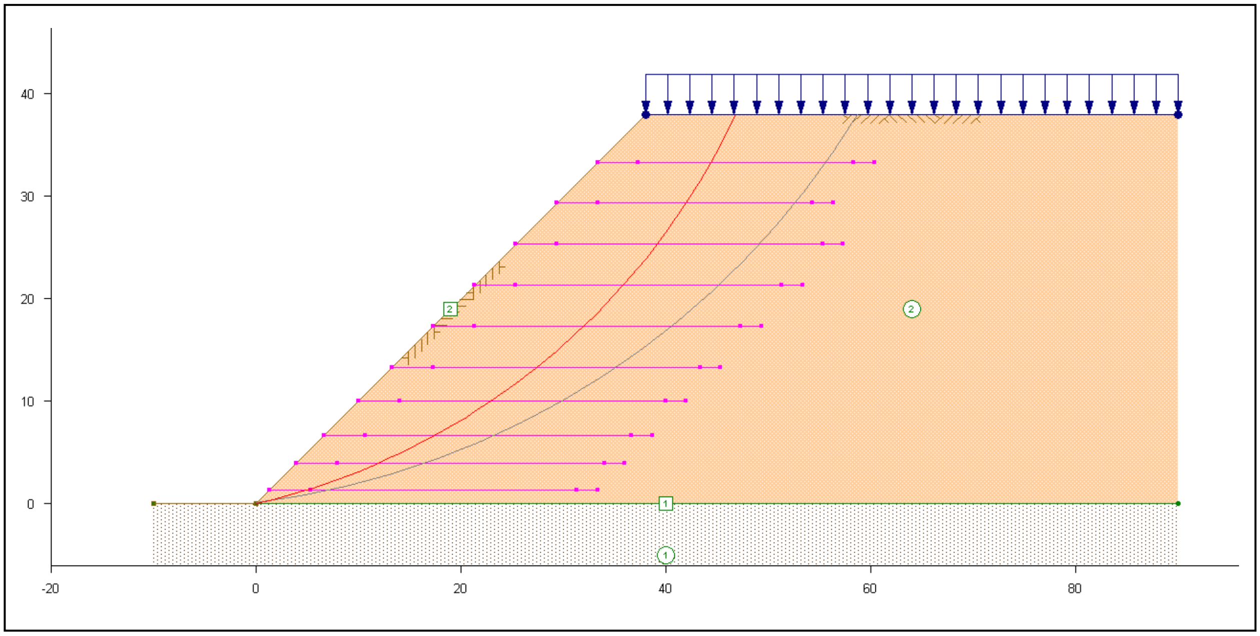
v. 9.2

GMS 9.2 Tutorial

UTEXAS – Reinforced Slope

Build a UTEXAS model that uses soil reinforcement to strengthen a slope



Objectives

Learn how to build a UTEXAS model in GMS that uses soil reinforcement to stabilize a slope. This tutorial is similar to tutorial number five in the UTEXAS tutorial manual. (“UTEXPREP4 Preprocessor for UTEXAS4 Slope Stability Software” by Stephen G. Wright, Shinoak Software, Austin Texas, 2003.)

Prerequisite Tutorials

- None

Required Components

- GIS
- Map
- UTEXAS

Time

- 30-60 minutes

1 Contents

2 Introduction

This tutorial illustrates how to build a UTEXAS model in GMS that uses soil reinforcement to stabilize a slope. This tutorial is similar to tutorial number five in the UTEXAS tutorial manual ("UTEXPREP4 Preprocessor for UTEXAS4 Slope Stability Software" by Stephen G. Wright, Shinoak Software, Austin Texas, 2003.).

The problem is illustrated on page 1. A fairly steep embankment is subjected to loading. The slope has several reinforcement elements to help strengthen it against failure. Reinforcement elements include geotextiles, nails, piers etc.

The *UTEXAS – Embankment on Soft Clay* tutorial explains more about UTEXAS and provides a good introduction to the GMS/UTEXAS interface. You may wish to complete it before attempting this tutorial.

2.1 Outline

In this tutorial, we'll be examining a reinforced embankment problem that looks like the one shown in on page 1. This is what you will do:

1. Create the slope.
2. Create material properties and assign materials to polygons.
3. Assign the distributed load.
4. Create the reinforcement lines.
5. Set the analysis options.
6. Export the UTEXAS input file, run UTEXAS, and view the solution in GMS.

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 a set of GIS features (points, lines, and polygons) that are used to define the model input. The data in the conceptual model are organized into a set of layers or groups called *coverages*. Each coverage is used to define a portion of the input and the properties that are assigned to the features in a coverage are dependent on the coverage type. GMS 2D allows us to quickly and easily define all of the coverages needed for our conceptual model using the *New Project* window.

2. Change the *Conceptual model name* to **Reinforced Slope**.

3. Turn **off** the *SEEP2D* option in the *Numerical models* section.

4. Select the following coverage options:

   Profile lines
   Distributed loads
   Reinforcement

5. Select the *OK* button.

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. Select **ft** for the *Length* units.

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

4. Select the *OK* button.

6 Save the GMS Project File

Before continuing, we will save our project to a GMS project file.

1. Select the *File* | *Save As* command.

2. Path to a location where you wish to save your project

3. Locate and open the directory entitled **Tutorials\UTEXAS\reinforcement**

4. Enter a name for the project file (ex. "**reinforced slope.gpr**") and select the *Save* button.

As you continue with the tutorial, click on the Save macro frequently to save your changes.

7 Create the Slope

The first step is to create the GIS features defining the embankment geometry. We will begin by entering a set of points corresponding to the key locations in the geometry. We will then connect the points with lines called "arcs" to define the outline of the embankment. We will then convert the arcs to a closed polygon defining our problem domain.

7.1 Create the Profile Geometry

First, we will create the profile lines defining the slope.

Create the Points

The locations of the points defining the slope were determined beforehand. We will simply enter the points and then connect them with arcs.

1. Click on the *Profile lines* coverage to make it active.
2. Right-click on the **Profile lines** coverage and select the *Attribute Table* command from the pop-up menu. This brings up the coverage *Properties* dialog.
3. Make sure the *Feature type* is **Points**.
4. Make sure the *Show point coordinates* option is turned **on**.
5. Enter the X and Y coordinates shown in the table below. If you are viewing this tutorial electronically, you can copy and paste these values into the GMS spreadsheet.

X	Y
-10	0
90	0
0	0
38	38
90	38
-10	-10
90	-10

6. Click *OK* to exit the dialog.
7. Click the Frame button to center the view on the new points.

You should now see the seven points defining the corners of the slope.

Connect the Points to Create Arcs

Next, we will connect the points to form arcs:

8. Select the *Create Arcs* tool.

9. 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.

10. Using Figure 1 as a guide, click on the points to connect them with arcs to create the slope.

Create the Polygons

Now that the arcs are created, we can use the arcs to build polygons representing the regions enclosed by the arcs. Later in this tutorial we will use the polygons to assign material properties. To build the polygons:

11. Select the *Build Polygons* macro at the top of the GMS window (or select the *Feature Objects* | *Build Polygons* command).

At this point you should see something like Figure 1 below.

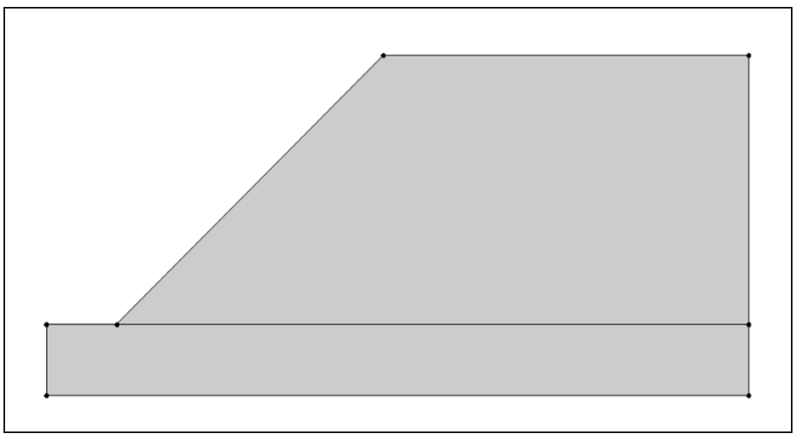


Figure 1. The basic slope.

8 Material Properties

The next step is to define the properties associated with the soil materials.

8.1 Create the Materials

1. Select the Materials macro (or select the *Edit* | *Materials* menu command).

2. Select the *UTEXAS* tab.

3. Click on the material named “material_1” and rename it “**Bedrock”**.

4. Click on the *Color/Pattern* button and change the color to **Dark yellow** or some other attractive color.

5. Create a new material by entering "**Sand**" in the *Name* column of the blank row at the bottom of the spreadsheet.

6. Change the material color to **Light orange**.

7. Make sure the *UTEXAS* tab is displayed.

8. Change the material properties to those shown in the following table:

Unit Weight Stage 1	Shear Strength Method Stage 1	Cohesion Stage 1	Angle of Internal Friction Phi Stage 1
160	Very Strong material		
120	Conventional	0	32

9. Make sure the *Pore Water Pressure Method Stage 1* is set to **No Pore Pressure** for the second material (Sand). You may have to scroll the spreadsheet to the right to see this column.

10. Leave the other settings at the defaults.

11. Click *OK* to exit the dialog.

8.2 Assign Materials to Polygons

Next, we will associate the materials with the polygons defining the soil zones in the profile:

1. Switch to the *Select* tool .

2. Double-click on the upper polygon (the big one).

3. Change the *Material* to **Sand** and click *OK* to exit the dialog.

4. Double-click on the lower polygon.

5. Change the *Material* to **Bedrock** (it may already be set) and click *OK* to exit the dialog.

9 Assign the Distributed Load

Now we'll set up the distributed load.

Create the Points

We will simply enter the points and then connect them with an arc.

1. Turn **off** the *Profile lines* coverage by unselecting its toggle.

2. Click on the **Distributed loads** coverage to make it active.

3. Right-click on the **Distributed loads** coverage and select the *Attribute Table* command from the pop-up menu. This brings up the coverage *Properties* dialog.

4. Make sure the *Feature type* is **Points**.

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

6. Enter the X and Y coordinates shown in the table below. If you are viewing this tutorial electronically, you can copy and paste these values into the GMS spreadsheet.

X	Y
38	38
90	38

7. Click *OK* to exit the dialog.

Connect the Points to Create Arcs

Next, we will connect points to form an arc representing a distributed load:

8. Select the *Create Arcs* tool.

9. Click on the two points representing the top of the sand embankment to connect them with an arc. This will create an arc representing a distributed load as shown below in Figure 2.

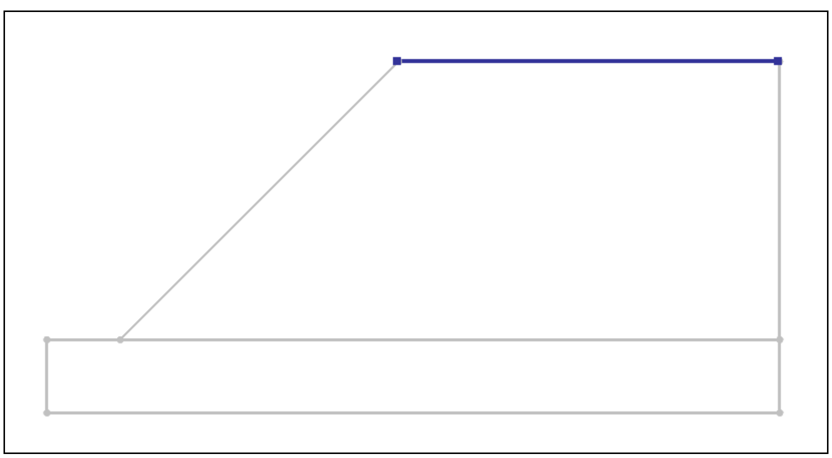


Figure 2. Selecting the distributed load arc.

10. Switch to the *Select* tool .

11. Double click on the distributed load arc you just created.

12. Change *Beg. Load Stage 1* to **240.0**.

13. Change *End Load Stage 1* to **240.0**.

14. Leave the other values alone and click *OK* to exit the dialog.

15. Turn **on** the display toggle for the *Profile lines* coverage.

You should now see the arrow heads indicating there is a distributed load.

10 Create Reinforcement Lines

Now we need to create the arcs for the reinforcement. The following diagram from the UTEXAS tutorials shows the location of the reinforcement:

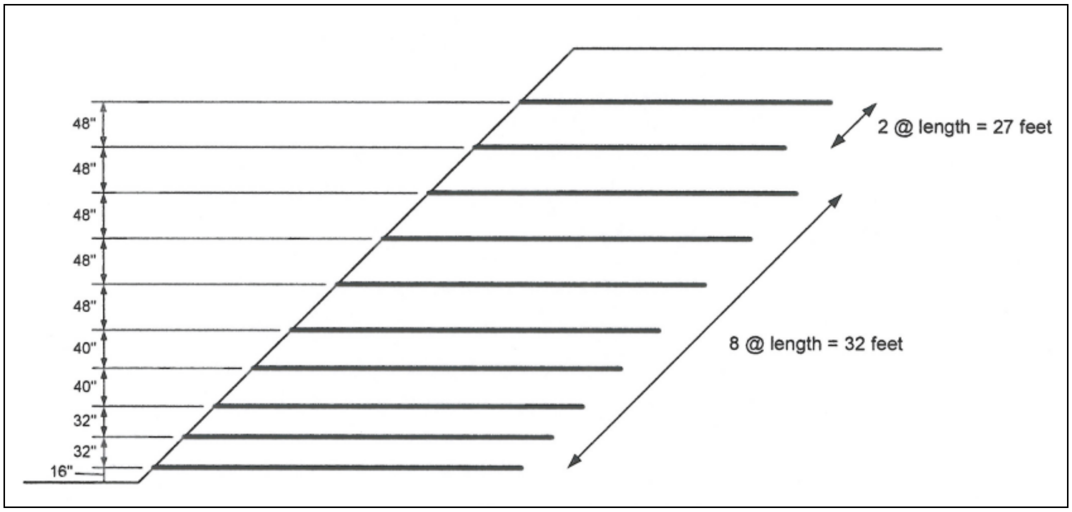


Figure 3. Reinforcement layout diagram, from “UTEXPREP4 Preprocessor For UTEXAS4 Slope Stability Software” by Stephen G. Wright, Shinoak Software, Austin Texas, 2003.

10.1 Enter the Points

The XY coordinates of the reinforcement lines have been computed for you and are provided below. You simply need to enter them in to GMS.

1. Click on the **Reinforcement lines** coverage to make it active.

2. Right-click on the **Reinforcement lines** coverage and select the *Attribute Table* command from the pop-up menu.

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

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

5. Enter the X and Y coordinates shown in the table below. If you are viewing this tutorial electronically, you can copy and paste these values into the GMS spreadsheet.

X	Y
1.33	1.33
5.33	1.33
31.33	1.33
33.33	1.33
6.67	6.67
10.67	6.67
36.67	6.67
38.67	6.67
13.33	13.33
17.33	13.33
43.33	13.33
45.33	13.33
17.33	17.33
21.33	17.33
47.33	17.33
49.33	17.33
21.33	21.33
25.33	21.33
51.33	21.33
53.33	21.33
25.33	25.33
29.33	25.33
55.33	25.33
57.33	25.33
29.33	29.33
33.33	29.33
54.33	29.33
56.33	29.33
33.33	33.33
37.33	33.33
58.33	33.33
60.33	33.33
4	4
8	4
34	4
36	4
10	10
14	10
40	10
42	10

6. Click *OK* to exit the dialog.

10.2 Connect the Points to Create Arcs

Now we'll connect the points to create arcs.

1. Select the *Create Arcs* 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 4 (below) as a guide, click on the points to connect them with arcs to create the slope.

At this point you should see something like Figure 4 below.

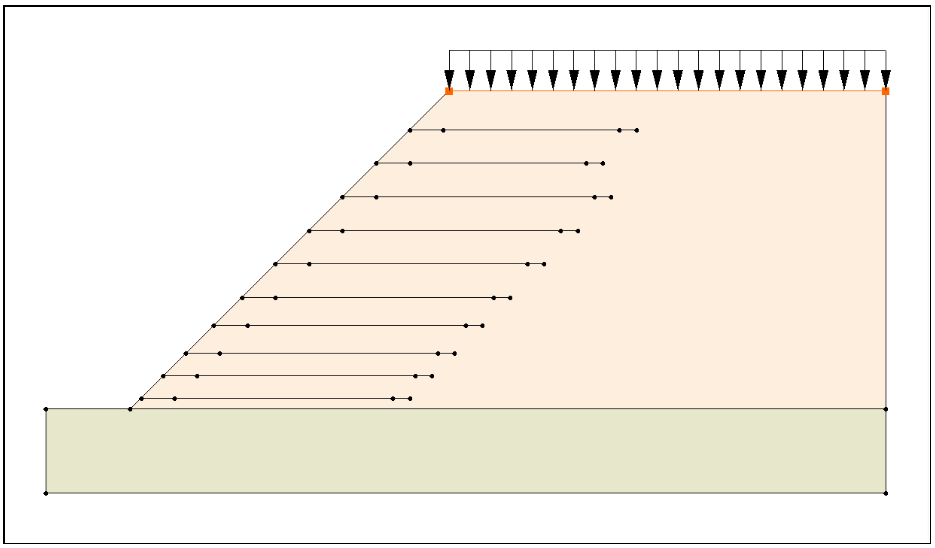


Figure 4. After adding reinforcement lines.

10.3 Assign the Forces at the Nodes

At each point along the reinforcement line, UTEXAS requires that we specify a longitudinal force and a transverse force. We will assign those now.

Nodes Along the Slope

1. Switch to the *Select Points/Nodes* tool.

2. Hold down the shift key and select all the reinforcement nodes on the left ends of the reinforcement arcs – the nodes that intersect the slope – as shown in Figure 5 below.

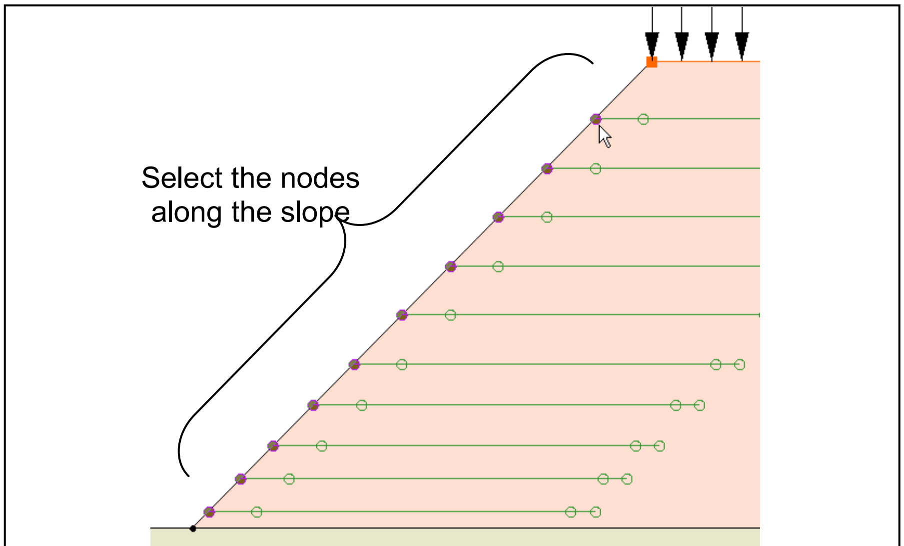


Figure 5. Selecting the reinforcement nodes along the slope.

3. Select the *Properties* button to open the coverage properties dialog.

4. In the *All* row at the top of the spreadsheet, change the *Long. Force* to **500** and hit the tab key. This should change the *Long. Force* to 500 in all the rows.

5. Leave the *Trans. Force* at **0.0**.

6. Click *OK* to exit the dialog.

7. Click anywhere not on the model to unselect the nodes.

Nodes Just to the Right of the Slope

1. Repeat the above procedure to assign a *Long. Force* of **1000** to all the nodes just inside the slope – all the nodes just to the right of the ones you just selected.

2. Again repeat the above procedure to assign a *Long. Force* of **1000** to the nodes to the right of the ones you just assigned – not the nodes on the right ends of the arcs but the nodes just to the left of those.

The nodes on the right ends of the reinforcement arcs are supposed to have both forces at 0.0. That is the default so we don't need to change those nodes.

11 Analysis Options

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

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 settings to match those shown in the following figure.

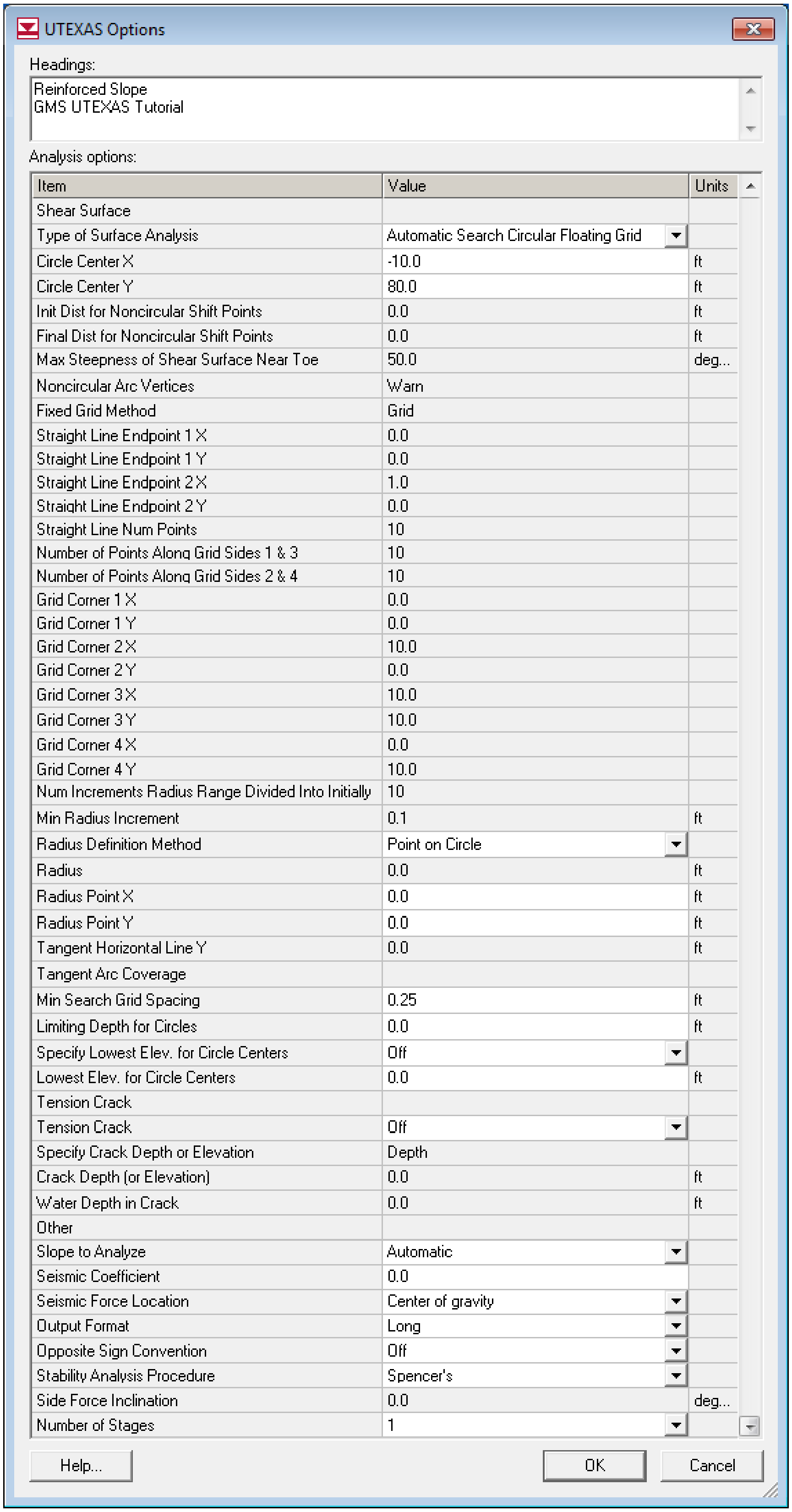


Figure 6. UTEXAS Options.

3. When you're done, click *OK*.

At this point you should see the starting circle displayed.

12 Save the GMS file

Before continuing, we will save the GMS project file.

1. Select the *File* | *Save* command.

13 Run UTEXAS

We're ready to export and run the model in UTEXAS.

13.1 Export the Model

To export the model:

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\reinforcement**.
3. Change the *File name* to **reinforced slope** and click *Save*.

13.2 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 **reinforced slope.utx** file you just saved (in the **Tutorials\UTEXAS\reinforcement**) folder and open it.
5. 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.

13.3 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 **reinforced slope.OUT**.

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

14 Conclusion

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

- You can use GMS to build reinforced slopes for analysis by UTEXAS.

- Reinforcement lines can be placed alone in a separate coverage or included in an existing coverage.

- In GMS, the forces applied along the length of the reinforcement lines are specified at the nodes.

