XtremeGeo

USER GUIDE

FLATIRONS™
REFRACTION STATICS

Flatirons User Guide

June 2025

Version 2

PO Box 350021
Westminster, CO 80035 USA

xtgeo.com
720-505-6149

Table of Contents

Table of Contents

1 Introduction

Warning: This document is a partial draft. Chapters are incomplete, some chapters are missing, appendices are incomplete or missing and no editing has been done. Many figures are simply placeholders and do not represent proper usage. Other than that, enjoy!

Introductory material for Flatirons. Overview of Flatirons – refraction statics, delay time, GLI, tomo. Note: This User Guide was written from a functional viewpoint. In Part II, each chapter proceeds through the steps required to perform some task. For example, Chapter TBD walks the user through the task of building a new Flatirons project from SEG---Y data. Etc.

2 Creating a Project

In order for Flatirons to compute statics, a project has to be created. In this Chapter a new project will be created from a SEG---Y data file using a wizard dialog. A previously created SEG---Y dictionary file will be assumed. For details on how to create a SEG---Y dictionary, see Chapter 3.2.

2.1 Begin Flatirons

In order to run a license server and install/set up Flatirons or Phoenix:

Place the license file in the extracted licensing toolkit folder (FlexLM-D.tar.gz). Navigate to that folder in the terminal, and use the command:

./lmgrd -c (license name).lic

The software can be downloaded from our website at www.xtgeo.com/resource-center/downloads. Download the latest version of Flatirons++ and extract into the same folder as the license file.

Before starting Flatirons, the following environment variables will need to be set on the user's computer:
XT_FLEXPORT (set to 27011; change this if you change the port number in the license file)
XT_FLEXHOST (set to the IP address of the license server)

To run the Flatirons directly without using Flatirons++ in Linux/Max, open a terminal window and change directory to the application installation directory. Enter the following command

java –Xmx4000M –Xms4000M –jar Flatirons.jar

to start Flatirons. The -Xmx and -Xms settings specify the amount of memory to allocate for Java total, and the amount of that memory to use at startup, respectively. In this example I allocate 4 GB, which is enough for most surveys. However, surveys with high trace counts will require more memory, especially if tomography is being performed. We used 12 GB to handle a 2 billion trace survey. The first window gives the option of opening an existing project or creates a new project in several ways. An additional option is to open the SEG---Y viewer and dictionary editor, which will be covered elsewhere.

For this example, we select **Create a new project based on SEG---Y files** as shown in Figure 2---1.

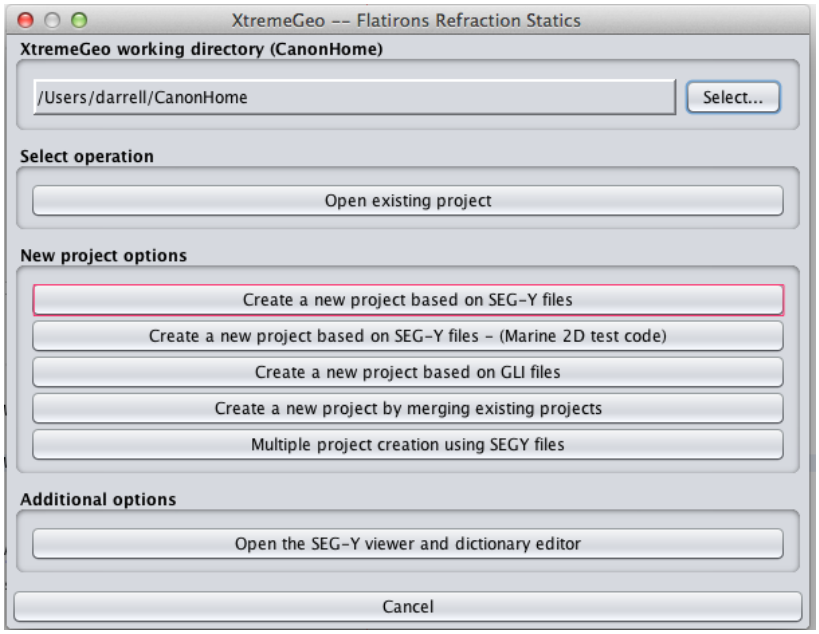


Figure 2---1 Opening window for Flatirons. The starting point for opening or creating a project.

The first window of the project creation wizard is the SEG---Y import panel (Figure 2---2). Project options for name, dimension and units are chosen, as well as various options for a marine survey.

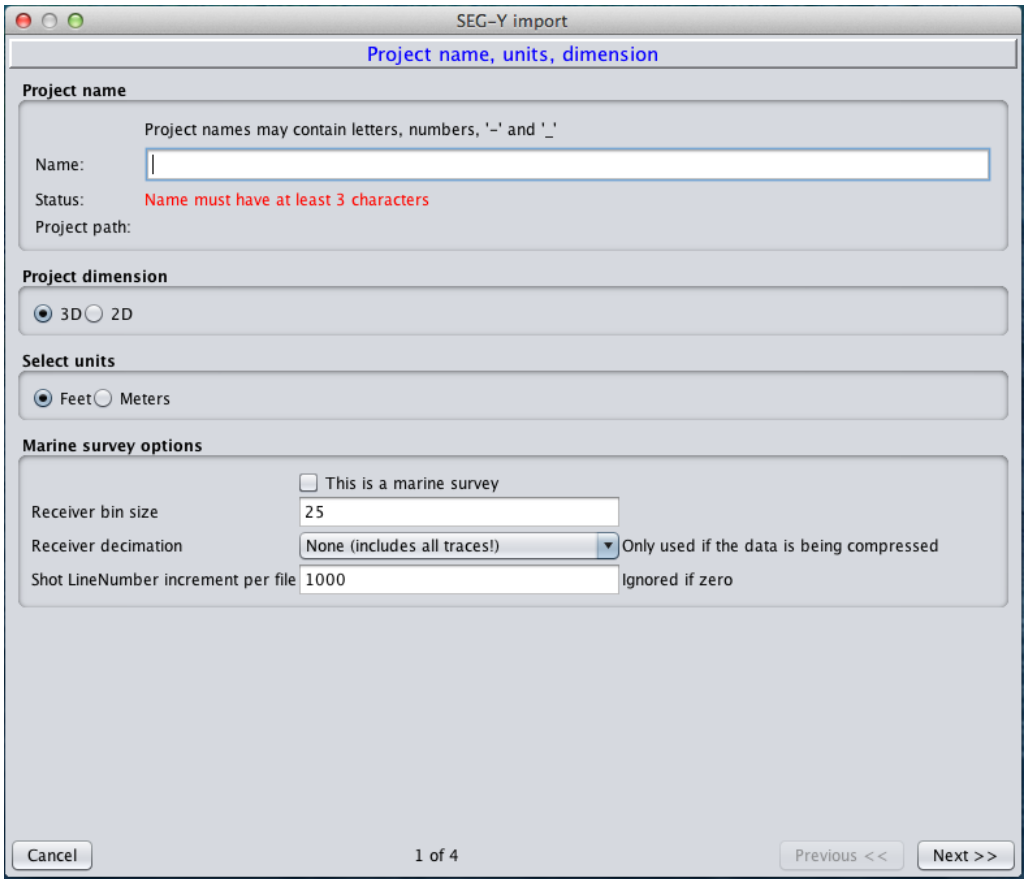
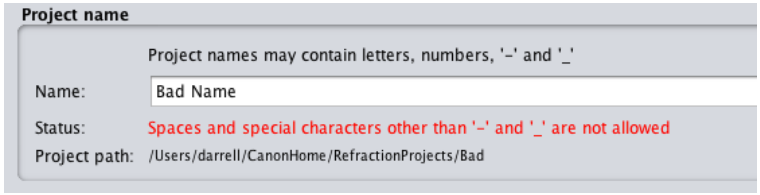


Figure 2---2 First window in the project creation wizard is the SEG---Y import window.

For the project name, the project path and status are indicated. The project name cannot have spaces or special characters (number are okay). Figure 2---3 shows the display for both acceptable and unacceptable project names.



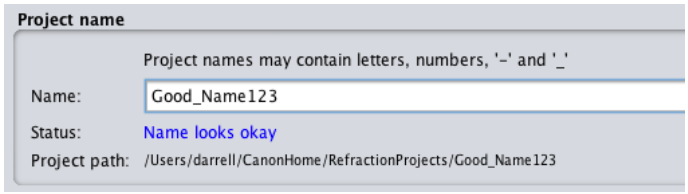


Figure 2---3 Project name text box. Examples for both acceptable and unacceptable names are shown.

The next window, **Compression options**, has compression, CMP and noisy traces options (Figure 2---4).

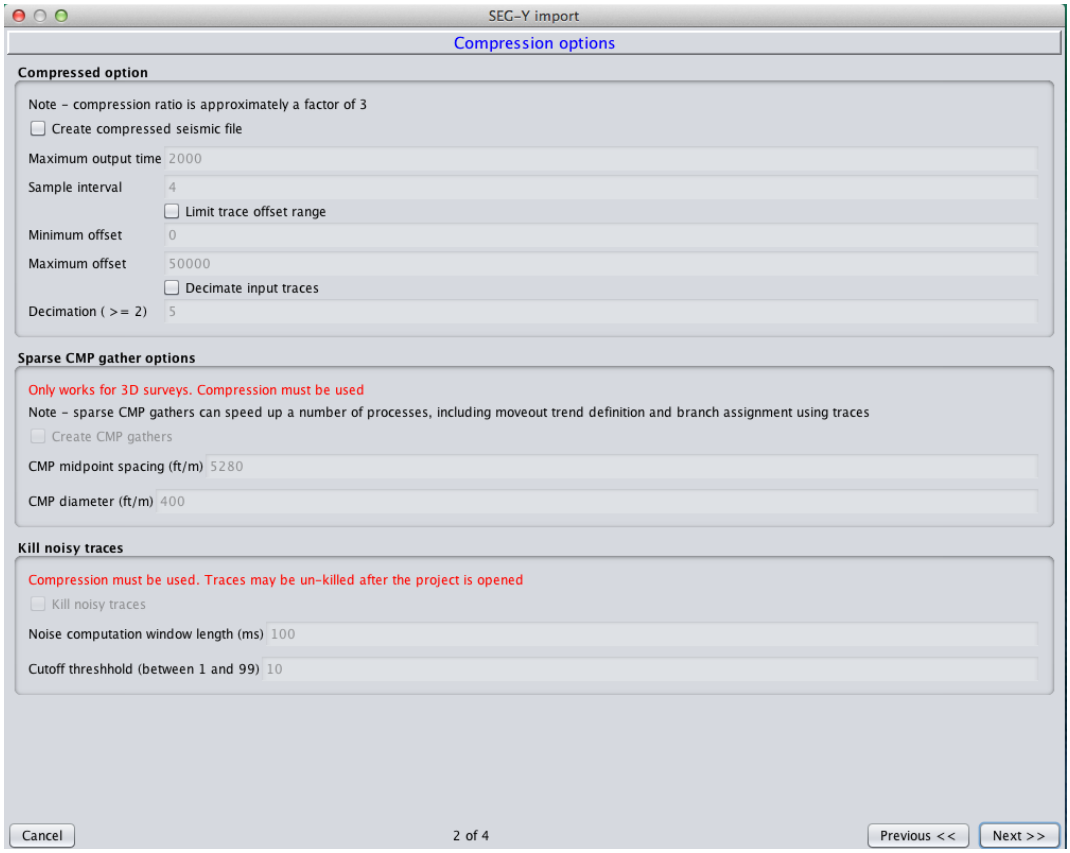


Figure 2---4 Compression options when creating a new project.

2.2 Compressed option

- **Create compressed seismic file**. Seismic data is compressed and saved to a new file. The original file is unaffected. Compression ratio is typically about three.
- **Maximum output time**. Default is 2000 milliseconds. Can be used to truncate, but not pad, the seismic data. (Only available if compression has been selected.)
- **Sample interval**. Default is 4 milliseconds. (Only available if compression has been selected.)
- **Limit trace offset range**. The minimum and maximum offset range, in the units previously chosen, can be selected. (Only available if compression has been selected.)
- **Decimate input traces**. Reduce the number of traces by an integer factor. For example, a decimation of 5 will keep every 5th trace.

Sparse CMP gather options (Applies only to 3D surveys and only if the compression option was selected.)

- **Create CMP gathers**. Create CMP gathers at specified spacing with specified diameter. This option speeds up a number of processes, including moveout trend definition and branch assignment using traces. Although this option adds a small

amount of overhead at the beginning of the process, the speed gains are quite significant. This option is highly recommended. Note that spacing and diameter selected here can be modified at a later time – there is no need to create a new project.

- **CMP midpoint spacing (ft/m)**. The distance between midpoints of the sparse CMPs.
- **CMP diameter (ft/m)**. The diameter of the sparse CMPs.

Kill noisy traces (Compression must be used. Traces may be un---killed after the project is opened.)

- **Kill noisy traces**. This option applies a simple algorithm to identify and flag noisy traces. This option is not recommended for typical data.
- **Noise computation window length (ms)**. Length of the window used by the algorithm.
- **Cutoff threshold (between 1 and 99)**. Threshold for identification of noisy traces. Small values kill more traces while large values kill fewer traces.

The third window, **Select SEG---Y files**, is used to choose the SEG---Y data files required for the project. One or more files, from one or more directories are selected using the **Add files(s)** button, as shown in Figure 2---5. All SEG---Y files selected must use the same header dictionary.

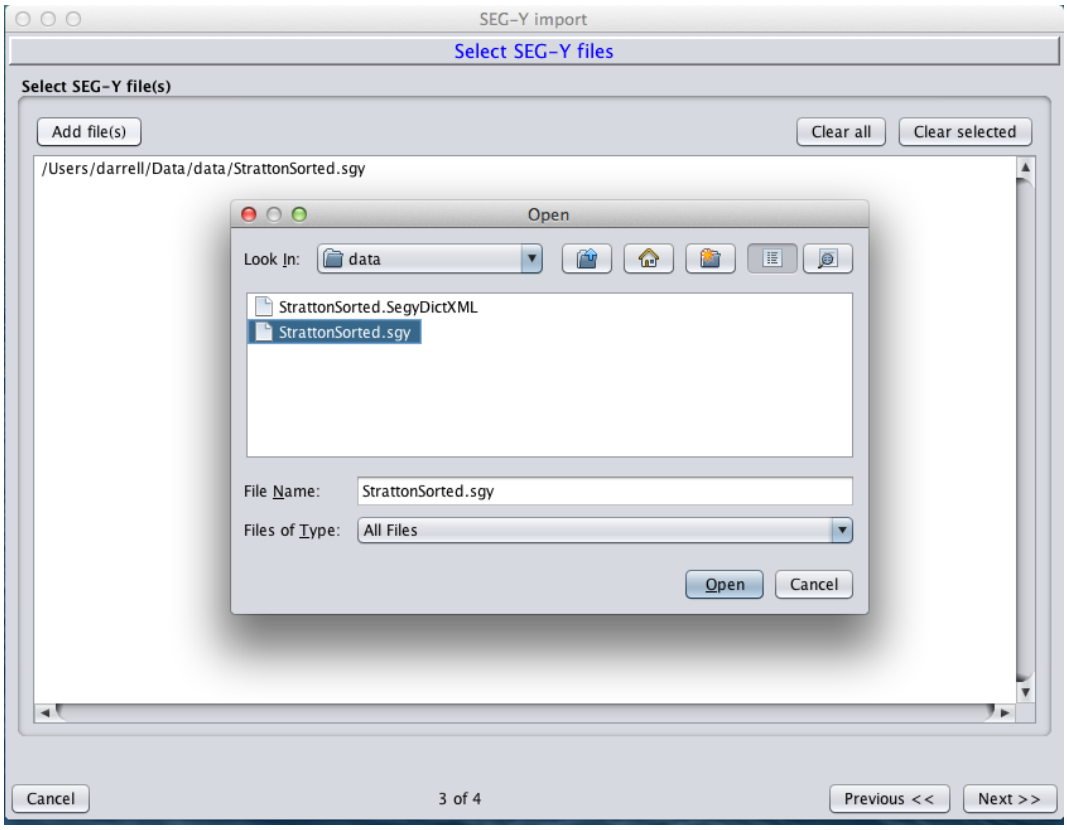


Figure 2---5 The Select SEG---Y files window with Add files dialog box.

In the fourth window, **SEG---Y header dictionary file name**, a header file is opened, created or edited (Figure 2---6).

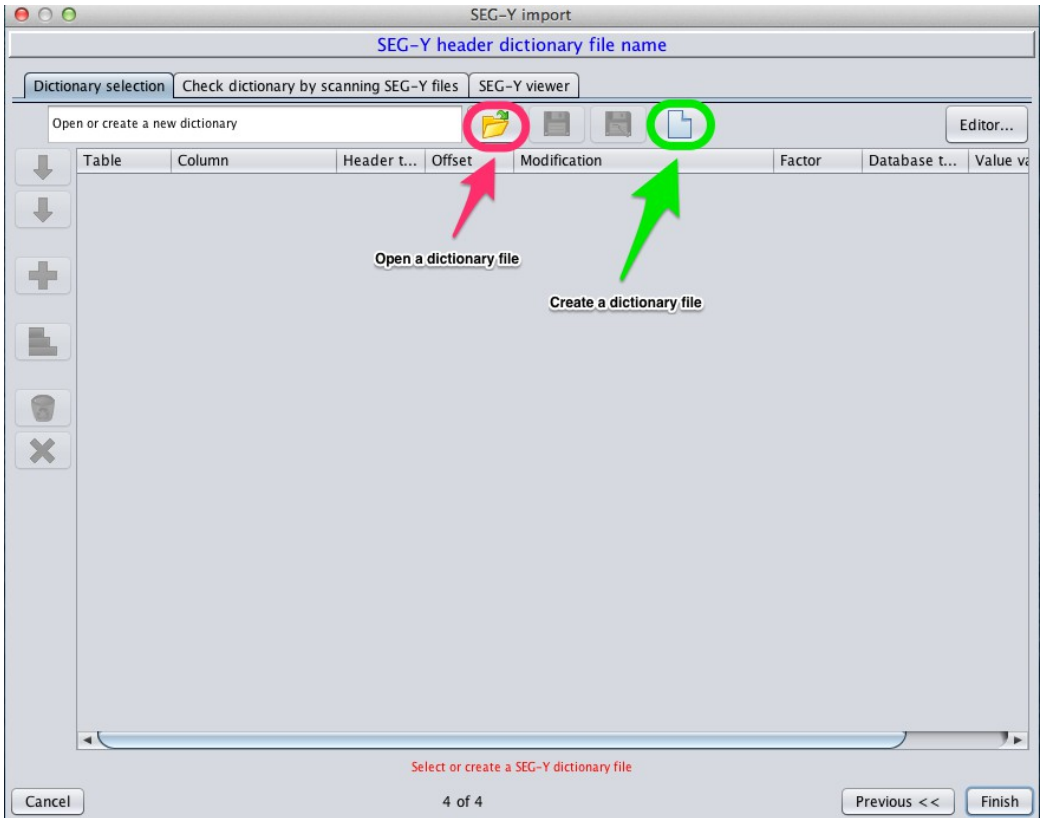


Figure 2---6 For SEG---Y import, this window provides for opening, creating or editing a dictionary file.

Click the **File Dialog** icon (📂) to select a previously defined dictionary file. If this icon is not visible, make sure that the first tab, **Dictionary selection,** is active. Once a file has been selected the panel will be populated with rows and columns from the dictionary. On input the application will attempt to determine if the dictionary is in the proper format and indicate the status at the bottom of the panel (Figure 2---7).

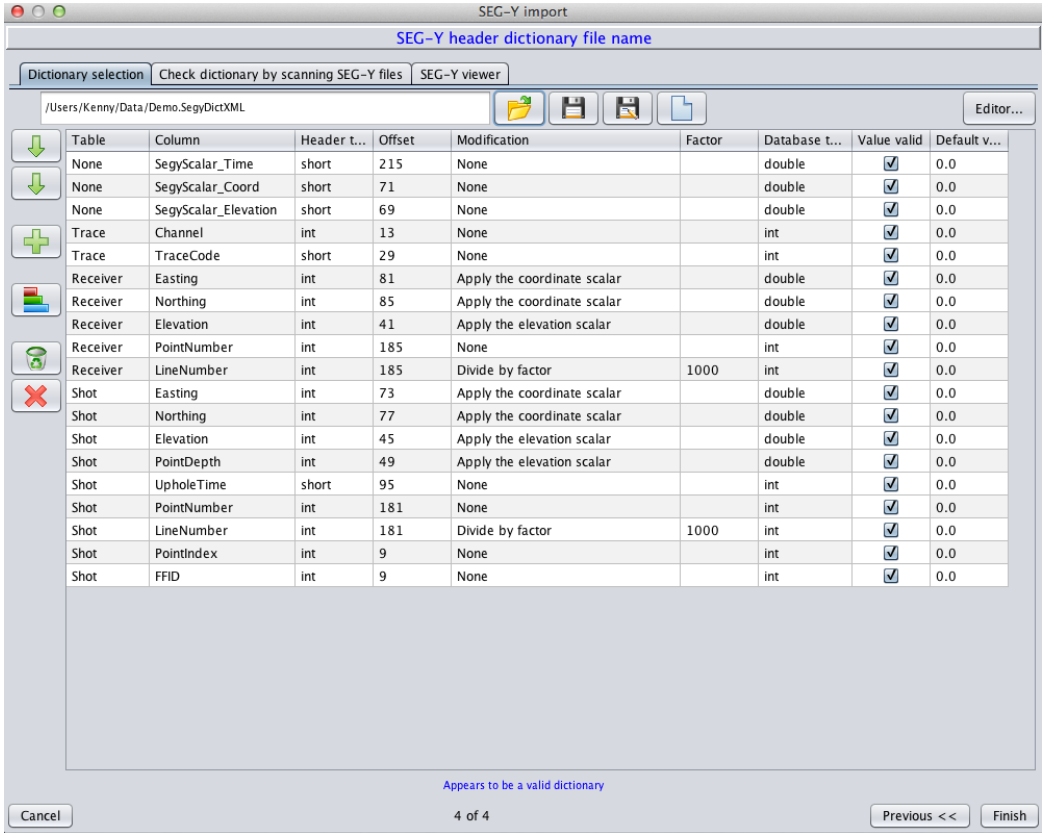


Table	Column	Header t...	Offset	Modification	Factor	Database t...	Value valid	Default v...
None	SegyScalar_Time	short	215	None		double	☑	0.0
None	SegyScalar_Coord	short	71	None		double	☑	0.0
None	SegyScalar_Elevation	short	69	None		double	☑	0.0
Trace	Channel	int	13	None		int	☑	0.0
Trace	TraceCode	short	29	None		int	☑	0.0
Receiver	Easting	int	81	Apply the coordinate scalar		double	☑	0.0
Receiver	Northing	int	85	Apply the coordinate scalar		double	☑	0.0
Receiver	Elevation	int	41	Apply the elevation scalar		double	☑	0.0
Receiver	PointNumber	int	185	None		int	☑	0.0
Receiver	LineNumber	int	185	Divide by factor	1000	int	☑	0.0
Shot	Easting	int	73	Apply the coordinate scalar		double	☑	0.0
Shot	Northing	int	77	Apply the coordinate scalar		double	☑	0.0
Shot	Elevation	int	45	Apply the elevation scalar		double	☑	0.0
Shot	PointDepth	int	49	Apply the elevation scalar		double	☑	0.0
Shot	UpholeTime	short	95	None		int	☑	0.0
Shot	PointNumber	int	181	None		int	☑	0.0
Shot	LineNumber	int	181	Divide by factor	1000	int	☑	0.0
Shot	PointIndex	int	9	None		int	☑	0.0
Shot	FFID	int	9	None		int	☑	0.0

Figure 2---7 A SEG---Y dictionary file. Individual cells can be edited. Icons on the left add functionality.

Several icons on the left side of the panel add functionality. From top to bottom they are: Sort by Table/Column ( ), Sort by Offset ( ), Add an entry ( ), Use default post---stack map ( ) (this icon is not used by Flatirons), Remove all entries ( ) and Remove selected entry ( ).

At this point, clicking on the **Finish** button will create the project. However, to more thoroughly verify the format of the dictionary, select the second tab **Check dictionary by scanning SEG---Y files**. Then click on the **Begin scan (cannot interrupt!)** button. The default setting will cause the first 100,000 traces to be read from each SEG---Y file. The header information is parsed and checked for conflicts and inconsistencies. The user can select the number of traces to be read, including all traces (Figure 2---8).

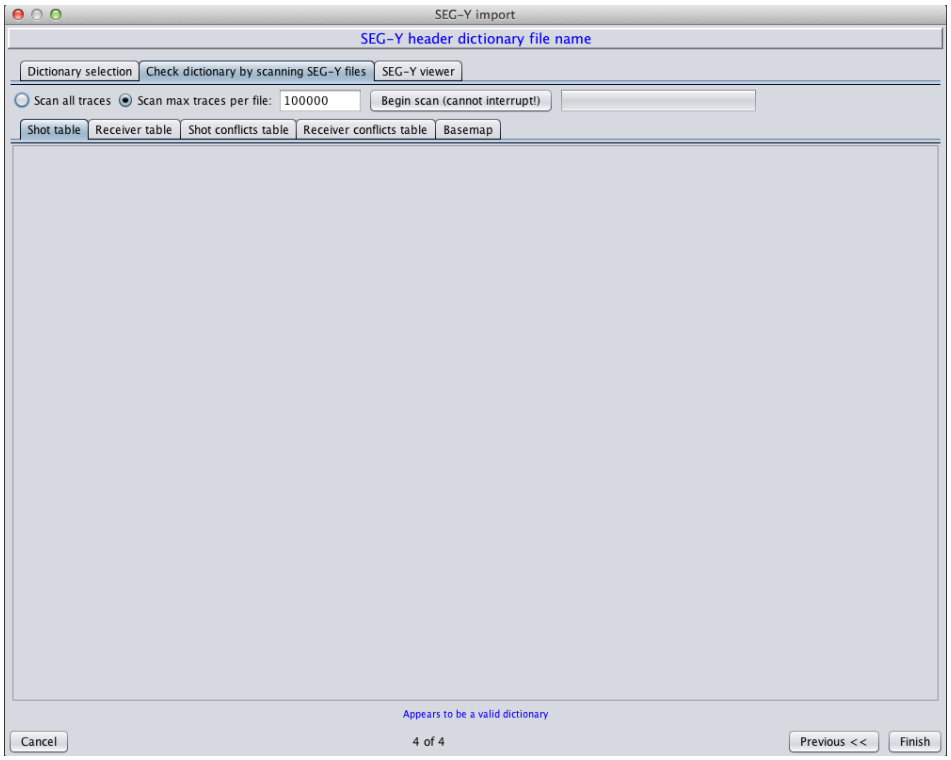
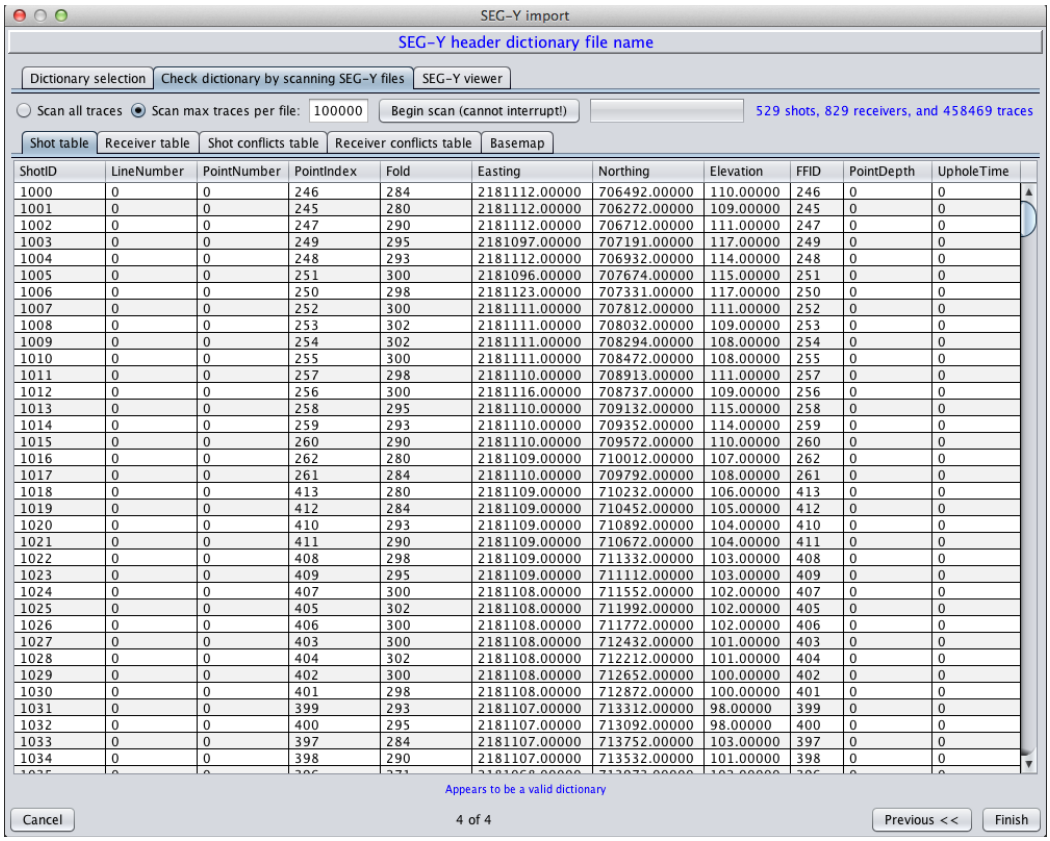


Figure 2---8 Check the veracity of the SEG---Y file by scanning a subset of the file and checking for conflicts and inconsistencies.

When the scan is complete the total number of shot, receivers and traces is displayed adjacent to the scan button. The **Shot table** and **Receiver table** tabs are now populated with shot and receiver header information, respectively. Any identified conflicts are listed in the **Shot conflicts** tab and the **Receiver conflicts** tab (Figure 2---9).



ShotID	LineNumber	PointNumber	PointIndex	Fold	Easting	Northing	Elevation	FFID	PointDepth	UpholeTime
1000	0	0	246	284	2181112.00000	706492.00000	110.00000	246	0	0
1001	0	0	245	280	2181112.00000	706272.00000	109.00000	245	0	0
1002	0	0	247	290	2181112.00000	706712.00000	111.00000	247	0	0
1003	0	0	249	295	2181097.00000	707191.00000	117.00000	249	0	0
1004	0	0	248	293	2181112.00000	706932.00000	114.00000	248	0	0
1005	0	0	251	300	2181096.00000	707674.00000	115.00000	251	0	0
1006	0	0	250	298	2181123.00000	707331.00000	117.00000	250	0	0
1007	0	0	252	300	2181111.00000	707812.00000	111.00000	252	0	0
1008	0	0	253	302	2181111.00000	708032.00000	109.00000	253	0	0
1009	0	0	254	302	2181111.00000	708294.00000	108.00000	254	0	0
1010	0	0	255	300	2181111.00000	708472.00000	108.00000	255	0	0
1011	0	0	257	298	2181110.00000	708913.00000	111.00000	257	0	0
1012	0	0	256	300	2181116.00000	708737.00000	109.00000	256	0	0
1013	0	0	258	295	2181110.00000	709132.00000	115.00000	258	0	0
1014	0	0	259	293	2181110.00000	709352.00000	114.00000	259	0	0
1015	0	0	260	290	2181110.00000	709572.00000	110.00000	260	0	0
1016	0	0	262	280	2181109.00000	710012.00000	107.00000	262	0	0
1017	0	0	261	284	2181110.00000	709792.00000	108.00000	261	0	0
1018	0	0	413	280	2181109.00000	710232.00000	106.00000	413	0	0
1019	0	0	412	284	2181109.00000	710452.00000	105.00000	412	0	0
1020	0	0	410	293	2181109.00000	710892.00000	104.00000	410	0	0
1021	0	0	411	290	2181109.00000	710672.00000	104.00000	411	0	0
1022	0	0	408	298	2181109.00000	711332.00000	103.00000	408	0	0
1023	0	0	409	295	2181109.00000	711112.00000	103.00000	409	0	0
1024	0	0	407	300	2181108.00000	711552.00000	102.00000	407	0	0
1025	0	0	405	302	2181108.00000	711992.00000	102.00000	405	0	0
1026	0	0	406	300	2181108.00000	711772.00000	102.00000	406	0	0
1027	0	0	403	300	2181108.00000	712432.00000	101.00000	403	0	0
1028	0	0	404	302	2181108.00000	712212.00000	101.00000	404	0	0
1029	0	0	402	300	2181108.00000	712652.00000	100.00000	402	0	0
1030	0	0	401	298	2181108.00000	712872.00000	100.00000	401	0	0
1031	0	0	399	293	2181107.00000	713312.00000	98.00000	399	0	0
1032	0	0	400	295	2181107.00000	713092.00000	98.00000	400	0	0
1033	0	0	397	284	2181107.00000	713752.00000	103.00000	397	0	0
1034	0	0	398	290	2181107.00000	713532.00000	101.00000	398	0	0

Figure 2---9 A completed scan of the SEG---Y file. Any identified conflicts are listed in the Shot/Receiver tabs.

Finally, the **Basemap** tab displays a basemap of shots and receivers with default color scaled by elevation. It provides a quick visual check to see if the data looks reasonable (Figure 2---10).

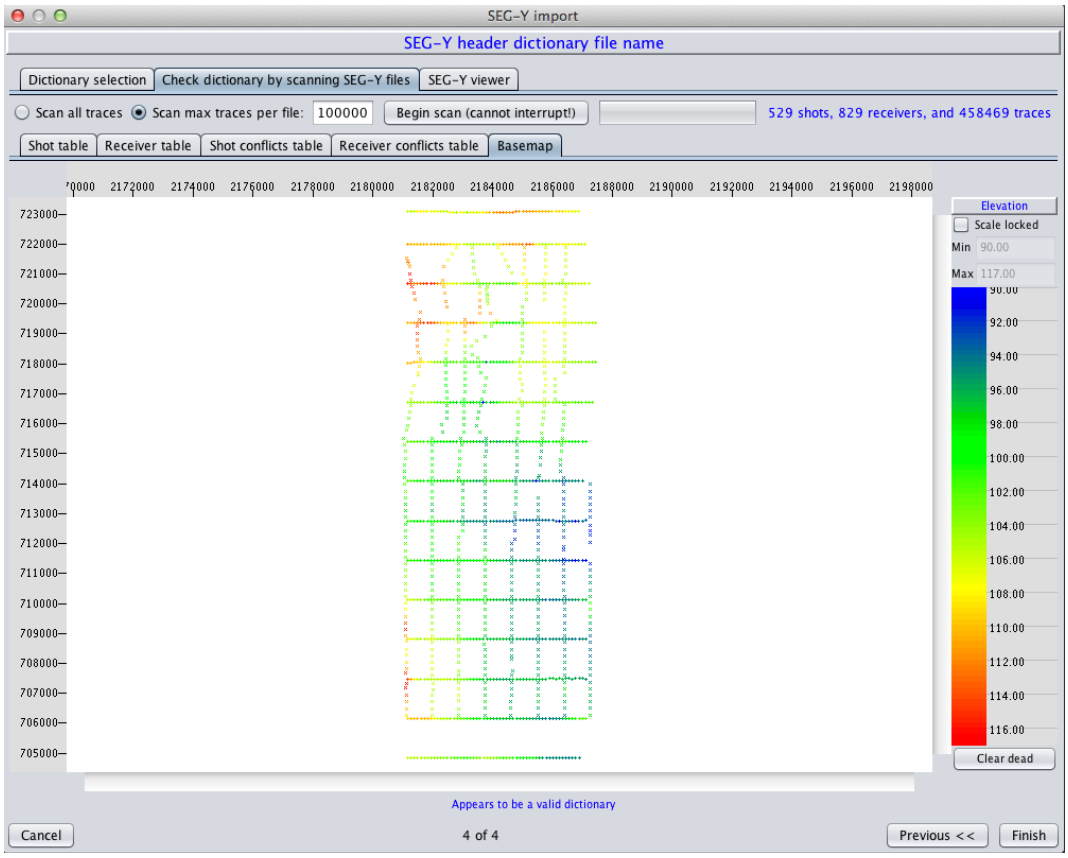


Figure 2---10 The Basemap tab can be used as another check on the integrity of the SEG---Y data.

At this point, if all the data looks satisfactory, click the **Finish** button at the bottom right corner of the panel and the project will be created. While Flatirons is building the project a window will open, displaying a progress bar and other status information. This information includes the current cumulative number of traces, shots and receivers; the number of geometry conflicts for both shots and receivers, the run time, free memory and the estimated time remaining to finish building the project (Figure 2---11). Be aware that an indication of zero geometry conflicts does not necessarily mean that none exist! When the process is complete, a brief image of the basemap is displayed and then the initial project window opens.

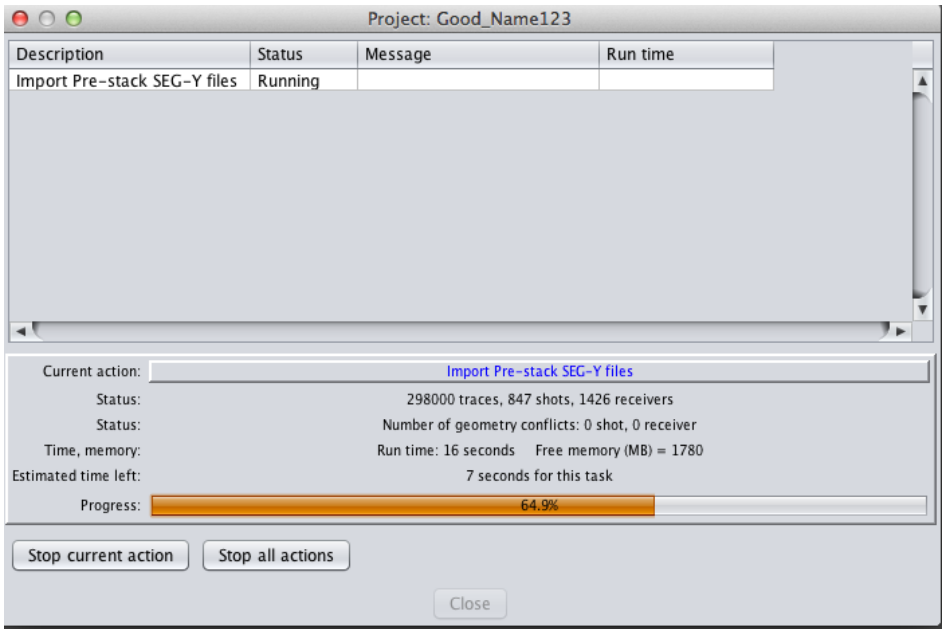


Figure 2---11 Popup window showing the progress of the creation of a project.

The initial project window defaults to the **Multimap** tab, shown in Figure 2---12. At this point the user has successfully created a Flatirons project and the data is ready for analysis. Subsequent chapters will cover the menus, features and functions in Flatirons to do this analysis.

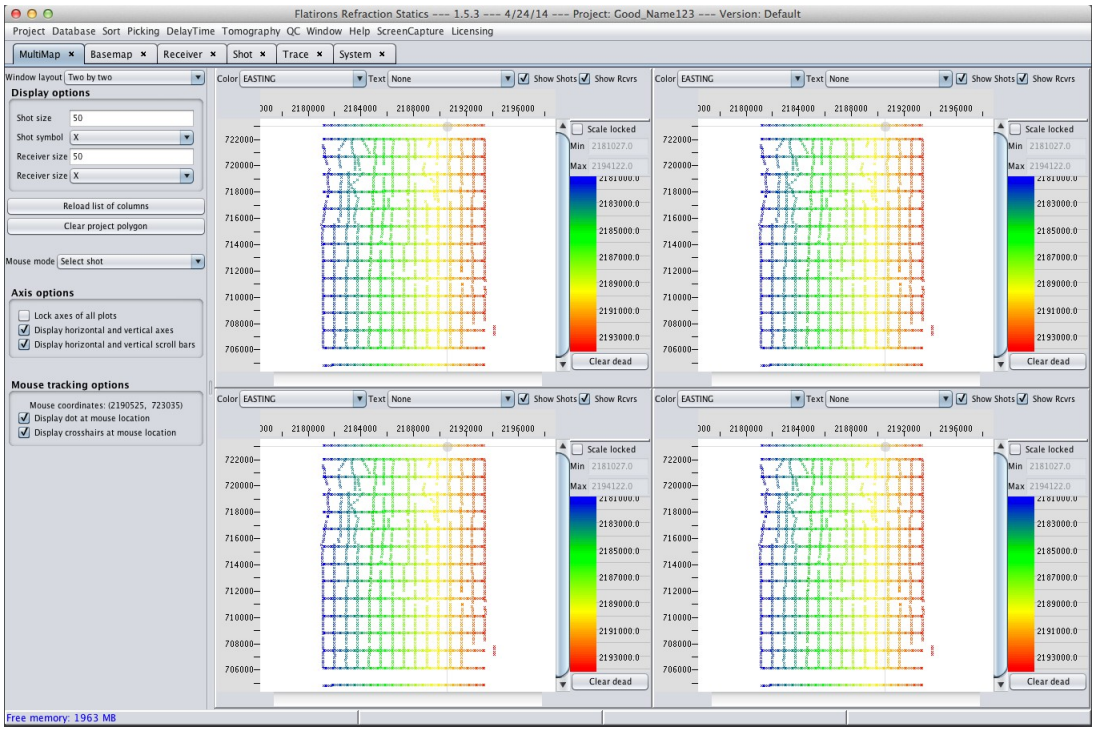


Figure 2---12 The initial window upon successful creation of a Flatirons project. The default view is the Multimap tab

3 Project

The **Project** menu has items for opening a project, a SEG---Y viewer or a JavaSeis viewer. The first two options open a separate window, to select project or file. The last option opens a tab window called **JavaSeis** for viewing a JavaSeis dataset. Note that opening a Flatirons project automatically closes the current project.

3.1 Open Project

Any pre---existing Flatirons projects can be opened from this menu item. It opens the **Select project** window shown in Figure 3---1. Each project is listed by row and each column lists information about the project. Not all the columns are shown in the figure. To open a project, click anywhere in the desired row and then click the **Open selected project** button. Prior to opening a project, the user can check **Load saved tabs**; this will open the project with the same tabs that the project had open when it was closed. Otherwise the project is opened with the default tab set. Generally, the state of Flatirons is saved whenever an operation writes to the database; this includes opening or closing tabs.

To switch between projects simply open a different project or create a new project. The current project is automatically saved. Only one project can be open at a time in a single Flatirons application. Closing a project is equivalent to closing Flatirons, using the standard window close button.

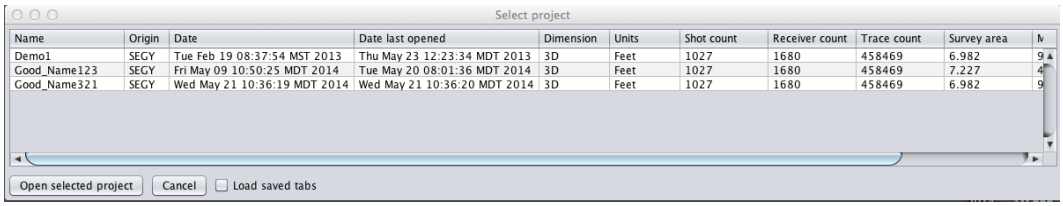


Figure 3---1 The 'Select project' window from the Open Project. All previously saved projects are listed.

3.2 Open SEG---Y Viewer/Dictionary Editor

This item opens the SEG---Y viewer and editor, as well as an assortment of other features. Opening the viewer brings up the window shown in Figure 3---2, with the default tab **File selection/dictionary editor**. The top left panel is used to open a SEG---Y file and view the file header. The top right panel is used to scan through the SEG---Y file, trace---by---trace and byte---by---byte. The bottom panel is used to open and edit an existing dictionary file.

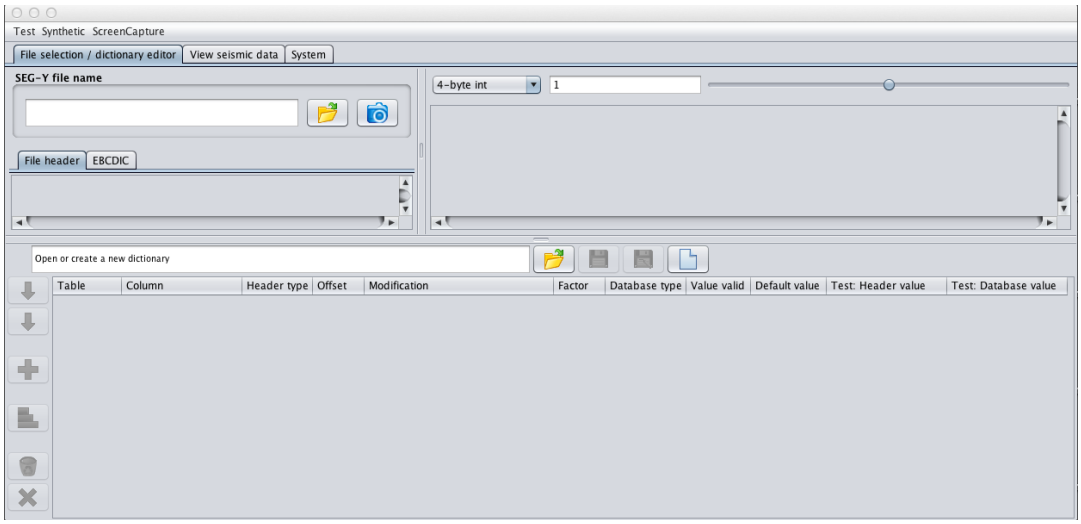


Figure 3---2 The default window in the SEG---Y Viewer/Dictionary editor.

Figure 3---3 shows the same window, after a SEG---Y file and dictionary file have been opened. The **EBCDIC** tab (**E**xtended **B**inary **C**oded **D**ecimal **I**nterchange **C**ode) permits the desired 8-bit encoding scheme to be selected. [Note: Canonical Geosciences LLC?] The snapshot icon ( ) allows the user to create a partial SEG--- Y file containing just the first 1,000 traces. The partial file has the same name as the full SEG---Y file but with the extension ".sgy_first".

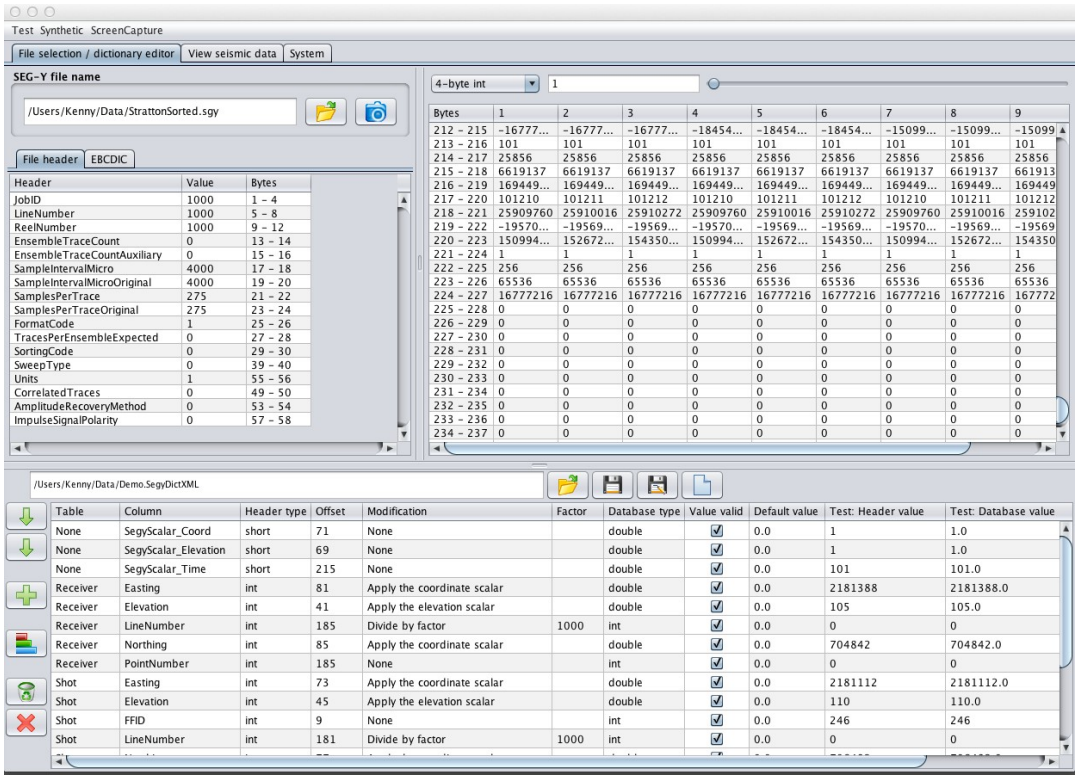


Figure 3---3 The SEG---Y Viewer/Dictionary editor after a SEG---Y file and dictionary have been opened.

The bottom panel allows the Dictionary file to be edited. The function of the icons on the left edge are: Upper down arrow ( ) – sort by Table/Column, lower down arrow ( ) – sort by Offset, plus sign ( ) – add an entry, post---stack map ( ) – the default post---stack map, trash ( ) – remove all entries, delete ( ) – remove selected entry; to select an entry just click anywhere on the desired row. Selection is indicated by highlighting of the row.

All columns can be edited, with the exception of the last two Test columns. Most columns have a pre---defined set of options, selected by a drop---down list that appears when a particular cell is clicked. For example, **Database type** can only be int, float or double. Other columns, such as **Offset** or **Default value**, can be changed with keyboard input. Once editing is complete the Dictionary file can be saved ( ) or saved to a new file ( ).

The **View seismic data** tab opens a modified version of the standard trace display window. The left panel has a trace display and text/graphical headers while the right panel presents the **Ensemble processing sequence** as shown in Figure 3---4. The purpose of this tab is to quickly review the traces to see if they appear reasonable. The Ensemble processing sequence is explained in greater detail in a later chapter (see Figure 6---6). In this context, various functions can be added to the sequence in order to modify the appearance of traces to aid in review.

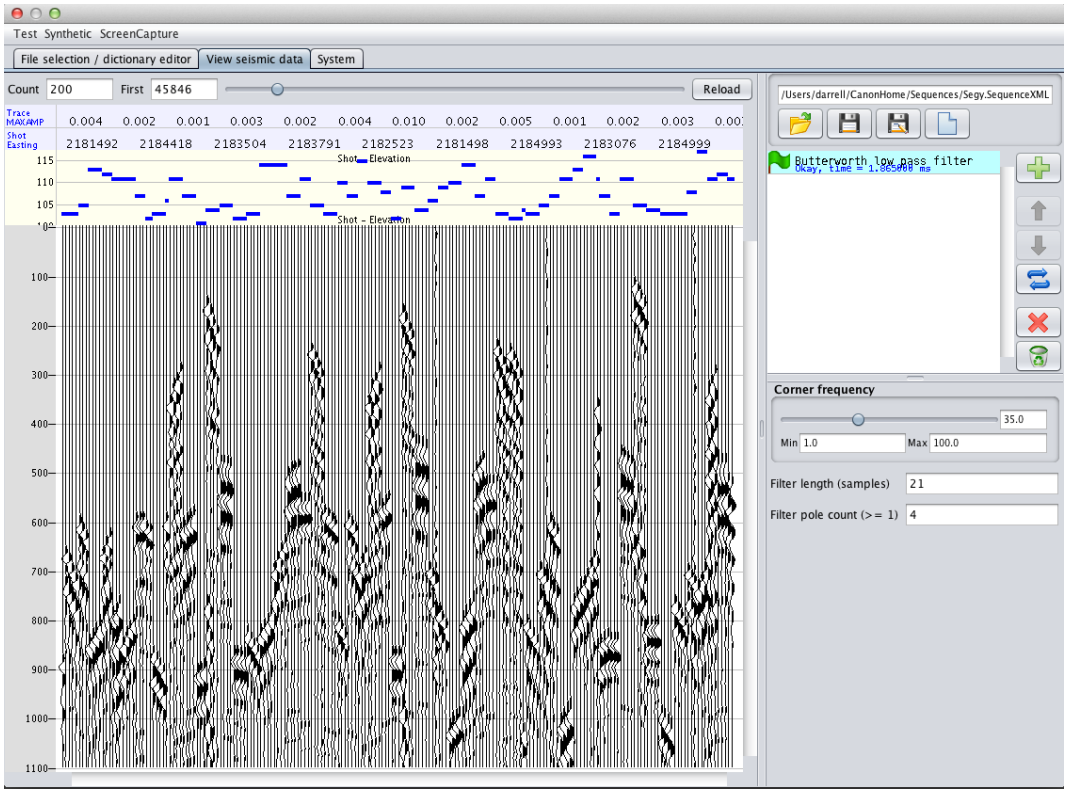


Figure 3---4 The View seismic data tab with trace display on the left and ensemble processing sequence widget on the right.

Two checkboxes allow for customizing the display. **Count** determines how many traces to show in the display. Any values below the minimum (10) or above the maximum (400) appear in red and won't be applied. **First** determines the trace number of the first trace in the set of **Count** traces to display, typically used for fine control of the display. The slider bar permits quick scanning of all traces, in chunks of **Count** traces. Finally, the **Reload** button reads the trace data from the SEG---Y file again.

The **System** tab is used to display information about the application and system. It has four tabs: **Application exceptions**, **Java system properties**, **Environmental variables** and **qc**. The Application exceptions tab list errors encountered by the application. By selecting an item on the list, a lower pane is populated with details of the exception, as shown in Figure 3---5.

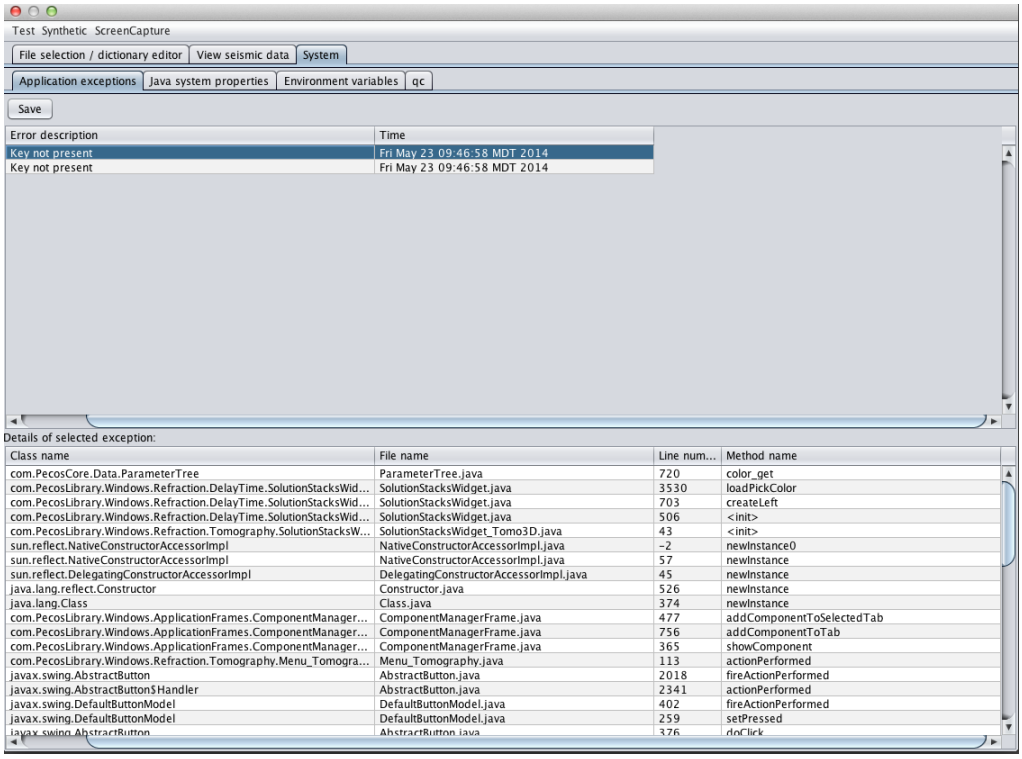
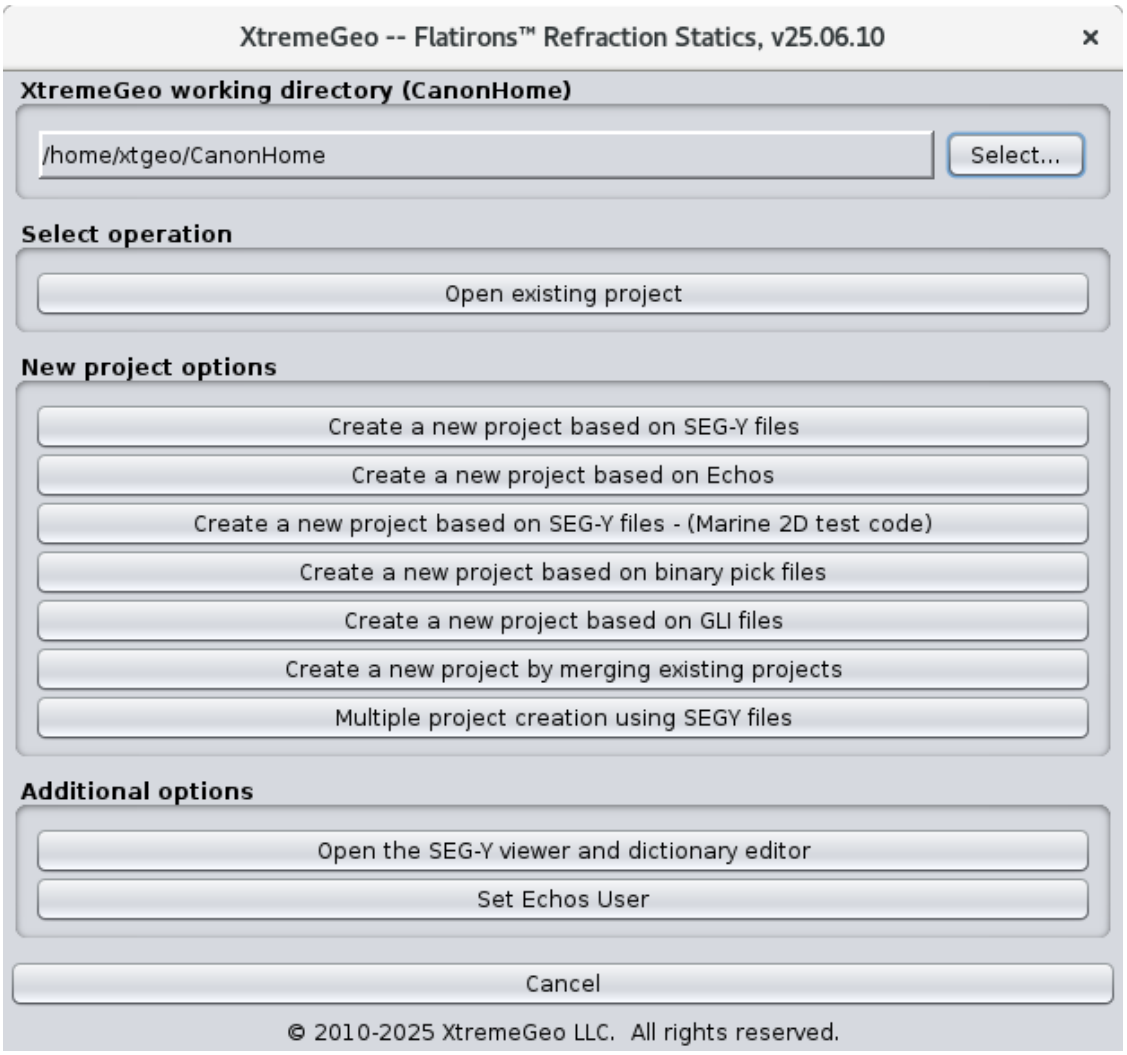


Figure 3---5 The Application exceptions tab in the SEG---Y viewer. Lower panel shows details of the selected exception.

The **Java system properties tab** (not shown) is a two---column table listing Property names and values related to the installed Java system and user. Approximately 50 names are listed, including class.path, library.path, vendor, operating system info and virtual machine info. The **Environmental variables** tab (not shown) is also a two---column table listing Property names and values related to the system. Examples include HOME, PATH, SHELL, USER.

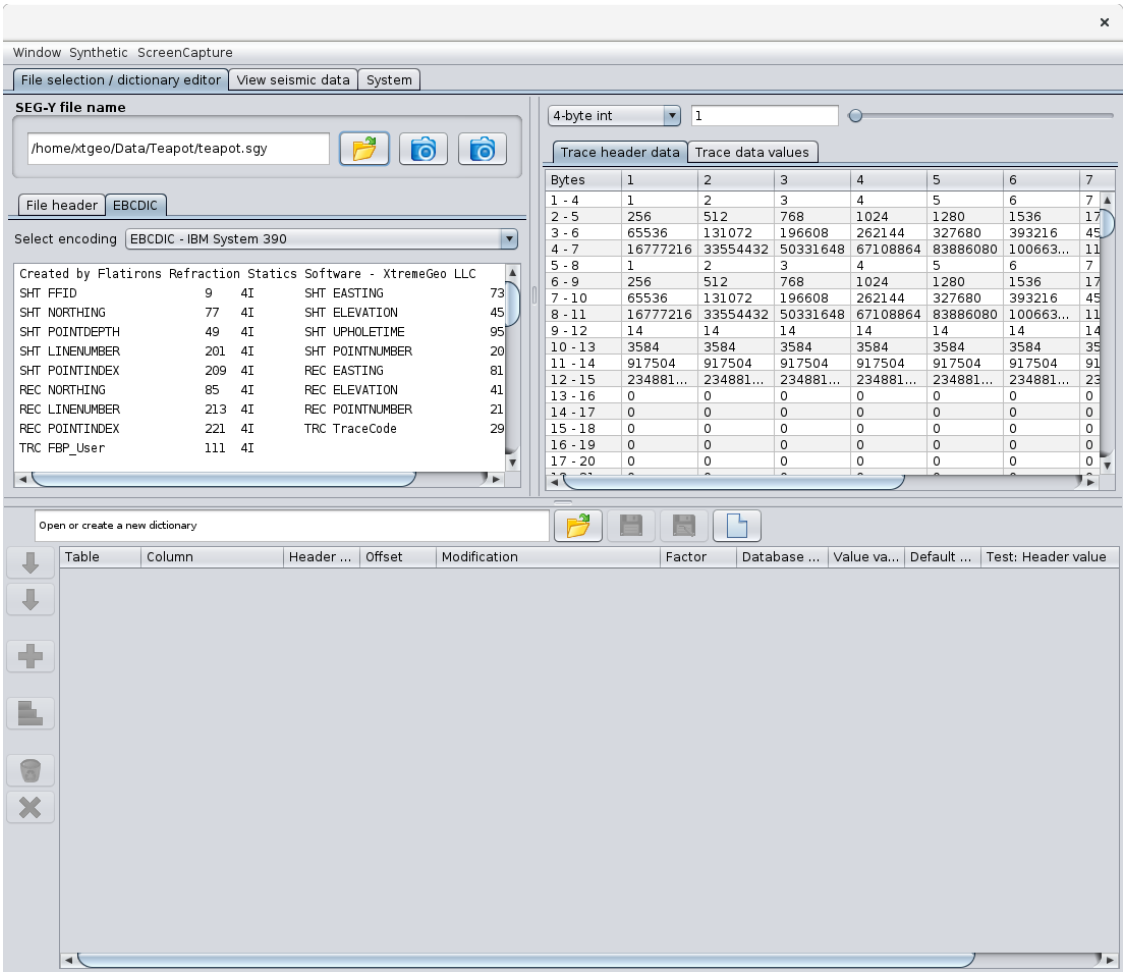
3.3 Segy header map definition

Select “Open the SEG-Y viewer and dictionary editor” on the main Flatirons display:



flatirons

When the dialog opens select a SEG-Y file. If the file is valid a number of displays will be populated, including information from the 400 byte file header and the 3200 byte EBCDIC header. The EBCDIC header may contain information about the location of header data:



Map 1

The upper-right displays raw header information. The header data may be interpreted in a number of formats (4 byte in, IEEE float, etc) to help find and QC header values.
Next create a new header map - click the white “New dictionary” icon. You will be prompted for a file name, and then the map will be populated with default entries:

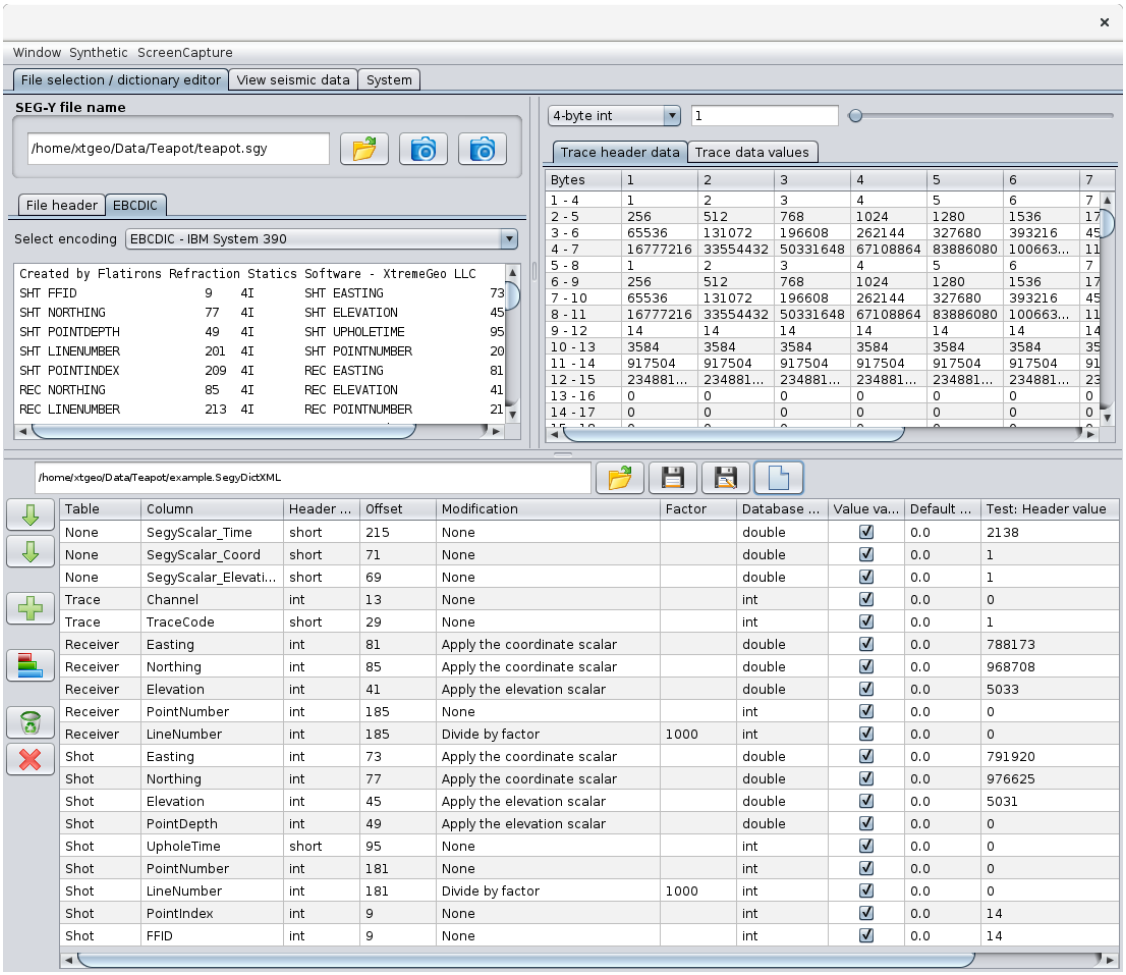


Table	Column	Header ...	Offset	Modification	Factor	Database ...	Value va...	Default ...	Test: Header value
None	SegyScalar_Time	short	215	None		double	☑	0.0	2138
None	SegyScalar_Coord	short	71	None		double	☑	0.0	1
None	SegyScalar_Elevati...	short	69	None		double	☑	0.0	1
Trace	Channel	int	13	None		int	☑	0.0	0
Trace	TraceCode	short	29	None		int	☑	0.0	1
Receiver	Easting	int	81	Apply the coordinate scalar		double	☑	0.0	788173
Receiver	Northing	int	85	Apply the coordinate scalar		double	☑	0.0	968708
Receiver	Elevation	int	41	Apply the elevation scalar		double	☑	0.0	5033
Receiver	PointNumber	int	185	None		int	☑	0.0	0
Receiver	LineNumber	int	185	Divide by factor	1000	int	☑	0.0	0
Shot	Easting	int	73	Apply the coordinate scalar		double	☑	0.0	791920
Shot	Northing	int	77	Apply the coordinate scalar		double	☑	0.0	976625
Shot	Elevation	int	45	Apply the elevation scalar		double	☑	0.0	5031
Shot	PointDepth	int	49	Apply the elevation scalar		double	☑	0.0	0
Shot	UpholeTime	short	95	None		int	☑	0.0	0
Shot	PointNumber	int	181	None		int	☑	0.0	0
Shot	LineNumber	int	181	Divide by factor	1000	int	☑	0.0	0
Shot	PointIndex	int	9	None		int	☑	0.0	14
Shot	FFID	int	9	None		int	☑	0.0	14

map2

Following the SEG SPS standard, shots and receivers are uniquely identified by a combination of (LineNumber, PointNumber, PointIndex). LineNumber and PointIndex may be omitted in PointNumber uniquely identifies that shot or receiver.
There are four types of "tables" in a dictionary:

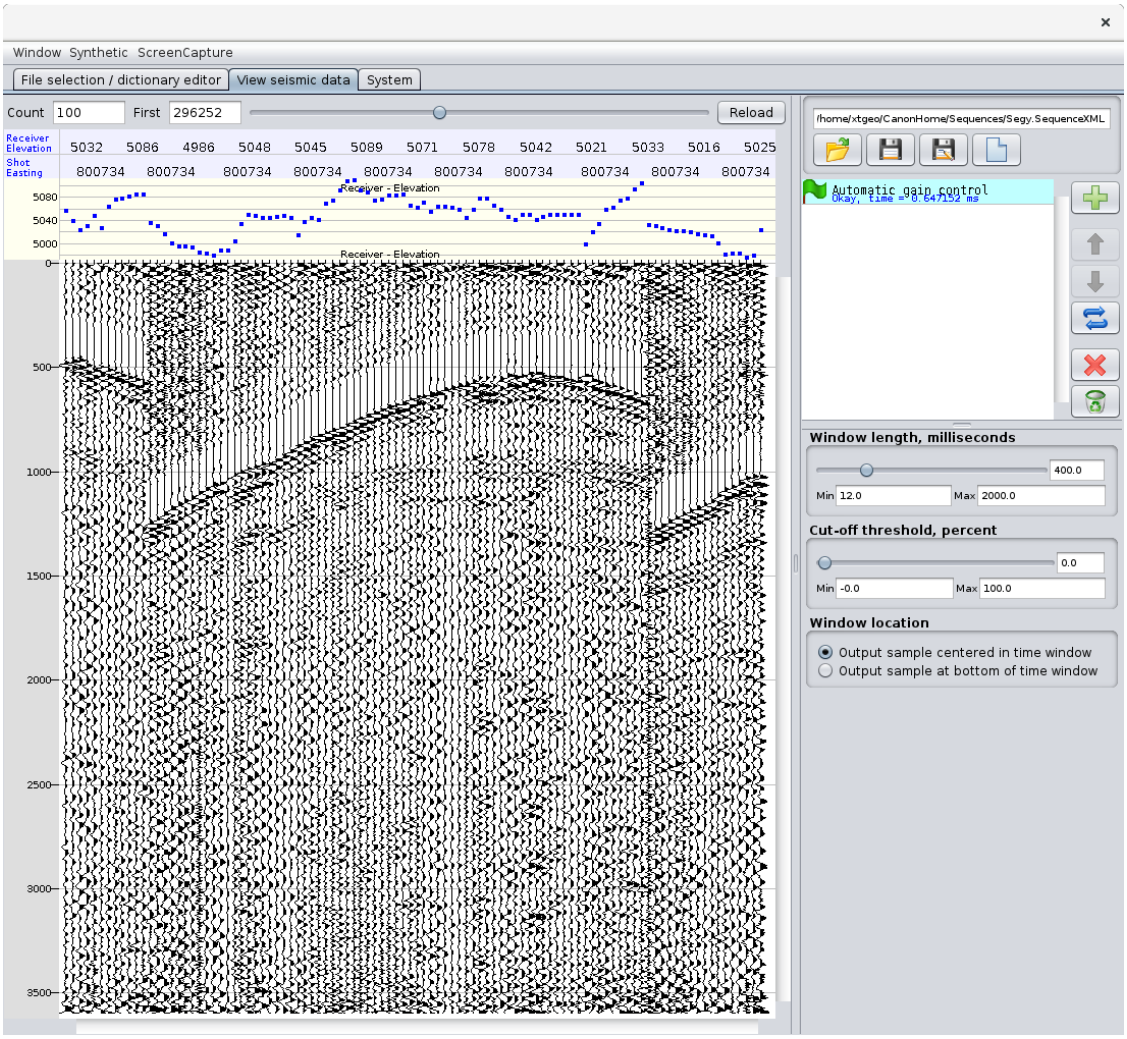
- None. This is used for things like elevation scalar information. These values are not imported into the database.
- Trace. Used for TraceCode, picks, channel, etc
- Shot.
- Receiver.

Here's a list of headers of interest to Flatirons. Only some of these are required.

Table	Column	
None	SegyScalar_Coord	Optional but recommnded. Scalar applied to coordinates
None	SegyScalar_Elev	Optional but recommnded. Scalar applied to elevations
Trace	Channel	Recommended but not required
Trace	TraceCode	Required. Flatirons requires TraceCode = 1
Shot	LineNumber	Optional. Recommended for 3D

Shot	PointNumber	Required
Shot	PointIndex	Optional. Very often FFID is used
Shot	Easting	Required. SegyScalar_Coord may need to be applied
Shot	Northing	Required. SegyScalar_Coord may need to be applied
Shot	Elevation	Required. SegyScalar_Elev may need to be applied
Shot	PointDepth	Optional. SegyScalar_Elev may need to be applied
Shot	WaterDepth	Optional. SegyScalar_Elev may need to be applied
Shot	UpholeTime	Optional
Receiver	LineNumber	Optional. Recommended for 3D
Receiver	PointNumber	Required
Receiver	PointIndex	Optional. Used for replants
Receiver	Easting	Required. SegyScalar_Coord may need to be applied
Receiver	Northing	Required. SegyScalar_Coord may need to be applied
Receiver	Elevation	Required. SegyScalar_Elev may need to be applied
Receiver	PointDepth	Optional. SegyScalar_Elev may need to be applied
Receiver	WaterDepth	Optional. SegyScalar_Elev may need to be applied
Receiver	UpholeTime	Optional, very rarely used

As you define the header be sure to check the “Test header value” on the far right.
Once the map has been defined it is a very good idea to check the map by viewing gathers. Select the “View seismic gathers” tab.



map3

You may need to click “Reload” for the header data to refresh.
Click on the header displays and view different database values to ensure that they match expectations.

3.4 Open JavaSeis Viewer

This menu item opens the **JavaSeis** tab window displayed in Figure 3---6. The purpose of the JavaSeis viewer is to ensure that the dataset has the proper format and/or that Flatirons can read the dataset correctly.

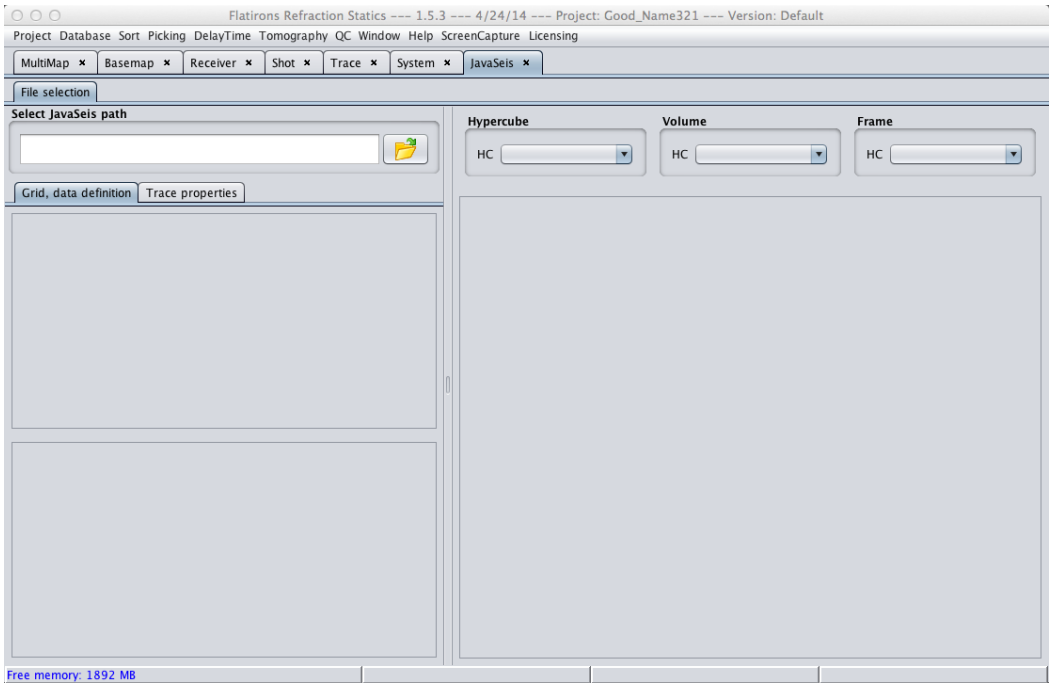


Figure 3---6 JavaSeis tab window.

3.5 Exit Flatirons

This menu item closes the Flatirons program. If a job is currently running this option is not available. Once the job is complete, the current project is automatically saved and this option becomes available.

4 Database

The Database menu item has a large number of sub---menu items as indicated in Figure 4---1, grouped by topic. Most, but not all, of the items are related to database functions. This chapter will proceed through these menu items, mostly in order and grouped by functionality, and describe the purpose or action of each item.

4.1 Refresh all data

The first item, **Refresh all data**, is deprecated and will be removed in later versions of **Flatirons.** Data no longer needs to be refreshed because that action is performed automatically, as needed. This refers to databases and files. Display plots often use drop---down lists that may need to be updated when the underlying database is updated. In this case a **Reload** button appears somewhere next to the plot.

4.2 Double---click time interval

The next two items allow the user to set and use their own preferred mouse double---click speed. The **Use custom double---click time interval** item toggles the status of the user---defined interval, with 'ON' status indicated with a checkmark next to the menu item. The **Select double---click time interval** item opens a dialog box (not shown) where the user enters a custom time, in milliseconds.

4.3 Open specific tab windows

The next six items open up separate tabs in the window:

1. **Pattern analysis**. Inactive. This item will open the **Patterns** tab, which can be used to identify pattern errors related to misidentified or mislabelled receivers.
2. **Basemap**. Opens the Basemap tab for viewing the survey in map view. This tab is covered in detail in Section 14.2.
3. **Multiple basemap display**. Opens the Multimap tab which is similar to the Basemap tab, except up to nine (in a 3x3 grid) basemaps can be displayed at once. This tab is covered in detail in Section 14.1.
4. **Trace table**. Opens the Trace tab, for viewing the trace table as well as performing related operations. This tab is covered in detail in Section 14.4
5. **Shot table**. Opens the Shot tab, for viewing selected Columns from the Shot table using SQL queries. Other functions include Column modification, export options, and error checking. This tab is covered in detail in Section 14.3.
6. **Receiver table**. Opens the Receiver tab. Essentially the same as the Shot tab, but for receivers. This tab is covered in detail in Section 14.3.

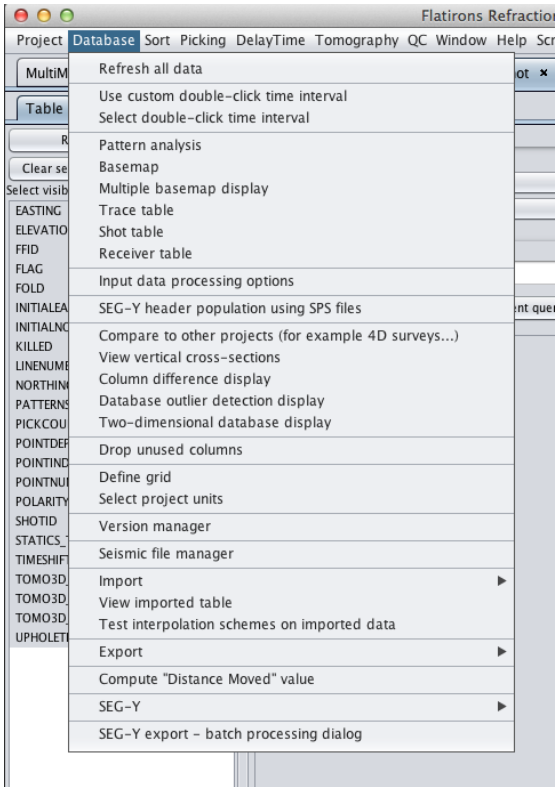


Figure 4---1 Sub---menu items for the Database menu item.

4.4 Processing options

The **Input data processing options** item opens a window (Figure 4---2) in which a processing sequence can be defined. There are also options for setting Offset and Azimuth ranges.

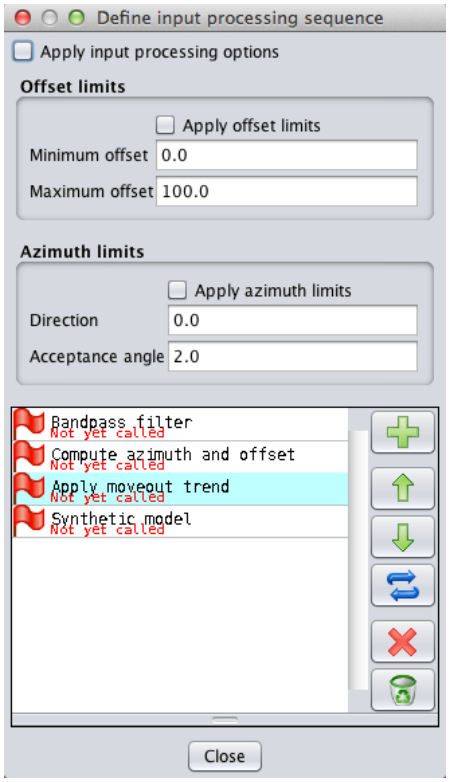


Figure 4---2 Pop---up window for the Input data processing options menu item.

4.5 SPS files

The **SEG---Y header population using SPS files** item opens the **Geom/SPS** tab shown in Figure 4---3. **SPS** refers to the Shell Processing Support format.

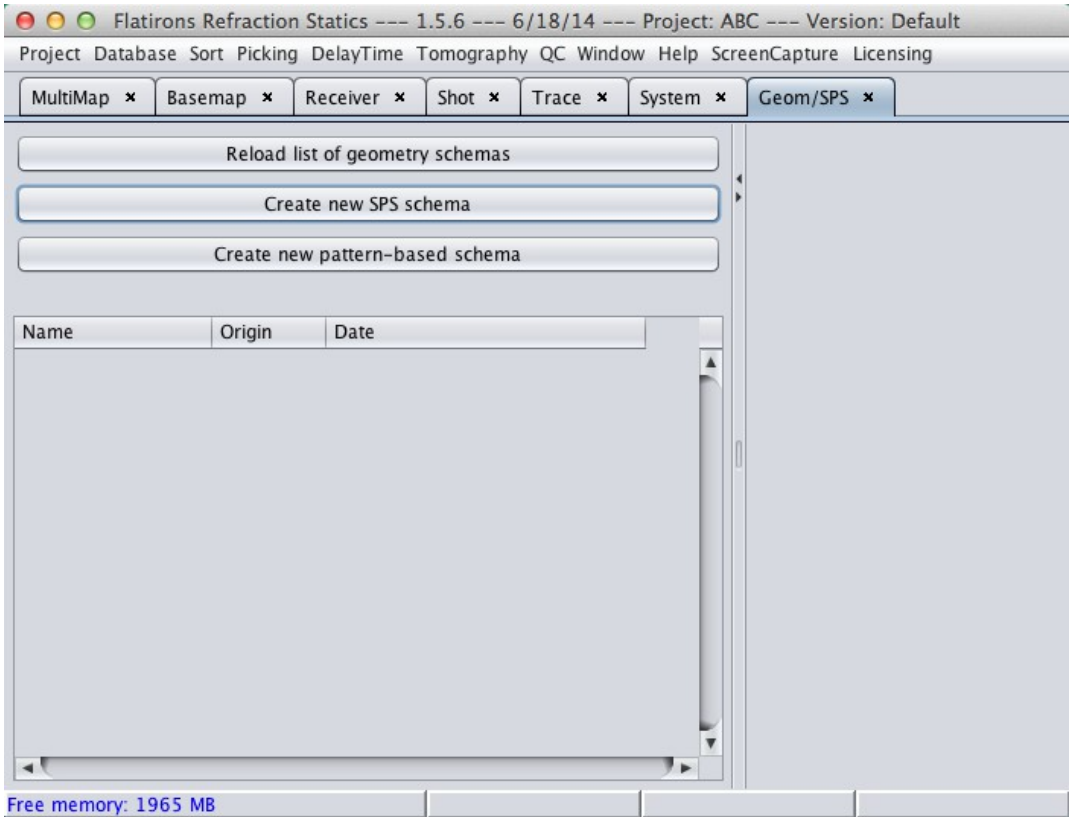


Figure 4---3 The Geom/SPS tab window.

Ff

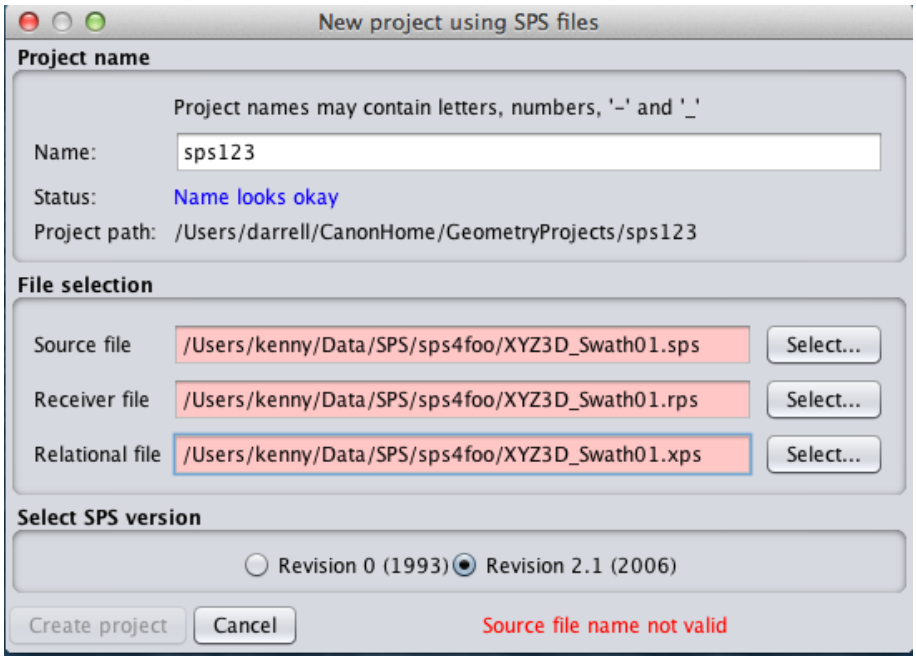


Figure 4---4 Dialog box from 'Create new SPS schema button in the Geom/SPS window.

The next group of five menu items relate to displays used for comparison of data in some way.

4.6 Compare to other project (for example 4D surveys)

The purpose of this item is to compare two projects. As noted previously, **Flatirons** does not permit two projects to be opened at the same time. But there are still few ways in which data from another project can be read and displayed. This is one of them. Selecting this menu item opens the **4D Comp** tab comprising the typical basemap view and selection options, as shown in Figure 4---5.

As the name implies this item is primarily intended to compare time---lapse or '4D' data sets. However, the difference between any two columns from separate projects can be plotted. For the current project, select the column from the **Compare with this column** drop---down list. For the other, unopened project, drop---down lists are used to select the desired project (the currently open project is not available), the version of that project and one Column to be used for comparison.

A textbox can be used to set a maximum allowed distance. This restricts the distance allowed between shot/receiver positions that are being compared. For example, in the figure, several shot positions are displayed as empty circles. This indicates that a shot in one survey was not within the maximum allowed distance (30 ft) of a corresponding shot in the other survey. As both projects are using the same survey, a few of the shot positions were altered in one project.

Checkboxes permit display of either shot or receiver positions, or both. There are buttons to reload the other projects or columns, as needed. Finally, whenever the Columns to be compared are changed, the **Reload plot** button needs to be clicked to update the basemap.

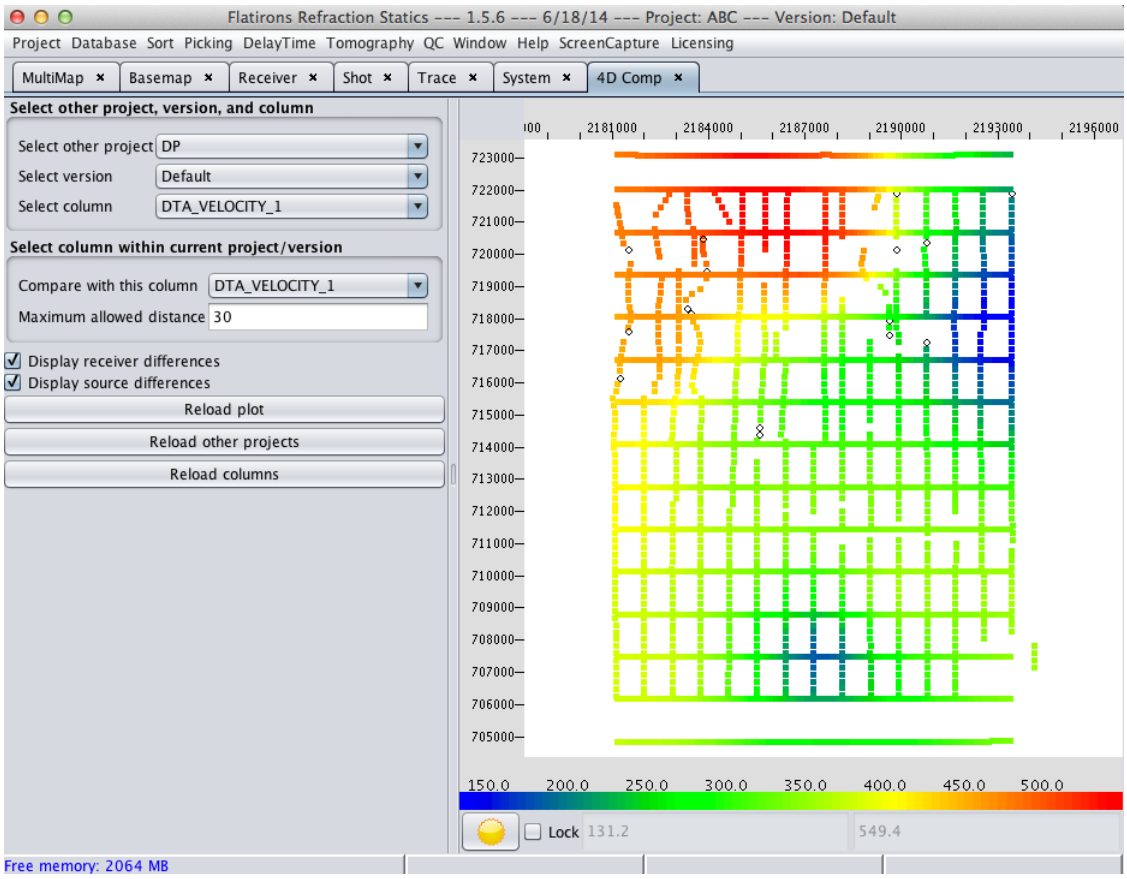


Figure 4---5 The 4D Comp tab for comparing specific data in different projects. Here, the difference between columns DTA_VELOCITY_1 in each project is displayed on a basemap.

4.7 View vertical cross sections

This item opens the **Profile**s tab in which multiple user---selected vertical cross sections of data can be displayed. Figure 4---6 displays the default window with a set of example profiles. Note the tab labels: **Profile #1** and **Profile #2**, demonstrating that multiple **Profile** tabs can be opened. The default window is set to have four plots visible but anywhere from 1---10 can be selected from a drop---down list.

To select what Column(s) to plot, click the left mouse button anywhere in the plot region and a pop---up list of available Columns will appear. Figure 4---6 shows the pop---list for the top plot, in which FOLD was selected. The selected Column is identified in the top left of the plot. As shown in the bottom plot, multiple Columns can be plotted; each plot is color---coded and labeled.

To clear a plot, display the pop---up list and select the first item – **Clear selected columns**. This will remove all Columns from that single plot. The unlocked ( ) and locked ( ) icons refer to the vertical scale of the plot and appear only after one Column has been selected. The default is unlocked, which means that whatever Column(s) are selected for the plot, the vertical scale will adjust to show all the data. However, if the scale is set to 'locked', then the scale is fixed to that of the first Column plotted. So it's possible that the second plot is not

visible at all. But the scale can be unlocked by clicking on the locked icon. The icon changes to unlocked and the scale automatically changes to show all data. (Clicking on the unlock icon has no effect – it will not change scale to hide data.) The same logic applies when adding a third Column, etc.

To select a profile use the basemap on the lower left of the window. Click, drag and release the left mouse button across the map. All defined plots will be populated immediately. Repeating the click---and---drag process generates plots of the same Columns for the new profile.

The **Plot options** section pertains to the basemap. The Column to display on the basemap is chosen from a drop---down list. The map can be black---and---white (No colors) or colored by the database value of the chosen Column. The standard color bar features can be used to adjust the colors. Note that the range for the basemap will not necessarily be identical to the same Column in a plot. For example, in Figure 4---6 the STATICS_TOMO3D was selected for both the basemap and the third plot. The basemap has a range (---15.9, 32.7) while the plot has a range of about (0, 6.1). Clearly, this is because the selected profile extends over a limited section of the basemap.

The **Interpolation options** section applies to the plots. In order to have continuous plots over the defined profile, interpolation is required. Two options are available, **Inverse square** and **Inverse fourth**. These refer to the weight given to nearby points – inversely proportional to the distance square or to the fourth power. So, inverse square more strongly weights nearby points. Toggle back and forth between the options to immediately see the effect they have on the plots. The **Interpolation radius** defines what are the nearby points. Generally, a larger radius results in a smoother curve. Choose a value and press Enter/Return to see the effect on the plots.

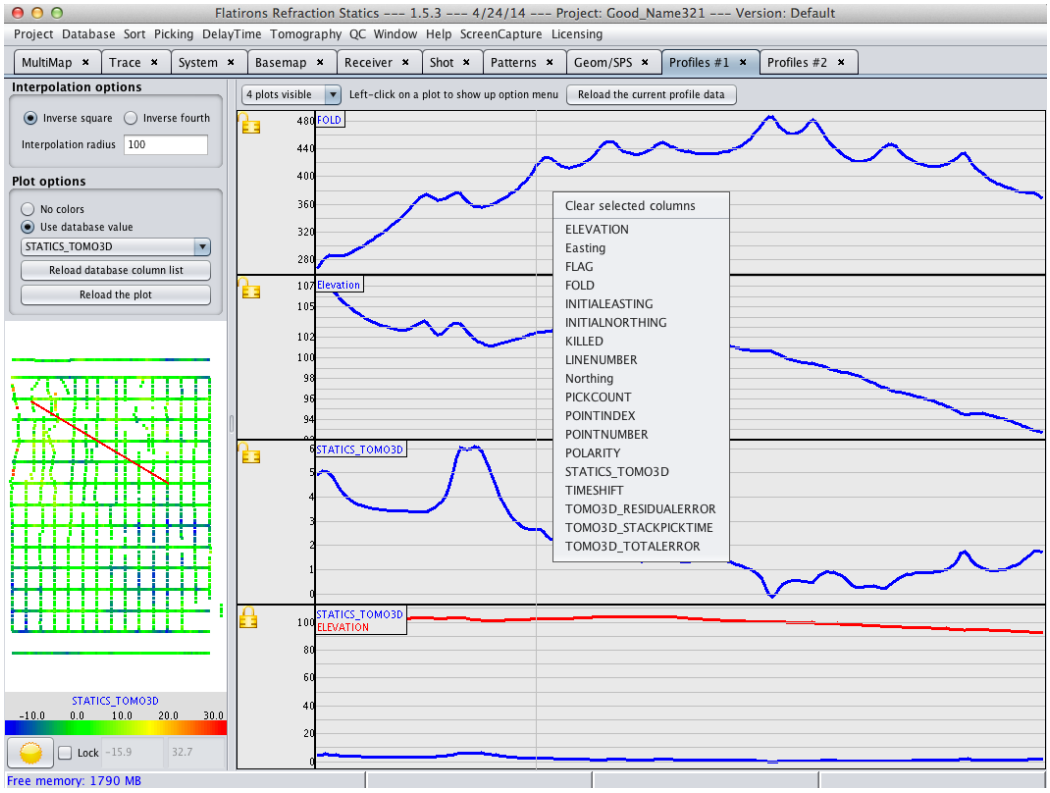


Figure 4---6 The Profiles tab available from the Database → View vertical cross sections menu.

4.8 Column difference display

This menu item opens up the **Diff plot** tab in which the difference between two columns is displayed on a basemap. Multiple **Diff plot** tabs can be open at the same time. The panel on the left in Figure 4---7 has drop---down lists for the first and second Columns to compare. The source of the Columns must be the same project, but different versions of the project can be selected. In the figure, the difference between two versions of statics is being displayed. The histogram in the lower left of the figure shows the same data as the basemap but is more convenient for showing the distribution of values.

The **Mouse mode** is a method of selecting an individual shot or receiver and obtaining more information. To use **Mouse mode** select a shot, first open a new Picker window (**Picking →New picker window – shot/receiver/bin plane**) and make sure shot plane is selected.
Then, on the basemap in the **Diff plot** tab select a specific shot. It may be helpful to zoom in first. Return to the Picker window and it will be populated with information for that specific shot. That same behavior applies to selecting a receiver.

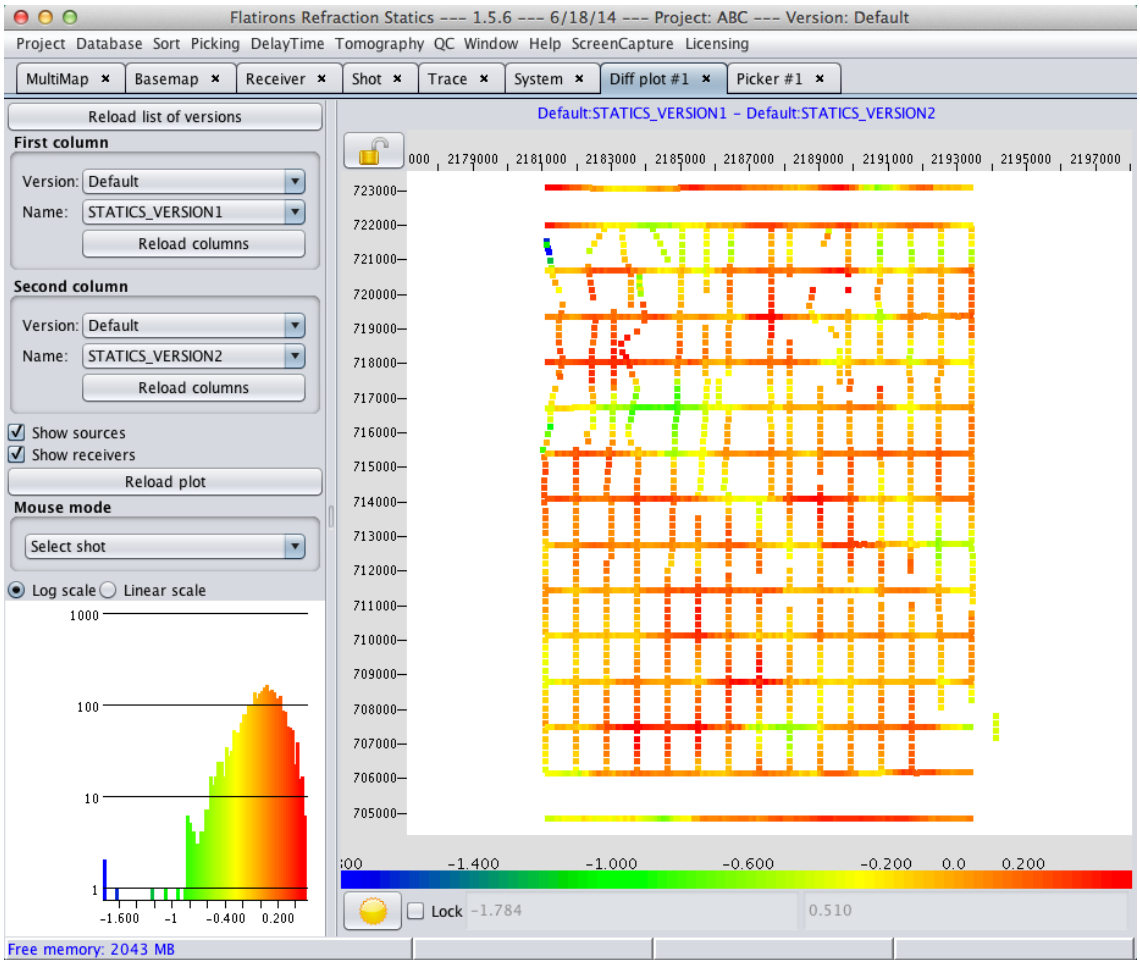


Figure 4---7 Difference plot tab from the Database → Column difference display menu item.

4.9 Database outlier detection display

This menu item opens the **Outliers** tab, shown in Figure 4---8. Multiple Outliers tab can be open at the same time. By displaying the difference between a columnar value and its smoothed value over a basemap, any outliers are readily apparent on the basemap display. In a project, the version and Column to investigate are selected from drop---down lists. Values for the **Smoothing radius** and a **Minimum valid value** are assigned in textboxes. The purpose of the **Minimum valid value** is to exclude, from the smoothed grid computation and display, of any values below the desired minimum. Often, before a Column value has been computed, **Flatirons** may assign it a large negative default value (i.e., ---9999) and the user may not want to include such default values. Column values at the Shot and/or Receiver locations can be used to generate the smoothed grid.

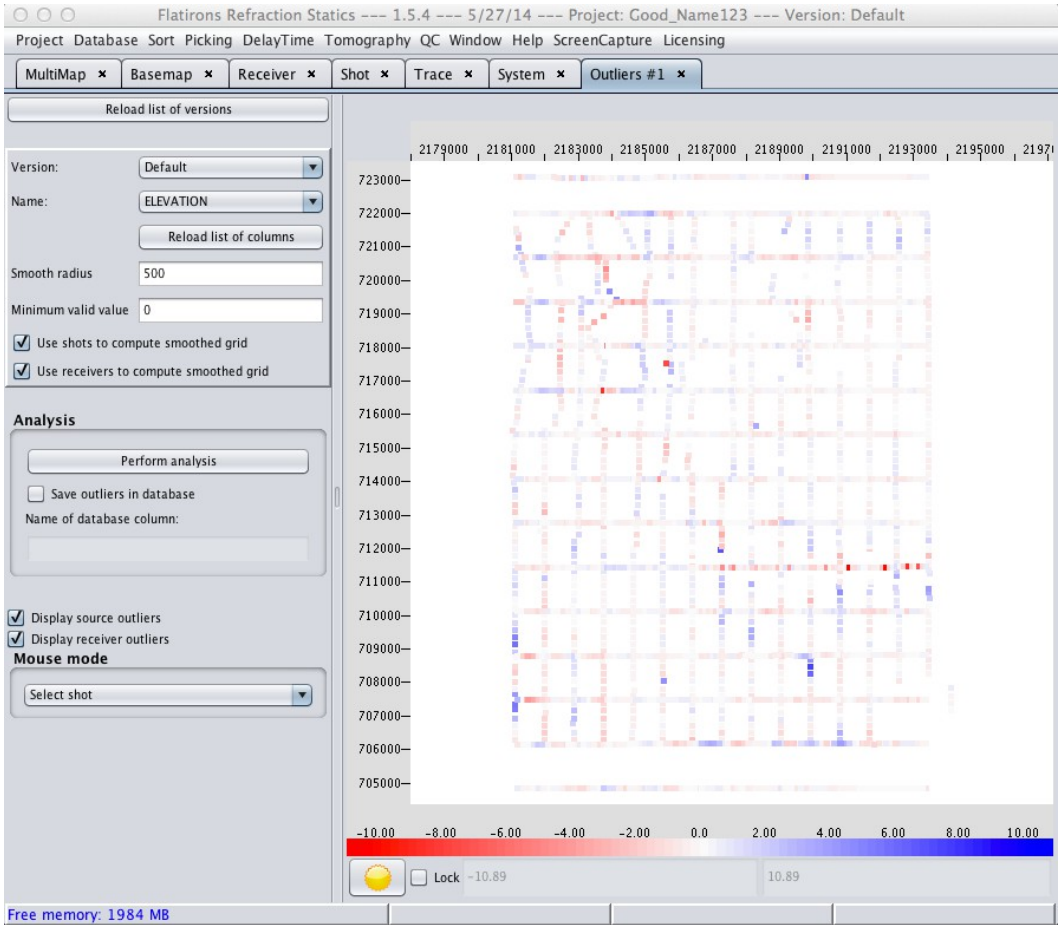


Figure 4---8 Outliers tab from the Database outliers detection display.

In the example in Figure 4---8, the chosen Column is Elevation. The outliers from the smoothed grid, created using both Shot and Receivers is about +/--- 11 feet. If the Minimum valid value had been set to 50, for example, the plot would be unchanged. This is because all Elevations are greater than 50, as can be readily verified by running a query for Elevation at the Shot and Receiver tabs.

In order to run the calculation for outliers and update the display, the **Perform analysis** has to be clicked. Also, the results of the calculation can be saved to a user---specified database column. The computed outlier value for all shots and receivers are saved.

To enhance the appearance of the outliers, the color bar can be used to change the color scheme and/or exclude the coloring of non---outlier values.

Finally, note the distinction between this outlier computation and **Interpolate to grid** in the **Basemap** tab. In the latter case, a Column value is interpolated, possibly with smoothing, between shots and receivers. For the outlier calculation, however, it is the Column value's distance 'above' and 'below' (in a 3D sense, along the Column axis) from the smoothed value at the shot and receiver location that is determined.

4.10 Two---dimensional database display

This menu item opens a **2D plots** tab. Multiple **2D plots** tabs can be open at the same time. The function of this tab is to simply plot Column X vs. Column Y. Up to five such 2D plots are available in a single tab window, with the restriction that each plot must have the same horizontal axis. Of course, to have multiple plots with a different horizontal axis, simply open another **2D plots** tab. Figure 4---9 shows an arbitrary example with four 2D plots. In the left panel **Display options** consist of Plot count, symbol size and symbols for shot and receiver. There is also a drop---down list to select the Column to be used for the horizontal axis on all plots. Also in the left panel is the **Select column** panel, where the vertical axis for each 2D plot is selected. For each plot there are checkboxes to toggle the display of symbols for source (SRC) and/or receiver (REC).

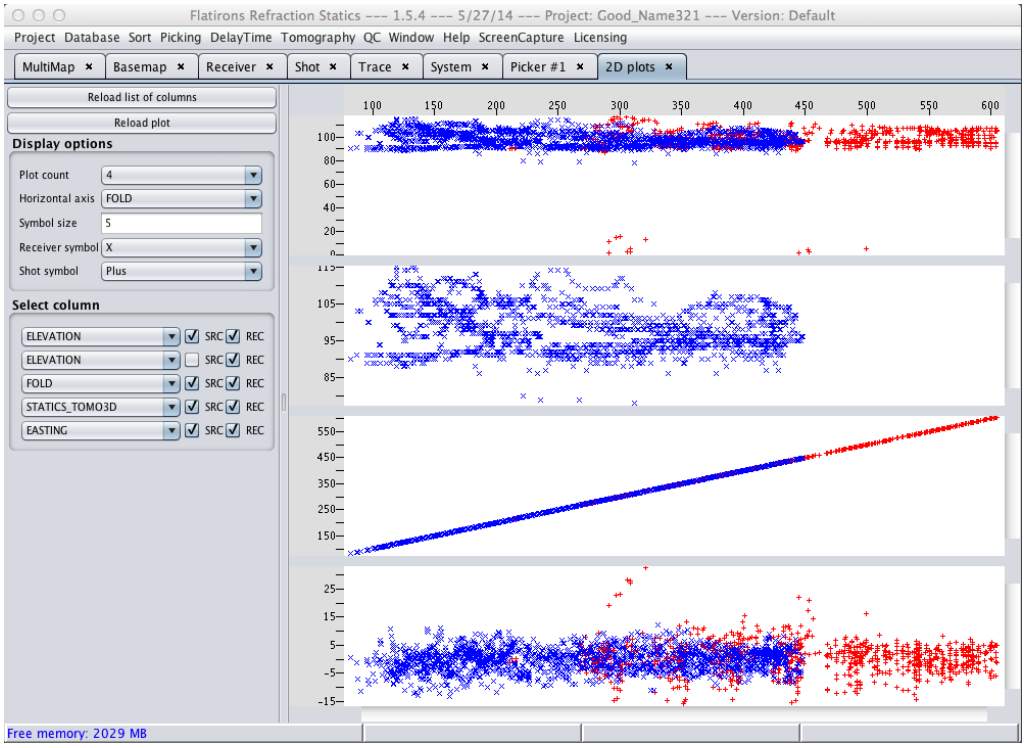


Figure 4---9 The 2D Plots tab, from the Two---dimensional database display menu item. Up to five plots can be displayed.

Plot 1, at the top, is Elevation vs. Fold. Plot 2 is the same, except only receiver symbols are plotted. Plot 3 is Fold vs. Fold, to verify the linear output. Because there are only four plots, the fifth Column, Easting, in the **Select column** panel, has no effect.

When any changes are made to either the **Display options** or **Select Column** sections, the **Reload plot** button must be clicked in order to update the 2D plots.

4.11 Drop unused columns

This menu item opens the pop---up window shown in Figure 4---10. From this window, one or more Columns can be deleted from the desired table(s). Multiple columns can be selected. For contiguous columns, click on a column, move the mouse to another column and Shift---click. The selected columns as well as all columns in between are highlighted for deletion. To select multiple non---contiguous columns, click while pressing the Control/Command key.

Once the desired columns have been selected, click on the appropriate button to delete them from the shot table, receiver table, or both tables.

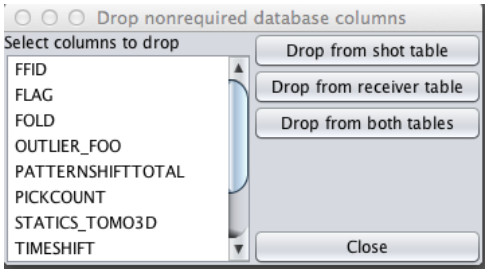


Figure 4---10 Pop---up window from the Drop unused columns menu item.

4.12 Define grid

This menu item permits changing the grid definition, including orientation and spacing. After selecting this menu item, a window is opened, displaying a simple basemap as shown in Figure 4---11(except initially without the dialog box). The first step is to define the azimuth by using the mouse to left---click and drag, generating a line about which the azimuth is measured. Releasing the mouse opens up a dialog box, as shown. Here the user sets the desired inline and crossline bin spacing.

When complete, click the **Apply changes** button. If the changes are unsatisfactory, simply repeat the process. For fine---tuning the azimuth, once the dialog box is opened, just type in the value for the desired Inline angle. After all changes are acceptable, click the **Apply** button at the bottom of the basemap display to complete the process.

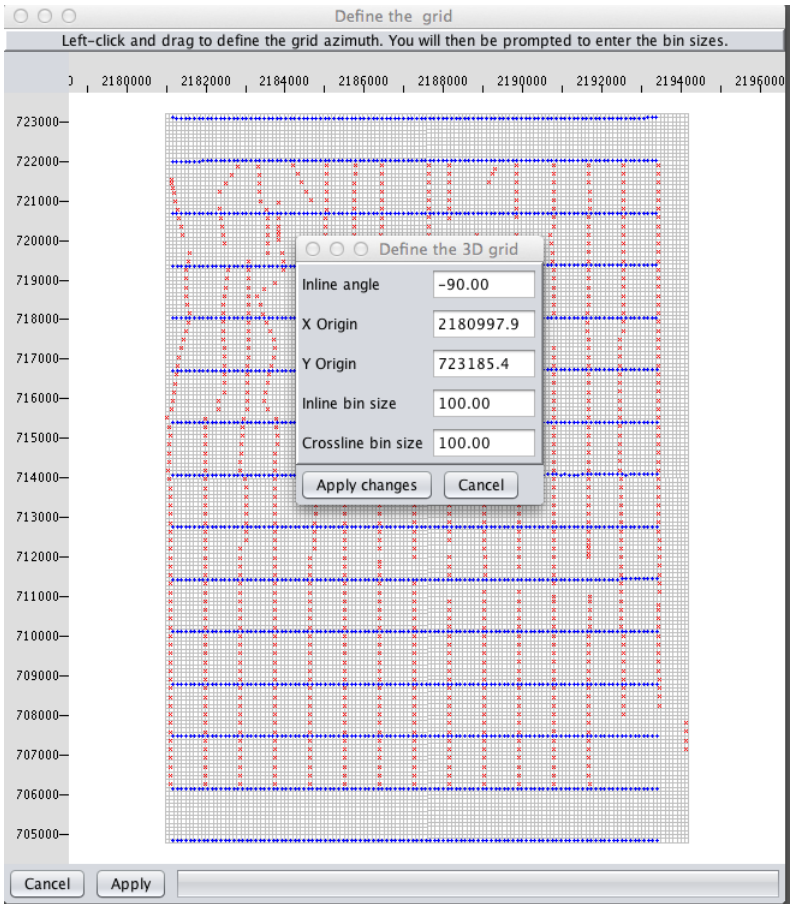


Figure 4---11 Grid definition window, opened with the Database → Define grid menu item.

4.13 Select project units

To set the desired unit of length for the project, use this menu item. From a pop---up dialog box (Figure 4---12) the user selects either **Meters** or **Feet**.

Figure 4---12 Dialog box for selecting project units, from the Database → Select project units menu item.

4.14 Version manager

This menu item is used to manage different version of the same project. Differing versions use the same trace table but effectively everything else is separate. Just as only one project can be open at a time, only a single version can be open at a time. (As mentioned previously, saved data from another project/version can be read and displayed in some windows.) The associated dialog box is shown in Figure 4---13. In this example, version Ver2_12---25---2000 is

currently open, as indicated, and clicking on the **Open selected version** button will open the Default version.

Figure 4---13 Dialog box for the Database → Version manager menu item. Used to open an existing version or create a new one.

To create a new version of the current project, click the **Create a new version** button. This will bring up the message box shown on the left in Figure 4---14. Clicking the **OK** button will

Figure 4---14 Message box and dialog box when creating a new version of the current project.

open the dialog box on the right, where the user enters a new version name. For easy reference, the name of the current version is displayed at the top of the Flatirons application window, just to the right of the project name.

Note: Deleting a version of a project requires removing its directory, either from a File Manager or command line. Generally, the path to this folder is

username/CanonHome/RefractionProjects/ProjectName/Version/VersionName

Where, of course, ProjectName is name of the project and VersionName is the name of the version. To delete the version, the VersionName folder should be deleted. Similarly, to delete a project, the ProjectName folder should be deleted. This will delete all versions of the project.

Seismic file manager. Occasionally the path to a seismic file (SEG---Y) may be altered. This menu item simply opens a window to let the user locate the file and re---associate the correct

path for Flatirons. As Figure 4---15 indicates, separate buttons allow the path to be set for all files or just the selected files.

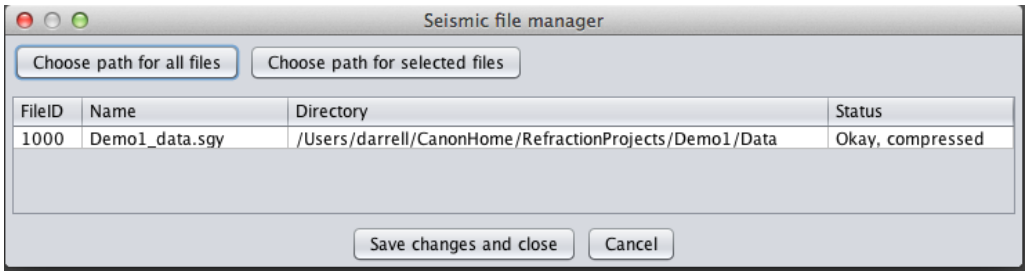


Figure 4---15 Seismic file manager window from the Database menu.

The next three Database menu items relate to importing ASCII data.

From the **Import** item there are two options: **ASCII data table** and **Open ASCII import window**. The first option opens a dialog box (Figure 4---16) to select the ASCII file.

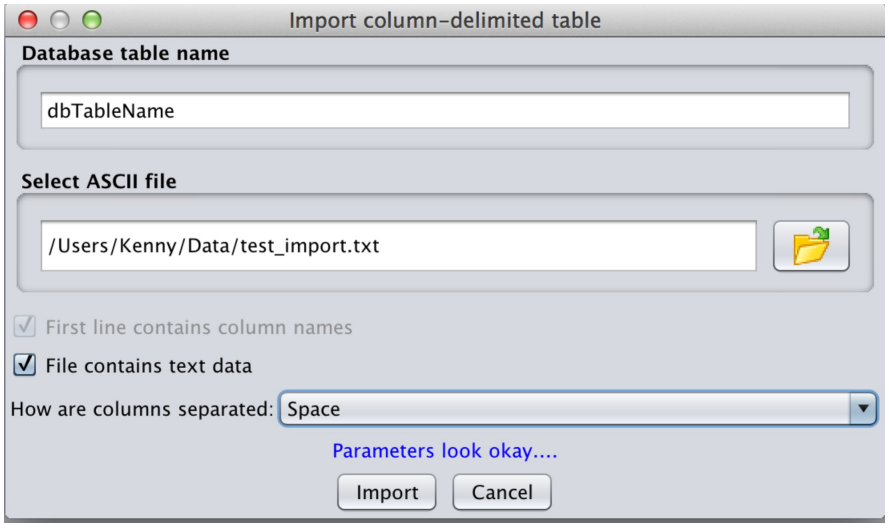


Figure 4---16 Database → Import data → Import ASCII data table.

The second option opens a tab window called **ASCII #1** with two panels as shown in Figure 4---17. The right panel is used to select the ASCII file and view the data. From the **ASCII file selection** tab, locate the ASCII file an import it. The display window will be populated with the ASCII data, in columnar format. The left panel is used to open the column definition file name, if it exists. Once opened, it can be edited and saved.

If the Column definition file does not exist, one must be created. If the data is in an acceptable format the file can be imported and saved to a database. Using the icons, a new entry (row) can be added or removed. Each entry requires a name, type and From/To fields to define the width of the data type. Each cell in a row is manually edited to match the data.

Alternatively, clicking on the first column of a entry in the display window will set both the From and To cells to the selected column and only the To field has to be manually entered. Once the From/To cells are set, the columns for that entry are automatically highlighted in yellow.

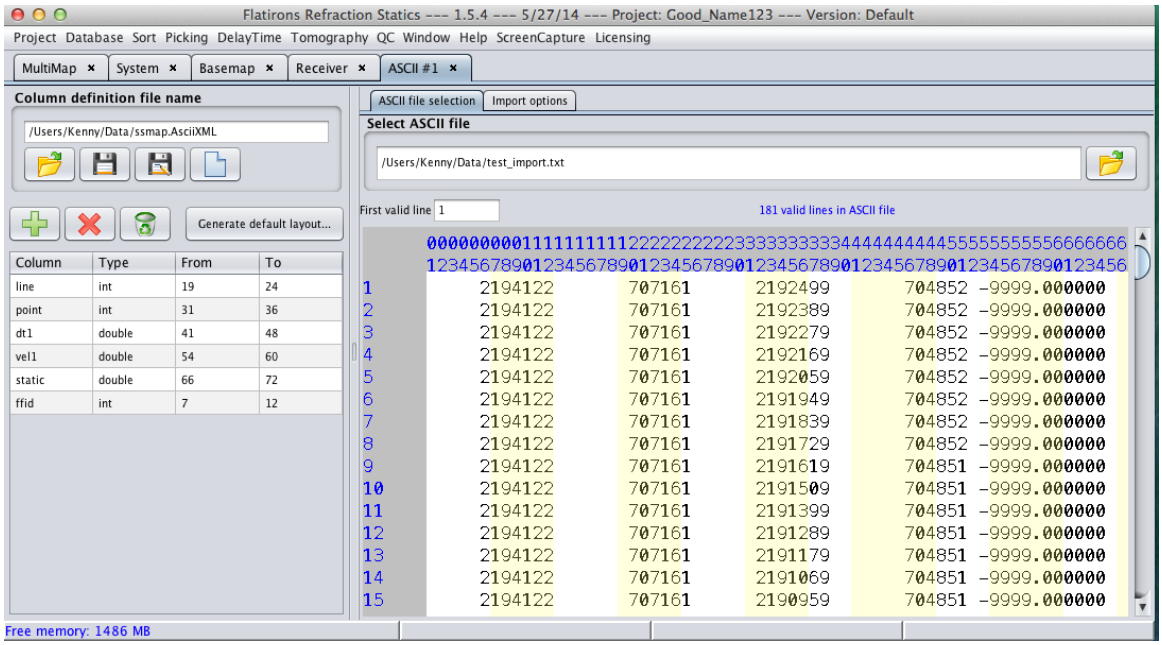
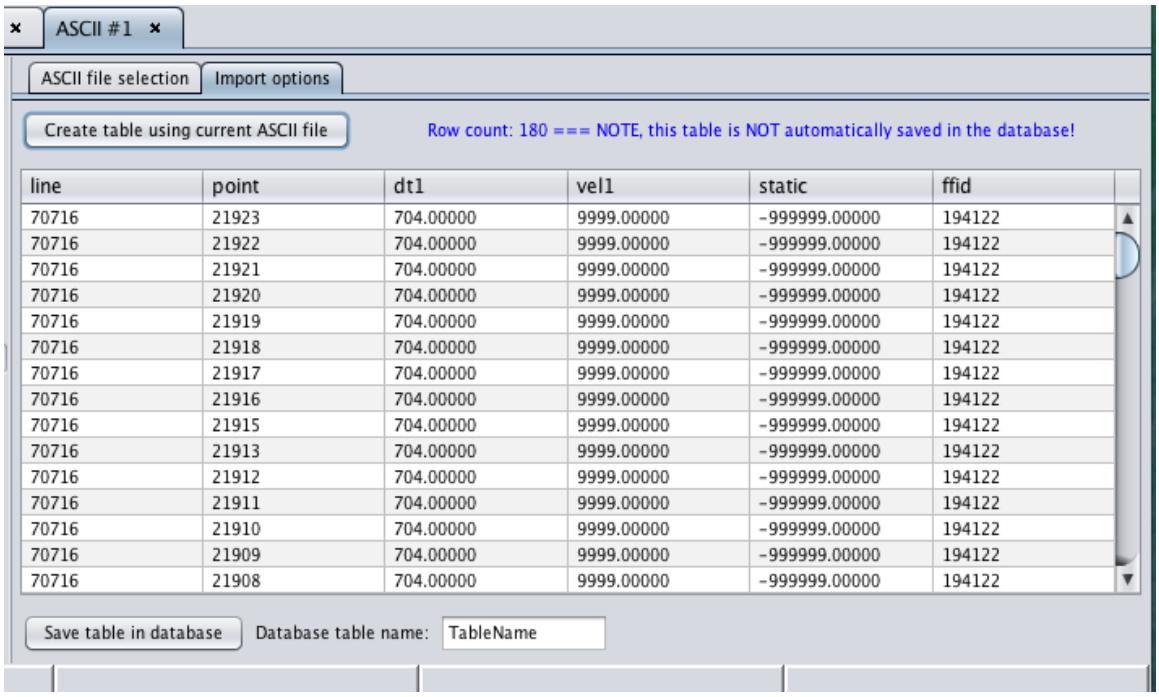


Figure 4---17 The tab window opened from the Database → Import → Open ASCII import window menu.

Figure 4---18 shows just the **Import options** tab in the right panel of the ASCII import window. At the top of the panel is a button to create a table using the ASCII data. This is not sufficient to actually save the data to a database. If the newly generated table appears okay, a button at the bottom of the panel is used to save the table to a database. The name of the table is specified in a textbox.



line	point	dt1	vel1	static	ffid
70716	21923	704.00000	9999.00000	-999999.00000	194122
70716	21922	704.00000	9999.00000	-999999.00000	194122
70716	21921	704.00000	9999.00000	-999999.00000	194122
70716	21920	704.00000	9999.00000	-999999.00000	194122
70716	21919	704.00000	9999.00000	-999999.00000	194122
70716	21918	704.00000	9999.00000	-999999.00000	194122
70716	21917	704.00000	9999.00000	-999999.00000	194122
70716	21916	704.00000	9999.00000	-999999.00000	194122
70716	21915	704.00000	9999.00000	-999999.00000	194122
70716	21913	704.00000	9999.00000	-999999.00000	194122
70716	21912	704.00000	9999.00000	-999999.00000	194122
70716	21911	704.00000	9999.00000	-999999.00000	194122
70716	21910	704.00000	9999.00000	-999999.00000	194122
70716	21909	704.00000	9999.00000	-999999.00000	194122
70716	21908	704.00000	9999.00000	-999999.00000	194122

Figure 4---18 The Import options tab in the ASCII import window.

Database → View imported table opens a dialog box (not shown) that allows the user to select the desired table from a list. After the selection a tab window opens, with the tab name the same as the table name. Figure 4---19 shows an example window for importing the IMPORTCOL table. The description of the functionality of this window is deferred to Section 14.3, which deals with operations on the Shot/Receiver tables. To close this tab window, make sure it is the active tab and simply click on the **Window → Close window** menu item.

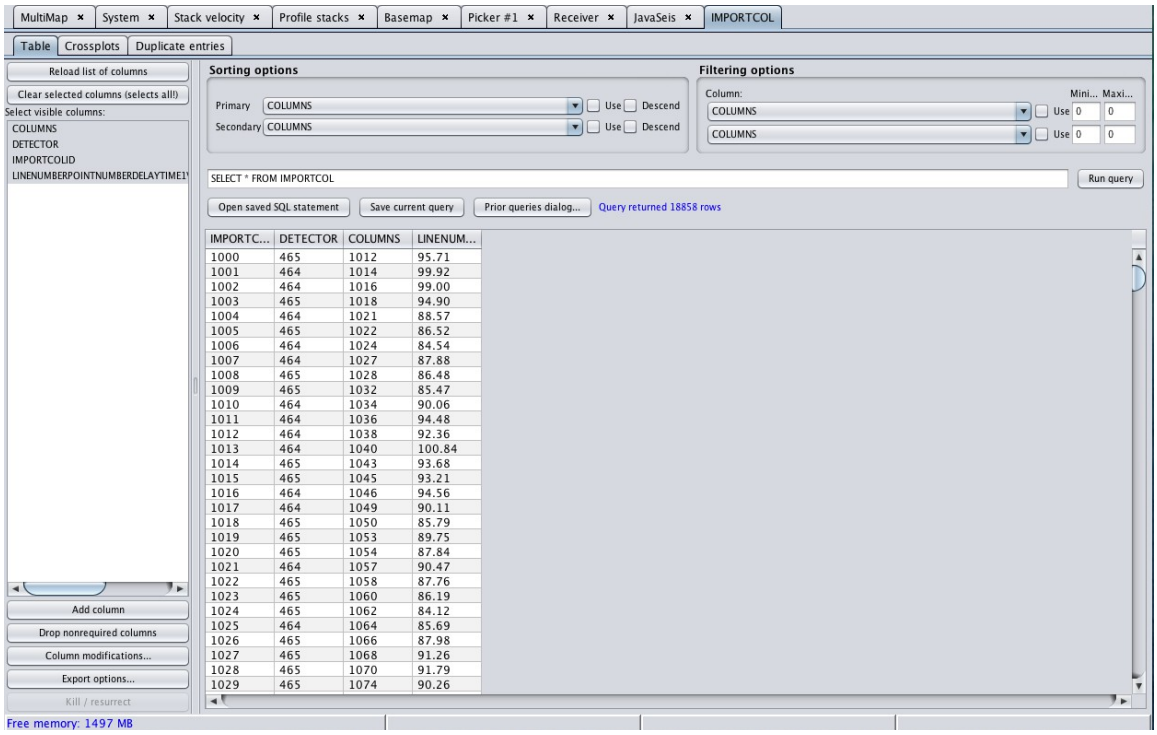


Figure 4---19 The tab window for the IMPORTCOL table. From the Import ASCCII data table menu.

The last of the import options is **Database → Test interpolation schemes on imported data**. Selecting this option opens the **Interpolation test** tab window. There are two panels in the window, the left is for selecting the data and operations and the right is for displaying the interpolated results. See Figure 4---20 for an example In the left panel a drop---down list is used to select the imported table. Also in the left panel, a Selected axes section has drop---down lists to select the desired column for each of the three axes. The grid size to use in the interpolation process is entered in a textbox. Generally, the smaller the grid size, the smoother the interpolation and the longer it takes.

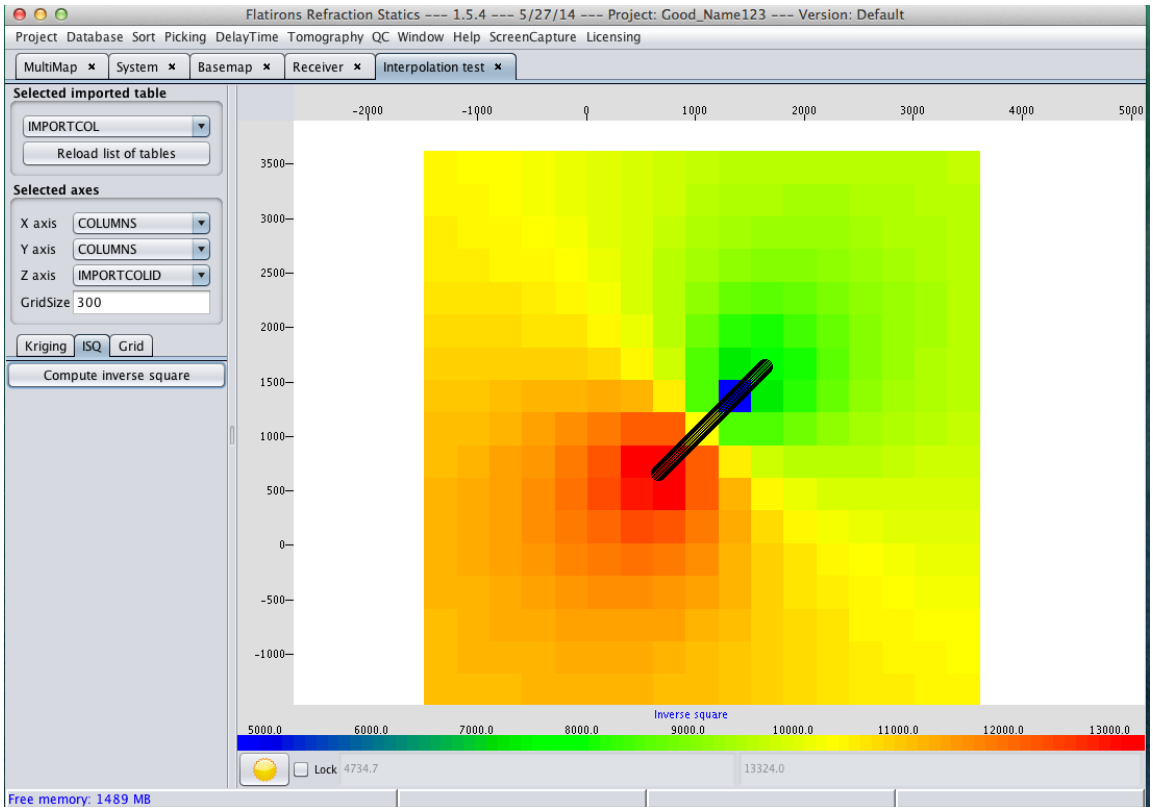


Figure 4---20 The Interpolation test tab.

Finally, three tabs are used to select the interpolation method. The **Kriging** tab (not shown) has textboxes for constants **C0** and **C1** and the **Decay distance**. Once these are specified the user clicks the Krige button to begin the computation. The button will maintain the 'pressed' appearance (from white to gray) until the process is complete. The Inverse Square tab (**ISQ**) has a single button to perform the interpolation using an inverse square weighting. Finally, the **Grid** tab (not shown) also has a single button, to perform the interpolation using a grid---base smoother.

Once the interpolation is complete the result is plotted in the right panel. The axes are labeled according to the specified axis and the type of interpolation is stated beneath the image. The standard color bar is available but zooming is not.

From the **Export** item there are two options: **Export geometry changes for this project** and **Export geometry changes for all projects**. The first option creates two ASCII files containing information about shots that have been moved or killed. The names of the files are ProjectName_shot_moves.txt and ProjectName_shot_kills.txt, respectively. The format of both files is column---delimited columns containing identifying information, distance moved, and killed status. The second option is the same, except the process is repeated for all projects. Figure 4---21 shows the dialog box for selecting the path of the ASCII file destination for either option.

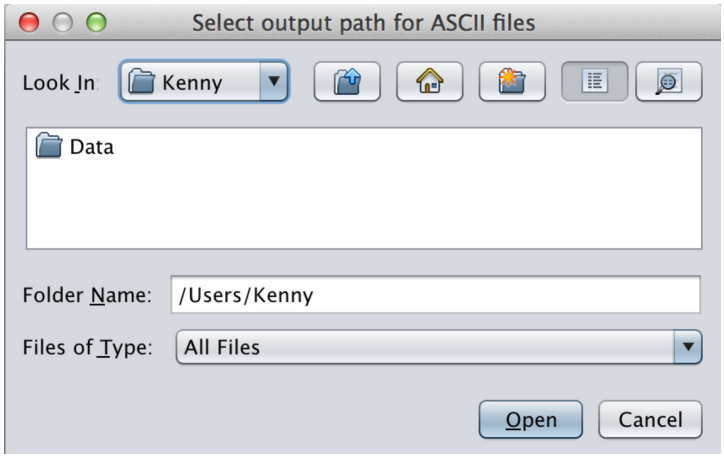


Figure 4---21 Dialog box for exporting geometry changes to ASCII files.

5 Sort

The **Sort** menu item permits creation of sorted bins or receivers or sparse CMP gathers for use elsewhere in the application. Usually it is preferred to generate sparse CMP gathers when creating a project but if that step was skipped this is the place to do it. Or, if different settings for the sparse CMPs are desired it can be done in this menu. Figure 5---1 is the pop---up window where CMP spacing, bin radius and offset range are specified. Selecting **Create bin sort** or **Create receiver sort** opens the **Job** tab that shows a progress bar and current status. Selecting **Remove bin sort** or **Remove receiver sort** simply deletes the corresponding folder where the sorted data was stored. It does not otherwise affect the project.

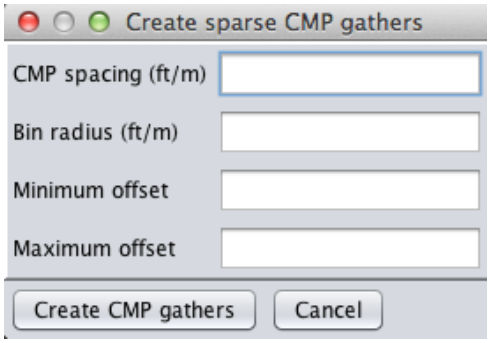


Figure 5---1 Pop---up window for creating sparse CMP gathers.

6 Picking

The Picking menu has all things related to first break picking. The menu items are shown in Figure 6---1. There are six **New picker window…** items and each opens up a tab window specialized to each item. The two **Define linear moveout trend**… items open a separate window where a linear moveout correction can be determined and applied.

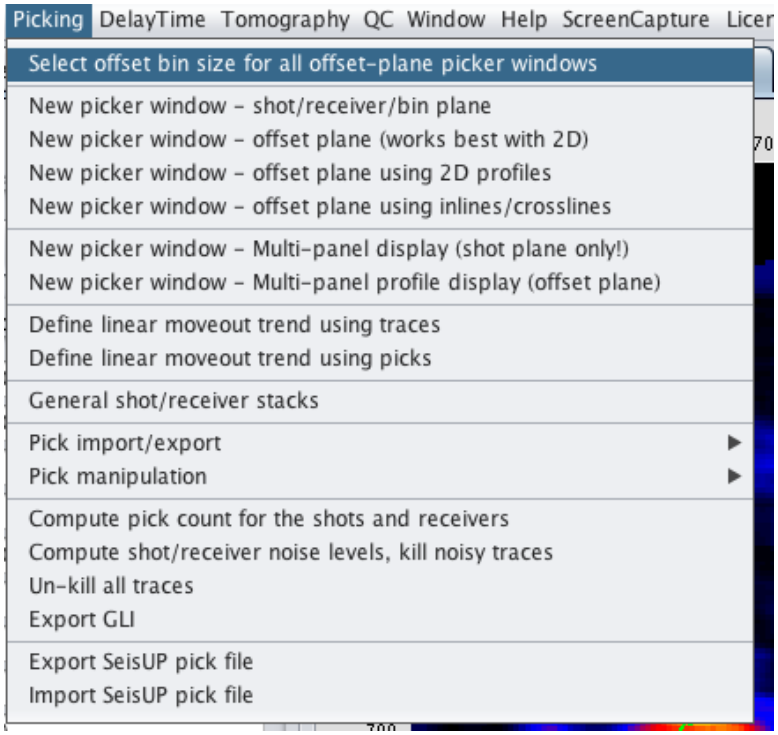


Figure 6---1 The items located in the Picking menu.

6.1 New picker window – shot/receiver/bin plane

The shot/receiver/bin plane is the usual choice for first break picking. So this section will cover in detail this particular Picker window, explaining the various components, icons, buttons and functions. The remaining sections of this chapter will focus on differences in the other Picker windows. The Picker window is one of the main tabs that is not opened by default on a newly created project and is not covered in Chapter 14.

To begin picking on an opened project, go to the **Picking** menu and from the drop---down menu select **New picker window – shot/receiver/bin plane**. Figure 6---2 shows an example default picker window. Notice that the active tab is labeled **Picker #1**, correctly indicating that multiple picker windows (same or different types) can be open at the same time.

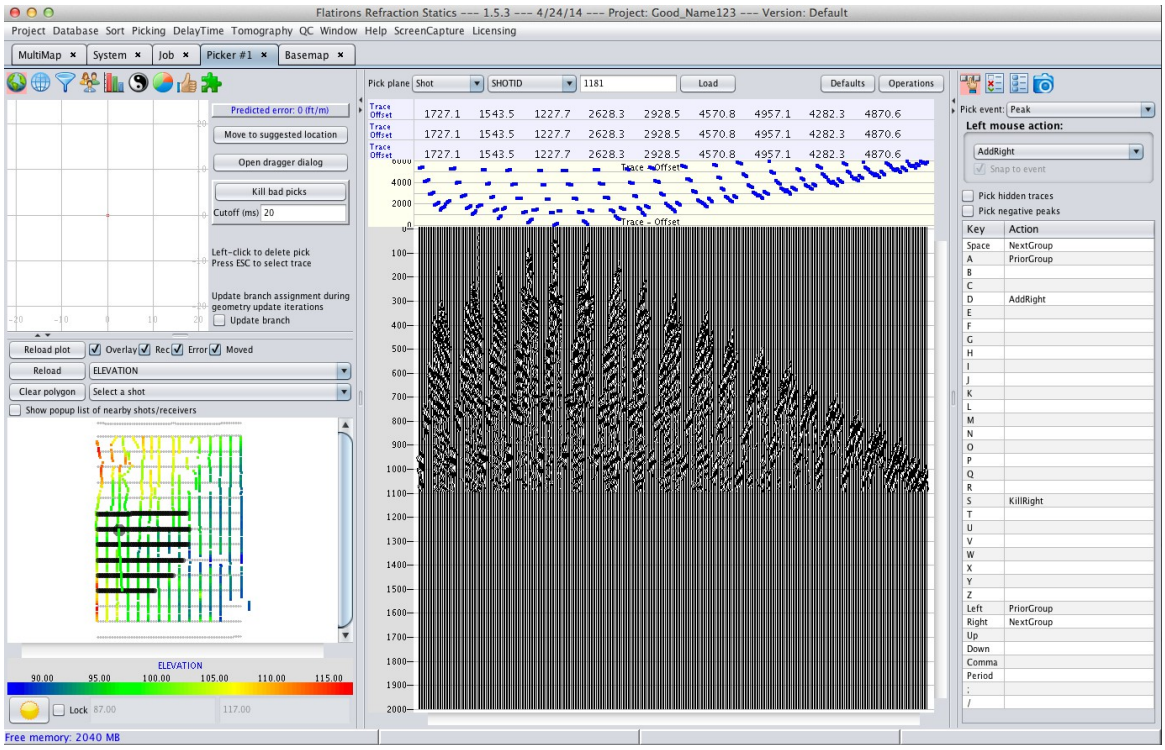


Figure 6---2 Picker window. The window is made up of three panels. The center panel displays traces, while the left and right panels have a set of icon tabs that allow quick access to picking---related actions and functions.

The picker window comprises three panels. The panels can be resized by dragging the vertical border between panels (but decreased only to a point, determined by the size of whatever buttons are in the panels). Also, either side panel can be temporarily closed or opened by clicking on the small arrow markers ( ) appearing near the top of the borders.

The center plane displays traces and is where the major work done in picking will be performed. Which traces to display is determined by the option selected in the drop---down **Pick plane** located at the top left of the panel. Choices include Shot, Receiver, Bin and Sparse CMP. A text box permits entering a specific value. For example, in Figure 6---2 the shot plane with SHOTID 1181 has been selected.

Directly above the trace display are two rectangular regions for displaying trace related information. The light---blue region, **text header**, has by default three rows of user---selected information. The number of rows can be selected in the **Visible picks, header plots** icon tab ( ). Only textual information is displayed in this header. A left or right click anywhere in this are produces a pop---up window from which any Column from the Shot, Receiver or Trace table can be selected. The values for each Column corresponding to the trace displayed directly below it are populated in this region. Because of the density of traces, only information for every n^{th} trace is shown, where 'n' depends on the number of traces displayed and the zoom level.

Similarly, the yellow rectangular region, **plot header**, is a plot of one of the Columns from any of the three tables. Up to four (or zero) plot headers can be selected in the **Visible picks, header plots** icon tab ( ). Only graphical information is displayed in this header. Because the plot uses markers instead of text, a marker for every trace in the display is shown in this region, regardless of the zoom level.

The **Defaults** button pertains to preparing data for various tasks including autopicking, delay time analysis and 3D tomographic analysis.

The **Operations** button opens a pop---up box providing several quick global operations on the data.

Flip polarity. Reverses the polarity of all traces. Selecting this option a second time returns the traces to their original polarity.

Kill current shot or receiver. Sets the current shot or receiver, for the respective plane and indicated by the current ID, to the 'killed' status. For example, if ShotID = 1616 is killed, then a SQL query 'SELECT SHOTID, KILLED FROM Shot' will produce a two column spreadsheet and the row with SHOTID = 1616 with have 'true' in the KILLED column.

Undo geometry corrections.

Unkill traces for current shot or receiver.

Export trace headers to ASCII.

The left panel has nine icon tabs, each of which modifies what is displayed in this panel. One of the main reasons for using icons instead on named tabs is to save on screen real---estate. This is important because during analysis the user will frequently move back and forth between several icon tabs, in both the left and right panels, in order to alter the trace display in the center panel.

Figure 6---3 Icon tabs for left panel in a Picker window.

The selected icon is indicated by a change to a light red background behind the icon. In Figure 6---3 the leftmost icon, Basemap, has been selected. Hovering the mouse over the icon produces a pop---up label or description of the icon. Note that the order of these icons is not fixed.

Basemap ( ). The **Basemap** icon tab is comprised of two panels arranged vertically. The size of each can be adjusted by sliding the horizontal border between the panels. Alternatively, either panel can be temporarily opened or closed using the arrows ( ) located near the left edge of the border. The upper panel relates to geometry corrections. The lower panel displays the basemap. This is very similar to the

Basemap tab, but with different options. More importantly, this view is tied to the trace display in the center panel. So any mouse clicks on the basemap immediately updates the traces displayed in the center panel.

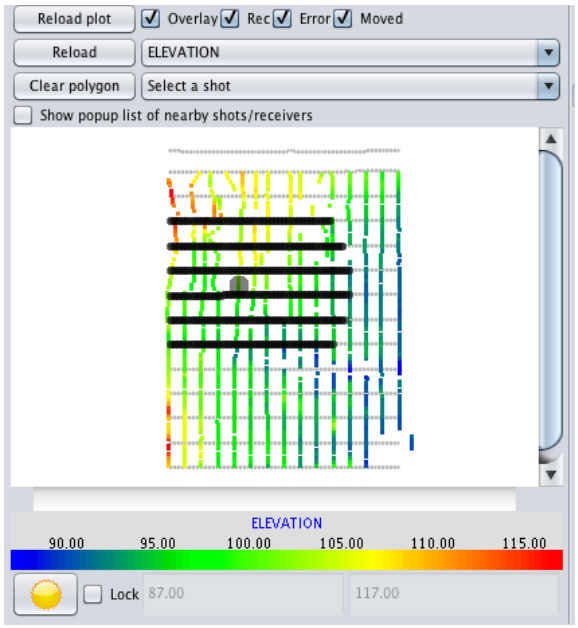


Figure 6---4 Basemap icon tab, lower panel. The selected shot is identified by the light gray circle and the associated receivers are identified with a larger black marker, appearing here as horizontal thick black lines. These black lines are referred to as the Overlay.

In addition to the basemap and usual color bar, there is a drop---down menu to determine the mouse function: select a shot, define a polygon or drag a shot. Another drop---down is use to select the Column used to color the shot markers. There are four checkboxes to toggle what is displayed in the basemap.

When the mouse function **Select a shot** is chosen, a left---click on the basemap will highlight the nearest shot to the mouse position with a light gray circle and identify all associated receivers, as indicated in Figure 6---4. At the same time the center panel is populated with the associated traces. If the trace display was currently in a zoomed view, it will revert to the default unzoomed view. When the mouse function **Drag a shot** is chosen, a left---click---and---hold will cause the yellow region in the center panel to be updated.

The checkbox Overlay toggles the display of associated receivers, identified by a larger black marker. The checkbox **Rec** toggles the display of all receivers, not including the Overlay. The Error checkbox toggles.

Ensemble Navigation ( ). The order of traces displayed and restrictions on what traces to display can be set in the **Ensemble Navigation** icon tab (Figure 6---5). Moving from top to bottom on the window, there are two tabs, one for **Shot** and one for **Receiver**, which are identical in layout, differing only in the Columns available in the respective tables. Note that

the **Pick plane** drop---down in the center panel overrides this tab selection. For example, if the **Shot** tab is current in the **Ensemble Navigation** icon tab, choosing **Receiver** in the **Pick plane** drop---down will cause the **Ensemble Navigation** icon tab to switch to the **Receiver** tab.

Next there are two buttons: **Reload columns**, in case the tables were modified or Columns added, and the data needs to be reloaded, and **Apply options**, for applying the navigation and filtering options made in the window.

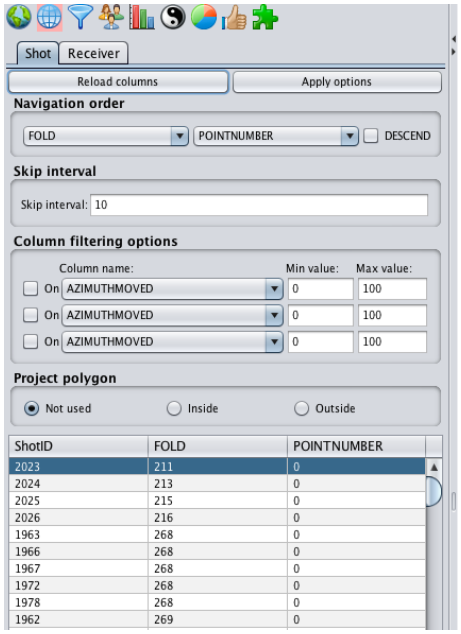


Figure 6---5 Ensemble Navigation icon tab, with Shot tab selected.

The **Navigation order** second has options for two---level sorting, by choosing Columns from the drop---down. Sort order is Ascending by default, with a checkbox to sort in Descending order.

The **Skip interval** section has a box to enter an integer (n) that will result in the center panel displaying every n^{th} trace.

The **Column filtering options** section is a way to restrict what traces are displayed. Up to three separate Columns restrictions can be applied, by supplying a minimum and maximum value for each Column.

The Project polygon is another way to restrict the displayed traces. By default it is not used. If a polygon has been defined (such as by the **Define project polygon** in **Mouse Mode** icon tab of the **Basemap** tab), then traces only inside (or outside) of the polygon can be shown.

Finally, the Shot table is displayed in spreadsheet format, with three columns. The first column has the ShotID (or ReceiverID, if in the **Receiver** tab) while the second and third columns consist of the two Columns selected in the **Navigation order** section. Clicking on

any row (or using up/down arrows, if the spreadsheet is active), thereby selecting a particular ShotID, will cause the relevant traces to be immediately displayed in the center panel.

REMEMBER: For navigation order and filtering options to take effect, the **Apply options** button must be clicked!

Ensemble processing sequence (). 'The funnel'. The purpose of this tab is to prepare and launch batch picking jobs. A processing sequence typically involves applying a series of functions or actions that modify the data in some way. Examples of such functions include moveout, filters, gain, statics, etc. This tab is designed to quickly build, modify and save such sequences.

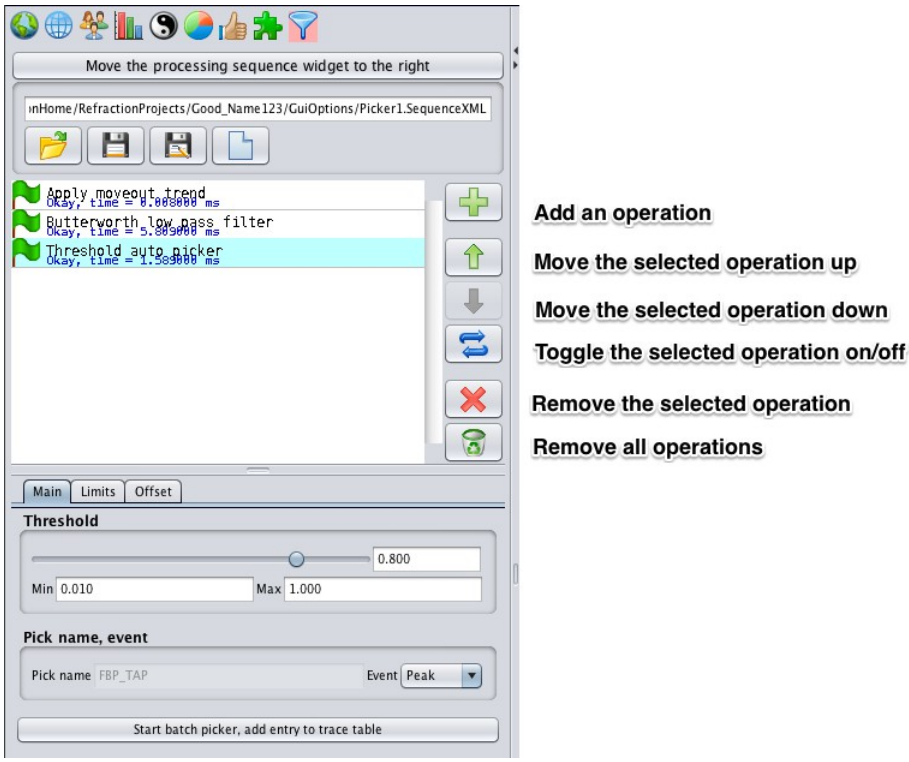


Figure 6---6 The Ensemble processing sequence. In this example, three operations have already been added. The current operation has a cyan background. Any options appearing beneath this window apply to the current operation.

In the example shown in Figure 6---6, three operations have already been added. Icons to the right of the window permit adding, moving or removing operations. A green flag () next to an operation means that it is turned on, while a yellow flag () means that the operation is currently turned off. Beneath the name of each operation is a single line of text. If the operation is toggled off the text states 'Turned off' in red. Otherwise, the text states 'Okay, time = 0.n ms' in blue, where the number of milliseconds is the time taken to run the operation on the current set of traces. Immediately beneath the sequence window are any options available for the currently selected operation. For example, in Figure 6---6 the

Threshold slider in the Main tab apply to the **Threshold auto picker** operation highlighted in cyan. Because operations in the processing sequence are applied to the traces displayed in the center panel, settings adjustments made to an operation are immediately displayed in the center panel. In this way the effect of the processing sequence and settings for each operation on the traces can be seen before running a batch job. Note that the user can move to the **Ensemble Navigation** icon tab, navigate around the Shot (or Receiver) plane to see the effects of the current sequence, and then return to the **Ensemble processing sequence** icon tab to continue editing the sequence.

Above the sequence window are icons for the usual I/O operations: Open existing sequence (), Save sequence (), Save to new file () and New sequence (). This allows for a standard set of processing sequences to be saved and reused, rather than rebuilt each time.

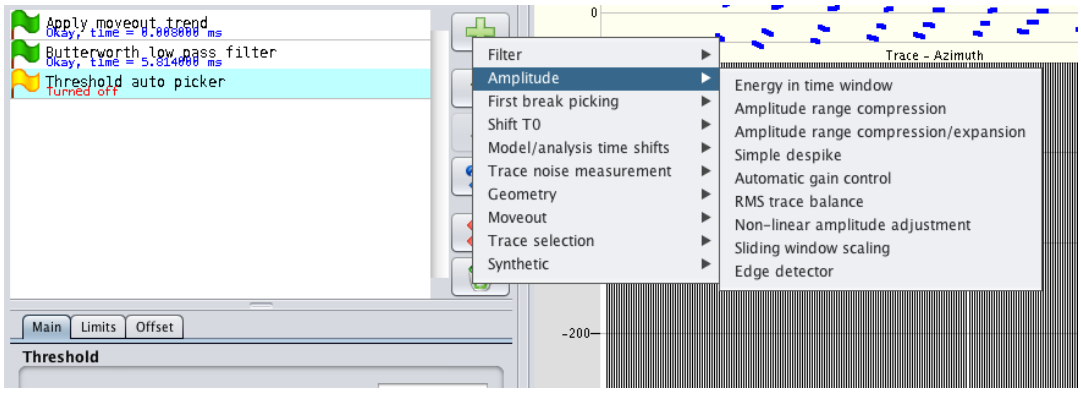


Figure 6---7 Example of submenus available for adding operations.

Flatirons has a large assortment of operations that can be added to a processing sequence. Currently the operations are grouped into ten categories. Figure 6---7 shows the categories, as well as all the operations in the category Amplitude.

Ensemble grouping options (). This icon tab is where the user can select how to group traces when navigating through the survey. By default grouping is off. In this case the group is the entire gather. As indicated in Figure 6---8 there is a checkbox for grouping traces by trace count. Or, the traces can be grouped using a header value. Drop---down lists permit the selection of table and column. Grouping traces together aids in navigation; the right panel in the **Picker** window has predefined keyboard shortcuts for quickly moving from one group to the next. This panel also has a separate tab to list all groups so that any desired group can be navigated to immediately.

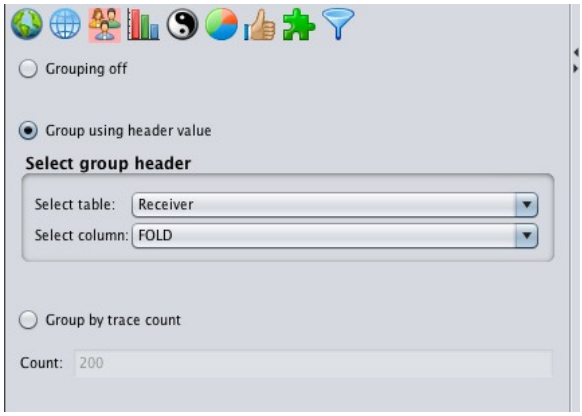


Figure 6---8 The Ensemble grouping options tab.

Horizontal axis selection (). This self---explanatory icon tab is used to set the horizontal axis in the trace display of the center panel in the **Picker** window. Figure 6---9 shows two methods to set the horizontal axis. The first is by selecting a primary and secondary sort order using drop---down lists for table and column. Traces are displayed based on this order and automatically evenly spaced. The other method, Axis Header, sets the horizontal axis to a single Column from the desired table. The traces are plotted strictly according to the values for this Column so spacing of the traces will typically not be uniform.

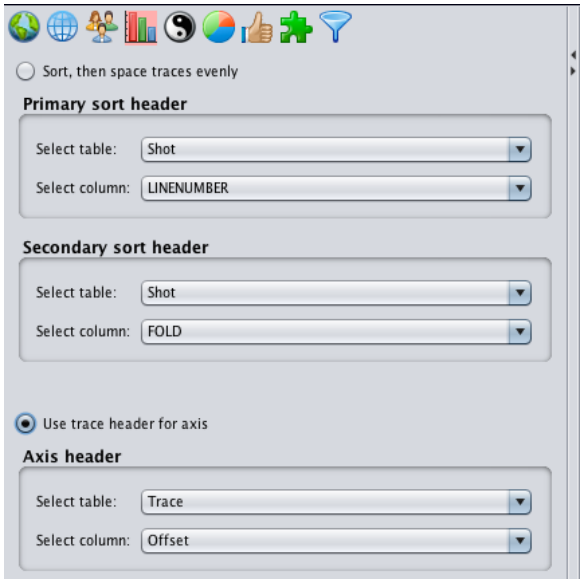


Figure 6---9 The Horizontal axis selection tab determines how traces are arranged horizontally in the trace display of the center panel of the Picker window.

Wiggle display options (). This icon tab is used to select how to display individual traces. The Display type option allows the choice between 'wiggles' or Gray scale. As noted 'Gray scale display only available if horizontal axis is Sort, then space traces evenly.' Remember, the Sort option is set in the **Horizontal axis selection** icon tab. If **Gray scale** is selected, no

other options on this tab apply. If the Display type is **Wiggles**, then the **Wiggles display options** shown in Figure 6---10 apply.

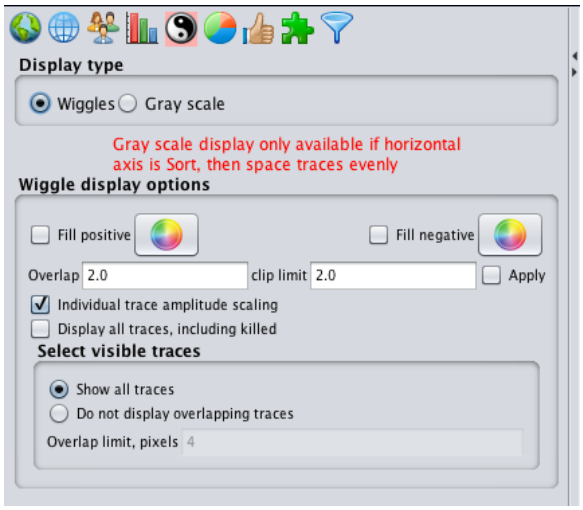


Figure 6---10 Wiggle display options, including gray scale.

Standard fill options are shown in Figure 6---11. Positive fill has a default color of black and

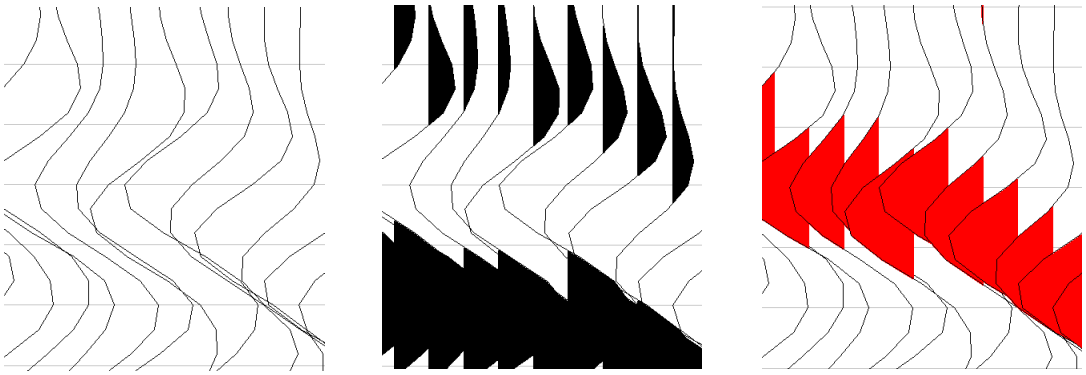


Figure 6---11 Wiggle display options. Left: no fill, Middle: fill positive (default black), Right: fill negative (default red). Not shown: Fill positive and negative.

negative fill is red, although the user can choose any color from the **Select positive color** and **Select negative color** buttons ( ). **Overlap** determines how much the traces are allowed to overlap and works in conjunction with **clip limit**, which will chop off trace peaks above this limit. Other options toggle whether to display killed traces, all traces, and if individual trace amplitude scaling is desired.

Visible picks, header plots ( ). This icon tab (Figure 6---12) combines options for coloring picks and selecting the number of text and header plots to display above the trace display in the center panel.

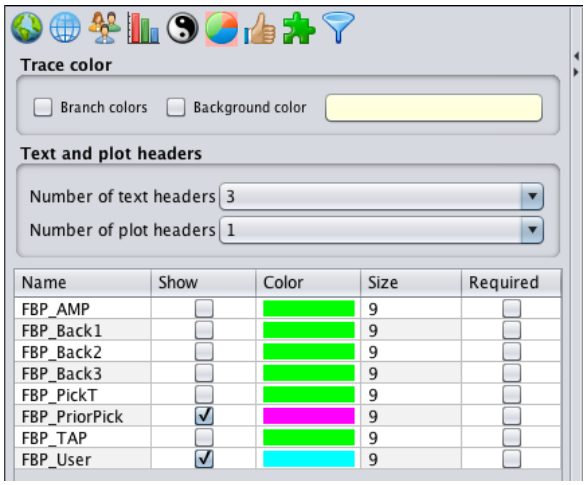


Figure 6---12 Visible picks, header plots icon tab.

The spreadsheet format lists different named picks ("FBP" means First Break Picks), whether to display them on traces, the color and size of the marker, and if display of a specific pick---type is required. Similarly, the Background color checkbox turns on background coloring in the trace display. The background actually alternates between the default gray and the color selected here. Flatirons identifies incremental steps in the selection for Horizontal Axis and alternated the background color for each step. For example, if the **Primary sort header** in the **Horizontal axis selection** is Table: Receiver, Column: LineNumber then the background color is alternated by Receiver line. This feature is useful to quickly identify intrinsic groupings that may not be immediately obvious from the displayed traces.

Select visible traces (👍). This icon tab is used to limit what traces to display. By default all traces are displayed. Trace display can be limited by branch, either from **delay time** of **hybrid GLI** analysis or by **header values**. Limitations are selected by radio buttons as shown in Figure 6---13. For both delay time and hybrid GLI, a particular branch or all branches can be selected from a drop---down list. If the limitation is by header value, the table and column are chosen from a drop---down list also. Additionally, a range limitation applied to the column value can be set by entering a Minimum and Maximum value.

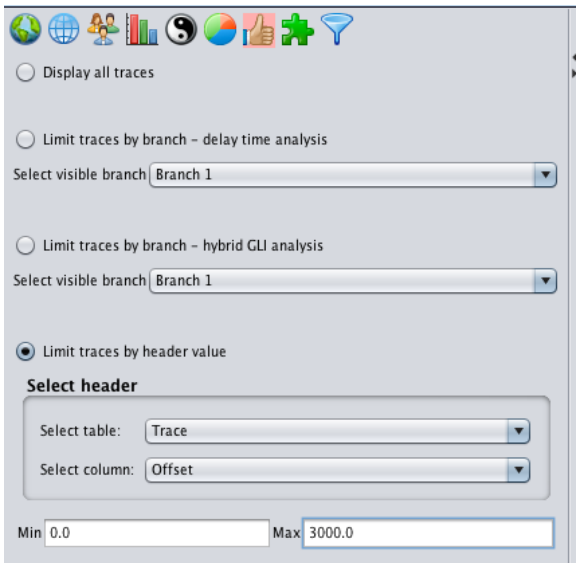


Figure 6---13 Select visible traces icon tab. Default is display all traces but three methods to limit the traces are provided.

Be aware that the Min/Max values apply only to the traces displayed. For example, let these values be set to Min=1000, Max=6000 for Table: Trace and Column: Offset. Also, suppose the **Axis header** is set to Table: Trace and Column: Offset in the **Horizontal axis selection** tab icon. Finally, in the **Axis locks, group info** tab icon ( ) in the right panel (covered in detail below) the axis is locked at Min=0, Max=7000. Then the trace display and headers in the center panel will appear as in Figure 6---14. As the trace header, plot header and trace display all show, only traces with offset ranging from 1000---6000 feet are plotted, while the axis ranges from 0---7000 feet.

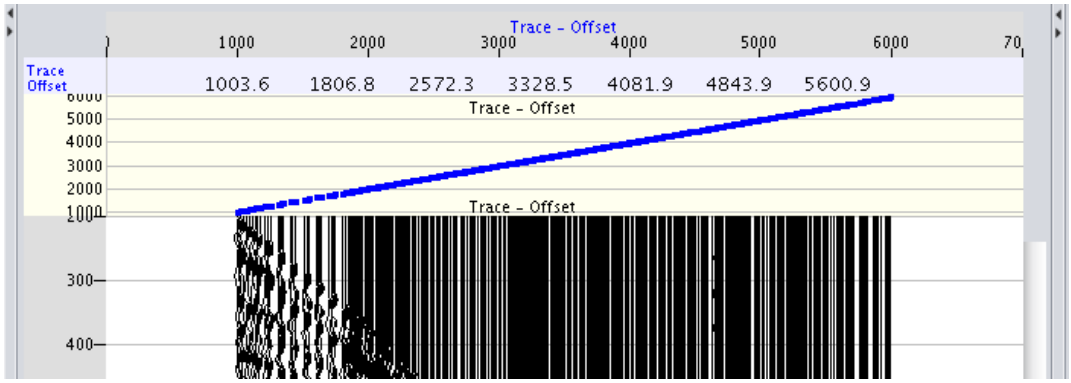


Figure 6---14 Partial trace display with headers, showing effects of various offset settings.

Prior pick usage ( ).

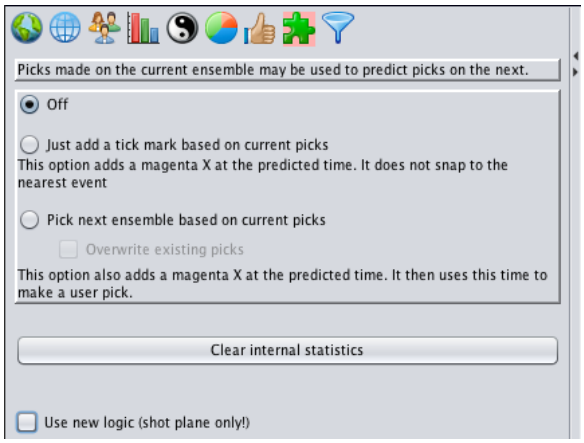


Figure 6---15 Prior picks usage icon tab. [TBD]

The right panel on the picker window has four icon tabs that aid the user in picking events.

Figure 6---16 Icons tabs in right panel of picker window.

The selected icon is indicated by a change to a light red background behind the icon. In Figure 6---16 the leftmost icon, **Picker controls**, has been selected. Hovering the mouse over the icon produces a pop---up label or description of the icon. Note that the order of these icons is not fixed.

Picker controls (). This icon tab shown in Figure 6---17 has settings for the **Left mouse action** and keyboard shortcuts, or hot---keys. (The right mouse button is always used to control zooming.) See Appendix 1 for a detailed list of these actions. At the top of the tab is a drop---down list to select which pick event is desired: Peak, Trough, NoEvent. The NoEvent option picks a time sample of the nearest trace to the current mouse position. The **Left mouse action** has a drop---down list of pre---defined actions. See Appendix 1 for a detailed list of these actions. When the left mouse button is clicked somewhere over the trace display window, the action defined here will take place. For some actions a **Snap to event** option applies and a checkbox is available to toggle this option.

Two other options, toggled with checkboxes, are available: **Pick hidden traces** and **Pick negative peaks**. Remember that not all traces may be displayed – the **Wiggle display options** icon tab () has an option to hide overlapping traces. Checking the **Pick hidden traces** checkbox will ensure that any hidden traces are also picked. If the **Pick negative peaks** option is checked then the picking algorithm is permitted to pick peaks with negative amplitude.

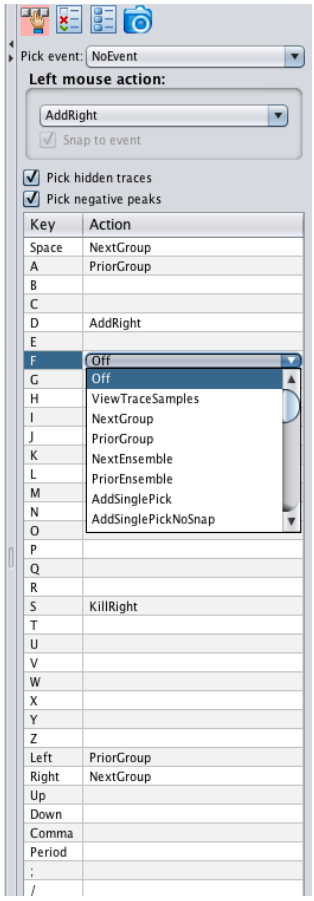


Figure 6---17 Picker controls icon tab is used to select event to pick on, left mouse action and keyboard shortcuts.

Figure 6---18 shows an example of a **Left mouse action**. In this case, the Pick event is set to Peak and the Left mouse action is **AddRight**. A left mouse click in the red circle has resulted in the nearest peak on every trace to the right being picked. Note that to the left of the mouse click, the previously picked peaks on a separate earlier event are unchanged. Note that if **NoEvent** had been selected, the blue dots to the right of the red circle would form a horizontal line to the right.

Referring again to Figure 6---17 the bottom section of the **Picker controls** icon tab is a two---column listing of keyboard keys and associated defined actions. When the mouse is over the trace display, entering a key will perform the associated action. Because grouping, determined in **Ensemble grouping options** (), is so frequently used, keyboard shortcuts for related to group actions are defined for the Space bar and Left/Right arrow keys.

The assignment of an action to any of the listed keys is set by a drop---down list, made visible by clicking in the Action column. In the figure the 'F' key is currently Off (nothing in the Action column). There are nearly 50 defined actions. If desired, multiple keys can be set to the same action. An example is if a user has a preferred key for a particular action that already has been defined; there is no need to un---define a predefined key.

It is important to realize that when an action affects the traces in some way (e.g., picking an event), the action is generally only applied to traces currently in the display window, that is, at the current zoom level. This feature was added to prevent changes 'off screen' that and therefore difficult to follow. One exception is the **KillPicksOutsideZoom** action.

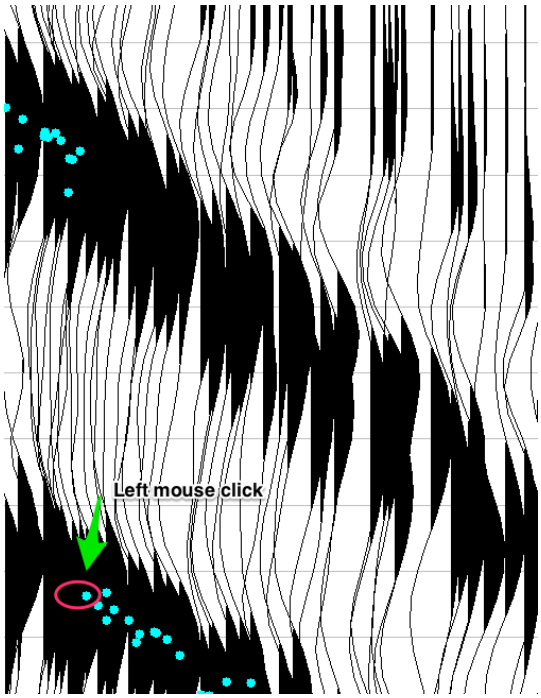


Figure 6---18 Example of the AddRight left mouse action with Pick event set to Peak.

Axis locks, group info (). This icon tab serves the dual purpose of locking the vertical (Time) and horizontal axes in the trace display. Both axes have a toggle for axis locking as well as setting for Minimum and Maximum values. The units for the Time axis are milliseconds. The units for the Horizontal axis are determined by the Column chosen for the Horizontal axis. Remember that this is set in the Axis header of the **Horizontal axis selection** () icon tab.

The second purpose of this tab is to display Grouping information. The bottom section of the tab displays the groups set in the **Ensemble grouping options** () icon tab. If no group was selected, the **Groups** section is empty. If **Group by trace count** was selected then each group is listed, where the group size is based on the user---defined **Count**. An example of the other grouping option, by group header, is shown in Figure 6---19. In this case grouping is by table: Trace, column: Crossline. Each row has two values: the first is the Column value for that group, the second, in parentheses indicates the number of traces in that group.

Clicking on any particular group cause the trace display to jump to that group. Using the up/down arrows allow the user to quickly cycle through all the groups. Note that depending on what order the analysis has proceeded among the icon tabs, if might be necessary to toggle grouping off/on in **Ensemble grouping options** for the Groups in **Axis locks, group info** to be updated.

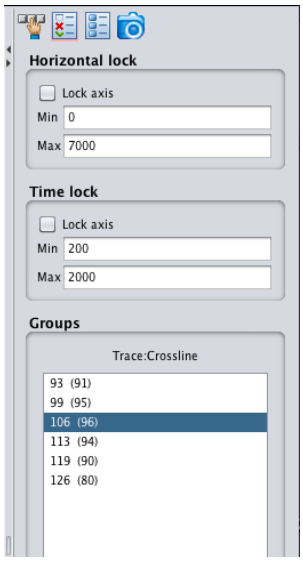
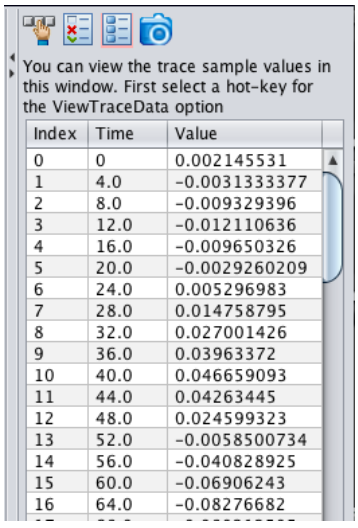


Figure 6---19 Axis locks, group info tab icon. Horizontal and Time (vertical) ranges can be set and locked. A listing of all groups, if defined are also displayed.

View traces samples ( ). This icon tab is used to display the actual data, sample by sample, for a single user---selected trace. Figure 6---20 shows a portion of this scrollable tab. The data is listed in a three---column format, with each row giving the sample's index, time (milliseconds) and value.

The method to populate the table is by placing the mouse over the desired trace and hitting a hotkey that has been assigned the **ViewTraceSamples** Action (in the **Picker controls** icon tab). Using a hotkey to select the trace permits the left and right mouse buttons to retain their functions (picking and zooming).



Index	Time	Value
0	0	0.002145531
1	4.0	-0.0031333377
2	8.0	-0.009329396
3	12.0	-0.012110636
4	16.0	-0.009650326
5	20.0	-0.0029260209
6	24.0	0.005296983
7	28.0	0.014758795
8	32.0	0.027001426
9	36.0	0.03963372
10	40.0	0.046659093
11	44.0	0.04263445
12	48.0	0.024599323
13	52.0	-0.0058500734
14	56.0	-0.040828925
15	60.0	-0.06906243
16	64.0	-0.08276682

Figure 6---20 A portion of the View traces sample icon tab. Each sample of the selected traces is displayed in spreadsheet format.

Snapshot creation (). The final icon tab on the right panel of the Picker window is for taking a snapshot of each shot. The Create snapshots button will create a subdirectory in the **SnapshotVersions** folder and for **every** shot a PNG file is generated, containing an image of every trace associated with that shot. The user sets the dimension of the image in pixels. As noted in Figure 6---21, there are two restrictions. First, **Grouping** must be turned off (in the **Ensemble grouping options** icon tab). Second, the trace display in the center panel of the Picker window must be set to the Shot plane.

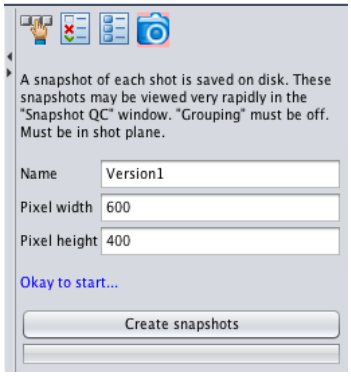


Figure 6---21 Snapshot creation icon tab.

Once the Snapshot has been created the images can be quickly scanned in the **Snapshot scanner** tab, which is opened from **QC** → **Ensemble snapshot viewer**. Figure 6---22 is an example of this tab. Various controls for manual or automatic scanning are available in the left panel. There is also a convenient slider bar above the image display for quick navigation.

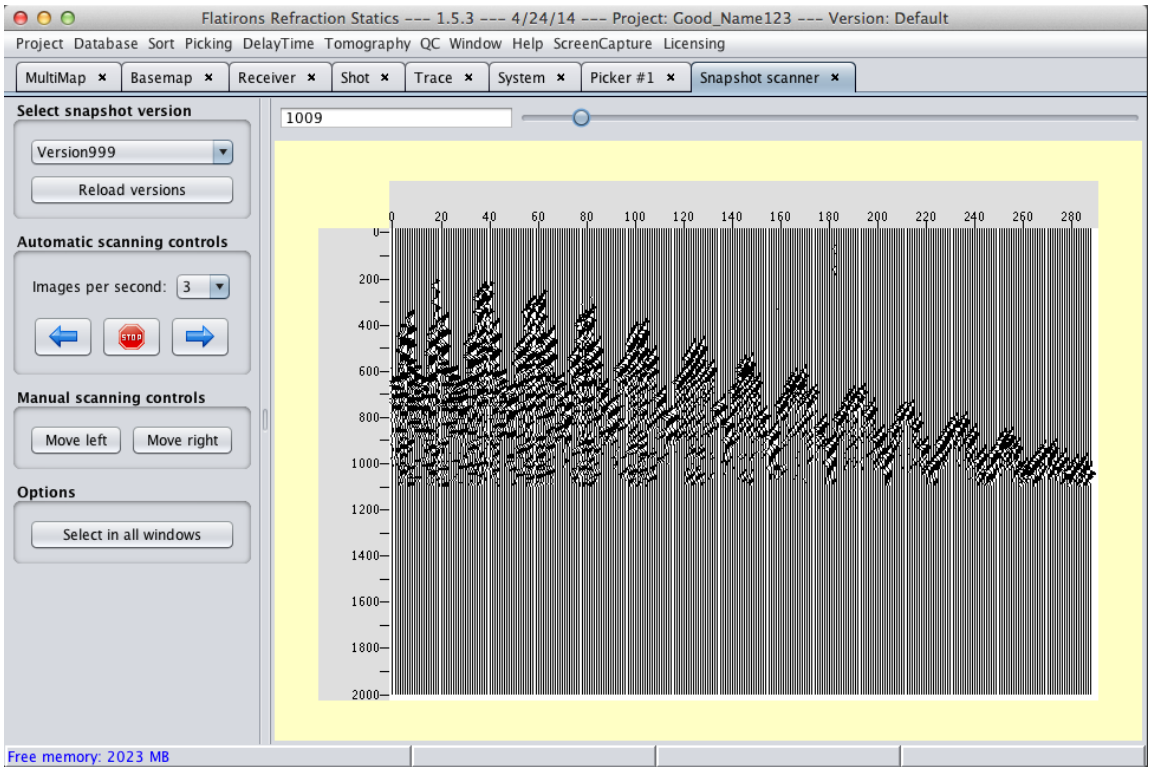


Figure 6---22 Snapshot scanner tab, displaying shot ensemble images created in the Picker window.

6.2 New picker window – offset plane (works best with 2D)

The offset picker window is designed to bin traces by offset. It is similar to the standard Picker window with a few differences (Figure 6---23). There are buttons for navigation to previous/next offset bins (outlined in red box). Traces for the current offset bin are identified by black lines on the basemap. To prepare for offset binning, the bin size must be set. The **Offset bin selection** icon ( ) in the left panel, shown in Figure 6---24, has a text box for entering the bin size. Clicking on the **Perform binning** button computes the offset bins. The rest of the panel displays the bins with number of trace in each offset bin. Clicking on a row will update the trace display with traces in that bin.

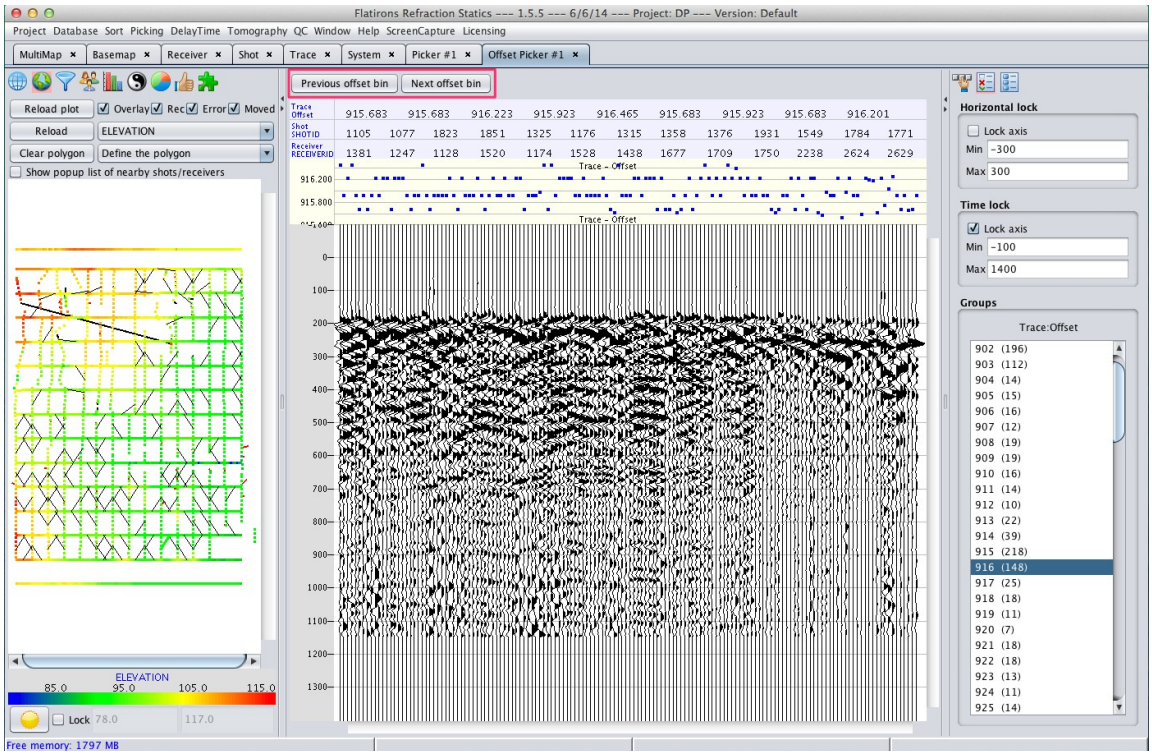


Figure 6---23 Offset Picker window.

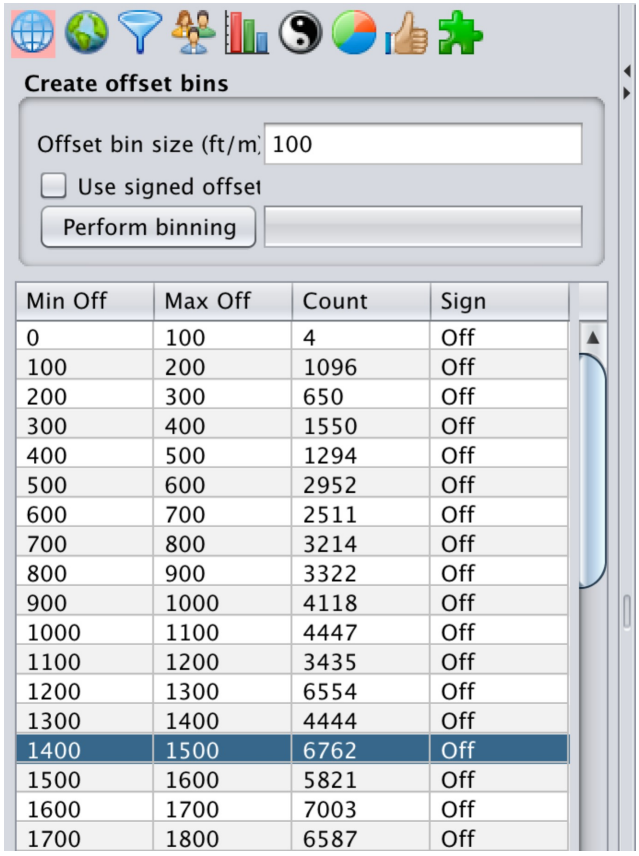


Figure 6---24 Offset bin selection icon. Use this to set the bin size and perform the binning.

6.3 New picker window – offset plane using 2D profiles

This picker window also groups traces by offset but the traces are restricted to those that intersect a 2D profile across the survey. Figure 6---25 shows the picker window with a profile already defined. The default icon tab in the left panel is the **Profile selection** ( ) icon tab. There are just a few differences in this variation of the Picker window.

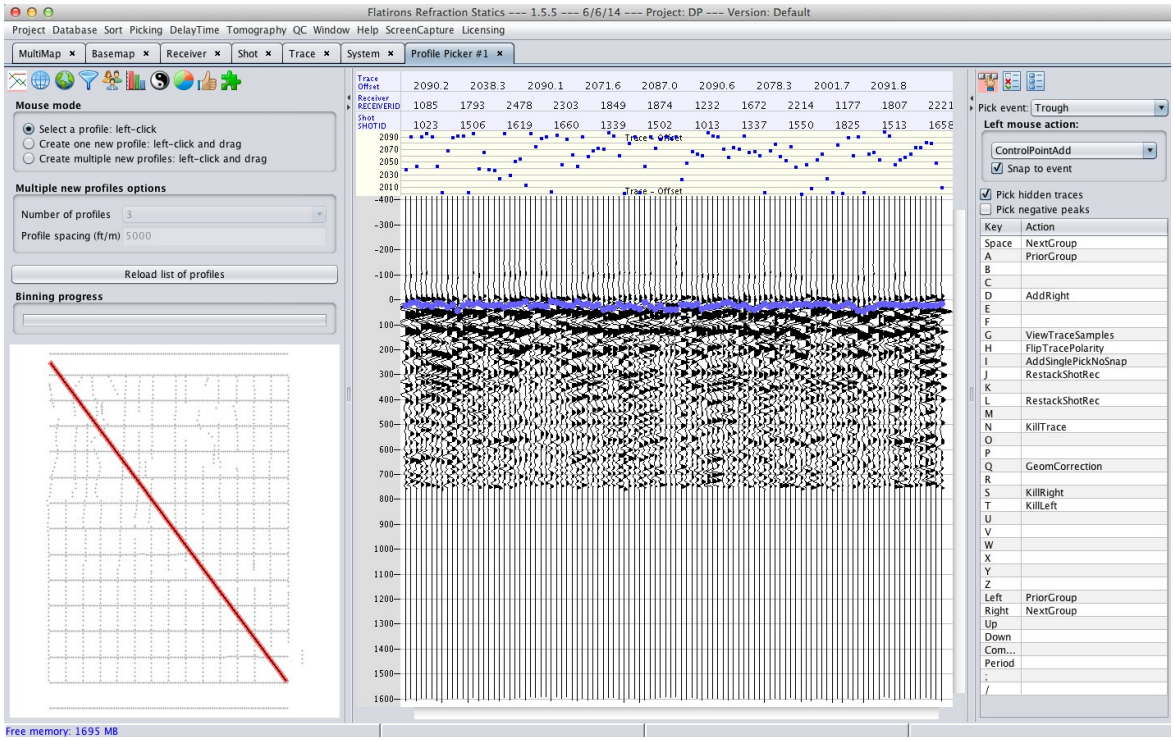


Figure 6---25 New picker window --- offset plane using 2D profiles.

Figure 6---26 shows a close---up of the **Profile selection** tab with two different Mouse mode settings. Three **Mouse mode** settings are available. The first is simply select a profile; this mode is used only if a profile has been previously created. The second mode is used to create a single profile. To do this, just left---click and drag a line across the basemap. Creating a profile requires re---binning the data so a progress bar indicates the progress of this process.

The third mode is used to create multiple profiles at once. These profiles will all be parallel and are generated by the same left---click and drag method. However, the user has two additional options that become active once this mode is selected. The first is the number of profiles, selected from a drop---down list and the second is the spacing between profiles.

The number of profiles is restricted to be an odd number. When the set of profiles is being created, one profile is generated at the mouse position while an even number is created on either side. For example in Figure 6---26, a set of five profiles were created, with two automatically generated on either side of the central one.

Additional options relate to the bin settings and offset. After the left mouse button is released (for single or multiple profiles), a popup window, again see Figure 6---26, has settings for bin spacing, bin size, minimum and maximum offset. Once these are set selecting the **Create new seismic profile** button creates the profile(s).

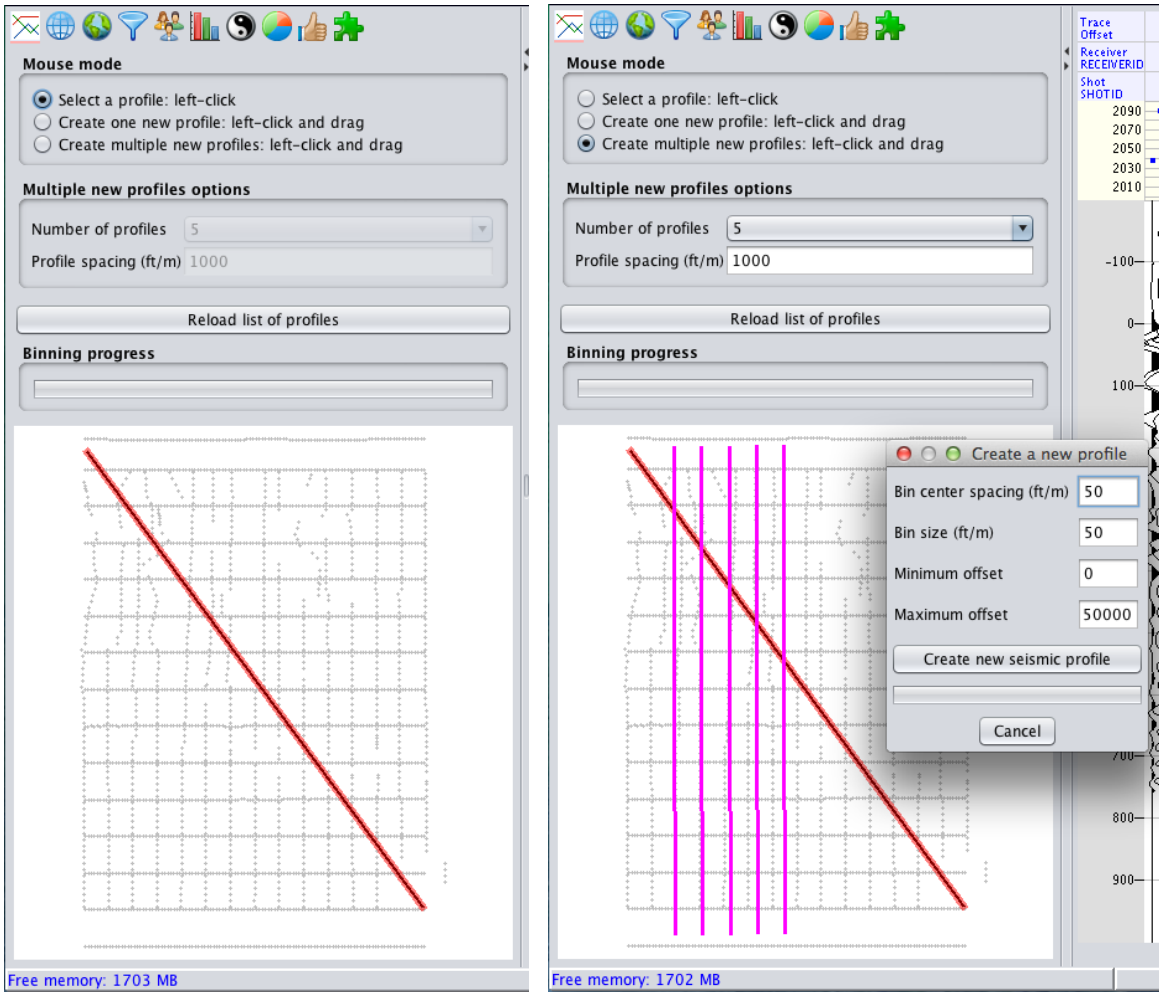


Figure 6---26 The Profile selection icon tab in the Picker window. Left: Mouse mode to select a profile. Right: Mouse mode to create multiple profiles, with popup window to specify parameters.

Figure 6---27 shows the **Navigation** icon tab (called **Offset bin selection** here) and the **Basemap** icon tab. A particular offset bin (3600---3700 ft) with 134 traces has been selected. On the basemap, these traces are shown in black; notice that they all cross the specified profile. Also note that no azimuth restrictions have been applied. Therefore, traces of any azimuth are selected, as indicated by the solid green circle.

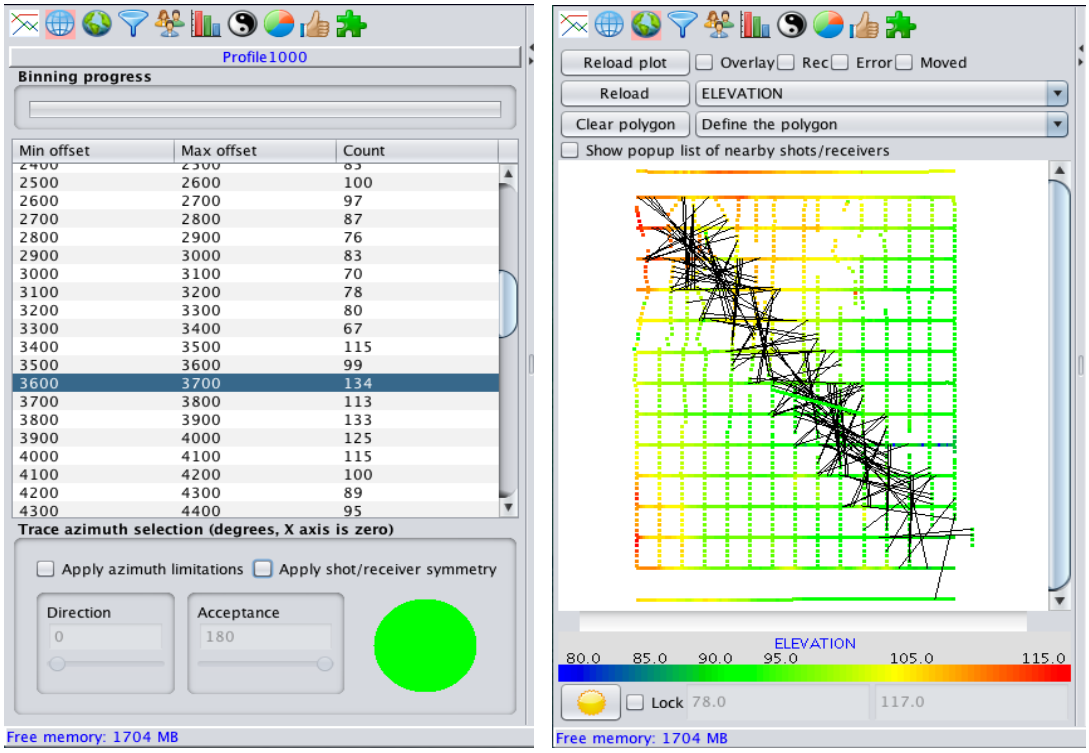


Figure 6---27 Example of offset binning with no azimuth restrictions. On the basemap, all traces in the selected bin that cross the specified profile are displayed.

Figure 6---28 shows the same example as Figure 6---27 with two changes. First, azimuth restrictions have been applied. To do this, first specify the **Direction,** which is the angular direction, relative to Easting, used to limit the azimuth. Second, specify the **Acceptance**, which is the opening angle, or 'cone', about the **Direction**, which all trace azimuth's must lie. In Figure 6---28, the Direction is 0° and the Acceptance is 45°. This azimuth limitation is indicated by the green 'cone' in the red circle.

But notice there is also a green 'cone' to the left, or a Direction equal to 180°. This is because the **Apply shot/receiver symmetry** option has been checked. In the basemap, notice that only traces that lie within the green 'cones', that is, satisfy the azimuth restrictions, are plotted.

Figure 6---29 shows the same example as Figure 6---28 except the shot/receiver symmetry option has been unchecked. So now the only traces selected are those with shot---to---receiver azimuth within the 45° Acceptance cone, about the 0° Direction. Obviously, this cuts the number of selected traces in this offset to bin to approximately half those in Figure 6---28.

Only one profile can be active at a time. For example, if the process of creating the five new profiles demonstrated in Figure 6---26 was completed, then only one could be active. Follow the same procedure as discussed above: put the Mouse mode in 'Select a profile' and left---click on the desired profile. The active profile will turn red and re---binning will proceed. Regardless of how many profiles are defined, only traces that cross the active profile are included.

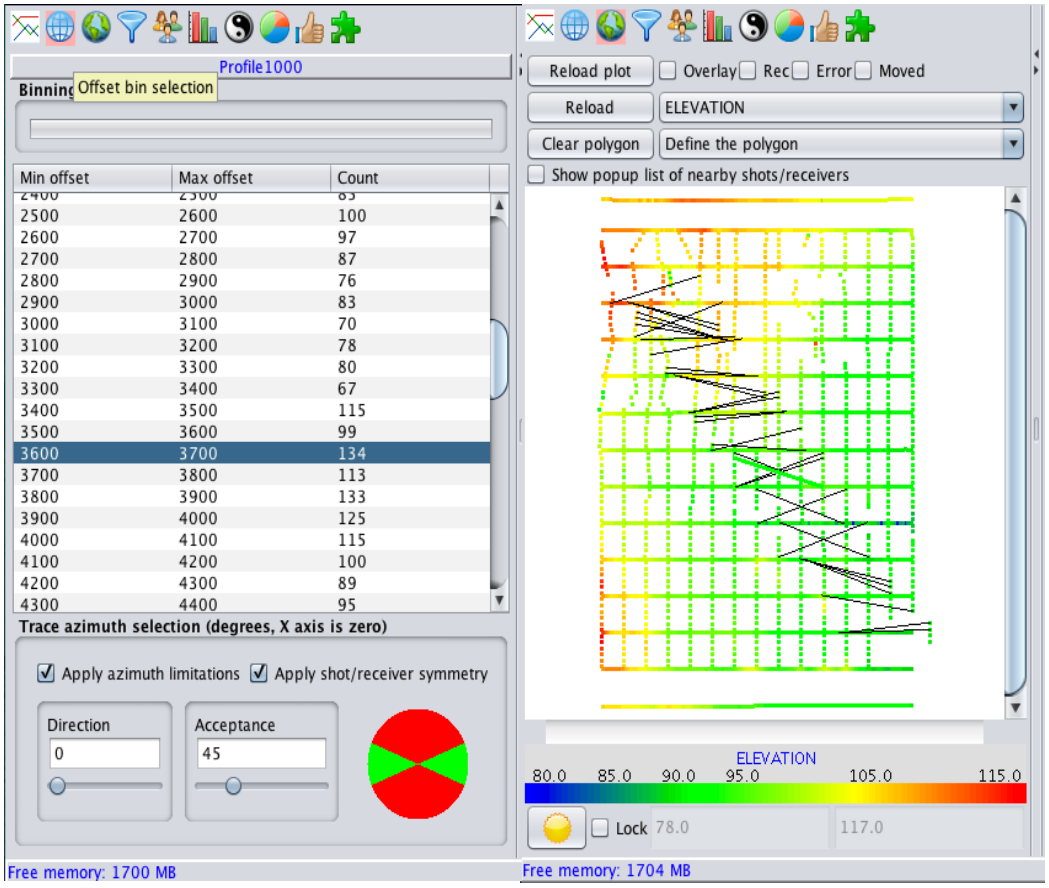


Figure 6---28 Same example as shown in Figure 6---27 except that azimuth restrictions are applied and with shot/receiver symmetry accepted.

Other than these differences the 'offset plane using 2D profiles' picker window is the same as the 'shot/receiver/bin plane' picker window.

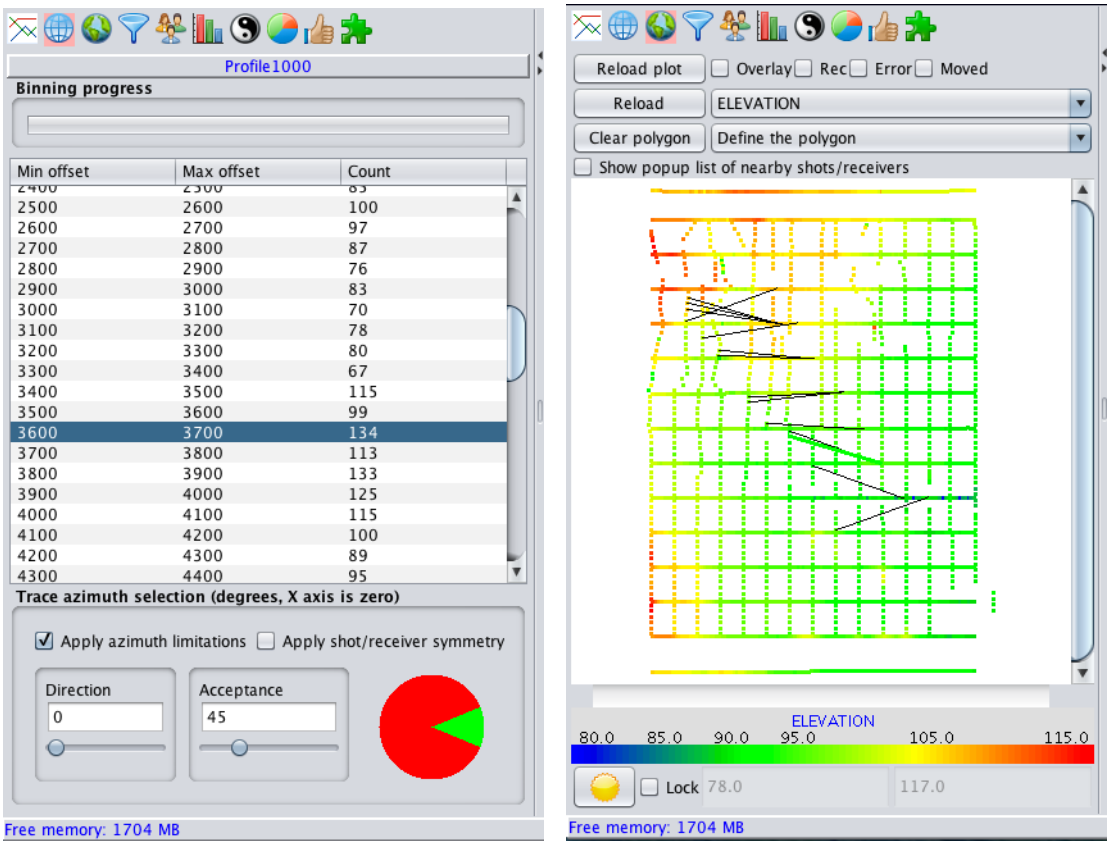


Figure 6---29 Same example as in Figure 6---28 except that shot/receiver symmetry is not applied. This cuts the number of acceptable traces approximately in half as indicated in the basemap.

6.4 New picker window – offset plane using inlines/crosslines

Another option for picking is to organize traces by offset and navigate using inlines or crosslines. To open this picker window go to **Picking → New picker window – offset plane using inlines/crosslines**. A new tab window called **I/C Picker** will open, as shown in Figure 6---30. Notice that the Navigation icon ( ), called **Offset bin selection** in this window, is in the right panel now. As the name implies, navigation is by inline or crossline, selected from a drop---down list, indicated by the red box in the figure. Buttons permit going to the next or previous inline/crossline and the current inline/crossline is displayed between the drop---down list and navigation buttons.

As shown in the figure, at the bottom of the Navigation tab are the Azimuth restriction options, although none are set for this example. Also shown on the basemap are the traces in the current offset bin.

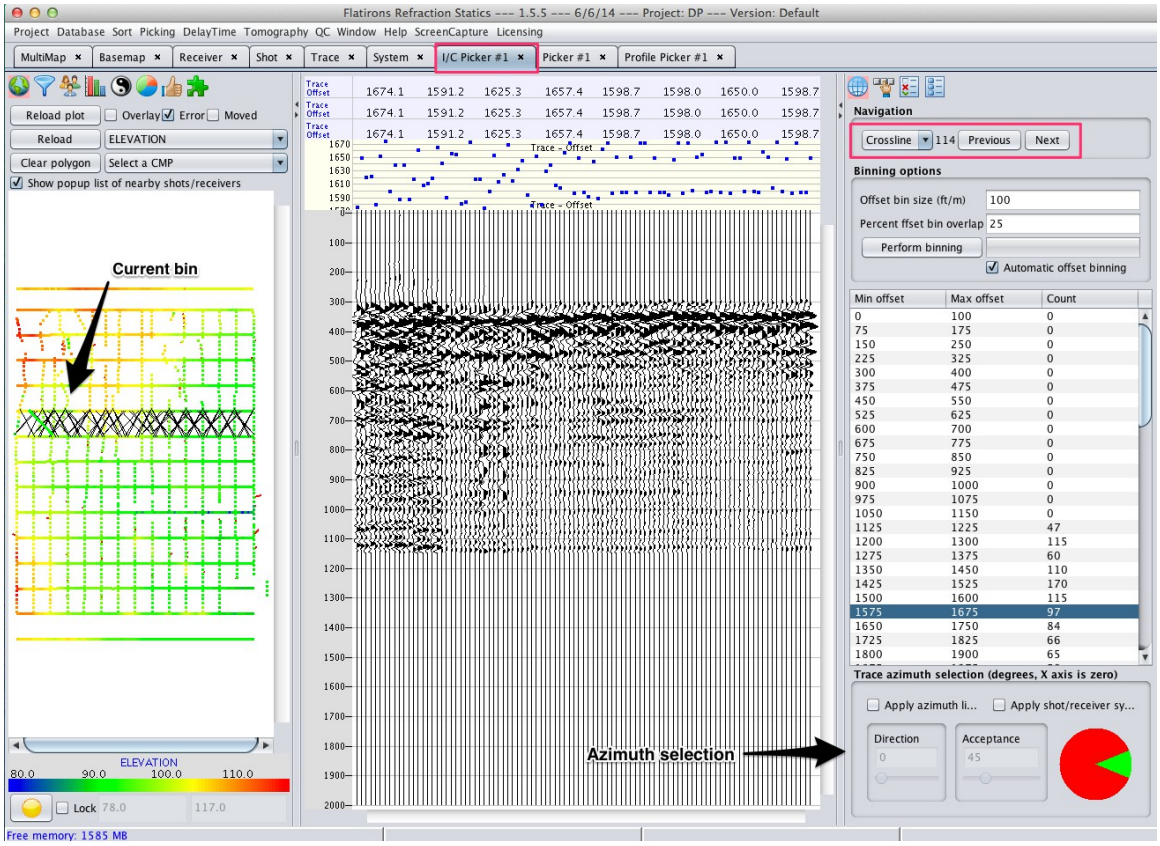


Figure 6---30 New picker window --- offset plane using inlines/crosslines. Key features are highlighted.

Figure 6---31 has a close---up of the Navigation icon tab. Besides navigation there are also binning options. Both the bin size and the amount that bins are allowed to overlap, in percent, can be modified. If either of these options are changed the bins must be re---computed by clicking on the **Perform binning** button. Finally, a checkbox toggles automatic binning. When checked, offset binning is automatically performed whenever a new incline or crossline is selected.

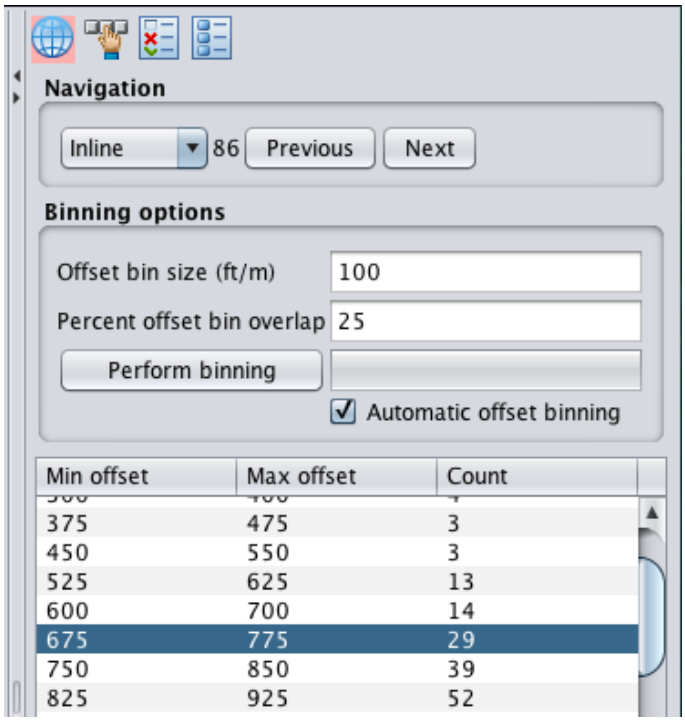


Figure 6---31 Close---up of the Navigation icon tab showing the navigation and binning options.

6.5 New picker window – Multi---panel display (shot plane only)

A variation on the standard picker window is to display multiple shots at once. This option is found at the **Picking → New picker window – Multi---panel display (shot plane only!)** menu item. A new tab window called **MultiPicker** will open, as shown Figure 6---32. Readily observable differences from the standard picker window are the absence of the Ensemble grouping options icon ( ) from the left panel and View traces samples ( ) and Snapshot creation ( ) icons from the right panel.

More significantly, the Basemap icon tab has been re---configured to permit selecting multiple shots. A close---up of this tab is shown in Figure 6---33. In this close---up and the main window, notice the color---coding: the shot in the basemap is marked with a colored circle, the shot selection box has the same background color and the vertical axis in the trace display also has the same color.

Configuration options are set in the **Plot count, plot headers** section. Up to six plots can be display simultaneously. From none to four text headers and none to four plot headers can appear above each plot. As usual, the information each header display can be independently configured. If some combination of plot count and plot/text headers exceeds the vertical resolution of the screen, the application will simply refuse to plot anything and the trace display in the center panel will remain gray. If this happens, reducing the number of plot headers incrementally usually resolves the issue.

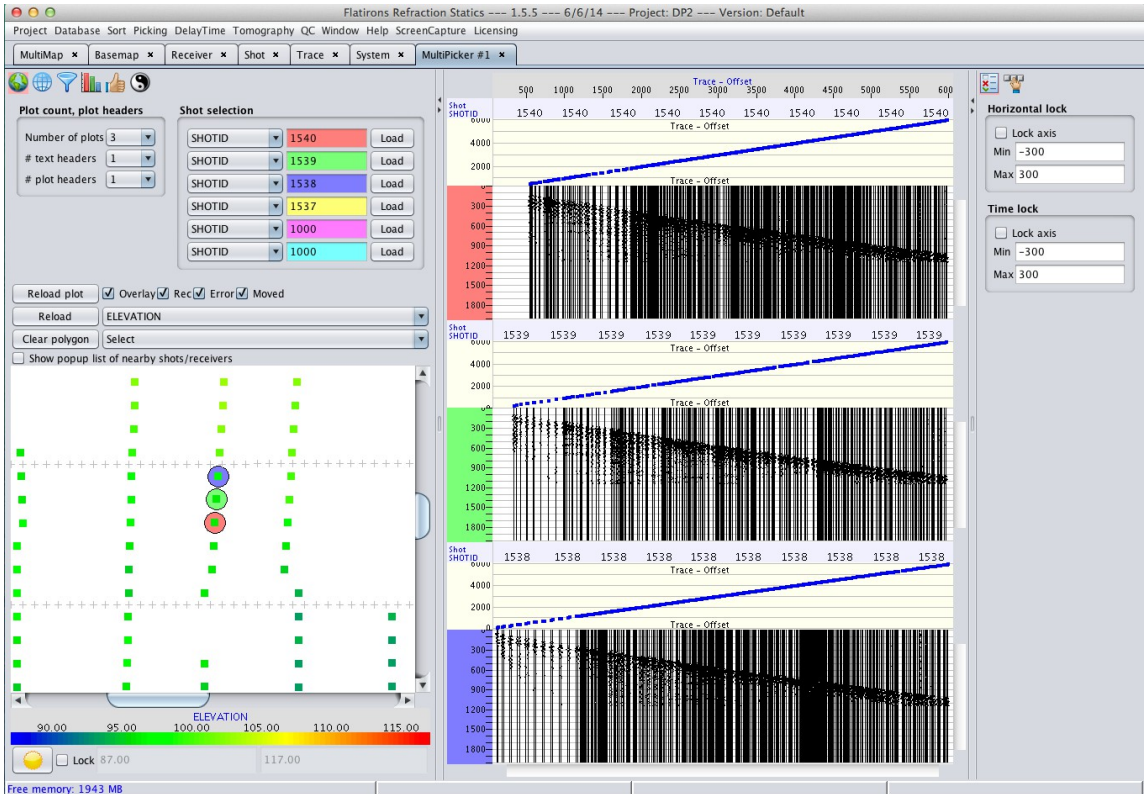


Figure 6---32 MultiPicker window for displaying multiples shots at once.

In the **Shot selection** section there are six drop---down lists, which default to **SHOTID**. Other options in the drop---down are indicated in the sixth drop---down list in Figure 6---33. Although six slots are always displayed, the number of plots determines the number that is active. In this example, three plots were selected so only the first (top) three slots are in effect.

The drop---down list selects the Column while the text box is used to select the entry. A **Load** button is used to update the display after a change is made. Another option for shot selection is to simply left---click somewhere on the basemap. The nearest shot and 'nearest' shots, in some sense are identified and the trace display automatically populated.

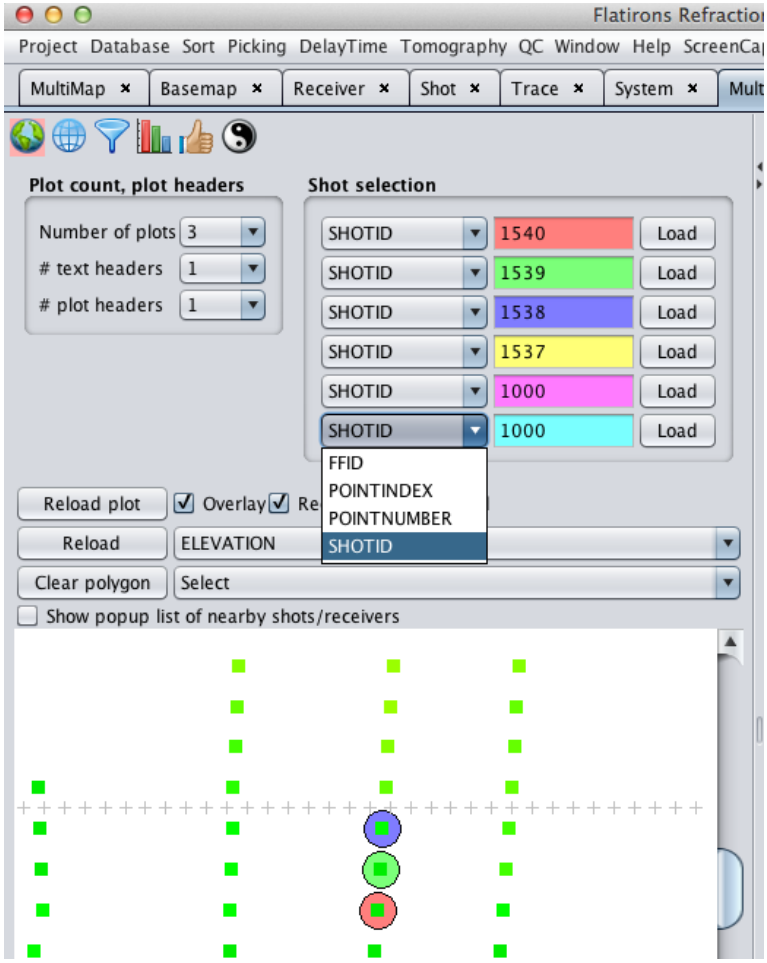


Figure 6---33 Close---up of the Basemap icon tab in the MultiPicker window.

The **Ensemble processing sequence** ( ) icon tab for the Multi---panel shot---plane window is generalized from its normal appearance. Because there are multiple shots, the user may want to apply different sequences to the ensembles in different panels. Figure 6---34 show the upper portion to the Ensemble processing sequence window. Notice the separate numbered tabs and as well as the default **Shared** tab. Naturally, the label and the color correspond to the specific shot displayed in the panels. The **Shared** tab holds any processes that the user wants to apply to all panels. Also, any processes in the **Shared** tab will be applied before processes in the individual tabs.

In the individual tabs, the normal functionality of the Ensemble processing sequence applies. Individual processes can be added, removed, temporarily turned off, moved up or down the sequence, etc. Whatever ensemble is chosen, it will be applied only to the set of traces that correspond to that particular shot. By default, six individual tabs are displayed but only those having active shots, determined by the number of plots selected by the user, are in effect.

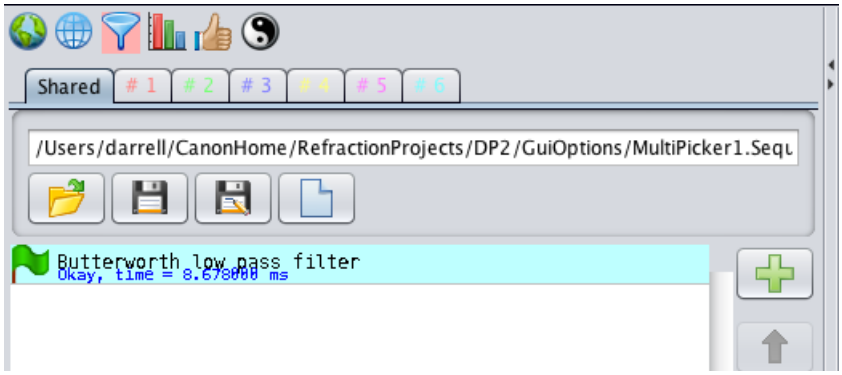


Figure 6---34 Ensemble processing sequence icon tab in the MultiPicker window.

6.6 New picker window – Multi---panel profile (offset plane)

This picker window is similar in functionality to the 'offset plane using 2D profiles' picker window except it is designed to display multiple profiles at the same time. Figure 6---35 shows an example of five profiles displayed at once.

Each panel is color---coded – the color of the vertical axis is matched to the color of the associated profile, displayed in the basemap view. In Figure 6---35, the one profile that is not active is the far right vertical profile, and it remains black.

The trace display can be modified using the Plot count tab in the Profile selection ( ) icon tab. Figure 6---36 shows this tab and the options available. The number of plots, 1---6 (default: 4) is selected from a drop---down list. The other two options are the number of header plots, None---4, of both text headers (purple) and plot headers (alternating yellow and blue) that are located above each set of traces in the trace display. In this example, both headers are turned off for all profiles, to give more space to the trace display.

Generally, multiple text and plot headers will not be used in the multi---panel profile mode. However, given that up to four of each are allowed in each panel, it's possible to exceed the available vertical minimum space on a computer screen. If that happens, the entire trace display will remain gray. Simple solutions to this problem are to maximize the window vertically, if possible, temporarily reducing the number of active profiles or temporarily reducing the number of plot headers (text header too, but they require less vertical space).

The **Azimuth selection** tab has the options for azimuth restrictions, including setting the **Direction** and **Acceptance**. These options were covered in Section 6.3. Whatever options are set apply to all active profiles. Similarly, the Profile creation tab is used to create single and multiple profiles. This functionality was also covered in Section 6.3.

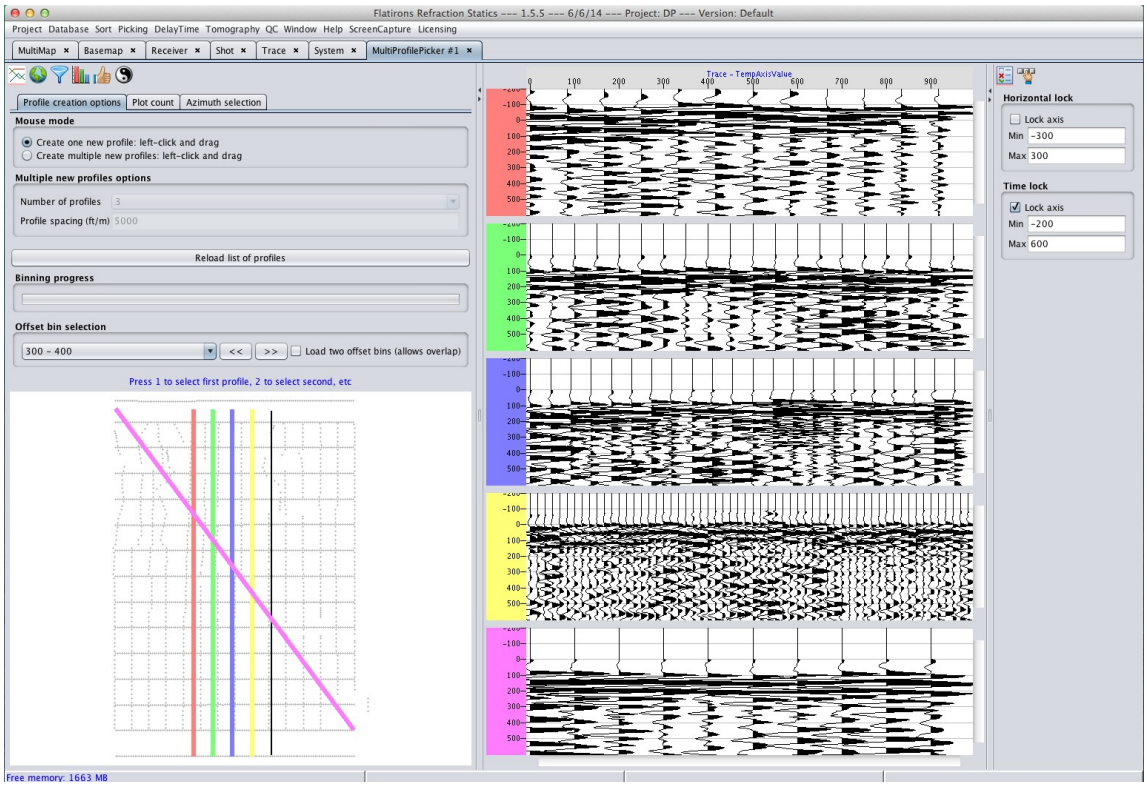


Figure 6---35 The Multi---panel profile (offset plane) picker window. Profiles have already been selected and the panels in the trace display are populated. Note the color---coding for each profile.

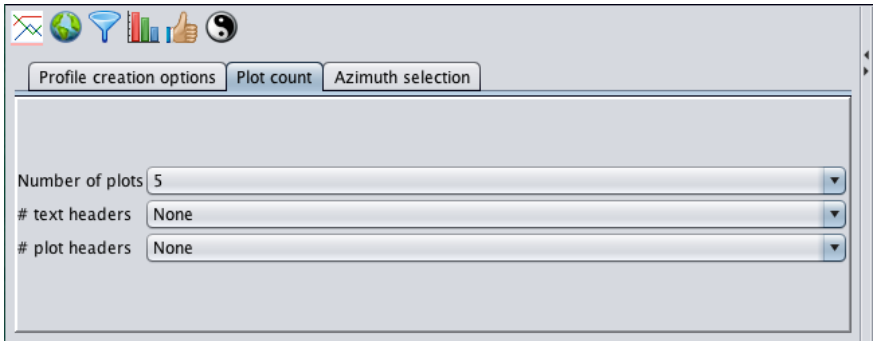


Figure 6---36 The Plot count tab in the Profile selection icon tab.

The procedure for selecting the active profiles is straightforward. On the basemap display in the **Profile selection** icon tab, move the mouse over the first desired profile and press the '1' key. That profile will instantly change color to 'red', the color allocated for the first profile. Repeat for the second profile, pressing the '2' key and the profile will change color to 'green'. Continue until all panels are assigned to a profile. Each panel must be assigned a different active profile. To change the ordering, simply move the mouse over a profile and select a different number key.

To navigate through the offset bins there is a drop---down list above the basemap display to select a specific bin (Figure 6---37). Buttons also permit moving forward and backward

through the bins. Finally, there is an option to load and display two offset bins at the same time.

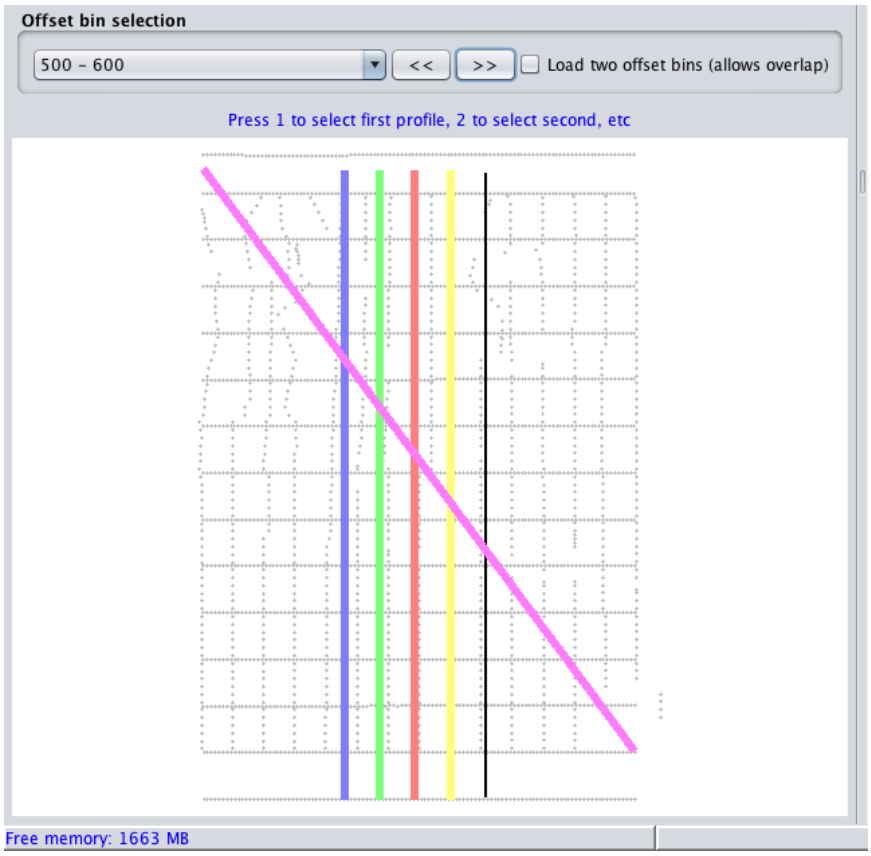


Figure 6---37 Offset bin selection in the Profile selection icon tab.

The **Ensemble processing sequence** ( ) icon tab for the Multi---panel profile window is generalized from its normal appearance. Because there are multiple profiles, the user may want to apply different sequences to different profiles. Figure 6---38 shows the upper portion of the Ensemble processing sequence window. Notice the separate numbered tabs and as well as the default **Shared** tab. Naturally, the label and the color correspond to the specific profile displayed in the panels. The **Shared** tab holds any processes that the user wants to apply to all profiles. Also, any processes in the **Shared** tab will be applied before processes in the individual tabs.

In the individual tabs, the normal functionality of the Ensemble processing sequence applies. Individual processes can be added, removed, temporarily turned off, moved up or down the sequence, etc. Whatever ensemble is chosen, it will be applied only to the set of traces that correspond to that particular profile. By default, six individual tabs are displayed but only those having active profiles assigned by the user are in effect.

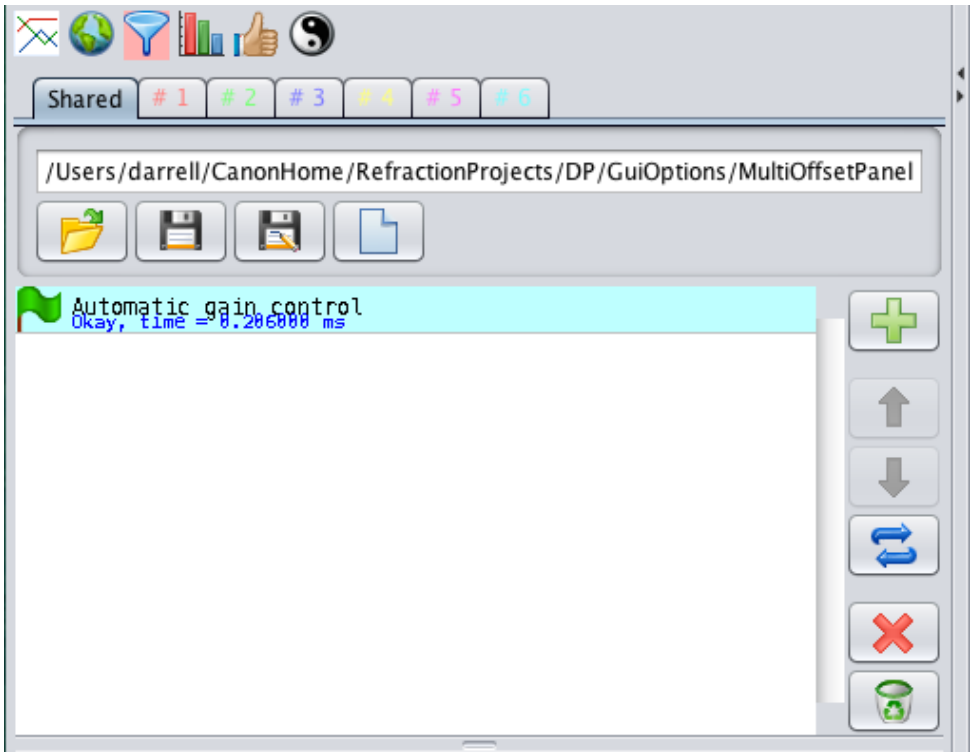


Figure 6---38 Ensemble processing sequence in the Multi---panel profile window.

Another minor difference from the normal picker window displays is in the **Horizontal axis selection** (). If the Sort option is checked, normally there no axis displayed because the sorted traces (by whatever Column was selected) are spaced evenly. In the Multi---panel profile window the traces, in every panel, are still sorted by the Column the user selected. However, the axis is labeled **Trace – TempAxisValue**, and the scale is 0---1000. This is simply a reminder that every panel may have a different scaling. Of course, if the user selects **Use trace header for axis**, the actual Column value is used for plotting the horizontal axis, as usual.

6.7 Define linear moveout trend using traces

One of the processes available for the Ensemble processing sequence is '**Apply moveout trend**' as shown in Figure 6---39. What is this trend and where did it come from? Before we answer this question, notice in the figure the menu option '**Apply linear moveout**'. This option allows the user to specify a linear moveout correction to all traces in the survey based on a single velocity. For a more sophisticated, spatially variable velocity, Flatirons allows the user to define a trend using traces. (In next menu item, covered below, picks are used to compute this trend.) This trend is then used to perform the linear moveout correction. Where it comes from is the **Picking → Define linear moveout trend using traces** menu item. Selecting this item opens a separate '**Linear moveout trend**' window, shown in Figure 6---40.

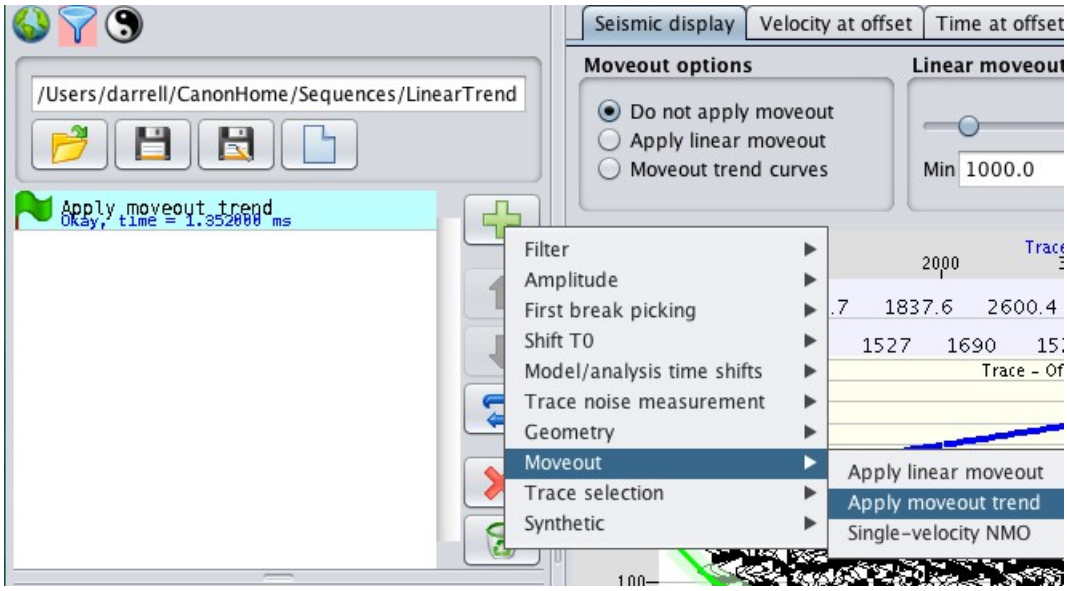


Figure 6---39 The 'Apply moveout trend' process option added to the Ensemble processing sequence. Never add both 'Apply linear moveout' and 'Apply moveout trend' to the same sequence!

On the basemap, the light red circles indicate the sparse CMP gathers that were generated when the project was created. See Chapter 1 for details. If the option to generate sparse CMPs at the time of project creation was declined, they can still be created at any time in an opened project. This open is located in the **Sort → Create sparse CMP gathers**. See Chapter 1 for details. However, using precomputed sparse CMP gathers is not required. A checkbox turns off this option. The current CMP is displayed in a slightly larger bright red circle. For CMPs in which the moveout trend as been defined, the circles are green.

The procedure for defining the moveout trend on a particular CMP is straightforward. In the trace display, a 'live' yellow line connects the (0,0) position with the current mouse position. On the left---most trace, the user identifies an event and left---clicks to mark it. Progressing across the display the user tracks the same event, intermittently left---clicking. The yellow line connects the positions of the clicks to form a segmented line. See Figure 6---41 for an example. Here, the current trend line (yellow) is wildly inaccurate and exaggerated to show the progess of six separate clicks, including the left end of the line.

When the user is finished, a double left---click set the trend line and turns it green, as shown in Figure 6---40. If the resulting trend line is unacceptable, simply repeat the process. After the moveout trend has been computed based two CMP trend lines, subsequent trace displays will show the interpolated moveout trend at that CMP. The trend will appear as a dashed yellow line as shown in Figure 6---42. And green band can be used as a visual guide. It will have more use when using picks to define a moveout trend.

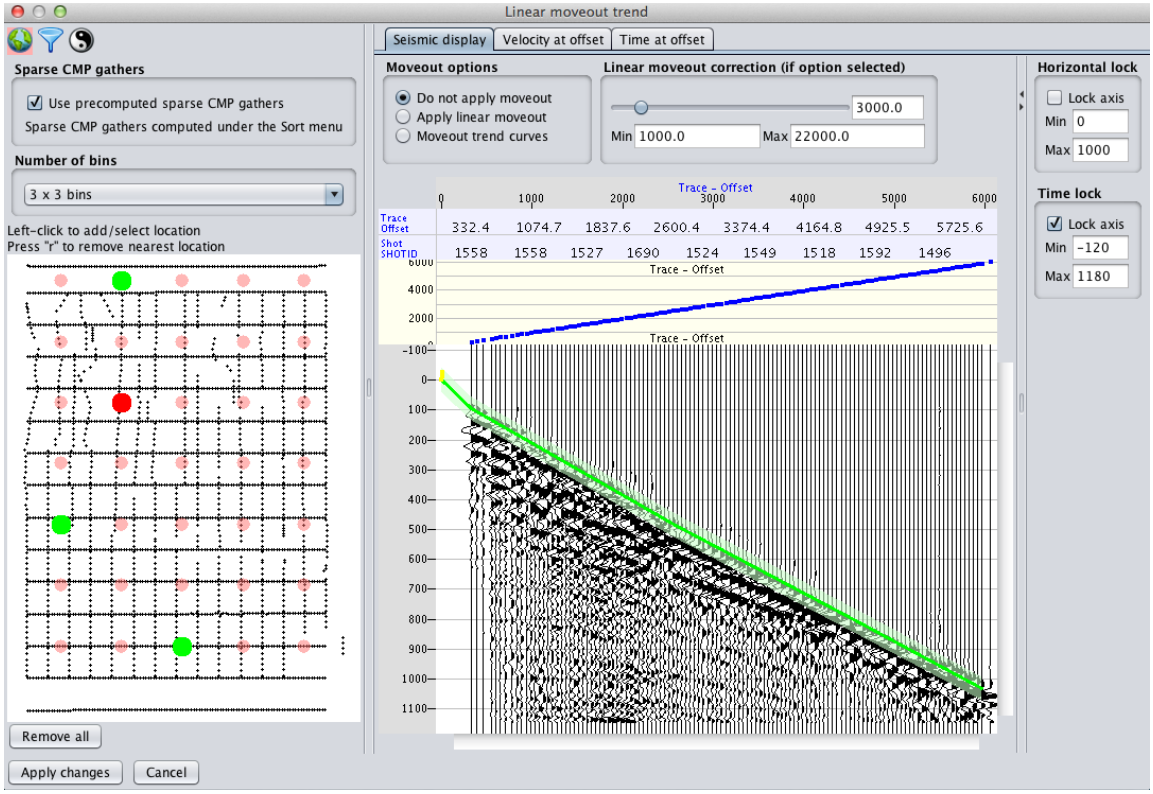


Figure 6---40 The Linear moveout trend window.

As mentioned above, a trend line in a particular CMP can be re---done by repeating the process of defining the line. But what if a user wants to remove a trend line? There are two ways. First, the **Remove all** button deletes all the trend line data, for every CMP. Second, pressing the 'r' key when the mouse is over the basemap will remove the trend line calculation for the one CMP closest to the mouse pointer (the dot will change from green to light red).

The number of CMPs that need to be processed is up to the user. By randomly selecting CMPs the user can check how well the survey---wide trend fits the data. Once the user is satisfied with the trend, clicking the **Apply changes** button will set a moveout trend for the survey and close the window. To see the effect of the trend, go to any window that has the **Ensemble processing sequence** ( ) icon tab available and add the **Apply moveout trend** item, as described in Figure 6---39 at the beginning of this section.

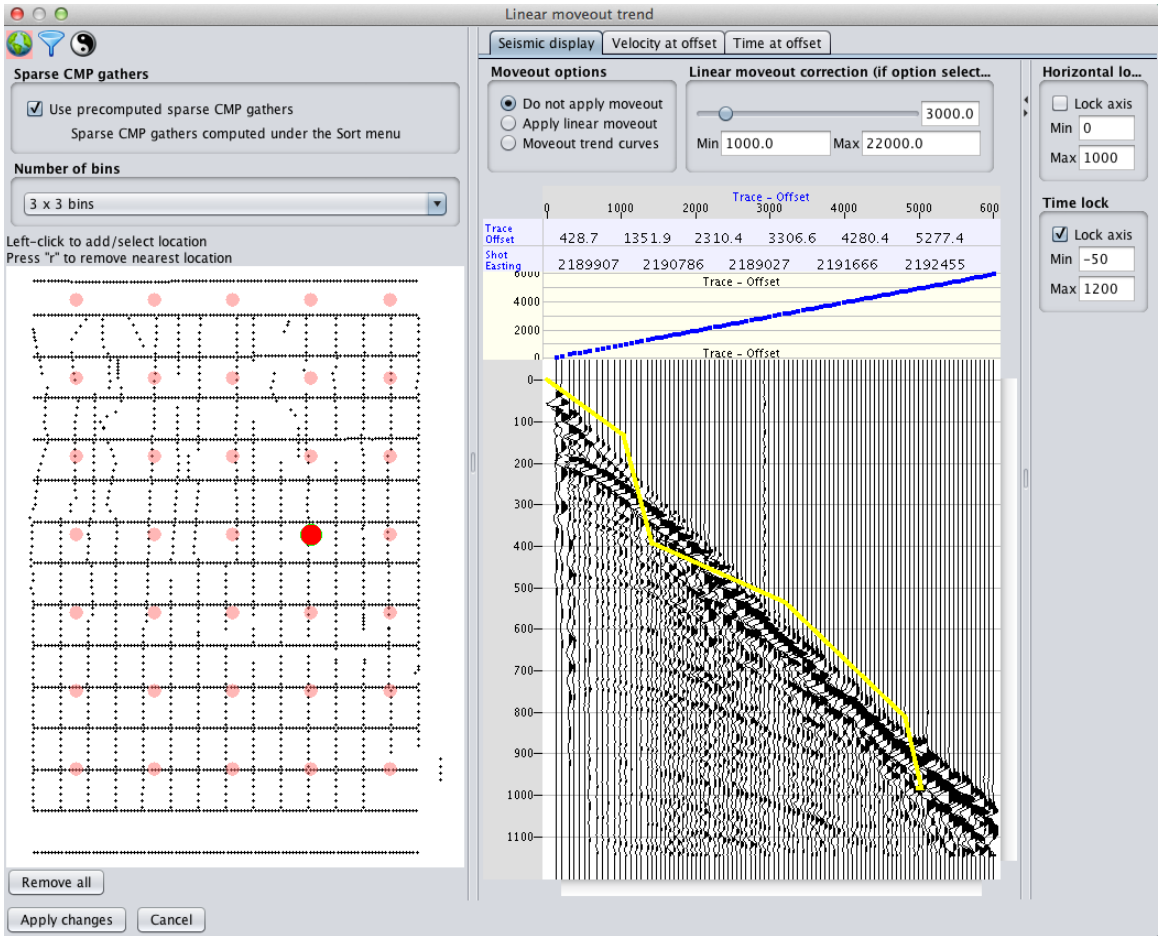


Figure 6---41 Defining a moveout treand with left mouse clicks, moving left to right. The obviously inaccurate trend line (yellow) is exaggerated to show six separate clicks. A double left---click will set the trend for this CMP and turn the line green.

Figure 6---43 shows a close---up of the top of the center pane, above the trace display. The default tab is **Seismic display**, where options for applying moveout are set. Earlier in this section, all the examples used the default option of **Do not apply moveout**. However, it may be preferable to apply a simple linear moveout with the **Apply linear moveout** option and then selecting a velocity using the slider, until the desired event is approximately horizontal.

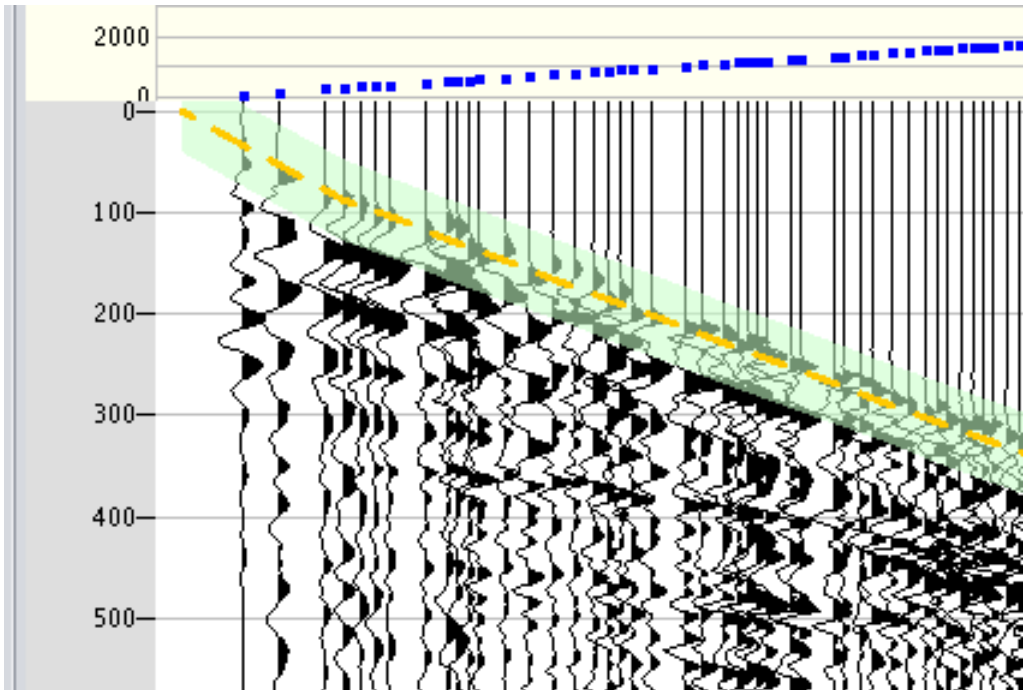


Figure 6---42 Close---up of the trace display showing the current interpolated trend line (dashed yellow) and a +/--- error band (green).

Figure 6---44 displays part of the **Linear moveout window**, where linear moveout has been applied and the user has defined a moveout trend line. Notice that the vertical axis has been locked. So applying the linear moveout effectively permits the view to be zoomed in to make identifying the event to trend on somewhat easier.

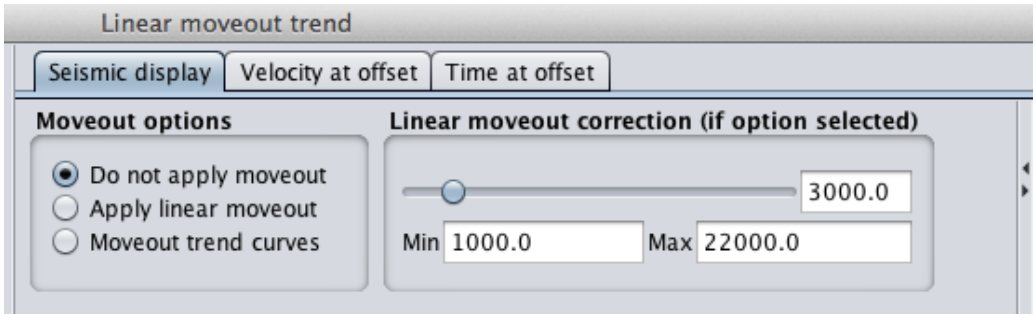


Figure 6---43 Top section of center pane above trace display. There are options for applying moveout and displaying velocity or time at a given offset.

To see how effective the Moveout trend is, the user can select a CMP that has not had a trend line defined and check the **Moveout trend curves** option. This will apply the moveout trend that has been interpolated from all the other trend---defined CMPs.

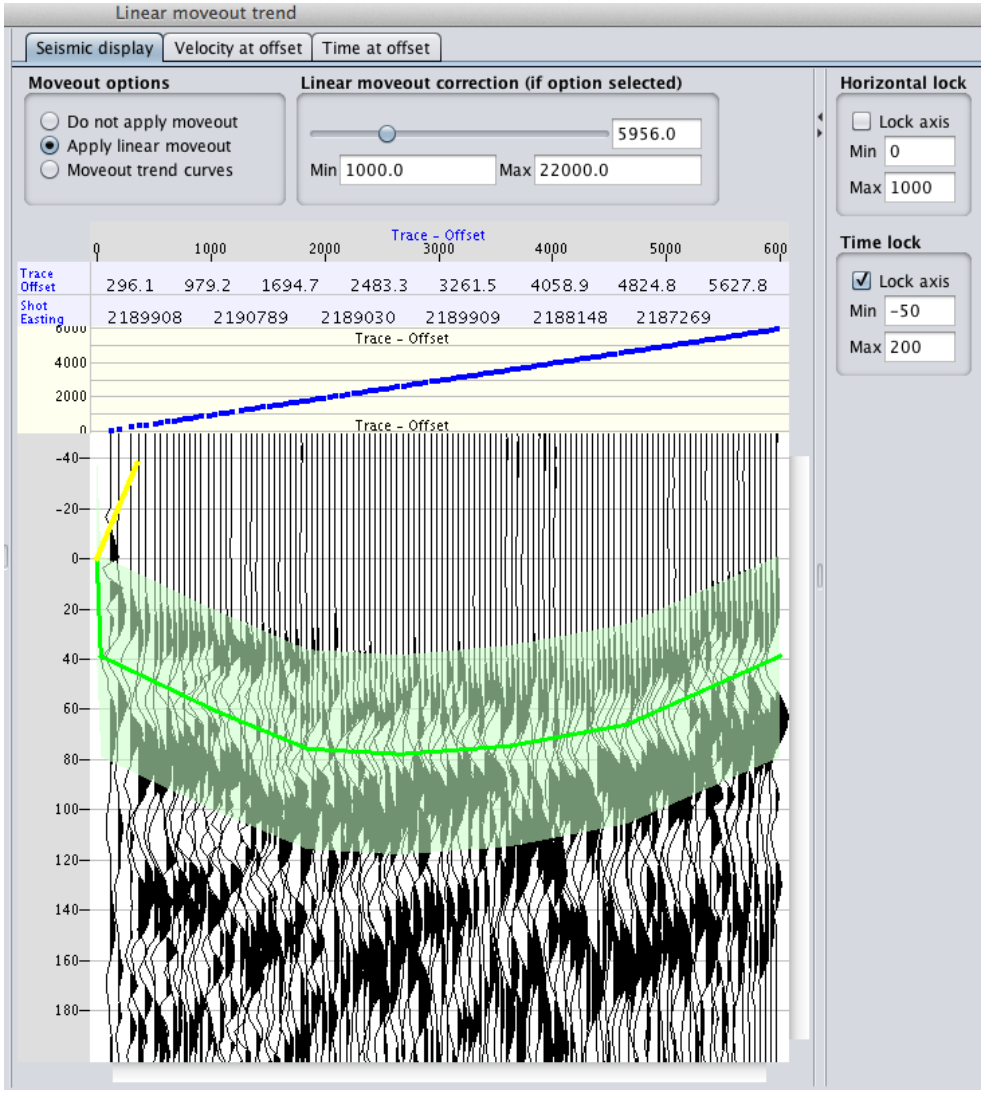


Figure 6---44 Defining a moveout trend with simple linear moveout applied. The slider was used to select a reasonable velocity.

Figure 6---45 shows the difference between a simple linear moveout and a user---defined moveout trend. The moveout trend for this particular CMP was not directly defined by the user but instead is interpolated from a moveout trend defined at other CMPs.

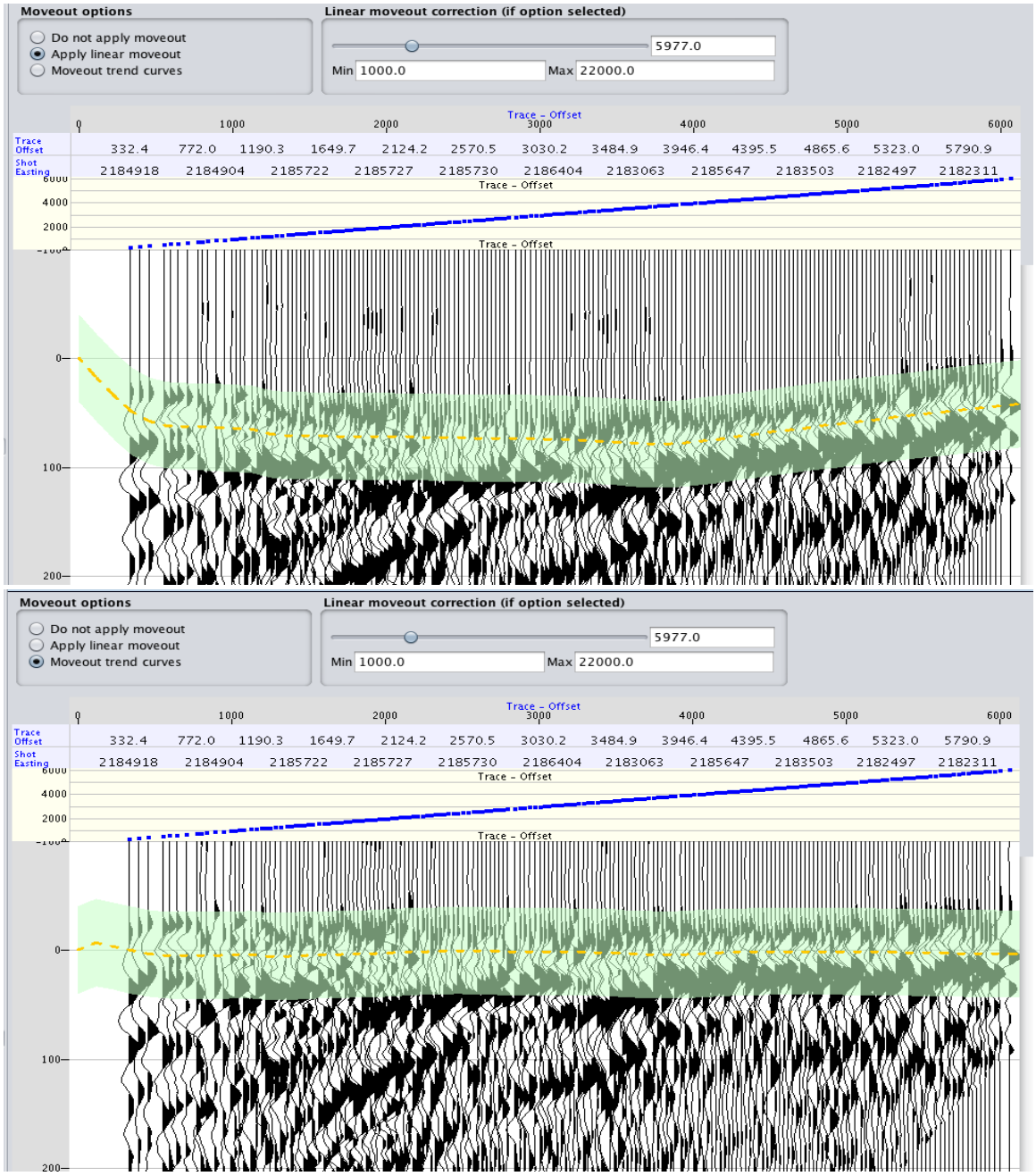


Figure 6---45 Example of the difference between a simple linear moveout (top) and a user define moveout trend (bottom).

The second tab (**Velocity at offset**) and third tab (**Time at offset**) in Figure 6---43 provide a quick visual quality control check of the current defined moveout trend. Figure 6---46 shows an example of the velocity at a specified offset (3400 ft). This is the velocity field of the moveout trend defined by the user. It will change considerably in appearance as more CMPs are added to the trend computation. Similarly the image will change considerably with offset. The QC aspect should really focus on the range of the velocities being plotted, which in this example, is not large.

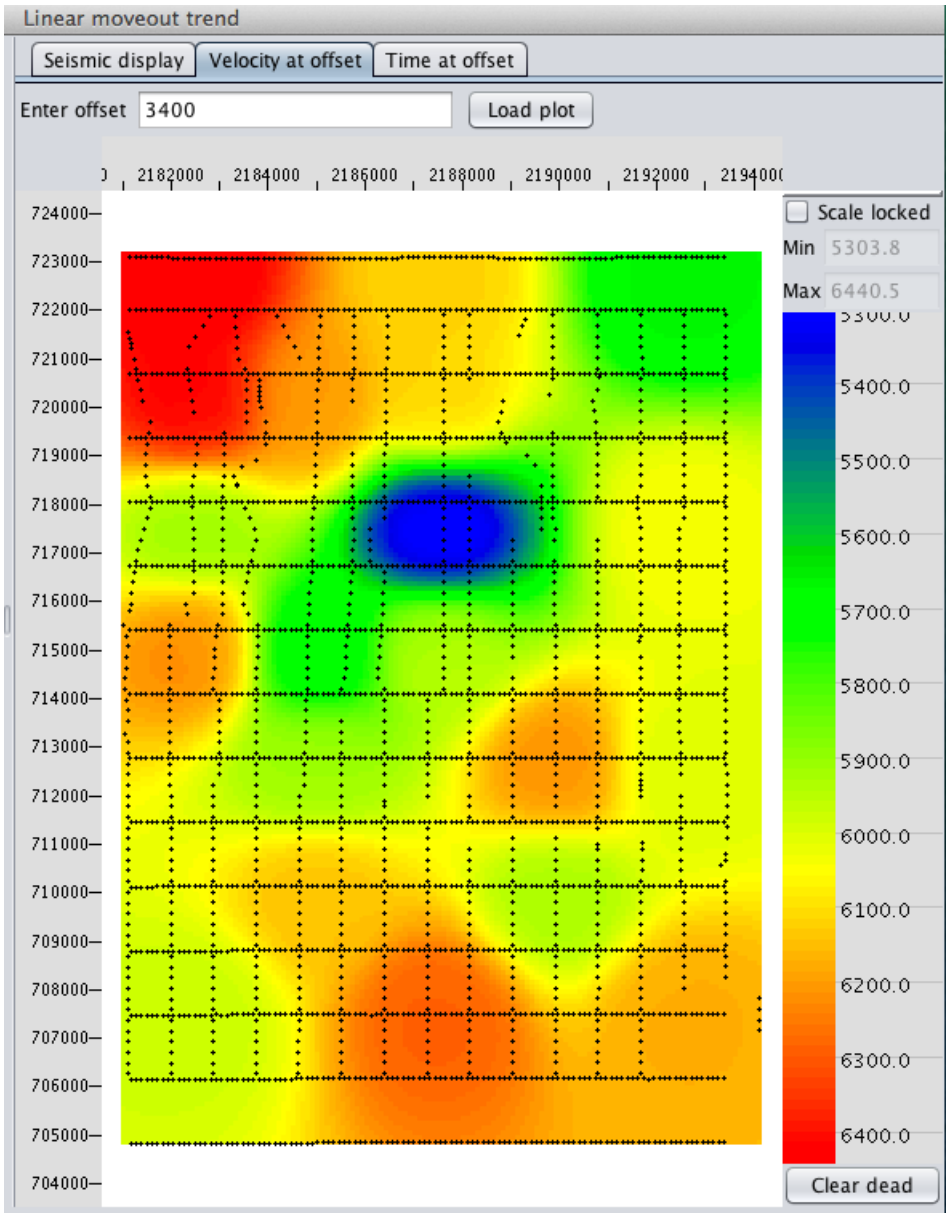


Figure 6---46 Velocity at offset QC check. The image displays the velocity of the user---defined moveout trend at a specified offset.

Figure 6---47 shows an example of the time at a specified offset (3400 ft). This is the time of the event, at the specified offset, chosen by the user when drawing the trend line. Of course, the image is an interpolation over all the defined CMPs. Like the velocity map, it will change considerably in appearance as more CMPs are added to the trend computation. Similarly the image will change considerably with offset. The QC aspect should really focus on the range of the times being plotted and a reasonable increase in times with offset.

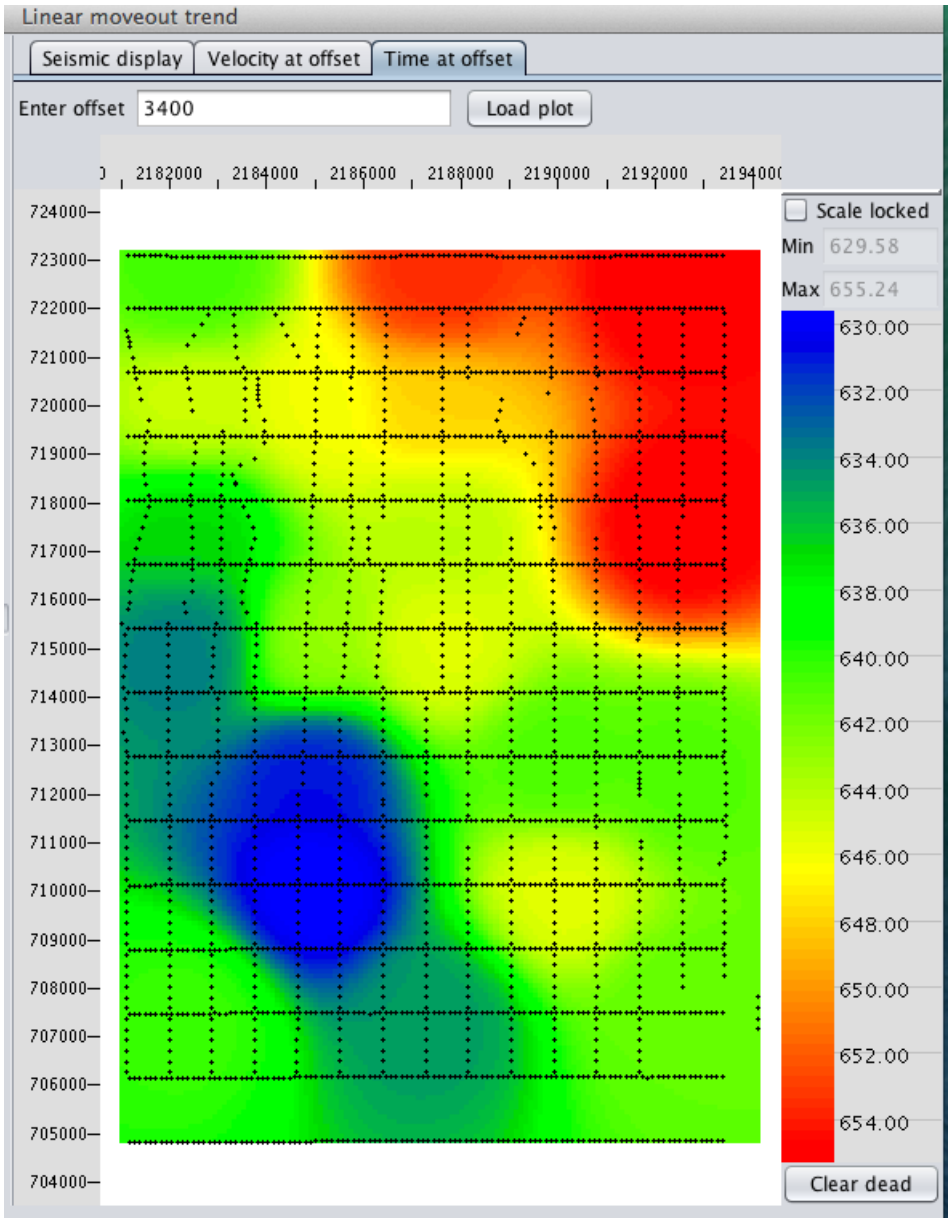


Figure 6---47 Time at offset QC check. The image displays the time where user---defined trend line intersects the specified offset.

As usual for basemap displays, both the velocity at offset and time at offset images can be zoomed by pressing the right mouse button and dragging out a rectangular region. This process can be repeated to continue zooming. A single right click returns the image back to normal view.

6.8 Define linear moveout trend using picks

If picks have been made, another option for defining a moveout trend becomes available. Selecting the **Picking → Define linear moveout trend using traces** menu item will open the window shown in Figure 6---48, except one pick gather has already been selected on the basemap in order to populate the display window with picks. Note that in this example, pre---

computed pick gathers are not being used. The procedure for defining a trend is the same as described in the previous section, except for additional options that will be described presently.

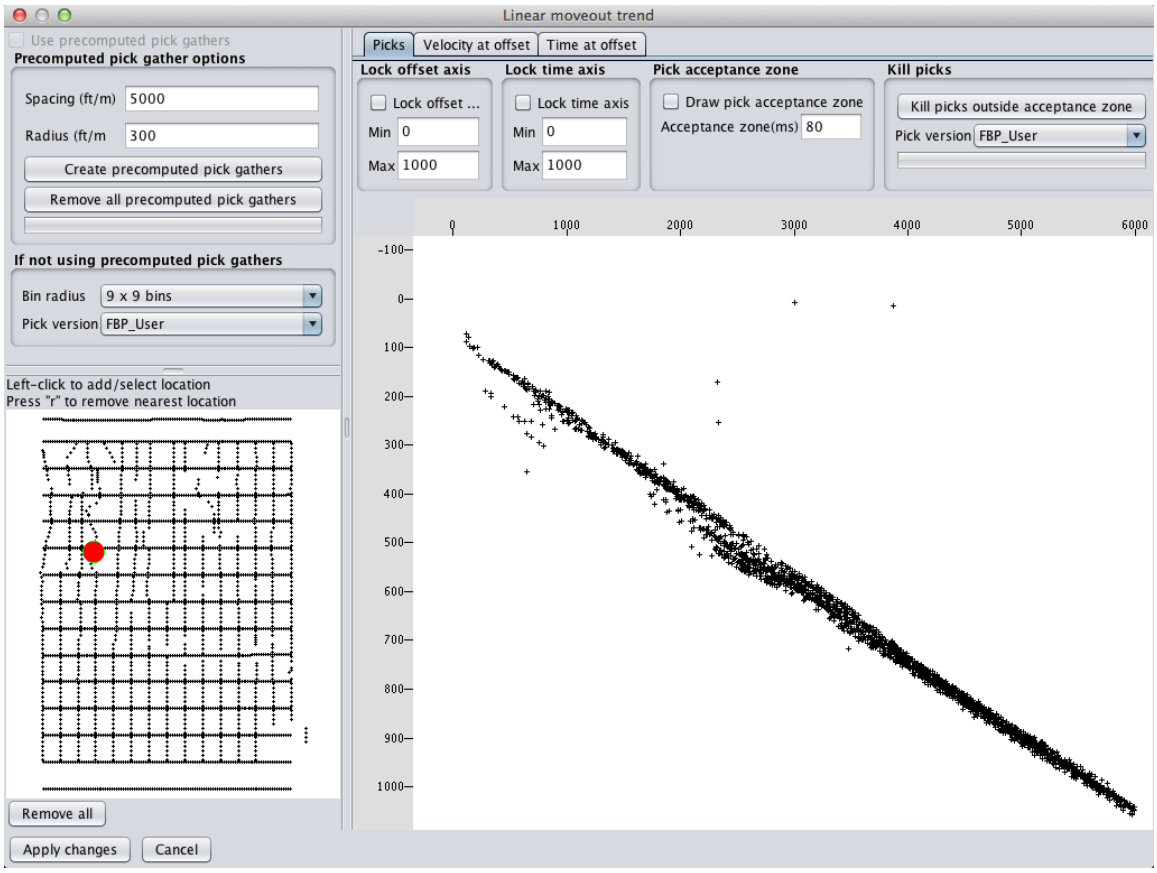


Figure 6---48 The opening window from selecting 'Define linear moveout trend using traces' in the Picking menu. A first CMP has been selected on the basemap to fill the display window with picks.

Figure 6---49 shows a trend line defined for a fourth pick gather. The green band is an acceptance zone. Its width (vertically, in milliseconds) is user---defined and can be toggled on or off using the **Pick acceptance zone** section, highlighted by the red box (left) in the figure. The purpose of the acceptance zone is that it can be used to kill (remove) picks that the user deems unacceptable.

To kill picks, the user first selects which pick version to apply, from a drop---down list. Then the user clicks the **Kill picks outside acceptance zone** button. These options are highlighted by the blue box (right) in Figure 6---49. This button does just what is says: any picks, across the entire survey, that lie outside the acceptance zone are marked killed in the database and not used in computations. So it is a good idea to bounce around the survey and be sure to choose a useful pick acceptance zone.

After the **kill picks**... button is clicked, a pop---up window requests confirmation of the kill action. Once the database is updated another pop---window informs the user that the changes won't be visible until another pick gather has been made.

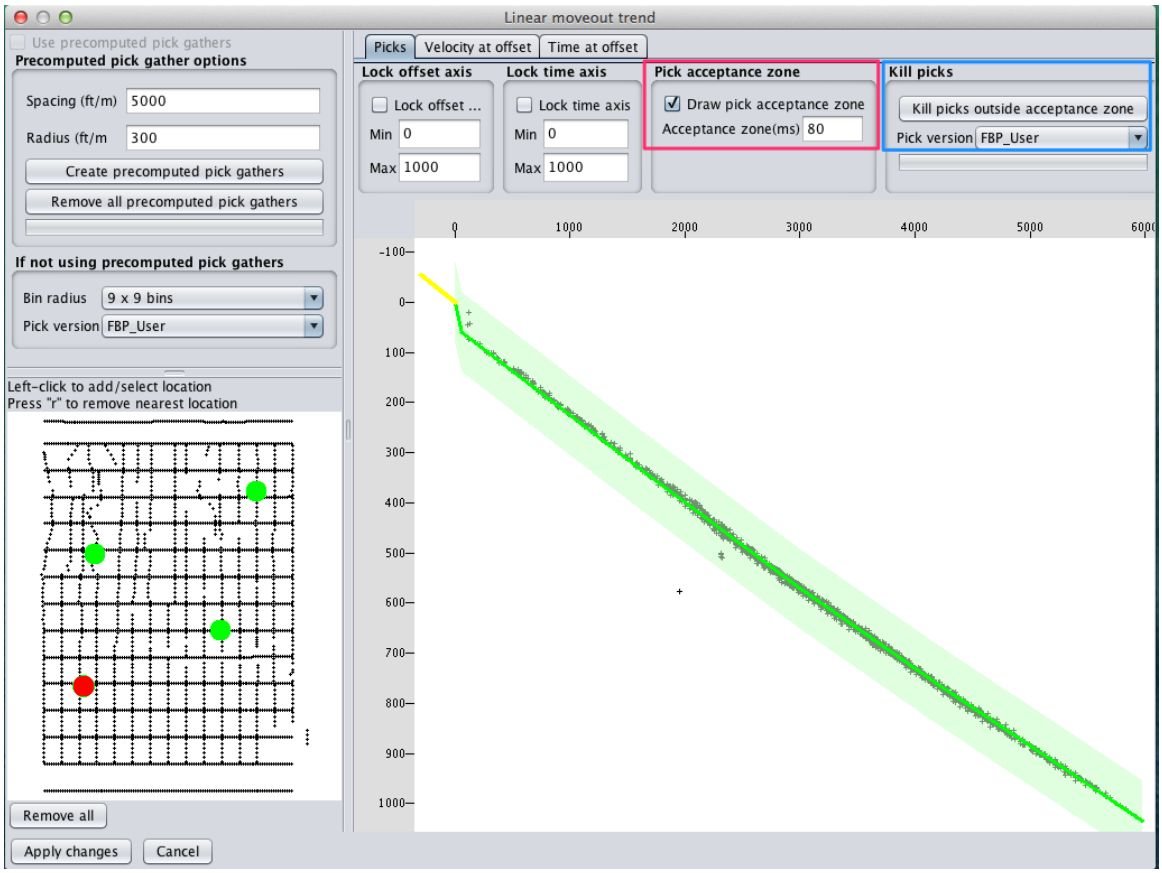


Figure 6---49 Trend line defined for fourth CMP. Red box (left) has the pick acceptance zone options, while the blue box (right) has the kill picks options.

The **Linear moveout trend** for picks window has **Velocity at offset** and **Time at offset** tabs (Figure 6---49), just as the window for traces does. Their function is the same; see the previous section for details. After completion of defining the trend and killing picks, the user clicks the **Apply changes** button to finish. Doing so will cause this window to immediately close and return control back to the main Flatirons window.

6.9 General shot/receiver stacks

An alternative method of picking is to stack shots and receivers and pick the stacks rather than individual traces. In some instances, with particularly noisy data, this may be the best route to obtaining reasonable picks. Even if some other method of picking was used first, the stack picking method can be used to improve those picks.

NOTE: A prerequisite for this method is that a linear moveout trend has been defined and applied.

To begin the process of stack picking, select the **Picking → General shot/receiver stacks** menu item. This will open the window Figure 6---52. Before picking, three steps must be completed. All three are set in the left panel of Figure 6---52. A close---up of this panel is shown in Figure 6---53. This panel has four icon tabs, with the default being the **Create new stacks** ( ) tab. See Section 6.1 for descriptions of the other icon tabs. One variation is that the

Wiggle display options icon ( ) has an additional section called **Select pick color**. It is very simple, as Figure 6---50 indicates. Clicking on the blue rectangle opens a dialog box, named **Pick a color**, based on Java's JColorChooser class (Figure 6---51).

Figure 6---50 Wiggle display options, highlighting the Select pick color option.

From this dialog box, the pick color can be chosen any number of ways. See Java's documentation for more details about the JColorChooser class. To return the pick color to the default blue, simply click the **Reset** button.

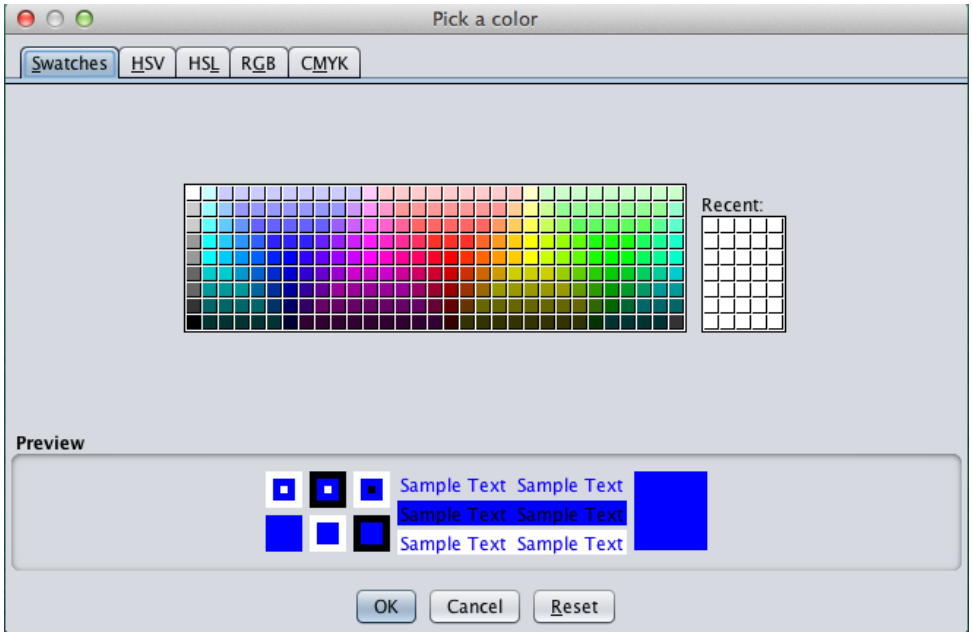


Figure 6---51 Dialog box for setting the pick color. Based on Java's JColorChooser class.

Returning now to the process of picking stacks, some information has to be provided so Flatirons can create the stacks.

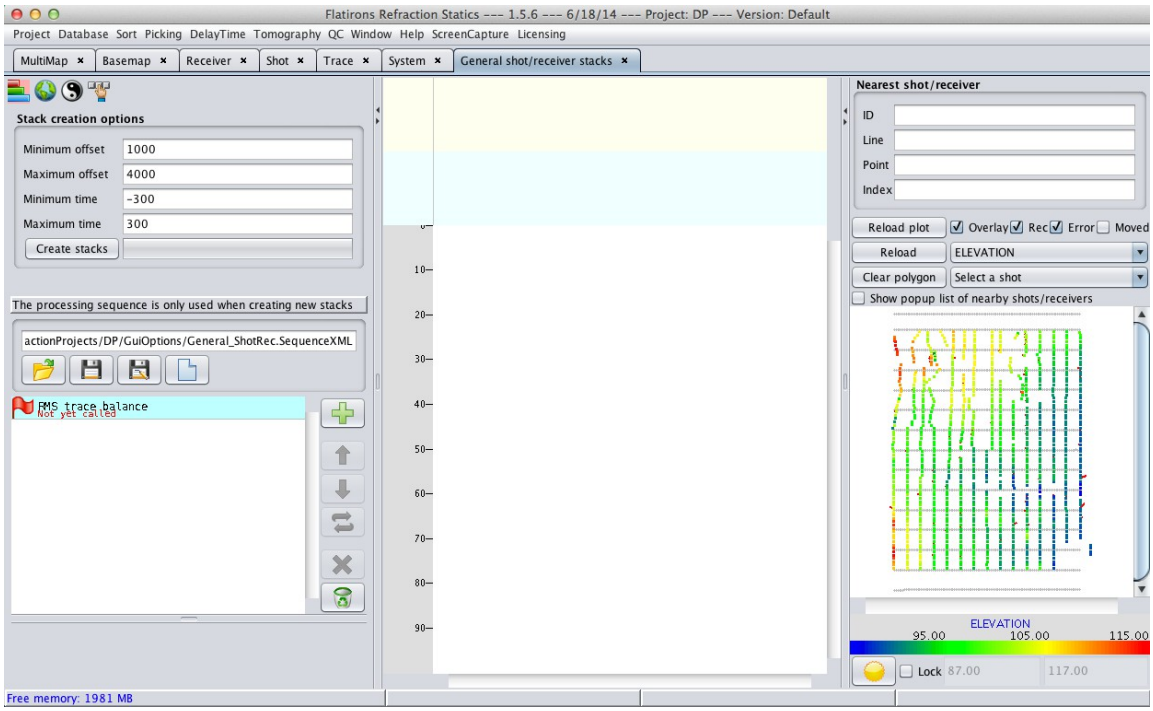


Figure 6---52 Default window from Picking → General shot/receiver stacks.

First, the **Stack creation options** need to be entered. These are the minimum/maximum offset and the minimum/maximum time. Generally, a narrow time window about zero time will give best results. Obviously, for stacking to be effective, moveout needs to be corrected. While it is possible to apply a simple linear moveout, such a constant moveout over the entire survey is unlikely. Better to go through the process of defining and applying a moveout trend. See the previous two sections for how to do that using either traces or picks, respectively.

The second step is to add the moveout trend to the ensemble processing sequence, as shown in Figure 6---53. Of course, other processes, such as filters, may also be added to the sequence. Note in the figure that underneath each process is the phrase “Not yet called” in red.

The third step is to create the shot/receiver stacks by clicking on the **Create stacks** button. A progress bar next to the button will show the progress of the stacking process. Once complete the window will appear something like that in Figure 6---54. The phrase underneath each process should now indicate “Okay” in blue, along with a time to apply that process.

Note that in the figure, the number of headers has been changed to one and **Shot – SharedPickTime** is being displayed. This column is where the picks from the stack picking method are stored. Since picking has not yet begun, all values are zero. Also, the header is

only updated when moving to the next group. So to see the current values of picks, move to the next group, and then return by moving to the previous group.

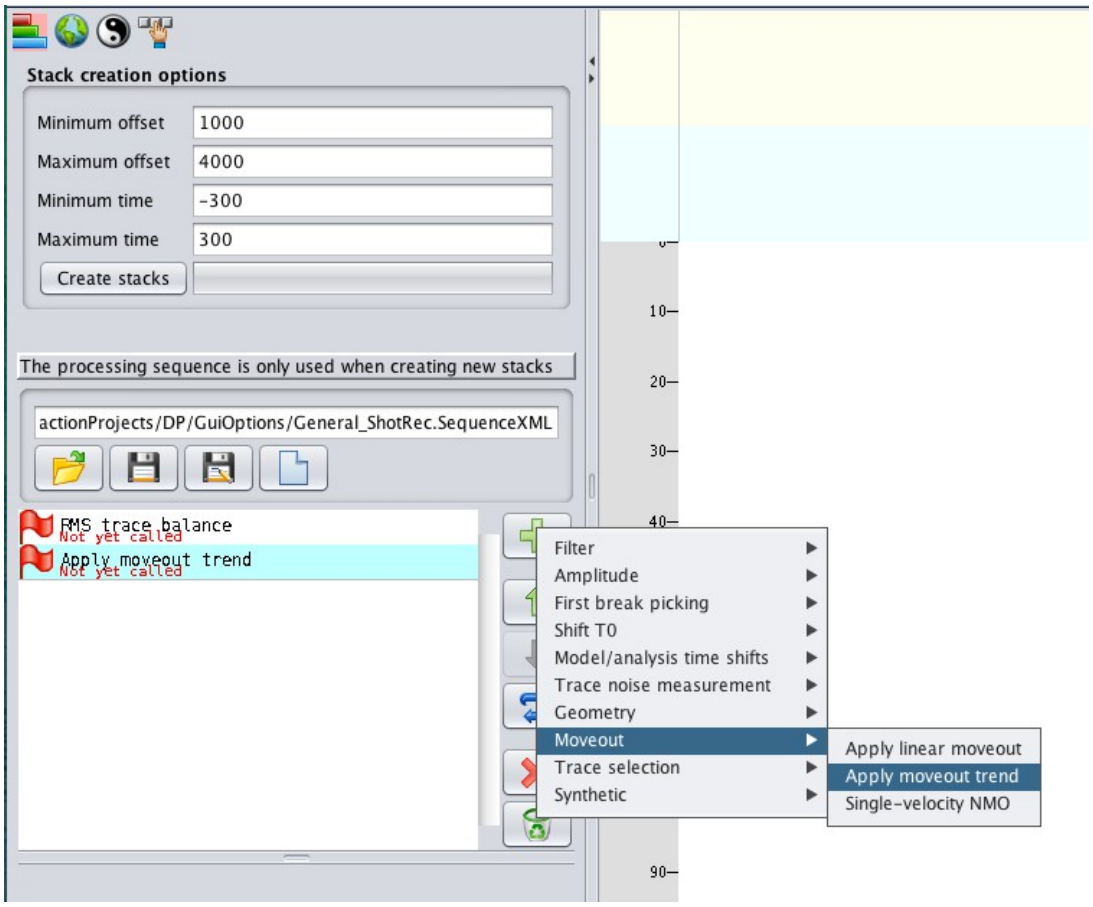


Figure 6---53 Close---up of left panel in the General shot/receiver stacks window. The menu shows the 'Apply moveout trend' process being added.

Notes: setting **Stack time modification percent** to 50 works better than 100. Apparently, the method attempts to shift the shots/receivers in such a way as to move the stacks to zero time. Picking a shot stack means at 30ms means what? The shot gets shifted --- 15ms and all receivers also ---15? But if mod % is set to 100 then shot is shifted ---30ms?
Unknown.

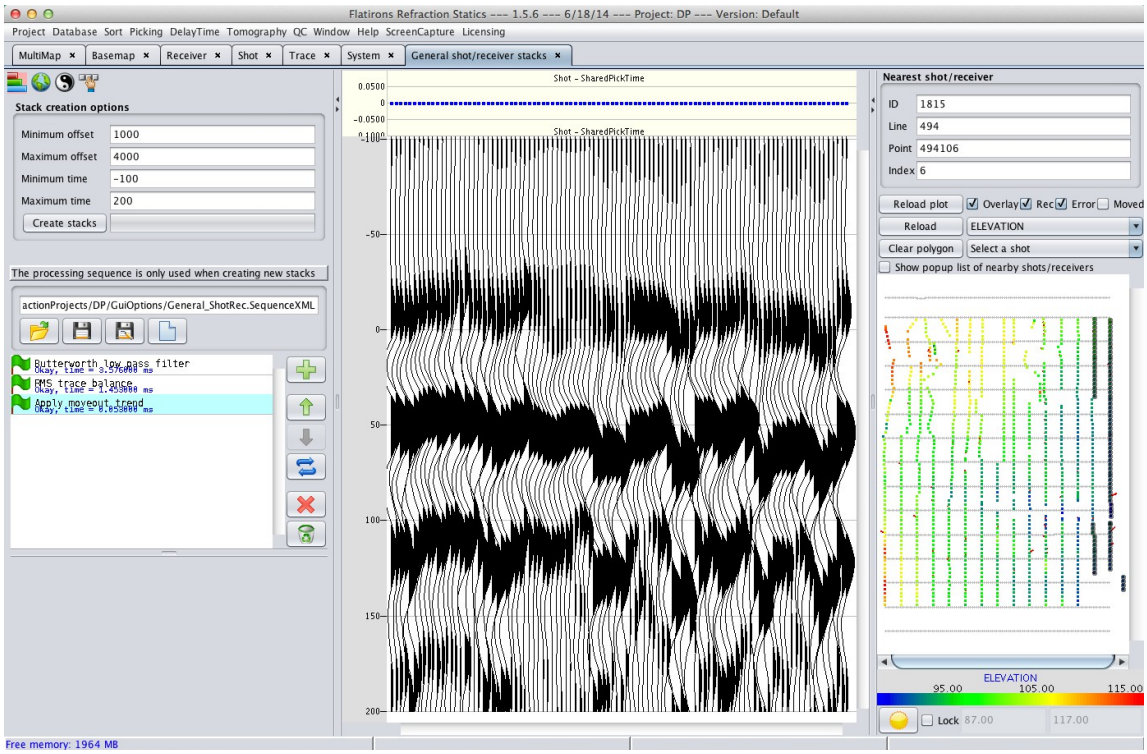


Figure 6--54 After the stacks have been created but before picking has started.

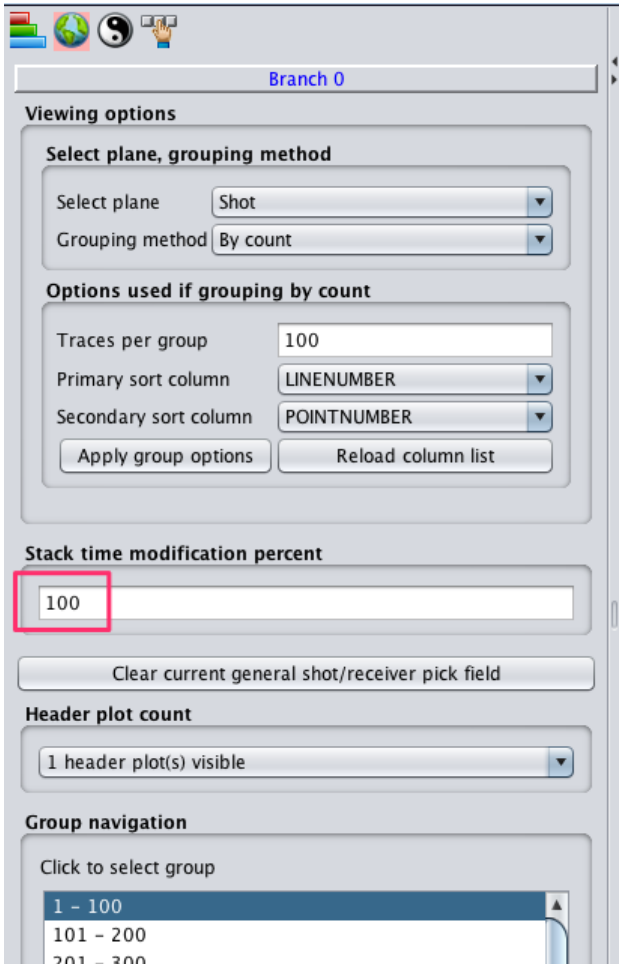


Figure 6---55 The Basemap icon tab in the left panel of the General shot/receiver stacks window. The Stack time modification percent ...[TBD]

6.10 Pick import/export

Flatirons provides to method for importing/exporting picks from ASCII files: either using coordinates or FFID/Channel. Figure 6---56 shows the **Picking → Pick Import/export** menu item and each option. Clicking **Import picks using coordinates** opens the dialog box shown in Figure 6---57.

The procedure is straightforward: select the pick column name to import into (typically **FBP_User**), choose whether to overwrite any existing valid picks and use the **File Dialog** icon ( ) to select the picks file.

It is possible to import picks into a new column. For this option, click the **Add pick version** button. A dialog box will open, as shown in Figure 6---57. Enter a **Pick name** of at least three characters. As the figure indicates, the column name will be prepended with '**FBP_**' to help the user identify First Break Picks columns. Be aware that the drop---down list used to select a column for import is not immediately updated with a newly defined column. In this situation, simply click the **Close** button (without importing) and repeat the import process

from the **Picking** menu. Alternatively, the user could import the picks into an existing column, go the Trace table, add a new column and apply column math to move the picks to the new column. Click on the **Create pick column** button to complete the process.

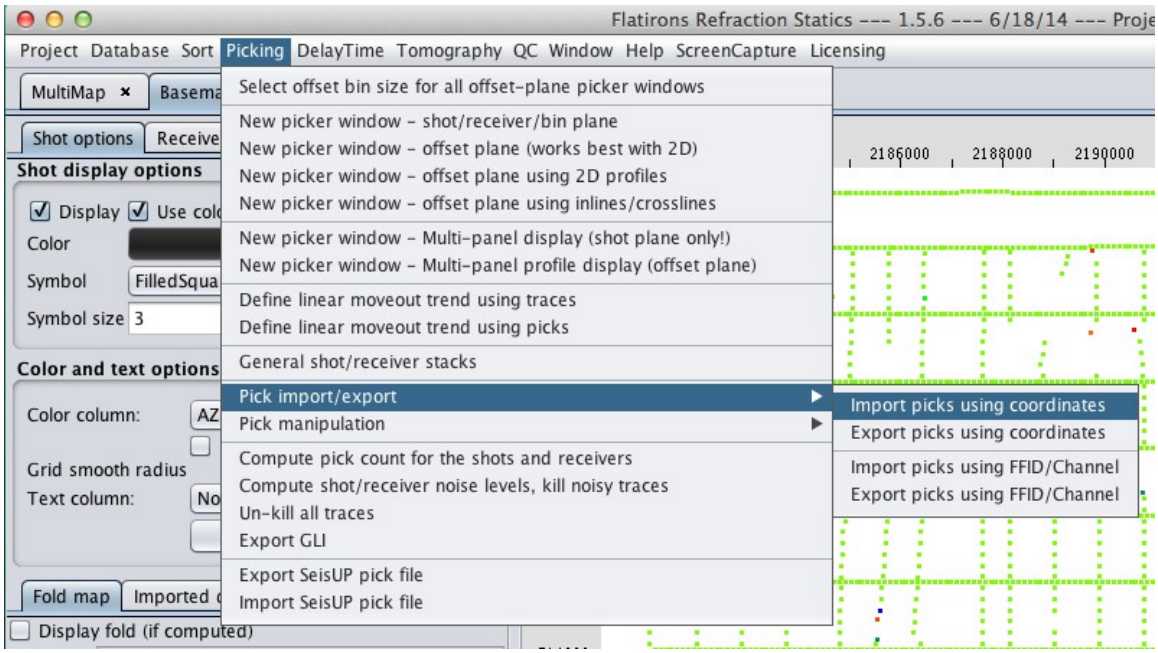


Figure 6---56 The Picking → Pick import/export menu item has options for import/export via coordinates or FFID/Channel.

The procedure for exporting picks is approximately the same. Clicking **Export picks using coordinates** opens a dialog box (not shown). In this case the **File Dialog** icon ( ) is used to find the file to export to or create a new file name. Be aware that the usual '.txt' extension is not automatically

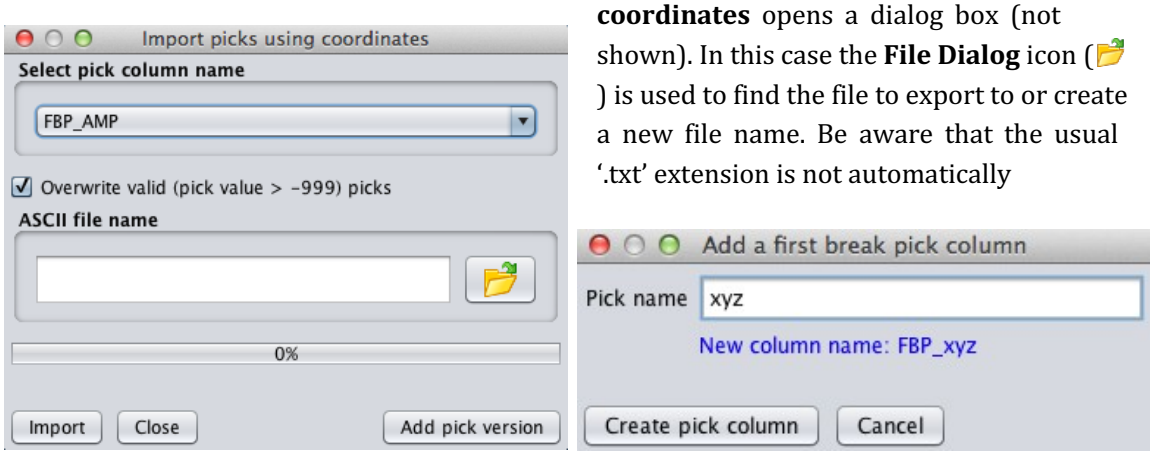


Figure 6---57 Import picks using coordinates pop---up dialog, on the left. Clicking on the Add pick version button opens the dialog on the right.

appended. Optionally, picks for dead shots and receivers can be included in the export. They have the default value of ---9999. Click the **Export** button to begin the export. A progress bar in the dialog box indicates the progress of the export. When complete, the dialog box closes.

The result of the export is a column---delimited ASCII file with five columns including: shot coordinates in the first two columns, receiver coordinates in the third and fourth column and pick times in the fifth column. The file has no column headings.

The other method for import/export of picks is using the FFID/Channel information and is accessed from the **Picking → Pick Import/export** menu item (Figure 6---56). The procedure is the same as import/export using coordinates. The only difference is that the ASCII file has three columns, in order: trace FFID, trace CHANNEL, trace pick time.

6.11 Pick manipulation

Under this general menu item (Figure 6---58) are several actions related to picks.

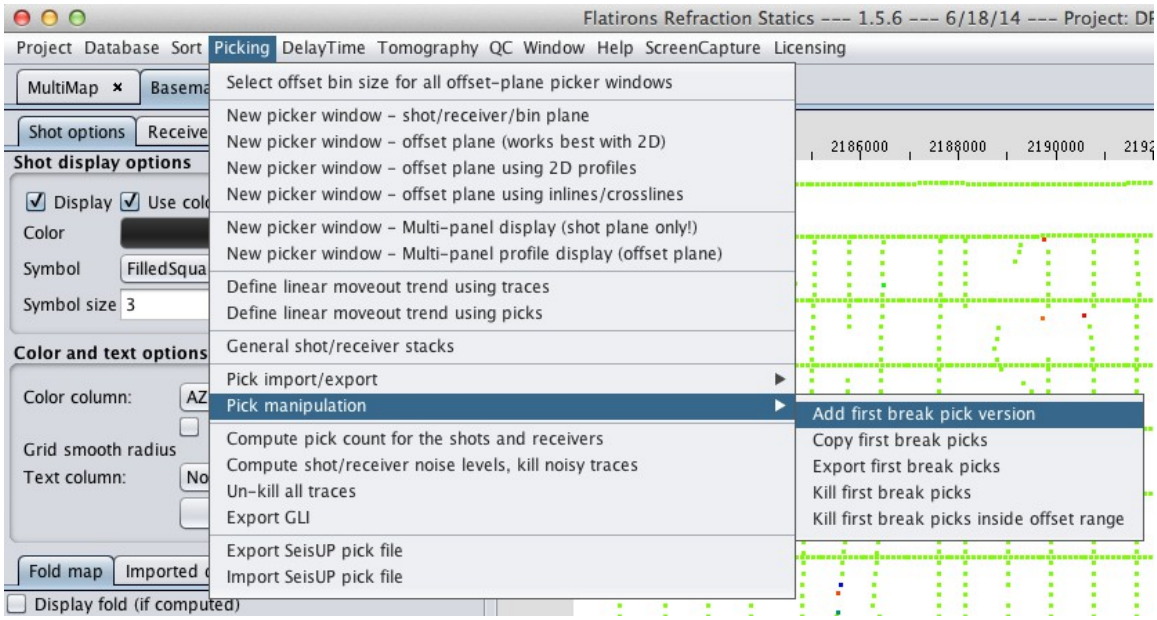


Figure 6---58 Options related to pick in the Picking → Pick manipulation menu.

6.11.1 Add first break pick version

This option is used to create a new Column in the Trace table for the purpose of storing another version of first break picks. It works the same way as creating a new pick version when importing picks, as described in the previous section. Choosing this option opens a dialog box (same as on the right in Figure 6---57). Whatever name is entered is automatically prepended with '**FBP_**' to identify the type of data in the column.

6.11.2 Copy first break picks

To copy one set or version of first break picks to another version, use this menu option. In effect, this procedure just copies data in one column of the Trace table to another column. However, as Figure 6---59 shows, the data that gets copied can be restricted and limited. Also, the copy procedure is not limited to first break pick columns, although that is the intended purpose. Limitations are toggled with check boxes and include copying only picks greater than zero, copy picks associated with traces assigned to a particular branch, or overwrite

valid pick times. This last limitation is TRUE by default, so be sure this is intended before proceeding! Clicking on the **Copy column** button will open a dialog box requiring the user to confirm the copy.

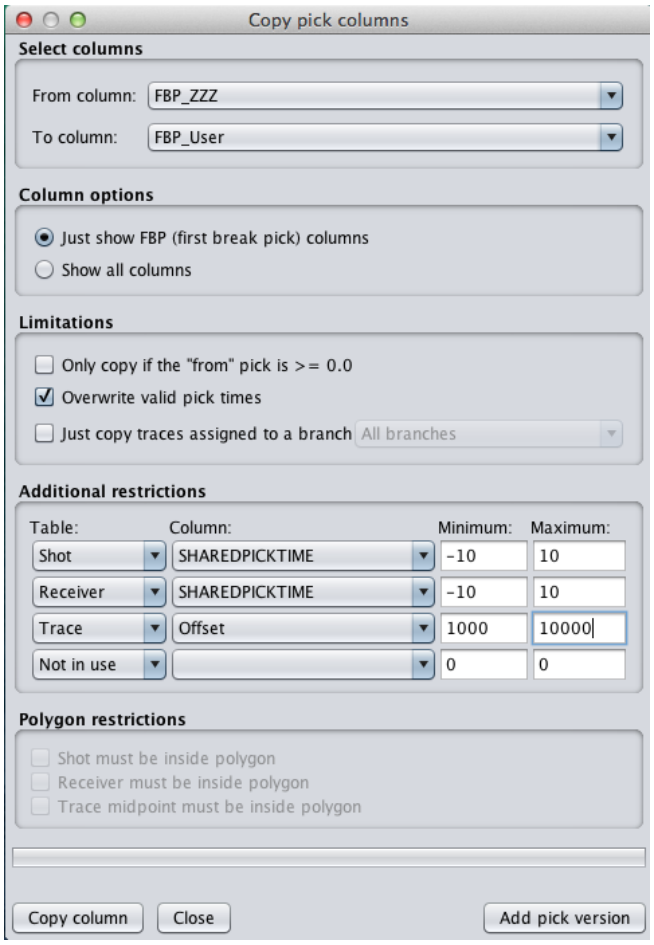


Figure 6---59 Dialog box for copying first break picks.

The restrictions comprise setting minimum/maximum values of columns in the Shot, Receiver or Trace tables. The **Polygon restrictions** option is only in effect if polygon the user has previously defined a polygon, typically in the Basemap tab. See, for example, Figure 14---7 and associated text. If there is a polygon defined, then options are available for copying only picks that have shots, receivers or midpoints within the polygon.

6.11.3 Export first break picks

This option opens up a dialog box (Figure 6---60) for the user to select which version of first break picks to export. The column is selected from a drop---down list and clicking on the **Export column** button completes the process. The result is an ASCII file containing a single column – the pick values, with no header. The name of the file is the column name with '_export.txt' appended. For example, in the example shown the created file is named 'FBP_ZZZ_export.txt'. The file is located in the TraceTable folder.

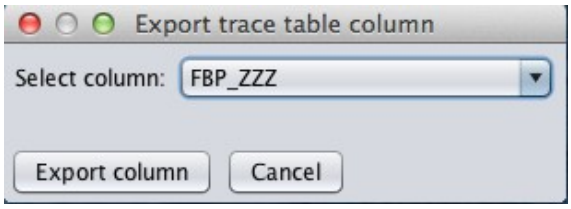


Figure 6---60 Dialog box for exporting picks.

6.11.4 Kill first break picks

Occasionally the need arises to delete some picks. These next two options in the Pick manipulation menu item do just that. The term 'kill picks' means to set the pick value to the default, nonsensical value of ---9999.0. The procedure for killing picks proceeds as follows. Following Figure 6---61, the user selects the version of first break picks (i.e., the Column in the Trace table) on which to operate. Optionally kill only picks that do not have associated traces assigned to a branch. If a polygon has been assigned, optionally apply desired restrictions. Then, apply **Column filter options**. These options are similar to those when copying picks, described above (see Figure 6---59). However, there are separate **Apply** checkboxes for each filter option that have to be checked to put the filter in effect.

Up to four separate filters can be applied, with min/max ranges set on columns in the shot, receiver or trace tables. In the example shown just one filter is used: all picks in version FBP_ZZZ associated with SHOTID = 1000 will be killed. Clicking on the **Kill picks** button will open a dialog box requiring the user to confirm the procedure.

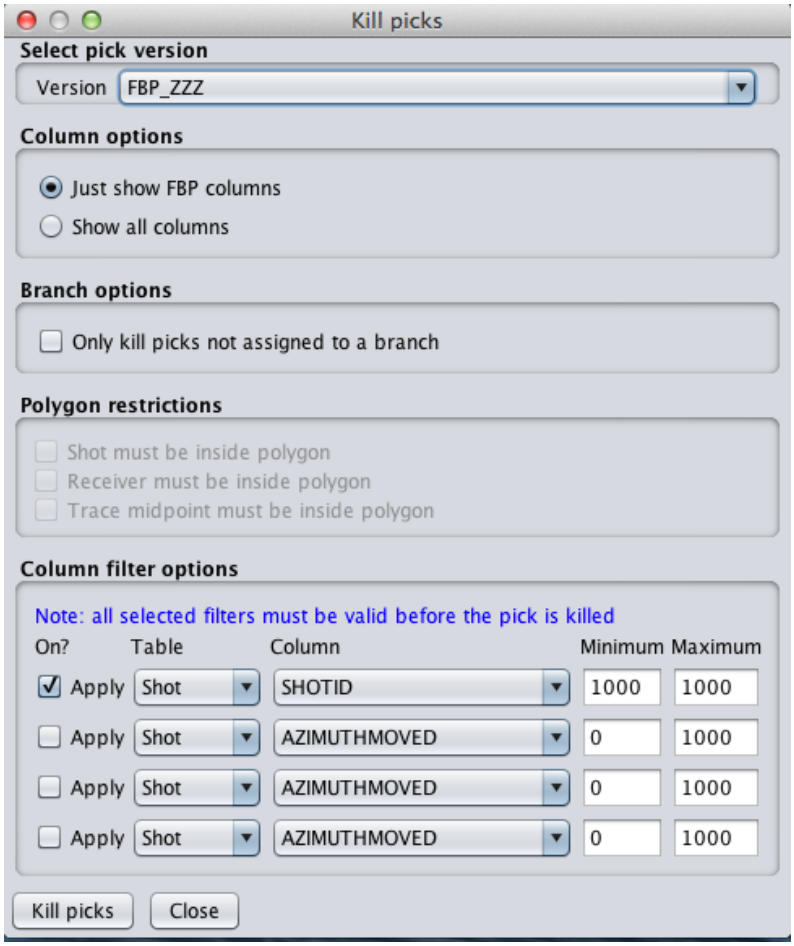


Figure 6---61 The Kill first break picks option from the Pick manipulation menu item.

6.11.5 Kill first break picks inside offset range

This option is also used to kill picks but the criteria are different and simpler. The only filter is the trace offset. Figure 6---62 shows the dialog box for this procedure. Aside from selecting the pick version, the only filter is sett by entering the min/max offset. Click the **Kill picks** button to proceed, and click the **OK** button on the subsequent **Confirm pick kill** dialog window.

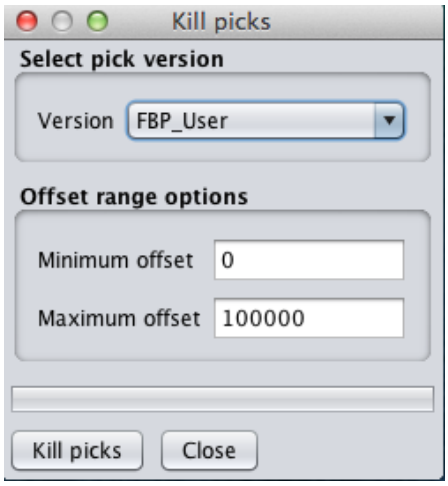
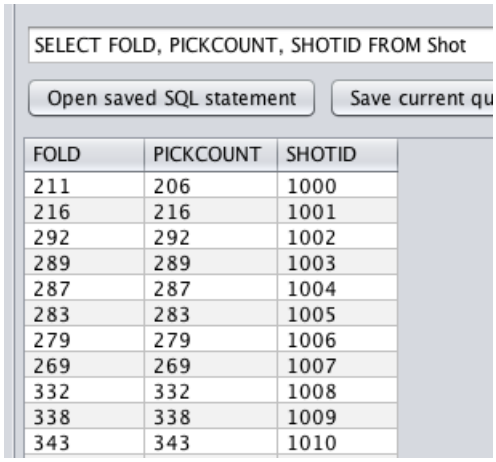


Figure 6---62 Dialog box for killing picks based on an offset range.

6.11.6 Compute pick count for shots and receivers

This option calculates the number of picks that have been made on the shots and receivers. The count is saved in the **PICKCOUNT** column for both Shot and Receiver tables. Regardless of how many pick versions exist (i.e., columns beginning with '**FBP**', either Flatirons default or user---added) the count is based on the **FBP_User** column, in the Trace table.

Generally, the PICKCOUNT will equal the FOLD, if every trace was picked. Figure 6---63 shows an example of this, displaying the result of a simple query on the Shot table. The exception is for SHOTID 1000, where five traces do not have picks. Likely they were killed when demonstrating the pick---killing options in previous sections of this chapter.



FOLD	PICKCOUNT	SHOTID
211	206	1000
216	216	1001
292	292	1002
289	289	1003
287	287	1004
283	283	1005
279	279	1006
269	269	1007
332	332	1008
338	338	1009
343	343	1010

Figure 6---63 Partial result of query on Shot table. If every trace associated with a given shot was picked, then PICKCOUNT will equal FOLD.

6.11.7 Compute shot/receiver noise levels, kill noisy traces

This option is used to compute noise levels on traces, and kill traces that are too noisy or have too much DC bias. Figure 6---64 shows the dialog box that opens upon choosing this option. The result of the computation is a noise level value, stored in a Column called NoiseLevel in both the shot and receiver tables.

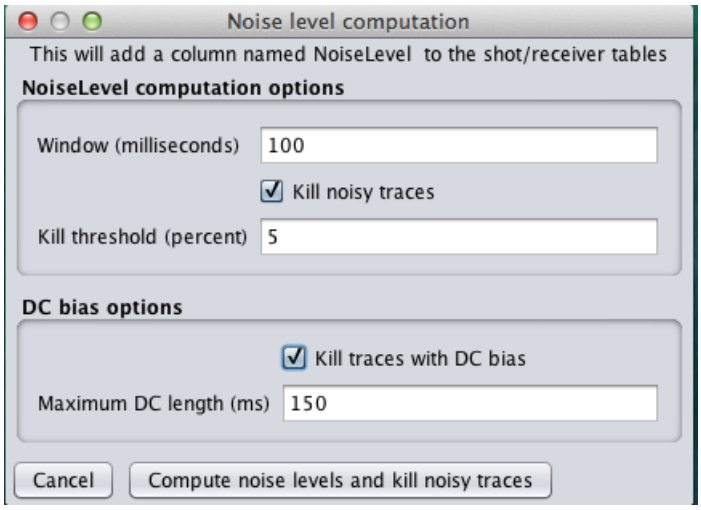


Figure 6---64 Dialog box for menu item Picking → Compute shot/receiver noise levels, kill noisy traces.

6.11.8 Un---kill all traces

There are no dialog boxes or user choices for this option. It simply sets every entry in the KILLED column in the Trace table to zero. That is, for every trace the 'killed---status' is set to False. It does not matter how or for what reason the traces were killed, after running this option, they will all be 'Un---killed.'

7 DelayTime

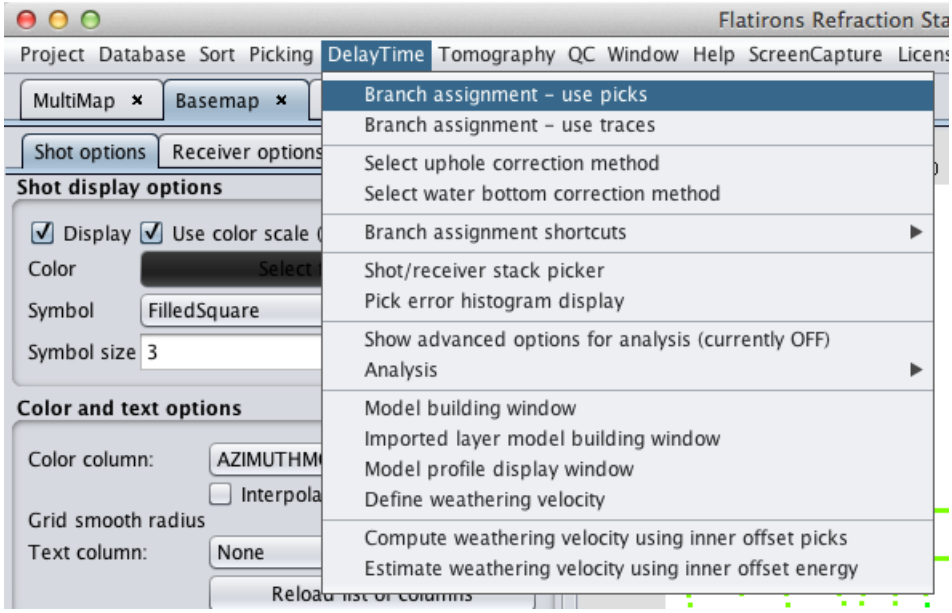


Figure 7---1 Options for the DelayTime menu item.

7.1.1 Branch assignment – use picks

A delay time solution requires a refractor… If a set of picks for a survey exists, then branch assignment can be done using those picks. The alternative method, branch assignment using traces, is described in the next section. From the DelayTime menu item, selecting **Branch assignment – use picks** will open the **Refractor branch assignment** window shown in Figure 7---2. The window has three panels; most of the work will be done in the center panel. The right panel has two basemap displays for showing information related to branch assignments. The left panel defaults to the **Map** tab, and has options to adjust the branch assignment process as well as the standard basemap display for selecting CMPs over the survey.

In the figure, the center panel has two sections highlighted with red boxes. **Linear moveout corrections** and **Lock Time axis** will almost certainly need to be applied for correct branch assignment.

The general procedure for branch assignment is to select a location on the basemap in the left panel. The picks associated with the CMP at that location would appear in the center panel. Then, up to five branches are identified, characterized and defined. This process is repeated until the user is satisfied that the entire survey has been sufficiently covered.

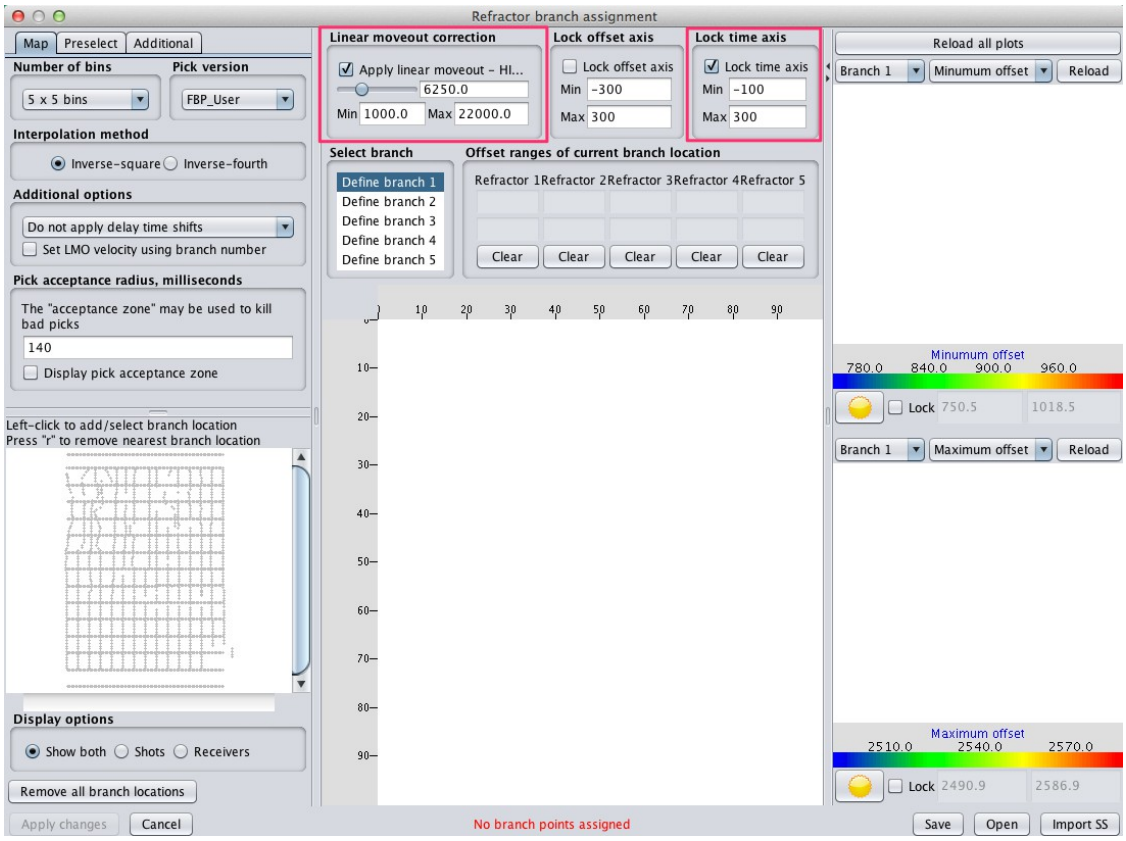


Figure 7---2 The Refractor branch assignment window.

The picks that are displayed depend on a couple of additional options that can be set in the left panel. These are the **Number of bins** (i.e., size of the CMP) and the **Pick version**, both set via drop---down lists.

Once a CMP is chosen, the center panel will appear similar to that shown in Figure 7---3, except the figure shows branch assignment already in progress. Note that linear moveout and a locked time axis have been applied.

To assign a branch, the user must decide which branch and/or how many branches to assign. In the figure, there are three branches, with the second branch being assigned. Note that although this is the second of three branches visible at this CMP, it is being defined as Branch 1 because in this example, only one branch is needed. To select the branch number, either click on the desired branch in the **Select branch** section, or press the number desired on the keyboard.

Defining a branch involves the usual left---click and drag mouse method. Dragging either left or right will work. First, select the location of the desired branch (offset and time). Then left---click and drag the mouse to the other end of the branch. Releasing the mouse will cause the branch to be defined. In the display above the picks, the offset range of the branch just defined will become populated. In Figure 7---3, Refractor 1 has minimum offset = 1999 and maximum offset 4353. If desired, these offset fields can be manually edited: enter a new

offset min/max and hit the **Enter/Return** key to update the display with the new offset range.

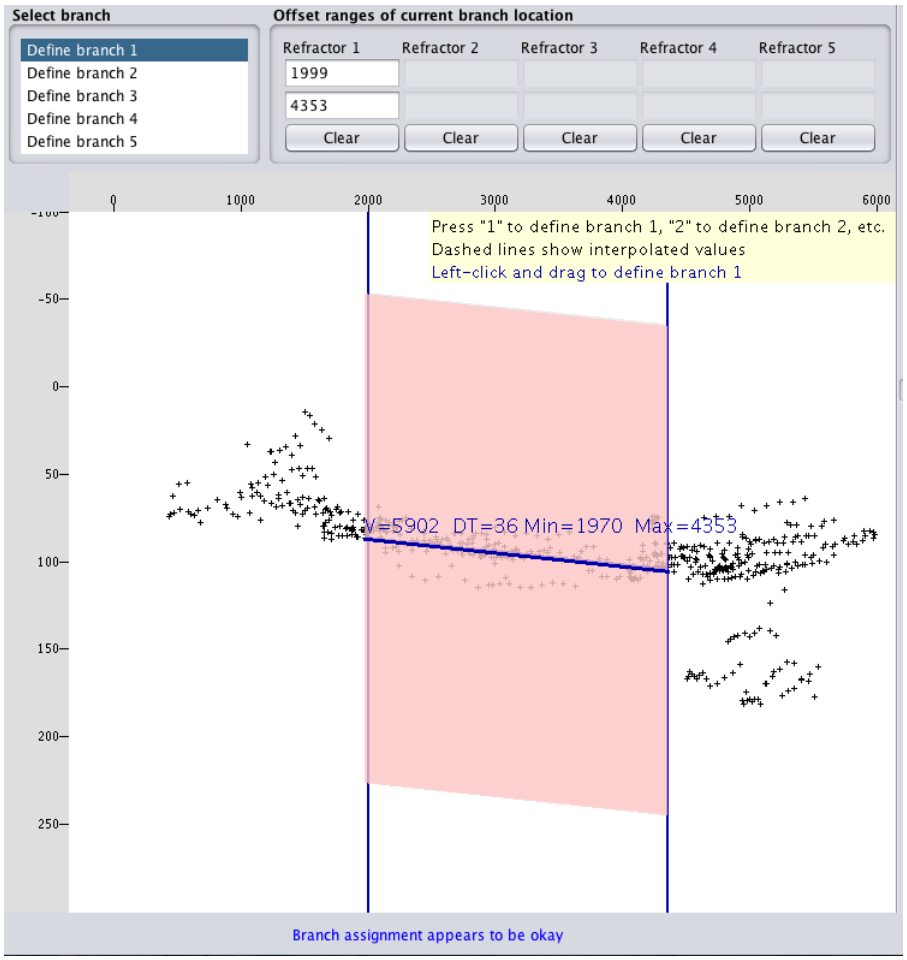


Figure 7---3 Center panel of Refractor branch assignment window displaying picks for a CMP. Branch 1 is in the process of being assigned.

If the branch assignment is unsatisfactory, simply repeat the process. Make sure the correct is being re---assigned! In the figure, Branch 1 is being re---defined, to indicate the display during the process of branch assignment. Alternatively, to remove a particular branch assignment, simply click the **Clear** button for the appropriate Refractor.

During branch assignment, two vertical lines are drawn to emphasize the offset range. A roughly horizontal line is drawn as a guide through the picks. Above this line are four numbers. The first (v) is the refractor velocity, the second (DT) is the estimated Delay Time, and the next two (Min, Max) are the offset range. These numbers change in real time as the mouse cursor is moved, until the left mouse button is released. Also a pink band is displayed along the branch line. This band is related to the **Pick acceptance** option, to be described later in this section. Upon releasing the button, the color turns a light gray.

As a side note, when defining a branch, the branch line can be temporarily extended to zero offset. The intercept with the time axis will be twice the delay time.

Each defined refractor is color coded as follows: branch 1 (Blue), branch 2 (Green), branch 3 (Red), branch 4 (Magenta) and branch 5 (Gray). Below the trace display are helpful hints such as “No branch points assigned”, “Overlapping branch offset range”, “Cannot define branch 2 without also defining branch 1”, “Branch assignment appears to be okay”, etc.

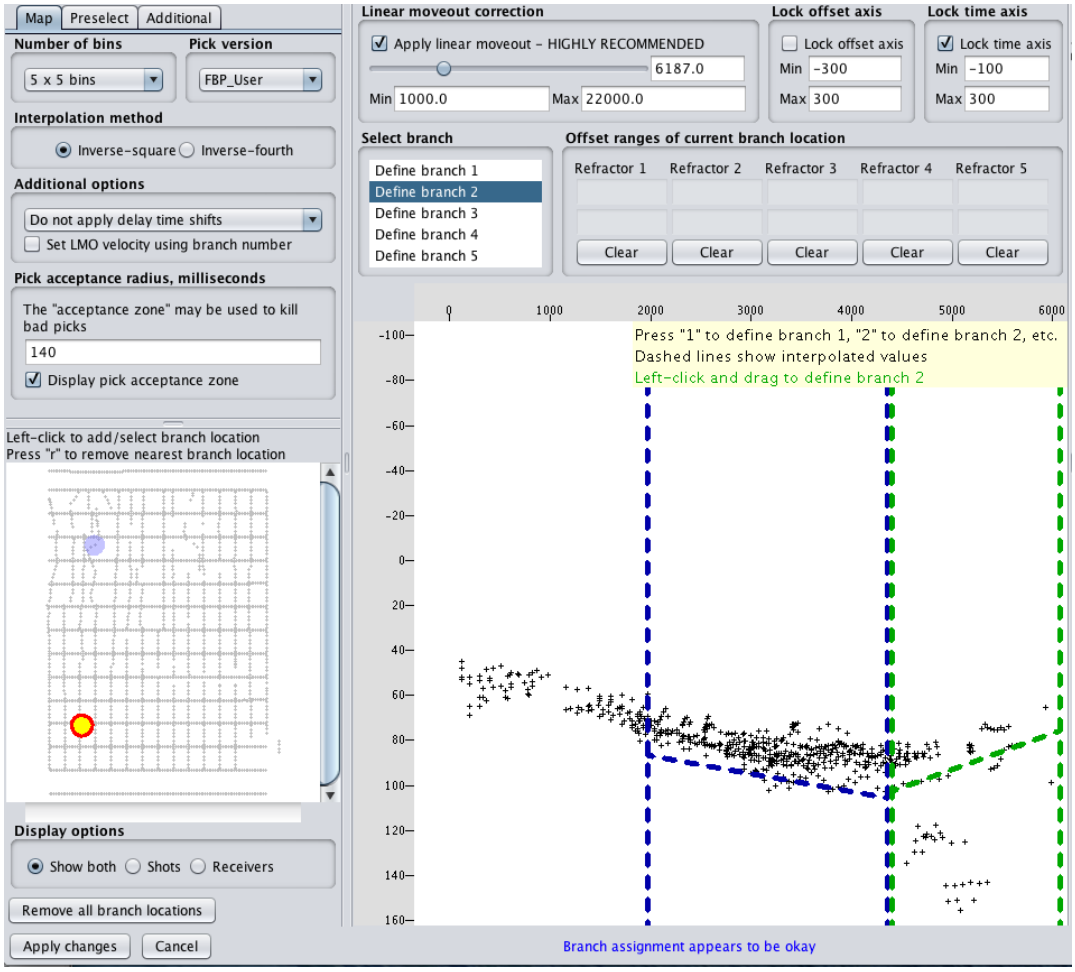


Figure 7---4 Another close---up view of the Branch assignment window. A second CMP has been selected. The dotted lines indicate extrapolated branches from previous CMPs.

After defining all desired branches at one particular CMP, continue on to another CMP. The picks display will appear as in Figure 7---4. The extrapolated branches from all previous CMPs are displayed as dotted lines, with consistent coloring. When a branch is defined on the current CMP, the dotted lines are replaced with the usual solid lines (branch line and vertical lines). If the **Clear** button for a particular refractor is clicked, the dotted lines reappear.

When moving around the basemap, from CMP to CMP, it is perfectly acceptable to adjust the linear moveout and/or time axis, as desired. These setting have no effect on the defined branches.

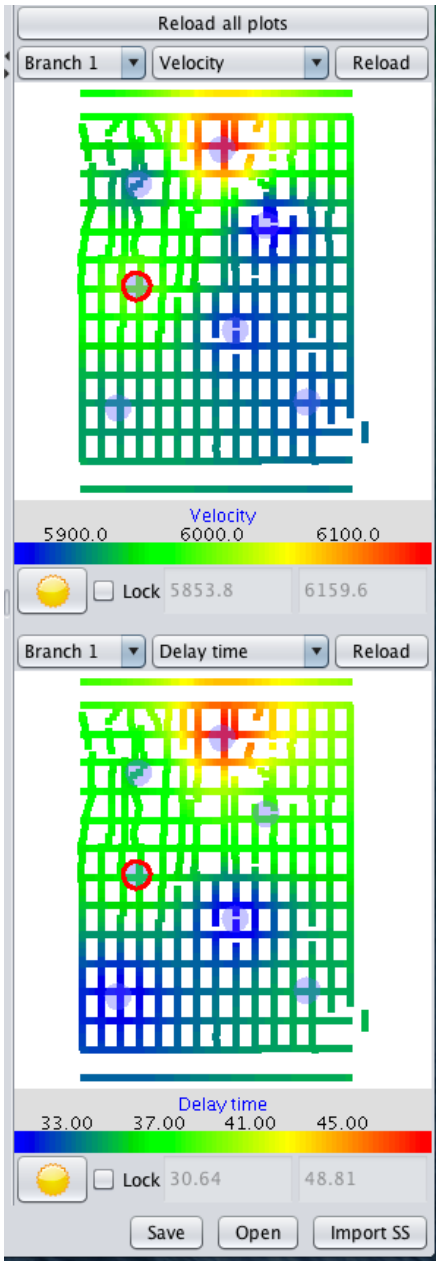


Figure 7---5 Right panel of the Refractor branch assignment window. Two separately configurable basemap views are available. Here, the refractor velocity and delay time associated with Branch 1 are being shown.

As the user defines branches at the CMPs, the right panel is updated. There are two display areas, each showing a basemap view (Figure 7---5). The user can select which branch and what information pertaining to that branch to display. In the figure, the velocity and delay time associated with Branch 1 was chosen.

Three buttons at the bottom of the panel permit file I/O of branch assignment data. Although branch information is automatically saved, the option exists for a user to specify a separate file to store the data. Clicking the **Save** button will open a standard file dialog box, prompting the user to enter the location and file name. The data is saved in XML format and has the '.xml' extension automatically appended. The **Open** button is used to import a previously saved branch assignment file.

Finally, the **Import SS** button is specifically designed to import branch data files that were created in the **Seismic Studio** software package.

Figure 7---6 shows the left panel of the Refractor branch assignment window, with the default view opened to the **Basemap** ( ) icon tab. There are two other standard icon tabs: the **Ensemble processing sequence** ( ) and **Wiggle Display options** ( ), both relating to how the traces appear. See Figure 6---6 and Figure 6---10 and associated text for details on the **Ensemble processing sequence** and **Wiggle Display options**, respectively.

The **Basemap** icon tab has three tabs – **Map**, **Preselect** and **Additional**. The **Additional** tab has two buttons. The first, **Copy first branch to second branch**, does exactly as the name implies. Selecting this button essentially renames Branch 1 as Branch 2 (throughout the survey, not just on one CMP). If there was a previously defined Branch 2, it is deleted. After clicking the button, the trace display will go blank – to repopulate, return to the **Map** tab and select any CMP. If the branch copy operation results in an incomplete or incorrect branch assignment at a CMP, the CMP will appear as a yellow dot on the basemap. Notice that the color---coding remains consistent – what was blue (Branch 1) is now green (Branch 2).

The second button on the Additional tab is **Set offset range for a given branch**. Again, the name is self---explanatory. Selecting this button will open a dialog box (not shown) in which the user selects the desired branch and enters a minimum and maximum offset for that branch. Clicking the **Reset offsets** button in the dialog box will make the desired offset change. This change is immediately reflected in the appropriate branch in the **Offset ranges of current branch location** section in the center panel. The change is applied to all CMPs for which the selected branch was previously defined. If a CMP is revisited and the selected branch is redefined, the offset range is updated for that CMP only – all other CMPs keep the specified offset range.

The **Preselect** tab includes options related to pick gathers. These are the sparse CMP gathers that were optionally generated when the project was created. If that step was omitted, this tab can be used to generate them before branch assignment. The CMP computation requires both spacing and radius, which are entered via text boxes.

It may be desirable to recreate the gathers, for whatever reason. If so, simply click the **Remove all precomputed pick gathers** button, enter the desired CMP parameters and click the **Create precomputed pick gathers** button. Finally, a toggle box sets whether or not branch assignment uses the precomputed gathers. If this box is checked, the location of

sparse CMP appear on the basemap in the **Map** tab, and clicking anywhere in the basemap causes the nearest sparse CMP to be selected for branch assignment.

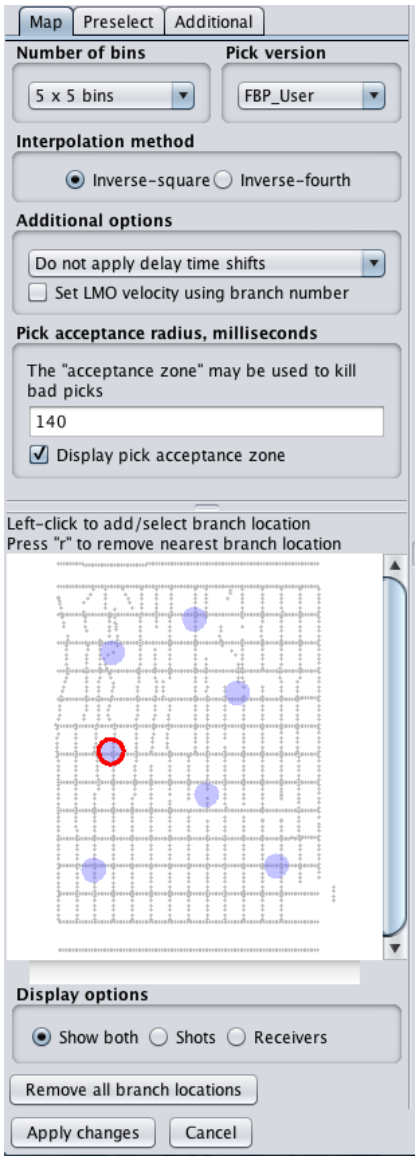


Figure 7-6 Left panel of the Refractor branch assignment window.

The **Map** tab has a number of options in addition to the basemap display. At the top is the **Number of bins** drop-down list. This is used to set the bin size of the CMPs and is in effect only if precomputed sparse CMPs are not being used.

The **Interpolation method** option has a checkbox for either **Inverse-square** or **Inverse-fourth** interpolation. The offset ranges, refractor velocities and delay times will all be interpolated across the survey according to the selected interpolation method.

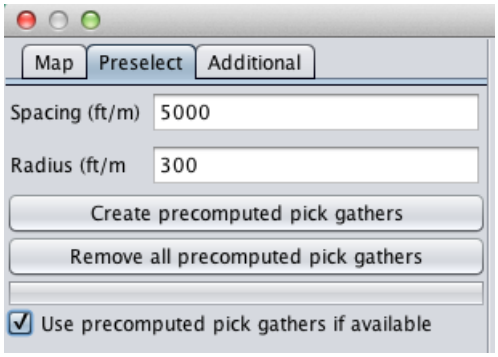


Figure 7---7 The Preselect tab in the left panel of the Refractor branch assignment window.

The next section is called **Additional Options**. It has a checkbox for setting LMO velocity according to branch number. There is also a drop---down list for shifting traces based on delay times. In the example shown in Figure 7---8 all traces will be shifted according to Branch 1 delay times. This is useful if a delay time solution already exists. By applying these shifts, the quality of the branch assignments is easier to determine.

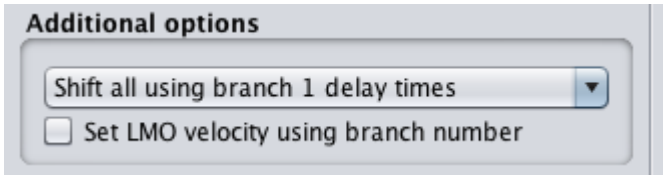


Figure 7---8 Additional options in the left panel of the Refractor branch assignment window.

The process of defining branches can be used to kill picks. Figure 7---9 shows the **Pick acceptance radius** section in the Branch assignment window. The 'acceptance zone' is a time (in milliseconds) above and below the defined branch. Picks outside this zone will be (optionally) killed. So in the example shown, the total acceptance zone is 280 ms. A checkbox toggles the display of the zone. It is the pink zone when a branch is being defined and turns to light gray after the branch has been defined. Picks will not be killed automatically. The user will be asked to confirm whether picks should be killed after branch assignment is finished.

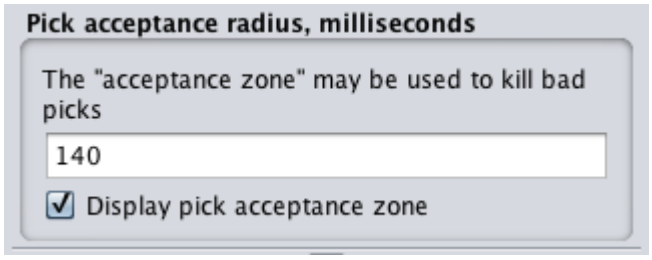


Figure 7---9 Pick acceptance option in the Refractor branch assignment window.

Once all branches are defined to the user satisfaction, regardless of how many CMPs have been considered, click on the **Apply changes** button at the bottom left of the window. A popup window will appear (not shown), asking if the defined fields should be used to

interpolate velocities and delay times. The default is **Yes** and is highly recommended. This will populate the shot and receiver tables with refractor velocities and delay times. If a refractor solution already exists from a previous session, it will be overwritten with a new solution.

Next, there is be a popup window asking if picks should be killed based on branch assignment information. The default is **Yes**. It may be best to make a backup of the picks before proceeding.

A third popup window will appear as shown in Figure 7---18. If the **Not now** button is selected the **Refractor branch assignment** window closes and branch assignment is complete. If there currently are no saved picks it does not make sense to execute delay time analysis.

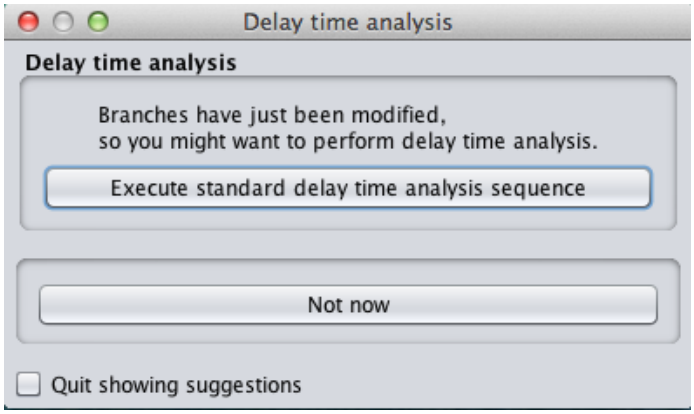


Figure 7---10 Dialog box after branch assignment, asking if delay time analysis should be executed.

7.1.2 Branch assignment – use traces

There is a way to obtain a delay time solution without having to pick the first breaks on individual traces. But it still requires branch assignment. Because the branch assignment method is very similar to that using picks, the information presented here will be very similar to the previous section. The **Branch assignment – use traces** menu item opens the **Refractor branch assignment** window shown in Figure 7---11. For display purposes, a linear moveout trend has been applied, the time axis is locked at a narrow range and a sparse CMP has been selected in order to populate the display with traces. Up to five branches, and therefore five refractors, can be defined.

Typically, it is best to lock the time axis and apply linear moveout so that the separate refractors are identifiable by the different slopes of an event with respect to offset. There is no requirement for which refractor is selected for which branch assignment. For example, if a user wants to use a deeper refractor, at larger offset, it can be defined as branch 1. However, it is not allowed to define a branch of a higher number (for example, branch 2) without also defining a lower numbered branch (in this example, branch 1).

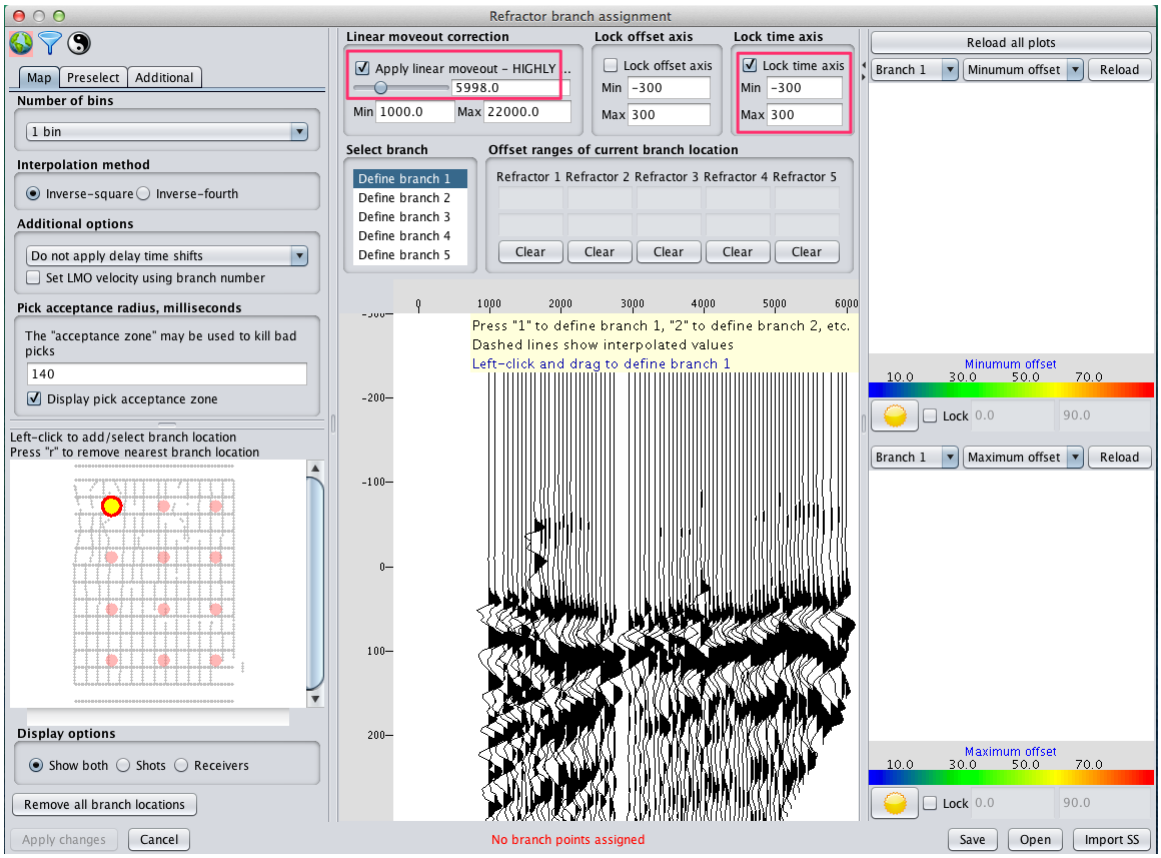


Figure 7---11 Opening window upon selection of the Branch assignment --- use traces menu item. Linear moveout and Lock time axis have been applied as shown in the red boxes. Also, the top left sparse CMP has been clicked as indicated by the yellow circle on the basemap.

Figure 7---12 shows an example of branch definition in progress. To define a branch, the user selects which branch to define, either in the **Select branch** box or by pressing a number key. Then, left---click and drag a best---fit line along an event, releasing the mouse key when finished. While dragging, the current minimum/maximum offset, estimated refractor velocity and estimated delay time are displayed above the line.

When the left mouse key is released the definition of the selected branch is done. Vertical lines are added to highlight the offset range and min/max offset are inserted into the boxes for the designated refractor, above the trace display. If a given branch assignment is unsatisfactory, simply click the **Clear** button for that refractor.

To review: branch assignment consists of setting a refractor offset range, estimating a refractor velocity and estimating a delay time. The latter two are done implicitly, based on the slope of the best---fit branch line.

Each defined refractor is color coded as follows: branch 1 (Blue), branch 2 (Green), branch 3 (Red), branch 4 (Magenta) and branch 5 (Gray). Below the trace display are helpful hints such as “No branch points assigned”, “Overlapping branch offset range”, “Cannot define branch 2 without also defining branch 1”, “Branch assignment appears to be okay”, etc.

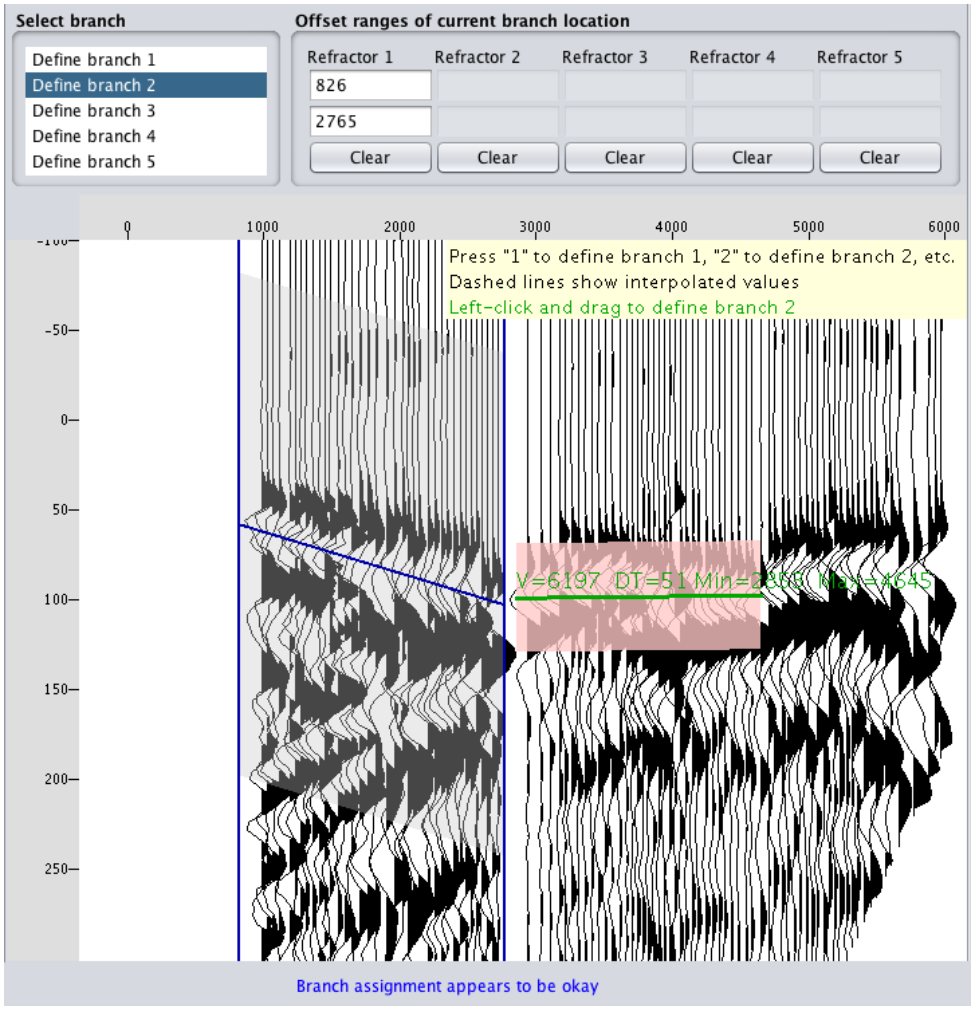


Figure 7---12 Close---up of Branch assignment window, where the second branch of a particular CMP is being defined. The first branch has already been defined and its offset ranges automatically displayed.

After defining all desired branches at one particular CMP, continue on to another CMP. The trace display will appear as in Figure 7---13. The extrapolated branches from all previous CMPs are displayed as dotted lines, with consistent coloring. When a branch is defined on the current CMP, the dotted lines are replaced with the usual solid lines (branch line and vertical lines). If the **Clear** button for that particular refractor is clicked, the dotted lines reappear.

Note that when moving around the basemap, from CMP to CMP, it is perfectly acceptable to adjust the linear moveout and/or time axis, as desired.

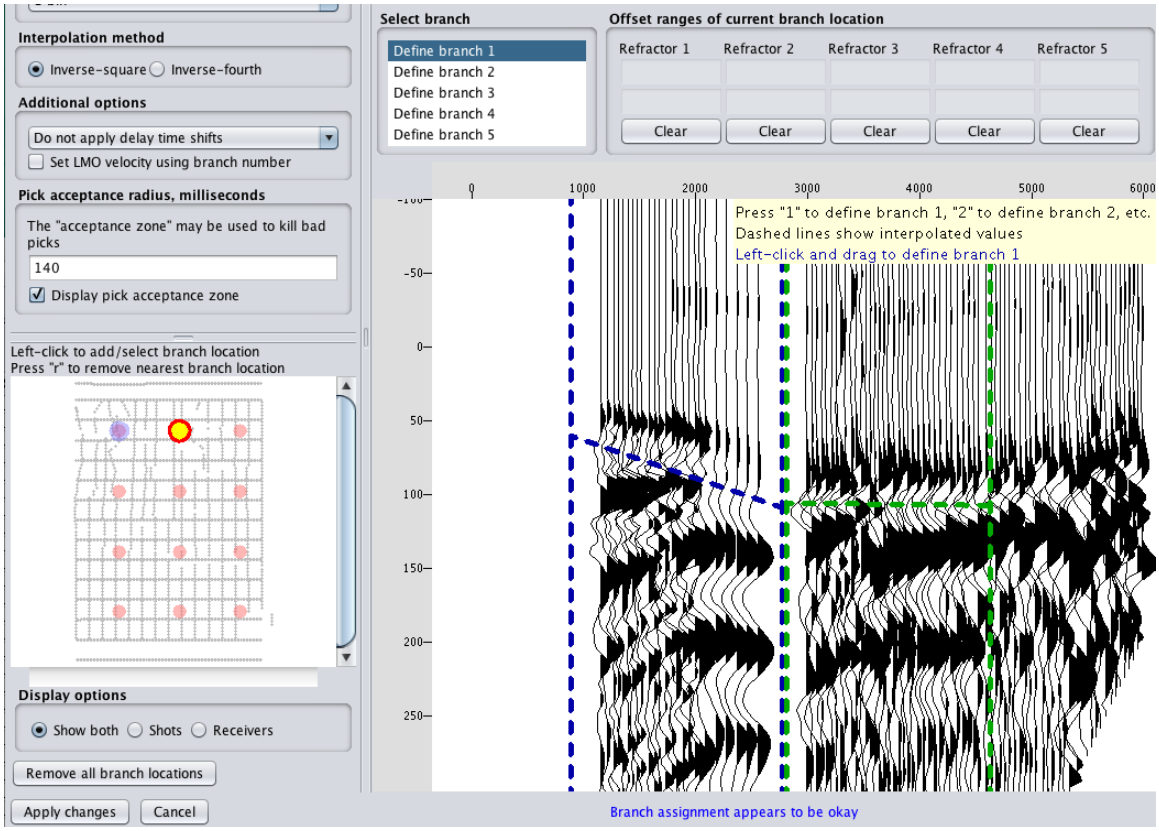


Figure 7---13 Another close---up of the Branch assignment window. Dotted lines indicate extrapolated branches from previous CMPs.

As the user defines branches at the CMPs, the right panel is updated. There are two display areas, each showing a basemap view (Figure 7---14). The user can select which branch and what information pertaining to that branch to display. In the figure, the velocity and delay time associated with Branch 2 was chosen.

Three buttons at the bottom of the panel permit file I/O of branch assignment data. Although branch information is automatically saved, the option exists for a user to specify a separate file to store the data. Clicking the **Save** button will open a standard file dialog box, prompting the user to enter the location and file name. The data is saved in XML format and has the '.xml' extension automatically appended. The **Open** button is used to import a previously saved branch assignment file.

Finally, the **Import SS** button is specifically designed to import branch data files that were created in the **Seismic Studio** software package.

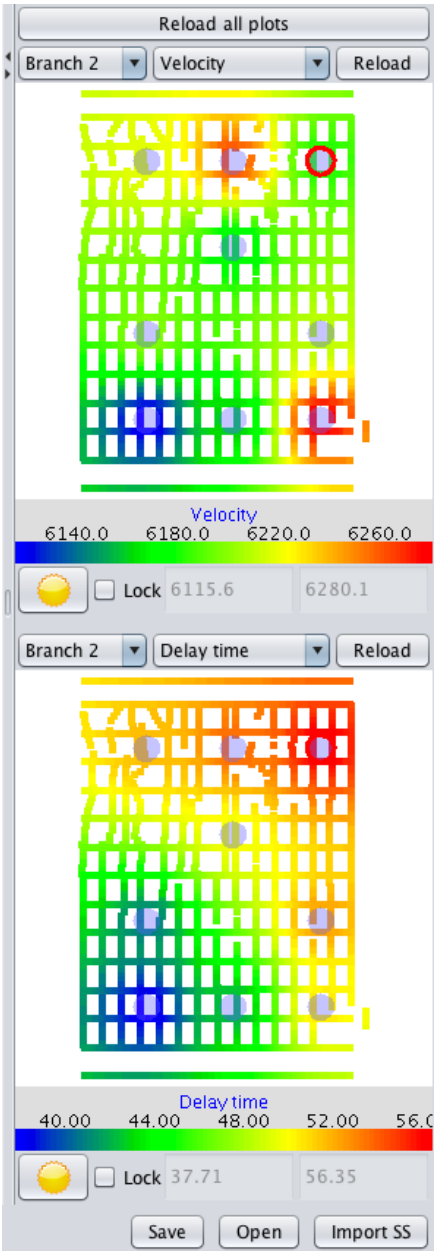


Figure 7---14 Right panel of the Refractor branch assignment window. Two separately configurable basemap views are available. Here, the refractor velocity and delay time associated with Branch 2 are being shown.

Figure 7---15 shows the left panel of the Refractor branch assignment window, with the default view opened to the **Basemap** ( ) icon tab. There are two other standard icon tabs: the **Ensemble processing sequence** ( ) and **Wiggle Display options** ( ), both relating to how the traces appear. See Figure 6---6 and Figure 6---10 and associated text for details on the **Ensemble processing sequence** and **Wiggle Display options**, respectively.

The **Basemap** icon tab has three tabs – **Map**, **Preselect** and **Additional**. The **Additional** tab has two buttons. The first, **Copy first branch to second branch**, does exactly as the name

implies. Selecting this button essentially renames Branch 1 as Branch 2 (throughout the survey, not just on one CMP). If there was a previously defined Branch 2, it is deleted. After clicking the button, the trace display will go blank – to repopulate, return to the **Map** tab and select any CMP. If the branch copy operation results in an incomplete or incorrect branch assignment at a CMP, the CMP will appear as a yellow dot on the basemap. Notice that the color---coding remains consistent – what was blue (Branch 1) is now green (Branch 2).

The second button on the Additional tab is **Set offset range for a given branch**. Again, the name is self---explanatory. Selecting this button will open a dialog box (not shown) in which the user selects the desired branch and enters a minimum and maximum offset for that branch. Clicking the **Reset offsets** button in the dialog box will make the desired offset change. This change is immediately reflected in the appropriate branch in the **Offset ranges of current branch location** section in the center panel; see Figure 7---13 for example. The change is applied to all CMPs for which the selected branch was previously defined. If a CMP is revisited and the selected branch is redefined, the offset range is updated for that CMP only – all other CMPs keep the specified offset range.

The **Preselect** tab currently has just one item – a toggle box for whether to use precomputed sparse CMP gathers. If sparse CMP gathers were not precomputed when the project was created then they can be computed at any time via the **Sort** menu item.

The **Map** tab has a number of options in addition to the basemap display. At the top is the **Number of bins** drop---down list. This is used to set the bin size of the CMPs and is in effect only if precomputed sparse CMPs are not being used.

The **Interpolation method** option has a checkbox for either **Inverse---square** or **Inverse---fourth** interpolation. The offset ranges, refractor velocities and delay times will all be interpolated across the survey according to the selected interpolation method.

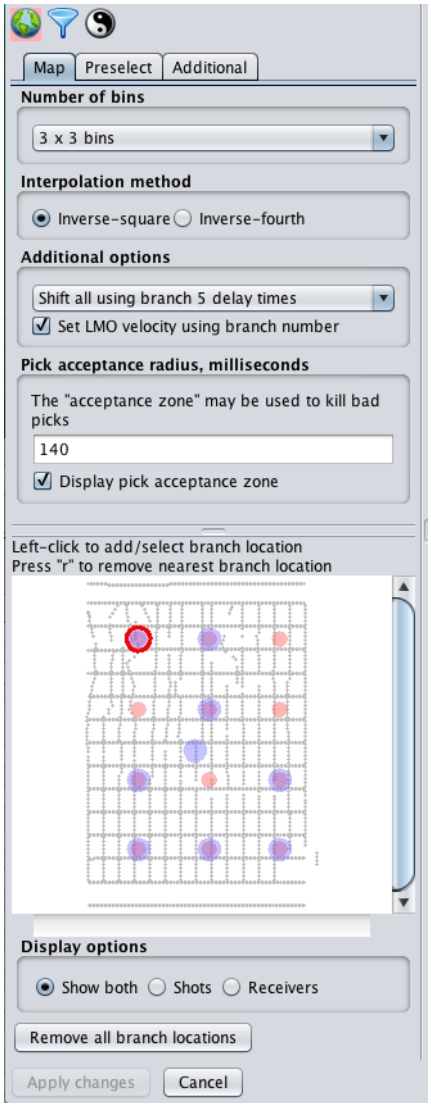


Figure 7---15 Left panel of the Refractor branch assignment window.

The next section is called **Additional Options**. It has a checkbox for setting LMO velocity according to branch number. There is also a drop---down list for shifting traces based on delay times. In the example shown in Figure 7---16 all traces will be shifted according to Branch 1 delay times. This is useful if a delay time solution already exists. By applying these shifts, the quality of the branch assignments is easier to determine.

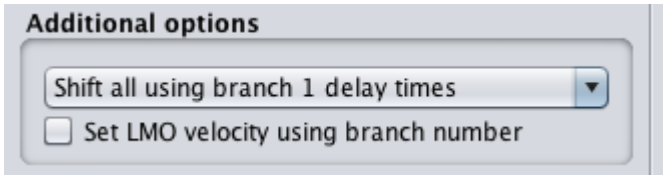


Figure 7---16 Additional options in the left panel of the Refractor branch assignment window.

The process of defining branches can be used to kill picks. Figure 7---17 shows the **Pick acceptance radius** section in the Branch assignment window. The 'acceptance zone' is a time (in milliseconds) above and below the defined branch. Picks outside this zone will be (optionally) killed. So in the example shown, the total acceptance zone is 280 ms. A checkbox toggles the display of the zone. It is a lightly shaded gray area and only appears once a branch has been defined. See Figure 7---12 and Branch 1 for an example. Picks will not be killed automatically. The user will be asked to confirm whether picks should be killed after branch assignment is finished.

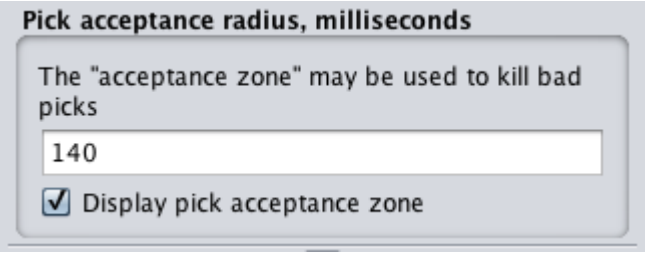


Figure 7---17 Pick acceptance option in the Refractor branch assignment window.

Once all branches are defined to the user satisfaction, regardless of how many CMPs have been considered, click on the **Apply changes** button at the bottom left of the window. A popup window will appear, asking if the defined fields should be used to interpolate velocities and delay times. The default is **Yes** and is highly recommended. Next, there is be a popup window asking if picks should be killed based on branch assignment information. The default is **Yes**. It may be best to make a backup of the picks before proceeding.

A third popup window will appear as shown in Figure 7---18. If the **Not now** button is selected the **Refractor branch assignment** window closes and branch assignment is complete. If there currently are no saved picks it does not make sense to execute delay time analysis.

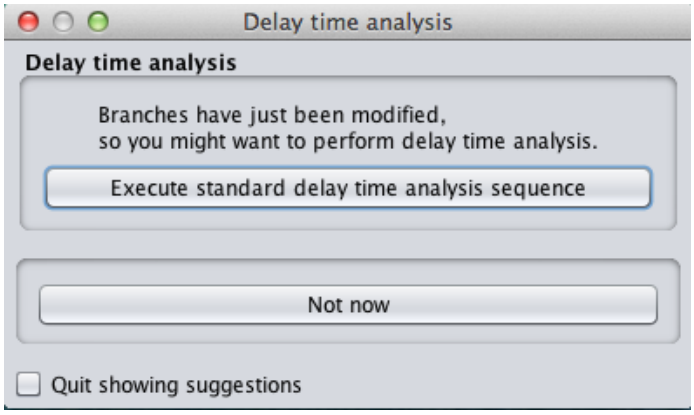


Figure 7---18 Dialog box after branch assignment, asking if delay time analysis should be executed.

7.1.3 Select uphole correction method

An uphole correction is required when shots are buried beneath the surface, as is the case for dynamite surveys. This menu option lets the user select how to make the correction. Figure 7---19 shows the dialog box that opens when selecting this menu item. If the **Use the uphole time** option is checked, the program will use values stored in the **Upholetime** column in the Shot table. These values are typically imported into Flatirons via the SEG---Y file but can be imported separately from an ASCII file. The other correction option is the **Raypath correction**, using a delay time solution. This is the default option and is recommended due to better performance.

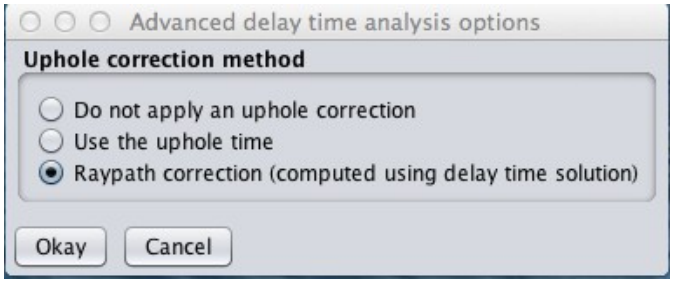


Figure 7---19 Dialog box from the Select uphole correction method menu item.

7.1.4 Select water bottom correction method

Similar to uphole corrections, for marine surveys a correction related to ocean bottom cable (OBC) data is required. The **Select water bottom correction method** menu item is used to apply this correction. Selecting the item opens the dialog box shown in Figure 7---20. Again, there are two separate methods to apply the correction. Both require a water velocity --- the default velocity is 5000 ft/sec (or 1485 m/sec) but the user can enter any desired value in the text box. The first method is **Use travel time from shot (or receiver) to water bottom**. This method uses values in the **WaterDepth** column, typically read in from a SEG---Y file. The other method uses a **Raypath correction**, similar to the uphole correction described above.

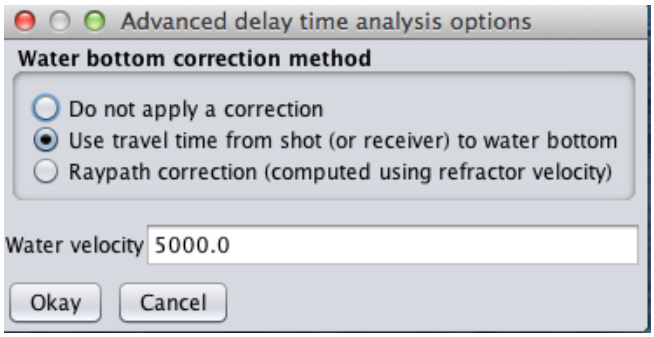


Figure 7---20 Dialog box from the Select water bottom correction method.

7.1.5 Shot/receiver stack picker

[TBD] If branch assignment has been done, shot/receiver stacks can be used to pick delay times. The menu item **Picking → Shot/receiver stack picker** opens the **DT Shot/Rec stacks** window shown in Figure 7---21. The window has the usual three---panel setup. The center panel, with variable number of text and plot headers, displays the shot/receiver stacks. The right panel has a basemap display along with several display options. Above the basemap is a section called **Nearest shot/receiver**. Whenever the mouse cursor is over the stack display in the center panel, this section is continually updated with information about the shot/receiver nearest the mouse cursor. The information included is the ID, LineNumber, PointNumber and Index. Of course, if the center panel is on the shot plane, the nearest shot to the mouse cursor is identified. Similarly, a receiver is identified for the receiver plane.

The left panel includes four icon tabs. The last three were covered in Sections 6.1 and 6.9. The first, and default, icon tab is **Create new stacks** (). Figure 7---22 shows this tab. Under **Stack creation options** the user selects which branch (i.e., refractor) to use when stacking from a drop---down list. Obviously, only previously defined branches can be used for stacking. If an undefined branch is accidentally selected, a dialog box will open, warning that the selected branch is not valid.

The user also needs to select a time window by entering a minimum and maximum time. At this point, clicking on the **Create stacks** button will start the job to create stacks. However, the user may choose to modify the default processing sequence before creating stacks. As a reminder, the red flags next to the individual processes (and the red 'Not yet called' text) simply indicate that stack creation has not yet been performed. Notice that one process is **Apply delay time analysis shifts**. So, with the chosen refractor, the delay times and refractor velocity are incorporated into the stacking process. Of course, the first time stacks are created there may not be any delay times yet.

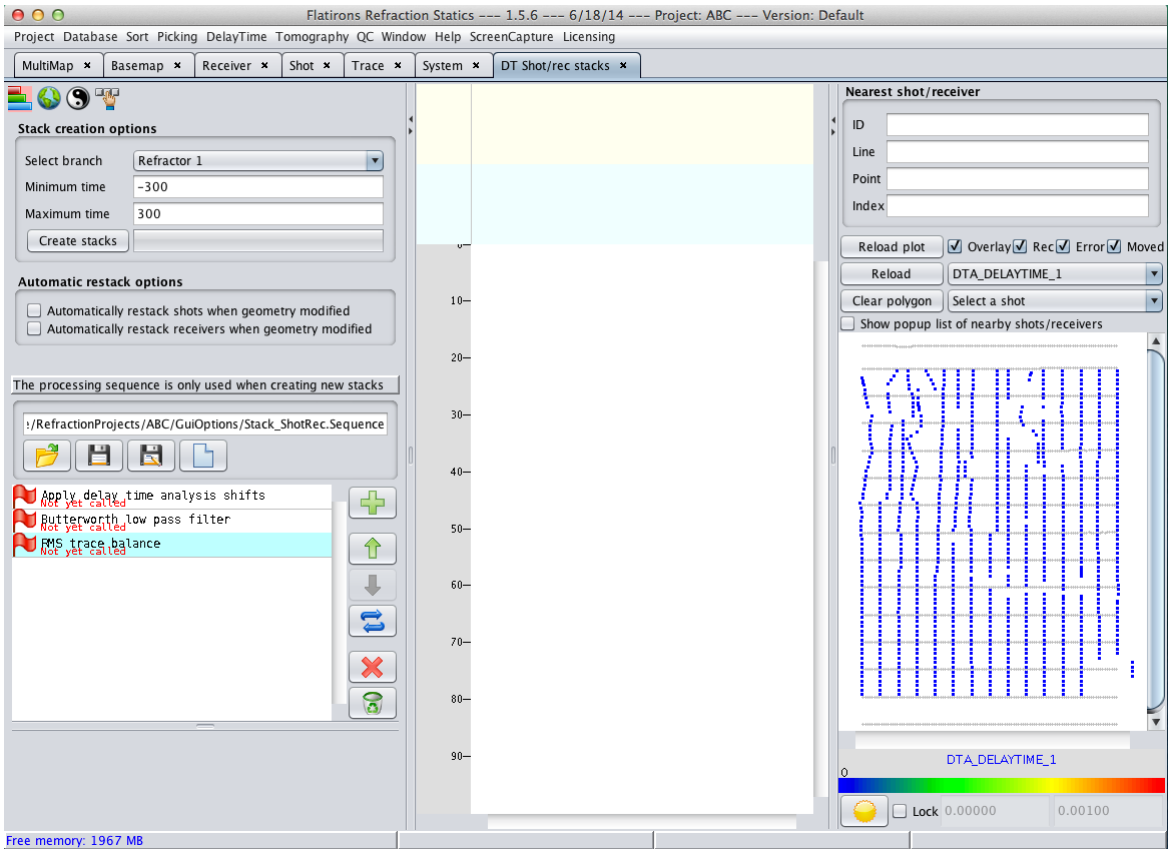


Figure 7-21 The DT Shot/rec stacks opening window from the Picking → Shot/rec stack picker menu item.

The remaining section in this panel is the **Automatic restack options** section. The just have checkboxes for whether to restack shots and/or receivers automatically if the geometry is modified.

Clicking on the **Create stacks** button will cause a progress bar to appear to the right of the button. When the job is finished the red flags in the processing sequence will turn green.

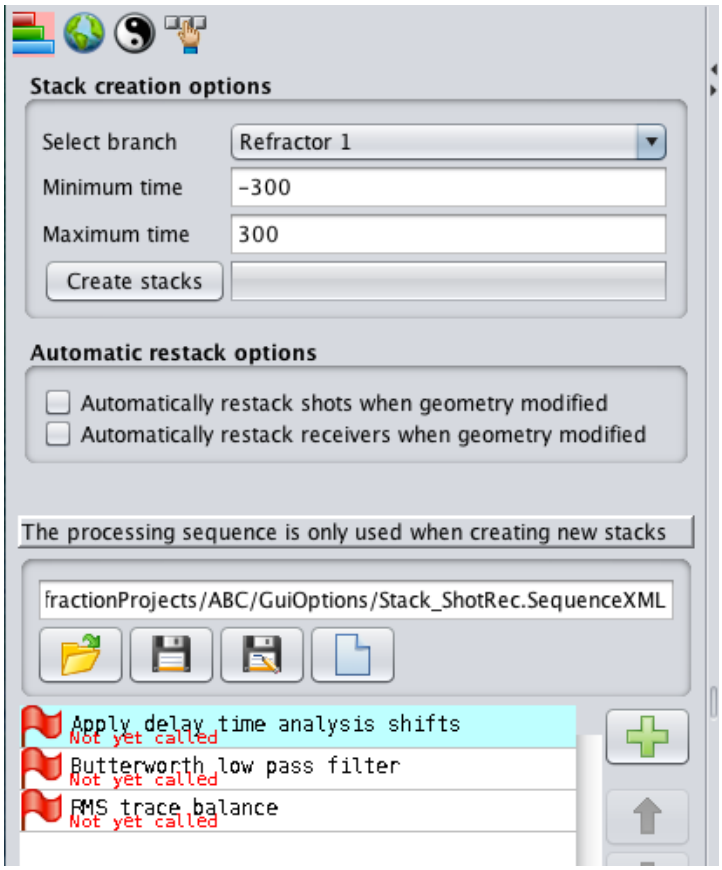


Figure 7---22 Closeup of the 'Create new stacks' icon in the left panel of the 'DT Shot/rec stacks' window.

The procedure for obtaining the best delay times using this window is: Create stacks ⟶ pick shot stacks ⟶ pick receiver stacks ⟶ Create stacks ... and repeat as necessary. In order to do this the user will navigate among the icon tabs to select an event to pick, select shot/receiver planes and other viewing options and use pick controls. How to do these tasks have been described elsewhere in the manual.

After iterating through picking and stacking a few times, the event being picked should approach zero time for all stacks. Figure 7---23 show a partial view of the **DT Shot/rec stacks** window after two iterations. The basemap has been set to display delay times (DTA_DELAYTIME_1) for shots. The '**_1**' appended to the delay time indicates that these times refer to Refractor 1. Similarly for '_2', etc. The two plot headers show display time and velocity. The event picked was selected to be the trough and indeed it does closely line up at zero time.

Two final reminders: one, only those traces that were assigned to the selected branch (Refractor 1 in this example) have been picked in this process; and two, those picks are based on a rough refractor velocity from hand---selected sparse CMPs. To obtain a proper delay time solution a complete delay time solution has to be computed. That will be covered later in this chapter, under a different menu item: **Picking ⟶ Analysis.**

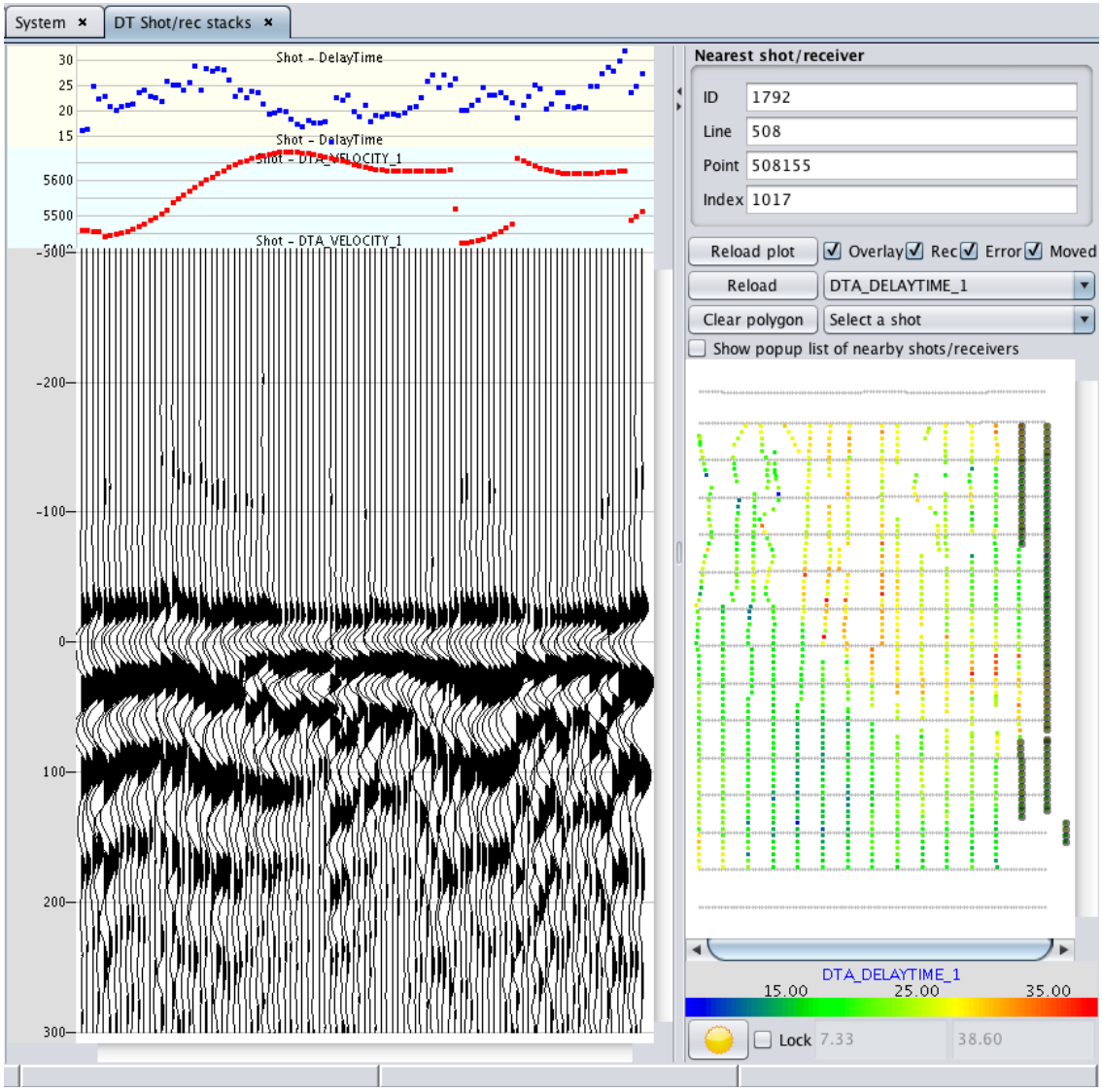


Figure 7---23 The DT Shot/rec stacks window after a couple iterations of picking and stacking.

7.1.6 Pick error histogram display

This menu item opens a tab window called Pick histogram which displays up to two histograms of pick errors (Figure 7---24). These errors are the magnitude difference between the actual pick times and the pick time predicted by delay time theory. As was mentioned in the previous section, if the pick error was zero, the picked events would line up at zero time. As shown in Figure 7---23, those events are close to, but not exactly zero. The histogram display provides a quick overview of the quality of the stack picking.

Drop down lists allow the user to select both pick version and branch. If the delay time analysis incorporated an anisotropy term, the pick errors can be optionally computed with this term.

A section of the window called **Info from cursor location** displays in real time information related to the current mouse location. This information includes: the cutoff error (that is, where on the horizontal axis the mouse is located), the number of picks with error less than the cutoff error (i.e., cumulative number of picks to the left), the number of picks with error greater than the cutoff error (i.e., cumulative number of picks to the right) and the percent less (percent of picks to the left of the mouse cursor). In the example in the figure, about 95% of the picks have a pick error less than 10 ms.

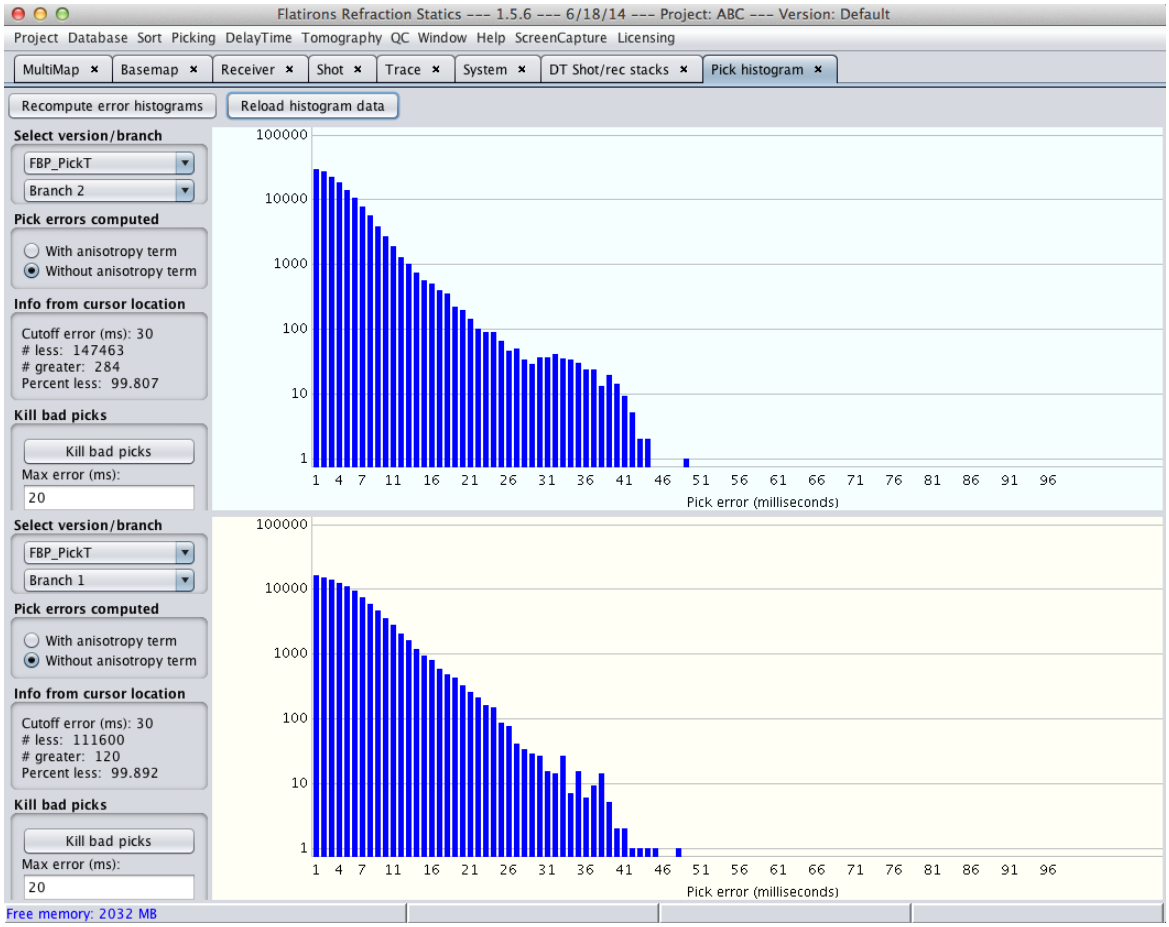


Figure 7---24 The Pick histogram tab, from the Pick error histogram display menu item.

Another useful feature of the histogram window is the option to kill bad picks. The user enters a Maximum error value (in ms) and clicks the **Kill bad picks** button. This will cause a job to start and **Flatirons** will temporarily change focus to the Job tab to show the progress of the job. All picks exceeding the maximum error will be killed. This is accomplished by setting pick times to ---9999 in the trace table, for the selected version (for example, FBP_PickT).

To display the updated histogram, click the **Reload histogram data** button. If a significant number of picks were killed, the delay time solution should be recomputed. Preferably the killed picks should be identified and picked correctly, if possible, before recomputing.

7.1.7 Show advanced options for analysis (currently OFF)

This menu item is simply a toggle for whether to show advanced analysis options. Once this menu item is selected, a checkmark will be added to the left of the text and the text will change to **Show advanced options for analysis (currently ON)**. By default, advanced options are off.

The effect of toggling this item is seen in the behavior of the next menu item, **Analysis**. Specifically, when advanced options are off, selecting the option **Analysis → Default analysis sequence** automatically runs a default delay time analysis sequence (see Section 7.1.9 for details). However, when advanced options are **ON**, selecting this option opens the window shown in Figure 7---25. It has no effect on any other menu items.

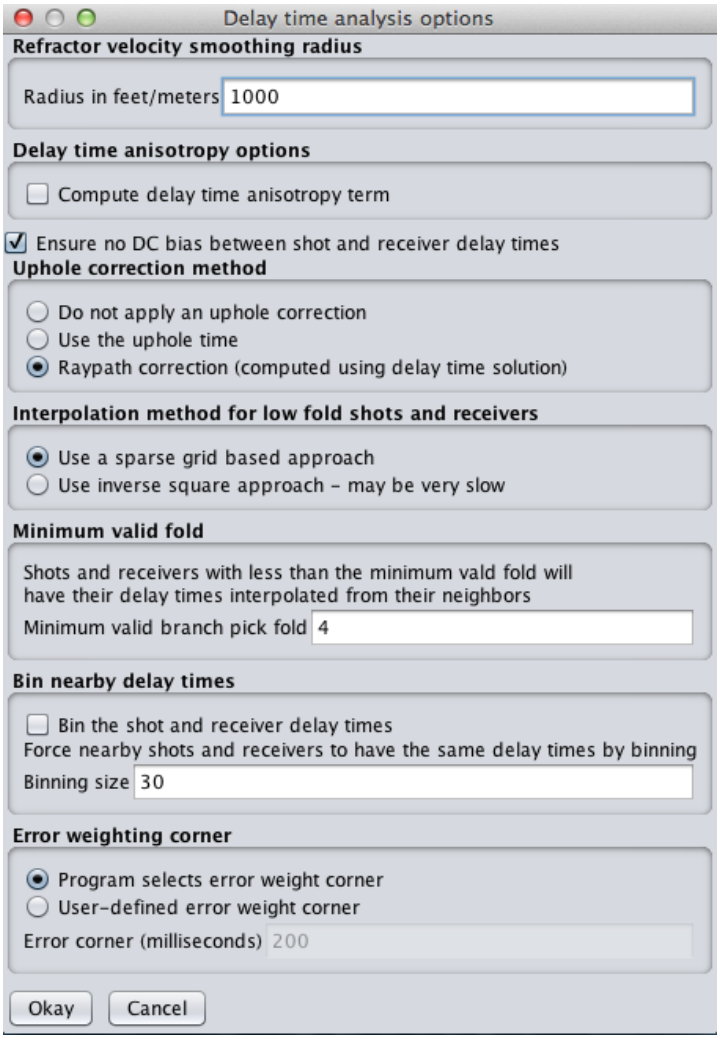


Figure 7---25 Advanced options from the Analysis → Default analysis sequence menu item. Appears only if Advanced Options are turned on.

Here, several parameters used in the delay time analysis can be modified. If all the default values in this window are left unchanged, it is equvialent to running the default analysis sequence (that is selecting **Analysis → Default analysis sequence** with advanced options turned OFF).

7.1.8 Analysis

This menu item has several sub---menu items (Figure 7---26) related to the method of updating the velocity and inclusion of anisotropy.

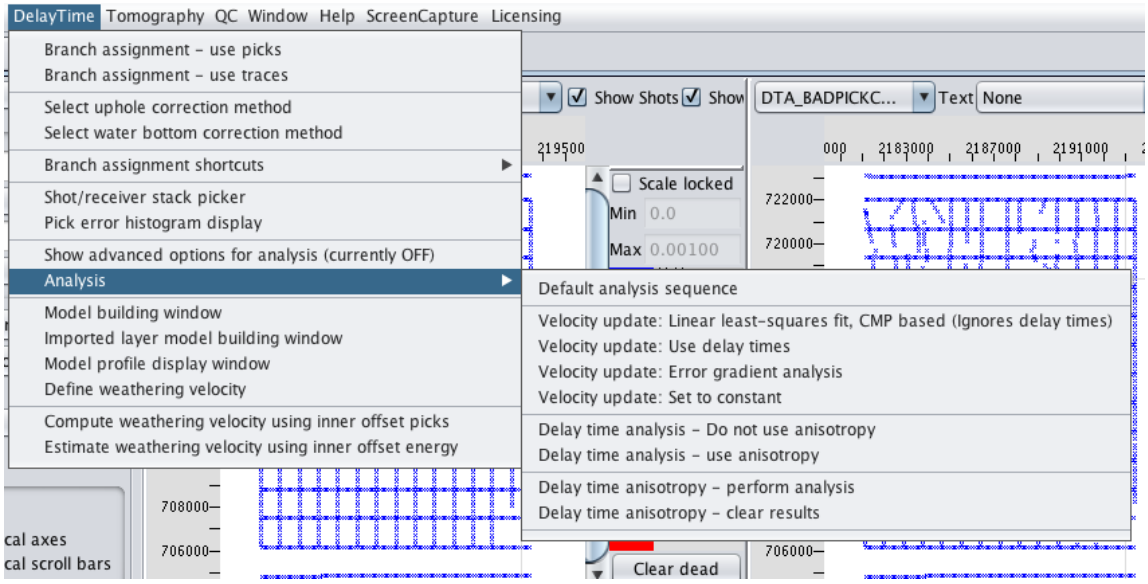


Figure 7---26 Sub---menu items from the DelayTime → Analysis menu.

If advanced options are OFF (see Section 7.1.8) selecting the Default analysis sequence immediately executes a job comprising a default sequence of analysis steps. The Job tab (Figure 7---27**Error! Reference source not found.**) opens and displays the progress of the analysis sequence.

The next four sub---menu items are used to update the refractor velocity. It is up to the user to determine if the velocity needs to be updated and which method to apply.

The remaining items pertain to anisotropy. Selecting item **Delay time analysis – Do not use anisotropy** opens the window shown in Figure 7---28. Compare this figure with Figure 7---25. The effect of selecting this menu item is the same as the Advanced options except the default refractor smoothing radius is automatically used and there is no option turn on the anisotropy computation. Which makes sense, because the user specifically requested to not use anisotropy. Otherwise, the rest of the advanced options are available.

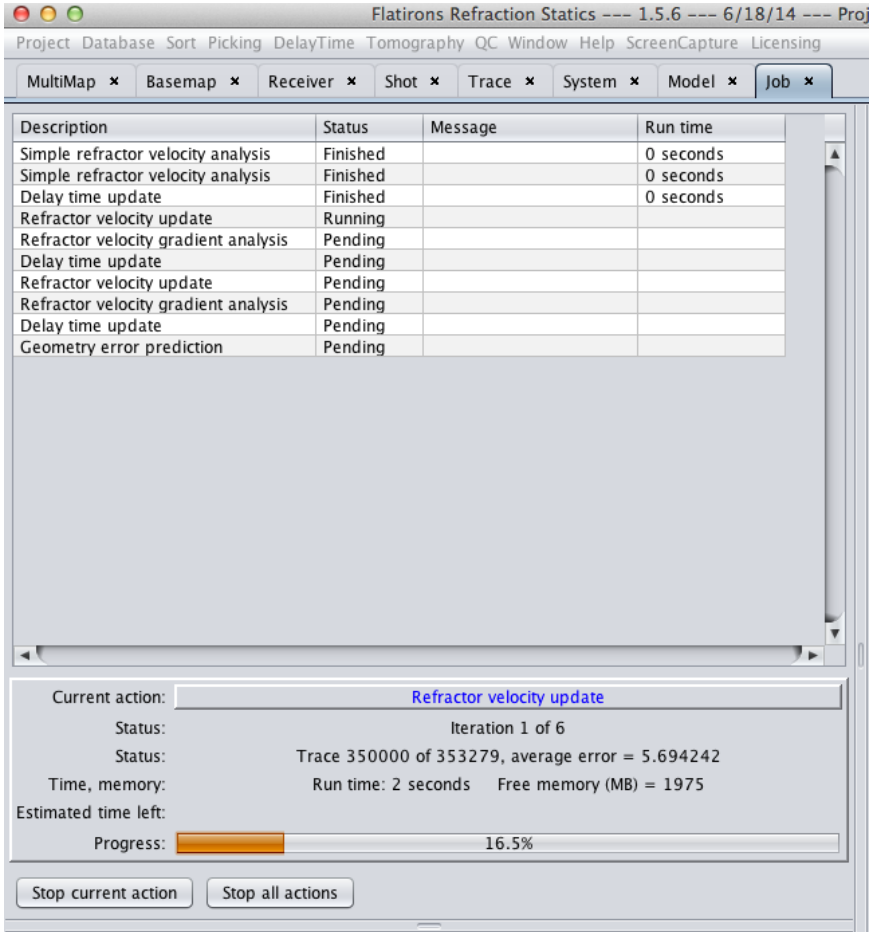


Figure 7---27 Job tab showing progress of the default sequence for delay time analysis.

Instead of selecting the item item **Delay time analysis – use anisotropy** just use **Analysis → Default analysis sequence** with advanced options turned ON and be sure to check the anisotropy option.

The next menu item is **Delay time anisotropy – perform analysis**. Selecting this item causes the anisotropy calculations to be performed. The **Job** tab opens and displays the sequence of operations, similar to Figure 7---27, except that just anisotropy terms are computed.

The result of the computation is that a magnitude and azimuth term is computed for every defined refractor. The values are saved in database columns **DTA_ANISMAG_n** and **DTA_ANISAZ_n**, respectively, where '**n**' refers to the refractor number.

Figure 7---29 displays sample anisotropy data for Refractor 1. The basemap in the **Model** tab window is colored by anisotropy magnitude (**DTA_ANISMAG_1**). Additionally the **Delay time ansisotropy overlay** option is ON. This option overlays the basemap with 'sticks', that indicate the magnitude and direction of the anisotropy. The stick length can be adjusted from the **Max stick length** drop---down list. For weak anistropy, increase the stick length to scale up the overlay display. In the figure, note the correlation between the color and stick

length. If there are multiple defined refractors, be sure to select the correct refractor in the overlay section.

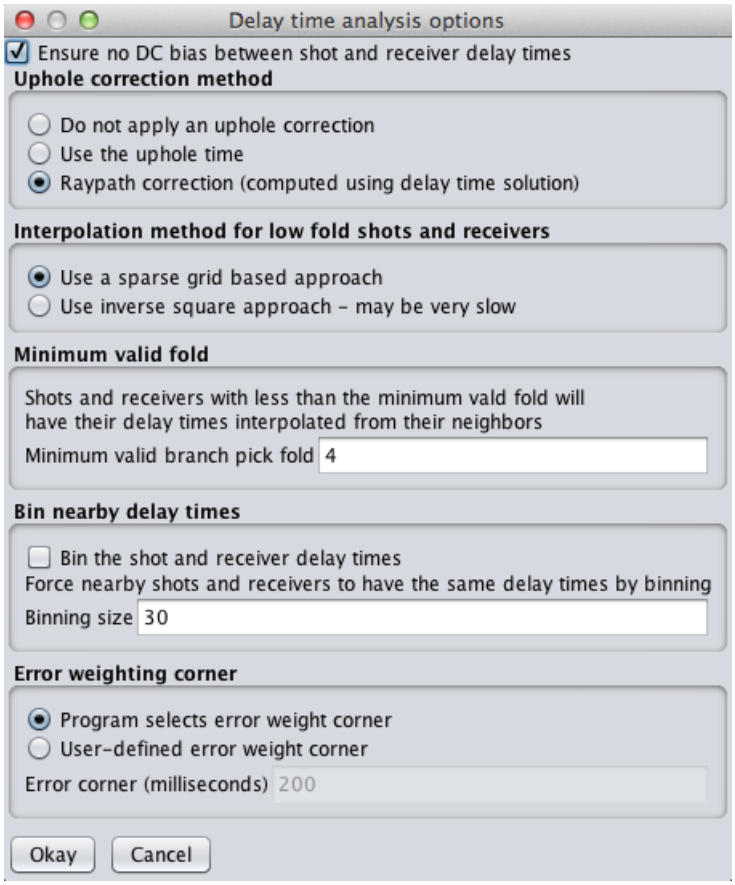


Figure 7---28 Pop---up window when selecting the Delay time analysis – Do not use anisotropy sub---menu item.

The last sub---menu item under **Analysis** is **Delay time analysis – clear results**. This item is use to delete all anisotropy data. Specifically the columns containing anistropy magnitude and azimuth information are reset to zero for all refractors in the Shot and Receiver tables.

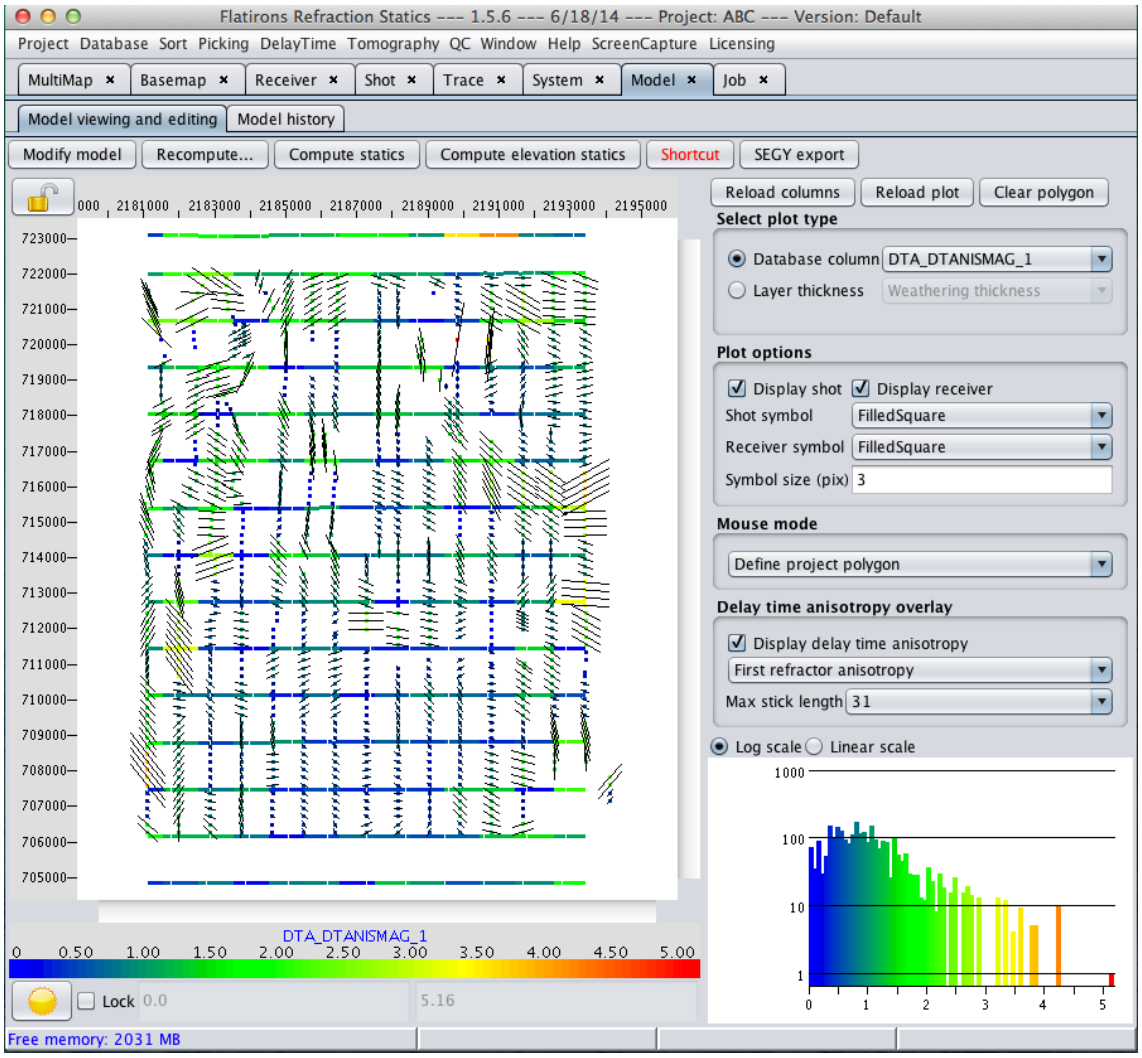


Figure 7---29 Model tab window with basemap colored by Refractor 1 anisotropy magnitude and anisotropy overlay. The 'sticks' display magnitude and direction of the anisotropy.

7.1.9 Model building window

This menu item is used to construct a refractor model based on refractor velocity and delay times obtained from delay time analysis. Selecting this item first brings up a pop---up box (Figure 7---30), reminding the user that the refractor model requires that a weathering velocity be defined and that is should be approximately one---half the refractor velocity. Next, the **Model** tab shown in Figure 7---31 is opened.

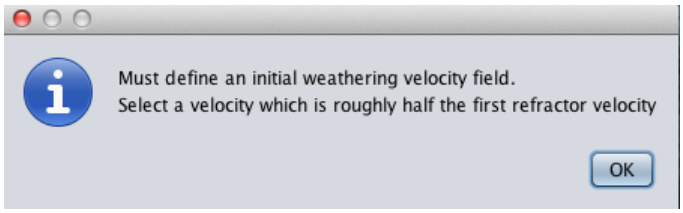


Figure 7---30 Pop---up box reminding the user to define an initial weathering velocity that is approximately one---half the first refractor velocity.

The **Model** window has two main tabs: **Model viewing and editing** and **Model history**. The **Model history** tab displays a textual history of actions related to the model, including information related to branch assignment, delay time analysis and refractor velocity computations. The information displayed is just the contents of an ASCII file called History.txt.

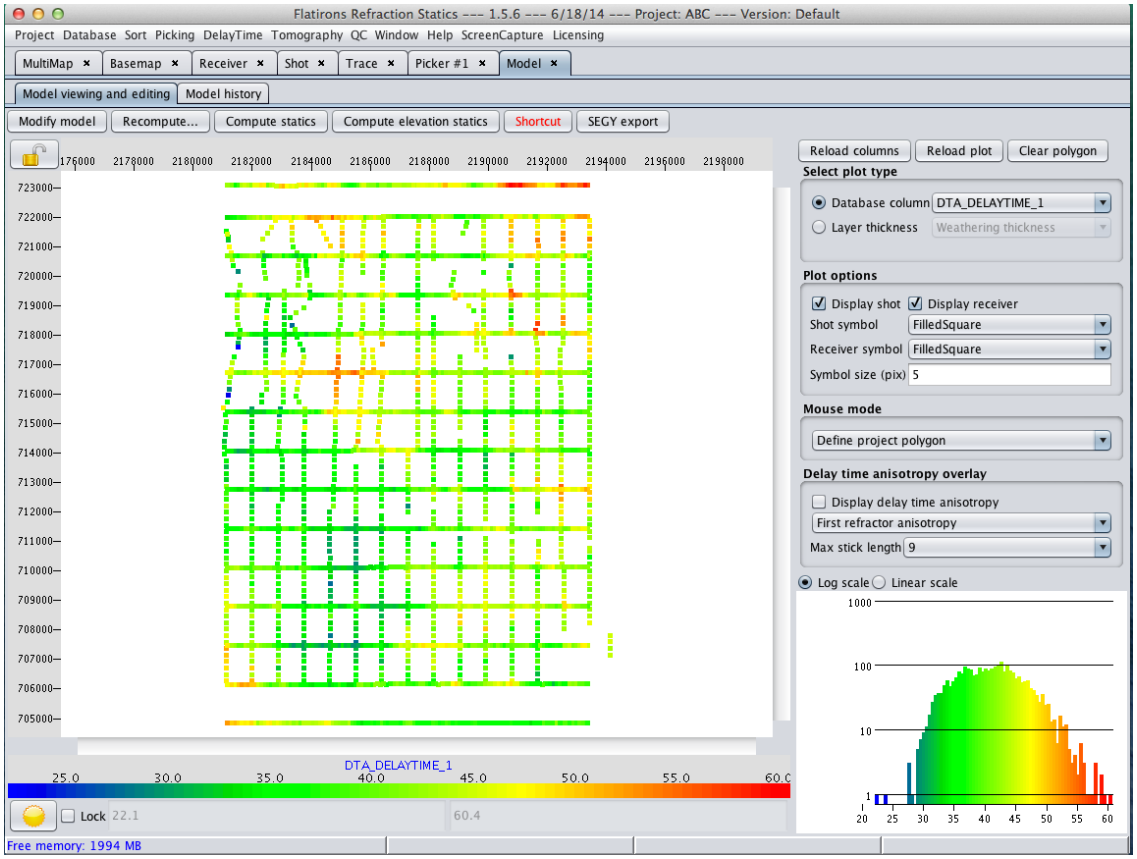


Figure 7---31 The Model tab from the DelayTime → Model building window menu item. The default view is Elevation but Delay Time was selected for this example.

The **Model viewing and editing** tab contains six buttons above the basemap display and a section for adjusting display parameters and setting plot options. The first button, **Modify model**, is used to begin the process of model building.

NOTE: For most situations model building can be performed using the shortcut method via the Shortcut button. See the end of this section for details.

Clicking the **Modify model** button opens a series of three pop---up windows where parameters needed for the model are set. The first of the three windows, called **Select attribute**, is used to set weathering velocity (Figure 7---32). When creating the refractor model, weathering velocity is the only attribute available; so simply click the **Next** button to move on to the second step.

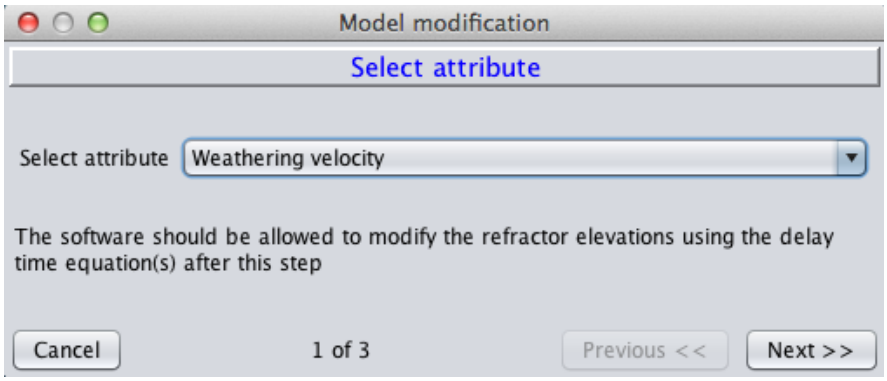


Figure 7---32 Step 1 of the Model modification process. Select the weathering velocity attribute.

The second step, **Modification method**, has several options available, as indicated in Figure 7---33. Selecting an option will adjust the window, as needed, to enter the required parameters.

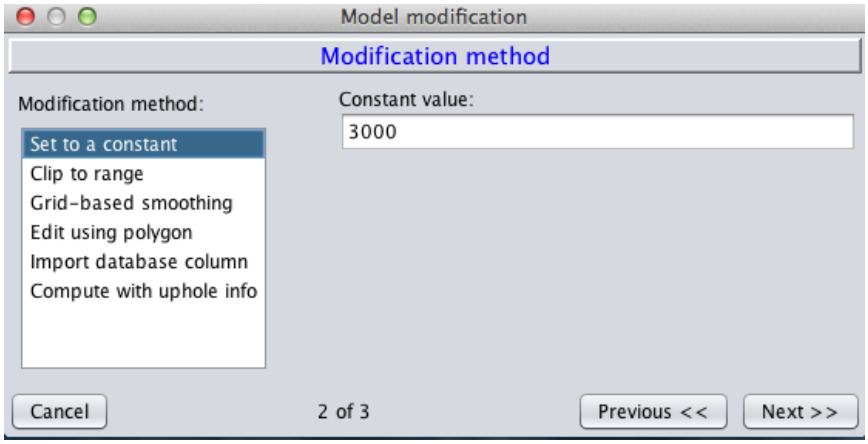


Figure 7---33 Step 2 of the Model modification process. Select the modification method. Several options are available. Here, the weathering velocity is set to a constant 3,000 ft/sec.

The Modification methods include:

1. **Set to a constant**. Set the weathering velocity to a constant value. It is recommended to select a value that is one---half the refractor velocity obtained in branch assignment procedure.
2. **Clip to a range**. Enter a minimum and maximum value for the weathering velocity.

3. **Grid---based smoothing**. Apply smoothing, with user---selected smoothing radius and optionally, select a database column to weight the smoothing. On the first pass through the model building process there is no weathering velocity to smooth, so this option does not apply.
4. **Edit using polygon**. Delete a region either inside or outside a defined polygon. Values in the deleted region are interpolated. On the first pass through the model building process there is no weathering velocity to smooth, so this option does not apply.
5. **Import database column**. If there exists a database column containing weathering velocities, perhaps imported by the user into Flatirons previously, use these values for the weathering velocity.
6. **Compute with uphole info**. If this data is available, use depth and uphole time to compute an initial weathering velocity. Optionally, restrict these values by entering minimum and maximum allowed values.

In this example, the weathering velocity is being set to a constant value of 3,000 ft/sec. On the third step, the **Attribute** (Weathering velocity) and selected **Modification** are listed for review. The only option on this window is whether to let Flatirons modify the refractor velocity using the delay time equations. Although not required, it is highly recommended to choose this option (Figure 7---34). Otherwise the refractor model will not be internally consistent.

To create the initial refractor model, click the **Finish** button. Although the pop---up window closes, nothing else appears to happen. However, to view the result of the initial calculation, select the refractor elevation (DTA_Elevation_1) for the Database column in the **Model** tab (Figure 7---31). Or, select the initial weathering velocity (DTA_Velocity_0) to check that it agrees with the modification value set earlier (in this example, a constant 3,000 ft/sec).

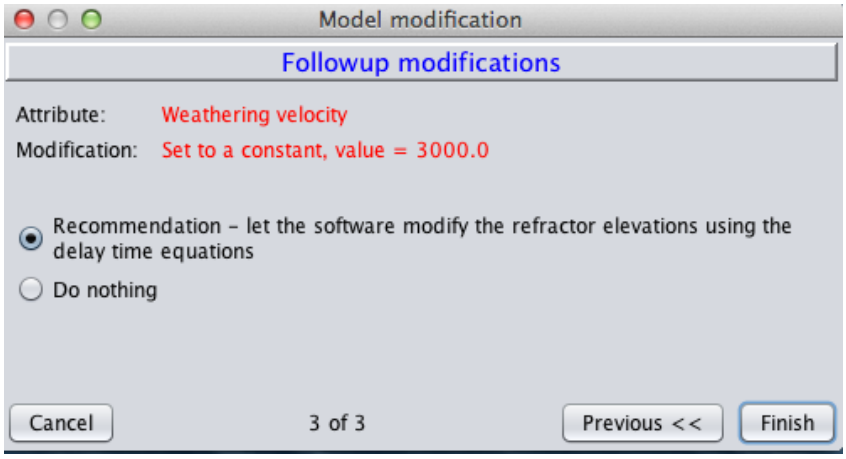


Figure 7---34 Step 3 of the Model modification process. Optionally, use delay time equations to modify the refractor elevation. Highly recommended.

Clicking on the **Modify model** button again and selecting another attribute to modify can improve the initial refractor model. Note that the drop---down list of attributes now has several attributes available, not just the **weathering velocity**. In this example, the refractor shape will be smoothed. For this, the **Refractor 1 elevation** attribute is selected (Figure

7---35). The effect of smoothing the refractor will be to alter the weathering velocity, allowing it to deviate from the (arbitrary) constant velocity chosen when creating the model.

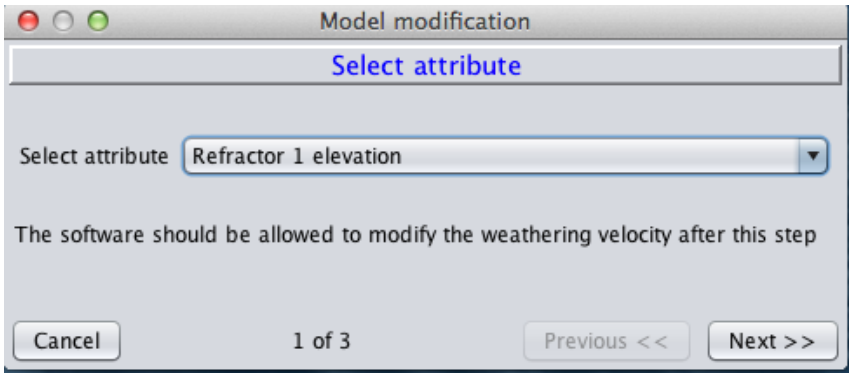


Figure 7---35 Second pass through Model modification. Attributes other than weathering velocity are now available to be modified.

In the next step, the Modification method chosen is **Grid---based smoothing** (Figure 7---36) with a smoothing radius of 2000 ft.

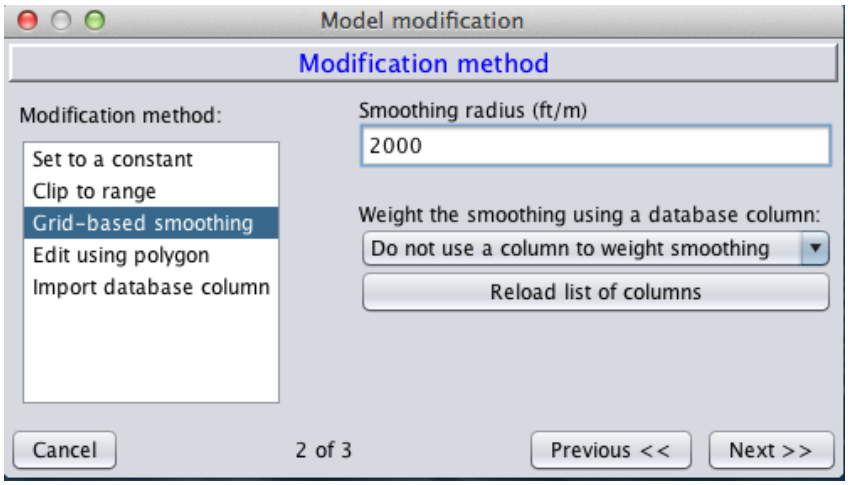


Figure 7---36 Applying smoothing to refractor 1.

Again, the third step reviews the attribute and applied modification and recommends that Flatirons be allowed to maintain a consistent model (Figure 7---37). In this case the weathering velocity will be altered as the refractor is smoothed.

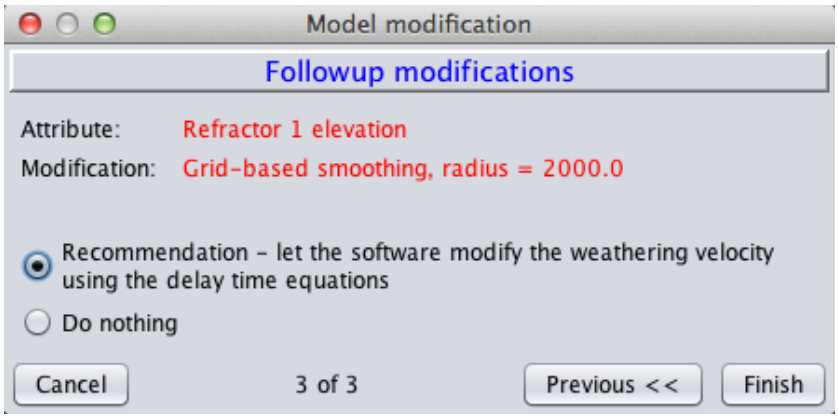


Figure 7---37 The third step of Model modification. The recommendation is to let Flatirons alter the weathering velocity in the process of smoothing the refractor.

Click the **Finish** button to complete the modification. To see the changes select the refractor elevation (DTA_Elevation_1) for the Database column in the **Model** tab (Figure 7---31). Or, select the weathering velocity (DTA_Velocity_0) to verify that it now varies from the constant value it had initially.

At this point, profiles of the model can be viewed by selecting the **DelayTime → Model profile display window** menu item. See Section 7.1.12 for details.

The remainder of this section will cover the five remaining buttons in the **Model viewing and editing** tab window.

The **Recompute** button can be used to recompute model parameters if needed. Clicking on this button opens a selection box (Figure 7---38) to force a calculation of refractor elevations based on the current weathering velocity or vice---versa. This can be useful if the underlying data has been altered.

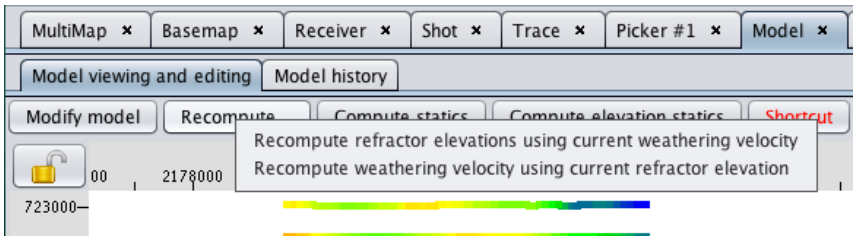


Figure 7---38 The Recompute button opens a selection box to recompute refractor elevations or weathering velocity.

The **Compute statics** button begins a four---step procedure to compute statics based on the current model. Clicking on the button opens up the first of four dialog boxes. Called **Name of statics column**, this dialog is used to specify the database column where the statics will be saved (Figure 7---39). The actual database colunm name will have '**STATICS_**' prepended to whatever the user specifies.

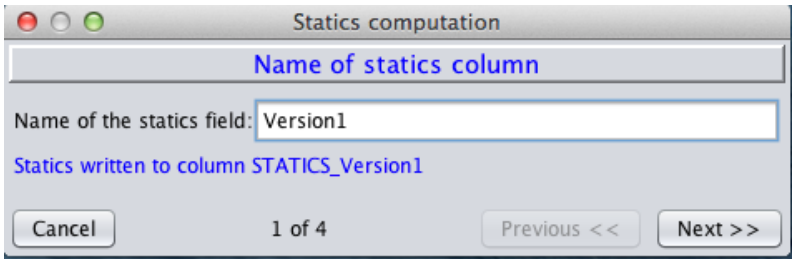


Figure 7---39 Dialog box 1 of 4 for statics computation. Specify the database column where the statics are saved.

The second step in computing statics is specifying the intermediate datum. Figure 7---40 shows the dialog box for this step. The drop---down list is expanded to show the four options available. Depending on which option is selected, the dialog box will be updated for the relevant parameters. The dialog box shows the minimum and maximum elevations for the surface and all defined refractors in the model.

In this example, the elevation of Refractor 1 has a ranged of about (---85, ---18) feet. So, choosing an intermediate datum at constant elevation a reasonable fixed elevation is ---60 ft.

Note that at any step in the process, the user can return to a previous step and alter the configuration of the statics computation.

The third step in computing statics is specifying the final datum. Figure 7---41 shows the dialog box for this step. In this case there are only two options for the final datum: constant elevation and fixed depth below a smoothed surface. In addition to the elevations that the previous dialog box displayed, this dialog box displays the minimum and maximum velocities of the weathering layer and all defined refractors.

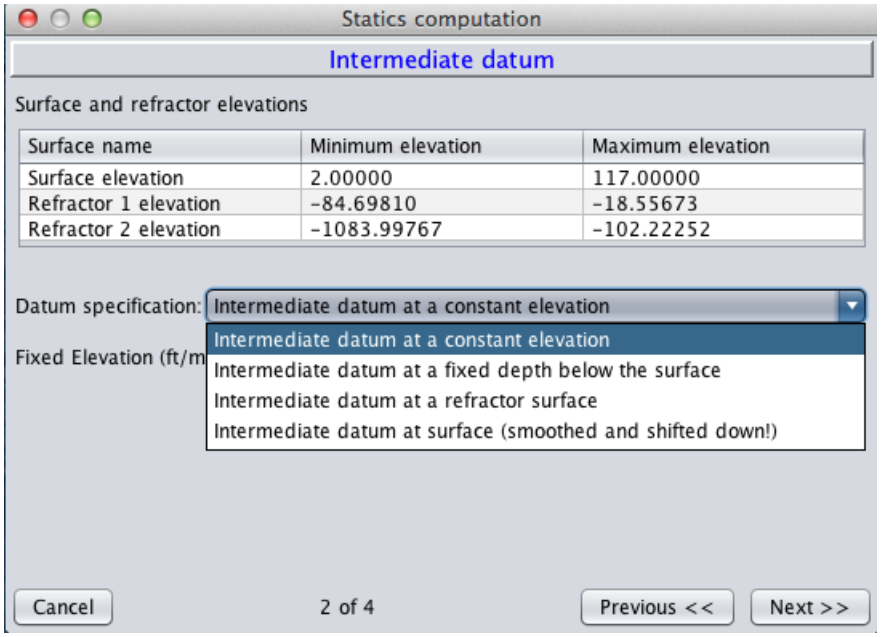


Figure 7---40 Dialog box 2 of 4 for statics computation. Specify the Intermediate datum. Drop---down list is expanded to show all options.

In this example, the final datum is chosen to be at a fixed elevation of 120 ft. The replacement velocity is chosen to be approximately the velocity of Refractor 1 for which 6,000 ft/sec seems reasonable.

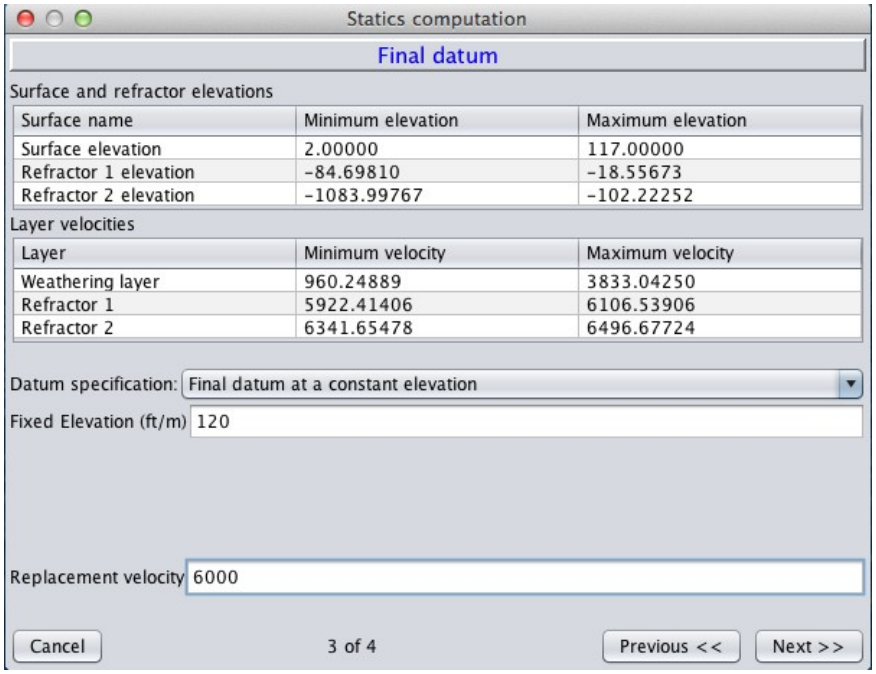


Figure 7---41 Dialog box 3 of 4 for statics computation. Specify the final datum and replacement velocity.

The fourth and final step in computing statics is optionally applying a bulk time shift. Generally, the final product would not have a bulk time shift applied. However, for

comparison purposes it may be useful to shift the final statics to a zero mean or the same mean as some other statics field. Figure 7---42 shows the final dialog box, with toggle buttons for choosing the bulk shift option. If the third option is chosen, to tie the statics to some other statics field, the dialog box opens a field for the user to select the desired database column containing the other set of statics.

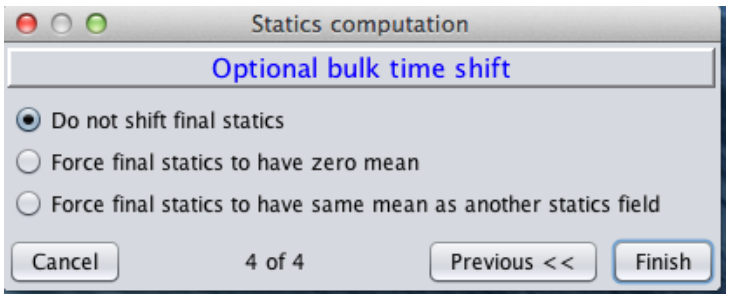


Figure 7---42 Dialog box 4 of 4 for statics computation. Optionally apply a bulk time shift.

After clicking the **Finish** button, Flatirons will compute statics (but not give any indication of having done so). To see the statics, in the **Model** tab, click on the **Reload columns** button to update the drop---down list, and then select the database column containing the statics. In step 1, the statics field was named 'Version1' so the database column is **STATICS_VERSION1** as shown in Figure 7---43. The statics are displayed on the basemap and do appear to have zero mean. The histogram at the bottom right clearly shows a few outliers at the high end; these are almost certainly due to uncorrected geometry errors.

The basemap display in the **Model** tab window has the usual options like locked axes, color bar features, plot options, etc. One additional option is the **Delay time anisotropy overlay**. Because anisotropy was not included in the delay time analysis for this example, this option has no effect. Later sections in this chapter will return to this feature.

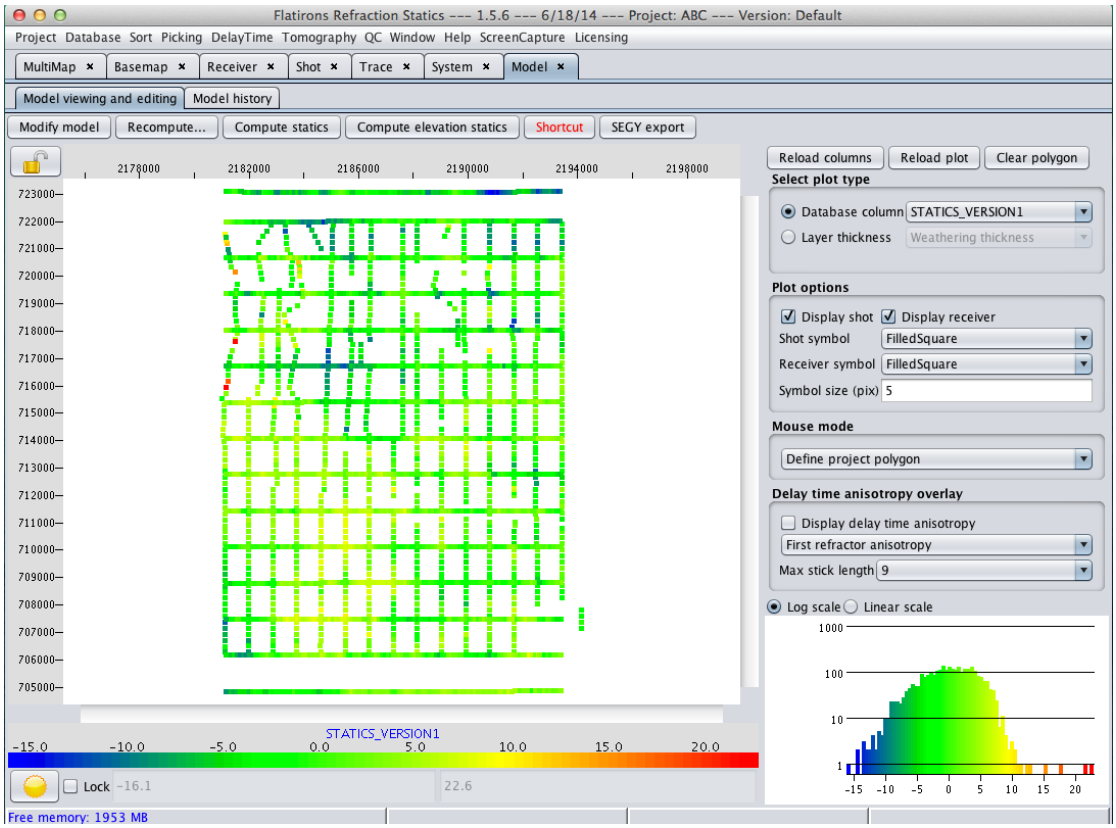


Figure 7---43 Result of the statics computation. The statics do appear to have zero mean.

The next button is **Compute elevation statics** and its function is, of course, to compute elevation statics. Clicking on this button opens the dialog box shown in Figure 7---44. The user enters the statics name, used to determine the name of the database column, and the final datum elevation and replacement velocity. Optionally, these statics can be forced to be zero mean.

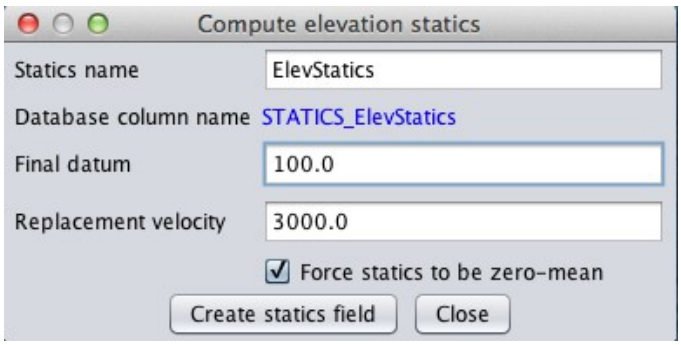


Figure 7---44 Dialog box for computing elevation statics.

Click the button to close the dialog box and perform the computation. Again, **Flatirons** gives no indication that statics were computed. To see the elevation statics, in the **Model** tab, click on the **Reload columns** button to update the drop---down list, and then select the database

column containing the statics. The statics field is **STATICS_ELEVSTATICS** as shown in Figure 7---45.

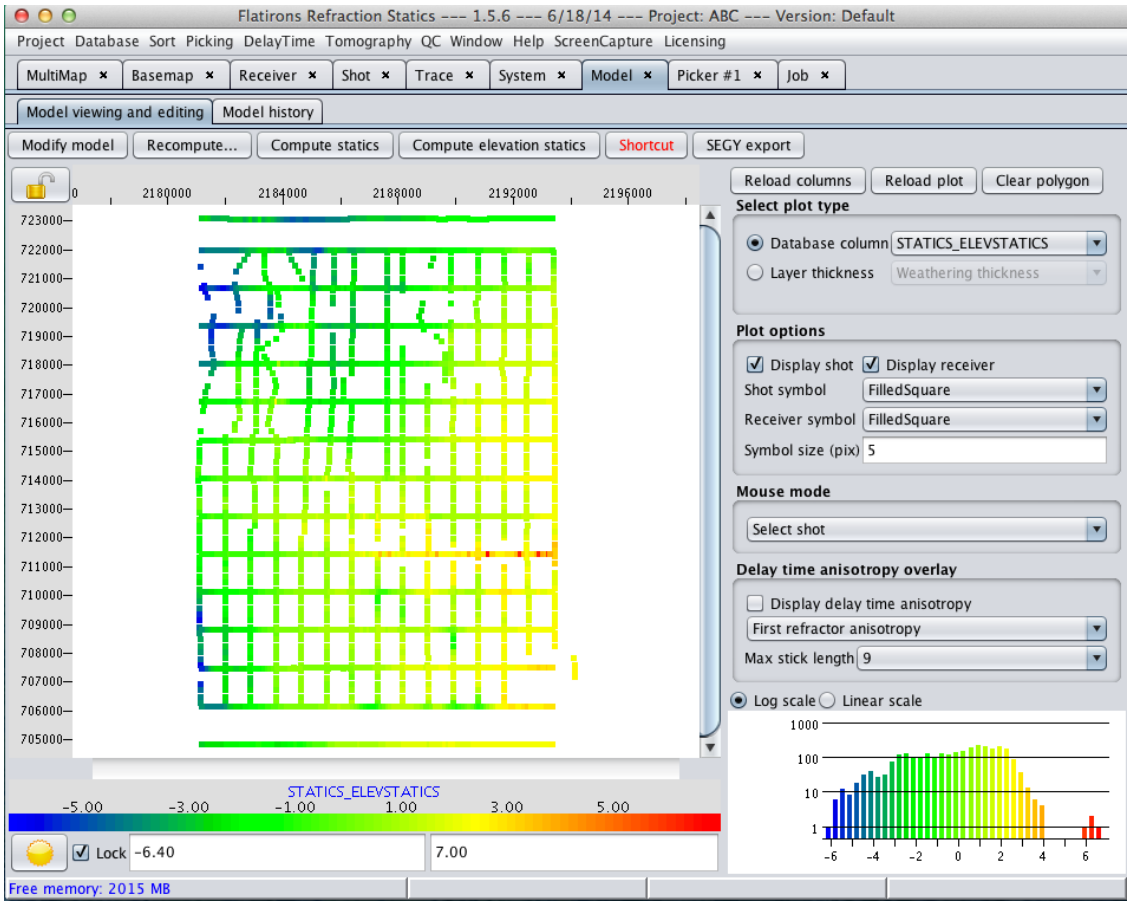


Figure 7---45 Model tab with basemap set to display elevation statics.

The **Shortcut** button is used to quickly build a velocity model. If no advanced options, such as using a polygon (see, for example, Figure 7---36) then the shortcut method is recommened. Clicking on this button opens a dialog box (Figure 7---46) requesting the weather velocity and the refractor smoothing radius.

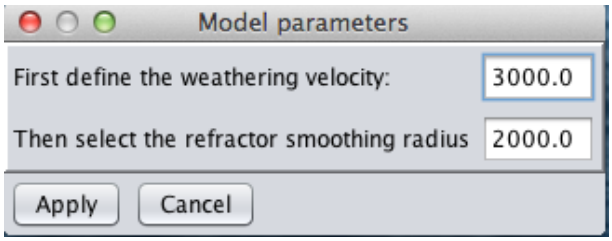


Figure 7---46 Dialog box for the Shortcut method of model building.

Click the **Apply** button to complete the process. Figure 7---47 shows the Refractor 1 elevation after applying the shortcut method, using the parameters shown in Figure 7---46. These are the same values for weathering velocity and refractor smoothing radius that was used for

the regular model building method (see for example, Figure 7---34 and Figure 7---37). The refractor 1 elevation range can be read from the axis in Figure 7---47. It is the same range as was obtained by the regular method (see Figure 7---41): approximately (---84.6, ---18.5).

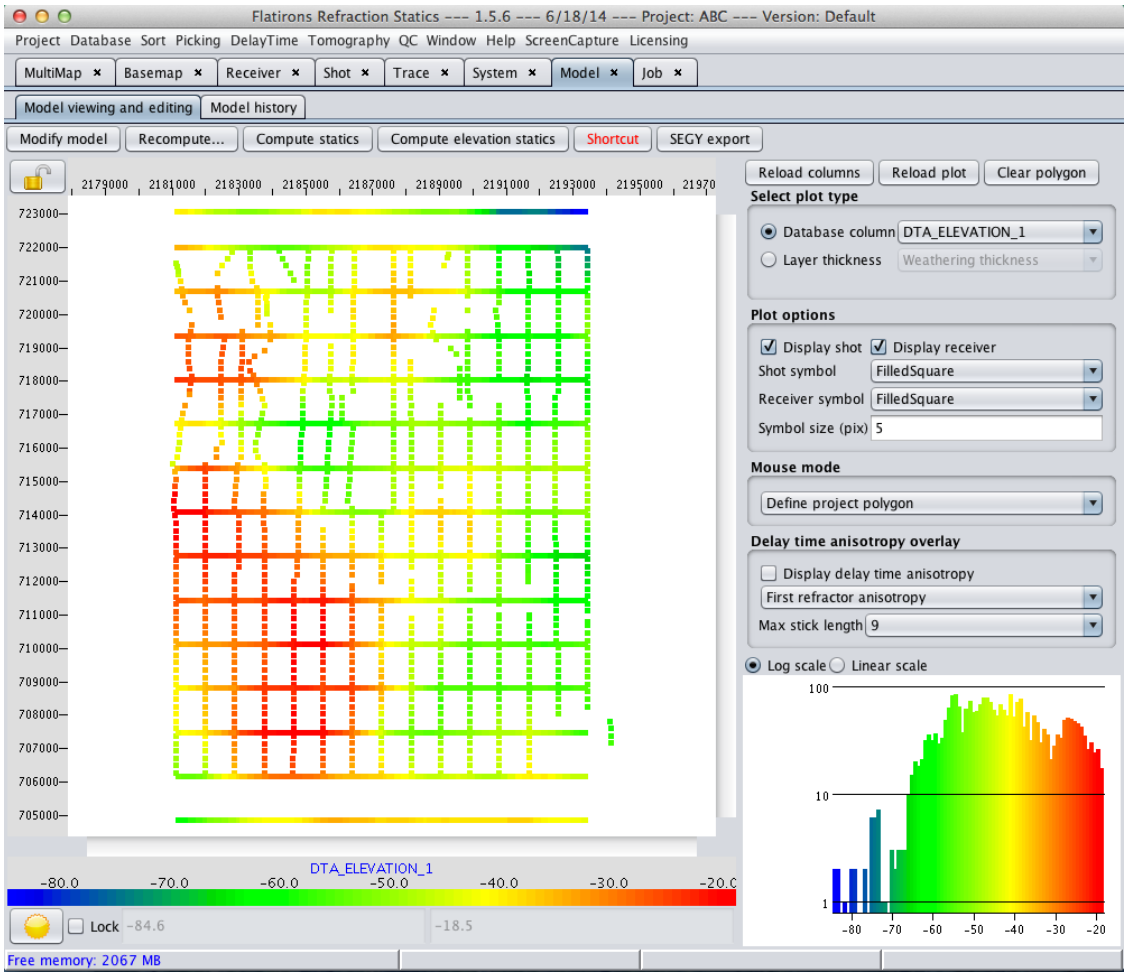


Figure 7---47 Refractor 1 elevation from the Shortcut method.

Finally, the **SEGY export** button is used to export the velocity model to a SEGY file. Clicking on the button automatically performs the export and saves the model data to the file **ModelVel.segy** in the project folder.

7.1.10 Model profile display window

Once a delay time solution has been computed and a refractor model built, the model can be studied by viewing it along profiles across the survey. The **Model profile display window** menu item opens the **DT model profiles** tab window as shown in Figure 7---48. By default, three plots are visible but 1---4 plots can be displayed via a drop---down list. Each plot has individual plot options to the right of the plot, including scale locking and the standard color bar features.

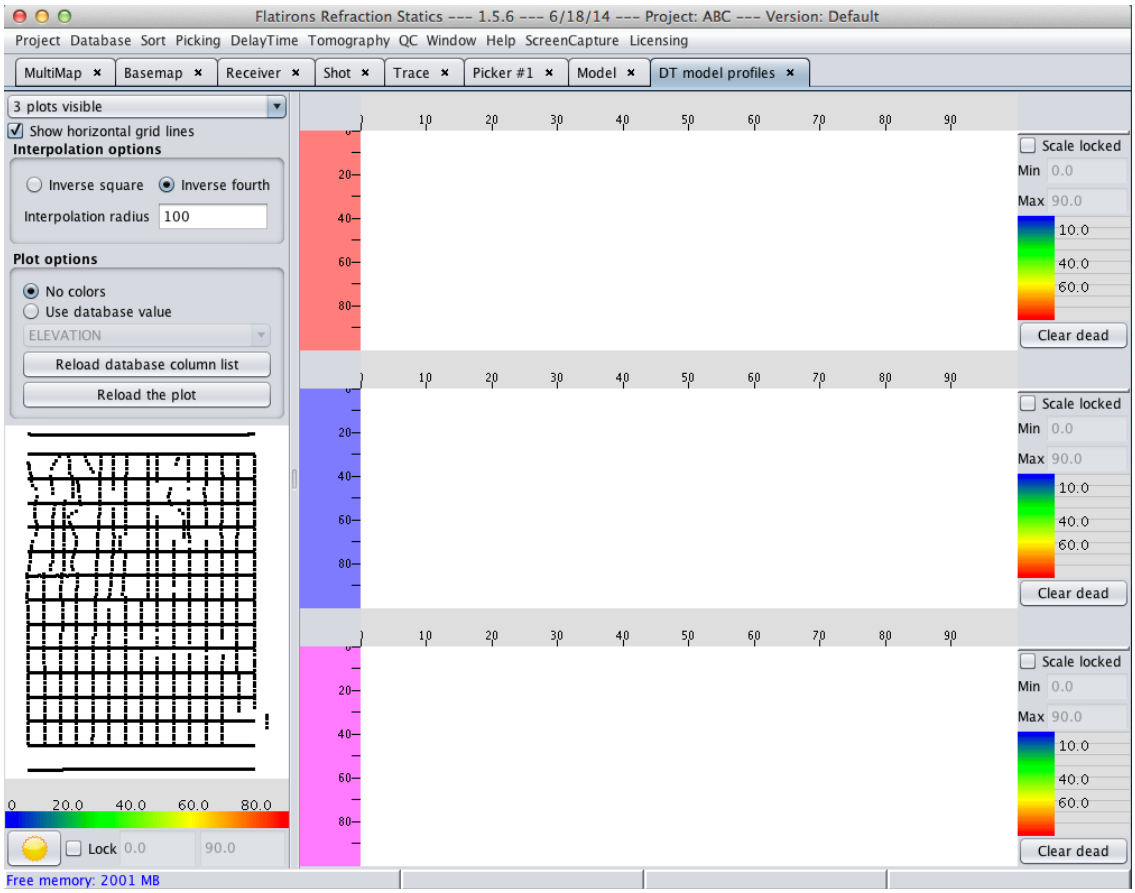


Figure 7---48 The default Model profile display window.

Figure 7---49 shows the Model profile display window with three profiles. To define a profile, left---click and drag a line across the basemap. Release the mouse button to complete the definition. The appropriate plot will painted with the model data along that profile. The profiles are color---coded: the color of the profile line on the basemap matches the color of the vertical axis on the plots (in order: red, blue, magenta, yellow).

The profiles in the example shown in Figure 7---49 have two refractors, each with constant but different velocities (red vs. orange). The elevations of the refractors are non---constant, with the deeper refractor having much greater variability. The weathering velocity (blue/green) varies across the profiles but in general is about half the first refractor velocity.

The profile data can be smoothed by setting the **Interpolation options**. Either **Inverse square** or **Inverse fourth** smoothing can be selected, as well as the **Interpolation radius**. These options must be set before defining a profile.

The **Plot options** section applies to the basemap. It can have no color (default) or be colored by values in a user---selected database column.

Profiles are defined in a cyclical manner. For example, if three plots are visible, all with populated profiles, defining a fourth profile will cause the first profile (top plot) to be replaced by the new profile. Defining a fifth profile will replace the second profile, and so on.

At any time the number of visible plots can be reduced and the remaining plots will expand to fill the display. The profile information is temporarily saved however, so increasing the number of visible plots will re---paint the plot of the recently removed profile.

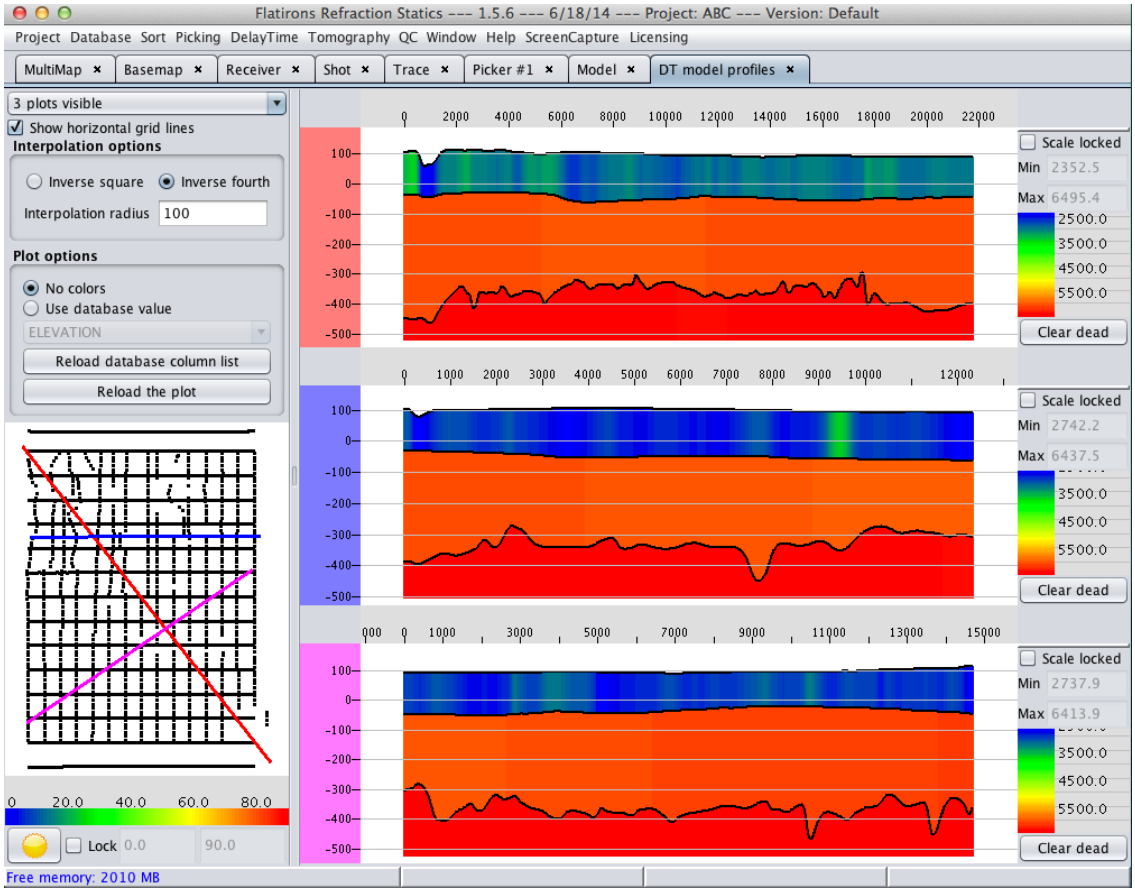


Figure 7---49 The Model profile display window with three profiles.

Figure 7---50 shows a contrived example of a two---refractor model to highlight other features of the window. The horizontal axis marker lines have been toggled off, only one plot is visible, and the basemap is colored by the second refractor's elevation (DTA_Elevation_2). The basemap is also zoomed to show a very short profile. In the basemap, the two blue arrows point to two large dips in the second refractor elevation near the middle of the profile. Both dips are quite obvious in the profile display.

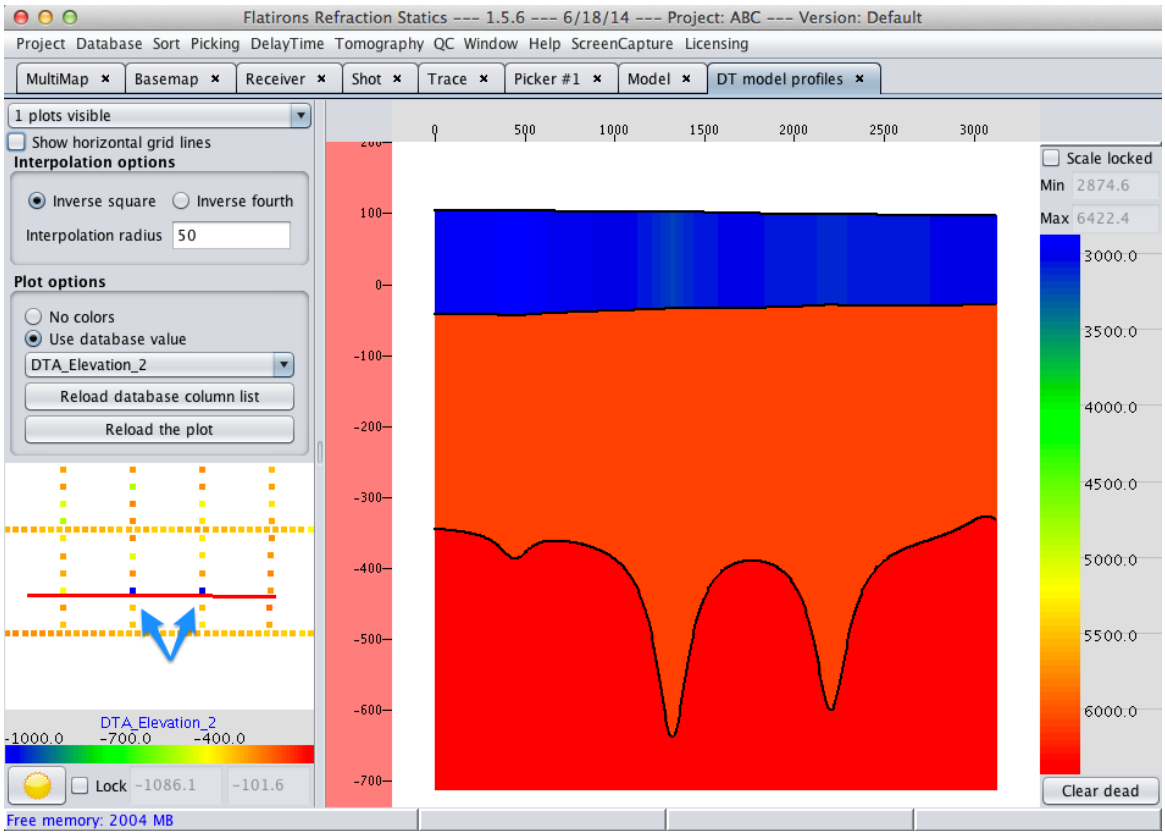


Figure 7---50 Another view of the profile window showing various features. The blue arrows overlaid on the basemap indicate two dips in the second refractor's elevation.

8 Tomography

The Tomography menu items accesses the 3D tomography (tomo) module. The Flatirons tomography method is based on a very fast implementation of an Eikonal solver and has options for node spacings, maximum offsets, iterations and display options.

Finite difference, 3D same as 2D, get velocity from branch assignment window(?), ray density (where rays spend most of time(?) Or hit count?), multithreaded, approx. 100M traces in about 10 hours.

From tutorial: QCing the resulting tomo model using the Picker window. Second check (?): Shot/Receiver tomo stacks.

Tomography → 3D Eikonal Tomography

The Tomography module is accessed from the **Tomography → 3D Eikonal Tomography** menu item. A new tab called Tomo3D is opened as shown in Figure 8---1. The window has two panels, each with multiple tabs. In the right panel the Map View tab has a drop---down list to select several tomo---related features, such as velocity, node hit count or statistical weight. Depending on the feature selected, an elevation (positive or negative) can be selected or a depth can be entered. Checkboxes allow for the positions of Shots and Receivers to be displayed also.

The **Profile** tab, as displayed in Figure 8---1, has two plots: the top one is the tomo velocity model and the bottom one is a plot, called **Node sampling**, of either the approximate node hit count or the statistical weight. The approximate node count is an interpolated measure of the number of rays passing through an elemental volume of the tomo model. The statistical weight is similar to node count but incorporates the error between ray travel time and pick time. Generally, the greater the node count or statistical weight in a region, the more accurate the calculated velocity will be. More precisely, the plots represent a vertical section, or profile, of the tomo model.

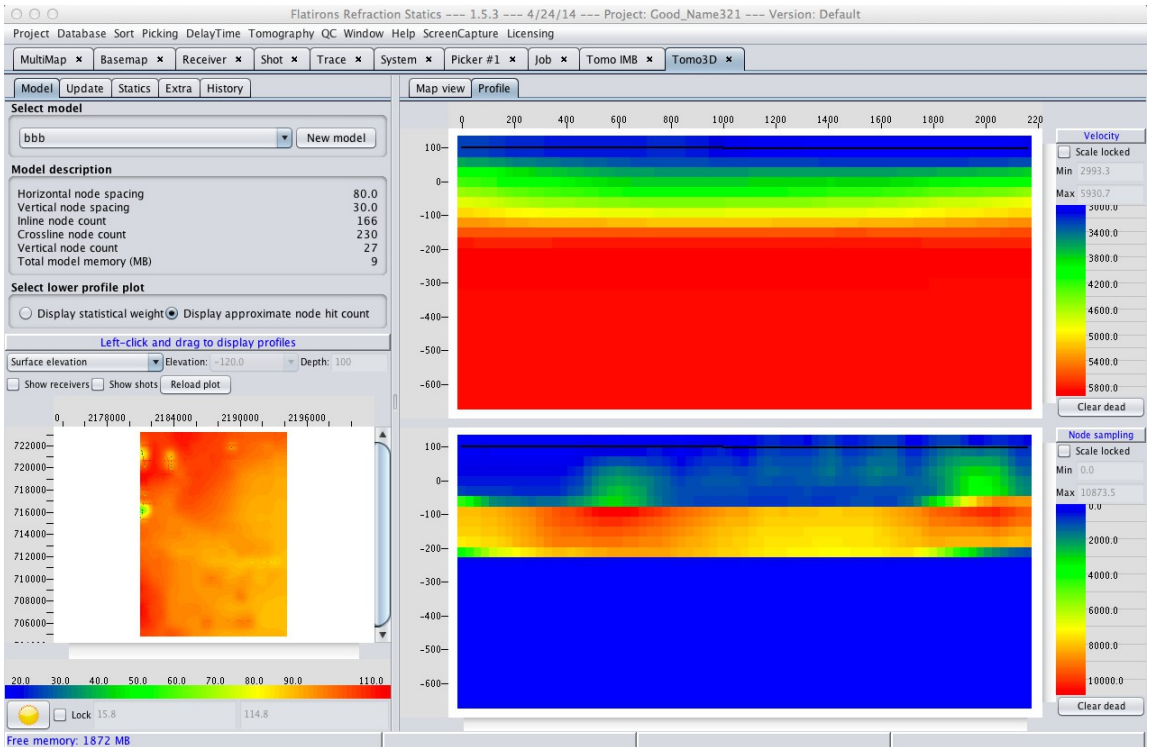


Figure 8---1 The Tomo3D tab, for creating and viewing tomo models.

The left panel has five tabs, relating to different aspects of the tomo model. The **Model** tab is used for viewing the results of a model or creating a new model. A map view, with the same settings as in the right panel appears on the bottom section. To define a profile, the user uses the mouse to left---click and drag a line across the map. The profiles in the right panel are updated continuously, until the left mouse button is released. Any previously created tomo model can be selected from a drop---down list or a new model can be created.

Clicking on the **New model** button opens the Create new **3D tomography model** window as shown in Figure 8---2. The tomo model is constructed using a rectangular grid of nodes with user specified horizontal and vertical spacing. More precisely the 3D model is a rectangular cuboid. The advantage of the rectangular cuboid over a cubical model is that vertical grid spacing can be set to a much smaller length than the horizontal dimension, which, for the near surface, is more natural. In this way the depth can be sampled at a high resolution without wasting computational resources on unnecessary 'horizontal' nodes.

The user sets the orientation of the rectangle by dragging a single line segment across the base map. A rectangle is automatically generated that encompasses the survey. The Grid angle has no intrinsic meaning – the purpose is to define the smallest rectangular region necessary so that computational resources are not wasted on unused nodes. Re---defining the rectangle can be repeated as often as desired. Optionally, a Grid angle value can be entered directly.

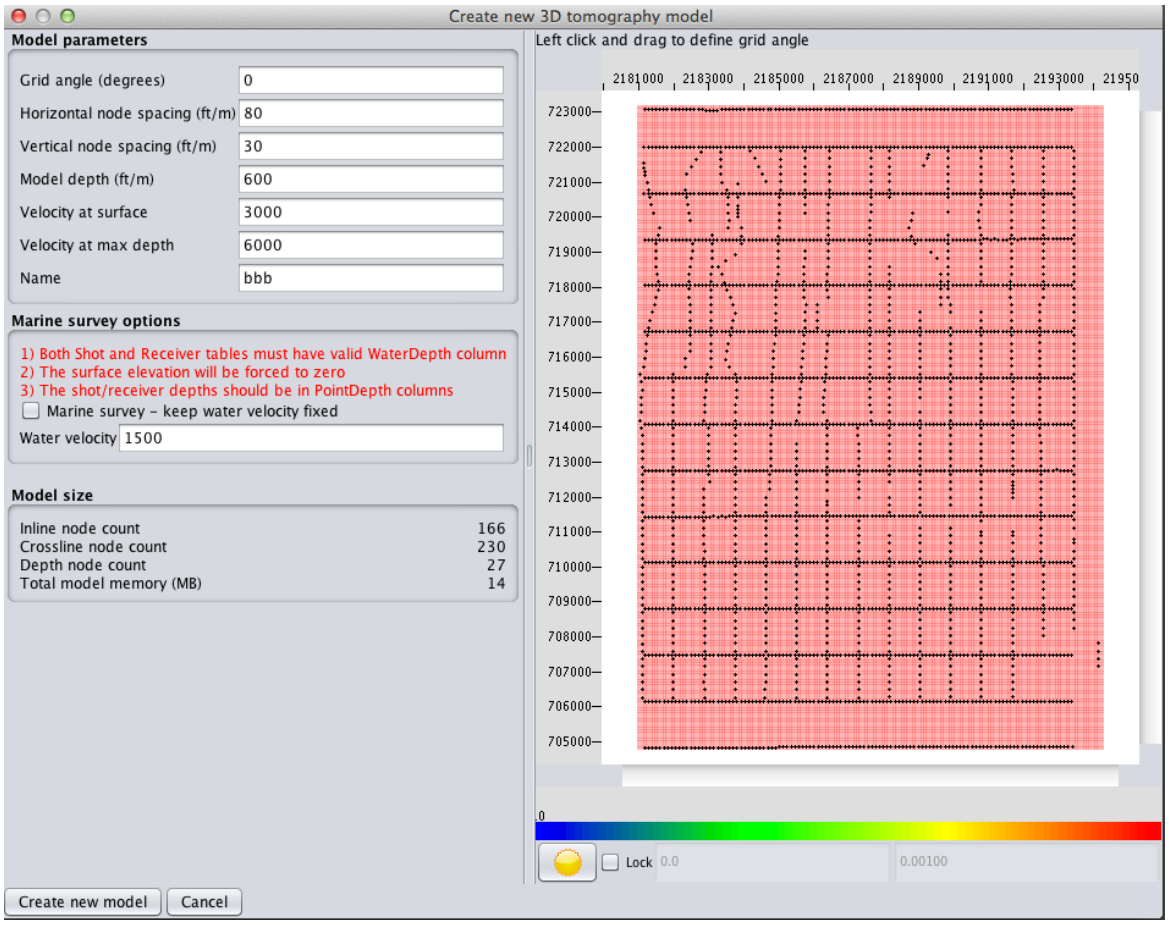


Figure 8---2 The window for creating a new tomo model.

After setting the grid angle, values for the horizontal and vertical node spacing, total model depth, velocity at the surface and maximum depth and name of the model are entered. This defines the tomo model.

For marine surveys there is an additional option – whether to keep the water velocity fixed. To apply this option it is necessary to have a valid WaterDepth column in the shot/receiver tables. If this option is chosen, the user must enter the desired water velocity. After all data has been entered the model dimensions are computed and displayed in the **Model size** box. The dimensions consist of node counts for Inline, Crossline and Depth, as well as the amount of memory required for the tomo model. If all parameters and dimensions are satisfactory the model is built (but not yet run) by selecting the **Create new model** button.

To actually run a tomo model, the **Update** tab is used. In this tab, various options are set that determine how the model is applied. As shown in Figure 8---3, in Batch update options, the offsets, number of iteration and number of threads (up to the CPU maximum) are set. Two settings related to the tomo algorithm are **Offset weight corner** and **Smooth radius**. The first setting adjusts an offset---related weighting factor that is applied to traces. Increasing this value increases the importance of larger offsets. The **Smooth radius** is a scale factor for smoothing the velocity model. Any saved set of picks can be applied, using a

drop---down list. **Pick shift** is a user---specified time shift to apply to all traces. Conceptually, the reason for having such a shift available is because tomography identifies first arrival of a ray but user picks generally do not. That is, the user picks generally correspond to a separate event, such as maximum amplitude (peak or trough) that occurs a small time later than the first arrival. This difference in time is assumed to be the same for all traces and the **Pick shift** compensates for the lag. So, typically, **Pick shift** will be some small negative value.

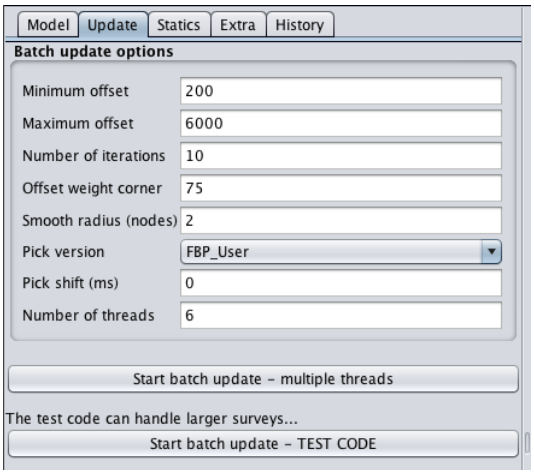
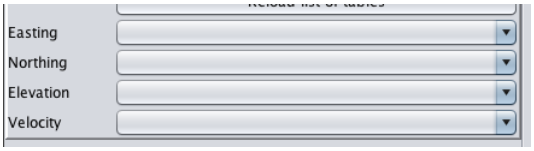
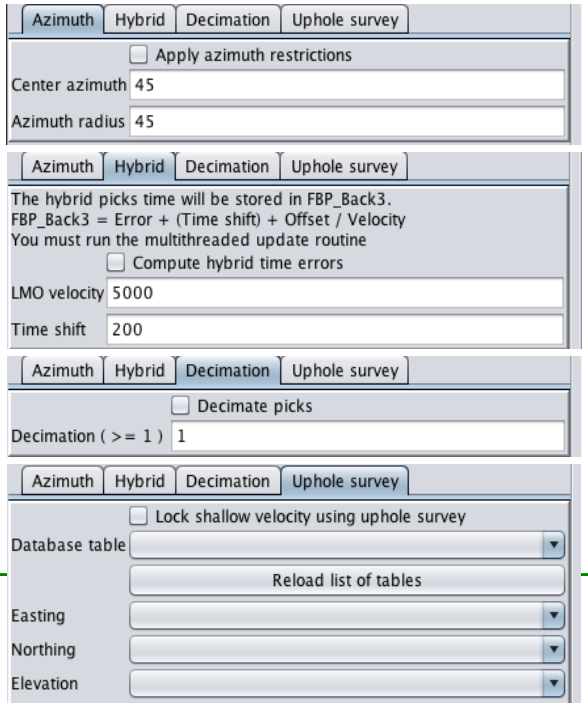


Figure 8---3 The Update tab for Tomo3D. Options for the tomo model and batch process are set here.



One of two buttons begins the batch process that builds a velocity model using tomography. The first is **Start batch update – multiple threads** and the other is **Start batch update – TEST CODE**. The former is the standard method, even if there is only one thread. The threaded update works by preparing a separate tomo model for each thread. The advantage is speed, while the disadvantage is memory usage. The so---called 'TEST CODE' method uses only one tomo model in memory and the threads have synchronized access to this one model. While it is slower it does allow for much larger surveys. Which model is preferable depends on the computing resources available to the user. This is why the Model description on the **Model** tab displays an estimate of the total memory required by the model.



At the bottom of this panel is another set of tabs for specialized tasks related to the tomo application, see Figure 8---4. **Azimuth** has options to apply azimuth restrictions to the tomo model. This option restricts traces that have the given **Center azimuth** plus/minus the **Azimuth radius**. For example, if Center azimuth = 45° and Azimuth radius = 15° then only traces with azimuth in the range 30° --- 60° are included (and, by symmetry 210° --- 240°). The **Hybrid** option uses a combination of tomo and delay time analysis. **Decimation**

permits restricting the number of picks to use. For example, if Decimation = 4 then only 25% (every 4th) of the picks are used. This option is useful for doing a quick check on the tomo model configuration. Finally, **Uphole survey** can be used to pin the tomo velocities to provided uphole velocities. The uphole information is required to be imported into a database table and each of the appropriate columns in that table matched to Easting, Northing, Elevation and Velocity using drop---down lists.

The **Statics** tab is, of course, used to prepare for and compute statics based on the tomo model. Proceeding down the tab (Figure 8---5), the user first selects the desired positioning of the intermediate datum using a fixed elevation, depth or velocity. Next, final datum options include the position of the final datum and the replacement velocity.

The Residual correction options relate to the discrete nature of the tomo model and its inherent scale lengths. By checking Apply residual correction the application will adjust the statics for identified errors, generally high---frequency in nature. The Residual multiplier determines how much correction to apply. Specifically, it multiplies the value stored in the TOMO3D_RESIDUALERROR column in the Shot/Receiver tables. These columns are populated if the residual correction option is selected and computed during the batch update.

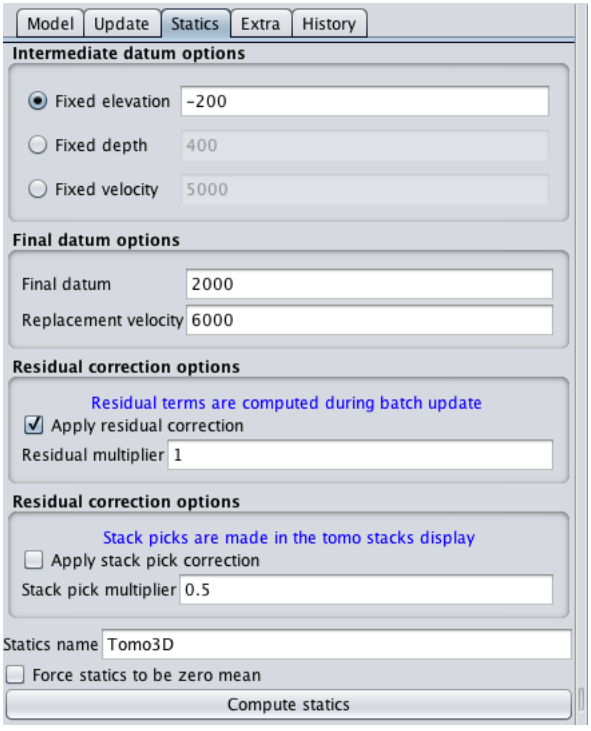


Figure 8---5 The Statics tab, used for preparing datum and residual correction options and computing statics.

The other type of residual correction is selected using the **Apply stack pick correction** checkbox. The stack pick correction is stored in the TOMO3D_STACKPICKTIME column in the Shot/Receiver tables. These columns are populated only if tomo QC has been applied from the **Tomography → Shot and receiver stacks using 3D Eikonal tomography solution** menu option. This option, the associated window and stack pick times will be covered later in the chapter.

Each set of statics is given its own name by the user, separate from the tomo model name. Finally, a checkbox can force the statics to be zero mean. This can be useful if comparing to other models. When all options have been selected, click on the **Compute statics** button. Although nothing appears to happen, the STATICS_TOMO3D columns in the Shot/Receiver tables are populated.

The **Extra** tab holds options that didn't belong elsewhere. First, a weathering velocity can be computed and applied to a delay time model. The user can select the desired depth and whether the new velocity should be used to update the first refractor elevation in the delay model.

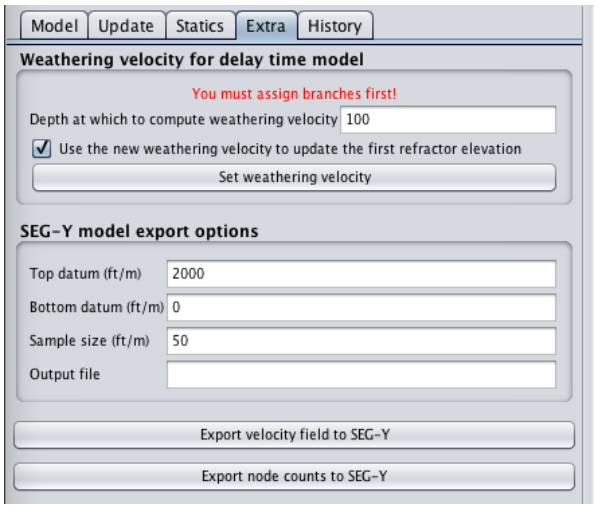


Figure 8---6 The Extra tab, with options for setting a weather velocity and exporting tomo data.

The other option pertains to exporting the results of the tomo calculation. As shown in Figure 8---6 the export is done in SEG---Y format. To export the velocity field, first enter the Top datum, the Bottom datum, the Sample size and the name of the Output file. Then click the **Export velocity field to SEG---Y** button. Two things to note: one, for **Output file**, the full path for the desired directory and file name must be specified; two, exporting the node counts will save the data to the same output file, overwriting the velocity field SEG---Y file. So, if exporting both velocity field and node counts, be sure to specify a different file name in between the two exports.

Finally, the History tab displays a history of the current model, from the date it was created to the present. Included in the history are model parameters set on the **Update** tab, batch

update information such as iterations and errors, and statics options selected in the **Statics** tab. Figure 8---7 shows a partial listing of the history for a tomo model.

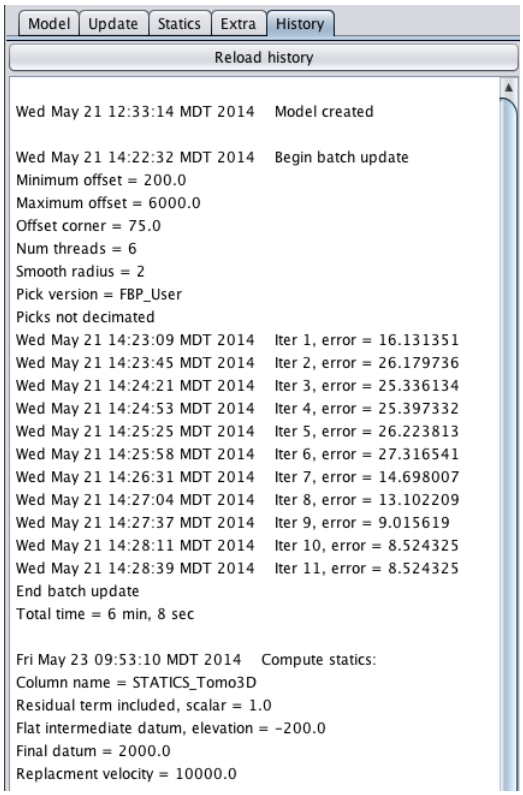


Figure 8---7 Partial listing of the History tab for the current tomo model.

9 Stacking Velocity Picker

Opens the **Stack velocity** tab. Need a velocity model in order to create stacks. Semblance velocities, CMP velocities. Note: velocities derived here are not appropriate for processing; they have not been de---convolved, etc. Velocity parameters in the **Velocity function options** panel default to quite large ranges; the user will almost certainly need to adjust these settings.

10 Window

This menu item has an assortment of options related to the appearance of windows. Figure 10---1 shows these options. By 'window' it is meant the windows associated with named tabs. For example, if the current active tab is **Basemap**, then selecting the **Close window** option with close the **Basemap** tab. Similarly, **Close all windows** will close all the currently open tabs. The option **System window (errors, environment...)** opens the **System** tab, described in more detail in section 14.5. The **Show kill button on tabs** option toggles whether to add the kill button '**x**' to the tab, to the right of the tab name. A left---click on the button will close the tab and is equivalent to the **Close window** option, regardless of whether it is the active window.

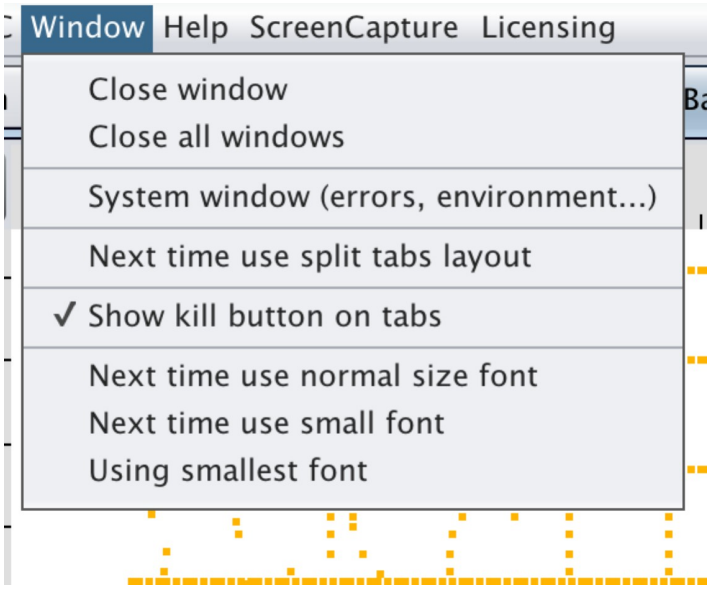


Figure 10---1 Window menu items.

The remaining options modify how the tabs and text will appear the next time Flatirons opens.

11 ScreenCapture

This menu item provides a capability to take a screen capture and save the image. Three options are available; the first two are **Entire screen** and **Active window**. A third option **Ask if the image should be saved to disk** (default: ON) toggles whether the user wants to select the file name and destination. Selecting either option takes the screen capture of the desired area. If the third option is ON (i.e., checked) a dialog box (not shown) is opened for naming and saving the image. The file type of the image is JPEG (with .jpg extension). If the third option is OFF (i.e., unchecked), the image is simply saved to the clipboard.

12 Licensing

See section 2. In order for Flatirons to run a licensing server must be set up. This menu item is deprecated, with other means of configuring the license server being preferred. Please consult your IT department for any questions about licensing.

For completeness, the Licensing menu item has two similar options: **Set primary licensing server** and **Set secondary licensing server**. Both open a dialog box (not shown) where the user enters the port used by the XtremeGeo (CanonGeo?) vendor daemon. [Are both required? Are they necessary in order to run Flatirons? Explain.]

13 Main Project Window

The main project window (**Error! Reference source not found.**) is the window displayed when a project is created or opened. It is the starting point for analysis of the data and it is from this window that the various tools available in Flatirons are selected.

In this chapter each tab window (not menu items) is described in order. Most of the features available in each tab are covered, but only at the highest level. Other chapters will explore the features in depth. The purpose of this chapter is to serve as a reference, not as a logical flow in the analysis of the data.

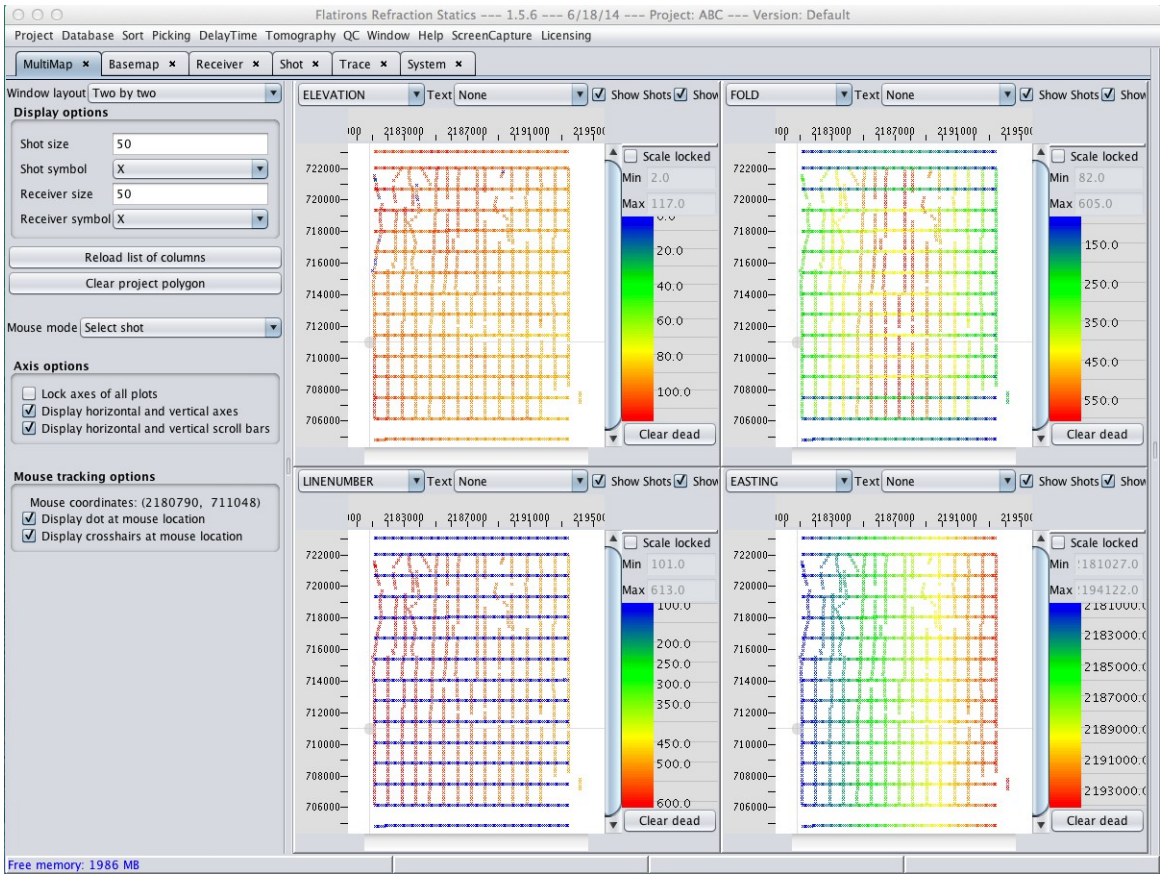


Figure 14---1 Main project window. When opening a new project, the default tab window in Flatirons is the Multimap tab with four configurable Basemap displays.

The main project window is roughly comprised of a menu bar, tabs, display areas and display options. The display options generally are on the left hand side of the window and change depending on the selected tab. There are currently six tabs opened by default. At any time an individual tab can be closed. To re---open the MultiMap, Basemap, Receiver, Shot or Trace tabs, go to the Database menu and select the appropriate item. Note that the re---opened tab will be added to the right of all currently opened tabs. Thus the order of the opened tabs may change in the course of data analysis.

13.1 Multimap

The Multimap tab has, by default, four independently configurable display panels. It is also the active tab when Flatirons opens, as shown in **Error! Reference source not found.**. Each panel shows a basemap with elements indicated by color or text (Figure 14---2). In the example below, ELEVATION has been selected in the drop---down for Color, while FOLD was selected in the drop---down for Text. Markers for both shots and receivers have been selected. More than 20 elements are available for the Color/Text indicators, although some depend on the task being applied to the data.

To zoom in on the map, right---click---and---drag to form a greyed rectangle. Releasing the right mouse button sets the zoom area. This process can be repeated to continue zooming in. To return to the normal view, apply a single right click with the mouse arrow over the display to be un---zoomed. Each display area can be independently zoomed. The number of panels displayed can be set by the **Window layout** option on the left edge of the main window. A drop---down allows the used to select from a single panel up to nine panels in a 3x3 arrangement.

The color bar on the right hand side of the display panel determines settings for the Color option. The scale can be locked to a user---set minimum/maximum value for the chosen element. Additionally, right---clicking on the color bar opens a display box with various color schemes (not shown). Left click on the color scheme of choice to instantly change to the new scheme.

A left---click anywhere in the color bar opens a Color panel to allow the user to set the color for that single color bin. A left---click---hold---drag---release modifies the color scheme by interpolating the colors from the top to the bottom of the selected region. To reset any of these changes, mouse over the color bar, right---click and select Default colors.

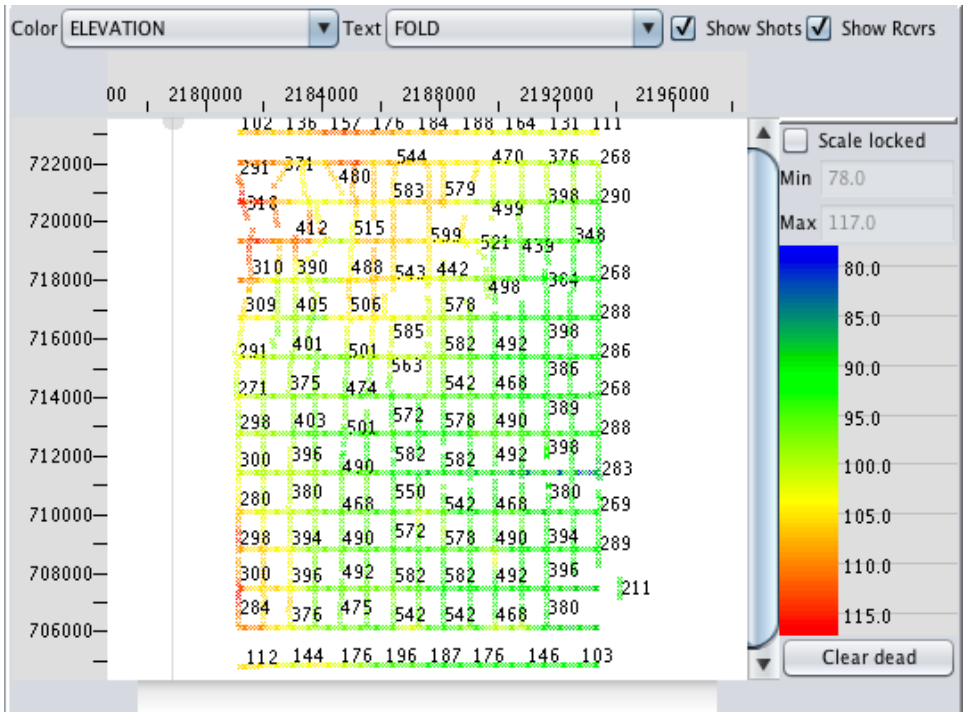


Figure 14---2 One panel in the MultiMap view displaying a basemap and associated color bar. Each item for Color and Text is independently selected. In this example Color indicates Elevation and Text indicates Fold. Both shots and receivers are displayed.

Finally, shots or receivers can be removed from the display using the Color option. By using left---click---drag---release on the scale to the right of the color bar, any shot/receiver having an element in that selected color region is removed from the display. For example, in Figure 14---3, the element selected for Color option is SHOTID. Two separate regions, indicated by the red rectangular areas to the right of the color bar, have been used to eliminate Shot IDs of ~1200---1300 (dark green) and ~1500---1800 (yellow). To restore these 'dead' shots to the display, simply click the **Clear dead** button. This functionality is typically present in all displays with color bars, whether vertical or horizontal.

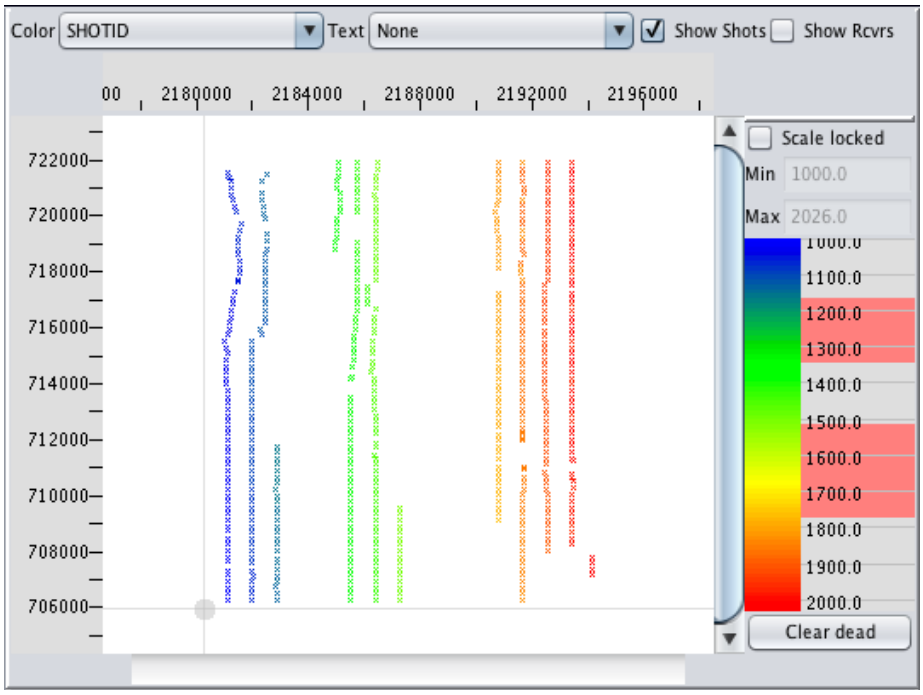


Figure 14---3 Using the color bar to omit selected regions; in this example two different ranges of ShotID are removed from the basemap.

Display Options

Unlike the Color/Text drop---down options in each panel, the display options apply to all panels simultaneously. The **Display Options** comprise the **Window layout** mode, **Display Options**, **Mouse Mode**, **Axis options**, and **Mouse tracking options** (Figure 14---4).

- Window layout – allows for 1---9 displays.
- Display options – choice of shot/receiver size and symbol.
- Reload list of columns – only applies if user added a column in the database when creating the project. This button incorporates the added column(s) to the Color/Text drop---downs.
- Clear project polygon – clears the user---defined polygon (if any – see Mouse mode).
- Mouse mode
    - Define project polygon – user defines a polygon by left---clicking points on the display, closing the polygon with a double---click. The purpose of the project polygon is to target a specific sub---set of the data for subsequent analysis.
    - Select shot – The user left---clicks in the display and the nearest shot is selected. The selected shot is not indicated with a marker in the Multimap tab but is it visible in the Basemap tab.
    - Select receiver – Same as Select shot but the nearest receiver is selected.
- Axis options

- Lock axes of all plots – with this option unchecked, the zoom operation can be performed independently on all displays. Otherwise, a zoom operation applied in one display automatically is applied of all displays.
- Display horizontal and vertical axis – toggle whether axis information is shown.
- Display horizontal and vertical scroll bars – toggle whether to show scroll bars. Typically used to scan across a survey when in zoom mode.

- Mouse tracking options
  - Mouse coordinates – map coordinates of the current mouse position is continuously updated here.
  - Display dot a mouse location – toggle the display of a small gray circle at the current mouse position.
  - Display crosshairs at mouse location – toggle the display of light gray horizontal and vertical lines, crossing at the current mouse position.

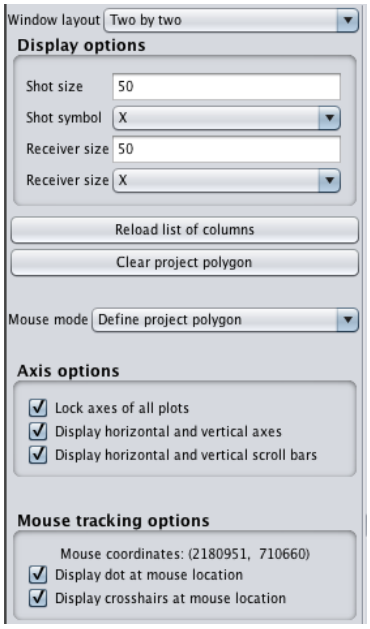


Figure 14---4 The left panel of the MultiMap tab allows for a wide range of display options.

13.2 Basemap

The basemap tab is quite similar to the Multimap tab but with increased functionality for analysis related to the basemap view. The window is comprised of three panels, with the basemap displayed in the center panel, display options in the left panel and in the right panel, various methods to allow interaction with the basemap and extract information (Figure 14---5). As in most windows in Flatirons, the panel widths can be adjusted using a left---click---and---drag anywhere along the vertical boundary of the two panels to be re---sized. In this view, the color bar is horizontal but the same standard filtering effects are available.

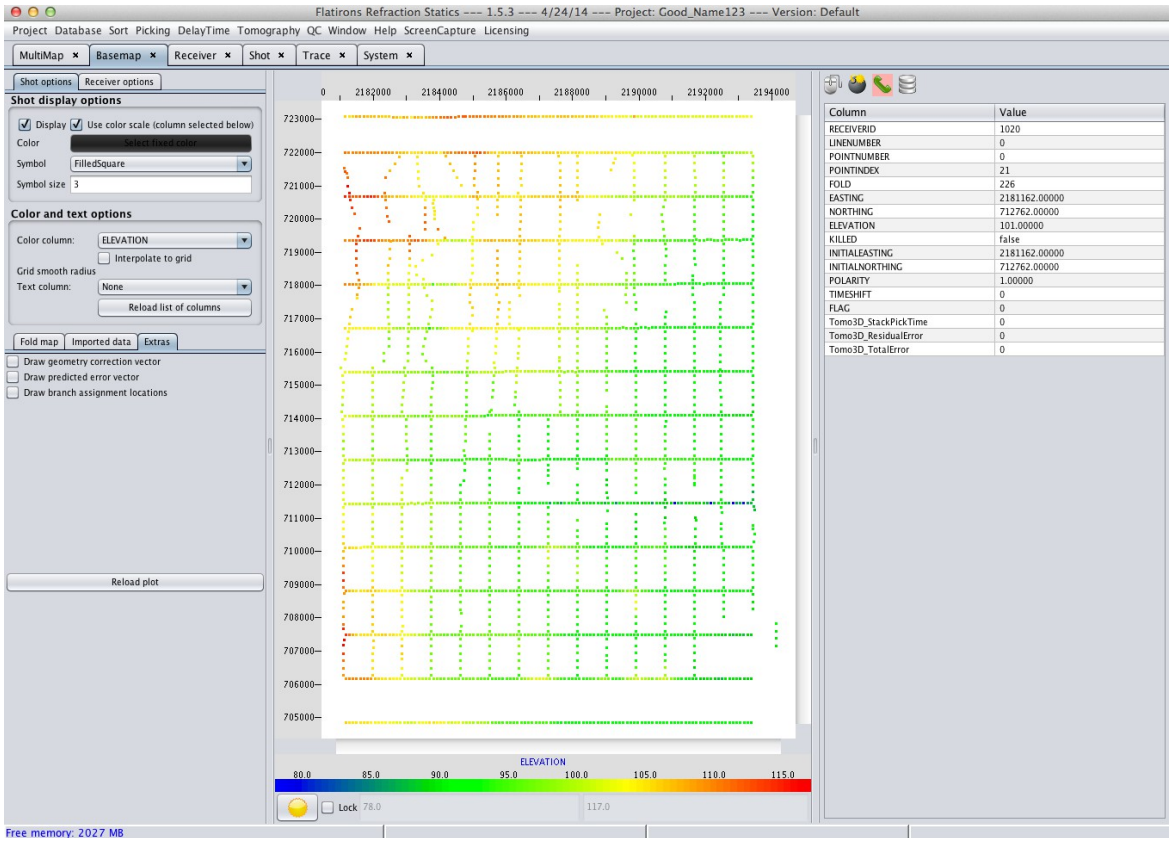


Figure 14---5 The default view of the Basemap tab with three panels. The widths of the panels can be adjusted using vertical sliders.

The display options panel is similar to that of the Multimap tab, with a few differences. In this case the display options panels has two tabs, one for Shot options and one for Receiver options as shown in Figure 14---6 Display options for Basemap tab. The shot and receiver options are identical. The options for shot and receiver are identical. One difference from the Multimap tab options is the **Interpolate to grid** checkbox. After this option is checked the **Grid smooth radius** option becomes active. When this checkbox is selected the Color column element is interpolated across the entire grid. For example, with ELEVATION chosen, an elevation map of the entire survey is created by interpolating the Shot and Receiver elevations. In order that the Shot and Receiver locations are still visible, they are set to black. The scale of the interpolation is a user---selected number of bins in the **Grid smooth radius** drop---down. Of course, zooming operations are still available.

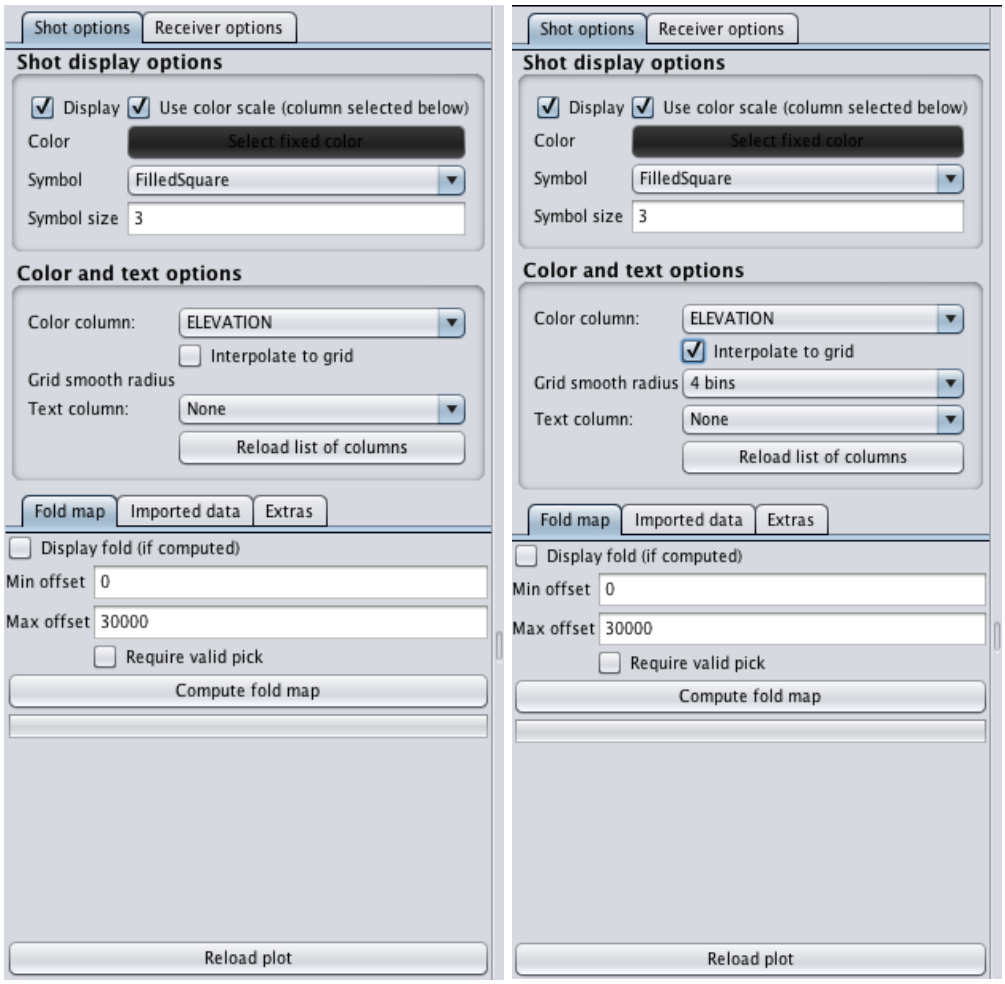


Figure 14---6 Display options for Basemap tab. The shot and receiver options are identical. On the left is the default view, on the right is with Interpolate to grid checked. Note that the Grid smooth radius option becomes active.

The bottom section of the display options panels has three additional tabs --- **Fold map**, **Imported data** and **Extras**, each with its own options. Finally, at the bottom of the display options panel is a **Reload plot** button. Often, when changes are made to the basemap involving altered data, the map has to be re---displayed manually, to signal to Flatirons that the alterations are complete.

The right panel in the Basemap tab has four alternative display options, indicated by icons instead of tabs. They are **Mouse mode** (), **Shot table** (), **Receiver table** () and **Imported data table** (). These four options relate, generally, to dynamic display of information related to the current basemap using display options in the left panel.

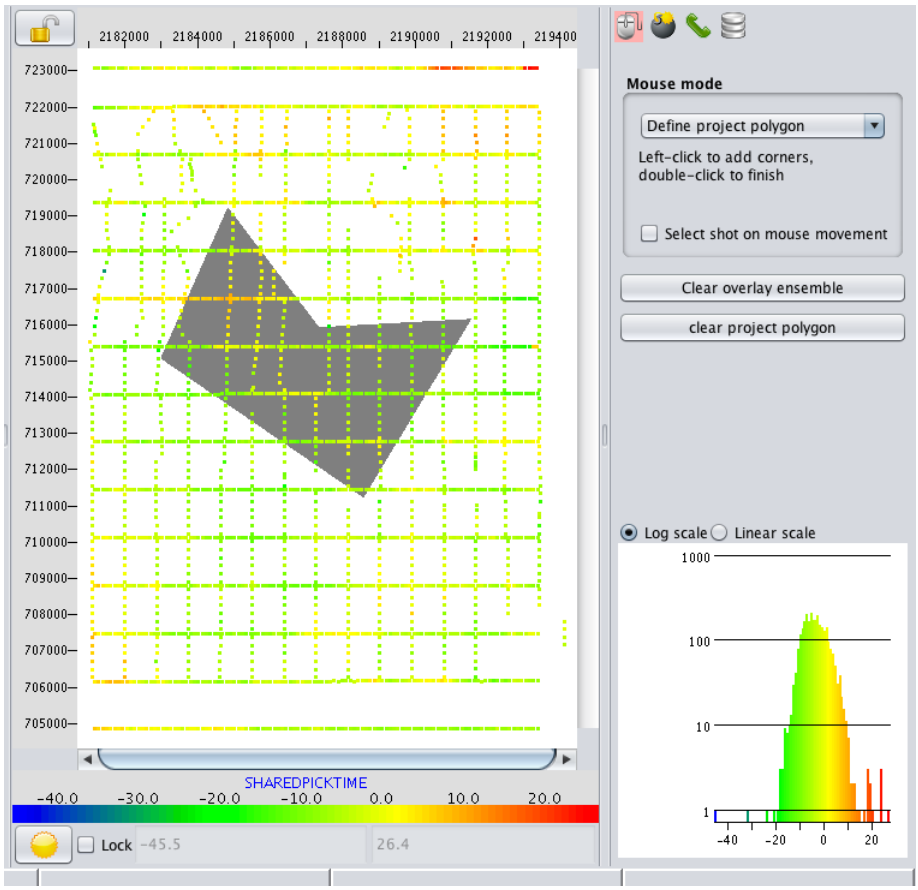


Figure 14---7 Defining a project polygon in Mouse mode in the Basemap tab.

The **Mouse mode** option allows for selection of shot, receiver, midpoint or a user---defined polygon. Figure 14---7 shows an example of defining a polygon. Simply left---click at a point on the basemap, repeat at each desired polygon corner and double---click to finish. If the resulting polygon is not satisfactory, simply click the **clear project polygon** button and try again.

If the mouse mode is **Select Shot** then a left mouse click will cause Flatirons to locate the shot closest to the current position of the mouse and then highlight every receiver associated with that shot by a red marker. Similarly, if **Select Receiver** is chosen, every shot associated with that recevier is highlighted in blue. Finally, if mouse mode is **Select midpoint** then every shot and receiver associated with the midpoint near the mouse---click is highlighted in blue and red, respectively. Optionally, by holding down the Control key first, the selected shot or receiver is updated in all open windows. Note that when grid interpolation is on, the basemap display is not updated as shown in Figure 14---8 but all other open windows are still updated.

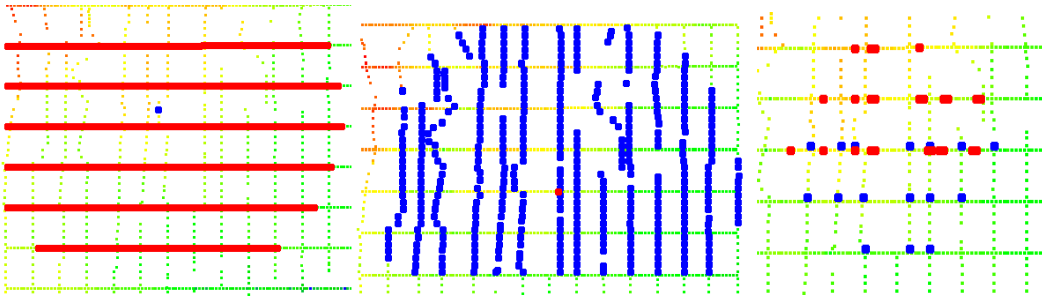


Figure 14---8 Mouse mode selections for shot, receiver and midpoint, respectively. On the left, the single shot is the blue dot and associated receivers are in red. In the middle, the single receiver is the red dot and the associated shots are in blue. On the right are the shots (blue) and receivers (red) associated with the selected midpoint (not shown).

There is also a checkbox for **Select on mouse movement**. When this option is selected the first display, select shot, the left---most image in Figure 14---8, is animated. That is, the currently selected shot is determined not by a mouse click but by the current mouse position. However, a left mouse click will momentarily override the animation and display the Mouse mode selection. In this way, it is easy to rapidly alternate between shot/receiver mode and shot/midpoint mode.

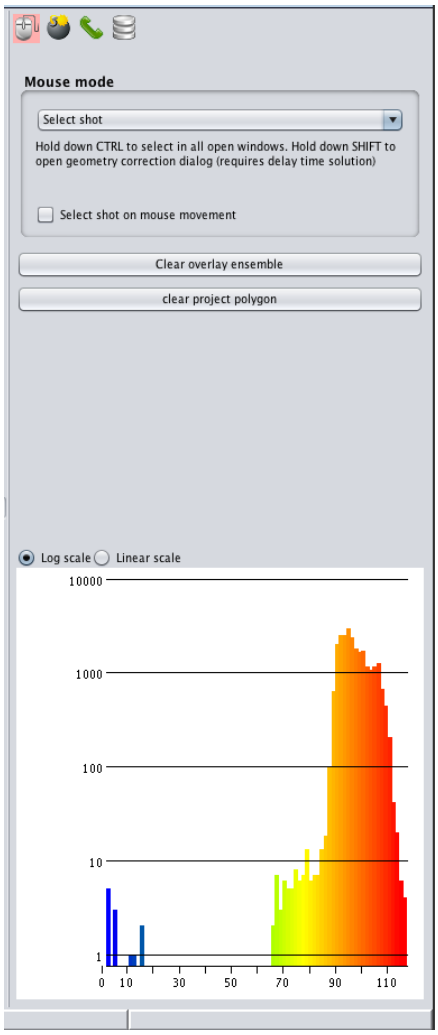


Figure 14---9 Mouse mode panel in the Basemap tab. The bottom figure is a histogram of the Column displayed on the basemap.

Two additional buttons permit clearing the highlighted selections (**Clear overlay ensemble**) or any user---defined polygon (**clear project polygon**). Finally, there is a histogram plot of the same Column being displayed in the basemap, with the same color scale and axis as the basemap color bar (Figure 14---9). Especially when set to **Log scale**, the default, extreme values that are difficult to see on the basemap are readily apparent. If the basemap is zoomed, the histogram updates to include only the zoomed region.

The **Shot table** option shows the Columns from the shot data table in tabular form. As the mouse indicator is moved over the basemap, the displayed table is continually updated with the data associated with the shot nearest the mouse indicator. Similarly, the **Receiver table** option shows realtime data from the receiver data table. Finally, the **Imported data table** option would show similar information from any separate, imported data table. For example, an independent uphole survey may be imported and viewed by scanning over the basemap with this feature.

13.3 Shot/Receiver

The **Receiver** tab permits viewing of the Receiver data table as well as showing other receiver---related information. The **Shot** tab has the identical format and function, for shots. The **Receiver** tab has four tabs itself: **Table**, **Import Conflicts**, **Crossplots** and **Duplicate Entries**.

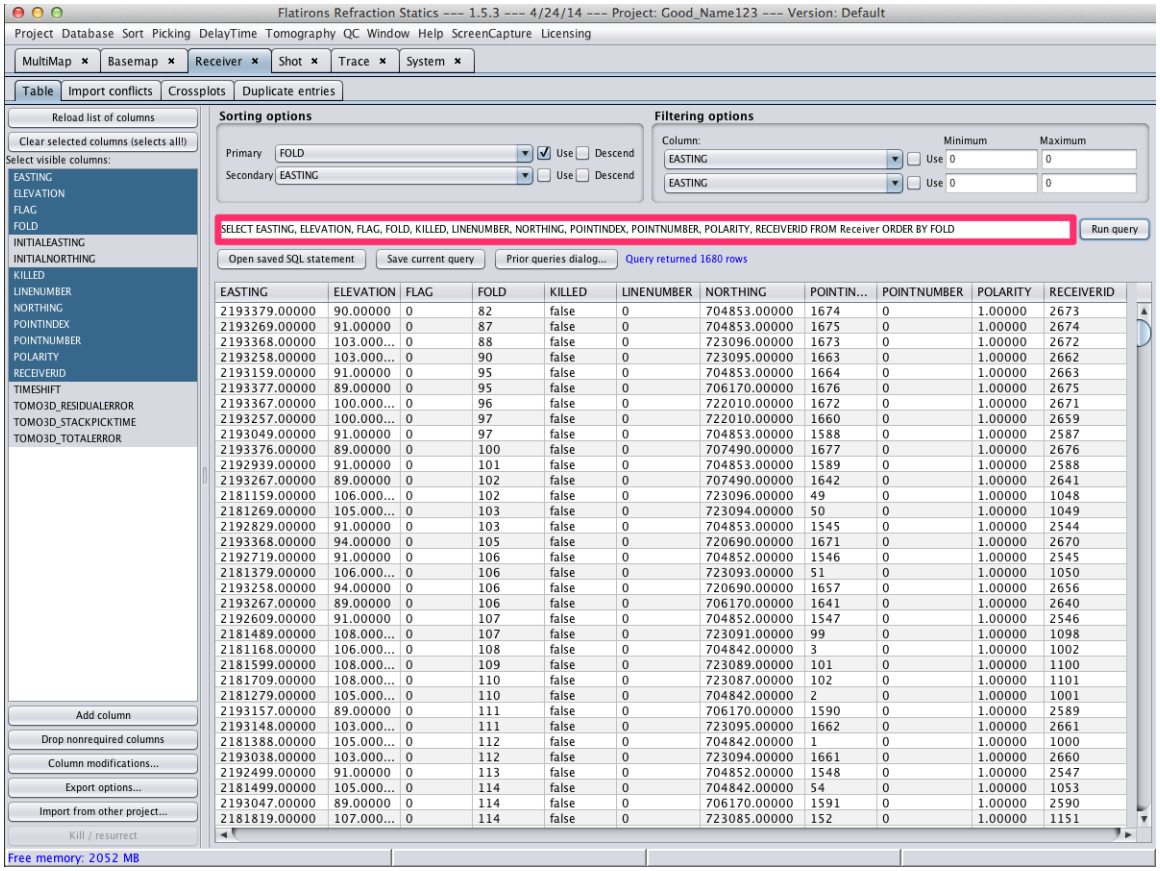


Figure 14---10 Table tab displaying a spreadsheet of selected fields from the Receiver table. Retrieved data is based on an SQL statement, outlined in red.

Table. The Table tab displays a spreadsheet containing the result of an SQL operation on the Receiver table (Figure 14---10). The left panel lists all the columns in the Receiver table. The columns to be displayed are selected by repeatedly clicking on the desired column, using Shift and/or Control/Command (Mac) keys to select multiple columns. The **Sorting options** section is used to sort the spreadsheet by primary and secondary keys (columns). The **Filtering options** section is used to filter the data (i.e., eliminate rows) using a specified minimum/maximum range on columns. Note that when entering minimum/maximum values into the box, the Return/Enter key has to be pressed for the changes to take effect.

As the columns are selected, the sorting options chosen and the filtering options applied, the SQL statement is generated and displayed. When all options have been completed, clicking on the **Run query** button will execute the SQL statement as a Query on the database. The

number of rows satisfied by the Query is displayed in blue. If no columns are selected and no sorting or filtering options applied, the default SQL statement is simply

SELECT * FROM Receiver

which means read in and display the entire Receiver table. Initially, before any processing has been done, the available Columns are just from the raw data. However, as delay time or tomographic solutions are computed, for example, additional Columns are added.

NOTE: When creating a project it is highly recommended a Query be run on both the Shot and Recevier tables to ensure that the data was imported correctly and the number of shots and receivers is correct.

Three buttons beneath the SQL box allow for operations on the Query. **Open saved SQL statement** will open a dialog to permit the user to read in and execute as previously saved query, if any. The **Save current query** button automatically save the current query, with a pop---up box notifying the user that the query was saved. Note that Flatirons does not check that each saved query is unique. The **Prior queries dialog…** button opens a dialog box listing of queries, along with the date previously executed and the number of rows satisfying the query. From here an existing query can be selected and immediately executed.

In the spreadsheet, the data can be sorted by any Column, in ascending or descending order simply by clicking on the Heading of the desired Column. Also, any cell in the spreadsheet can be edited except for the RECEVIERID/SHOTID.

The left panel has a few more options related to the data tables.

- **Add column**. From a pop---up dialog box a user can add a new Column to the Shot and/or Receiver table, by specifying a (unique) Column name and Data type.
- Drop non---required columns. From a pop---up dialog box a user can remove a selected Column from the Shot and/or Receiver table, by selecting one of the existing Columns.
- **Column modifications…** From a pop---up box, an assortment of options are available that perform some type of modification on Columns (Figure 14---11). Some of the options require prerequisites. For example, to modify data within a polygon, the user must have pre---defined a polygon – remember, this was an option in **Mouse Mode** under the **Basemap** tab. As another example, to use data from an imported table, such a table must exist.

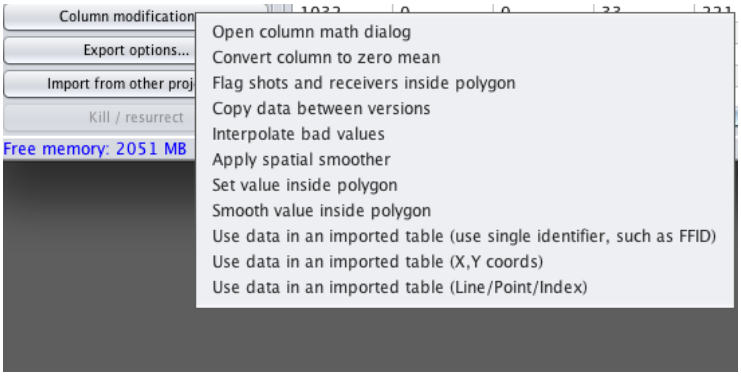


Figure 14---11 Options for the Column modifications item.

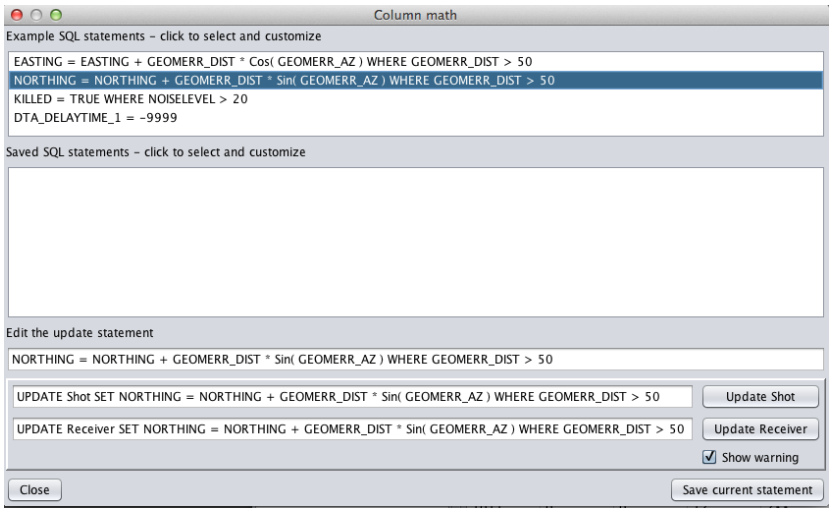


Figure 14---12 Column math dialog box, from the Column modification... button.

The **Open column math dialog** option opens the Column math dialog box, shown in Figure 14---12. From here, SQL statements, previously saved or built from scratch, can be applied to the data. In the example shown, a saved SQL statement has been selected and automatically applied to both Shot and Receiver tables. The SQL statement has not been edited, as can be seen by comparing the saved statement with the one in the **Edit the update statement** box. If the SQL had been edited it could be saved via the **Save current statement** button. To actually perform the update on the Shot and/or Receiver tables, the **Update Shot** and/or **Update Receiver** buttons must be selected, respectively.

- **Export options...** Allows the user to export selected data, based on the result of an SQL query, or the entire table in various formats.
- **Import from other project...** Allows the user to import data from another project. Figure 14---13 shows the dialog box used to import the data.

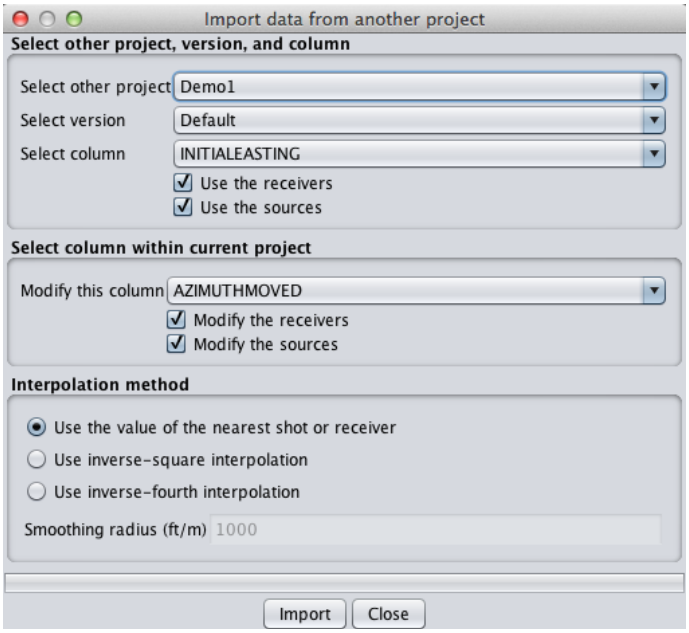


Figure 14---13 Dialog box for importing data from another project.

- Kill / resurrect

Import conflicts. If there were any suspected geometry errors when the project was created, they would show up in this window.

Crossplots. Any two Columns from the Receiver table (or Shot table, if the Shot tab has been selected), can be plotted together, on the the X (Horizontal) axis, the other on the Y (Vertical) axis. Each Receiver is then plotted by its selected (X, Y) values and colored by another user---selected Column. As usual, a color bar allows for filtering based on this third dimension. Additionally, the data can be limited by selecting a minimum/maximum range for yet another Column. When selecting a new Column or data limitation, be sure to click the **Reload plot** button for the changes to take effect. In Figure 14---14 a crossplot of receivers shows FOLD vs EASTING, colored by ELEVATION. No limitations of the data have been made, either by the color bar or by setting a min/max on a Column. Crossplots can be repeatedly zoomed and unzoomed.

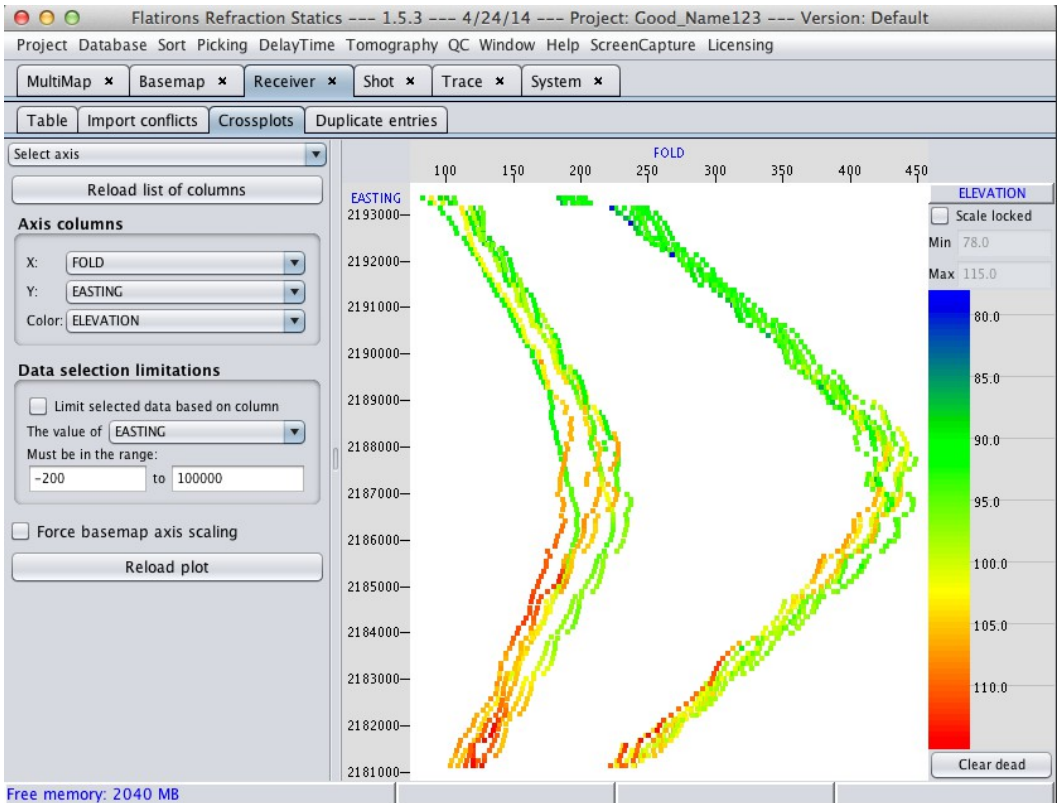


Figure 14---14 Example crossplot showing FOLD vs EASTING, colored by ELEVATION with no limitations on the data. The axes for all three dimensions are labeled in blue.

13.4 Trace

The Trace table is displayed in spreadsheet form. Several operations on trace data are available through buttons above the table as shown in Figure 14---14.

Refresh. Refresh the spreadsheet after operations on the trace table.

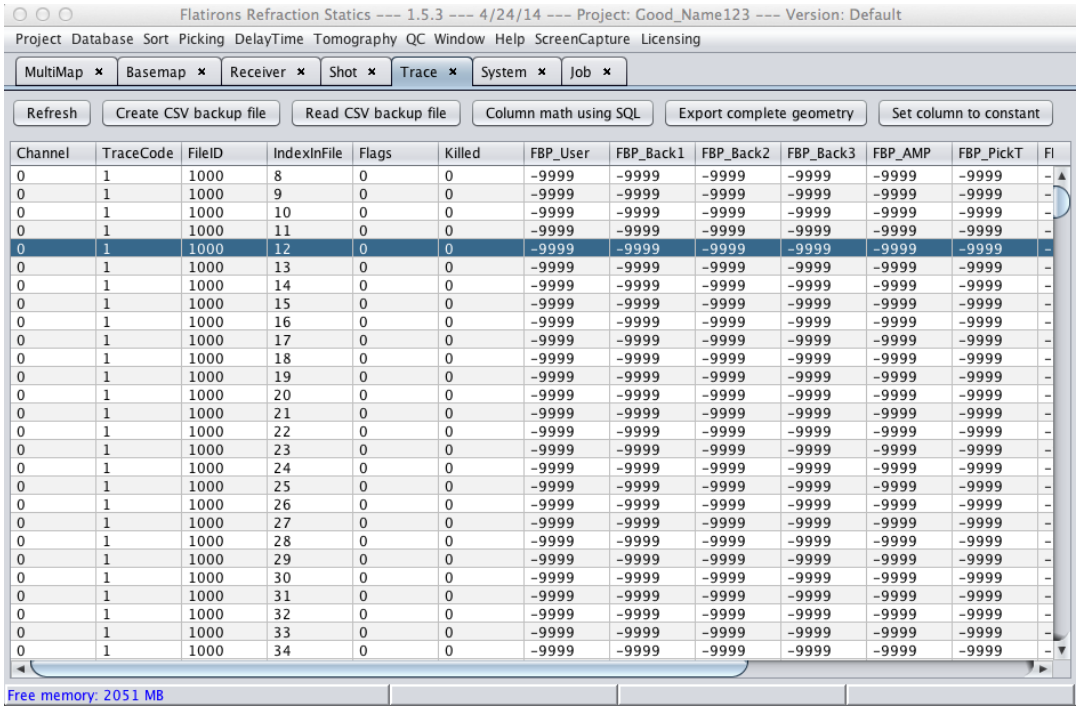
Create CSV backup file. Clicking on this button will cause Flatirons to temporarily jump to the **Job** tab and display the progress of the creation of the backup CSV file.

Read CSV backup file. Clicking on this button will cause Flatirons to temporarily jump to the **Job** tab and display the progress of reading the backup CSV file.

Column math using SQL. Opens a window, similar to that shown in **Error! Reference source not found.**, for performing trace table column math. Saved SQL statements can be executed, with or without editing, or new SQL statements can be created, executed and saved.

Export complete geometry. Opens a window in which shot columns, receiver columns and trace columns can be independently selected and output to an ASCII file in CSV format. Figure 14---15 shows an export in progress.

Set column to constant. Opens a dialog box in which a new value can be assigned to a single selected column. Every row in the column is set to this value.



Channel	TraceCode	FileID	IndexInFile	Flags	Killed	FBP_User	FBP_Back1	FBP_Back2	FBP_Back3	FBP_AMP	FBP_PickT	Fl
0	1	1000	8	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	9	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	10	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	11	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	12	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	13	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	14	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	15	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	16	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	17	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	18	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	19	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	20	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	21	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	22	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	23	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	24	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	25	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	26	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	27	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	28	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	29	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	30	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	31	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	32	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	33	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-
0	1	1000	34	0	0	-9999	-9999	-9999	-9999	-9999	-9999	-

Figure 14---14 Trace table. Buttons open dialogs to perform operations on the trace data.

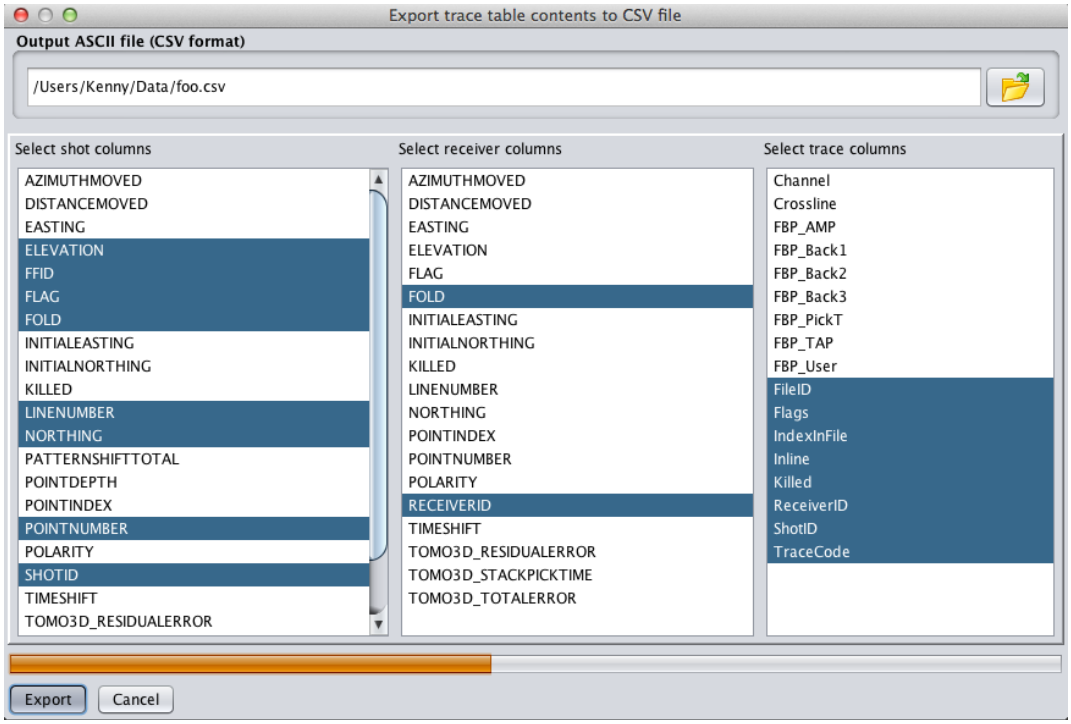


Figure 14---15 Exporting data to a CSV file. Several shot, receiver and trace columns have been independently selected.

13.5 System

The System tab has four tabs that provide information about the Flatirons application and Java properties.

Application exceptions. This window displays a list of errors, with description and date the exception occurred. The list of errors can be saved to a file.

Java system properties. This read---only window displays Java property name and values. Examples include the runtime version, classpath, encoding, timezone.

Environment variables. This read---only window displays defined Environment variables by name and value. Examples include HOME, PATH, USER, SHELL.

Change log. This read---only window displays the version history of Flatirons along with a short list of changes and bug fixes for that version.

13.6 Job

This tab displays the progress of an ongoing job, such as file I/O. Some operations that create a job will automatically jump to this tab, temporarily displaying the progress of the current job, before returning to the prior state.

This concludes the overview of the main window in Flatirons. This chapter omitted any discussion of the menu items.

14 First Break Picking Using PickX on Flatirons Projects

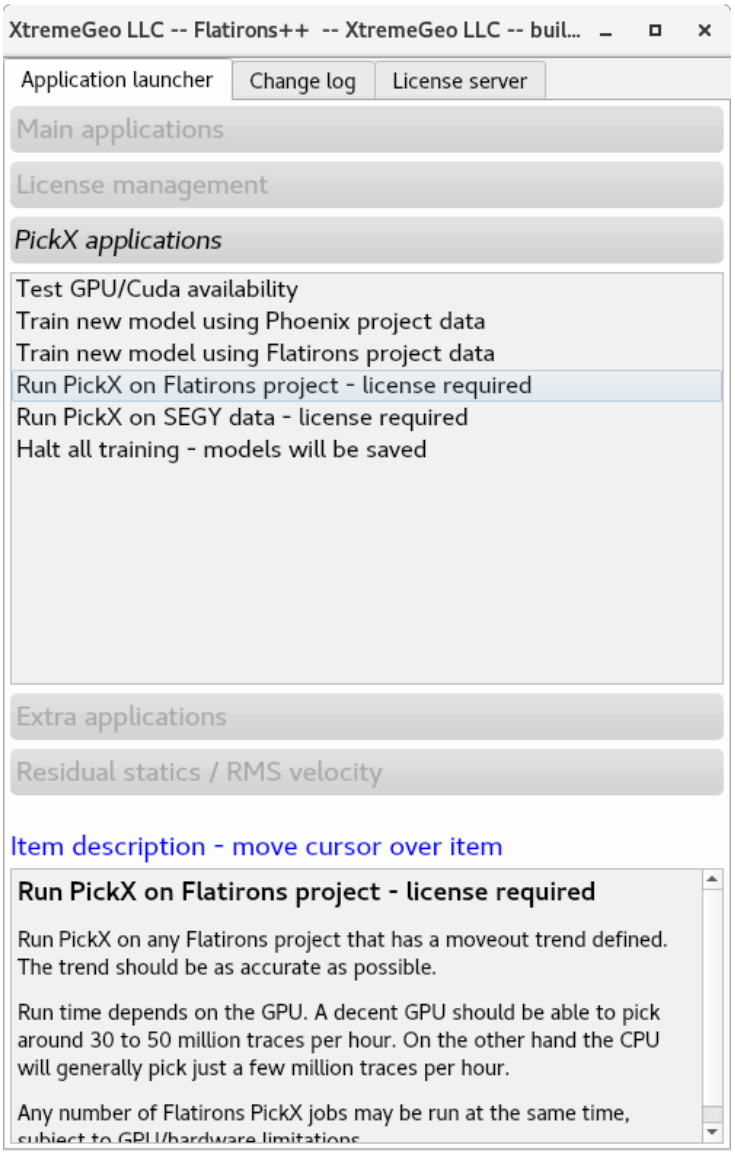
A valid PickX license is required to perform this operation.

The Flatirons project must meet these two requirements:

- Shot ordered data
- A decent moveout trend must be defined

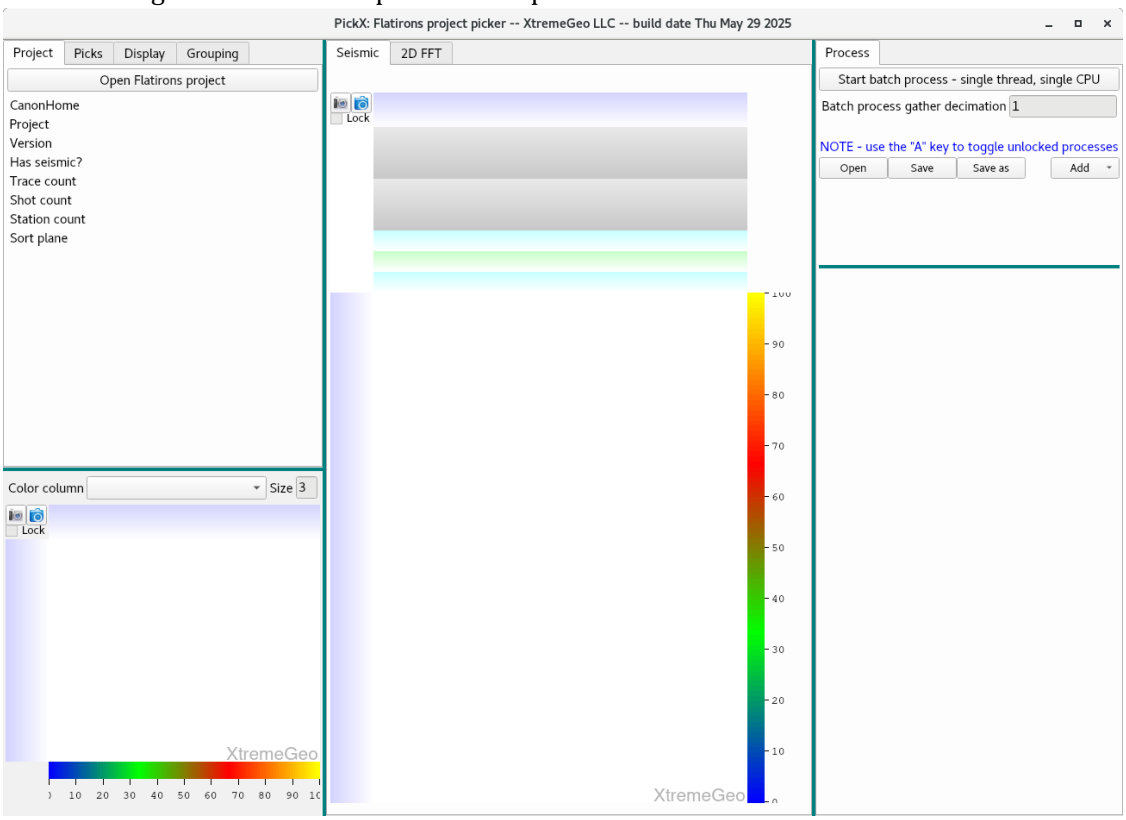
Launch the Picker

Start the Flatirons++ application launcher and select “Run PickX on Flatirons project - license required”.



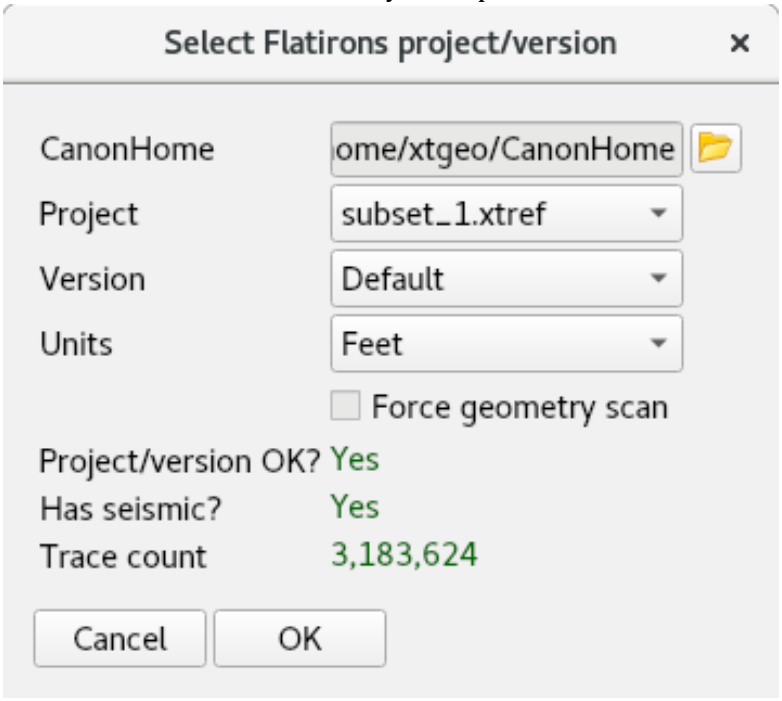
Flatirons++

The following window will be opened in a separate executable:

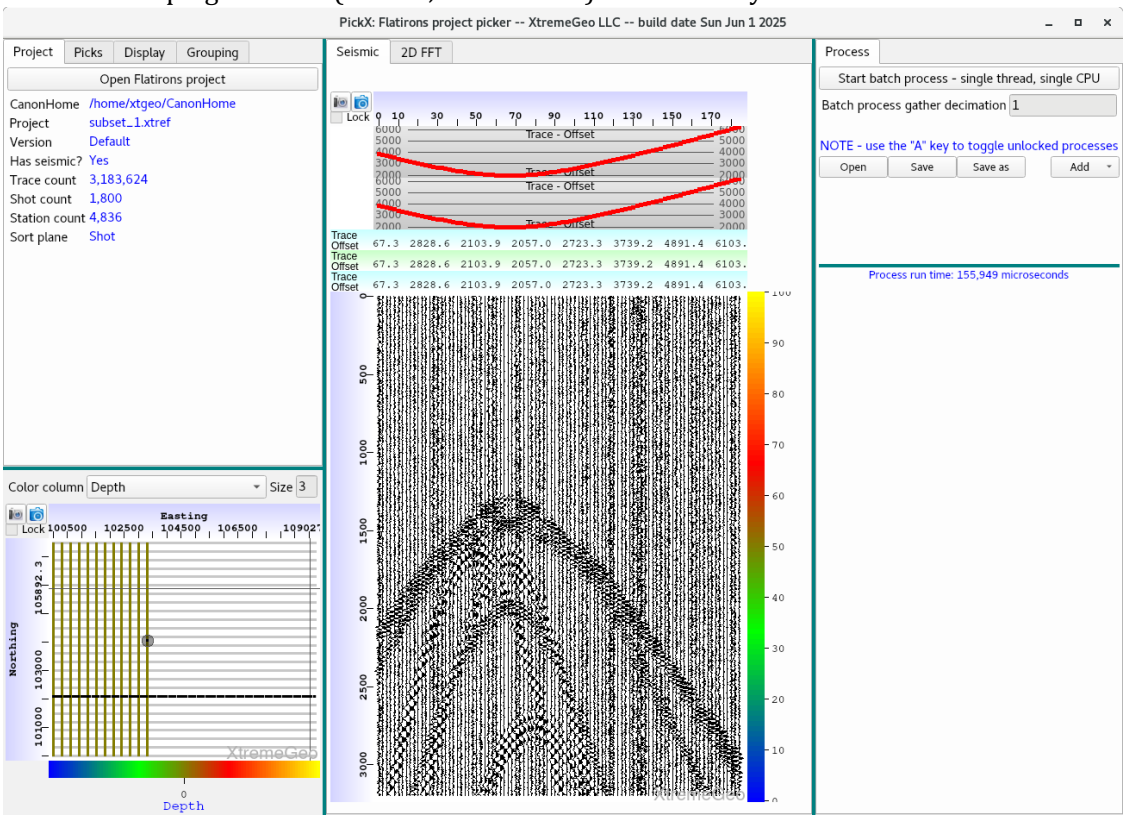


Flatirons++

Click “Open Flatirons project” then select the CanonHome directory, the project and version.
NOTE - the CanonHome directory is the parent of the RefactionProjects directory!

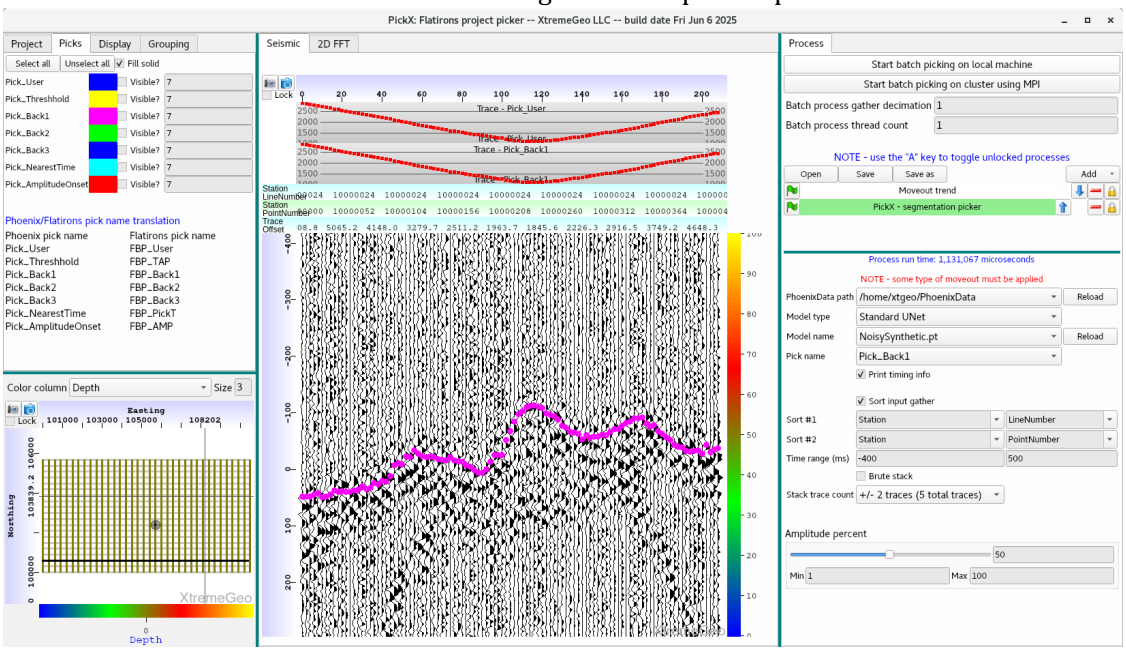


The program will scan the Flatirons project data to generate shot lookup information. This step may take a few minutes for large surveys, but will only be done the first time to project is opened.
Once the scan is completed a shot will be selected and displayed.
Be sure “Grouping” is set to (Station,LineNumber) for 3D surveys.



Flatirons++

Next add the “Moveout trend” and “PickX - segmentation picker” processes to the flow builder on the right.



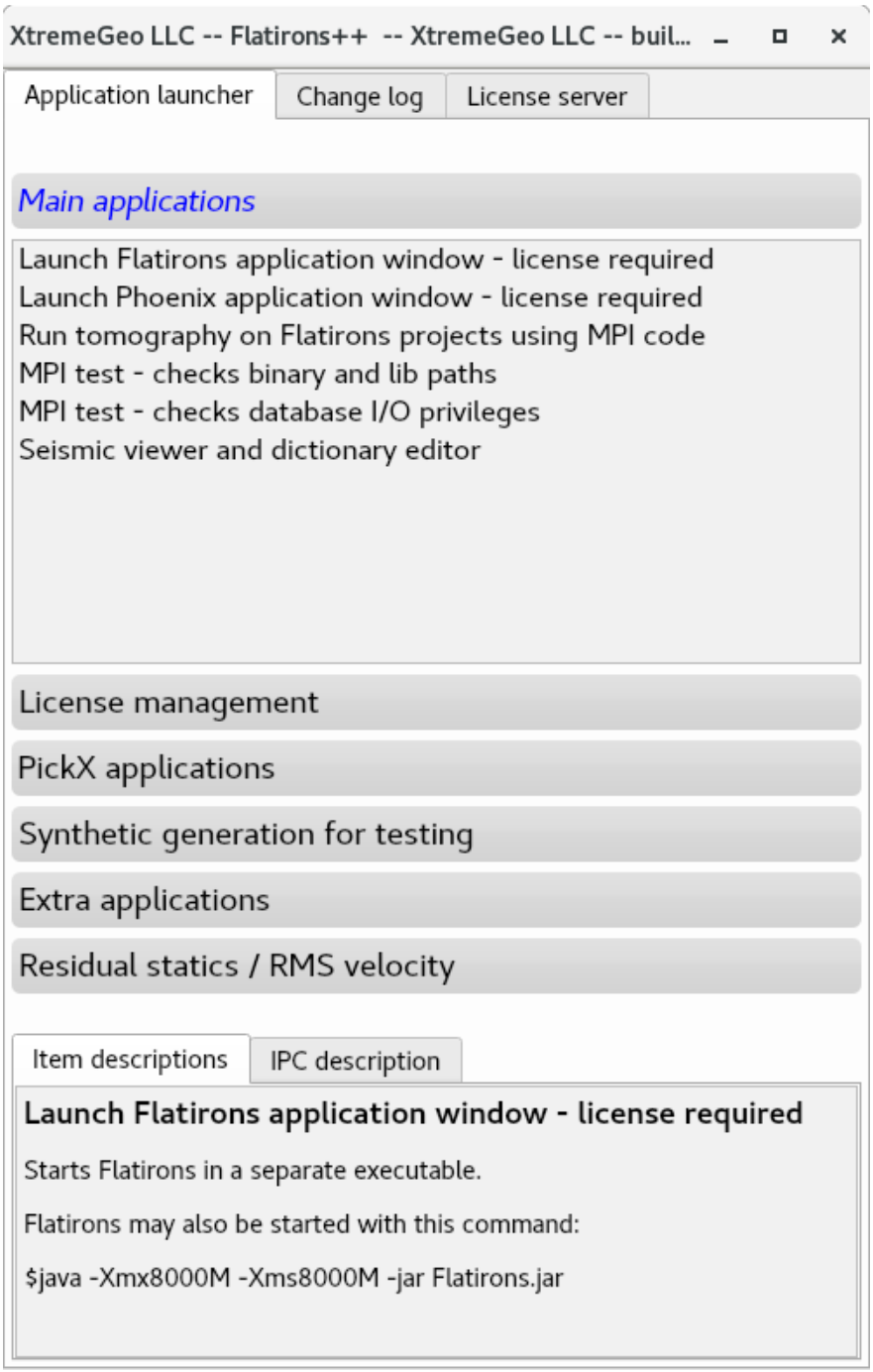
Flatirons++

You will eventually have more than one trained model, so be sure to select the correct “Model name”.
The only other important option is “Pick name”. See the Flatirons/Phoenix name translator under the “Picks” tab in the upper left.
FInally start the batch picker. Running on the local machine can be quite slow if you’re running the CPU version of PickX, so MPI should be selected if a cluster is available.
Once the batch job has been started another Flatirons project may be selected and picked.

15 Merging Flatirons projects in Phoenix

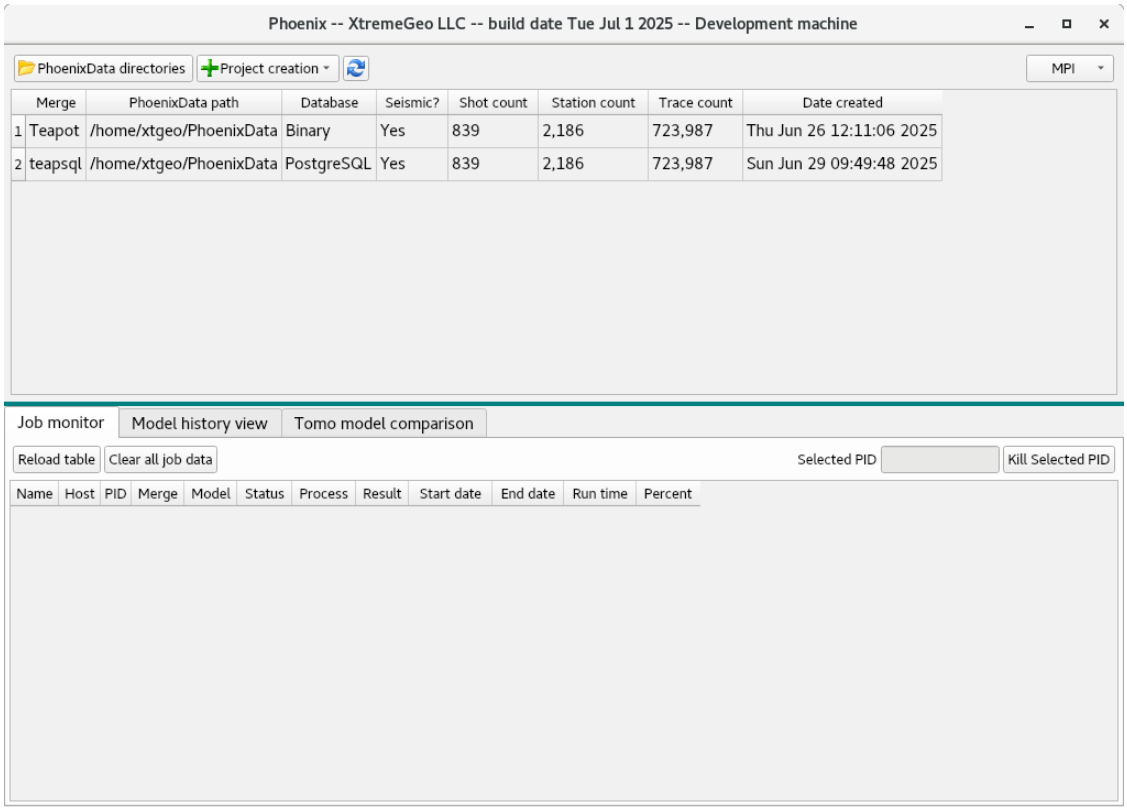
When creating a merge all Flatirons project data is left “in-place” - it is NOT converted to internal Phoenix format. The merging process simply creates new shot and station tables. This means that creating merges in Phoenix is very fast.

In what follows LPI is an abbreviation for (LineNumber,PointNumber,PointIndex). Both Flatirons and Phoenix uses the SEG/SPS standard to identify shots and stations.
Open the Flatirons++ application launcher and select “Launch Phoenix application window”:



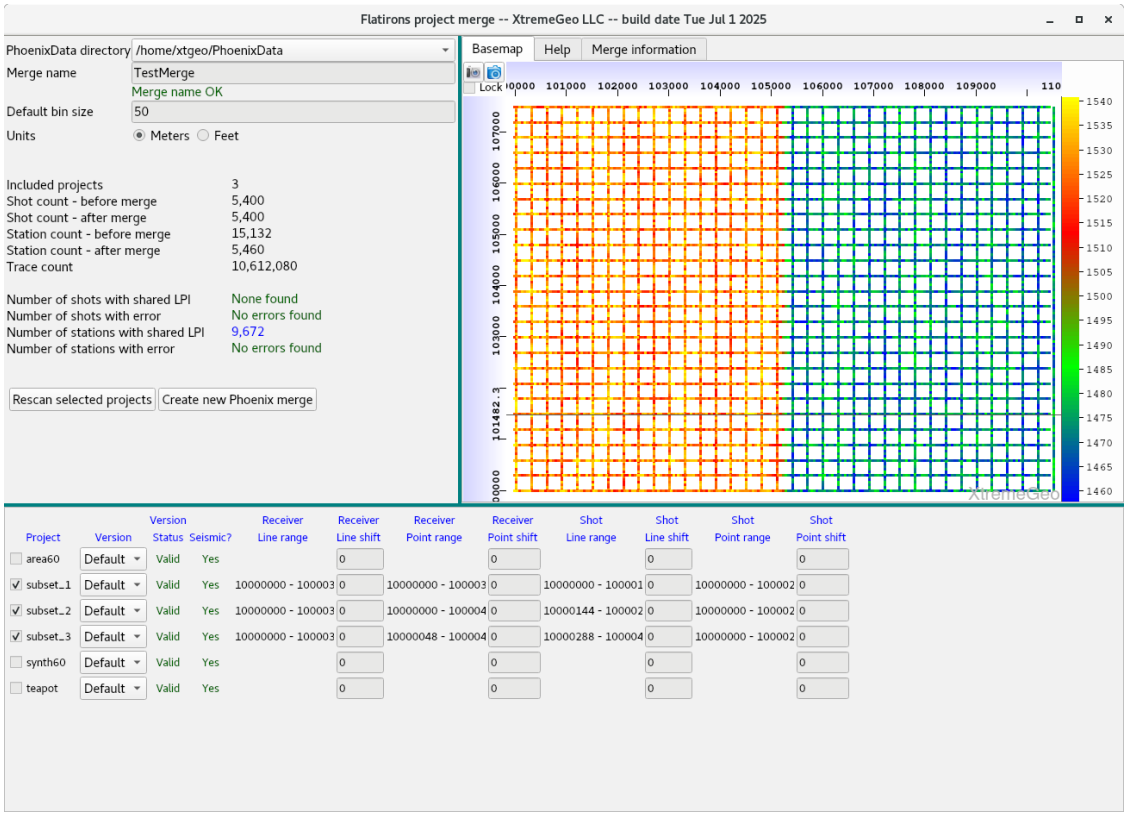
Flatirons++

This should start the main Phoenix display. Note that a license is required.



Phoenix

Click “Project creation” and then select “Merge Flatirons projects. This approach does not convert Flatirons into Phoenix data”.
A dialog will open to select the CanonHome directory - this directory is the **parent** of the RefractionProjects directory.



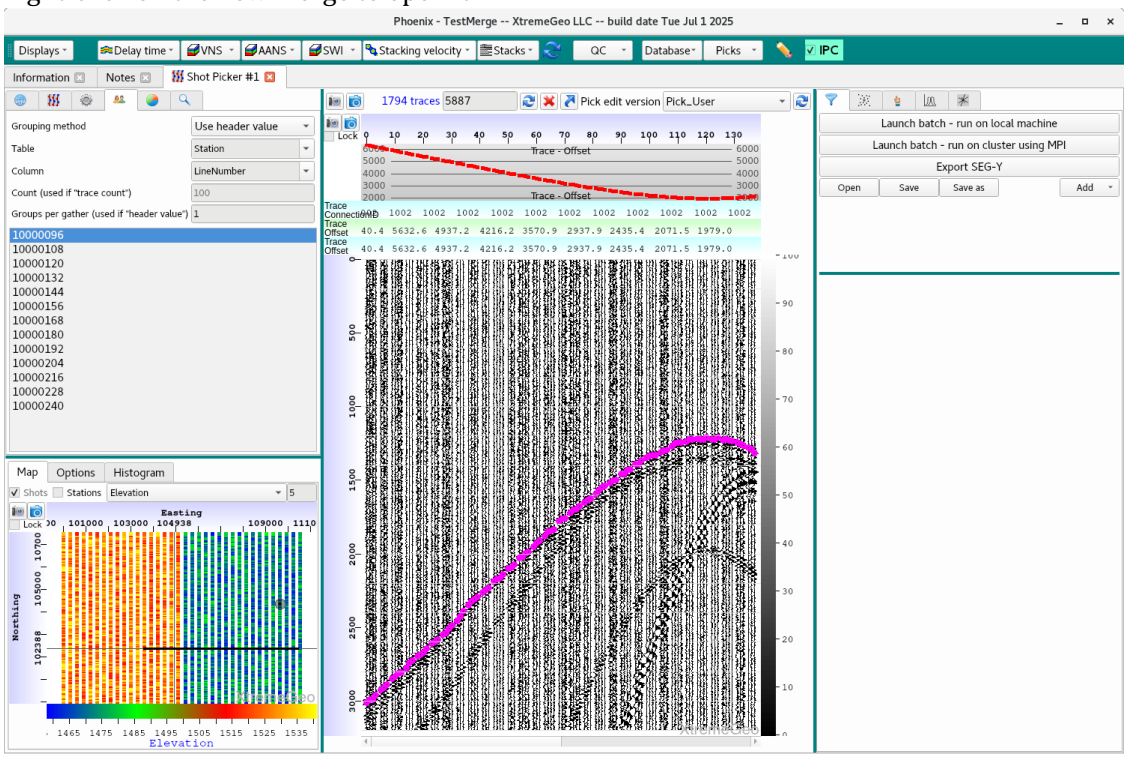
Merge

Select the output PhoenixData directory, give the new merge a name, select the units and enter a default bin size.
Then select the Flatirons projects to be merged in the lower part of the dialog. As projects are selected the basemap will be updated and the program will scan for geometry conflicts.
A conflict occurs when a station in two different projects has the same LPI but different coordinates. If this occurs, set the "Receiver Line Shift" or "Receiver point shift" for one of the projects. You may need to try a number of shifts before the conflict is resolved.
The upper-left displays information about the merged geometry.

- Shot count - before merge. This is just the sum of the shots in the underlying Flatirons projects
- Shot count - after merge. This is the number of shots with unique LPI, after applying the LineNumber and PointNumber shifts
- Station count - before merge. This is just the sum of the stations in the underlying Flatirons projects
- Station count - after merge. This is the number of station with unique LPI, after applying the LineNumber and PointNumber shifts
- Trace count. The total number of traces
- Number of shots with shared LPI
- Number of shots with error. This is the number of shots with shared LPI but different coordinates
- Number of stations with shared LPI
- Number of stations with error. This is the number of stations with shared LPI but different coordinates

When everything looks OK, click “Create new Phoenix Merge”. The application will compute raypath coverage next, which takes about 1 minute per 50 million traces.

When the merge is finished it should become visible in the Phoenix merge table. If it does not click on the “Reload table” button.
Right-click on the new merge to open it.



Phoenix

16 Training new PickX models

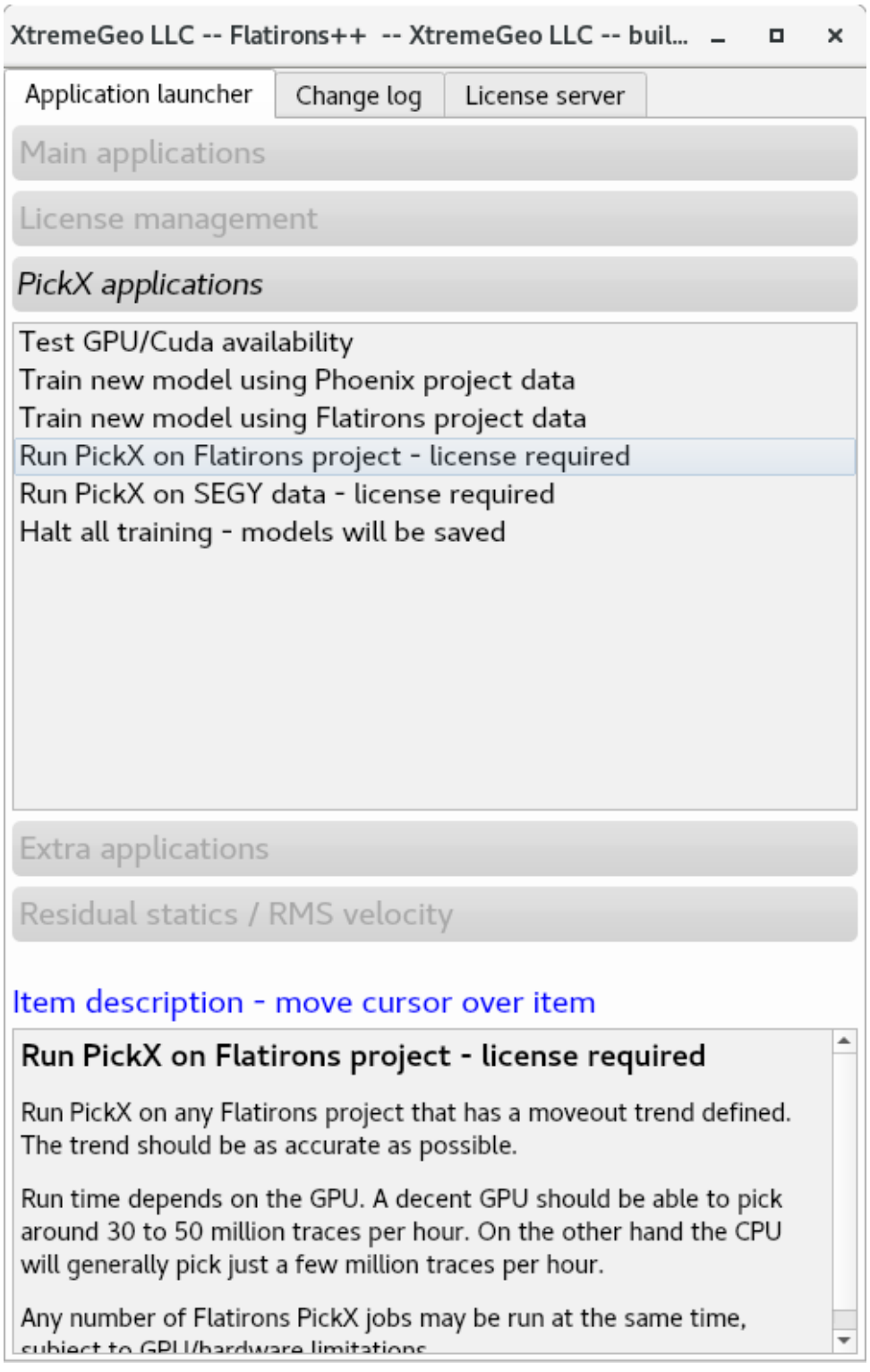
The current release of Flatirons++, May 27 2025, does not require a license to train new PickX models. This means that Flatirons++ may be downloaded and installed on any number of machines and training run on all of them, freeing up licensed machines for other work.

Training data, whether from Flatirons or Phoenix, must have the following:

- A good set of first break picks covering all offsets. The picks don’t have to be perfect.
- Moveout trends defined. Again, these don’t have to be perfect. A bit of “wobble” in the curves may actually be beneficial.

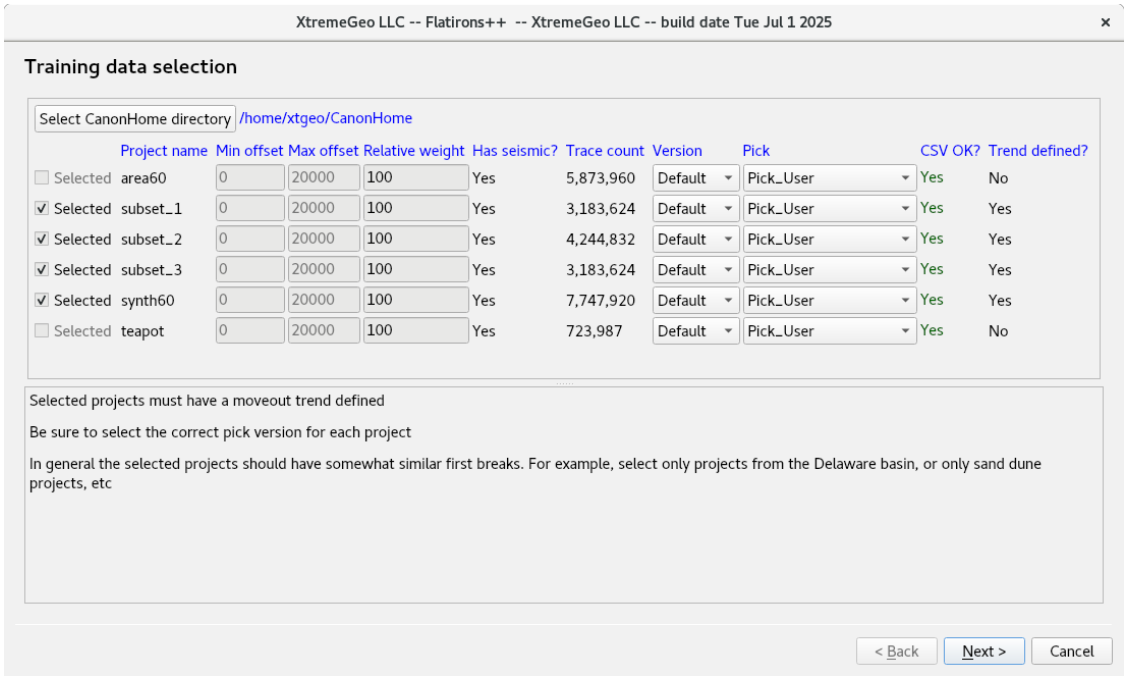
Select type of training data

Start the Flatirons++ application launcher and select “Train new model using Flatirons project data”.



Flatirons++

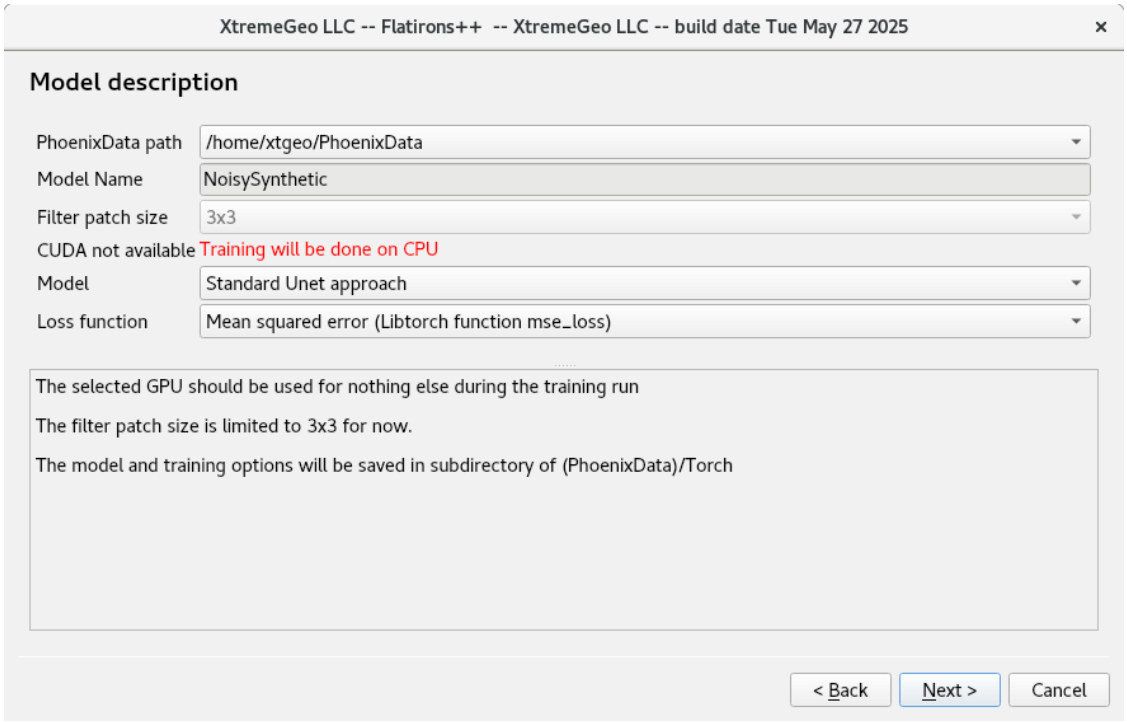
As a rule projects from the same general area should be selected, and if possible a mixture of projects with clean data and noisy data. For this example several synthetic projects have been selected. Note that the CanonHome directory is the **parent** of the RefractionProjects directory!



Flatirons++

Model description page

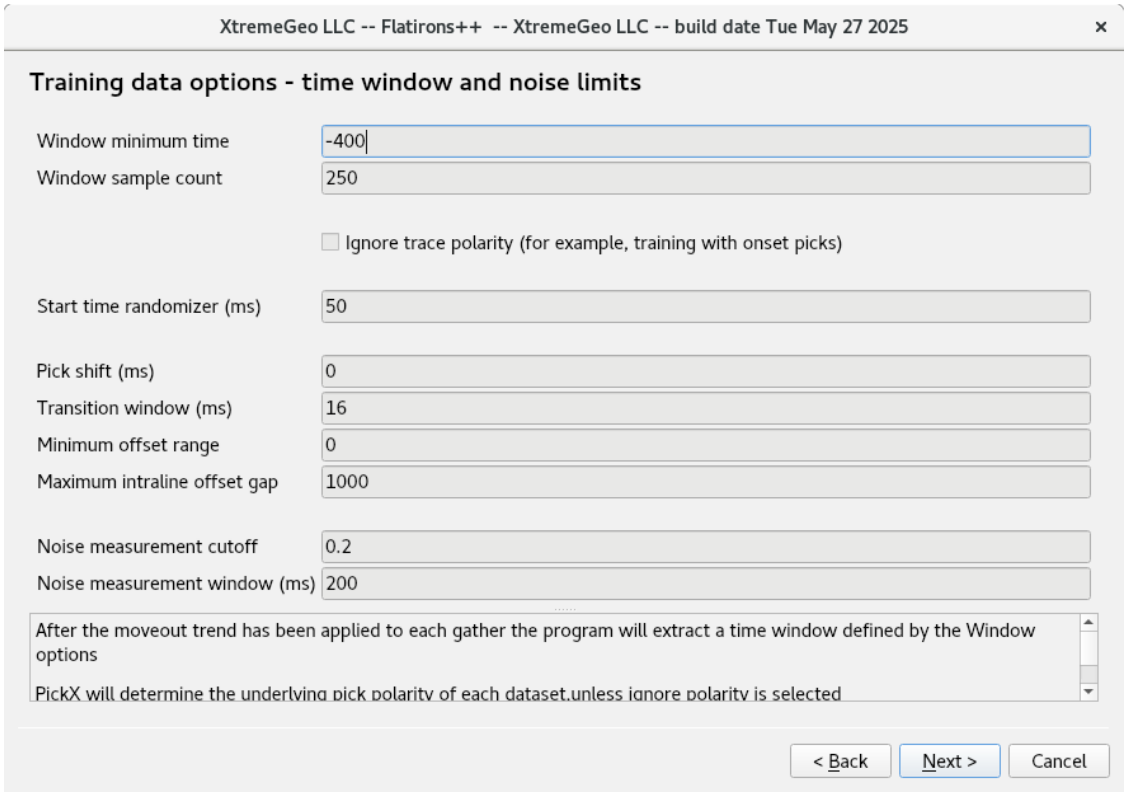
On the next page of the wizard please select “Standard Unet approach” for the model and “Mean squared error” for the loss function.



Flatirons++

Training data options - time windows, noise limits

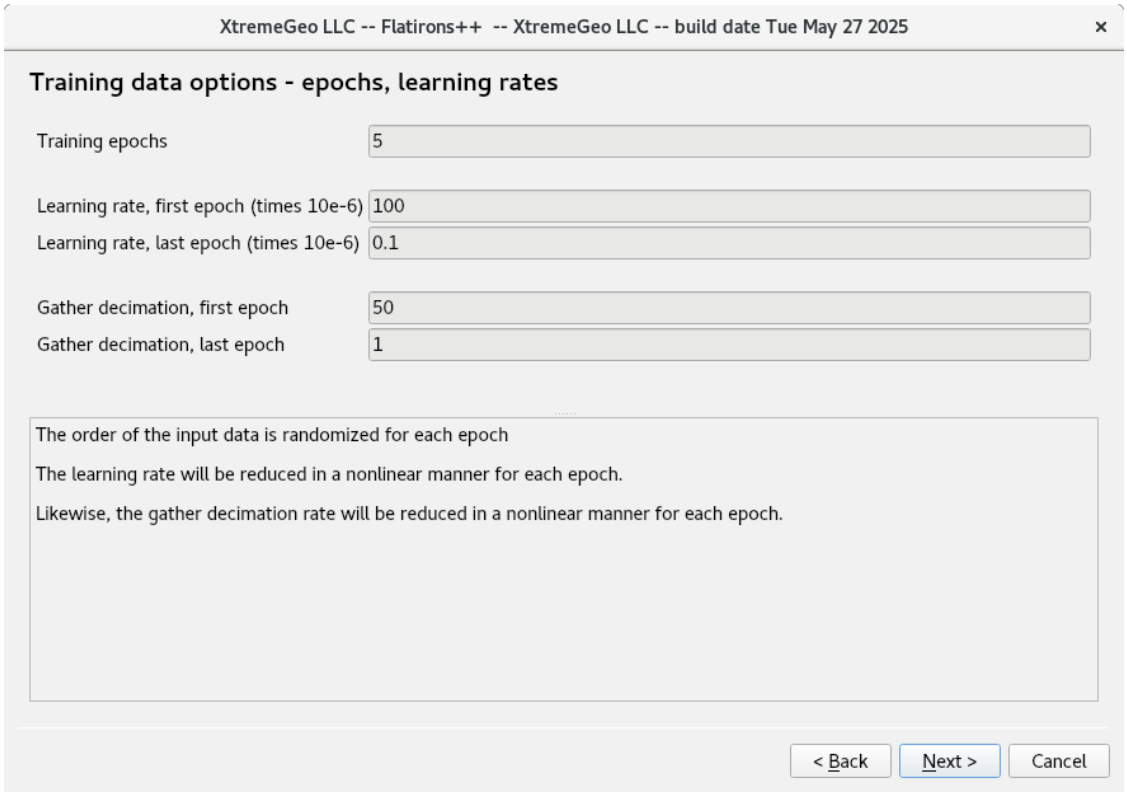
- Window options - after moveout has been applied a narrow (usually 800ms) time window is extracted from the gather. The defaults are usually fine
- Ignore trace polarity. Select this option if the training data was picked on amplitude onset
- Start time randomizer. This option stabilizes the model by adding some wobble to the trace start times after moveout has been applied.
- Noise measurement cutoff. This is a way of excluding picks on very noisy traces. A value of 1.0 is pure noise. The default value of 0.2 is probably too small for most surveys



Flatirons++

Training data options - Epochs and learning rates

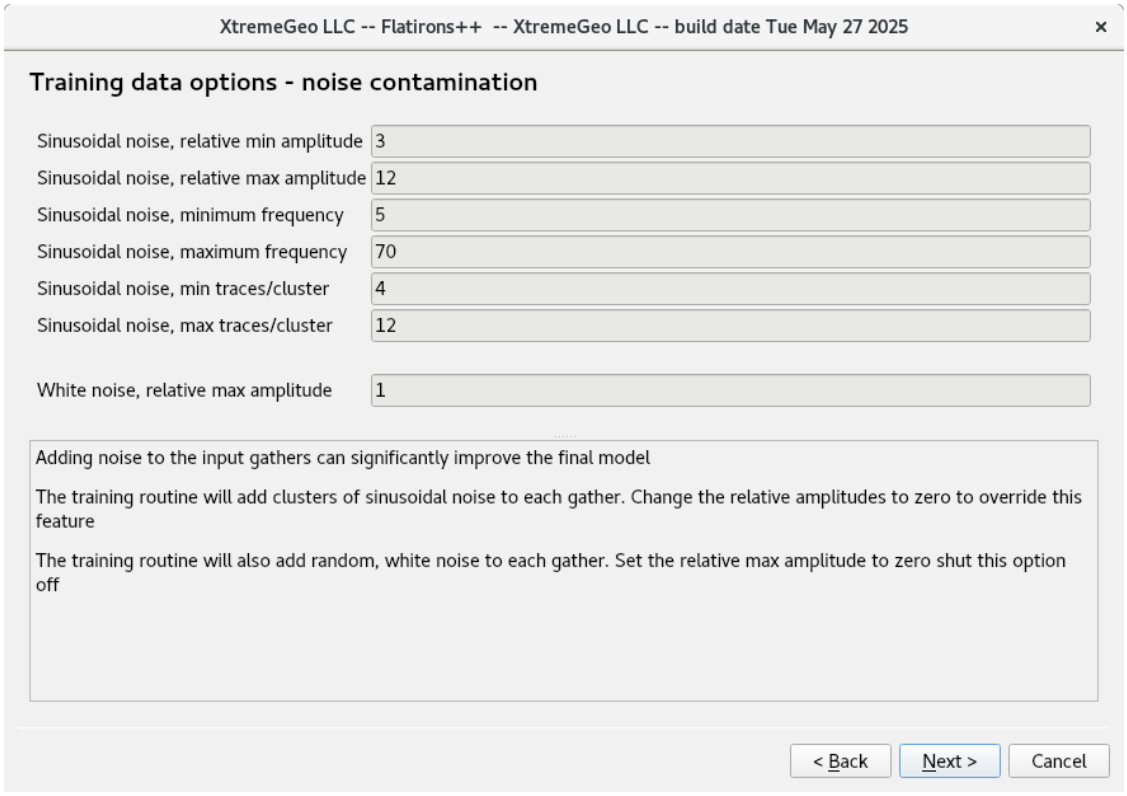
- Training epochs. During each epoch the training rate is reduced and the number of gathers is increased.
- The default learning rates are fine
- Good values for the gather decimation rates depend on the number of gathers in all the selected surveys. If the selected surveys have more than a couple hundred thousand gathers then the decimations should be increased to something like 500 and 10.



Flatirons++

Training data options - noise contamination

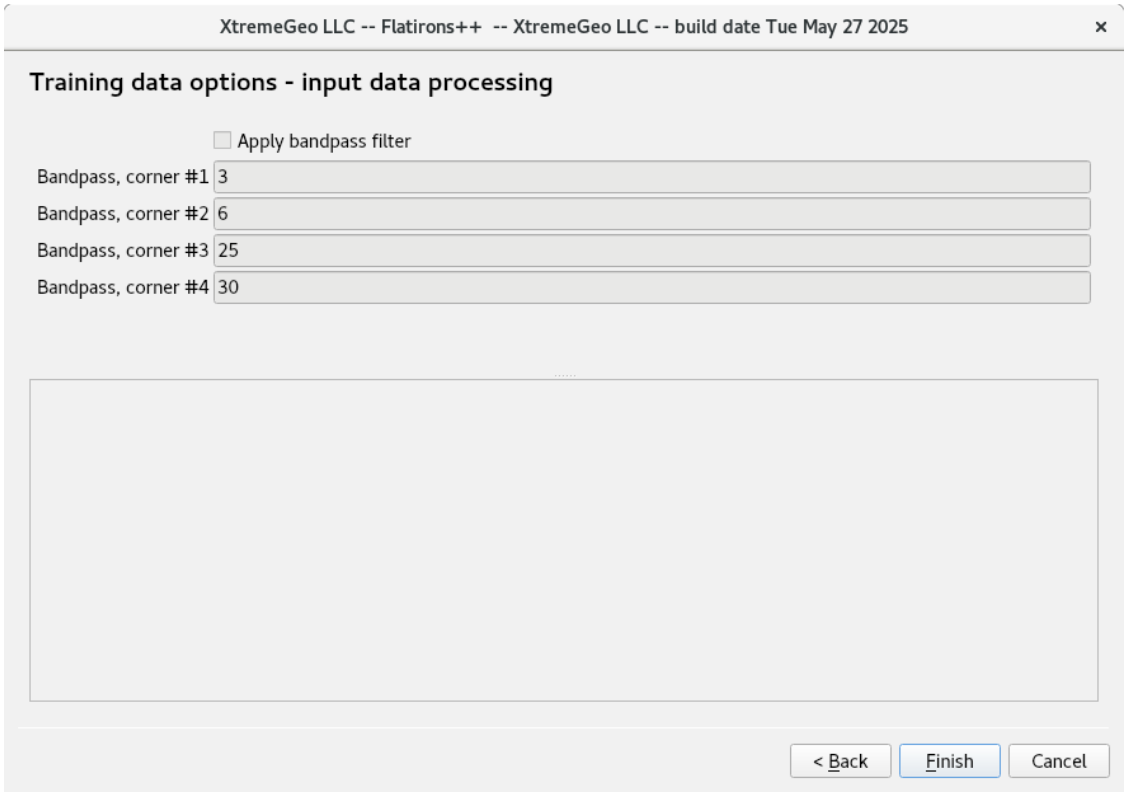
These options are crucial to obtain a good model. The “sinusoidal noise” options mimic contamination from nearby shots. - Amplitudes - these should be decreased from the defaults to about 1.0 and 3.0 - The dominant frequency will be chosen at random for each gather - The number of traces on the sinusoidal cluster, chosen at random for each gather



Flatirons++

Training data option - input data filter

This option is rarely used, since in general noisy data can help stabilize the training.

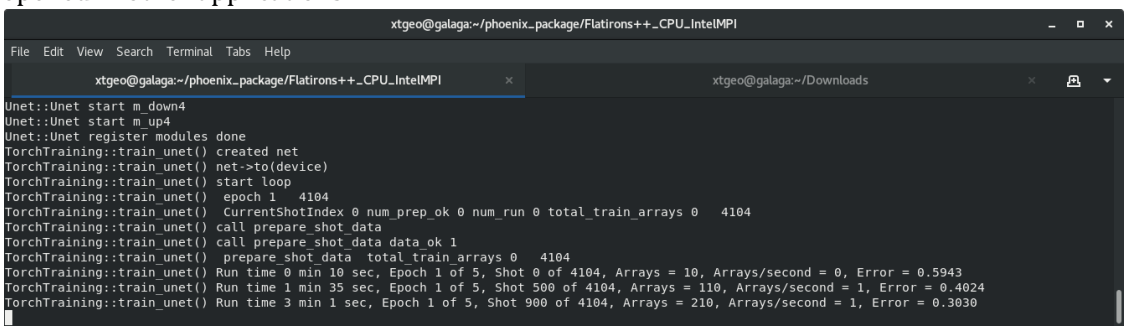


Flatirons++

Start the training run and monitor training progress

Click “Finish”. On a CPU the training will take around 12 to 24 hours.

The progress of the training may be viewed in the terminal window used to launch Flatirons++. The print out will show epoch, shot within epoch, total arrays (gathers) using for training, and Error.
The Error should drop to roughly 0.02 fairly quickly and hover there. But don’t be fooled - much more training is required!
A decent machine should use about 4 arrays per second, and it takes a minimum of 50,000 gathers to obtain a good model.
NOTE - You don’t have to wait for training to finish to use the model. It is saved every few minutes and may be opened in other applications



Flatirons++

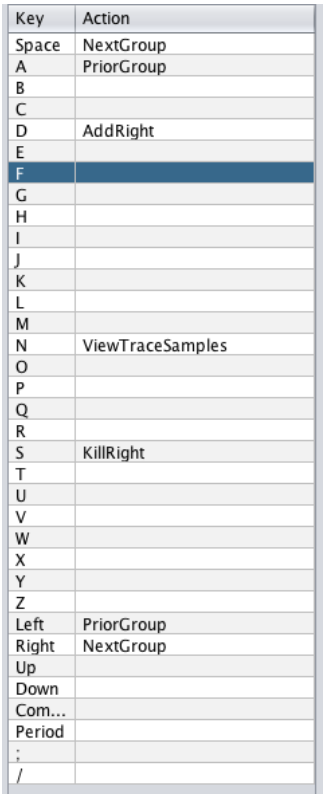
17 The Icons

A comprehensive listing of all icons used in Flatirons, preferably in 2(?) column tabular format: Icon & Description. Perhaps have column for location in Flatirons?

It is important to realize that the icon tabs are not necessarily identical for the same icon. That is, different tab windows may contain the same icon tab but the options, appearance and functionality is not identical. For example, the **Basemap** tab icon in the **Picker** window is not the same as in the **Stack velocity** window.

18 Picker Control: Hotkeys

The Picker controls icon tab ( ) has several keys with pre---defined actions to aid in picking events. Figure 20---1 shows a screen capture of the default Hotkeys panel.



Key	Action
Space	NextGroup
A	PriorGroup
B	
C	
D	AddRight
E	
F	
G	
H	
I	
J	
K	
L	
M	
N	ViewTraceSamples
O	
P	
Q	
R	
S	KillRight
T	
U	
V	
W	
X	
Y	
Z	
Left	PriorGroup
Right	NextGroup
Up	
Down	
Com...	
Period	
;	
/	

Figure 20---1 Pre---defined hotkeys in the Picker controls icon tab.

Table 20---1 lists all the Actions that are built---in to Flatirons. Any of these Actions can be assigned to a key to form a hotkey. The same Action can be assigned to multiple keys. Unless stated otherwise, all Actions apply to the single trace nearest the current mouse cursor. For actions that apply to multiple traces, unless indicated, only traces in the current window are affected. Specifically, if currently in a zoomed---in view, any traces outside the current display are unaffected.

Table 20---1 Actions and definitions for defined hotkeys in the Picker controls icon tab.

Shortcut Name (Action)	**Definition**
Off	Key is undefined. Action field is blank.
ViewTraceSamples	Used in conjunction with **View trace samples** icon tab ( ). Updates this tab with current trace.
NextGroup	If grouping is on, move to the next group.

PriorGroup	If grouping is on, move to the prior group.
NextEnsemble	?
PriorEnsemble	?
AddSinglePick	Pick a single trace. If Event is Peak or Trough, pick will snap to Event.
AddSinglePickNoSnap	Pick a single trace at time location of mouse cursor.
KillSinglePick	Delete the pick.
AddRight	Pick every trace to the right of mouse cursor, in the current window.
KillRight	Delete all picks to the right of the mouse cursor, in the current window only.
AddLeft	Pick every trace to the left of mouse cursor, in the current window only.
KillLeft	Delete all picks to the left of the mouse cursor, in the current window only.
AddBoth	Pick every trace in the current window.
KillBoth	Delete every trace in the current window.
GeomCorrection	?
CrudeGeometryUpdate	?
PickZeroTime	Pick event nearest to zero time, for every trace in current window.
PickCursorTime	Pick event nearest to time of mouse cursor, for every trace in current window.
KillTrace	For the trace nearest the mouse cursor, set its status to Killed and remove from display.
FlipPolarity	Flip the polarity of a single trace.
FlipTracePolarity	?
NextInline	?
PriorInline	?

NextCrossline	?
PriorCrossline	?
ShiftPatternUp	? (Only for 2D shot gathers)
ShiftPatternDown	? (Only for 2D shot gathers)
CopyAutoPickToUser	? Copy auto picks to user pick
CopyAutoPickToUserOverwrite	
RubberBandStart	Begin RubberBand at current mouse cursor position.
RubberBandAddPoint	
RubberBandCancel	Cancel the current RubberBand action.
RubberBandAddPicksSnapToEvent	
RubberBandAddPicksAtLine	
RubberBandKillPicks	Kill all picks within the RubberBand section.
RubberBandKillTraces	Kill all traces within the RubberBand section.
SetFlag0	
SetFlag1	
SetFlag2	
SetFlag3	
SetFlag4	
KillCurrentShotOrReceiver	
KillPicksInZoom	Kill all picks in current zoom zoom window.
KillPicksOutsideZoom	Kill all picks outside current zoom window.
RestackShotRec	

19 Picker Control: Left Mouse Action

The Picker controls icon tab ( ) has a drop---down list for defining the action of a left mouse click. Occasionally a user may prefer to use the Left Mouse button instead of Hotkeys for some Actions. So several of the most common actions described in Appendix 1 are also available for the Left Mouse button. Table 21---1lists the possible Actions and associated definition.

Table 21---1 Items in drop---down list for Left mouse action in Picker control icon tab.

Shortcut Name (Action)	Definition
ControlPointAdd	Single clicks create multi---segmented red line as mouse moves across trace display. Double click causes trace to be picked along red line; snap to event if enabled (default).
ControlPointKill	Opposite of ControlPointKill --- picks are deleted. If red line crosses a trace multiple times, only the last crossing is applied.
ControlPointSelect	Similar to ControlPointKill except double click opens popup window giving option to Kill or Flip polarity to all visible traces or all traces.
AddRight	Pick every trace to the right of mouse cursor, in the current window.
KillRight	Delete all picks to the right of the mouse cursor, in the current window only.
AddLeft	Pick every trace to the left of mouse cursor, in the current window only.
KillLeft	Delete all picks to the left of the mouse cursor, in the current window only.
AddBoth	Pick every trace in the current window.
KillBoth	Delete every trace in the current window.
AddSinglePick	Pick a single trace. If Event is Peak or Trough, pick will snap to Event.
KillSinglePick	Delete the pick.
KillTrace	For the trace nearest the mouse cursor, set its status to Killed and remove from display.
FlipPolarity	Flip the polarity of a single trace.
PenMode	Left---click---and---drag will pick traces that the 'pen' crosses. Moving too fast will cause some traces to be skipped. Pressing CTRL key in pen mode will delete picks.