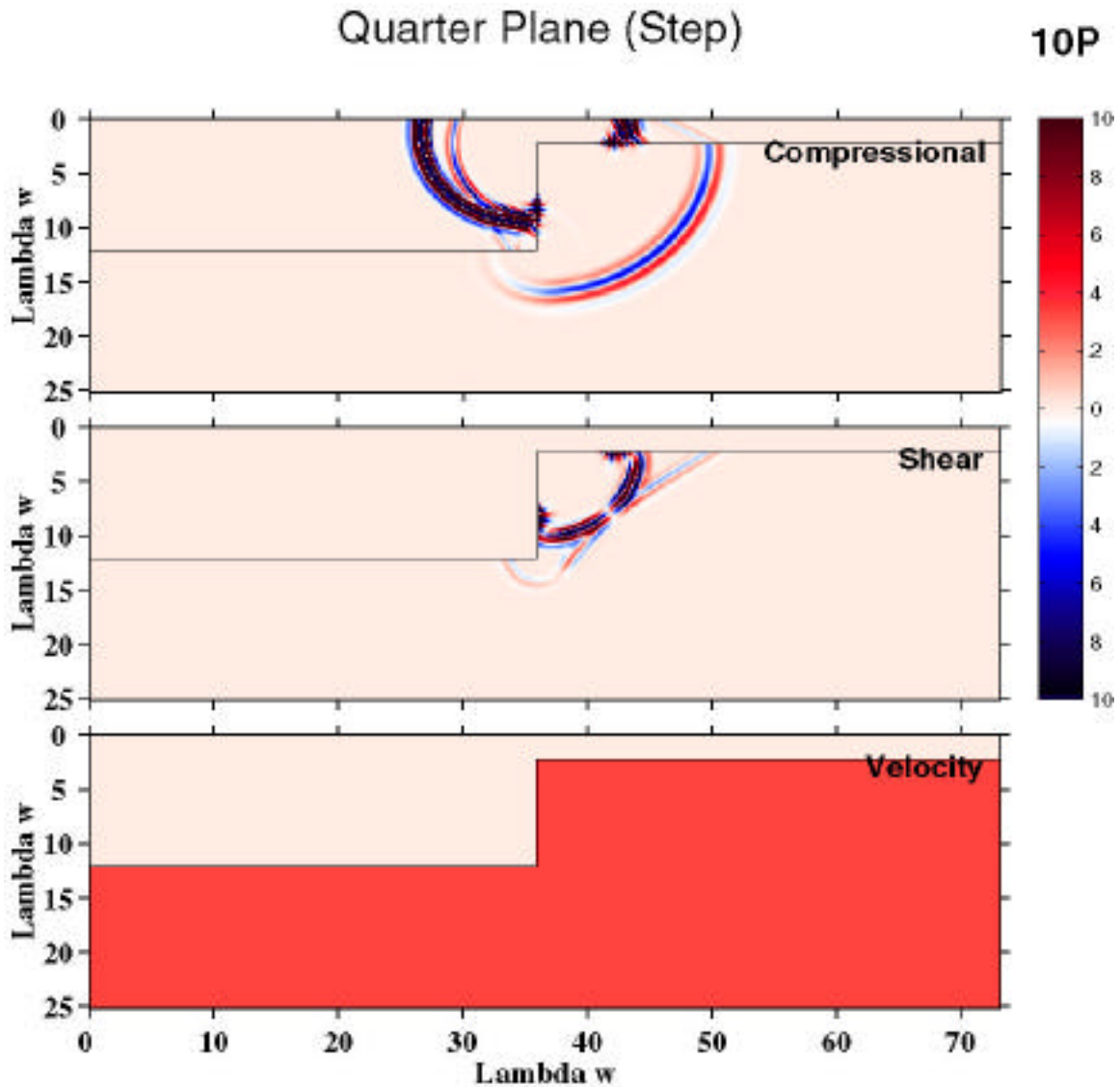


USER's GUIDE
for
PLOT_FINDIF_1
S. T. Bolmer & R. A. Stephen
November 2004



ras07_100 plotted on: 10/28/2004 10:45:37

PLOT_FINDIF_1

Plot_findif_1.m is a MATLAB 6 script, which is used to plot output from the Woods Hole Oceanographic Institution Finite Difference Program. This script will plot both the snap shots and the time series output from the Finite Difference program. It is part of a set of 32 MATLAB functions to do this (see list of m-files at the end of this document). To run this you must have MATLAB 6 and the suite of m-files. This code works pretty well but there are some complaints that MATLAB makes which end up being benign.

This works with Matlab 6. The one time I tried this with Matlab 7 I had problems. I have not had time to check this code's compatibility with Matlab 7 yet.

Tom Bolmer

11/01/04

tbolmer@whoi.edu

TABLE of CONTENTS

<i>M-FILES LOCATION</i>	5
<i>GETTING STARTED</i>	5
Selecting a DIRECTORY	5
Figure 1. The main GUI window for <i>plot_findif_1</i> at startup.	6
Figure 2. Select Directory popup window.	7
<i>SNAP SHOT PLOTTING</i>	7
TIME STEP and NORMAL Model files	7
MAX and MIN Points	7
Figure 3. Snap Shot GUI control section.	8
Adjustment FACTORS for Depth and Range	8
AXES LABELS	9
TIME STEP LABEL Factor	9
Managing TITLES	9
Figure 4. Manage Titles popup window.	10
Figure 5. Selecting a file with titles in it.	11
Figure 6. Editing and adding a new title in the edit box.	12
VELOCITY PLOT and SeaFloor/Sediments	12
FULL or WINDOW Plot	13
CREATE a SNAP SHOT Plot	13
Figure 7. An Example Snap Shot Plot.	14
<i>TIME SERIES PLOTTING</i>	15
Figure 8. The Time Series GUI section.	15
FILE EXTENT	15
Show MODEL Space	15
Figure 9. Plot of the Model Space.	16
HOW to SELECT Which TRACES to Plot	16
TRACE Numbers or MODEL Space to Plot	16
Frequency of Traces to Plot	17
What to Plot TRACES Against	17
Values to Adjust Model space with	17
Physical SIZE of the Plot	17
SCALING of the Data	18
REDUCTION VELOCITY	18
CLIPPING	18
EXTRA TITLE	18
New SAMPLING RATE	18

AXES LABELS	18
CREATE a TIME SERIES PLOT	19
Figure 10a. An Example of plotting only the traces in the top set of receivers in the TSP file.	20
Figure 10b. An Example of plotting only the traces in the bottom set of receivers in the TSP file.	21
Figure 10c. An Example of plotting only the traces in the bottom set of receivers in the TSS file.	22
Figure 11. An example of filling only the Negative section of the trace.	23
Figure 12. An example of filling only the positive section of the trace.	24
Figure 13. An example of filling both the negative and positive sections of the trace.	25
Figure 14. An example of filling the trace and showing the trace also.	26
<i>List of Matlab m-files needed to run the function.</i>	27

M-FILES LOCATION

To be able to run this you must have the location of the MATLAB functions in your path. If you have installed them in a place that is not part of your normal MATLAB path, you must use the *addpath* command in MATLAB to do this. The *plot_findif_1* script can be run from any directory but the easiest place to run this from is the directory in which the data to plot are located.

GETTING STARTED

To start the function in MATLAB type: `[lfpar2]=plot_findif_1;` This will bring up the GUI window shown in Figure 1. When the function starts up, it gets its initial parameter settings from *lastfindif.mat*, if the file exists in the current directory. If the file does not exist, then the default parameter settings in the function are used. By working in the data directory the file *lastfindif.mat* will be saved there and the latest plotting parameters will be available to you the next time you wish to make plots of this data. This will save time and help you remember the parameters last used.

On startup the code looks in the directory selected, if there is a *lastfindif.par* file in the directory, or in the local directory, if there is *no lastfindif.par* file in the current working directory, for a file with the extent of .par or .PAR. If that file exists, then the model name box (***MODEL (use 5 characters)***) will be filled in, the ***TIME STEP*** pull down menu will be filled with the time steps available, and the Time Series files extents (***Data file Extent***) pull down menus will be filled in. The code does not accept any files that are compressed. If you do not see the files you want and you know they are there, be sure that they are uncompressed.

Selecting a DIRECTORY

To select a new data directory, you can either fill in the full path name in the Directory box or use the ***Select Data Directory*** button (see Figure 2). The default startup directory is the current working directory. Once you select another directory, the Model name, Time Step numbers, and the Time Series file extents will be updated. In the ***Select Data Directory*** popup window, one click will change to a new directory and list the directories in that directory. Only directories are shown in this window. If you select the / directory, the code will complain, but do not worry. Close the popup window and re-select it. You will be at the directory where you left from and you can continue. I can't figure out why this happens but getting in and out of the popup works. Once you get to the directory you want, you will only see the "." and ".." entries in the directory. Click on the ***Accept*** button and then click the ***Quit*** button. When you click the ***Accept*** button, you may have an error message "beep" in the MATLAB window. Again, it is unknown why this happens but it seems to be benign.

Having selected a directory and seen that the proper Model Name, Time Step values, and Time Series file extents have been filled in, you are now ready to plot. The large GUI window will plot both the ***Snap Shots*** and the ***Time Series*** created by the FINDIF program. The controls from one section don't affect those of the others. Only selecting the

directory changes settings in the two types of plot menus.



Figure 1. The main GUI window for *plot_findif_1* at startup.

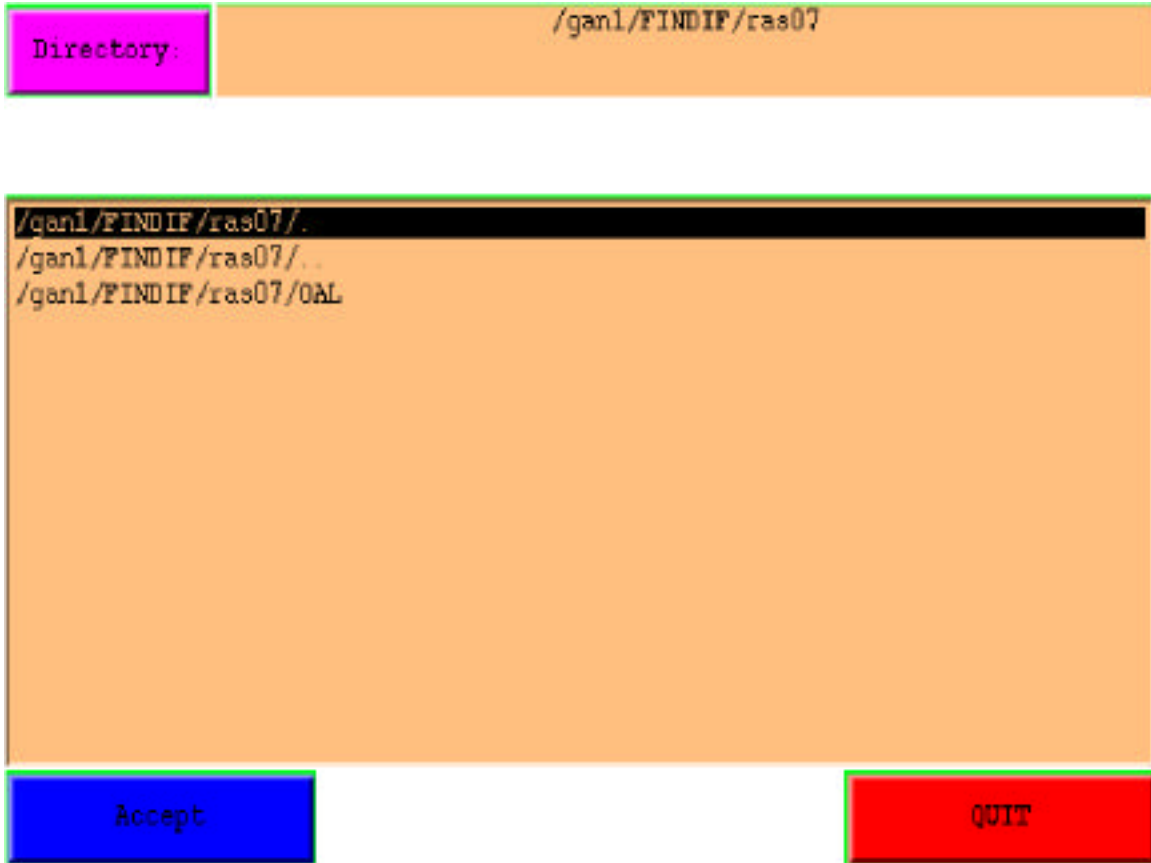


Figure 2. Select Directory popup window.

SNAP SHOT PLOTTING

TIME STEP and NORMAL Model files

To plot a snap shot work from the top to the bottom of the menu. First, select the Time Step you want to plot. Hold the left mouse button on the Time Step pull down menu and slide the mouse to the proper Time Step number you desire. The numbers here will be those that the directory selection process found in the area you chose. Once you have selected a time step, be sure the button to the right shows that ***Normal model files*** are to be used. Toggle this to select the ***Normal model files*** if it is not already set.

MAX and MIN Points

You next will want to edit the Max and Min Points boxes. These values are the Maximum and Minimum values that will be contoured in your plot. These are the values

that appear at the top and bottom of the color bar on the snap shot figure. The color contouring will fill in the points greater and less than this set of Max and Min values with the closest color value to it on the color table. The right *Min points value* box will be automatically set to the negative value of the left *Max points value* box. If you want a different Minimum value, you must explicitly edit that in yourself.

The screenshot shows a GUI control section for 'SNAP SHOTS'. At the top, there are fields for 'Directory' (set to '/gen1/FINDIF/ras07') and 'MODEL' (set to 'ras07'), with a 'Select Data Directory' button. Below this is a large green 'PLOT the Snap Shot' button. The main control area includes a 'TIME STEP' dropdown (set to 100), a 'Normal model files' dropdown, and input boxes for 'Max points value' (10) and 'Min points value' (-10). There are also fields for 'factor into DEPTH values' (1), 'Depth label' (Lambda w), 'factor into RANGE values' (1), 'Range label' (Lambda w), and 'factor into time Step for P label' (10). A 'Plot Title' dropdown is set to 'Quarter Plane (Step)', with a 'Manage Titles' button. Below these are three checkboxes: 'Plot Velocity' (checked), 'Plot Full Model' (unchecked), and 'Plot SeaFloor' (unchecked). At the bottom, there are input boxes for 'Top Depth Value' (0), 'Left Range Value' (0), 'Right Range Value' (1000), and 'Bottom Depth Value' (1000).

Figure 3. Snap Shot GUI control section.

Adjustment FACTORS for Depth and Range

Sometimes the range and depth values in the snapshot file are not written in the right units. The edit boxes *factor into DEPTH values* and *factor into RANGE values* can be used to adjust the axis labeling. This is the value to divide into the depth and range values passed in the snapshot file. When the *factor into DEPTH values* is changed *the factor into RANGE values* is automatically set to that value. If you want a different value for the *factor into RANGE values*, then you must be sure to change the RANGE factor also.

AXES LABELS

At present, the Axis Labels can be set to three different labels. To select these hold the left mouse button on the GUI button for the ***Depth label*** or ***Range label*** and highlight the label you desire. When you release the mouse button the highlighted option is now selected. These options are currently:

- Depth (or Range) w
- Depth (or Range) (Kilometers),
- Depth (or Range) (Meters).

As with the factors above, the ***RANGE label*** will change to match the ***DEPTH label*** when the ***DEPTH label*** is selected. You must change the ***RANGE label*** separately, if you wish a different RANGE axis label from the ***DEPTH label***.

TIME STEP LABEL Factor

It is also possible that the file's TIME STEP number may not be right for the actual time unit. The ***factor into the Time Step for P label*** corrects that for labeling purposes. This value is divided into the TIME STEP value selected above for plot labeling purposes.

Managing TITLES

You can use any title you want for the plot. This has been a less than easy to use option. *Just remember that if all else fails you can edit the title in the MATLAB figure window.* There is a core set of titles that are in the title pull down menu. These probably won't contain the title you want to use, since your model will be different from the core titles we used when this code was written. You can use the ***Manage Titles*** button to get a popup window to create or load your own titles file (see Figure 4).

You can create your own file to save titles you use frequently. If you click on the button ***open <your user name> Titles***, you will see the files in your default login area in the Matlab directory (~/.Matlab) that are named *.title. You must select one of these files to open even, if you see one of them already highlighted (see Figure 5). Once you click on the file name, that file is now loaded into the ***Manage Titles*** window. You should click on the ***QUIT*** button to close this window. Now you can select a new title in the ***Manage Titles*** pull down menu.

You can also add and edit titles in the ***Manage Titles*** window (see Figure 6). ***The EDIT selection*** does not work very well, but you should be able to select a title from the scroll window to edit and change in the edit box. This change will be reflected in your title bar on the main GUI window. You can also add a new title in the same general way. To save your changes for the future you can hit the ***SAVE*** button. This will bring up a new window to select a file to save with this new title. In this Save/Select a file window you MUST select a file even if it looks like one is highlighted. When you do this, you will see the ***SAVE TITLES*** button come up.

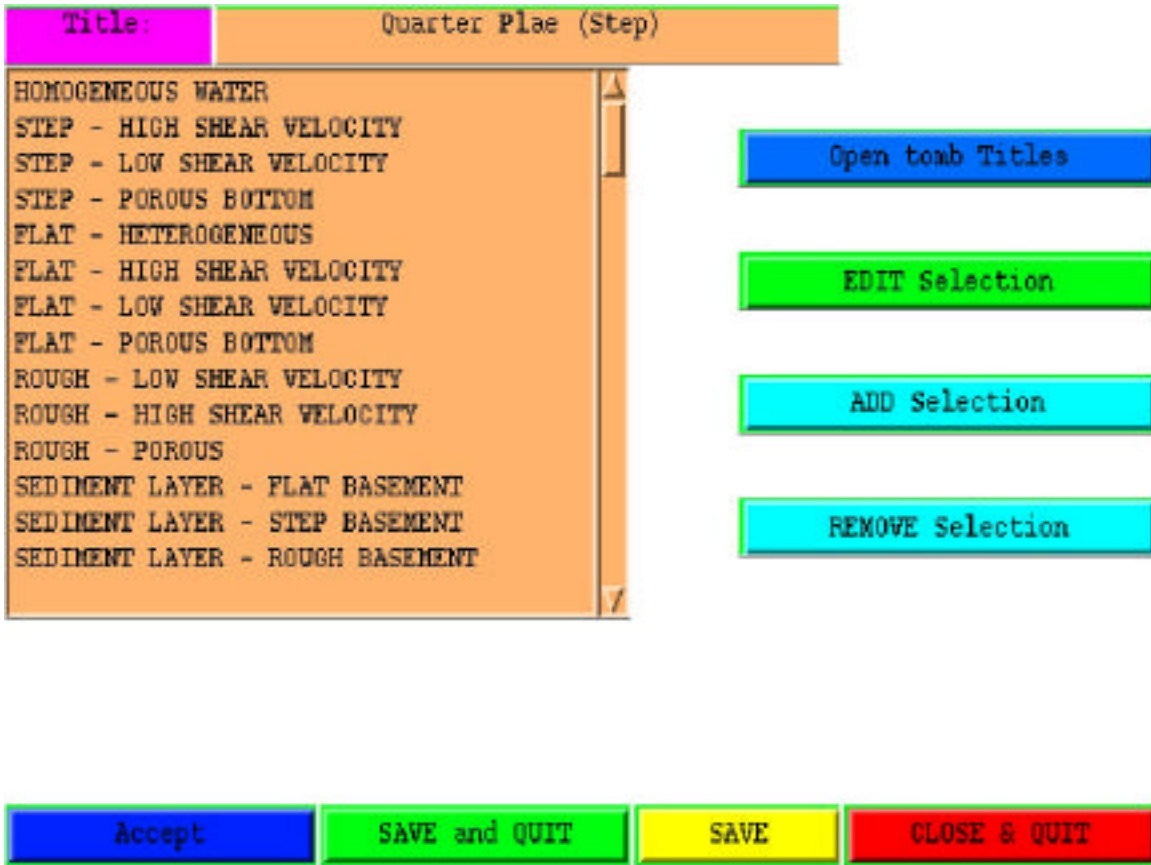


Figure 4. Manage Titles popup window.

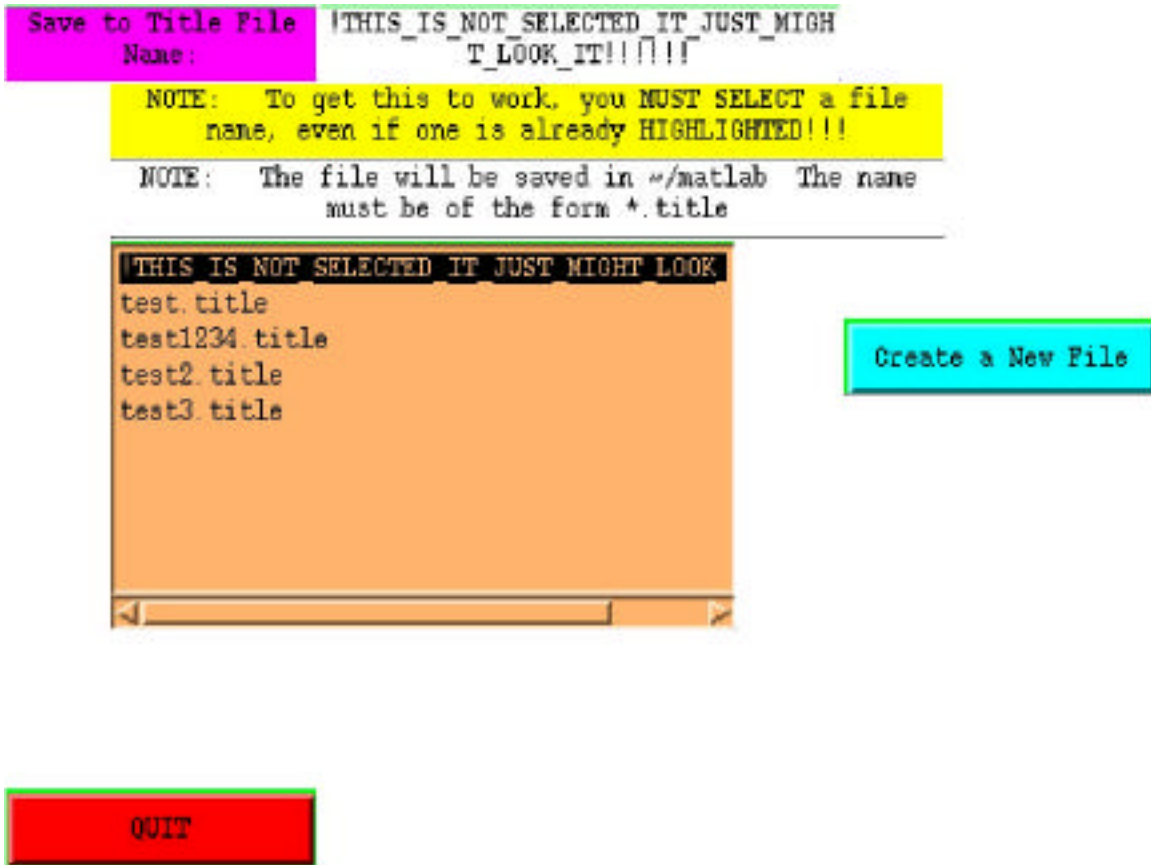


Figure 5. Selecting a file with titles in it.

The **SAVE TITLES** button is not there until you do select a file. You can either click the **Create a New File** button or click the **SAVE TITLES** button to save this set of titles to the selected file. If you are saving to an existing file, you will get a warning message asking you if you want to overwrite this file or not. You must decide whether you will save to that file or not. If you opt **NOT** to save, you must re-enter the **SAVE** window again. Once you have saved your file you should click on the **ACCEPT** button. The **ACCEPT** button may also give you a warning beep, which to date has been benign. After that you should click on the **CLOSE & QUIT** button. The **SAVE AND QUIT** button combines all the above into one step. The **CLOSE & QUIT** button only closes the window and does not load your new title, if you did not **ACCEPT** it. It just might look like it was accepted to you.

I strongly recommend closing this **MANAGE TITLES** window and any other windows before proceeding any further to keep the computer screen less cluttered and not force Matlab into keeping too many windows active.

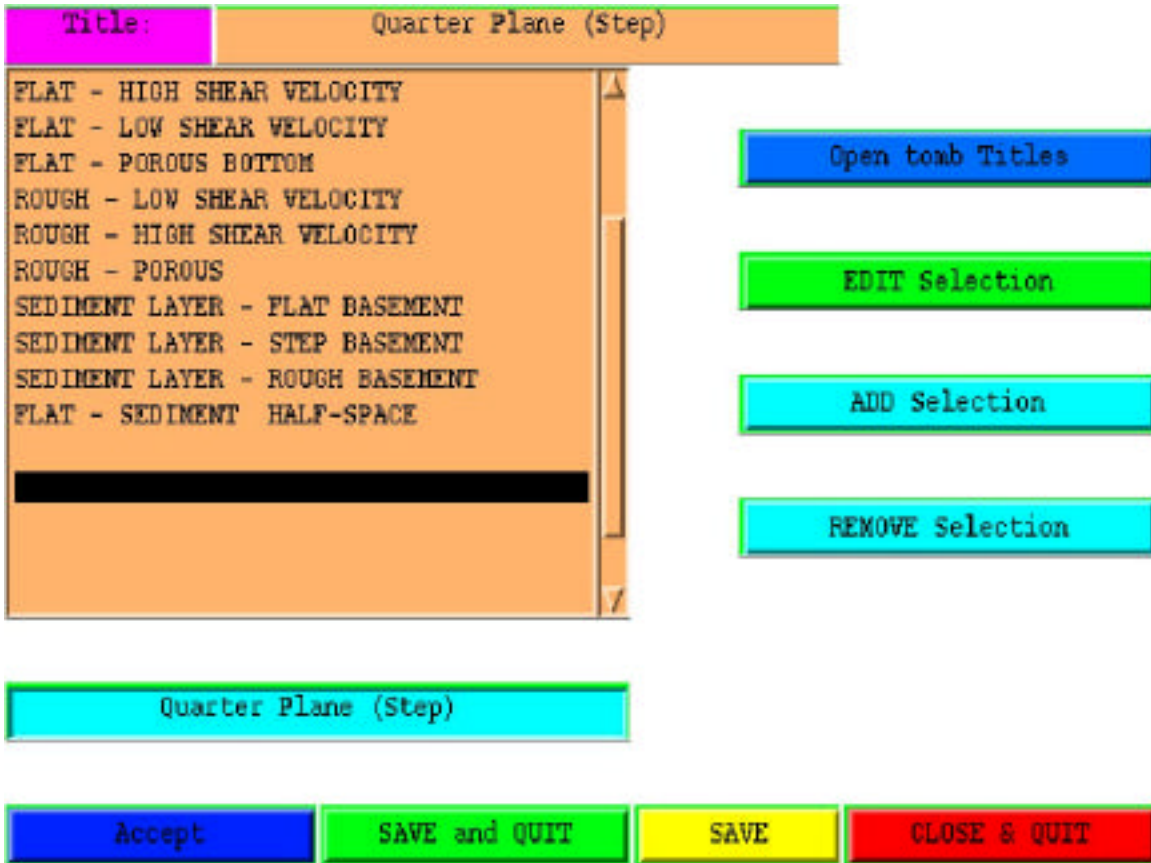


Figure 6. Editing and adding a new title in the edit box.

VELOCITY PLOT and SeaFloor/Sediments

The *Plot Velocity* toggle switch switches between *Plot Velocity* and *Do NOT Plot Velocity*. If you select to plot the velocity, a third box will be plotted showing a colored snapshot of the velocity structure used to create this model. You MUST see a file named by using the model name with the .VEL extension in the data directory. Again, if this is compressed, you must uncompress it. The program will not run properly if there is no .VEL file.

You have two options for showing the bottom in your plots. You can toggle between the *Plot SeaFloor* and the *Plot Bottom of Sediments*. The information to plot this data will be gotten from the Velocity file mentioned above. You will ALWAYS have the SeaFloor plotted but can select not to plot the Sediment bottom. To get these layers the values in the Velocity file are searched to find changes.

FULL or WINDOW Plot

You can select to plot a part of the model area. Toggle the ***Plot Full Model*** button to get the ***Plot Window*** menu as shown in Figure 3. You must enter the values in model **RANGE** and **DEPTH** values space of the window box you want to plot. The default is to plot the whole model. This is a useful option to highlight and blowup a specific region of your model.

The ***Plot Window*** option can be confusing. It only takes in the **RANGE** and **DEPTH** values created from the factors, which you have entered above. The original intent of how to use this option was to look at a full scale plot of the time step and then select the window you want to zoom into. With this in mind, the axis values are much more useful than remembering the size of the rows and columns in the data files. Since the program searches this window using **RANGE** and **DEPTH** units this can be a confusing option, since you are now using a different model "space". If this does not work easily, you can try plotting the full model with the **DEPTH** and **RANGE** factors as 1 to get the units you wish to use to blow an area up.

CREATE a SNAP SHOT Plot

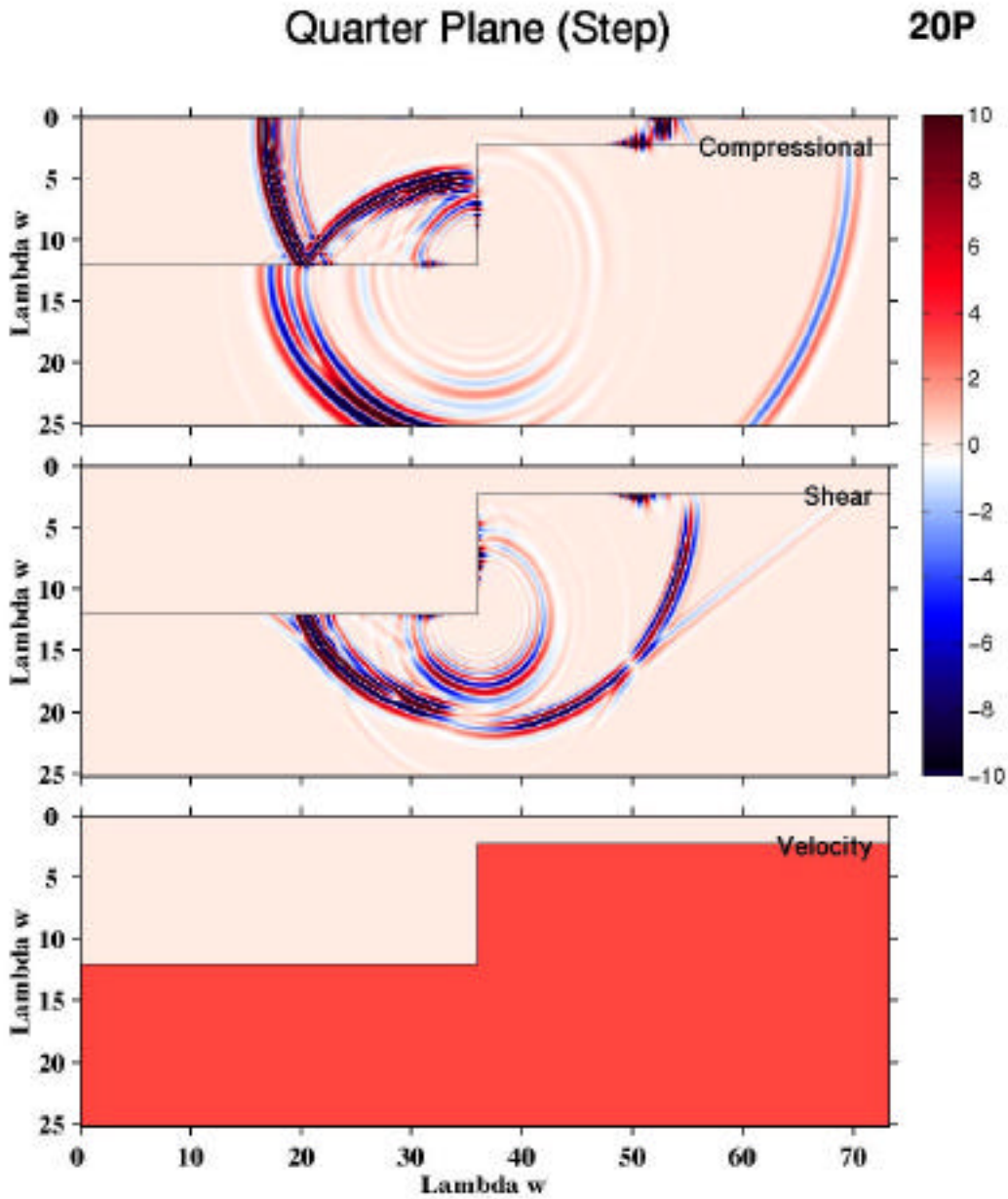
Once all of the above is completed, you may hit the ***PLOT the Snap Shot*** button to create a plot (see figure 7). This takes a little bit of time to work. You will see a red warning in the ***SNAP SHOTS*** box that the data is being read in. When the plot comes up you may decide whether to keep it or not. The plot comes up in the MATLAB figure window. To get rid of this plot you use the ***close*** option under the ***file menu*** in the figure window. You may use all the tools and options MATLAB provides here to print, save, or edit the figure.

I recommend not keeping too many figure windows open, so that your MATLAB program doesn't crash. The program has worked pretty successfully for me but does need some attention. You should learn the functions idiosyncrasies and learn to work with them. The code will only run the way it was written and not the way you think it should.

Parameters used for the below plot (Figure 7) are:

Time Step:	200	<i>Normal model files</i>
Max points value:	10	Min points value: -10
Factor into DEPTH values:	1	Depth label: <i>Lamda</i>
Factor into RANGE values:	1	Range label: <i>Lamda</i>
Factor into time Step for P label:	10	
Plot Title:	<i>Quarter Plane (Step)</i>	

<i>Plot Velocity</i>	<i>Plot Full Model</i>	<i>Plot Seafloor</i>
Left Range Value:	Top Depth Value: 0	Right Range Value: 1000
	Bottom Depth Value: 1000	



ras07 200 plotted on: 09/21/2004 09:01:57

Figure 7. An Example Snap Shot Plot.

(The parameters to create this plot are given on the preceding page)

TIME SERIES PLOTTING

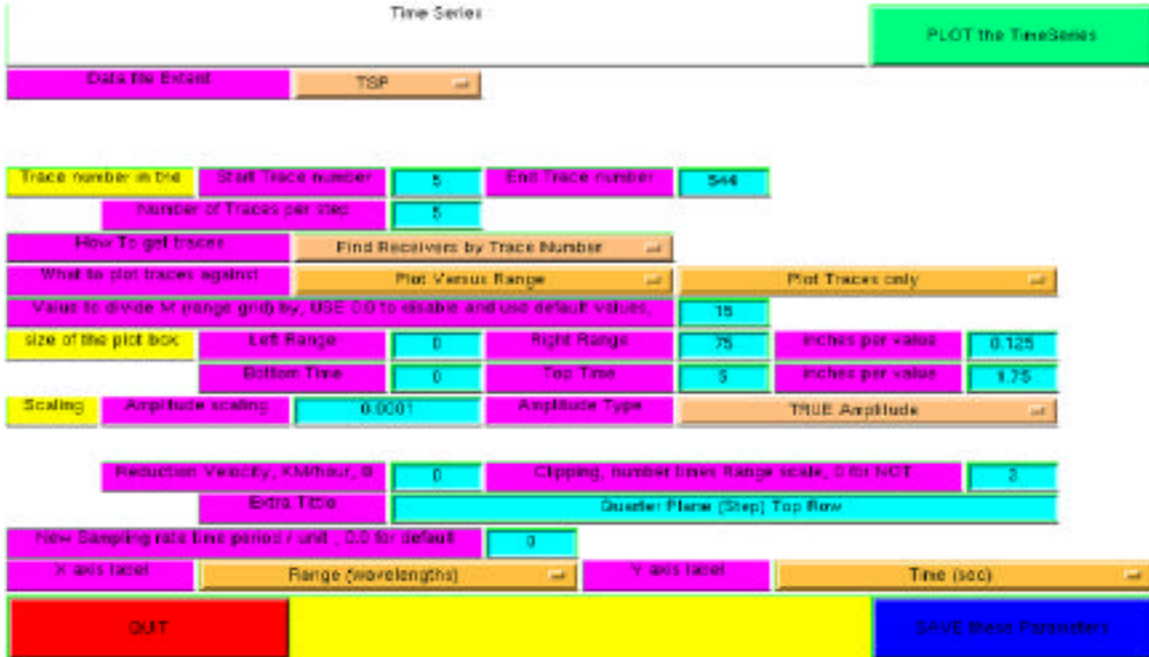


Figure 8. The Time Series GUI section.

FILE EXTENT

To plot a Time Series work from the top to the bottom of the menu. First, select the Time Series file Extent you want to plot. Hold the left mouse button on the Time Series *Data file Extent* pull down menu and slide the mouse to the proper file extent you desire. The values here will be those that the directory selection process found in the area you chose. I will use trace number and receiver number interchangeably in the below sections.

Show MODEL Space

This section depends on you knowing something about how the traces are stored in the Time Series file. You need to know the locations of the receivers you are interested in the file's M and N space (x and y, respectively). The locations and the numbers of receivers varies with a FINDIF model's parameters. You may also plot the traces in the order they are in the file (Trace Number). This can be very confusing at first. To get a summary of the locations of these receivers in the file you can make a plot of this information using the *Show model Space* button. This will read (and load) the whole file. A red warning the file is being read will come up while the data file is read. It makes two plots to help show where the receivers are located (figure 9). Use this figure to help you enter the arguments needed

in the options below.

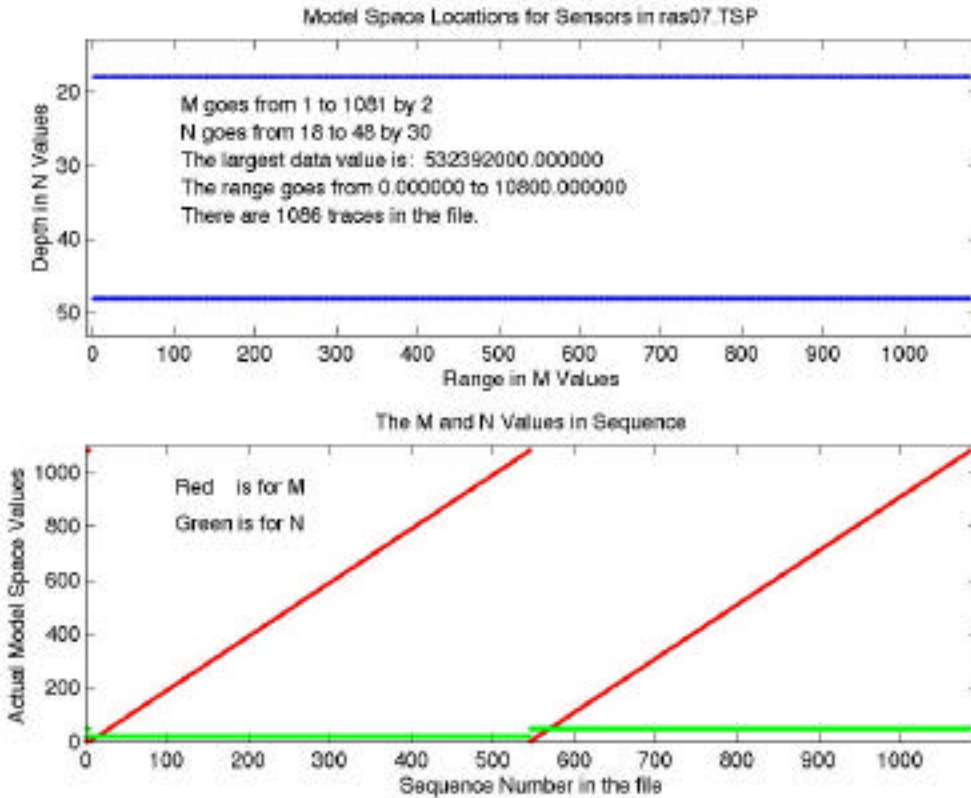


Figure 9. Plot of the Model Space.

HOW to SELECT Which TRACES to Plot

In *the How To get traces* button you have the option to either find the receivers *by Trace Number* or to *find receivers by Model Points*. Both of these options have separate boxes for parameter entry and are only shown depending on the option selected.

TRACE Numbers or MODEL Space to Plot

To plot using the *trace number* you need to know the receiver's sequential location in the file. Fill in the *Start Trace number* and *End trace number* in the boxes that are shown. To plot using the *Model Space* points you need to enter the *Upper* and *Lower* depth points and the *Left* and *Right* range points in the boxes available. These points will form a box around the receivers you want to plot. If a receiver is inside or on the side of this box, then it will be plotted. Be careful to limit this plot to one row of receivers.

Frequency of Traces to Plot

You may select the frequency of plotting the receivers. Every Nth trace will be plotted. Enter this in the ***Number of Traces per step*** box. When using the Model point option to find your receivers remember that every Nth trace is found in the order it is read in the file.

What to Plot TRACES Against

You may plot these data relative to range, time or sequence number in the file. For the section ***What to plot traces against*** there is a pull down menu to select the option you want.

To the right of the pull down menu to select how to find the receivers is a pull down menu to select how to show the traces. There are 5 options available, which are all shown in figures 10 - 14. These options are:

Plot Traces

Fill Negative Wiggles

Fill Positive Wiggles

Fill Wiggles and Traces

Fill Positive and Negative Wiggles

The method how to plots the traces is pretty obvious by the names of the options. The negative fills will be in Red and the positive in Blue. As more traces are filled the time to create the plot will increase. The fastest option is plot only the traces. Use this until you get the windows and amplification values you think are right, then try filling the wiggles.

Values to Adjust Model space with

Sometimes the **Range** and **Depths** are not written in the right units. You can change this by entering a number into the ***Value to divide M by...*** This number will be divided into the Model point value of this receiver against which the plot is being made, either M for range or N for depth. This will replace the range value passed for this receiver. To not use this option enter 0.0 When the ***plot versus trace number*** option is used, this value is set to -12 automatically.

Physical SIZE of the Plot

The plot can also be limited in both the time range and the traces plotted with ***the size of the plot box*** arguments. This section will also control the physical size of the plot in inches. You can set the left and right X axis limits and the inches per step value. Also the start and stop times along with the inches per second scale can be set. These parameters will limit the time window and the traces show. **A WARNING**: Though you have set the Axes limits and plot size here, some of the options can ignore these parameters and fit the computer screen. I have not figured out what is happening here but it works properly the bulk of the time. This is a clear example of why to use the traces only option until you tune your parameters a bit.

SCALING of the Data

There are several options for scaling the data. All three of these options multiply a factor times the raw data *passed from the FINDIF program*:

The ***TRUE*** Amplitude scaling factor is simply a number that is multiplied times the data.

The ***Equal Maximum Value Amplitude*** option will scale each trace so that it's largest value is the size of the Amplitude scaling value in inches. This is done regardless of any range related effects there are in the raw data.

The ***Range Weighted Amplitude*** option applies a scale factor relative to the range. This option will bring up two ***Range Weighting*** parameters. The range value is divided by the ***Ro*** value. The result of this division has the exponent of ***Alpha*** applied to it. And then all of this is multiplied by the ***Amplitude Scaling factor***. This option only has the range weighting applied to ranges greater than ***Ro***.

REDUCTION VELOCITY

You can also apply a ***Reduction Velocity*** to the trace. The number used here is in Kilometers/Second. The value entered here is divided into the range value of the trace. The range value is divided by the Reduction Velocity. This result is the number of seconds to shift the trace downwards. You must adjust the start and stop times in seconds accordingly for this option. Enter 0 for this option not to be used.

CLIPPING

Since some traces will have some very large values in sections you are not concentrating on, you may use a clipping value to limit the size of the trace. This value is in inches. Any part of a trace larger than this value or less than the negative of this value is set to those limits. This can be used bring out fainter sections of the trace. To not use the ***Clipping*** option you must set it to 0.0 to disable it.

EXTRA TITLE

The plot will have the name of the file plotted used as a title. You may add text in the ***Extra Title*** box to use in the title before the file name is used.

New SAMPLING RATE

It is possible that the FINDIF code may not pass the right sampling rate in seconds in the Time Series file. At present the ***New Sampling rate time period*** is **NOT** operational.

AXES LABELS

Some of X and Y axes labels are automatically selected by some of the options you have selected but some are not. You may set those labels in the ***X axis label*** and ***Y axis label*** pull down menus. You should double check that these labels reflect the options you have selected. If you wish any different labels you may use the Matlab figure tool to change the axes labels.

CREATE a TIME SERIES PLOT

When you select the ***PLOT the TimeSeries*** button the Time Series plotting section title will turn red and say it is reading the data. Once the data is read in, then this will become white again. The time series file remains loaded into the program until you toggle the ***Data file Extent*** pull down menu or the ***Select Data Directory*** selection button. It takes a bit of time to read the data in but once it is read in the plots come up pretty quickly, since the data file does not need to be re-read.

The Time Series plot comes up in it's own Matlab figure window. You may manipulate this figure like any other Matlab figure. I suggest that you not keep too many figure windows open so that both the computer and Matlab do not lock up.

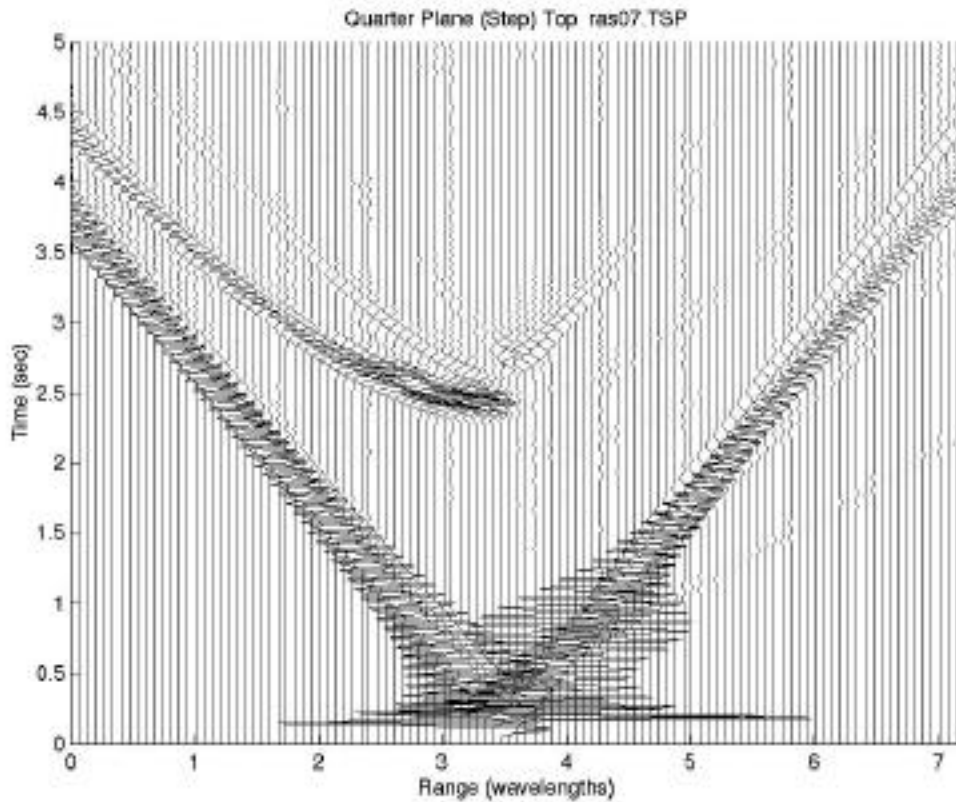


Figure 10a. An Example of plotting only the traces in the top set of receivers in the TSP file.

Parameters used for the above plot are:

Data Extent:	<i>TSP</i>		
Start Trace Number:	<i>5</i>	End Trace Number:	<i>545</i>
Number traces/step:	<i>5</i>		
<i>Find Receivers by Trace Number</i>			
<i>Plot Versus Range</i>		<i>Plot Traces Only</i>	
Divide M by:	<i>150</i>		
Range: Left:	<i>0</i>	Right:	<i>7.2</i> inches per value: <i>1.5</i>
Time: Bottom:	<i>0</i>	Top:	<i>5</i> inches per value: <i>1.5</i>
Amplitude Scaling:	<i>0.0000002</i>		<i>True Amplitude</i>
Reduction Velocity:	<i>0</i>	Clipping:	<i>3</i>
Extra Title:	<i>Quarter Plane (Step) Top</i>		
New Sample Rate:	<i>0</i>		
X axis Label: <i>Range (wavelengths)</i> Time axis label: <i>Time (sec)</i>			



Figure 10b. An Example of plotting only the traces in the bottom set of receivers in the TSP file.

Parameters used for the above plot are:

Data Extent:	<i>TSP</i>		
Start Trace Number:	<i>546</i>	End Trace Number:	<i>1081</i>
Number traces/step:	<i>5</i>		
<i>Find Receivers by Trace Number</i>			
<i>Plot Versus Range</i>		<i>Plot Traces Only</i>	
Divide M by:	<i>150</i>		
Range: Left:	<i>0</i>	Right:	<i>7.2</i> inches per value: <i>1.5</i>
Time: Bottom:	<i>0</i>	Top:	<i>5</i> inches per value: <i>1.5</i>
Amplitude Scaling:	<i>0.0000002</i>		<i>True Amplitude</i>
Reduction Velocity:	<i>0</i>	Clipping:	<i>3</i>
Extra Title:	<i>Quarter Plane (Step) Bottom</i>		
New Sample Rate:	<i>0</i>		
X axis Label: <i>Range (wavelengths)</i> Time axis label: <i>Time (sec)</i>			

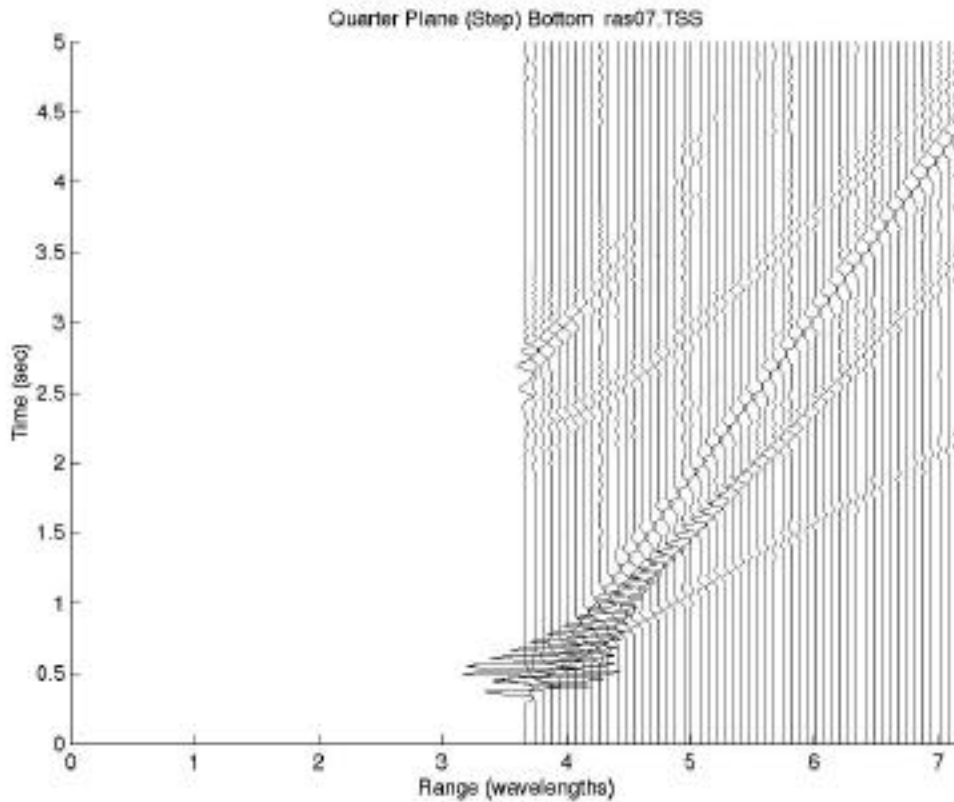


Figure 10c. An Example of plotting only the traces in the bottom set of receivers in the TSS file.

Parameters used for the above plot are:

Data Extent:	TSS		
Start Trace Number:	546	End Trace Number:	1081
Number traces/step:	5		
<i>Find Receivers by Trace Number</i>			
<i>Plot Versus Range</i>		<i>Plot Traces Only</i>	
Divide M by:	150		
Range: Left:	0	Right:	7.2 inches per value: 1.5
Time: Bottom:	0	Top:	5 inches per value: 1.5
Amplitude Scaling:	0.0000002		True Amplitude
Reduction Velocity:	0	Clipping:	3
Extra Title:	<i>Quarter Plane (Step) Bottom</i>		
New Sample Rate:	0		
X axis Label: <i>Range (wavelengths)</i> Time axis label: <i>Time (sec)</i>			

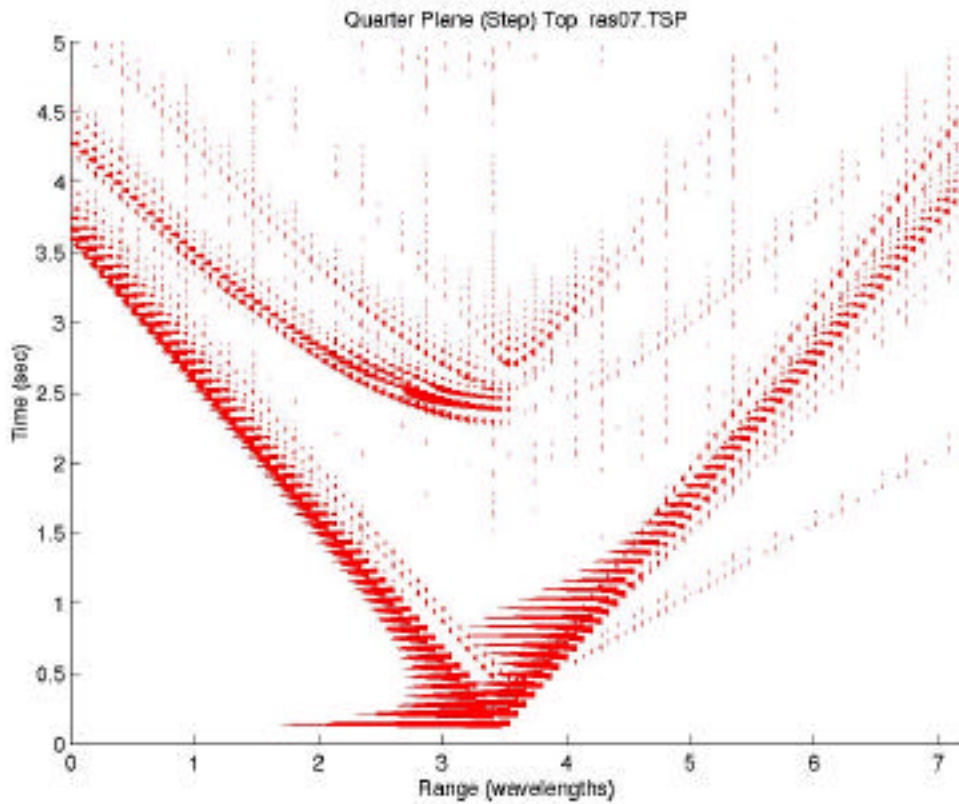


Figure 11. An example of filling only the Negative section of the trace.

Parameters used for the above plot are:

Data Extent: *TSP*
Start Trace Number: *5* **End Trace Number:** *545*
Number traces/step: *5*
Find Receivers by Trace Number
Plot Versus Range **Fill Negative Wiggles**
Divide M by: *150*
Range: Left: *0* **Right:** *7.2* **inches per value:** *1.5*
Time: Bottom: *0* **Top:** *5* **inches per value:** *1.5*
Amplitude Scaling: *0.0000002* **True Amplitude**
Reduction Velocity: *0* **Clipping:** *3*
Extra Title: *Quarter Plane (Step) Top*
New Sample Rate: *0*
X axis Label: *Range (wavelengths)* **Time axis label:** *Time (sec)*

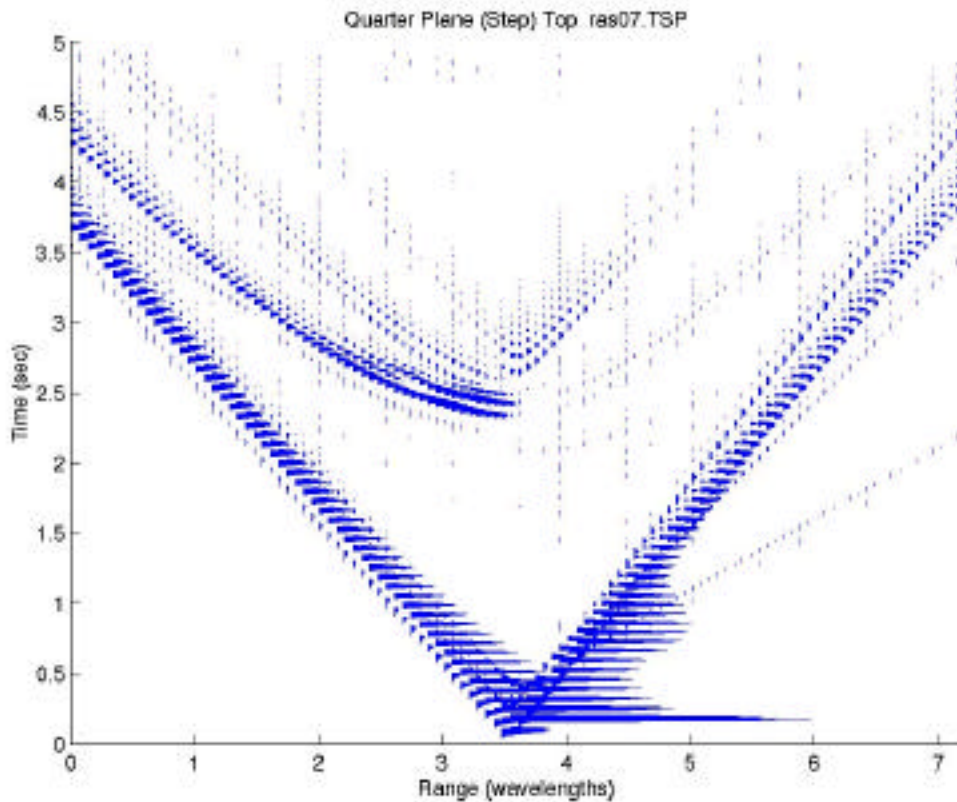


Figure 12. An example of filling only the positive section of the trace.

Parameters used for the above plot are:

Data Extent: *TSP*
Start Trace Number: *5* **End Trace Number:** *545*
Number traces/step: *5*
Find Receivers by Trace Number
Plot Versus Range *Fill Positive Wiggles*
Divide M by: *150*
Range: Left: *0* **Right:** *7.2* **inches per value:** *1.5*
Time: Bottom: *0* **Top:** *5* **inches per value:** *1.5*
Amplitude Scaling: *0.0000002* **True Amplitude**
Reduction Velocity: *0* **Clipping:** *3*
Extra Title: *Quarter Plane (Step) Top*
New Sample Rate: *0*
X axis Label: *Range (wavelengths)* **Time axis label:** *Time (sec)*

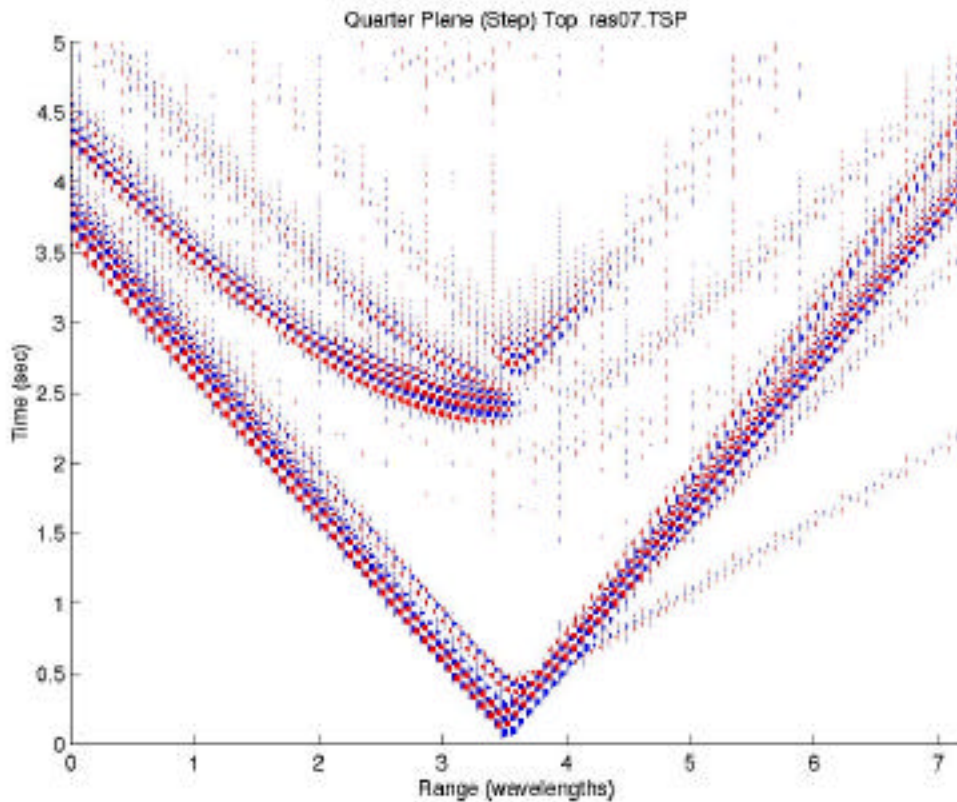


Figure 13. An example of filling both the negative and positive sections of the trace.

Parameters used for the above plot are:

Data Extent:	<i>TSP</i>		
Start Trace Number:	<i>5</i>	End Trace Number:	<i>545</i>
Number traces/step:	<i>5</i>		
<i>Find Receivers by Trace Number</i>			
<i>Plot Versus Range</i>		<i>Fill Positive and Negative Wiggles</i>	
Divide M by:	<i>150</i>		
Range: Left:	<i>0</i>	Right:	<i>7.2</i> inches per value: <i>1.5</i>
Time: Bottom:	<i>0</i>	Top:	<i>5</i> inches per value: <i>1.5</i>
Amplitude Scaling:	<i>0.0000002</i>		<i>True Amplitude</i>
Reduction Velocity:	<i>0</i>	Clipping:	<i>0.03</i>
Extra Title:	<i>Quarter Plane (Step) Top</i>		
New Sample Rate:	<i>0</i>		
X axis Label: <i>Range (wavelengths)</i> Time axis label: <i>Time (sec)</i>			

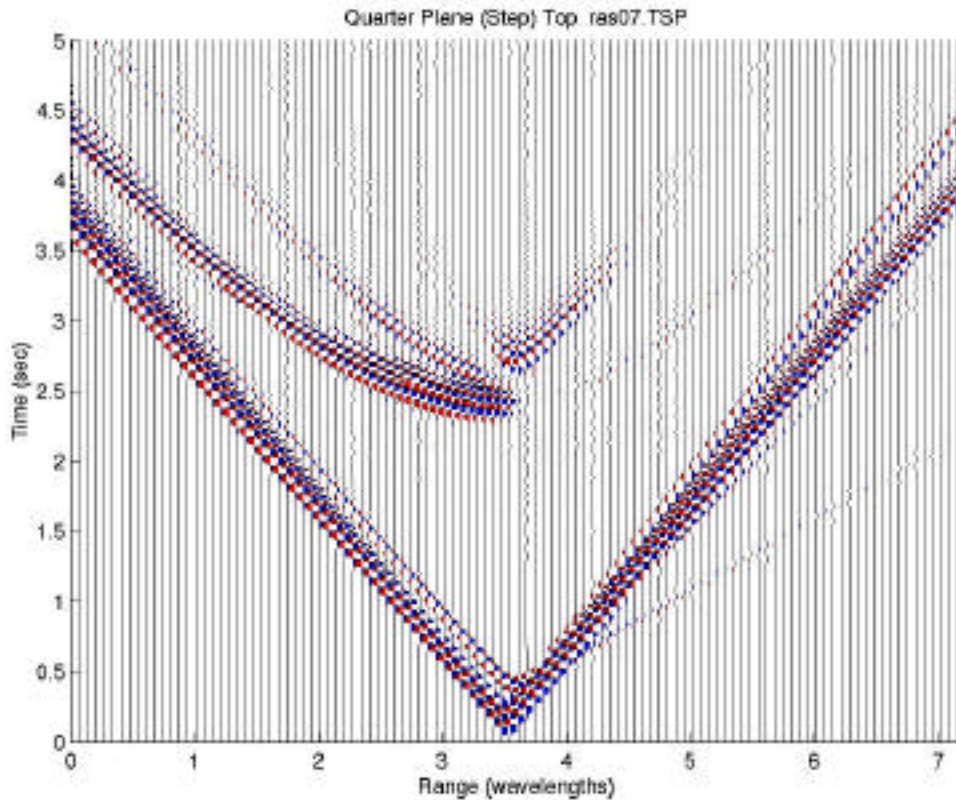


Figure 14. An example of filling the trace and showing the trace also.

Parameters used for the above plot are:

Data Extent:	<i>TSP</i>		
Start Trace Number:	<i>5</i>	End Trace Number:	<i>545</i>
Number traces/step:	<i>5</i>		
<i>Find Receivers by Trace Number</i>			
<i>Plot Versus Range</i>		<i>Fill Wiggles and Traces</i>	
Divide M by:	<i>150</i>		
Range: Left:	<i>0</i>	Right:	<i>7.2</i> inches per value: <i>1.5</i>
Time: Bottom:	<i>0</i>	Top:	<i>5</i> inches per value: <i>1.5</i>
Amplitude Scaling:	<i>0.0000002</i>		<i>True Amplitude</i>
Reduction Velocity:	<i>0</i>	Clipping:	<i>0.03</i>
Extra Title:	<i>Quarter Plane (Step) Top</i>		
New Sample Rate:	<i>0</i>		
X axis Label: <i>Range (wavelengths)</i> Time axis label: <i>Time (sec)</i>			

List of Matlab m-files needed to run the function.

REFRESH_TSPfind_1.m
REFRESH_plotsnap_3.m
Refresh_IPLOT2.m
edit_title.m
getTSPlist_1.m
get_EXTENT_1.m
get_TSPdir.m
get_dir.m
get_dir_1.m
get_rangetitle_1.m
get_savename.m
get_step_1.m
get_title_1.m
get_titlefile.m
get_titlenames.m
get_titles_1.m
getsnplist_1.m
getsnplist_2.m
listadir.m
plot_findif_1.m
plot_lfazo_2.m
plot_model_box_1.m
plotsnap14.m
relabel.m
relabelsnap.m
run_plot_find_6.m
samefile.m
save_title.m
seismiccolors.dat
start_all1.m
tomuilayout.m
warningw.m