GMS TUTORIALS

SEEP2D – Confined

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, sheet piles, etc.

SEEP2D can be used for both confined and unconfined problems. Accordingly, the SEEP2D tutorials are divided into two parts. The tutorial in this chapter 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 the following chapter. The two SEEP2D tutorials are entirely independent and can be completed in any order. However, it is recommended that the first tutorial be completed before the second tutorial since the motivation behind many of the steps in the model definition process is described in more detail in the first tutorial.

1.1 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.

1.2 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 *File* | *Register* command.

2 Description of Problem

The problem we will be solving in this tutorial is shown in Figure 2-1.

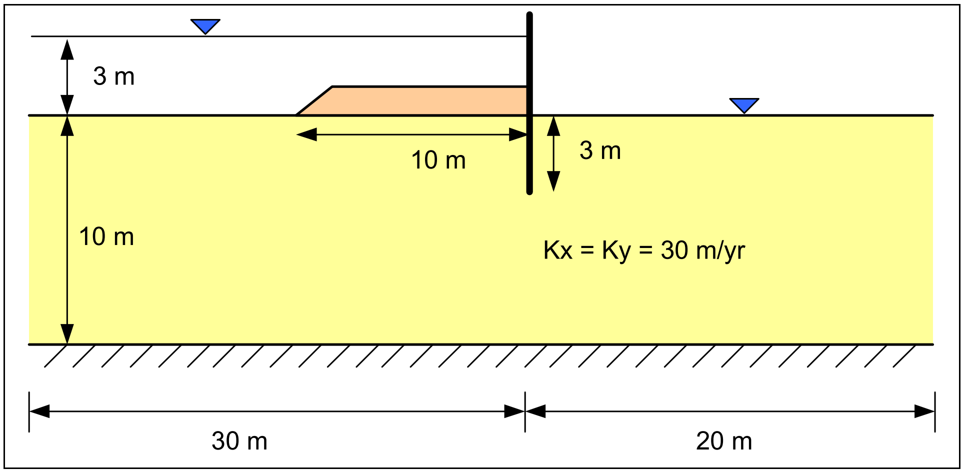

Figure 2-1 Confined Flow Problem.

The problem involves a partially penetrating sheet pile wall with an impervious clay blanket on the upstream side. The sheet pile 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.

A variety of methods are available in GMS for constructing a 2D mesh. The approach we will use here will be to define the boundary of the mesh using map objects, 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 sheet pile at the top of the bedrock as shown in Figure 5-1.

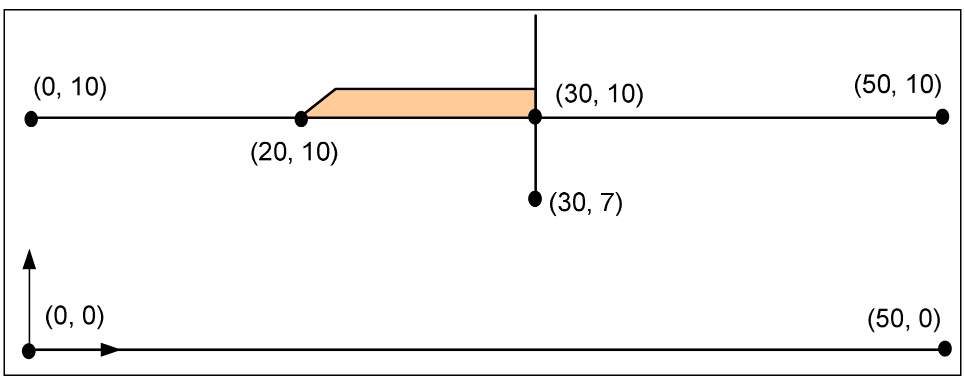

Figure 5-1 Coordinate System.

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**.

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. Turn on the following properties:

- *Refinement*

- *Head*

8. Turn off the *Single Head Value for Arcs* property.

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. Select the *Create Point* tool from the *Tool Palette*.

2. Click anywhere on the graphics window to create a point. Then edit the coordinates using the edit fields at the top of the GMS window. Change the *X*: value to **0.0** and the *Y*: value to **0.0**.

3. Create another point by clicking on the graphics window and change the x and y coordinates to **50.0** and **10.0** respectively.

4. Now select the *Frame* macro.

5. Now create points at the following locations using the same steps as before:

X	y
0	10
20	10
30	10
30	7
50	0

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. You can also select points and delete them using the *Delete* key on the keyboard or the *Delete* command in the *Edit* menu.

The nodes you have created should resemble the nodes shown in Figure 5-2.

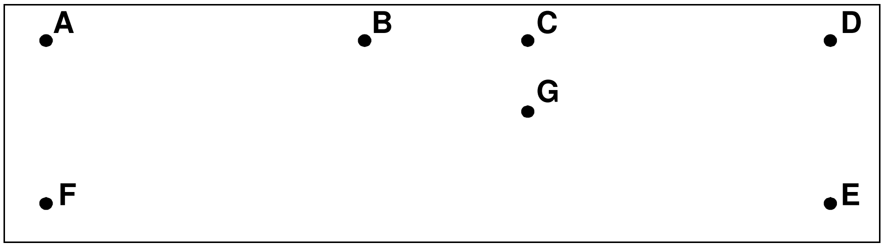

Figure 5-2 Points Created with the Create Point Tool

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. Click on the existing points to create arcs between the points around the perimeter of the model. Start by clicking on point A, as shown in Figure 5-3, then click on B and so on to point F to A.
3. Also create an arc for the sheet pile. Start by clicking on point C and click on point G to finish.
4. Your model should now look like Figure 5-3.

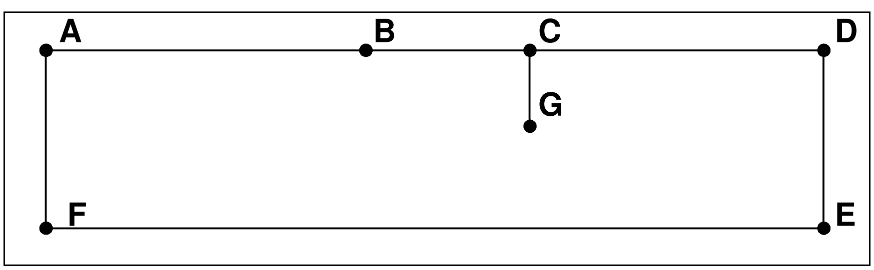

Figure 5-3 Arcs Connecting Points

5.5 Modeling the Sheet Pile

To model the sheet pile, we need a no-flow boundary. However, SEEP2D doesn't have a no-flow boundary condition that we can apply. Thus, we will put a break in the mesh at the location of the sheet pile, as shown in Figure 5-4.

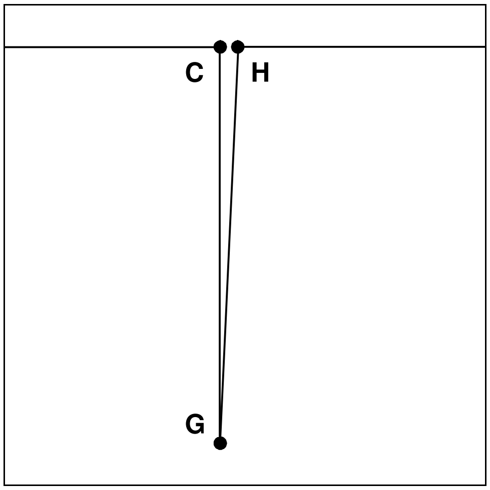

Figure 5-4 Sheet Pile Zoomed

To create the break, we need to add another point and another arc as shown in Figure 5-4. To create the point:

1. Use the *Zoom* tool to zoom up on the sheet pile area shown in Figure 5-4.
2. Choose the *Create Vertex* tool .
3. Create point **H** by clicking anywhere on the arc to the right of node **C**.
4. With the vertex still selected, change the vertex coordinates to *x* = **30.1** and *y* = **10.0** using the x and y fields in the Edit Window.
5. With the vertex still selected, select the *Feature Objects* | *Vertices ←→ Nodes* command.

This turns the vertex into a node and splits the arc into 2 arcs. Now we can delete the small arc between points **C** and **H**.

1. Select the *Select Arcs* tool .
2. Delete the small arc between points **C** and **H** by selecting it and hitting the *Delete* key.
3. Select the *Create Arc* tool .
4. Create the vertical arc between points **G** and **H** as shown in Figure 5-4 by clicking on point **G** and then clicking on point **H**.
5. Select the *Frame* macro to see the entire model.

5.6 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.

5.7 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 Renumbering the Mesh

The next step in the model definition process is to renumber the nodes and elements. The sequence that the nodes are numbered has a dramatic impact on the amount of time required by SEEP2D to find a solution. The nodes and elements have been numbered in the order that they were created. This numbering is not optimal. The relative quality of the node numbering is indicated by the nodal half band width. The time required for a solution is quadratically proportional to the nodal half band width. To view the nodal half band width:

1. Select the *2D Mesh Data* folder in the *Project Explorer*.
2. In the *Project Explorer*, right-click on the *2D Mesh Data* folder and select the *Properties* command from the pop-up menu.

3. Note the *Max node half band width*.

4. Select the *Done* button.

The ID's of each of the nodes in the mesh can be displayed as follows:

5. Select the *Display Options* button.

6. Turn on *Node numbers*.

7. Select the *OK* button.

To renumber the nodes and elements, we must first select a string of nodes on the boundary. This string of nodes defines a starting point for the renumbering process.

8. Choose the *Select Node Strings* tool.

9. Select the leftmost column of nodes in sequence from top to bottom. This can be done one of two ways:

10. (a) Click on the top node, then the next node down and so on until all of the nodes in the column have been selected. Or,

11. (b) Click on the top node in the column, and while holding down the *Control* key, click on the bottom node in the column. All of the intermediate nodes will automatically be selected.

12. Select the *Mesh* | *Renumber* command.

Notice how the node numbering has changed. Bring up the *Properties* dialog again to examine the new nodal half band width. Generally, the best node half band width occurs when a sequence of renumbering nodes is selected along the minor axis of the mesh. When in doubt, the mesh can be renumbered using several different strings of nodes to find the best numbering sequence.

To turn off the node numbers:

13. Select the *Display Options* button.

14. Turn the *Node numbers* option off.

15. Select the *OK* button.

7 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, select the Confined option.

3. Select the OK button to exit the dialog.

8 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 8-1.

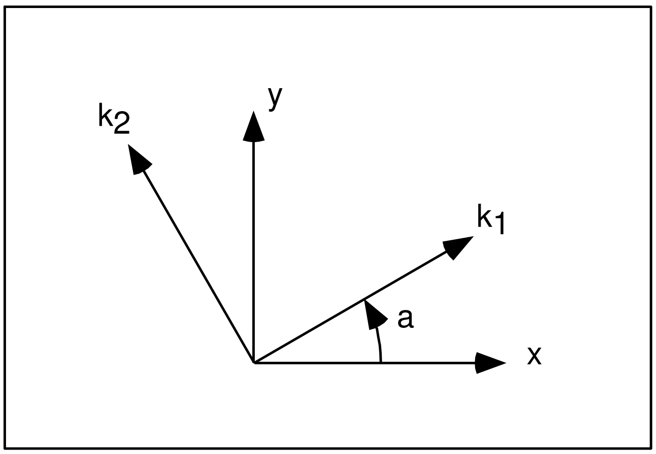

Figure 8-1 Definition of Hydraulic Conductivity Angle.

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. Select the SEEP2D | Material Properties command.

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.

9 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.

There are two ways to apply boundary conditions for the SEEP2D model. The boundary conditions are applied directly to the mesh in the 2D mesh module using node strings, or they can be applied to the conceptual model as arc properties which can then be mapped to the 2D Mesh using *Map → SEEP2D*. We will Use the conceptual model approach to assign the boundary conditions.

9.1 Constant Head Boundaries

The constant head boundary conditions for our mesh are shown in Figure 9-1.

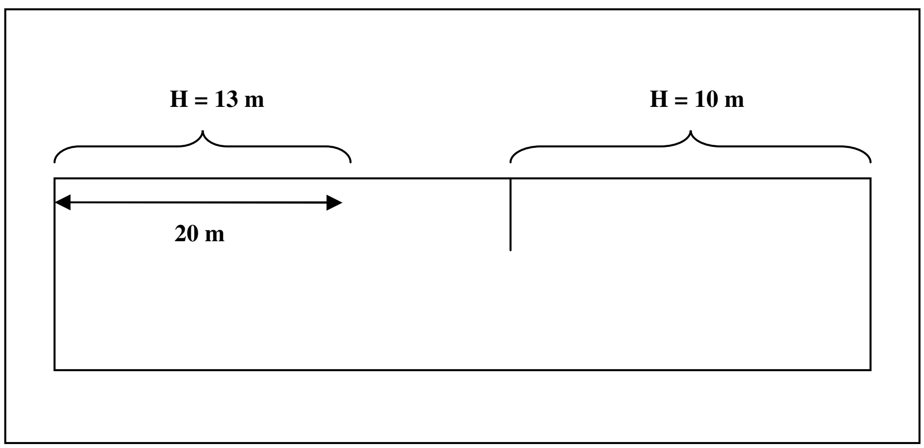

Figure 9-1 Constant Head Boundary Conditions.

The region on the left in Figure 9-1 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. While holding down the Shift key, click on arc between points **A** and **B** and between points **H** and **D**.

4. Select the *Properties* button.

5. In the *All* row in the spreadsheet, change the *Type* to **head**. This assigns this type to both arcs.

6. Select the *OK* button.

Note that no head values were assigned to the arcs. The head values are assigned to the nodes at the ends of the arc.

7. Select the *Select Points/Nodes* tool.

8. Select nodes **A** and **B** and select the *Properties* button.

9. In the *All* row in the spreadsheet, change the *Head* to **13**. This assigns the head to both nodes.

10. Select the *OK* button.

11. Now set the head for nodes **H** and **D** to be **10**.

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

12. 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.

10 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.

11 Running SEEP2D

To run SEEP2D:

1. Select the *2D Mesh Data* folder in the *Project Explorer*.

2. Select the *SEEP2D | Run SEEP2D* command. At this point SEEP2D is launched in a new window.

3. 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:

4. Select the *Display Options* button .

5. Select the *SEEP2D* tab.

6. Turn on the *Title* and *Total flow rate* options.

7. Select the *OK* button.

12 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.

- SEEP2D uses a banded matrix solver so renumbering the nodes can help reduce the amount of memory used.

