# GMS TUTORIALS

# *UTEXAS – Dam with Seepage*

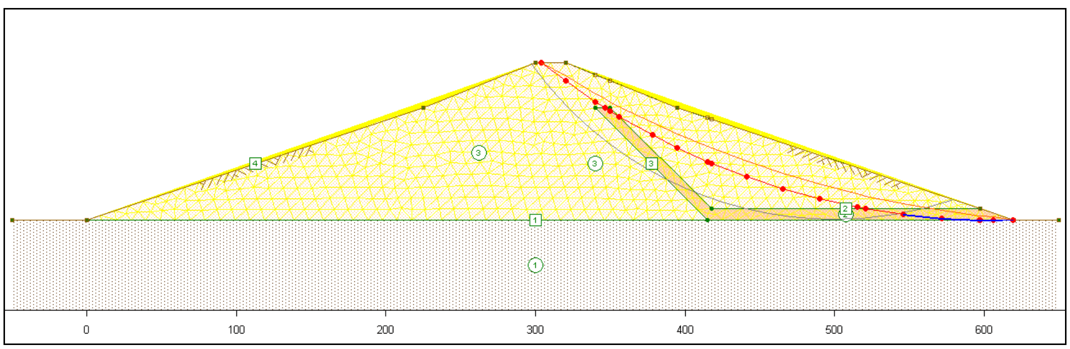


*Figure 1. Dam with Seepage.*

## 1 Introduction

This tutorial illustrates how to build an integrated SEEP2D/UTEXAS model in GMS. This tutorial is similar to tutorial number three in the UTEXAS tutorial manual ("UTEXPREP4 Preprocessor For UTEXAS4 Slope Stability Software" by Stephen G. Wright, Shinoak Software, Austin Texas, 2003.).

For many slopes, the distribution of pore pressures throughout the slope has a significant effect on the stability of the slope. The SEEP2D model can be used to quickly and easily model seepage and pore pressure distributions. The UTEXAS4 and SEEP2D interfaces in GMS are integrated in such a manner that the output from SEEP2D can automatically be used as input to UTEXAS4. This coupled analysis process is illustrated in this tutorial.

The *UTEXAS – Embankment on Soft Clay* tutorial explains more about UTEXAS and provides a good introduction. The *SEEP2D – Unconfined* tutorial explains more about SEEP2D. You may wish to complete these tutorials before starting this tutorial.

## 1.1 Contents



## 1.2 Outline

In this tutorial, we'll be examining the earth dam shown in Figure 1. We will use both UTEXAS and SEEP2D together to analyze the stability of the dam.

This is what you will do:

1. Create a SEEP2D / UTEXAS conceptual model.
2. Create a SEEP2D mesh.
3. Run SEEP2D.
4. Run UTEXAS.
5. View the results.

 

### 1.3 Required Modules/Interfaces

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

- Map
- SEEP2D
- UTEXAS

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

## 2 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.

## 3 Set 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 **ft** for the *Length* units.
3. Select **lb** for the *Force* units.
4. Select the *OK* button.

## 4 Save the GMS Project File

Before continuing, we will save what we have done so far to a GMS project file:

1. Select the *Save* button. This brings up the *Save As* dialog.
2. Navigate to the **tutfiles/UTEXAS/dam seepage** folder.
3. Enter a name for the project file (ex. "**embank-utexas.gpr**") and select the *Save* button.

You may wish to select the *Save* button occasionally to save your work as you continue with the tutorial.

# 5 Create the Embankment

The model we will create will be an integrated SEEP2D/UTEXAS model. The input for both the SEEP2D and UTEXAS models is entered using a common conceptual model, allowing both simulations to use the same geometry.

## 5.1 Create the Conceptual Model

1. Right-click in the *Project Explorer* and select the *New* | *Conceptual Model* menu command.

2. In the *Conceptual Model Properties* dialog, change the options as shown in the figure below and click *OK*.

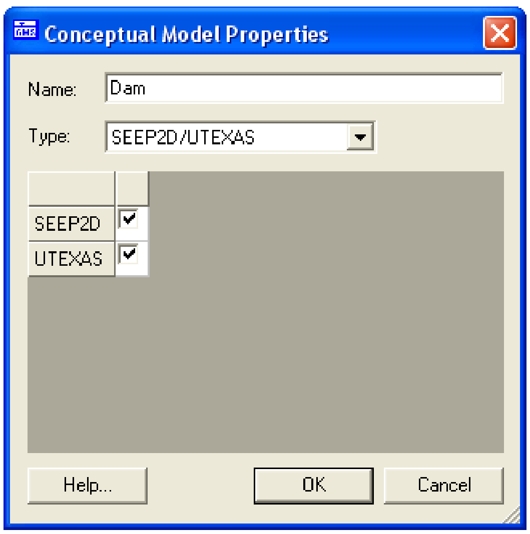


*Figure 2. The Conceptual Model Properties dialog.*

## 5.2 Create a New Coverage

We'll create the coverage for the arcs and polygons which define the embankment.

1. In the *Project Explorer,* right-click on the *Dam* conceptual model you just created and select the *New Coverage* command from the pop-up menu.

2. In the *Coverage Setup* dialog, change the *Coverage Name* to **Profile**.

3. In the *Coverage Setup* dialog, turn on the following options:

   - *Head*
   - *Single Head Value for Arcs (this one is turned on automatically with Head)*
   - *Exit Face*

 

- *Meshing options*

The dialog should appear as shown in Figure 3 below.

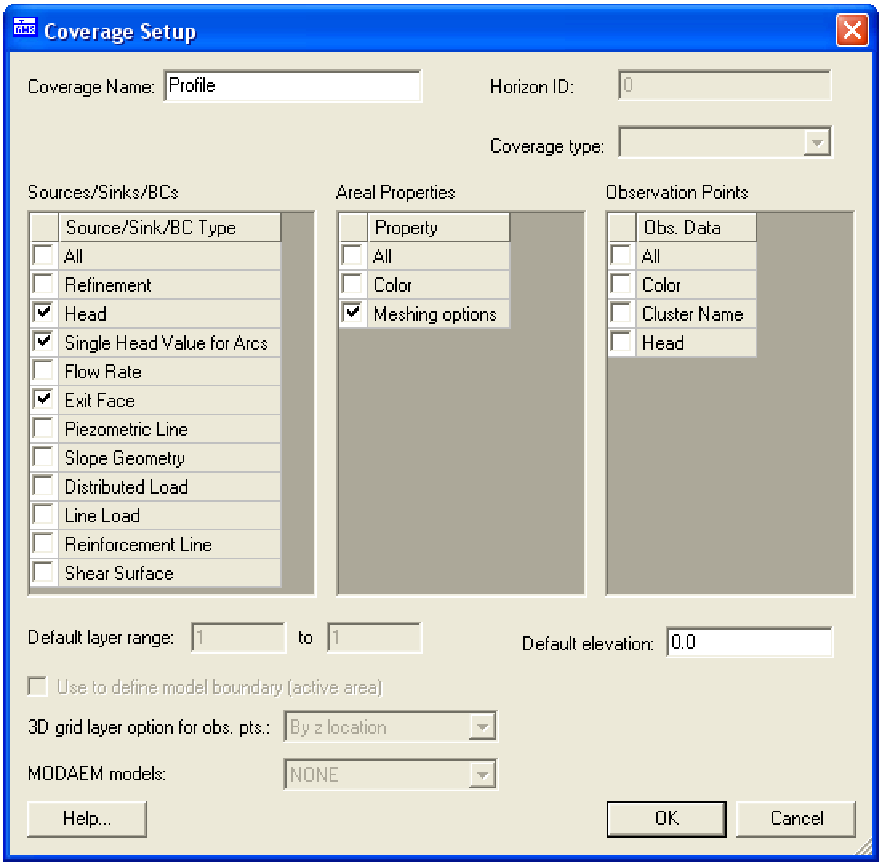


*Figure 3. Coverage Setup dialog.*

4. Click *OK* to exit the dialog.

## 5.3 Create the Points

The XY coordinates of the points defining the geometry of the zones of the dam were determined in advance. We'll enter them now.

1. In the *Project Explorer*, right-click on the **Profile** coverage and select the *Attribute Table* command from the pop-up menu.

2. In the dialog, change the *Feature type* to **Points**.

3. Make sure the *Show point coordinates* option is turned **on**.

4. Enter the X and Y coordinates show in the table below. If you are viewing this tutorial electronically, you can copy and paste these values into the GMS spreadsheet. Don't worry about the Z coordinates. They are ignored by both SEEP2D and UTEXAS.

 

| x | y |
|---|---|
| -50 | -21 |
| -50 | 0 |
| 0 | 0 |
| 225 | 75 |
| 300 | 105 |
| 320 | 105 |
| 340 | 75 |
| 350 | 75 |
| 395 | 75 |
| 415 | 0 |
| 417.5 | 7.5 |
| 597.5 | 7.5 |
| 620 | 0 |
| 650 | -21 |
| 650 | 0 |

5. Verify that the dialog looks like the figure below and click *OK*.

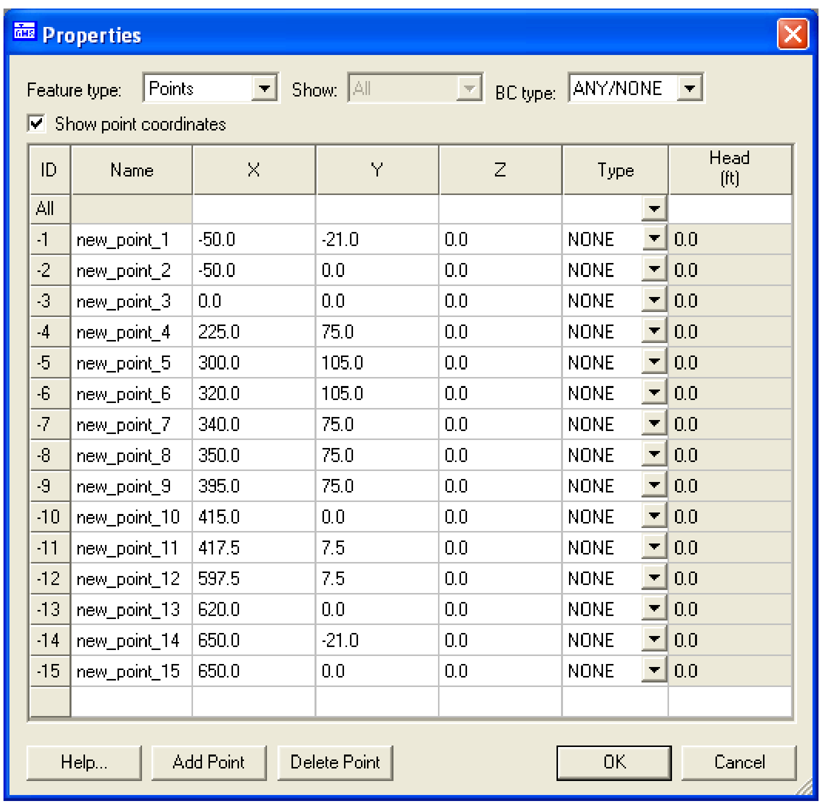


*Figure 4. Entering the Point Coordinates*

6. Now select the *Frame* macro .

You should now see the points on the screen.

## 5.4 Create the Arcs and Polygons

Now we need to connect the points to form arcs.

1. Select the *Create Arc* tool .
2. Hold down the *Shift* key. This makes it so that you can create multiple arcs continuously without having to stop and restart at each point. Double-click whenever you want to stop creating arcs.
3. Using Figure 5 below as a guide, click on the points to connect them with arcs to create the slope.
4. Select the *Feature Objects | Build Polygons* menu command.

Your model should now look like the figure below:

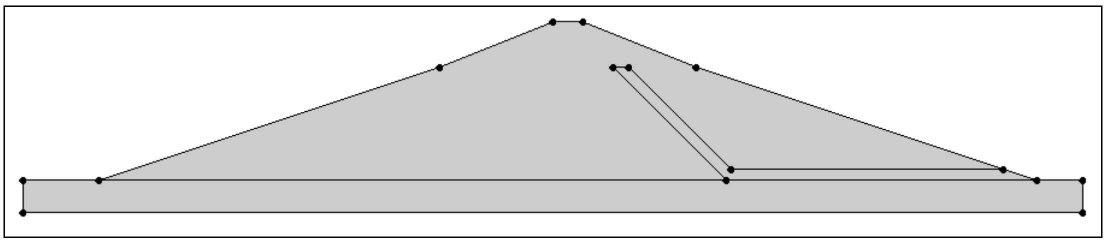

*Figure 5. Arcs Connecting Points*

## 5.5 Assigning Head Arcs

We'll assign specified head to the outer arcs on the upstream (left) side of the dam. The head will correspond to the level of the water in the reservoir.

1. Switch to the *Select Arcs* tool.
2. Double-click on one of the arcs on the upstream (left) side of the dam, as shown in the Figure 6 below.

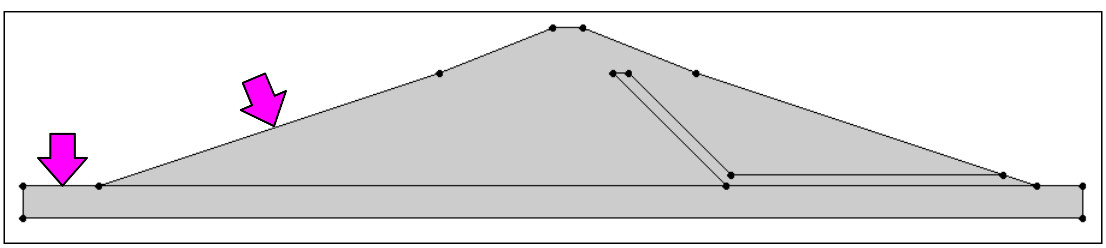

*Figure 6. Assigning the Head Arcs*

3. In the *Properties* dialog change the type to **head**.
4. Change the *Head* value to **75.** Leave the options for stage 2 head alone. When done, click *OK* to exit the dialog.

5. Repeat steps 2 – 4 for the other arc.

As you assign the head attribute to the arcs, GMS fills in the area above the arcs up to the head elevation to represent the water, as shown in Figure 7 below. You can turn this display option off or change the fill color by accessing the *Display Options* dialog.

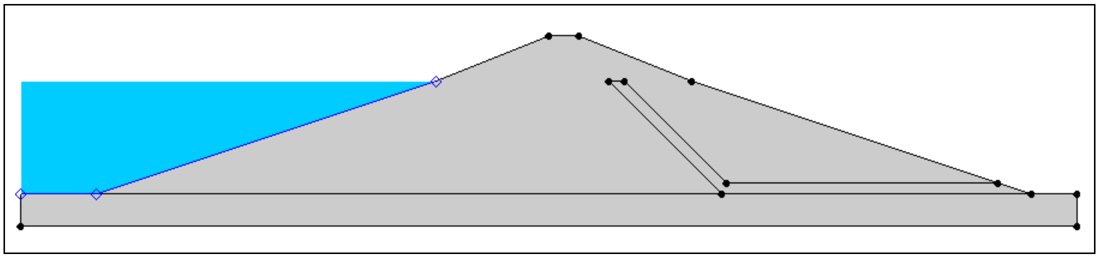

*Figure 7. Filling the Area Above the Head Arcs.*

## 5.6 Redistributing Vertices

We will be creating a mesh for SEEP2D and the mesh density will be determined by the vertex spacing on the arcs. We need to redistribute the arc vertices so that our mesh will be sufficiently dense.

1. Switch to the *Select Arc* tool.
2. Select the *Edit* | *Select All* menu command.
3. Select the *Feature Objects* | *Redistribute Vertices* command.
4. Change the *Spacing* to **10** and click *OK*.

It is not immediately evident that anything has changed but we have indeed added vertices. To see them:

5. Switch to the *Select Vertices* tool.

Now the new vertices should be displayed.

## 5.7 Assigning the Exit Face Arcs

We need to specify the downstream (right) side of the dam as an exit face boundary condition. On an exit face boundary, SEEP2D sets the head to be equal to the elevation of the nodes along the boundary.

1. Switch to the *Select Arc* tool.
2. Double-click one of the three arcs highlighted in Figure 8 below.

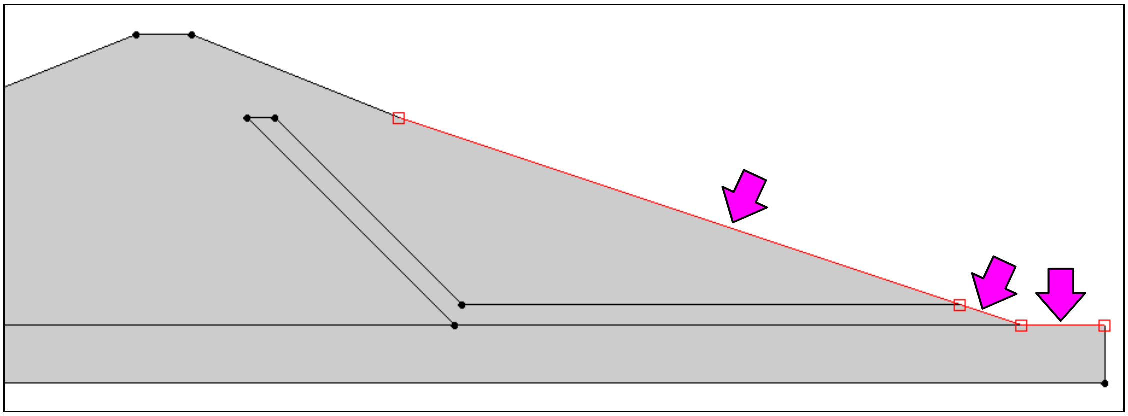

*Figure 8. The Exit Face Arcs.*

3. In the *Properties* dialog change the type to **exit face** and click *OK*.

4. Repeat steps 2 – 3 for the other two arcs.

# 6 Material Properties – SEEP 2D

Each zone in the dam consists of a different type of soil material. We need to create the materials and then assign them to the polygons. We'll set up the SEEP2D material properties now and the UTEXAS properties later.

1. Select the *Edit* | *Materials* menu command.

2. Click on **material_1** in the spreadsheet and rename it "**Foundation Rock**".

3. Change *k1* to **25** and *k2* to **5**. These values correspond to horizontal and vertical hydraulic conductivity, respectively, in feet per day. Don't change the other values.

To create the next material:

4. Type "**Embankment**" in the *Name* column of the blank row at the bottom of the spreadsheet.

5. Change *k1* to **100** and *k2* to **20** in the row with ID equal 2.

To create the final material:

6. Type "**Drain**" in the *Name* column of the blank row at the bottom of the spreadsheet.

7. Change *k1* to **1000** and *k2* to **1000** in the row with ID equal to 3.

8. When you are finished, click *OK* to exit the dialog.

 

# 7 Assign Materials to Polygons

Now that the materials are all defined, we can assign the appropriate material to each polygon.

1. Select the *Select Polygons* tool.
2. Double-click on the embankment which is zone 2 in Figure 9 below.

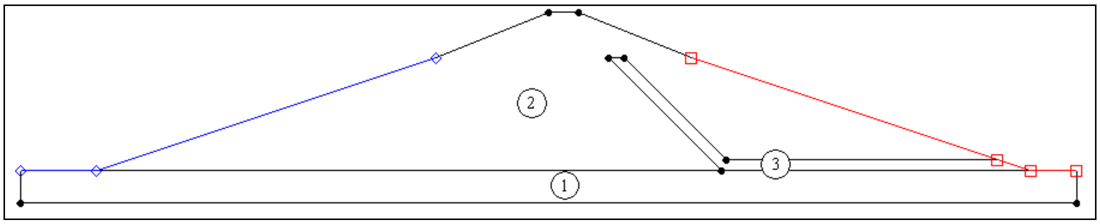


*Figure 9. Polygons and their Material IDs.*

3. In the *Properties* dialog, change the *Material* to **Embankment** and click *OK*.
4. Repeat steps 2 – 3 for the **Drain**, which is zone 3. (The polygon for zone 1 should already be set to **Foundation Rock**.)
5. Click anywhere in the graphics window to unselect the polygon.

# 8 Create the 2D Mesh

We're ready to create the mesh for SEEP2D.

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

After several seconds you should see the mesh appear. Let's turn the mesh nodes off so the display doesn't look so cluttered.

2. Select the *Display Options* button.
3. Select the *2D Mesh Data* item from the list on the left.
4. Turn off the *Nodes*.
5. Click *OK*.

# 9 Map → SEEP2D

Now we need to transfer the arc attributes in the conceptual model to the mesh. First, we will initialize the SEEP2D simulation.

### 9.1 Initializing SEEP2D

1. In the *Project Explorer,* click on the *2D Mesh Data* folder.
2. Select the *SEEP2D* | *New Simulation* menu command.
3. Make sure the *Model type* is set to **Saturated/Unsaturated with linear front**.
4. Leave the other settings at their defaults and click *OK* to exit the dialog.

### 9.2 Map → SEEP2D

Next, we will transfer the SEEP2D boundary conditions from the conceptual model to the mesh.

1. In the *Project Explorer,* right-click on the *Dam* conceptual model and select the *Map To* | *SEEP2D* command from the pop-up menu.

At this point, you should see some symbols appear on the mesh nodes on the upstream and downstream face indicating that the boundary conditions have been assigned to the mesh nodes.

## 10 Run SEEP2D

We're ready to save the project and run SEEP2D.

### 10.1 Save the Project

1. Click the *Save* button.

### 10.2 Run SEEP2D

1. In the *Project Explorer,* click on the *2D Mesh Data* folder.
2. Select the *SEEP2D* | *Run SEEP2D* menu command.

GMS launches the model wrapper. It may take several seconds for SEEP2D to complete the analysis.

3. When the model wrapper shows that SEEP2D terminated successfully, select the *Close* button.

GMS automatically reads in the solution and contours the results. The blue lines you see are flow lines.

 

# 11 Display Options

Let's change the display options so we can view the solution better.

1. Select the *Display Options* button.
2. Turn **off** the *Element edges* option.
3. Select the *SEEP2D* tab.
4. Turn **off** the *Head BC* and *Exit face BC* options.
5. Change the *Base* material to **Foundation Rock**. (this controls how many flow lines are displayed)
6. Click *OK*.

You should be able to see the head contours and flow lines as shown in Figure 10 below. The thick black line is the piezometric line computed by SEEP2D. Notice the flow lines concentrated in the drain portion of the dam.

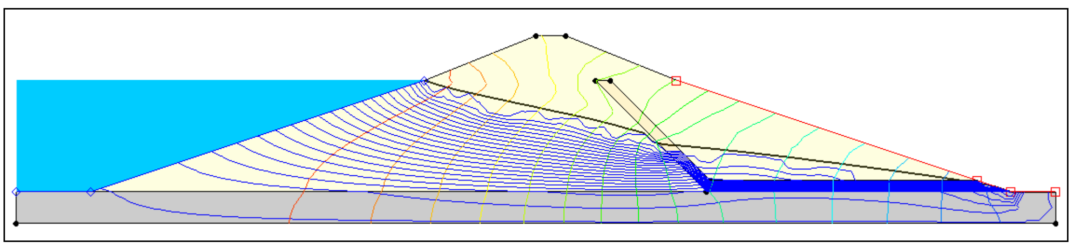

*Figure 10. SEEP2D Solution.*

We'll turn off the flow lines now so that we can see and work with the model easier.

7. Select the *Display Options* button.
8. Make sure *2D Mesh Data* is selected from the list on the left and the *SEEP2D* tab is visible on the right.
9. Turn **off** the *Flow Lines* option.

# 12 Material Properties – UTEXAS

Now that we have a set of pore pressures computed by SEEP2D, we are ready to continue to the second phase of our analysis and set up the data required for the UTEXAS4 slope stability analysis. Since the site geometry is already defined using the feature objects in the conceptual model, we mainly need to enter the shear strength properties for the soils and define some UTEXAS4 analysis options.

Earlier we specified the SEEP2D material properties. Now we need to specify the UTEXAS material properties.

 

1. Select the *Edit* | *Materials* menu command.

This brings up the *Material Properties* dialog.

2. On the right side of the dialog select the *UTEXAS* tab.

3. Make sure the *Show stage 1* option is **on**, and the *Show stage 2* option is **off**.

4. Enter the following material properties.

| Material Properties – UTEXAS Stage 1 | | | | | |
|---|---|---|---|---|---|
| Id | Name | Unit Weight | Shear Strength | | |
| | | | Method | Cohesion c | Friction angle |
| 1 | Foundation Rock | 160 | Very Strong | NA | NA |
| 2 | Embankment | 128 | Conventional | 200 | 34 |
| 3 | Drain | 130 | Conventional | 0 | 37 |

5. In rows 2 and 3, change the *Pore Water Pressure Method* to **Interpolate Pressures**. With this method, UTEXAS uses the pore pressures computed by SEEP2D.

Look at the *Pore Pressure Data Set Stage 1* entry for row 2 (you may need to expand the column). GMS automatically found the SEEP2D pore pressure data set and listed it in this row. To see how you would manually change this:

6. Click on the ... button in the *Pore Pressure Data Set Stage 1* column for row 2 and 3. This brings up the *Select Data Set* dialog.

7. Select the data set named "*pore pressure*" under the *dam profile (SEEP2D)* solution (it should already be selected by default).

8. Click *OK* to exit the *Select Data Set* dialog.

# 13 Analysis Options

The only thing left to do before we save and run the model is to set the UTEXAS analysis options. We will do an automatic search for the critical factor of safety using Spencer's Method and circular failure surfaces.

1. In the *Project Explorer*, right-click on the **UTEXAS** model and select the *Analysis Options* command from the pop-up menu.

2. Change the options to match those shown in the dialog below.

3. When you're finished, click *OK* to exit the dialog.

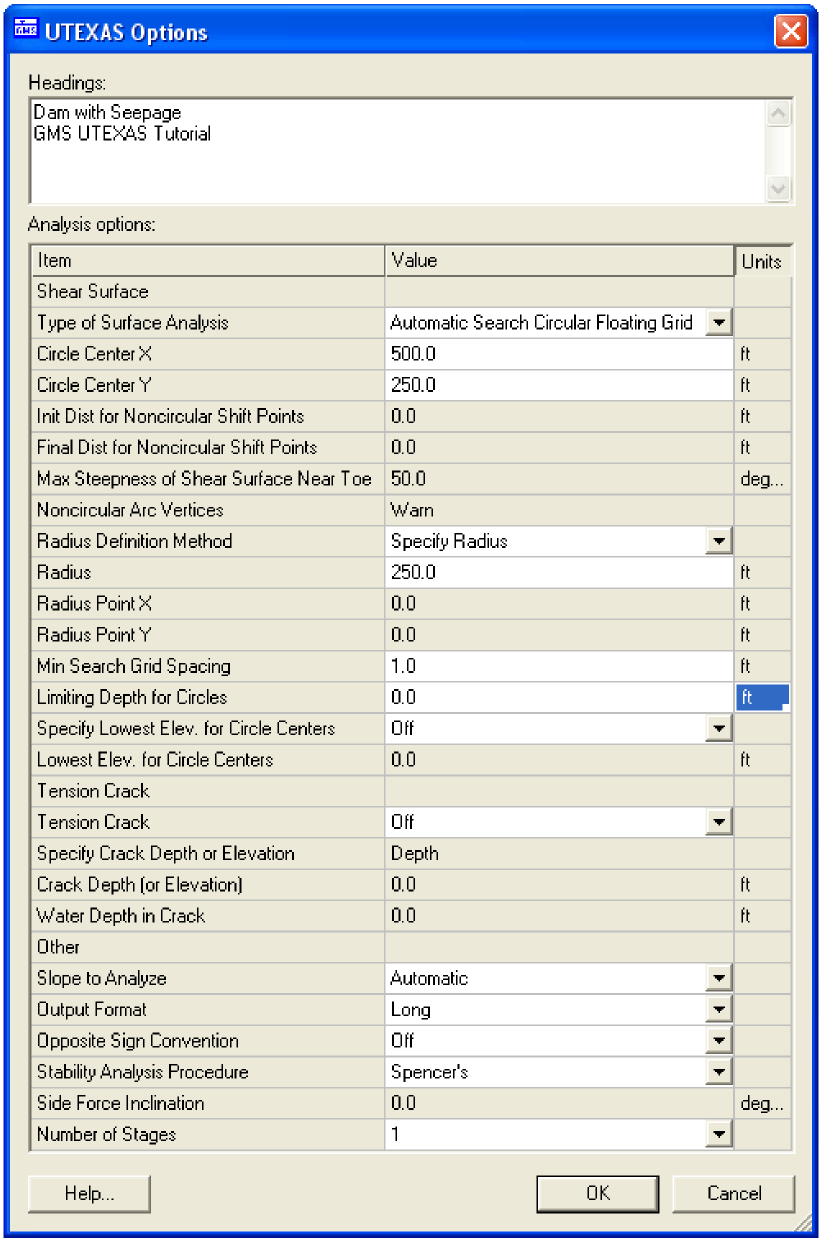


*Figure 11. UTEXAS Options*

At this point you should see the starting circle displayed.

# 14 Export the Model

We're ready to export the model for use in UTEXAS.

1. In the *Project Explorer*, right-click on the **UTEXAS** model  and select the *Export* command from the pop-up menu.

2. If necessary, locate and open the directory entitled **tutfiles\UTEXAS\dam seepage** (you should already be there).

3. Change the *File name* to **UTseepage** and click *Save*.

# 15 Run UTEXAS

Now that we've saved the UTEXAS input file, we're ready to run UTEXAS.

1. In the *Project Explorer*, right-click on the **UTEXAS** model and select the *Launch UTEXAS4* command from the pop-up menu. This should bring up the UTEXAS4 program.

2. In UTEXAS4, select the *Open File* button.

3. Change the *Files of type* to **All Files (*.*)**.

4. Locate the **UTseepage.utx** file you just saved (in the **tutfiles\ UTEXAS\dam seepage**) folder and open it.

5. When UTEXAS4 finishes, look at the things mentioned in the *Errors, Warnings* window, then close the window.

# 16 Read the Solution

Now we need to read the UTEXAS solution.

1. In the *Project Explorer*, right-click on the **UTEXAS** model and select the *Read Solution* command from the pop-up menu.

2. Locate and open the file named **UTseepage.out**.

You should now see a line representing the critical failure surface, and the factor of safety.

# 17 Conclusion

This concludes the tutorial. Here are some of the key concepts in this tutorial:

- SEEP2D and UTEXAS can be used together in GMS to do a slope stability analysis.

- The UTEXAS material properties can refer to a SEEP2D solution to define the pore pressures.

- An arc whose type is *head* serves for both the SEEP2D head and the UTEXAS distributed load.