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

TOUGH2 Example:

Production from a Geothermal Reservoir (EWASG)

PetraSim 5

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.

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.

Production from a Geothermal Reservoir (EWASG)

Description

This example is Problem 12 - Production from a Geothermal Reservoir with Hypersaline Brine and CO2 (EWASG) described in the TOUGH2 User's Manual (1). This problem examines production from a hypothetical geothermal reservoir with high salinity and CO2. Fluid withdrawal causes pressure to drop near the production well. Boiling of reservoir fluid gives rise to dilution of CO2 in the gas phase and to increased concentrations of dissolved NaCl, which begins to precipitate when the aqueous solubility limit is reached. As the boiling front receded from the well, solid precipitate fills approximately 10% of the original void space, causing permeability to decline to approximately 28% of its original value.

The mesh uses a 1D radial geometry.

Create an EWASG Model

We will first create a new model using the EWASG EOS.

From the **File** menu, select **New…** to open the **New Model** dialog.

1. For the **Simulator Mode**, choose **TOUGH2**.
2. For the **Equation of State (EOS)**, choose **EWASG**.
3. For the **Model Bounds**, enter the values from Table 1.

Table 1: Model boundary dimensions

Axis	Min (m)	Max (m)
X	5.0	1000.0
Y	0.0	1.0
Z	-500.0	0.0

Click **OK** to close the **New Model** dialog and create the model.

Specify the Solution Mesh

The solution mesh must be specified in two steps. First the Z divisions must be specified per layer. Then the mesh can be created using the **Create Mesh** dialog.

Specify Z Divisions

We must first specify one Z division for the default layer. To do so, we will open the **Edit Layers** dialog. On the **Model** menu, select **Edit Layers…**.

1. In the layers list in the left-hand pane, select the **Default** layer.
2. In the right-hand pane, select **Regular** for **Dz**.
3. In the **Cells** box, type *1*.
4. In the **Factor** box, type *1.0*.

Click **OK** to apply the changes and close the **Edit Layers** dialog.

Create the Mesh

We will now create the radial solution mesh using the **Create Mesh** dialog. To open the dialog, on the **Model** menu, select **Create Mesh…**.

1. For the **Mesh Type**, select **Radial**.
2. For the **Divisions**, select **Regular**.
3. In the **Radial Cells** box, type *100*.
4. In the **Factor** box, type *1.03705*.

Click **OK** to create the mesh.

We have selected Radial Mesh as the mesh type. This means that a 2D cylindrical RZ mesh will be created in which X corresponds to cylindrical R, Z corresponds to cylindrical Z, and the Y coordinate corresponds to cylindrical Theta (which is not used). 100 cells extend in the R direction, with an X Factor of 1.03705. This means that each cell is 1.03705 times the size of the previous cell in the R direction. These were chosen to give a first cell length of 2.0 m, which is the value used in the TOUGH2 example. See the PetraSim User Manual for instructions on how to calculate the appropriate factor.

The resulting mesh is displayed in Figure 1. Click the **Front View** button on the toolbar or rotate the model until the front is visible.

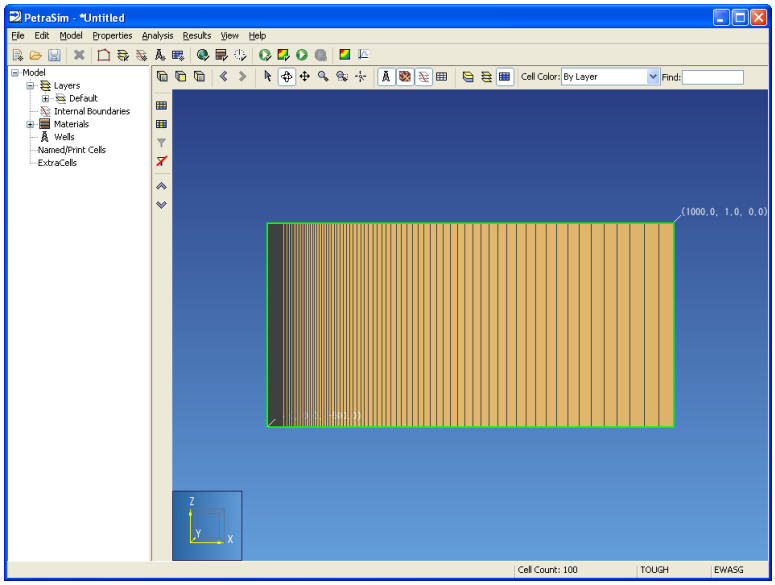

Figure 1: The resulting model

Global Properties

Global properties are those properties that apply to the entire model. We will make changes to some of the EOS options, including activating molecular diffusion. To edit global properties, you use the **Global Properties** dialog.

On the **Properties** menu, click **Global Properties…** (or click on the main toolbar).

1. In the **Global Properties** dialog, select the **Analysis** tab.
2. In the **Name** box, type *Geothermal Production, Brine and CO2*.
3. In the **Global Properties** dialog, select the **EOS** tab.
4. Select **Non-Isothermal**.
5. In the **Type of NCG** list, select **CO2**.
6. For **Dependence of Permeability on Pore Space**, select **Tubes in Series**.
7. In the **phi** box, type *0.8*.
8. In the **G** box, type *0.8*.
9. In the **Property Dependence on Salinity** list, select **Full Dependence**.
10. In the **Brine Enthalpy Correction** list, select **Michaelides**.
11. Select the **Include Vapor Pressure Lowering** checkbox.
12. Do NOT select **Molecular Diffusion**.
13. Click **OK** to close the **Global Properties** dialog.

Material Properties

To specify the material properties, use the **Material Data** dialog.

1. On the **Properties** menu, click **Edit Materials…** (or click on the toolbar).
2. In the **Name** box, type **POMED**.
3. In the **Density** box, type *2600.0*.
4. In the **Porosity** box, type *0.05*.
5. In all three **Permeability** boxes (X, Y, and Z), type *50e-15*.
6. In the **Wet Heat Conductivity** box, type *2.0*.
7. In the **Specific Heat** box, type *1000.0*.
8. Click **Apply** to save the changes.

In addition to the physical rock parameters, we also need to specify the relative permeability and capillary pressure functions for this material. These options can be found in the **Additional Material Data** dialog. To open this dialog, click the **Additional Material Data…** button.

To specify the relative permeability function

1. Select the **Relative Perm** tab.
2. In the **Relative Permeability** list, select **Corey's Curves**.
3. In the $\mathbf{S_{lr}}$ box, type *0.3*.
4. In the $\mathbf{S_{gr}}$ box, type *0.05*.

The default capillary pressure (none) is correct for this example.

Click **OK** to exit the **Advanced Material Data** dialog.

Click **OK** again to save your settings and exit the **Material Data** dialog.

Initial Conditions

The initial state of each cell in the model must be defined. To specify global initial conditions that will be used as the default for all cells in the model, on the **Properties** menu, click **Initial Conditions...** (or click on the toolbar).

To set the initial conditions

1. Select the **Two Fluid Phases (P, Xsm, Sg, T)** state option.
2. In the **Pressure** box, type *6.0E6*.
3. In the **Temperature** box, type *275.55*.
4. In the **Gas Saturation** box, type *0.45*.
5. In the **Salt Mass Fraction** box, type *0.3*.

Click **OK** to close the **Default Initial Conditions** dialog.

Define Reservoir Production

In this model, fluid is produced from the inner cell.

1. In the **3D View**, click the **Front View** button on the toolbar or rotate the model until the front is visible.
2. Use the **Zoom Box** tool to zoom in on the left side of the model until the left-most cell is visible.
3. Using the **3D Orbit Navigation** tool, right click the left-most cell and from the context menu, select **Edit Cells...** as shown in Figure 2.
4. In the **Cell Name** box, type *Production*.
5. Click the **Sources/Sinks** tab.
6. In the **Production** section, select the **Mass Out** check box.
7. In the **Rate** box, type *65*.
8. Click the **Print Options** tab.
9. Select the **Print Time Dependent Flow and Generation (BC) Data**.
10. Click **OK** to close the **Edit Cell Data** dialog.

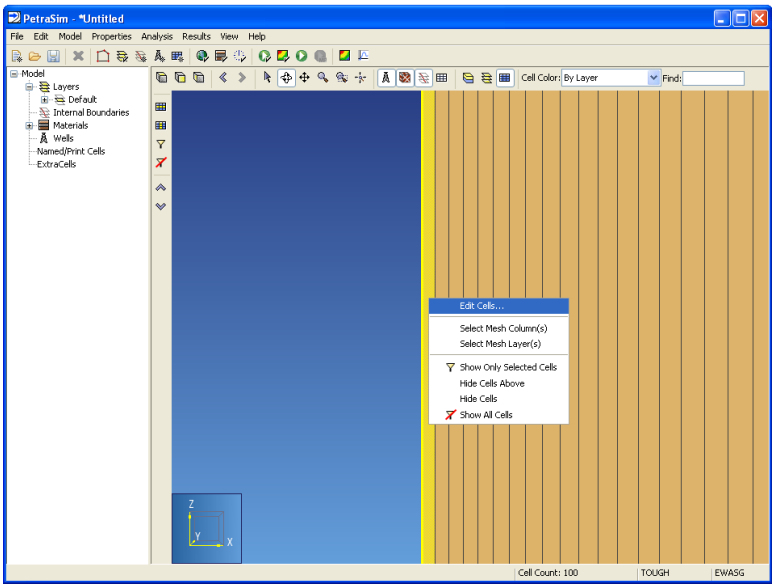

Figure 2: The cell to be selected

Solution Controls

We will now define the solution control options.

1. On the **Analysis** menu, click **Solution Controls…** (or click on the main toolbar).
2. In the **End Time** box, type *2.0E6* (approximately 23 days).
3. In the **Time Step** list, ensure that *Single Value* is selected.
4. In the **Time Step** box, type *1000*. This is the initial time step.
5. Select **Enable Automatic Time Step Adjustment**.
6. Click **OK** to exit the **Solution Controls** dialog.

Output Controls

By default, the simulation will print output every 100 time steps. We can increase the frequency of the output in the **Output Controls** dialog.

1. On the **Analysis** menu, click **Output Controls…** (or click on the main toolbar).
2. In the **Print and Plot Every # Steps** box, type *10*.
3. Next to the **Additional Print & Plot Times**, click the **Edit…** button.
4. In the **Additional Print Times** table, type *.50E5*. This will force a solution printout at this time.
5. Click **OK** to exit the **Additional Print Times** dialog.
6. Click **OK** to exit the **Output Controls** dialog.

Save and Run

The input is complete and you can run the simulation. If you haven't already, you may want to save your model in a new directory. For example

1. On the **File** menu, click **Save** (or click on the main toolbar).

2. Create a new folder named *ewasg_geothermal* and in the **File Name** box, type *geothermal.sim*.
3. Click **Save**.

To run the simulation, on the **Analysis** menu, click **Run TOUGH2** (or click on the main toolbar).

During the solution, a graph will display the time step size.

View 3D Results

To view the 3D results for a simulation, on the **Results** menu, click **3D Results** (or click on the main toolbar). The data for the current simulation will be automatically loaded and displayed.

To show gas saturation contours for the last time step

1. In the **Time(s)** list, select the last entry (**2.0E6**).
2. In the **Scalar** list, select **SG**.
3. Click to clear the **Show Isosurfaces** checkbox.

To add a slice plane on which contours will be displayed, click **Slice Planes…**. For this example we will show one slice plane. To configure the slice plane

1. In the **Axis** list, select **Y**.
2. In the **Coord** box, type *0.5*.
3. Select the **Scalar** check box.
4. Click **Close**.

To orient and view the model with mesh

1. Click the **Front View** button on the toolbar or rotate the model until the front is visible.
2. On the **View** menu, click **Show Mesh**.

The view can be zoomed and panned by holding the Alt and Shift keys, respectively, and dragging the mouse. After moving the view to the upper left corner of the model, the resulting contour plot is shown in Figure 3.

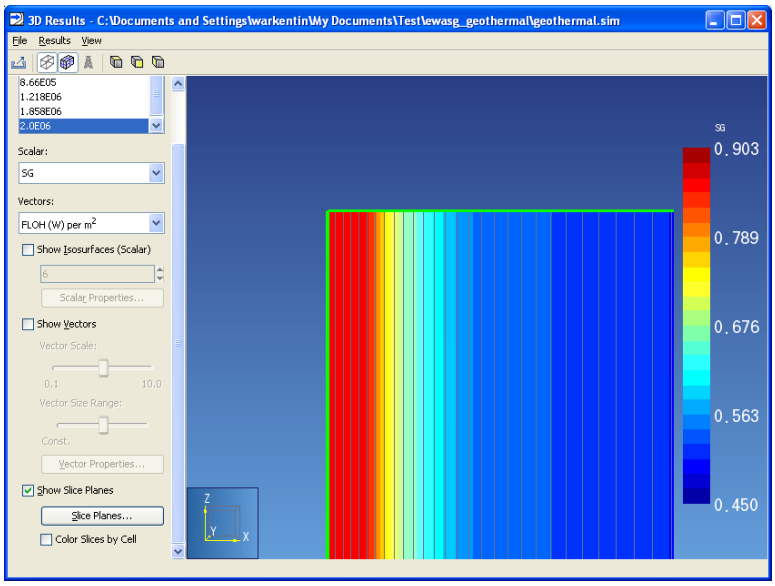

Figure 3: 3D representation of the saturation of gas

Close the **3D Results** window.

View Cell History Plots

You can view time history plots with the **Cell History** dialog. On the **Results** menu, click **Cell History Plots** (or click on the main toolbar).

The **Cell History** dialog will be displayed. In this window, you can display time history data using a plotting parameter and a list of cells. For example, to view the gas saturation in the Heat Source cell

1. In the **Variable** list, select **SG**.
2. In the **Cell Name** list, select **Production**.

Figure 4 shows the time history plot of SG at the production cell. The user can export this data for plotting in spreadsheets.

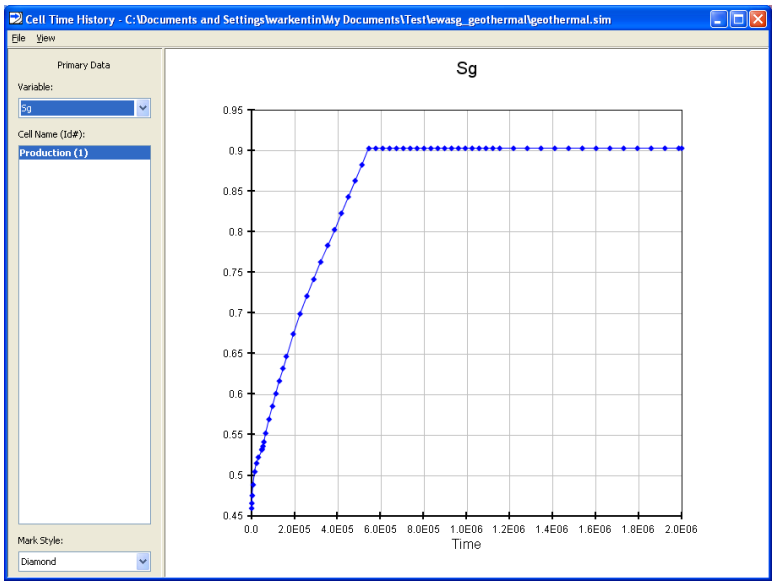

Figure 4: The cell history plot for gas saturation in the production cell

Close the **Cell History** window.

Line Plots

PetraSim also supports line plots, where you define a line in 3D space and the data is interpolated along the line. To make a line plot

1. First open the 3D results view. On the **Results** menu, click **3D Results** (or click on the main toolbar).
2. On the **File** menu, click **Line Plot…**.
3. In the **Point 1** coordinate boxes (**X, Y,** and **Z**) type *5.0, 0.5,* and *-250.0*, respectively.
4. In the **Point 2** coordinate boxes (**X, Y,** and **Z**) type *1000.0, 0.5,* and *-250.0*.
5. Click **OK** to close the **Line Plot** dialog.
6. This will open a **Line Plot** window.
7. In the **Variable** list, select **P (Pa)**.
8. In the **Time** list, select **2.0E06** (the last time).

The plot is shown in Figure 5. You can export this data to a comma separated value file for import into a spreadsheet.

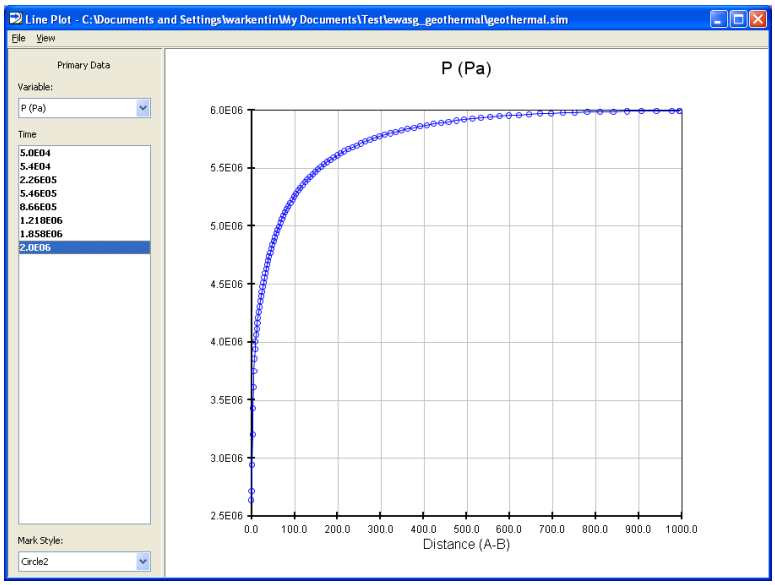

Figure 5: Line plot of pressure

Close the **Line Plot** window.

Data saved using the line plot are shown in the same format as in the TOUGH2 user's manual in Figure 6. The calculated results compare well with the manual results.

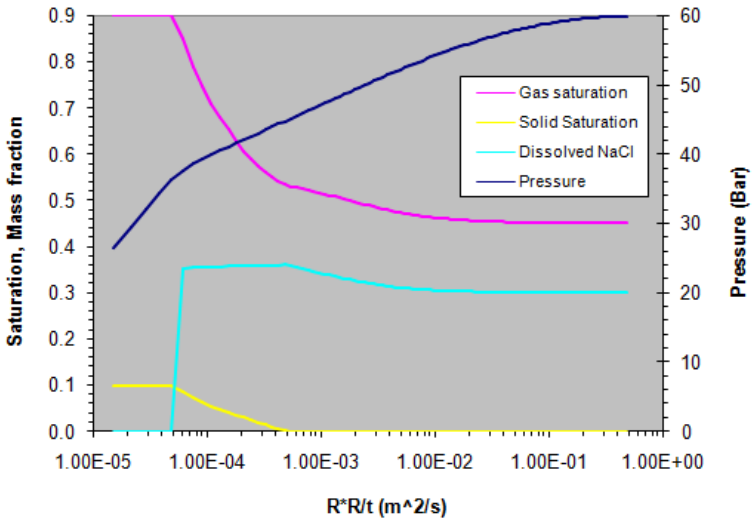

Figure 6: Results for comparison with TOUGH2 manual

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.