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

TOUGH2 Example:

Five-Spot Geothermal Production and Injection

PetraSim 2016.1

Table of Contents

Disclaimer

Thunderhead Engineering makes no warranty, expressed or implied, to users of PetraSim, and accepts no responsibility for its use. Users of PetraSim assume sole responsibility under Federal law for determining the appropriateness of its use in any particular application; for any conclusions drawn from the results of its use; and for any actions taken or not taken as a result of analyses performed using these tools.

Users are warned that PetraSim is intended for use only by those competent in the field of multi-phase, multi-component fluid flow in porous and fractured media. PetraSim is intended only to supplement the informed judgment of the qualified user. The software package is a computer model that may or may not have predictive capability when applied to a specific set of factual circumstances. Lack of accurate predictions by the model could lead to erroneous conclusions. All results should be evaluated by an informed user.

Throughout this document, the mention of computer hardware or commercial software does not constitute endorsement by Thunderhead Engineering, nor does it indicate that the products are necessarily those best suited for the intended purpose.

Five-Spot Geothermal Production and Injection

This example is *Problem 4 - Five Spot Geothermal Production and Injection (EOS1, EOS2)* described in the TOUGH2 User's Manual (1). It simulates a five-spot pattern of injection and production from a geothermal reservoir. Only one quarter of the reservoir is included, since the solution is symmetric, as shown in Figure 1. The mesh uses cells of uniform 50 m x 50 m size.

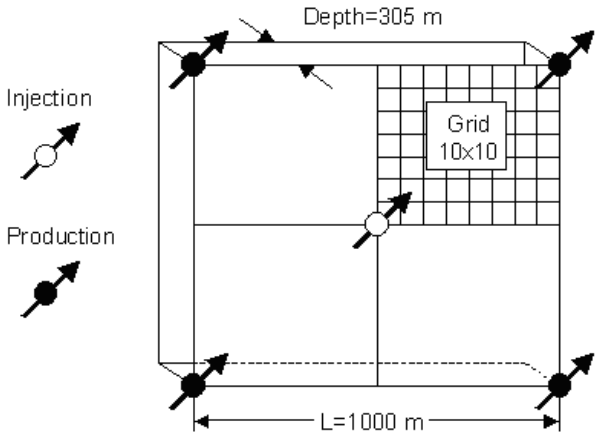

Figure 1: Diagram of reservoir for Five-Spot Injection

We will solve this example using a porous media assumption for the formation.

Create a New Model

This example will use EOS1, the equation of state module that models multi-phase water and tracer. The dimensions of the model will be 500x500x305 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 **TOUGH**.
3. In the **Equation of State (EOS)** list, select **EOS1.**
4. In the **X Min** box, type **0.**
5. In the **X Max** box, type **500.**
6. In the **Y Min** box, type **0.**
7. In the **Y Max** box, type **500.**
8. In the **Z Min** box, type **0.**
9. In the **Z Max** box, type **305.**
10. Click **OK** to close the dialog and create the model.

Default Units

By default, PetraSim uses the TOUGH2 metric unit system (for example, the TOUGH2 system specifies permeability in units of m^2). We will use the default system. You can select other units for the input parameters. If you do this, PetraSim will remember your selection the next time you open a model:

1. On the **View** menu, click **Unit System.**
2. Select the units you wish to use for each parameter.
3. To reset to the default TOUGH2 units, click **Reset**.

Note: You can always enter values in any of the optional units supported by PetraSim. For example, you can type **10 years** for time or **5 mD** for permeability. PetraSim will convert to the units being used in the model.

Save the Model

It is good practice to save the model initially. Some of the files created by TOUGH have the same name for every simulation, so it is good practice to create a separate folder for simulation.

To save your model:

1. On the **File** menu, click **Save.**
2. Create a folder named Five Spot, then type the file name as **five spot** and click **Save.**

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 and a single cell through the layer. This matches our desired input, so no changes need to be made to the layers.

Create Mesh

To create a 10 x 10 regular mesh:

1. On the **Model** menu, click **Create Mesh.**
2. In the **X Cells** box, type *10.*
3. In the **Y Cells** box, type *10.*
4. Click **OK** to create the mesh (Figure 2) **.**

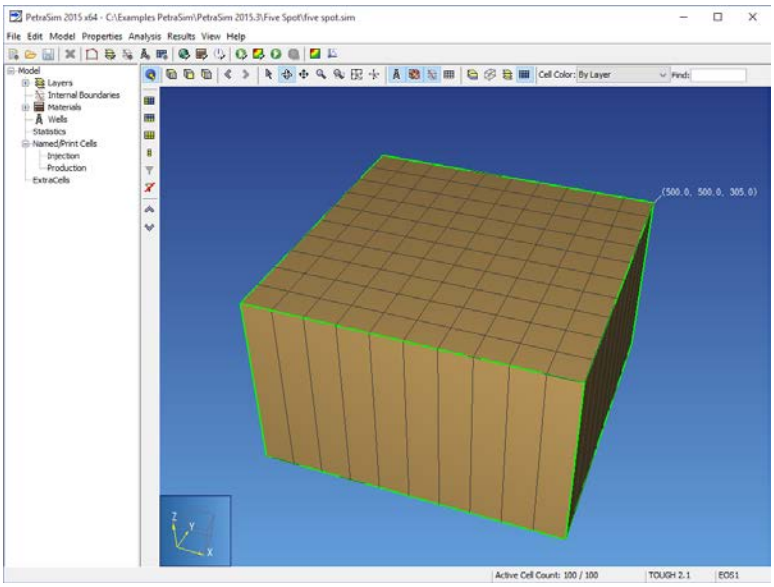

Figure 2: After creating the mesh

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:

4. On the **Properties** menu, click **Global Properties.**
5. In the **Name** box, type **Five Spot Production and Injection.**
6. 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 all three **Permeability** boxes (X, Y, and Z), type **6e-15.**

For this simulation we will define relative permeability using 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.** The default input parameters are satisfactory.
3. Click **OK.**
4. Click **OK** to save changes and exit the **Edit Materials** 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 typical for this reservoir. TOUGH will calculate the pressure using the equation of state to correspond to the small gas saturation. In some cases, having a small gas component will speed convergence by reducing the number of state transitions (two-phase to single-phase or single-phase to two-phase) that make convergence more difficult.

Edit Injection and Production Cells

The total injection/production from each five spot pattern is 30 kg/sec. Since we are using quarter symmetry, the production and injection mass flow rates will be 7.5 kg/sec. In this example we edit specific cells to specify injection and production parameters.

When editing individual cells, it is sometimes convenient to use the top view:

1. In the **View** menu, click **Top View**.

To define the injection cell:

1. Right-click the lower left cell (ID=001), in the popup menu click **Edit Cells.**
2. In the **Cell Name** box, type *Injection.*
3. Click the **Sources/Sinks** tab.
4. Under **Injection**, select the **Water/Steam** check box**.**
5. In the **Rate** box, type **7.5.**
6. In the **Enthalpy** box, type **5E5.**

7. Click the **Print Options** tab.
8. Click **Print Time Dependent Flow and Generation Data.**
9. Click **OK** to save changes and close the dialog.

To define the production cell:

1. Right-click the upper right cell (ID=100), in the popup menu click **Edit Cells.**
2. In the **Cell Name** box, type **Production.**
3. Click the **Sources/Sinks** tab.
4. Under **Production**, click **Mass Out.**
5. In the **Rate** box, type **7.5.**
6. Click the **Print Options** tab.
7. Click **Print Time Dependent Flow and Generation Data.**
8. Click **OK** to save changes and close the dialog.

The injection and production cells now appear in the tree at the left under the **Named/Print Cells** heading. Any cell that has been selected for time dependent output or explicitly named will appear in this list.

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. The simulator will generate numerous output files (e.g. FOFT, mesh.csv, etc.). As previously noted, since these files have the same name for every simulation, it is usually a good idea to create a folder specifically for each model.

To save your model:

1. On the **File** menu, click **Save.**

To run the simulation:

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

You should see a graph showing the increase in time step size as the simulation converges. If there are any problems, you can view the log. Output that has been identified as errors will appear in red.

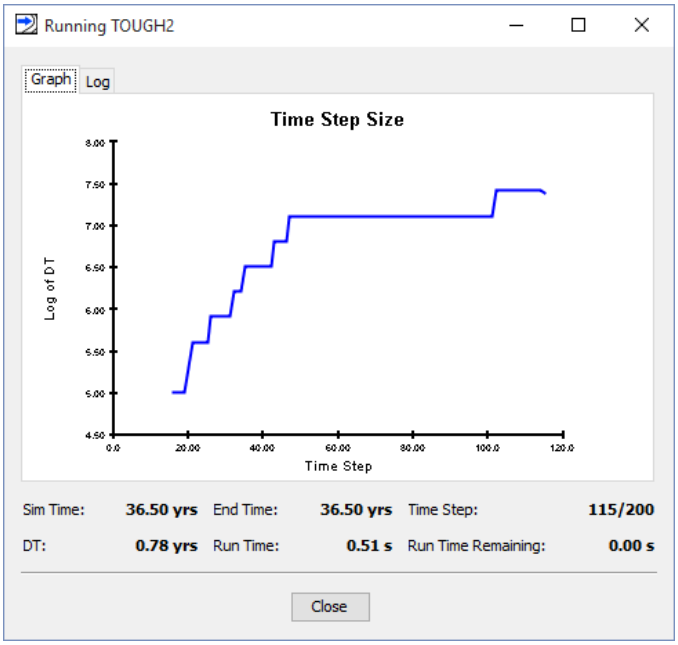

Figure 3: The Running TOUGH2 dialog shows a graph of time step size.

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 = **1.15183E09**).

To show scalar data on a slice plane:

1. Click **Slice Planes.**
2. In the **Axis** list, click **Z.**

3. In the **Coord** box, type **100.**
4. Click **Close.**

To remove isosurfaces and show only slice data:

1. Clear the **Show Isosurfaces** check box**.**

The resulting visualization is shown below. Note the temperature drop at the production well. This is a result of specifying a production flow rate, which results in a lower pressure than would physically occur in a real well. This will be discussed further in the next example.

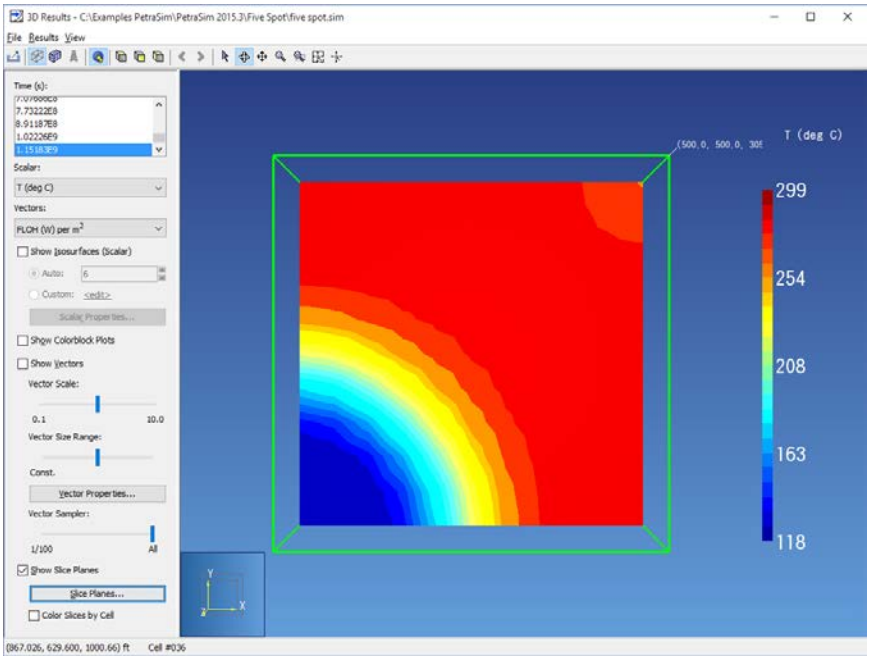

Figure 4: Temperature contours at end of solution

When finished, you can close the **3D Results** dialog.

View Time History Plots

To view time history plots:

1. On the PetraSim **Results** menu, click **Cell History Plots.**
2. In the **Variable** list, click **T (deg C)**.
3. In the **Cell Name** list, click **Injection.**

The resulting plot is shown below.

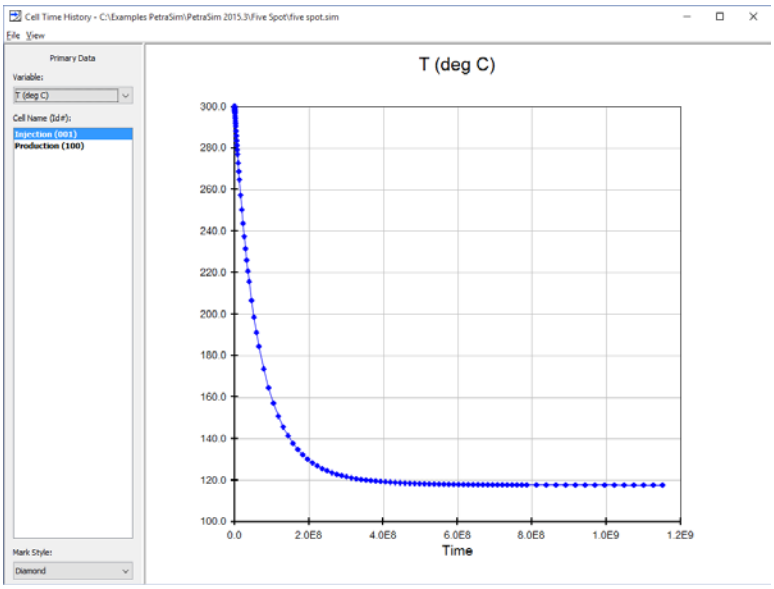

Figure 5: Temperature history of injection cell.

When finished, you can close the **Cell History** dialog.

References

1. **Falta, Ronald, et al., et al.** *T2VOC User's Guide.* Berkeley, CA, USA : Earth Sciences Division, Lawrence Berkeley National Laboratory, March 1995. LBNL-36400.

1. **Pruess, Karsten, Oldenburg, Curt and Moridis, George.** *TOUGH2 User's Guide, Version 2.0.* Berkeley, CA, USA : Earth Sciences Division, Lawrence Berkeley National Laboratory, November 1999. LBNL-43134.

2. **Falta, Ronald, et al.** *T2VOC User's Guide.* Berkeley, CA, USA : Earth Sciences Division, Lawrence Berkeley National Laboratory, March 1995. LBNL-36400.