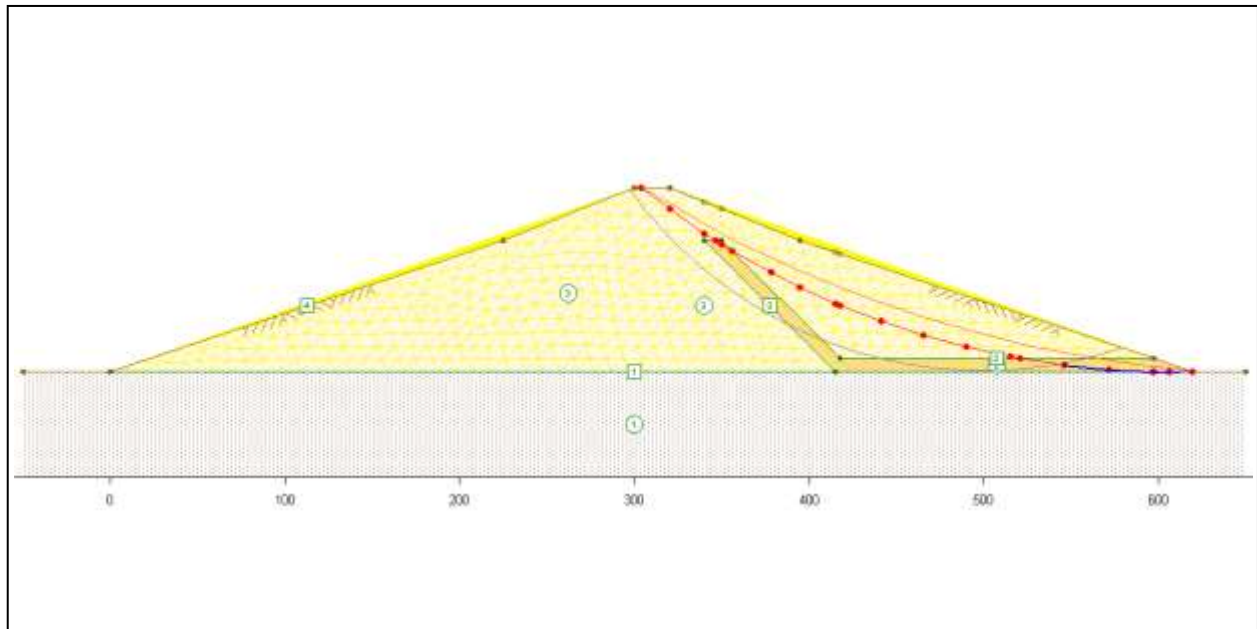


GMS 10.0 Tutorial

UTEXAS – Dam with Seepage

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



Objectives

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

Prerequisite Tutorials

- SEEP2D – Earth Dam
- UTEXAS – Natural Slope

Required Components

- GIS
- Map Module
- Mesh Module
- SEEP2D
- UTEXAS

Time

- 25–40 minutes



1	Introduction	2
1.1	Outline.....	3
2	Program Mode	3
3	Getting Started	3
4	Set the Units	4
5	Save the GMS Project File	4
6	Create the Conceptual Model Features	5
6.1	Create the Points	5
6.2	Create the Arcs and Polygons.....	6
6.3	Assign the Specified Head Arcs	7
6.4	Assign the Exit Face Arcs	8
6.5	Redistribute the Vertices	8
7	Material Properties – SEEP 2D	9
8	Assign Materials to Polygons	10
9	Create the 2D Mesh	10
10	Run SEEP2D	11
11	Display Options	11
12	Material Properties – UTEXAS	12
13	Analysis Options	13
14	Surface Water Forces	14
15	Export the Model	15
16	Run UTEXAS	15
17	Read the Solution	15
18	Conclusion	16

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.¹

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 – Earth Dam” tutorial explains more about SEEP2D. The user may wish to complete these tutorials before starting this tutorial.

1. Wright, S.G. (2003). UTEXPREP4 Preprocessor for UTEXAS4 Slope Stability Software. (Shinoak Software, Austin, Texas.)

1.1 Outline

In this tutorial, the user will be examining the earth dam shown on page 1. The user will use both UTEXAS and SEEP2D together to analyze the stability of the dam.

Here are the steps of this tutorial:

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

2 Program Mode

This tutorial assumes that the user is operating in the GMS 2D mode. If the user is already in GMS 2D mode, skip ahead to the next section. If the user is not already in GMS 2D mode, do the following:

1. Launch GMS.
2. Select the *Edit* / **P**references 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* dialog.
8. Then select the *File* / **E**xit command to exit GMS.

3 Getting Started

Do the following to get started:

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

At this point, the user should see the *New Project* dialog. 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.” The user 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, only a “Profile lines” coverage is needed.

3. Change the *Conceptual model name* to “Combined Model.”
4. Make sure that both the *SEEP2D* and *UTEXAS* options are turned on in the *Numerical models* section.
5. Make sure the *Profile lines* option is still selected.
6. Click **OK**.

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


4 Set the Units

Before continuing, the user must establish the units to be used. GMS will display the appropriate units label next to each of the input fields to remind the user to be sure to use consistent units.

1. Select the *Edit / Units* command.
2. For the Units select the “...” button, and, if necessary, change the units for both vertical and horizontal to “feet.”
3. If necessary, select “lb” for the *Force* units.
4. Select the **OK** button.

5 Save the GMS Project File

Before continuing, save the work as 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 (e.g., “dam -combined.gpr”).
4. Select the **Save** button.

Select the **Save**  button occasionally to save the developing work.

6 Create the Conceptual Model Features

The model the user 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.

6.1 Create the Points

The XY coordinates of the points defining the geometry of the zones of the dam were determined in advance. Enter them now according to the following instructions:

1. In the Project Explorer, right-click on the “Profile lines” coverage.
2. Select the **Attribute Table** command from the pop-up menu to open the *Attribute Table* dialog.
3. Make sure the *Show coordinates* option is turned on.
4. Enter the X and Y coordinates show in the table below. If the user is viewing this tutorial electronically, he or she 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.
6. 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

7. Now select the **Frame**  macro.

The user should now see the points on the screen.

6.2 Create the Arcs and Polygons

Now the user needs 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 allows the user to create multiple arcs continuously without having to stop and restart at each point. Double-click to stop creating an arc.
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.

The model should now look like the figure below:

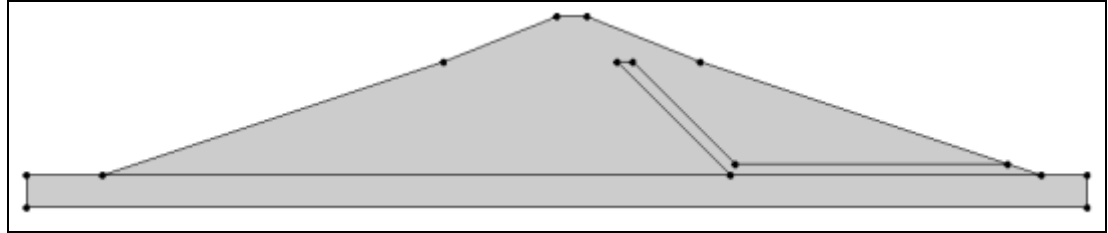



Figure 2 Arcs connecting points

6.3 Assign the Specified Head Arcs

Next the user will 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.

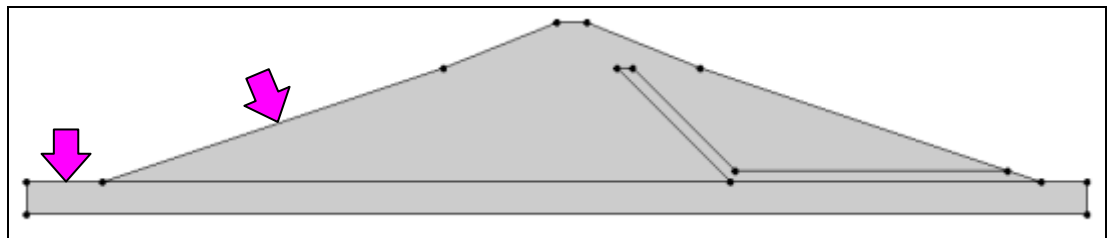


Figure 3 Assigning the head arcs

3. In the *Attribute Table* dialog, change the *Type* to “head.”
4. Change the *Head* value to “75.” Leave the options for stage 2 head alone.
5. When done, click **OK** to exit the dialog.
6. Repeat steps 2–5 for the other arc.

As the user assigns 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. The user can turn this display option off or change the fill color by accessing the *Display Options* dialog.

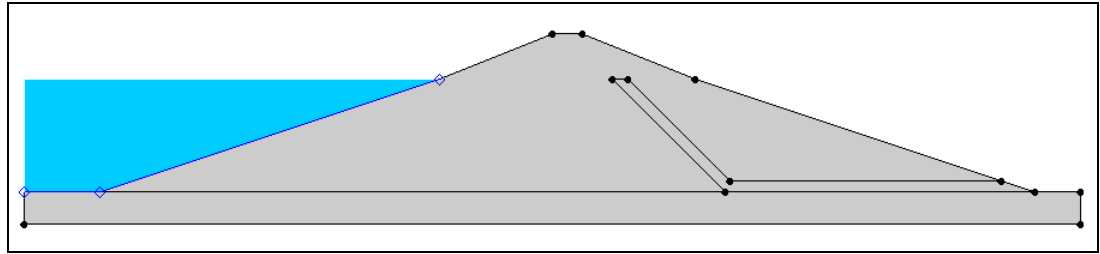



Figure 4 Filling the area above the head arcs

6.4 Assign the Exit Face Arcs

Next the user 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.

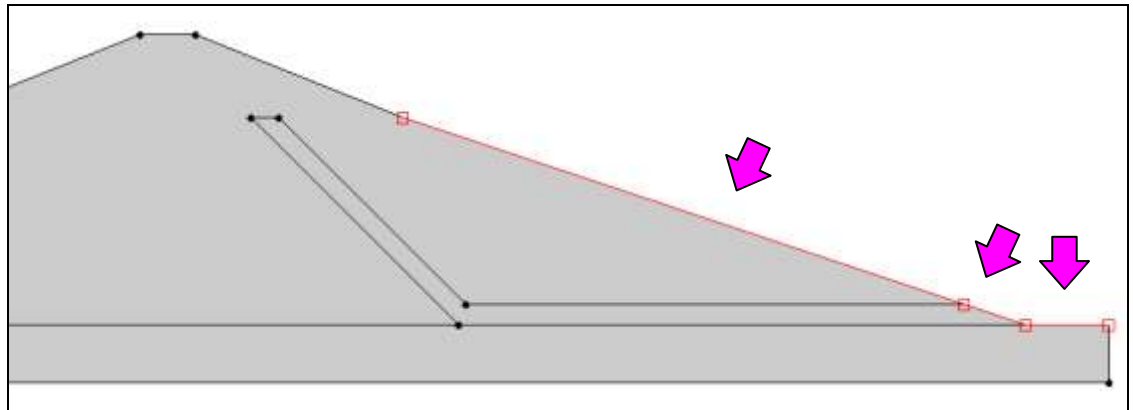


Figure 5 The exit face arcs

3. In the *Attribute Table* dialog, change the *Type* to “exit face.”
4. Click **OK**.
5. Repeat steps 2–4 for the other two arcs.

6.5 Redistribute the Vertices

The user will be creating a mesh for SEEP2D, and the mesh density will be determined by the vertex spacing on the arcs. The user needs to redistribute the arc vertices so that the 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 *Average spacing* to “10.”
5. Click **OK**.


It is not immediately evident that anything has changed but the user has indeed added vertices. To see them, do the following:

6. Switch to the **Select Vertices**  tool.

Now the new vertices should be displayed.

7 Material Properties – SEEP 2D

Each zone in the dam consists of a different type of soil material. The user needs to create the materials and then assign them to the polygons. The user will 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). This will open the *Materials* dialog.
2. Click on “material_1” in the spreadsheet and rename it “Foundation Rock.”
3. Optionally, the user may wish to select a new color for the material.
4. Change *k1* to “25.”
5. Change *k2* to “5.” The *k1* and *k2* values correspond to horizontal and vertical hydraulic conductivity, respectively, in feet per day. Don’t change the other values.

To create the next material:

6. Type “Embankment” in the *Name* column of the blank row at the bottom of the spreadsheet.
7. Change *k1* to “100” in the row with the ID equal to 2.
8. Change *k2* to “20” in the row with the ID equal to 2.


To create the final material:

9. Type “Drain” in the *Name* column of the blank row at the bottom of the spreadsheet.
10. Change *k1* to “1000” in the row with the ID equal to 3.

11. Change k_2 to “1000” in the row with the ID equal to 3.
12. When finished, click **OK** to exit the dialog.

8 Assign Materials to Polygons

Now that the materials are all defined, the user 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.

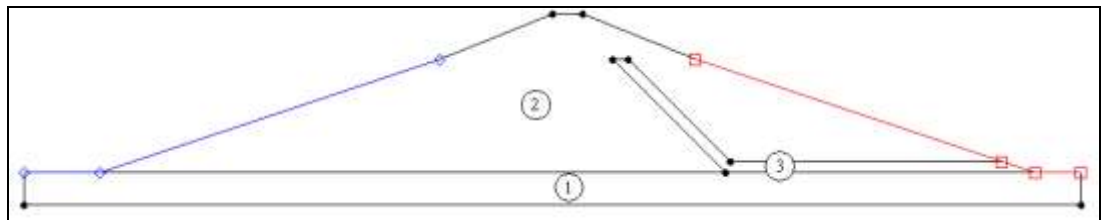



Figure 6 Polygons and their material IDs


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

9 Create the 2D Mesh

It is now possible to create the mesh for SEEP2D.


1. Select the **Map → 2D Mesh**  macro at the top of the GMS window (or the user can select the *Feature Objects* / **Map → 2D Mesh** command).

After several seconds, the user should see the mesh appear. Next, turn the mesh nodes off so the display doesn’t look so cluttered.

2. Select the **Display Options**  button to open the *Display Options* dialog.
3. Select the *2D Mesh Data* item from the list on the left.
4. Turn off the *Nodes* option if it is on.

5. Click **OK**.




Now the user needs 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 the user can select the *Feature Objects* | **Map → SEEP2D** command)

At this point, the user 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

Now it is possible 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 has terminated successfully, select the **Close** button.

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

11 Display Options

Next, change the display options so the solution is more easily viewable.

1. Select the **Display Options**  button to open the *Display Options* dialog.
2. Make sure that *2D Mesh Data* is selected in the list on the left.
3. Turn off the *Element edges* option.
4. Select the *SEEP2D* tab.
5. Turn off the *Head BC* and *Exit face BC* options.

6. If necessary, change the *Base* material to “Foundation Rock” (this controls how many flow lines are displayed).
7. Click **OK**.

The user 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.

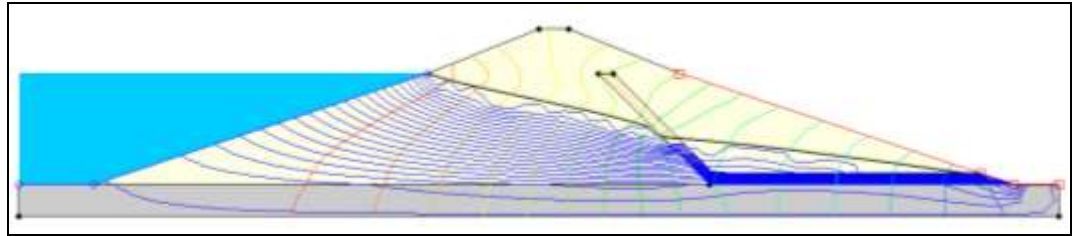



Figure 7. SEEP2D solution.


Now turn off the flow lines now so that it is easier to see and work with the model.

8. Select the **Display Options**  button to open the *Display Options* dialog.
9. Make sure *2D Mesh Data* is selected from the list on the left.
10. Select the *SEEP2D* tab.
11. Turn off the *Flow Lines* option.
12. Click **OK**.

12 Material Properties – UTEXAS

Now that the user has a set of pore pressures computed by SEEP2D, it is possible to continue to the second phase of the 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, the user mainly needs to enter the shear strength properties for the soils and define some UTEXAS4 analysis options.

Earlier, the user specified the SEEP2D material properties. Now, the user needs to specify the UTEXAS material properties.

1. Select the **Materials**  macro to open the *Materials* dialog.
2. Select the *UTEXAS* tab.
3. Make sure the *Show stage 1* option is on.
4. Make sure the *Show stage 2* option is off.

- Enter the following material properties.

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

- 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 Dataset Stage 1* entry for rows 2 and 3 (the user may need to expand the column). GMS automatically found the SEEP2D pore pressure dataset and listed it in this row.

13 Analysis Options

The only thing left to do before saving and running the model is to set the UTEXAS analysis options. The user will do an automatic search for the critical factor of safety using Spencer’s Method and circular failure surfaces.

- In the Project Explorer, double-click on the “UTEXAS” model.
- Make sure the *Type of Surface Analysis* option is set to “Automatic Search Circular Floating Grid.”
- Change the *Circle Center X* value to “500.”
- Change the *Circle Center Y* value to “250.”
- Scroll so that the lower half of the options is visible.
- Change *Radius* to “250.”
- Change *Limiting Depth for Circles* to “0.0”
- The entries should match those shown in Figure 8.
- When finished, click **OK** to exit the dialog.

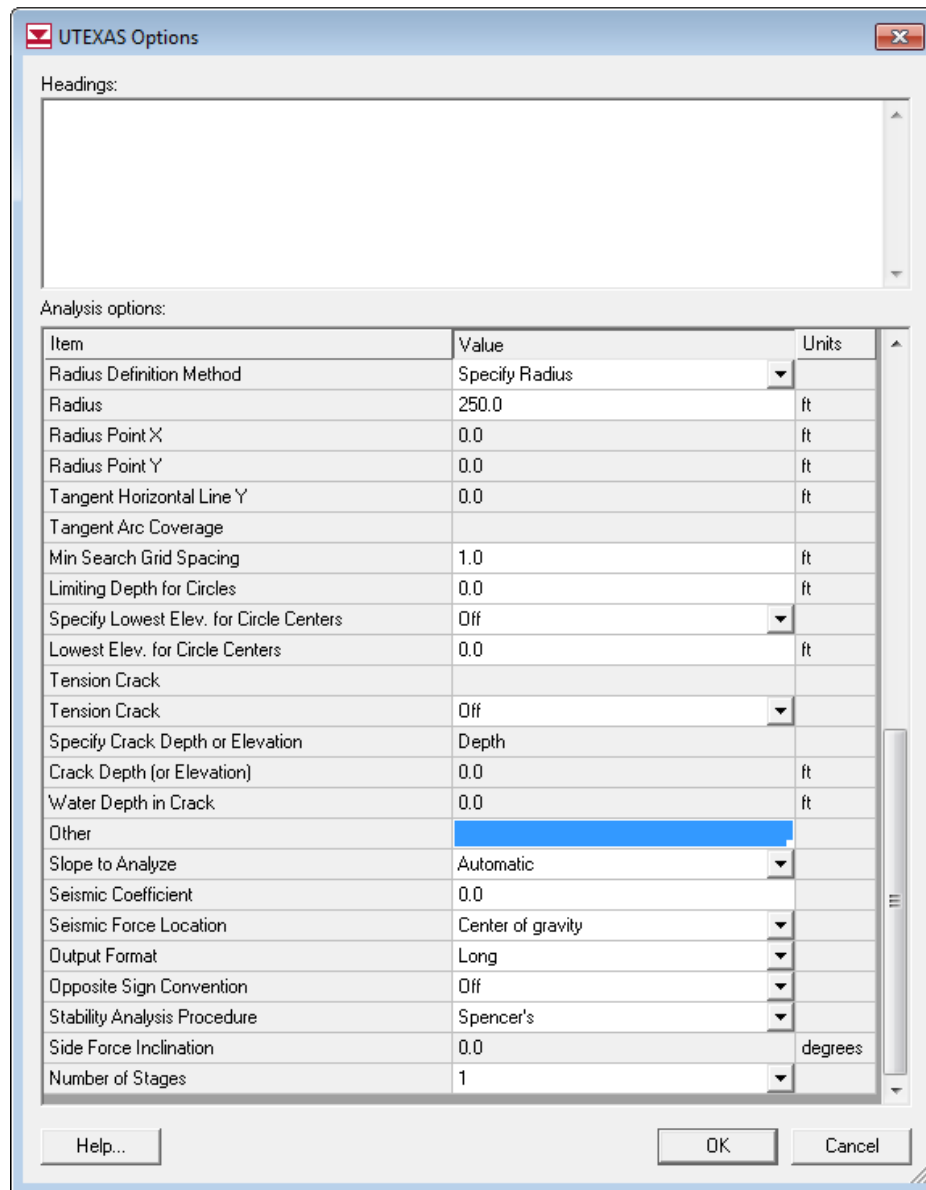


Figure 8. UTEXAS Options dialog.


At this point, the user should see the starting circle displayed.

14 Surface Water Forces

Before running UTEXAS, please note that the upstream slope is underwater. Therefore, the user needs 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 the user were doing a UTEXAS problem without SEEP2D, he or she 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.



15 Export the Model

It is now possible to export the model for use in UTEXAS.

1. In the Project Explorer, right-click on the “UTEXAS”  model.
2. Select the **Export** command from the pop-up menu.
3. If necessary, locate and open the directory entitled *Tutorials\UTEXAS\dam_seepage*.
4. Change the *File name* to “UTseepage.”
5. Click **Save**.


16 Run UTEXAS

Now that the UTEXAS input file has been saved, it is possible to run UTEXAS.

1. In the Project Explorer, right-click on the “UTEXAS”  model.
2. Select the **Launch UTEXAS4** command from the pop-up menu. This should bring up the UTEXAS4 program.
3. In UTEXAS4, select the **Open File**  button.
4. Change the *Files of type* to “All Files (*.*)”
5. Locate the “UTseepage.utx” file that was just saved (in the *Tutorials\UTEXAS\dam_seepage* folder).
6. Click **Open**.
7. When UTEXAS4 finishes, press **Save** in the *Open file for graphics output* dialog box. This will save a TexGraf4 output file.
8. Look at the things mentioned in the *Errors, Warnings* window, then close the window.
9. Close the UTEXAS window as well.

17 Read the Solution

Now it is necessary to read in the UTEXAS solution.

1. In the Project Explorer, right-click on the “UTEXAS”  model.

2. Select the **Read Solution** command from the pop-up menu.
3. Locate the file named “UTseepage.OUT.”
4. Click **Open**.

The user should now see a line representing the critical failure surface and the factor of safety.

18 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.