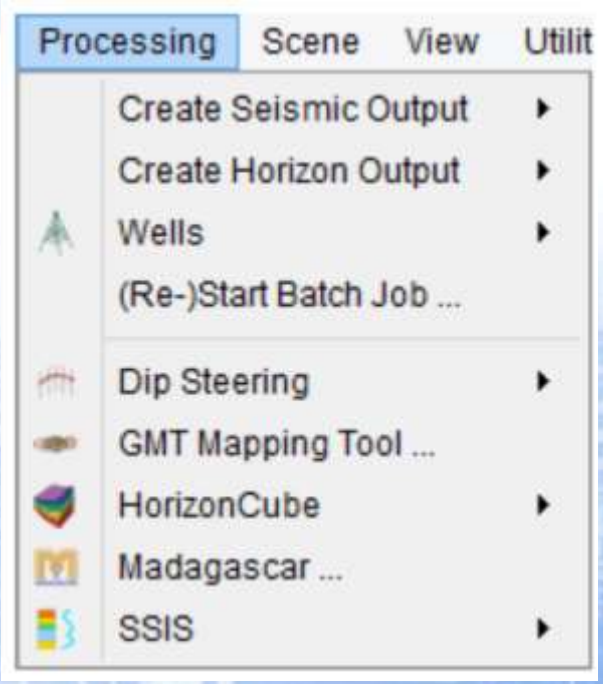
6 处理(Processing)

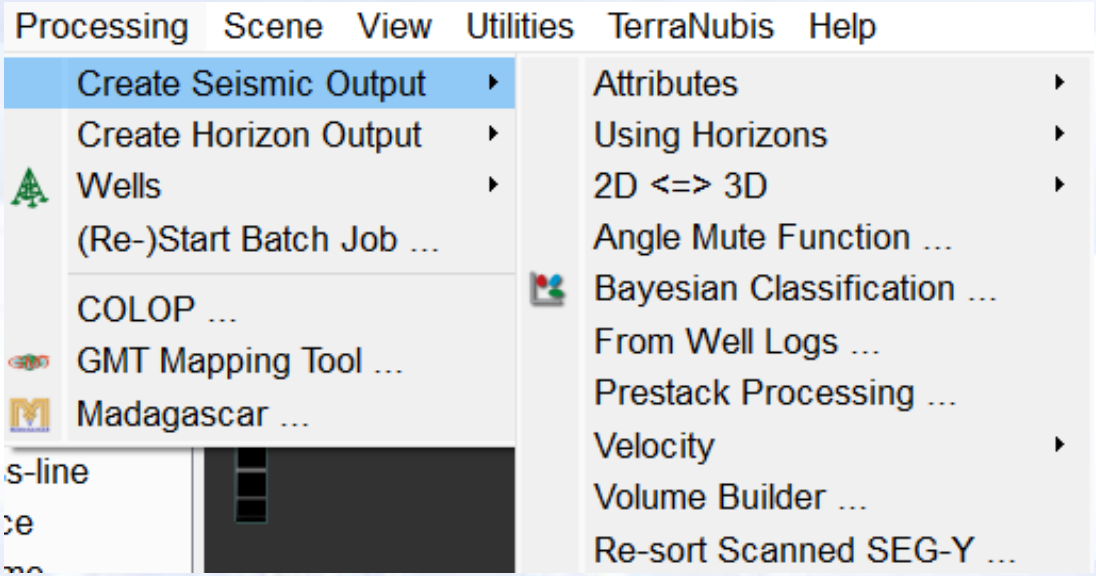
6.1创建地震输出

地震属性的计算和输出是重点，但受到许可证的诸多限制。需要用开源程序来补充，或者购买许可证。



6.1创建地震输出

The Create Seismic Output option leads to a number of further choices, detailed in the following subsections:



6.1.1 Attributes

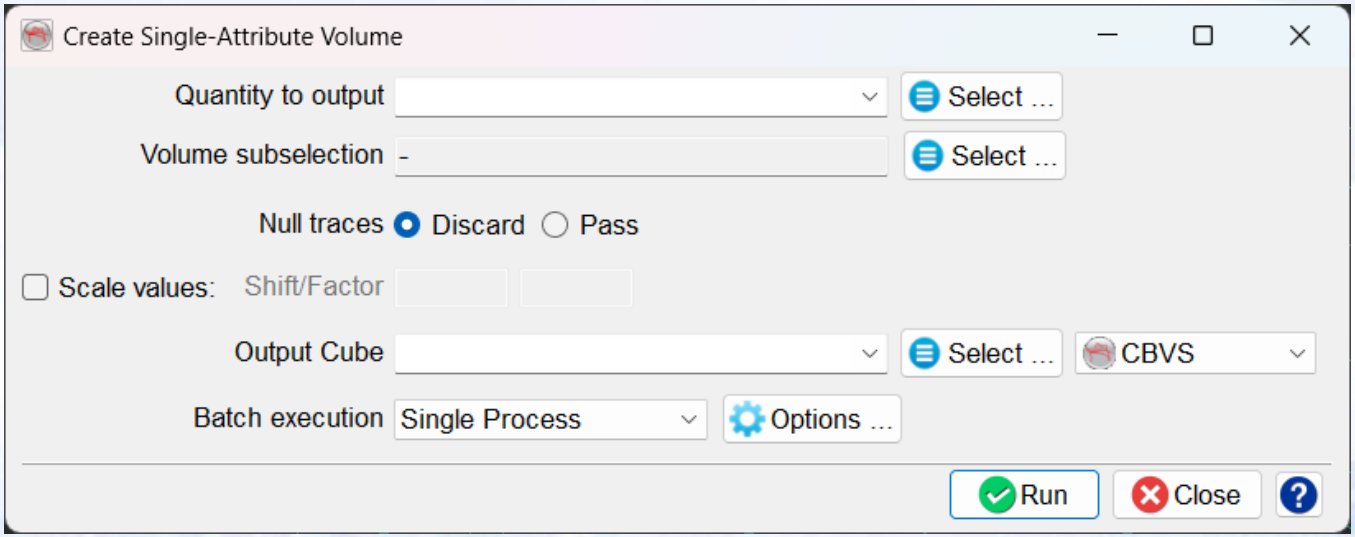
In this module any (attribute) volume can be calculated and saved to disk. In case of *2D attributes*, the output is a new data set. The volume output module can be run in batch mode, allowing to continue working in the main window while the processing is running.

This module creates, for example, attribute cubes, neural network cubes, or filtered data cubes.

本功能模块负责计算（属性）数据体并保存到磁盘。
对于2D属性，输出是新的数据集。
可以批处理运行数据体输出，允许正在运行Processing，同时在主窗口下继续工作。

6.1.1.1 Single Attribute

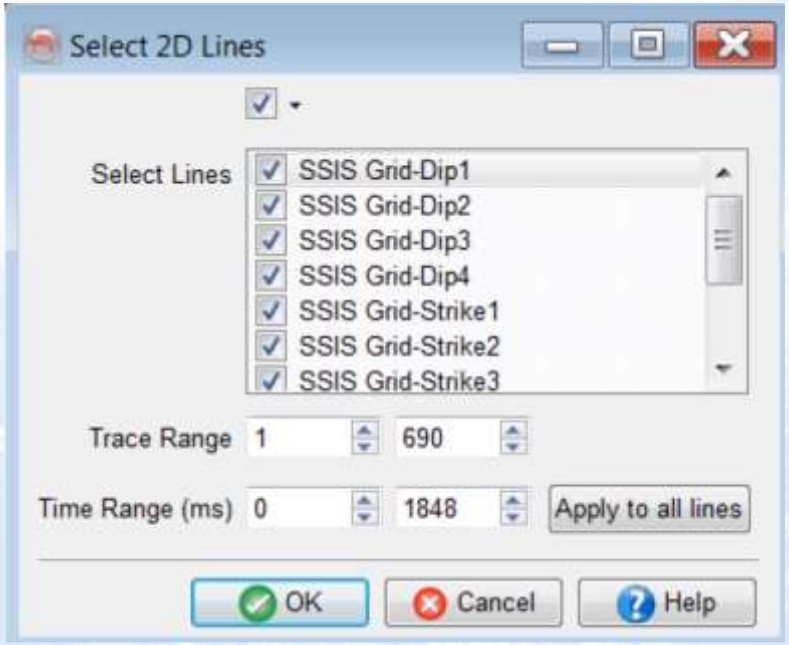
> Attribute > Single Attribute



First, *Select* the output quantity : it can be either a stored 2D volume or an attribute from the active 2D attribute set.

Note that only attributes from the current attribute set can be selected in the "*Select quantity to output*" window. To output an attribute from another attribute set, you must select this attribute set in the attribute module.

Though the default is set to '*All Lines*', a selection of lines can be specified by clicking '*Select*':

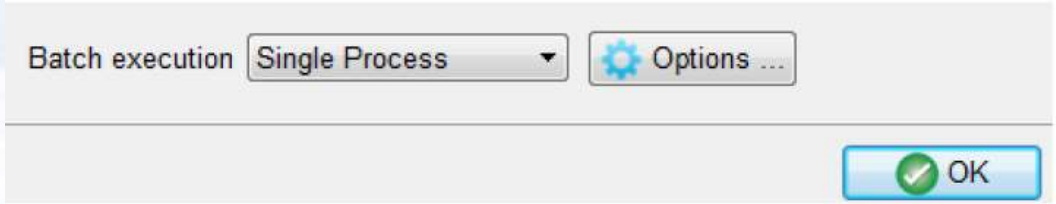


Optionally, the output can be scaled with a *Shift* and a *Factor*:

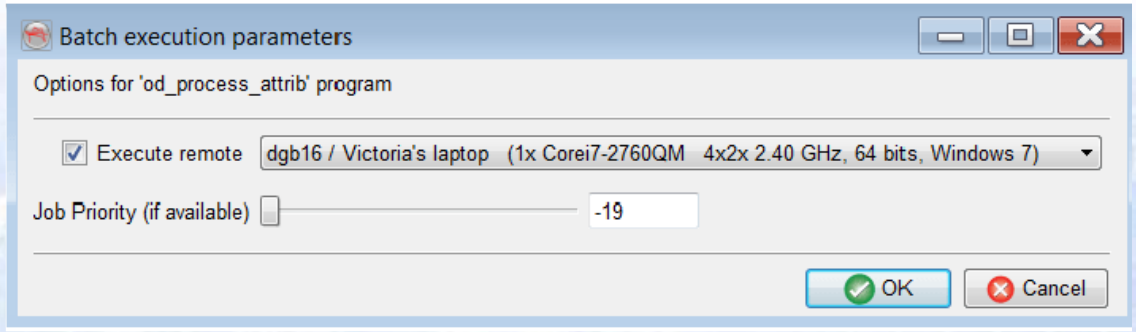
Output = Factor * Input + Shift

Null traces can be discarded.

It is also possible to choose between *Single Process* and *Multi-Job/Machine* processing:

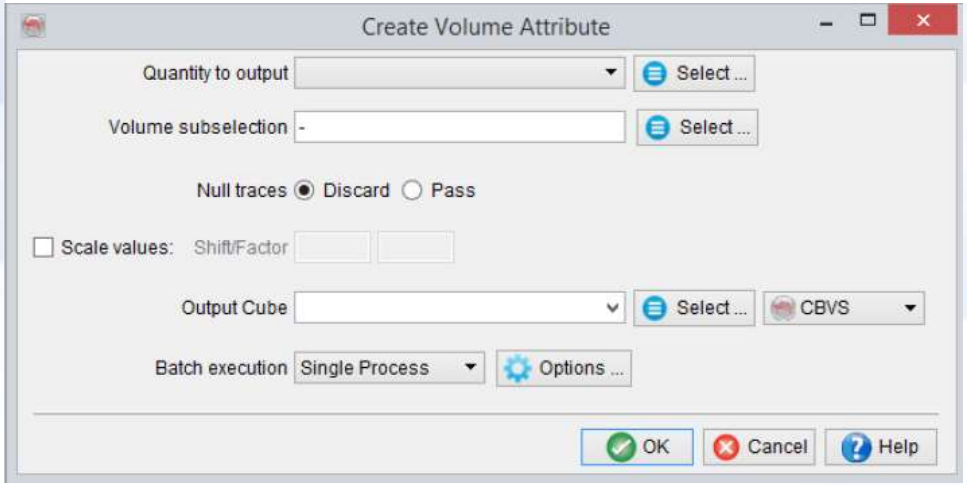


And to set the job priority (if deemed necessary):



6.1.1.2 3D - Create Output

To create a 3D seismic output from an attribute, follow the path *Processing* > *Create Seismic Output* > *Attribute* > *3D*.



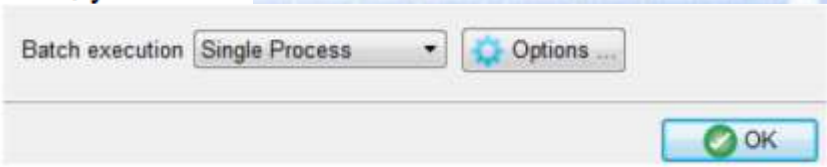
GNU（只有Single Attribute），与手册的不一样。
是否是因为F3_Demo是3D Survey?

First, *Select* the output quantity. Optionally, a *sub-volume* can be specified.

Note that only attributes from the current attribute set can be selected in the '*Select output quantity*' window. To output an attribute from another attribute set, you must select this attribute set in the attribute module.

Null traces can be discarded.

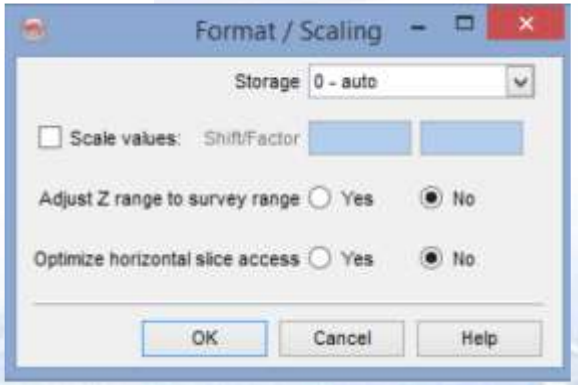
It is also possible to choose between Single Process and Multi-Job/Machine processing:



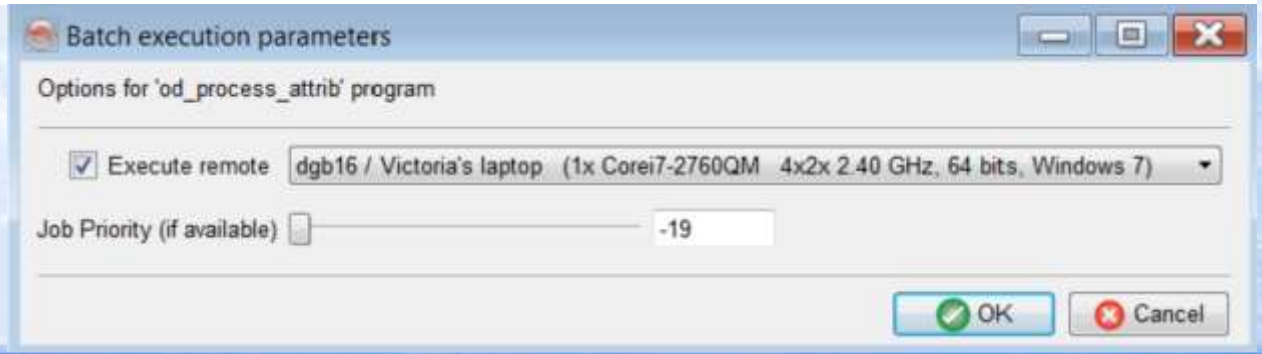
Optionally, the output can be scaled with a Shift and a Factor:

Output = Factor * Input + Shift

Storage: OpendTect can store data internally in 8-, 16-, 32-, and 64-bit seismic data formats. 8-bit signed has a data range between -127 and +127. 8-bit unsigned ranges between 0 and 255. Similarly, 16-bit signed ranges between -32767 and +32767 (unsigned 0 - 65535). The data is stored in the same byte-format as the input by default (Storage is set to 0 - auto). This is chosen when specifying the *Format/Scaling*.



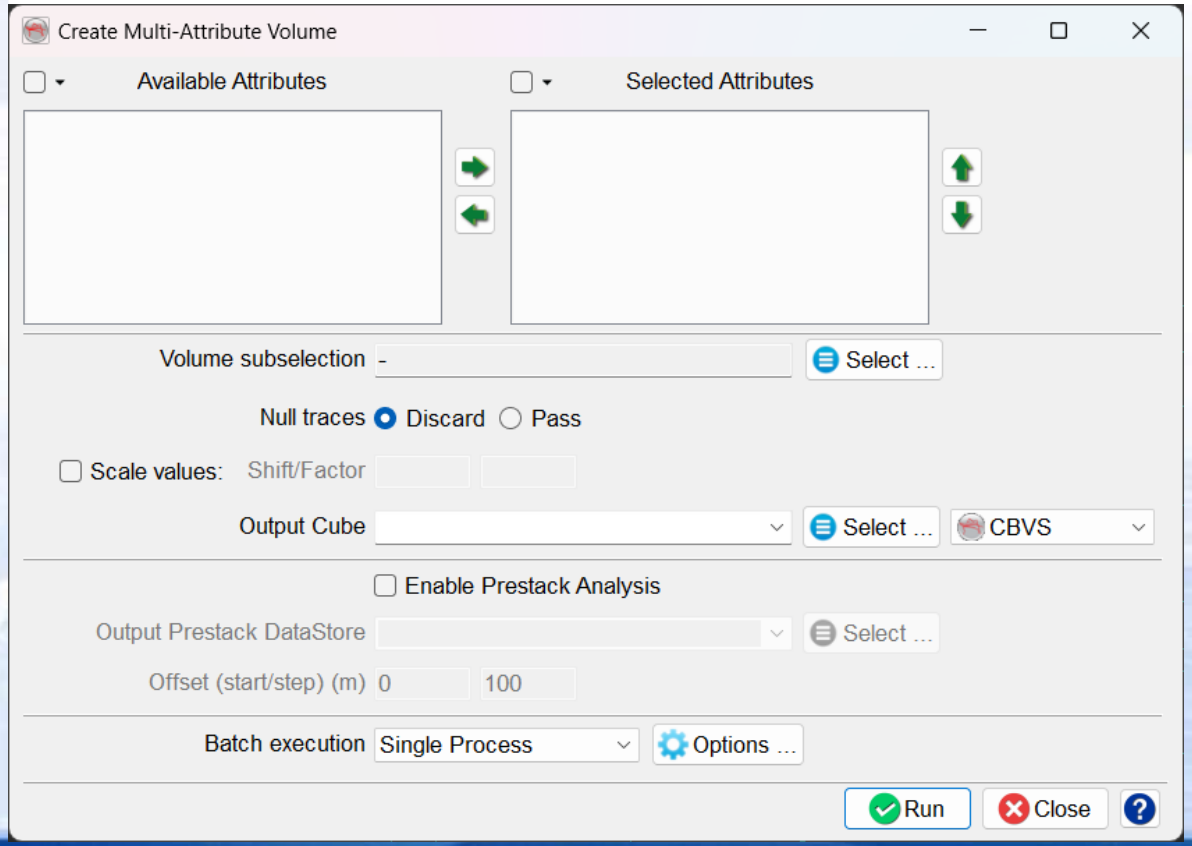
And to set the job priority (if deemed necessary). This is equivalent to the '*Nice level*' on Linux, determining how much priority the process will take on the remote machine:



批处理运行参数

6.1.1.3 Multi Attribute

Multi-attribute output enables the user to create a volume with several attributes. '*Available Attributes*' lists all possible components of the possible multi-attribute output and any combination can be selected. Once moved across into the '*Selected Attributes*' list, these attributes are processed into a single volume.

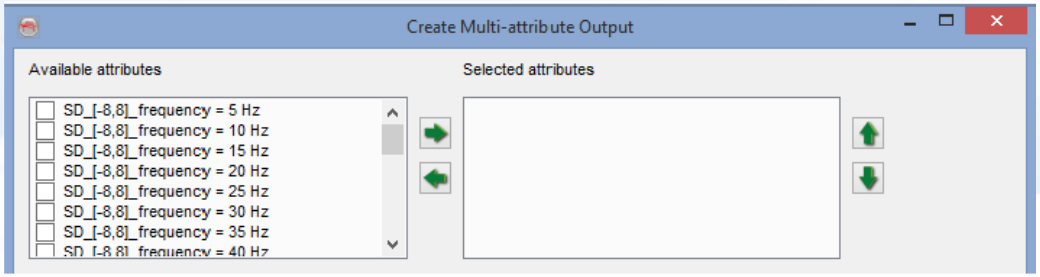


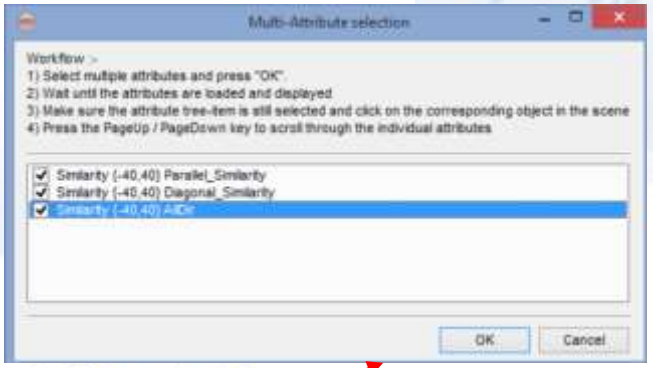
用户可以创建一个Volume具有多种地震属性。Available Attributes列表中应该列出多种可使用的多属性输出，可以选择联合使用。移动到Selected Attributes后，这些属性将处理为单个Volume。

注：地震属性计算受许可证的限制很多。

It is not necessary to check the boxes next to the attributes in the Selected Attributes list in order for them to be included in the output volume. All attributes in the 'Selected' list are included by default.

Also note that a Spectral Decomposition attribute will show a list of all possible outputs in the 'Available attributes' field:





On loading the volume onto an inline (for example), the user is prompted to select which of the available attributes he wishes to use:

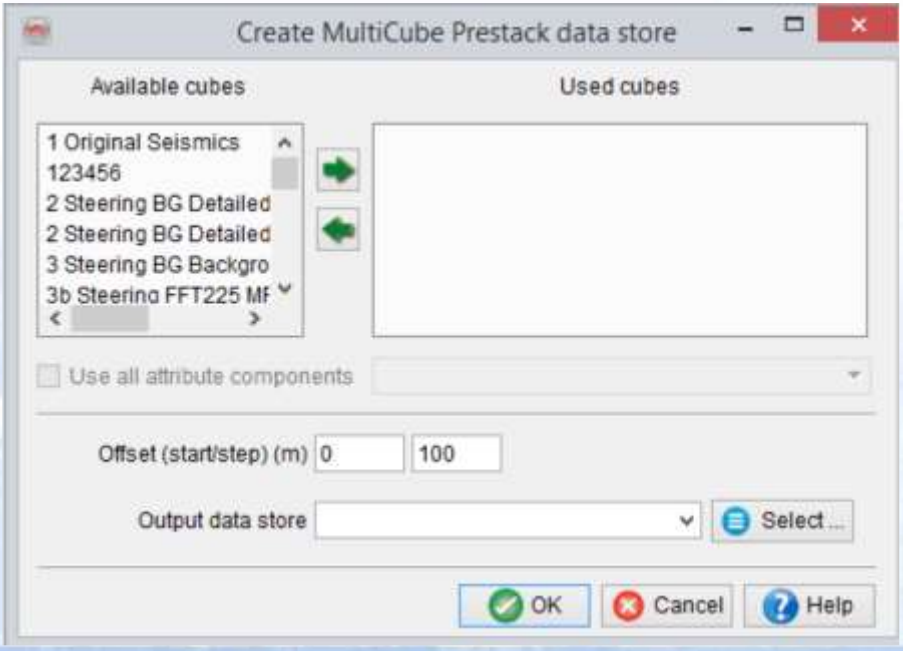
The component of the multi-attribute cube that is actually being displayed in the scene is appended on the volume name in the tree:

6.1.1.4 Multi-Cube Data Store

Multi-components cubes for some attributes (e.g Spectral Decomposition, Steering attributes) can be created. 一些地震属性的Multi-components cubes

First, create your attribute in the attribute engine.

Create the volume output: Go to *Processing* > *Create Seismic Output* > *Attribute* > *Multi-cube data store* and select a volume that contains multiple components (here, Spectral Decomposition as example):

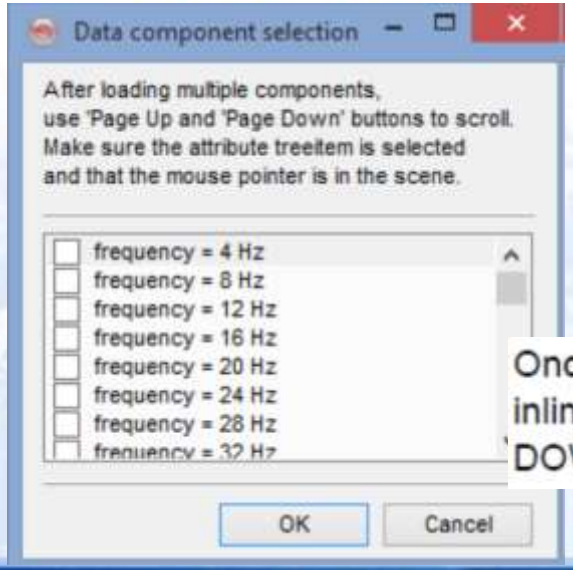


Select a component of the cube and assign it a 'pseudo-offset' value. Repeat this process of pseudo-offset assignment for all of the components that you wish to be present in the output, name it and press '*Go*'.

Multi-component cubes can be exported as SEG-Y or simple ASCII file but only one component per output. The choice of which component is given to the user during the export process.

How to display the Multi Component Volumes?

Whenever displaying a multi-component attribute volume on an inline/crossline, the component selection dialog box (below) will pop up. In this dialog one or more outputs to be displayed on the same section must be selected.



Once several components are selected and displayed for an element (e.g. an inline), place the mouse in the scene, and use the keyboard's PAGE-UP/PAGE-DOWN keys to view the next/previous slice in real-time.

6.1.1.5 Along Horizon

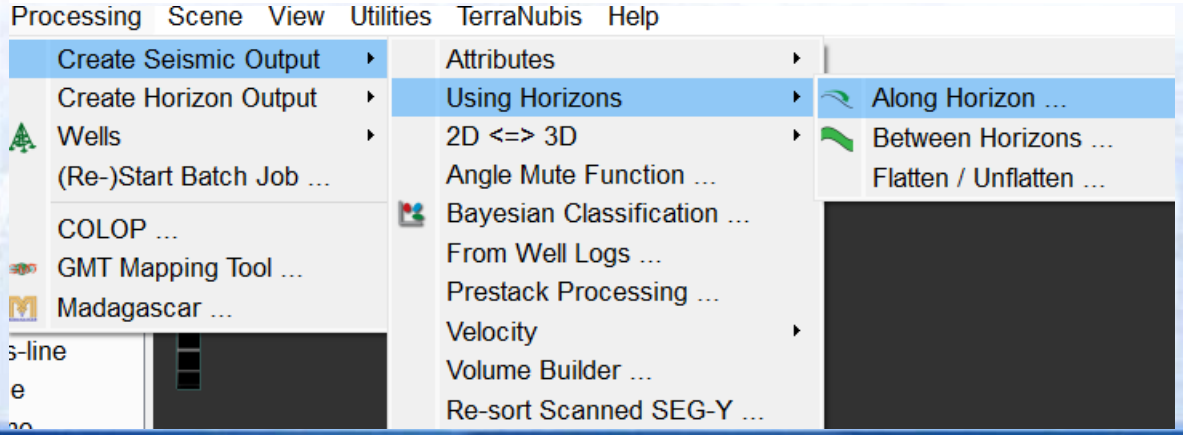
Using Horizon > Along Horizon

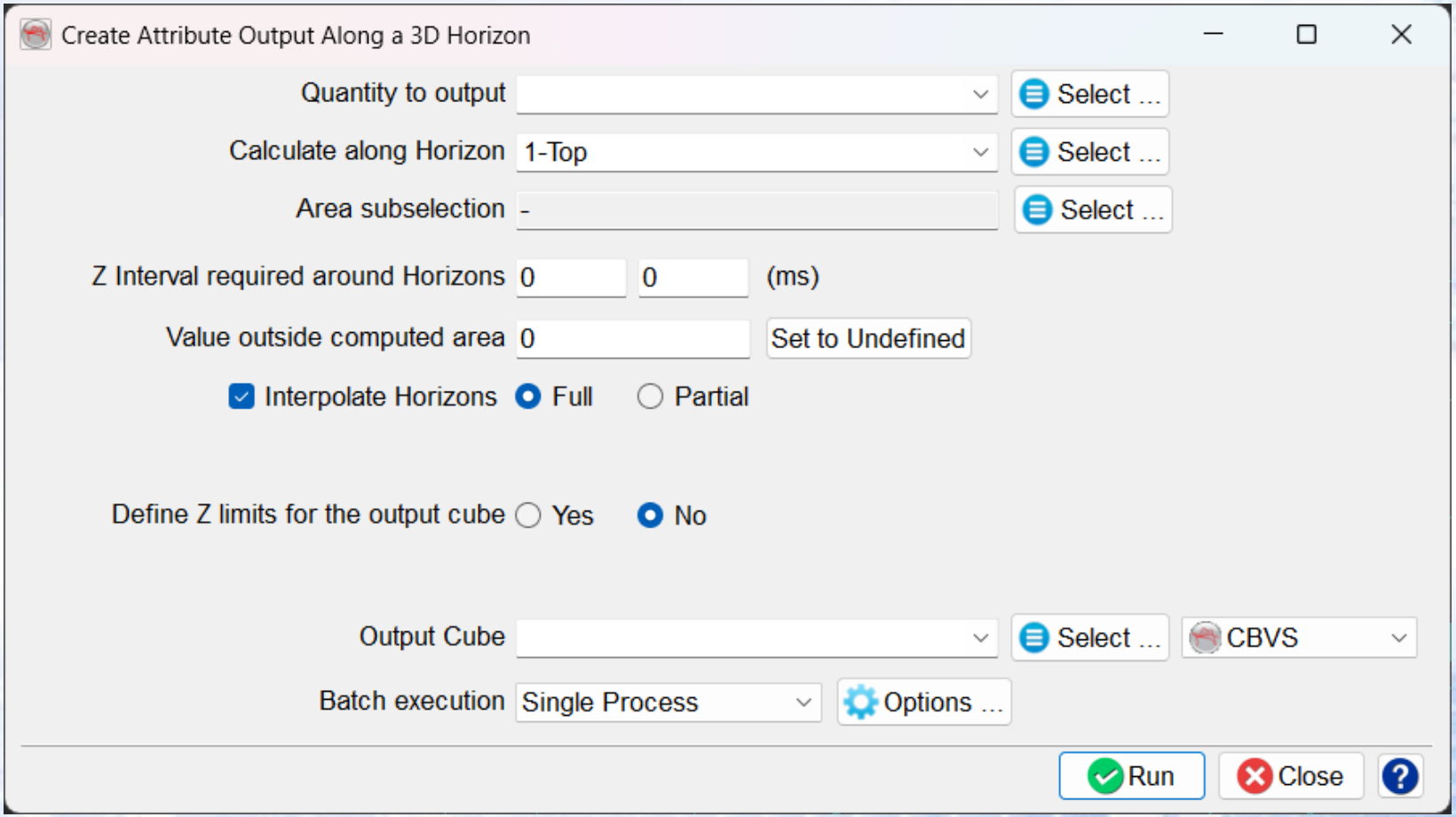
To create a seismic output in a time interval relative to a single horizon, the quantity to output has first to be selected from the list of stored data or attributes from the current attribute set. Specify the horizon and the Z interval relative to this horizon. A sub-area can be specified.

The *Value outside the computed area* is the undefined value. The standard undefined value in OpendTect is 1e30, but any other value can be specified.

Optionally, the horizons can be interpolated. The interpolation can be full or partial.

A calculation parameter file is automatically created with a default name in the *Store Processing Specification* field. This file allows to re-start the calculation process easily if needed. The '*Execution Options*' can be set to use either single or multiple machines processing.



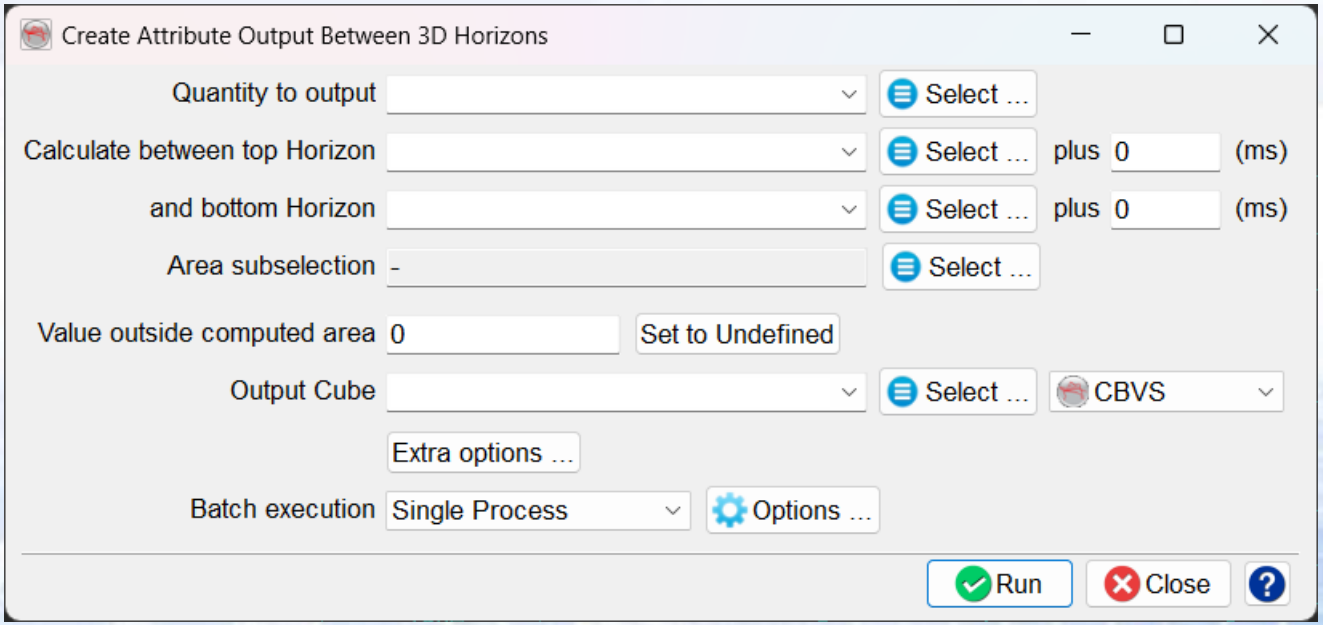


Along Horizon

6.1.1.6 Between Horizons

Using Horizon > Between Horizons

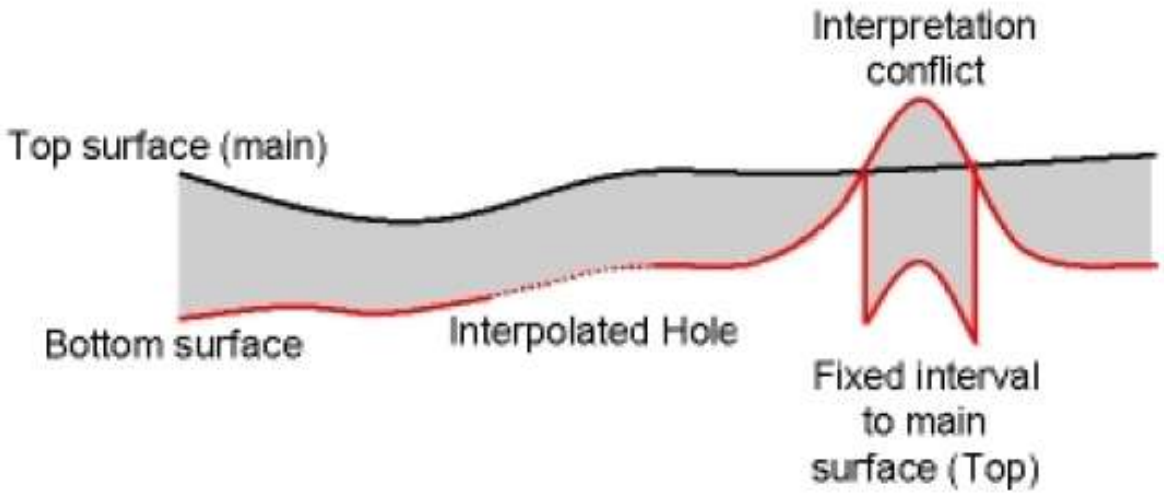
To create a seismic output between two horizons, first the quantity to output has to be selected from the list of stored data or attributes. Specify the horizons that form the upper and lower boundaries of the output volume. A *Z shift* to be specified can be applied to the upper boundary and/or to the lower boundary. A sub-area can be specified.



The *Value outside the computed area* is the undefined value. The standard undefined value in OpendTect is 1e30 or Undef, but any value can be specified.

In *Extra options*, horizons can be interpolated. The interpolation can be full or partial. When partial, only the gaps smaller than a user-defined number of traces are filled. A *fixed Z interval* length can be added to the leading horizon when the second surface is missing or in case of conflict during the interpolation. To constraint the interpolation, the *Z limits* for the output cube can also be specified.

For '*Execution Options*', please see: Batch Execution



6.1.2 2D to 3D Conversions

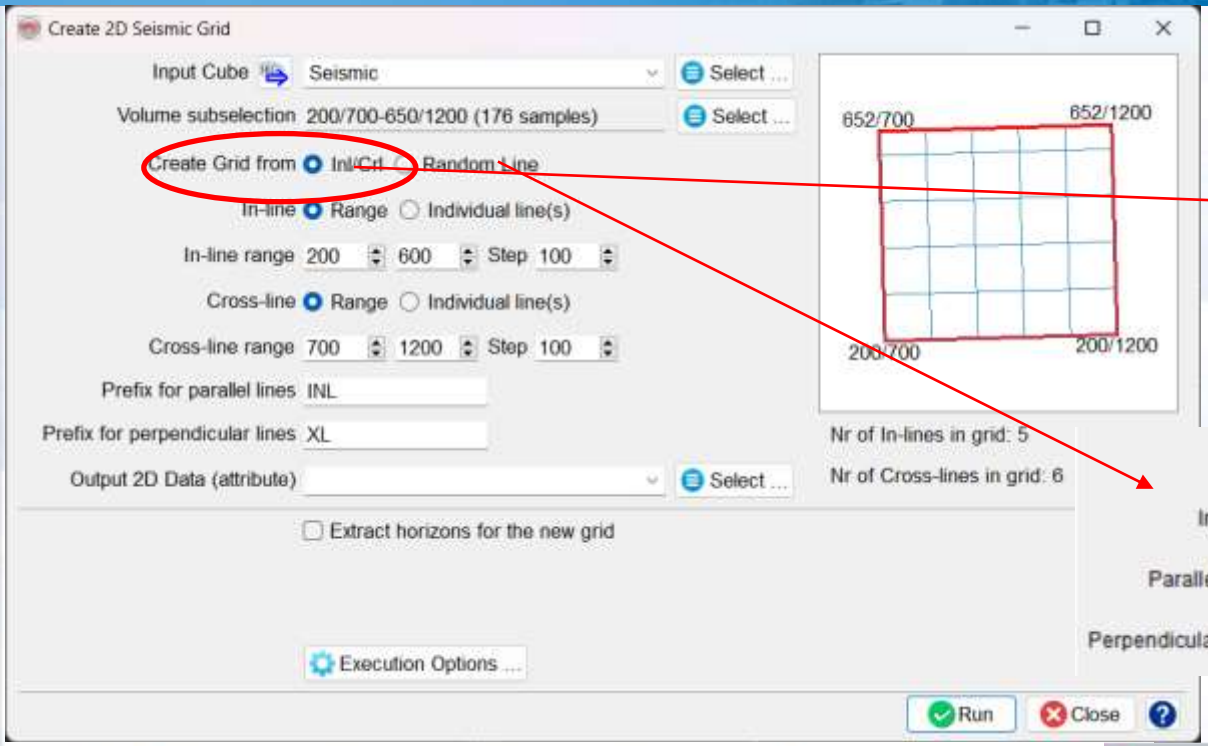
几种将数据在2D与3D之间转换的方式：

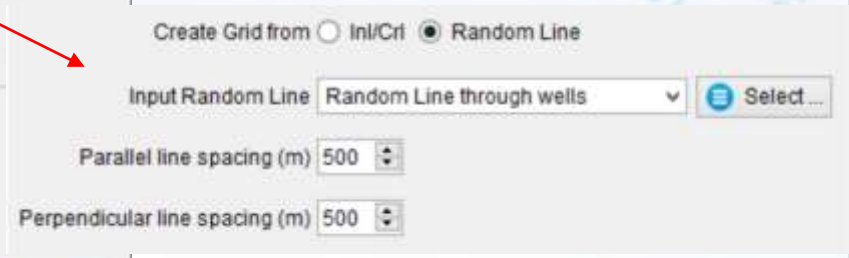
6.1.2.1 Create 2D Grid

The *Create 2D from 3D* option is an interactive tool for creating 2D-lattices from a 3D volumes.

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

- **Input Cube:** 3D seismic data for conversion into 2D line-set.
- **Volume Subselection:** Restricts the extent of the 3D volume (optional). Choose inline/cross-line/time ranges for the selected volume.
- **Create Grid from:** Define the output grid geometry and orientation of the 2D lines. If *Inl/Crl* is selected, the inline/cross-line orientation (or geometry) of the selected 3D data will be used within the defined inline/cross-line/z-range and corresponding steps. Optionally, the inline/crossline range can be edited manually by using loosely spaced inline/cross-line numbers separated by commas.



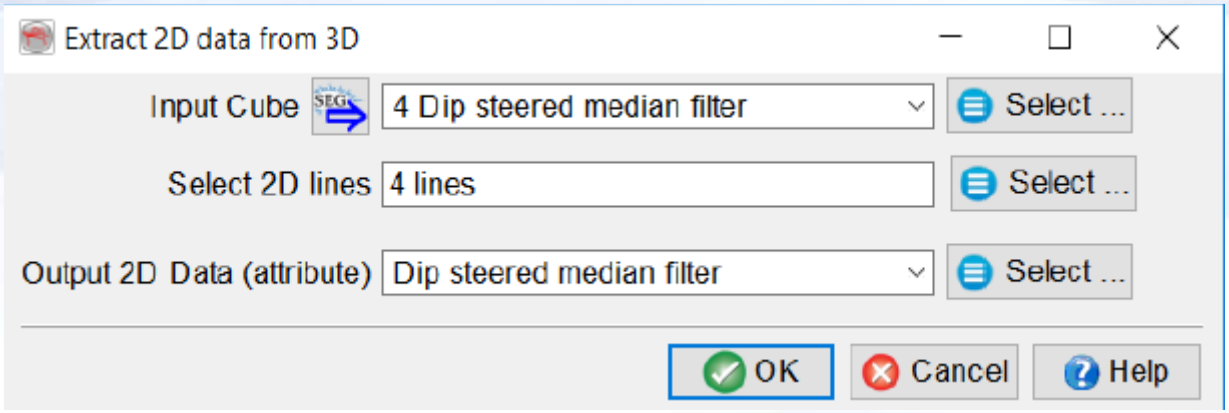


Another option is to define the 2D grid geometry based on a *Random Line*. The grid can be created both parallel- and perpendicular to the direction of the random line. Set the fixed spacing in the *line spacing(m)* fields.

- **Prefix:** Label the 2D-line names.
- **Output Data Set:** Output for the 2D line-set. Provide a name for the line and the seismic data (attribute).
- **Extract Horizons:** Convert existing 3D horizons into 2D horizons by checking the box (optional). In the *Select horizons* list, select one or more horizons and, if desired, set a prefix for these selected horizons.
- **Execution Options:** Launches the 'Batch Execution Parameters' window and gives the option to execute the job remotely:

6.1.2.2 Extract 2D from 3D

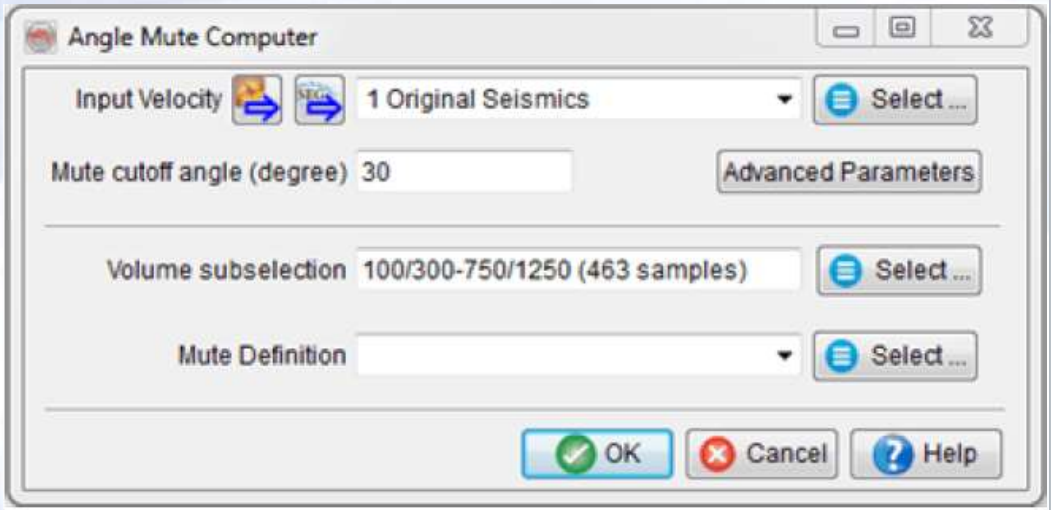
Extract 3D data onto selected 2D lines. Input data is required in the form of a stored 3D volume. One or more 2D lines can be selected for the 3D data to be extracted onto. The output data set requires naming:

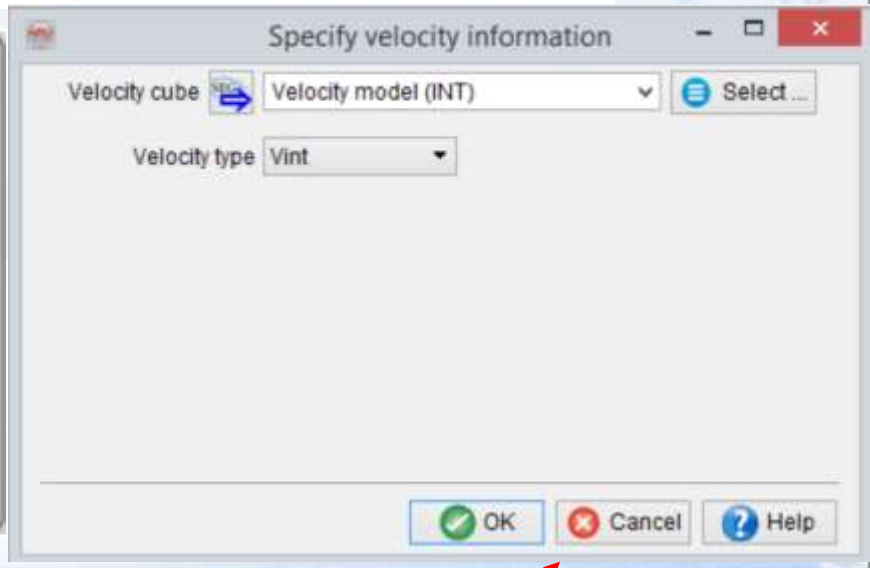


If just one line is selected, you may also sub-select a trace range.

6.1.3 Angle Mute Function

This module creates angle-based Z-Offset functions from velocity volumes. The primary input is a velocity model that provides the time-depth relation.





If you have no velocity volume available, click '*Create*'. This will bring up a window in which you can select a volume and tag it with a velocity type:

Vertical 1D ray-tracing is performed assuming a fully flat, isotropic earth model. The travel-time 0ms corresponds to the depth of the Seismic Reference Datum, defined in the survey definition window.

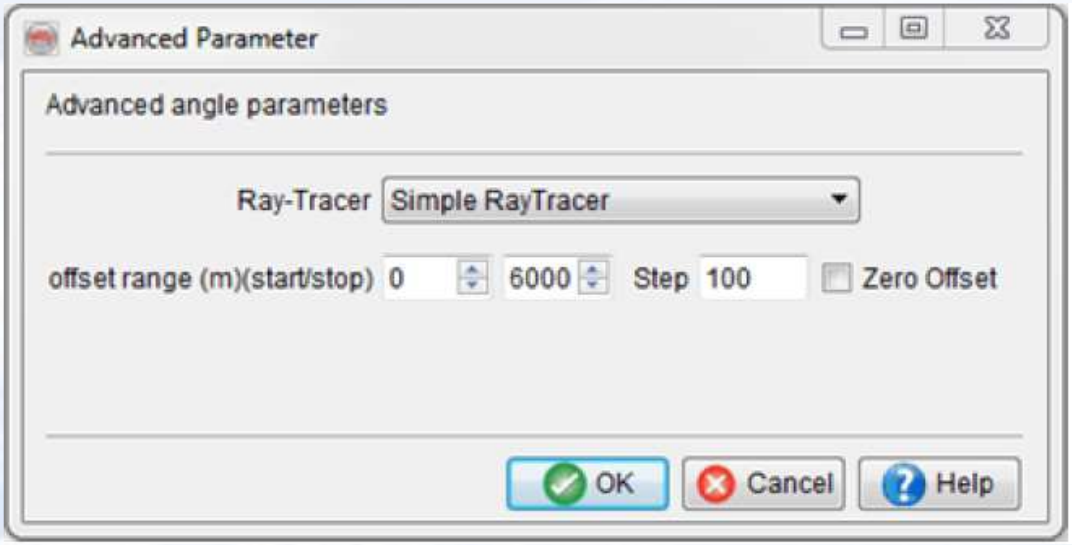
The offset range must be provided since the angle mute computer is not aware of any prestack datastore. It does not necessarily need to match the prestack data. The output function will have one point at the start and stop of the Z range, and one point at each offset specified by the offset range parameters.

The main output parameter is the incidence angle in degrees at which the mute function must be computed. By default the functions will be computed on a relaxed grid, every 20th time the inline/crossline stepout. This can be changed by selecting other "Volume subselection" parameters. In general it is not necessary to decrease that stepout.

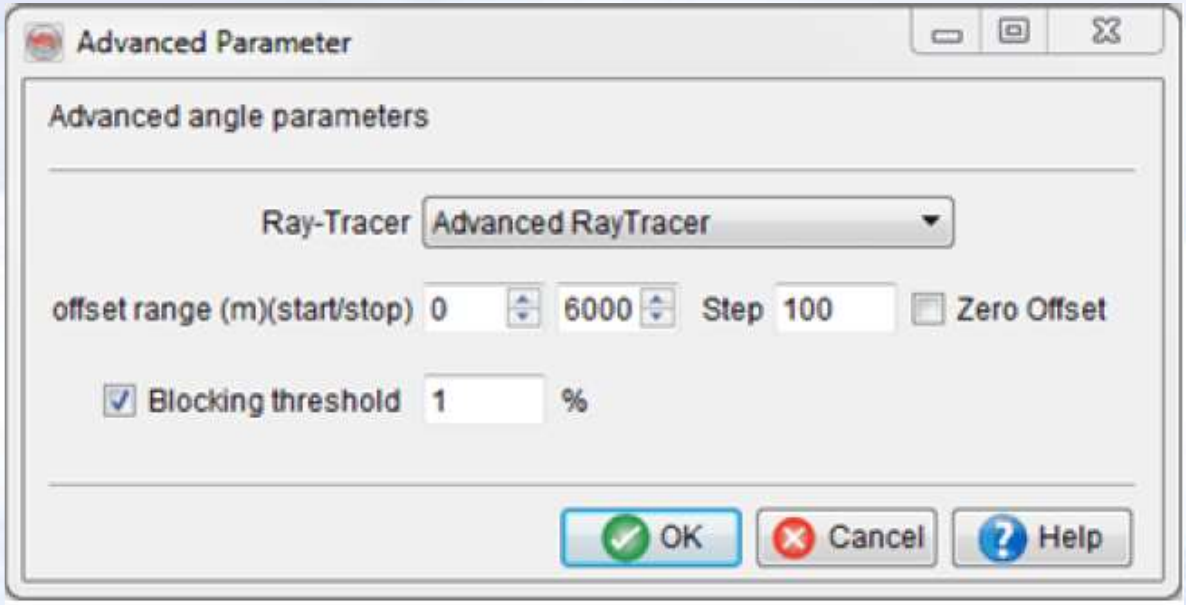
6.1.3.1 Advanced Angle Mute Parameters

The ray-tracing can be performed in two ways:

Simple: The ray is going directly from the source to the depth of the target layer, and up to the receiver in the same way. This does not account for ray bending, or velocity inversions.



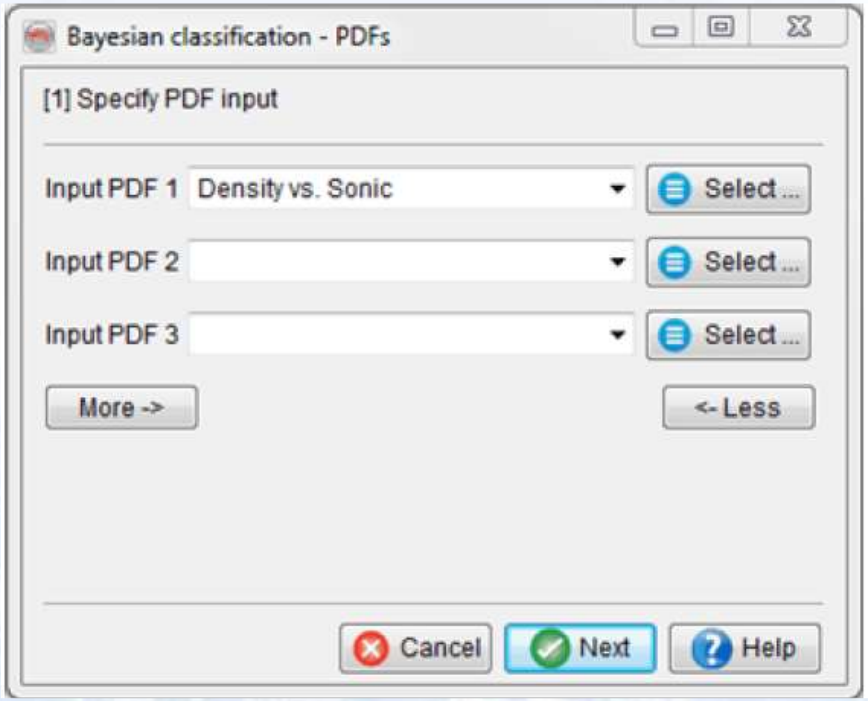
Advanced: Will honour the ray bending according to Snell's law and thus velocity inversions as well. To reduce the processing time, the layers may be blocked: Consecutive layers with similar Vp, (and density, Vs if present) values are concatenated together. The ray is propagated in a straight line inside a concatenated layer.



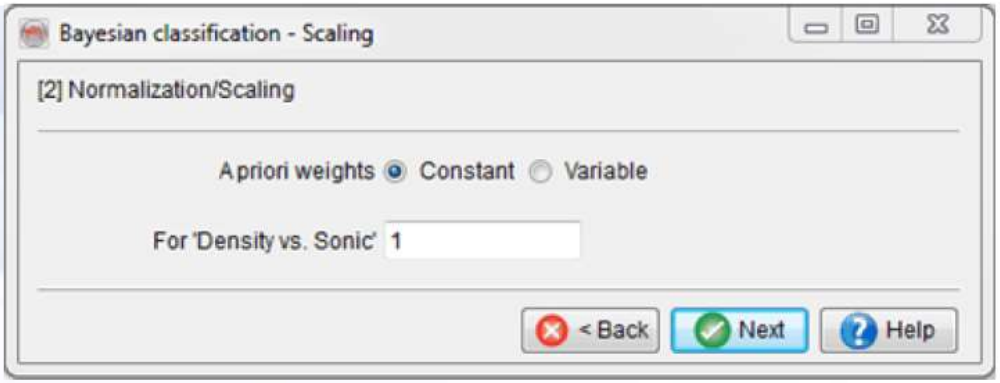
6.1.4 Bayesian Classification

Bayesian classifications are used to link several attributes based on one or several Probability Density Functions.

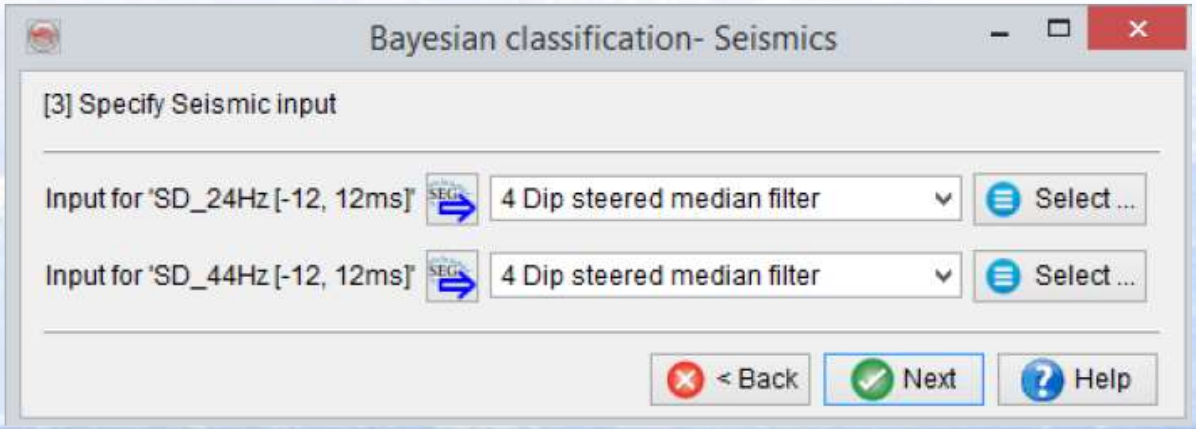
First, one or several PDF(s) need to be provided. *More* will allow to select more PDFs.



After clicking on *Next* >>, the PDF(s) can then optionally be normalized based on a priori weights per PDF. The a priori weights can be provided by attribute volumes which will vary at every sample location.

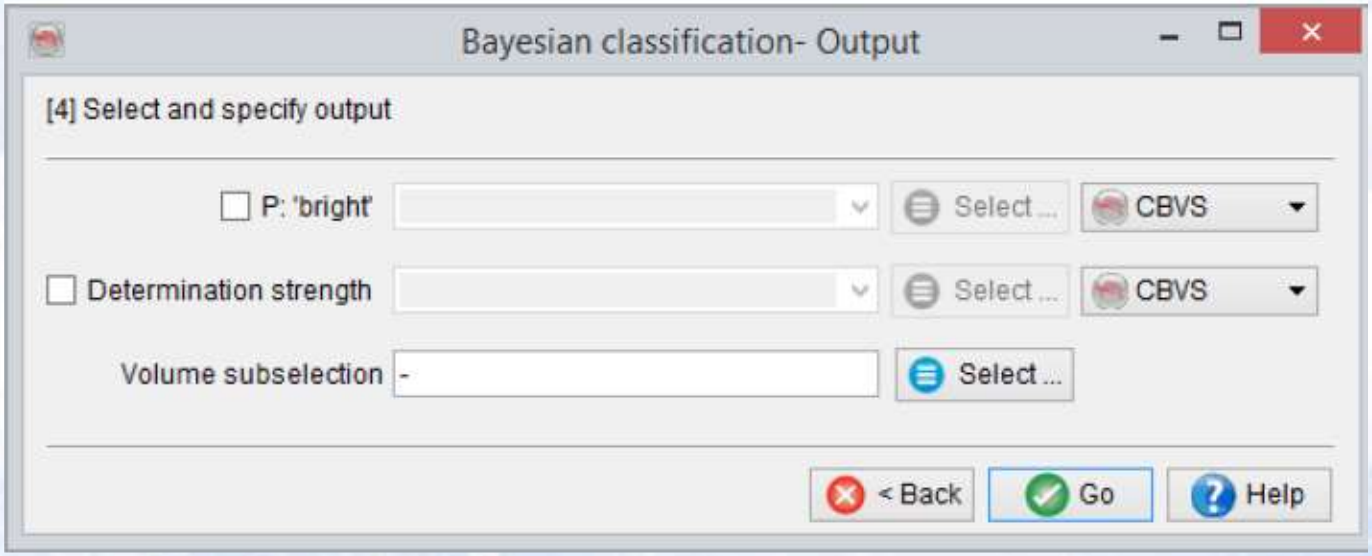


The processing will be based on (inverted) stored volumes that should correspond to the variables used for generating the PDF. Please note that OpendTect cannot make this check.



If only one PDF was used as input, the Bayesian classification volume will generate two volumes:

1. A probability volume "P:PDF name"
2. A determination strength volume, mainly for QC.



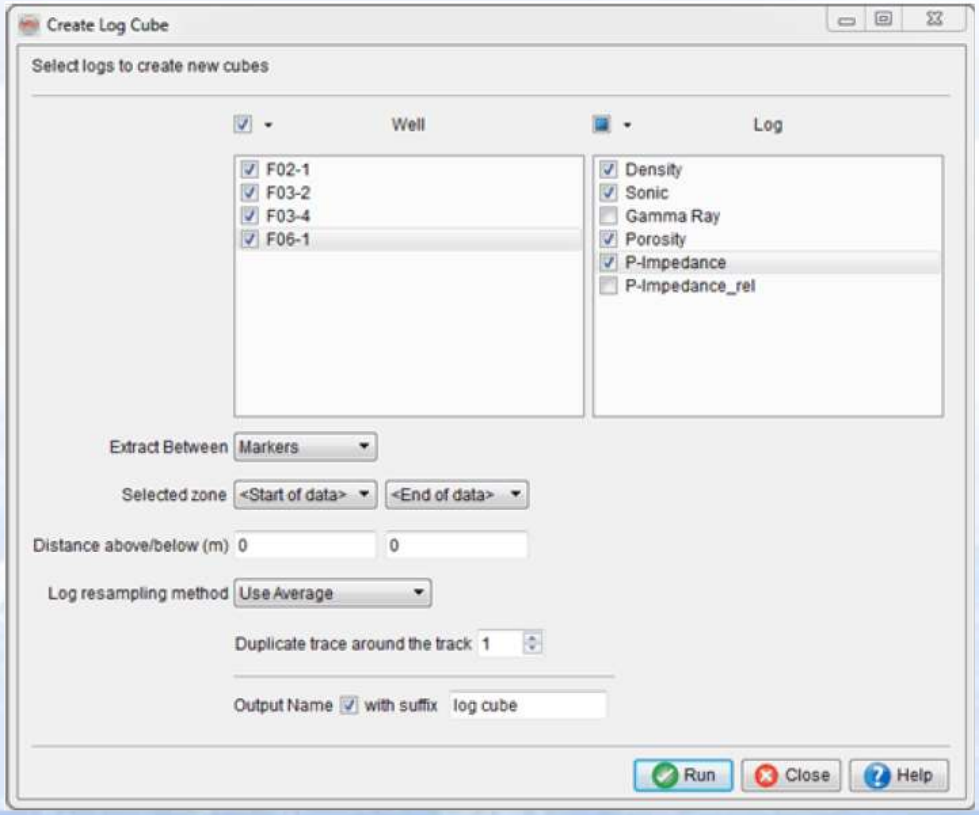
If more than one PDF were used as input, the Bayesian classification volume will generate:

1. A probability volume "P:PDF1 name"
2. A probability volume "P:PDF2 name" (one for each input PDF)
3. Two classification volumes : Class (with the most likely class) and Confidence (the difference between the two most likely)
4. A determination strength volume, mainly for QC.

6.1.5 Create from Well Logs

The *Create seismic output from wells* option writes loaded logs as seismic volumes.

The dialog is accessed from the *Processing* menu in the main toolbar; choose the *Create from Wells..* option.



Select the well(s) in the left and the log(s) in the right list, note that several wells and logs can be selected at once.

The log extraction can be done either between *markers* or by selecting a *depth range*. For the latter, select start/stop depths in meters. For extraction between markers select the wanted markers and add *distance above/below* (optional) for including intervals above and/or below the marker depths.

The logs need to be resampled in order to display correctly on seismic sections and volumes. Choose a resampling method from the dropdown list:

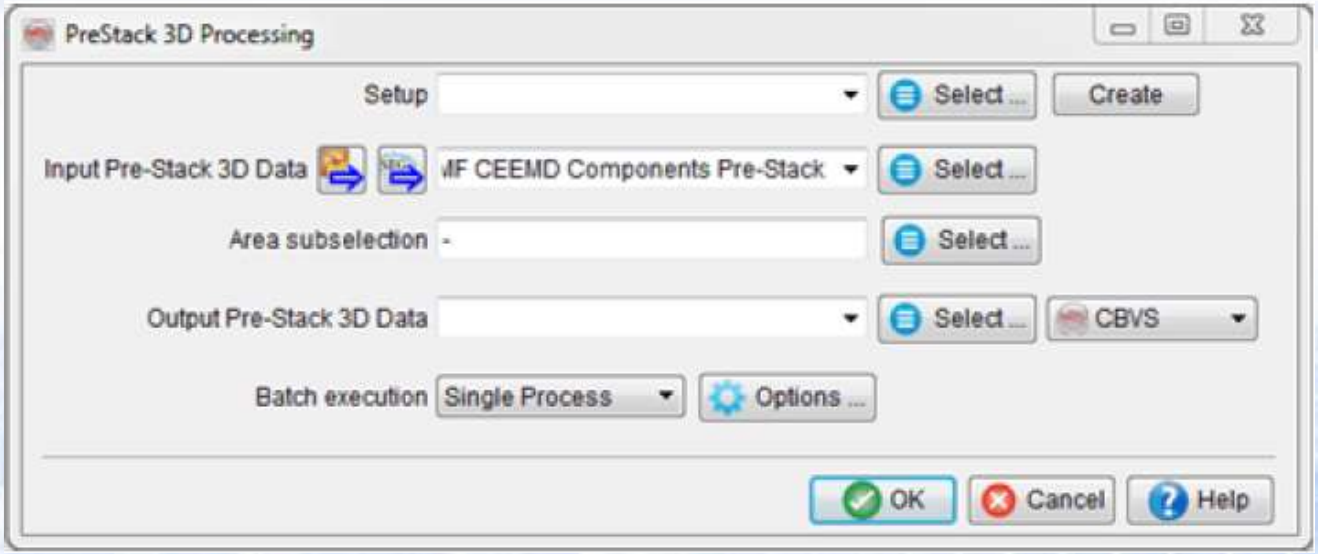
- Take Nearest Sample
- Use Average
- Use Median
- Use RMS
- Use Most Frequent

Next, choose how many traces should be duplicated around the well track. Essentially this will determine the dimension and geometry of the output cube. Finally give a name (suffix) to the CBVS volume (seismic volume), the volume itself will automatically be named according to logs selected in the list.

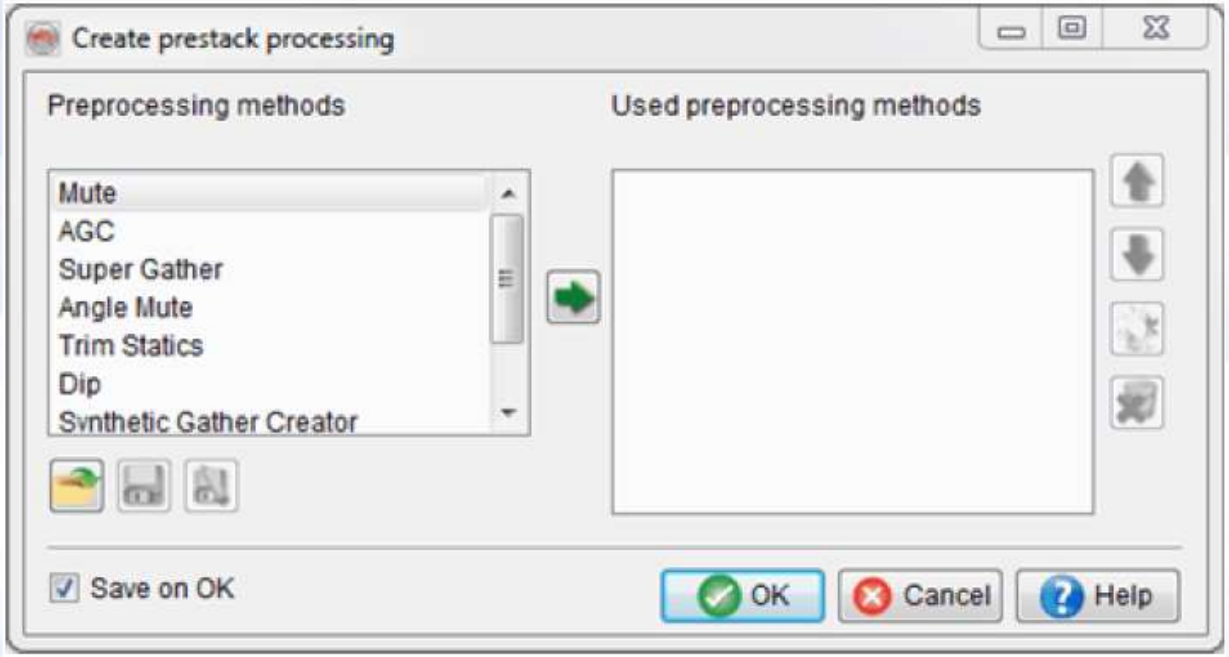
6.1.6 Pre-Stack Processing

Prestack processing can be applied using different methods. It is the only place where prestack data can be output in OpendTect based on another prestack data-store.

Open the Pre Stack processing window: *Processing* > *Create Seismic Output* > *Pre-Stack Processing*

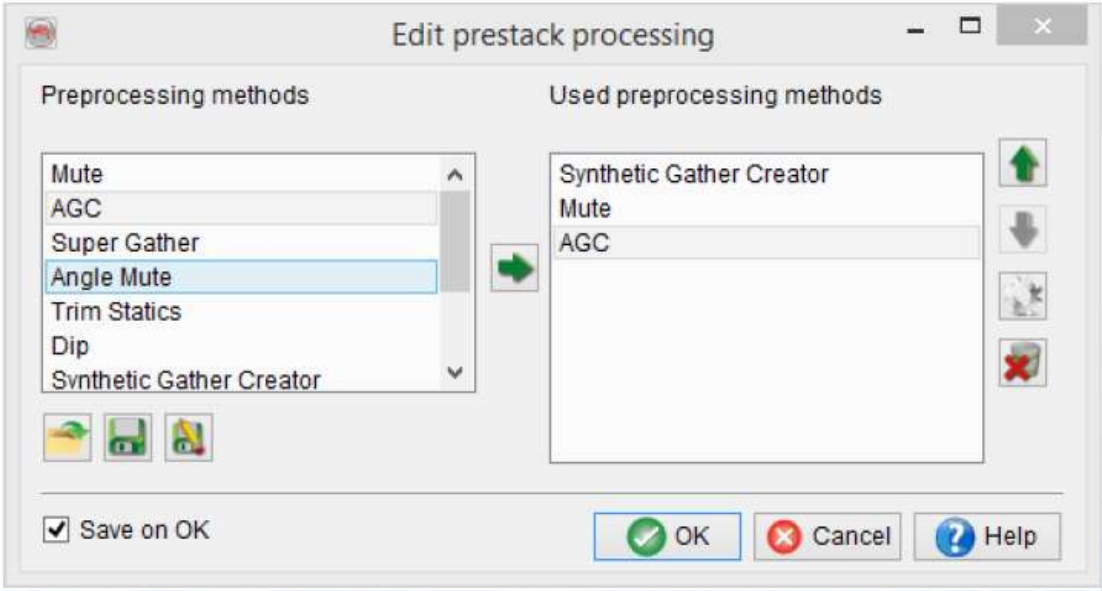


The processing can be done by a number of sequential steps. Either select a pre-defined set up (as above, which may be further '*Edit*-ed') or press '*Create*'...



Which brings you to the following window...

To use a preprocessing step, select it in the *preprocessing methods* and click on Add. It will then be listed in the used *preprocessing methods*, click on *Properties* to define the step parameters. Move up or Move down to change the preprocessing steps order for the processing.

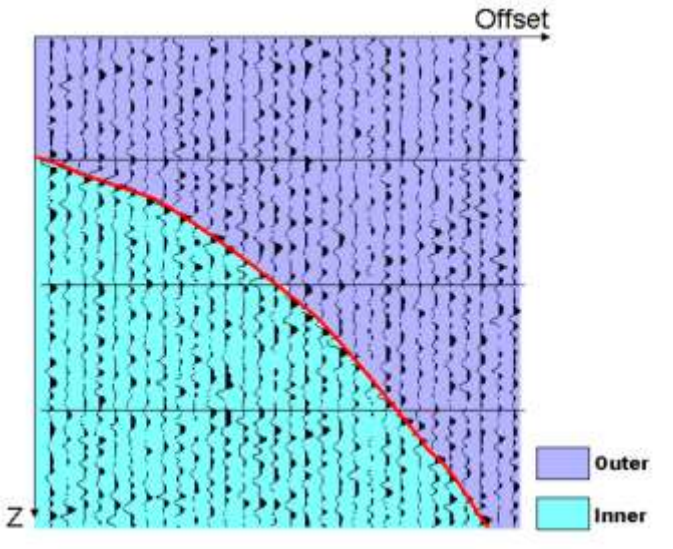


The following sections describe the available different steps.

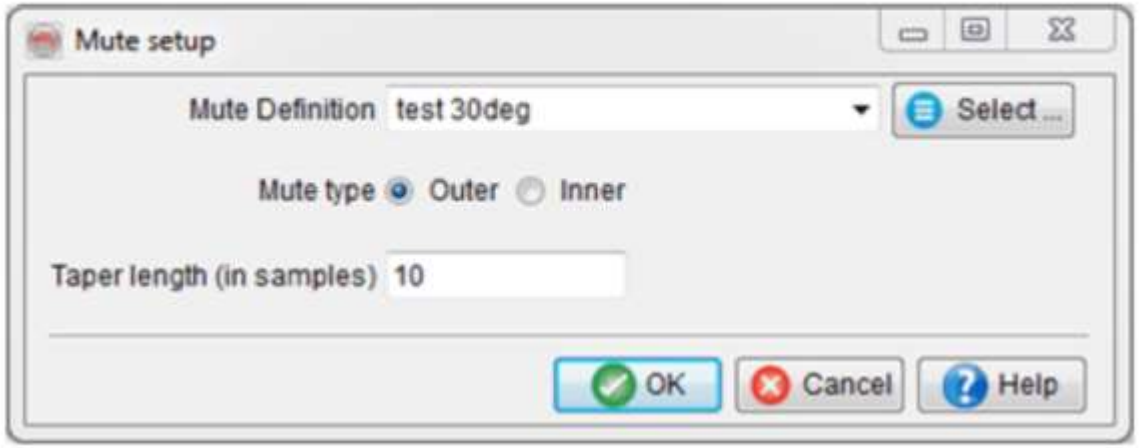
6.1.6.1 Mute

Mute functions may be applied to *Prestack gathers*. This window will allow you to choose the mute definition, as well as to specify settings such as:

Mute type: Outter (top) or Inner (tail)

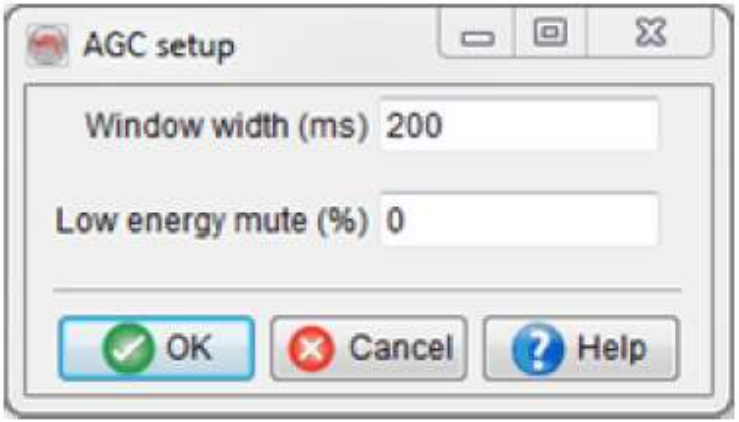


Taper length (in samples)



6.1.6.2 Automatic Gain Control

Automatic Gain Control (AGC) is one of the processing methods available for *Prestack gathers*. It will adjust the amplitude level using a sliding window of user-defined size (window width). Optionally, part of the lowest energy may be discarded from the amplitude level computation (in percent of the amplitude distribution).



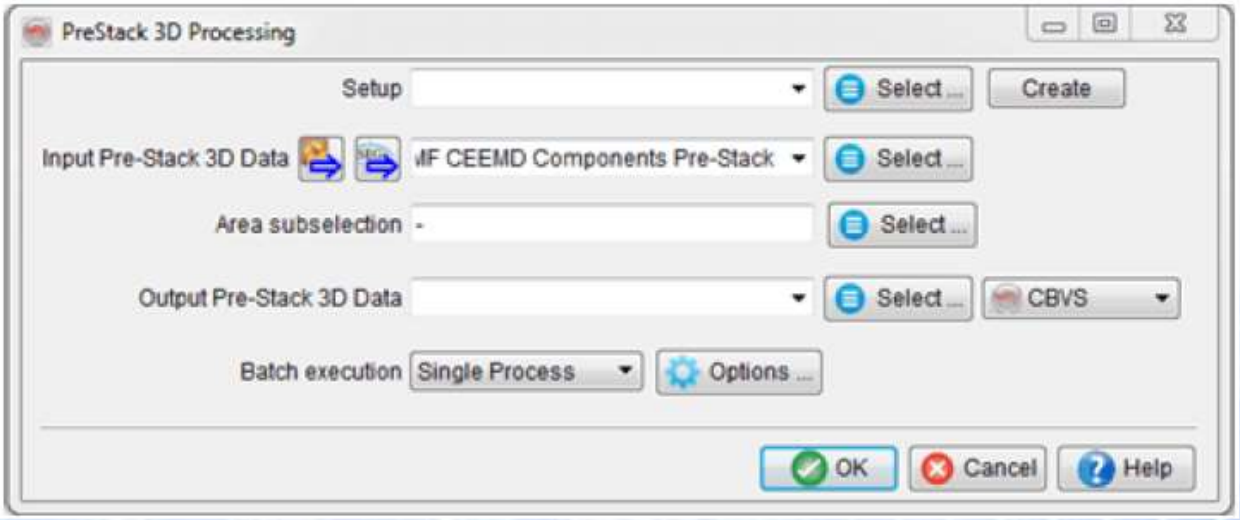
6.1.6.3 Super Gather

A *Super Gather* may be used to laterally stack the traces in order to increase signal-to-noise ratio of *Prestack gathers*. The stack is controlled by an inline/crossline stepout, and the Shape (Cross or Rectangle). The computation is similar to a (non-steered) volume statistics attribute with a zero time-gate.

6.1.6.4 Angle Mute

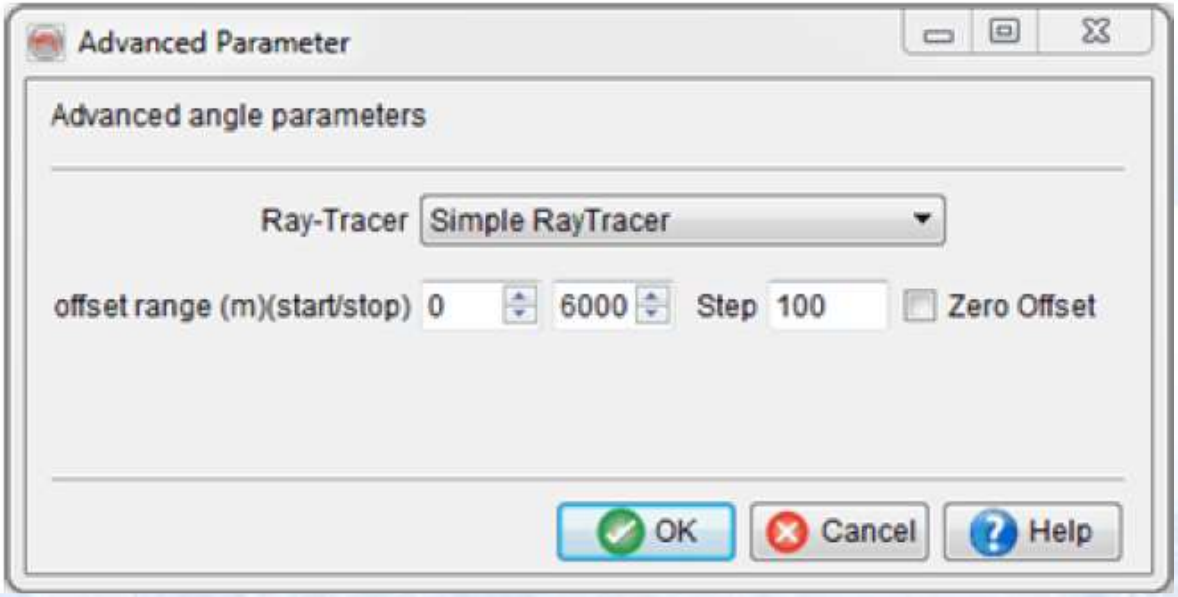
This processing method computes and applies a mute function. See the processing method documentation. The only difference is that this method reads the offset range from the input prestack datastore, such that there is need to specify it. See here where and when the input velocity source needs to be edited.

The application of the computed mute function is strictly identical to the application of a stored mute function.

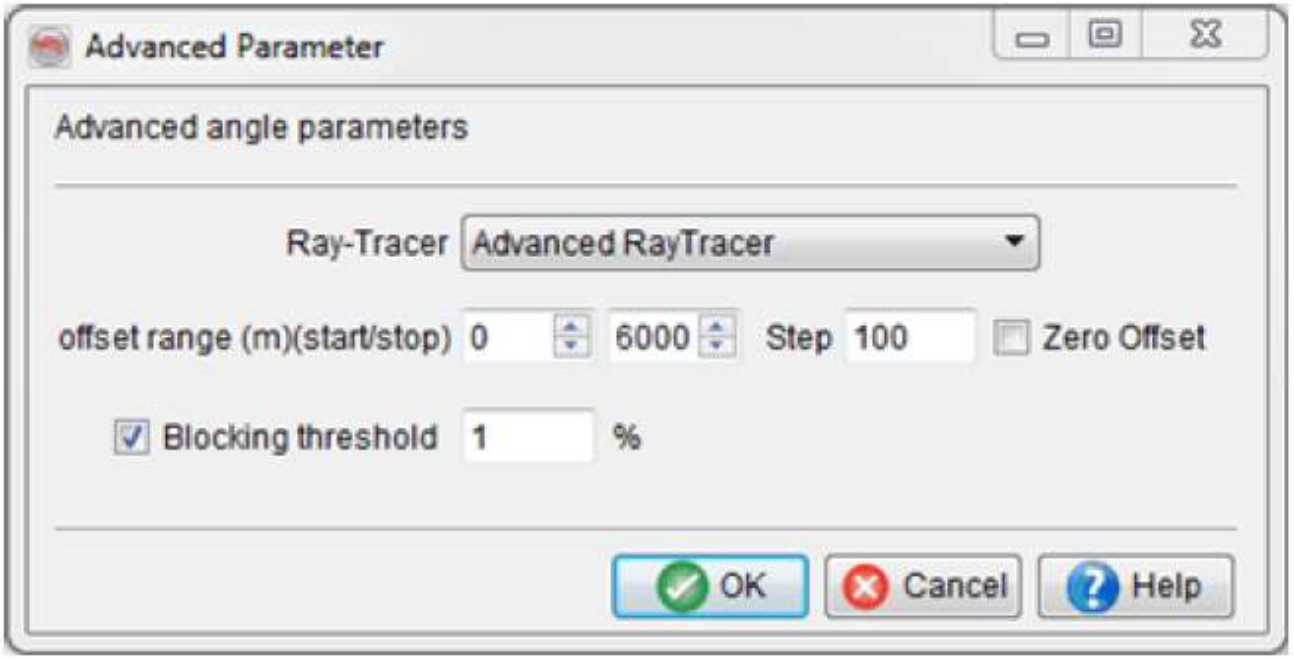


The ray-tracing can be performed in two ways:

Simple: The ray is going directly from the source to the depth of the target layer, and up to the receiver in the same way. This does not account for ray bending, or velocity inversions.

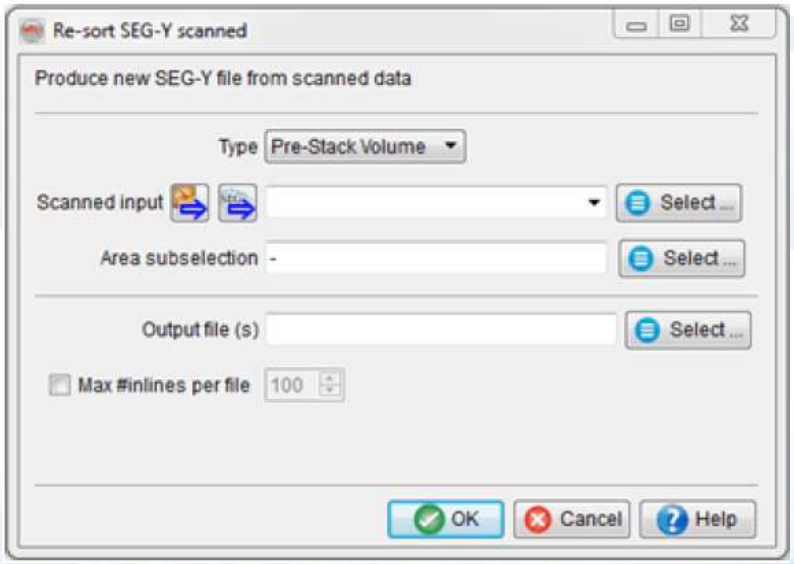


Advanced (not in the GPL version): Will honour the ray bending according to Snell's law and thus velocity inversions as well. To reduce the processing time, the layers may be blocked: Consecutive layers with similar Vp, (and density, Vs if present) values are concatenated together. The ray is propagated in a straight line inside a concatenated layer.



6.1.7 SEG-Y Scanned Re-Sort

The *SEG-Y Scanned re-sort* uses a scanned SEG-Y file and outputs it as a new file and re-writes the file-header. This tool is useful in case information in the header is poor or poorly sorted.



In the *Type* field select the type of volume, either *Pre-Stack* or *3D volume*.

Next, select the *scanned input file* and optionally *area sub-selection*. Note that a SEG-Y file must be scanned prior to resorting. Choose a name for the *output file* and (optionally) restrict the number of inlines to be written per file. In case the latter option is used, multiple files will be written to disk, either using sequential numbers or the inline ranges included in the separate files.

6.1.8 Velocity

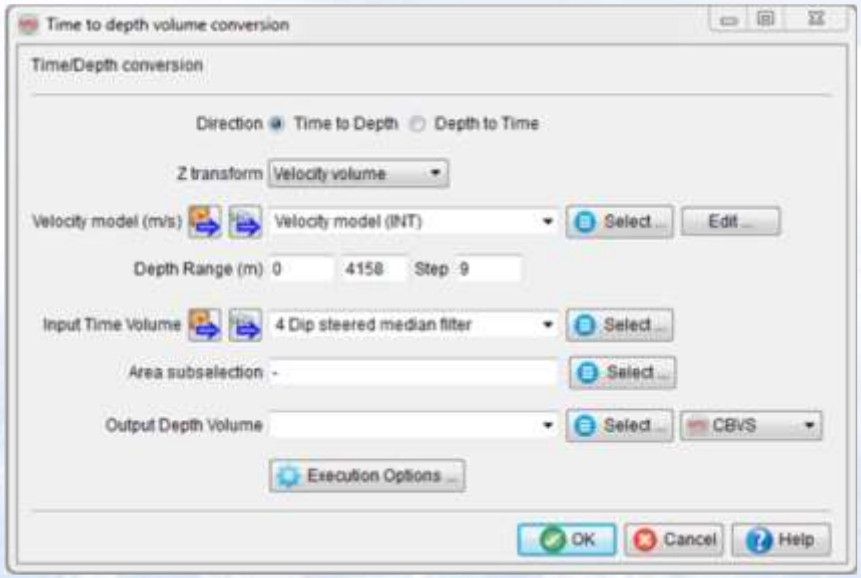
Under 'velocity' sit two velocity-based conversion options:

截图？

6.1.8.1 Time-Depth Conversion

To create an time-depth converted output, follow: *Processing* > *Create Seismic Output* > *Time-Depth Conversion...*

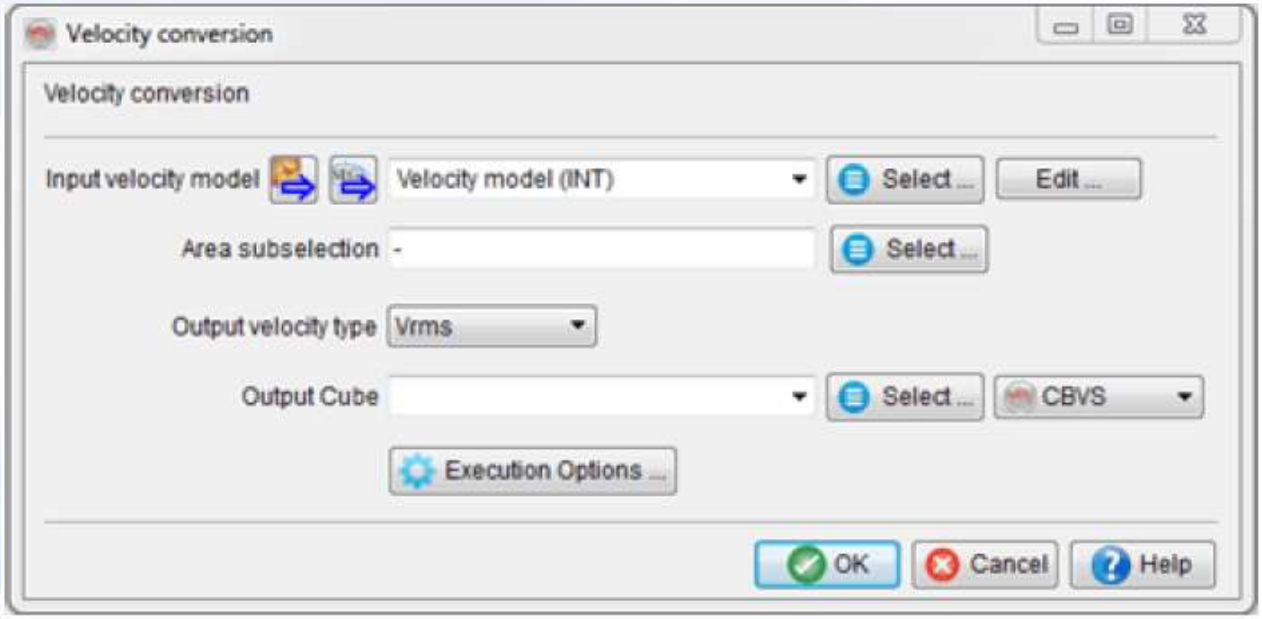
Time-depth conversion is done by applying Dix's equation, based on a interval- or RMS velocity volume. A velocity model and input time volume must be provided, and the direction of the conversion has to be set.



It is also possible to convert from Depth to Time. Instead of an input Time volume, a input Depth model has to be provided.

6.1.8.2 Velocity Conversion

This tool is started from *Processing* > *Create Seismic Output* > *Velocity conversion*. It can be used to convert interval velocity volumes to RMS velocity volumes and vice versa. The conversion is applied using Dix's formula. Please note that for this reason it can only be applied in the time domain.



6.1.9 Volume Builder Output

Volume-builder creator window can be launched from *Processing* > *Create Seismic Output* > *Volume Builder*. It is used to create the output volume that has been defined in a volume builder setup. Optionally, if the initial volume builder setup is not defined, press the *Edit* button to define the setup. Press '*Ok*' to launch a batch processing window.