

Tsunami Toolbox: Horizon, Seismic,
and Velocity Modeling tools



**Tsunami
Development**

Horizon Tools.....	6
HORIZON SMOOTHING 3D	7
<i>INTRODUCTION</i>	7
<i>Setup</i>	7
<i>Using Horizon Smoother 3D</i>	7
APPENDIX- HORIZON SMOOTHING 3D	9
<i>HORIZON NAMES</i>	9
<i>SMOOTHING OPTIONS</i>	9
<i>HOW TO CREATE HORIZONS</i>	9
HORIZON DEPTH CORRECTION	10
<i>INTRODUCTION</i>	10
<i>SETUP</i>	10
<i>USING HORIZON DEPTH CORRECTION</i>	10
APPENDIX.....	12
<i>HORIZON NAMES</i>	12
<i>Convert Horizons</i>	12
<i>Files</i>	12
<i>HOW TO CREATE HORIZONS</i>	12
HORIZON INTERPOLATION	13
<i>Introduction</i>	13
Seismic Tools	15
ANGLE MUTE.....	16
<i>Introduction</i>	16
<i>Angle Mute Dialog</i>	16
FLOATING DATUM.....	18
<i>INTRODUCTION</i>	18
<i>SETUP</i>	18
<i>USING THE FLOATING DATUM UTILITY</i>	18
APPENDIX	20
<i>FILES</i>	20
<i>DATA TYPE</i>	20
<i>FLOATING DATUM</i>	20

<i>PLUG-IN INDEXING FORMATS</i>	20
SEISMIC MUTE	21
<i>Introduction</i>	21
<i>Setup</i>	21
<i>Using the Mute Plug-In</i>	21
APPENDIX	25
<i>INPUT PARAMS.</i>	25
<i>MUTE OPTIONS</i>	25
<i>OUTPUT OPTIONS</i>	25
STACK	27
<i>INTRODUCTION</i>	27
<i>SETUP</i>	27
<i>USING STACK</i>	27
<i>NORMALIZING</i>	27
<i>USE MINIMUM MIGRATION DISTANCE FOR STACK</i>	27
<i>STACKING POWER</i>	27
APPENDIX	28
<i>FILES</i>	28
<i>STACKING</i>	28
<i>STACK INDEXING FORMATS</i>	28
Time Or Depth Conversion	29
<i>INTRODUCTION</i>	29
<i>SETUP</i>	29
<i>USING THE TIME OR DEPTH CONVERSION PLUG-IN</i>	29
APPENDIX	31
<i>FILES</i>	31
<i>INPUT PARAMETERS</i>	31
<i>FLOATING DATUM</i>	31
<i>OUTPUT PARAMETERS</i>	31
<i>TIME OR DEPTH CONVERSION INDEXING FORMATS</i>	31
Velocity Tools	32
ASCII CONVERTER	33
<i>INTRODUCTION</i>	33

<i>SETUP</i>	33
<i>USING ASCII CONVERTER</i>	33
<i>LOADING A DEFAULT FILE FORMAT</i>	34
<i>LOADING A CUSTOM FILE FORMAT</i>	34
APPENDIX.....	35
<i>PARTIAL CONTENT OF INPUT FILE</i>	35
<i>INPUT FILE DESCRIPTION</i>	35
<i>SKIP LINE</i>	35
<i>DEFAULT COLUMNS</i>	35
<i>OUTPUT PARAMETERS</i>	36
<i>VERTICAL UNIT (Z)</i>	36
P VELOCITY TO S VELOCITY CONVERTER.....	37
<i>INTRODUCTION</i>	37
<i>SETUP</i>	37
<i>USING P VELOCITY TO S VELOCITY CONVERTER</i>	37
APPENDIX.....	38
<i>FILES</i>	38
<i>P VELOCITY TO S VELOCITY CONVERTER INDEXING FORMATS</i>	38
Velocity Model resampler.....	39
<i>INTRODUCTION</i>	39
<i>SETUP</i>	39
<i>USING VELOCITY RESAMPLER</i>	39
APPENDIX.....	41
<i>FILES</i>	41
<i>VELOCITY MODEL TYPE</i>	41
<i>DATASET DIMENSIONS</i>	41
<i>VELOCITY RESAMPLER INDEXING FORMATS</i>	41
VELOCITY MODEL UPDATE.....	41
<i>Introduction</i>	41
<i>USING THE FLOOD OPTION</i>	42
<i>USING THE TOP/BOTTOM OPTION</i>	44

The Information contained in this document is subject to change without notice. Tsunami Development assumes no responsibility for any error that may appear in this manual. Tsunami Development does not warrant that this document is error free. Please report any errors in this document in writing to Tsunami Development. Some states or jurisdictions do not allow disclaimer of expressed or implied warranties in certain transactions; therefore this statement may not apply.

HORIZON TOOLS

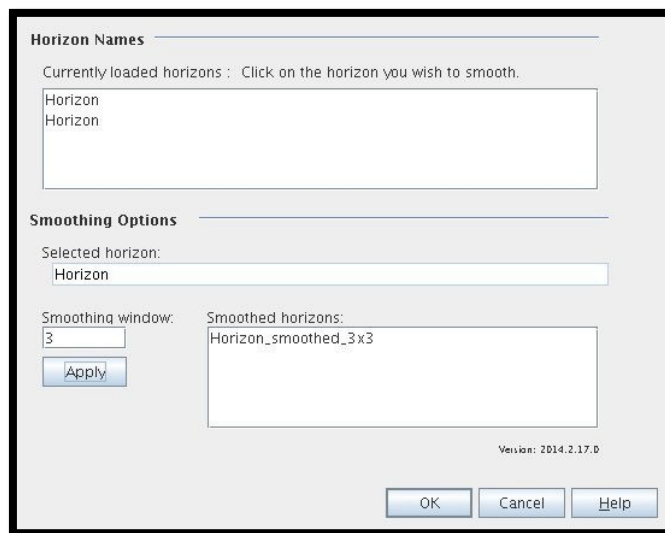
HORIZON SMOOTHING 3D

INTRODUCTION

The 3D Horizon Smoother takes an input horizon (20 or 30) and smooths the horizon by the specified smoothing window. These horizons are also interpolated in all directions when smoothed. Horizons picked on a 10x10 grid will be smoothed and output on a 1x1 grid.

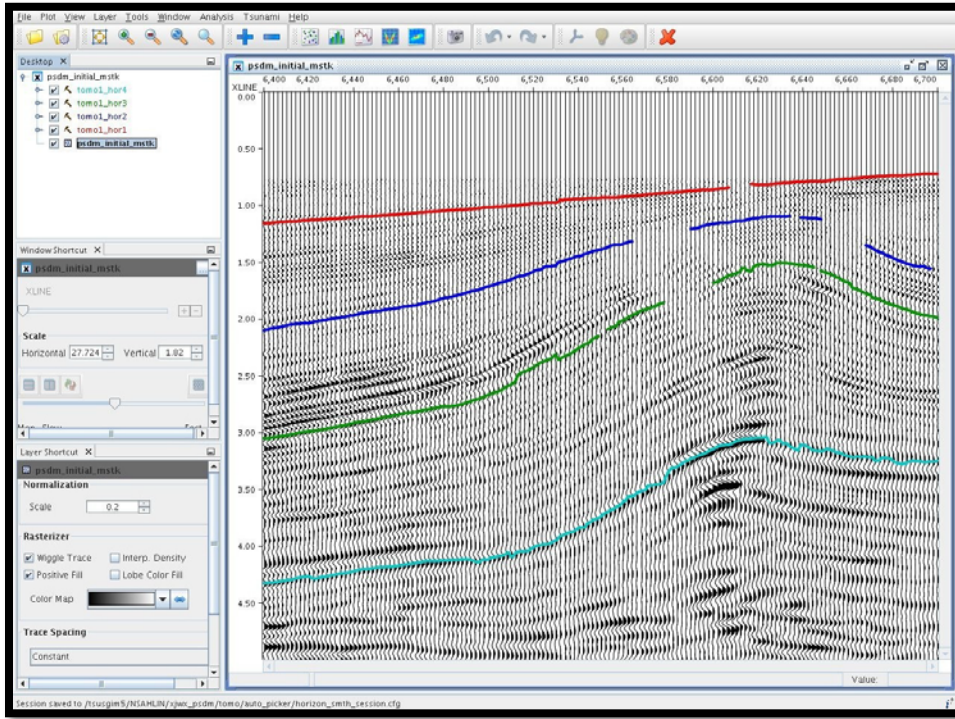
SETUP

The 3D horizon smoother needs a seismic dataset with horizons layers to open. The horizons available for smoothing will be shown in the "Currently loaded horizons:" text box.

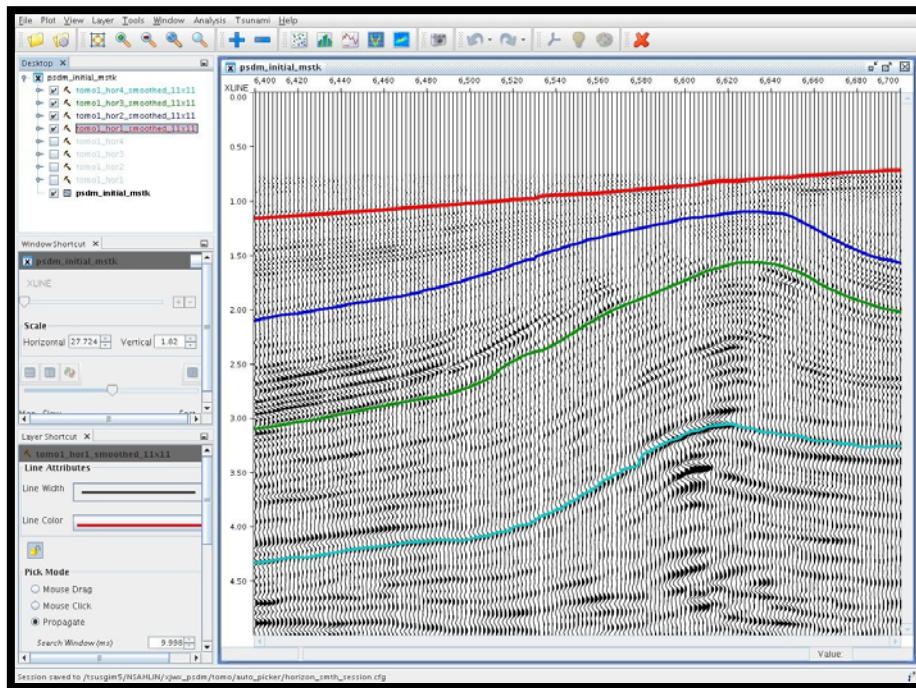


USING HORIZON SMOOTHER 3D

To smooth a horizon, first click on the horizon name in the "Currently loaded horizons:" text box. The selected horizon's name will appear in the "Selected Horizon" text box below. Then input the smoothing window desired. The smoothing window does not have a maximum limit, but does need to be an odd number starting with a window size of three (3,5,7 etc...). Click the "Apply" button to output the smoothed horizon. Repeat this process for multiple horizons or different smoothing windows. Larger smoothing windows will create more gently sloping horizons.



Example Input Horizons



Example Output Horizons

APPENDIX- HORIZON SMOOTHING 3D

HORIZON NAMES

Currently loaded horizons: The horizons available for smoothing which are shown as layers on the active dataset. Click on a horizon to select it for smoothing.

SMOOTHING OPTIONS

Selected horizon: The horizon currently selected for smoothing. Change the horizon by clicking on a different horizon in the "Currently loaded horizons" text box.

Smoothing Window: The window that the smoother will use to create and interpolate the new horizon. This parameter must be an odd number and cannot be smaller than a size of 3.

Apply: Click this button to apply the smoothing operator to the selected horizon.

Smoothed Horizons: A list of the horizons that have already been smoothed by the plug-in. The user may not replicate a horizon-smoother pair that has already been processed.

HOW TO CREATE HORIZONS

Please see the "Create New Horizon" INTViewer help file for more information about horizons.

HORIZON DEPTH CORRECTION

INTRODUCTION

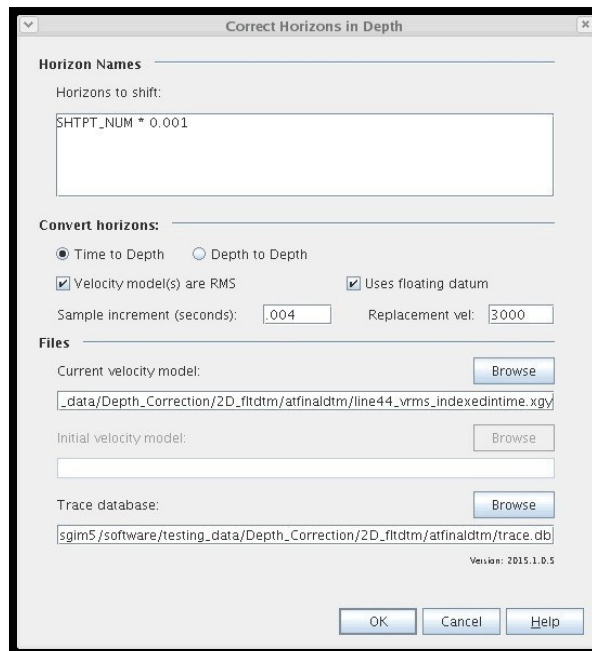
The Horizon Depth Correction plug-in is used to convert horizons from time to depth or from a previous depth model to a new depth model. These horizons are used in many Tsunami processing workflows such as converting an RMS velocity model to an interval velocity model. Plug-ins that need depth corrected horizons will state this in their own respective help files.

SETUP

The Horizon Depth Correction plug-in needs at least one horizon present on a dataset (velocity or seismic). When the plug-in starts the horizon(s) that will be corrected will appear in the "Horizons to shift:" textbox.

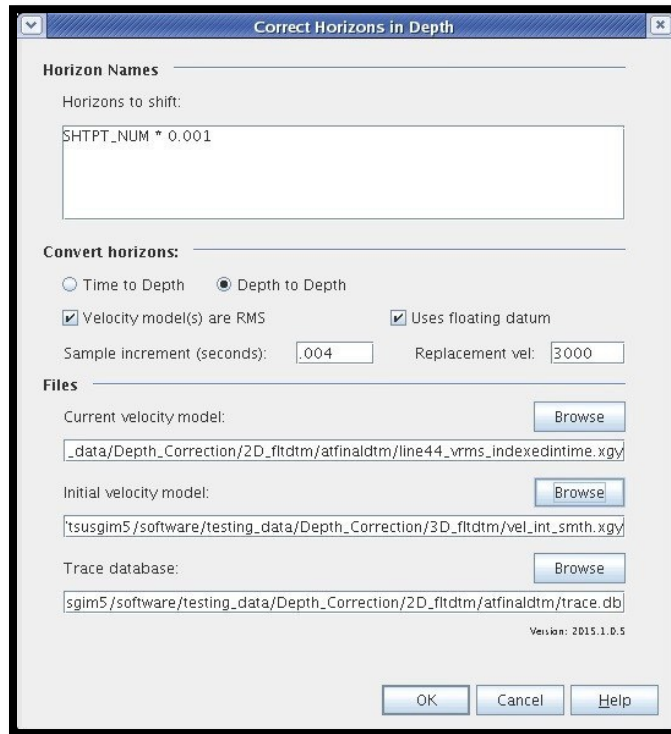
USING HORIZON DEPTH CORRECTION

Input horizons can be in depth or time, however both conversion options will lead to a depth solution. It is suggested to have a fine velocity model (5x5, 2x2, 1x1) when using this plug-in for better accuracy. If the velocity model has a sample increment of less than 10x10 it is probably best to use the [Velocity Resampler](#) plug-in to achieve a finer sample increment.



Input Horizons are in Time

When this option is selected only one velocity model needs to be applied. The velocity model can be in time (RMS) or in depth (interval). In general an interval velocity model will give more accurate results. After the model is supplied, press the "OK" button to generate the corrected horizons in depth. Save these horizons using INTViewer.



Horizons are in Depth

Choose this situation when the velocity model used for migration has been updated. For example, if the user has made an initial velocity model and then updated that velocity model using tomography then these two models will be needed to update the depth horizons. Specify the initial model (used when creating horizon) and the updated model (Current velocity model). Press "OK" to generate the updated depth horizons and save the horizons using INTViewer.

APPENDIX

HORIZON NAMES

Horizons to shift: Horizons specified in this text box will be corrected by the plug-in. The user cannot select or deselect horizons in this box. If a desired horizon is not shown in this box make sure that the horizon is active on the data selected before starting the plug-in.

CONVERT HORIZONS

Time to Depth: Choose this option when the horizons to be shifted are made in the time domain. An RMS (time) or Interval (depth) velocity model can be used to shift these horizons to depth.

- Depth to Depth: Choose this option when the horizons to be shifted are made in the depth domain. The initial and current velocity models specified must be interval velocity models for a correct depth shift.
- Sample increment: This is the sample increment in seconds of the dataset the horizon was picked on.
- Uses floating datum: Choose this option when using an RMS velocity model to prevent any inversion problems in the time to depth calculations when velocity model is a final datum.

FILES

Current Velocity Model: For Time to Depth: The velocity model used to create the horizons. For Depth to Depth: the velocity model used to update the horizons.

Initial Velocity Model: The velocity model used to create the initial horizons.

HOW TO CREATE HORIZONS

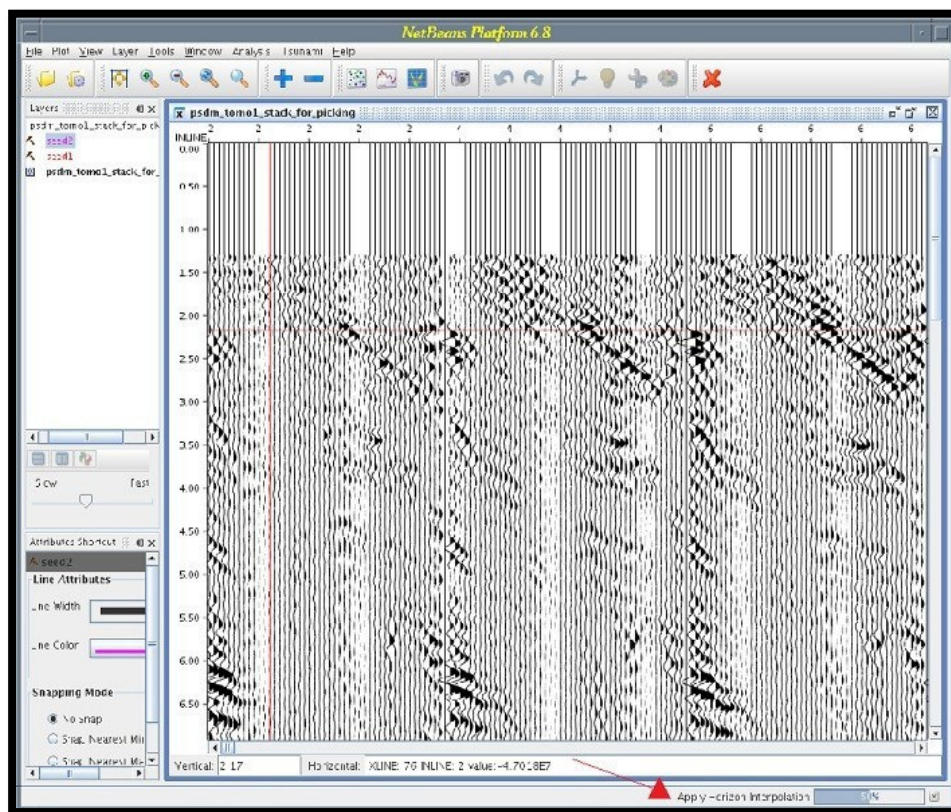
Please see the "Create New Horizon" INTViewer help file for more information about [horizons](#)

HORIZON INTERPOLATION

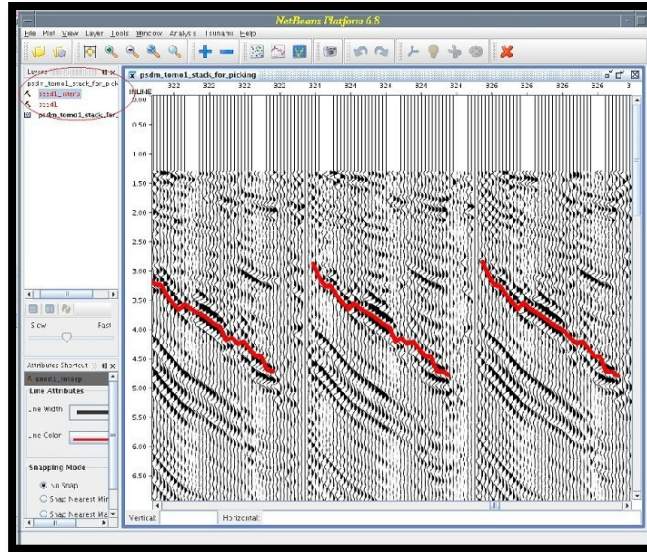
INTRODUCTION

The Horizon Interpolation option provides horizon interpolation capabilities to INTViewer. A seismic dataset needs to be loaded into the INTViewer and the horizons you want to be interpolated added to the seismic view as layers. This module will take all horizon layers displayed in the selected XSection window and loop through each horizon interpolating between picked points. A status bar is displayed in the lower right hand corner of the viewer to track the interpolation process. The interpolation is run as a background process so you may continue to use the INTViewer.

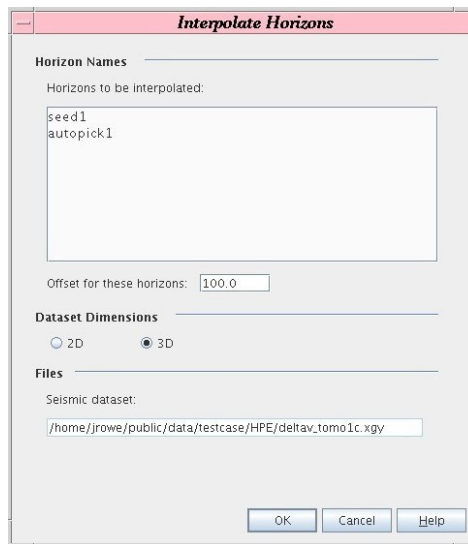
Please note: If the INTViewer is closed before completion, the work will be lost.



When all interpolation is complete new horizons with an interp extension to the name are created and displayed in the XSection window. Please note, the interpolated horizons must be saved using the menu option Layer->Save horizon...



The list of Horizons to be interpolated are taken from the selected XSection window and cannot be edited. The horizons need to be added or deleted from the selected XSection window.



To add or subtract horizons from the list, close the Horizon Interpolation dialog by clicking the Cancel button. To add horizons to the selected XSection window, select menu option Layer->Add Layer->Horizon....

To delete horizons from the selected XSection window, move the cursor over the horizon name in the Layers window to the left of the XSection window and click the right mouse button. Then on the menu, select Delete Layer. When the proper horizons are displayed, reselect the menu option Tsunami->Horizon->Interpolation to reopen the dialog box. The new list of horizons will be displayed. Offset is used to supply the newly interpolated horizon with the appropriate offset header value

Select either 2D interpolation or 3D interpolation for interpolating between the user's picked points.

Click the OK button to begin the interpolation.

SEISMIC TOOLS

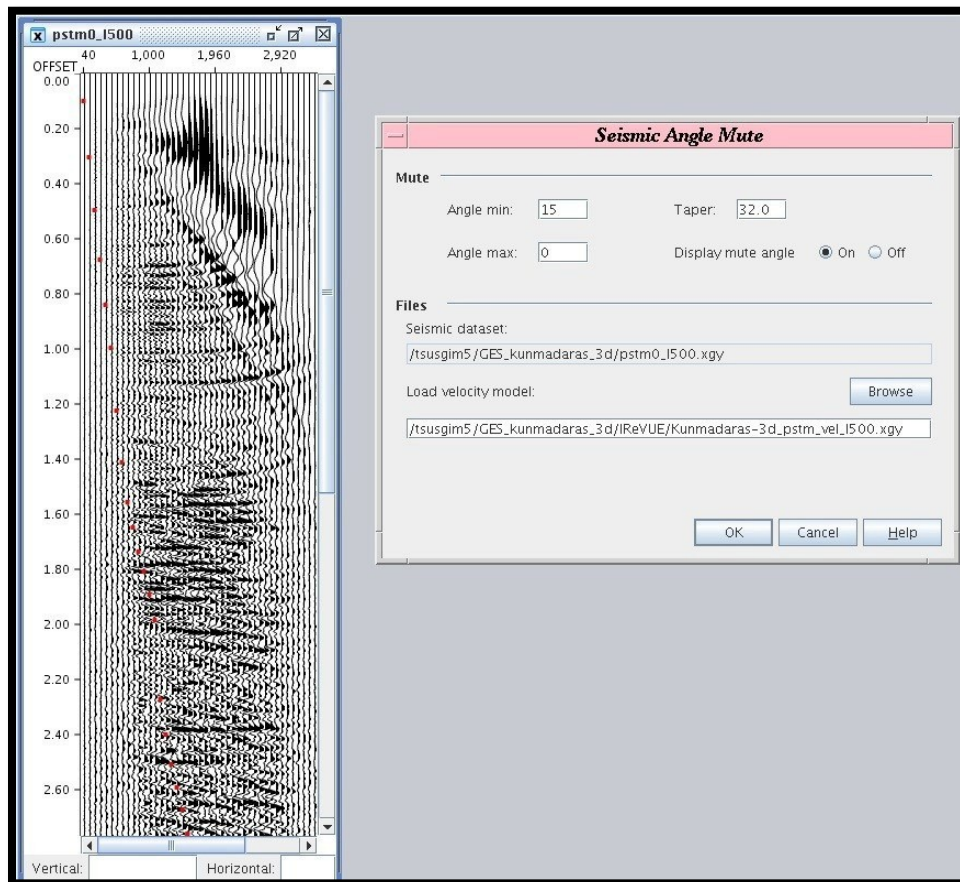
ANGLE MUTE

INTRODUCTION

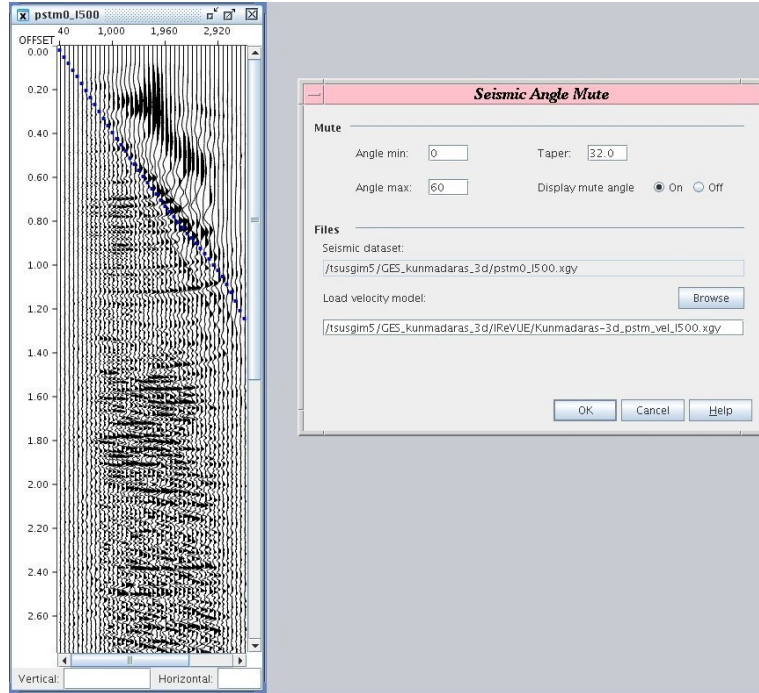
The Angle Mute plugin allows the user to apply an angle mute to seismic gathers. The mute is calculated by using the RMS velocity model.

ANGLE MUTE DIALOG

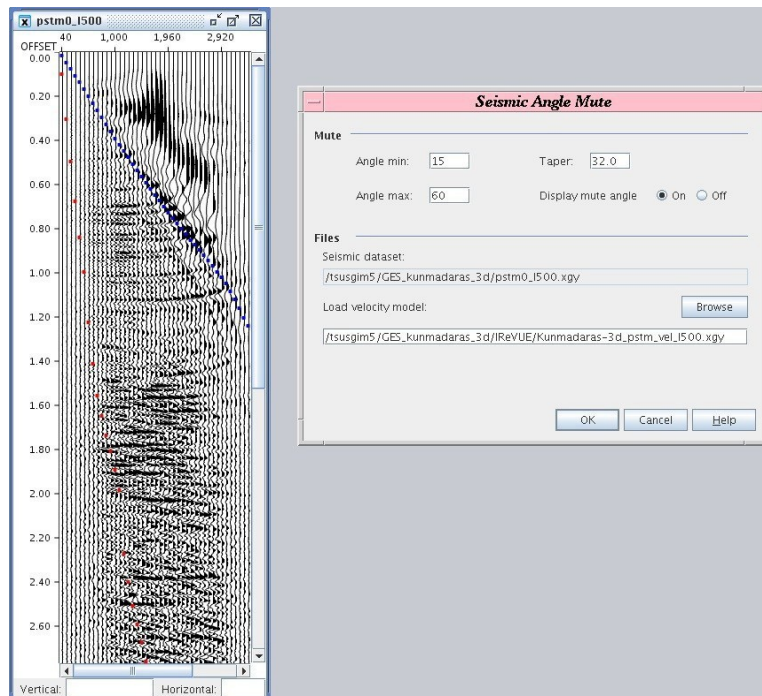
A minimum mute angle of 15 degrees is shown below.



A maximum mute angle of 60 degrees is shown below.



A minimum mute angle of 15 degrees and a maximum mute angle of 60 degrees is shown below.



A gather with a minimum mute angle of 15 degrees and a maximum mute angle of 60 degrees applied.

FLOATING DATUM

INTRODUCTION

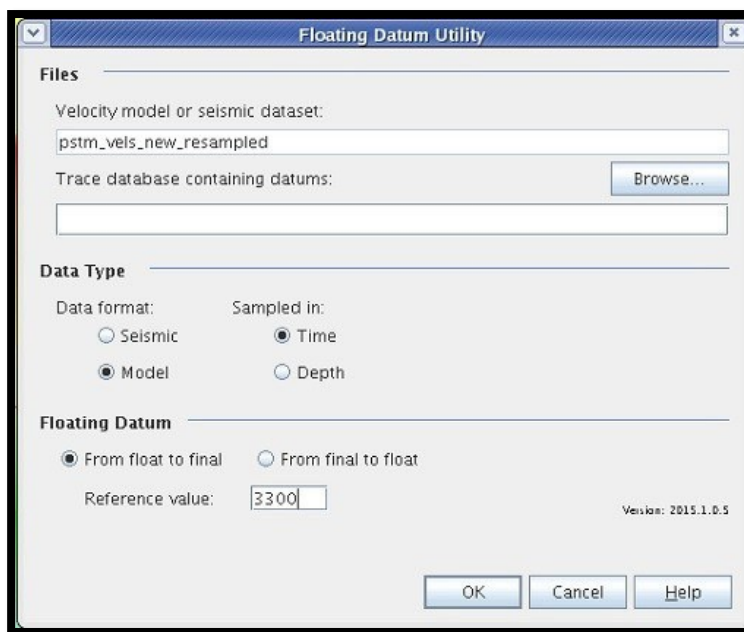
The floating datum utility can be used to shift a velocity model (RMS or interval) or stacked seismic data by the floating datum value. When shifting velocity models from floating datum to final datum the plug-in will also input a reference velocity chosen by the user in the space created.

SETUP

Use one of Tsunami's Modules (PSTM, PSDM, RTM) to build a trace database. This can be done by selecting the "Build trace database only" option on the "FILES" tab. When building the trace.db file the user must also specify the floating datum information on the "INPUT" tab in order to include this information in the trace database. Open the Indexed velocity model/Seismic data (indexed by INLINE and XLINE) to begin using the floating datum utility.

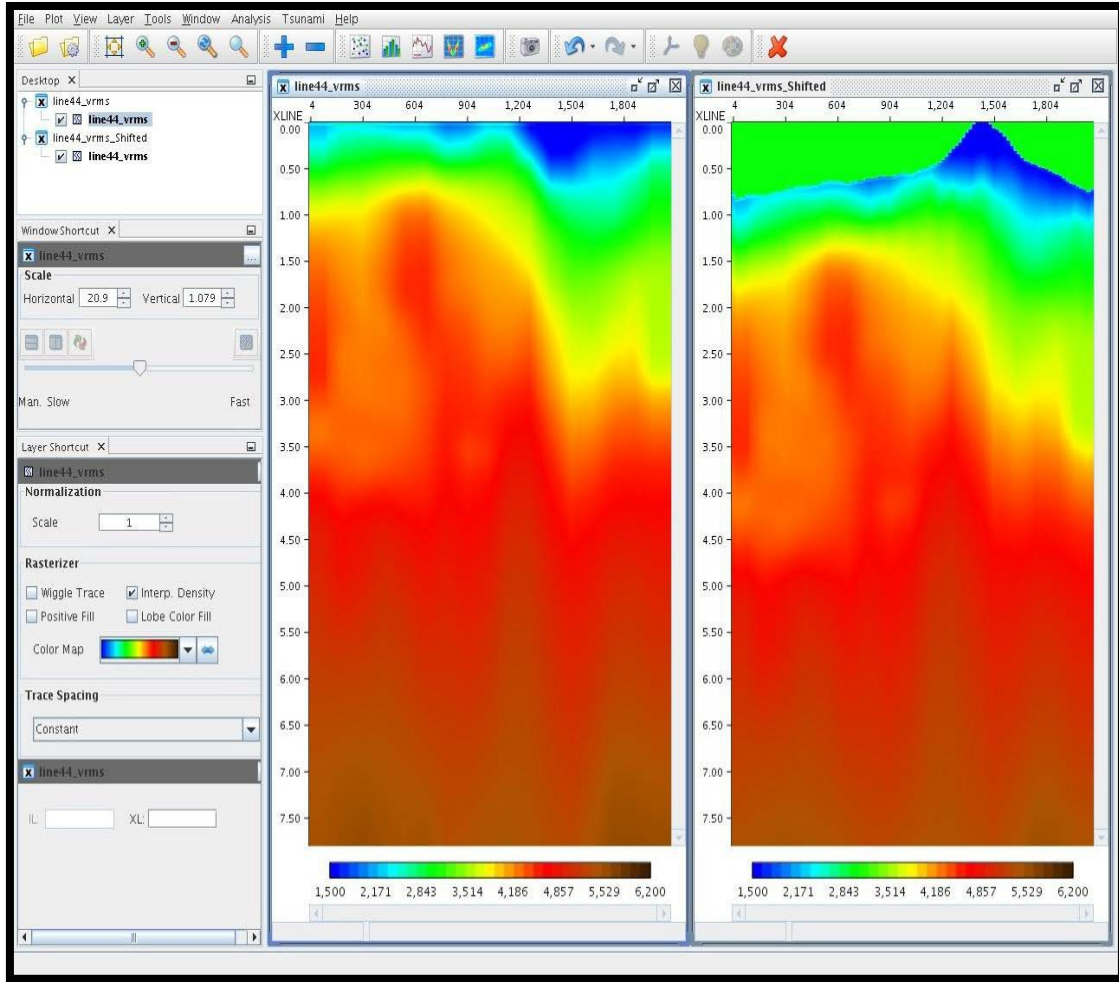
USING THE FLOATING DATUM UTILITY

Begin by selecting the trace.db output file from the Tsunami module containing the floating datum information. The field "RMS velocity model or seismic dataset:" should have the name of the data file needed to be shifted. Select the correct Data type and domain to be converted. Input the reference velocity and indicate how to handle negative datum values. The example below shows the velocity model is RMS (time domain), has a reference velocity of 3000, and negative datum values will be pulled upwards.



Floating Datum Utility Panel

When all parameters are correct press the "OK" button to run the plug-in. A new data window will appear which will be the original data with the specified shift. Save this data for further use.



Floating Datum Example Output

APPENDIX

FILES

- Trace Database Containing Datums: The path to the trace database (trace.db) file created using one of the following Tsunami Modules: PSTM, PSDM, RTM. This database contains the floating datum information which is specified on the "INPUT" tab of the Tsunami Module creating the trace database.
- RMS velocity model or seismic dataset: This field is filled in when starting the Floating Datum Utility plug-in. The name specified will be the data that was currently active/selected when starting the plug-in.

DATA TYPE

- RMS: Select this option when shifting velocity models in the time domain. Note, when using RMS only the Time option may be used.
- Seismic: Select this option when shifting seismic data. Time or Depth may be used when selecting Seismic.
- Interval Velocity: Select this option when shifting velocity models in the depth domain. Note, when using Interval Velocity only the Depth option may be used.

FLOATING DATUM

- Reference Velocity: The corresponding reference velocity used to shift and fill when shifting data.
- Negative datum values:
 1. Traces pulled upward: Select this option when negative datum values should be shifted toward the top in a time or depth cross section.
 2. Traces pulled downward: Select this option when negative datum values should be shifted toward the bottom in a time or depth cross section.

PLUG-IN INDEXING FORMATS

This plug-in must have the data indexed by these header keys for different types of data. Note, data in depth is still indexed in time. The order in which the indexes are selected is irrelevant.

- 2D Stacked Data Header Keys: Xline, Time
- 3D Stacked Data Header Keys: Inline, Xline, Time Back to Top

SEISMIC MUTE

INTRODUCTION

The Seismic Mute plug-in allows the user to mute seismic gathers by zeroing out the samples above the user picks. A linear interpolation can be applied to gathers using the points picked by the user. If the gathers are at final datum a trace.db file must be supplied to insure proper interpolation between gathers. User picks and interpolated picks may be saved to an output file. The muted gathers may be saved to a SEG-Y file, which will be indexed for the INTViewer on output.

SETUP

Load a dataset of seismic gathers.

USING THE MUTE PLUG-IN

Begin by selecting whether a new mute will be created or a previously picked mute will be loaded. If a new mute is selected you will need to select either the "Top mute" or "Bottom mute" radio button. Select whether the gathers are in time or depth by choosing the appropriate radio button. You may now start picking points on the dataset.

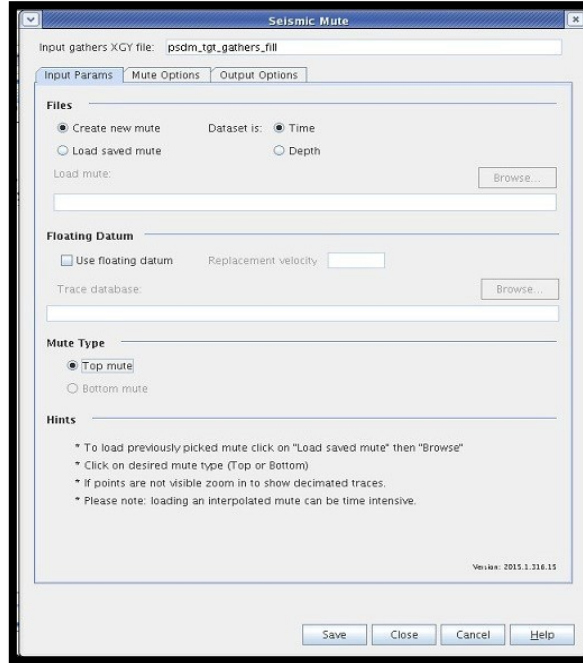
Move the cursor to the point on the trace you wish to mute and press the left mouse button. If the point is not where you want it you may press the right mouse button to remove the point.

Under the Mute Options tab you may adjust the mute taper by supplying a new value in milliseconds and pressing the Update button.

If you wish to see the mute interpolated across the dataset select the Interpolate check box. If you wish to see the data that has been muted out you may select the Display samples above mute checkbox.

Under the Output Options you have the options of save the user picks to a mute file, saving the interpolated picks to a mute file and saving the muted gathers to an indexed SEG-Y file. Any or all of these choices may be selected. Once you press the Save button at the bottom of the dialog you will see browser boxes open for each of the choices you have made so you may supply output filenames.

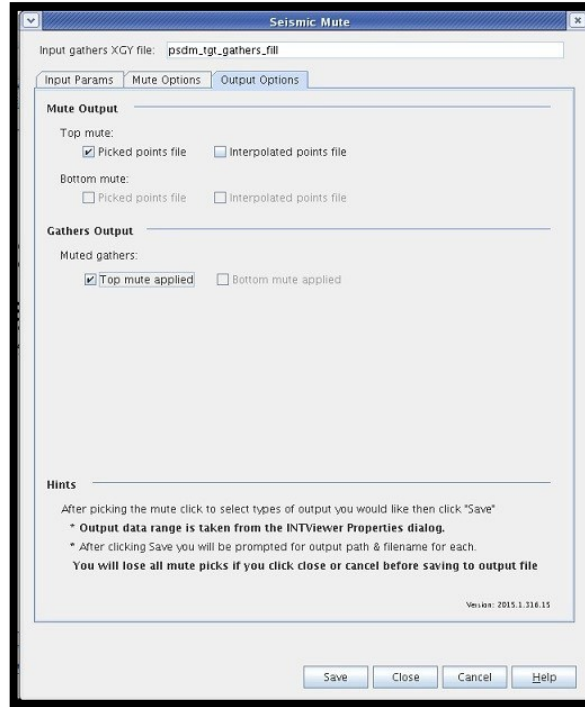
The mute dialog may be opened and closed without having to reload your parameters. They will be remembered. You may also click the right mouse button and in the menu that opens select Save Parameters. This file can be loaded for a later mute session by clicking the right mouse button and selecting the Load Parameters option.



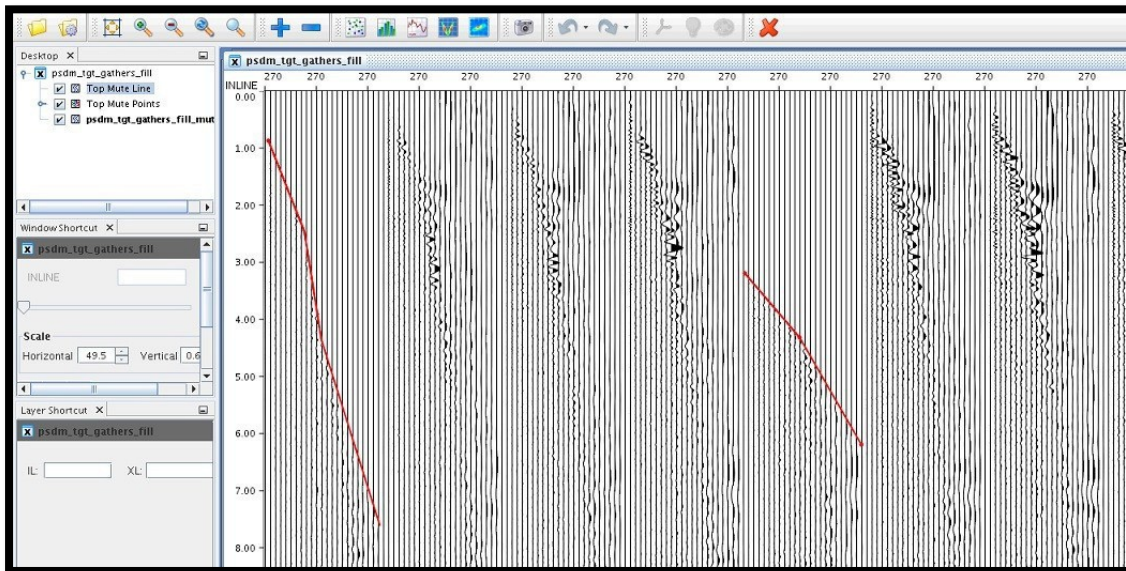
Mute Panel Input Params Tab



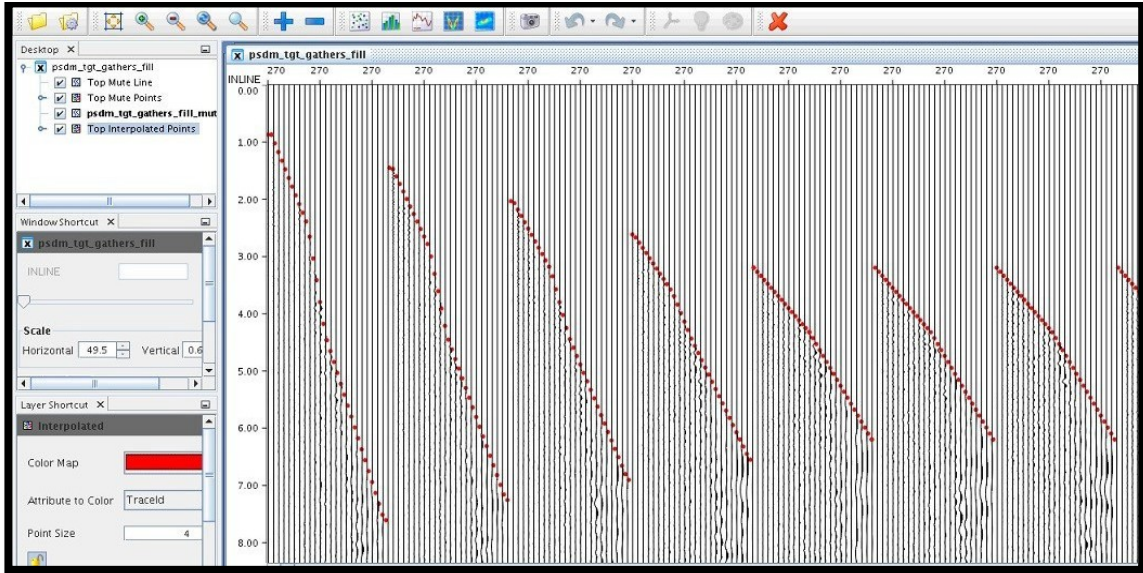
Mute Panel Mute Options Tab



Mute Panel Output Options Tab



Mute Example User Picks



Mute Example Interpolated Picks

APPENDIX

INPUT PARAMS

- Input gathers XGY file: This field is filled in when starting the Mute plug-in. The name specified will be the data that is currently active/selected when starting the plug-in.
- Create new mute: This radio button is chosen when starting a muting session for the first time.
- Load saved mute: This radio button is chosen when loading a saved mute from a previous muting session.
- Dataset is Time: This radio button is chosen when the seismic data is sampled in Time.
- Dataset is Depth: This radio button is chosen when the seismic data is sampled in Depth.
- Load mute: This field will be filled when the user has selected the Load saved mute radio button and then pressed the Browse... button to open a browser dialog and chosen a previously picked mute file.
- User floating datum: This checkbox is chosen when the seismic dataset is at final datum.
- Replacement velocity: The replacement velocity will need to be supplied if the dataset is sampled in Depth.
- Trace database: The path to the trace database (trace.db) file created using one of the following Tsunami Modules: PSTM, PSDM, RTM. This database contains the floating datum information which is specified on the "INPUT" tab of the Tsunami Module creating the tracedatabase.
- Top mute: This radio button is chosen when designing a top mute.
- Bottom mute: This radio button is chosen when designing a bottom mute.

MUTE OPTIONS

- Picking on: This radio button is selected while user is picking mute points. If focus is on a layer other than the Top (or Bottom) Mute Line layer you cannot pick. Clicking this radio button puts focus back in the picking layer.
- Picking off: This radio button is selected automatically when the user selects the interpolation check box.
- Taper(ms): This field must be supplied by user to have the mute taper on. The default taper length is 32 milliseconds.
- Update: This button only needs to be selected if the Taper value is changed.
- Interpolate: Select this check box to generate an interpolated mute between user picked mute points.
- Display samples above mute: Select this check box to toggle back and forth between the zeroed samples and original data samples.

OUTPUT OPTIONS

- Top mute - Picked points file: Select this check box to output your user picks for a Top Mute to a mute file (extension .mut)
- Top mute - Interpolated points file: Select this check box to output the interpolated picks for a Top Mute to a mute file for use in the Stack plug-in.
- Bottom mute - Picked points file: Select this check box to output your user picks for a Bottom Mute to a mute file (extension .mut)
- Bottom mute - Interpolated points file: Select this check box to output the interpolated picks for a Bottom Mute to a mute file for use in the Stack plug-in.
- Muted gathers - Top mute applied: Select this check box to output the interpolated Top muted gathers SEGY file (extension .xgy). This file will be indexed for the INTViewer automatically.
- Muted gathers - Bottom mute applied: Select this check box to output the interpolated Bottom muted gathers SEGY file (extension .xgy). This file will be indexed for the INTViewer automatically.

At the bottom of each tabbed page are hints that will help you during your mute session.

- To load previously picked mute click on "Load saved mute" then "Browse"
- click on the desired mute type (Top or Bottom)
- If points are not visible zoom in to show decimated traces.
- Please note: loading an interpolated mute can be time intensive.
- click interpolate to see interpolated mute, click again to remove interpolation
- click to show samples above mute for QC; click again to remove
- Once complete click Output tab to save mute picks
- Output range is taken from the INTViewer Properties dialog. This means that if a subsection of the dataset has been selected for viewing, that is all you will save.
- After clicking on the Save button you will be prompted for output file names.

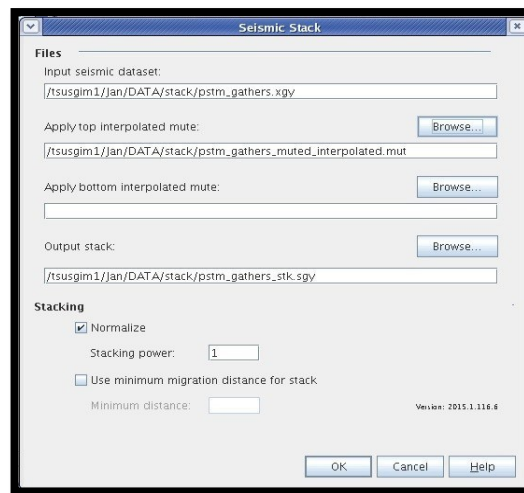
STACK

INTRODUCTION

The Stack plug-in adds together common Inline/Xline (CDP) offsets into a single trace. Top and bottom mutes may be applied which are made through Tsunami's Mute plug-in. Additional options available are normalization, minimum migration distance, and stacking power.

SETUP

Stack requires seismic gathers to open, please check the Stack Indexing Formats for correct indexing options.



Stack Plug-in Panel

USING STACK

Once the Stack plug-in is open it is possible to output a stacked volume. The user may apply a top and/or bottom mute by filling in the path or browsing to the .mut files. The name/location of the output file may also be changed by altering the specified path or browsing to the desired directory and filling in the file name.

NORMALIZING

Checking the Normalize box will make stack divide by the number of non-zero samples.

USE MINIMUM MIGRATION DISTANCE FOR STACK

Checking this box will allow the user to input a minimum migration distance. When the stack is performed, any trace with a minimum migration distance larger than the specified value will not be included in the stacked traces.

STACKING POWER

The stacking power parameter multiplies the number of non-zero samples by the number specified before stacking.

APPENDIX

FILES

- Input Seismic Dataset: Displays the path to the active data set in INTViewer.
- Apply top interpolated mute: The path to the top .mut file made with Tsunami's Mute plug-in. This file must be saved as an interpolated mute in the Mute plug-in.
- Apply bottom interpolated mute: The path to the bottom .mut file made with Tsunami's Mute plug-in. This file must be saved as an interpolated mute in the Mute plug-in.
- Output Stack: The path to the output stacked file.

STACKING

- Normalize: When checked, Stack will divide the output samples by the number of nonzero input samples.
- Use Minimum Migration Distance for Stack: The distance in which a migrated trace is allowed to have to be included in the stack. Any traces with distances greater than this parameter will not be used.
- Stacking Power: A value which multiplies the number of non-zero traces before normalization.

STACK INDEXING FORMATS

This plug-in must have the data indexed by these header keys for different types of data. Note, data in depth is still indexed in time. The order in which the indexes are selected is irrelevant.

- 2D gathers: Xline, Offset, Time
- 3D gathers: Inline, Xline, Offset, Time

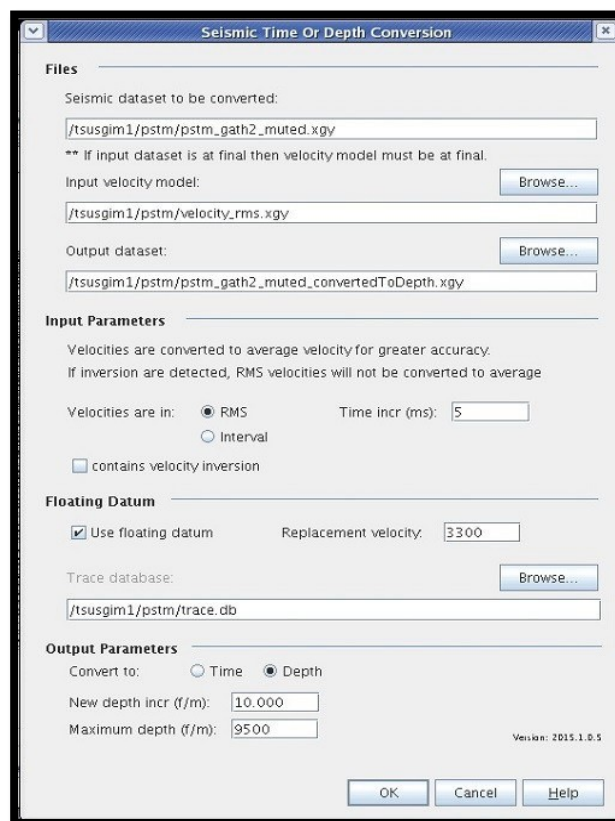
TIME OR DEPTH CONVERSION

INTRODUCTION

The Time or Depth Conversion Plug-in allows the user to convert seismic data in the time domain to the depth domain and vice versa. An interval or RMS velocity model may be used and floating datum statics are accounted for by inputting the Tsunami Trace.db file and reference velocity. Floating datum does not need to be used if the seismic and velocity model are both at the same datum (final or floating).

SETUP

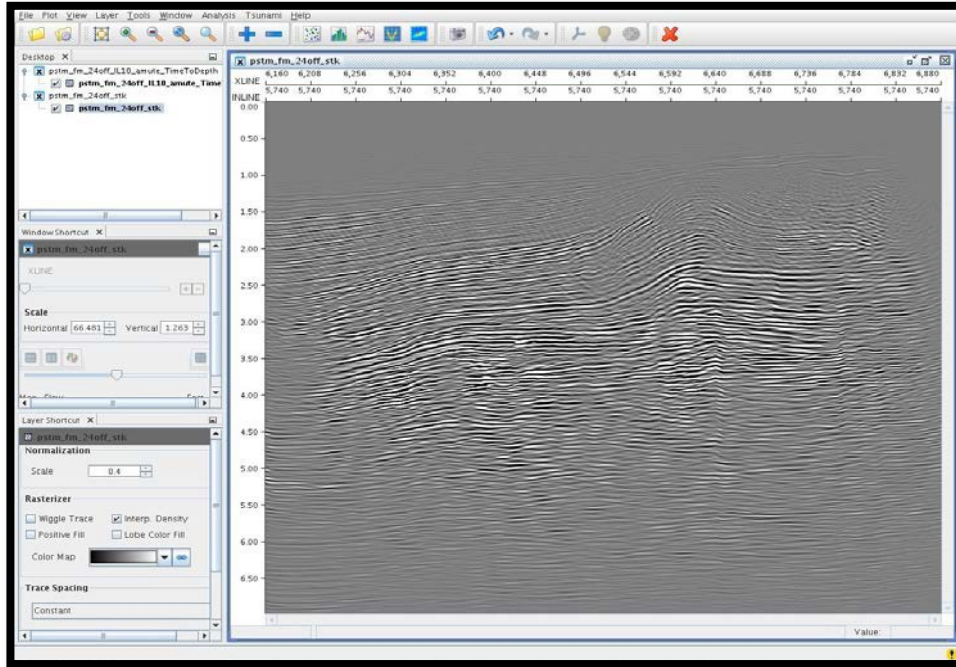
To begin using the Time or Depth Conversion plug-in a stacked indexed seismic dataset must be open. Please check the Time or Depth Conversion Indexing Formats for correct indexing options.



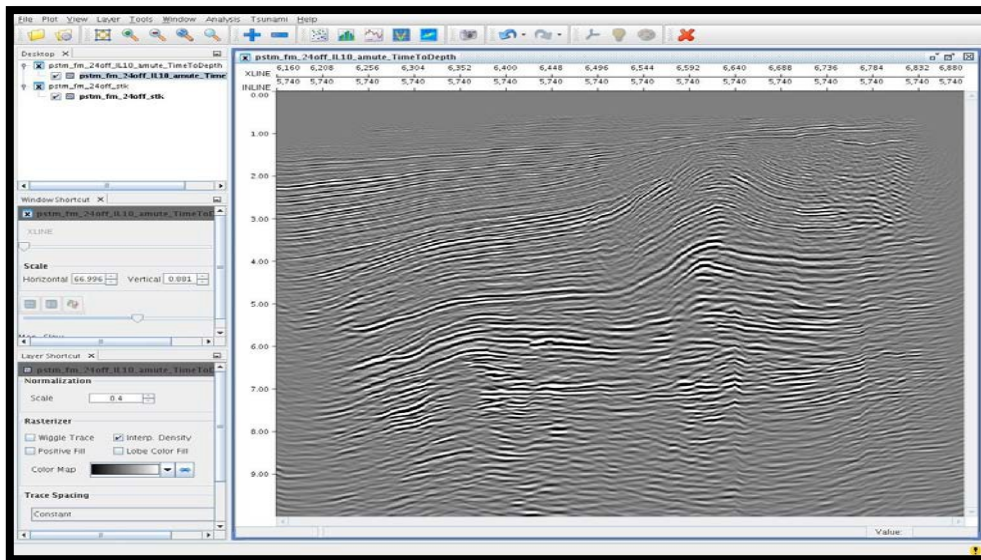
USING THE TIME OR DEPTH CONVERSION PLUG-IN

On start-up the plug-in needs a velocity model. The user may select an RMS or interval velocity model to convert to the desired domain however, generally both conversions are more stable when using an interval velocity model. Give the corresponding time/depth increment in the specified units. If either model is at a different datum than the other, the floating datum information must be provided. Then specify the domain and parameters desired for the output data. Press the "OK" button to generate the data.

The check box marked "contains velocity inversion" is used to indicate whether a reversal in the gradient of the model is present.



Example Input Time Seismic



Example Output Depth Seismic

APPENDIX

FILES

- Input velocity model: The path to the velocity model to be used in the conversion from one domain to the other.
- Seismic dataset to be converted: The path to the seismic dataset to be converted. This is selected when the plug-in starts and will be the active data in INTviewer.

INPUT PARAMETERS

- Velocities are in:
 1. RMS: Select this parameter when the input velocity model is an RMS velocity model in the time domain.
 2. Interval: Select this parameter when the input velocity model is an interval velocity model in the depth domain.
- Time incr (seconds): The velocity model sampling increment in seconds. Example: A velocity model sampled at two milliseconds will have an input value of "0.002"
- Depth incr (ft/m): The velocity model sampling increment in feet or meters. Example: A velocity model sampled every five meters would have an input value of "5.0".
- Contains velocity inversion: This check box needs to be selected when the velocity model has higher velocities above lower velocities (when the gradient of the model is negative).

FLOATING DATUM

- Velocity at Float/Seismic at Final Check Box: Check this box when the velocity model is at floating datum but the seismic data is at final datum. Checking this box will allow the user to input the floating datum information.
- Reference Velocity: The reference velocity used when building the Tsunami Trace.db file.
- Trace database: The path to the Trace.db file made from Tsunami's PSTM, PSDM, or RTM modules.

OUTPUT PARAMETERS

- Convert to:
 1. Time: Select this option when converting from depth to time domains.
 2. Depth: Select this option when converting from time to depth domains.
- New Time incr (seconds)/New Depth incr (ft/m): The desired output increment of the seismic data in each respective domain.
- Maximum Time (seconds)/Maximum Depth (ft/m): The vertical extent of the seismic that will be output after the conversion.

TIME OR DEPTH CONVERSION INDEXING FORMATS

This plug-in must have the data indexed by these header keys for different types of data. Note, data in depth is still indexed in time. The order in which the indexes are selected is irrelevant.

- 20 Stacked Oata: Xline, Time
- 30 Stacked Oata: Inline, Xline, Time Back to Top

VELOCITY TOOLS

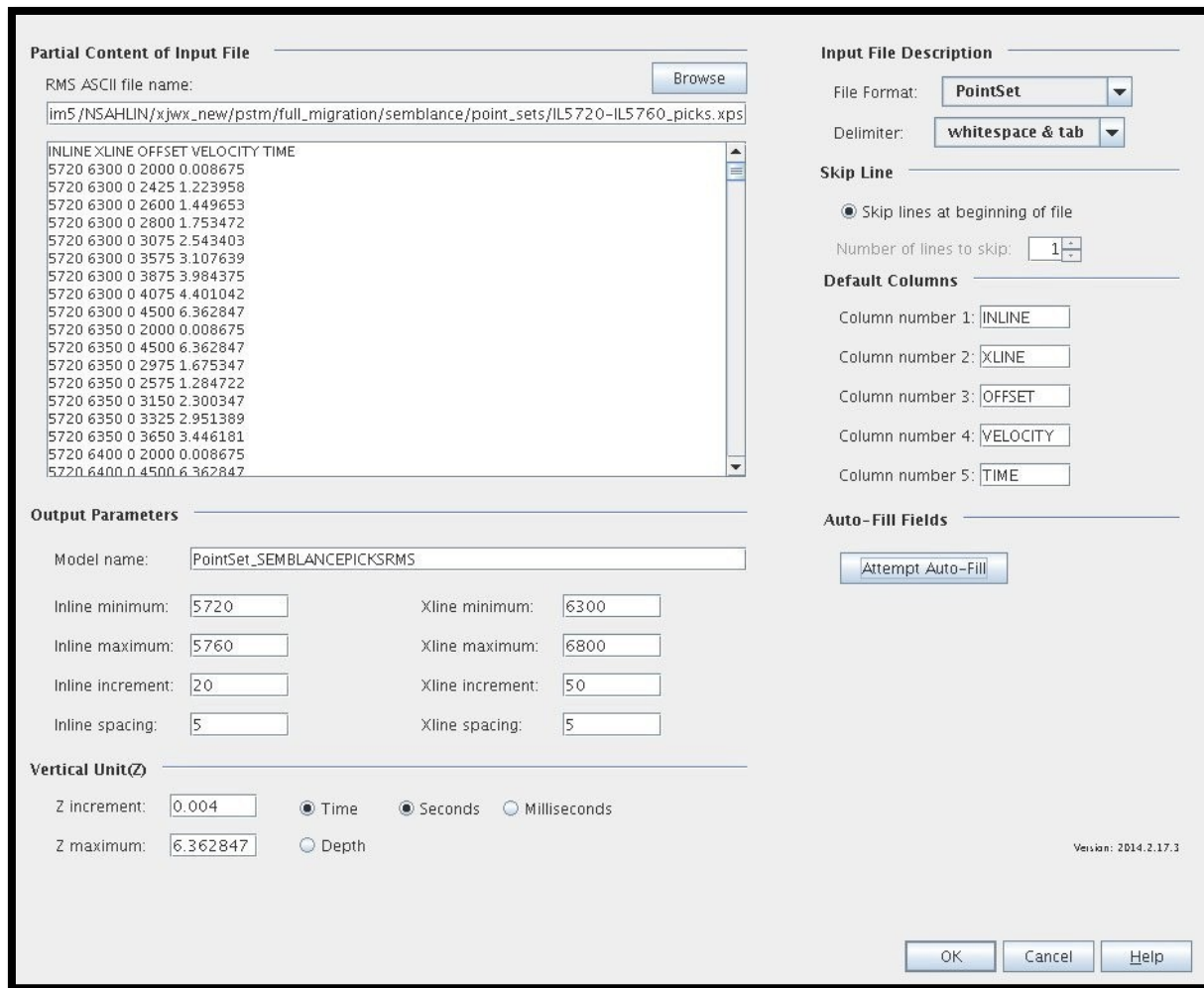
ASCII CONVERTER

INTRODUCTION

The ASCII Converter allows the user to create models from an ASCII text file. This plug-in is intended to create velocity, Eta, and gamma (Vp/Vs) models from simple time or depth data. Custom formats may be used as well as Promax and point set formats.

SETUP

The user must have an ASCII file which provides the Inline (if 30), Xline, Time, and Velocity information. An example of this is shown in the screen shot below. The input file provided is a point set.



USING ASCII CONVERTER

To use the plug-in, simply use Browse button associated with the "RMS ASCII file name:" to load the RMS ASCII file. If the format of the file is one of the three default file formats (PointSet, HANDVEL, SeisUP) the plug-in will recognize the file as such and ask to load the file using the default parameters for that type of file. If the file is not in any standard format the data will be loaded and the file format will be "Custom".

LOADING A DEFAULT FILE FORMAT

When loading a default file format type, it is recommended to use the "Attempt Auto-Fill" button to automatically fill in all of the parameter fields. Review these parameters to make sure they are correct. Press the "OK" button to output the RMS velocity model in INTViewer.

LOADING A CUSTOMFILE FORMAT

When loading a Custom file format the user must input all parameters. An explanation of all the parameters can be found below in th Appendix. After filling out all of the parameters the user should press the "OK" button to output the RMS velocity model in INTViewer.

APPENDIX

PARTIAL CONTENT OF INPUT FILE

- RMS ASCII file name: The path to the input ASCII file to be converted to SEG-Y format.

INPUT FILE DESCRIPTION

- File Format:
 1. Custom: Choose this option when importing a custom made ASCII file that is not one of the default options in the drop down list.
 2. SeisUP: Choose this option when importing a SeisUP file format. The ASCII converter should automatically recognize and select this option upon import.
 3. HANDVEL: Choose this option when importing a HANDVEL file format. The ASCII converter should automatically recognize and select this option upon import.
 4. PointSet: Choose this option when importing a PointSet file format made in INTViewer. The ASCII converter should automatically recognize and select this option upon import.
- Delimiter: The delimiter drop down menu allows the user to select the appropriate delimiter to differentiate between columns.
 1. whitespace & tab- Delimitate by any amount of spaces or tabs.
 2. ::- Delimitate by two colons.
 3. :- Delimitate by one colon.
 - |- Delimitate by pipe.
 4. ;- Delimitate by one semicolon.
 5. ,- Delimitate by one comma.

SKIP LINE

- Skip Lines at beginning of file radio button: Select this option when there are comments or headers at the beginning of the ASCII file.
- Number of lines to skip: The number of lines to skip before reading the input data columns.

DEFAULT COLUMNS

- Column Number (#): The header key associated with the appropriate column number from left to right. These header keys are used to define the dimensions of the output velocity model. If the user has selected to Auto-Fill the parameters these header keys will be input from the file header. The following are the necessary header keys:
 - INLINE
 - XLINE
 - VELOCITY (ETA, GAMMA)
 - TIME
 - CDP

OUTPUT PARAMETERS

- **Model Name:** The path to the output file name for the velocity SEG Y file. This name may be changed when saving the SEG Y file in INTViewer.
- **Inline minimum/maximum:** The extent in the inline direction the velocity model extends to. These may be changed to larger or smaller extents than the velocity model data. Changing the extents beyond the original data will use a distance weighted interpolation to create the new velocity traces.
- **Xline minimum/maximum:** The extent in the xline direction the velocity model extends to. These may be changed to larger or smaller extents than the velocity model data. Changing the extents beyond the original data will use a distance weighted interpolation to create the new velocity traces.
- **Inline/Xline increment:** The increment in which the inline/xline directions will be sampled. The increment may be sampled on a finer or coarser scale by decreasing or increasing the increment parameter respectively.
- **Inline/Xline spacing:** The spacing between one inline/xline and the next in feet or meters. These two parameters are necessary for correct interpolation when creating new velocity traces.

VERTICAL UNIT (Z)

- **Z increment:** The increment in time or depth which the velocity model is sampled. Choose a finer or coarser increment by decreasing or increasing this parameter.
- **Z maximum:** The maximum time or depth which the velocity model extends. This parameter may be changed to stretch or squeeze the velocity model.

P VELOCITY TO S VELOCITY CONVERTER

INTRODUCTION

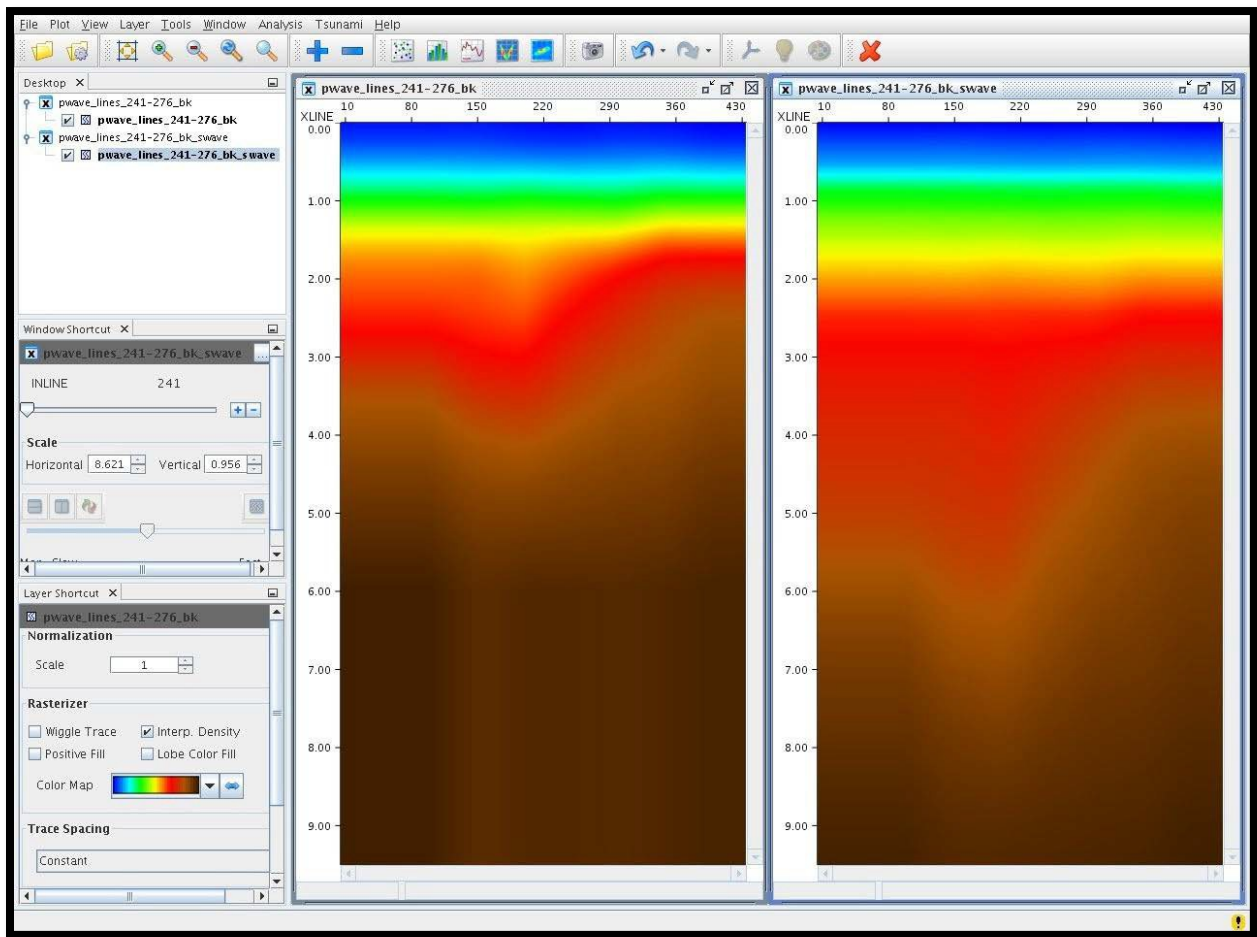
The P-wave to S-wave Converter plug-in uses a P-wave velocity model and corresponding V_p/V_s ratio model to produce an S-wave velocity model.

SETUP

The P Velocity to S Velocity Converter requires a velocity model indexed in Inline and/or Xline and time to open. When launching the plug-in the highlighted velocity model will be the input P-wave model.

USING P VELOCITY TO S VELOCITY CONVERTER

Once the plug-in has been launched, use the "Browse" button to select the appropriate gamma (V_p/V_s ratio) velocity model. Then press the "OK" button to output the corresponding S-wave velocity model.



Example Input Pwave (left) and Output Swave (right) Models

APPENDIX

FILES

- P Wave Velocity Model: The starting velocity model to be used in the conversion. This model must be indexed by Inline/Xline and Time.
- Vp/Vs Ratio (Gamma) model for velocity conversion: The Vp/Vs Ratio (Gamma) model used to convert the starting P-wave model. This model must be indexed by Inline/Xline and Time.

P VELOCITY TO S VELOCITY CONVERTER INDEXING FORMATS

This plug-in must have the data indexed by these header keys for different types of data. Note, data in depth is still indexed in time. The order in which the indexes are selected is irrelevant.

- 20 velocity Data: Xline, Time
- 30 velocity Data: Inline, Xline,

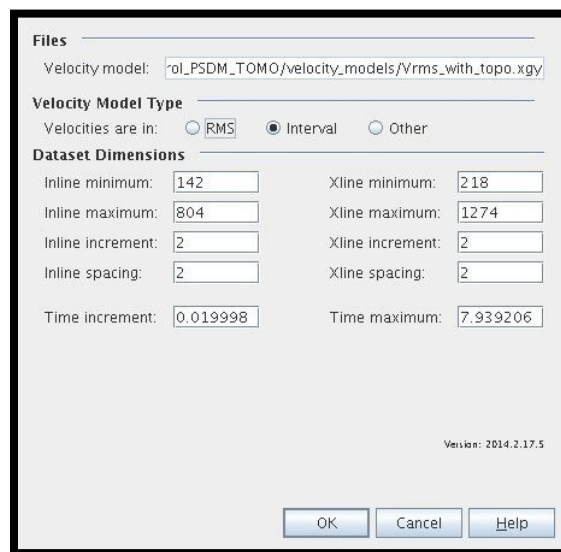
VELOCITY MODEL RESAMPLER

INTRODUCTION

The Velocity Resampler plug-in samples velocity models to finer or coarser increments in space (Inline, Xline and vertical distance). RMS velocity models may only be resampled in Inline and Xline space. Additionally a 20 model (Xline) may be converted to a 30 model by filling in the Inline/Xline min/max and increment.

SETUP

Velocity Resampler requires a properly indexed model opened in INTViewer to be opened. The plug-in panel is shown below.



Velocity Resample Plug-in

USING VELOCITY RESAMPLER

The Velocity Resample plug-in may be used to perform several tasks.

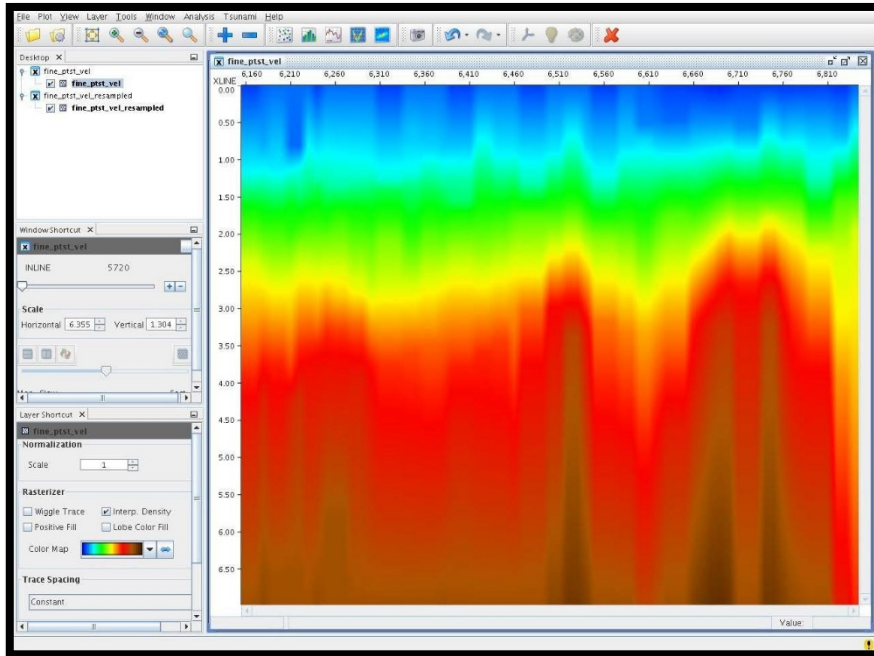
1. Resample a velocity model to a finer/coarser increment in Inline/Xlinespace.
2. Resample a 20 model to a 30 model or pad a velocity model further outwards in either Inline/Xline space.
3. Resample the depth increment when using an interval velocity model.

Combinations of these tasks can be easily performed by filling in the required parameters simultaneously. When sampling to increments which do not currently have existing traces the plug-in performs an inverse distance weighted interpolation (IOWI) based upon the Inline/Xline spacing parameters. Below are instructions on how to perform resampling tasks.

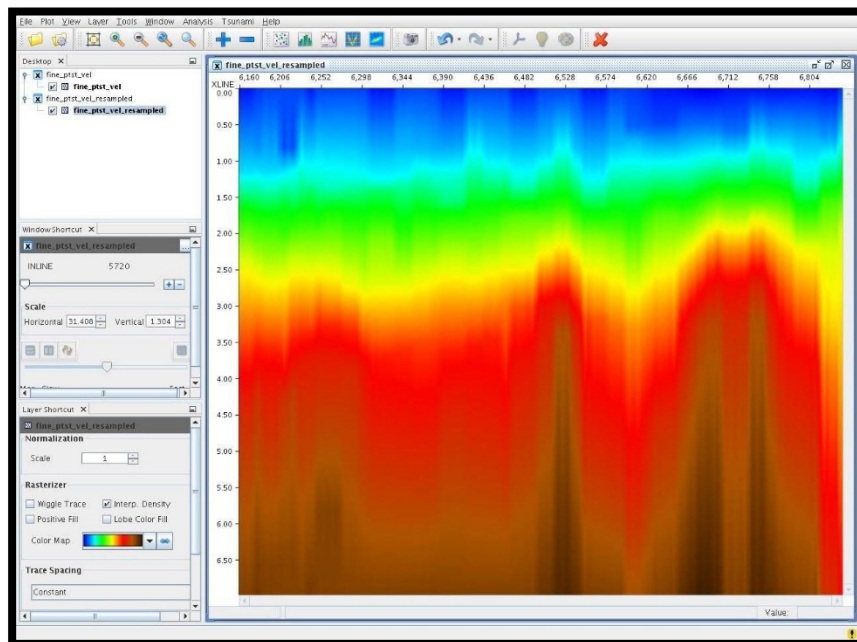
1. Resampling to a finer interval: Replace the value in the Inline/Xline increment parameter with a smaller number than the increment specified in INTViewer.
2. Resampling to a coarser interval: Replace the value in the Inline/Xline increment parameter with a larger number than the increment specified in INTViewer.
3. Resampling a 20 to 30 and padding outwards: Replace the Inline/Xline minimum/maximum

parameters to the desired extents. These extents may not be negative, and when padding outward on a 3D model the plug-in will use the IOWI spacing parameters.

4. Resample the Depth increment of an interval velocity model: Replace the Time increment parameter with the desired output increment.
5. When all parameters are correctly filled press the "OK" button to run the plug-in. The plug-in panel will close and a new velocity model will appear in the INTViewer. To save the velocity model use the INTViewer option "Save Seismic Data."



Example Input Model (Sampled 20x20)



Example Output Model (Sampled 1x1)

APPENDIX

FILES

- Velocity Model: The path to the velocity model that will be resampled.

VELOCITY MODEL TYPE

- RMS: Select this option if the velocity model is in Time.
- Interval: Select this option if the velocity model is in Depth.
- Other: Select this option if the velocity model is ETA, Azimuth, Dip.

DATASET DIMENSIONS

- Inline/Xline minimum: The minimum spacial extent in the Inline/Xline directions. Changing this parameter will result in a velocity model padded outwards or cut off for the minimum extent.
- Inline/Xline maximum: The maximum spacial extent in the Inline/Xline directions. Changing this parameter will result in a velocity model padded outwards or cut off for the maximum extent.
- Inline/Xline increment: Initially this parameter will be automatically filled with the initial increment of the data set which can be seen in the INTViewer properties menu. Changing these parameters will cause the Velocity Resample plug-in to replicate the velocity model with the new increment.
- Inline/Xline spacing: The spacing in feet or meters between the each Inline/Xline. This parameter is necessary for the inverse distance weighted interpolation when creating new velocity traces.
- Time increment: The increment between samples in time or depth. When using depth velocity models the increment should still be filled in as a time in seconds. Example: a depth increment of 5 meters should be filled in as "0.005"
- Time maximum: The maximum time or depth the velocity model extends to. Change this parameter to cut off or add time/depth to the velocity model.

VELOCITY RESAMPLER INDEXING FORMATS

This plug-in must have the data indexed by these header keys for different types of data. Note, data in depth is still indexed in time. The order in which the indexes are selected is irrelevant.

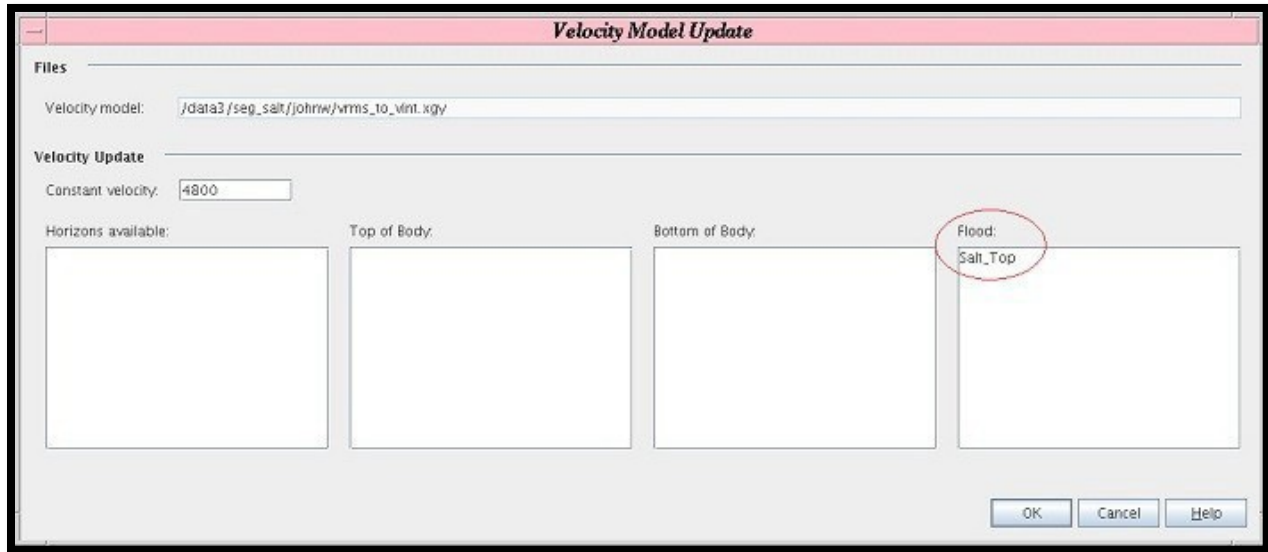
- 2D: Xline, Time
 - 3D: Inline, Xline, Time
-

VELOCITY MODEL UPDATE

INTRODUCTION

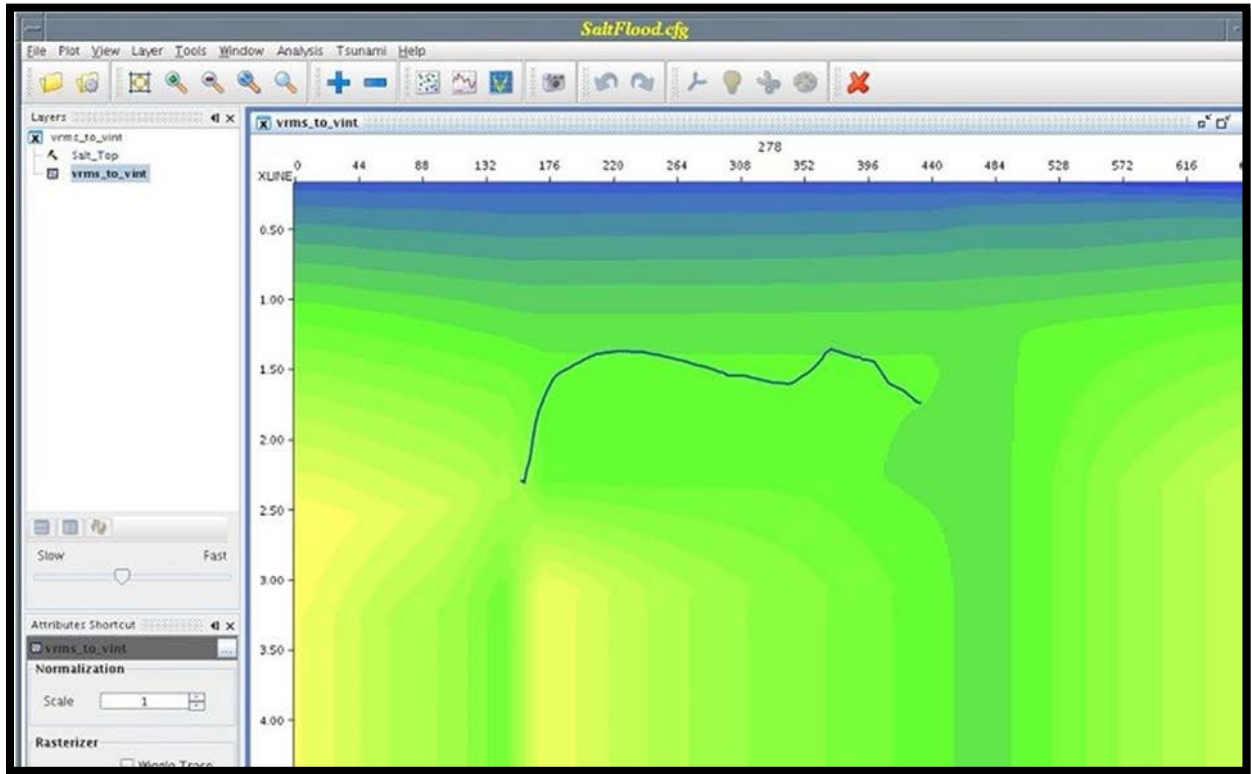
The Velocity Model Update plugin allows the user to delineate an area using either 1 (flood) or 2 horizons (top/bottom) per area and replacing the velocity within that area with a user defined velocity. This plugin takes the velocity model from the selected X-Window as well as the horizons that are displayed on the seismic dataset.

USING THE FLOOD OPTION

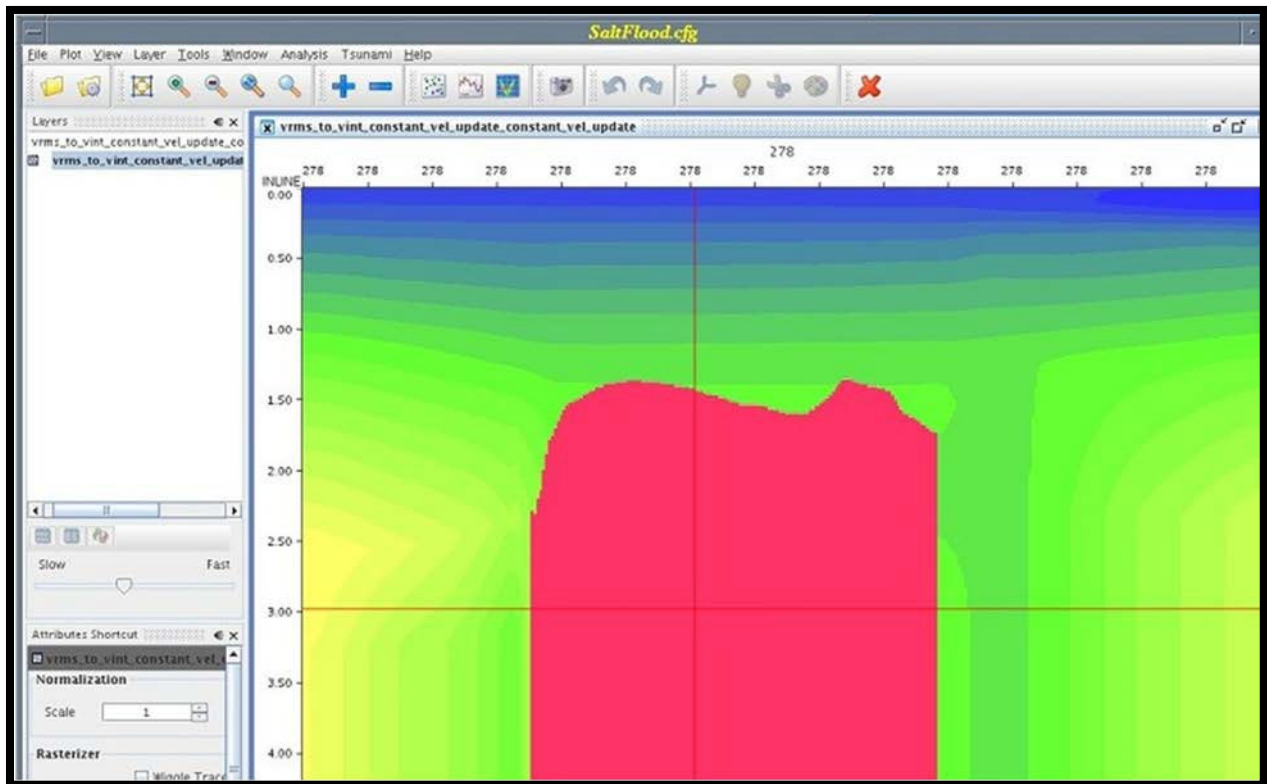


- Supply a constant velocity to replace all velocities below the flood horizon.
- Select a horizon from the Available horizon list and drag it into the Flood list.
- Click OK button

The Velocity Model Update plugin takes that horizon and interpolates to have a point for every trace. Once the interpolation is complete, the velocity model will be rebuilt by replacing all velocities lying below the flood horizon with the constant velocity supplied by the user.

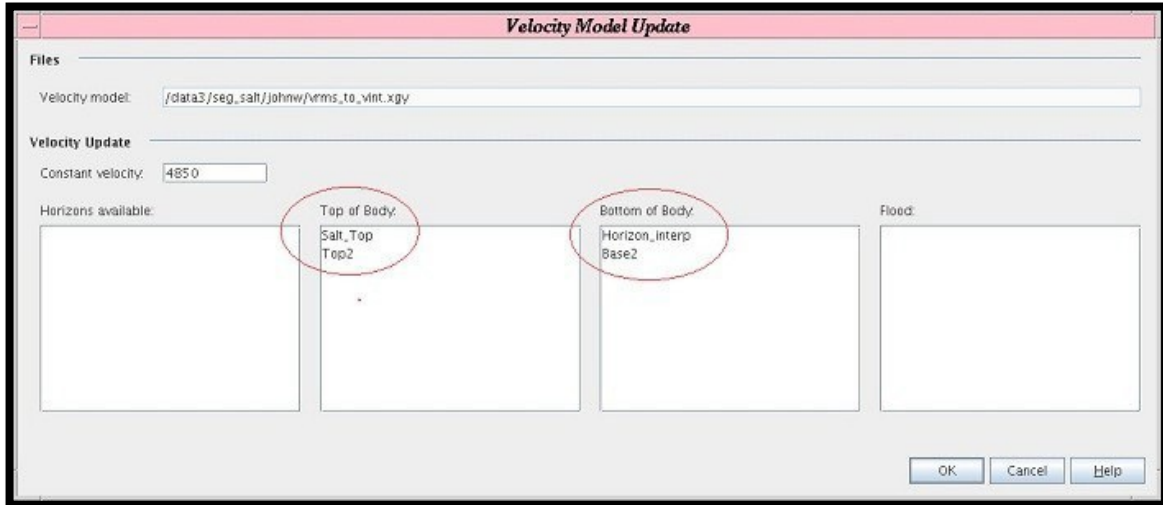


Before



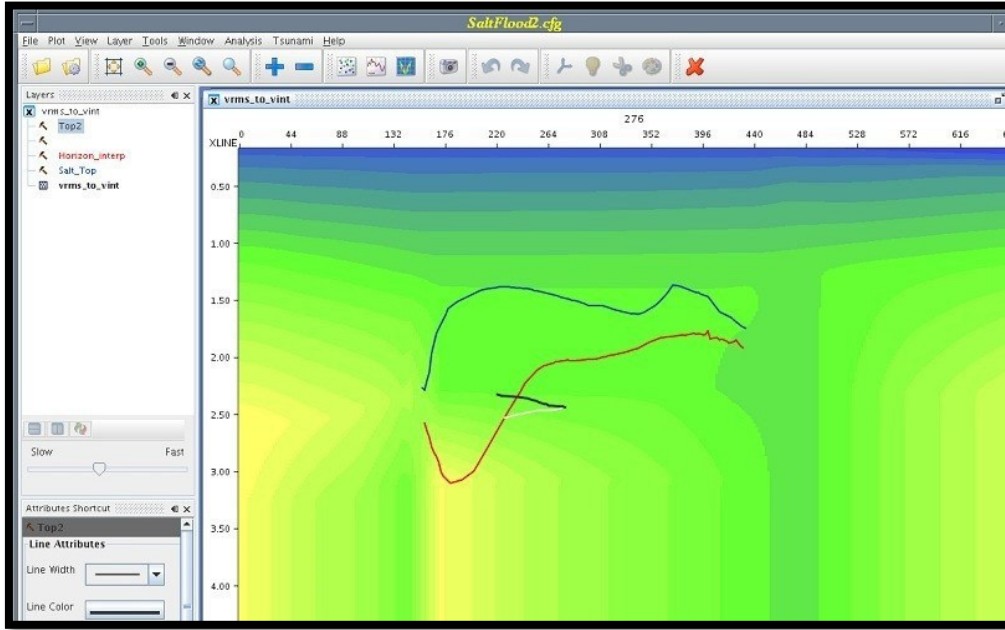
After

USING THE TOP/BOTTOM OPTION

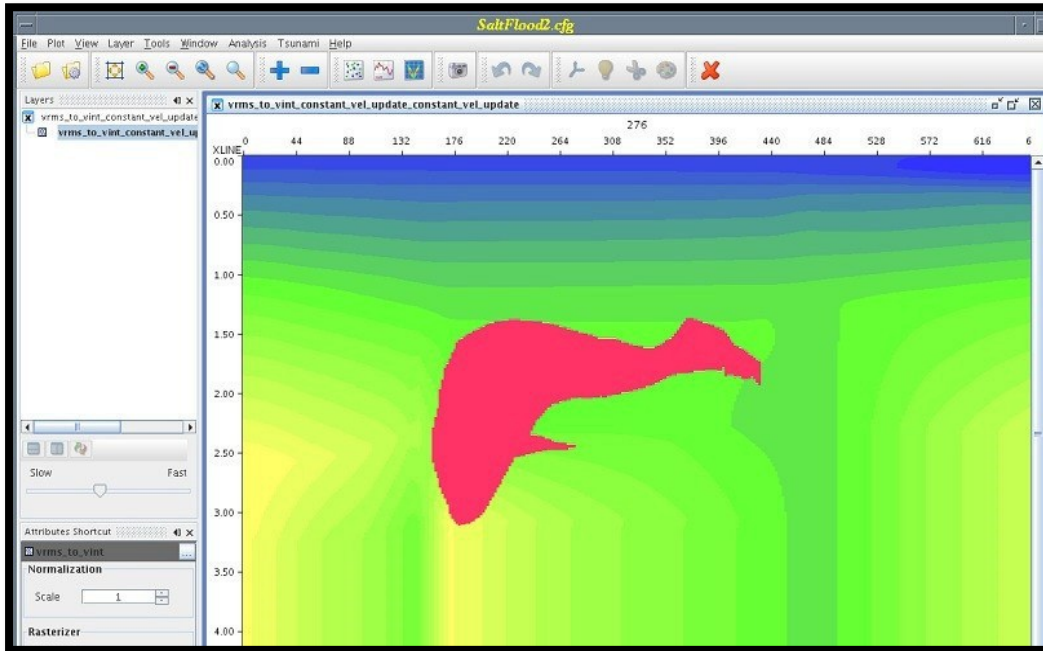


- Supply a constant velocity to replace all velocities within the area defined by the top and bottom horizons.
- Select a horizon from the Available horizon list and drag it into the Top list.
- Select a horizon from the Available horizon list and drag it into the Bottom list.
- Click OK button

The Velocity Model Update plugin takes the top and bottom horizons, where the first top horizon is paired with the first bottom horizon (etc.) and interpolates both horizons to have a top and bottom point for every trace. Once the interpolation is complete, the velocity model will be rebuilt by replacing all velocities between top and bottom horizons with the constant velocity supplied by the user.



Before



After