THUNDERHEAD
ENGINEERING

403 Poyntz Avenue, Suite B
Manhattan, KS 66502
USA
+1.785.770.8511
www.thunderheadeng.com

PetraSim Examples:

3D Five-Spot Geothermal Injection and Production

2010

Table of Contents

3D Five Spot Geothermal Production and Injection (EOS1)

This example is another variation of the five-spot example in the TOUGH2 User's Manual. In this analysis, we use a 3D mesh and model injection and production wells. An image is added to the model to illustrate the surface of the earth. The model represents 1/2 of the repeating five-spot pattern.

Create a New Model

This example will use EOS1, the equation of state module that models multi-phase water and tracer. The extent of the model will be 1000x500x305 meters. This information can be entered using the **New** dialog.

To create the model:

1. On the **File** menu, click **New**
2. In the **Simulator Mode** list, select **TOUGH2**
3. In the **Equation of State (EOS)** list, select **EOS1**
4. In the **X Min** box, type *-500*
5. In the **X Max** box, type *500*
6. In the **Y Max** box, type *500*
7. In the **Z Max** box, type *305*
8. Click **OK** to close the dialog and create the model

Edit Layers

Before creating the mesh, we must first define the model layers. Layers in PetraSim define depth-based boundaries that can provide a convenient way to manage multiple rock layers. When the mesh is generated, the layer boundary will become the boundary between cells and each layer can be divided into an arbitrary number of internal divisions – these internal divisions will also become cell boundaries.

By default, the model is created with a single layer. To set the default layer:

1. In the **Tree View**, expand the **Layers** node and click **Default**
2. On the **Edit** menu, click **Properties...**
3. In the **Cells** box, type *6*
4. Click **OK** to save changes to the default layer

Add Wells

To add wells:

1. On the **Model** menu, click **Add Well…**
2. In the **Name** box, type *Inject*
3. In the **Ordered Well Coordinates** table, type these values:

X	Y	Z
0.0	0.0	310.0
0.0	0.0	0.0

4. Click **OK** to create the well
5. Repeat this for the production well in the upper left corner. Use the name *ProdUL* and the following coordinates:

X	Y	Z
-500.0	500.0	310.0
-500.0	500.0	0.0

6. Repeat this for the production well in the upper right corner. Use the name *ProdUR* and the following coordinates:

X	Y	Z
500.0	500.0	310.0
500.0	500.0	0.0

Create Mesh

To create a polygonal (Voronoi) mesh:

1. On the **Model** menu, click **Create Mesh**
2. For **Mesh Type**, select *Polygonal*
3. In the **Maximum Cell Area** box, type *2500*
4. In the **Max Area Near Wells** box, type *250*
5. Click **OK** to create the mesh

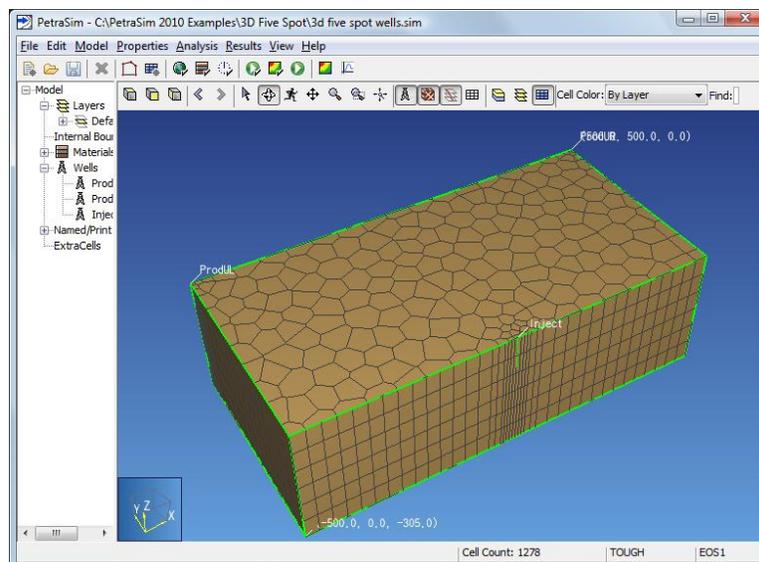Your mesh should be refined near the wells, Figure 1.

Figure 1: 3D Five Spot model

Edit Global Properties

Global properties apply to the entire model. In this example, the only item to change is the analysis name. To set the simulation name:

1. On the **Properties** menu, click **Global Properties**
2. In the **Name** box, type *3D Five Spot Production and Injection*
3. Click **OK**

Edit Materials

This simulation will require only one material. To specify material properties:

1. On the **Properties** menu, click **Edit Materials...**
2. In the **Name** box, type *POMED*
3. In the **Density** box, type *2650.0*
4. In the **Porosity** box, type *0.01*
5. In the two **Permeability** boxes (XY and Z), type *6e-15*

By default, materials use linear relative permeability. However, for this simulation we will use Corey's Curves. To specify the relative permeability for this material:

1. Click **Additional Material Data...**
2. In the **Relative Permeability** list, click **Corey's Curves**
3. Click **OK**

Click **OK** to save changes and exit the **Material Data** dialog.

Initial Conditions

When specifying parameters such as initial conditions, PetraSim uses a tiered system that allows cells to inherit values from the containing region, regions from the containing layer, and layers from the global defaults. Once we set the default initial conditions, these values will be used by all cells except those where we have explicitly set values.

Correct specification of initial conditions is essential for simulation convergence. The initial conditions must be physically meaningful. Often this requires an initial state analysis in which a model is run to obtain initial equilibrium conditions before the analysis of interest (geothermal production, VOC spill, etc.) is run.

However, in 2D models with no pressure gradient (such as this) this process is simplified somewhat.

To set the global initial conditions:

1. On the **Properties** menu, click **Initial Conditions...**
2. In the **EOS1** list, select **Two-Phase (T, Sg)**
3. In the **Temperature** box, type *300*
4. In the **Gas Saturation** box, type *0.01*

5. Click **OK** to save changes and exit the dialog

Selecting a two-phase initial state with very little gas saturation is an appropriate simplification for this geothermal analysis. Since we have only one element in the vertical direction, we have selected a temperature of 300 °C as a typical for this reservoir. The pressure will be calculated by the equation of state to correspond to the small gas saturation. Having all cells in a two-phase state will also reduce the number of state transitions (two-phase to single-phase or single-phase to two-phase) that make convergence more difficult.

Edit Well Flow Rates

To define the injection rate:

1. In the **Tree View**, right-click on the **Inject** well and click **Properties…**
2. On the **Geometry** tab, in the **Top Completion Depth** box, type *305.0* and in the **Bottom Completion Depth** box, type *100.0*. This means that only the range 100≤Z≤305 is open for injection.
3. On the **Flow** tab, in the **Injection** section, select the **Water/Steam** check box
4. In the **Rate** box, type *15.0*
5. In the **Enthalpy** box, type *50000* (corresponding to about 6 °C)
6. Click the **Print Options** tab
7. Click to select the **Print Time Dependent Flow and Generation Data** check box
8. Click **OK** to save changes and close the dialog

Define the production wells using the Productivity Index:

1. In the **Tree View**, right-click on the **ProdUL** well and click **Properties…**
2. On the **Geometry** tab, in the **Top Completion Depth** box, type *100.0* and in the **Bottom Completion Depth** box, type *0.0*. This means that only the range 0≤Z≤100 is open for production.
3. On the **Flow** tab, in the **Production** section, select the **Well on Deliv** check box
4. In the **Productivity Index** box, type *2.0E-12* (based on calculation described in the TOUGH2 User Manual)
5. In the **Pressure** box, type *8.6E6*
6. For the **Gradient**, select the **Well Model** check box (this means that TOUGH2 will use the density to calculate the pressure gradient in the well above the specified bottom pressure.
7. Click the **Print Options** tab and click to select the **Print Time Dependent Flow and Generation Data** check box
8. Click **OK** to save changes and close the dialog

Repeat with the same input for the well in the upper right corner, using the name **ProdUR**.

Edit Solution Controls

Parameters relating to the solver and time stepping can be found in the Solution Controls dialog.

To specify the simulation end time:

1. On the **Analysis** menu, click **Solution Controls**
2. In the **End Time** list, click **User Defined** and type *36.5 years*
3. Click **OK**

Edit Output Controls

By default, the simulation will print output every 100 time steps. For this simulation, we will specify output every 5 time steps.

To specify the output frequency:

1. On the **Analysis** menu, click **Output Controls**
2. In the **Print and Plot Every # Steps** box, type *5*
3. Click **OK**

Save and Run

The input is complete and you can run the simulation. Save your model and run the simulation:

1. On the **Analysis** menu, click **Run TOUGH2**

The **Simulation Complete** dialog will notify you when the end time has been reached. Click **OK** to dismiss the notification and click **Close** to exit the **Running TOUGH2** dialog.

View 3D Results

To open the **3D Results** dialog:

1. On the **Results** menu, click **3D Results**

By default, the display will show isosurfaces corresponding to pressure for the first output step.

To show temperature isosurfaces for the last time step:

1. In the **Scalar** list, click **T**
2. In the **Time(s)** list, click the last entry (t = **1.15183E9**)

To show scalar data on a slice plane:

1. Click **Slice Planes...**
2. In the **Axis** list, click **Z** and in the **Coord** box, type *1*
3. In the **Axis** list, click **X** and in the **Coord** box, type *0*
4. In the **Axis** list, click **Y** and in the **Coord** box, type *500*
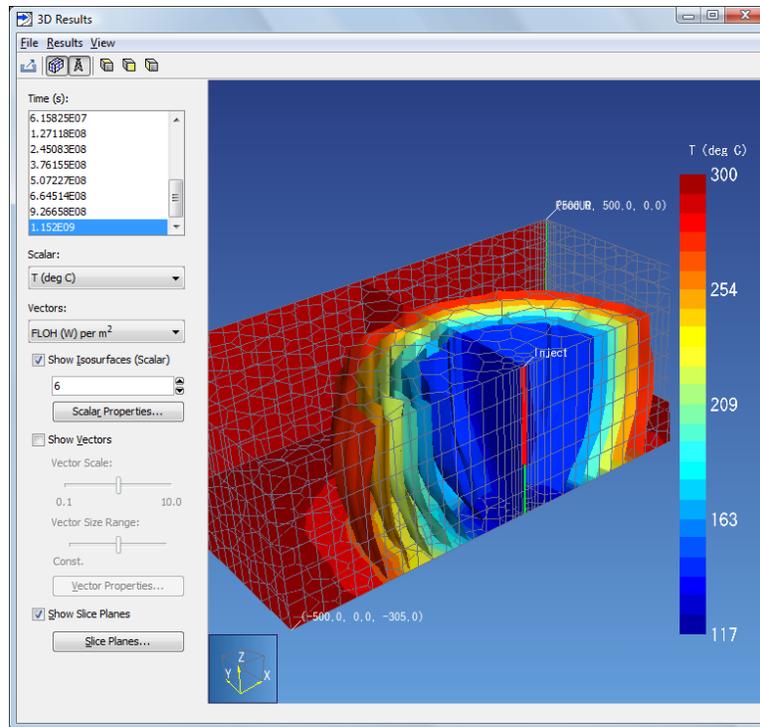5. Click **Close**

The resulting visualization is shown below:

Figure 2: 3D Five Spot temperature isosurfaces at end of solution

When finished, you can close the **3D Results** dialog.
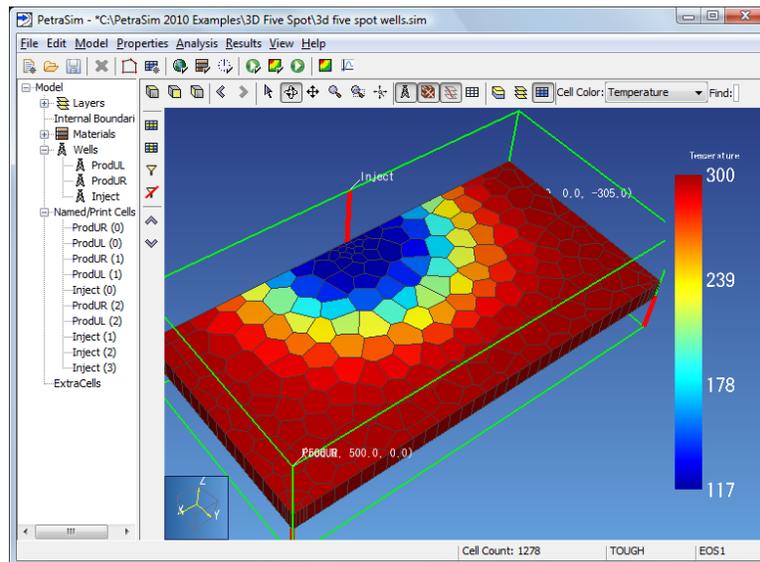
Importing the SAVE data into the model makes it possible to display results by cell:

Figure 3: Temperature display by cell