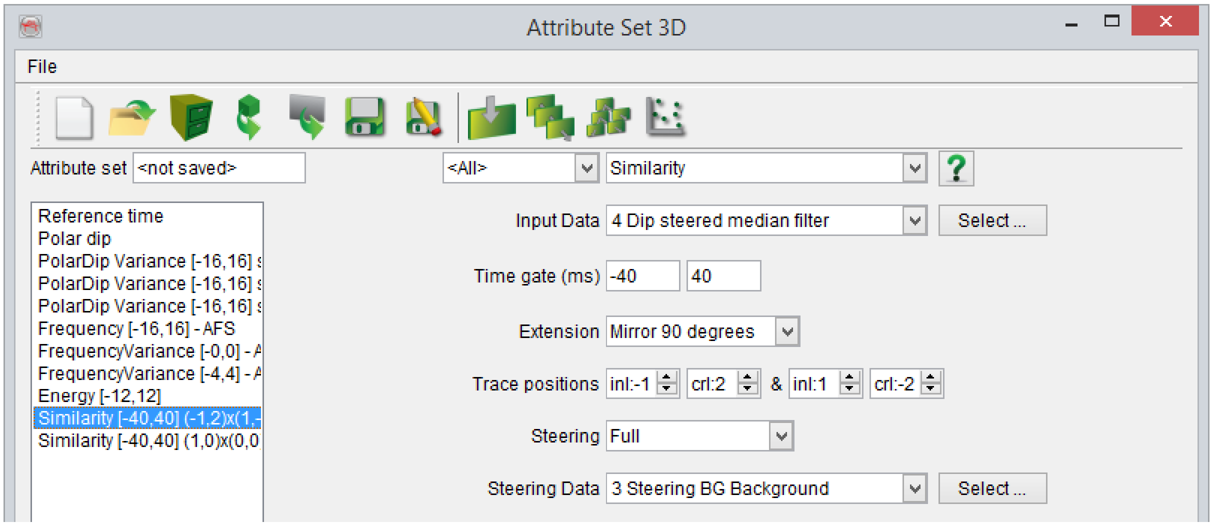


Figure 7. Result after applying a neural network with the NN SaltCube as input. The salt probability is displayed as overlay over the seismic data, with blue indicating low salt probability and yellow to red indicating a high salt probability.

NN Slump Cube

This attribute set was created by Andrew Wilson of *BG International* for quickly mapping turbidite slumps. Slumps are characterized by chaotic reflection patterns and frequency losses; therefore, tops and bottoms are difficult to map using conventional techniques. In Figure 8 a conventional view of the seismic data (left) and slump detection results (right) is shown, and in Figure 9 the slump detection results are presented as a 3D body.



Attribute set <not saved>

<All> Similarity

Reference time
Polar dip
PolarDip Variance [-16,16] s
PolarDip Variance [-16,16] s
PolarDip Variance [-16,16] s
Frequency [-16,16] - AFS
FrequencyVariance [-0,0] - A
FrequencyVariance [-4,4] - A
Energy [-12,12]
Similarity [-40,40] (-1,2)x(1,-
Similarity [-40,40] (1,0)x(0,0

Input Data: 4 Dip steered median filter

Time gate (ms): -40 40

Extension: Mirror 90 degrees

Trace positions: inl:-1 crl:2 & inl:1 crl:-2

Steering: Full

Steering Data: 3 Steering BG Background

"SlumpCube" default attribute-set

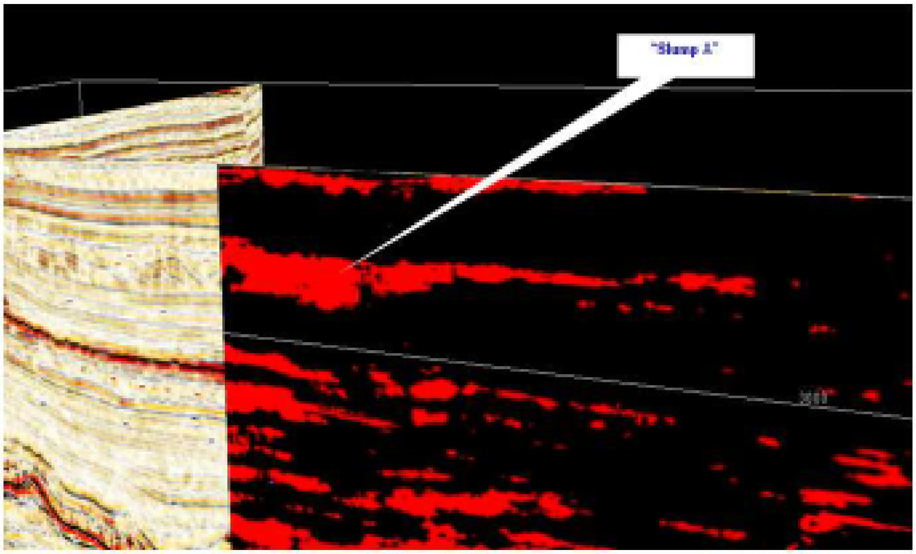


Figure 8: Left seismic data with slumps, on the right the result of the neural network with the NN Slump Cube attribute set as input. Courtesy Andrew Wilson BG.

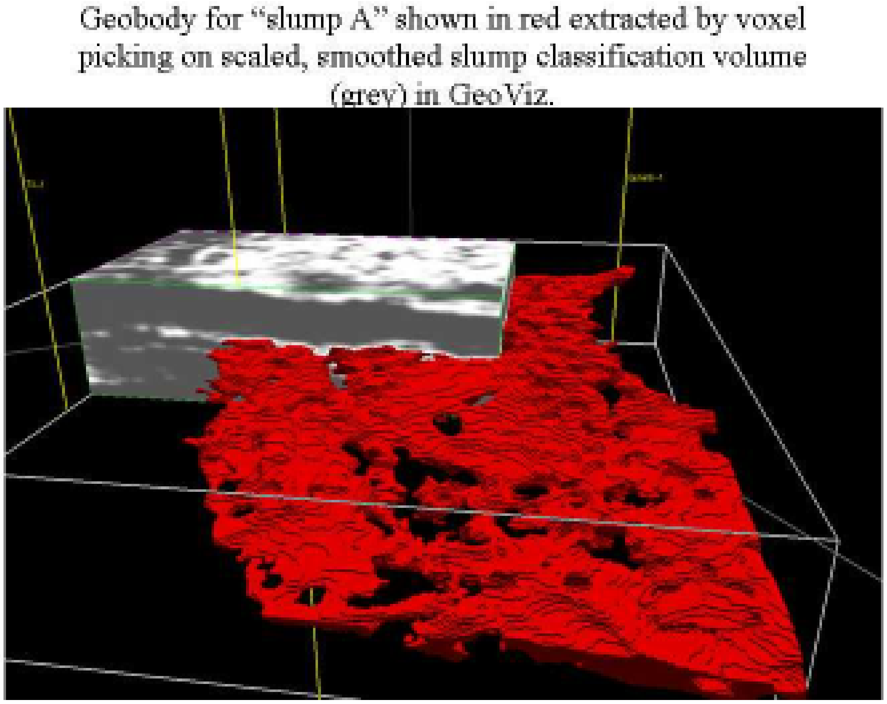
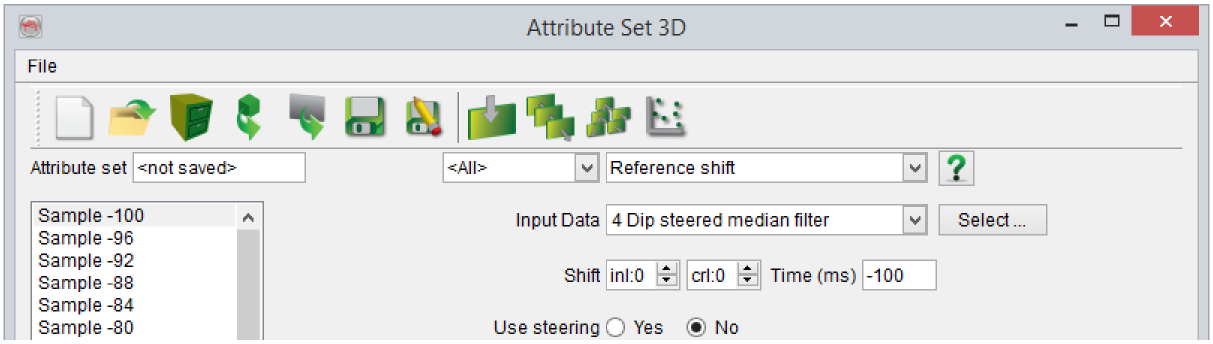


Figure 9: Result after applying a neural network with the NN Slump Cube as input. The slump volume is displayed as a three dimensional body. Courtesy Andrew Wilson BG.

Unsupervised Waveform Segmentation

This attribute set contains a number of samples from the seismic data volume above and below the sample position. The set of samples describes the seismic waveform, and can be used in horizon based unsupervised segmentations. The workflow is as follows: 1) create a set of random picks along the horizon (Pickset menu), 2) train a UVQ network on examples extracted at the random pick locations (use (part of) the waveform as input), and 3) apply the trained network to the horizon (horizon menu).



Attribute set <not saved>

<All> Reference shift

Sample -100
Sample -96
Sample -92
Sample -88
Sample -84
Sample -80

Input Data 4 Dip steered median filter

Shift inl:0 crl:0 Time (ms) -100

Use steering ○ Yes ● No

"UVQ" default attribute-set

A horizon may cover a number of geological (sedimentary) environments. Generally each geological environment will generate a specific seismic response. An unsupervised neural network learns to segment the different seismic responses into different classes. Operating in this way channels, sand bars, bright spots, and other geological bodies might be detected. Note that the default set should not be used for segmentation of volumes because the input changes dramatically when we modify the extraction time position. For segmentation of 3D bodies, you should use phase-independent attributes (e.g. energy, similarity etc). In Figure 8-10 an example of horizon based segmentation in 8 classes is shown.

The default set should be modified such that the sample rate of the attribute set corresponds with the sample rate of the data, and that the sampled window covers the seismic response of the level of interest. In the default attribute set the segmentation is based on the waveform; another approach would be to segment on basis of a number of attributes such as energy, frequency, etc.

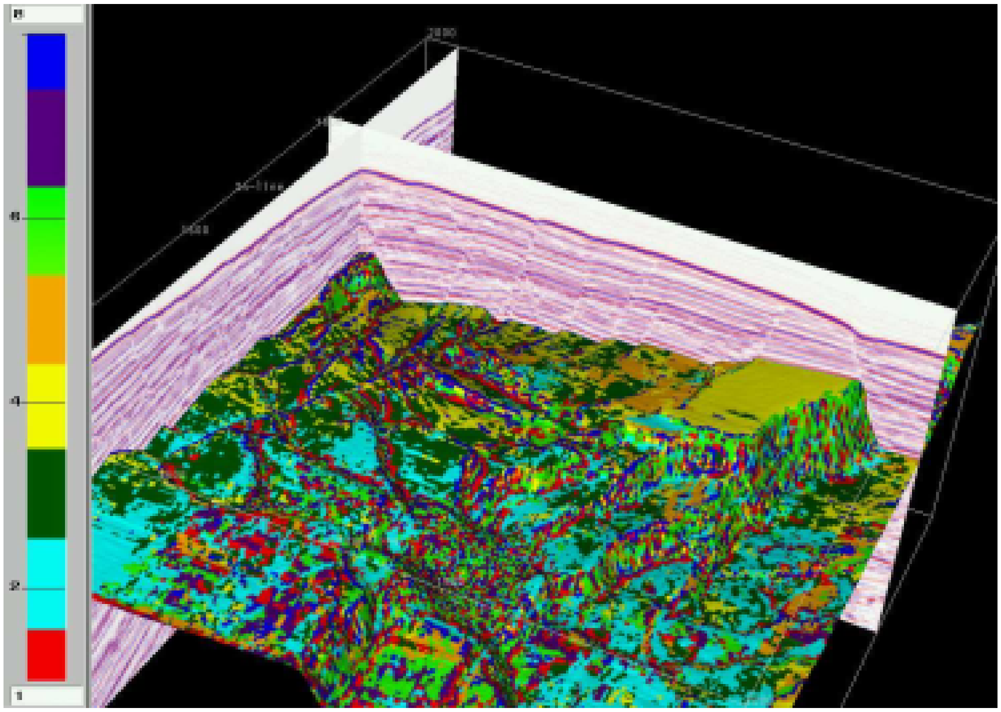


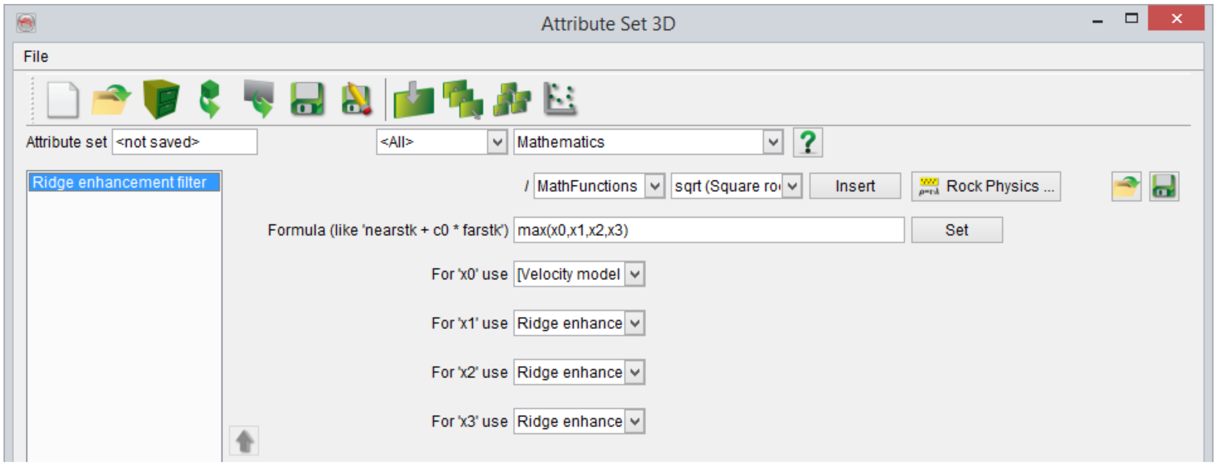
Figure 10. Horizon based unsupervised segmentation

To quickly create, apply and display an UVQ network, use this link: **Quick UVQ**

Ridge-enhancement Filter

This filter sharpen ridges in a similarity cube.

The filter compares, in the time-slice domain, neighboring similarity values in four different directions,(inline direction, crossline, 45 degrees and 135 degrees), then outputs the largest ridge value. The ridge in each direction is the: sum (values on either side) / 2 - center value. In most evaluation points, there are no ridges and the values, thus, tend to be small; but when you cross a fault, there will be a large ridge perpendicular to the fault direction. The filter outputs the largest value, i.e. the ridge corresponding to the perpendicular direction.



Attribute set <not saved>

<All>

Mathematics

Ridge enhancement filter

/ MathFunctions sqrt (Square ro

Formula (like 'nearstk + c0 * farstk') max(x0,x1,x2,x3)

For 'x0' use [Velocity model

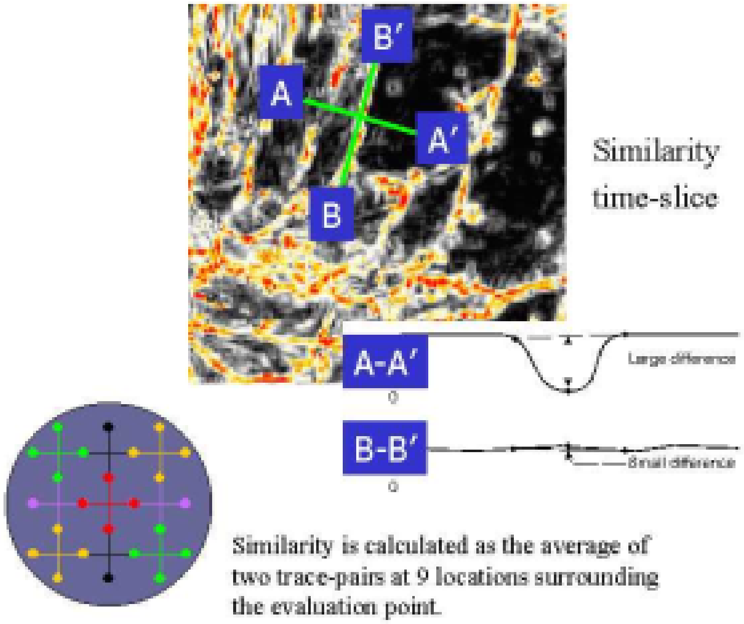
For 'x1' use Ridge enhance

For 'x2' use Ridge enhance

For 'x3' use Ridge enhance

"Ridge Enhancement Filter" Attribute-set

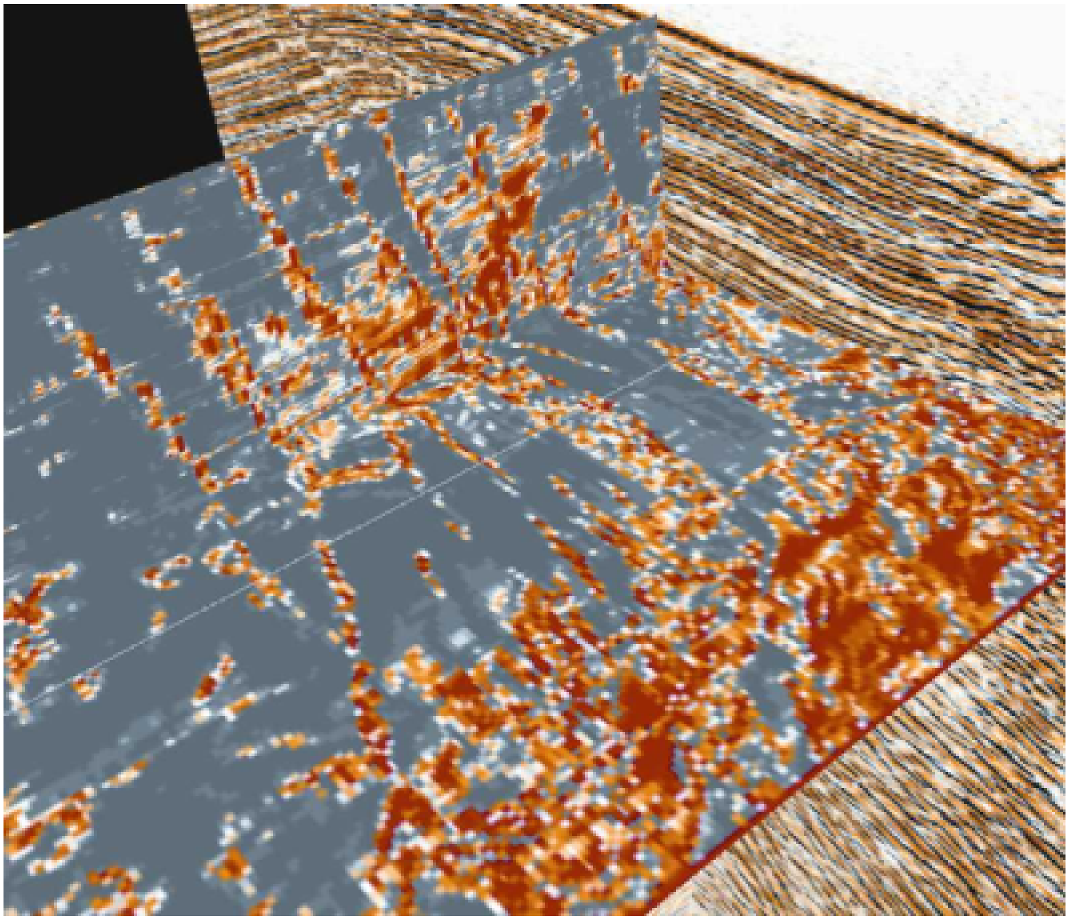
The output of this attribute set is the meta attribute *Ridge-enhancement attribute* at the bottom of the attribute set. All other attributes are intermediate attributes, used in the calculation of the final attribute. The construction of the attribute detects lateral lineaments in time slices of steered similarity, yet ignores bodies of low similarity. The idea behind the ridge-enhancement attribute is explained in figure below.

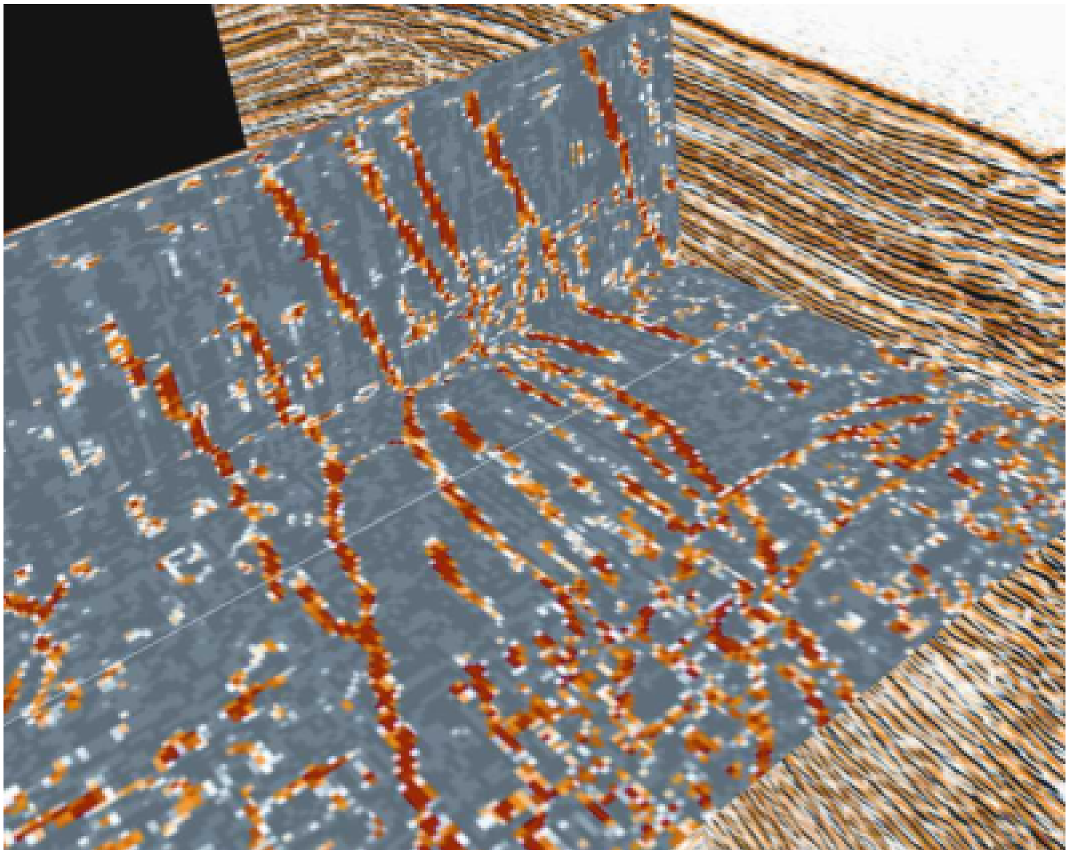


Similarity is calculated as the average of two trace-pairs at 9 locations surrounding the evaluation point.

When we slice through a similarity cube and we cross a fault we observe a large difference in attribute response between the value at the fault position and the values on either side. In the ridge enhancement set we calculate 9 similarity attributes surrounding the evaluation point. We then scan in different directions to find the largest difference, which is the desired output. In following figures the output of the similarity attribute is compared with the output of the ridge-enhancement cube. The bodies of high similarity have disappeared and the faults are also sharper. The users can optimize this attribute by fine tuning the parameters of the similarity attribute such that faults are optimally detected. For example parameters can be adjusted to the width and orientation of the faults, according to: wider faults or faulted zones must have longer windows and larger step-outs. More flat lying faults are better detected using smaller (vertical) windows.

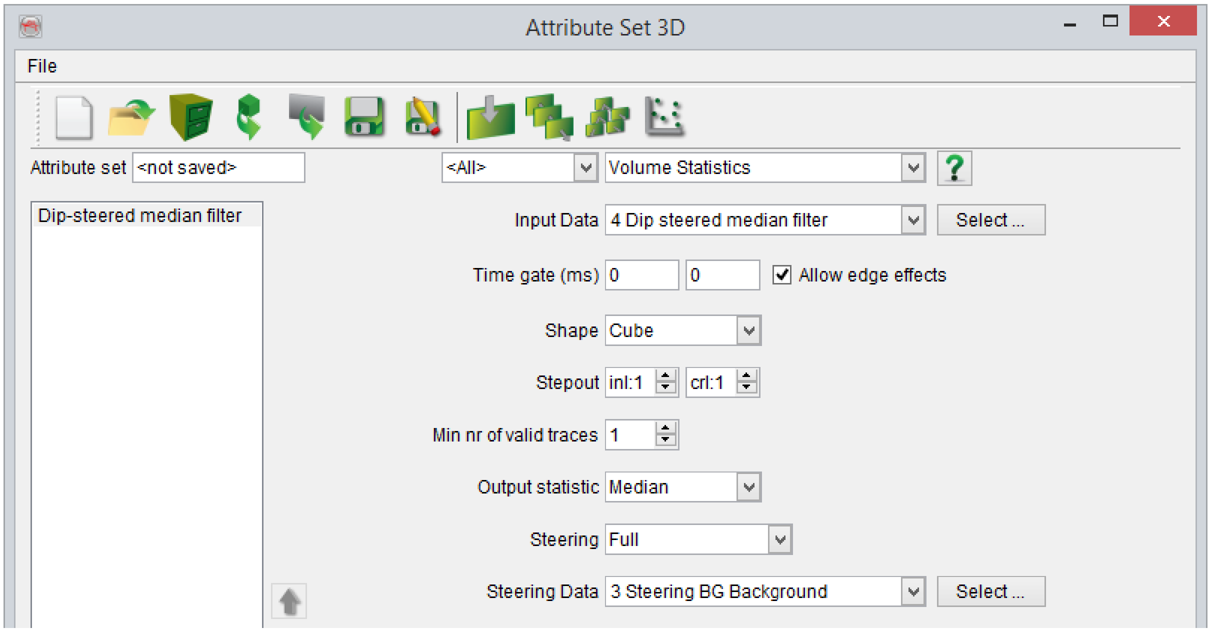
TIP: you can decrease the processing time by almost a factor 9 if you store a similarity cube first and use this as input. Instead of calculating the similarity attribute 9 times on the fly, you calculate and store the similarity once and retrieve it. After you have stored your similarity cube can either trick the system by changing the first attribute (for instance by making it a Mathematics, Formula: x0 and as input your similarirty cube) or by removing the first attribute and changing the attribute input of the following attributes from the removed attribute to your stored similarity cube.





Dip-Steered Median Filter

The *Dip-Steered median filter* removes random noise and enhances laterally continuous seismic events by filtering along the structural dip. In median filtering, the center amplitude in a dip-steered circle is replaced by the median amplitude within the extraction circle. The effect is an edge-preserving smoothing of the seismic data.



Attribute set <not saved>

<All> Volume Statistics

Dip-steered median filter

Input Data 4 Dip steered median filter

Time gate (ms) 0 0 ☑ Allow edge effects

Shape Cube

Stepout inl:1 crl:1

Min nr of valid traces 1

Output statistic Median

Steering Full

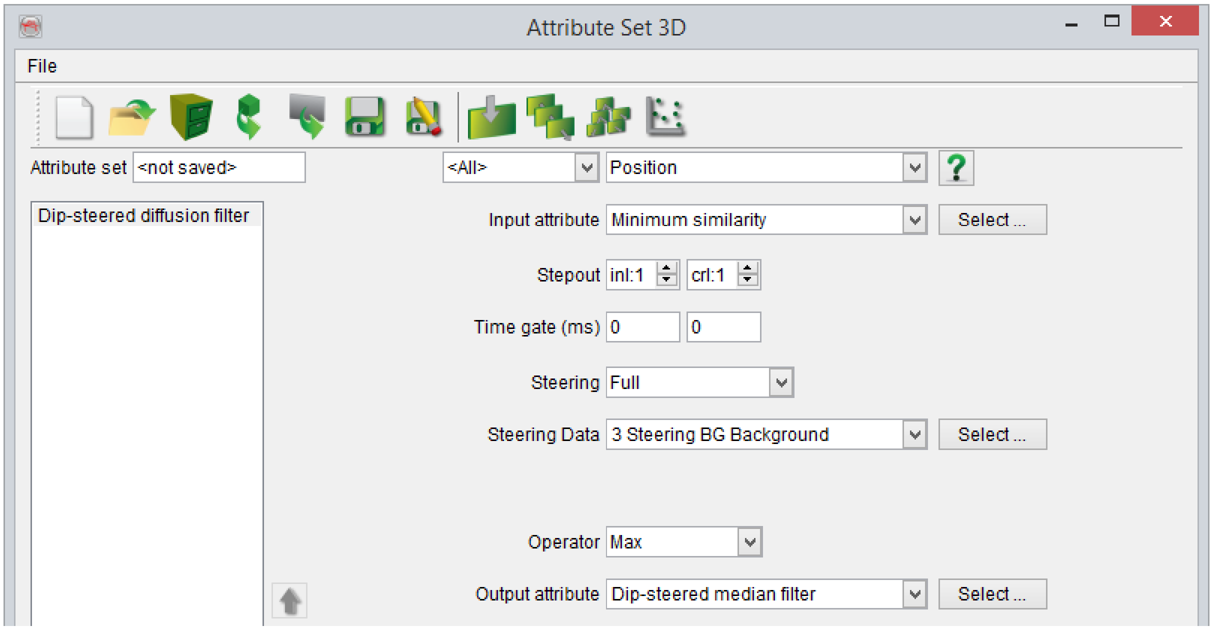
Steering Data 3 Steering BG Background

"Dip-Steered median filter" default attribute-set

For more details on workflow and tips, look at **dGB Tutorial Videos**

Dip-Steered Diffusion Filter

A "*Dip-Steered Diffusion Filter*" is a default attribute set used to sharpen faults in Fault/Fracture analysis. "Position" is the attribute to use; it 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).



Attribute set <not saved>

<All> Position

Dip-steered diffusion filter

Input attribute Minimum similarity

Stepout inl:1 crl:1

Time gate (ms) 0 0

Steering Full

Steering Data 3 Steering BG Background

Operator Max

Output attribute Dip-steered median filter

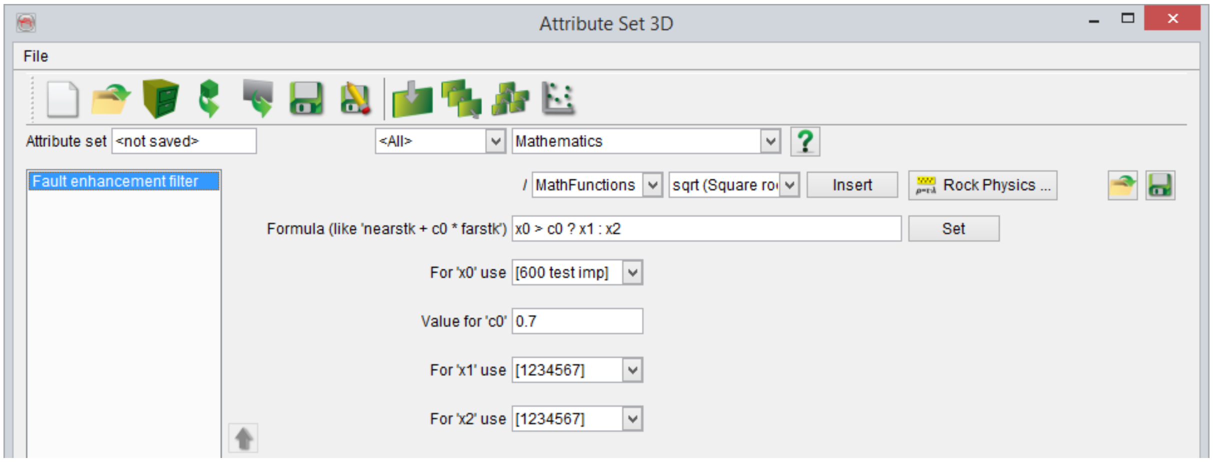
"Dip-Steered Diffusion Filter" default attribute-set

For more informations on "Position" attribute look at : **Position attribute**.

Fault Enhancement Filter

The "*Fault Enhancement Filter*" sharpens edges (faults) by means of median or diffusion filtering, along the structural dip.

In Fault enhancement filtering the quality of the seismic data in a dip-steered circle is evaluated. If the quality is good (Similarity is high) a dip-steered median filter can be applied. If the quality is low (near faults) a dip-steered diffusion filter should be used. The effect is smoothed seismic with sharp fault breaks.



Attribute set <not saved>

<All>

Mathematics

Fault enhancement filter

/ MathFunctions

sqrt (Square ro

Formula (like 'nearstk + c0 * farstk') x0 > c0 ? x1 : x2

For 'x0' use [600 test imp]

Value for 'c0' 0.7

For 'x1' use [1234567]

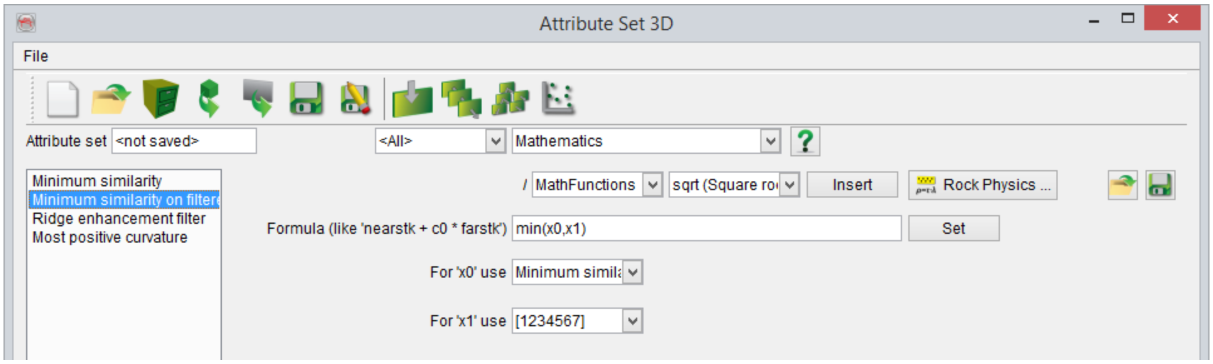
For 'x2' use [1234567]

"Fault Enhancement Filter" default attribute set

For more detail information about workflows and tips look at: **dGB tutorial videos**

Fault Enhancement Attributes

The "*Fault Enhancement Attributes*" default attribute set is a setup of four attributes for Fault/Fracture analysis.



Attribute set <not saved>

<All>

Mathematics

Minimum similarity
Minimum similarity on filtere
Ridge enhancement filter
Most positive curvature

/ MathFunctions

sqrt (Square ro

Formula (like 'nearstk + c0 * farstk') min(x0,x1)

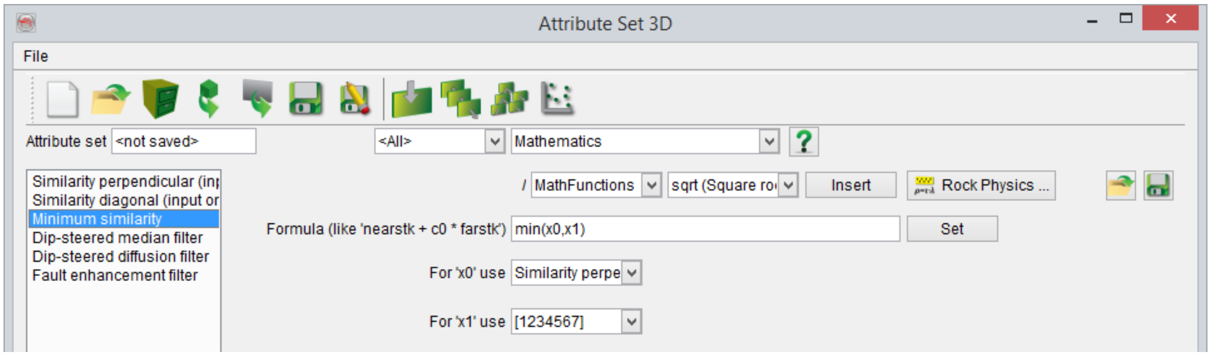
For 'x0' use Minimum simil

For 'x1' use [1234567]

"Fault Enhancement Attributes" default attribute set.

Seismic Filters Median Diffusion Fault Enhancement

This is the same as the "*Fault Enhancement Filter*", but it enables the user to visualize and modify the parameters of the dip-steered median filter, dip-steered diffusion filter, and Fault Enhancement Filter; on the other hand, the "Fault Enhancement Filter" is a "ready to use" default attribute set.



Attribute set <not saved>

<All>

Mathematics

Similarity perpendicular (inp
Similarity diagonal (input or
Minimum similarity
Dip-steered median filter
Dip-steered diffusion filter
Fault enhancement filter

/ MathFunctions

sqrt (Square ro

Formula (like 'nearstk + c0 * farstk') min(x0,x1)

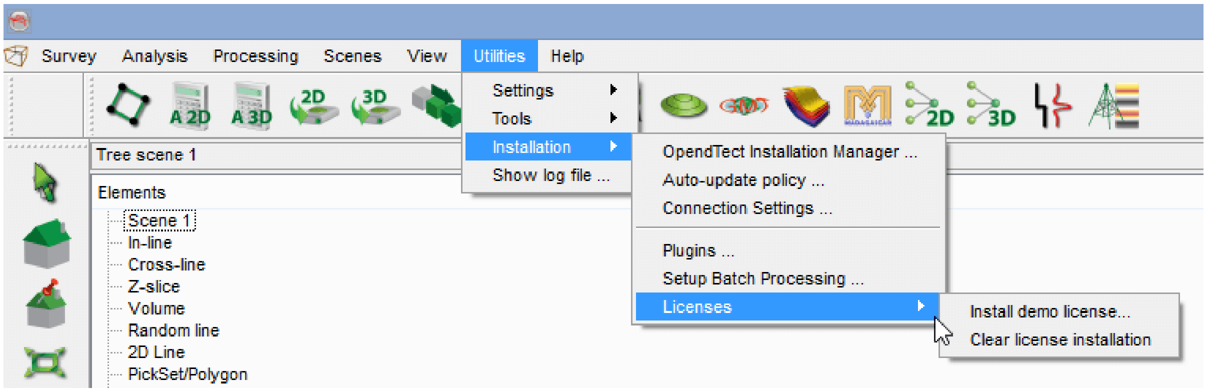
For 'x0' use Similarity perpe

For 'x1' use [1234567]

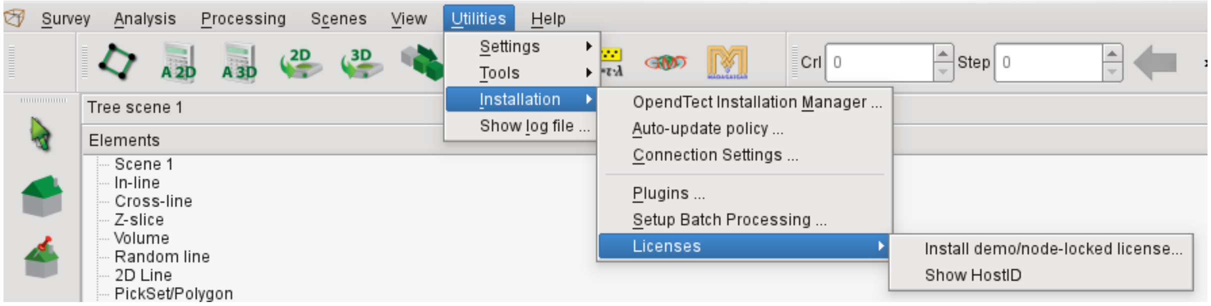
Seismic Filters Median-Diffusion-Fault-Enhancement" default attribute set.

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 Host ID**

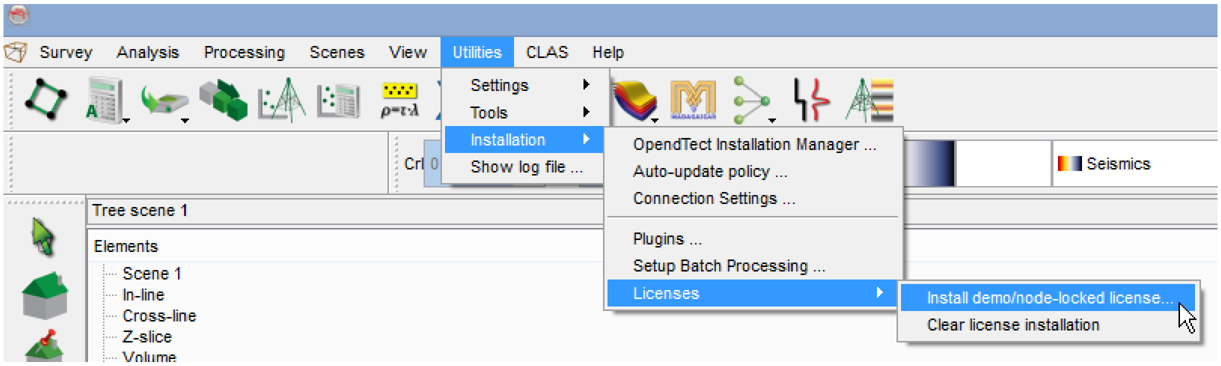
For information about floating or server-based licenses, please refer to: **http://opendtect.org/lic/doc/flexnet_installation_guide.html**

For more general information about OpendTect licensing options, please see: http://opendtect.org/index.php/support/licenses

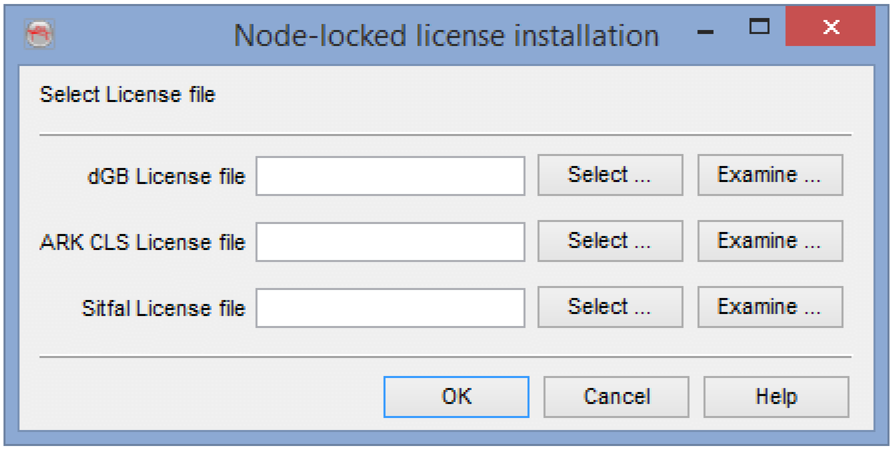
A more complete explanation of OpendTect license Installation can be found in the License Installation Webinar , available on OpendTect's YouTube Channel or via: http://opendtect.org/index.php/support/tutorials/tutorial-videos/webinars

Install demo/node-locked license

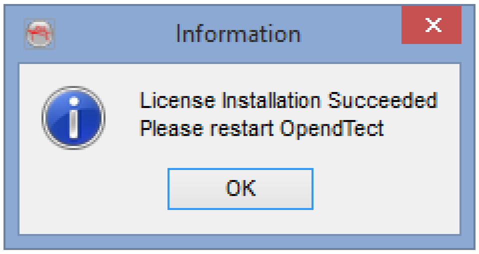
Plugins to OpendTect can be run either by using a license server or by using demo (evaluation) licenses. This second case is case called "node-locked license installation".



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:



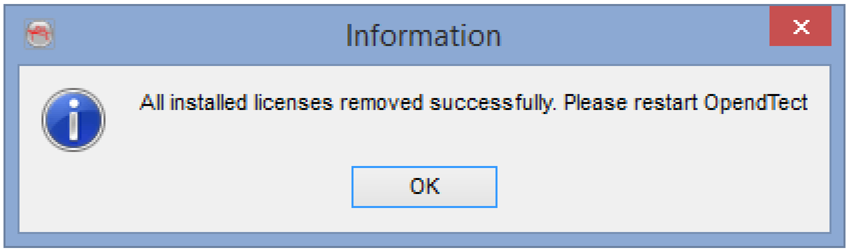
License Installation Succeeded
Please restart OpendTect

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:



All installed licenses removed successfully. Please restart OpendTect

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,

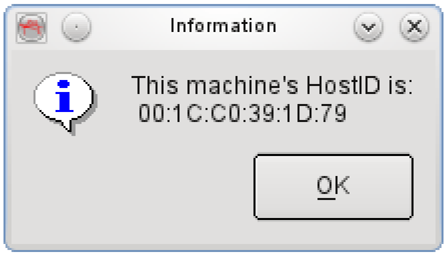
ARKCLS, SITFAL)

- 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.

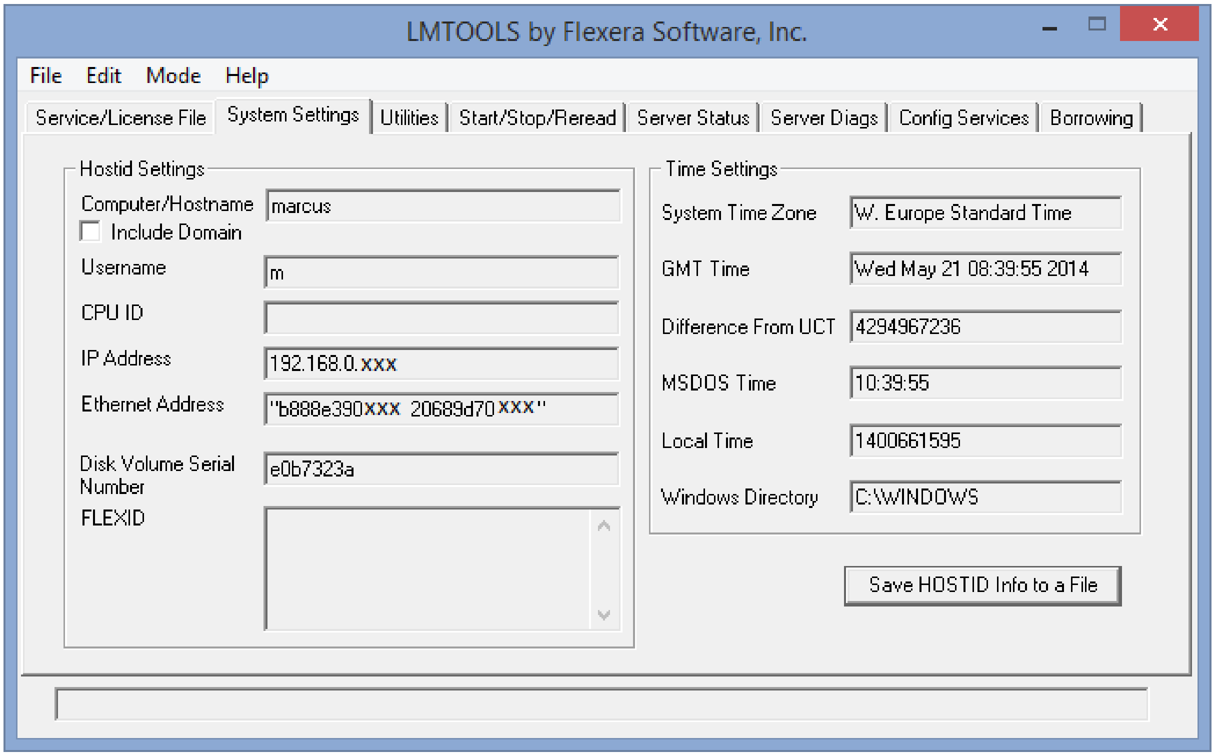
Show HostID

Clicking this option (only available on Linux systems) will pop up a simple dialogue showing the HostID of the machine:



This machine's HostID is:
00:1C:C0:39:1D:79

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):



Hostid Settings

Computer/Hostname: marcus

☐ Include Domain

Username: m

CPU ID:

IP Address: 192.168.0.xxx

Ethernet Address: "b888e390xxx 20689d70xxx"

Disk Volume Serial Number: e0b7323a

FLEXID:

Time Settings

System Time Zone: W. Europe Standard Time

GMT Time: Wed May 21 08:39:55 2014

Difference From UCT: 4294967236

MSDOS Time: 10:39:55

Local Time: 1400661595

Windows Directory: C:\WINDOWS

The option 'Save HOSTID Info to a file' will simply save the information displayed above into a .txt file for reference.

References

- Aminzadeh, F., de Groot, P., Berge, T. and Valentini, G., 2001. Using Gas Chimneys as an exploration tool. Part I - concepts and procedures, Part II - examples. World Oil, May 2001 and June 2001.
- de Groot, P.F.M., Ligtenberg, H., Heggland, R. and Meldahl, P., 2001. Selecting and combining attributes to enhance the detection of seismic objects. 63d. EAGE conference, Amsterdam. 11-15 June 2001.
- Heggland, R., Meldahl, P., Groot, P. and Bril, B., 2000. Detection of seismic chimneys by neural networks - A new prospect evaluation tool. 62nd EAGE conference, 29 May - 2 June 2000. Glasgow.
- Heggland, R., Meldahl, P., de Groot, P. and Aminzadeh, F., 2000. Seismic chimney interpretation examples from the North Sea and the Gulf of Mexico.The American Oil & Gas Reporter, Feb. 2000.
- Heggland, R., Meldahl, P., Bril, B, and de Groot, P., 1999. The chimney cube, an example of semi-automated detection of seismic objects by directive attributes neural networks: Part II; Interpretation. 69th SEG conference, Oct. 31 - Nov . 5, 1999. Houston.
- Marfurt, K.J., and R. L. Kirlin, 2000, 3D Broadband estimates of reflector dip and amplitude: Geophysics, 65, 304-320.
- Meldahl, P., Heggland, R., Bril, B, and de Groot, P., 2001. An iterative method for identifying seismic objects by their texture, orientation and size. SEG international Exposition and 71st. Annual Meeting. San Antonio, Texas, USA. 9-14 Sep. 2001.
- Meldahl, P., Heggland, R., Bril, B, and de Groot, P., 2000. Semi-automated detection

of seismic objects by directive attributes and neural networks, method and application. 62nd EAGE conference, 29 May - 2 June 2000, Glasgow.

- Meldahl, P., Heggland, R., Bril, B, and de Groot, P., 1999. The chimney cube, an example of semi-automated detection of seismic objects by directive attributes neural networks: Part I; Methodology. 69th SEG conference, Oct. 31 - Nov. 5, 1999. Houston.
- Roberts, Andy, 2001, Curvature attributes and their application to 3D interpreted horizons, First Break, 19(2), 85-100, First Break, 19(2), 85-100.

Appendix A - Synthetic Data Generation

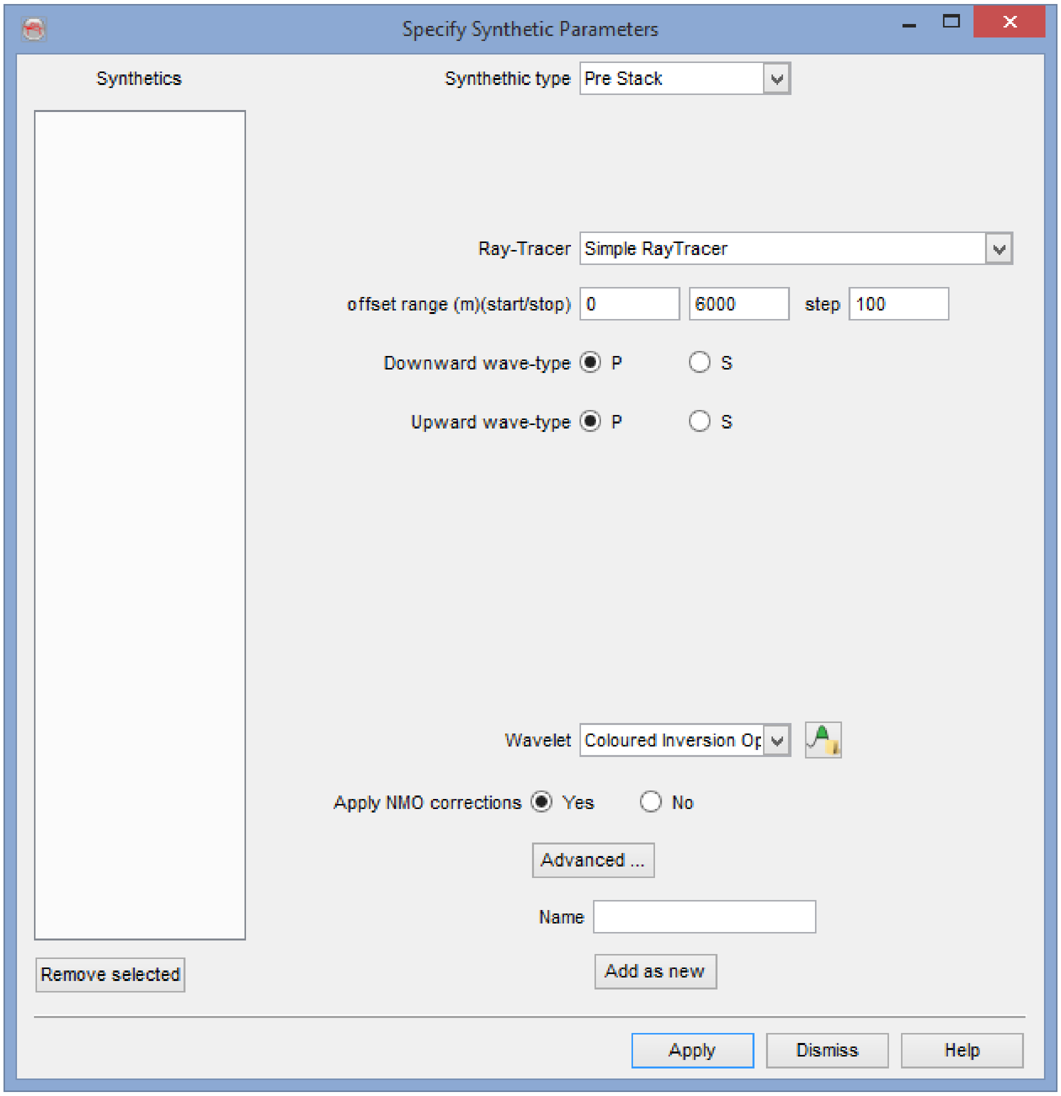
Ray Tracing

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:



Synthethic type: Pre Stack

Ray-Tracer: Simple RayTracer

offset range (m)(start/stop): 0 6000 step 100

Downward wave-type ● P ○ S

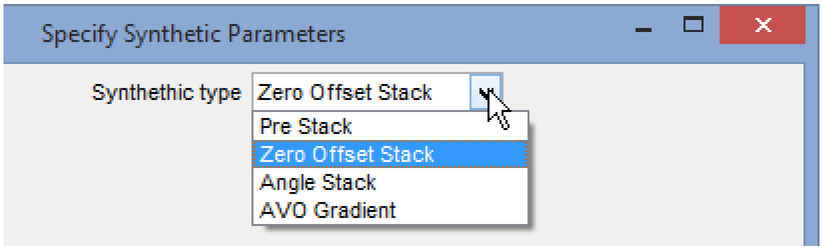
Upward wave-type ● P ○ S

Wavelet: Coloured Inversion Op

Apply NMO corrections ● Yes ○ No

Name:

Here various types of synthetic data can be generated: Zero Offset Stack, Pre Stack gathers, Angle Stack and AVO Gradient:



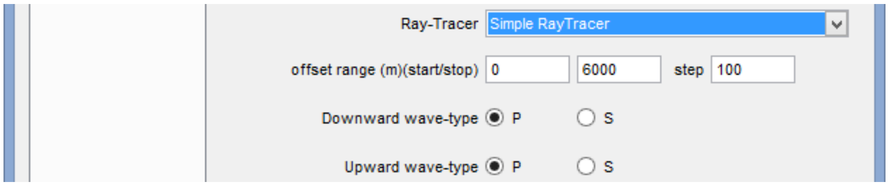
Synthethic type: Zero Offset Stack

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).



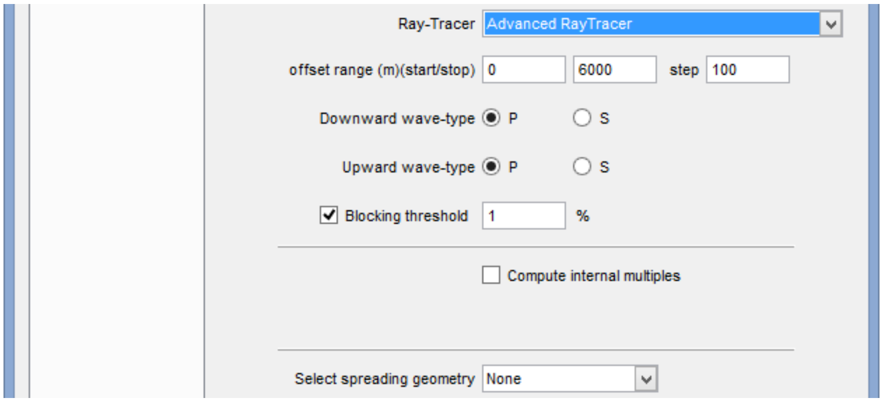
Ray-Tracer: Simple RayTracer
offset range (m)(start/stop): 0 6000 step 100
Downward wave-type ● P ○ S
Upward wave-type ● P ○ S

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" .



Ray-Tracer: Advanced RayTracer

offset range (m)(start/stop): 0 6000 step 100

Downward wave-type: ● P ○ S

Upward wave-type: ● P ○ S

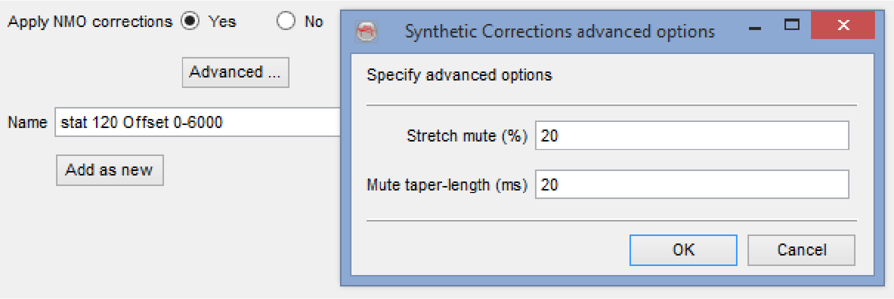
☑ Blocking threshold: 1 %

☐ Compute internal multiples

Select spreading geometry: None

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:



Apply NMO corrections ● Yes ○ No

Name stat 120 Offset 0-6000

Stretch mute (%) 20

Mute taper-length (ms) 20

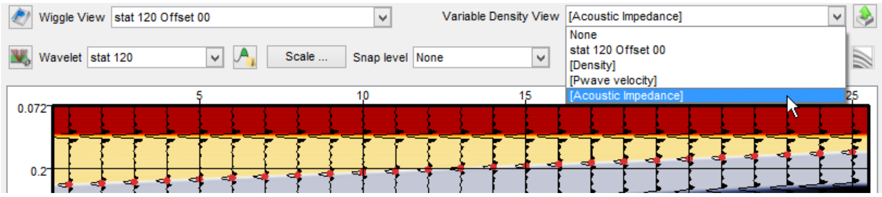
Advanced RayTracer: Advanced corrections options

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:


Wiggle View: stat 120 Offset 00

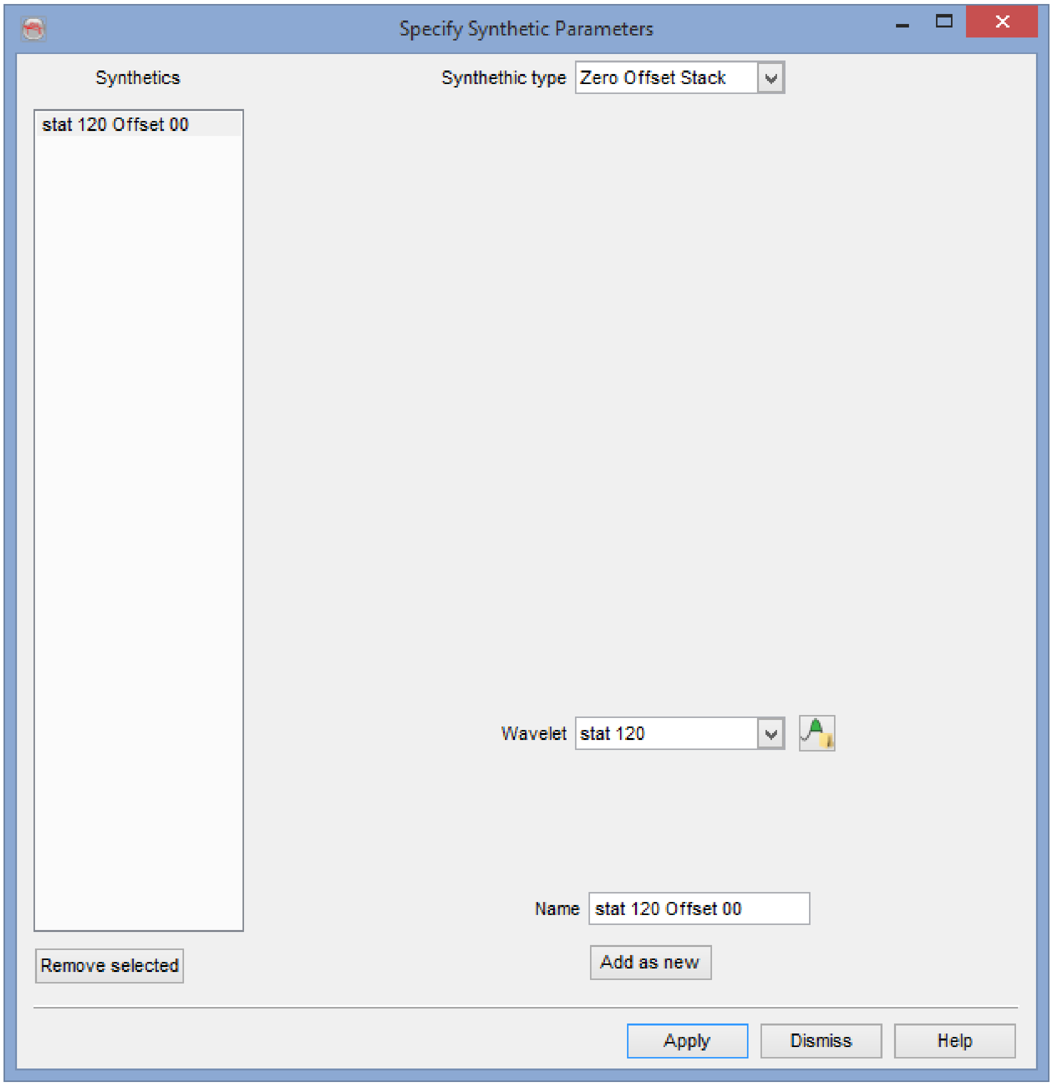
Variable Density View: [Acoustic Impedance]

Wavelet: stat 120

Snap level: None

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:


Synthetics: stat 120 Offset 00

Synthethic type: Zero Offset Stack

Wavelet: stat 120

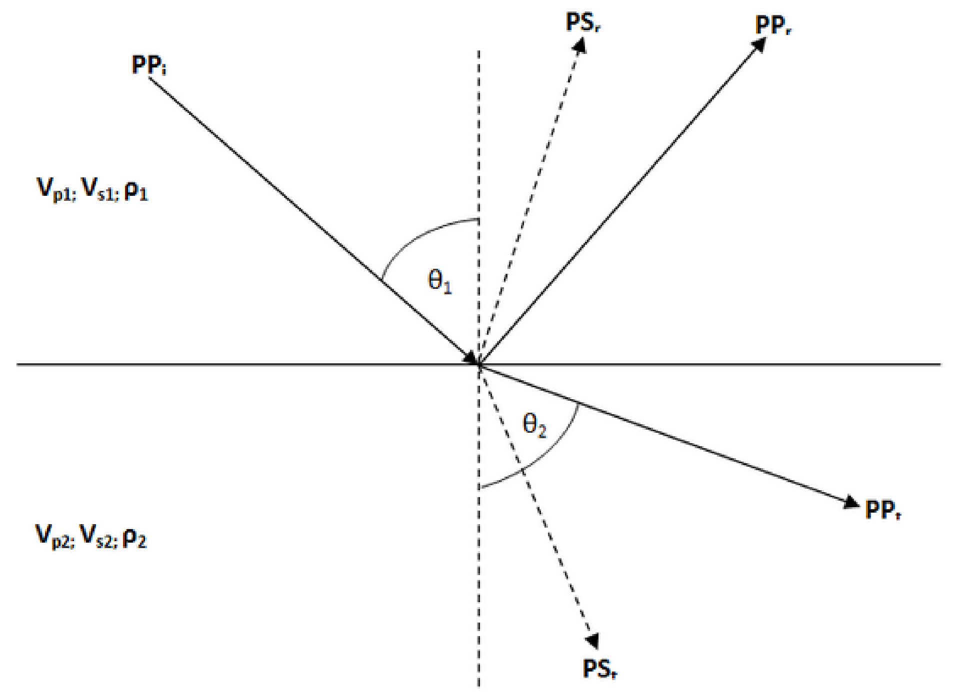
Name: stat 120 Offset 00

Computation of the Reflection Coefficient at any non-zero offset 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:

(above images are from Wikipedia)

$$\begin{bmatrix} R_\mathrm{P} \\ R_\mathrm{S} \\ T_\mathrm{P} \\ T_\mathrm{S} \end{bmatrix} = \begin{bmatrix} -\sin\theta_1 & -\cos\phi_1 & \sin\theta_2 & \cos\phi_2 \\ \cos\theta_1 & -\sin\phi_1 & \cos\theta_2 & -\sin\phi_2 \\ \sin 2\theta_1 & \frac{V_\mathrm{P1}}{V_\mathrm{S1}}\cos 2\phi_1 & \frac{\rho_2 V_\mathrm{S2}^2 V_\mathrm{P1}}{\rho_1 V_\mathrm{S1}^2 V_\mathrm{P2}}\cos 2\phi_1 & \frac{\rho_2 V_\mathrm{S2} V_\mathrm{P1}}{\rho_1 V_\mathrm{S1}^2}\cos 2\phi_2 \\ -\cos 2\phi_1 & \frac{V_\mathrm{S1}}{V_\mathrm{P1}}\sin 2\phi_1 & \frac{\rho_2 V_{P2}}{\rho_1 V_{P1}}\cos 2\phi_2 & \frac{\rho_2 V_{S2}}{\rho_1 V_{P1}}\sin 2\phi_2 \end{bmatrix}^{-1} \begin{bmatrix} \sin\theta_1 \\ \cos\theta_1 \\ \sin 2\theta_1 \\ \cos 2\phi_1 \end{bmatrix}$$

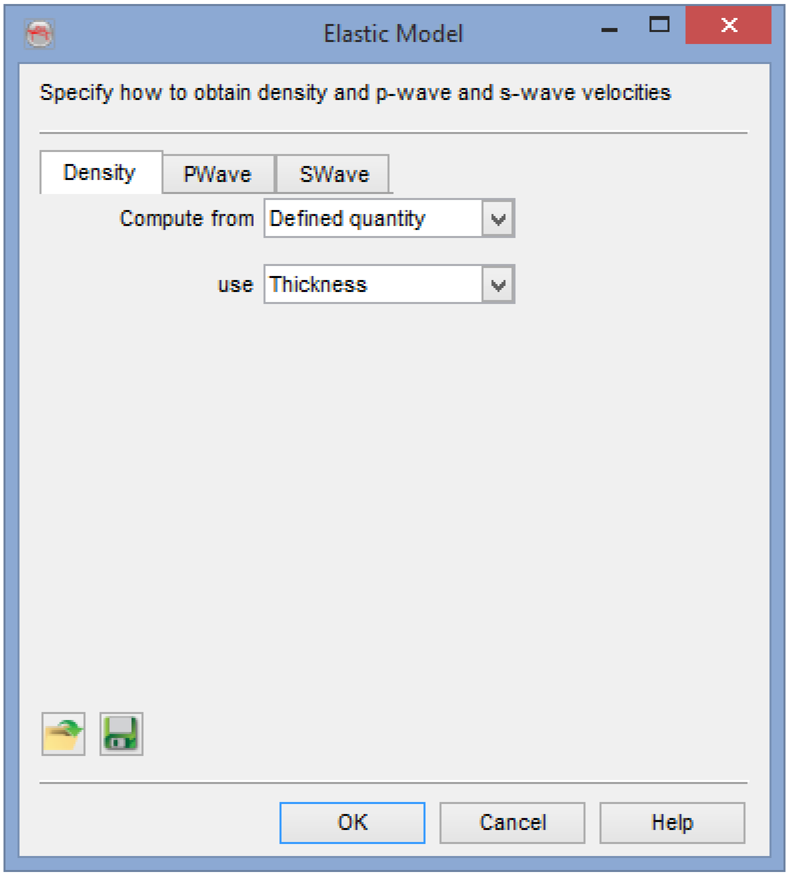


A P-wave (PP_i) propagating through a medium of density, ρ_1, P-wave velocity, V_{p1}, and S-wave velocity, V_{s1}, is incident upon an interface with a medium of density, ρ_2, P-wave velocity, V_{p2}, and S-wave velocity, V_{s2}, at an angle, θ_1. Mode conversions occur resulting in reflected P- and 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:

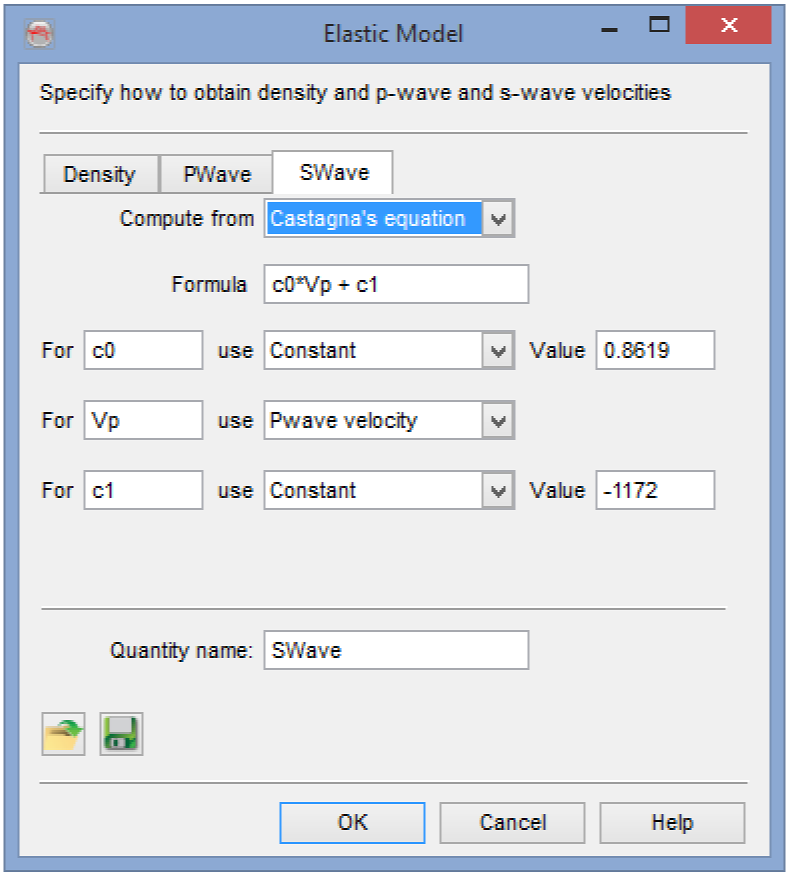


Specify how to obtain density and p-wave and s-wave velocities

Compute from: Defined quantity

use: Thickness

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:



Specify how to obtain density and p-wave and s-wave velocities

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

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

A

Attribute 41, 44, 105, 113, 144, 323, 388

Attribute Sets 373

E

Elastic Model 402

F

FCF 10, 355-357, 362

Fluid Replacement 275

H

HorizonCube 8, 13, 21, 46-47, 49-52, 58-59, 62, 64, 68-69, 72-74, 76, 79, 83-84, 89, 91, 95-99, 101, 105, 110, 114, 116-122, 127, 136, 139-140, 143, 145, 147, 156, 158

N

Neural Networks 8, 48, 149, 291, 293-294

S

SSIS 8, 47, 56, 103, 119, 121-122, 126, 137, 139-140, 144-145, 148, 150, 158

SynthRock 9, 165, 168, 196, 209, 216, 227, 249, 274, 402

V

VMB 10, 324-325, 339, 348, 351, 353

W

WCP 8, 151-152, 157, 163