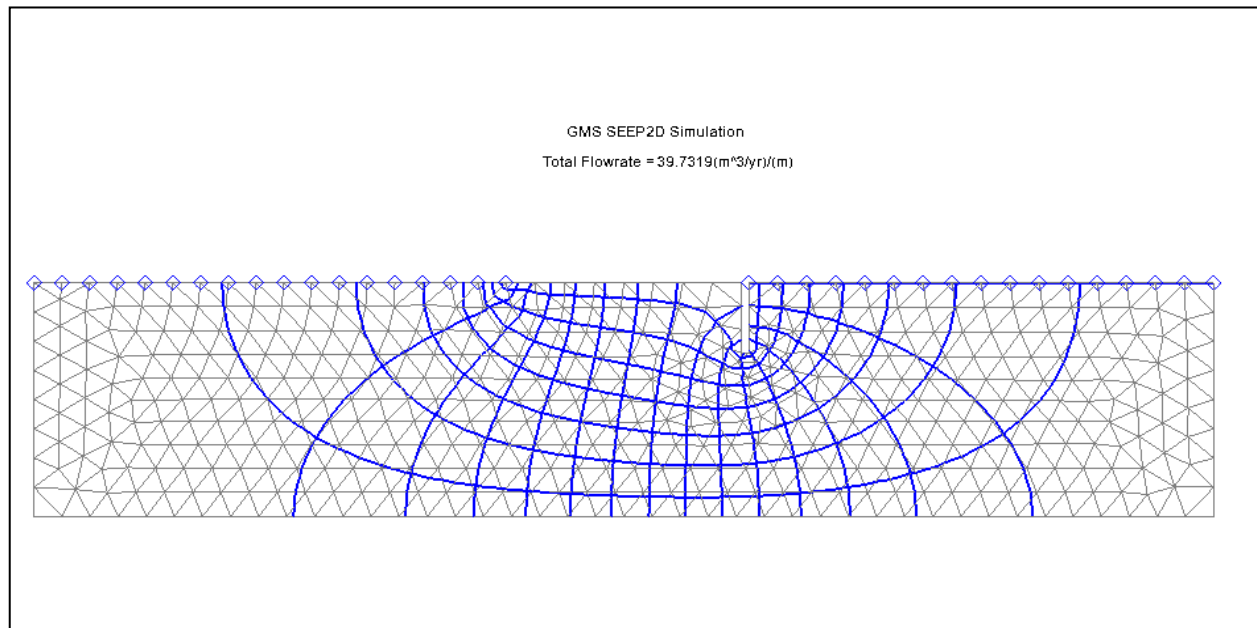


GMS 10.3 Tutorial

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

- Feature Objects

Required Components

- GIS
- Map Module
- Mesh Module
- SEEP2D

Time

- 30–45 minutes



1	Introduction	2
2	Setup	3
2.1	Program Mode.....	3
2.2	Getting Started	3
2.3	Setting the Units	4
2.4	Saving the Project	4
3	Creating the Conceptual Model Features	5
3.1	Defining a Coordinate System.....	5
3.2	Creating the Corner Points	5
3.3	Creating the Arcs.....	6
3.4	Creating the Polygons	8
3.5	Assigning the Material Properties and Zones	8
4	Assigning Boundary Conditions	9
4.1	Constant-Head Boundaries.....	9
4.2	Building the Finite-Element Mesh.....	11
5	Running SEEP2D	12
6	Conclusion	13

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 that is symmetric in the third dimension). Examples of profile models include earth dams, levees, sheet piles, etc.

The problem in this tutorial involves a partially penetrating sheet pile wall with an impervious clay blanket on the upstream side (Figure 1). The sheet pile is driven into a silty sand deposit underlain by bedrock at a depth of 10 meters.

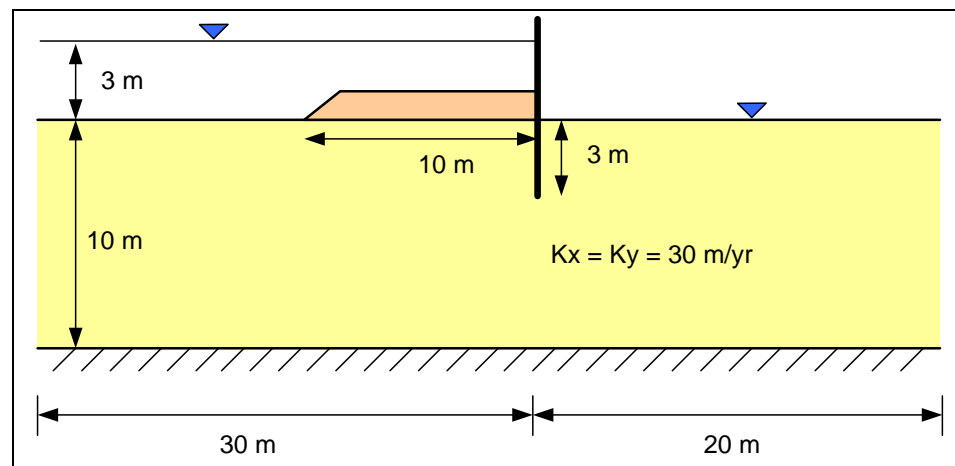


Figure 1 Confined flow problem

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.

This tutorial discusses and demonstrates creating a SEEP2D conceptual model, mapping the model to a 2D mesh, running SEEP2D, and viewing the solution.

2 Setup

2.1 Program Mode

The GMS interface can be modified by selecting a *Program Mode*. When GMS is first installed and runs, 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 can only be completed in the GMS 2D mode.

Once the mode is changed, GMS can be exited and restarted repeatedly and the interface stays in the same mode until changed. Thus, it’s only necessary to change the mode once if intending to repeatedly solve SEEP2D/UTEXAS problems. If already in GMS 2D mode, skip ahead to the Getting Started section. If not already in GMS 2D mode, do the following:

1. Launch GMS.
2. Select *Edit / Preferences...* to bring up the *Preferences* dialog.
3. Select “Program Mode” from the list on the left.
4. Select “GMS 2D” from the *Program Mode* drop-down.
5. Click **OK** to close the *Preferences* dialog.
6. Click **Yes** in response to the warning about all data being deleted. After a moment, the *New Project* dialog will appear.
7. Click **OK** to close the *New Project* dialog.
8. Select *File / Exit* to exit GMS.

2.2 Getting Started


Do the following to get started:

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

The *New Project* dialog will appear. This dialog is used to set up a GMS conceptual model. A conceptual model is organized into a set of layers or groups called “coverages”. GMS 2D allows quickly and easily defining all of the coverages needed for the conceptual model using the *New Project* dialog. Most of the options seen in the window are related to UTEXAS.


For SEEP2D models, the only coverage needed is the *Profile lines* coverage. This allows defining the geometry of the mesh, the boundary conditions, and the material zones.

3. Select *Create a new project* and enter “Sheetpile Model” as the *Conceptual model name*.
4. In the *Numerical models* section, turn off *UTEXAS*.
5. In the *Create coverages* section, select only *Profile lines*.
6. Click **OK** to close the *New Project* dialog.

A new “ Sheetpile Model” conceptual model object should appear in the Project Explorer.

2.3 Setting the Units

Before continuing, it is necessary to establish the units to be used. GMS will display the appropriate units label next to each of the input fields as a reminder to be consistent.

1. Select *Edit | Units...* to open the *Units* dialog.
2. Click the  button to the right of *Length* to bring up the *Display Projection* dialog.
3. In both the *Horizontal* and *Vertical* sections, select “Meters” from the *Units* drop-down.
4. Click **OK** to close the *Display Projection* dialog.
5. Select “yr” from the *Time* drop-down.
6. Select “kg” from the *Mass* drop-down.
7. Click **OK** to close the *Units* dialog.

2.4 Saving the Project

Before continuing, save the project to a GMS project file.

1. Select *File | Save As...* to bring up the *Save As* dialog.
2. Browse to the *\Tutorials\SEEP2D* directory.
3. Select “Project Files (*.grp)” from the *Save as type* drop-down.
4. Enter “s2con.gpr” as the *File name*.
5. Click **Save** to create the project file and close the *Save As* dialog.

It is recommended to use the **Save**  macro frequently while working on any project.

3 Creating the Conceptual Model Features

The first step in setting up the problem is to create the GIS features defining the problem geometry. Begin by entering a set of points corresponding to the key locations in the geometry. Then connect the points with lines called “arcs” to define the outline of the problem. Next, convert the arcs to a closed polygon defining the 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.

3.1 Defining a Coordinate System

Before constructing the conceptual model features, first establish a coordinate system. Use a coordinate system with the origin 30 meters upstream of the sheet pile at the top of the bedrock as shown in Figure 2.

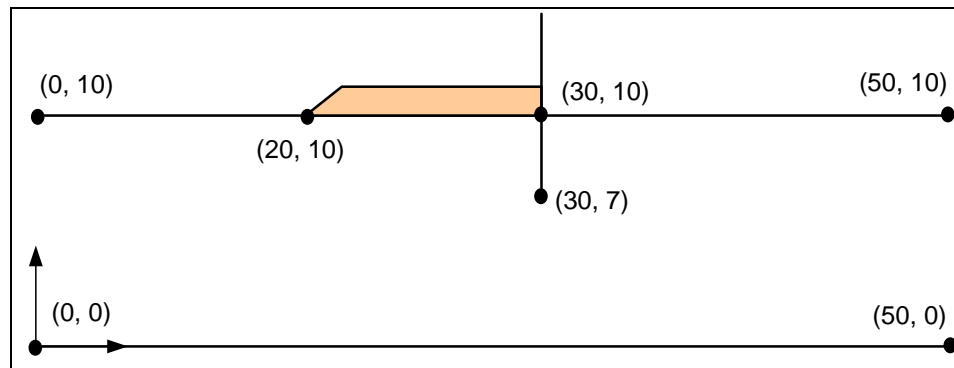





Figure 2 Coordinate system

3.2 Creating the Corner Points

It is now possible 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 **Attribute Table...** to open the *Attribute Table* dialog.
2. Change the *Feature type* to “Points”.
3. Enter into the spreadsheet the coordinates from the table to the right. The names of the points will be automatically generated by GMS as the coordinates are entered for each.
4. Click **OK** to exit the *Attribute Table* dialog.
5. Click **Frame Image**  to center the view on the new points (Figure 3).

Name	X	Y
new_point_1	0.0	0.0
new_point_2	0.0	10.0
new_point_3	20.0	10.0
new_point_4	30.0	10.0
new_point_5	30.0	7.0
new_point_6	30.3	10.0
new_point_7	30.3	7.0
new_point_8	50.0	10.0
new_point_9	50.0	0.0

Points at 30, 10 and at 30.3, 10 have been entered in order to model the sheet pile, which is about 0.3 m thick. If the coordinates of an individual point need to be edited, use the **Select Points/Nodes**  tool and double-click on the desired node to bring up the *Attribute Table* dialog for that point. Points can be deleted by selecting the desired points and pressing the *Delete* key on the keyboard or by selecting *Edit* | **Delete**.

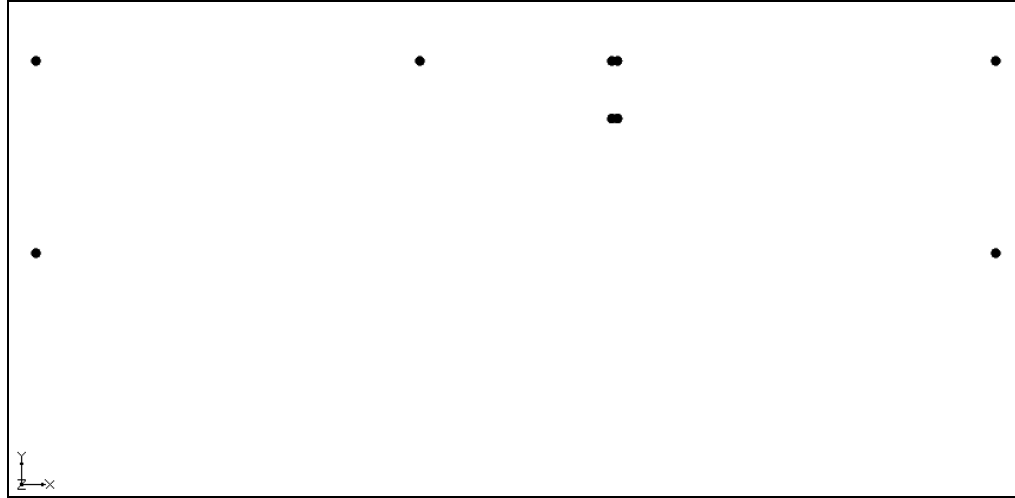




Figure 3 Points created in the profile lines coverage.

3.3 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 it active.
2. Using the **Create Arc**  tool, create the arcs indicated with arrows in Figure 4.

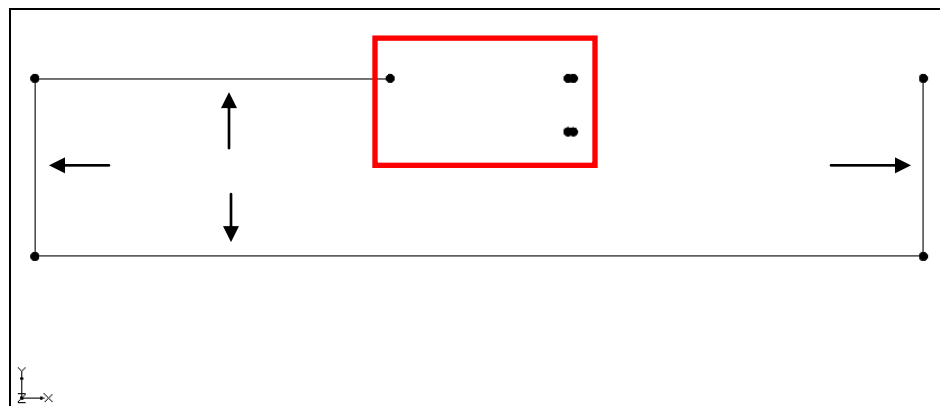




Figure 4 Initial arcs

3. Using the **Zoom**  tool, drag a box similar to that shown by the red box in Figure 4.
4. Using the **Create Arc**  tool, create the arcs indicated with arrows in Figure 5.

Notice that there is no arc created at the top of the sheet pile.

5. Click **Frame Image** .

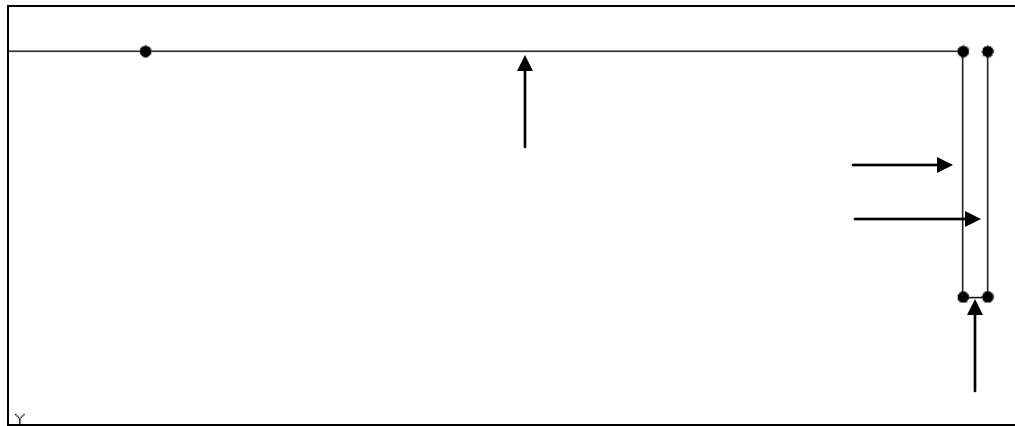



Figure 5 Arcs including the sheet pile

6. Using the **Zoom**  tool, drag a box as shown by the red box in Figure 6.

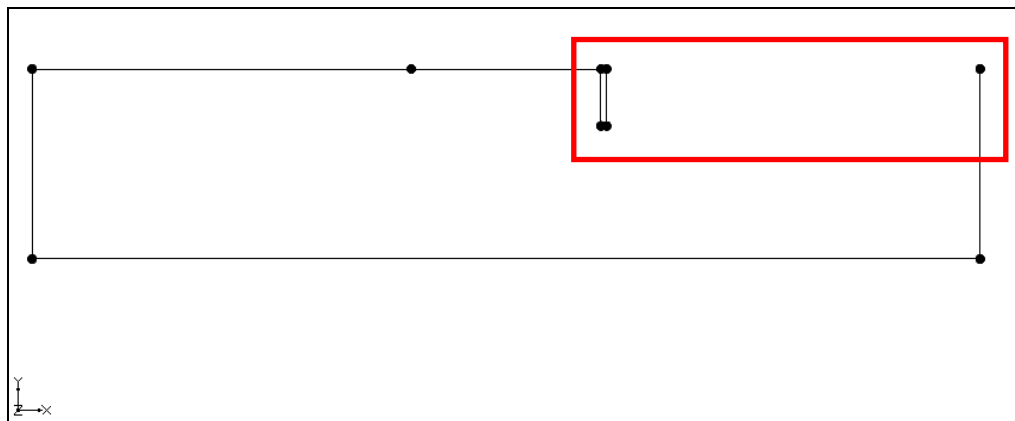


Figure 6 Zoom in to this area

7. Create the arc indicated by the arrow in Figure 7. Notice there is no arc across the top of the sheet pile.

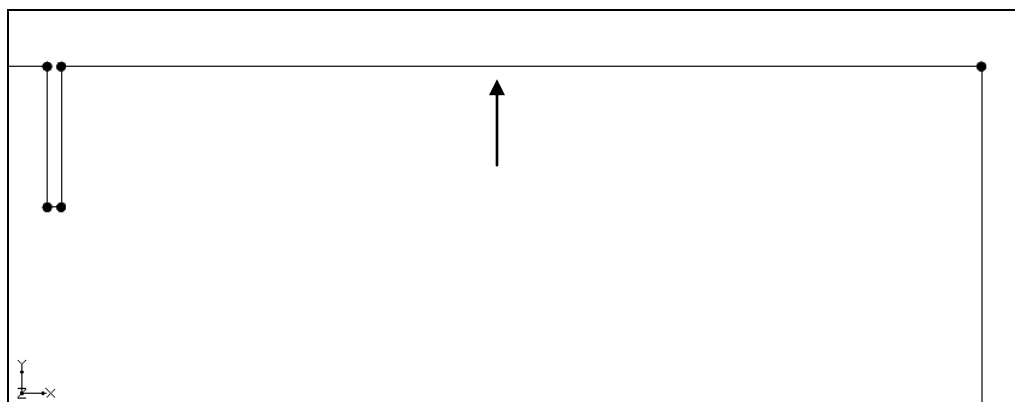


Figure 7 Create the final arc

8. Click the **Frame Image**  macro. The model should now look like Figure 8.

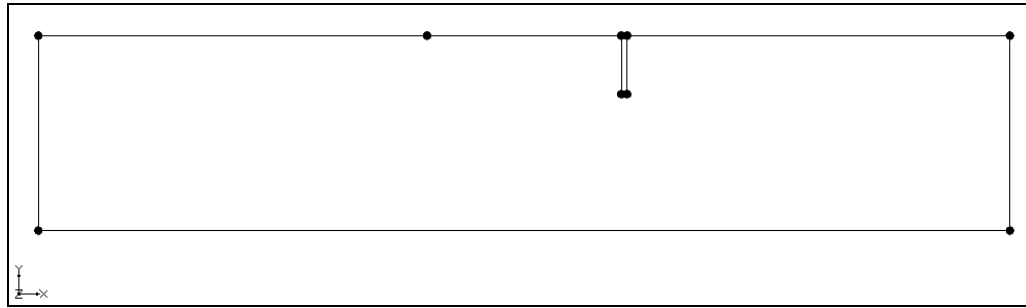


Figure 8 Final view with all the arcs created

3.4 Creating the Polygons

Now that the arcs are created, use the arcs to build a polygon representing the region enclosed by the arcs. For problems with multiple materials, use the polygons to assign the material zones. There is only one material in this case, but it is still necessary to create a polygon. To build the polygon, do the following:

1. Select *Feature Objects* / **Build Polygons**.

The polygon will have a gray background (Figure 9).

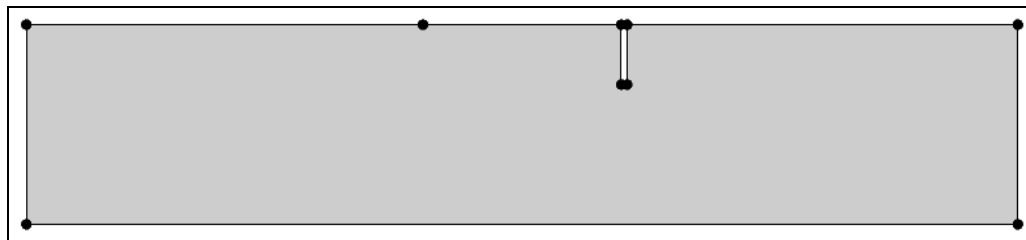



Figure 9 The polygon is indicated by the gray background

3.5 Assigning the Material Properties and Zones

Next, assign the material properties and zones. To edit the material properties, do the following:

1. Click **Materials**  at the top of the GMS window to bring up the *Materials* dialog.
2. Select the *SEEP2D* tab.

At this point, a material for each of the zones in the problem would normally be created and given a unique name and color. But since there is only one material, 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 10.

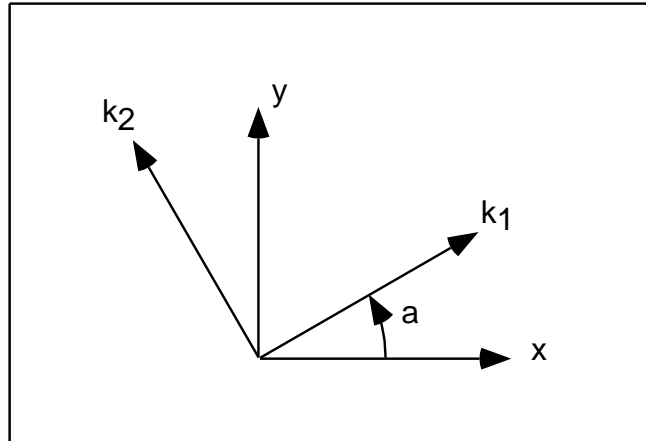


Figure 10 Definition of hydraulic conductivity angle

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 “30” for both $k1$ and $k2$ (assume that the material is isotropic).
4. Click **OK** to close the *Materials* dialog.

For problems with multiple materials, double-click on the polygons at this point and assign the material zones. Since there is only one material in this example, the polygon inherits the zone by default.

4 Assigning Boundary Conditions

The next step in defining the model is to assign boundary conditions to the conceptual model. For the problem being modeled, 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.

4.1 Constant-Head Boundaries

The constant-head boundary conditions for the problem are shown in Figure 11.

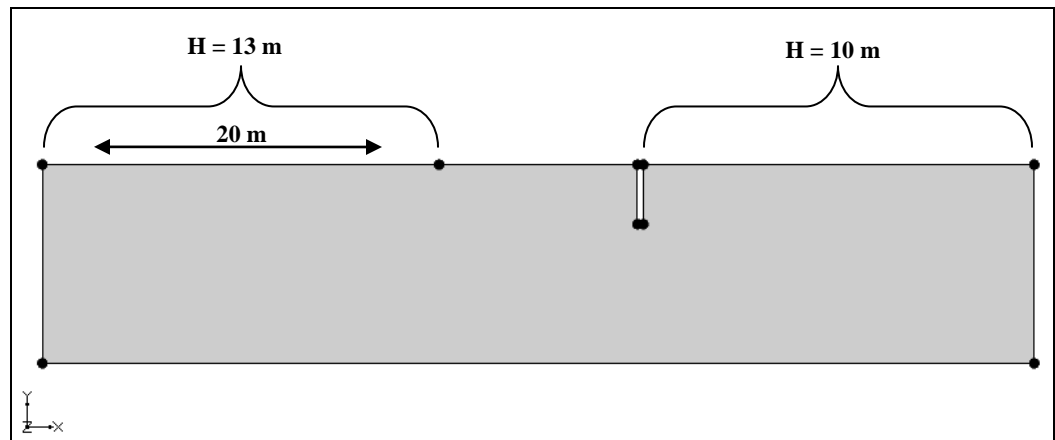




Figure 11 Constant-head boundary conditions

The region on the left in Figure 11 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.

Enter the constant-head boundary conditions for the arc on the left:

1. Select the “ Profile lines” coverage in the Project Explorer to make it active.
2. Using the **Select Arcs**  tool, double-click on the top left arc of the model to open the *Attribute Table* dialog.
3. Select “head” from the drop-down in the *Type* column.
4. Enter “13.0” in the *Head (m)* field.
5. Click **OK** to exit the *Attribute Table* dialog. Notice that the arc is now blue, indicating a constant-head boundary condition (Figure 12).

Enter the constant-head boundary conditions for the arc on the right:

6. Repeat steps 2–5 for the top right arc, entering “10.0” in the *Head (m)* field. Notice this arc has also changed blue.

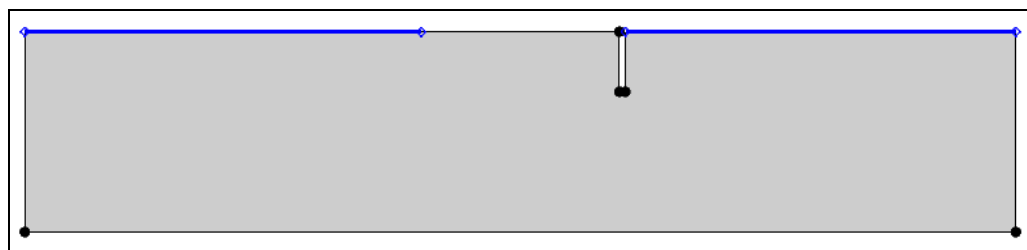




Figure 12 Constant-head boundaries marked in blue

4.2 Building the Finite-Element Mesh


It is now possible 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 the arcs have one edge and zero vertices.


To subdivide the arcs to create appropriately sized edges, do the following:

1. After selecting the **Select Arcs**  tool, select *Edit* | **Select All** to select all the arcs.
2. Select *Feature Objects* | **Redistribute Vertices...** to bring up the *Redistribute Vertices* dialog.
3. Select “Specified spacing” from the *Specify* drop-down.
4. Enter “1.2” for the *Average spacing*.
5. Click **OK** to close the *Redistribute Vertices* dialog.
6. To see the vertices, switch to the **Select Vertices**  tool.

To make the vertices always visible, do the following:

7. Click **Display Options**  to bring up the *Display Options* dialog.
8. Select “Map Data” from the list on the left.
9. On the *Map* tab, turn on *Vertices* and click **OK** to close the *Display Options* dialog.

At this point, construct the mesh.

10. Click the **Map** → **2D Mesh**  macro at the top of the GMS window (or select *Feature Objects* | **Map** → **2D Mesh**).

A 2D mesh should now be visible (Figure 13).

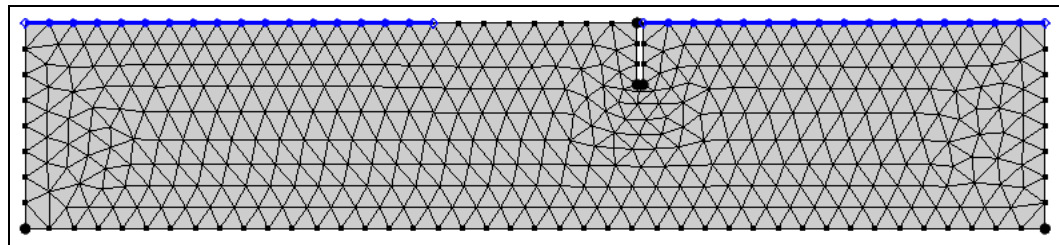



Figure 13 The 2D mesh created from the vertices



Finally, 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.

11. Click the **Map** → **SEEP2D**  macro at the top of the GMS window (or select *Feature Objects* | **Map** → **SEEP2D**)

A set of blue symbols should appear along the constant-head boundaries indicating that the boundary conditions have been assigned.

5 Running SEEP2D

Now save the changes and run SEEP2D.

1. Click **Save** .
2. Click the **Run SEEP2D**  macro (or select *SEEP2D* | **Run SEEP2D**) to open the *SEEP2D* model wrapper dialog.
3. When the solution is finished, turn on *Read solution on exit* and click **Close** to close the *SEEP2D* model wrapper dialog.

GMS automatically reads in the SEEP2D solution. The solution should be visible as a flow net. The flow net consists of equipotential lines (total head contours) and flow lines (Figure 14).

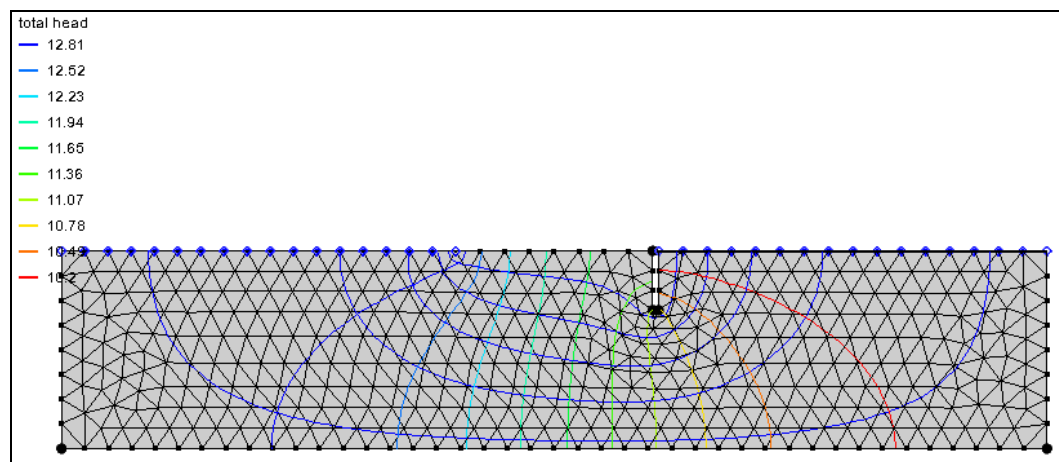



Figure 14 Total head contours are visible

To turn on the display of the total flow through the cross section, do the following:

4. Click **Display Options**  to bring up the *Display Options* dialog.
5. Select “2D Mesh Data” from the list on the left.
6. On the *2D Mesh* tab, turn off *Nodes* and *Element Edges*, and turn on *Mesh Boundary*.

7. On the *SEEP2D* tab, turn on *Title* and *Total flow rate*.
8. Click **OK** to close the *Display Options* dialog. The display should appear similar to Figure 15.

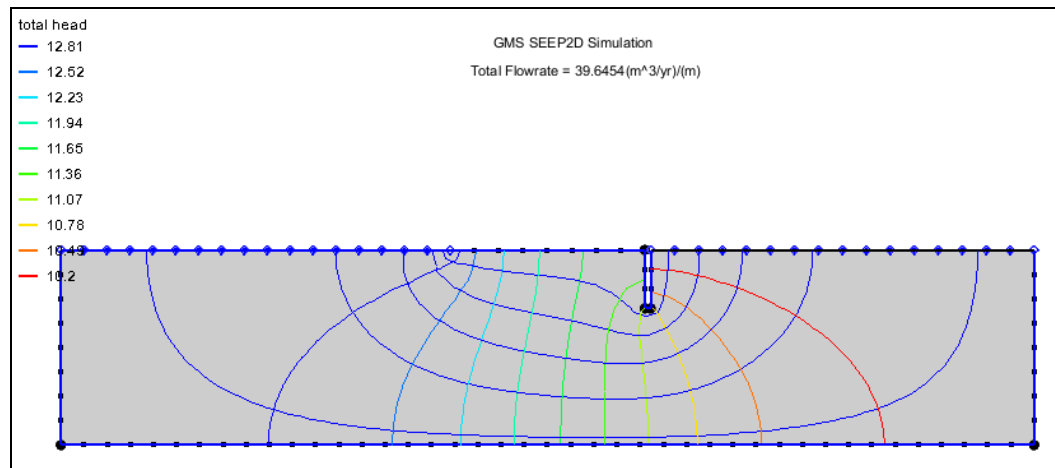


Figure 15 The final view

6 Conclusion

This concludes the “SEEP2D – Sheet Pile” tutorial. The following topics were discussed and demonstrated:

- SEEP2D is a 2D, finite-element seepage model.
- It is possible to use a conceptual model to create a 2D mesh.