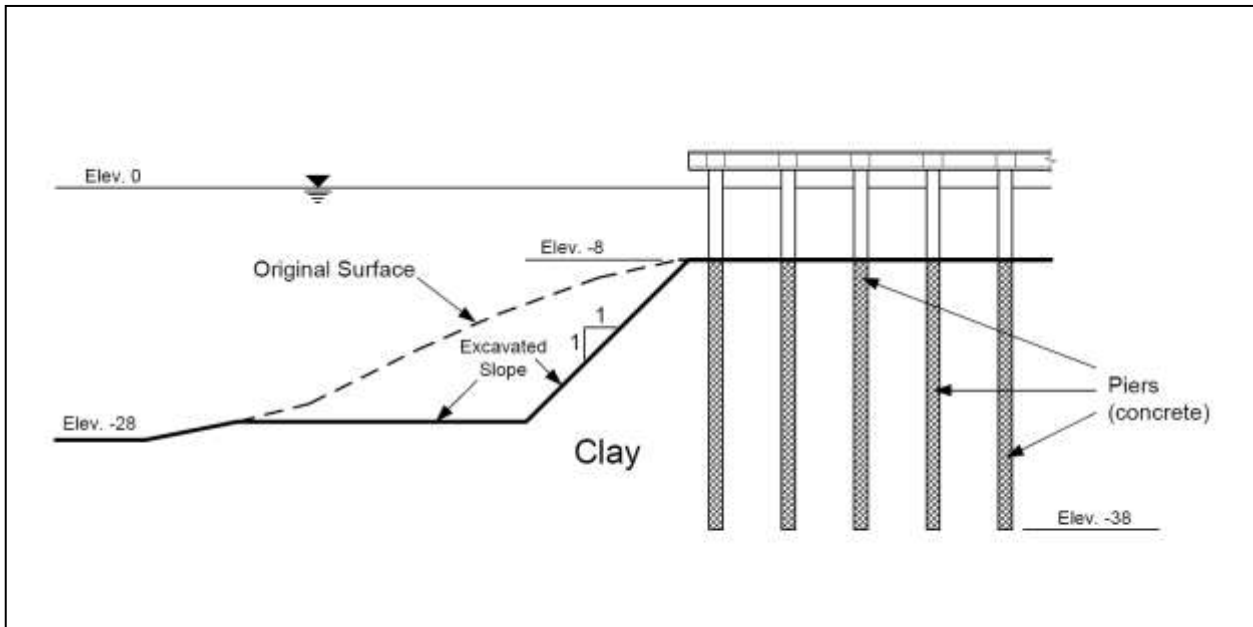


GMS 10.0 Tutorial

UTEXAS – Underwater Slope

Use a slope geometry line to build a model in UTEXAS of an underwater slope



Objectives

Illustrate how to build a UTEXAS model in GMS that uses a slope geometry line. This tutorial is similar to tutorial number eight in the UTEXAS tutorial manual manual (Wright, Stephen G., “UTEXPREP4 Preprocessor for UTEXAS4 Slope Stability Software,” Shinoak Software, Austin, Texas: 2003).

Prerequisite Tutorials

- UTEXAS – Embankment on Soft Clay
- UTEXAS – Natural Slope
- Feature Objects

Required Components

- GIS Module
- Map Module
- UTEXAS

Time

- 30–45 minutes



1	Introduction	2
1.1	Outline.....	3
2	Program Mode	3
3	Getting Started	4
4	Set the Units	4
5	Save the GMS Project File	5
6	Create the Conceptual Model Features	5
6.1	Create the Points	5
6.2	Create the Arcs and Polygon	6
7	Material Properties	7
8	Create the Piezometric Line	7
8.1	Create the Arc	8
9	Create the Slope Geometry Arc	9
9.1	Create the Points	9
9.2	Create the Arc	9
9.3	Convert Nodes to Vertices	10
10	Create Line Loads	11
10.1	Enter the Points	11
11	Analysis Options	12
12	Save the GMS file	12
13	Run UTEXAS	12
13.1	Export the Model.....	13
13.2	Run UTEXAS	13
13.3	Read the Solution	13
14	Alternate Model	14
15	Reinforcement Lines	14
15.1	Enter the Points	14
15.2	Connect the Points to Create Arcs.....	15
15.3	Assign the Node Attributes	16
16	Save the GMS file	17
17	Run UTEXAS	17
17.1	Export the Model.....	17
17.2	Run UTEXAS	18
17.3	Read the Solution	18
18	Compare the Solutions	19
19	Conclusion	19

1 Introduction

This tutorial illustrates how to build a UTEXAS model in GMS that uses a slope geometry line. This tutorial is similar to tutorial number eight in the UTEXAS tutorial manual.¹

The problem is illustrated on page 1. A dock supported by concrete piers is being built on an underwater slope. The slope will then be excavated below the original surface as shown in the figure. The slope will be modeled using two different approaches. First, the user will use line loads on the top of the slope to represent the weight of the dock

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

structure on the slope. Then the user will add reinforcement lines representing the concrete piers to determine the impact they have on the stability of the slope.

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 this tutorial and the “UTEXAS – Natural Slope” tutorial before beginning this tutorial. The user should also be familiar with feature objects and should have completed the “Feature Objects” tutorial.

1.1 Outline

In this tutorial, the user will be examining the underwater slope problem illustrated on page 1. Here are the steps to this tutorial:

1. Create the basic slope profile.
2. Define material properties.
3. Create the slope geometry line.
4. Create the distributed loads and line loads.
5. Create a starting circle.
6. Set up the analysis options.
7. Save the model, run UTEXAS4 to get a solution, and view the solution in GMS.
8. Add reinforcement lines to the model.
9. 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:

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* dialog.
8. 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 users 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 “Underwater Slope.”
4. Turn off the *SEEP2D* option in the *Numerical models* section.
5. Select the following coverage options:
 - *Profile lines*
 - *Piezometric line*
 - *Slope geometry*
 - *Line loads*
 - *Reinforcement*
6. Click **OK** to exit the dialog.

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


4 Set the Units


Before continuing, the user will 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** menu command.
2. If necessary, for the length, click the “...” button, and 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. Locate and open the directory entitled *Tutorials\UTEXAS\underwater*.
3. Enter a name for the project file (e.g., “uwslope-utexas.gpr”).
4. Select the **Save** button.


Select the **Save**  button occasionally to save the work as it develops.

6 Create the Conceptual Model Features

The first step is to create the GIS features defining the model 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 slope. Next, the user will convert the arcs to a closed polygon defining the problem domain.


6.1 Create the Points

To create the points:

1. Click on the “Profile lines”  coverage to make it active.
2. Right-click on the “Profile lines” coverage.
3. Select the *Attribute Table* command from the pop-up menu. This brings up the *Attribute Table* dialog.
4. Make sure the *Feature type* is “Points.”
5. Make sure the *Show coordinates* option is turned on.



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

X	Y
0	-28
10	-28
20	-26
28	-24
40	-18
49	-14
60	-10
69	-8
110	-8
0	-40
110	-40

7. Click **OK** to exit the dialog.
8. Click the **Frame**  button to center the view on the newly created points.

The user should now see the eleven points defining the outline of the slope.

6.2 Create the Arcs and Polygon

1. Select the **Create Arcs**  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. Using Figure 1 (below) as a guide, click on the points to connect them with arcs to create the slope. It doesn't matter where the user starts or which direction the user goes.
4. Select the **Build Polygons**  macro at the top of the GMS window (or select the *Feature Objects* / **Build Polygons** command).

At this point, the user should see something like Figure 1 below.

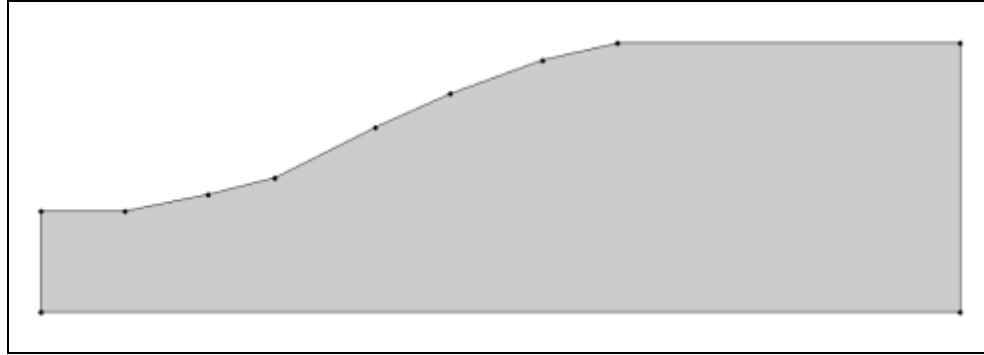




Figure 1 The underwater slope before excavation

7 Material Properties

The next step is to define the properties associated with the soil material. The user will model the shear strength of the clay using the linear strength method, which is similar to a c/p approach. The user will enter an initial undrained strength of 100 lb/ft^2 for the ground surface and a rate of change defining the increase in shear strength vs. depth ($10 \text{ lb/ft}^2/\text{ft}$).

1. In the Project Explorer, double-click on “Material Properties”  under “UTEXAS” . This brings up the *Materials* dialog.
2. Click on the material named “material_1” and rename it to “Clay.”
3. Click on the *Color/Pattern* button and change the color to Orange or some other attractive color.
4. Change the material properties to those shown in the following table:



Unit Weight Stage 1	Shear Strength Method Stage 1	Linear Increase Strength Stage 1	Linear Increase Rate Stage 1
100	Linear	100	10

5. Make sure the *Pore Water Pressure Method Stage 1* is set to “No Pore Pressure.” The user may have to scroll the spreadsheet to the right to see this column.
6. Click **OK** to exit the dialog.

8 Create the Piezometric Line

The next step is to create a piezometric line. Since this is a total stress analysis, UTEXAS does not use the piezometric line to compute pore pressures. However, it does use the line to compute the total stresses.

8.1 Create the Arc

1. Select the “Piezometric line”  coverage to make it the active coverage
2. Select the **Create Arcs**  tool.
3. Create a new arc above the model, as shown in **Error! Reference source not found.** below. Don’t worry about the exact location of the arc as the user will reposition it in the next steps.

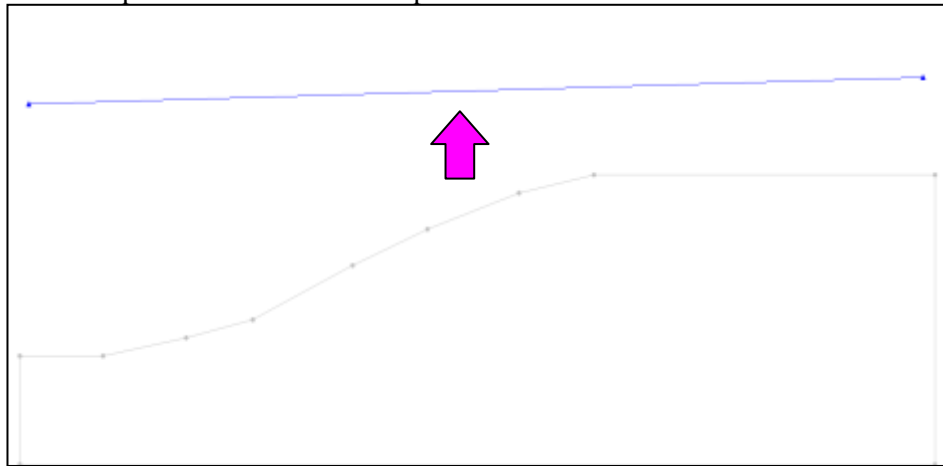




Figure 2 The initial piezometric line

4. Switch to the **Select Points/Nodes**  tool.
5. Select the node on the left of the new arc that was just created.
6. In the *GMS Edit Window*, change each of the XYZ coordinates to “0.”
7. Similarly, select the node on the right side of the new arc and change its XYZ coordinates to “110,” “0,” and “0.”

Finally, it is necessary to set an option in the arc properties to indicate that the line will be used to define distributed loads on the surface of the slope.


8. Switch to the **Select Arcs**  tool.
9. Double-click on the arc to bring up the *Attribute Table* dialog.
10. Turn on the *Dist. Load Stage 1* option.
11. Click on the **OK** button to exit.

9 Create the Slope Geometry Arc

This model uses a slope geometry arc in addition to the other arcs to define the model geometry. A slope geometry arc is typically used when modeling a “cut” or excavation into an existing slope. The native soil layers are modeled with one set of lines and the cut line is overlaid on top defining the new slope geometry. This approach makes it possible to define a slope using the original, unexcavated geometry and then test the impact of various excavations without having to change all of the profiles associated with the slope. Only the slope geometry line needs to be altered.

9.1 Create the Points

To enter the points defining the excavation:

1. Select the “Slope geometry”  coverage to make it active.
2. Right-click on the “Slope geometry” coverage and select the **Attribute Table** command from the pop-up menu. This will open the *Attribute Table* dialog.
3. In the dialog, make sure the *Feature type* is set to “Points.”
4. Make sure the *Show coordinates* option is turned on.
5. Enter the X and Y coordinates shown in the table below. If the user is viewing this tutorial electronically, the user can copy and paste these values into the GMS spreadsheet.

X	Y
0	-28
10	-28
20	-26
52	-26
70	-8
110	-8

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

The user should see the new points. Some might be obscured by the inactive coverages. That will now be fixed.

7. In the Project Explorer, turn off the “Profile lines” and “Piezometric line” coverages.

9.2 Create the Arc

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

1. Select the **Create Arcs**  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. Using Figure 3 (below) as a guide, click on the points to connect them with arcs to create the slope.

At this point, the user should see something like Figure 3 below.

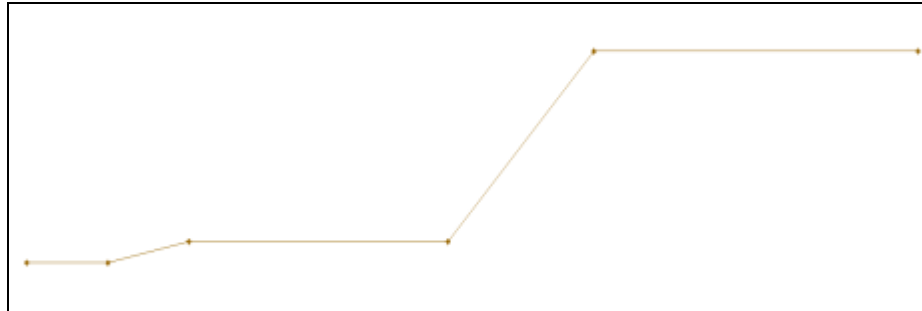



Figure 3 The slope geometry arcs

9.3 Convert Nodes to Vertices

Now the user has five slope geometry arcs. However, the user can only have one since each arc will be exported as a slope geometry line in the UTEXAS input file, and UTEXAS can only handle one slope geometry line. The user will change it so that there is only one slope geometry arc.

1. Select the **Select Points/Nodes**  tool.
2. Drag a box around the four middle nodes, as shown in Figure 4 below.

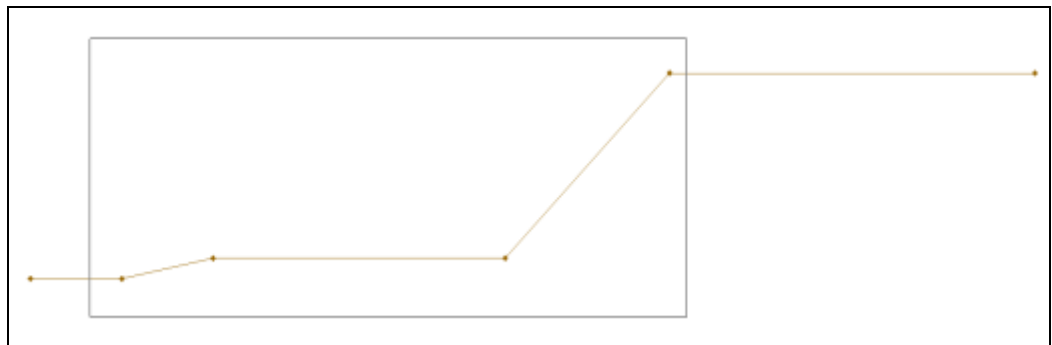



Figure 4 Selecting the interior nodes by dragging a box

3. Select the *Feature Objects* / **Nodes** → **Vertices** menu command.
4. In the Project Explorer, turn on the “Profile lines” and “Piezometric line” coverages.

10 Create Line Loads

The weight of the dock structure on the top of the slope will be represented using line loads. Each line load represents a row of piers in a direction perpendicular to the plane of the slope and will be applied at points along the top of the slope. The user will apply an average line load of 2000 lbs per foot.

10.1 Enter the Points


1. Select the “Line loads”  coverage to make it active.
2. Right-click on the “Line loads” coverage.
3. Select the **Attribute Table** command from the pop-up menu to open the *Attribute Table* dialog.
4. In the dialog, make sure the *Feature type* is set to “Points.”
5. Make sure the *Show coordinates* option is turned on.
6. Enter the X and Y coordinates shown in the table below. If the user is viewing this tutorial electronically, the user can copy and paste these values into the GMS spreadsheet.

X	Y
73	-8
81	-8
89	-8
97	-8
105	-8

7. Note that the *Type* has already been assigned as “line load.”
8. In the *All* row, change the *Line Load Method Stage 1* to “Horz. and Vert. Forces.” This should change it for all the rows.
9. In the *All* row, enter a value of “-2000” for the *V Force or Incl Stage 1* and hit *Enter*. This should enter the value in all the rows. Leave the other values at the default.
10. Click **OK** to exit the dialog.

11 Analysis Options

The only thing left to do before saving and running the model is to set the UTEXAS analysis options.

1. In the Project Explorer, right-click on the “UTEXAS”  model.
2. Select the **Analysis Options** command from the pop-up menu to open the *UTEXAS Options* dialog.
3. In the *Headings* section, enter the following headings:
 - “Underwater Slope”
 - “GMS UTEXAS Tutorial”
4. Change the *Type of Surface Analysis* to “Automatic Search Circular Floating Grid.”
5. Change the *Circle Center X* to “50.”
6. Change the *Circle Center Y* to “25.”
7. Change the *Radius Definition Method* to “Point on Circle.”
8. Change the *Radius Point X* to “52.”
9. Change the *Radius Point Y* to “-26.”
10. Change the *Min Search Grid Spacing* to “0.1.”
11. Change the *Limiting Depth for Circles* to “-100.”
12. Click **OK** to exit the dialog.

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

12 Save the GMS file

Before continuing, save the GMS project file.


1. Select the *File / Save* command.

13 Run UTEXAS

Now it is possible to export and run the model in UTEXAS.



13.1 Export the Model

Do the following to export the model.

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\underwater*.
4. Change the *File name* to “uwslope-run1.”
5. Click **Save**.


13.2 Run UTEXAS

Now that the user has saved the UTEXAS input file, 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 “uwslope-run1.utx” file that was just saved (in the *Tutorials\UTEXAS\underwater* folder).
6. Click **Open**.
7. Press **Save** on *Open file for graphic output* dialog box.
8. When UTEXAS4 finishes, look at the things mentioned in the *Errors, Warnings* window, then close the window.
9. Close the UTEXAS window as well.

13.3 Read the Solution

Now the user needs to read in 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 the file named “uwslope-run1.OUT.”

3. Click **Open**.


The user should now see a line representing the critical failure surface, and the factor of safety. Note that the critical circle exits at the location of one of the reinforcement lines. This problem has several local minima, and the user will see the solution converge to different solutions if the user starts the initial circle near one of the other line loads. The user can adjust the location of the starting circle by dragging the radius arc endpoints using the **Select Point/Node**  tool.

14 Alternate Model

The first part of this tutorial is complete. The user should see a factor of safety of 1.09. Now the user will modify the model to include the affect of the internal reinforcement in the slope from the concrete piers. The compressive force in the piers will decrease from the top to the bottom due to load transfer from the piers to the soil. Including the reinforcement should increase the factor of safety.

15 Reinforcement Lines

15.1 Enter the Points

1. Select the “Reinforcement lines”  coverage to make it the active coverage
2. Right-click on the “Reinforcement lines” coverage.
3. Select the **Attribute Table** command from the pop-up menu to open the *Attribute Table* dialog.
4. In the dialog, make sure the *Feature type* is set to “Points.”
5. Make sure the *Show coordinates* option is turned on.
6. Enter the X and Y coordinates shown in the table below. If the user is viewing this tutorial electronically, the user can copy and paste these values into the GMS spreadsheet.



X	Y
73	-8
73	-13
73	-18
73	-23
73	-28
73	-33
73	-38
81	-8
81	-13

81	-18
81	-23
81	-28
81	-33
81	-38
89	-8
89	-13
89	-18
89	-23
89	-28
89	-33
89	-38
97	-8
97	-13
97	-18
97	-23
97	-28
97	-33
97	-38
105	-8
105	-13
105	-18
105	-23
105	-28
105	-33
105	-38

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

15.2 Connect the Points to Create Arcs

To create the arcs representing the concrete piers:

1. Use the **Zoom**  tool to zoom in on the new points by dragging a box around the points.
2. Select the **Create Arcs**  tool.
3. 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.
4. Using Figure 5 (below) as a guide, click on the points to connect them with arcs to create the slope.

At this point, the user should see something like Figure 5 below.

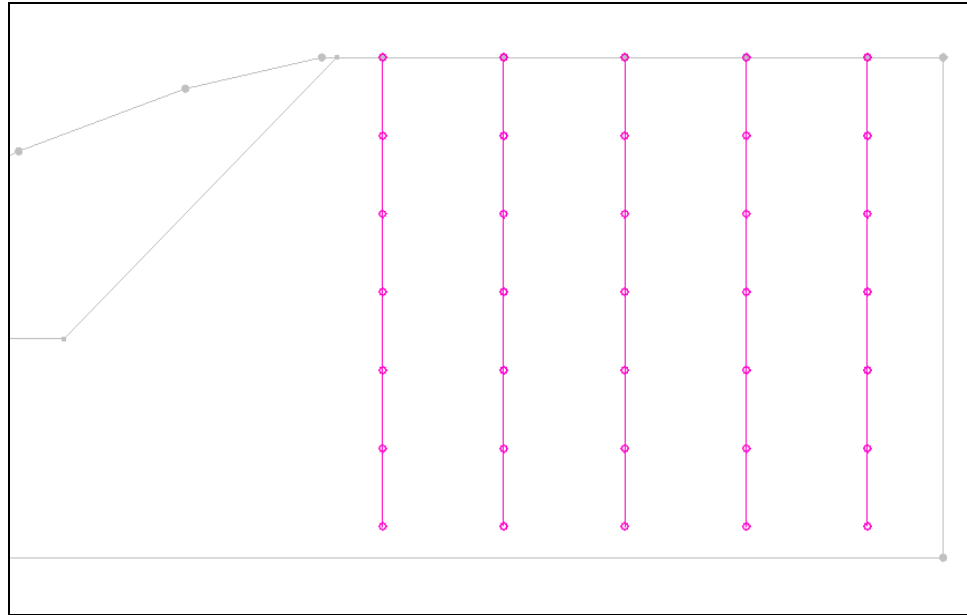



Figure 5 The piers represented as reinforcement lines

15.3 Assign the Node Attributes

The forces on the reinforcement lines are specified at the nodes. We'll assign them now.

1. Switch to the **Select Points/Nodes**  tool.
2. Select all the top nodes by dragging a box around them as shown in Figure 6 below.

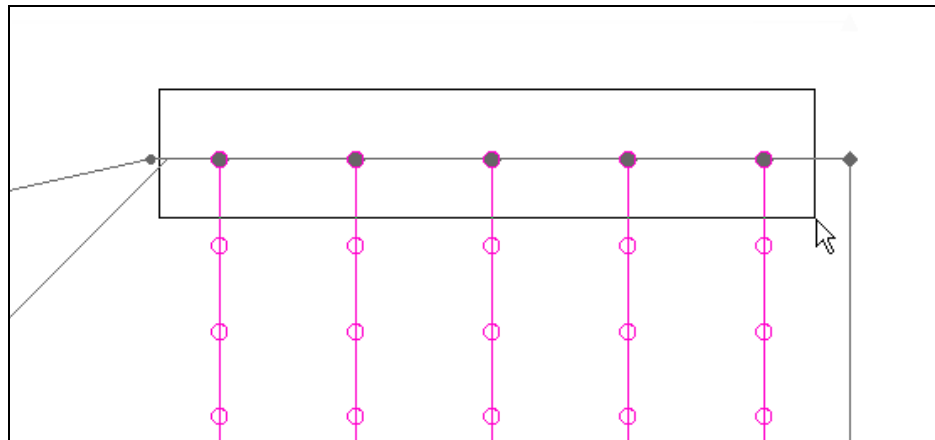




Figure 6 Selecting the top reinforcement nodes by dragging a box

3. Select the **Properties**  button. This brings up the *Attribute Table* dialog.
4. In the *All* row, change the *Long. Force* to be “-2000” and hit *Enter*. This should change the value in all the rows.

5. Leave the *Trans. Force* at “0.”
6. Click **OK** to exit the dialog.

Now the user will repeat the above procedure several times to assign all the node values.

7. Repeat the above steps to assign the *Long. Force* values shown in Figure 7 below.
8. When finished, select the **Frame**  macro.

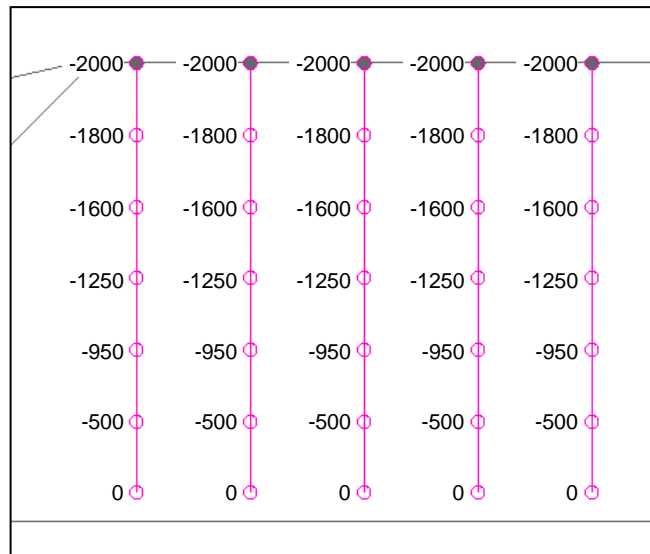


Figure 7 Longitudinal forces assigned to the reinforcement lines

16 Save the GMS file


Before continuing, save the GMS project file.

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

17 Run UTEXAS

Now it is possible to save the changes and run UTEXAS again.



17.1 Export the Model

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\underwater*.
4. Change the *File name* to “uwslope-run2.”
5. Click **Save**.


17.2 Run UTEXAS

Now that the user has saved the UTEXAS input file, 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 “uwslope-run2.utx” file that was just saved (in the *Tutorials\UTEXAS\underwater* folder).
6. Click **Open**.
7. Press **Save** on *Open file for graphic output* dialog box.
8. When UTEXAS4 finishes, look at the things mentioned in the *Errors, Warnings* window, then close the window.
9. Close the UTEXAS window as well.


17.3 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 “uwslope-run2.OUT.”
4. Click **Open**.

18 Compare the Solutions

Now the user has two solutions in memory. The user should see two failure surface lines. Both factors of safety are being displayed, and since they are in the same place, they are obscuring each other. Do the following to look at the solutions one at a time:

1. In the Project Explorer, turn off the “uwslope-run1 (UTEXAS)”  solution.

Now the user is seeing just one of the solutions being displayed.

2. In the Project Explorer, turn on the “uwslope-run1 (UTEXAS)”  solution
3. Turn off the “uwslope-run2 (UTEXAS)”  solution.

The factor of safety for the second case should be about 1.71 (compared to 1.09 for the first model). Both line loads and the reinforcement act in the vertical direction only. The reinforcement lines tend to balance out the effect of the line loads at the surface. In other words, the net load applied at the failure surface is the difference between the line load and the reinforcement line force. The reinforcement loads starts at 2000 at the ground surface and decreases with depth. Thus, the reinforcement load at the top of the slope (in the area where the failure surface intersects the piers) is close to the line load value resulting in a small net load and a larger factor of safety.

19 Conclusion

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

- It is possible to use GMS to build slopes with slope geometry lines for analysis by UTEXAS. Slope geometry lines are helpful in cut or fill scenarios.
- Vertical reinforcement lines can be used to represent concrete piers.