Phoenix + DeepTrace Startup Usage Guide*

A.I. and physics working together

prepared by Front Range Geosciences

*this guide will cover the basics of picking and creating tomo models

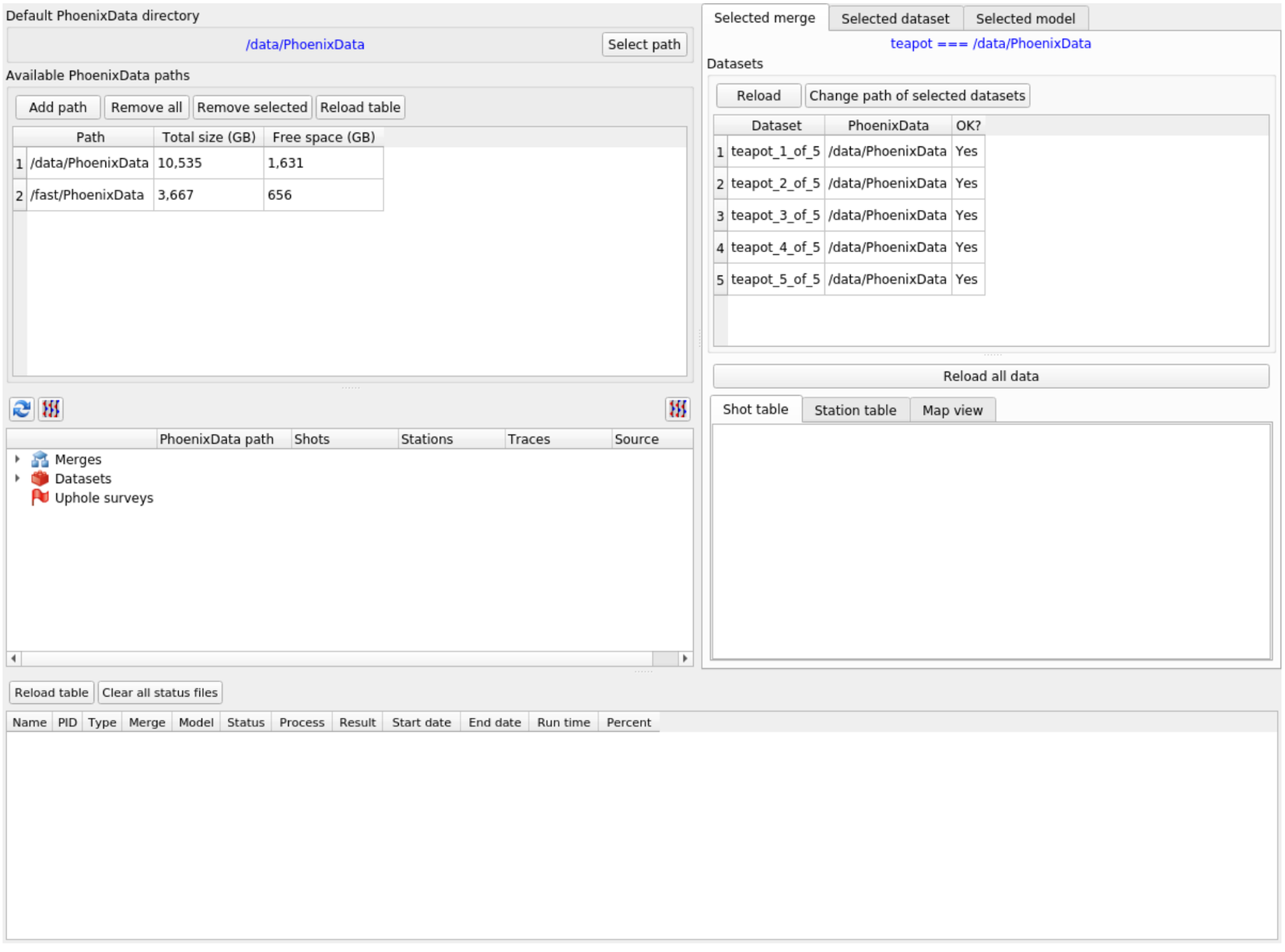
Contents

Creating a Project

SEGY Import

This is the Phoenix directory. From it you can create new projects and open existing ones.

To start a new project, first create a Dataset by right-clicking on Datasets and choosing “import SEGY files”.

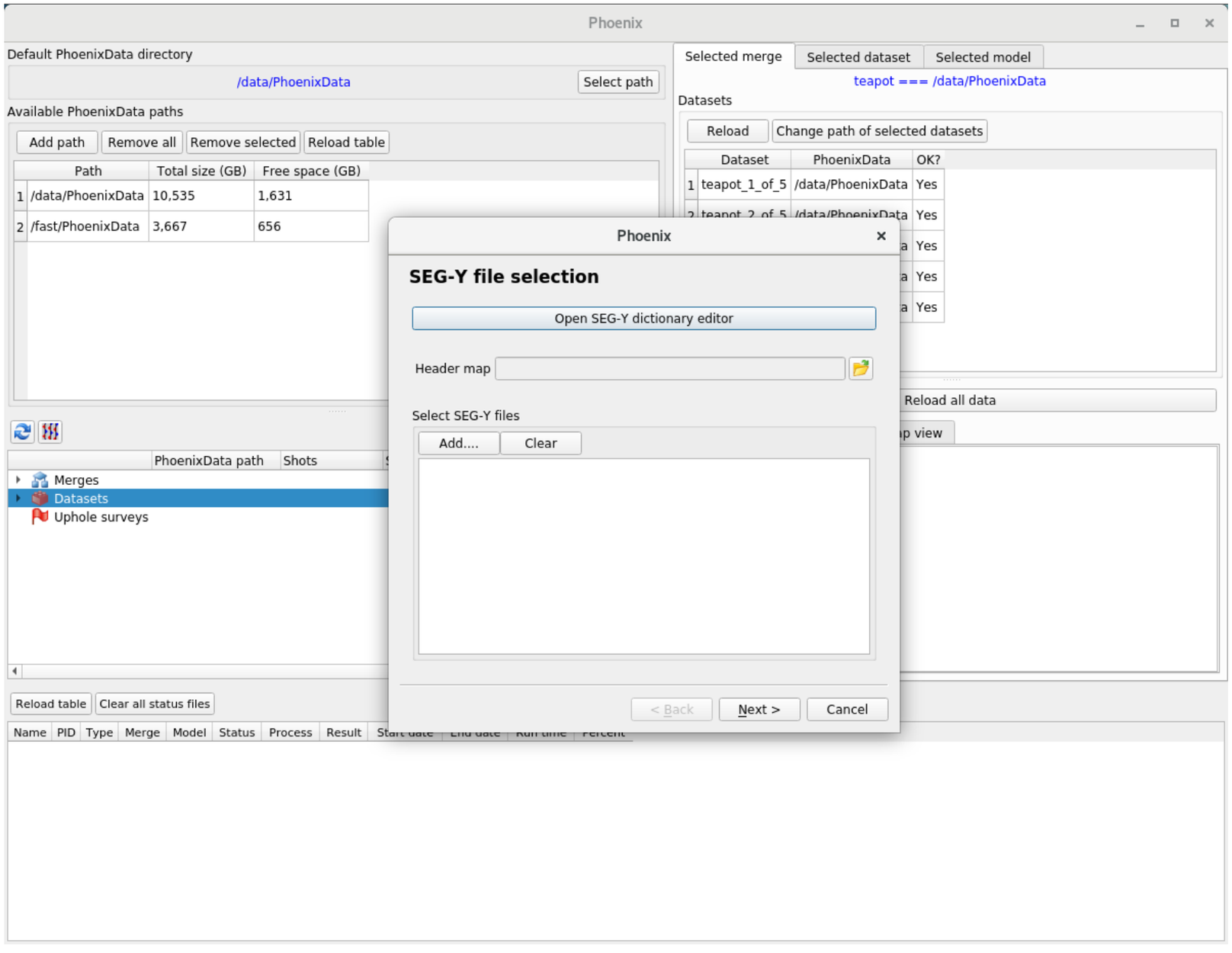


SEGY Import

In the SEG-Y file selection window, you can select as many SEG-Y files as you wish to create your project with. They must all share the same header map.

The header map is a csv-style file that defines the byte locations of various properties in the SEG-Y files. To create a new one, click the “Open SEG-Y dictionary editor” button.

If you already have a header map defined, you can select it and move on.



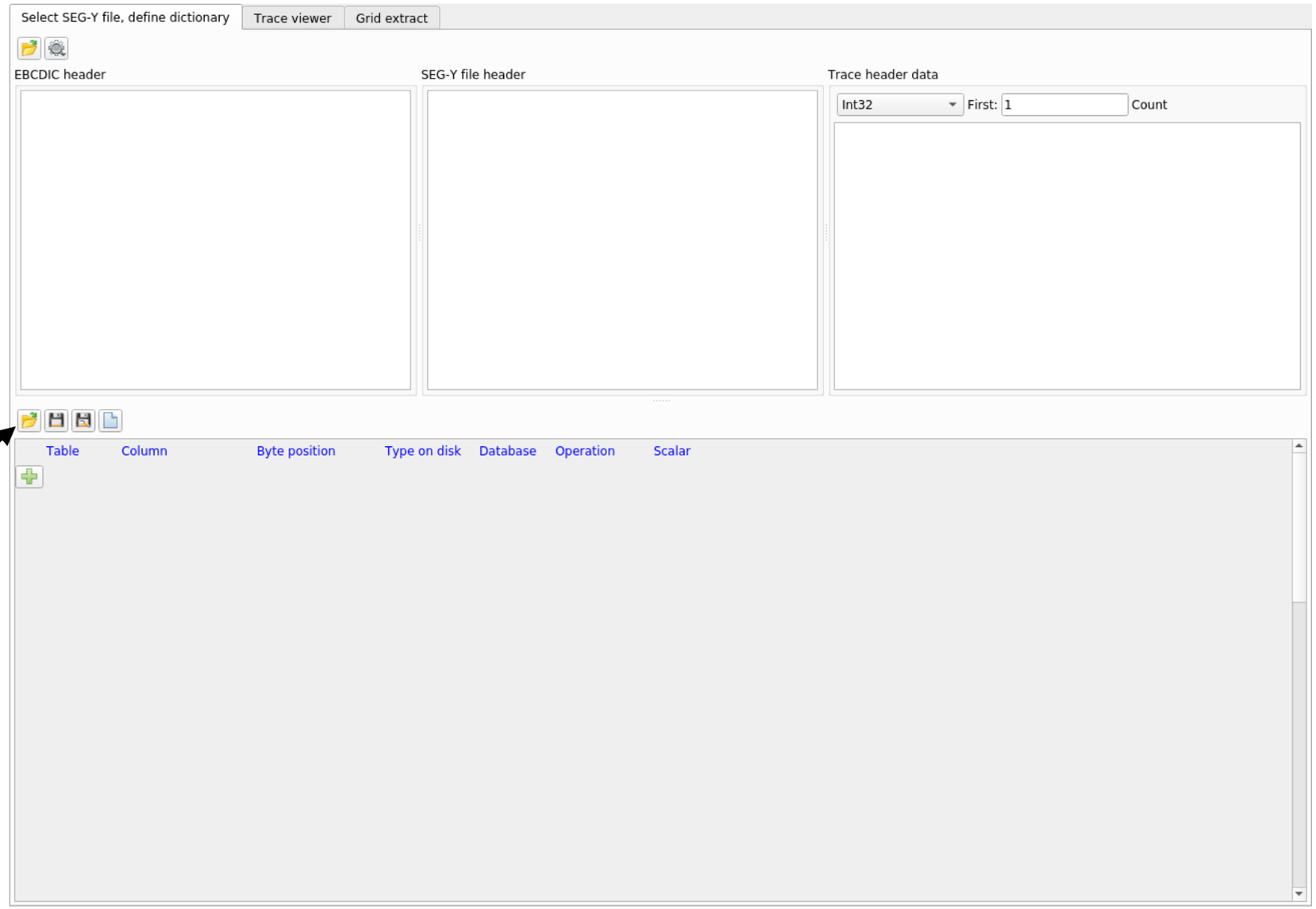
SEGY Import

In the SEG-Y dictionary editor, you can define a new header map.

The top left file button is used to choose a SEG-Y file.

The middle left file button is used to open an existing header map.

The blank paper icon starts a new header map.



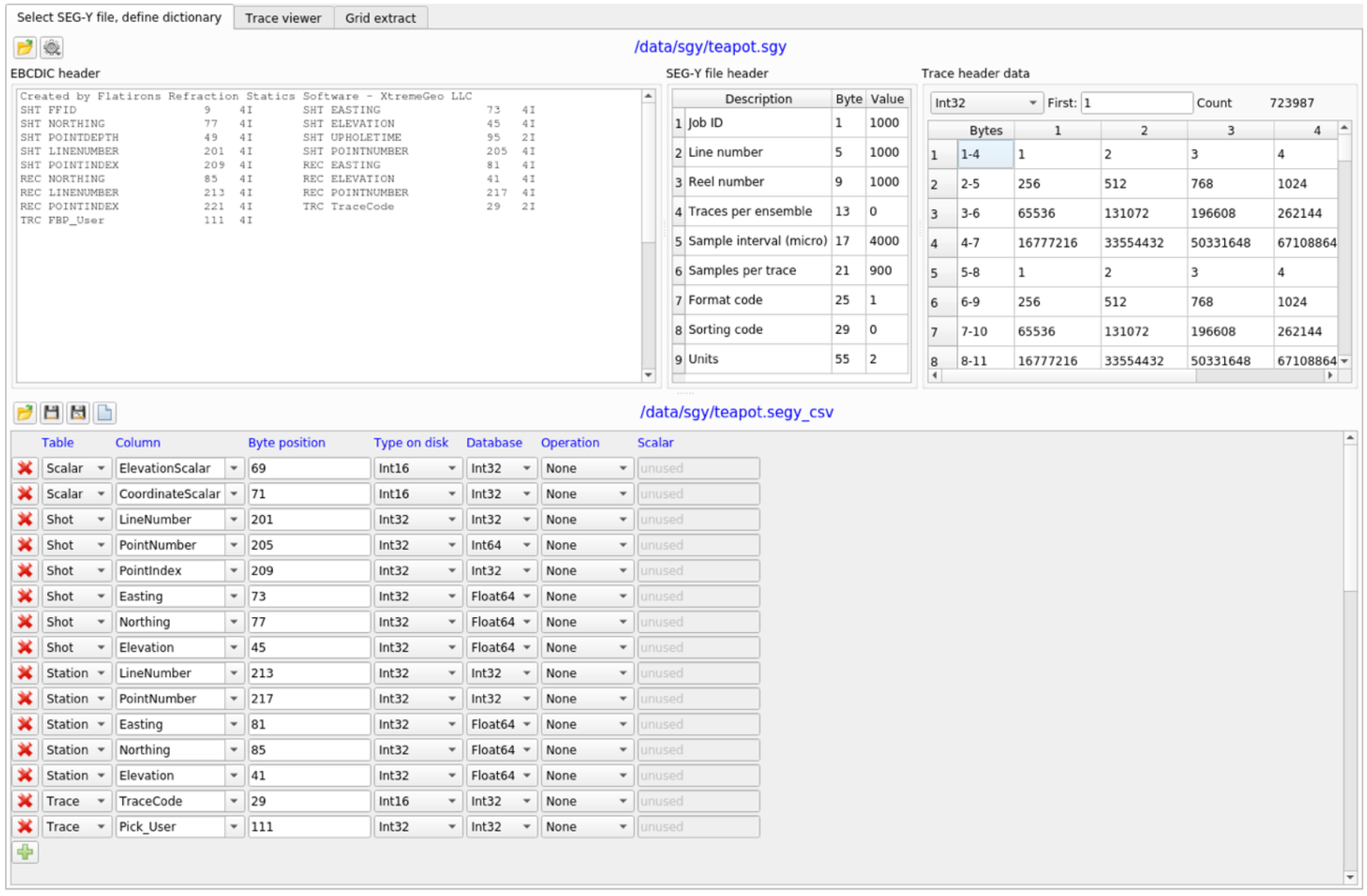
SEGY Import

After a SEG-Y file has been selected, you can begin to define a header map.

Click the green plus button to add a new row to your header map.

This image shows an example header map which matches the EBCIDIC found in the SEG-Y file.

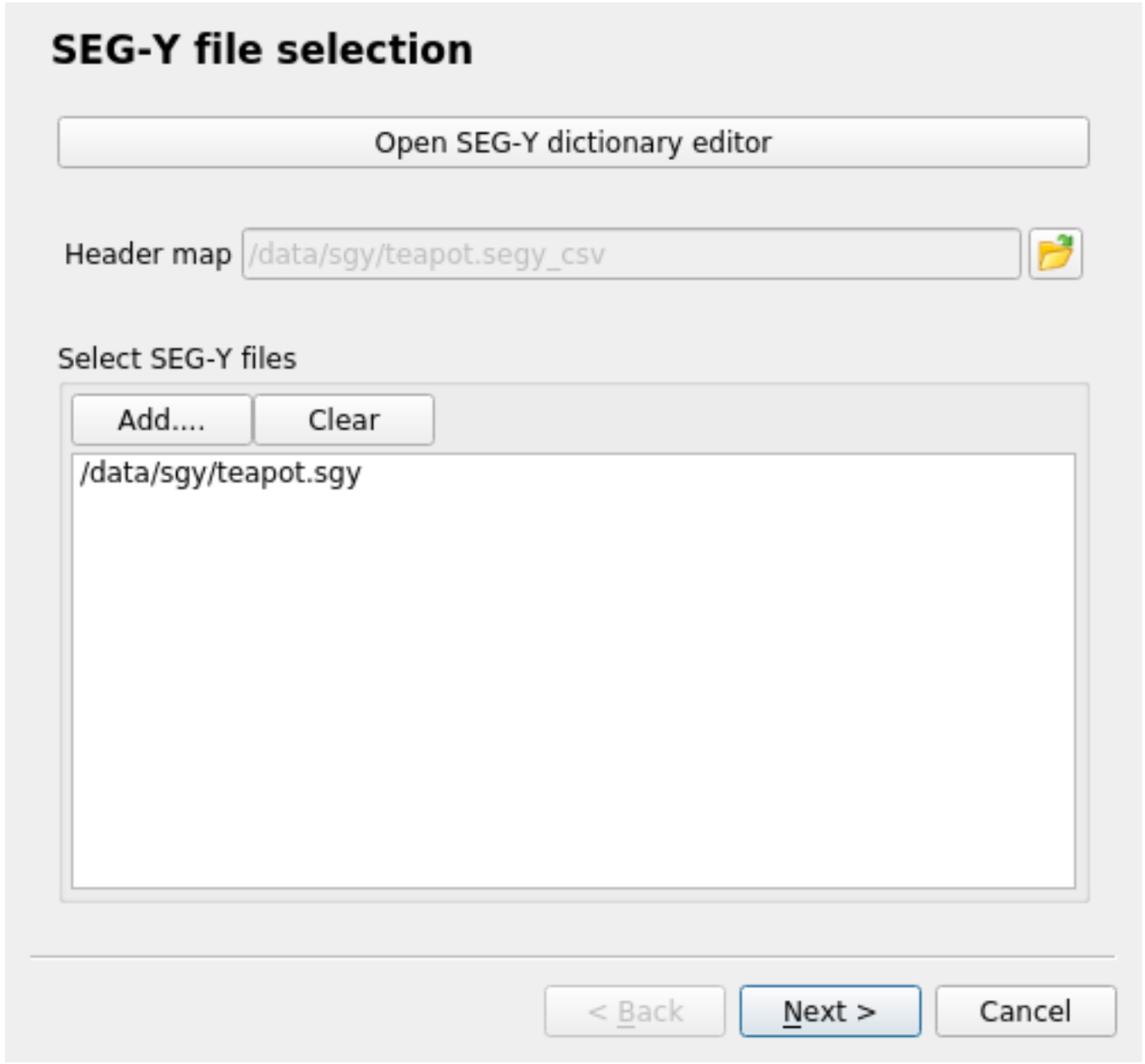
Save your header map using the right “save as” button. We use .segy_csv as the file extension for these header maps, but you can name them whatever you wish.



SEGY Import

After you have defined your header map, go back to the SEG-Y file selection screen and choose your SEG-Y file(s) and the header map you created.

When you've selected the files, click next.



SEGY Import

Choose a name for your dataset.

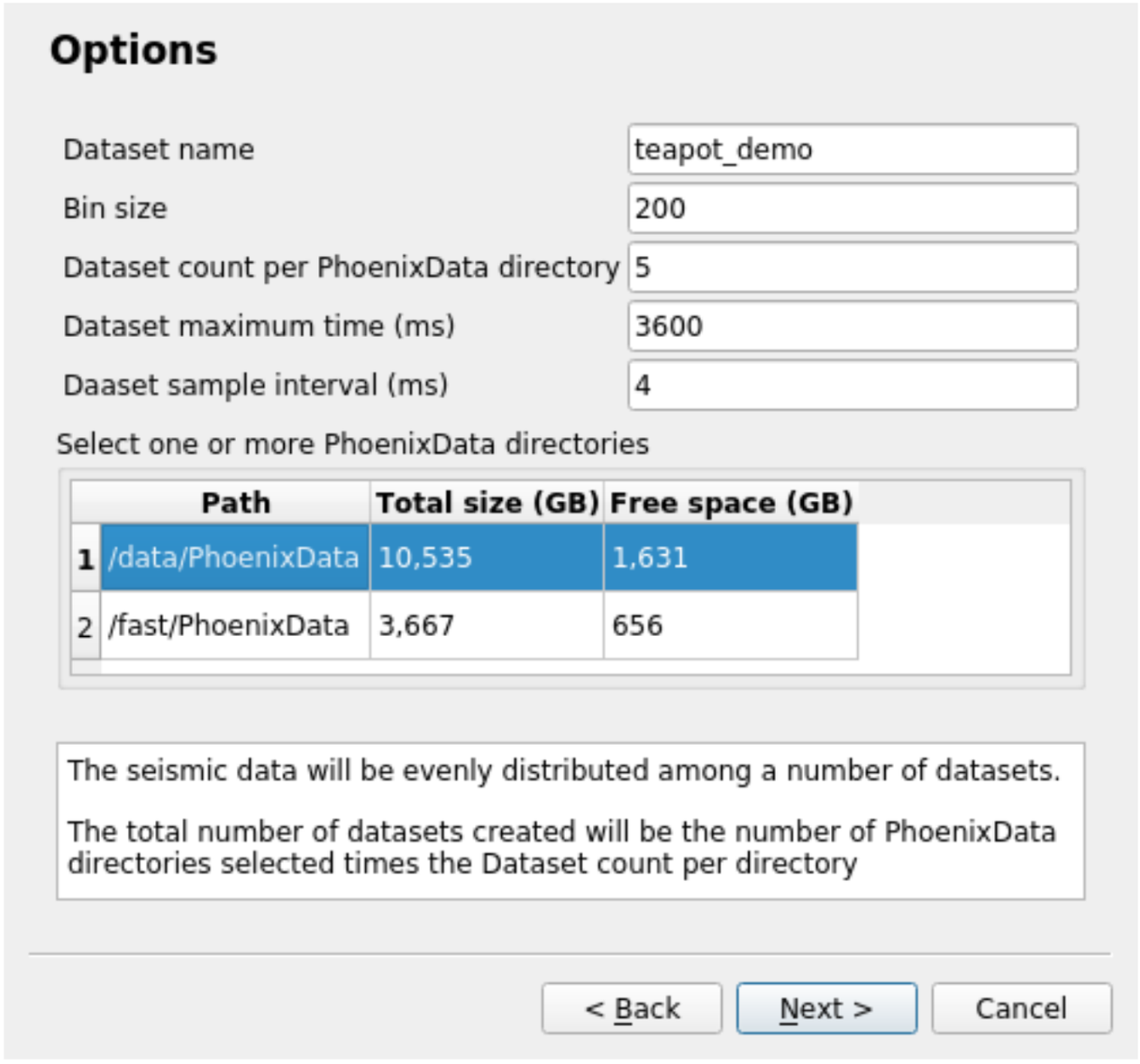
The data will be spread across the “Dataset count” number of collections. Many processes in Phoenix are threaded based off of this number.

In general, having more datasets can speed up processes. The default value of 5 will be fine for most projects.

Set the maximum time you want the seismic data to go out too. This should be at max (# samples * sampleRate), but may be any number smaller.

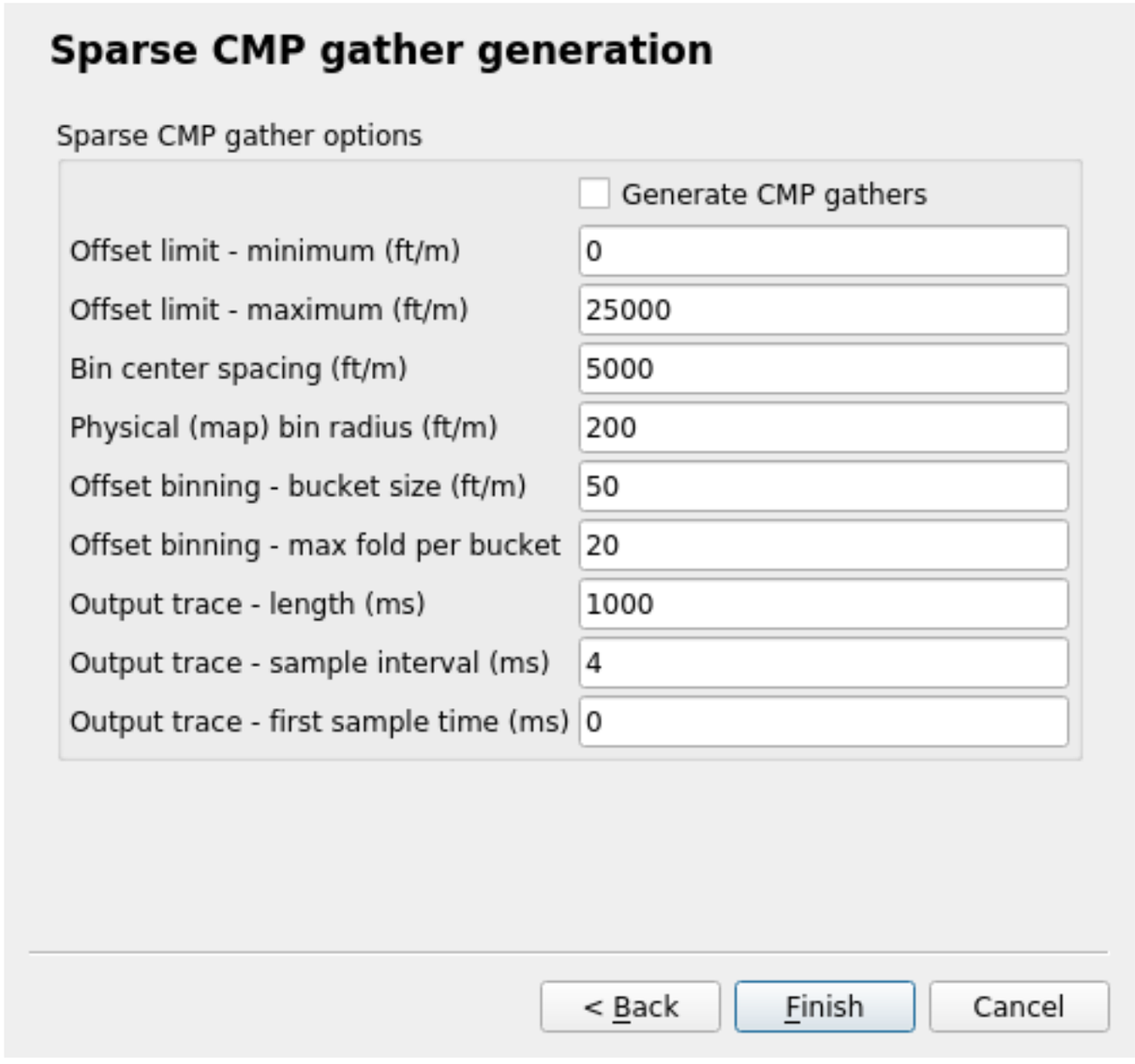
Choose the PhoenixData directory to store it in if you have more than one.

Click next.



SEGY Import

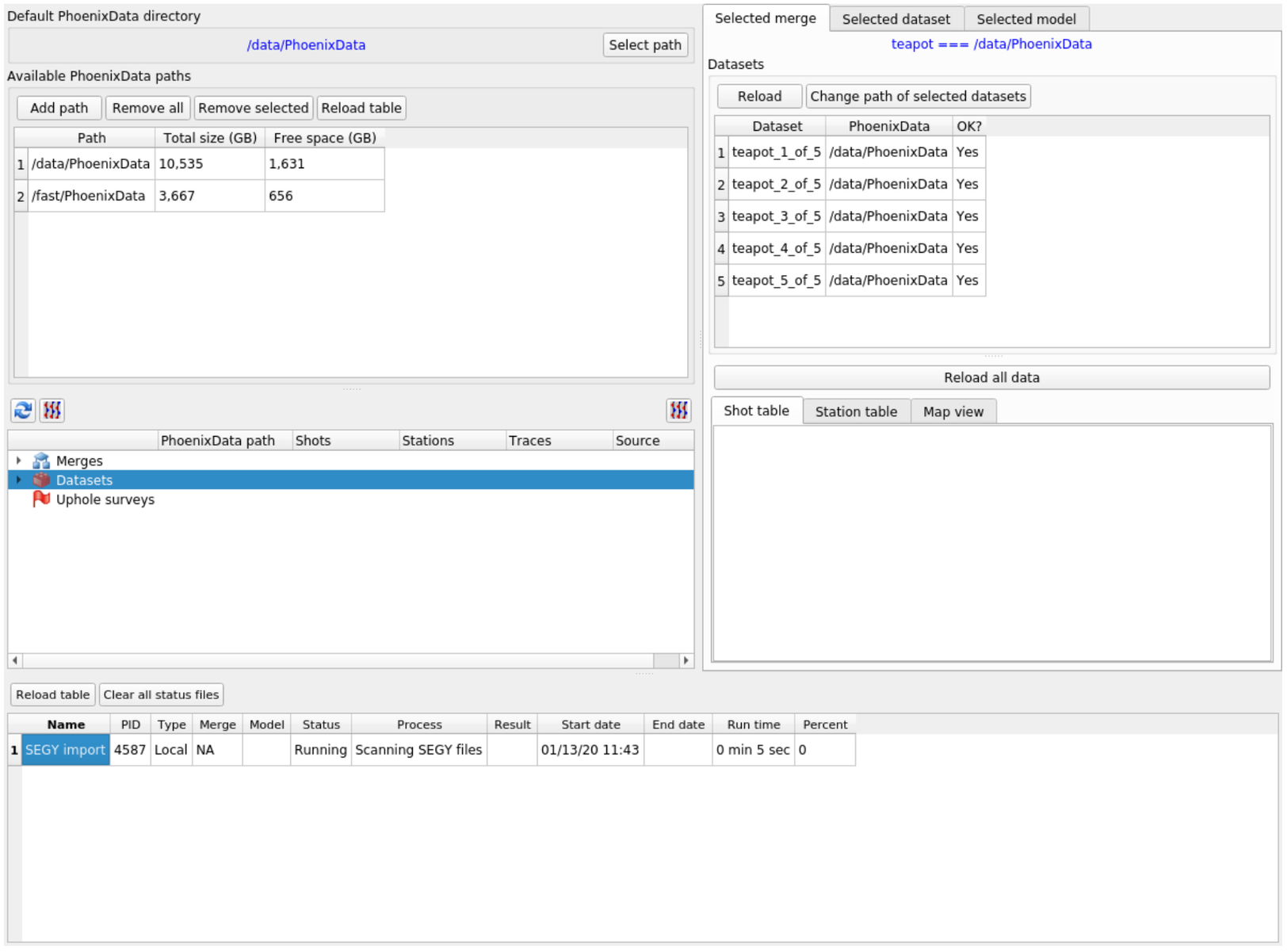
For now, uncheck “Generate CMP gathers” and click Finish.



SEGY Import

The datasets are now being imported by Phoenix. You can monitor the status of the import at the bottom of the window.

You can also monitor the status by clicking the Datasets drop down button and refreshing.

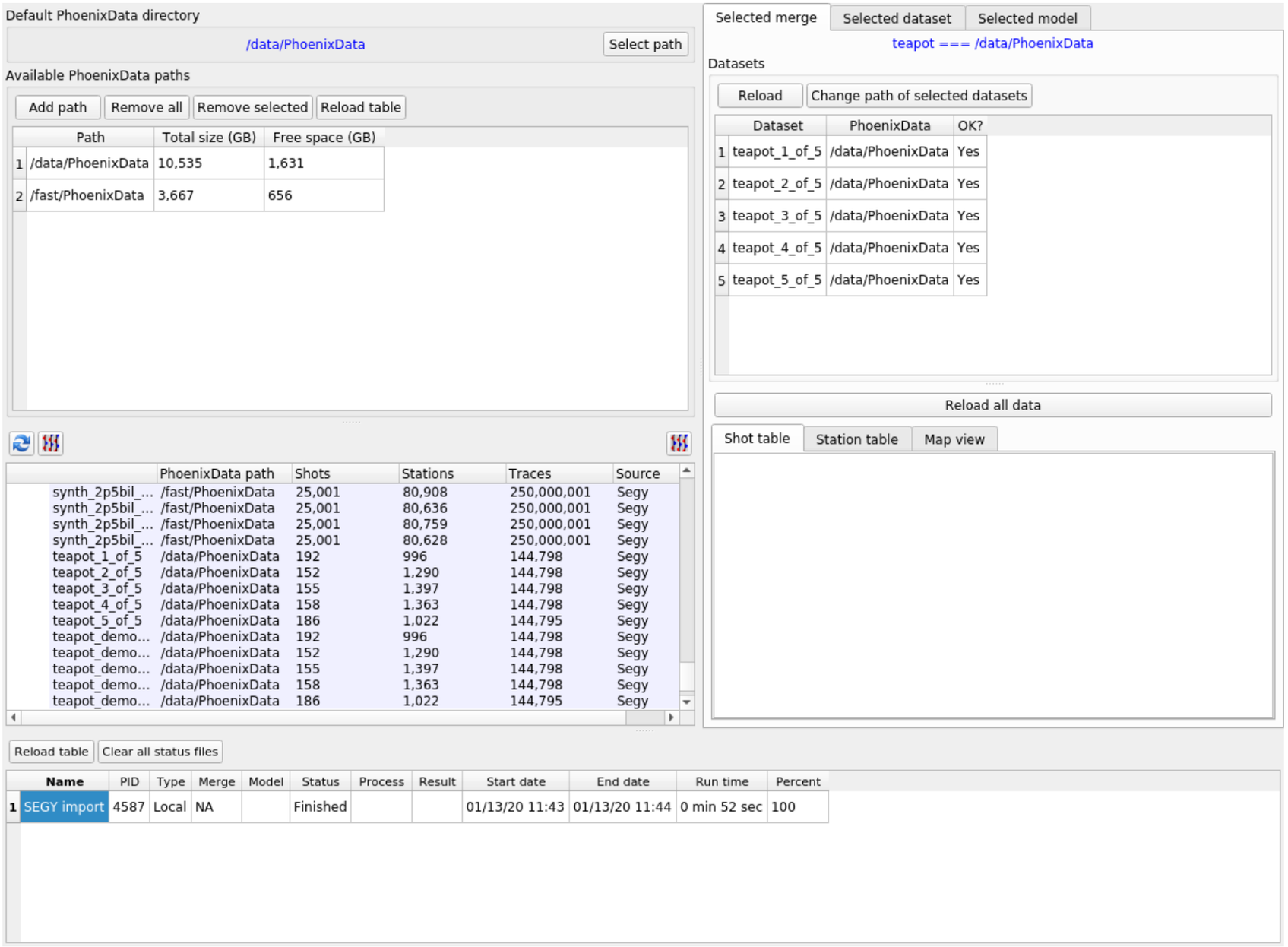


SEGY Import

Here we created the teapot_demo datasets (5).

Next we will create a Merge. A Merge can be composed of any Datasets whose shots and receivers are consistent.

Merges correspond to projects in similar software products.

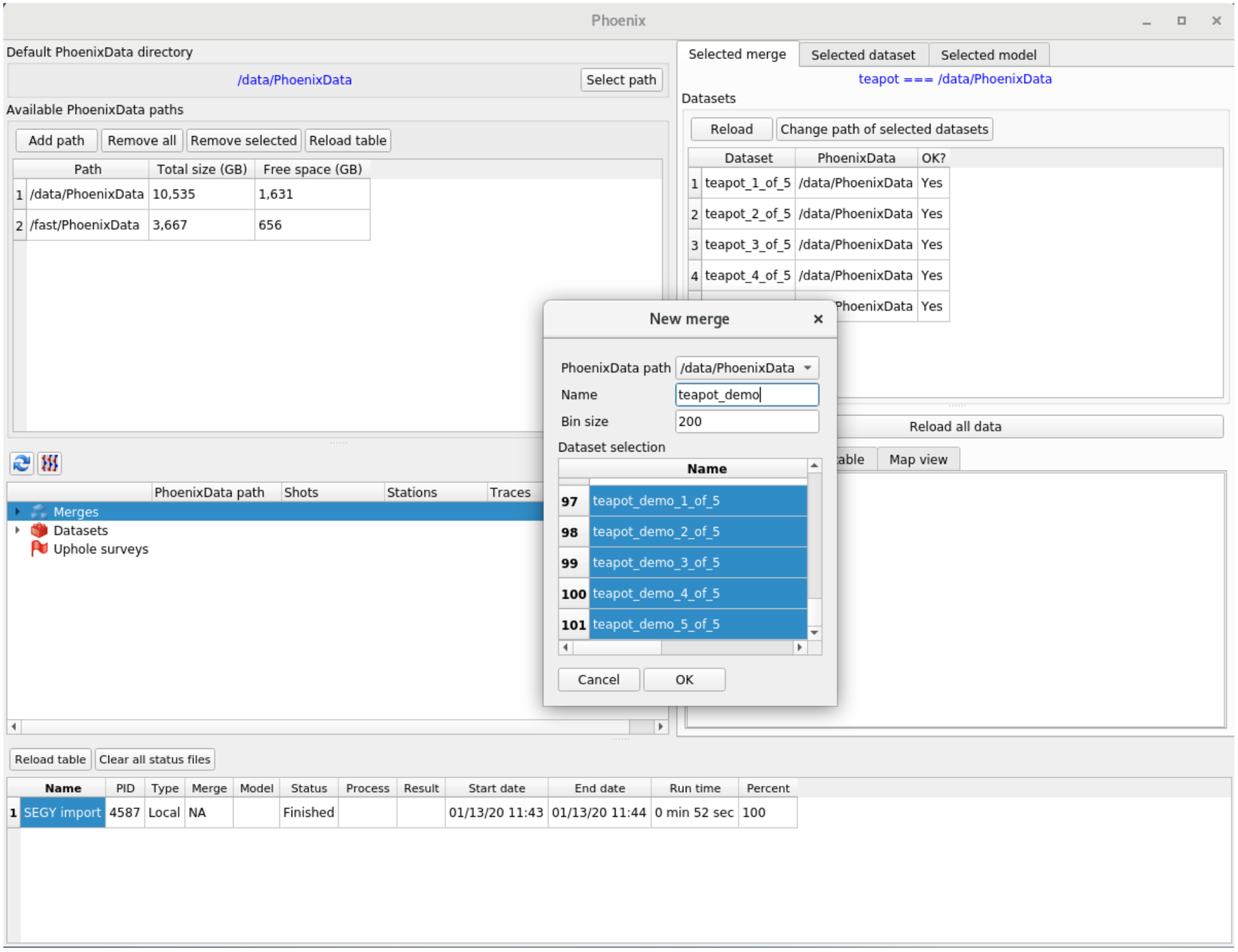


SEGY Import

To create a Merge, right click on the “Merges” drop-down list and select “Create new merge”.

Name your merge, and select the Datasets it will be composed of. We’ll call this Merge teapot_demo and select the 5 teapot_demo datasets we created earlier.

After you have created a Merge, right click on it under the Merge drop down and select “Open merge manager window”.



Merge Manager

Now we are in the Merge manager window.

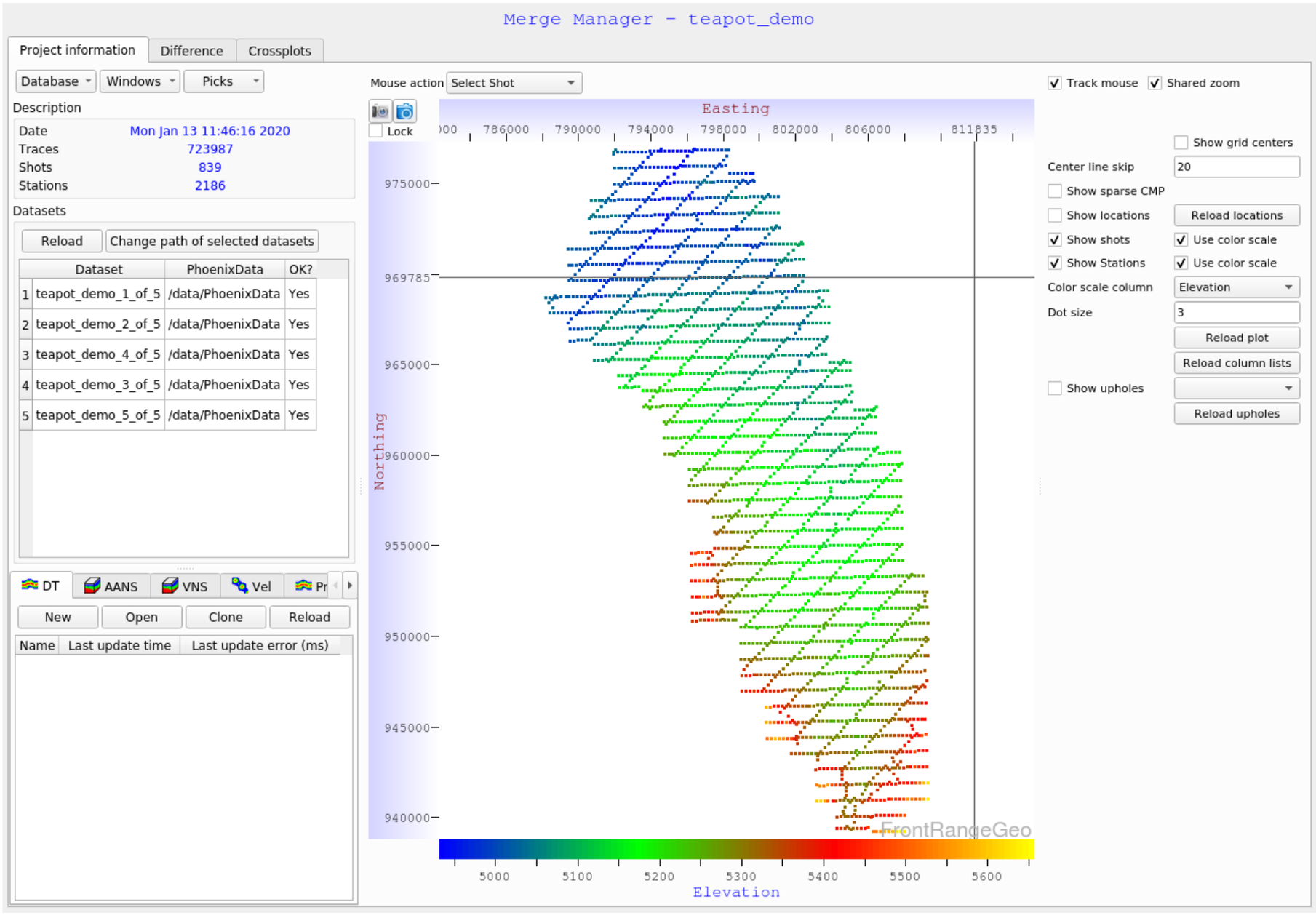
In the center we see a base map.

At the top left there are three tabs, Database, Windows, and Picks.

At the bottom left, there are tabs for our various models: DT (Delay Time), AANS (Auto-Adaptive Node Spacing), VNS (Variable Node Spacing), and others.

ANS is the new tomography in Phoenix. We'll come back to that later.

For now, click on the Windows tab in the top left and select "Open move out trend display - azimuth version".



Moveout Trend

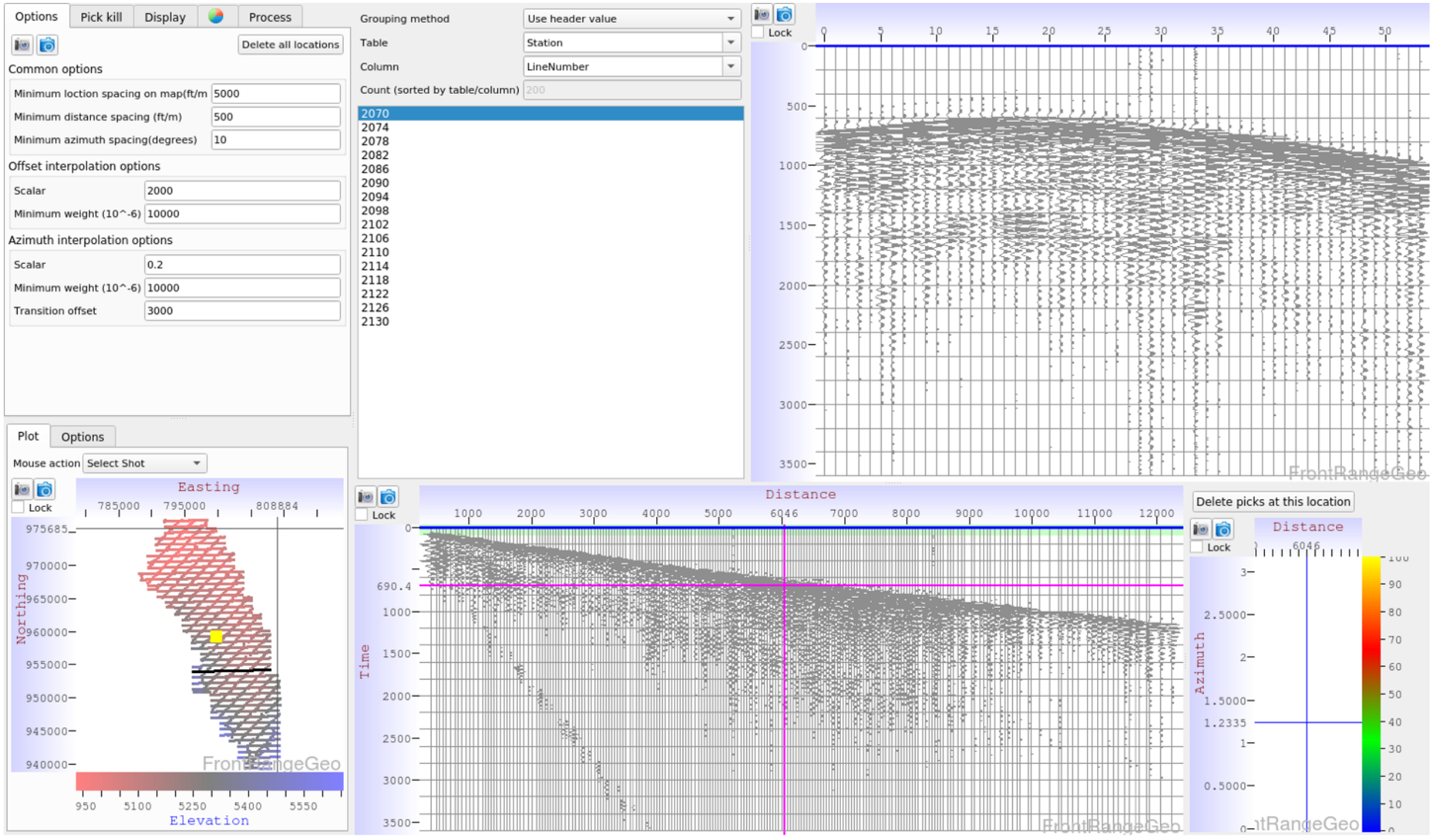
A move out trend is a function that maps an (offset, azimuth) pair to a time shift.

We use move out trends to flatten out seismic data.

Click anywhere on the base map to select a shot gather.

In the “Grouping method” tab at the middle top, select “Use header value” and select Station and Line Number as in the image.

Click on arrival times on the seismic data until it is relatively flat.



Moveout Trend

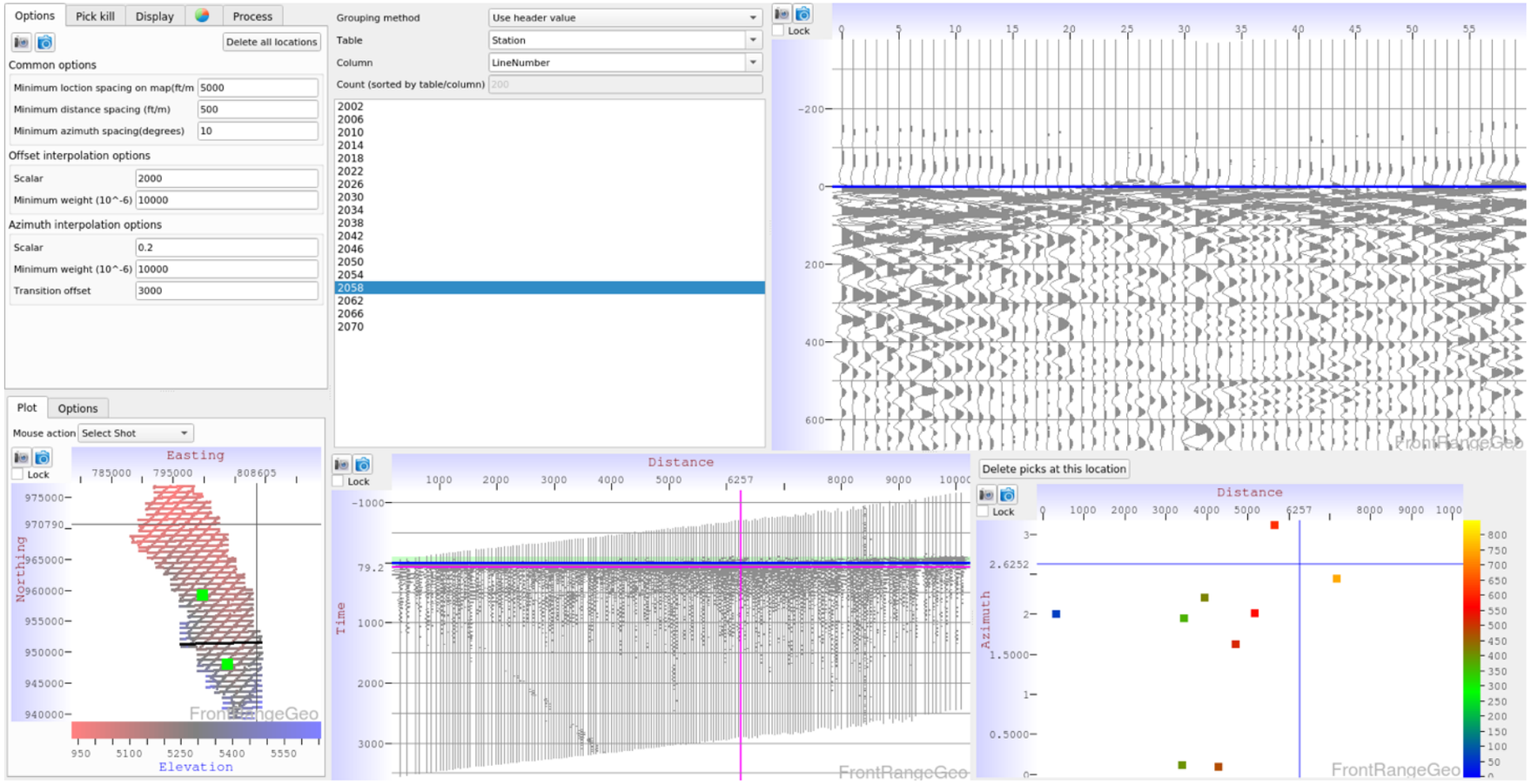
When you are satisfied that the data is relatively flat, click on another line number (2058 in the image) to move to another line in the shot you selected, and repeat the process.

Click around on the base map at various locations and repeat this process.

Choose as many locations as you can, and get the seismic as flat as you can.

The better you can make this move out trend, the easier it will be to pick your data.

When you are finished, simply close the window.



Picking

Now that we have a move out trend, let's open the picker window.

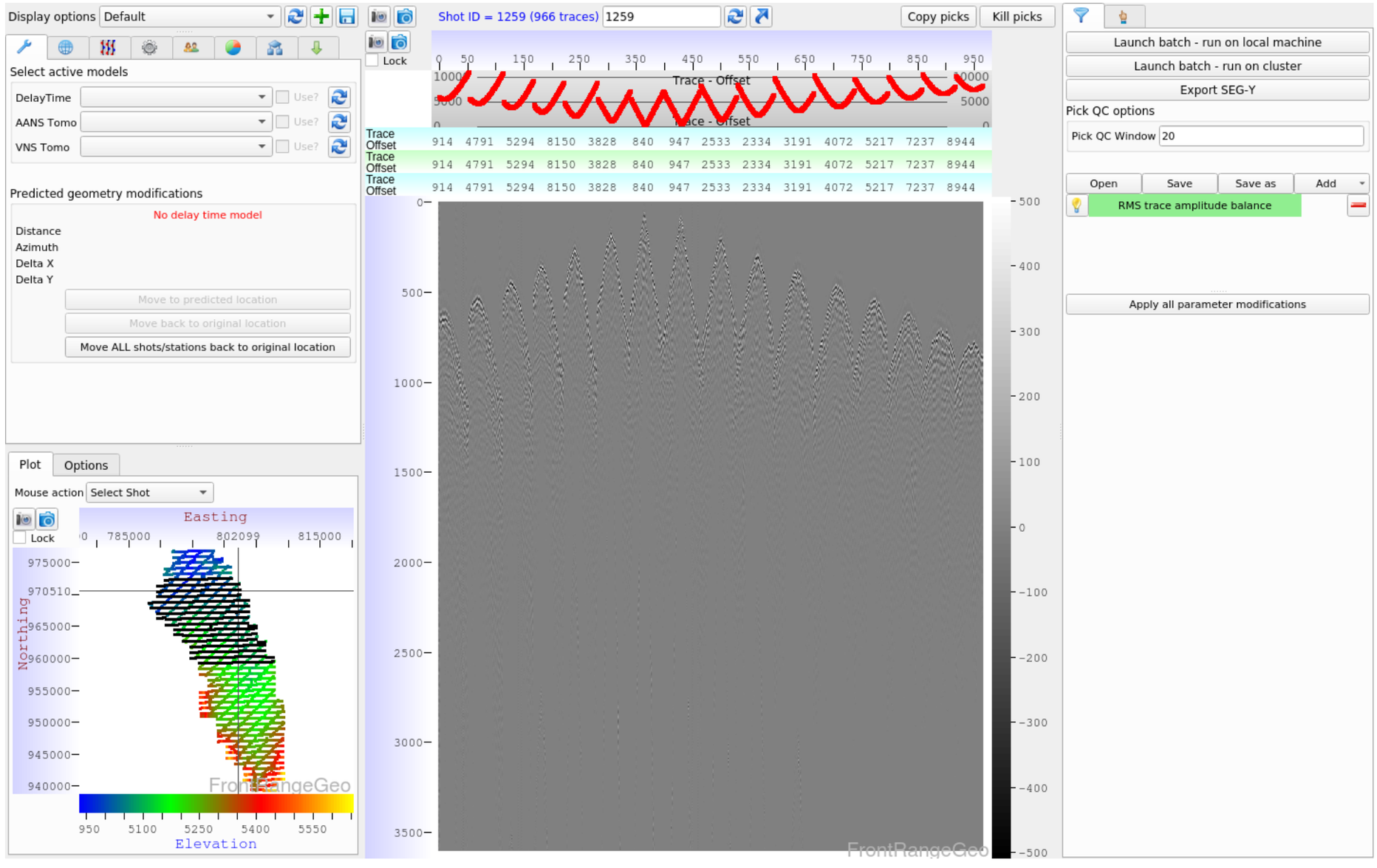
From the Merge Manager, select "Windows-> Open picker window".

This is the picker window.

On the bottom left, we have the base map. you can select shots by clicking at various locations.

At the top left there are various tabs. (Wrench, Globe, Squiggles, Gear, Group, Color Wheel, Network Graph, Arrow).

The wrench tab selects an existing velocity model. We can select a DelayTime, ANS Tomo, or VNS Tomo model. Click the refresh button and select a model from the dropdown to load it into memory, if one exists.



Picking

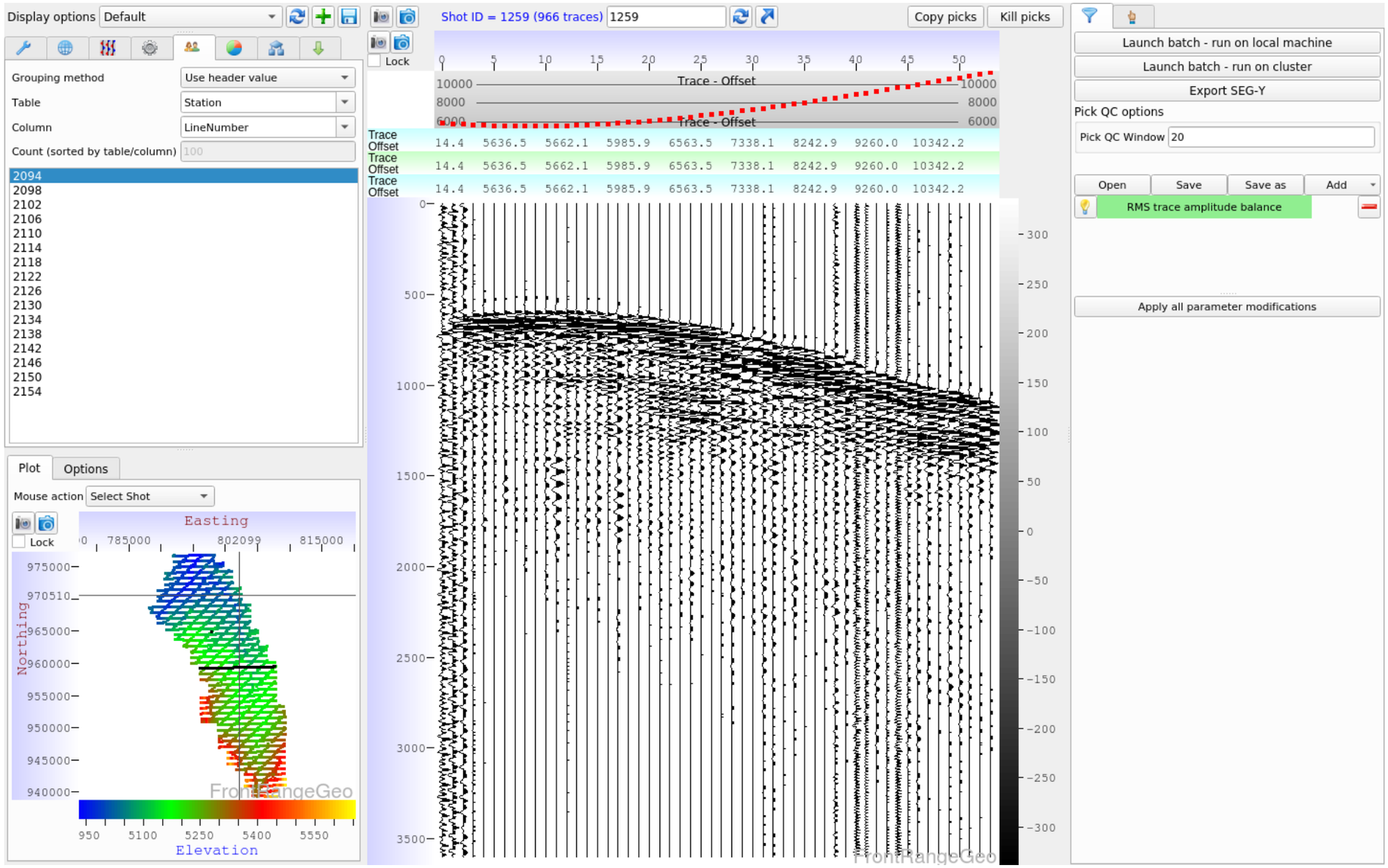
Click on the grouping tab.

Under grouping method, select “Use header value” and then “Station”, and “LineNumber” as in the image.

Now we are looking at a single line gather. The Line and Shot are represented on the base map as a Black Line and Dot respectively.

Now that we have selected a line, we can apply processing to it.

(You can apply processing to an entire shot gather as well, but for this guide we will go on a line-by-line basis as it’s easier to see.)



Picking

On the right side of the picker window is the “Process Flow”. This is an extremely powerful and flexible set of processing steps that will be applied to the seismic data.

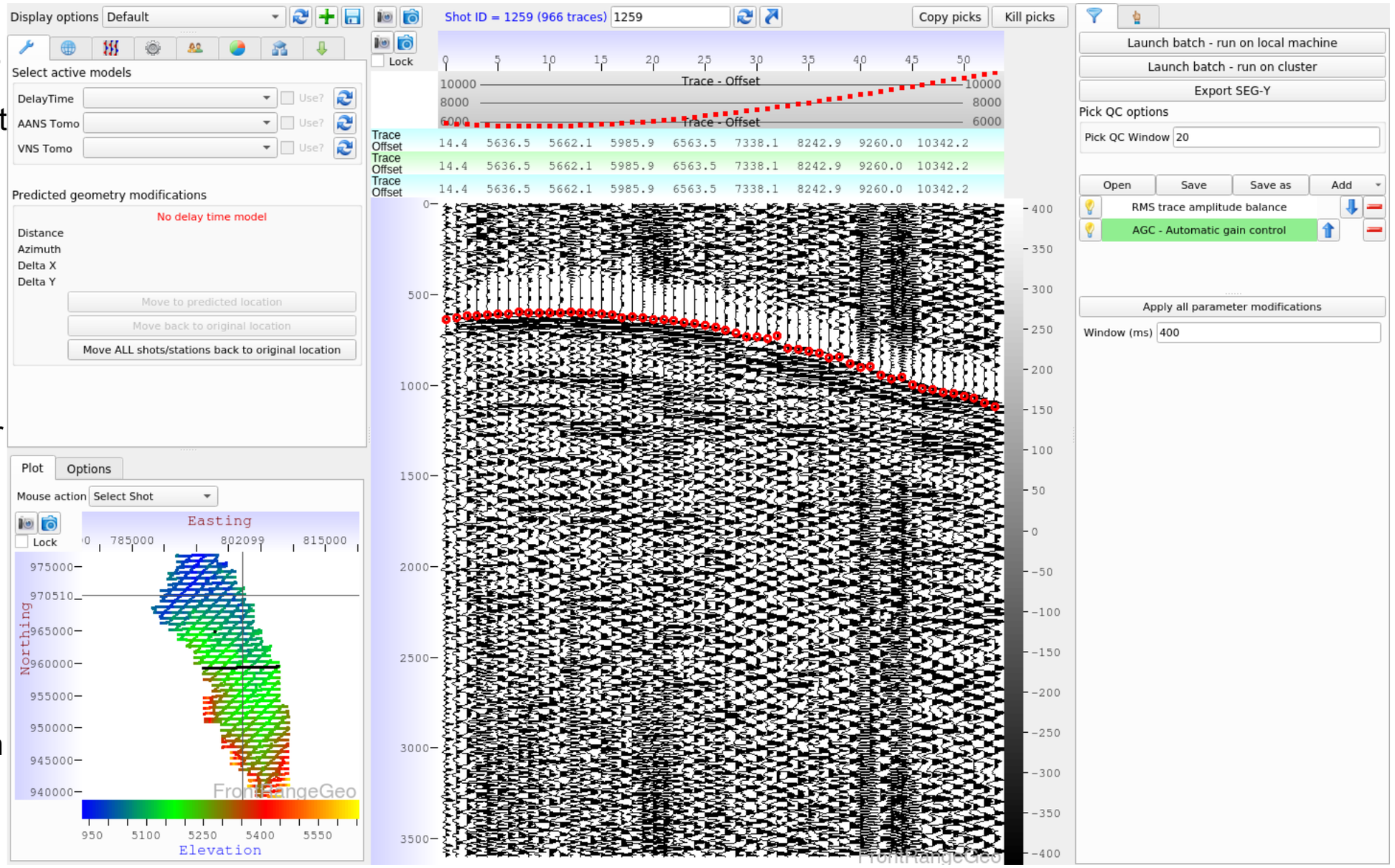
Currently we have selected “RMS trace amplitude balance” and “AGC”.

Processing flow steps are applied in the order they appear. The order can be changed using the arrow buttons.

Processing flow steps can be added by clicking on the “Add” tab and selecting a process.

The light bulbs at the sides of each step can be turned on and off to test how that processing step affects the seismic without removing it from the Processing Flow window.

To completely remove a step from the flow, click the red line button for that step.



Picking

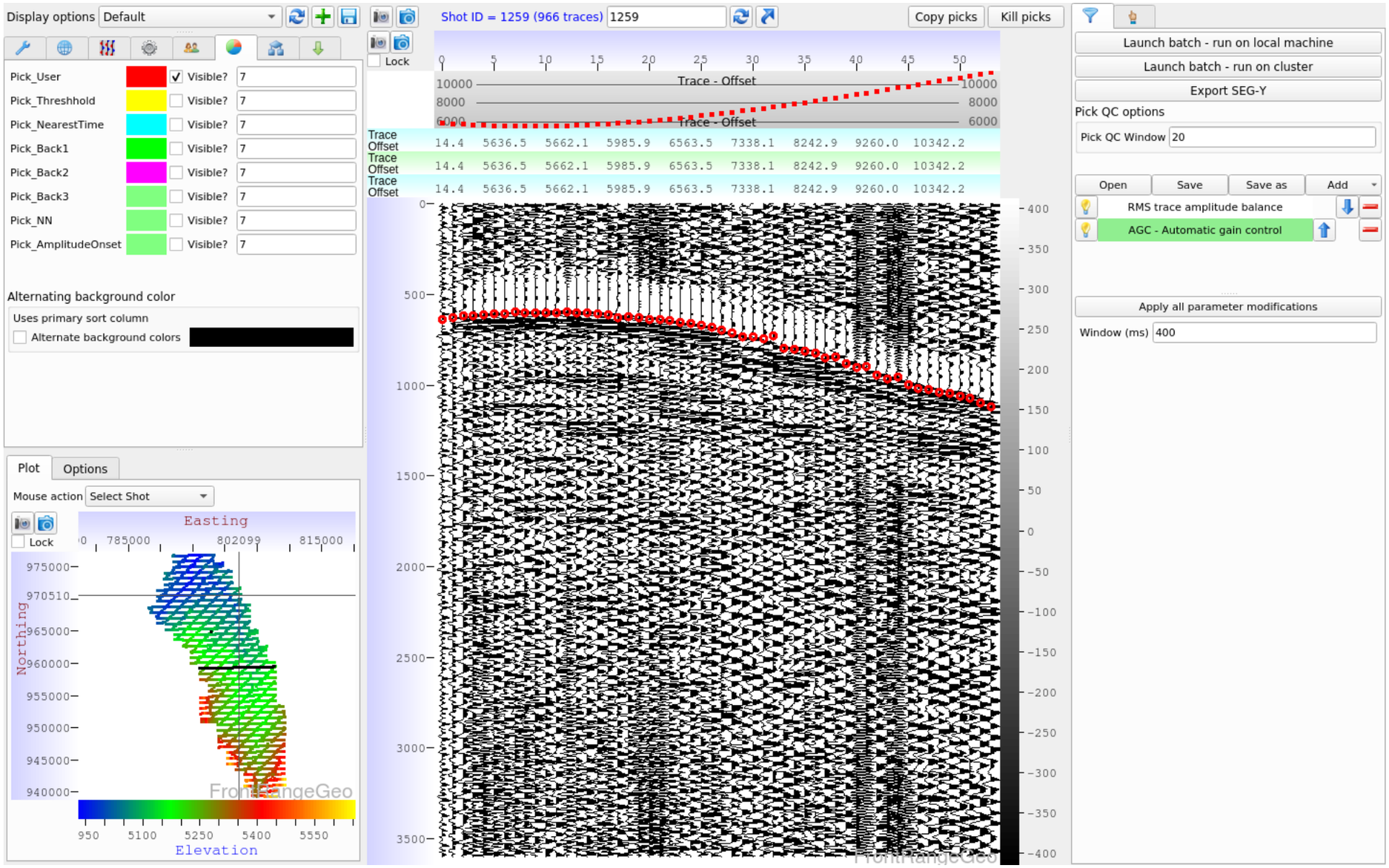
Select the color wheel tab from the top right.

These are the various pick sets which exist in your Merge project.

We imported picks with the SEG-Y file, so they are in Pick_User.

Check the various Pick “Visible?” boxes to make them appear on the seismic data.

The number to the right of the Pick columns scales the size of the pick circles.



Picking

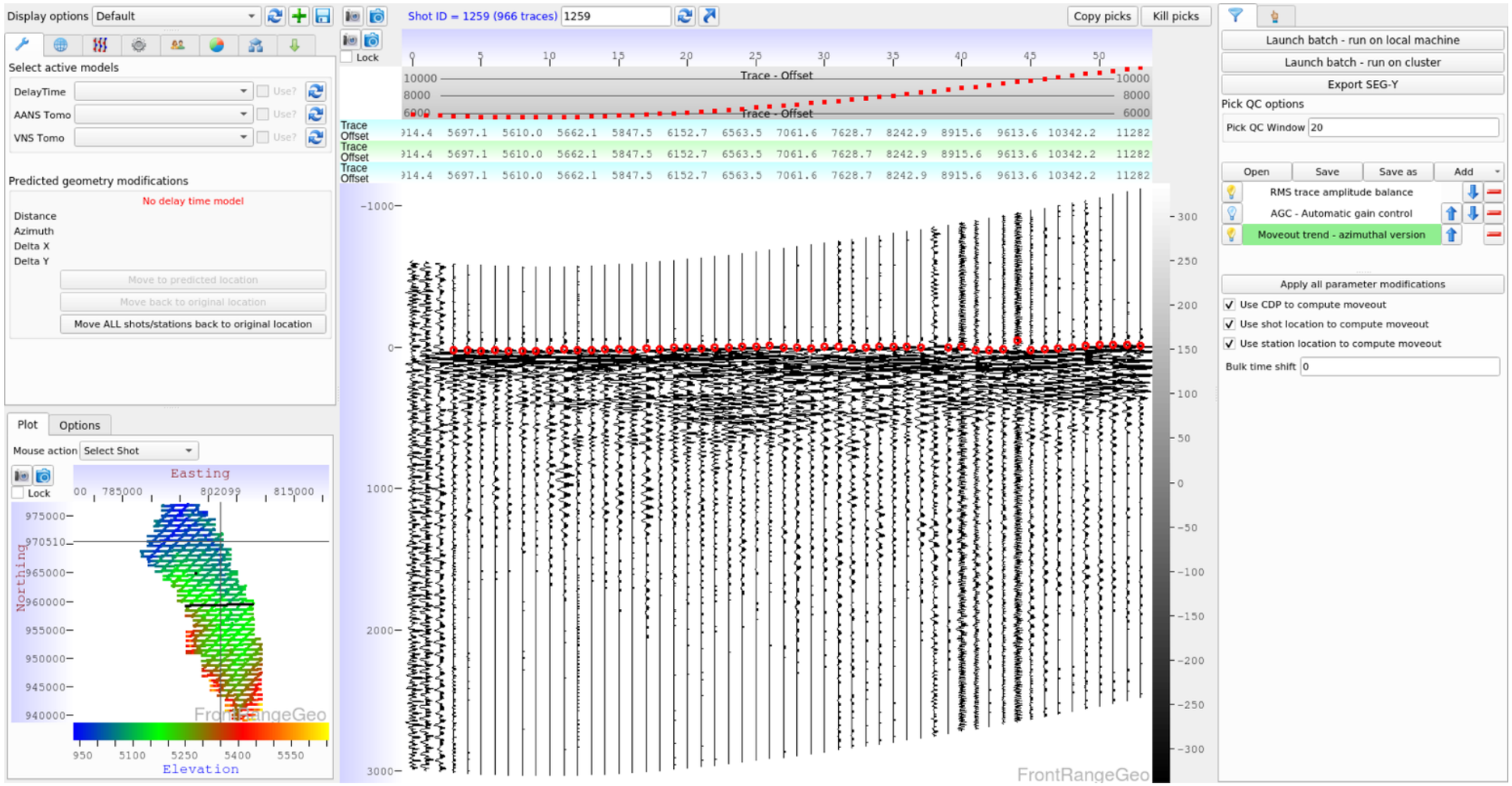
Now let’s use the processing flow.

In the processing flow, select “Add”, then “Moveout”, then “Moveout trend - azimuthal version”.

This will apply the move out trend we picked earlier to the seismic data.

Now we can try to pick the data with various methods.

One could at this point choose “Add->Picking-> Pick event nearest time”, to pick the sample which has been moved-out to zero time by the move out trend.



Picking

(If you do not have DeepTrace installed and running on a computer, you can skip this stage)

Let’s pick with DeepTrace.

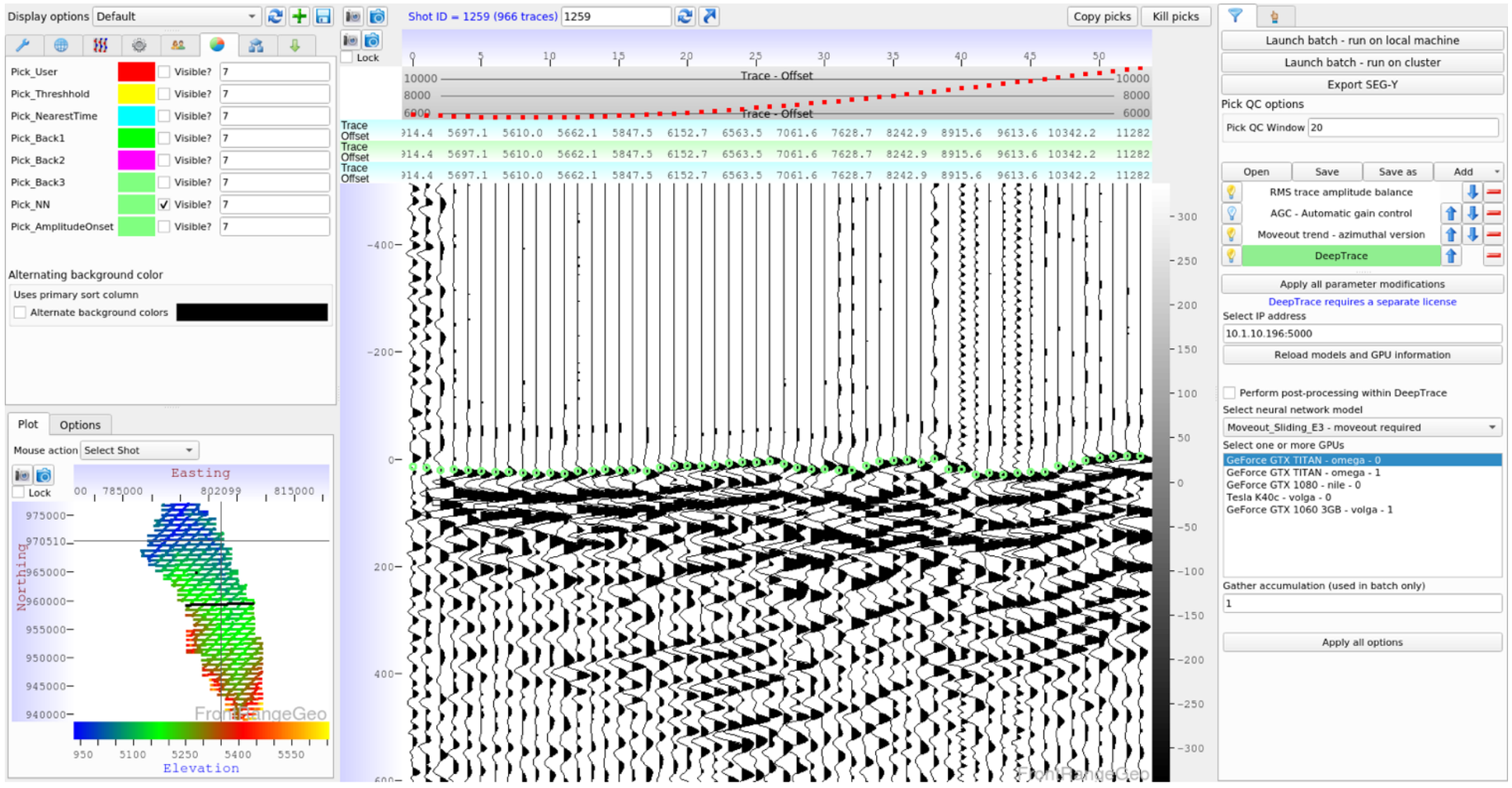
Add to your processing flow Add->Picking->DeepTrace.

Input the IP address of the computer running DeepTrace, and click “Reload models and GPU information”.

Select which DeepTrace model you would like to use from the Drop-down menu, and then select which GPU(s) to send the request to.

Click “Apply all Options” and turn the DeepTrace lightbulb on.

DeepTrace picks are put into Pick_NN by default, so make sure you have that visible from the pick tab.



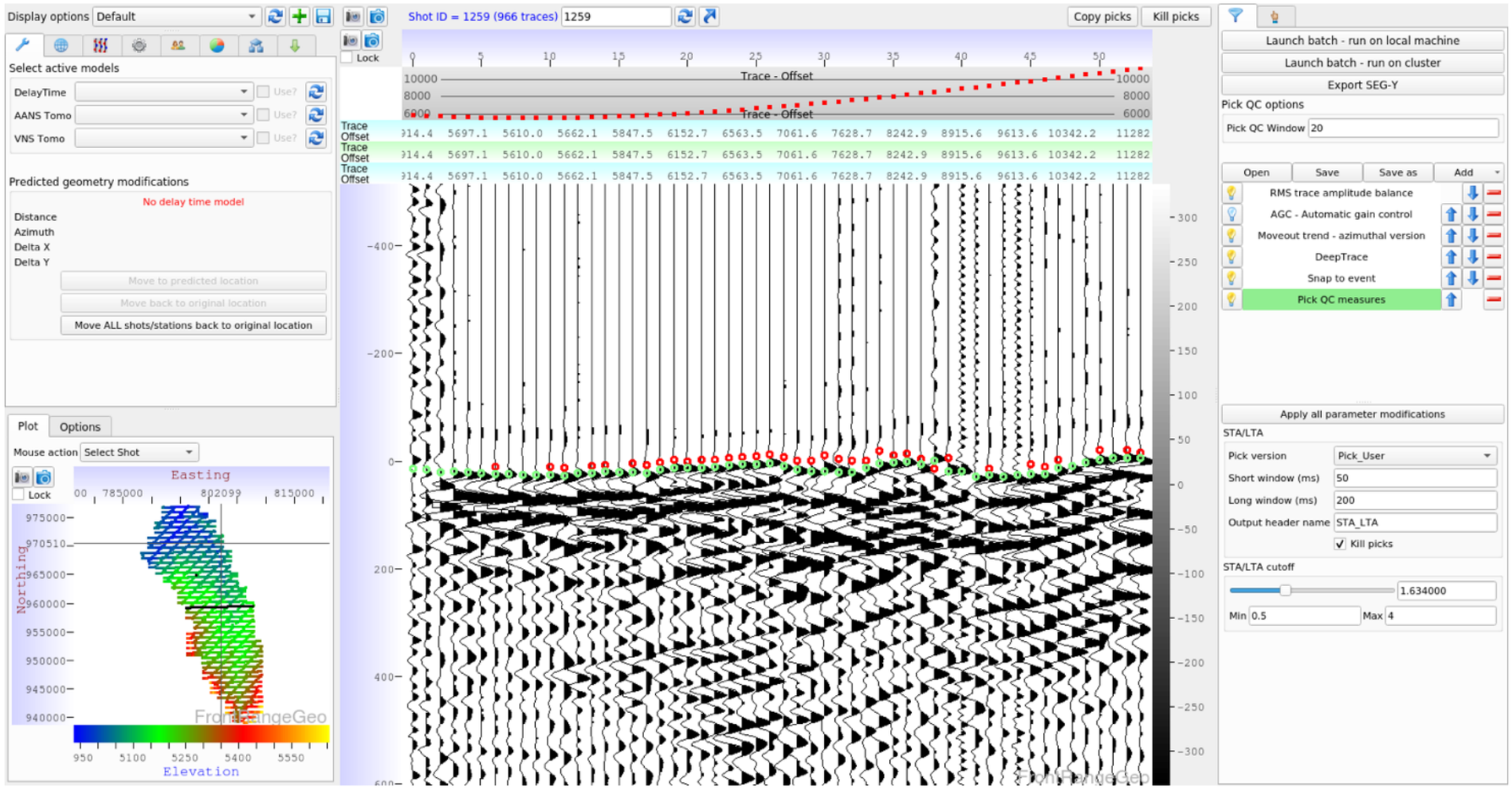
Picking

Next, we'll add some more processing steps to modify our picks.

After DeepTrace, (or Pick event nearest time if you are not using DeepTrace), add the "Snap to event" and "Pick QC measures" processing steps.

After selecting "Snap to event" in the processing flow, change its options to, "Source pick name = Pick_NN", "Output pick name = Pick_User" and "Event = Trough".

This will take the picks in Pick_NN, snap them to trough, and put them in Pick_User.



Picking

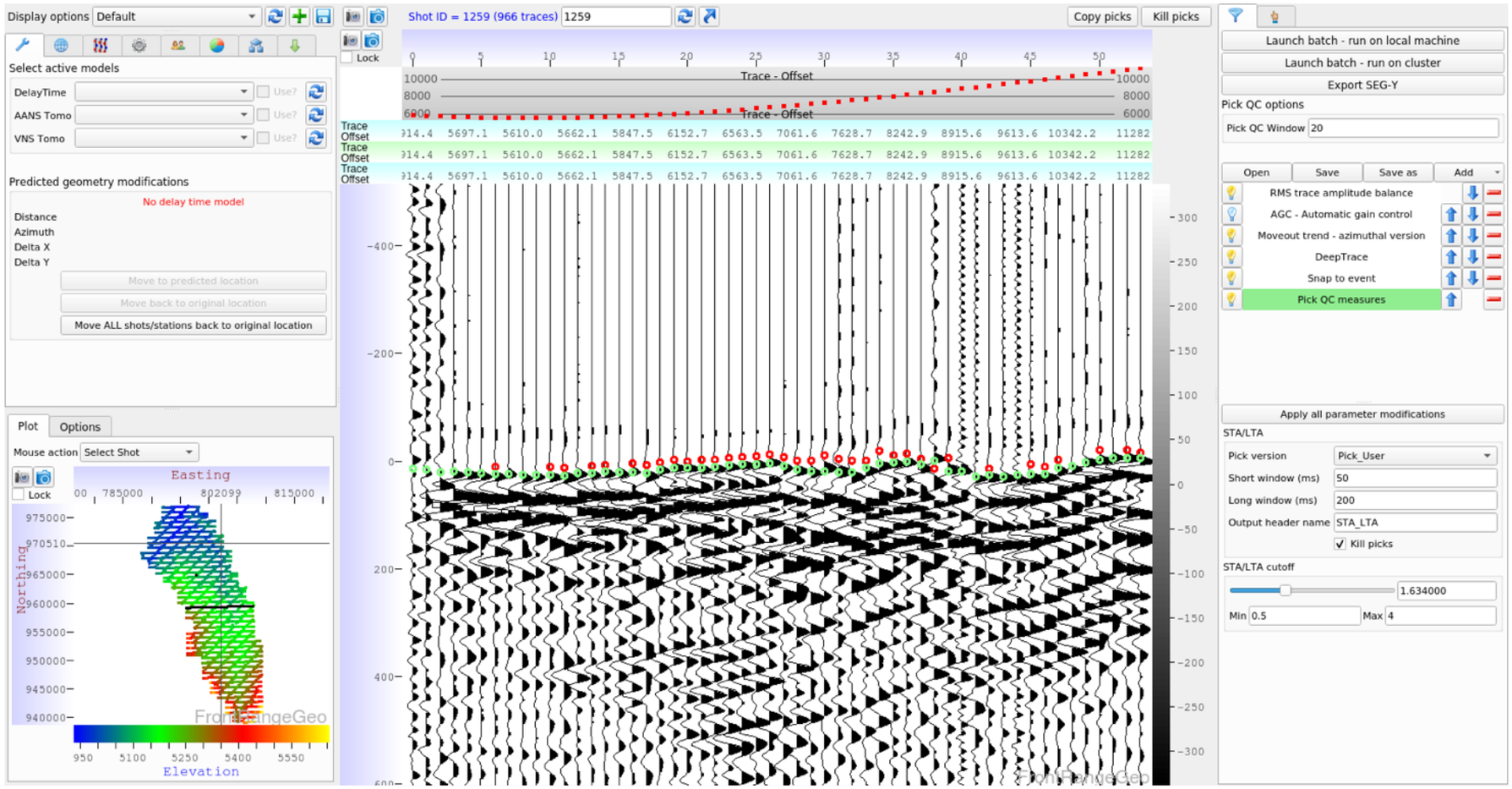
“Pick QC measures” lets us calculate a signal to noise ratio around our pick (using the STA/LTA method), and kill it if the ratio does not meet a minimum threshold.

Select the Pick QC measures processing step and modify its options as in the image.

Pick version = Pick User.

Next, play around with the STA/LTA cutoff. You will see that picks in areas with a low signal/noise ratio get killed.

This is an example of how the processing flow can be used to pick seismic data.

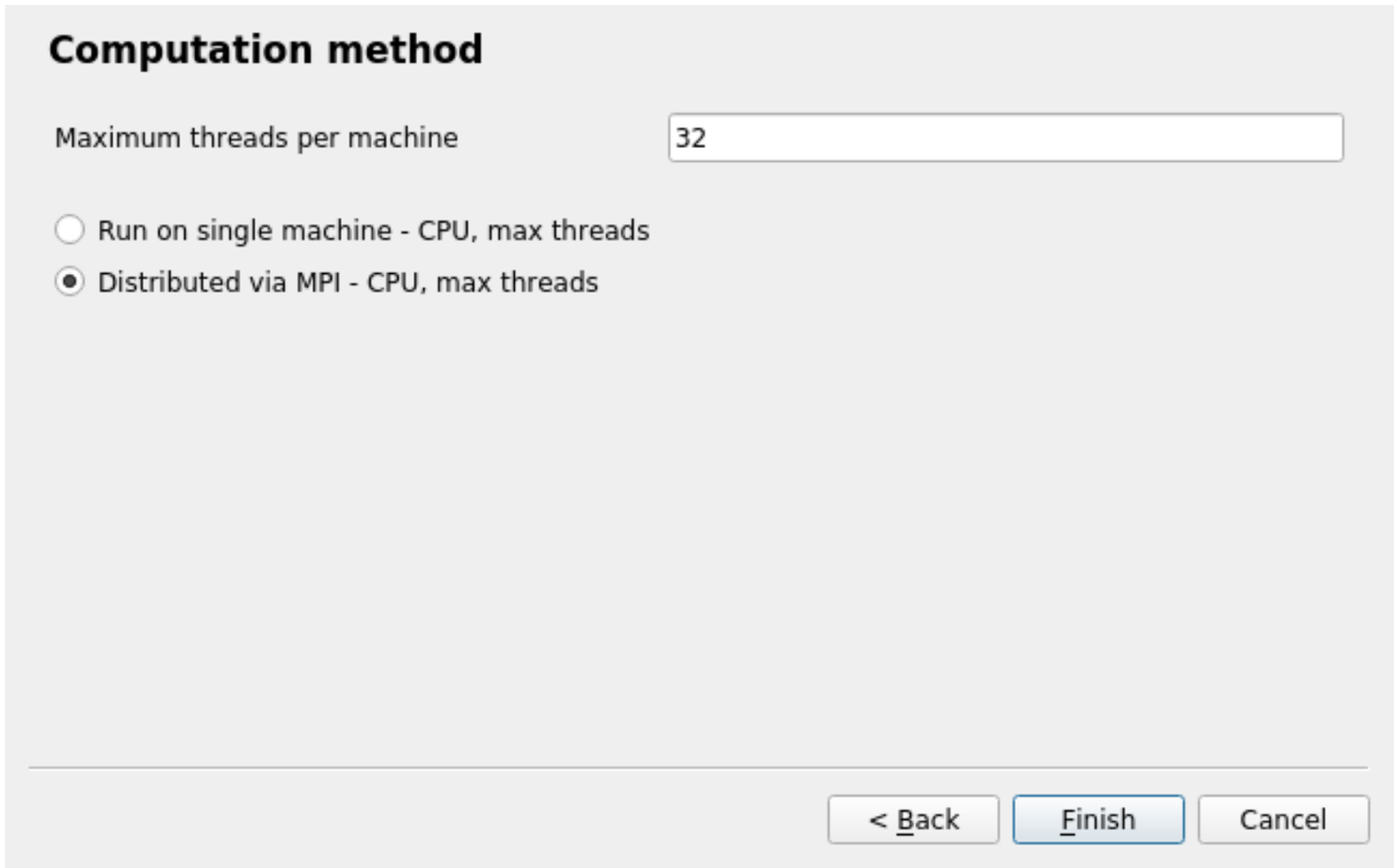


Picking

The picks shown on-screen are not saved until a batch job is run.

To run the processing flow on the entire Merge, select “Launch batch” at the top of the processing flow.

If you’ve configured Phoenix to run on a distributed cluster, select the “run on cluster” option. To simply run on the current machine, select “run on local machine”.

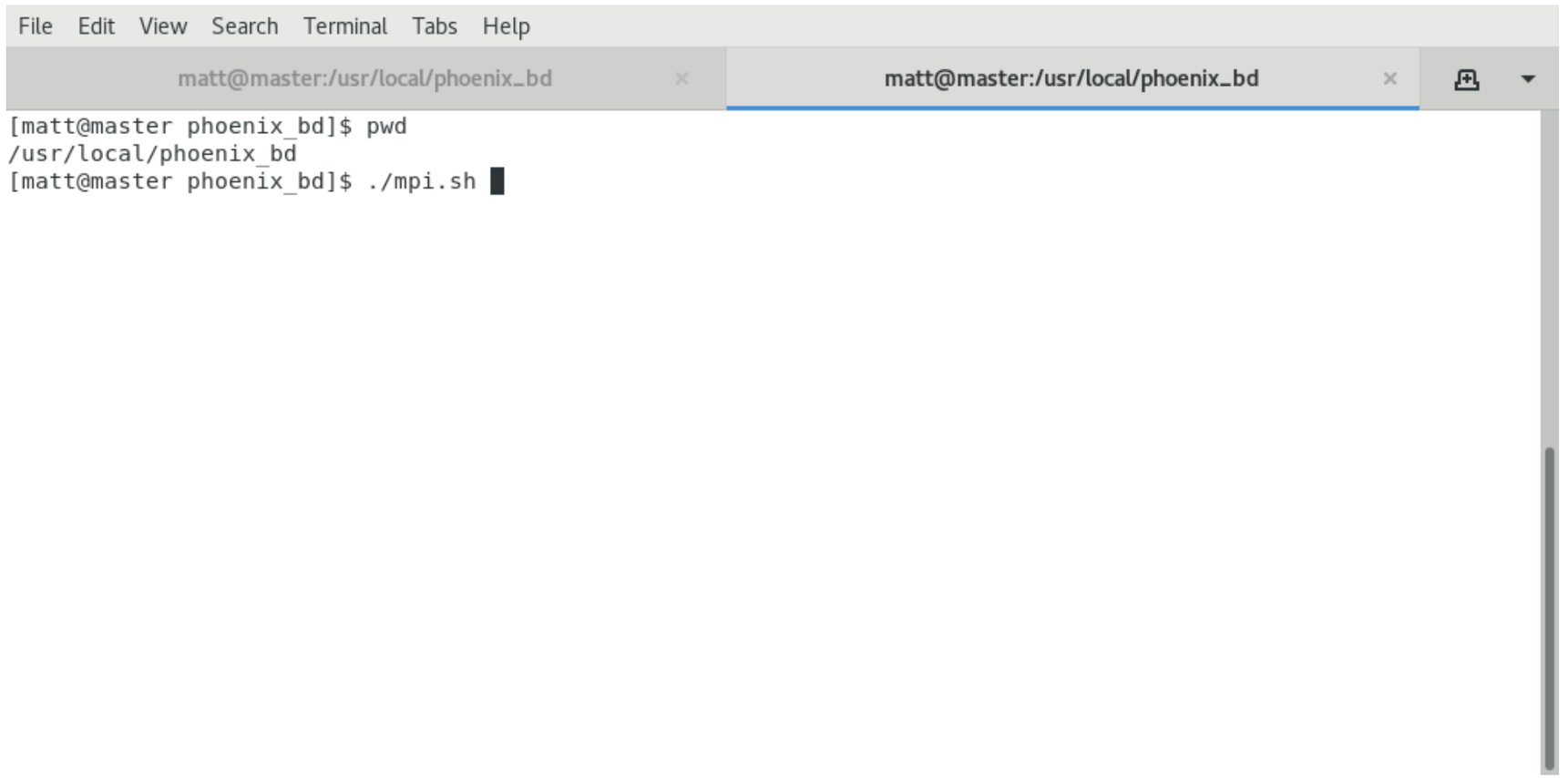


Picking

If you select ‘run on local machine’, you’ll just need to decide how many threads of your machine to give Phoenix, and what rate to decimate picks at (1= no decimation, 2= every other pick decimated, etc…). After you’ve made your selections, Phoenix will start the batch processing.

If you’ve selected the ‘run on cluster’ options, Phoenix will write out to the mpi.sh file in your phoenix directory.

Navigate to that directory and run the batch job on the cluster by executing it as in the image.



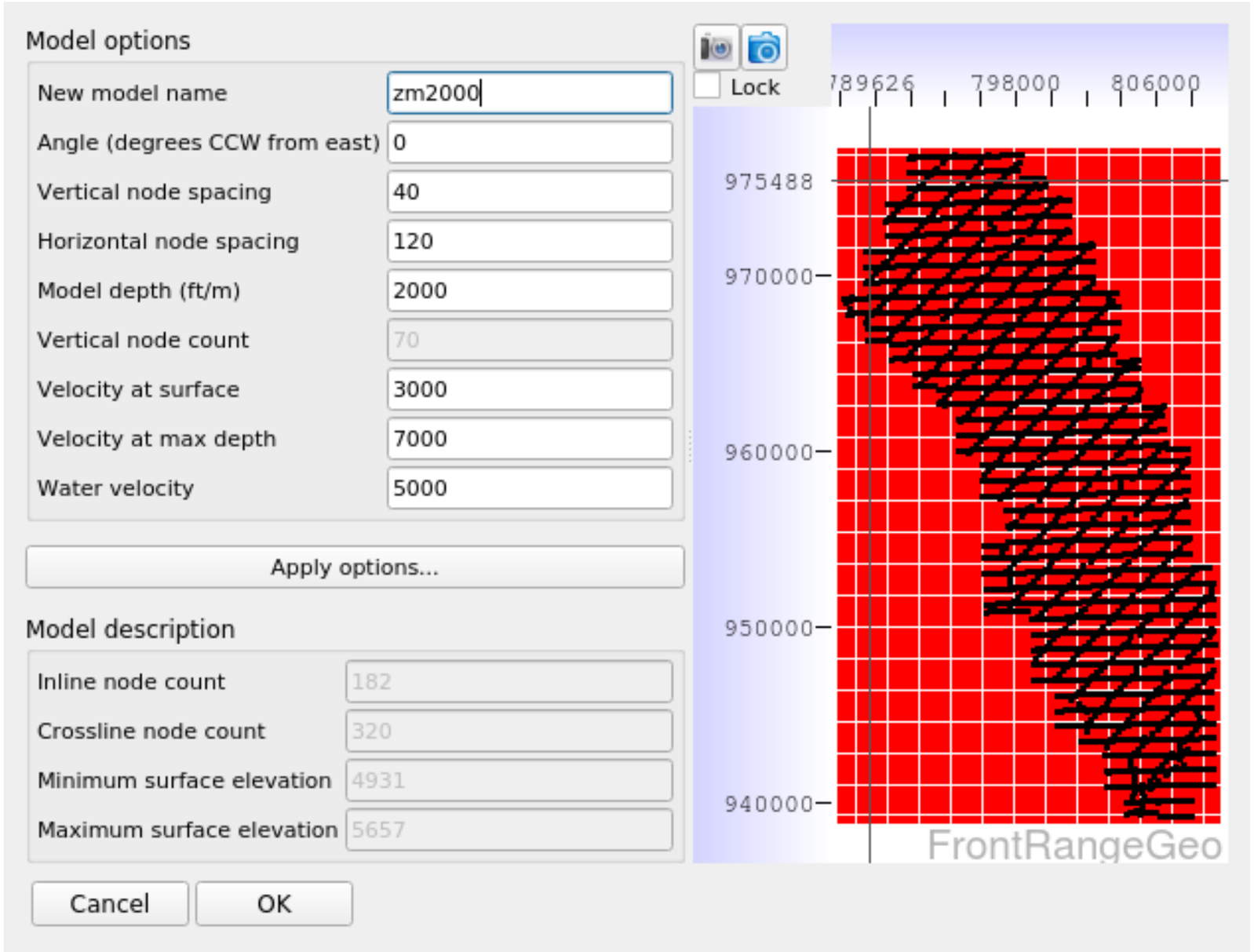
Modeling

Now that we have some picks, let's create a tomo model with the new ANS algorithm.

Navigate back to the Merge Manager window and select the AANS tab from the bottom left. Click "New".

Give your ANS model a name and adjust the options to your liking. (The default spacing and velocity values should be changed to about 1/2 to 1/3rd of their default if your distance units are in meters.)

After you are satisfied with the model parameters click "OK".

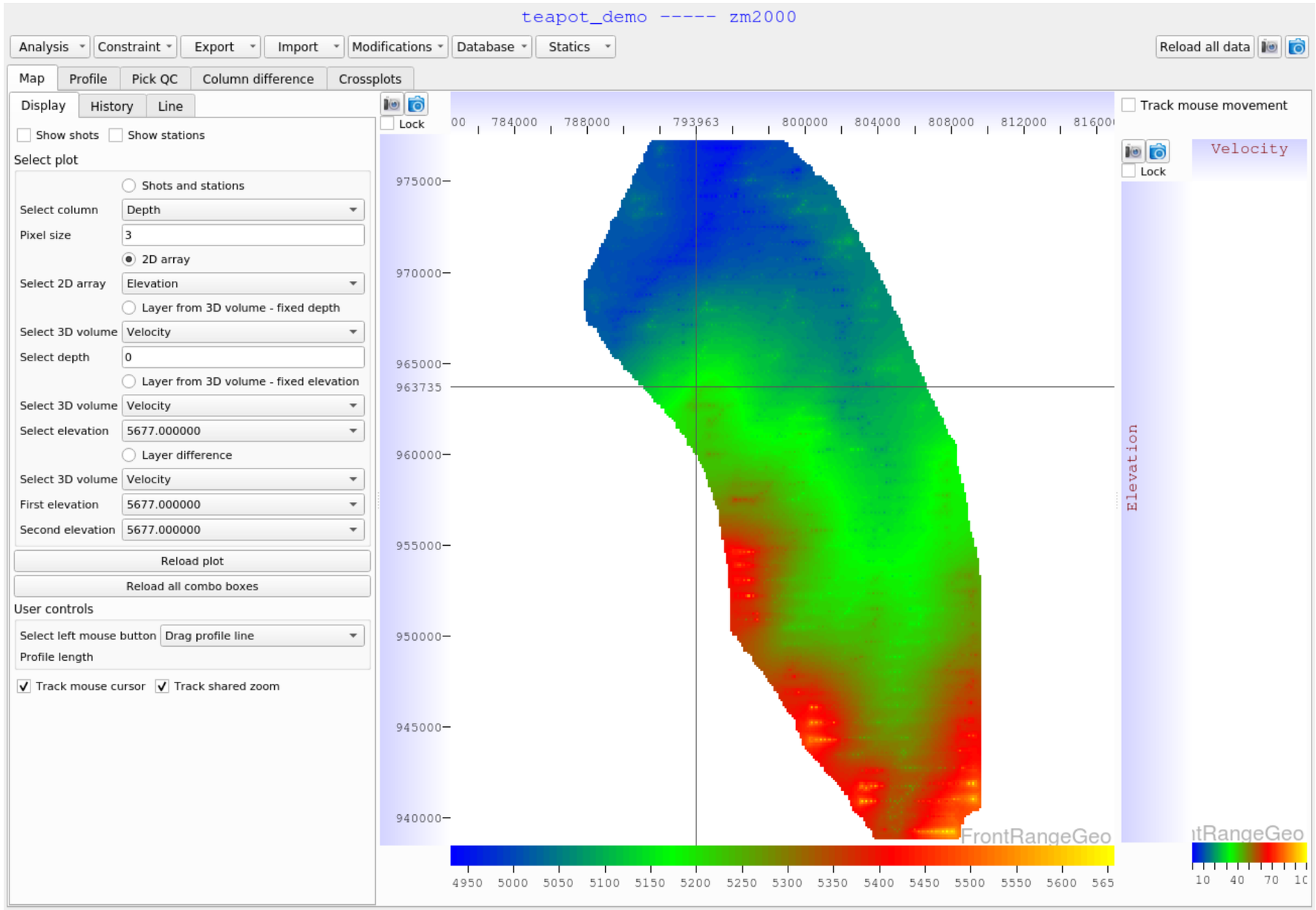


Modeling

The model will take a moment to instantiate. Then you can select it from the AANS tab and click Open.

The model window will appear.

The model starts off with some initial guessed velocities, but needs to be updated. Click “Analysis” in the top left and “Batch job: Launch model update”.



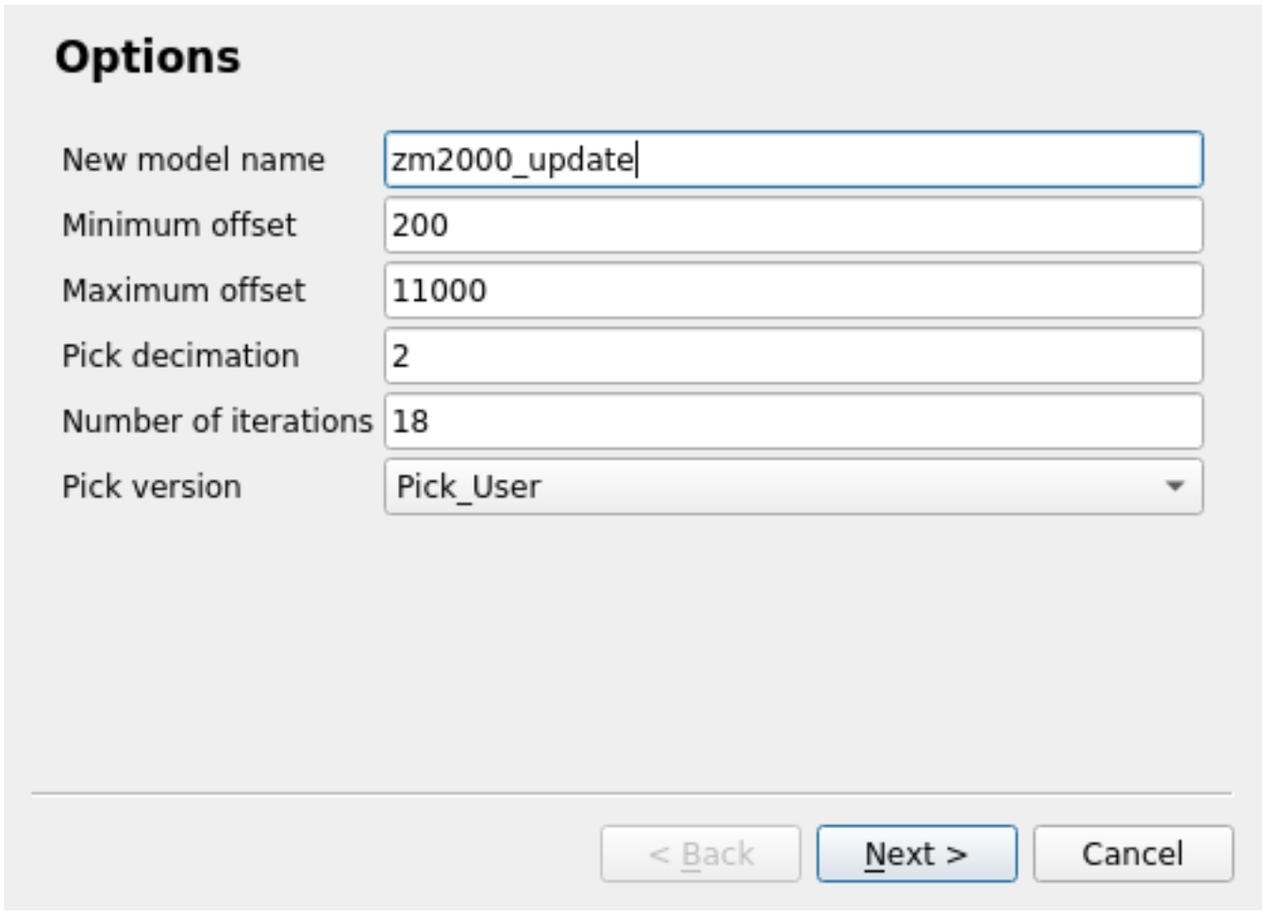
Modeling

Give your updated model a name.

Select the minimum and maximum offset from which picks should be used for modeling. (We typically do not trust picks at the very nearest and farthest offsets.)

These options are fairly self-explanatory.

Click next, and for now, leave all the values on their default settings.



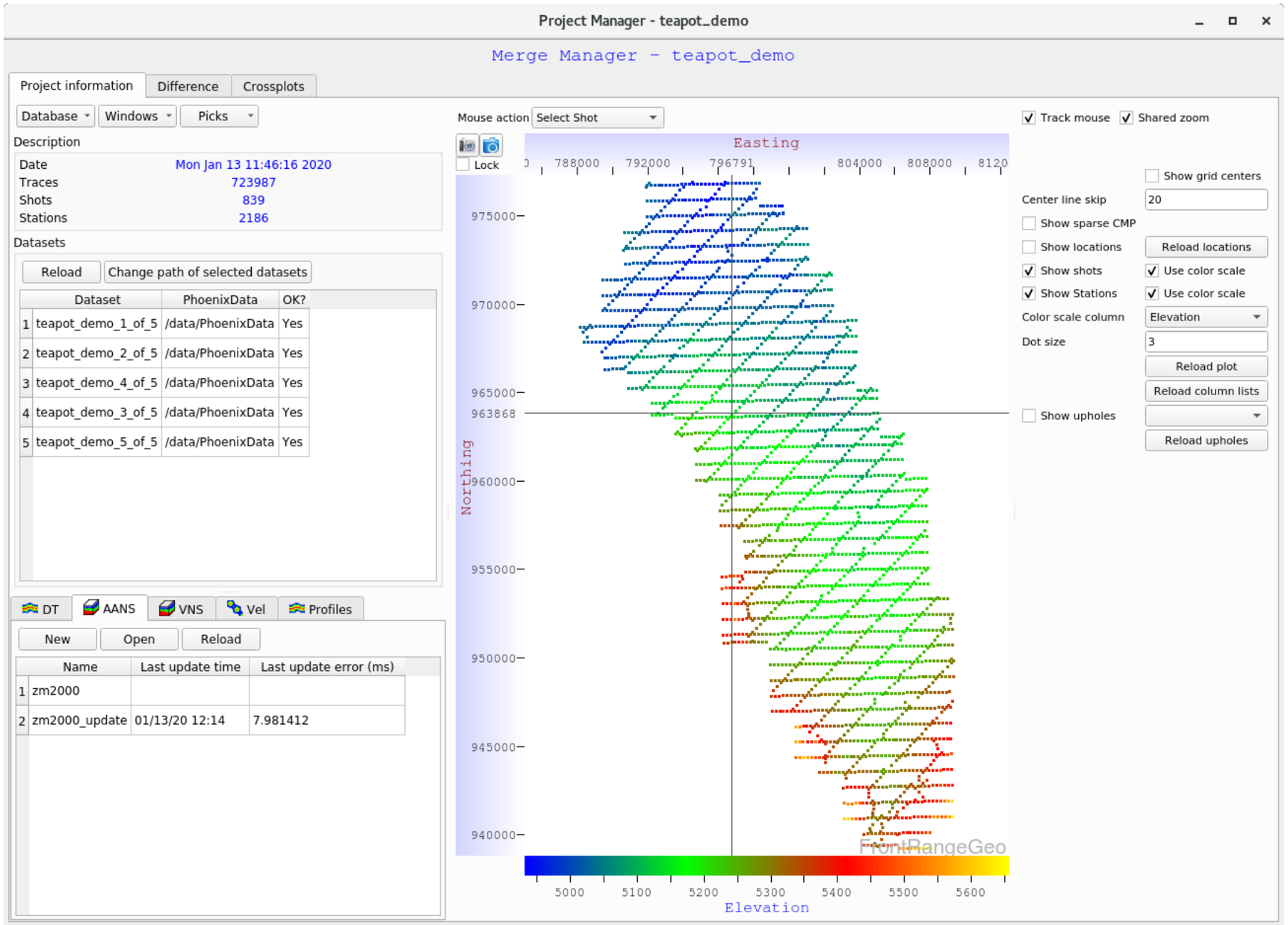
Modeling

Now the model is being updated.

We can monitor its progress and latest error in the Merge Manager window.

After a few iterations, our model reached an error of 7.9ms. Let's have a look at it. This is the average absolute difference between the picked value and the simulated ray travel time in the model.

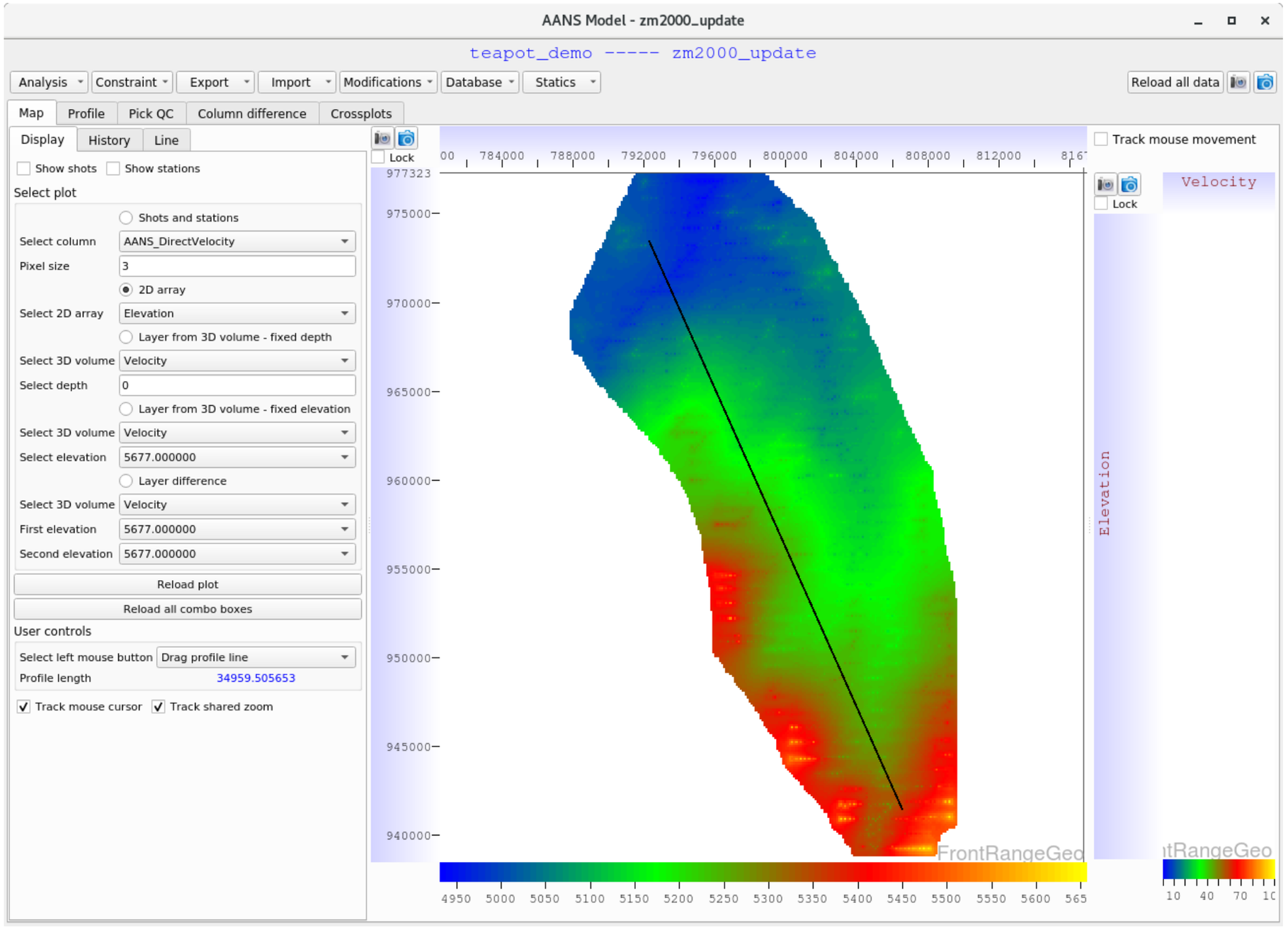
Open the model.



Modeling

In the updated model window, drag out a line across the map. We will look at a profile along this line by navigating to the “Profile” tab next to the “Map” tab.

You can view the model update history in the “History” tab. It’s useful to see how quickly the model converged during the update.



Modeling

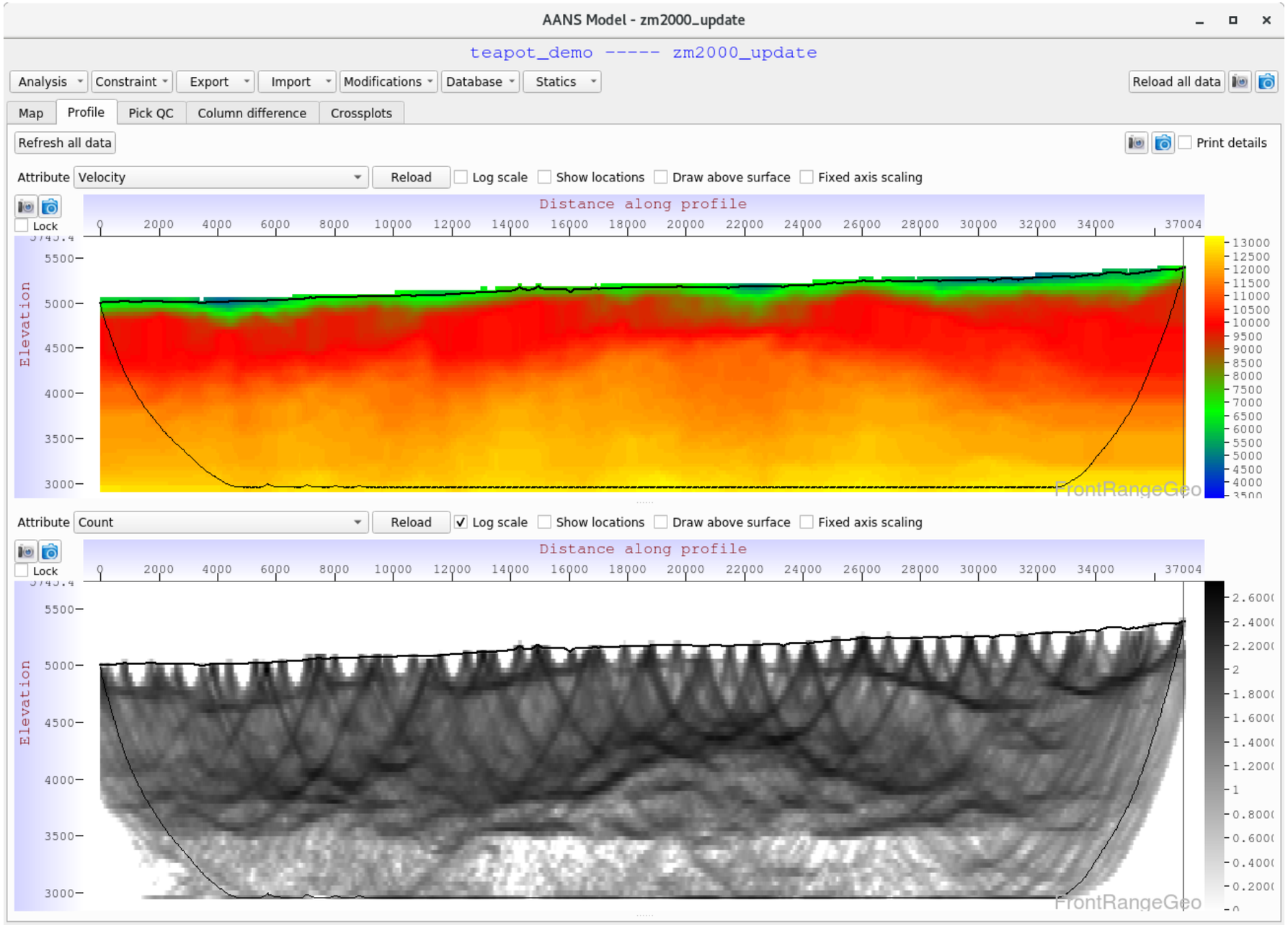
This profile slice corresponds with the line we dragged on the map.

You can look at three different profiles at once. For now, let's drag the third one down to the bottom, and change the second one to "Count".

Check "Log Scale".

Right-click on the scale on the right side, and select "White Black".

This plot shows the node hit count during the update. Nodes are hit when a simulated ray path travels through them. By examining the node hit count, we get an idea of where the rays are spending time. These plots can be very useful for visualizing refractor boundaries.



Modeling

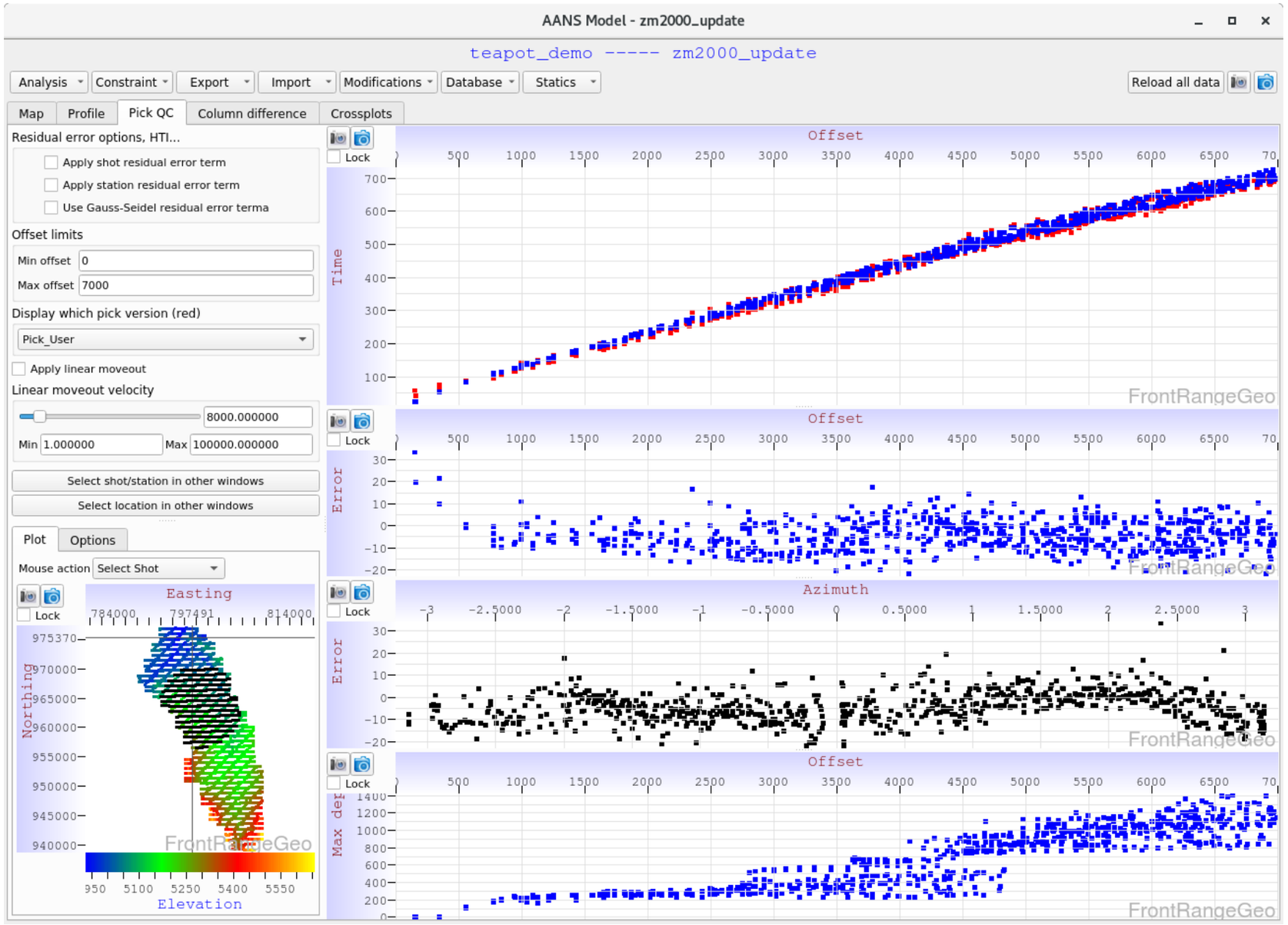
Navigate to the Pick QC tab of the model window.

Click an area on the base map.

These plots show how picks from the model agree with the user-supplied picks. The top plot shows an Offset vs. Time graph of the user picks (red) and the model picks (blue).

The middle two plots show pick error (model-user pick) as a function of offset and azimuth.

The bottom plot shows the maximum depth achieved by simulated rays through the model as a function of offset. This plot is useful to tell how deep we need to model to see various refractors.



Re-Picking

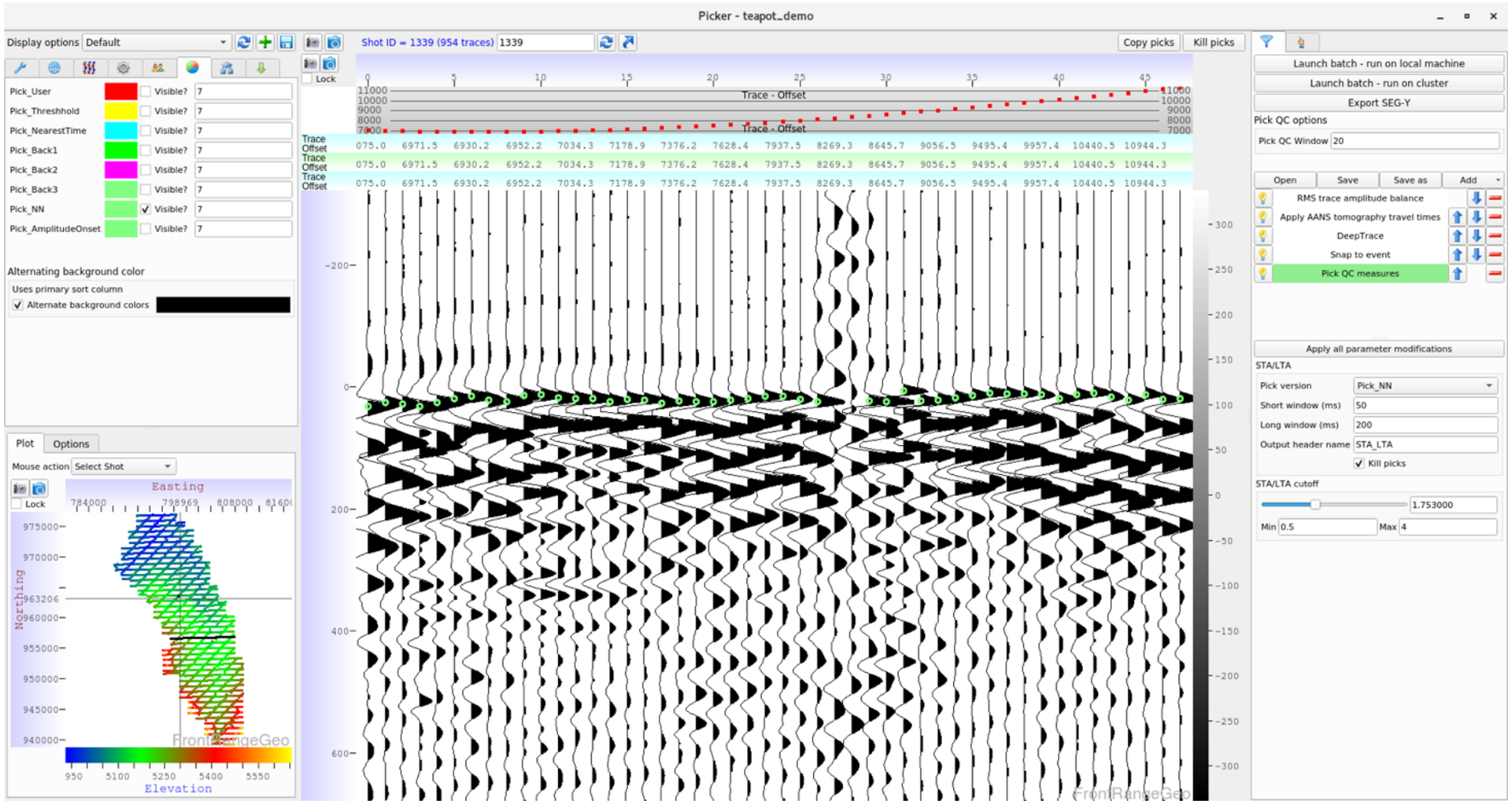
Now that we have an ANS tomography model, we can use it to modify our picks.

Head back to the picker window and replace the move out trend with “Add -> Moveout -> Apply AANS tomography travel times”.

Make sure you’ve selected your updated model under the wrench tab and selected “Use?”.

Now the move out trend is replaced by simulated travel times from the model - much more accurate!

We can re-pick using this flow to get better picks.



This is a quick guide to get users going with their first project. Phoenix has a wide range of functionality beyond the scope of this guide, such as picking stacking velocities, creating statics and stacks, importing and comparing Well-log data with velocity models, and much more.

We encourage you to explore all the possibilities Phoenix has to offer now that you understand the basics.