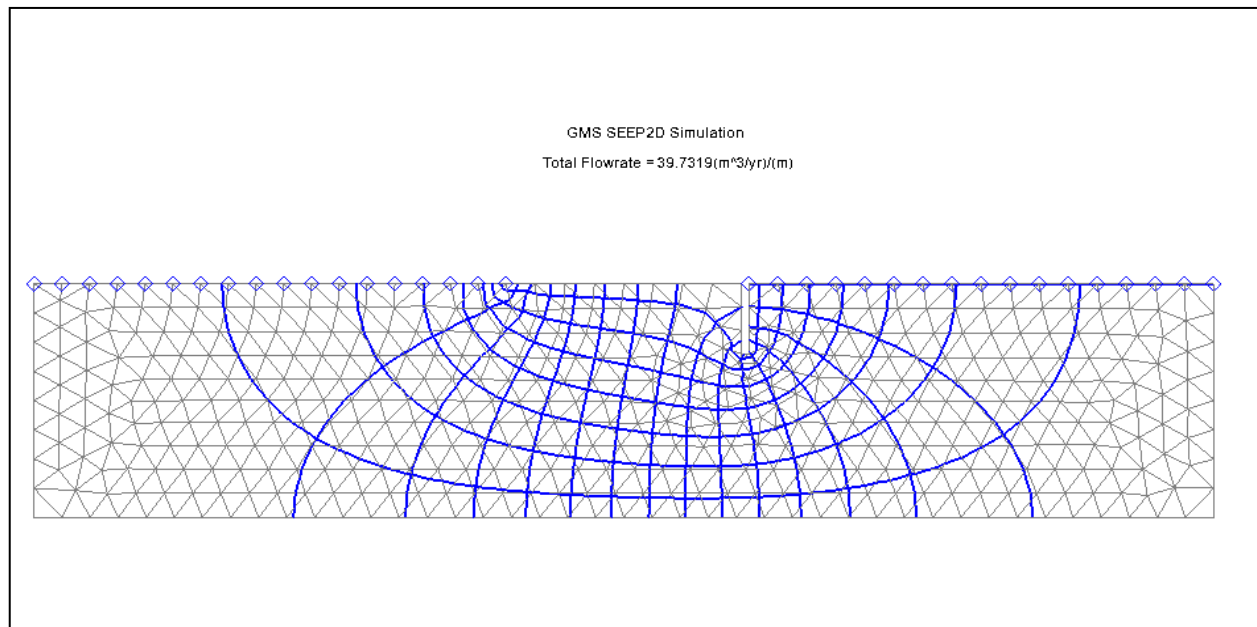


GMS 9.2 Tutorial

SEEP2D – Sheet Pile

Use SEEP2D to create a flow net around a sheet pile



Objectives

Learn how to set up and solve a seepage problem involving flow around a sheet pile using the SEEP2D interface in GMS.

Prerequisite Tutorials

- None

Required Components

- GIS
- Map
- Mesh
- SEEP2D

Time

- 30-60 minutes



1 Contents

1	Contents	2
2	Introduction.....	2
2.1	Outline	2
3	Description of Problem	2
4	Program Mode.....	3
5	Getting Started	4
6	Setting the Units	4
7	Saving the Project	5
8	Creating the Conceptual Model Features	5
8.1	Defining a Coordinate System	5
8.2	Creating the Corner Points.....	5
8.3	Creating the Arcs	7
8.4	Creating the Polygons.....	9
8.5	Assigning the Material Properties and Zones	9
9	Assigning Boundary Conditions	10
9.1	Constant Head Boundaries	10
9.2	Building the Finite Element Mesh	11
10	Running SEEP2D	12
11	Conclusion.....	13

2 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.

2.1 Outline

Follow these steps to complete this tutorial:

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

3 Description of Problem

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

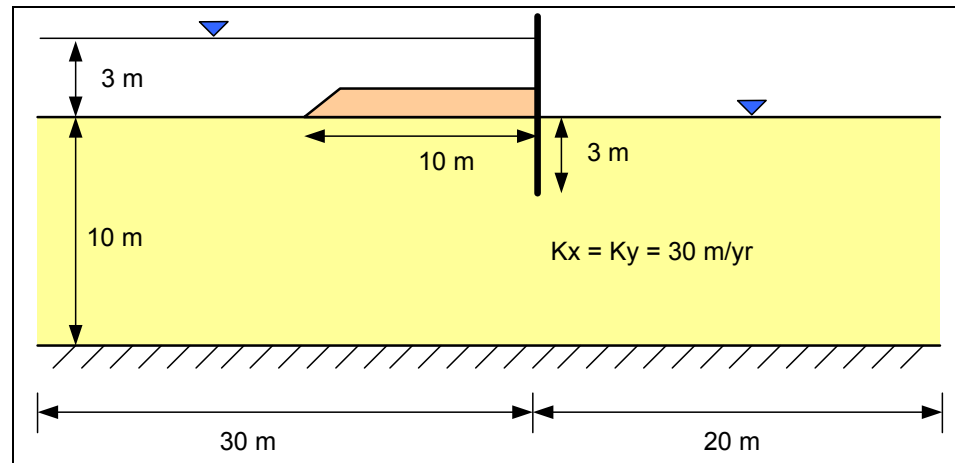


Figure 1. Confined flow problem.

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.

4 Program Mode

The GMS interface can be modified by selecting a *Program Mode*. When you first install and run GMS it is in the standard or "GMS" mode which provides access to the complete GMS interface, including all of the MODFLOW tools. The "GMS 2D" mode provides a greatly simplified interface to the SEEP2D and UTEXAS codes. This mode hides all of the tools and menu commands not related to SEEP2D and UTEXAS. This tutorial assumes that we are operating in the GMS 2D mode. Once the mode is changed, you can exit and restart GMS repeatedly and the interface stays in the same mode until you change it back. Thus, you only need to change the mode once if you intend to repeatedly solve SEEP2D/UTEXAS problems. If you are not already in GMS 2D mode, do the following. If you are already in GMS 2D mode, you can skip ahead to the next section.

1. Launch GMS.
2. Select the *Edit | Preferences* command.
3. Select the *Program Mode* option on the left side of the dialog.
4. On the right side of the dialog, change the mode to *GMS 2D*.
5. Click on the *OK* button.

6. Click *Yes* in response to the warning.
7. Click *OK* to get rid of the *New Project* window and then select the *File | Exit* command to exit GMS.

5 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.

At this point, you should see the *New Project* window. This window is used to set up a GMS conceptual model. A conceptual model is a set of GIS features (points, lines, and polygons) that are used to define the model input. The data in the conceptual model are organized into a set of layers or groups called *coverages*. Each coverage is used to define a portion of the input and the properties that are assigned to the features in a coverage are dependent on the coverage type. GMS 2D allows us to quickly and easily define all of the coverages needed for our conceptual model using the *New Project* window. Most of the options you see in the window are related to UTEXAS. For SEEP2D models, the only coverage we need is the *Profile lines* coverage. This allows us to define the geometry of the mesh, the boundary conditions, and the material zones.

2. Change the *Conceptual model name* to **Sheetpile Model**.
3. Turn **off** the *UTEXAS* option in the *Numerical models* section.
4. Make sure the *Profile lines* option is still selected and click *OK*.

You should see a new conceptual model object appear in the Project Explorer.

6 Setting the Units


Before we continue, we will establish the units we will be using. GMS will display the appropriate units label next to each of the input fields to remind us to be sure and use consistent units.

1. Select the *Edit | Units* command.
2. For the length select the “...” button, and change the units for both horizontal and vertical to **meters**.
3. Select **yr** for the *Time* units.
4. Select **kg** for the *Mass* units.
5. Select the *OK* button.

7 Saving the Project

Before continuing, we will save our project to a GMS project file.

1. Select the *File | Save As* command.
2. Path to a location where you wish to save your project
3. Change the name of the project file to **s2con**.
4. Click on the *Save* button.

As you continue with the tutorial, click on the Save macro  frequently to save your changes.

8 Creating the Conceptual Model Features

The first step in setting up the problem is to create the GIS features defining our problem geometry. We will begin by entering a set of points corresponding to the key locations in the geometry. We will then connect the points with lines called "arcs" to define the outline of the problem. We will then convert the arcs to a closed polygon defining our problem domain. Once this is complete, the arcs and the polygon will be used to build the finite element mesh and define the boundary conditions to the problem.

8.1 Defining a Coordinate System

Before we construct the conceptual model features, 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.

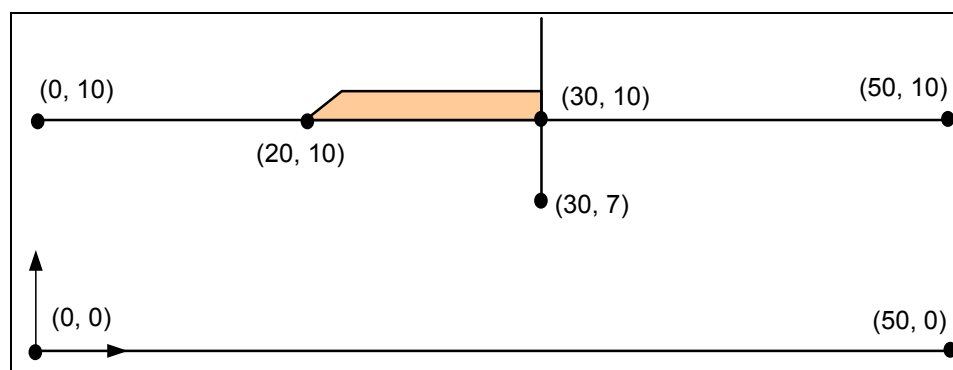



Figure 2. Coordinate system.


8.2 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 model boundary.

1. Right-click on the **Profile lines** coverage in the *Project Explorer* and select the *Attribute Table* command.
2. 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

3. Click *OK* to exit the dialog.
4. Now select the *Frame* macro  to center the view on the new points.

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.

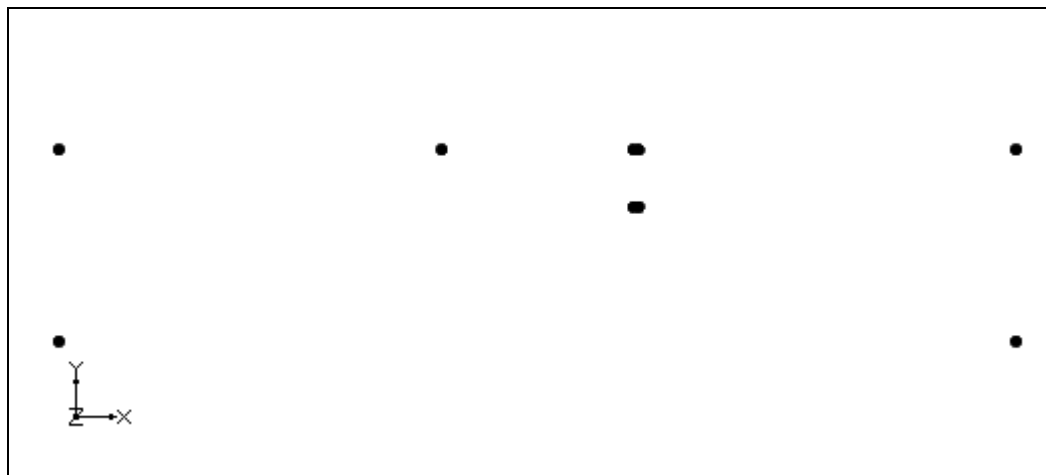



Figure 3. Points created in the profile lines coverage.

8.3 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. Click on the *Profile lines* coverage to make sure it is active.
2. Select the *Create Arc* tool .
3. Create the arcs shown in the figure below.

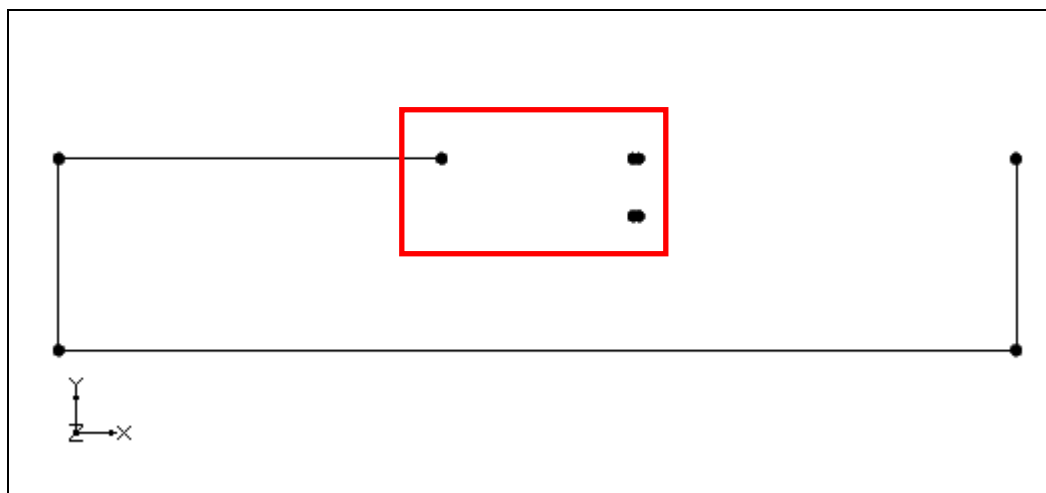




Figure 4. Arcs connecting points.

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

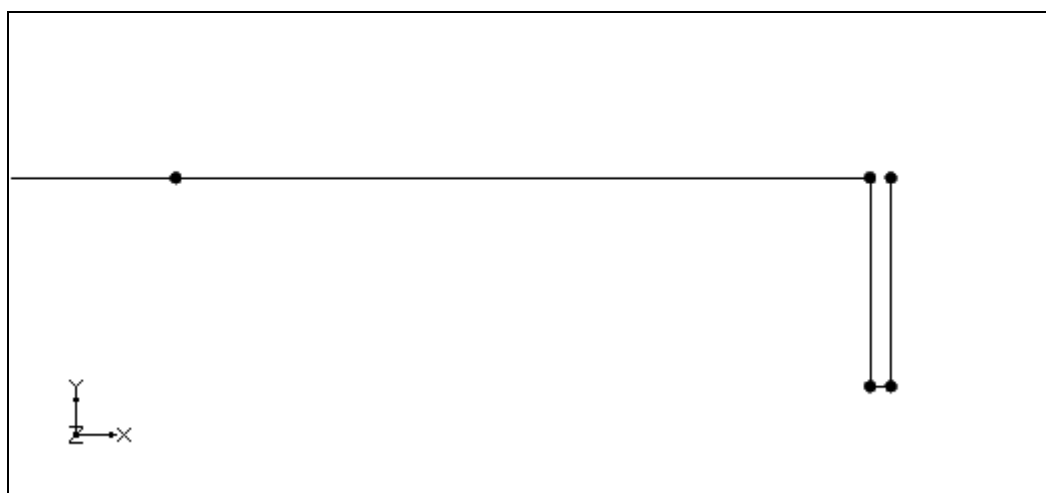




Figure 5. Arcs connecting points.

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

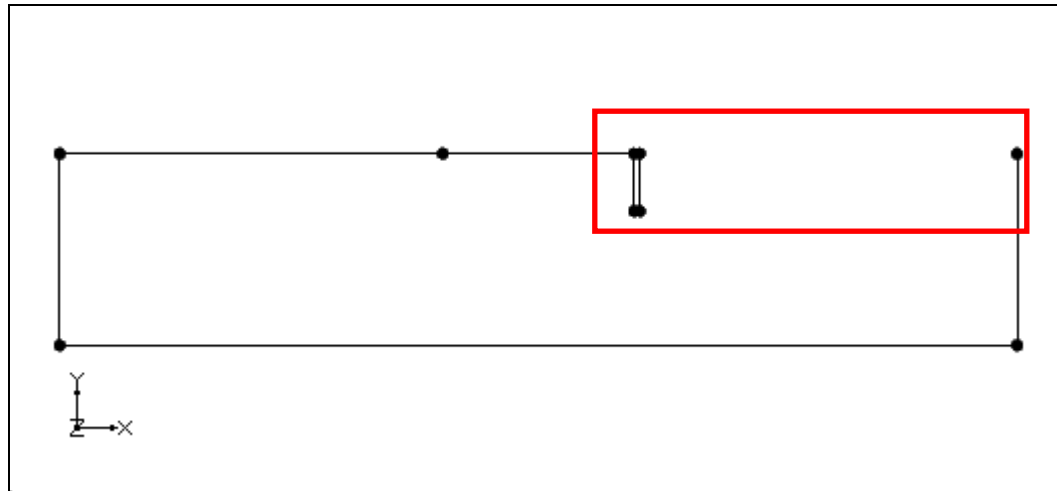


Figure 6. Arcs connecting points.

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

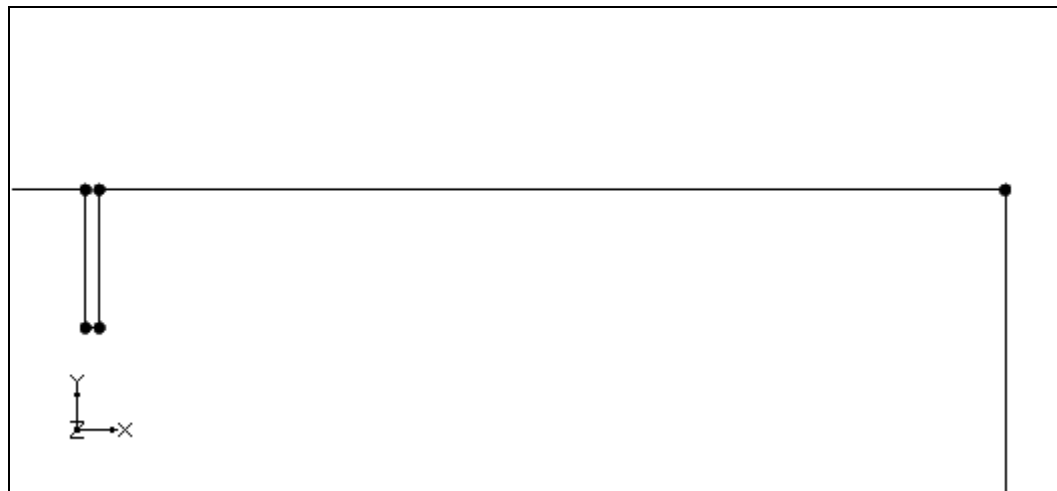



Figure 7. Arcs connecting points.

10. Now select the *Frame* macro 
11. Your model should now look like Figure 8.

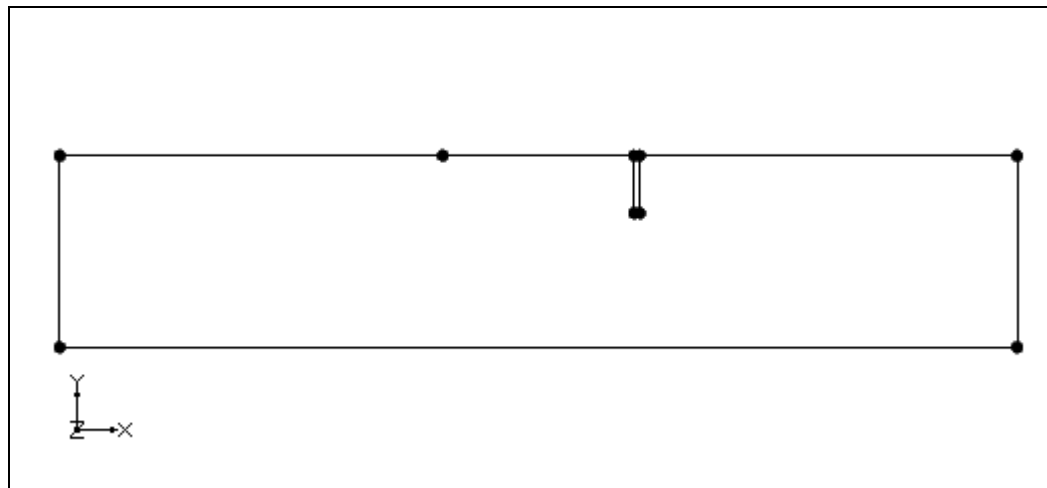


Figure 8. Arcs connecting points.

8.4 Creating the Polygons


Now that the arcs are created, we can use the arcs to build a polygon representing the region enclosed by the arcs. For problems with multiple materials, we use the polygons to assign the material zones. We only have one material in this case, but we still need to create a polygon. To build the polygon:

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

You should see a polygon appear in the background.

8.5 Assigning the Material Properties and Zones

Next we will assign the material properties and zones. To edit the material properties:

1. Click on the Materials icon  at the top of the GMS window.
2. Make sure the SEEP2D tab is selected.

At this point we would normally create a material for each of the zones in the problem and give each material a unique name and color. But since we only have one material, we can use the default name and color and simply edit the properties. 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.

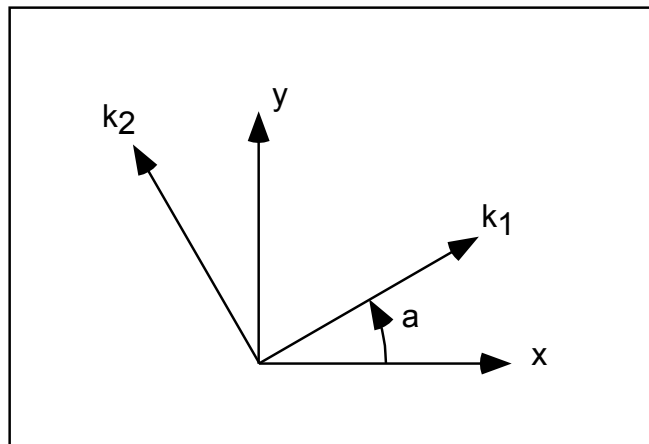


Figure 9. Definition of hydraulic conductivity angle.

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. 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.

3. Enter a value of **30** for both $k1$ and $k2$ (we will assume that the material is isotropic).
4. Select the *OK* button.

For problems with multiple materials, we would double-click on the polygons at this point and assign the material zones. But since we only have one material, the polygon inherits the zone by default and we can continue to the next section.

9 Assigning Boundary Conditions

The next step in defining the model is to assign boundary conditions to the conceptual model. 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.

9.1 Constant Head Boundaries

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

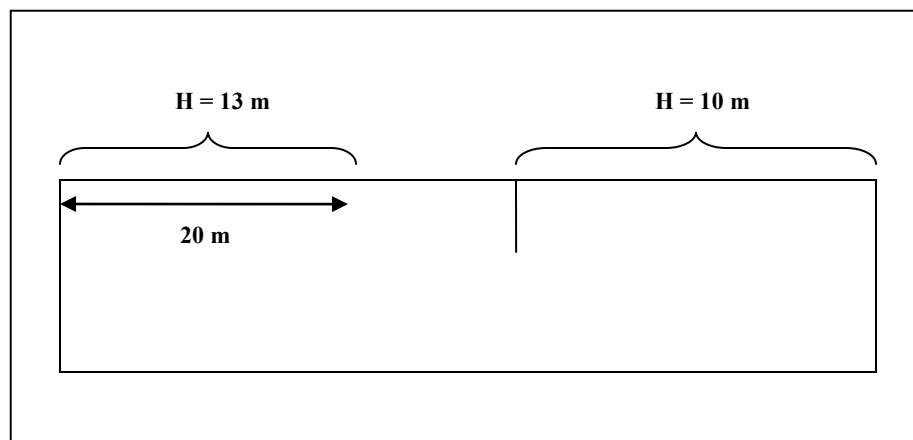



Figure 10. Constant head boundary conditions.

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 have a no-flow boundary condition by default.


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

1. Make sure the *Profile lines* coverage is selected 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-right of the model.
7. Change the *Type* to **head** and enter **10.0** in the *Head* field.

9.2 Building the Finite Element Mesh

We are now ready to build the finite element mesh used by SEEP2D. The mesh is automatically constructed from the conceptual model. The size of the elements in the mesh is controlled by the spacing of vertices along the length of the arcs making up the boundary of the model domain. Arcs are composed of both nodes and vertices. The nodes are the two end points of the arc. The vertices are intermediate points between the nodes. The gaps between vertices are called edges. At this point, all of our arcs have one edge and zero vertices. 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 .

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

8. Select the *Map->2D Mesh* macro  at the top of the GMS window (or you can select the *Feature Objects | Map → 2D Mesh* command).

You should now see a 2D mesh.


Finally, we will convert the conceptual model to the SEEP2D numerical model. This will assign all of the boundary conditions defined on the feature objects to the node-based boundary conditions required by SEEP2D.

9. Select the *Map->SEEP2D* macro  at the top of the GMS window (or you can select the *Feature Objects | Map → SEEP2D* command)


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

10 Running SEEP2D

We are now ready to save and run SEEP2D. Before running SEEP2D, make sure all of our changes have been saved to disk:


1. Select the *Save* macro  at the top of the GMS window (or select the *File | Save* command).

To run SEEP2D:

2. Select the *Run SEEP2D* macro  (or 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 *2D Mesh Data* from the list on the left of the dialog.
6. Turn off the *Nodes* and *Element Edges* options.
7. Turn on the *Mesh Boundary* option.
8. Select the *SEEP2D* tab.
9. Turn on the *Title* and *Total flow rate* options.
10. 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.