THUNDERHEAD
ENGINEERING

Tutorial: Using Wells and Polygonal Mesh Refinement

PetraSim 5

Introduction

Wells are a mesh-independent way to specify source/sink information in PetraSim. This tutorial will show how to use wells in PetraSim. In addition, we will demonstrate a well-based refinement feature of PetraSim's polygonal mesh and well-specific time history plots.
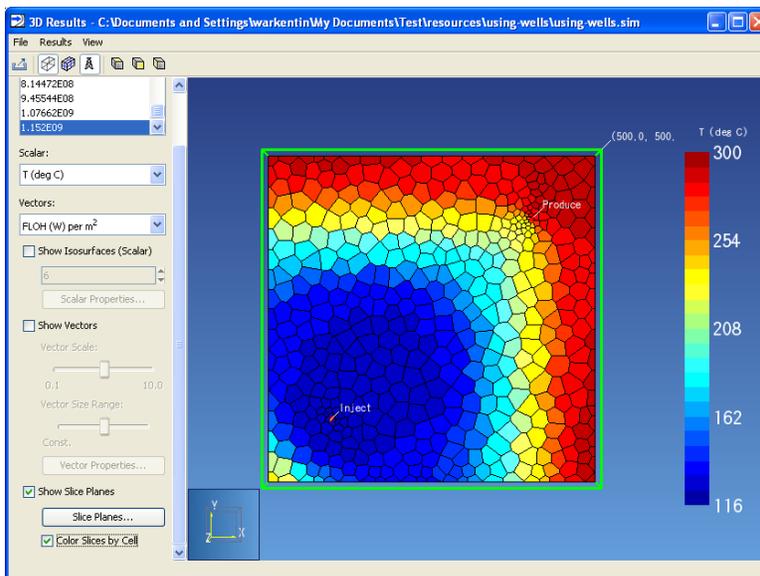

Figure 1: 3D Results view showing the end result of this tutorial.

The design of this problem is similar to the Five Spot TOUGH2 example problem. In this problem cold water (6 °C) is injected into a field containing hot water (300 °C). A production element removes the water from the opposite corner of the model. Here, we will use wells instead of elements for injection and production. The locations of the production and removal sites will also be moved inward, breaking the symmetry of the original problem, but providing a more interesting demonstration of PetraSim's mesh refinement.

Load Initial Configuration

This tutorial begins after some initial configuration steps. An archive of example and tutorial files is provided on the PetraSim support web site. The folder corresponding to this problem is named "using-wells".

To load the initial configuration of this problem:

1. Start PetraSim
2. Load the file *using-wells\using-wells.sim*

This will initialize the model dimensions, material properties, global initial conditions, and some additional simulation parameters.

Add Wells

In PetraSim, adding wells is a two step process. First the wells are inserted at the desired location, then well parameters can be edited.

To add the injection well to the model:

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

X	Y	Z
100.0	100.0	100.0
100.0	100.0	0.0

4. Click **OK** to create the well.

To edit flow parameters for the injection well:

1. In the **Tree View**, right-click the **Inject** well and click **Properties...**
2. On the **Flow** tab, in the **Injection** section, select the **Water/Steam** check box
3. In the **Rate** box, type *7.5*
4. In the **Enthalpy** box, type *5e5* (corresponding to about 6 °C)
5. On the **Print Options** tab, select **Print Time Dependent Flow and Generation (BC) Data**
6. Click **OK** to save changes and exit

To add the production well to the model:

1. On the **Model** menu, click **Add Well...**
2. In the **Name** box, type *Produce*
3. In the **Ordered Well Coordinates** table, enter the following values:

X	Y	Z
400.0	400.0	100.0
400.0	400.0	0.0

4. Click **OK** to create the well.

This example will use TOUGH2 "well on deliverability" feature which allows production to occur against a specified bottomhole pressure. To use this feature, we must calculate a productivity index (PI). The equation given in the TOUGH2 User's Guide for calculating PI (assuming steady radial flow) is shown below.

$$PI_l = \frac{2\pi(k\Delta Z_l))}{ln(r_e/r_w) + s + 0.5}$$

Using the following variable assignments:

Permeability (k) = 6E-15 m^2
Layer Thickness (dZ_l) = 100 m
Element Radius (r_e) / Well Radius (r_w) = 1.0
Skin Factor ($s)$ = 6.0

We can calculate a permeability index (PI_l) of 5.8E-13.

To edit flow parameters for the production well:

1. In the **Tree View**, right-click the **Produce** well and click **Properties...**
2. On the **Flow** tab, in the **Production** section, select **Well On Deliv.**
3. In the **Productivity Index** box, type *5.8e-13*
4. In the **Pressure** box, type *8.6e6*
5. In the **Gradient** box, select **Well Model**
7. On the **Print Options** tab, select **Print Time Dependent Flow and Generation (BC) Data**
8. Click **OK** to save changes and exit

Create Mesh

This model will use a polygonal mesh that preserves the Voronoi property required by the TOUGH2 solver. PetraSim can also automatically refine this mesh near wells.

To create the mesh for this model:

1. On the **Model** menu, click **Create Mesh**
2. In the **Mesh Type** box, select **Polygonal**
3. In the **Maximum Cell Area** box, type *500*
4. In the **Max Area near Wells** box, type *25*
5. Click **OK** to create the mesh

The values entered in the **Max Area near Wells** box will control the size of the cells near any wells in the model. The area measurement is the 2-dimensional top surface of the cell. The **Maximum Cell Area** will determine how large cells can become when nothing nearby is causing refinement. The resulting mesh is shown in Figure 2.
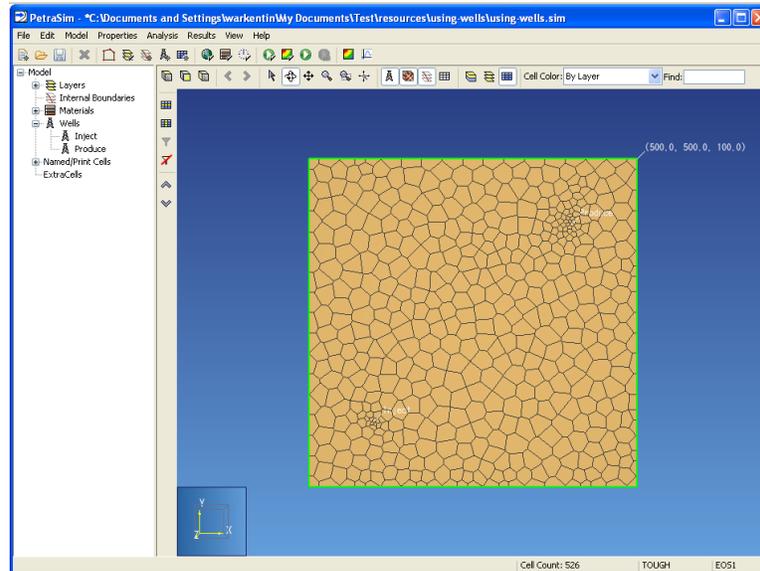
Figure 2: The simulation mesh with refinement at wells.

Run Simulation

The problem setup is now complete. To run the simulation: On the **Analysis** menu, click **Run TOUGH2**.

View 3D Results

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

Wells have no special visualization options in the 3D results window. To regenerate the image shown at the beginning of this tutorial, use the following steps:

1. In the **Time(s)** list, select the last time step
2. In the **Scalar** box, select **T (deg C)**
3. Click to disable (un-check) the **Show Isosurfaces (Scalar)** option
4. Click **Slice Planes...**
5. In the **Axis box**, click **Z**
6. In the **Coord Box**, type *50*
7. Click **Close** to create the slice plane
8. Click to enable (check) the **Color Slices by Cell** option

When finished, close the **3D Results** window.

View Well Plots

When wells are present in the model and have been marked for additional output, PetraSim can show time history plots of well flow rates. These values are accumulated across cells running the length of the well.

To view well plots: On the **Results** menu, click **Well Plots**

The Flow Rate (kg/s) for the Inject well will be shown. Note that the injection rate is a constant 7.5 kg/s. You can verify that the production well matched this rate by clicking **Produce** in the Well Name list.
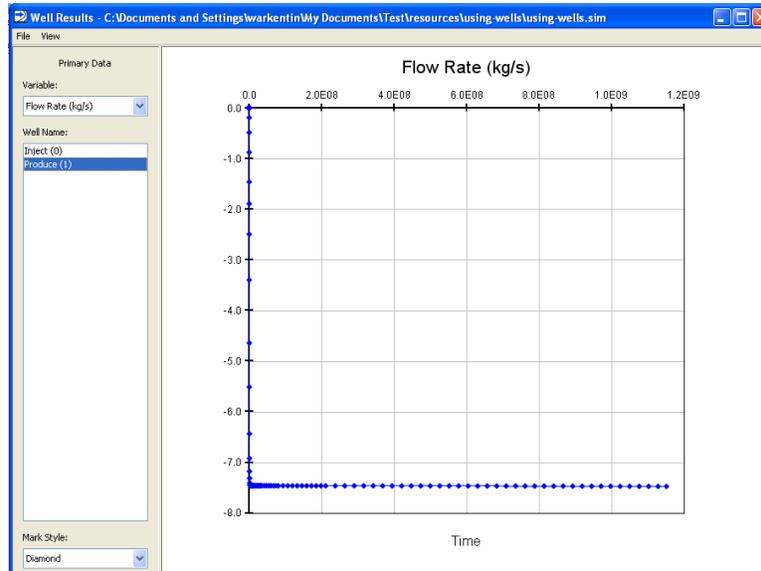

Figure 3: Mass flow rate for the production well.