



Flatirons™ Refraction Statics

Training Manual

Revised October 24, 2018

Table of Contents

SEG-Y Import/Project Creation.....	2
Picker Window.....	6
Delay Time Analysis.....	13
VNS Tomography.....	24
Stack QC (Statics QC).....	32
Database.....	36

SEG-Y Import/Project Creation

Projects in Flatirons™ are created from prestack SEG-Y files (other methods for project creation using proprietary formats are available but are not covered in this tutorial).

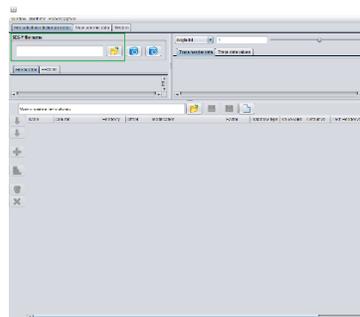
Summary of Workflow:

- Create SEG-Y dictionary
- Set project name, units, dimensions
- Set compression options
- Select SEG-Y file(s)
- Scan SEG-Y file using SEG-Y dictionary

1. Create SEG-Y Dictionary



The program launch window is shown above. In order to import a SEG-Y file and create a project, a SEG-Y dictionary (header map) must first be created to define byte locations in the trace headers. Begin by clicking on the menu option to “Open the SEG-Y viewer and dictionary editor.”



The Dictionary Editor will appear as a new popup window. First select the SEG-Y file by clicking on the open folder icon. Note: if using multiple SEG-Y files, be aware that all files must have the same trace headers in the same byte locations. Only one of the files needs to be used in this step.

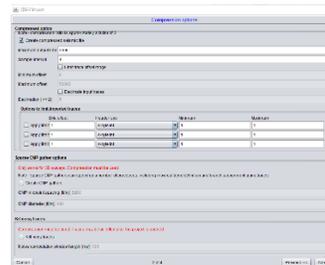
2. Set Project Name, Units, Dimensions



On this first page, set the project name. Select the units (feet or meters) and dimensions (2D or 3D). Note: the 2D option should only be used for reasonably straight 2D lines. Crooked or turning lines should be imported as 3D projects.

Set whether the marine options should be used. These options are for marine streamer data only, not OBC or land surveys. For marine streamer surveys, the program will bin receivers by proximity in order to have more than one trace per “receiver.”

3. Set Compression Options



The imported data may be compressed and stored in the project so that the original data will no longer be needed. This offers a 3 to 1 compression ratio on disk and speeds up a number of processes (particularly the loading of trace gathers), but the import process will take longer to complete. Traces may be resampled and trace length may be modified during the compression if the user chooses to do so.

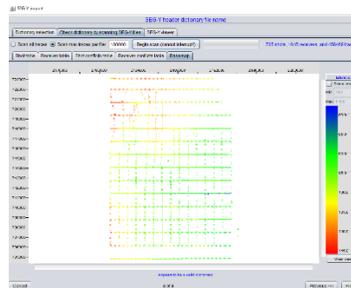
If the compression option is used, precomputed sparse CMP gathers may also be created. These will speed up a number of processes as well.

4. Select SEG-Y File(s)



In the next window, select the SEG-Y file(s) to import. As mentioned earlier, all header byte locations must be the same across all files being imported.

5. Scan SEG-Y File(s) Using Dictionary



In the first tab, "Dictionary selection," select the SEG-Y dictionary file created at the beginning of this tutorial.

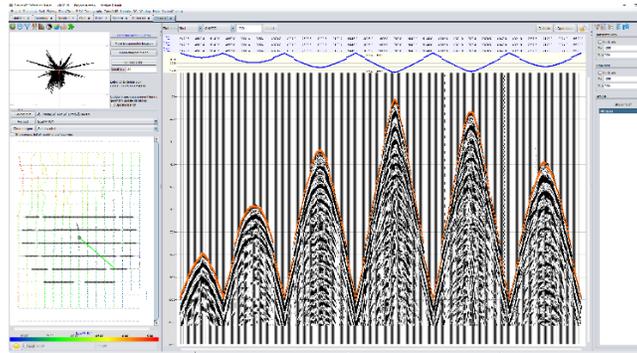
In the second tab, scan the SEG-Y file(s) to check the validity of the dictionary file. It is possible to scan all traces; however, we recommend simply scanning the first 100000 traces of each file as that will provide sufficient information.

Look for abnormal numbers of shots and/or receivers. Shot and receiver conflicts will show up in the relevant "Conflicts" tab. Check the basemap tab to ensure that the partial map looks reasonable.

If all parameters are correct, click the Finish button. Import will begin.

Picker Window

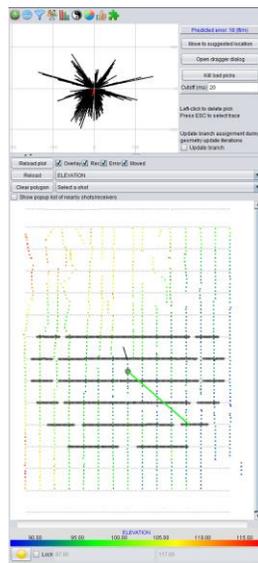
To open a Picker window, go to the Picking menu and select "New picker window - shot/receiver/bin plane". This will bring up the primary Picker.



There are three panels in this window. On the left side is the navigation and display panel. The icons here allow the user to select trace gathers from the basemap and choose how the traces will be displayed. The middle panel is the trace display panel. The right panel is the pick control panel. The options here allow the user to select tools for first break picking.

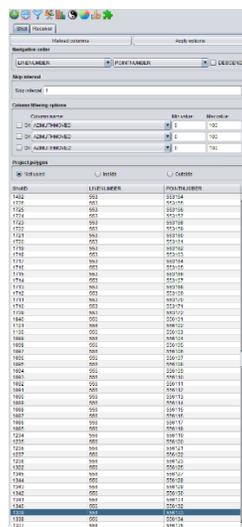
Navigation and Display Panel

Basemap Options

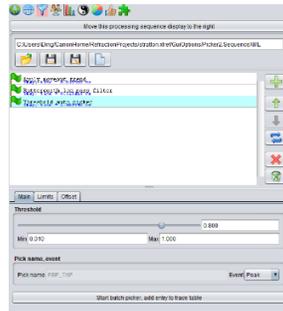


Click a location on the basemap to select a trace gather. The type of gather selected is dependent upon the Pick Plane (explained below). Any database column may be used to color the basemap. The visible shots/receivers in the picker window will be displayed on the basemap. The vector display at the top is used to display the predicted geometry error (this is explained in the Pick QC section of the Delay Time Analysis tutorial).

Navigation Options



Processing Options



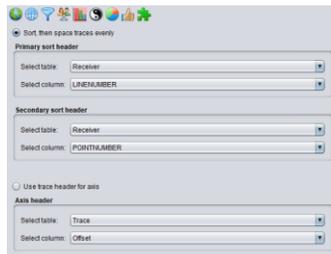
Set the interactive processing sequence for the trace display. Batch jobs (such as autopickers) are launched from this flow as well. If an autopicker is active in the flow, the predicted picks will be displayed on the traces.

Grouping Options



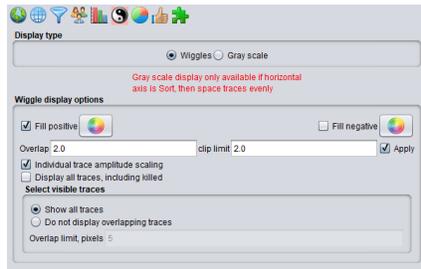
Split a gather into smaller groups for display purposes. This is most commonly used to split a shot gather into receiver lines.

Horizontal Axis Options



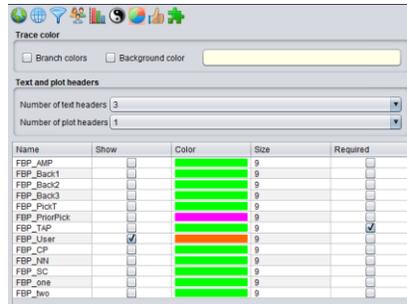
Set the order in which traces are organized from left to right. If using a trace attribute, the traces may be spaced irregularly.

Trace Wiggle Display Options



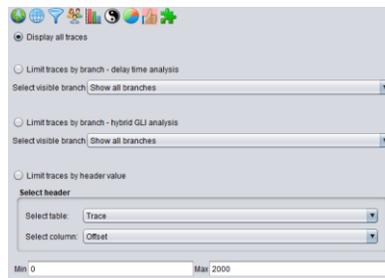
Set options for displaying traces on screen.

Trace Color Display Options



Set options for first break pick overlay (including the color and size of picks), as well as trace coloring based on branch assignment.

Visible Pick Options

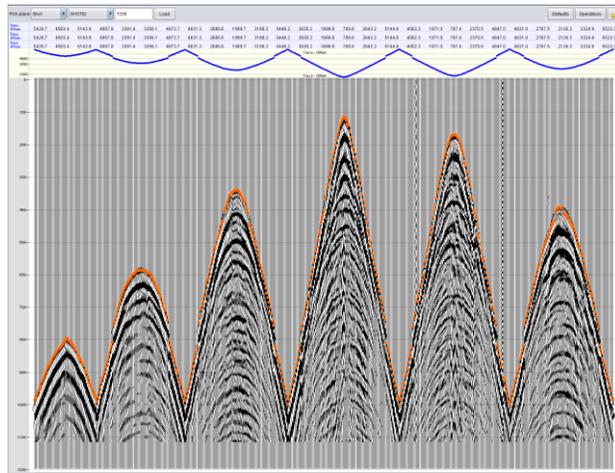


Set which traces to display.

Prior Pick Usage

Allows the use of existing picks to predict future picks. Not recommended in most cases as it is not reliable.

Trace Display Panel



At the top left of this panel is the Pick Plane selector. By default this is set to Shot, meaning the traces are organized in a shot gather, with each trace in the gather having a corresponding receiver. The Pick Plane may be changed to Receiver or CDP.

Next to the Pick Plane selector is a text box that allows the user to specify a gather by ID and load it. To the right are menus for picking defaults and additional operations.

Under these buttons are three rows for displaying trace header text values, and one row for plotted values. Select the header to display by right clicking on the row.

The traces are displayed in the main part of this panel. To use a picking tool with either the mouse or keyboard, move the mouse over the desired pick location and press the key or the mouse button.

Right click and drag to zoom in. Right click once to zoom out.

In order to select a corresponding shot/receiver of a displayed trace, you must have another Picker window open in the correct Pick Plane. Move the mouse over the desired trace and press the ESC key.

Pick Control Panel

Picker Options



Set event to pick (Peak, Trough, None), along with the mouse and keyboard picker command tools. Some useful tools are mentioned below.

Axis Locks



Set zoom locks for the display axes. Selected group info is also displayed here.

Trace Header Values

Display header values for a selected trace. Must have "ViewTraceHeaders" set to a key.

Trace Sample Values

Display trace sample values for a selected trace. Must have "ViewTraceData" set to a key.

Snapshots

Create a snapshot video of all shots. Intended for quick QC. Grouping must be turned off to use this option.

Useful Manual Picking Tools

The following are some of the more useful manual picking tools that can be assigned to the mouse and keyboard:

ControlPointAdd

This should be selected for the left mouse action. Click once to set a starting control point. Click again to set another, and so on, until all control points are set. Double click to pick along the control point lines.

AddLeft

This picks every event to the left of the mouse cursor location.

AddRight

This picks every event to the right of the mouse cursor location.

AddBoth

This picks every event to the left and right of the mouse cursor location.

GeomCorrection

This moves the current shot/receiver to the predicted spatial location.

FlipTracePolarity

This flips the polarity of the trace closest to the mouse cursor location.

KillTrace

This kills the trace closest to the mouse cursor location.

PickEventNearsetZero

This automatically picks the event closest to zero time on all visible traces. This is used when the traces have been flattened by applying model traveltime time shifts.

Delay Time Analysis

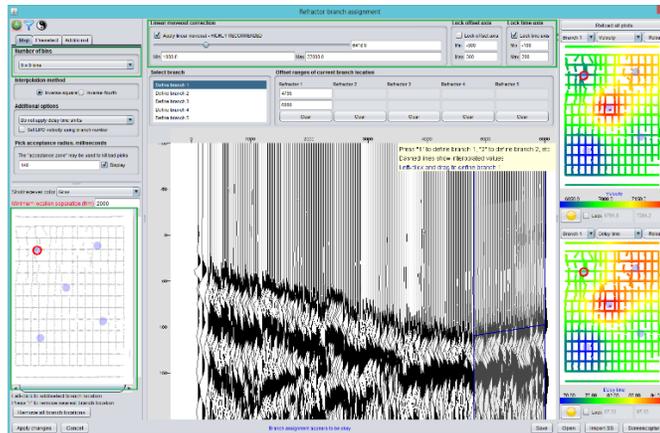
Delay Time Analysis is a layer-based approach to solving near-surface statics. Using either picks or traces, the user specifies the number of layers as well as some initial parameters for the velocity and delay times (which will relate to layer depth).

Summary of Workflow:

- Branch assignment using traces (or picks)
- Refine delay times using shot/receiver stack picker
- Optional: Generate picks based on delay times
- Run delay time analysis sequence
- Perform geometry and pick QC
- Re-run delay time analysis sequence
- Create near-surface model
- Compute statics

1. Branch Assignment

To open the branch assignment window, go to the DelayTime menu and select “Branch assignment - use traces.” If using picks, select “Branch assignment - use picks.” These instructions will assume you are using traces.



If you are using precomputed CMP gathers, you may select whether to use them using the “Preselect” tab in the top left corner.

Select a location on the basemap. The program will load a trace gather for that location. You can use a larger bin size to show more data in the gather, or a smaller bin size to show less data.

The goal is now to find refractors. As the traces go from smaller to larger offset (left to right), we see that the slope of the first arrival changes. Each of these sections with a unique slope is treated as a refractor.

To pick a refractor, first select the refractor number by clicking in the “Select branch” box or by using the numerical keys on your keyboard. Then, click the mouse and drag along the first arrivals for the section of traces, and release the mouse when finished. You’ll see vertical lines at the start and end of the refractor.

Next, select another location on the basemap. It is necessary to track the changes in the refractor along different parts of the survey (Note: in some cases it may be impossible to track a given refractor across the survey. They will sometimes disappear, especially in cases of dipping refractors and other geological anomalies). We will repeat this process for several locations across the basemap.

The color maps on the right hand side are a useful guide when picking refractors. Generally speaking, the refractor velocity, delay times, min and max offsets should not change too drastically across the survey. The maps provide a visual QC to make sure the correct refractor is being tracked.

It is not necessary to pick all the refractors in order to generate an acceptable solution. Oftentimes, an adequate solution can be found by only using one or two refractors. However, it should be noted that picks will only be generated for offset ranges which have refractors picked.

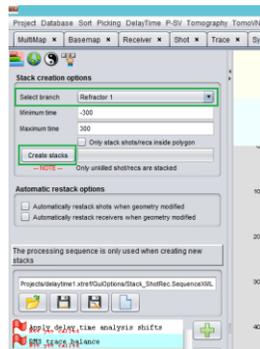
Once the refractors have all been picked across the survey, click the “Apply changes” button in the bottom left corner. A series of dialog boxes will come up:

- First, you will be prompted to interpolate refractor velocities and delay times based on the branch assignment. If you do not already have picks, and have not already done branch assignment prior to this, you should click “YES.”
- Next, you will be prompted to kill picks using the branch assignment information. This is not a recommended way to kill picks. You’ll almost always want to click “NO” here.
- Finally, you will be prompted to run the standard delay time analysis sequence. This will compute a much more accurate delay time model based on the picks. However, this only works if you have picks. For the purposes of this tutorial, we do not have picks yet, so you should click “NO.”

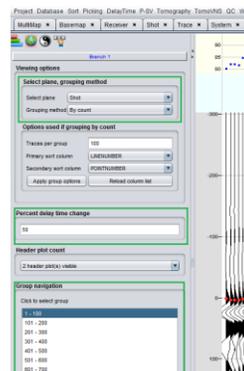
2. Shot/Receiver Stack Picking

Now that branches have been assigned, we can create some shot and receiver stacks in order to refine the delay times.

Go to the DelayTime menu, and select “Shot/receiver stack picker.” This will open up the stack picker window. On the left hand side, check that Refractor 1 is selected, then click the “Create stacks” button.



Select the “globe” icon to view the navigation and sort options:

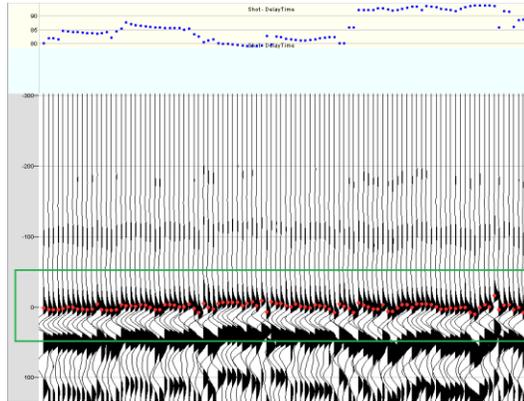


Each “trace” displayed in this window is not a trace, but either a shot or a receiver stack. The stacked trace is made by stacking all traces within a single shot or receiver that are within the specified branch, with a time shift applied to each trace based on the delay time. The more accurate the delay time, the closer the picked event will be to zero.

It is possible to select a shot or receiver in the stack picker and view it in a Picker window. First, make sure there is a Picker window open, with the Plane set to “Shot” or “Receiver” (whichever one you want to view). Then, in the stack picker, mouse over the shot or receiver and press the “ESC” key. A green highlight bar should appear over that shot or receiver in the stack picker.

Go through all of the stacks and pick along the first arrivals. Some useful picking tools are “PickEventNearestTime,” “AddLeft,” “AddRight,” and “AddBoth” on the keyboard, as well as “ControlPointAdd” using the mouse. The delay times are updated instantaneously upon picking.

Once the shots are picked, switch the navigation to receivers and pick those as well (TIP: it is easiest to display shots by count by receivers by LineNumber).



Once all shots and receivers are picked, go back to the stacks icon and click the “Create stacks” button again. When the program finishes reloading the stacks, the events should now look flatter.

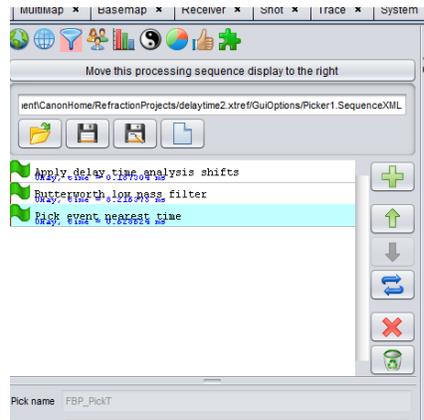
Pick all of the shots and receivers again, then create the stacks again. At this point, the events may be flat enough to move on. If not, repeat the process again.

Repeat this entire process for each refractor picked in branch assignment.

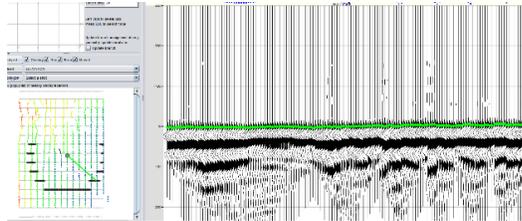
3. Generate Picks (optional) and Run Delay Time Analysis Sequence

While this step is optional if you already have picks, for the purposes of this tutorial, we will cover this step.

In a Picker window (open one if necessary), click the “Defaults” button, then select “Prepare options for delay time analysis.” This will add some items to the interactive processing flow (which can be viewed by clicking the Funnel icon on the top left), as well as set up some display options.



Click around the survey and confirm that the picks are acceptable:



Select the “Pick event nearest time” item in the flow window, then click “Start batch picker, add entry to trace table” at the bottom. This will start the autopicker.

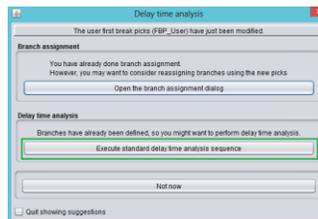


When the autopicker has finished running, a dialog box will pop up asking to copy the picks. The autopicker saves the picks to a trace table column called “FBP_PickT.” In order for picks to be used in analysis, the picks must be copied into a column called “FBP_User.” Click “Open pick copy dialog” on this prompt. A new dialog box will appear:

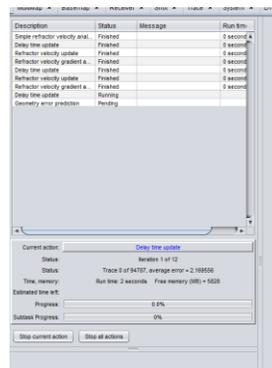


The program will automatically select the columns to copy. Confirm that they are correct and click the “Copy column” button.

Once the program is finished copying the picks, a new dialog box will pop up, asking to reassign branches and/or run the delay time analysis sequence. It’s not necessary to reassign the branches at this point. Click “Execute standard delay time analysis sequence” to begin.



The program will now run the analysis sequence. The progress of the job can be viewed while it’s running:

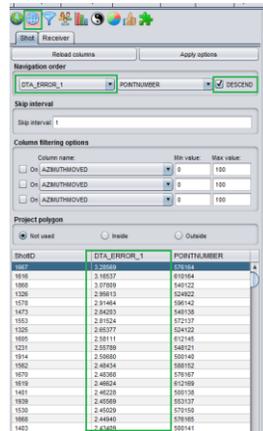


The standard delay time analysis sequence can also be run by going to the DelayTime menu and selecting “Analysis → Default analysis sequence.”

4. Geometry and Pick QC

Now that the delay time analysis computations are complete, the solution can be used to QC both the geometry and picks.

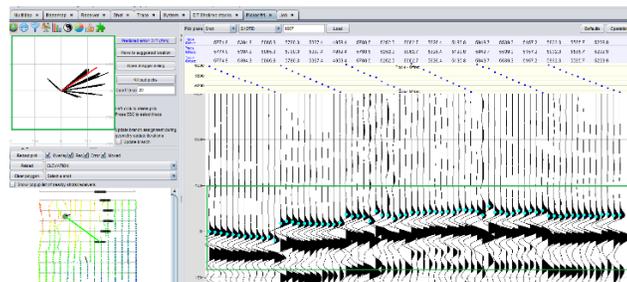
Open a Picker window, and set it to Shot Plane. Select the navigation options (second from left icon). Change the Primary navigation order to “DTA_Error_1” – this is the delay time error term for Refractor 1. Check the “Descend” box so the program displays the larger errors first, then click “Apply options.” The shots are now ordered by highest error.



ShotID	DTA_ERROR_1	POINTNUMBER
1817	3.31079	57114
1818	3.31027	57116
1869	3.31009	540122
1526	2.89615	524922
1578	2.91464	568142
1473	2.84203	540138
1553	2.91524	572127
1525	2.85377	524122
1885	2.98111	571145
1521	2.93789	540121
1814	2.91085	568143
1562	2.88434	568182
1670	2.45385	570187
1639	2.46824	602186
1481	2.45228	568138
1638	2.45589	602187
1530	2.45529	570150
1666	2.44945	540185
1483	2.43483	500141

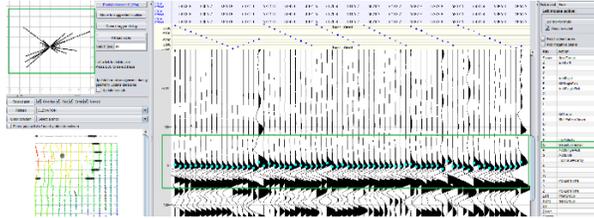
Navigate through shots using the list of error terms. Fix any bad picks that are visible. Ideally, the error terms should be under 5. However, this is not always possible.

Geometry errors can also be identified during this process. Assuming all traces are picked correctly, the vector plot in the top left shows the predicted geometry error from each trace. A composite red vector shows the total predicted move. When the vectors show a strong trend such as in the image below, the shot (shots are more likely to have geometry errors) is a good candidate for geometry correction.



In order to automatically move a shot to the predicted location, the “GeomCorrection” picker key command must be assigned to a key. With the mouse cursor over the trace display panel, press the GeomCorrection key to move the shot. This operation can be undone by clicking the “Operations” button and selecting “Undo geometry correction.”

Once a shot has been moved, the traces that fall within its branch offset limits will change. This means that some traces will now be in the gather which have not yet been picked. If this occurs, pick any unpicked traces and fix any bad picks that are visible. At this point, there may still be a predicted geometry error. Continue using the GeomCorrection key, along with fixing picks, until the shot is in the correct location.



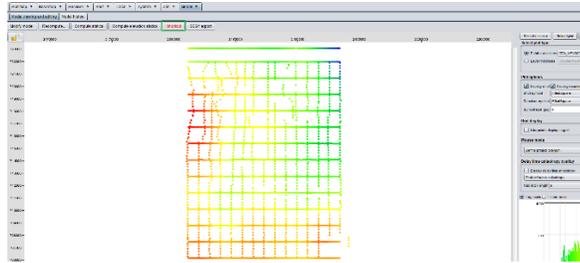
Once the shots have been checked (it is not practical in most cases to check all of them, so we recommend just checking the ones with the largest error terms), switch the Pick Plane to Receiver, and make sure the navigation order is set to “DTA_Error_1,” Descending, for the Receiver table. Go through the receivers with the largest error terms and fix any bad picks that are visible. Generally speaking, receiver geometry errors are less common, but they do occur, so keep an eye out for those as well.

If there are multiple refractors assigned, repeat this QC process for the other refractors using “DTA_Error_2,” etc.

Any time the picks and/or geometry are modified, the delay time analysis sequence should be run again. This can be done by going to the DelayTime menu and selecting “Analysis → Default analysis sequence.”

5. Delay Time Model Building

The delay time information can now be used to construct a model of the near-surface. Go to the DelayTime menu and select “Model building window.”



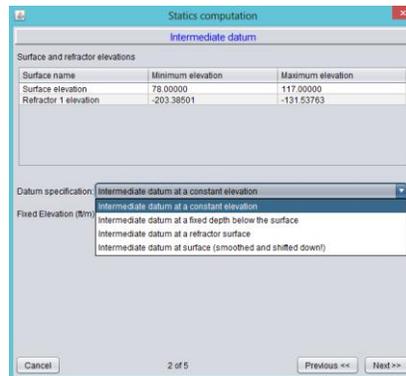
Click the “Shortcut” button. It will handle everything automatically with only a small amount of parameterization. A dialog box will pop up. Enter the desired value for the weathering velocity. This needs to be lower than the first refractor velocity, which can be viewed in the “DTA_Velocity_1” column in the database. The basemap can be colored using this value. The refractor smoothing radius can also be set; a lower value results in less smoothing, although the default is recommended.

6. Delay Time Statics Computation

In the Delay Time Model Building window from the previous step, click the “Compute statics” button. A dialog box will appear.

On the first page, enter a name for this set of statics. The program will show the database column name where it will be saved.

The next page will display the minimum and maximum values for the elevations of the surface and all refractors. Specify an intermediate datum. The intermediate datum should be set just below the bottom-most refractor.



The page after this displays the elevations from the prior page as well as refractor velocities. Specify the final datum and replacement velocity. These values should be provided by the client.

On the fourth page, select the method of calculating the traveltimes to the intermediate datum. The differences between the methods are explained in the dialog box. There is only one case where there is an actual difference in the result, and it's not necessary to overthink this step.

On the final stage, specify whether to bulk shift the statics – this is useful for comparing statics, and is covered in the Stack QC tutorial. Please note: for production statics, do not bulk shift using this step.

The statics values will be saved to the Shot and Receiver database tables.

VNS Tomography

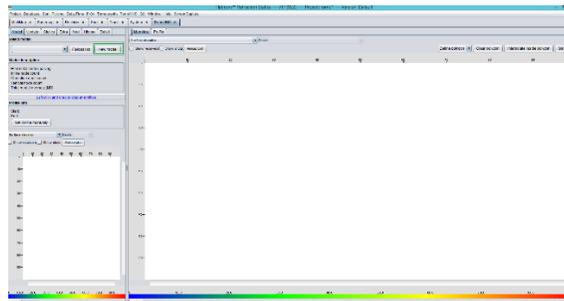
Tomography uses a node-based approach to model the near surface. Typically, these nodes are evenly spaced in depth. However, Flatirons™ contains both a standard (evenly spaced) as well as variable spaced tomography module. This tutorial focuses on the variable node spacing (VNS) tomography.

Preparation

Tomography requires that good, accurate first break picks be available. The more offsets covered by these picks, the better the expected result – however, the priority should be accuracy in the near offsets, as these picks are treated as more important by the algorithm.

For the purposes of this tutorial, we will assume that first break picks have already been generated.

To open the VNS Tomography window, go to the TomoVNS menu, and select “Variable node spacing (VNS) tomography.”



Summary of Workflow:

- Create coarse model
- Update coarse model
- QC coarse model
- Create fine model
- Update fine model
- QC fine model
- Repeat above as needed
- Compute statics
- (optional) Export velocity field to SEG-Y

1. Create Coarse Model

To create a new model, click the “New model” button on the left panel. A dialog box will appear.



Set the angle of the grid by dragging a line on the map. The purpose of this is to minimize the amount of unused space in the model, so it’s advisable to line the grid up with the survey.

Set the horizontal grid spacing for the model, and give it a name. A good starting point for a coarse model is between 150 and 250 feet (40 to 80 meters). The coarser the model, the faster it will run, and the first models should be fast.

Set the vertical grid spacing at the surface and at max depth. This is the key to VNS. The grid spacing is finer near the surface, where velocities are expected to change more rapidly, but coarser near the base of the model, where velocities are expected to change less rapidly.

Set the initial velocity at the surface and max depth, and select how the velocity is calculated at each node – using elevation or depth.

If this is a marine streamer or OBC survey, check the “Marine survey” box and set the water velocity. Note that the Shot and Receiver tables must have valid “WaterDepth” columns in order to use this option.

2. Update Coarse Model

Select the Update tab (second from left).

The screenshot shows the 'Batch update options' dialog box. The 'Update' tab is selected. The following parameters are visible:

- Minimum offset: 200
- Maximum offset: 6000
- Number of iterations: 10
- Horizontal smooth radius (nodes): 2
- Weight at max radius (<= 1): 0.2
- Vertical smooth radius (nodes): 0
- Pick version: FBP_User
- Pick shift (ms): 0
- Number of threads: 2
- Allow velocity inversions:
- Accuracy (2 - 5): 2

Buttons and options below the input fields:

- Shallow velocity lock:
- Uphole survey:
- Azimuth:
- Hybrid:
- Decimation:
- Apply azimuth restrictions:
- Center azimuth: 45
- Azimuth radius: 45

A 'Start batch update' button is located at the bottom of the dialog.

Set the minimum and maximum offsets used to update the model. Recall that the near offsets are the most important for tomography.

Select the number of iterations to run. Since this is the first model, it is advisable to run less iterations at first, to avoid wasting time if the parameters are not good. The number of iterations can be increased as the process is repeated.

The default smoothing radius and weight values should be used.

Select which pick version to use.

Picks should not be shifted unless there is a specific reason to do so.

Set the number of threads to use – more threads is faster; the maximum number is limited to twice the number of processor cores available on the CPU. If a value larger than the maximum is used, it will default to the maximum.

Set the accuracy value – this may result in a marginal benefit but at the cost of a great deal of extra time. We recommend always using 2 unless time is not an issue.

Once parameters are acceptable, click “Start batch update” to begin updating the model.

This may take some time depending on the number of picks and the size of the model.

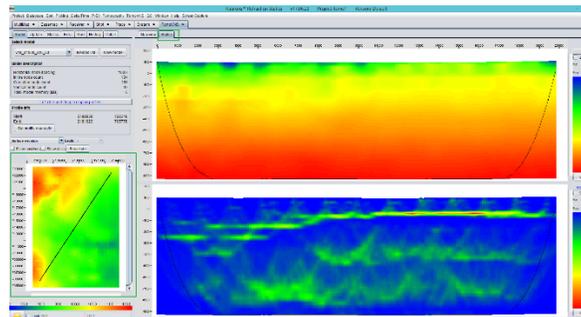
3. QC Coarse Model

There are several options available for to QC tomography models. They will be covered here. These QC methods should be used for the initial coarse model along with any finer models that are created and updated hereafter.

Profile View

In the TomoVNS window, select the Model tab on the left panel and the Profile tab in the middle panel.

Click and drag a line across the basemap in the bottom left. A 2D depth view of the profile will be displayed in the main panel. The top plot is the velocity model. The bottom plot is the node hit count, or how many times a ray was traced through that node. A larger hit count means that a node has been updated more by the algorithm. It gives an idea of far down the traces are traveling through the model.



Error Term

Select the “History” tab on the left. You may need to click the “Reload history” button.

The average error (in milliseconds) in each iteration is listed. When a model has converged to a solution, the change in error between iterations should be fairly small.

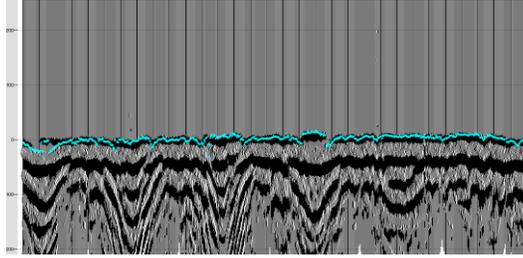
Picker Window

In a Picker window, click the Defaults button and select “Prepare options for TomoVNS analysis.” This will set up the processing flow to apply VNS model time shifts to the traces.

The model currently selected in the TomoVNS window will be used to compute these time shifts.

If the model is perfectly correct, the gathers should be completely flat, with the picks all on zero time.

Select different locations on the basemap and determine the quality of the model based on how well they flatten gathers.



TomoVNS Stack Picker

Go to the TomoVNS menu and select “Shot and receiver stacks using TomoVNS solution”

Make sure the correct model is selected in the TomoVNS window before creating stacks.

These work much like the Delay Time stacks, but uses the TomoVNS model time shifts instead of delay times. However, unlike the Delay Time stacks, you cannot iteratively pick and restack these. These stacks may only be picked once to generate a residual error term.

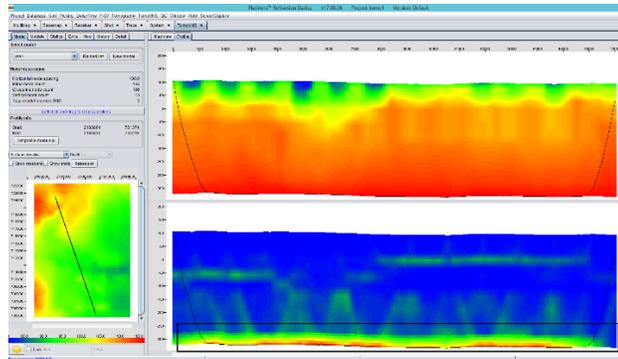
This is primarily a visual QC tool, much like the Picker window option. A better solution will create flatter stacks.

4. Create, Update, and QC finer Models

Repeat the above process, but using finer node spacings. If the models are not converging, increase the number of iterations run. It may require creating and updating several models before an acceptable solution is found.

5. Common Mistakes

Refer to the following model:



Note the bottom of the hit count display – the rays are going all the way to the bottom of the model. There is no way of knowing if they should go even lower, because the model is not deep enough. In this case, a new model must be created with a greater depth.

6. Compute TomoVNS Statics

Select the Statics tab. All parameters for statics computation are edited on this page.

Model Update Statics Extra Mod History Detail

Rebuild list of database columns

Intermediate datum options

- Fixed elevation: -850
- Fixed depth: 400
- Fixed velocity: 5000
- Database column: CROSSLINE

Final datum options

- Fixed elevation: 120
- Database column: CROSSLINE
- Replacement velocity: 6000

Residual correction options

Residual terms are computed during batch update

- Apply residual correction
- Residual multiplier: 1

Bulk shift options

- Do not modify the statics
- Convert the statics to zero mean
- Tie the statics to another column

CROSSLINE

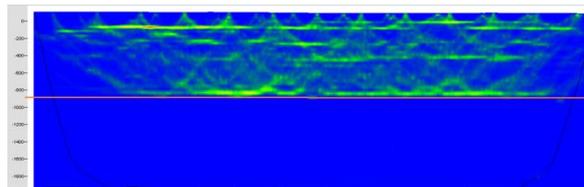
Statics name: TomoVNS

Compute statics

Refer to the node hit count chart on the profile.

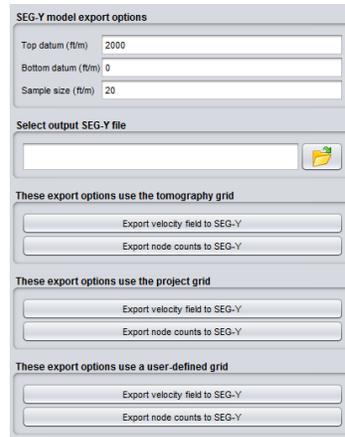
If there are no rays updating the bottom part of the model, that part of the model should not be used for computing statics, because it has not been updated – it's simply the starting velocity model. The "Highlight intermediate datum" checkbox may be selected to display the selected datum on the profile.

Refer to the below image. In this example, the intermediate datum should be set around an elevation of -850 or -900, where the orange line is drawn:



7. Export velocity field to SEG-Y (optional)

It may sometimes be desirable to export the TomoVNS velocity field to an interval velocity model in SEG-Y form. This may be done in the Extra tab.



The screenshot shows a dialog box titled "SEG-Y model export options". It contains three input fields: "Top datum (ft/m)" with the value "2000", "Bottom datum (ft/m)" with the value "0", and "Sample size (ft/m)" with the value "20". Below these fields is a section titled "Select output SEG-Y file" with an empty text box and a file selection icon. The dialog is divided into three sections based on the grid type used for export:

- These export options use the tomography grid:** Two buttons labeled "Export velocity field to SEG-Y" and "Export node counts to SEG-Y".
- These export options use the project grid:** Two buttons labeled "Export velocity field to SEG-Y" and "Export node counts to SEG-Y".
- These export options use a user-defined grid:** Two buttons labeled "Export velocity field to SEG-Y" and "Export node counts to SEG-Y".

Set the elevation at the top and bottom datums to define the bounds of the exported model. Also set the sample interval.

Both the velocity and node counts may be exported.

There are three options for the output grid:

- The tomography grid (x and y spacing defined at model creation)
- The project grid (uses the internal Inline and Crossline values in the project)
- User defined grid (a dialog box will appear asking for a three corner grid definition)

Stack QC (Statics QC)

The QC menu contains options for statics QC using internally generated stacks. This allows the user to QC statics solutions before importing them into other processing software.

Summary of Workflow:

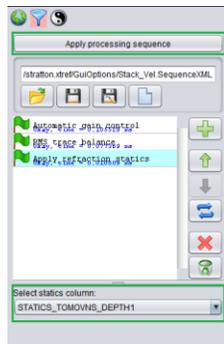
- Pick stacking velocity field
- Create profile lines and stacks along profiles
- Compare stacks

1. Stacking Velocity Picker

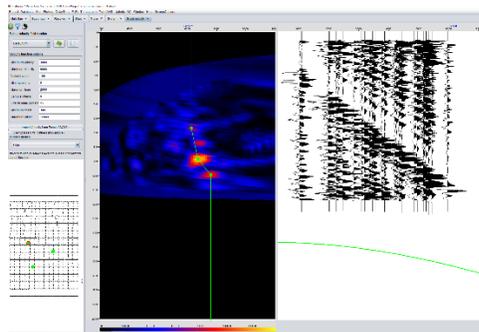
From the “QC” menu, select “Stacking velocity picker”. This will open a window for picking a rough velocity model for QC purposes.

Please note that this is not intended to be a full-featured velocity picking tool. It is only used for a very rough approximation purely for internal QC.

First, apply a refraction statics solution. Click on the processing funnel icon, then click the plus sign, and select Shift T0 → Apply refraction statics. Select the preferred statics solution from the drop down selector under the processing flow, then click “Apply processing sequence.”



In the velocity picker, select a location to pick velocities by clicking on the basemap. Precomputed CMP gathers may be used.



The “heat map” display shows the velocities that should be picked, with velocity plotted against time. Click on a spot in order to pick that velocity. Velocities should increase with depth/time. If the surface velocity is known, a pick near zero time at that velocity may help the coherence of the stacks.

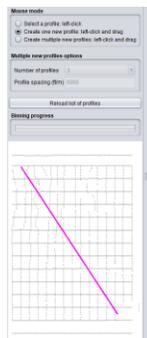
Pick several locations. Some locations will not show a coherent semblance anywhere; if that is the case try to pick a nearby location. Make sure to pick velocities in locations that are known to have geological differences, if possible.

There is no need to save or do anything else once the velocities have been picked; all picks are saved as soon as they are made.

2. Create Profile Lines and Stacks along Profiles

From the QC menu, select “View stacks along profile.” This will open the stack profile creation/comparison window.

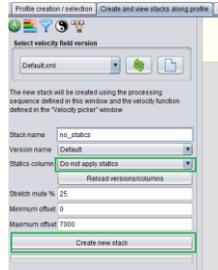
In the first tab of this window, “Profile creation/selection,” click the box to “create a new profile: left click and drag.” Then select the profile by using the mouse on the basemap.



A dialog box will pop up asking for parameters. Click “Create new profile” to start. If processing a crooked or turning 2D line as a 3D project, check the box to “snap all traces to the profile line.” Do not do this for actual 3D surveys.

Once a profile has been created, move to the second tab in the profile menu, “Create/view stacks along profile.” Click the bars icon (second from the left).

The stack creation options are here. Note the “Statics column” field. Stacks may be created with any statics solution, or with no statics applied.



First create a stack with no statics applied to serve as a point of comparison. It may also be informative to create a stack with elevation statics applied.

Next, create stacks with any statics solutions to be compared. Stacks do not need to be created for statics solutions that will not be compared or used.

Note: we recommend that for comparison stacks, statics tied to the same average value (generally tied to the elevation statics field) be used. Clean statics (not bulk-shifted for comparison) can be recomputed after the comparisons are complete.

3. Compare statics

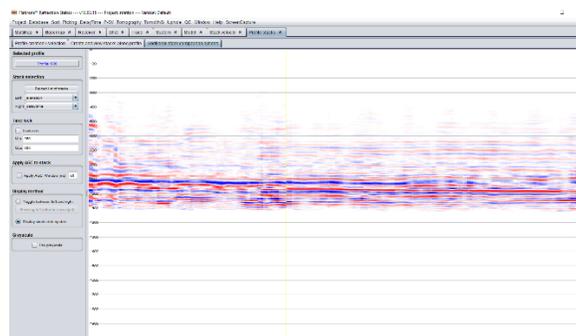
The third tab in this window, “Additional stack comparison options,” provides the display functionality for stack QC.

First, click the button to “Reload list of stacks.” Since stacks have just been created, they need to be loaded into this window.

Next, select the two stacks to compare. “Left” and “right” refer to the secondary option for display (side-by-side) which is covered below.

The first option for comparison is to toggle between the two stacks. Click the button to switch between them.

The second option allows the user to display both using a sweeping cursor. The left stack is displayed to the left of the mouse cursor and the right stack is displayed to the right.



Greyscale is available if the default color scheme is not preferred.

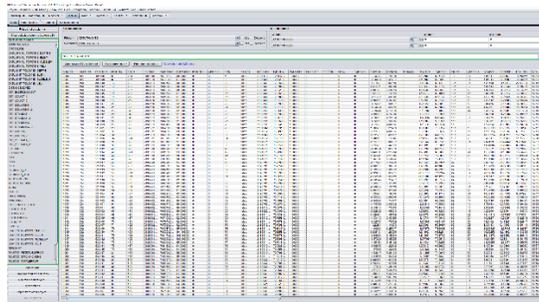
This is primarily a visual comparison. Look for the coherence of events below the surface to judge the quality of the statics solutions.

Refraction statics stacks should be compared to one another as well as to the elevation statics and “no statics” stacks.

Database

The refraction database contains all of the geometry information for the project. By default, this consists of the Shot and Receiver tables. Additional tables may be imported, but that will not be covered in this tutorial; other tutorials are available that cover additional table import. Please note that the Trace table is not contained in this database – it’s stored in a binary file and does not have the functionality described in this tutorial.

The Shot and Receiver tables are both open by default. If they are closed, they can be reopened by going to the Database menu and selecting “Shot table” or “Receiver table.”



The database is modified and displayed using SQL commands; however, the display functionality may be controlled by simply using the mouse and selecting options.

Select columns to display, then click on the “Run query” button in order to show the selected columns. Clicking on the header of a column will sort the table using the data in that column. Click again to switch between value ascending and value descending for the sort.

Additional options are found in the bottom left corner. To add a column to the database, click the “Add column” button to bring up a dialog box. Columns may be added to either or both the Shot and Receiver tables.

The “Column modifications” button has a number of other database tools.

Overview:

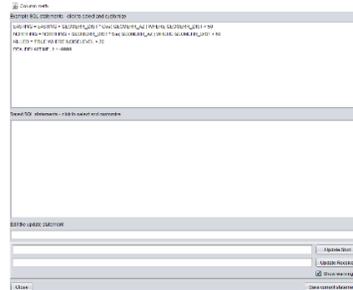
- Column math

- Interpolating columns

- Statics export

Column Math

The column math dialog box is used to run SQL commands directly on the database in order to modify its contents. To open this dialog, click the “Column modifications” button in the bottom left and select “Open column math dialog.”



Example statements are listed. User-defined statements may be saved for later use. Once the command has been typed in, select the Shot/Receiver checkboxes to mark which databases to modify. Click the button to run the SQL modification.

Interpolating Columns

There are several different options available for interpolating data between columns. All of them are available from the “Column modifications” button.

Exporting Statics and Other Information

In order to export information from the database, first select the columns to export, then click “Run query” in the main database window. Once the columns are displayed, click the “Export options” button in the bottom left corner, then select the desired output format. CSV uses commas, and column-delimited uses spaces.

A dialog box will appear simply asking for the file location to save. Please note, this process only exports the table that’s selected (Shot or Receiver), so it must be done for both tables.