5.2 Volume Builder Setup

5.2.1 Body Shape Painter

5.2.2 Lateral Smoother

5.2.3 Smoother

5.2.4 Horizon-Based Painter

5.2.5 Velocity Gridder

5.2.6 Input Volume

5.2.7 Voxel Connectivity Filter

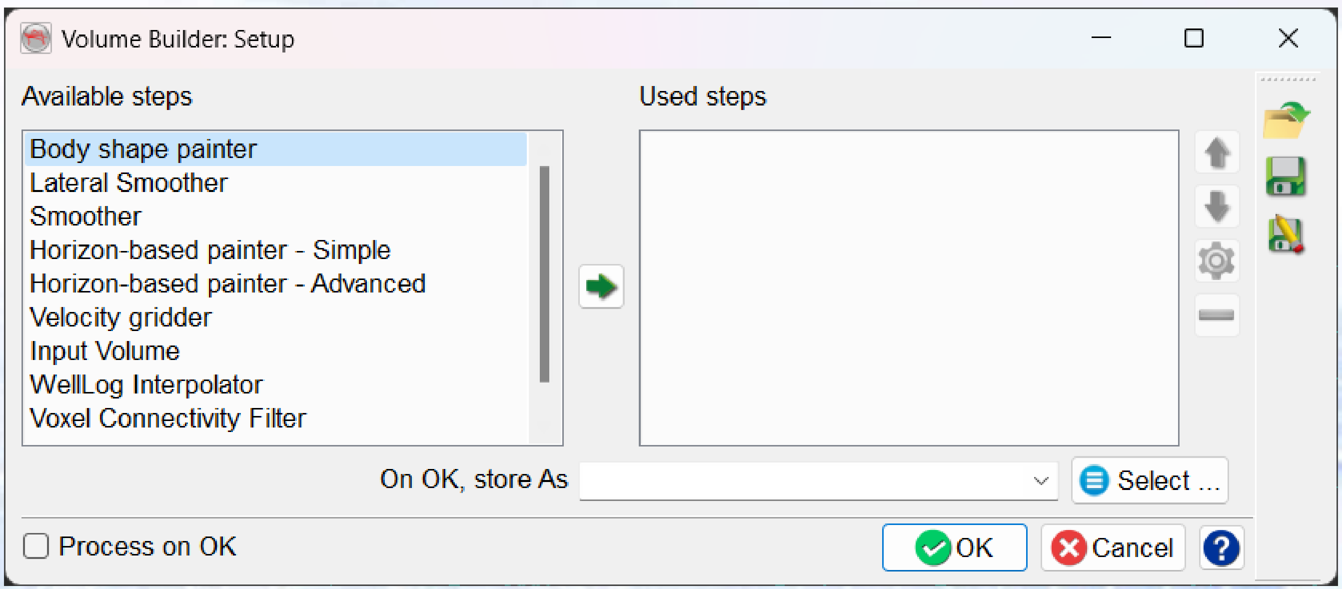
5.2.8 Well Log Interpolator

5.2 Volume Builder Setup

The volume builder setup is used to apply volume-based operations, unlike the attributes that work trace-by-trace. The setup is launched via the *Analysis > Volume Builder* menu. The setup is a very useful tool for gridding velocities or other rock properties.

Analysis > Volume Builder

对网格化速度模型或计算其他岩石特性很有用。



实施基于数据体的操作，与属性的基于trace-by-trace的方式不同。

The volume builder setup window contains several available steps that are applied sequentially to generate a volume. Any particular step can be selected from the Available steps by double-clicking on it. The later steps may replace the earlier ones, therefore care must be taken when ordering and setting up the workflow.

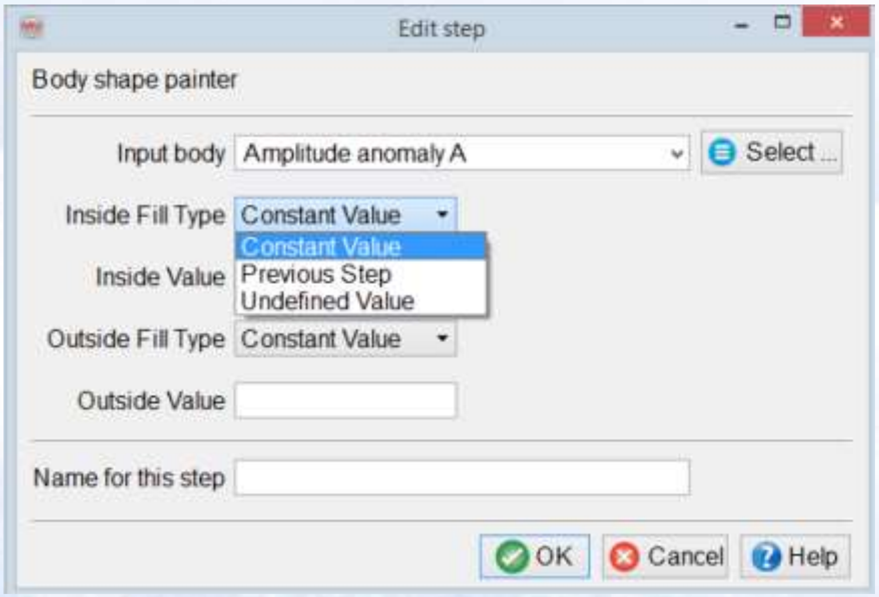
Once your workflow is defined, you will need to save this setup. This can be done by writing the name of your setup.

The computation and storage of a volume processing setup can be found under OpendTect's Processing menu: *Processing > Create Seismic Output > Volume Builder...*

顺序地执行若干步骤生成一个体(Volume)。双击，选择某具体步骤。之后操作步骤可能会替代之前的操作，因此需要注意操作顺序和设置工作流程。
一旦定义好工作流后，可以保存这个设置。可以为设置命名。
可以在Processing菜单下找到计算和存储一个volume处理的操作设置：
Processing > Create Seismic Output > Volume Builder..

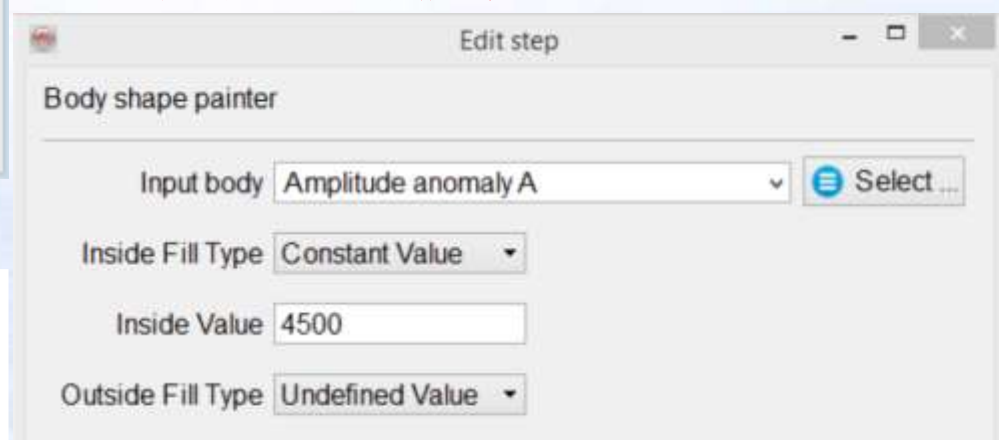
5.2.1 Body Shape Painter

The body shape painter is used to define the inside and outside values for an OpendTect body. The options for 'Inside Fill Type' are shown in the image below. The same options are available for 'Outside Fill Type':



第1种： Constant Value

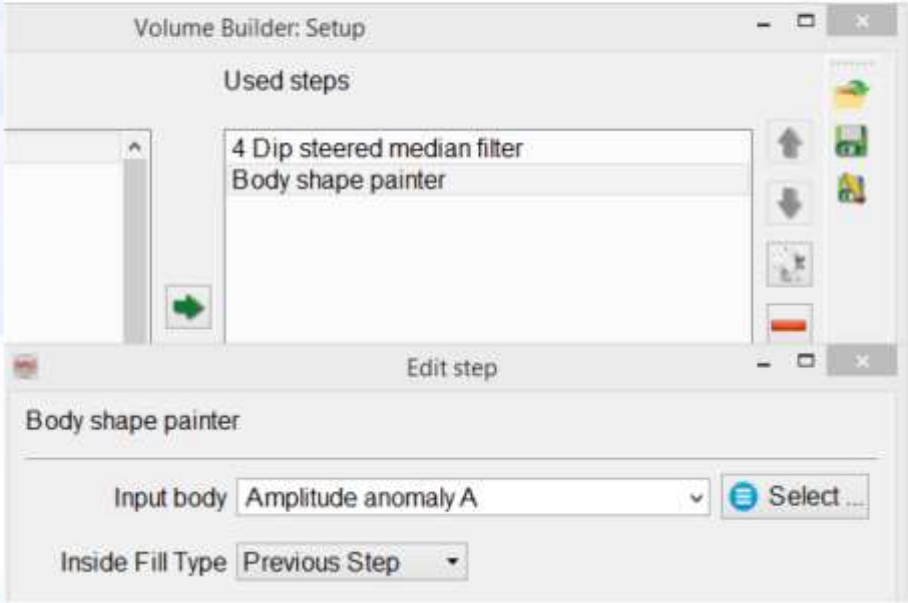
用户设置的填充值：



Fill Types:

Constant Value: Single, user-defined value:

第2种：Previous step: takes the values of the previous step in the Volume Builder setup (ie: a stored volume):



第3种：Undefined Value: Equivalent to 'transparency'

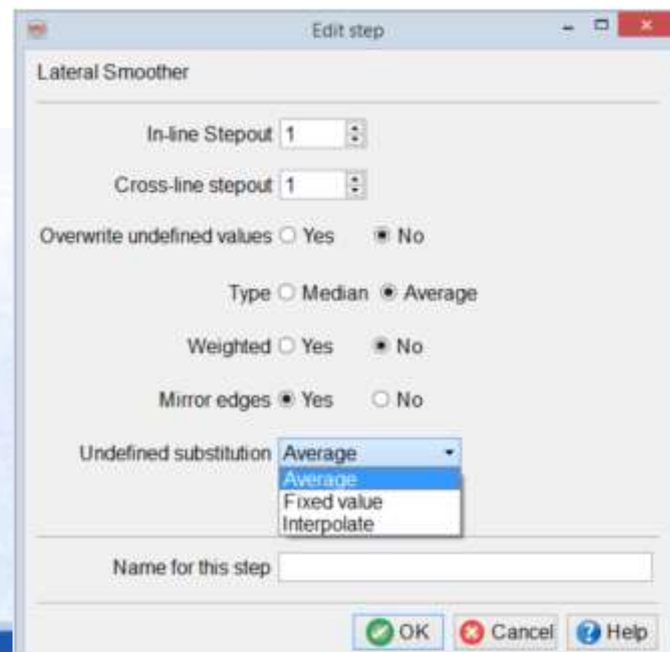
One example of use would be: if one wants to create a salt velocity cube, the values inside can be filled with a salt velocity and outside can be set to previous step. If no other step exists the undefined value is written.

5.2.2 Lateral Smoother

The lateral smoothing is a rectangular two-dimension smoothing filtering method of the volume.

The average filtering will be done in the frequency domain by applying a 2D FFT to the Z slices. This requires a rectangular dataset, while the input can be irregular. The filtering type can be chosen between "Median" or "Average" which can optionally be "Weighted". Positions without data or with undefined values are first replaced by the option "Mirror edges" and "Undefined substitution". The "Undefined substitution" can be done by taking *Average* values between the defined points, *Fixed value* (one value needs to be specified which will be used everywhere) or using *Interpolate* between the defined points.

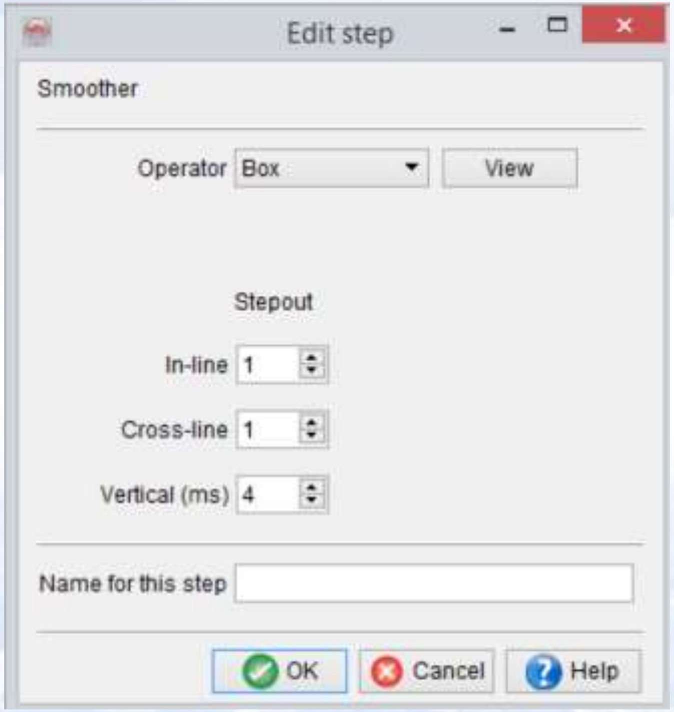
Finally, this step can be saved by giving a name.

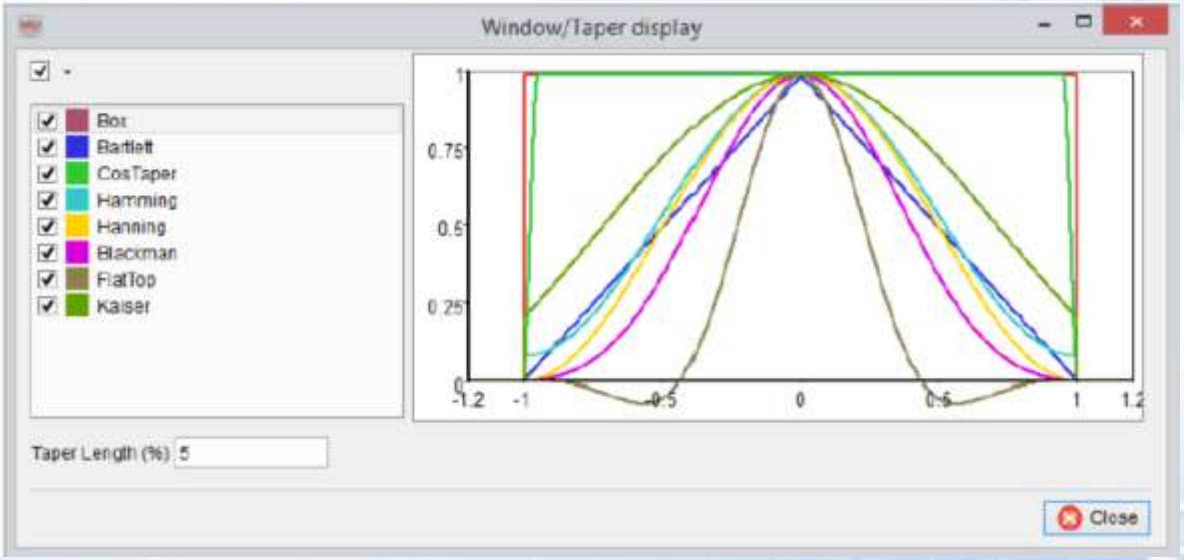


侧向光滑，是矩形区域的2D光滑过滤方法。

5.2.3 Smoother

The smoother step is used to apply a three dimensional smoothing operator by specifying In-line, Cross-line and Vertical (ms) stepouts for 3D or two dimensional smoothing with Trace and Vertical (ms) stepouts in case of 2D.

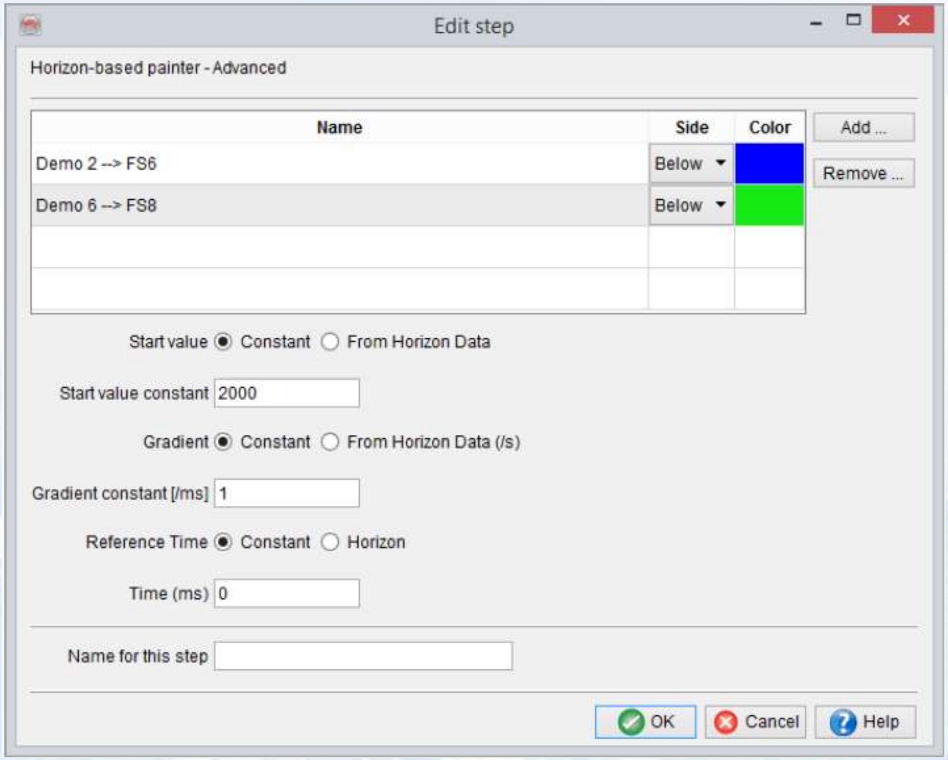




Various operator shapes can be chosen (e.g. Hamming) and can be visualized by pressing on the View button. The CosTaper also requires specification of a "Taper Length (%)".

5.2.4 Horizon-Based Painter

The horizon-based painter is used to create a model between two surfaces. The initial top and bottom values are necessary to be filled in the input. The intermediate values are interpolated to a survey or a horizon. In this window, horizons have to be selected as top/base values. The slope type is used in interpolation to define a slope.



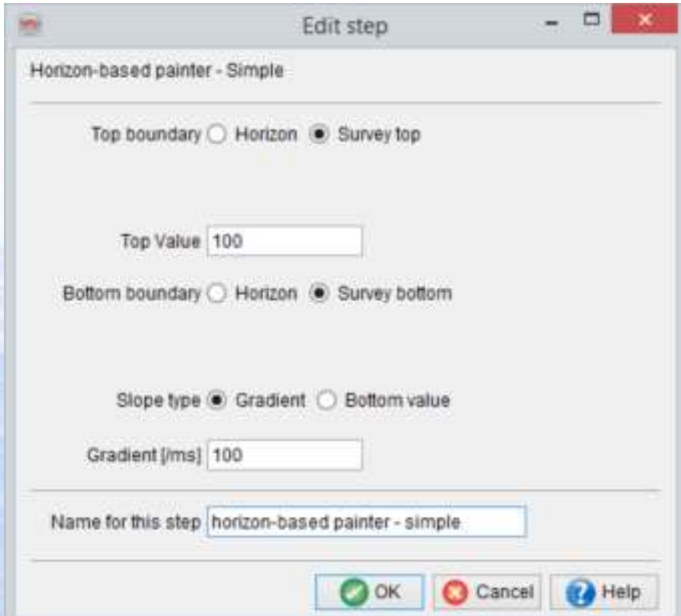
基于层位的painter，创建2个面之间的模型。

The horizon-based painter paints velocities in a 3D area or on a 2D line. Geometry of the velocities is defined by one or more horizons.

The painted velocities are referenced to a specific time. This time can be either constant (user-defined), or retrieved from a horizon and not necessarily from one of the horizons defining the limit of the body. For example, an intermediary horizon could be used.

Then velocities are painted from that reference time. The velocity must be provided as a velocity/gradient pair. The values are once again either user-defined or extracted from a surface data (grid) attached to a horizon.

There is also a much simplified Horizon-based painter also available:



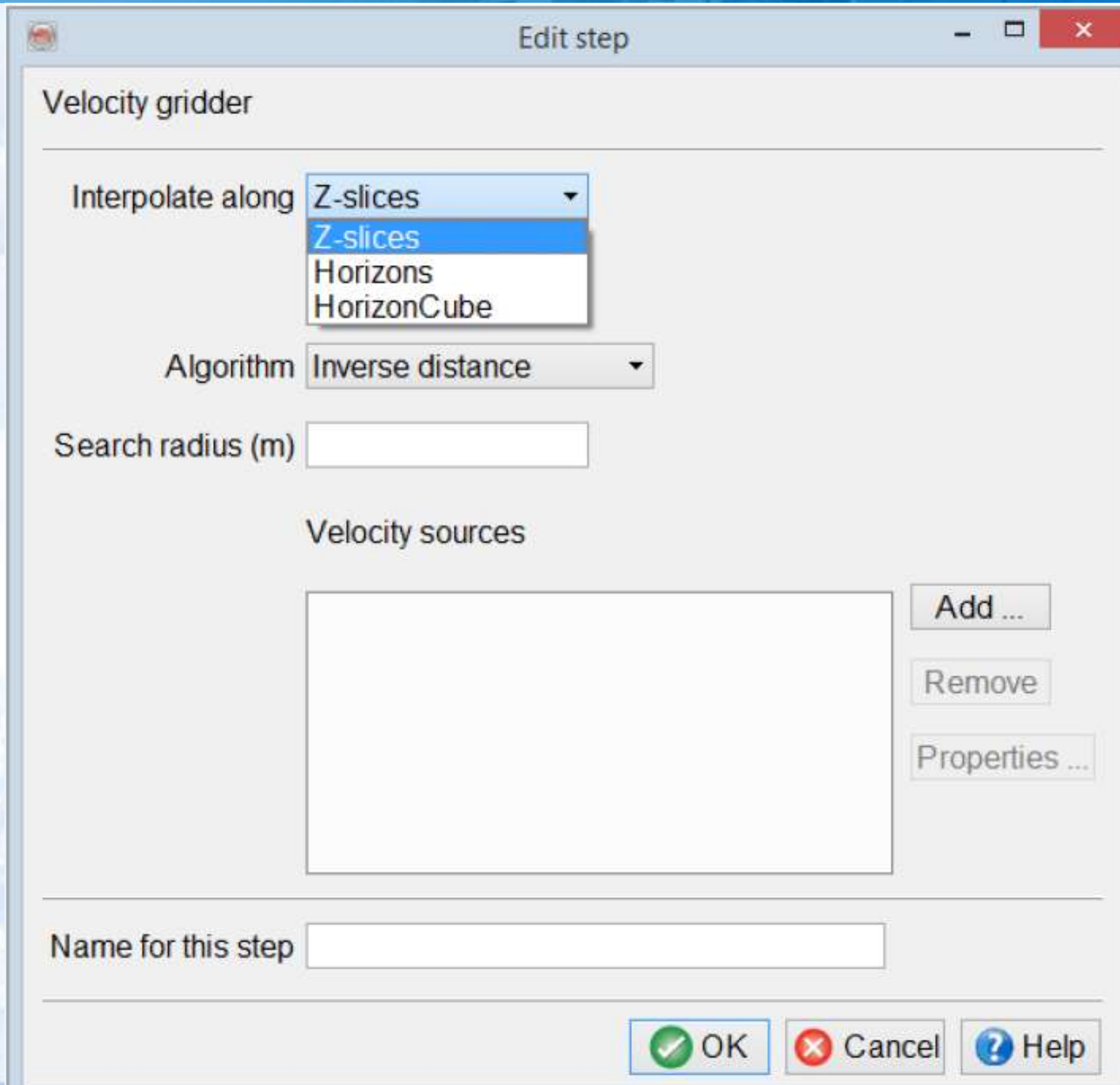
5.2.5 Velocity Gridder

The gridding will create a volume out of a sparsely sampled dataset. **The input source MUST be tagged with a velocity type.** Indeed the gridding is applied to the time-depth relation hold by the velocity source and not on the amplitudes of the velocity source. This preserves the time-depth relation and blocky-ness of interval velocity models. The gridded time-depth relation is converted back to the input velocity type in the output volume.

However any other data type could be gridded by this module (Thomsen parameters, temperatures, ...). These other types must be tagged as delta, epsilon or eta before being used for gridding. The functions will be vertically interpolated using a linear 1D interpolation before the lateral gridding.

Two interpolation methods are available for gridding: inverse distance interpolation or triangulation. The first method is designed for the interpolation of sparse dataset, while the second algorithm should be preferred if the input exists on a regular (but coarse) grid. In general gridding is always followed by some filtering.

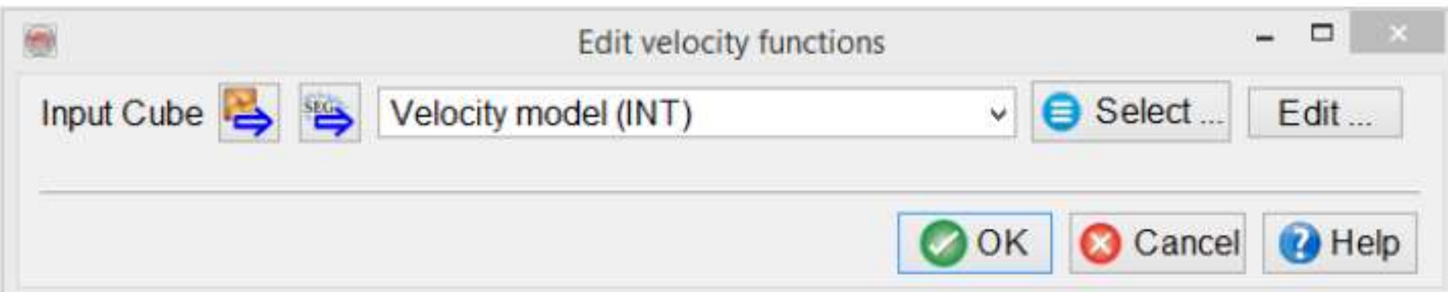
速度模型网格化，使稀疏采样的数据集变为规则的网格数据集。输入的数据源必须标记为velocity类型。使用2种插值方法：反距离插值与三角化插值。通常网格化之后，还要执行一些过滤。


Edit step
Velocity gridder
Interpolate along
Z-slices
Z-slices
Horizons
HorizonCube
Algorithm
Inverse distance
Search radius (m)
Velocity sources
Add ...
Remove
Properties ...
Name for this step
OK
Cancel
Help

The input data can be either a (coarse SEG-Y imported) volume, stored (ascii) functions or velocity points (requires the Velocity Model Building plugin)

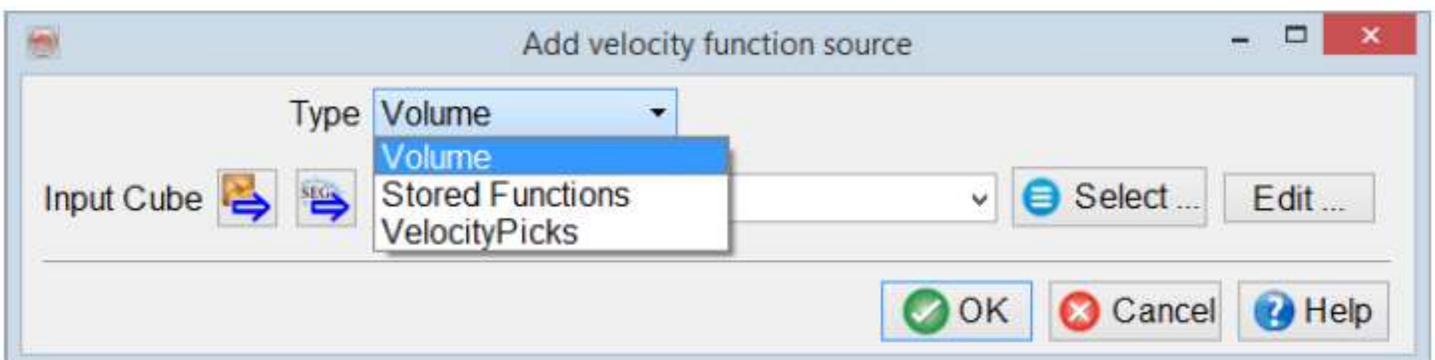
输入数据可以是：SEG-Y的数据体、ASCII的函数或速度点

Clicking on the '*Properties*' button will allow you to change the selected input for this step:

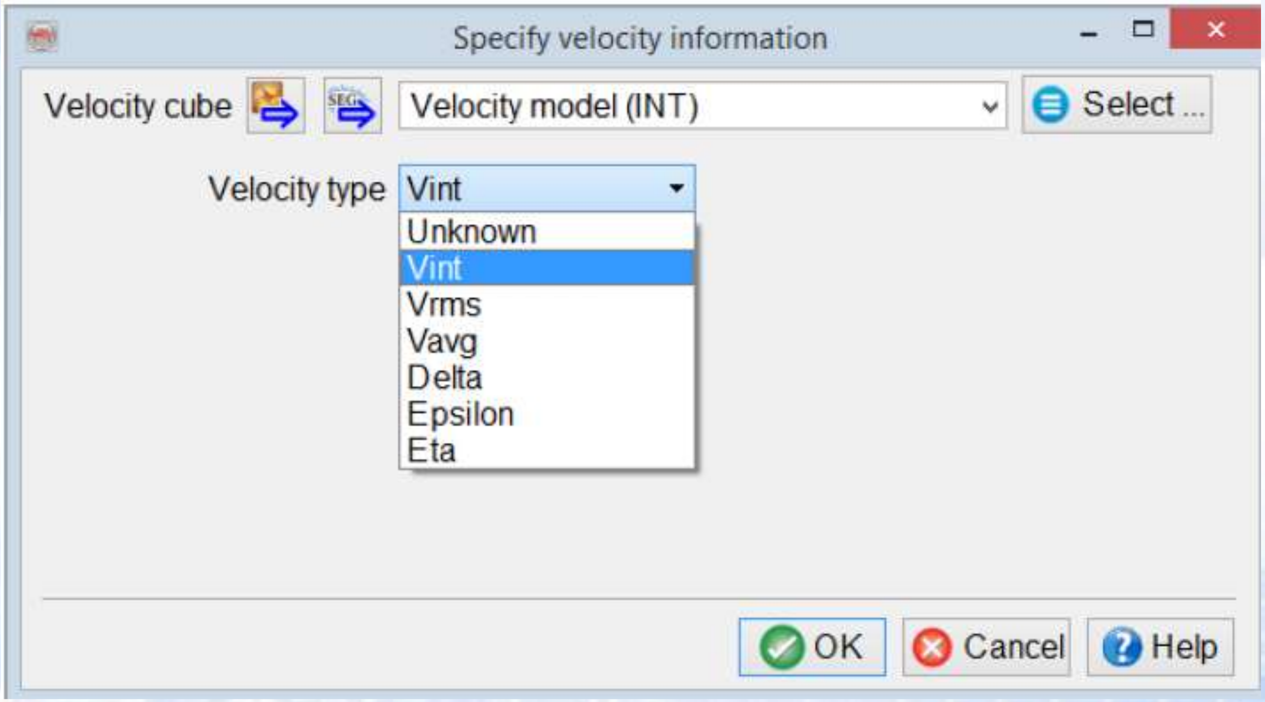


Velocity volumes have to be tagged to recognize their type.

Tip: If you have no velocity volume available, press '*Add*' in the '*Edit Step*' window and then, in the window that pops up (shown below), press '*Create*':



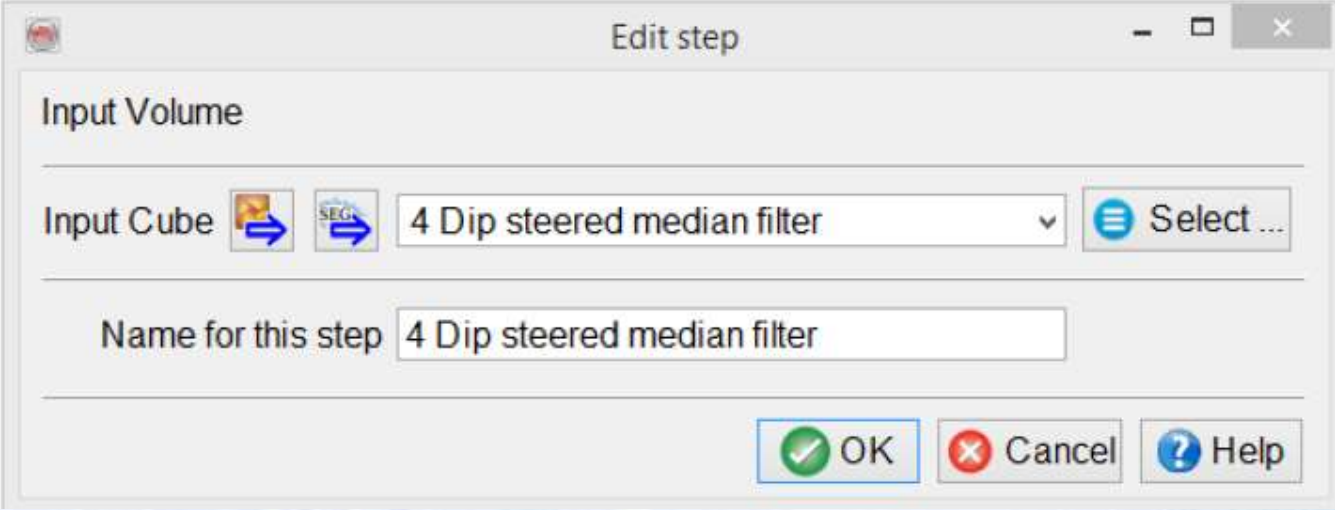
Here you may tag a volume with a velocity type, so that it can be used as input for the gridding step. Or change the tag that the volume has. This can be useful for interpolation of velocity cubes (for example, we strongly advise against trying to interpolate Vint or Vrms, but Eta-tagged volumes can be interpolated):



对导入的数据体，标记为某种速度类型。

5.2.6 Input Volume

This step is in general used to provide a background volume or 2D data attribute before using spatially constrained steps, for instance before the body shape painter or horizon-based painter.



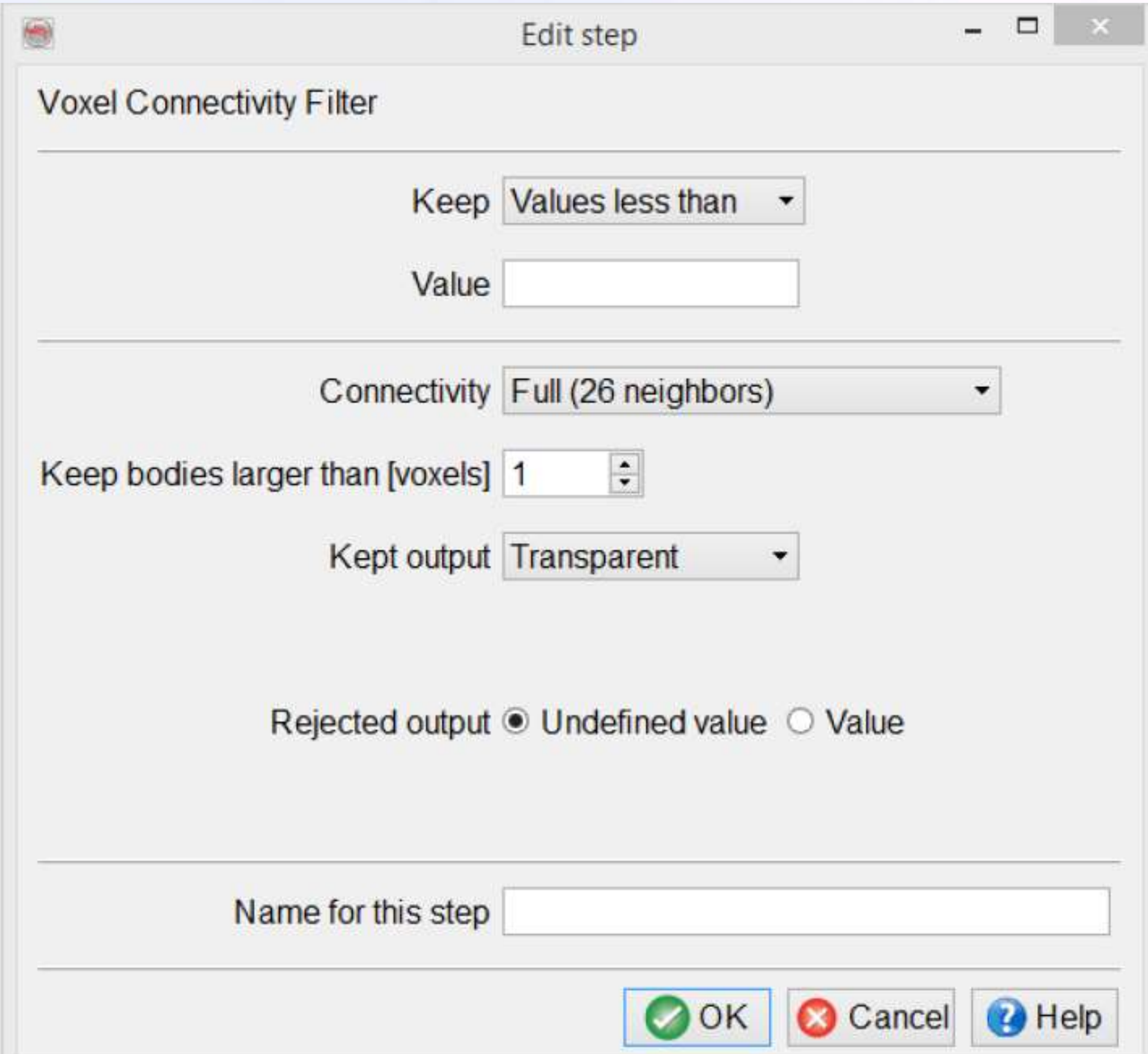
5.2.7 Voxel Connectivity Filter

Voxel Connextivity Filter is a special tool to create continuous bodies based on the amplitudes in a stored volume. A 'voxel' is defined as the volume around one sample. It is thus linked to the survey bin size and sampling rate.

This volume builder step must be preceded by a step providing the necessary input data, like "Stored volume".

This volume builder step implies a volumetric calculation. The result of the application on a single inline will differ from the result of the application to the whole volume.

The filter is based on a user-defined amplitude selection to compute the bodies. The samples interconnection is computed based on an amplitude criteria and geometrical spreading settings. It is a very useful tool to visualize seismic attributes in 3D. Other benefits of this tool are to get a volume of several bodies and visualize them in 3D or use it as an input to supervised Neural Network. A general and most popular use of this tool can be the DHIs detection. For instance, if creating a volume that represents DHIs only, it may be interesting to clip the amplitudes to visualize the DHIs present in the seismic data. For such case, when having the seismic amplitude attribute as a volume, this filter can be used to create new DHI volume.


Edit step
Voxel Connectivity Filter
Keep
Values less than
Value
Connectivity
Full (26 neighbors)
Keep bodies larger than [voxels]
1
Kept output
Transparent
Rejected output
Undefined value
Value
Name for this step
OK
Cancel
Help

The voxel connectivity filter has a number of parameters to set:

Keep: Specifies the part of the input dataset used to compute the bodies, based on their amplitudes.

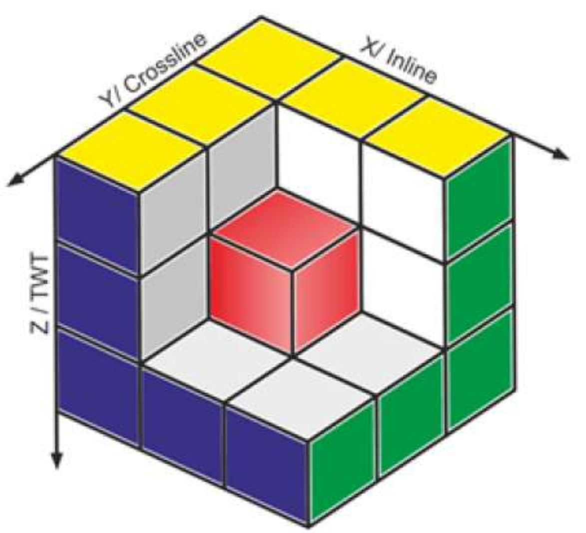
- Values more than: The envelope of the amplitudes higher than the given value define the bodies to be computed. Example: 0 will select all positive amplitudes.
- Values less than: The envelope of the amplitudes lower than the given value define the bodies to be computed. Example: 0 will select all negative amplitudes.
- Values between: The envelope of the amplitudes between inside the given range define the bodies to be computed. Example: 9000, 14000 will select all values in between, like 12000.

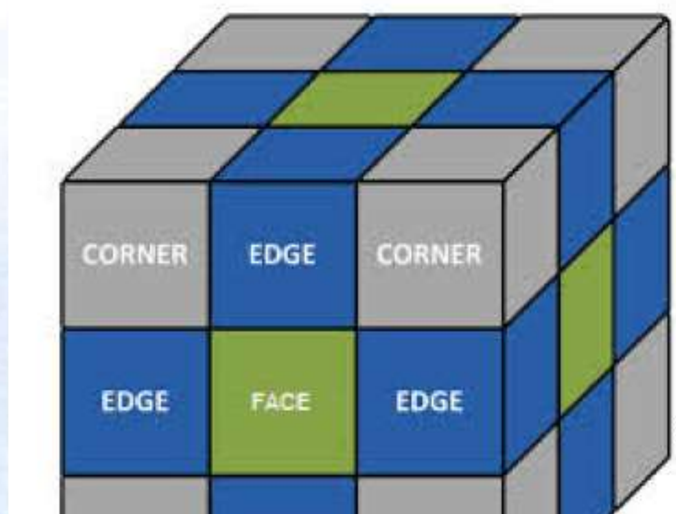
- Values outside: The envelope of the amplitudes between outside the given range define the bodies to be computed. Example: -10000, 10000 will select all values lower than -10000 or larger than +10000 (the extremes).

Connectivity: Selects the method used to connect different voxels when computing the bodies. Each sample in the input volume acts like a seed.

- Common Faces (6 neighbours): The propagation is done by strictly using the 6 faces adjacent to the current seed.
- Common Edges (18 neighbours): The propagation is done by using the 6 faces and the 12 edges adjacent to the current seed.
- Full (26 neighbours): The propagation is done is all directions, using the 6 faces, 12 edges and 8 corners adjacent to the current seed. This is the default mode.

The easiest way to visualize the connectivity is to imagine the reference voxel as the central voxel in a 3x3x3 cube, such as the red one in the first image. Then the second image shows the Face-, Edge- and Full (corner) conecctions:





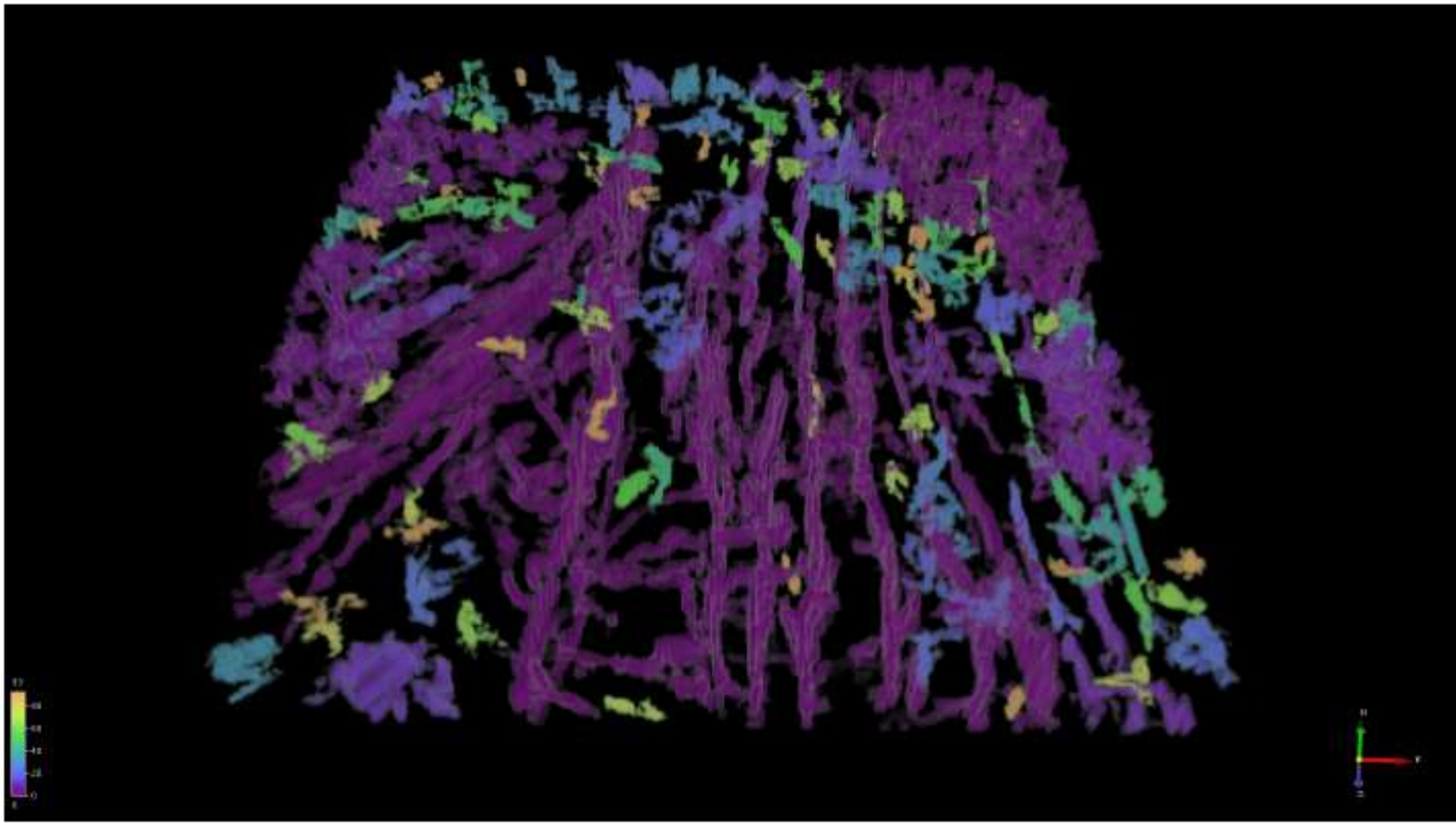
Keep bodies larger than [voxels]: It defines the minimum number of voxels required to output a body. Actually all bodies are computed in the first pass. The smallest bodies are then dismissed. Minimum allowed is one.

Keep output: The following value(s) will be output on the samples inside the computed bodies:

Body-size rank:The output value is an integer with a constant, different value for each body. The values are sorted by decreasing body size, starting at zero: 0 is the largest body, 1 the second largest...

The example below is created using a similarity attribute to locate faults and fractures in a volume. It is set-up to create bodies connecting low similarity values (threshold of 0.5). All values that are above this threshold are ignored. Furthermore, it is also ignores the very small bodies (size < 10 voxels).

It shows number of connected bodies (purple being the largest ones) in a volume. Such a result can directly show which faults are connected and those that are not. Visualizing such a VCF result can be a valuable method in performing direct interpretation.



Body-size: The output value is the size in number of voxels of each body. This gives an approximation of the real-world volume, when multiplying by the bin size. For example, a body of 2500 voxels (10 inlines, 50 crosslines, 5 samples), with a bin size 25m x 12.5m, at 4ms sampling with a constant velocity of 2000 m/s: Vol = 2500 * 25 * 12.5 * 2000*0.004/2 = 10000 m3...

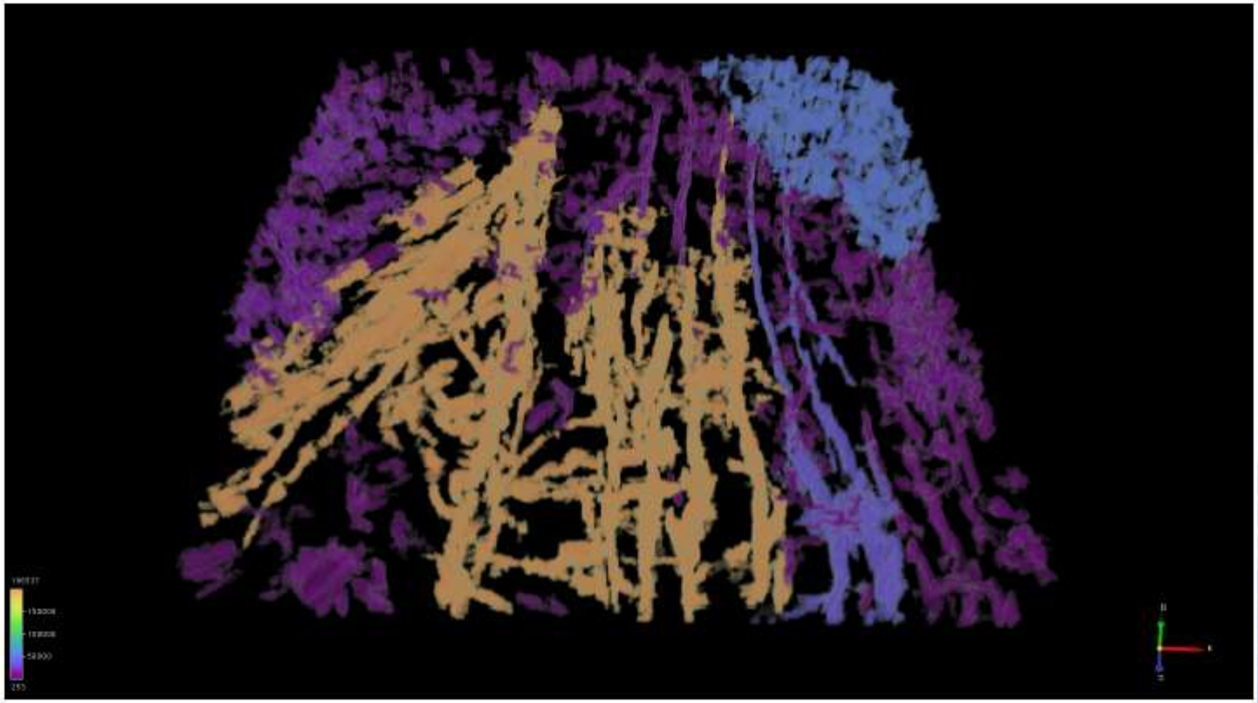
In this second example, the same volume is being processed for Body-size. It shows the same patterns suggesting that the prediction is identical to the earlier result. However, the predicted voxels are being filled differently. Here the same bodies are defined by largest volume in cubic meters (m3).

Generally speaking, areas of higher faults/fractures density allow greater connectivity between bodies.This example below shows this case.

- Value: The output value is a user-defined value specified in the "Kept value" field underneath.
- Transparent: The output value is taken from the amplitude in the input volume.

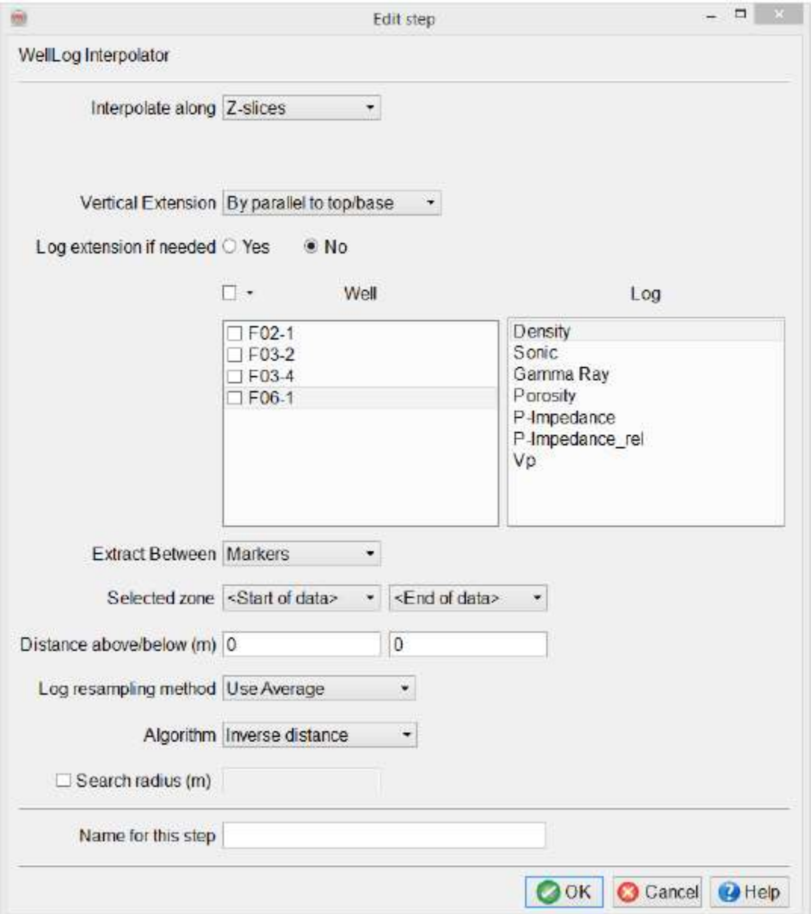
Rejected output: The value outside the computed bodies can be either the undefined value or a user-defined value specified by the field "Rejected value" underneath.

Name for this step: Provide a user-defined name for this volume builder step that will appear in the Used-steps list of the Volume builder.



5.2.8 Well Log Interpolator

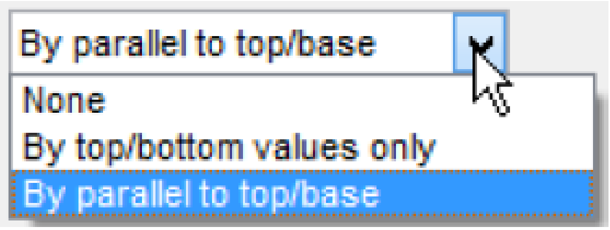
This gridding step is used to populate a 3D volume using well logs by interpolating along Z-slices.



插值测井数据，沿着Z-slices

Vertical Extension: Select the method of vertical extension from the following options:

- **Log extension if needed:** Extend the logs (if required) to match the *Selected zone*
- **Extract Between:** Extract data from a marker-defined, depth-defined or time-defined range. You may also toggle on the option to extract the data in time.
- **Selected zone:** Set the extraction zone suing either markers or start and end of data (or combination thereof).
- **Distance above/below:** Extend, if desired, the extraction zone above and below the selected zone.
- **Algorithm:** Choose between inverse distance or triangulation.
- **Search radius:** For inverse distance only - set an optional maximum search radius for the algorithm.



After the selection of well(s) and log, parameters and algorithm, provide a name for this step at the bottom and proceed to the Volume Builder by pressing *'OK'*.