User's Manual
SeisOpt® ReMi ™ Version 3.0

For support contact support@optimsoftware.com

Optim LLC
UNR – MS 174
1664 N. Virginia St.
Reno, NV 89557-0141
USA

Phone: 775.784.6613 * Fax: 775.784.1833

CONTENTS

1.0 SeisOpt® ReMi™ Software Overview

SeisOpt ReMi software uses refraction microtremor recordings from standard refraction equipment to estimate 30 m (100 ft) average shear wave velocities and one- and two-dimensional shear wave profile down to depths of 100 m with 5%-15% accuracy, with the accuracy decreasing with depth. The software is based on the refraction microtremor (ReMi) technique described by Louie (2001) (http://www.optimsoftware.com/white_papers/images/remi.pdf). A field tutorial on how to acquire ReMi data is provided in the insert of this user's manual. Before collecting the data for ReMi software analysis, please read the field tutorial carefully and contact support@optimsoftware.com with any questions about the acquisition parameters. The frequency of the available geophones, distance between geophone takeouts, total length of the recording array, and the distribution of subsurface velocities will determine the maximum depth up to which shear wave velocities can be derived.

Current techniques of estimating 30 m (100 ft) average shear wave velocities ($V_{s100'}$) and one-dimensional shear velocities for assessment of earthquake site response are too costly for use at most construction sites. They require large sources to be effective in noisy urban settings, or specialized independent recorders laid out in an extensive array. SeisOpt ReMi software uses microtremor noise recordings made on 200-m-long lines of seismic refraction equipment can estimate shear velocity with 5-15% accuracy, often to 100-m depths. Shorter lines and higher frequency geophones will yield velocities down to shallower depths. It uses standard refraction equipment, simple recording with no source, a wavefield transformation data processing technique (ReMi Vspect module, see Section 3.0), and an interactive Rayleigh-wave dispersion-modeling tool (ReMi Disper module, see Section 4.0). Thus, the refraction microtremor exploits the most effective aspects of the microtremor, spectral analysis of surface wave (SASW), and multi channel analysis of surface wave (MASW) techniques (Louie, 2001). The slowness-frequency (slowness is inverse of velocity) wavefield transformation is particularly effective in allowing accurate picking of Rayleigh-wave phase-velocity dispersion curves despite the presence of waves propagating across the linear array at high apparent velocities, higher-mode Rayleigh waves, body waves, airwaves, and incoherent noise (Louie, 2001).

2.0 Installing and Getting Started with SeisOpt ReMi

2.1 Installing SeisOpt ReMi

Begin the installation of SeisOpt ReMi by inserting the installation CD into the CD drive. The install process should start automatically. If it does not, you can go to the CD drive and click on the Setup.exe file, which has the icon, shown in Figure 1.

Figure 1: Setup.exe icon. Click this to start the installation, if it does not start automatically on insertion of the install CD into the CD drive.

When the install is complete, a ‘‘Setup Complete’’ window will pop open. You have the option of choosing to read the README file or clicking on 'Finish' to end the installation. The README file gives you directions on how to obtain your license from Optim to run SeisOpt@2D. The default installation directory is **C:\Optim\ReMiv30**. A successful installation will create three SeisOpt ReMi desktop icons, as follows:

- SeisOpt ReMi Vspect module (Figure 2a, see Section 3.0), which has tools to read in the recorded data and create a Rayleigh wave phase-velocity dispersion curve.

Figure 2a: SeisOpt ReMi Vspect module icon

- SeisOpt ReMi Disper module (Figure 2b, see Section 4.0), which is the interactive, one-dimensional shear wave velocity-modeling tool.

Figure 2b: SeisOpt ReMi Disper module icon

- SeisOpt ReMi Registration tool (Figure 2c), which has to be used to register the software before use.

Figure 2c: SeisOpt ReMi Registration icon

2.2 Authorizing and Registering SeisOpt ReMi

SeisOpt ReMi software modules are protected by encryption software and so you need to obtain the proper authorization from Optim to be able to start using them. The following sections outline this procedure:

2.2.1 Registering SeisOpt ReMi software

1. The first step is to register SeisOpt ReMi. Note that you can do the authorization and registration at the same time. That is, send the registration key (Section 2.2.1 and site code at the same time to Optim.
2. To register double-click on the "Register_ReMi" icon (Figure 2c) that appears on the desktop after installation.
3. A command (MS-DOS) window will open up (Figure 3), displaying the "Registration Code".
4. Send this code, along with your company name and address to Optim, via email to support@optimsfotware.com, phone or fax.
5. **DO NOT** quit the registration window until after you have entered the registration key. Else, you will not be able to run this module.
6. A registration key will be returned, which you will need to enter to authorize the program.
7. On successful registration window will either automatically close or you can close the window.

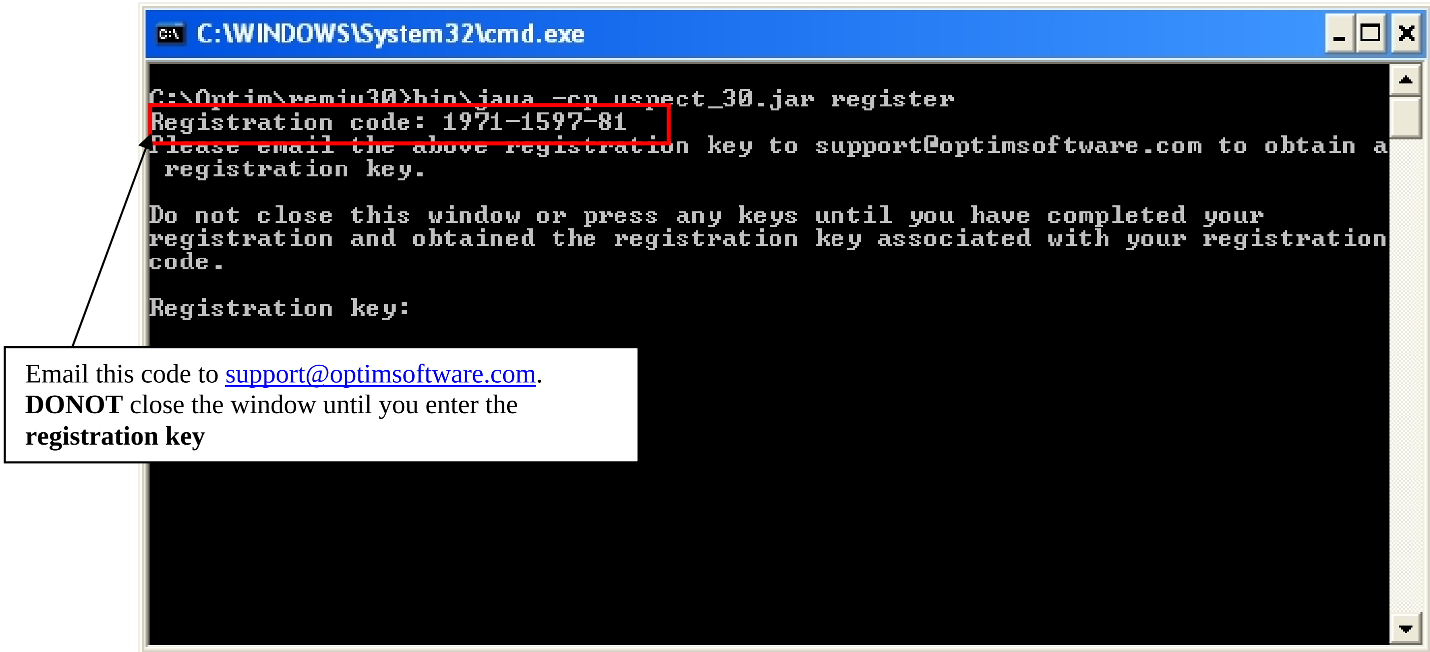


Figure 3: A command (MS-DOS) window opens up with the ''Registration Code''

2.2.2 Authorizing SeisOpt ReMi Vspect and Disper Modules

The next step is to authorize the license for SeisOpt ReMi Vspect and Disper modules. Note that you can do the authorization and registration at the same time. That is, send the registration key (Section 2.2.1 and site code at the same time to Optim. To do so, following the steps below:

1. Copy the file 'Cks.exe' that is present in the installation CD to the install directory C:\Optim\ReMiv30\
2. If you are installing and running SeisOpt ReMi on a Windows **NT**, **2000, or XP** machine you need to log in as the **local administrator** and run **setupex.** Find the 'setupex' file in the installation directory (globe icon show in Figure 4). Run 'setupex' by double clicking the globe icon from a Windows-explorer window.

Figure 4: Setupex icon. Double-click on this after installation, if the host is an NT, 2000, or XP machine.

3. Click on the ReMi Disper icon (Figure 2b) on the desktop.
4. When the license configuration window appears (Figure 5), a **Site Code** should be displayed, as shown in Figure 5. If not, click the **'Display Site Code'** button.

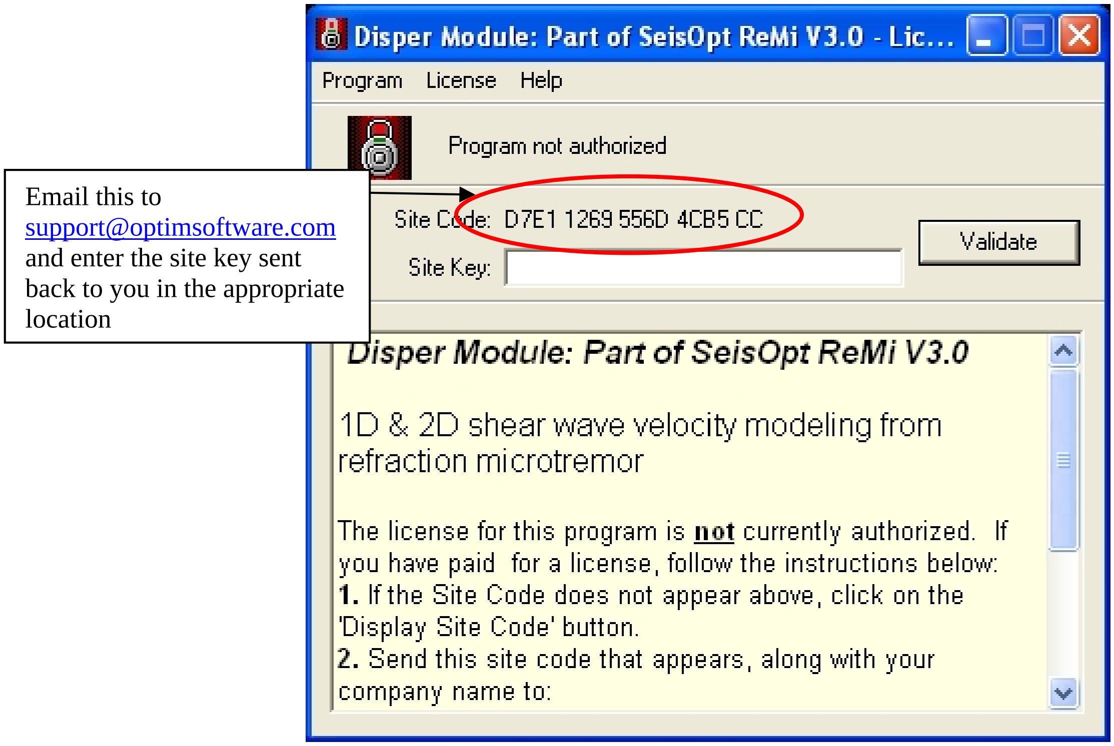


Figure 5: License configuration window for ReMi Disper module displaying the 'Site Code'

5. Send this site code to Optim, by email (support@optimsoftware.com), phone (775.784.6613), or fax (775.784.1833). This same **Site Code** will reappear each time the license configuration window is opened until a **Site Key** is entered.
6. Upon receiving your **Site Code**, Optim will generate a **Site Key** and send it to you. Enter this **Site Key** in the blank line below the site code, and click **'Validate'**.
7. Each time you run ReMi Disper, a small window briefly appears (Figure 6) describing the status of the license, then closes, and the program starts. DO NOT hit `ENTER/RETURN` when this window appears.

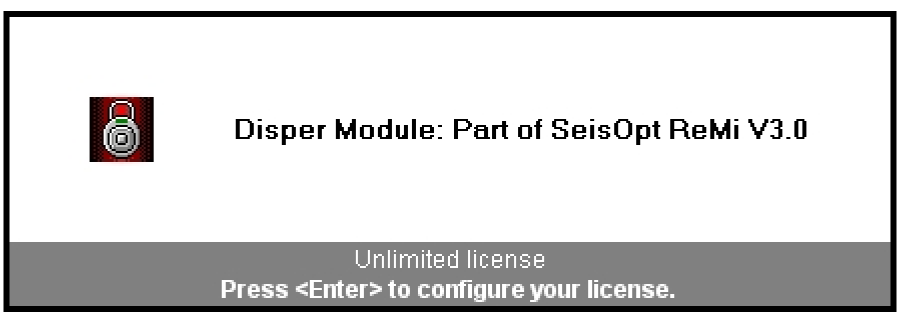


Figure 6: License check window appears each time ReMi Vspect or ReMi Disper module is run

8. SeisOpt ReMi will not run on any other computer, without obtaining another Registration Key and Site Key or authorization to transfer the license (sees Section 6.2).

2.2.3 Preventing loss of license

SeisOpt ReMi Disper module software protection mechanism works by storing hidden files in the PC. As a result, the user should make sure these files are not erased during disk de-fragmentation or

while running an anti-virus scan. For example, this is known to happen when running Speed Disk, a de-fragmentation utility included in Symantec's Norton Utilities. This also happens while running Norton Anti-Virus Utility. Loss of these files will result in the loss of license to run SeisOpt.
To prevent this loss, do the following:

1. Open Speed Disk, and choose **File, Options, Customize,** and **Unmovable Files.**
2. Specify that the *.ENT, *.RST, .KEY, and.41S files cannot be moved.

3.0 SeisOpt ReMi Vspect Module

Double-click on the desktop icon shown in Figure 2a to launch the ReMi Vspect module. The graphical user interface shown in Figure 7 will open up. The Vspect module imports the microtremor data recorded in the field and performs a wavefield transformation, creating a ''velocity spectrum'' in the slowness-frequency (p-f) domain.

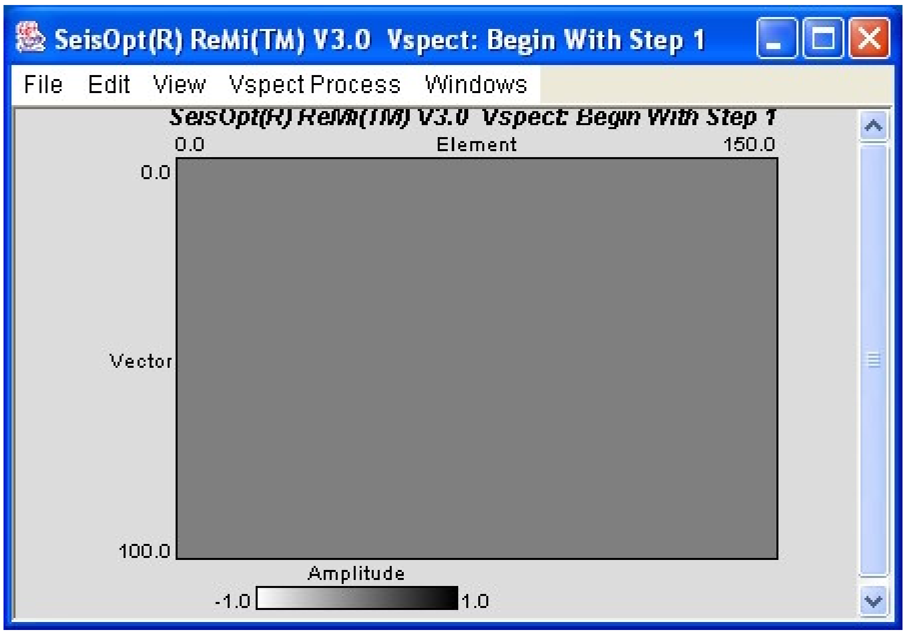


Figure 7: Starting graphical user interface of SeisOpt ReMi Vspect module

This module contains tools to do the following:

- Convert refraction microtremor data recorded in the field (SEG-2 format) to SEG-Y.
- Import, visualize and plot the data (Step 1).
- Perform some pre-processing on the data (Step 2).
- Erase or apply geometry to the microtremor records (Step 3).
- Perform wavefield transformation on the data in and generate a velocity spectrum in the slowness-frequency domain (Step 4). This yields the Rayleigh wave phase-velocity dispersion curve.
- Visualize and plot the velocity spectrum.
- Stack and sum velocity spectrum from individual records (gathers) to one spectrum (Step 5).
- Pick the dispersion curve and export the picks for interactive one-dimensional velocity modeling using SeisOpt ReMi Disper module (Step 6).
- In addition it has also options to write out images in JPEG, SEG-Y, or Binary formats.

Each of these steps launches a new window. For ease of use it is recommended that the user close any windows that are not needed, by selecting ‘‘Close Window’’ under the ‘‘File’’ menu. That is, for example, if the user is performing Step 4, he or she can close windows that were opened during Step 1, Step 2 and Step 3.

Note that the title bar on each window indicates which step of the SeisOpt ReMi Vspect processing the user is at, making it easier to keep track of the processing sequence. When proceeding to the next step, use the select the options from the current window. If you do have windows from all the processing steps open, click on the frame of the window you want to activate and bring to the front.

3.1 Step 1: Converting, importing and visualizing field data

The first step is to convert the microtremor data recorded in the field to SEG-Y format since current version of the ReMi Vspect module can only read in field data in SEG-Y format. If the seismic recorder can save data in SEG-Y format, then this step can be skipped.

For the convenience of the user, the module comes with a program that will convert data recorded in SEG-2 format to SEG-Y. If you wish to convert other data formats to SEG-Y (like from ABEM Mark 3 or OYO), use SeisOpt® Picker which is available from Optim. SEG-2 (usually, with .DAT extension) is the format of data recorded by several seismographs and is one of the most common formats used for recording seismic data. The conversion program can handle only data that has 16,000 samples per trace or less. Hence, set the recording length and sampling rate to make sure this limit is not exceeded. Contact support@optimsoftware.com if you have any questions before acquiring the data. There are several commercial programs available (for example, IXSeg2Segy © Interpex Limited, Golden, CO, USA) that can handle longer trace data.

3.1.1 Converting SEG-2 data to SEG-Y format

If your seismograph records in SEG-2 format, then the first step is to convert it to SEG-Y format. To do so, choose the ‘‘Convert SEG2 Files to SEGY…’’ item from the File menu of the Vspect module, as shown in Figure 8.

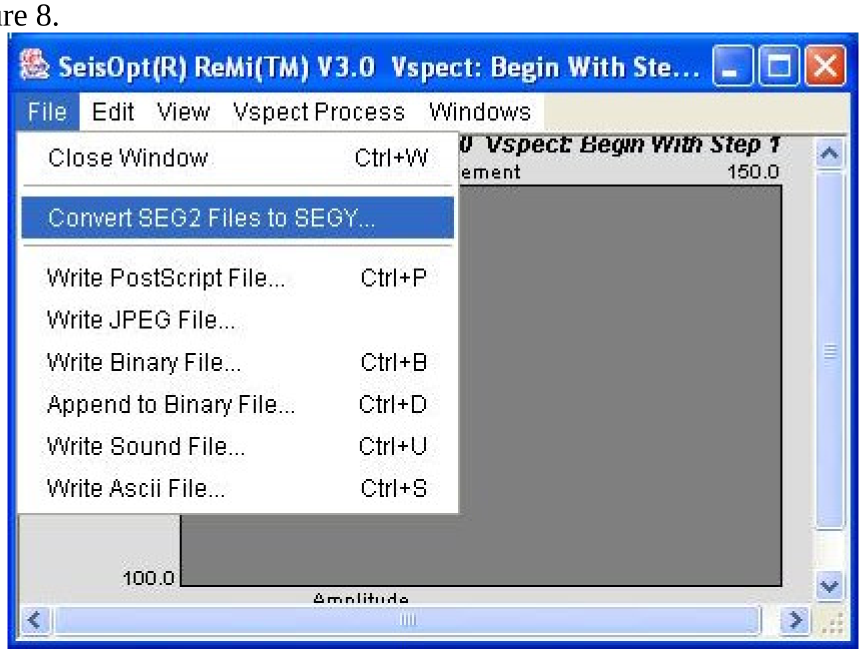


Figure 8: Choose ‘‘Convert SEG2 Files to SEGY…’’ to start the process of converting SEG-2 files

The dialog window shown in Figure 9 opens up, with the message ‘‘Select first SEG File…’’ in its title bar. Go to the directory containing the SEG-2 data files (.DAT extension) and choose all the records that need to be converted. Make sure all the files that are being converted are recorded along one profile. Note that for this example, the data files RM2011.DAT to RM2021.DAT residing in the subdirectory ‘‘demo’’ was chosen.

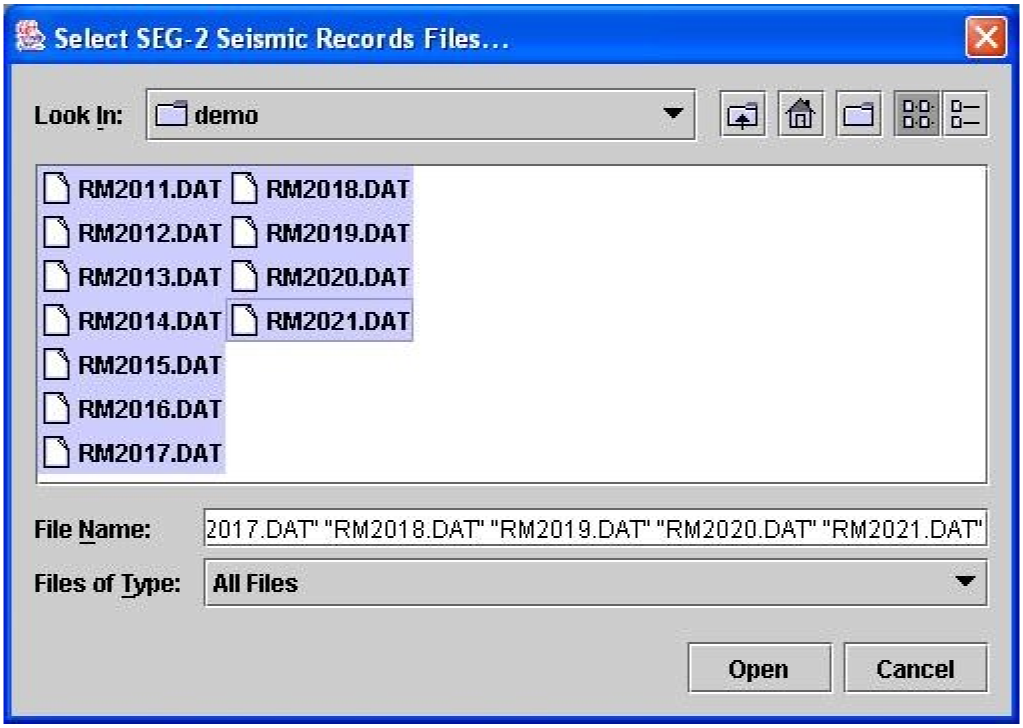


Figure 9: Choose all the SEG-2 files to be converted and click ‘‘Open’’

If the conversion is successful, a window displaying the message ‘‘Conversion completed’’ will open up (Figure 10).

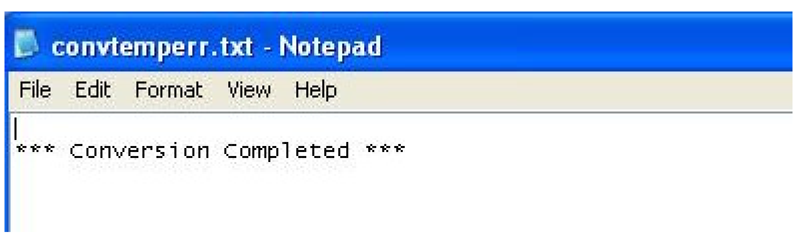


Figure 10: On successful conversion, the message above will be displayed on a Notepad/Wordpad window

Any errors that might occur during the conversion are also displayed in this window. A SEG-Y file (.SGY extension) containing all the files (records) and which has the name of the first SEG-2 file will be written to the directory containing the SEG-2 files. The conversion is almost instantaneous (few seconds) and so, if the window shown in Figure 10 does not appear, it indicates a problem in converting the files. If this is the case, try converting one file at a time. This will create one SEG-Y file for each SEG-2 file. The individual SEG-Y files can then be read in together in Step 1 of the Vspect processing sequence (Section 3.1.2).

The conversion is performed by the seg2segy routine, slightly modified to adapt to Vspect module, and is available from the Colorado School of Mines (Copyright, 1992-1999). The following rules and limitations apply to the conversion:

- The SEG-2 files should not exceed 16,000 samples per trace. For a typical SeisOpt ReMi data acquisition, it is sufficient to record for 30 seconds, at 2 ms sample interval, resulting in 15,000 samples per trace. One can record noise for 30 seconds at 2 ms sample interval and the data can be converted using the above module.

3.1.2 Importing and Visualizing SEG-Y Files

Once the SEG-2 files have been successfully converted to SEG-Y format, the next step is to import and visualize them using the ReMi Vspect module. You also use this step to read in individual SEG-Y files (records) that have been recorded in the field. To do so, go the ‘‘Vspect Process’’ menu item and choose ‘‘Step 1.b: Open SEG-Y Seismic Records…’’ as shown in Figure 11.

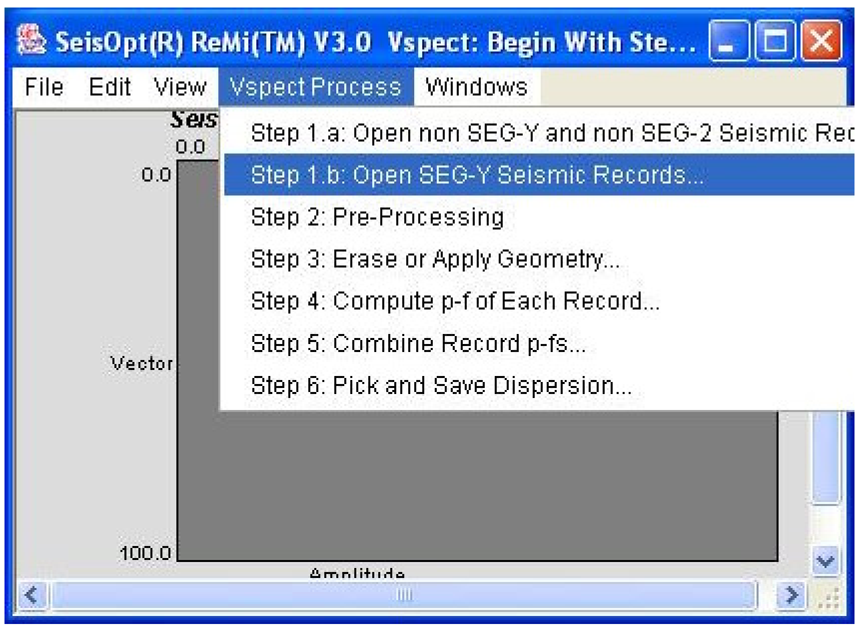


Figure 11: Choose ‘‘Step 1.b: Open SEG-Y Seismic Records…’’ to read in the microtremor data in SEG-Y format

Note that Step 1.a should be used only for data saved or recorded in a format that is **NOT** SEG-Y or SEG-2.

When the Step 1.b is selected, the dialog box shown in Figure 12a will open up. Go the directory that contains the data file in SEG-Y format, select it, and click ‘‘Open’’. If you are reading in individual SEG-Y files created in the field then you simply choose all the SEG-Y files to be read in as show in Figure 12b. Note that you use this option only when each SEG-Y file contains only ONE record. If they contain multiple records, see Section 3.1.5 on how to read them in.

In both cases, the format verification window shown in Figure 13 will open up. The default data type of the converted SEG-Y traces is 32-bit integer. Use this the option when the SEG-Y file is created using the SEG-2 conversion module (Section 3.1.1) or when using DAQLink II or the SeisOpt ReMi recording unit manufactured by Seismic Source Company. The program can also read in SEG-Y data whose traces could be in any of the formats listed in Figure 14. Usually, the seismic recording instrument manual should specify the trace type of the data being recorded. Next, make sure the number of traces to be read in is correct. The default value is 24, but if you have a 12-channel system this should be changed to 12. Similarly, when recording more channels this should be change to reflect the total number of traces being analyzed. Click ‘‘Read Binary’’ to read in the SEG-Y data.

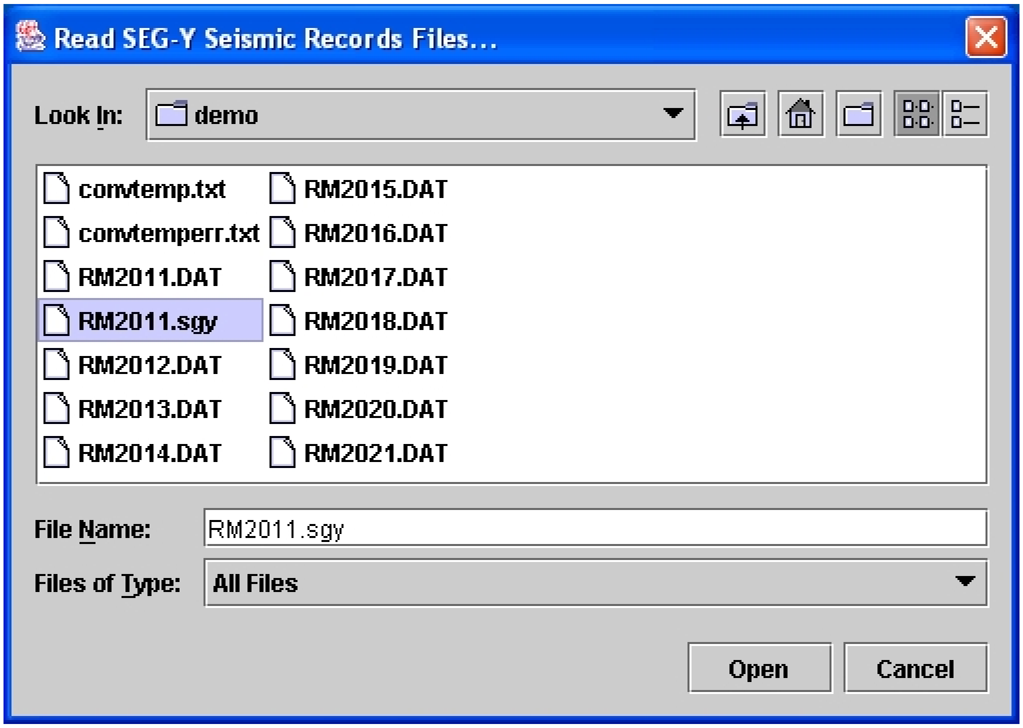


Figure 12a: Choose the SEG-Y file to be imported in ReMi Vspect for analysis

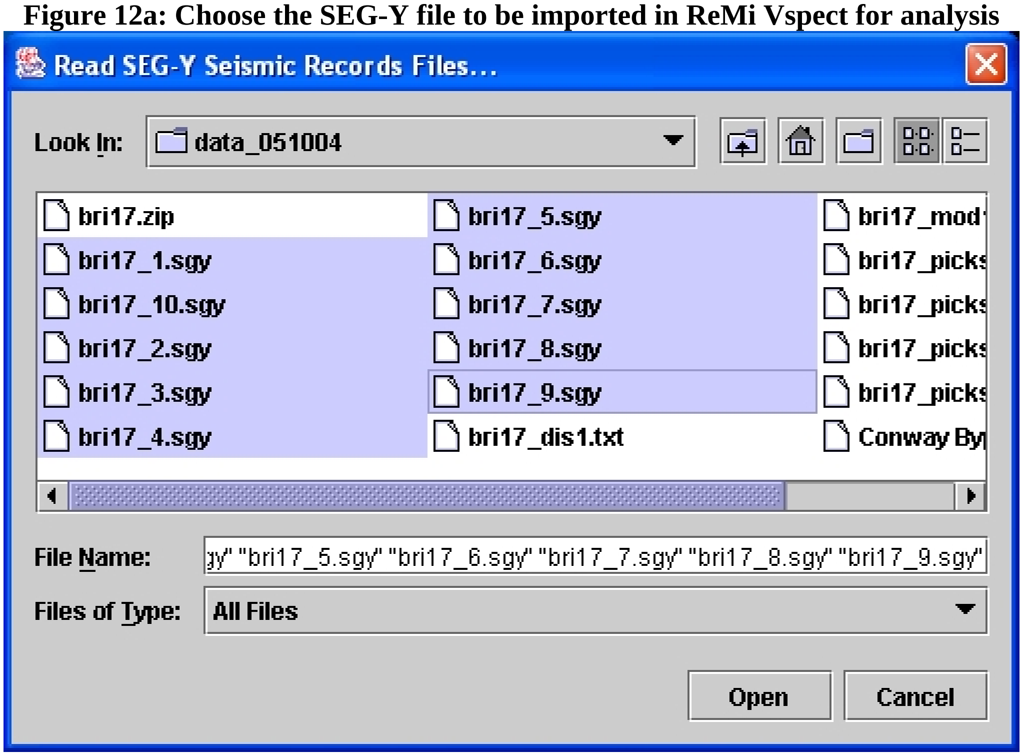


Figure 12a: When reading in several individual SEG-Y files, each containing only ONE record, select all of them and click ‘Open’.

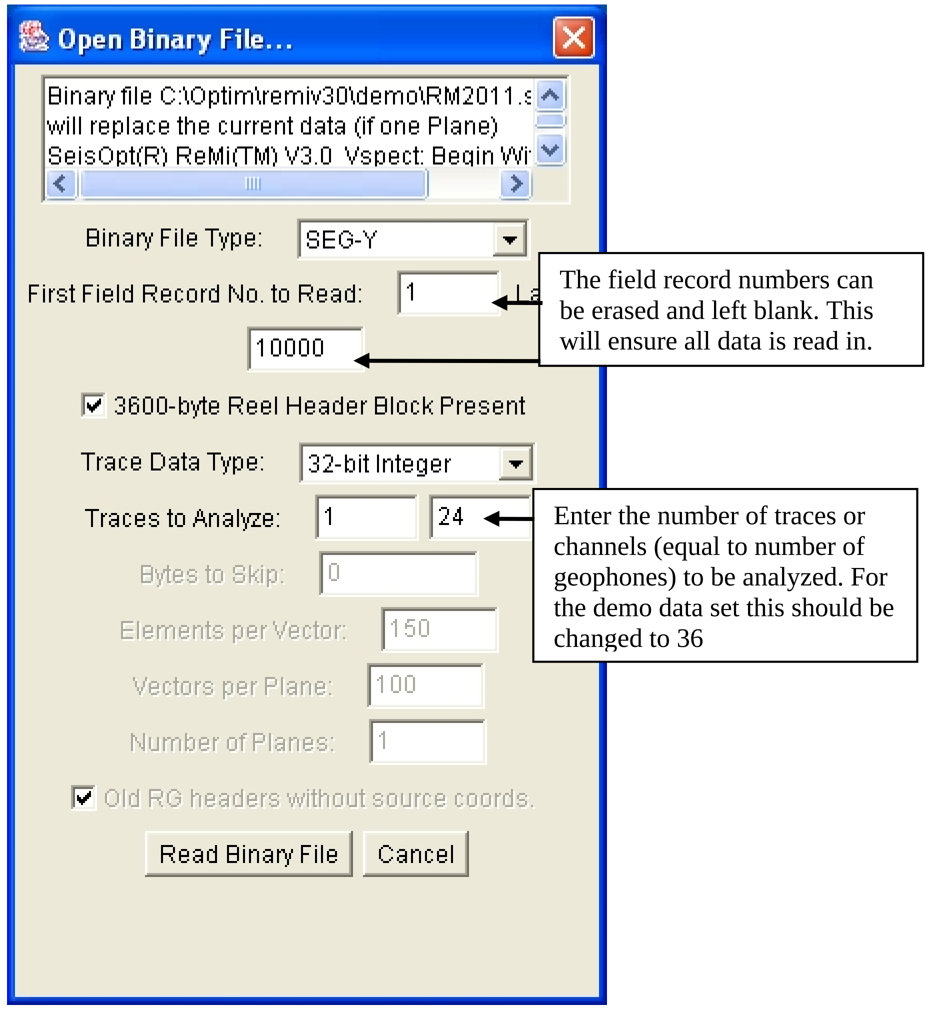


Figure 13: SEG-Y data verification window. The default trace data type (32-bit integer) may needed to be changed only if the SEG-Y data is created by a process other than the one described in Section 3.1.1

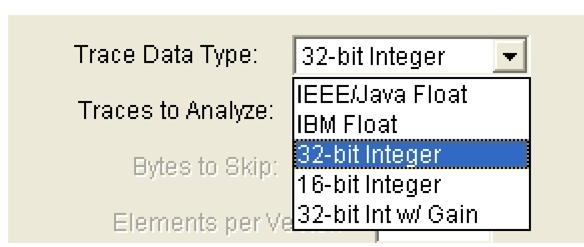


Figure 14: Different ''Trace Data Type'' can be chosen using the pull-down menu

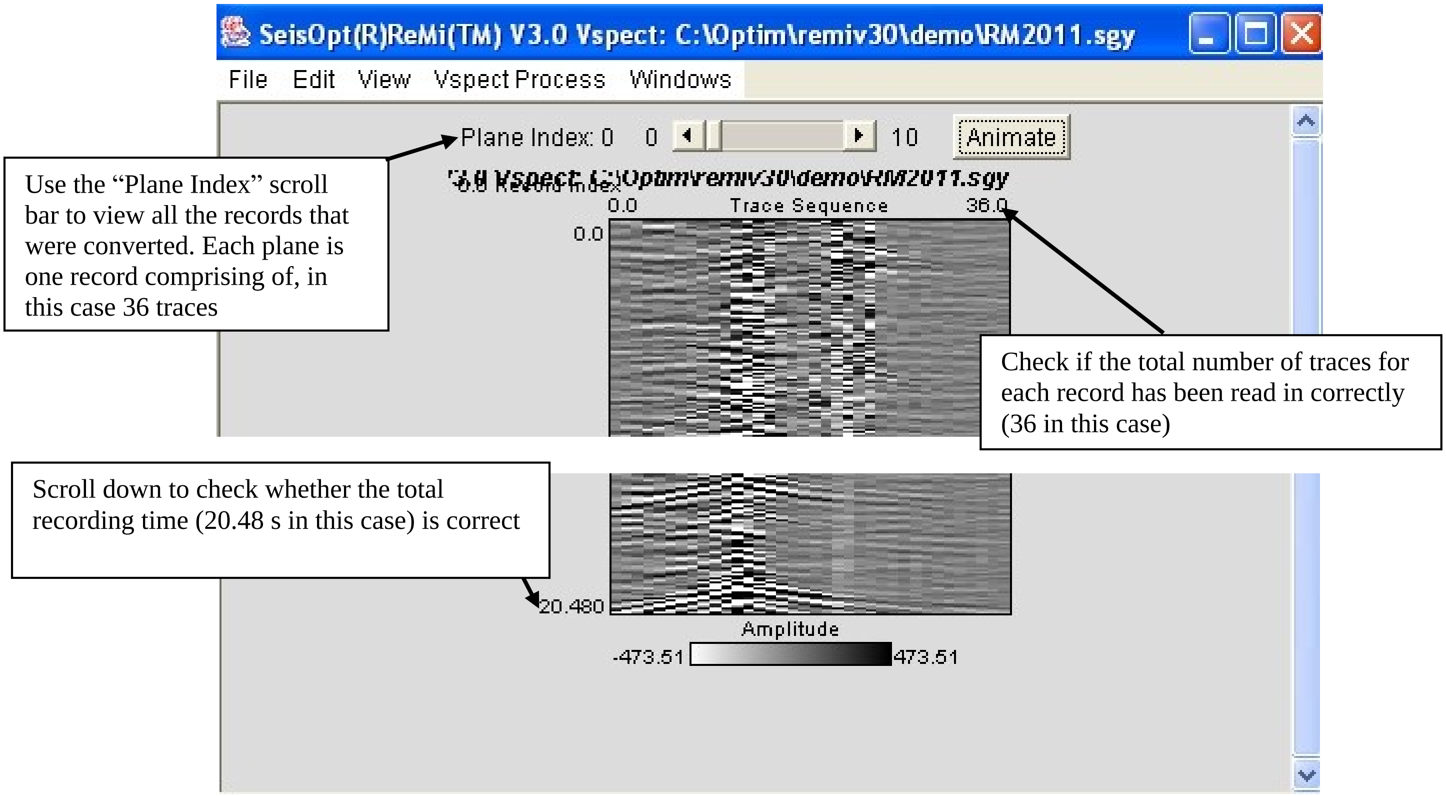


Figure 15: (a) SEG-Y data imported into ReMi Vspect module (Step 1) (b) Scroll down the bottom of the data to make sure that the total recording time and the amplitude of the traces are correct. An incorrect trace data type will result in very large amplitudes for the traces

For example, if the SEG-Y traces were created by a procedure other than the one described in Section 3.1.1 then it is recommended that the user change the trace data type to either IEEE or IBM float. One can detect if the trace data type is correct by looking at amplitude scale (Figure 15b) of the data. Incorrect trace data type will result in amplitudes being very large (e.g. 1e-8 to 1e+8).

The data will open up as shown in Figure 15. Do the following to check for integrity of the imported data:

1. Make sure to scroll down to the bottom of the window to check that the total recording time, displayed in seconds, is correct (for this example it is 20.48 seconds).
2. Check the amplitude scale and make sure they are reasonable. Incorrect trace data type will result in very large amplitudes.

Click on the ''Animate'' button or use the scroll bar to view all the files (records) that were converted. The SEG-Y file should contain all the SEG-2 files that were converted. The ''Plane Index'' counter will increment from ''0'' as each record is displayed. For example, for the demo data 11 planes (Plane Index 0 to 10) consisting of SEG-Y data from records RM2011 to RM2021 will be present.

Note that messages will appear on the MS-DOS window, while the data is being read in, including any error messages. Since SeisOpt ReMi Vspect user interface is Java based, it is possible that an ''Out of Memory'' error might appear when reading in large data sets, with multiple records. As a rule of thumb, the available RAM on the machine must be about four times the number of bytes in the SEG-Y file. See Section 6.0 on how to increase available RAM for Java.

The data size of the display window can be adjusted by choosing an appropriate zoom percentage under ‘‘View’’ menu (Figure 16).

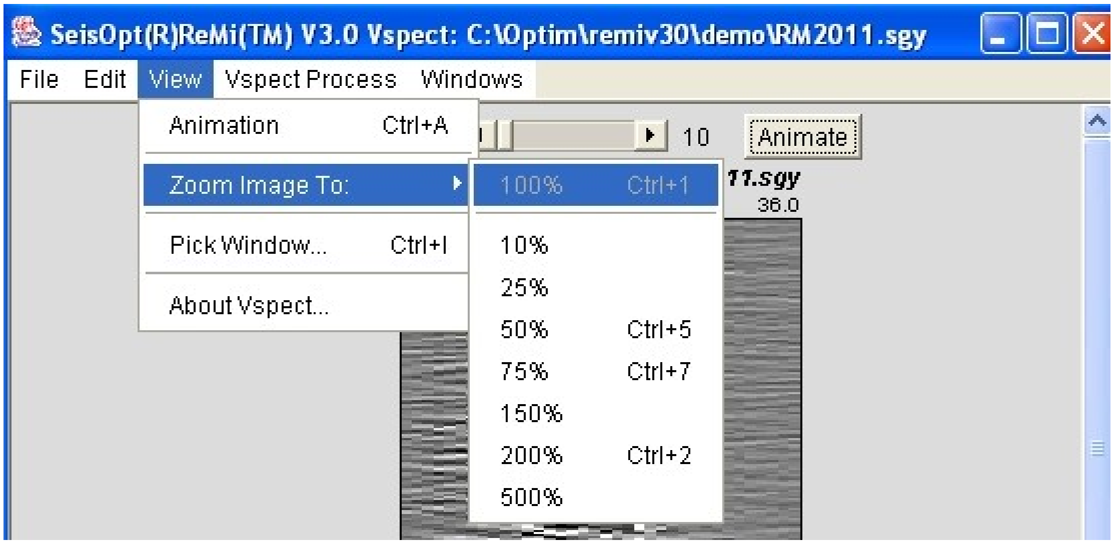


Figure 16: Adjust window size using the Zoom options under the ‘‘View’’ menu

3.1.3 Setting Plot Parameters of the Imported Data

The parameters that control the display of the data on the window and the output can be viewed and edited by choosing the ‘‘Plot Parameters…’’ option under the ‘‘Edit’’ menu (Figure 17). Choosing this will open up the plot property window shown in Figure 18. If the window panel does not look like the one show in Figure 18, or if it appears to be missing the lower items, just drag a corner with the mouse to enlarge it a little. Setting the plot parameters correctly for the record will allow you to analyze it more quickly, as some of the geometry information will be correctly set as well. The following describes the plot parameters:

- Rows Point Down: If checked, renders the image with the time axis going downwards.
- Vert. Exagg. (Recommended between 0.01 and 0.1): controls the size of the plot.
- Positive Only: If checked, renders only positive amplitudes.
- Amplitude Clip: Values above this are clipped. Best to set it at ``3*rms"
- Units: Type in label for scale of image.
- Color Table for Image: Choose different color palettes for the image.
- Element Zero: Starting time for traces
- Element Delta: Sampling interval of the traces in seconds
- Element Units: Label for the time axis.
- Vector Zero: Starting value for horizontal axis (number of traces or distance).
- Vector Delta: Geophone spacing (in meters) or trace spacing.
- Vector Units: Label for the x (horizontal) axis (e.g.: ‘‘Offset, meters'‘’).
- Planes Zero: Index for start of planes, if multiple gathers (records) are being displayed.
- Planes Delta: Increment for labeling subsequent planes.

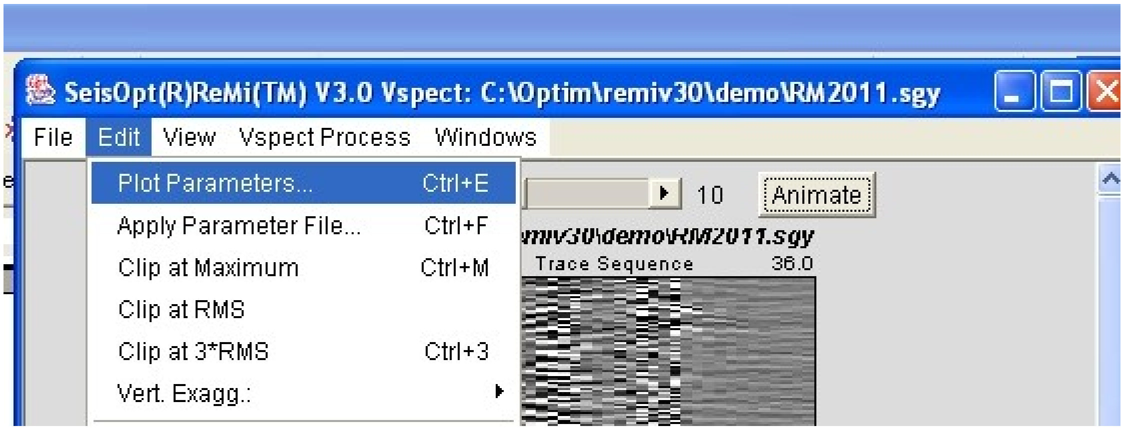


Figure 17: View and edit plot parameters for the display of data by choosing ''Plot Parameters…'' under the ''Edit'' menu

Click the ''Apply Changes'' button for the changes to be applied. Click ''Reset Values'' to delete the changes and ''Cancel'' to close the window.

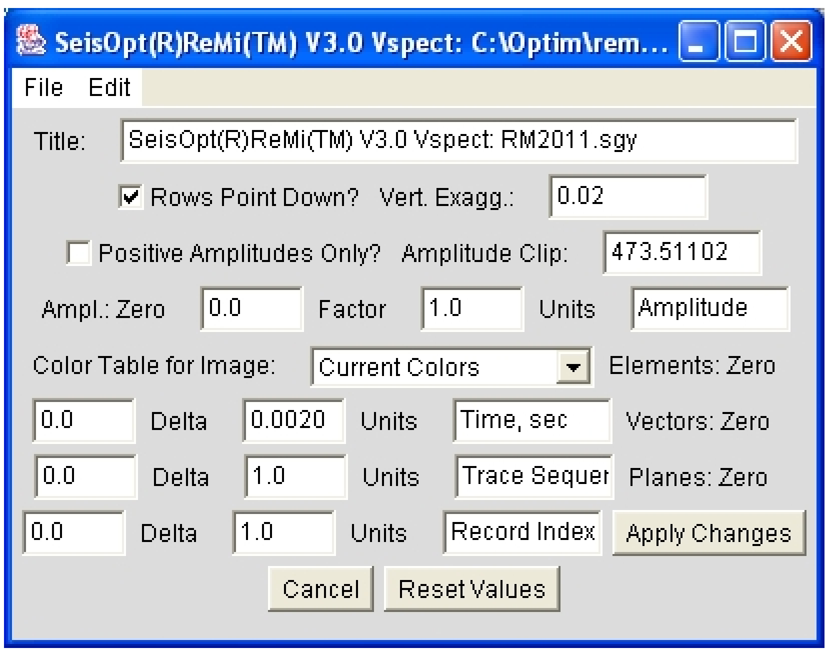


Figure 18: Edit and save plot parameters

The plot parameters can be saved to file by choosing the ''Save Parameter File…'' option under the ''File'' menu of the properties window. A new dialog appears asking you to find a directory and suggesting a name for the parameter file. A saved parameter file can be read in using the ''Open Parameter File…'' option.

3.1.4 Exporting Data Files for Printing

To save or print a plot of the record, display the appropriate record the, and then select ''File->Write PostScript File...'' from the menu bar (Figure 19). Note that only the displayed file will be output as a PostScript file. If multiple files are present, the correct file must be first displayed by choosing using the ''Planes Index'' scroll bar at the top. PostScript files can be either displayed, edited or printed using drawing programs like Adobe Illustrator. To view and/or print these files a program capable of reading EPS format files is needed. If you do not have such a program, you can download a set of free ones, called GhostScript (Copyright © Aladdin Enterprises) and GSView (Copyright © 2000 Ghostgum Software Pty Ltd.), from www.cs.wisc.edu/~ghost/. GSView is the visual interface to GhostScript. See Section 3.4.5 on how to import the PostScript file into an MS-WORD document. Version 2.0 allows to you output the file in JPEG format. Select "File->Write JPEG File…" from the menu (Figure 19) to output the rendered image in JPEG.

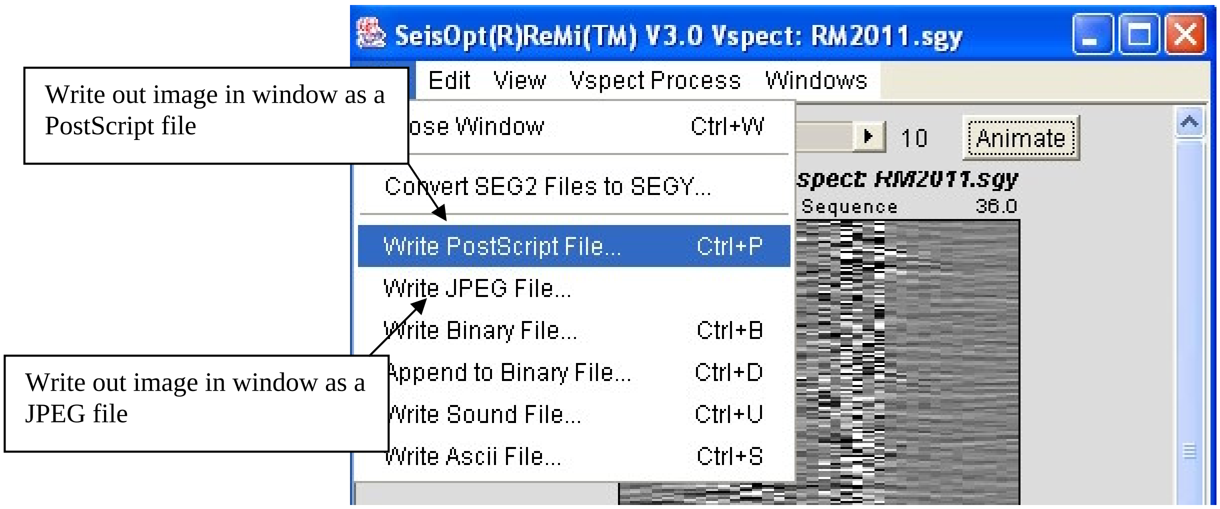


Figure 19: "Write PostScript File…" option outputs a PostScript file of the data displayed in the user interface while "Write JPEG File…" writes out a jpeg image.

3.1.5 Importing/Exporting files in another format and combining SEG-Y files

Use the "Step 1.a. Open non SEG-Y and non SEG-2 Seismic Records…" option under 'Vspect Process' if you are importing data that is not in SEG-Y or SEG-2 format. When doing so, make sure you choose the correct binary file type option.

There could also be a situation where you need to combine SEG-Y files created by converting group of SEG-2 files (using the conversion module), into one SEG-Y file. That is you need to combine several SEG-Y files, each containing multiple records. This Section describes how to do this.

Read in each SEGY file using the "Step 1.b.: Open SEG-Y seismic records…" option under 'Vspect Process' option. From the window that displays the SEG-Y record, choose the "Write Binary File…." option under the File menu as shown in Figure 19a.

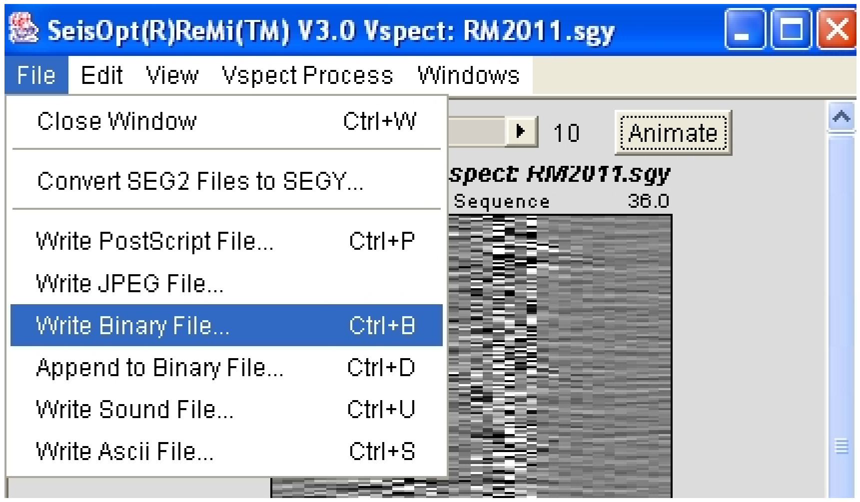


Figure 19a: Choose the 'Write Binary File…' option to export the SEG-Y file as a binary file.

The dialog window shown in Figure 19b opens up. Enter a file name with any extension you choose.

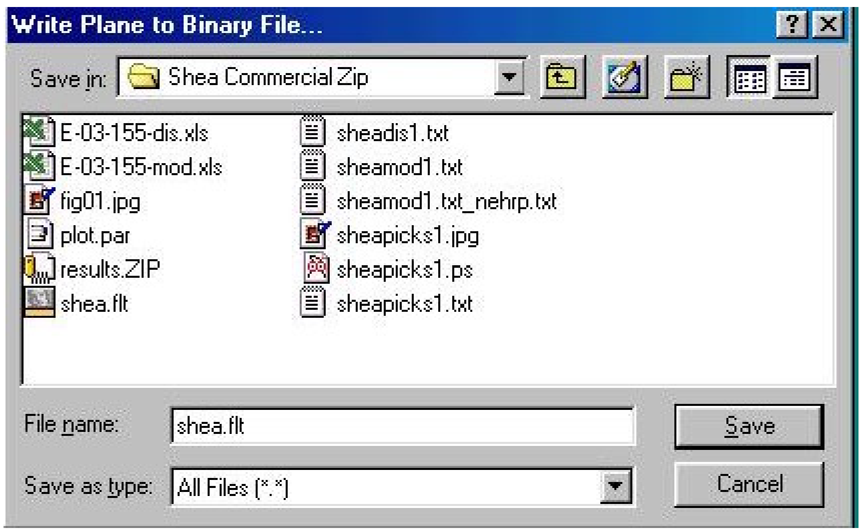


Figure 19b: Provide a filename, with an extension of your choice, for the binary file to be written to.

Press the "Save" button. The window shown in Figure 19c will open up. Choose the "Raw Float" option for the binary file type. The default option is "RG", so you need to scroll up to choose the "Raw Float" option. Click on "Create Binary File".

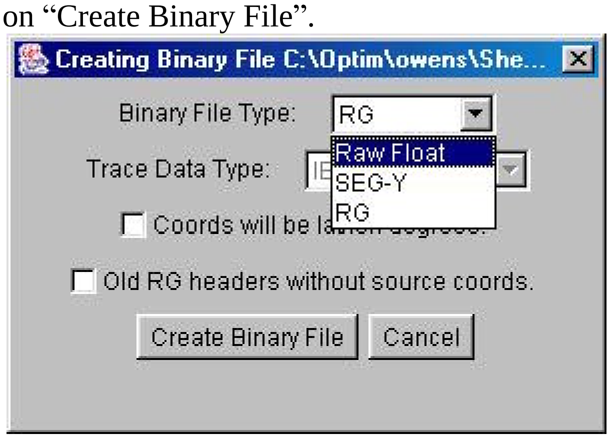


Figure 19c: Select the "Raw Float" option and click on "Create Binary File" .

Now open up the next SEG-Y file to be combined using the "Open seismic records…" option under Vspect Process menu. From the window that displays the SEGY traces, choose the "Append to Binary File…" option as shown in Figure 19d.

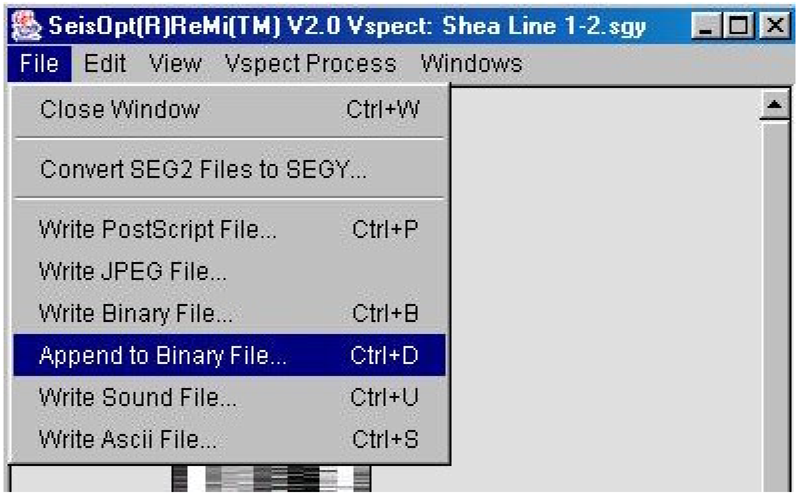


Figure 19d: After reading in the next file to be combined, choose 'Append to Binary File…' option.

A dialog window will open up. Choose the ***same file name*** that was used to create the binary file for the 1st record. The 2nd record will be appended to the 1st one. As before, choose "Raw Float" for the binary file type (Figure 19e) and click on "Append Binary File" to append the second record to the first one.

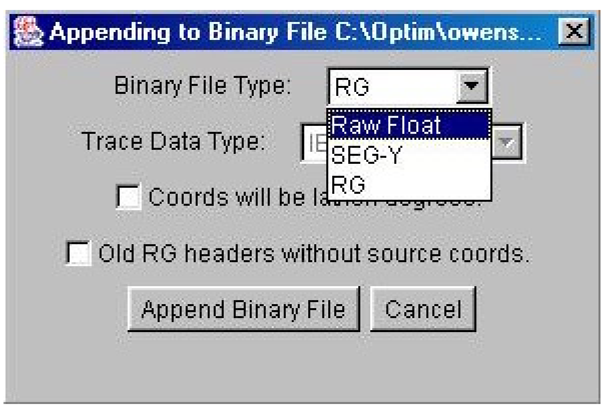


Figure 19e: Choose the "Raw Float" option and click 'Append Binary File'.

Repeat this process for the remaining SEGY files, each time appending the binary file to the existing binary file. Make sure you choose the "Raw Float" option for the binary file type each time.

While displaying any one of these SEGY files, make sure you save the plot parameters. This will allow you to display the appended file properly. To do so, choose "Plot Parameters…" from the Edit menu in the window that displays any one of the SEGY file as shown in Figure 19f.

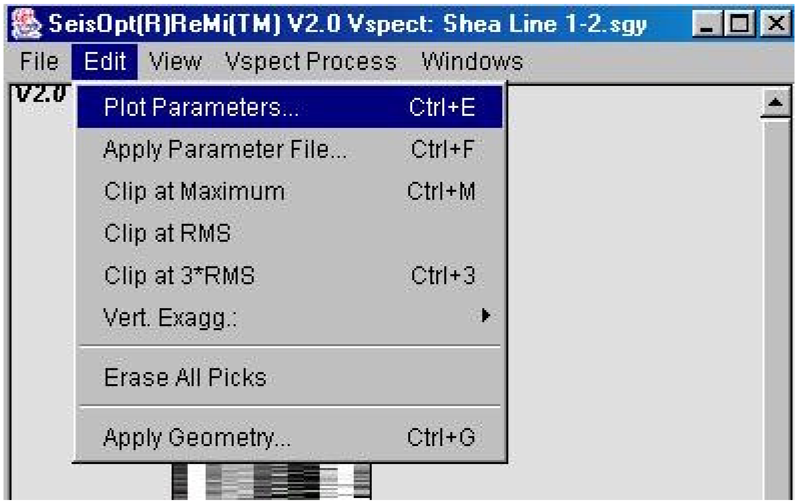


Figure 19f: Select 'Plot Parameters…' option from the 'Edit' menu

Now click on "Save Parameter File…" under the File menu in the parameter window that opens up (Figure 19g) and save the parameter file (Figure 19h).

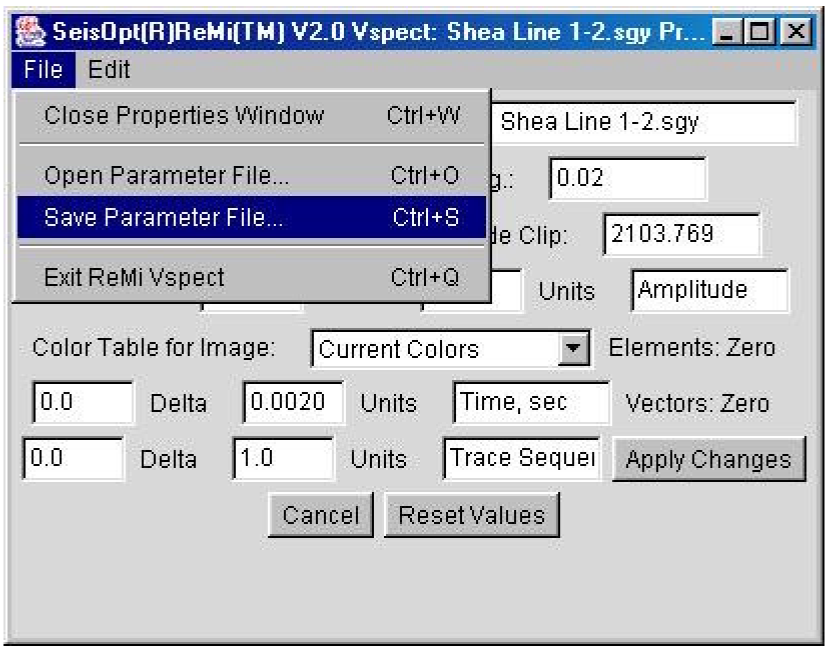


Figure 19g: Select the 'Save Parameter File…' to save the plot parameters to a file.

Once you have appended all the files and created a new binary file, you can read it back in. To do so, use "Step 1: Open Seismic Records…" option under Vspect Process. Select the combined binary file. Set the "Binary File Type" to "Raw Float". You also need to set the following as shown in Figure 19i:

Elements per vector = Total recording time / sample interval. For example, if the total recording time is 30 seconds and sampling interval is 2ms, then elements per vector = 15000

Vectors per plane = Number of channels per record (all traces recorded NOT the ones analyzed).

Number of planes = Number of records that were combined.

Once the data is read in displayed, you can adjust the plot parameters by reading in the parameter file saved in Figure 19h. To do this select the "Apply Parameter File…" option under the Edit menu from the window that displays the records (Figure 19j) and read in the saved parameter file (plot.par in this case).

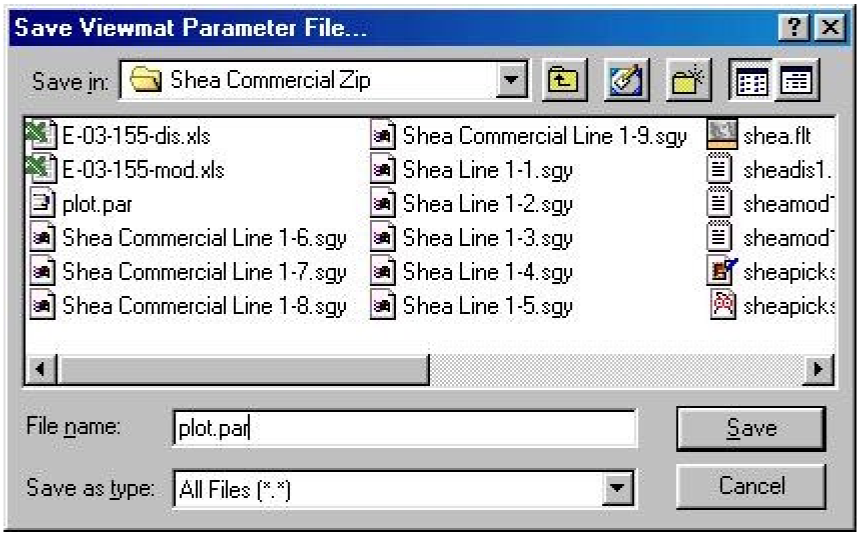


Figure 19h: Save the plot parameters to a file.

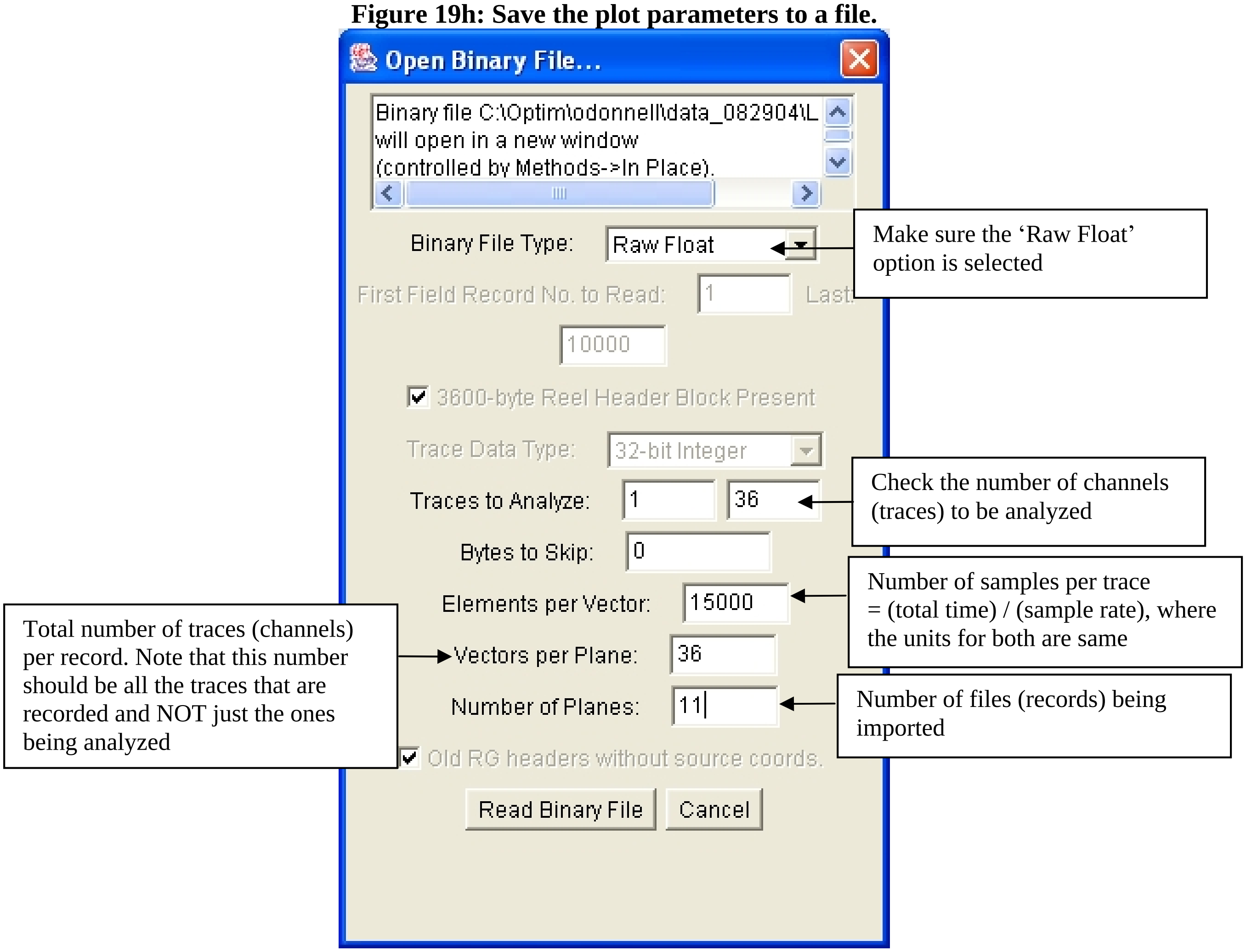


Figure 19i: Read in the combined SEG-Y file by selecting the "Raw Float" option for the binary file type and entering the correct values for elements per vector, vectors per plane, and number of planes.

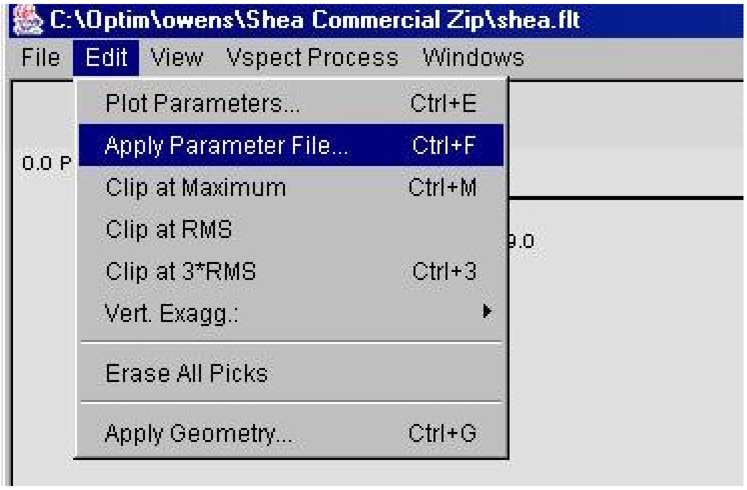


Figure 19j: Render the data properly, by applying the parameter file saved in an earlier step.

Now all the records should be displayed properly as separate planes and the Vspect processing can be continued.

3.2 Step 2: Pre-processing the Data

The next step in the SeisOpt ReMi Vspect analysis is pre-processing (Figure 20) the imported SEG-Y data. This step is not necessary but is useful to do it. It does a trace-equalization gain and centering of all the traces in the records.

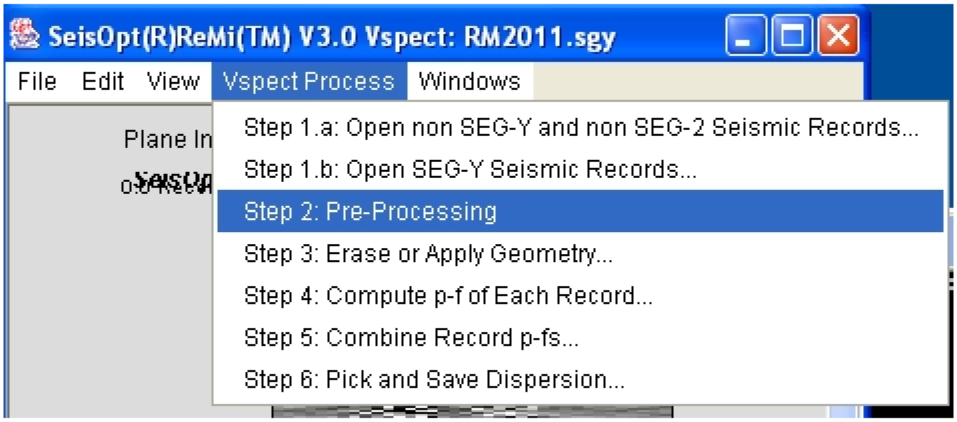


Figure 20: Pre-processing of imported microtremor data

Trace-equalization gain includes applying an automatic gain control whose length equals the number of time samples per traces, on all the traces. The centering function averages out any deviations or spikes in the data. Most records will provide a more coherent velocity spectrum if a trace-equalization gain is applied first. This prevents high-amplitude near-offset traces from dominating the spectra. Once the pre-processing is completed, the pre-processed data (Figure 21) will be displayed in the same window as the original data. The title bar on this window will indicate that the process has been completed by displaying ‘ SeisOpt® ReMi™ V3.0 Vspect: SEG-Y filename + Step 2’’.

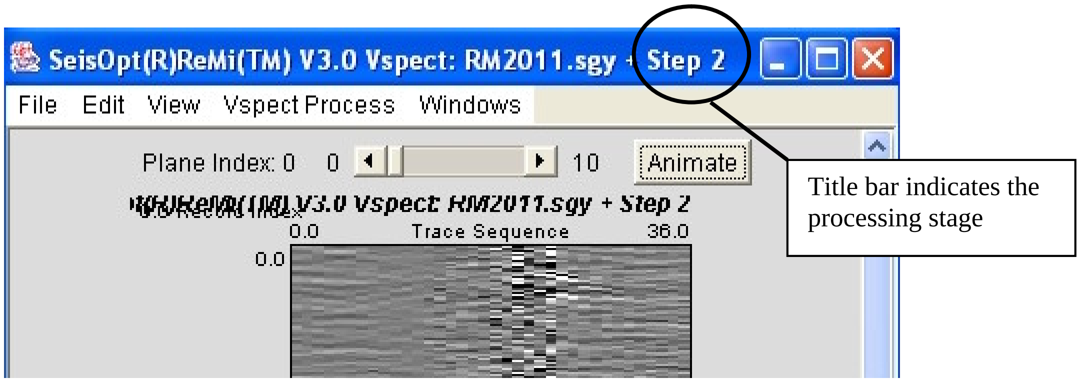


Figure 21: Data shown in Figure 15 after Step 2, pre-processing

3.2.1 Writing out Pre-Processed Data to disk

If you wish to save the pre-processed data, and save it to disk, use the ‘‘Write Binary File…’’ under the ‘‘File’’ menu (Figure 22).

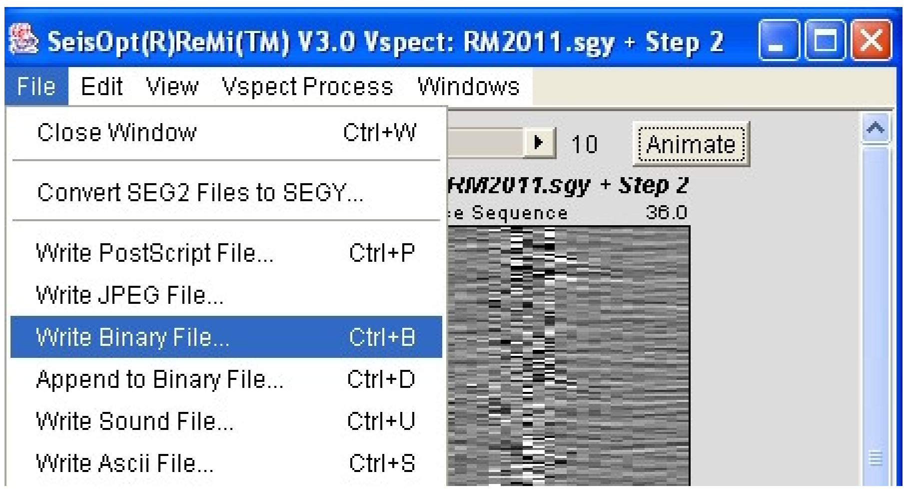


Figure 22: Use the ‘‘Write Binary File…’’ option under the ‘‘File’’ menu to write out the pre-processed data in SEG-Y format

Provide the desired file name, making sure not to overwrite any existing files in the directory, in the dialog window with a ‘‘.SGY’’ extension, since the output file will have be in the SEG-Y format. Choose the ‘‘SEG-Y’’ option for the binary file type and 32-bit integer for the trace data type (Figure 23). Click on ‘‘Create Binary File’’ button to save the file to disk.

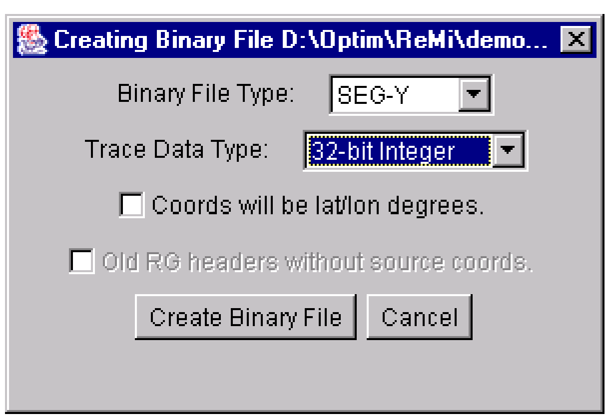


Figure 23: Select SEG-Y for the file type and 32-bit integer for the data type and click ‘‘Create Binary File’’ to write out the file

3.3 Step 3: Erase or Apply Geometry

The next step is to enter the survey or erase the geometry present in the trace headers of the microtremor data. Choose ‘‘Step 3: Erase or Apply Geometry…’’ item from the Vspect Process menu (Figure 24) on the window that has the displayed data.

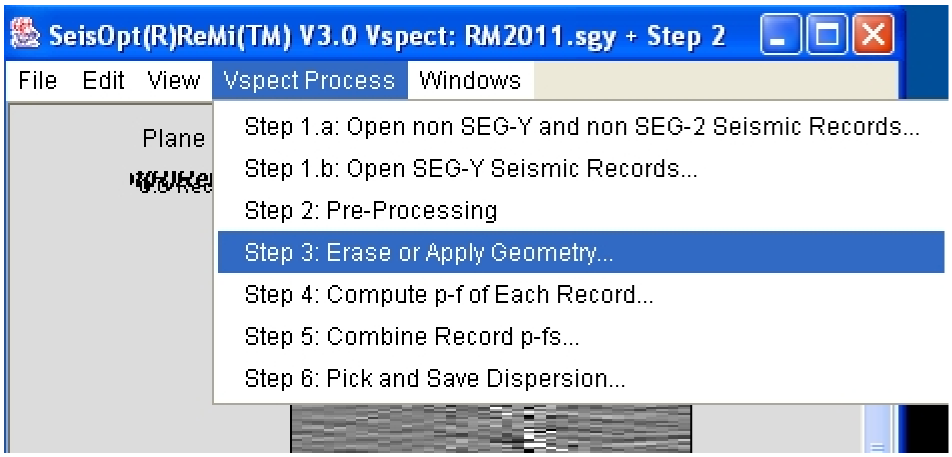


Figure 24: Choose ‘‘Step 3’’ to erase or apply geometry to the microtremor data

The window shown in Figure 25 will open. Make sure all the buttons appear as shown. If they do not, use the mouse, drag the lower right hand corner of the window and make it longer so all the buttons are visible.

3.3.1 Erasing Geometry from the Trace Headers: Recommended for typical ReMi survey

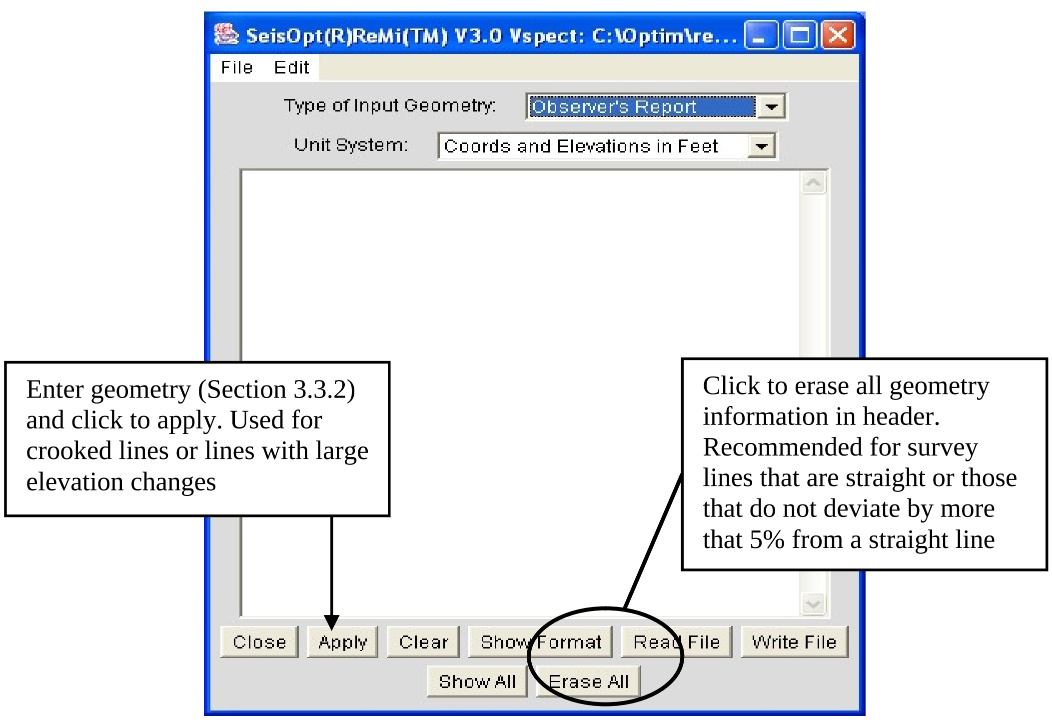


Figure 25: SeisOpt ReMi Vspect geometry tool. It is recommended that for a typical refraction microtremor survey, all geometry information contained in headers be erased by pressing the ‘‘Erase All’’ button

For a typical ReMi survey, with approximately evenly spaced geophones and relatively straight line with gradual elevation changes, it is recommended that the geometry information contained in the trace headers be erased. This is done by clicking on the "Erase All" button in the window shown in Figure 25. The dialog window shown in Figure 26 will open up. Click ''OK'' for the geometry information to be erased. This procedure is **recommended** when the survey line does not deviate by more than 5% of the length of the line (for example, 10m for a 200m long line) or if the elevation change is gradual and not more than 5% of the total length of the line.

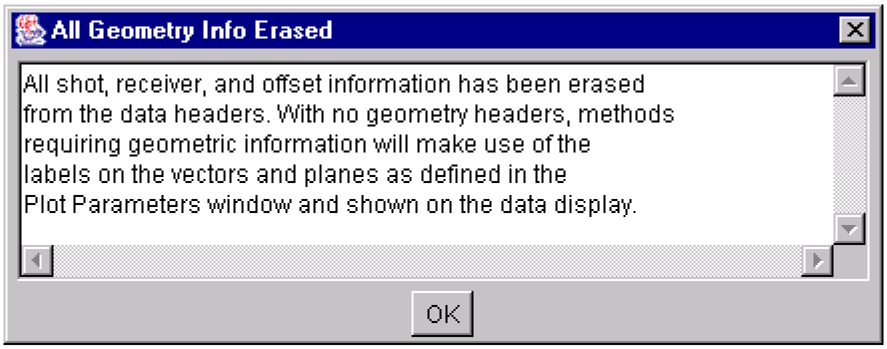


Figure 26: The message window above appears. Click ''OK'' to erase all geometry information

3.3.2 Applying Geometry to Trace Headers for Crooked line ReMi survey

If the survey line is very crooked and there are rapid elevation changes then the correct geometry must be applied. To do so, the user needs to create the observer and surveyor's reports in plain text using Wordpad and **copy and paste** the text into the text area of the Geometry tool (Figure 25).

First, create an observer's report according to the format given in the tool. You will need one line of observer's report text per record, in the order the records were read from the SEG-Y file. For example, if the data has 11 records (noise gathers) each having 36 channels and the stations are numbered 1 to 36 then the observer's report will have the following lines:

```
1     1     1     1     36
2     1     1     1     36
3     1     1     1     36
4     1     1     1     36
5     1     1     1     36
6     1     1     1     36
7     1     1     1     36
8     1     1     1     36
9     1     1     1     36
10    1     1     1     36
11    1     1     1     36
```

Copy it to the clipboard from the text editor (like Wordpad) and paste it to the Geometry tool's text area. Make sure the ''Type of Input Geometry'' is set to Observer's Report (Figure 27) and click the ''Apply'' button below.

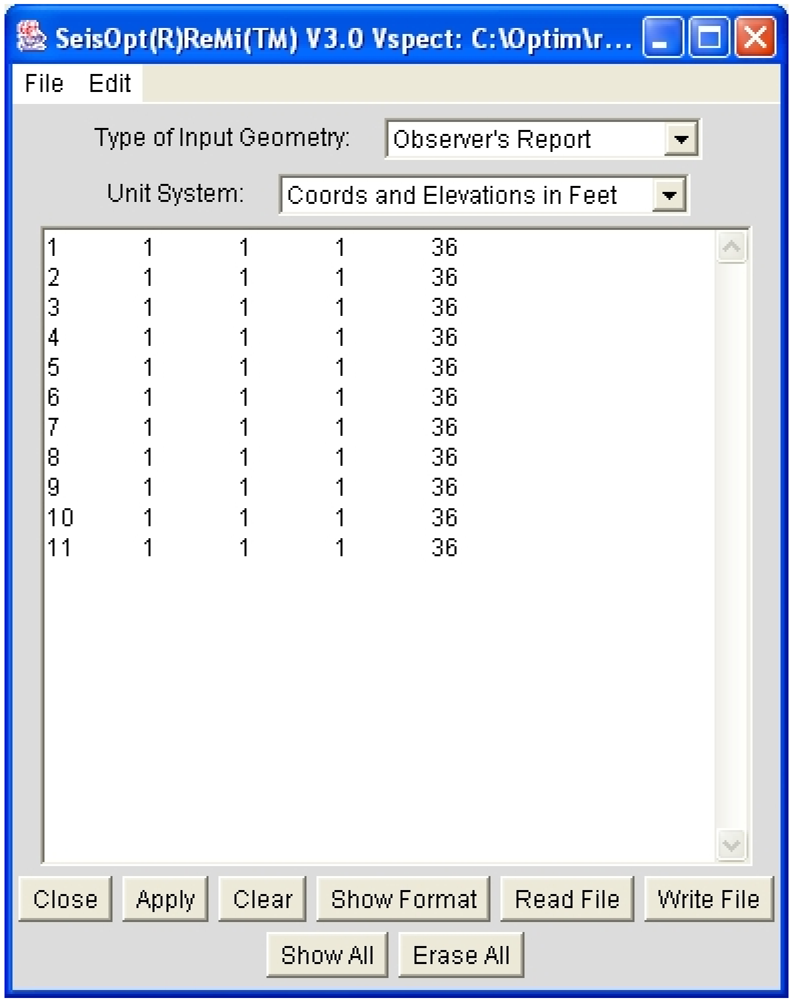


Figure 27: Enter the information for each record contained in the file and click 'Apply' to apply the observer's report.

Next, create a surveyor's report or station coordinates list according to the format given in the tool. You will need one line of station coordinate text per record, in any order. Note that there are two orders for the coordinate text accepted (N-X-Y-Z or N-Z-Y-X). The station numbers in the observer's report and surveyor's report must match. For example, if you choose the (N-X-Y-Z) option then the surveyor's information will look like the one below for a 36-channel survey and with geophone spacing 10 feet and elevation along the profile is flat.

```
1 0.0 0.0 0.0
2 10.0 0.0 0.0
3 20.0 0.0 0.0
4 30.0 0.0 0.0
5 40.0 0.0 0.0
6 50.0 0.0 0.0
7 60.0 0.0 0.0
8 70.0 0.0 0.0
9 80.0 0.0 0.0
10 90.0 0.0 0.0
11 100.0 0.0 0.0
12 110.0 0.0 0.0
13 120.0 0.0 0.0
14 130.0 0.0 0.0
15 140.0 0.0 0.0
```

```
16 150.0 0.0 0.0
17 160.0 0.0 0.0
18 170.0 0.0 0.0
19 180.0 0.0 0.0
20 190.0 0.0 0.0
21 200.0 0.0 0.0
22 210.0 0.0 0.0
23 220.0 0.0 0.0
24 230.0 0.0 0.0
25 240.0 0.0 0.0
26 250.0 0.0 0.0
27 260.0 0.0 0.0
28 270.0 0.0 0.0
29 280.0 0.0 0.0
30 290.0 0.0 0.0
31 300.0 0.0 0.0
32 310.0 0.0 0.0
33 320.0 0.0 0.0
34 330.0 0.0 0.0
35 340.0 0.0 0.0
36 350.0 0.0 0.0
```

Station ID (N) X Y Z

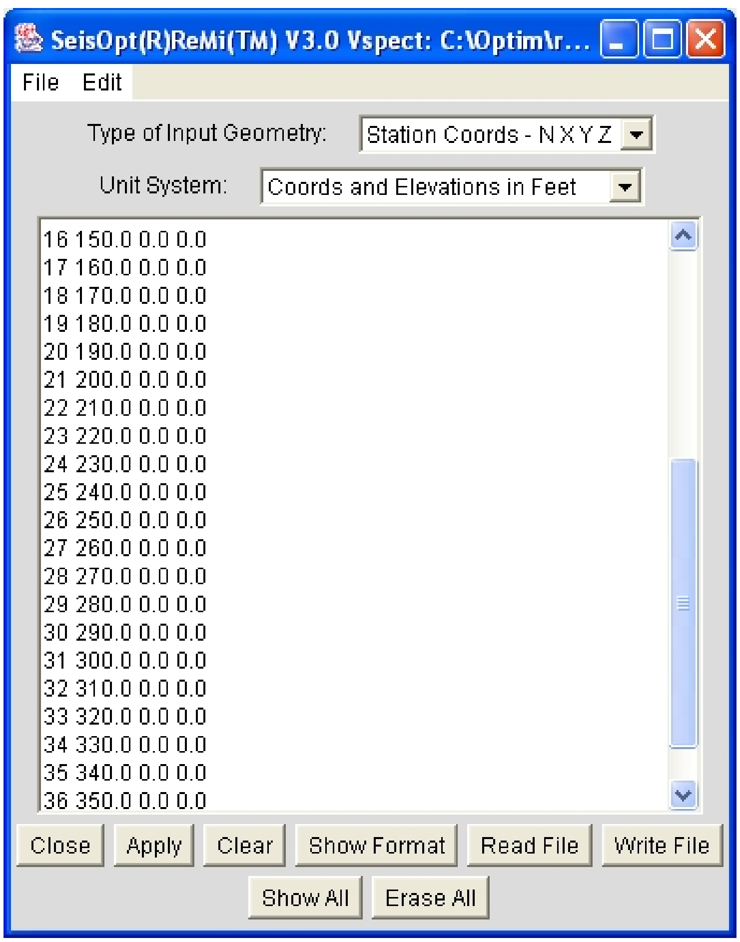


Figure 28: Enter the coordinates for each geophone location and click 'Apply' to apply the surveyor's report.

Paste it to the text area of the Geometry tool. Make sure the ''Type of Input Geometry'' is set to the appropriate ''Station Coordinates'' option, that the ''Unit System'' is set according to your input units, and click the ''Apply'' button at the bottom of the Geometry tool (Figure 28). Close the geometry tool by clicking the ''Close'' button.

To get more detailed information on what the observer's and surveyor's reports are, select the appropriate entry from ''Type of Input Geometry'' above the text field using the pull-down menu (Figure 29), and click the ''Show Format'' button at the bottom of the window.

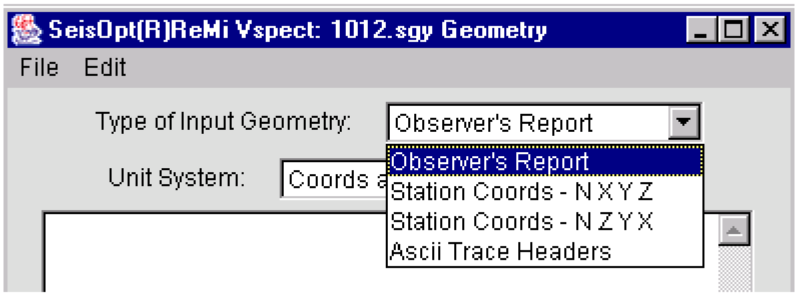


Figure 29: Choose the type of input geometry and click ''Show Format'' to see the format of the input files needed to apply the geometry

3.4 Step 4: Computing the velocity spectrum for each record: p-f transformation of the data

Once the geometry has been erased (or entered for a very crooked line), the next step is to compute the velocity spectrum. This process involves computing a surface-wave, phase-velocity dispersion spectral-ratio image by p-tau and Fourier transform across all vectors. This is described in detail in Louie (2001, see Section 1.0). The resulting image will be in the slowness-frequency (p-f) domain.

To start the process, choose ''Step 4: Compute p-f of Each Record…'' from the ''Vspect Process menu (Figure 30) on the window on which the geometry process (Step 3) was done. To make sure, verify that the title bar on the window displays ''SeisOpt® ReMi V3.0 Vspect: SEG-Y filename+ Step 2, 3'' (Figure 30).

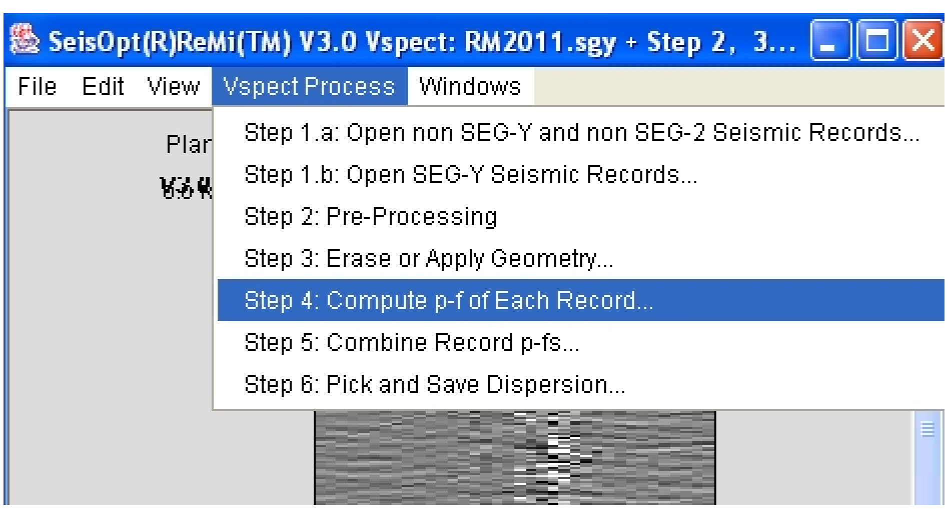


Figure 30: Choose ''Step 4: Compute p-f of Each Record…'' to start the velocity spectrum calculation of each record. Make sure you perform this action on the window which displays the data and has had 'Step 3' applied to it.

This will perform the p-f transformation on each plane or record of the data sets. For example, for this demo data (RM2011.sgy), each of the 11records (RM2011 to RM2021) will be transformed individually.

3.4.1 Setting Parameters and Starting ReMi Vspect Processing

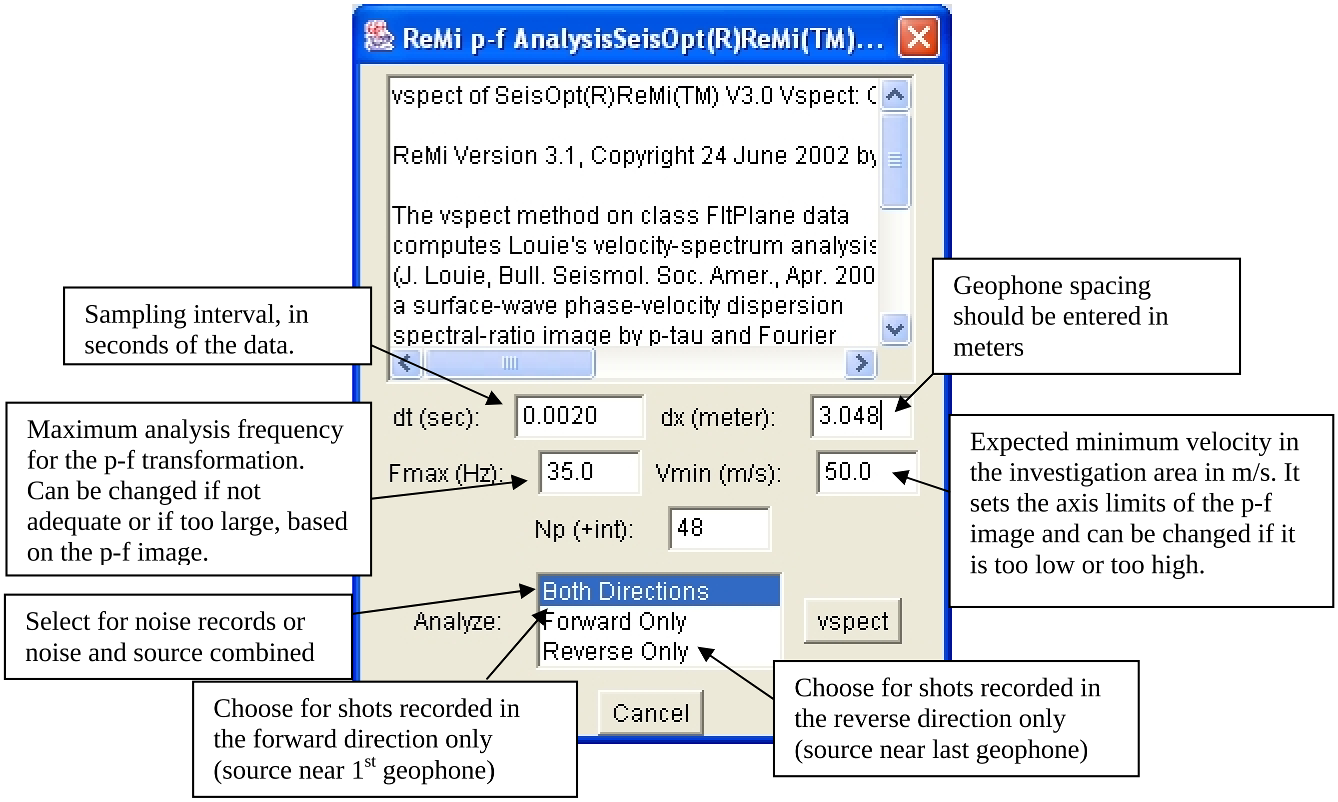


Figure 31: Set the parameters and click the ''vspect'' to start computing the p-f image for each record

On choosing this step, the dialog window shown in Figure 31 will open up. Note that if the geometry has been entered correctly (see Section 3.3.2) then the window shown in Figure 31 will not have the 'dx' entry; it will be automatically read in from the trace header information. Hence, it is important to enter the geometry correctly. If you are unsure about the geometry, it is recommended that it be erased (see Section 3.3.1) from the trace headers and enter and average geophone spacing before performing the 'Vspect' processing.

The settings in the dialog window are as follows:

- dt is the time sampling interval in seconds. For this example, it was 2ms (0.002 s)
- dx is the geophone spacing in meters. For this example, the spacing was 10 feet or 3.048 meters.
- Fmax is the maximum frequency you are interested in analyzing, in hertz. The default 25 Hz is the maximum used in analyzing data from 8-Hz geophones, and gives some resolution of velocities in the upper few meters. If you had used higher frequency phones and were interested in velocities in the uppermost meter, you might use a higher value for Fmax. Conversely, if you had data from lower-frequency geophones spread over larger

offsets you might make Fmax lower. Note that these values can be changed and Step 4 rerun, if the energy is observed at higher frequencies.

- Vmin is the minimum apparent or phase velocity your record might show. 200 m/s is default. It is useful to change this to 100 m/s or, as in this example, to 50 m/s. Once again, this value can be changed after the initial run.
- Np is the number of different slowness to include in the analysis. This defaults to 48 and usually a higher number results in increase of slowness resolution of higher velocities. However, slowness resolution is a complicated function of frequency, the number of traces, their spacing, and the total offset range. Hence, in most cases, the default value should be sufficient.
- Analyze:
  - ⇒ ***All Directions***: Use this option when computing the p-f transformation of **noise only or noise and source records** . This assumes surface waves are coming from all directions with relation to the ReMi acquisition array. Note that if you are analyzing records that might contain both noise and source induced energy, then this option should be used.
  - ⇒ ***Forward Only***: Use this option when computing the dispersion curve for records that were collected using a source that is located near the first geophone
  - ⇒ ***Reverse Only***: Use this option when computing the p-f image for records that were collected using a source that was located near the last geophone in the ReMi array.

Once the parameters are set, click the ‘‘vspect’’ button to start the wavefield (p-f) transformation. A moderate-sized record can take a couple of minutes and a multi-record volume several. The progress will be indicated on the original MS-DOS or command window (Figure 32). Once the computing is completed and the velocity spectrum calculated, the window displaying the p-f image (Figure 33) will open up. Note that this is an entirely new window, and it is safe to close all other ReMi Vspect windows by clicking ‘‘Close Window’’ option under the ‘‘File’’ menu.

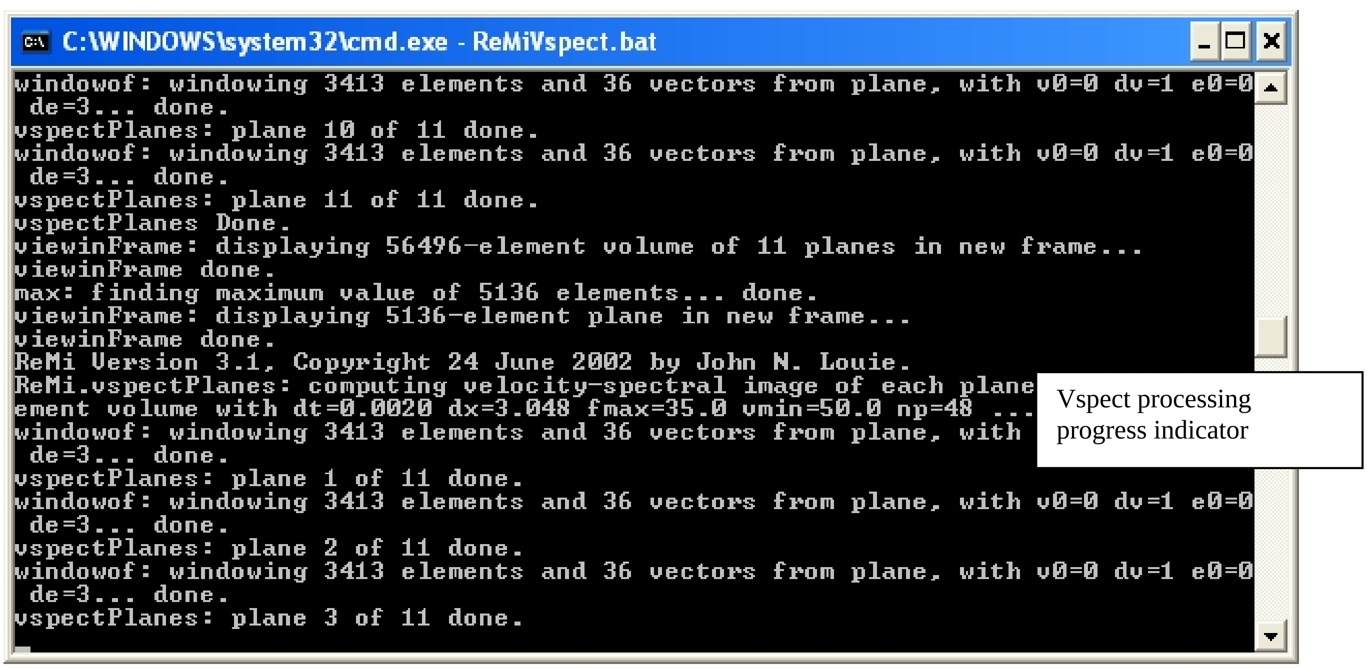


Figure 32: The progress of the p-f transformation will be displayed in the MS-DOS window

3.4.2 Visualizing Results of ReMi Vspect Processing: The Velocity Spectrum (p-f image)

Once the ‘‘vspect’’ processing is completed, a window displaying the p-f image will open up. We are looking for trends of high spectral ratio that slope down to the right, giving lower phase velocities at higher frequencies, as is typical of surface-wave dispersion.

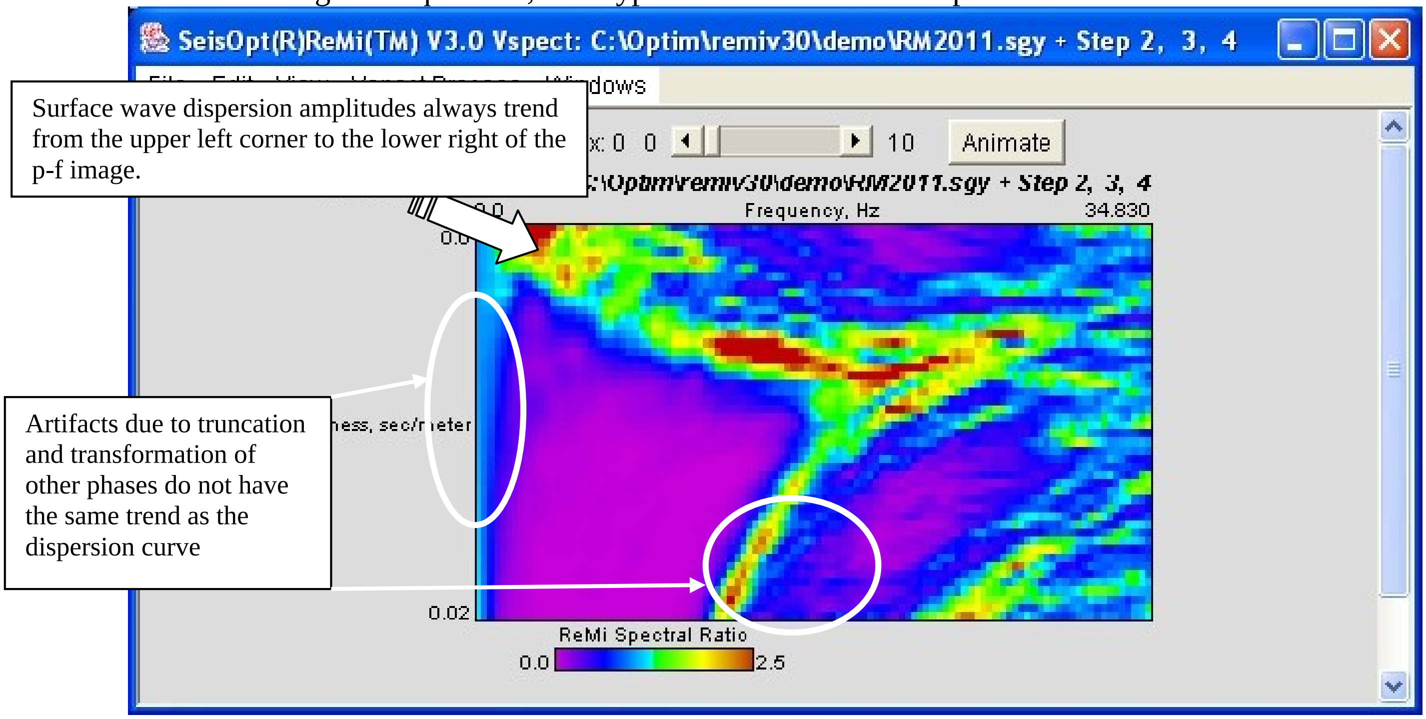


Figure 33: On completion of the computation the window displaying the velocity spectrum for each record will be open up

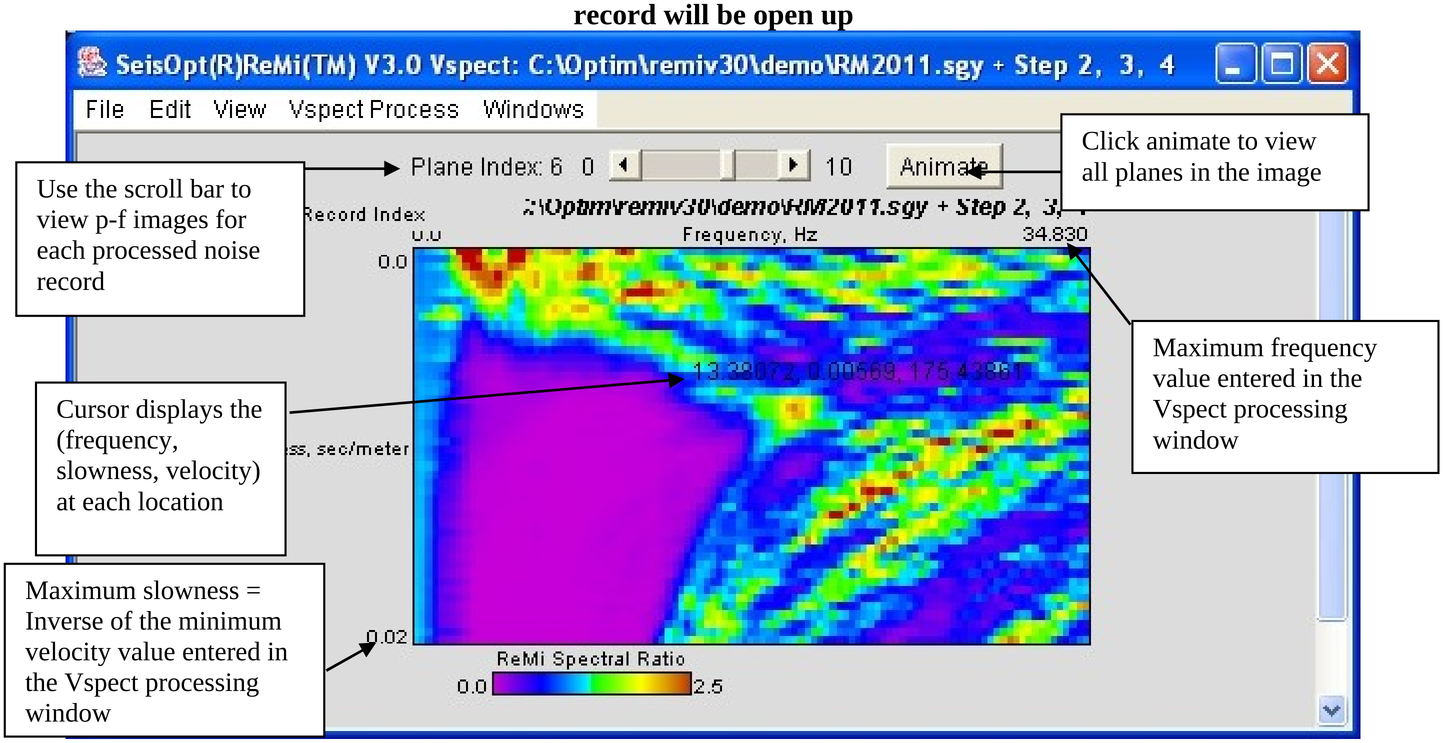


Figure 34: Step 4 computes the p-f image (velocity spectrum) for each individual record and is stored in individual planes. Use the scroll bar to view each one.

The velocity spectrum is calculated for each record and so the p-f image (Figure 33) will have the same number of planes as the number of records in the data set, with the p-f image for each record stored in separate planes. Like for the data set display (Section 3.1.2) click on the ‘‘Animate’’ button to go through all the planes of the image. Use the scroll bar on the top to look at each image individually.

Depending on the level of noise and its frequency content, each data, the p-f image (velocity spectrum) will be different for each record. For example, Figure 34 shows the p-f image of the seventh record (plane index 6) of the data set. The p-f image for each record is stored in different planes. If the seismic record has 11 records (as in this case) the p-f image will have 11 planes (Plane Index 0 to Plane Index 10) corresponding to the velocity spectrum from data sets RM2011 to RM2021 (in this case).

The image is rendered with the frequency (in Hz) plotted along the horizontal (x) axis and the slowness (in seconds/meter, inverse of velocity) plotted down the vertical (y) axis. The origin (0, 0) of the image is the top left-hand corner. Since the highest frequency we used for the analysis was 35 Hz, the frequency scale goes from 0 Hz to 35 Hz. Since 50 m/s was the lowest velocity used (Figure 31), the slowness goes from 0 s/m to 0.02 s/m (1/50). The horizontal and vertical axes are linear in frequency and slowness, respectively. Consequently, the vertical axis is nonlinear in velocity. For instance, halfway down the plot at the left would be at half the slowness, 0.01 s/m, or 100 m/s; a quarter of the way down would be 0.005 s/m, or 200 m/s; and so on. The top row of the plot represents infinite velocity or, an arrival simultaneous at all the geophones.

Details about how to pick the dispersion curve from the p-f image are given in Section 3.6. The important thing to note is that the Rayleigh wave dispersion trends from the top left to the lower right, while the aliasing and wavefield truncation artifacts track from left to right.

3.5 Step 5: Combining Individual p-f Records into one Image

The next step in the ReMi Vspect processing sequence is to combine the p-f image created for each record into one image. This can be achieved by choosing ‘‘Step 5: Combine Record p-fs…’’ from the ‘‘Vspect Process’’ menu (Figure 35). The window shown in Figure 35a will open up. The user has the option of either using all the records or choosing the ones to be used to create the combined p-f image. To use all the records, click on the “Use all planes” radio button and click ‘Ok’. To select the records to be used in the summation, click on “Use selected planes” and choose the planes or records to be used by keeping the ‘Ctrl’ key pressed while clicking on the appropriate “Planes” (Figure 35b). Remember that ‘Plane 0’ corresponds to ‘Plane 0’ in the p-f volume created during Step 4 (Figure 34), ‘Plane 1’ corresponds to ‘Plane 1’ of the p-f image and so on. The combined p-f image shown in Figure 35b was created using planes 0, 4, 6, and 8. When you make the selection, make sure that the desired plane is highlighted in blue and the ‘Ctrl’ key is pressed down when selecting multiple planes.

The option to choose the pf images that go into the combined image allows the user to exclude records that might have incoherent dispersion response or ones that show too many artifacts masking the continuity of the dispersion energy. For example, in the demo, only four out of the eleven records seem to have a dispersion curve that was coherent and continuous across a wide frequency band. Consequently only those four were used to produce the combined pf image.

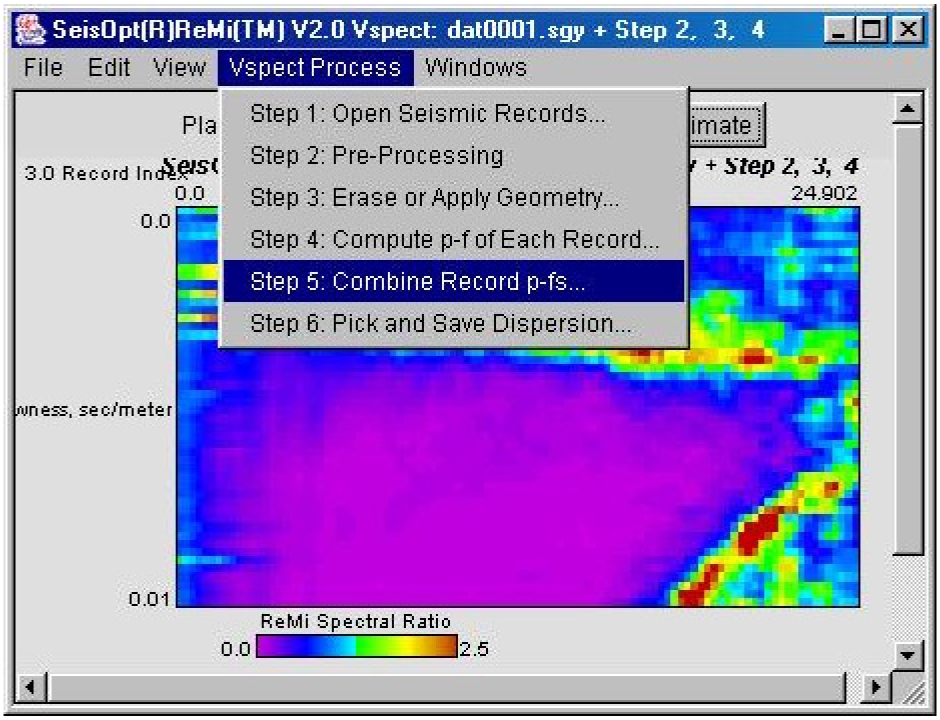


Figure 35: Choose ''Combine Record p-fs..'' to combine individual p-f images into one image.

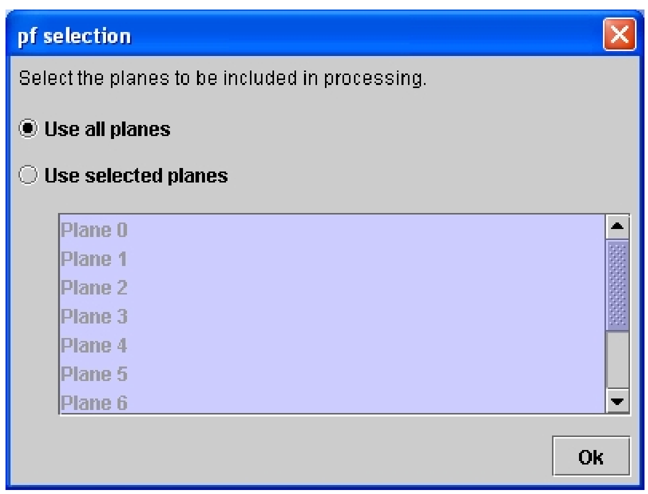


Figure 35a: Choose either all records or select the records to be used to create the combined p-f image.

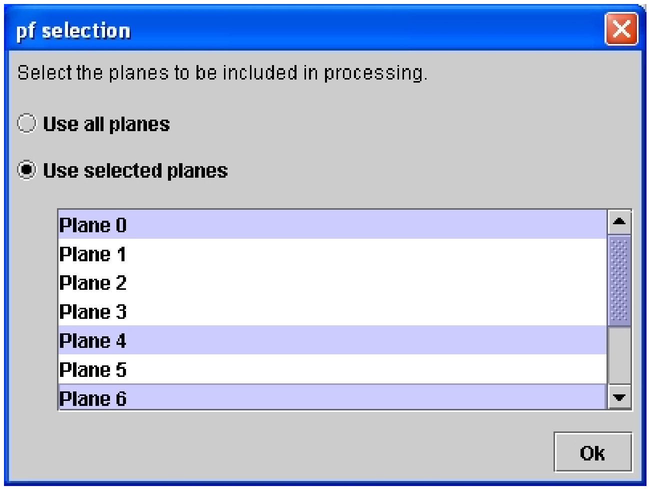


Figure 35b: Planes 0, 4, 6 and 8 were selected to produce the image shown in Figure 36.

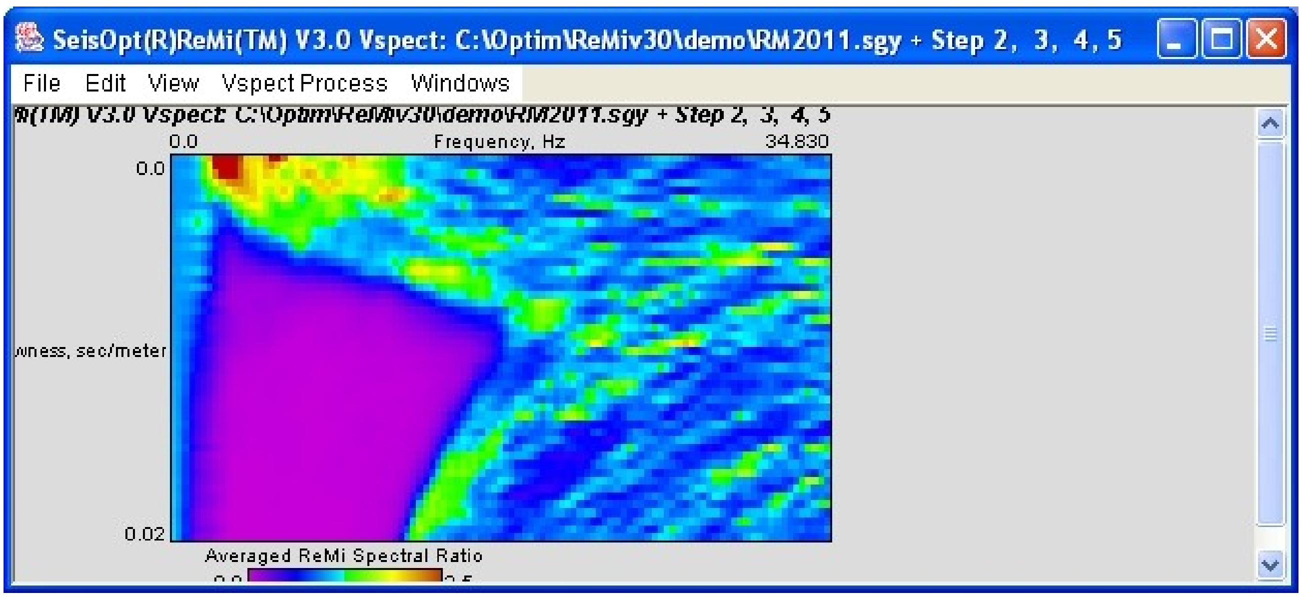


Figure 36: Step 5 combines the individual wavefield transformed images for each microtremor record into one p-f image. In this example, planes 0, 4, 6, and 8 were used to create this image.

ReMi Vspect will stack and sum the individual velocity spectrum into a single p-f image (Figure 36). Examine the individual p-f images (Figure 34) and the combined one and choose the one that shows the Rayleigh wave dispersion the best. That is look for the trend of warm colors that go from the upper-left part of the image to the lower right. We are looking for trends of high spectral ratio that slope down to the right, giving lower phase velocities at higher frequencies, as is typical of surface-wave dispersion. The color palette used in Figure 36 show the higher spectral ratios as warmer colors. In some cases, an individual image will show a larger amplitude trend than the combined one.

3.6 Step 6: Pick and Save Dispersion

After choosing the p-f image that displays the most coherent Rayleigh-wave dispersion, the next step is to pick the dispersion curve and save the picks for input into the interactive velocity-modeling module, SeisOpt ReMi Disper.

3.6.1 Picking the Dispersion Curve

The following steps describe the picking procedure

1. The first step in picking the dispersion curve is to identify the normal-mode dispersion trend, down to the right (starting from the upper left) and distinct from the aliasing and wavefield-transformation truncation artifact trends, which are down to the left. Make sure you are following the trend of the high spectral-ratio band that has the lowest velocity, or the highest slowness. That is the trend that is towards the bottom of the image. Note the velocity and frequency ranges of the identifiable dispersion trend.
2. It is sometimes useful to magnify the image using the ‘‘Zoom’’ options under the ‘‘View’’ menu (Figure 37).

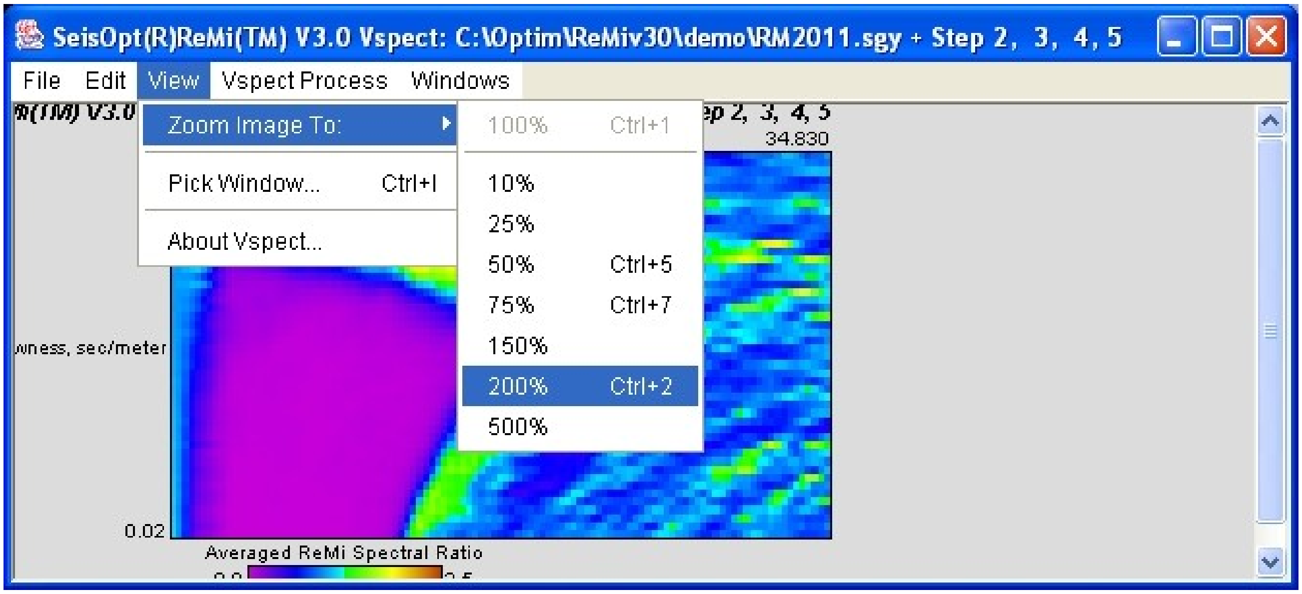


Figure 37: Choose an appropriate ‘‘Zoom’’ percentage to magnify or shrink the p-f image.

3. Pick several frequency-velocity pairs along the trend by left-clicking the mouse on the image (Figure 38)

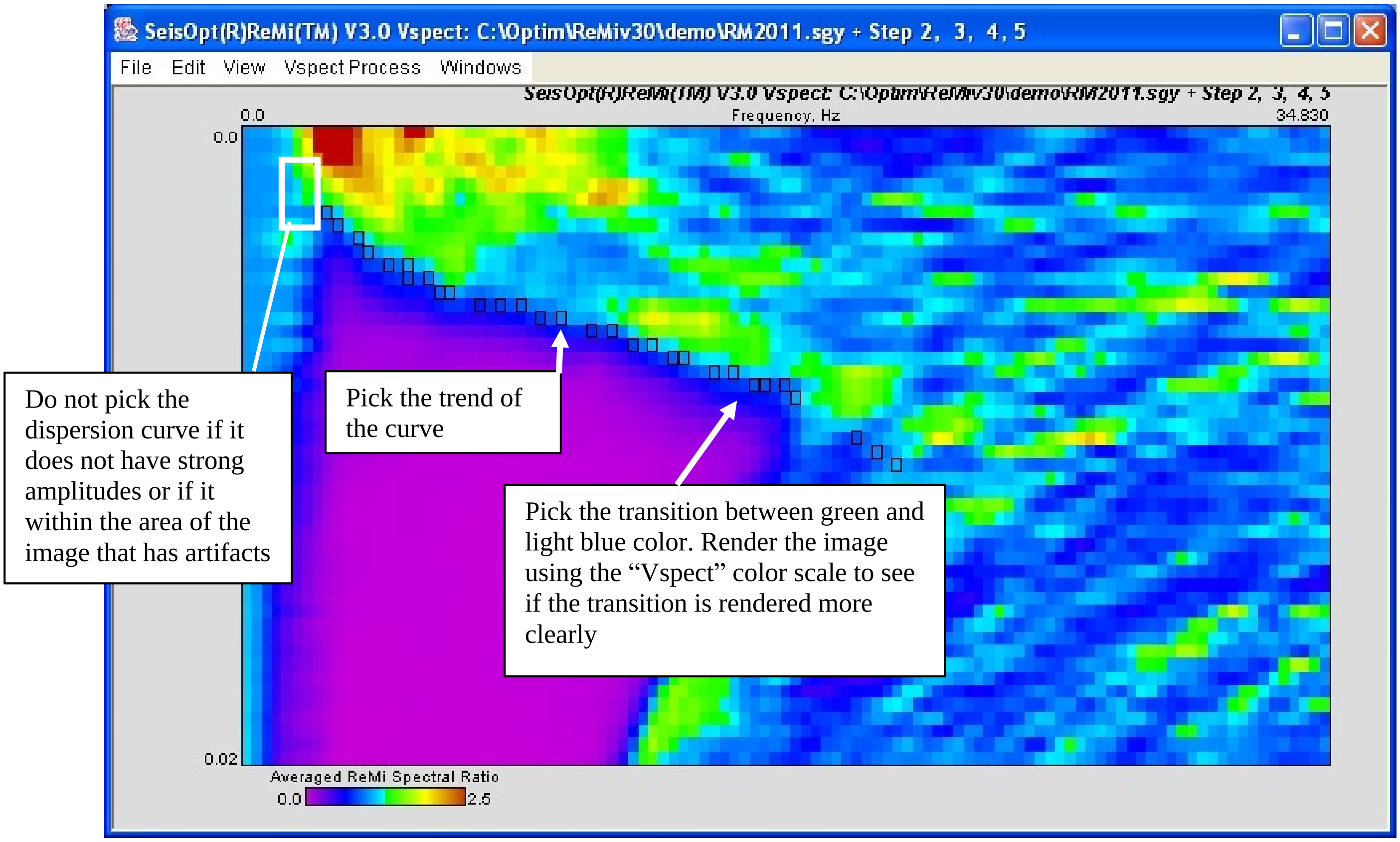


Figure 38: Window showing dispersion picks made on the p-f (velocity spectrum) image.

4. While picking microtremor records, try to pick the lowest energy bound of the high-amplitude (high spectral ratio's) trend. This stays closest to the real velocities, and below the higher apparent velocities of waves traveling not along but obliquely to the geophone array.
5. While picking follow the zone where the warmer colors start to blend with the cooler ones (the blue-green color).
6. It is not necessary that only one pick have to be made for a particular frequency. You can make more than one slowness pick for any given frequency. The range of slowness will give a bound on the velocities that are modeled. For example note that the picks in Figure 38 have some scatter to them. Since SeisOpt ReMi Disper determines the average shear-wave velocity, the picks do not have to be very accurate.
7. Change the color palette to "Vspect" to see if this defines the transition better. To change the color palette, do the following:
   - Select "Plot Parameters" option from the "Edit" menu (Figure 39).
   - Choose "Vspect™" color palette as shown in Figure 40.
   - Click "Apply Changes" to render the image shown in Figure 41.
   - If you wish to go back to the original color palette, choose "Topographic, no clip" and click "Apply Changes".

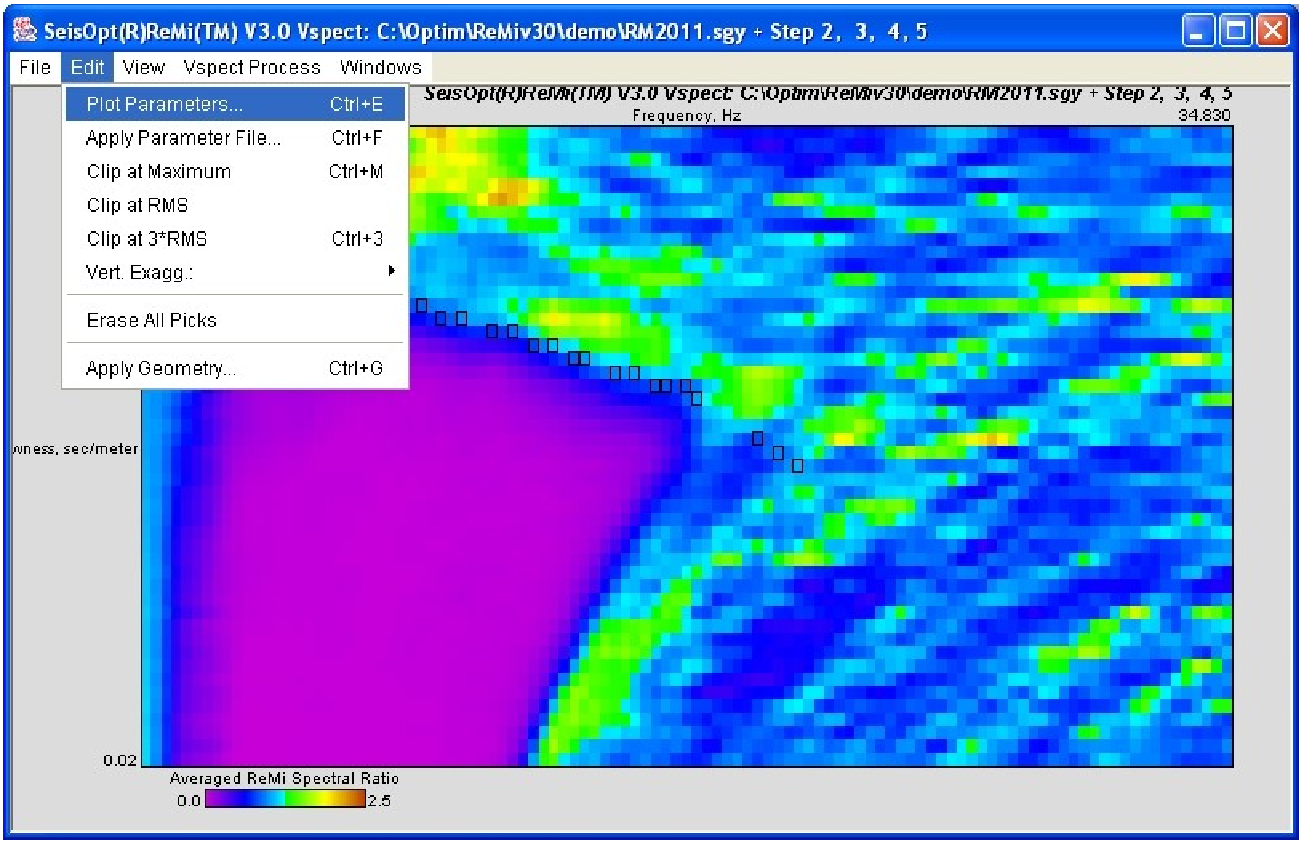


Figure 39: Select "Plot Parameters" from the "Edit" menu to bring up the plot properties window.

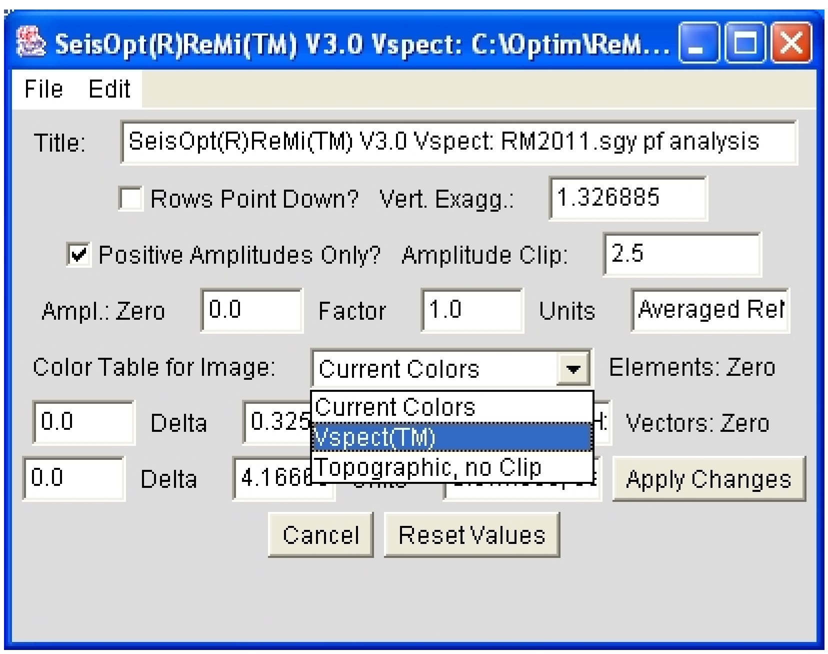


Figure 40: Choose "Vspect™" as shown above. To go back to the original color palette choose "Topographic, no Clip" option.

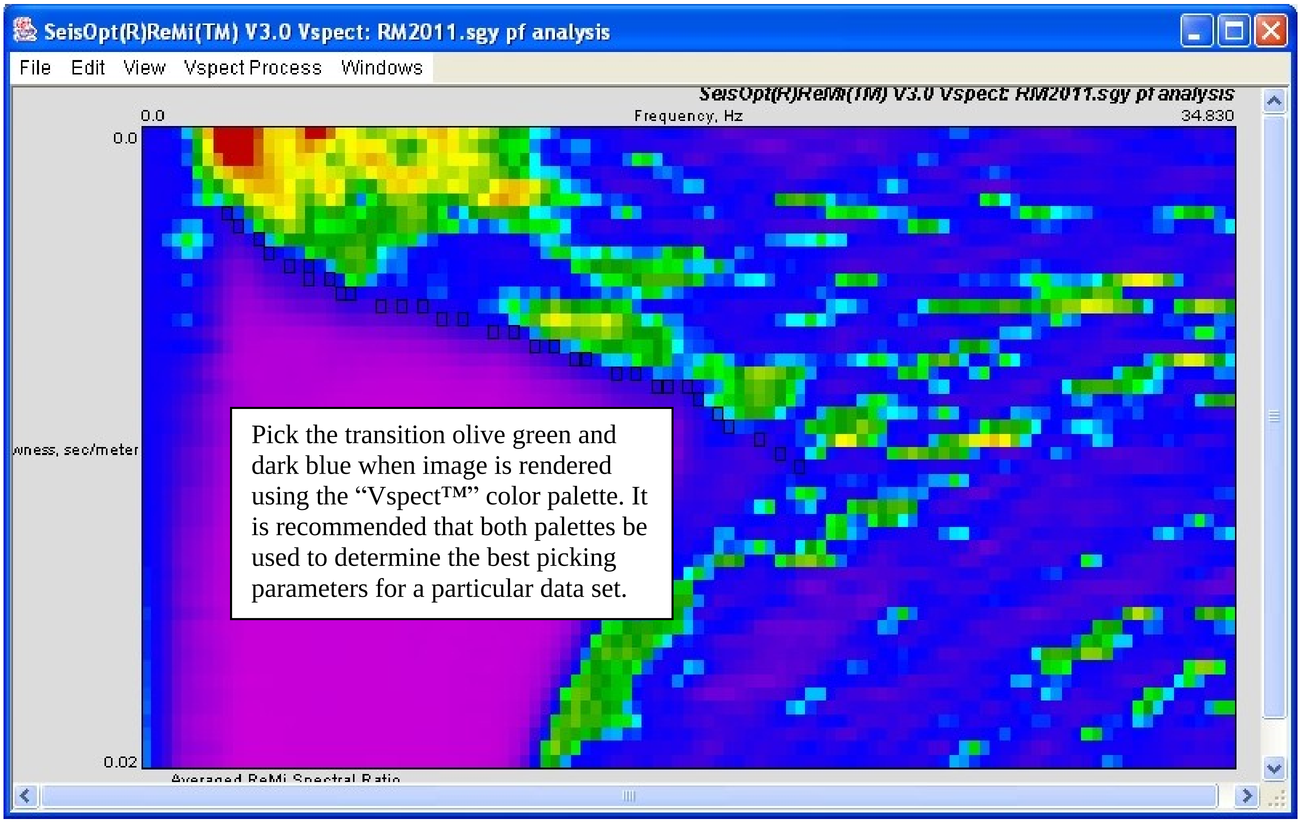


Figure 41: p-f image rendered using the "Vspect™" color palette. Depending on the frequency and amplitude content in the noise records, this color palette will provide a better rendering of the transition from the high to low spectral ratios that need to be picked.

8. It is better to be conservative while picking the data. That is, do not attempt make picks in areas where the warmer colors are absent (lack of high spectral ratio) or where there are gaps in the trend.
9. On the other hand, if there is distinct 'kink' in the dispersion curve, and it is present in all the p-f images of the data set, then it probably implies the presence of a high or low velocity layer, depending on the shape of the 'kink' (see Figure 41a).

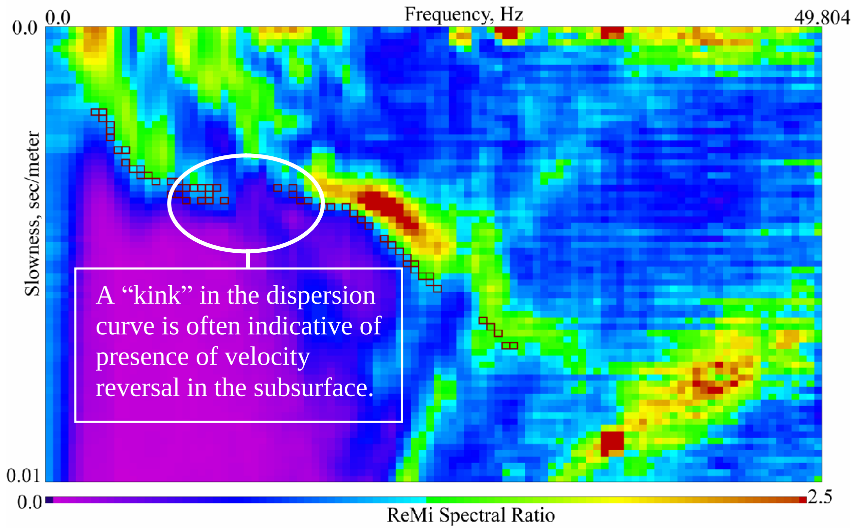


Figure 41a: Example p-f image showing a "kink" in the dispersion curve indicative of a velocity reversal.

10. If you are picking p-f images created from a source (Forward Only or Reverse Only options used for Analysis, see Section 3.4.1), then you need to pick the peak of the dispersion curve as shown in Figure 41b.

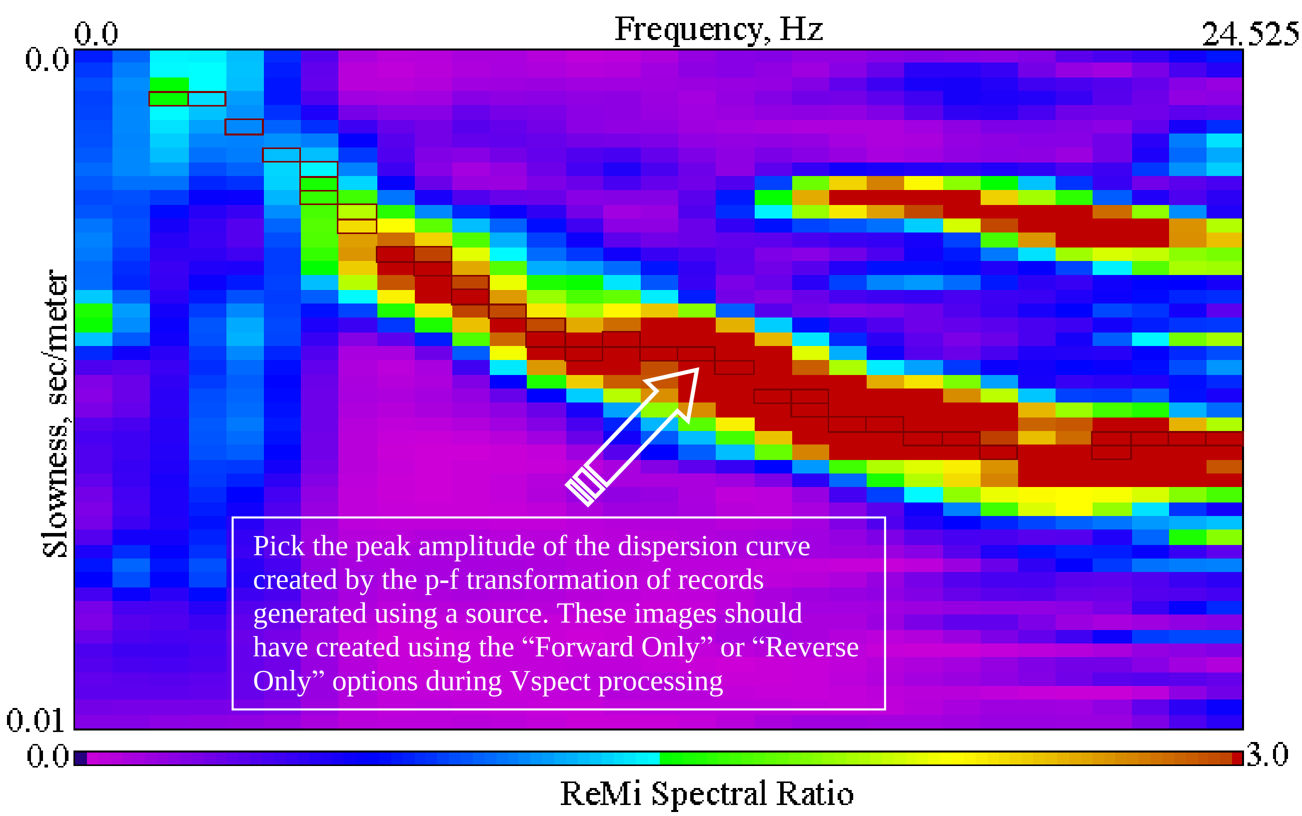


Figure 41b: When picking the p-f image generated from using sources, then the picks need to be made along the peak amplitude and not the low-energy bound.

11. Save the p-f image as a binary file (.SGY) using the ''Write Binary File…''´option under the ''File'' menu. See Section 3.2.1 for details.

12. Optim recommends and encourages users to send their first data set and any difficult data set to support@optimsoftware.com. We will perform the analysis and train you on how to pick and/or verify whether the picks you made are correct.

3.6.2 Exporting and Saving Picks

Now that the picks have been made, the next step is to export and save them to a file. These picks can then be imported into the SeisOpt ReMi Disper module (Section 4.0) for interactive modeling of the one-dimensional shear-wave velocity.

To export the picks choose ‘‘Pick and Save Dispersion…’’ option from the ‘‘Vspect Process’’ menu (Figure 42). This will open up the pick window shown in Figure 43.

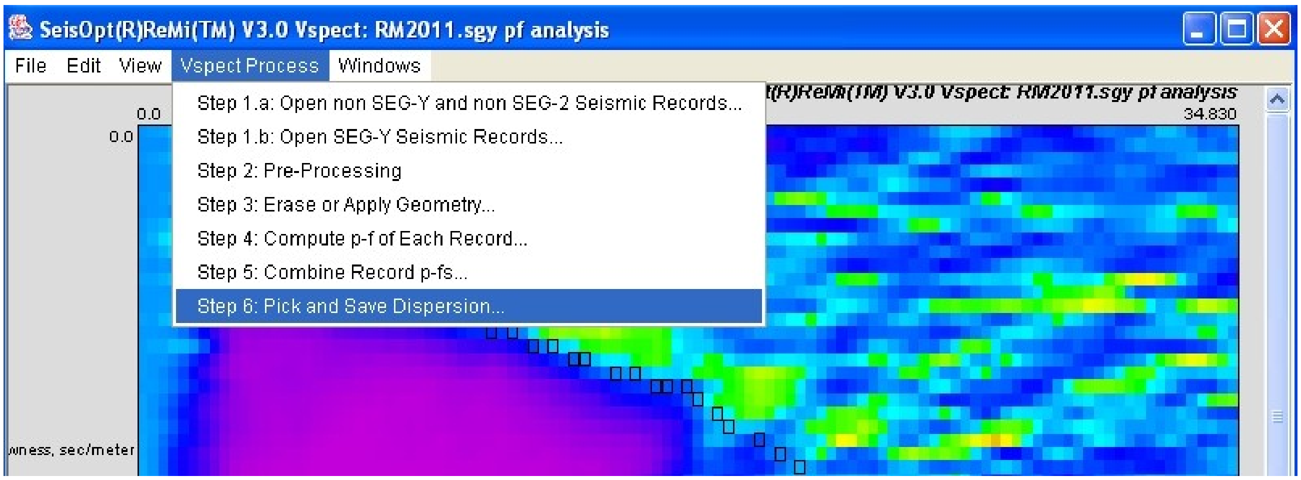


Figure 42: Choose the ‘‘Pick window…’’ option to display and save the dispersion picks.

Click on the ‘‘Save’’ button to bring up the dialog box shown in Figure 44. Choose a file name for the picks. Giving it a ‘.txt’ extension will allow the file to be viewed using Wordpad or Notepad. Clicking on ‘‘Erase All’’ will erase all the picks. The picks that are displayed in the window are ordered according to the order the picks were made. You can delete any unwanted picks by highlighting it and clicking the keyboard ‘Delete’ button (Figure 43a). When deleting picks, make sure that are no blank lines (Figure 43b) between pick entries in the pick window. It is recommended that it is best to erase all the picks and start over if you are unsure of any picks. The best way to pick the dispersion curve is to follow the trend in one sitting.

Once the picks have been saved, you are now ready to perform interactive modeling of the one-dimensional shear wave velocities using the SeisOpt ReMi Disper module.

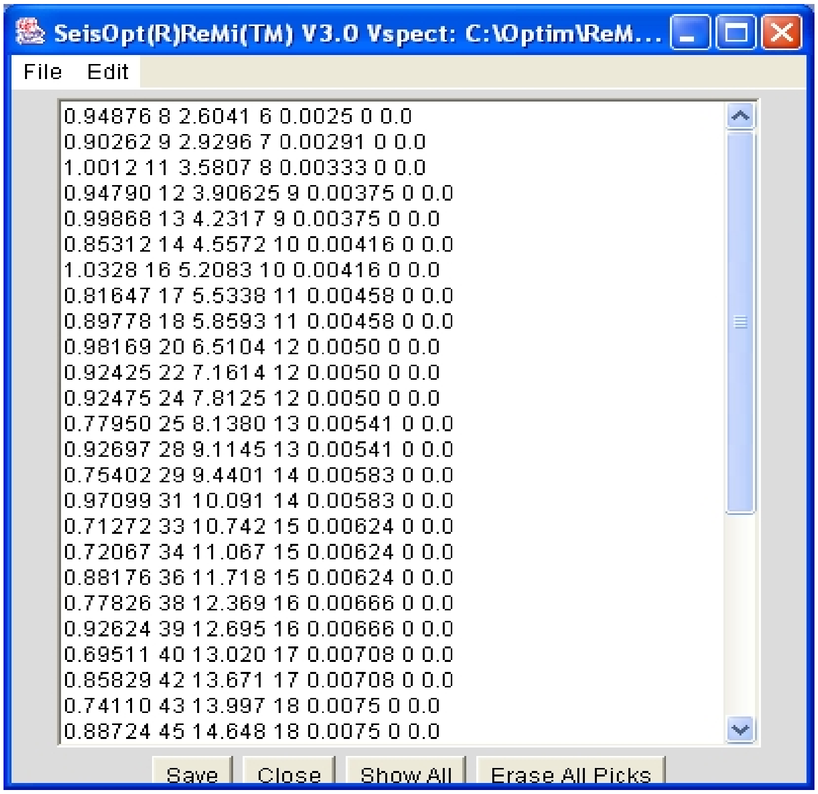


Figure 43: Pick window showing the dispersion curve picks made on the p-f (velocity spectrum) image.

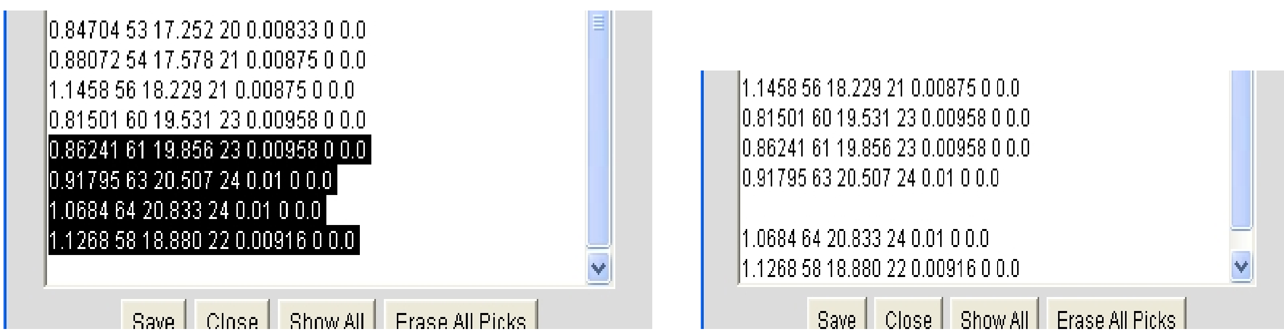


Figure 43(a): Select the picks to be deleted and click 'Delete' to erase them. When deleting, make sure there are no blank lines between entries as shown on the right (b).

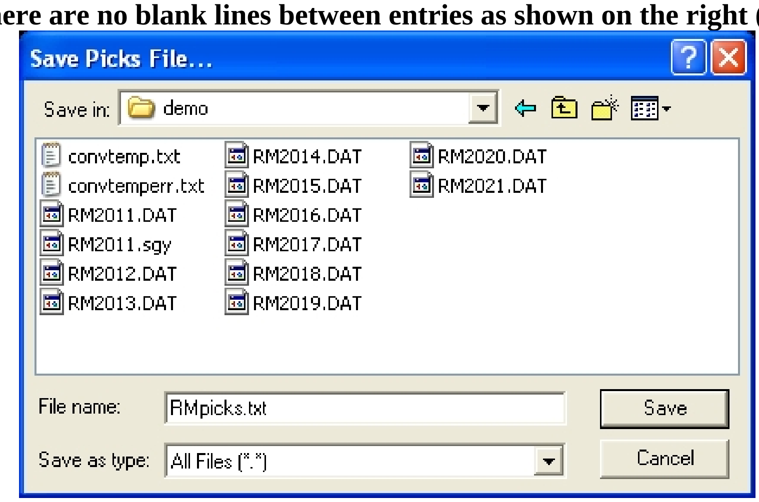


Figure 44: Save the pick file. This can be imported into the SeisOpt ReMi Disper module for interactive modeling of one-dimensional shear-wave velocities.

3.7 Making Report Quality Output of Velocity Spectrum Image

SeisOpt ReMi allows the user to write out the images rendered either as a PostScript (.ps extension) file or as a JPEG file. It outputs the image is displayed in the ReMiVspect window.

So, if you wish to plot the p-f image with the picks you have made on it, then you should use the "Write PostScript File…" option or the "Write JPEG File…" under the "File" menu before you close this window.

The default plot parameters used to render the velocity spectrum (p-f image) can be viewed and edited by choosing the ''Plot Parameters…'' option from the ''Edit'' menu (Figure 39 and 40). This window is similar to the plot parameter window for displaying and plotting the microtremor records. See Section 3.1.3 for a description of each parameter in the window. One can change the title of the plot or the axes if desired. The aspect ratio and the color palette can also be changed. The following sections describe how to create report quality output of the p-f image, with our without the picks.

3.7.1 Creating Output Velocity Spectrum Image

After setting the desired plot parameters as described in the previous section, there are two ways to create a file containing the image shown in Figure 39 that can output to a printer or imported into a report. The first option is to use the "Write JPEG File…" option under the "File" menu (Figure 46) to write out the image in jpeg format (Figure 45). Make sure to provide a '.jpg' extension to the file name.

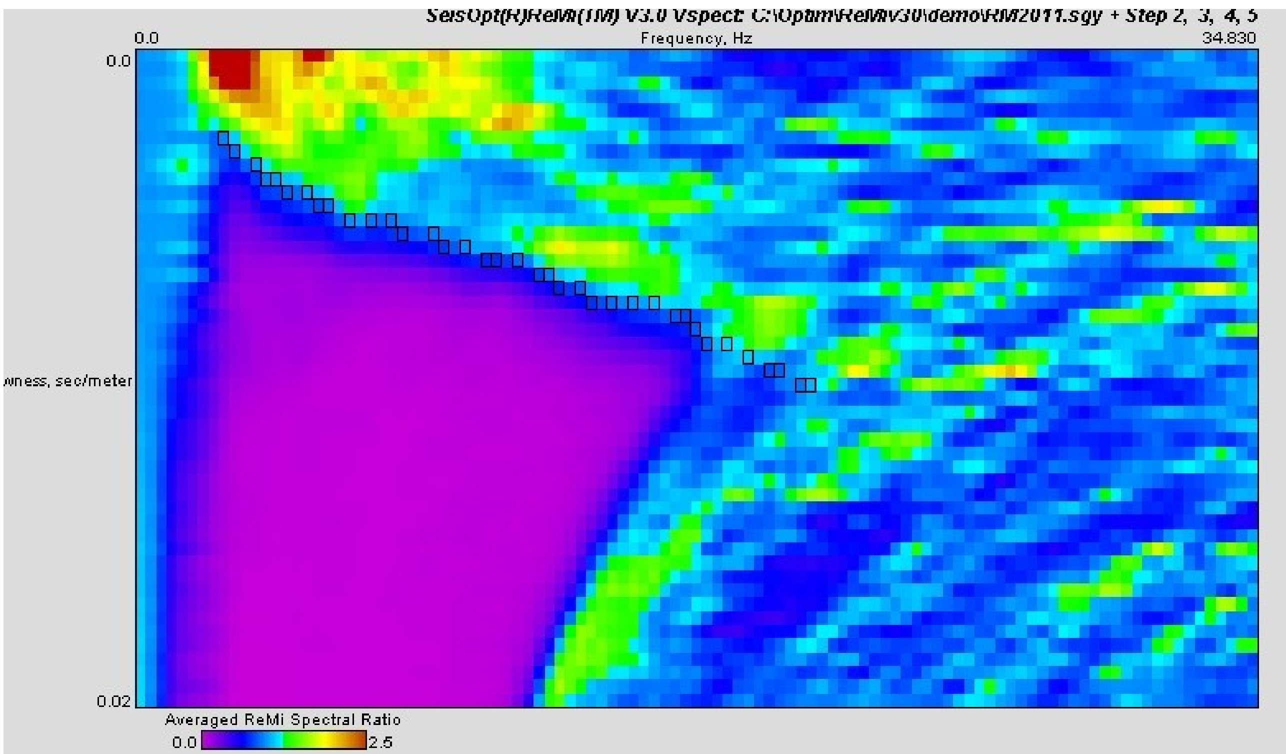


Figure 45: Select ''Write JPEG File…'' under the ''File'' menu to create a jpeg image of the p-f image with the picks.

The second option is to write out a PostScript file. PostScript format files can be read into programs such as Adobe Illustrator and Corel Draw (Version 7.0 or less). This format is useful because, unlike other graphics formats, text and other elements of the image are preserved as discrete objects (e.g., text is still text, not a raster image of text), which makes subsequent editing and customization of these files very easy. To view and/or print these files a program capable of reading EPS format files is needed. If you do not have such a program, you can download a set of free ones, called GhostScript (Copyright © Aladdin Enterprises) and GSView (Copyright © 2000 Ghostgum Software Pty Ltd.), from www.cs.wisc.edu/~ghost/. GSView is the visual interface to GhostScript. It allows viewing, printing, and manipulating the PostScript file for importing into an MS-WORD document (**see Section 3.7.1.1**). GhostScript/GSView can also be used to convert the EPS file into BMP, GIF, PDF, and several other formats. In addition, a quick Internet search will reveal that there are several shareware programs available that allow EPS files to be converted to GIF or BMP format files.

To write out a PostScript file, choose the "Write PostScript File…" option from the "File" menu (Figure 46). A dialog window will open up. Choose an appropriate name with the '.ps' extension.

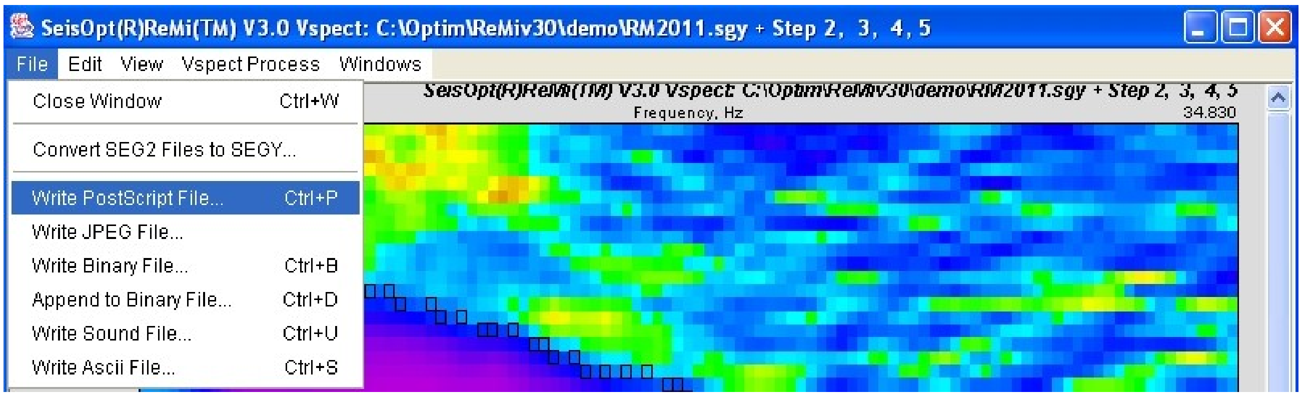


Figure 46: Select the "Write PostScript File…" to write out a PostScript file of the image.

3.7.1.1 Importing PostScript files into an MS WORD document

The PostScript file created above can be modified using the freeware program GhostScript/GSView and inserted into an MS-WORD document. Essentially, it involves using GSView to create an EPS file with a Windows Metafile or TIFF preview. Here are the steps to follow to do the above:

1. Open the PostScript file (.ps) written out by ReMi Vspect using GSView. Click "OK" when warning messages come up. Do not click "Ignore all DSC".
2. Go to the ''Options'' menu and select 'EPS Clip' and 'Show Bounding Box' options (Figure 46a).
3. Make sure the "Ignore DSC" option is not selected.
4. If the entire image is not displayed on the screen, go to 'Media' menu and choose '11x17' option or something appropriate so the entire image is visible.

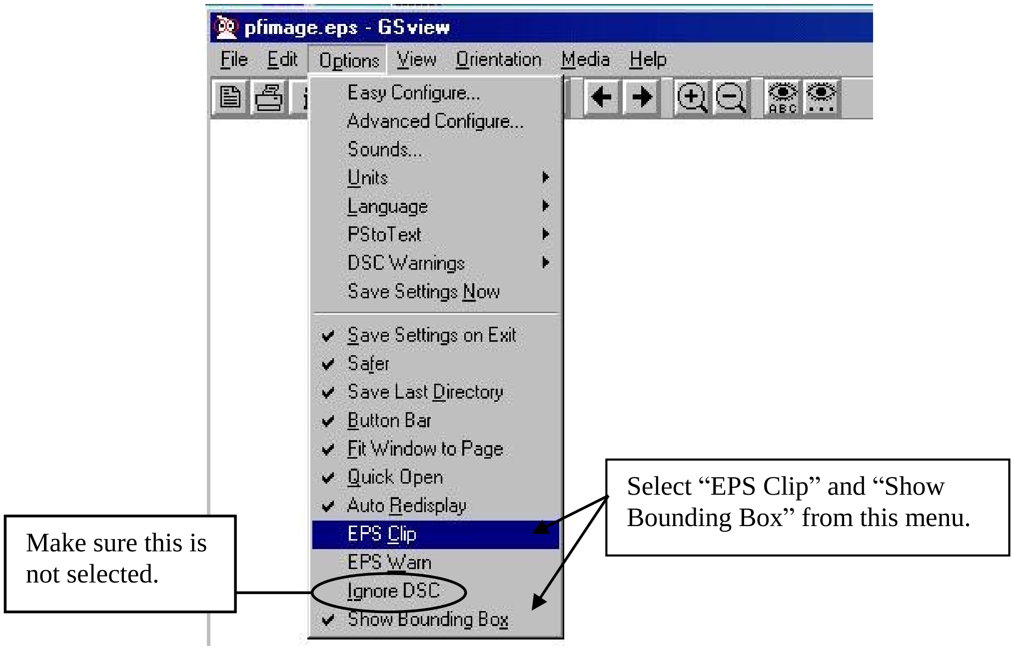


Figure 46a: After opening the PostScript file using GSView, make sure the "EPS Clip" and "Show Bounding Box" options are set as shown above.

5. Go to the ''File'' menu and select 'PS to EPS' option (Figure 46b).

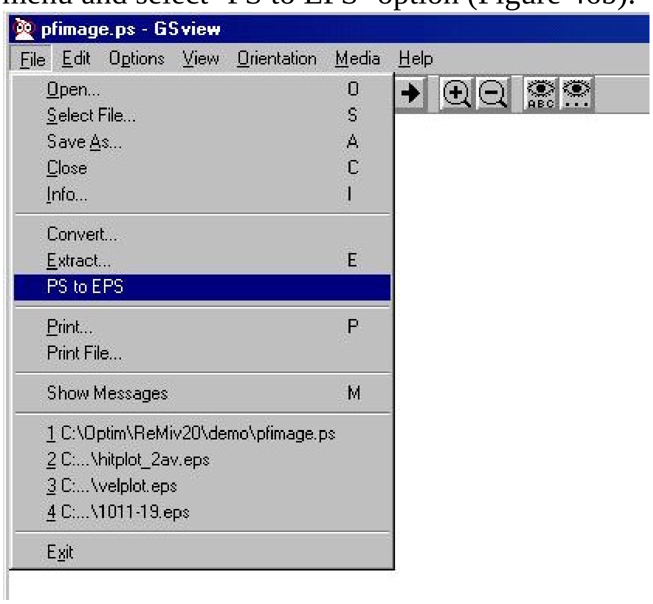


Figure 46b: Choose "PS to EPS" and convert the PostScript image to Encapsulate PostScript format.

6. Make sure the option "Automatically calculate Bounding Box is selected as shown in Figure 46c

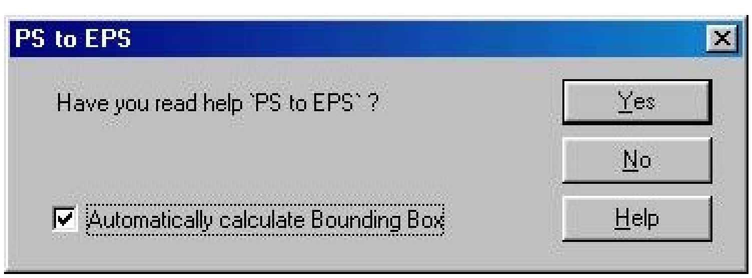


Figure 46c: Select the option "Automatically calculate Bounding Box" in the dialog box.

7. Provide a filename with the '.eps' extension.
8. Read in the saved EPS file.
9. You will notice that the 'Bounding Box' (dashed box) now surrounds only the image.
10. Now go to the ''Edit'' menu and choose ''Add EPS preview'' option, and select the preview type (Windows Metafile or TIFF is the best option) (Figure 46d).

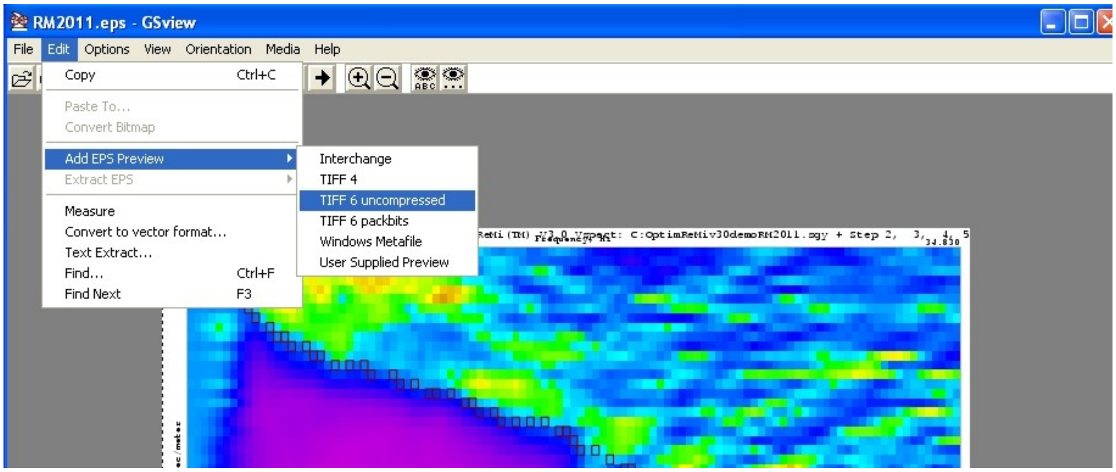


Figure 46d: Choose the preview format to be "Windows Metafile" or "TIFF 6 uncompressed"

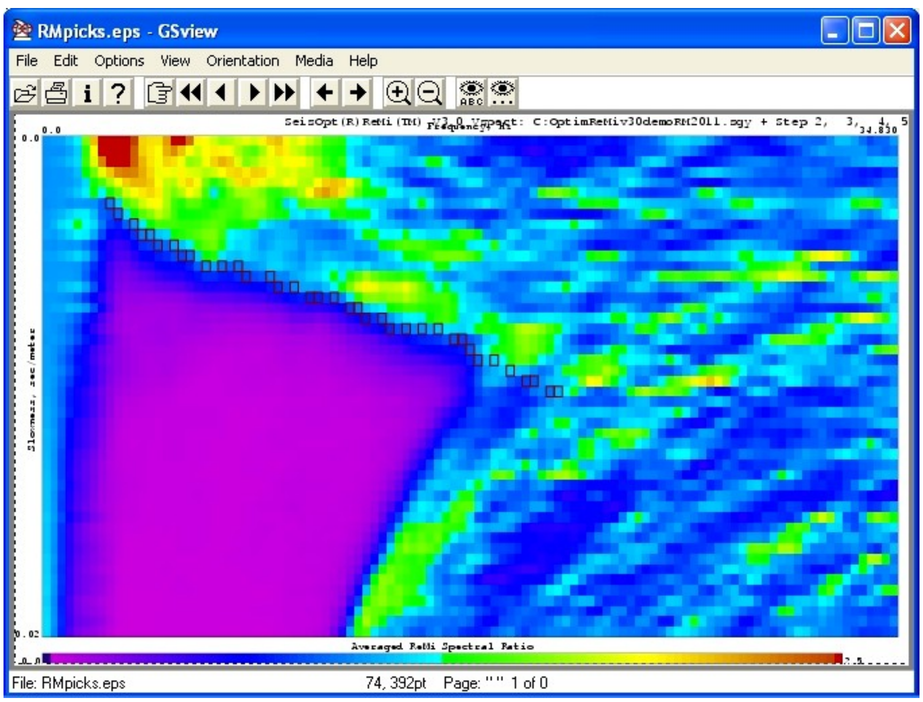


Figure 46e: The Encapsulated PostScript (.EPS) file created can be viewed using GSView or imported into MS-WORD document.

11. Provide a new file name for the EPS file with the preview. Make sure you type in the extension '.eps' for the file name.
12. Save the file as an EPS file.

Now, you will be able to insert the EPS file saved in step 9 into an MS-Word document using the insert-picture option or sent to a printer directly from GSView.

4.0 SeisOpt ReMi Disper module

SeisOpt ReMi Disper uses an iterative dispersion-modeling algorithm based on the method described by Saito (1979) to model the dispersion picks made using ReMi Vspect module. Double-click on the ReMiDisper desktop icon (Figure 2b) to start the Disper module. The module consists of three separate windows shown in Figures 47, 48, and 49. The objective is to model the picked dispersion curve (red filled squares in Figure 48) by clicking and dragging the shear-wave velocity profiles (red lines in Figure 49) so that the calculated dispersion curve (blue lines in Figure 48) matches the picks (red squares). The control window (Figure 47) can be used to toggle between different units (meters/second or feet/second), control the depth of the model, and magnify the image to better view and model the variations in the dispersion curve.

4.1 Description of ReMi Disper Module Windows

The Disper module has three windows and they are described in detail in the following sections.

4.1.1 Controls Window

This will open up at the top left-hand corner of the screen (Figure 47). It contains functions that control the plot parameters of the other two windows, read / write functionality, and parameters that control the convergence and sampling criteria of the modeling algorithm. The following is a description of the parameters:

- Period:
  - Min: Minimum period, in seconds, of the dispersion curve being modeled. This will control the horizontal axis in the 'Dispersion Curve' window (Figure 48).
  - Max: Maximum period, in seconds, of the dispersion curve being modeled. This will control the extent of horizontal axis in the 'Dispersion Curve' window (Figure 48).
  - No. Steps: This controls the number of steps in the calculated period. That is more the number of steps, smoother is the calculated dispersion curve (blue line) shown in 'Dispersion Curve' window (Figure 48). The default value is 20, and usually this is sufficient for modeling picks made from a typical p-f image. But, if the user wishes to model slight variations, one could increase it to 100. Be aware that increasing the number of steps will increase the time to render each new dispersion curve. It is best to increase the number of steps after the dispersion curve has been fit and before the calculated dispersion curve is written out to disk (see Section 4.4).
- Velocity:
  - Min: Minimum velocity of the model and dispersion curve. This will control the vertical axis of the 'Dispersion Curve' window (Figure 48). Also, make sure this value is less than the minimum velocity used while performing the interactive velocity modeling in the 'Model Profiles' window (Figure 49). Else, a 'root not found' error will occur. The values are either in meters/second or feet/seconds, depending on the choice made by the user.
  - Max: Maximum velocity of the model and dispersion curve.
  - Step: This controls the convergence criteria of the modeling algorithm. In most cases, this will not have to be changed. The only scenario this may have to be changed is when the

velocities are very low (< 100m/s) and the interactive module fails to model the picked dispersion values very well. In this case, the step size can be decrease. We strongly recommend that you contact support@optimsoftware.com in such cases for suggestions on how to model set this parameter.

- Calculate:
  - ➢ Rayleigh Dispersion: This is the default option and must be the one used. This models the picked dispersion curve, using the Rayleigh-wave phase-velocity dispersion analysis criteria. The one-dimensional shear-wave velocity derived using this option will be closer to the real earth shear wave velocity.
  - ➢ Quarter-Wave Approx.: This option models the picks assuming the quarter-wavelength approximation. That is, it assumes the average shear-wave velocity is approximately equal to the one-quarter of the dominant wavelength. This option should be used only for instructional purposes.

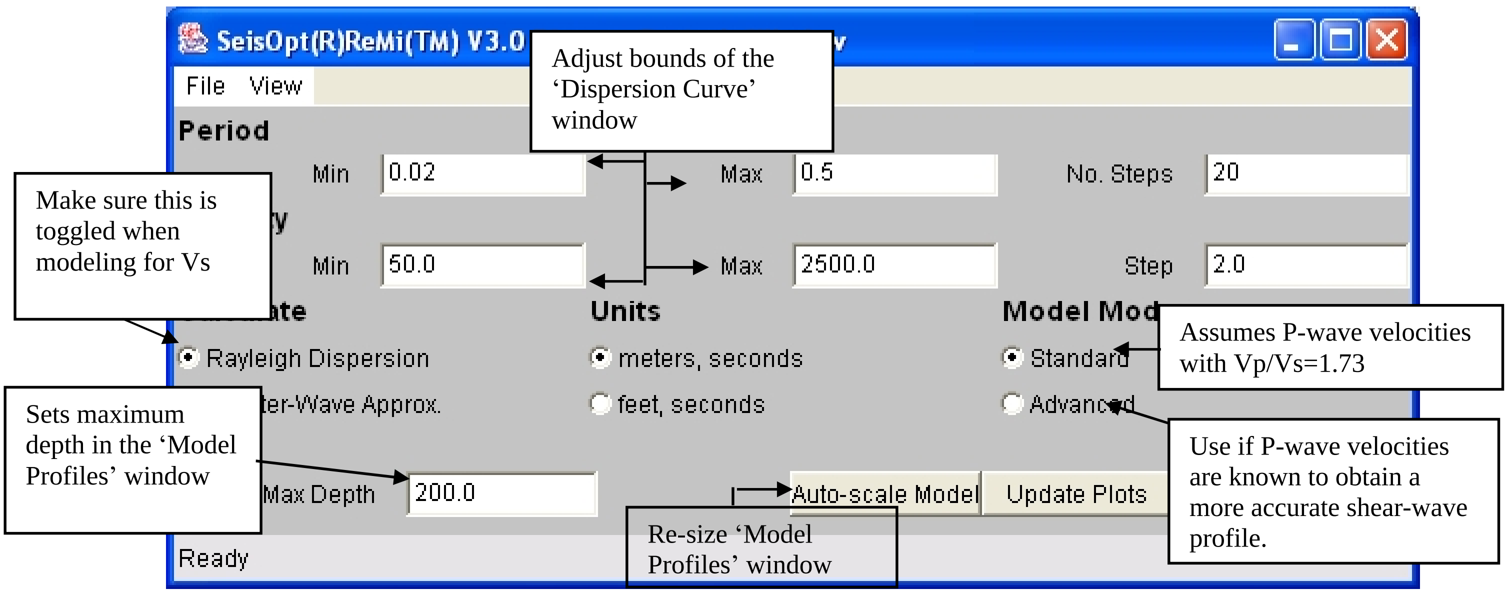


Figure 47: Set parameters for interactive velocity modeling and dispersion curve plotting.

- Units:
  The user has the option to toggle between meters/second and feet/second. The default setting is to meters/second. The output velocity model will always be written out in the unit that was selected during the modeling process.
- Max Depth:
  This controls the maximum depth of the velocity model shown in the 'Model Profiles' window (Figure 49). Make sure that this value is set to be greater than the deepest interface in the model.
- Auto-scale Model:
  Clicking this will automatically re-size both the 'Model Profiles' and 'Dispersion Curve' windows to fit the maximum and minimum velocity limits in the modeled data.
- Update Plots:
  Clicking this button will refresh the velocity models in the 'Model Profiles' window and the dispersion curves in the 'Dispersion Curve' window.
- Model Mode:

- The “Standard” mode will display only the shear-wave velocity (red vertical line) and density (vertical blue line) for each layer. It assumes the P-wave velocities are about 1.73 times that of the shear-wave velocity (Poisson solid assumption).
- The “Advanced” model will display the P-wave velocity (green vertical line) for each layer in addition to the shear-wave and density profiles. This mode can be used when P-wave velocities for the site are available either from refraction surveys or well log. If so, this will improve the accuracy of the modeled shear-wave profiles.

- Pull-down menus:

  Across the top of this window under the ‘‘File’’ menu are options to open (import) and save (export) the model and the dispersion curves. The “Convert File” options can be used to convert dispersion files created from other programs (like SASW) to format for ReMiDisper (Section 4.1.1a). “Create 2D Profile” option is to create a 2D profile from a series of 1D profiles. This will be explained in a later section. The ‘Quit’ option is also under this menu.

4.1.1a Importing dispersion picks from third party software

The “Convert File” option under the “File” menu (Figure 47a) in the ReMi ‘Controls’ window gives the user the ability to import dispersion picks made using other analysis software into SeisOpt ReMi Disper module for shear-wave velocity modeling.

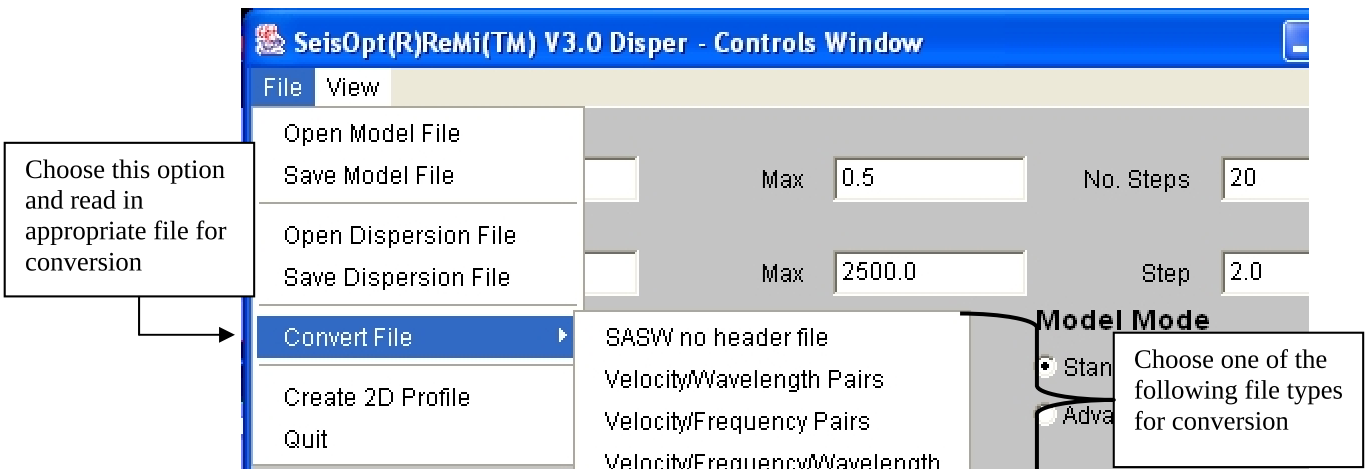


Figure 47a: Select the file type for converting dispersion picks made using other software for input into ReMi Disper module

Currently, the following types of dispersion files can be read in and converted:

1. SASW (no header) file: Dispersion picks output from SASW (Spectral Analysis of Surface Waves) software. It assumes the file has no header information and there is only a single dispersion curve in the file being converted.
2. Velocity/Wavelength pair: ASCII file containing two columns, with the first column being velocities and the second column being wavelengths.
3. Velocity/Frequency pair: ASCII file containing two columns, with the first column being velocities and the second column being frequencies.
4. Velocity/Frequency/Wavelength: ASCII file containing three columns, with the first column being velocities, the second column being frequencies and the third being wavelengths.

Select the appropriate option to bring the “Browse” window (Figure 47b).

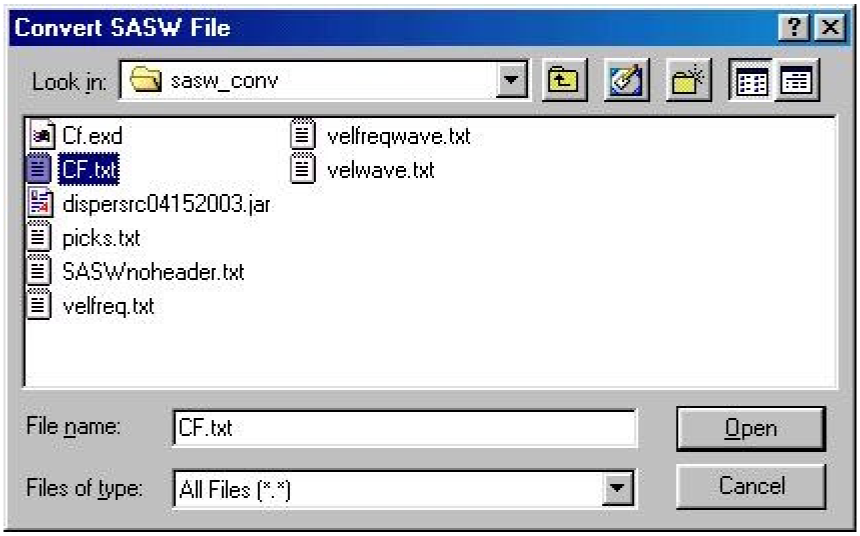


Figure 47b: Choose the appropriate file for conversion

The program will convert the input file into a dispersion file with extension "_disper.txt" which can be read into ReMi Disper.

4.1.2 Dispersion Curve Window

This window will open up below the 'Controls' window. When first opened, this will have a blue line showing the modeled dispersion curve (Figure 48).

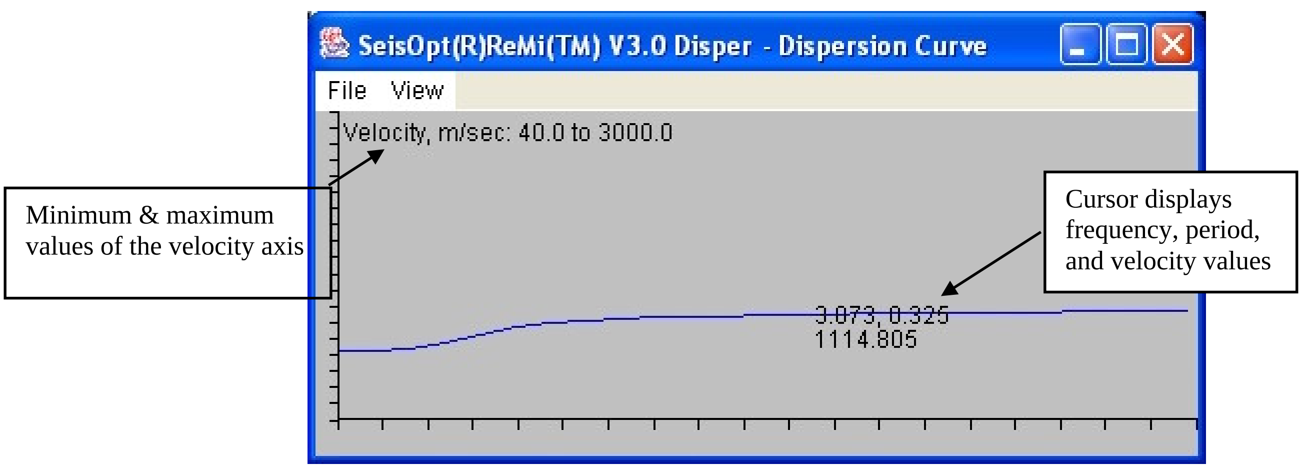


Figure 48: Dispersion Curve window of the ReMi Disper module. This window displays the modeled and picked dispersion values.

Unlike the p-f image (Figure 34), the horizontal axis in this plot is the period (inverse of the frequency) in seconds and the vertical axis is phase-velocity, in meters/second or feet/second, depending on the units chosen in the 'Controls' window (see Section 4.1.1). The minimum velocity and minimum period (maximum frequency) are at the left lower corner of the plot. Velocities and period increase linearly in the vertical and horizontal direction, respectively. The maximum and minimum values of the period, frequency, and velocity are displayed on the screen. Changing the limits in the 'Control' window (Figure 47, Section 4.1.1) can change these values. Moving the cursor anywhere in the window will display the frequency, period, and velocity values. When the picked dispersion values are imported into the ReMi Disper module, they will be displayed as filled red squares. The objective then is to match and overlay the blue line on top of the red squares. Section 4.2 describes the modeling process in detail.

4.1.3 Model Profiles Window

Figure 49 shows the 'Model Profiles' window. The window in the "Standard" model mode is shown on the left while the window in the "Advanced" mode is shown on the right. The default display is a three-layer model.

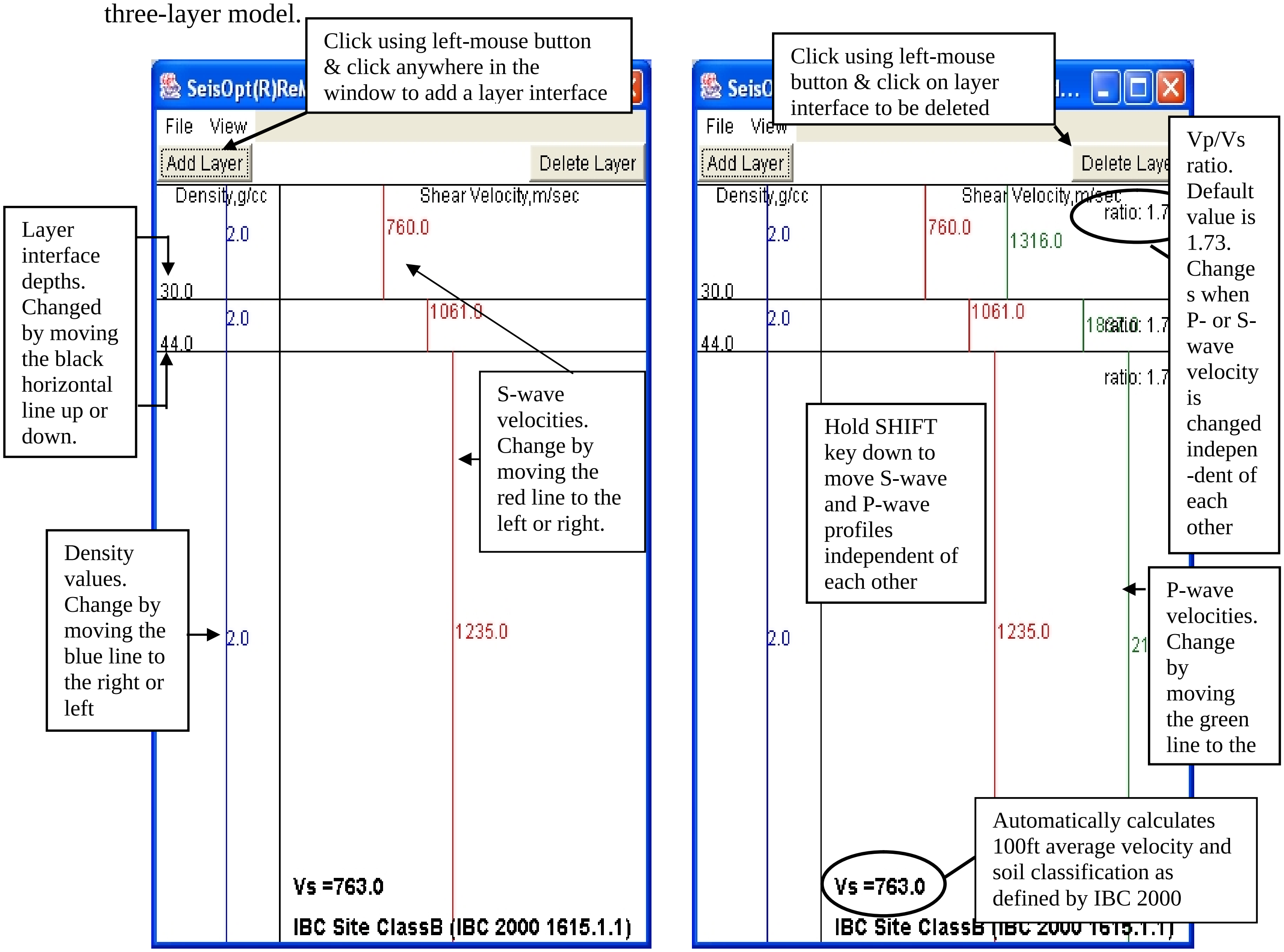


Figure 49: 'Model Profiles' window. This window is used to interactively model the one-dimensional shear-wave velocity of the subsurface to match the dispersion picks. The figure of the left is the window when used in the "Standard" mode while on the right is the default window when used in the "Advanced" model mode.

- Vertical red lines: These are the shear-wave velocities for each layer. The value is either in meters/second or feet/second and is set in the 'Controls' window (Section 4.1.1). One can manipulate the velocity model by clicking on the line with the left mouse button and dragging the mouse over the plot with the left button held down. Each segment of the red line, indicating velocity of a particular layer, can be move independent of the other. As the velocity profile is being changed, the synthetic (calculated) dispersion curve (blue line in Figure 48) in the

‘Dispersion Curve’ window will change. The velocity values for each layer are displayed to the right of the red line and they change as the lines are moved (Figure 49). Note that in the “Standard” model mode only the red S-wave velocities appear. The modeling assumes the subsurface has a Vp/Vs ratio of 1.73. If the ratios are known to be different for the site under investigation or if reliable P-wave velocities are available from refraction surveys, logs, etc., then the “Advanced” model mode can be used.

- Horizontal black lines: These indicate the layer interface depths. Clicking the left mouse on any interface and dragging the mouse over the plot with the left mouse button held down will change the depth of the layer. The depth value is shown on top of each interface towards the left of the plot (Figure 49).
- Blue vertical line: The blue vertical line towards the left of the plot indicates the density within each layer. They are assumed to be of a constant density of 2.0 g/cc for all the layers. This can be changed if reliable density values are available. For most soil types it has been shown that density variations are within 5% to 10% of 2.0g/cc and these have negligible effect on the shear-wave velocity profile. But, if density values are available they should be used for modeling the shear-wav velocity profile.
- Green vertical line: These are the P-wave velocities for each layer. The value is either in meters/second or feet/second and is set in the ‘Controls’ window (Section 4.1.1). These appear only when the “Advanced” model mode is selected in the ‘Controls’ window and should be used only when reliable P-wave velocity information is available (like from refraction surveys, logs, etc). Using accurate P-wave velocity will lead to a more accurate shear-wave (S-wave) velocity profile. One can manipulate the velocity model by clicking on the green line with the left mouse button and dragging the mouse over the plot with the left button held down. Each segment of the green line, indicating the P-wave velocity of a particular layer, can be moved independent of the other. Doing so, will alter the S-wave velocity also, keeping the Vp/Vs ratio constant at 1.73. To move either the P-wave or S-wave profile only, press and hold down the SHIFT button while using the left mouse button to move either the P-wave (green line) or S-wave (red line) profiles. Thus, layers with Vp/Vs ratios greater or less than 1.73 can be modeled.
- Add Layer button: The ‘‘Add Layer’’ button on the top left allows the user to add a layer interface to the model. To do so, left-click on the button and then left-click at the location where the layer has to be added. For example, to add a layer interface at, say depth of 59 m, left click on the ‘‘Add Layer’’ button and then left click at about 59 m within the ‘‘Model Profiles’’ window. The correct depth can be achieved by dragging the interface after it has been added.
- Delete Layer button: To delete a layer interface, left-click on the ‘‘Delete Layer’’ button and left-click on the interface that has to be removed.
- Vs: The 100 ft average velocity is continuously calculated and display at the bottom of the plot window. This forms the basis for classifying the soil as per IBC 2000. The soil classification is also shown in the window. **Important: There are situations when the soil classification may have to modified based on presence of low-velocities etc. Please consult a certified professional engineer before confirming the soil classification. ReMi only provides the 100 feet average shear-velocity calculated as per IBC 2000.**

4.2 Interactive Modeling using ReMi Disper

The following steps enumerate how to perform interactive modeling using the Disper module:

1. Open the dispersion pick file created during Step 6 of the ‘Vspect Processing’ sequence (Section 3.6.2) using the ‘‘Open Dispersion File’’ under the ‘‘File’’ menu in the ‘Controls

Window' (Figure 50). You can also load the file using the "Open Dispersion File" under the "File" menu in the 'Dispersion Curve' window. If you plan to model the velocity in units of feet/second, make sure you toggle the 'Units' option to that in the 'Controls Window' (Figure 53). It is recommended that the modeling be done using the "Standard" model mode first. If P-wave velocities are available, then the shear-wave velocities can be refined in the "Advanced" model mode.

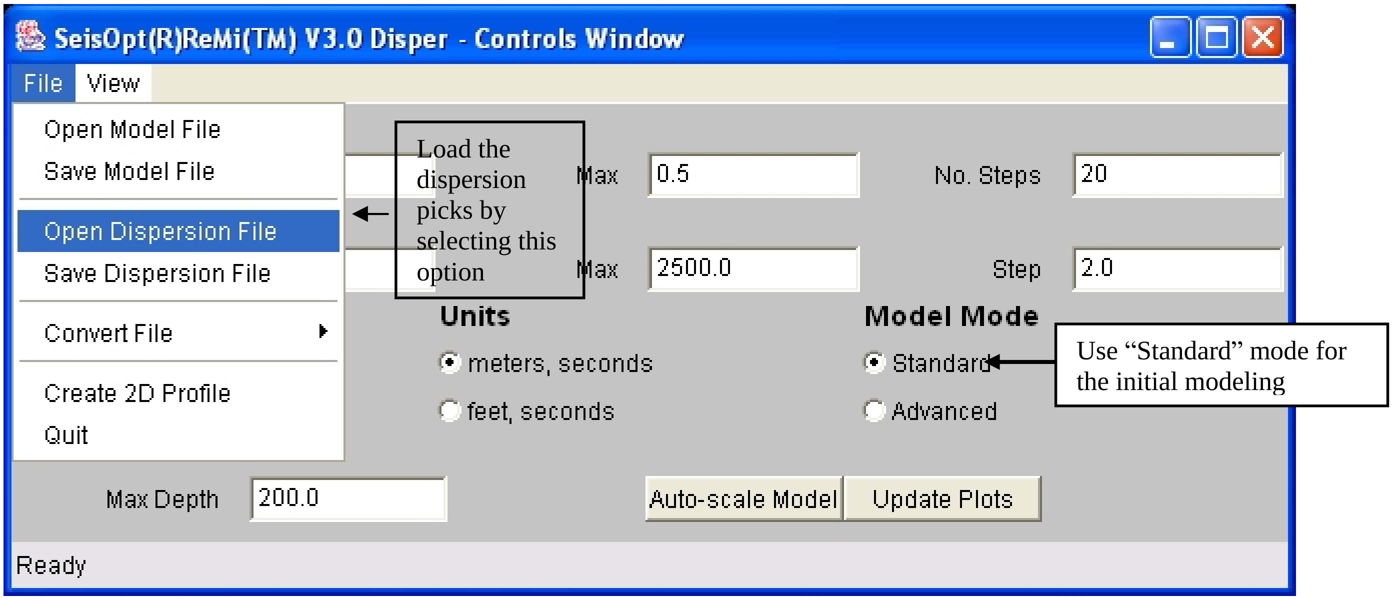


Figure 50: Choose the Open Dispersion File option to load the dispersion picks

2. The dialog window shown in Figure 51 will open up. Choose the appropriate pick file that was save from the ReMi Vspect module (Section 3.6.2, Figure 44). For the ''demo'' data, this file is 'RMpicks.txt'.

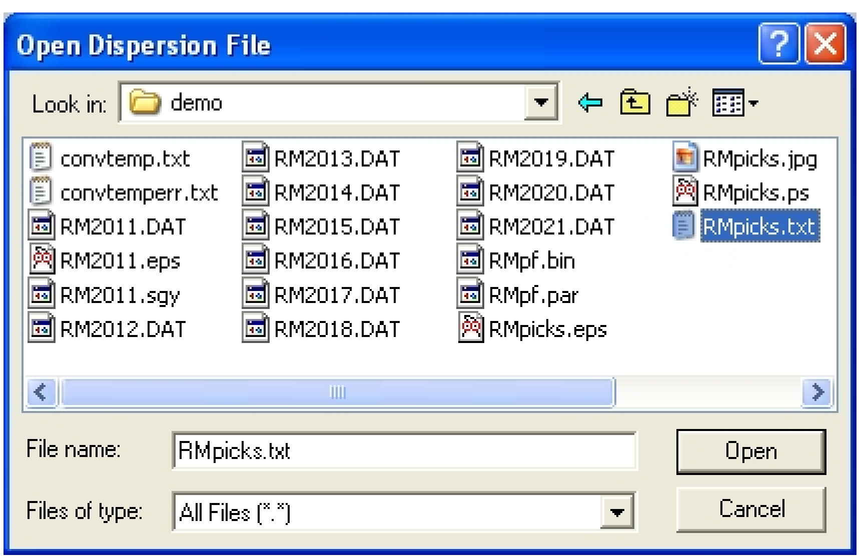


Figure 51: Select the pick file that was created during Step 6 of the Vspect processing sequence.

3. The dispersion picks will now be displayed as red filled squares in the 'Dispersion Curve' window (Figure 52).

4. If you wish to model the velocities in feet/sec, toggle the 'Units' option in the 'Controls Window' to feet per second (Figure 53). The velocities displayed in the 'Model Profile' (Figure 54a) and the 'Dispersion Curve' window will change accordingly (Figure 54a). Note that the 'Velocity Step' in the 'Controls Window' will also change. It is recommended that this value not be changed during processing. The only time this might have to be changed is if the shear-wave velocities are very low (<100 m/s) and if the interactive modeling fails to fit the picked picks. If such a situation arises, contact support@optimsoftware.com for suggestions.

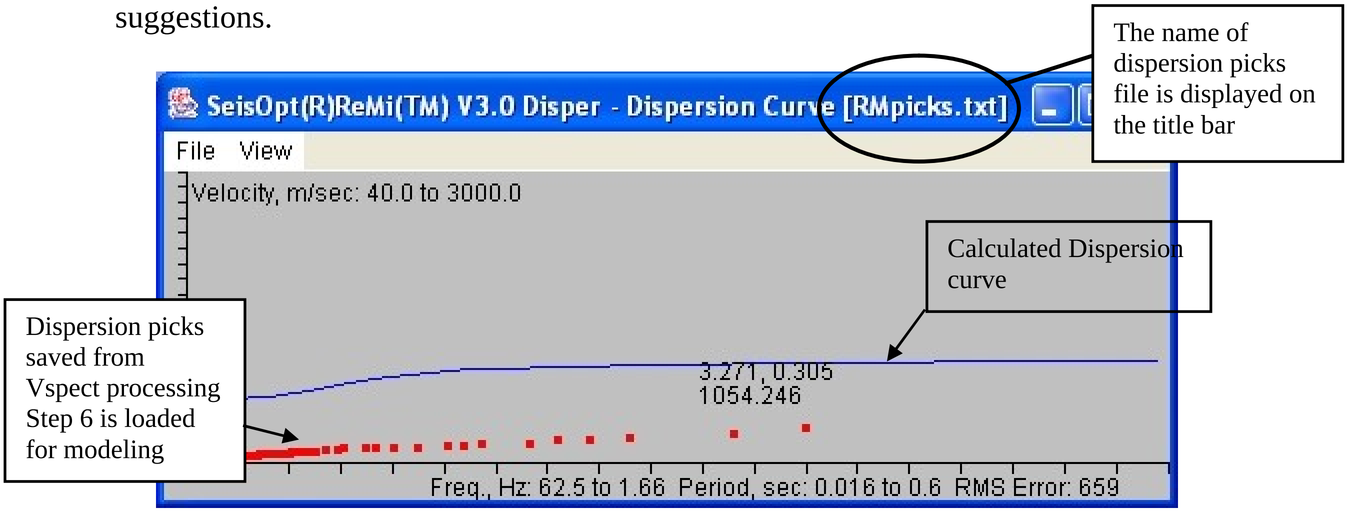


Figure 52: The dispersion picks are displayed as red filled squares in the 'Dispersion Curve' window.

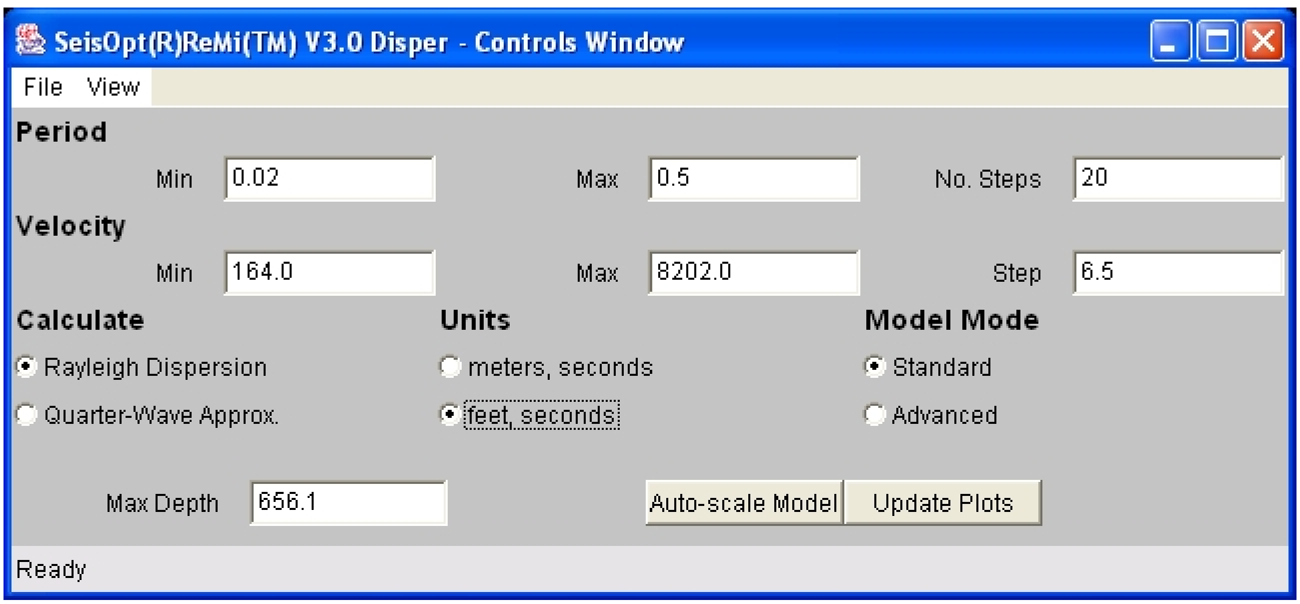


Figure 53: Select 'feet, seconds' to do the modeling in units of feet/second

5. Now click and drag the velocity profile (vertical red lines in the 'Model Profiles' window) and the layer interfaces (horizontal black lines in the 'Model Profiles' window) until the modeled dispersion curve (blue line the 'Dispersion Curve' window) overlay the dispersion picks (red filled squares). The RMS error that appears at the bottom right hand corner of the 'Dispersion Curve' window is a quantitative estimate of the fit. Lower the error, better the fit for a particular data set.

6. It is recommended that the modeling start from the top down. Take an iterative, top-down approach to interactively modeling the dispersion picks. Figures 54a, b, c, d, e, and f illustrate the steps involved in modeling.
7. First fit the lower periods (higher frequency) picks first, since they sample the shallow horizons. Since the shallow horizons are lower velocities associated with the top layer, keep changing the layer velocities and interface depth of the first layer until the first few picks (shorter period picks) are matched well (Figure 54a). Change the velocity and the interface depth.
8. Also increase the size of the 'Dispersion Curve' window by dragging the right-lower corner. This will increase the spacing between the uploaded (red squares) picks, enabling them to be fit better.

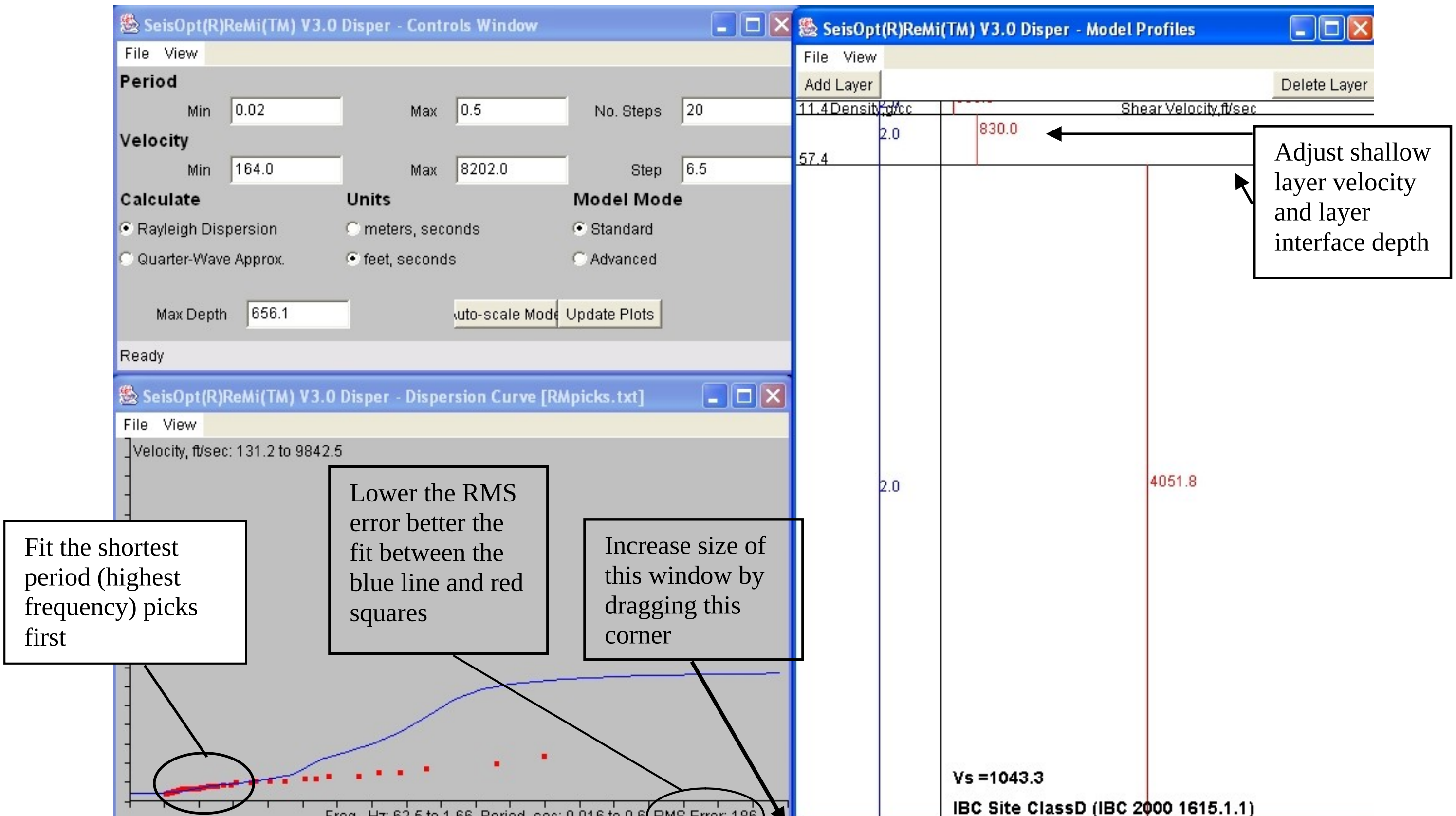


Figure 54a: The first step is to model the shorter-period picks. Adjust shallow layer velocity and interface depth so that the blue line matches the high-frequency picks (short period) red squares.

9. Note that you can fit as few or as many of the picks in one step as you wish. Going for too few would result in many layers, while too many may result in too few layers. Try to model about one-third to one-fourth of the picks at one step, starting from the shortest period. You can fine-tune the fits later.
10. Once the shorter period (higher frequency) picks are matched, then change the velocities an interface of the second layer until a few middle-period picks at the higher periods (lower frequencies) are matched. Figure 54b shows all three analyses windows from this step.
11. Repeat this process until all the picks are matched (Figure 54c). Usually the last few, longer period, picks are least constrained, since the sample the deepest part of the model. There is bound to be trade-off between the layer velocity and the interface depth, especially for the deeper layers.

12. Once you are satisfied with the picks, add a layer to the bottom of the model. Adjust the velocity of the deepest layer and the one above it and see if you can improve the picks.

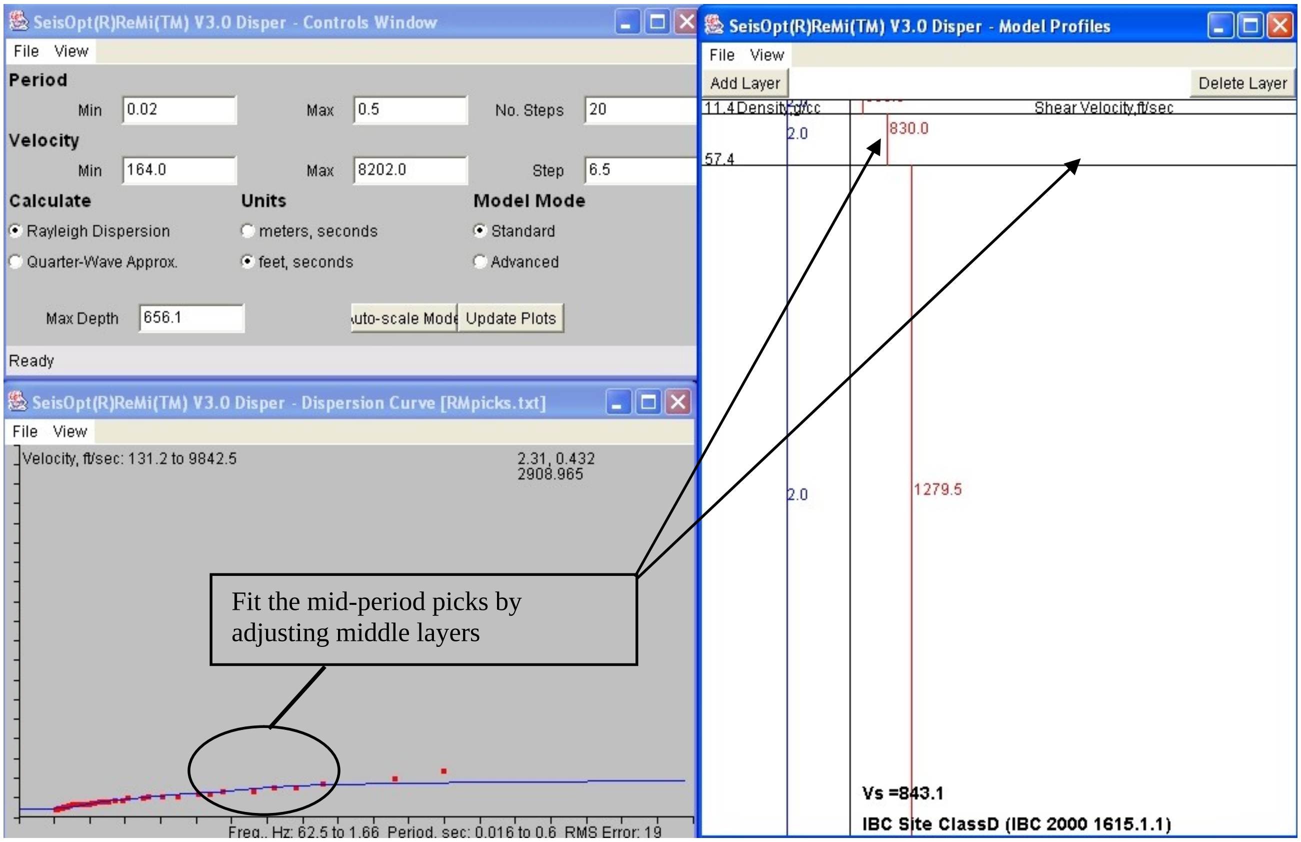


Figure 54b: The next step is try fitting the middle-period picks. Adjust the middle layer velocity and interface depth to achieve this.

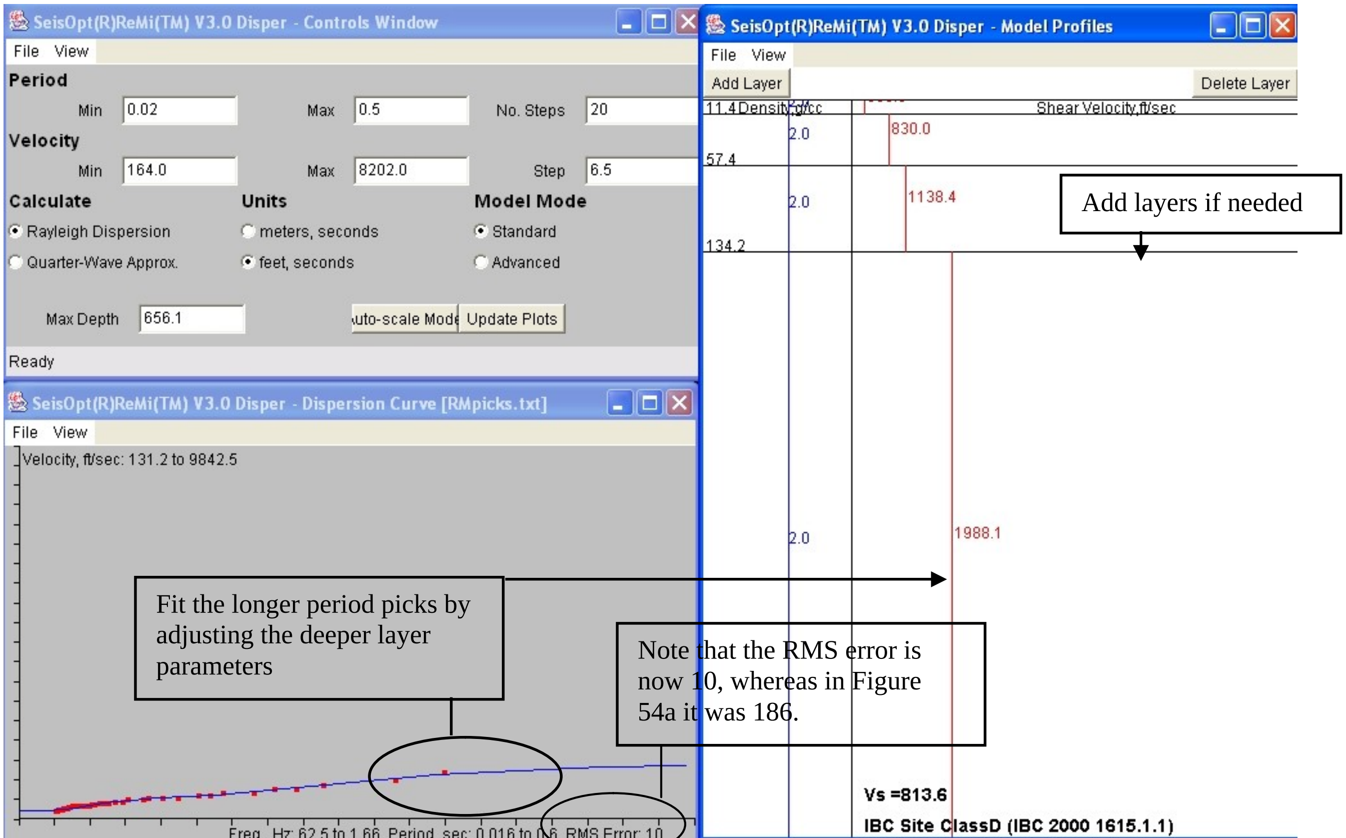


Figure 54c: Lastly fit the longer-period picks by adjusting the deeper layers

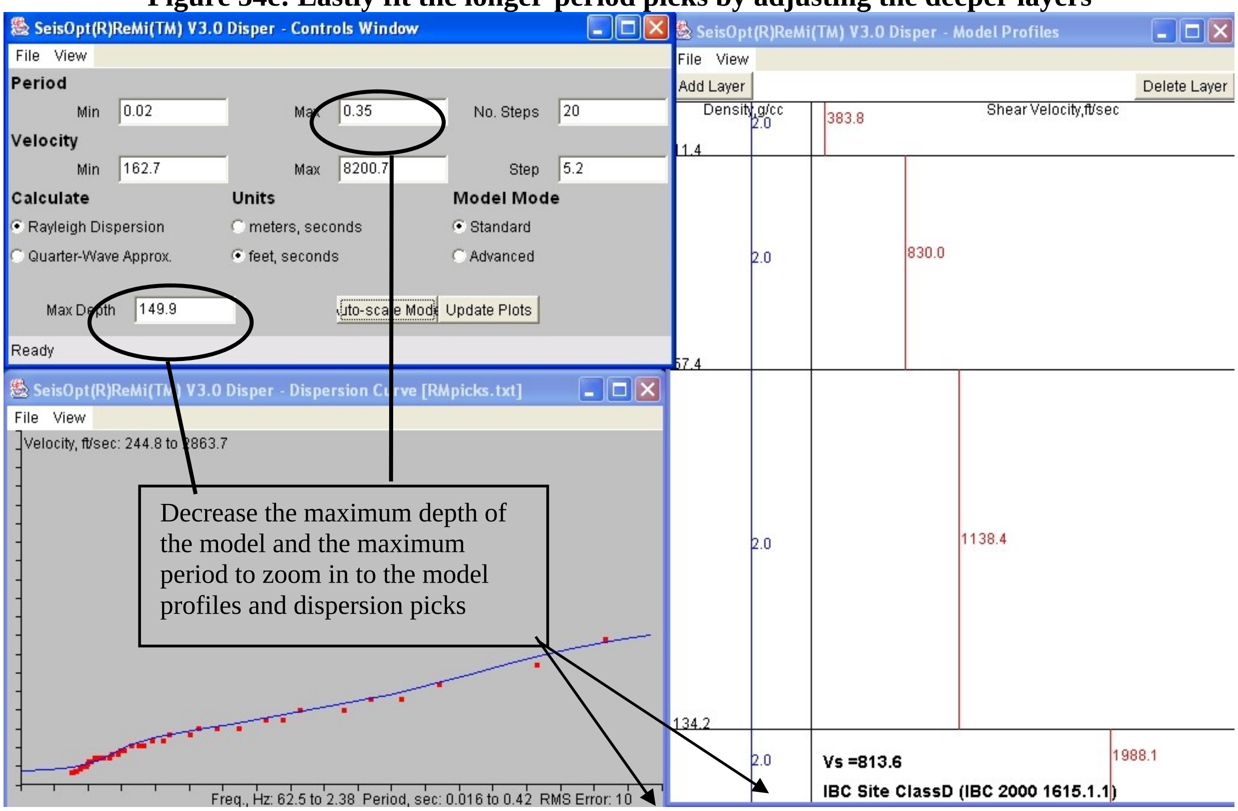


Figure 54d: Decrease the "Max Depth" parameter and the maximum velocity value so that the model profile is magnified in the 'Model Profiles' window. The maximum period can also be adjusted to match the data available.

13. Once all the picks are fit, that is, when the error is as low as possible, the next step is to fine-tune the model.
14. Set the Period Min and Max bounds to values outside the range of periods in your picks. Move the cursor to the pick location to find out these values. For the dispersion picks shown in Figure 54c these might be 0.02 to 0.35 seconds.
15. Set the Max Depth to a lower value, if needed. In this case it was set at 150 feet.
16. Click on 'Auto-scale' model to automatically scale the 'Dispersion Curve' and 'Model Profiles' window.
17. Click on the ''Update Plots'' button to register these changes. Figure 54d shows the updated windows.
18. Click and drag the lower-right corner of the 'Dispersion Curve' window to enlarge it, if necessary.
19. If you wish to fine tune the shallow layers and need to zoom in, decrease the 'Max Depth' and the 'Max Period' parameters. This will display the shallow layers in greater detail (Figure 54e) and adjust them to fit the high-frequency (low period) picks better, thus decreasing the overall RMS error.

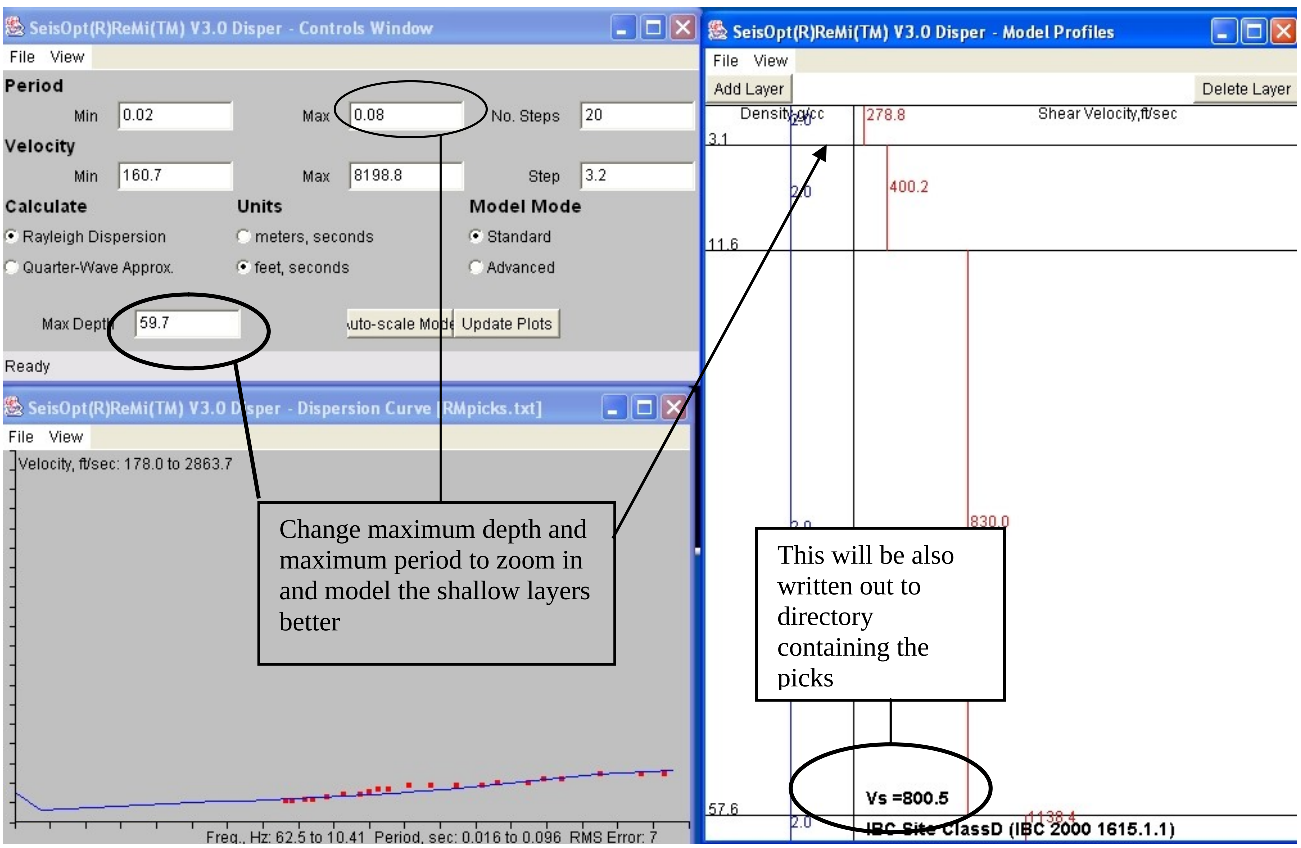


Figure 54e: Decrease 'Max Depth' in 'Controls Window' to zoom in to the shallow layers in order to fine tune them.

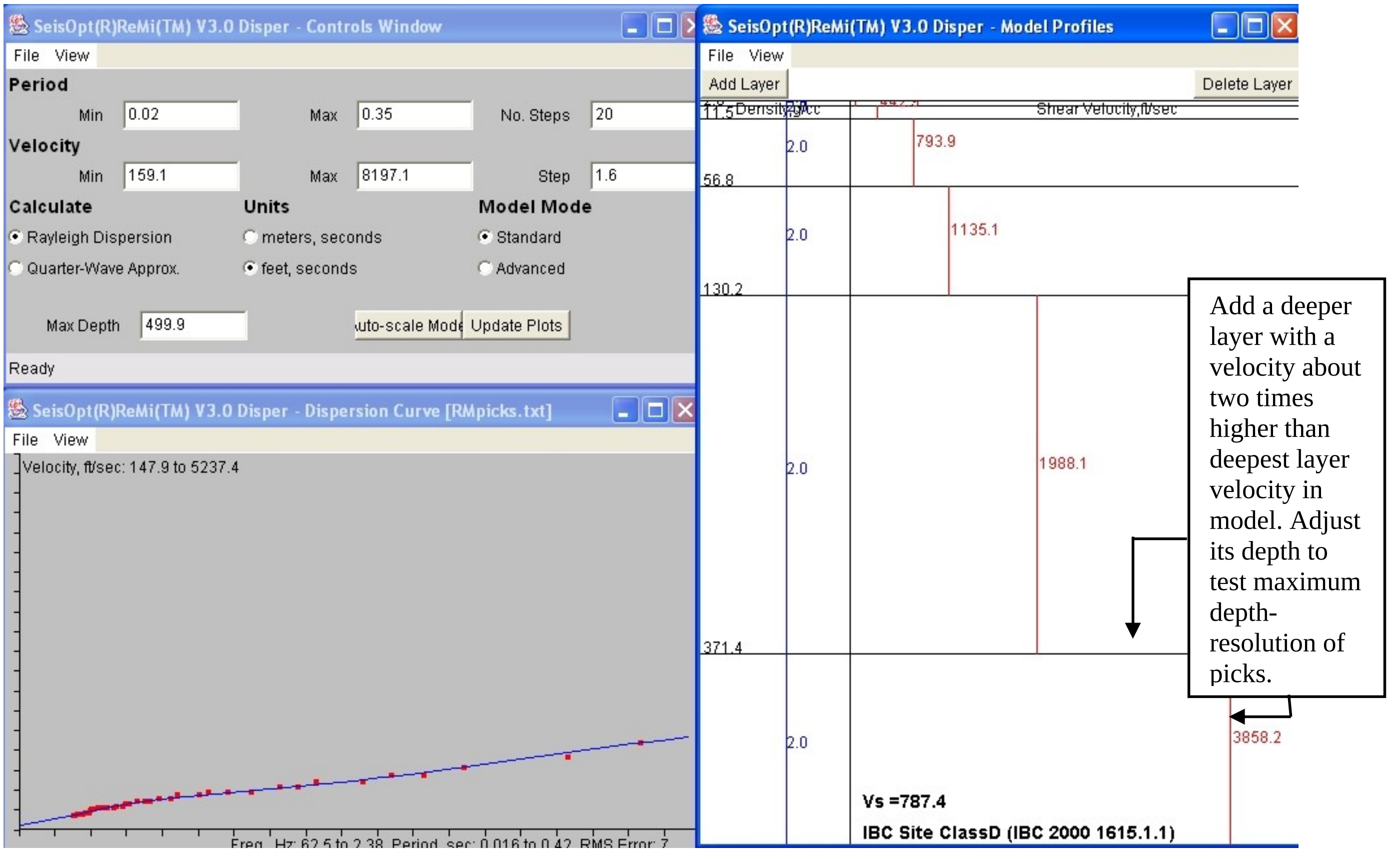


Figure 54f: Once satisfied with fits, add a deeper layer with a velocity about two times greater than the layer above it and adjust to see the maximum depth of resolution provided by the dispersion picks.

20. Unless you have good evidence for a velocity inversion, try not to introduce one. The governing criteria while modeling the picks should be that the simplest model that fits the data well is the best estimate of the average shear wave velocity.
21. As you adjust depths and velocities, note that there may be a range of these parameters in each layer that seem to fit the dispersion data equally well. These become the indeterminacies of your data. There is trade-off between the thickness and velocity of for every layer. Try to find the two end-member combinations of large thickness and high velocity, and small thickness and low velocity, that both fit the data equally well. This will give you limits or bounds of the velocity model.
22. To determined the maximum depth to which the dispersion picks can resolve the velocities do the following test: Add a layer with a velocity about two times greater than the deepest layer velocity. Now vary its depth slowly to see how it affects the picks. Determine the depth at which it stops affecting the picks. Now take the average of this depth and the depth of layer above. This is approximately the deepest the picks can resolve shear-wave velocities to.
23. The 100 ft average shear-wave velocity (V_s), along with the IBC 2000 (International Building Code) soil classification, is automatically calculated and displayed at the bottom of the ‘Model Profiles’ window. This is also written out to a file called “[name of the model file]_ibc.txt. If for example the model file is “mod.txt”, the IBC value will be written out to a file called “mod.txt_ibc.txt” in the same directory to which the model file was written.
24. Keep in mind that SeisOpt ReMi calculates the average one-dimensional shear wave velocities, so the user does not have to fit every pick. The objective is to fit the general trend of the picks, with the blue line passing through the center of the picks, resulting in an average shear-wave velocity profile. It is best to introduce velocity reversals only if you have other

evidence suggesting its presence or if quite a few p-f images (velocity spectra, Figure 55) show velocity reversal signatures.

25. Figure 55 shows an example velocity spectrum (p-f image) that shows a velocity reversal. Figure 55a and b show the picks and the model profiles that fit the dispersion picks. This data set shows a clear ‘‘kink’’ in the picked dispersion values (red squares) that is indicative of the presence of a high velocity layer and thus it is safe to model it as a velocity reversal.

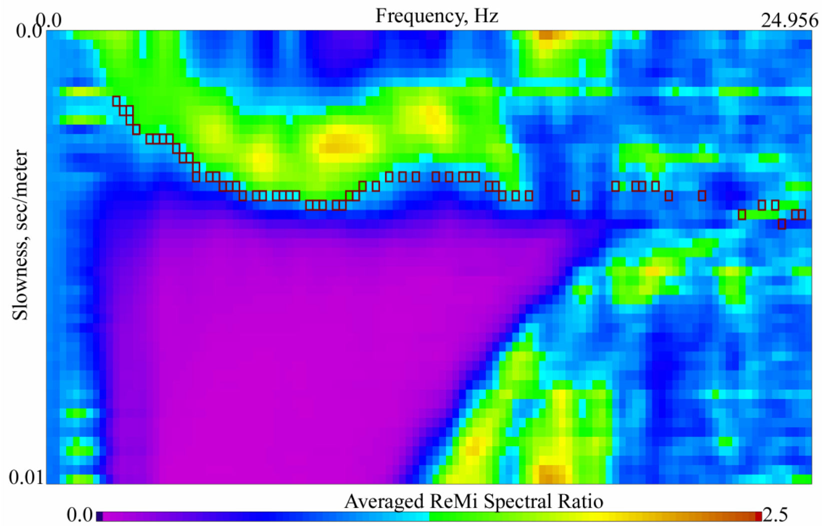


Figure 55: Example p-f image showing the dispersion picks for a model with velocity reversal.

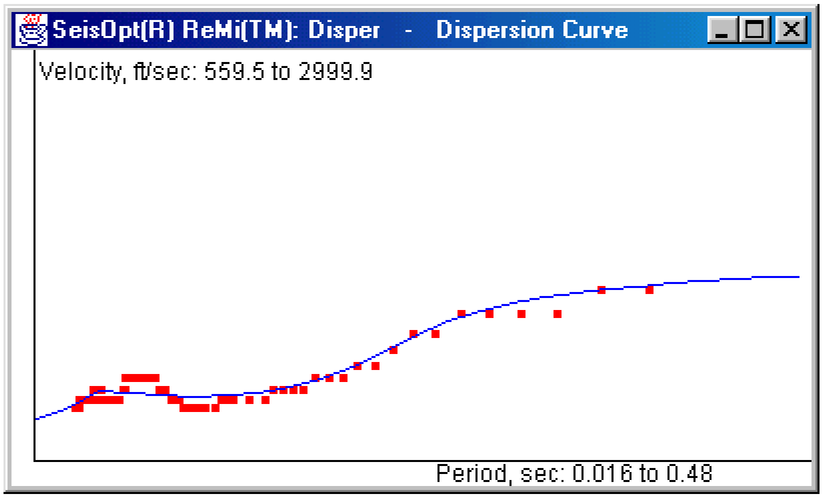


Figure 55a: Dispersion picks (red squares) made from the p-f image shown in Figure 55 and the fits (blue line) produced by the model shown in Figure 55b.

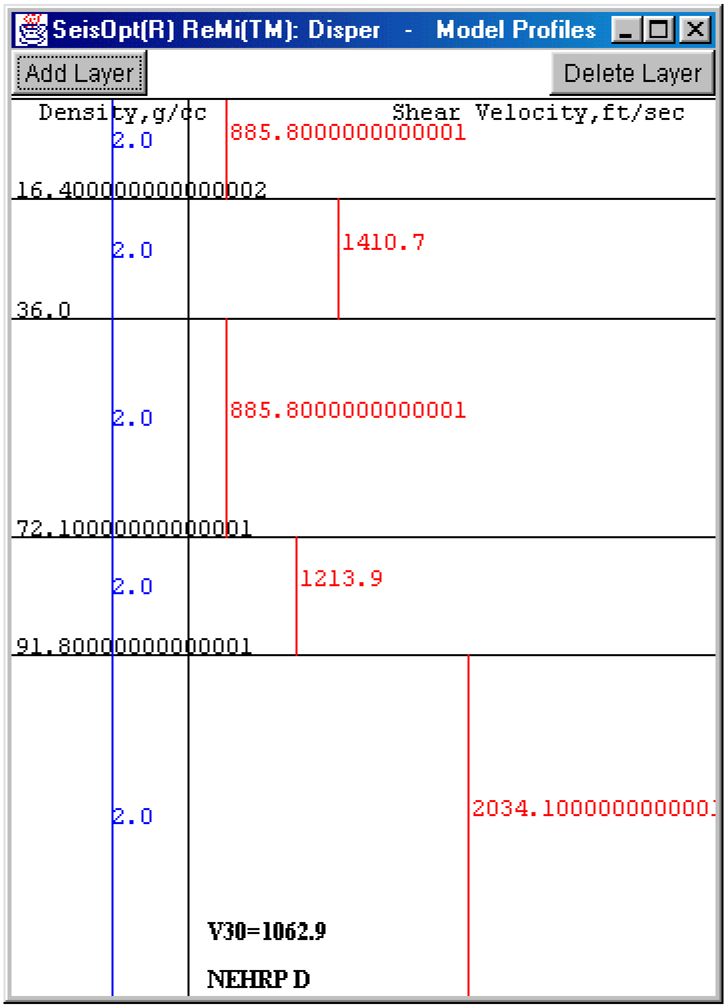


Figure 55b: Shear-wave velocity model with velocity reversal obtained by fitting the picks shown in Figure 55a.

4.2.1 Using the "Advanced" Model Mode

This version of ReMi allows the user to use P-wave velocity information, if available, to improve the accuracy of the modeled shear-wave profile.

- In the "Standard" model mode, SeisOpt ReMi Disper assumes that the Vp/Vs ratio is 1.73. This is may not be necessarily true for all soil types. For example very soft clays are known to have Vp/Vs ratios of over 5.
- If P-wave velocities for the site are known (from refraction surveys, logs, etc) then they can be used to model the shear-wave velocities better.
- To do so, select the "Advanced" model mode in the 'Controls Window' (Figure 56). The P-wave velocities for will be displayed as green lines.
- Click on "Auto-Fit Model" in the 'Controls' window to adjust the 'Model Profile' window so all the layer velocities are visible. As mentioned above, the default P-wave velocities have a Vp/Vs ration of 1.73.

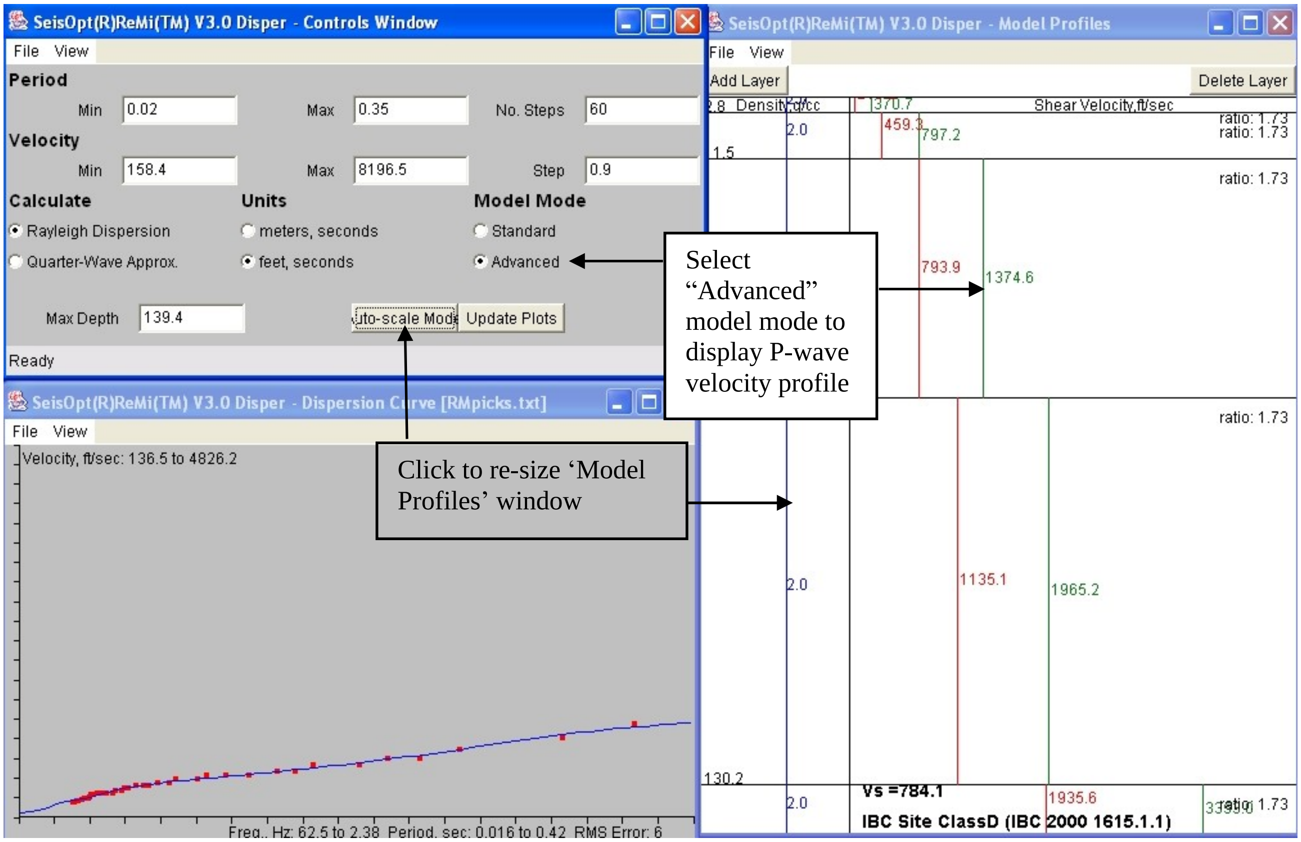


Figure 56: Select the “Advanced” model mode in the ‘Controls’ window to model the shear-wave velocity profile with P-wave velocity constraints.

- Now the P-wave velocity can be adjusted independent of the shear-wave velocity by holding down the SHIFT key and moving the green lines to the appropriate values for each layer using the left mouse button.
- You will notice that the fits, that is, the blue curve in the ‘Dispersion’ window, will change for Vp/Vs ratios much different from 1.73.
- Keeping the P-wave velocities fixed (since they are known) adjust the shear-wave velocities so that the fit between the red squares and blue line is again satisfactory.
- One may also have to add more layers to account for the variations. Figure 57 shows a modified shear-wave profile, assuming P-wave velocities. Note that this is just for demo purposes – Vp/Vs ratios may not be as high as shown.
- Vp/Vs ratios can be used for determining Poisson’s ratio and other soil properties.

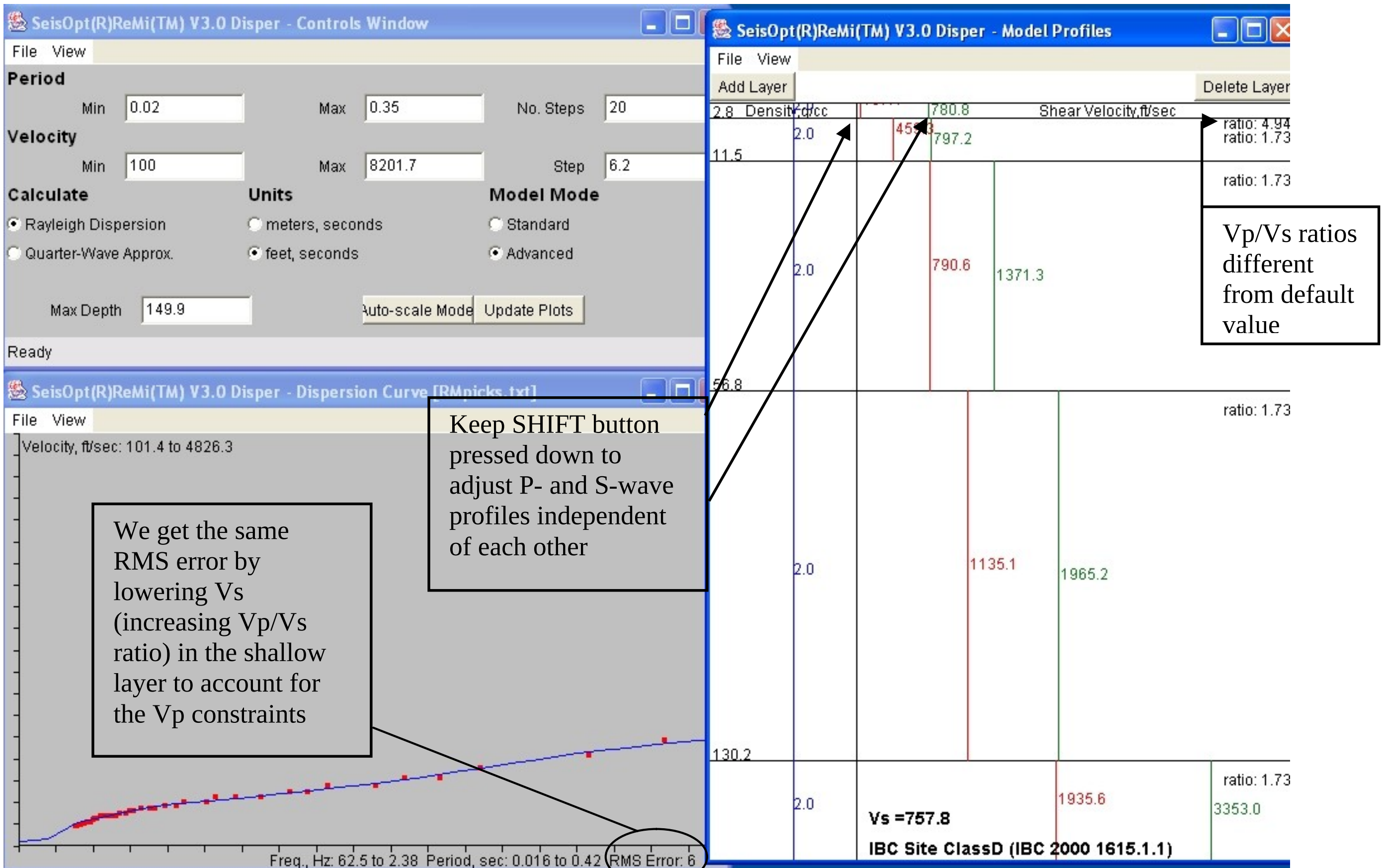


Figure 57: The shear-wave velocities need to adjusted to fit the picks based on the P-wave velocity information.

4.3 Exporting the Model Profiles and Modeled Dispersion Curve

Once the model has been derived, the next step is to save it to disk. To do so, choose the ‘‘Save Model File’’ option from the ‘‘File’’ menu (Figure 58). A dialog window opens up (Figure 59) where the file name for the model can be entered. Using a ‘.txt’ extension for the model file will make it easy for it to be opened using a file editor like Notepad or Wordpad.

The saved model can be read back into the ReMi Disper module, by using the ‘‘Open Model File’’ under the ‘‘File’’ menu (Figure 60). The model will be rendered in the ‘Model Profiles’ window. Make sure you select the desired ‘Units’ parameter when saving the model. The model file output by the ReMi Disper module writes depth and velocities in the selected units on the ‘Controls’ window. The first line in the saved model file indicates whether it is in units of “m,s” or “ft,s”. The first column in the file is the depth of the layer, the second column is the density (in g/cc), third column are the P-wave velocities, and the fourth column is the shear-wave velocity of the layers. If the “Standard” model mode is used, the Vp values written out will be zero. The actual values will be output when the “Advanced” model mode is selected. When the model is saved, the IBC soil classification for the profile is also written out to a file. The file will have use the name of the model file as prefix follow by the suffix “ibc.txt”. So if the model file name is “RMmodel.txt” then the IBC soil classification will be written out to a file called “RMmodel.txt_ibc.txt”. This will be written out to the directory that contains the dispersion picks.

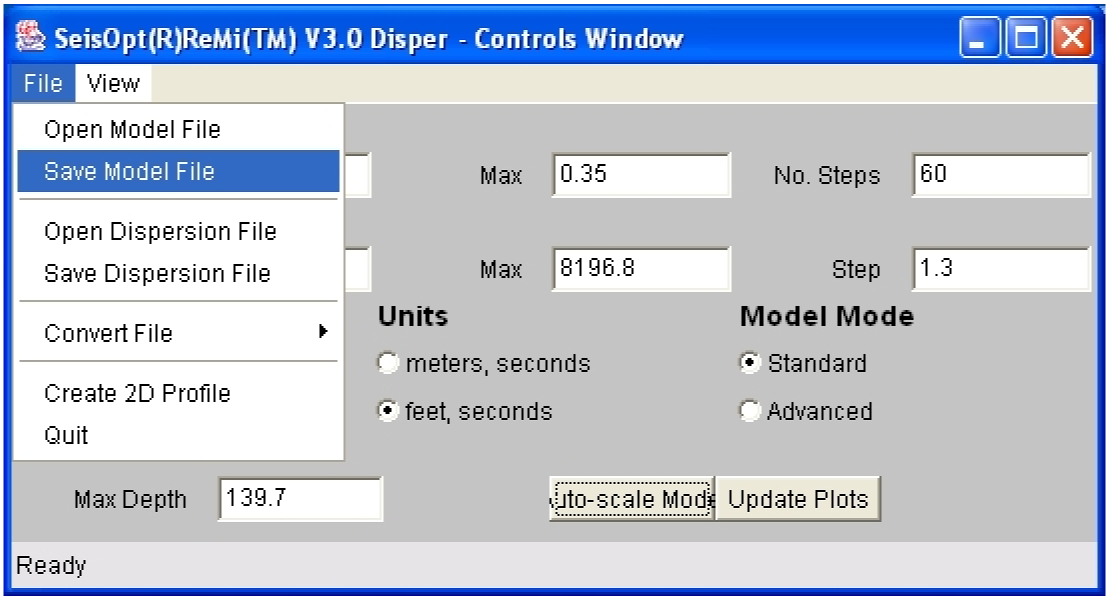


Figure 58: Choose the ''Save Model File'' to save the one-dimensional velocity model to disk.

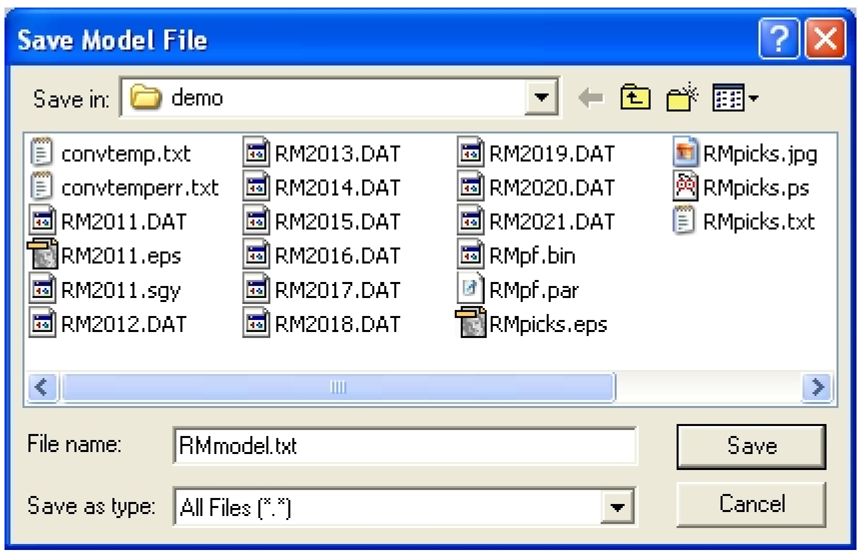


Figure 59: Enter the file name for the model to be saved to. The IBC soil classification is written out to a file with a suffix "ibc.txt" to the directory that contains the dispersion picks.

One can also save the modeled (calculated) dispersion curve (blue line in the 'Dispersion Curve' window) to a file (Figure 61). This is useful if you wish to plot the picked dispersion curve against the calculated dispersion pick using MS-Excel (see Section 4.5). Note that if you increase the number of steps ('No. Steps') parameter in the 'Controls window' from the default 20 to 60 and click ''Update Plots'', then more dispersion points will be calculated, resulting in a smoother calculated dispersion curve (blue line).

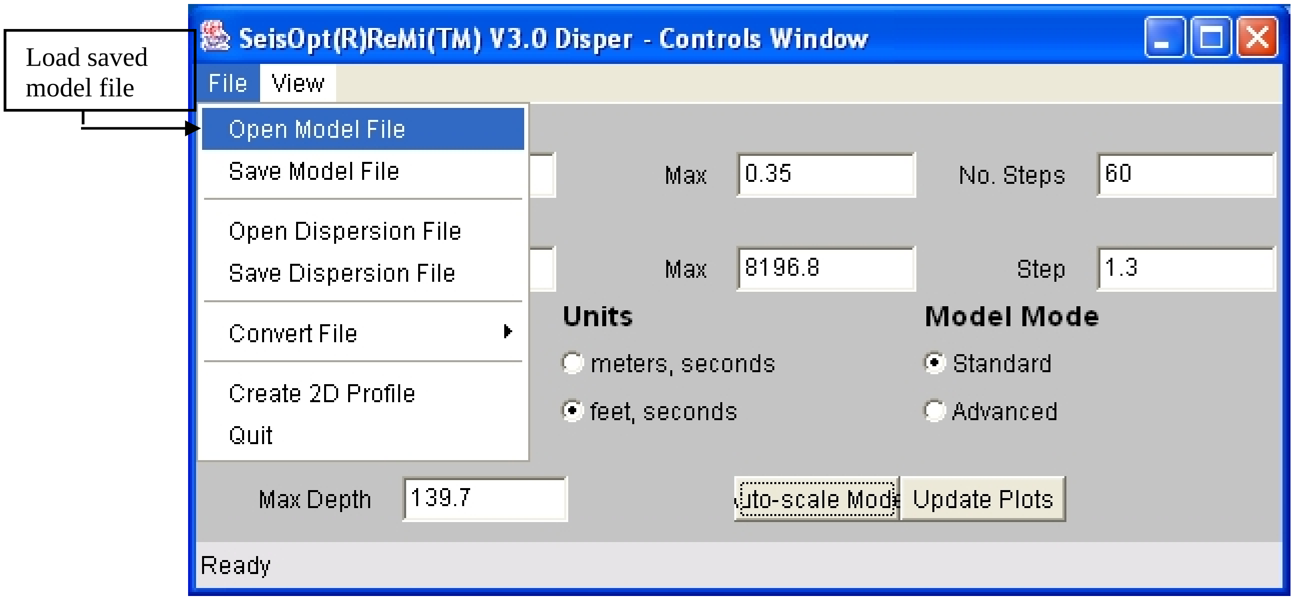


Figure 60: The saved model can be imported into ReMi Disper by using the ''Open Model File'' option under the ''File'' menu.

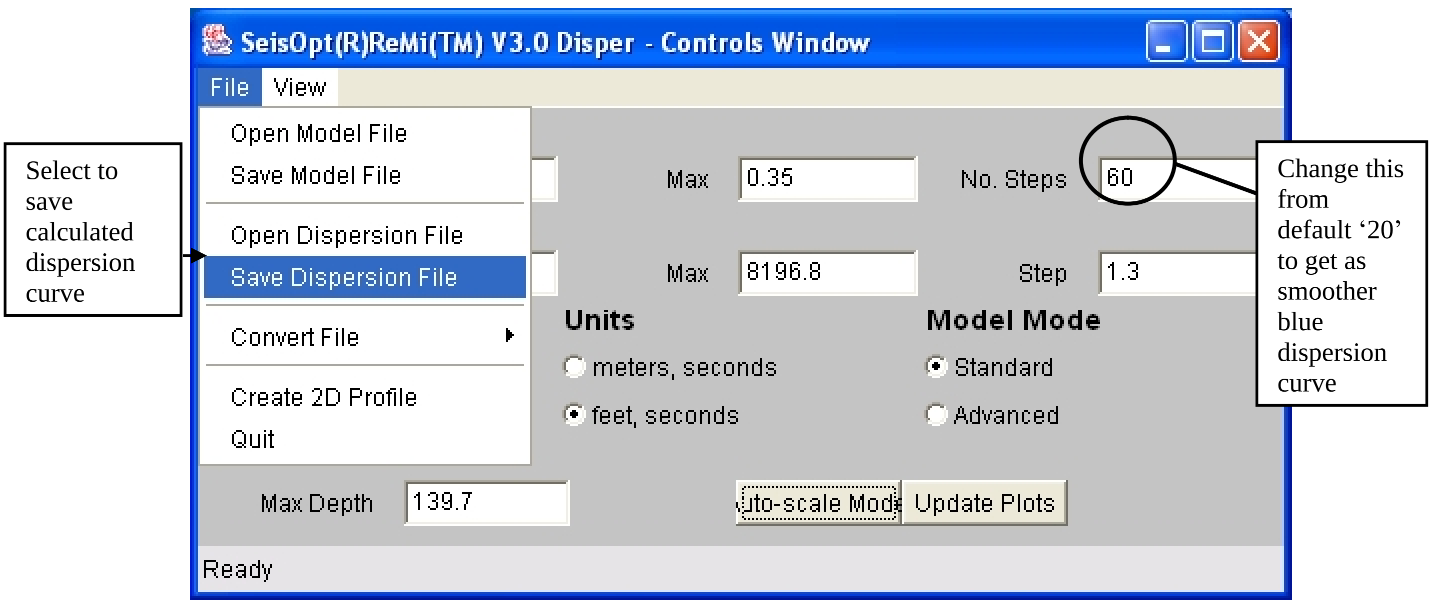


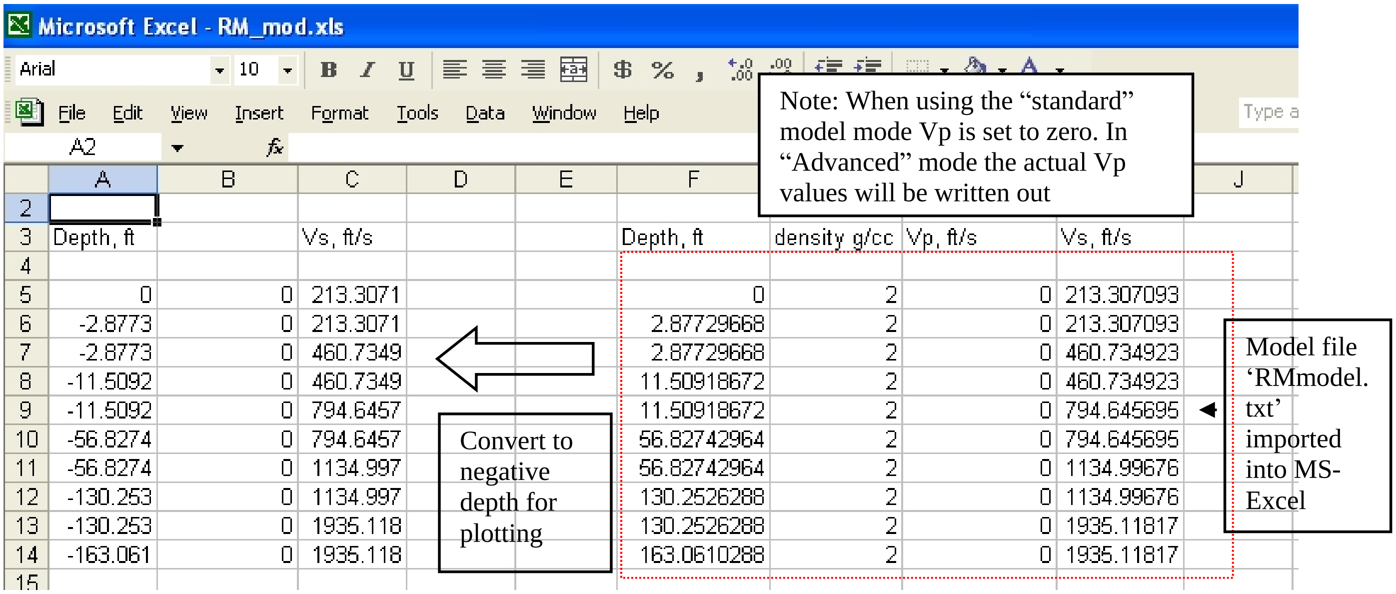
Figure 61: Use the ''Save Dispersion File'' option to save the calculated dispersion picks (blue line in the 'Dispersion Curve' window.

4.4 Making Report Quality Output from the ReMi Disper Module

Once the model and dispersion files are saved to a local disk, they can be imported into a spreadsheet program like MS-Excel to plot the profile and dispersion fits, respectively.

4.4.1 Plotting the Model Profile

The sample spreadsheet (RM_mod.xls) included in the 'demo' sub-directory in the SeisOpt ReMi installation directory (C:\Optim\ReMiv30) (Figure 62) shows the model file after it has been imported into Excel. The first columns, which are the depths to the layer interfaces, have been modified to be negative, so the plots are rendered correctly. Next, use the graph module of MS-Excel to plot the profile shown in Figure 63. Make sure to use ‘space’ delimit option when cutting and pasting the ‘RMmodel.txt’ (saved model file) into the spreadsheet.



	A	B	C	D	E	F	G	H	I
2									
3	Depth, ft		Vs, ft/s			Depth, ft	density g/cc	Vp, ft/s	Vs, ft/s
4									
5	0	0	213.3071			0	2	0	213.307093
6	-2.8773	0	213.3071			2.87729668	2	0	213.307093
7	-2.8773	0	460.7349			2.87729668	2	0	460.734923
8	-11.5092	0	460.7349			11.50918672	2	0	460.734923
9	-11.5092	0	794.6457			11.50918672	2	0	794.645695
10	-56.8274	0	794.6457			56.82742964	2	0	794.645695
11	-56.8274	0	1134.997			56.82742964	2	0	1134.99676
12	-130.253	0	1134.997			130.2526288	2	0	1134.99676
13	-130.253	0	1935.118			130.2526288	2	0	1935.11817
14	-163.061	0	1935.118			163.0610288	2	0	1935.11817

Figure 62: Import the saved model file into MS-Excel to plot the one-dimensional shear-wave profile. This file (RMmod.xls) is included in the ‘demo’ subdirectory.

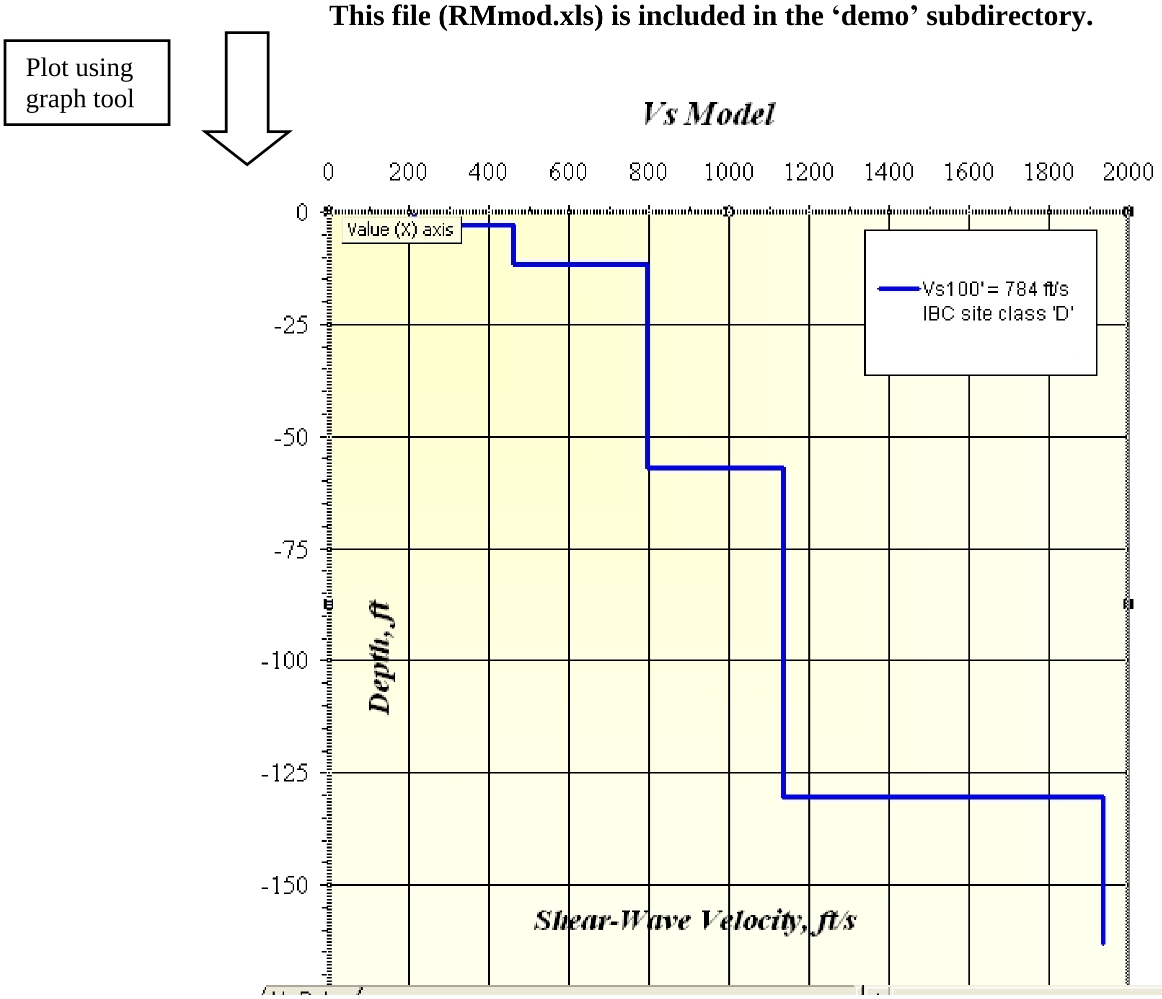


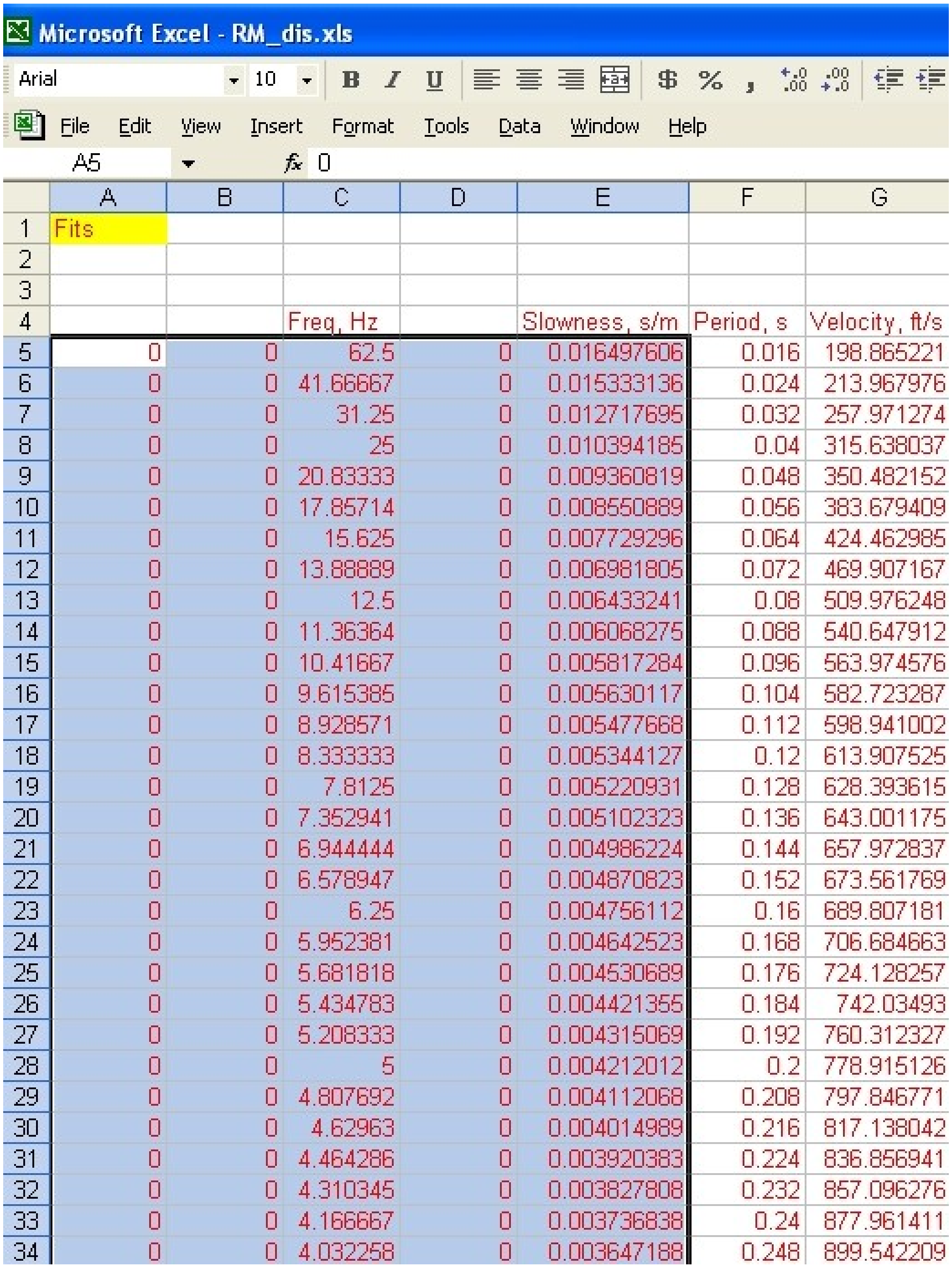
Figure 63: Plot of the one-dimensional shear-wave velocity created using MS-Excel. The spreadsheet is included in the ‘demo’ subdirectory.

4.4.2 Plotting the Dispersion Curve

In addition, one can plot the picked (red squares) versus calculated dispersion curve (blue lines) by importing the picked file (for the demo data, 'RMpicks.txt') and the saved dispersion data. The MS-Excel file 'RMcalcdis.xls' (Figures 64a and 64b) included in the ''demo'' subdirectory shows the imported picks (numbers in black) and saved dispersion (numbers in red) file. The picked and saved dispersion values are in frequency (Hz) and slowness (seconds/meter). Use the spreadsheet functions to convert it to desired values you wish to plot. That is, if you wish to plot frequency versus Rayleigh-wave phase velocity, then convert slowness (for example, column E in dis.xls) to velocity (for example, column G in the dis.xls) by taking the inverse of the output slowness values. Similarly, frequency can be converted to period by taking its inverse (Period = 1/Frequency).

Picks								
		Freq, Hz		Slowness, s/m			Period, s	Velocity, ft/s
0.94876	8	2.6041	6	0.0025	0	0	0.38401	1312.32
0.90262	9	2.9296	7	0.00291	0	0	0.341344	1127.42268
1.0012	11	3.5807	8	0.00333	0	0	0.279275	985.2252252
0.9479	12	3.90625	9	0.00375	0	0	0.256	874.88
0.99868	13	4.2317	9	0.00375	0	0	0.236312	874.88
0.85312	14	4.5572	10	0.00416	0	0	0.219433	788.6538462
1.0328	16	5.2083	10	0.00416	0	0	0.192001	788.6538462
0.81647	17	5.5338	11	0.00458	0	0	0.180708	716.3318777
0.89778	18	5.8593	11	0.00458	0	0	0.170669	716.3318777
0.98169	20	6.5104	12	0.005	0	0	0.1536	656.16
0.92425	22	7.1614	12	0.005	0	0	0.139638	656.16
0.92475	24	7.8125	12	0.005	0	0	0.128	656.16
0.7795	25	8.138	13	0.00541	0	0	0.12288	606.4325323
0.92697	28	9.1145	13	0.00541	0	0	0.109715	606.4325323
0.75402	29	9.4401	14	0.00583	0	0	0.105931	562.7444254
0.97099	31	10.091	14	0.00583	0	0	0.099098	562.7444254
0.71272	33	10.742	15	0.00624	0	0	0.093093	525.7692308
0.72067	34	11.067	15	0.00624	0	0	0.090359	525.7692308
0.88176	36	11.718	15	0.00624	0	0	0.085339	525.7692308
0.77826	38	12.369	16	0.00666	0	0	0.080847	492.6126126
0.92624	39	12.695	16	0.00666	0	0	0.078771	492.6126126
0.69511	40	13.02	17	0.00708	0	0	0.076805	463.3898305
0.85829	42	13.671	17	0.00708	0	0	0.073148	463.3898305
0.7411	43	13.997	18	0.0075	0	0	0.071444	437.44
0.88724	45	14.648	18	0.0075	0	0	0.068269	437.44
0.99235	47	15.299	18	0.0075	0	0	0.065364	437.44
1.1019	49	15.95	18	0.0075	0	0	0.062696	437.44
0.84157	51	16.601	19	0.00791	0	0	0.060237	414.7661188
0.80595	52	16.927	19	0.00791	0	0	0.059077	414.7661188
0.84704	53	17.252	20	0.00833	0	0	0.057964	393.8535414

Figure 64a: Import the dispersion picks file (RMpicks.txt in this case) into MS-Excel to plot the dispersion curve (Figure 65). The blue color filled columns are the imported picks. The 'Period' and 'Velocity' values are calculated.



	A	B	C	D	E	F	G
1	Fits						
2							
3							
4			Freq, Hz		Slowness, s/m	Period, s	Velocity, ft/s
5	0	0	62.5	0	0.016497606	0.016	198.865221
6	0	0	41.66667	0	0.015333136	0.024	213.967976
7	0	0	31.25	0	0.012717695	0.032	257.971274
8	0	0	25	0	0.010394185	0.04	315.638037
9	0	0	20.83333	0	0.009360819	0.048	350.482152
10	0	0	17.85714	0	0.008550889	0.056	383.679409
11	0	0	15.625	0	0.007729296	0.064	424.462985
12	0	0	13.88889	0	0.006981805	0.072	469.907167
13	0	0	12.5	0	0.006433241	0.08	509.976248
14	0	0	11.36364	0	0.006068275	0.088	540.647912
15	0	0	10.41667	0	0.005817284	0.096	563.974576
16	0	0	9.615385	0	0.005630117	0.104	582.723287
17	0	0	8.928571	0	0.005477668	0.112	598.941002
18	0	0	8.333333	0	0.005344127	0.12	613.907525
19	0	0	7.8125	0	0.005220931	0.128	628.393615
20	0	0	7.352941	0	0.005102323	0.136	643.001175
21	0	0	6.944444	0	0.004986224	0.144	657.972837
22	0	0	6.578947	0	0.004870823	0.152	673.561769
23	0	0	6.25	0	0.004756112	0.16	689.807181
24	0	0	5.952381	0	0.004642523	0.168	706.684663
25	0	0	5.681818	0	0.004530689	0.176	724.128257
26	0	0	5.434783	0	0.004421355	0.184	742.03493
27	0	0	5.208333	0	0.004315069	0.192	760.312327
28	0	0	5	0	0.004212012	0.2	778.915126
29	0	0	4.807692	0	0.004112068	0.208	797.846771
30	0	0	4.62963	0	0.004014989	0.216	817.138042
31	0	0	4.464286	0	0.003920383	0.224	836.856941
32	0	0	4.310345	0	0.003827808	0.232	857.096276
33	0	0	4.166667	0	0.003736838	0.24	877.961411
34	0	0	4.032258	0	0.003647188	0.248	899.542209

Figure 64a: Import the calculated dispersion file (RMcalcdis.txt in this case) into MS-Excel to plot the dispersion curve (Figure 65). The blue color filled columns are the imported picks. The 'Period' and 'Velocity' values are calculated.

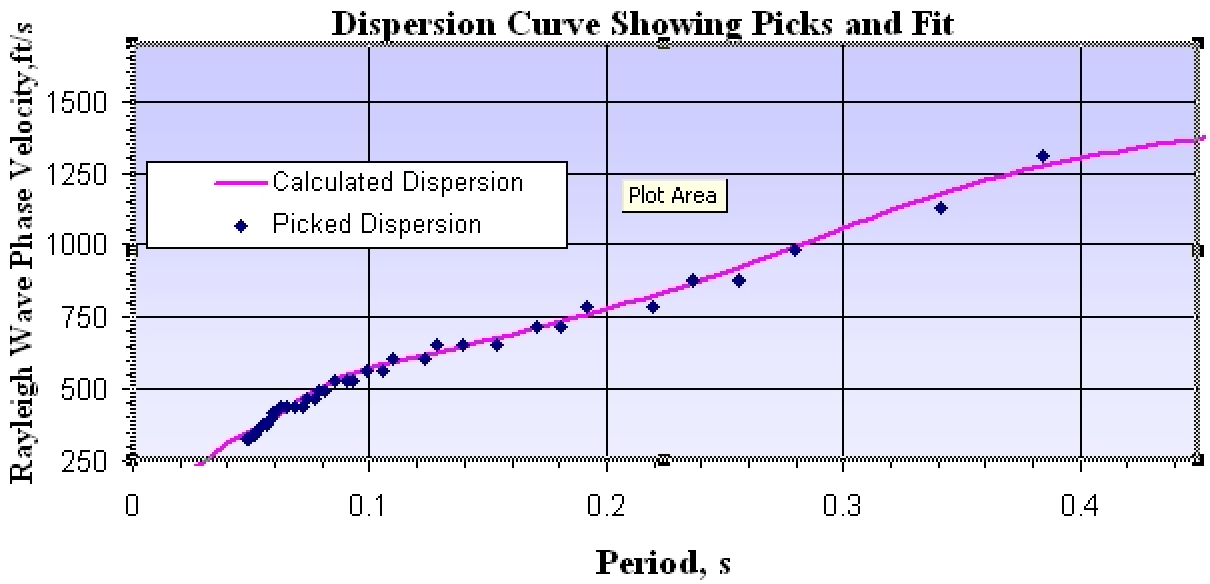


Figure 65: Dispersion curve plot created using the graph module of MS-Excel. The example file 'RM_dis.xls' included in the 'demo' directory produces this plot.

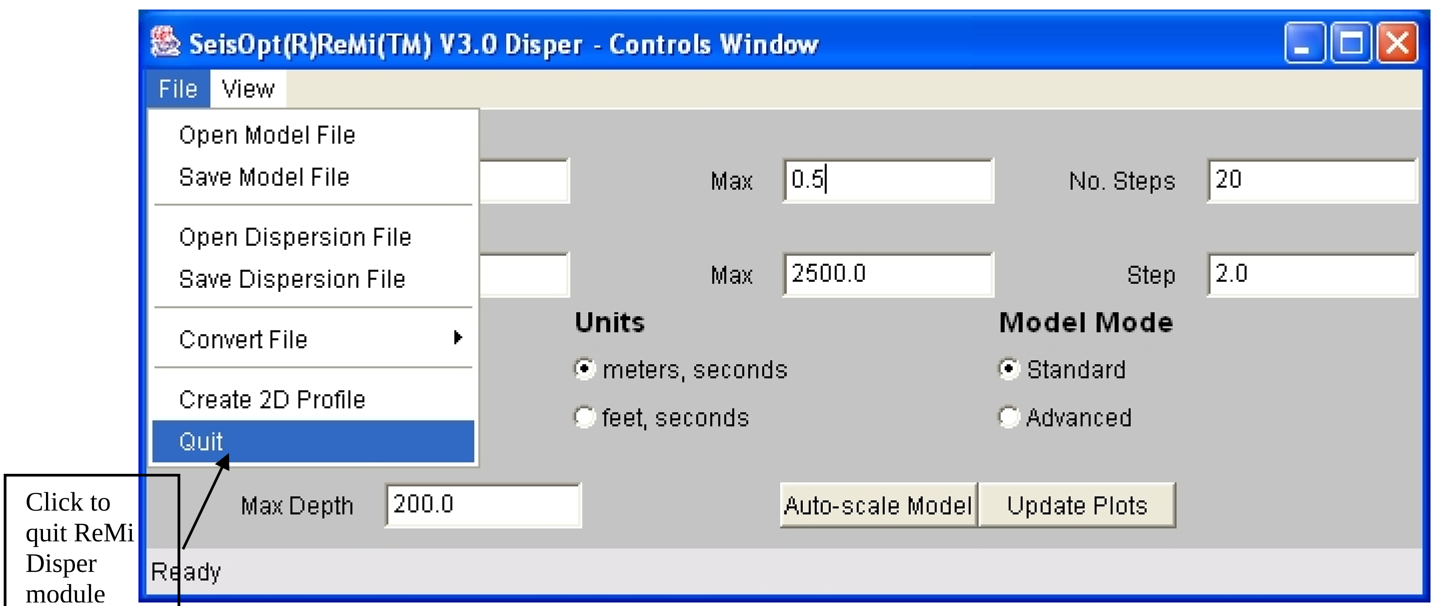


Figure 66: Click on 'Quit' to exit the ReMi Disper module.

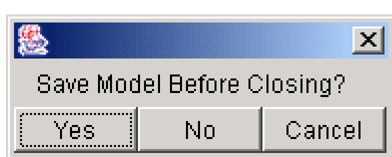


Figure 67: Click "Yes" to save model file or "No" to quit without saving. Click "Cancel" to continue using Disper.

Use the 'Quit' button under the 'File' menu (Figure 66) to quit the ReMi Disper module. The dialog window shown in Figure 67 will open up. If you have not saved the model, click "Yes" and enter a model file name. Click "Cancel" if you wish to continue using ReMi Disper or "No" to quit without saving the model file.

5.0 Creating 2D Velocity Models Using SeisOpt ReMi

The new version of SeisOpt ReMi allows the user to create 2D cross-section by analyzing traces recorded on groups of geophones deployed along the array. The software comes with the capability that allows the user to perform the "Vspect" analysis on groups of traces (for example, trace 1-6, 3-8,…and so on) and derive a 1D model for each group. Then the 1D models can be put together to create a 2D cross-section.

Creating a 2D cross-section will allow the user to image the lateral variations in the shear-wave velocity. It can be very useful to map isolated low-velocity zones, get constraints on depth to higher-velocity layers (for example, bedrock), map velocity discontinuities, like faults, and in general get constraints on velocity model obtained from standard refraction survey. Since ReMi can be done in noisy urban areas, the 2D image allows the user to get some constraints on velocities in areas where the high noise level would render a standard refraction survey useless.

5.1 Importing and processing data for 2D analysis

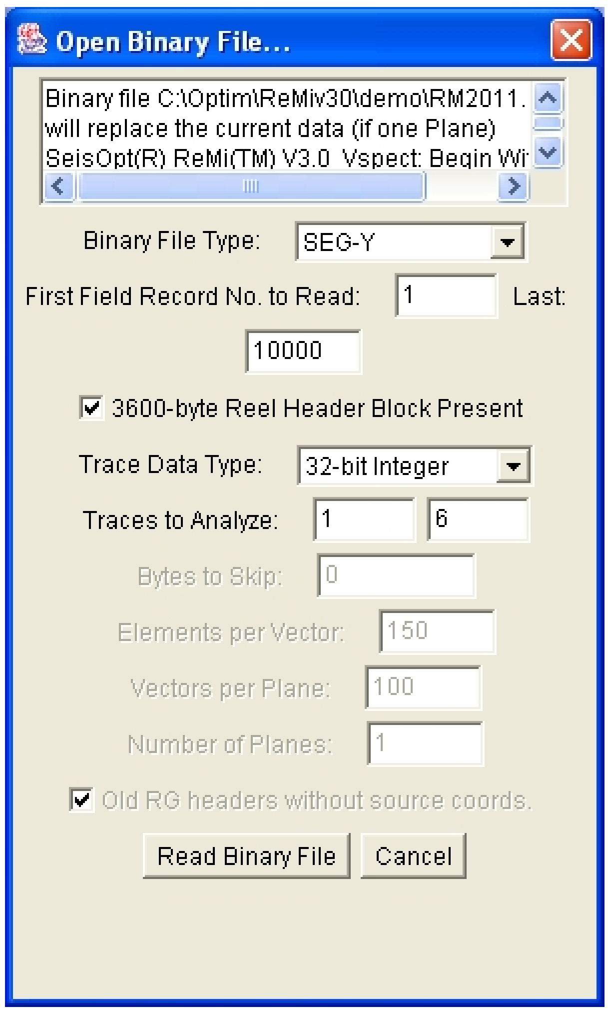


Figure 68: Import traces 1-6 for 2D analysis

The first step is to import data as described in Section 3.1.1 and 3.1.2. The objective to analyze groups of traces (channels) instead of processing data recorded at all the geophones at the same time. That is, only selected traces are imported and analyzed. In this example, the field data has 36 channels. So for the 2D analysis, traces 1-6, 3-8, 5-10, 7-12, 9-14, 11-16, 13-18, 15-20, 17-22, 19-24, 21-26, 23-28, 25-30, 27-32, 29-34, 31-36 are analyzed. A 1D model is derived for each group of traces. Then the 1D models are put together to obtain a 2D cross-section. It is recommended that at least 6 traces be used for analysis. The overlap of 70% of traces (4 traces in this case) gives the user a better control on the lateral velocity variation. If the subsurface velocities are expected to be layer-cake, with little or no lateral variation, then more traces at a time and less overlap can be used for analysis. One consequence of using fewer traces is that the maximum offset is reduced. As a result the lower-frequencies in the pf image (Figure 70b) are not well defined. This, in turn, causes the depth of the velocity models obtained to be much less than when using all the traces. But, it is important to remember that most of the velocity variations do occur at shallow depths. So, the 2D image created will map the velocity variations were they occur the most.

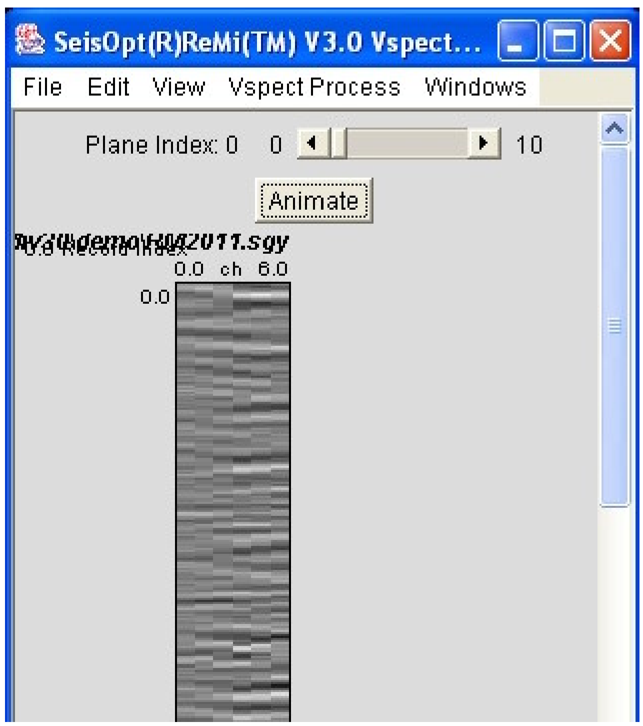


Figure 69: Traces 1-6 are read in and displayed

As shown in Figure 68, traces 1-6 are read in. The first six traces are displayed in Figure 69. Next, use the "ReMiVspect" module and follow the steps described in Sections 3.2, 3.3, 3.4, 3.5 and 3.6 to create a pf image and pick the dispersion curve (Figures 70a and 70b). When performing the "Vspect" processing on fewer traces, it is best to use the same parameters as those used when processing all the traces.

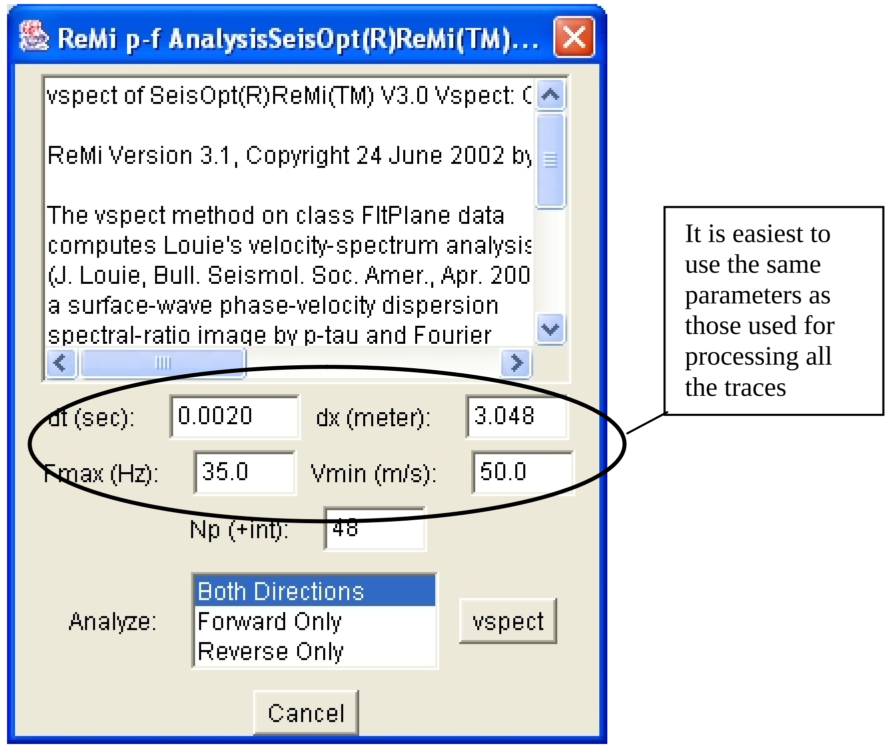


Figure 70a: Process the traces using the "ReMiVspect" module. It is easiest to use the same parameters used to process all the traces when processing groups of traces

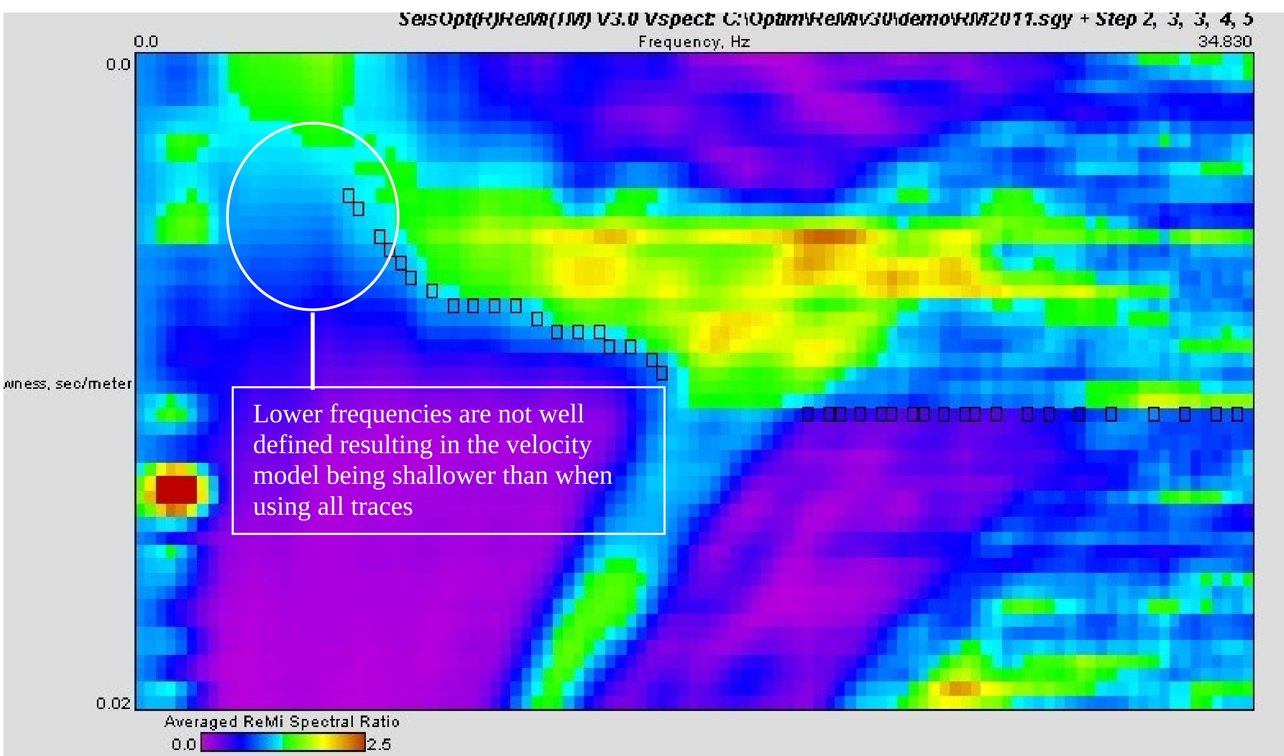


Figure 70b: pf image showing the picks for traces 1-6. Note that the fewer traces used causes the dispersion curve to be less well defined at the lower frequencies, consequently limiting the depth of the velocity model.

Next follow the steps outlines in Section 4.2 to generate a 1D model using the "ReMiDisper" module. The best procedure to model fewer traces is to start of with the 1D model generated using all the traces (used "Open Model" to read in the saved 1D model) and then adjust the layer velocities and layer interface depths until the model fits the picks (minimum RMS error). Since fewer traces are being used the maximum depth of the model will be less than the depth of the model obtained using all the traces. Consequently, as in this example, the deeper layer will probably not imaged (Figure 71).

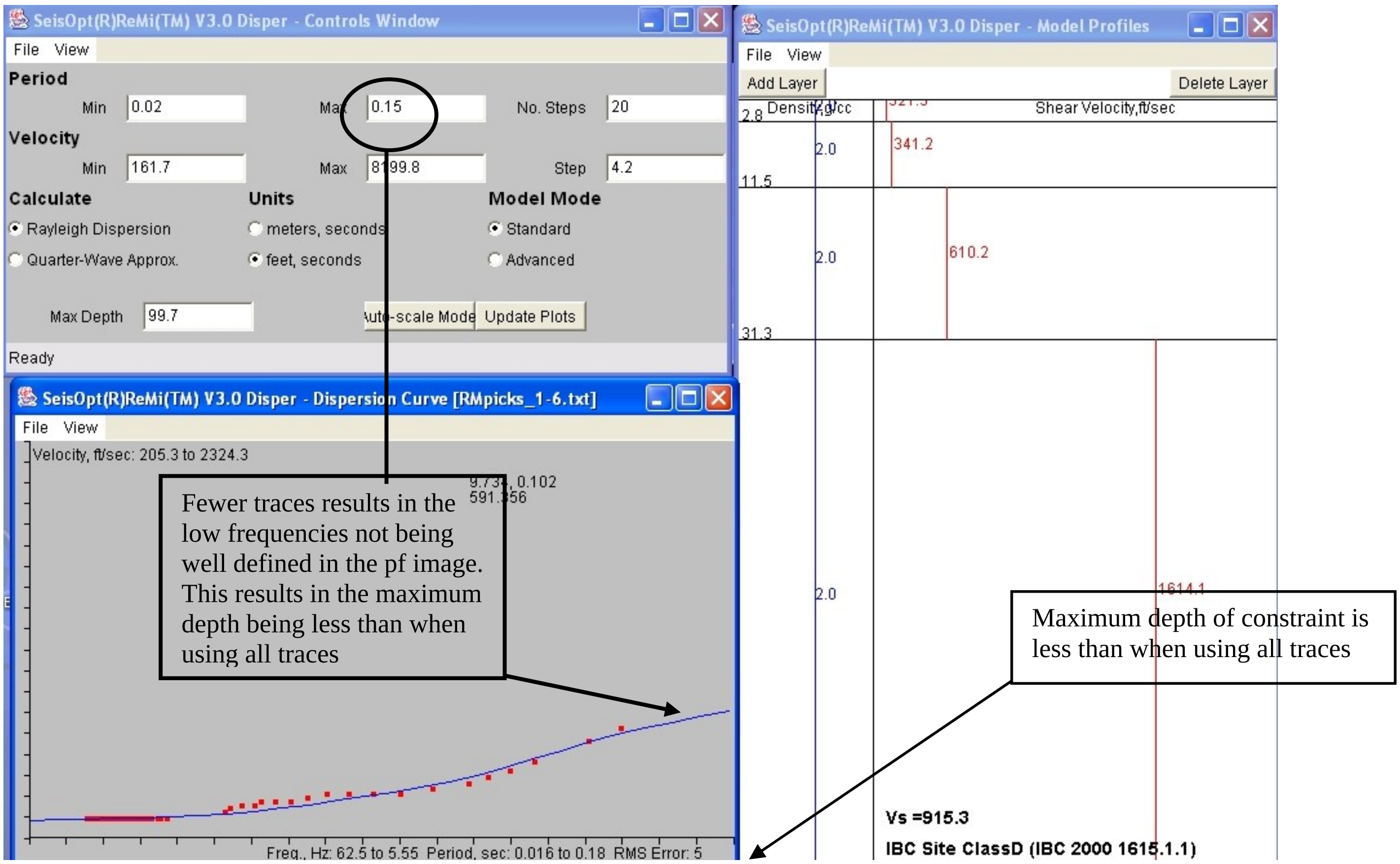


Figure 71: Picks are modeled using "ReMiDisper" module. Note that the maximum depth of constraint is now much less (about 60 feet) compared to when all traces are used (about 150 feet). Consequently the deepest layer is not imaged.

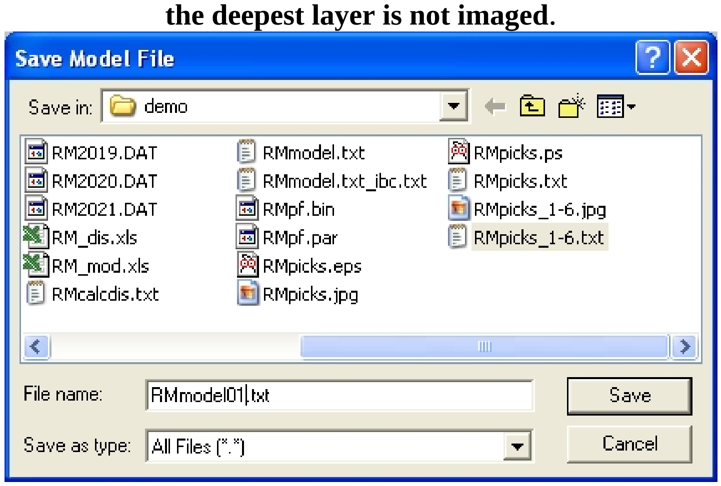


Figure 72: Save the model to a file with numeric component. In this case, it is save to file 'RMmodel01.txt'.

The next step is to save the model to a file that is named a certain way. It should have a numeric part to it, preferably to the end before the extension. For example, the model file can be called 'RMmodel01.txt", or 'Vs0001.txt' or 'model_01.txt'. The idea is to name the files sequentially for each set of traces being processed as one goes from lower to higher geophone numbers.

In the above case, model derived by analyzing traces 1-6 is saved to file called 'RMmodel01.txt'. Repeat the steps for traces 3-8, 5-10, 7-12, 9-14, 11-16, 13-18, 15-20, 17-22, 19-24, 21-26, 23-28, 25-30, 27-32, 29-34, and 31-36. Models derived from each set should be saved to file that has the numeric value sequentially increasing with increasing geophone numbers. In this case the model files were called 'RMmodel02.txt', 'RMmodel03.txt',…,`RMmodel16.txt'.

5.2 Assembling a 2D Cross-section

Once the 1D models for each set of traces have been derived, they can be assembled to create a 2D shear-wave velocity cross-section under the profile. To do this, the saved model files have to be edited to enter the value of the horizontal distance of the midpoint of the traces being analyzed and the elevation, if any, at that location.

For example, open 'RMmodel01.txt' (Figure 73). Edit the first line and enter the horizontal distance and elevation of the midpoint of the traces being analyzed. Since the geophone spacing was 10 feet, the midpoint between trace 1, which is at 0 feet, and trace 6, which is at 50 feet, will be 25 feet. Enter the value "25.0 0.0" (since it is flat elevation) in the first line (Figure 73). Repeat this step for each of the saved model files, each time changing the horizontal distance and the elevation values. Figure 74 shows the change made for the last model file (traces 31-36, file 'RMmdoel16.txt'). The horizontal distance for this will be 325 feet.

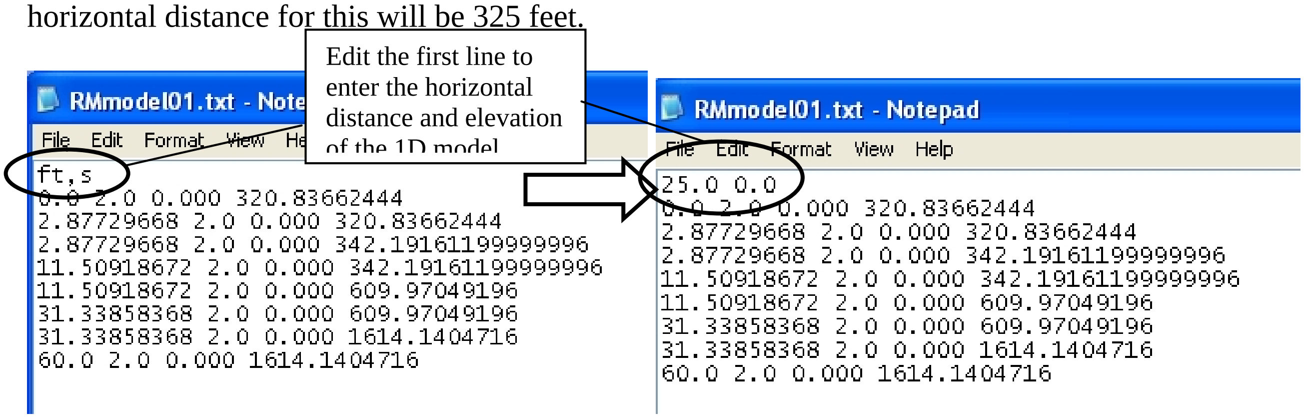


Figure 73: Saved model file for traces 1-6, 'RMmodel01.txt', is edited to enter the horizontal distance and elevation of the model location. Note that the location is the midpoint of the traces being analyzed.

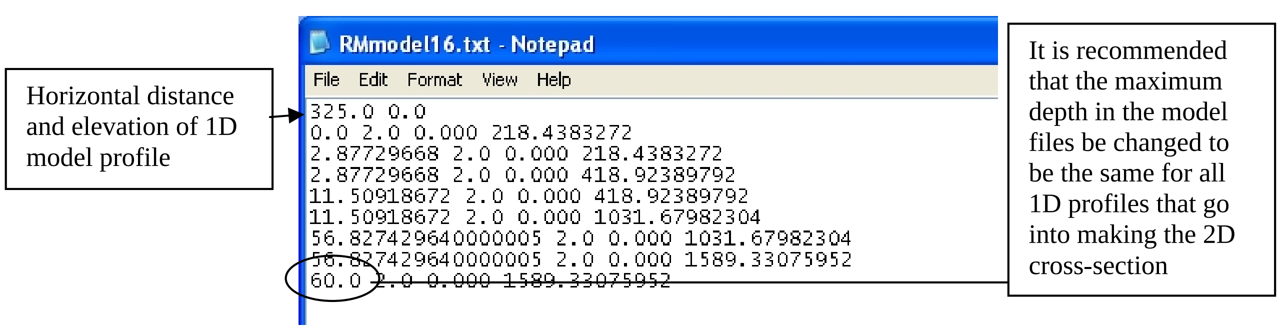


Figure 74: Model file 'RMmodel16.txt' is edited to include the location of the last set of traces that were analyzed (traces 31-36).

It is also recommended that the maximum depth in the 1D profiles (last value in the 1st column, Figure 74), be edited so that it is the same in all the files. This will ensure that the 2D cross-section will be rendered properly.

The next step is to read in the edited 1D model files and generate a 2D cross-section. Select the 'Create 2D profile' option under the 'File' menu from either the 'Controls' window or the 'Model Profiles' window (Figure 75).

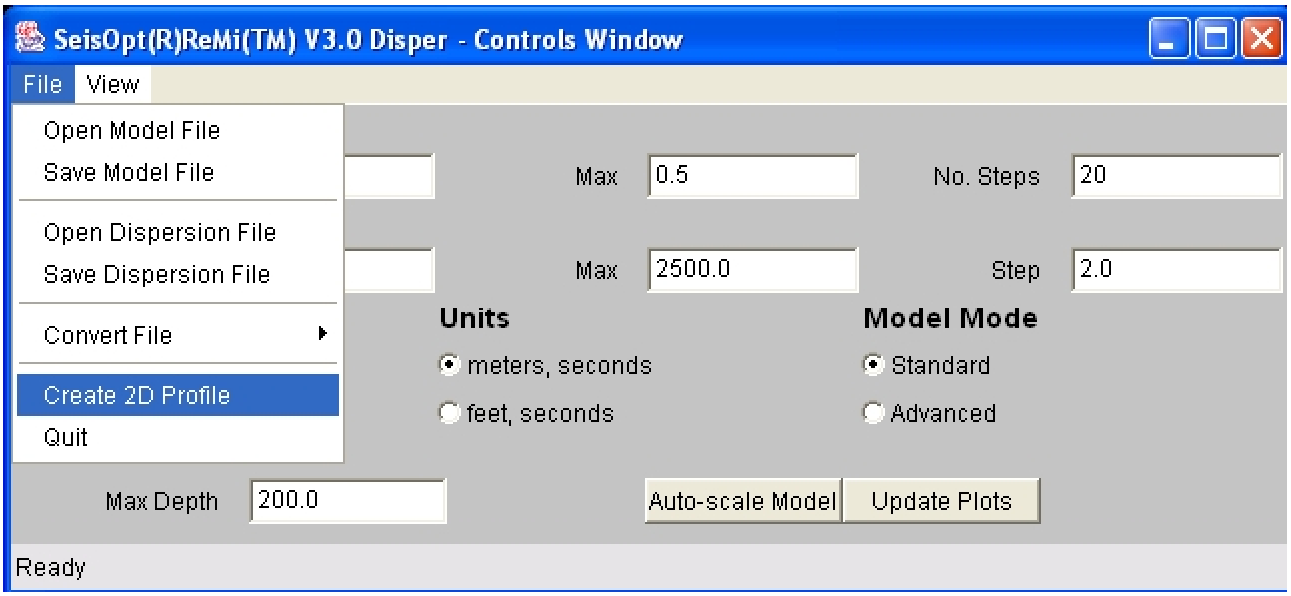


Figure 75: Choose the 'Create 2D Profile' option from the 'File' menu to start the 2D assembly process.

The 'File browse' window shown in Figure 76 will open up. As the title bar on the window indicates, choose the first 1D profile model file that will go into creating the 2D cross-section. In this example, the file is the edited file 'RMmodel01.txt'. Select the file and click 'Open'.

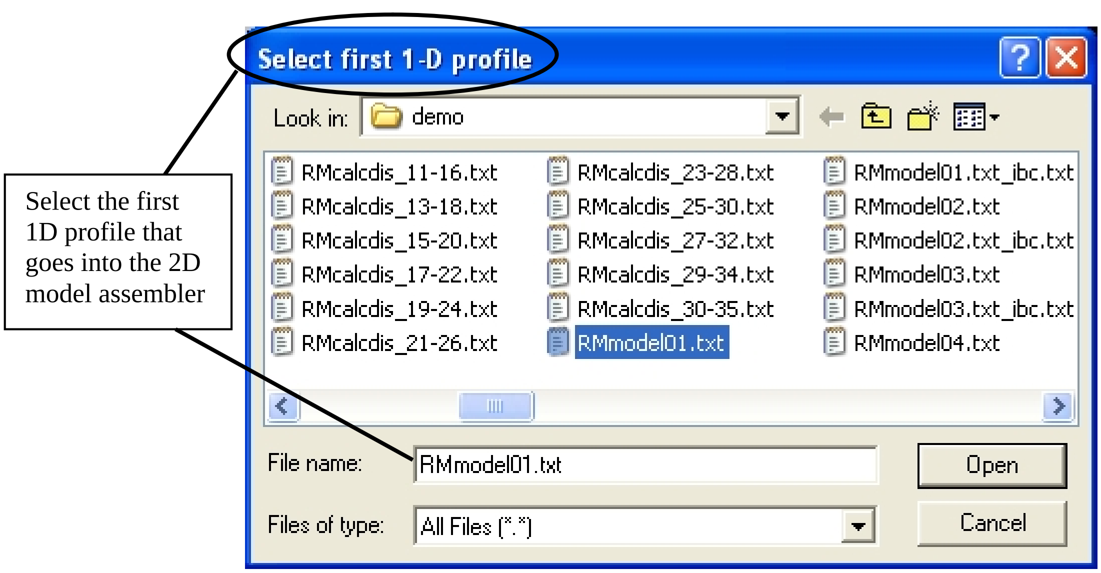


Figure 76: Select the first 1D profile to be read in and click 'Open'.

Another window will open up requesting the last 1D profile model file be chosen (Figure 77). Select the last file ('RMmodel16.txt' in this example) and click 'Open'. If the models have been successfully read in, the dialog window shown in Figure 78 will open. **Note that the assembler will not work if the model files are not named sequentially and if the numeric component of the file name does not increment by 1.**

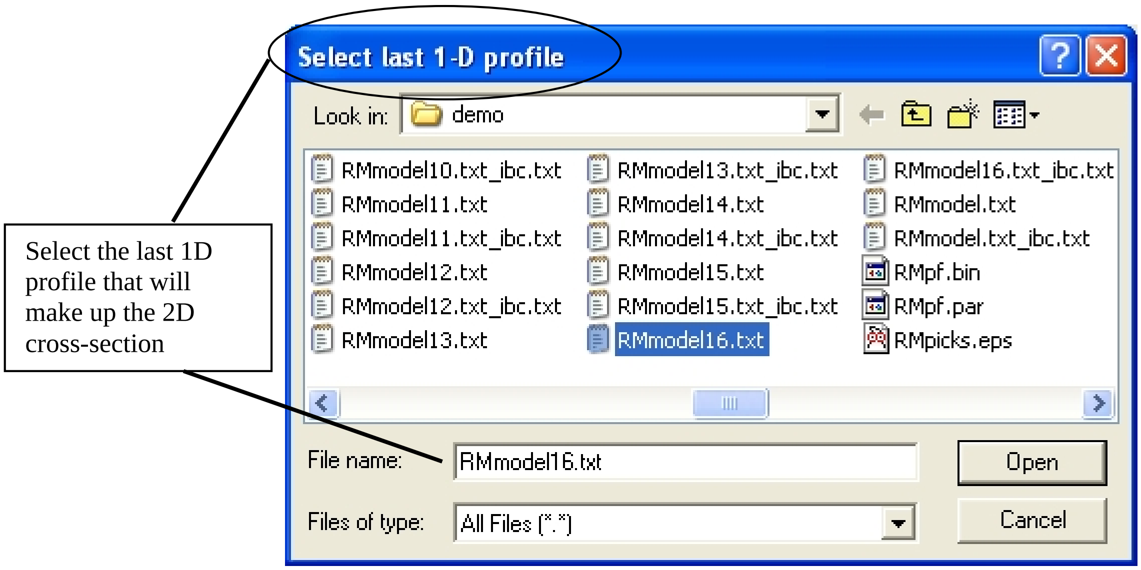


Figure 77: Select the last 1D profile to be read in to create the 2D cross-section.

The dialog window shown in Figure 78 allows the user to set the following plotting parameters:

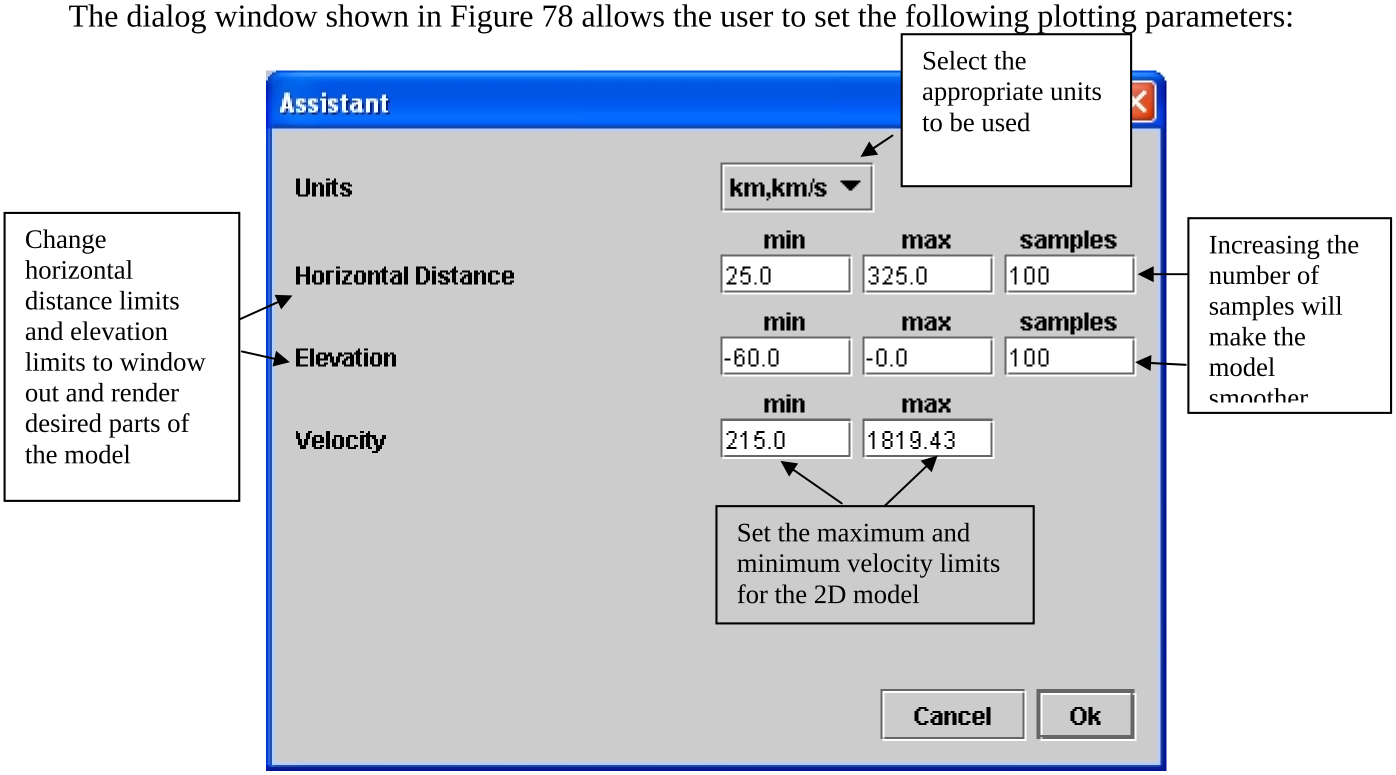


Figure 78: Set the plot parameters in this window and click 'OK'.

- Units: Use the pull-down menu to select the appropriate unit for the plot. These should be units that were used when saving the 1D model profiles. It can be 'km,km/s', 'm,m/s', of 'ft,ft/s'.

- Horizontal Distance: The module automatically estimates the minimum and maximum horizontal distances of the 2D cross-section. These values will be displayed in the 'min' and 'max' box. They can be changed to window out parts of the model to be displayed. The 'Samples' box shows the number of samples used when interpolating between the each 1D profile. More the samples, smoother the model in the horizontal direction.
- Elevation: The maximum elevation is displayed in the 'max' box and will be the top of the model (zero depth). The minimum elevation is the bottom of the model and should be less the maximum elevation value. 'Samples' box shows the number of samples used when interpolating between the each 1D profile in depth. More the samples, smoother the model in depth.
- Velocity: The default maximum and minimum values in the velocity model are displayed in the window. The velocity limits can be changed to according to user needs.

Figure 79, shows the 2D cross-section. One can click anywhere on the image to view the location and the velocity value. The amplitude scale can be adjusted to bring display the desired features better. In this case the maximum and minimum values were change to 1,500 ft/s and 300 ft/s to bring out the low-velocity shallow layer and higher velocity material to the right.

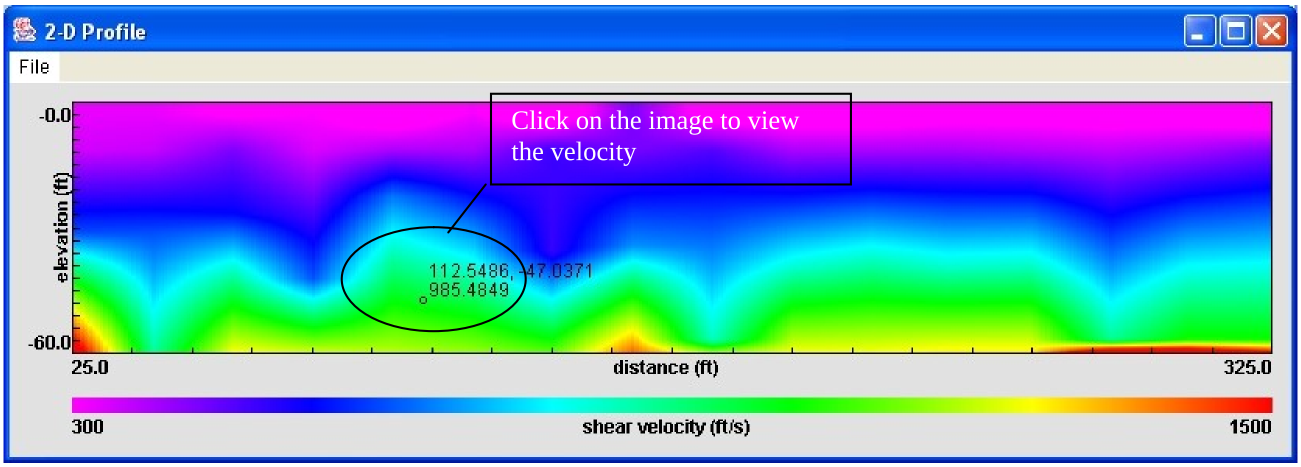


Figure 79: 2D cross-section created along a 36-geophone array with 10 feet geophone spacing.

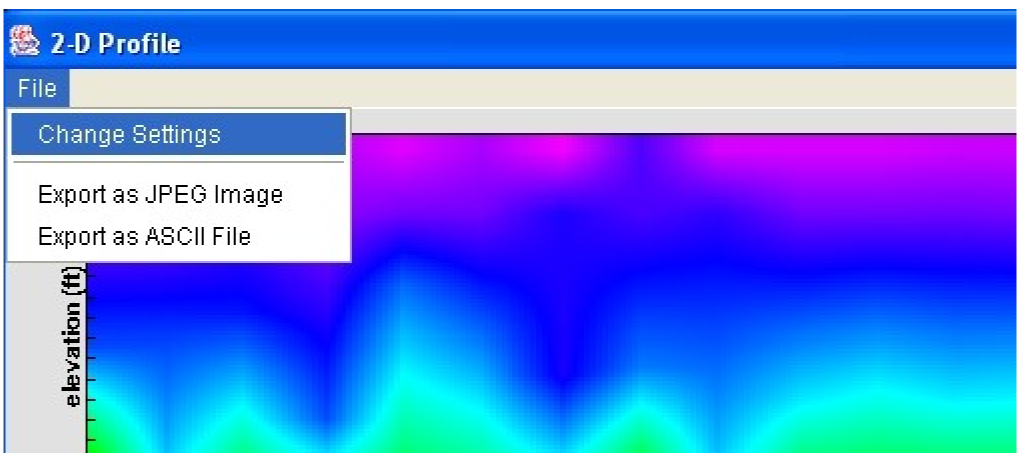
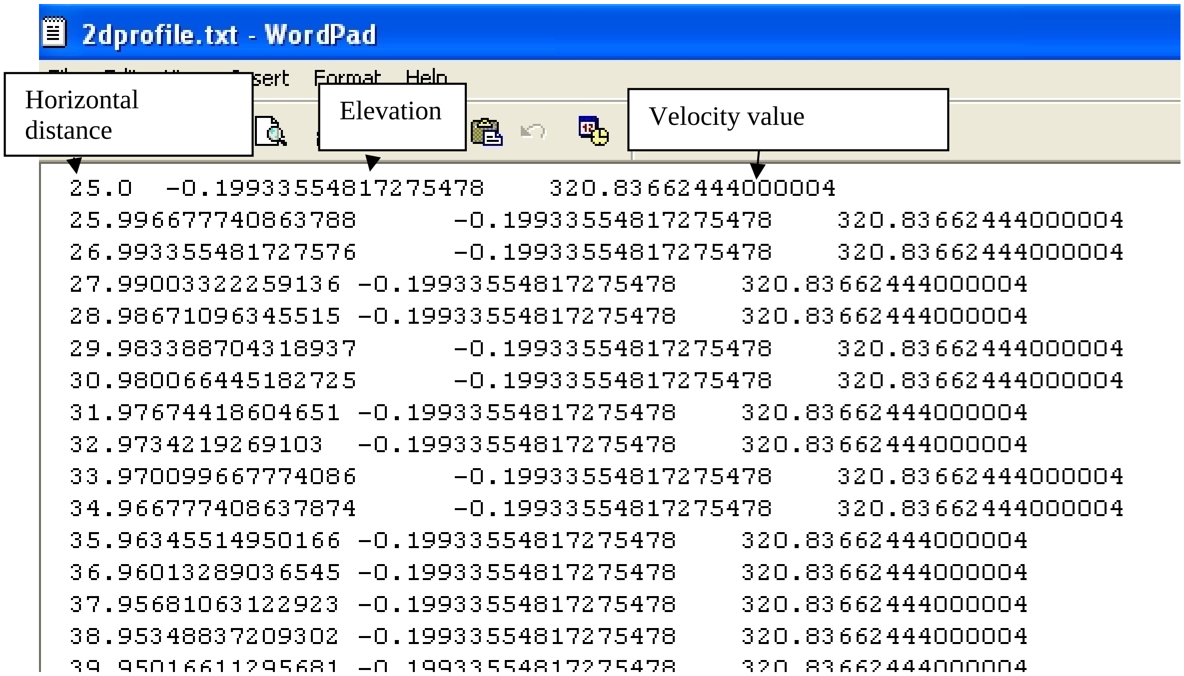


Figure 80: Choose 'Change Settings' to open the dialog window and change the plot parameters. Select 'Export as JPEG Image' to save the model as a jpeg file and choose 'Export as ASCII File' to save the write out the ASCII velocity values and its horizontal location and elevation within the subsurface.

The plot parameter window (Figure 78) can be viewed by selecting ‘Change Settings’ from the ‘File’ menu in the 2-D Profile window (Figure 80). The plot settings can then be changed and the image rendered accordingly. Choosing the ‘Export as JPEG Image’ option allows the user to save the 2D image as a jpeg file. The program also offers the option of saving the velocity grid as a 3-column ASCII file, with first column being the horizontal distance, second column being the elevation and third the velocity value at that location (Figure 81). This file can be imported into drawing programs like ‘Surfer’ to create contoured images of the velocity model.




```
25.0  -0.19933554817275478    320.83662444000004
25.996677740863788      -0.19933554817275478    320.83662444000004
26.993355481727576      -0.19933554817275478    320.83662444000004
27.99003322259136 -0.19933554817275478    320.83662444000004
28.98671096345515 -0.19933554817275478    320.83662444000004
29.983388704318937      -0.19933554817275478    320.83662444000004
30.980066445182725      -0.19933554817275478    320.83662444000004
31.97674418604651 -0.19933554817275478    320.83662444000004
32.9734219269103  -0.19933554817275478    320.83662444000004
33.970099667774086      -0.19933554817275478    320.83662444000004
34.966777408637874      -0.19933554817275478    320.83662444000004
35.96345514950166 -0.19933554817275478    320.83662444000004
36.96013289036545 -0.19933554817275478    320.83662444000004
37.95681063122923 -0.19933554817275478    320.83662444000004
38.95348837209302 -0.19933554817275478    320.83662444000004
39.95016611295681 -0.19933554817275478    320.83662444000004
```

Figure 81: ASCII velocity file saved using the ‘Export ASCII file’ option under the ‘File’ menu in the 2-D profile window.

6.0 Solutions to Possible Problems and Other Useful Information

It is possible to encounter some problems while running SeisOpt ReMi software, mostly because of memory issues or using incorrect file formats. Below we list some trouble shooting tips.

6.1 Authorizing the license

SeisOpt ReMi will not run without a valid license and registration. Follow the steps outlined in Section 2.0.

The licensing software used by ReMi uses hidden files to keep track of the license. If an ‘unlimited’ license ever becomes disabled, it is probably because one or more of the hidden files was inadvertently removed, possibly by a hard-disk utility program. For example, this is known to happen when running Speed Disk, a de-fragmentation utility included in Symantec’s Norton Utilities. This also happens while running Norton Anti-Virus Utility. Loss of these files will result in the loss of license to run SeisOpt ReMi

To prevent these losses do the following:

1. Open Speed Disk, and choose **File, Options, Customize,** and **Unmovable Files.**
2. Specify that the *.ENT, *.RST, .KEY, and.41S files cannot be moved.

6.2 Transferring the license

SeisOpt ReMi Vspect and Disper module license can be transferred from one computer to another. ***But the user will still have the Register ReMi by following the instructions in Section 2.0***. The transfer process only transfers the licenses but not the registration. The transfer should be done before uninstalling SeisOpt ReMi on the old computer. Once the license has been transferred, SeisOpt ReMi will not run on the old computer. Here are the steps to follow for transferring the ReMi license from one computer to another:

1. Install SeisOpt ReMi on new computer.
2. Put a new floppy disk into the disk drive
3. Click on the SeisOpt ReMi Disper desktop icon (Figure 2b). A license window will open.
4. Go the ‘‘License’’ menu and choose ‘‘Transfer in from another computer’’
5. The program imprints its registration on the disk.
6. Now remove the floppy disk and put it into the old computer on which SeisOpt ReMi was installed.
7. Start SeisOpt ReMi on the old computer and hit return when the ‘‘Check License’’ window comes up.
8. A license window will open up.
9. From the license window, go to License menu and choose ‘‘Transfer Out of Computer’’. Supply the floppy disk path.
10. Remove the floppy disk and go back to new computer.
11. Click ‘‘Transfer into Computer’’ to complete the transfer and discard the intermediate imprint files on the floppy disk.

6.3 Out of Memory error

Memory shortage may cause large SEG-Y files not to be displayed by the ReMi Vspect module. If such a scenario arises, an ‘‘Out of memory’’ error message will be displayed on the console (MS-DOS) window. If this happens, send email to support@optimsoftware.com mentioning the amount of RAM available on your PC. A replacement “vspect” and “disper” script will be sent to you for your use.

6.4 Cannot Find Root Error

This error might occur when the velocity limits for the model in the ‘Controls window’ of the ReMi Disper module are set to be inside the limits of the profiles displayed in the ‘Model Profiles’ window. For example, if the minimum velocity was set to be greater than minimum velocity in the profile and the maximum velocity less than the maximum velocity of the profile, then a ‘‘Cannot find root error’’ will appear in the ‘Dispersion Curve’ window. The ‘Model Profiles’ window will not display a velocity model (no red lines). To avoid this, make sure the velocity limits are outside the maximum and minimum velocities in the ‘Model Profiles’ window.

6.5 ReMi Disper Module: Shape of the Calculated Dispersion Curve

As shown in the examples included in the manual, the calculated dispersion curve (blue line the 'Dispersion Curve' window) should be smooth. Sometimes, the curve might display sharp changes, especially towards the edge of the windows. This is an artifact of the limits placed on the velocity model, period, and maximum depth in the 'Controls Window' of the Disper module. Change these and click 'Update'. If it doesn't make any difference reload the dispersion curve and model file and re-render the calculated dispersion curve.

7.0 References

Louie, J. N., 2001, Faster, Better: Shear-wave velocity to 100 meters depth from refraction microtremor arrays, Bulletin of the Seismological Society of America, Vol. 91, 347-364.

Pullammanappallil, S., Honjas, B., Louie, J., Siemens, J. A., and Miura, H., 2003, Comparative study of the refraction microtremor method: Using seismic noise and standard P-wave refraction equipment for deriving 1D shear-wave profiles, Proceedings of the 6th SEGJ International Symposium (January 2003, Tokyo), 192-197.

Saito, M., 1979, Computations of reflectivity and surface wave dispersion curves for layered media. I. Sound wave and SH wave, Butsuri-Tansa, Vol. 32, no. 5, 15-26.