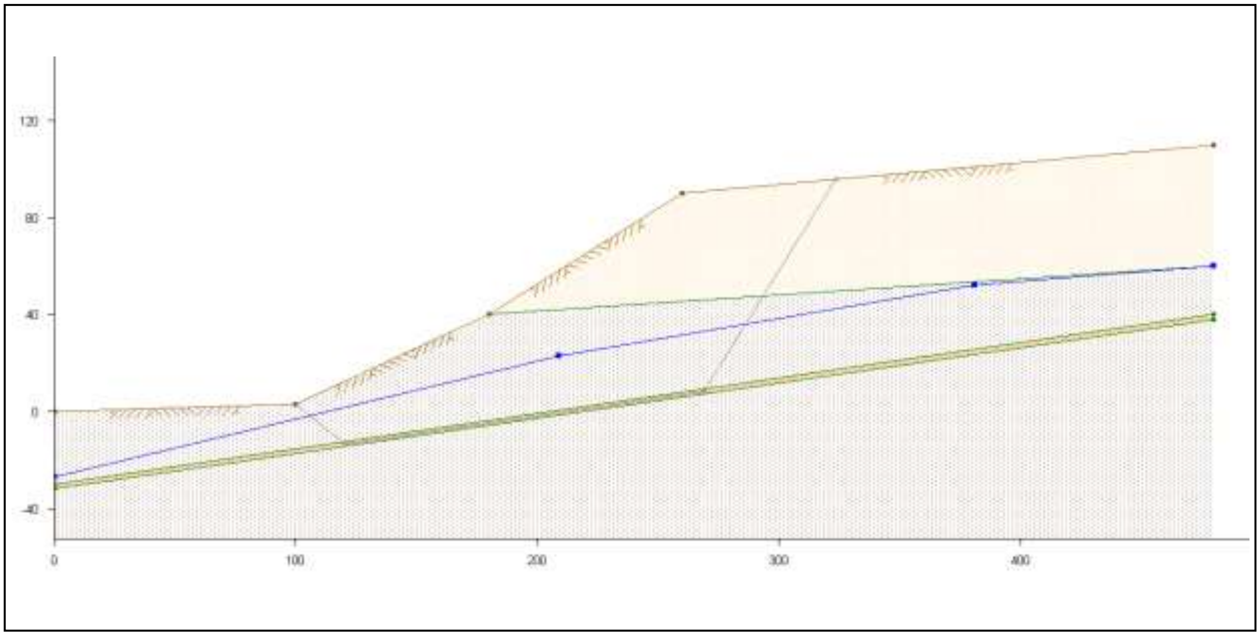
GMS

v. 10.0

GMS 10.0 Tutorial

UTEXAS – Natural Slope

Use the noncircular, shear surface option with UTEXAS



Objectives

Learn how to use GMS to create a UTEXAS model that uses a noncircular, shear surface to model a weak soil layer. This tutorial is similar to tutorial number two in the UTEXAS tutorial manual (Wright, Stephen G., "UTEXPREP4 Preprocessor for UTEXAS4 Slope Stability Software," Shinoak Software, Austin, Texas: 2003.)

Prerequisite Tutorials

- Feature Objects
- UTEXAS – Embankment on Soft Clay

Required Components

- GIS
- Map Module
- UTEXAS

Time

- 15–30 minutes

AQUAVEO™

1 Introduction

This tutorial illustrates how to use GMS to create a UTEXAS model that uses a noncircular shear surface to model a weak soil layer. This tutorial is similar to tutorial number two in the UTEXAS tutorial manual.[1]

The problem is illustrated on page 1. The user will analyze a natural slope that includes a weak clay seam for long-term stability.

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

1.1 Outline

Here are the main steps in this tutorial:

1. Create a UTEXAS4 model in GMS with a noncircular shear surface.

---

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

2. Create Piezometric lines defining the pore water pressures.
3. Assign attributes to the model and adjust the analysis options.
4. Save the model, run UTEXAS4 to get a solution, and view the solution in GMS.

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.

5. Launch GMS.
6. Select the *Edit* | **Preferences** command.
7. Select the *Program Mode* option on the left side of the dialog.
8. On the right side of the dialog, change the mode to "GMS 2D."
9. Click on the **OK** button.
10. Click **Yes** in response to the warning.
11. Click **OK** to get rid of the *New Project* dialog.
12. Then select the *File* | **Exit** 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* | **New** 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 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 the conceptual model using the *New Project* dialog.

3. Change the *Conceptual model name* to "Nat Slope Model."

4. Turn off the *SEEP2D* option in the *Numerical models* section.
5. Select the following coverage options:
   - *Profile lines*
   - *Piezometric line*
   - *Shear surface*
6. Select the **OK** button.

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

4 Set the Units

Before continuing, it is necessary to 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. Select "ft" for the *Length* units.
3. Select "lb" for the *Force* units.
4. Select the **OK** button.

5 Save the GMS Project File

Before continuing, save the project as a GMS project file.

1. Select the *File* | **Save As** command.
2. Locate and open the directory entitled *Tutorials\UTEXAS\natural_slope*.
3. Change the name of the project file to "embankment."
4. Click on the **Save** button.

Click on the **Save** macro frequently to save all changes.

6 Create the Embankment

The first step is to create the GIS features defining the embankment geometry. The user will begin by entering a set of points corresponding to the key locations in the geometry.

The user will then connect the points with lines called "arcs" to define the outline of the embankment. Next, the user will convert the arcs to a closed polygon defining the problem domain.

6.1 Create the Profile

The locations of the points defining the slope were determined beforehand. The user 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 will open the *Attribute Table* dialog.
3. Make sure the *Feature type* is set to "Points."
4. Make sure the *Show coordinates* option is on.
5. Enter points at the following XY locations. If the user is viewing this tutorial electronically, he or she can simply copy and paste these values into GMS. Don't worry about the Z coordinates.

x	y
0	-32
0	-50
0	-30
0	0
100	3
180	40
260	90
480	-50
480	40
480	60
480	110
480	38

6. Click **OK**.
7. Select the **Frame** macro to center the view on the new points.

The user should now see the points.

6.2 Connect the Points with Arcs

Now the user will connect the points to form arcs and create the polygons.

1. Select the **Create Arc** tool.

2. Hold down the *Shift* key. This makes it so that the user can create multiple arcs continuously without having to stop and restart at each point. Double-click to stop creating an arc.

3. Connect the points to make the profile look like the figure below. The user may need to use the **Zoom** tool to connect the points that are close together.

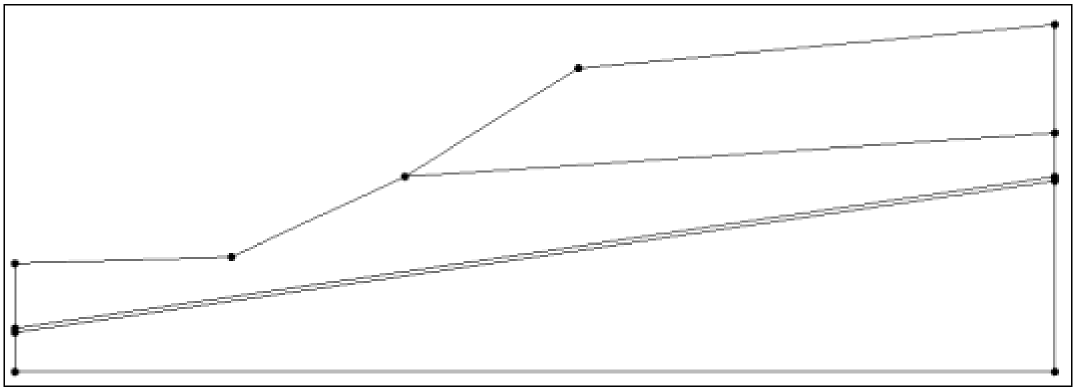


Figure 1 Natural slope profile

6.3 Create the Polygons

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

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

The model should now have gray polygons in the background.

7 Create Piezometric Line

In this model, the user will use a piezometric line to define the pore water pressures (Figure 2).

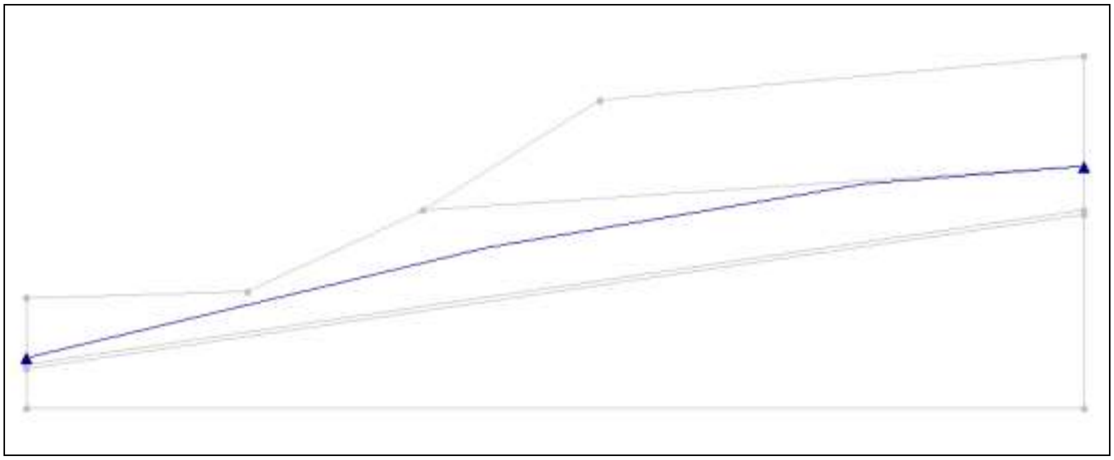


Figure 2 Piezometric line

7.1 Create the Points

Again, the user will enter the points first and then later connect them to form arcs.

1. Click on the “Piezometric line” coverage to make it active.
2. Right-click on the “Piezometric line” coverage and select the **Attribute Table** command from the pop-up menu to open the *Attribute Table* dialog.
3. Make sure the *Feature type* is set to “Points.”
4. Make sure the *Show coordinates* option is on.
5. Enter points at the following XY locations. If you are viewing this tutorial electronically you can simply copy and paste these values into GMS.

x	y
0	-27
209	23
381	52
480	60

6. Click **OK** to exit the dialog.

The user should now see the points. Some of the points may be obscured by the other coverage. The user will turn it off.

7. In the Project Explorer, turn off the “Profile lines” coverage.

7.2 Connect the Points with Arcs

Next, the user will connect the points to create arcs.

1. In the Project Explorer, click on the *Piezometric line* coverage to activate it.

2. Select the **Create Arc** tool.

3. Connect the points to form the arcs shown in Figure 2.

When the user starts with a set of four points and connects them to form arcs, the end result is three separate arcs. However, the UTEXAS input requires the use of a single arc to define the piezometric line. To convert the three arcs to a single arc, the user will select the two interior nodes and convert them to vertices.

4. Using the **Select Node** tool, select the two middle nodes (by dragging a box around them or holding down the shift key).

5. Right-click on either one of the nodes and select the **Node → Vertex** command from the pop-up menu.

Now the three arcs have been converted into a single arc.

8 Create Trial Shear Surface

The user will now create an initial noncircular slip surface that lies partially in the weak soil zone. UTEXAS uses this as the initial guess at the failure plane. Then it iteratively moves the surface to find the critical surface (i.e., the surface with the minimal factor of safety).

8.1 Create the Points

1. Click on the "Shear surface" coverage to make it active.

2. Right-click on the "Shear Surface" coverage and select the **Attribute Table** command from the pop-up menu to open the *Attribute Table* dialog.

3. Make sure the *Feature type* is set to "Points."

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

5. Enter points at the following XY locations. If the user is viewing this tutorial electronically, he or she can simply copy and paste these values into GMS.

x	y
100	3
120.5	-13.5
269	8.2
326	96

6. Click **OK** to exit the dialog.

The user should now see the points.

8.2 Connect the Points with Arcs

Next, the user will connect the points to form arcs defining the shear surface.

1. In the Project Explorer, click on the "Shear surface" coverage to activate it.
2. Select the **Create Arc** tool and connect the nodes (the user may want to hold down the shift key so that it isn't necessary to stop and restart at each point).

The shear surface should look like this:

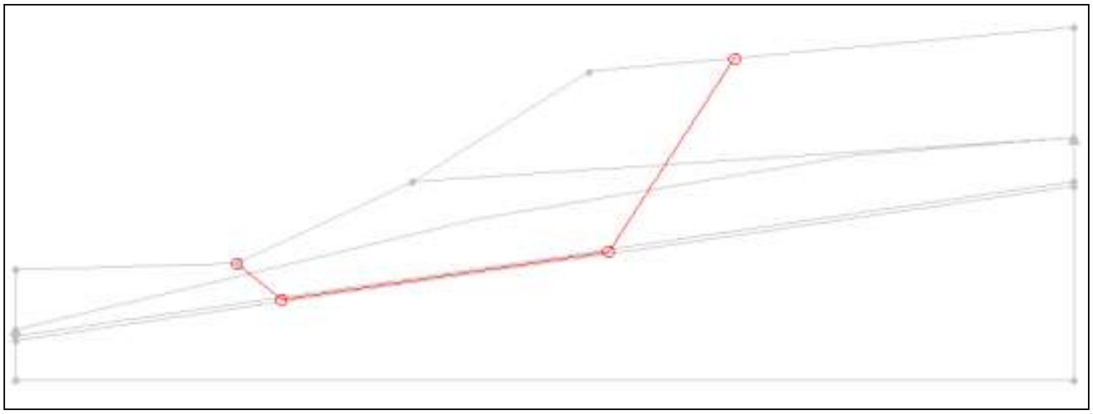


Figure 3. Initial shear surface.

In this case, it is not desirable to merge the three arcs into one. It is necessary to have separate arcs so that the user can assign attributes to the nodes.

8.3 Assign the Node Attributes

UTEXAS moves the initial surface according to the options that are specified at the nodes. The user can tell UTEXAS that a node is fixed, that it can only move in a specified direction, or that UTEXAS can move it in any direction it wishes.

1. Using the **Select Node** tool, select the two middle nodes by clicking on one and holding the shift key to click on the other or by dragging a box around the two nodes.
2. Select the **Properties** tool to bring up the *Attribute Table* dialog.
3. In the *All* row under *Point Shift*, enter "Specified."
4. In the *All* row, enter **"**8.4**"** for *Angle*.

This option will allow these two nodes to move only along the weak clay seam. The other nodes will be left at their default settings, which is "Automatic." For nodes on the slope exterior, UTEXAS moves the nodes such that they move laterally along the ground surface.

5. Click **OK** to close the dialog.

9 Material Properties

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

9.1 Create the Materials

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

2. Select the *UTEXAS* tab.

3. Double-click on "material_1" in the list in the upper left of the dialog.

4. Change the name to "Silty Clay."

5. Change the *Color/Pattern* from black to a lighter color, like Gold.

Do the following to create a new material:

6. Type "Highly Plastic Clay" in the *Name* column of the blank row at the bottom of the spreadsheet.

7. Change its color to Lime or some other color.

8. Create another material named "Sandy Gravel."

9. Change the color of this material to Brown or some other color.

9.2 Set the Material Properties

1. Change the material properties to the following (make sure the Silty Clay ID is 1, Highly Plastic ID is 2, and Sandy Gravel ID is 3):

ID	Unit Weight Stage 1	Shear Strength Method Stage 1	Cohesion Stage 1	Angle of Internal Friction Phi Stage 1	Pore Water Pressure Method Stage 1
1	126	Conventional	100	29	Piezometric Line
2	115	Conventional	0	14	Piezometric Line
3	131	Conventional	0	38	Piezometric Line

The above table does not show the column titled *Piezometric Line Coverage Stage 1* because the values in that column are set automatically when the user changes the *Pore Water Pressure Method* to "Piezometric Line." Thus, the user doesn't need to worry about it.

2. Click **OK** to exit the dialog.

9.3 Assign Materials to Polygons

Now the user will change each polygon in the "Profile lines" coverage to be associated with the correct material.

1. In the Project Explorer, select the "Profile lines" coverage to make it the active coverage. Also, turn it on so it's visible again.
2. Select the **Select Polygons**  tool.
3. Double-click on the top polygon to bring up its properties.
4. In the *Attribute Table* dialog, change the *Material* to "Sandy Gravel."
5. Click **OK**.
6. Do the same for the other three polygons, assigning them materials according to the following diagram:

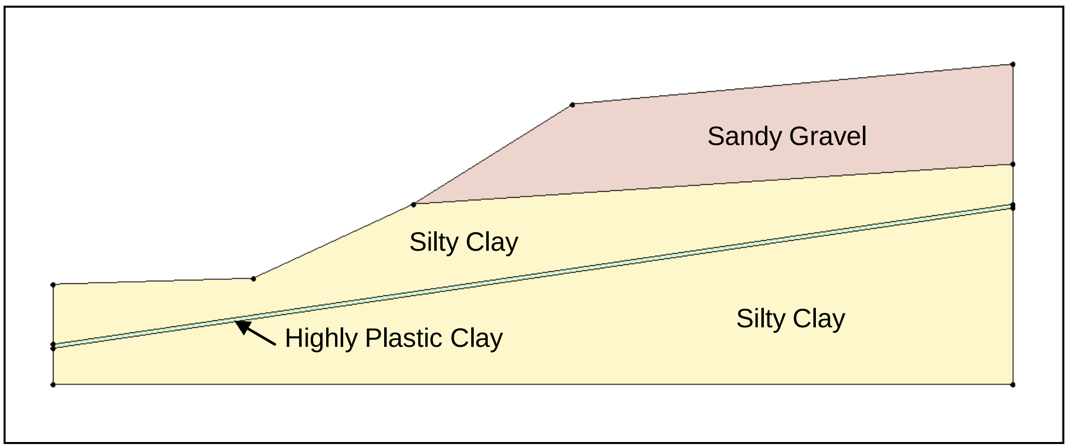
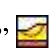


Figure 4 Materials for each polygon

10 Analysis Options

The only thing left to do before saving and running the model is to set the UTEXAS analysis options. The user will perform an automatic search using a non-circular surface 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. This will open the *UTEXAS Options* dialog.
2. In the *Headings* section, enter the following headings:

   "Natural Slope"
   "GMS UTEXAS Tutorial"

3. Change *Type of Surface Analysis* to “Automatic Search Noncircular.”
4. Change *Init Dist for Noncircular Shift Points* to “5.0.”
5. Change *Final Dist for Noncircular Shift Points* to “1.0.”
6. Change *Max Steepness of Shear Surface Near Toe* to “50.0.”
7. The options should match those shown in the dialog below.

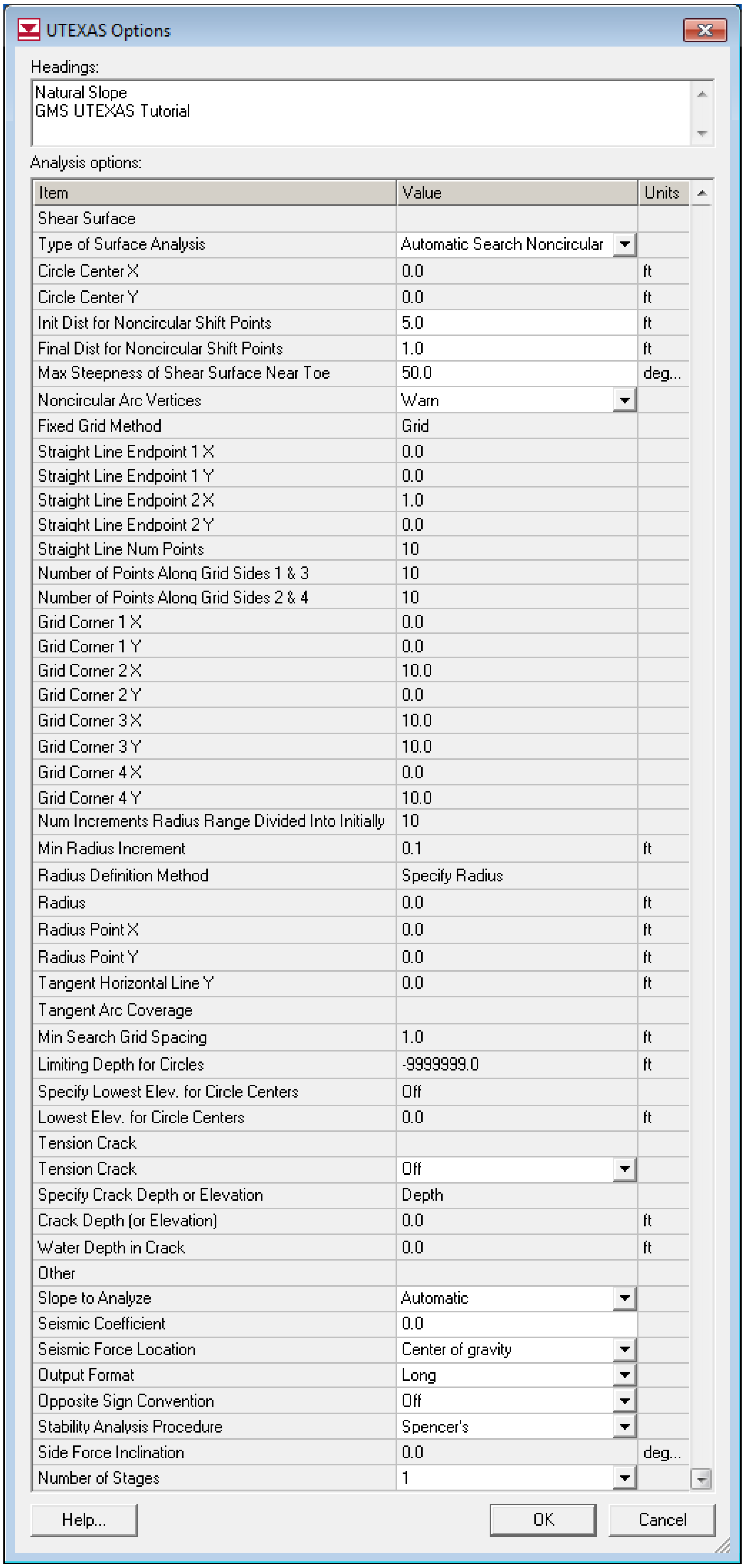


Figure 5. UTEXAS Options.

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

11 Save the GMS file

Before continuing, save the GMS project file.

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

12 Export the Model

Now it is possible to export the model for use in UTEXAS.

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

13 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 “nat_slope.utx” file that was just saved (in the *Tutorials\UTEXAS\natural_slope* folder).
6. Click **Open**.
7. 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.

14 Read the Solution

Now the user needs to read 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 “nat_slope.OUT.”
4. Click **Open.**

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

15 Conclusion

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

- UTEXAS can calculate a noncircular slip surface, given the slope angle and an initial guess at the shear surface.
- It is possible to tell UTEXAS how to move the initial shear surface by specifying options at the nodes of the shear surface arcs.

