Multiparameter Numerical Medium
for Seismic Modeling, Planning,
Imaging & Interpretation
Worldwide

Tesseral Geo Modeling

Tesseral 2D Tutorial

Stepwise instructions for getting started

Contents

USEFUL INFORMATION

- ✓ Use menu **Help/ User Documentation** for introductory information.

… can be found on the company's website ***www.tesseral-geo.com***:

- ✓ ***Primer*** provides introductory information about the modeling in seismic and its different applications.
- ✓ ***Troubleshooting & FAQ*** provides explanations, answers and extended presentations to Frequently Asked Questions which may occur at using the package.
- ✓ You can also use ***training movies*** http://www.tesseral-geo.com/support.en.php .
- ✓ It is advised to download supplied with the package ***library of prebuilt models***

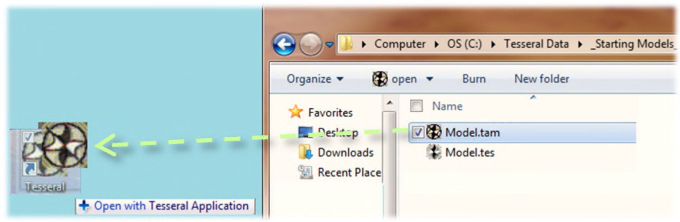
http://www.tesseral-geo.com/download.en.php /Tesseral Data Samples

…. look through *overview of prebuilt models*:

http://www.tesseral-geo.com/documentation/en/general/DataSamples.pdf

1 HOW TO START

- ✓ You can start the program using its icon (shortcut or pinned to taskbar) icon or by the sequence *Start/Programs/Tesseral/ Tesseral 2D*.
- ✓ The initial model data are associated with the ***Modelbuilder***, and resulting – with the ***Visualizer***. A mouse click on these files from *Explorer* calls and executes the *Tesseral 2D* (*2D* - here, an installed variant of the package) with relevant data. You can run the package, also using the method of drag-and-drop method.

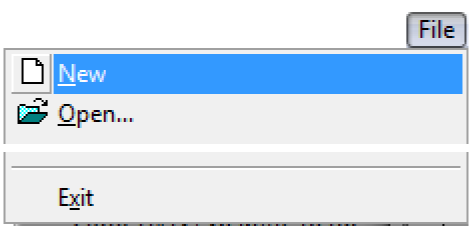


2 TO MAKE A NEW MODEL

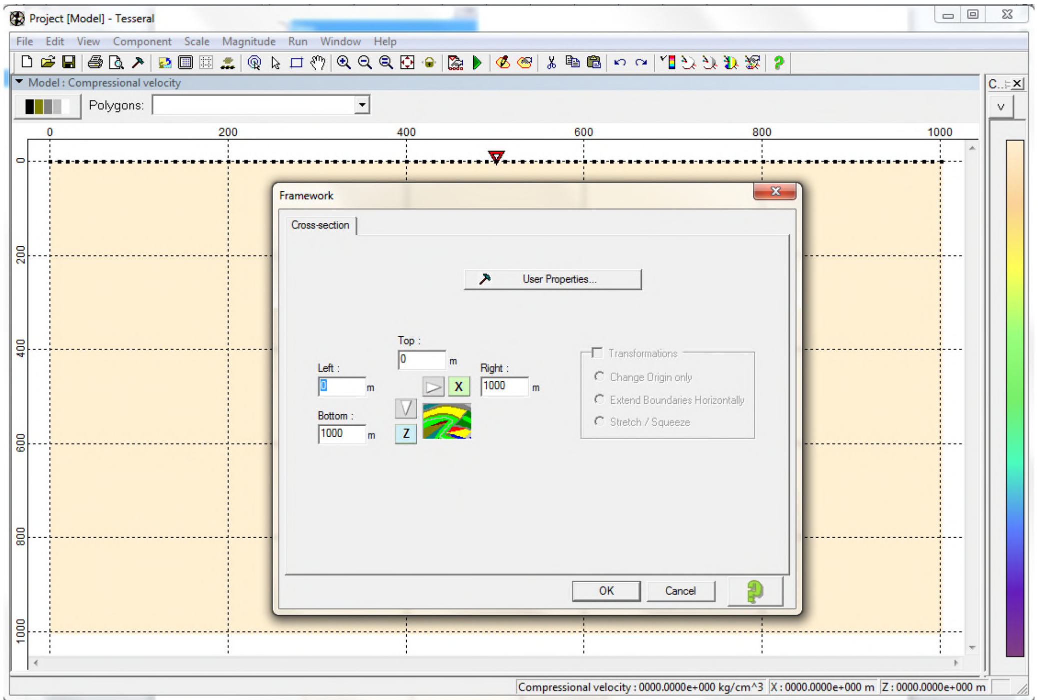
… we have to:

- Define the model boundaries;
- Build polygon layers inside the model boundaries;
- Define the physical parameters of the layers;
- Engage the proper modeling algorithm for our purposes;

➢ When program ***first starts*** or you create a new model using ***File/New*** menu:



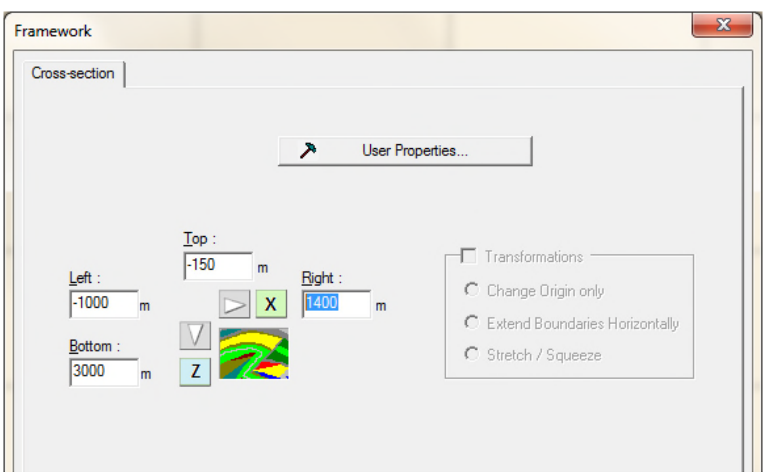
A default model appears. The ***Framework/Cross-section*** dialog and will be used to define the ***model rectangle*** (cross-section boundaries).



We want to:

- build a new model,
- define the (modeled) survey parameters and
- run seismic (numerical) modeling.

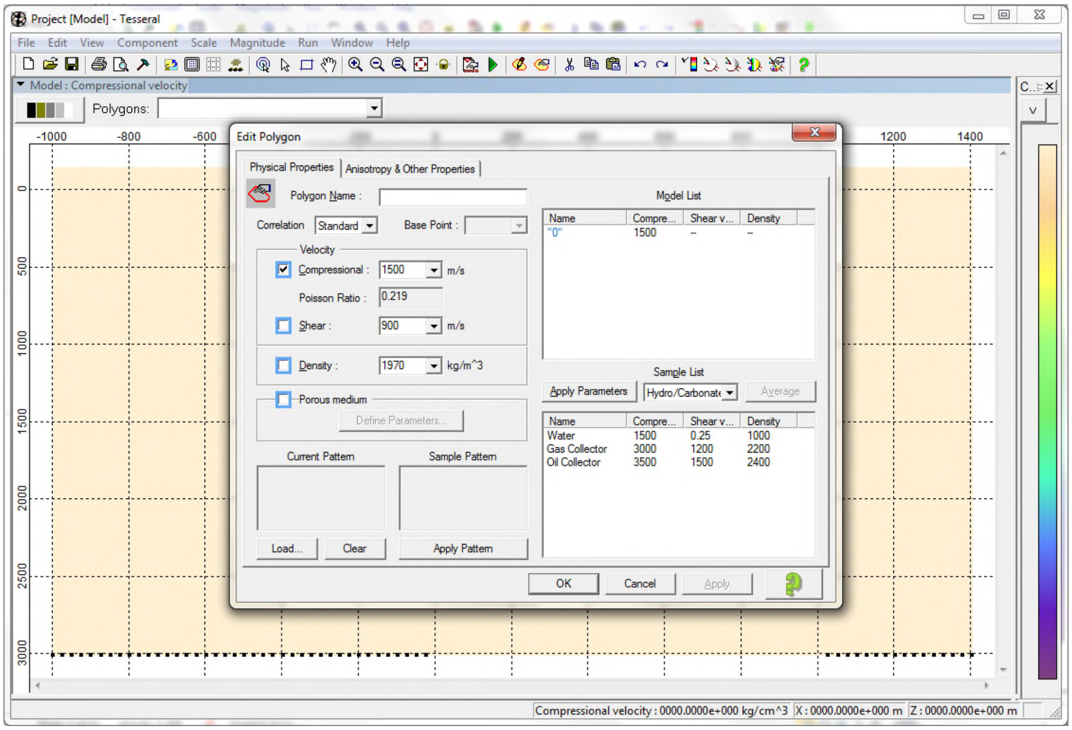
Let's define the model boundaries to be –150 on the top, 3000 m down to the bottom, -1000 m to the left of our 0, and 1400 m to the East.



➢ After we click OK, *zero-polygon* (number 0), which covers the whole model rectangle, is automatically built and then is produced dialog for setting polygon parameters …

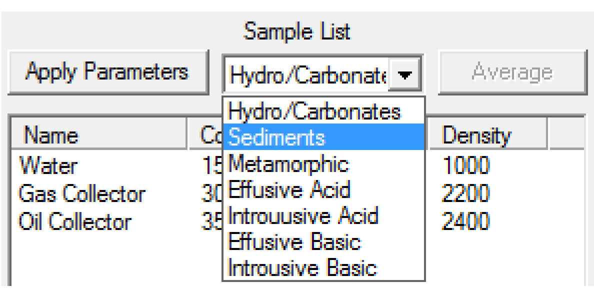
3 MAKING THE FIRST MODEL POLYGON

… When we fill this in, and then click on *OK*, we can define our 1st polygon.

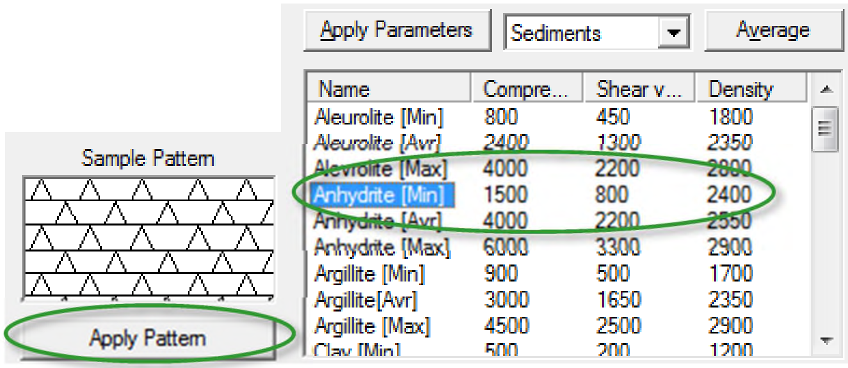


In the menu, we can start by defining the *P-wave velocity*. The *S-wave (shear) velocity* and density will assume default values (the defaults will be used when the *blue boxes* are unchecked).

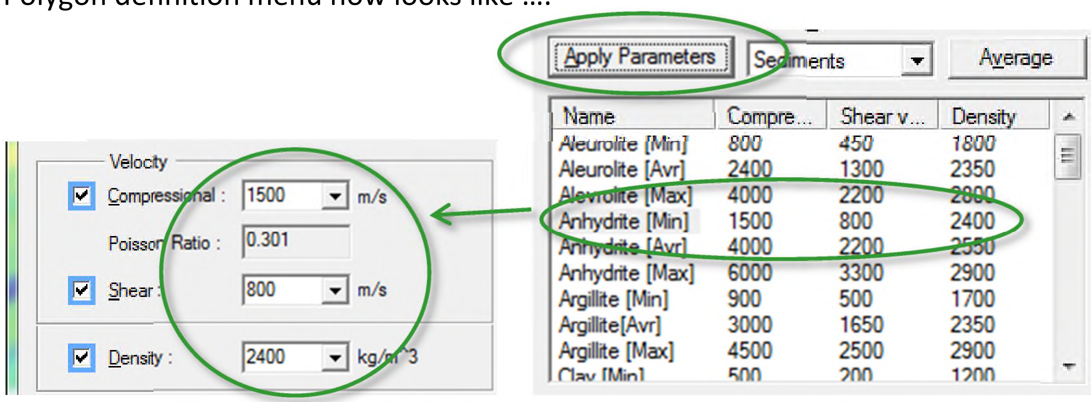
One of the options that we have is to define the layer physical properties using the *table of sample properties*. The table has been set up for various rock types with their minimum, average and maximum *P-* and *S-wave velocities* and *Densities*.



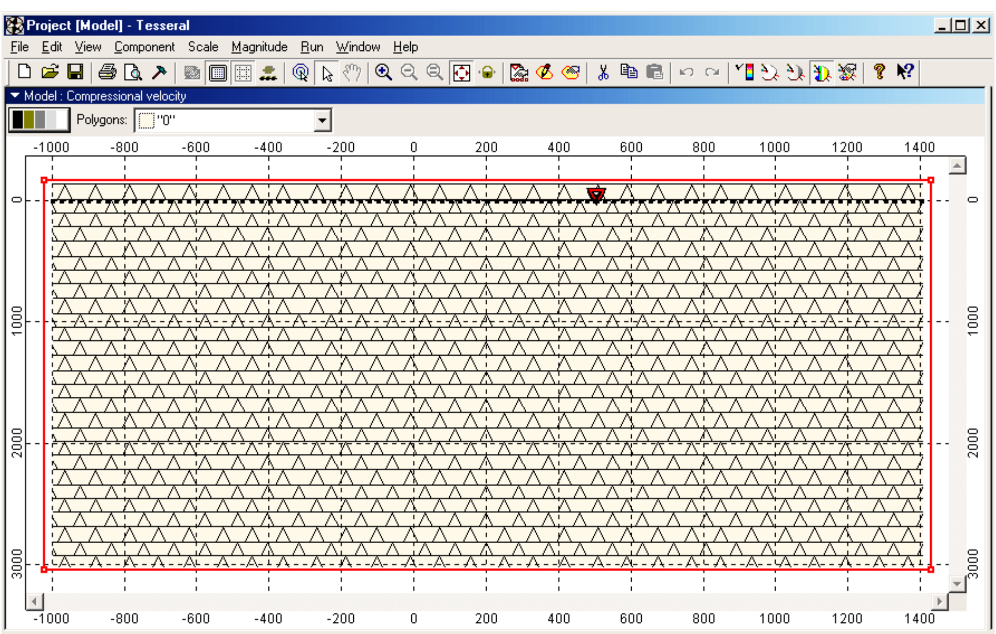
➢ We pull down the sample list and select, in this case, *Sediments* …



- ➢ *Let's select Anhydrite [min].*
- ➢ We can apply a pattern to the layer by clicking on *Apply Pattern*. The pattern for the sediments appears in the pattern box.
- ➢ Next, let's transfer our Sample list properties to the *Velocities* and *Density* entries. The altered Polygon definition menu now looks like ….

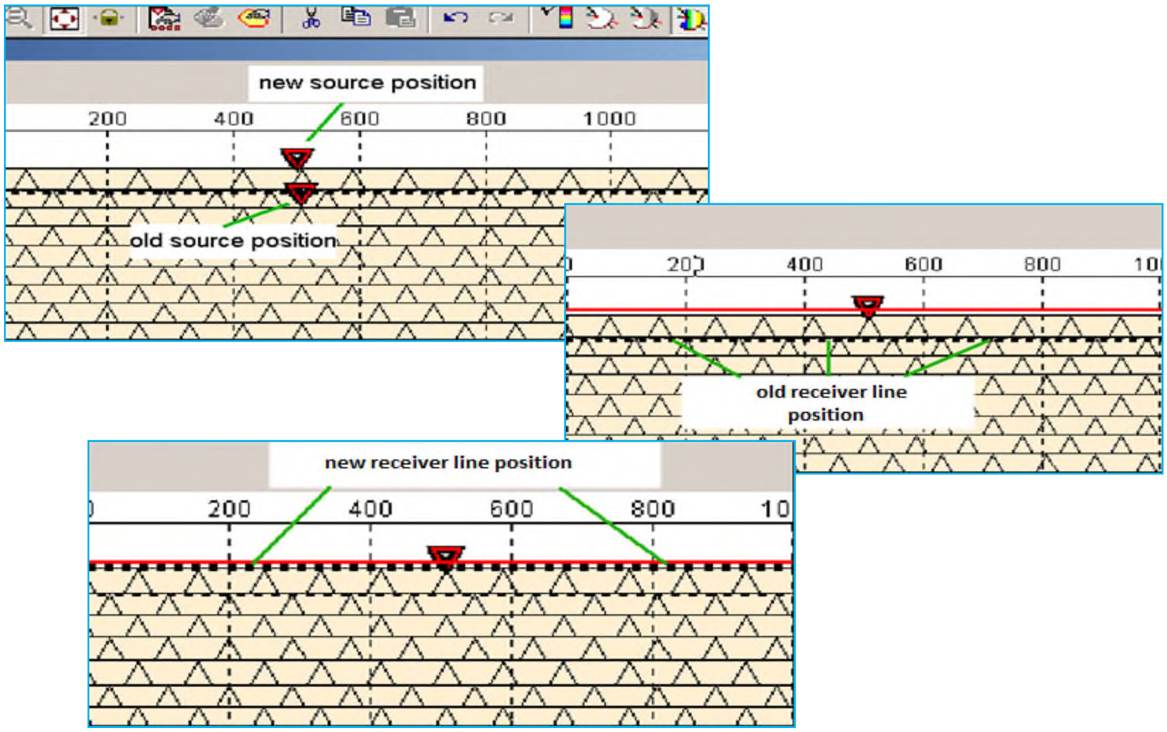


- ➢ When we click *OK*, the *zero-polygon* layer appears:



We want to show how to move the source and receiver line to a new elevation.

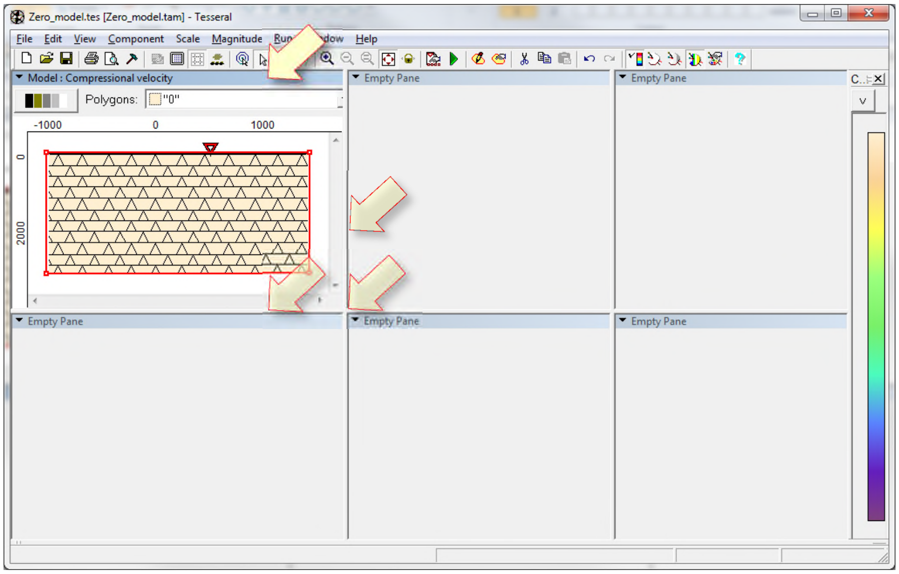
- To move the source, we place the cursor over the source (shown as a red triangle … delta) and pull the source to our new location.
- To perform the same task for the receivers, we place the cursor on top of a receiver, click *mb1* and pull the entire line up receivers up to the source location …



4 MANIPULATIONS WITH WINDOW PANES

Let’s agree on the notation used: *mb1* - left mouse button, *mb2* - right mouse button (*mb* - mouse button).

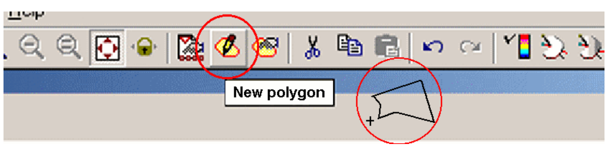
- *Program window* can be divided into several *panes*. Each of them may have (show) its data.
- If you double-click *mb1* on the title bar (blue area), there will be shown six panes.

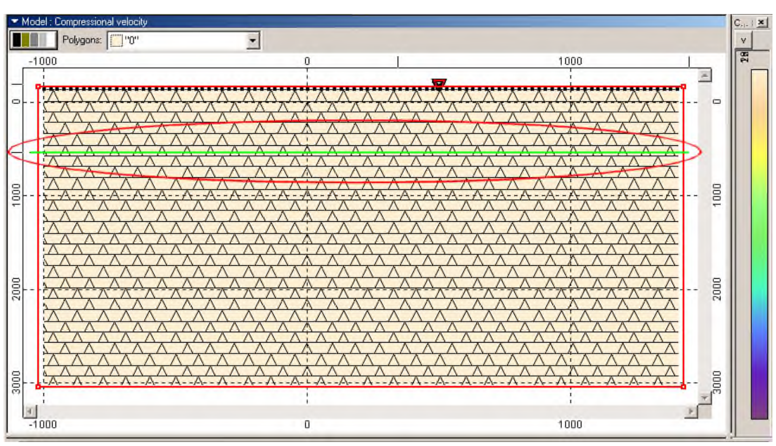


- You can also manually change the (relative) size of the pane by dragging its edges.
- With *mb1* double click on the first screen title we return to one pane window.

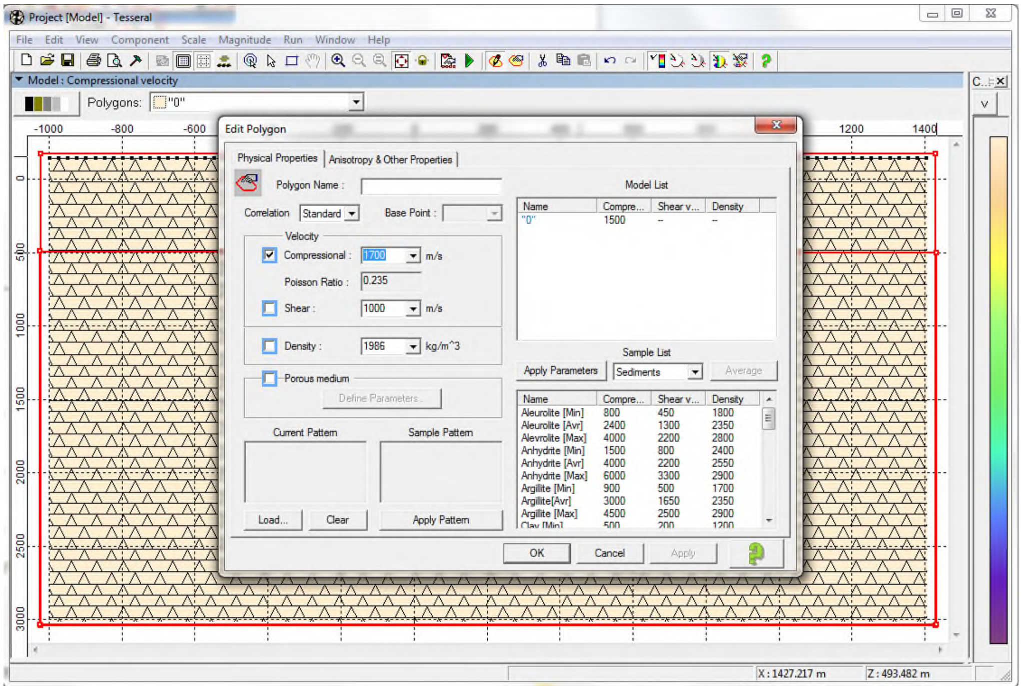
5 MAKING THE SECOND POLYGON (SIMPLE FLAT LAYER)

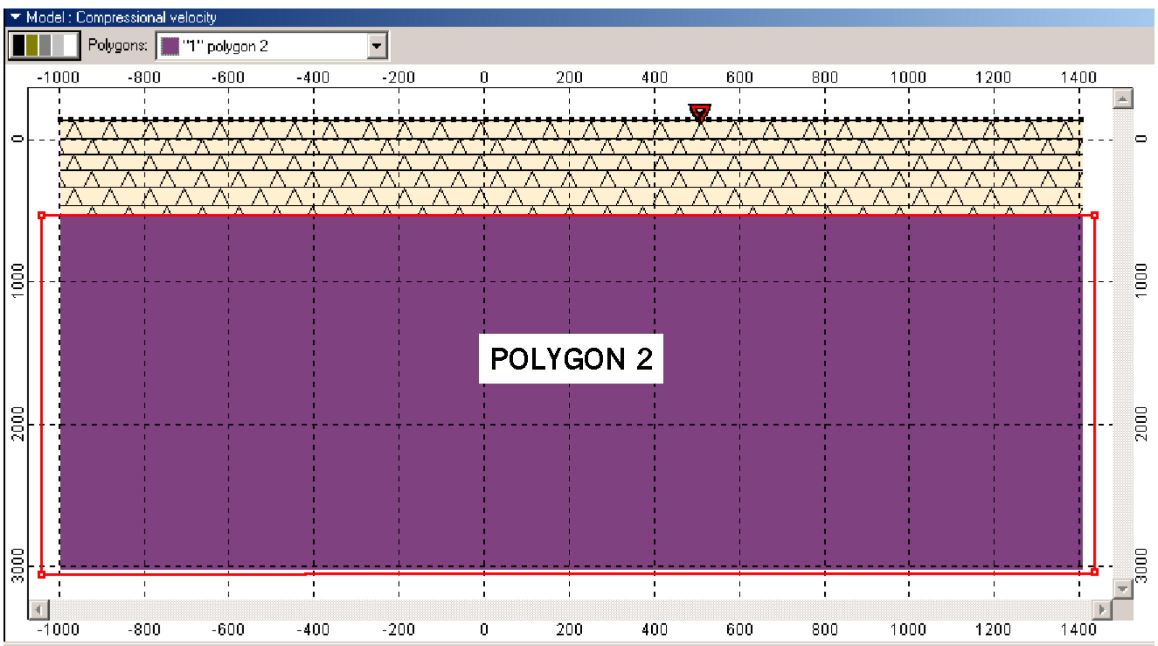
- Place the cursor in the New Polygon icon (shown below) and click mb1. The cursor turns into a polygon shape with a plus sign below it. Locate the cursor outside of the left boundary at about the 500 m mark (see the Z value at the bottom of the Tesseral window).
- Click once and extend the line horizontally over to a position just outside of the right boundary. Double clicking *mb1* will close the polygon down to the bottom of the model. This is shown below …





- After the double-click, the Polygon property menu appears. Fill in 1700 m/s for the P-wave velocity …

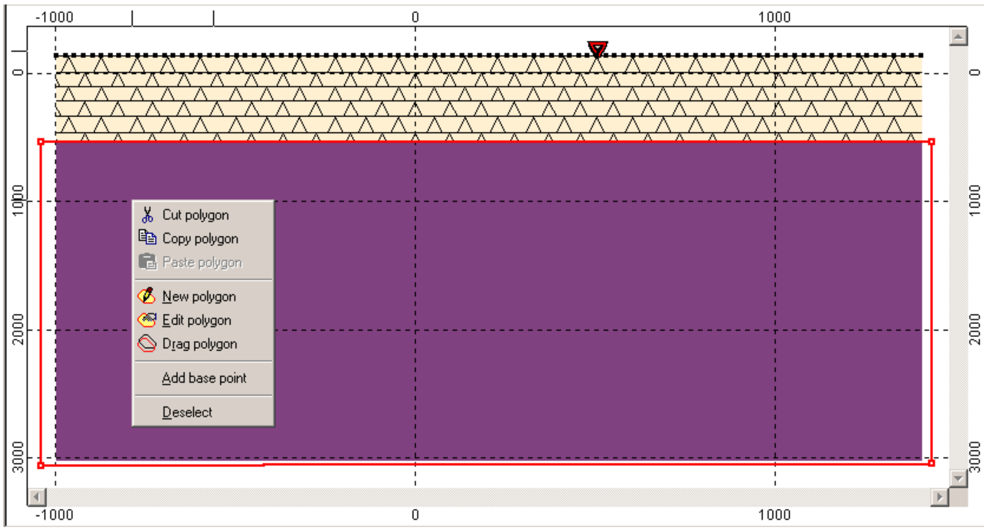




Notice the boundaries of the second polygon.

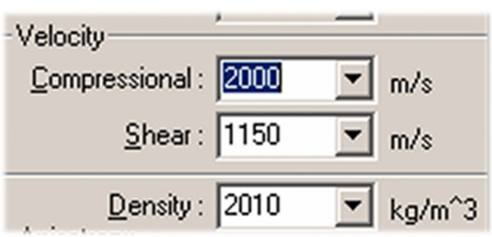
➢ After filling in the Velocities (defaulting the S-wave velocity and density), press *OK* to complete the model entry.

6 DESIGNING THE THIRD AND FOURTH POLYGON

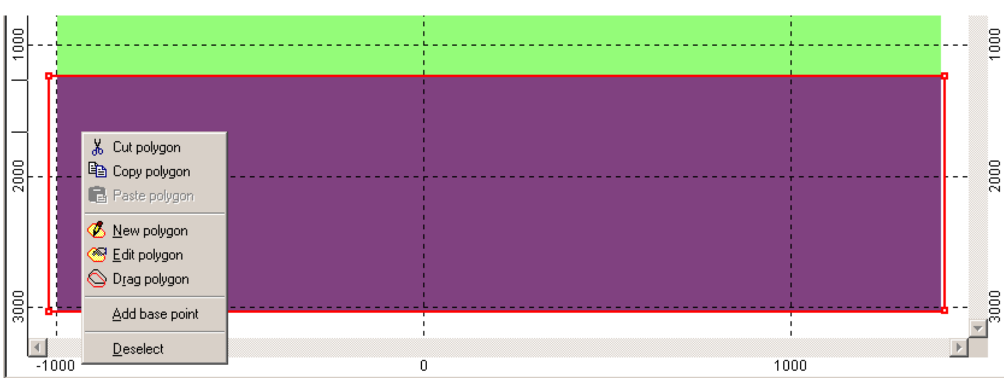


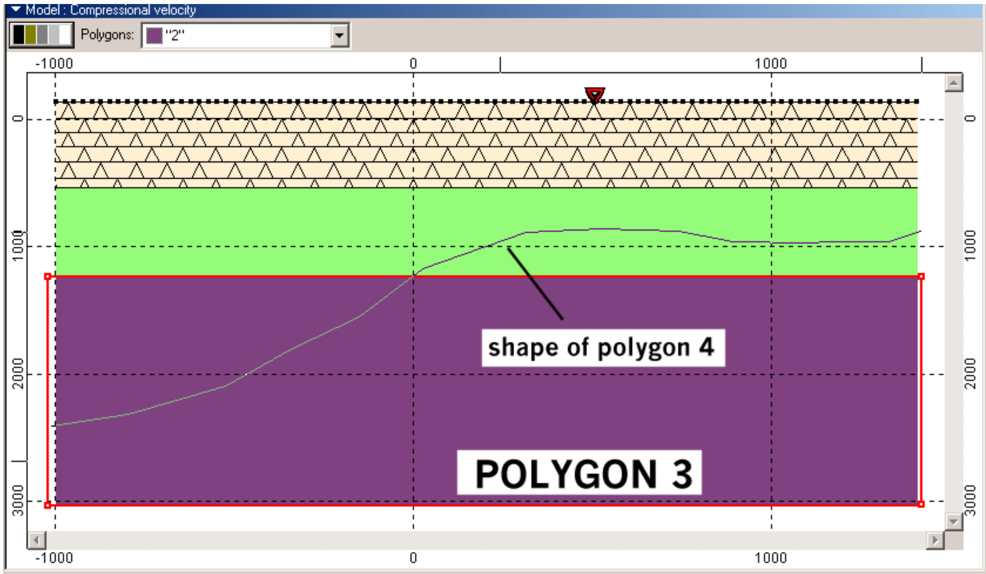
Polygon 3 will start from the left side of the model and truncate against Polygon 4. How do we do this? We first input *Polygon 3*. Then we construct *Polygon 4* so that it cuts through *Polygon 3*.

- Let's put *in Polygon 3*. Place the cursor inside *Polygon 2* (where the new polygon will be created) and click *mb3* (*left mouse button*). Choose the *New polygon* menu item …
- Place the cursor at 1222 m on the outside of the left boundary, click *mb1*, bring the cursor directly across the model (also at 1222 m depth) to the outside of the right boundary, double-click *mb1* to complete the polygon and bring up the *Properties* menu … fill in the *P-wave* velocity of 2000 m/s.

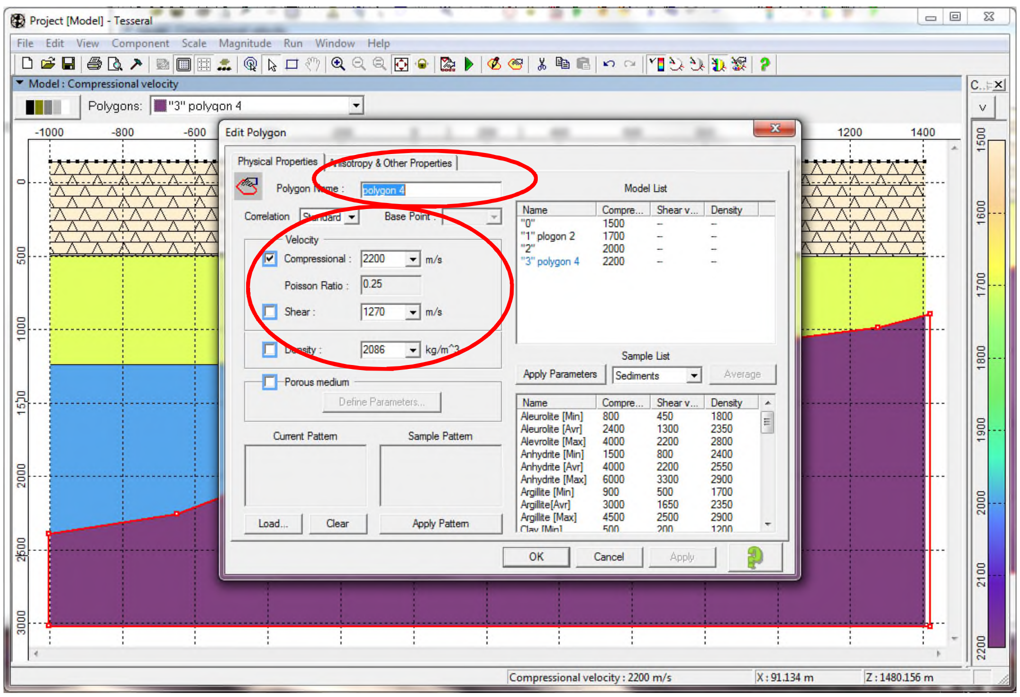


- Click Ok to see the new layer 3. Inside of the 3rd layer, click mb3 and select New Polygon … Let's start drawing the new polygon …





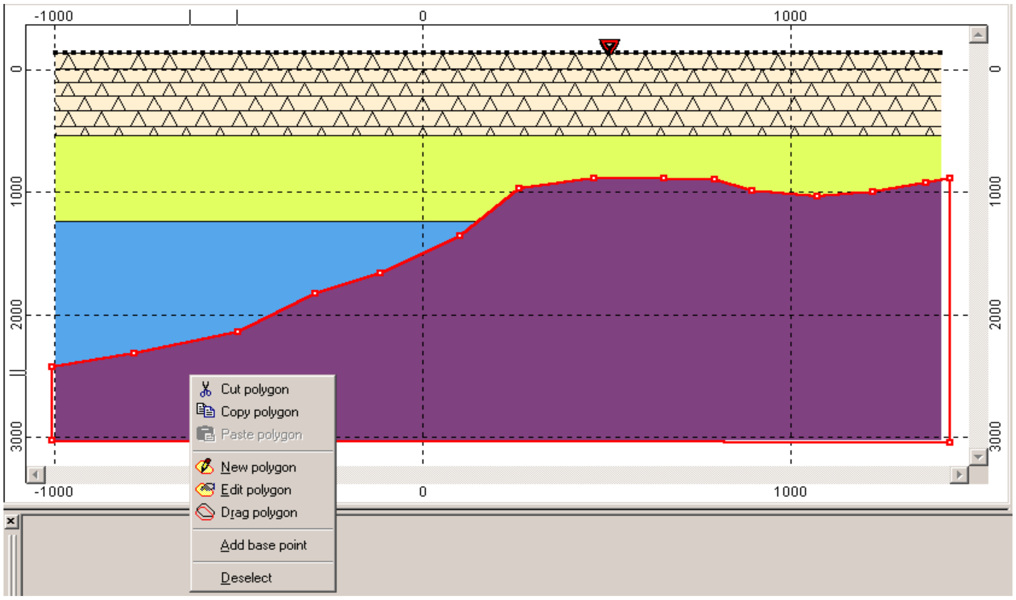
- The polygon is drawn over to the right boundary (just outside of it). Double click *mb1* to close the polygon. Define the velocities and densities. Name it the *Polygon 4* as shown below …



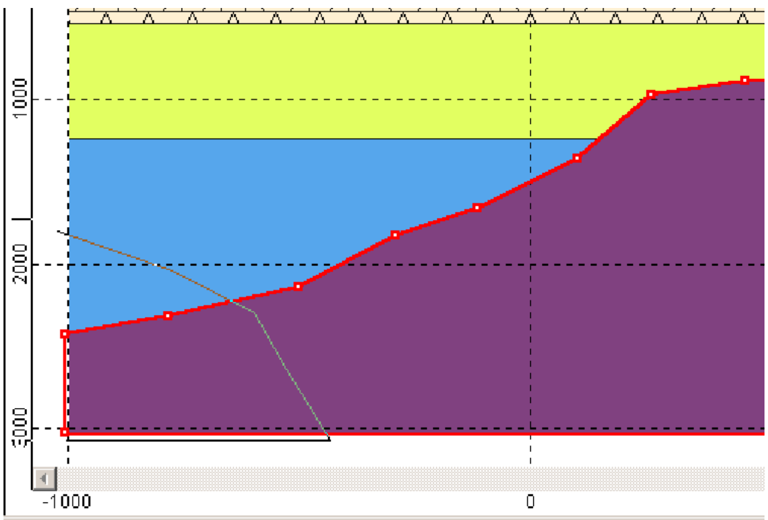
- Click Ok to see model. The third layer is truncated against layer 4. This is similar to the shales abutting against the carbonate reefs.

7 FINAL LAYER CUTTING ACROSS TWO PREVIOUSLY DEFINED LAYERS

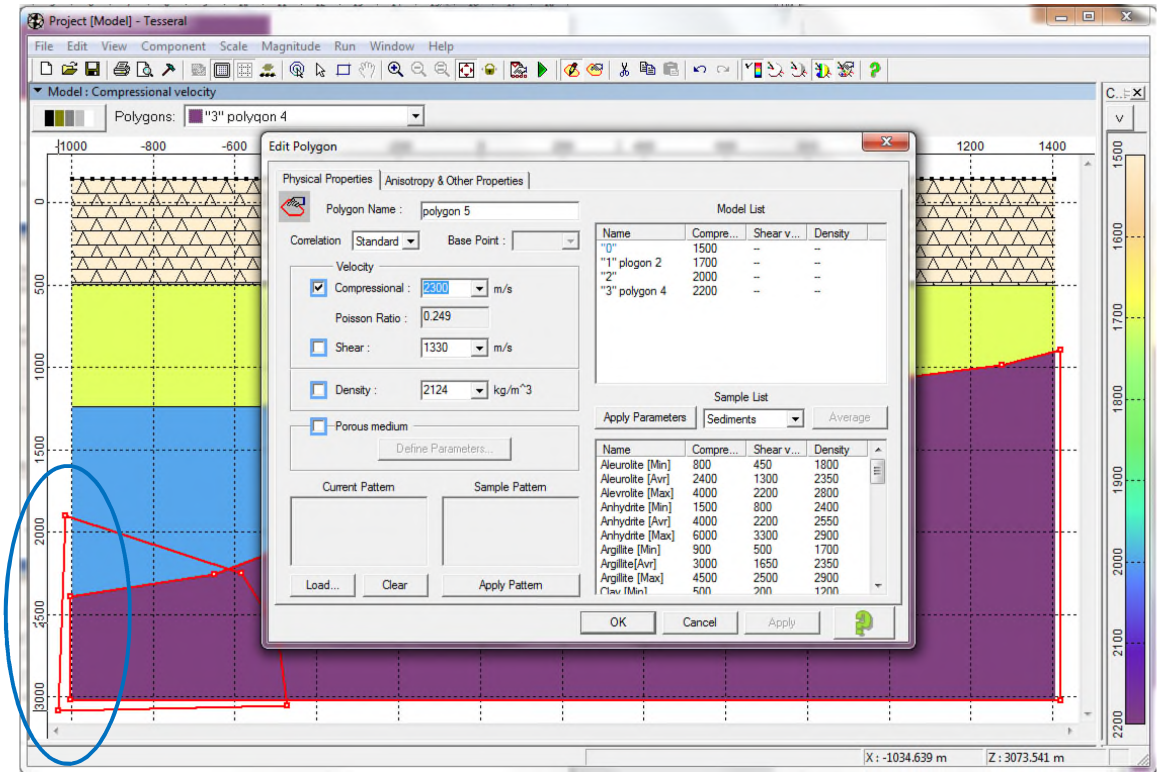
- In this layer input, the layer penetrates both polygons 3 and 4. Initiate the *New Polygon cursor* by clicking *mb3* inside layer 4 … and clicking the *New Polygon* icon …



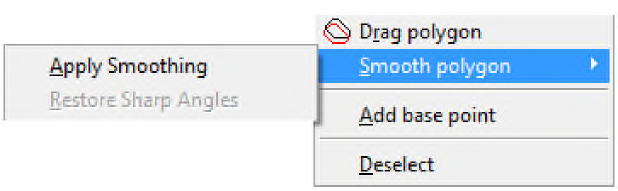
- Draw the polygon starting just outside the left boundary at ~ 1800 m, draw the line through layers 3 and 4 as shown, draw the line to just below the model boundary and then extend the line back to the outside of the left model boundary (see below) …

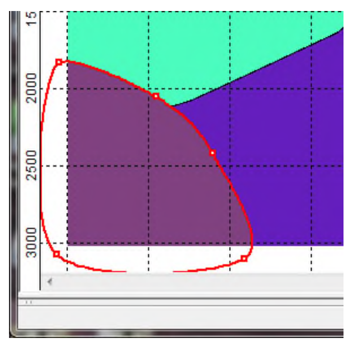


- When you place the cursor to the left of the left boundary and double click *mb1*, the polygon closes (see blue ellipse) and the Polygon menu box appears.



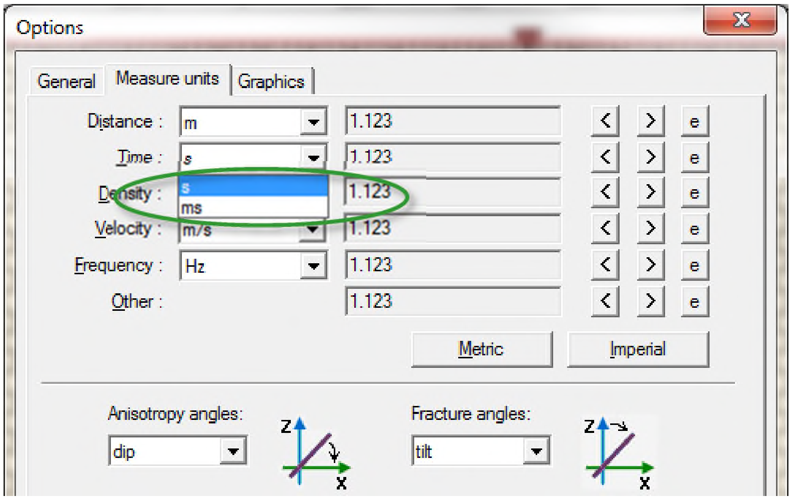
- ➢ For polygon 5, fill in the P-velocity values (2300 m/s) as shown above (and then click *OK*). Polygon 5 is done.
- ➢ You can smooth polygon boundary using *Smooth polygon* option of *mb2* menu.





8 DOUBLE CHECKING THE UNITS OF MEASUREMENT

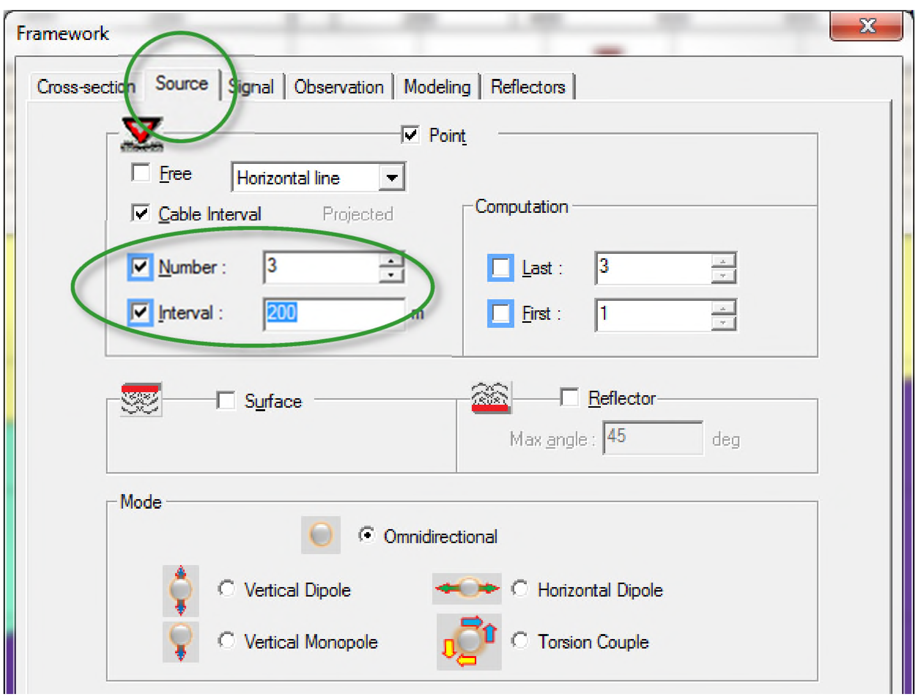
- ➢ Push the toolbar button and bring up the ***Properties*** dialog
- ➢ Click on the ***Units*** tab to reveal the measurement system used in the modeling…



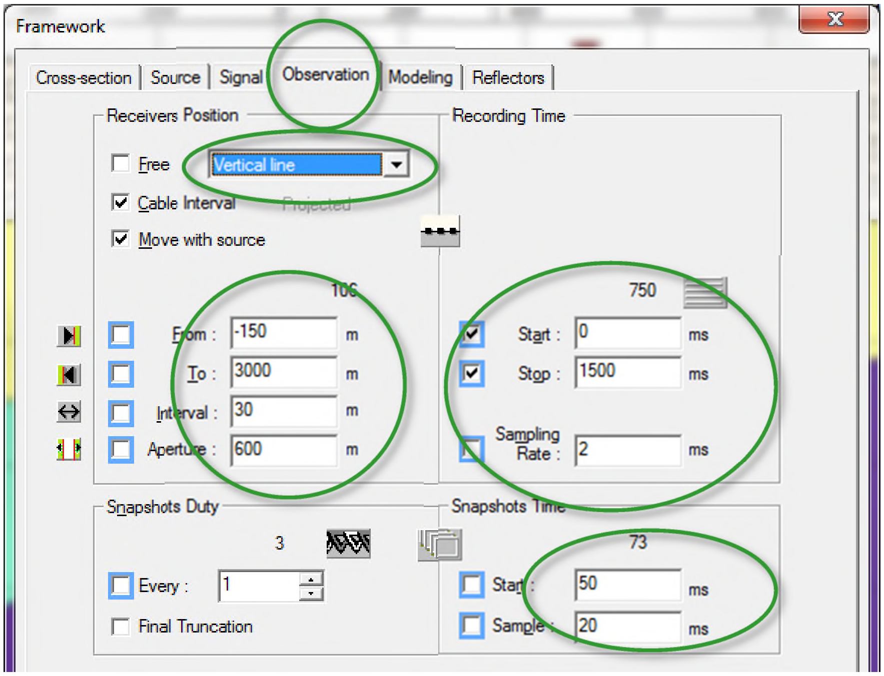
- Let's look at how to change the units. Click on the arrow located on the right hand side of the *Time* box. It reveals that we can choose to enter time units in *s (seconds)* or *ms (milliseconds)*.

9 SETTING THE SOURCES (ON THE SURFACE) AND RECEIVERS (IN THE BOREHOLE)

- Push the toolbar button . The ***Framework*** dialog appears …
- Click on the ***Source*** tab to define the source configuration as it is shown below …

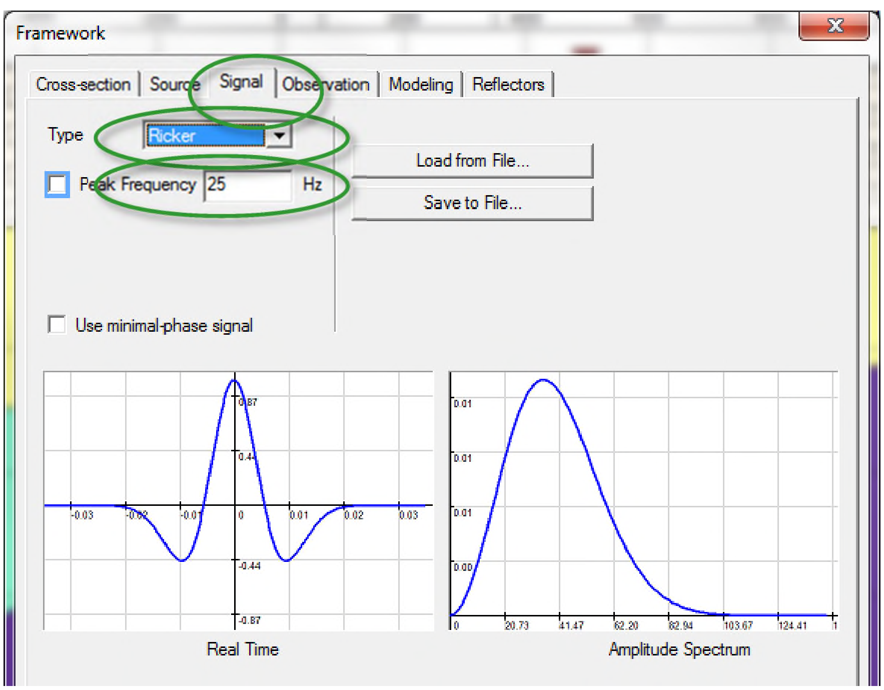


- To set the receiver locations, click the ***Observation*** *tab* and select *Vertical line*:



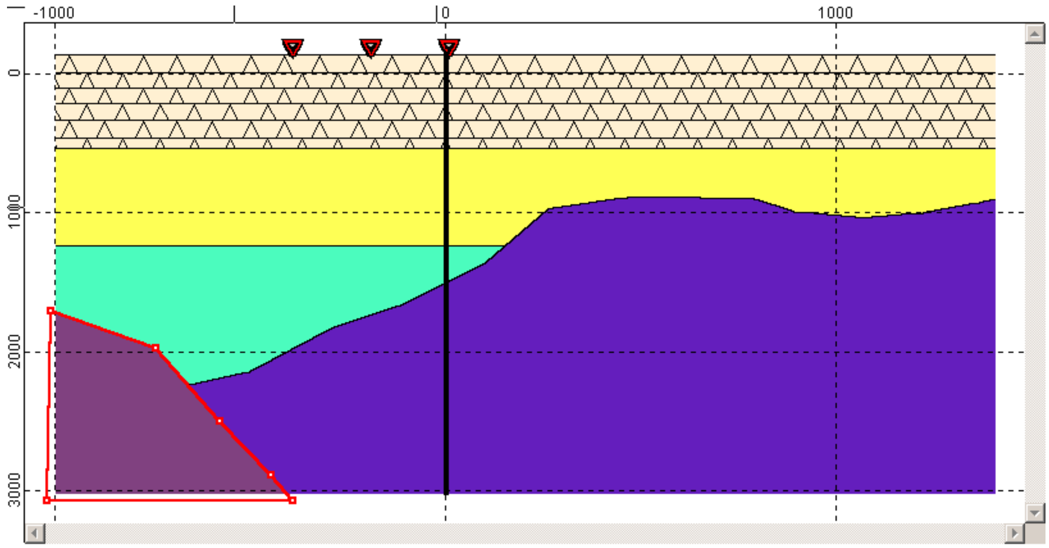
- Set the downhole receivers to start at -150 m to 3000 m depth at a sonde depth interval of 30 m. Start the recording at 0 ms and stop at 1500 ms. The sample rate for the recording is 2 ms. Note that the number of depth levels, time samples, sources and pictures are shown on top of the menu boxes…

To set the signal parameters, click the ***Signal*** Tab and define it as shown below ….

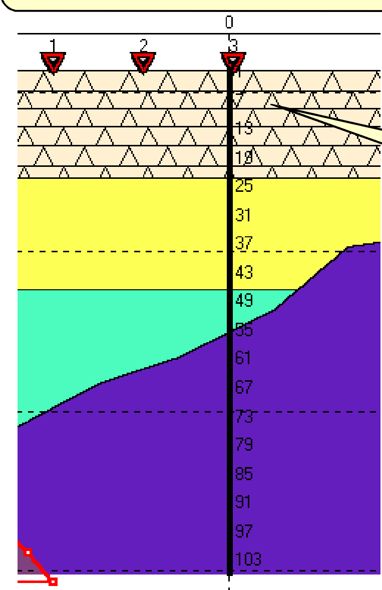


- After setting of all *Framework* parameters is finished click *OK*.

The setup for the sources and receivers now looks like …

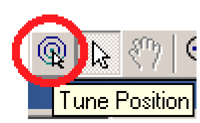


➢ Let us see the shots and receivers using the ***Show receiver Numbers***



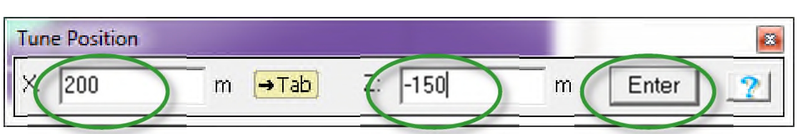
The shots and receivers now appear with numerical labels …

➢ We now want to fine tune the positions of the sources using the fine tune icon …

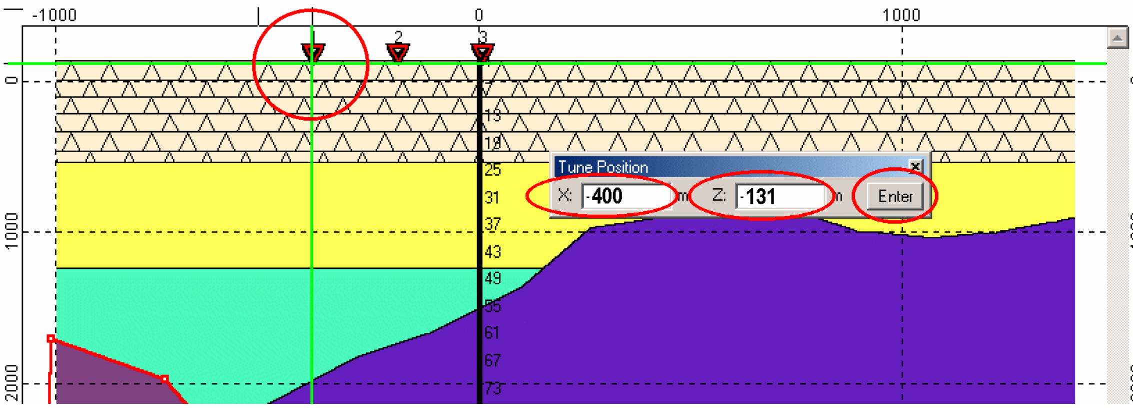


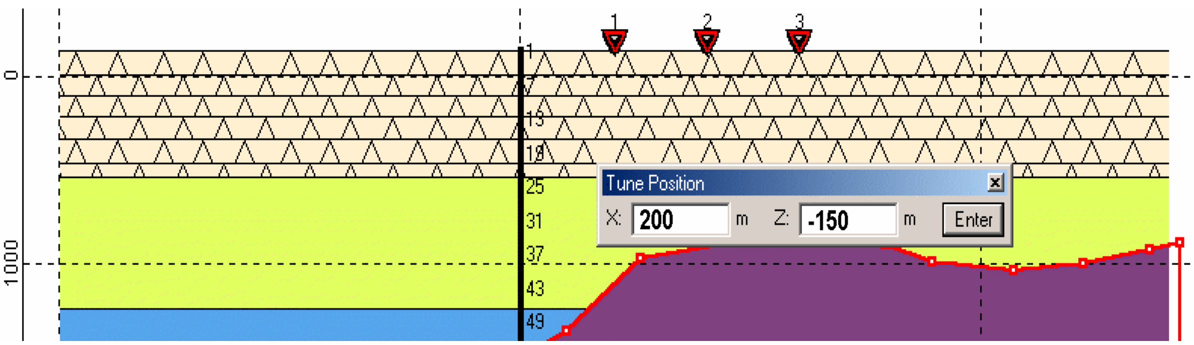
The procedure is to locate the cursor at the location of the object that you want to move, a faint cursor image lies on top of the object, type in the new coordinates for the object and then click enter. In this example, we want to move the entire row of sources.

➢ Moving the cursor to the first source locates the first source at x= -400 and z= -150 m, as shown below. With the source highlighted by the cursor, type in 200 for X and –150 for Z (to shift into Z window use ***Tab*** keyboard button) in the Tune menu …



- Press ***Enter*** in the Tune Position menu to move the sources to the right.

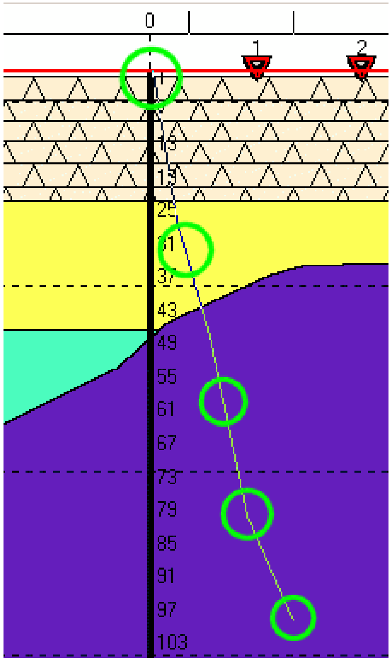




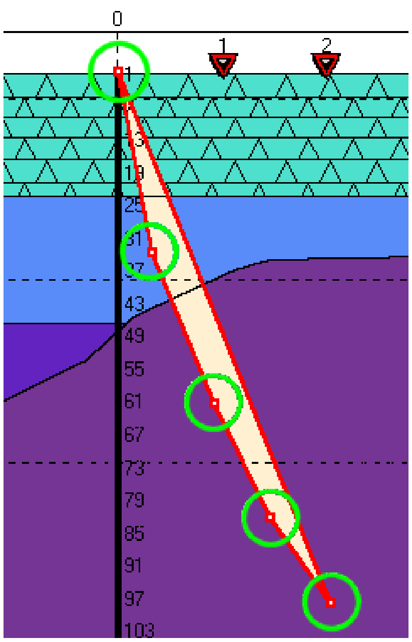
- The leftmost source now has coordinates of X=200 m and Z=-150 m. Close the fine tune box by clicking on the **x** in the upper right hand corner.

10 INPUTTING A DEVIATED BOREHOLE

- Let's start by creating a new polygon using the  *New polygon* button.
- Draw in the deviated well (use the X and Z values shown at the bottom bar). The green circles show the chosen *break in slope* points in the deviated well. Our task is to create an artificial polygon and then project the polygon onto the deviated well.



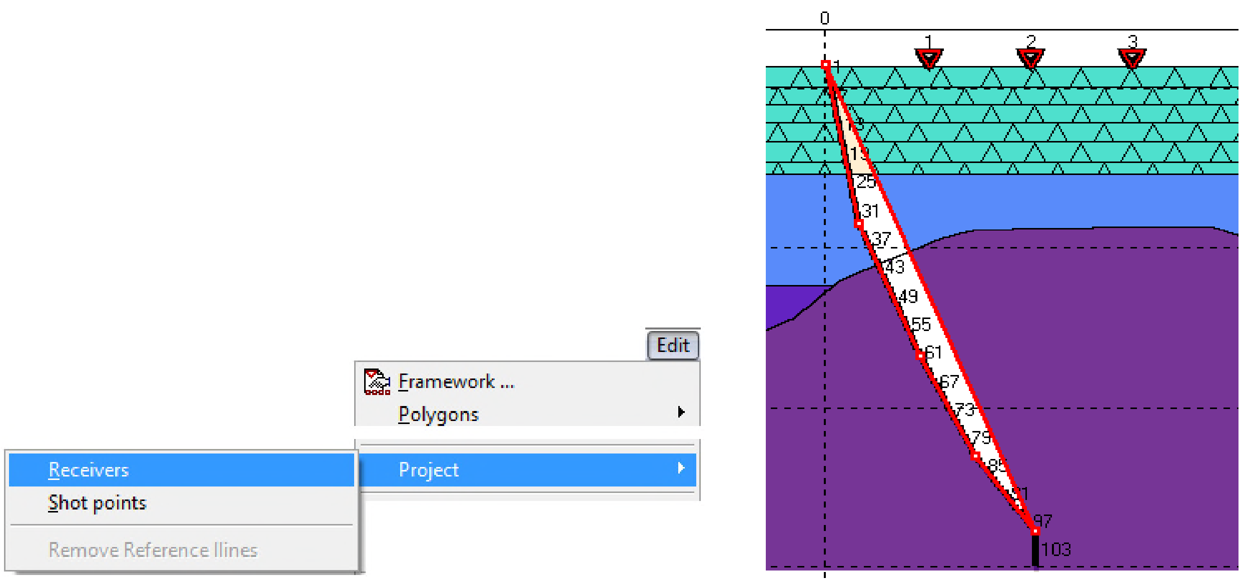
➢ Double click mb1 at the bottom of the deviated well to bring up the Polygon edit dialog. Click the OK box because this is a dummy polygon and the parameters do not need to be specified.



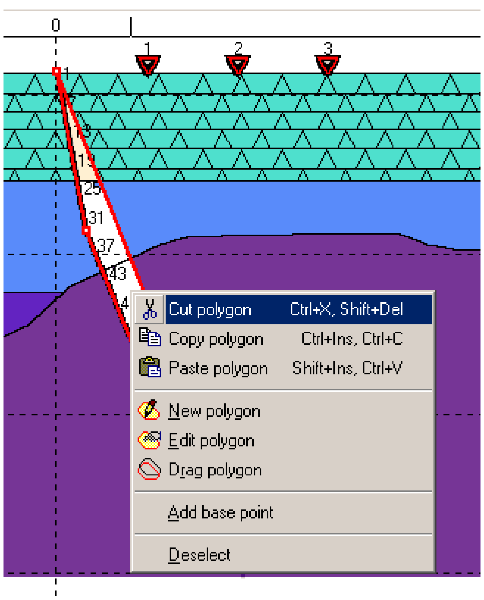
The polygon is completed, as shown below.

Now we want to collapse the polygon onto the deviated well trace.

➢ To do this, we use a menu ***Edit/Project/Receivers*** to project the receivers onto the trace of the borehole as it is shown below …

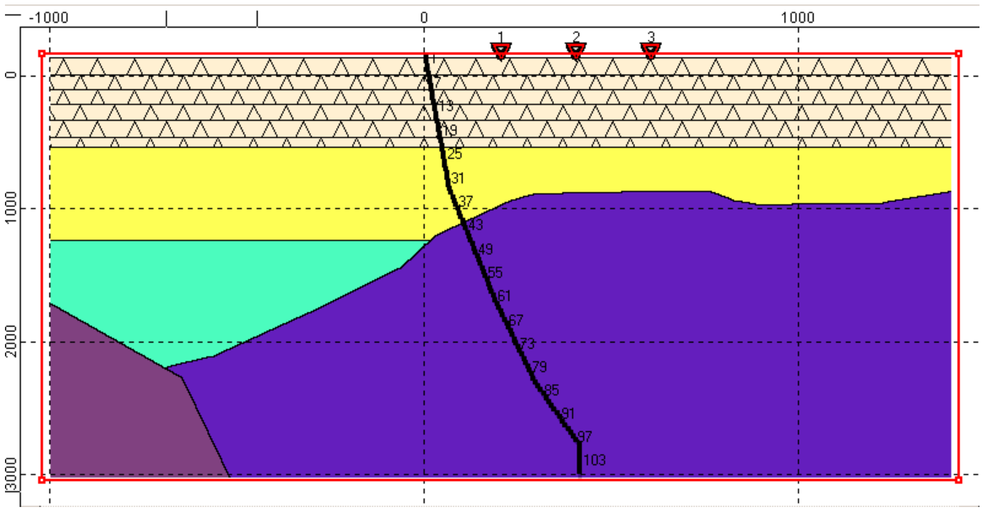


➢ We can now collapse the polygon into a single line. Place the cursor inside the polygon that you just made using the deviated borehole. Click *mb2* to bring the menu shown below …



➢ Select *Cut Polygon*

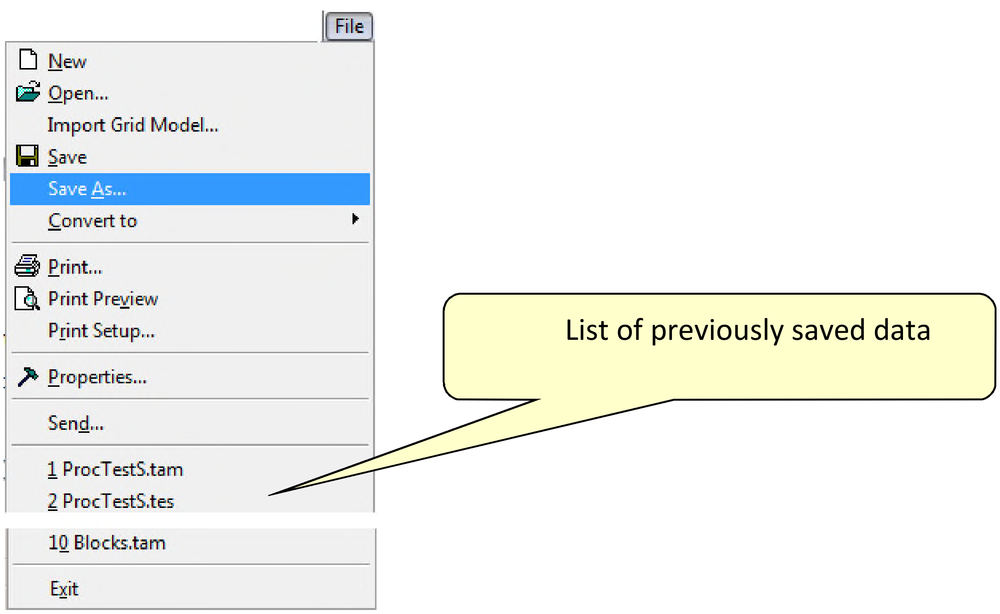
Next what should appear is the trace of the deviated borehole with receiver levels superimposed onto the trace …



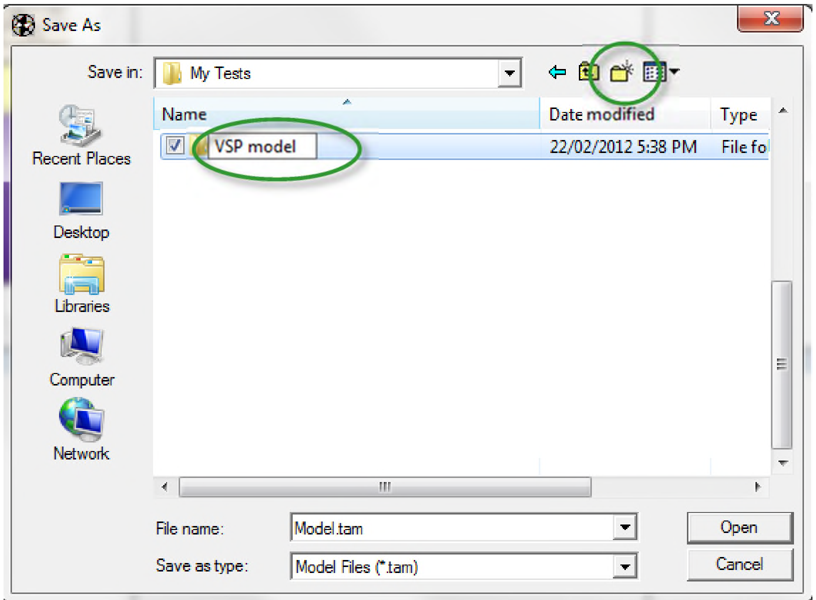
We are now ready to run the Finite Difference modeling software.

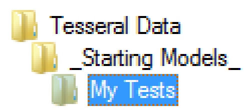
11 SAVING CREATED MODEL

A good practice is to save the model that you just created. To do this, select on ***File/ Save As…*** menu item …

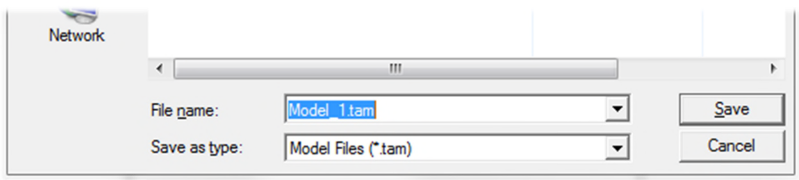


The ***Save As…*** dialog appears …





- Create new folder as it is shown. Rename the folder by placing the cursor inside the name box, click *mb1*, edit in a new name and click *mb1* outside of the name box …

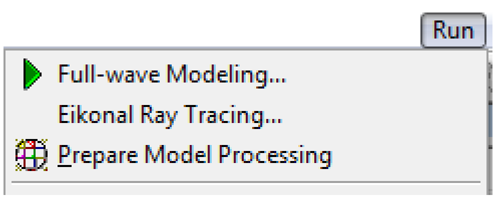


- Edit the File Name to be Model_1 and click Save button. This puts the model file into the selected folder.

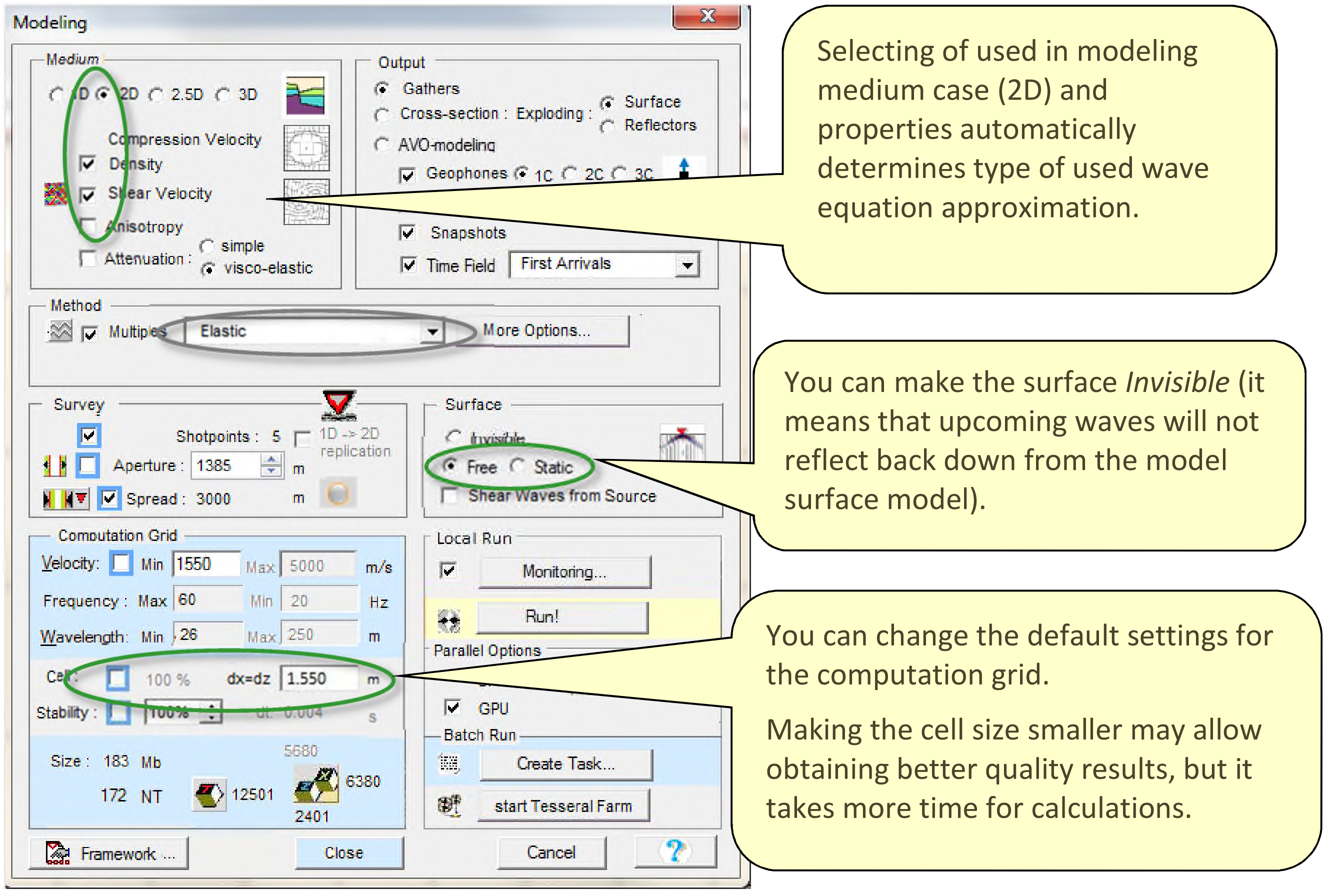
Now that the model has been saved, we can initiate the modeling software.

12 RUNNING THE MODELING PROCEDURE

Select ***Run/Full-wave Modeling*** menu item …

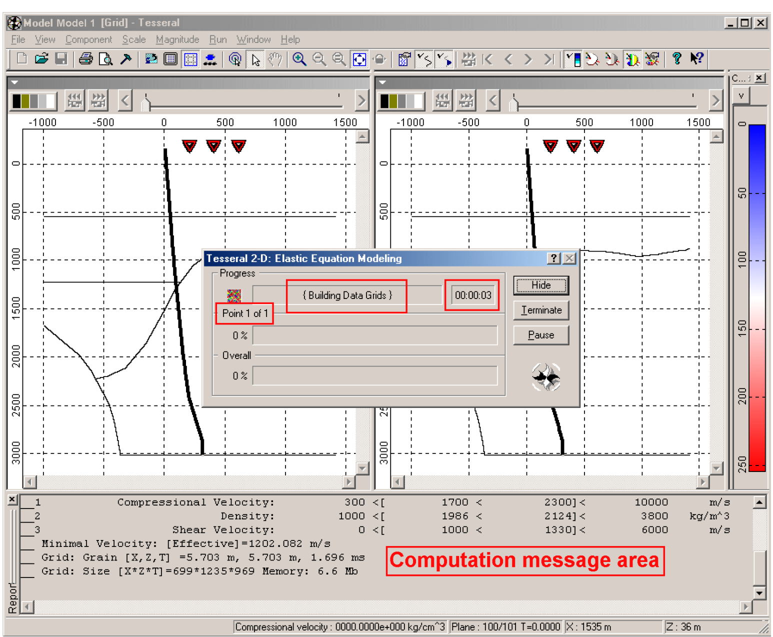


The ***Modeling*** dialog appears …

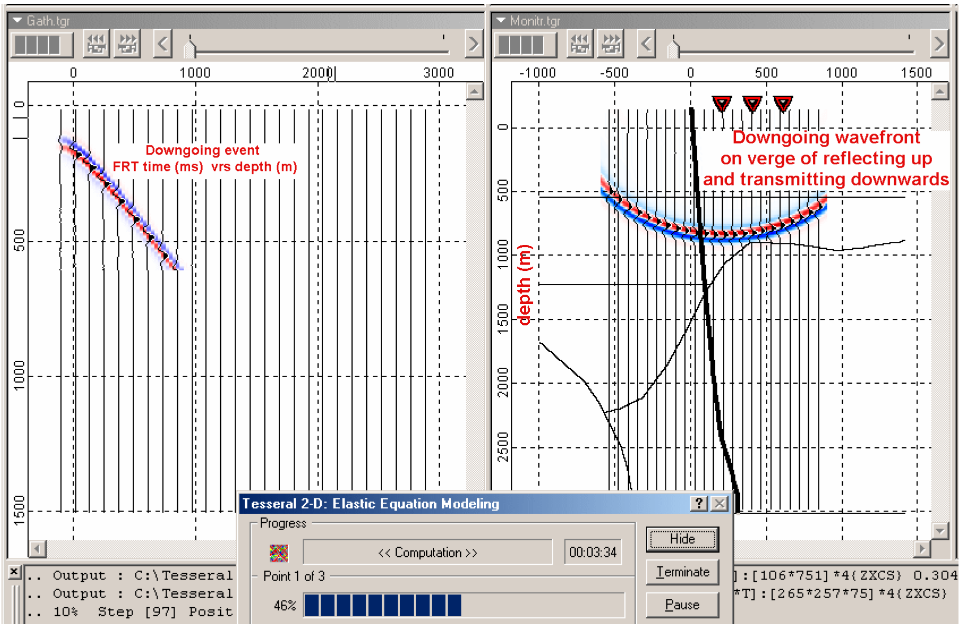


➢ Let's use shown above options and parameters. Push ***Run!*** button …

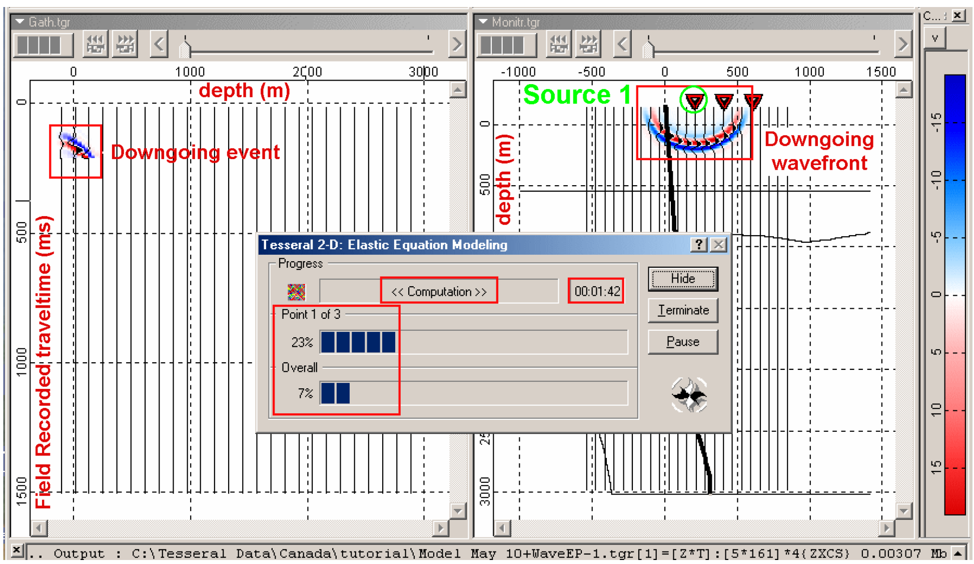
The first step in the *Full-wave modeling (FWM)* routine is to build the model grid. The grid spacing is determined by the dominant (peak) frequency of the source wavelet. The grid spacing is chosen (by default) to avoid *grid aliasing* computation artifacts.

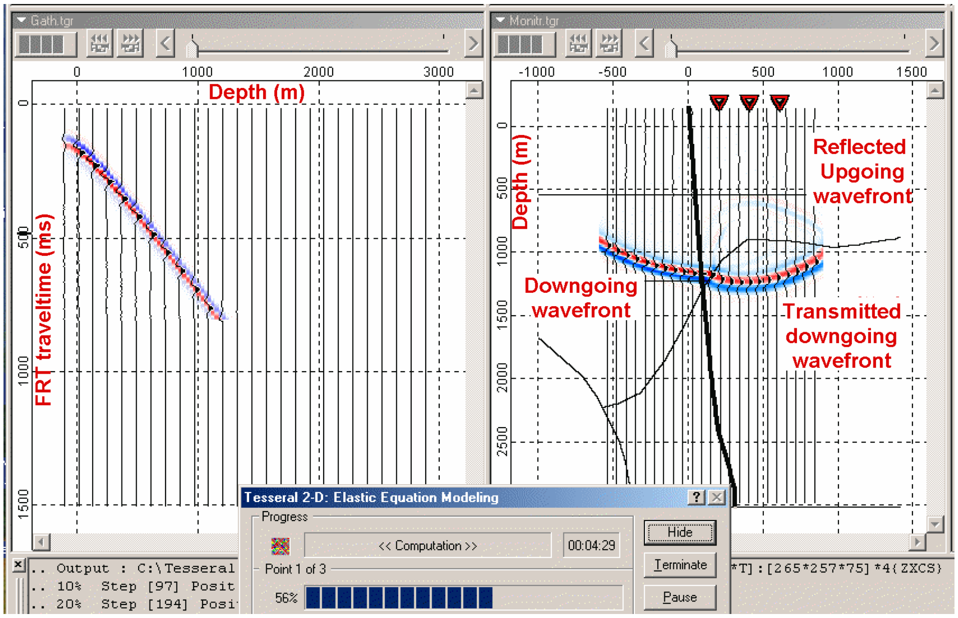


Once the grid is built, the computation begins for source number 1. Note the propagation of the *modeled* wavefront and the downgoing events recorded in the well.



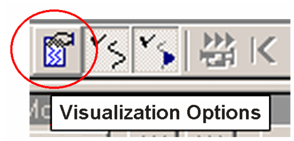
As the computation continues, we see that the downgoing wavefield is about to reflect from the *"reef-like"* structure.



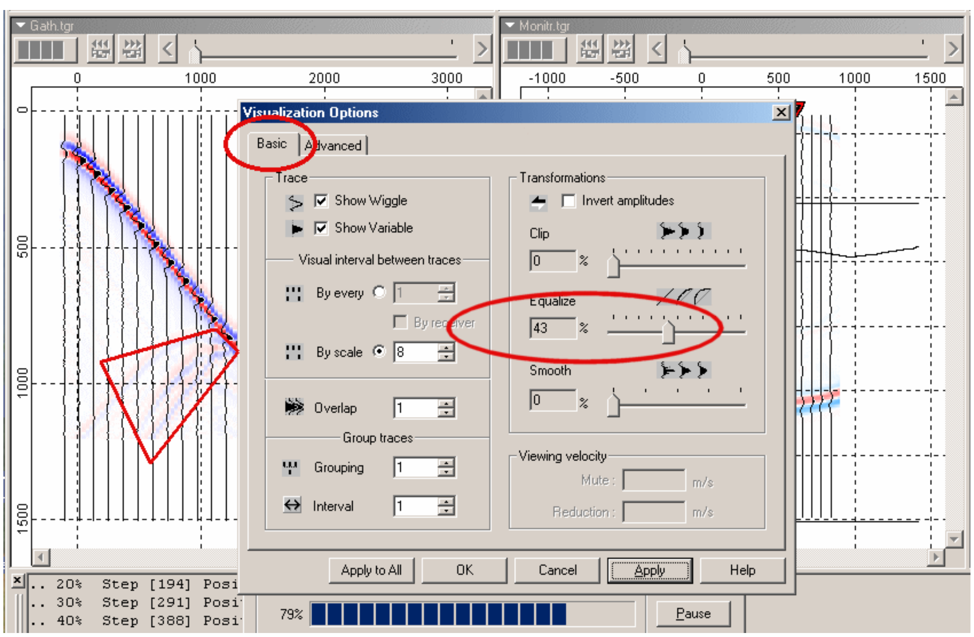


Note the reflected upgoing wavefront. The amplitude of the downgoing event can be 100 times larger than the upgoing events … so the upgoing VSP event is difficult to see on the VSP (on the left) without proper scaling.

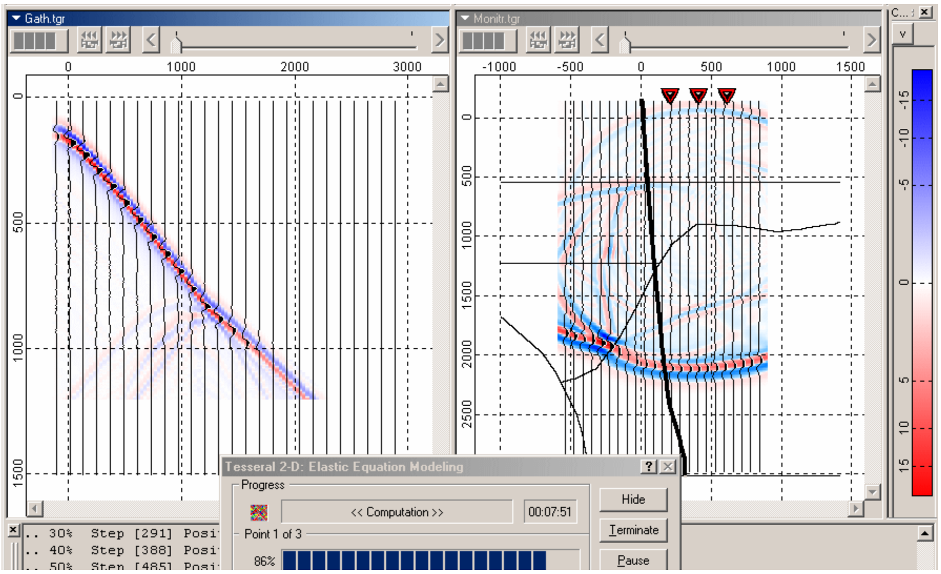
➢ To equalize the up- and downgoing VSP events and wavefronts for display purposes, we use the ***Visualization options*** … use the icon shown (or double-click inside of the pane).



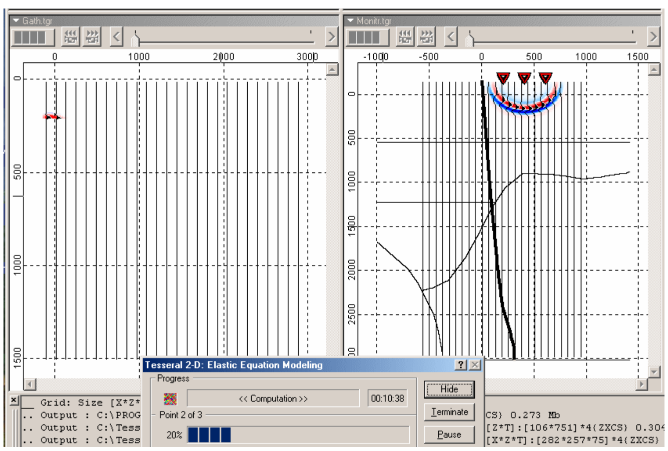
➢ Increase the equalize percentage to view the VSP and wavefront data …

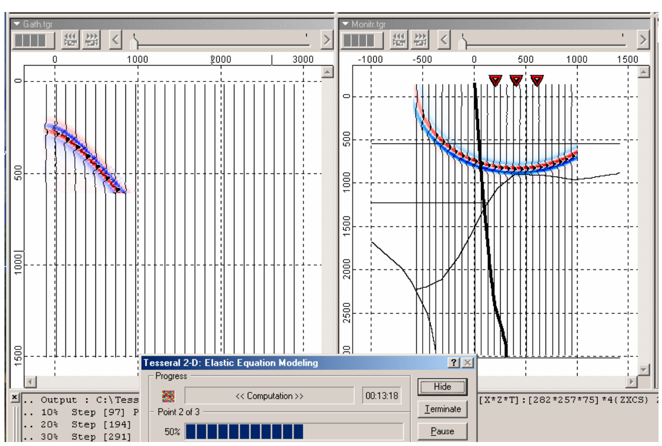


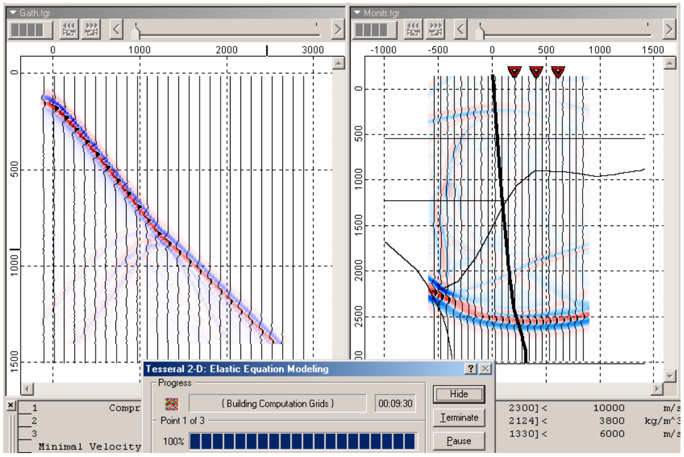
➢ Press ***Apply to All*** …



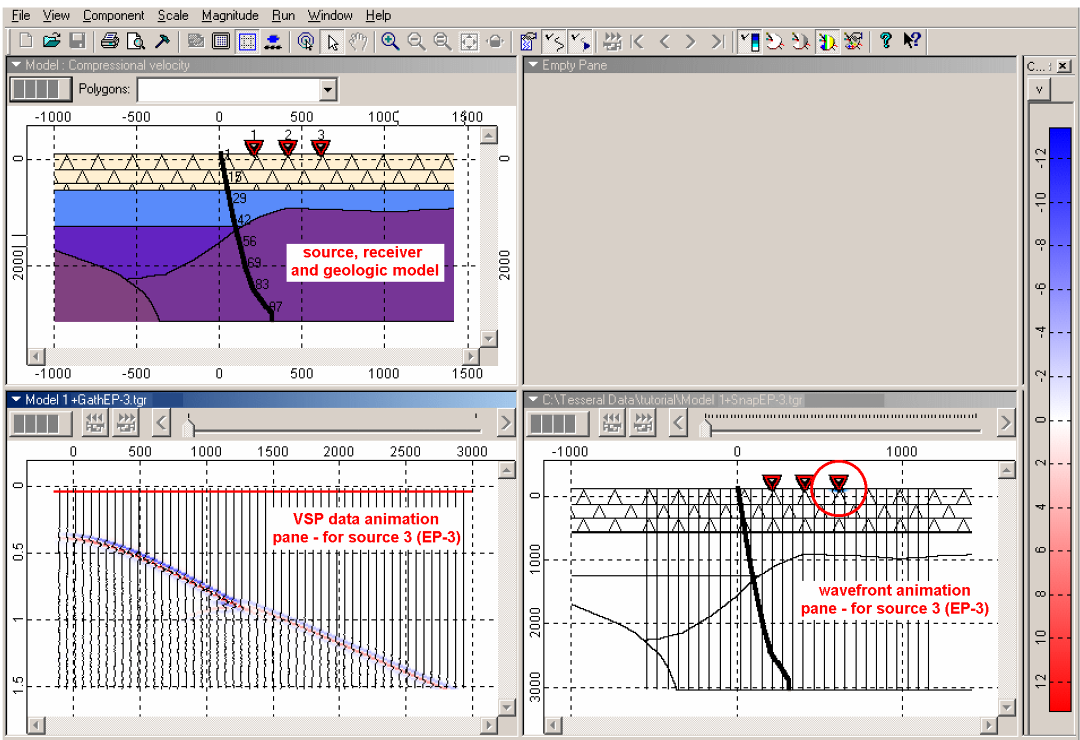
The computation continues from the first source to the third source …



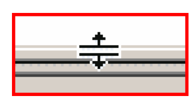




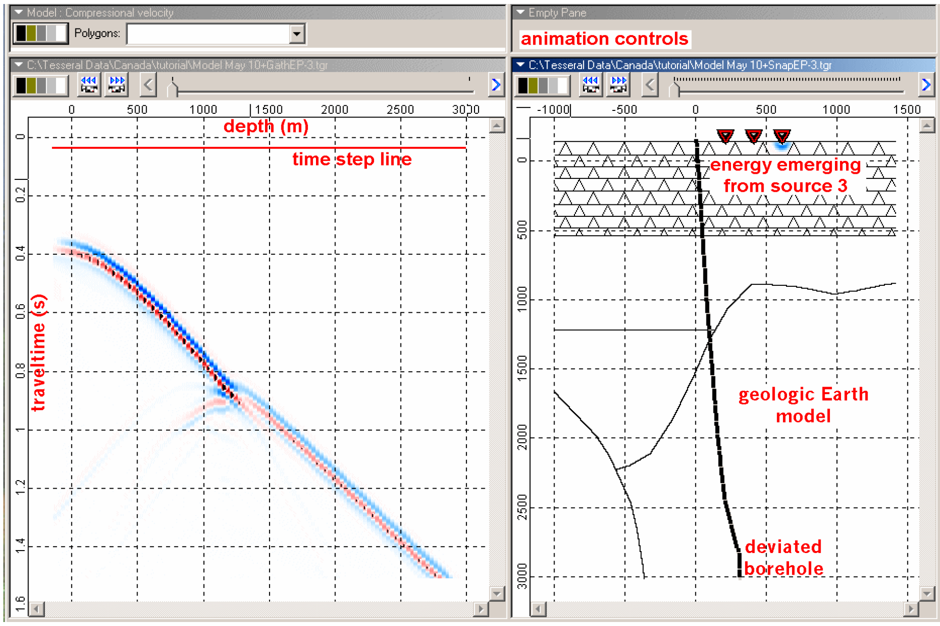
Following the computation, the screen shows the model, last computed VSP data and the model with animation options …

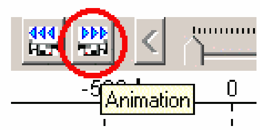


- Place the cursor between the top and bottoms pane until it change shape (two horizontal lines). This allows one to expand the bottom panes to fill almost the entire working area by dragging up the pane boundaries.



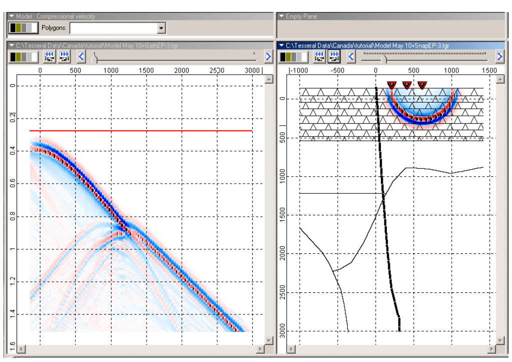
- Pull up the pane border … to create the picture like shown below …

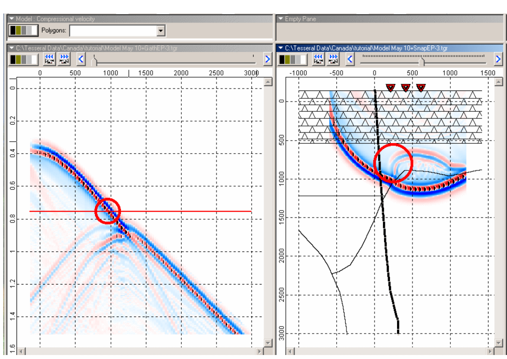


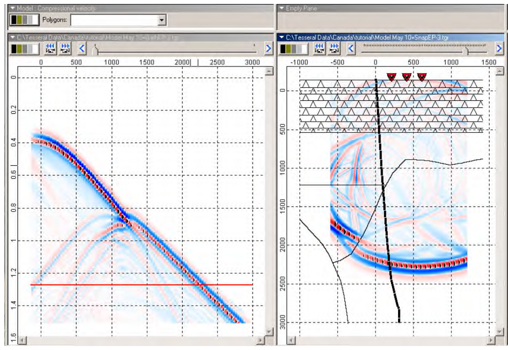


We see a time step line on the VSP data. This is used during the animation process. The animation process will replay the generation of the wavefronts in a movie fashion. The time step line shows how the VSP data is being generated as the wavefronts in the movie propagate. The animation icons in the two panes look like

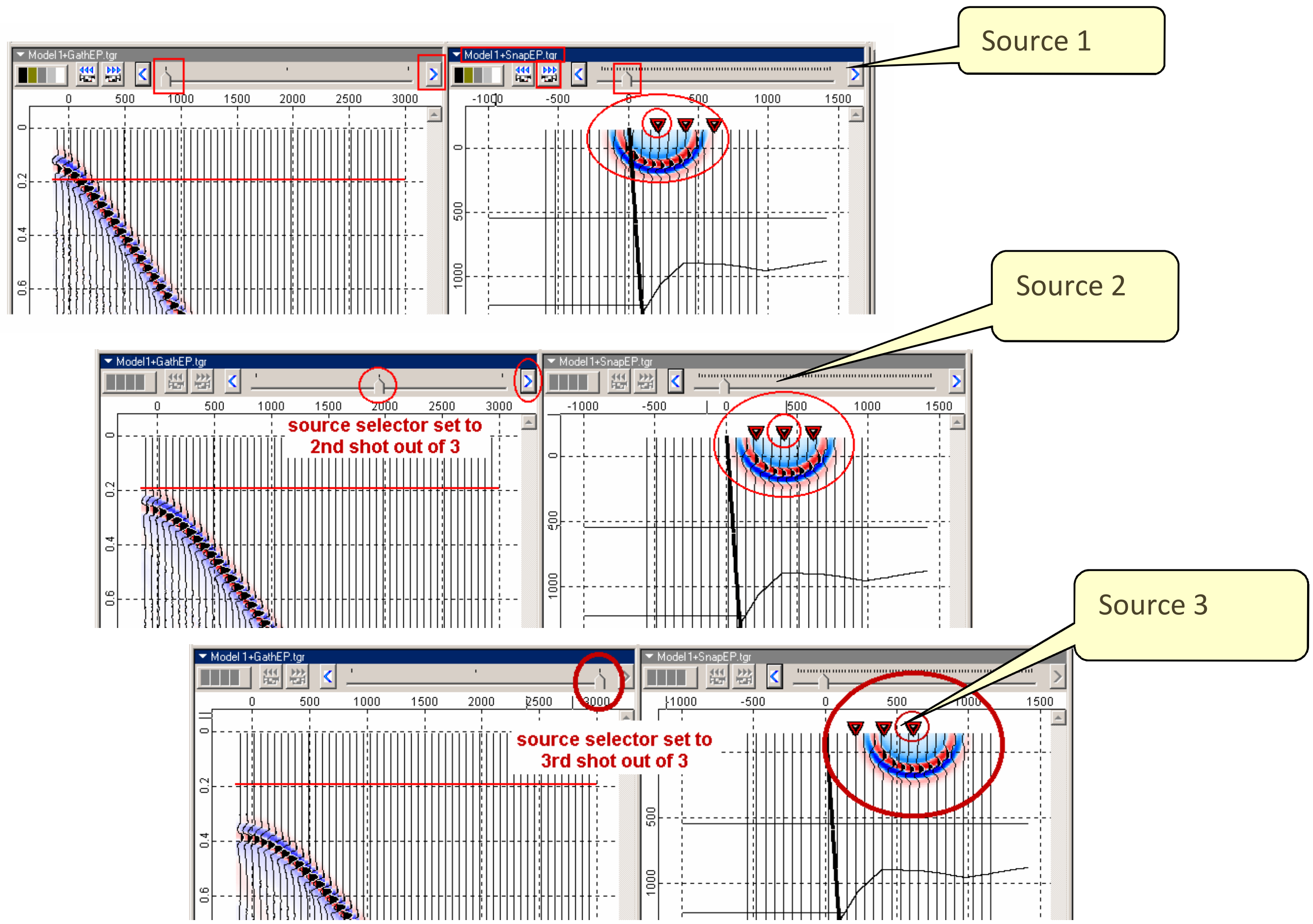
➢ To animate the wavefront propagation, click *mb1* on the animation button. Some example displays are shown below:





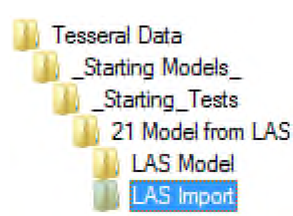


We *can look* at the three shots, *one at a time*, by using the sliding bar on the *Gath-EP.tgr* pane … as shown below …



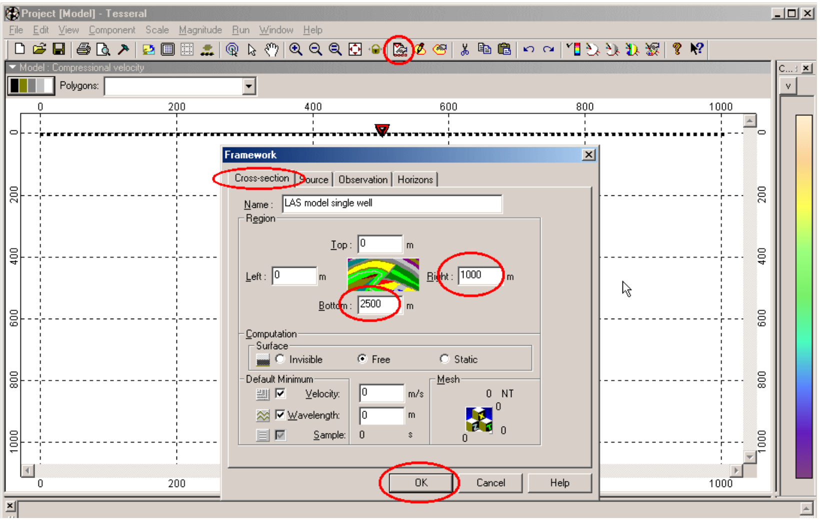
13 APPENDIX A INPUTTING LOG CURVES IN LAS FORMAT AND MODEL BUILDING

In this exercise you can use well-log data supplied with library of prebuilt models.

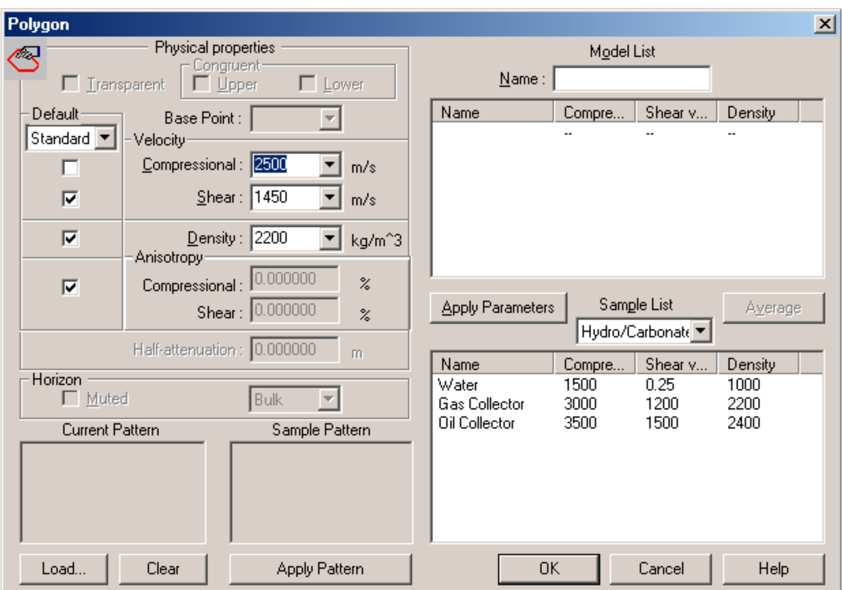


To load LAS files into Tesseral, we need to have a *zero-model* already defined. We then input a log and ask the program to generate a model from the log (using P- and SV-velocities derived from the input sonic logs and the density taken from the input density log).

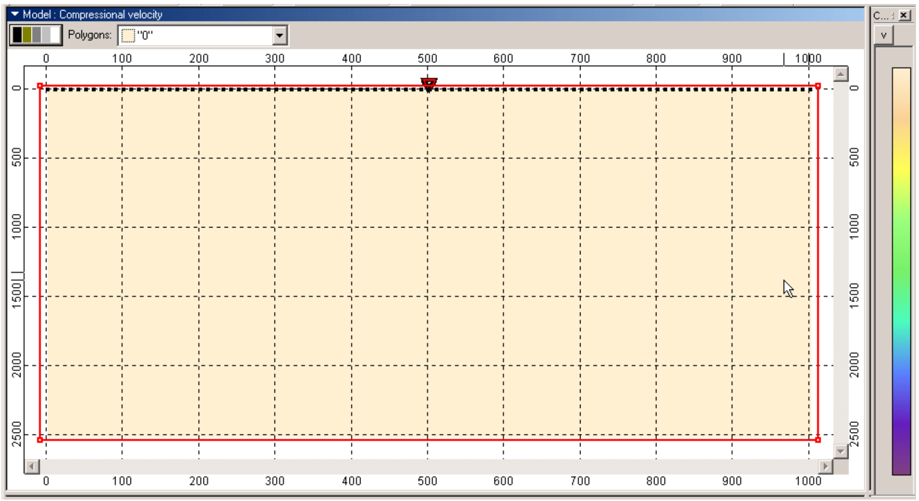
- To generate the *zero-model*, let us start at the beginning … initiate the *Tesseral 2D* package - as it is described in introductory part of this *Tutorial.*



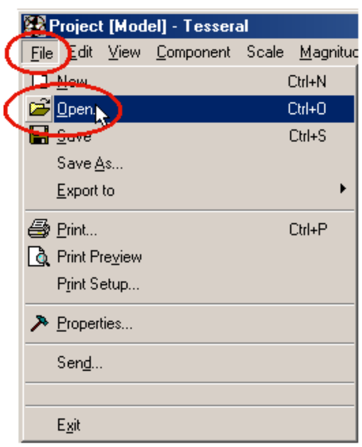
- The main Tesseral menu appears … Click mb1 on the Framework icon (circled in red), … , click on the Observations tab, change the bottom and right boundary distances and click OK.

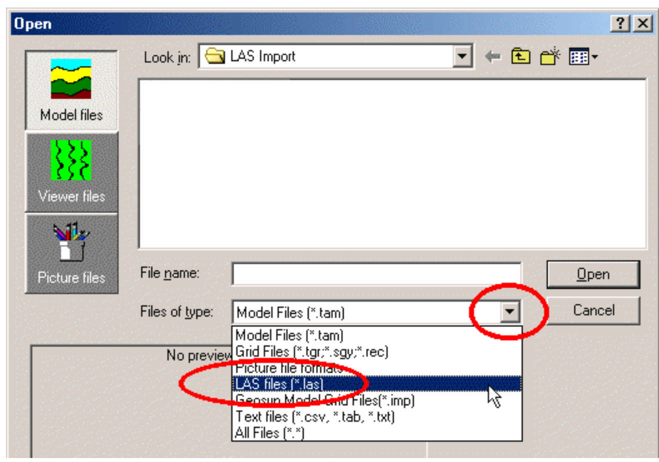


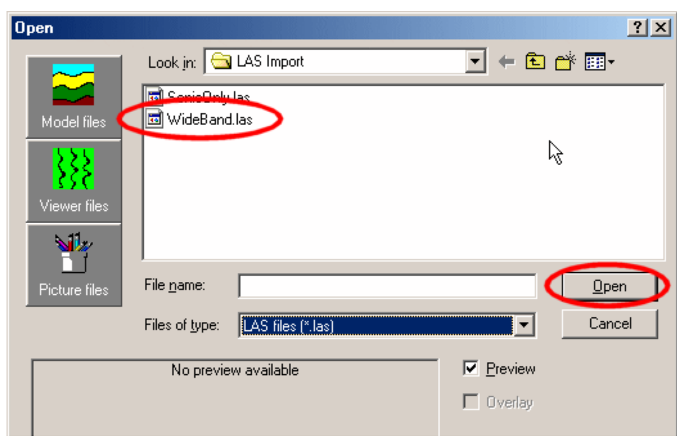
- Enter the polygon properties as shown and press *OK*. This will create a *zero-model* (with *zero-polygon* only) to place the well logs into. NOTE: the depth of the model should be deeper than the deepest log depth.



To bring in the LAS file, use ***File/Open*** as shown below … Search for LAS files using ***Files of type*** …

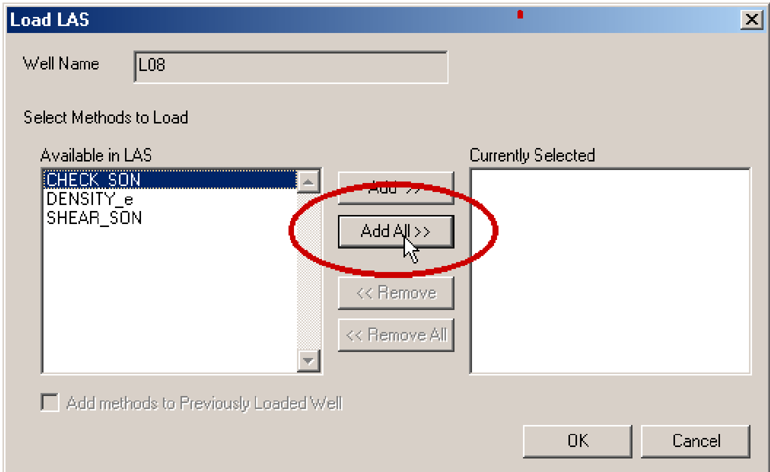


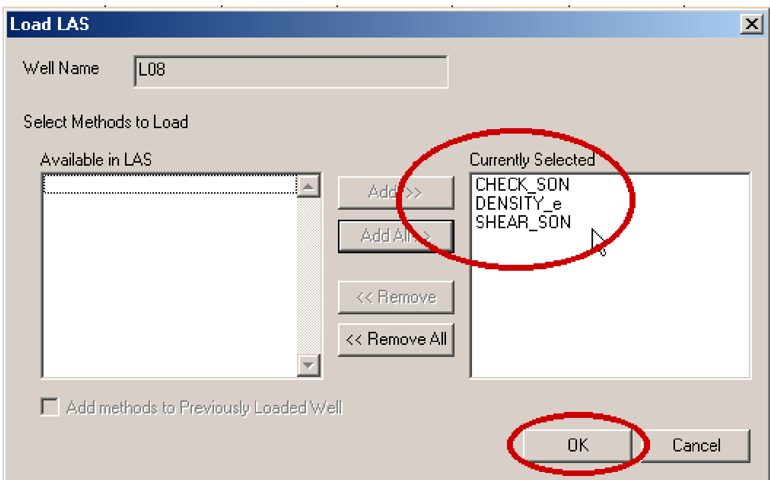




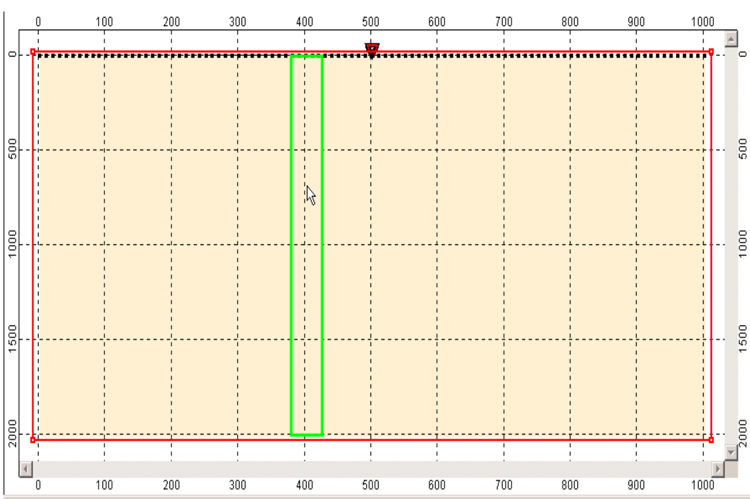
- Choose the WideBand.las file…

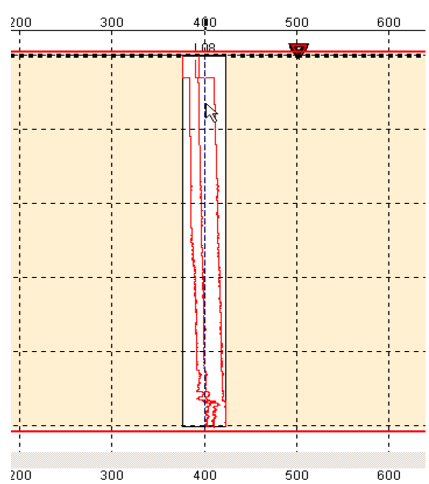
- The next menu allows you to select which curves from the well you want to put into the model … Choose ***Add All*** …



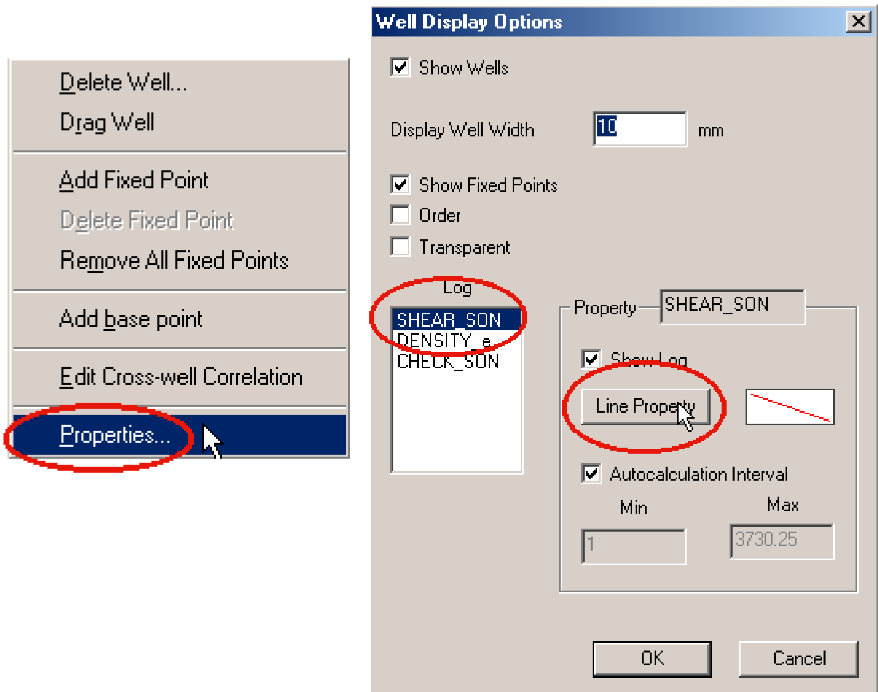


- A rectangle appears inside the model which can be positioned where you want to place it by dragging the box with the cursor …

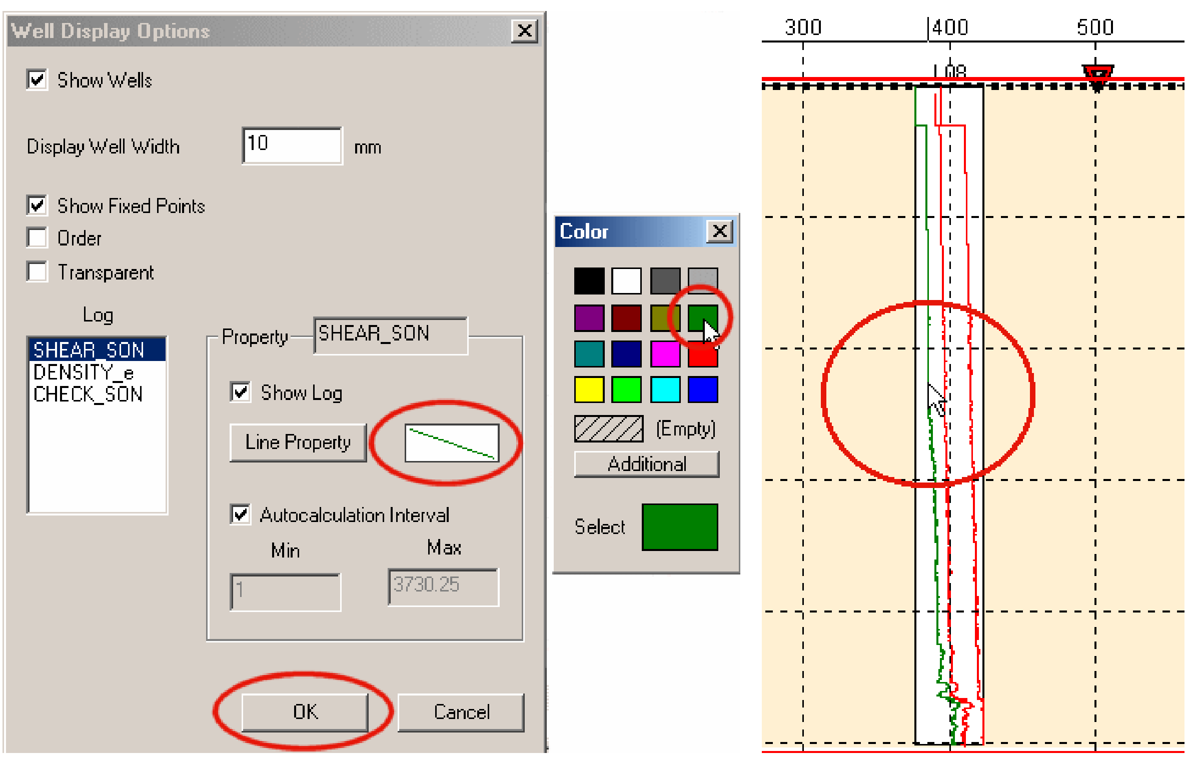




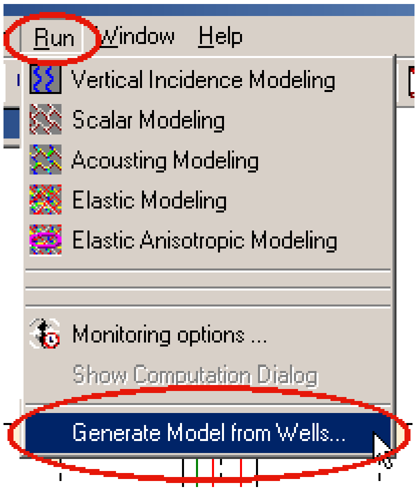
- once the rectangle is in position, click mb1 to bring in the logs …
- Click *mb3* inside the well rectangle to bring up the menu that has the Properties option …



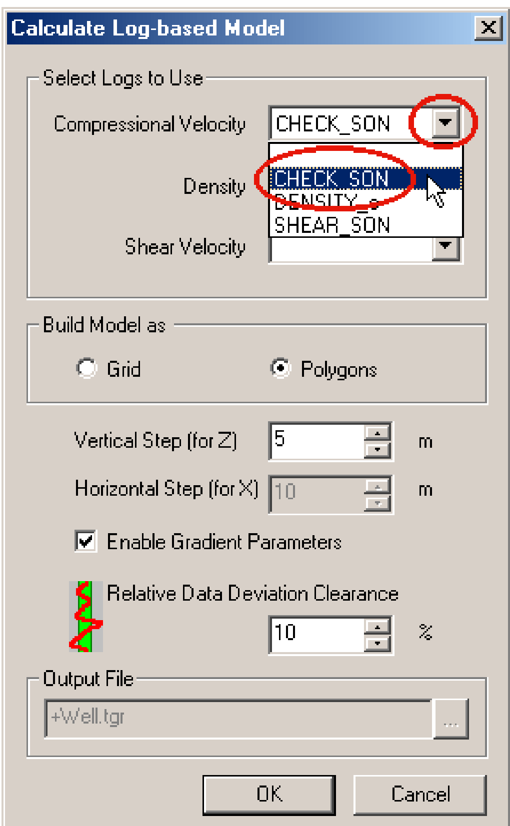
- In this menu, we can change the *Line Property* of the chosen *Log*. This will assist us in telling one curve from the other …
- Select the *Shear sonic* curve and click on Line Property… Choose the green color …
- Press *Ok* to see the curve change color in the model display …

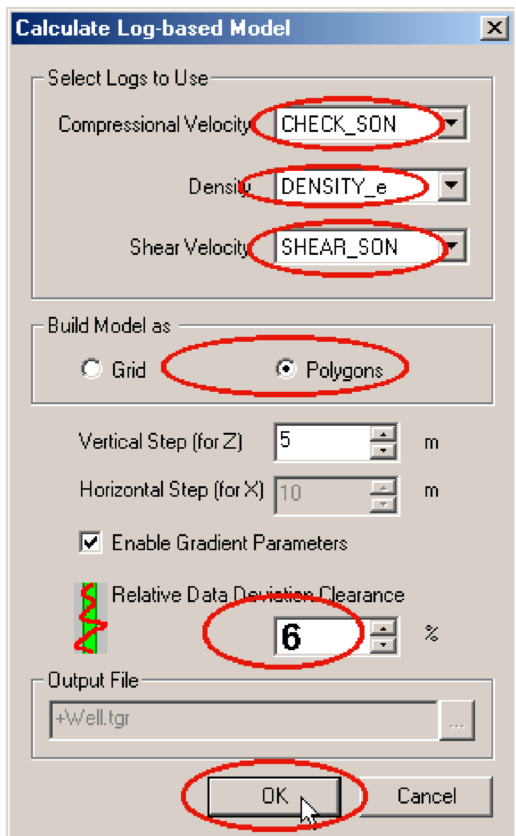


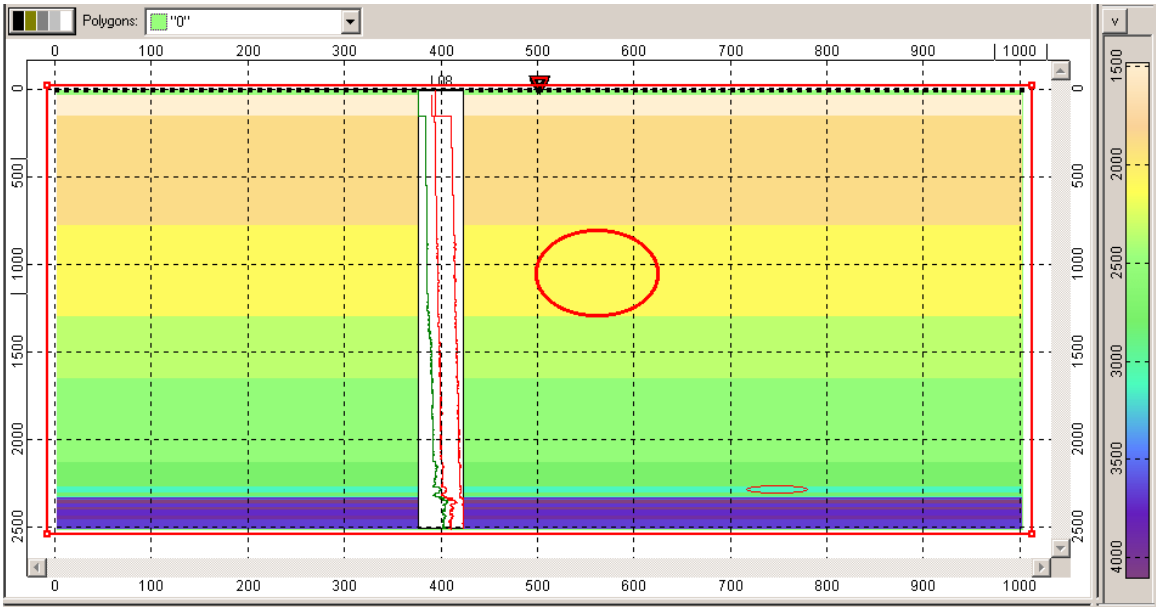
Next, we want to generate the model using the well log curves as input … the model will obviously be composed of horizontal layers …



- Choose the curve names that match the model property and choose 6 (versus 10 for the deviation clearence) as the curves do not show drastic jumps in their overall range of values …

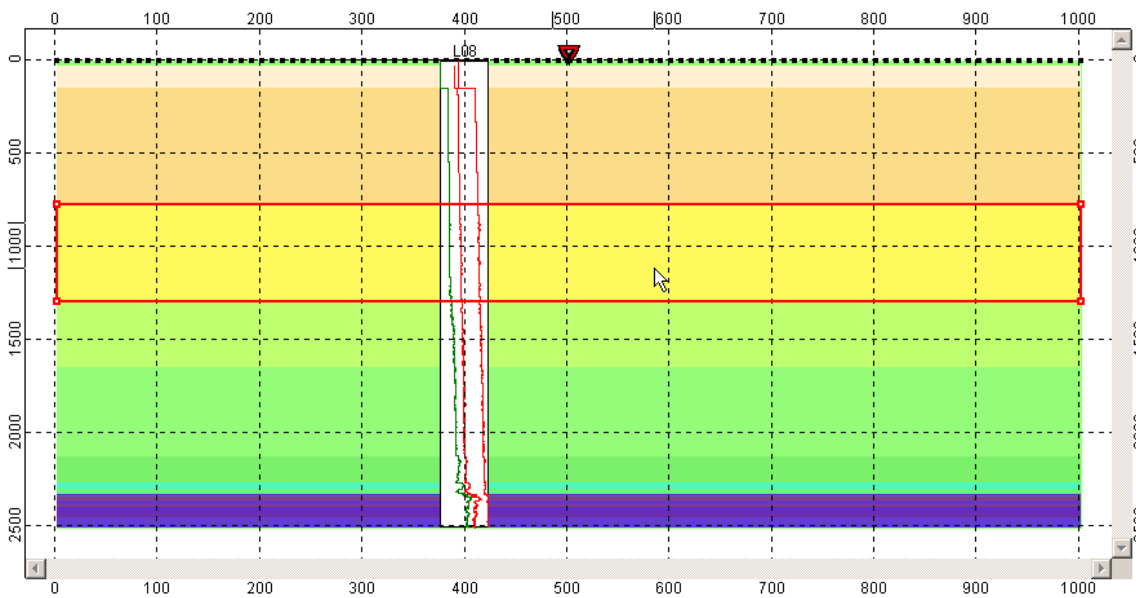




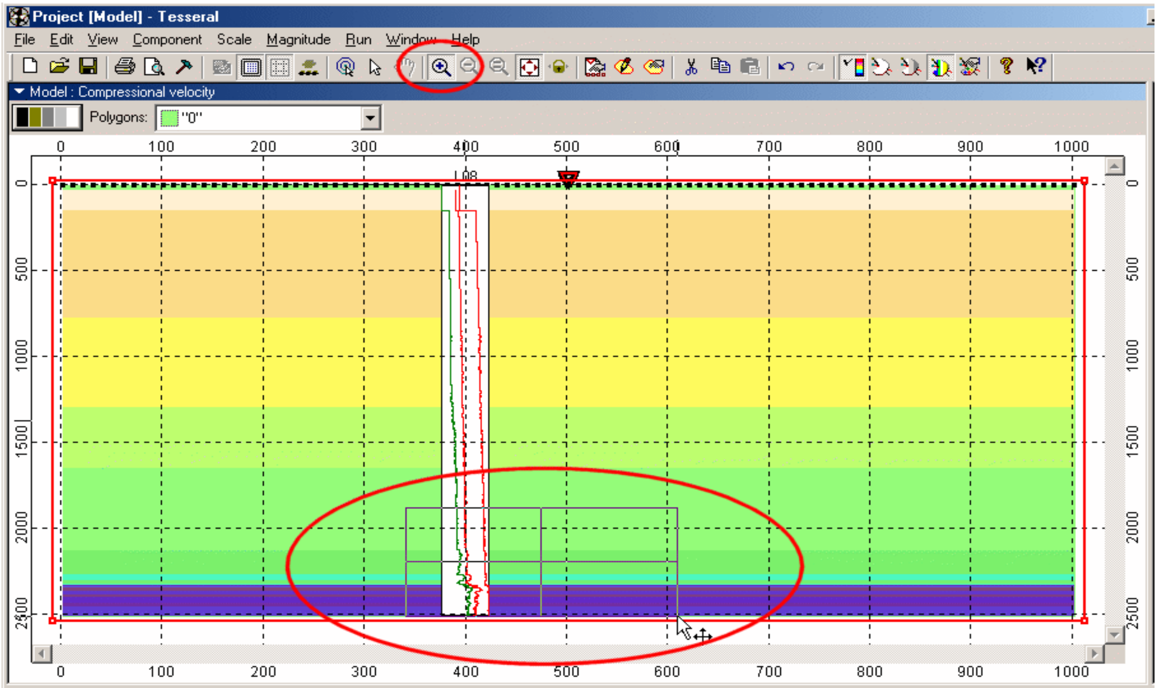


Here we see the model … some of the layers are thick and some are thin. The thin layers are nearer to the bottom of the borehole ..

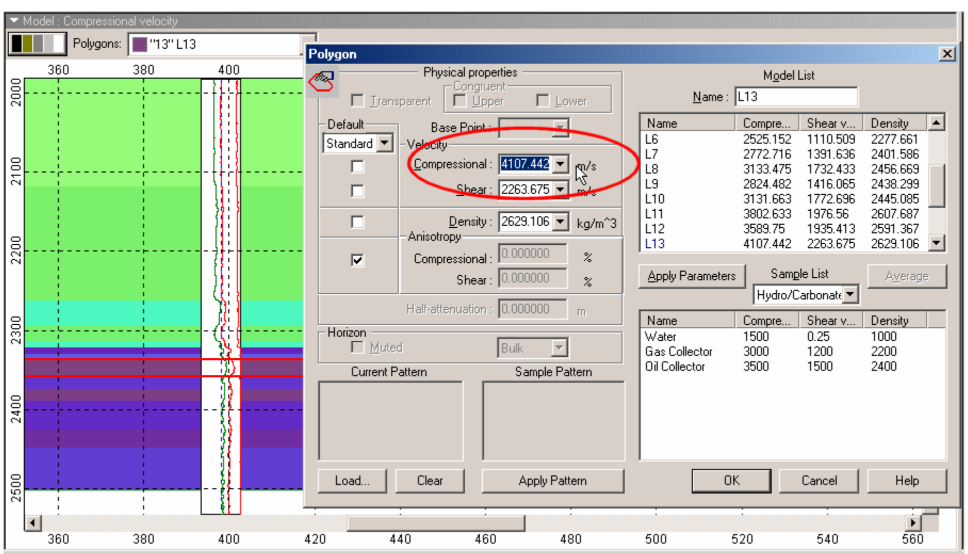
➢ Each layer's polygon can be seen by clicking *mb1* within the layer



➢ To zoom in on one of the thin layers near the bottom of the borehole, we use the magnifying tool … Once the magnifying tool is active, click *mb1* to start one side of the zoom box, drag to enlarge the box and then click mb1 to close the box …



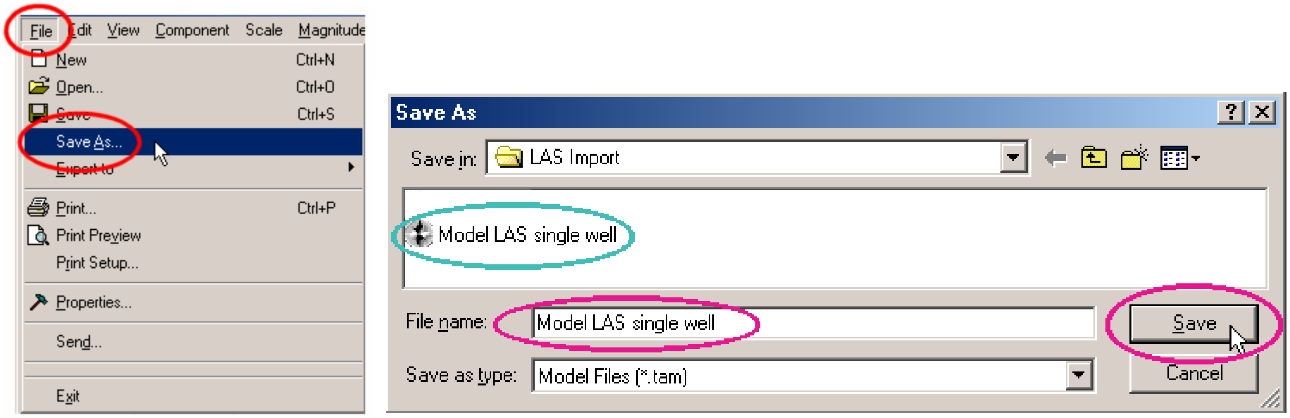
➢ To exit from magnification mode click mb2 or chose (pointing)  from toolbar. To choose a layer, click mb1 on the layer. To bring up the Polygon properties menu, double click mb1 on the selected layer. You can now change the layer properties …



Name	Compre...	Shear v...	Density
L6	2525.152	1110.509	2277.661
L7	2772.716	1391.636	2401.586
L8	3133.475	1732.433	2456.669
L9	2824.482	1416.065	2438.299
L10	3131.663	1772.696	2445.085
L11	3802.633	1976.56	2607.687
L12	3589.75	1935.413	2591.367
L13	4107.442	2263.675	2629.106

Name	Compre...	Shear v...	Density
Water	1500	0.25	1000
Gas Collector	3000	1200	2200
Oil Collector	3500	1500	2400

➢ Let's save the model we have created … to do this Click on *File/Save As…*



➢ Within the ***Save As***... menu, type in the model name and Save