



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 5

Table of Contents

Acknowledgements	iv
Five-Spot Geothermal Production and Injection	1
Create a New Model	1
Edit Layers	1
Create Mesh.....	2
Edit Global Properties	2
Edit Materials.....	2
Initial Conditions	3
Edit Injection and Production Cells.....	3
Edit Solution Controls	4
Edit Output Controls	4
Save and Run.....	4
View 3D Results.....	5
View Time History Plots	6
References	8

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.

Acknowledgements

We thank Karsten Pruess, Tianfu Xu, George Moridis, Michael Kowalsky, Curt Oldenburg, and Stefan Finsterle in the Earth Sciences Division of Lawrence Berkeley National Laboratory for their gracious responses to our many questions. We also thank Ron Falta at Clemson University and Alfredo Battistelli at Aquater S.p.A., Italy, for their help with T2VOC and TMVOC. Without TOUGH2, T2VOC, TOUGHREACT, and TOUGH-Fx/HYDRATE, PetraSim would not exist.

In preparing this manual, we have liberally used descriptions from the user manuals for the TOUGH family of codes. Links to download the TOUGH manuals are given at <http://www.petrasim.com>. More information about the TOUGH family of codes can be found at: <http://www-esd.lbl.gov/TOUGH2/>. Printed copies of the user manuals may be obtained from Karsten Pruess at <K_Pruess@lbl.gov>.

The original development of PetraSim was funded by a Small Business Innovative Research grant from the U.S. Department of Energy. Additional funding was provided by a private consortium for the TOUGHREACT version and by the U.S. Department of Energy NETL for the TOUGH-Fx/HYDRATE version.

We most sincerely thank our users for their feedback and support.

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.

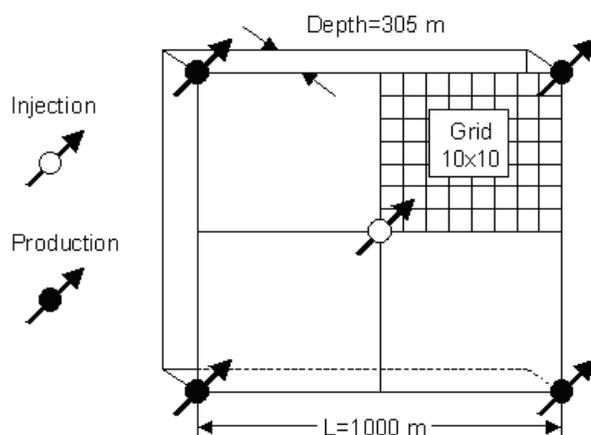


Figure 1: Diagram of reservoir for Five-Spot Injection

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 **TOUGH2**
3. In the **Equation of State (EOS)** list, select **EOS1**
4. In the **X Max** box, type *500*
5. In the **Y Max** box, type *500*
6. In the **Z Max** box, type *305*
7. 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. Because this example contains only a single cell in the Z-direction, the layer must be edited to contain only a single cell.

To set the default layer to use a single vertical cell:

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 *1*
4. Click **OK** to save changes to the default layer

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*
1. Click **OK** to create 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:

1. On the **Properties** menu, click **Global Properties**
2. In the **Name** box, type *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 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**
3. Click **OK**

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 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 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. To switch to the top view: in the **View** menu, click **Top View**.

To define the injection cell:

1. Right-click the lower left cell (ID=1), 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 *50000*

-
7. Click the **Print Options** tab
 8. Click to select all time dependent output check boxes
 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**, select the **Mass Out** check box
5. In the **Rate** box, type 7.5
6. Click the **Print Options** tab
7. Click to select both time dependent output check boxes
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.). Since these files have the same name for every simulation, it is usually a good idea to create a folder specifically for a particular model.

To save your model:

1. On the **File** menu, click **Save**
2. Select a location, then 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.

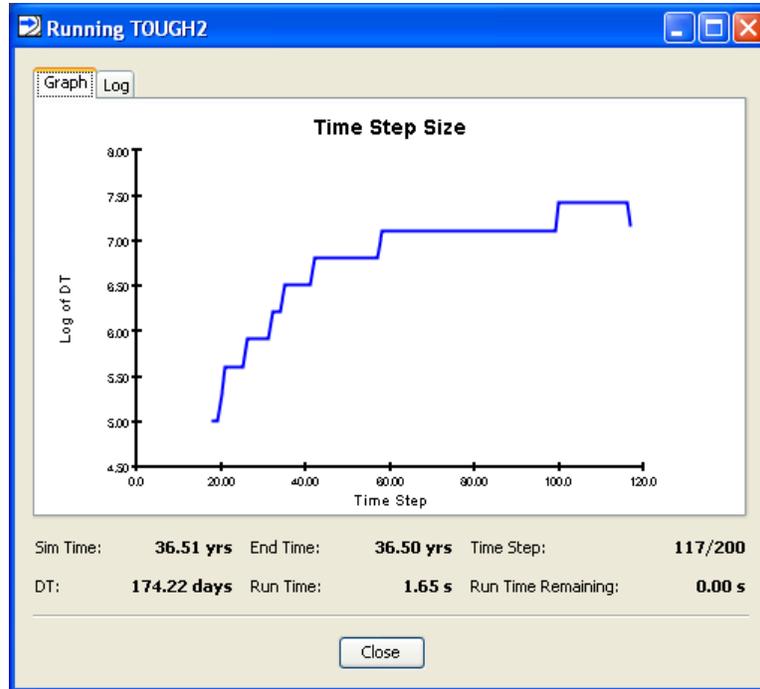


Figure 2: The Running TOUGH2 dialog shows a graph of time step size.

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 = **1.152E09**)

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. Click to 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.

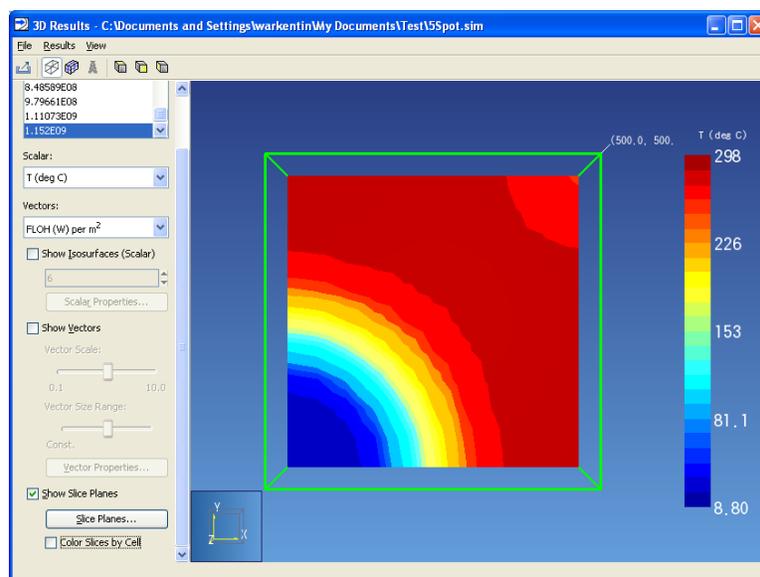


Figure 3: Temperature contours at end of solution

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.

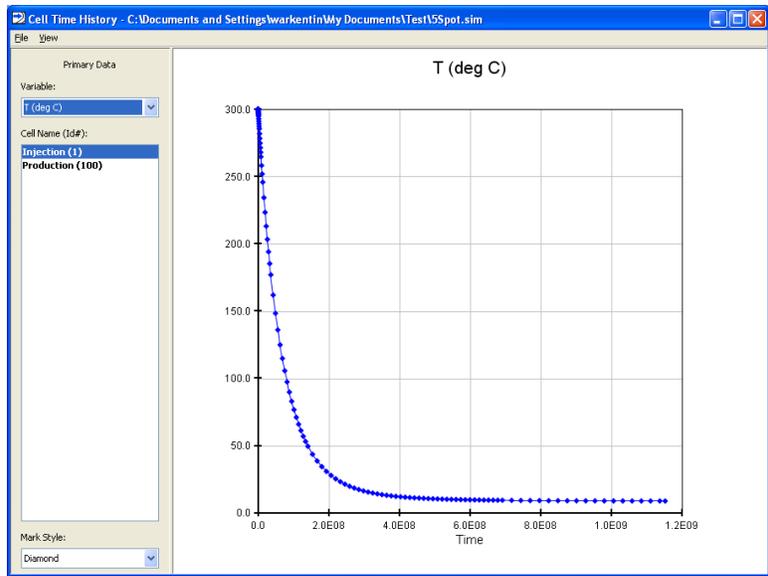


Figure 4: Temperature history of injection cell.

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

References

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.