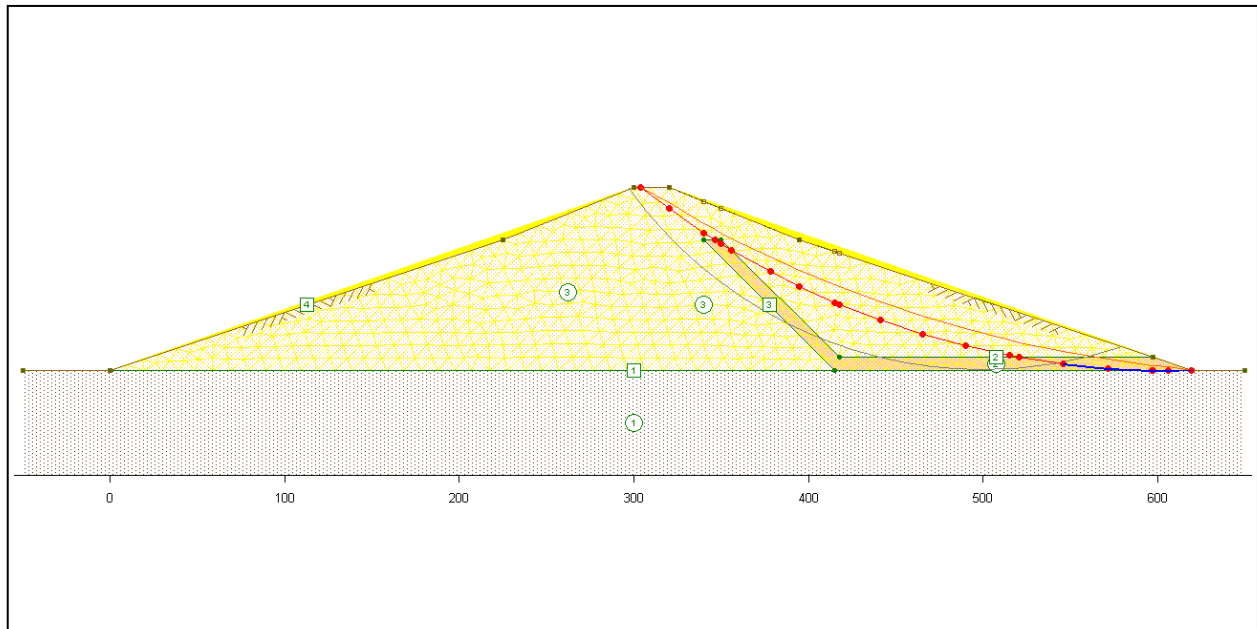*GMS 9.2 Tutorial*

UTEXAS – Dam with Seepage

Use SEEP2D and UTEXAS to model seepage and slope stability of a earth dam

Objectives

Learn how to build an integrated SEEP2D/UTEXAS model in GMS.

Prerequisite Tutorials	Required Components	Time
• None	• GIS • Map • Mesh • SEEP2D • UTEXAS	• 30-60 minutes

AQUAVEO™

1   Contents

2   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 to the GMS/UTEXAS interface. The *SEEP2D – Unconfined* tutorial explains more about SEEP2D. You may wish to complete these tutorials before starting this tutorial.

2.1   Outline

In this tutorial, we'll be examining the earth dam shown on page 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.

3   Program Mode

This tutorial assumes that we are operating in the GMS 2D mode. 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.

4   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 organized into a set of layers or groups called *coverages*. We can use the same conceptual model to prepare the input to both

SEEP2D and UTEXAS. Since this is a simple UTEXAS model and the pore pressures will be inherited from the SEEP2D solution, we only need a *Profile lines* coverage.

2. Change the *Conceptual model name* to **Combined Model**.

3. Make sure that both the *SEEP2D* and *UTEXAS* options are turned **on** 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.

5   Set 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 Units select the "…" button, and change the units for both vertical and horizontal to **feet**.

3. Select **lb** for the *Force* units.

4. Select the *OK* button.

6   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 **Tutorials/UTEXAS/dam seepage** folder.

3. Enter a name for the project file (ex. "**dam -combined.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.

7   Create the Conceptual Model Features

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.

7.1   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 lines** coverage and select the *Attribute Table* command from the pop-up menu.

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

3. 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

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

ID	Name	X	Y	Z	Type	Head (ft)
All					▾	
-1	new_point_1	-50.0	-21.0	0.0	NONE ▾	0.0
-2	new_point_2	-50.0	0.0	0.0	NONE ▾	0.0
-3	new_point_3	0.0	0.0	0.0	NONE ▾	0.0
-4	new_point_4	225.0	75.0	0.0	NONE ▾	0.0
-5	new_point_5	300.0	105.0	0.0	NONE ▾	0.0
-6	new_point_6	320.0	105.0	0.0	NONE ▾	0.0
-7	new_point_7	340.0	75.0	0.0	NONE ▾	0.0
-8	new_point_8	350.0	75.0	0.0	NONE ▾	0.0
-9	new_point_9	395.0	75.0	0.0	NONE ▾	0.0
-10	new_point_10	415.0	0.0	0.0	NONE ▾	0.0
-11	new_point_11	417.5	7.5	0.0	NONE ▾	0.0
-12	new_point_12	597.5	7.5	0.0	NONE ▾	0.0
-13	new_point_13	620.0	0.0	0.0	NONE ▾	0.0
-14	new_point_14	650.0	-21.0	0.0	NONE ▾	0.0
-15	new_point_15	650.0	0.0	0.0	NONE ▾	0.0

Figure 1.   Entering the point coordinates.

> 5. Now select the *Frame* macro 🔍.

You should now see the points on the screen.

7.2   Create the Arcs and Polygons

Now we need to connect the points to form arcs and build the polygons.

> 1. Click on the Profile lines coverage to make sure it is active.
>
> 2. Select the *Create Arc* tool ⌇.
>
> 3. 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.
>
> 4. Using Figure 2 below as a guide, click on the points to connect them with arcs to create the slope.
>
> 5. Select the *Feature Objects | Build Polygons* menu command.

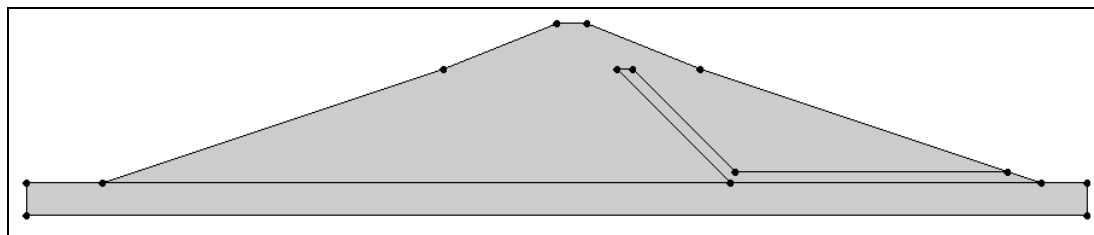Your model should now look like the figure below:

Figure 2.   Arcs connecting points.

7.3   Assign the Specified Head Arcs

Next 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 3 below.
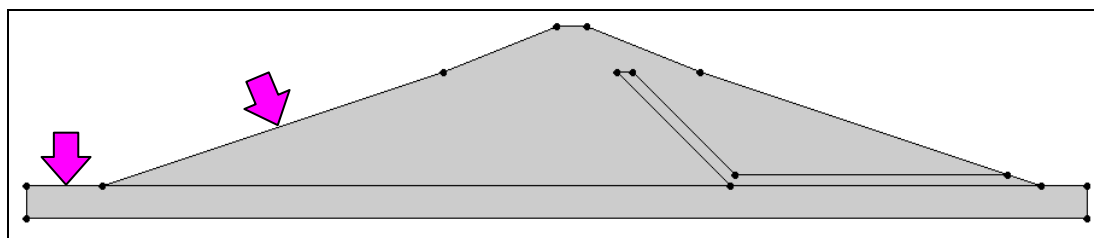

Figure 3.   Assigning the head arcs.

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 4 below. You can turn this display option off or change the fill color by accessing the *Display Options* dialog.
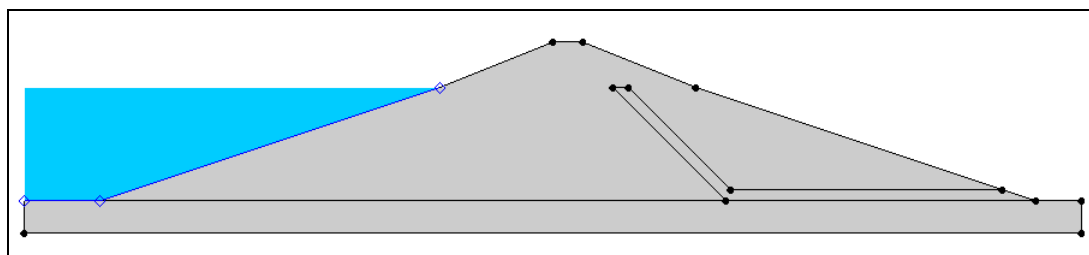

Figure 4.   Filling the area above the head arcs.

7.4   Assign the Exit Face Arcs

Next we will 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.

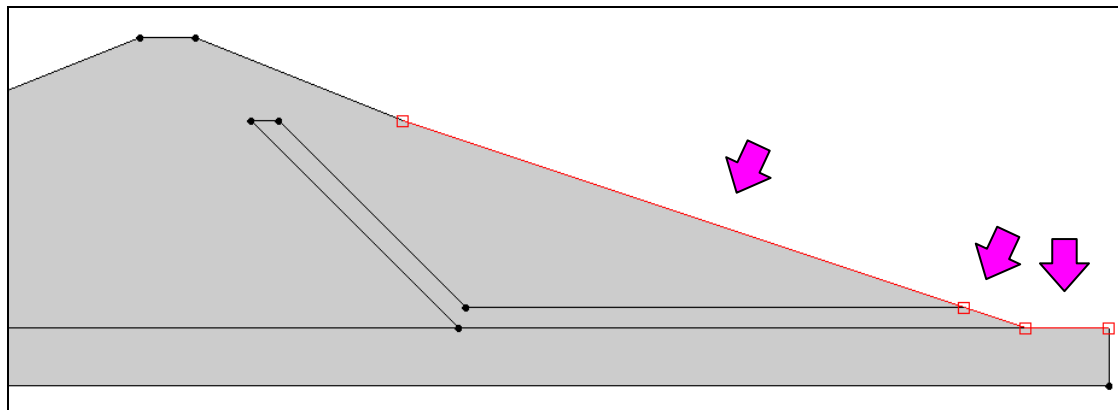2. Double-click one of the three arcs highlighted in Figure 6 below.

Figure 5.   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.

7.5   Redistribute the 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.

8   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. Click on the Materials macro ⊞ (or select the *Edit | Materials* menu command).

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

3. Optionally, you may wish to select a new color for the material.

4. 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:

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

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

To create the final material:

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

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

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

9   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.

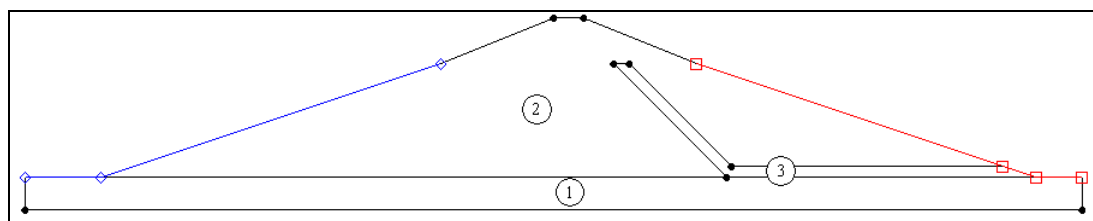2. Double-click on the embankment which is zone 2 in Figure 6 below.

Figure 6.   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.

10   Create the 2D Mesh

We're ready to create the mesh for SEEP2D.

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

Now we need to transfer the boundary conditions in the conceptual model to the mesh.

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

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.

11   Run SEEP2D

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

1. Click the *Save* 💾 button.

2. In the *Project Explorer,* click on the *2D Mesh Data* 📁 folder.

3. Select the *Run SEEP2D* macro 🔺 (or select the *SEEP2D | Run SEEP2D* command). At this point SEEP2D is launched in a new window.

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

4. 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.

12   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 7 below. The thick black line is the piezometric line computed by SEEP2D. Notice the flow lines concentrated in the drain portion of the dam.
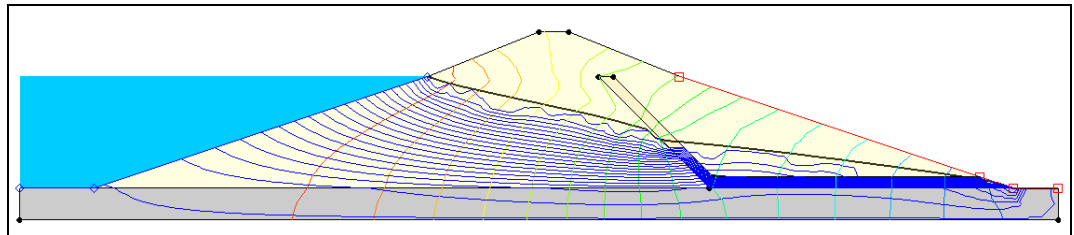

Figure 7.   SEEP2D solution.

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 selected.

9. Turn **off** the *Flow Lines* option.

13   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 *Materials* macro 🔲.

2. 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 (Embankment and Drain), 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 rows 2 and 3 (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-combined (SEEP2D)* solution (it should already be selected by default).

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

14   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*, double-click on the **UTEXAS** model.

2. Make sure the *Type of Surface Analysis* option is set to **Automatic Search Circular Floating Grid**.

3. Change the *Circle Center X* value to **500** and change the *Circle Center Y* value to **250**.

4. Scroll so that the lower half of the options are visible and change the options to match those shown in Figure 8.

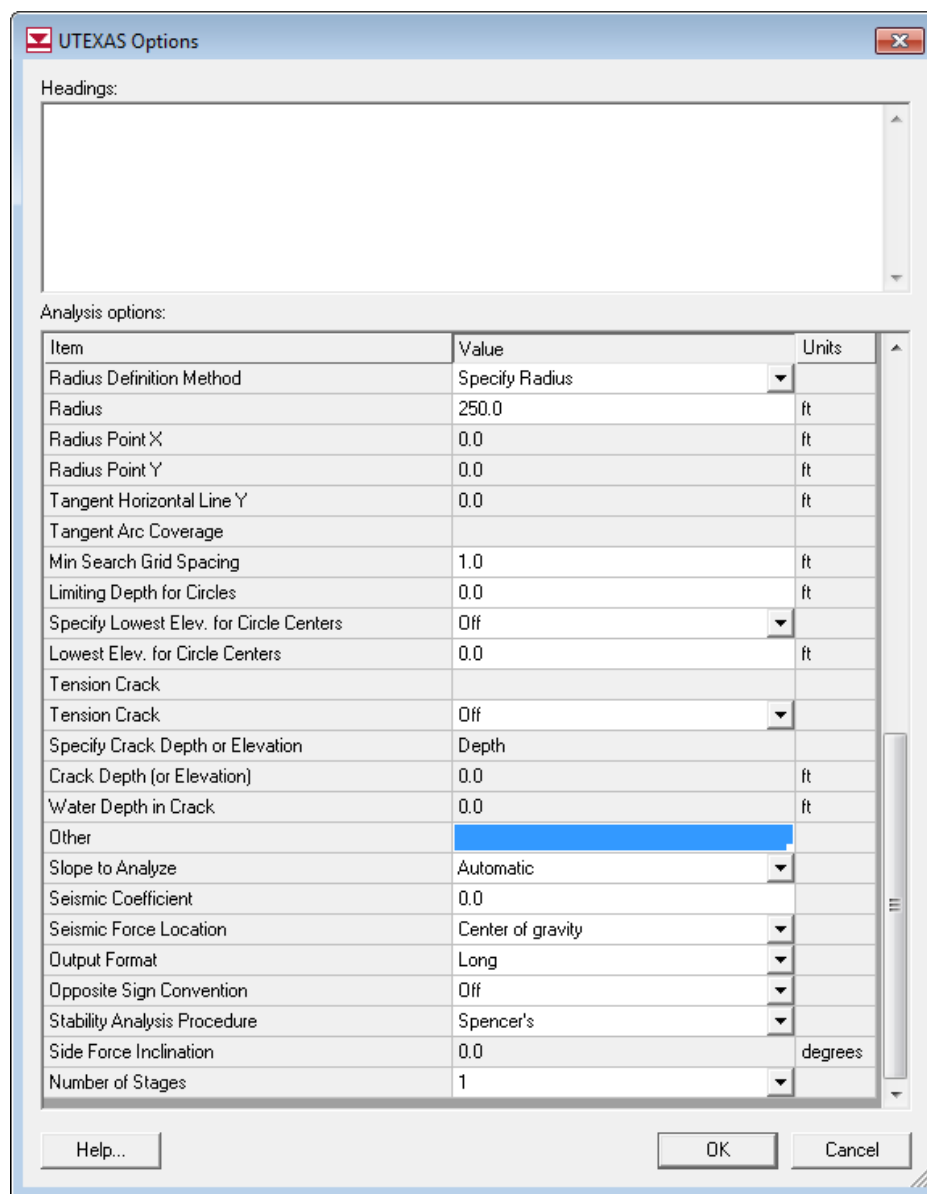5. When you're finished, click *OK* to exit the dialog.

Figure 8.   UTEXAS Options dialog.

At this point you should see the starting circle displayed.

15   Surface Water Forces

Before running UTEXAS, please note that the upstream slope is underwater. Therefore, we need to supply distributed forces along the top surface of the soil in order to ensure that the total stresses are properly computed in that portion of the problem domain. If we were doing a UTEXAS problem without SEEP2D, we would create a distributed load coverage and create two arcs to assign the loads. However, when performing a coupled SEEP2D/UTEXAS problem, GMS automatically creates the distributed loads using the specified head boundary condition used for SEEP2D. The distributed loads are automatically written to the UTEXAS input file when it is exported.

16   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 **Tutorials\UTEXAS\dam_seepage** (you should already be there).

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

17   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 **Tutorials\ UTEXAS\dam_seepage**) folder and open it.

5. When UTEXAS4 finishes, press *Save* in the *Open file for graphics output* dialog box. This will save a TexGraf4 output file.

6. Look at the things mentioned in the *Errors, Warnings* window, then close the window.

18   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.

19   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.