User's Manual
SeisOpt® @2D™ Version 5.0

For support contact support@optimsoftware.com

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

Phone: 775.784.6613 * Fax: 775.784.1833

CONTENTS

0.0 Installing SeisOpt @2D

Begin the installation of SeisOpt @2D 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 1a.

Figure 1a: 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. Select ‘Yes, Launch the program file’ and then click 'Finish' to complete the installation procedure (Figure 1b). Ignore the message that appears (Figure 1c) and click ‘OK’.

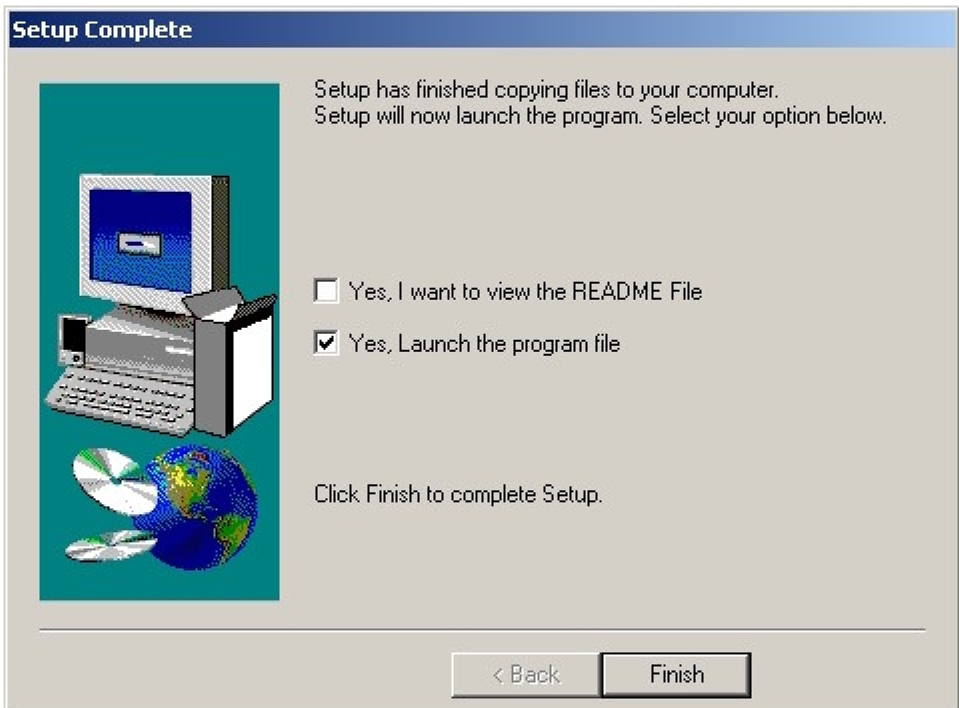


Figure 1b: Select ‘Yes, launch the program file’ and then click on ‘Finish’ to complete the installation.

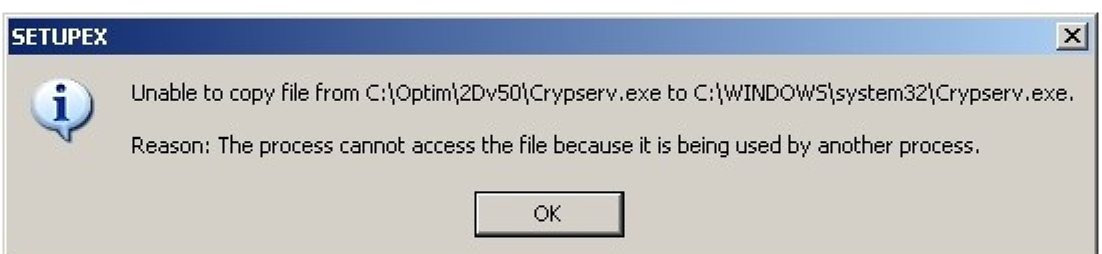


Figure 1c: Ignore the message and click ‘OK’.

The README file gives you directions on how to obtain your license from Optim to run SeisOpt @2D. The installation directory is **C:\Optim\2Dv50**. A successful installation will create a SeisOpt @2D desktop icon as shown in Figure 1d.

Figure 1d Desktop icon for SeisOpt@2D

SeisOpt @2D is protected either by encryption software or by a USB dongle.

To authorize the software protected version, do the following:

1. Click on either the SeisOpt @2D icon on the desktop.
2. When the license configuration window appears (Figure 1e), a **Site Code** should be displayed, as shown in Figure 1e. If not, click the **'Display Site Code'** button.
3. Send this site code to Optim, by email (support@optimsoftware.com). This same **Site Code** will reappear each time the license configuration window is opened until a **Site Key** is entered.
4. 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'**.
5. Each time you run SeisOpt @2D, a window briefly appears (Figure 1f) describing the status of the license, then closes, and the program starts. DO NOT hit `ENTER/RETURN` when this window appears.
6. SeisOpt @2D will not run on any other computer, without obtaining another Site Key or without performing the license transfer as described in Section 9.2

Figure 1e: Email the 'Site Code to support@optimsoftware.com and a 'Site Key' will be returned.

Figure 1f: License check window appears each time SeisOpt @2D is run.

To authorize the USB dongle protected version, do the following:

1. In addition to the desktop icon, another a "Cbsetup" icon will be placed on the desktop
2. Click on the "CbSsetup" icon.
3. Follow the steps to install the CB drivers and update Windows registry. Make sure you choose USB for the type of dongle key (should be the default setting).
4. Now insert the provided USB dongle into the USB port.
5. The PC or laptop should detect the new hardware, and a Wizard will open up that will update the drivers.

1.0 SeisOpt @2D Overview

SeisOpt @2D is an automatic refraction interpretation package that contains modules for performing velocity optimization and visualization, virtual survey design and output report quality images. Seisopt@2D is the Graphical User Interface to the suite of modules for performing the velocity model optimization and visualization, interactive seismic survey design, and outputting postscript images of the results for printing.

SeisOpt@2D uses only the first-arrival travel times and the survey geometry to derive subsurface velocity information. For this reason, accurate picks are important. It uses a nonlinear optimization technique called adaptive simulated annealing and it involves forward modeling. Test velocity models are created, through which travel times are calculated. These calculated travel times are compared with the observed data. Testing every possible velocity model would take far too long, so SeisOpt@2D uses Optim's proprietary algorithm (visit www.optimsoftware.com to view and download peer-reviewed references) to search through only a small percentage of the many possible models, yet still find the best model. It is called an optimization because the discrepancy, or error, between the calculated and observed travel times is optimized. In this case the optimal solution is the velocity model with the minimum travel-time error.

Within SeisOpt@2D, the velocity models are represented as discrete square or rectangular cells. The term resolution is used to refer to the relative number of cells used to represent a given model. A low- resolution model would have a relatively low number of large cells, compared to a high-resolution model, which would be comprised of a large number of small cells. In SeisOpt@2D, the resolution can be adjusted. In general, the factors that determine the appropriate resolution are the receiver spacing and the size of the target subsurface features. SeisOpt@2D comes with several preset resolutions that are based on the receiver spacing of the current data set. Optionally, any resolution can be specified. However, increasing the resolution beyond the highest preset can cause artifacts, or false velocity features, in the final velocity model. Running the optimization on the same data with different resolutions and with different depth ranges at the same resolution of the velocity model can increase confidence in the results. Features that are present in models with different resolutions can be assumed to be real with much greater confidence.

For each velocity model optimization, SeisOpt@2D produces a file showing the number of times each cell in the model was sampled. This determines which parts of the velocity model are actually controlled by the data. Any gaps in the sampling at the end of the model correspond to unconstrained areas of the velocity model. Therefore, this sampling information should be considered before interpreting the optimized velocity model.

1.1 Why Use SeisOpt @2D?

SeisOpt @2D is refraction velocity optimization software. The only input it requires is the first arrival picks and array geometry, making it an ideal tool to use in an area where very little information is available about the subsurface velocity structure. It can handle irregular topography and does not require any elevation correction of the data before analyses. SeisOpt does not require constant user interaction and can be set up to run several jobs in batches (see Section 4.6). This leads to savings in time and money during processing. SeisOpt @2D predicts subsurface morphology, maps strong lateral velocity variations, outputs a map of the subsurface ray coverage, contains tools to visualize and analyze the results, quantitatively compare the data (picks) and model, and interactively or automatically optimize the array geometry for sampling the desired subsurface target. If used in the field during data acquisition, SeisOpt @2D can lead to savings by enabling the user to determine if they are sampling to desired target depths and thus avoid costly redeployment costs. Its MakeEPS™ module allows the user to manipulate the output images and produce report quality color output. The files can be imported into drawing programs like Adobe Illustrator and Corel Draw (7.0 or less) or viewed and printed using the freeware programs GhostScript and GSView (can be downloaded from www.cs.wisc.edu/~ghost/, see Section 7.2). SeisOpt @2D also produces ASCII files of the velocity models and 'blanking' files that can be imported into contouring programs like Surfer (Section 7.3). The "tuning" module can be used to fine-tune the final optimized velocity model as desired (see Section 8.0).

1.1.1 Capabilities of SeisOpt @2D

SeisOpt @2D will allow you to process up to 28 shots and unlimited receivers, deployed along the surface. It is possible to increase your capabilities for processing seismic data, beyond 28 shots, by either getting SeisOpt® Pro™.

Use SeisOpt @2D if you need to:

- Process up to 28 shots and unlimited number of receivers deployed along the surface
- Interactively and automatically design your survey, in the field or in the office
- Make report quality output
- Tune and build models, for analysis or bidding
- Set velocity limits to constrain your model
- Re-optimize models and increase resolution
- Determine velocities over length of entire line, including underneath offend shots
- Export ASCII velocity values for import into contouring programs like Surfer™
- Determine interface boundaries through the gradient model

1.1.2 Upgrade features for SeisOpt @2D v5.0

SeisOpt @2D v5.0 contains the following upgrade features:

- Faster optimization scheme: Implementation of adaptive simulated annealing that leads to faster convergence of the optimization process. For similar data sets the new optimization scheme is at least twice as fast as the one used in earlier versions.
- Advanced fine-tuning features: Version 5.0 allows user to draw polygons of any shape and change the velocity within it. In addition to the gradient and constant velocity changes, the velocities within the area can also be averaged. This enables smoothing of isolated high or low velocity cells within the model.
- Variable grid optimization: The new version allows the user to re-run the final optimized model using a variable grid mesh. The user needs to first run the model using a uniform grid and then use that as an input to a variable grid run, where the grid sizes can change either horizontally or in depth. This feature is useful when dealing with long source-receiver offsets for off-end shots or for increasing the resolution (number of grids) at shallow depths.
- Editing Pickfiles: The Pickfile produced from an optimization run can be loaded into the 'Pickfile' display and bad picks either deleted or moved to their correct position. The edits are recorded and new input files (rec, obs, and src files) are created that can be used for a new run.
- MakeEPS module upgrade: The user can specify velocity ranges when now creating encapsulated PostScript (EPS) images of the velocity models using the MakeEPS module. The resulting model will have specific colors assigned to specific velocity ranges, thus creating a layered model effect.
- Make JPEG: The 'Make JPEG' button on each of the Velfile, Hitfile, Pickfile and Surveyfile displays allows the user to create JPEG images of the displayed files directly.

1.2 Things to Remember When Using SeisOpt @2D

The speed of processing data using SeisOpt @2D is proportional to the speed of the processor in the user's computer. It is highly recommended that the software be installed on a computer having at least a Pentium III processor, or its equivalent. The speed of processing is also proportional to the size of the velocity model and the number of sources in the survey. The model size is determined by the following factors: resolution setting (Higher the resolution, smaller the grid spacing, and hence, larger the model) and length of the seismic array (longer the array, larger the model). The software produces the best results when run at the 'High' or 'Highest' resolution settings. These factors make it practical to use SeisOpt @2D for processing data from surveys containing up to 28 shots, depending on user tolerance. Sampling density is another important factor that controls the robustness of the results obtained from SeisOpt @2D. Increasing the number of sources and receivers will result in better the sampling and thus better the result. As a rule of thumb, the software resolves structures with dimensions about half the receiver spacing, but this can vary depending on the presence or absence of strong lateral and vertical velocity variations. SeisOpt @2D, unlike traditional refraction software, images velocity gradients in the subsurface. The software will introduce an appropriate gradient between horizons defined by discrete velocities. The interface between layers is the depth at which the gradient change is the steepest. In general, this will be shallower than the actual depth at which the layer velocity is encountered in the SeisOpt @2D image. This should be taken into account when comparing @2D results with results from methods that produce ‘‘layer’’ models. For example, the interface between layers that have velocities of 500 ft/s and 2000 ft/s, will not be at 2000 ft/s, but rather shallower where velocities are about 1,250-1,500 ft/s. As often the case in reality SeisOpt @2D reveals subsurface velocities as gradients and not solid layers. The software allows the user to compute the depth (or elevation) at which the interfaces between layers occur. Given the layer velocities, the ‘DetLayer’ (Section 6.0) module will compute the interface depth between each pair of layers. This can then be plotted on top of the velocity model using the ‘MakeEPS’ module or using Surfer™.

1.3 Optimal Survey Parameters

As described in the above section, SeisOpt @2D produces the best results when the sampling density of the subsurface is high. As a rule of thumb, the software, when run at high or highest resolution setting, can resolve structures that have dimensions of about half the receiver (geophone) spacing. For resolving depths accurately, there must be at least 7 to 10 grids (or cells) to the estimated target depth. For example, if the estimated depth to basement is 10 m, SeisOpt @2D must be run with a grid spacing of 1m, which means that the receiver spacing must be about 2 m. More the number of sources within the receiver array, better the model. A minimum of 7 sources is recommended for high quality results.

2.0 Input Data Format

The input for SeisOpt@2D must be in a certain format and they can be created by:

- Using SeisOptPicker™
- Conversion Programs
- Manually

2.1 Using SeisOptPicker to generate SeisOpt@2D input

SeisOptPicker is Optim's automatic and manual first-break picking module. It can read in field data recorded in several formats like SEG-Y, SEG-2, SEG-D, and ABEM Terraloc and perform an automatic or manual first break picking. In addition, it has modules for geometry entry, header manipulation, filtering and AGC, and creating report quality output files. The user can view, edit and export the picks, along with the survey information, to files that can directly read into SeisOpt @2D. Contact Optim for more information about this module.

2.2 Using a conversion program to generate SeisOpt@2D input

Programs to convert the output from some picking programs into the format required by SeisOpt@2D are available. The graphical user interfaces includes the following programs for extracting input files needed for SeisOpt @2D from the following common picking modules (Figure 2):

- **sip2opt**: Extracts input files for SeisOpt @2D from .SIP files

NOTE: There are several flavors of SIP files that are written out depending on the SIPx edit file you use. So, when using sip2opt please review the “src” and “rec” files that are written out. Make sure the source and receiver coordinates match the original. If not, send the *.sip file to support@optimsoftware.com **and you will be sent the proper sip2opt conversion program.**

- **vs2opt**: Extracts input files for SeisOpt @2D from ViewSeis .PRN files

- **grm2opt**: Extracts input files for SeisOpt @2D from Gremix GRM files
- **imager2opt**: Extracts input files for SeisOpt @2D from Seisimager output files
- **terra2opt**, a conversion program that converts ABEM Terraloc MK6 *.fir files, is also provided, but not included as part of the graphical interface. To use this, the user has to open up an MS-DOS window and type terra2opt.

To use any one of the above conversion programs, click on the appropriate button. A dialog window, with a 'Browse' button will open up. Choose the appropriate input file (.SIP, .GRM or .PRN file). And click OK to execute the conversion. The 'Progress Window' will indicate the location of the three input files needed to run SeisOpt @2D. It will be written to the directory that contained the input (.SIP, .GRM. or .PRN) file. Also the parameter file called riotsinput (see Section 4.1), which can be read in modified using 'RIOTS Settings' (see Section 4.1), will be written to the SeisOpt install directory.

2.3 Format of input data files

The input to SeisOpt@2D consists of 3 ASCII (text) files. One contains the source location information, one contains the receiver location information, and one contains the first-arrival pick times. The format of these files is as follows. If you have used a conversion program to generate these files, they should already be in the correct format. If you are creating these files manually, follow the specifications exactly.

The **source information file** has three columns separated by white space: *Column 1*: Relative or absolute distance of each source along the horizontal direction in physical units (ft, m, and km). These can be either distance measured along the ground (tape measure) or projected horizontal distances that take into account the elevation along the profile. The appropriate option should be chosen in the 'RIOTS Settings' window. *Column 2*: Elevation (z) of each source in physical units. Note that elevation decreases with depth. The top of the model has to have the maximum elevation. *Column 3*: Number of picks (or number of recording receivers) associated with each source.

The **receiver information file** has two columns that should be separated by white space: *Column1*: Relative or absolute Distance of each receiver in the X (horizontal) direction in physical units (ft, m, and km). These can be either distance measured along the ground (tape measure) or projected horizontal distances that take into account the elevation along the profile. The appropriate option should be chosen in the 'RIOTS Settings' window. They should be consisted with units entered in the source information file. *Column 2*: Elevation of receivers (geophones) in the same units as above. Note that elevation decreases with depth. The top of the model has to have the maximum elevation.
The number of entries in this file must equal the sum of the entries in the 3rd column of the **source information file**. That is, for each source, one must enter the co-ordinates of all the recording receivers (there must be a pick associated with it), even if receivers repeat from one source to the next.

The **pick times** (observations) **file** consists of one column, with one pick on each line. These times can be in seconds or milliseconds (specify the units in the input settings file, **Section 4.1**). The number of entries in this file should be the same as the number of entries in the **receiver information file.**

Important: The units and order of the information in each file must be consistent between all files. The units must be specified in the input settings file (**Section 4.1**). The order of the pick times must follow the order of the receiver locations in the receiver file, and these, in turn, must follow the order of the sources in the source file. For the input to be consistent, the sum of the numbers of picks (or recording receivers) per source (the 3rd column of the source file) must be the same as the total number of receivers in the receiver file and the total number of picks in the pick file.

3.0 The SeisOpt@2D User Interface

Double-clicking on the SeisOpt@2D desktop icon (Figure 1) brings up the SeisOpt@2D user interface (Figure 2c). But before this, a browse window comes up (Figure 2a) allows user to load in the three input files (obs, rec, src). This feature is for editing the Pickfiles generated from the optimization run (Section 5.3.3.2). If Pickfiles are not going to be edited (as would be the case when making the first run), simple click "Cancel". Window shown in Figure 2b appears. Click "OK". And the main graphical user interface shown in Figure 2c opens up. The main display window is blank, showing the phrase 'No data', until files from a velocity optimization are read in and displayed.

It includes buttons for velocity optimization (red 'RIOTS' button), setting the input parameter file for running RIOTS ('RIOTS Settings'), calculating layer interfaces given the layer velocities (green 'DetLayer' button), setting up batch file for batch processing ('Set up Batch File' and the red 'Run RIOTS Batch File' button), button for making encapsulated PostScript (EPS) files ('MakeEPS'), and buttons for converting output files from SIP, GREMIX, and ViewSeis to input for SeisOpt @2D. In addition it has buttons for screen dump ('Make JPEG') and tuning the velocity model (see Section 8.0).

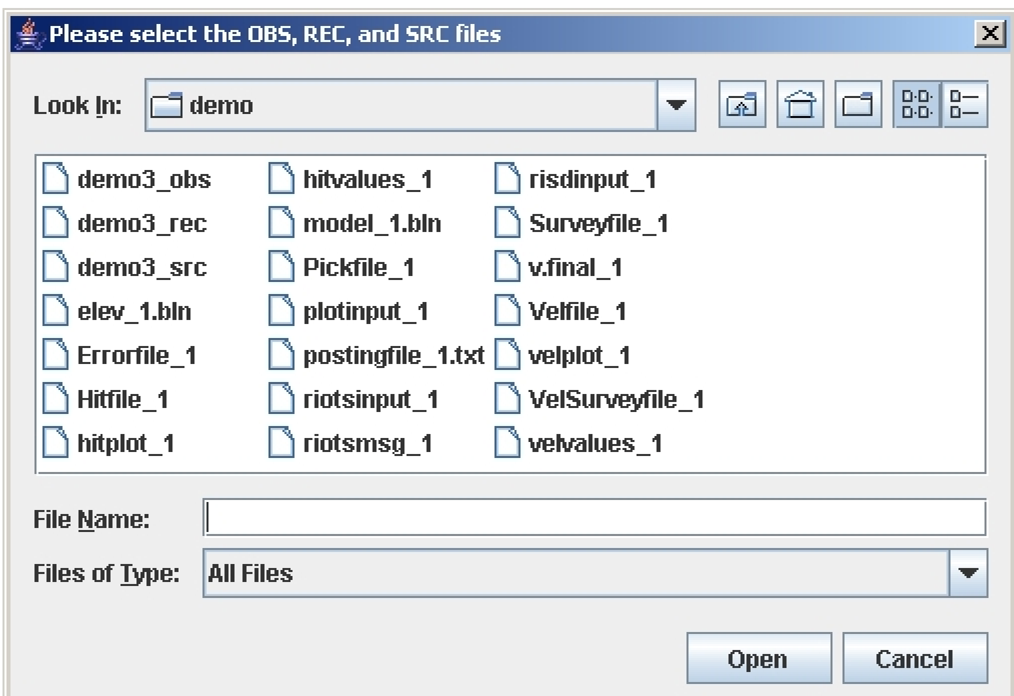


Figure 2a: Browse window that allows user to read in the input files for SeisOpt @2D runs. The files need to be selected only if the Pickfiles are going to be edited. If not click "Cancel" and the window shown in Figure 2b will appear.

Figure 2b: Click 'OK' if the Pickfiles are not going to be modified.

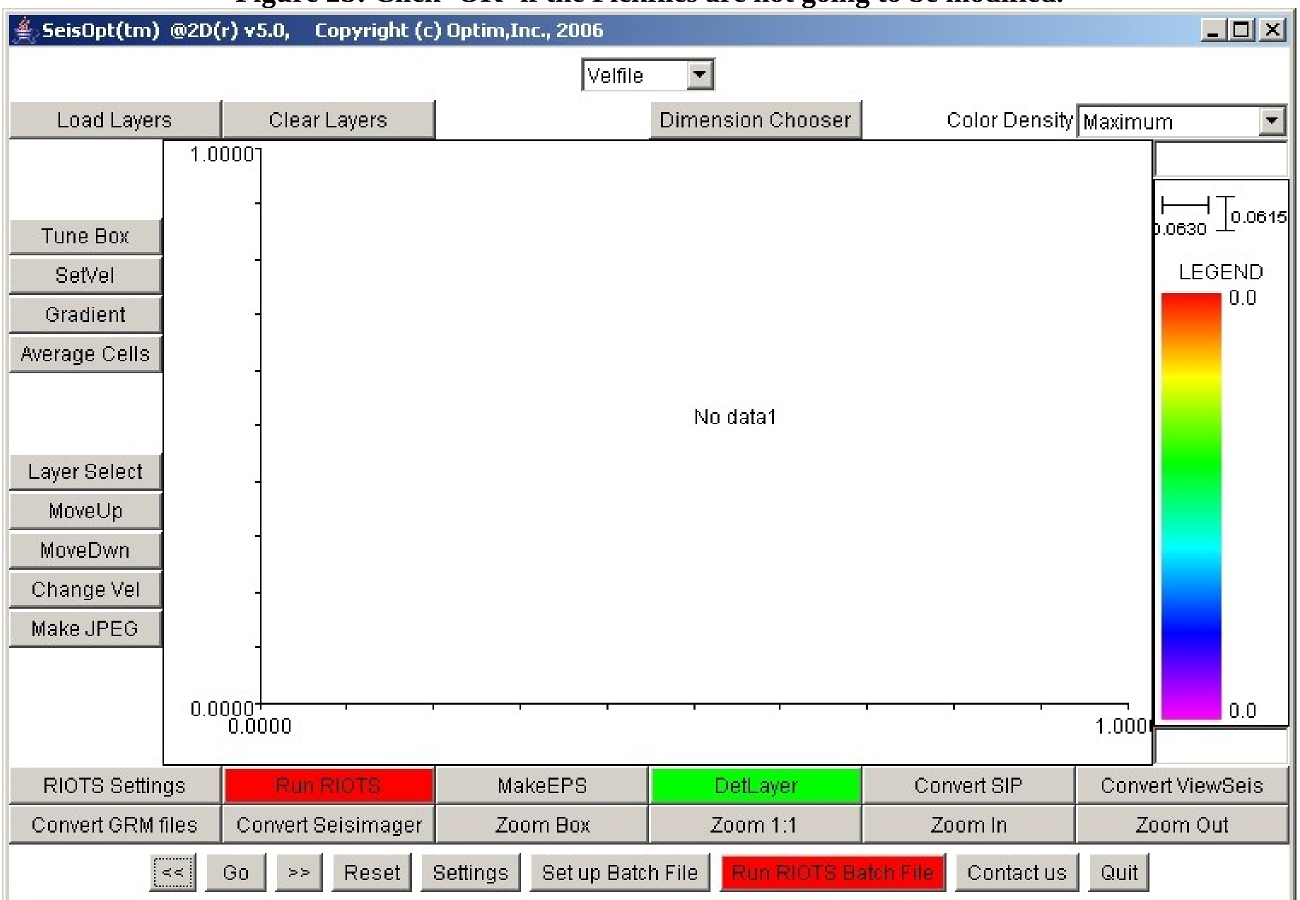


Figure 2c: SeisOpt@2D user interface

4.0 Invoking RIOTS, the Velocity Optimization Module

The velocity optimization module is named RIOTS™, for Refraction Inversion OpTimization. The first step in the velocity model optimization is setting up the input parameters. To do this, click on the 'RIOTS Settings' button. This brings up the Riots Settings window, shown in Figure 3. If using a conversion program provided by Optim to convert output from a picking program to SeisOpt@2D format, see the **Section 2.1**. The conversion program will set these values automatically. Make sure to copy the file 'riotsinput' produced by the conversion program to the install directory (C:\Optim\2Dv50).

4.1 RIOTS input parameters: the riotsinput file produced by 'RIOTS Settings' window

Autocal: This check box toggles between manual and automatic selection of the resolution parameter. When this box is checked, then autocal is on and there are 5 preset levels of resolution to choose from (see 'Resolution' below). When autocal is off, nx, nz, and h must be set manually (see 'nx, nz, h' below for details).

Units: Choose between feet, meters, or kilometers from this pull-down menu for the distance units. These units should match the units used in the input files.

Time Units: Choose between seconds and milliseconds from this pull-down menu for the units of the picks in the pick file.

Source file: Enter the name of the source information input file, including the path, or click 'Browse…' to find the file.

Receiver's file: Enter the name of the receiver information file, including the path.

Picks file: Enter the name of the first arrival time picks file, including the path.

Resolution: This pull-down menu is active when autocal is on. Select between the 5 preset resolutions. The following preset resolutions are based on the receiver spacing of the input data:

Highest : hx = 0.375 * receiver spacing
hz = 0.1875 * receiver spacing
High : hx = hz = 0.375 * receiver spacing
Med : hx = hz = 0.5 * receiver spacing
Low : hx = hz = 1.0 * receiver spacing
Lowest : hx = hz = 2.0 * receiver spacing

nx, nz, hx,hz: These values are for manually selecting the model resolution, and become active when autocal is off. nx is the number of cells in the horizontal direction, and nz is the number of cells in depth. hx and hz is the dimension of the grids in horizontal (x) and vertical (z) direction, respectively, in physical coordinates (ft, m, or km - whichever units are selected in the units pull-down menu). Do not attempt to set these manually before running RIOTS at least once with autocal on, to determine the approximate range of appropriate values. See Section 4.4 for a description of this procedure.

Horizontal Distance: Select this option if the input source and receiver distances are horizontal distances. De-select (uncheck the box) this if the input distances are those measured along the ground and have **NOT** been converted (projected) to horizontal distances.

Set velocity bounds: Select this if you want to set the velocity limits for the RIOTS run. If not selected, the default velocity limits will be used.

Max. Velocity: Enter value for the desired maximum velocity during the RIOTS run. This is active only when the 'Set velocity bounds' option is selected.

Min. Velocity: Enter value for the desired minimum velocity during the RIOTS run. This is active only when the 'Set velocity bounds' option is selected.

Source count: The number of sources in the source input file.

Pick count: The total number of first arrival time picks.

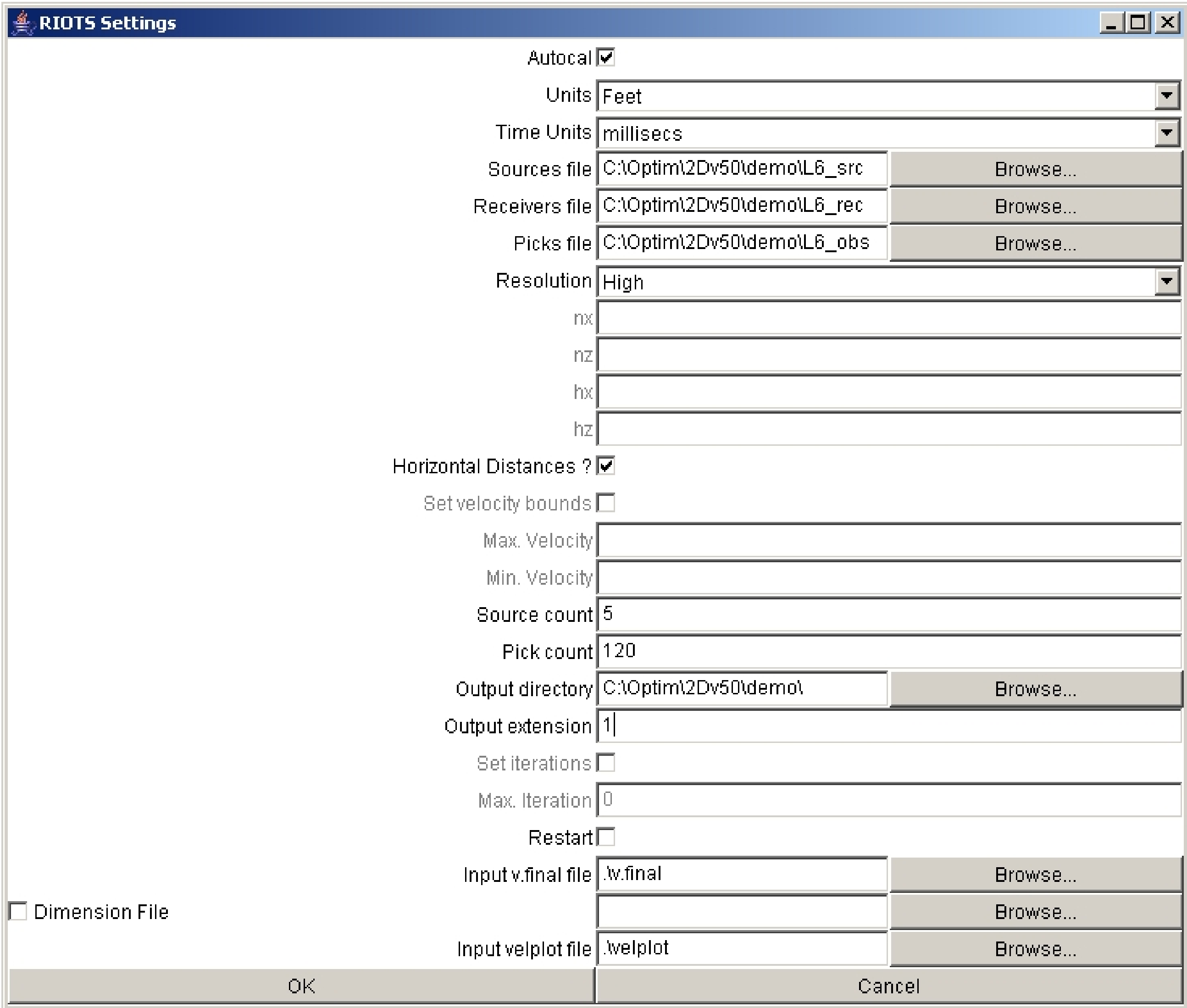


Figure 3: 'RIOTS Settings' window. Clicking 'OK' produces riotsinput - the input parameter file to run RIOTS

Output directory: Enter the path where output files from the optimization should be written.

Output extension: This value (or character string) is appended to the end of the optimization output file names. The purpose of this extension is to distinguish between the results from different runs. The default extension is a timestamp, but it can be set to anything. If this field is blank, 0 is used for the extension. Note that the extensions will be of the form '_[extension]'.

Set iterations: Choose this option, when you re-run the model after 'tuning'. You can set the maximum number of iterations for the RIOTS run.

Max. Iteration: Set the maximum number of iterations for the RIOTS run, if 'Set iterations' option (above) is selected. It should be an integer value and it is strongly recommended that this be used only when re-running RIOTS after tuning the model or with an input velocity model.

Restart: If this option is selected, one can re-run RIOTS with an input velocity model produced from a previous run or created by the user after tuning. If not selected, RIOTS runs with the default constant velocity input model.

Input v.final file: The name of the ASCII velocity file used by SeisOpt@2D during a RIOTS re-run. If the 'Restart' option (above) is selected, RIOTS reads in the velocity model contained in this file. If the velocity model is not in the installation folder (C:\Optim\2Dv50\) and is not called v.final, use the 'Browse' button and select the appropriate 'v.final' file. This file is also used during the 'tuning' process (see Section 8.0).

Input velplot file: The name of the 'velplot' velocity file used by SeisOpt@2D during a RIOTS run. If the 'Restart' option is selected, RIOTS reads in the parameters contained in this file. If the velplot file is not in the installation folder (C:\Optim\2Dv50\) and is not called velplot, use the 'Browse' button and select the appropriate velplot file. This file is also used during the 'tuning' process (see Section 8.0).

Dimension file: This file is contains information on the variable grid created using the 'Dimension Chooser' Module. The variable grid option is used for fine tuning a model run using uniform grid layout. The user can interactively change the horizontal and vertical dimensions of the grids anywhere within the model and re-run optimization using an input velocity model generated from the uniform grid run.

Clicking 'OK' in the 'RIOTS Settings' window saves the current settings of these parameters to a file called **riotsinput** which is saved to the C:\Optim\2dv50\ folder. This file can also be manually edited. The following list shows the contents of this file, for the 'RIOTS settings' window shown in Figure 3:

```
units=1
tunits=2
autocal=0
res=4
hdist=0
setvel=0
nPicks=120
nSources=5
obfil=C:\Optim\2DV50\demo\L6_obs
srcfil=C:\Optim\2DV50\demo\L6_src
recfil=C:\Optim\2DV50\demo\L6_rec
outdir=C:\Optim\2DV50\demo\
ext=1
setiter=0
restart=0
inmod=.\v.final
vinmod=.\velplot
```

For the purposes of editing this file manually (instead of using 'RIOTS Settings' to do it), use the following guide:

units: 1 = ft, 2 = meters, 3 = km
tunits: 1=seconds, 2=milliseconds
autocal: 0 = use presets, 1 = set manually

res: 1 to 5, where 1 = lowest, 5 = highest resolution
[if autocal = 1, replace res with nx,nz,h, each on a separate line; see above for details]
hdist: 0=input distances are horizontal distances, =1 input distances are ground distances
setvel: 0=use default velocity limits, 1=set velocity bounds
[if setvel=1, maxvel= maximum velocity in the model, minvel=minimum velocity]
nPicks: total number of time picks
nSources: number of sources
obfil: path and filename of pick times input file
srcfil: path and filename of source location input file
recfil: path and filename of receiver location input file
outdir: path of directory in which output from the optimization is to be written
ext: extension to append to output filenames. The extension will be of the form "_[ext]"
setiter: 0=use default number of iterations, 1=set number of iterations
[if setiter=1, set maxiter=maximum number of iterations for RIOTS run]
restart: 0=start with default constant velocity model, 1=restart with velocity model inmod]
inmod: name of ASCII velocity file (v.final). Use if restart chosen and in tuning (**Section 8.0**)
vinmod: name of 'velplot' file. Use if restart chosen

4.2 Starting the Optimization

Once the input parameters have been set in 'RIOTS settings', click the red 'Run RIOTS' button. This starts the velocity optimization and brings up a window that keeps track of its progress.

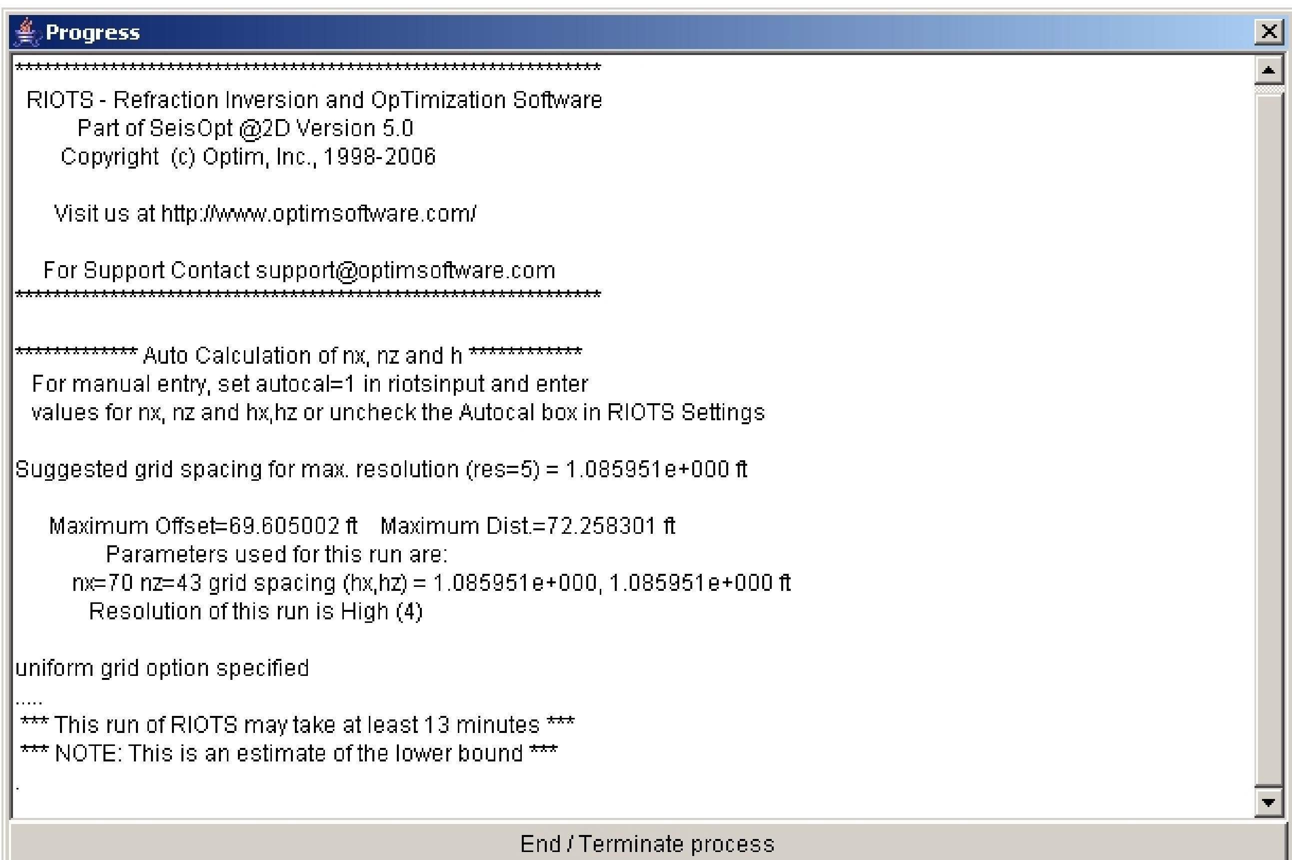


Figure 4: Beginning of Progress window during a sample RIOTS run

Figure 4 shows a progress window for a sample run. Clicking on the 'End / Terminate Process' button will stop the optimization. In order to terminate the optimization after it has been running for a while, it is best to use the ''Task Manager'' (invoked by pressing Ctrl-Alt-Del) and choose the process 'riots' for termination.

As RIOTS begins the optimization, it prints information regarding the resolution and the units of both the time and distance information. Make sure they are correct. In the example above, 'Autocal' is on, and the resolution is set to the 'High' value. The nx, nz, hx, and hz values can be used to determine appropriate values for manually setting the resolution (Section 4.4). As the optimization progresses dots slowly appear across the screen. This means the program is running properly. After about 30 seconds, an estimate of the total time running time of the optimization is displayed. As mentioned in the previous Section, it is strongly recommended that, at first, RIOTS be run with the default velocity bounds and for the default number of iterations. The velocity limits for the optimization can be set if it is determined that the model produced from the optimization run is vastly different the known maximum and minimum velocities expected in the region.

4.2.1 RIOTS running time

The total time to complete an optimization run can vary widely because it depends on several factors. These factors include the number of cells in the model, the number of sources (and to a lesser extent the number of receivers), the number of iterations needed to converge on a solution, and the processor-speed of the computer being used. The number of cells in the model is controlled by the resolution, but since the resolution is based on the receiver spacing (unless manually chosen), the relationship between the receiver spacing and the overall length of the survey will affect the total number of cells in the model. The resolution is the only factor affecting the running time under the user's control. The number of sources is obviously determined by the survey specifications, and not subject to change. The number of iterations of the optimization process is dependent on the behavior of the travel-time errors for a particular model, and cannot be controlled by the user (the dots that display during the optimization are related to the number of iterations). How fast a given computer will perform a certain optimization, with all other variables fixed, depends mainly on the clock speed of the processor (a number expressed in MHz). Therefore a Pentium-4 processor will complete the optimization approximately twice as fast as a Pentium-3. We recommend using at least a Pentium-3, but the only real drawback of using a slower processor is the increase in running time.

The estimate of the running time displayed in the progress window is designed to give the user an estimate of the time to complete the optimization. The reason it is an estimate is because the number of iterations required to converge is different for every model. If this time is very large, and only a preliminary result is needed, the optimization could be stopped and re-started with a lower resolution. To give an idea of actual running times that can be expected, high resolution models with about 100 time picks might take only a few minutes, while high resolution models with more than 1000 picks could take several hours.

When the optimization is finished, information appears regarding the results of the run, followed by the word 'Done', as shown in Figure 5. This means the optimization has been completed successfully. The minimum and maximum velocities present in the final velocity model are displayed. The resolution parameters are again displayed along with the least-square error between the observed first arrival (picks) and times through the final, optimized velocity model. This information is useful for quantitatively evaluating the nature of the fits and choosing the best model from several @2D runs, on the same data set (Section 4.5). Finally, the location of the output files is given, along with the chosen output extension.

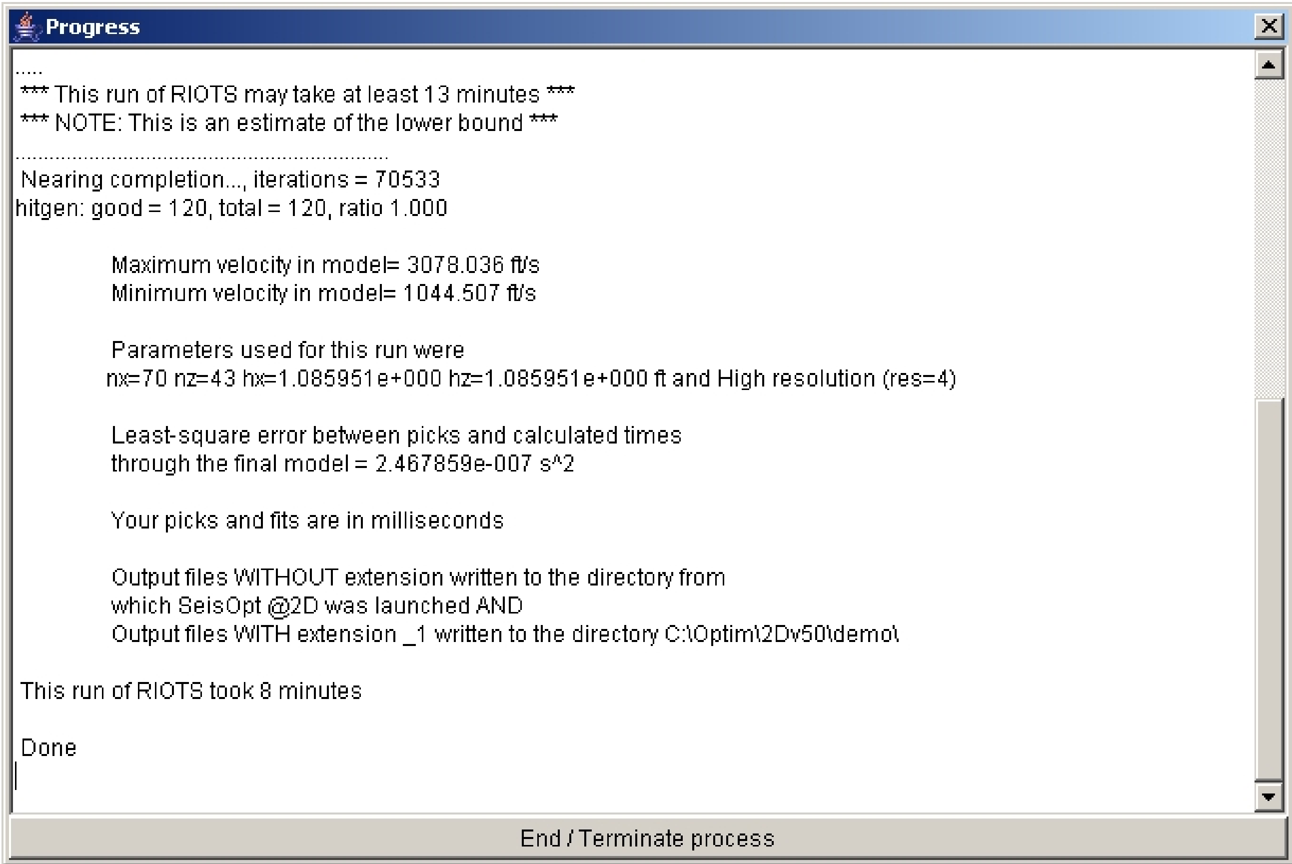


Figure 5: End of Progress window during a sample RIOTS run

Once one RIOTS run is done, you can re-run it by using the 'Restart' option or run another run by setting the velocity bounds and selecting the appropriate 'v.final' and 'velplot' files.

4.3 Output from RIOTS

Upon completion of the optimization, RIOTS writes out several files to the output directory specified in RIOTS settings. **<u>The file extension chosen in RIOTS settings is appended to the file names listed below</u>**. SeisOpt@2D uses the first six files listed below for displaying the results of the optimization. The rest are either ASCII text versions of the optimization output or for internal use by SeisOpt@2D.

Velfile: Contains final velocity model.

Hitfile: Ray coverage of the final velocity model. It is similar to the Velfile, but instead of velocity values for each cell, it contains the number of times each cell was sampled. Areas that are not sampled at all are unconstrained and zeroed out in the velocity model.

Pickfile: Contains the observed picks and the picks calculated from the final velocity model, for visually inspecting the quality of the fit. The observed picks can be edited.

Surveyfile: This file contains the Hitfile information, as well as the source and receiver geometry for use with the interactive survey design feature of SeisOpt@2D.

VelSurveyfile: It is the same as Velfile, but with source and receiver location information as well.

Errorfile: Contains the least-square error between the picks and the first arrival times through the models as a function of the iterations of the optimization process. The last value in this file is the least-square error of the final velocity model. Lower this error, tighter the fit and, in general, better the final velocity model.

riotsinput: This text file is created by riots settings, and contains all the parameters used in the optimization.

riotsmsg: Contains the messages displayed in the progress window during the optimization process. This file is overwritten out with the specified extension only to the output directory specified in 'RIOTS Settings'.

velvalues: An ASCII (text) version of the final velocity model in 3-column format. (x, z, velocity). This can be imported into contouring programs, like Surfer™, for rendering the results in a different format.

hitvalues: An ASCII (text) version of the final hit count model in 3-column format. (x, z, hits). This can be imported into contouring programs, like Surfer™, for rendering the results in a different format.

velplot/hitplot: velocity and hit files used by the encapsulated PostScript (MakeEPS module) output feature of SeisOpt@2D. This file is also need when using the 'restart' option.

v.final: File used by SeisOpt@2D during the 'restart' option. Changes to the velocity model during g the tuning process is also saved in this file.

risdinput: File used by SeisOpt@2D during the interactive and automatic survey design process

plotinput: The EPS plotting module, MakeEPS™ uses this file to create encapsulated PostScript files.

elev.bln: "Blanking file" containing the elevation coordinates. This file can be imported into Surfer™ to draw the elevation profile on top of the velocity model.

model.bln: "Blanking file" containing the boundary of the velocity model . Importing this into Surfer™ will zero out velocity values below the envelope of the ray coverage producing a plot similar to that one created by the MakeEPS module.

postingfile.txt: File containing the shot and geophone locations and label. This file can also be imported into Surfer for labeling purposes.

4.4 Procedure for determining nx, nz, hx, and hz

If you want to set your own horizontal and vertical resolution (hx and hz) for the optimization instead of choosing a preset option, uncheck the 'Autocal' button in 'RIOTS Settings' (see Section 4.1). Then, enter values in the spaces for nx, nz, hx and hz. The following procedure describes how you determine these values:

Use L6 data files in the 'demo' folder as an example. In Figure 4, L6 was run with a pre-set 'High' resolution setting (res=4). From the progress window, one can see the values used by RIOTS for this run are nx=70, nz=43, hx=1.085951 (feet), and hz=1.085951 (feet). Now, say you want to set the resolution of your optimization (cell size) to hx=1 (feet) and hz=0.5 (feet). The nx you will need to enter is determined by using the following formula:

nx = (integer) [(Maximum Distance)/hx] + 3

where, Maximum Distance is the maximum length of the survey (end to end) and is written out in the 'Progress' window or in the appropriate riotsmsg file, when RIOTS is run at any of the five default resolution settings. For this example, nx for an optimization with cell size 1.0 (feet), will be:

nx = (integer)[72.258301/1.0] + 3 = **75**

nx has to be an integer. So, if it is a floating point, make sure it is rounded off to the next highest number.

To determine the value of nz, use the following formula

$$\begin{aligned} \text{new nz} &= (\text{old nz} * \text{old hz}) / (\text{new hz}) \\ &= (43 * 1.085951) / 0.5 \\ &= 93.39 \end{aligned}$$

This can be rounded to the next highest integer **93.**

So the values you need to enter are nx=75, nz=93, hx=1.0, and hz=0.5. It is recommended that for a data set, run the optimization at the lowest resolution to obtain the values of nx, nz, hx, and hz used by RIOTS. Then use them to determine nx and nz for your choice of hx and hz.

If you are manually editing the riotsinput file, you need to set autocal=1, and replace res=[value] parameter with four lines in the order nx=[value], nz=[value], hx=[value], hz=[value].

4.5 Getting the Best Results Using SeisOpt @2D

SeisOpt @2D is designed for velocity optimization with minimum user input and interaction. Nevertheless, it is useful to remember a few tips to use it effectively and get the best possible velocity model from your data.

In a refraction survey, it is hard to determine how deep the first-arrival rays penetrate before hand. This is because the depth of penetration is a function of the source-receiver offset used during the survey and the subsurface velocities. The non-linear relationship between them precludes determining how deep the rays would penetrate before running the optimization. As a result, the depth of model is the biggest unknown. In other words, the number of cells in the vertical direction, nz, cannot be predetermined before RIOTS is run. One can set nz to a very high value, but that would result in waste to computing time. The default nz values that are used when RIOTS is run using the default resolution settings (that is, when run at 'High', 'Highest' etc), is approximated one-third the maximum offsets in the survey. This nz setting is not necessarily the one that will give the best result. ***Hence, it is imperative that RIOTS be run with different nz values to ensure that you***

get the best result for a particular data set. The following paragraph and subsection describe how to do this automatically using the "batch processing" feature.

First, decide the default resolution setting that will provide you with the best solution without any artifacts. To do this, run RIOTS at 'Medium', 'High', and 'Highest' settings. During each run, you will notice that the velocities in the regions not constrained by the rays (zero hit counts) will vary from one run to the next. Repeating the runs gives a user a handle on these velocities. For typical geo-technical surveys, the 'Highest' setting gives the best result. But there are cases, for example, when there are large elevation changes, when a 'High' resolution run will give you a better result. So, first run it at 'Highest' preset resolution. Carefully look at the parameters that print on the "Progress" window (Figure 4). If the model parameters, nx and nz, are large (say, $nx * nz > 50{,}000$) or if the estimated time to complete the run is very long, reconsider running it at 'High' or 'Medium' resolution setting. If you do decide to terminate the current run, use the 'Task manager' and end 'riots'.

After you have settled on the 'hx' and 'hz' values to use, now you need to run the same data set with different values of nz, keeping hx, hz, and nx fixed. Section 4.6 describes how to automate this by using the "batch processing" feature, remembering to give a different extension for each run to prevent overwriting output files from the previous runs.

Try at least 10 runs with 10 different values of nz, each time changing it in increments or decrements of 1. For example, you can run 5 with values of nz less than the default nz and, 5 with nz greater than the default nz, and one at the default setting. You will notice the least-square error increasing and/or decreasing as the nz's are changed. Look at the Velfile, Hitfile, Pickfile, and the final error of the model (it appears on the 'Progress' window and is also written to the riotsmsg file) from all the runs and choose the model you think best fit your data and known geology. In general, lower the final error, better the model.

The more runs you can make with different nz values, the better. The batch processing feature enables this to be done overnight or while the user is away from the computer.

4.6 Batch Processing

The more runs you perform on one data set, the better your ability to get the best velocity model and to be certain that you have a robust model. Batch processing feature allows the user to automate these runs. The following are the steps to use this feature:

1. The first step is to create several 'riotsinput' files containing the different model parameters. To do this follow steps 2 through 9.
2. First, run 'RIOTS' at a default resolution setting, say at 'High' (res=4 in the riotsinput file).
3. You can either let it run to completion, or terminate it immediately and run it later as part of the batch processing.
4. View the riotsmsg_[ext] file created by SeisOpt@2D run at the default setting (Step 2), to get the default values of nx, nz, hx, and hz. Alternatively, you can just look at the values that appear in the 'Progress' window.
5. Now, you will need to run the same data set, using the manual option and using the same values of nx, hx and hz as in Step 3, but different values of nz.

6. To do this, choose the manual option (Deselect ‘autocal’ button in RIOTS Settings window) and
   Enter the same value for nx as the default setting
   Decrease (or increase) the value of nz by 1, from the default setting
   Enter the same values for hx and hz as the default setting (Figure 5b).

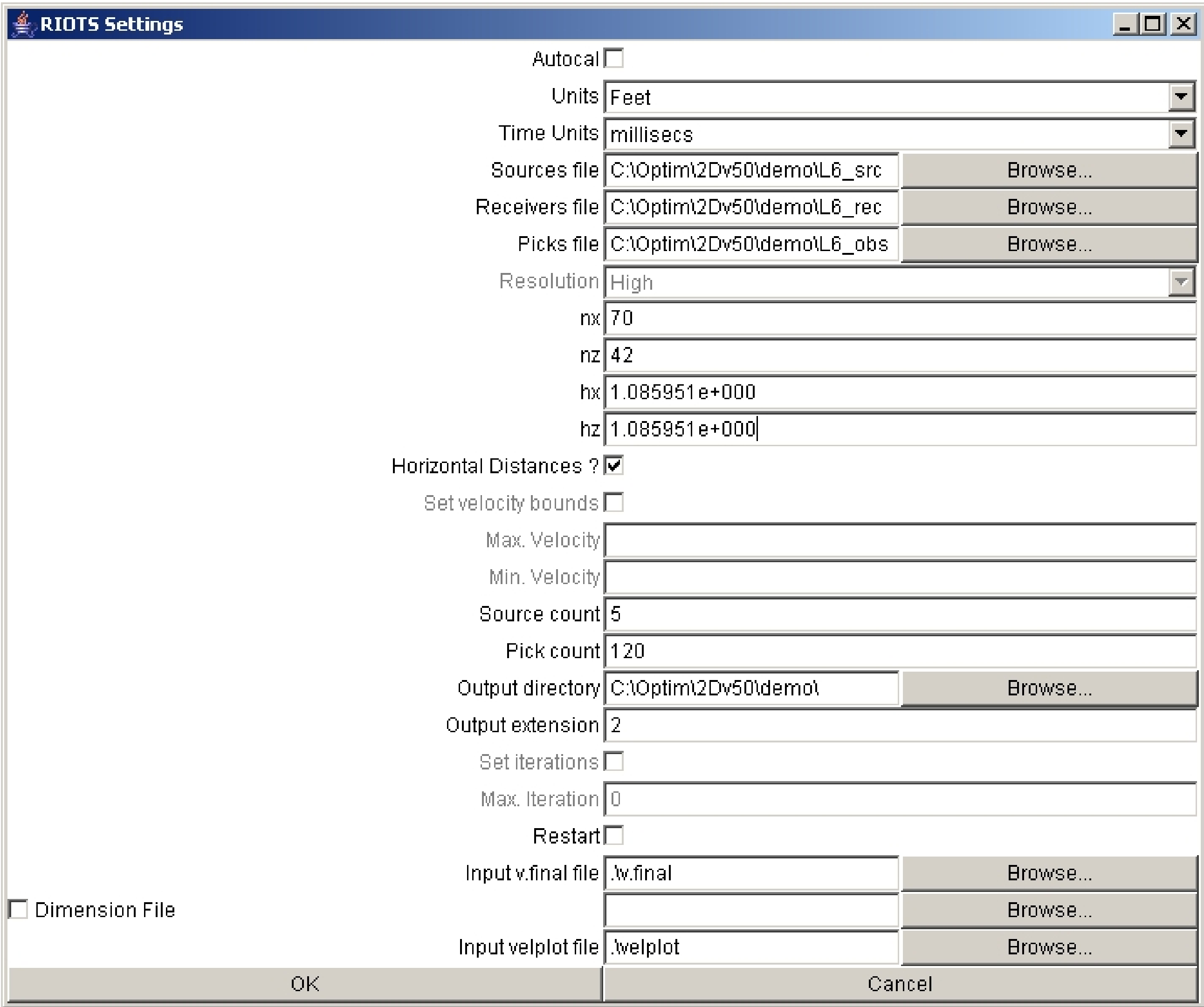


Figure 5b: RIOTS setting window showing how to enter values for nx, nz, hx and hz manually (de-select ‘autocal’). Also note that extension for the output files has been changed.

7. Enter a new extension for this run of RIOTS (any alphanumeric character or text) so as to prevent files from one run overwriting existing files from a previous run (Figure 5b).
8. Click OK. A file called riotsinput_[given extension] will be written to the output folder specified in ‘RIOTS Settings’ (C:\optim\2dv50\demo\ in this case).
9. Repeat steps 5-7 for at least 6 different values of nz, 5 values greater than the default nz and 5 smaller than the default nz. More nz values you can test the better.
10. After creating all the riotsinput files, click on the ‘Set up Batch file’ button found in the lower row of the main (Velfile or Hitfile display) graphical user interface (Figure 2c).
11. The window shown in Figure 6 will open up.

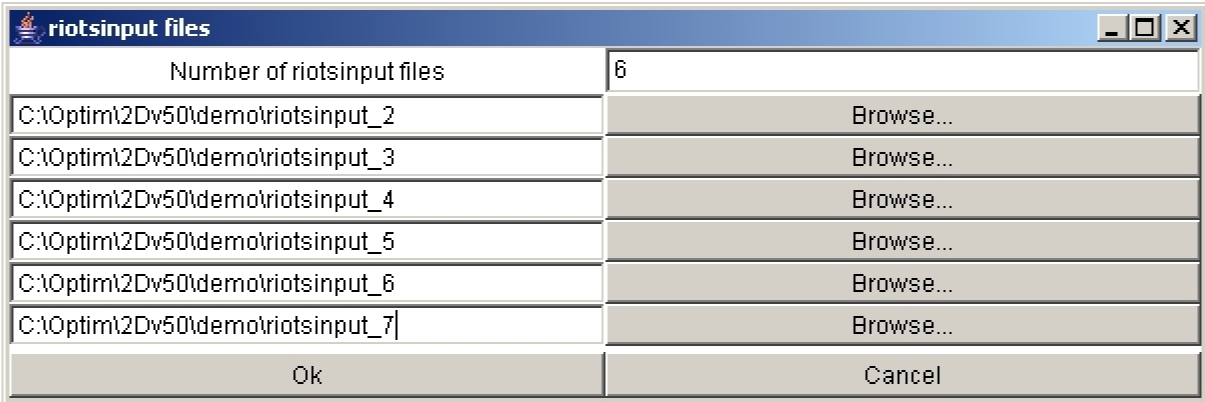


Figure 6: Set up batch file for batch processing by choosing the appropriate riotsinput files and clicking 'Ok'.

Enter the number of riotsinput files (number of runs) you will be using to batch process and hit `<ENTER> or <RETURN>`. The appropriate number of riotsinput entry spaces should open up. Note that a blank 'Progress' window will also open up at the same time. You can close this window. Use the 'Browse' button to choose the appropriate riotsinput files with extensions.

12. Click 'Ok'. The window shown in Figure 7, with the message riotsbat file written will open up. Now you are ready to start batch processing.

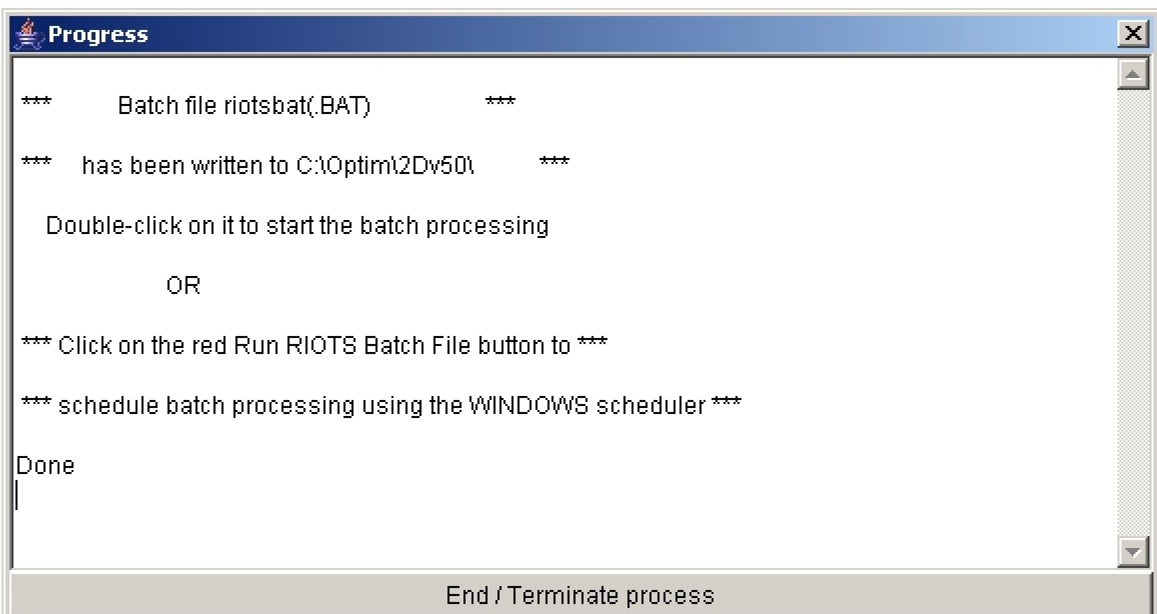


Figure 7: Progress window showing creation of batch file. Now you are ready to start batch processing.

13. To start batch processing, you can either click on 'riotsbat.BAT' (Figure 7a) created in C:\optim\2dv50\ or use Windows scheduler.

Figure 7a: Click on the 'riotsbat.BAT' file in C:\optim\2dv50\ to start batch processing.

14. If you click on the 'riotsbat.BAT' file a MS-DOS windows opens up showing the runs (Figure 7b).

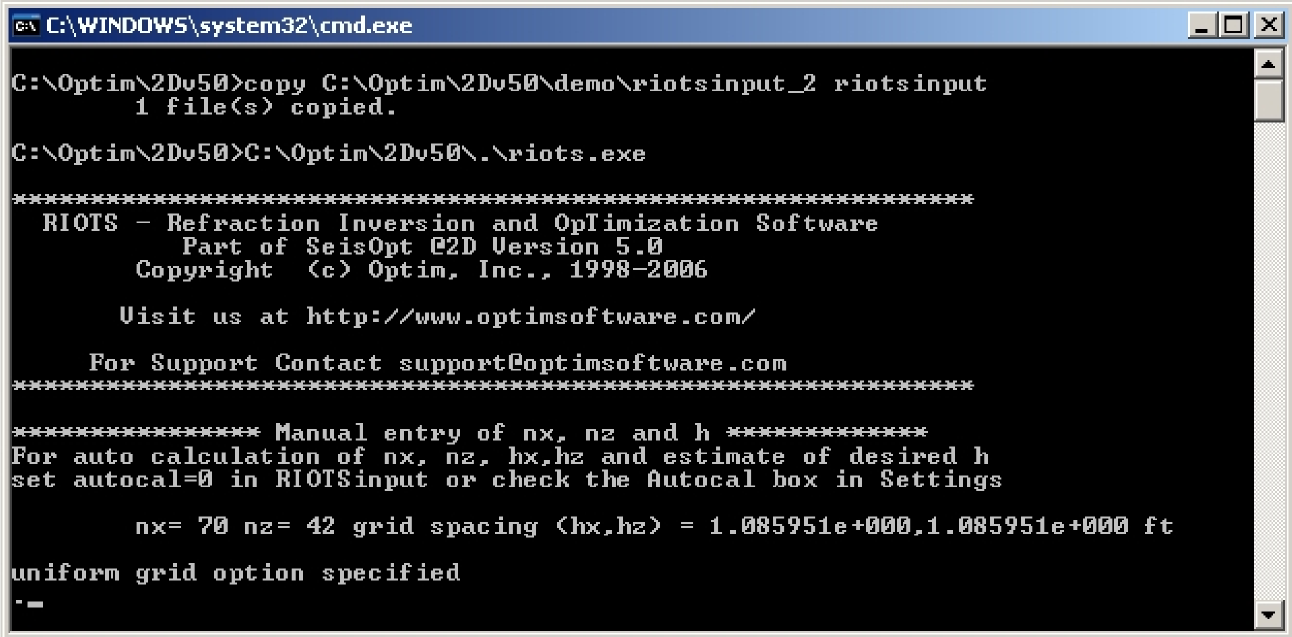


Figure 7b: On clicking 'riotsbat.BAT' file, MS-DOS windows opens up showing the progress of the batch processing. On completion, the window closes automatically.

15. To use the Windows scheduler, click on the red 'Run RIOTS Batch File' button in the lower row of the main (Velfile and Hitfile display) graphical user interface (Figure 2c). The Windows Task Manager shown in Figure 8 opens up.

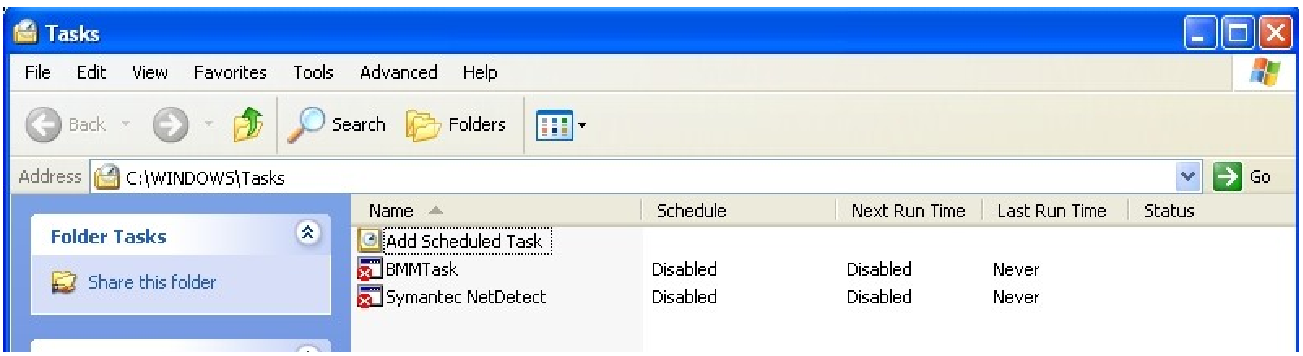


Figure 8: Windows Task Manager opens up when the 'Run RIOTS Batch File' button is clicked.

16. Click on the 'Add Scheduled task' icon to bring up the 'Schedule Task Wizard' shown in Figure 9.

Figure 9: The Schedule Task Wizard helps schedule batch processing jobs.

17. Click 'Next' to bring up the file selection dialog box (Figure 10). Go to the SeisOpt @2D installation directory (default directory is C:\Optim\2Dv50) and choose the file '**riotsbat.BAT.**' DO NOT choose the file called riotsbat_temp.

Figure 10: Click on the 'Browse' button and choose the riotsbat (BAT file) file in the SeisOpt @2D installation directory (default: C:\Optim\2Dv50)

Figure 11: Select the 'One time only' option and change the task name if desired.

18. Choose 'One time only' option in the window shown in Figure 11. The task name can be changed if desired.
19. Next, choose the time and date the job needs to start (Figure 12). The default setting shows the current time and date. Make sure you change the time to be a little later than the displayed time (say, 10 minutes later). If you do wish to start the job immediately, see Step 19. The time can be changed so that the batch processing can start at any time.

Figure 12: Set the time and date for the batch processing to start. The default setting is for the job to start immediately. Change this to be at least 10 minutes after the displayed time. If you wish to start the job immediately, click on the 'Run' option in Figure 14.

20. Click ‘Next’ to bring up the dialog window shown in Figure 13. If you choose the advance properties option, the window shown in Figure 14 will open up on clicking the ‘Finish’ button.

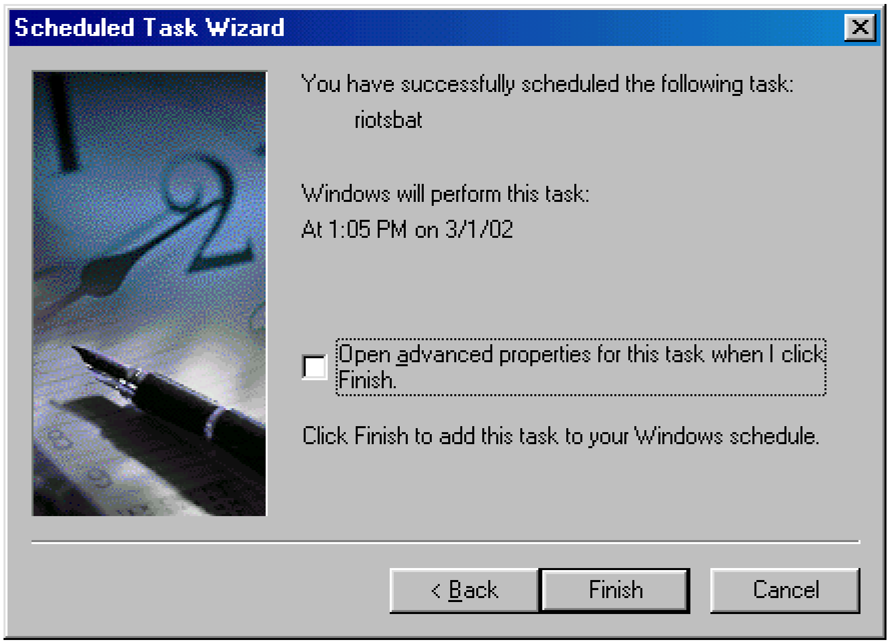


Figure 13: Click ‘Finish’ to end the batch processing set up.

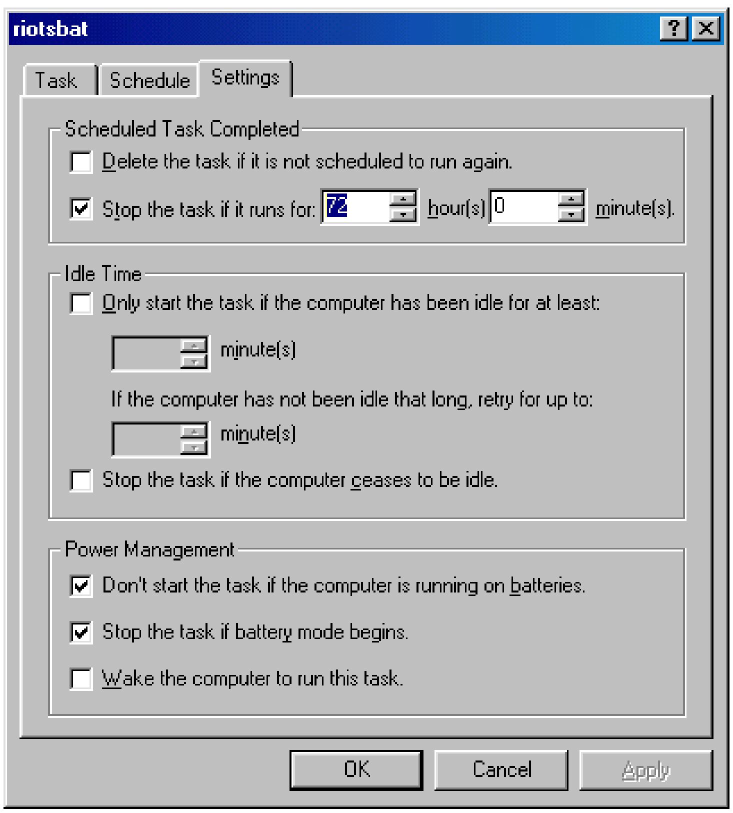


Figure 14: For large models with many shots, it is a good idea to change the ‘Stop task’ time to be 5 days (120 hours)

Click on the 'Settings' tab on this window and it is good to change the option that specifies how long the task will run before it is deleted. Since some RIOTS runs take a long time, it is good to change the number of hours to be longer.

21. On clicking 'Finish' (Figure 13) or 'Ok' (Figure 14), the riotsbat file will be scheduled for batch processing. It will appear in the Tasks window as shown in Figure 15. If you wish to start the job immediately, highlight the riotsbat name and choose 'Run' from the File menu. To delete it from the queue, highlight riotsbat and choose 'Delete' from the File menu. Batch processing will start at the specified time and date.

Once the runs are complete, view the 'riotsmsg' files from each run. You will notice the least-square error increasing and/or decreasing as the nz's are changed. In most cases, the run that had the lowest error is the best model. To be sure look through the Velfile, Hitfile, and Pickfile from all the runs and choose the model that fits any prior constraints, in addition to the lowest error criterion.

4.7 Using the 'Restart' option in RIOTS settings

The 'Restart' option is used for the following scenarios:

- Set the velocity bounds and re-run the optimization.
- Tune a model and re-run using the tuned velocity model as input.
- Change the grid spacing to a variable grid mesh and re-run the optimization. In this case the model run using the uniform grid layout must be read in to create the variable grid mesh (see Section 8.3). Then the 'Dimension File' that contains the variable grid information and the v.final and velplot file from the uniform grid runs must be read in.

In all cases, the appropriate 'v.final' and 'velplot' files must be read in. The number of iterations can be set to '1' if the object is to just to see the effects of doing the actions listed above. To run the full optimization, using the new velocity model as input, do not set the number of iterations.

5.0 Visualizing RIOTS Output

5.1 Selecting the results of the most recent optimization for viewing

Output files from the optimization runs are saved to the output directory (folder) specified in 'RIOTS Settings'. The files will have the extension specified by the user. By default, the extension is the date and time when 'RIOTS settings' window was opened for entering the parameters.

Typically, four files are loaded from each run. They are Velfile, Hitfile, Pickfile, and Surveyfile. Of these the ones of most interest are the Velfile, Pickfile, and Hitfile. Make sure to press the 'Reset' button before loading from new runs.

To load the files click on the 'Settings' button. The window shown in Figure 15 opens up. Browse and select the appropriate files.

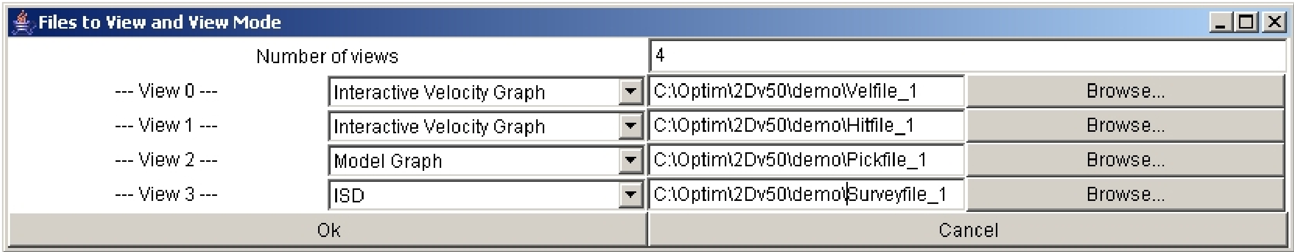


Figure 15: Click on 'Settings' and browse and select the files to be viewed.

Click 'OK'. If you wish to modify the Pickfiles created during the optimization run, make sure you select the SeisOpt @2D input files (obs, rec, src files) as shown in Figure 15a.

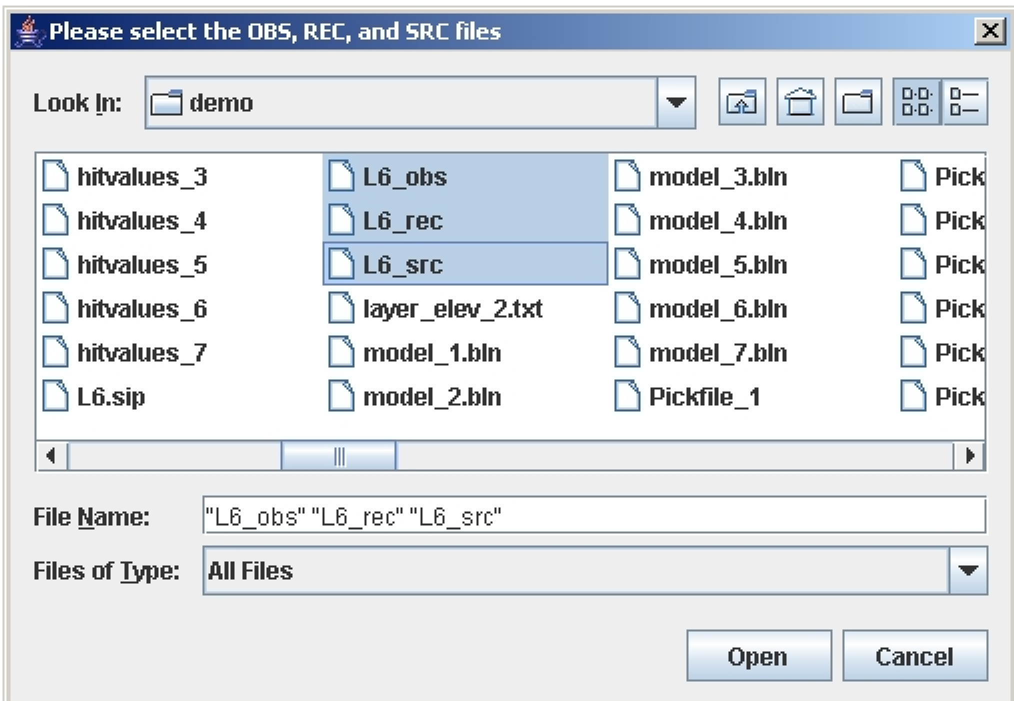


Figure 15a: To be able to modify the Pickfiles you will have to read in the corresponding obs, rec, and src files (input files used for the optimization run).

The 'Number of views' field is for entering the number of files to be read. The default is 4, corresponding to Velfile, Hitfile, Pickfile and Surveyfile, but it can be any number. The next lines, labeled 'View 0', 'View 1', etc., are for entering the names of the files to be read. Pressing 'Ok' after entering the number of views adjusts the number blank view fields to correspond to the 'Number of views' parameter. Either enter the full path and file name, or browse for the file using the 'Browse…' button. To the left of each file name window is a pull-down menu for selecting the type of display. They are preset for the default file names, but may need to be adjusted if different files are selected. Use the following guide to choose the appropriate display type:

File type	Display type
Velfile:	Interactive velocity graph
Hitfile:	Interactive velocity graph
Surveyfile:	ISD (Interactive Survey Design)
Pickfile:	Model Graph
VelSurveyfile:	Interactive velocity graph

5.2 Viewing Selected Files

Once a file has been selected for viewing, click the ‘Go’ button to display them. This may take a few moments for large files. A dialog box appears if SeisOpt@2D cannot find the selected file. In this case, make sure the file name is correct, and that it is present in the specified directory. However, a shortage of memory could also cause large files to not load. In this case try clicking the ‘Reset’ button, which clears the memory, and select one file at a time to view. If that fails, use the procedure described in **Section 9.4** to increase the memory available to SeisOpt@2D.

Make JPEG button

The ‘Make JPEG’ button is common to all Views and it can be used to produces a jpeg file of the displayed image. **Section 7.1** describes how to use this feature.

Zoom buttons

The following list explains the zoom controls common to all display types:

> **Zoom Box**: Clicking this key allows you to define a zoom region by using the mouse to draw a box. Do this by clicking and holding the left mouse button while dragging the mouse pointer over the desired region. The display will zoom to the boundaries of the box.
>
> **Zoom 1:1**: Clicking this button restores the display to the default state of zoom, in which the entire file fits within the display window.
>
> **Zoom In**: After clicking this button, left click on a region of the display to zoom in on. To continue zooming in, keep clicking on the display. To stop zooming in and return to the default display mode, click on ‘Stop Zooming’. Use ‘Zoom 1:1’ to return to the original display.
>
> **Zoom Out**: Works just like ‘Zoom In’. Click on the button, then on the display, and the image will zoom out centered on the point that was clicked.

5.2.1 Visualizing Velocity models

This section explains the display features of the ‘Interactive Velocity Graph’ display mode, used for displaying the Velfile output of SeisOpt@2D. On selecting a Velfile and clicking ‘Go’, the velocity model appears in the display window. Figure 16 shows an example velocity model. The velocity model is computed over a rectangular array of square cells, but only the region constrained by the ray paths is displayed. Therefore, it is the geometry of the paths of the energy traveling from sources to receivers that determines the size and shape of the resulting velocity model. (These paths can be viewed in Hitfile)

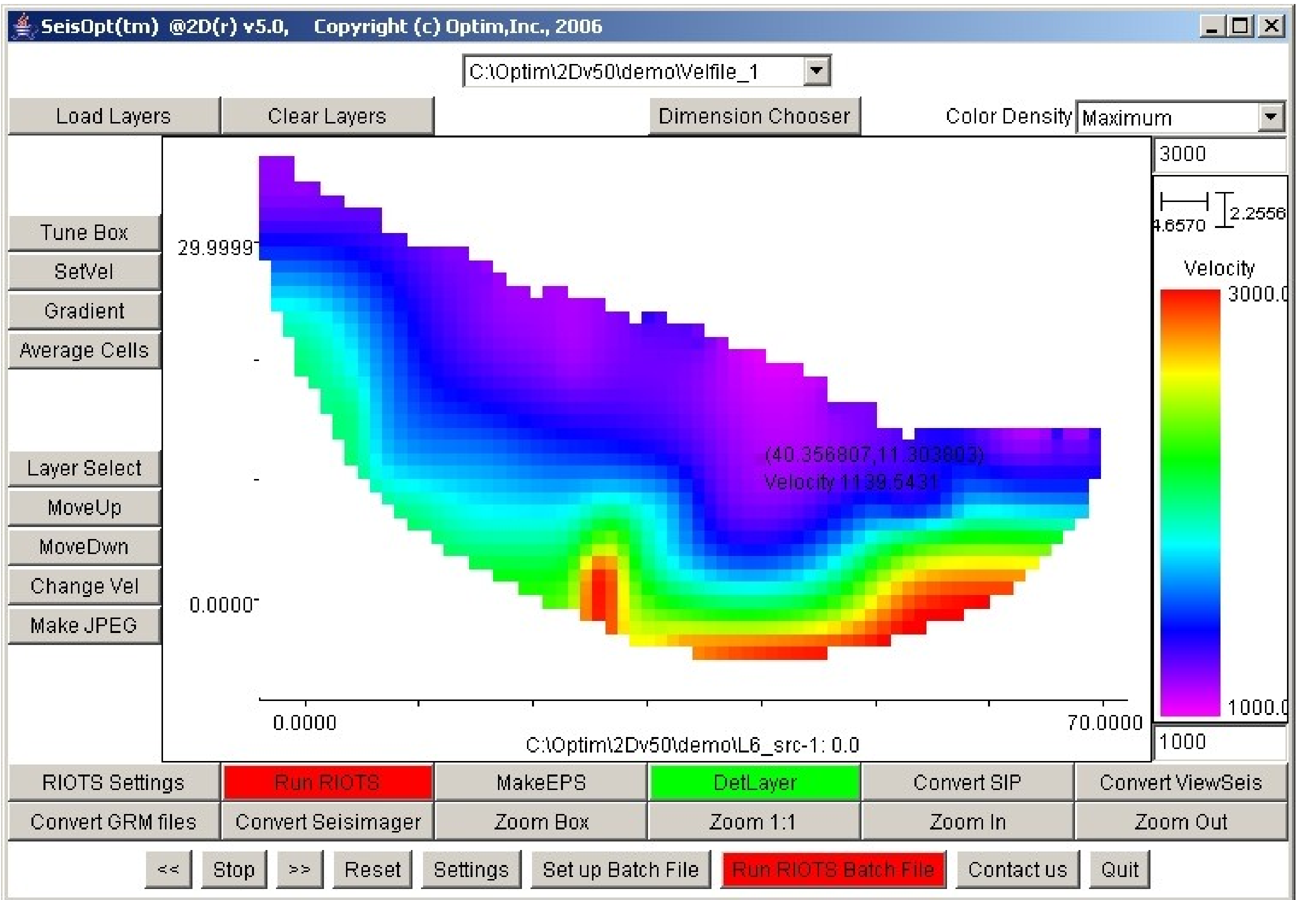


Figure 16: Visualizing Velfiles

Features of the Velocity model display (refer to Figure 16)

Mouse: Left clicking with the mouse anywhere on the velocity model displays the coordinates of that point in the physical units (offset from upper left corner of the model), and the velocity of that cell.

X-axis (horizontal): The X-axis displays the length or offset of the survey in the physical units used in RIOTS settings (i.e. feet, meters, and kilometers).

Y-axis (vertical): The Y-axis corresponds to the relative elevation of the survey. The top of the axis is labeled with the true maximum elevation of the survey in the units specified in 'RIOTS Settings', the same units used in the X-axis. The bottom of the Y-axis is labeled with the distance below the top of the survey. The elevation of the bottom of the survey can therefore be determined by subtracting the lower Y-axis label value from the upper value.

Vertical exaggeration: The vertical exaggeration of the display varies from model to model, because it depends on the size of the model (number of cells in each dimension) relative to the size of the SeisOpt@2D window. The SeisOpt@2D window can be resized however, and by carefully examining the x- and y-axes, a 1:1 display can be achieved. Here is how: Look at the length of the axis symbols on top of the amplitude bar. It indicates the length between the tick marks along the horizontal axis and vertical axis. For example, in Figure 16, the length between tick marks along the horizontal axis is 4.6570 (ft) while along the vertical axis it is 2.2556 (ft). In order to make a 1:1

display, we need to resize the window so that the horizontal length is about two times the vertical length of these lines. Then click on ‘Reset’, reload the src, obs, and rec files (if you wish to edit the Pickfile) and click ‘Go’ to view the Velfile.

Amplitude scale bar: The scale bar, which is labeled velocity and plotted to the left of the velocity model, shows the correspondence between the colors used in the velocity model plot and velocity. The velocity values are in the units specified in ‘RIOTS settings’ per second. Clicking the left mouse button anywhere on the scale bar displays the exact velocity of that color. The default range of the scale bar is from 0 to the maximum value in the model. The color/velocity correspondence of the scale bar can be adjusted. In the blank spaces at the top and bottom of the scale bar, velocity values corresponding to the maximum and minimum, respectively, of the color spectrum can be entered manually. For example, unless the lowest velocity in the model is 0, the full range of colors is not used in the default display. By entering the actual minimum velocity of the model (look in the ‘riotsmsg’ file) in the space at the low end of the scale bar, the full range of colors will be used in the display. By entering velocity values within the range present in the model, the color display can be ‘clipped’.

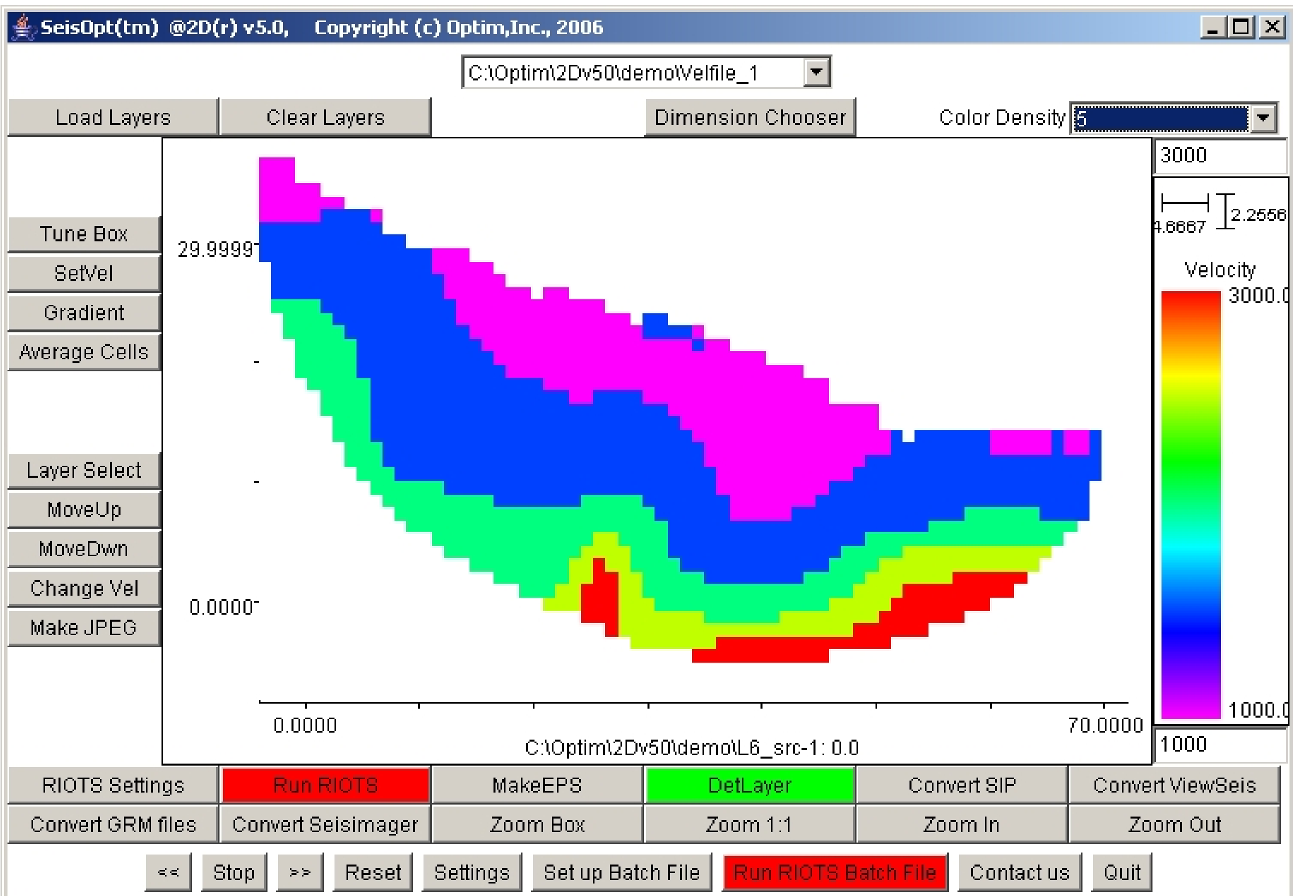


Figure 17: Changing the number of colors.

Color Density: The color density refers to the number of colors used in the velocity model display. The default setting is “Maximum”. In general SeisOpt@2D avoids the limitations imposed by assuming layered structures in velocity models, but this is a common feature of traditional velocity modeling programs. To cause to velocity model display to more closely resemble a simple layered structure, the number of colors used to represent the velocities can be reduced. Figure 17 shows the layer density pull-down menu. The numbers in this menu effectively refer to the number of colors that will be used in the velocity model plot. Scroll down the menu to find the lowest color density, 3.

Make JPEG: Click on Make JPEG to create a JPEG image of the displayed velocity model. Enter a file name for the output file. The program queries for the height and width of the image in pixels (Figure 17a, b, and c). If you wish to maintain the ratio of the displayed image, make sure that you specify the number of pixels to be in the same ratio as the numbers shown above the amplitude scale bar. This will create a JPEG image shown in Figure 17c.

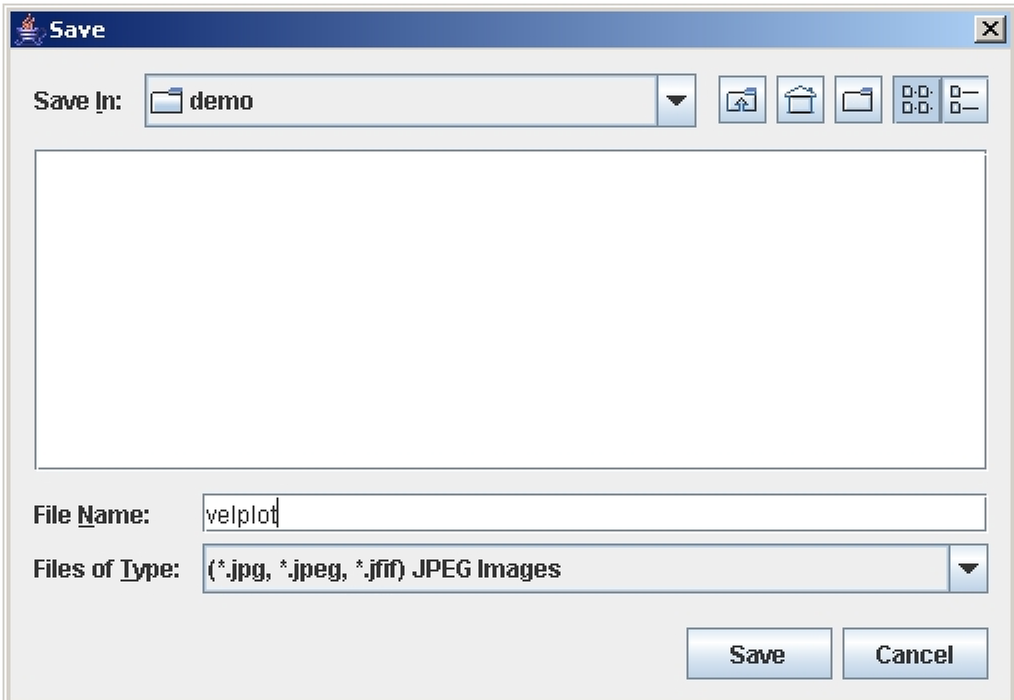


Figure 17a: Enter file name for the output JPEG image

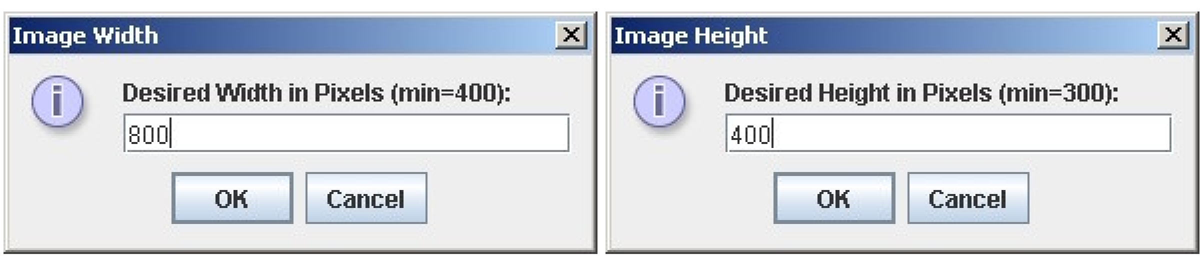


Figure 17b: Enter width of the output JPEG image in pixels. To maintain the aspect ratio of the displayed image, the ratio between the height and width (in pixels) should be similar to the numbers above the amplitude scale bar of the 'Velfile' display.

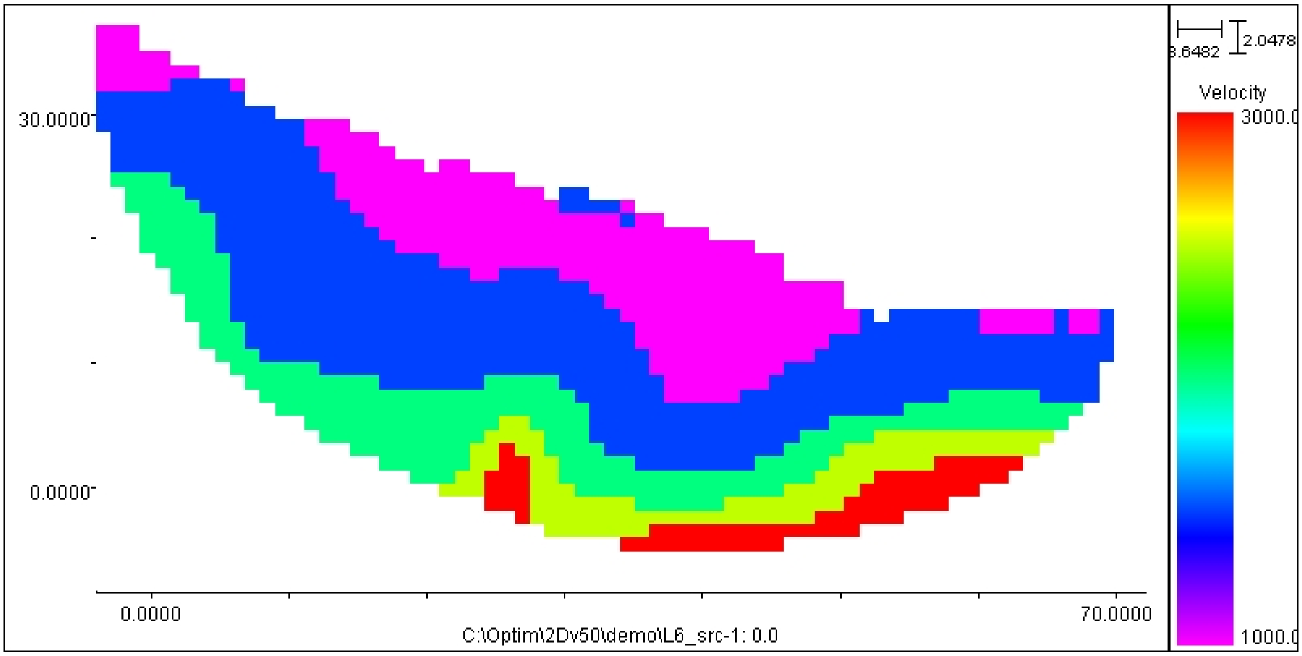


Figure 17c: JPEG image created using the 'Make JPEG' module.

5.3.2 Visualizing Hitfiles

Once a Hitfile has been selected for viewing, click 'Go' to display it. Figure 18 shows the Hitfile corresponding to the Velfile in Figure 16. Hitfiles contain information about the how the subsurface was sampled by the seismic survey, based on the optimized velocity model. SeisOpt@2D computes the ray paths for each source/receiver pair, and then determines the number of times each cell in the model is crossed ('hit') by a ray path. Hitfile displays this image. The envelope of the 'hit' count image determines what region of the optimized velocity model is constrained, and therefore displayed.

Note that the Hitfile in Figure 18 contains white cells within the plot. These areas have no hits but are surrounded by areas that have several hits are not unconstrained if they appear towards the middle of the model. Such areas should be interpreted with caution only if they appear towards the end of the array, say, between an off end shot and the geophone spread. Hitfiles use the same 'Interactive Velocity Graph' display mode as Velfiles, so the essential features of the display are the same.

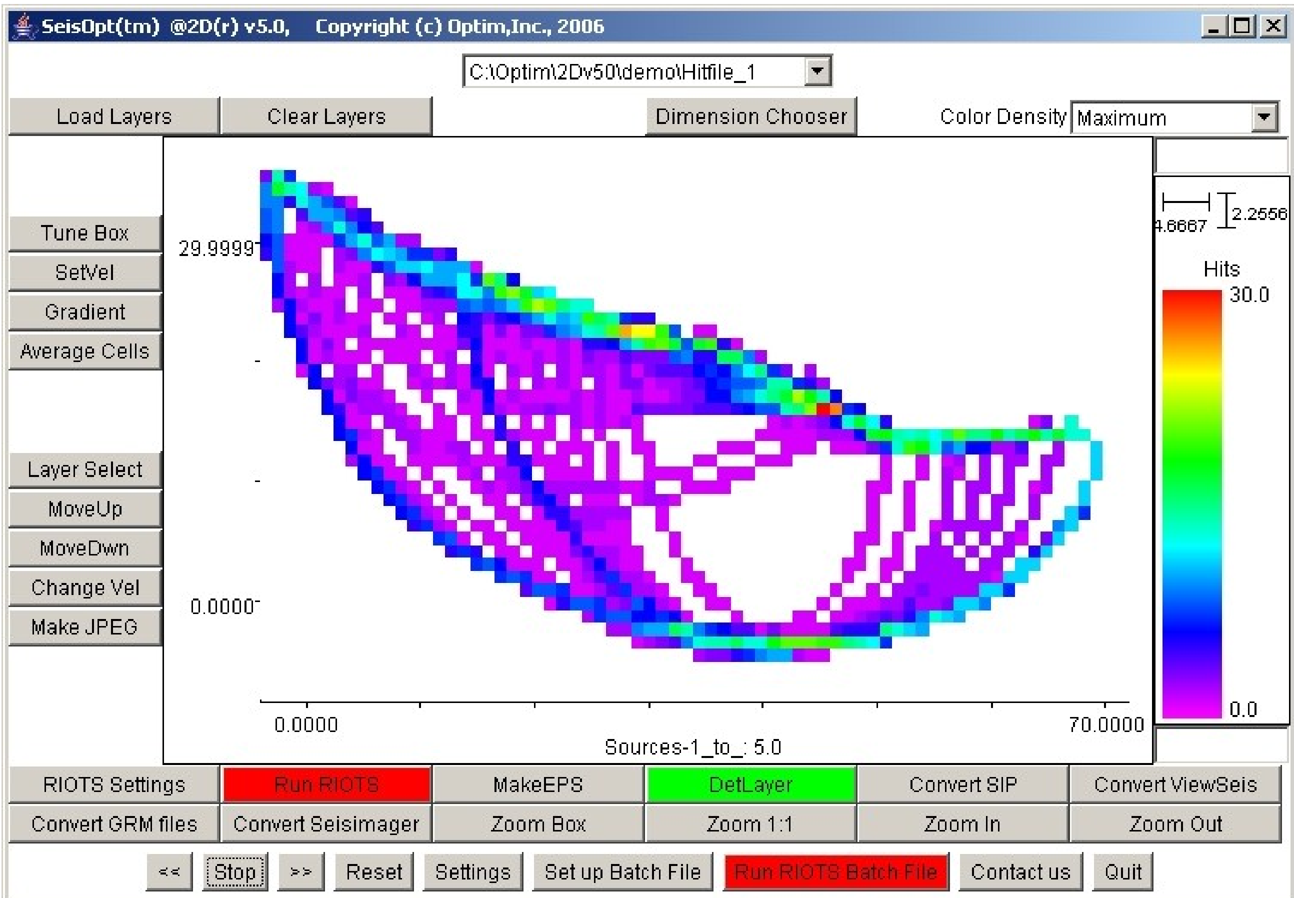


Figure 18: Visualizing Hitfiles

Features of the Hitfile Display (refer to Figure 18)

Mouse: Click anywhere on the displayed Hitfile to see the coordinates and number of hits for any cell in the model.

X, Y-axes: Same as velocity model display. (See 'Features of the Velfiles Display')
Amplitude scale bar: The scale bar shows the correspondence between the number of hits and color. Left click on the scale bar to see the hit value for specific colors. Light purple = 1 hit, white = 0 hits

Layer Density: This controls the number of colors that are used in the display. While useful for viewing Velfiles, this feature is not useful for Hitfiles.
<<, >>: Use these buttons to cycle the display between the ray paths for different sources. When a Hitfile is initially loaded, all the sources are displayed. Use '<<' to remove one source at a time from the display. Use '>>' to cumulatively add sources to the display.

Make JPEG: Click on Make JPEG to create a JPEG image of the displayed velocity model. Enter a file name for the output file. See the Velfile section to know the details.

5.3.3 Visualizing Pickfiles

Once a Pickfile has been selected for viewing, click 'Go' to display it. Pickfiles contain the arrival time pick data used in the velocity optimization and the first arrival times calculated from optimized velocity model. The discrepancy between these two sets of times is what SeisOpt@2D minimizes during the optimization. Pickfiles are used to visually inspect how close these two sets of times are. The least-square error that is displayed in the 'Progress' window and written out to the riotsmsg file is a quantitative measure of how well the times through the final model fit the picks. This value is also written to the Errorfile. One blue line and one black line are plotted for each source, corresponding to the observed and calculated first breaks, respectively. The black squares represent the observed (picked) first arrival times, and the blue triangles represent the calculated times, for each receiver. Figure 19 shows the times for source number 1 and Figure 19a times for all sources of the example data shown in Figures 16 and 18.

Features of the Pickfile display (refer to Figures 19 and 19a)

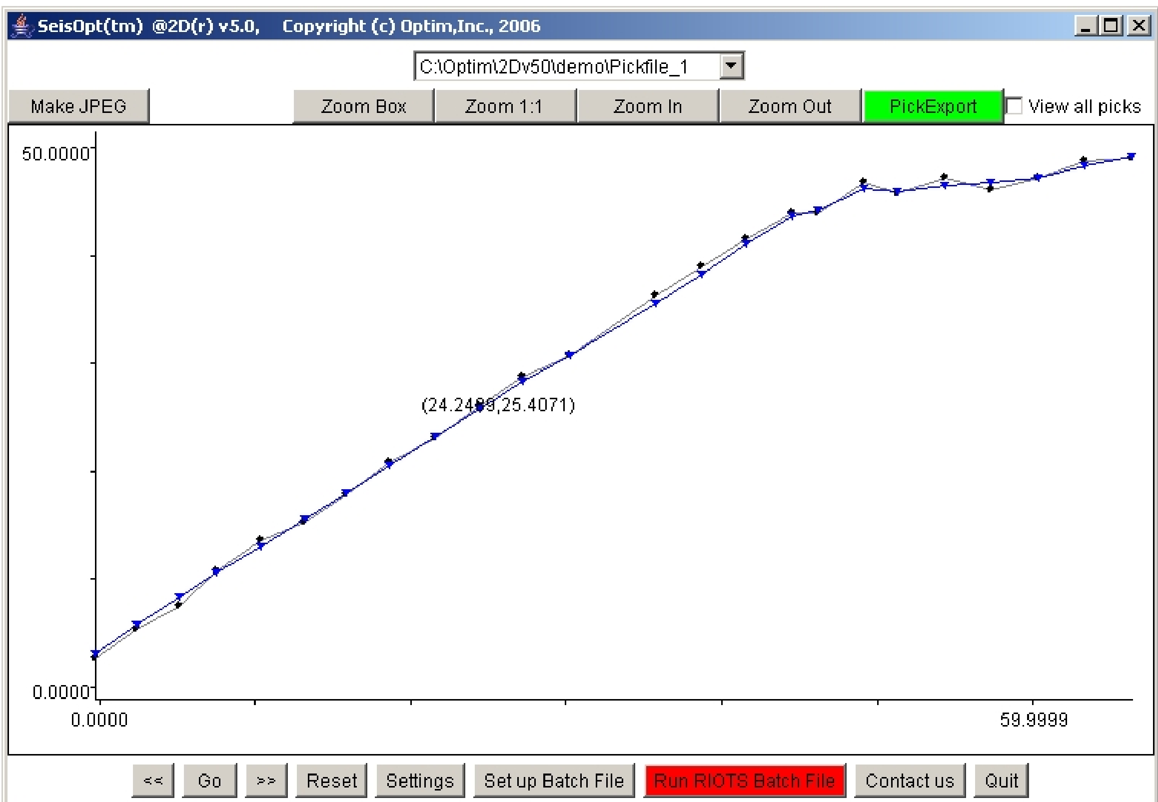


Figure 19: Visualizing Pickfiles - displaying picks and fits from one shot

Mouse: Left clicking the mouse on any point (square or triangle), displays the physical coordinates of that receiver, and the travel time to that receiver for the associated source.

X-axis (horizontal): Offset in physical units, the same as the Velfile/Hitfile display x-axis.

Y-axis (vertical): First arrival time in seconds or milliseconds.

View all picks check box: This check box controls whether sources are displayed singly or all together. The default setting for this feature is off (not checked), and only the first source is shown when the file is first displayed. Check this box with the left mouse button and the times for all sources will be displayed (Figure 19a).

<<, >>: When viewing one source at a time, use these buttons to cycle through the different sources. The current source number is shown at the bottom of the viewing window.

Make JPEG: Click on Make JPEG to create a JPEG image of the displayed velocity model. Enter a file name for the output file. See the Velfile section to know the details.

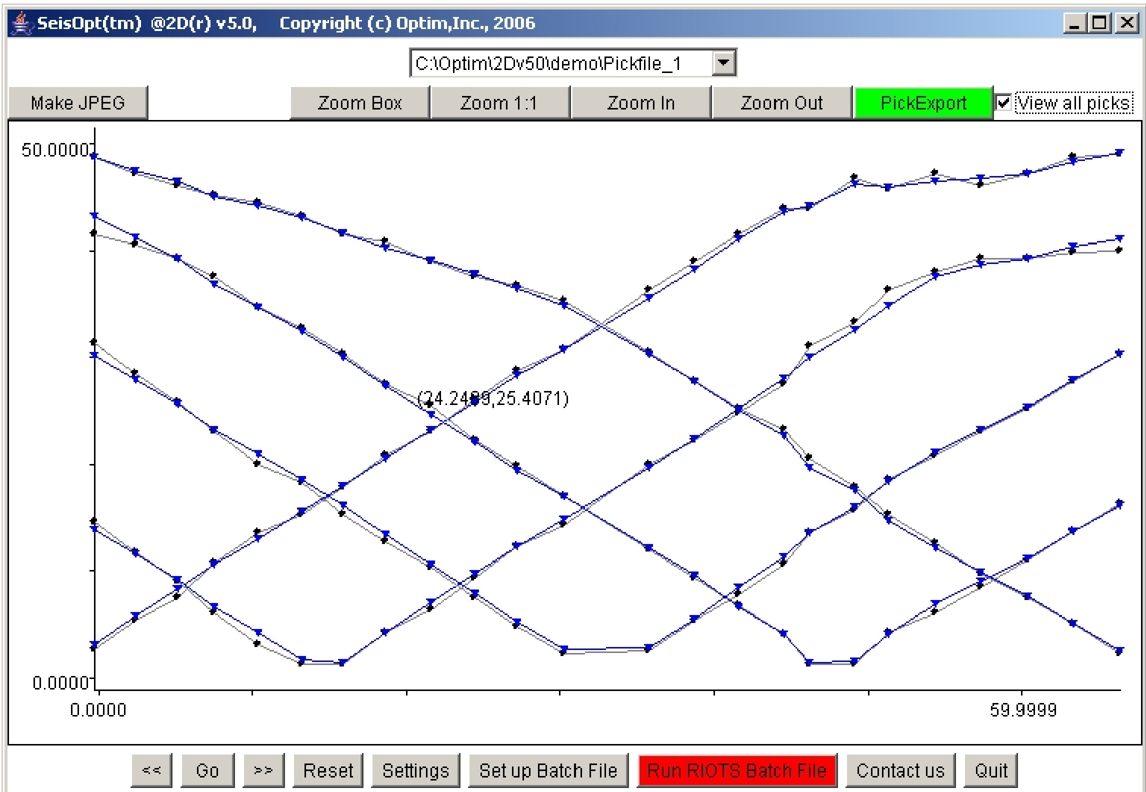


Figure 19a: Visualizing Pickfiles - viewing picks and fits from all the shots

5.3.3.1 Exporting/Printing Pickfiles: The PickExport module

The Pickfile generated from RIOTS can either be output to a printer (screen dump) by clicking on the ‘‘Print’’ button or exported as a 3 column ASCII file by clicking on the green colored ‘‘PickExport’’ button (Figure 19). When this button is clicked, the dialog window shown in Figure 20 opens up.

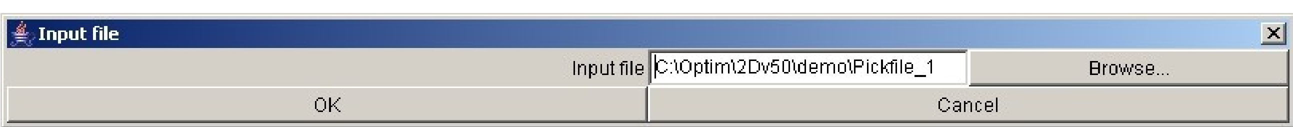


Figure 20: Choose the appropriate Pickfile and click ‘OK’ to write out a 3-column ASCII ‘picks.txt’ file

The input file that appears in the PickExport dialog window will be the Pickfile that is loaded and displayed in the Pickfile window (Figure 19). For example, in Figure 19 the Pickfile loaded for display is ‘Pickfile_1’ and so the input file is for PickExport is also ‘Pickfile_1’.Use the ‘Browse’ button to choose the appropriate Pickfile if different from the one displayed in the window. Click

'OK' to execute the PickExport function. The progress window shown in Figure 21 will appear. A file called picks_[ext].txt will be written to the directory that contains the input Pickfile. The picks.txt file uses the extension ([ext]) of the input Pickfile. If the input file has no extension, then the picks file will not have an extension either.

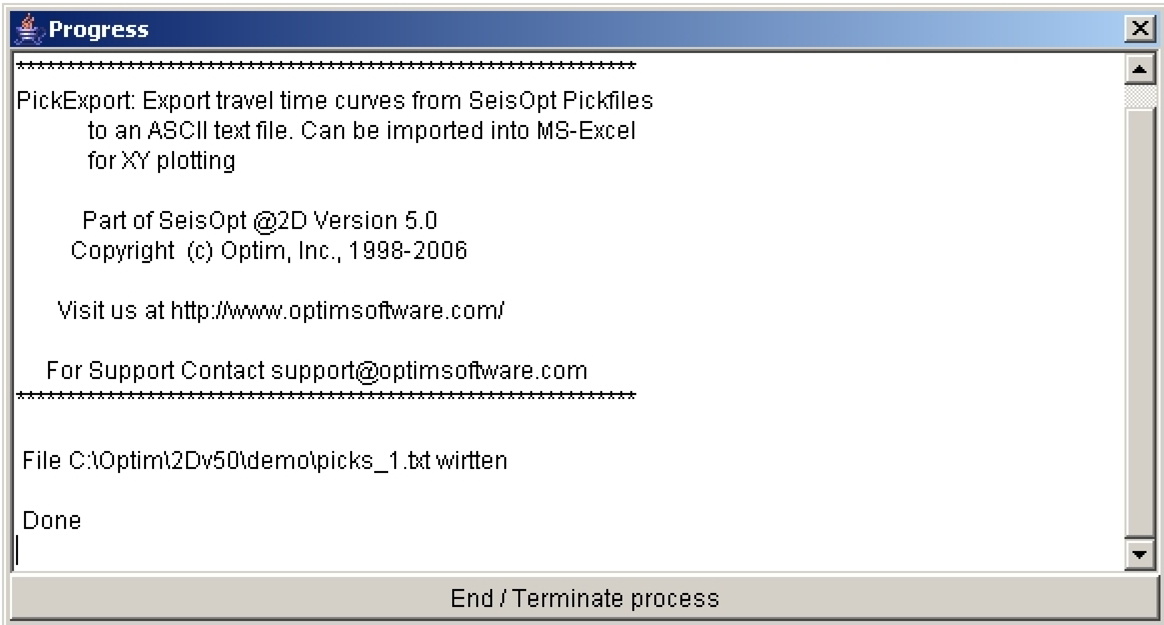


Figure 21: Progress window showing completion of the 'PickExport' function

The file produced by PickExport can then be imported into a graphing program like Microsoft Excel to view/edit/plot the picks.

5.3.3.2 Editing Pickfiles

The observed picks (black squares) shown in the Pickfile display can be either deleted or moved and the changes saved to the three input files (src, obs, rec) needed to run 'RIOTS'. Here are the steps involved:

- View the Pickfile. Changes can be made either to the single source file view mode or the 'View all Picks' mode.
- To move an observed pick (black square), click on it using the left mouse button and drag it either up or down to the desired time (Figure 21a). Note that the 'y' mouse coordinate indicates the travel time. Hit <ENTER> to record the changes. **NOTE that no changes can be undone.**

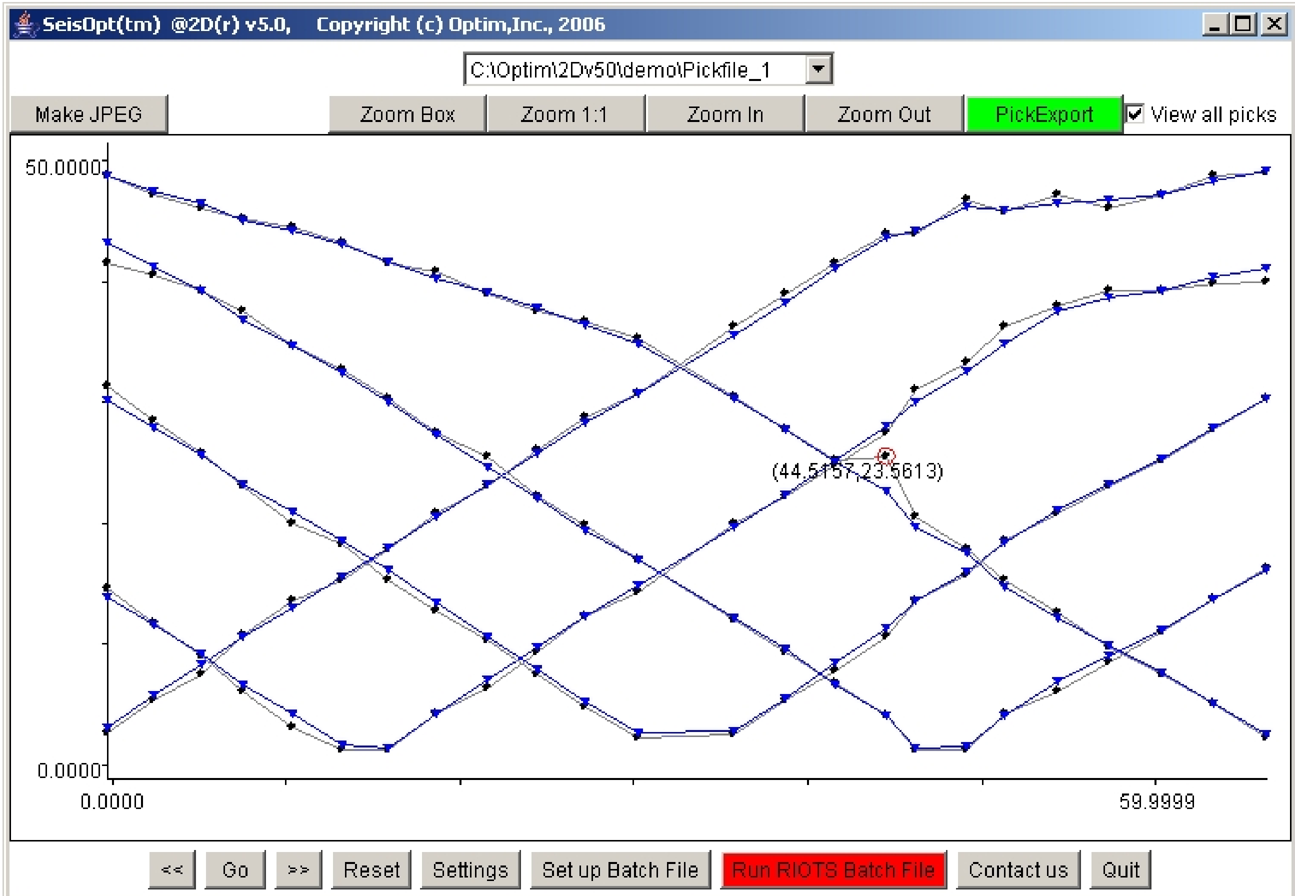


Figure 21a: Left-click on a pick to be change and either drag it up or down to move it or hit the <DELTE> key on the keyboard to delete it. Hit <ENTER> to record the changes

- The MS-DOS window that opens up when SeisOpt@2D is launched will show the new file names to of the new src, rec, obs and the edited Pickfile (Figure 21b). The new input files will have the word '.new' appended to them. For example, if the input files that are read in are L6_src, L6_rec, and L6_obs, the changed files will have the names L6_src.new, L6_rec.new, and L6_obs.new, respectively. If another change is done, then the new files will be called L6_src.new.new, L6_rec.new.new, and L6_obs.new.new and so on.

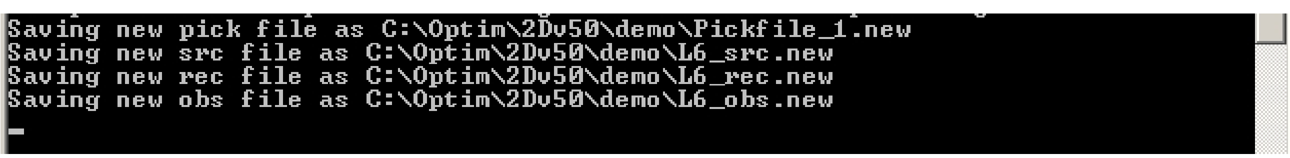


Figure 21b: On hitting <ENTER> the changes to the Pickfile are recorded as new input files for 'RIOTS'. The word '.new' is appended to the saved src, rec, and obs files.

- To delete a pick, select it using the left mouse button and hit the <Delete> key on the keyboard. Once again hit <ENTER> to record the changes and write out the new input files for RIOTS run. Make sure you keep track of the number of picks you delete. This 'Number of picks' parameter in 'RIOTS Settings' should be changed accordingly when re-running 'RIOTS' using the edited input files.

5.3.4 Visualizing Surveyfiles/Performing Interactive Survey Design

Once a Surveyfile has been selected for viewing, click 'Go' to display it. Figure 22 shows a sample Surveyfile corresponding to the example files used in the previous sections. Surveyfiles contain the same subsurface sampling information as Hitfiles, but they also include the seismic array geometry. Surveyfiles are used to visualize the changes in subsurface ray path coverage when the array geometry is altered. Sources and receivers can be manually moved, added, or deleted in 'Interactive' mode, or they can be arranged automatically in 'Auto' mode to improve ray coverage in a specified area. Whenever the array has been changed, the subsurface ray path coverage can be recomputed to reflect the array changes.

Features of the Surveyfile Display/Interactive Survey Design controls (refer to Figure 22)

Mouse, X-axis, Y-axis, Scale Bar, and Layer Density functionalities are the same as in the Hitfile display.

Sources and Receivers: Sources and receivers are plotted along the top of the ray path coverage image, connected by a line representing the approximate elevation profile. Sources are shown as 6-pointed stars and receivers are represented by downward triangles.

Associations: This button is used to display and alter the receivers associated with (recording) a given source. Click this button and left-click on any source to see the association. That source and all the receivers currently associated with it turn black. Any receivers not associated with that source remain gray. Now, click on any receiver to toggle between associated and not associated.

Associate All Recs.: This button is used to associate every receiver with a given source. It is most useful when adding new sources. After clicking this button left-click on any source, and all the receivers will be associated with it.

Move: The move button allows sources and/or receivers to be moved using the mouse. Click on the 'Move' button, then click and hold the left mouse button on the source/receiver while dragging it to the new position.

Add Source: Use this button to add a source. Click 'Add Source' and then click on any location on the coverage display to add a source at that location. After adding a source, receivers must be associated with it. Use the 'Associations' or 'Associate All Recs' buttons to do this.

Del Source: Use this button to delete sources. Click this button and then on the source to be deleted.

Add Receiver: Use this button to add receivers. Click this button and then on the location where a new receiver is to be added. After adding new receivers, they must then be associated with sources. Use the 'Associate' or 'Associate All Recs' buttons to do this.

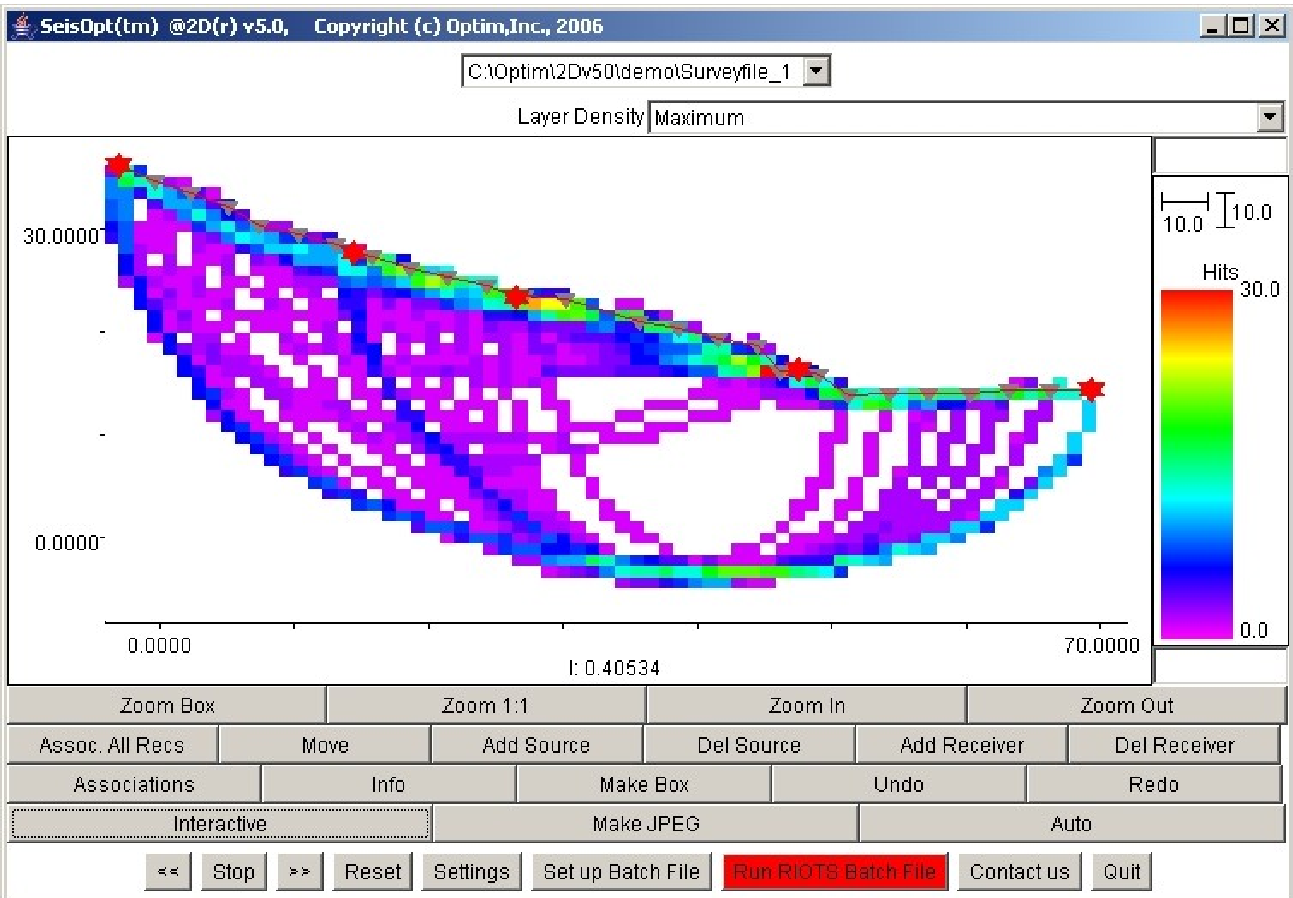


Figure 22a: Visualizing Surveyfiles for performing interactive and automatic survey design

Del Receiver: Use this button to delete receivers. Click this button and then on the receiver to be deleted.

Undo: Click this button to undo the most recent change to the array.

Redo: Restore the last undo.
Info: Click this button to enable mouse clicks on the model to display coordinate and hit information.

Interactive: This button recalculates the ray coverage. Click it after making changes to the array geometry (such as moving sources/receivers, deleting/adding sources/receivers), and a progress window opens while SeisOpt@2D calculates new ray paths. Once the calculation is finished, the word ‘Done’ appears. Figure 23 shows a progress window for a completed calculation. The name of the file with the new ray paths, which also appears in the progress window, is derived from the number of sources and receivers. In this case the filename is 'Surveyfile-5s120r', meaning there are 5 sources and 120 total receivers (In this case, 5 sources recording into 24 receivers each). Select this new Surveyfile for viewing using the ‘Settings’ button. When it is no longer needed, click ‘End / Terminate process’ on the progress window to close it.

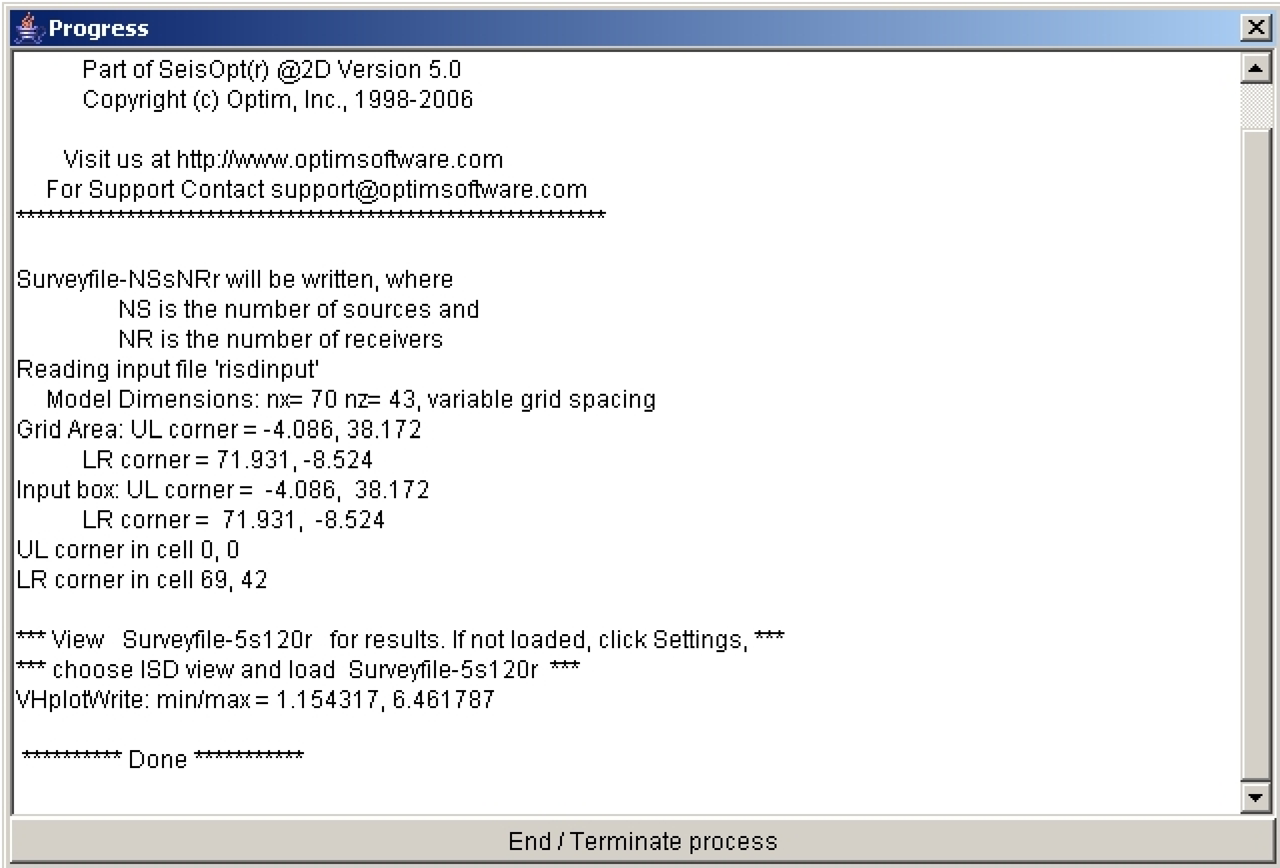


Figure 23: Progress window showing performance of an interactive survey design operation

Make Box: Use this feature with 'Auto' to automatically rearrange the array geometry to optimize ray coverage in a certain area. After clicking 'Make Box', then click and hold the left mouse button on the ray coverage display. While holding he left mouse button, drag the mouse to draw a box over the area of the model where an increase in coverage is desired. Then click 'Auto', and SeisOpt@2D will adjust the positions of sources and/or receivers in an attempt to maximize the coverage within the box drawn with 'Make Box' (Figure 24a).

Auto: This button automatically rearranges the array geometry and computes the new subsurface ray coverage. Use this feature after drawing a box using the 'Make Box' button (explained above) to draw a box around a region of the subsurface where an increase in coverage is desired. Once 'Auto' has been clicked, a progress window opens to track the progress of the calculation of the new array geometry and ray path coverage. Once the calculation is complete, select the new Survey file, whose name is given in the progress window, using the 'Settings' button. Note the new Surveyfile name, in this case 'Surveyfile-5s120r'. The extension appended to Surveyfile is derived from the number of sources and total number receivers (or picks).

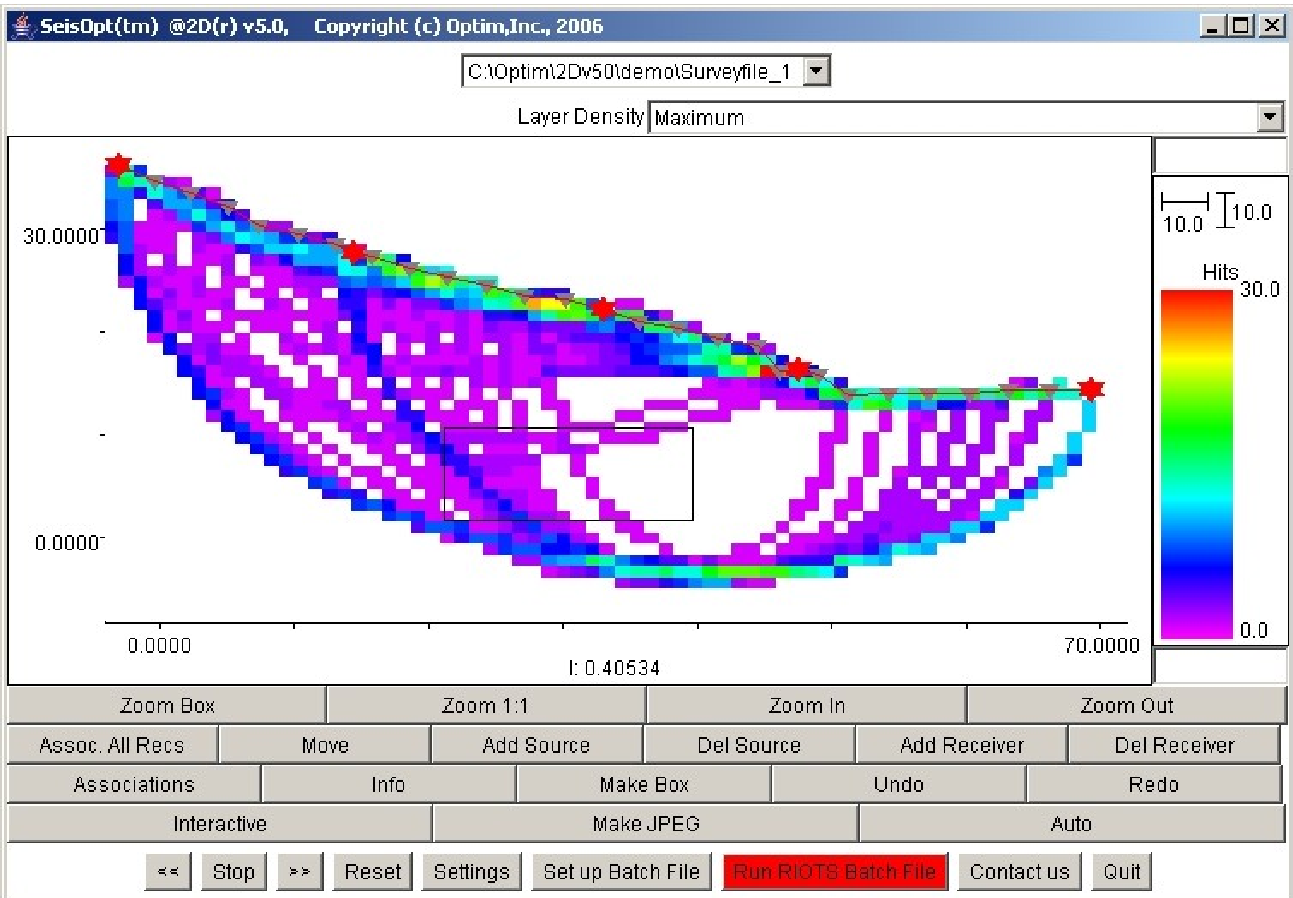


Figure 24: Click on 'Make Box' on draw a box around the region where the hit counts need to be maximized.

Summary of steps for 2 common Interactive Survey Design tasks

Add a source:

Step 1: Click 'Add Source', and left click at the location where the new source is to be added. A six-pointed star should appear at that spot, indicating a source.
Step 2: Click 'Associate All Recs', and click on the new source. This causes all the receivers to record the new source (and create ray paths from the new source to all of the receivers).
Step 3: Click 'Interactive' to recalculate the ray coverage that will include the new source. This opens a progress window.
Step 4: When the progress window indicates that the computation is done, note the new filename containing the updated ray coverage, of the form Surveyfile-#s#r.
Step 5: Select this file for viewing using the 'Settings' button.
Step 6: Once the file is selected for viewing, display it by pressing 'Go'.

Automatically rearrange the seismic array to increase coverage in a specified area:

Step 1: Click 'Make Box'. Then draw a box around the region of the model where an increase in coverage is needed. Do this by clicking and holding the left mouse button while dragging the mouse.

Step 2: Once a box has been drawn, click ‘Auto’. This opens a progress window.
Step 3: When the progress window indicates that the computation is done, note the new filename containing the updated ray coverage, of the form Surveyfile-#s#r.
Step 4: Select this file for viewing using the 'Settings' button.
Step 5: Once the file is selected for viewing, display it by pressing ‘Go’.

Note: The survey design module is set up to perform the above functions using the velocity model most recently created using RIOTS. If you want to do so with a previously created velocity model, go to the output directory of that optimization run and copy over the ‘v.final_[ext]’ file and ‘risdinput_[ext]’ file to the SeisOpt directory and re-name them ‘v.final’ and risdinput’ respectively. After each survey design run (interactive or automatic) a ‘plotinput_survey’ file is created in the directory, which can then be used for creating an EPS image file. To do this rename ‘plotinput_survey’ as ‘plotinput’ and use the ‘MakeEPS’ button in the Velfile display (see **Section 7.0**).

6.0 The DetLayer™ module: Determine layer interfaces given layer velocities

The ‘DetLayer’ module allows the user to calculate and plot interface depth as defined by layer based methods like GRM through the gradient model that is output by SeisOpt @2D. There are instances when a project calls for estimation of layer depths. Since, SeisOpt @2D reveals subsurface gradients, the question of how to determine the depth (or elevation) at which the transition between one layer and another occurs is answered by the ‘DetLayer’ module. Given the layer velocities (which can be estimated from GRM-type analysis, slope breaks in the travel time curve or bore-hole logs), the ‘DetLayer’ module (green button) on the Velfile and Hitfile display (Figure 2) calculates the elevation of the interface between the layers. Thus, this requires that the users either know or assume the velocity and the number of layers (up to a maximum of 10). It should be noted that the layer elevations calculated using ‘DetLayer’ is an estimate. Like any geophysical analysis it should corroborated with other measurements.

To use this module, click the green ‘DetLayer’ button. Choose the appropriate ‘velplot’ file that was generated from the RIOTS run (Section 4.3) through which the layer interface depths need to be determined in the ‘DetLayer Settings’ window that opens up (Figure 25). Then enter the minimum and maximum extent of the velocity model through which the calculation has to be done. This can either be the minimum and maximum x-coordinate of the survey, including the off end shots, or smaller extents, say, just under the receiver spread. The next step is to enter the number of layers and the desired layer velocities, up to a maximum of 10. Enter only the desired number of velocities (equal to the number of layers), and make sure that all the layer velocities are non-zero. After entering the velocities, click ‘OK’. The progress window shown in Figure 26a shows the execution of the program and reports the writing of the output files and their location. Three files are output by this module - one is called ‘velplot_[ext]_layer’ , the second is called ‘layer_elev_[ext].txt’, and the third ‘surfer_layer_[layer#].txt’ where [ext] is the extension of the input velplot file that was used by ‘DetLayer’ and [layer#] corresponds to the layer. If the input velplot file did not have any extensions, then the output files will not have extensions either. For example, the extension of the input file read in Figure 25 is ‘6’, and so the output files are named velplot_6_layer and layer_elev_6.txt to the input file directory C:\Optim\2dv50\demo\. Import the ‘velplot_layer’ file into the ‘MakeEPS’ module (Section 7.2) to create and encapsulated PostScript (EPS) file (Figure 29) of the velocity model with the layers superimposed on it. This file can also be imported into the ‘Velfile’ display and plotted and the layers plotted on the velocity model (Section 6.1).

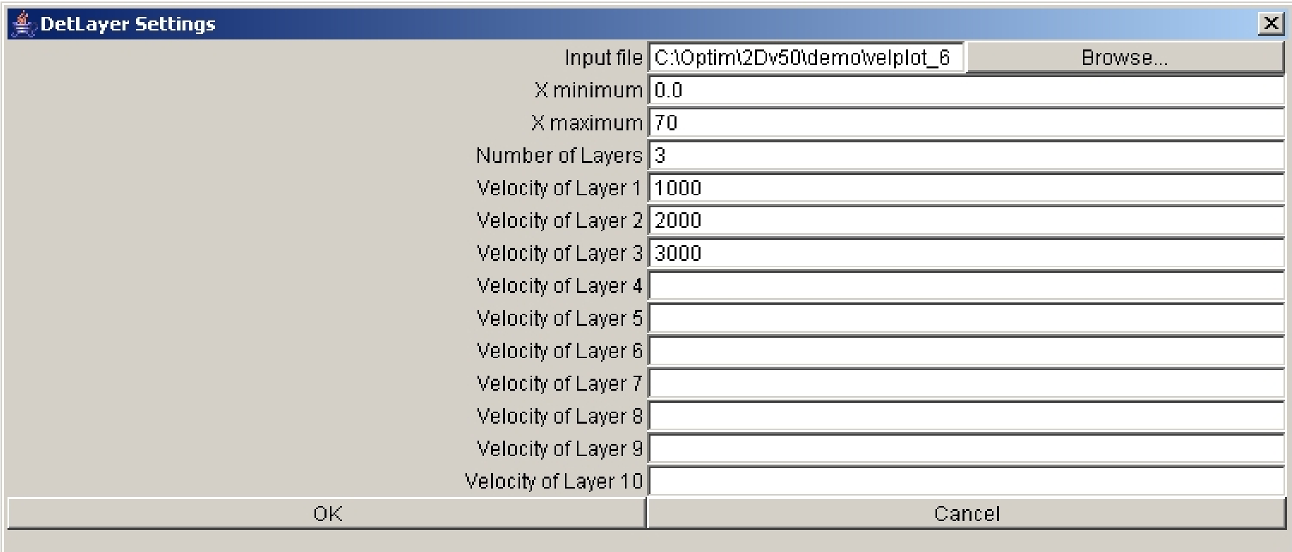


Figure 25: Set the layer velocities, maximum and minimum x-coordinates, and load the input velplot file in the 'DetLayer Settings' window.

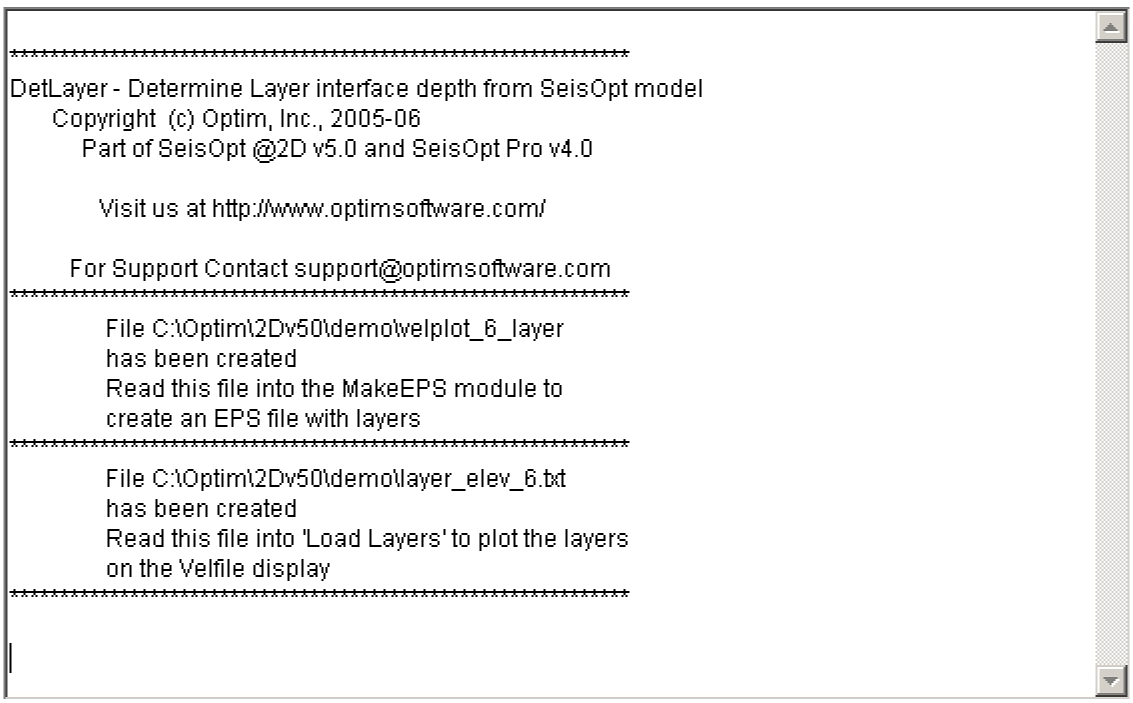


Figure 26a: Output files for use with the 'MakeEPS' module and 'Velfile' display are written to the input file directory. The .txt file can also be viewed using Notepad or WordPad and used as input into other drawing programs.

The 'layer_elev' file is an ASCII '.txt' file, and so can be viewed using Windows Notepad or WordPad, and with editing, could be imported into other drawing programs, if needed. The 'surfer_layer' files contain the (x,z) coordinates for each layer can be imported into programs like 'Surfer'.

6.1 Plotting layers on the 'Velfile' display

The 'layer_elev_[ext].txt' file created by the 'DetLayer' module can be imported into the Velfile display and the layers plotted on the velocity model. To do this, click on 'Load Layers' button on the Velfile display (Figure 26b). Browse for the appropriate 'layer_elev' file and click 'OK' (Figure 26c). The layers determined by 'DetLayer' will be plotted on the velocity model (Figure 26d).

Figure 26b: Click on 'Load Layers' button on the Velfile display panel to load the 'layer_elev' file created by the 'DetLayer' module. This will plot the layers on the velocity model. Use the 'Clear Layers' button to remove the plotted layers.

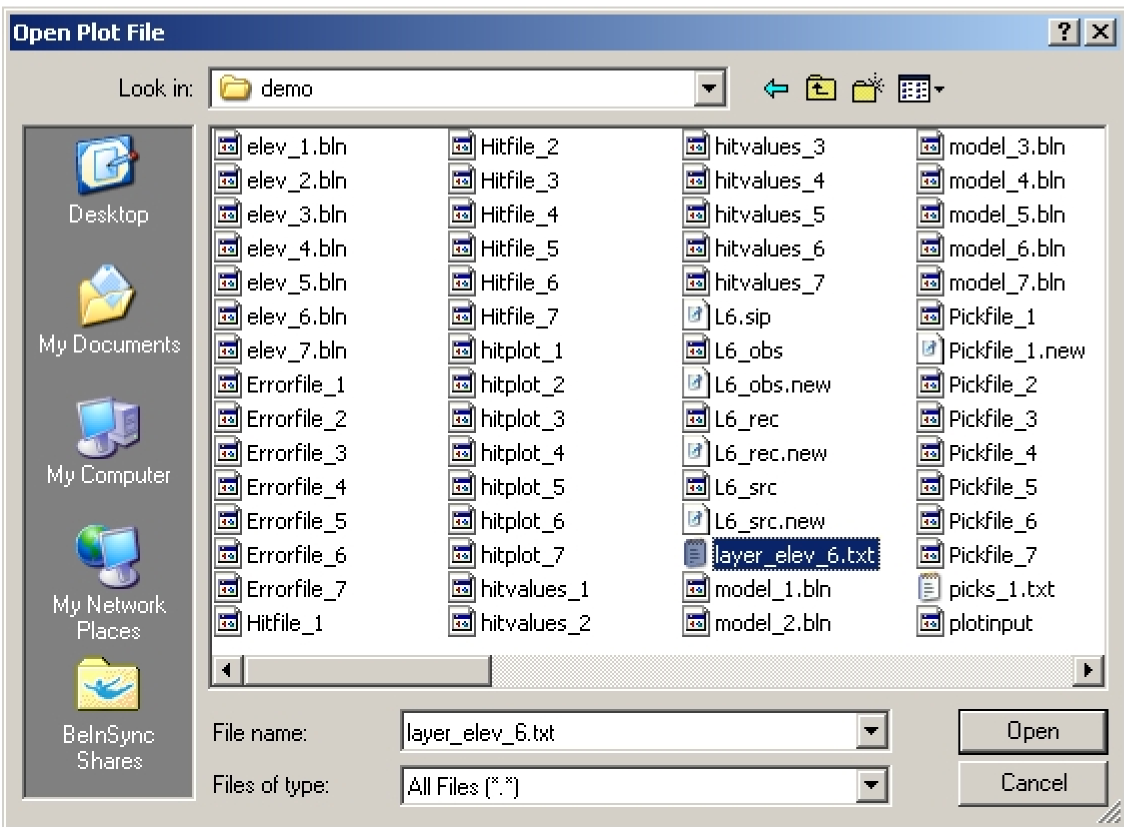


Figure 26c: Browse for the appropriate 'layer_elev' file that was created by the 'DetLayer' module.

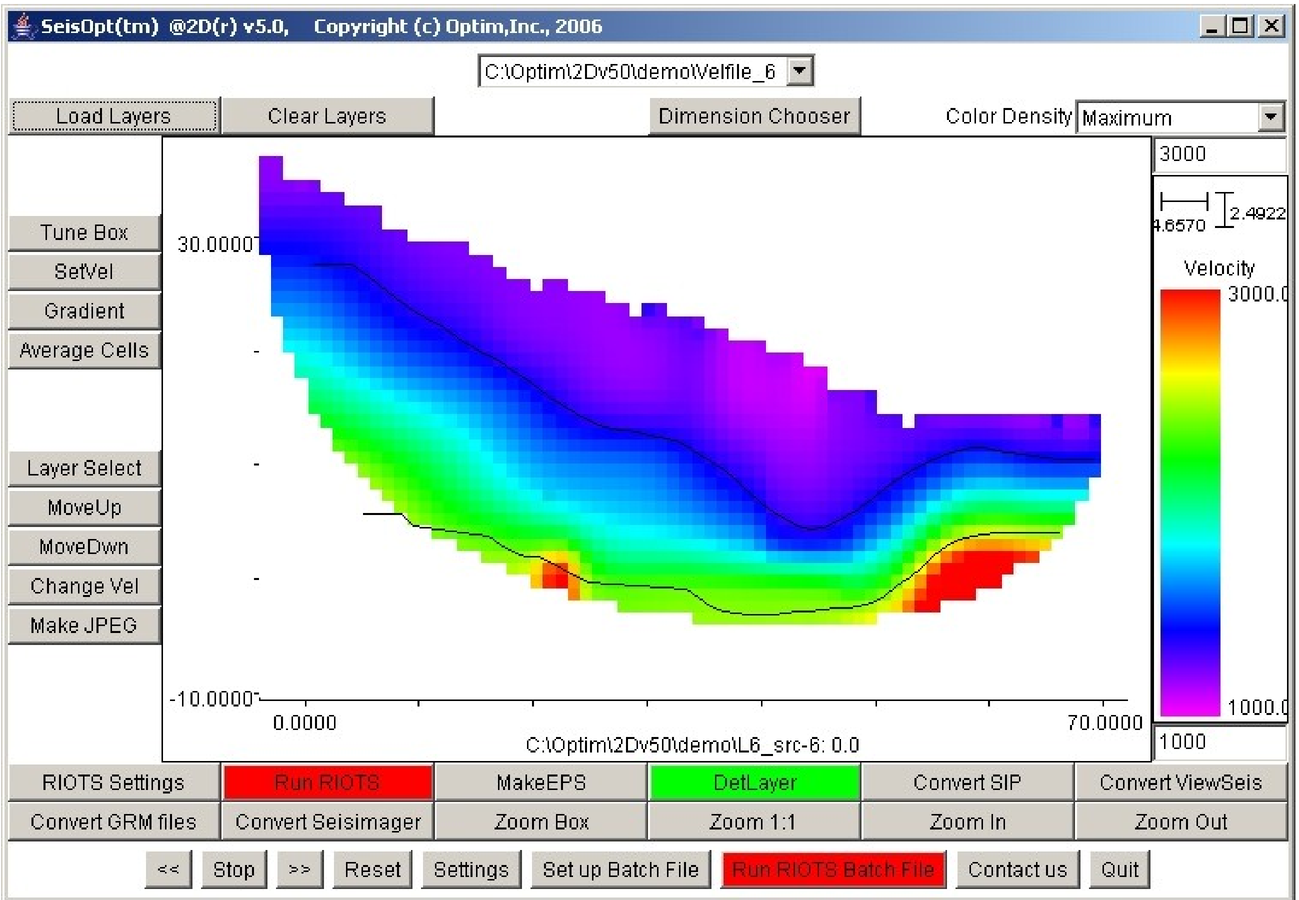


Figure 26d: The layer determined by the 'DetLayer' module will be plotted on the velocity model in the Velfile display. Use the 'Clear Layers' button to remove the layers.

7.0 Generating output images of the optimization results

7.1 Make JPEG

The quick and dirty way to output images of the optimization results is to click the 'Make JPEG' button. Enter a file name for the output file. The program queries for the height and width of the image in pixels (Figure 26e, f, and g). If you wish to maintain the ratio of the displayed image, make sure that you specify the number of pixels to be in the same ratio as the numbers shown above the amplitude scale bar. This will create a JPEG image shown in Figure 26g.

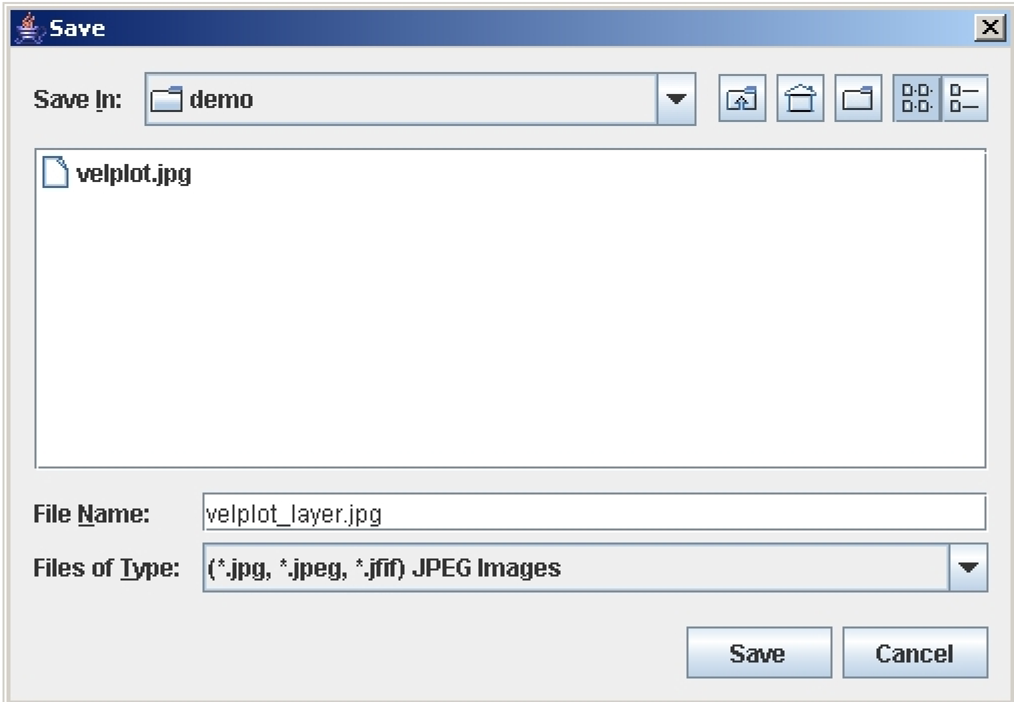


Figure 26e: Enter file name for the output JPEG image

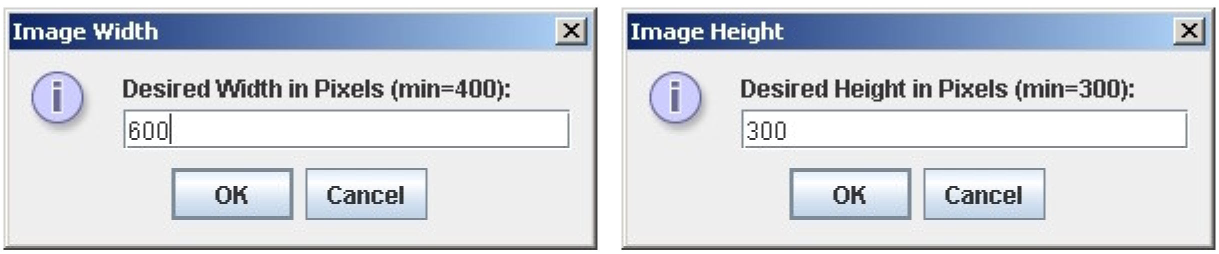


Figure 26f: Enter width of the output JPEG image in pixels. To maintain the aspect ratio of the displayed image, the ratio between the height and width (in pixels) should be similar to the numbers above the amplitude scale bar of the 'Velfile' display.

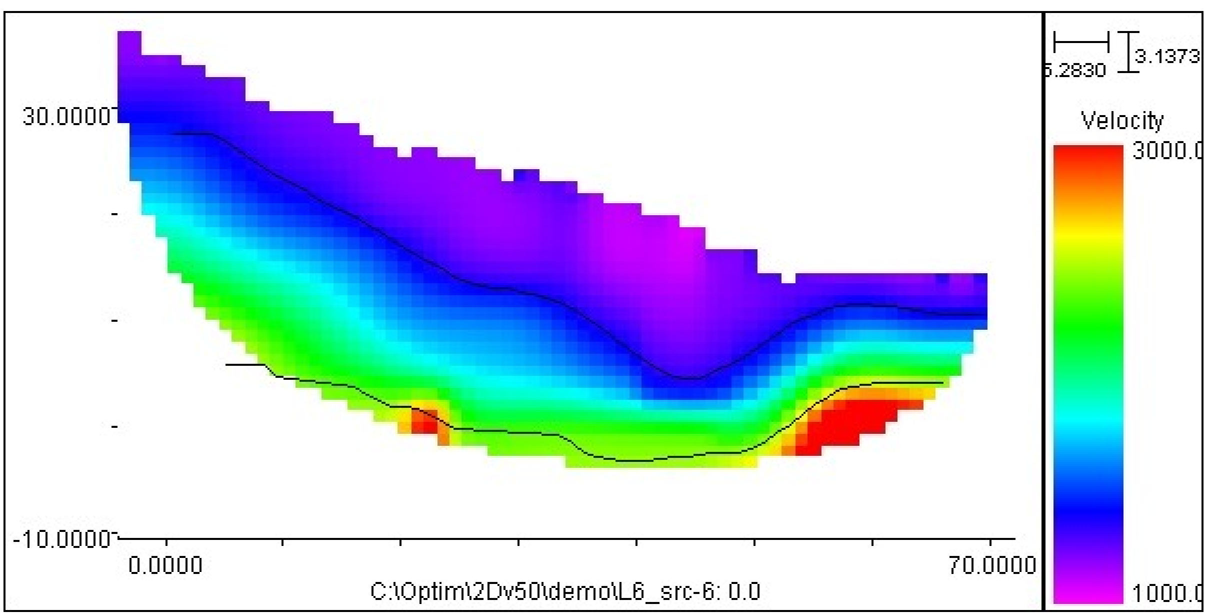


Figure 26g: JPEG image created using 'Make JPEG'.

7.2 The MakeEPS module: Create Report Quality Encapsulated Postscript files

To create an encapsulated PostScript (EPS) file that can be subsequently edited, use the MakeEPS™ function. Encapsulated 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 http://www.cs.wisc.edu/~ghost/ . GSView is the visual interface to GhostScript. It allows viewing, printing, and manipulating the EPS file for importing into an MS-WORD document (see Section 7.2.1). 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 begin the EPS file creation, click the 'MakeEPS' button, which can be seen when viewing Velfiles or Hitfiles ('Interactive Velocity Graph' view in 'Settings'). This brings up the 'MakeEPS Settings' window, shown in Figure 27. This reads in the 'plotinput' file present in the SeisOpt directory. The plotinput file is created by RIOTS. To load a 'plotinput' file from a previous run, copy the appropriate plotinput file with extension to C:\optim\2dv50\ and rename it as 'plotinput' after deleting the existing 'plotinput' file. The settings in Figure 27 correspond to the file created after the RIOTS run shown in Figure 26g, with the velplot file ('velplot_layer_6') generated by running the DetLayer module (Figure 26). This window is used to set the attributes of the file to be created. The following is a guide to the fields in this window.

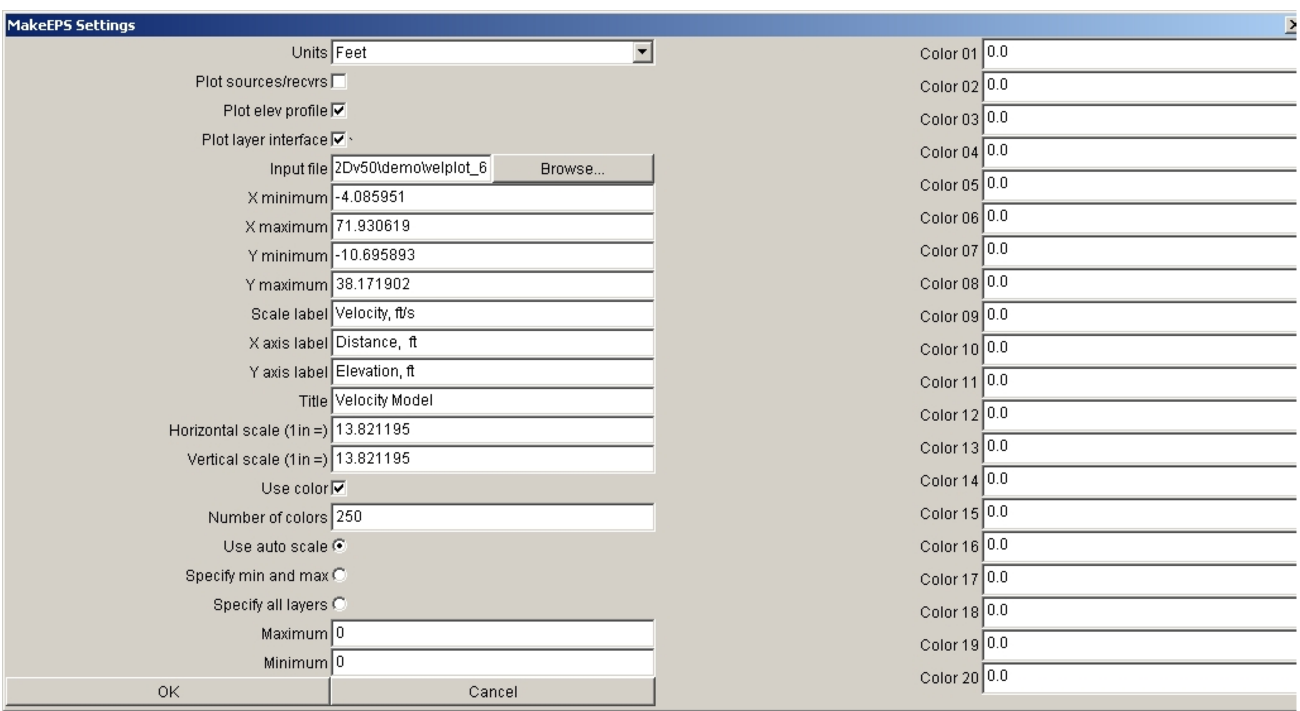


Figure 27: MakeEPS Settings window. Clicking OK will execute the MakeEPS module.

Units: Choose the appropriate units (km, ft, or m). The default unit displayed in the window is of the data set from the last RIOTS run.

Plot sources/recvrs: Click this check box to cause sources and receivers to appear in the file, as they appear in the Surveyfile display.

Plot elev profile: Click this check box to cause the approximate elevation profile to appear in the file, as it does in the Surveyfile display.

Plot layer interface: Click this check box to plot the layer interfaces computed using the 'DetLayer' module on the velocity model. Note: If this is checked, then the appropriate 'velplot' input file (with 'layer' extension, velplot_layer_1 in this case) should be selected.

Input file: Select the path and filename of the file to be used to generate the EPS file, or click 'Browse…' to find the file. These files can be velplot, hitplot, or surveyplot files. **Note**: in addition to the velplot/hitplot/surveyplot file, MakeEPS also requires a 'plotinput' file. Since the optimization process and the interactive survey design process both generate this file, it is normally always present. However, when making EPS plots of data generated during previous SeisOpt@2D sessions, it may be necessary to copy the appropriate 'plotinput' file from the corresponding output directory (it is saved there with the extension used for the optimization) into the SeisOpt@2D directory. For the MakeEPS settings to read it, the file has to be called 'plotinput'.

Also, if the 'Plot layer interface' option is checked, the velplot file with the 'layer' extension must be selected.

Note: The purpose of the following four fields is to select a sub region of the total image for inclusion in the EPS file.

X minimum: Minimum value of x-coordinate (horizontal distance in physical units - ft, m, km) to start plotting the image from. The default setting is the smallest possible x-coordinate value. It should only be increased, if desired.

X maximum: Maximum value of the x-coordinate to stop plotting the image. This number must always be greater than X minimum. Note also that the default setting for this value is slightly greater than the actual total length of the array and is the largest possible value. If changed, it should only be to a smaller value.

Y minimum: Minimum value of the y-coordinate (elevation in physical units) to plot. Note: this refers to the lowest level of the model to include and not the upper limit. It should only be changed to a numerically larger value, if needed.

Y maximum: Maximum value of the y-coordinate (elevation) to plot. Note: this refers to the upper limit of elevation of the model to include. Any change should be to a smaller value.

Scale Label: Enter the label to use for the scale bar (e.g., Velocity, km/s).

X-axis label: Label for the x (horizontal) axis. (e.g., Distance, km).

Y-axis label: Label for the y (vertical) axis (e.g., Distance, km).

Title: Enter the title that will appear on the plot.

Horizontal Scale: This value specifies how the horizontal extent of the image maps to the actual physical units. In particular, 1 inch on the plotted image corresponds to the value specified by this entry in units of distance used during the RIOTS run. Smaller this value larger the horizontal scale of the image. One can control the aspect ratio of the image by manipulating this, along with the vertical scale.

Vertical Scale: This value specifies how the vertical extent of the image maps to the actual physical units. In particular, 1 inch on the plotted image corresponds to the value specified by this entry in units of distance used during the RIOTS run. Smaller this value larger the vertical scale of the image. One can control the aspect ratio of the image by manipulating this, along with the horizontal scale.

Use Color: Check this box to create a color EPS file.

The following parameters can be used in different combinations to create 'layered' or 'contoured' images of the velocity model.

Combination 1 (Figure 27a)

- **Number of Colors:** Enter the number of colors to use in the model. This can range from 2 to 255.
- **Use Auto scale**: Select this option to use the default amplitude scale limits.

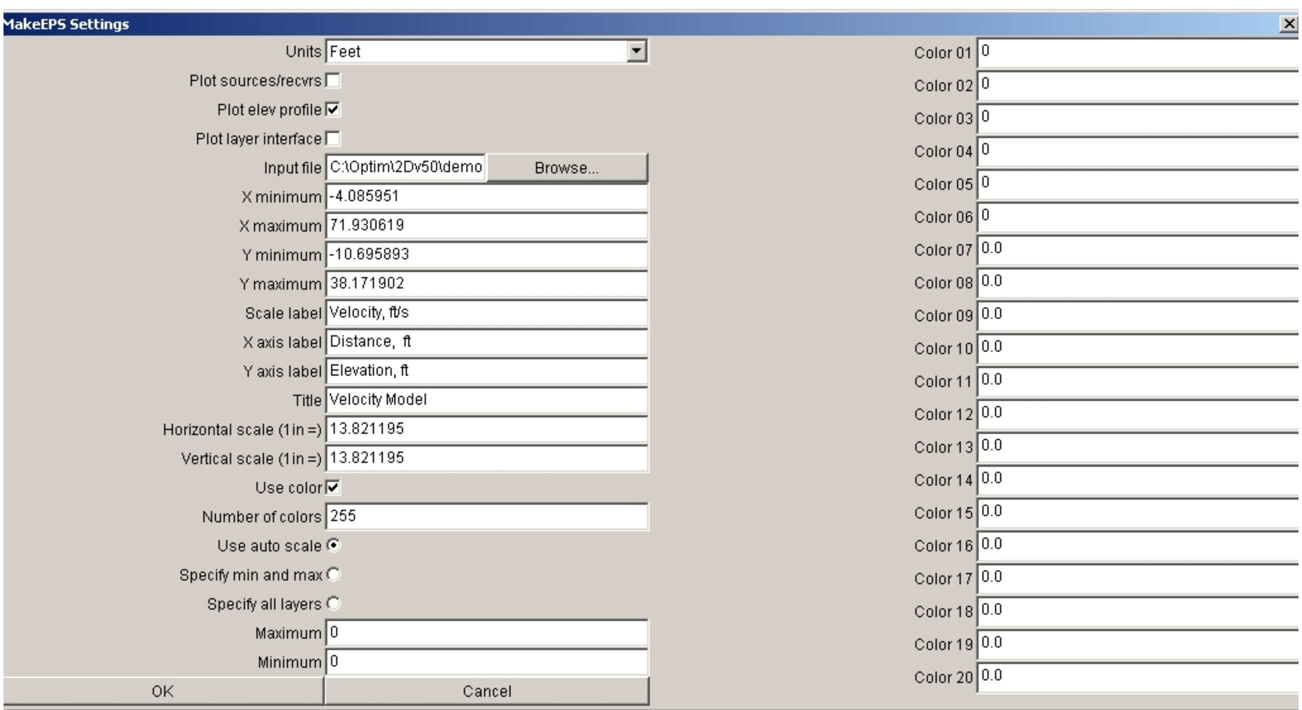


Figure 27b: MakeEPS settings for creating EPS file using Combination 1

Combination 2 (Figure 27b)

- **Number of Colors:** Enter the number of colors to use in the model. This can range from 2 to 255.
- **Use Auto scale**: De-select this option to use the default amplitude scale limits.
- **Specify min and max**: Enter minimum and maximum velocities for the model after de-selecting ‘Use Auto scale’.

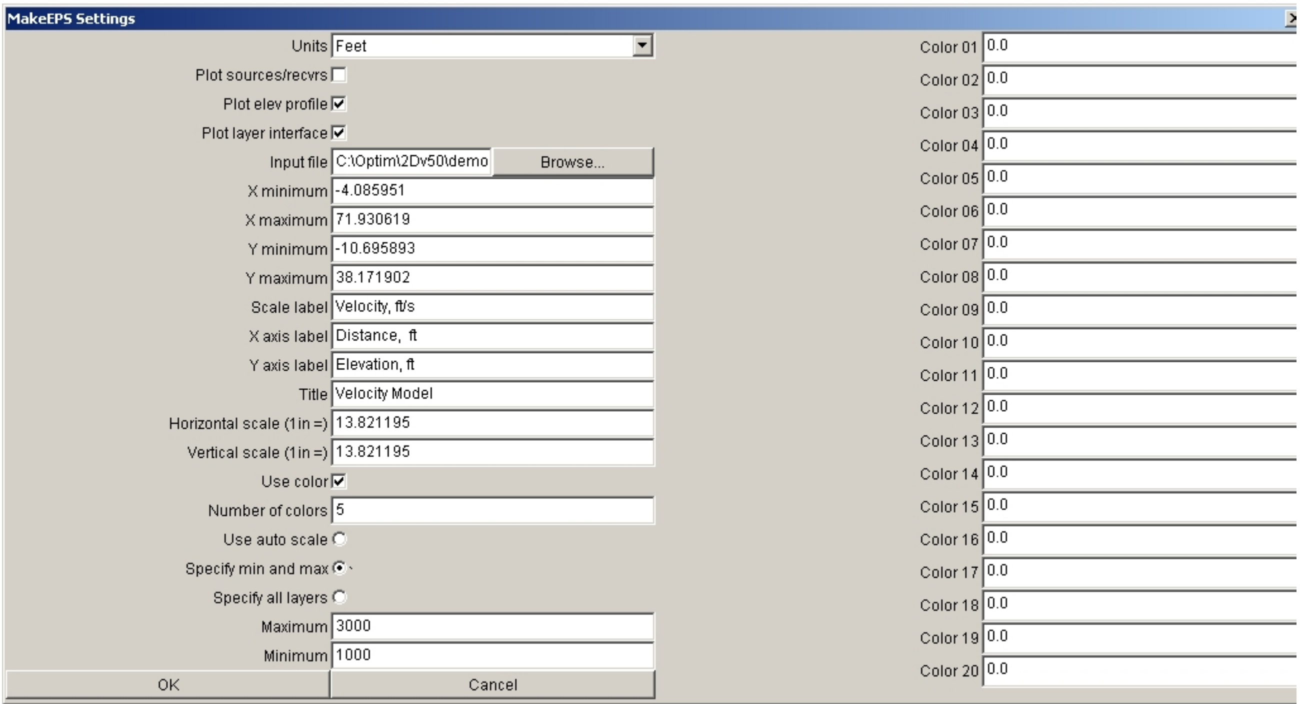
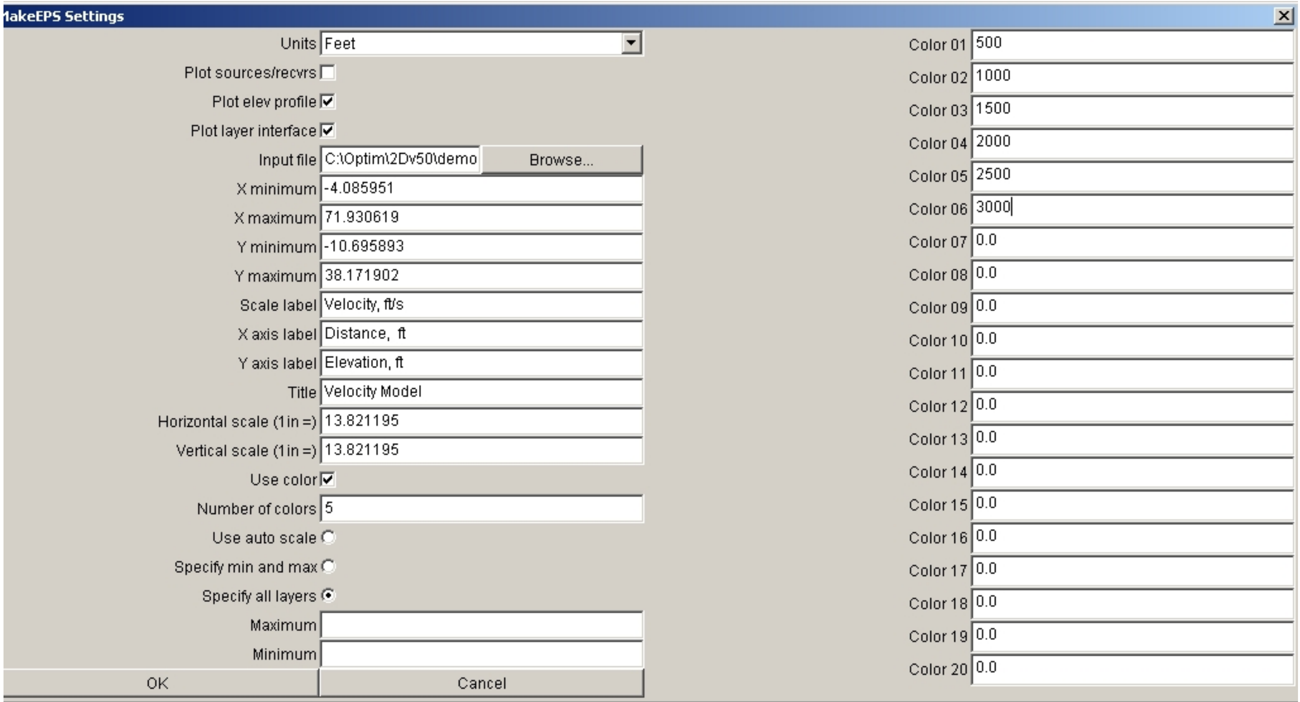


Figure 27b: MakeEPS settings for creating EPS file using Combination 2

Combination 3 (Figure 27c)

- **Number of Colors:** Enter the number of colors to use in the model. This can range from 2 to 255.
- **Specify all layers:** Select this option
- **Color01, Color 02,…Color 20:** Enter the velocity ranges for each color, one more than the number of colors specified in ‘Number of Colors’. For example if the ‘Number of Colors’ specified is 5, the user needs to enter six color values (up to maximum of 20). The resulting EPS plot will have 5 colors each representing the velocity ranges specified in Color 01 to Color 02, Color 02 to Color03, Color03 to Color04, Color04 to Color 05 and Color 05 to Color 06. In this example, the five colors will correspond to velocity ranges 500-1000, 1000-1500, 1500-2000, 2000-2500 and 2500-3000 ft/s.



27c: MakeEPS settings for creating EPS file using Combination 3

Once all fields have been set, click 'OK' to create the EPS file. A progress window will open, indicating when the file creation is complete (Figure 28). Additionally, the name of the EPS file appears in this window. Click 'End / Terminate Process' at the bottom of the progress window to close it.

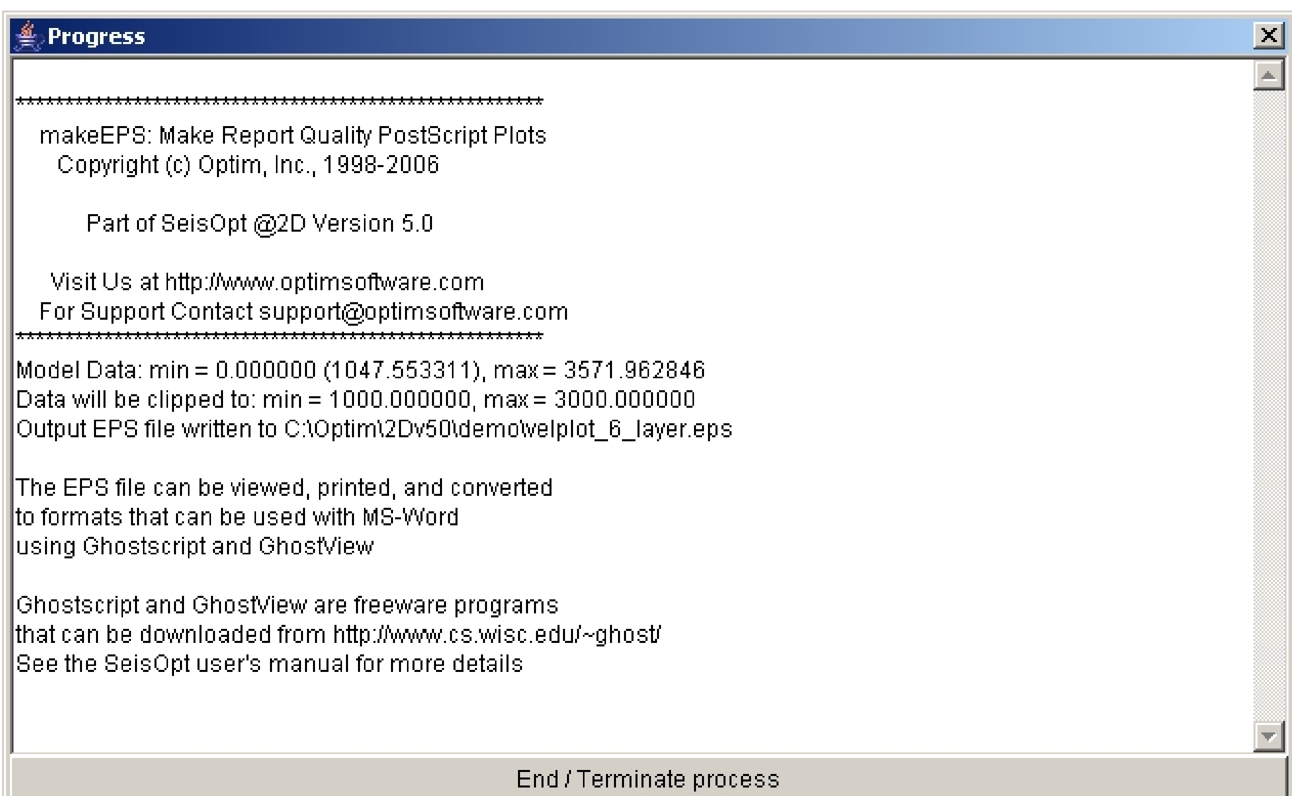


Figure 28: Progress window showing the execution of MakeEPS

The output EPS is always written to the directory from which the input file was read in. For example, in Figure 27, the input file was in subdirectory demo and so the output file is also written to the subdirectory demo. Also note that if the 'Plot sources/recvrs' option is chosen, the output EPS will have a '_c.eps' extension instead of just '.eps' extension. Figure 29a shows an EPS file created with the MakeEPS Settings shown in Figure 27b (Combination 2). This has the 'Plot layer interface' option chosen. If this is not checked, the layer interfaces will not be plotted on top of the velocity model. Figure 29b shows the EPS file created using settings specified in 'Combination 3' (Figure 27c).

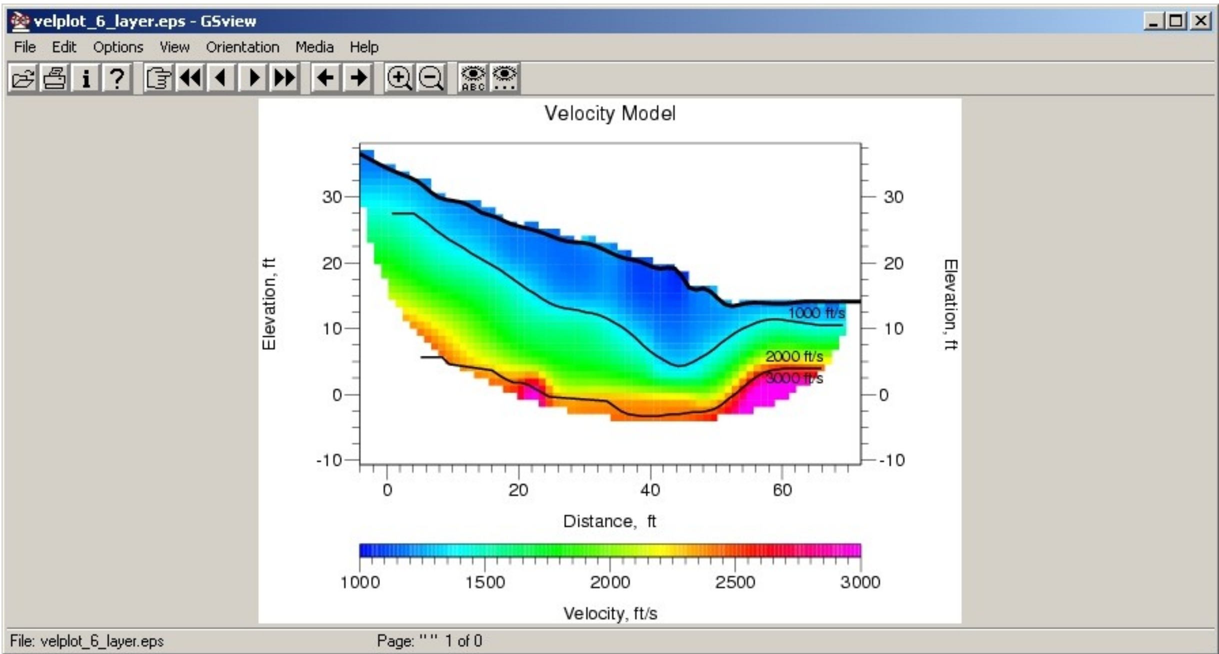


Figure 29a: Output from the MakeEPS module showing an optimized velocity model loaded into GSView program. The plot was created using settings specified by 'Combination 2'.

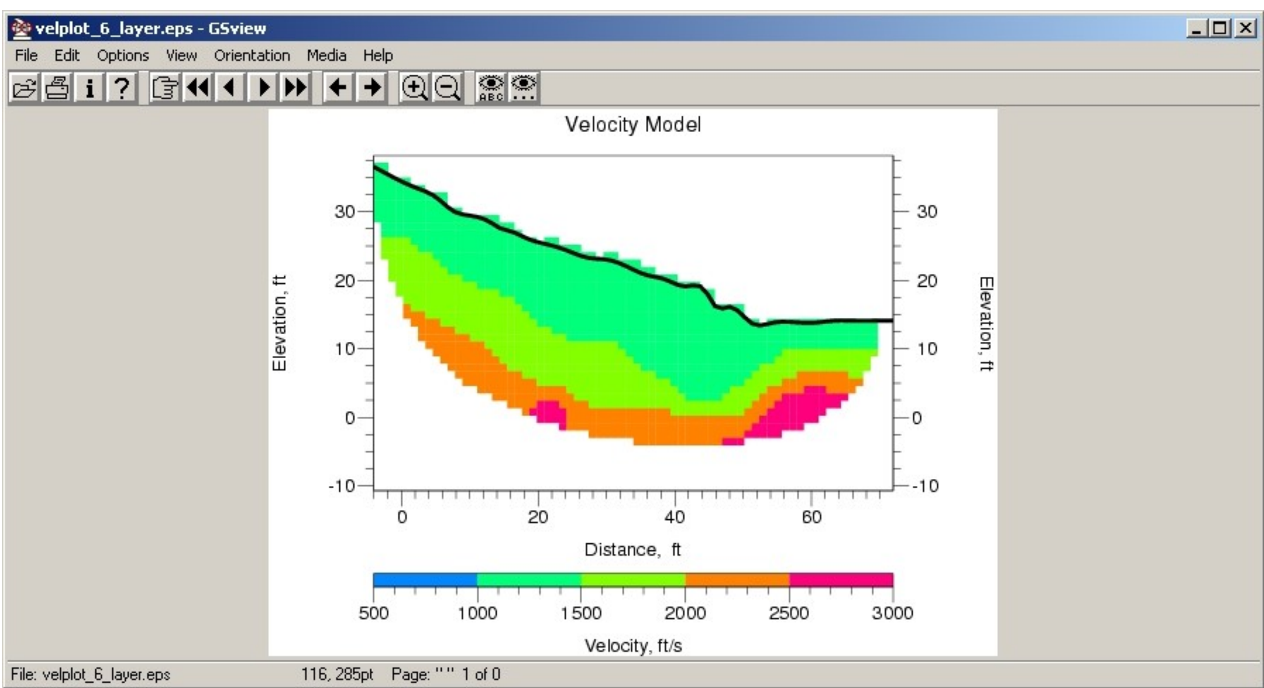


Figure 29a: Output from the MakeEPS module created using 'Combination 3' settings.

7.2.1 Importing EPS files into an MS WORD document

The EPS file created using the MakeEPS module can be modified using the freeware program GhostScript/GSView (see Section 7.2, 1st paragraph) and inserted into an MS-WORD document. There are two ways to do this.

The first method is to convert the EPS file into bitmap format using GSView. Here are the steps to that:

1. Open the EPS file (.eps) written out by SeisOpt @2D using GSView
2. Go to the "File" menu and select the "Convert" option. A dialog window will open up.
3. In this window choose 'bmp256' for the 'Device' option and '120' for the "Resolution" option (Figure 29a). You will notice there are other options for file formats. You can try these if you have a drawing program that will render them. The reason to choose bitmap is because it can be rendered using MSPaint, which is supplied with all Windows OS.

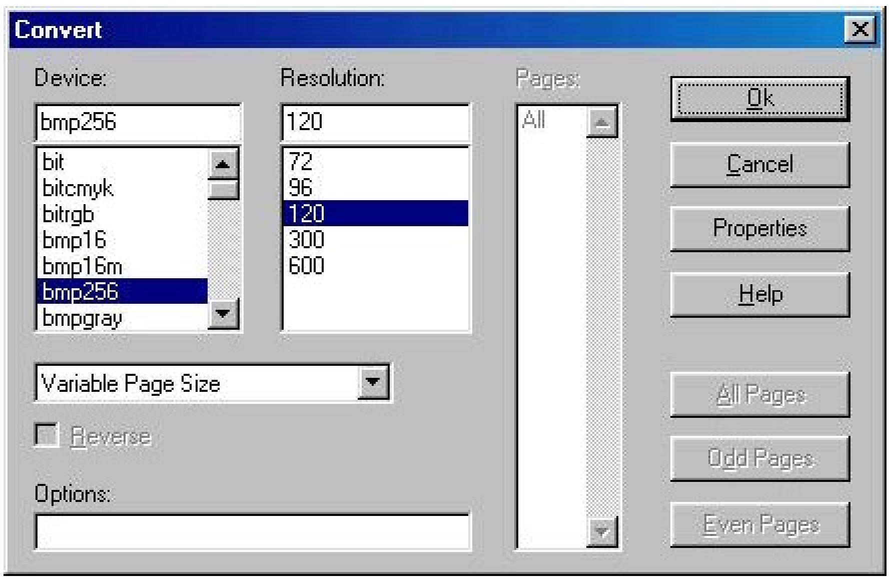


Figure 29a: Choose 'bmp256' for the Device option and '120' for the Resolution. One can go with higher resolution if desired. Note that higher the resolution large the file size.

4. Click on "Properties" button to set the page offsets as shown below. This will ensure the image, when opened up using MSPaint will be rendered at the top left corner. These values can be changed to suit the page size.

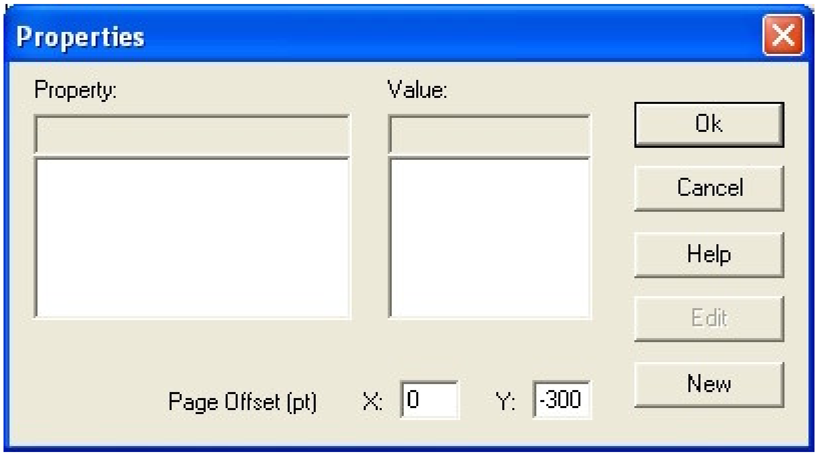


Figure 29b: Click on the 'Properties' window to bring up the above window. Set the 'Page Offset' as shown so the image is rendered in the left top corner of the page.

5. Save the file name with a '.bmp' extension. Load the file into MSPaint (just double-click on it and it should open up in MSPaint). You may have to crop the image so as to remove the white spaces around the image. If you have problems contact support@optimsoftware.com to find out how to do this.

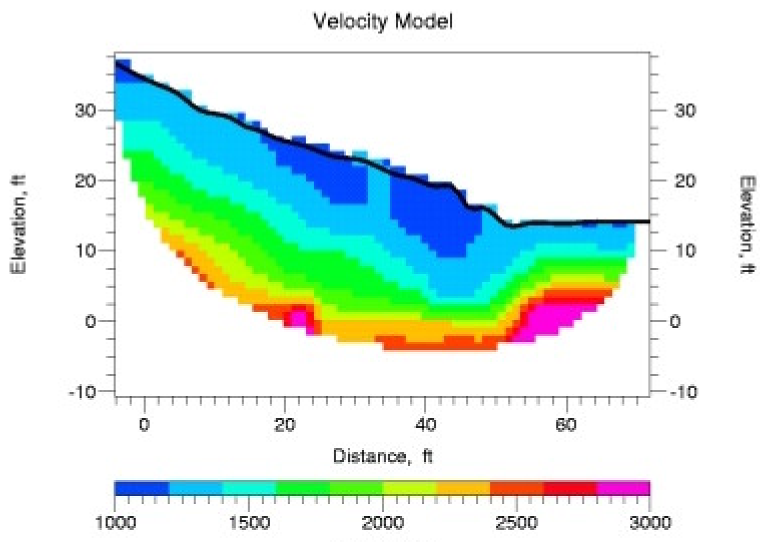


Figure 29c: Bitmap image created using steps 1-5. This can be imported into a MS-WORD document.

6. The bitmap image (Figure 29c) can then be imported into an MS-WORD document.

The second 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:

a. Open the EPS file (.eps) written out by SeisOpt @2D using GSView.
b. Go to the ''Options'' menu and select 'EPS Clip' and 'Show Bounding Box' options.
c. You will notice that the 'Bounding Box' (dashed box) now surrounds only the image.
d. 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).
e. Provide a new file name for the EPS file with the preview. Make sure you type in the extension '.eps' for the file name.

f. Save the file as an EPS file.

Now, you will be able to insert the EPS file saved in step ‘f’ into an MS-Word document using the insert-picture option. Once you have it Word, you should be able to print it to your printer.

7.3 Using Surfer™ to create contour plots

SeisOpt @2D creates output files that can be imported into “Surfer™” for plotting purposes. The files that can be used with Surfer are:

1. “velvalues” (three column ASCII file containing the velocity values in (x,z,vel) format)
2. model_[ext].bln - blanking file that blanks model below ray coverage
3. elev_{ext}.bln - blanking file that blanks model above topography
4. postingfile_[ext].txt - shot/geophone labels.
5. Also if using “DetLayer” module, a "layer_[ext].bln" file is written that can be imported into Surfer for plotting the layer interfaces.

8.0 SeisOpt Tuner, the Tuning Module

SeisOpt Tuner module has several functions. The user can use it to (1) fine tune an optimized velocity model to make the fits between the observed and calculated fits tighter (2) construct a new velocity model and use it with the survey design module to visualize subsurface sampling. (3) Re-run the optimization using a variable grid mesh. The third option is useful if there are large ‘holes’ in the model or when the off-end source-receiver offsets are very large.

8.1 Tuning an Optimized Velocity Model

The buttons used for tuning an optimized velocity model are present along the left-hand side of the SeisOpt@2D user interface and the ‘Dimension Chooser’ button along the top. (Figure 40).

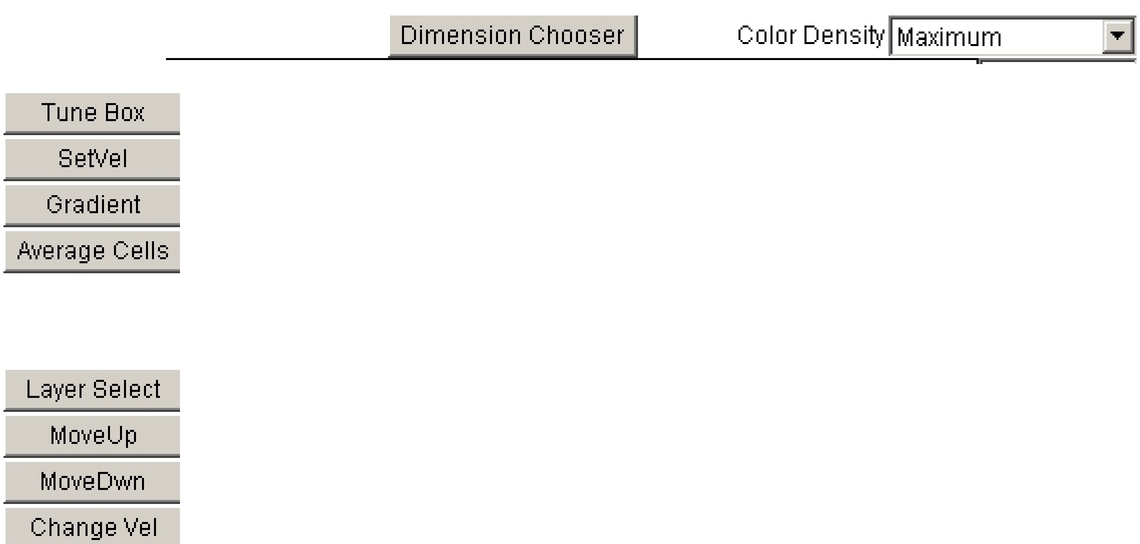


Figure 40: Buttons to use for tuning an optimized velocity model or constructing a new velocity model

Steps to tune a velocity model using 'Tune Box'

The following steps show how to change the velocity of an area within the velocity model to be tuned either by a constant new velocity, adding a gradient or averaging velocities within the tune polygon.

- Load and display the desired optimized velocity model (see Section 5.3.1) to be tuned
- Make sure that the appropriate 'riotsinput' file (without extension) that generated the displayed velocity model is present in C:\optim\2dv50\ folder. To be sure click on 'RIOTS Settings' and verify if the input parameters and input files are correct.
- Click on 'Tune Box' (the button labeling will change to 'Boxing') and draw any shaped polygon by clicking the left mouse button, around the area to be tuned (Figure 40b).
- The grids that will be affected will be shaded.
- Hit <ENTER>.
- The polygon boundaries will turn from grey to black.

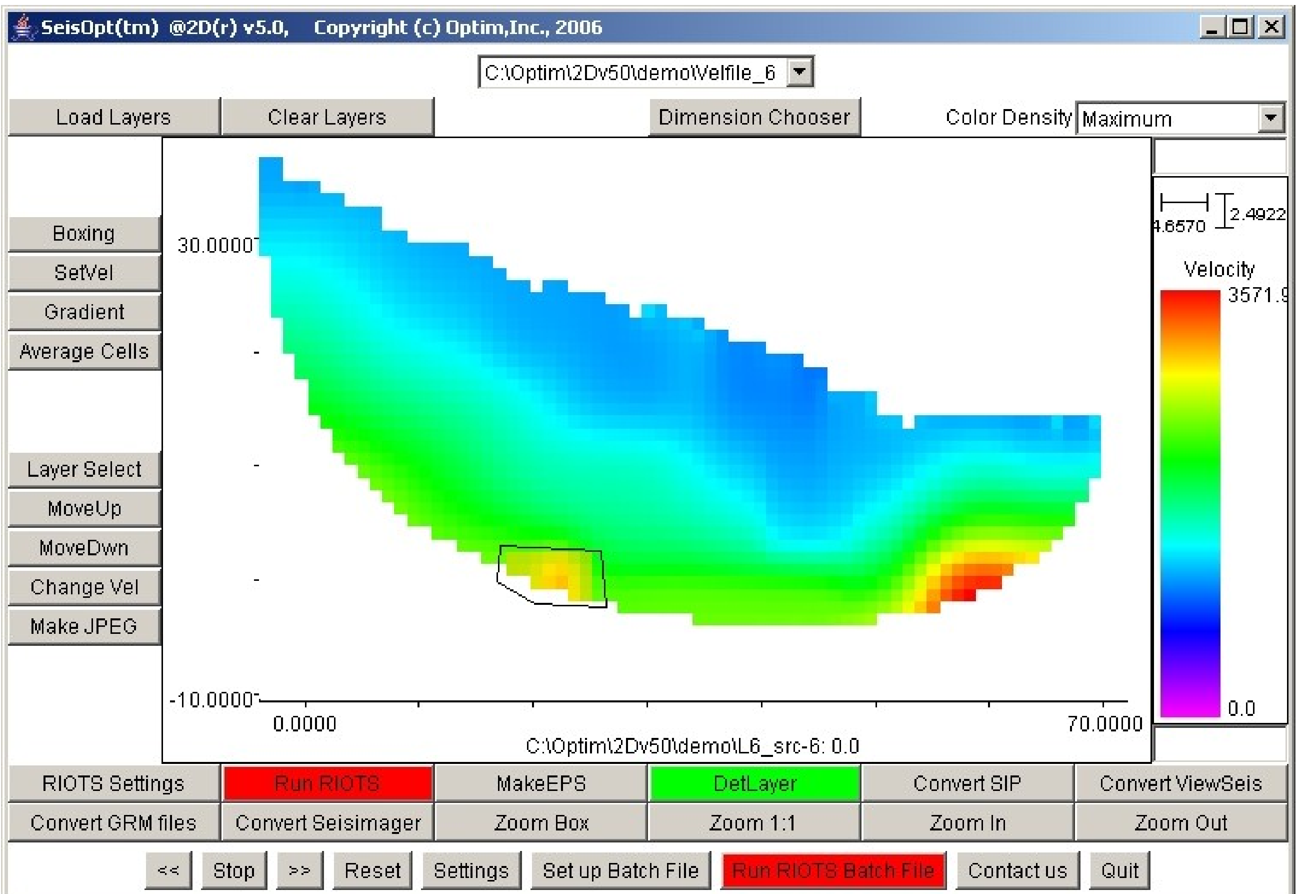


Figure 40b: Click on 'Tune Box' button and draw a box around the region to be tuned.

- To enter a new constant velocity, click on 'SetVel' button. The window shown in Figure 41 opens up

Figure 41: Window to enter new constant velocity for area to be tuned

- Enter the value of the new constant velocity for the area to be tuned
- Click 'Done' to execute the change (Figure 41b).

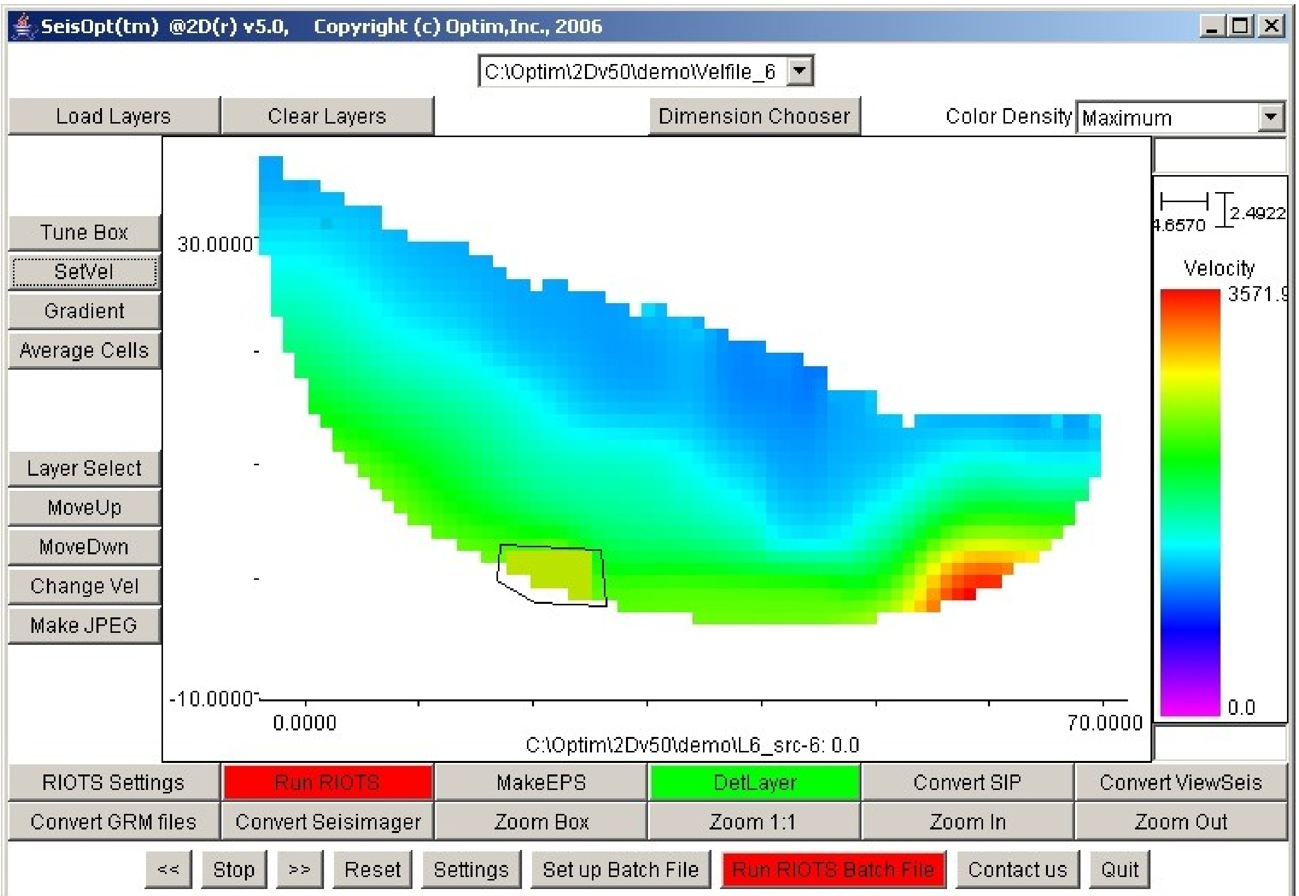


Figure 41b: Velocity value inside the 'Tune Box' changes to the entered value.

- To enter a gradient, click the 'Gradient' button. The window shown in Figure 42 opens up

Figure 42: Window to enter values to calculate the new velocity gradient for the area to be tuned

- Enter the velocity for the top of the area to be tuned next to 'Top Velocity' and velocity for the bottom of the area next to 'Bottom Velocity'
- Click 'Done' to execute the changes
- If you wish to average the velocities with the polygon, click the 'Average Cells' button.

- Click again on ‘Tune Box’ to complete the area selection
- The modifications to the velocity model are saved in ‘v.final’ file that is written to C:\optim\2dv50\
- Click on ‘RIOTS Settings’ to bring up the window shown in Figure 42a. Change the output file extension, if needed, to prevent overwriting any existing files. In this example the extension has been changed to ‘6_tune’.
- Make sure the ‘Input v.final file’ is set to ‘v.final’ and the ‘Input velplot file’ is set to the appropriate velplot file (in this case it will be velplot_6 since we are tuning the output from run 6 of batch processing).
- Click ‘Set Iterations’ and change the ‘Max. Iteration’ to 1 to run the model just once and view the effects of tuning (Figure 42a). If you wish to run the model through the full optimization do not check the ‘Set Iterations’ box.
- Click ‘OK’
- Click ‘Run RIOTS’ to run the optimization using the tune velocity model as input.
- The progress window shown in Figure 43 opens up. Note that the optimization was run for only one iteration.
- Load the new Velfile, Hitfile and Pickfile
- Verify if tuning had the desired effect by comparing the Pickfiles of the tuned and those of the model before tuning and by comparing the least-square errors of the final models.

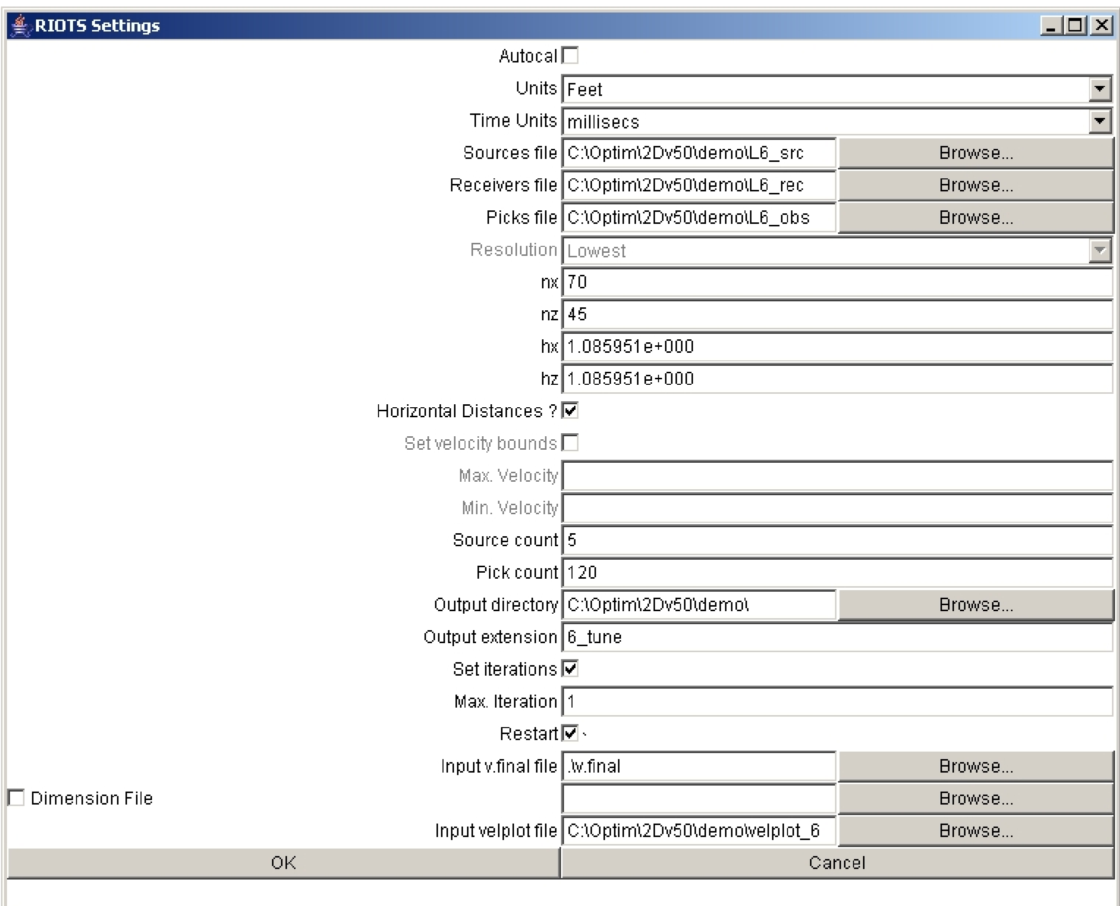


Figure 42a: ‘RIOTS Settings’ parameters for performing the optimization on the tuned velocity model.

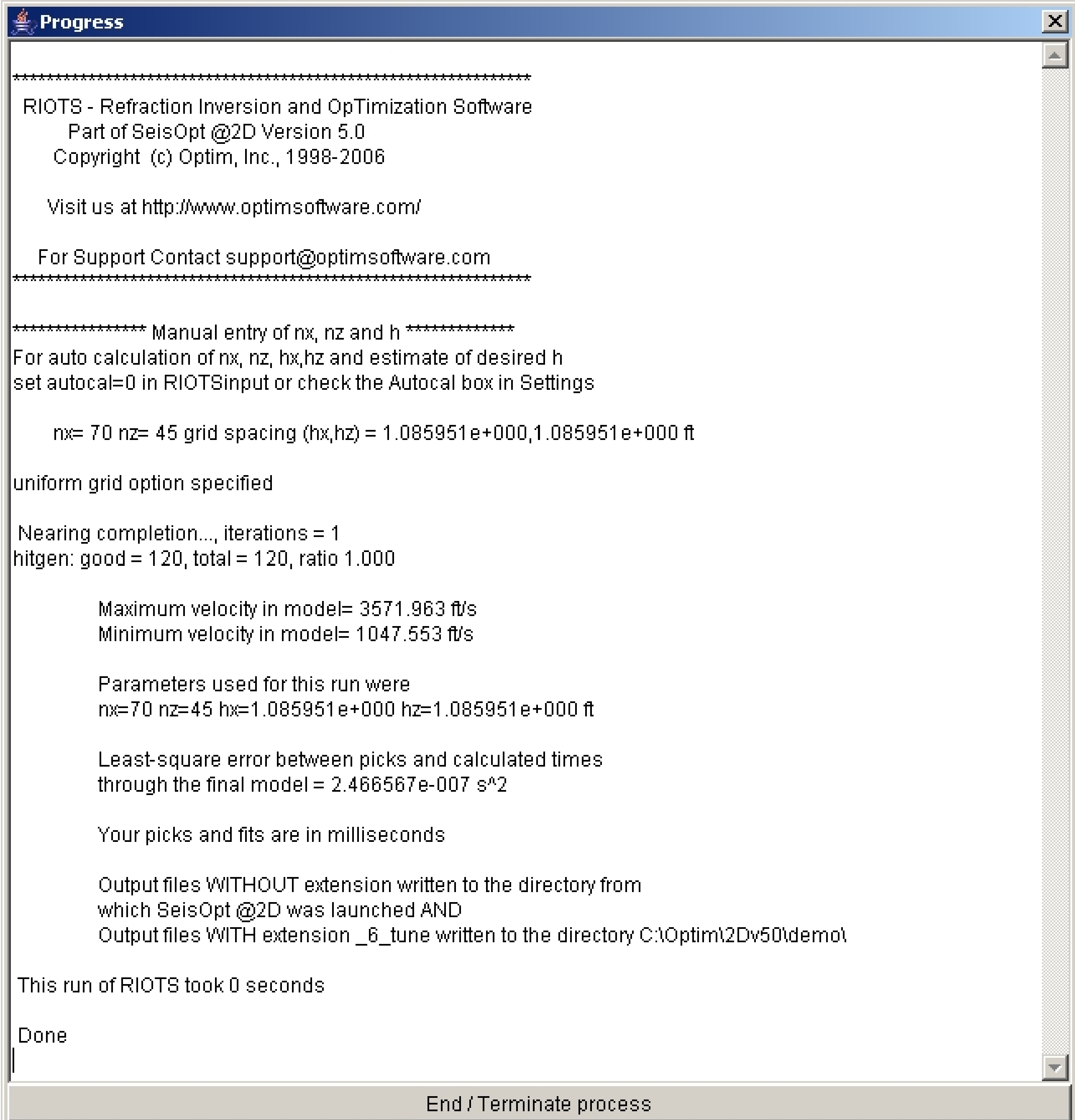


Figure 43: Progress window showing completion of the 'RIOTS' run for one iteration using the tuned model as input.

Steps to tune a velocity model using 'Layer Select'

The following steps enumerate how to choose a layer in the velocity model to be tuned, move the layer up, down, or change just change the velocity of the layer:

- Load and display the desired optimized velocity model (see Section 5.3.1) to be tuned
- Make sure that the appropriate 'riotsinput' file that generated the displayed velocity model is present in the installation directory. To be sure click on 'RIOTS Settings' and verify if the input parameters and input files are correct.

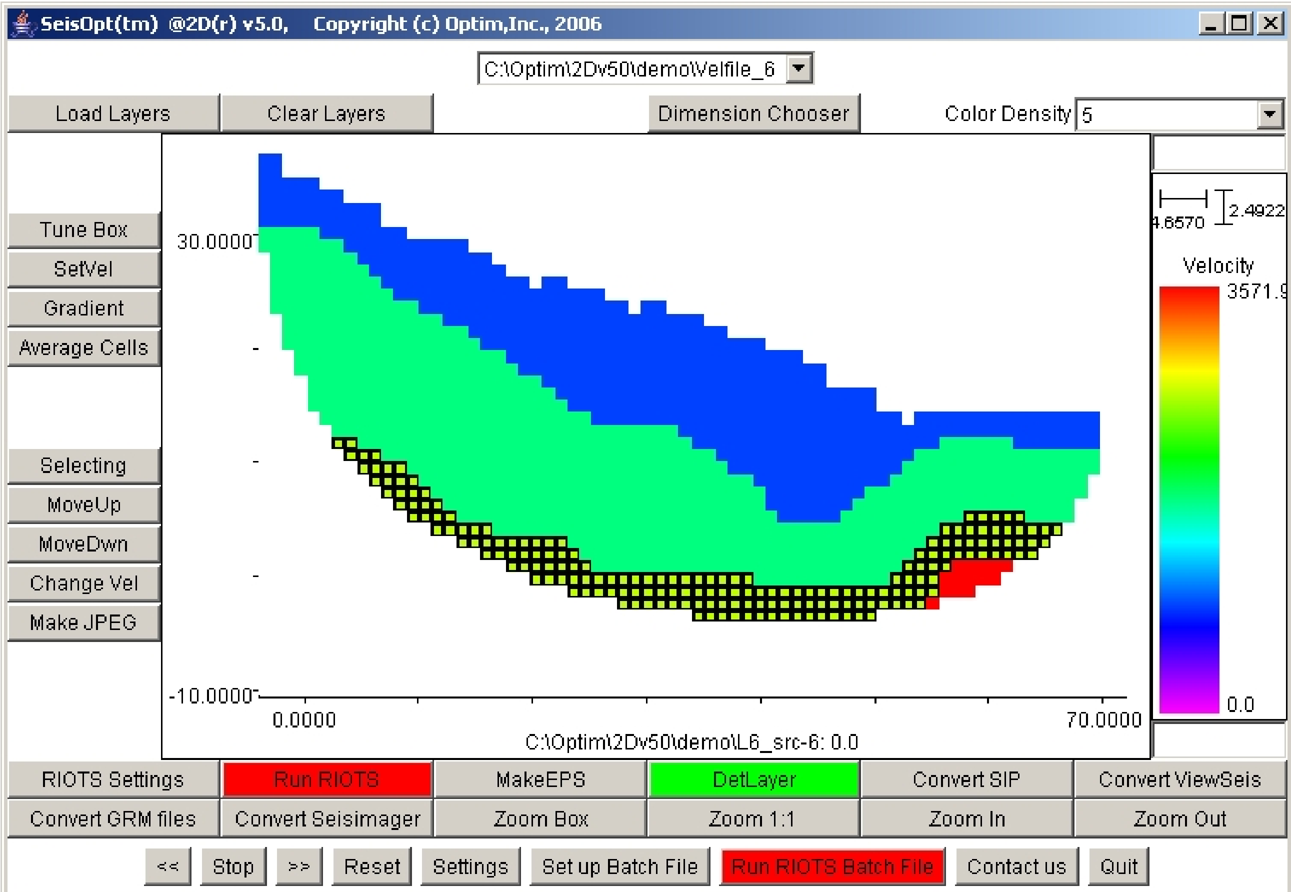


Figure 44: Velocity model after the 'Layer Select' action

- Use the 'Layer Density' pull-down menu (Figure 17) and change the number of layers in the model. Choose among the different layer density options and select the number that best shows the layer which has to be moved or whose velocity has to be tuned
- Click on 'Layer Select' and click on the layer which has to be moved or whose velocity has to be changed. Figure 44 shows an example of a velocity model after such an action
- Click on 'MoveUp' to move the selected layer up
- Click on 'MoveDwn' to move the selected layer down

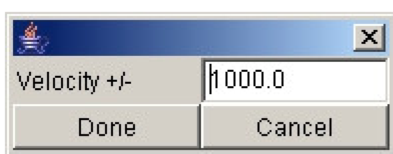


Figure 45: Increment or decrement the velocity of the chosen layer by a specific amount

- Click on 'Change Vel' in order to increment or decrement the velocity of the layer. Figure 45 shows the window that opens up when this option is chosen. To increase the velocity in the layer, enter the value by which it has to be increased. To decrease the velocity, enter the negative velocity value by which it has to be reduced
- Click on 'Done' to execute the changes
- Click again on 'Layer Select' to stop the layer selection process

- Click on ‘RIOTS Settings’ to bring up the window shown in Figure 42a. Change the output file extension, if needed, to prevent overwriting any existing files.
- Make sure the ‘Input v.final file’ is set to ‘v.final’ and the ‘Input velplot file’ is set to the appropriate velplot file (in this case it will be velplot_6 since we are tuning the output from run 6 of batch processing).
- Click ‘Set Iterations’ and change the ‘Max. Iteration’ to 1 to run the model just once and view the effects of tuning (Figure 42a). If you wish to run the model through the full optimization do not check the ‘Set Iterations’ box.
- Click ‘OK’
- Click ‘Run RIOTS’ to run the optimization using the tune velocity model as input.
- Load the new Velfile, Hitfile and Pickfile
- Verify if tuning had the desired effect by comparing the Pickfiles of the tuned and those of the model before tuning and by comparing the least-square errors of the final models.

8.2 Creating a New Velocity Model

The steps outlined in **Section 8.1** can be used to create a new velocity model. The ‘v.final’ file created during this process can then be used as an initial model in a RIOTS optimization run (see **Section 4.6**), if needed. The Hitfile generated when the tuned model is run can then be used to determine the subsurface sampling and if the desired target has been sampled by the deployed array. If the array geometry needs to be changed, load the Surveyfile created from this new model into the ISD view (see Section 5.3.4). Perform an interactive and/or automatic survey design to optimize the array geometry in order to sample the desired target depth. Thus, this can be used as a ‘‘bidding’’ module.

8.3 Using Dimension Chooser and the Variable grid optimization process

SeisOpt velocity optimizations by default use uniform grids. That is the hx and hz values are the same through out the entire model. Now it is possible to change this and use variable grid spacing to define the model space. <u>It is recommended to always use the velocity model (v.final and velplot) files from a uniform grid spacing run as input to a variable grid spacing optimization.</u>

Here are the steps to define a variable grid model:

- Click on ‘Dimension Chooser’ button.
- The Window shown in Figure 46 opens up

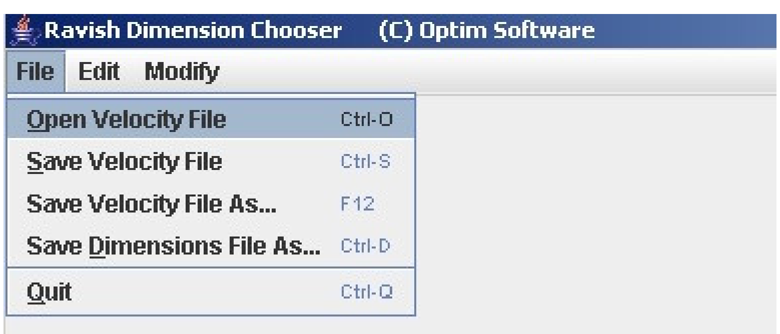


Figure 46: Click on ‘Dimension Chooser’ to bring up the window to specify variable grids

- Select the Velfile generated from the uniform grid spacing run (File-> Open Velocity File, Figure 47)

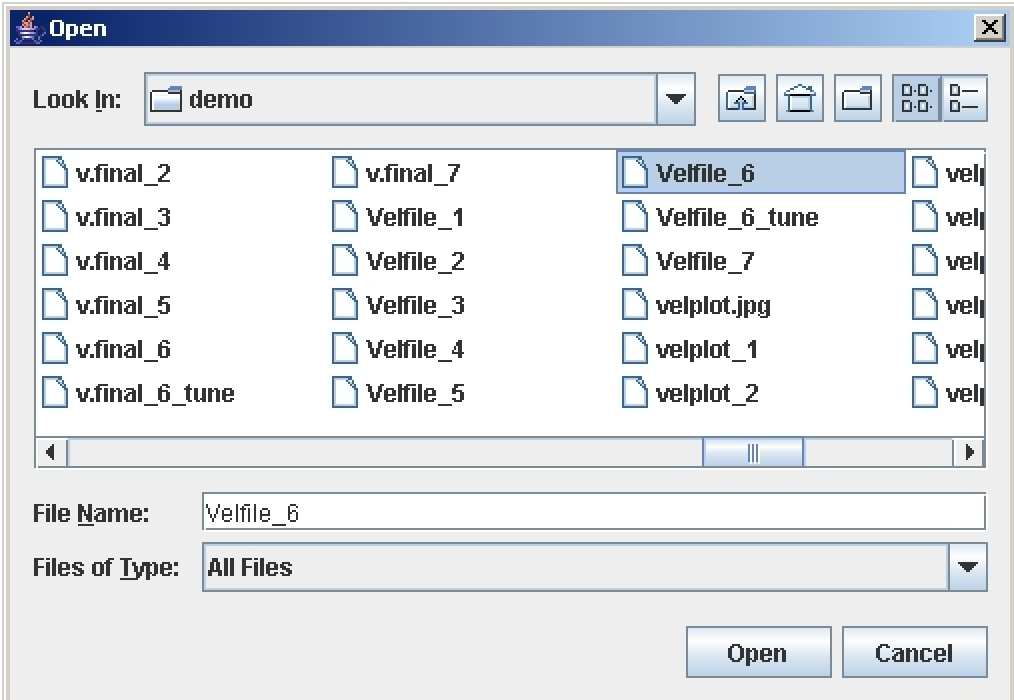


Figure 47: Select the uniform grid Velfile that needs to be modified.

The plot shown in Figure 48 is rendered showing the grid mesh of the input Velfile. Note that the colors do not represent velocities.

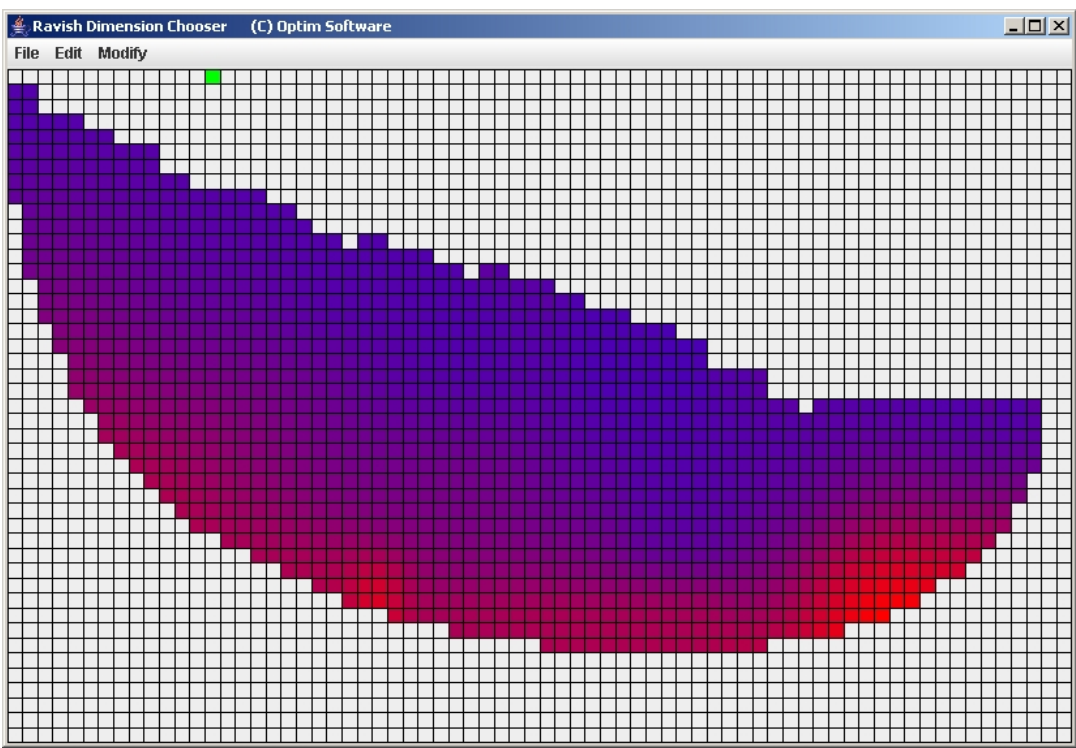


Figure 47: Uniform grid Velfile is loaded and can be modified to have variable grid sizes.

- To modify the grid spacing, select the rows or columns that need to be changed and then choose from among different options under Modify menu (Figure 49). If multiple rows or columns are to be changed then make sure the 'CTRL' button on the keyboard is pressed while selecting the desired rows and columns.

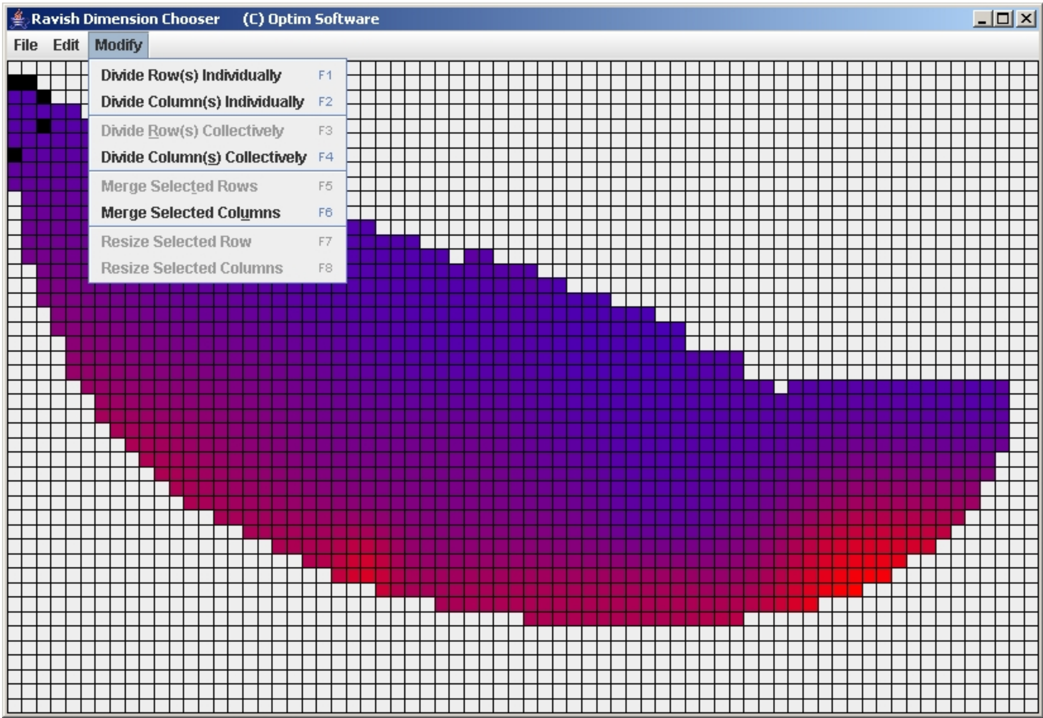


Figure 49: Choose the different options under the Modify menu to specify the desired variable grid mesh.

- The possible modification options are:
  - o Divide the selected rows: This option will increase the vertical resolution of the model by increasing the number of cells in the vertical direction. This operation can either be done for each row (divide selected rows individually option) or at once on several rows (collectively option).
  - o Divide the selected column: This option will increase the horizontal resolution of the model by increasing the number of cells in the horizontal direction. As with rows, this operation can either be done on each column individually or on several columns at the same time (collectively) if more than one column is selected.
  - o Merge selected rows: This operation will decrease the vertical resolution by merging selected rows.
  - o Merge selected columns: This operation will increase the horizontal resolution by merging selected columns.
  - o Re-size selected row: One can enter the physical dimension of the cells in the vertical direction using this option. For example if the uniform run had an hz of 1.0, this option can be selected (for one row only) and hz changed to 0.5 or 1.5 or any other number.
  - o Re-seize selected column: One can enter the physical dimension of the cells in the horizontal direction using this option. For example if the uniform run had an hx of

1.0, this option can be selected (for one column only) and hz changed to 0.5 or 1.5 or any other number.

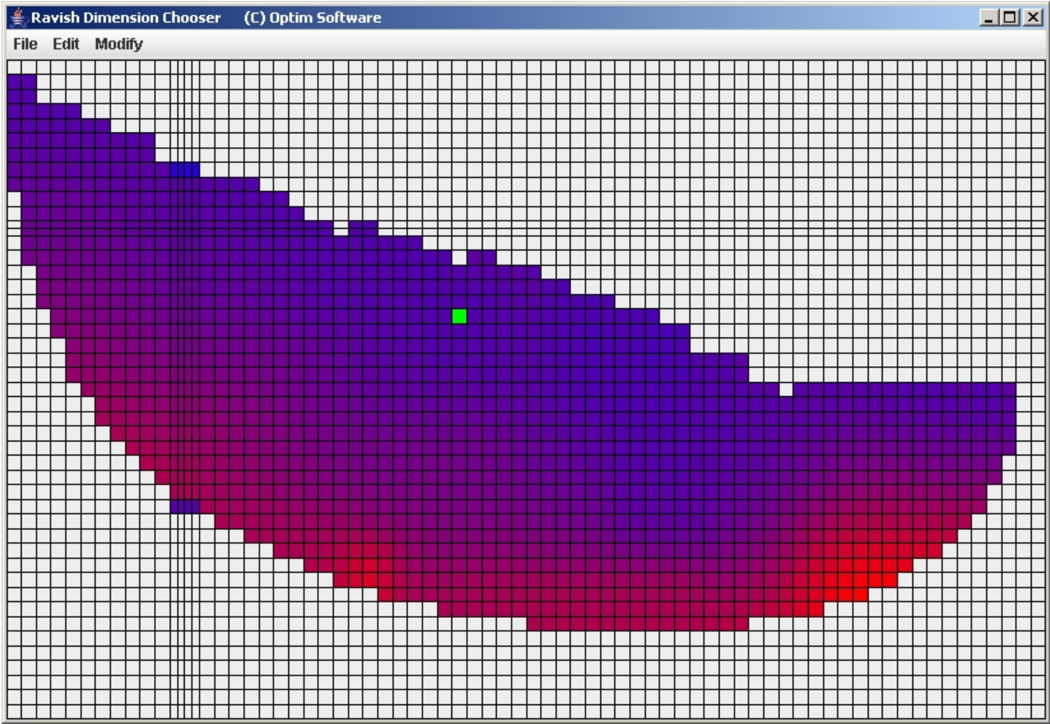


Figure 50: Variable grid mesh showing the effect of dividing rows and columns which increases the vertical and horizontal resolution respectively.

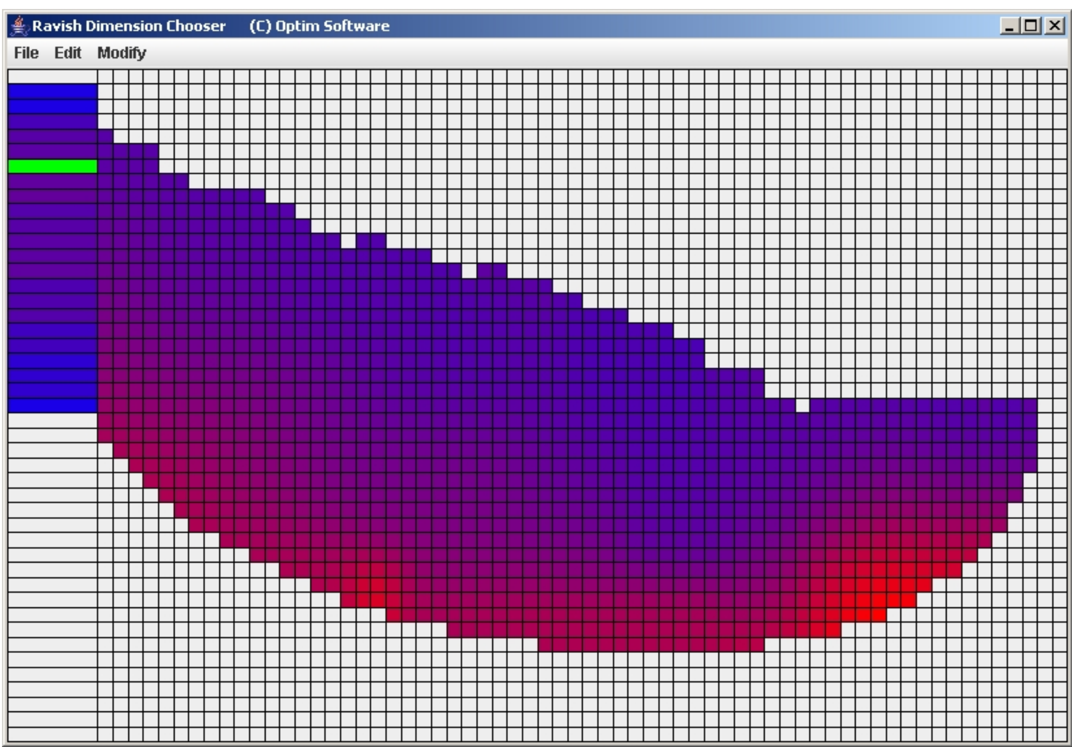


Figure 51: Variable grid mesh showing the effect of merging columns which decreases the horizontal resolution. Similarly rows can be merged to decrease vertical resolution.

Figure 52: The module allows user to specify the physical units of the rows (vertical resolution).

- Once the desired changes are made, save the dimension file that contains information about the variable grid mesh. To do this select 'Save dimension file as' under the File menu and specify a name (Figures 53 and 54).

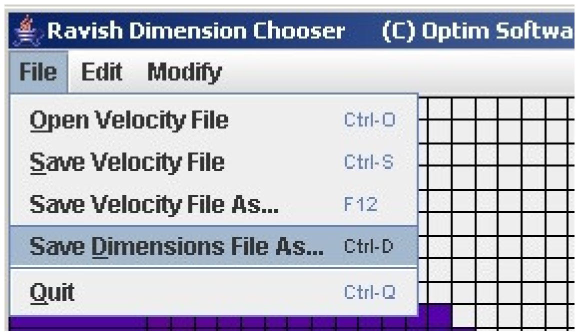


Figure 53: Save the dimension file that needs to be input into 'RIOTS Settings' to run the variable grid optimization.

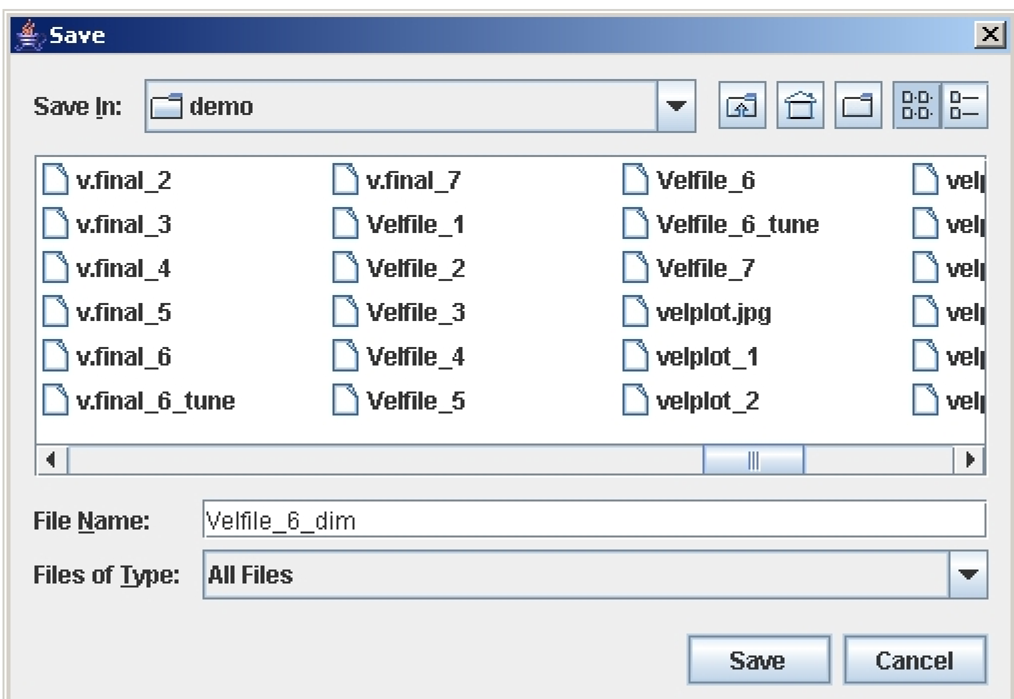


Figure 54: Save the dimension file

- This file is input into 'RIOTS Settings' (Figure 55). Make sure the correct v.final file and velplot file for the uniform run are selected.

- Since the uniform grid velocity model is being used as a constraint, the 'Restart' box should be checked. The 'Set Iteration' box can be check and the 'Max. Iteration' set to 1 to see the effect of the variable grid change. To run the full optimization, do not check the 'Set Iteration' box.
- Click 'Run RIOTS' to run the variable grid optimization (Figure 56) and view the modified Velfile output (Figure 57).

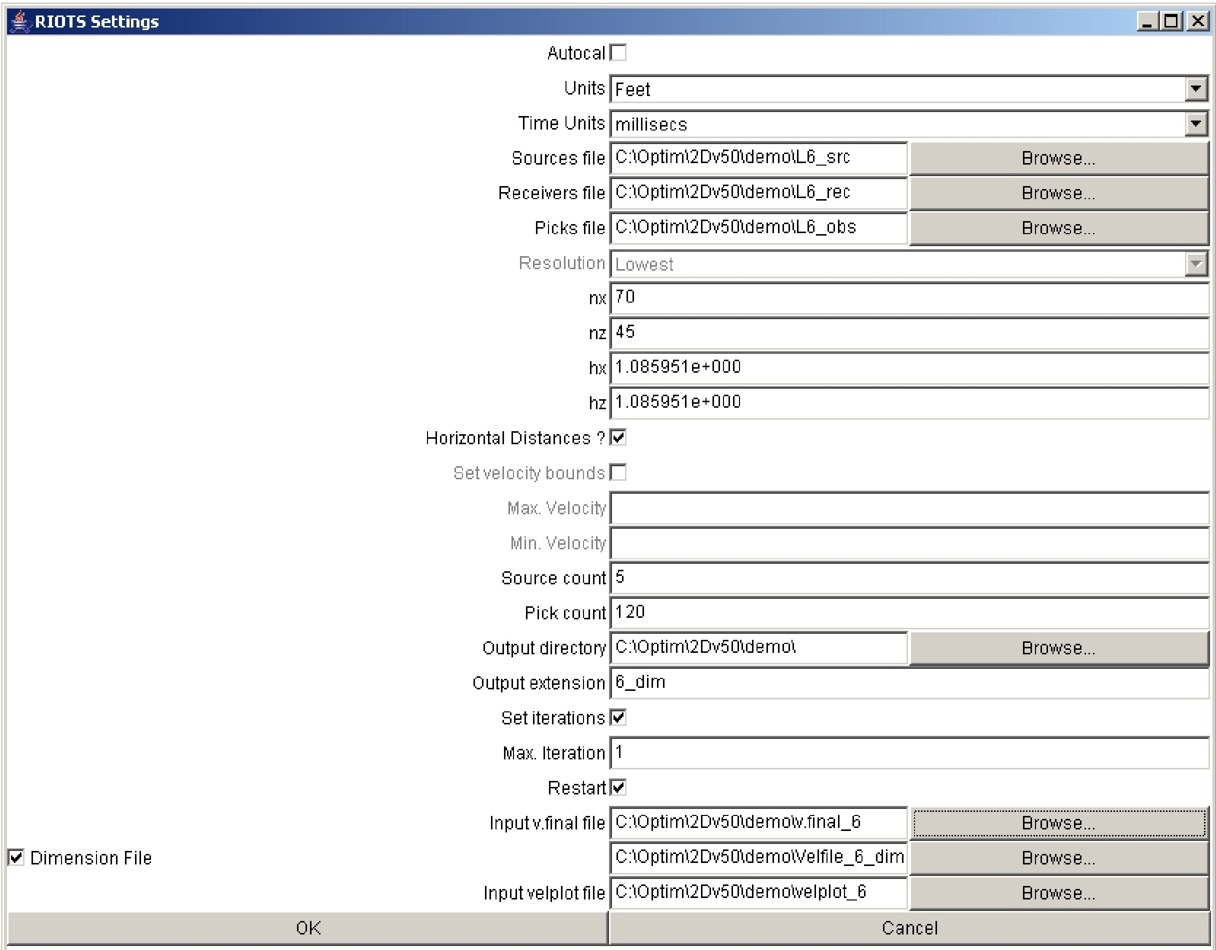


Figure 55: RIOTS Settings window for the variable grid optimization. Choose the appropriate dimension file, v.final, and velplot files. Check the 'Restart' box.

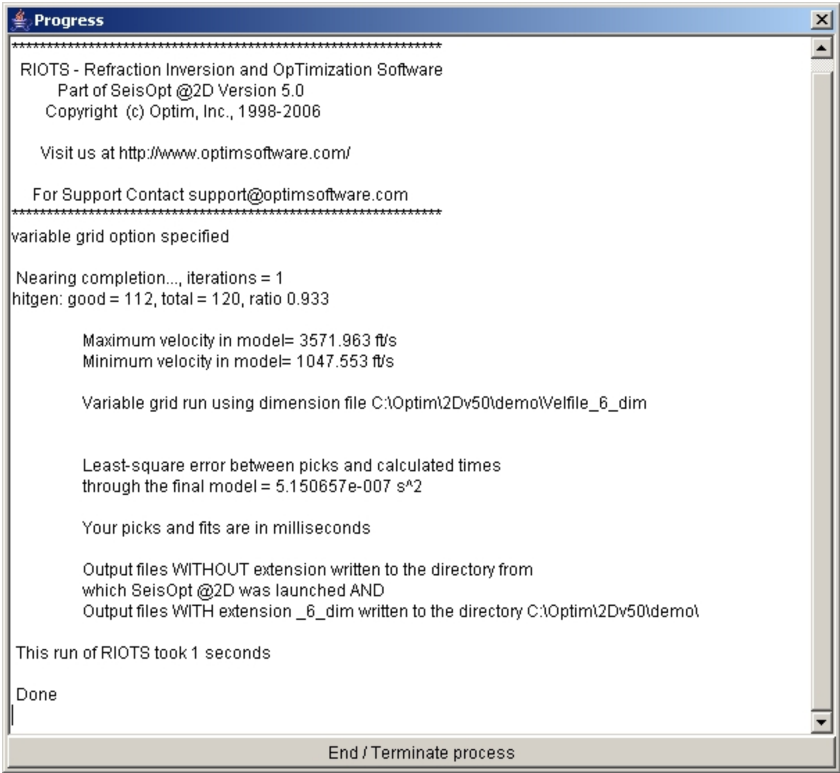


Figure 56: Progress window showing variable grid optimization run for one iteration.

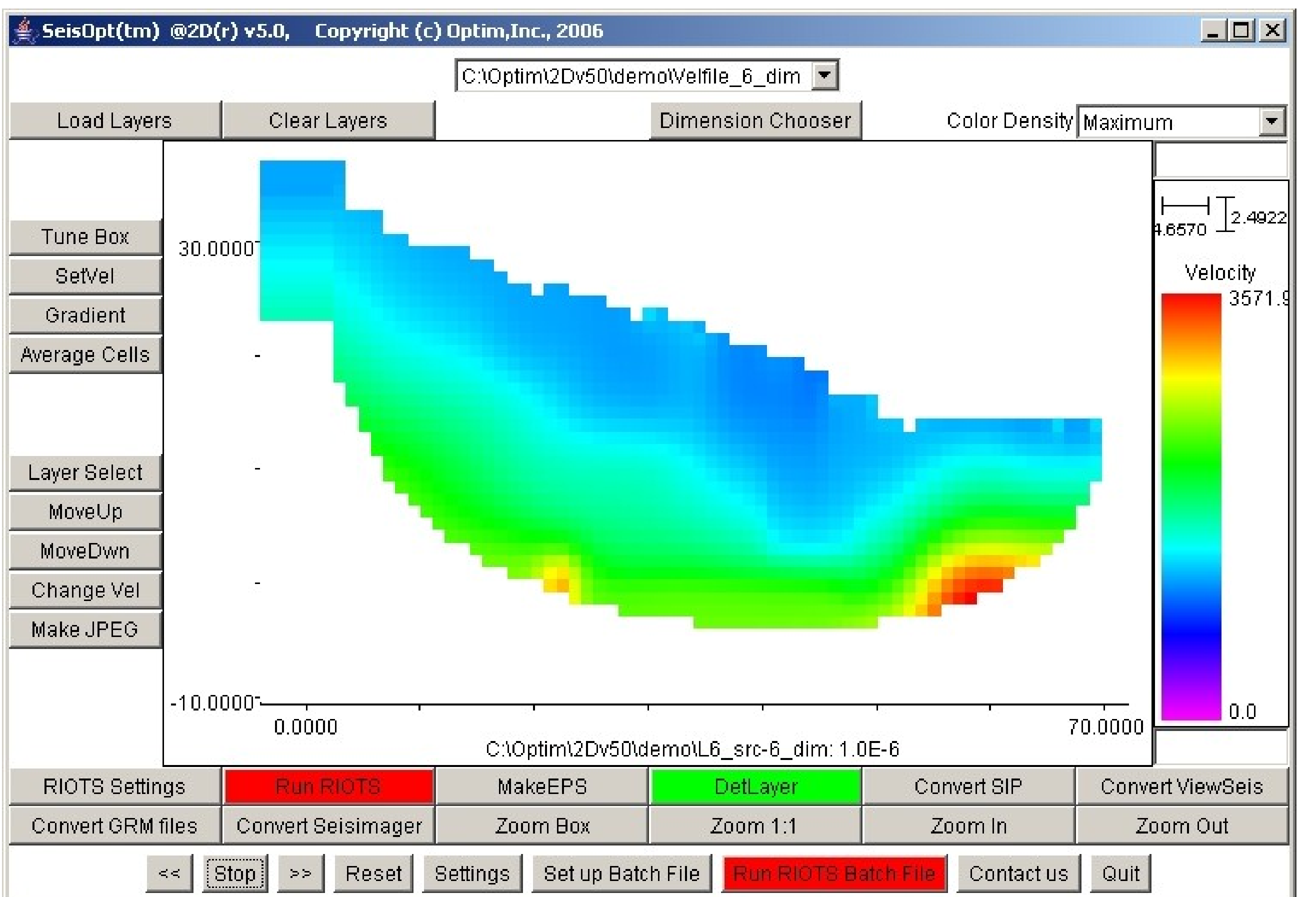


Figure 57: Output from variable grid run with the columns on the left merged into one.

9.0 Solutions to Possible Problems and Other Useful Information

It is possible to encounter some problems while running SeisOpt@2D, mostly because of memory issues or using incorrect file formats. Below we list some trouble shooting tips.

9.1 Authorizing the license

SeisOpt@2D will not run without a valid license, and separate licenses are required for each computer on which SeisOpt@2D runs. If a valid license is not present, a message saying 'program not authorized' appears when SeisOpt @2D is started. Configuring the license involves an exchange of 2 numbers between the licensee and Optim, a ‘Site Code’, and a ‘Site Key’. The details of this procedure are given in the Section 0.0. To avoid problems with incorrect site codes/keys, always try to use Windows' ‘copy’ and ‘paste’ functions when transferring these numbers. If the message 'not a valid site key' appears when validating the license, verify that the site key entered exactly matches that sent by **Optim**.

SeisOpt @2D's license software 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. In this case, contact **Optim** for a new license.

9.2 Transferring the license

SeisOpt @2D license can be transferred from one computer to another. The transfer should be done before uninstalling SeisOpt @2D on the old computer. Once the license has been transferred, SeisOpt @2D will not run on the old computer. Here are the steps to follow for transferring the SeisOpt license from one computer to another:

1. Install SeisOpt @2D on new computer.
2. Put a new floppy disk into the disk drive
3. Click on the SeisOpt @2D desktop icons. 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 @2D was installed.
7. Start SeisOpt @2D 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.

9.3 Running RIOTS

In order to run, RIOTS needs 4 files, the parameter file ‘riotsinput’, and the 3 data files, described in Section 2.2. The ‘riotsinput’ file must be present in the directory from which SeisOpt@2D is run, or RIOTS will not start. If any of the file names and/or paths of the 3 data files are not specified correctly in ‘riotsinput’, RIOTS will not start. A message in the RIOTS Progress window appears if RIOTS cannot find any of the 4 required input files. Use ‘RIOTS Settings’ to correct any incorrect path or files names.

9.4 Unable to display Velfile/Hitfile

Memory shortage will cause the Velfile and Hitfile generated by SeisOpt@2D not to be displayed on the SeisOpt@2D GUI. Contact support@optimsoftware.com should you run into such a problem.

9.5 Terminating RIOTS

Clicking on End/Terminate button at the bottom of the Progress window (Figure 4) may lead to the computer screen going blank and may require a re-boot of the computer. It is best to terminate the process by using the 'Task Manager'. This can be invoked by clicking ‘Ctrl-Alt-Del’. Choose the process ‘riots’ and click ‘End Task’ to terminate RIOTS.

9.6 Preventing loss of license

Optim uses software protection mechanism to protect and license its SeisOpt products. The software protection works by storing hidden files in the installation directory. 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.