GMS 7.0 TUTORIALS

SEEP2D – Confined

1 Introduction

SEEP2D is a 2D finite element, steady state, flow model. It is typically used for profile models, i.e., cross-section models representing a vertical slice through a flow system which is symmetric in the third dimension. Examples include earth dams, levees, sheetpiles, etc.

SEEP2D can be used for both confined and unconfined problems. Accordingly, the SEEP2D tutorials are divided into two parts. This tutorial describes how to set up and solve a confined seepage problem for SEEP2D using GMS. The steps required for simulating the unconfined condition are described in a separate tutorial. The two SEEP2D tutorials are entirely independent and can be completed in any order. However, it is recommended that this tutorial be completed before the other tutorial since the motivation behind many of the steps in the model definition process is described in more detail in this tutorial.

1.1 Contents

1.2 Outline

Follow these steps to complete this tutorial:

1. Create a SEEP2D conceptual model.
2. Map the model to a 2D mesh.
3. Define conditions.
4. Run SEEP2D.

1.3 Required Modules/Interfaces

You will need the following components enabled to complete this tutorial:

- Mesh
- Map
- SEEP2D

You can see if these components are enabled by selecting the *Help* | *Register* command.

2 Description of Problem

The problem we will be solving in this tutorial is shown in Figure 1.

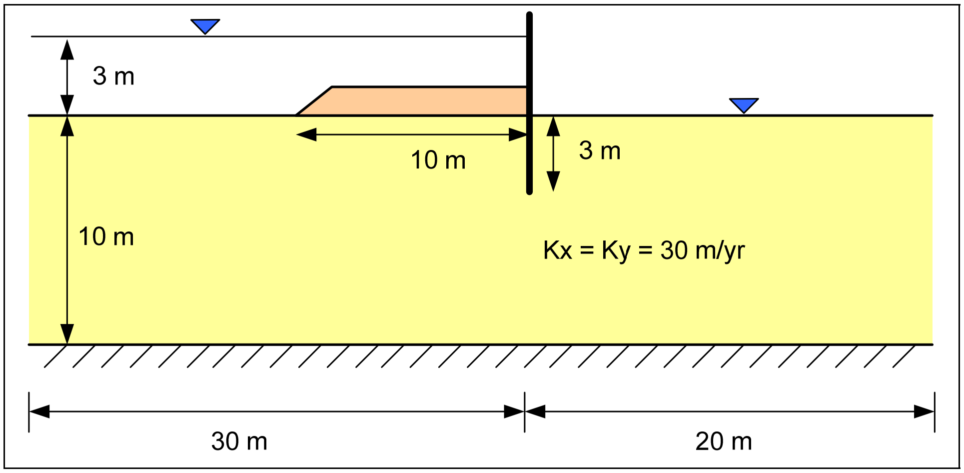

Figure 1. Confined Flow Problem.

The problem involves a partially penetrating sheetpile wall with an impervious clay blanket on the upstream side. The sheetpile is driven into a silty sand deposit underlain by bedrock at a depth of 10 m.

From a SEEP2D viewpoint, this problem is a "confined" problem. For SEEP2D, a problem is confined if it is completely saturated. A problem is unconfined if it is partially saturated.

3 Getting Started

Let's get started.

1. If necessary, launch GMS. If GMS is already running, select the *File* | *New* command to ensure that the program settings are restored to their default state.

4 Setting the Units

We will start by setting the units we are using. GMS will display the units we select next to the input fields to remind us what they are.

1. Select the *Edit* | *Units* command.
2. Select **m** for the *Length* units.
3. Select **yr** for the *Time* units.
4. Select **kg** for the *Mass* units.
5. Select the *OK* button.

5 Creating the Mesh

The first step in setting up the problem is to create the finite element mesh. Two types of elements can be used with SEEP2D: three node triangular elements and four node quadrilateral elements. We will use a conceptual model to define our model domain and then use the *Map → 2D Mesh* command to automatically fill in the elements and nodes.

5.1 Defining a Coordinate System

Before we construct the mesh, we must first establish a coordinate system. We will use a coordinate system with the origin 30 meters upstream of the sheetpile at the top of the bedrock as shown in Figure 2.

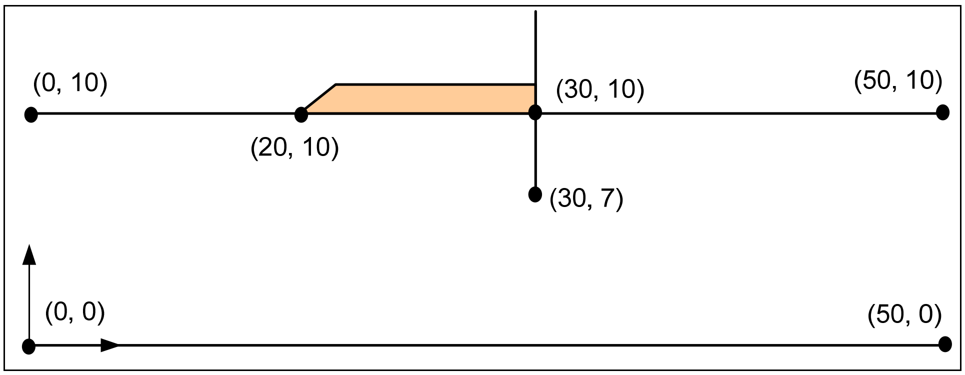

Figure 2. Coordinate System.

5.2 Creating the Conceptual Model

First we'll create a SEEP2D conceptual model. Then we'll create a coverage in that conceptual model.

1. In the *Project Explorer* right-click on the empty space and then, from the pop-up menu, select the *New* | *Conceptual Model* command
2. Change the *Name* to "**Confined**".
3. Change the *Model* to **SEEP2D/UTEXAS**.
4. Uncheck the *UTEXAS* toggle.
5. Click *OK*.
6. In the *Project Explorer*, right-click on the **Confined** conceptual model and select the *New Coverage* command from the pop-up menu.
7. Change the name to **boundary**.
8. Turn on the following properties:

- *Refinement*
- *Head*

9. Click *OK*.

5.3 Creating the Corner Points

We are now ready to create some points at key corner locations. These points will then be used to guide the construction of a set of arcs defining the mesh boundary.

1. Right-click on the **boundary** coverage in the *Project Explorer* and select the *Attribute Table* command.

2. Turn on the *Show point coordinates* toggle.

3. Enter the following coordinates in the spreadsheet.

X	Y
0	0
0	10
20	10
30	10
30	7
30.3	10
30.3	7
50	10
50	0

4. Click *OK* to exit the dialog.

5. Now select the *Frame* macro.

If you need to edit the node coordinates this can be done by using the *Select Points/Nodes* tool. When this tool is active you can select points and change the coordinates using the edit fields at the top of the GMS window. You can also select points and delete them using the *Delete* key on the keyboard or the *Delete* command in the *Edit* menu.

We have entered points at 30, 10 and at 30.3, 10. We have done this to model the sheet pile which is about 0.3 m thick.

The nodes you have created should resemble the nodes shown in Figure 3.

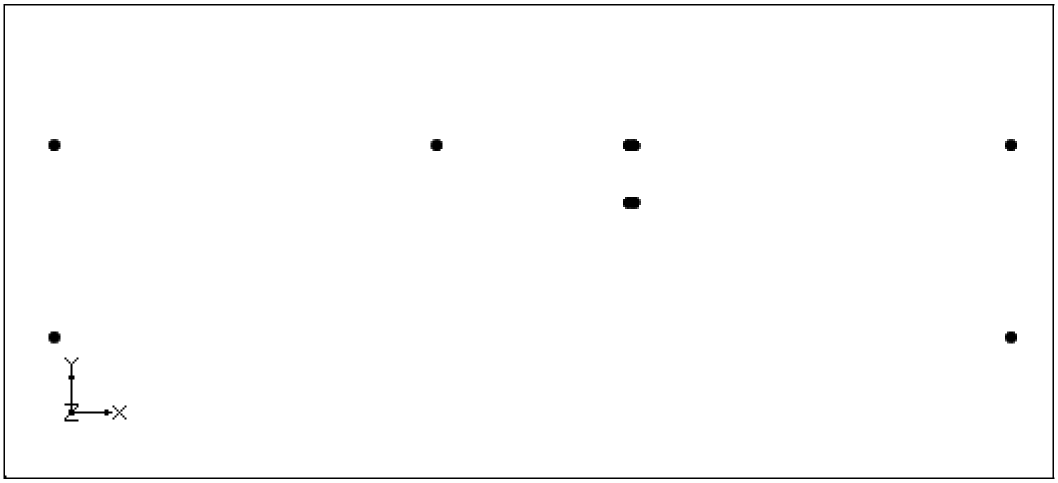

Figure 3. Points Created in the Boundary Coverage

5.4 Creating the Arcs

Now that the corner nodes are created, the next step is to create the arcs. This can be accomplished as follows:

1. Select the *Create Arc* tool .
2. Create the arcs shown in the figure below.

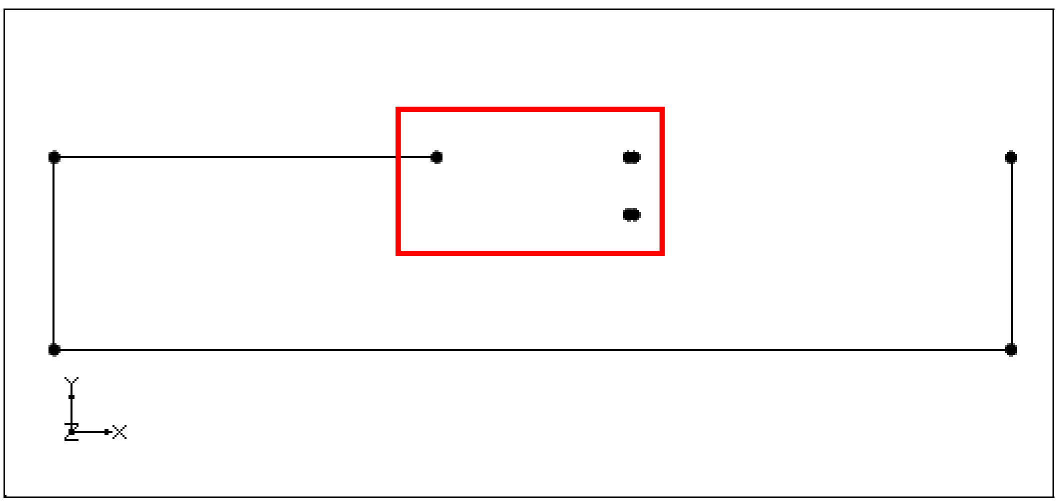

Figure 4. Arcs Connecting Points

3. Select the *Zoom* tool and drag a box as shown by the red box in Figure 4.
4. Select the *Create Arc* tool .
5. Create the arcs shown in the following figure.

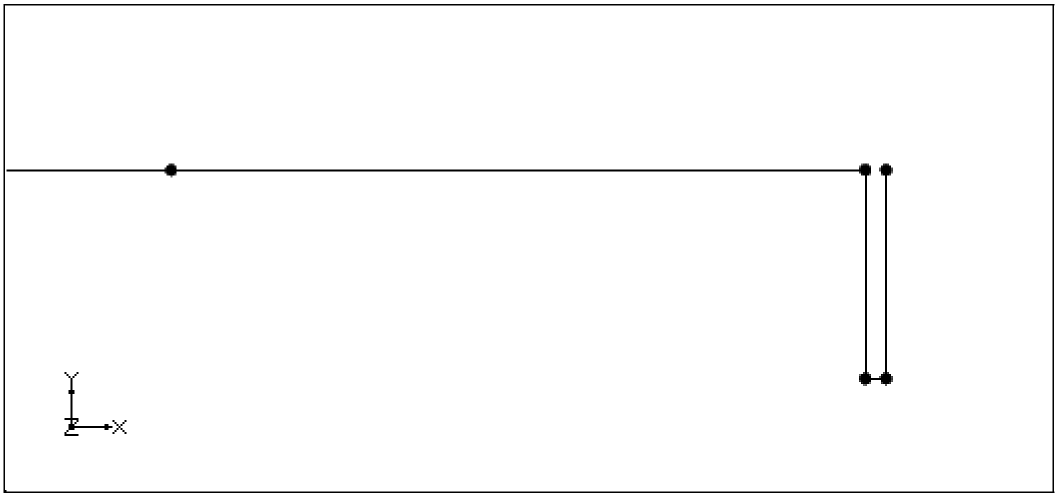

Figure 5. Arcs Connecting Points

6. Now select the *Frame* macro.
7. Select the *Zoom* tool and drag a box as shown by the red box in Figure 6.

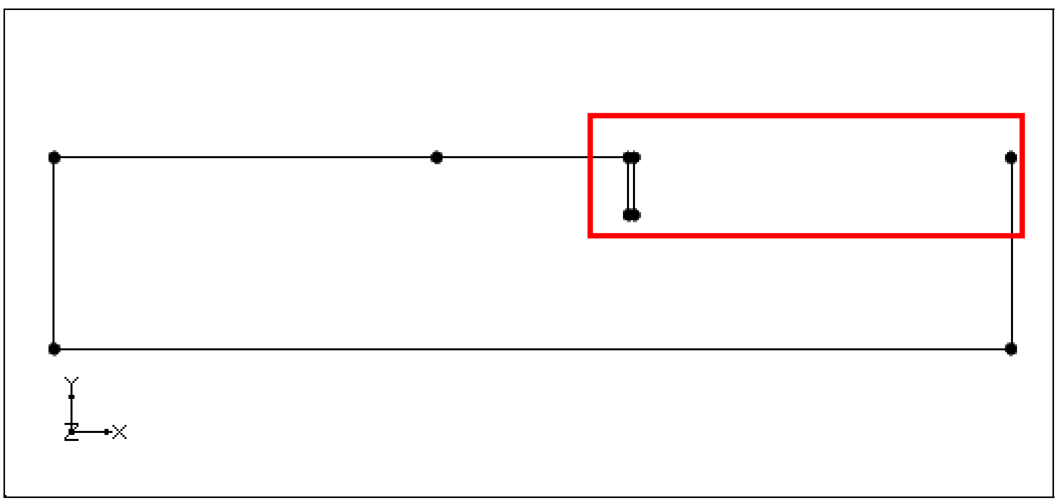

Figure 6. Arcs Connecting Points

8. Create the last arc shown in the following figure.

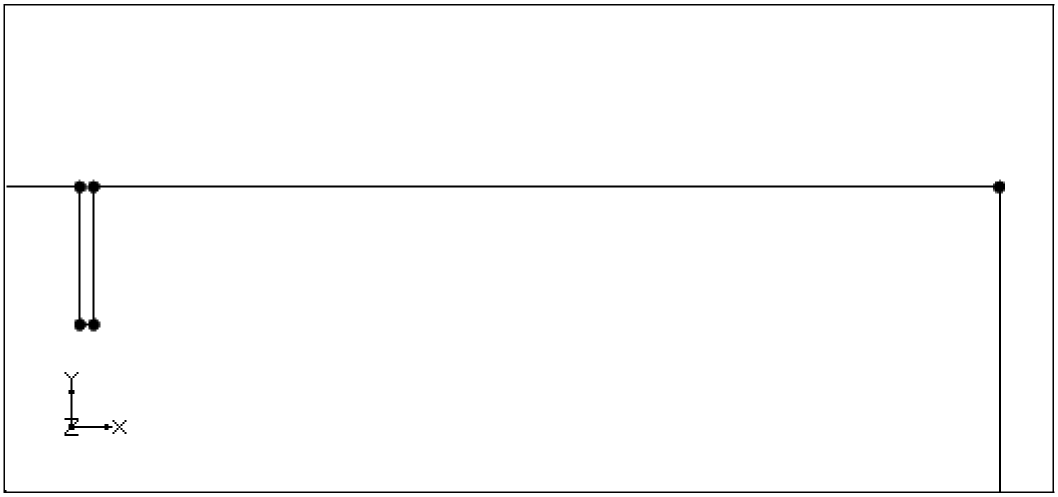

Figure 7. Arcs Connecting Points

9. Now select the *Frame* macro.

10. Your model should now look like Figure 8.

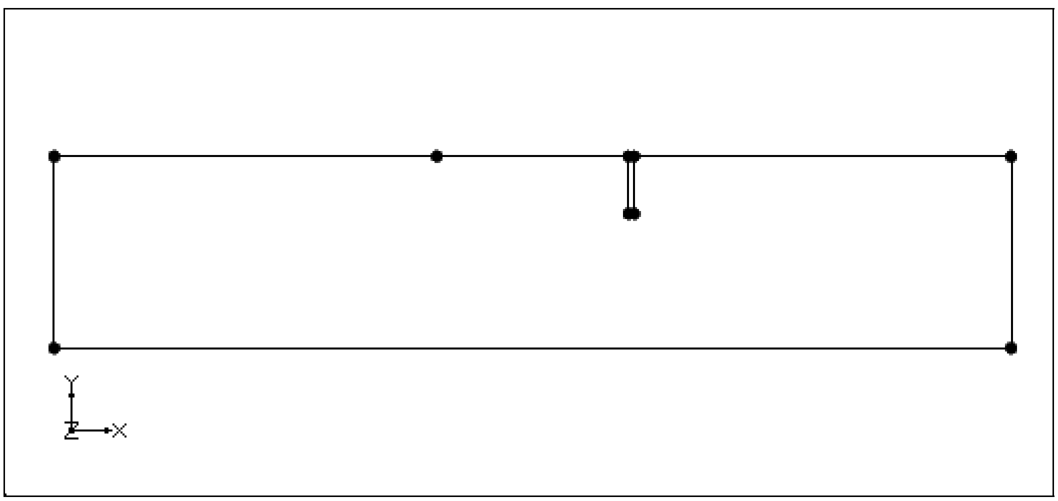

Figure 8. Arcs Connecting Points

5.5 Redistributing Vertices

At this point, all of our arcs have one edge and zero vertices. When we issue the *Map → 2D Mesh* command, the density of the elements in the interior of the mesh is controlled by the edge spacing along the arcs. Thus, we will subdivide the arcs to create appropriately sized edges.

1. Choose the *Select Arcs* tool.

2. Select the *Edit | Select All* command.

3. Select the *Feature Objects | Redistribute Vertices* command.

4. Select the *Specified spacing* option.

5. Enter a value of **1.2** for the spacing.

6. Select the *OK* button.

7. To see the vertices, switch to the *Select Vertices* tool .

5.6 Creating the Polygons and Building the Mesh

Before we can create the mesh, we first have to build a polygon. Simply creating the arcs does not create the polygon. We must explicitly create the polygons using the *Build Polygons* command.

1. Select the *Feature Objects* | *Build Polygons* command.

At this point, we are ready to construct the mesh.

2. Select the *Feature Objects* | *Map → 2D Mesh* command.

You should now see a 2D mesh.

6 Initializing SEEP2D

Now that the mesh is constructed, we can begin to enter the SEEP2D data.

1. Select the *SEEP2D* | *New Simulation* command.

2. In the *Model Type* section, make sure the *Saturated/Unsaturated with linear front* option is selected.

3. Select the *OK* button to exit the dialog.

7 Assigning Material Properties

The next step in creating the model is to define material properties. There is one set of material properties for each zone of the mesh. The material properties are k_1, k_2, and an angle. The values k_1 and k_2 represent the two principal hydraulic conductivities and the angle is the angle from the x-axis to the direction of the major principle hydraulic conductivity measured counter-clockwise as shown in Figure 9.

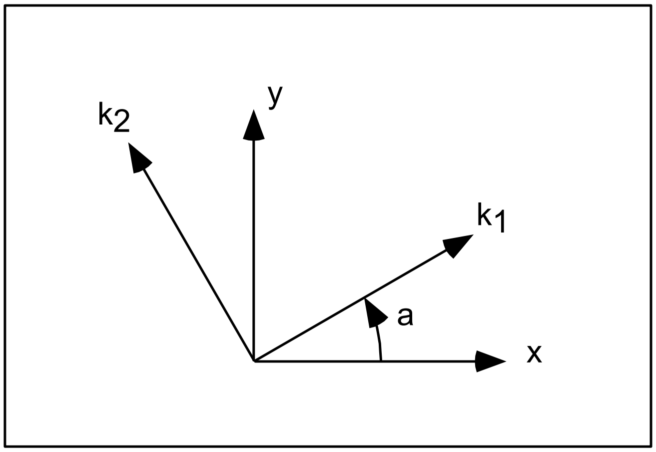

Figure 9. Definition of Hydraulic Conductivity Angle.

With most natural soil deposits, the major principal hydraulic conductivity is in the x direction, the minor principal hydraulic conductivity is in the y direction, and the angle is zero.

Each element in the mesh is assigned a material id. The material id is an index into a list of material properties. All of the elements we have created so far have a default material id of 1. This is sufficient since we only have one soil type in our problem. To enter the properties for material #1:

1. Double click on the *Material Properties* in the *Project Explorer*.
2. Enter a value of **30** for both *k1* and *k2*.
3. Select the *OK* button.

Note: The units for hydraulic conductivity are L/T (length / time). The length units should always be consistent with the units used in defining the mesh geometry. Time units can be used in any format. However, small time units (such as seconds) will result in very small velocity values and may make it difficult to display velocity vectors. It is recommended that time units of days or years be used.

8 Assigning Boundary Conditions

The final step in defining the model is to assign boundary conditions to the mesh. For the problem we are modeling there are two types of boundary conditions: constant head and no-flow (flow is parallel to the boundary). With the finite element method, not assigning a boundary condition is equivalent to assigning a no-flow boundary condition. Therefore, all of the boundaries have a no-flow boundary condition by default and all that is necessary in this case is to assign the constant head boundary conditions.

8.1 Constant Head Boundaries

The constant head boundary conditions for our mesh are shown in Figure 10.

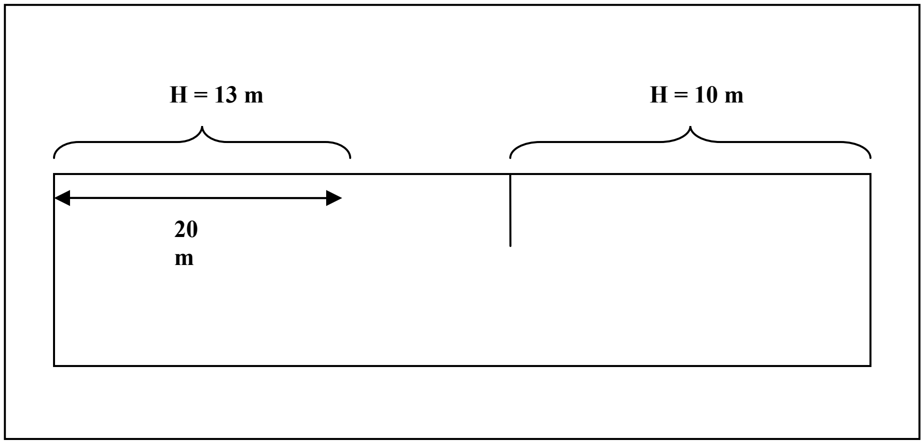

Figure 10. Constant Head Boundary Conditions.

The region on the left in Figure 10 represents the top of the mesh on the upstream side that is not covered with the clay blanket. The region on the right represents the downstream side of the mesh. Using a datum of zero, the total head in either case is simply the elevation of the water. As mentioned above, all other boundaries on the mesh have a no-flow boundary condition by default.

To enter the constant head boundary conditions for the region on the left:

1. Select the *Map Data* folder in the *Project Explorer.*

2. Choose the *Select Arcs* tool .

3. Double-click on the arc on the top-left of the model.

4. Change the *Type* to **head** and enter **13.0** in the *Head* field.

5. Select OK to exit the dialog.

6. Double-click on the arc on the top-left of the model.

7. Change the *Type* to **head** and enter **10.0** in the *Head* field.

Now we are ready to convert the conceptual model to the SEEP2D numerical model. This will assign all of the boundary conditions using the data defined on the feature objects.

8. In the *Project Explorer* right-click on the *Confined* conceptual model and select *Map to →SEEP2D* command from the pop-up menu.

A set of symbols should appear indicating that the boundary conditions have been assigned.

9 Saving the Simulation

We are now ready to save the simulation.

1. Select the *File* | *Save As* command.
2. Locate and open the directory entitled **tutfiles\SEEP2D\s2con**.
3. Enter **blanket** for the file name.
4. Select the *Save* button.

10 Running SEEP2D

To run SEEP2D:

1. Select the *SEEP2D* | *Run SEEP2D* command. At this point SEEP2D is launched in a new window.
2. When the solution is finished, select the *Close* button.

GMS automatically reads in the SEEP2D solution. You should see the solution as a flow net. The flow net consists of equipotential lines (total head contours) and flow lines.

You can also view the total flow through the cross section. To turn on the display of the total flow through the cross section, do the following:

3. Select the *Display Options* button.
4. Select 2D Mesh Data from the list on the left of the dialog.
5. Select the *SEEP2D* tab.
6. Turn on the *Title* and *Total flow rate* options.
7. Select the *OK* button.

11 Conclusion

This concludes the tutorial. Here are the things that you should have learned in this tutorial:

- SEEP2D is a 2D finite element seepage model.
- You can use a conceptual model to create a 2D mesh.