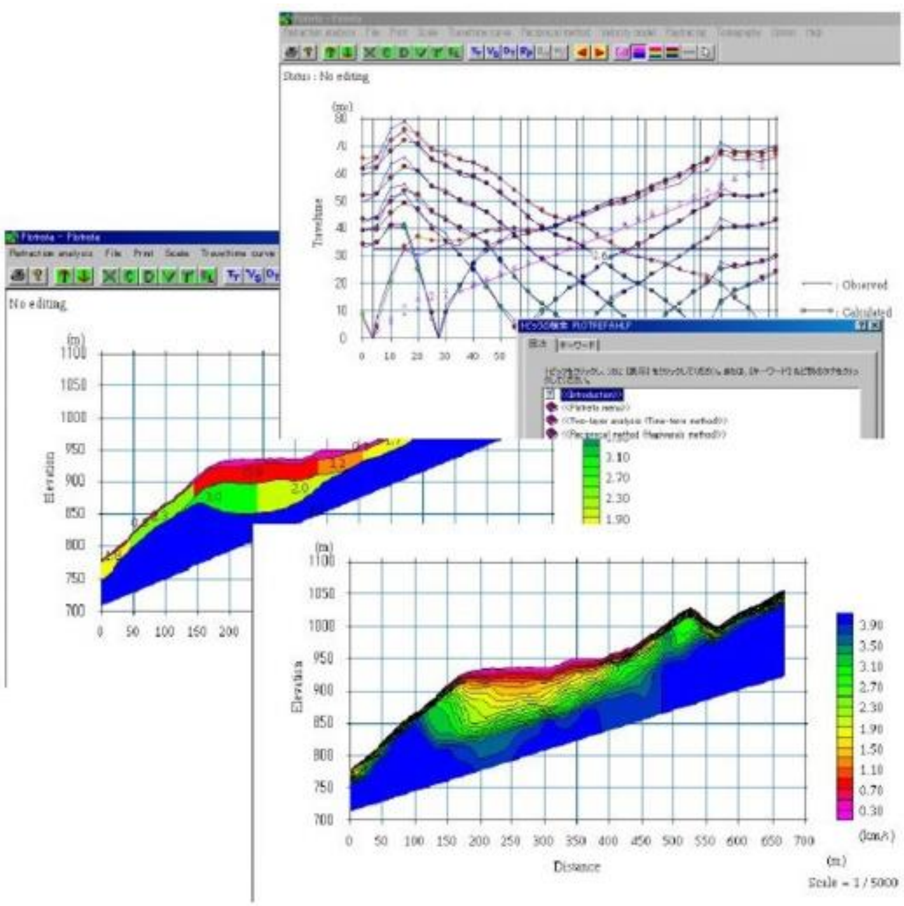
SEISIMAGER/2D
USER'S MANUAL

Software for the Analysis of
Seismic Refraction Data

Version 4.0
P/N 770-00117-01

July 2024



Craig Lippus / Deb Underwood

TABLE OF CONTENTS

APPENDICES

1 INTRODUCTION

Welcome to SeisImager/2D™! SeisImager/2D is an easy-to-use, yet powerful program that allows you to:

- Read in and display your refraction data.
- Control how your data is displayed.
- Make changes/corrections to your data files and save them.
- Pick the first breaks and save them.
- Invert your data to a velocity section.
- Output a travel time plot, velocity section, and other graphics.

SeisImager™ is the master program that consists of seven modules for refraction, downhole, and surface-wave data analysis. The individual modules are Pickwin™, Plotrefa™, WaveEq™, PSLog™, SPACPlus™, and GeoPlot™. The Surface-wave Analysis Wizard™ is not a separate module, but automatically calls on specific functions from Pickwin, WaveEq, and GeoPlot. The overall structure of SeisImager is shown below:

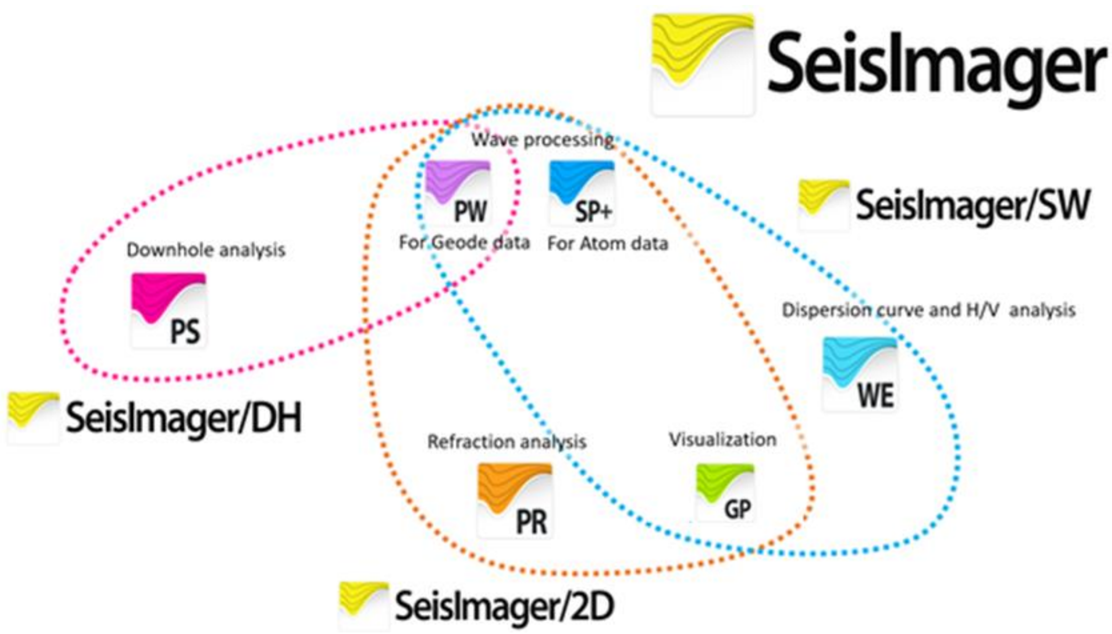


Figure 1: SeisImager family of applications.

Pickwin and Plotrefa are the modules used for refraction analysis, making up the program called SeisImager/2D. GeoPlot can be used to visualize the data beyond the tools included in SeisImager/2D. Pickwin is the first break picking module and Plotrefa is the main analysis program. Though we touch on some refraction theory in this manual (see appendices), this is not meant to be a treatise on seismic refraction. It is assumed that the user has a reasonable grasp of the main principles of seismic refraction, especially those behind the specific analysis techniques employed by this software. Please see the recommended reading list (Appendix G) for some good primers on seismic refraction theory and inversion techniques. See Page 9 and Page 124 for links to useful video clips. Also see this treatise on seismic refraction field methods.

SeisImager/2D is a very powerful refraction package. It offers three separate inversion techniques: the Time-term Method, the Reciprocal Method, and tomography. Both the Time-term and Reciprocal Methods are based on "delay-times" (see Appendix E for a discussion of this all-important concept). The main difference between the two is the method by which the delay-times are calculated. In the Time-term Method, the delay-times are calculated automatically (via a linear least-squares inversion technique). In the Reciprocal Time method, the delay-times are calculated manually – much more user input is required. Each technique is unique, and which technique you should use depends on the goals of the survey and the character of the data. SeisImager/2D also contains many ancillary tools that we hope you will find useful.

Section 2 describes the software installation process. Section 3 describes the process of picking first breaks with Pickwin. Section 4 describes in detail the various inversion techniques available in Plotrefa. Appendix A and Appendix B provide a number of Pickwin and Plotrefa examples, and Appendix C gives an overview of seismic refraction theory. Appendix D, Appendix E, and Appendix F describe some of the particulars of the three inversion algorithms. Finally, Appendix G provides a list of references for further reading on seismic refraction. A separate booklet of examples, *SeisImager/2D Examples*, along with example data files and video clips, is available for download on our web site. Please visit our site often for manual updates and free updates of the software.

Although this manual can be printed, **it was designed as an online resource, and includes many internal and external hyperlinks.** It will be updated on a semi-regular basis, and a current version will always be available for download on our site. Be sure to display the navigation toolbar in Acrobat Reader (as of this writing, the toggle switch was F8) to simplify navigation:

Figure 2: Acrobat Reader navigation toolbar.

If your version of Acrobat Reader does not have the above tool bar, use *Alt+Left Arrow* to return to the previous view after clicking on a link.

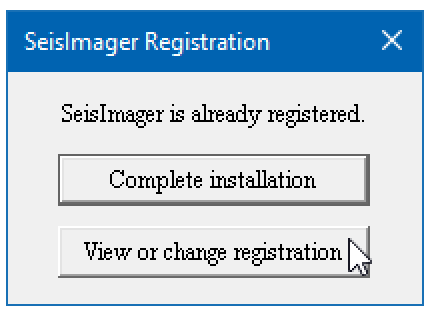
The manual makes liberal use of color, so if you elect to print it, using color is highly recommended. There are also links to online videos, so an internet connection is useful (some of these videos are also available here and can be stored on your hard drive for offline viewing).

Finally, we are very interested in your constructive criticism of both this manual and the software itself. Please contact us at sales@geometrics.com with any comments you might have.

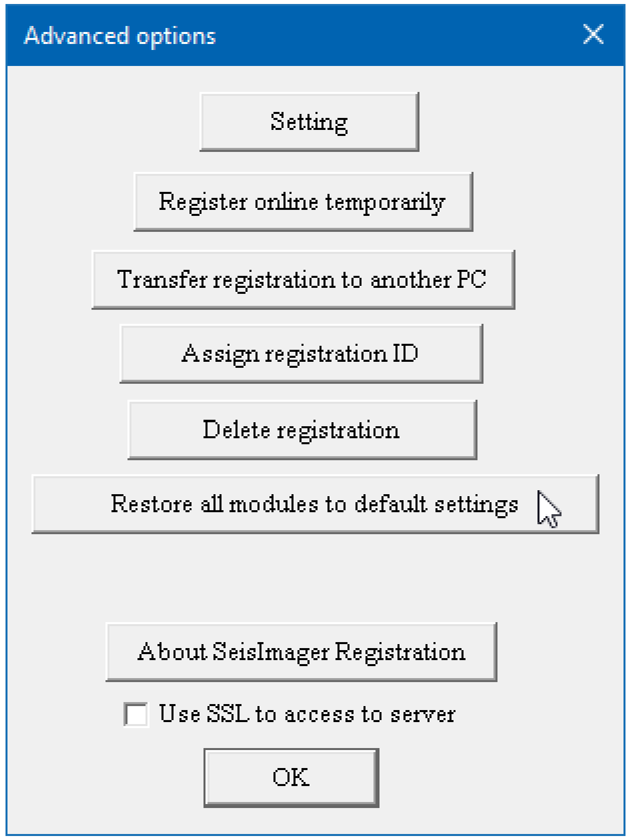
__Note:__ SeisImager/2D is very complex software "under the hood" and may grow unstable and give spurious results if many different models are run in one session or if there are unit conflicts between modules. It is therefore best to begin a new instance of the application to run new models. If the program does exhibit instabilities, follow this procedure:

- Close all SeisImager modules.
- In the Windows search box,

Type in "SeisImager Registration." You will see the following:



- Select *View or change registration*. You will be presented with the **SeisImager Registration** dialog box.
- Select *Advanced options* (upper right) and then press *Restore all modules to default settings* and then press *OK*.



Note: *Throughout this manual, you will find that certain menu items are greyed out. There are two reasons for this. The most common is that the feature is not applicable to that dataset or that point in time. The other reason is that some items may not be available under your license.*

2 INSTALLING THE SOFTWARE

The SeisImager USB stick is supplied (1) for trial evaluation of the programs, (2) for purchase, rental, or upgrade of one or more of the programs, or (3) with purchase of an ES-3000™, Geode™, StrataVisor NZXP™, or Atom™ seismograph, which all include the Lite version of SeisImager/2D. The USB contains all programs and all documentation.

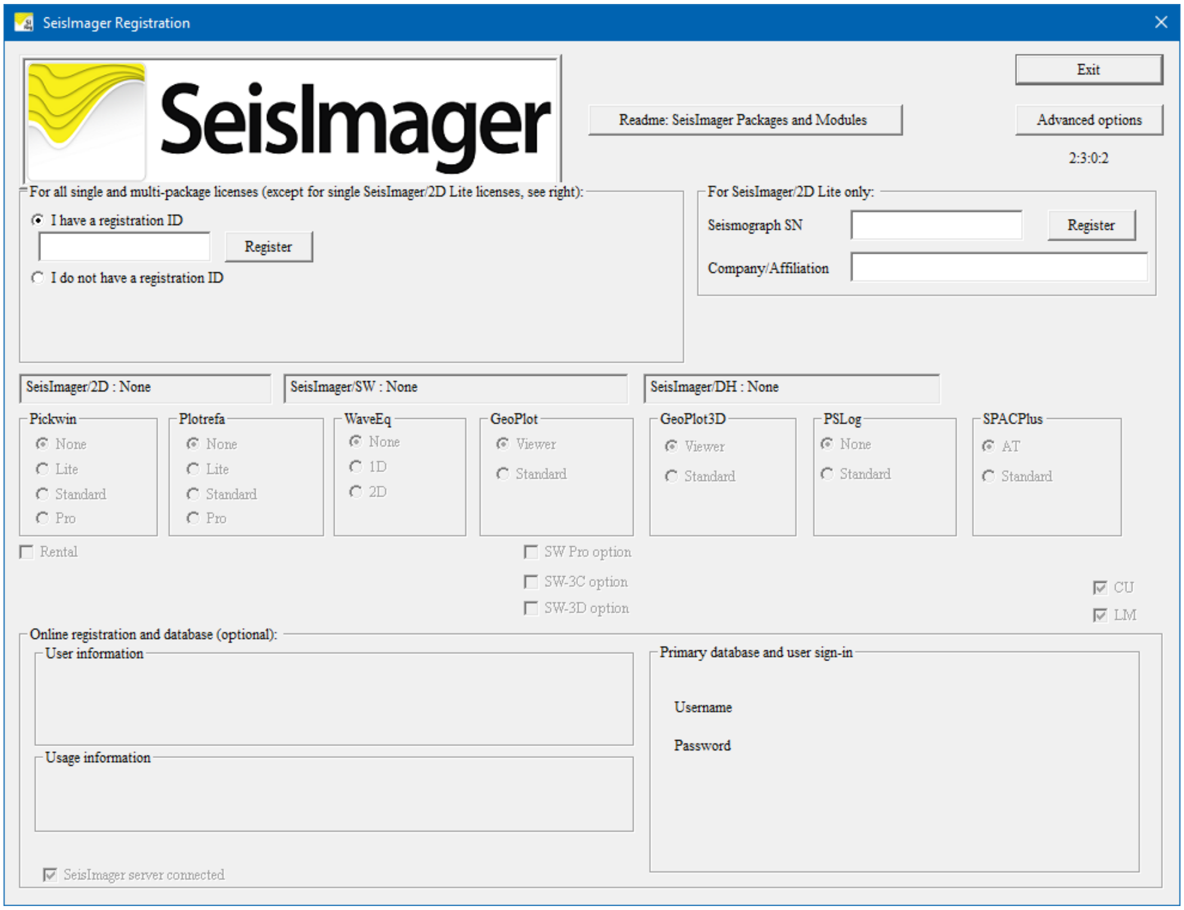
Occasionally, there will be a software release in between USB releases. In this situation, the USB will be labeled with a notice to download the latest version.

Note: *Even if the USB is not labeled with instructions to do so, it is best practice to download and install the latest software prior to installation, as SeisImager is updated frequently. The USB is convenient, especially if you do not have an internet connection. However, if you* ***do*** *have an internet connection, we highly recommend that you skip the USB for installation altogether. If you do so, be sure to download the newest documentation as well.*

Note: *You must have administrator rights to install the software. After installation by an administrator, users with lower-level privileges can use the software.*

To install or update the software, click on the file named **SeisImager.msi** (or SeisImager_xxxx.msi). If SeisImager is already installed on your computer, you will be prompted to remove it or repair it. Remove the software, run SeisImager.msi again, and then simply follow the prompts.

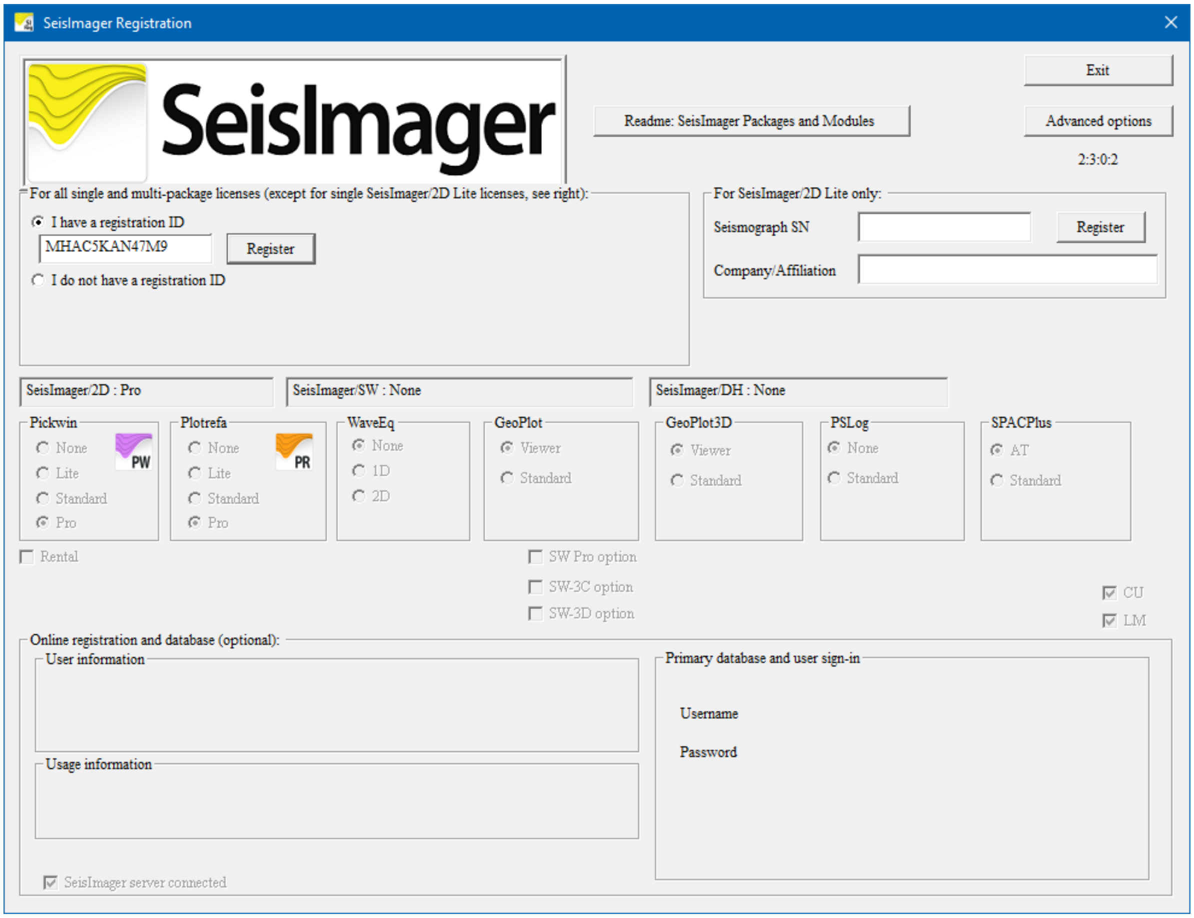
After the installation is complete, you will be presented with the registration screen:



If you already have a registration ID, indicate as such, type it in, and press *Register*. If you do not have an ID, *click I do not have a registration ID*, and send your keyword and order number or seismograph serial number to support@geometrics.com. You will be given a registration ID that will enable the products that you purchased or rented. You may return to this screen later by typing “SeisImager Registration” into the Windows search box at the lower left of your desktop:

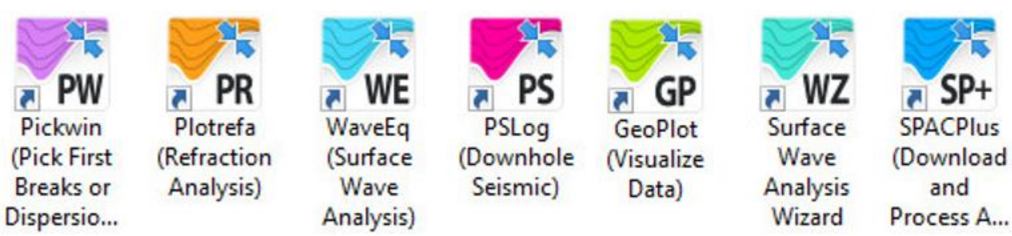
The programs enabled by the registration ID will be reported in a series of messages. Click *OK* to accept each message.

After these messages have appeared, the registration screen will reflect the programs that have been registered, as shown below. In this case, Pickwin Pro™ and Plotrefa Pro™ are the programs that have been registered.



Typically, installing an upgrade of the software does not require re-registration, but if you are upgrading from a version older than April 2007, you will need to re-register.

Once installed, the program modules can be opened directly through the desktop icons shown below:

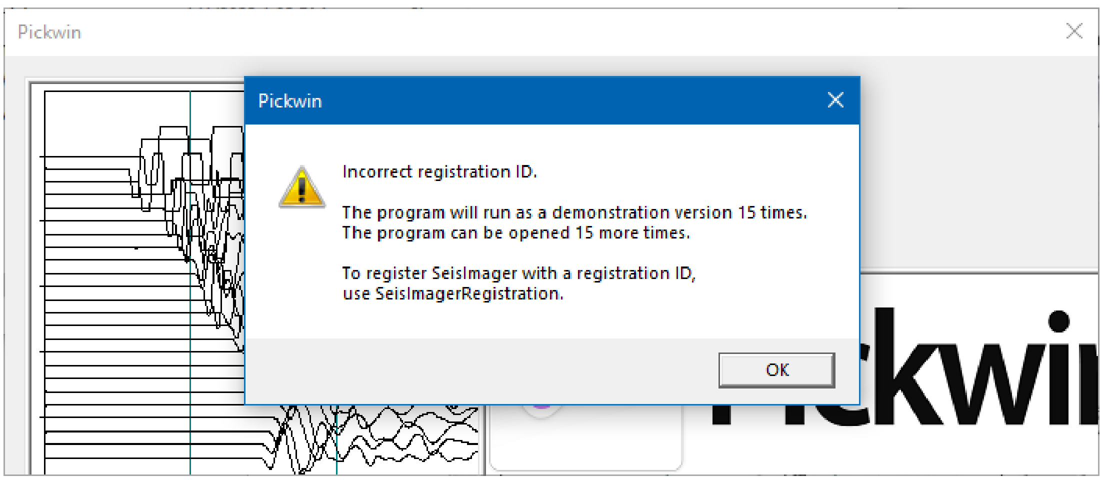


The Surface-wave Analysis Wizard is not a separate module but automatically calls on specific functions from Pickwin, WaveEq, and GeoPlot to walk you through the analysis process. <u>All of the icons (shortcuts) will be copied to your desktop regardless of which program(s) has been purchased.</u> You may wish to create a folder for the various shortcuts to avoid cluttering on your desktop. Alternatively, you may elect to simply delete the shortcuts that you did not purchase/rent the rights to.

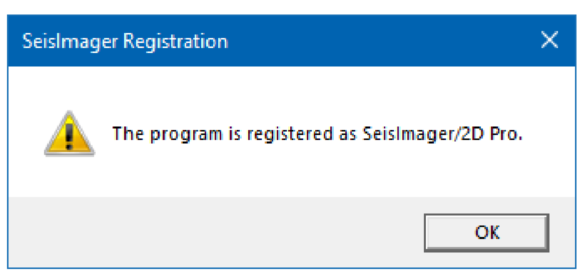
To begin using the software, double-click the appropriate shortcut.

For registered installations, the module opens and is ready for use. The other registered modules are ready for use as well.

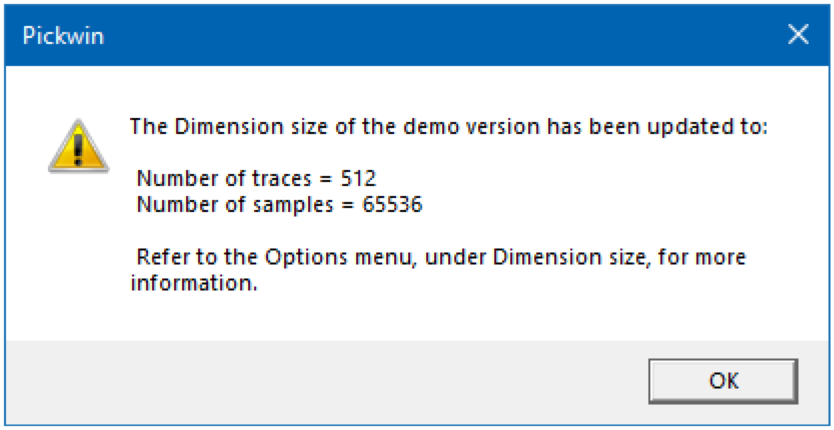
For unregistered installations running in demonstration mode, you will be presented with the message shown below. Press *OK*.



To enter a registration ID after your 15 times in demonstration mode, go to the Windows search box (lower left) and type "SeisImager" to find the SeisImager Registration  program as shown above. Open the register and email the keyword to support@geometrics.com with your order number and seismograph serial number (if you purchased the software with a seismograph), and we will reply with a registration ID to enable the version of the software you have purchased. Once received, enter the registration ID press *OK*. You will see a message like the following:



Once the software is registered, the data input dimensions of the demonstration version will be updated to reflect the limits of the program purchased. You will see a message like the one below. Press *OK*.



This completes the description of most, if not all, possible registration pathways.

As mentioned previously, the Lite version of SeisImager/2D comes free with all seismograph purchases, so if you have purchased SeisImager/S2D with a seismograph, you are also entitled to the Lite version of SeisImager/2D. If you do not already have a license for SeisImager/2D, Lite or otherwise, but would like to order a copy, please contact us at support@geometrics.com.

A general recommendation when using SeisImager is to close and reopen the software modules or open a second instance of the software modules to start new, separate analyses. The programs are efficient and launch quickly, so this is easy to do and will prevent complications when processing data.

Regarding making report graphics and documenting your data processing, SeisImager includes the ability to print graphics to a printer or pdf, as well as save images to PNG, JPG, BMP, or GIF format. You might also find it handy to have a screen capture program such as HyperSnap® from Hyperionics. Bitmap screen captures can be quickly and easily made at the desired stages of processing and saved for import into Microsoft Word® or other applications.

For more advanced graphics, consider Geometrics GeoPlot.

3 THE PICKWIN MODULE

Prior to reading this chapter, you might find it useful to view this 12-minute video on using the Pickwin module. It will provide valuable context for what follows.

If you are new to seismic refraction, it is highly recommended that you watch the following clips on data acquisition:-

Seismic refraction training 1-0: Geode Seismograph Field Setup (5:52)

Seismic refraction training 1-1: SCS Data Acquisition (7:41)

Seismic refraction training 1-2: SCS Data Acquisition (8:59)

Seismic refraction training 1-3: SCS Data Acquisition (6:10)

Although the above videos are based on the Geometrics Geode seismograph, they are applicable to all Geometrics seismographs and useful even with competing seismographs.

When doing seismic refraction, the main purpose of Pickwin is to help you identify your first breaks, pick them, and save them for input to the analysis program, Plotrefa. Once you have read your data in, (and edited it if necessary; see below), you may optimize the appearance of the data to enhance the appearance of the first breaks. Toward this end, you may filter the data, change the display gains, change the distance and time scales, change the trace style, and correct the record for timing errors. Once you have optimized the data, the program will automatically pick the first breaks at the touch of a button, which you will then have a chance to adjust. After the breaks have been picked, you may save them, read in the next SEG-2 file, and repeat until all files have been picked and the picks have been saved.

After reading in your data file, as mentioned above, you may edit it. For example, you may truncate it, de-sample it, or change the geometry information in the header. Once you have finished editing, you may save it in the same SEG-2 format. This is a useful feature for correcting any mistakes you may have made in the field. For instance, you may have used a much longer record length than you needed, resulting in very large files. This feature allows you to truncate the unnecessary part of the file and make the file smaller.

Note: *Never overwrite raw field data. Always save edited data with a different file name than that of the raw data.*

The general flow of Pickwin (for refraction) is depicted in the flow chart below:

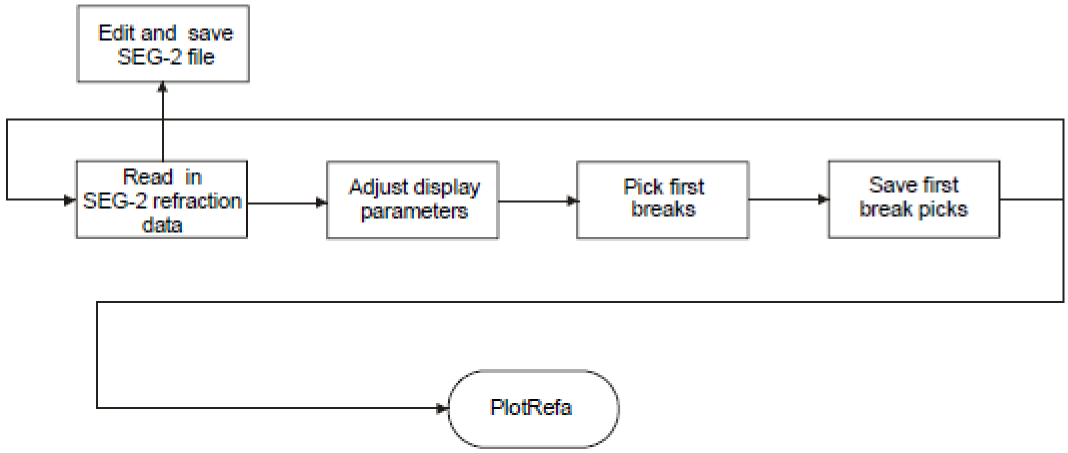
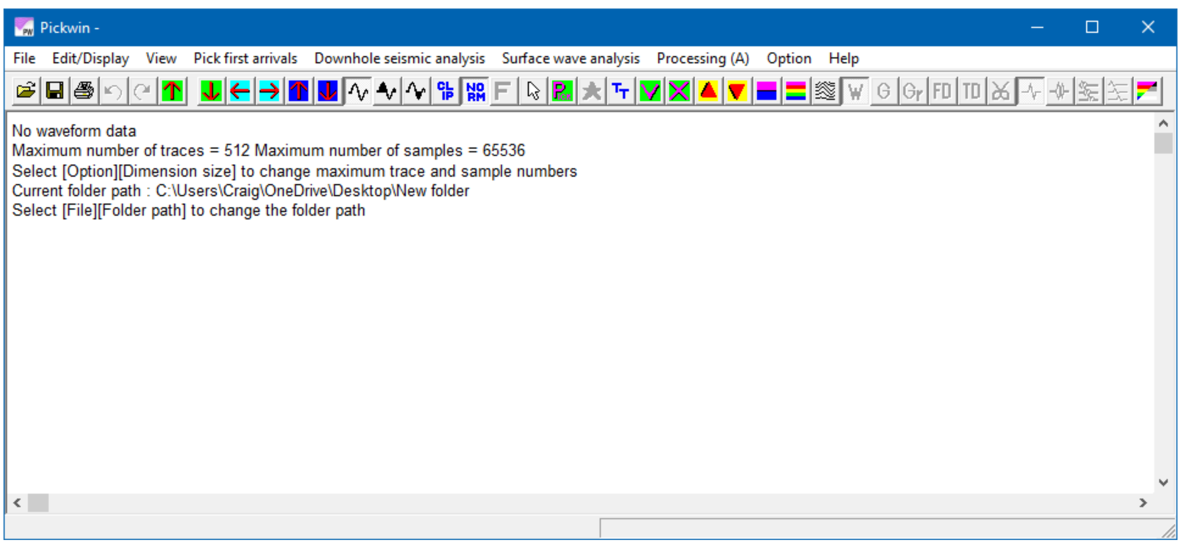


Figure 3: Pickwin flow chart.

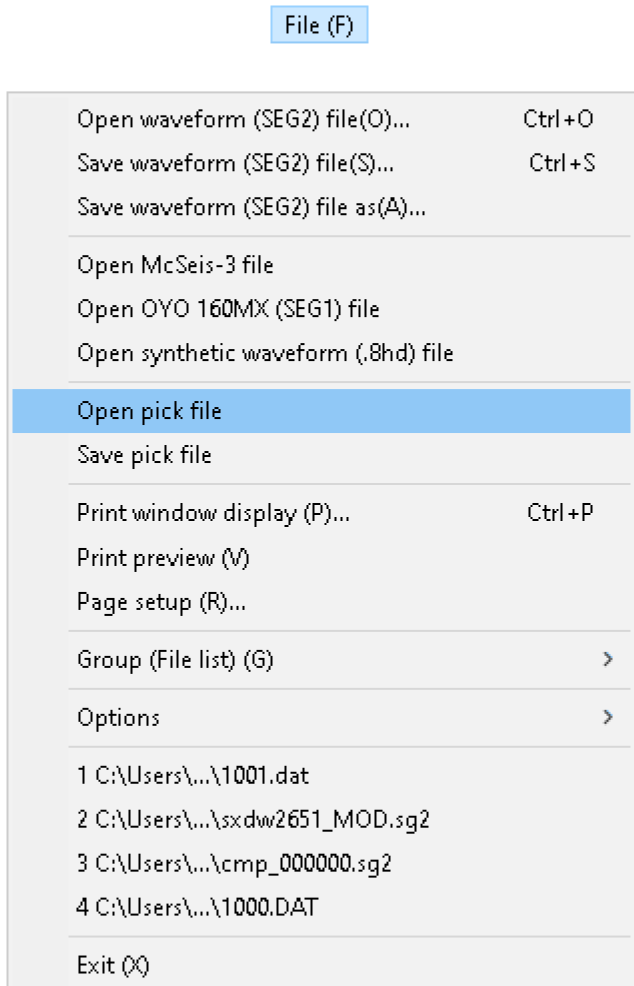
Click on the  shortcut to start Pickwin. You will see the following:



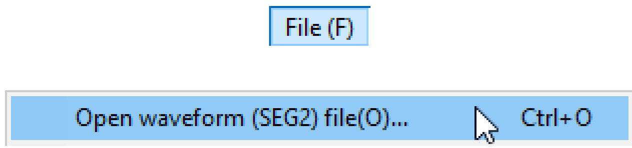
The user-interface of Pickwin consists of a series of menus along with a toolbar. We will now discuss in detail the various menus and features of Pickwin.

3.1 File Menu

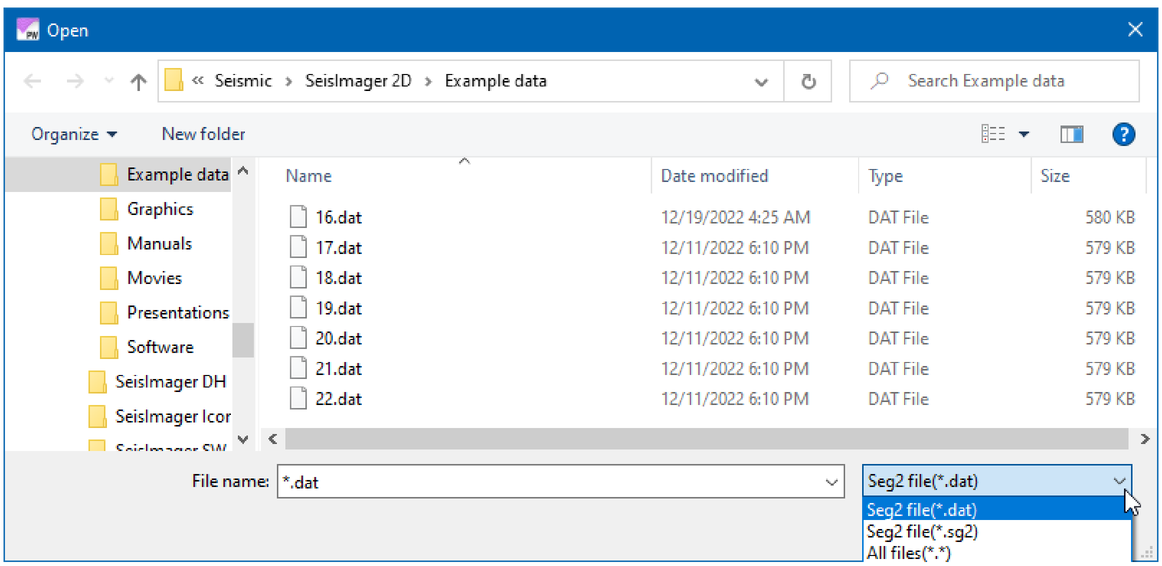
Click on *File* to reveal the **File** menu:



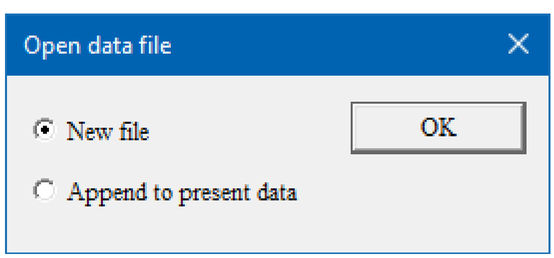
3.1.1 Open Waveform (SEG-2) File [CTRL+O]



Click on *Open waveform (SEG-2) file*, press *Ctrl+O*, or press the "Open File" tool button  to read in a record. You will see a dialog box like the one below:

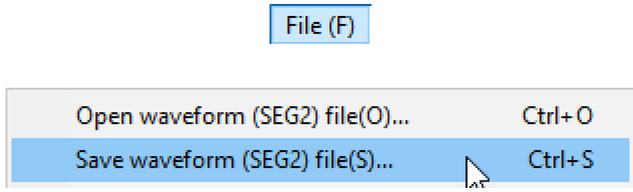


Find the folder your data resides in and open it. SEG-2 files from Geometrics seismographs have a .dat extension, so this is the default, and only .dat files will be displayed (another common extension is .sg2). Choose the file you wish to read in by double-clicking on it. If there is already data in memory, you will be presented with the following dialog box:



Generally, you will be reading in a "new" file, but you may also append records together. The "append" option is discussed in depth in Section 3.2.9 on Page 54. Also see Section 3.1.13.1 on Page 33.

3.1.2 Save Waveform (SEG-2) File [CTRL+S]

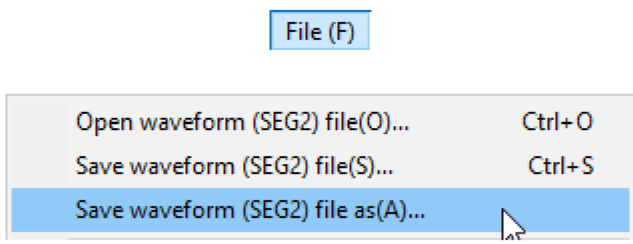


To save a SEG-2 file after editing, choose *Save waveform (SEG-2) file, press Ctrl+S,* or press the "Save File" tool button.  You will get a dialog box like the one above.

Type in a file name and press *Save*. The extension will default to the Geometrics-standard .dat.

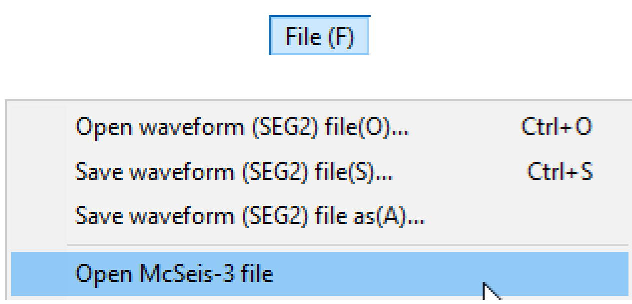
Note: *Again, be careful not to overwrite your raw field records. If you use this option, be sure to have a copy of the raw records stored elsewhere.*

3.1.3 Save Waveform (SEG-2) File As



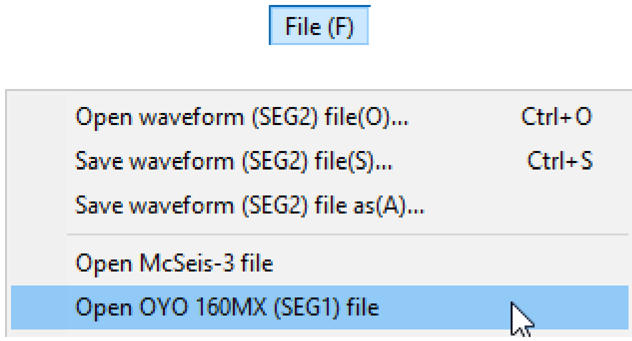
Use this option to save the waveform file with any name you wish. This is the recommended "save waveform" option, as it limits the possibility of overwriting your original raw records.

3.1.4 Open McSeis-3 File



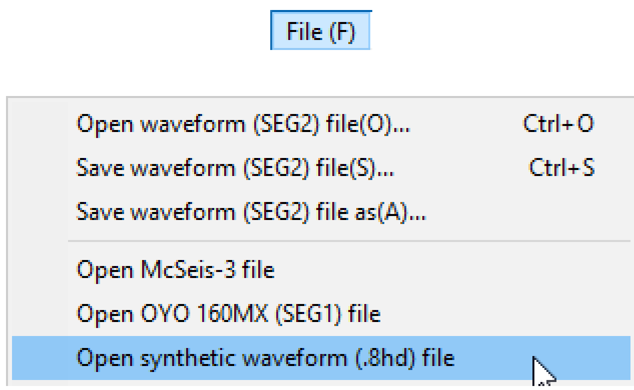
This is identical to that described in Section 3.1.1 above. It is used to open data acquired with the Oyo McSeis-3 seismograph. There is no particular file extension for McSeis files.

3.1.5 Open OYO 160MX (SEG1) File



This is identical to that described in Section 3.1.1 above. It is used to open data acquired with the Oyo 160MX seismograph. There is no particular file extension for 160MX files.

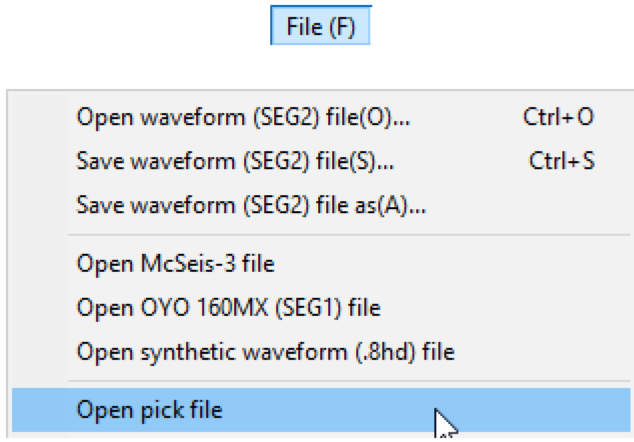
3.1.6 Open Synthetic Waveform (.8hd) File



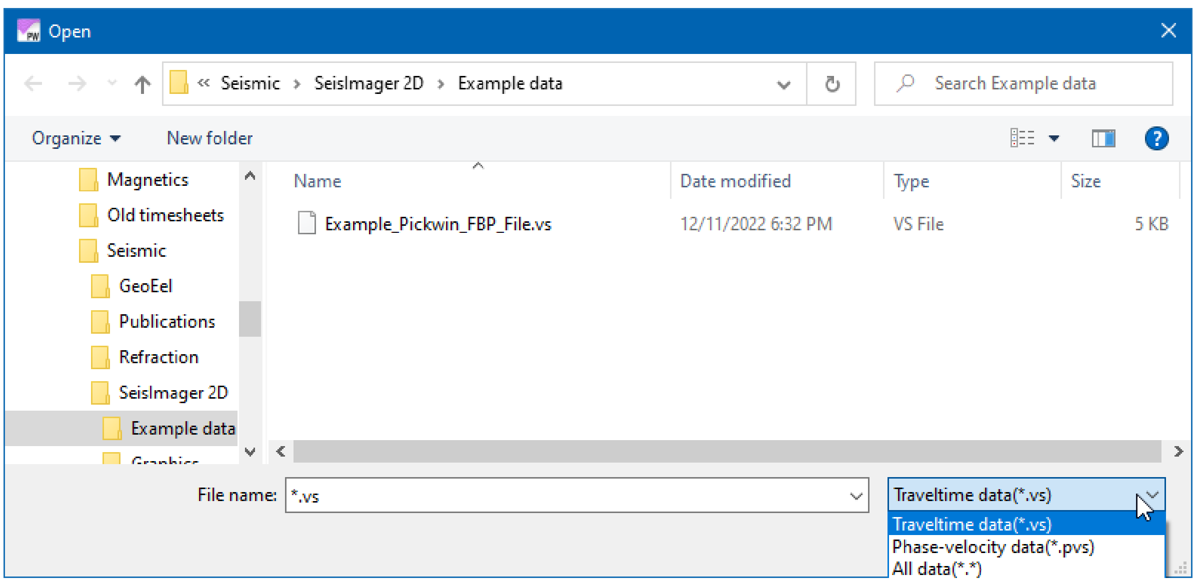
This is identical to that described in Section 3.1.1 above. It is used to open a waveform file generated by WaveEQ. Please refer to the separate SeisImager/SW manual for explanation of this menu item. It is generally not used for refraction.

3.1.7 Open Pick File

A SEG-2 file must be open to use this feature. Click on *Open pick file*.



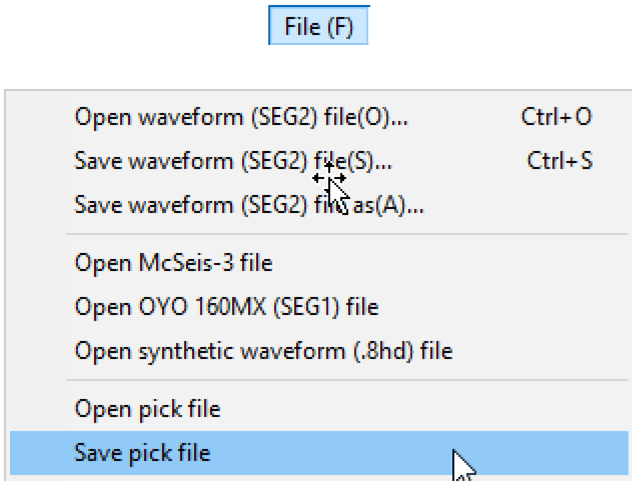
You will see the following dialog box:



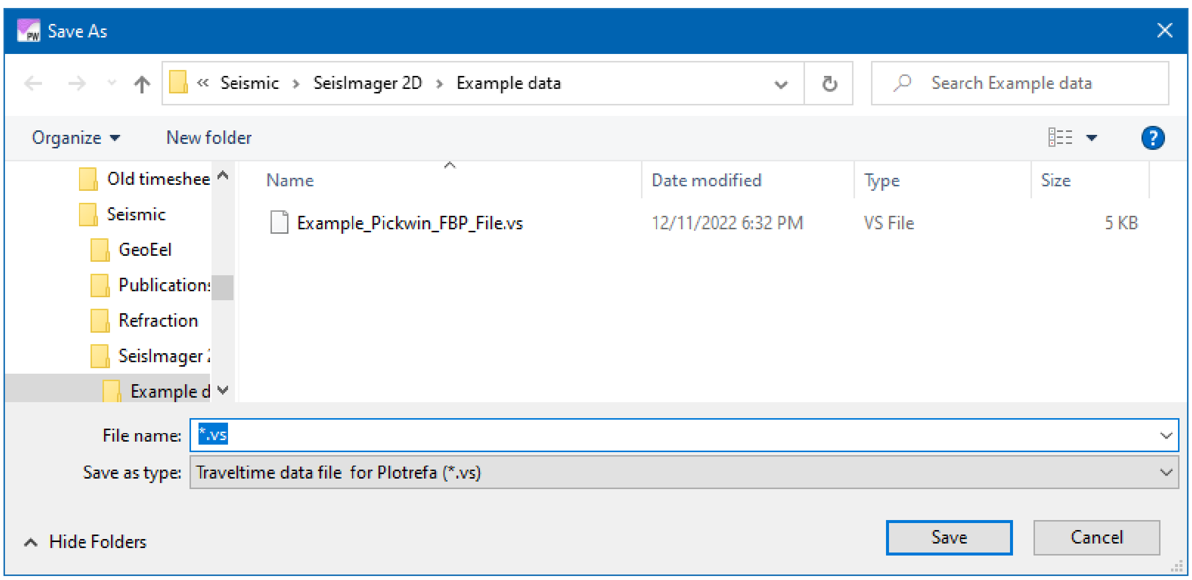
First break pick files that have been picked with Pickwin have a .vs extension. Choose the file you wish to read in by double-clicking on it.

3.1.8 SAVE PICK FILE

To save the first breaks, choose *Save pick file*.

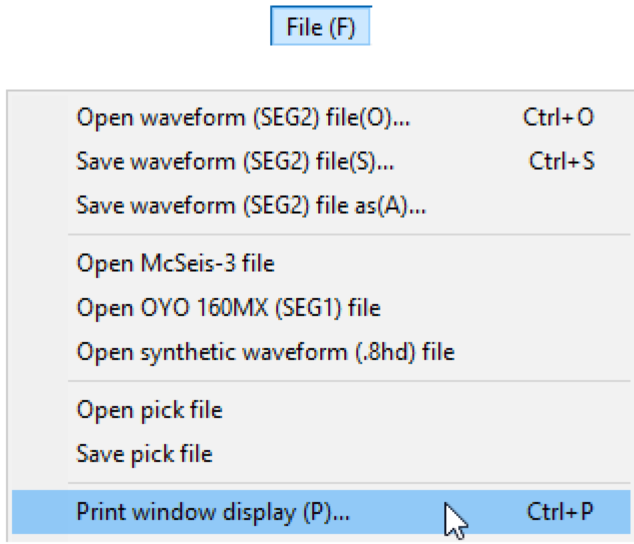


You will see the following dialog box:

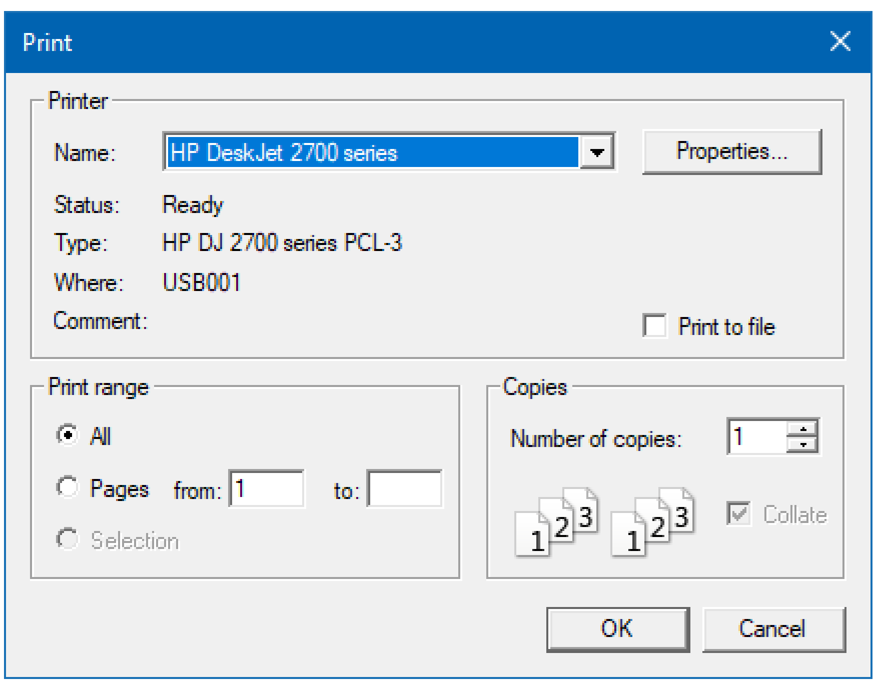


Type in a file name and press *Save*. The extension will default to .vs.

3.1.9 PRINT WINDOW DISPLAY [CTRL+P]

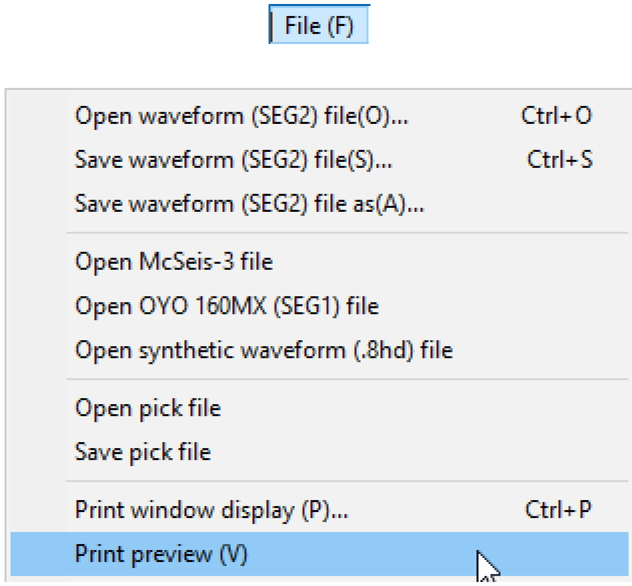


To print the window display of Pickwin, choose *Print window display*, press *Ctrl-P*, or press the "Print" tool button. You will see the print dialog box for your computer:



Press *Print* to print the current window display of Pickwin.

3.1.10 PRINT PREVIEW [V]



To preview the window display of Pickwin for printing, choose *Print preview*. You will see a preview of the window display that will be printed:

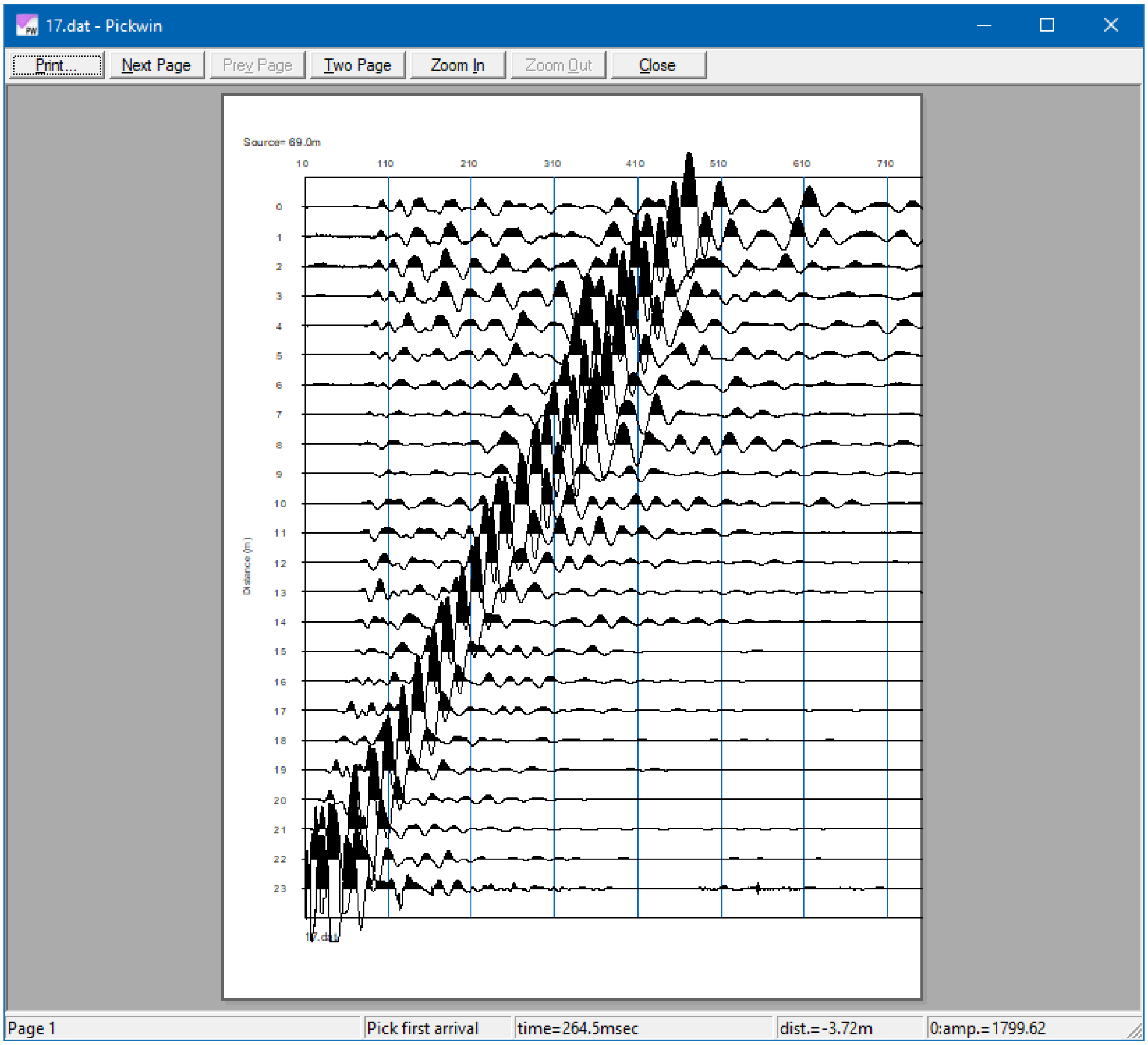
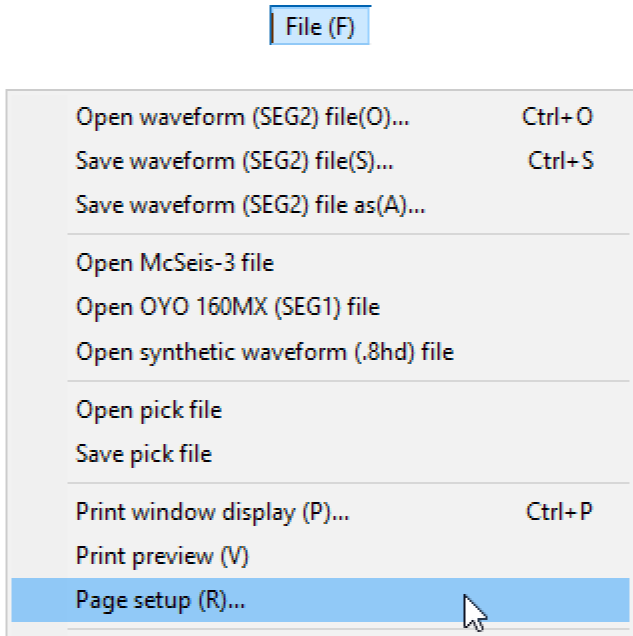


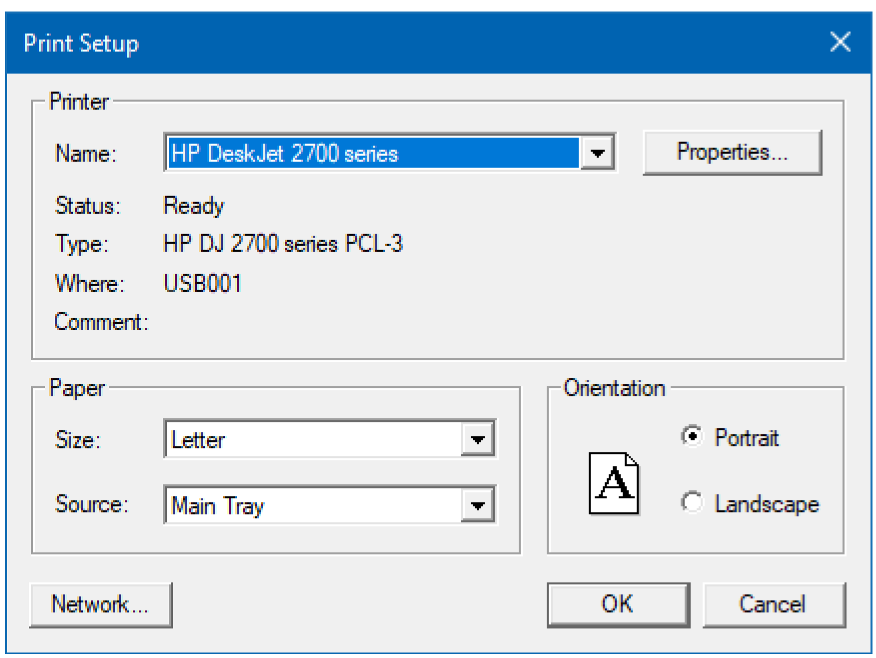
Figure 4: Waveform display ready for printing.

To print this display, press *Print*. To close this display, press *Close.*

3.1.11 PAGE SETUP [R]



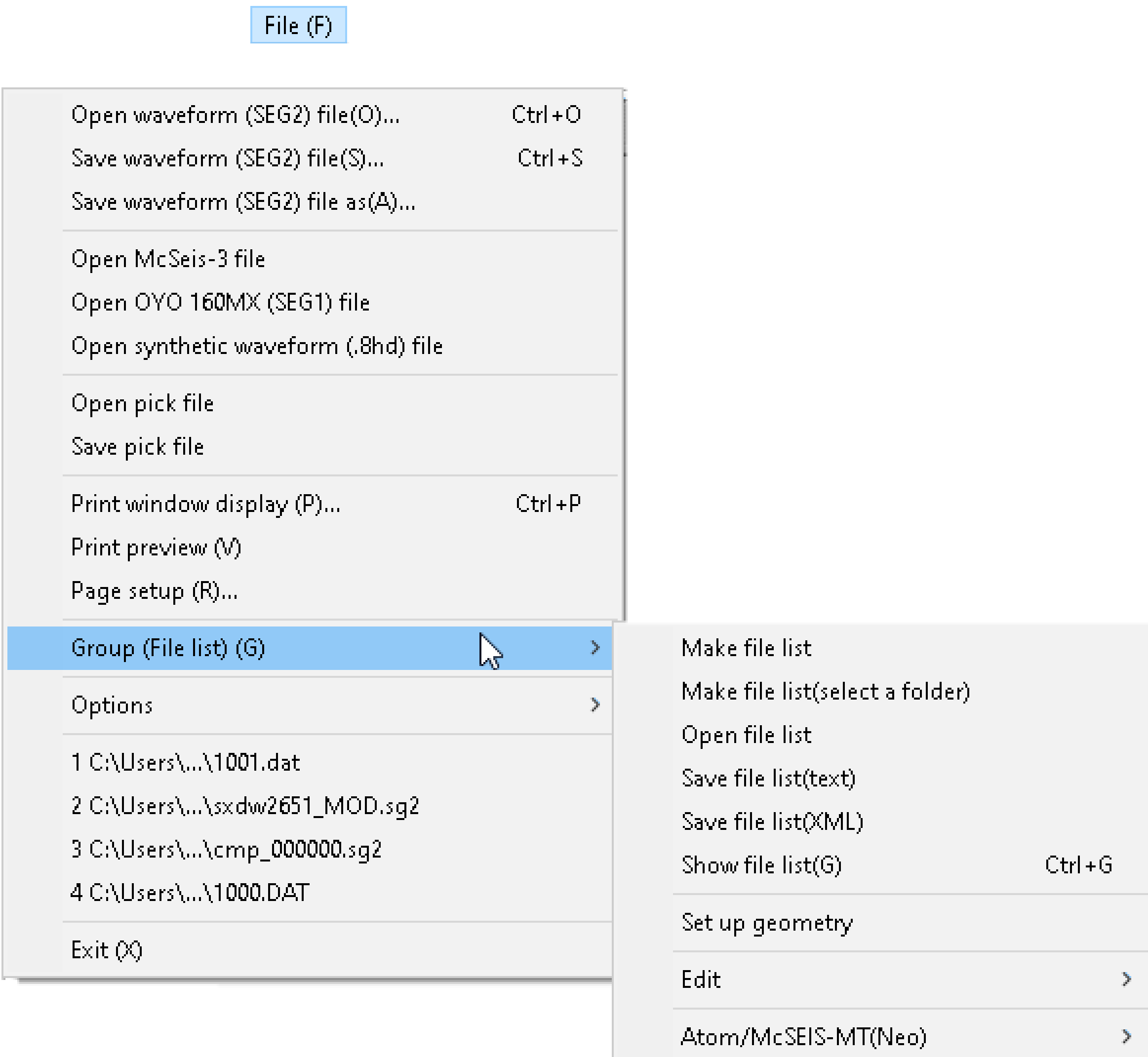
To prepare a page for printing, choose *Page setup*. You will see the print dialog box for your computer:



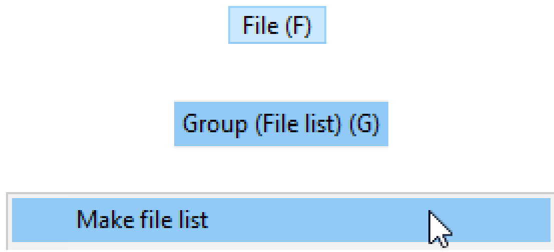
Adjust the properties for printing and press *OK* to print the current window display of Pickwin.

3.1.12 GROUP (FILE LIST) [G]

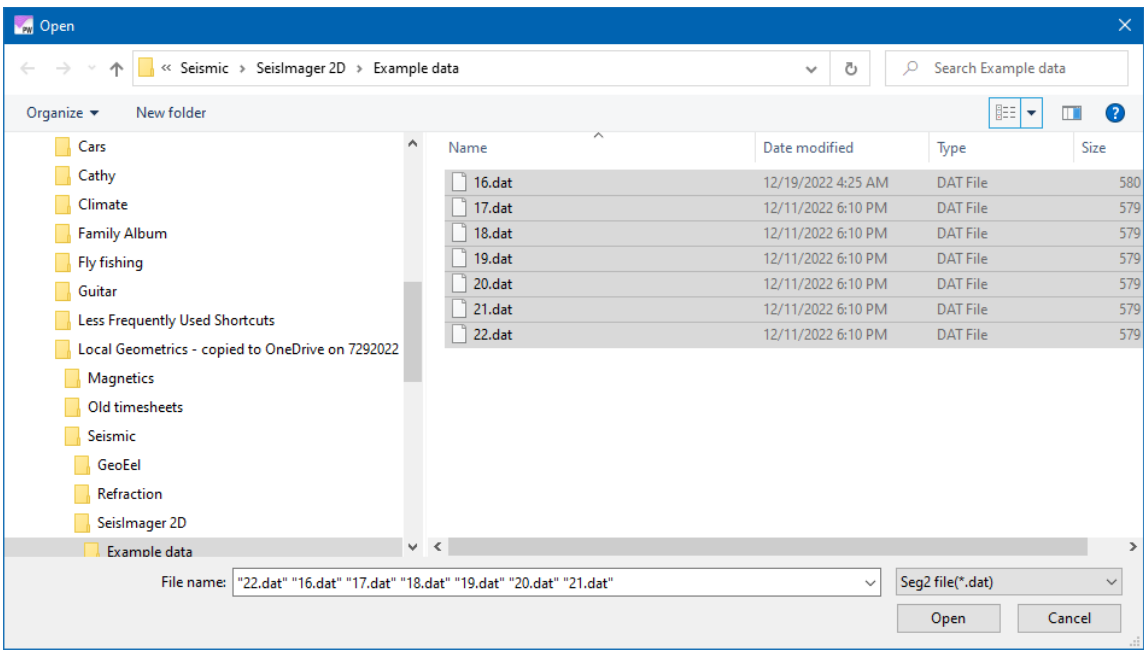
It is often convenient to read in a group of shot records, rather than one at a time. The **Group (File list)** menu can be used to group files together and read them in all at once.



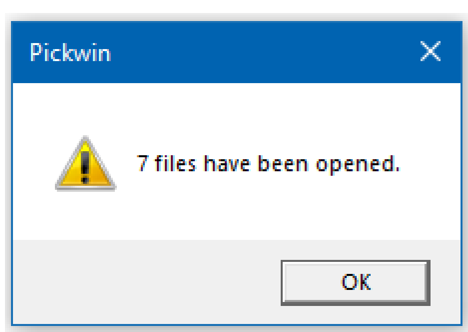
3.1.12.1 Make File List



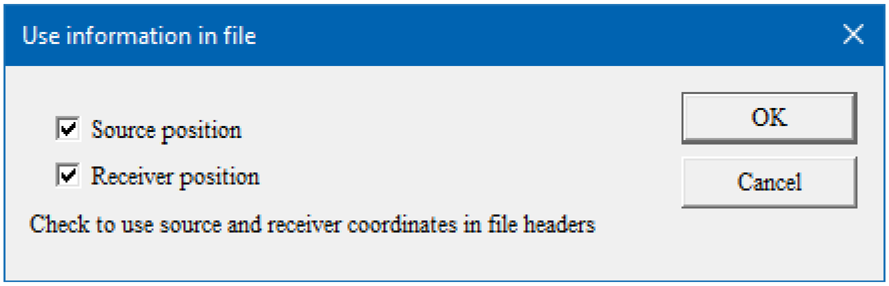
To make a group, select *Make file list* in the sub-menu. Select a folder and choose the files you wish to group by holding down the *Ctrl* or *Shift* key and clicking on them.



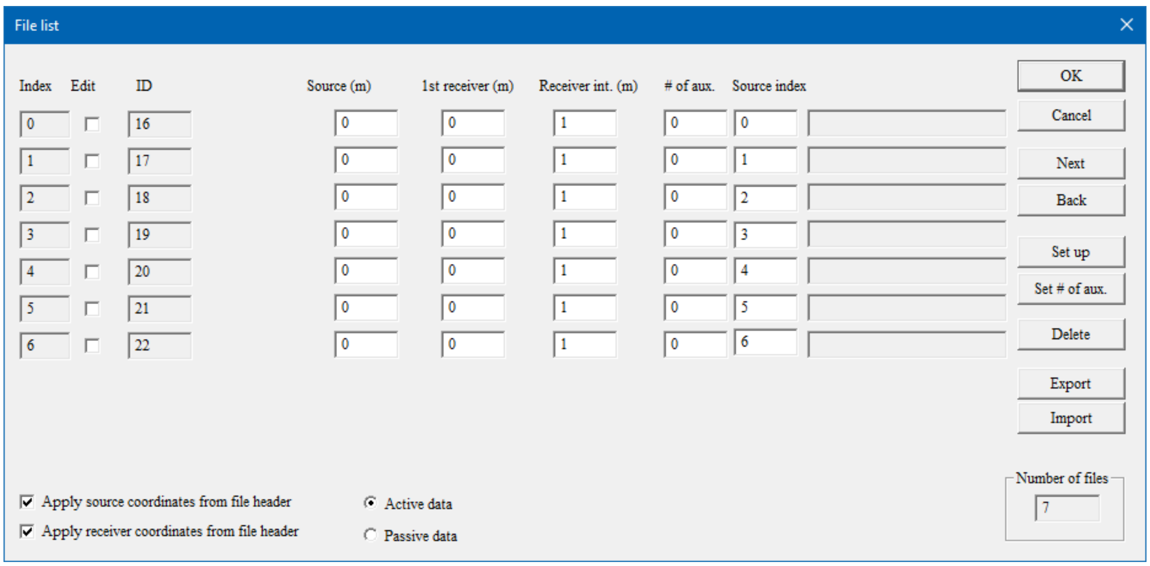
Press *Open* and you will see the following message:



Press *OK:*



If the correct source and receiver positions are included in the data file headers, check both boxes in the above dialog box. The geometry will be automatically read from the file headers. The following table will then be displayed:



If you read the geometry directly from the shot file headers, then **the geometry information displayed in the above table should be ignored**. However, this is a good place to confirm the shot records you have included in the group (see the *ID* column). If you wish to delete one of the files in the group, check the appropriate box in the *Edit* column and press the *Delete* button. Press *OK* to continue.

If the geometry data was *not* read from the shot file headers, this dialog box will allow you to set up the survey geometry. For each file, enter the source position, the location of the first receiver, and the receiver interval. For most refraction applications, you may ignore the *# of aux* column.

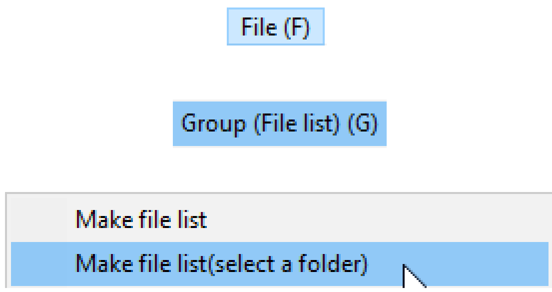
Use the *Next* and *Back* buttons to scroll up and down through the shot record IDs. Ignore the *Setup* button.

Press the *Export* button to export the list as a text (.txt) file for import later.

Press the *Import* button to import an existing text (.txt) list file.

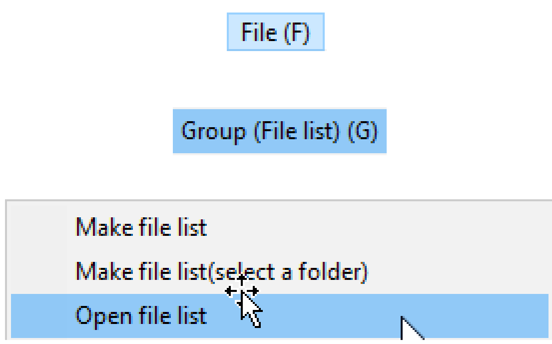
Note: *The above dialog box assumes constant geophone spacings. If you have variable* *geophone spacings,* ***and*** *the geometry data was* ***not*** *recorded in the shot record files, you may enter the geometry information under* ***Edit source/receiver locations, etc.*** *in the* ***Edit/Display*** *menu (see Section 3.2.14, Page 68). This must be done on a file-by-file basis – read in the file, fill in the geometry information, and save the file back out as a SEG-2 file.* ***It is generally advisable to set the geometry parameters in the field and record them in the shot file headers.***

3.1.12.2 MAKE FILE LIST (SELECT A FOLDER)

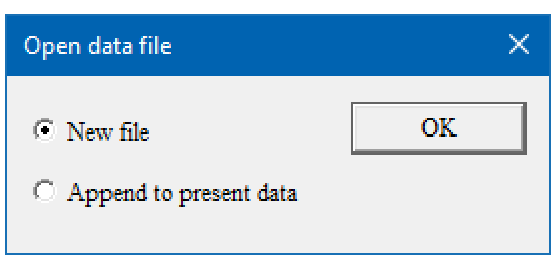


This feature is identical to *Make File List* above, except that in this case, all you must do is choose the folder that the seismic waveform files are in. You do not need to choose the files – all the waveform files in the folder will be read in. This is convenient when you have a large number of waveform files in a folder.

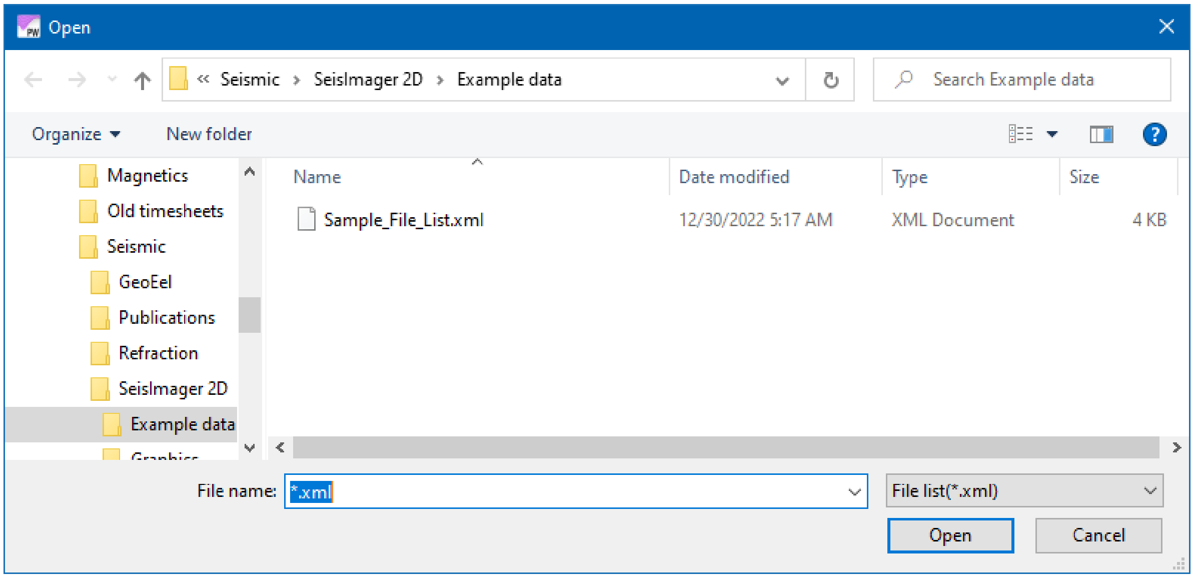
3.1.12.3 OPEN FILE LIST



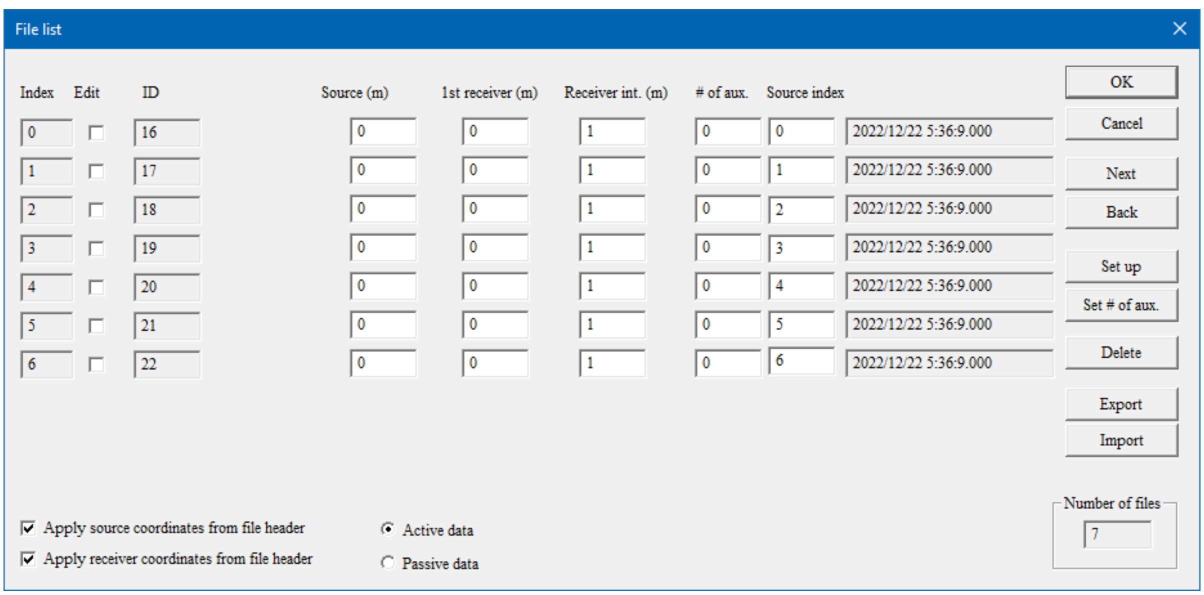
Once you have created a group, you may save it for future retrieval in .txt or .xml format (see below). To open a group, select *Open file list*. You will be presented with the following dialog box:



You can append file groups in the same way you can append individual files. In this case, choose *New file* and press *OK*:

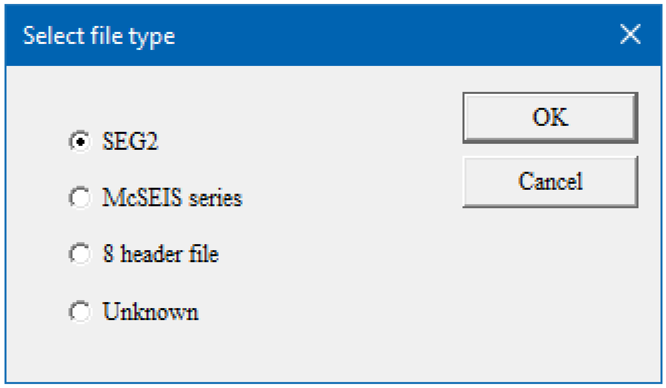


Choose the appropriate group file (the default file format is .xml). If the file is in .xml format, you will see the geometry dialog box:

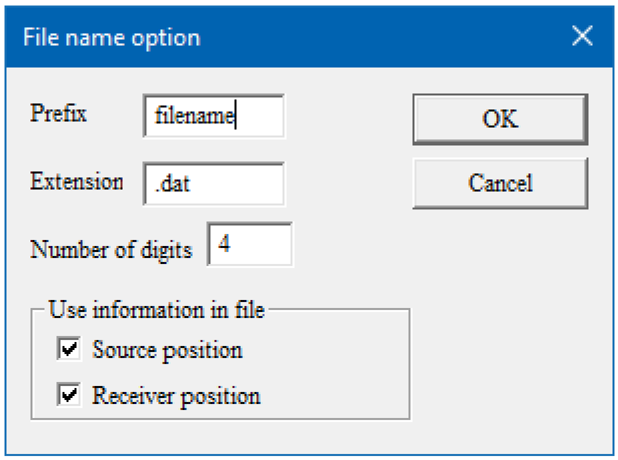


This is an opportunity to modify the geometry or to delete shot files from the group. Press *OK* to display the first shot record in the group.

If the group file is in a .txt format, you will be presented with the following dialog box:



Indicate the file type and press *OK.* The following dialog box will appear:



Prefix: Some seismographs put some sort of prefix before the file ID number, e.g., FILE2001.SG2. When the group file is in text format, you must enter any prefix manually. If there is no prefix (Geometrics seismographs), leave the prefix field blank.

Extension: Different seismographs use different file extensions, such as .dat (Geometrics-standard) or .sg2. Enter the correct extension.

Digits: Enter the number of digits in the file ID number.

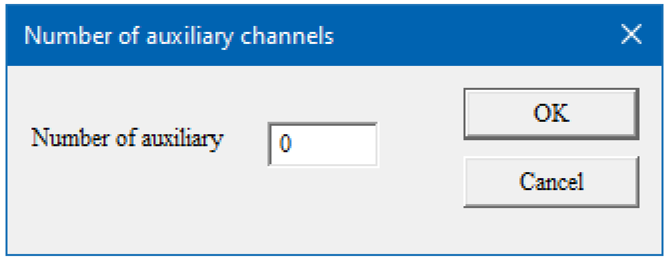
Finally, indicate whether the source and receiver positions should be read from the file headers.

Press *OK*, and you will be presented with the **Geometry** menu, as above. Press *OK* again to display the first shot record in the group.

Note: *In general, XML format tends to be the most convenient. Whenever you have the choice, save to XML.*

Once you reach the data display, you may now page through all the shot records in the group using the  and  tool buttons. This is extremely convenient in the first break picking process.

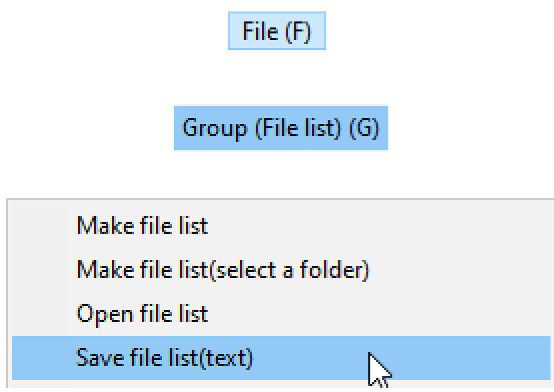
Next you will be prompted for the number of auxiliary channels:



Auxiliary channels are used to record non-standard data, such as the trigger signal coming from the source. In most refraction work, this will be set to zero.

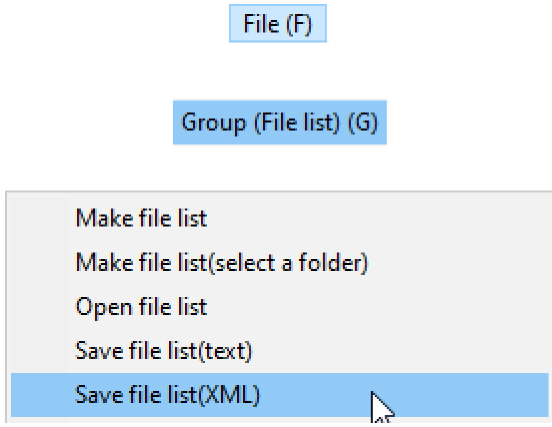
3.1.12.4 Save File List (text)

If you modify the file list, you must save it.



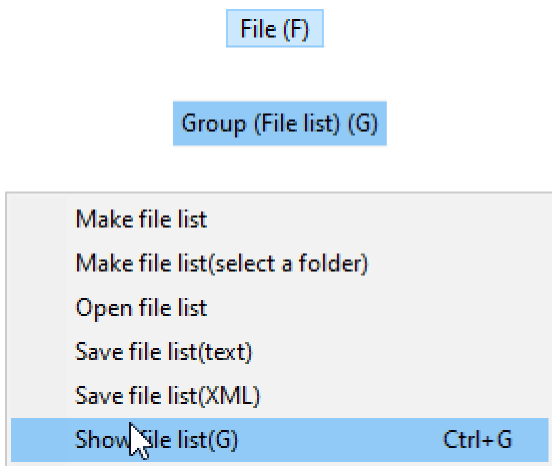
Choose *Save file list (text)* to save the group as a text file.

3.1.12.5 Save File List (XML)



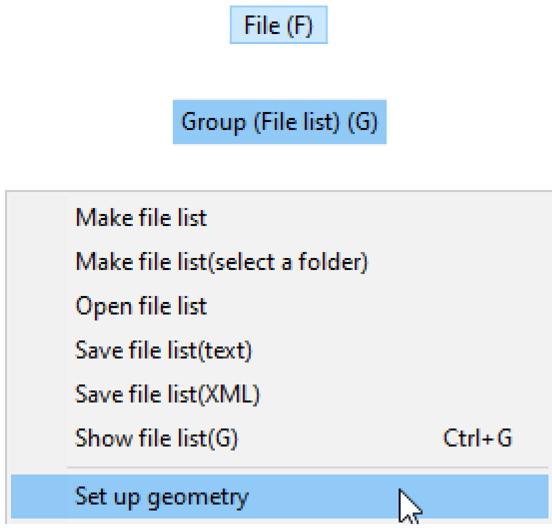
Choose *Save file list (XML)* to save the group as an XML file.

3.1.12.6 Show File List [CTRL+G]



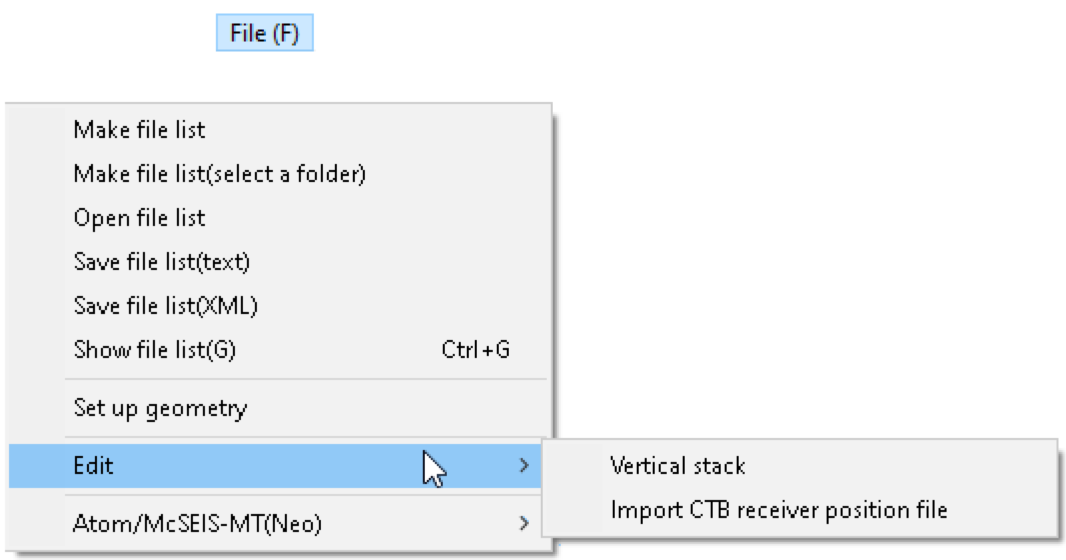
Select *Show file list* or press *Ctrl+G* to display the file list that you are working with. This can be a file list that you just created using *Make File List* (and have not saved), or it can be a file list that was created and saved at another time that you have imported.

3.1.12.7 SET UP GEOMETRY



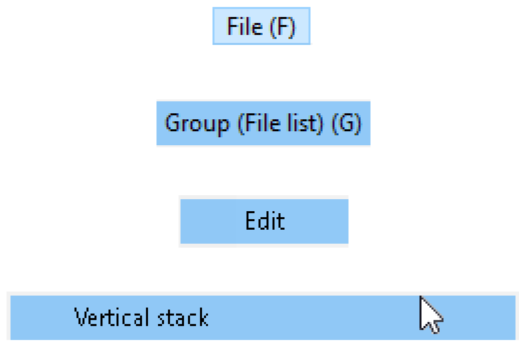
Please refer to the separate SeisImager/SW manual for explanation of this menu item. It is generally not used for refraction.

3.1.12.8 EDIT



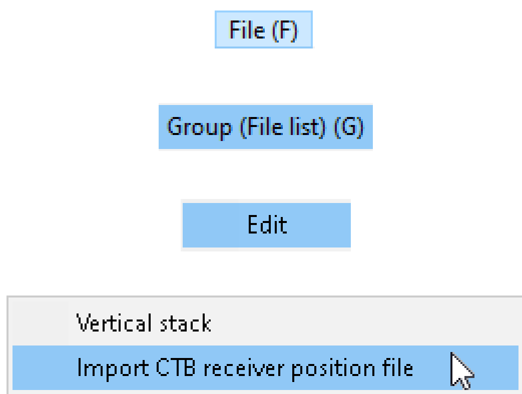
Continue.

3.1.12.8.1 Vertical Stack



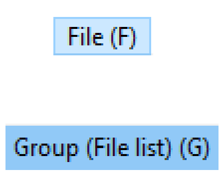
Please refer to the separate SeisImager/SW manual for explanation of this menu item. It is generally not used for refraction. See Section 3.7.3, Page 94, if you are doing refraction and need to vertically stack your records.

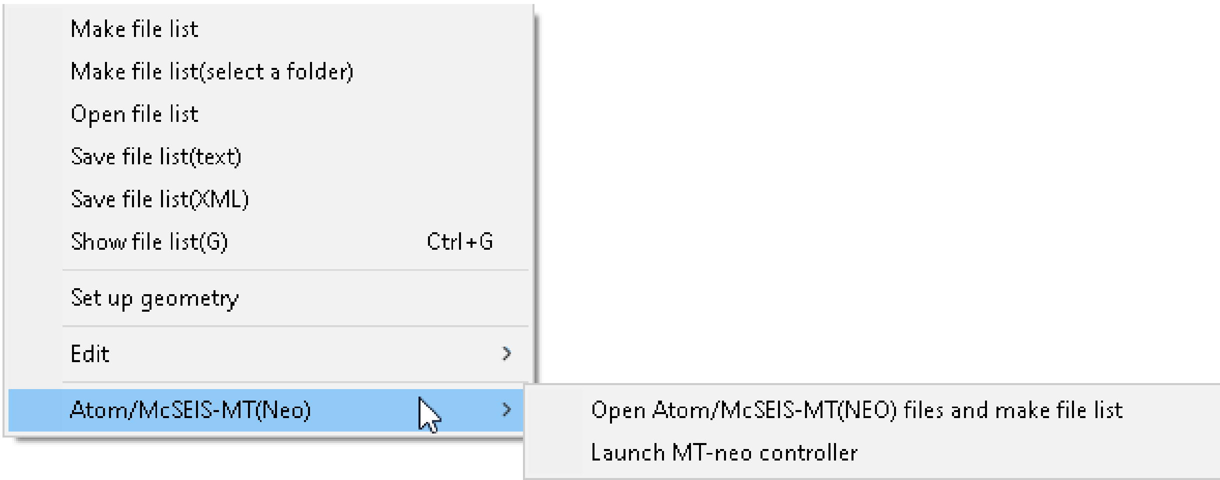
3.1.12.8.2 Import CTB Receiver Position File



Please refer to the separate SeisImager/SW manual for explanation of this menu item. It is not used for refraction.

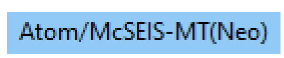
3.1.12.9 Atom/McSeis-MT (Neo)

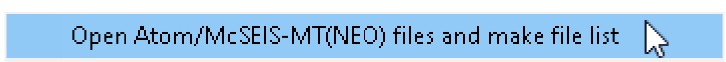




Continue.

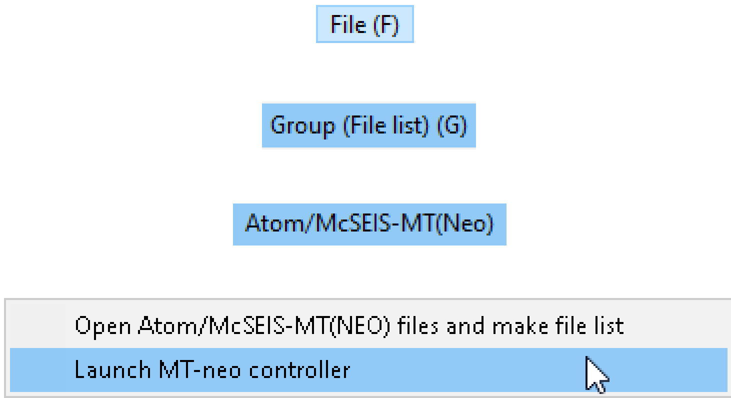
3.1.12.9.1 Open Atom/McSeis-MT (Neo) Files and Make a File List





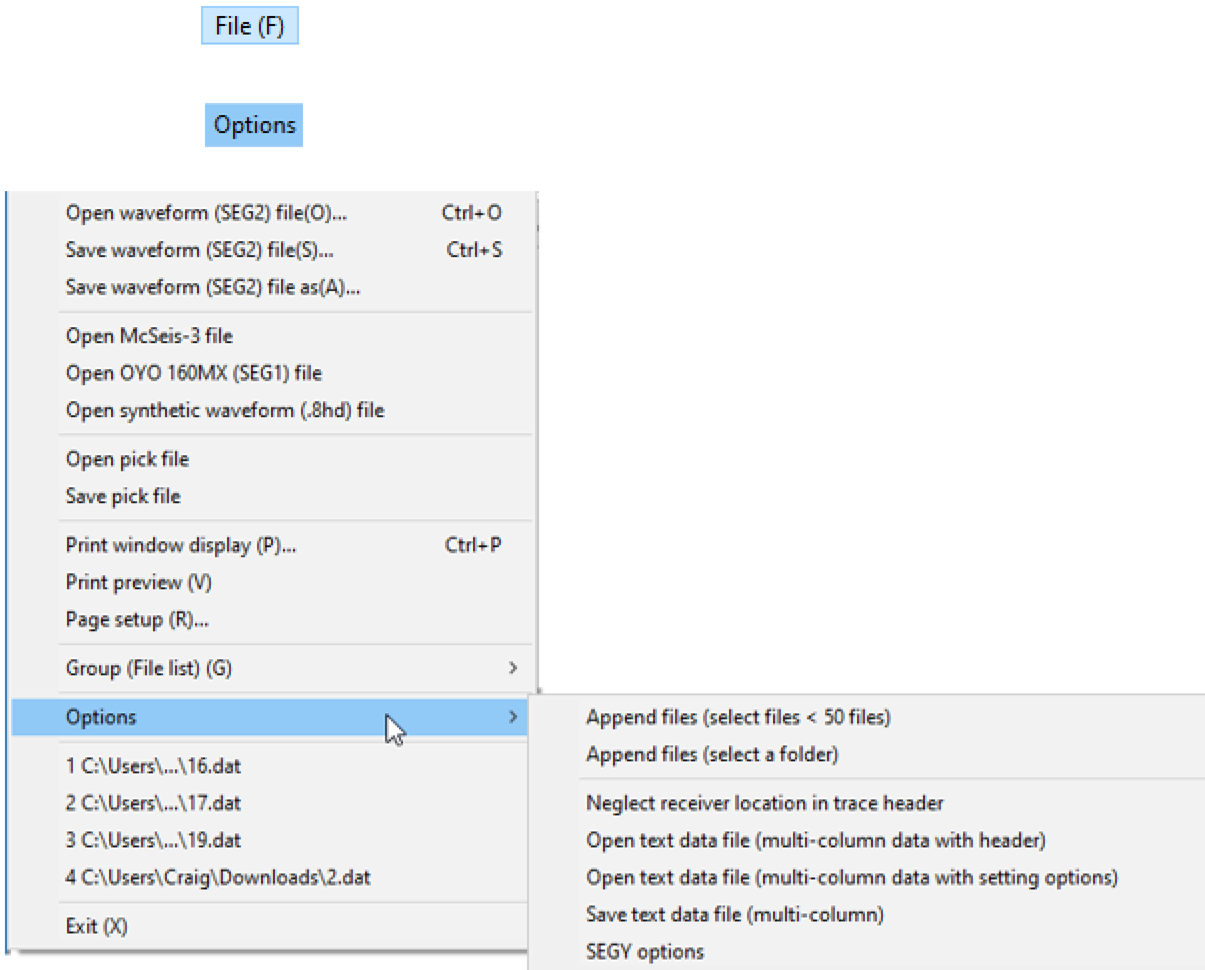
Please refer to the separate SeisImager/SW manual for explanation of this menu item. It is generally not used for refraction.

3.1.12.9.2 Launch MT-Neo Controller



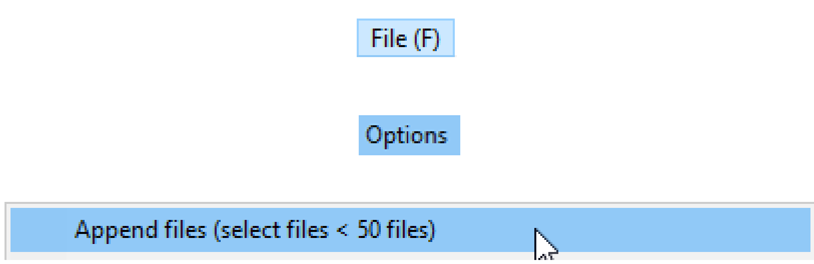
Please refer to the separate SeisImager/SW manual for explanation of this menu item. It is generally not used for refraction.

3.1.13 OPTIONS



Continue.

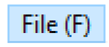
3.1.13.1 APPEND FILES (SELECT FILES < 50 FILES)

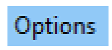


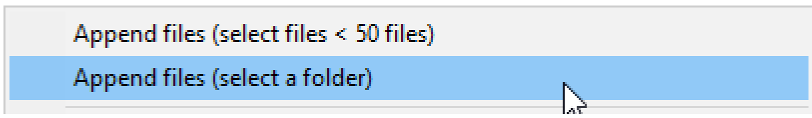
This feature opens a file-read dialog box and allows you to choose up to 49 SEG-2 files to append together.

Note: *Do not confuse "append" with "concatenate". See Section 3.2.9, Page 54, for a discussion of appending records. Concatenating records is used almost exclusively in surface-wave applications.*

3.1.13.2 Append Files (Select a Folder)

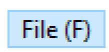


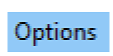


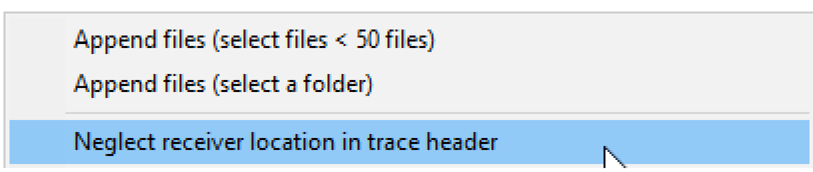


This feature appends together all the files in the selected folder. It is useful when you have many files to append together.

3.1.13.3 Neglect Receiver Location in Trace Header

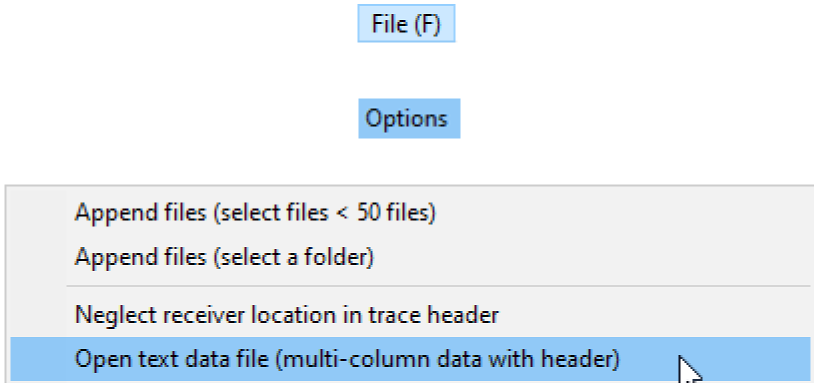






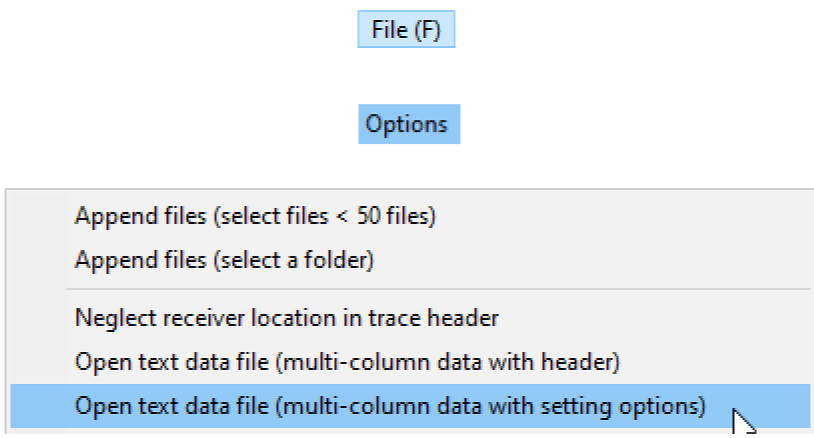
If you did not record the correct receiver locations in the trace headers while in the field, you can choose to ignore the trace headers by choosing this option.

3.1.13.4 Open Text Data File (Multi-column Data With Header)

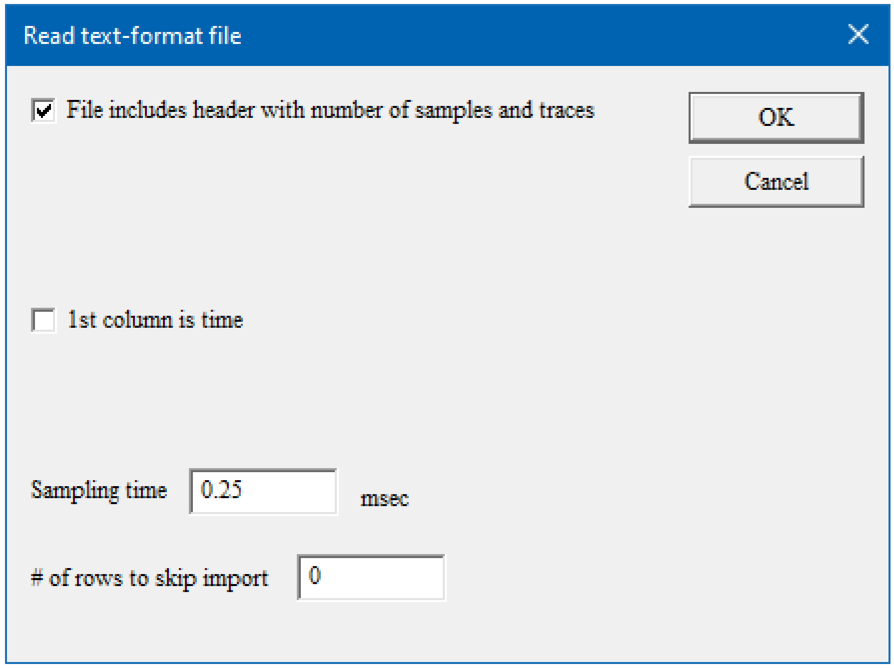


Select this option to open a previously saved waveform file that you have saved in ASCII text format (see Section 3.1.13.6 below).

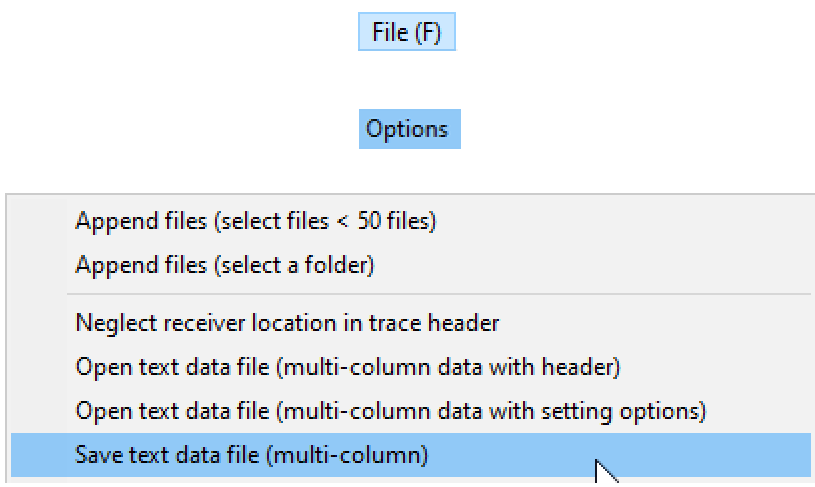
3.1.13.5 Open Text Data File (Multi-column Data With Setting Options)



Select this option to open a previously saved waveform file that you have saved in ASCII text format (see Section 3.1.13.6 below). In this case, you will be prompted for the following parameters:

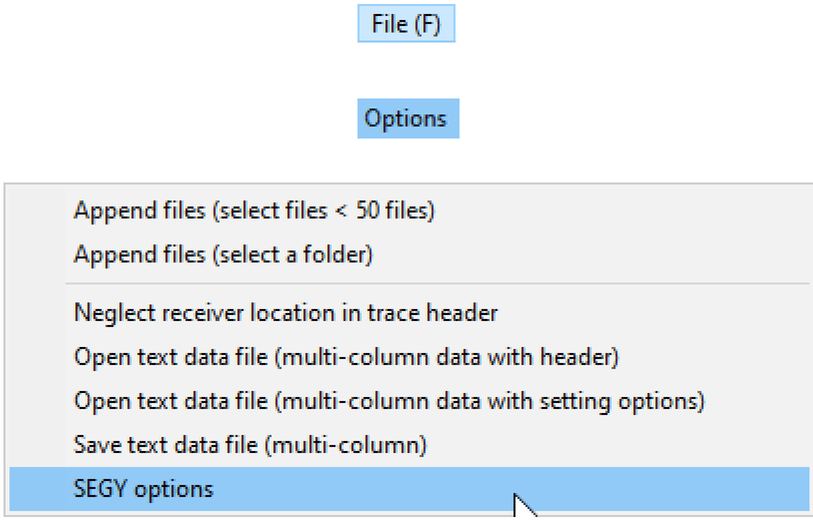


3.1.13.6 Save Text Data File (Multi-column)

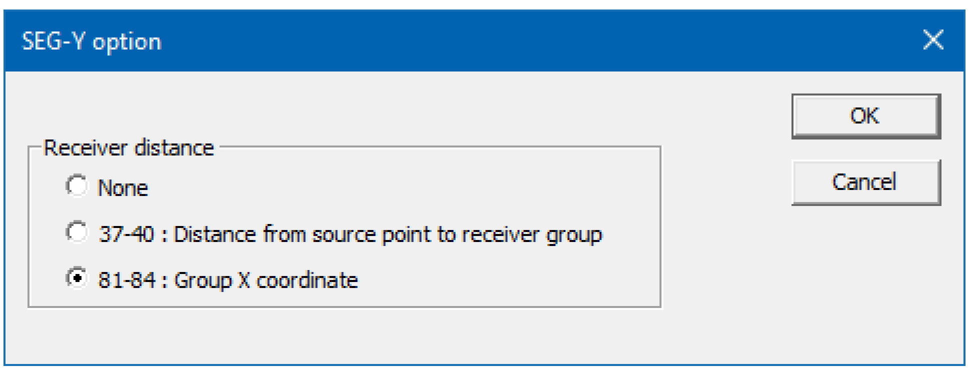


You may save any waveform file as an ASCII text file. Simply read in the waveform file and select *Save text data file (multi-column)*. You will be prompted for a file name.

3.1.13.7 SEGY Options



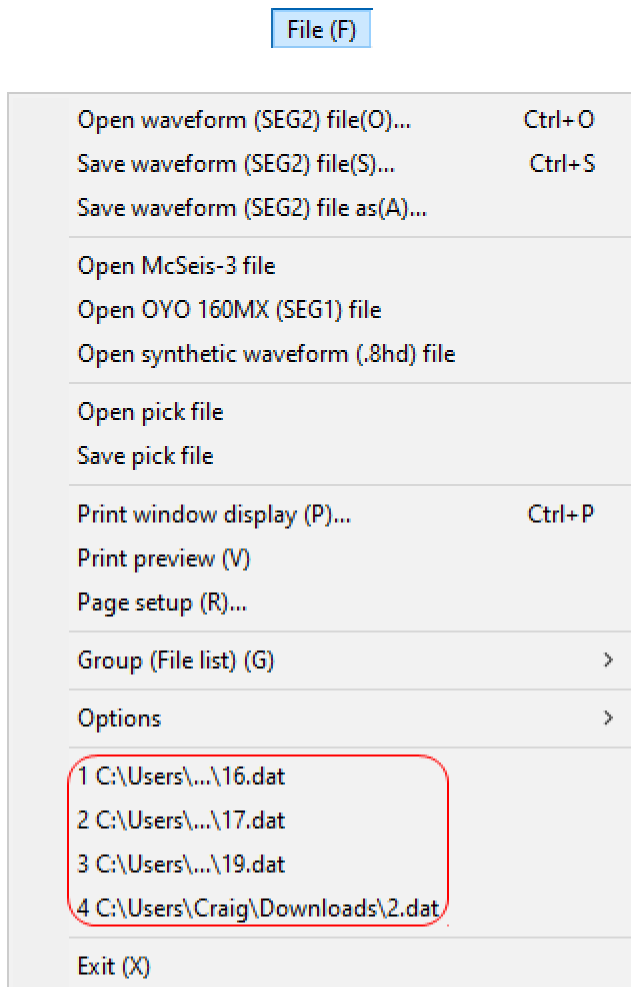
Selecting this item will reveal the following dialog:



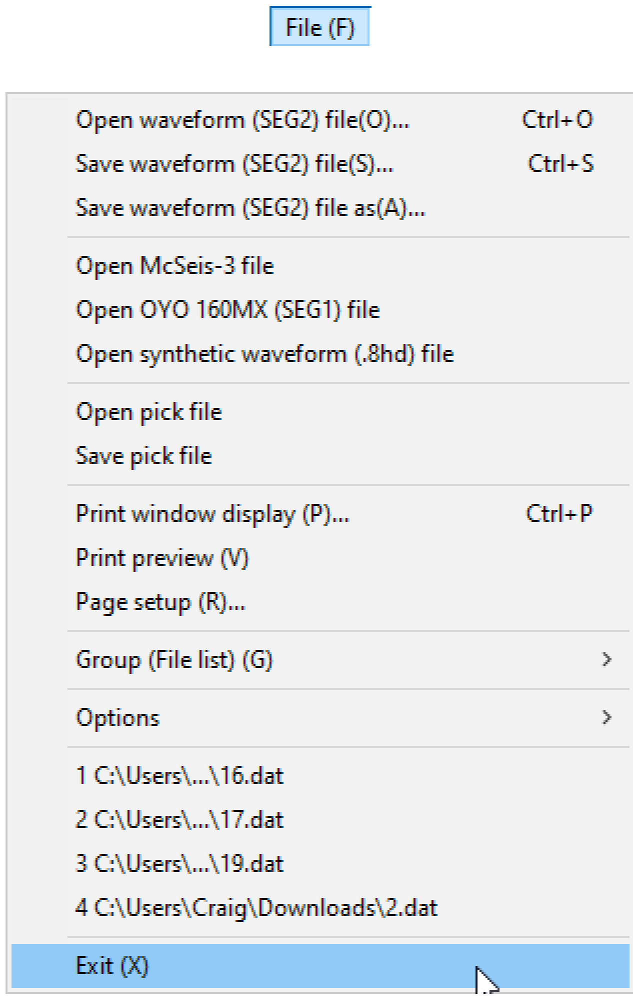
Choose the appropriate header locations for the receiver distances or coordinates. If they were not recorded in the field, select *None*.

3.1.14 Recent Files

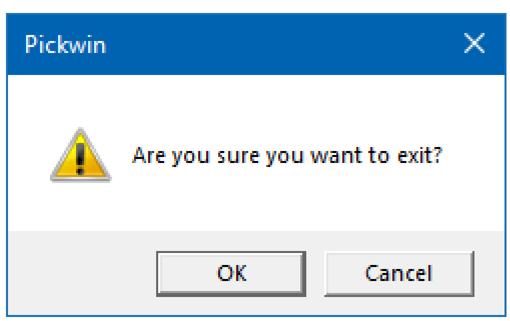
The last four files opened in Pickwin will be displayed in the **File** menu. To open any of these files, just click on the file.



3.1.15 EXIT [X]



To exit the Pickwin module, choose *Exit*. You will see the following dialog box:

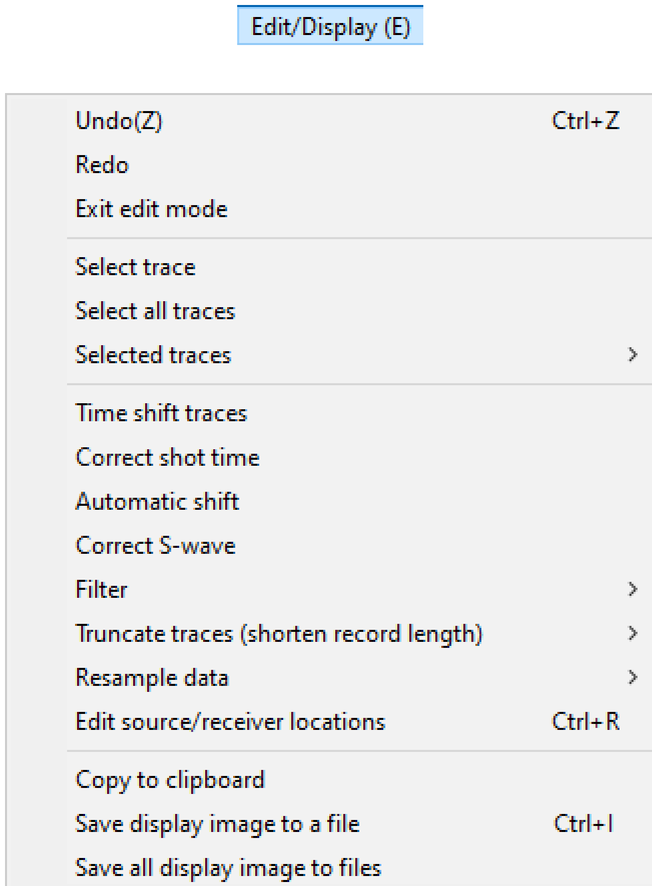


Press *OK* to exit Pickwin or press *Cancel* to continue using Pickwin.

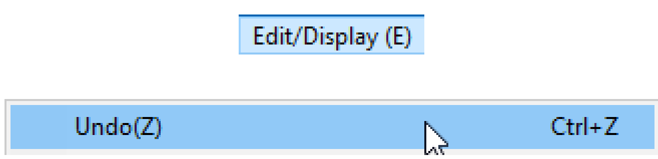
3.2 EDIT/DISPLAY MENU

Note: *Be sure to do any trace editing* ***before*** *picking your first breaks.*

Click on *Edit/Display* to reveal the **Edit/Display** menu:

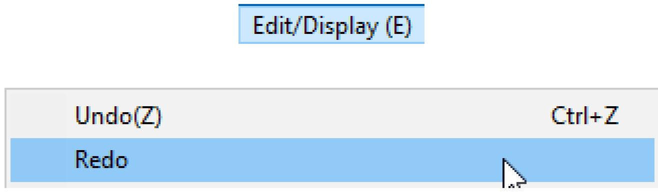


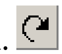
3.2.1 UNDO [CTRL+Z]



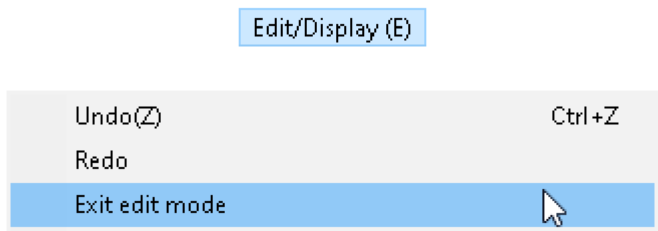
To undo the last command performed, click on *Undo*, press *Ctrl+Z*, or press the "Undo" tool button.  The last operation performed by Pickwin will be undone.

3.2.2 REDO

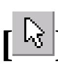


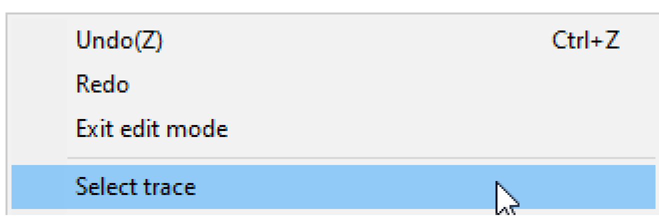
To re-do a command that was undone, click on *Redo* or press the "Redo" tool button.  The last operation that was undone will now be redone.

3.2.3 EXIT EDIT MODE



This is the same as pressing *Cancel* or *Escape*. There are many edit modes, such as picking first arrivals, shifting waveforms, etc. *Exit edit mode* terminates all such edit modes and brings the application to neutral.

3.2.4 SELECT TRACE []



Certain operations can be performed on individual traces. It is therefore necessary to first choose which traces you wish to perform these operations on. Specifically, a trace must be selected before it can be reversed in polarity, killed, or deleted. To enable individual trace selection, click on *Select trace.*

Alternatively, you may enable/disable trace selection by pressing the  tool button.

Select a trace for editing by clicking on it. The trace will change from black to red when it is selected.

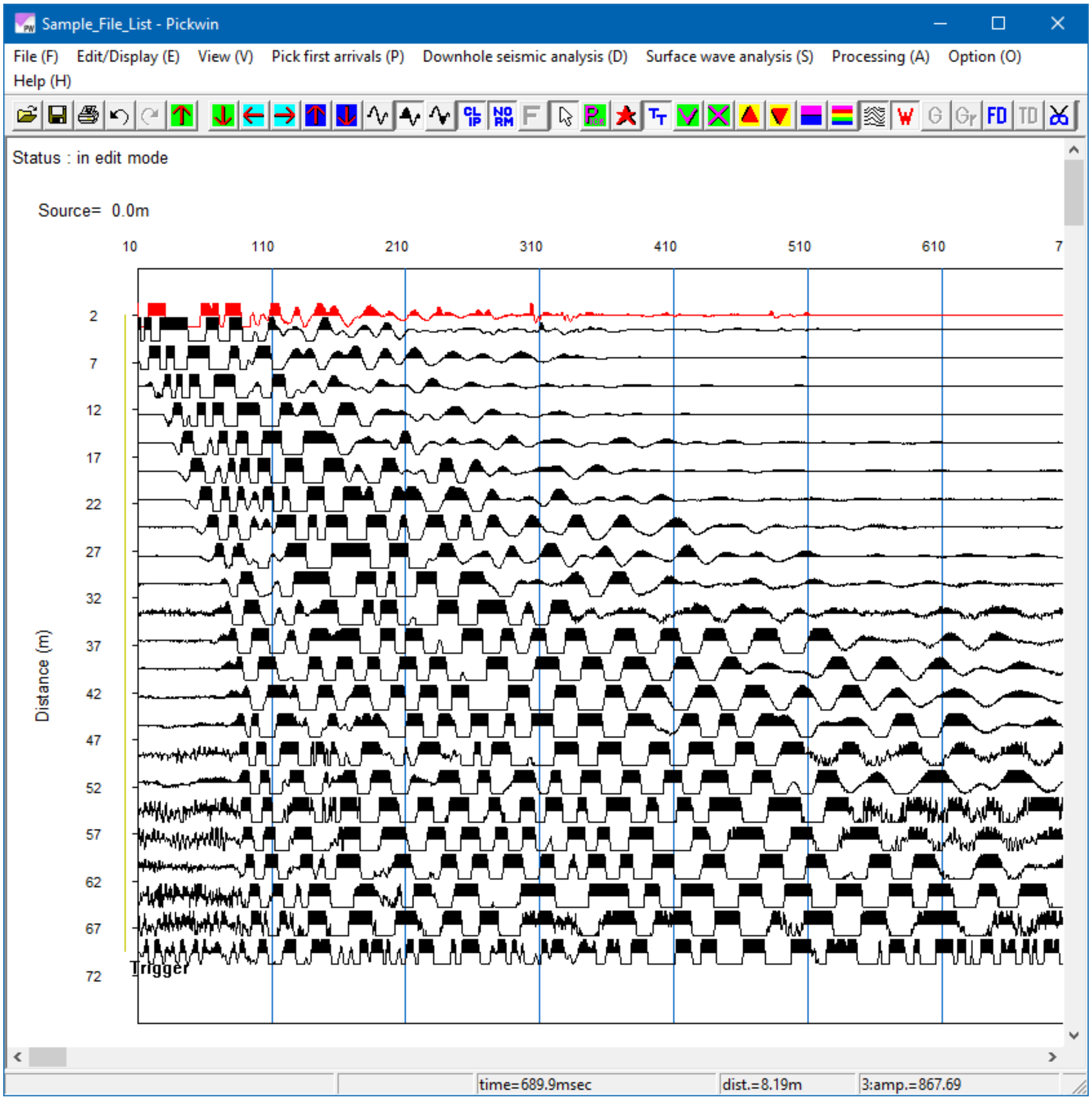
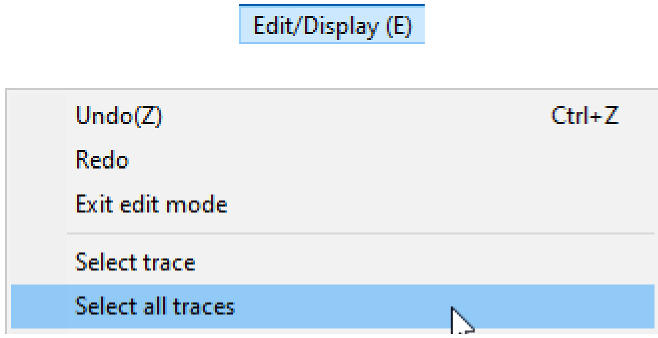


Figure 5: Trace selection. In the above figure, Trace #1 has been selected.

3.2.5 SELECT ALL TRACES



To select all the traces at once, click on *Select all traces*. All the traces will turn red, and a check mark will now be displayed next to the **Select trace** menu item:

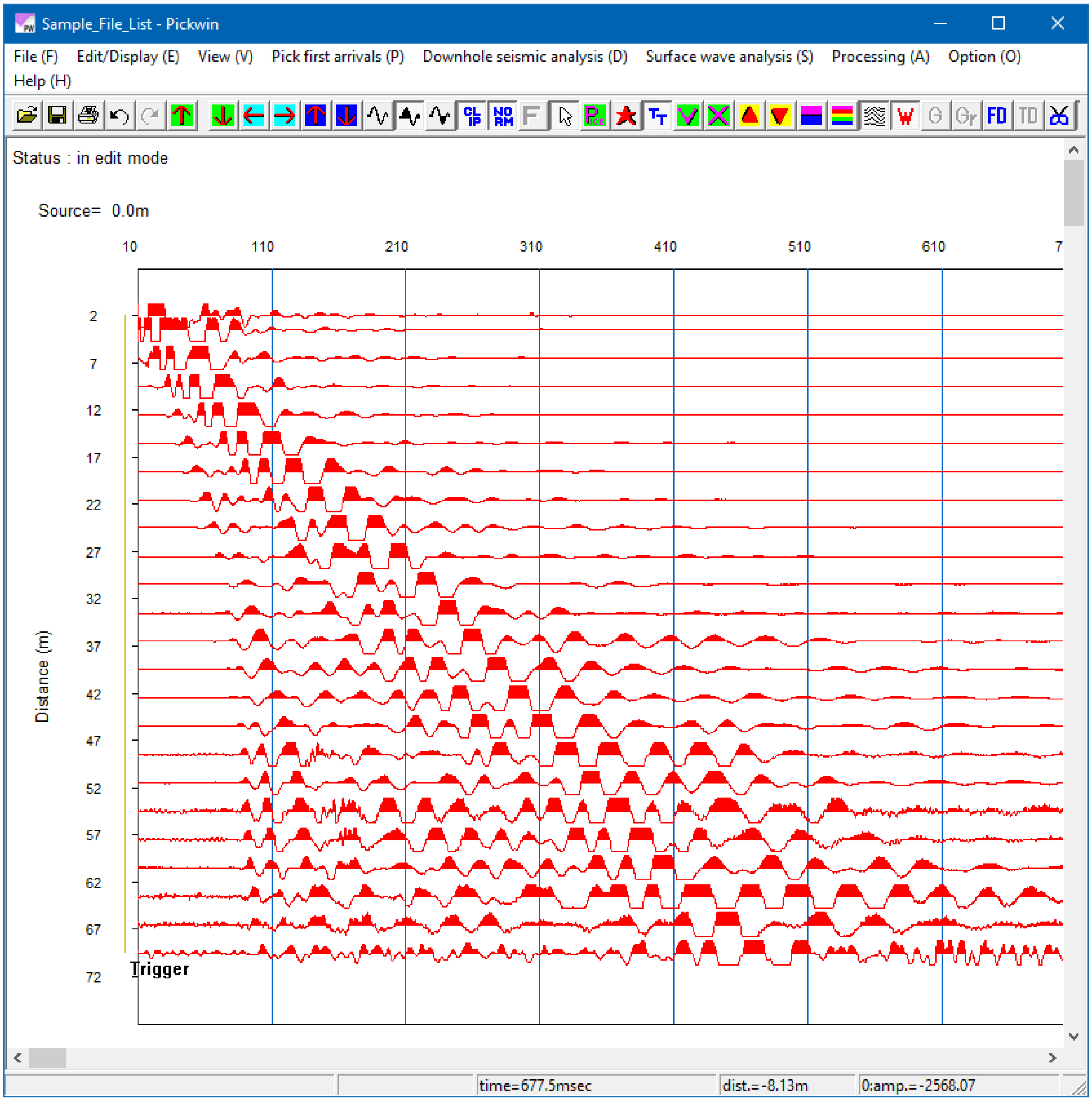


Figure 6: Trace selection. In the above figure, all traces have been selected.

Alternatively, you can press the  button and then drag your mouse over some or all of the traces. This is a convenient way to select a group of traces:

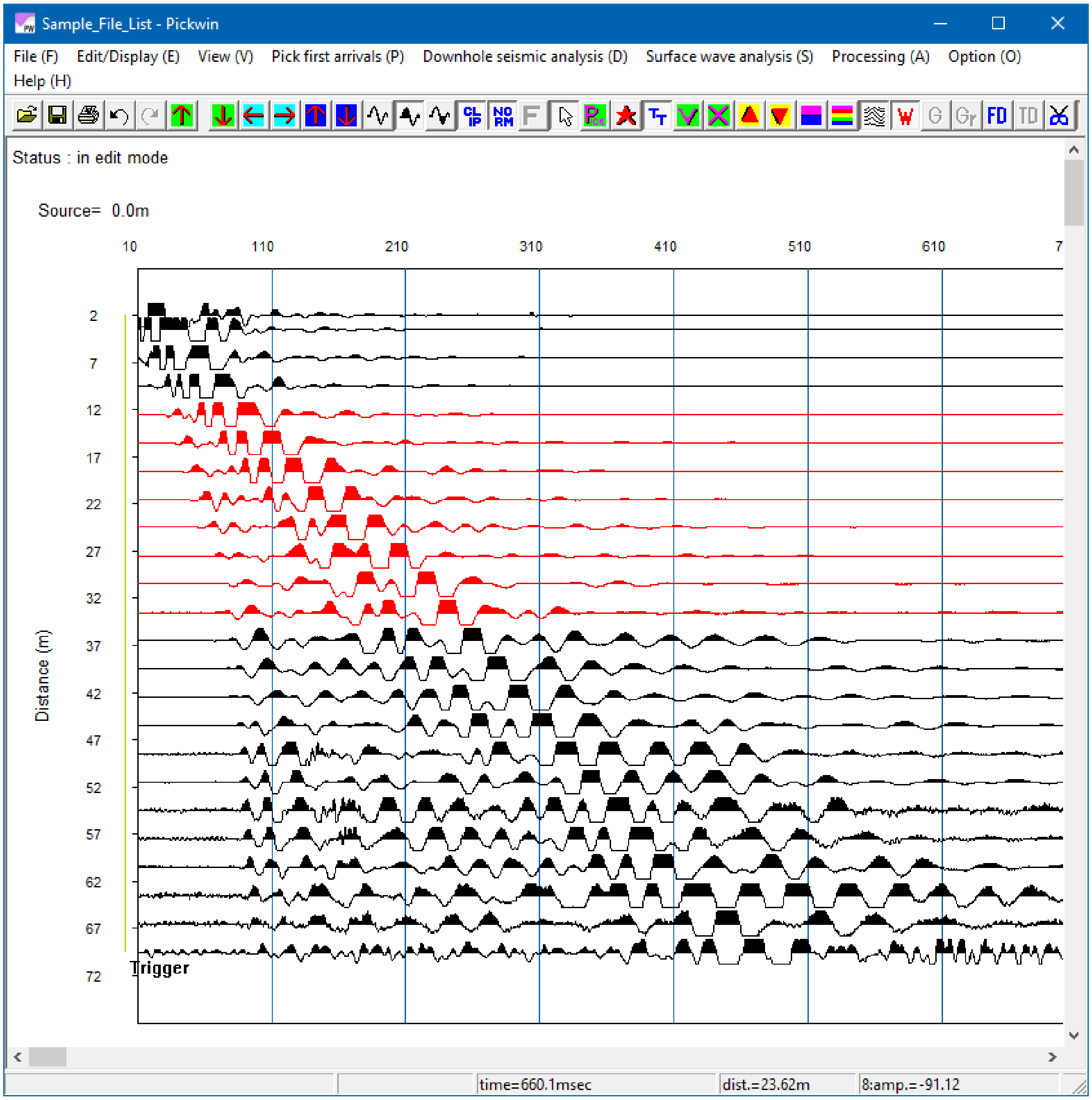
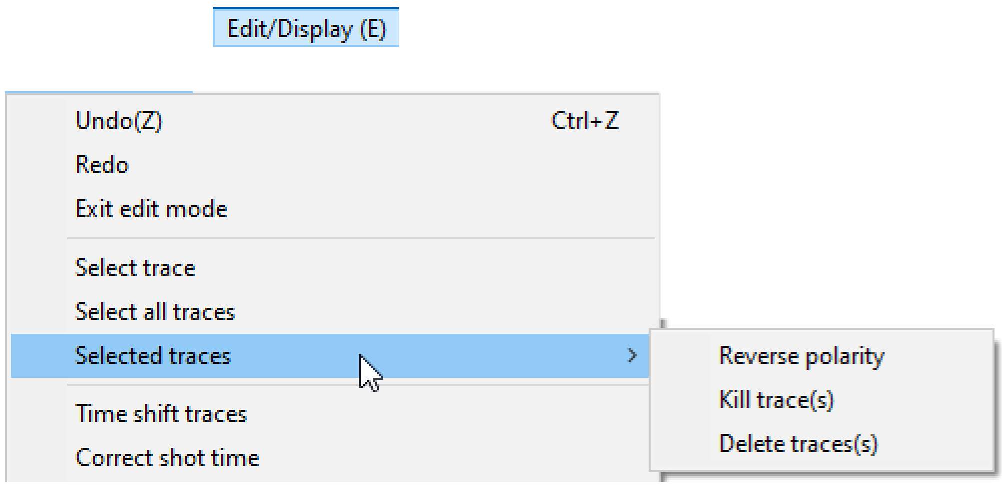


Figure 7: Trace selection. In the above figure, a contiguous group has been selected by clicking and dragging the mouse.

To de-select the traces, press the  button, or click on *Select trace* in the **Edit/Display** menu.

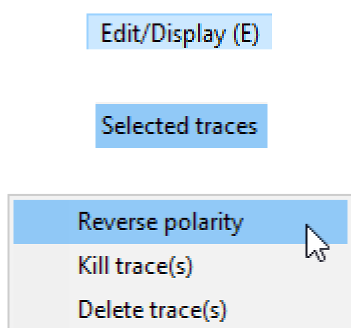
3.2.6 Selected Traces



When a trace or traces are selected for editing, the selected trace(s) can have the polarity reversed, can be killed, or deleted:

3.2.6.1 Reverse Polarity

To reverse the polarity of a selected trace(s), click on *Reverse polarity* in the sub-menu:



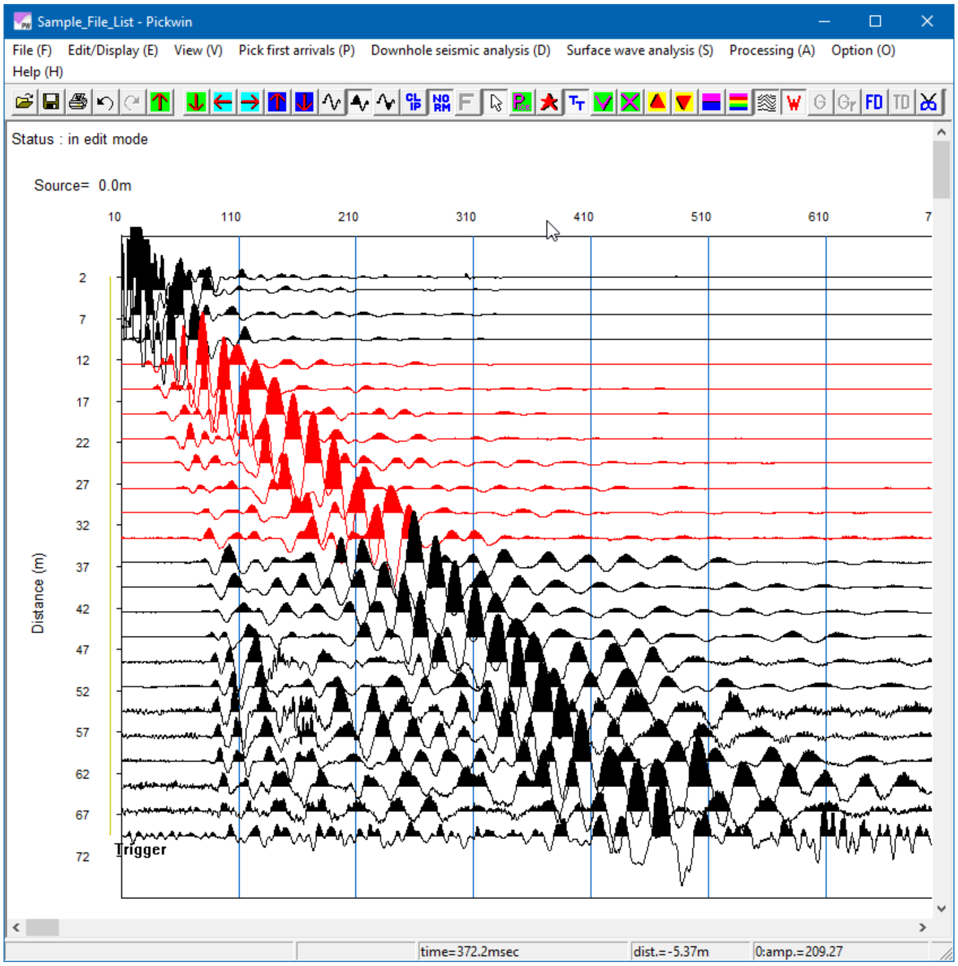
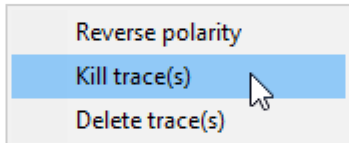


Figure 8: Polarity reversal. Red traces have been reversed.

Note that the polarity of the group of traces selected earlier has now been reversed.

3.2.6.2 KILL TRACE(S)

To “kill” a selected trace(s), click on *Kill trace(s)* in the sub-menu:



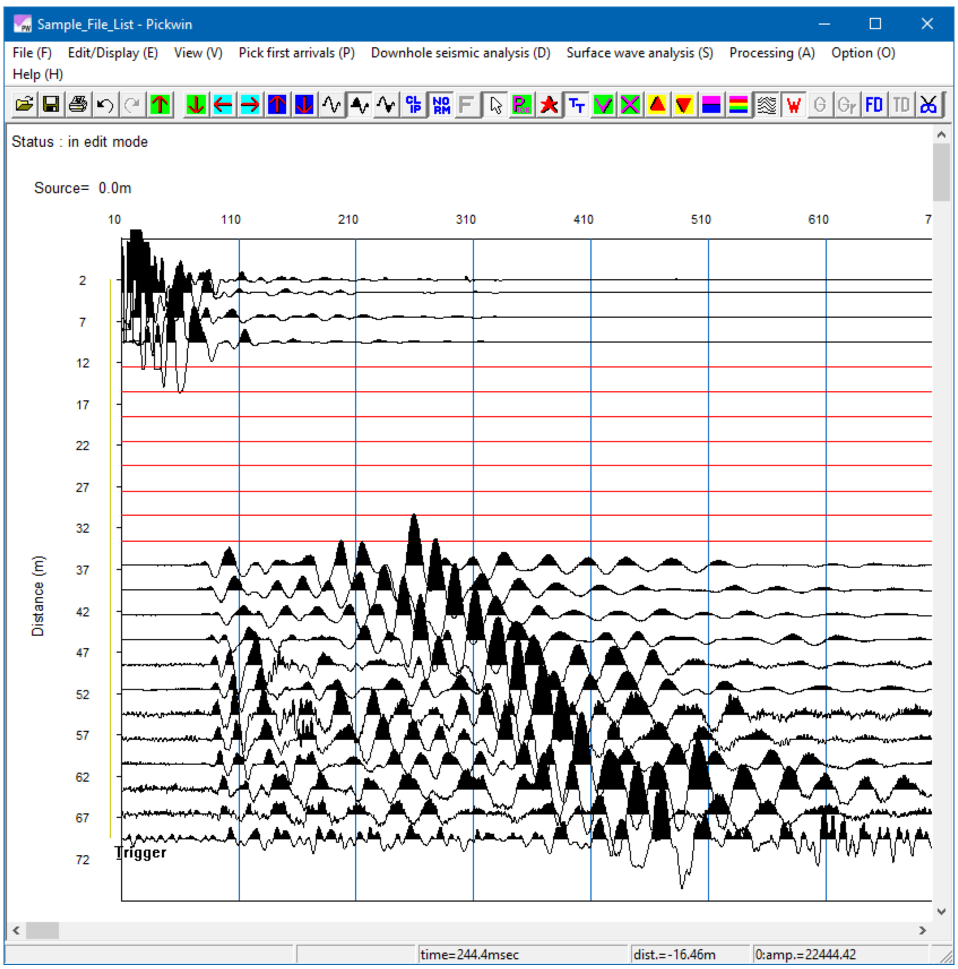
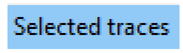


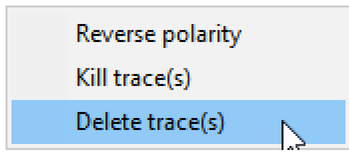
Figure 9: Trace kill. Red traces have been killed.

The selected trace(s) will now be “killed” (zeroed), as shown above.

3.2.6.3 DELETE TRACE(S)

To delete a selected trace(s), click on *Delete trace(s)* in the sub-menu:





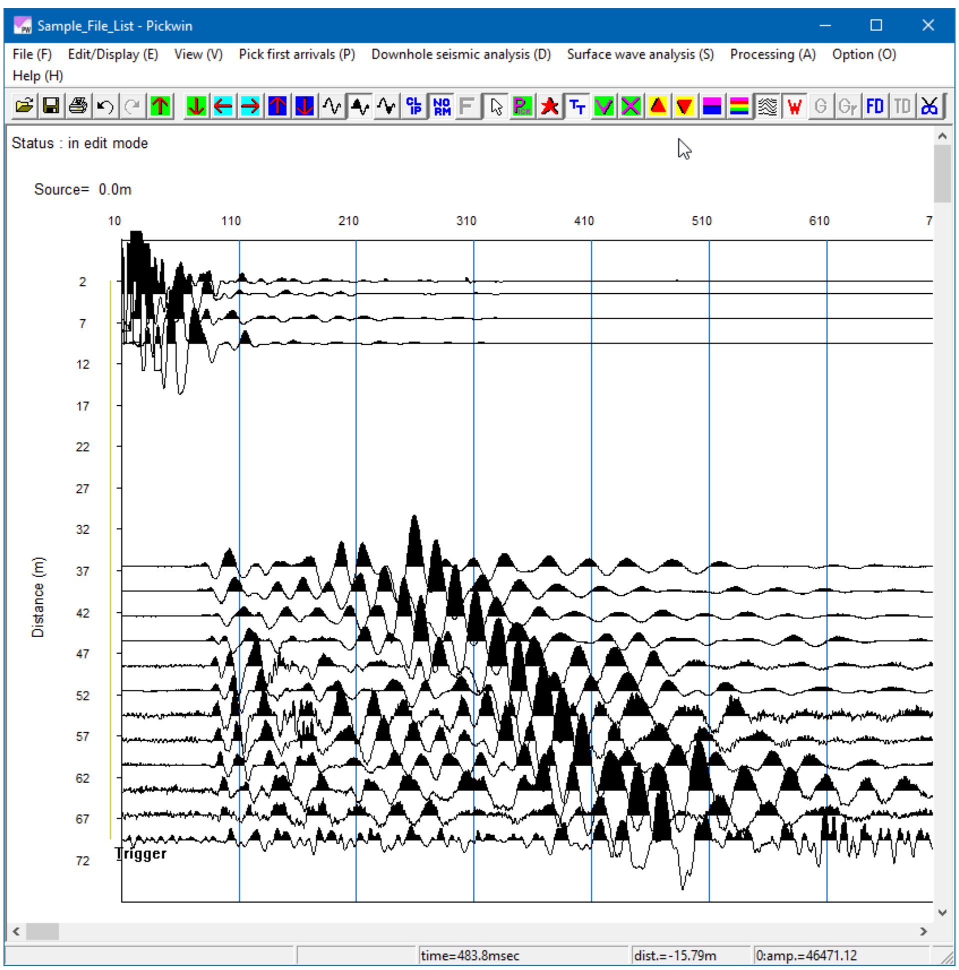
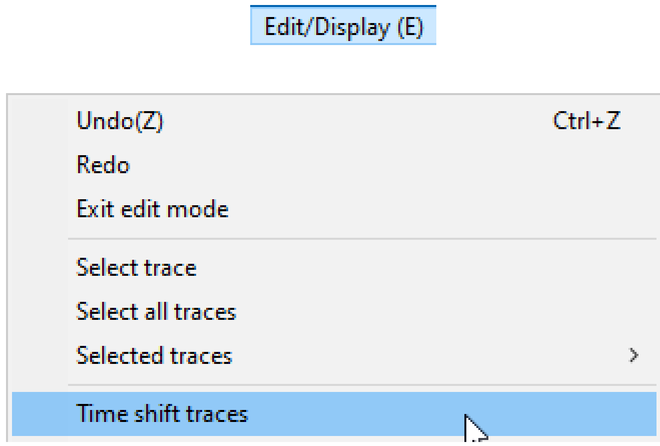


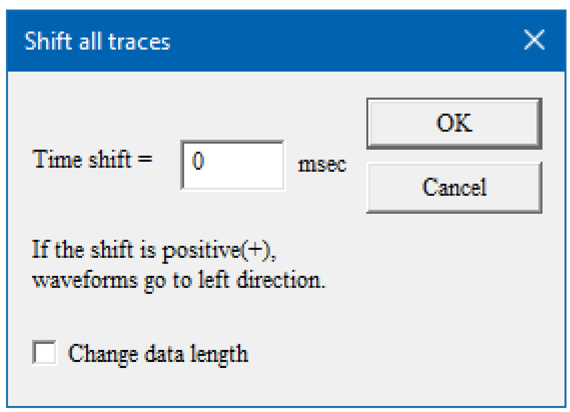
Figure 10: Trace deletion. Selected traces have been deleted.

The selected trace(s) will be deleted, as shown above.

3.2.7 TIME-SHIFT TRACES



To shift the time axis of all the traces, choose *Time shift traces*. You will see the following dialog box:



Choose an amount of time (in milliseconds) to shift the record and press *OK*. The record may be shifted in a positive or negative time direction. In the example shown below, a negative 100 msec shift has been applied.

If you wish the traces to be padded with zeros at the end, check the *Change data length* box.

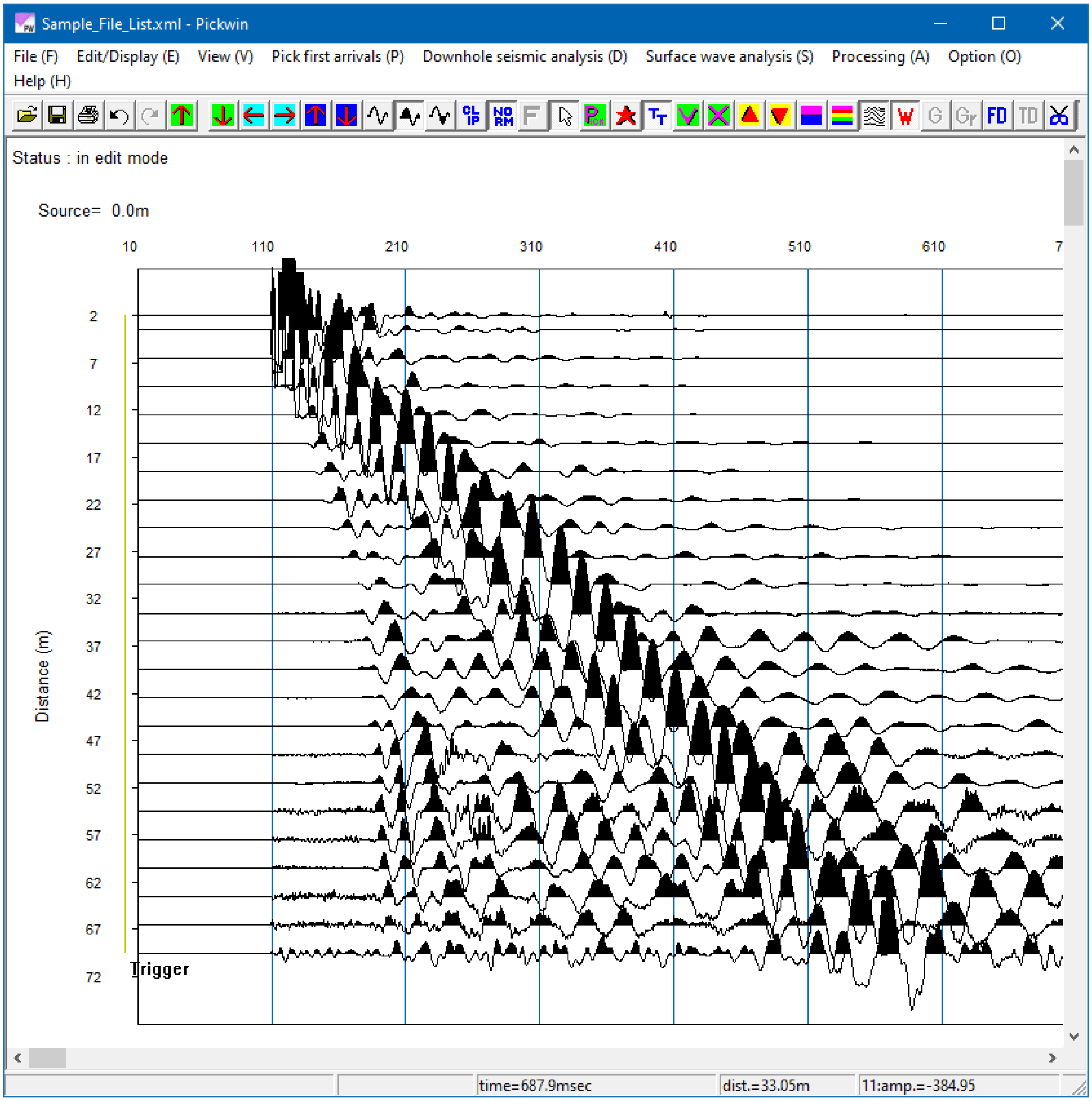
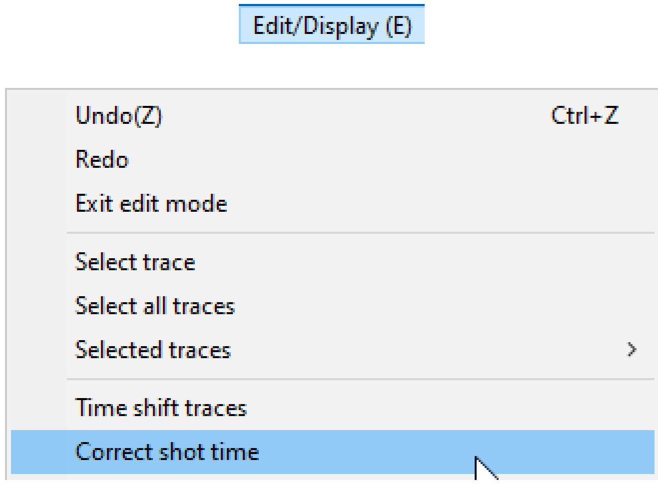


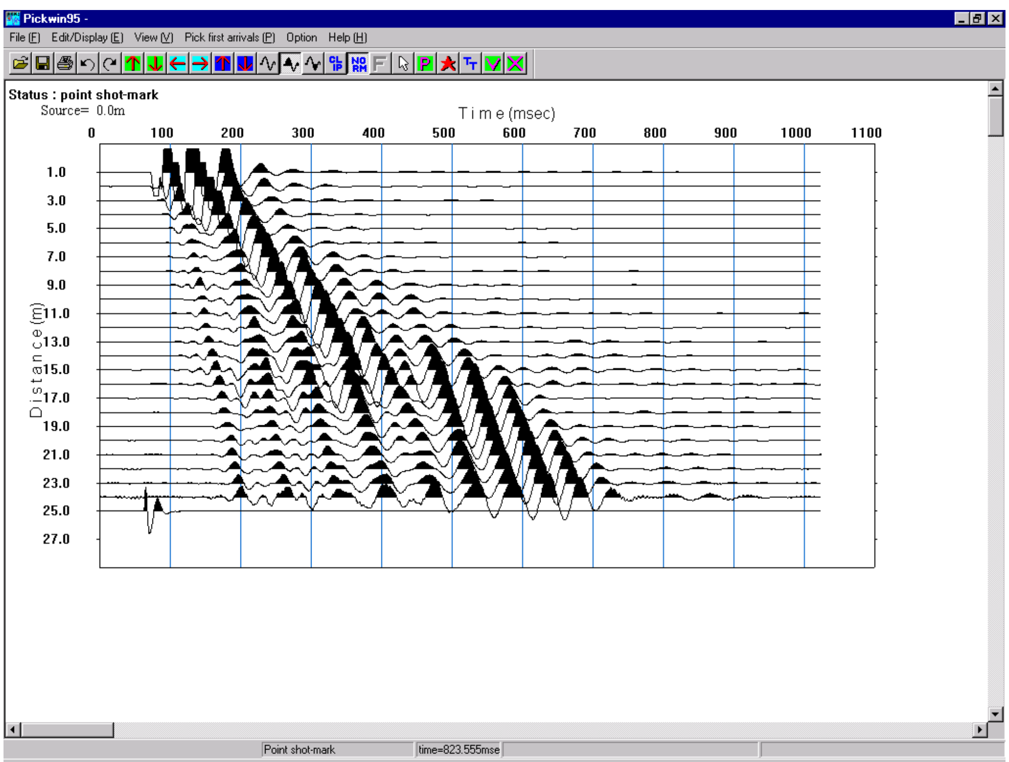
Figure 11: Trace time shift.

Note: *A positive (+) value will shift the record to the left and shorten the record time of the traces. A negative (-) value will shift the record to the right and increase the record time of the traces.*

3.2.8 CORRECT SHOT TIME



Depending on your triggering methodology, you may need to correct the time of the shot. This is often the case when you use a geophone to trigger the seismograph. In the example below, channel 25 has been connected to a geophone placed next to the source to record the actual source time (which, in this case, came about 80 msec after the seismograph triggered).



To correct the shot time, choose *Correct shot time* in the **Edit/Display** menu (notice that "point-shot mark" is now displayed in the editing status mode in the upper left-hand corner). Position the cursor along the time record to where you would like to set the correct time of the shot and click. The time-position of the cursor will be shown at the bottom of the window.

The traces will be adjusted for the corrected shot time, as shown below:

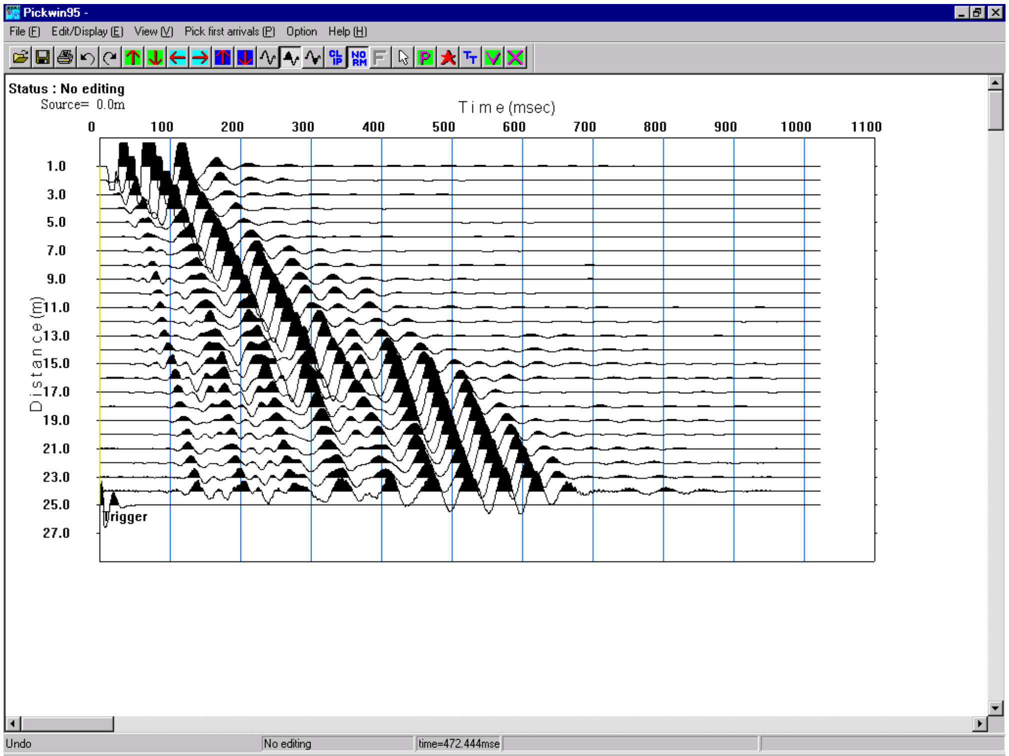
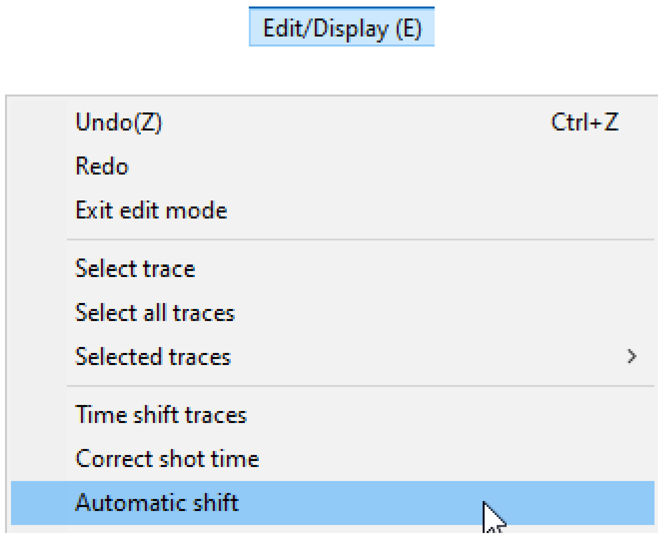


Figure 12: Corrected shot time.

3.2.9 AUTOMATIC SHIFT



Some styles of surveying require the ability to append shot records together. For instance, if the goals of your survey require more channels than are available, you may overcome this by laying out several individual spreads end-to-end, and re-occupying some or all the shot points. As an example, suppose you wish to do a 48-channel, five-shot spread, but you only have a 24-channel seismograph available. You may simulate a 48-channel spread (and seismograph) through the following procedure:

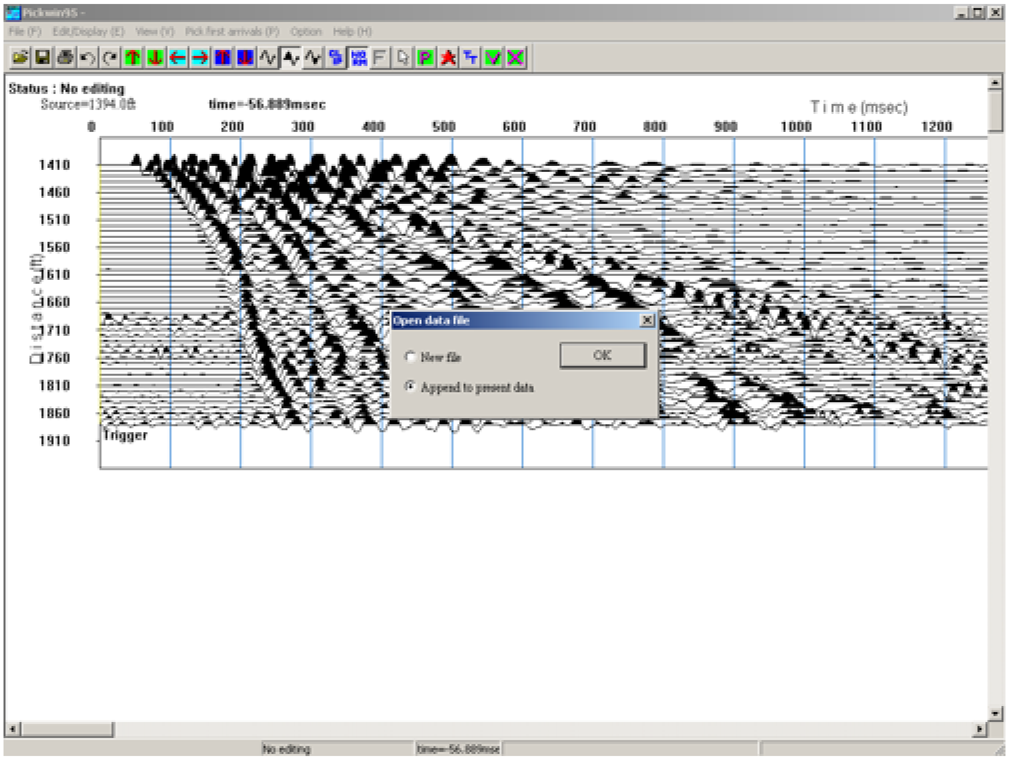
- Lay out the “left” half of the spread (all 24 of your channels).
- Do your five shots *as if the entire 48-channel spread* is on the ground.
- Pick up the 24 geophones and attendant cables and move them over to the “right” half of the spread.
- Re-occupy your five shots.

Once this has been completed, you have acquired the exact same data that you would have acquired had you laid out 48 channels all at once and simply done five shots. It’s the same result, but twice the work.

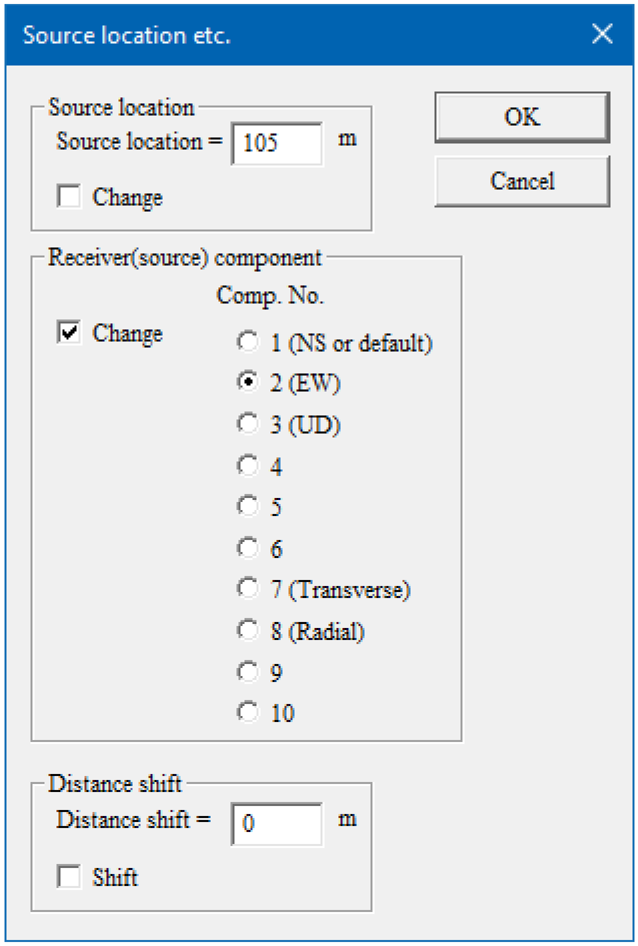
When conducting more than one shot at the same location, the physical properties of the earth can be altered, leading to slight differences in local velocities. This, in turn, can lead to a slight difference in travel times to equivalent geophone stations between the first and subsequent shots. This is especially true when using explosives. To account and correct for this, it is best to overlap one or two geophones when acquiring data in this fashion.

The *Automatic shift* in SeisImager/2D can then be used to correct for any change in travel times from one occupation to the next. This is demonstrated in the following example.

Read in the first half of the spread. Next, read in the second half. You will be asked if it is new data, or if you would like to append it to the present data:



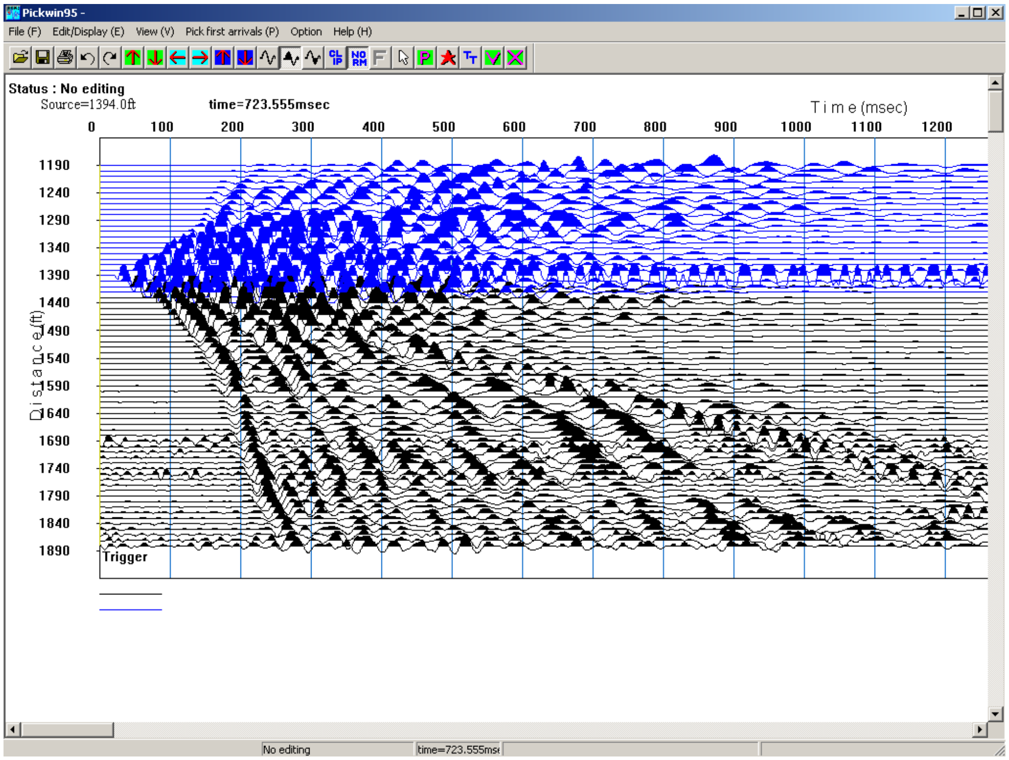
Choose *Append to present data*. Next, you will be presented with the following dialog box:



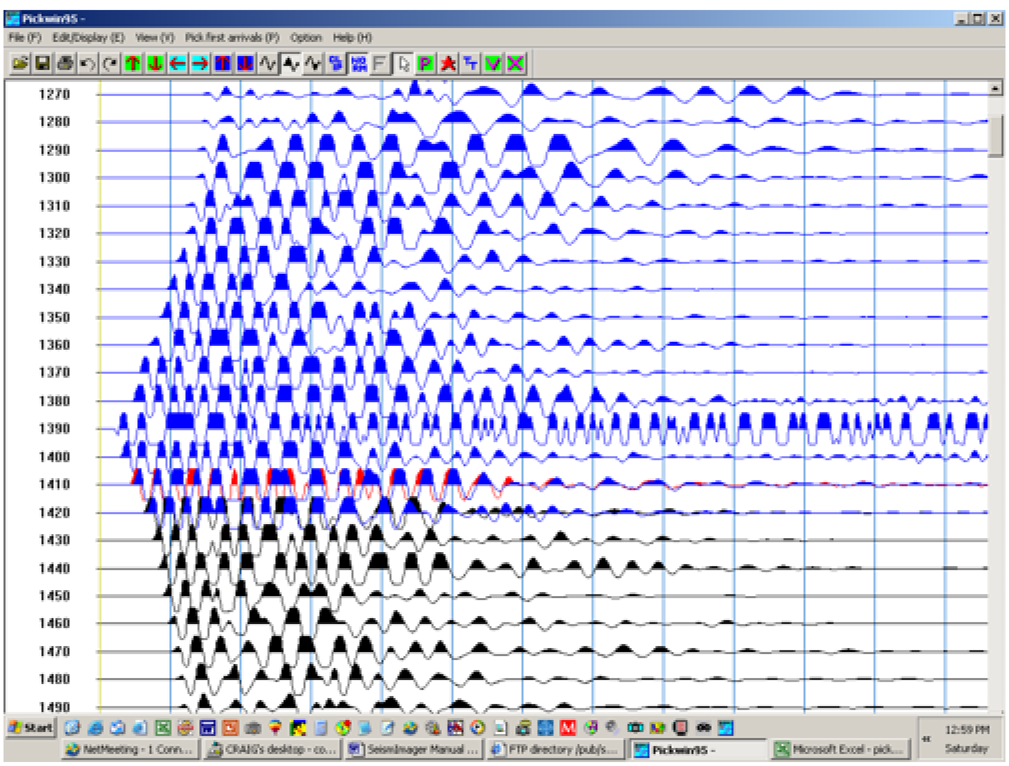
If you need to correct the source location (read from the SEG-2 file header), you may do so here. The component number ("Comp. No.") is used to keep different spreads separate and will automatically increment each time you append a new file. In this case, the component number defaulted to "2" (ignore the "EW"). You may append up to 10 files (but see discussion beginning with Section 3.1.13.1, Page 33).

Note: *The* ***Change*** *check box* ***must be checked*** *for any changes you make in the above menu to actually take effect.*

Press *OK* and the next file will be appended to the first:



If we zoom in, we can see that we have overlap at station 1410:



Note that there is a slight time shift between the two. To eliminate this, select one of the overlapping traces (the red trace shown above), and then select *Automatic shift*:

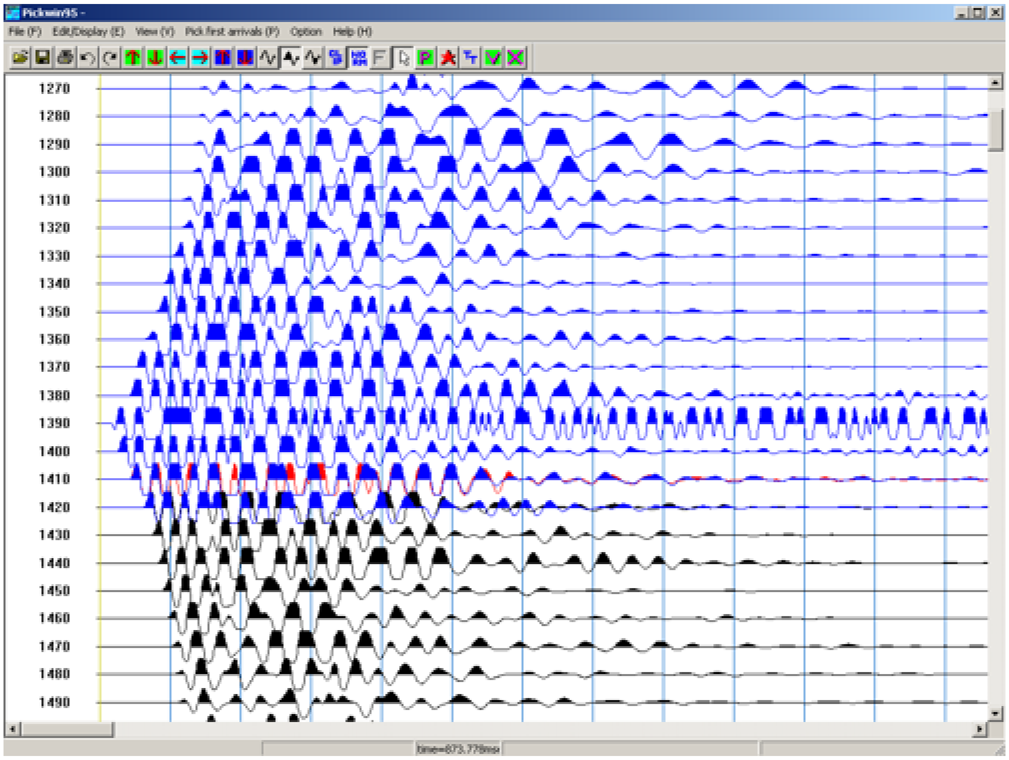
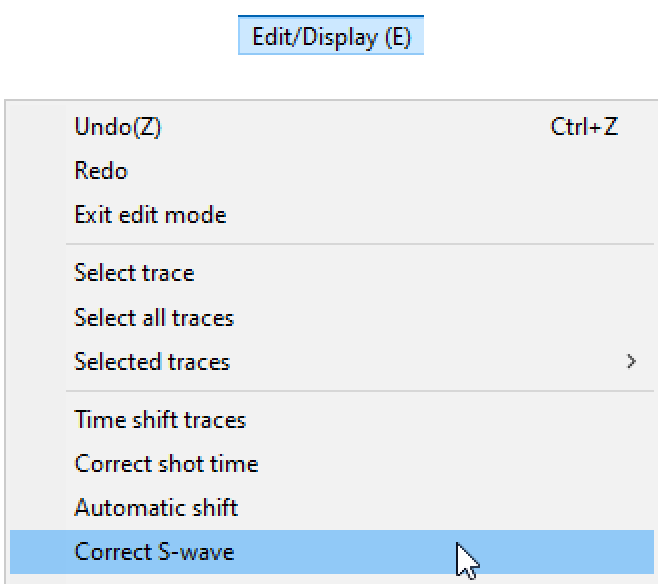


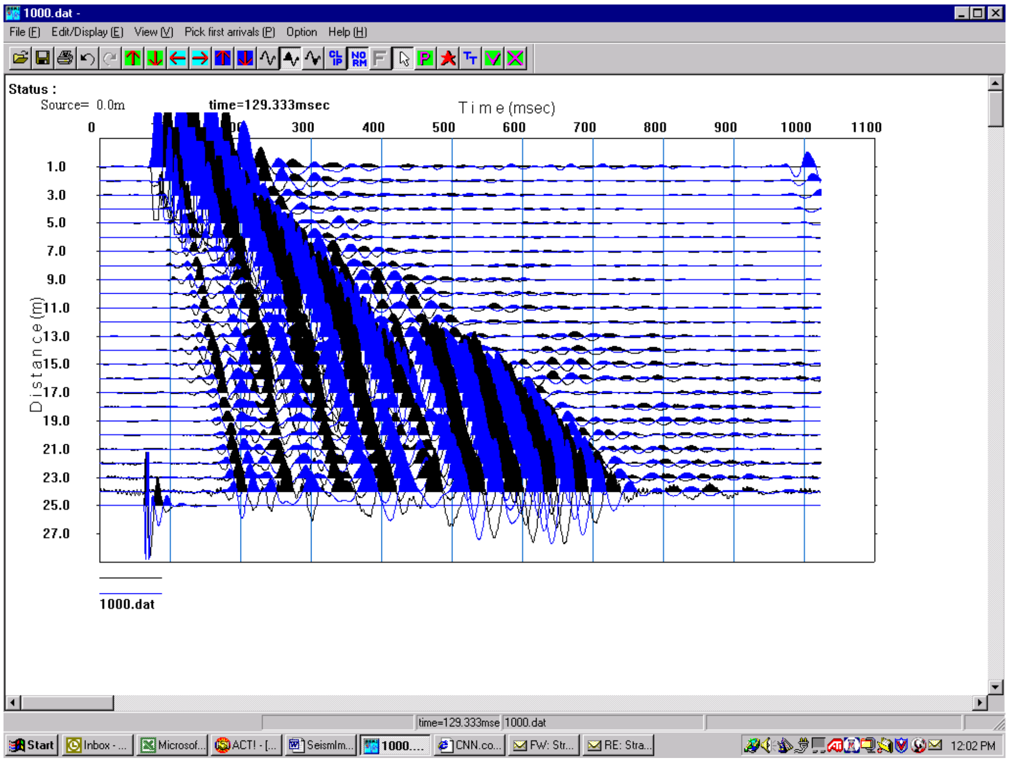
Figure 13: Trace time shift.

The entire second spread has been shifted in time to correspond with the first.

3.2.10 Correct S-Wave

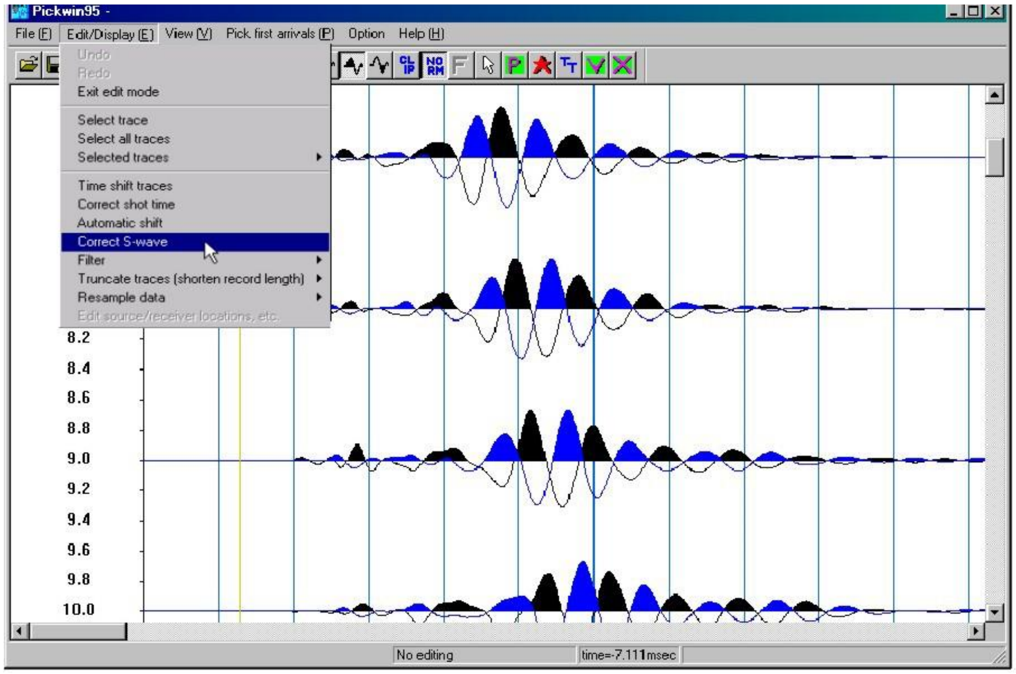


When doing a shear or "s-wave" survey, it is common practice to do reverse-polarity shots to facilitate the identification of shear wave arrivals. It is useful to overlay reverse-polarity shots from the same shot point. This can be done by reading in the first shot, and then appending the second, resulting in an overlay like the one shown below:



Ideally, the first shear wave arrival times will be identical for both records. However, it is often the case that they are not – one is often shifted slightly in time. This is quite common when the shear wave source consists of a long plank of wood or other non-point source.

To correct the s-waves to coincide at the same arrival times, click on *Correct S-wave* in the **Edit/Display** menu:



A cross-correlation of the oppositely polarized traces will be done in an attempt to better align first breaks. An example of the effect is shown below:

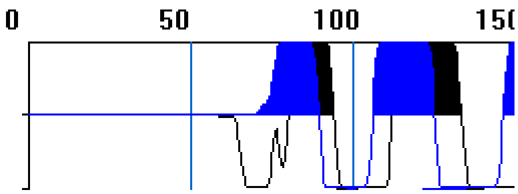


Figure 14: Uncorrected shear wave pair.

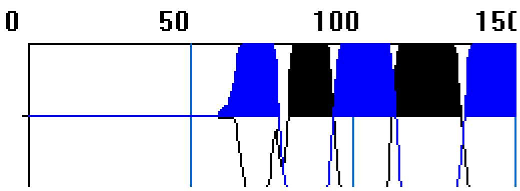
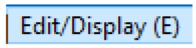
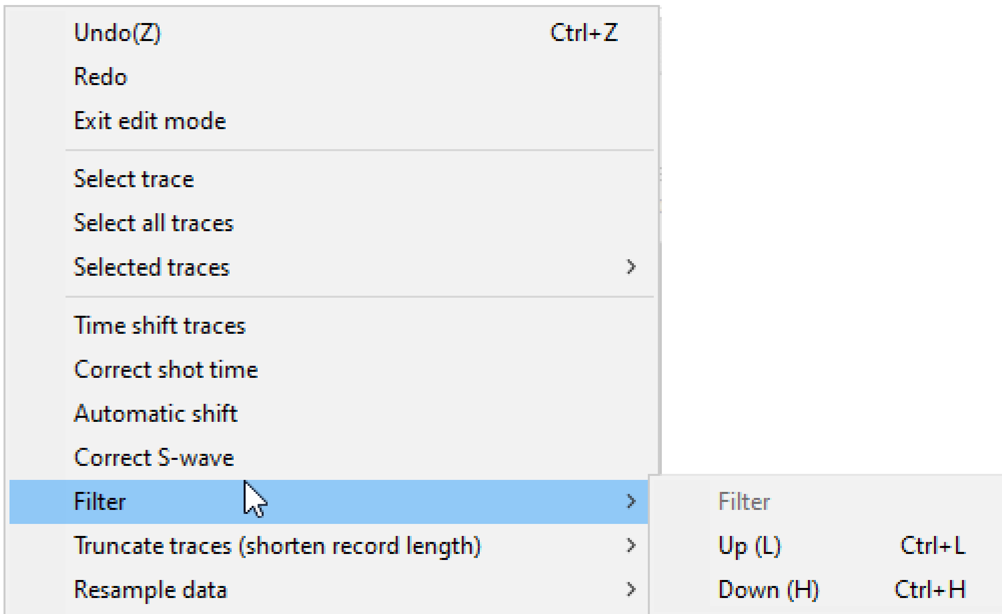


Figure 15: Shear wave pair aligned by cross-correlation.

3.2.11 FILTER 





Filters can be used to remove noise caused by wind, traffic, and other sources. You may apply high-cut filters and/or low-cut filters. To apply a 1000 Hz high-cut filter, press *CTRL+H*. Each subsequent press of *Ctrl+H* will multiply the corner frequency by 0.8, so that the second press applies an 800 Hz filter, the third press applies a 640 Hz filter, and so on.

To set a 5 Hz low cut filter, press *Ctrl+L*. In a similar fashion to that described above, each subsequent press increases the corner frequency by 1.5.

Below is an unfiltered record with some high-frequency noise in the early part of the record:

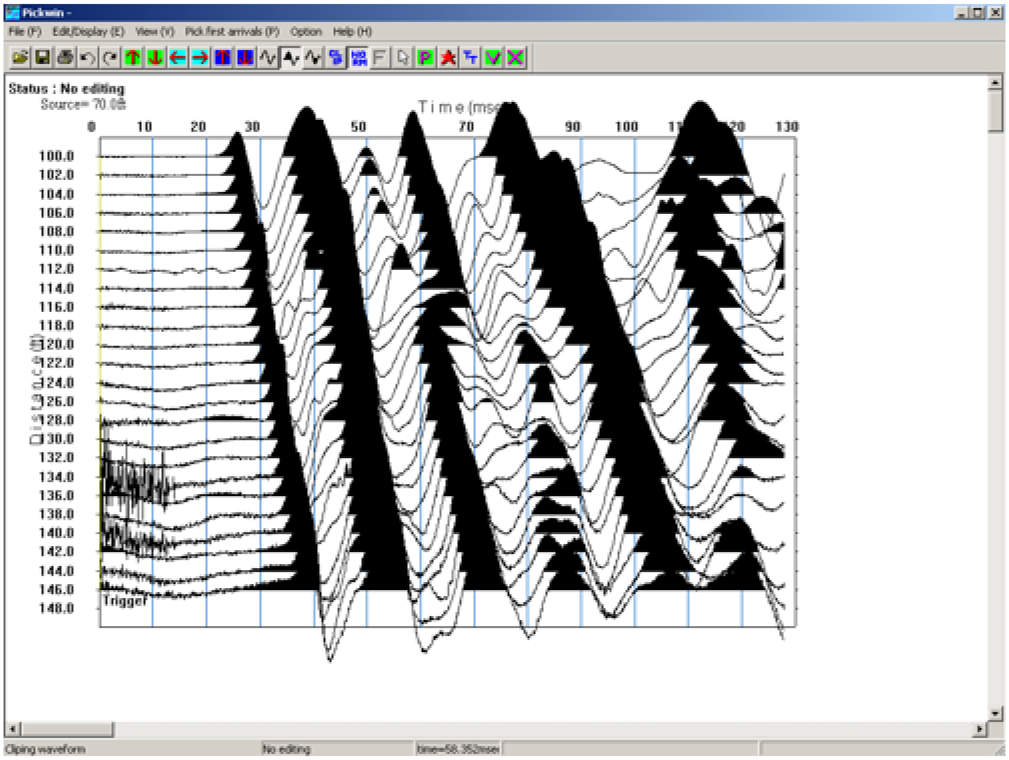


Figure 16: Unfiltered refraction record.

Here is the same record after applying a 512 Hz high-cut filter (four presses of *Ctrl+H):*

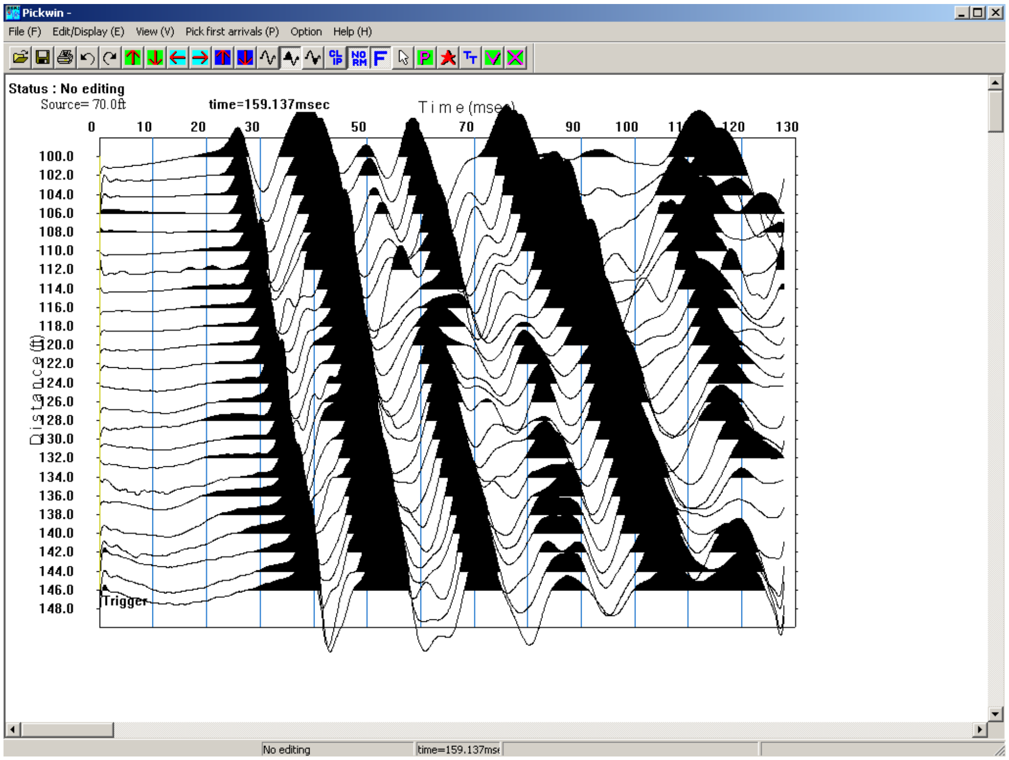


Figure 17: Filtered refraction record.

Below is an unfiltered record with some low-frequency noise in the far channels:

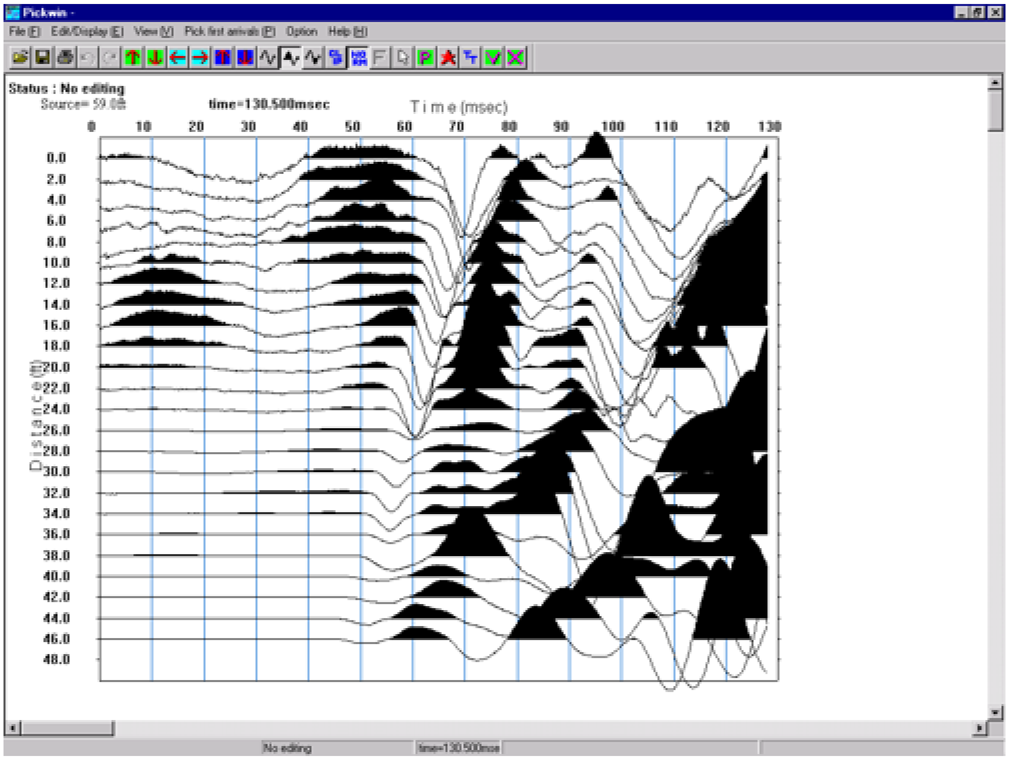


Figure 18: Unfiltered refraction record.

Here is the same record after applying a 38 Hz low-cut filter (six presses of *Ctrl+L):*

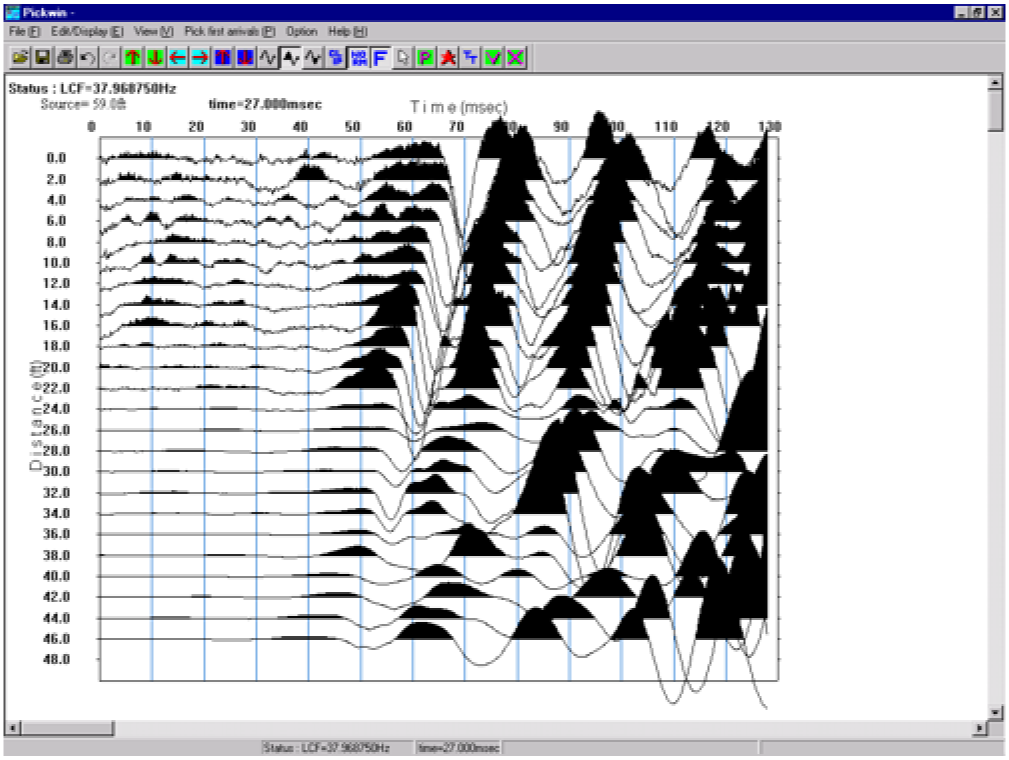


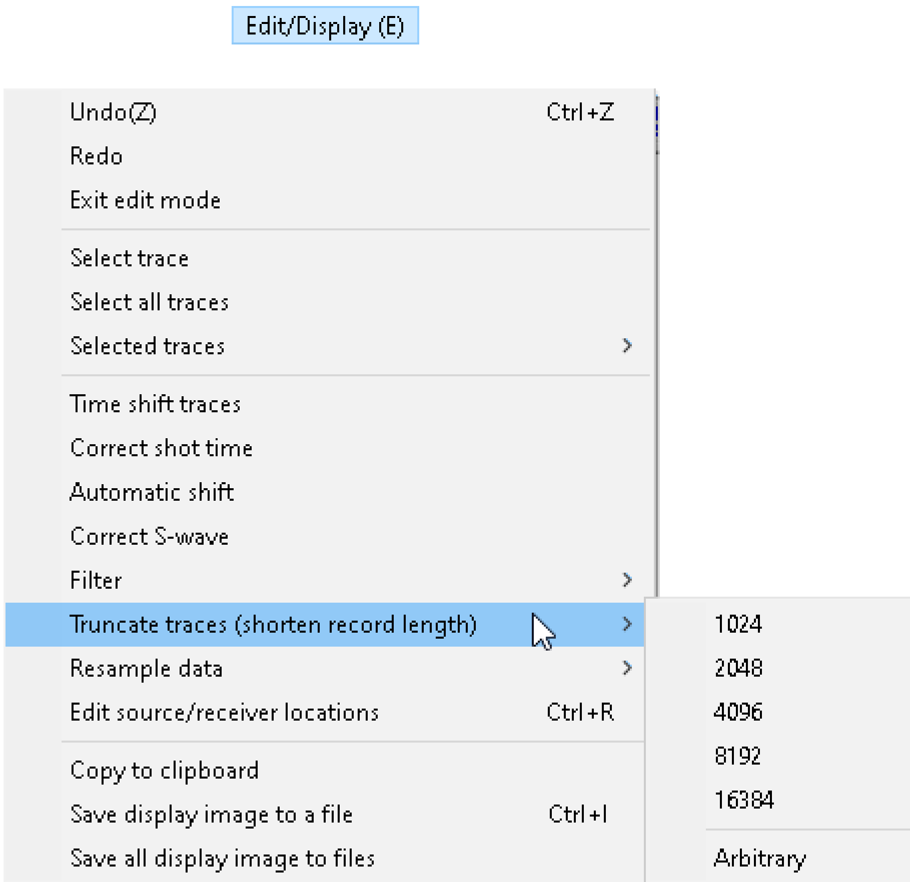
Figure 19: Filtered refraction record.

To disable all filters and return to the raw data, press the  tool button.

See Section 3.7.1, Page 93, for more on filtering.

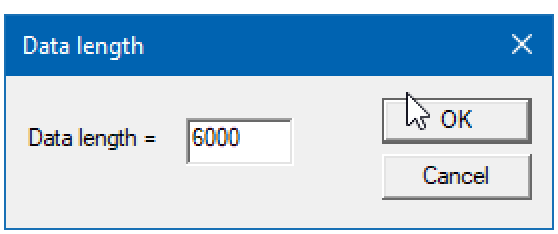
Note: *Use caution when applying filters. If the cutoff frequency is too close to the dominant frequency of the first arrivals, the arrivals will look like they are coming in later than they actually are.*

3.2.12 Truncate Traces (Shorten Record Length)



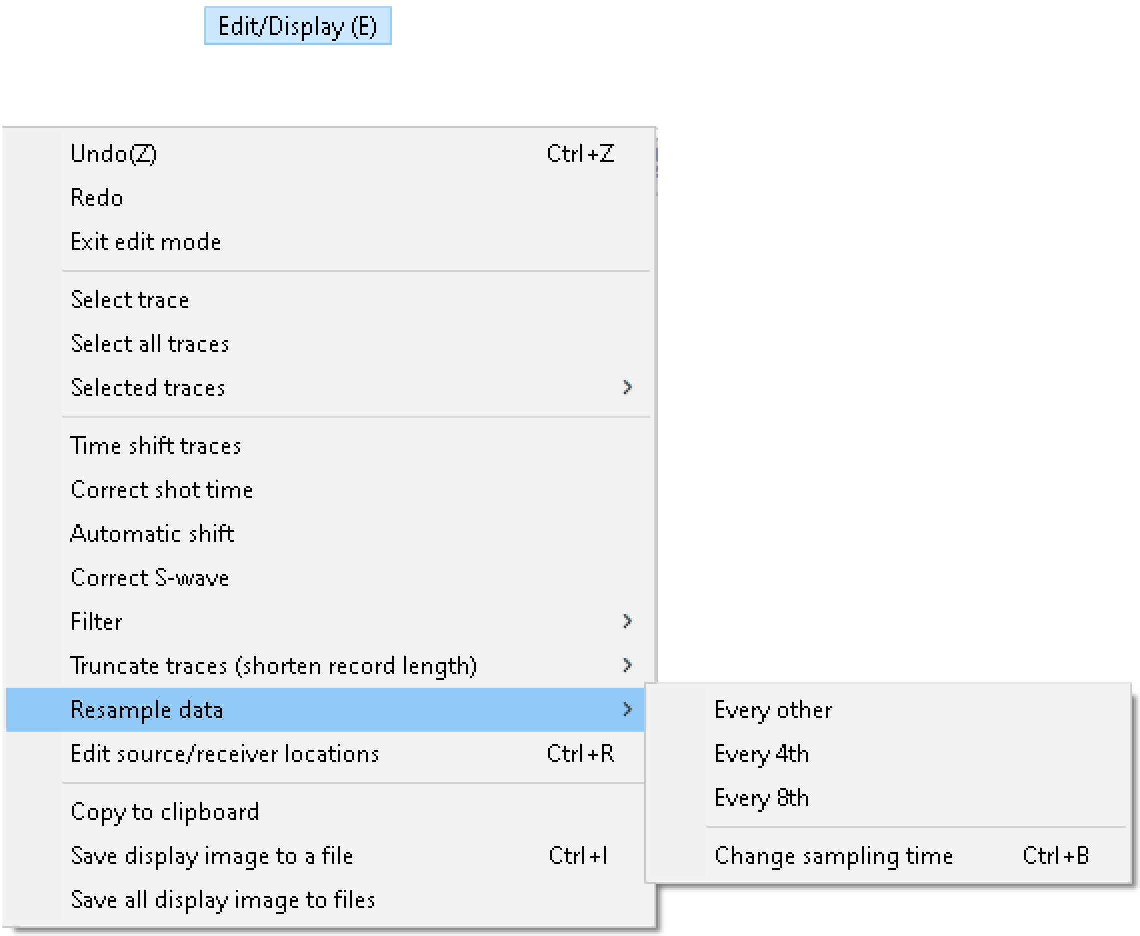
To shorten or truncate the record length, click on *Truncate traces (shorten record length)*. A sub-menu with the Pickwin default options for truncating traces will be displayed.

Click on a default truncation of 1024, 2048, 4096, 8192, or 16384 samples to truncate the traces to the respective record length. Clicking on *Arbitrary* allows you to specify an arbitrary trace length. If an arbitrary truncation of a trace is chosen, the following dialog box will be displayed:



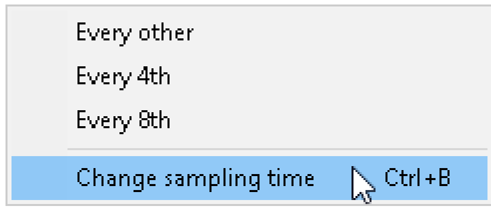
Type the desired data length in number of samples for the traces. Click *OK* and the traces will be shortened accordingly. This can be useful if the record length was much longer than it needed to be to record first breaks.

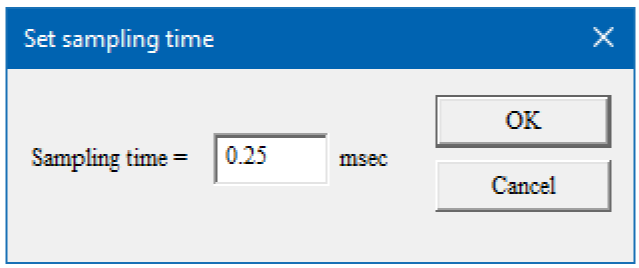
3.2.13 RESAMPLE DATA



To resample data, click on *Resample data*. From the sub-menu, click on one of the default re-sampling options: *Every other*, *Every 4th*, or *Every 8th*. This can be useful if the data has been oversampled and you wish to make the data files smaller.

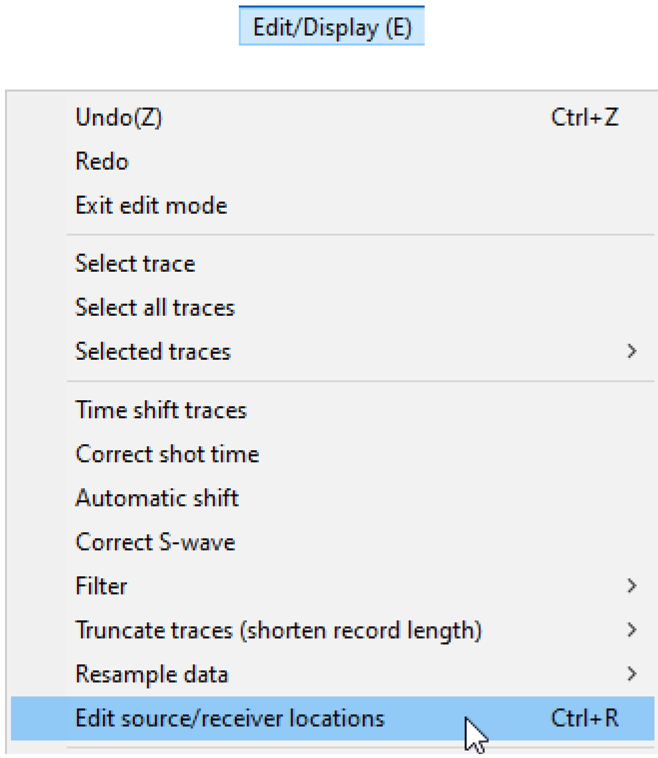
3.2.13.1 CHANGE SAMPLING TIME [CTRL+B]



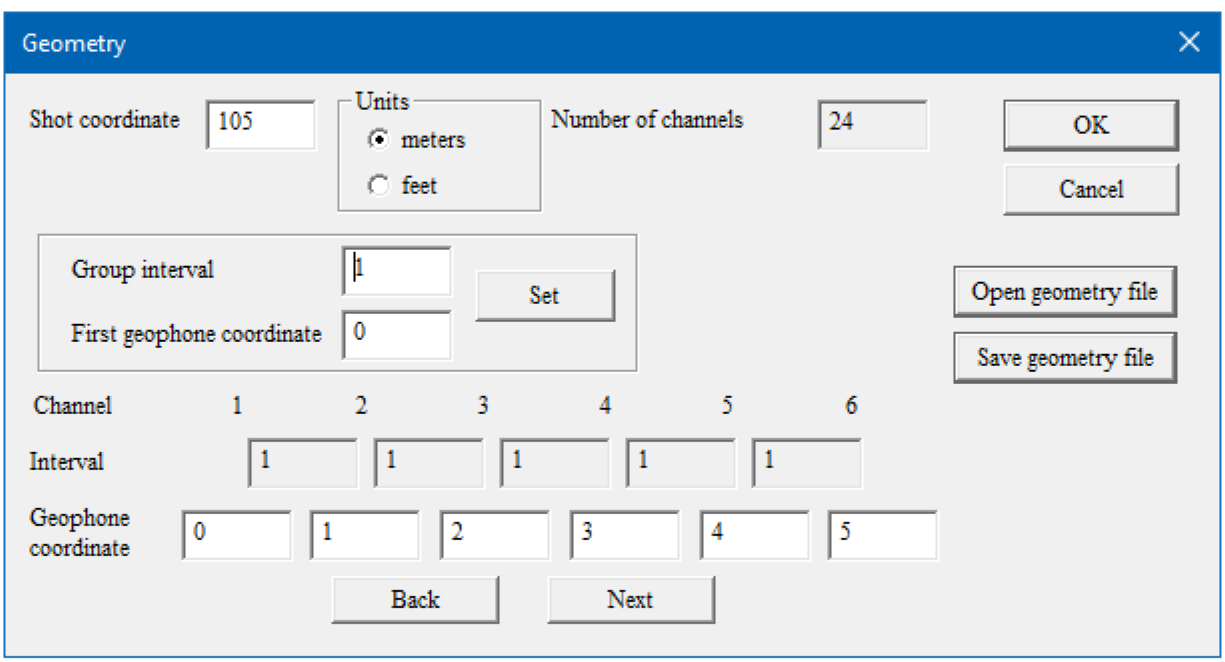


This feature allows you to resample the waveform at whatever rate you wish. This can be useful if the waveform file is in ASCII format and the sampling rate is (somehow) incorrect. Example: If you have a record containing 1024 samples and the sampling interval is 1 msec, the record length is 1.024 seconds. If you change the sampling time to 2 msec, the record length will be 2.048 seconds.

3.2.14 EDIT SOURCE/RECEIVER LOCATIONS, ETC. [CTRL+R]

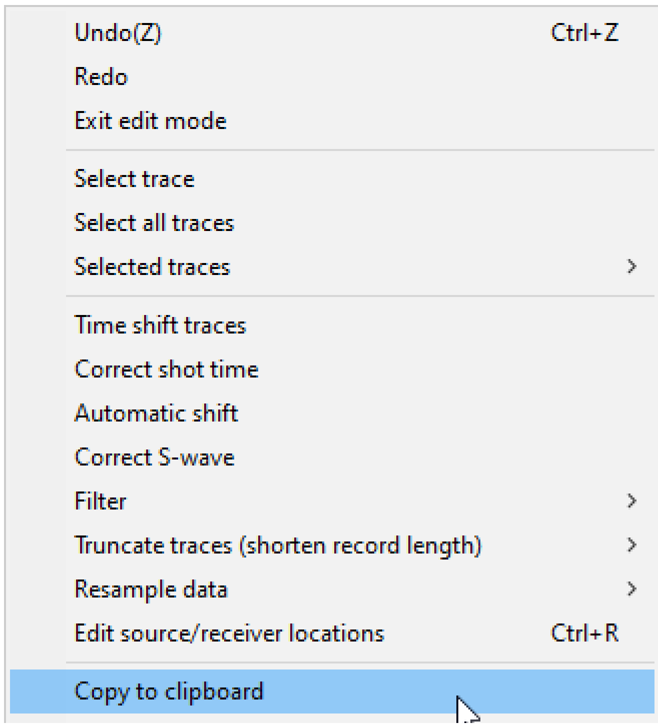


To edit source and receiver locations of the file to reflect that of the actual survey, click on *Edit source/receiver locations, etc.* or press *Ctrl+R.* The following dialog box, with appropriate parameters respective to your survey, will be displayed:



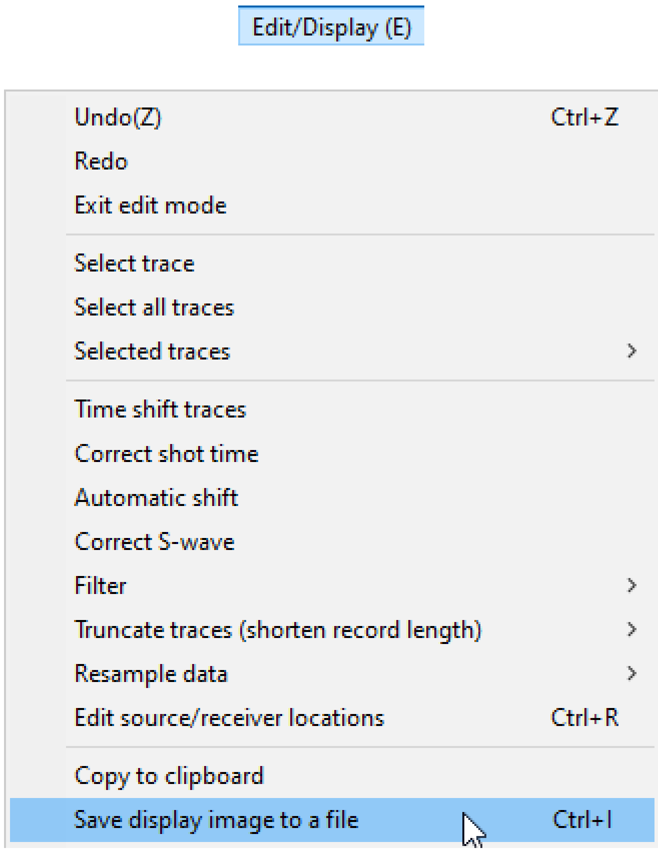
Edit the geometry of the survey by clicking in a box and typing in the new value. If you change the *Group interval* or *First geophone coordinate*, you must press the *Set* button to affect the change. Only six geophones are displayed at a time. Use the *Back* and *Next* buttons to scroll through the geophones. Click *OK* when changes are complete.

3.2.15 COPY TO CLIPBOARD



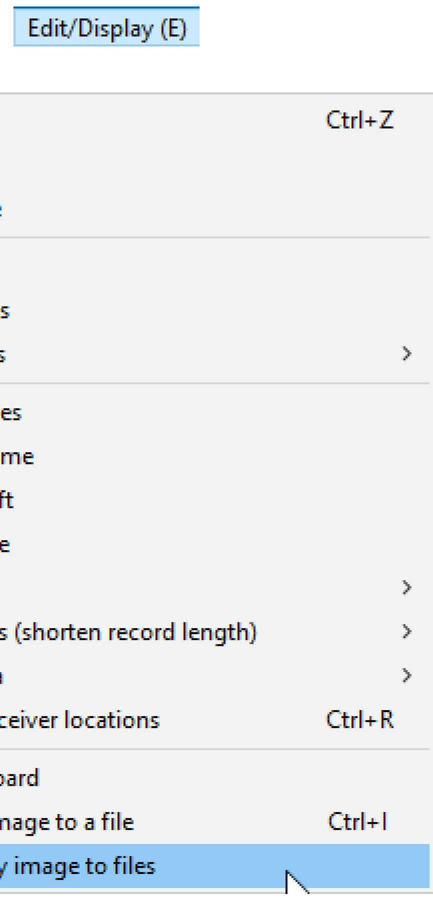
This feature copies whatever is on the screen to the clipboard. It can then be pasted into any third-party application, such as Microsoft Word.

3.2.16 SAVE DISPLAY IMAGE TO A FILE [CTRL+I]



This feature saves whatever is on the screen to a PNG, JPG, BMP, or GIF file.

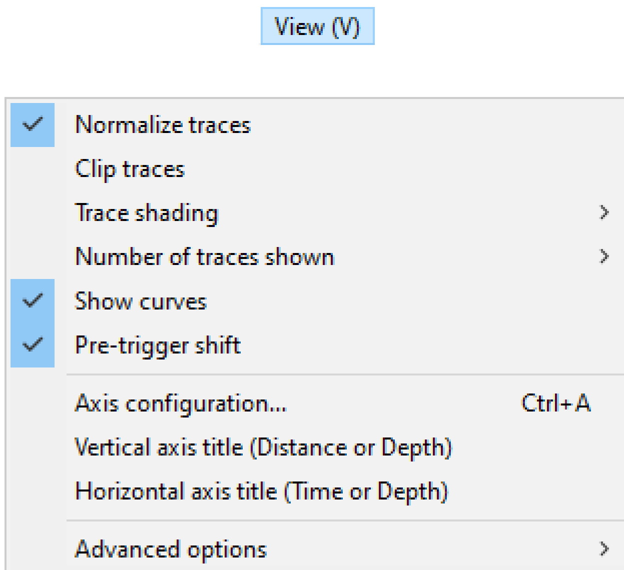
3.2.17 SAVE ALL DISPLAY IMAGES TO FILES



Please refer to the separate SeisImager/SW manual for explanation of this menu item. It is not used for refraction.

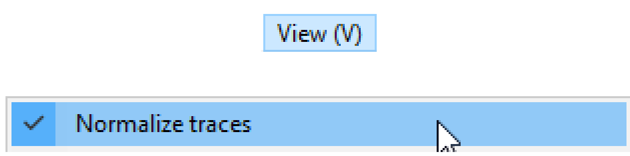
3.3 View Menu

Click on *View* to reveal the **View** menu:



Many of the features in this menu are toggle switches – clicking on them either enables (signified by a "✓" next to the selection) or disables the feature. Most toggle switch items also have buttons on the tool bars, and all work the same way. In the discussion below, toggle switches are identified by a "✓" and their tool bar button, if they have one.

3.3.1 Normalize Traces ✓ []



When traces are normalized, the maximum amplitude of each trace will be equalized. Lower-amplitude traces (those farther from the source) will be "turned up" so that their maximum amplitude is equal to that of higher-amplitude traces. This has the effect of optimizing the appearance of the first breaks across the record and is recommended for picking first breaks.

An example of normalized traces is shown below, followed by a record with normalization disabled.

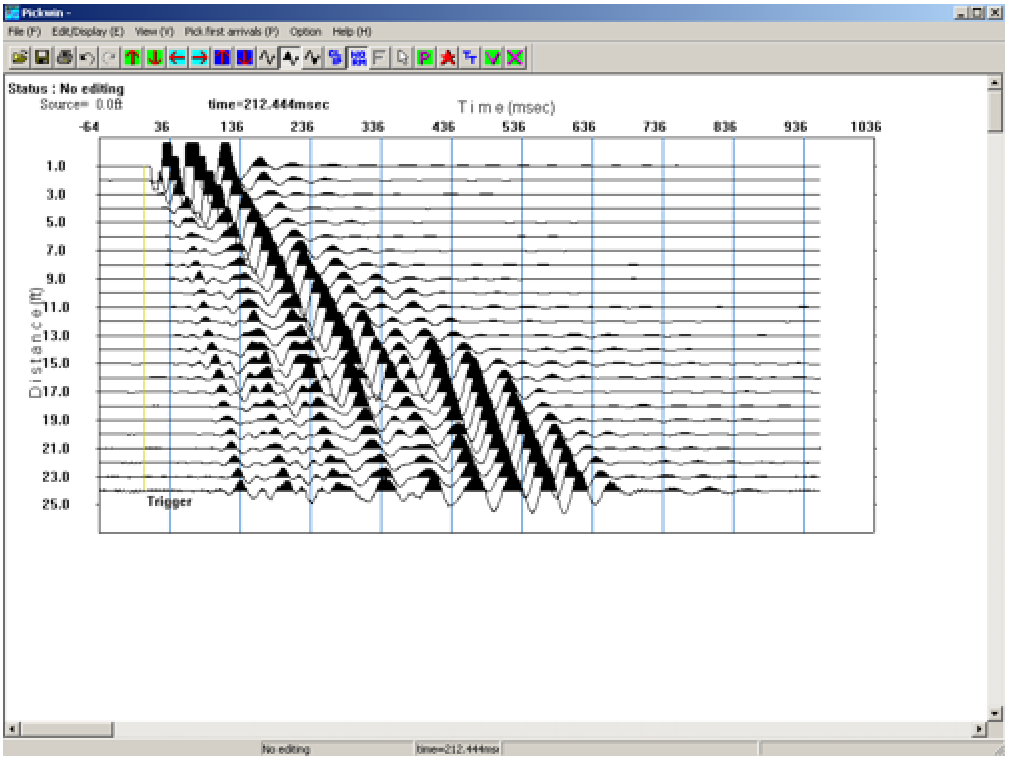


Figure 20: Refraction record with trace normalization applied.

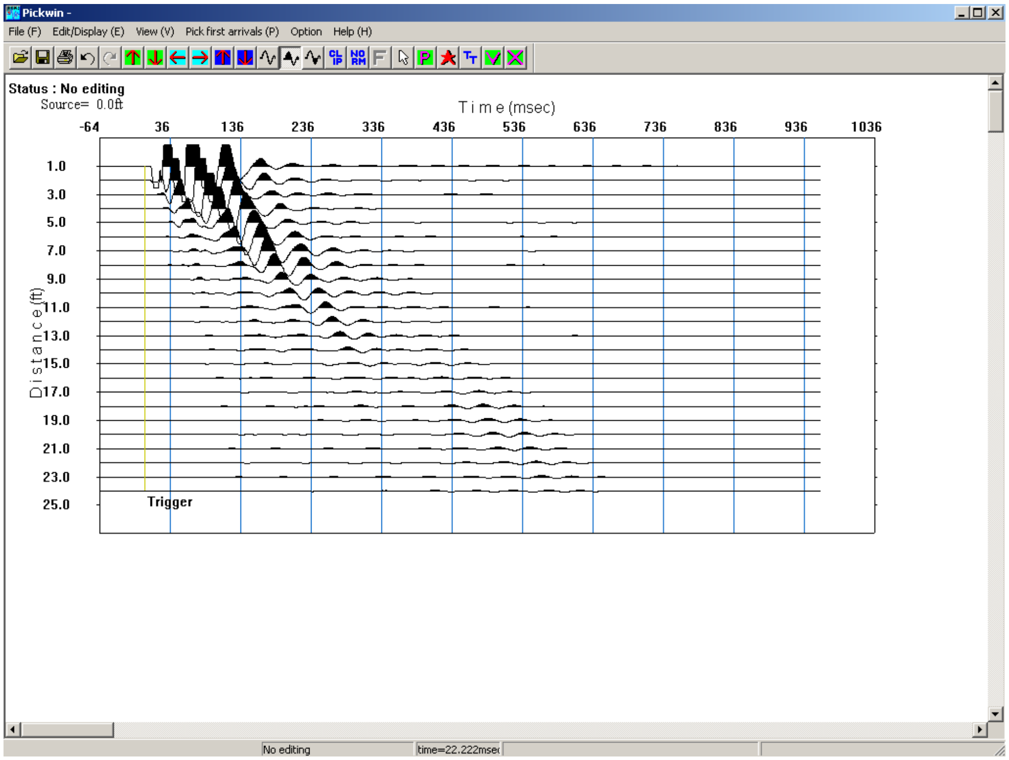
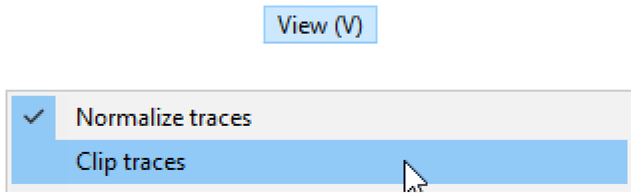


Figure 21: Refraction record with trace normalization disabled.

3.3.2 Clip Traces ✓ [ ]



The *Clip traces* feature is useful in preventing adjacent traces from interfering with each other and obscuring the first breaks. An example of clipped traces is shown below.

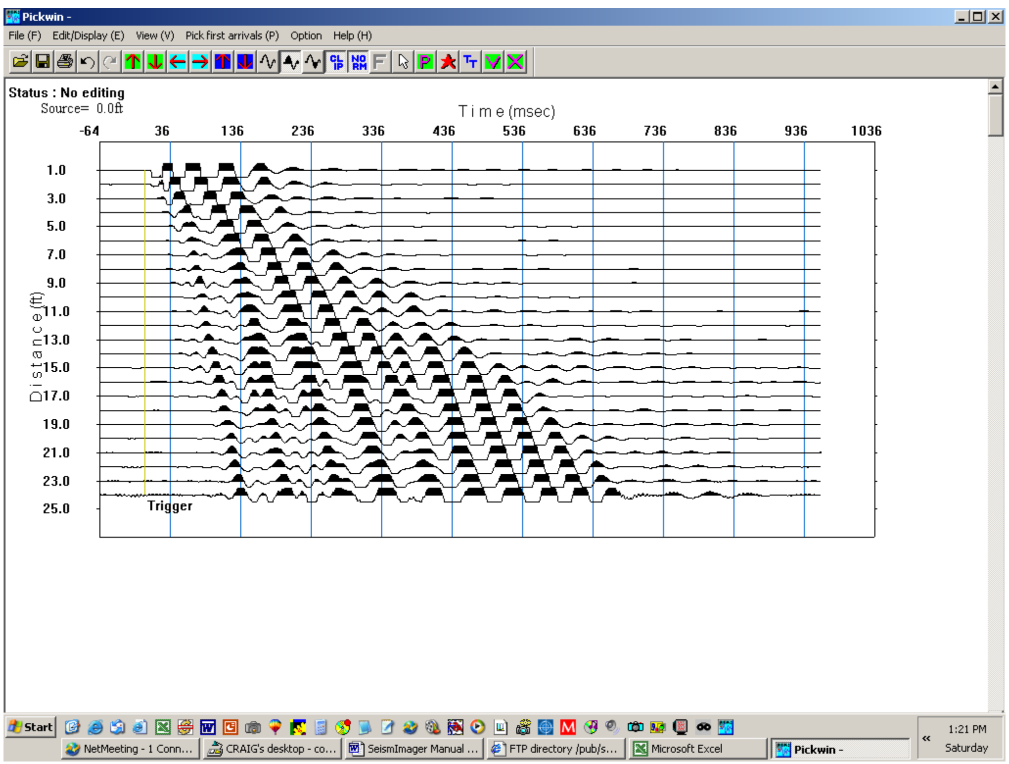
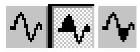
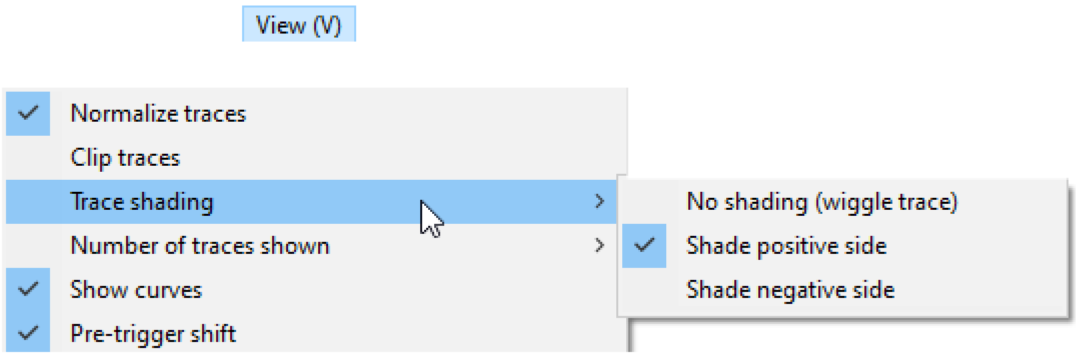


Figure 22: Refraction record with trace clipping applied.

3.3.3 TRACE SHADING []



In addition to the positive-shaded trace display used in the previous examples, you may also shade negative amplitudes, or seismic traces may be displayed as simple wiggle traces. The trace style may be changed via the **Trace shading** sub-menu or with the appropriate tool buttons shown above.

An example of negative amplitude shading is shown below, followed by a wiggle-trace plot.

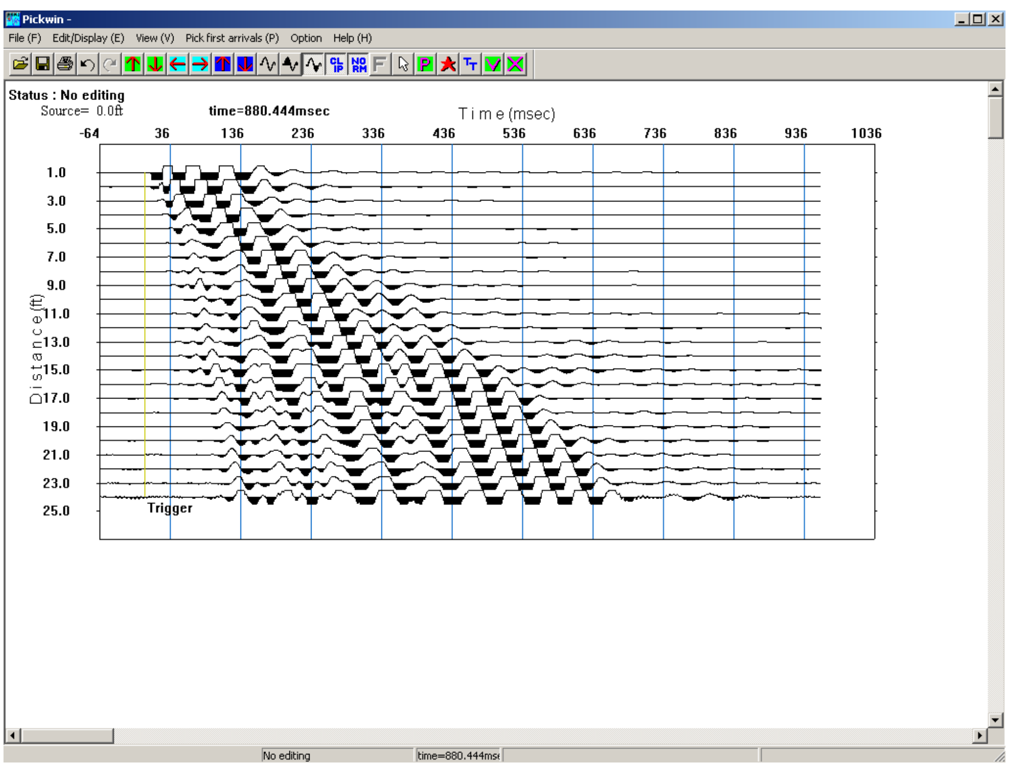


Figure 23: Example of negative trace shading, with clipping applied.

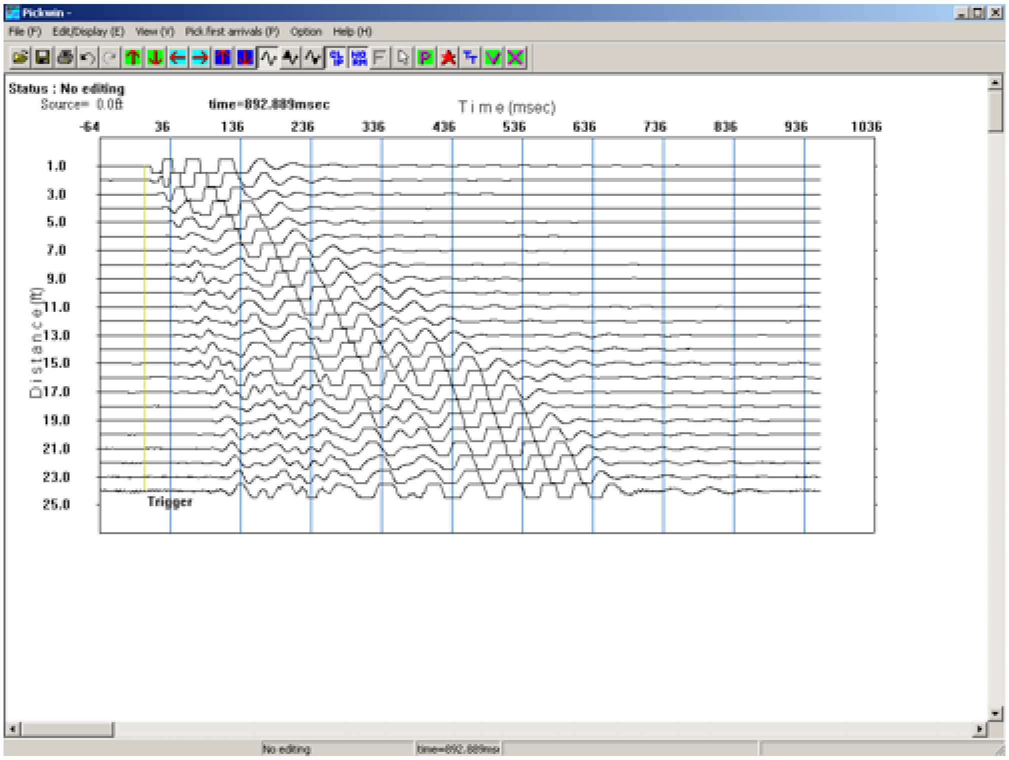
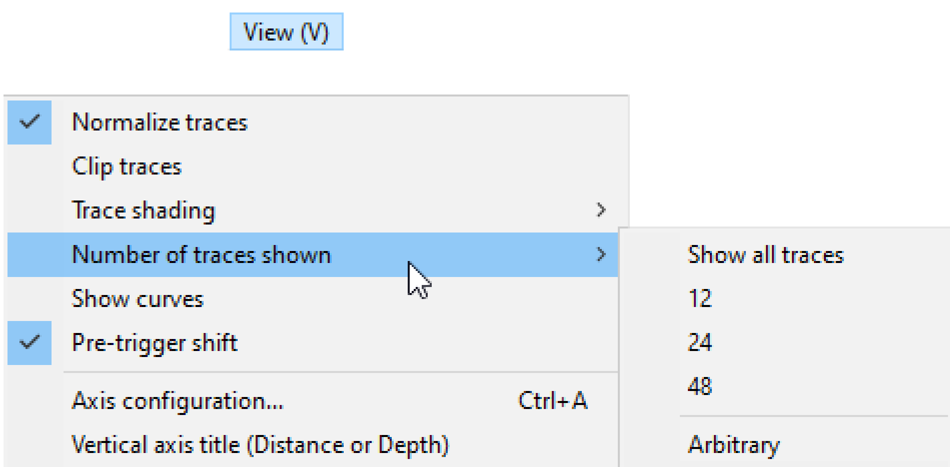


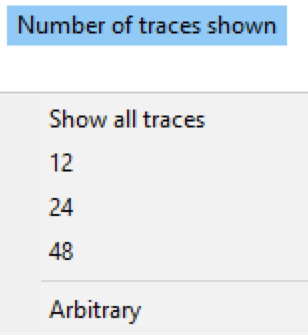
Figure 24: Wiggle trace display.

3.3.4 NUMBER OF TRACES SHOWN



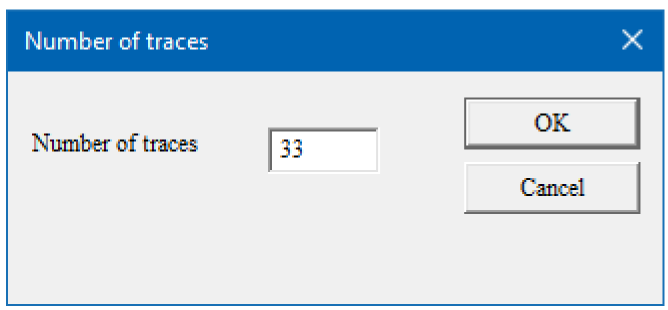
Note: *Pickwin displays all the seismic traces of a file by default.*

To change the number of traces that are displayed, click on *Number of traces shown*. The following sub-menu will appear with the following choices:



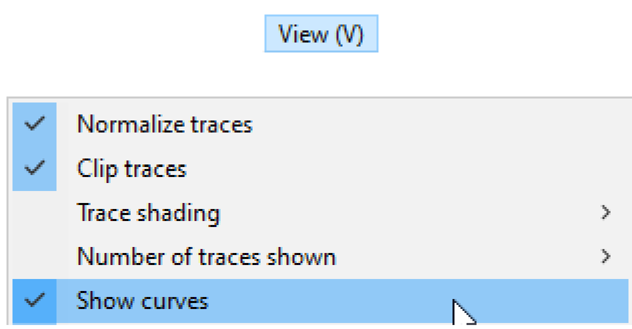
Select the number of traces you wish to display. Note that whatever is chosen, trace number 1 will be the first trace displayed. For instance, if you choose *12*, traces 1-12 will be displayed.

If *Arbitrary* is selected, the following dialog box will appear:



Enter the number of traces to display and press *OK*.

3.3.5 SHOW CURVES ✓ [ ]



When picking first breaks, it is often helpful to display the first break picks of prior records in the survey as a reference. An example of this is shown below. The red line indicates the first breaks of the current record, while the green lines represent the first breaks of several prior records from the same seismic spread. Pressing the tool button toggles the green travel time curves on and off.

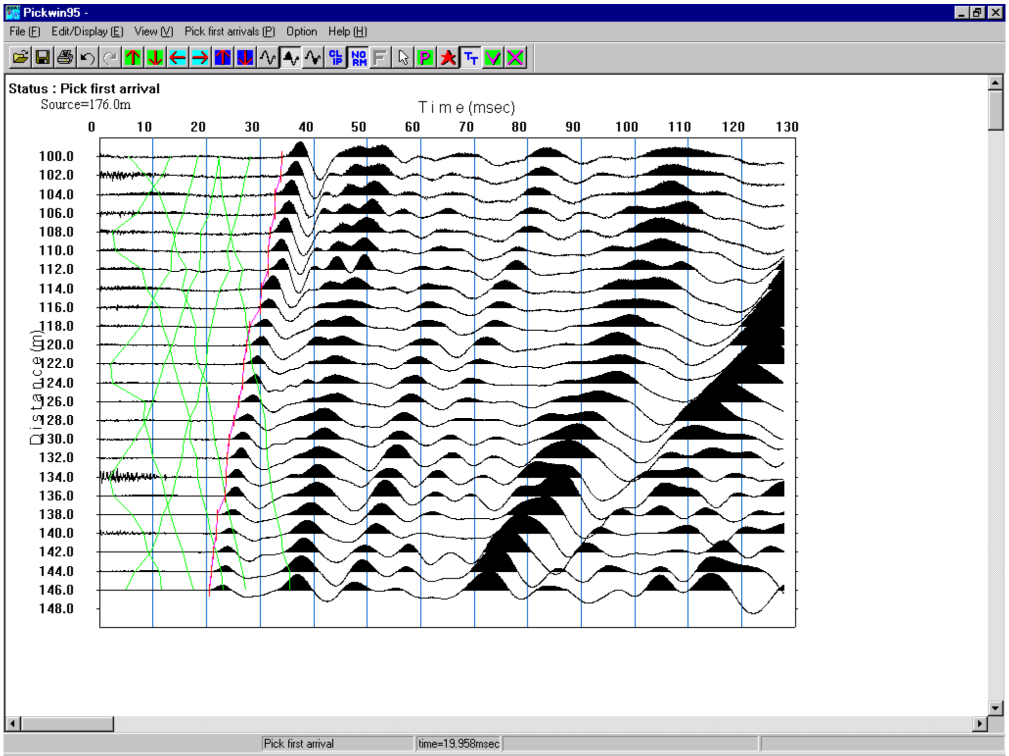
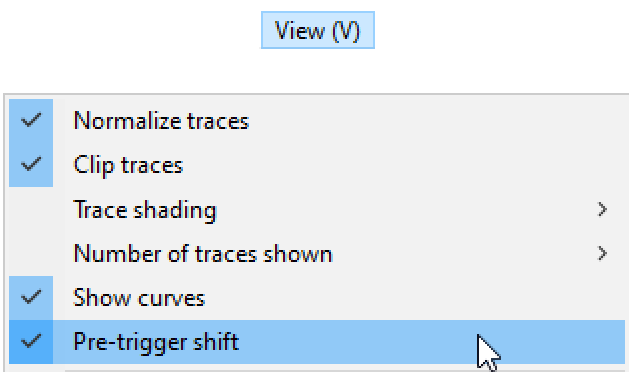


Figure 25: Refraction record with first arrival picks (red). Green lines represent first arrivals of previously picked records.

3.3.6 PRE-TRIGGER SHIFT ✓



If you have recorded pre-trigger data (accomplished by setting a “negative delay” on the seismograph), you may choose whether to display it (generally, you should). In the example below, the pre-trigger data is displayed, and the shot- or zero-time is indicated by the vertical tan line.

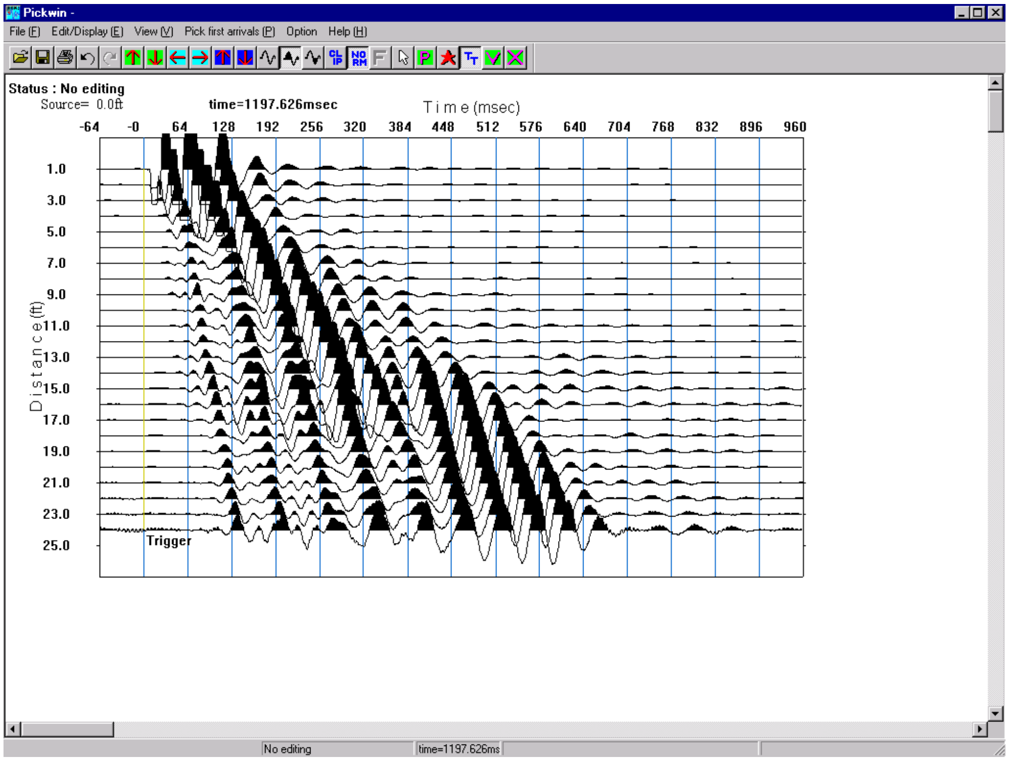
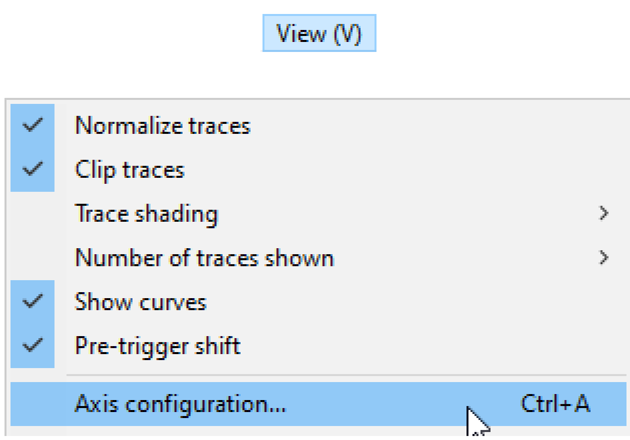


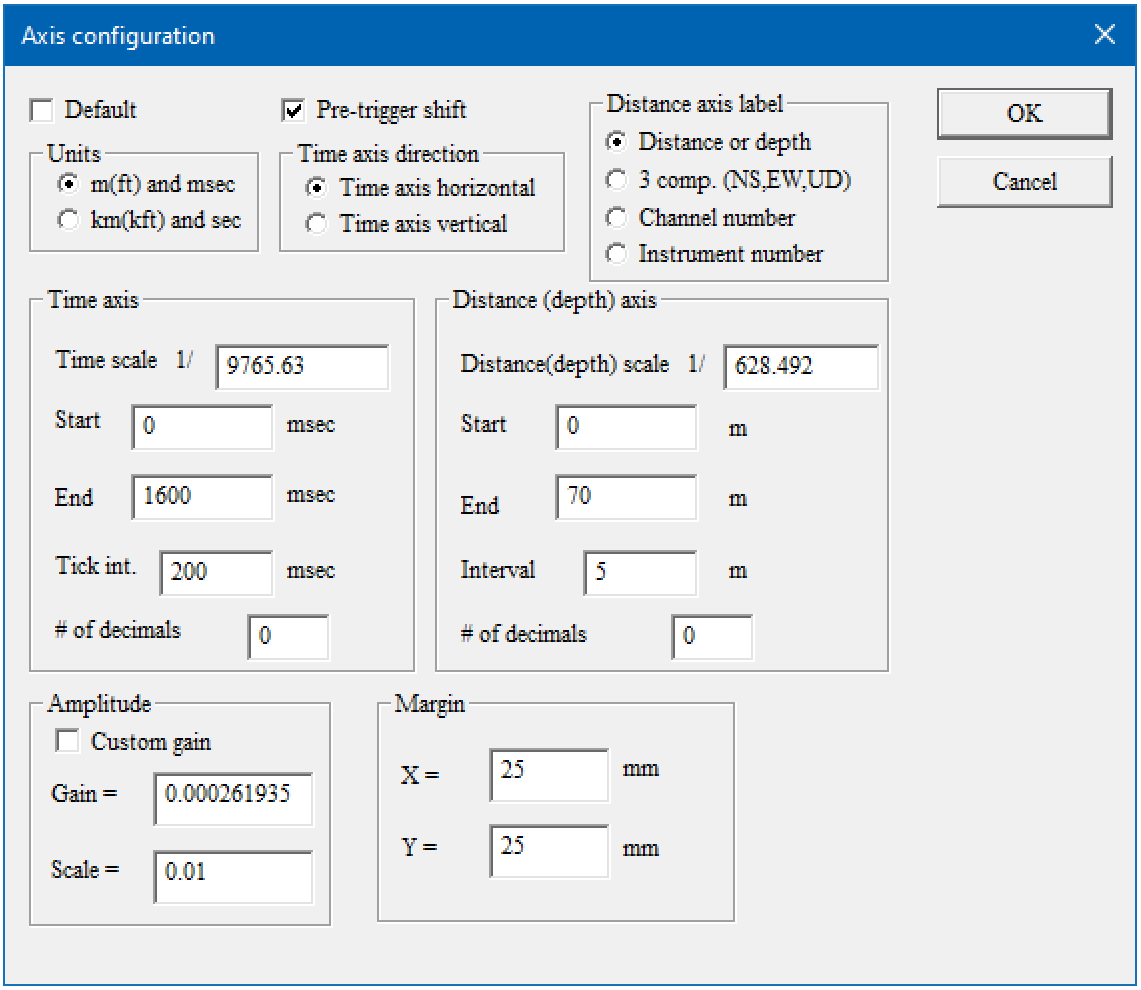
Figure 26: Refraction record recorded with pre-trigger data.

The pre-trigger shift setting can also be controlled from the **Axis Configuration** dialog box.

3.3.7 AXIS CONFIGURATION [CTRL+A]

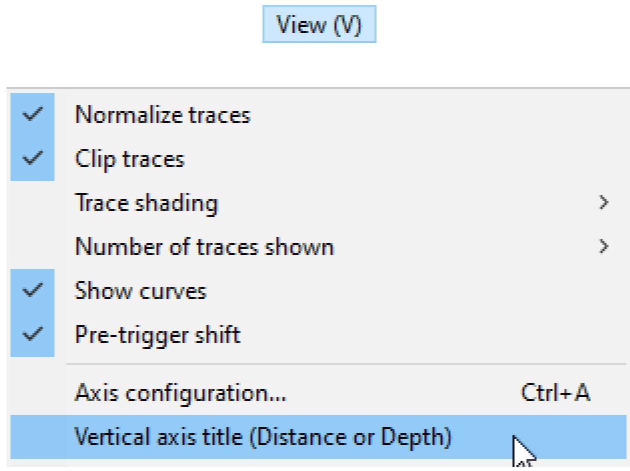


To change the display of the time (horizontal) axis or the distance (vertical) axis, click on *Axis configuration*. The following dialog box will appear:



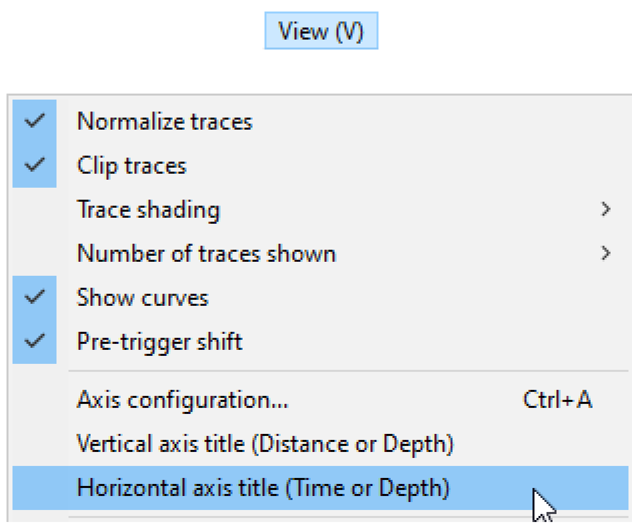
The functions of most of the parameters in the above dialog box are self-evident or can be deduced by simple trial and error. Configure the axes to your liking and press *OK* when finished.

3.3.8 VERTICAL AXIS TITLE (DISTANCE OR DEPTH) ✓



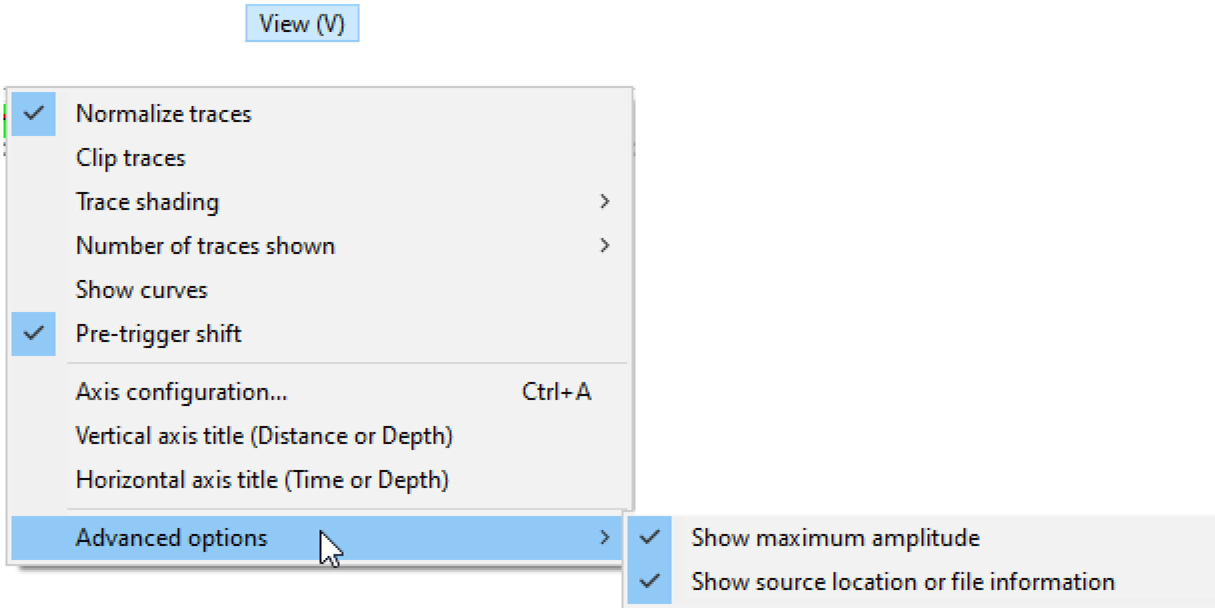
This is a switch that toggles the vertical axis label on the travel time plot between distance (surface survey) and depth (downhole survey).

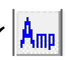
3.3.9 HORIZONTAL AXIS TITLE (TIME OR DEPTH) ✓

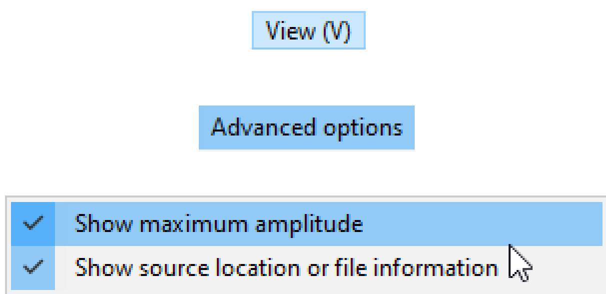


This is a switch that toggles the horizontal axis label on the travel time plot between time and depth.

3.3.10 ADVANCED OPTIONS



3.3.10.1 SHOW MAXIMUM AMPLITUDE ✓ 



This is a switch that toggles on/off the numerical maximum amplitude of each trace. The maximum amplitude is shown at the end of each trace:

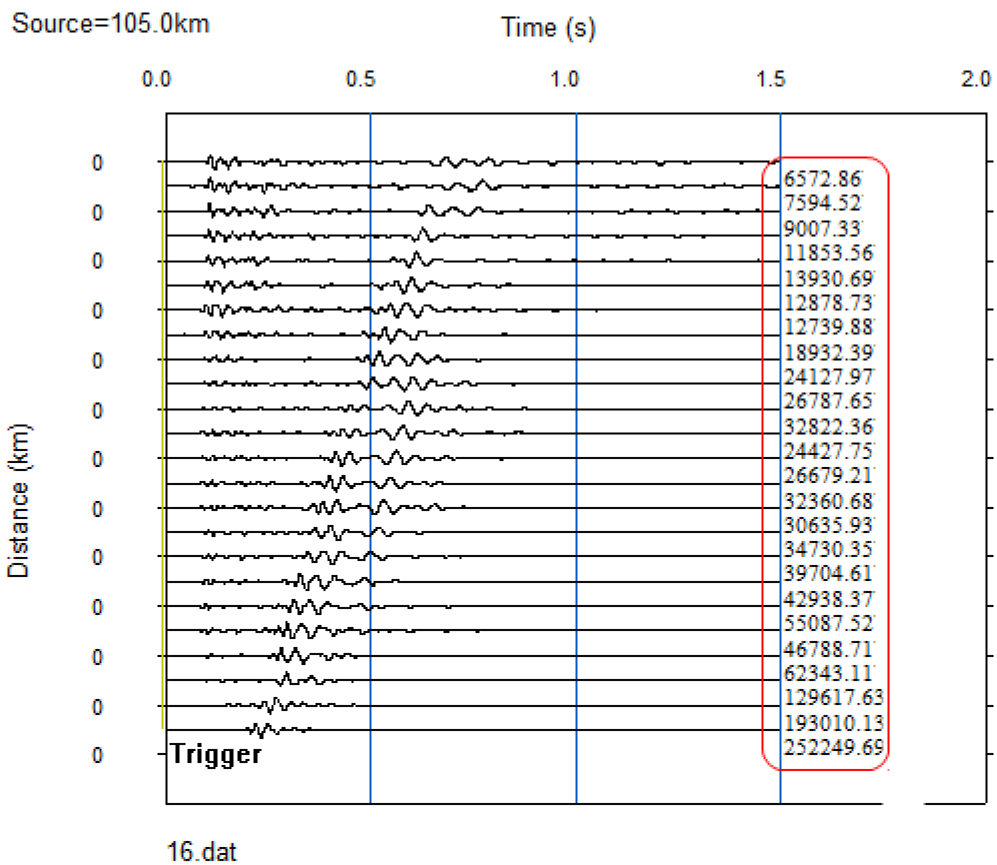
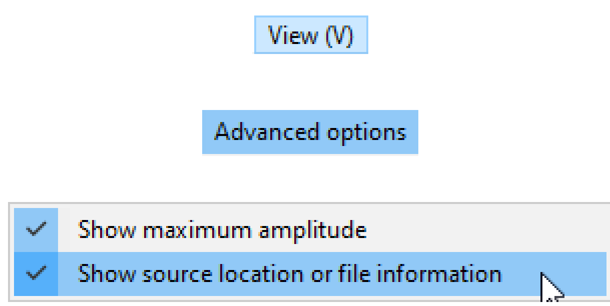


Figure 27: Seismic record with amplitude values shown.

3.3.10.2 Show Source Location or File Information ✓



This toggles on/off the source location label for a travel time plot:

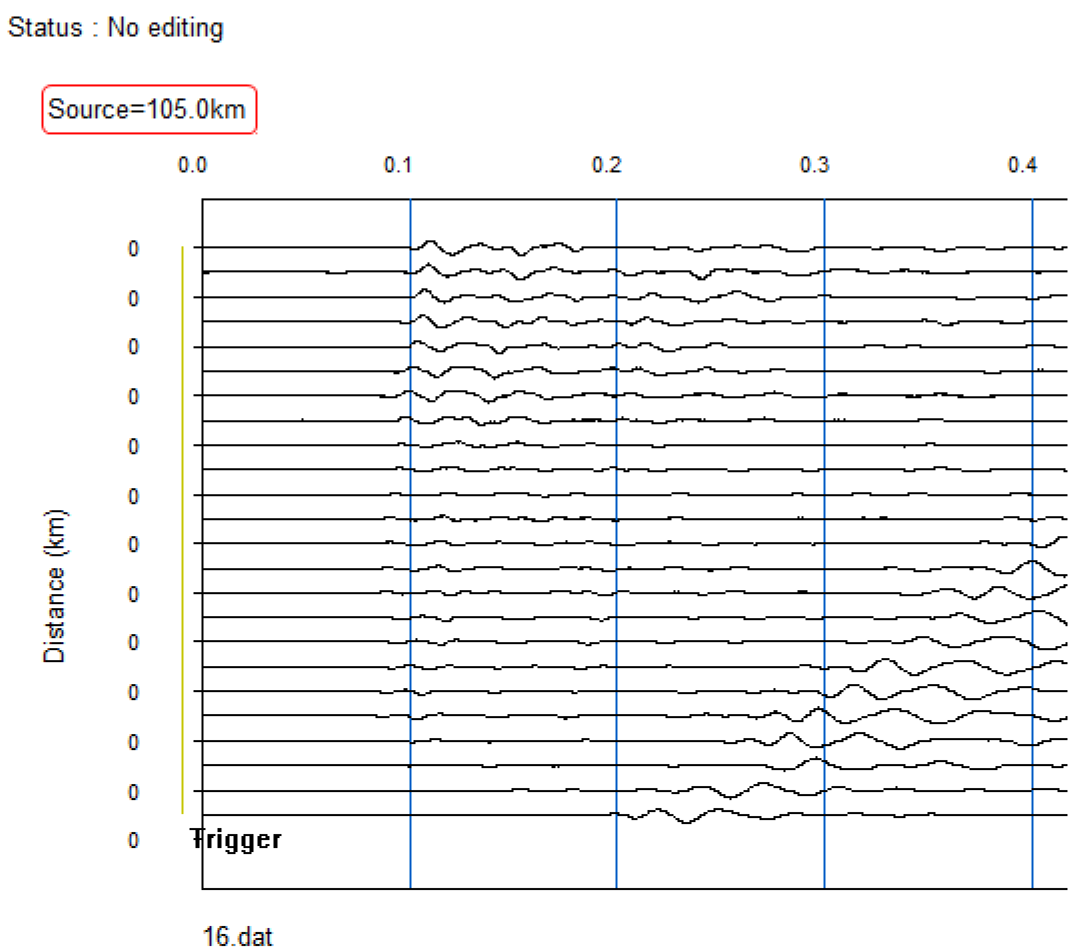
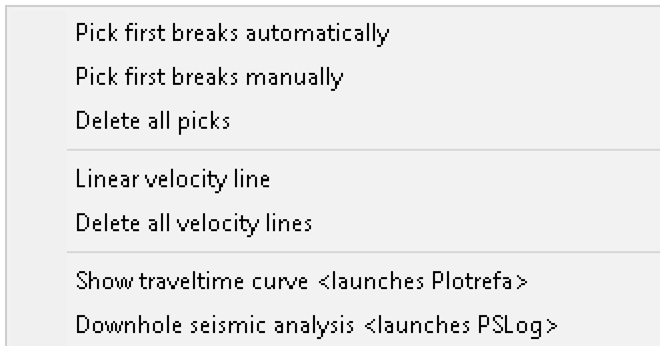


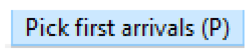
Figure 28: Seismic record with source location displayed.

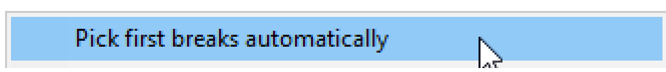
3.4 Pick First arrivals Menu

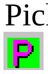
Click on *Pick first arrivals* to reveal the **Pick first arrivals** menu.



3.4.1 Pick First Breaks Automatically []





To have the Pickwin module pick the first breaks, click on *Pick first breaks automatically*, or click on the  button on the tool bar.

The first break picks chosen by Pickwin will appear as vertical red lines, as shown below:

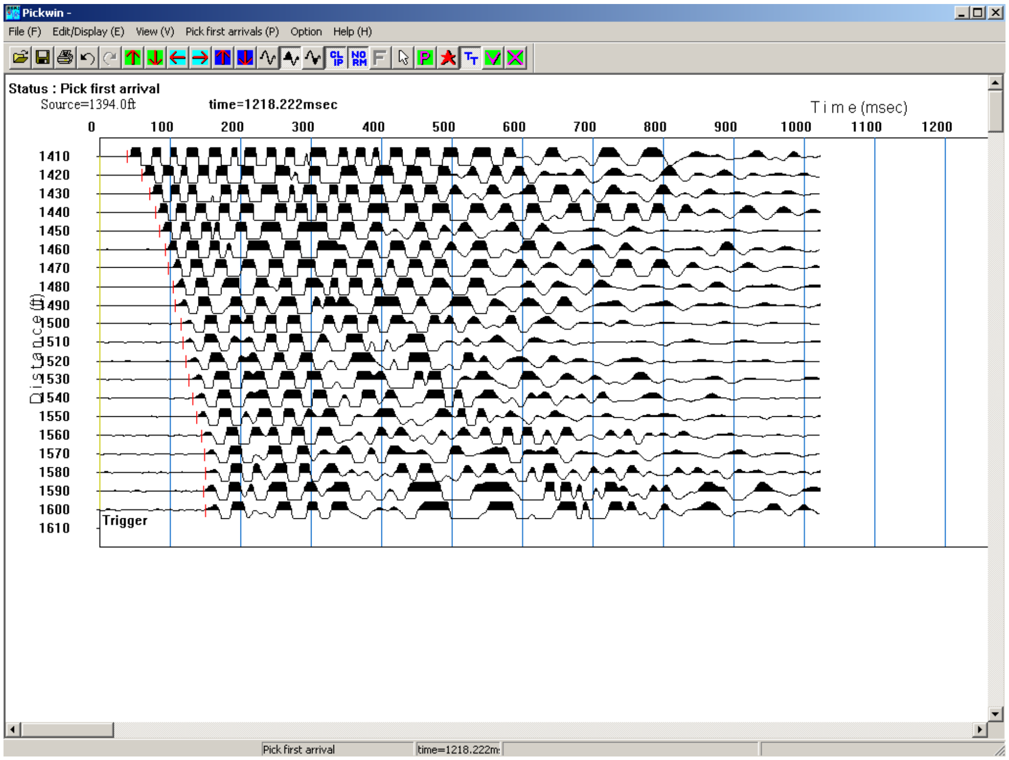
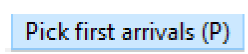
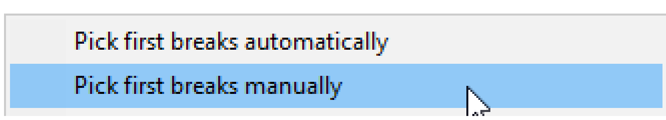


Figure 29: Seismic record showing first break picks.

Once Pickwin has automatically picked the first arrivals, these picks may be manually adjusted. Simply position the mouse at the desired location and click. The first break pick will be updated. Repeat until you are satisfied that the first breaks have been assigned correctly to all traces. See here for a video on the first break picking procedure (be sure to turn up the volume).

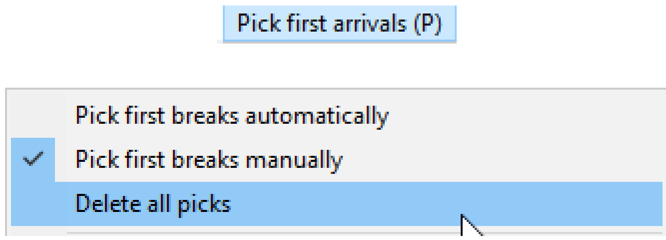
3.4.2 PICK FIRST BREAKS MANUALLY ✓



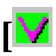


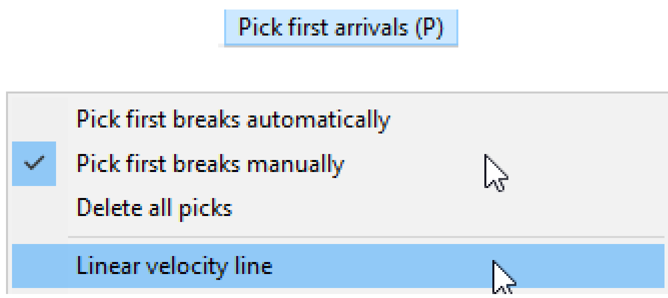
This option skips the above step of attempting to pick the first breaks automatically. You must pick all the breaks manually from the start.

3.4.3 DELETE ALL PICKS



If you wish to delete all your first break picks and start over, select this option.

3.4.4 LINEAR VELOCITY LINE []



To measure the apparent velocity of a series of first break picks, click on *Linear velocity line*, or press the  tool button.

Next, left click at the beginning of the series, and drag the cursor to the last break in the series. To fix the line, right click. The line will be labeled with the apparent velocity, as shown below:

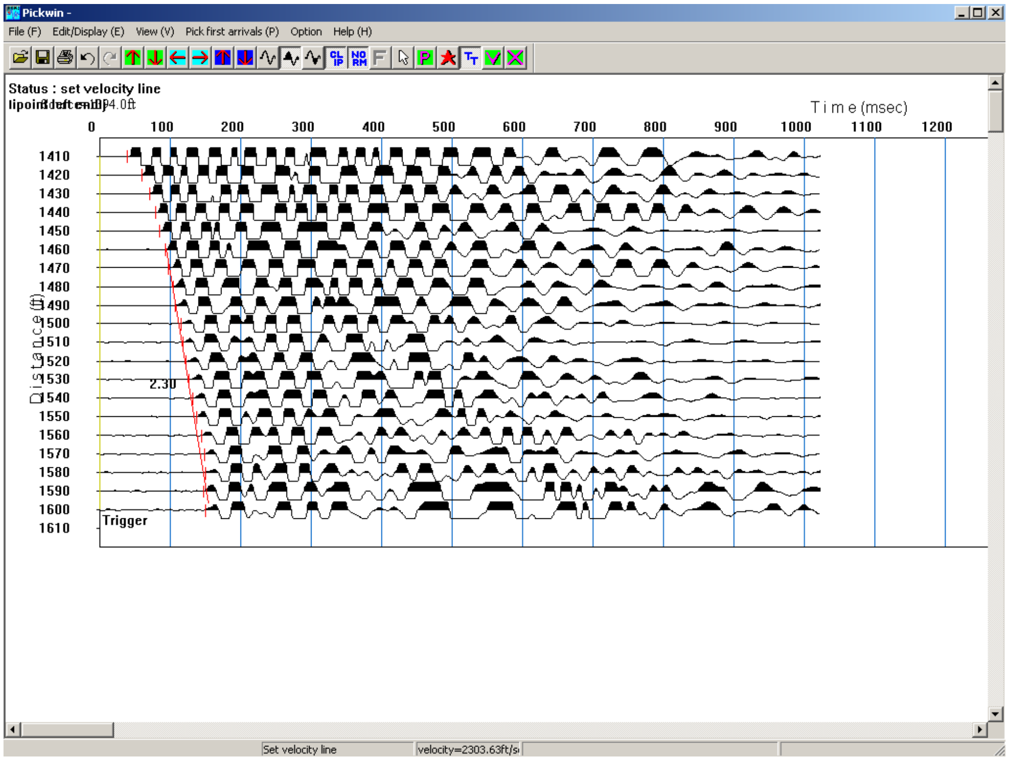
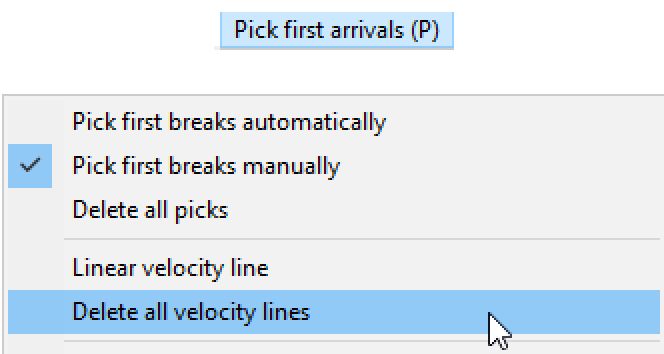


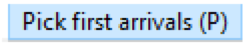
Figure 30: Apparent velocity line drawn on refraction record.

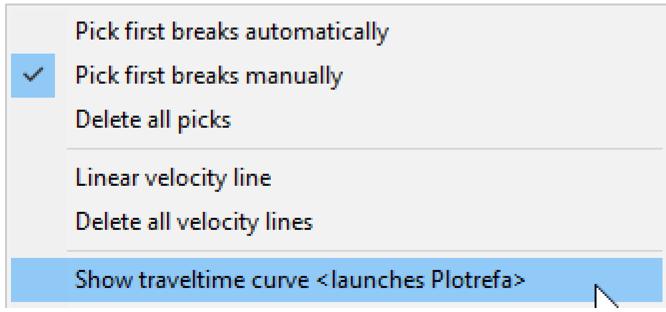
3.4.5 Delete All Velocity Lines



Choose *Delete all velocity lines* to remove any velocity lines from the display.

3.4.6 SHOW TRAVEL TIME CURVE <LAUNCHES PLOTREFA>





Choosing this option will launch Plotrefa and allow Plotrefa and Pickwin to operate interactively during the picking process. Follow these steps:

1. Create a file list (see Section 3.1.12.1, Page 22).

2. Pick the first breaks of one of the records on your list.

3. Press the button to connect them (see Section 3.10.18, Page 116).

4. Select *Pick first arrivals* | *Show travel time curve <launches Plotrefa>*. Plotrefa will open and automatically display the first breaks in a travel time plot.

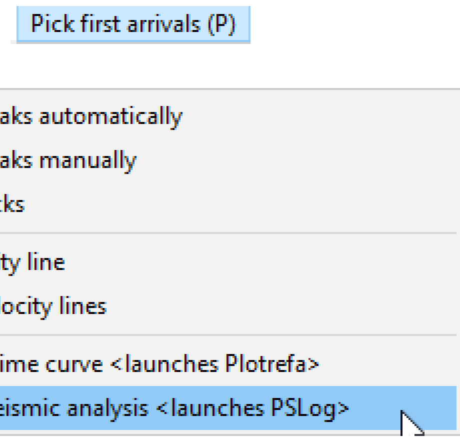
5. Now, in Pickwin, make any necessary adjustments to the first breaks. Press to connect the new picks to the existing picks.

6. In Plotrefa, press , and the travel time curve will be updated with the new picks.

7. Once you are finished with the shot record, scroll in Pickwin (using the tool buttons) to another waveform file and repeat.

8. When all the waveform files have been picked, you may proceed with Plotrefa to invert the data and create a velocity section.

3.4.7 DOWNHOLE SEISMIC ANALYSIS <LAUNCHES PSLOG>



Please refer to the separate SeisImager/DH manual for explanation of this menu item.

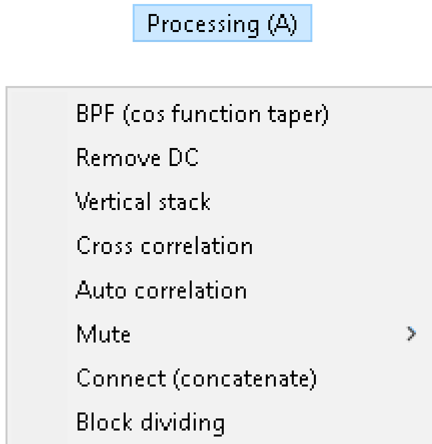
3.5 DOWNHOLE SEISMIC ANALYSIS MENU

Please refer to the separate SeisImager/DH manual for explanation of this menu.

3.6 SURFACE-WAVE ANALYSIS MENU

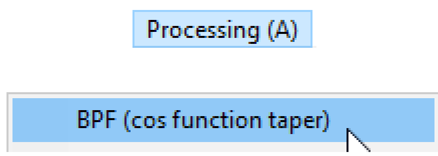
Please refer to the separate SeisImager/SW manual for explanation of this menu.

3.7 Processing Menu

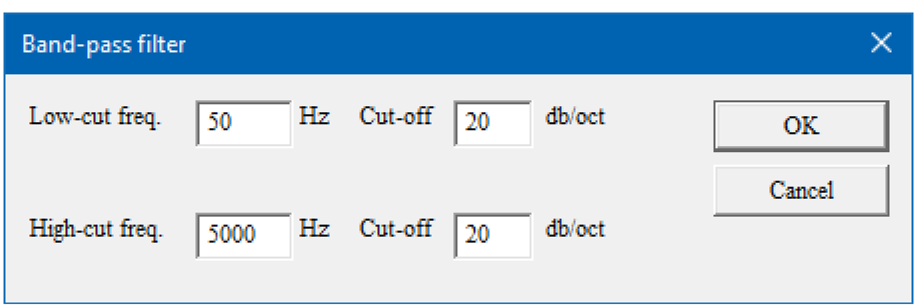


This menu includes various data processing tools that can be useful for both refraction and surface-wave data.

3.7.1 BPF (cos function taper)



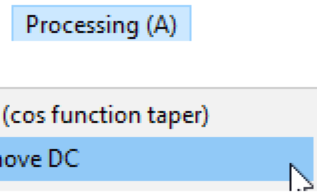
This is a bandpass filter for which you can set the specific filtering parameters:



You must filter each waveform file individually. To disable the bandpass filter, press the *Undo*  button.

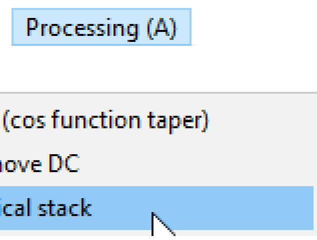
Note: *If you filter your data and save it, it will be saved in a filtered state.* ***Be sure not to overwrite your original data.***

3.7.2 REMOVE DC



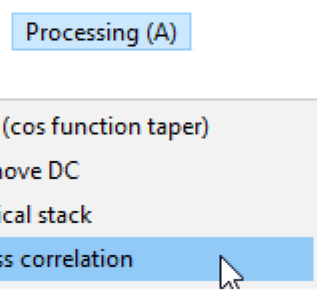
This feature removes any DC offset that might be present in your traces.

3.7.3 VERTICAL STACK



You may vertically stack (sum) two or more records together. This is handy if stacking was not done in the field. Simply append (Page 55) two or more files together, then select *Processing | Vertical Stack*.

3.7.4 CROSS CORRELATION



You may cross-correlate all the traces against any one trace. To do so, you must select an operator by pressing the  button and clicking on the desired trace:

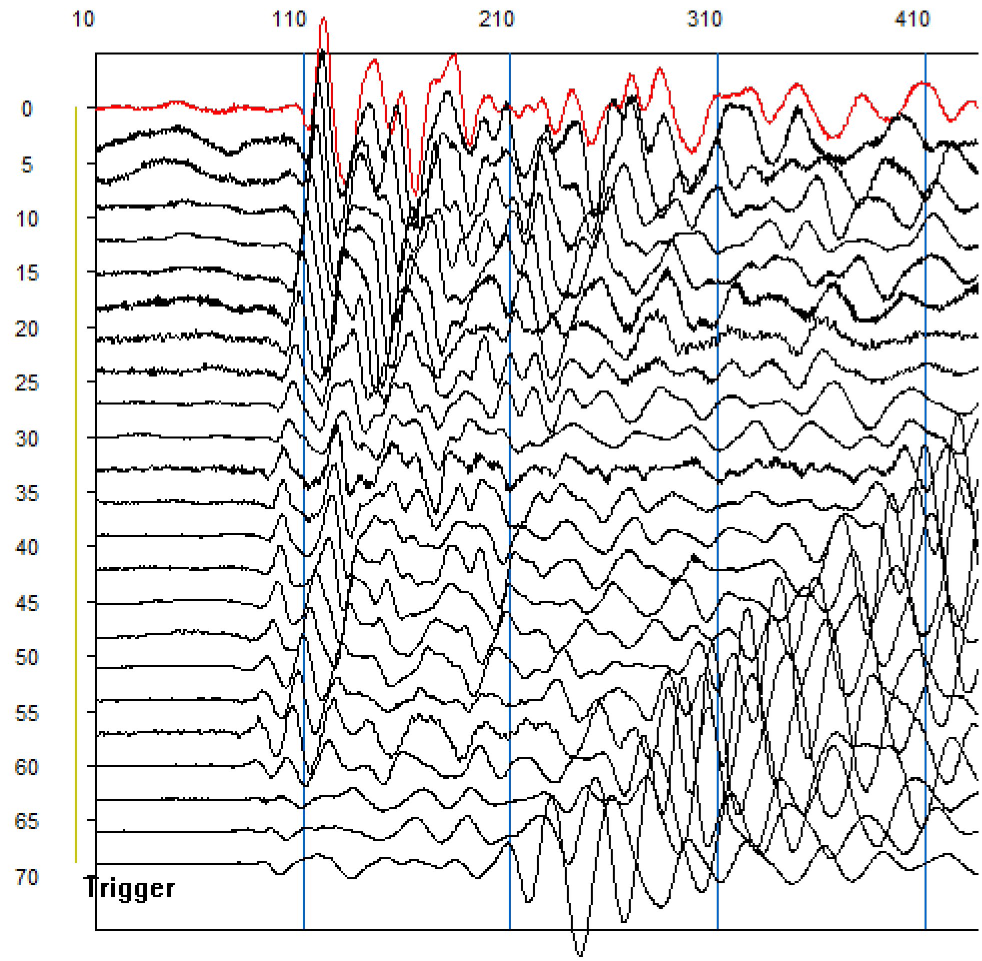
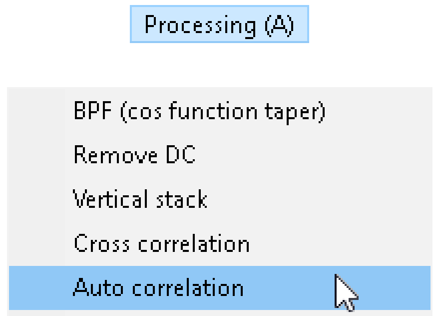


Figure 31: Operator (red) selected for cross-correlation.

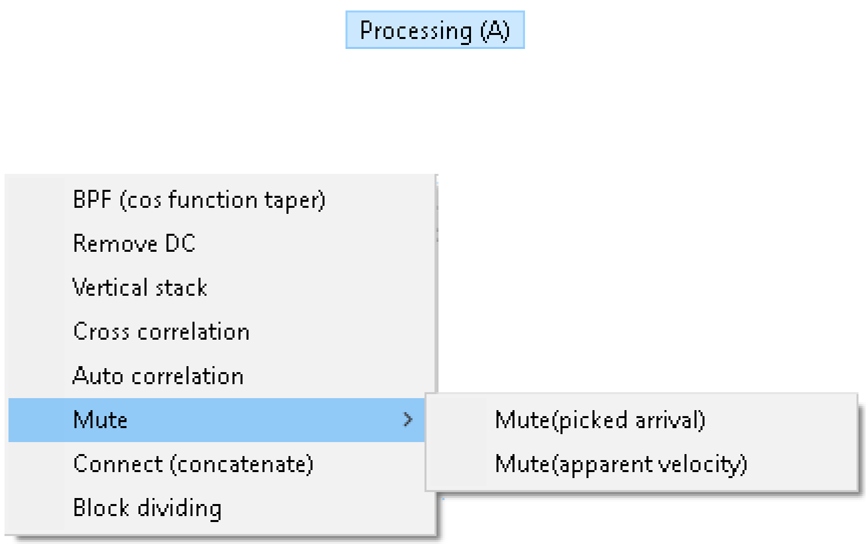
The selected trace will turn red; this is your operator. Now select *Cross correlation*, and all the traces in the record will be cross correlated with the operator.

3.7.5 AUTO CORRELATION



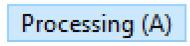
This feature auto correlates each trace with itself.

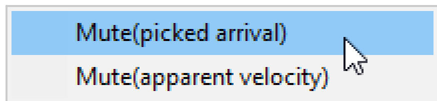
3.7.6 MUTE



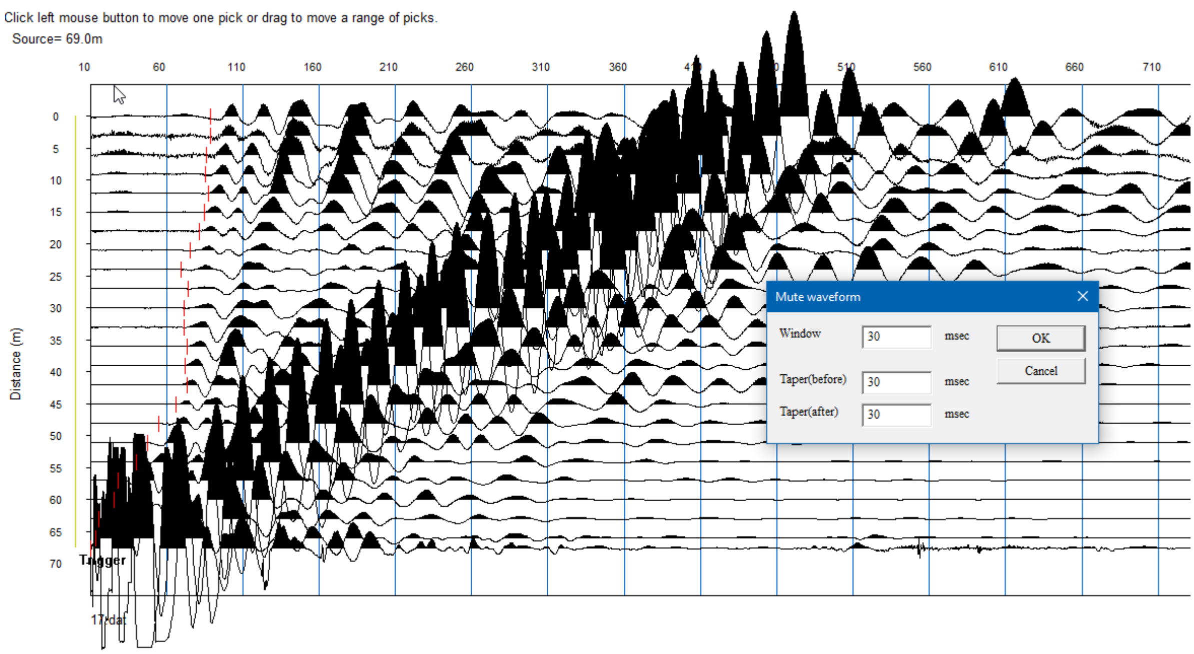
You may mute the record based on picked arrivals or apparent velocity.

3.7.6.1 Mute (Picked Arrival)





To mute based on first arrivals, pick the arrivals and then select *Mute (picked arrivals)*.



Set your mute parameters (see above) and press *OK*. Your record will be muted according to these parameters. In this example, we chose to mute the traces before and after the first breaks:

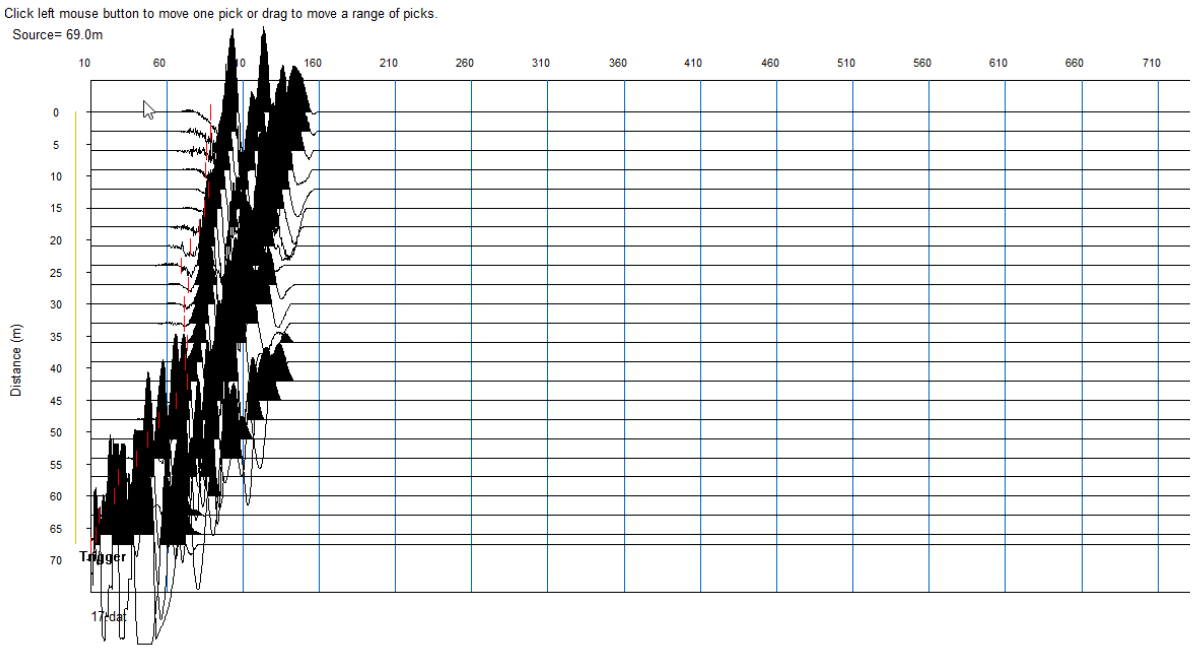
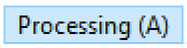
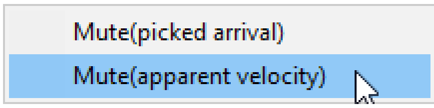


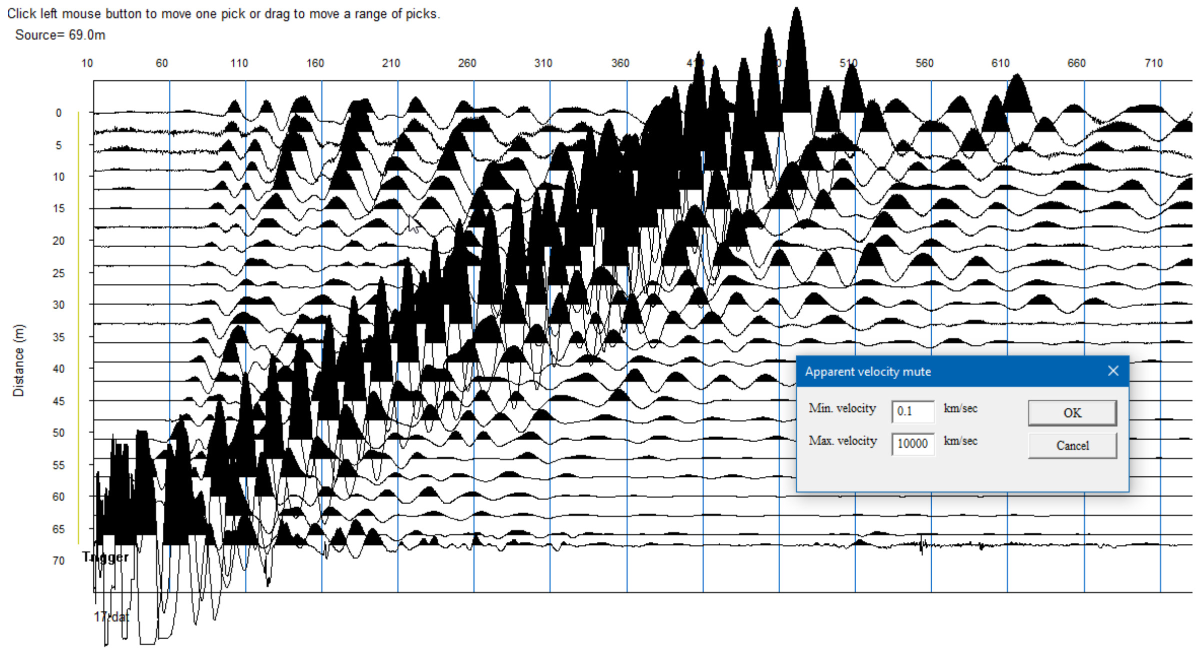
Figure 32: Refraction record muted based on the first arrivals.

3.7.6.2 Mute (Apparent Velocity)





You may also mute traces based on apparent velocity. This is most useful when working in the frequency domain (such as surface-wave analysis) but can be applied to refraction data as well. Enter the range of velocities you wish to preserve:



Press *OK*.

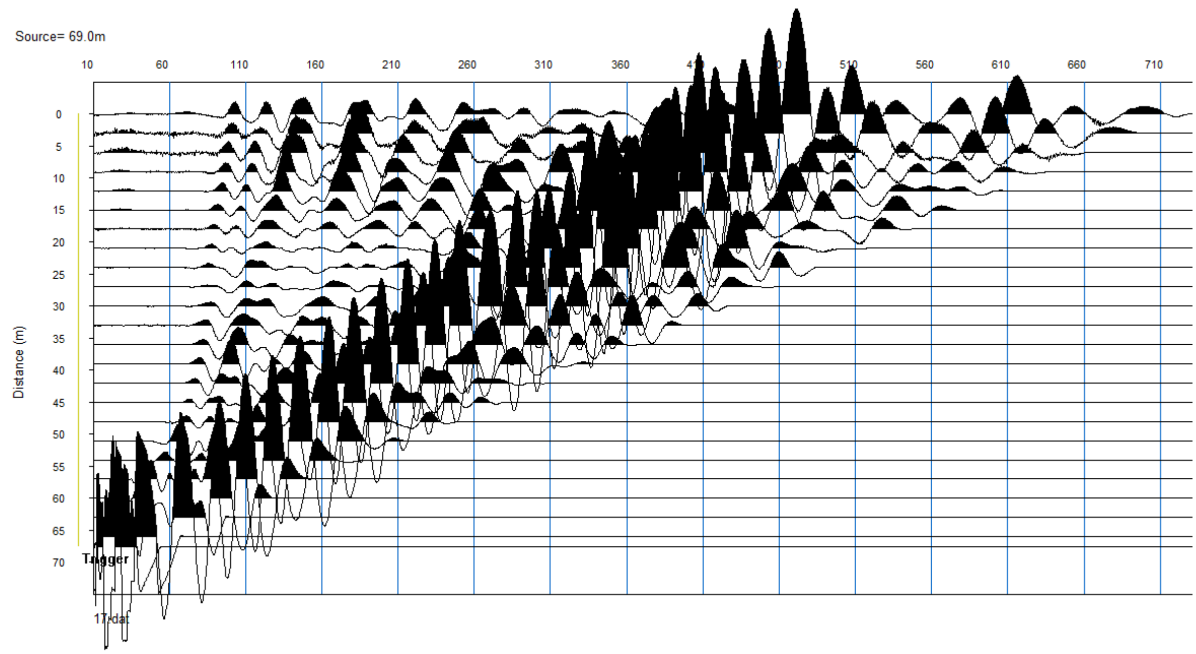
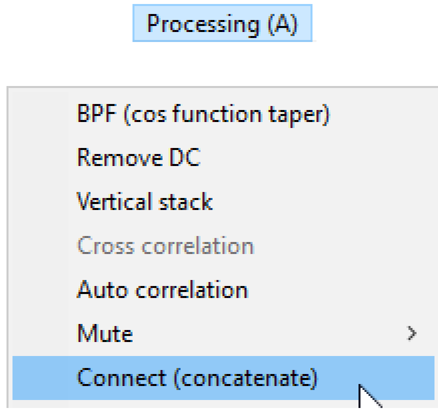


Figure 33: Refraction record muted by velocity.

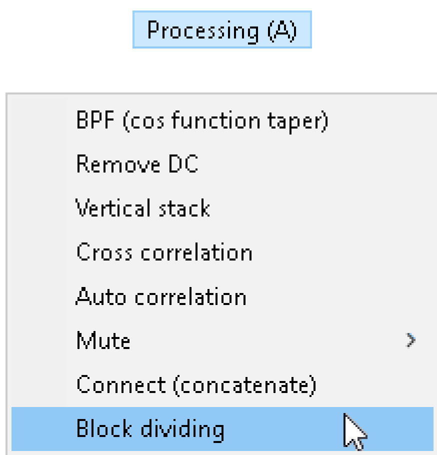
In general, muting is rarely applied to refraction data.

3.7.7 CONNECT (CONCATENATE)



Please refer to the separate SeisImager/SW manual for explanation of this menu item. It is generally not used for refraction.

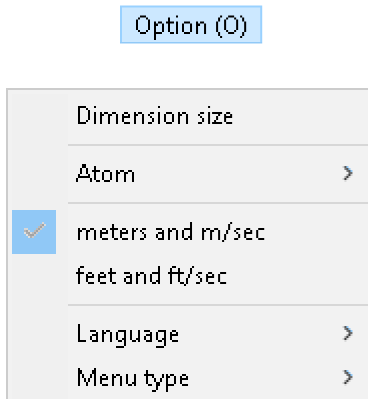
3.7.8 BLOCK DIVIDING



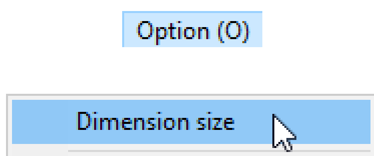
Please refer to the separate SeisImager/SW manual for explanation of this menu item. It is generally not used for refraction.

3.8 OPTION MENU

Click on *Option* to reveal the **Option** menu:

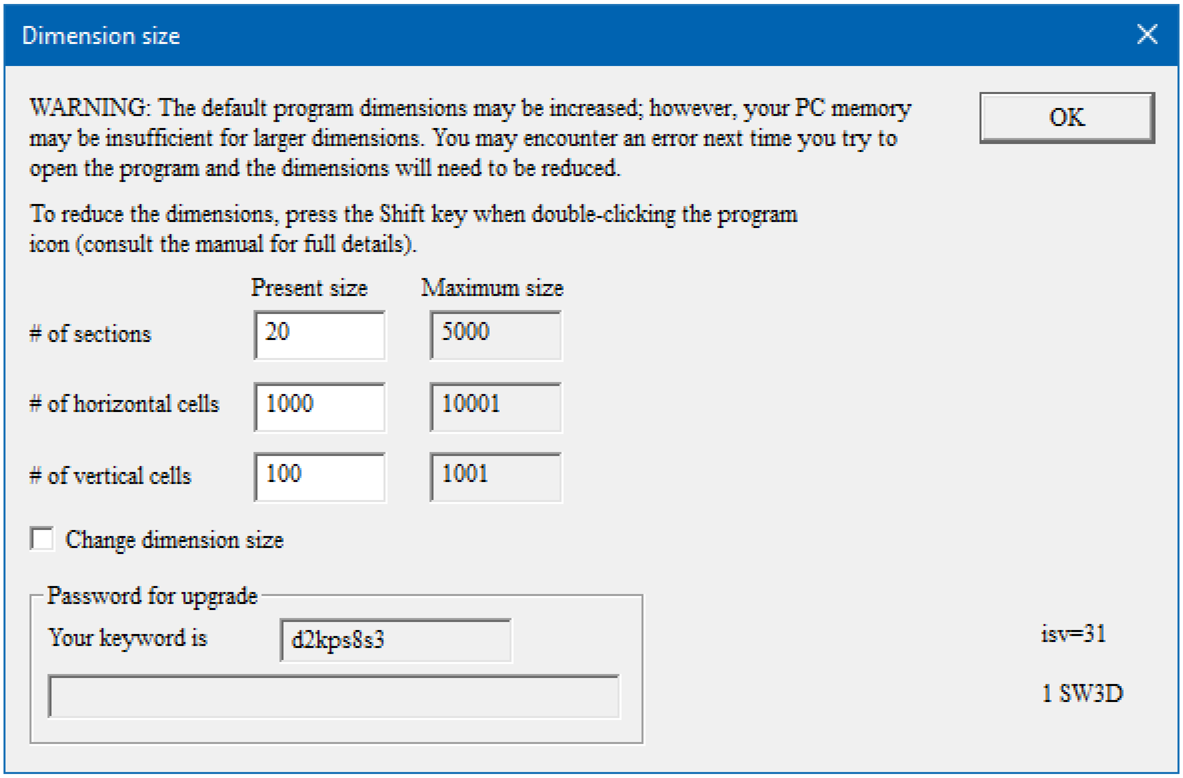


3.8.1 DIMENSION SIZE

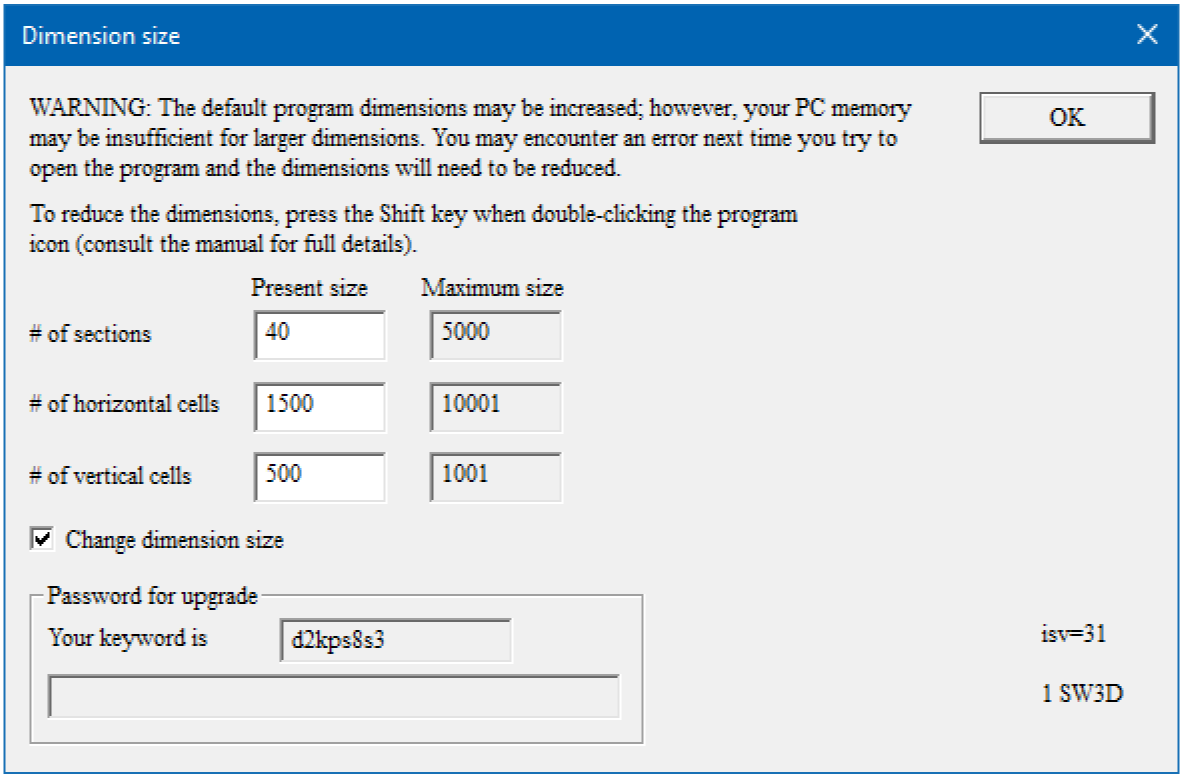


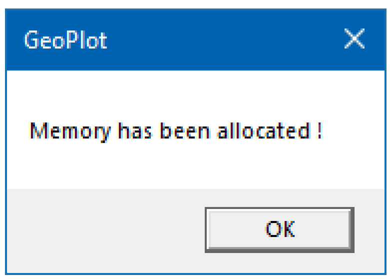
To view or change the program data input allowances, select *Dimension size.*

Present size reflects the current dimensions for *# of sections, # of horizontal cells,* and *# of vertical cells*. *Maximum size* shows the largest possible dimensions.



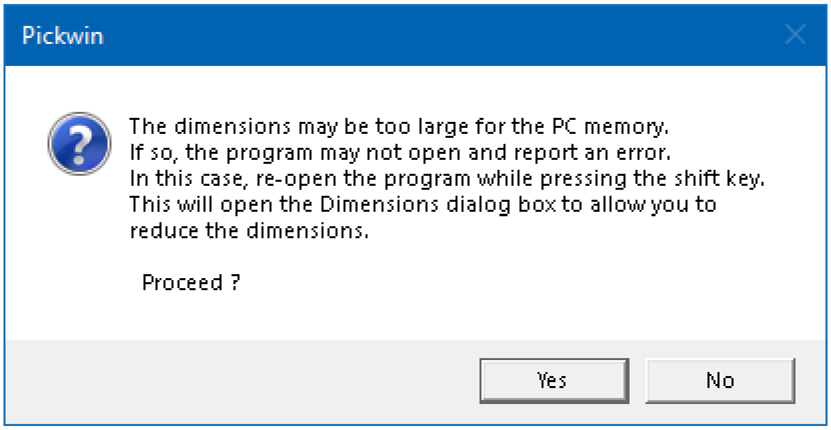
To change the dimensions, enter the new value(s), check *Change dimension size*, and press *OK*.





Press *OK* and restart the program.

Note: *If a very large value is entered, a warning message will appear before you are allowed to restart the program. It is recommended that you do not proceed; select* **No** *and reduce the dimensions.*



If you proceed and indeed the PC has insufficient memory, the program will no longer be able to open. To lower the values and recover the program, open the **Dimension size** dialog box directly by pressing the *Shift* key while double-clicking the program icon.

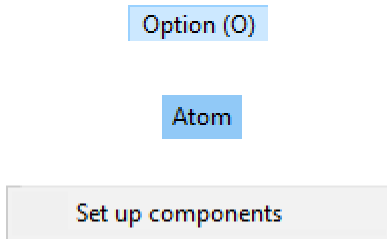
Note: *Sometimes the program will simply crash rather than post the above message. If that happens, use the procedure outlined on Page* *3* *to restore the system defaults.*

If a program upgrade is purchased, the new registration password can be directly entered in the **Dimension size** dialog box in the *Password for upgrade* field; however, it is strongly recommended to upgrade via the SeisImager Registration program instead.

3.8.2 ATOM

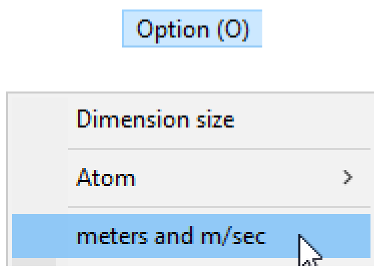
Continue.

3.8.2.1 Set up Components



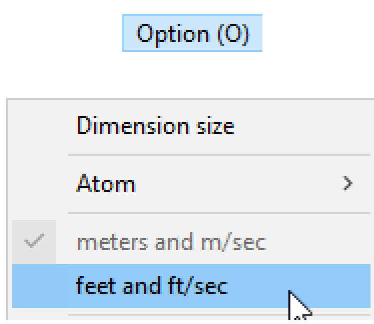
Please refer to the separate SeisImager/SW manual for explanation of this menu item. It is generally not used for refraction.

3.8.3 Meters and m/sec ✓



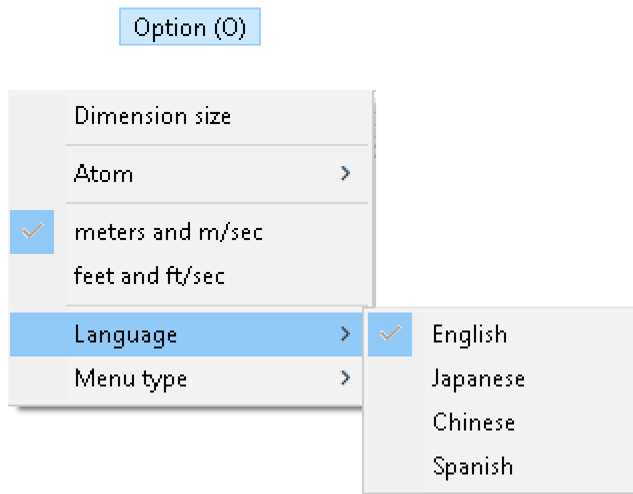
If you are working in metric units, choose this option.

3.8.4 Feet and ft/sec ✓



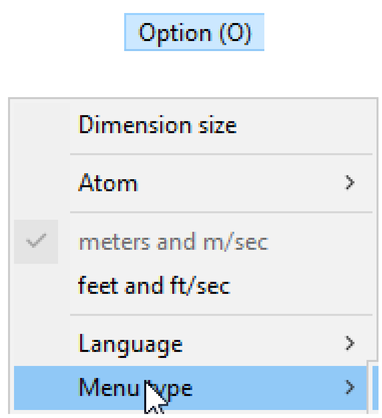
If you are working in British units, choose this option.

3.8.5 LANGUAGE



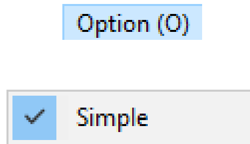
Choose your preferred language. At the time of this writing (July 2024), Chinese and Spanish were under construction.

3.8.6 MENU TYPE

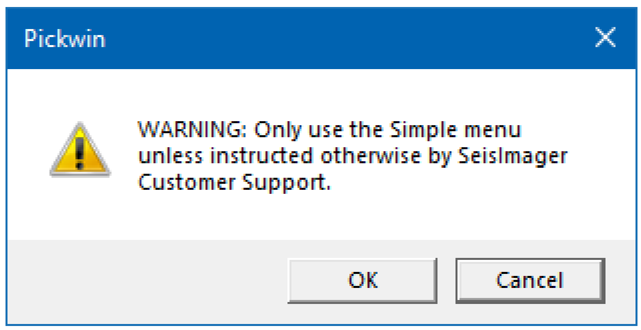


Continue.

3.8.6.1 SIMPLE ✓



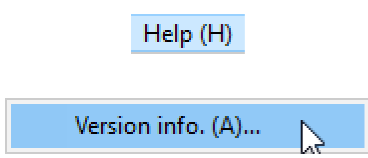
SeisImager includes advanced options for very special applications that are only available if *Simple* is **unchecked**. These are undocumented and should only be used under supervision from Geometrics (contact support@seisimager.com). If *Simple* is unchecked, you will see the following warning:



Pressing *OK* above will give you access to these features (mainly processing). To return to the **Simple** menu, select *Option* | *etc.* | *Menu type* | *Simple*.

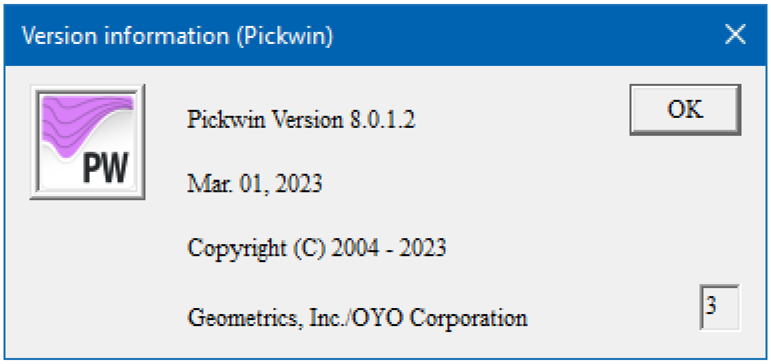
3.9 HELP MENU

Click on *Help* to reveal the **Help** menu:



3.9.1 VERSION INFO

Click on *Version info* to display the Pickwin Version number:



3.10 TOOL BUTTONS AND HOT KEYS

Figure 34: Pickwin Tool Buttons.

Pickwin includes numerous tool buttons and "hot keys", which are detailed below. Some have been mentioned before.

3.10.1 OPEN A FILE 

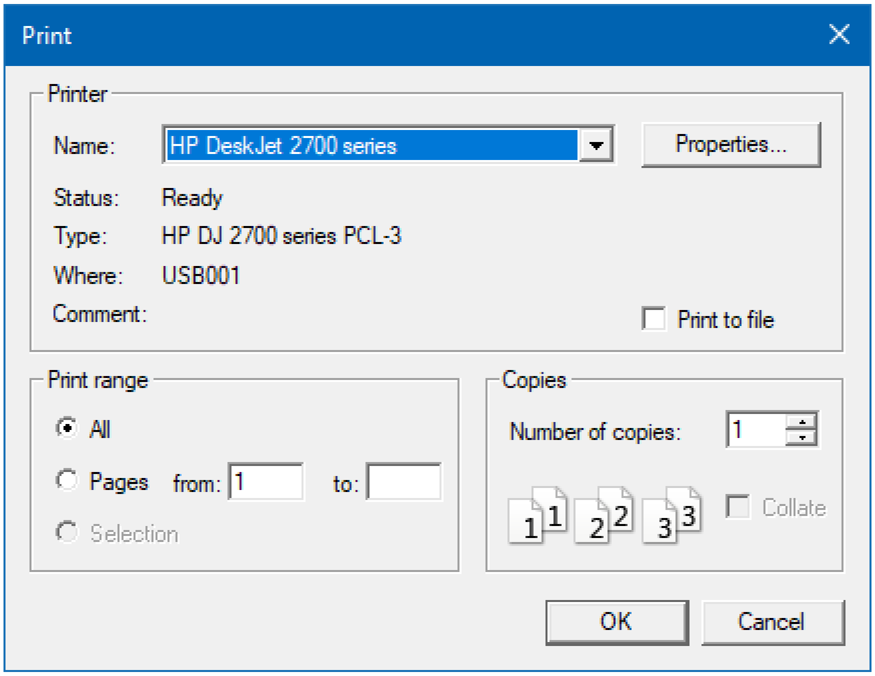
Use this button to open a SEG-2 file (typically, .dat or .sg2). It is the same as choosing *File* | *Open Waveform (SEG) File*.

3.10.2 SAVE A SEG-2 FILE 

Use this button to save a SEG-2 file that you have edited. Be sure to choose a file name such that you do not overwrite your original file. It is the same as choosing *File* | *Save Waveform (SEG-2) File*.

3.10.3 PRINT 

Pressing the *Print* button will display the **Print** dialog:



Choose your print preferences and press *OK*; whatever is on the display will be printed. This is the same as selecting *File* | *Print window display*.

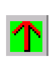
3.10.4 UNDO

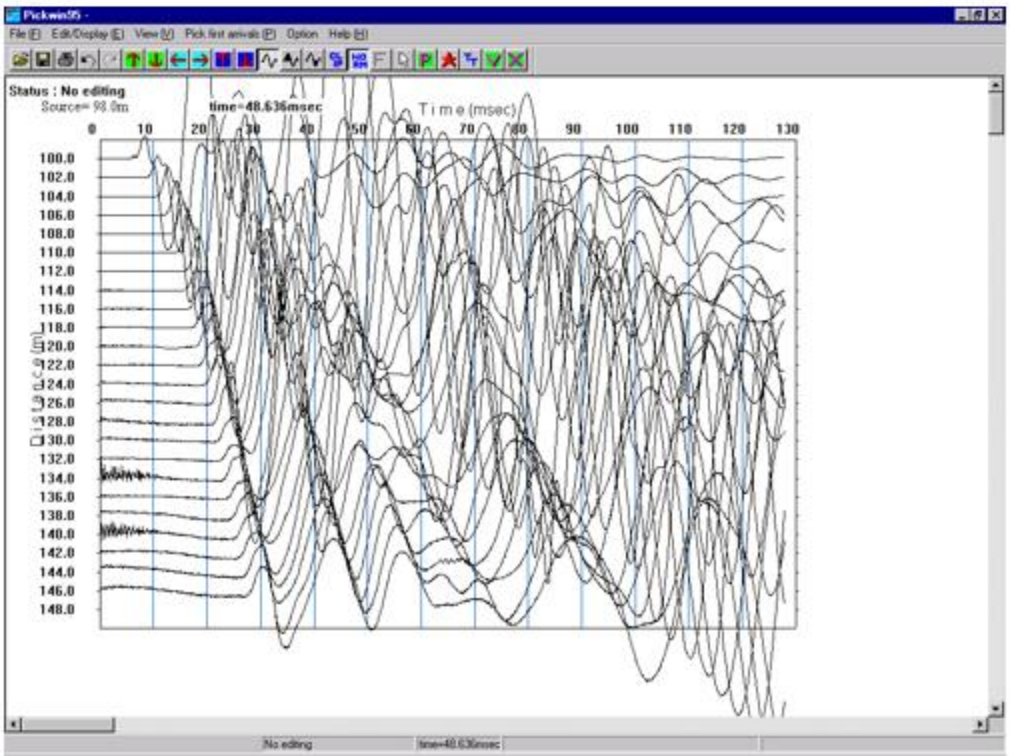
To undo the last command performed, click on .

3.10.5 REDO

To reverse the Undo button, press .

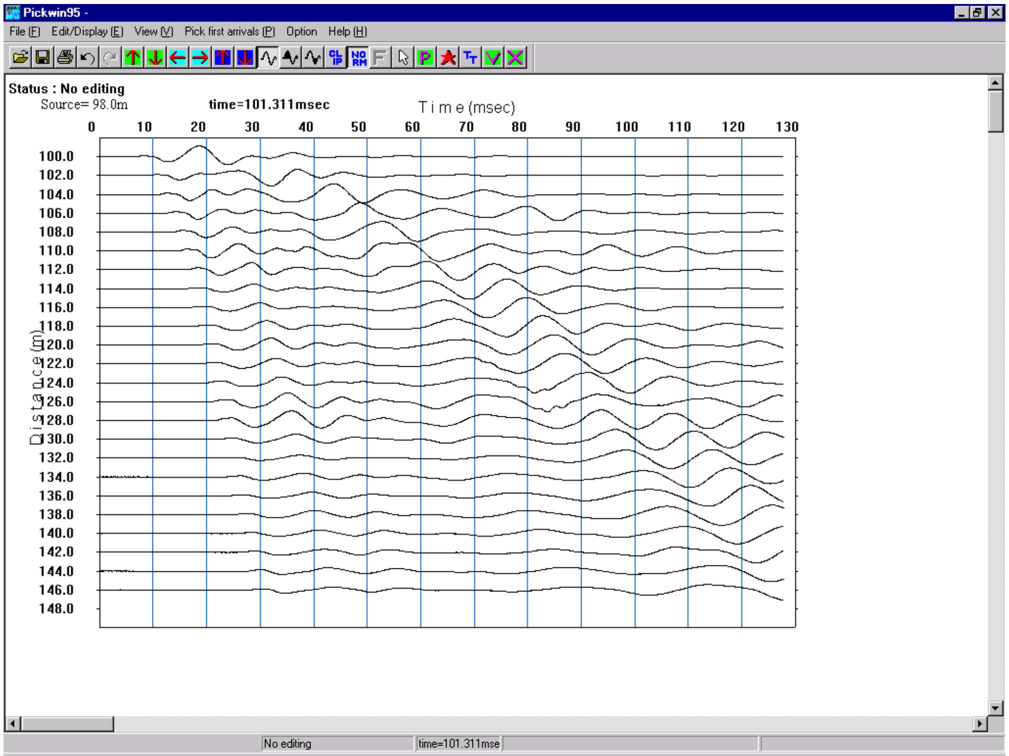
3.10.6 INCREASE AMPLITUDE TOOL BUTTON AND HOT KEY []

The "Increase amplitude" tool button  increases the amplitude of all the traces. The *up-arrow* key (↑) on the keyboard accomplishes the same thing.



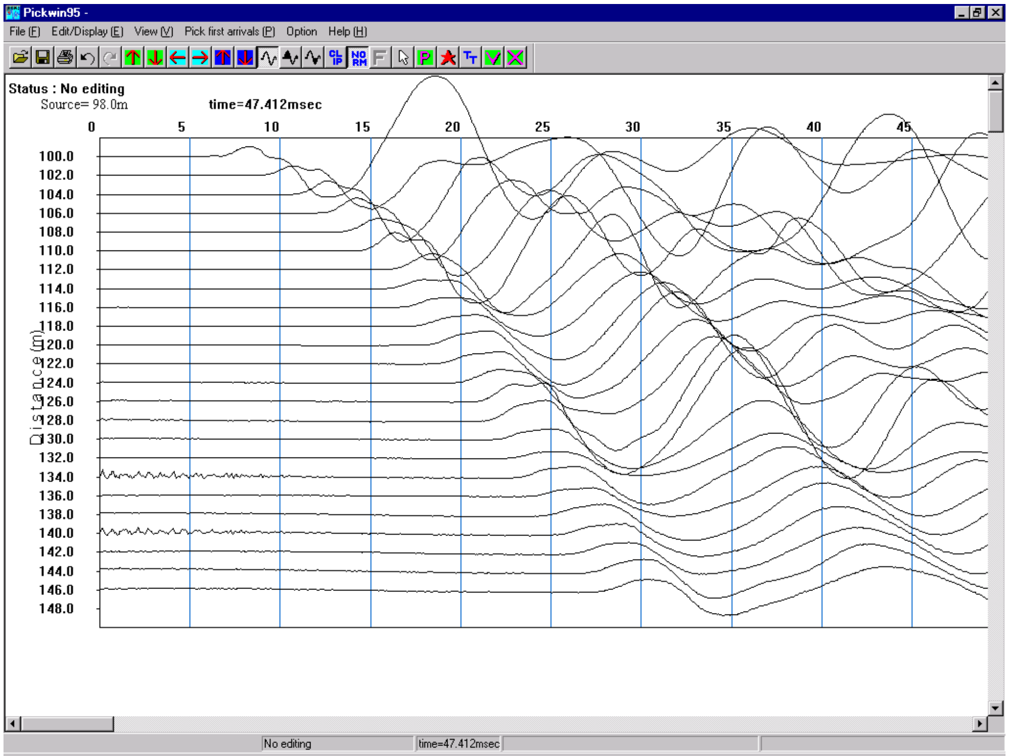
3.10.7 DECREASE AMPLITUDE TOOL BUTTON AND HOT KEY [ ]

The “Decrease amplitude” tool button decreases the amplitudes of all the traces. The *down-arrow* key (↓) on the keyboard accomplishes the same thing.



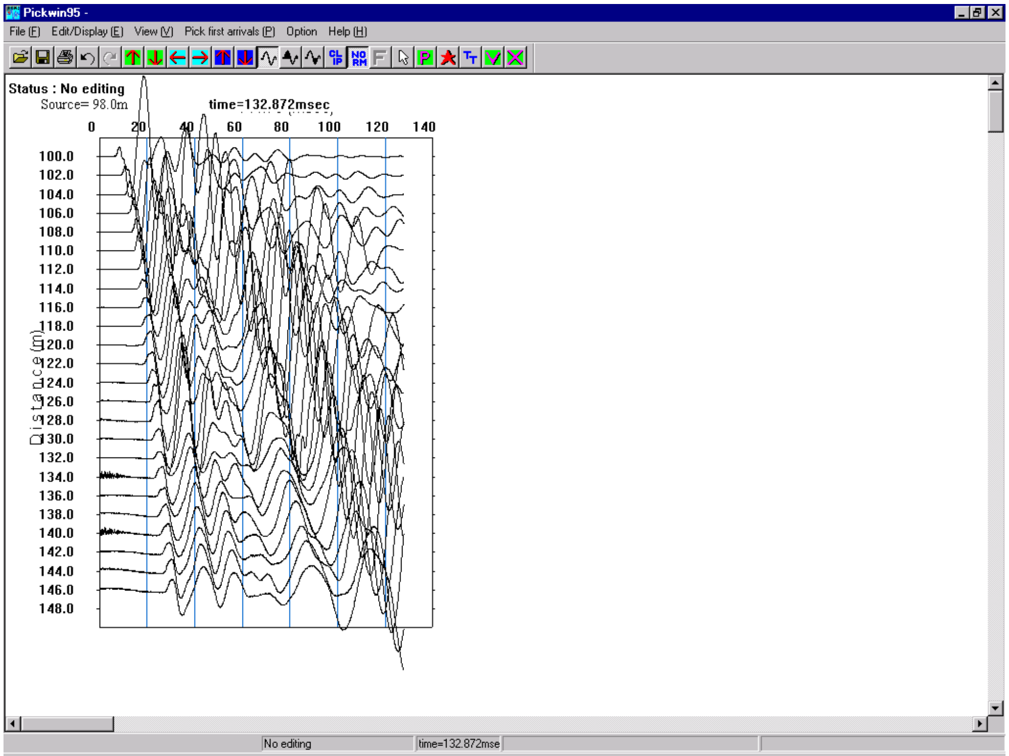
3.10.8 INCREASE HORIZONTAL AXIS TOOL BUTTON AND HOT KEY [ ]

The “Increase horizontal axis” tool button increases the length of the horizontal (time) axis. The *right-arrow* key (→) on the keyboard accomplishes the same thing.



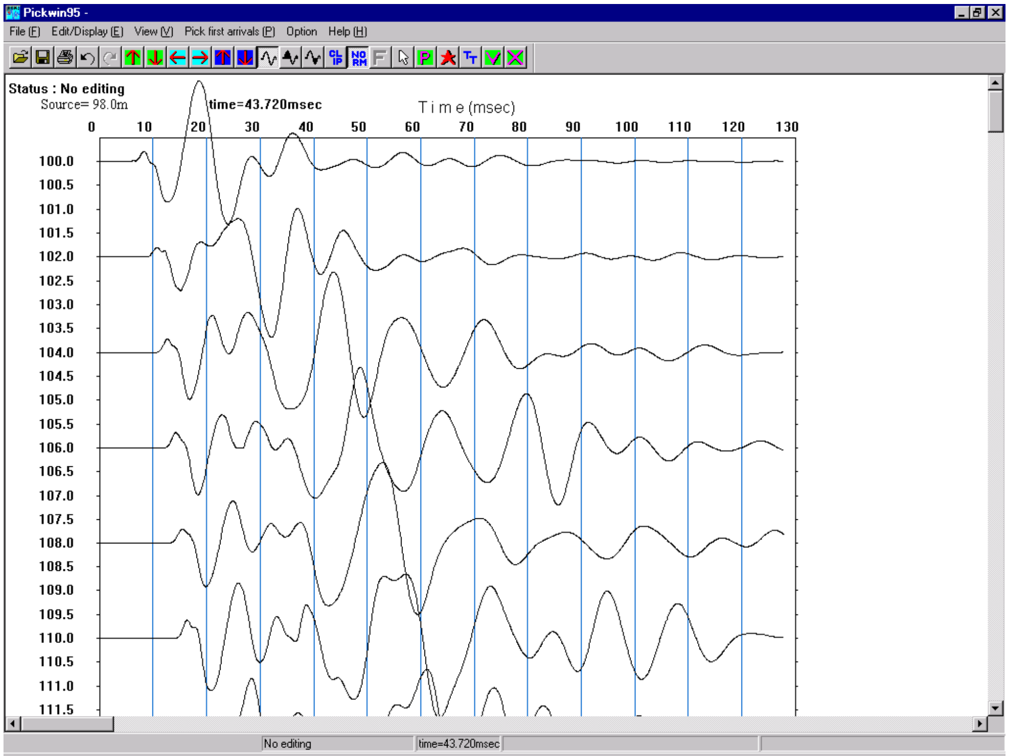
3.10.9 Decrease Horizontal Axis Tool Button and Hot Key [ ]

The "Decrease horizontal axis" tool button decreases the length of the horizontal (time) axis. The *left-arrow* key (←) on the keyboard accomplishes the same thing.



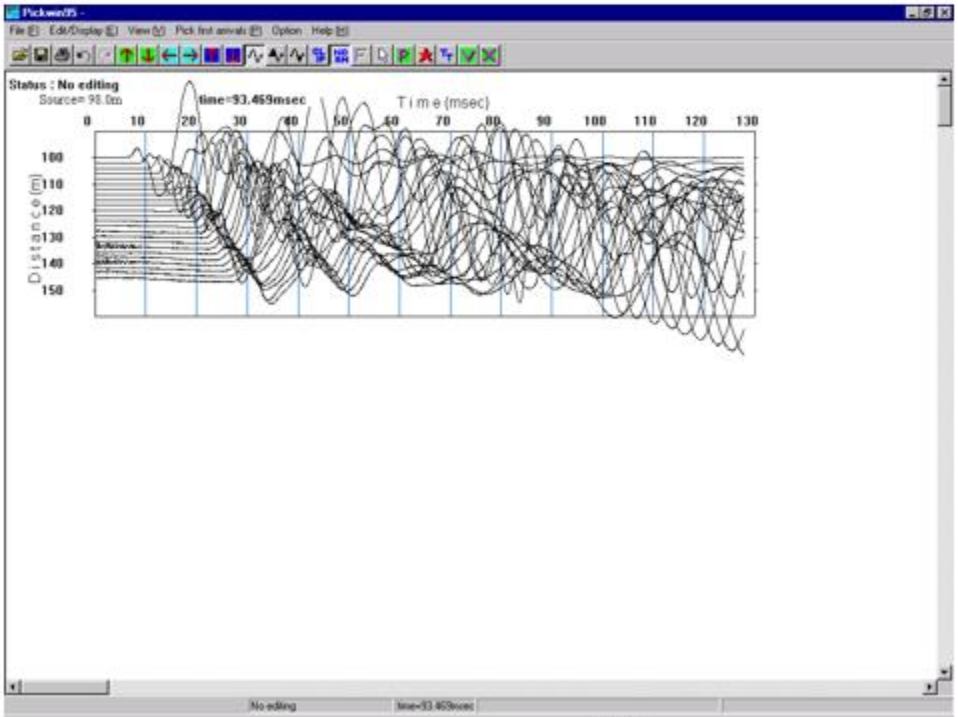
3.10.10 INCREASE VERTICAL AXIS TOOL BUTTON AND HOT KEY [ ]

The "Increase vertical axis" tool button increases the length of the vertical (distance or depth) axis. Pressing the *up-arrow* key (↑) on the keyboard while holding down the *Shift* key accomplishes the same thing.



3.10.11 DECREASE VERTICAL AXIS TOOL BUTTON AND HOT KEY [ ]

The “Decrease vertical axis” tool button decreases the length of the vertical (distance or depth) axis. Pressing the *down-arrow* key (↓) on the keyboard while holding down the *Shift* key accomplishes the same thing.



3.10.12 Trace Shading Tool Buttons 

See Section 3.3.3, Page 77.

3.10.13 Clip traces Tool Button 

See Section 3.3.2, Page 76.

3.10.14 Normalize Traces Tool Button 

See Section 3.3.1, Page 73.

3.10.15 Filter Tool Button 

See Section 3.2.11, Page 61.

3.10.16 Select Traces Tool Button

See discussion beginning in Section 3.2.4, Page 41.

3.10.17 Pick First arrivals Tool Button

See discussion beginning in Section 3.4, Page 87.

3.10.18 Register Travel times Tool Button [ ]

Once first breaks have been picked for a file, press the button. This will connect the first break picks with a red line. This is useful in making sure that there are no outliers that you may have missed.

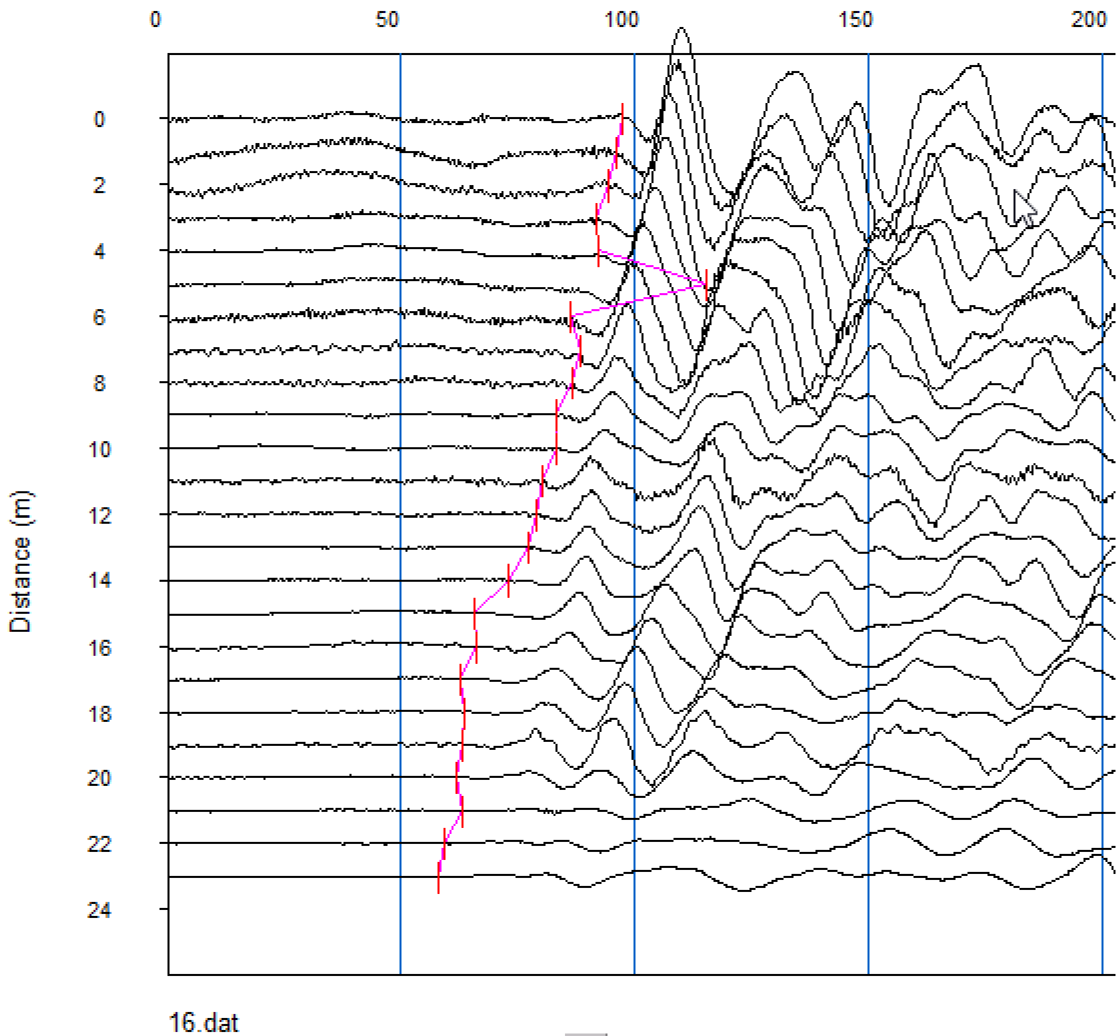


Figure 35: Picked first breaks; button flagged an outlier.

Note: *If a first break pick error is noticed after generating a travel time curve, simply reposition the first break pick by clicking at the appropriate position. Then, click on the button and the travel time curve will be readjusted.*

3.10.19 Show Travel time Curve Tool Button

See Section 3.3.5, Page 79.

3.10.20 Apparent Velocity Line Tool Button

At any time while displaying a shot record, you may press the button and draw an apparent velocity line (see below, light green). The apparent velocity will be reported at the top of the screen:

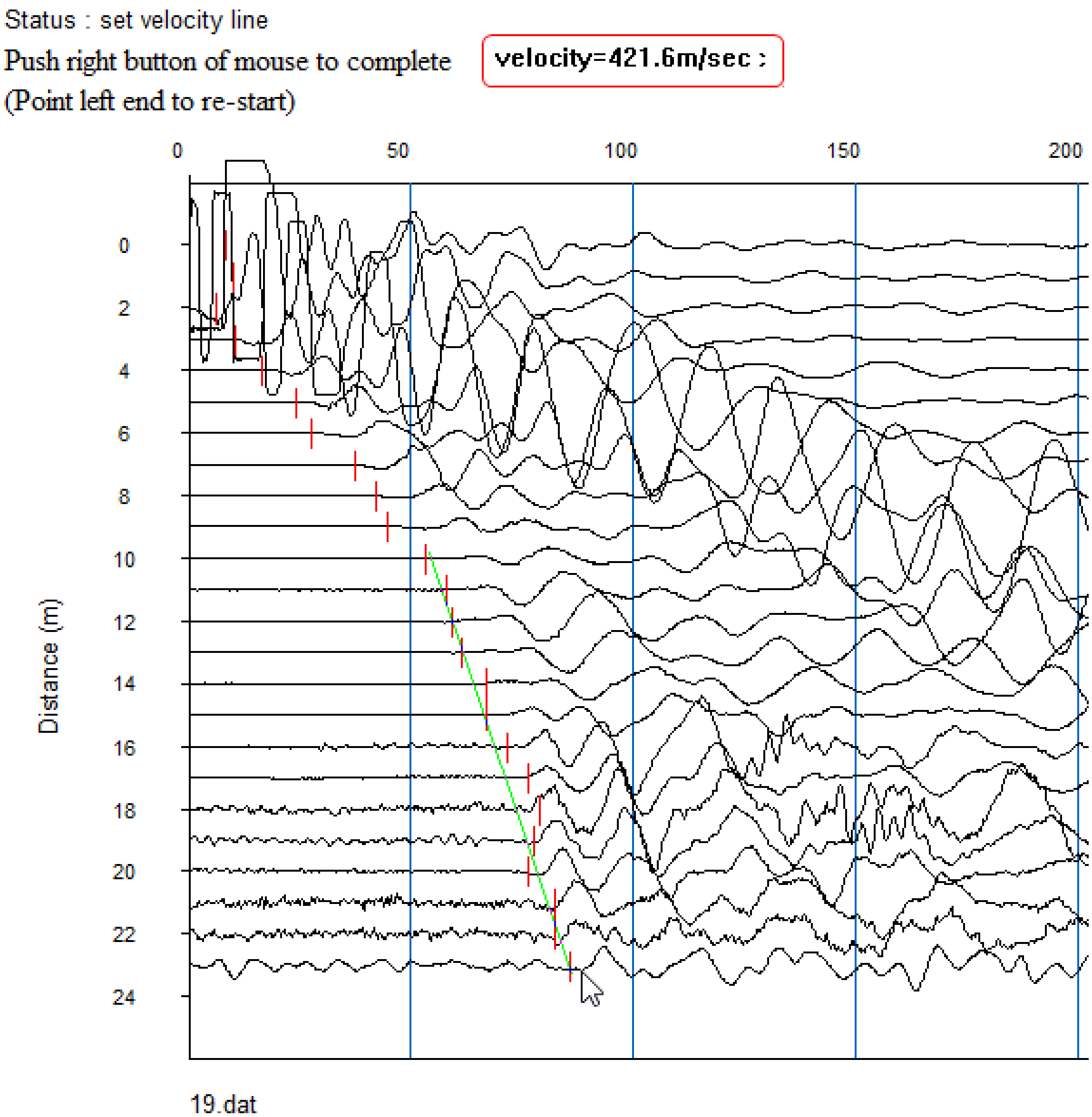


Figure 36: Apparent velocity line drawn on refractor.

Pressing the button again will remove the velocity line.

3.10.21 EXIT EDIT MODE TOOL BUTTON [ ]

This tool button exits from whatever "editing mode" you might be in. For instance, if you choose *Draw velocity line*, you are in an editing mode. To exit that edit mode (so you can, for instance, select traces), you must press the button.

3.10.22 PAGE UP TOOL BUTTON [ ]

If you have read in a group list of data files, you may page through from higher file number to lower file number by pressing the button.

3.10.23 PAGE DOWN TOOL BUTTON [ ]

If you have read in a group list of data files, you may page through from lower file number to higher file number by pressing the button.

3.10.24 COLOR CONTOUR TOOL BUTTON (COURSE)

Please refer to the separate SeisImager/SW manual for explanation of this menu item. It is generally not used for refraction.

3.10.25 COLOR CONTOUR TOOL BUTTON (FINE)

Please refer to the separate SeisImager/SW manual for explanation of this menu item. It is generally not used for refraction.

3.10.26 PLOT SEISMIC TRACES TOOL BUTTON

This button reverts to the standard seismic trace display:

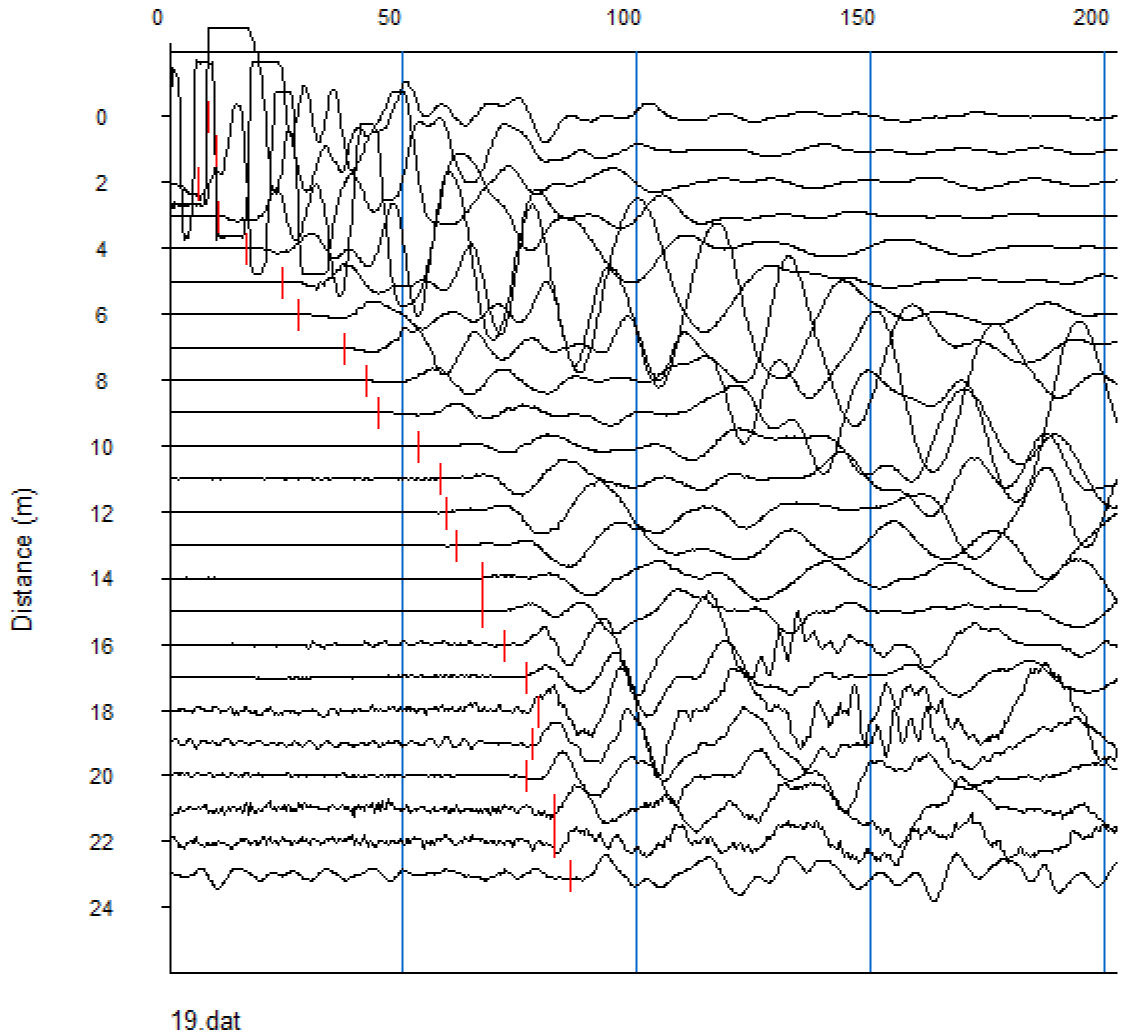


Figure 37: Trace display.

3.10.27 Show Waveforms Tool Button

Please refer to the separate SeisImager/SW manual for explanation of this menu item. It is generally not used for refraction.

3.10.28 Show Geometry Tool Button

Please refer to the separate SeisImager/SW manual for explanation of this menu item. It is generally not used for refraction.

3.10.29 Show Gridded Geometry Tool Button

Please refer to the separate SeisImager/SW manual for explanation of this menu item. It is generally not used for refraction.

3.10.30 Frequency Domain Tool Button

Press this button to display your data in the frequency domain. This is mainly a surface-wave analysis tool, but it can be handy and informative in refraction as well.

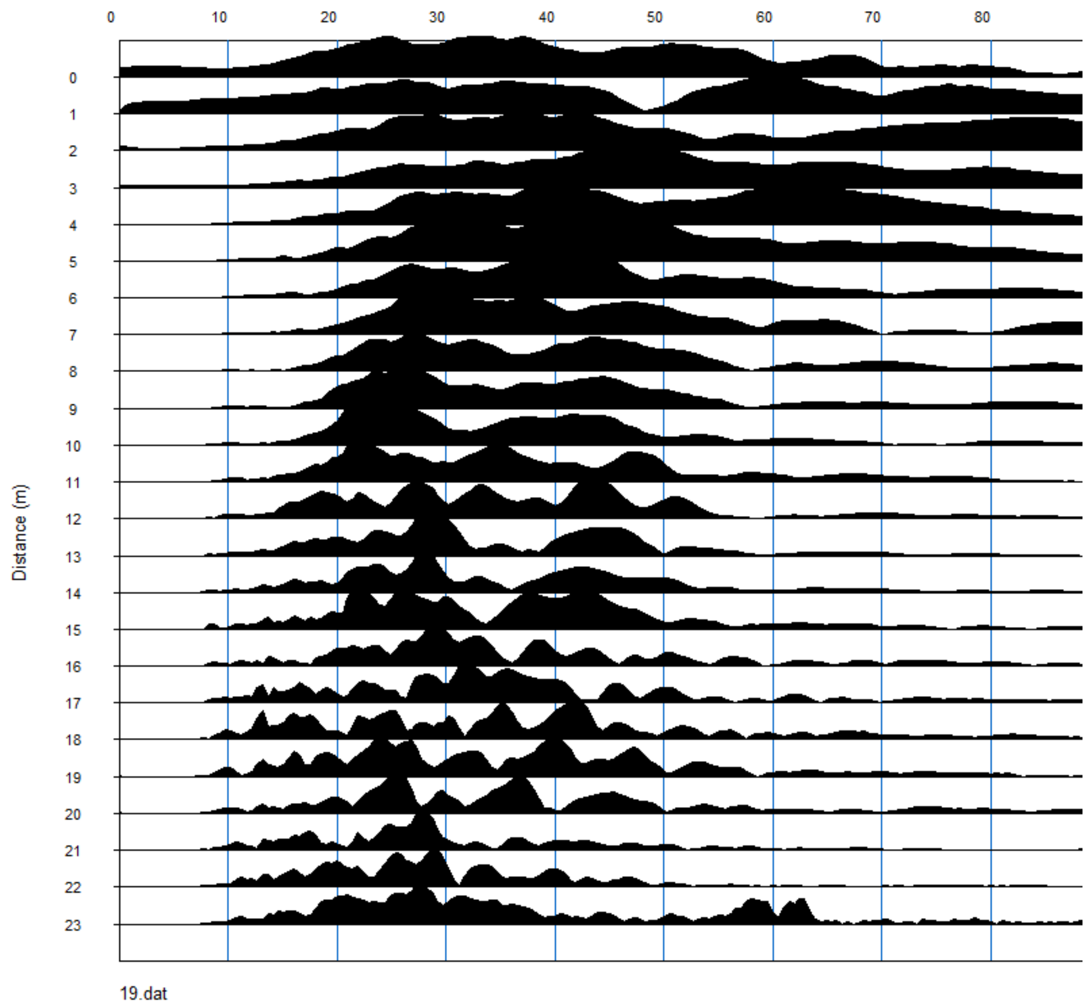
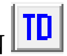


Figure 38: Seismic data shown in the frequency domain.

3.10.31 TIME DOMAIN TOOL BUTTON 

Press this button to revert to the time domain:

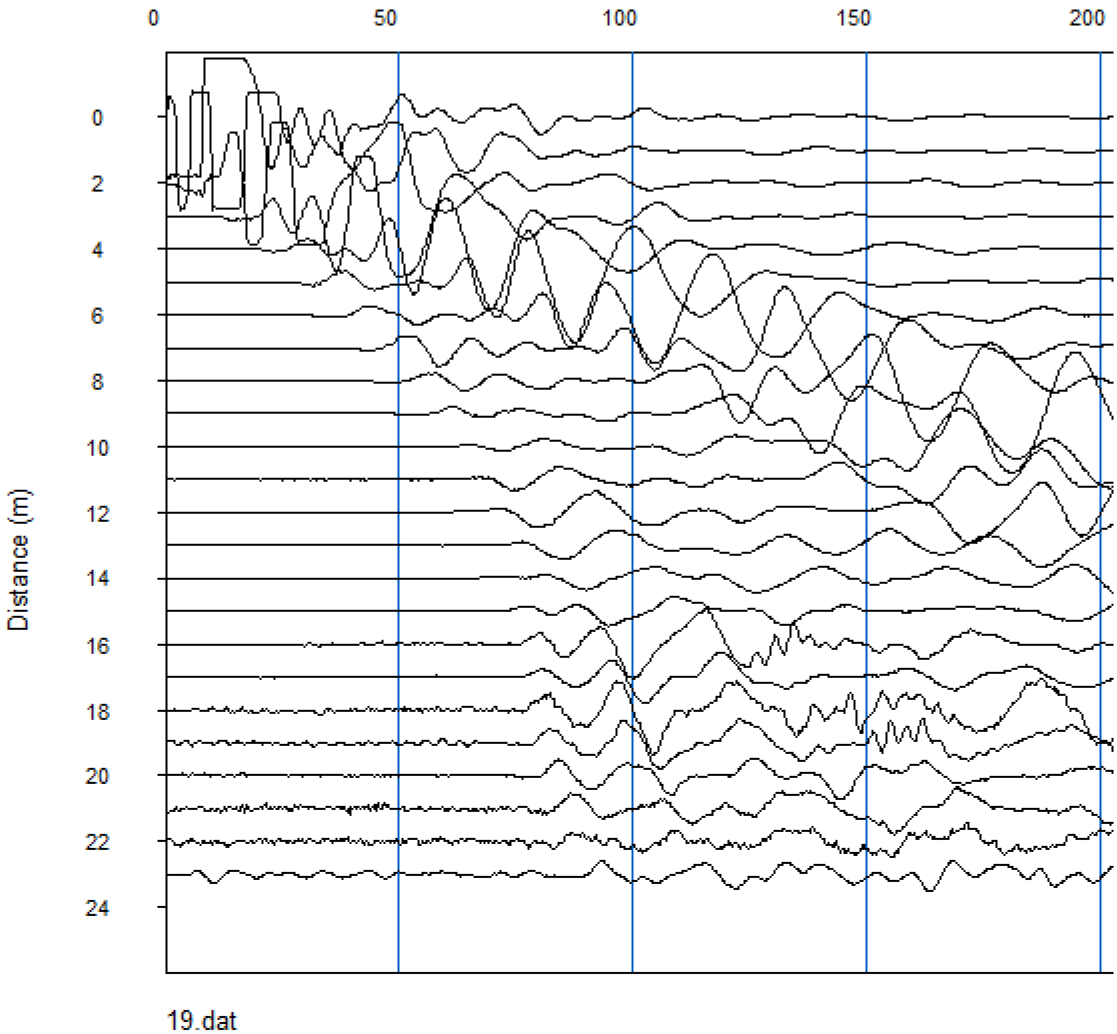
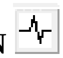


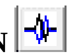
Figure 39: Trace display.

3.10.32 REMOVE FILE FROM FILE LIST TOOL BUTTON 

When working with a file list, use this button to remove a file from the list. Simply scroll to the file you wish to remove, and press . This can be handy if you make a group file list (see Section 3.1.12, Page 21) and then discover that one or more of the files is noisy or otherwise unusable.

3.10.33 Show Each Original File Tool button 

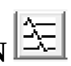
Please refer to the separate SeisImager/DH manual for explanation of this tool button. It generally does not apply to seismic refraction.

3.10.34 Show Left and Right Files Tool button 

Please refer to the separate SeisImager/DH manual for explanation of this tool button. It generally does not apply to seismic refraction.

3.10.35 Show Pre-processed Downhole Files Tool button 

Please refer to the separate SeisImager/DH manual for explanation of this tool button. It generally does not apply to seismic refraction.

3.10.36 Show P Waveform Tool button 

Please refer to the separate SeisImager/DH manual for explanation of this tool button. It generally does not apply to seismic refraction.

3.10.37 Color/Monochrome Tool Button 

Please refer to the separate SeisImager/SW manual for explanation of this menu item. It is generally not used for refraction.

3.10.38 Show Maximum Amplitude Tool Button 

See Section 3.3.10.1, Page 84.

4 THE PLOTREFA MODULE

Plotrefa is the interpretation module of SeisImager/2D. It takes the output of Pickwin as input, and through the application of one of the three available interpretation techniques, provides a velocity cross-section. It includes many useful tools for facilitating data interpretation. We will step through the various menu items in a fashion like that above, and then apply each of the three interpretation techniques to the same dataset.

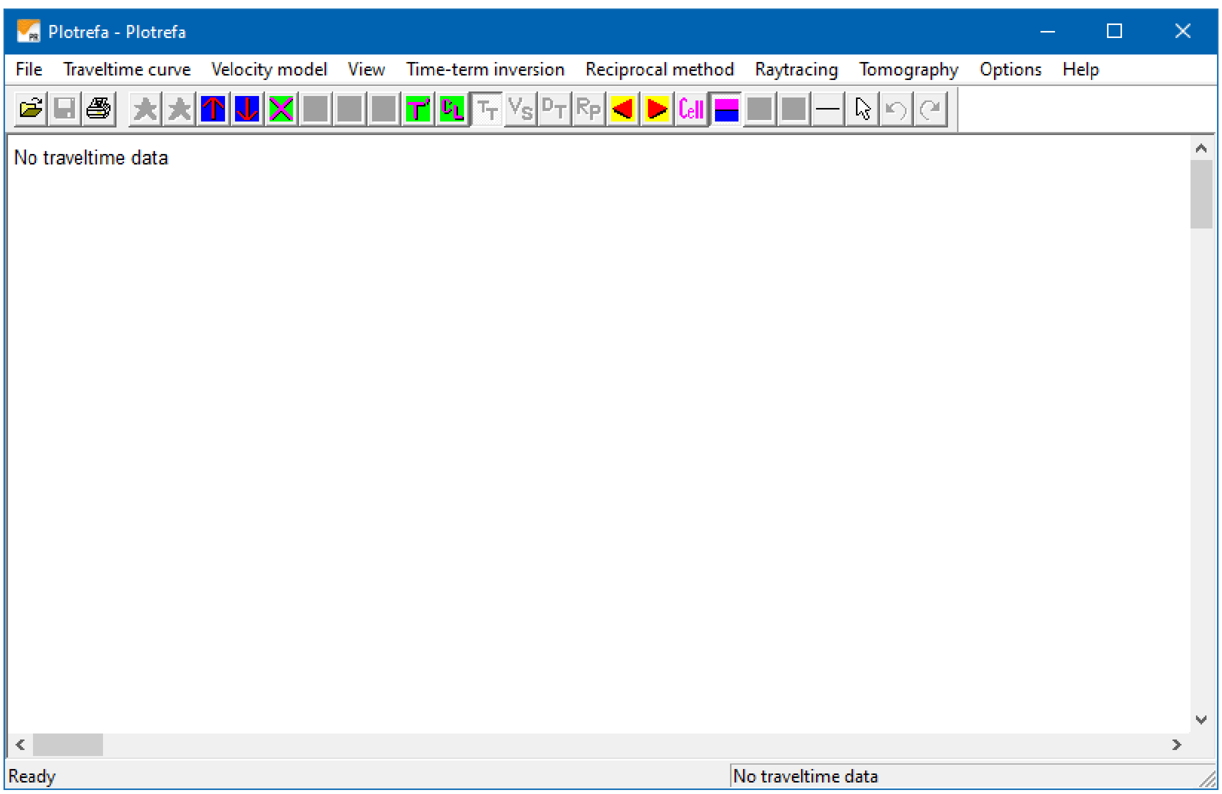
Prior to reading this chapter, you might find it useful to view the following video clips on using the Plotrefa module:

Seismic Refraction Training 2-2: Data Processing – Plotrefa (10:14)

Seismic Refraction Training 2-3: Data Processing – Plotrefa (7:32)

They will provide valuable context for what follows.

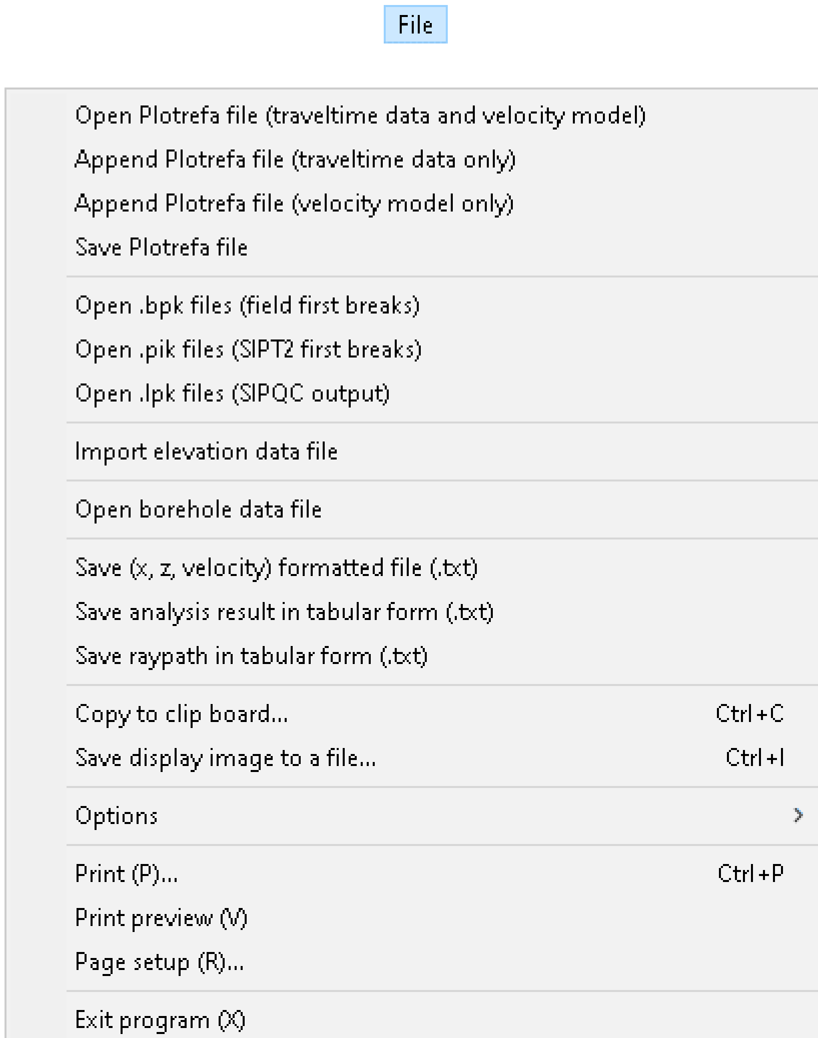
Click on the PR shortcut to start Plotrefa. You will see the following:



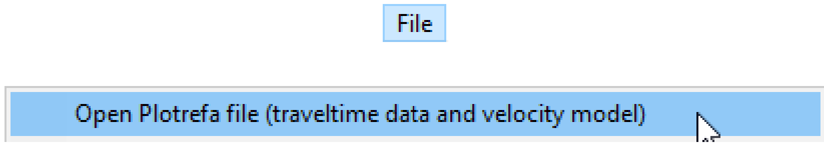
Like Pickwin, the user-interface of Plotrefa consists of a series of menus along with a toolbar. We will now discuss in detail the various menus of Plotrefa.

4.1 FILE MENU

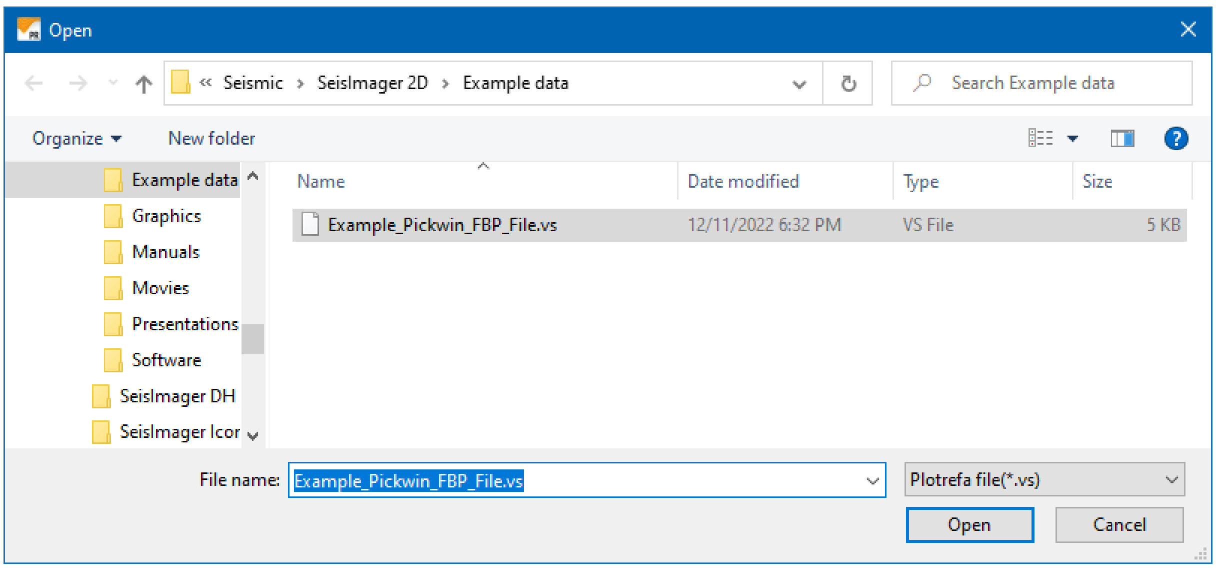
Click on *File* to reveal the **File** menu:



4.1.1 Open Plotrefa File (Travel Time Data and Velocity Model) []



To open a Plotrefa file, click on *Open Plotrefa file (travel time data and velocity model)* or press the "Open file" tool button  to read in a record. You will see a dialog box like the following:



Find the folder your data resides in and open it. Plotrefa files from Pickwin have a .vs extension, so this is the default, and only .vs files will be displayed. Choose the file you want to read in by double-clicking on it. You will see a travel time plot like the one below:

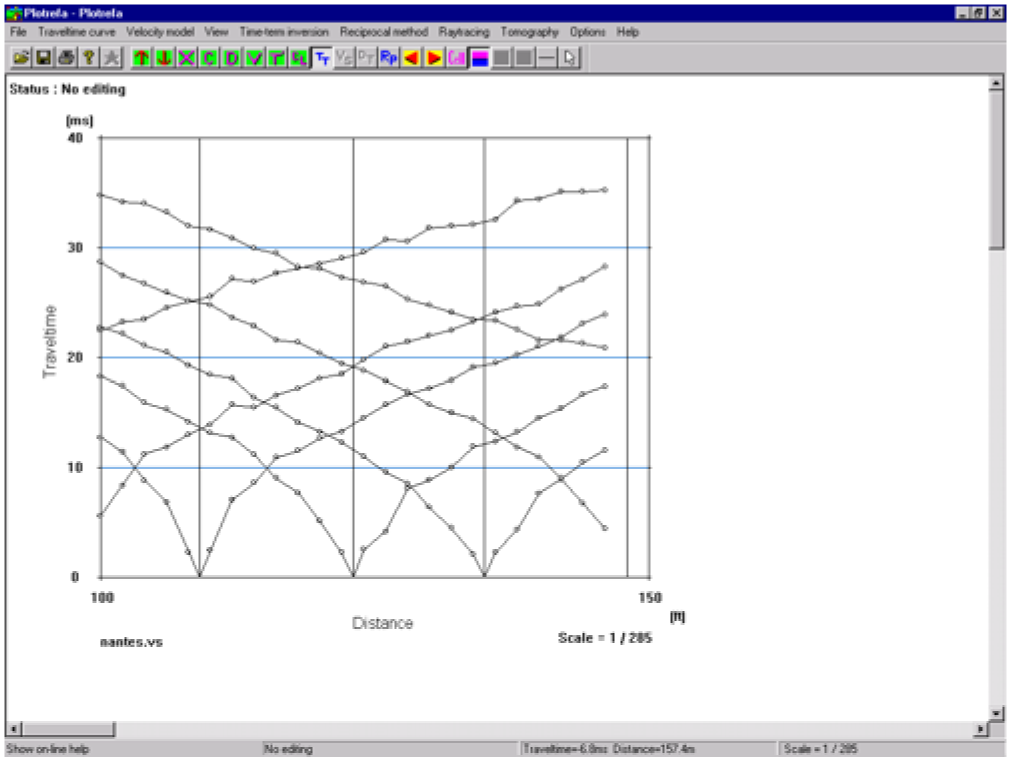
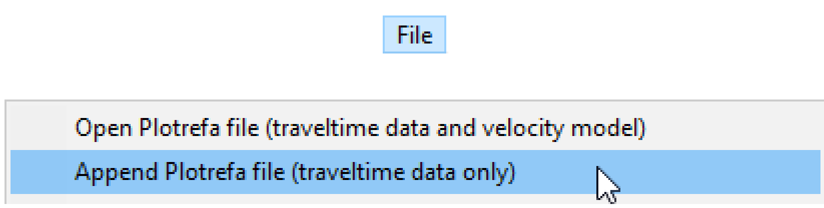


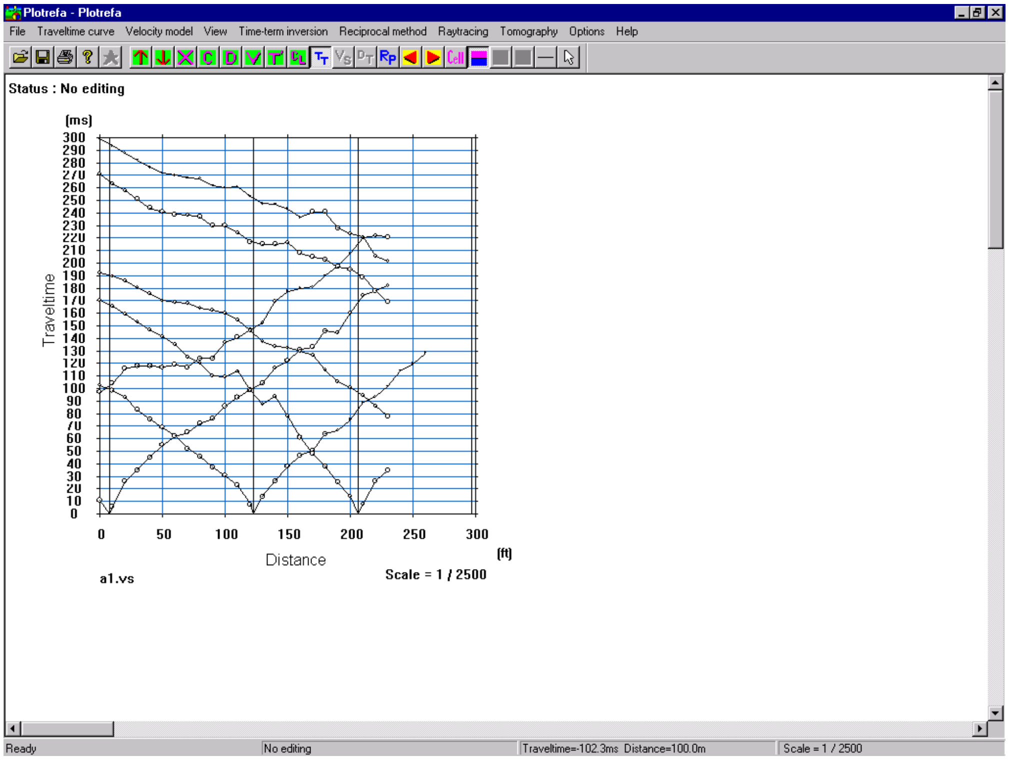
Figure 40: First arrival travel time plot.

Note: *Initial Plotrefa files written by Pickwin are travel time files only. As you advance through the interpretation, the Plotrefa file will have additional data added to it, such as elevations, layer assignments, and a velocity model.*

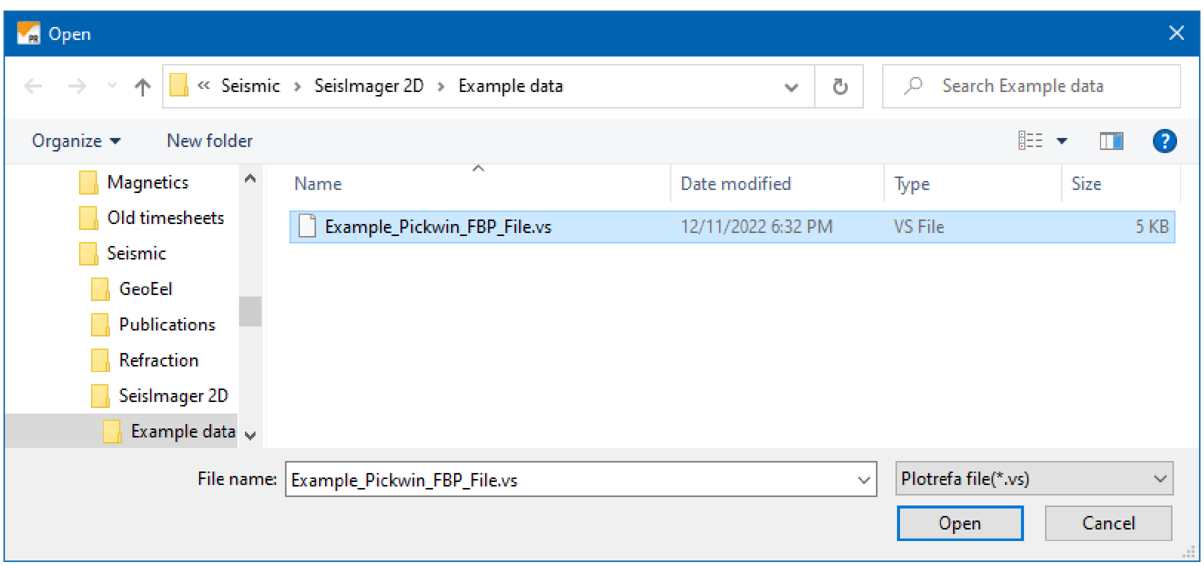
4.1.2 APPEND PLOTREFA FILE (TRAVEL TIME DATA ONLY)



If you have multiple spreads, you may append the travel time data together in Plotrefa. Open the first Plotrefa file:



Next, click on *Append Plotrefa file*. You will be presented with a dialog box like the one shown below:



Choose the appropriate .vs file to append, and double-click:

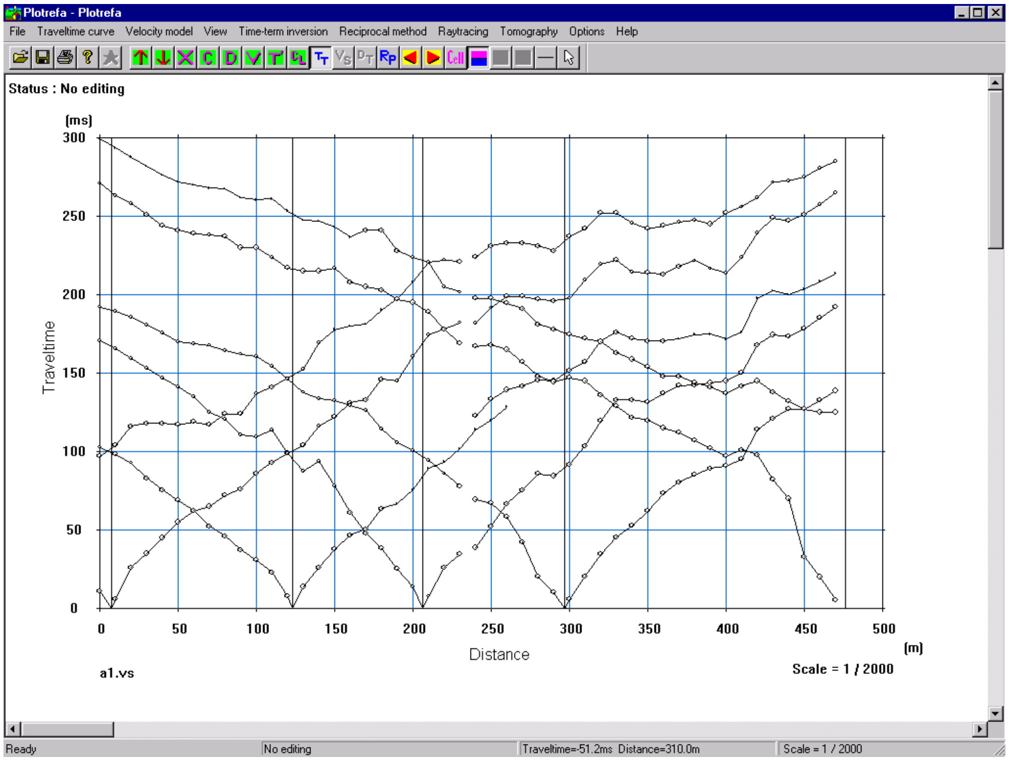
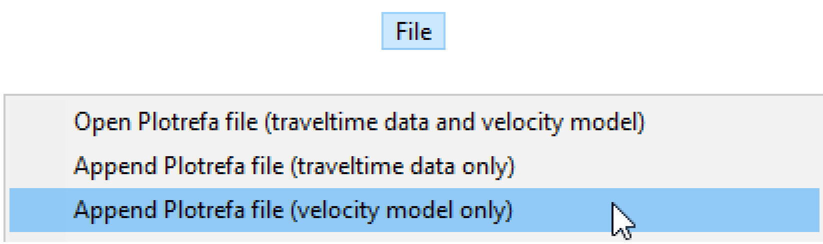


Figure 41: Appended seismic refraction spreads.

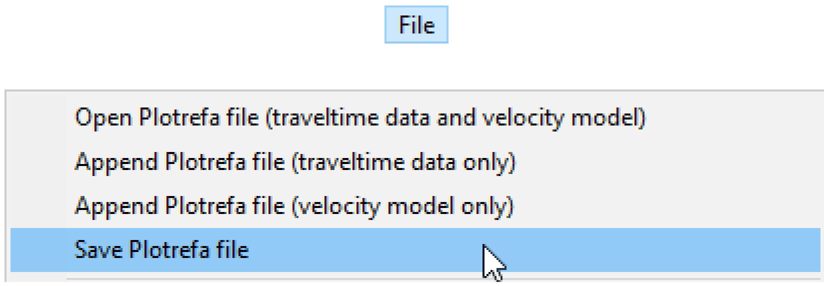
The two files will be appended together as shown above. You may append any number of files.

4.1.3 APPEND PLOTREFA FILE (VELOCITY MODEL ONLY)

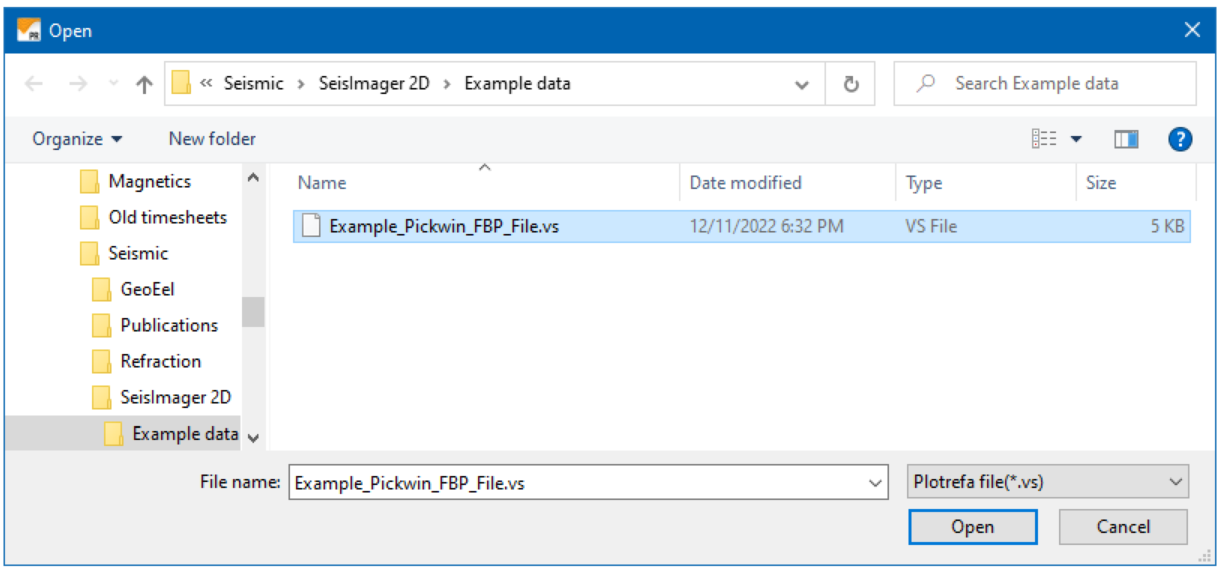


In a manner like that explained immediately above, you may append multiple velocity models together.

4.1.4 Save Plotrefa File [ ]

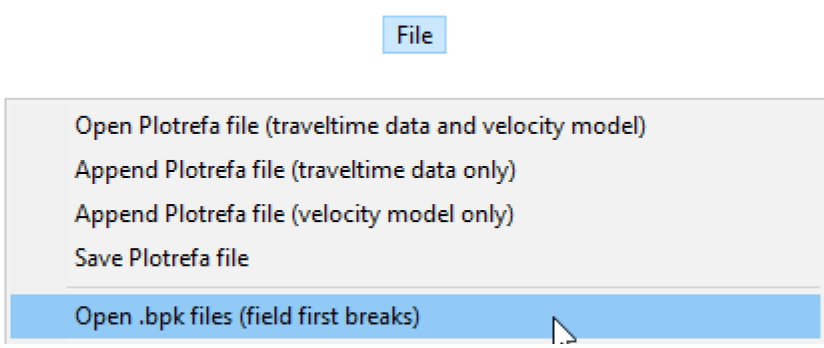


To save a Plotrefa file after editing, choose *Save Plotrefa file*, or press the "Save file" tool button. You will be presented with a dialog box like the one below:

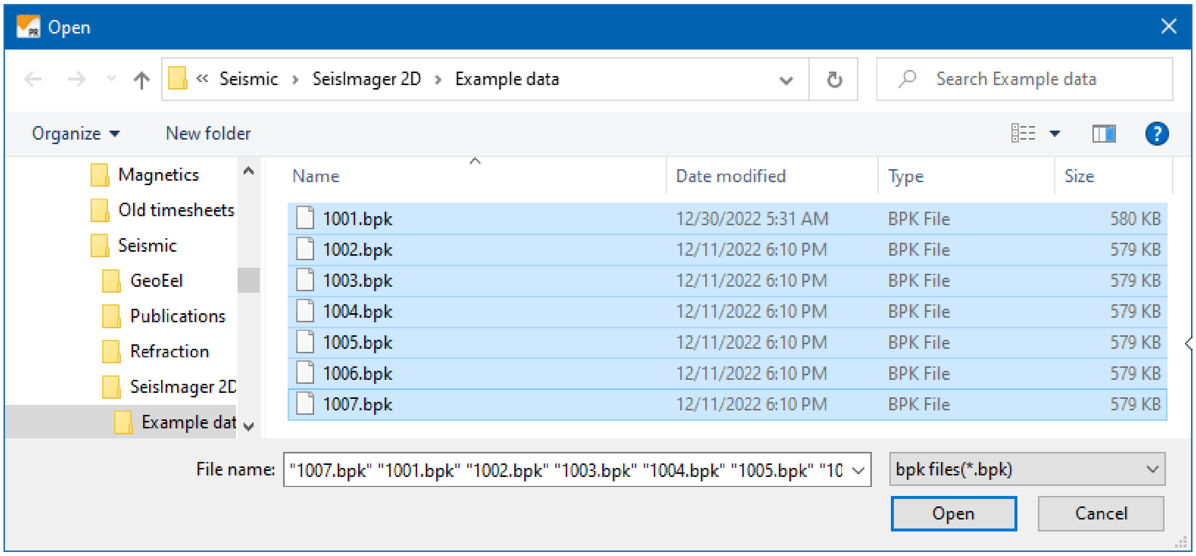


Type in a file name and press *Save*. The extension will default to .vs.

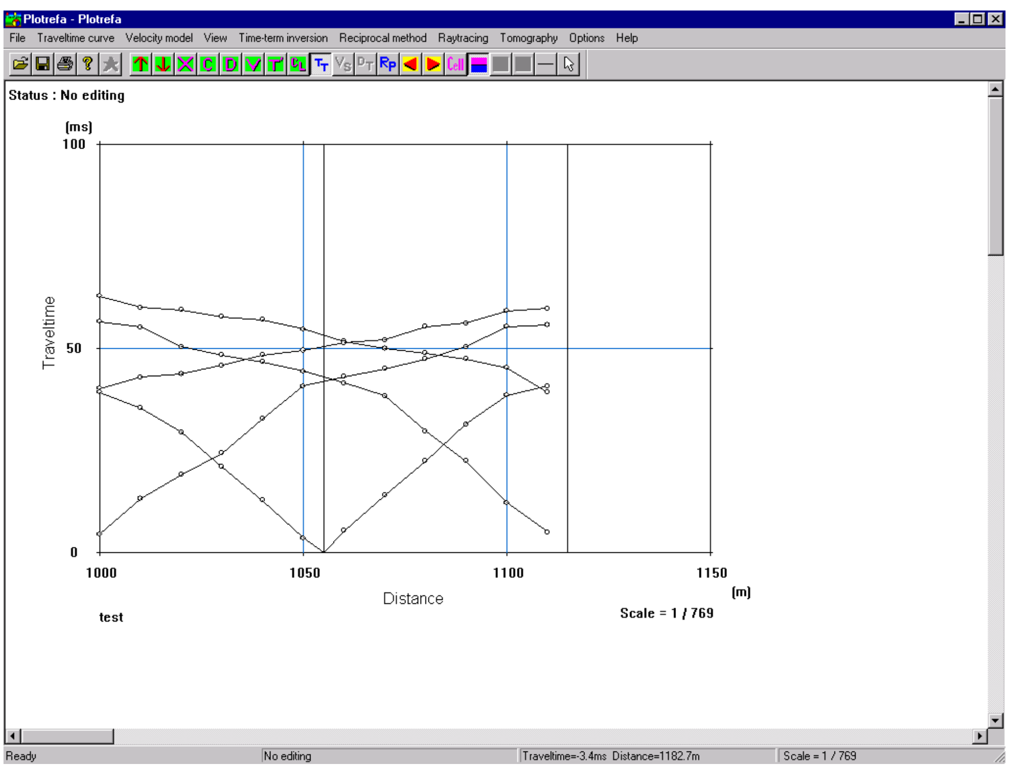
4.1.5 Open .bpk Files (Field First Breaks)



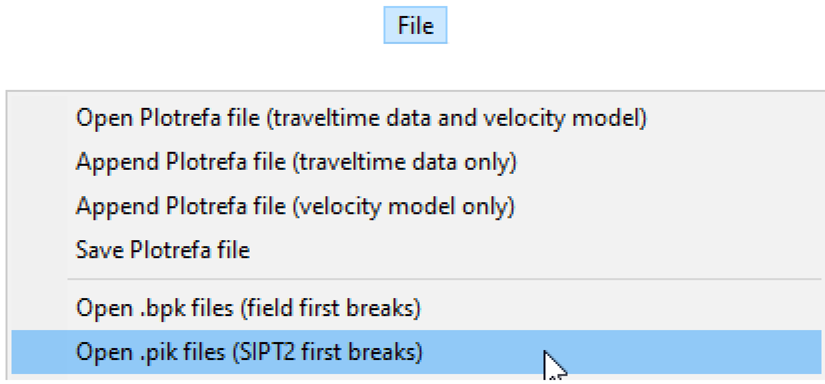
If you picked and saved your first breaks in the field using the Geometrics first break picker, you may read them in here – there is no need to re-pick them. Click on *Open bpk files (field first breaks)*. A dialog box will appear, displaying only .bpk files. Choose the files you would like to read in by holding down the *Control* key and clicking on them:



Press *Open* to display the travel times:

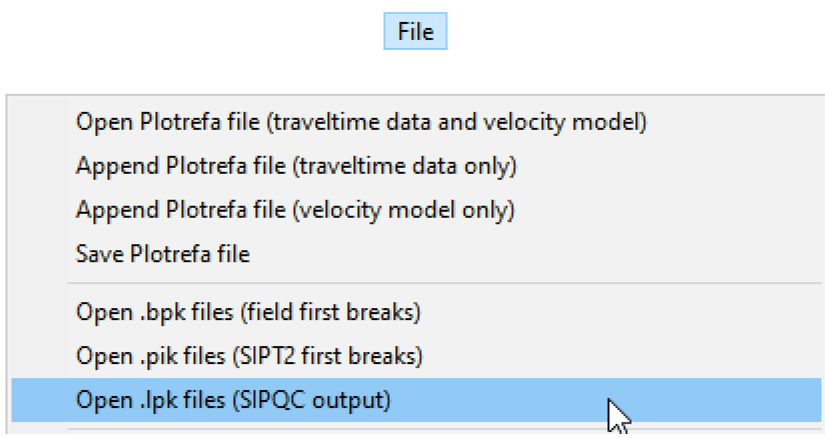


4.1.6 OPEN .PIK FILES (SIPT2 FIRST BREAKS)



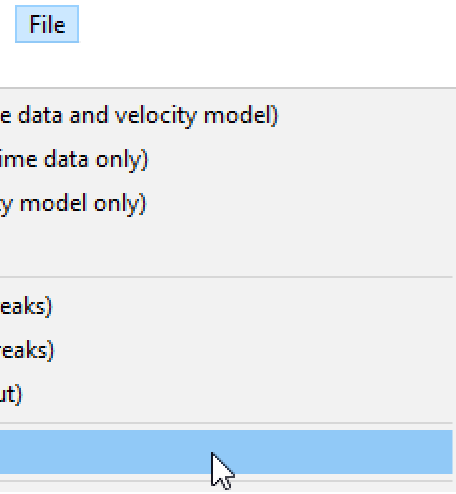
If the data has been picked with the RimRock Geophysics SIPT2 picker (older models of Geometrics seismographs), the resulting .pik files may be read in by Plotrefa in the same fashion as .bpk files (previous section).

4.1.7 OPEN .LPK FILES (SIPQC OUTPUT)

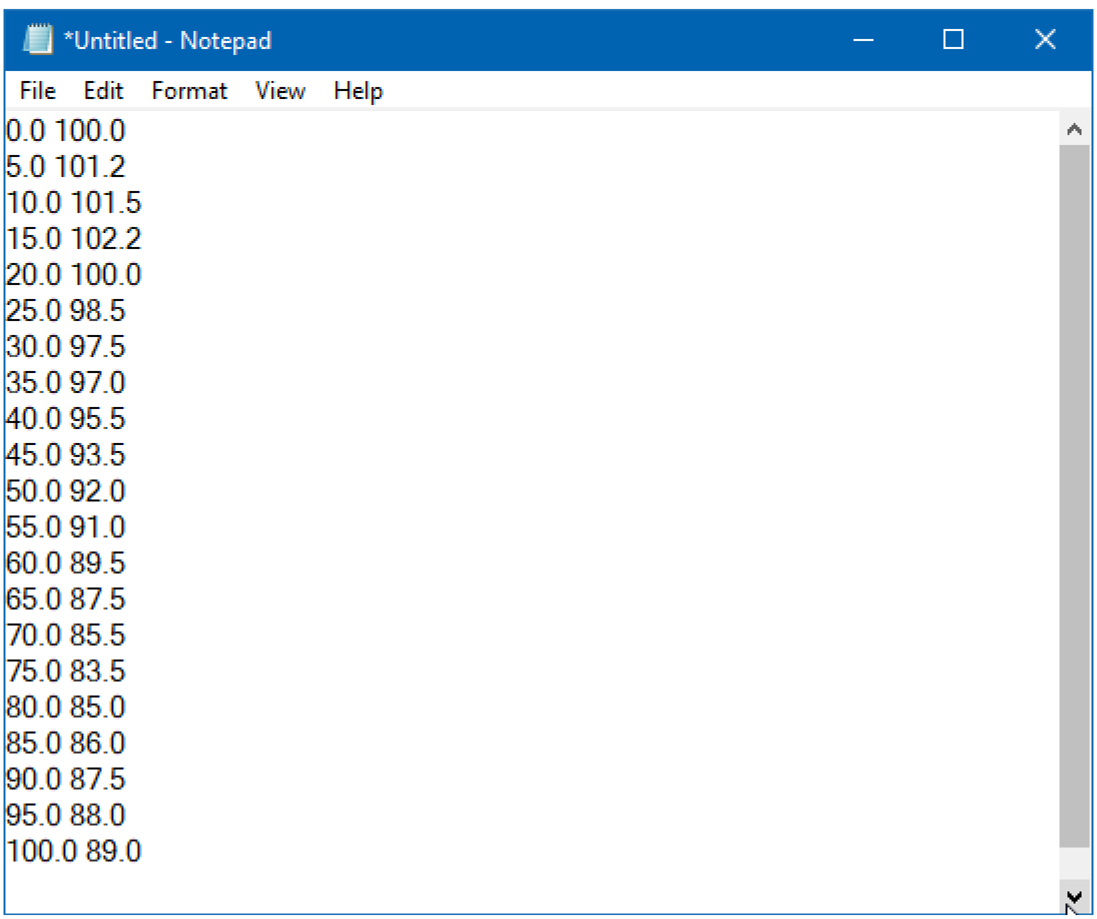


If you ran SIPQC in the field and assigned layers to the arrivals, an .lpk file was created. This simply groups together all the travel times for the spread into one file. To read in an .lpk file, choose *Open lpk files (SIPQC output)* and double-click on the appropriate .lpk file. You will see a travel time plot consisting of several shots, like the one shown in the previous section.

4.1.8 Import Elevation Data File



If you measured relative or absolute geophone elevations, these should be stored in an ASCII-columnar file as shown below:



The left column is the geophone location, and the right column is the elevation. You may read in this elevation file and incorporate it into your velocity model. Click on *Import elevation data file* and double-click on the appropriate file (there is no default extension for elevation files). The elevation profile will be displayed:

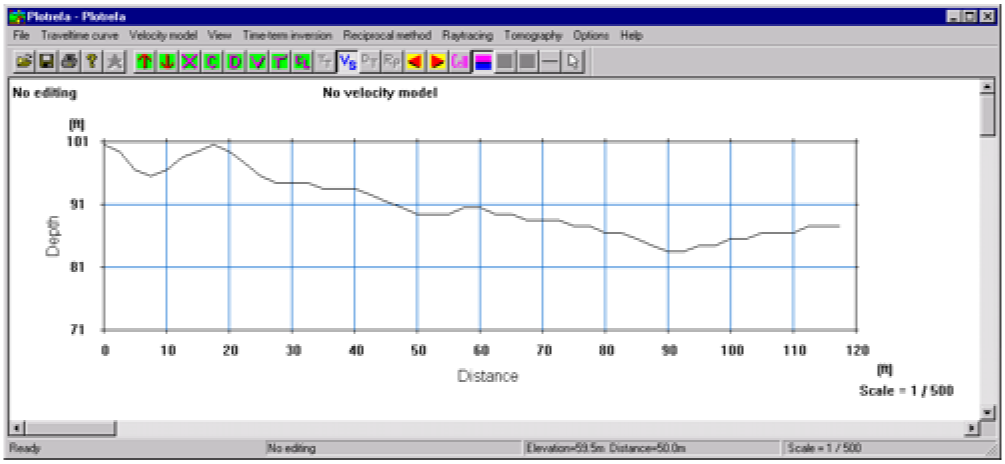


Figure 42: Surface topography profile.

Note: *Generally, it is advisable to have an elevation for each geophone. However, it is not necessary. The program will interpolate geophone elevations from the elevation profile if required. Shot elevations are not required.*

After interpreting your data and calculating the velocity structure, it will be drawn relative to the elevation profile, as shown below:

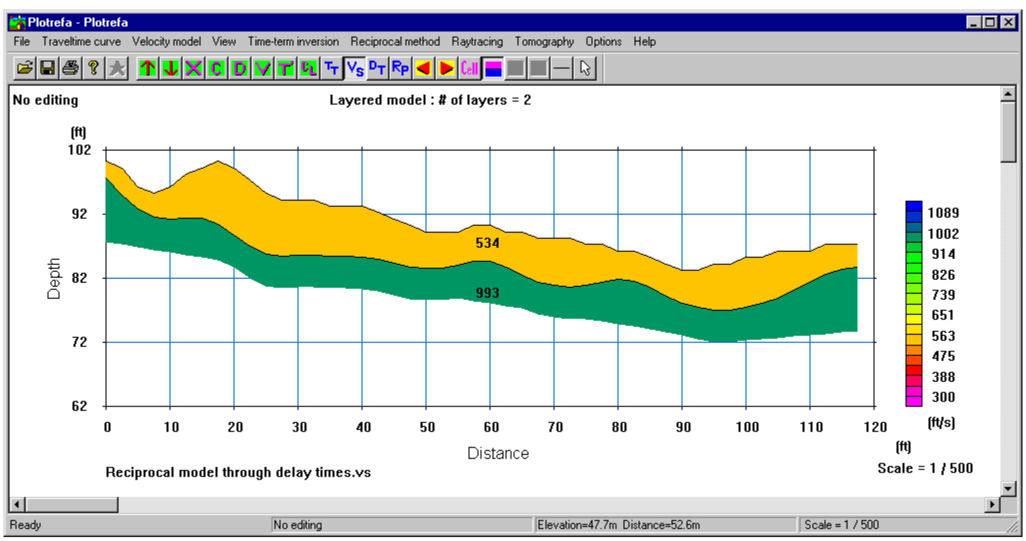
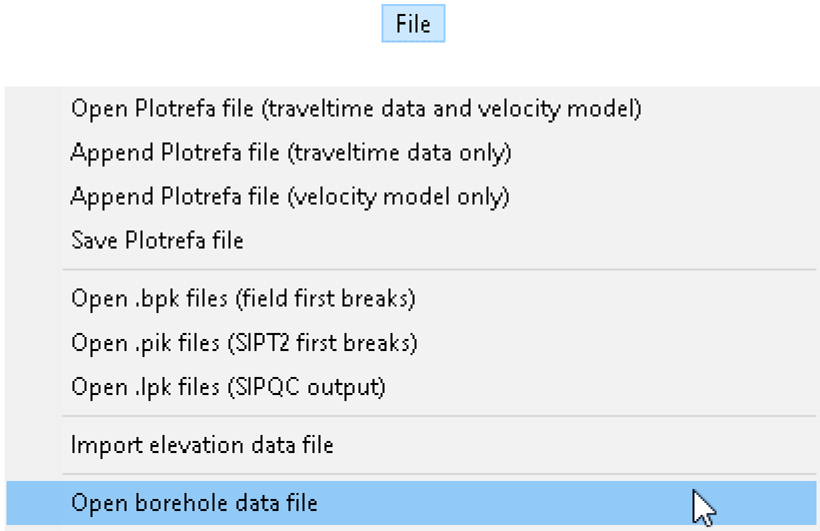


Figure 43: Seismic velocity profile output by Plotrefa.

4.1.9 OPEN BOREHOLE DATA FILE



SeisImager 2D can be used to do cross-hole tomography:

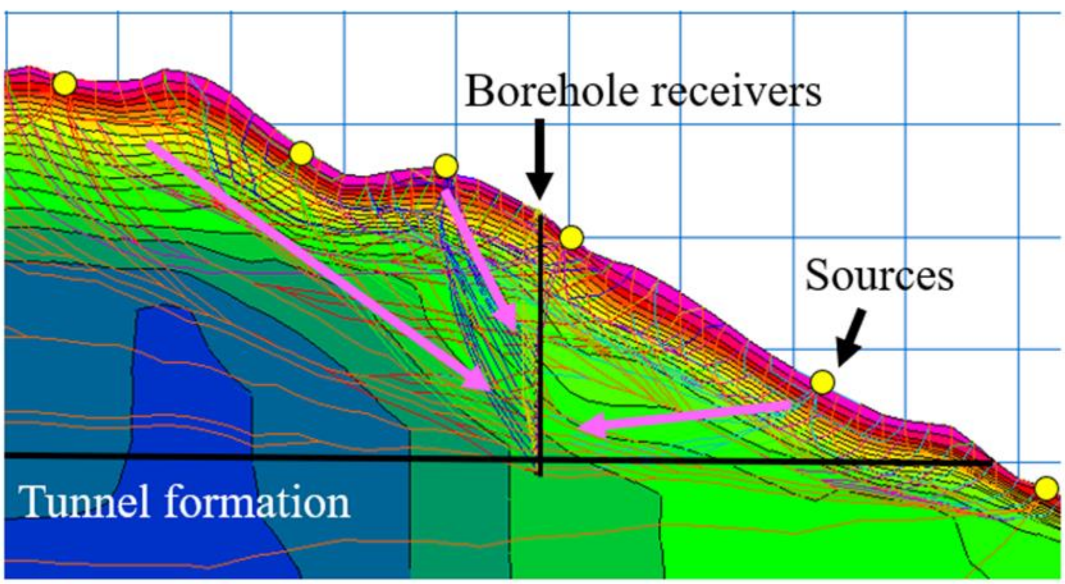


Figure 44: Cross-hole tomography with Plotrefa.

Select this option to open a borehole data file. An example is shown below:

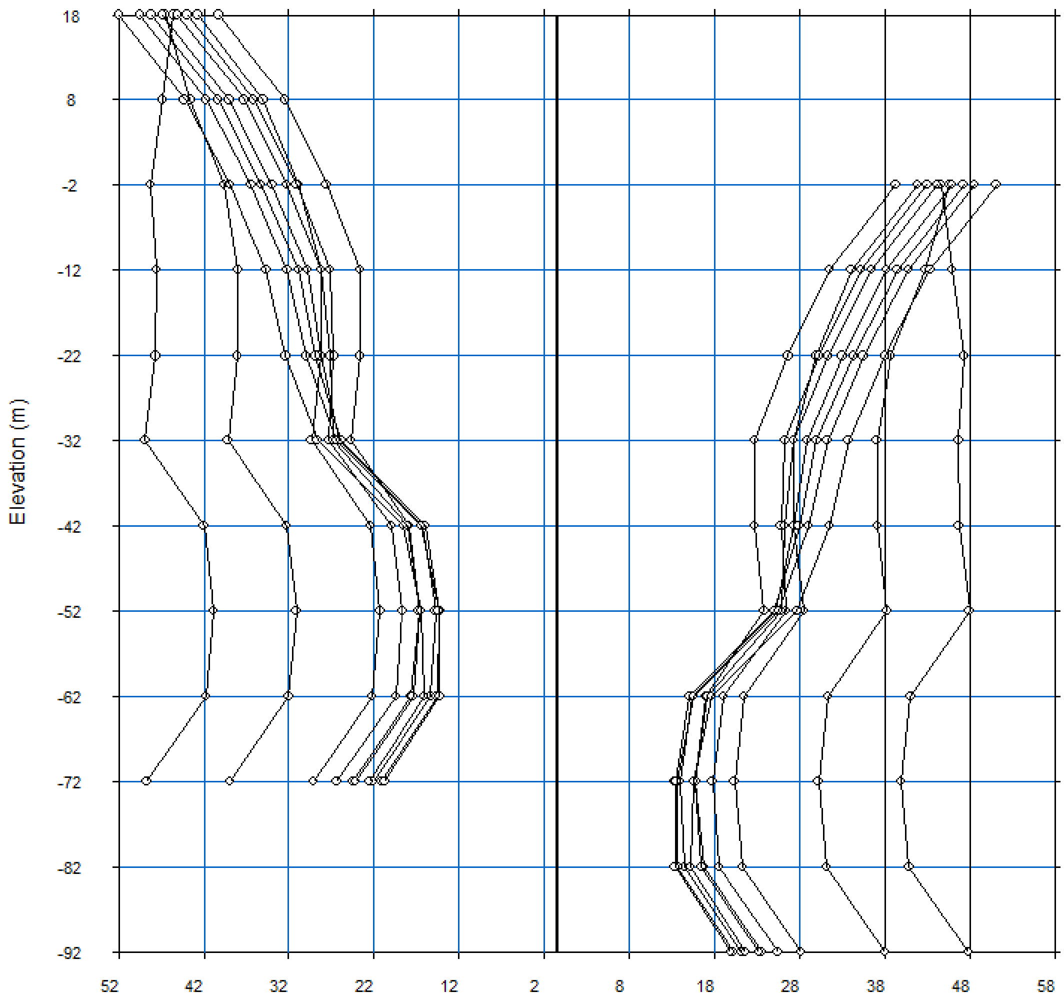
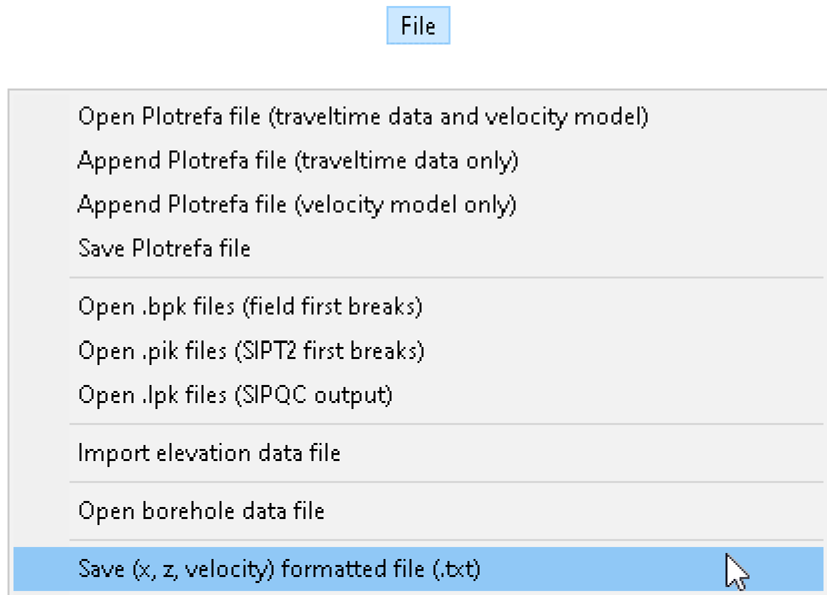


Figure 45: Sample borehole data.

This tutorial explains how to process cross-hole data with SeisImager/2D.

4.1.10 SAVE (X, Z, VELOCITY) FORMATTED FILE (.TXT)

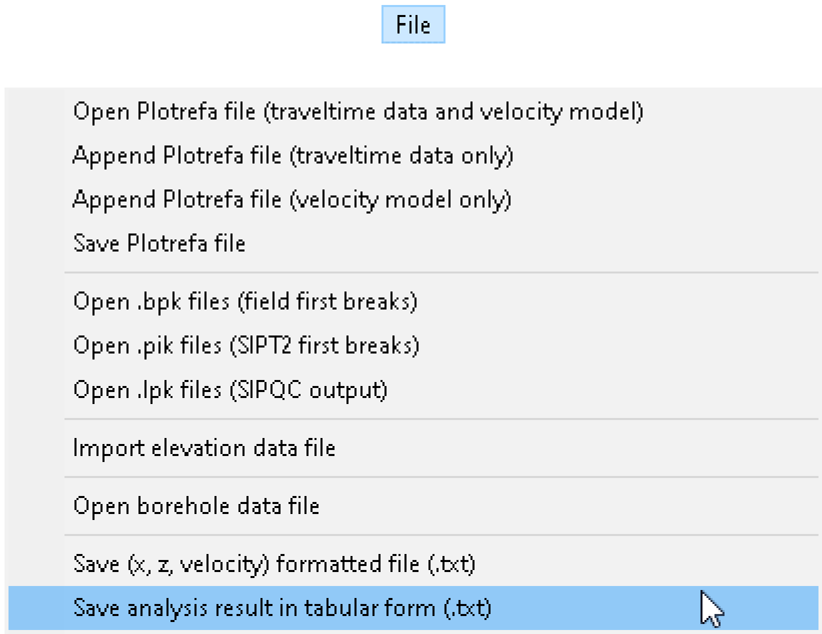


Once you have inverted your data, you may save your velocity model in an ASCII-columnar format for direct import into third-party graphics programs such as Surfer™. A partial x, z, velocity file is shown below.

70.000000	0.000000	0.286104
70.000000	-3.427992	1.359415
98.000000	0.000000	0.286104
98.000000	-3.175648	1.359415
100.000000	0.000000	0.286104
100.000000	-3.036011	1.359415
102.000000	0.000000	0.286104
102.000000	-2.982426	1.359415
104.000000	0.000000	0.286104
104.000000	-2.938553	1.359415
106.000000	0.000000	0.286104
106.000000	-2.863396	1.359415
108.000000	0.000000	0.286104
108.000000	-2.796365	1.359415
109.000000	0.000000	0.286104
109.000000	-2.757409	1.359415
110.000000	0.000000	0.286104
110.000000	-2.662040	1.359415
112.000000	0.000000	0.286104
112.000000	-2.482863	1.359415
114.000000	0.000000	0.286104
114.000000	-2.349191	1.359415
116.000000	0.000000	0.286104

Table 1: Velocity model in ASCII-columnar format.

4.1.11 SAVE ANALYSIS RESULT IN TABULAR FORM (.TXT)



If you wish to output your data in a tabular format, choose *Save analysis result in tabular form*:

The spread contains 7 shotpoints and 24 geophones

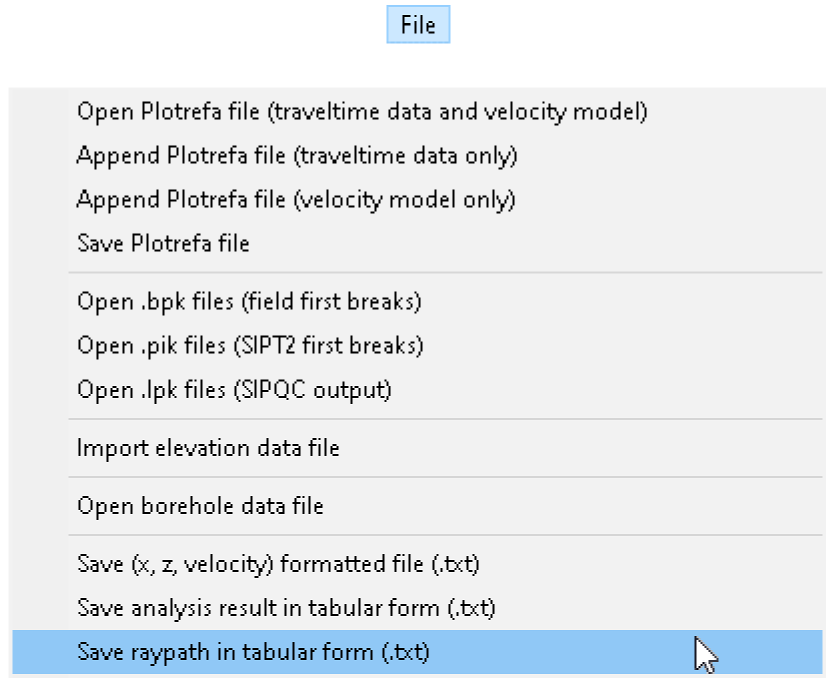
SP	Elev	X-loc	Y-Loc	Depth
1	-3.62	70.00	0.00	0.00
2	0.69	98.00	0.00	0.00
3	6.50	109.00	0.00	0.00
4	4.00	123.00	0.00	0.00
5	-5.00	135.00	0.00	0.00
6	1.37	148.00	0.00	0.00
7	6.54	176.00	0.00	0.00

Geo	Elev	X-loc	Y-Loc	SP 1	SP 2	SP 3	SP 4	SP 5	SP 6	SP 7
1	1.00	100.00	0.00	22.31 1	5.44 1	13.13 1	18.25 1	22.38 1	28.13 1	34.00 1
2	2.00	102.00	0.00	22.50 1	8.69 1	11.69 1	17.50 1	21.81 1	27.38 1	33.81 1
3	4.00	104.00	0.00	22.94 1	11.19 1	9.00 1	16.50 1	21.00 1	26.50 1	32.81 1
4	6.00	106.00	0.00	23.69 1	12.38 1	7.29 1	15.63 1	20.50 1	25.75 1	32.75 1
5	7.00	108.00	0.00	24.25 1	13.69 1	2.50 1	14.31 1	18.94 1	24.81 1	31.88 1
6	6.00	110.00	0.00	25.00 1	14.31 1	2.94 1	13.38 1	18.94 1	23.88 1	31.38 1
7	7.00	112.00	0.00	26.69 1	15.21 1	7.81 1	13.38 1	18.63 1	24.25 1	31.50 1
8	8.00	114.00	0.00	26.06 1	15.48 1	8.81 1	11.00 1	16.69 1	22.56 1	30.25 1
9	10.00	116.00	0.00	27.00 1	16.75 1	11.06 1	9.13 1	15.81 1	21.69 1	30.00 1
10	8.00	118.00	0.00	27.44 1	18.00 1	11.69 1	7.69 1	14.63 1	20.81 1	28.00 1
11	7.00	120.00	0.00	28.00 1	17.81 1	12.94 1	4.94 1	13.50 1	20.25 1	27.50 1
12	5.00	122.00	0.00	28.38 1	18.25 1	13.69 1	2.06 1	12.13 1	19.88 1	26.88 1
13	3.00	124.00	0.00	29.50 1	20.06 1	14.77 1	2.44 1	11.31 1	18.81 1	26.69 1
14	0.00	126.00	0.00	30.13 1	20.50 1	15.88 1	4.77 1	9.88 1	18.31 1	26.00 1
15	-2.00	128.00	0.00	30.88 1	21.06 1	16.94 1	8.31 1	8.69 1	16.63 1	25.13 1
16	-3.00	130.00	0.00	31.13 1	21.44 1	16.81 1	9.25 1	6.56 1	15.69 1	24.19 1
17	-4.00	132.00	0.00	31.31 1	22.19 1	18.19 1	9.88 1	4.31 1	15.38 1	23.75 1
18	-5.00	134.00	0.00	31.88 1	22.81 1	18.38 1	11.50 1	2.52 1	14.25 1	23.56 1

SP	X-loc	Layer 2
1	70.00	3.14
2	98.00	3.14
3	109.00	3.49
4	123.00	3.89
5	135.00	2.15
6	148.00	4.94
7	176.00	4.94
Geo		
1	100.00	3.14
2	102.00	3.43
3	104.00	4.11
4	106.00	4.49
5	108.00	4.28
6	110.00	2.70
7	112.00	3.30
8	114.00	3.45
9	116.00	4.77
10	118.00	3.21
11	120.00	3.67
12	122.00	3.76
13	124.00	4.01
14	126.00	2.87
15	128.00	2.26
16	130.00	2.39
17	132.00	2.36
18	134.00	2.01

Table 2: Analysis result in tabular form.

4.1.12 Save Raypath in Tabular Form (.txt)

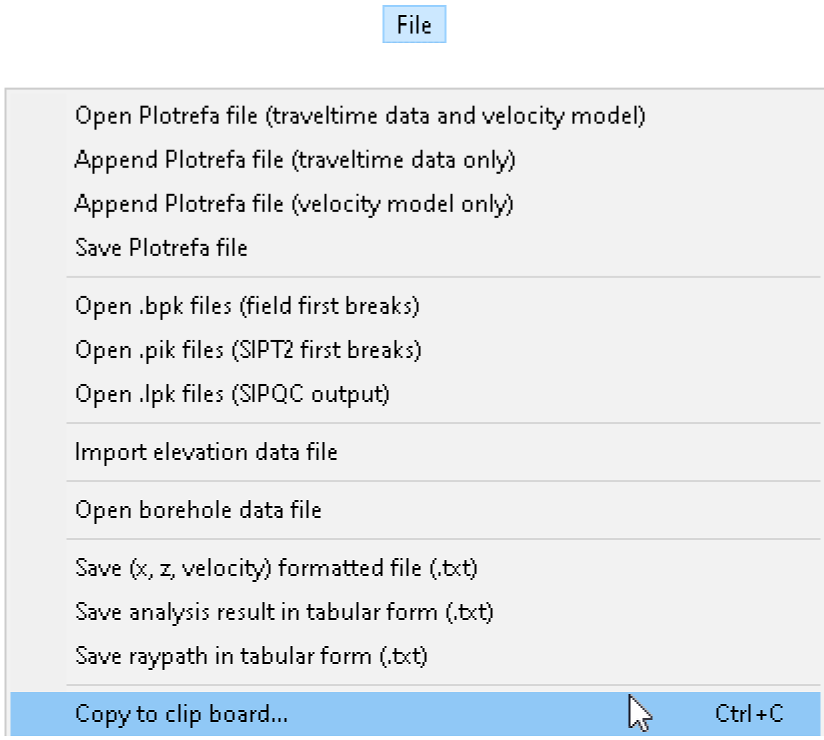


This option saves the raypath data in an ASCII file. A partial table is provided below. It can then be read into any drawing software, such as AutoCad®.

2	
70.000000	0.000000
70.000000	0.000000
16	
98.000000	0.000000
98.000000	-1.104166
98.000000	-1.545832
98.000000	-1.987498
98.000000	-2.429164
98.000000	-2.870831
98.000000	-3.312497
98.000000	-5.520829
92.400002	-7.729160
75.599998	-7.729160
70.000000	-5.520829
70.000000	-3.754164
70.000000	-3.312497
70.000000	-1.104166
70.000000	-0.662499
70.000000	0.000000
12	
100.000000	0.000000
100.000000	-1.104166
99.199997	-3.312497
98.000000	-5.520829
92.400002	-7.729160
75.599998	-7.729160
70.000000	-5.520829
70.000000	-3.754164
70.000000	-3.312497
70.000000	-1.104166
70.000000 -	0.662499
70.000000	0.000000
12	
102.000000	0.000000
102.000000	-1.104166
101.199997	-3.312497
100.000000	-5.520829
98.000000	-7.729160
75.599998	-7.729160
70.000000	-5.520829
70.000000	-3.754164
70.000000	-3.312497
70.000000	-1.104166
70.000000	-0.662499
70.000000	0.000000
14	

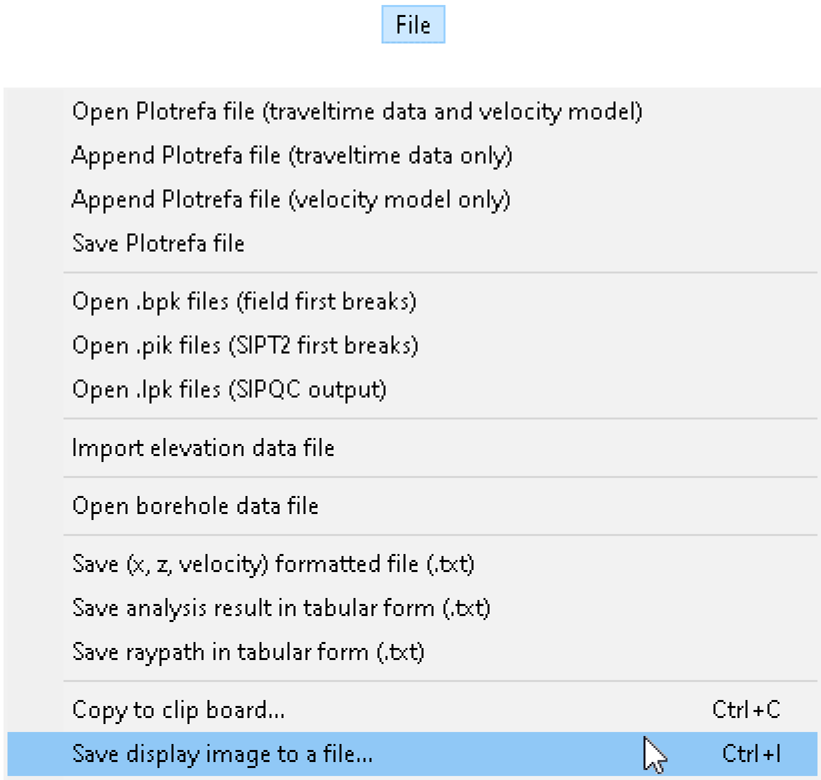
Table 3: Raypath data in tabular form.

4.1.13 Copy to Clipboard [CTRL+C]



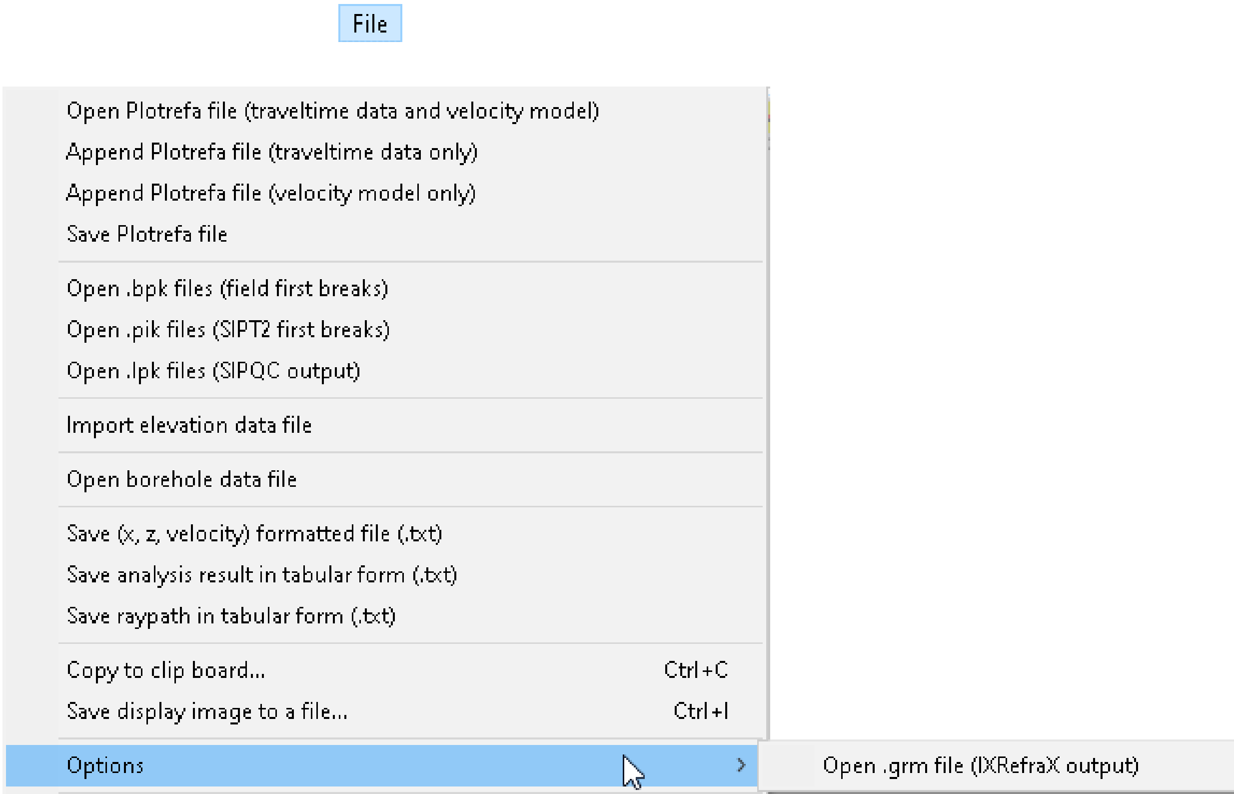
Choose this option to copy whatever is displayed on the screen to the clipboard. It can then be pasted into another application, such as a word processor.

4.1.14 SAVE DISPLAY IMAGE TO A FILE [CTRL+I]



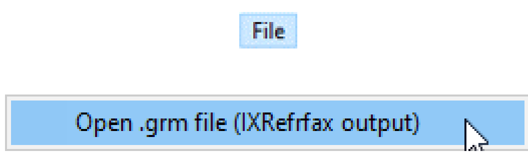
You may save whatever is displayed on the screen to a file. Choices include PNG, JPG, BMP, and GIF.

4.1.15 OPTIONS



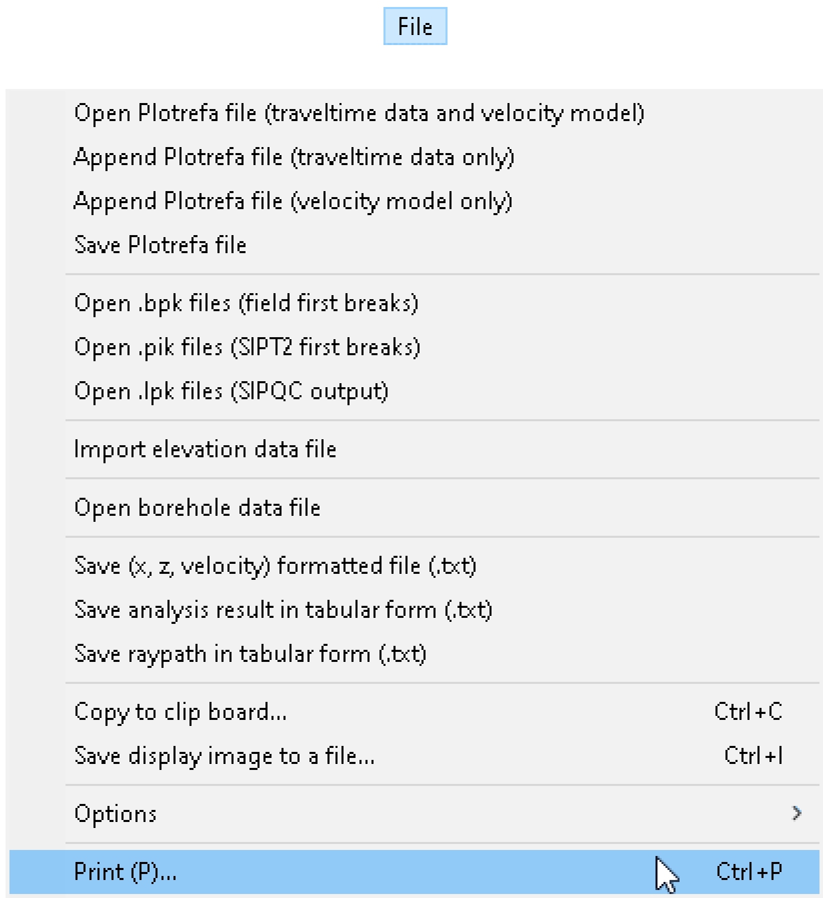
Continue.

4.1.15.1 OPEN .GRM FILE (IXREFRAX OUTPUT)

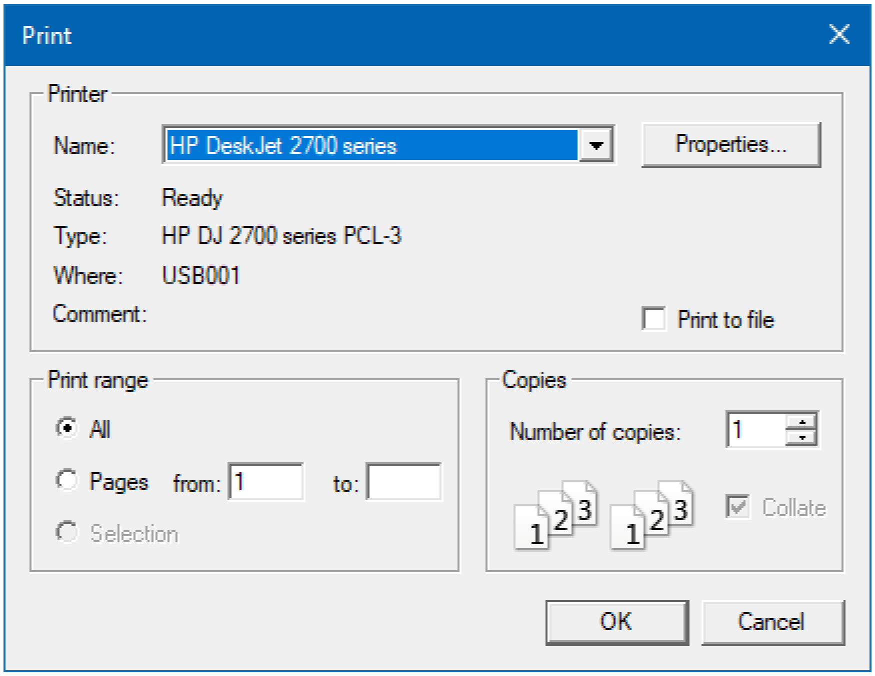


Choose this option to open an existing GRM file from the Interpex IXRefraX™ application. It can then be processed as usual.

4.1.16 PRINT [ ] [CTRL+P]

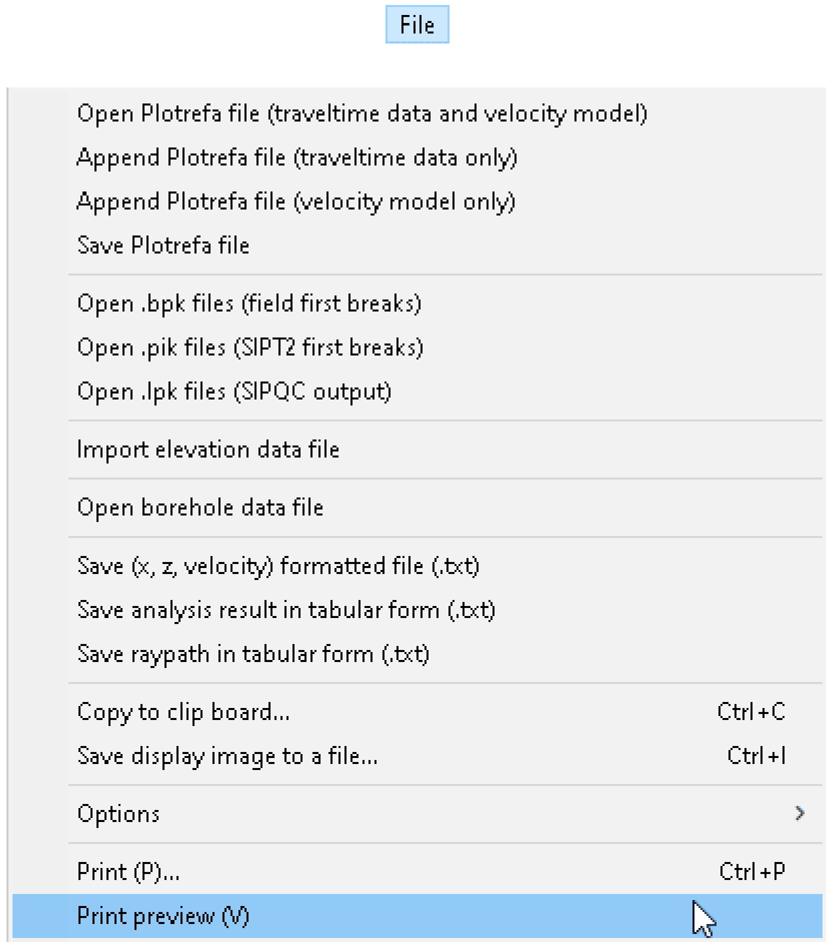


To print the window display of Plotrefa, choose *Print*, press *Ctrl-P*, or press the “Print” tool button. You will see the **Print** dialog box for your computer:



Press *Print* to print the current window display of Plotrefa.

4.1.17 PRINT PREVIEW



To preview the window display of Pickwin for printing, choose *Print preview*. You will see a preview of the window display that will be printed:

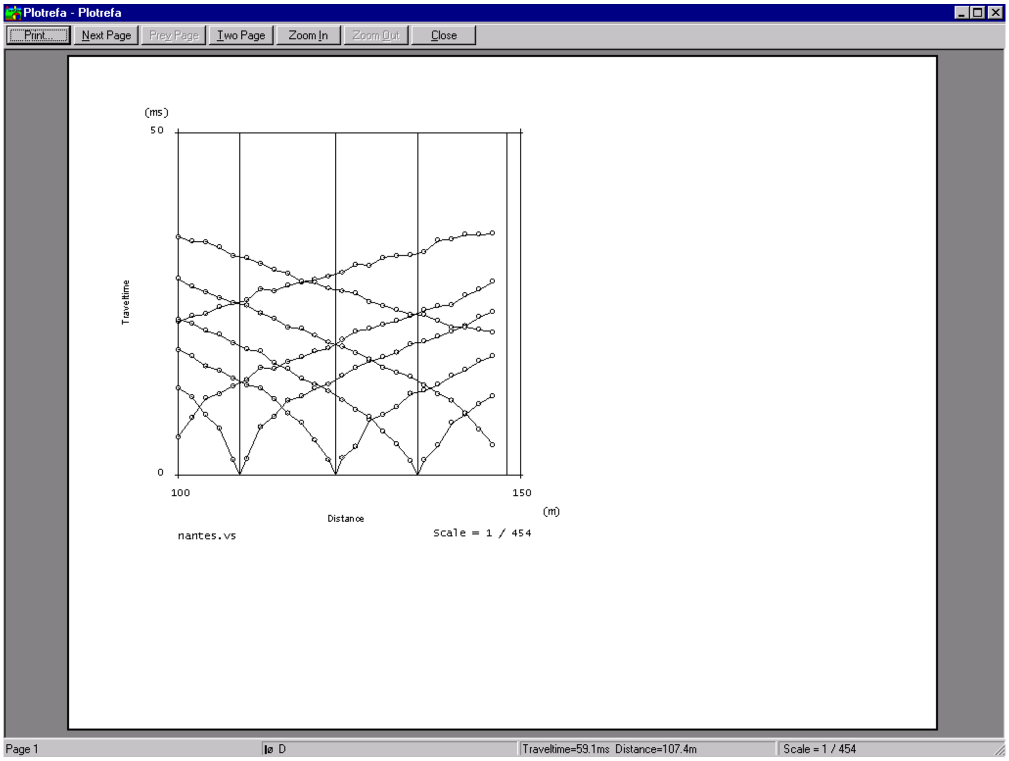
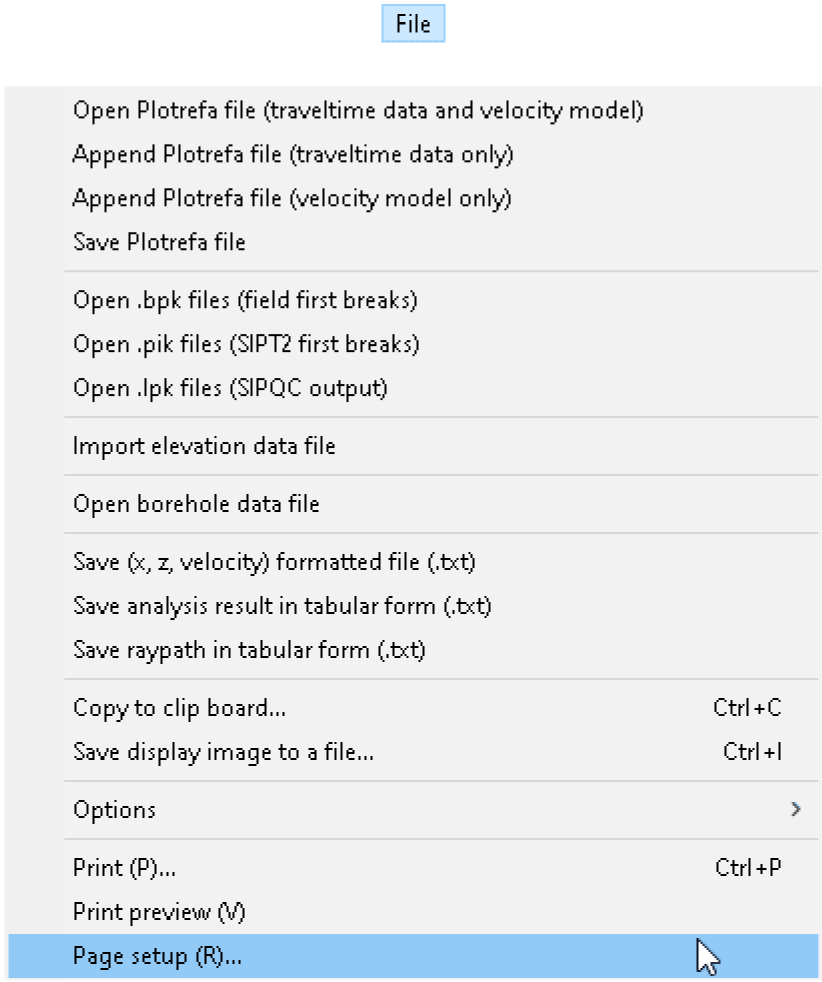


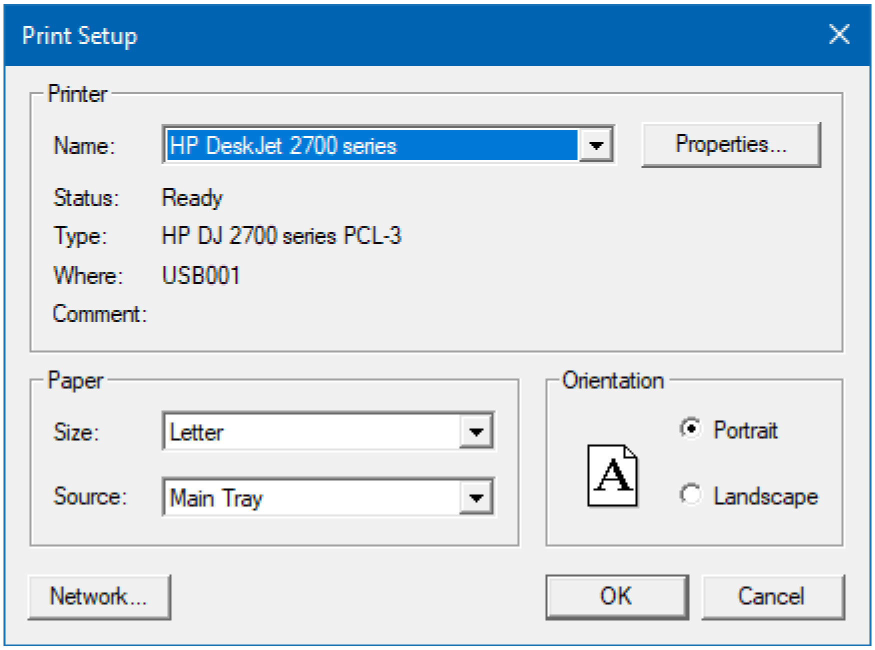
Figure 46: Travel time plot ready for printing.

To print this display, press *Print*. To close this display, press *Close.*

4.1.18 PAGE SETUP

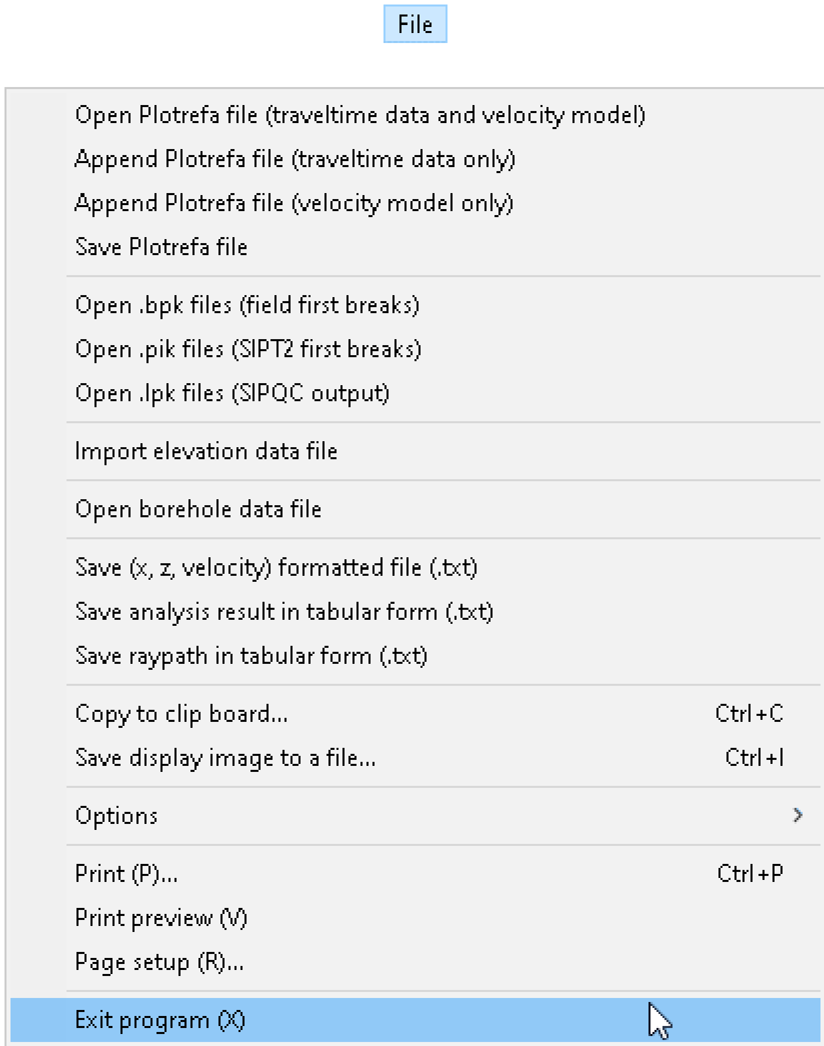


To prepare a page for printing, choose *Page setup*. You will see the **Print** dialog box for your computer:

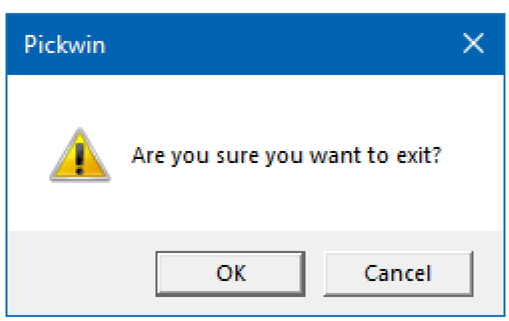


Adjust the properties for printing and press *OK* to print the current window display of Plotrefa.

4.1.19 EXIT PROGRAM



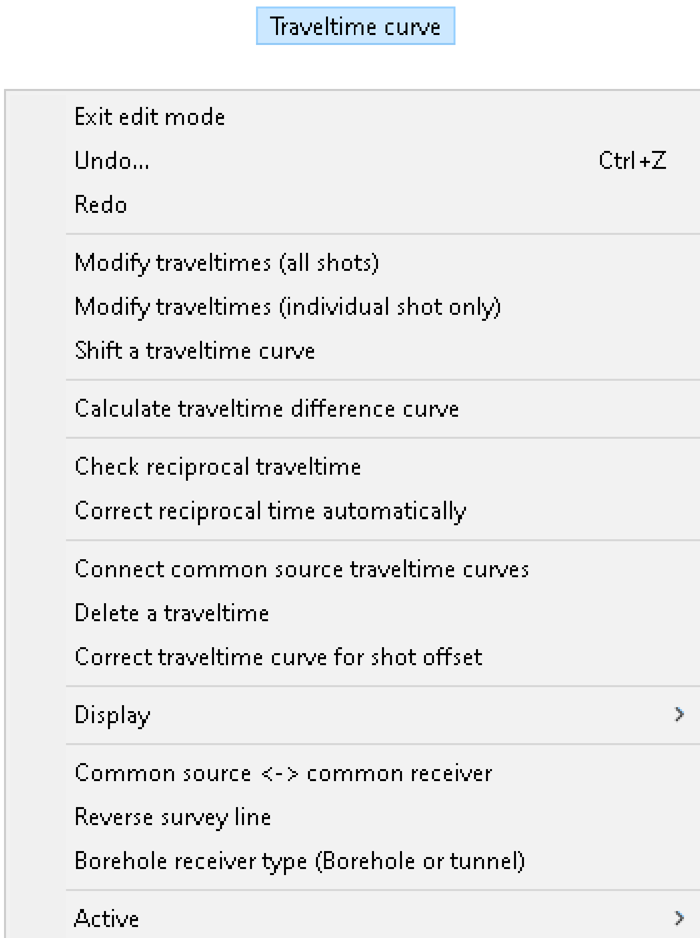
To exit the Plotrefa module, choose *Exit program*. You will see the following dialog box:



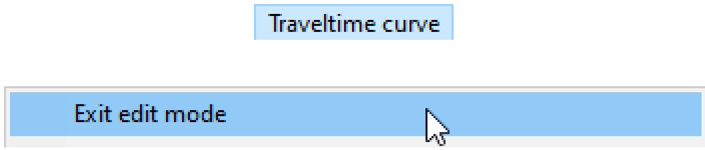
Press *OK* to exit Plotrefa or press *Cancel* to continue using Plotrefa.

4.2 TRAVEL TIME CURVE MENU

Click on *Travel time curve* to reveal the **Travel time Curve** menu:



4.2.1 EXIT EDIT MODE [ ]



When you are *not* in an edit mode, you can click and drag your mouse on the graph to measure time distances, as shown below:

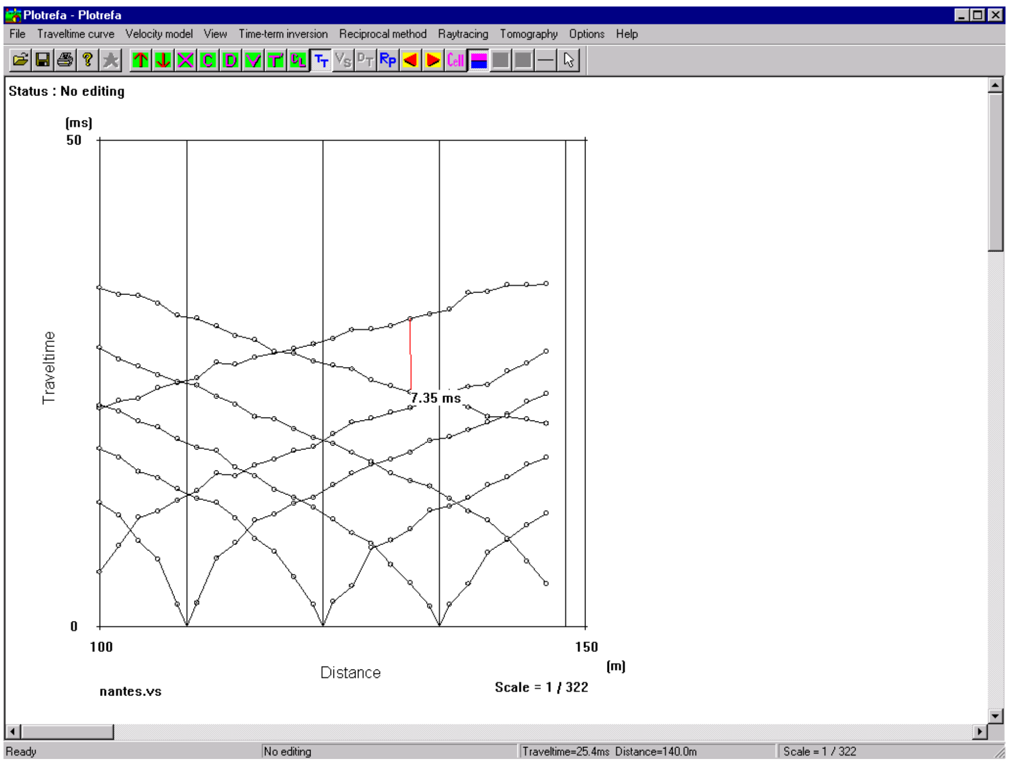
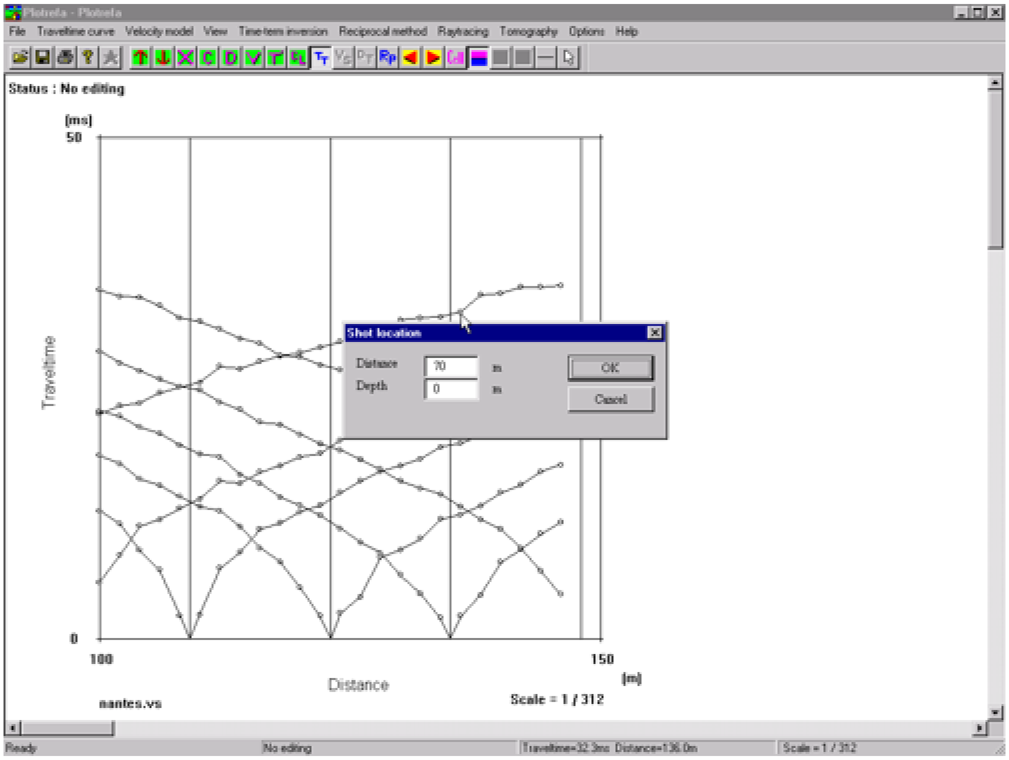


Figure 47: Measuring time-distance on a travel time plot.

Also, if you double-click on a travel time, the shot location and depth for that travel time curve will be displayed and can be edited if necessary:



You may also draw a velocity line on your travel time plot by clicking on the tool button and clicking and dragging your mouse. The velocity of the line will be displayed dynamically at the top of the display. Right click to "set" the velocity line:

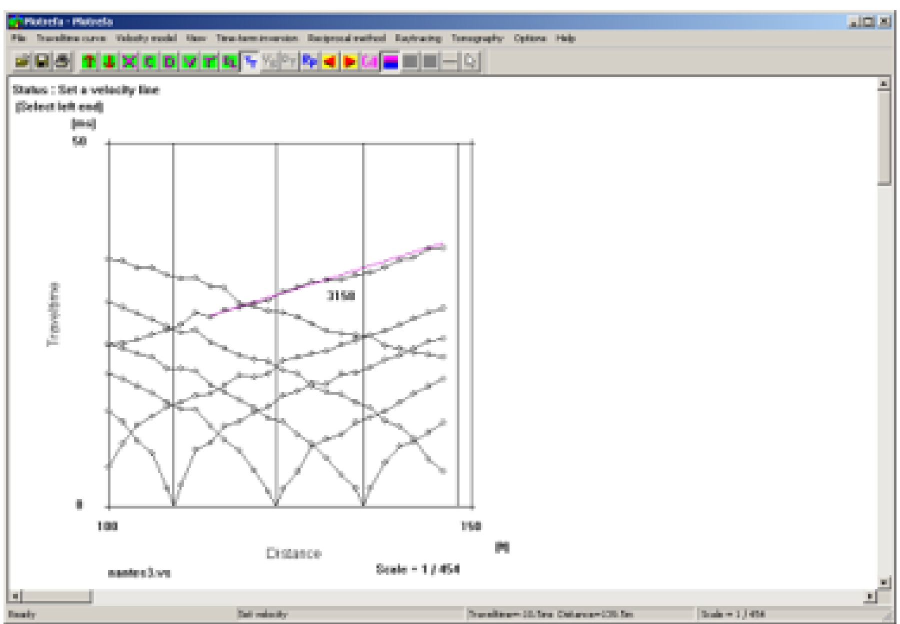
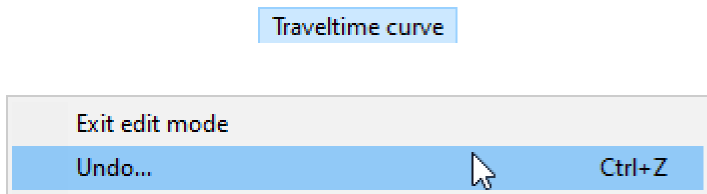


Figure 48: Measuring apparent velocity on a travel time plot.

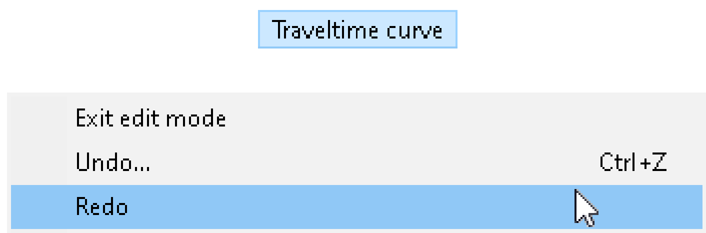
If you *are* in an edit mode (for instance, modifying travel times), clicking and dragging the mouse will alter your data, depending on the specific edit mode you are in. To get out of edit mode, choose *Exit edit mode*, or press the tool button.

4.2.2 UNDO [CTRL + Z] [ ]



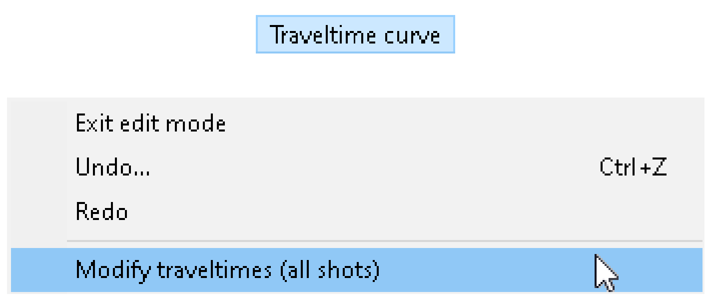
To undo the last command performed, click on *Undo*, press *Ctrl+Z*, or press the "Undo" tool button . The last applicable command performed by Plotrefa will be undone.

4.2.3 REDO [ ]



To re-do a command that was undone, click on *Redo* or press the "Redo" tool button . The last command that was undone will now be redone.

4.2.4 MODIFY TRAVEL TIMES (ALL SHOTS) [ ]



You may use Plotrefa to modify travel times if necessary. If you choose *Modify travel times (all* *shots)*, you can then click and drag any travel time to a new position. Simply point at the travel time you wish to change and click. The selected travel time will turn red:

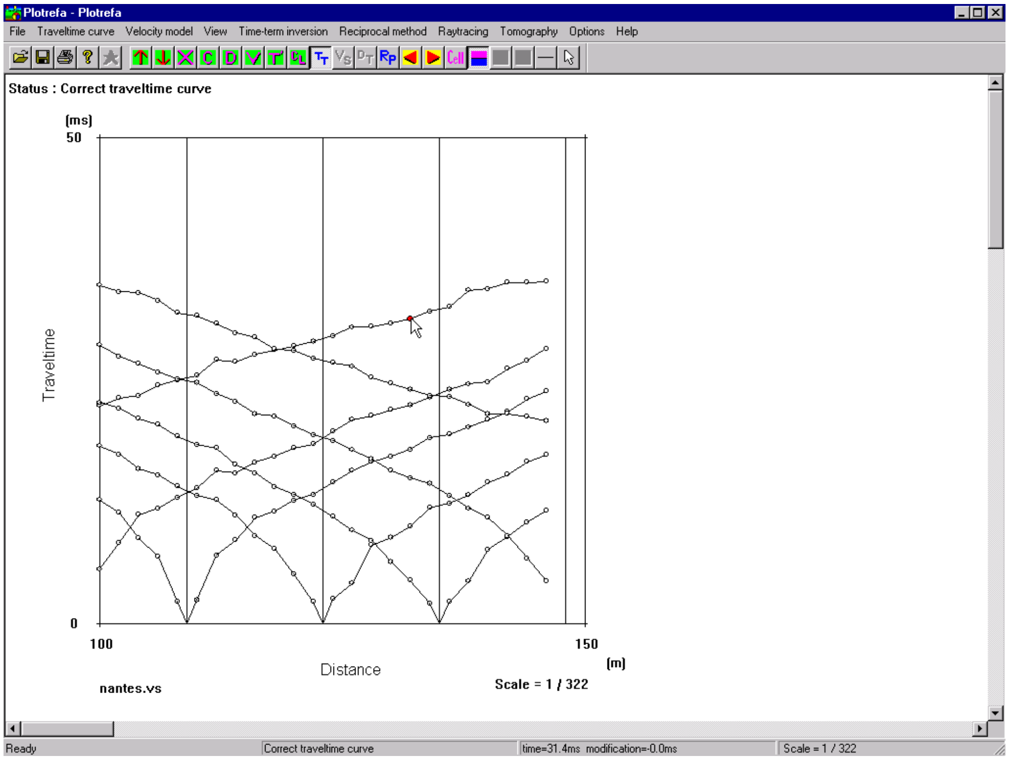


Figure 49: Selecting a travel time to adjust.

While holding the mouse button down, drag the cursor to where you want the travel time to be, and release:

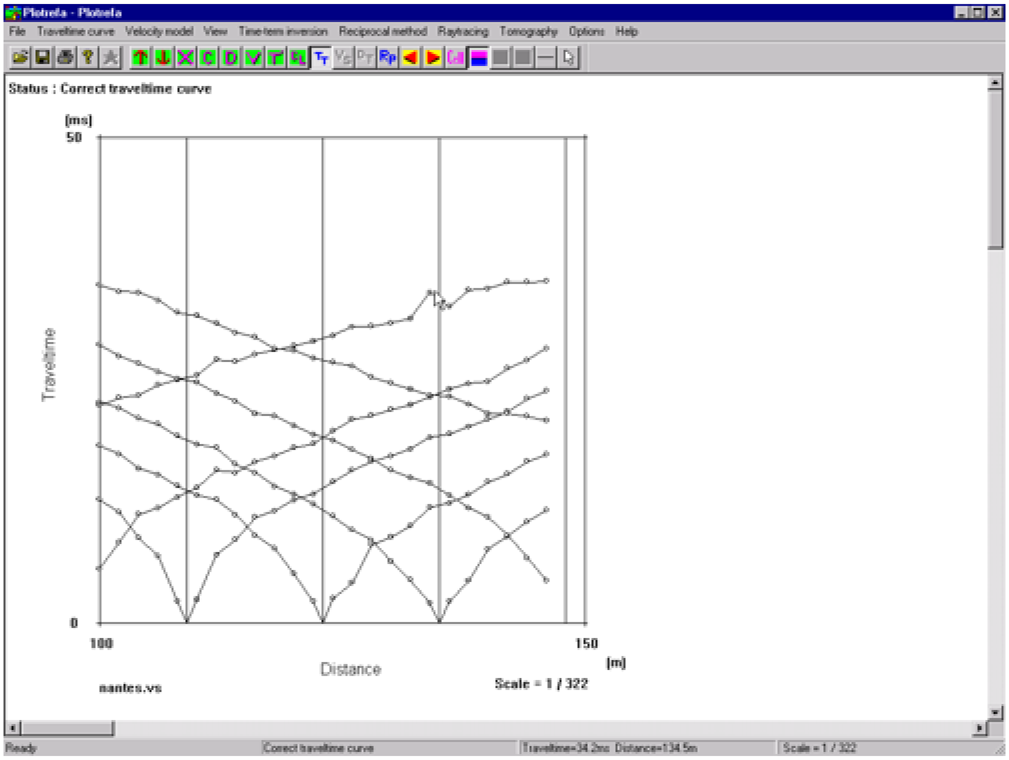
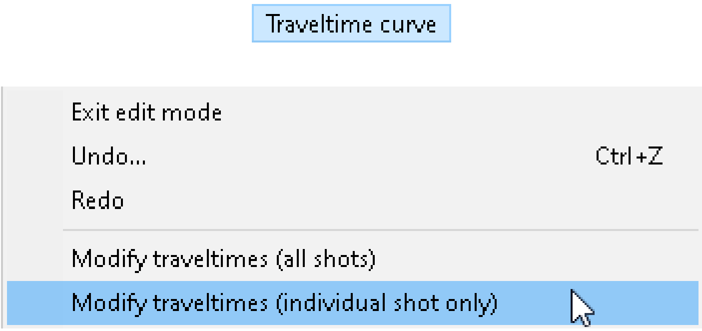
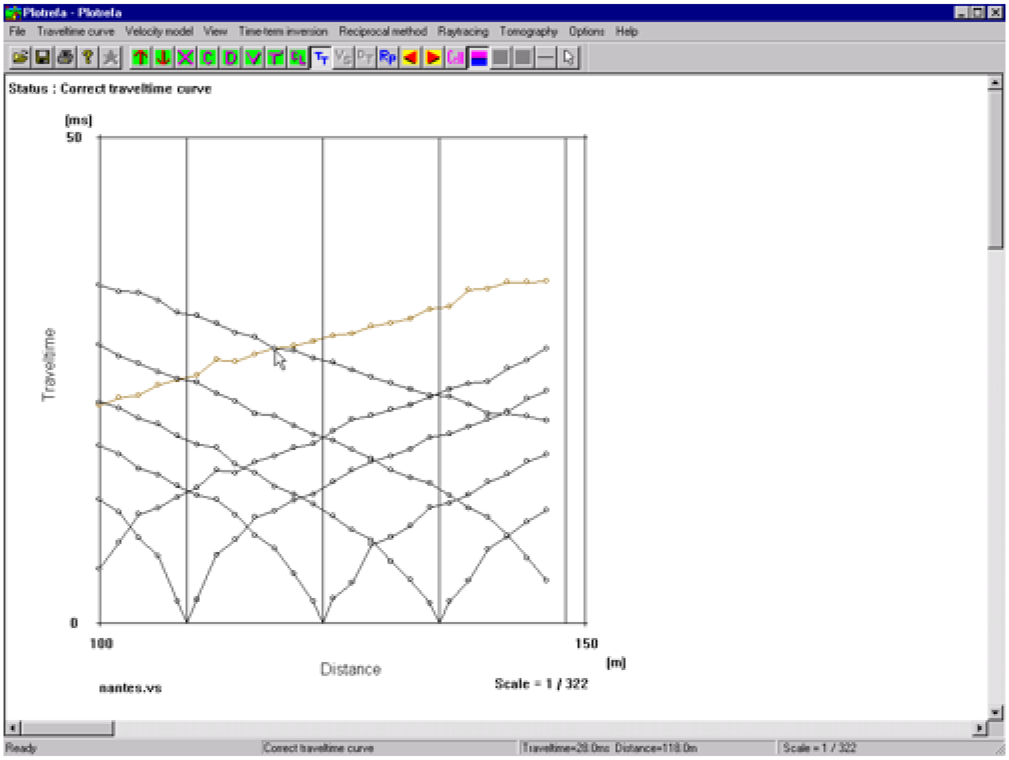


Figure 50: Modifying travel times on a travel time plot.

4.2.5 Modify travel times (individual shot only)



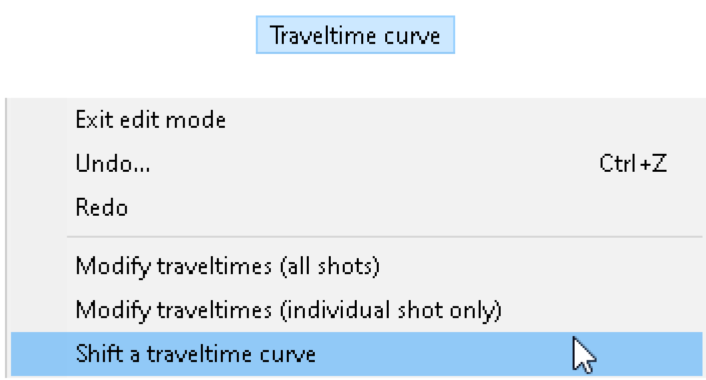
Sometimes two travel times plot on top of each other, making it difficult to take control of the one you want. When this happens, you can choose *Modify travel times (individual shot only)*. This allows you to first choose the travel time curve that contains the travel time you wish to modify. Simply click on the travel time curve, and it will change color:



Now, only travel times on that particular curve can be selected for modification. Note above that the cursor is pointing to a travel time that is coincident with a travel time on another curve. But only the one in the highlighted curve can be modified. Click and drag the travel time as described above.

Note: *You will notice that the curve is no longer highlighted. This feature turns itself off after adjusting one travel time, i.e.,* ***all*** *travel times are accessible after the first one is modified. You must choose* ***Modify travel times (individual shot only)*** *again to highlight another curve.*

4.2.6 SHIFT A TRAVEL TIME CURVE



You may also shift an entire travel time curve. Choose *Shift a travel time curve*, click and hold on the curve of interest (it will change colors), and drag it to the new position:

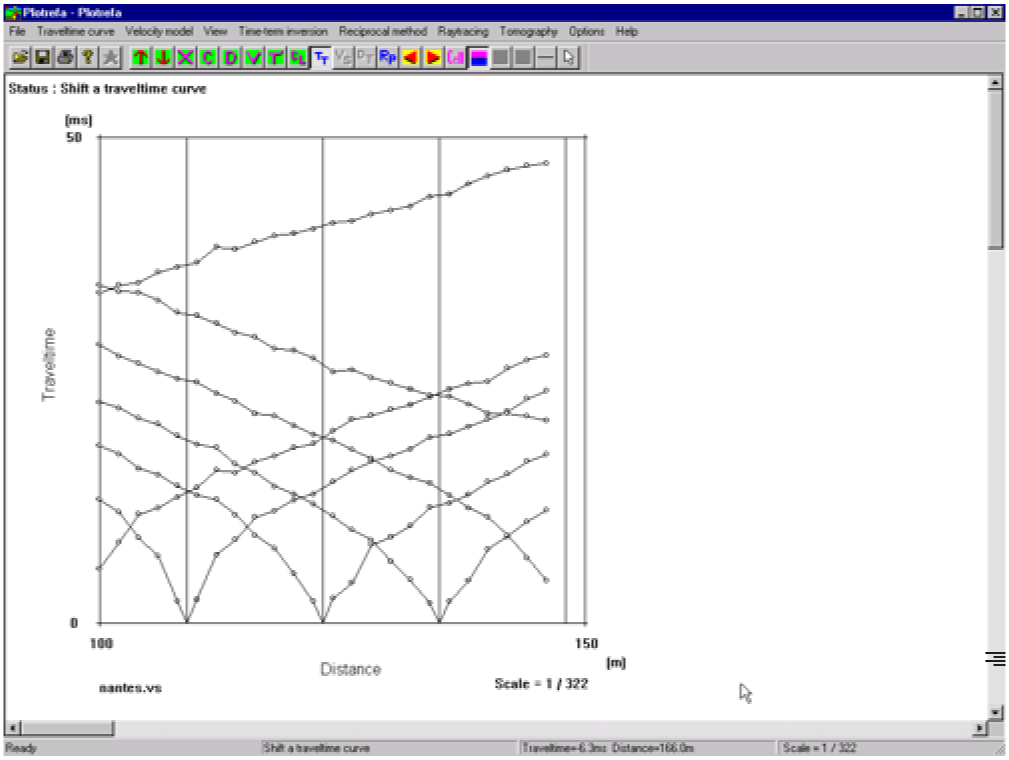
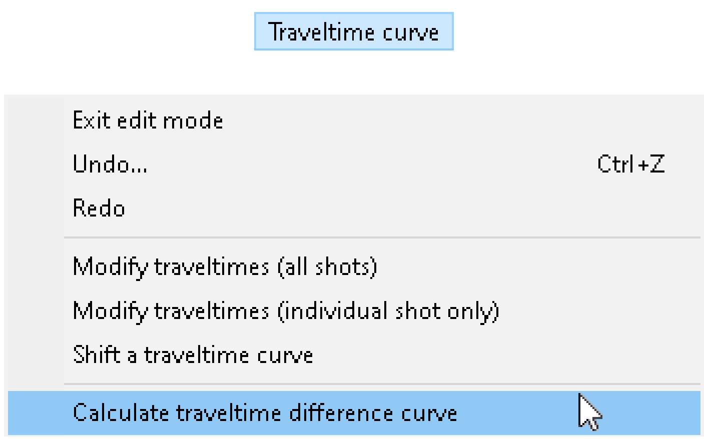


Figure 51: Shifting a travel time curve on a travel time plot.

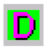
The entire curve highlighted in the previous section has been moved to a later time.

4.2.7 CALCULATE TRAVEL TIME DIFFERENCE CURVE [D]



When assigning layers to first arrivals, it is often useful to construct a travel time difference curve. The difference curve for two shots from the same direction should be flat where the travel times are coming from the same layer.

This can assist in determining crossover points.

To calculate a difference-time curve, choose *Calculate travel time difference curve*, or press the  tool button. Next, click on the two travel time curves you wish to calculate the difference-time curve for.

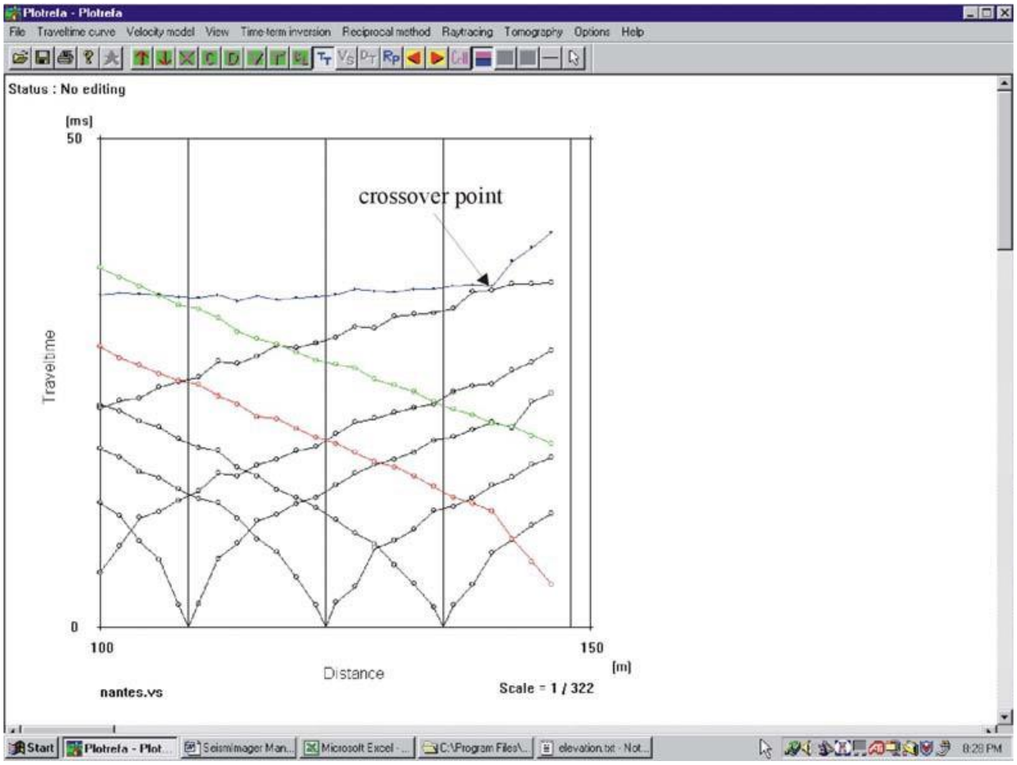
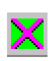


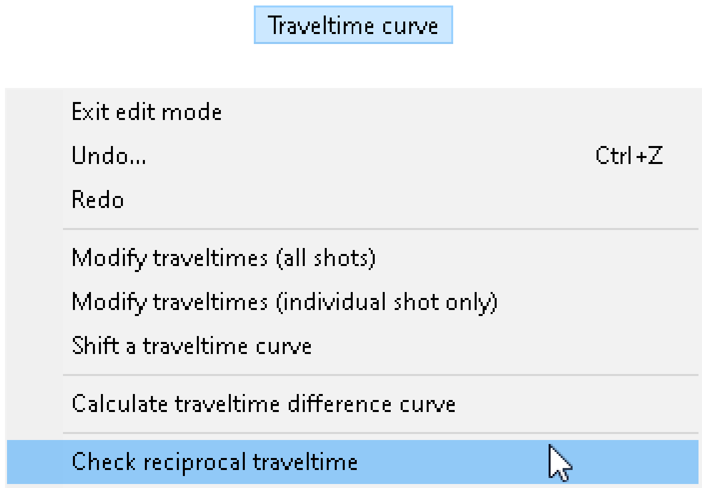
Figure 52: Calculating a difference-time curve.

In the example above, the blue curve represents the difference-time curve for the highlighted travel time curves. Note that the crossover point for the red curve is clearly delineated by the difference-time curve. This is an extremely useful tool when crossover points are difficult to determine.

To remove the difference-time curve, simply press the  tool button.

See here for a video on the difference-time curve calculation procedure (be sure to turn up the volume).

4.2.8 CHECK RECIPROCAL TRAVEL TIME



The *Principle of Reciprocity* states that the travel time measured between a source and receiver is independent of the direction of travel. In other words, if you invert the source and geophone, you must get the same travel time. This is true regardless of the subsurface conditions – in theory at least, **the travel times must be the same.**

Checking for reciprocity is an important step in ascertaining the quality of your data. If you don't have reciprocity within about 5%, you should recheck your travel times. Velocity models calculated from data exhibiting poor reciprocity are likely to be invalid.

Plotrefa will check reciprocity, where appropriate, automatically. Simply choose *Check reciprocal travel time*, and the program will examine the travel times and calculate reciprocity between shots in which the conditions of reciprocity are met.

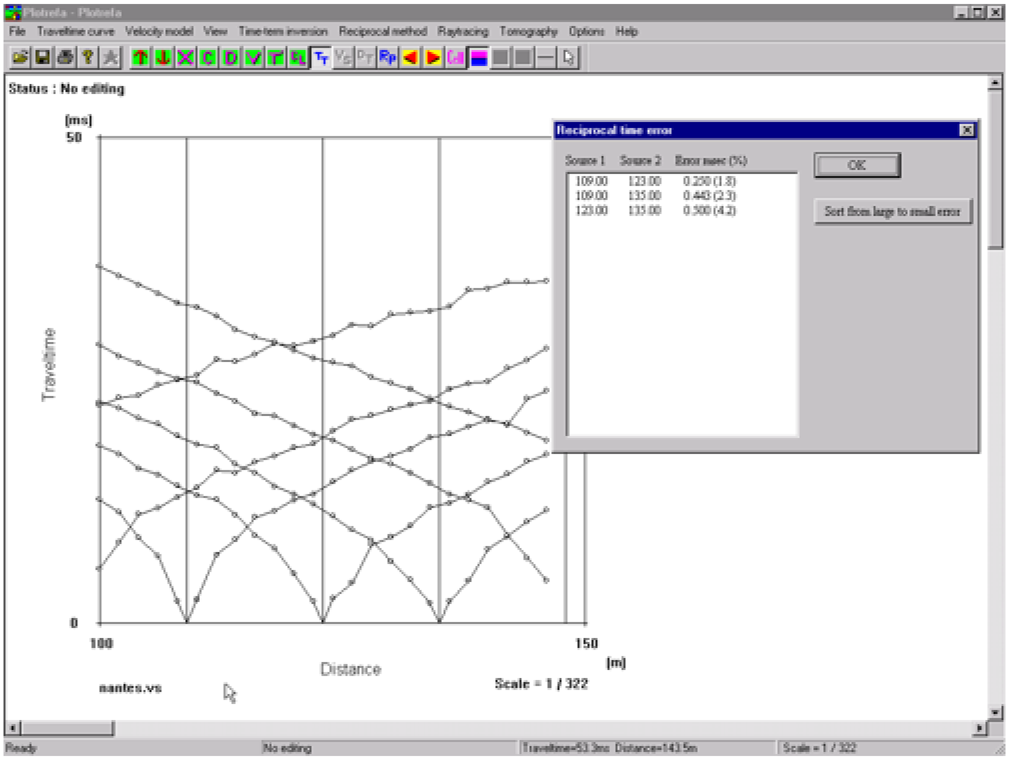
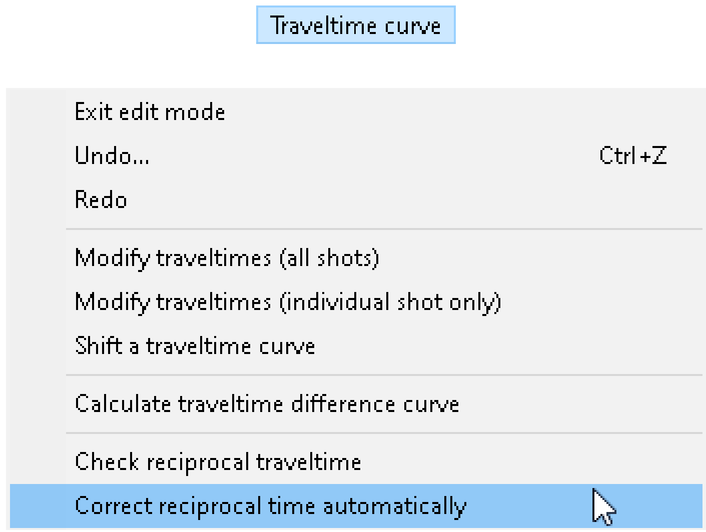


Figure 53: Checking the reciprocal time.

In the example above, the reciprocity has been reported for the three interior shots. Both the absolute and percentage errors are reported. The reciprocity report will be saved to a file called "reciprocity_check.txt" in the same folder in which your data is stored.

Note: *Reciprocal times are calculated only for shots that are* **within** *the geophone spread. For this reason, it is recommended that the shots at the end of the spread be located between the two end phones at either end of the line. For instance, with a 24-channel spread, the left "end-shot" would be between geophones 1 and 2, and the right "end-shot" would be between geophones 23 and 24. SeisImager/2D will interpolate to calculate the reciprocal times at the shot locations. If the shot points are outside the spread, the strict conditions of reciprocity are not met, and the program must extrapolate to estimate the reciprocity.*

4.2.9 CORRECT RECIPROCAL TIME AUTOMATICALLY



If the data quality is such that you cannot get better than a 5% reciprocity error, it is sometimes helpful to have the program correct the data. This, of course, is no substitute for picking the data correctly. It should only be used when true reciprocity cannot be achieved, because of difficulty in picking first breaks. The program will iteratively shift the travel time curves to spread the reciprocity error out as evenly as possible, and this will sometimes yield a better answer than if left alone.

To correct the reciprocal times, choose *Correct reciprocal time automatically*:

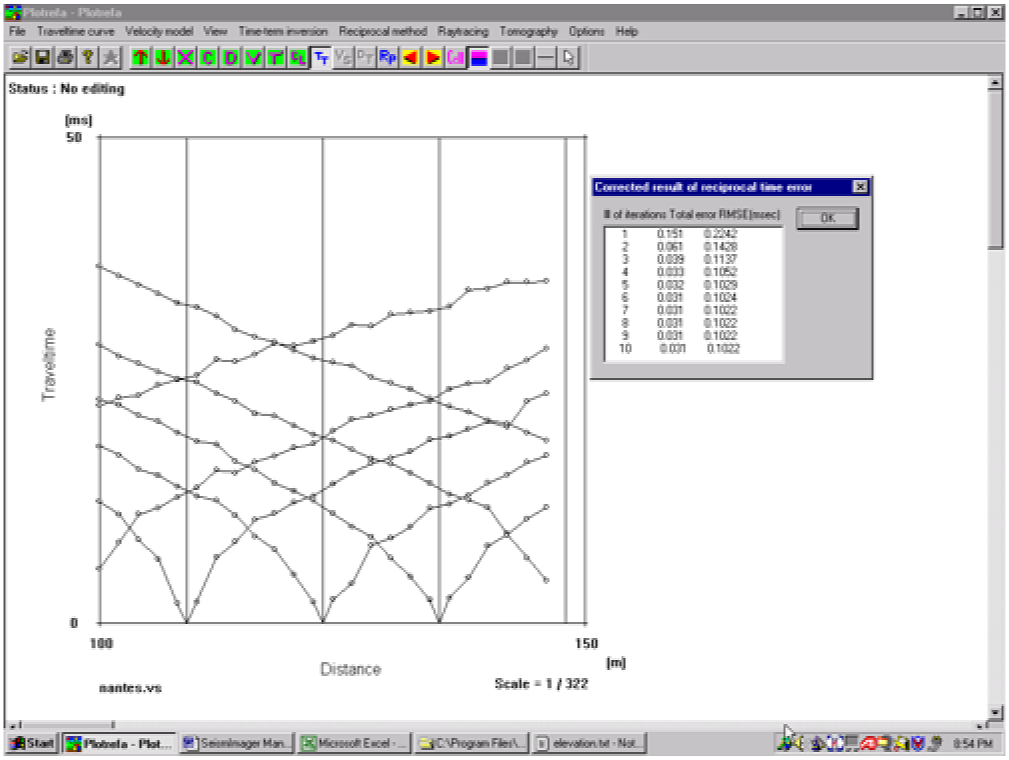


Figure 54: Correcting the reciprocal time.

The travel times will be shifted to minimize the errors, and a table of total error versus iteration number will be displayed. Press *OK*, and a new reciprocity report will be shown:

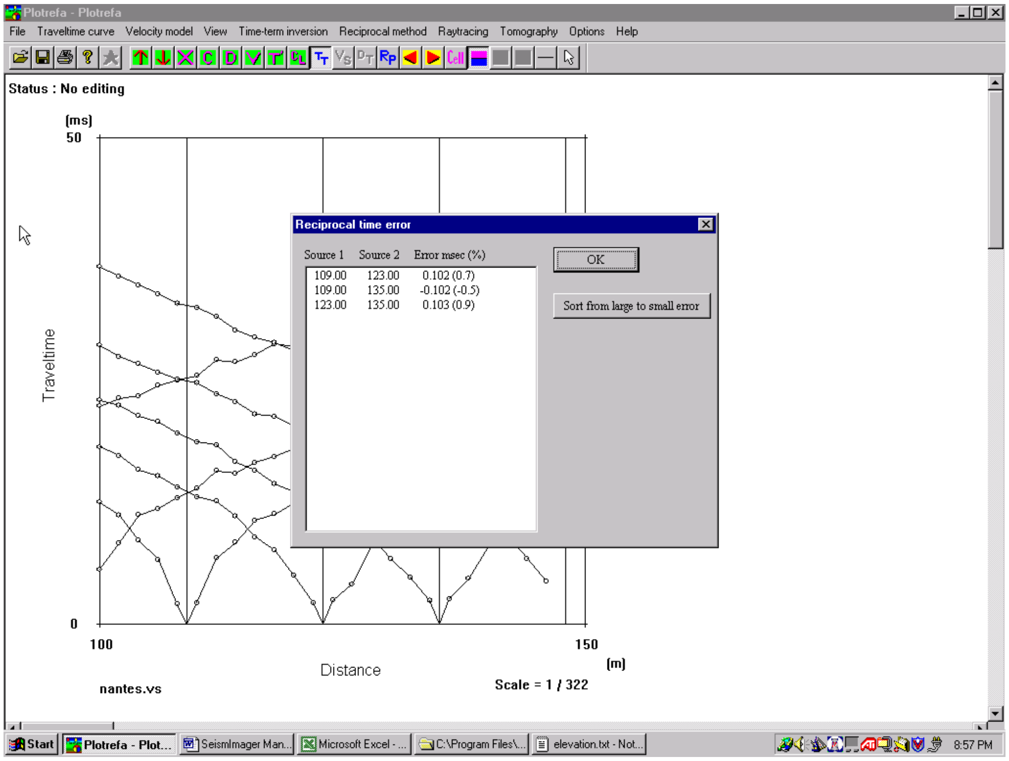
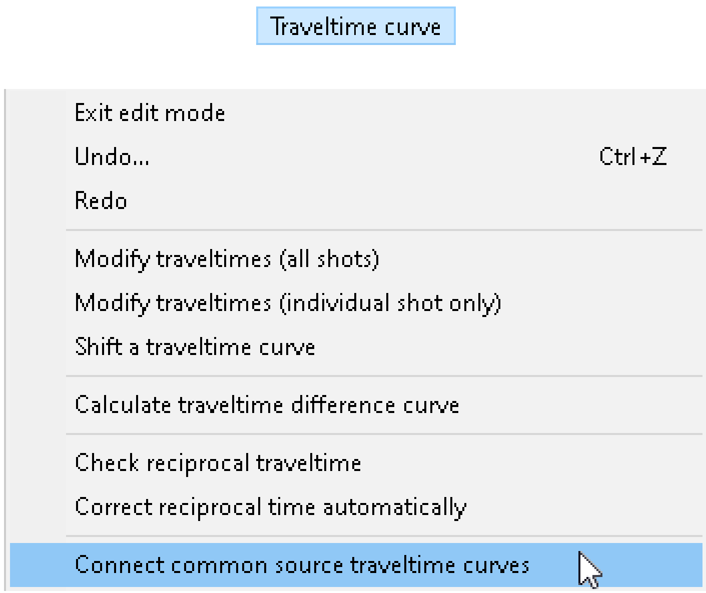


Figure 55: Reciprocal time report.

Note: *The amount of confidence in the resulting model should be inversely proportional to the level of correction required. A model calculated from modified data is always suspect.*

4.2.10 CONNECT COMMON SOURCE TRAVEL TIME CURVES



If you have appended Plotrefa files together, as discussed in Section 4.1.2 (Page 127), you may connect the travel time curves from common sources together. In the example below (the same used in Section 4.1.2), two separate Plotrefa files have been appended. However, they are still shown as separate curves (note the gaps in the middle). Before proceeding to the layer assignment phase, these travel time curves should be connected. You may do so by simply clicking on *Connect common source travel time curves*.

Note that this feature does **not** make any corrections to the data. If there are source-dependent offsets, such as those discussed in Section 3.2.8 (Page 52), they should be corrected manually by using the *Shift a travel time curve* function.

In the example below, two Plotrefa files are appended:

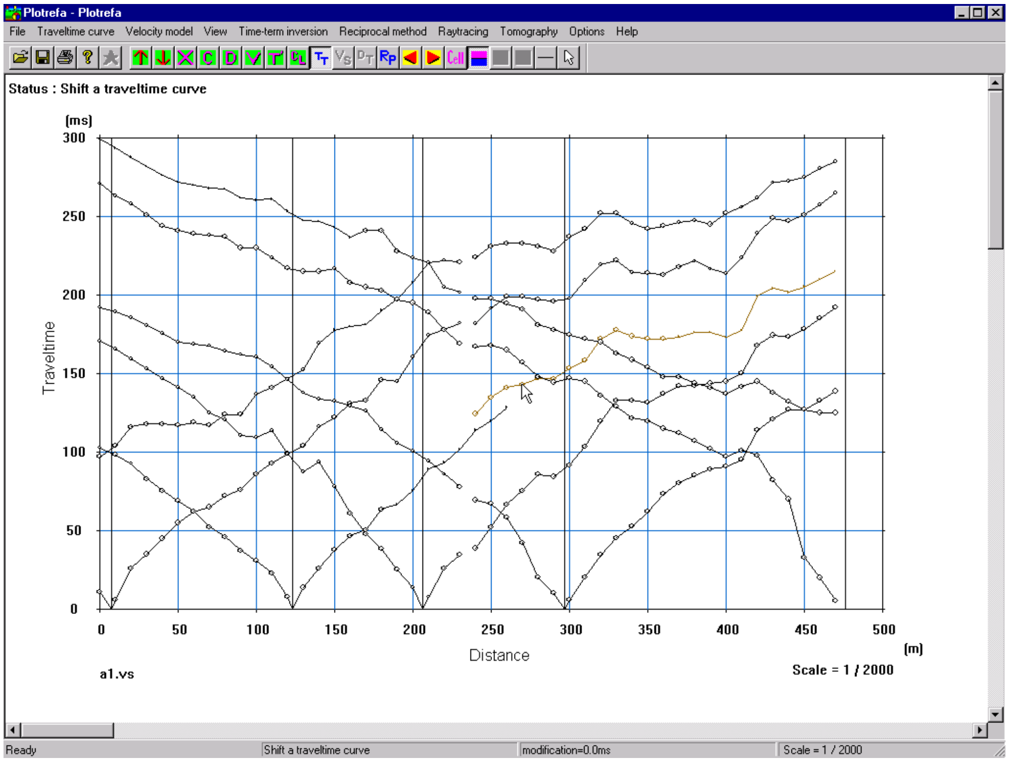


Figure 56: Highlighting a travel time curve for adjustment.

The highlighted travel time curve (see arrow) needs to be adjusted in time before connecting. Otherwise, the final interpretation will include an artifact due to a sudden and false jump in travel time at that location. In the figure below, the highlighted travel time curve has been moved down to better agree with its common-source data:

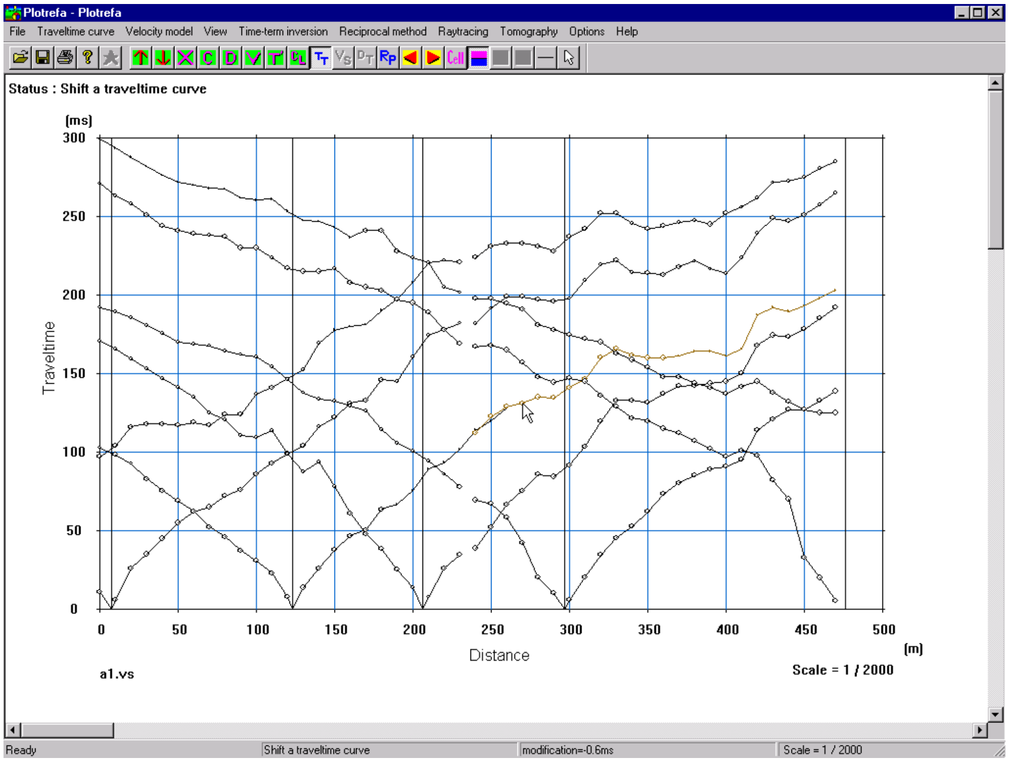


Figure 57: Adjusting a travel time curve.

After manually correcting where necessary, you may connect the travel time curves:

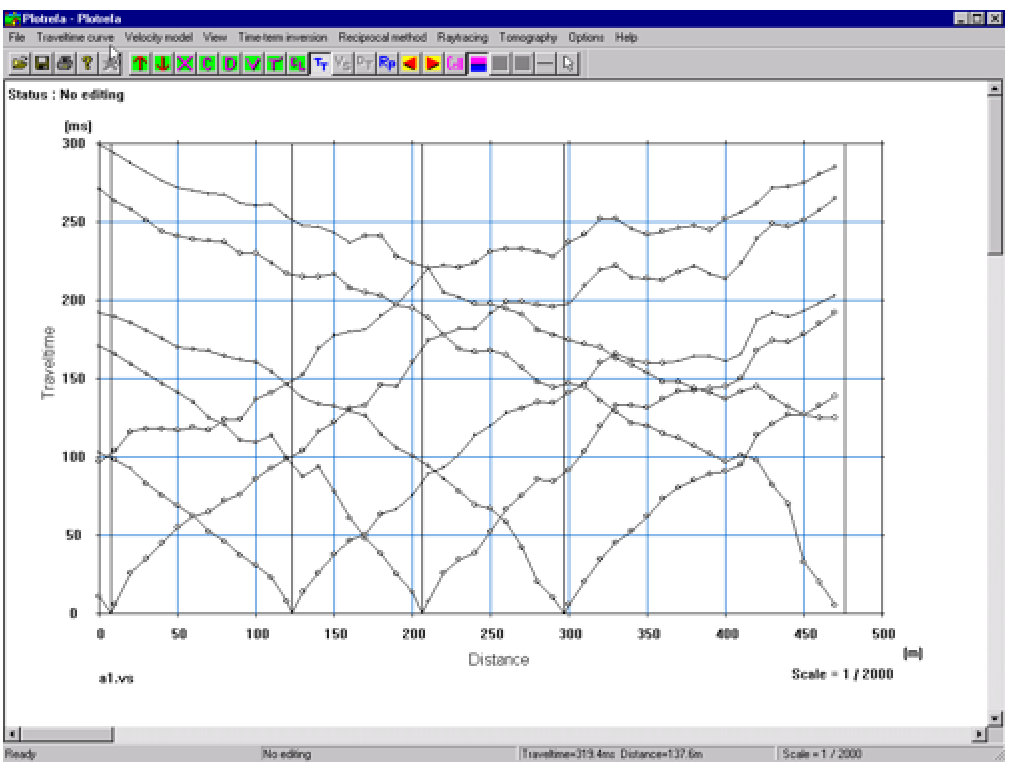
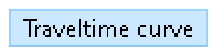


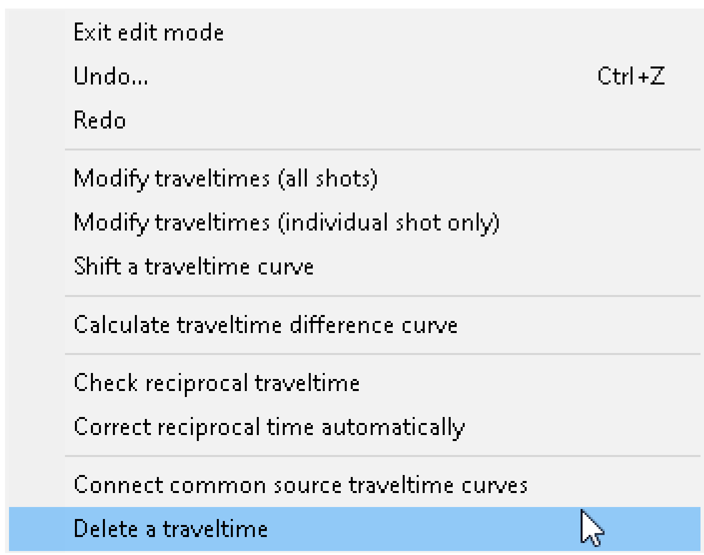
Figure 58: Connecting common-source travel time curves.

You now have a travel time plot that should be exactly the same as the one you would have achieved had you laid out the geophones once and occupied all of the shot points once.

At this point, it should be obvious why overlap is highly desired when using multiple spreads. As shown in Section 3.2.8 (Page 52), this same step can be accomplished in Pickwin. Where you do it is a matter of preference. But no matter how you do it, overlapping geophones is essential to correct for velocity inconsistencies at the shot point.

4.2.11 DELETE A TRAVEL TIME





You may delete a travel time by choosing *Delete a travel time* and clicking on it:

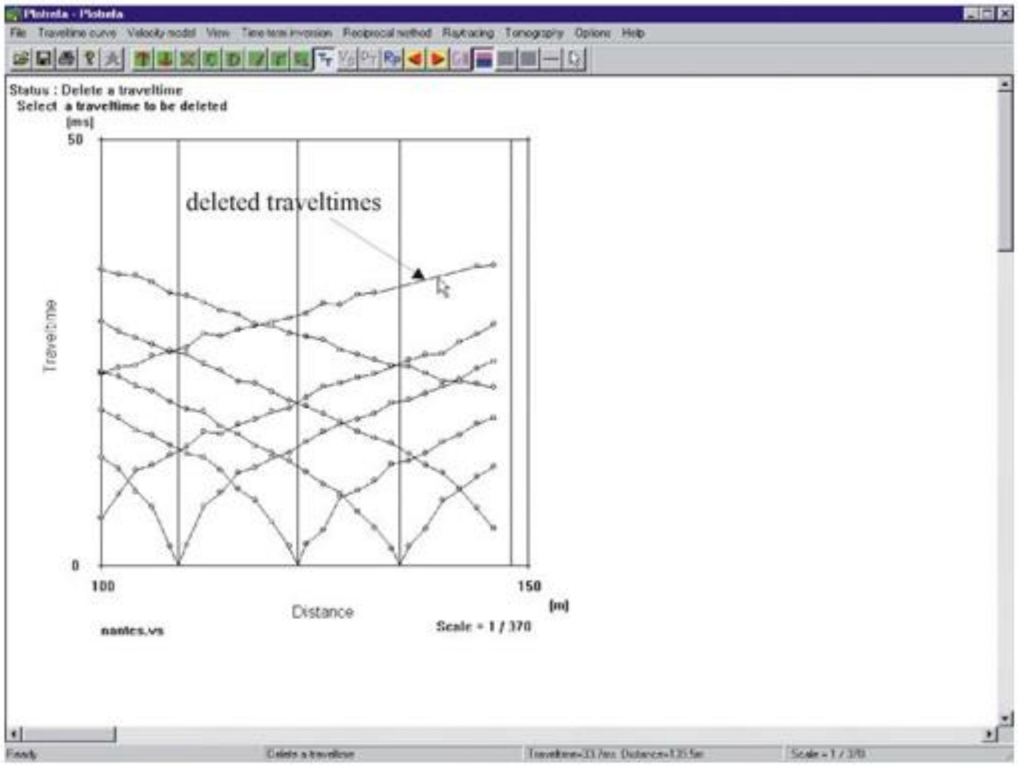
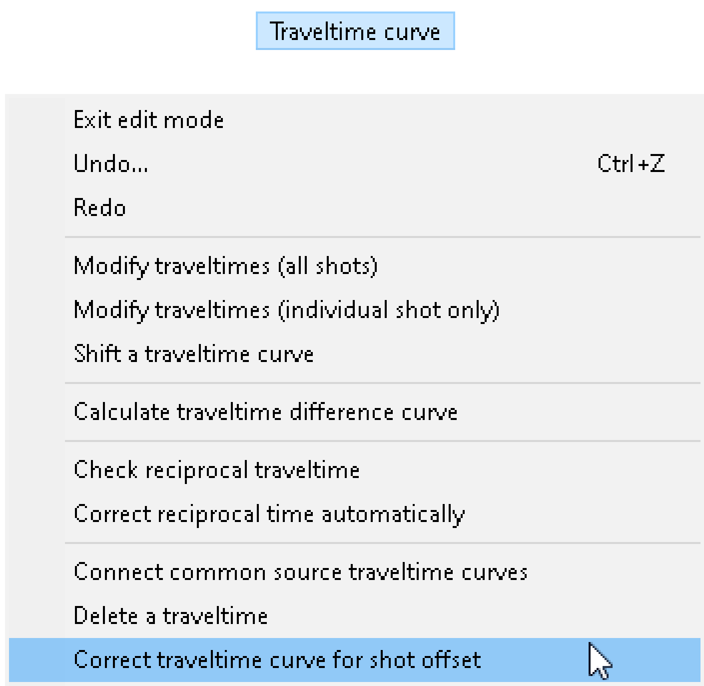


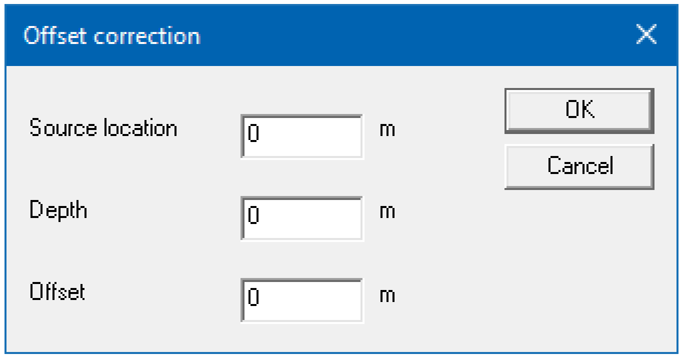
Figure 59: Deleting a travel time.

In the above figure, five travel times have been deleted.

4.2.12 CORRECT TRAVEL TIME CURVE FOR SHOT OFFSET



If the shot is beneath the surface and/or offset from the line, you may correct the travel time curve to account for it. In general, this is good practice when the shot depth/offset is more than 20% of the geophone spacing. Choose *Correct travel time curve for shot offset*, and you will see the following dialog box:



Input the source location (along the line), depth, and offset (perpendicular to the line), and the travel times will be corrected, using the near-surface velocity, to what they *would* be if the source were on the surface at zero perpendicular offset.

In the example below, the center shot of the dataset shown above has been corrected for a 2-meter offset:

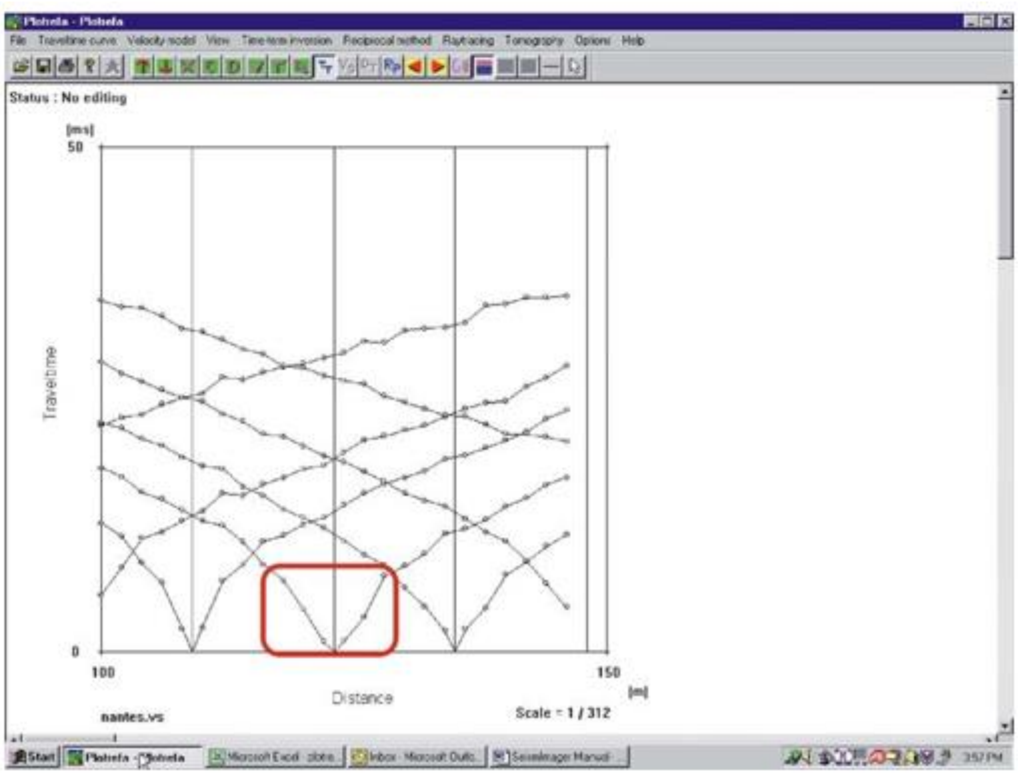
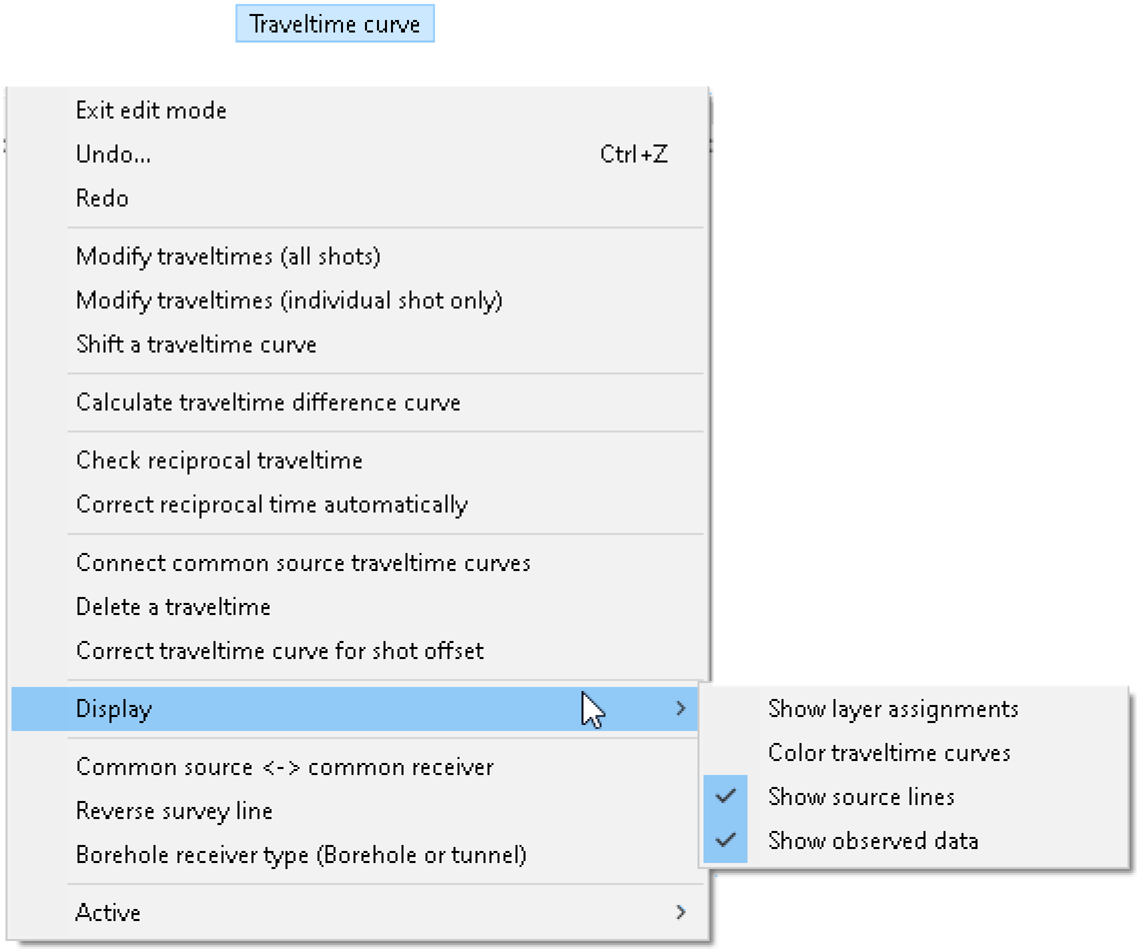


Figure 60: Correcting travel time curves for shot offset.

The geophones closest to the shot are most affected by this correction (compare to uncorrected data above, Figure 59).

4.2.13 DISPLAY



The display menu contains a sub-menu which allows you to control various display parameters of the travel time plot. All these choices are toggle switches; you simply click on them to turn them on or off.

If you have done your layer assignments, you may color-code them by choosing *Show layer assignments*:

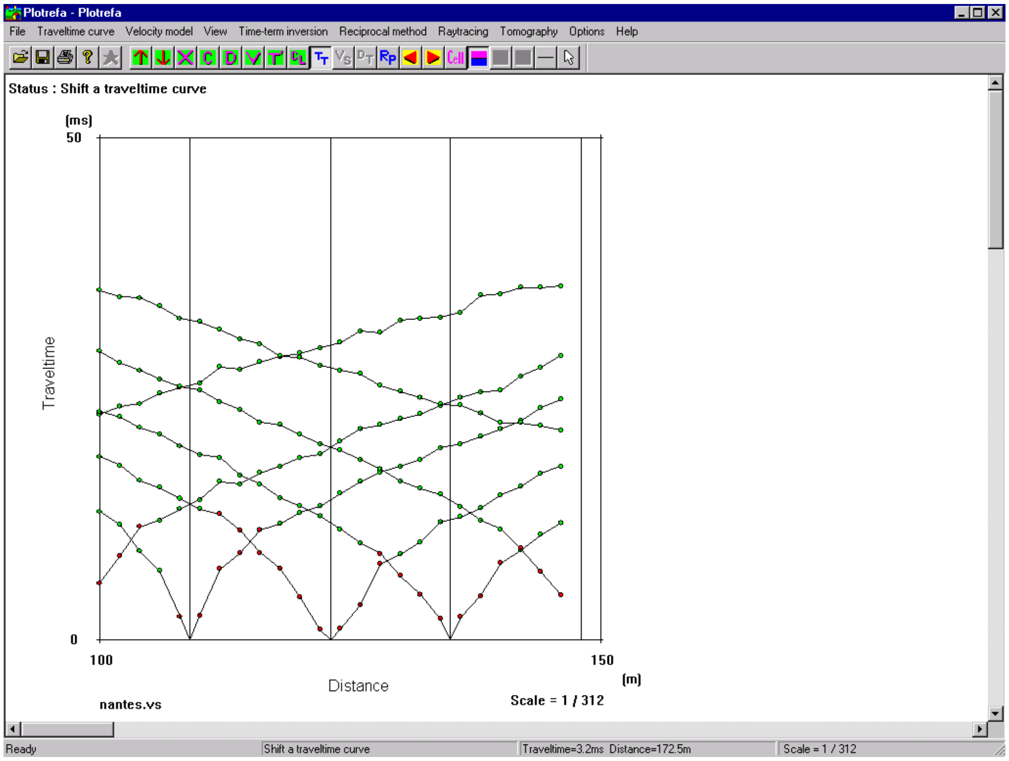


Figure 61: Drawing source lines on the travel time plot.

If you would like to differentiate the shot gathers, you may color them different colors. Just click on *Color travel time curves*:

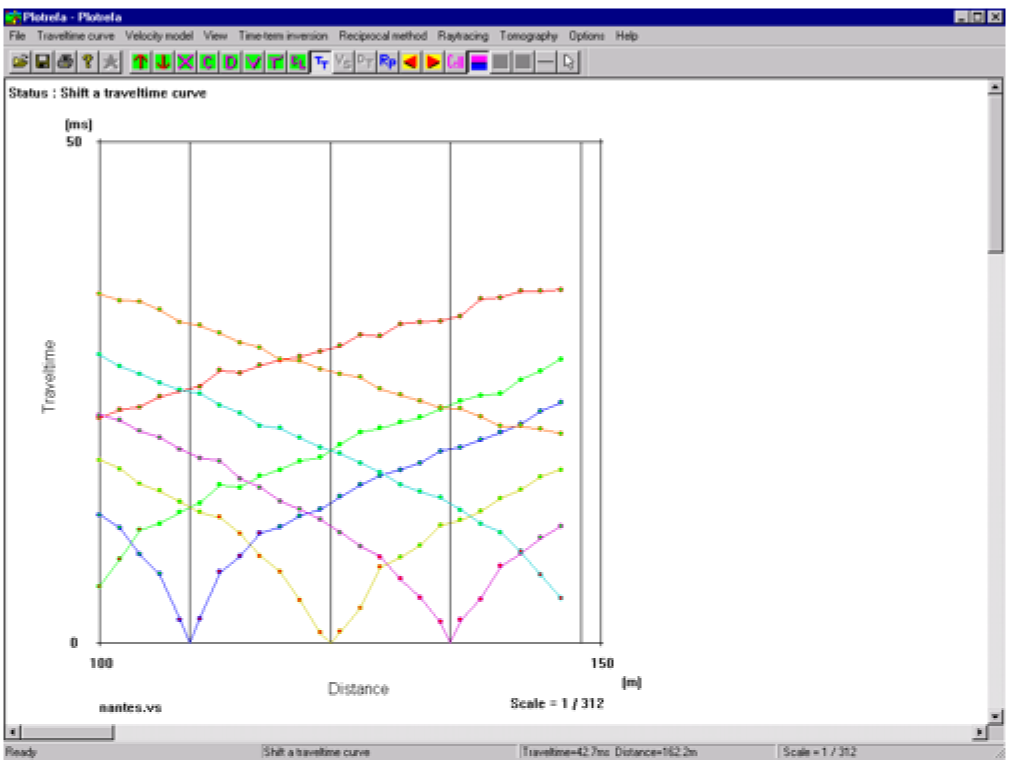


Figure 62: Using color in the travel time plot.

You may choose whether to connect the sources to the near geophones. Below is the same plot without the “source lines” drawn:

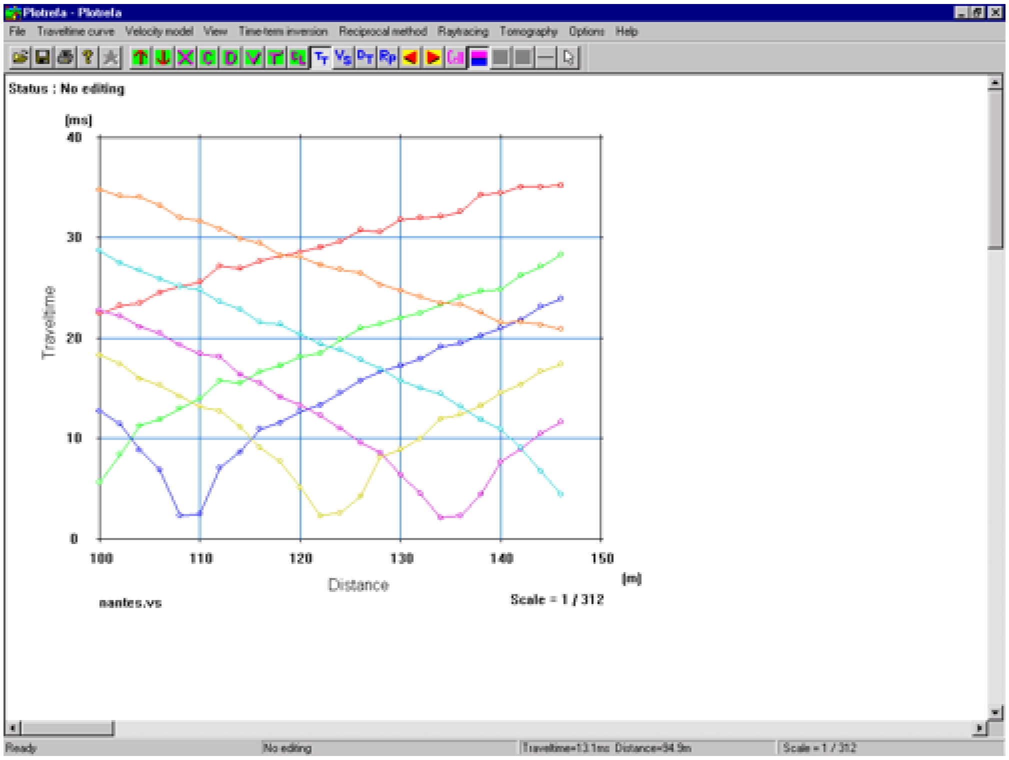
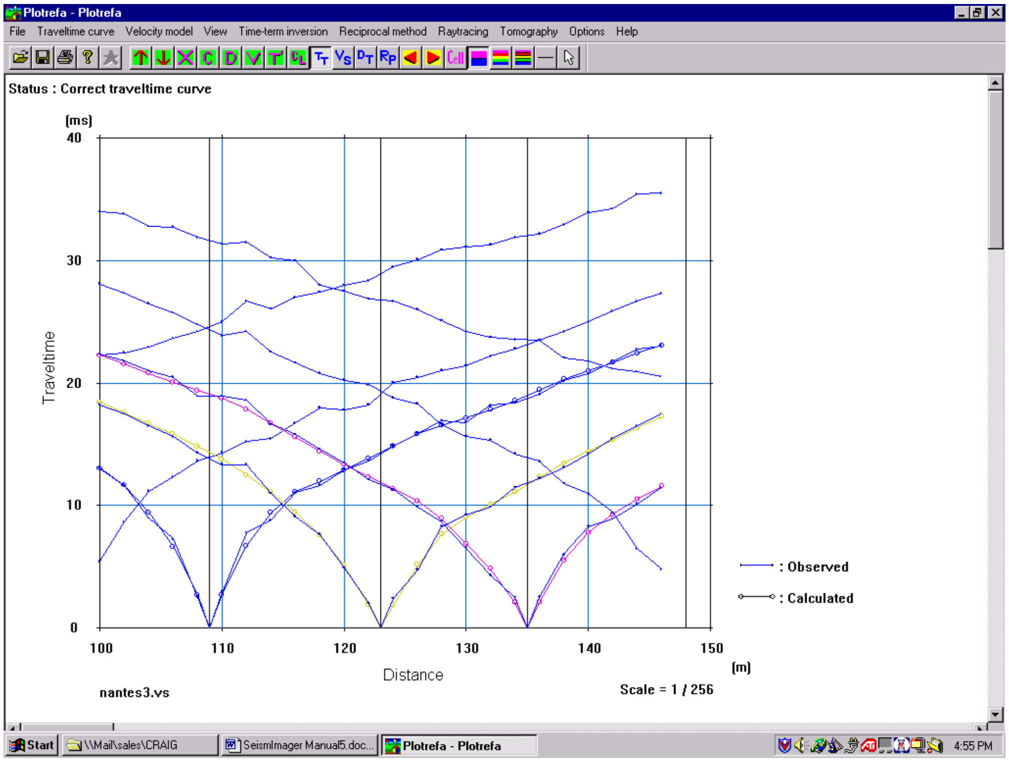


Figure 63: Travel time plot excluding source lines.

Note: *Source lines are only shown for shots within the geophone spread.*

If you have traced rays through your velocity model, the travel time plot will default to displaying both the observed and theoretical data:



If you wish to only see the theoretical data, click on the *Show observed data* toggle switch:

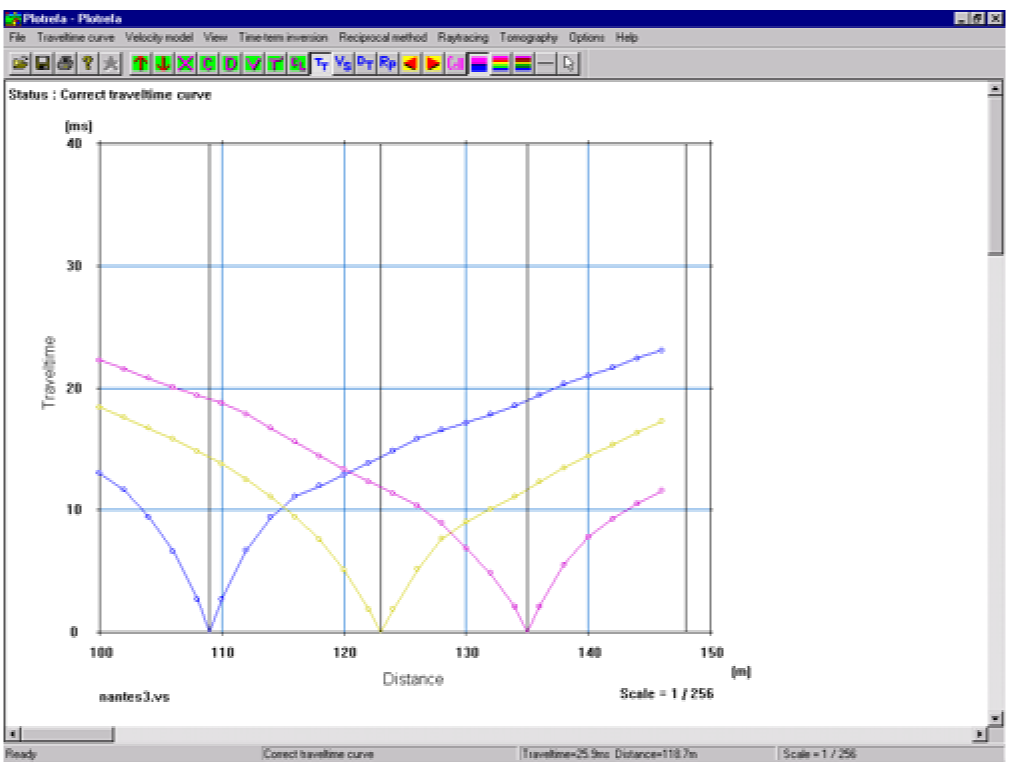
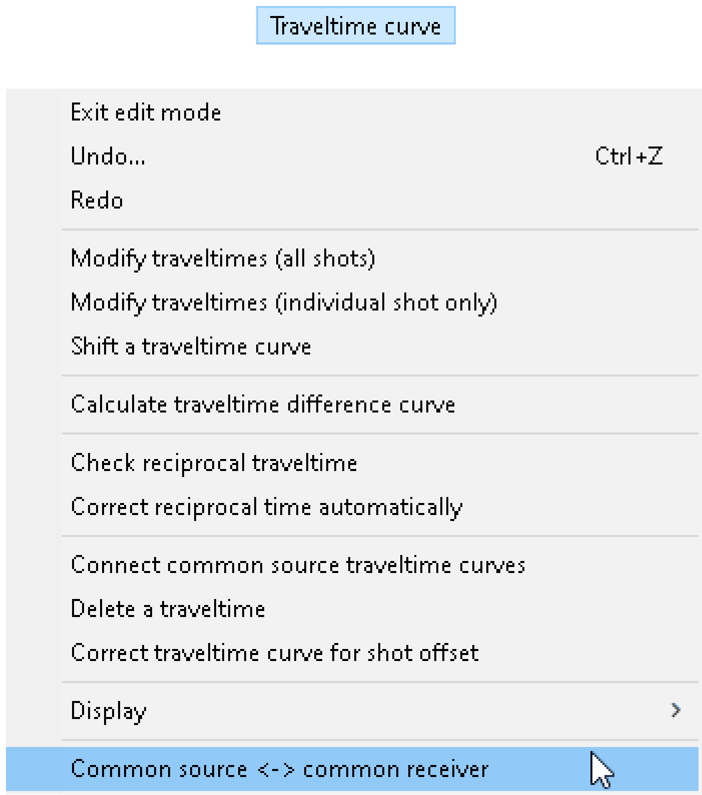
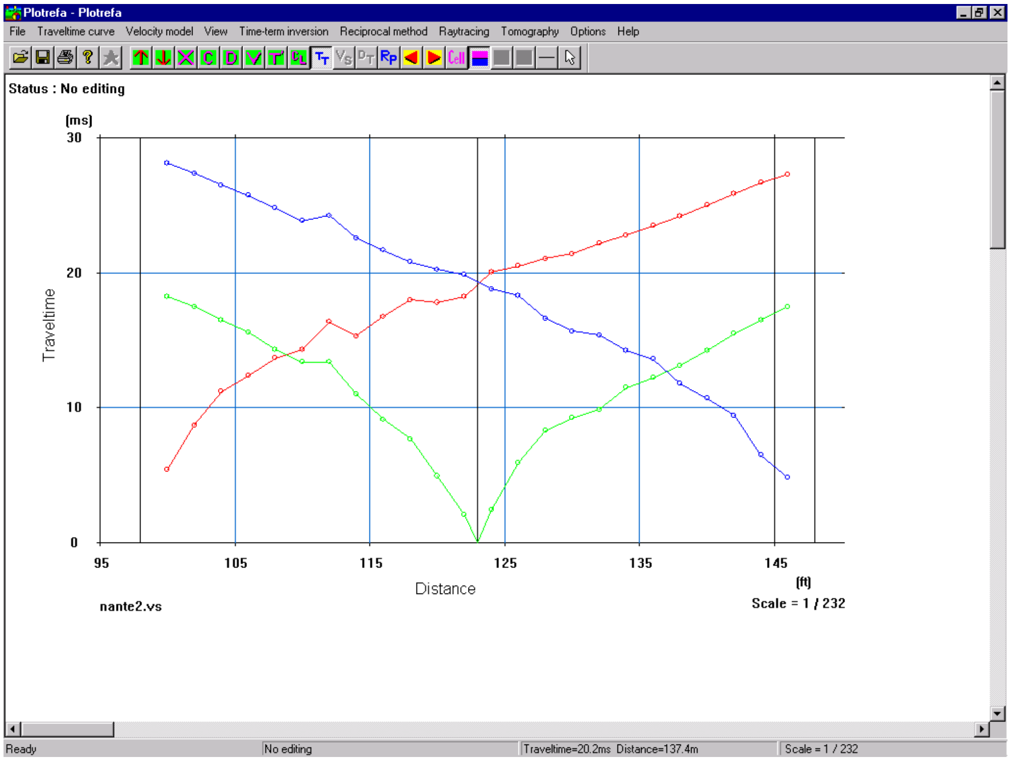


Figure 64: Displaying only theoretical data on the travel time plot.

4.2.14 Common Source <-> Common-receiver



It is sometimes useful to organize your travel time data in a common receiver gather rather than a common source gather. A typical common-source gather is displayed below:



To convert this to a common receiver gather, click on *Common source <-> common receiver*:

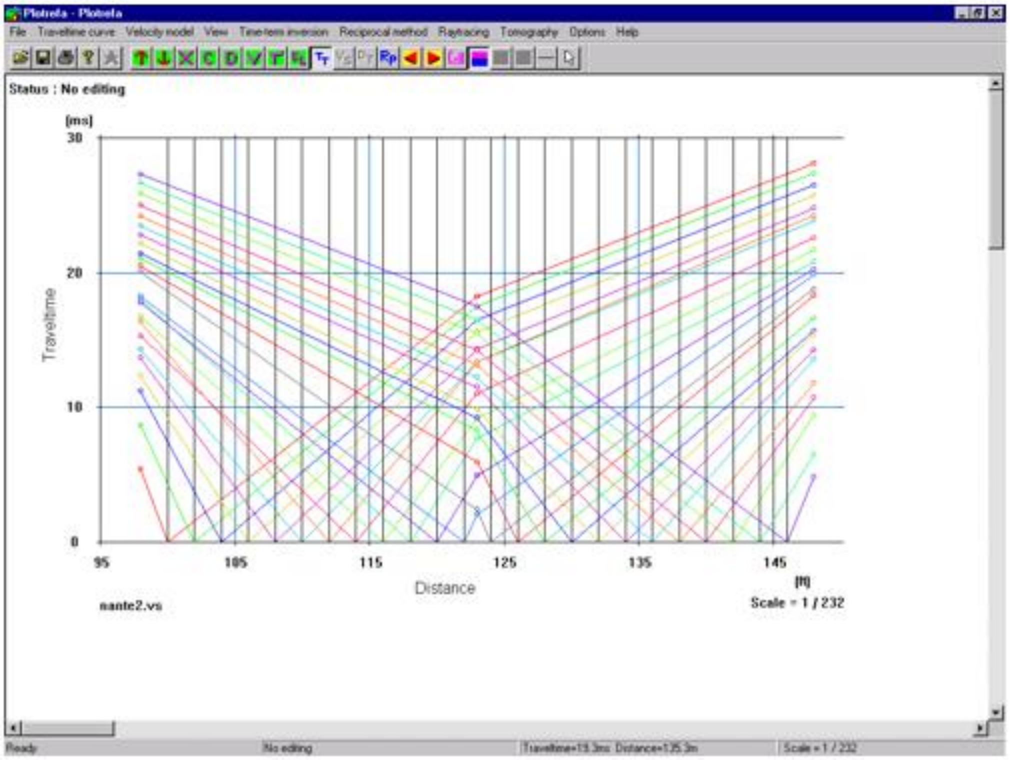
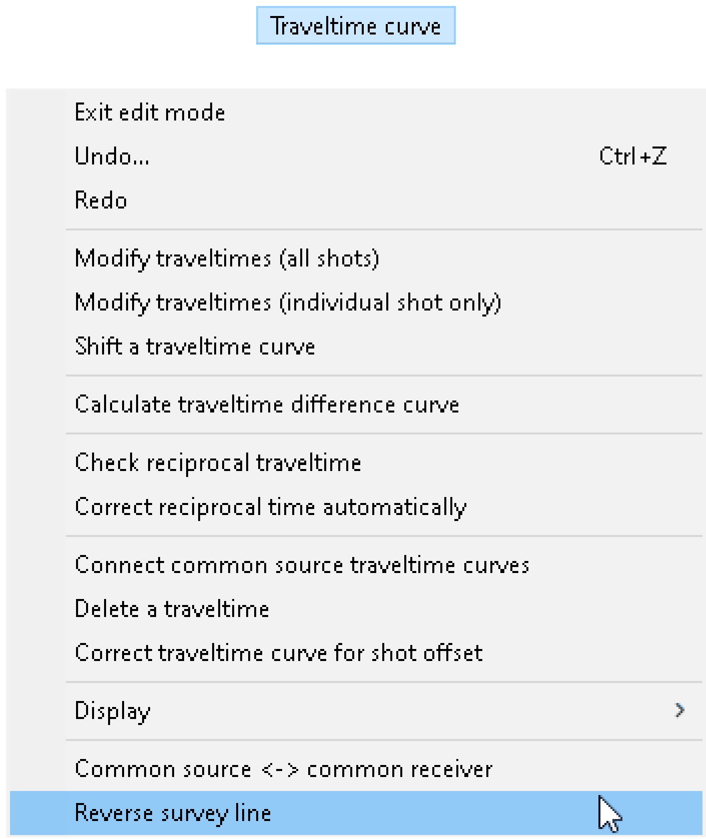


Figure 65: Common receiver gather.

4.2.15 REVERSE SURVEY LINE



To reverse the survey line, click on the *Reverse survey line* toggle:

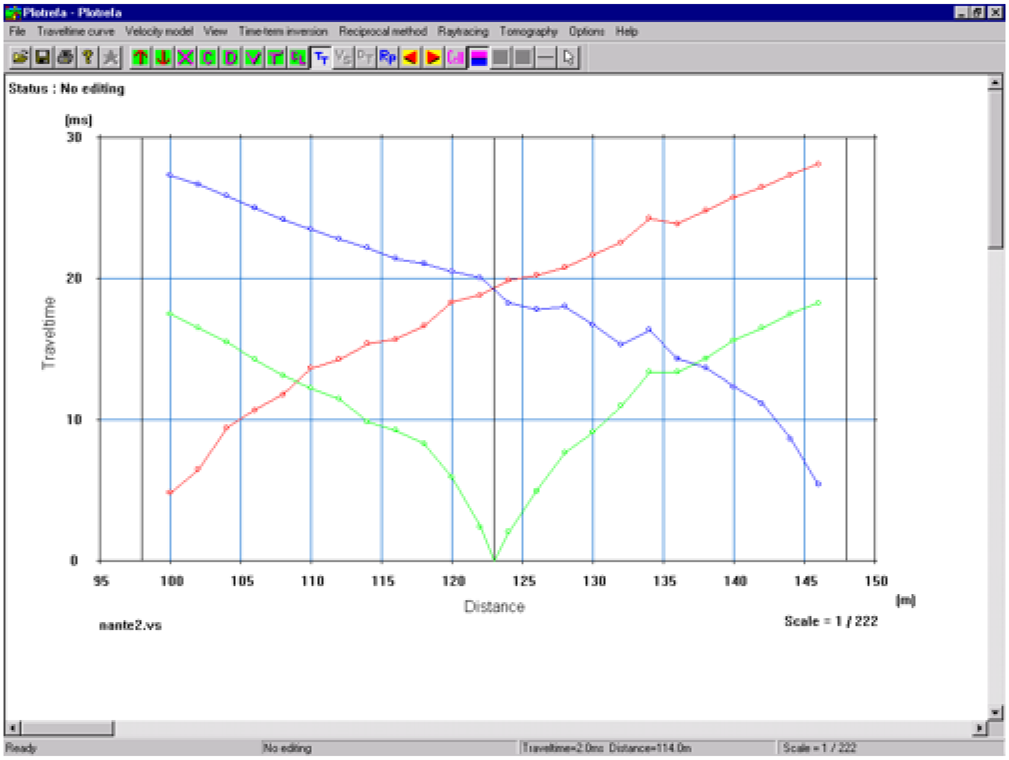
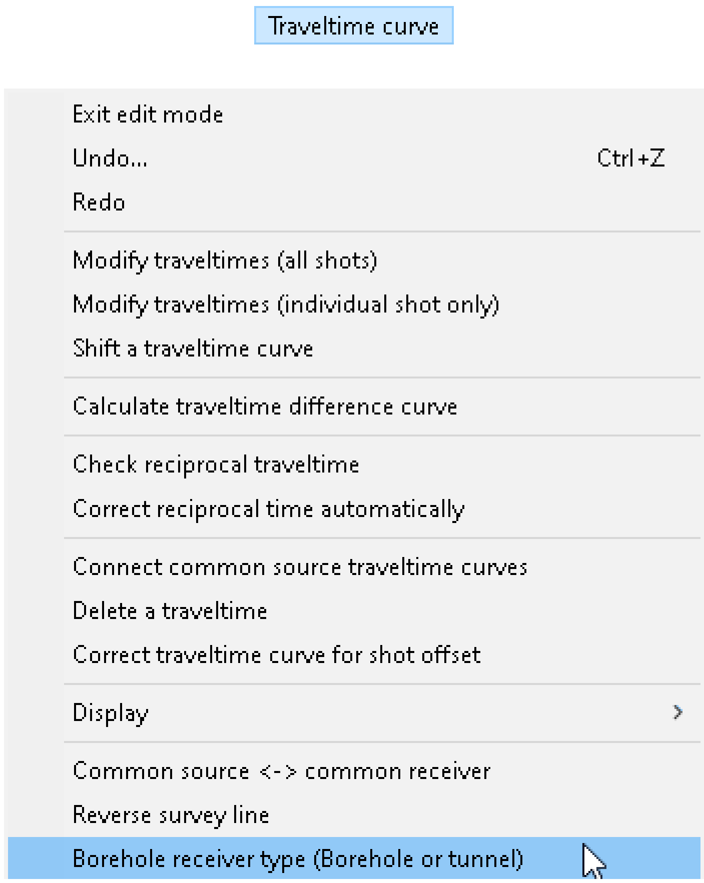


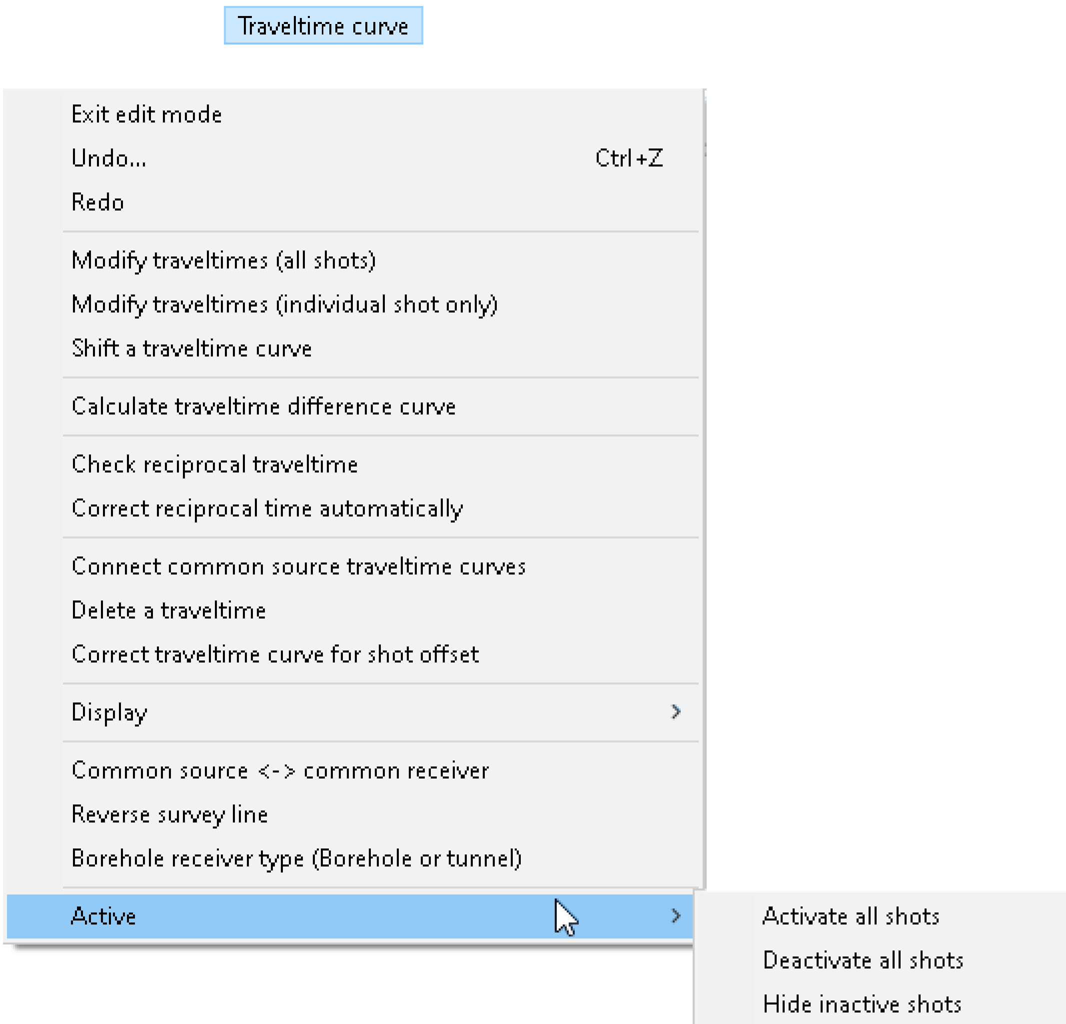
Figure 66: Reversing the survey line.

4.2.16 BOREHOLE RECEIVER TYPE (BOREHOLE OR TUNNEL)



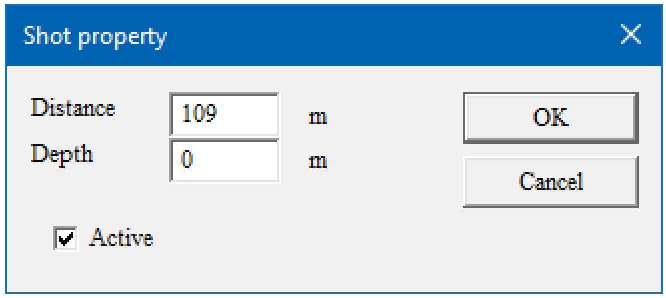
When you read in borehole data (see Section 4.1.9, Page 135), you are asked whether the borehole is vertical or horizontal. If you make a mistake (i.e., indicate vertical when it is horizontal or *vice versa*), you can correct it here. Simply select this option and it will toggle between vertical (borehole) and horizontal (tunnel).

4.2.17 ACTIVE



Once you have picked your first arrivals and have displayed your travel time plot, it is sometimes desirable to leave one or more shot records out of the interpretation. You might do this because of internal inconsistencies evident in the travel times, such as poor refractor parallelism.

To deactivate a shot record, double-click on one of the travel times to display the following dialog:



Note: *This dialog box also allows you to update the location and depth of the shot point.*

Uncheck the *Active* box. The shot record will either disappear altogether, or be displayed as follows:

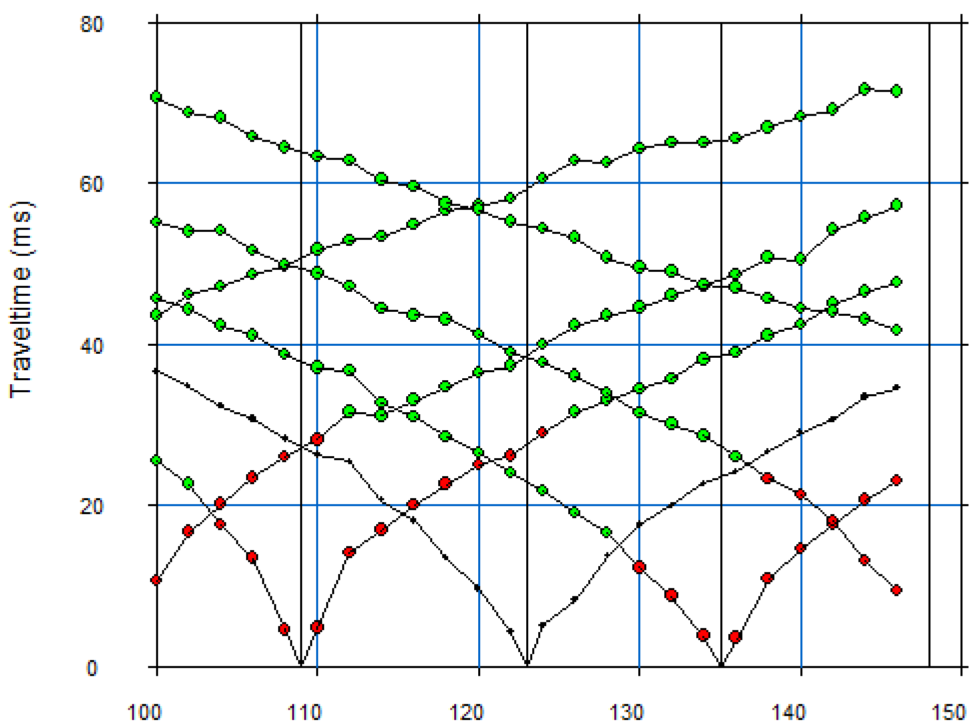
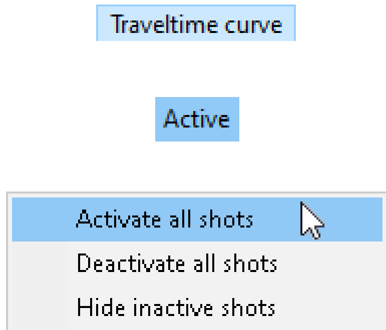


Figure 67: Deactivating a shot record in a spread.

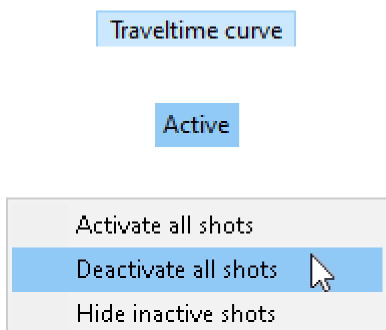
In the above travel time plot, the center shot has been deactivated. It will not be included in the calculation of the velocity section. If it disappears altogether, the *Hide inactive shots* toggle switch is thrown. Read on.

4.2.17.1 Activate All Shots



If you have deactivated some of or all the shot records, you may bring them back to active status by selecting *Activate all shots*.

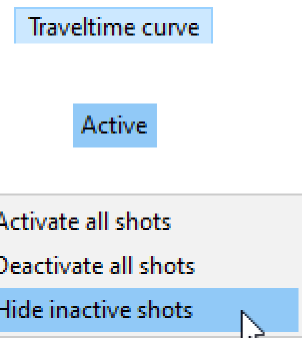
4.2.17.2 Deactivate All Shots



To deactivate all the shots in a shot record, select *Deactivate all shots*.

Note: *Deactivating all shots might seem counterintuitive. However, if you have 100 shots and only wish to view a small subset of them, it is more expedient to deactivate all of them and then activate the subset you wish to have active one-by-one.*

4.2.17.3 HIDE INACTIVE SHOTS ✓



This is a toggle switch that hides and un-hides inactive shots. In the figure below, the center shot has been deactivated and hidden. It may be made visible again by selecting *Hide inactive shots* again.

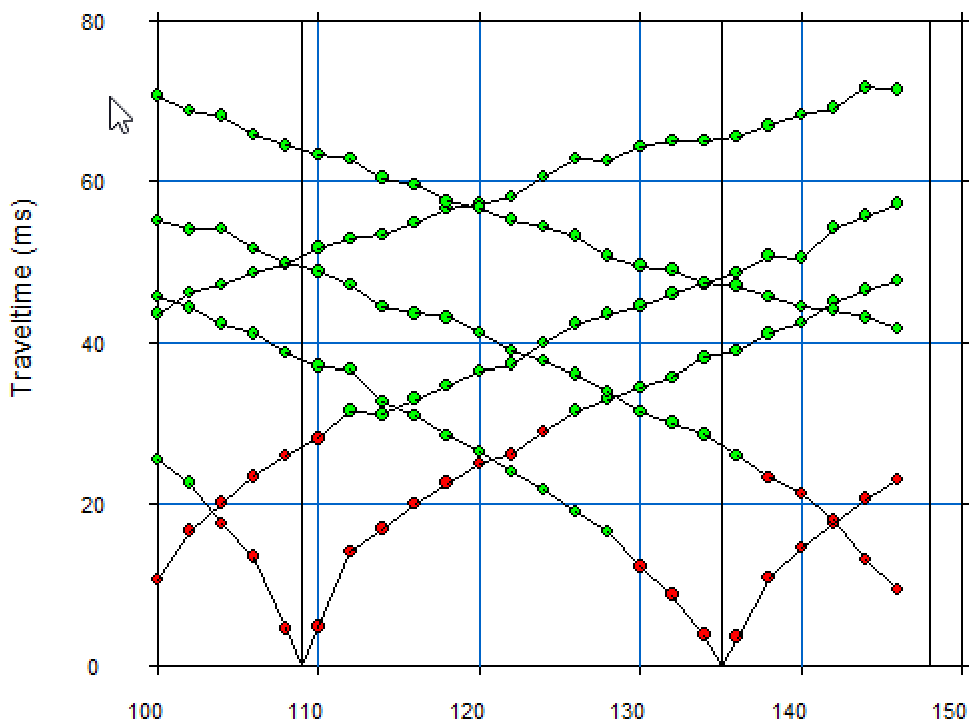
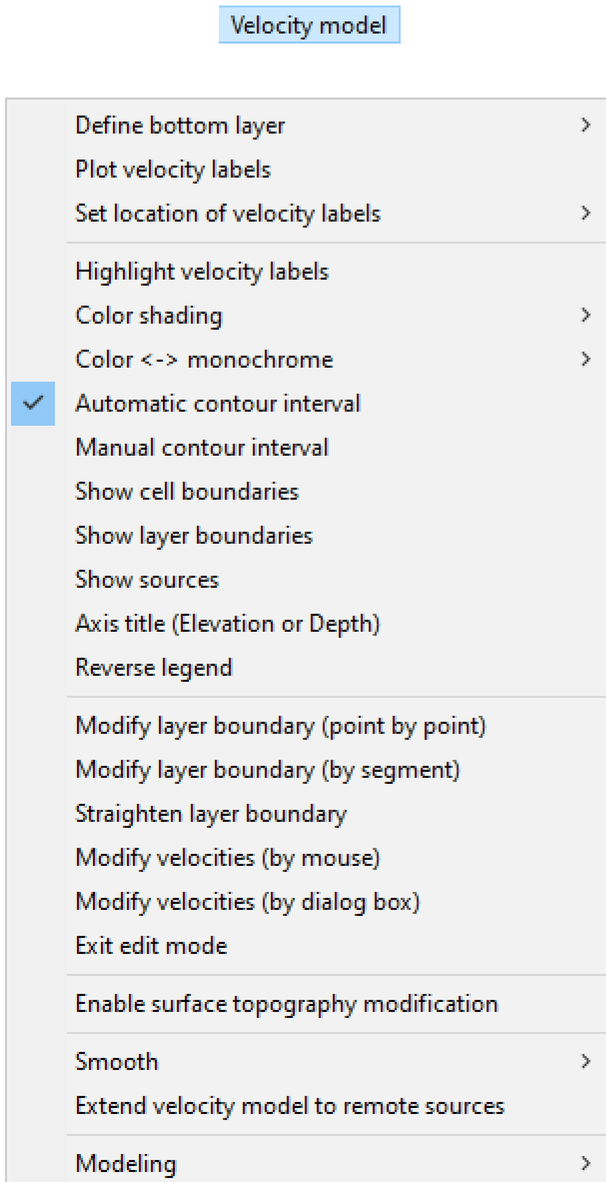


Figure 68: Travel time plot with deactivated center shot hidden.

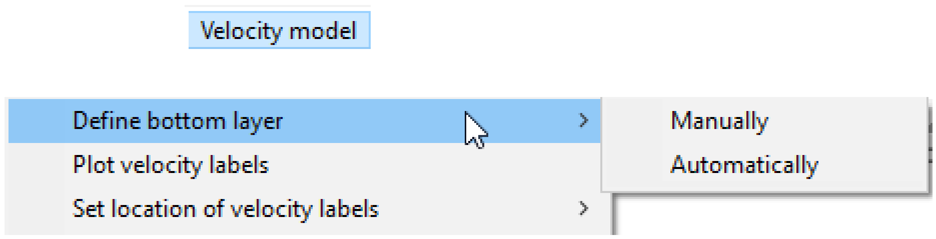
4.3 VELOCITY MODEL MENU

Click on *Velocity model* to reveal the **Velocity Model** menu:



The **Velocity Model** menu allows you to edit a velocity model and control its appearance. A velocity model can be generated synthetically with the modeling module (discussed in Section 4.3.23, Page 224), or it may be calculated from seismic data. A velocity model generated from the dataset above will be used for the purposes of illustrating the features of this menu.

4.3.1 DEFINE BOTTOM LAYER



The program automatically assigns a thickness to the bottom layer of an interpreted velocity model. But in a refraction survey, there is insufficient information to determine the thickness of the lowest layer detected; it is therefore assigned a "reasonable" thickness. However, by drawing the bottom layer with a certain thickness, it can give the impression that this thickness is known. It is therefore sometimes desirable to manually define the base of the bottom layer.

One way to deal with this is to determine the *maximum* thickness of the bottom layer. You can do this *if* you can make a reasonable estimate of the maximum velocity for the layer below it (perhaps you have borehole data or a deeper refraction survey nearby), and using a crossover distance equivalent to the greatest shot-geophone distance in your survey (i.e., you just missed seeing the next layer). Then compute the maximum depth from

$$depth = x_{cross} / 2\sqrt{(V_{n+1} - V_n / V_{n+1} + V_n)} ,$$

where x_{cross} is the assumed crossover distance, V_n is the velocity of the bottom layer, and V_{n+1} is the assumed maximum velocity.

Once you have computed a maximum depth, you may modify the base layer to reflect this. In the velocity model below, the assumed maximum thickness has been drawn on the model (red line).

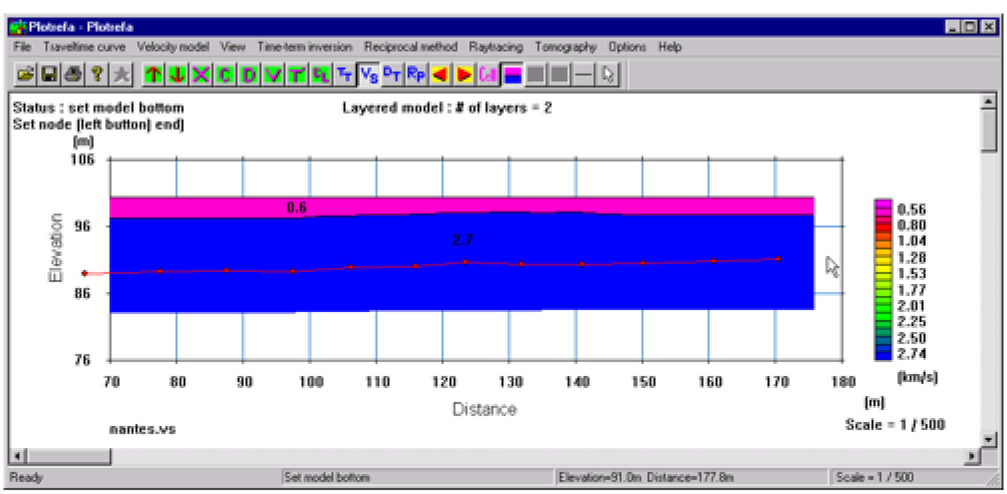
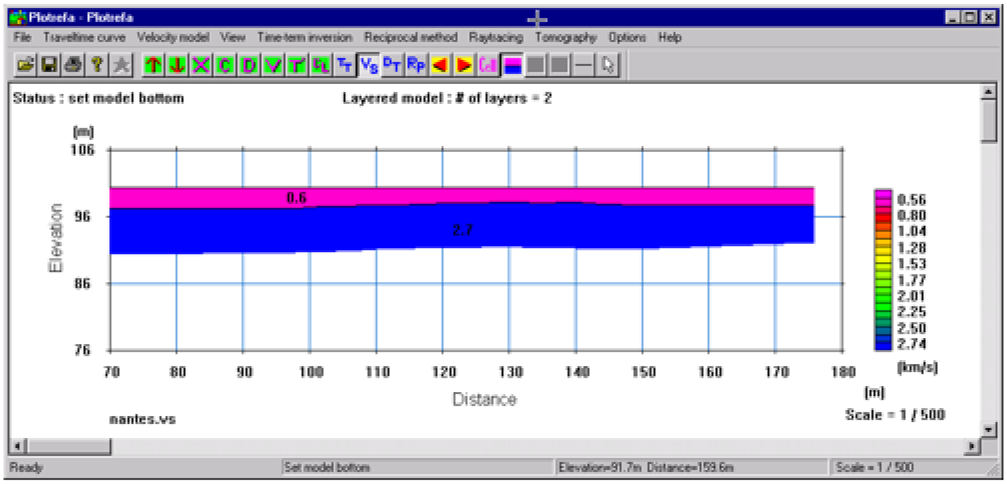


Figure 69: Defining the base of the bottom layer in a velocity section.

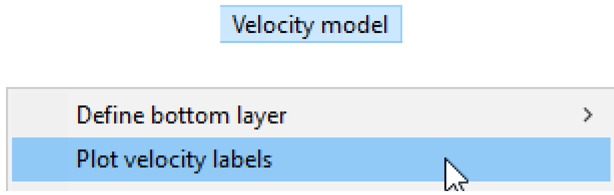
This is accomplished by clicking on *manually* in the sub-menu, and then clicking at various points along the line with the mouse. You must start outside the left edge of the velocity model, as shown above. To complete the process, click outside the right edge of the velocity model (see cursor) above:



To revert to the automatic thickness, simply select *automatically*.

Note that if you do compute a maximum thickness of the deepest layer, it is good practice to report the assumed velocity of layer n+1.

4.3.2 PLOT VELOCITY LABELS ✓



You may choose whether to show velocity labels in the model. Below is a model with velocity labels shown, followed by the same model without the velocity labels shown:

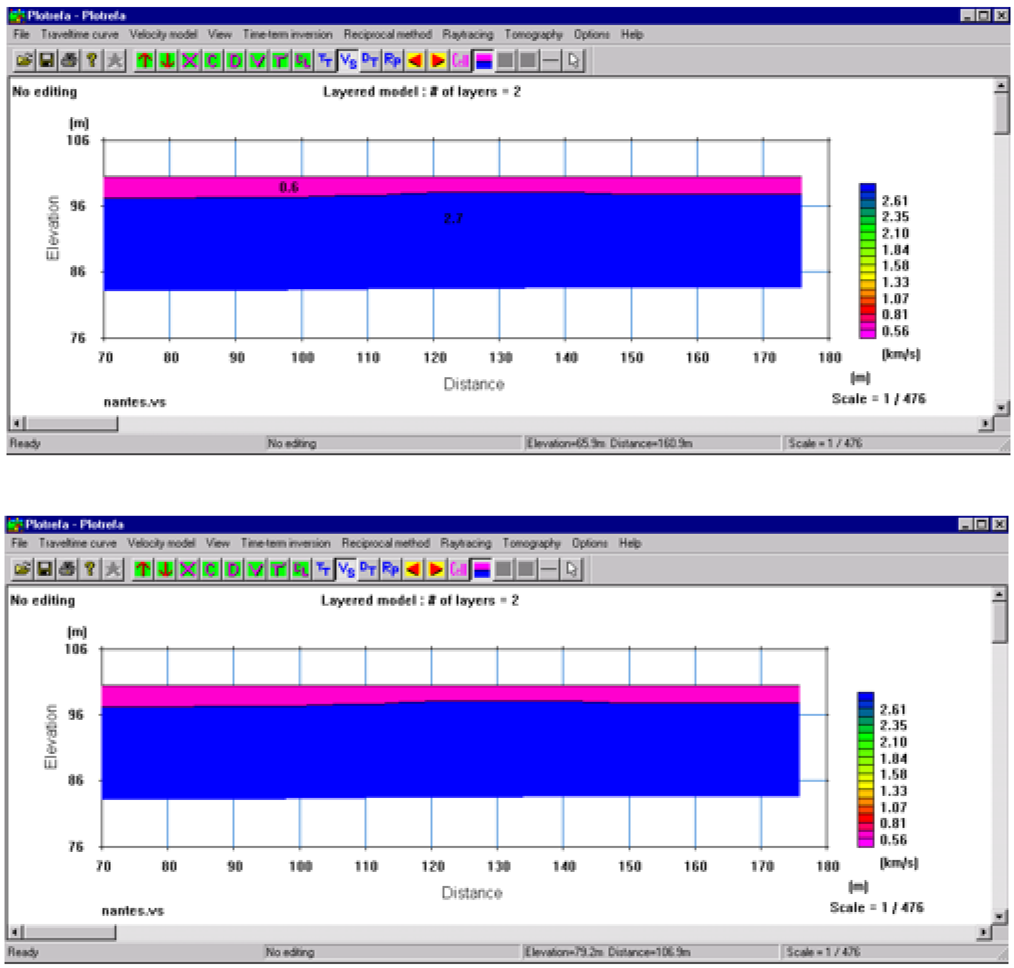
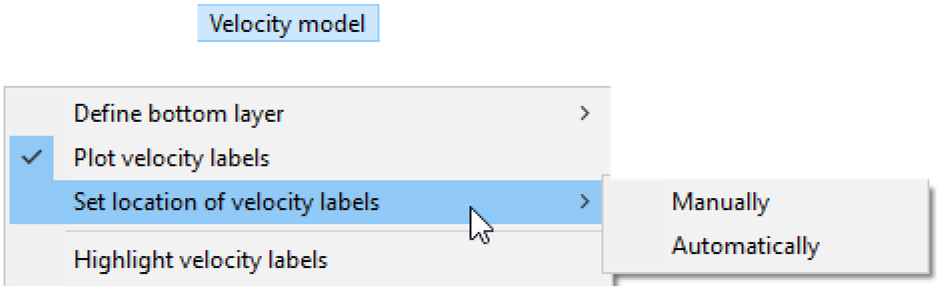


Figure 70: Top section: velocity labels included. Bottom section: velocity labels omitted.

4.3.3 Set Location of Velocity Labels



The locations of velocity labels may be set manually, or the program can set them automatically. To set it manually, simply click and hold on the center of the label, drag it to the desired location, and release:

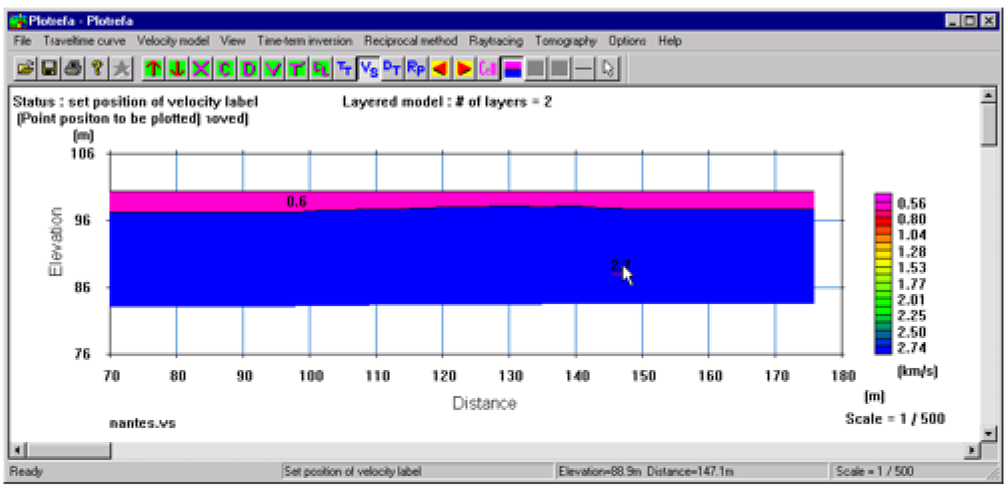
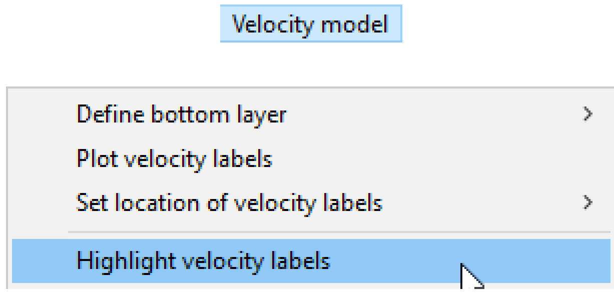


Figure 71: Positioning velocity labels on a cross-section.

Note: *A small red line will appear under the label to indicate that you have successfully grabbed it with the mouse.*

4.3.4 HIGHLIGHT VELOCITY LABELS ✓



Sometimes, depending on the colors used in the velocity model, the velocity labels may be difficult to see (the velocity model on Page 189 is a perfect example of this). As such, you may highlight the velocity labels to make them more visible. Simply choose *Highlight velocity labels* from the menu. Below is the same velocity model with the labels highlighted:

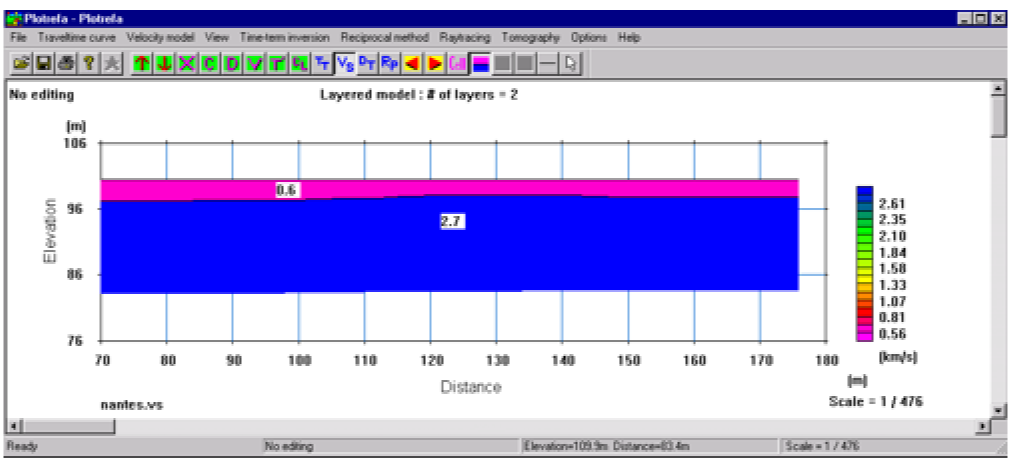
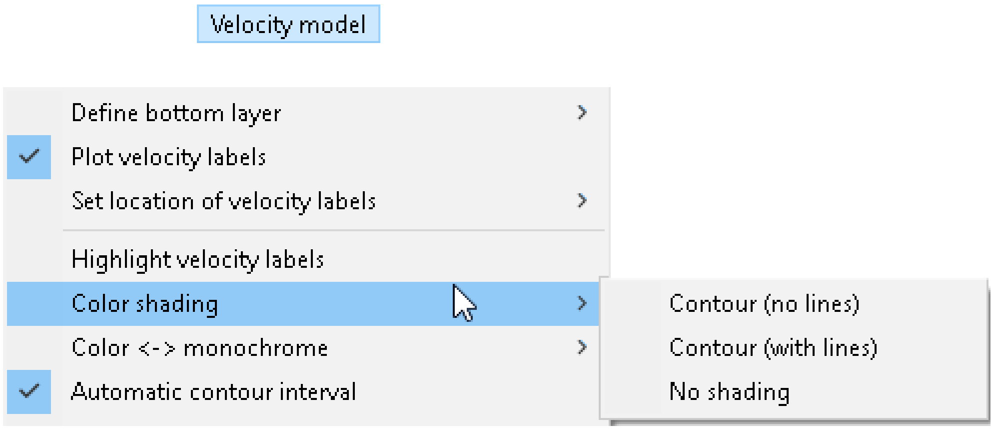


Figure 72: Highlighting velocity labels on a cross-section.

4.3.5 COLOR SHADING



If you have done a tomographic inversion, your velocity model will not consist of discrete layers of constant velocity like those discussed so far, but will instead resemble that shown below:

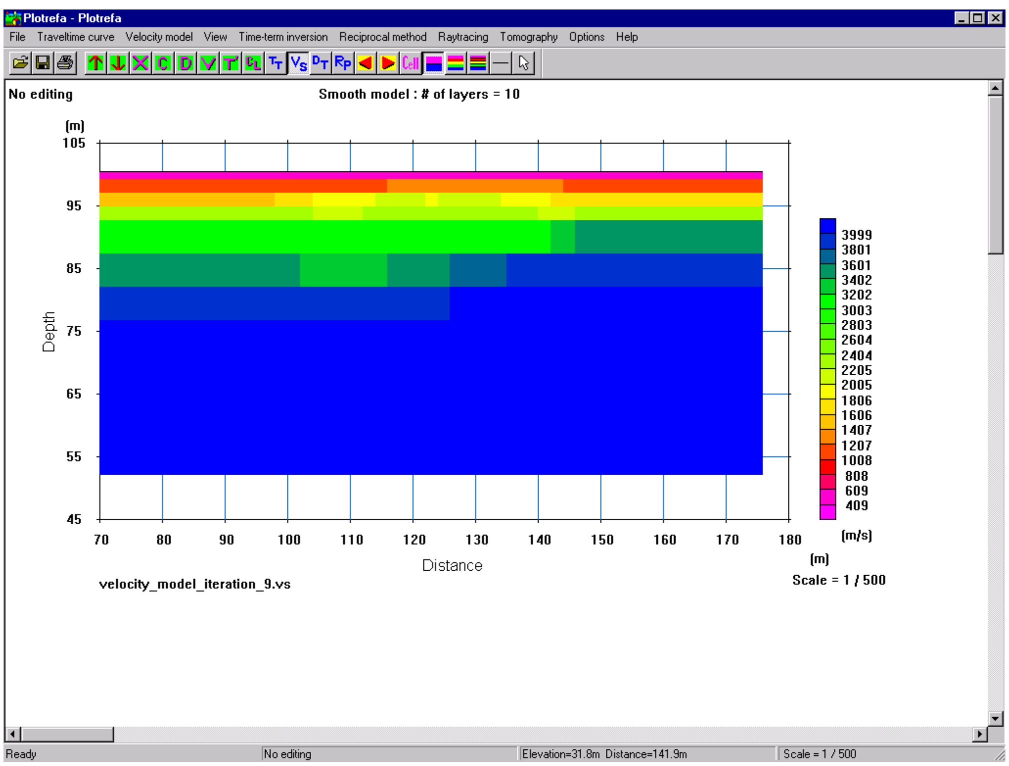
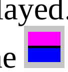


Figure 73: Displaying velocity cells in a cross-section.

In a tomographic inversion, the velocity model is divided into velocity “cells”. In the above model, the velocity for each cell is displayed. This is the default setting for tomographic inversions and is enabled by pressing the  tool button.

To create a more aesthetically pleasing velocity model, you may wish to contour the velocities. Choose *Contour (no lines)*, or press the tool button to contour the data:

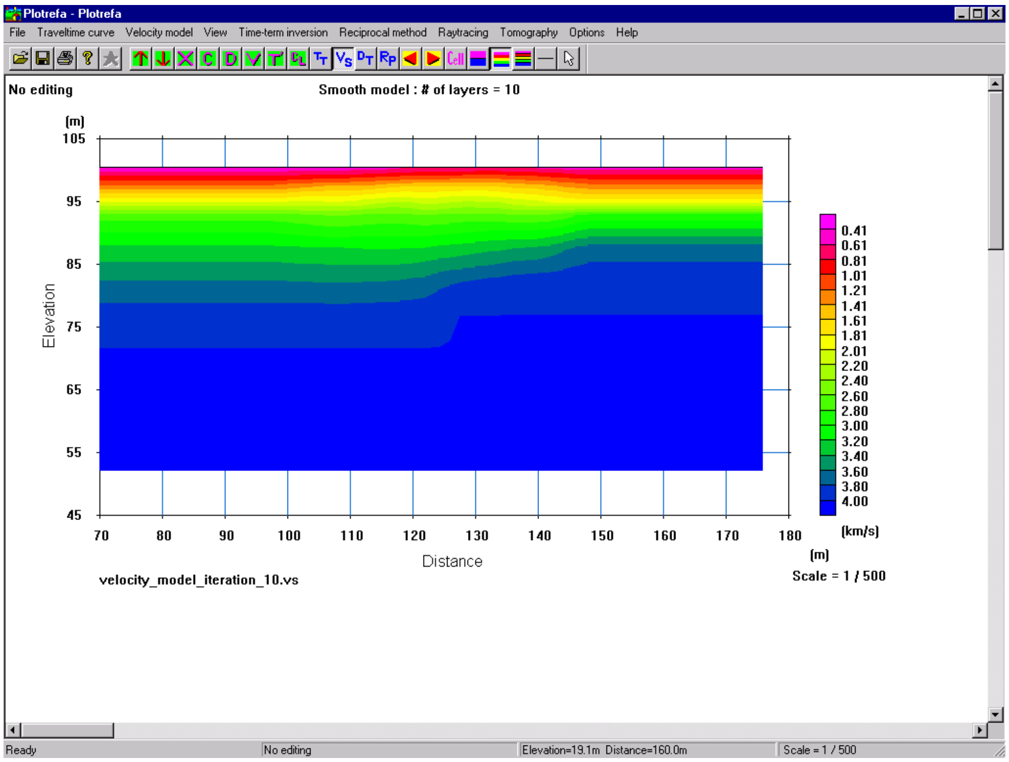


Figure 74: Contoured tomographic cross-section.

If you would like to include the actual contour lines, choose *Contour (with lines)*, or press the tool button:

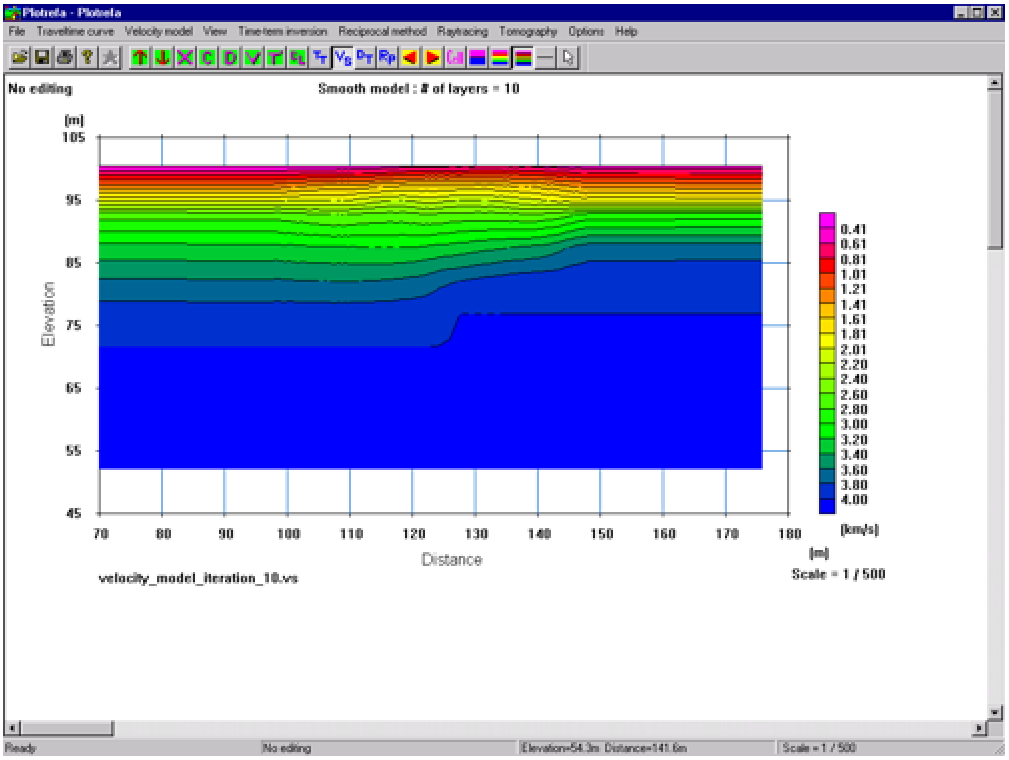


Figure 75: Contoured tomographic cross-section with contour lines.

If you have a layered model (defined as seven layers or less), like the one below,

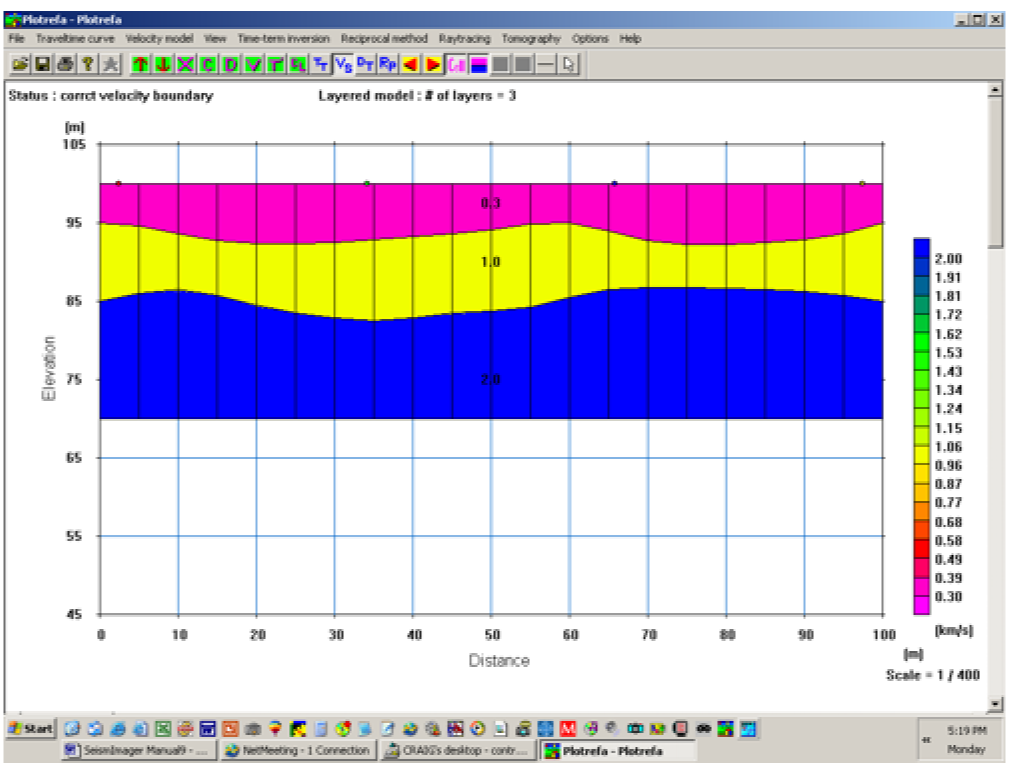


Figure 76: Discrete layered velocity model.

and you would like to remove the colors, choose *No shading*, or press the  button:

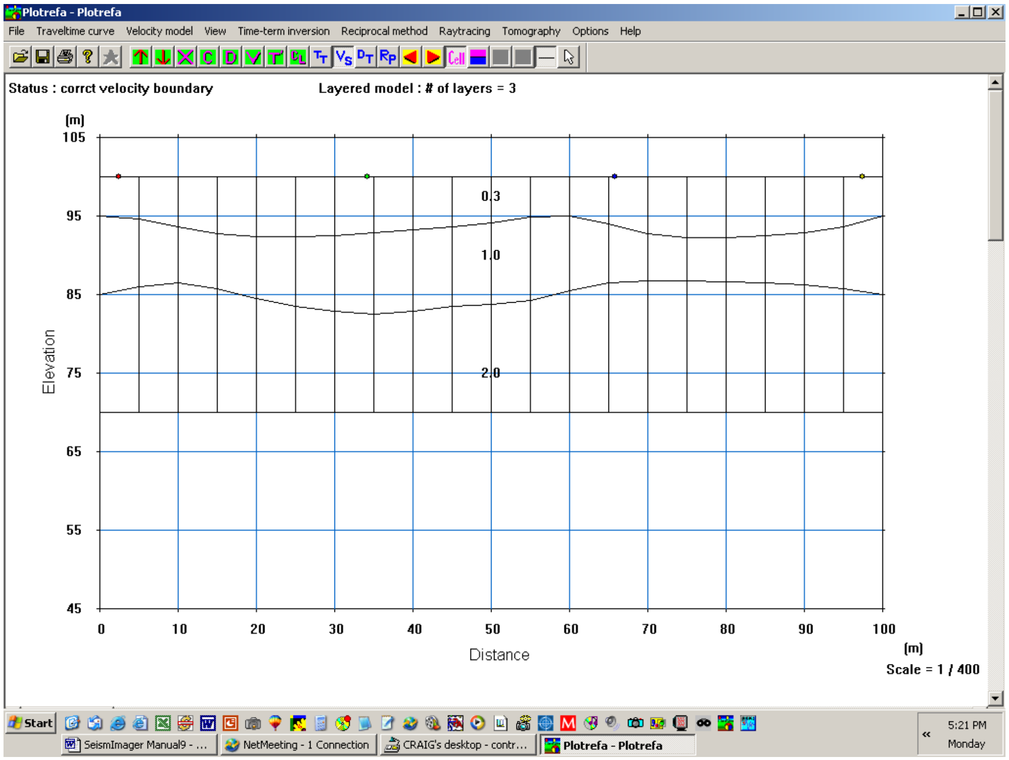
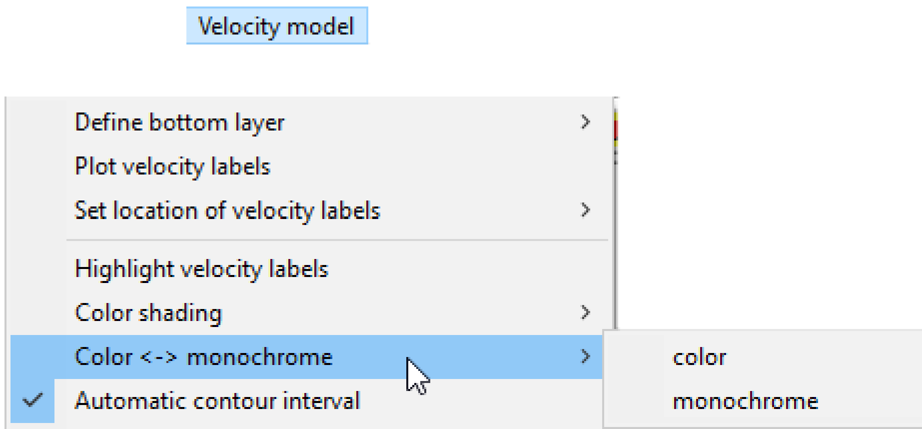


Figure 77: Layered model in black and white.

To turn color back on, press the tool button. 

Note: *Only layered models can be displayed in this manner; tomographic models cannot.*

4.3.6 COLOR <-> MONOCHROME



If you wish to show your velocity model in shades of gray, choose *monochrome*:

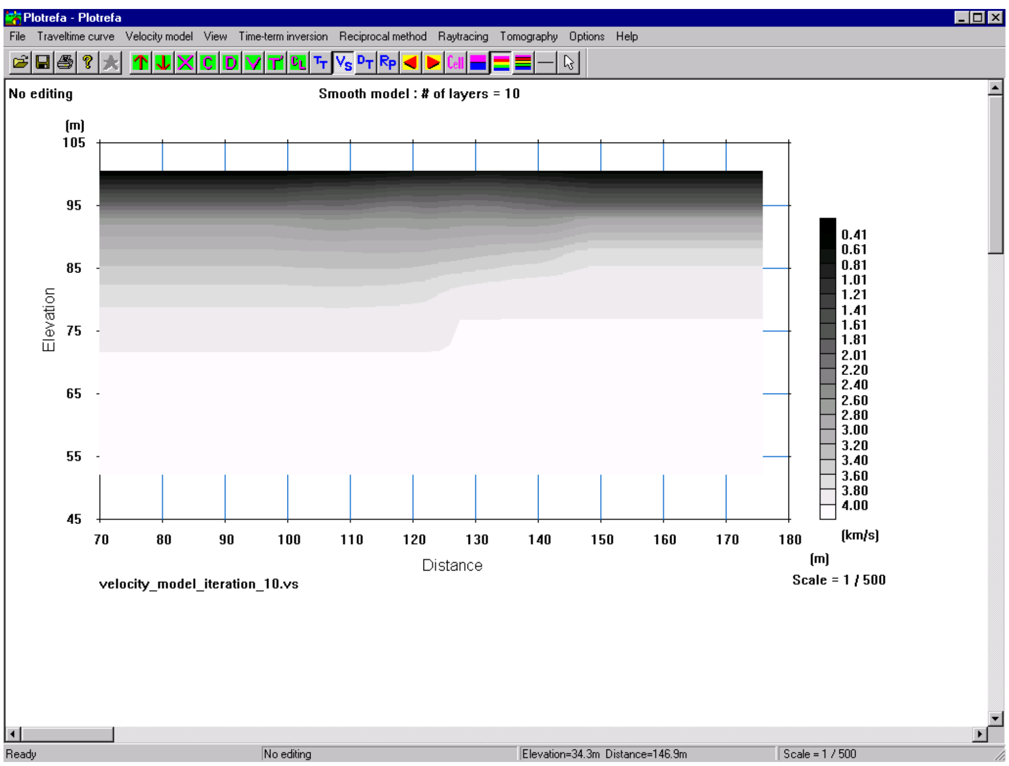
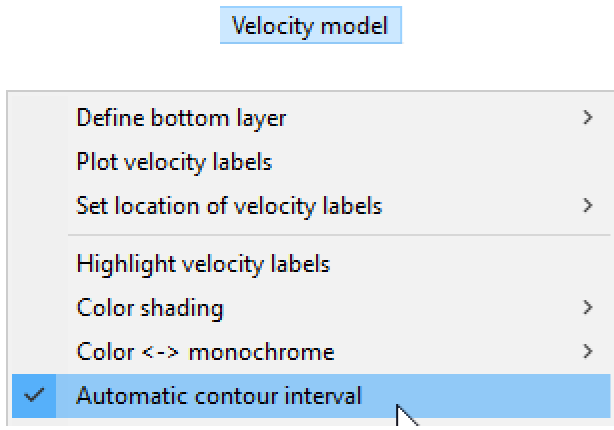


Figure 78: Tomographic velocity model shown in monochrome.

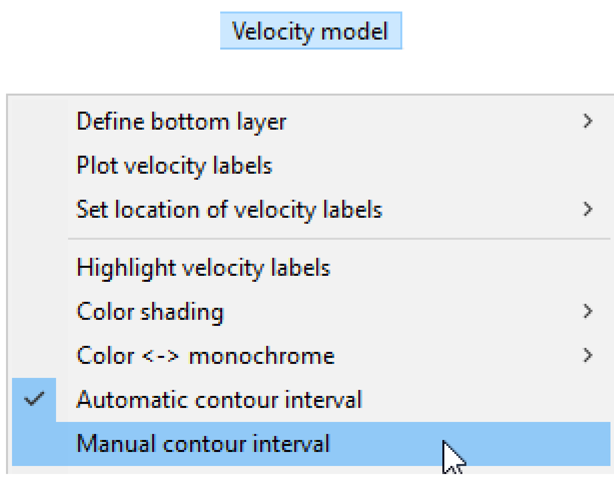
To change it back to a color display, choose *color*.

4.3.7 AUTOMATIC CONTOUR INTERVAL ✓

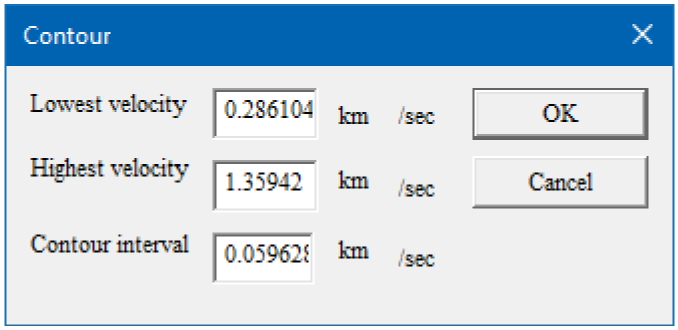


If you want the contour interval to be chosen by the program automatically, click on *Automatic contour interval*.

4.3.8 MANUAL CONTOUR INTERVAL

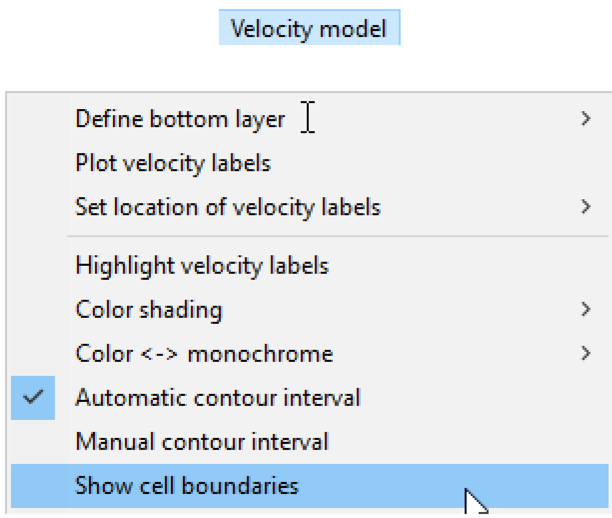


If you would like to set the contour interval manually, choose *Manual contour interval* to reveal the following dialog box:



Set the desired contouring parameters and press *OK.*

4.3.9 SHOW CELL BOUNDARIES []



If you would like to display the cell boundaries, click on *Show cell boundaries*, or press the  tool button:

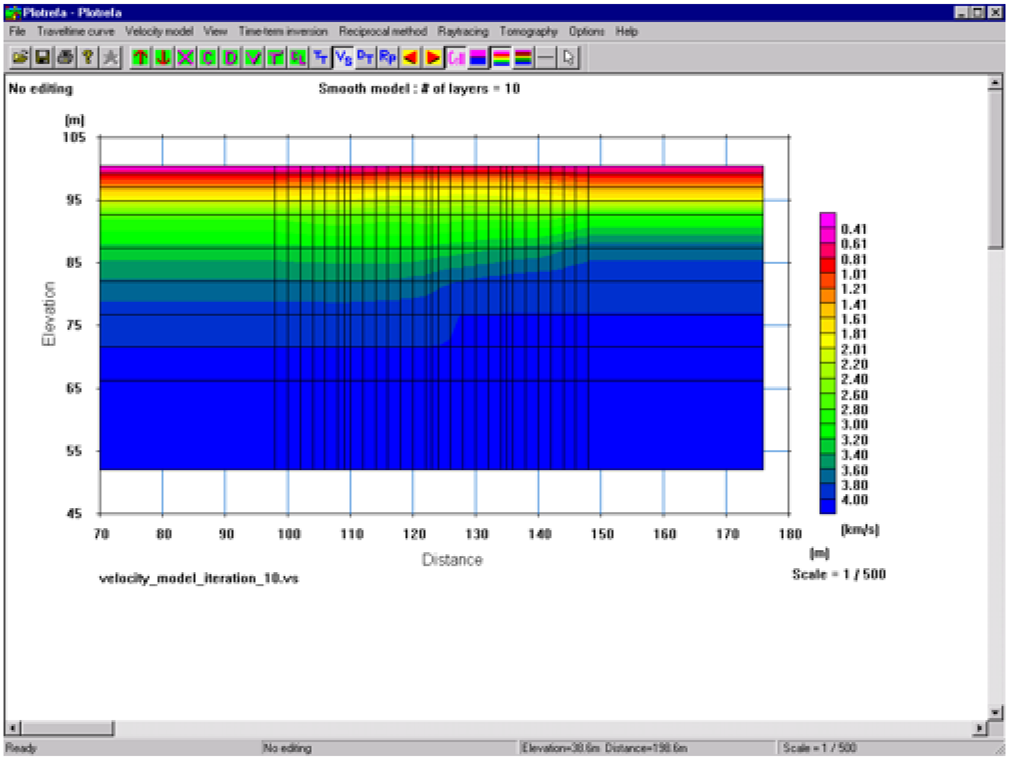
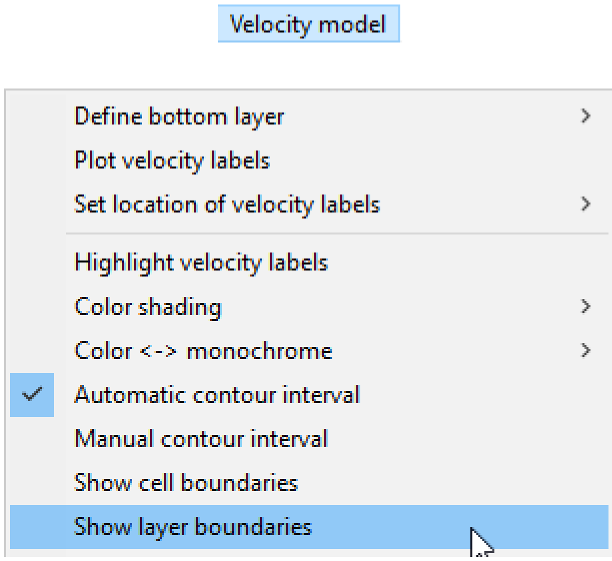


Figure 79: Displaying velocity cells.

4.3.10 Show Layer Boundaries



If you would like to display the layer boundaries of the initial model, disable the cell display and choose *Show layer boundaries*:

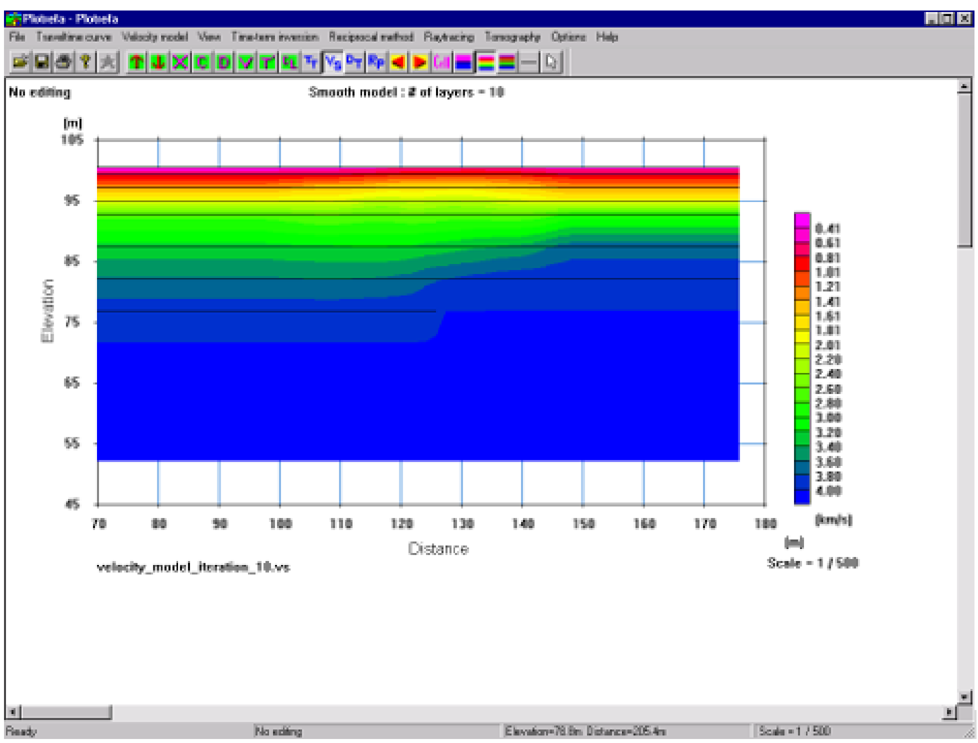
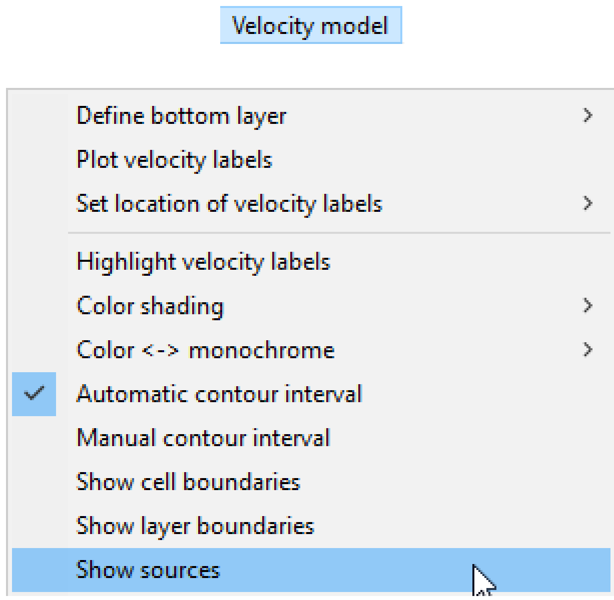


Figure 80: Displaying layer boundaries of initial model.

4.3.11 SHOW SOURCES ✓



If you wish to show where the sources are located, click on *Show sources*:

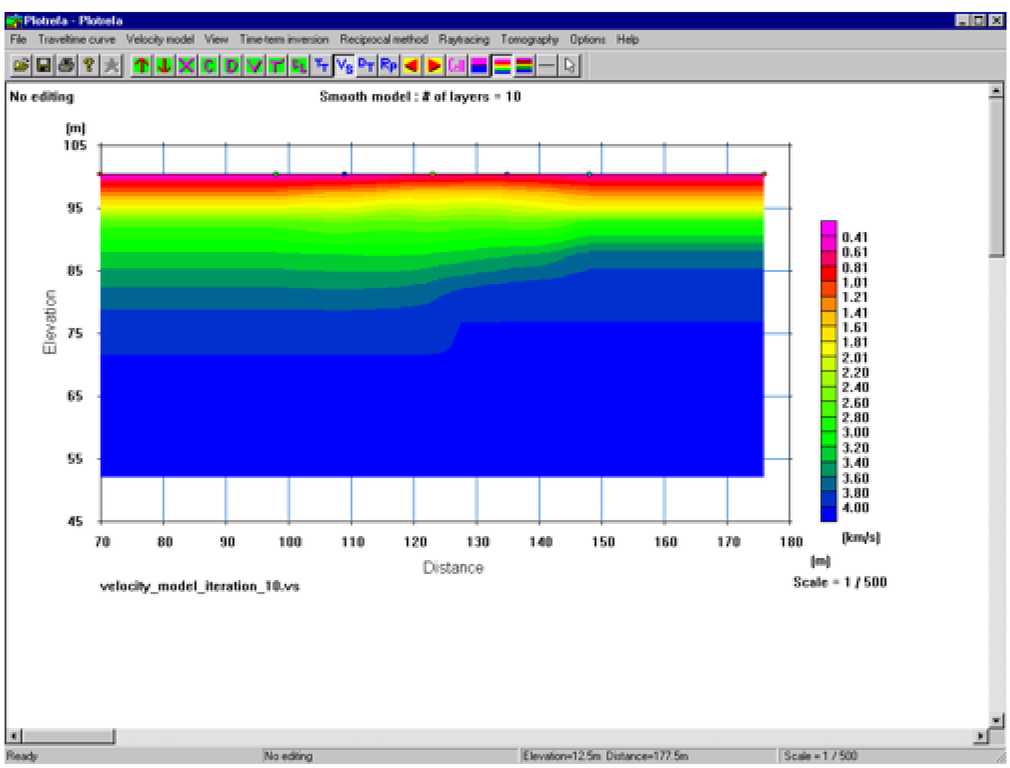
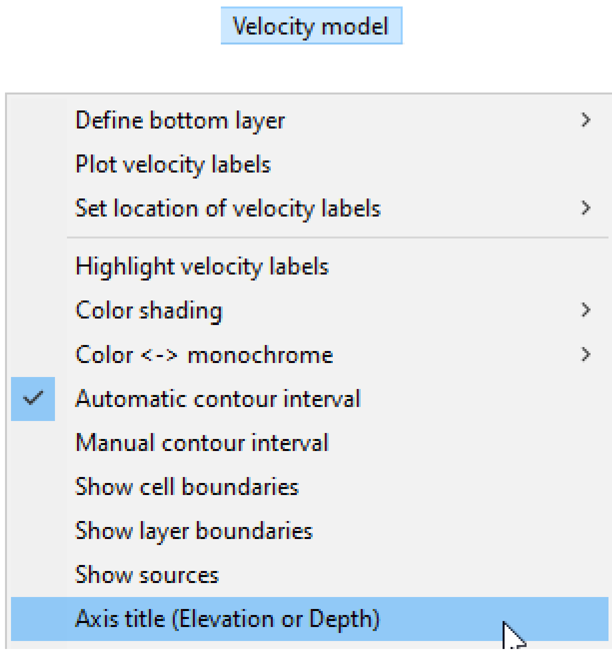


Figure 81: Displaying source locations on velocity model.

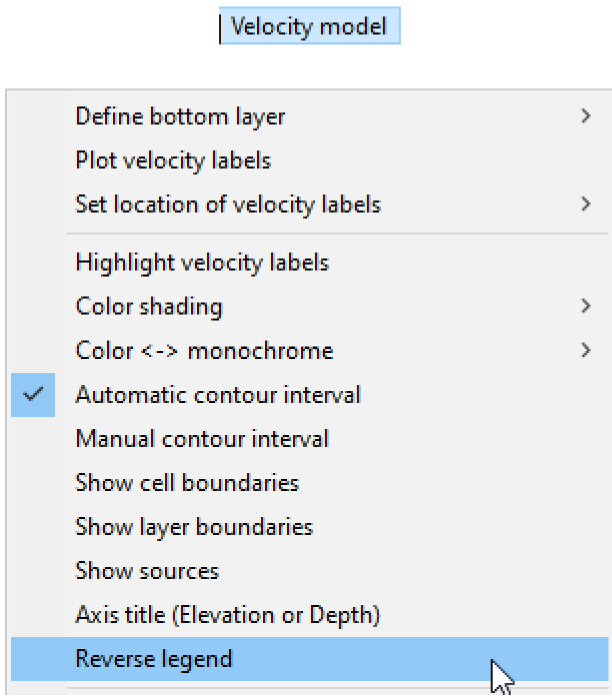
The source locations will be shown as small circles along the model surface.

4.3.12 AXIS TITLE (ELEVATION OR DEPTH)



If you have surveyed the actual elevations along the survey lines, you may label the vertical axis as an "Elevation" axis. Otherwise, you may label it a "Depth" axis. Simply click on *Axis Title (Elevation or Depth)* to toggle between the two options.

4.3.13 REVERSE LEGEND ✓



You may reverse the legend to put high velocities at the top and low velocities at the bottom, or *vice versa*. Simply click on *Reverse legend* to toggle between the two:

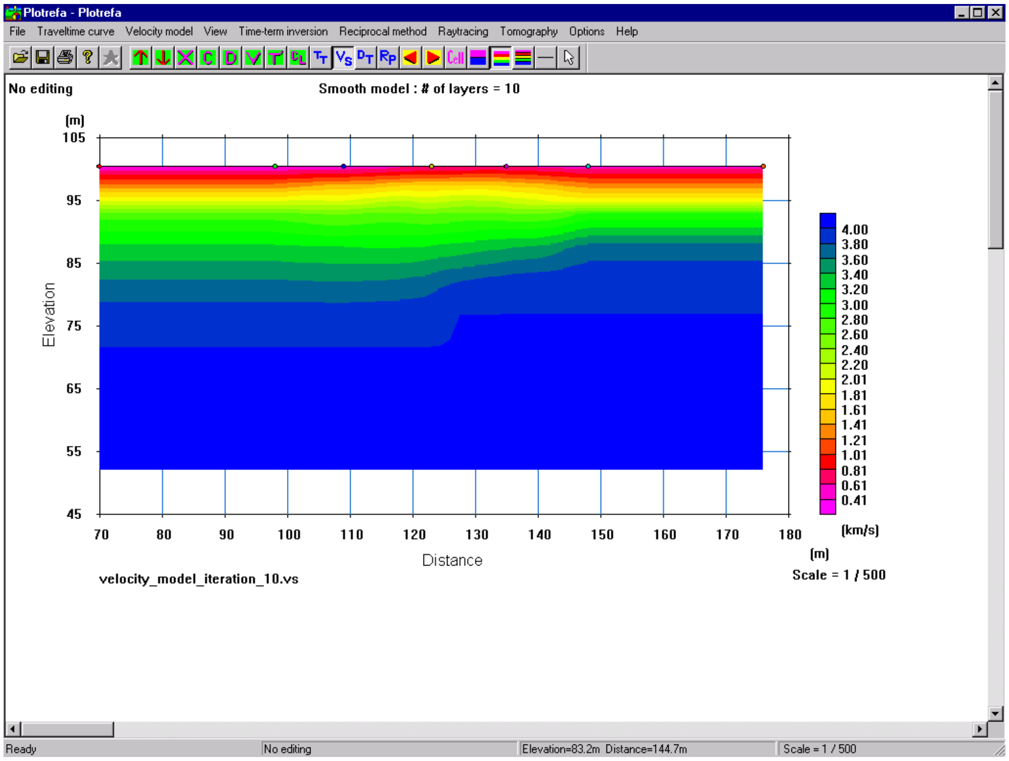
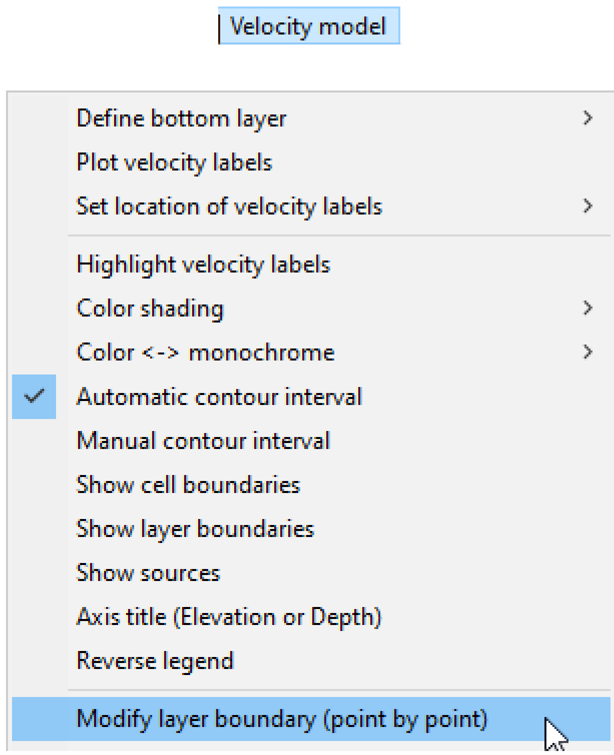


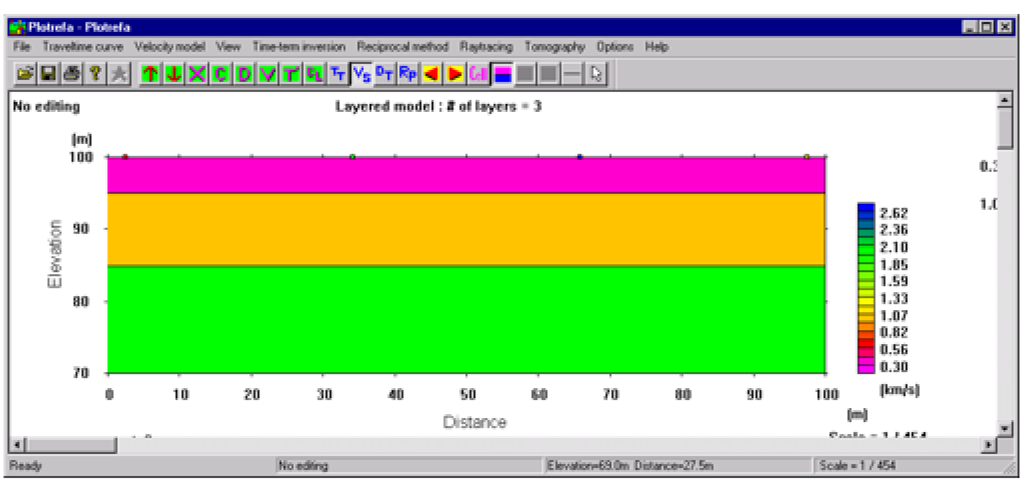
Figure 82: Labeling the vertical axis on a velocity model.

4.3.14 MODIFY LAYER BOUNDARY (POINT BY POINT) [ ]



You may modify the velocities and geometry of your velocity model. This is most useful when doing forward modeling (Section 4.3.23, Page 224).

Below is a synthetic velocity model:



To change the geometry of the velocity boundaries on a point-by-point basis, click on *Modify* *layer boundary (point by point)*, or press the  tool button. The individual velocity cells will be displayed. You may change the depth of any layer by clicking on a cell intersection and dragging the red dot to the desired depth:

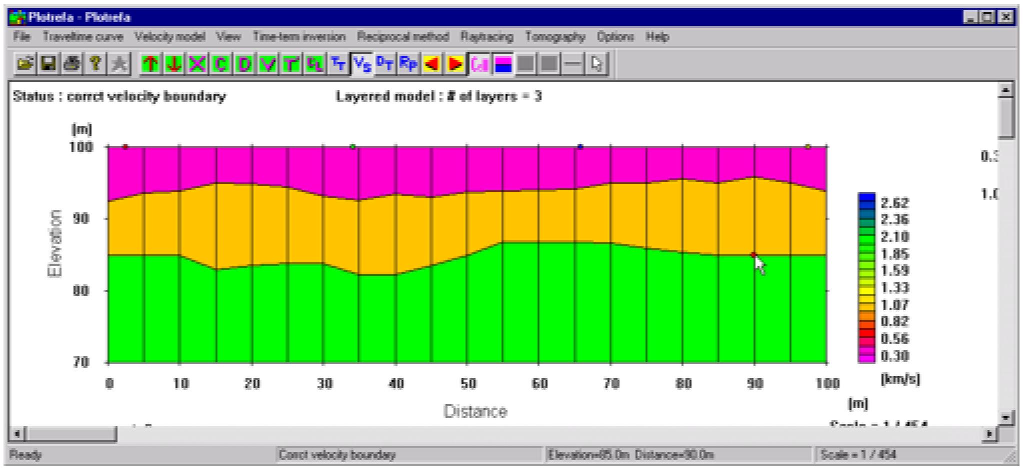
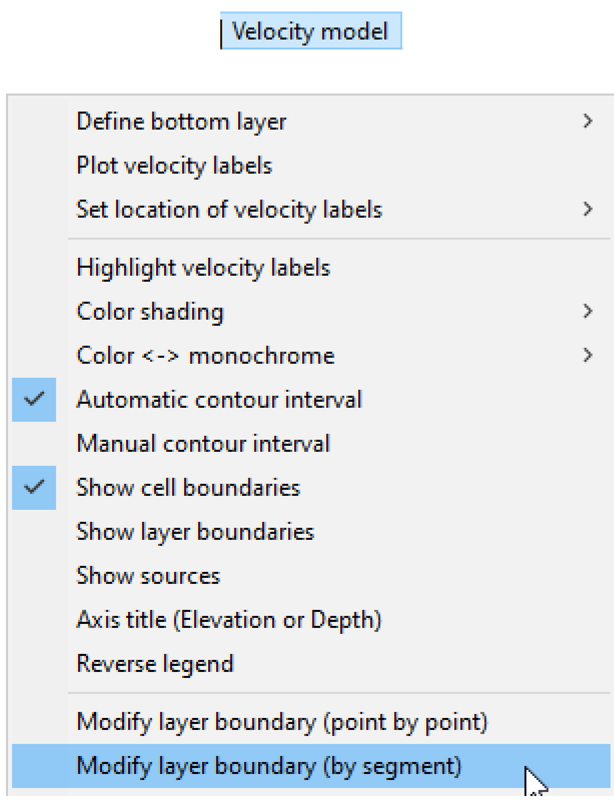
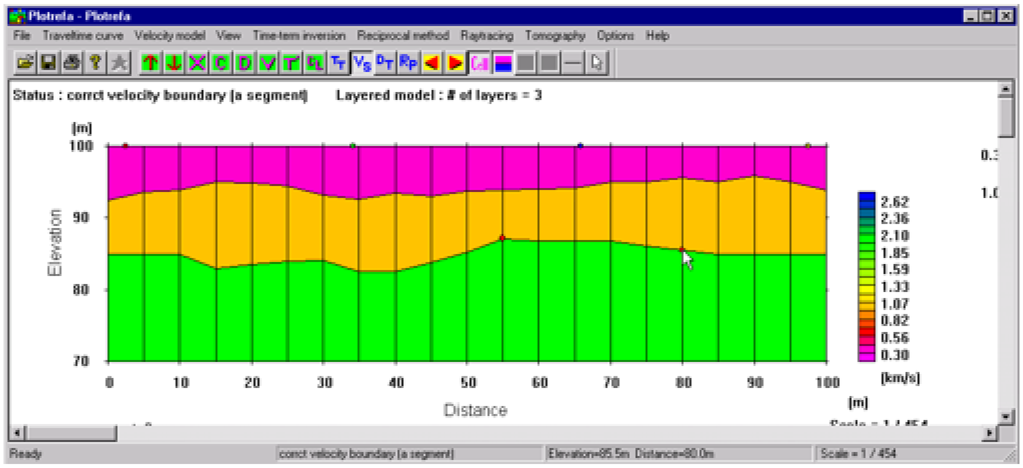


Figure 83: Modified layer boundaries using "point-to-point" option.

4.3.15 Modify Layer Boundary (by Segment)



In addition to moving individual points, you can also grab an entire segment of a boundary and move it. Choose *Modify layer boundary (by segment)*. Click on one end of the segment you wish to move. A red dot will be displayed. Now, click on the other end:



Drag the second red dot to the desired depth:

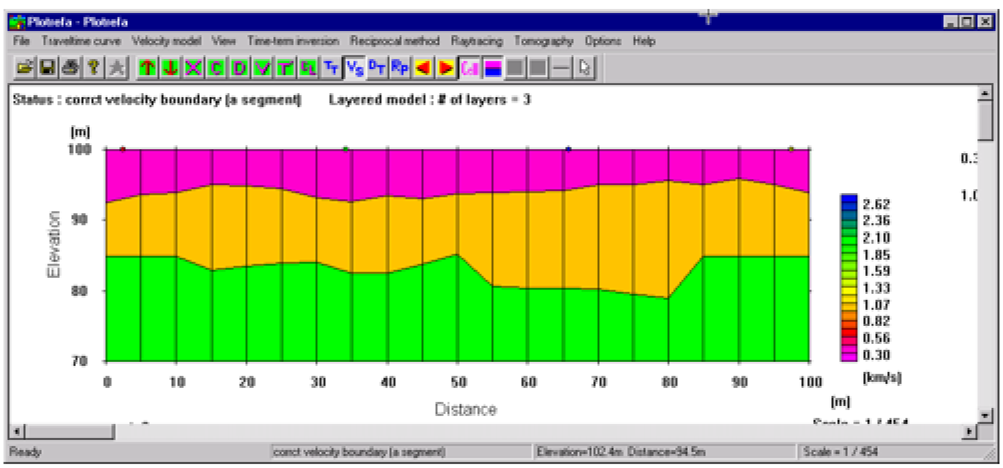
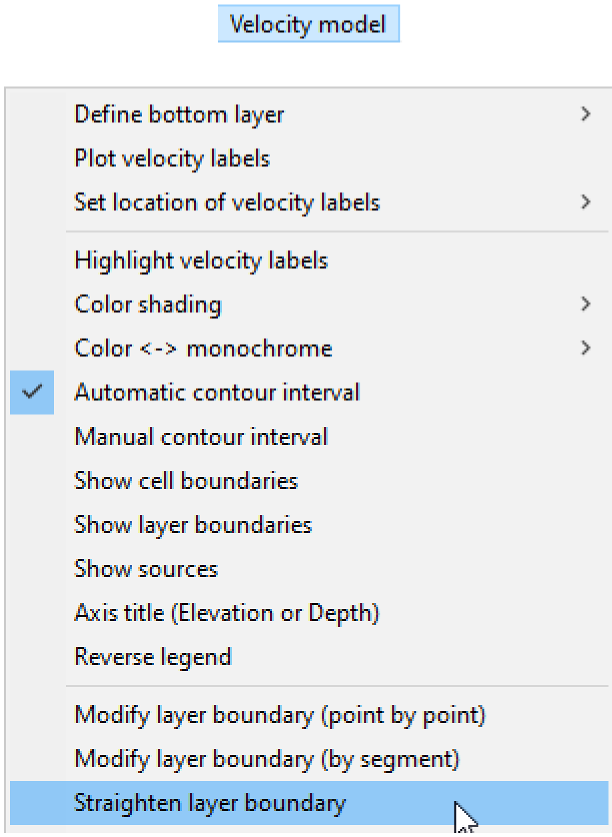
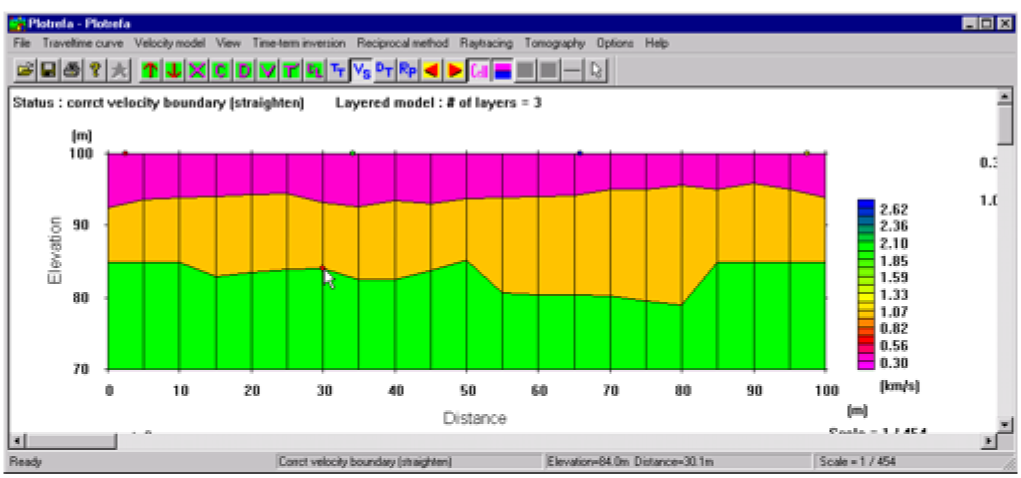


Figure 84: Modified layer boundaries using "by segment" option.

4.3.16 STRAIGHTEN LAYER BOUNDARY



If you wish to straighten the layer boundary between two points, choose *Straighten layer boundary*. Click on the first end of the segment you wish to straighten:



Then click on the other end of the segment:

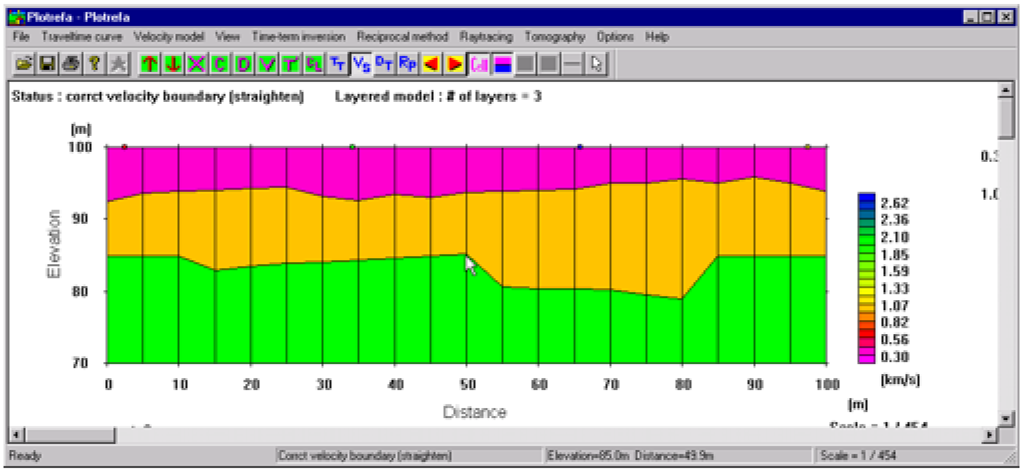
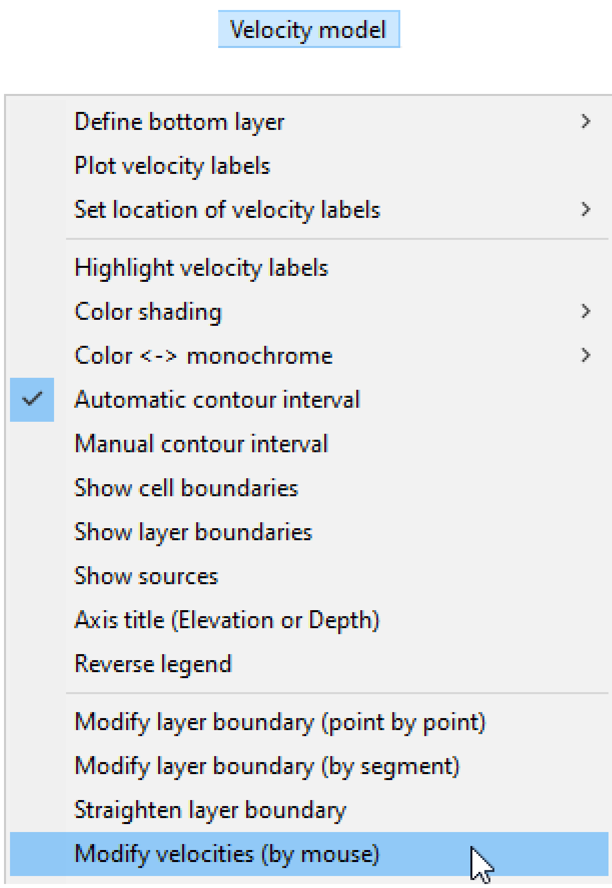


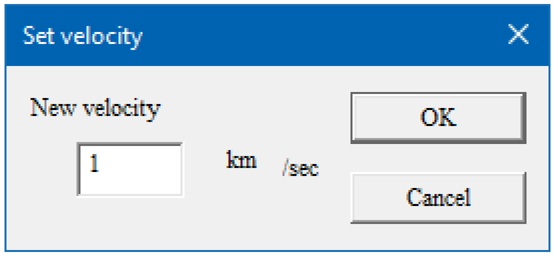
Figure 85: Straightening a layer boundary.

The layer segment will be a straight line between the two points.

4.3.17 MODIFY VELOCITIES (BY MOUSE)



In addition to editing the geometry of the model, you may also edit the velocities. You can do this via mouse or dialog box. To edit velocities using your mouse, choose *Modify velocities (by mouse)*. The following dialog box will be revealed:



Enter the desired velocity and press *OK*.

Now, click on the cells or click and drag your mouse over the region you wish to alter the velocity of:

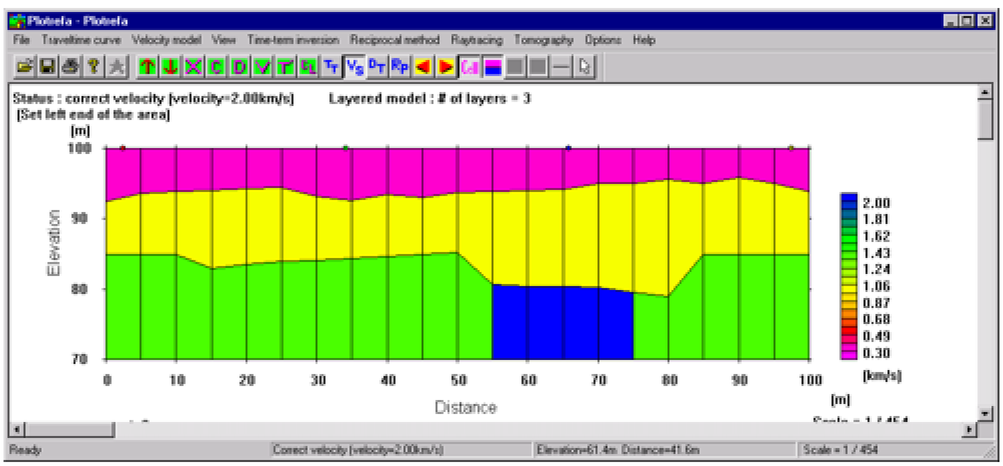
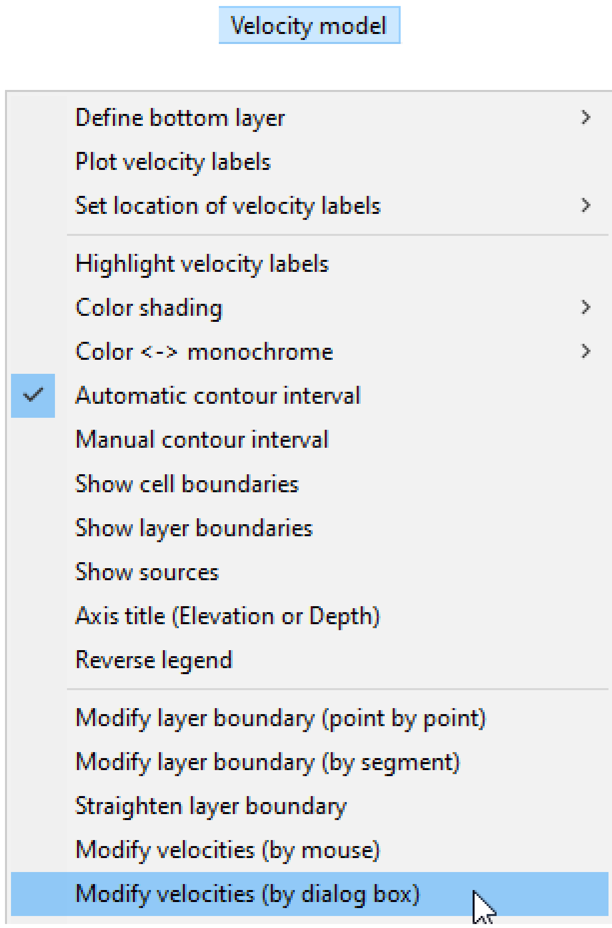
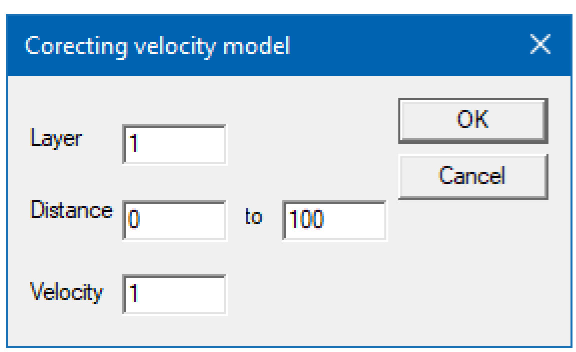


Figure 86: Modified velocity using mouse.

4.3.18 MODIFY VELOCITIES (BY DIALOG BOX)



To accomplish the above via dialog box, choose *Modify velocities (by dialog box)*. The following dialog box will appear (the values have already been filled in for this demonstration):



Indicate the layer, distance range, and new velocity, and press *OK*:

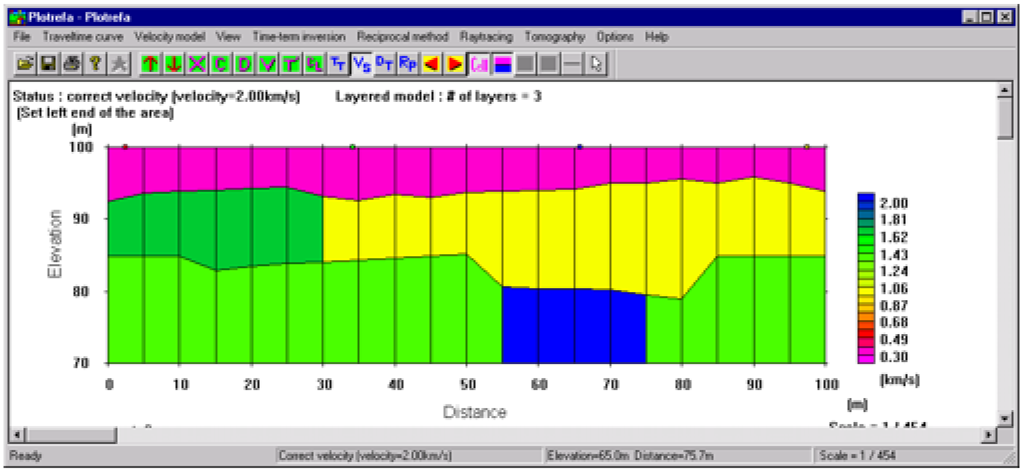
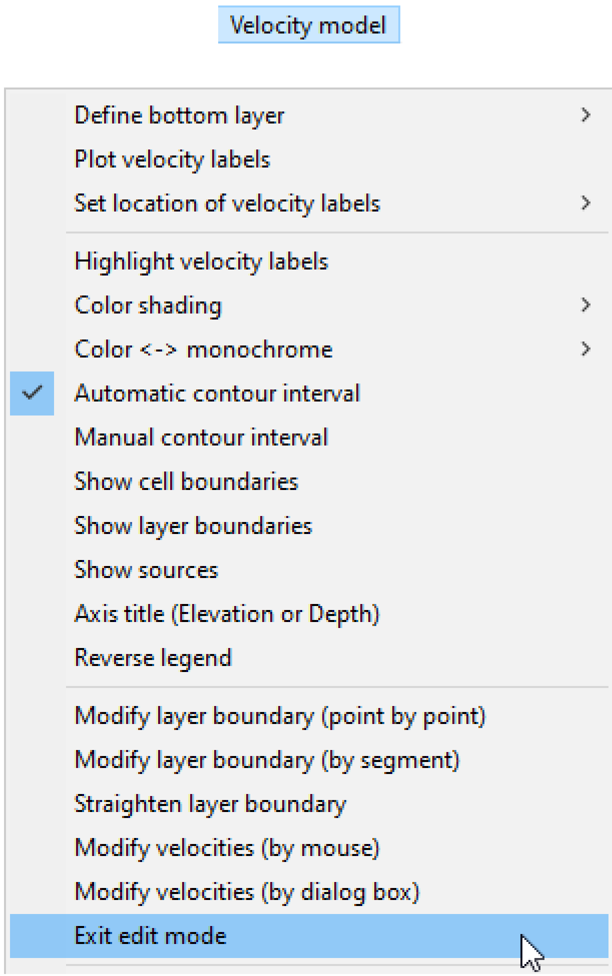


Figure 87: Modified velocity using dialog box.

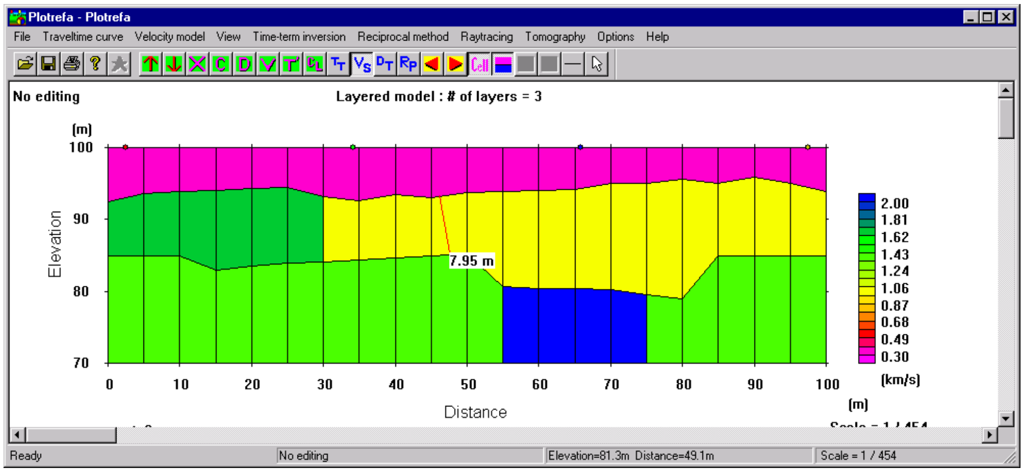
The velocity structure will be modified accordingly.

4.3.19 Exit Edit Mode [ ]

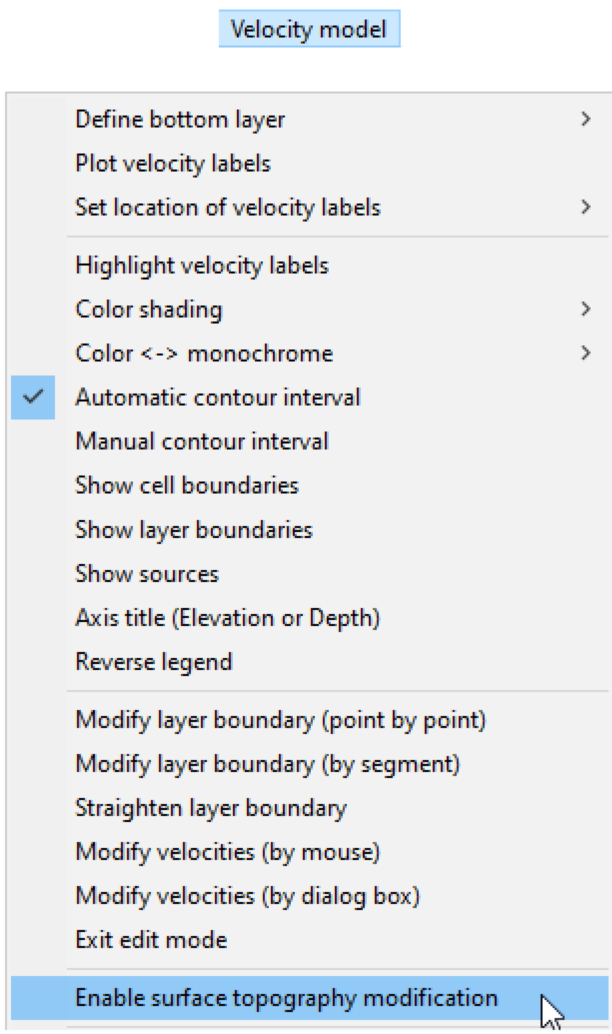


When modifying the geometry or velocities of the velocity model, you are in an "Edit" mode. To exit this mode, choose *Exit edit mode,* or just press the tool button.

In a manner similar to Pickwin (see Section 4.2.1, Page 154), when not in edit mode, you can use the mouse to measure vertical distances:

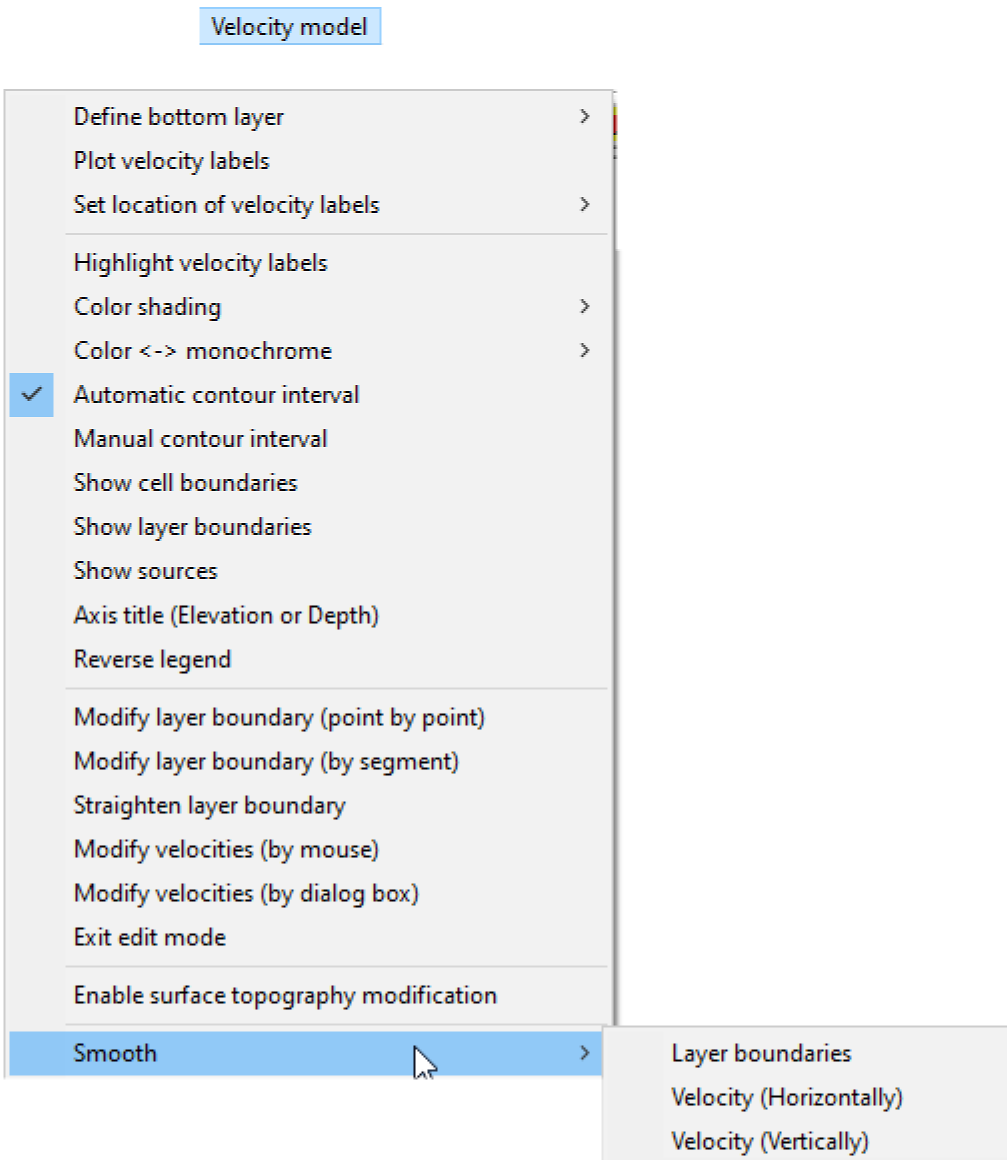


4.3.20 Enable Surface Topography Modification

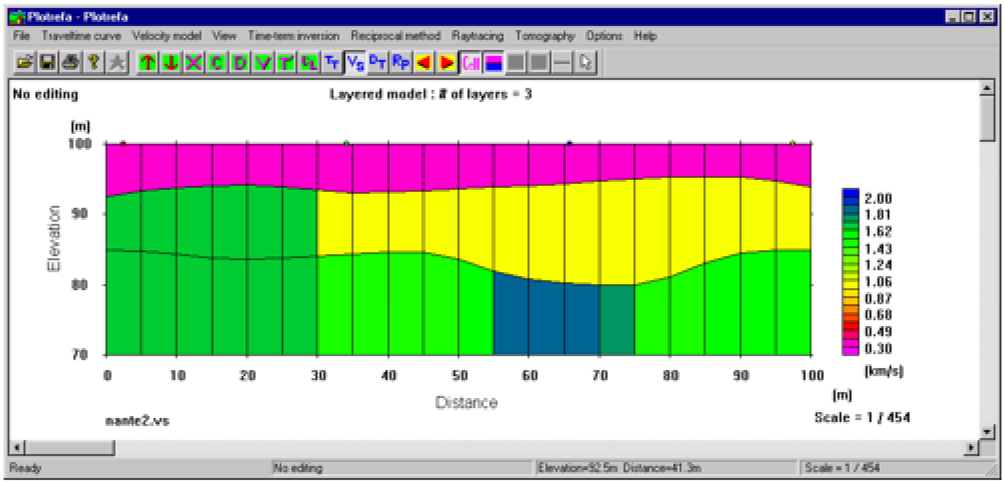


Modifying the topography is accomplished in the same manner as modifying velocity boundaries. However, you must enable this capability first. Click on *Enable surface topography modification*, then modify the surface as described in Sections 0, 4.3.15, and 0, starting on Page 204.

4.3.21 SMOOTH



The layer boundaries and velocity transitions can be smoothed. To smooth layer boundaries, choose *Layer boundaries* from the sub-menu:



To smooth out horizontal velocity changes, choose *Velocity (horizontally)*:

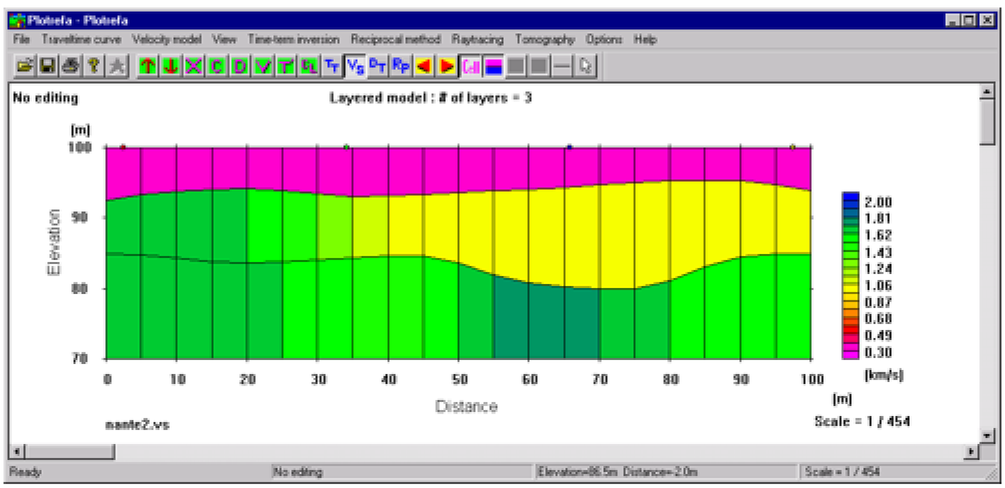


Figure 88: Horizontal velocity smoothing.

To smooth out vertical velocity changes, choose *Velocity (vertically)*:

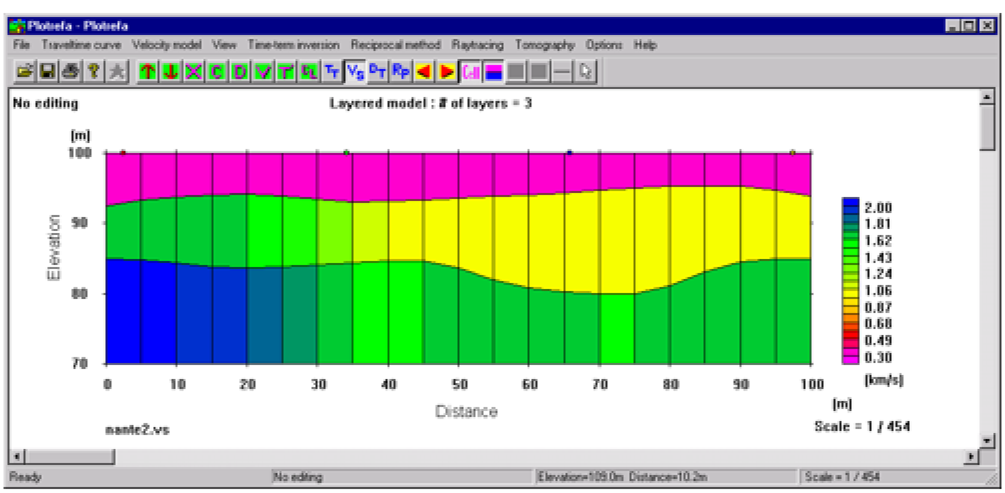


Figure 89: Vertical velocity smoothing.

With all three of the above smoothing operations, each time you click, a little more smoothing occurs. For instance, in the above model, the layers were smoothed twice. In the one below, it has been smoothed five times:

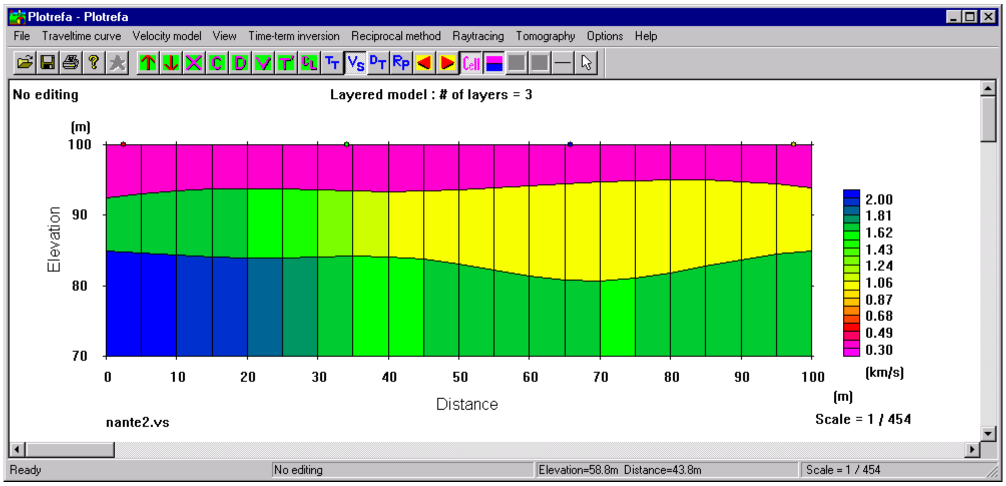
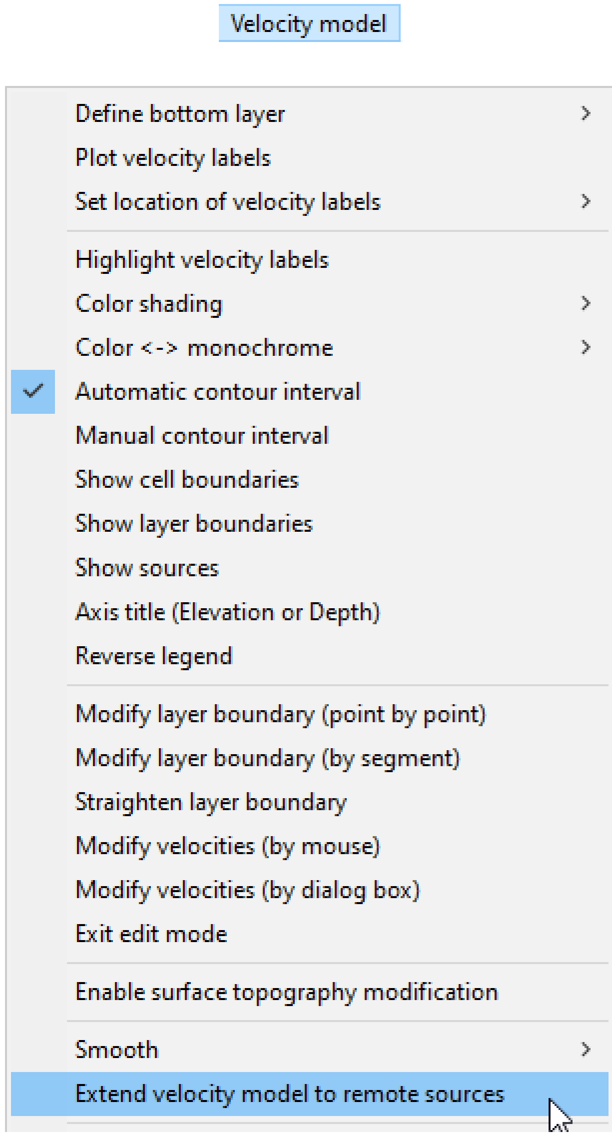


Figure 90: Smoothed velocity model.

4.3.22 Extend Velocity Model to Remote Sources

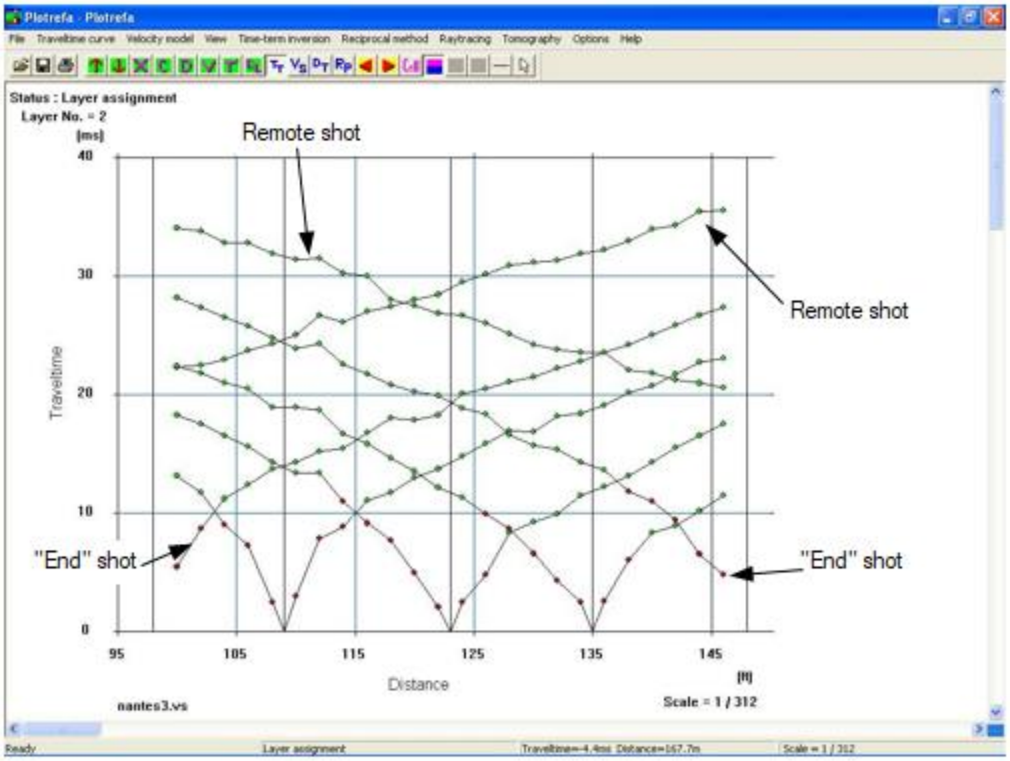


When using remote or “offset” shots, it is sometimes necessary to manually extend the velocity model to include them. If the velocity model is not extended to include the remote shots, the data from those shots cannot be used in the final inversion.

When the topography is *not* imported, the model will automatically extend to the remote sources, and they will be included in the inversion. However, ***if the topography was imported from an elevation file, the remote sources will only be included if their elevations are included in the elevation file.*** Since the elevations (and coordinates) of remote sources are not necessary parameters to record, they are often not measured. If they are not, you will have to extend the model manually. This is best illustrated by an example.

Below are the travel time data and the corresponding time-term inversion from a site with significant topography. Note that in addition to shots at the ends and within the spread, there is a shot well off either end ("remote shots").

Note: *SeisImager/2D treats any shot outside of the spread, even the two "end shots" (see below) as remote sources. See Note on Page* *163*.



The elevation file did not include the elevations of the end shots or the remote shots, and as a result, the data from these shots were not included in the time-term inversion:

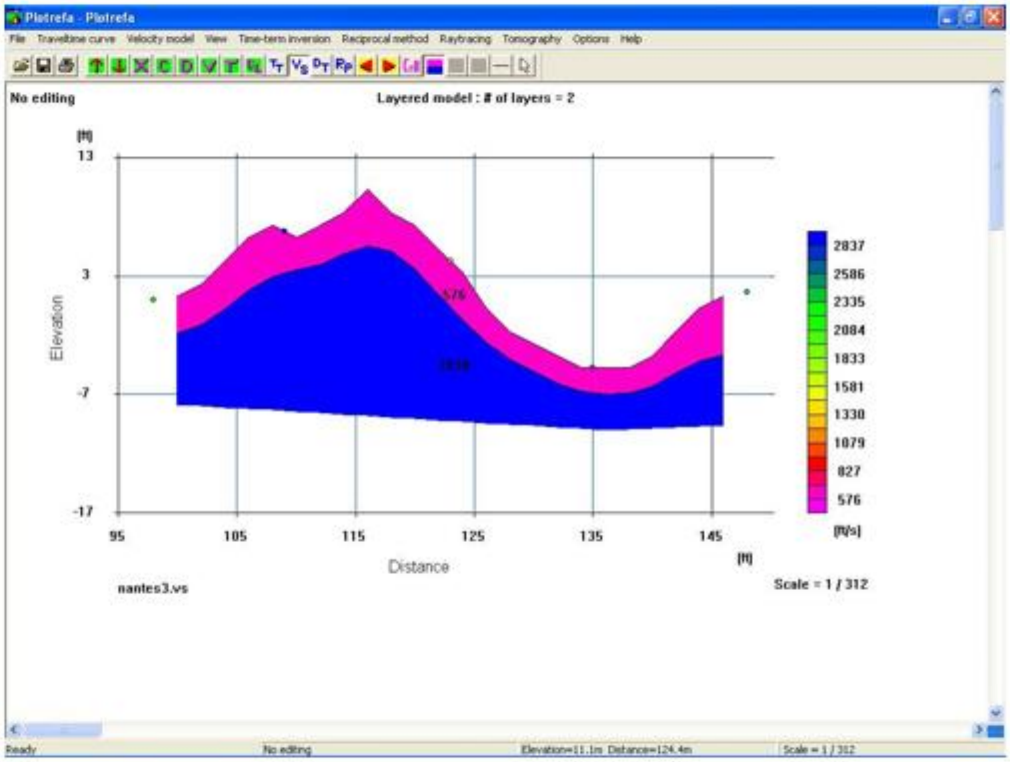
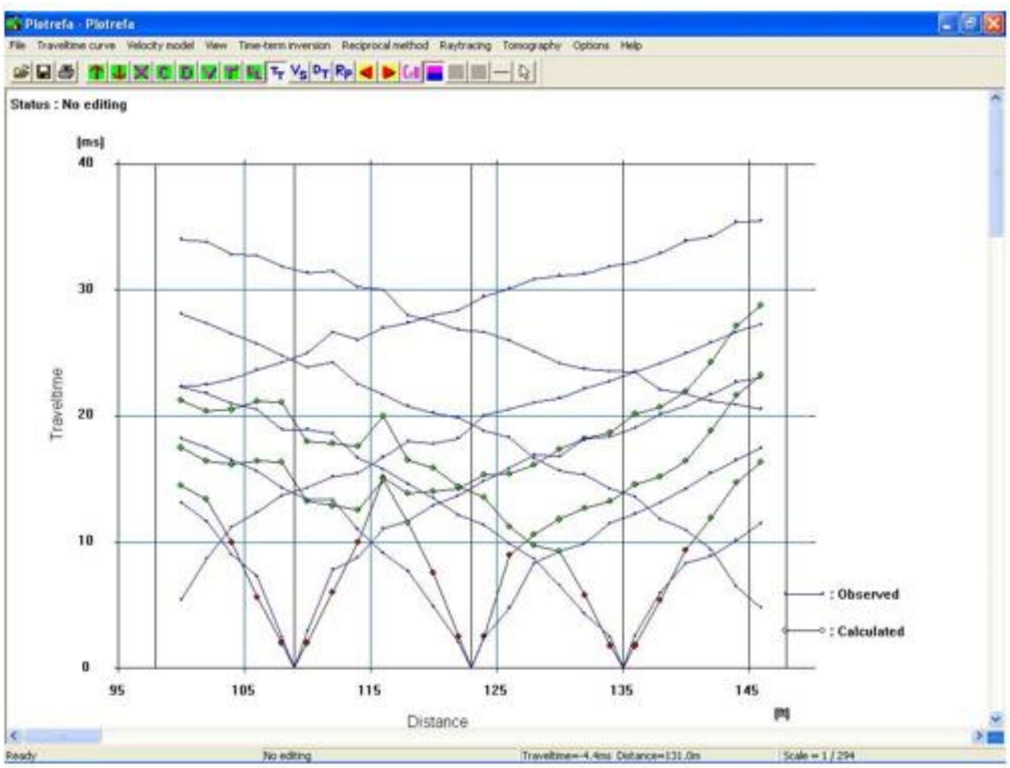


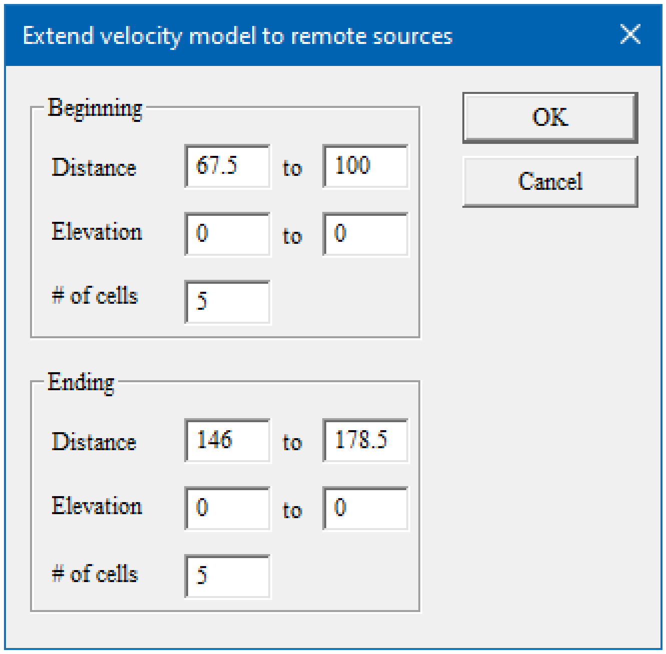
Figure 91: Time-term inversion. Data from "end shots" and "remote shots" have been ignored.

This can be demonstrated by running the raytracing routine through the above velocity model:



Note that no theoretical travel times have been computed for the end or remote sources. We must extend the velocity model to include them.

To extend the velocity model, choose *Extend velocity model to remote sources* to reveal the following dialog box:



The distance values will default to the locations of the farthest remote shots. Press *OK*, and the model will be extended to include them:

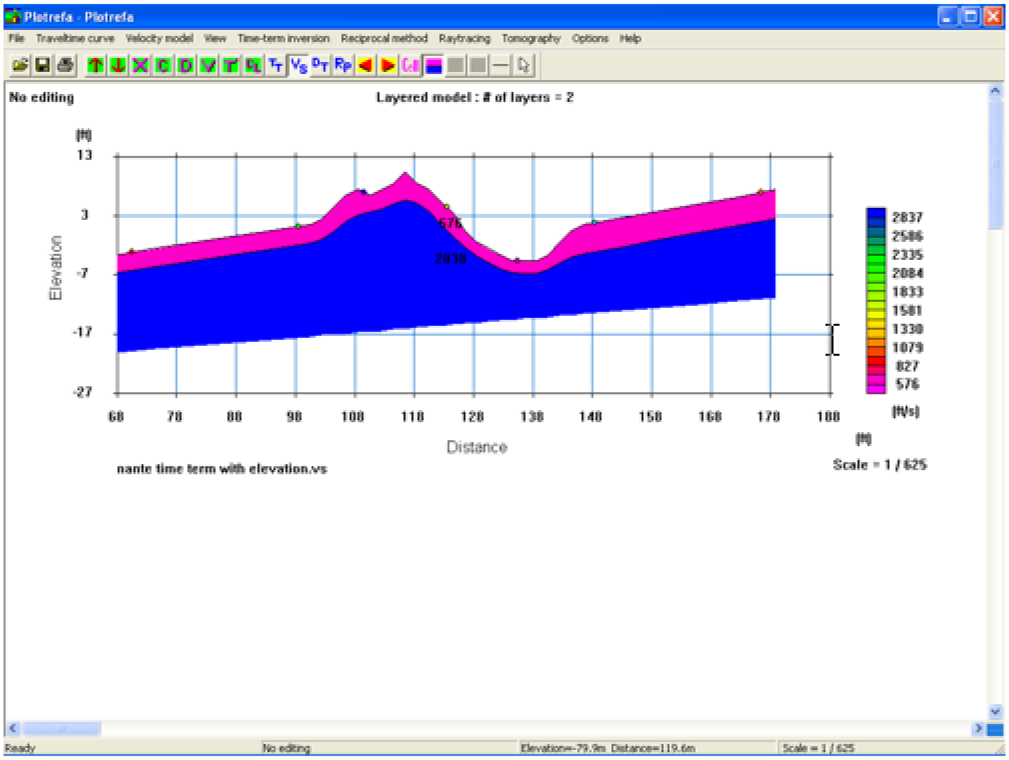


Figure 92: Extending velocity model to remote sources.

At this point you may refine the model as usual using the tomography module, and the remote sources will be included in the analysis. In this particular case, since the topography is significant, a tomographic analysis is the best approach. We use the above model as the initial model and invert:

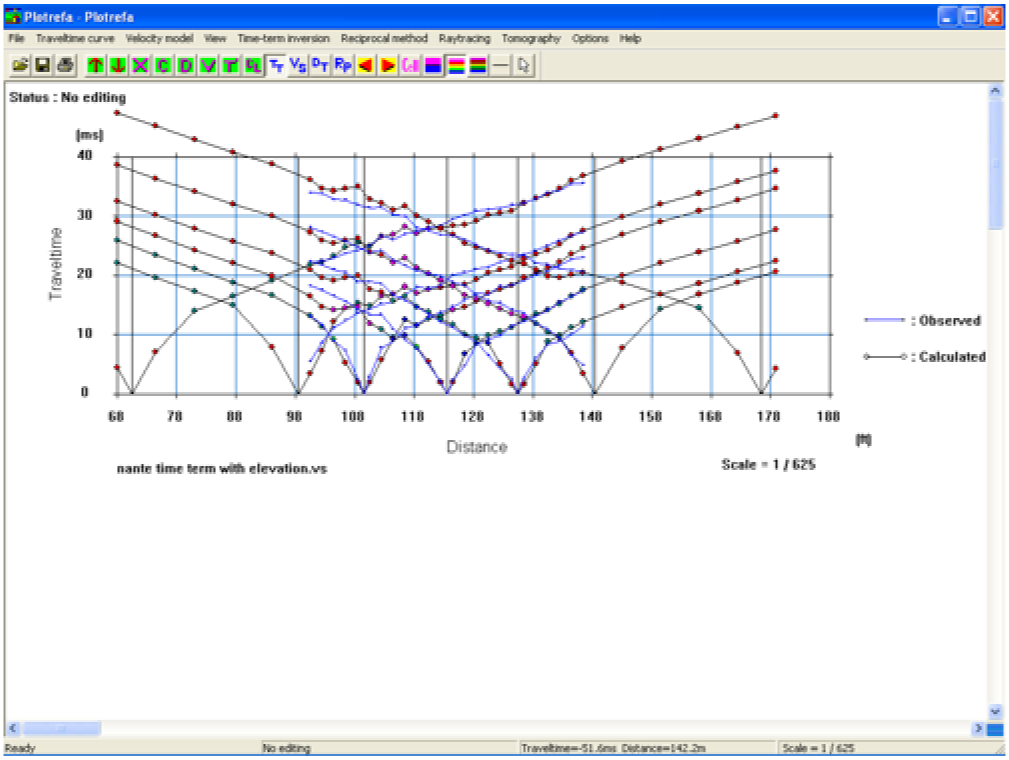


Figure 93: Travel time plot showing theoretical travel times for all sources, including remote sources.

Note now that theoretical travel time data have been calculated for all sources, including remote ones. The theoretical travel times outside of the geophone spread are calculated from extrapolated velocities and should be ignored.

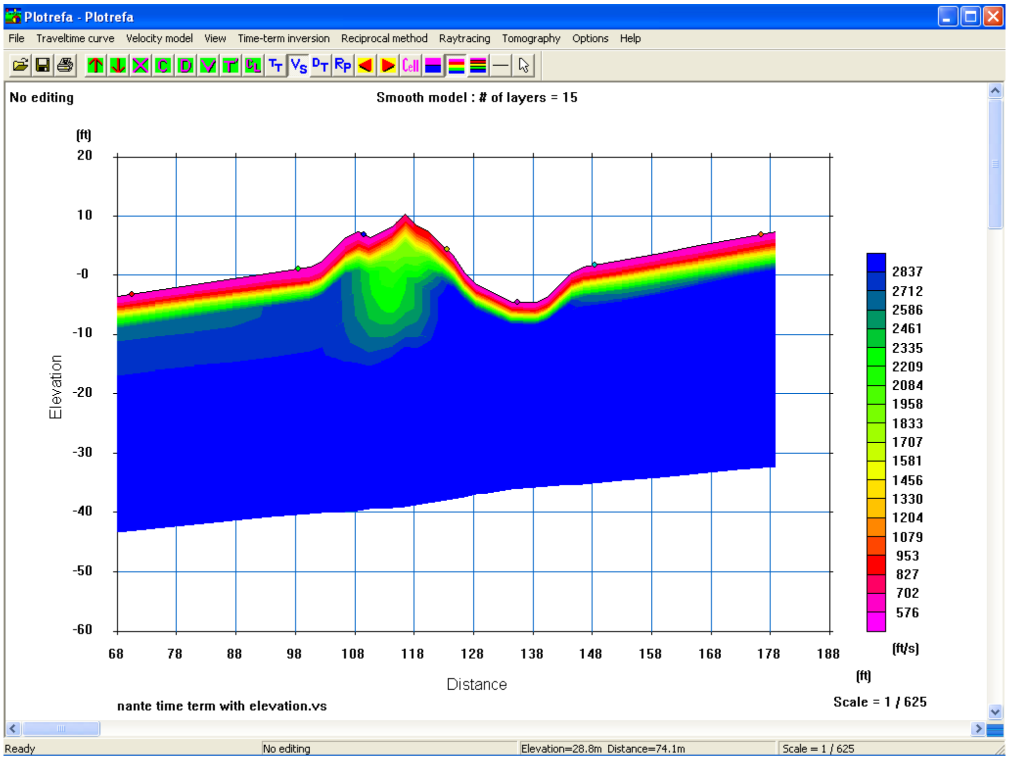


Figure 94: Velocity model extrapolated out to remote sources.

At this point, you can trim down the result to show only the zone within the geophone spread:

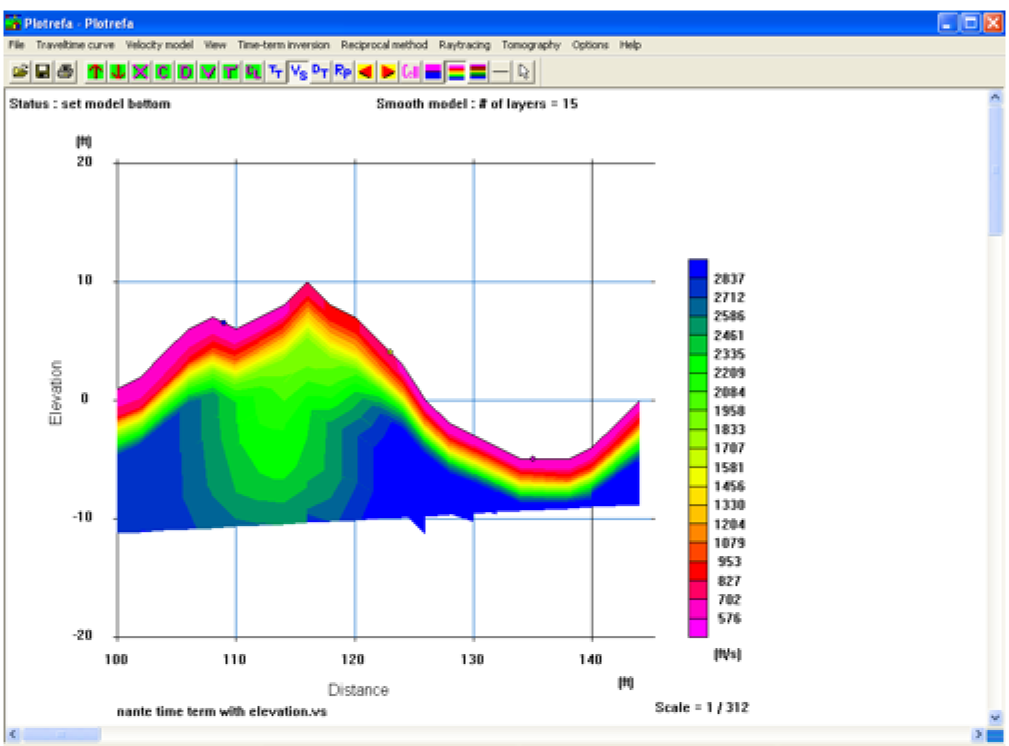
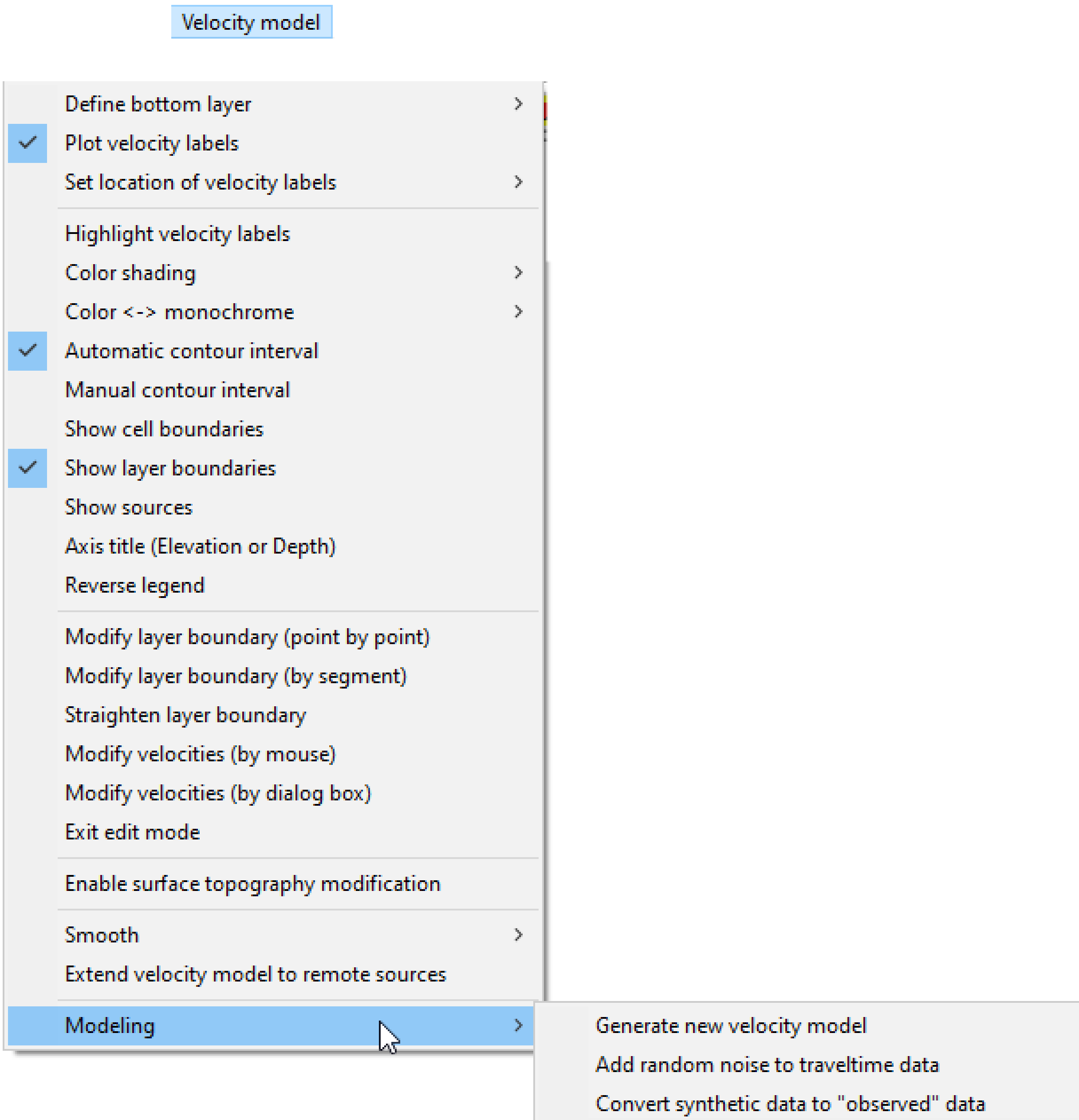


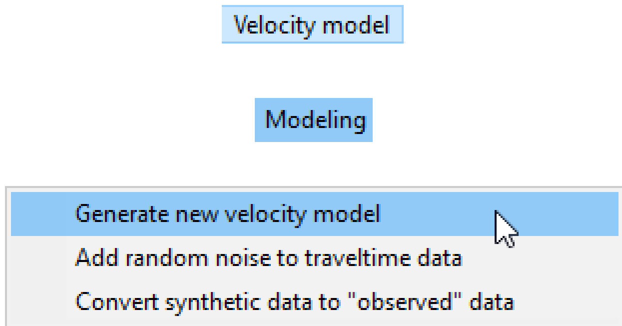
Figure 95: Trimmed result.

4.3.23 MODELING

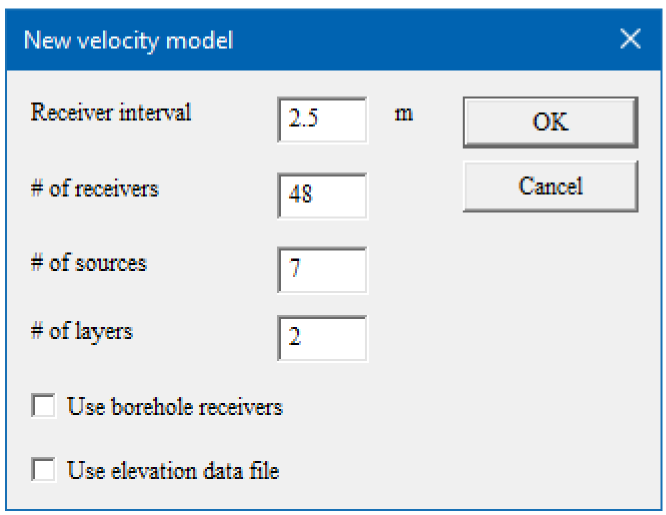


Plotrefa includes the capability of creating a custom velocity model for forward modeling purposes. You may create a simple layer-cake initial model, and then customize it further using the editing techniques discussed above. Once you have completed your model, you may use the raytracing routine (discussed in Section 4.7, Page 270) to compute theoretical travel times for the model.

4.3.23.1 Generate New Velocity Model



To make a new synthetic velocity model, choose *Generate new velocity model* to reveal the following dialog box:



Specify the necessary parameters, press *OK*, and a velocity model will be created with default velocities:

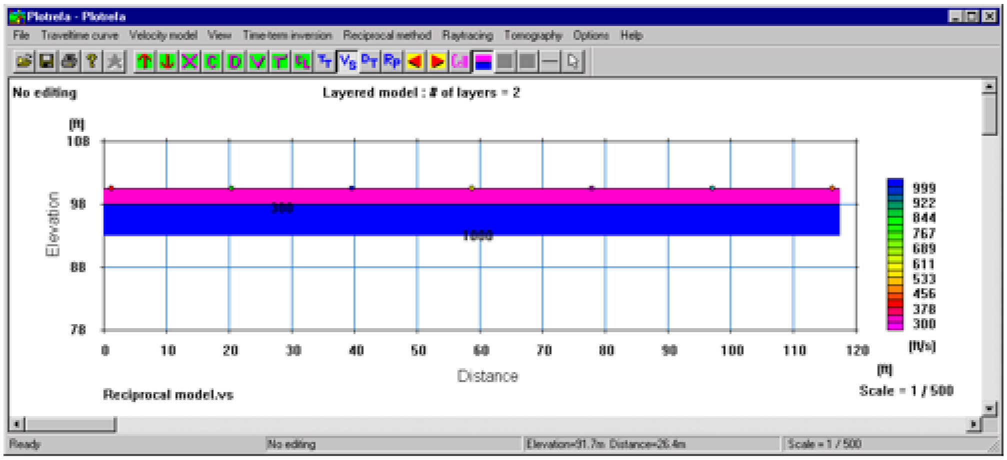


Figure 96: New velocity model.

You may now customize the model as needed using the tools in the **Velocity model** menu.

Below is a customized version of the initial model:

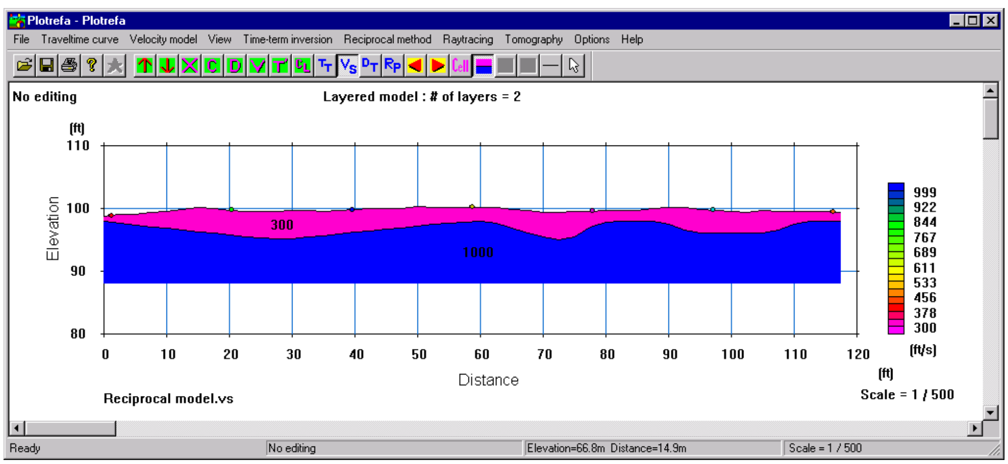


Figure 97: Customized velocity model.

If you calculate synthetic travel times using the **Raytracing** menu, you may add random noise to them. Below are the synthetic travel times generated for the model above:

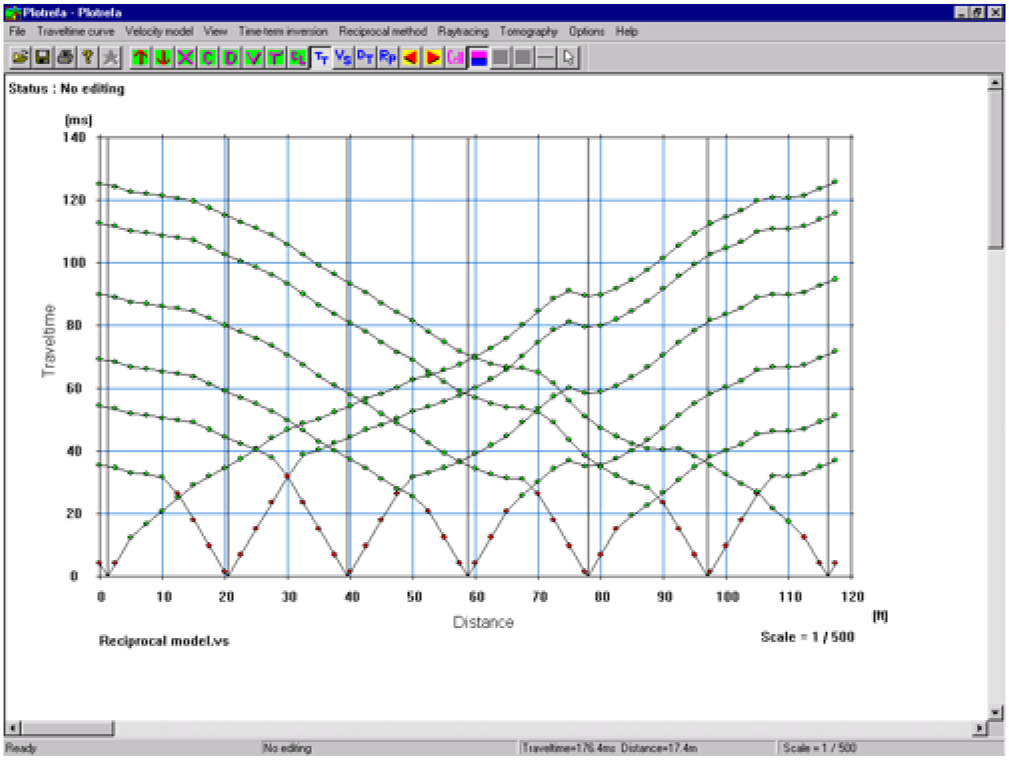
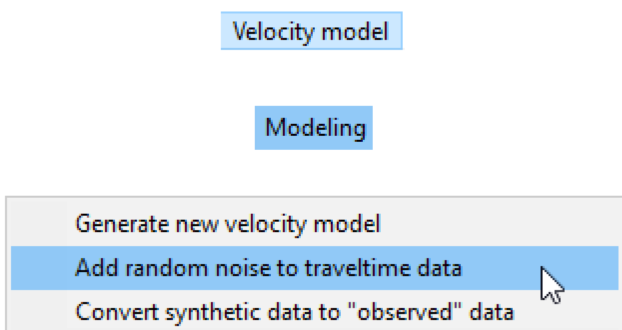
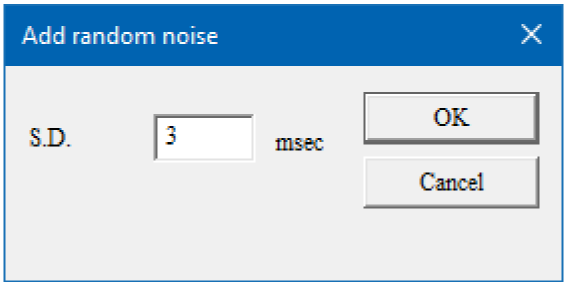


Figure 98: Travel time plot generated for synthetic velocity model.

4.3.23.2 Add Random Noise to Travel time Data



Choose *Add random noise to travel time data* to reveal the following dialog box:



Indicate the [standard deviation] noise range in milliseconds, and press *OK*:

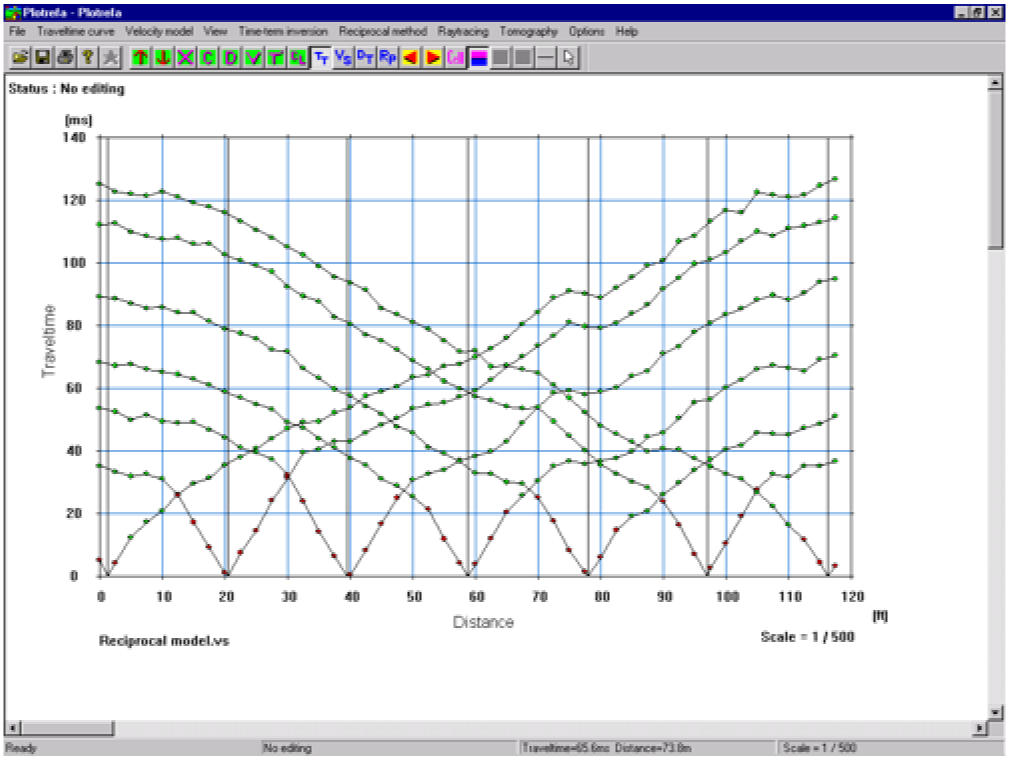
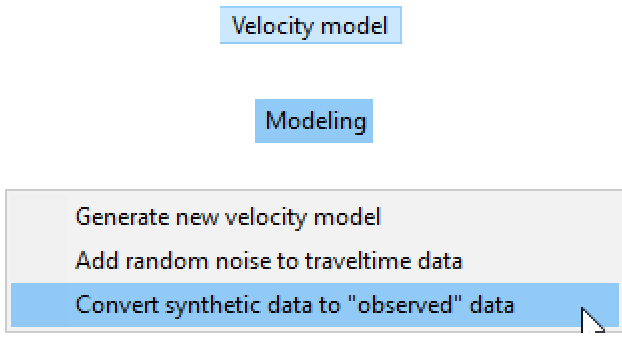


Figure 99: Random noise added to synthetic travel times.

Your data will now have a random noise component superimposed on it.

4.3.23.3 CONVERT SYNTHETIC DATA TO "OBSERVED" DATA



It is often useful to convert synthetic travel time data calculated from a synthetic model into "observed" data. This basically tricks the program into thinking that the synthetic data is real data, allowing you to treat it as such. This is a necessary step if you wish to invert this synthetic data and compare the resulting model to the original input model. This forward/inverse modeling can be very useful in testing the capabilities of the various inversion techniques on various types of seismic models. For instance, if you wanted to test the fault-detecting ability of tomography, you might take the following steps:

- Create a faulted velocity model.
- Use the **Raytracing** menu to calculate the synthetic travel times.
- Add a reasonable level of random noise to your data.
- Convert the synthetic data into "real" data.
- Do a tomographic inversion of the "real" data.
- Compare the initial model to the calculated model.

The new synthetic data will now be displayed along with the "observed" data (below).

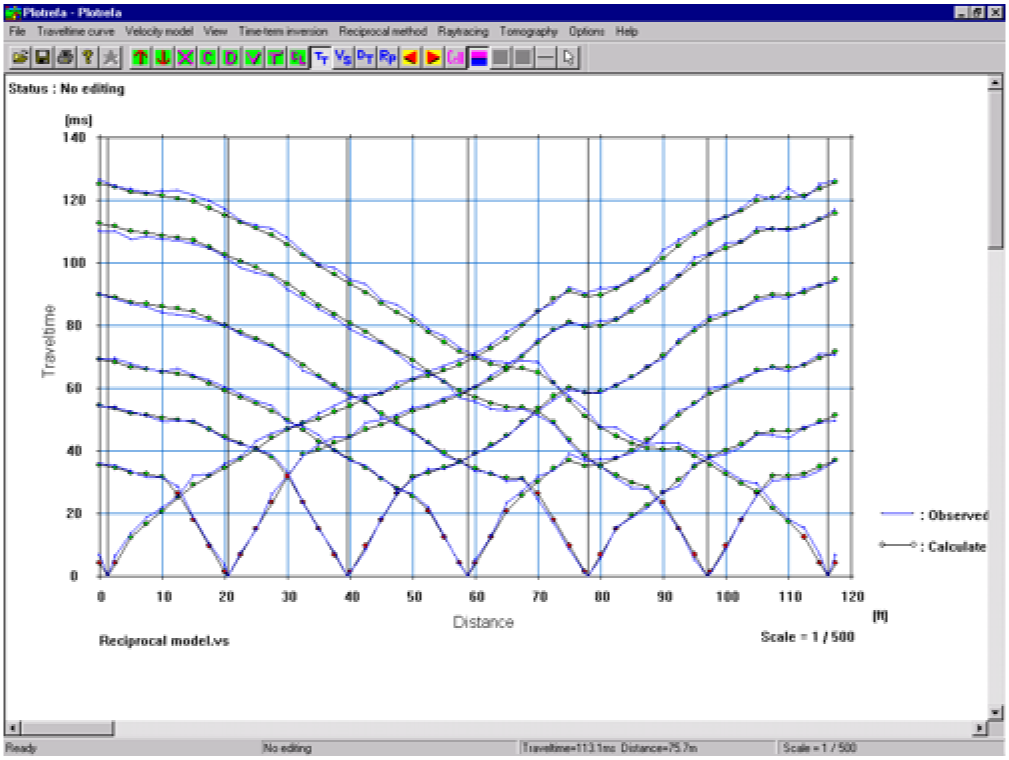
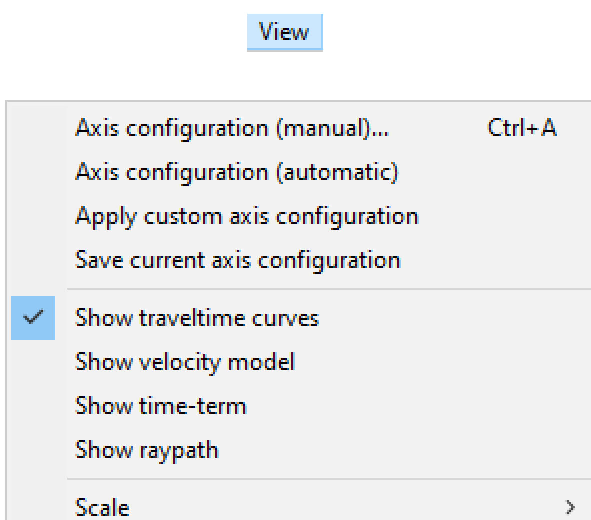


Figure 100: Synthetic and "observed" data.

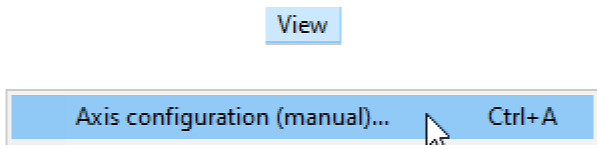
4.4 VIEW MENU

Click on *View* to display the **View** menu:

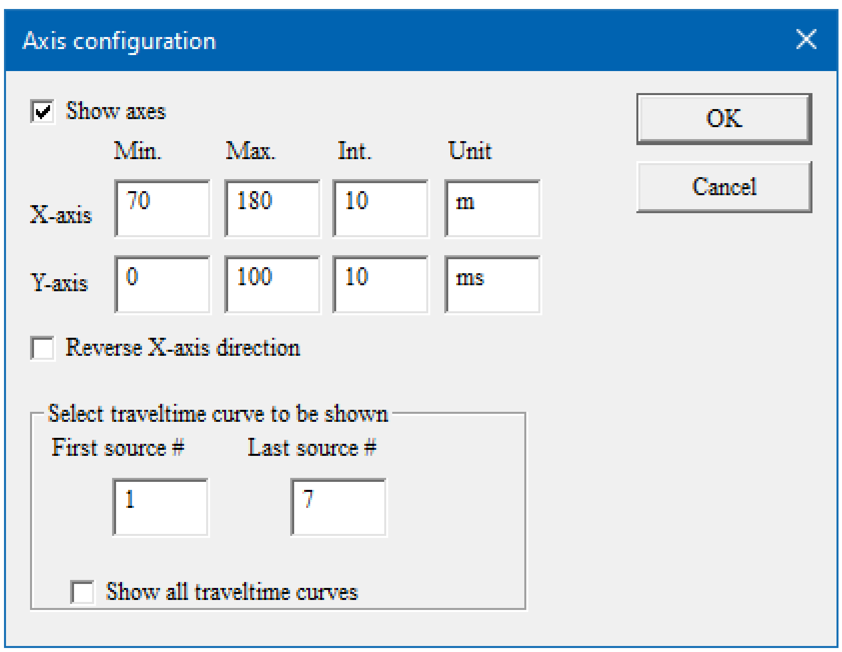


This menu gives you control over various display parameters. It allows you to configure the axes, determine which graphics to view, and set the scale.

4.4.1 AXIS CONFIGURATION (MANUAL) [CTRL+A]



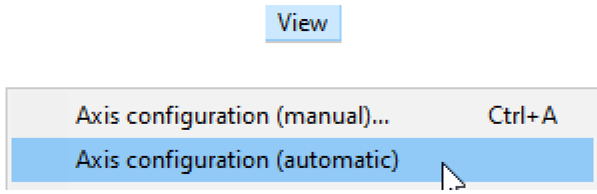
You may customize the axes of the travel time plot and the velocity section. Click on *Axis configuration (manual)* or press *Ctrl+A* to reveal the following dialog box:



The X-axis will always be in units of length (set in the main **Options** menu, discussed beginning in Section 4.9.2, Page 284); the units of the Y-axis will depend on what is being displayed when you choose to customize the axes. If the velocity section is displayed, the Y-axis will be in units of length. If the travel time plot is displayed, the Y-axis will be in units of time.

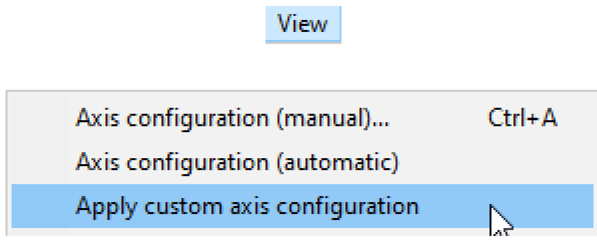
You may also control the number of travel time curves that are displayed. Experiment with the various parameters to see their effects.

4.4.2 AXIS CONFIGURATION (AUTOMATIC)

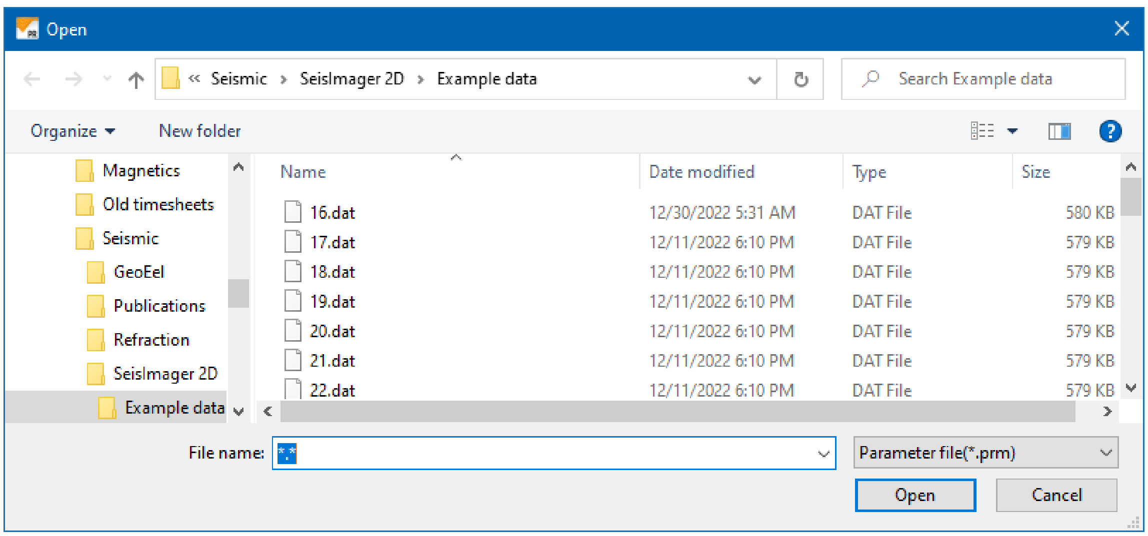


If you would like the axes to be configured automatically, choose *Axis configuration (automatic)*.

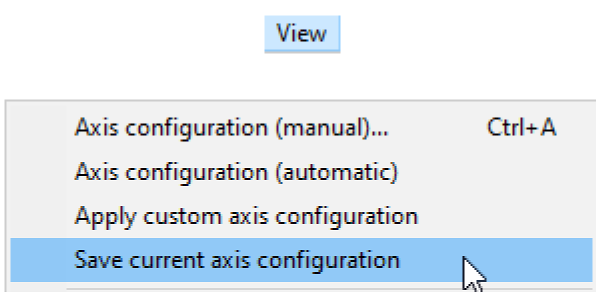
4.4.3 Apply Custom Axis Configuration



If you have created a custom axis file (see next section), you may apply that configuration to the current plot. Click on *Apply custom axis configuration*. Choose the appropriate .prm file, and the plot will be configured accordingly.

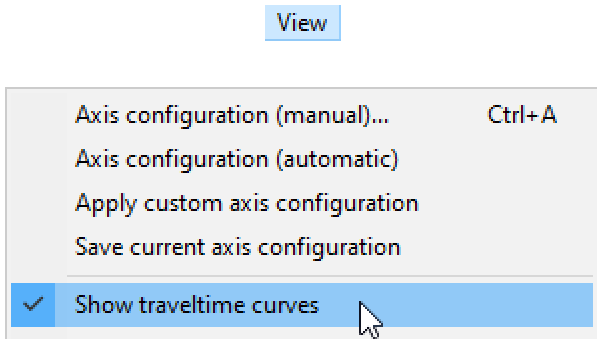


4.4.4 Save Current Axis Configuration



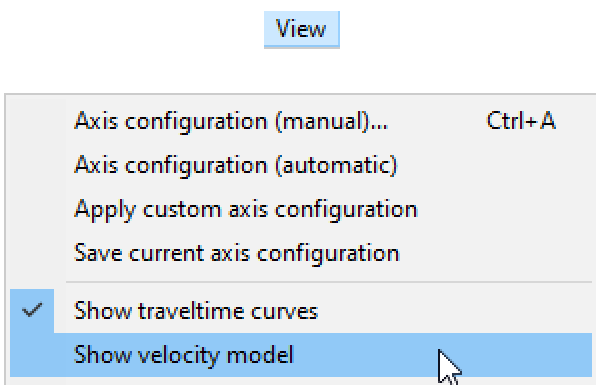
If you have created a custom axis configuration manually, you may save this configuration for use on subsequent models. Click on *Save current axis configuration*. You will be presented with a dialog box like that shown above. Provide a filename and press *Save*.

4.4.5 Show Travel time Curves [ ]



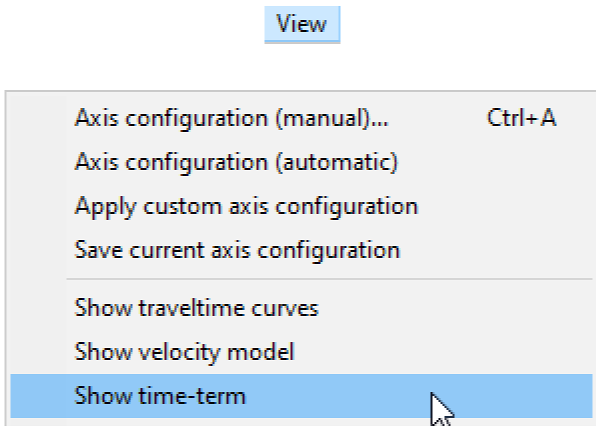
If you wish to view the travel time plot, click on *Show travel time curves*, or press the tool button.

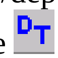
4.4.6 Show Velocity Model [ ]



If you wish to view the velocity model, click on *Show velocity model*, or press the tool button.

4.4.7 SHOW TIME-TERM []



As will be discussed in future sections, the time-term and Reciprocal Methods modules are based on the concept of "time-terms" or "delay-times". The calculated delay-times are used in conjunction with the associated velocities to generate the velocity/depth section. If you would like to view the delay-times, click on *Show time-term*, or press the  tool button. The delay-times will be presented in a plot like the one shown below:

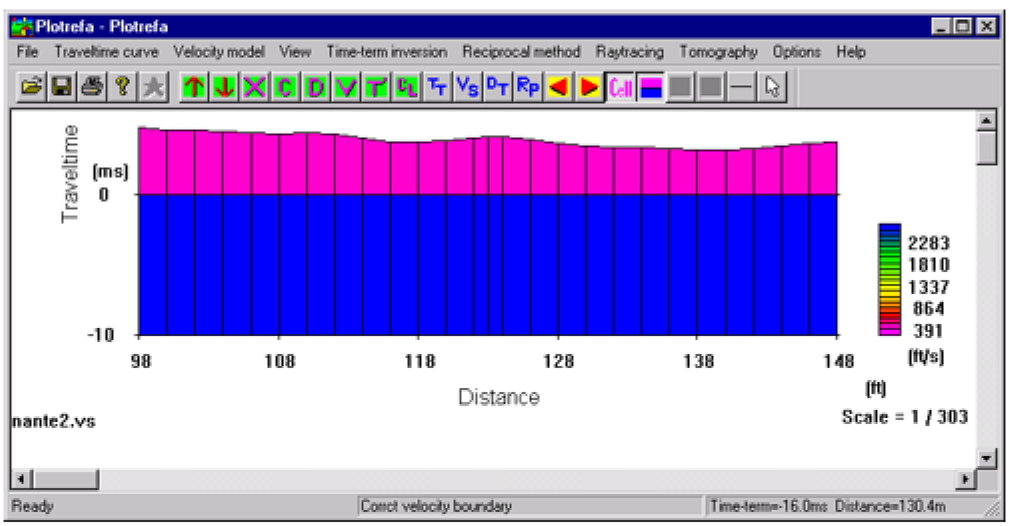
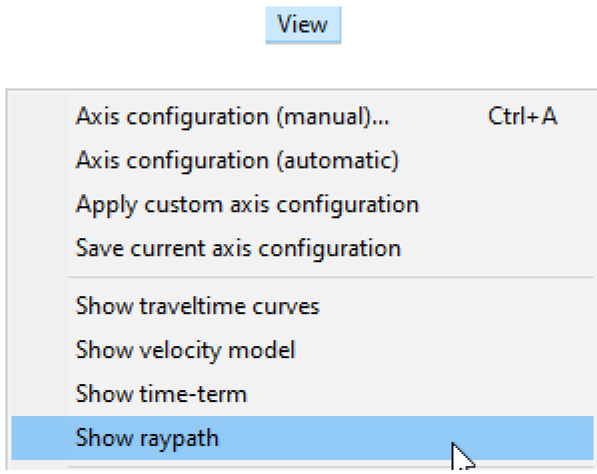


Figure 101: Time-term (delay-time) plot.

4.4.8 Show Raypath [ ]



If you have run your model through the raytracing routine (discussed in Section 4.7, Page 270), and you would like to view the raypaths, click on *Show Raypath* or press the tool button. A raypath diagram like that shown below will be displayed:

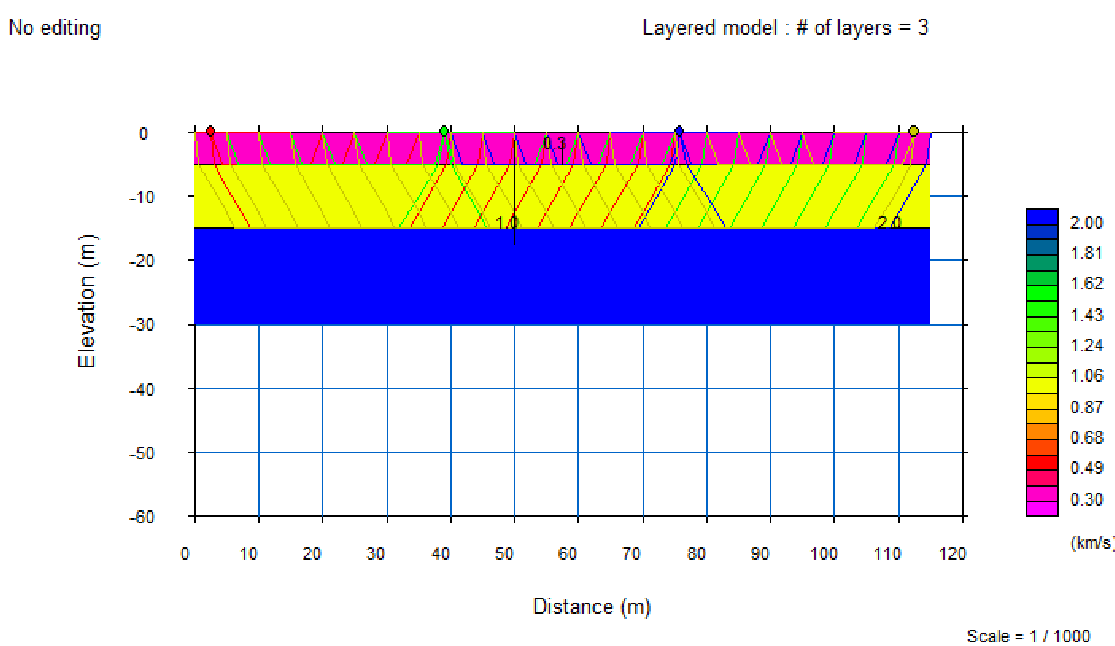
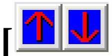
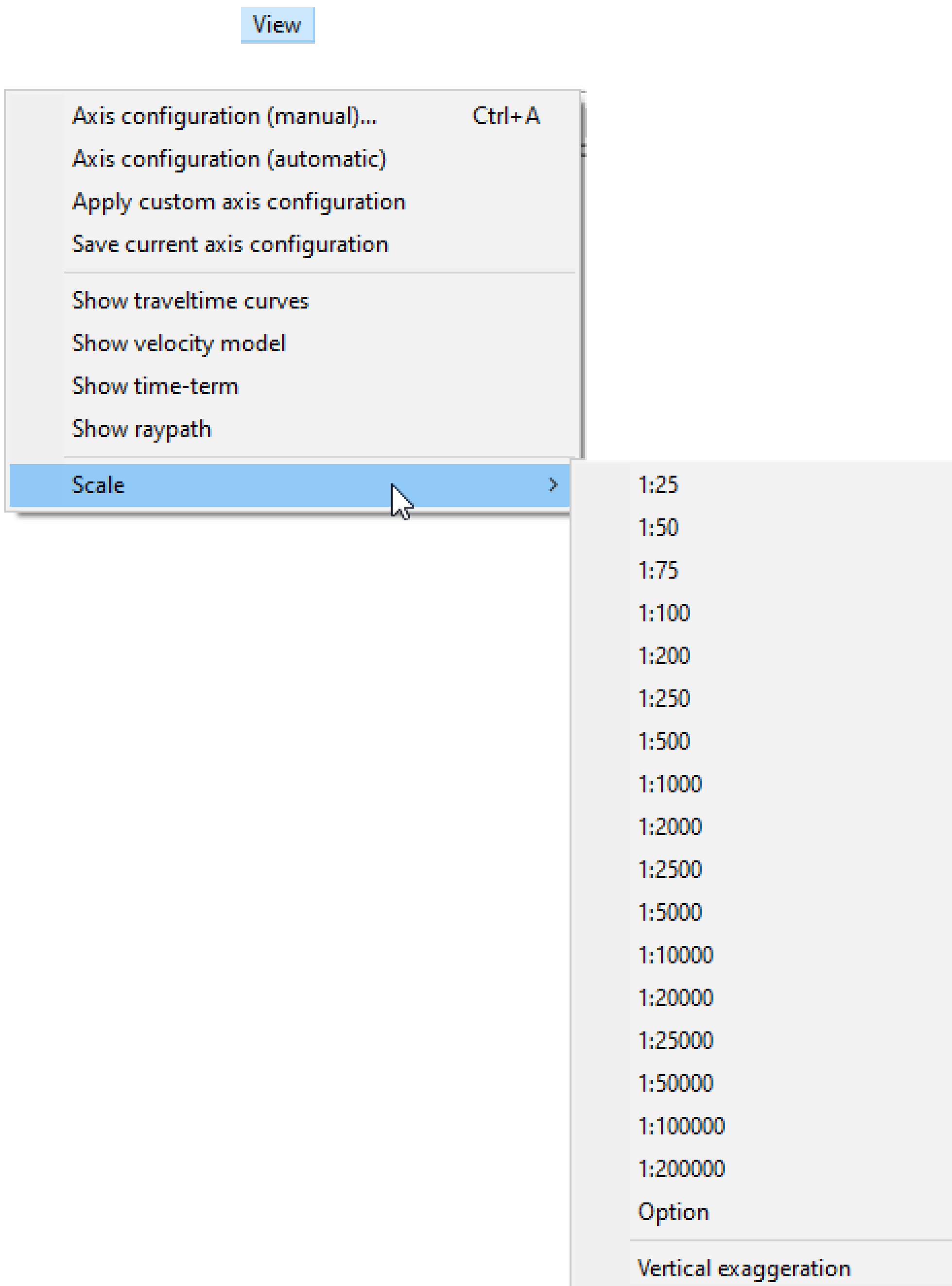


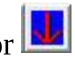
Figure 102: Raypath diagram.

4.4.9 SCALE []

There are several ways to set the scale of your output. If you click on *Scale*, you will see the following sub-menu:



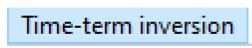
Choose from any one of the scales listed or click on *Option* and enter whatever scale you want.

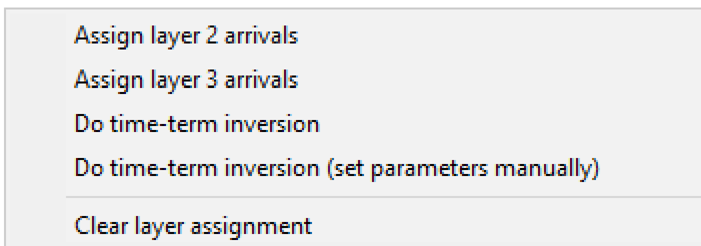
Alternatively, you may increase/decrease the scale by pressing the  or  tool buttons, respectively.

If you would like to change the aspect ratio, click on *Vertical exaggeration* and enter the desired ratio.

4.5 Time-term Inversion Menu

Click on *Time-term inversion* to reveal the **Time-term Inversion** menu:





We will now discuss the first of three inversion techniques. Which technique you employ will depend on the goals of your survey. In a fashion consistent with what we have done so far, each menu item for each technique will be discussed individually. Examples of each of the three interpretation techniques are given in the appendices.

The "Time-term" technique employs a combination of linear least-squares and delay-time analysis to invert the first arrivals for a velocity section. It is a good approach for lower-budget, simple refraction surveys, in which refractor detail is of lesser importance than gross velocities and depths. A good example might be a rippability survey. These types of surveys are typified by 12 or 24 channels, with as few as two shots per spread. The answer usually does not need to be a detailed one, and minimizing the time between fieldwork and the deliverable to the client tends to trump all.

A brief explanation of the time-term technique is given in Appendix D, Page D-1.

The general flow of the time-term technique is displayed in the flow chart below:

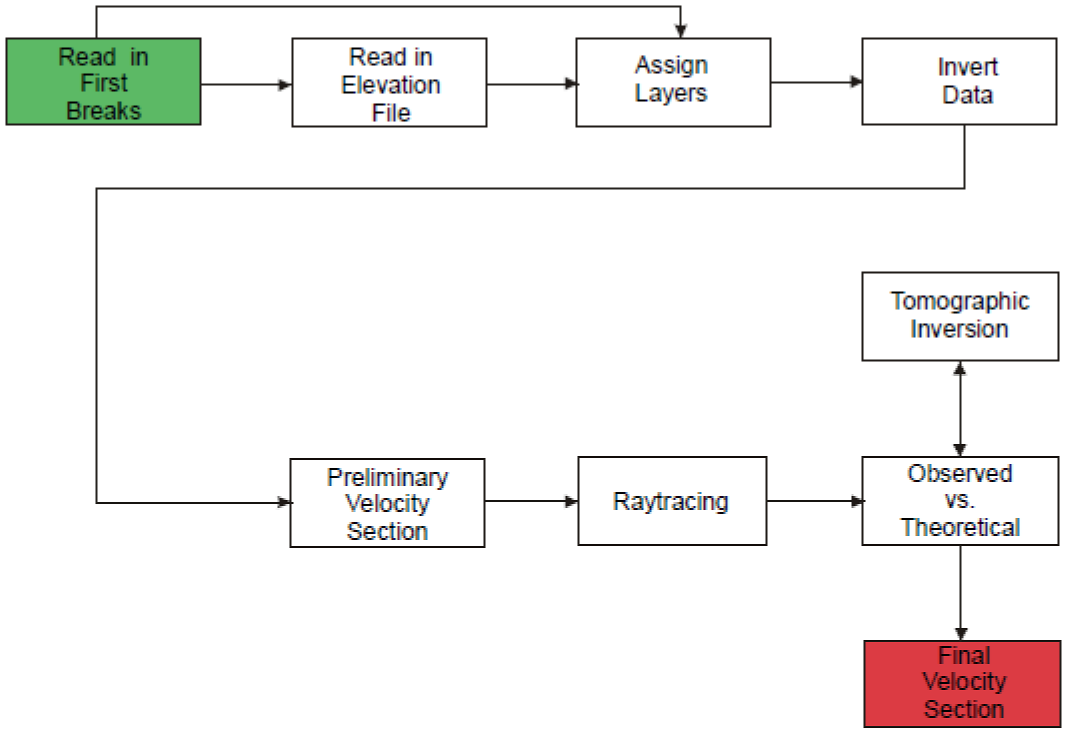
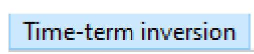
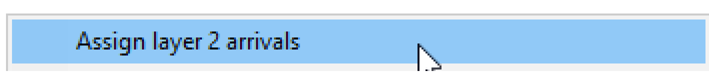


Figure 103: Flow chart for time-term interpretation technique.

4.5.1 ASSIGN LAYER 2 ARRIVALS





A simple time-term analysis allows a two- or three-layer interpretation. If you have a three-layer case, you should assign arrivals for the second layer first.

Read in your first breaks (.vs file) and click on *Assign layer 2 arrivals.* To assign layers, click on the travel time closest to the change in slope associated with the second layer. In the figure below, the cursor is pointing to the first travel time from the second layer for the left-most shot. Note that all subsequent travel times for that shot are now shown in green:

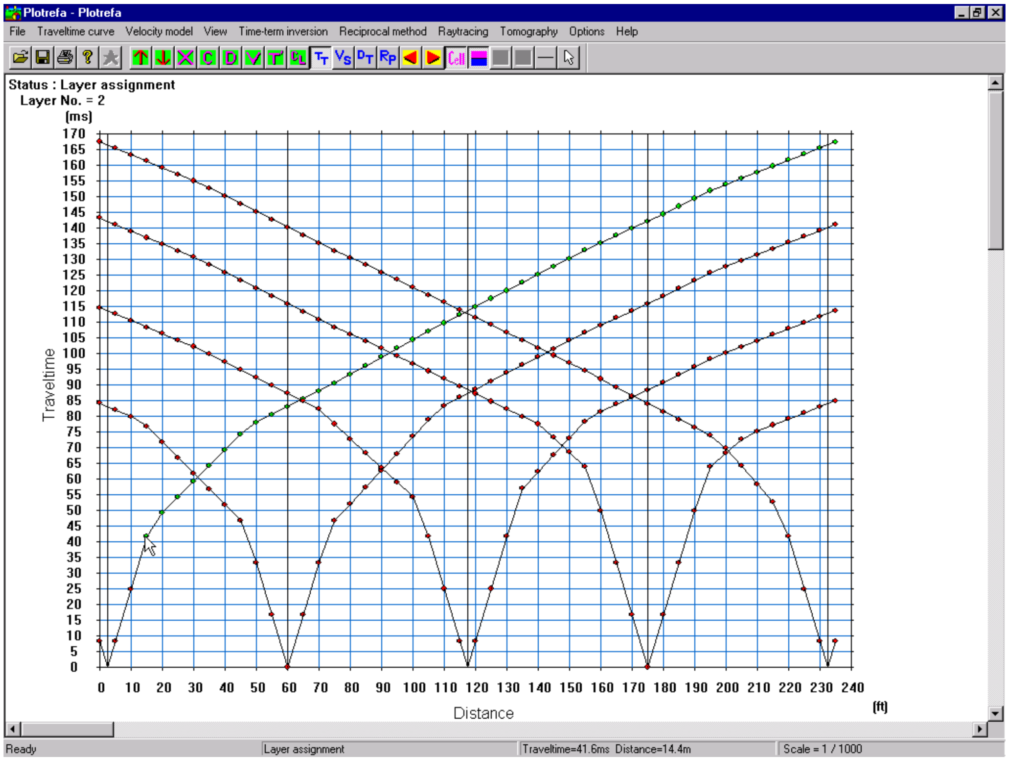


Figure 104: Assigning layer two arrivals.

Assign layer two to all shots:

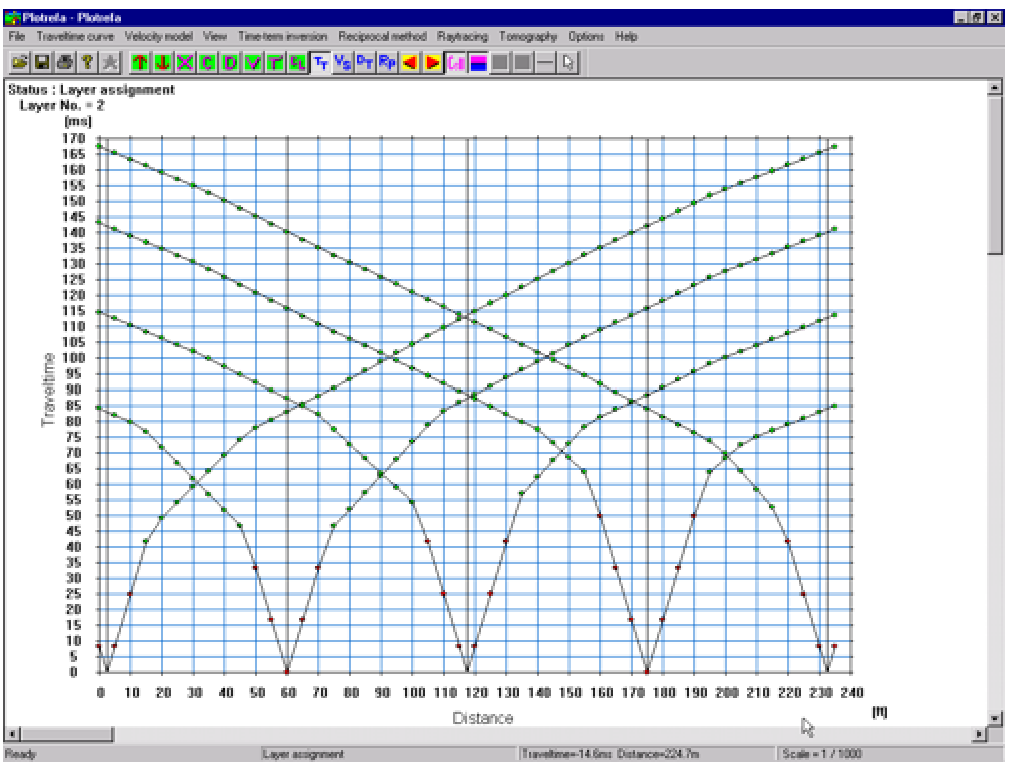
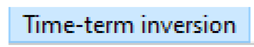
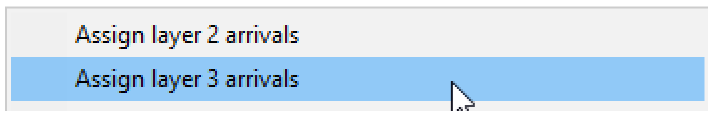


Figure 105: Layer two arrivals assigned.

If it is a two-layer case, you are finished assigning arrivals.

4.5.2 ASSIGN LAYER 3 ARRIVALS





If there is a third layer, you must repeat the process for layer three. Choose *Assign layer 3 arrivals* and follow the procedure detailed above. In the figure below, the cursor is pointing to the first layer-three arrival from the left-most shot:

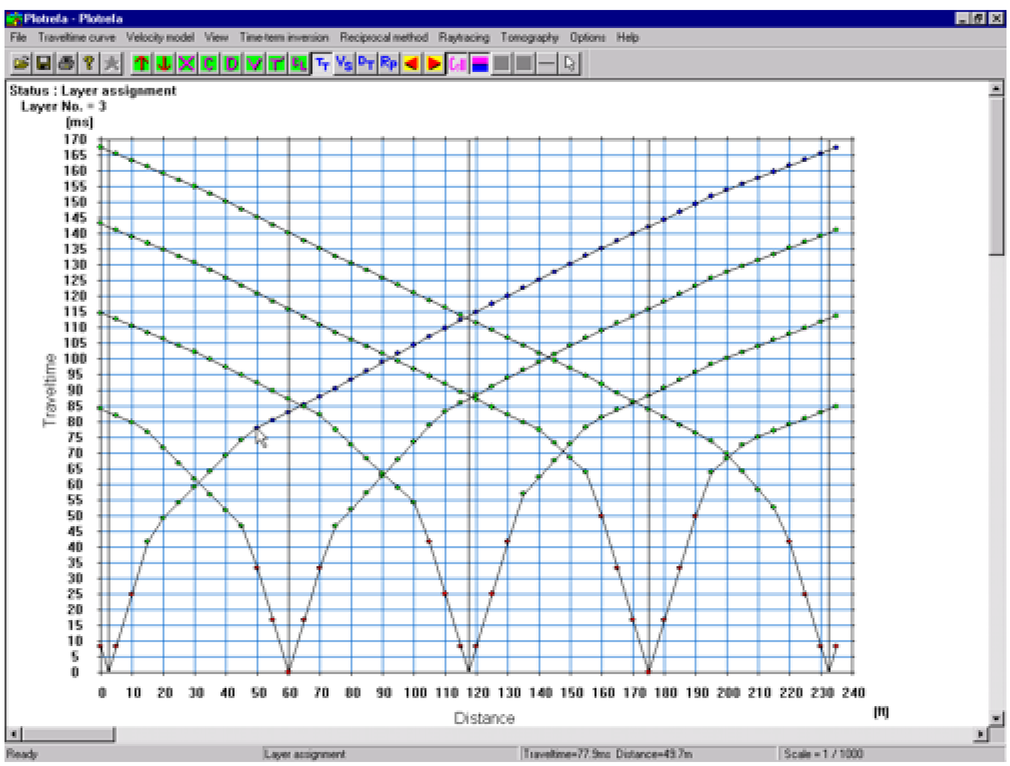


Figure 106: Assigning layer three arrivals.

Below is the full three-layer interpretation:

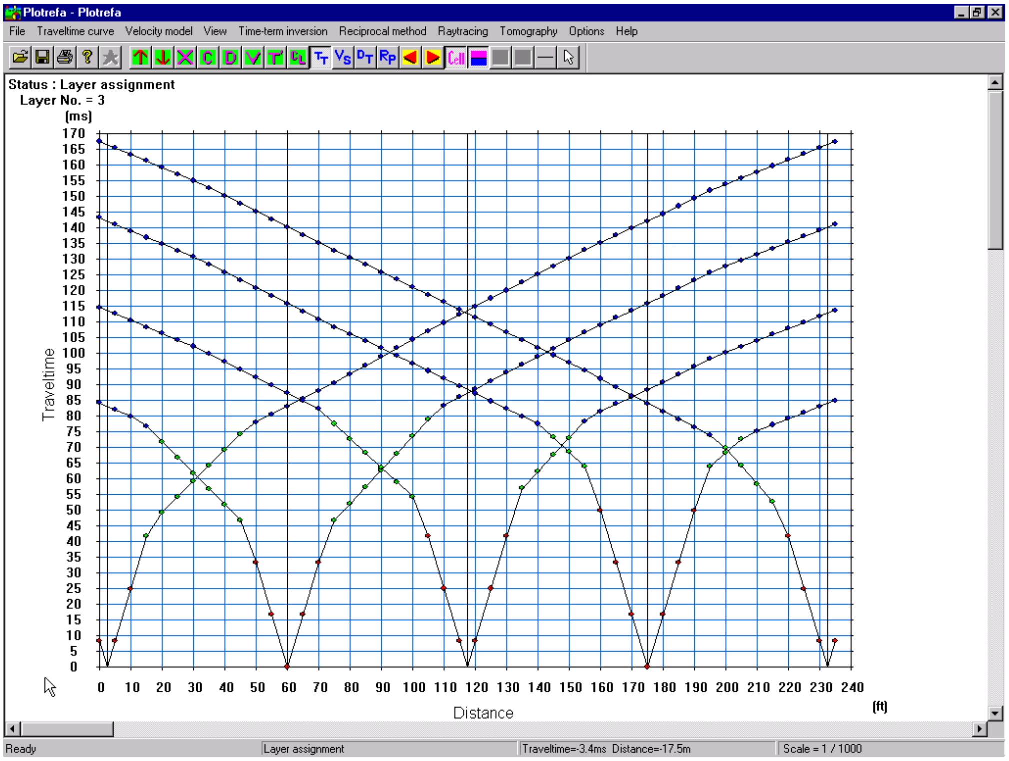
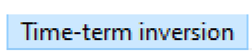


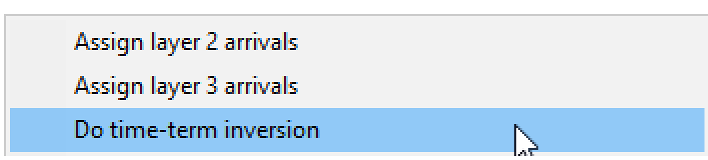
Figure 107: Layer three arrivals assigned.

Note: *When travel times from different shots coincide at a hinge point, it can be difficult to assign layers to both travel time curves. When this happens, the best remedy is to display a partial travel time plot, as discussed in Section* [*4.11.18*](#)*, Page 293.*

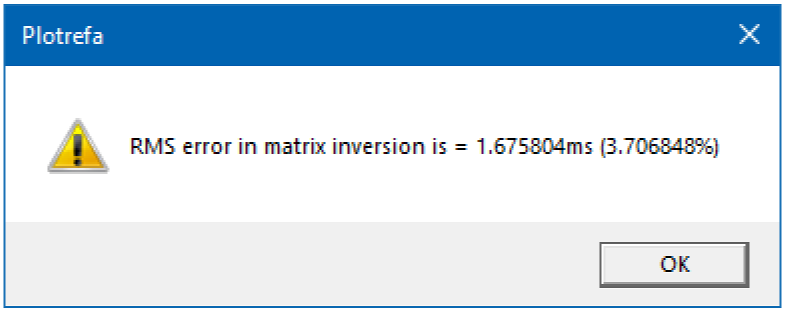
See [here](#) for a video on the layer assignment procedure (be sure to turn up the volume).

4.5.3 Do Time-term Inversion



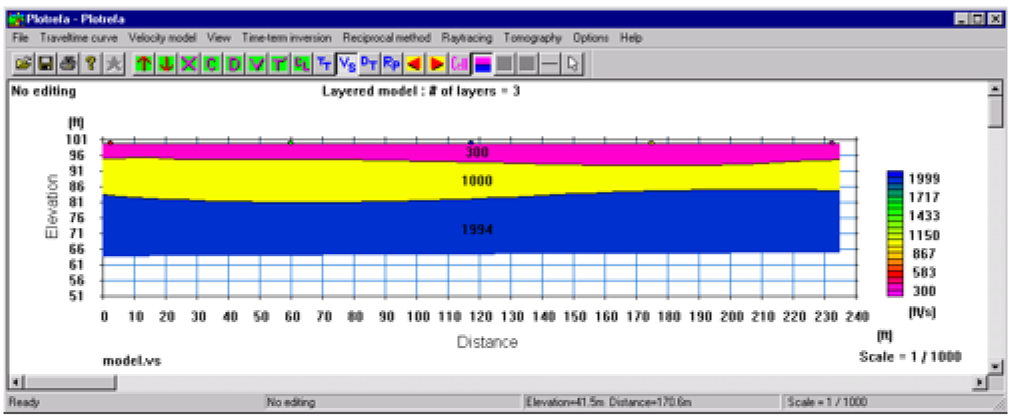


Once all the layers have been assigned, you are ready to invert the data for the velocity section. Click on *Do Time-term inversion*. The inversion RMS error will be displayed:

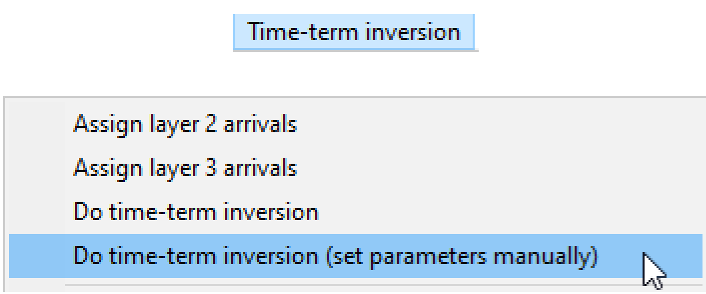


Note: *The message above does* **not** *indicate a failure. It is reported every time you do a time-term inversion. It is simply a measure of the quality of the least-squares inversion. The smaller the number, the better the fit.*

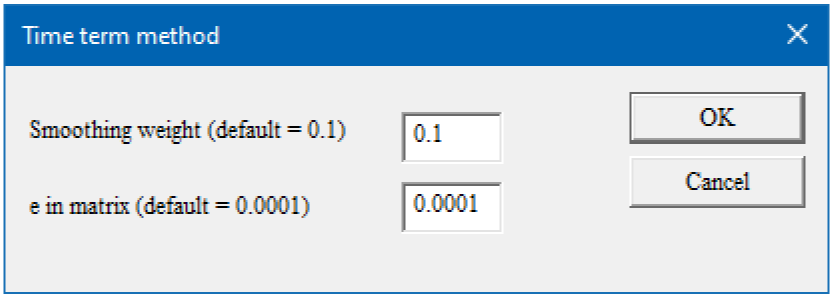
Press *OK* and the velocity section will be revealed:



4.5.4 Do Time-term Inversion (Set Parameters Manually)



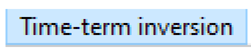
If you wish, you may set some of the time-term parameters yourself.

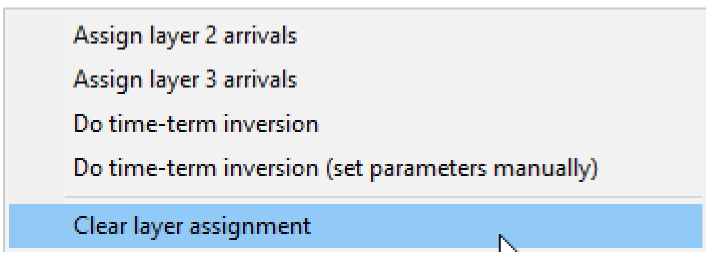


Increasing the *Smoothing weight* will make the velocity boundaries smoother.

Increasing *e in matrix* will make the inversion stable and it will generally result in a smoother velocity model. If you have an inversion failure, you can try increasing the *e in matrix* parameter.

4.5.5 CLEAR LAYER ASSIGNMENT

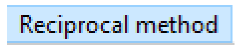


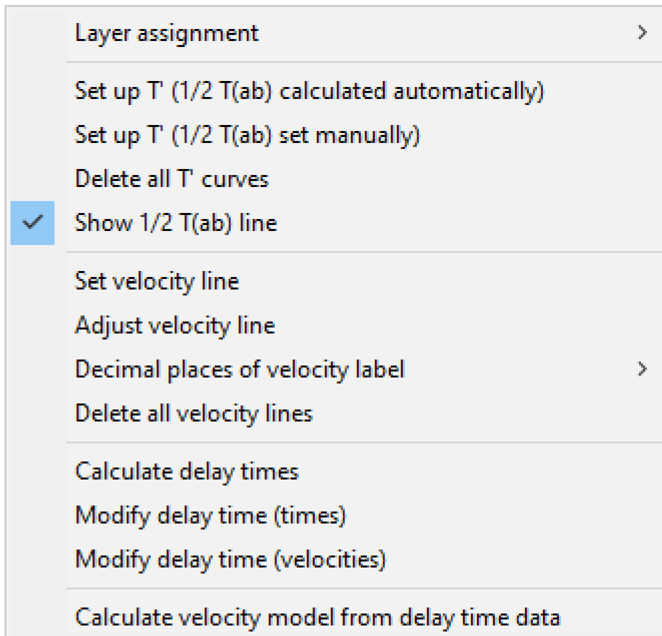


At any time, you may clear the current layer assignments and start over. Simply click on *Clear layer assignment*.

4.6 RECIPROCAL METHOD MENU

Click on *Reciprocal method* to reveal the **Reciprocal Method** menu:





The “Reciprocal Method” of interpretation is a powerful technique for solving more complex refraction problems. It works best with highly redundant data (many shots), 24 channels or more per shot, and requires a far greater degree of input from the interpreter compared with the Time-term method. Like the Time-term method, this technique can provide a refractor depth beneath each geophone, provided a delay-time for that geophone can be determined. This, in turn, requires “overlap” – ***to calculate a delay-time for a particular refractor for a particular geophone, you must have an arrival from that refractor from opposing directions***. This has implications for how the data is acquired in the field and will be discussed in further detail below.

The general flow of a reciprocal time inversion is shown in the following flow chart:

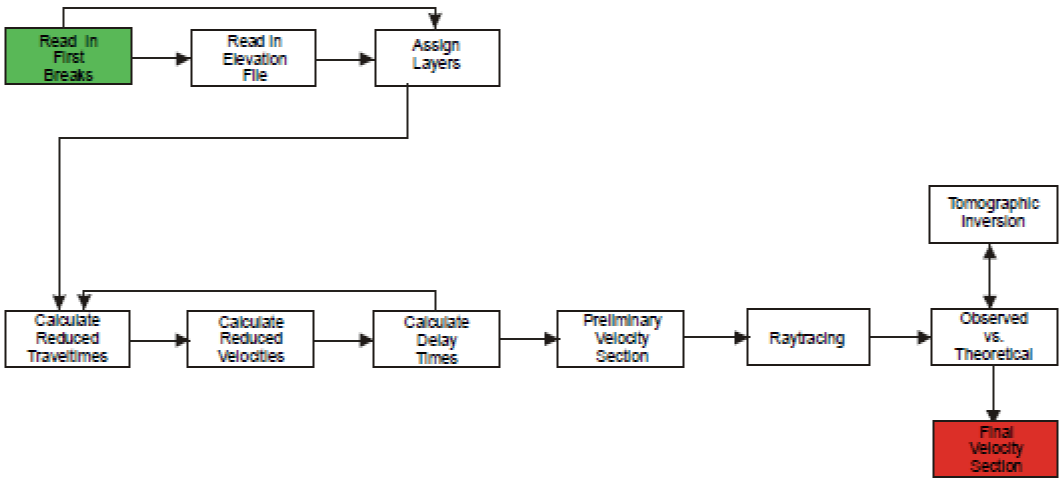


Figure 108: Flow chart for reciprocal time inversion.

If you are doing a simple rippability survey, the Reciprocal Method is generally overkill. You will do a lot more work and yield a marginally more useful answer. But if you are trying to image a fault or a buried stream channel, this technique can often provide a superior image. To learn more about the Reciprocal Method, see Redpath (1973), and Palmer (1980, 1981, 1990).

Below is an example of a refraction record that nicely lends itself to interpretation by the Reciprocal Method:

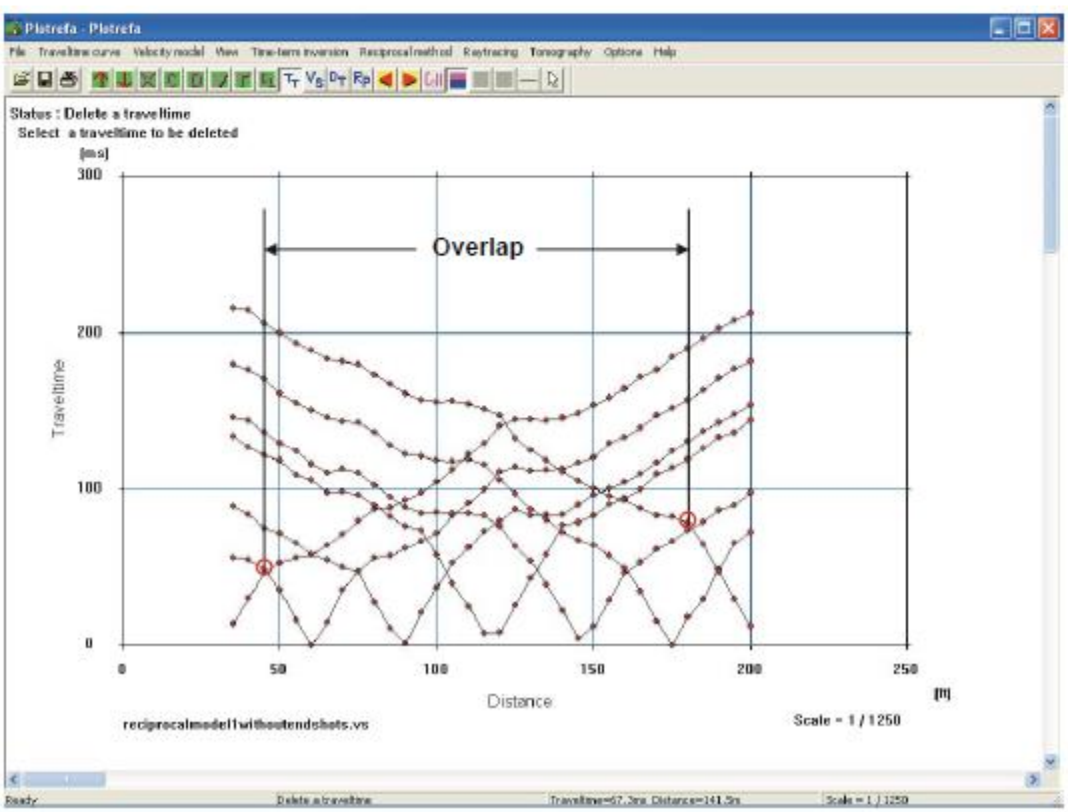


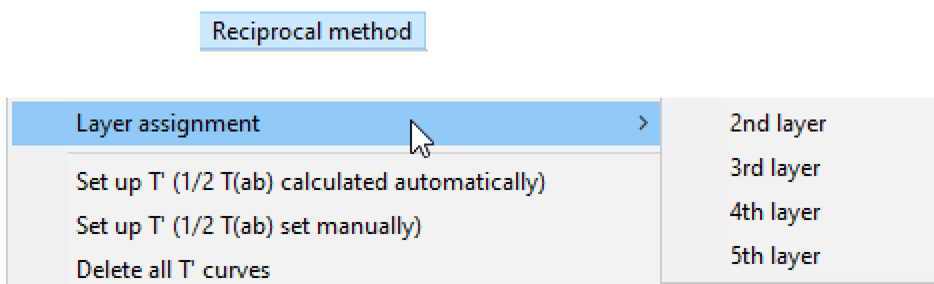
Figure 109: Refractor overlap.

Note the redundancy of the data. This is very important, because like the Time-term method, the Reciprocal Method makes much *use* of the scatter of the data about a best-fit line – in this type of interpretation, this is considered *signal*, not noise. It yields crucial information about the geometry of the refractor. But scatter about the best-fit line can have several other sources, not the least of which is errors in picking. For this reason, redundancy, achieved through numerous shots, is critical. It helps you to separate real structures from artifacts due to picking or other errors.

Also note the region of overlap. It is over this segment of the geophone spread where delay-times can be calculated for a particular pair of shots. Outside of this zone, the Reciprocal Method will not provide a solution.

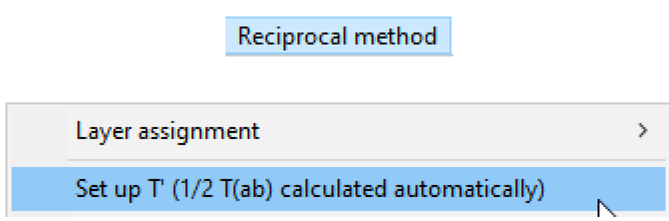
We will now step through each of the items in the **Reciprocal Method** menu.

4.6.1 LAYER ASSIGNMENT



Layers are assigned as discussed in Sections 4.5.1 and 4.5.2, beginning on Page 239. The **Reciprocal Method** menu differs from Time-term in this regard, allowing up to 5 layers to be interpreted.

4.6.2 SET UP T' (½ T(AB) CALCULATED AUTOMATICALLY) [ ]



The T' ("T-prime") or "reduced travel time curve" is a useful tool for determining refractor velocity. It essentially strips away the effects of the overlying layer, as if the shots and geophones were laid directly on the refractor. The T' curve is drawn relative to one-half the reciprocal time (½$T_{(ab)}$). If you wish ½ $T_{(ab)}$ to be calculated automatically, choose *Set up T' (½ $T_{(ab)}$ calculated automatically)*, or press the tool button. Next, click on the travel time curves for two opposing shots, in which there is significant refractor overlap. The two travel time curves will be highlighted, and the reduced travel times will be calculated and plotted:

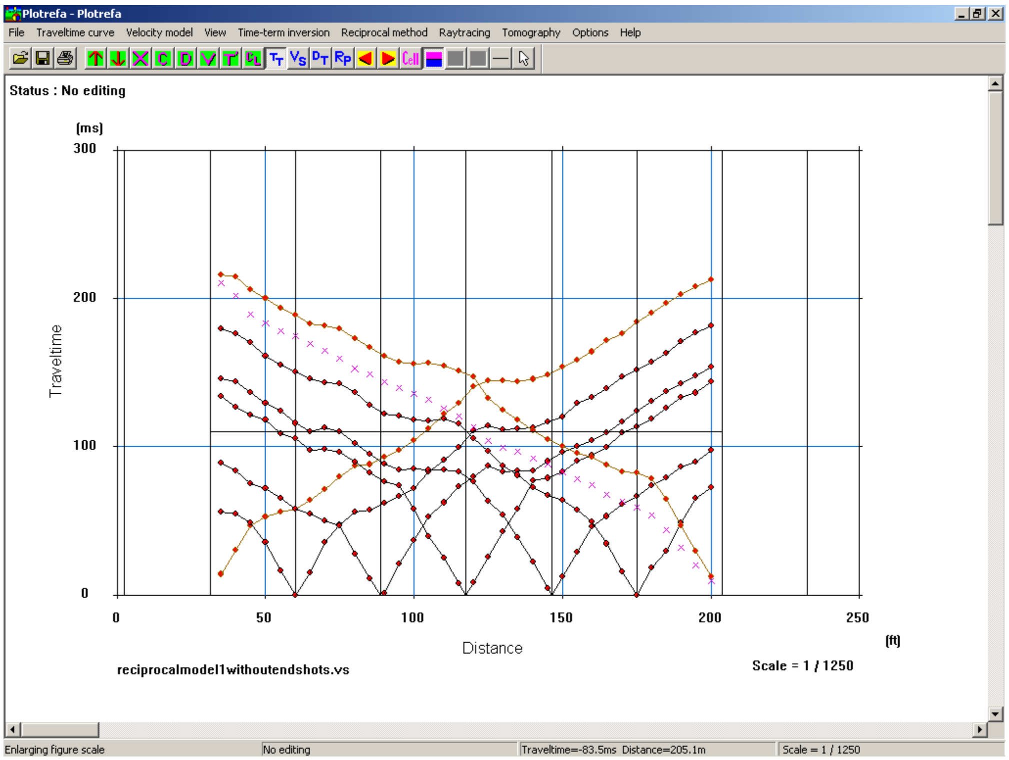


Figure 110: Reduced travel time plot (magenta).

Note: *The reduced travel time curve will be roughly parallel to the first travel time curve that you click on.*

In the example above, the two end-shots were chosen (right shot first), and the reduced travel times are shown in magenta. The ½ $T_{(ab)}$ line is shown in black (at about 110 msec).

See here for a video on setting up T' (be sure to turn up the volume).

4.6.3 SET UP T' (½ T(AB) SET MANUALLY)

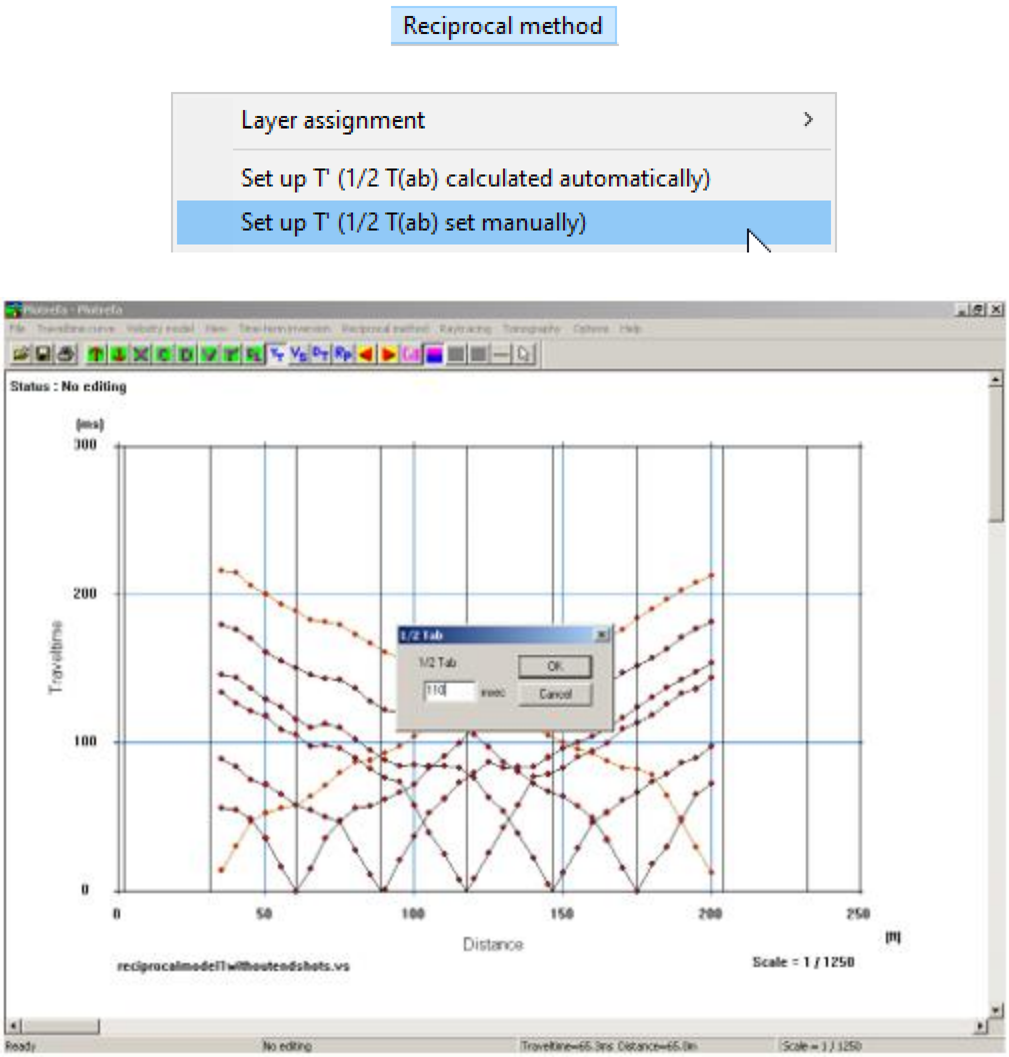
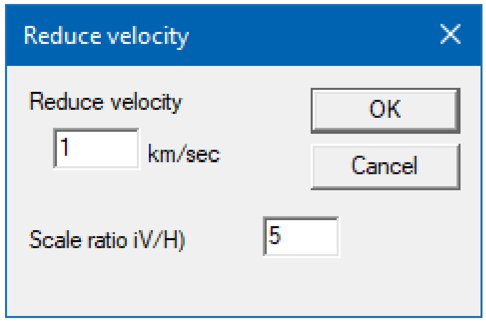


Figure 111: Setting up T.'

If you wish to set the ½ $T_{(ab)}$ value manually, click on *Set up T' (½ $T_{(ab)}$ set manually)*, choose the travel time curves, and you will be presented with a dialog box.

Enter the ½ $T_{(ab)}$ value and press *OK*. The reduced travel times will be calculated and presented as shown in the previous section.

Note that once you have calculated T', you may fit a velocity line to it to determine the reduced velocity (see Section 4.6.6, Page 251). If you then double-click on the T' curve, you will be presented with the following dialog box:



Type in the reduced velocity and press *OK*. The reduced travel time curve will appear:

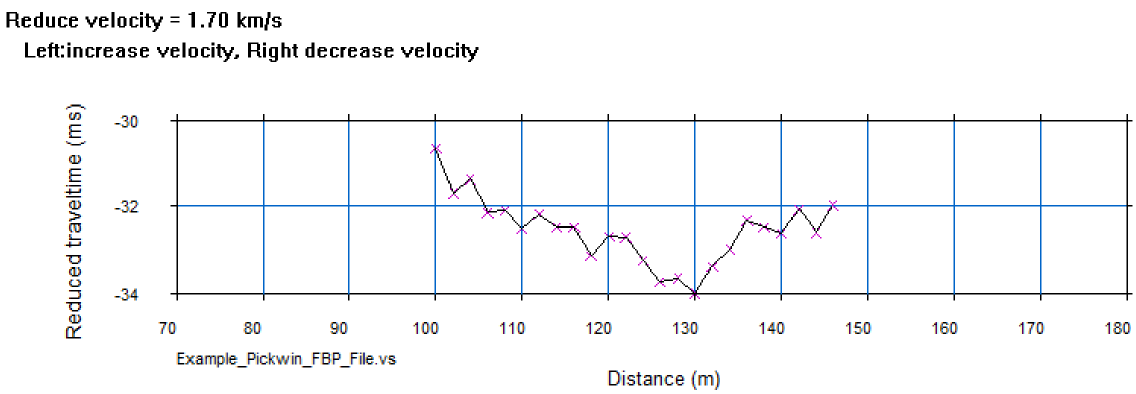
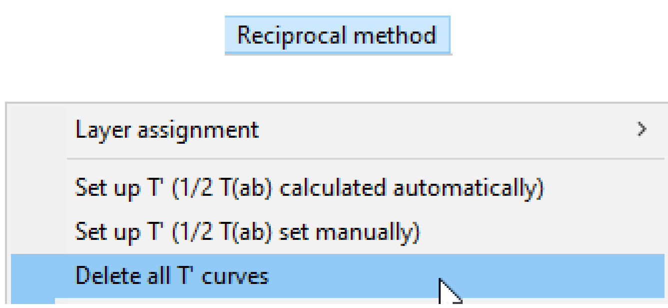


Figure 112: Reduced travel time plot.

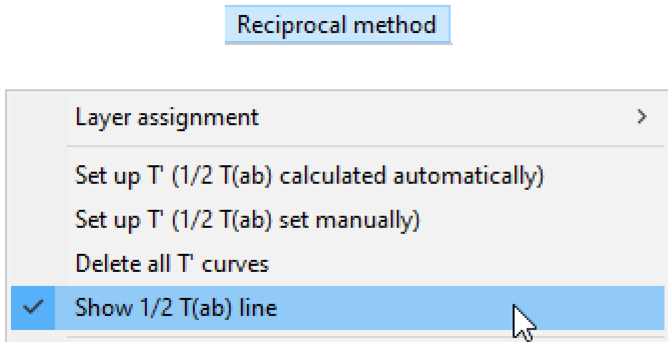
Pressing the left mouse button will increase the reduced velocity, and pressing the right mouse button will decrease the reduced velocity. Note the effect on the reduced travel time curve with varying reduced velocity.

4.6.4 Delete All T' Curves



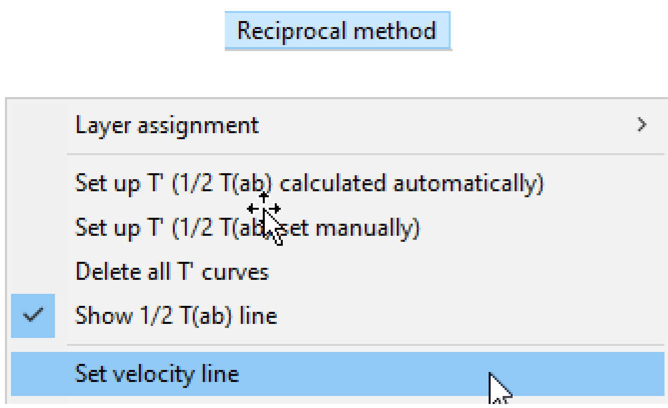
To make maximum use of the Reciprocal Method, you will need to calculate the reduced travel times for all opposing shots that have refractor overlap. Once you have done so, and have used them to calculate delay-times for a particular pair of shots (Section 4.6.10, Page 256), it is best to delete them to avoid confusion. To do so, click on *Delete all T' curves*.

4.6.5 SHOW ½ T(AB) LINE



You may choose whether to display the ½ $T_{(ab)}$ line on the travel time plot. Simply click on *Show ½ $T_{(ab)}$ line* to toggle it on or off.

4.6.6 SET VELOCITY LINE []



After calculating the reduced travel times, you must determine the refractor velocity. To do so, you must fit a line to the reduced travel times. Click on *Set velocity line*. Next, click on the left-most travel time, *within the region of refractor overlap*, and drag to the right-most travel time within this region:

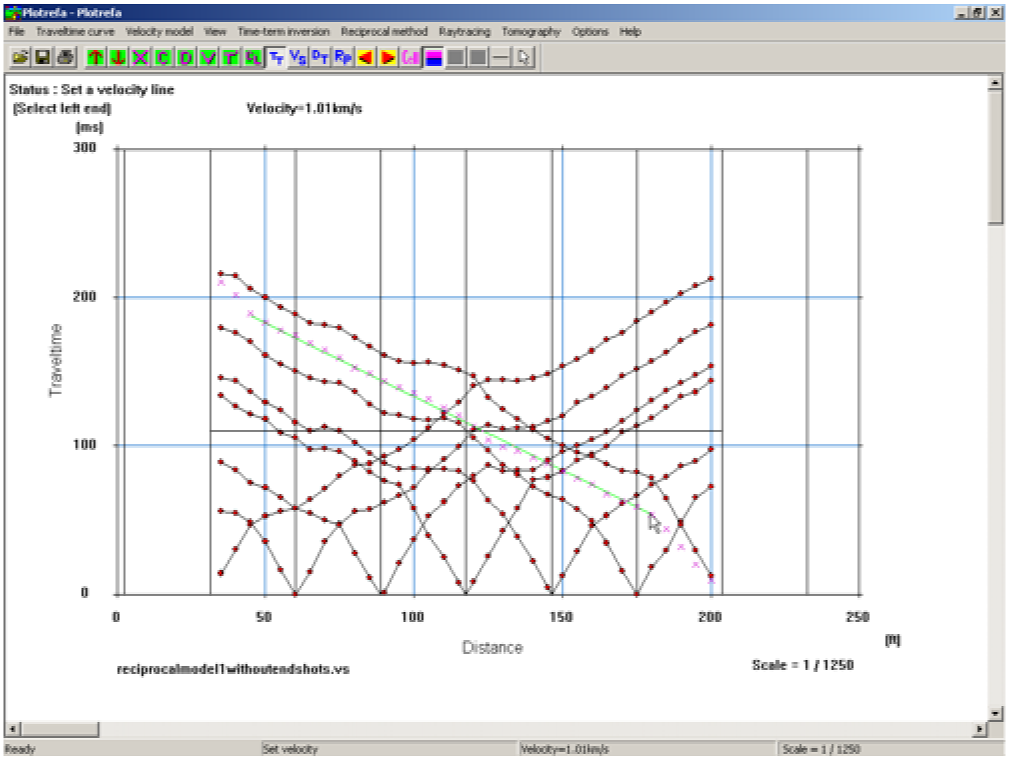


Figure 113: Setting velocity line.

Right click to complete the velocity line:

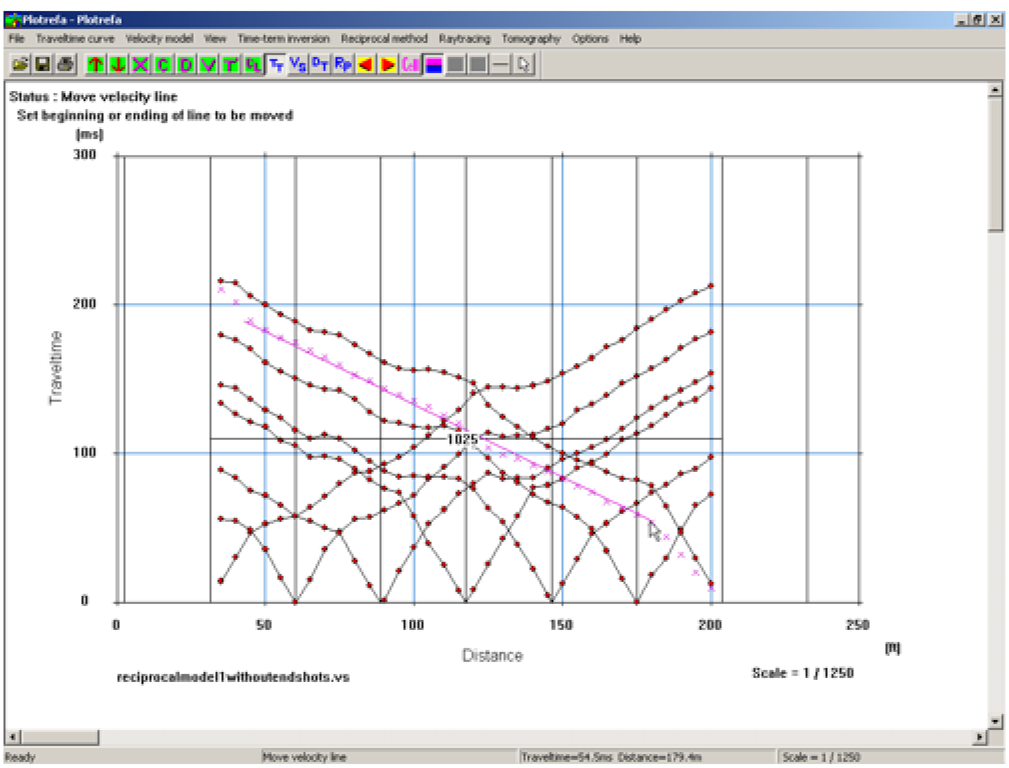


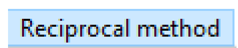
Figure 114: Measuring velocity from reduced travel time curve.

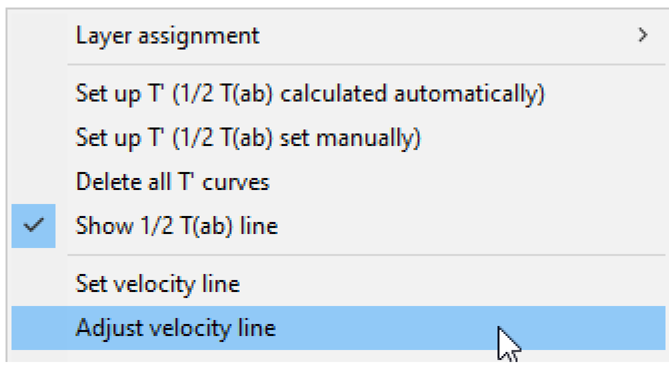
The velocity value will be displayed.

Notice how the reduced travel times at the ends are not included by the velocity line – they are out of the region of overlap.

See here for a video on setting a velocity line (be sure to turn up the volume).

4.6.7 Adjust Velocity Line





After you have drawn the velocity line, you may move it to improve the fit to the data, if necessary. Click on *Adjust velocity line*, then click on one end of the velocity line. A red dot will appear to indicate that you have control:

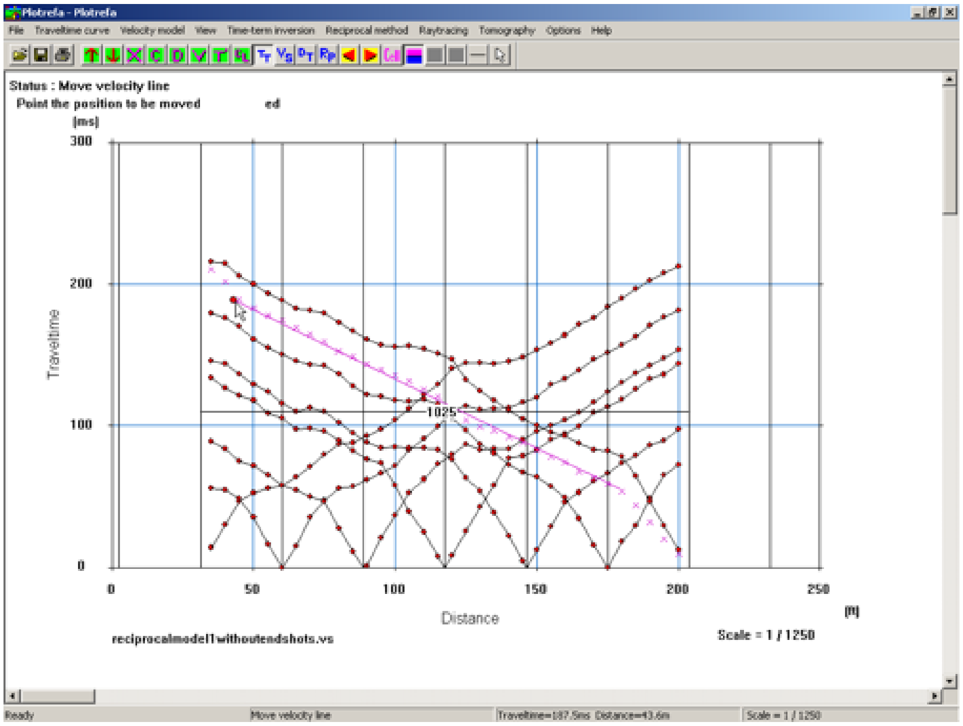


Figure 115: Adjusting velocity line.

Drag the end of the velocity line to the new location:

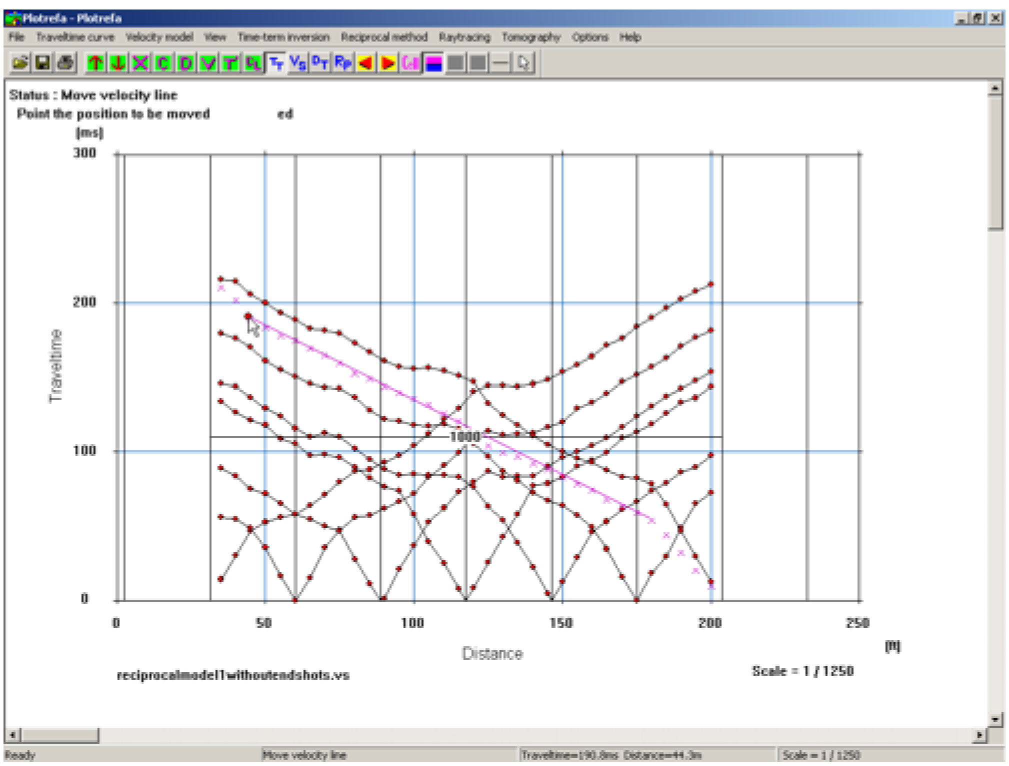
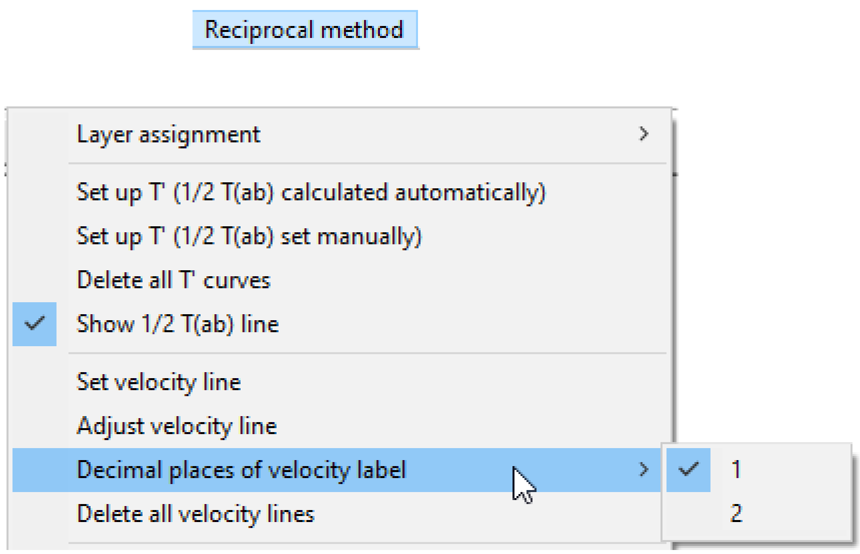


Figure 116: Velocity line adjusted.

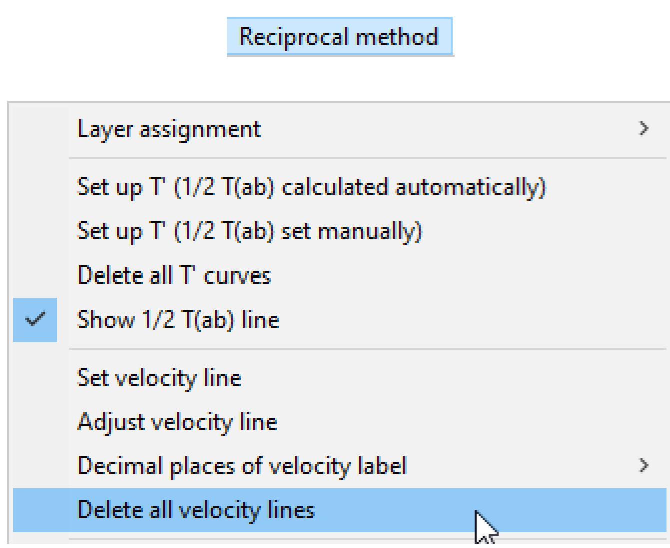
Repeat at the other end of the line if necessary.

4.6.8 Decimal Places of Velocity Label



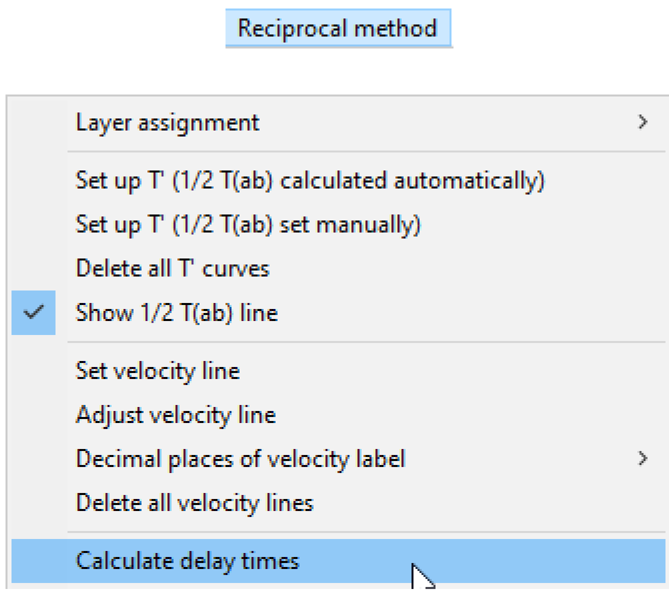
Click on *Decimal places of velocity label* and choose the desired number of decimal places. If you are working in units of kilometers, you will probably want to display velocities to at least one decimal point.

4.6.9 Delete All Velocity Lines



For the same reason you may want to delete reduced travel time curves, you may want to delete velocity lines once delay-times have been calculated for a particular shot pair. To do so, simply click on *Delete all velocity lines*.

4.6.10 Calculate Delay-times



Delay-times are calculated on a shot-by-shot basis. The delay-time is defined as ***the additional time required to traverse any raypath over the time that <u>would</u> be required to traverse the horizontal component at the highest velocity encountered on the raypath***. In the current context, the difference between the measured travel time and the associated reduced travel time is the delay-time (see Appendix E for more on the concept of delay-time). After computing the reduced travel time and setting the velocity line, click on *Calculate delay-times*. Next, click on the velocity label to select the velocity line. When selected, the velocity line will turn red:

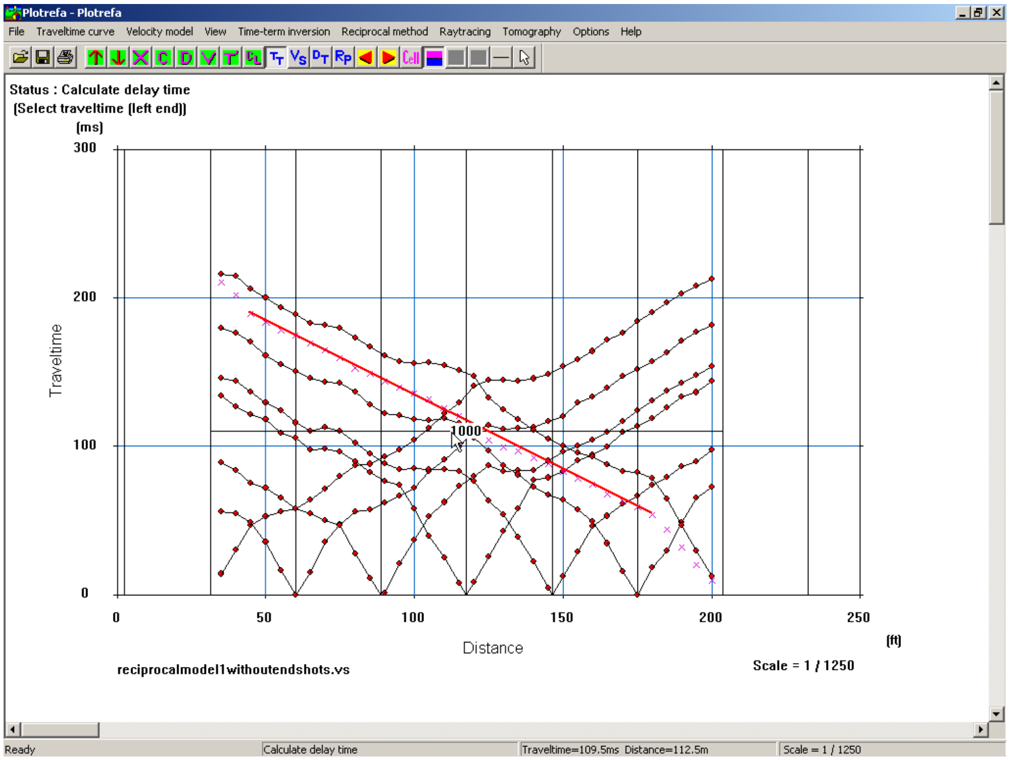


Figure 117: Selecting the velocity line.

Now you must indicate the portion of the curve to compute delay-times for. Like the velocity line, this should include only the region of overlap. Click on the left-most travel time within the region of overlap on the travel time curve parallel to the reduced travel time curve:

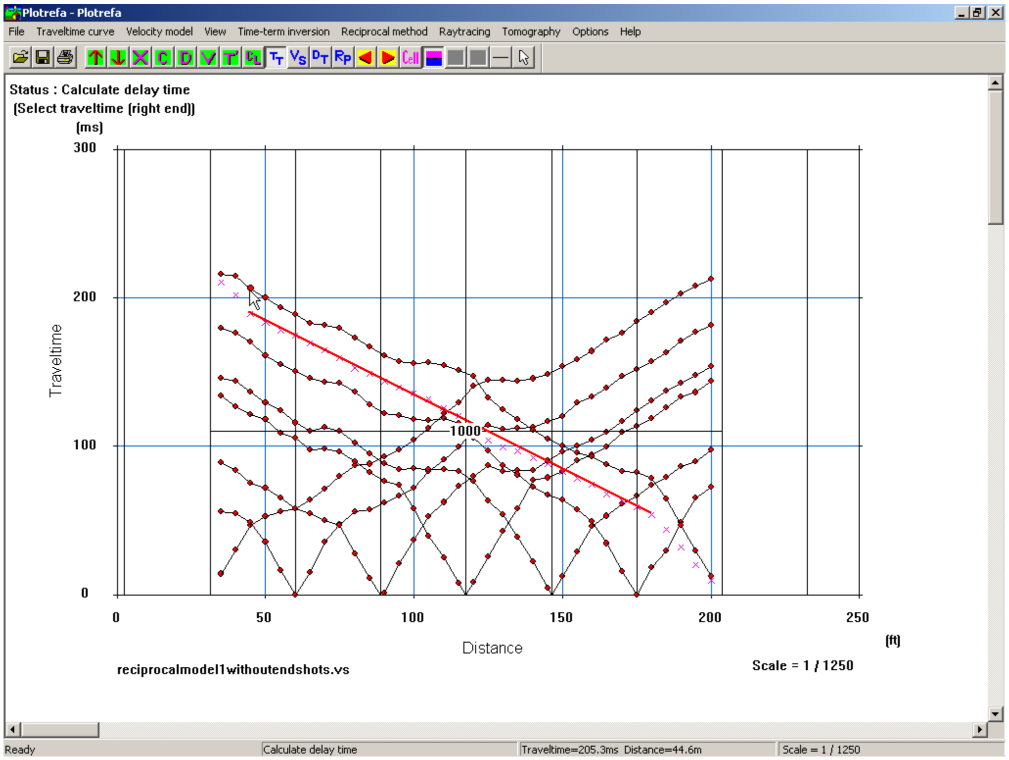


Figure 118: Indicating which portion of curve to compute delay-times for.

Next, click on the right-most travel time in the region of overlap:

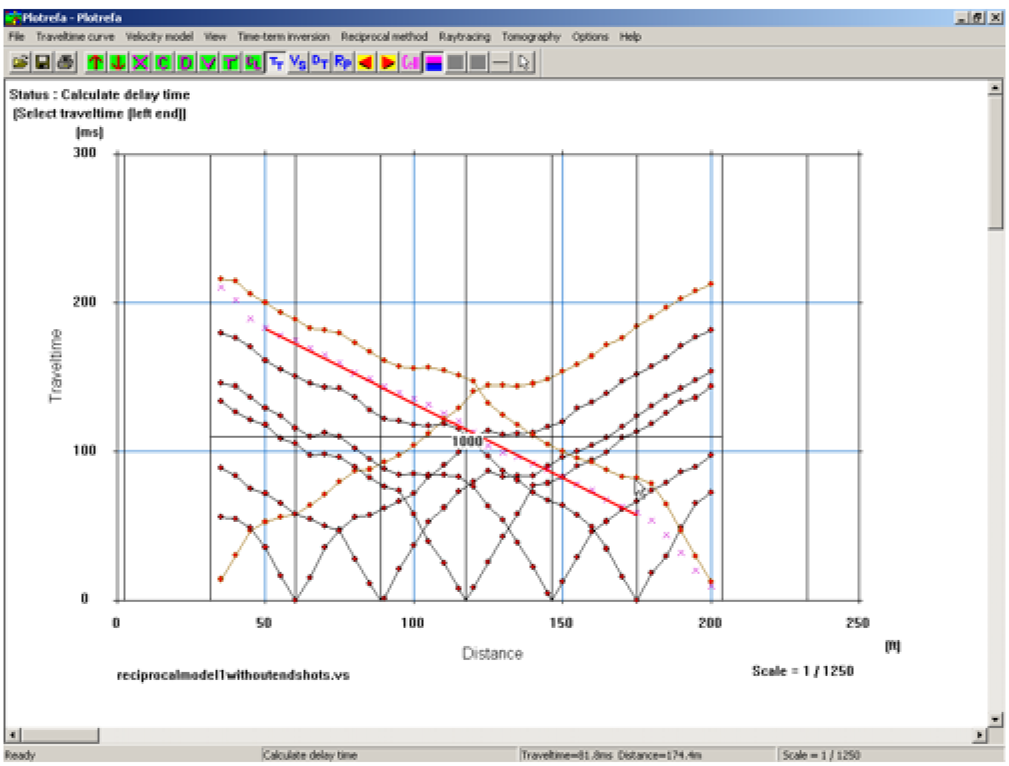


Figure 119: Section of curve for which to compute delay-times is selected.

You will be presented with the delay-times for that shot:

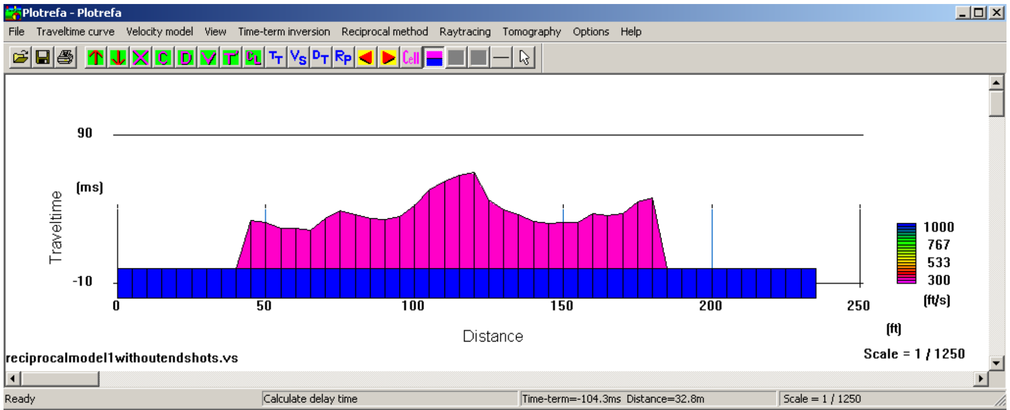


Figure 120: Delay-time plot.

You should do this for all opposing shots that have reasonable overlap, and for each pair, you should calculate the delay-times for *both* shots in the pair. To do so for the above shot pair, we will calculate the reduced travel times again, but this time we will click on the *left* shot first:

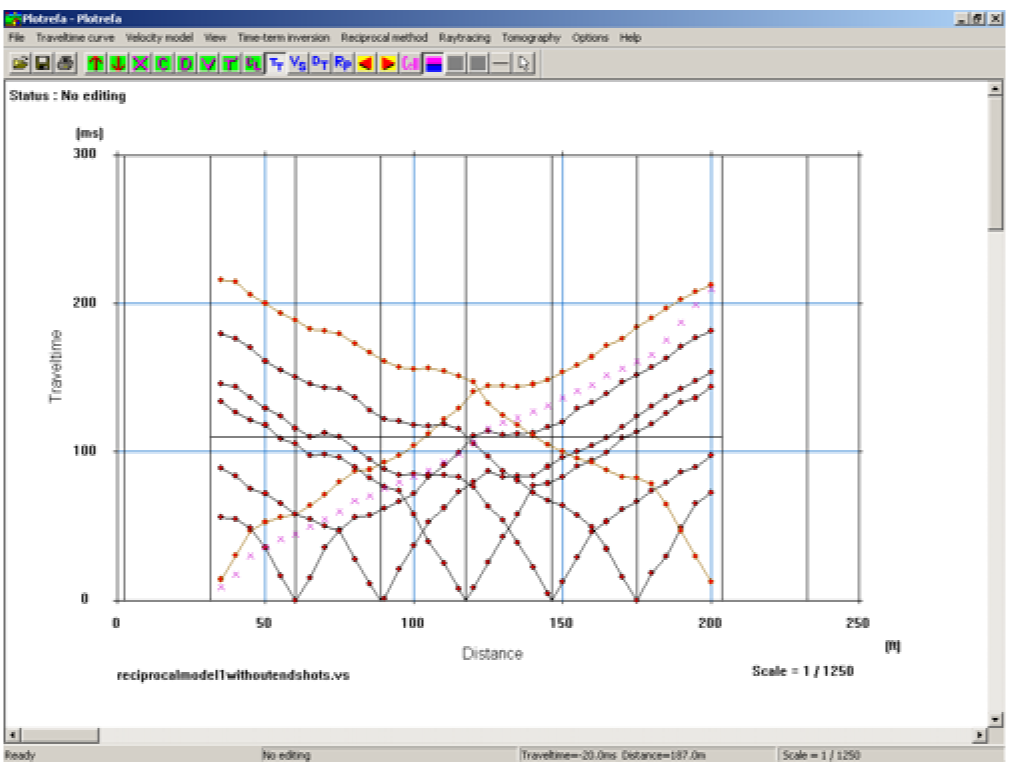


Figure 121: Calculating reduced travel time curve in opposing direction.

Note that the slope of the reduced travel time curve is now in the opposite direction – the reduced travel times are the same as they were before but reversed. The delay-times for the left end-shot shot can now be calculated from the differences between the left-hand-shot travel times and the reduced travel times. Simply follow the procedure detailed above for the right end-shot.

An alternative process for determining delay-times is as follows:

After setting the velocity line, press the  tool button. Then click on the velocity label to select the velocity line:

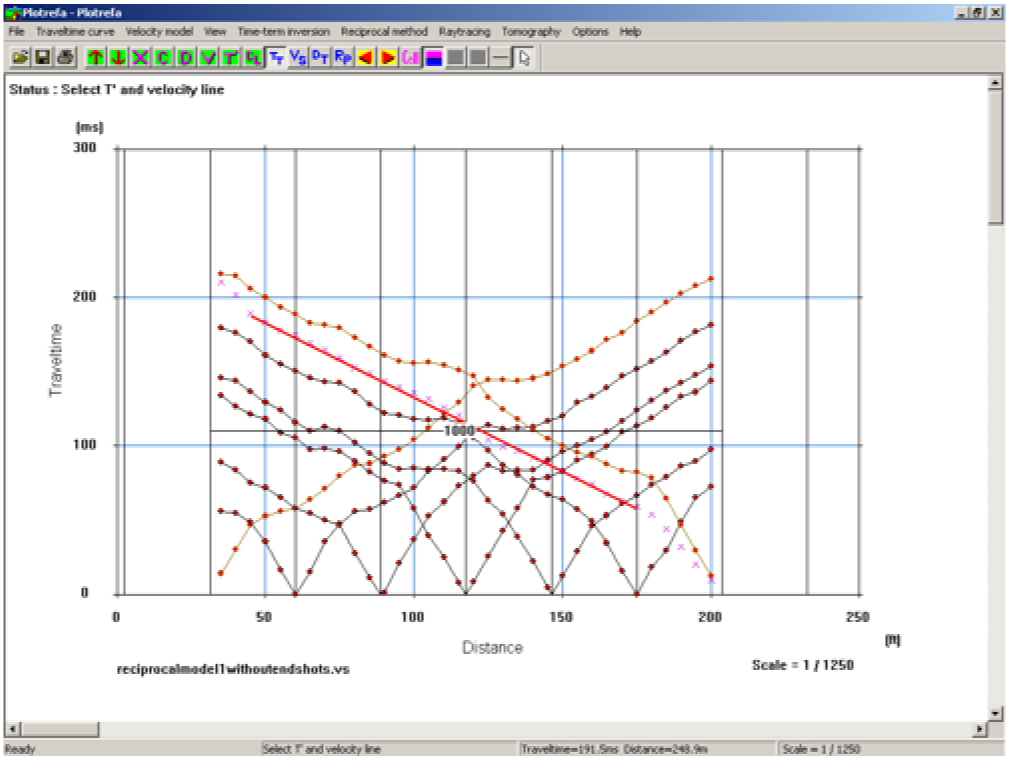


Figure 122: Selecting the velocity line.

Now, right click to bring up the sub-menu:

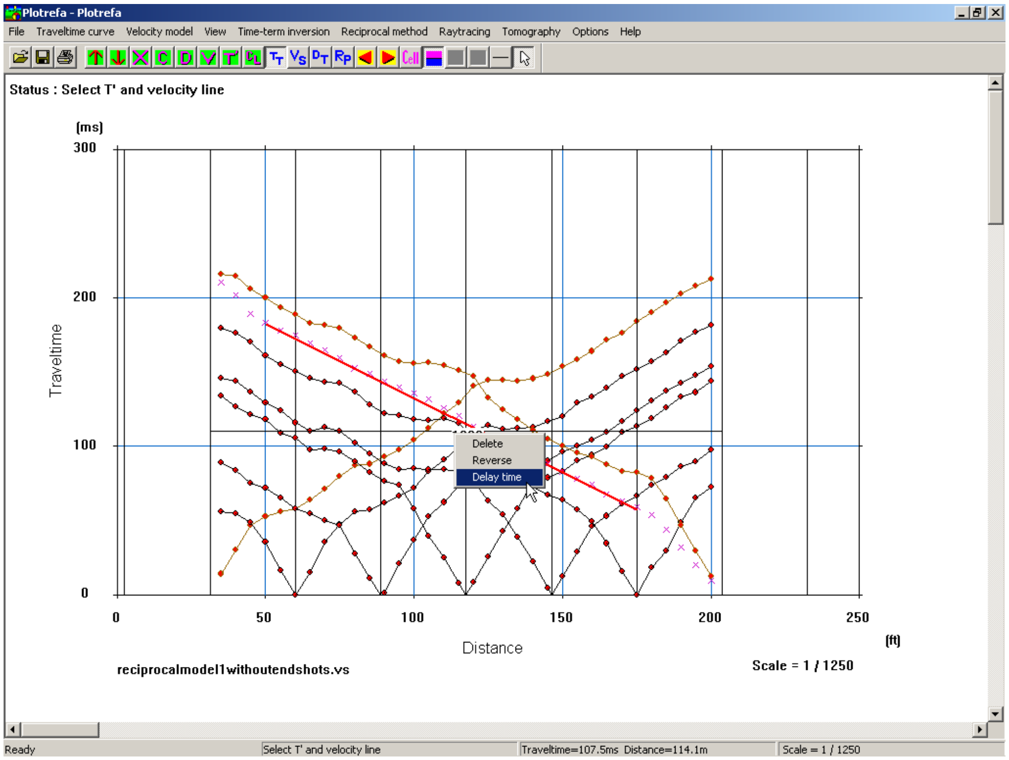


Figure 123: Calculating delay-times.

To calculate delay-times for the right shot, click on *Delay-time*, and then choose the segment of overlap on the travel time plot parallel to the reduced travel times and click. The delay-times for the right shot will be calculated and displayed:

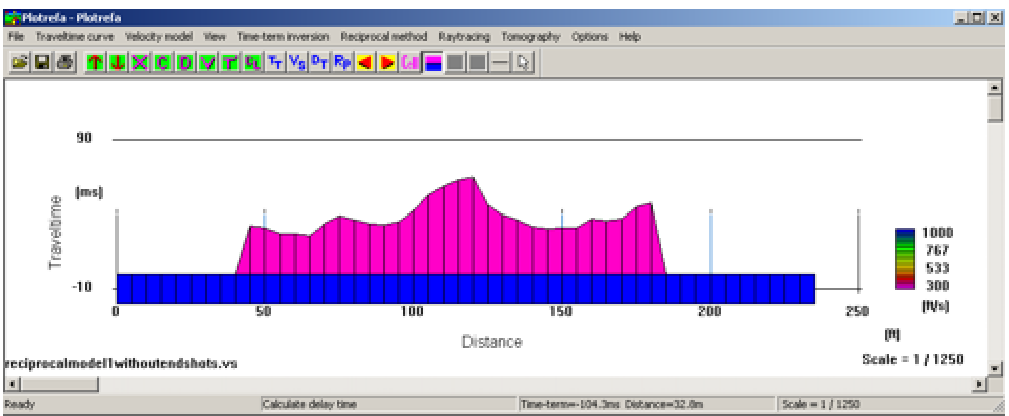


Figure 124: Delay-time plot.

See here for a video on delay-time determination (be sure to turn up the volume).

To calculate the delay-times for the left shot, right click on the velocity label to bring up the above menu again, and choose *Reverse*:

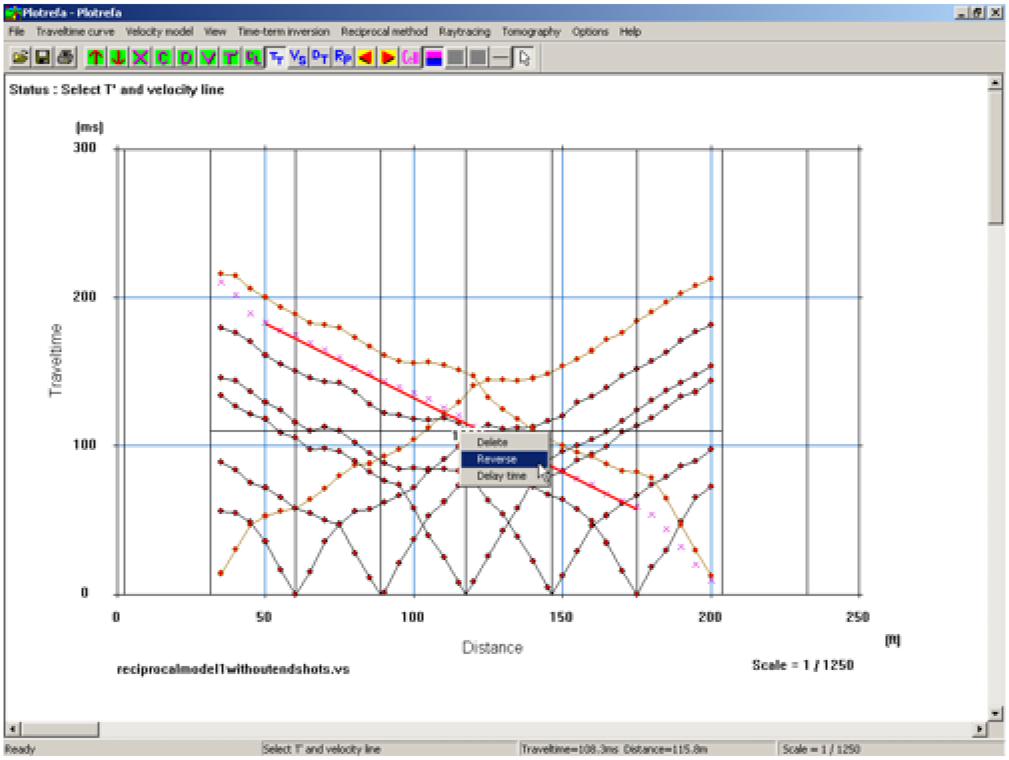


Figure 125: Calculating delay-times in the reverse direction.

Position the cursor on the ½ $T_{(ab)}$ line and click. The velocity line will be reversed:

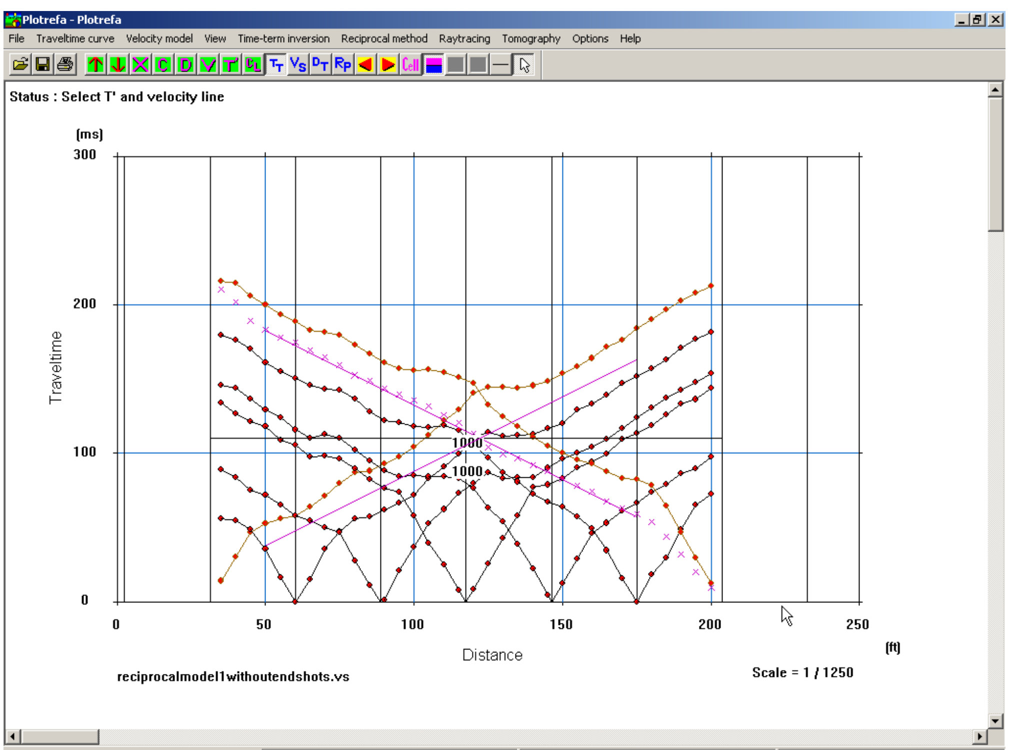


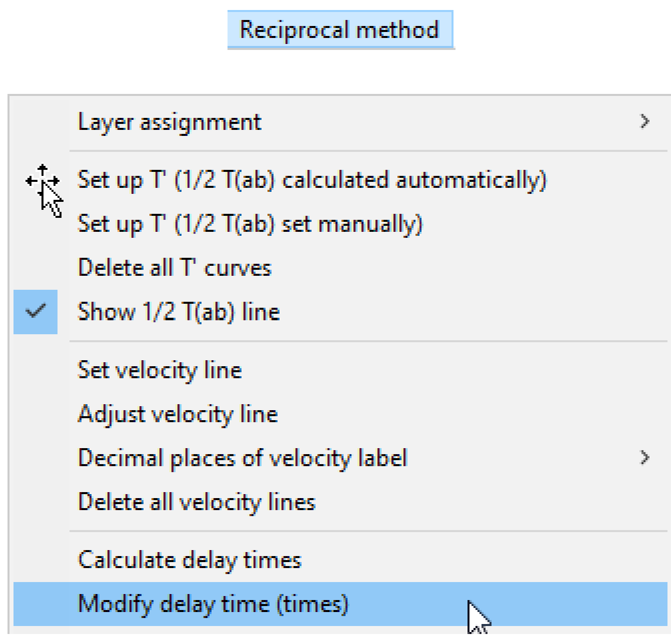
Figure 126: Reverse velocity line displayed.

Select the new velocity line, right click, and choose *Delay-times.* The delay-times for the left shot will be calculated and displayed.

See here for a video on the reverse-shot delay-time determination (be sure to turn up the volume).

See here for a video on the entire delay-time determination process (be sure to turn up the volume).

4.6.11 MODIFY DELAY-TIME (TIMES)



If necessary, you may modify the delay-times graphically, in the same manner that you can modify the layers in the velocity model (Sections 0, 4.3.15, and 0, beginning on Page 204).

Click on *Modify delay-time (times)*, and click on the point you wish to move:

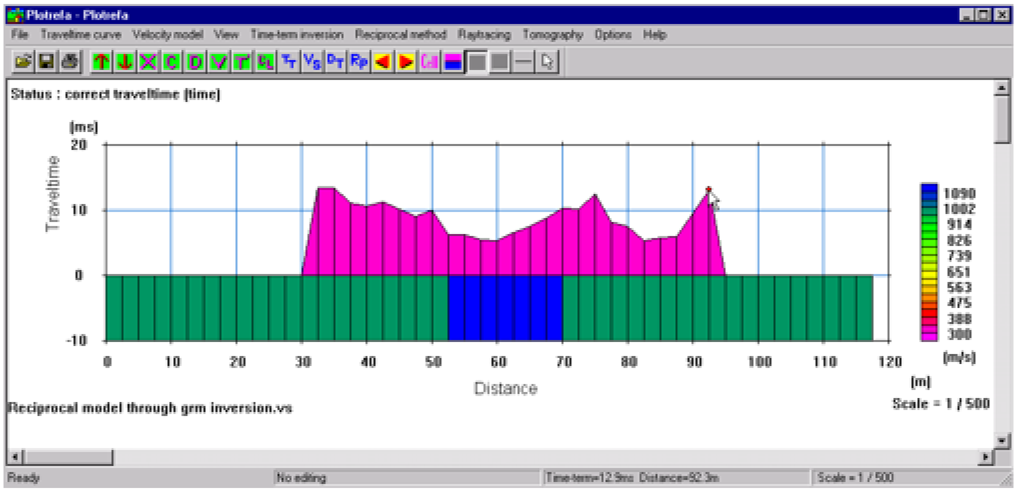


Figure 127: Selecting delay-time to be modified.

Drag the point to the desired delay-time and release:

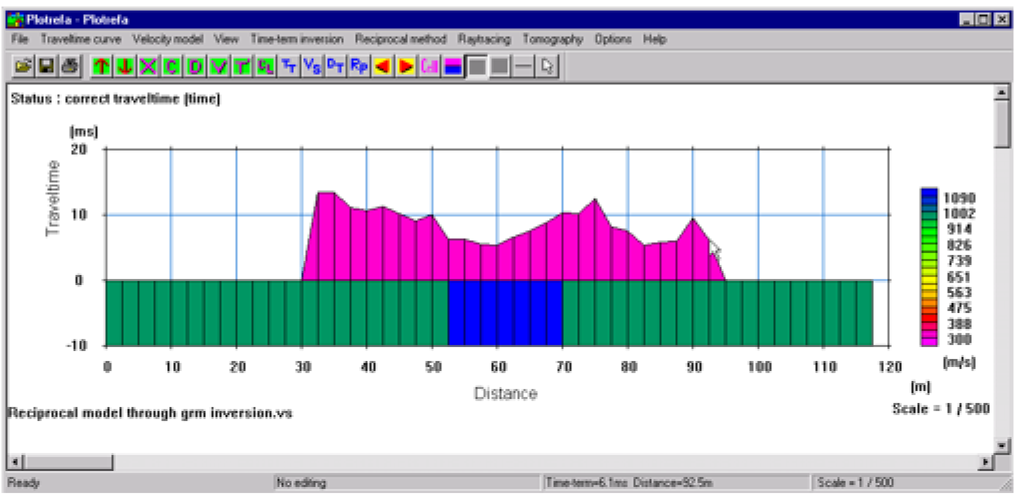
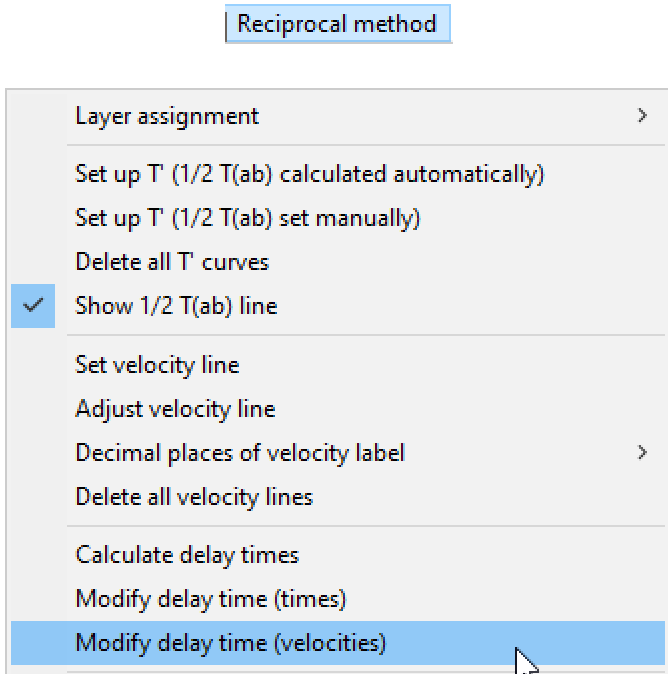
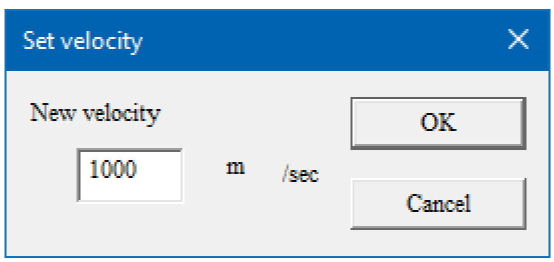


Figure 128: Modifying a delay-time.

4.6.12 Modify Delay-time (Velocities)



Modifying the velocities in the delay-time model is similar to modifying the velocities in the depth model (Sections 4.3.17 and 0, beginning on Page 208). Click on *Modify delay-time (velocities)*, and you will see the following dialog box:



Enter a velocity value, and then click on the cells you wish that velocity to be assigned to:

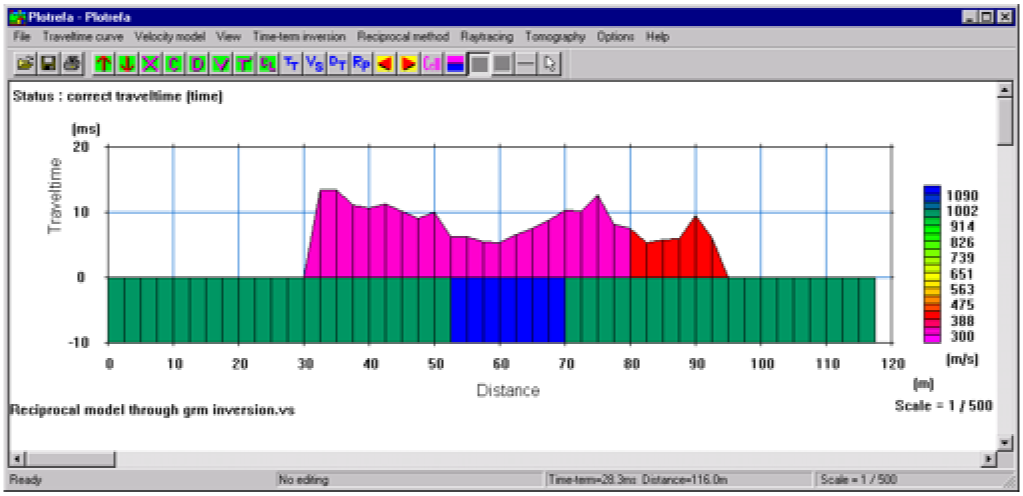
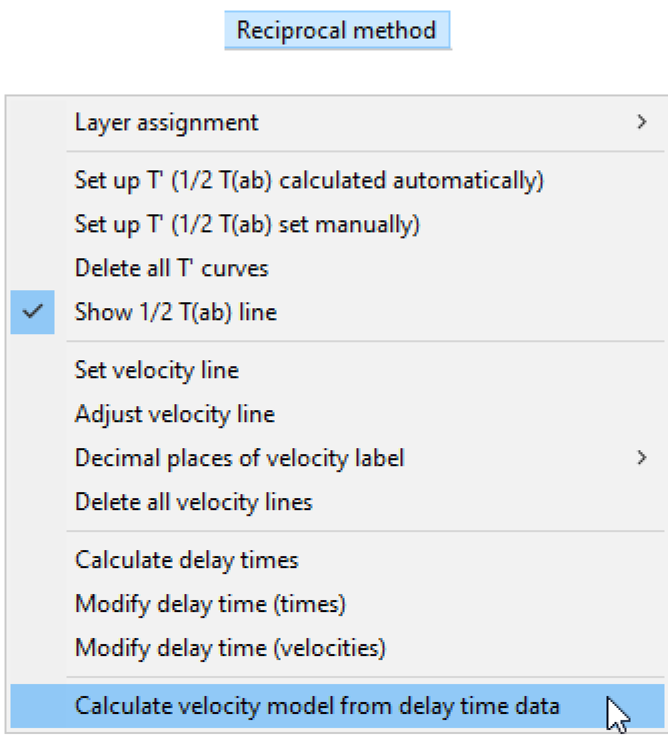
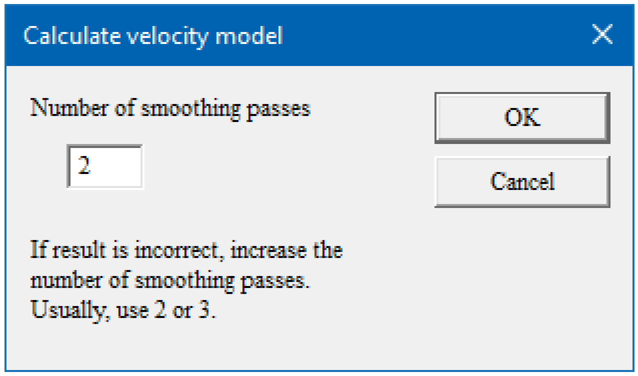


Figure 129: Modifying velocities in delay-time model.

4.6.13 Calculate Velocity Model From Delay-time Data



Once all the delay-times have been determined, you may calculate the velocity model. Click on *Calculate velocity model from delay-time data*. You will be presented with the following dialog box:



Choose a smoothing level (generally 2 or 3) and press *OK*. The velocity model will be calculated and displayed:

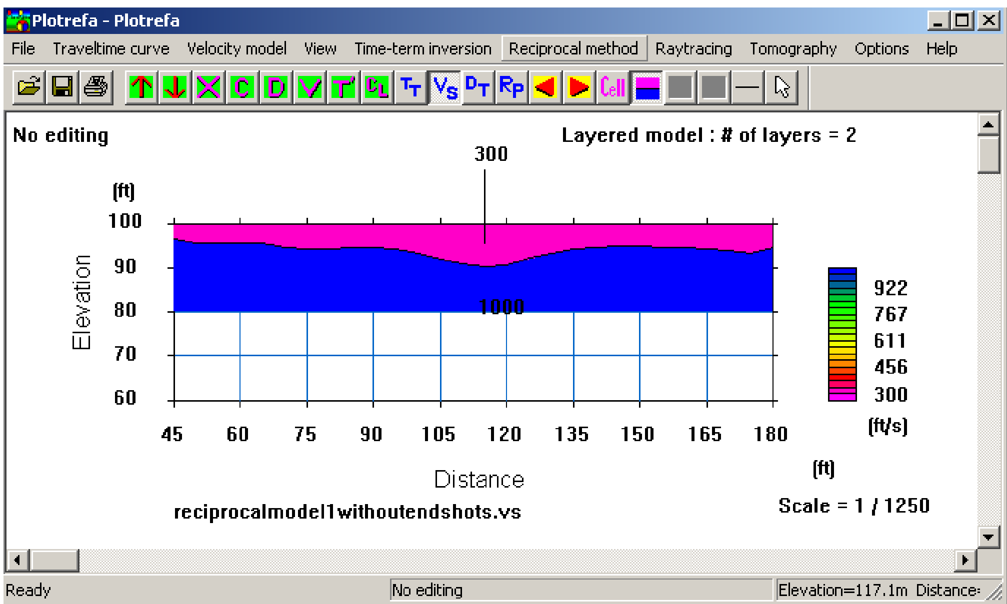


Figure 130: Velocity section computed from delay-times.

Note that the solution is limited to the zone of refractor overlap (45 to 180 feet). At the beginning of this section, it was noted that to calculate a delay-time for a refractor at a geophone, a refracted arrival from that layer is needed at the geophone from opposing directions. In this dataset, the first few arrivals from the end shots are direct arrivals. Hence this condition is not met toward the ends of the geophone spread, and delay-times cannot be calculated.

How can we make maximum use of the geophone spread? We must do “offset” or “remote” shots, as discussed in the time-term section. The idea of offset shots is to move the shot far enough off the end of the line such that *all* the first arrivals from that shot are refracted arrivals, including those nearest the shot. The distance from the offset shot to the nearest geophone should be equal to or greater than the crossover distance at that end of the line.

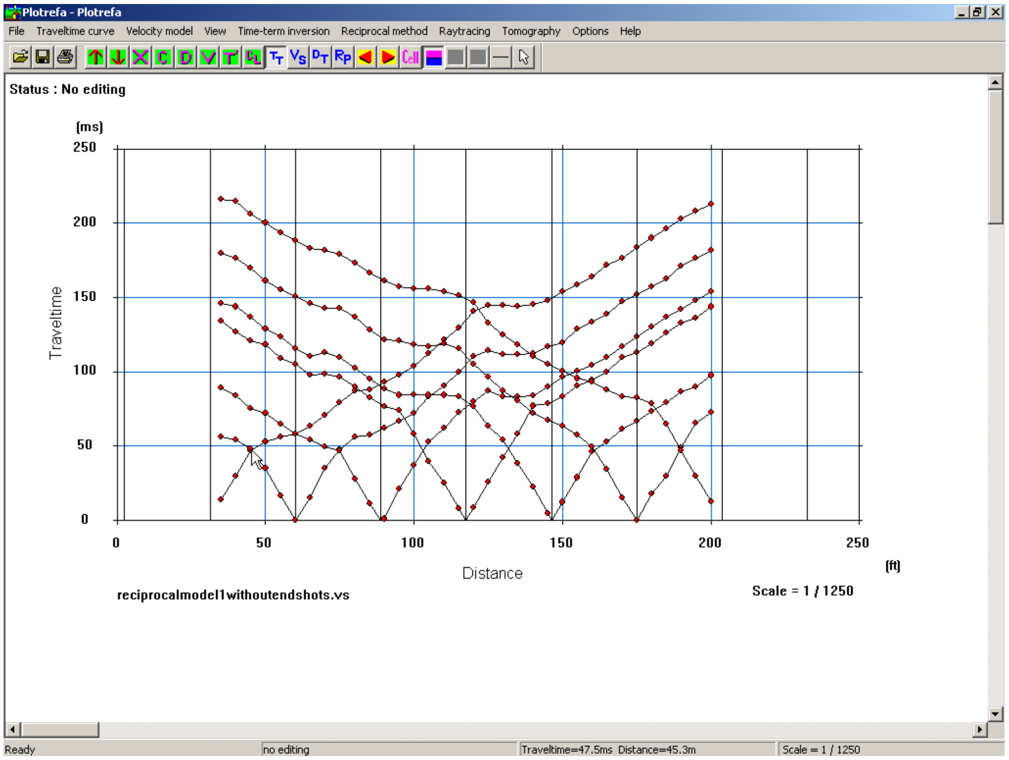


Figure 131: Original dataset.

Above is our dataset again. The left end-shot is at 31 feet. The crossover distance for that shot is at about the third geophone, or 45 feet. That means that the crossover distance is about 15 feet. As such, we want our offset shot to be at least 15 feet to the left of the left-most geophone (it is generally best to add 50% to account for any deepening of the refractor).

Similarly, at the right end, the crossover distance is about 25 feet. We therefore want to do a shot at least 25 feet to the right of the far-right geophone.

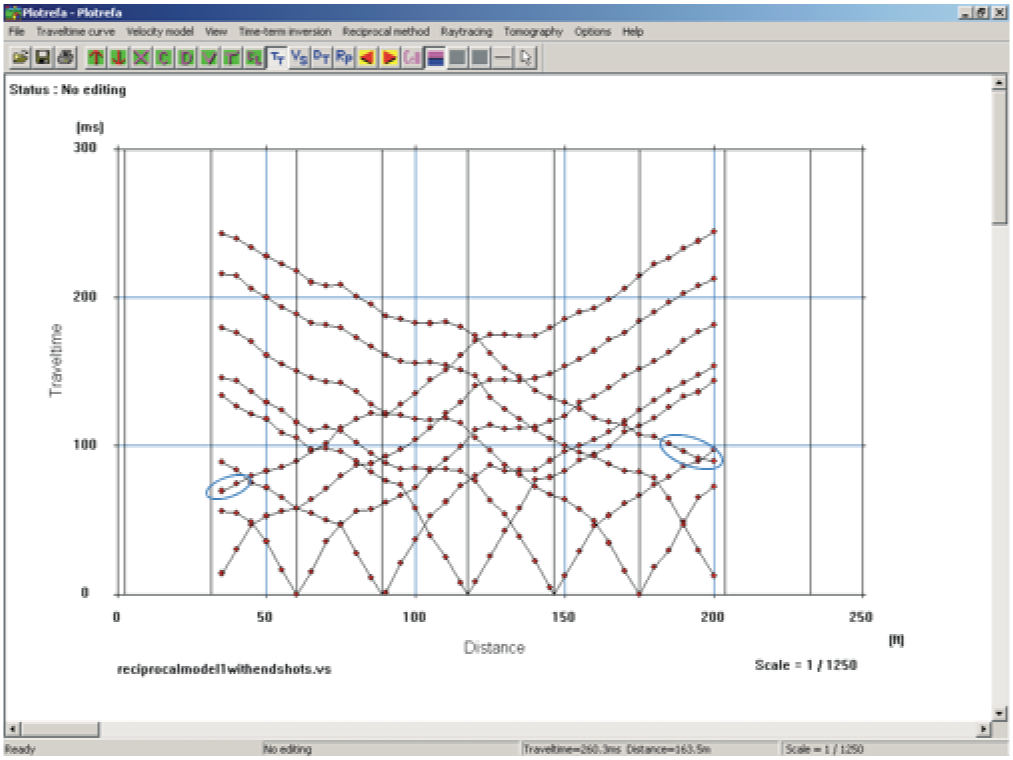


Figure 132: Original dataset with offset shots added.

Above is the same dataset with the addition of offset shots at 3 feet and 233 feet. The new information gained from the offset shots is indicated. We now have refractor overlap over the entire geophone spread and can calculate delay-times for all 48 geophones for these two shots.

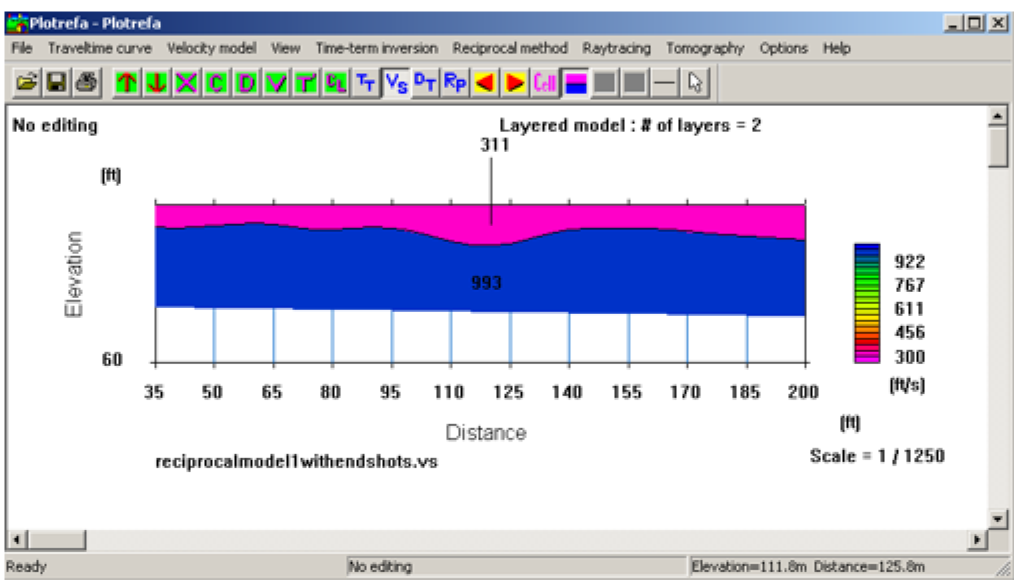
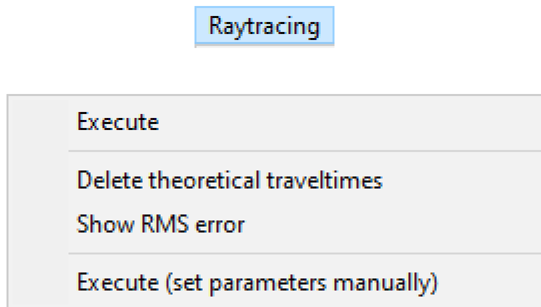


Figure 133: New velocity model incorporating data from offset shots.

We now have a velocity model that covers the entire geophone spread.

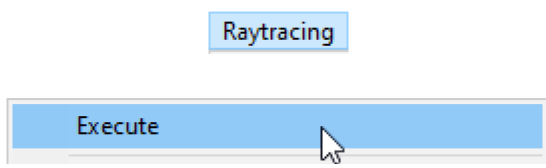
4.7 RAYTRACING MENU

Click on *Raytracing* to reveal the **Raytracing** menu:



As discussed in earlier sections, Plotrefa may be used to calculate theoretical travel times for any velocity model, real or synthetic. This is very useful for pre-survey planning, and for assessing the validity of an interpretation by either the Time-term or Reciprocal Method.

4.7.1 EXECUTE



To calculate the synthetic travel times, simply click on *Execute*. The travel times will be calculated and displayed along with the observed data, and the RMS error will be reported:

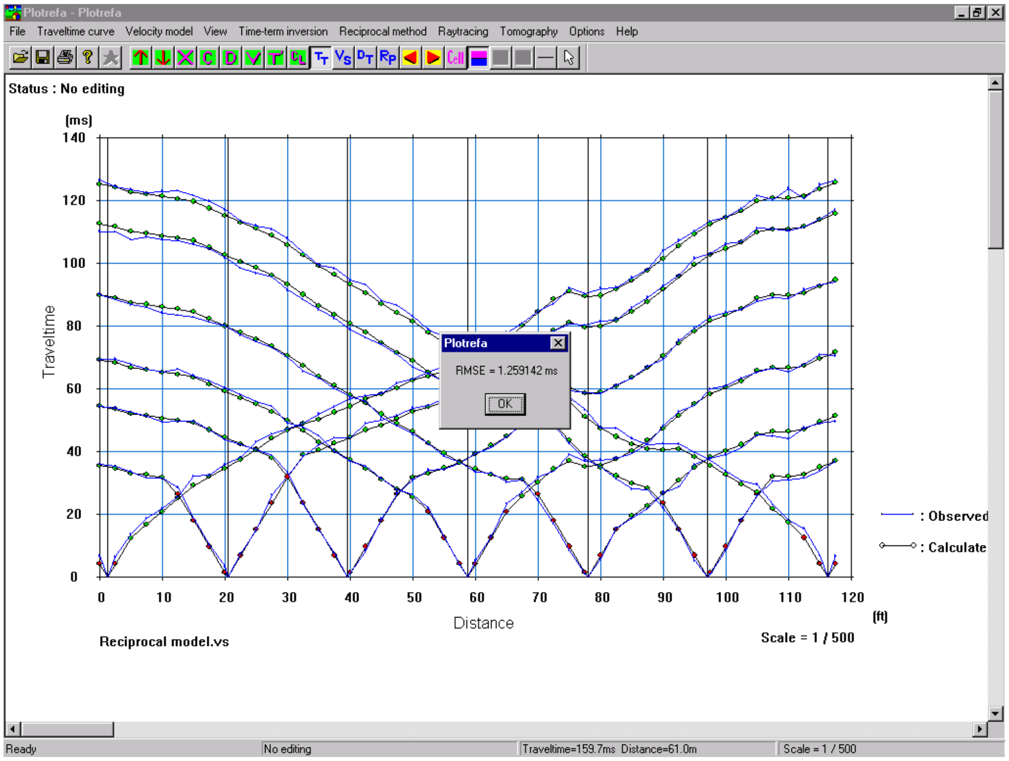
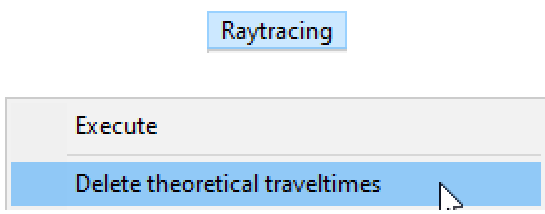


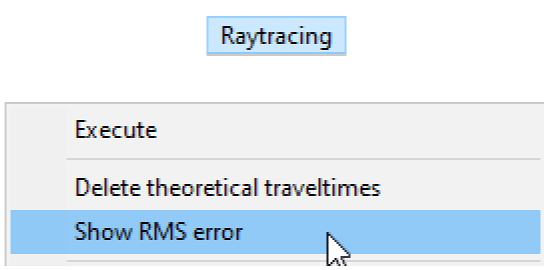
Figure 134: Observed vs. calculated data.

4.7.2 Delete Theoretical Travel times

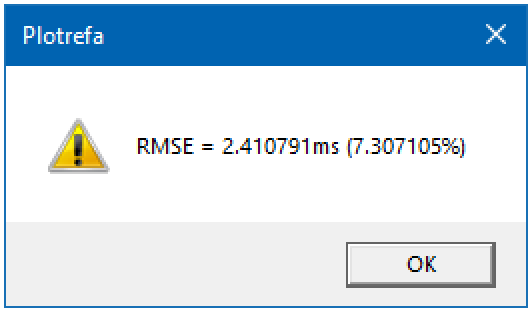


If you would like to delete the theoretical travel times, click on *Delete theoretical travel times*.

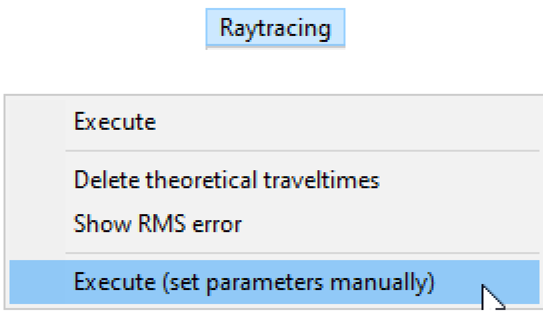
4.7.3 Show RMS Error



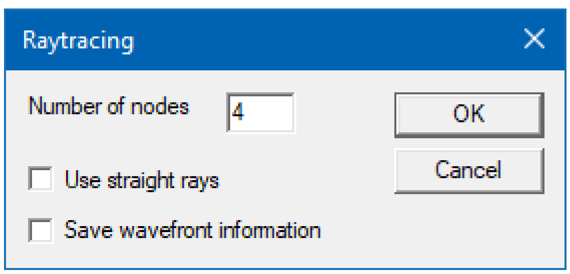
If you would like to check the RMS error of the theoretical travel times, choose *Show RMS error*:



4.7.4 EXECUTE (SET PARAMETERS MANUALLY)



You have a certain amount of control over the raytracing algorithm. Choosing this option will reveal the following dialog box:



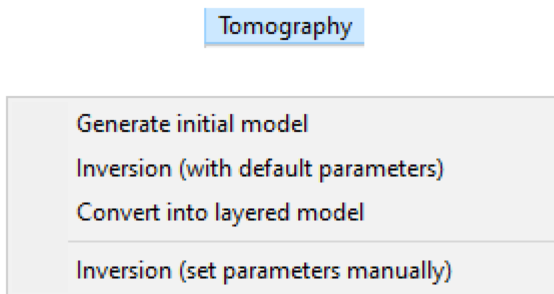
Enter the *Number of nodes* (see Section 4.8.4, Page 278), and indicate whether you want to use straight rays. If you wish to save the wave front information, check the box.

The greater the number of nodes, the more accurate the model, but the greater the computational time requirement.

In general, *Use straight rays* can be left unchecked. Checking the box increases the accuracy of calculation in layered models with complex surface topography and/or layer boundaries.

4.8 TOMOGRAPHY MENU

Click on *Tomography* to reveal the **Tomography** menu:



Tomographic inversion is the third interpretation technique provided by Plotrefa. This method starts with an initial velocity model (usually generated by a Time-term inversion), and iteratively traces rays through the model with the goal of minimizing the RMS error between the observed and calculated travel times. Layer assignments are not required (except in the Time-term inversion to generate the initial model).

Tomographic inversion is generally best used when velocity contrasts are known to be more gradational than discrete, when strong horizontal velocity variations are known to exist, and in extreme topography. All these cases can lead to erroneous results with the previous two interpretation techniques, depending on the severity.

The typical flow of a tomographic inversion is shown in the chart below:

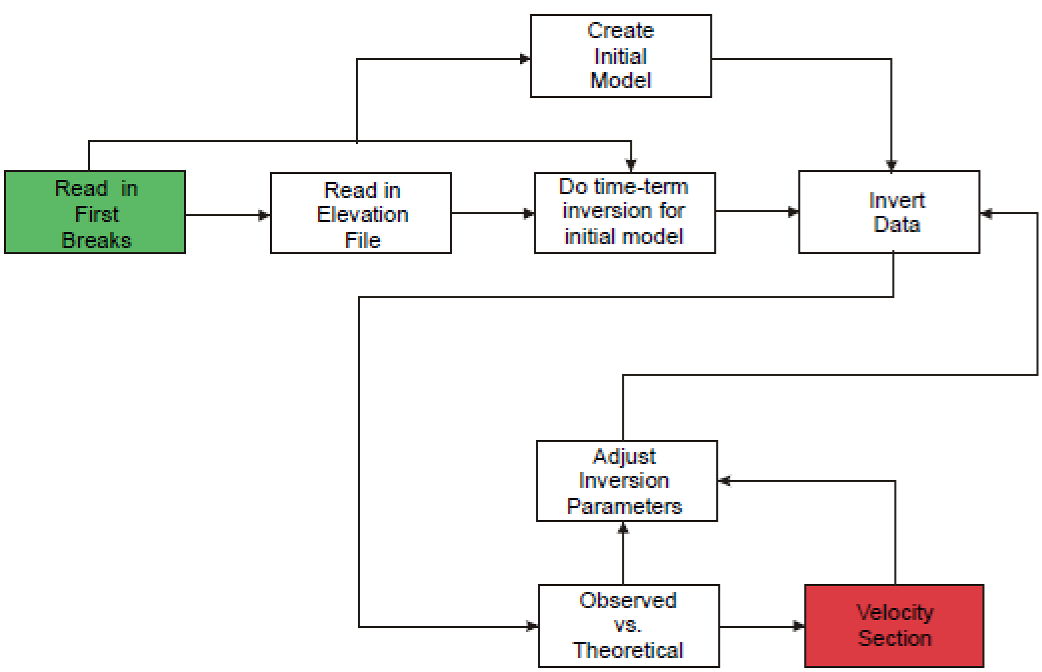
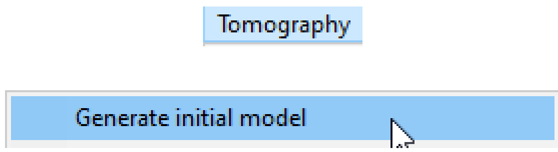
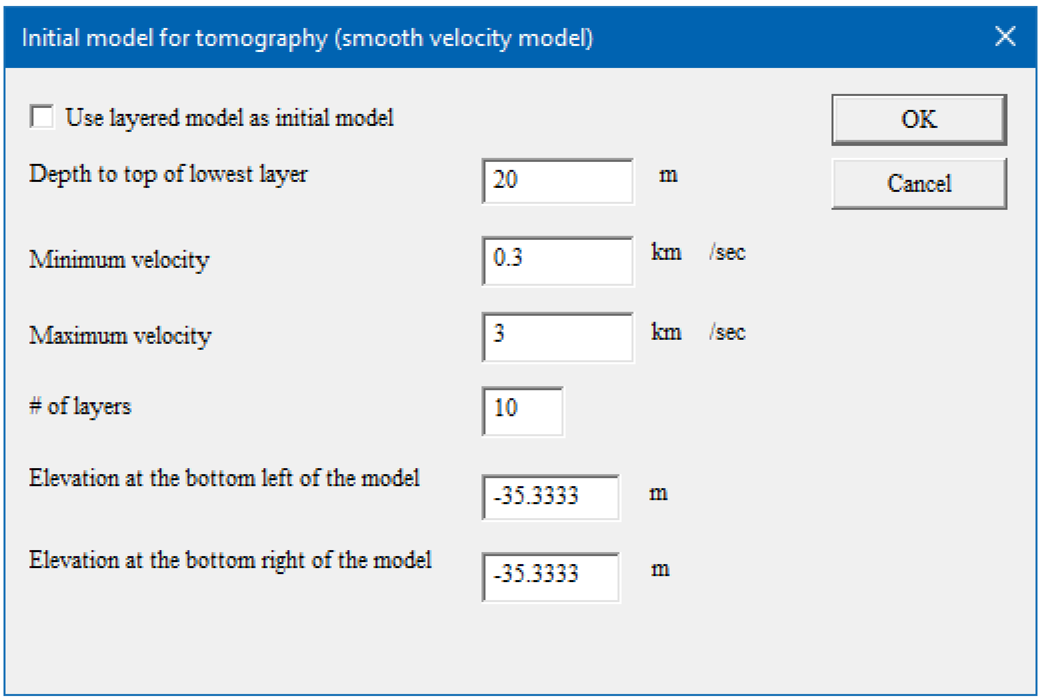


Figure 135: Flow chart for tomographic inversion.

4.8.1 GENERATE INITIAL MODEL



The first step is to create the initial model. Click on *Generate initial model* to reveal the following dialog box:



The chosen parameters for the initial model should bracket the possibilities. You can get an idea, for instance, of the minimum and maximum velocities from the raw travel time curves. From these and crossover distances, an idea of maximum depth of the deepest layer can be estimated.

Note: *By far the most important parameters to get right are the minimum and maximum velocities. If these do not bracket the actual velocities, the inversion will not converge. If you are setting these values manually, always err on the conservative side – the maximum velocity can be 20-30% higher than the real maximum, but it should not be lower. Similarly, the minimum velocity can be somewhat lower than the true minimum, but it should not be higher.*

In any case, the best way to generate the initial model is to do a quick time-term inversion of the data. Then, open the above dialog box and check the *Use layered model as initial model* checkbox. This overrides all the other settings in the dialog box, including the minimum and maximum velocities. If you have done a reasonable time-term inversion, the minimum and maximum velocities from this should provide a good tomographic inversion. After doing the inversion, you may change the minimum and maximum velocities and re-invert if necessary. See the tomography example (Example 4) in Appendix B.

Once you have entered the necessary parameters, press *OK*, and you will be presented with the initial velocity model:

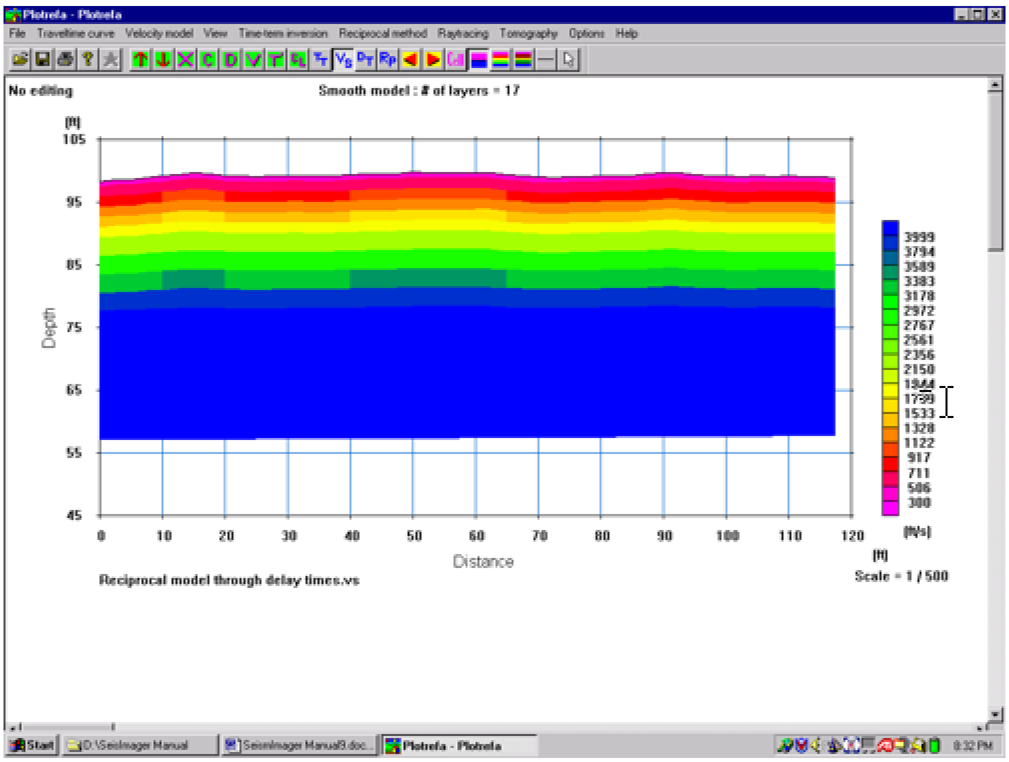
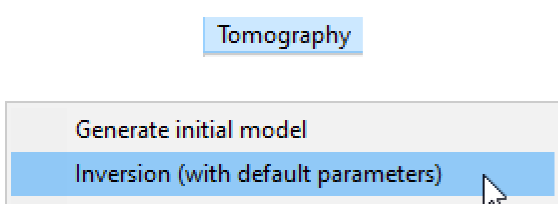


Figure 136: Initial model for tomographic inversion.

4.8.2 INVERSION (WITH DEFAULT PARAMETERS)



You may now conduct the tomographic inversion. If you would like to do so using default inversion parameters (most common), click on *Inversion (with default parameters)*. The inversion will begin. This can take a minute or two, depending on the speed of your processor. Progress will be shown in the upper left-hand corner. When the inversion is complete, the velocity model will be displayed:

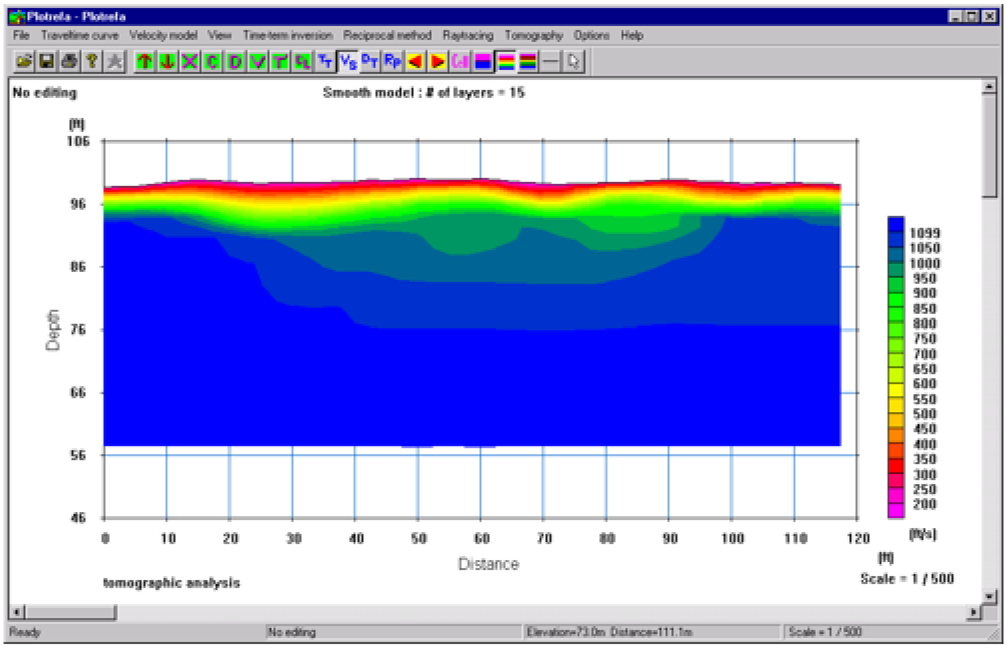


Figure 137: Tomographic inversion.

To see the agreement between the calculated and observed data, display the travel time curves by pressing the  tool button:

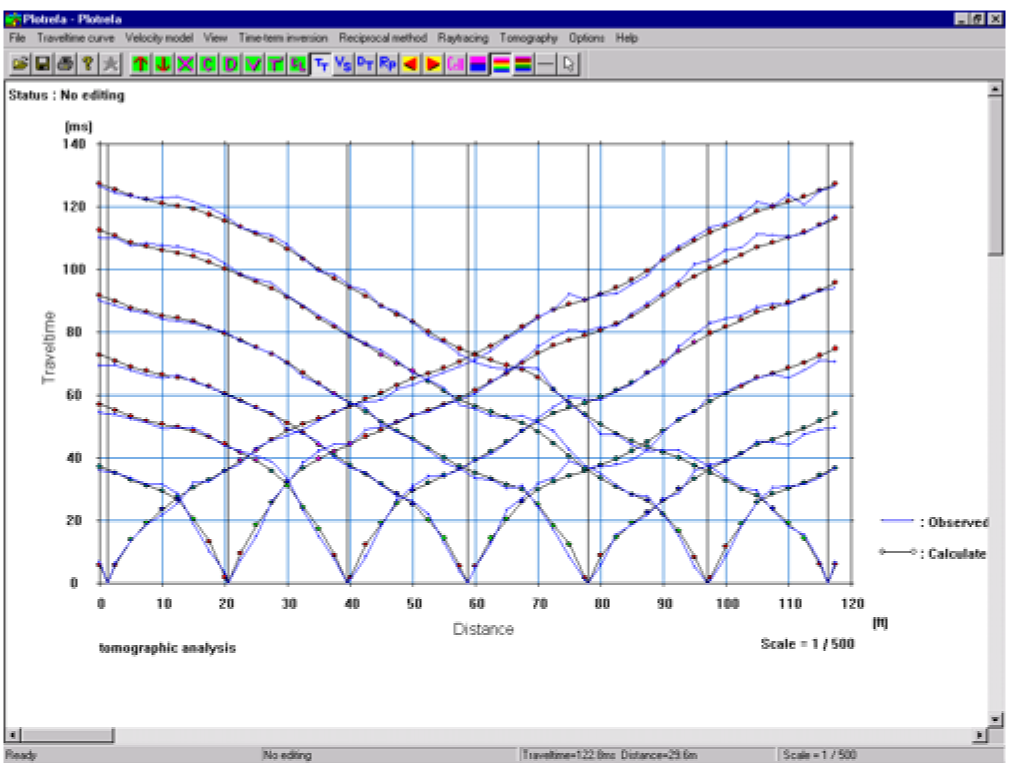
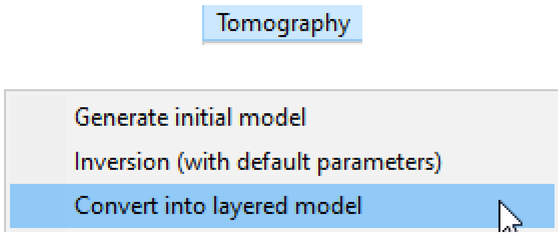


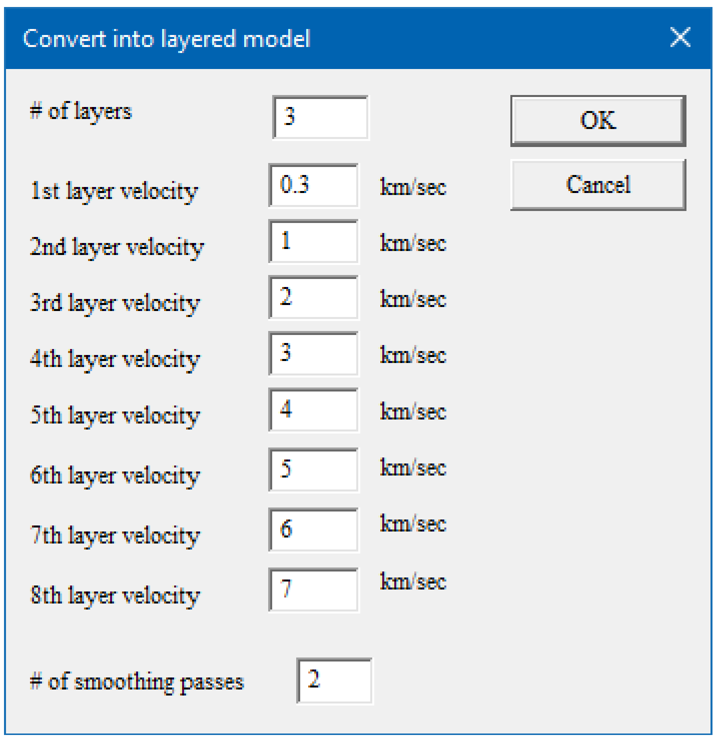
Figure 138: Calculated vs. observed data for tomogram.

4.8.3 Convert Into Layered Model



If you are working in extreme topography, it is often better to use a tomographic approach even in cases of very discrete velocity contrasts. You may then convert the tomogram into a layered model to better represent the layered nature of the geology.

To create a layered model, click on *Convert into layered model*:



You must provide the number of layers and the velocities you wish them to have. The program will divide the tomogram into the number of layers you specify, and the boundaries between them will divide layers having *bulk* velocities matching the specified velocities. Ideally, if you had done a layered interpretation from the start, this is what it would have looked like.

In the above example, examination of the travel time curves quickly indicates a two-layer case with approximate velocities of 300 and 1000 feet per second.

Entering this information into the above dialog box yields the following:

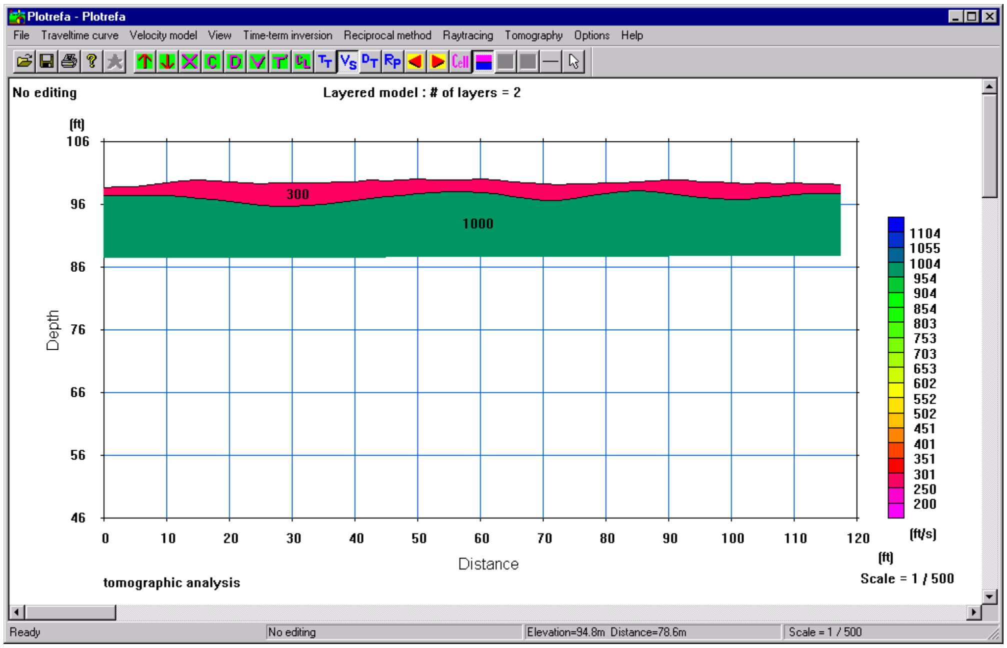
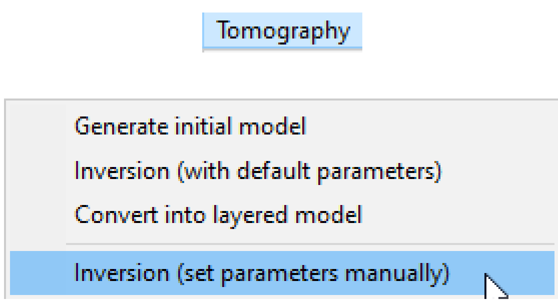


Figure 139: Layered model from tomographic inversion.

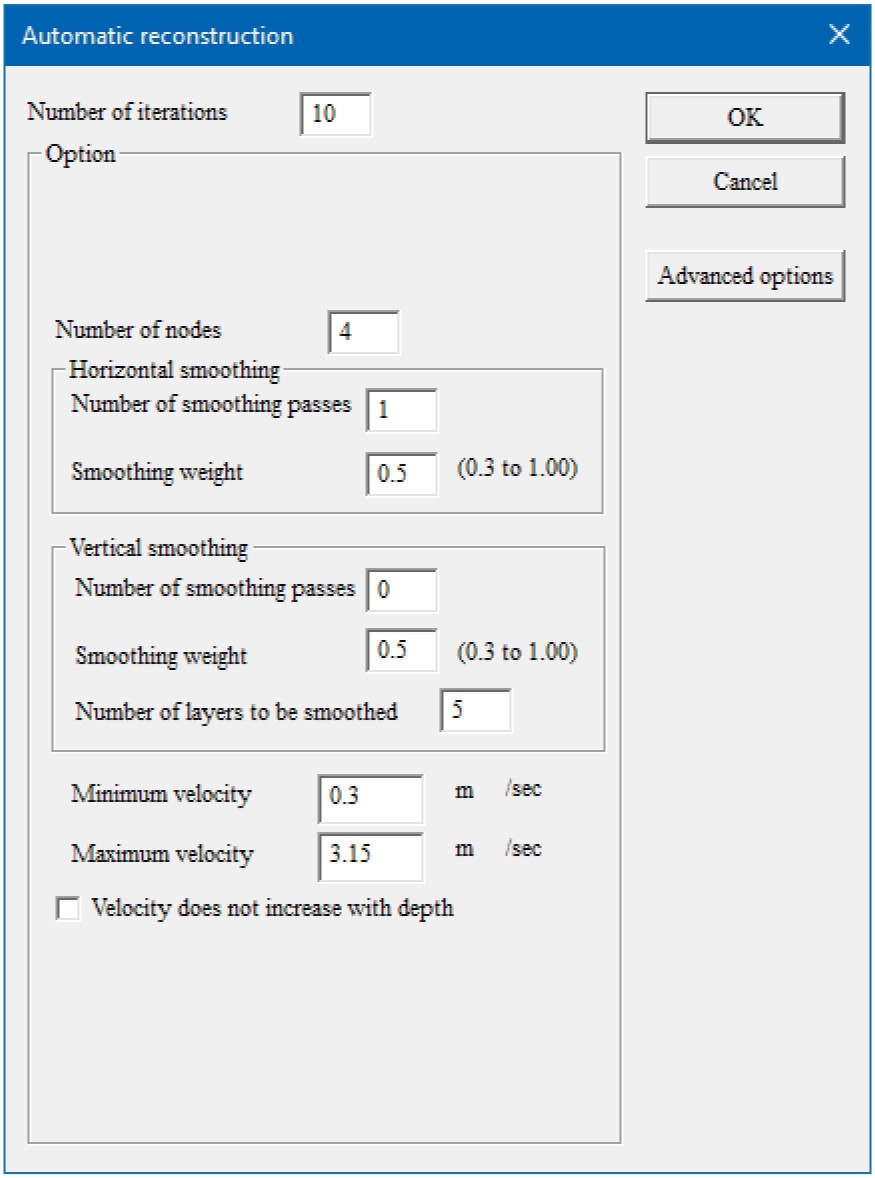
This procedure can be useful in improving the quality of any layered inversion, particularly when layer assignments are difficult.

Note: *If you wish to keep the tomographic inversion, you must save the Plotrefa file before you convert to a layered model.*

4.8.4 INVERSION (SET PARAMETERS MANUALLY)



If the tomographic inversion achieved with the default parameters needs improvement, you may modify the tomographic inversion parameters and try again. To do this, click on *Inversion (set parameters manually)*:



Number of iterations: The number of iterations defaults to 10. In general, the better the initial model, the fewer iterations required to arrive at an acceptable solution. If you are unsure about the quality of the initial model, you might want to compensate by increasing the number of iterations.

Note: *The number of iterations setting applies to each inversion, and subsequent inversions are cumulative. For example, if this parameter is set to 10, and after 10 iterations you decide to change one of the inversion parameters and run the inversion again, the cumulative number of inversions will be 20.*

Number of nodes: Tomography divides the velocity model into cells of constant velocity, and then traces rays through the model (see Appendix F). The number of nodes defines the density of rays – the more nodes, the more rays (and the longer the inversion takes). The corner of each cell is a node. In addition, there can be nodes along the sides of each cell. The number of nodes *per side* is what we refer to when we talk of the number of nodes. In the cell shown below, the number of nodes is one.

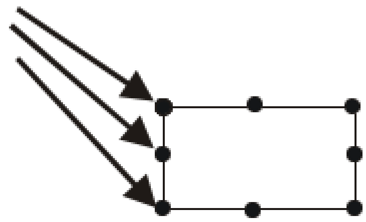


Figure 140: Single-node velocity cell.

The default value is three, as shown below.

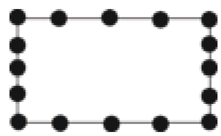


Figure 141: Three-node velocity cell.

Horizontal/Vertical Smoothing: It is generally desirable to apply some smoothing of the cell velocities, for two reasons: 1) it tends to produce a more pleasing velocity plot, and 2) it removes the inevitable small-scale velocity artifacts that might otherwise be interpreted as real. On the other hand, if you have extremely high-quality, redundant data, you may want to avoid smoothing so as *not* to obscure small-scale variations. In most cases, the default values will be suitable. Smoothing is accomplished by applying a three-term, weighted moving-average filter to the velocity cells. Smoothing in the horizontal and vertical directions is done independently.

Number of smoothing passes: This parameter controls the number of times the weighted average is applied in any one direction. You may run the same filter more than once. The more passes, the more smoothing.

Smoothing weight: This is the weight of the center term in the moving average. The basic filter equation is as follows:

$$V_2 = W_1 V_1 + W_2 V_2 + W_3 V_3$$

where $W_1 = W_3$ *and* $W_1 + W_2 + W_3 = 1$.

The default value of 0.5 for W_2 therefore weights the center term twice as much as the other two.

Note: *The* ***greater*** *the smoothing weight, the* ***less*** *the model will be smoothed. A smoothing weight of one will result in no smoothing at all. You may set the number of smoothing passes to zero if you wish not to smooth the model.*

Number of layers to be smoothed: This applies to vertical smoothing only. Since the resolving capabilities of any geophysical technique, including seismic tomography, tend to decrease with depth, it is often desirable to smooth the bottom layers more than the top layers. This parameter determines the number of layers *from the bottom* of the model to be smoothed. For instance, if the tomogram has 15 layers, setting this parameter to five will result in the bottom five layers being smoothed.

Minimum/maximum velocity: See above for explanation. If the match between observed and calculated data is poor, it may be that the minimum and maximum velocities need to be decreased and increased, respectively. You may change them in this dialog box and do the inversion again.

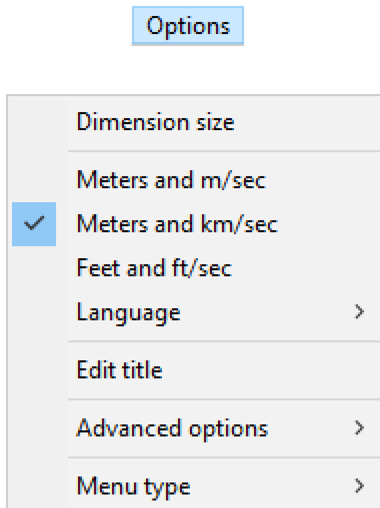
Note: *If you used the time-term model as your initial model, the minimum and maximum velocities in this dialog box will match those of the time-term inversion until you override them.*

Velocity vs. depth: In any surface refraction inversion technique, including tomography, it must be assumed that velocity increases with depth. However, this is not true in surface-to-borehole and borehole-to-borehole tomographic surveys. If you are doing a borehole survey, de-select *Velocity does not increase with depth*.

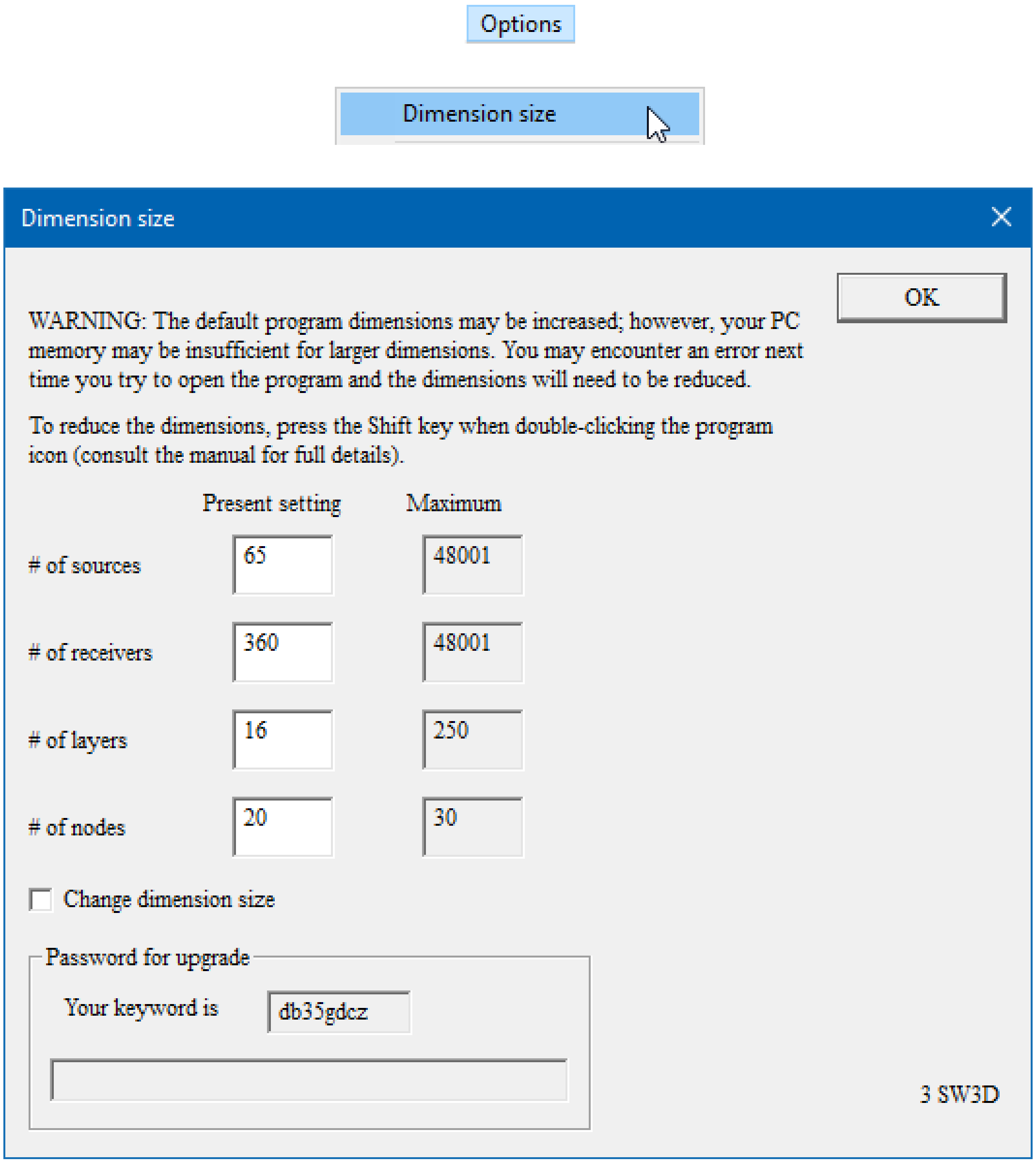
Note: *If you de-select the above parameter, run an inversion, and then decide to run a second inversion, be sure to de-select the parameter again, as it is selected by default.*

4.9 OPTIONS MENU

Click on *Options* to reveal the **Options** menu:



4.9.1 Dimension Size

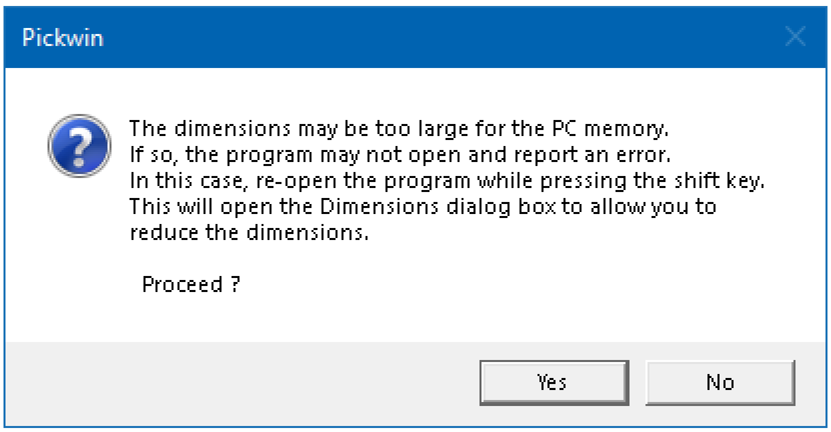


You must make sure that the program is dimensioned large enough for the dataset you are working with. Setting the values too small will result in errors. On the other hand, it is best not to set them much bigger than you really need, because memory is set aside to accommodate these dimensions. It is best to set them large enough, but not much larger than required. This is less of an issue than it used to be since most computers now come with Gb of memory. Still, it is best to set reasonable limits.

Note: *For changes to become effective, you must check the* ***Change dimension size*** *checkbox.*

__Note:__ The maximum allowable dimensions are displayed and are a function of which version of SeisImager/2D you purchased. If you choose to upgrade, we can provide a password that will increase the maximum dimension sizes.

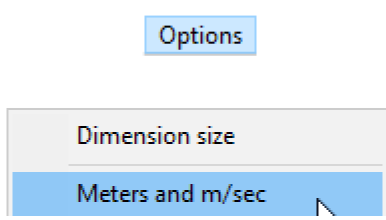
__Note:__ If a very large value is entered, a warning message will appear before you are allowed to restart the program. It is recommended that you do not proceed; select **No** *and reduce the dimensions.*



If you proceed and indeed the PC has insufficient memory, the program will no longer be able to open. To lower the values and recover the program, open the **Dimension size** dialog box directly by pressing the *Shift* key while double-clicking the program icon.

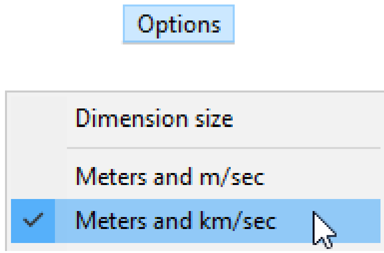
__Note:__ Sometimes the program will simply crash rather than post the above message. If that happens, use the procedure outlined on Page 3 to restore the system defaults.

4.9.2 METERS AND M/SEC



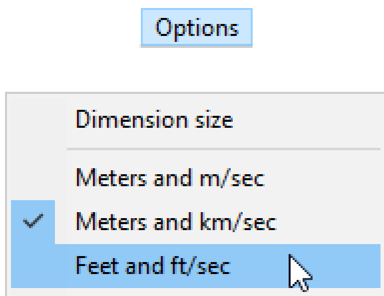
This enables metric units of distance and velocity (meters and meters per second).

4.9.3 METERS AND KM/SEC



This enables metric units of distance and velocity (meters and kilometers per second).

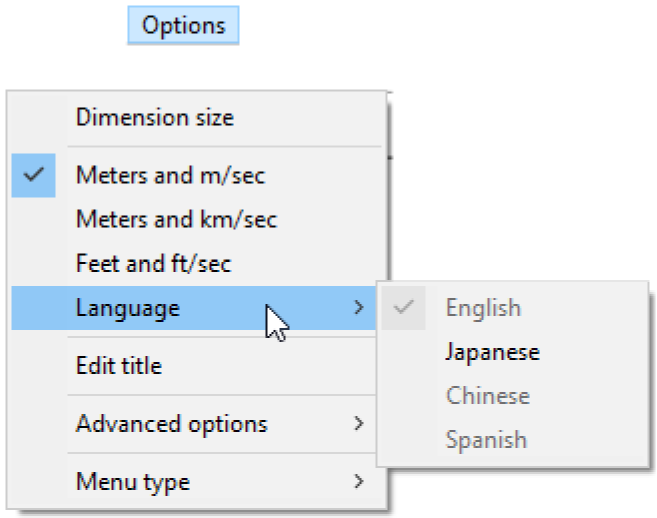
4.9.4 FEET AND FT/SEC



This enables British units of distance and velocity (feet and feet per second).

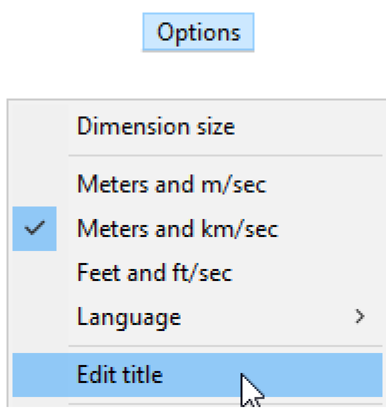
4.9.5 LANGUAGE

Choose your language:

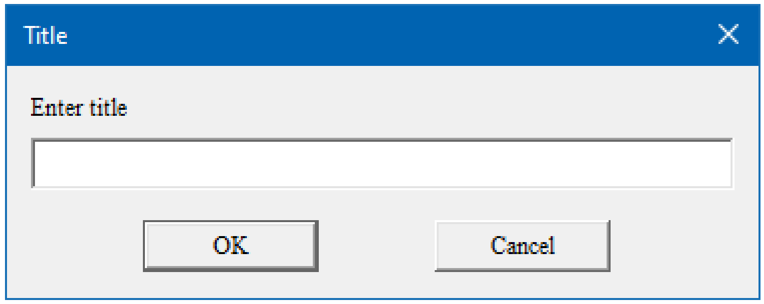


As of this writing (July 2024), Chinese and Spanish were under construction.

4.9.6 EDIT TITLE

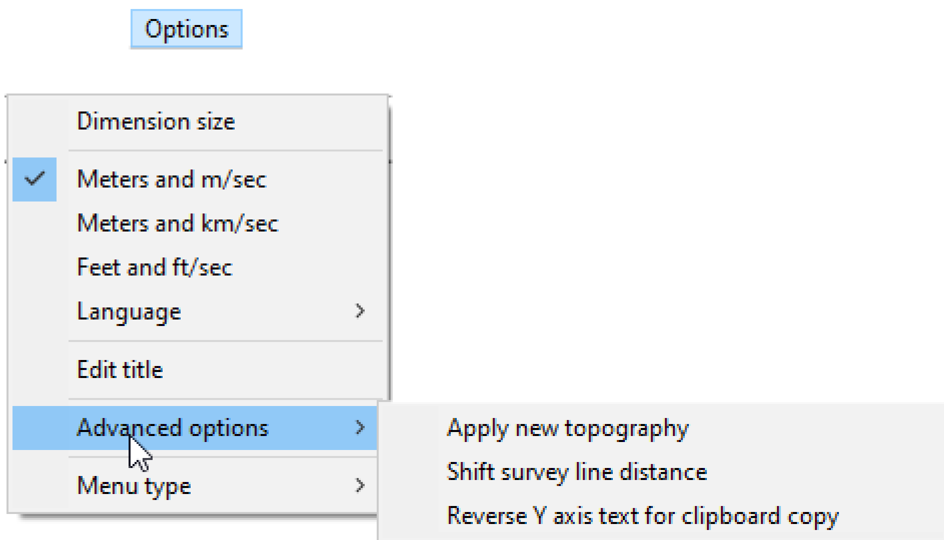


Clicking on *Edit title* will reveal the following dialog box:

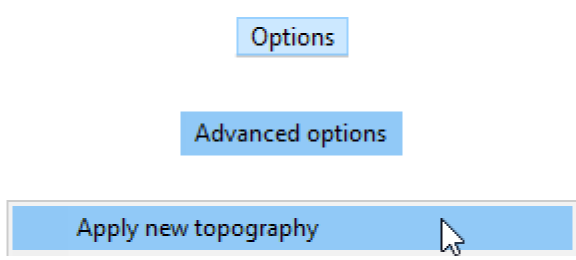


Enter the title you wish to have displayed on your output and press *OK*.

4.9.7 ADVANCED OPTIONS

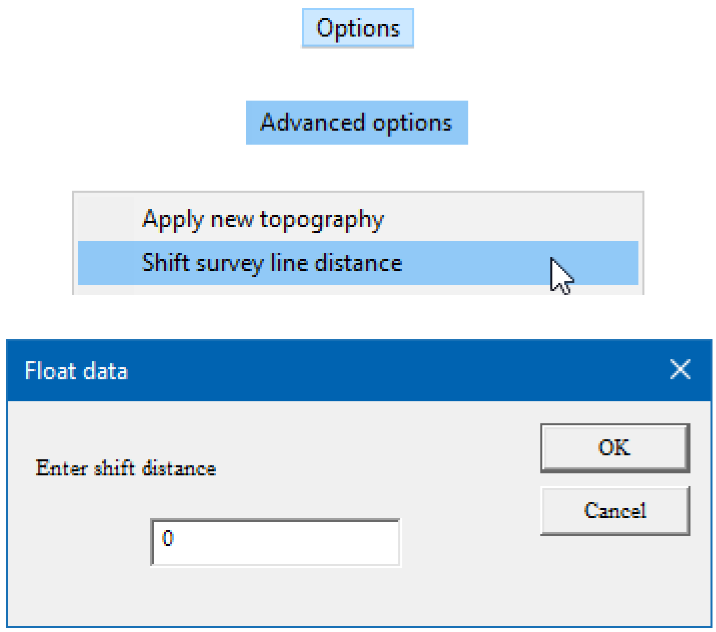


4.9.7.1 APPLY NEW TOPOGRAPHY



You may use this option to read in an elevation file. This will replace any topography data you might have read previously. See Section 4.1.8, Page 133. You will be prompted for a file name.

4.9.7.2 Shift Survey Line Distance



This option allows you to advance or retreat the survey line. For example, the spread below (left) was advanced by 100 feet (right).

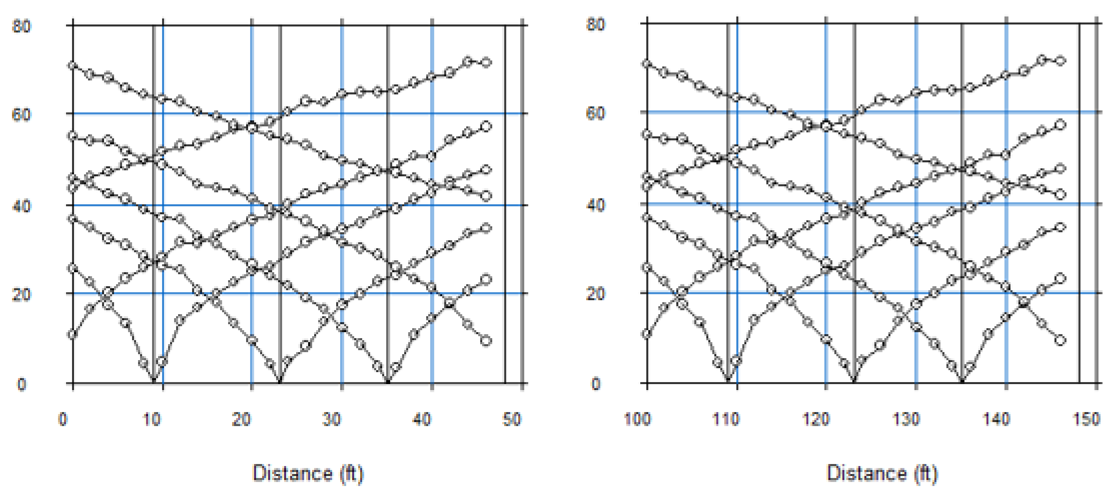
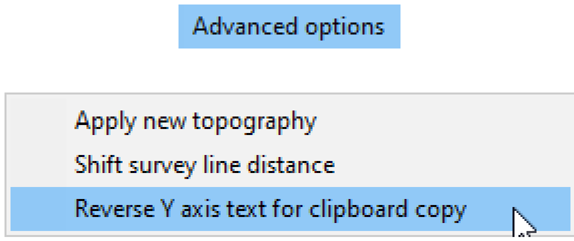


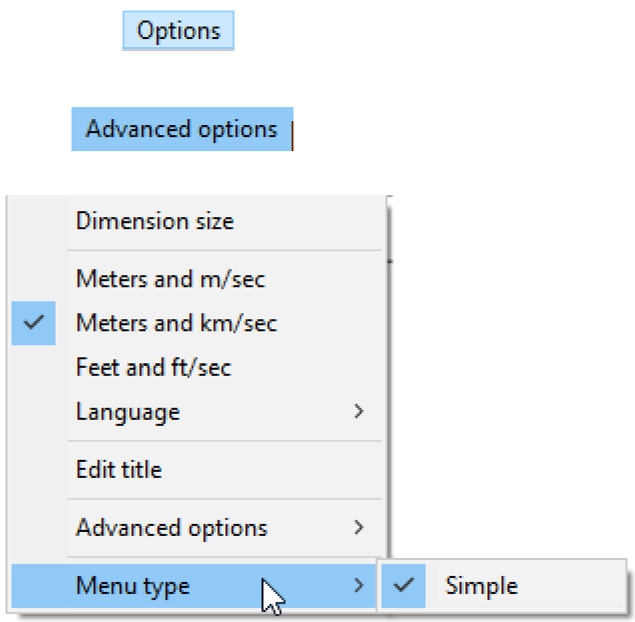
Figure 142: Advancement of seismic spread.

4.9.7.3 Reverse Y Axis Text for Clipboard Copy

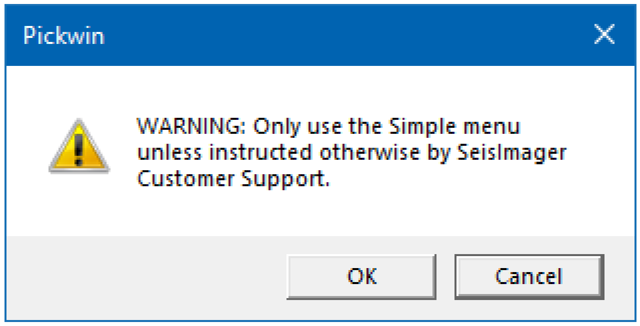


Reverse Y-axis text for clipboard copy controls how the label for the vertical axis appears when *Edit | Copy to clipboard* is selected. If *Reverse Y-axis text for clipboard copy* is selected, the vertical axis label appears written top to bottom. If not selected (the default), the vertical axis label appears written bottom to top.

4.9.8 Menu Type



SeisImager includes advanced options for very special applications that are only available if *Simple* is unchecked. These are undocumented and should only be used under supervision from Geometrics. If *Simple* is unchecked, you will see the following warning:

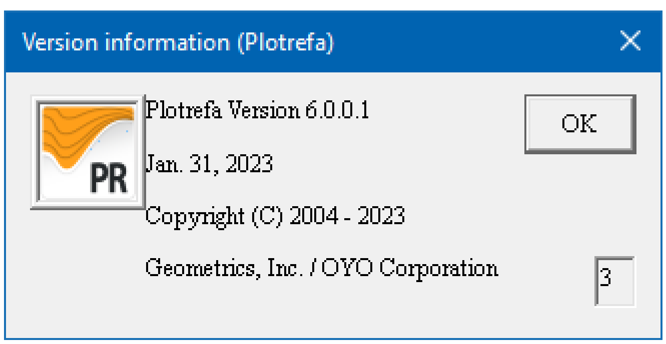


Pressing *OK* above will give you access to these features, most of which are in the **Options** menu. To return to the **Simple** menu, select *Option* | *etc.* | *Menu type* | *Simple*.

4.10 HELP MENU

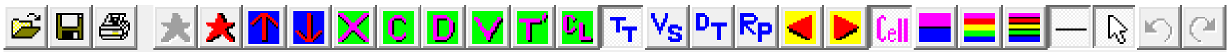
Continue.

4.10.1 VERSION INFO



This displays the software version.

4.11 Tool Buttons



Like Pickwin, Plotrefa includes numerous tool buttons, which are detailed below. Some have been mentioned before.

4.11.1 Open Plotrefa File Tool Button []

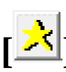
Use this button to open a Plotrefa (.vs) file. It is the same as selecting *File | Open Plotrefa File (travel time data and velocity model)*. See Section 4.1.1, Page 126.

4.11.2 Save Plotrefa File Tool Button []

This button saves a Plotrefa file. It is the same as selecting *File | Save Plotrefa File.* See Section 4.1.4, Page 130.

4.11.3 Print Figure Tool Button []

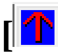
Select this button to print whatever is on the screen. It is the same as choosing *File | Print*. See Section 4.1.16, Page 146.

4.11.4 Update Travel time Curve Tool Button []

This button updates the travel time curve in Plotrefa when first arrivals are changed in Pickwin. See Section 3.4.6, Page 91.

4.11.5 Calculate Theoretical Travel times and Raypaths Tool Button []

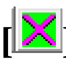
Selecting this button will calculate the theoretical travel times and raypaths for the velocity model. It is the same as choosing *Raytracing | Execute*. See Section 4.7.1, Page 270.

4.11.6 Enlarge Scale Tool Button []

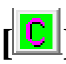
This button increases the scale of the travel time plot and velocity model. Also see Section 4.4.9, Page 237.

4.11.7 Reduce Scale Tool Button []

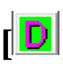
This button decreases the scale of the travel time plot and velocity model. Also see Section 4.4.9, Page 237.

4.11.8 Exit Current Editing Mode Tool Button []

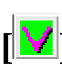
Select this button to exit the editing mode you might be in. It is the same as selecting *Velocity Model* | *Exit Edit Mode*. See Section 0, Page 212.

4.11.9 Correct Travel times Tool Button []

This tool button enables the editing of travel times in the travel time plot. It is the same as selecting *Travel time curve* | *Modify travel times (all shots)*. See Section 4.2.4, Page 156.

4.11.10 Calculate Difference-time Curve Tool Button []

Press this button to see the difference-time curve. It is the same as selecting *Travel time curve* | *Calculate travel time difference curve*. See Section 4.2.7, Page 160.

4.11.11 Set Velocity Line Tool Button []

Press this tool button to draw a velocity line on the travel time plot (see Page 155). It is the same as selecting *Reciprocal method* | *Set velocity line*. See Section 4.6.6, Page 251.

4.11.12 Set T' Curve Tool Button []

Press the T' tool button to draw a T' curve on the travel time plot. It is the same as selecting

Reciprocal Method | *Set up T' (½ T_{ab} calculated automatically)*. See Section 4.6.2, Page 247.

4.11.13 Correct Layer Boundary by Mouse Tool Button [ ]

This tool button allows you to edit velocity layer boundaries. It is the same as selecting *Velocity model* | *Modify layer boundary (point by point)*. See Section 0, Page 204.

4.11.14 Show Travel time Curves Tool Button [ ]

Pressing this button will display the travel time curves on the screen. It is the same as selecting *View* | *Show travel time curves*. See Section 4.4.5, Page 234.

4.11.15 Show Velocity Model Tool Button [ ]

Press this button to display the velocity model. It is the same as selecting *View* | *Show Velocity Model*. See Section 4.4.6, Page 234.

4.11.16 Show Time-term Tool Button [ ]

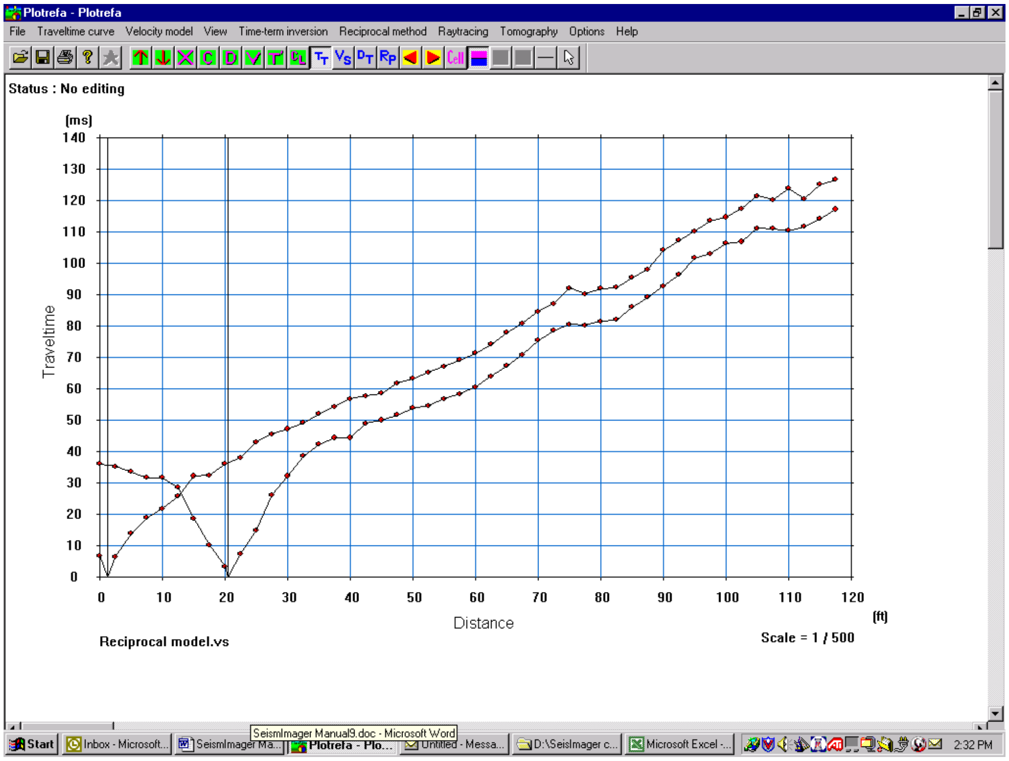
The time-term (delay-time) tool button displays the delay-times. It is the same as selecting *View* | *Show time-term*. See Section 0, Page 235.

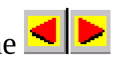
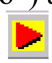
4.11.17 Show Raypath Tool Button [ ]

Choosing the "Show Raypath" tool button will display the seismic raypaths superimposed on the velocity model. It is the same as selecting *View* | *Show raypath*. See Section 0, Page 236.

4.11.18 Scroll Tool Buttons [ ]

If you have numerous shots in a spread, you may find it convenient to only display a subset of them at any given time. As explained in Section 4.4.1 (Page 231), you can control the number of displayed shots in the **Axis configuration (manual)** sub-menu. Using our dataset as an example, we will show only two travel time curves at a time:



The left-most two shots ("shot one" and "shot two") are displayed. We may now use the  buttons to scroll through the spread. Pressing the  button once will display shots two and three:

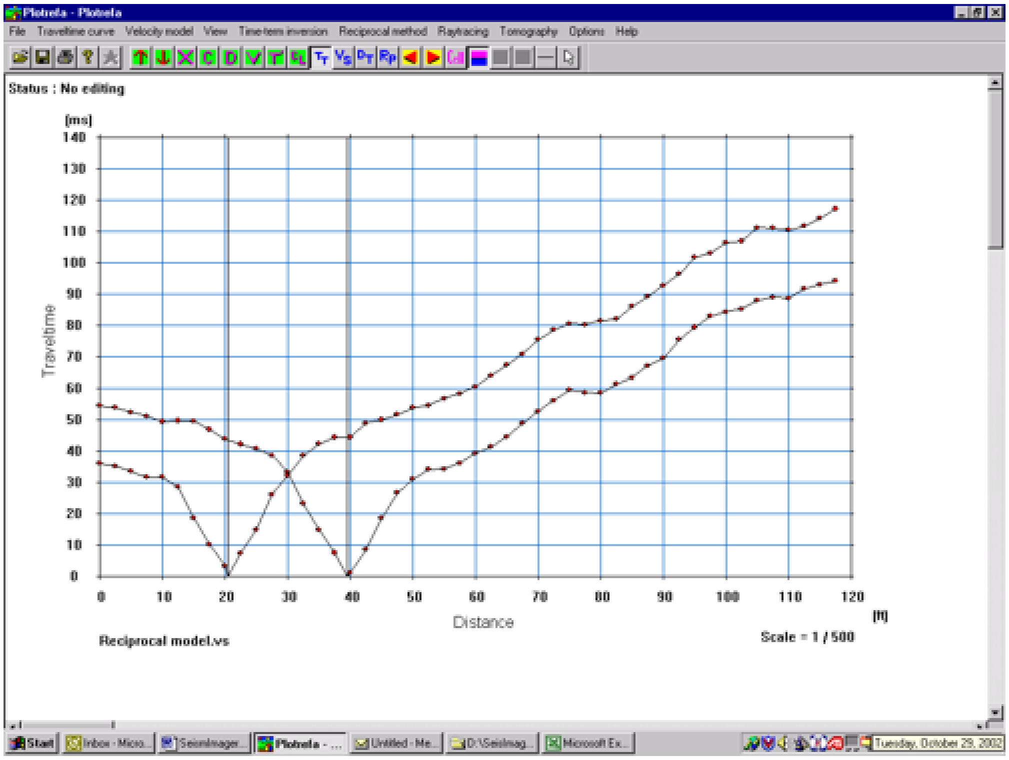


Figure 143: Displaying data for only shots two and three.

The number of shots added or removed with each press of the button is always one less than the total number of shots displayed. If we display three shots at a time, as shown below,

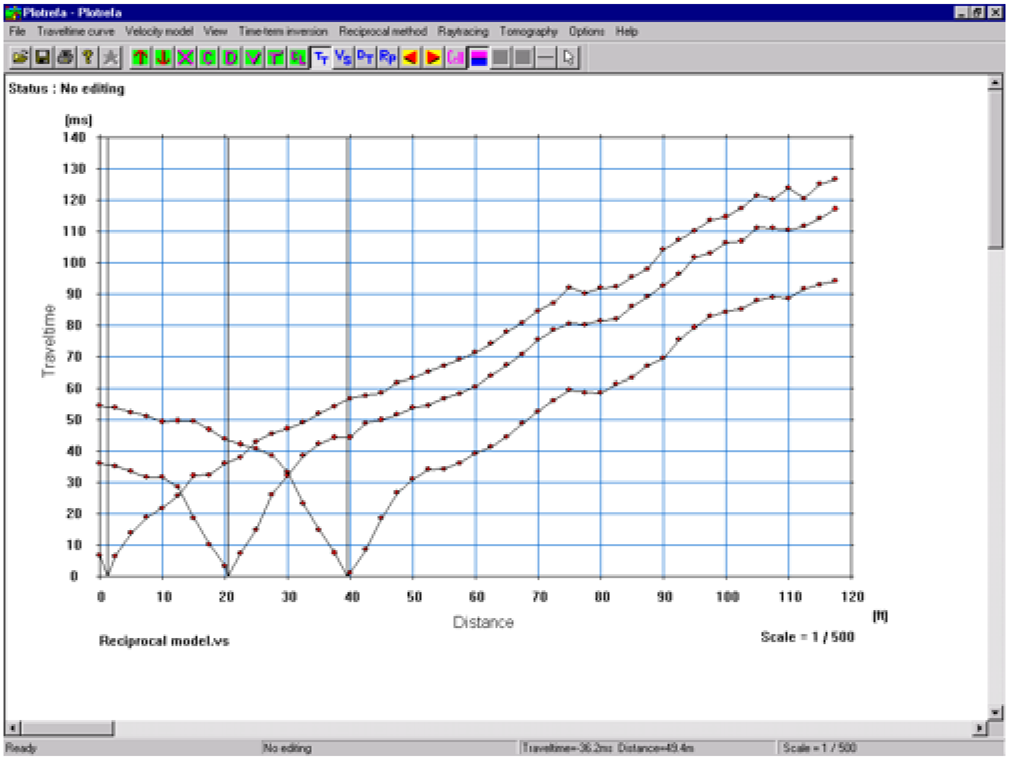
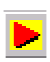


Figure 144: Displaying data for shots one, two, and three.

pressing the  button once will result in shots 3, 4, and 5 being displayed:

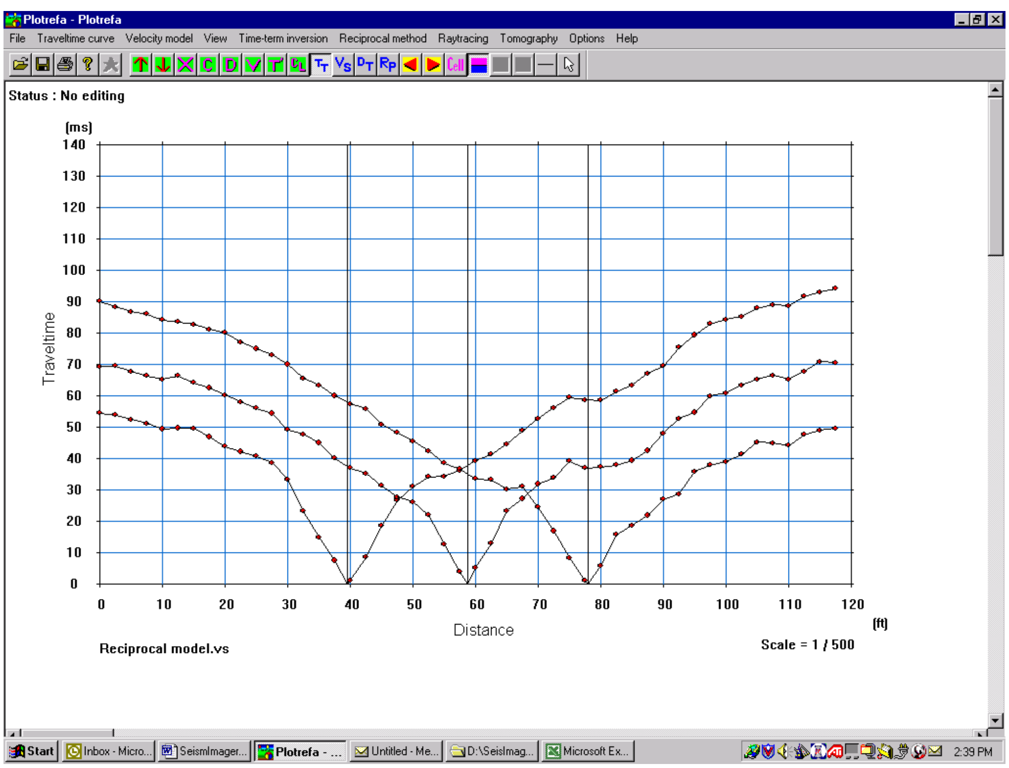


Figure 145: Displaying data for shots three, four, and five.

4.11.19 Show Cell Boundaries Tool Button [ ].

This tool button allows you to display the velocity cell boundaries. It is the same as selecting *Velocity model* | *Show cell boundaries.* See Section 4.3.9, Page 197.

4.11.20 Show Velocity Model as Cells Tool Button [ ]

This tool button will display the velocity cells that make up the velocity model, typically a model created by a tomographic inversion. An example is shown below:

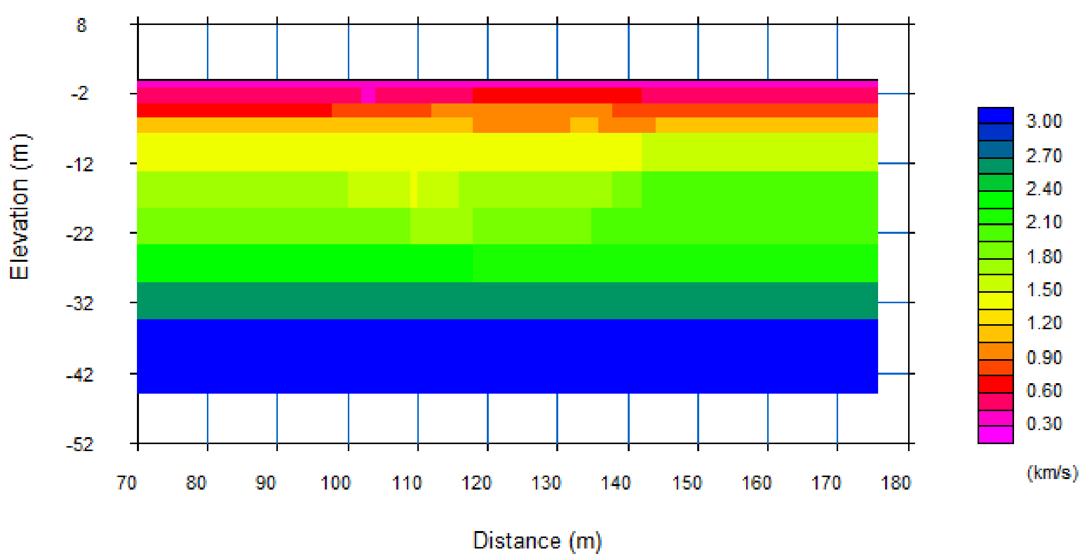


Figure 146: Displaying velocity model and cells.

4.11.21 Contour Map Tool Button [ ]

Selecting this button will display the velocity model as a contour map, which is essentially a smoothed version of the cell map above:

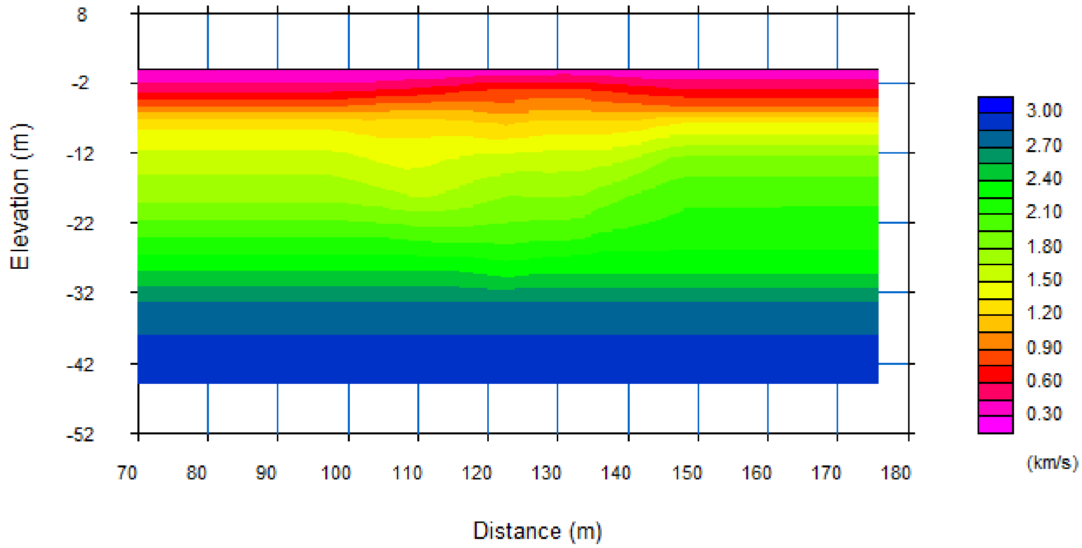


Figure 147: Displaying velocity model as contour map.

4.11.22 Contour Lines Tool Button []

To include contour lines in your velocity section, press the "Contour Lines" tool button:

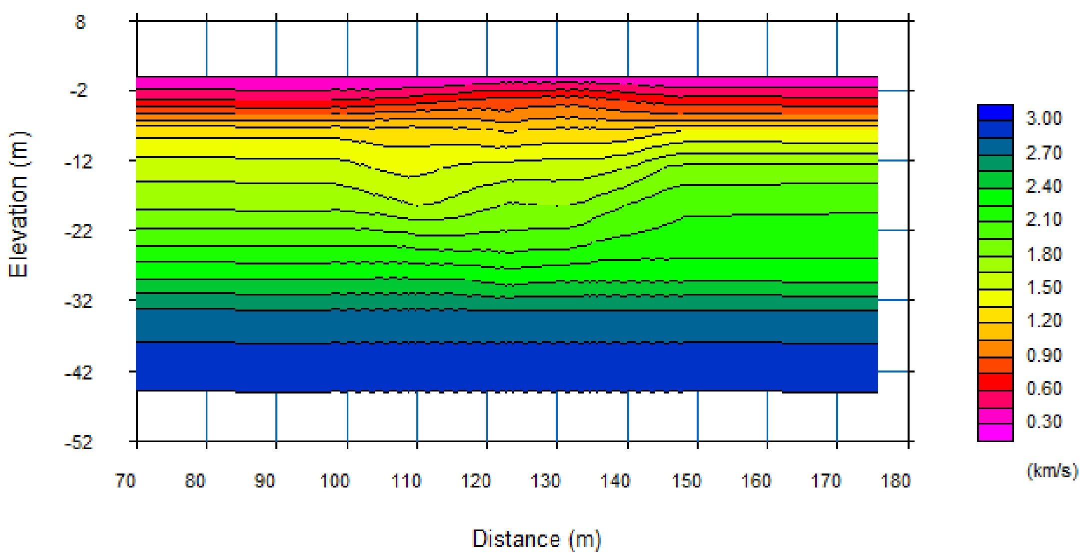


Figure 148: Displaying contour lines.

4.11.23 Velocity Boundary Lines Only Tool Button []

You may also draw your map with just velocity boundary lines (no coloring). To do so, press the "Velocity Boundary Lines Only" tool button:

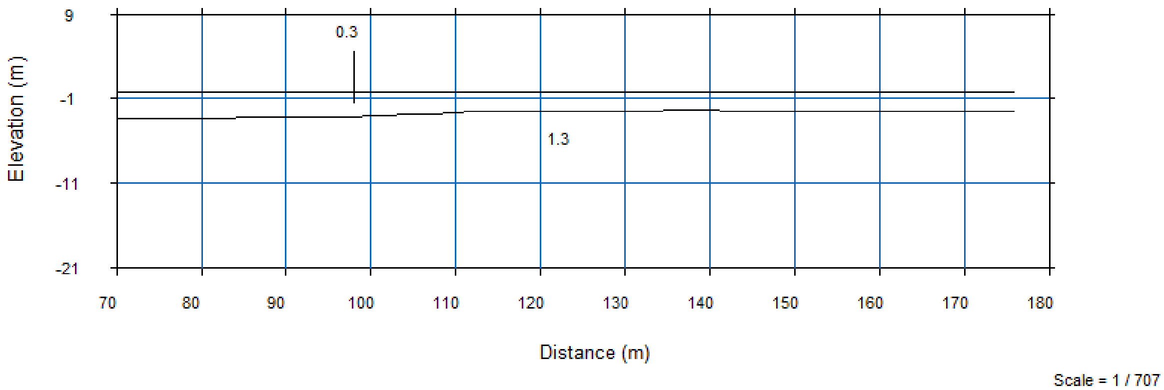


Figure 149: Displaying only velocity boundaries.

4.11.24 SELECT T' OR VELOCITY LINE TOOL BUTTON []

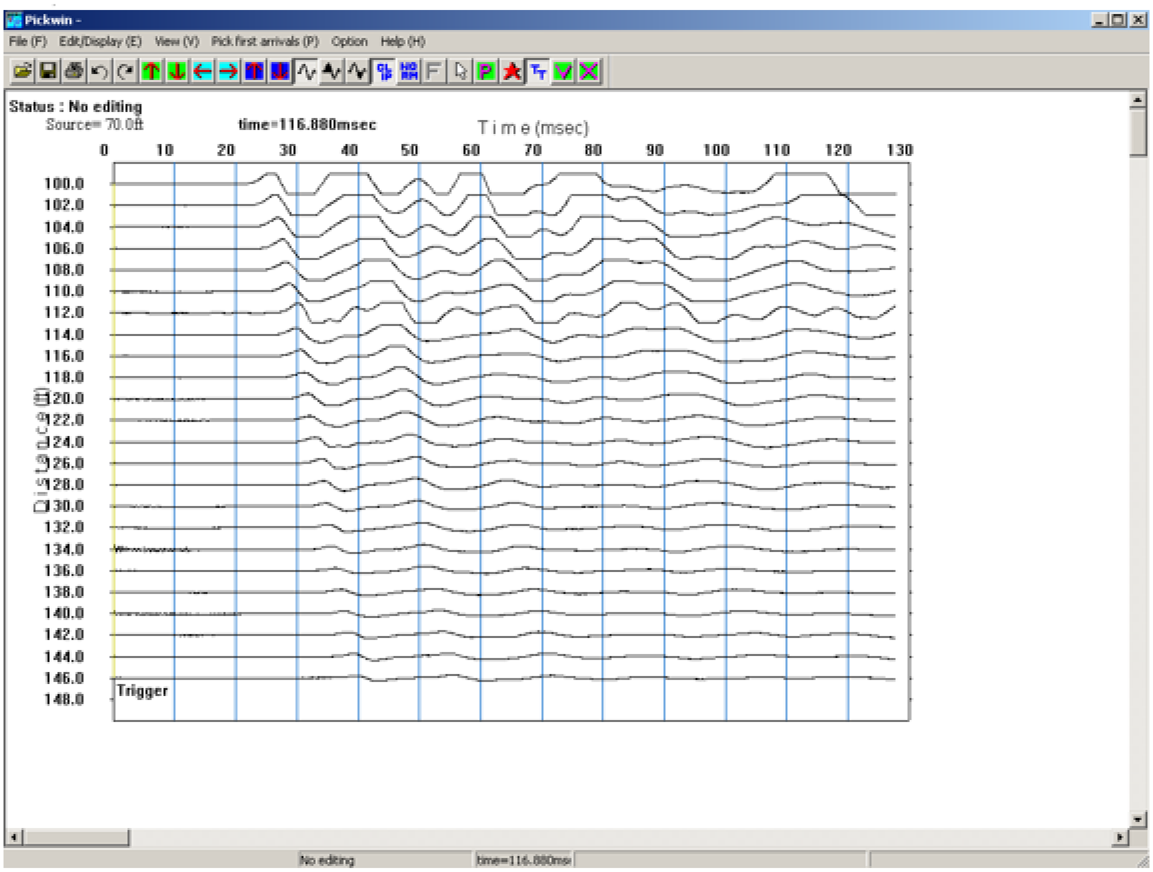
This is a “select” button. It is generally used to select travel time and velocity curves for various operations.

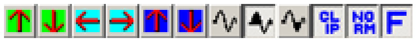
APPENDICES

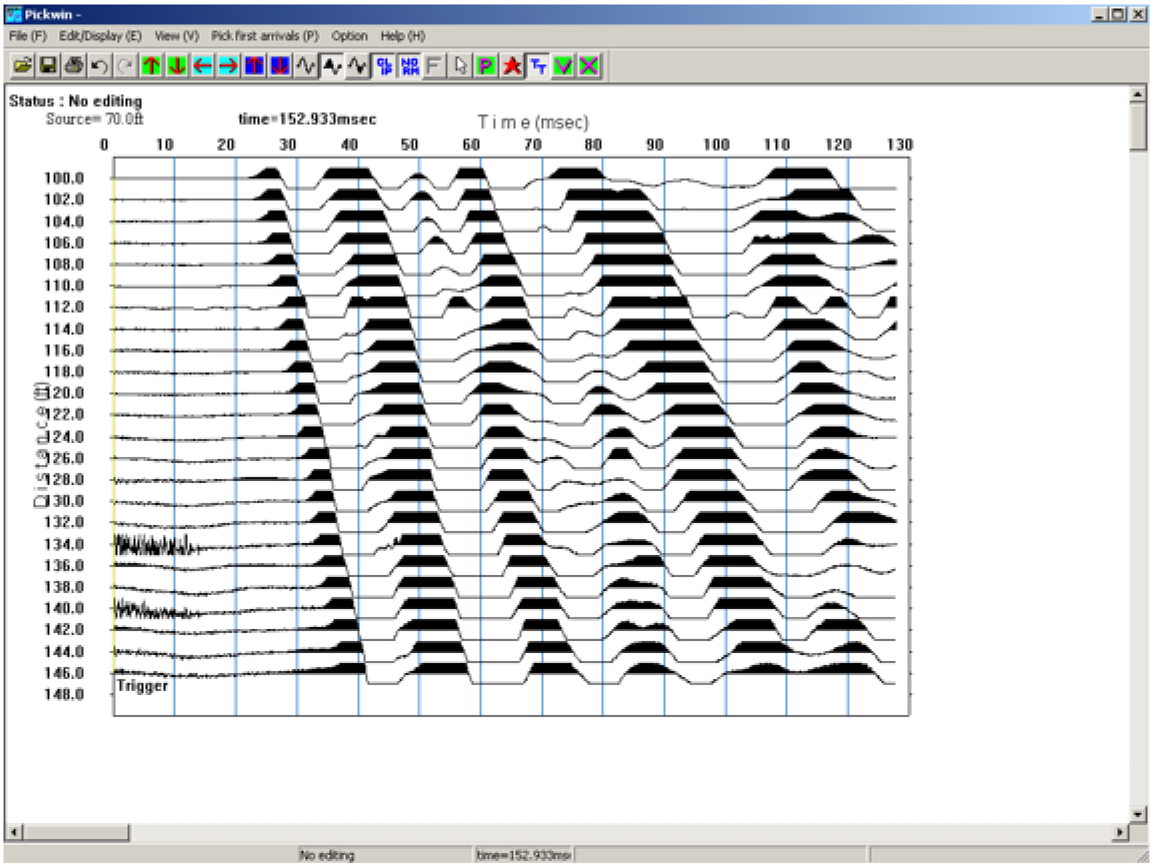
APPENDIX A PICKWIN EXAMPLES

Example 1: Pick a Simple 5-shot Spread

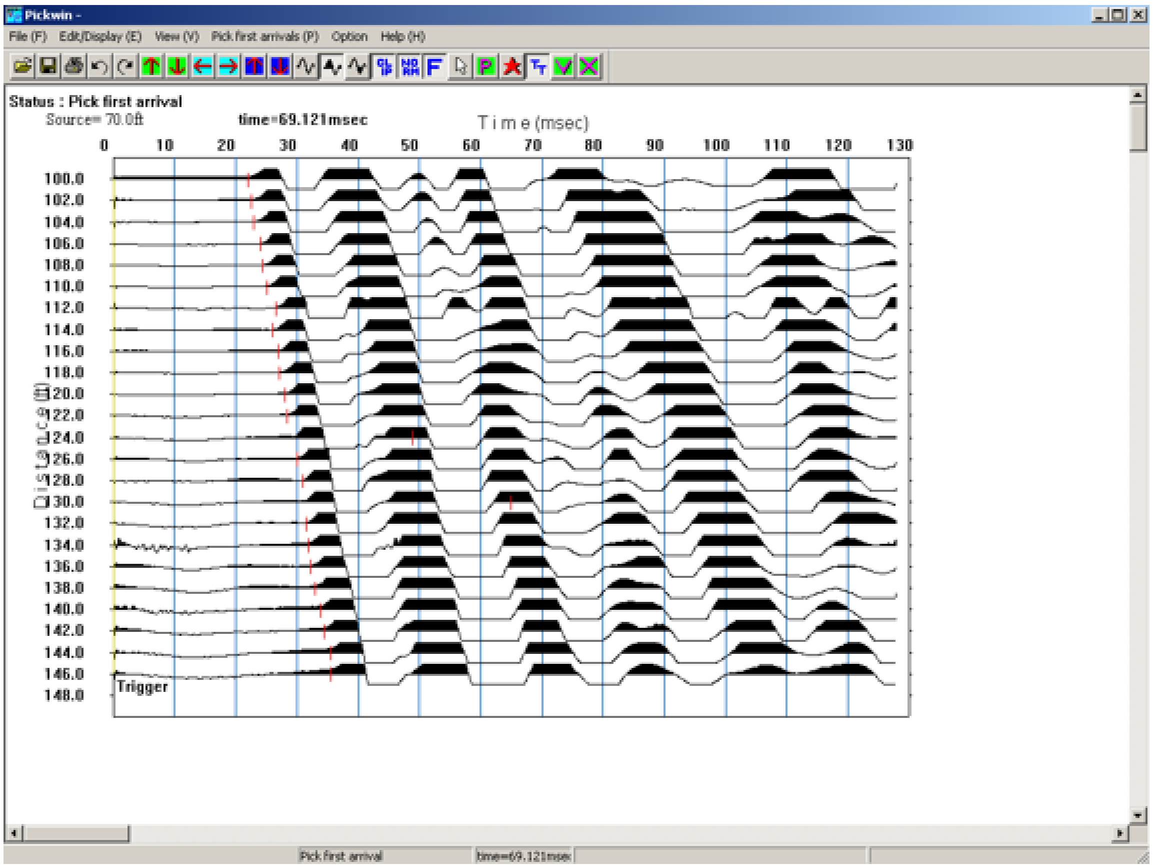
- Open a SEG-2 file: 



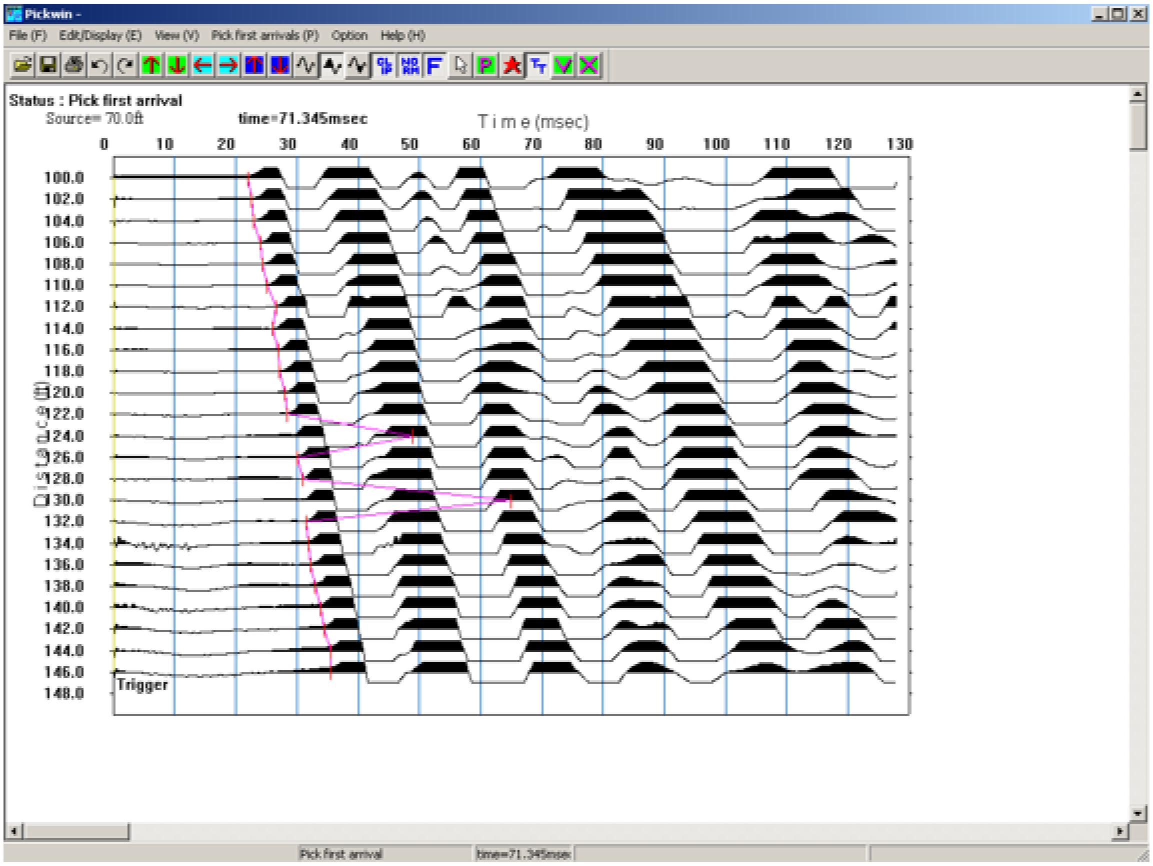
- Optimize display: 



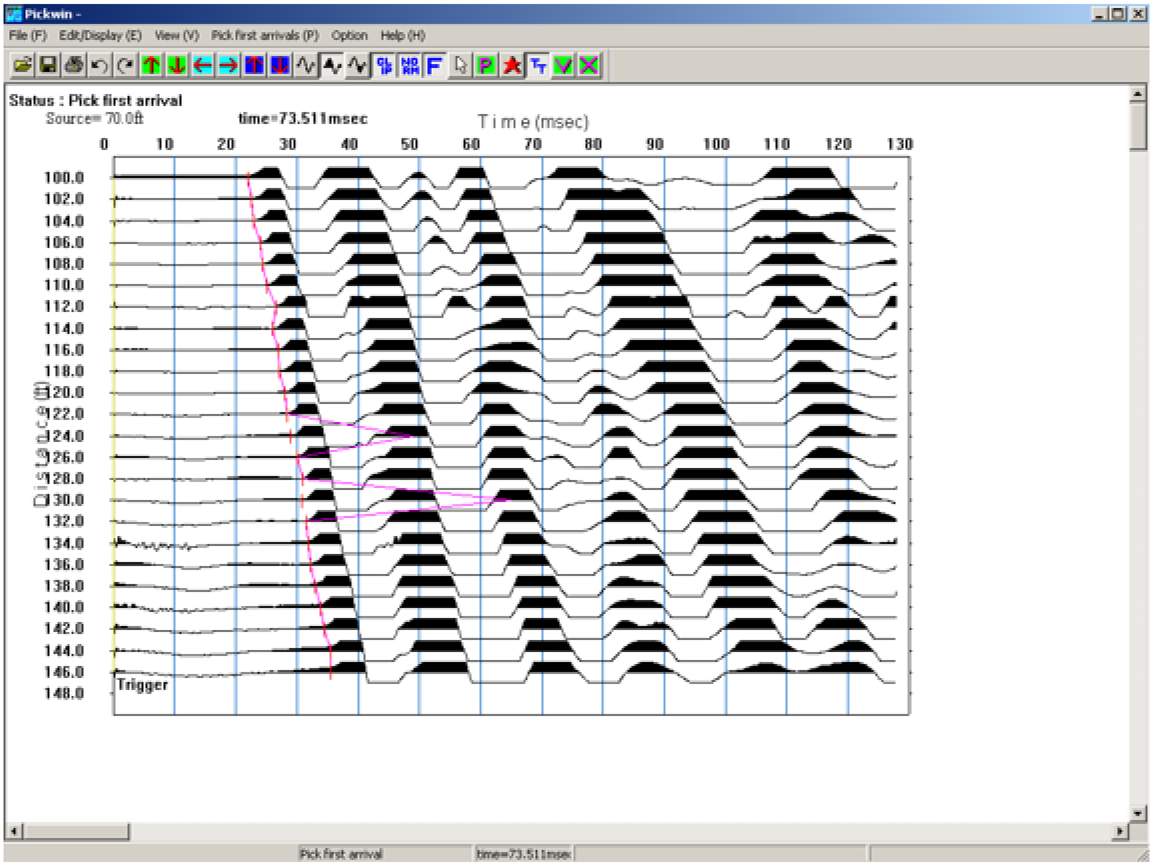
- Pick first breaks: 

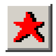


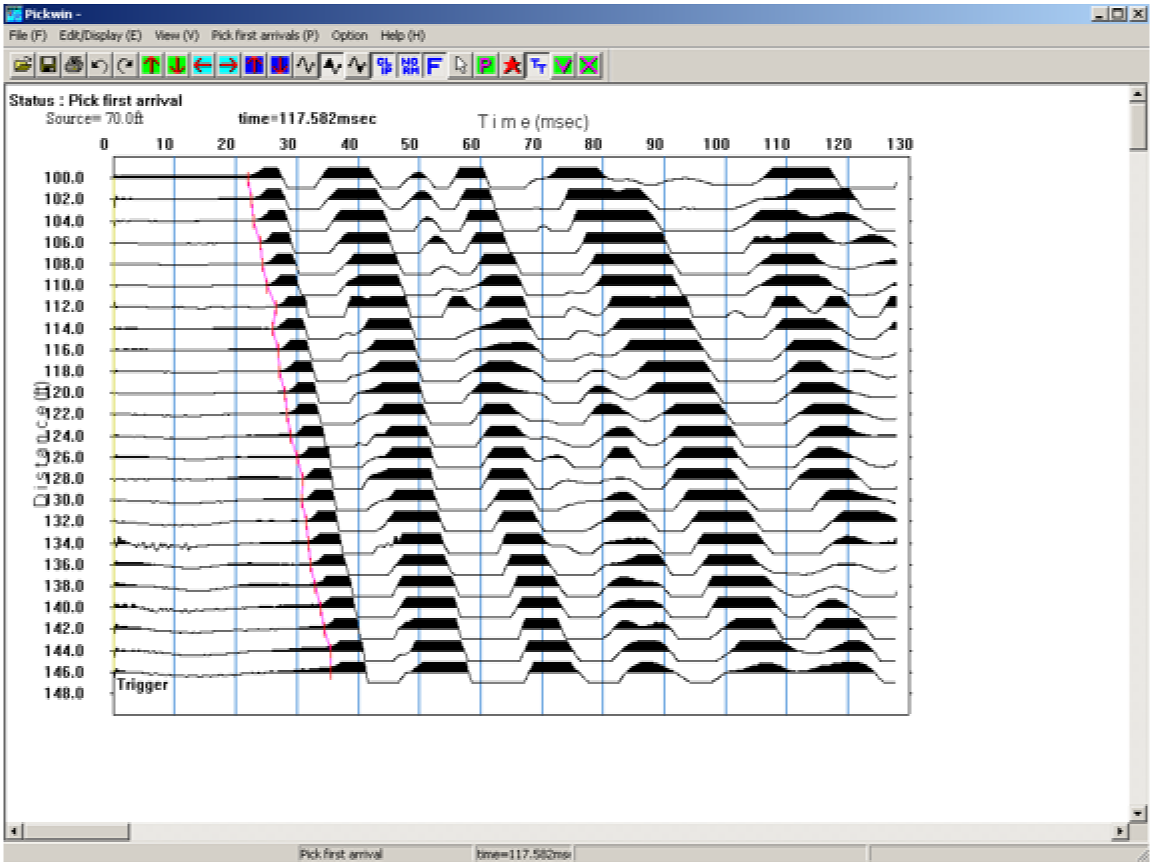
- Connect first breaks: 



- Manually adjust first breaks:

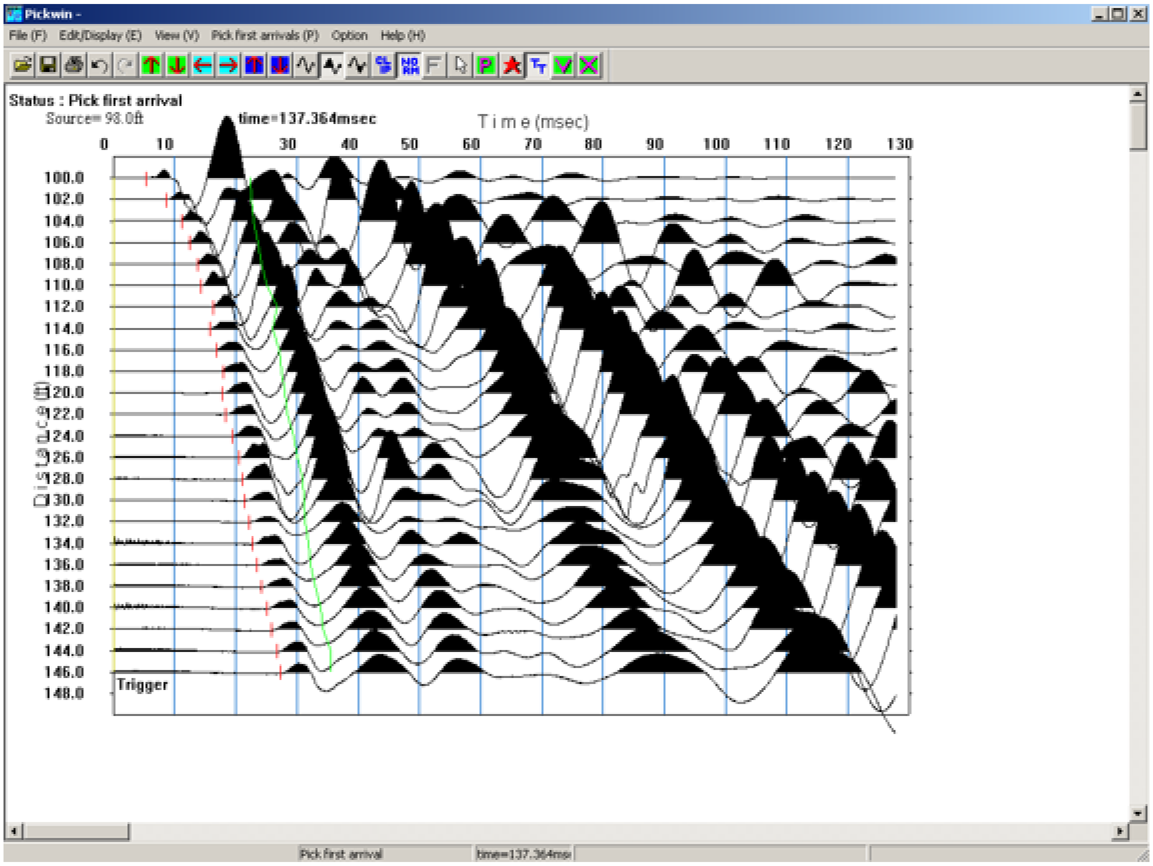


- Connect first breaks again: 

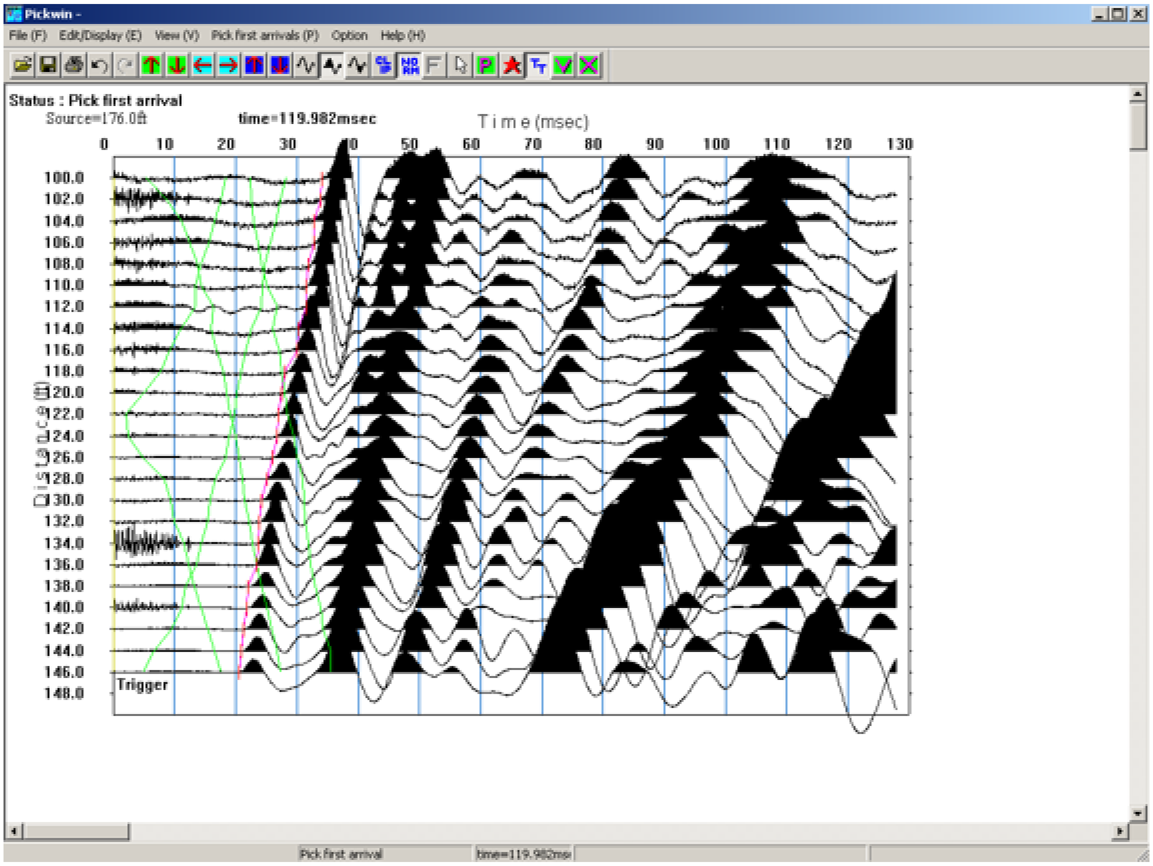


- Save first break picks file (optional but recommended).

- Read in next SEG-2 file as *New* file: 



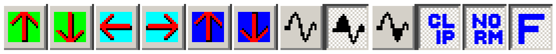
- Repeat above steps until all files have been picked:



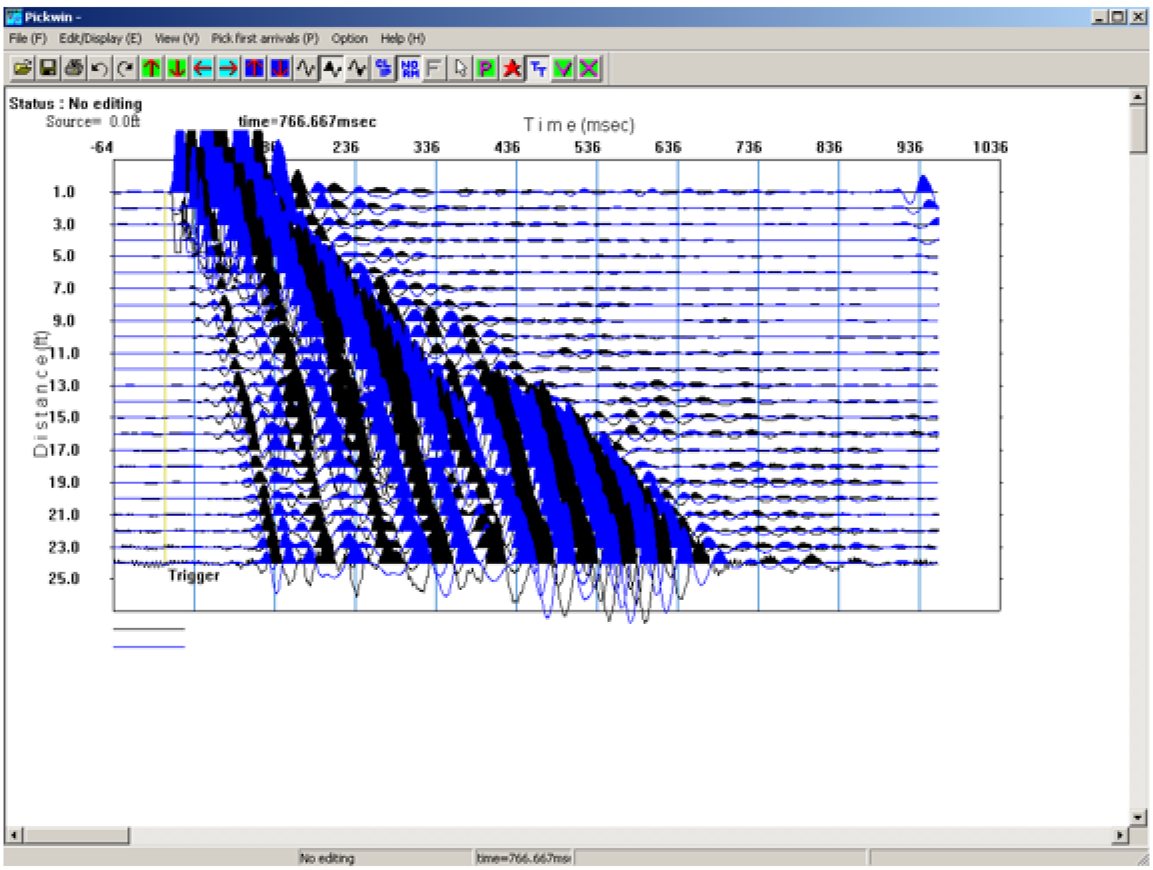
- When all files have been picked, save the first break picks file (mandatory!).

Example 2: Shear Wave Survey

1) Open a SEG-2 file: 

2) Optimize display: 

3) Open reverse-polarity record as an *appended* file (display parameters for appended will be the same as the first file):

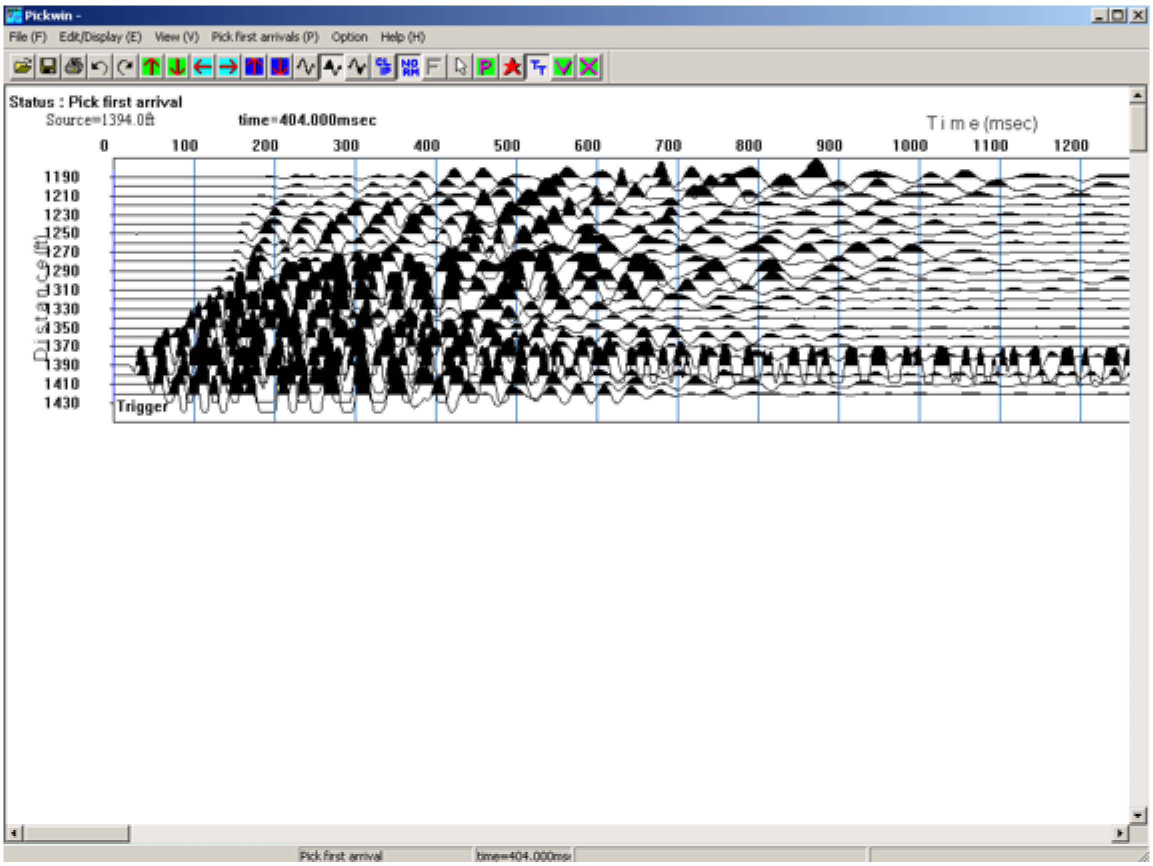


4) Pick the first breaks. 

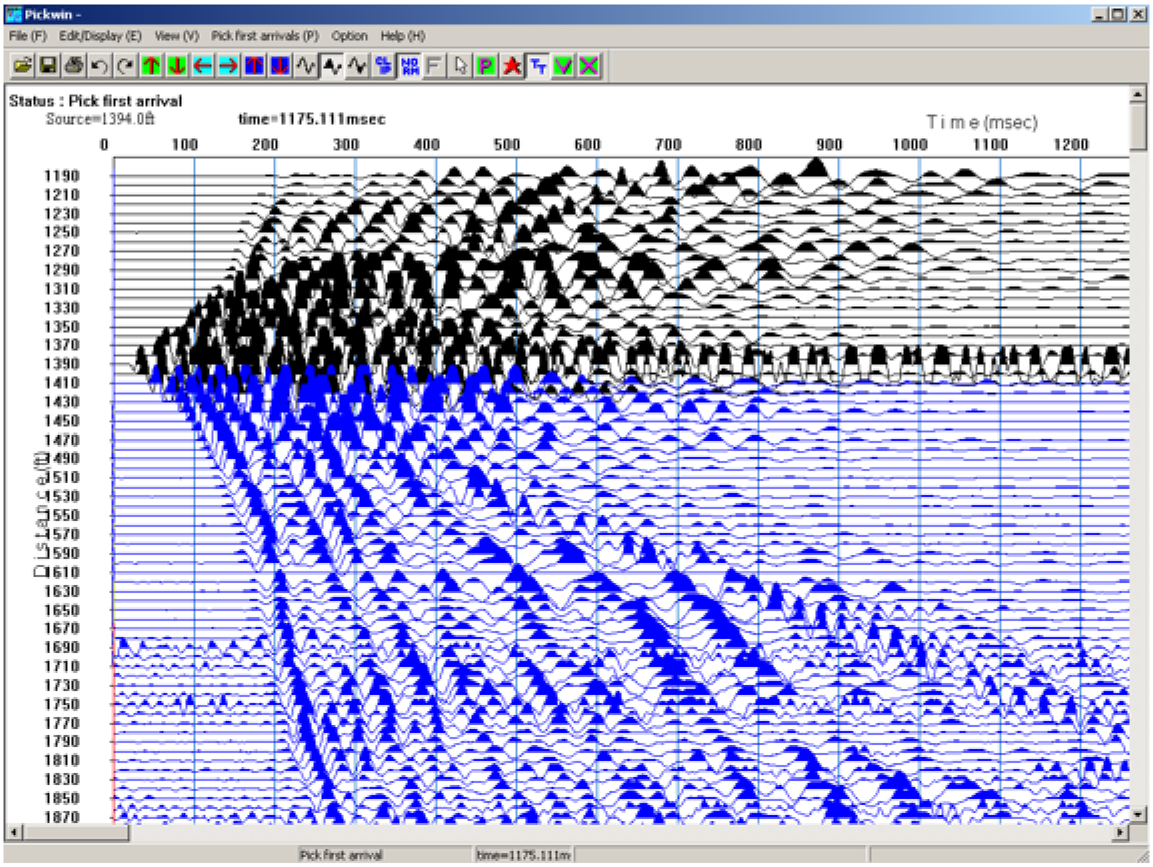
5) Follow the procedure described in Example 1 until all files have been picked.

Example 3: Multiple Spreads

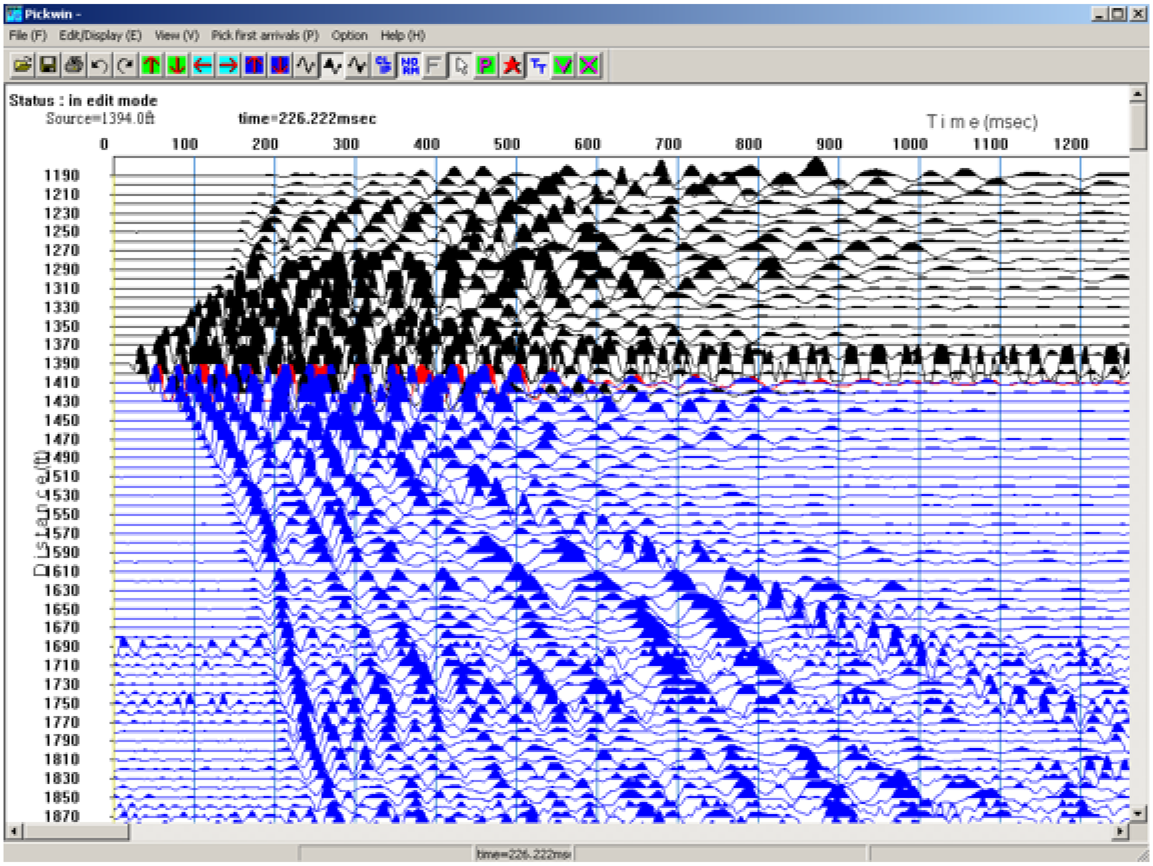
1) Open a SEG-2 file: 



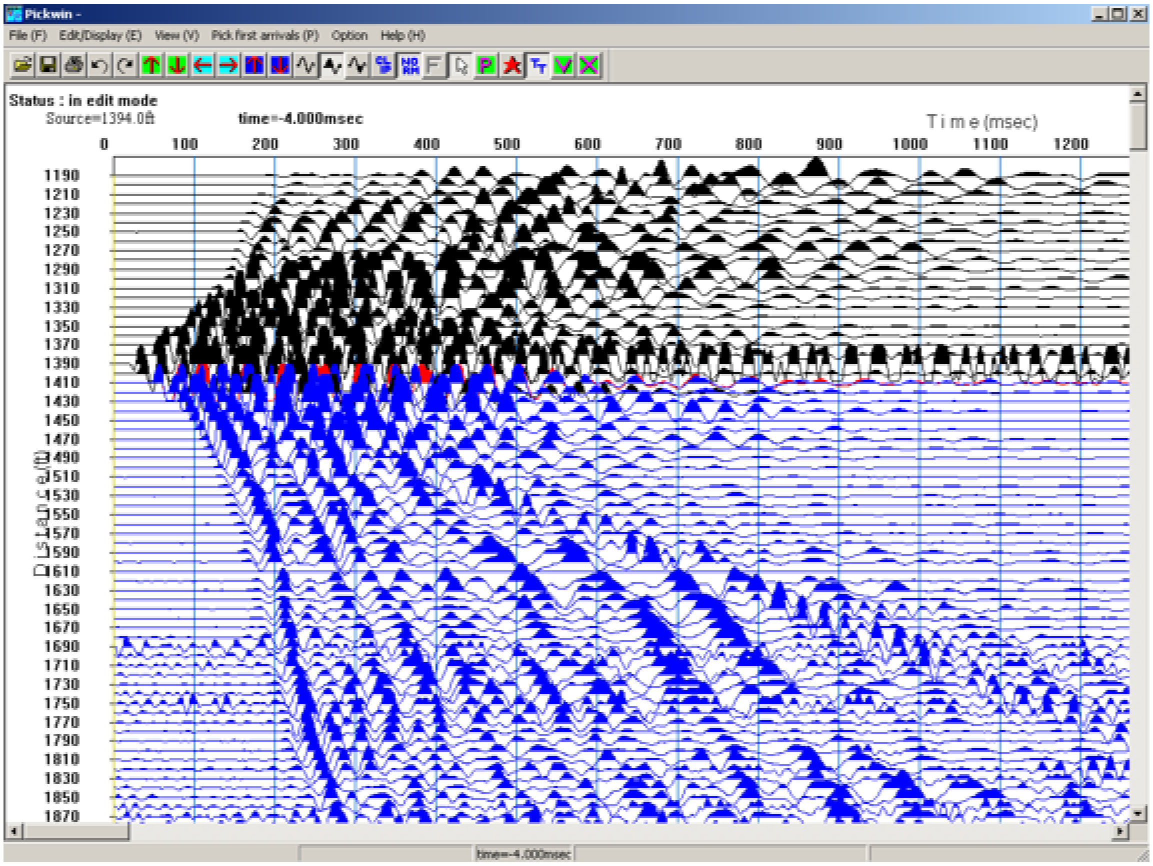
2) Open next SEG-2 file as appended file: 



3) If you have overlap, select overlapping traces:



4) Use the automatic shift function to correct data:



5) Pick the first breaks. 

6) Follow the procedure described in Example 1 until all files have been picked.

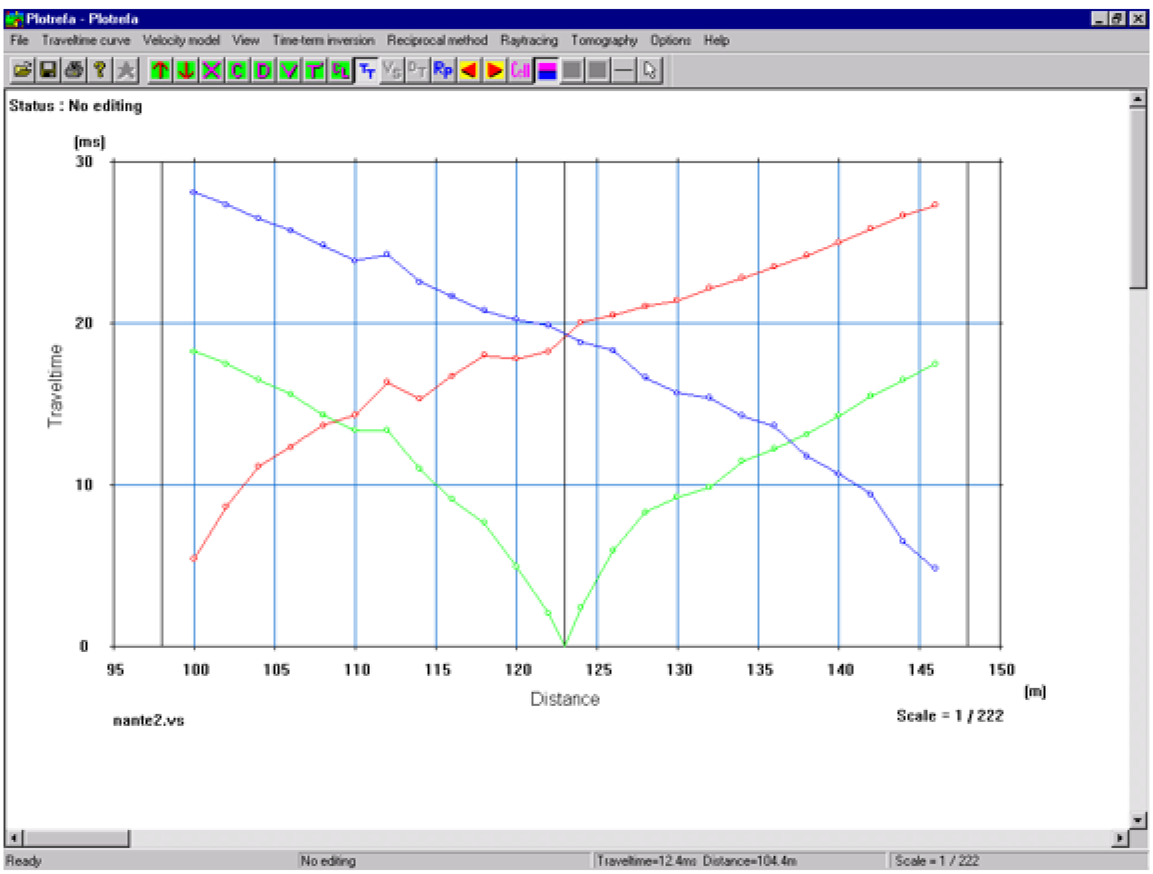
APPENDIX B PLOTREFA EXAMPLES

Example 1: Three-channel Refraction Survey

Most refraction surveys are conducted with seismographs consisting of 12 or more channels. Generally, this is the minimum number of travel times you need per shot to come up with a reasonable velocity model. Twelve or more geophones are laid out on the ground, and a series of shots are done at intervals along the line.

Twelve or more travel times are recorded for each shot.

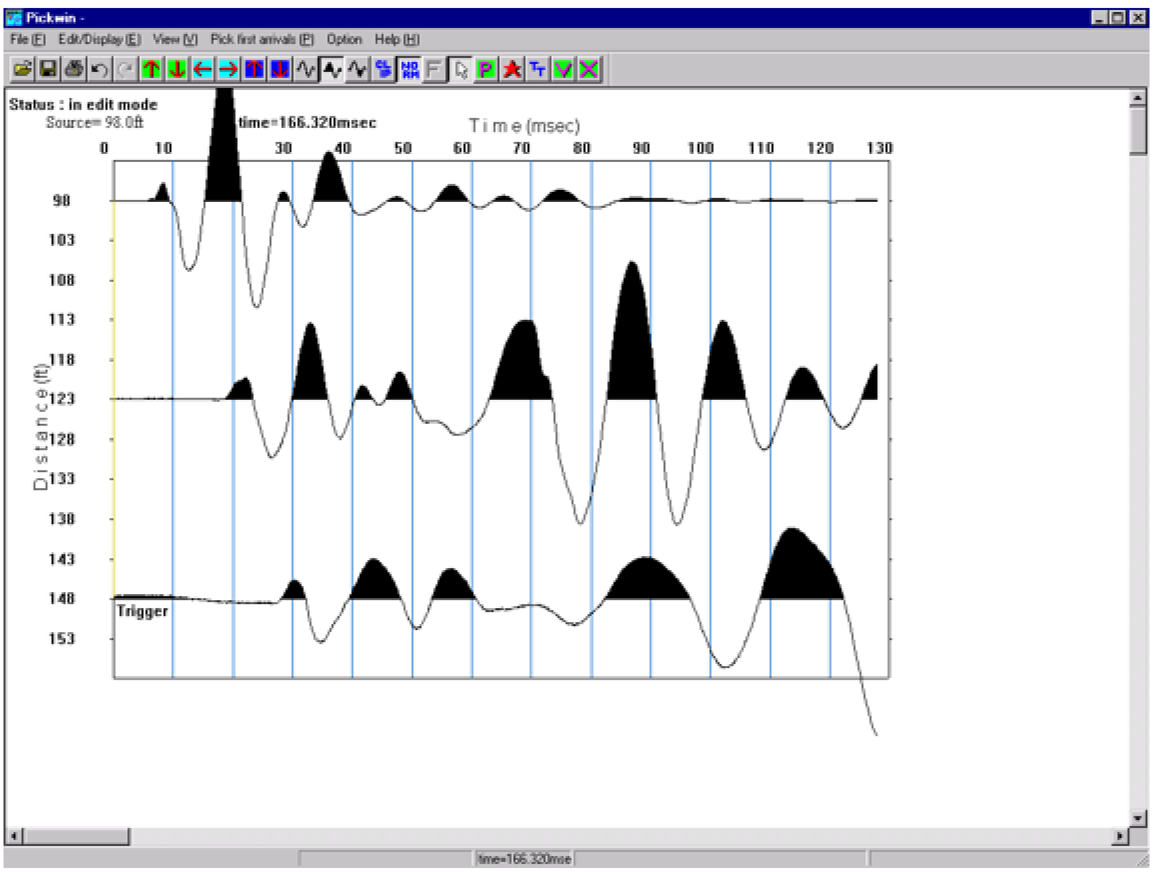
This style of surveying naturally leads to first arrivals being plotted relative to the *source* location. Plotting all the travel time curves together in one plot (below), is therefore referred to as a "common source gather".

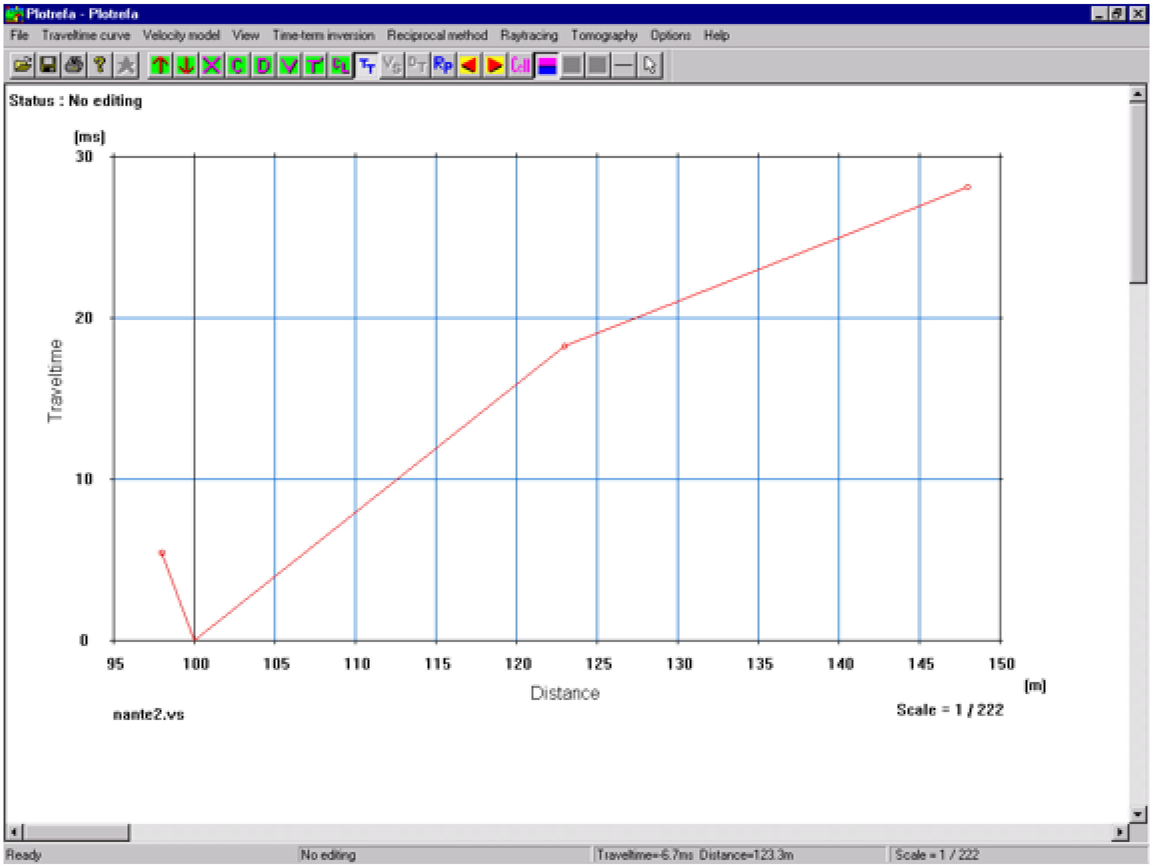


If the number of channels that you have available is less than the number of travel times you wish to record for each shot, you can take advantage of the principle of *reciprocity* and interchange the geophones and the shots. This is best illustrated by an example.

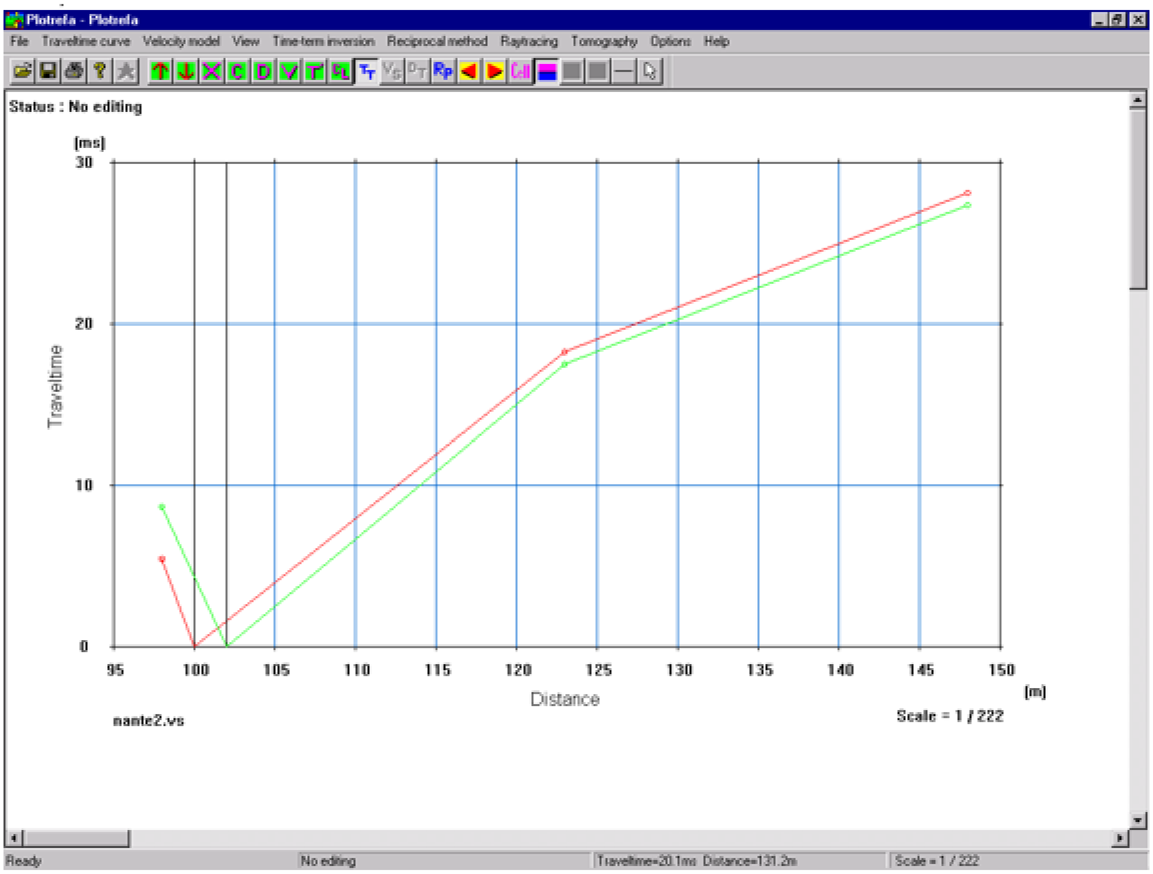
Suppose you have a 3-channel seismograph, but you would like to end up with a travel time plot like the one above, with 24 travel times per shot. Essentially, you must simulate having a 24-channel seismograph. You can do so by placing your three geophones where the shots would normally have been (in this example, at 98, 123 and 148 meters; see above), and doing your shots where the 24 geophones would normally have been. Instead of only shooting at three locations, you shoot at 24. It's more work but results in the same thing.

To simulate the above example, you would place your first shot at 100 meters (the location of the first geophone above). You would record three travel times, common to that source. You shot record would appear as shown below, and the common source plot follows:

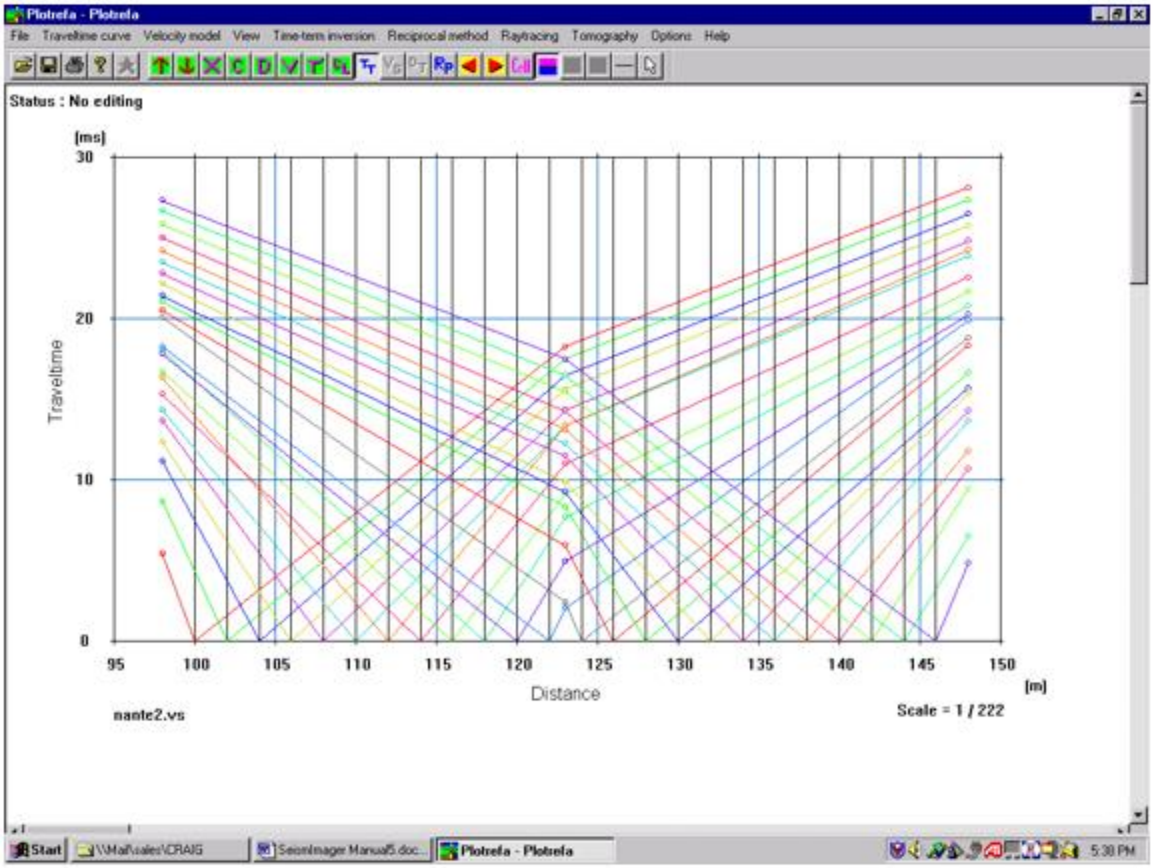



Plotrefa - Plotrefa
File Traveltime curve Velocity model View Time-term inversion Reciprocal method Raytracing Tomography Options Help
Status : No editing
[ms]
Traveltime
Distance
[m]
nante2.vs
Scale = 1 / 222
Ready
No editing
Traveltime=6.7ms Distance=123.3m
Scale = 1 / 222

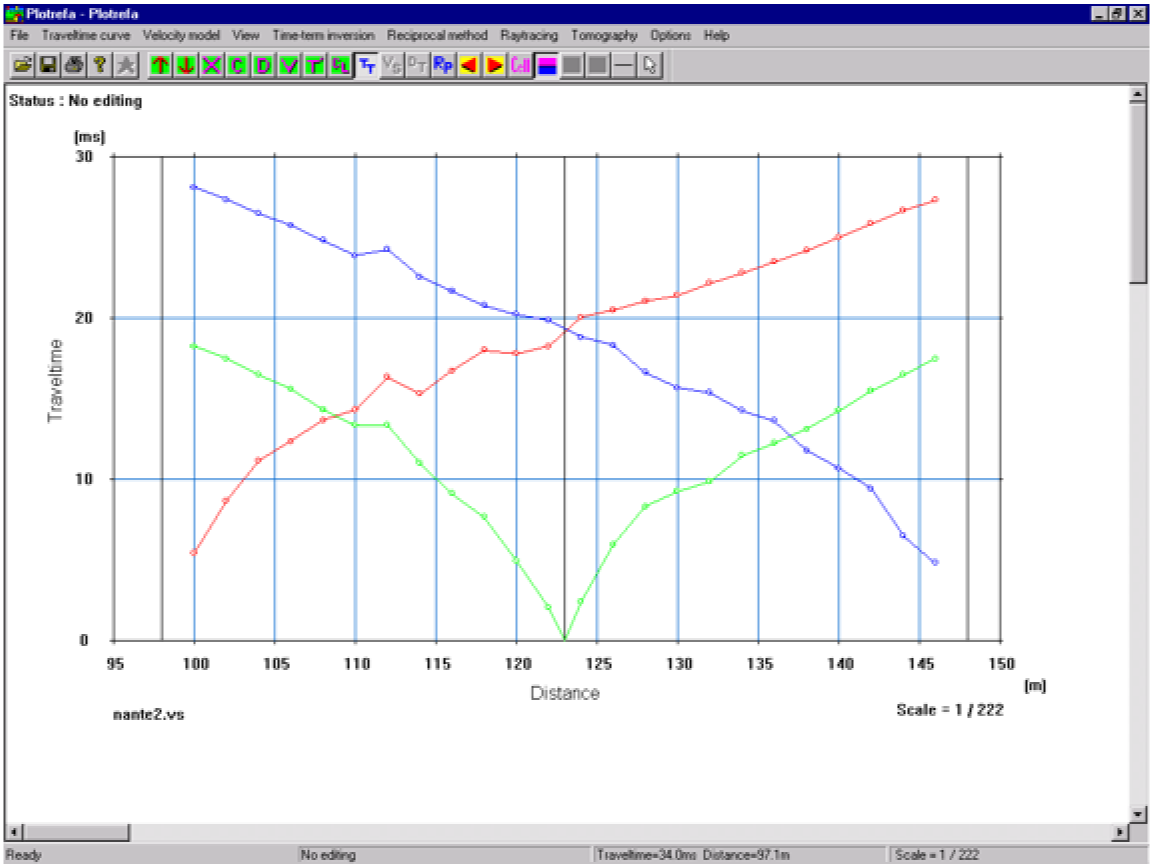
This is your first record. Next, you would move the source to 105 meters, and repeat:



You would repeat the process until you had occupied all the “geophone” stations. In the end, the common source *gather* would appear as follows:



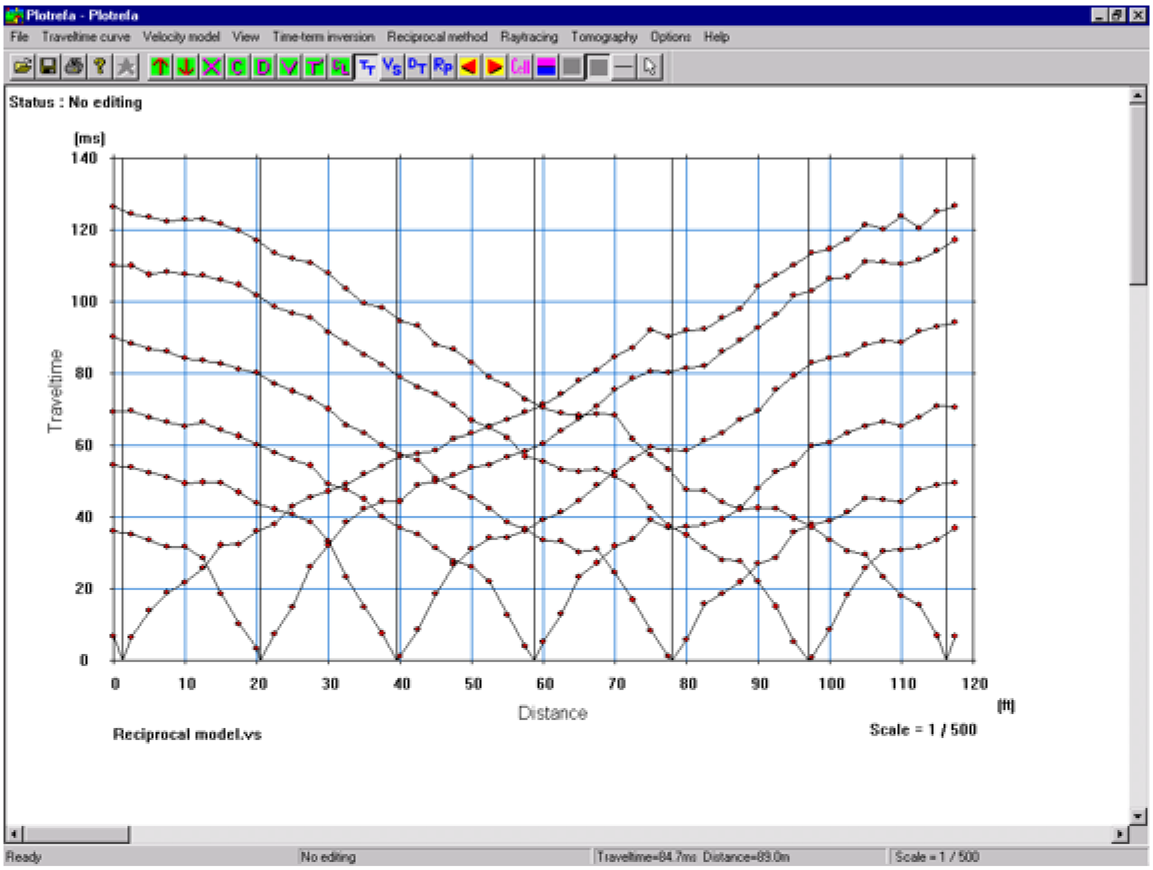
You have acquired 24 three-channel records, rather than three 24-channel records. But since you interchanged the sources and receivers, you must reorganize the above into a common *receiver* gather. To do so, simply click on the *Common source* <-> *common receiver* toggle switch (Section 4.2.14, Page 176):



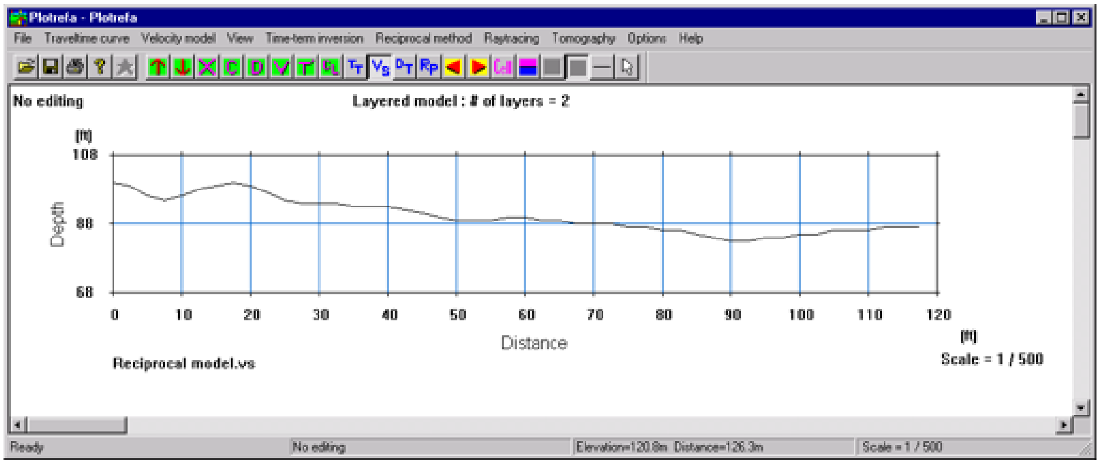
You can now interpret the data the same way you would have had you acquired it with three shots and a 24-channel seismograph.

Example 2: Two-layer Time-term Interpretation

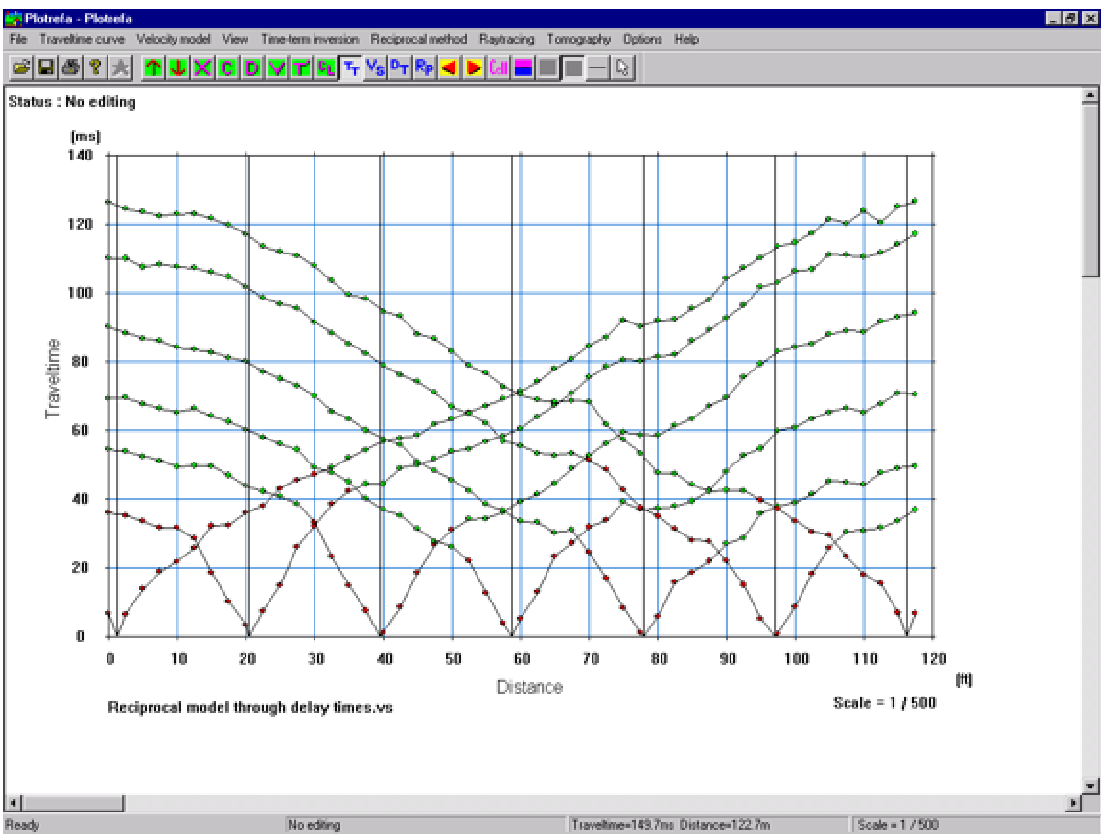
1) Open Plotrefa file: 



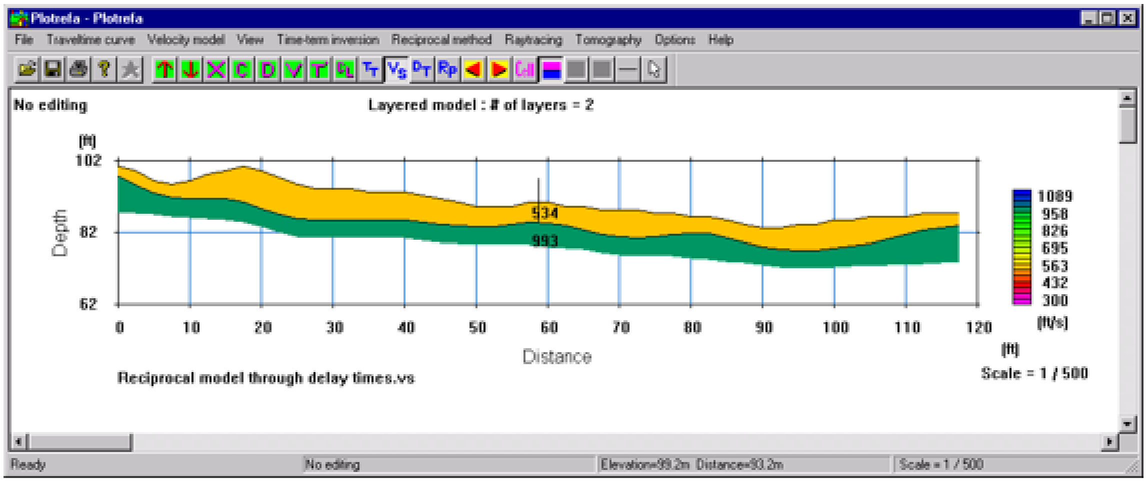
2) Import elevation file (if applicable):



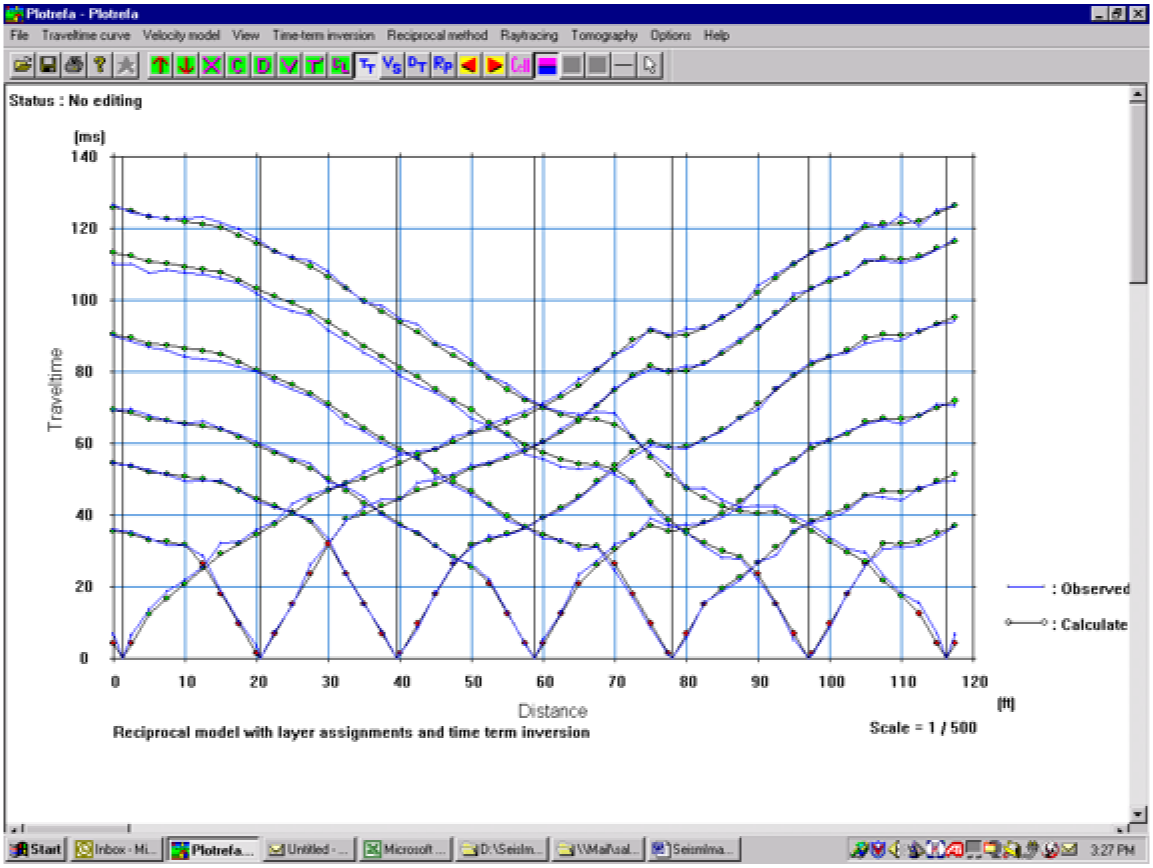
3) Assign layer 2 arrivals:



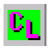
4) Do time-term inversion:



5) Run raytracing routine and compare theoretical travel times to observed travel times:

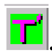


6) If necessary, adjust picks and layer assignments and repeat steps 4 and 5 until reasonable

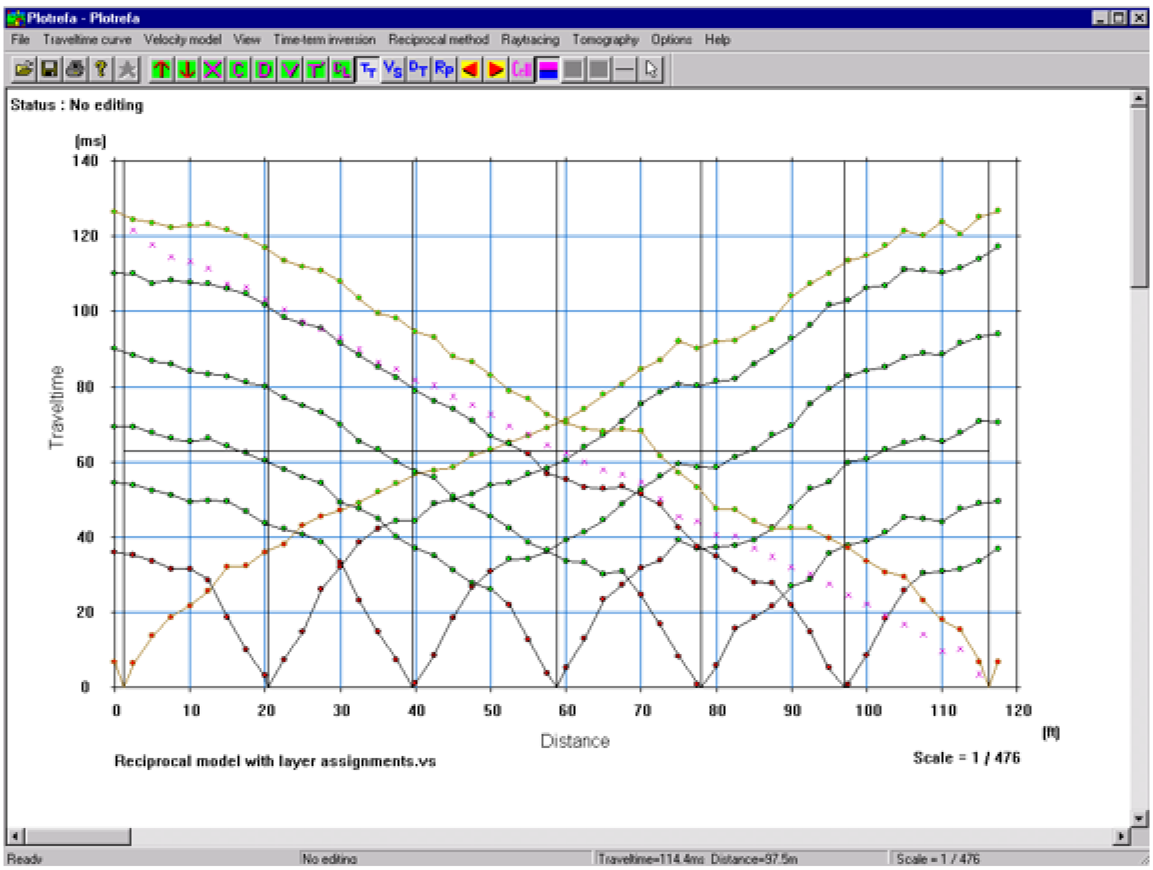
agreement is obtained between observed and theoretical travel times. Alternatively, press on the  tool button to modify the velocity section directly (see Section 0, Page 204). Run the raytracing routine after each set of modifications to see the effect on the travel time curves.

Example 3: Two-layer Reciprocal Method Interpretation

1) Calculate reduced travel times for opposing end-shots:

2) Click on .

3) Click on appropriate travel time curves.



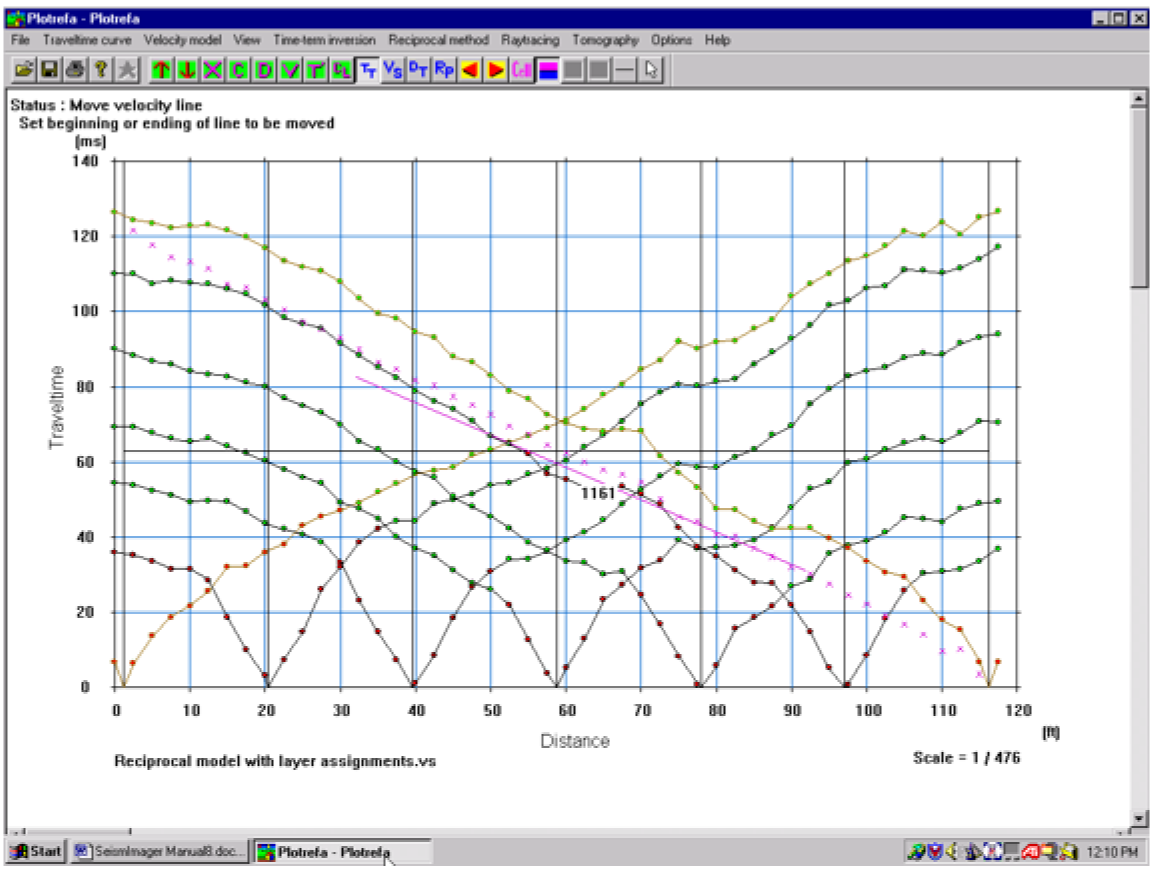
4) Fit velocity line to reduced travel time curve:

5) Click on .

6) Click on the reduced travel time curve at the left end of overlap zone.

7) Drag to right end of overlap zone to fit line.

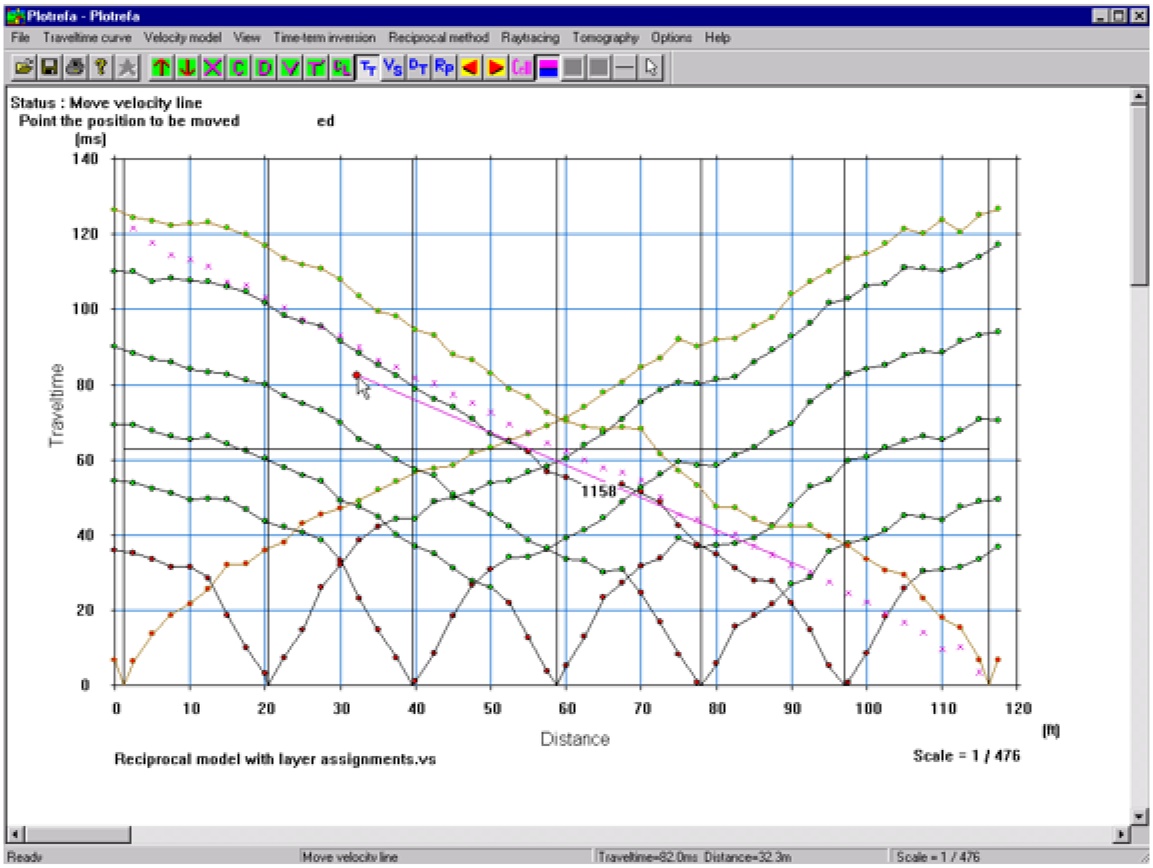
8) Right click to set velocity line.

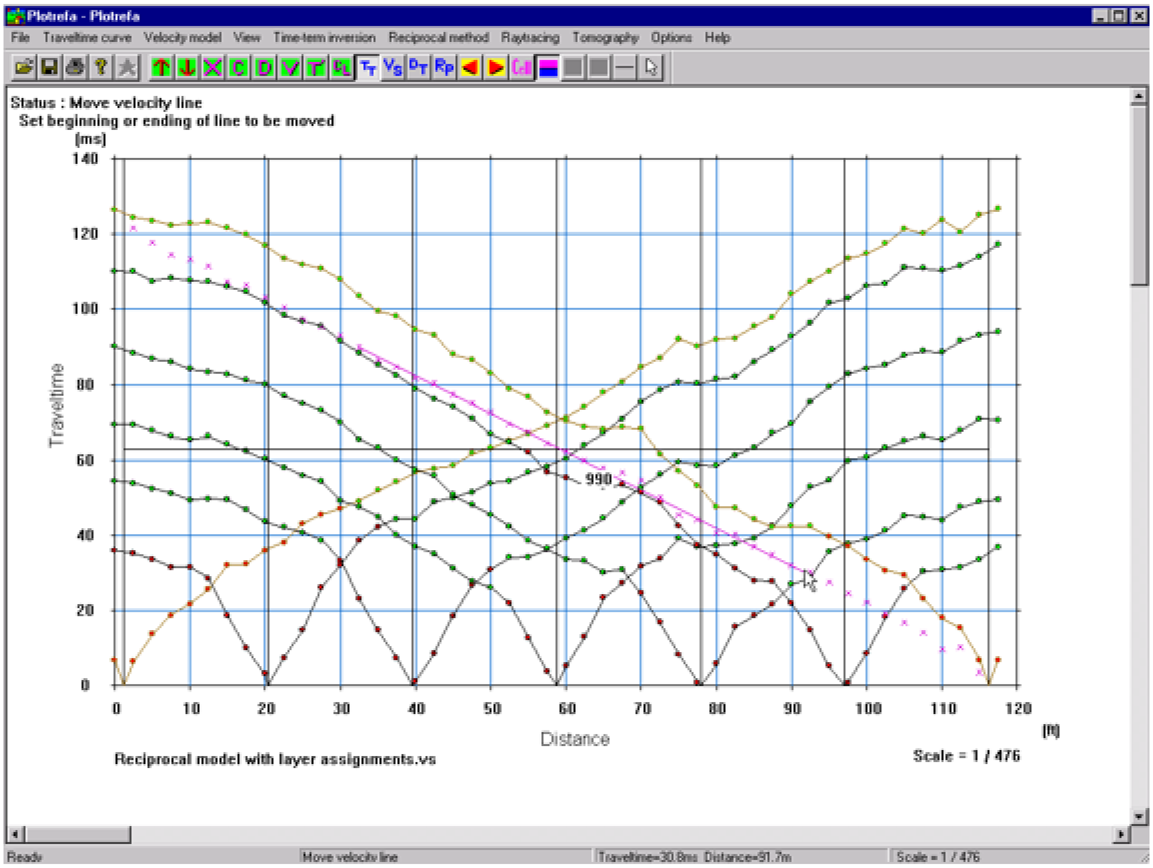


9) Adjust velocity line (if necessary):

10) Click on Adjust velocity line.

11) Click one end of velocity line and drag to new location.



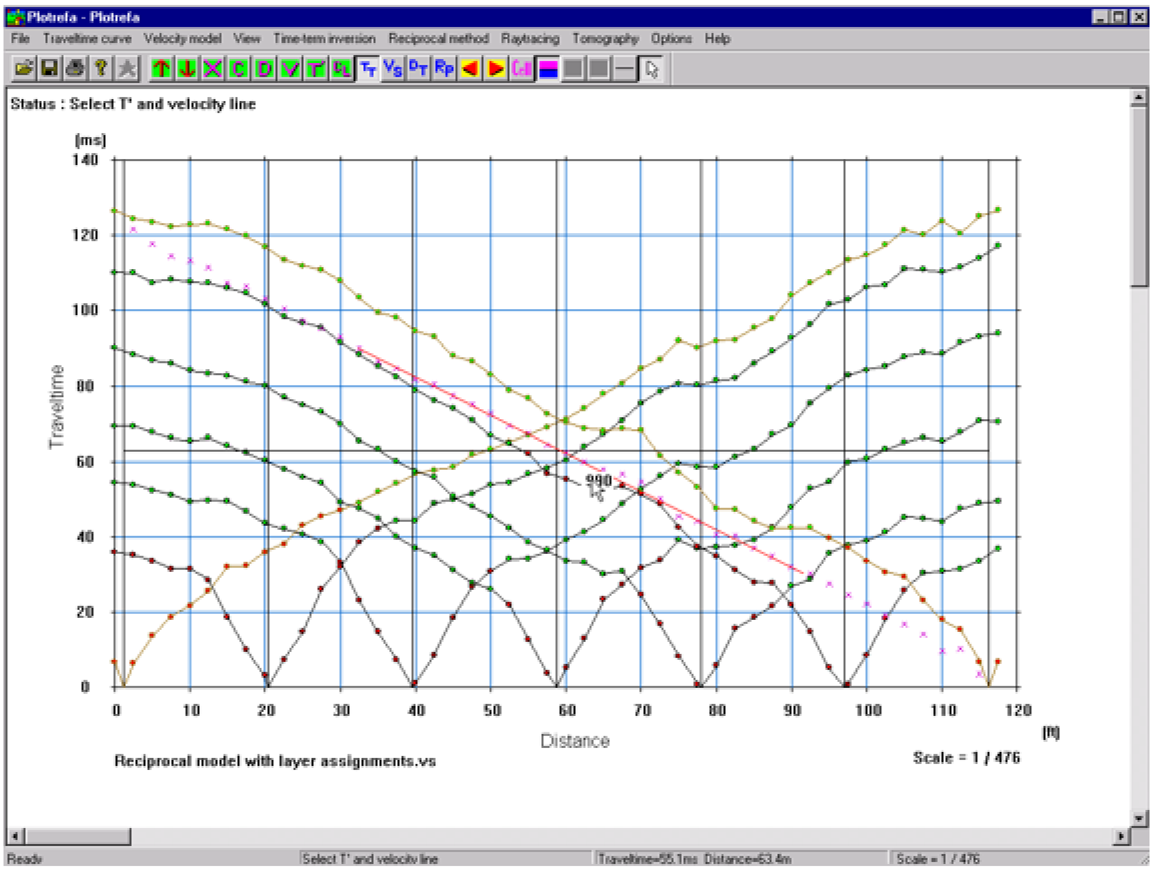


Plotrefa - Plotrefa
File Traveltime curve Velocity model View Time-term inversion Reciprocal method Raytracing Tomography Options Help
Status : Move velocity line
Set beginning or ending of line to be moved
[ms]
Traveltime
990
Distance
[ft]
Reciprocal model with layer assignments.vs
Scale = 1 / 476
Ready
Move velocity line
Traveltime=30.8ms Distance=91.7m
Scale = 1 / 476

12) Select velocity line:

13) Click on .

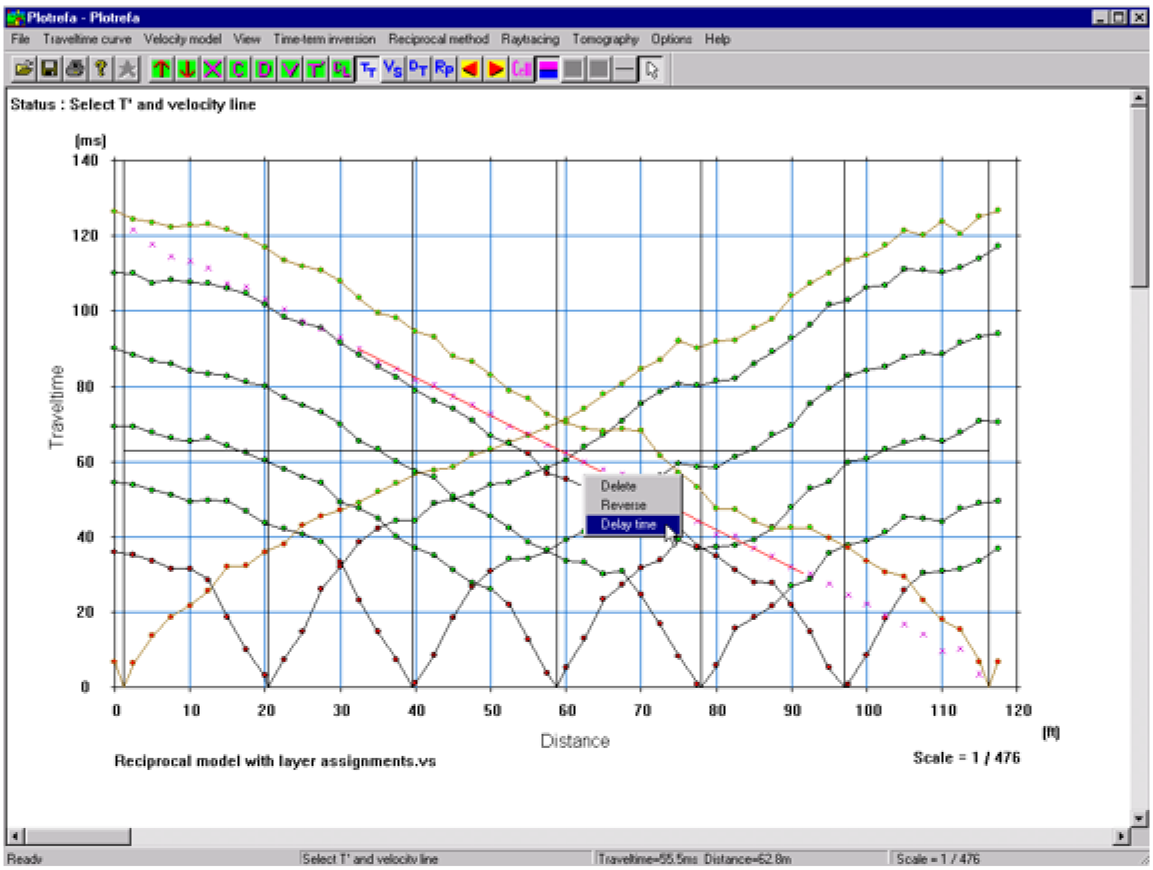
14) Click on velocity label; velocity line should turn red.



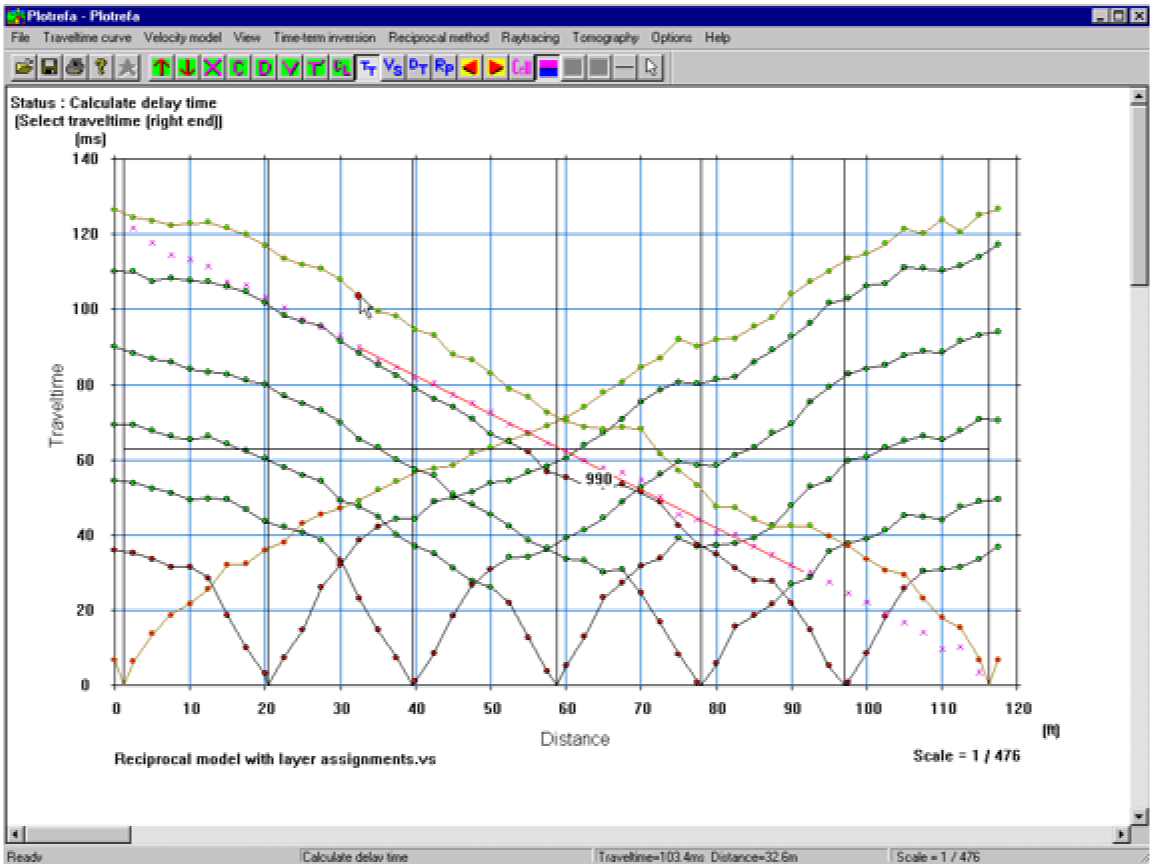
15) Calculate delay-times for first travel time curve:

16) Right click on velocity label to display menu.

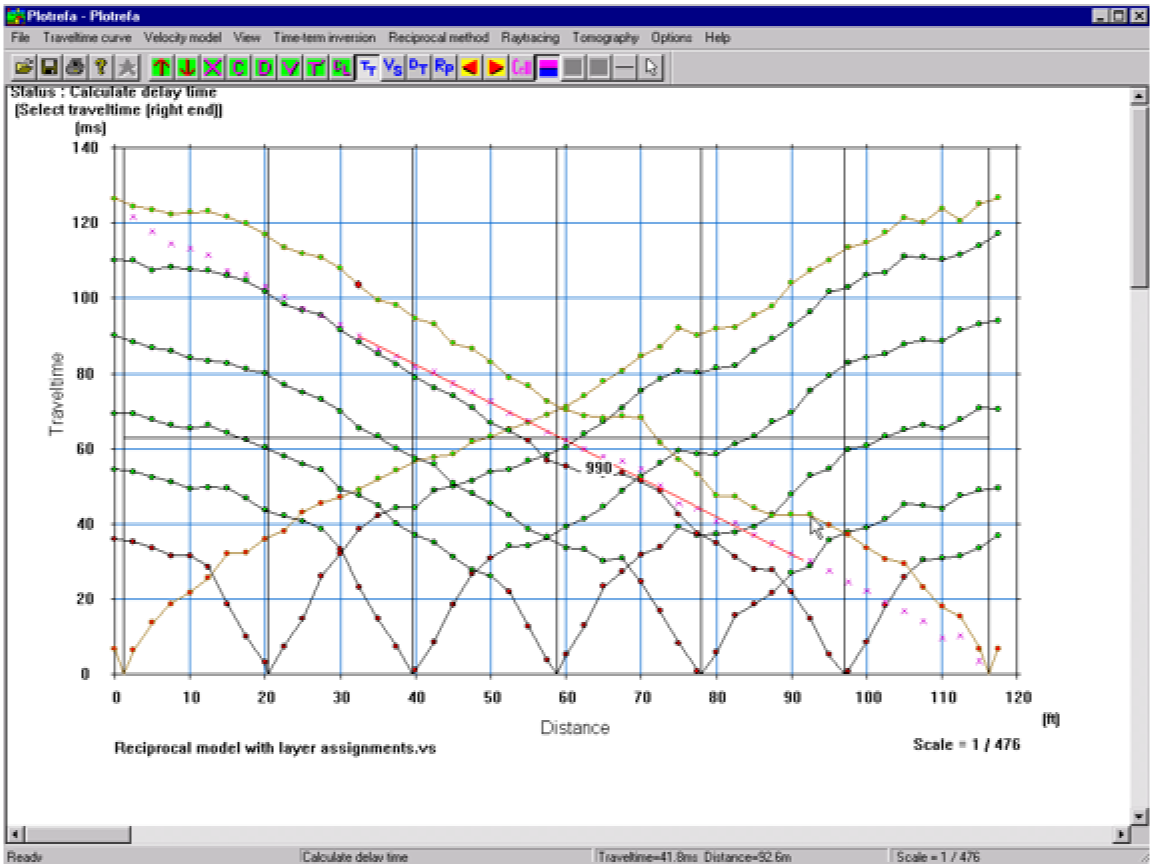
17) Click on Delay-time.



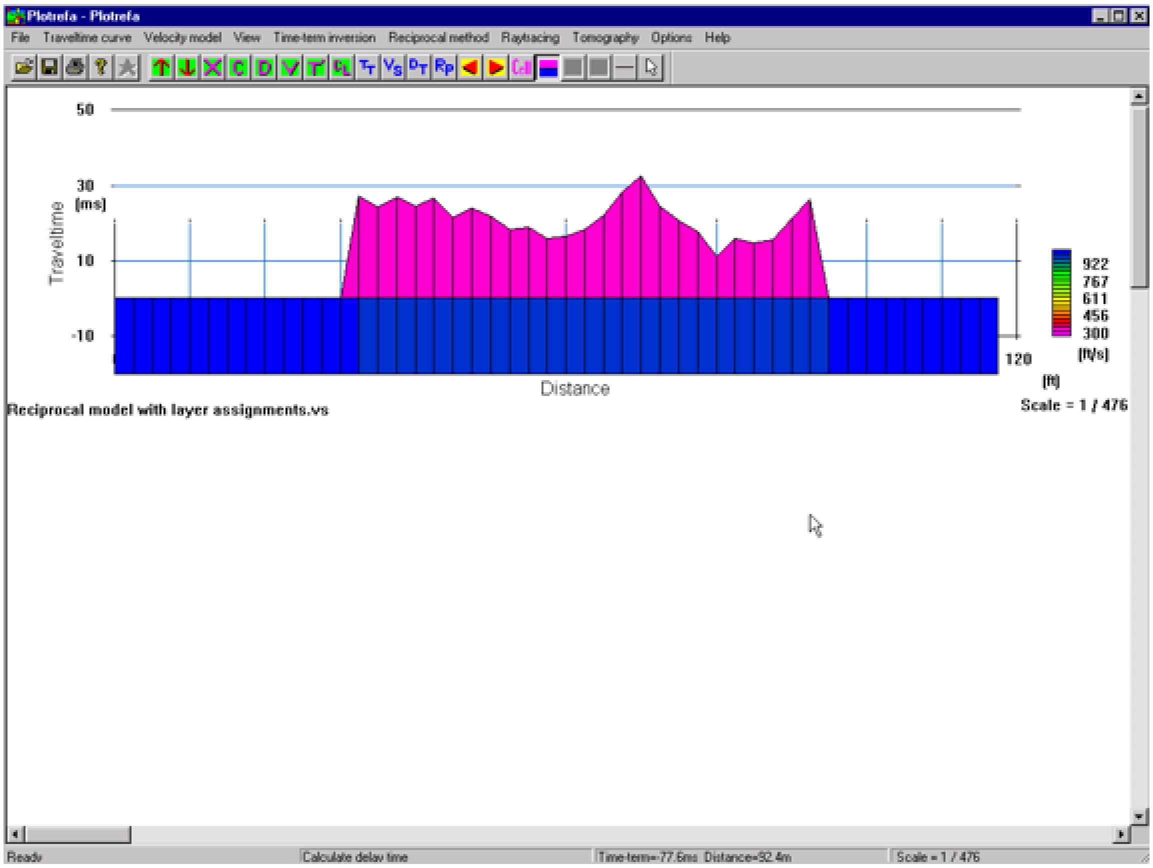
18) Click on left-end travel time within zone of overlap on travel time curve parallel to reduced time curve.



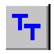
19) Click on right-end travel time within zone of overlap on travel time curve parallel to reduced time curve.



20) Delay-times will be displayed.

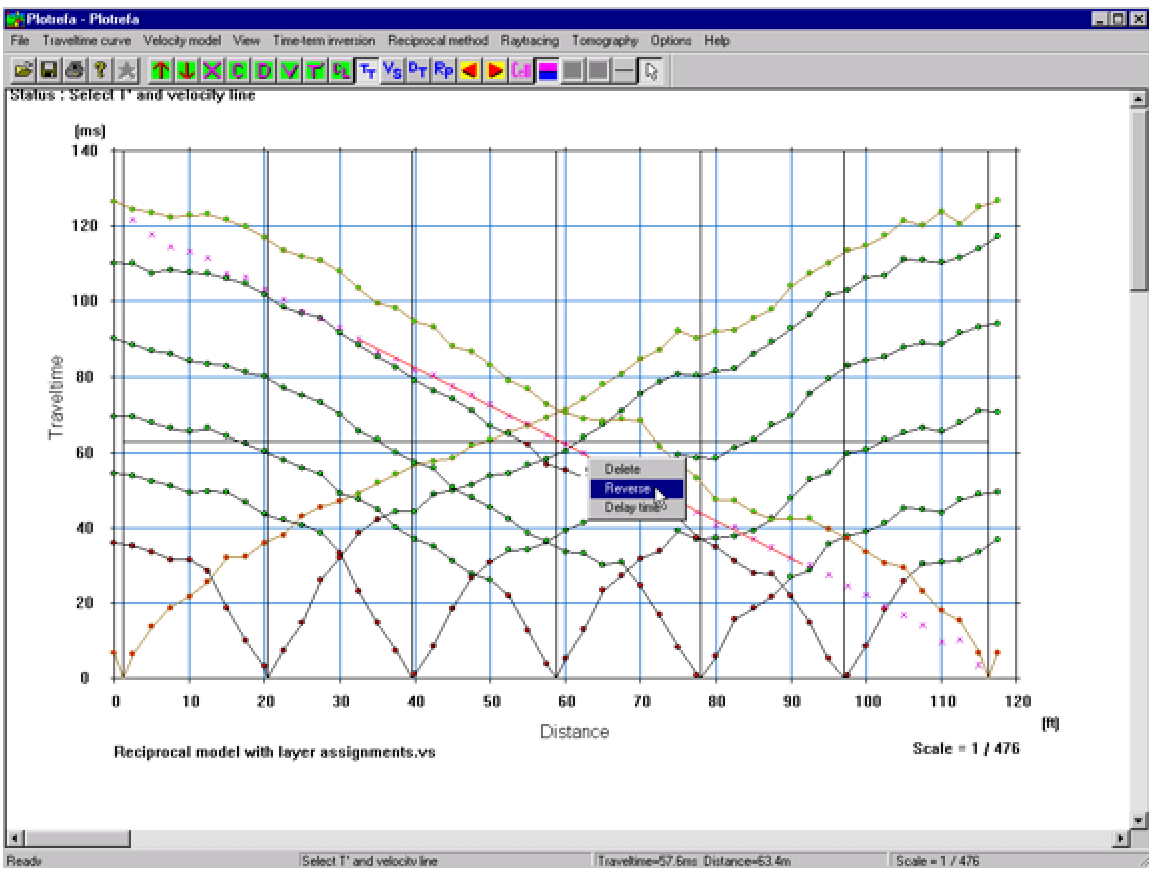


21) Calculate delay-times for opposing travel time curve:

22) Click on  to display travel time plot.

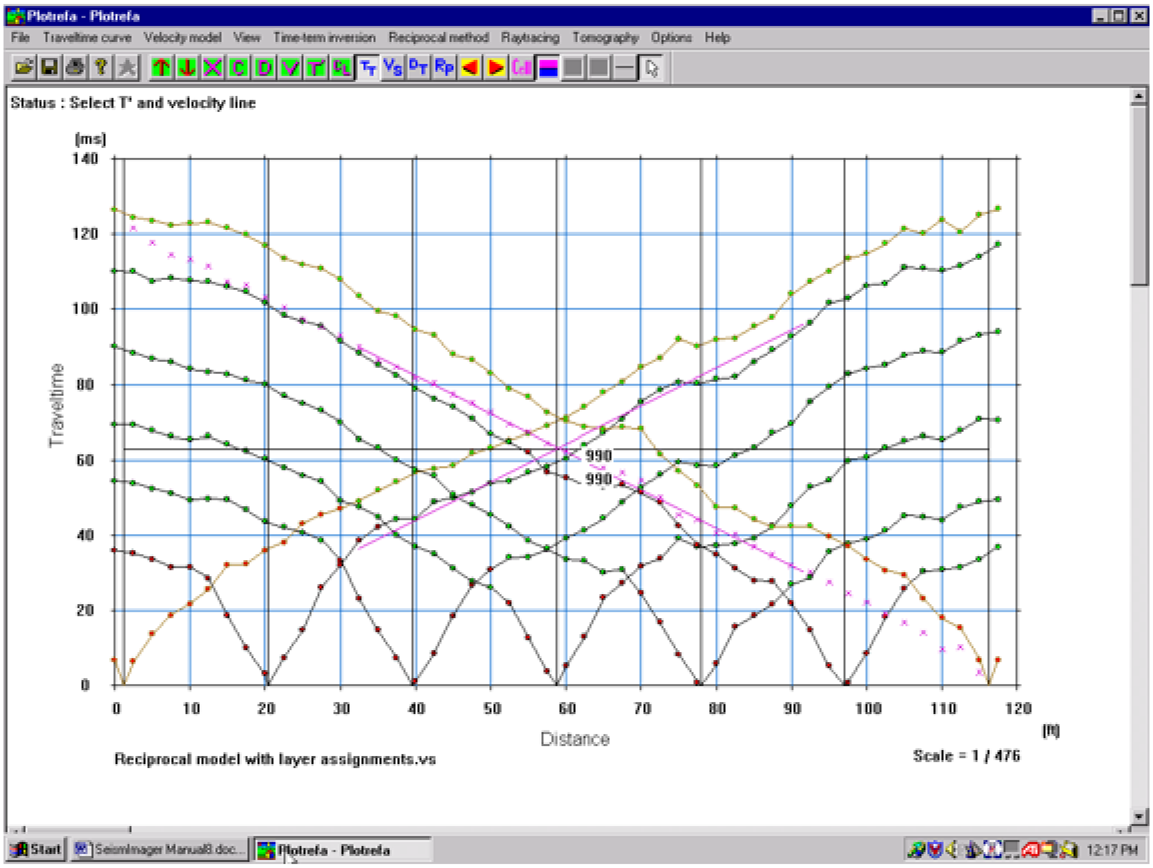
23) Click on  and then click on the velocity label to select the velocity line, which should turn red.

24) Right click on the velocity label to reveal the menu and choose *Reverse.*



25) Click on the ½ $T_{(ab)}$ line.

26) The velocity line will reverse.



27) Repeat (4) and (5) above to calculate delay-times for opposing shot record.

28) Delete all reduced travel time curves.

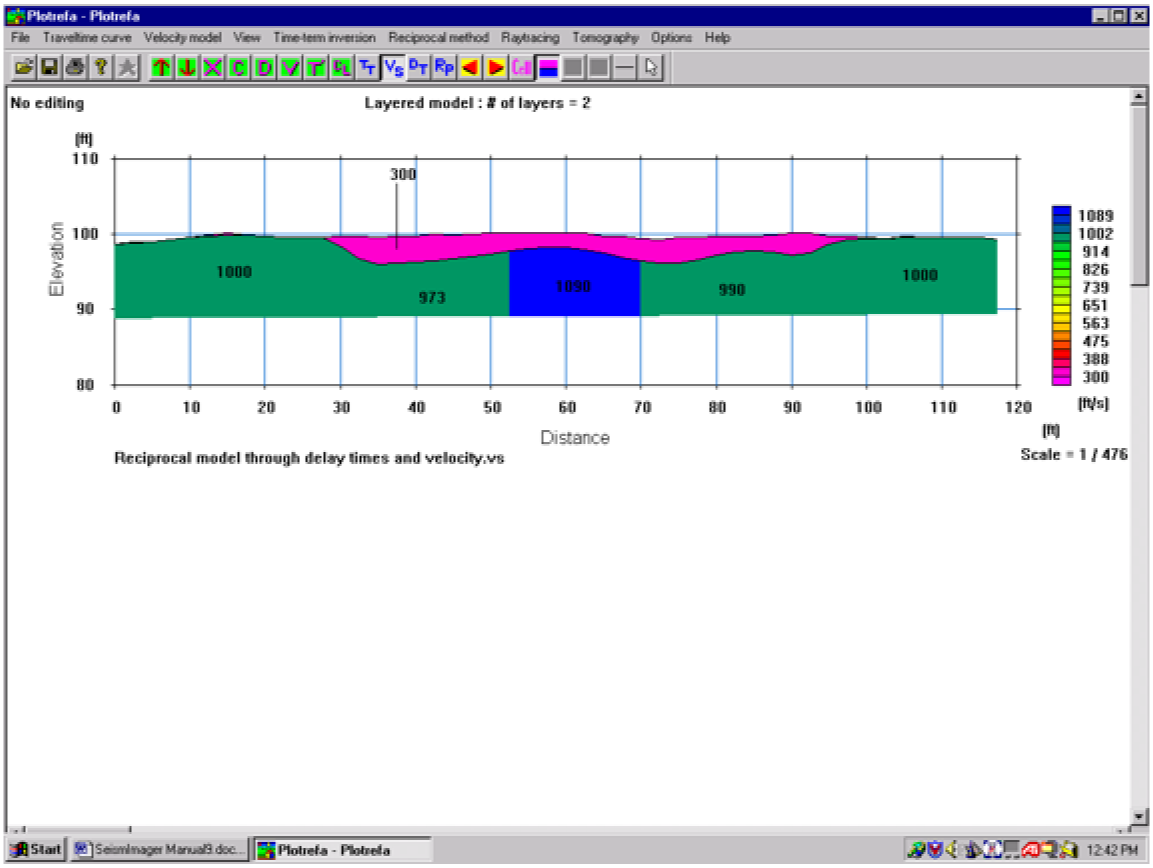
29) Delete all velocity lines.

30) Repeat (4), (5), and (6) for all other opposing shots having reasonable overlap.

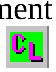
31) Create a velocity model.

32) Click on Calculate velocity model from delay-time data.

33) The velocity model will be displayed.

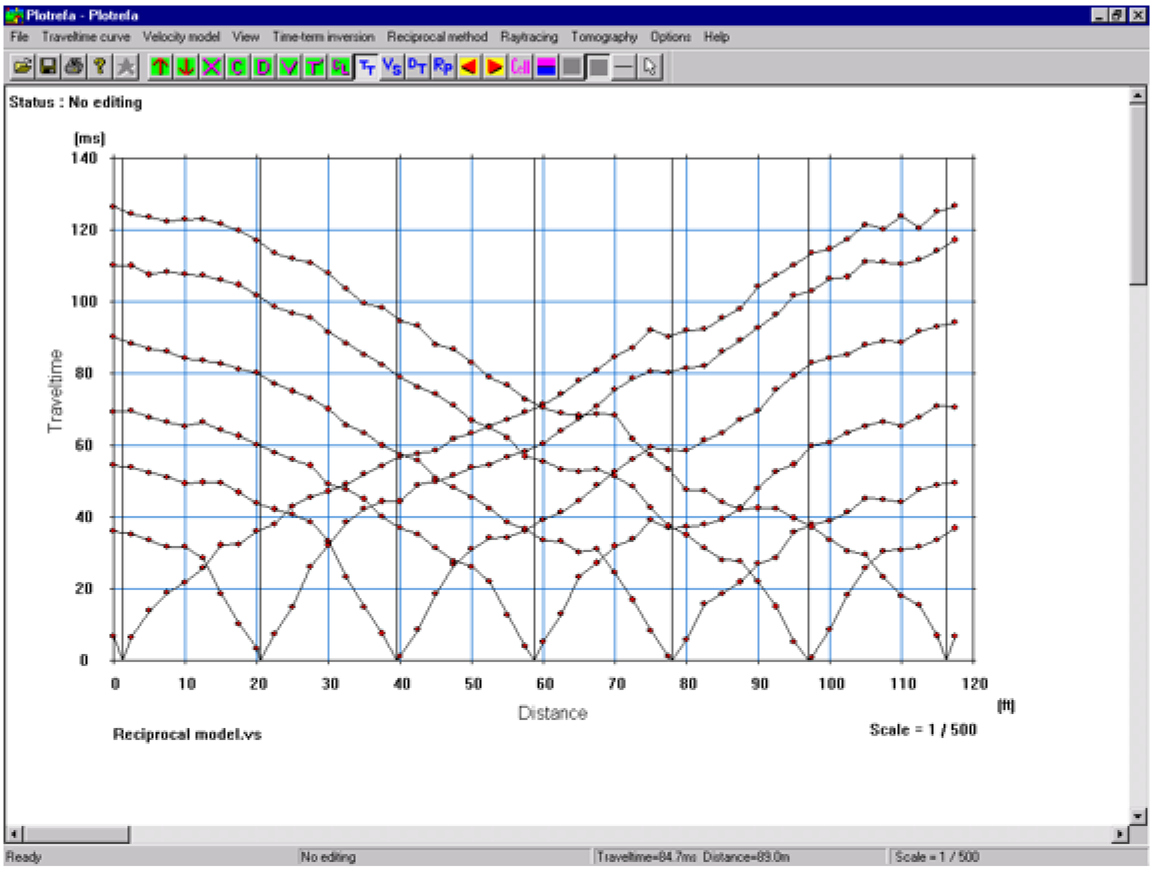


34) Run raytracing routine and compare theoretical travel times to observed travel times.

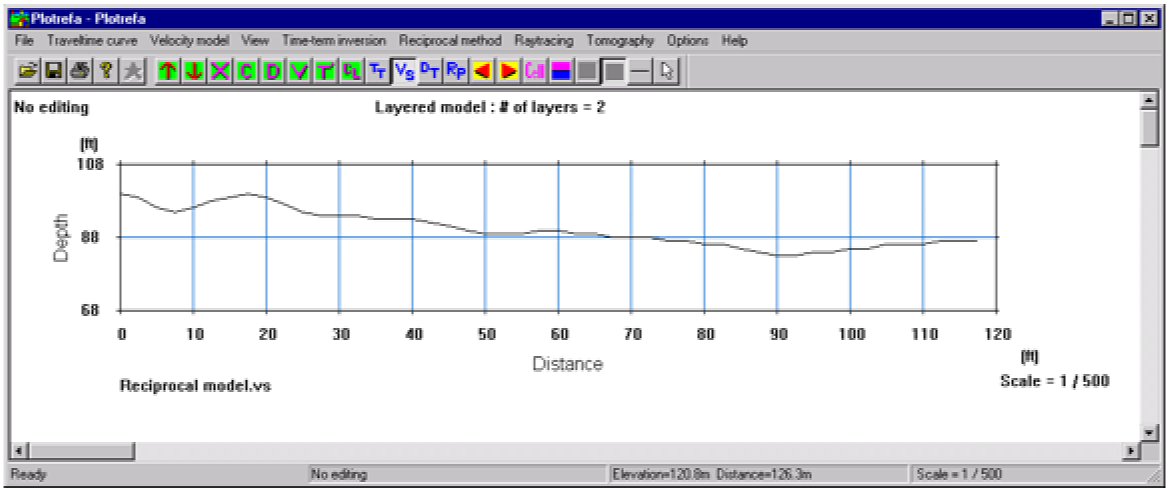
35) If necessary, adjust picks and layer assignments and repeat steps 4 and 5 until reasonable agreement is obtained between observed and theoretical travel times. Alternatively, press on the  tool button to modify the velocity section directly (see Section 0, Page 204). Run the raytracing routine after each set of modifications to see the effect on the theoretical travel time curves.

Example 4 – Tomographic Inversion

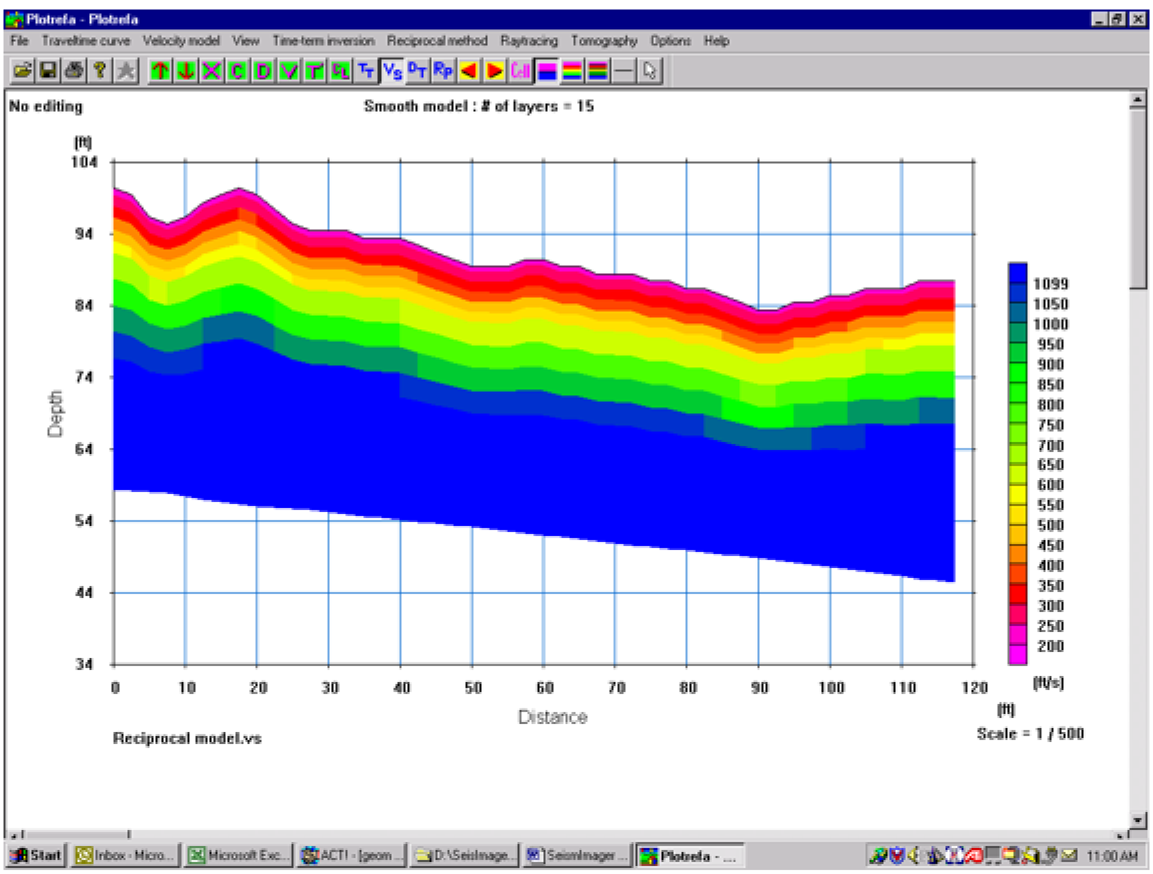
1) Open Plotrefa file: 



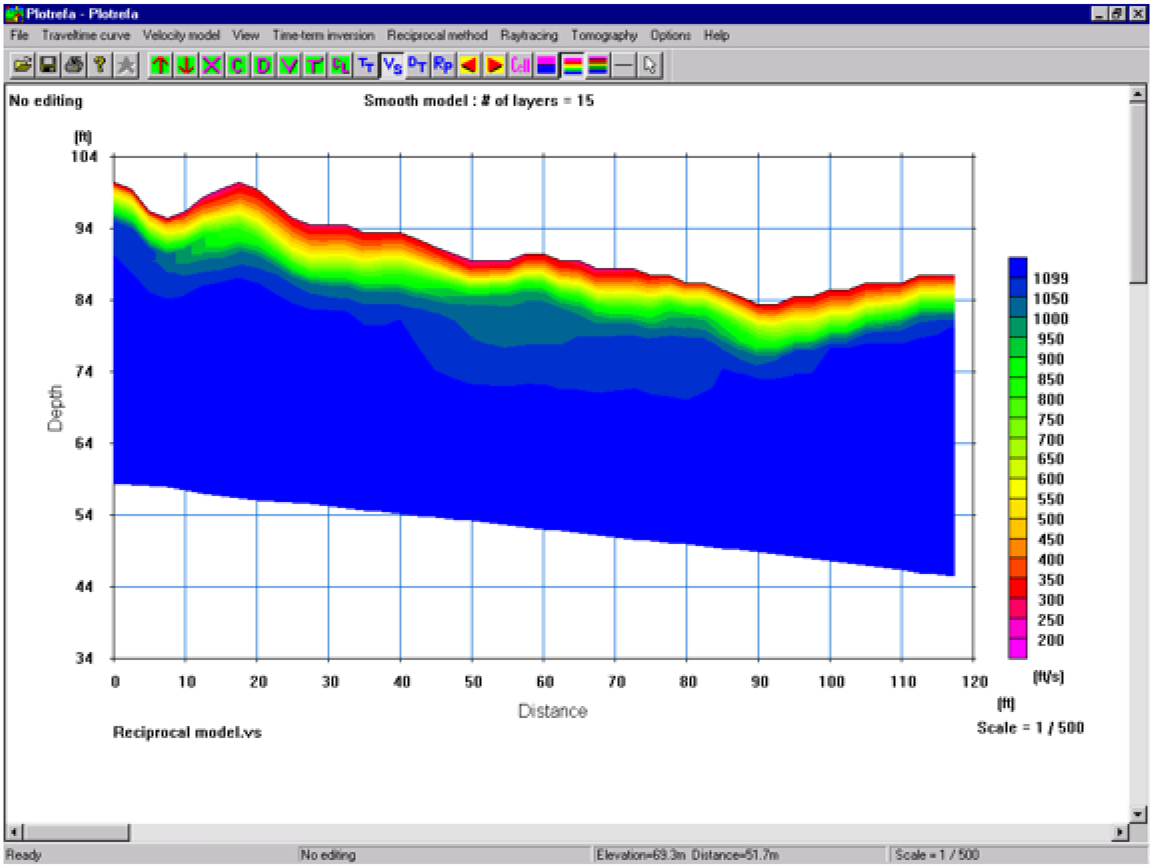
2) Import elevation file (if applicable):



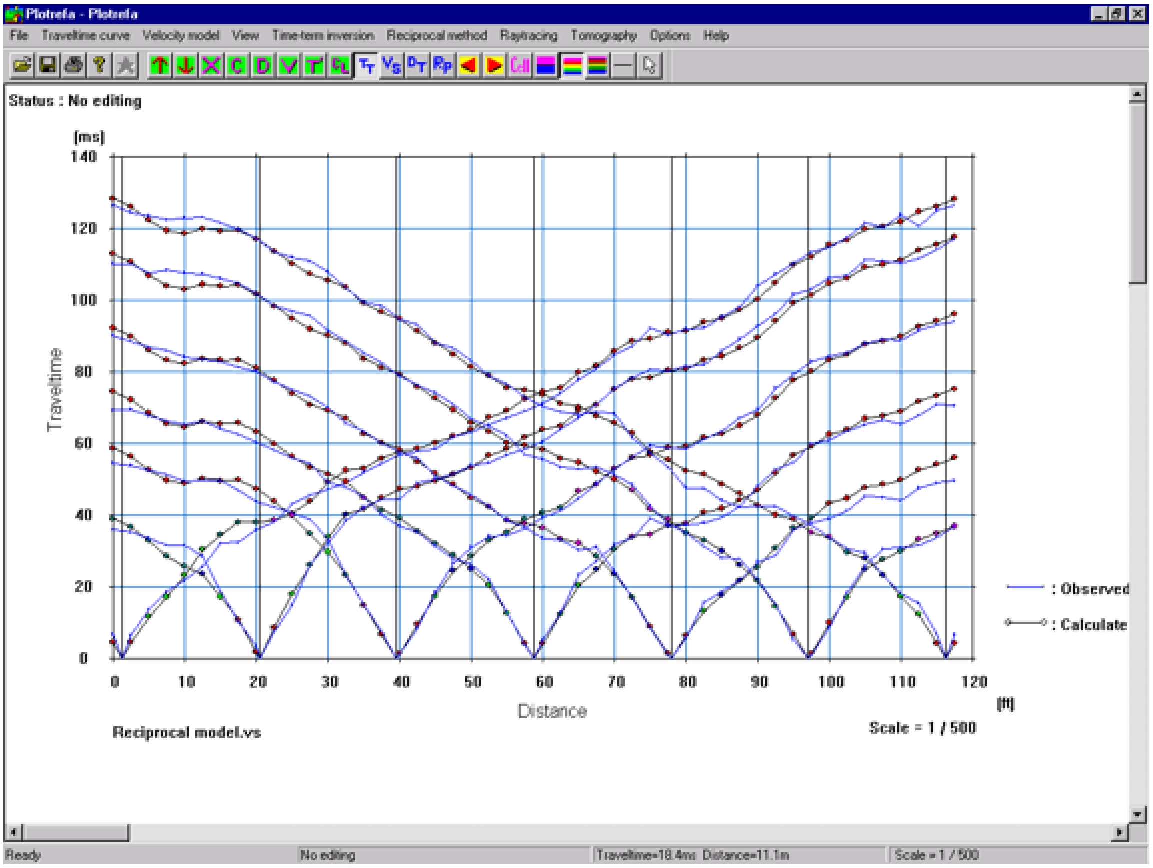
3) Create initial velocity model:



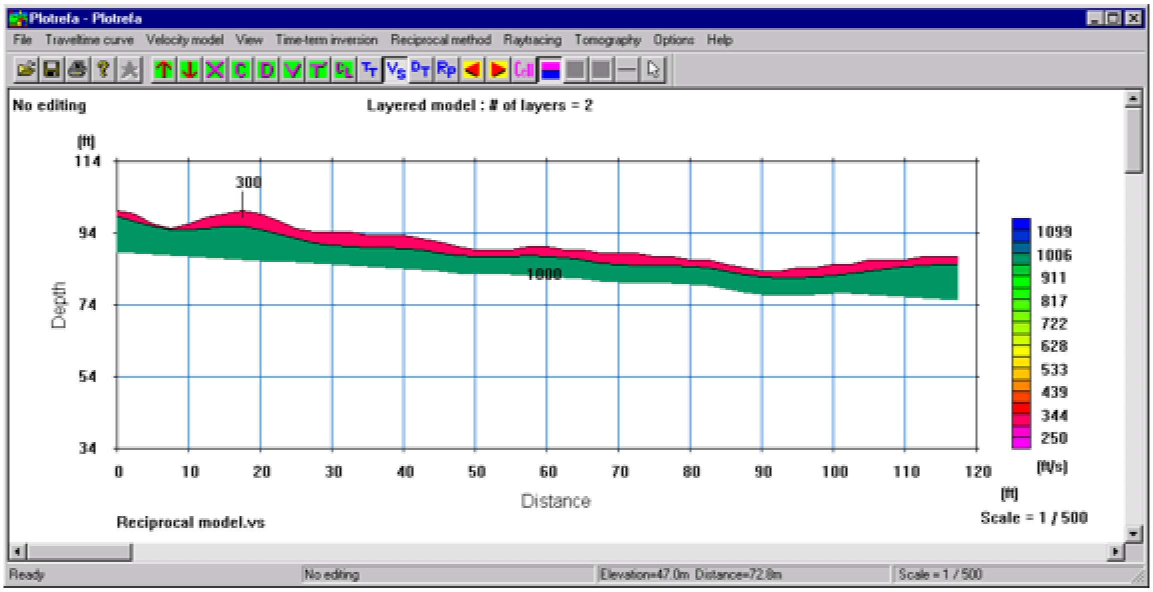
4) Do tomographic inversion:



5) Check agreement between measured and theoretical travel times; repeat analysis with different starting model and tomography parameters if necessary:



6) Convert to layered model (if applicable):



.

APPENDIX C FUNDAMENTALS OF SEISMIC REFRACTION

A simple two-layer velocity model along with its associated travel time curve is shown below. We will use this figure as a basis for discussing the fundamental principles underlying the seismic refraction technique.

Note: *This appendix discusses the very basics of seismic refraction and is not intended to be a complete treatment of the subject. For more in-depth discussions, see the recommended reading list in Appendix G. The following discussion borrows heavily from Redpath (1973). Also, the following links will be helpful in understanding what follows.*

What is a seismic wave? A seismic ray?

What is Huygen's Principle?

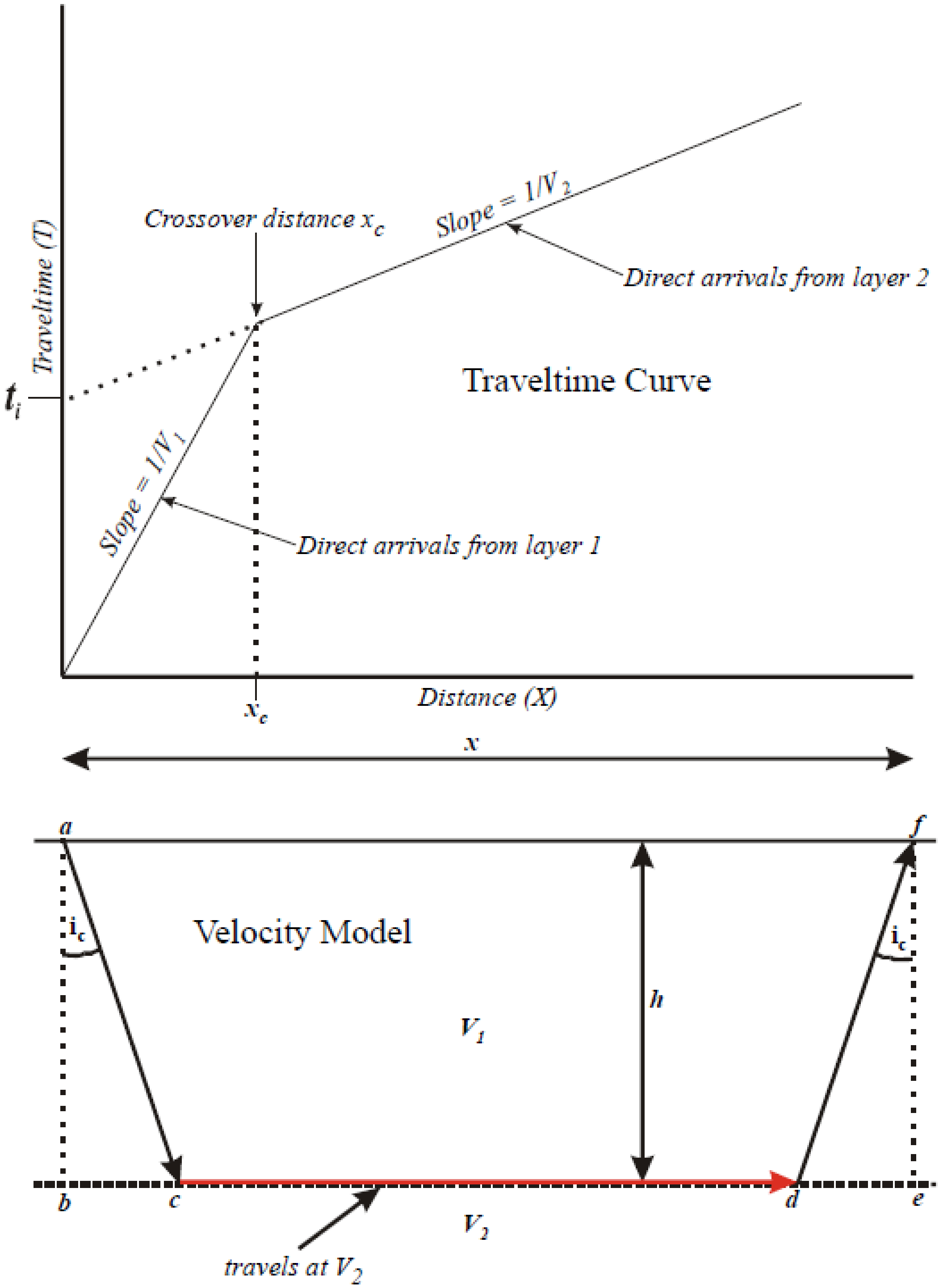
What is seismic refraction?

What is Snell's Law?

What is the crossover distance?

What is apparent velocity?

What is reciprocity?



The break in slope of the above travel time curve, which occurs at the “crossover distance”, marks the point at which travel times refracted from V_2 overtake direct arrivals traveling through V_1. The equation for the first segment T_1 is simply

$$T_1 = \frac{X}{V_1}$$

Equation C-1

The equation for T_2 is

$$T2 = \frac{\overline{ac}}{V_1} + \frac{\overline{cd}}{V_2} + \frac{\overline{df}}{V_1}.$$

From the figure above,

$$\overline{ac} = \overline{df} = hcos(i_c).$$

Substituting, we get

$$T_2 = \frac{2h}{V_1 \cos(i_c)} + \frac{\overline{cd}}{V_2}.$$

Equation C-2

Now,

$$\tan(i_c) = \frac{\overline{bc}}{h} = \frac{\overline{de}}{h},$$

or

$$\overline{bc} = \overline{de} = htan(i_c)\ .$$

Referring back to the figure,

$$\overline{cd} = x - \overline{bc} - \overline{dc} = x - 2htan(i_c).$$

Substituting into Equation C-2,

$$T_2 = \frac{2h}{V_1 \cos(i_c)} + \frac{x - 2htan(i_c)}{V_2}.$$

Equation C-3

Rearranging,

$$T_2 = \frac{2h}{V_1 \cos(i_c)} - \frac{2htan(i_c)}{V_2} + \frac{x}{V_2}\text{, }and$$

$$T_2 = 2h\left\{\frac{1}{V_1 \cos(i_c)} - \frac{\tan(i_c)}{V_2}\right\} + \frac{x}{V_2}.$$

Substituting *sin(i*$_c$*)/cos(i*$_c$*) for tan(i*$_c$*)*,

$$T_2 = 2h\left\{\frac{1}{V_1\, cos(i_c)} - \frac{sin(i_c)}{V_2\, cos(i_c)}\right\} + \frac{x}{V_2}.$$

Rearranging,

$$T_2 = 2h\left\{\frac{V_2}{V_1 V_2\, cos(i_c)} - \frac{V_1\, sin(i_c)}{V_1 V_2\, cos(i_c)}\right\} + \frac{x}{V_2}\text{, }and$$

$$T_2 = 2h\left\{\frac{V_2 - V_1\, sin(i_c)}{V_1 V_2\, cos(i_c)}\right\} + \frac{x}{V_2}.$$

Equation C-4

From Snell's Law,

$$sin(i_c) = \frac{V_1}{V_2}.$$

Equation C-5

Substituting $V_1/sin(i_c)$ for the V_2 term in the numerator of Equation C-4,

$$T_2 = 2h\left\{\frac{\frac{V_1}{sin(i_c)} - V_1\,sin(i_c)}{V_1V_2\,cos(i_c)}\right\} + \frac{x}{V_2}$$

$$T_2 = 2hV_1\left\{\frac{\frac{1}{sin(i_c)} - sin(i_c)}{V_1V_2\,cos(i_c)}\right\} + \frac{x}{V_2}$$

$$T_2 = 2h\left\{\frac{\frac{1}{sin(i_c)} - sin(i_c)}{V_2\,cos(i_c)}\right\} + \frac{x}{V_2}$$

$$T_2 = 2h\left\{\frac{\frac{1 - sin^2(i_c)}{sin(i_c)}}{V_2\,cos(i_c)}\right\} + \frac{x}{V_2}$$

$$T_2 = 2h\left\{\frac{1 - sin^2(\,i_c)}{V_2\,sin(i_c)\,con(i_c)}\right\} + \frac{x}{V_2}.$$

Since $1\text{-}sin^2(i_c)=cos^2(i_c)$,

$$T_2 = 2h\left\{\frac{cos^2(i_c)}{V_2\,sin(i_c)\,cos(i_c)}\right\} + \frac{x}{V_2}\ ,and$$

$$T_2 = 2h\left\{\frac{cos\,(i_c)}{V_2\,sin(i_c)}\right\} + \frac{x}{V_2}.$$

And from Snell's Law (Equation C-5), we substitute V_1 *for* $V_2 sin(i_c)$:

$$T_2 = 2h\frac{cos(i_c)}{V_1} + \frac{x}{V_2}.$$

Equation C-6

Since

$$cos(i_c) = \sqrt{1 - sin^2(i_c)}\,,$$

substituting into Equation C-6 gives us

$$T_2 = 2h\frac{\sqrt{1 - sin^2(i_c)}}{V_1} + \frac{x}{V_2}.$$

Equation C-7

From Snell's Law (Equation C-5)

$$sin(i_c) = \frac{V_1}{V_2},$$

so

$$sin^2(i_c) = \left(\frac{V_1}{V_2}\right)^2.$$

Substituting back into Equation C-7,

$$T_2 = 2h\frac{\sqrt{1 - \left(\frac{V_1}{V_2}\right)^2}}{V_1} + \frac{x}{V_2}$$

$$T_2 = 2h\frac{\sqrt{1 - \frac{V_1^2}{V_2^2}}}{V_1} + \frac{x}{V_2}$$

$$T_2 = 2h\frac{\sqrt{\frac{V_2^2 - V_1^2}{V_2^2}}}{V_1} + \frac{x}{V_2}$$

$$T_2 = 2h\frac{\left(\frac{\sqrt{V_2^2 - V_1^2}}{V_2}\right)}{V_1} + x/V_2$$

$$T_2 = 2h\frac{\sqrt{V_2^2 - V_1^2}}{V_1 V_2} + \frac{x}{V_2}.$$

For the special case of $x = 0$, we get

$$T_i = 2h\frac{\sqrt{V_2^2 - V_1^2}}{V_1 V_2},$$

Equation C-8

or, using Equation C-6,

$$T_i = \frac{2h\cos(i_c)}{V_1}.$$

Equation C-9

T_i is called the "intercept time".

Again, from Snell's Law (Equation C-5),

$$i_c = \sin^{-1}\frac{V_1}{V_2}.$$

Solving Equation C-9 for *h* and substituting,

$$h = 1/2T_i V_1/\cos\,(\sin^{-1} V_1/V_2),$$

and

$$\boldsymbol{h = \frac{1}{2}\frac{T_i V_1}{\cos\,(\sin^{-1}\frac{V_1}{V_2})}}.$$

Equation C-10

Alternatively, solving Equation C-8 for *h* yields

$$\boldsymbol{h = \frac{1}{2}\frac{T_i V_1 V_2}{\sqrt{V_2^2 - V_1^2}}}.$$

Equation C-11

Now, using Equations A-10 or A-11, we can calculate depth *(h)* by measuring T_i, V_1 and V_2 from the travel time graph.

Alternatively, the crossover distance can be used in lieu of the intercept time. At the crossover distance i_c, $T_1=T_2$, so we can equate Equations A-1 and A-8:

$$T_1 = \frac{X_c}{V_1} = \frac{2h\sqrt{V_2^2 - V_1^2}}{V_1 V_2} + \frac{x_c}{V_2} = T_2$$

$$\frac{X_c}{V_1} - \frac{x_c}{V_2} = \frac{2h\sqrt{V_2^2 - V_1^2}}{V_1 V_2}$$

$$\frac{x_c}{2h}\left(\frac{1}{V_1} - \frac{1}{V_2}\right) = \frac{\sqrt{V_2^2 - V_1^2}}{V_1 V_2}$$

$$\frac{x_c}{2h} = \frac{\frac{\sqrt{V_2^2 - V_1^2}}{V_1 V_2}}{\left(\frac{1}{V_1} - \frac{1}{V_2}\right)}$$

$$h = \frac{1}{2}\frac{\left(\frac{1}{V_1} - \frac{1}{V_2}\right)}{\frac{\sqrt{V_2^2 - V_1^2}}{V_1 V_2}} x_c$$

$$h = \frac{1}{2}\frac{\left(\frac{1}{V_1} - \frac{1}{V_2}\right) V_1 V_2}{\sqrt{V_2^2 - V_1^2}} x_c$$

$$h = \frac{1}{2}\frac{(\frac{V_2}{V_1V_2} - \frac{V_1}{V_1V_2})V_1V_2}{\sqrt{V_2^2 - V_1^2}}x_c$$

$$h = \frac{1}{2}\frac{\left(\frac{V_2 - V_1}{V_1V_2}\right)V_1V_2}{\sqrt{V_2^2 - V_1^2}}x_c$$

$$\frac{1}{2}\frac{(V_2 - V_1)}{\sqrt{V_2^2 - V_1^2}}x_c\,.$$

Squaring both sides,

$$h^2 = \frac{1}{4}\frac{(V_2 - V_1)^2}{V_2^2 - V_1^2}x_c^2\,.$$

Now, V_2^2-V_1^2 can be factored into

$$(V_2 + V_1)(V_2 - V_2).$$

Substituting,

$$h^2\frac{1}{4}\frac{(V_2 - V_1)^2}{(V_2 + V_1)(V_2 - V_1)}x_c^2$$

$$h^2 = \frac{1}{4}\frac{(V_2 - V_1)}{(V_2 + V_1)}x_{c^2}\,, and$$

$$\boldsymbol{h = \frac{1}{2}\sqrt{\frac{(V_2 - V_1)}{(V_2 + V_1)}}}x_c$$

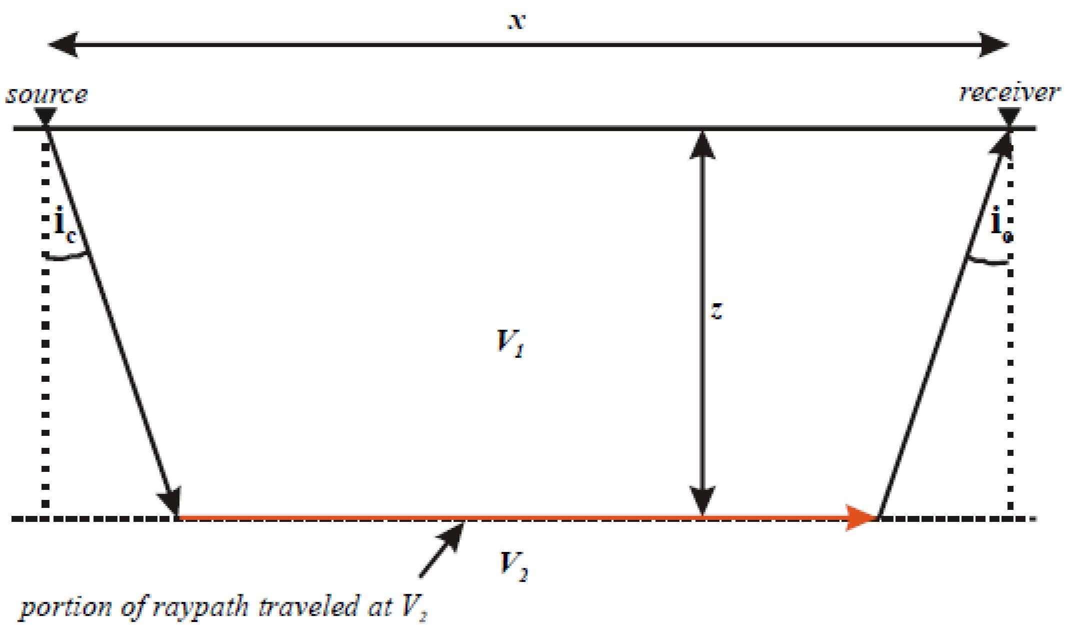
Equation C-12

Equations C-10 (or C-11) and C-12 are the most basic equations in seismic refraction, relating layer thickness to the travel time plot. Although valid only for constant layer thickness, understanding them and where they came from is essential to understanding seismic refraction in general. The above can be extended to any number of layers. See Redpath 1973 for a good example of a four-layer case.

The derivation for smoothly varying thickness with flat interfaces is much more complex, and beyond the scope of this discussion. A good treatment of the varying thickness case is also provided in the above-mentioned reference.

APPENDIX D THE TIME-TERM METHOD

The time-term technique is a linear least-squares approach to determining the best discrete-layer solution to the data. The math behind this technique is comparatively simple. Referring to the figure below,



we define the "slowness" *S* as the inverse velocity:

$$S_1 = \frac{1}{V_1}$$

$$S_2 = \frac{1}{V_2}.$$

From Snell's Law,

$$\sin(i_c) = \frac{S_2}{S_1}.$$

Referring back to the derivation in Appendix C (see math leading up to Equation C-6), the total travel time from source to receiver is then

$$t = 2S_1 \cos(i_c)\, z + xS_2 \,.$$

Now, if we define

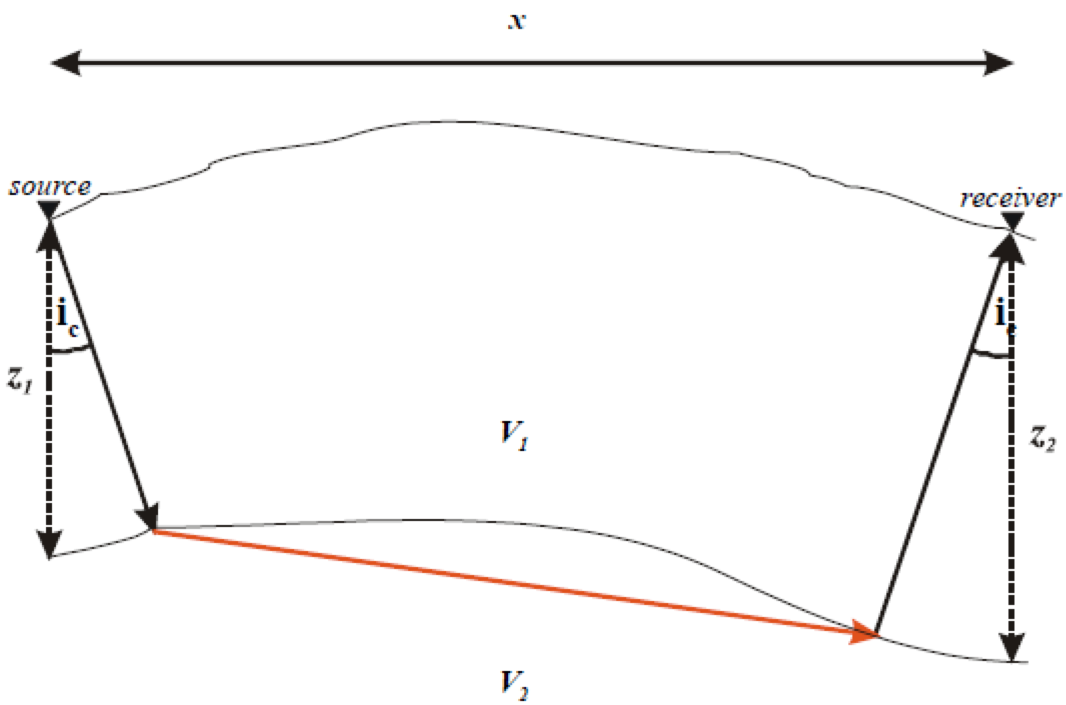
$$c = 2S_1 \cos(i_c) \,,$$

then

$$t = 2cz + xS_2$$

and z and S_2 are unknown.

The above example (Appendix A) assumes that the refractor is parallel to the ground surface. If we expand this to the general case – non-parallel, curved surfaces, as shown below – we end up with three unknowns rather than two: z_1, z_2, and S_2.



Now, we have

$$t = cz_1 + cz_2 + xS_2 \; .$$

Generalizing, we get

$$t_j = \sum_{k=1}^{n} c_{jk} z_k + x_j S_2 \; .$$

In matrix form, we get

$$\begin{pmatrix} c_{11} & c_{12} & c_{13} & \cdot & c_{1n} & x_1 \\ c_{21} & c_{22} & c_{23} & \cdot & c_{2n} & x_2 \\ c_{31} & c_{32} & c_{33} & \cdot & c_{3n} & x_3 \\ c_{41} & c_{42} & c_{43} & \cdot & c_{4n} & x_4 \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \\ c_{m1} & c_{m2} & c_{m3} & \cdot & c_{mn} & x_m \end{pmatrix} \begin{pmatrix} z_1 \\ z_2 \\ z_3 \\ \cdot \\ z_n \\ s_2 \end{pmatrix} = \begin{pmatrix} t_1 \\ t_2 \\ t_3 \\ t_4 \\ \cdot \\ t_m \end{pmatrix},$$

where m = number of travel times, and n = number of receivers (depths to be calculated). We can now solve for the matrix for $z_1 \ldots z_n$ and S_2.

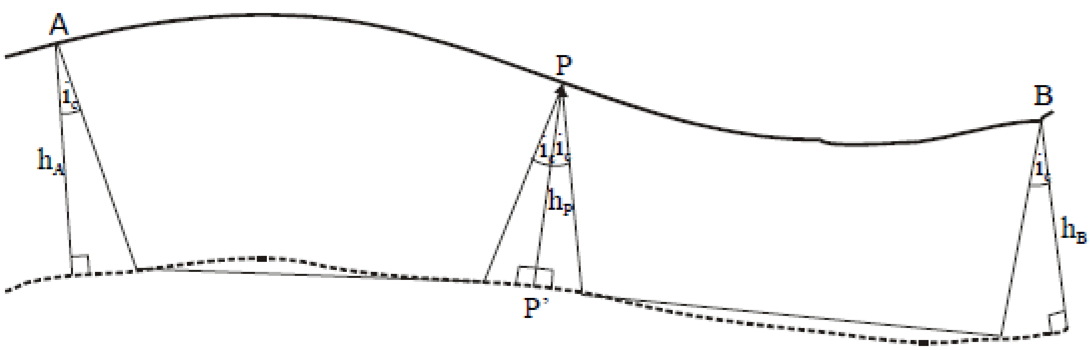
APPENDIX E THE RECIPROCAL TIME METHOD

The reciprocal time method is a much more "hands on" approach than the Time-term Method. Fewer assumptions are made, and the interpreter interacts with the software to a much greater degree, providing much more input.

Generally, the Reciprocal Method should be used when the desired result needs to be as detailed as possible. Read Palmer (1980) for a complete treatment of the Reciprocal Method.

The Reciprocal Method (and the Time-term Method, for that matter) generally requires more data because of its use of "delay-times", which require a refracted arrival from each direction (see Section 0, Page 245). Ideally, data is acquired such that a delay-time can be computed beneath each geophone. The depth is then computed from the delay-time and the velocity.

We will now introduce the concept of the **delay-time**.



Referring to the figure above, and following the derivation of Equation C-6 in Appendix C,

$$T_2 = 2h\frac{\cos(i_c)}{V_1} + \frac{x}{V_2}.$$

It is easy to show that

$$T_{AB} \cong \frac{h_A\cos(i_c)}{V_1} + \frac{\overline{AB}}{V_2} + \frac{h_B\cos(i_c)}{V_1}$$

$$T_{AP} \cong \frac{h_A \cos{(i_c)}}{V_1} + \frac{\overline{AP}}{V_2} + \frac{h_p \cos{(i_c)}}{V_1}$$

$$T_{BP} \cong \frac{h_B \cos(i_c)}{V_1} + \frac{\overline{BP}}{V_2} + \frac{h_p \cos(i_c)}{V_1}\,.$$

We define

$$t_0 = T_{AP} + T_{BP} - T_{AB}\,.$$

Equation E-1

Substituting,

$$t_0 = \left\{\frac{h_A \cos(i_c)}{V_1} + \frac{\overline{AP}}{V_2} + \frac{h_p \cos(i_c)}{V_1}\right\} + \left\{\frac{h_B \cos(i_c)}{V_1} + \frac{\overline{BP}}{V_2} + \frac{h_P \cos(i_c)}{V_1}\right\} - \left\{\frac{h_A \cos(i_c)}{V_1} + \frac{\overline{AB}}{V_2} + \frac{h_B \cos(i_c)}{V_1}\right\}$$

or

$$t_{0=} \frac{\overline{AP}}{V_2} + \frac{\overline{BP}}{V_2} + \frac{\overline{AB}}{V_2} + \frac{2h_p \cos{(i_c)}}{V_1}\,.$$

Referring to the figure, we see that

$$\overline{AB} = \overline{AP} + \overline{BP}\,.$$

Substituting,

$$t_0 = \frac{2h_P \cos(i_c)}{V_1}\,.$$

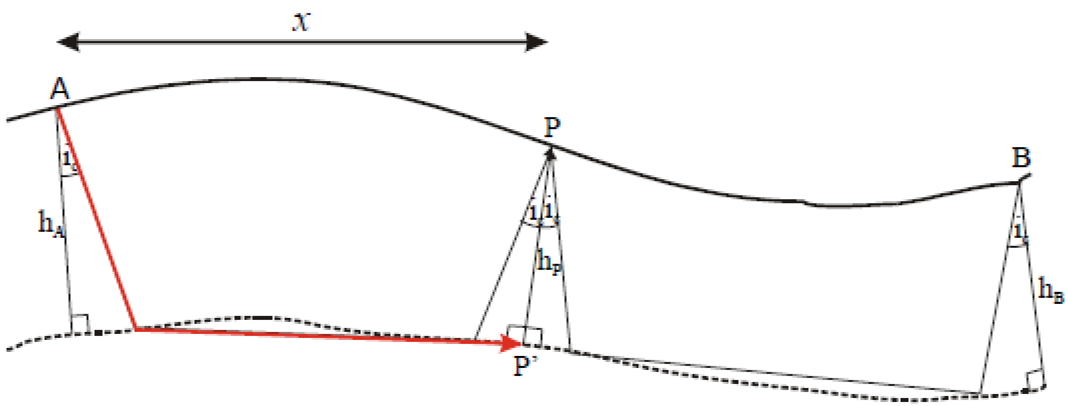
Now, t_0 is twice the time required for the seismic energy to travel from *P* to *P'*. We call $\boldsymbol{t_0/2}$ the "delay-time".

Delay-time D_T at point *P* =

$$D_T = \frac{t_0}{2} = \frac{h_P \cos(i_c)}{V_1}\,.$$

Equation E-2

Next, we shall examine the concept of **reduced travel times**. Computing reduced travel times is useful because it tends to remove the effect of changing layer thickness on the travel time curve and allows a better measurement of velocity. As will be seen, it also allows the computation of delay-time and hence, refractor depth.



Referring to the above figure, we define T'_{AP} (the reduced travel time at point P for a source at A) as $T_{AP'}$. This is represented by the red arrow. Upon examination, it should be apparent that a plot of T' vs. x, since all that changes with the position of P is the length of the ray traveling at V_2, will be roughly linear, unaffected by changes in thickness of the layer. Further, its slope will be $1/V_2$.

Mathematically, T_{AP}' can be expressed as follows:

$$T'_{AP} = T_{AP'} = T_{AP} - \frac{t_0}{2}.$$

From Equation E-1, we see that

$$T'_{AP} = T_{AP} - \frac{(T_{AP} + T_{BP} - T_{AB})}{2}.$$

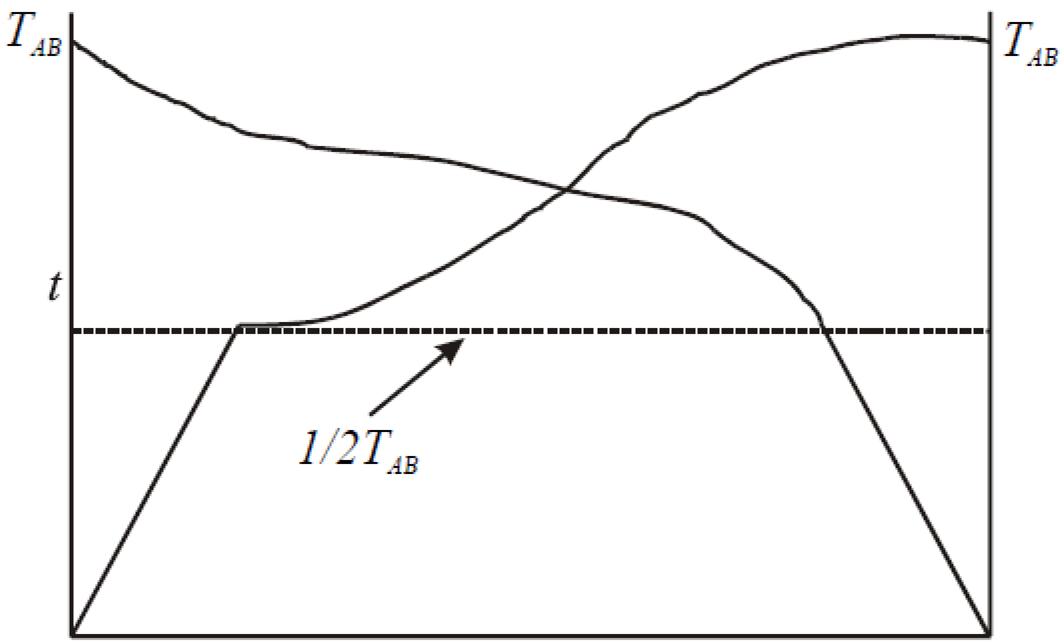
Equation E-3

Rearranging, we get

$$T'_{AP} = \frac{T_{AB}}{2} + \frac{(T_{AP} - T_{BP})}{2}.$$

Equation E-4

The above equation allows a graphical determination of the T' curve. Refer to the figure below:

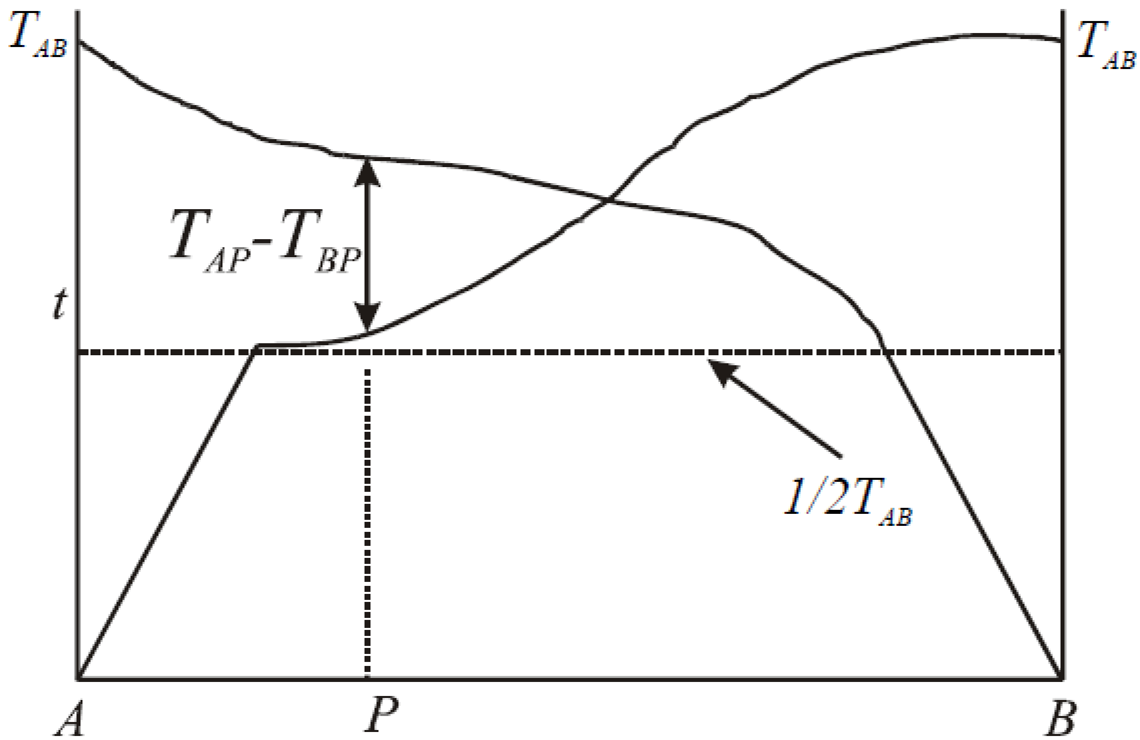


The travel time curves represent what you would expect to see from a velocity structure in which the thickness of layer 1 varies with *x*. T_{AB} is known as the "reciprocal time". We have drawn in ½T_{AB}, which is the first term in Equation E-4.

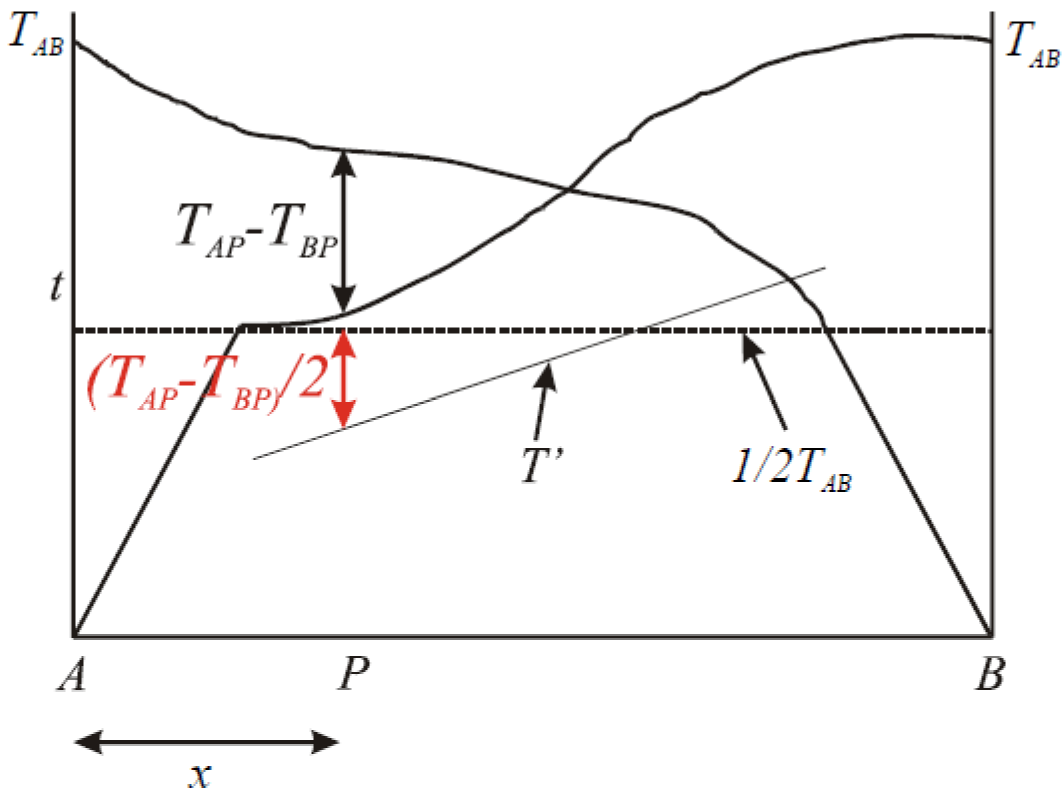
The second term,

$$\frac{(T_{AP} - T_{BP})}{2},$$

can also be determined graphically, as shown below:



Now, using Equation E-4, we can draw the reduced travel time curve by adding T_{AP}-$T_{BP}/2$ to the $T_{AB}/2$ line:



The slope of T' is $1/V_2$.

We now have everything we need to calculate the delay-time at point *P*. Combining Equations E-1,

$$t_0 = T_{AP} - \frac{(T_{AP} + T_{BP} - T_{AB})}{2},$$

and E-3,

$$T'_{AP} = T_{AP} - \frac{(T_{AP} + T_{BP} - T_{AB})}{2},$$

we see that

$$T'_{AP} = T_{AP} - \frac{t_0}{2}.$$

Combining with Equation E-2,

$$\frac{t_0}{2} = \frac{h_P \cos(i_c)}{V_1},$$

we get

$$T'_{AP} = T_{AP} - \frac{h_P \cos(i_c)}{V_1}.$$

Equation E-5

From Equation C-6 in Appendix C, it is fair to say that

$$T_{AP} \cong \frac{2h_P \cos(i_c)}{V_1} + \frac{x}{V_2}.$$

Equation E-6

Combining Equations E-5 and E-6 gives

$$T'_{AP} = \frac{h_P \cos{(i_c)}}{V_1} + \frac{x}{V_2}.$$

Equation E-7

We see from Equation E-2 that

$$D_{TP} = \frac{h_p \cos\ (i_c)}{V_1}.$$

Substituting into Equation E-7 yields

$$T'_{AP} = D_{TP} + \frac{x}{V_2}.$$

So the delay-time at point P is

$$D_{TP} = T'_{AP} - \frac{x}{V_2}.$$

Equation E-8

Depth can then be calculated by solving Equation E-2 for h_P:

$$h_p = \frac{D_{TP} V_1}{\cos\left(i_c\right)}.$$

Equation E-9

APPENDIX F THE TOMOGRAPHIC METHOD

The tomographic method involves the creation of an initial velocity model, and then iteratively tracing rays through the model, comparing the calculated travel times to the measured travel times, modifying the model, and repeating the process until the difference between calculated and measured travel times is minimized. The math is quite complex; what is presented here assumes a working understanding of upper-level calculus and linear algebra.

The essential goal is to find the minimum travel time between source and receiver for each source-receiver pair. This is accomplished by solving for *l* (raypath) and *s* (inverse velocity or "slowness"). Since we know neither, the problem is under-constrained, and we must use an iterative, least-squares approach.

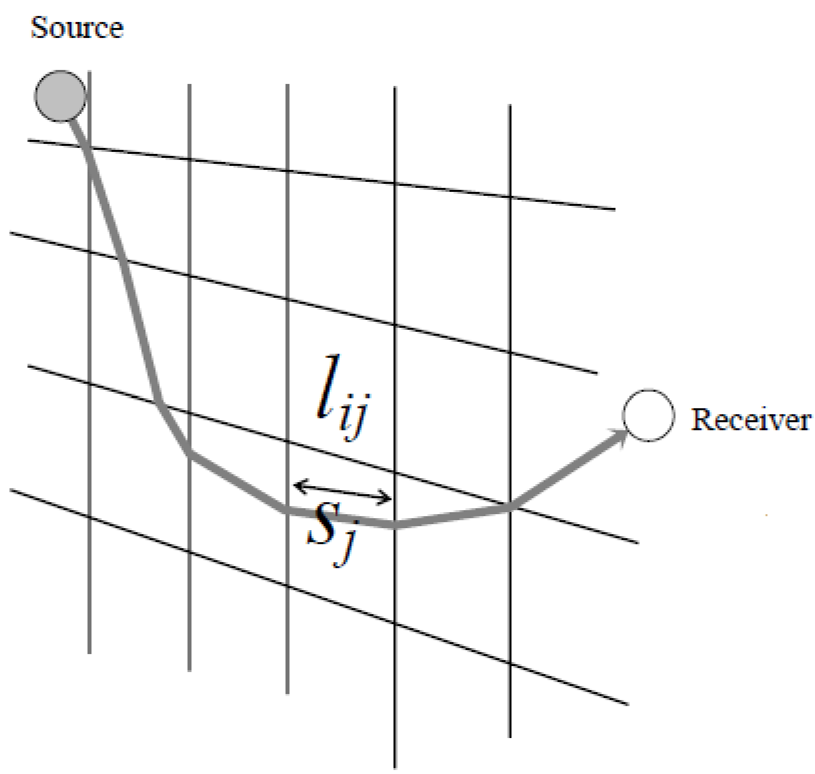


Figure F-1: Raypath through velocity cells.

Definition:

$$S = \frac{1}{v}.$$

S = "slowness"

v = velocity

l_{ij} = *raypath*

$$t_i = \int_x v(x) = \int_x S(x)dx$$

In discrete form, we get

$$t_i = S_1 l_{i1} + S_2 l_{i2} + S_3 l_{i3} + S_4 l_{i4} + \cdots + S_N l_N ,$$

or

$$t_i = \sum_{j=1}^{N} S_j l_{ij} .$$

We end up with *M* simultaneous equations (one for each travel time), and *N* unknowns:

$$t_1 = l_{11}S_1 + L_{12}S_2 + \cdots + l_{1N}S_N$$

$$t_2 = l_{21}S_1 + L_{22}S_2 + \cdots + l_{2N}S_N$$

$$t_3 = l_{31}S_1 + L_{32}S_2 + \cdots + l_{3N}S_N$$

• • • •

$$t_M = l_{M1}S_1 + L_{M2}S_2 + \cdots + l_{MN}S_N$$

In matrix notation, we get:

$$LS = \begin{pmatrix} l_{11} & l_{12} & . & . & l_{1N} \\ l_{21} & l_{22} & . & . & l_{2N} \\ l_{31} & l_{32} & . & . & l_{3N} \\ . & . & . & . & . \\ . & . & . & . & . \\ l_{M1} & l_{M2} & . & . & l_{MN} \end{pmatrix} \begin{pmatrix} S_1 \\ S_2 \\ . \\ . \\ S_N \end{pmatrix} = \begin{pmatrix} t_1 \\ t_2 \\ . \\ . \\ t_M \end{pmatrix} = T\ .$$

Raypaths Model Travel time

This is the **least-squares** method. Generally, $M > N$.

Example 1: Three equations, two unknowns:

$$2x_1 + x_2 = 11$$

$$4x_1 + x_2 = 17$$

$$6x_1 + x_2 = 23$$

Unknowns are x_1 and x_2. In matrix notation, we get:

$$AX = \begin{pmatrix} 2 & 1 \\ 4 & 1 \\ 6 & 1 \end{pmatrix} \begin{pmatrix} x_1 \\ x_2 \end{pmatrix} = \begin{pmatrix} 11 \\ 17 \\ 23 \end{pmatrix} = Y\ .$$

$$(AX = Y).$$

Matrix A is a Jacobian matrix:

$$f_1 = 2x_1 + x_2 - 11$$

$$f_2 = 4x_1 + x_2 - 17$$

$$f_3 = 6x_1 + x_2 - 23$$

or

$$A = \begin{pmatrix} 2 & 1 \\ 4 & 1 \\ 6 & 1 \end{pmatrix} = \begin{pmatrix} \frac{\partial f_1}{\partial x_1} & \frac{\partial f_1}{\partial x_2} \\ \frac{\partial f_2}{\partial x_1} & \frac{\partial f_2}{\partial x_2} \\ \frac{\partial f_3}{\partial x_1} & \frac{\partial f_3}{\partial x_2} \end{pmatrix}.$$

Error = $E = AX\text{-}Y$.

We wish to minimize the sum of squares errors:

$$E = (AX - Y)^T (AX - Y) = \|AX - Y\|^2 \rightarrow Minimize$$

We set the derivative of E to zero,

$$\frac{dE}{dX} = 2A^T (AX - Y) = 0 \,,$$

and solve for X:

$$(A^T A)X = A^T Y \,.$$

Back to our three equations,

$$f_1 = 2x_1 + x_2 - 11$$

$$f_2 = 4x_1 + x_2 - 17$$

$$f_3 = 6x_1 + x_2 - 23$$

and solving,

$$(A^T A)X = \begin{pmatrix} 2 & 4 & 6 \\ 1 & 1 & 1 \end{pmatrix} \begin{pmatrix} 2 & 1 \\ 4 & 1 \\ 6 & 1 \end{pmatrix} \begin{pmatrix} x_1 \\ x_2 \end{pmatrix} = \begin{pmatrix} 2 & 4 & 6 \\ 1 & 1 & 1 \end{pmatrix} \begin{pmatrix} 11 \\ 17 \\ 23 \end{pmatrix} = A^T Y ,$$

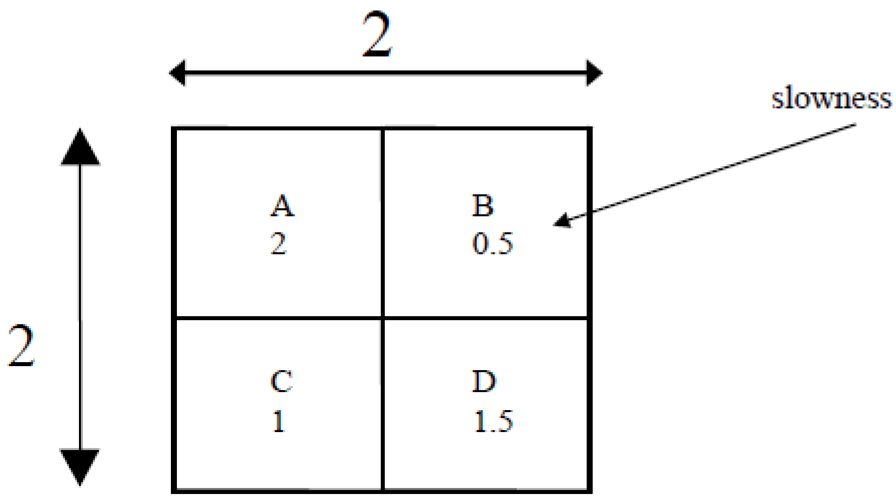
$$(A^T A)X = \begin{pmatrix} 56 & 12 \\ 12 & 3 \end{pmatrix} \begin{pmatrix} x_1 \\ x_2 \end{pmatrix} = \begin{pmatrix} 228 \\ 51 \end{pmatrix} = A^T Y ,$$

$$X = (A^T A)^{-1} A^T Y = \begin{pmatrix} .125 & -0.5 \\ -0.5 & 2.3333 \end{pmatrix} \begin{pmatrix} 228 \\ 51 \end{pmatrix} = \begin{pmatrix} 3 \\ 5 \end{pmatrix} .$$

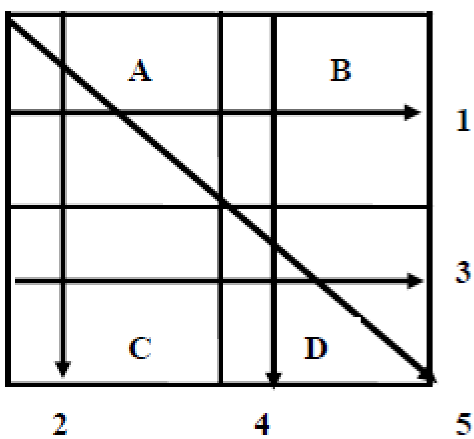
so $x_1 = 3$ and $x_2 = 5$.

Example 2:

Four cells (velocity unknown).



5 raypaths:



Observed travel times are:

$$T = \begin{pmatrix} t_1 \\ t_2 \\ t_3 \\ t_4 \\ t_5 \end{pmatrix} = \begin{pmatrix} 2 + 0.5 \\ 2 + 1 \\ 1 + 1.5 \\ 0.5 + 1.5 \\ 2\sqrt{2} + 1.5\sqrt{2} \end{pmatrix} = \begin{pmatrix} 2.5 \\ 3 \\ 2.5 \\ 2 \\ 4.95 \end{pmatrix}.$$

Jacobian matrix A (length of ray passing through each cell) is:

$$L = \begin{pmatrix} 1 & 1 & 0 & 0 \\ 1 & 0 & 1 & 0 \\ 0 & 0 & 1 & 1 \\ 0 & 1 & 0 & 1 \\ \sqrt{2} & 0 & 0 & \sqrt{2} \end{pmatrix}.$$

$$t_i = S_1 l_{i1} + S_2 l_{i2} + S_3 l_{i3} + S_4 l_{i4} + \cdots + S_N l_{iN}$$

$$\frac{\partial t_i}{\partial S_j} = l_{ij}$$

Equation to be solved:

$$LS = \begin{pmatrix} 1 & 1 & 0 & 0 \\ 1 & 0 & 1 & 0 \\ 0 & 0 & 1 & 1 \\ 0 & 1 & 0 & 1 \\ \sqrt{2} & 0 & 0 & \sqrt{2} \end{pmatrix} \begin{pmatrix} S_1 \\ S_2 \\ S_3 \\ S_4 \end{pmatrix} = \begin{pmatrix} 2.5 \\ 3 \\ 2.5 \\ 2 \\ 4.95 \end{pmatrix} = T\,.$$

Normal equation:

$$L^T LS = \begin{pmatrix} 4 & 1 & 1 & 2 \\ 1 & 2 & 0 & 1 \\ 1 & 0 & 2 & 1 \\ 2 & 1 & 1 & 4 \end{pmatrix} \begin{pmatrix} S_1 \\ S_2 \\ S_3 \\ S_4 \end{pmatrix} = \begin{pmatrix} 12.5 \\ 4.5 \\ 5.5 \\ 11.5 \end{pmatrix} = l^T T\ .$$

$$S_T = (s_1 \quad S_2 \quad S_3 \quad S_4) = (2 \quad 0.5 \quad 1 \quad 1.5)$$

- Jacobian matrix requires raypath.
- Raypath cannot be calculated without a velocity model.
- Cannot solve at once.
- Must use a **non-linear least-squares** method.

If the Jacobian matrix is not a constant,

$$y(z) = x_1 z - x_2 e^{-zx_3}$$

$$A = \begin{pmatrix} \frac{\partial y(z_1)}{\partial x_1} & \frac{\partial y(z_1)}{\partial x_2} & \frac{\partial y(z_1)}{\partial x_3} \\ \frac{\partial y(z_2)}{\partial x_1} & \frac{\partial y(z_2)}{\partial x_2} & \frac{\partial y(z_2)}{\partial x_3} \\ \vdots & \vdots & \vdots \\ \frac{\partial y(z_m)}{\partial x_1} & \frac{\partial y(z_m)}{\partial x_2} & \frac{\partial y(z_m)}{\partial x_3} \end{pmatrix} =$$

$$\begin{pmatrix} z_1 & -e^{-z_1 x_3} & -x_2 z_1 e^{-z_1 x_3} \\ z_2 & -e^{-z_2 x_3} & -x_2 z_2 e^{-z_2 x_3} \\ . & . & . \\ . & . & . \\ z_m & -e^{-z_m x_3} & -x_2 z_m e^{-z_m x_3} \end{pmatrix},$$

and parameter x is in matrix A.

Iterative Solution of a non-linear least-squares matrix:

1) Calculate theoretical value y_0 for initial value x_0.

$$y_0(z) = y(z, x_0)$$

2) Calculate residuals (Δy) between theoretical value y_0 and y.

$$\Delta y = y - y_0$$

3) Calculate correction value for $x(\Delta y)$ by the least-squares method.

$$(A^T A)\Delta \mathrm{x} = \mathrm{A}^T \Delta \mathrm{y}$$

4) Calculate new estimate for x_1.

5) Return to Step 1.

6) Stop when residual error reaches acceptable value.

Example 3:

Model:

$$y(z) = x_1 z - x_2 e^{-z x_3}$$

True solution:

$$x = \begin{pmatrix} x_1 \\ x_2 \\ x_3 \end{pmatrix} = \begin{pmatrix} 1 \\ 2 \\ 1 \end{pmatrix}.$$

Eleven observed data:

z	y(z)
0	-2
1	0.26421
2	1.729329
3	2.900426
4	3.963369
5	4.986524
6	5.995042
7	6.998176
8	7.999329
9	8.999753
10	9.999909

Partial differentiation:

$$\frac{\partial y}{\partial x_1} = z$$

$$\frac{\partial y}{\partial x_2} = -e^{-zx_3}$$

$$\frac{\partial y}{\partial x_3} = x_2 z e^{-zx_3}$$

Initial model:

$$x_0 = \begin{pmatrix} x_1 \\ x_2 \\ x_3 \end{pmatrix} = \begin{pmatrix} 12 \\ 23 \\ 2 \end{pmatrix}$$

Jacobian matrix *A*:

$$A_0 = \begin{pmatrix} \frac{\partial y(z_1)}{\partial x_1} & \frac{\partial y(z_1)}{\partial x_2} & \frac{\partial y(z_1)}{\partial x_3} \\ \frac{\partial y(z_2)}{\partial x_1} & \frac{\partial y(z_2)}{\partial x_2} & \frac{\partial y(z_2)}{\partial x_3} \\ . & . & . \\ . & . & . \\ \frac{\partial y(z_{11})}{\partial x_1} & \frac{\partial y(z_{11})}{\partial x_2} & \frac{\partial y(z_{11})}{\partial x_3} \end{pmatrix}$$

$$= \begin{pmatrix} z_1 & -e^{-z_1 x_3} & -x_2 z_1 e^{-z_1 x_3} \\ z_2 & -e^{-z_2 x_3} & -x_{2z_2} e^{-z_2 x_3} \\ . & \vdots & \vdots \\ . & \vdots & \vdots \\ z_{11} & -e^{-z_{11} x_3} & -x_2 z_{11} e^{-z_{11} x_3} \end{pmatrix}$$

$$= \begin{pmatrix} 0 & -1 & 0 \\ 1 & -0.1353352832 & 0.4060058497 \\ 2 & -0.0183156389 & 0.1098938333 \\ 3 & -0.0024787522 & 0.0223087696 \\ 4 & -0.0003354626 & 0.0040255515 \\ 5 & -0.0000453999 & 0.0006809989 \\ 6 & -0.0000061442 & 0.0001105958 \\ 7 & -0.0000008315 & 0.0000174621 \\ 8 & -0.0000001125 & 0.0000027008 \\ 9 & -0.0000000152 & 0.0000004112 \\ 10 & -0.0000000021 & 0.00000000618 \end{pmatrix}$$

Observed data:

$$Y^T = (-2.0000 \quad 0.264241 \quad 1.729329 \quad 2.900426 \quad 3.963369 \quad 4.986524 \quad 5.995042 \quad 6.998176 \quad 7.999329 \quad 8.999753 \quad 9.999909)$$

Theoretical data for initial model:

$$Y_0^T = (-3.0000 \quad 1.5940 \quad 3.9451 \quad 5.9926 \quad 7.9990 \quad 9.9999 \quad 12.0000 \quad 14.0000 \quad 16.0000 \quad 18.0000 \quad 20.0000)$$

Residual vector:

$$\Delta Y = Y_0 - Y$$

$$\Delta Y_0^T = (-1.0000 \quad 1.3298 \quad 2.2157 \quad 3.0921 \quad 4.0356 \quad 5.0133 \quad 6.0049 \quad 7.0018 \quad 8.0007 \quad 9.0002 \quad 10.0001)$$

RMSE (Root Mean Square Error):

$$RMSE_0 = \sqrt{\frac{\Delta Y_0^T \Delta Y_0}{11}} = 5.9449$$

$$A_0^T A_0 = \begin{pmatrix} 385 & -0.181 & 0.71304 \\ -0.181 & 1.0187 & -0.057 \\ 0.71304 & -0.057 & 0.17743 \end{pmatrix}$$

$$A_0^T \Delta Y_0 = \begin{pmatrix} 386.3 \\ 0.7702 \\ 0.8728 \end{pmatrix}$$

Solve:

$$(A_0^T A_0)\Delta X_0 = A_0^T \Delta Y_0$$

Get:

$$\Delta X_0 = \begin{pmatrix} 1.0016 \\ 1.0021 \\ 1.2162 \end{pmatrix}$$

New estimated value for $X(X_1)$:

$$X_1 = X_0 - \Delta X$$

$$X_1 = \begin{pmatrix} 2 \\ 3 \\ 2 \end{pmatrix} - \begin{pmatrix} 1.0016 \\ 1.0021 \\ 1.2162 \end{pmatrix} = \begin{pmatrix} 0.9984 \\ 1.9979 \\ 0.7838 \end{pmatrix}$$

Calculate residuals (RMSE) from new estimation of $X(X_1)$:

$$RMSE_1 = \sqrt{\frac{\Delta Y_1^T \Delta Y_1}{11}} = 0.0793$$

In second calculation,

$$A_1^T A_1 = \begin{pmatrix} 385 & -1.543 & 8.19332 \\ -1.543 & 1.2635 & -0.6652 \\ 8.19332 & -0.6652 & 2.02955 \end{pmatrix}$$

$$A_1^T \Delta Y_1 = \begin{pmatrix} -1.854 \\ 0.123 \\ -0.372 \end{pmatrix}$$

Correction is:

$$\Delta X_1 = \begin{pmatrix} -0.001 \\ 0.002 \\ -0.179 \end{pmatrix}$$

Corrected model is:

$$X_2 = \begin{pmatrix} 0.9984 \\ 1.9979 \\ 0.7838 \end{pmatrix} - \begin{pmatrix} -0.001 \\ 0.002 \\ -0.179 \end{pmatrix} = \begin{pmatrix} 0.9994 \\ 1.9959 \\ 0.9625 \end{pmatrix} \cong \begin{pmatrix} 1 \\ 2 \\ 1 \end{pmatrix}$$

Residuals are:

$$RMSE_2 = \sqrt{\frac{\Delta Y_2^T \Delta Y_2}{11}} = 0.0122 \cong 0$$

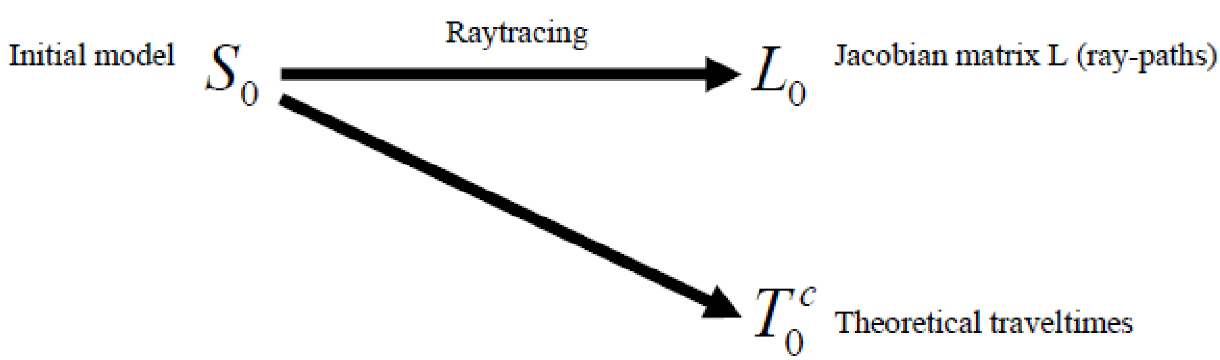
Summary

Simultaneous equations:

$$LS = T$$

L is a function of S (non-linear problem):

$$L(S)S = T$$



Then,

$$\Delta T_0 = T^0 - T_0^C = T^0 - L_0 S_0 \ .$$

Calculate correction:

$$L_0 \Delta S_0 = \Delta T_0$$

Correct model:

$$S_1 = S_0 + \Delta S_0$$

In the k^th iteration:

$$\Delta T_k = T^0 - T_k^c = T^0 - L_k S_k$$

$$L_k \Delta S_k = \Delta T_k$$

$$S_{k+1} = S_k + \Delta S_k$$

Solve large matrix:

Use diagonal:

$$L^T L\Delta S = \begin{pmatrix} \sum_{i-1}^{n} l_{i1}^2 & 0 & . & 0 \\ 0 & \sum_{i-1}^{n} l_{i2}^2 & . & 0 \\ . & . & . & . \\ 0 & 0 & . & \sum_{i-1}^{n} l_{im}^2 \end{pmatrix} \begin{pmatrix} \Delta S_1 \\ \Delta S_2 \\ . \\ \Delta S_m \end{pmatrix} = \begin{pmatrix} \sum_{i=1}^{n} \Delta t_i l_{i1} \\ \sum_{i=1}^{n} \Delta t_i l_{i2} \\ \sum_{i=1}^{n} \Delta t_i l_{i3} \end{pmatrix}$$

$$\Delta S_j = \frac{\sum_{i=1}^{n} \Delta T_i l_{\ddot{U}}}{\sum_{i=1}^{n} l_{\ddot{U}}^2} \cdot \alpha \cong \frac{\sum_{i=1}^{n} \left(\Delta t_i \frac{l_{\ddot{U}}}{L_i} \right)}{\sum_{i=1}^{n} l_{\ddot{U}}} = \frac{\sum_{i=1}^{n} \left(\Delta t_i \frac{t_{\ddot{U}}}{T_i^c} \right)}{\sum_{i=1}^{n} l_{\ddot{U}}}$$

$$L_i = \sum_{j=1}^{m} l_{\ddot{U}}$$

$$T_i = \sum_{j=1}^{m} t_{\ddot{U}}$$

Example 4:

From Example 2:

$$LS = \begin{pmatrix} 1 & 1 & 0 & 0 \\ 1 & 0 & 1 & 0 \\ 0 & 0 & 1 & 1 \\ 0 & 1 & 0 & 1 \\ \sqrt{2} & 0 & 0 & \sqrt{2} \end{pmatrix} \begin{pmatrix} s_1 \\ s_2 \\ s_3 \\ s_4 \end{pmatrix} = \begin{pmatrix} 2.5 \\ 3 \\ 2.5 \\ 2 \\ 4.949747 \end{pmatrix} = T$$

Initial model:

$$S_0^T = \begin{pmatrix} 1 & 1 & 1 & 1 \end{pmatrix}$$

Calculate residuals:

$$\Delta T_0 = T - T_0^c = \begin{pmatrix} 2.5 \\ 3 \\ 2.5 \\ 2 \\ 4.9497 \end{pmatrix} - \begin{pmatrix} 2 \\ 2 \\ 2 \\ 2 \\ 2.8284 \end{pmatrix} = \begin{pmatrix} 0.5 \\ 1 \\ 0.5 \\ 0 \\ 2.1213 \end{pmatrix}$$

RMSE:

$$RMSE_0 = \sqrt{\frac{\Delta T^T \Delta T}{5}} = 2.44949$$

e.g., in the first cell,

$$\Delta S_j = \frac{\sum_{i=1}^{n} \left({}^{\Delta t_i}/_{L_i} \right) L_{\ddot{U}}}{\sum_{i=1}^{n} L_{\ddot{U}}} = \frac{\frac{1}{2} \times 0.5 \times \frac{1}{2} \times 1 + \frac{\sqrt{2}}{2\sqrt{2}} \times 2.1213}{1 + 1 + \sqrt{2}}$$

$$= 0.53033$$

Similarly,

$$S_1 = S_0 + \Delta S = \begin{pmatrix} 1 \\ 1 \\ 1 \\ 1 \end{pmatrix} + \begin{pmatrix} 053033 \\ 0.125 \\ 0.375 \\ 0.383883 \end{pmatrix} = \begin{pmatrix} 1.53033 \\ 1.125 \\ 1.375 \\ 1.383883 \end{pmatrix}$$

(Normal equation is not required)

Second iteration:

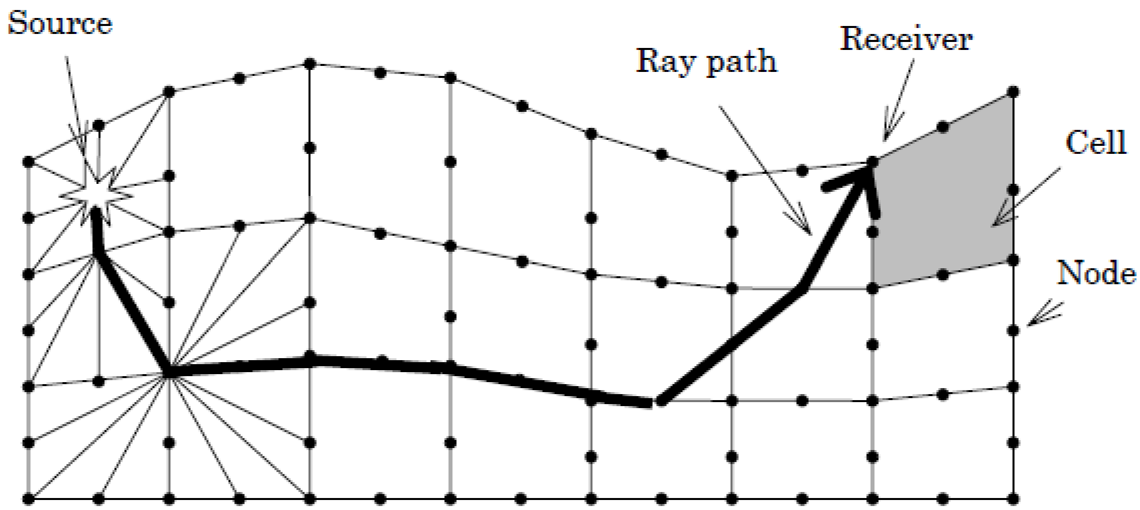
$$\Delta T_1 = T - T_1^c = \begin{pmatrix} 2.5 \\ 3 \\ 2.5 \\ 2 \\ 4.9497 \end{pmatrix} - \begin{pmatrix} 2.65533 \\ 2.90533 \\ 2.75888 \\ 2.50888 \\ 4.12132 \end{pmatrix} = \begin{pmatrix} -0.1553 \\ 0.09467 \\ -0.2589 \\ -0.5089 \\ 0.82843 \end{pmatrix}$$

RMSE:

$$RMSE_1 = \sqrt{\frac{\Delta T^T \Delta T}{5}} = 1.02243$$

General Summary

Calculation of travel times by raytracing,



we want to calculate the shortest path between A and B:

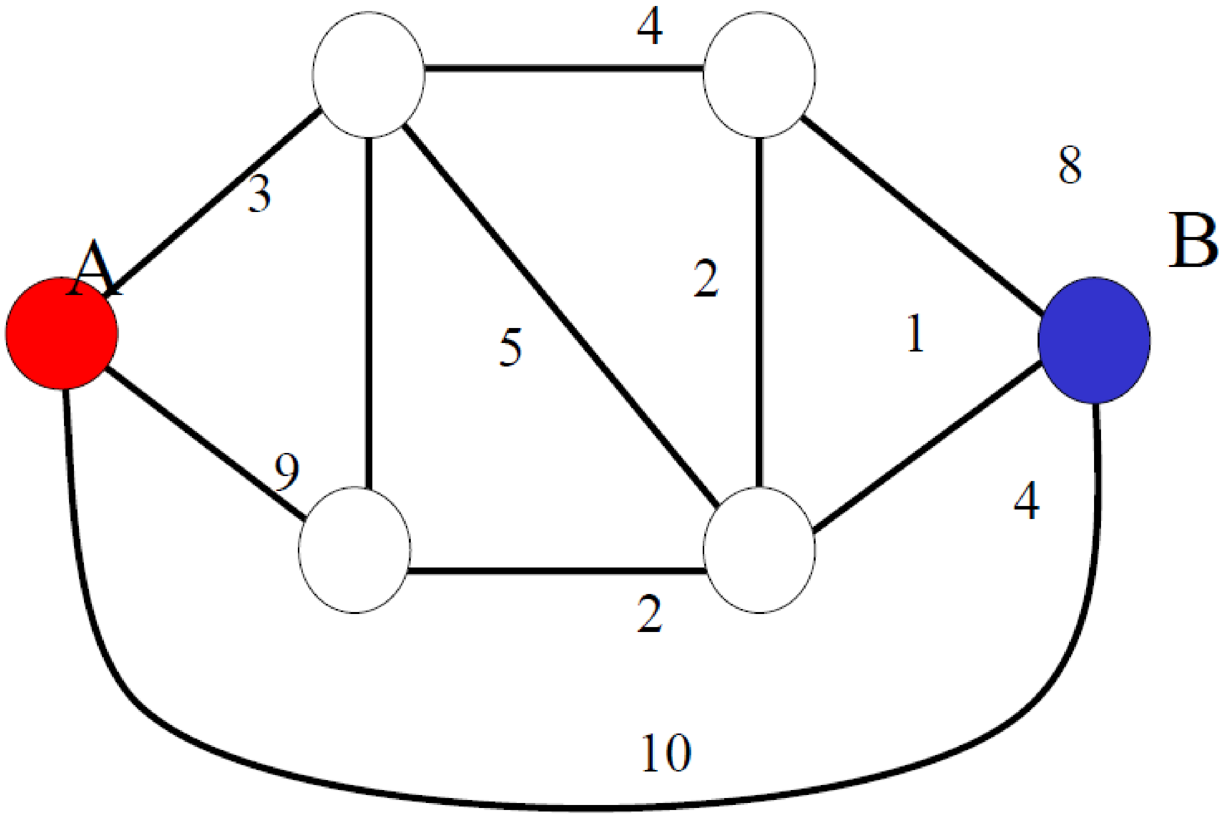
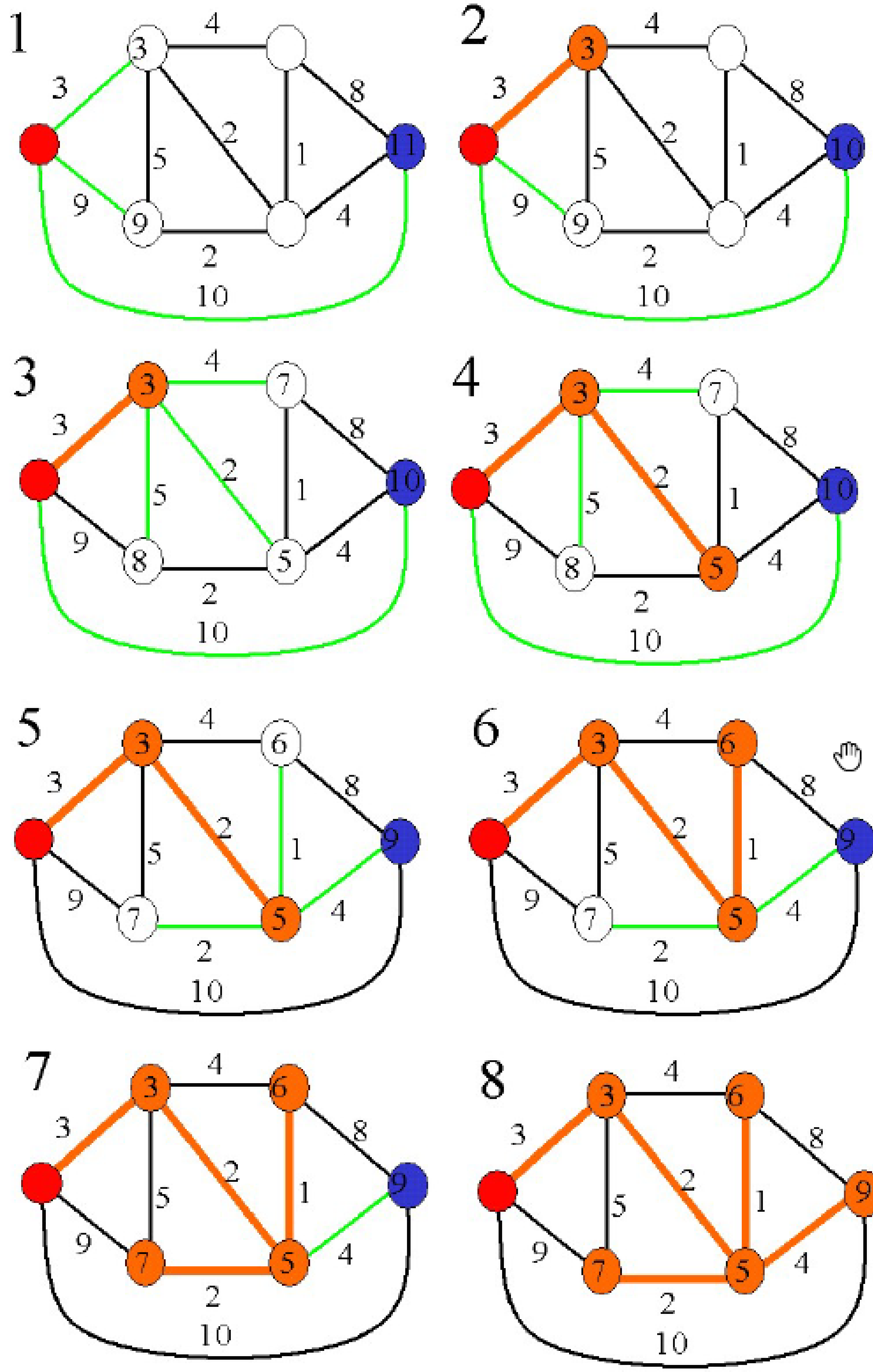
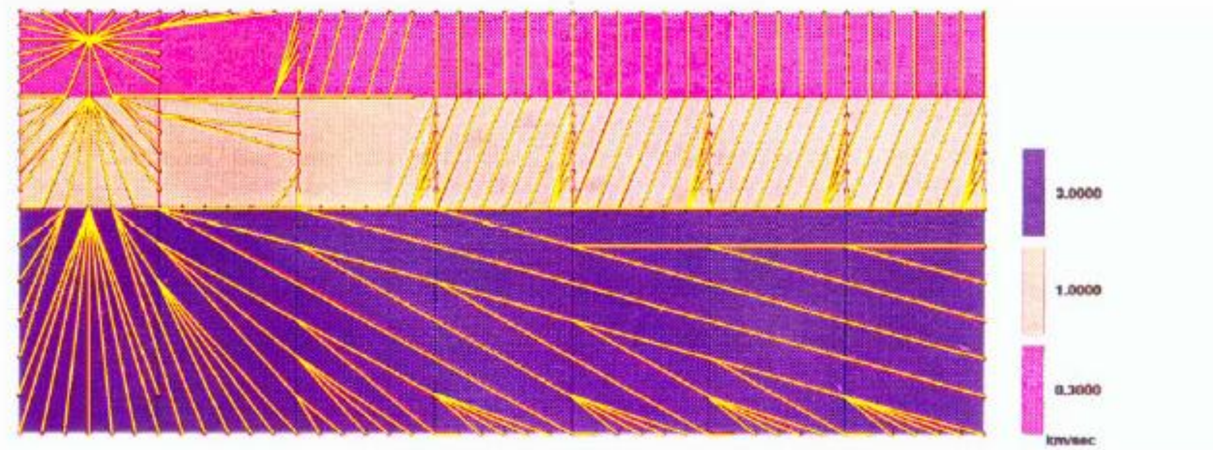


Figure F-2: Finding the shortest path between A and B.


1
3 4 3 8 11 2 5 1 9 9 4 2 10
2
3 4 3 8 10 2 5 1 9 9 4 2 10
3
3 4 3 7 8 10 2 5 1 9 8 5 4 2 10
4
3 4 3 7 8 10 2 5 1 9 8 5 4 2 10
5
3 4 3 6 8 9 2 5 1 9 7 5 4 2 10
6
3 4 3 6 8 9 2 5 1 9 7 5 4 2 10
7
3 4 3 6 8 9 2 5 1 9 7 5 4 2 10
8
3 4 3 6 8 9 2 5 1 9 7 5 4 2 10



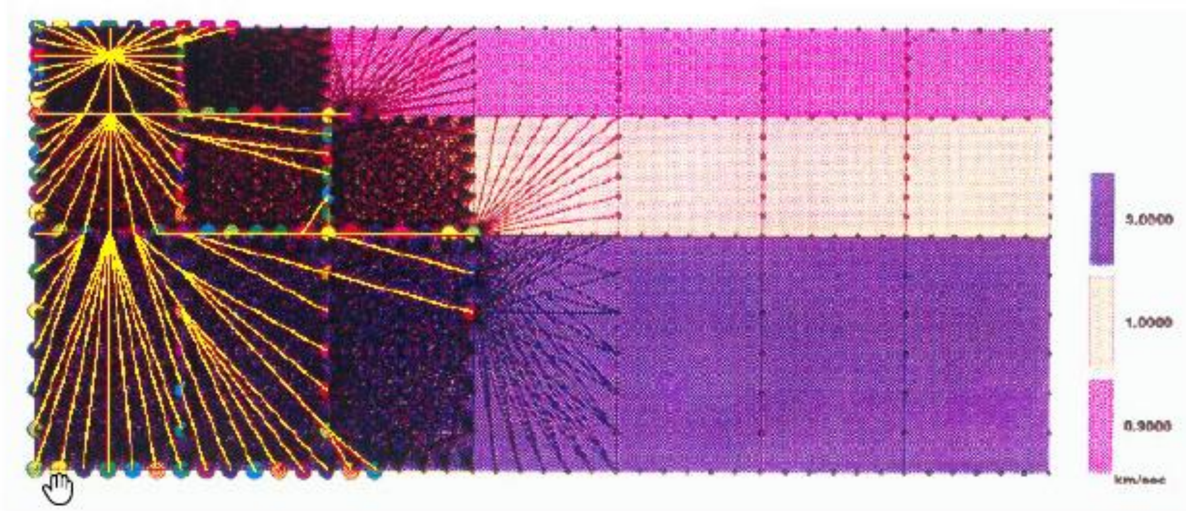


Figure F-3: Progression of raypaths through velocity model.

APPENDIX G RECOMMENDED READING

Dobrin, Milton B., *Introduction to Geophysical Prospecting, 629 p. 1976.*

Haeni, F.P., *Application of seismic-refraction techniques to hydrologic studies*, USGS TWRI Book 2, Chapter D2. 1988.

Laymon, Douglas E., and Robert H. Gilkeson, *Application of seismic refraction methods to evaluate regional ground-water resources.*

Palmer, Derecke, *The generalized Reciprocal Method of seismic refraction interpretation,* Society of Exploration Geophysicists, 104 p. 1980.

Redpath, Bruce B. *Seismic refraction exploration for engineering site investigations*, Explosive Excavation Research Laboratory, Livermore, California. 1973.

Visit our literature page for a complete list of available downloadable references.

Geometrics, Inc. takes no responsibility for any interpretation of data or effects resulting from use of the SeisImager software package or its support of the SeisImager software package.