# GMS 7.0 TUTORIALS

# SEEP 2D – Unconfined

## 1 Introduction

SEEP2D can be used for both confined and unconfined problems. The steps involved in performing a SEEP2D simulation for the unconfined condition are described in this tutorial.

It is recommended, but not required, that you complete the other SEEP2D tutorial (confined model) before doing this tutorial.

### 1.1 Contents



 

## 1.2 Outline

This is what you will do:

1. Create a SEEP2D conceptual model.
2. Map the model to a 2D mesh.
3. Define conditions for both a saturated and unsaturated zone.
4. Convert the model to SEEP2D and run SEEP2D.

## 1.3 Required Modules/Interfaces

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

- Mesh
- Map
- SEEP2D

# 2 Description of Problem

The problem we will be solving in this tutorial is shown in Figure 1. The problem consists of an earth dam with anisotropic soil and a low permeability core in the interior.

 

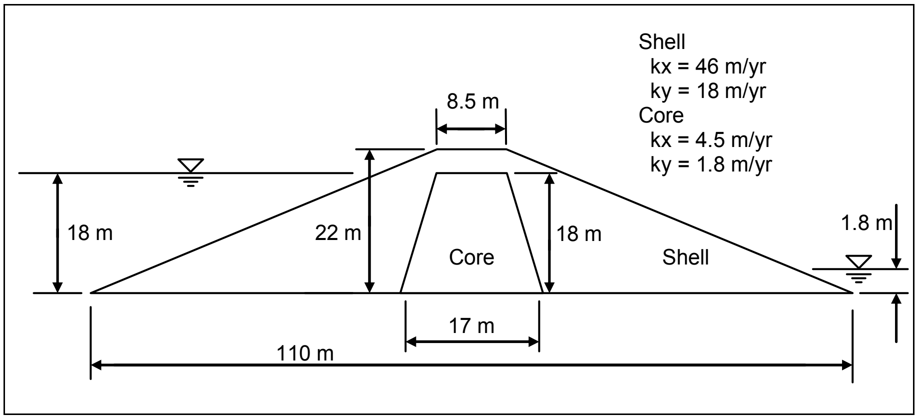


*Figure 1. Unconfined Flow Problem.*

# 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 **N** for the *Force* units.
6. Select the *OK* button.

# 5 Creating the Mesh

The first step in defining the model is to create the finite element mesh. The mesh will be constructed using the *Map* → *2D Mesh* command in the *Map* module. With this approach, we simply need to define the boundaries of the problem domain and GMS automatically constructs all of the interior nodes and elements.

 

## 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 at the lower left corner of the dam as shown in Figure 2.

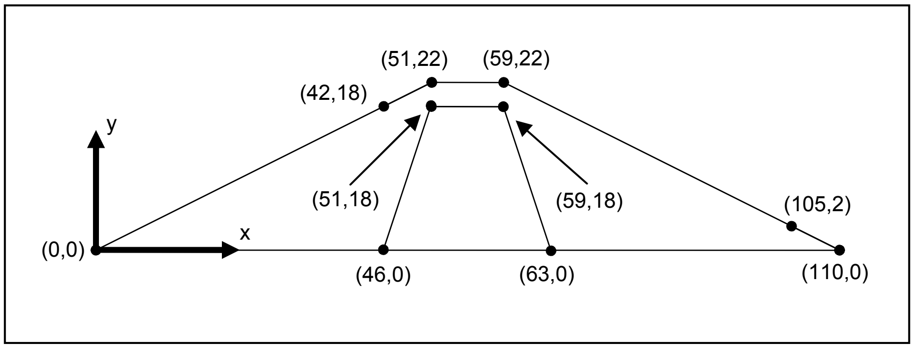


*Figure 2. Coordinate System.*

We will begin the mesh construction by creating some points at key locations in the mesh.

## 5.2 Creating a Coverage

Before we construct the feature objects defining the boundary of the mesh, we must first create a 2D mesh coverage.

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 **SEEP2D**.
3. Change the *Model* to **SEEP2D/UTEXAS**
4. Uncheck the *UTEXAS* option and click *OK*.
5. Right-click on the **SEEP2D** conceptual model  and select the *New Coverage* command from the pop-up menu.
6. Change the name to **seep2d**.
7. Turn on the following properties:
   - Meshing Options
   - Head
   - Exit Face
8. Select the *OK* button.

 

## 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 **59.0** and **22.0** respectively.

4. Create another point by clicking on the graphics window and change the x and y coordinates to **110.0** and **0.0** respectively.

5. Now select the *Frame* macro .

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

| X | Y |
|---|---|
| 42 | 18 |
| 51 | 22 |
| 51 | 18 |
| 59 | 18 |
| 46 | 0 |
| 63 | 0 |
| 105 | 2 |

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 3 (without the labels).

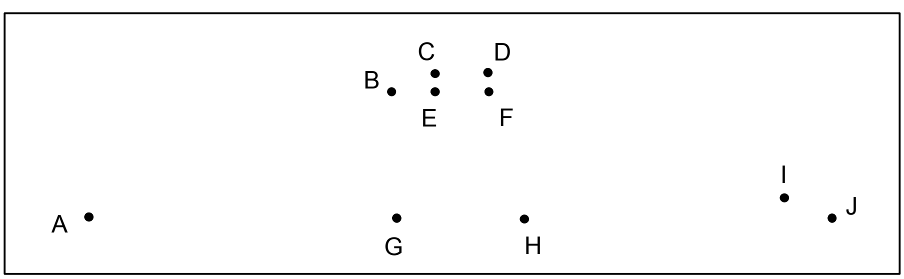


*Figure 3. Points Created with the Create Point Tool*

 

## 5.4 Creating the Arcs

Now that the corner nodes are created, the next step is to create the arcs defining the boundaries of both the shell and the core of the dam. This can be accomplished as follows:

1. Select the *Create Arc* tool .
2. Create a series of arcs around the perimeter of the dam by clicking on the following points in order: A-B-C-D-I-J-H-G-A.
3. Create the remaining arcs around the core boundary by clicking on the following points in order: G-E-F-H.

## 5.5 Redistributing Vertices

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. 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 all of the arcs by dragging a box that encloses all of the arcs.
3. Select the *Feature Objects | Redistribute Vertices* command.
4. Select the *Specified spacing* option.
5. Enter a value of **2.5** for the spacing.
6. Select the *OK* button.
7. Switch to the *Select Vertices* tool .

Notice the vertex spacing. The vertices are turned off by default but become visible when the *Select Vertices* tool is active. You can also turn them on in the display options so they are always visible.

In this particular problem most of the head loss will occur in the core of the dam. To more accurately model the core we will make the mesh more dense in the core of the dam.

8. Switch back to the *Select Arcs* tool .
9. Select the arcs that make up the core of the dam (G-E, E-F, F-H, H-G). Hold down the shift key to select multiple arcs at once.

 

10. Select the *Feature Objects | Redistribute Vertices* command.
11. Select the *Specified spacing* option.
12. Enter a value of **1.0** for the spacing.
13. Select the *OK* button.
14. Switch to the *Select Vertices* tool .

## 5.6 Creating the Polygons

Now we are ready to create the polygons defining the two material zones. Simply creating the arcs does not create the polygons. We must explicitly create the polygons using the arcs.

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

## 5.7 Assigning the Material Types

Now that the polygons are created, we need to define a material type to be associated with each polygon.

1. Select the *Edit | Materials* command.
2. Create a new material by clicking the *New* button .
3. Rename the materials to **"Shell"** and **"Core"**.
4. Click *OK*.
5. Choose the *Select Polygons* tool .
6. Double-click anywhere inside the polygon representing the shell of the dam.
7. Make sure the material associated with this polygon is the **Shell** material.
8. Select the *OK* button.

Now we will do the same for the other polygon.

9. Double-click anywhere inside the polygon representing the core of the dam.
10. Make sure that **Core** is selected as the material for this polygon.
11. Select the *OK* button.
12. Click anywhere on the *Graphics Window* to unselect the polygon.

 

### 5.8 Constructing the Mesh

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

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

## 6 Initializing the SEEP2D Solution

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

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

2. Update the value of the *Unit weight of water* to be **9810.0** (our current units are N/m^3).

Two methods are provided in SEEP2D for computing the relative conductivity in the unsaturated zone: the linear front method and the Van Genuchten method. With the linear front method, the relative conductivity varies linearly from the saturated value down to a user specified minimum at a given negative pressure head. With the Van Genuchten method, the Van Genuchten parameters are used to define the variation of the relative conductivity in the unsaturated zone. Both methods are described in detail in the SEEP2D Primer. We will use the linear front option.

3. Select the *Saturated/Unsaturated with linear front* option.

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

## 7 Assigning Material Properties

The next step is to define material properties. Each element in the mesh is assigned a material. The materials were automatically assigned to the elements using the polygons when the mesh was constructed.

1. Select the *Edit | Materials* command.

2. For the material entitled **Shell**, enter a value of **46** for *k1*, **18** *k2,* **-0.3** for *ho*, and **0.001** for *kro*.

3. For the material entitled **Core**, enter a value of **4.5** for *k1*, **1.8** *k2*, **-1.2** for *ho*, and **0.001** for *kro*.

4. Select the *OK* button.

## 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 three types of boundary conditions: (1) no-flow (flow is parallel to the boundary), (2) specified head, and (3) exit face. 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 is to assign the specified head and exit face boundary conditions.

## 8.1 Specified Head Boundary Conditions

The specified head boundary conditions for our mesh are shown in Figure 4.

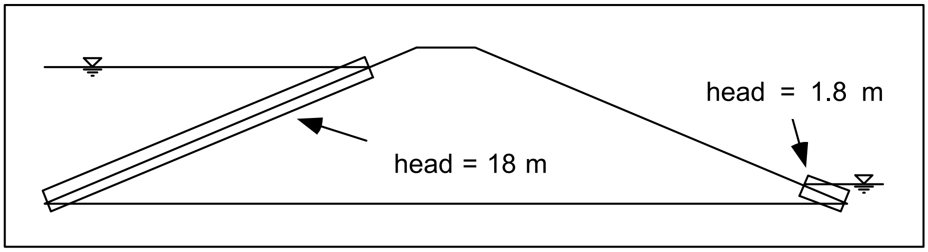


*Figure 4. The Specified Head Boundary Conditions.*

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

1. Choose the *Select Arcs* tool .
2. Double-click on the arc on the left face of the embankment. Change the *Type* to **head** and enter **18.0** in the *Head* field.
3. Select the *OK* button.
4. Repeat this process with the arc on the left side of the model and enter **1.8** in the *Head* field.

## 8.2 Exit Face Boundary Conditions

The remaining type of boundary condition to enter is the exit face definition. Since we are modeling an unconfined condition, SEEP2D will iterate to find the location of the phreatic surface. To guide the iteration process, we need to mark all of the nodes on the mesh where the phreatic surface may exit as exit face nodes. The region of the mesh where the phreatic surface may exit is illustrated in Figure 5.

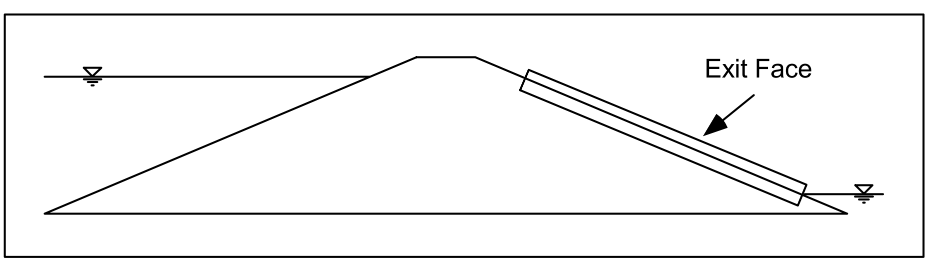


*Figure 5. The Exit Face Boundary Condition.*

To define the exit face boundary conditions:

1. Choose the *Select Arcs* tool .
2. Double-click on the arc on the upper right face of the embankment. Change the *Type* to **exit face** and exit the dialog.

### 8.3 Converting the Conceptual Model

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.

1. In the *Project Explorer* right-click on the *seep2d* coverage 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\s2unc**.
3. Enter **dam1** for the file name.
4. Select the *Save* button.

## 10 Running SEEP2D

We are now ready to run SEEP2D.

1. Select the *SEEP2D* | *Run SEEP2D* command.

At this point SEEP2D is launched in a new window. When the solution is finished, select the *Close* button. GMS will automatically read in the SEEP2D solution when this dialog is closed.

## 11 Viewing the Solution

You should now see a plot of the flow net. To better view the flow net, we will turn off the display of nodes and elements.

1. Select the *Display Options* button .

2. Select the *2D Mesh Data* item from the list on the left.

3. Turn off the *Nodes* and *Element edges* options.

4. Turn on the *Mesh boundary* option.

5. Select the *OK* button.

Note that there are only a small number of flow lines. GMS determines the number of flow lines to display based on how closely spaced the equipotential lines are in one of the materials. By default, the interval is computed based on the Shell material. To base the number of flow lines on the Core material:

6. Select the *Display Options* button.

7. Select the *2D Mesh Data* item.

8. Select the *SEEP2D* tab.

9. In the *Base Material* pull-down list, make sure that the **Core** material is selected.

10. Verify that the *Phreatic surface* option is turned on.

11. Select the *OK* button.

You should now see flow lines through your model as well as the phreatic surface.

# 12 Conclusion

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

- SEEP2D can do 2D confined and unconfined flow modeling.

- You can display the phreatic surface in GMS.

 