Thunderhead Engineering Consultants
I N C O R P O R A T E D

1006 Poyntz Avenue
Manhattan, KS 66502-5459
USA
www.thunderheadeng.com
(+1) 785-770-8511

Using the PetraSim
Pre- and Post-Processor
for TETRAD

March, 2003

Using the PetraSim
Pre- and Post-Processor
for TETRAD

Thunderhead Engineering Consultants, Inc.
1006 Poyntz Avenue
Manhattan, KS 66502-5459
USA
www.thunderheadeng.com
(+1) 785-770-8511

in joint venture with:

GeothermEx, Inc.
5221 Central Ave., Suite 201
Richmond, CA 94804-5829
USA
mw@geothermex.com
(+1) 510-527-9876

Table of Contents

Using the PetraSim
Pre- and Post-Processor

1. PetraSim

PetraSim is an interactive pre- and post-processor for the TOUGH2 and TETRAD simulators. Our goal is to provide the analyst with a tool to rapidly develop models and view results using those programs.

TOUGH2 (Lawrence Berkeley National Laboratory) and TETRAD (ADA International Consulting Ltd.) are general purpose computer programs for solving multi-phase fluid and heat flow problems in porous and fractured media. Typical applications include oil and geothermal reservoir engineering, nuclear waste isolation, and environmental simulations.

For TETRAD users, PetraSim provides the capability to create and edit InterSim files and to plot GridView (*.gv) and FPLOT (*.plt) time history files.

2. Preliminaries

2.1   Installation

If you download from the web, you will obtain a setup.exe file. Double click on this file to install PetraSim.

If you are installing from a CD, insert the CD in your CD-ROM drive and installation will begin automatically. (If installation does not start automatically, click START on the taskbar, then click RUN and type "D:\setup.exe" where "D:" is the drive letter for your CD-ROM.)

The installation will include PetraSim, and runtime copies of Java, and Java3D. The total disc space for PetraSim, Java, and Java3D is approximately 50 MB.

The CD installs PetraSim in a Microsoft Windows environment. Please contact Thunderhead Engineering to obtain Unix versions.

2.2   Rotate, Zoom, and Pan

Different views use somewhat different actions to move the image.

2.2.1 3D Main Window and Results Display

To rotate and spin the 3D display, click near the center of the image and move the mouse while keeping the left button depressed. The model will rotate or spin as though you had selected a point on a sphere surrounding the model.

To pan, press the "Shift" key while moving the mouse with the left button depressed.

To zoom in and out, press the "Alt" key while moving the mouse vertically with the left button depressed.

To reset the view, select the menu *View/Reset* or type the letter "r".

2.2.2 2D Grid Edit Display

To pan, click the pan button on the toolbar, then click on the image and drag. When finished, turn off dragging by clicking on the pan button again.

To zoom in and out, click the + or – magnifying glasses or use the zoom box tool on the toolbar.

To reset the view, press the reset view button on the toolbar.

2.3   Saving Images

To save an window image for inclusion in a report, select the window you want to save and press the Alt/Print Screen keys simultaneously. This puts the image in the clipboard

where it can then be pasted into a document. You can change to a white background by selecting *View/Color Scheme…* .

3. Creating a New Model

PetraSim can be used either to create a new model or to edit existing InterSim files (see Importing Legacy InterSim Files). It can also be used to view existing GridView contour data and plot file time history data (see Results Display). In this section, we describe the steps to create a new model and write a grid file in the InterSim format.

Before running these examples, make sure you have selected TETRAD as the simulator (Files/Preferences…, then restart PetraSim). This will be remembered for your next startup

3.1 Startup

Start PetraSim from the Start menu by selecting *Start/Programs/PetraSim*. You will be presented with a window, as shown in Figure 1.
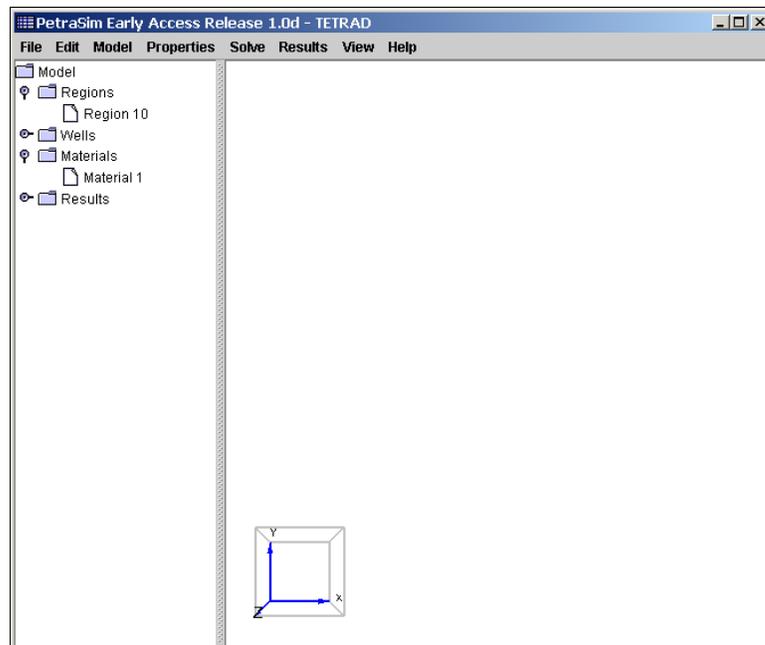

Figure 1: PetraSim startup window

The main window is the 3D display, which is the primary view of the model and provides reference to the user. The object tree displayed on the left is presently only illustrative. Capability will be added to select items in this tree, see them highlighted in the 3D display, and change their properties. This will provide a powerful way to interact with the model.

3.2 Define the Reservoir

The first step in creating the model is to define the reservoir boundary. Select *Model/Define Boundary*. This will display a dialog box in which the model boundaries can be defined, Figure 2. Select *OK* to accept the default data in the dialog box. A default material is assigned to the model. The model is now displayed in the 3D window, Figure 3.

The model can be manipulated using the mouse. To rotate and spin the model, click near the center of the image and move the mouse while keeping the left button depressed. To pan, press the "Shift" key while moving the mouse with the left button depressed. To zoom in and out, press the "Alt" key while moving the mouse vertically with the left button depressed.
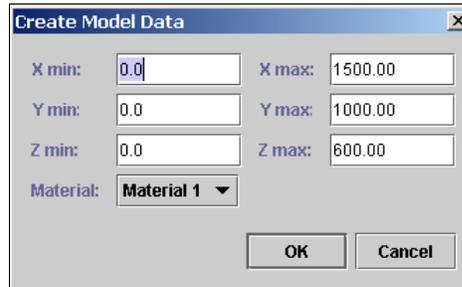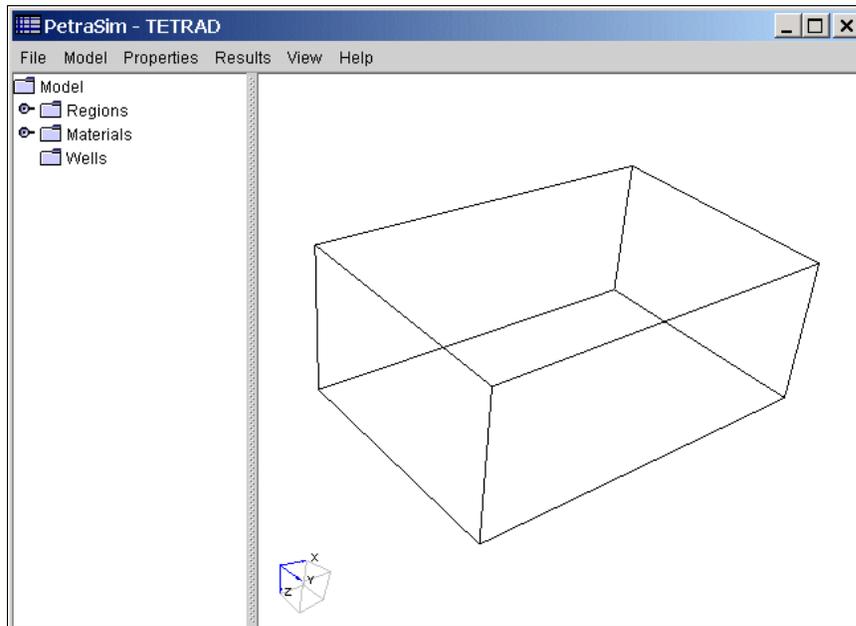

Figure 2: Create model data

Figure 3: Creation of model boundary

3.3   *Materials*

Materials allow the user to define material properties globally (for TETRAD the global properties include porosity, X permeability, Y permeability, Z permeability, and density). Materials can be assigned to regions in the model and all cells in the region will then inherit the material properties of the region. For any region or cell, the user can override the global property value by editing the local values for the region or cell (this will be demonstrated in the Grid Edit section).

To modify a material property, select ***Properties/Materials…***, the Material Data dialog box is shown in Figure 4. A default material was automatically created when the model was created.
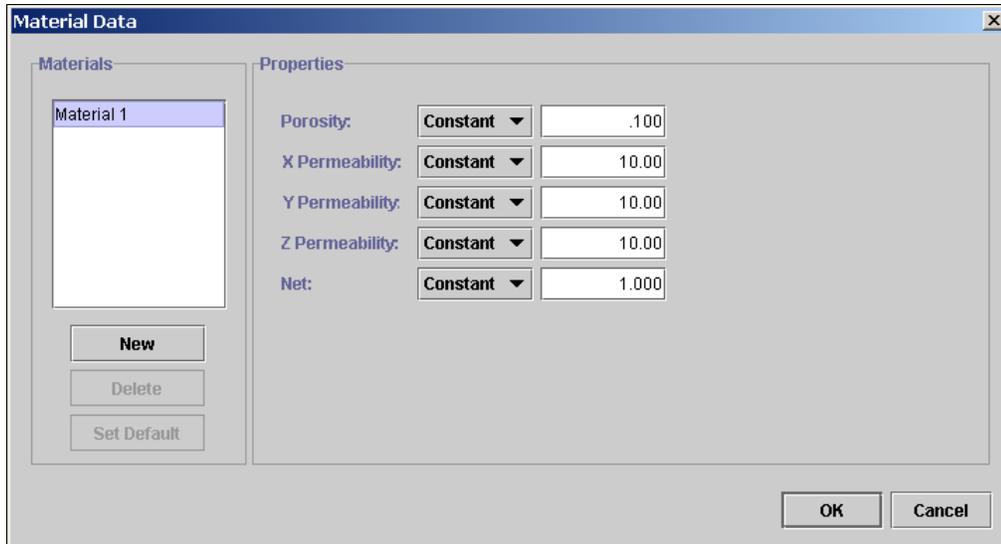

Figure 4: Material properties dialog

As shown in Figure 4, the default properties are constants; however, properties can also be linear functions or defined by contour map data. We will change the porosity so that it will be defined by a contour map. To do this, select the combo box to the right of the Porosity label. In the file selection dialog that is displayed, select the porosity.ctr file. After selecting this file, the dialog will change to indicate that porosity is now based on contour data, Figure 5. When the file is read, PetraSim copies the data into its own database.
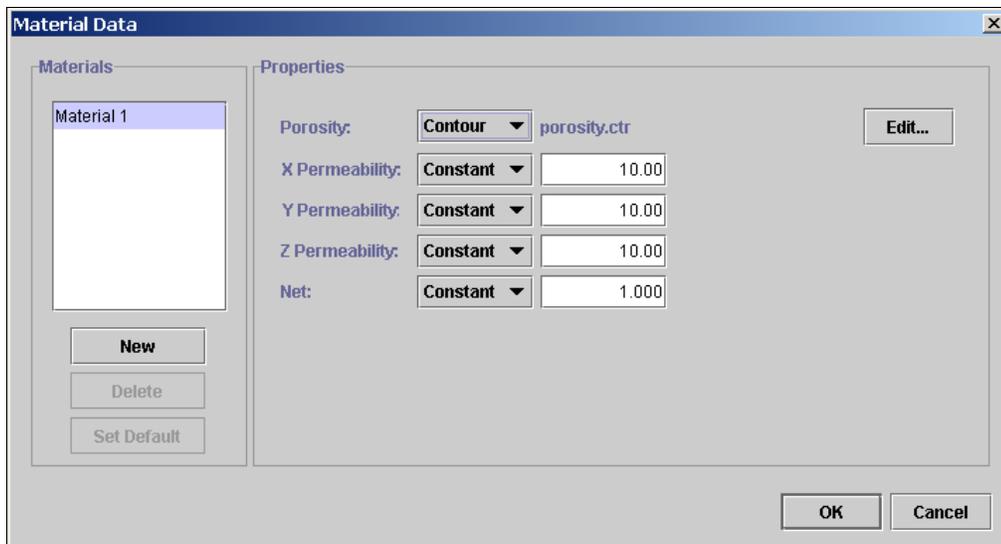

Figure 5: Porosity changed to contour data

The contents of the porosity.ctr file are shown in Table 1, including the definitions of the data. The pattern is to give a flag, followed by the data. Contour data are defined in 3D as 2D contour planes stacked in the Z (depth) direction. Linear interpolation is used within the 2D contour planes and between each plane in the Z direction. When a <depth> is defined, it is used for all following contour data until a new <depth> is defined. In this example, we use only one <depth>, so all data is uniform in the Z direction. The <contour> flag indicates the definition of contour data. Multiple <contour> data sets can be included.

Line	Data in File		Meaning
1	Example contour file		Title. Required.
2	<top origin>		Axis flag. Not required.
3	<depth>		Depth flag. May be repeated. Required.
4	0.0		Depth of contour data to follow. Used until new depth defined.
5	<contour>		Contour flag. May be repeated. Required.
6	0.1	4	First is contour value, second is number of X Y coordinates.
7	0.0	0.0	X Y coordinate.
8	1500.0	0.0	X Y coordinate.
9	0.0	1000.0	X Y coordinate.
10	1500.0	1000.0	X Y coordinate.
11	<contour>		Contour flag.
12	0.25	3	First is contour value, second is number of X Y coordinates.
13	500.0	0.0	X Y coordinate.
14	1000.0	0.0	X Y coordinate.
15	750.0	1000.0	X Y coordinate.

Table 1: Data in porosity.ctr file

To create a second material, select the NEW button. The New Material Dialog will allow the user to define the material name and to base the new material properties on an existing material, Figure 6. Select the OK button to create Material 2.

Figure 6: New Material dialog

Now there are two materials in the material list and Material 2 is selected by default. Notice that since Material 2 was based on Material 1, it currently has the same properties as Material 1. Now edit the X permeability of Material 2 to be 250, Figure 7, and select OK. We now have two materials. Material 1 uses a contour for porosity and has an X permeability of 10. Material 2 uses the same contour for porosity, but has an X permeability of 250.
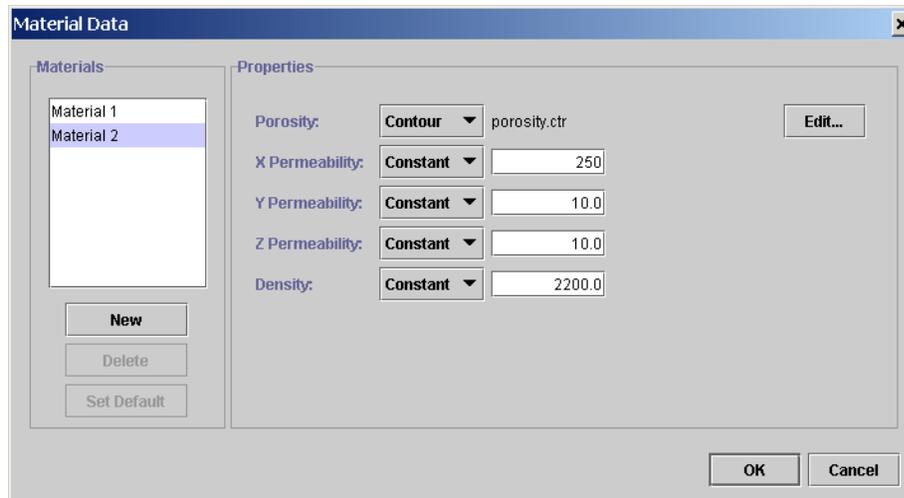
Figure 7: Material 2 values

3.4 Create a New Material Region

We now illustrate how different material regions can be defined at the model level. This is done by inserting a material boundary into the model and then defining the materials on either side of the boundary.

To add the plane, select **Model/Add Internal Boundary**. There are several options for creating a boundary, we will use the "Point + Normal" method. Select this option and the input the data for a point at (500, 500, 300) and the normal as (0.6, 0.4, 0.4), as shown in Figure 8. Make sure you enter all six numbers. You can view the material boundary as shown in Figure 9.
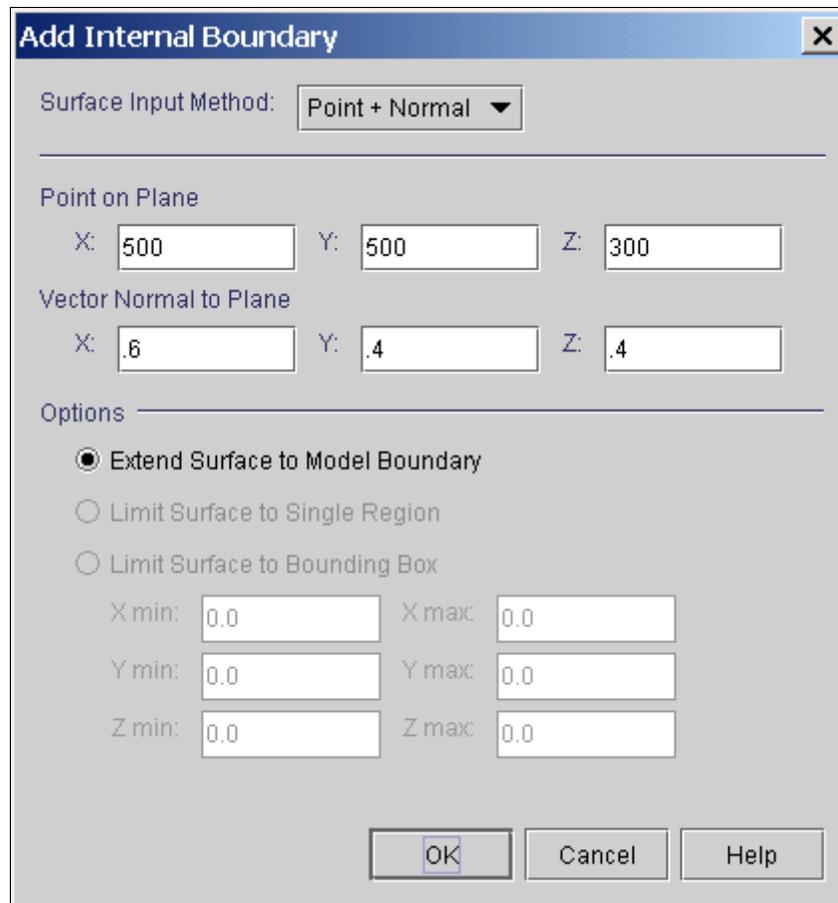
Figure 8: Inserting a material boundary
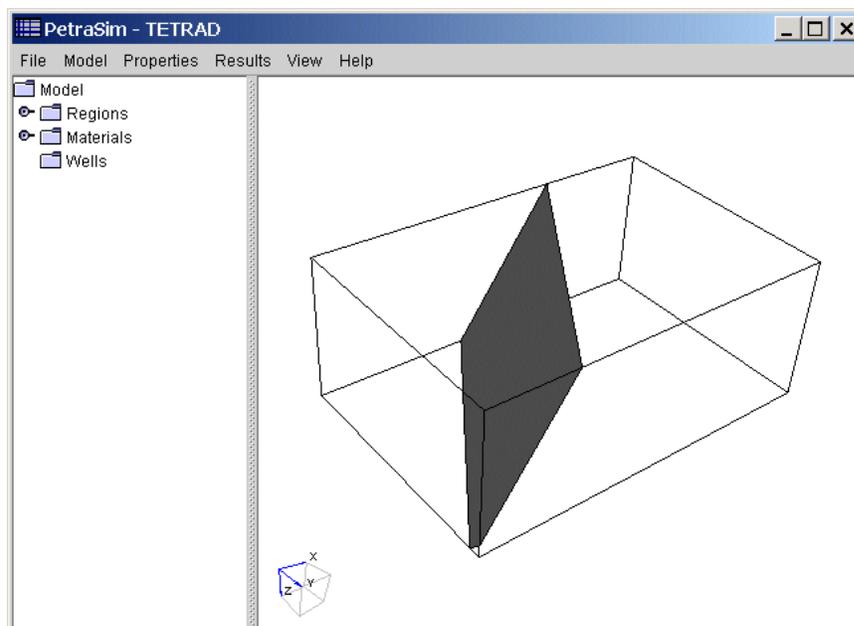
Figure 9: Internal view of material boundary

Now we will assign different material properties to one of the regions we created. With a single click, select the left region of the model. The selected region will be highlighted as shown in Figure 10.
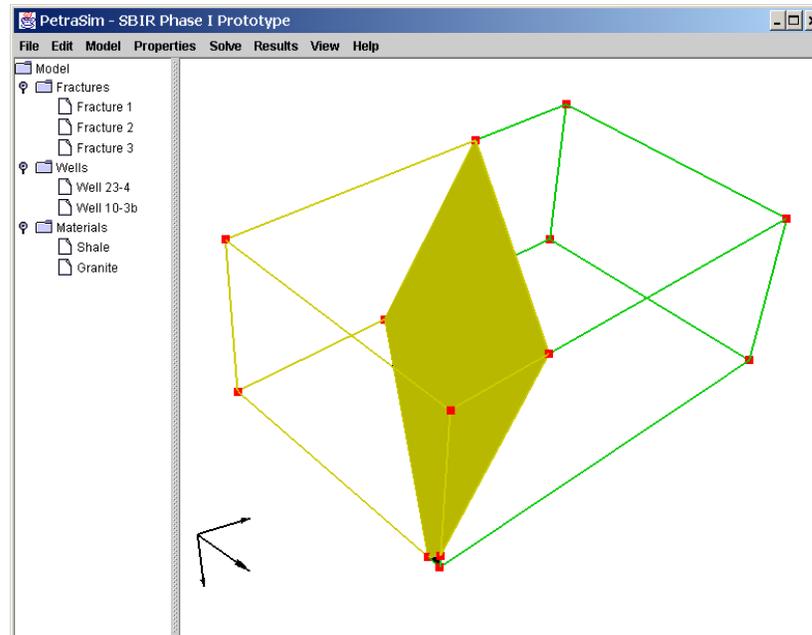
Figure 10: Selecting a region to define the material

Now select **Model/Edit Selection…** and the dialog shown in Figure 11 will be displayed. This dialog allows the user to assign a different material to a region. It also allows the user to change the value of a property (and initial condition) in the region. We want the selected region to use Material 2, so select Material 2 in the combo box. We will also change the Y Permeability in this region to be 25. With these changes, the dialog should be as shown in Figure 12.
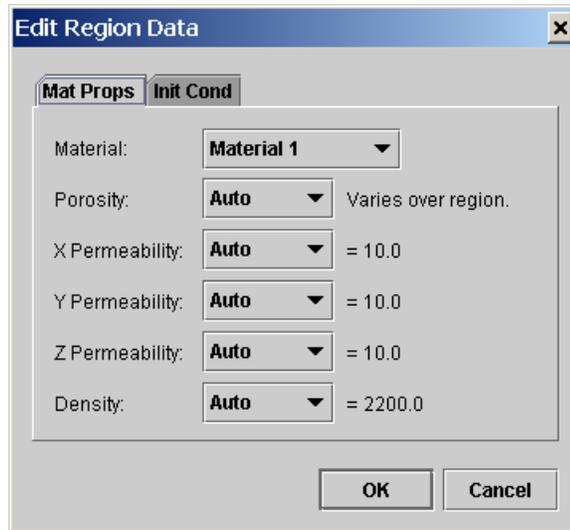
Figure 11: Properties of selected region
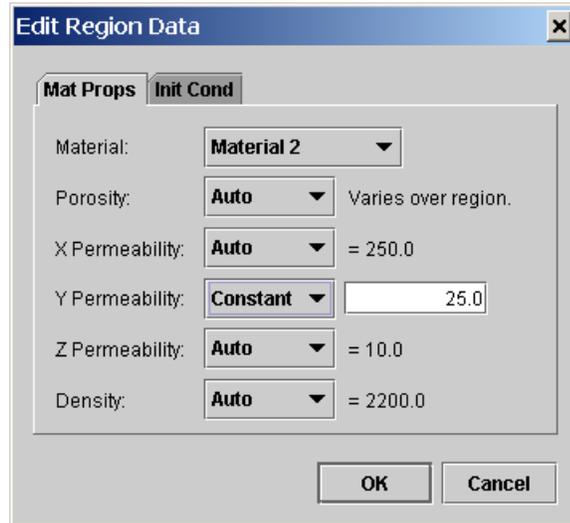
Figure 12: After selecting Material 2 and changing Y Permeability

3.5   Initial Conditions

Similar to material properties, initial conditions can be defined over the model and regions by constants, linear functions, or contour data.   We will change the initial pressure to vary linearly with depth. Select *Properties/Initial Conditions…* to see the dialog shown in Figure 13. Select the Function option in the Pressure combo box and change the values to those shown in Figure 14. After selecting OK, the initial conditions will be as shown in Figure 15. Select OK again to leave the Initial Conditions dialog.

Figure 13: Default initial conditions

Figure 14: Definition of linear distribution of pressure

Figure 15: After changing pressure to linear function

3.6   Add a Well

Let us now add two simple wells to the model. Select ***Model/Add Well*** and input the data shown in Figure 16. Repeat this, using the data in Figure 17. At this release, wells only are geometric data, capability to define boundary conditions associated with wells has not yet been added.

Figure 16: Adding first well

Figure 17: Adding second well

After adding both wells, the display should look like Figure 18. The user can add more wells or can create curved wells using connected line segments. The capability to read in the well geometry and completion intervals from a file will be added, simplifying the user's input.
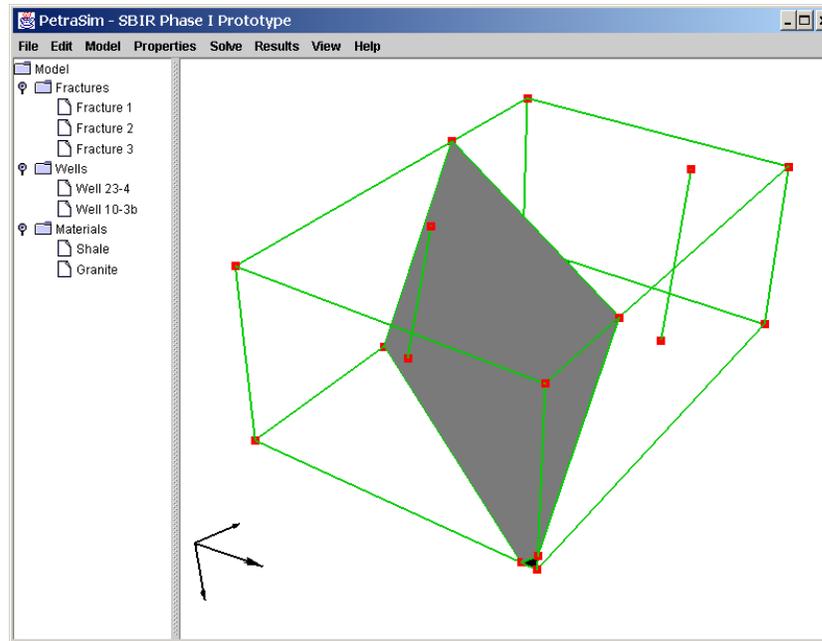
Figure 18: After adding wells

3.7   Create a Grid

To create a grid that lies on the model, select **Model/Create Grid** and give the grid divisions shown in Figure 19. We will create an XYZ Grid. The factors can be used to create a graded mesh spacing. We use a factor of 1.2 in the X direction.
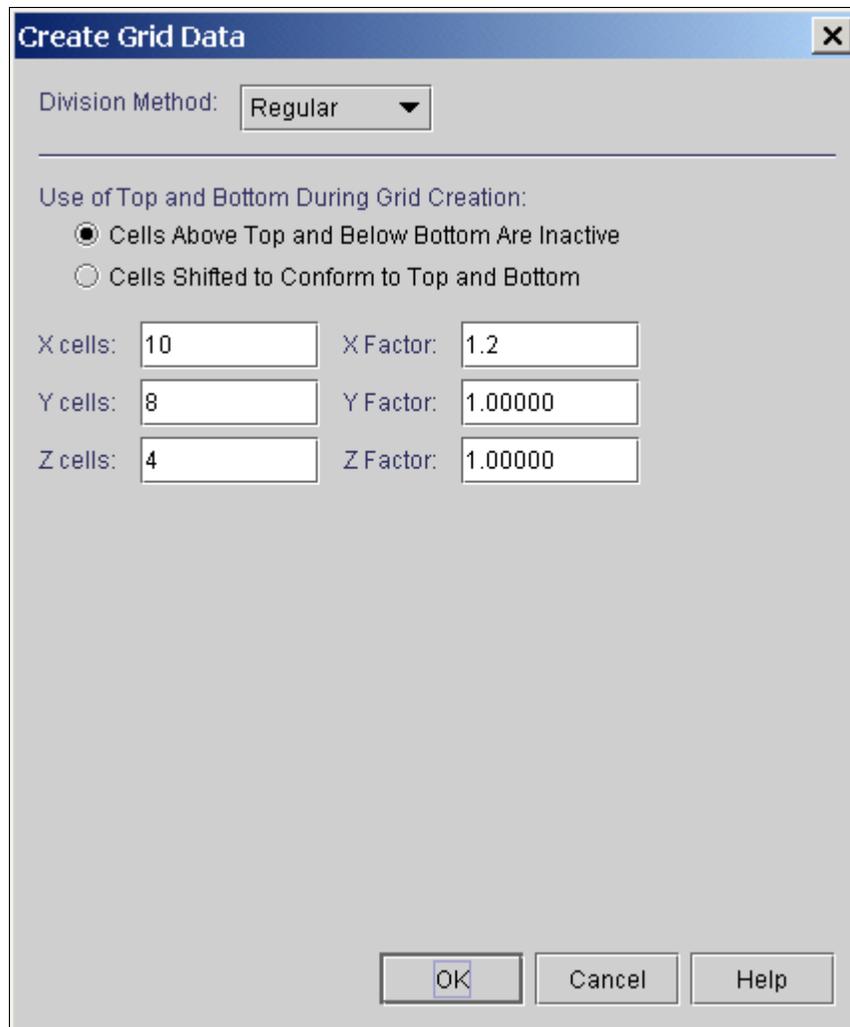
Figure 19: Grid divisions

Turn the boundary back on by selecting *View/Boundary*. You can rotate, pan, and zoom the model to obtain a better view, Figure 20.
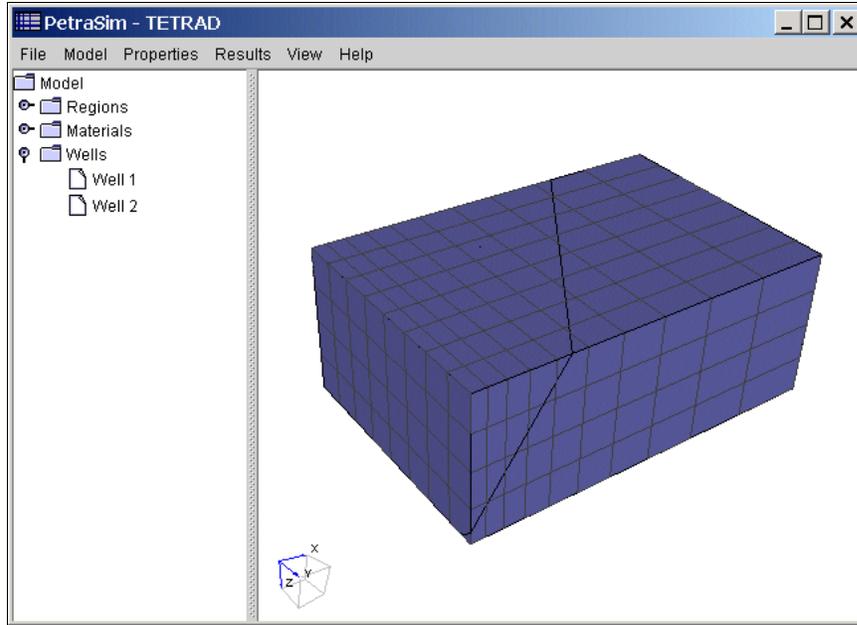
Figure 20: Model after creating grid

When the grid cells are created, they inherit the appropriate materials associated with the different model regions. In the following section we show how it is possible to edit the properties of individual cells.

3.8   Saving the Model and Grid

At any time the user can save the model *File/Save*. The user can also write an file in InterSim format by selecting *File/Write InterSim File…*. Do this now, using the file name "demo".

4. Editing the Grid

After the grid is created, it can be edited and the properties of each cell displayed. Select *Model/Edit Grid* to open a new window that allows the user to work with each layer of the grid. This editor allows detailed control of cells.

4.1   Viewing Cell Properties

When the grid was created, the cells inherited the properties of their associated regions. For example, the porosity was read from a contour file. To view porosity, select it in the Property combo box. The view should appear as in Figure 21.
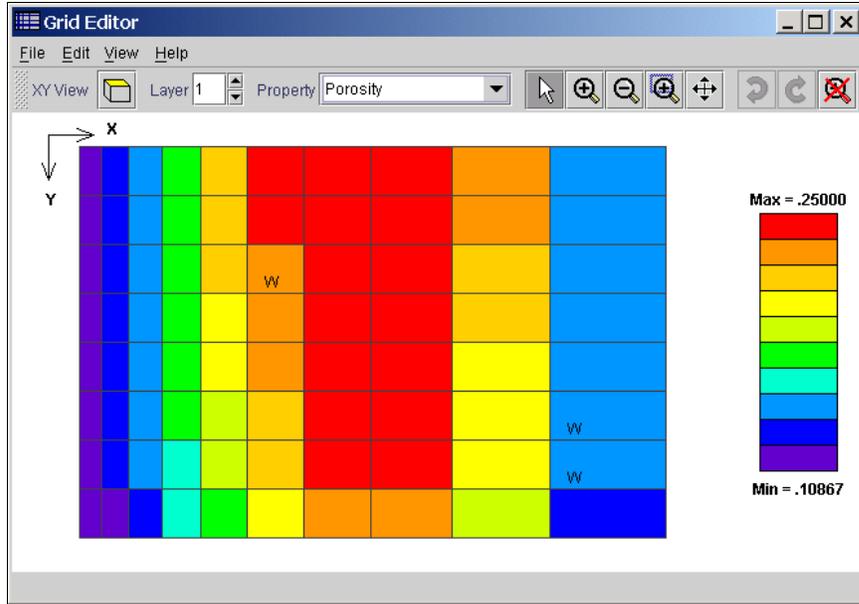
Figure 21: Display of porosity

As can be seen, the cells have different porosities based on their location in the model. The layer numbering starts at the origin, so this is the view of the top layer. In this example, there was no variation with depth, so all layers are the same. As the cursor is moved over the display, the X, Y, and Z coordinates and the property value are shown in the bottom message region of the window. The user can also select *View/Display Values* to see the value in each cell.

Recall that we defined X Permeability to be different in the two regions. The use can view this and move through the layers to see how the dipping material boundary changes the properties with depth.

To change the view from the XY plane to a different plane, click on the toolbar view box. Figure 22 shows the pressure in the XZ plane. The pressure was defined by a linear function that increased with depth.
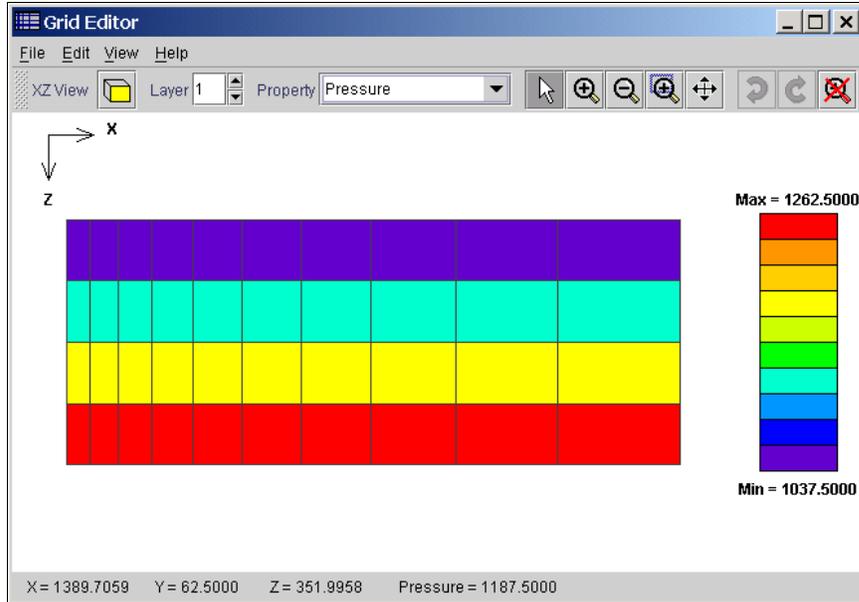
Figure 22: View of pressures in X Z plane

4.2   Changing the Cell Material

To change the property value of a single cell or of a group of cells, use the mouse to click and drag a box around the cells to be selected. The selected cells will be highlighted when the mouse is released. A right click and selection of the pop-up *Material…* menu allows the user to edit the selection. Alternately, select *Edit/Edit Selected Item*. Select a cell, right click, select Properties… and the Edit Cell dialog is displayed, Figure 23. This dialog gives complete control of data at a cell level. If desired, any of the properties can be changed only for that cell. In addition, it is possible to define whether the cell is active and grid refinement for the cell.
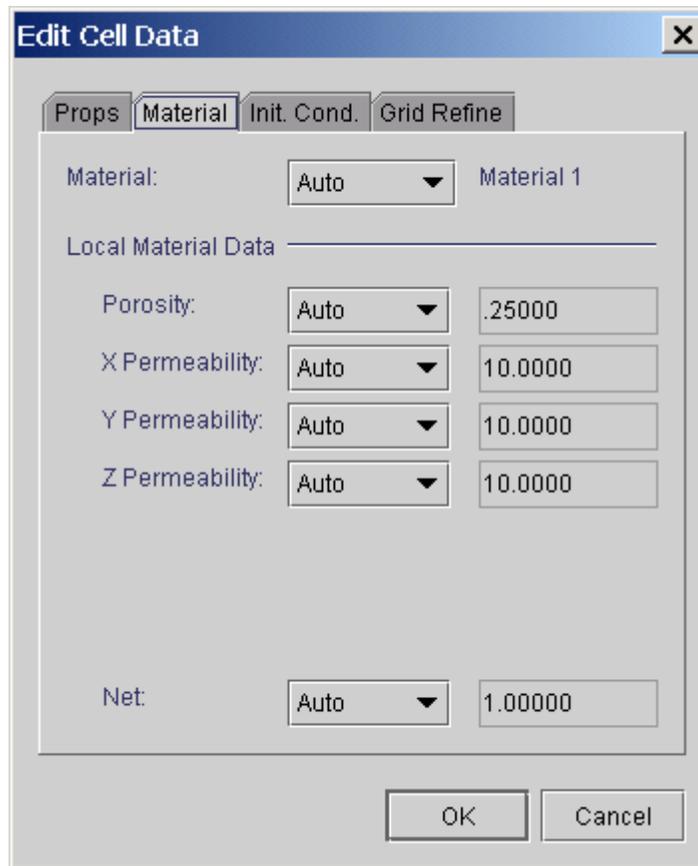
Figure 23: Edit cell dialog

4.3   *Dragging a Gridline*

When editing the grid, the user can select an existing grid line to highlight it, then select the Grid Editor menu ***Edit/Edit Selected Item***. Alternately, the user can double click or right click on an item to activate appropriate editing functions. Figure 24 shows the grid after moving gridlines. Note that the corresponding changes are displayed in the 3D view also, Figure 25.
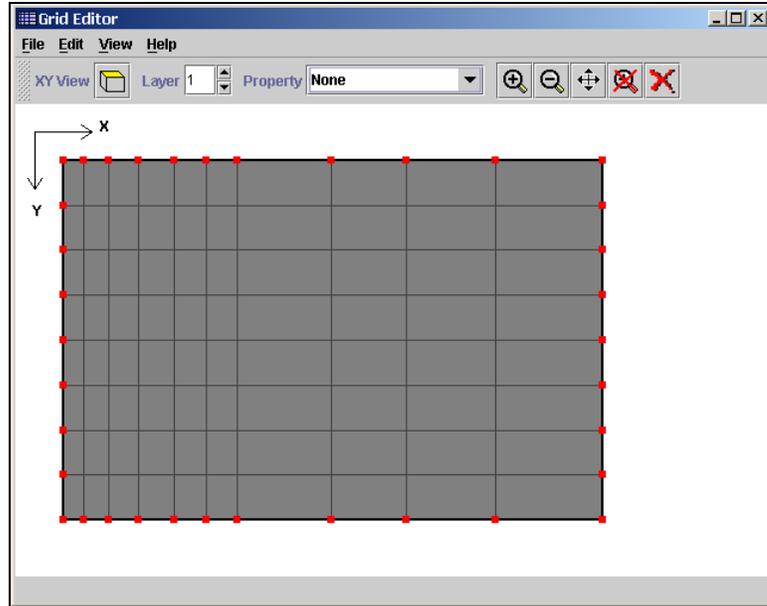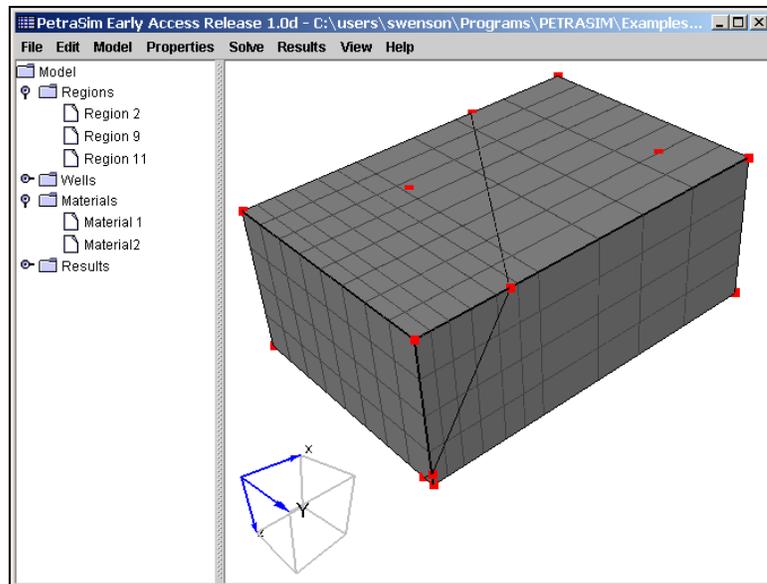
Figure 24: Grid after dragging gridlines

Figure 25: 3D display after dragging gridlines

4.4   Deleting a Gridline

Gridlines can be deleted by selecting a gridline with a single click (or right click) and then selecting the Grid Editor menu *Edit/Delete Selected Line*.

4.5   Inserting a Gridline

Gridlines can be inserted by the menus *Edit/Add Vertical Line* or *Edit/Add Horizontal Line*.

4.6   Closing the Grid Editor

Close the Grid Editor by selecting *File/Close* (or the X in the upper right corner of the window).

5. Importing Legacy InterSim Files

On startup (or after selecting *File/New*) the user can import existing InterSim files using *File/Read InterSim File*. A grid will be created corresponding to the InterSim file. The user can then edit cell properties using the Grid Editor.

The user can download the file named "example.is" and open it in PetraSim. The model will be as shown in Figure 26. Opening the Grid Editor and displaying the temperatures gives the plot shown in Figure 27.
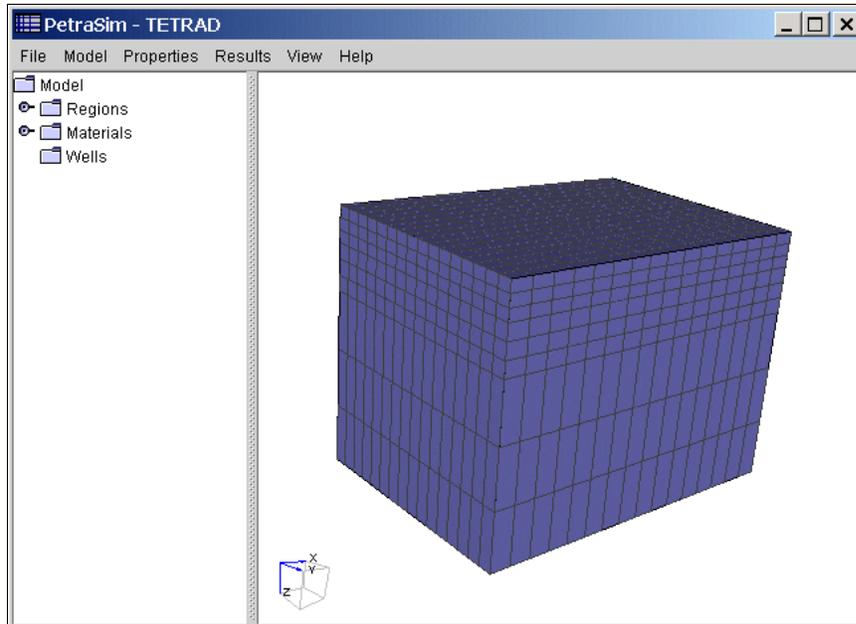

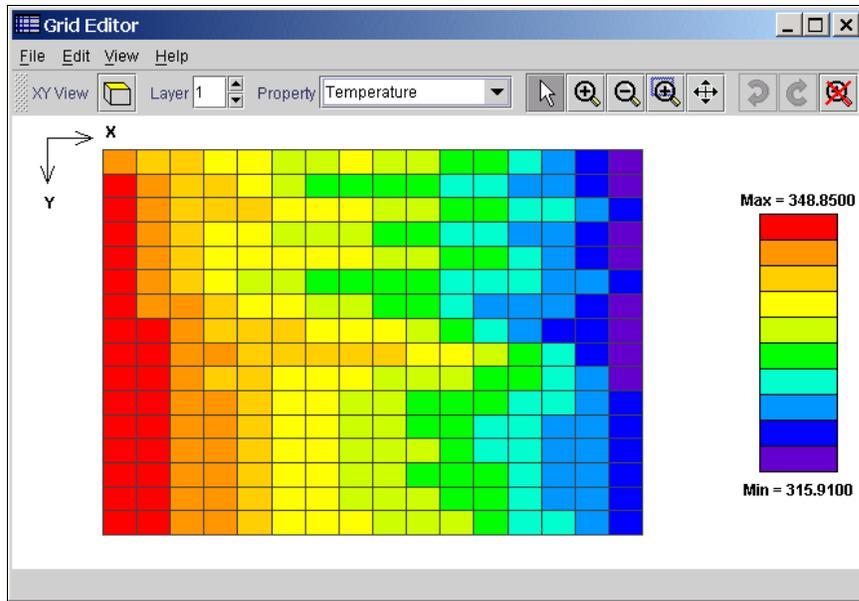Figure 26: Display of example InterSim file

Figure 27: Display of temperatures in Grid Editor

6. Plotting Legacy GridView and PLT Files

6.1 Plotting GridView Data

The user can also plot existing Gridview files. The user can download the "example.gv" file and open it using ***Results/3D Contours…*** . A window will open in which the user can select the variable to display, the time to display, and control other parameters such as the plot range. A plot of temperatures is shown in Figure 28. To make this plot, the temperature range was specified as shown in Figure 29. Slice planes can be defined on which contours will be drawn. In this example, the slice planes were as shown in Figure 30.
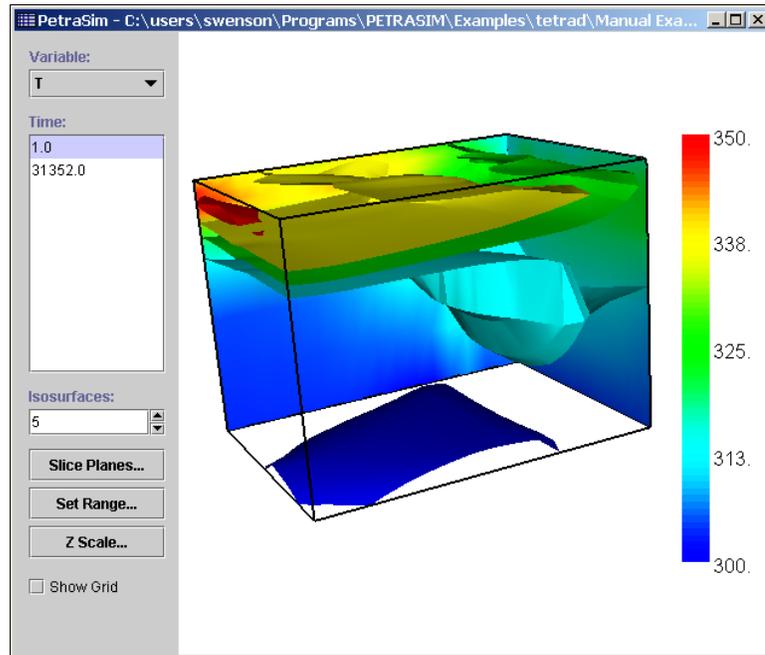

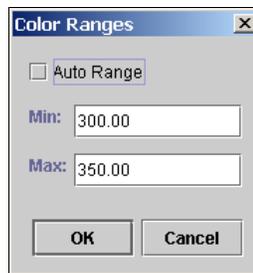Figure 28: Example plot of GridView

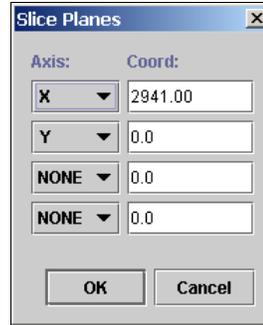Figure 29: Manually specify auto range from 300 to 350

Figure 30: Specify two slice planes

6.2   Plotting Time History Data

Time history plots of data stored in the FPLOT (*.plt) format can also be made. Using the "example.plt" file that can be downloaded from the web, make these plots by selecting **Results/Cell History Plots…** . The result is shown in Figure 31. The user can then write the plot data in a format for opening in a spreadsheet by selecting **File/Export Data…** .
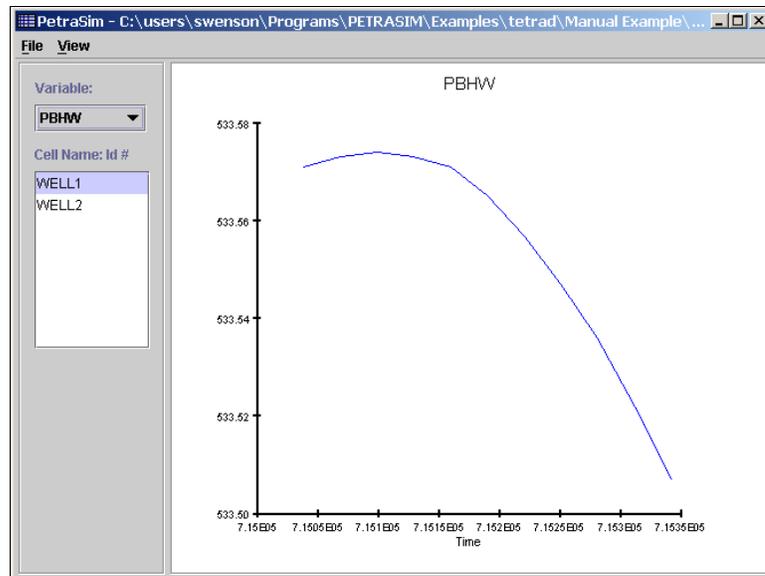

Figure 31: Example time history plot

7. Acknowledgments