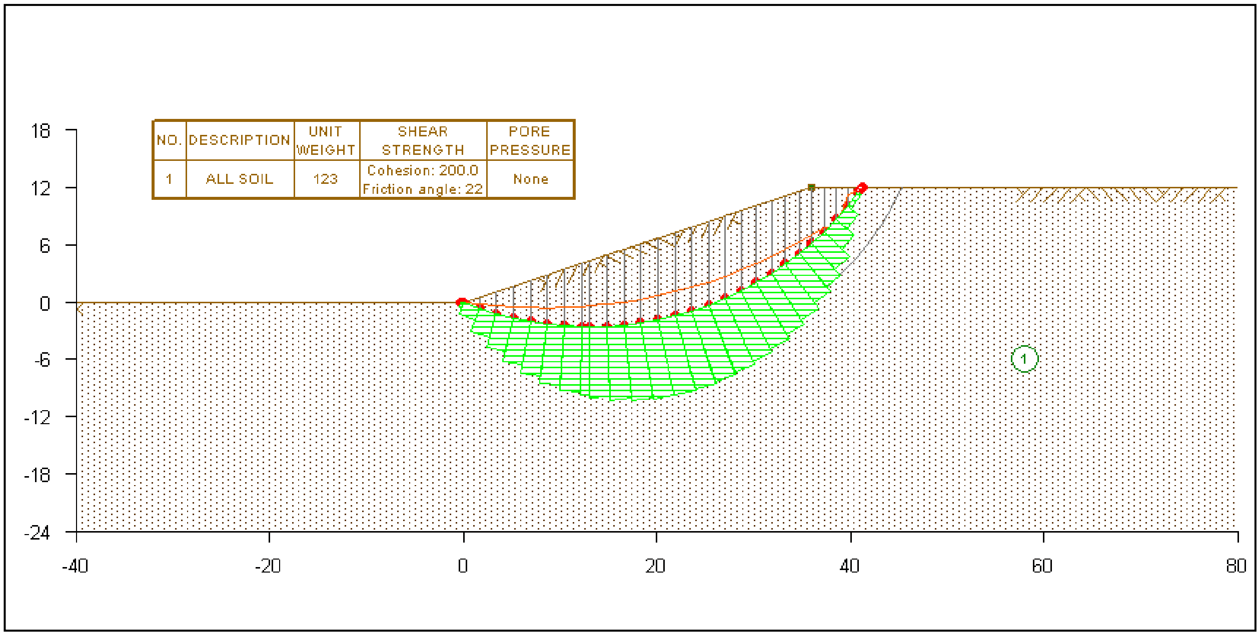
v. 10.0

*GMS 10.0 Tutorial*

# ***UTEXAS – Embankment on Soft Clay***

Introduction to the UTEXAS interface in GMS for a simple embankment analysis




## Objectives

Learn how to build a simple UTEXAS model in GMS.

## Prerequisite Tutorials

- Feature Objects

## Required Components

- GIS
- Map Module
- UTEXAS

## Time

- 30–45 minutes

# 1 Introduction

UTEXAS4 is a slope stability software package created by Dr. Stephen G. Wright of the University of Texas at Austin. UTEXAS4 is used to analyze slope stability using the limit equilibrium method. The user provides the geometry and shear strength parameters for the slope in question and UTEXAS4 computes a factor of safety against slope failure. The factor of safety for a candidate failure surface is computed as the forces driving failure along the surface divided by the shear resistance of the soils along the surface. UTEXAS4 is a state-of-the-art slope stability code and has been widely used in industry for many years.

This tutorial illustrates how to build a simple UTEXAS model in GMS. This tutorial is similar to the "Utexam1.dat" sample file distributed with UTEXAS and tutorial number one in the UTEXAS tutorial manual.[1]

The problem is illustrated on page 1. A simple embankment is being modeled to determine the factor of safety and critical failure surface.

This tutorial uses the GIS feature objects in the GMS Map module to build the geometric input to UTEXAS. The user may wish to complete the "Feature Objects" tutorial prior to beginning 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 examine different approaches for creating the input geometry required by UTEXAS4. Here are the steps to this tutorial:

1. Import an existing UTEXAS4 model into GMS.
2. Create a UTEXAS4 model in GMS by digitizing.
3. Create a UTEXAS4 model in GMS by typing point coordinates.
4. Assign attributes to the model and adjust the analysis options.
5. Save the model, run UTEXAS4 to get a solution, and view the solution in GMS.

## 2 Profile Lines vs. Arcs and Polygons

The input to UTEXAS4 is in the form of a two-dimensional description of the soils and pore water at the site in question (i.e., a vertical “slice” or cross-section). It is assumed that the geometry is relatively constant in the direction perpendicular to the cross-section. UTEXAS4 uses profile lines to define the soil stratigraphy. A profile line is a polyline representing the top of a soil unit. Each profile line is associated with a soil id (a material id). Profile lines are defined from left-to-right, and cannot overlap.

In GMS, the user doesn’t create profile lines—rather, the user creates arcs and polygonal zones from which the profile lines are later extracted automatically. For most users, it’s probably more intuitive to think in terms of soil zones than it is to think in terms of profile lines. GMS, therefore, simplifies the process of creating UTEXAS models.

For example, consider the figure below which is derived from an actual UTEXAS example and represents a dam cross section showing different zones for the shell, core, filter, riprap etc.

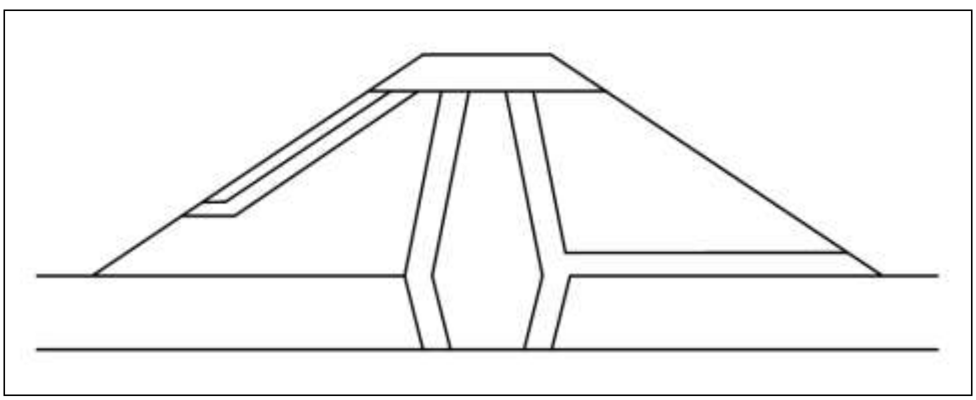


*Figure 1 A cross section diagram of a dam*

The profile lines corresponding to this model are shown in Figure 2 below.

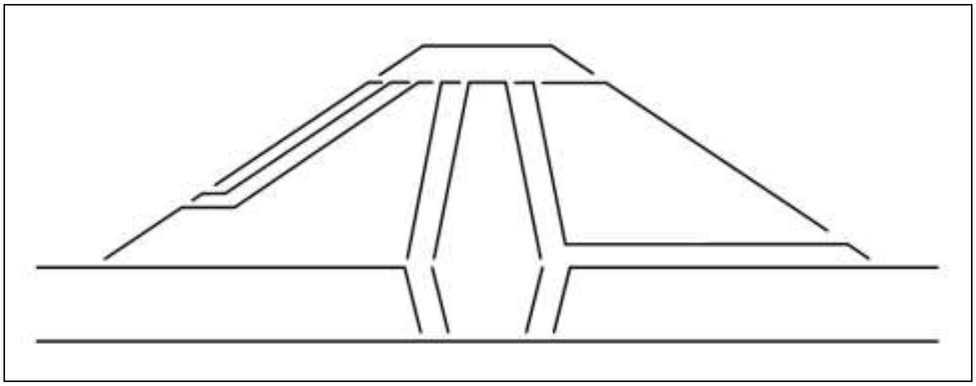


*Figure 2 The profile lines needed to define the cross section*

The profile lines in Figure 2 are shown with the lines separated at the endpoints to help illustrate where the profile lines begin and end (the actual profile lines will touch but not overlap). As the user can see from this example, trying to figure out how to define the profile lines on a complex model can be challenging. In GMS, the model is defined as shown in Figure 1, and GMS automatically determines how to define the profile lines.

# 3 Program Mode

The GMS interface can be modified by selecting a Program Mode. When the user first installs and runs GMS, it is in the standard or "GMS" mode, which provides access to the complete GMS interface, including all of the MODFLOW tools. There is another mode called "GMS 2D" that provides a greatly simplified interface to the SEEP2D and UTEXAS codes. This mode hides all of the tools and menu commands not related to SEEP2D and UTEXAS. This tutorial assumes that the user is operating in the GMS 2D mode. Once the mode is changed, the user can exit and restart GMS repeatedly and the interface stays in the same mode until the user changes it back. Thus, the user only needs to change the mode once if the user intends to repeatedly solve SEEP2D/UTEXAS problems. 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. Then select the *File* | **Exit** command to exit GMS.

# 4 Import an Existing UTEXAS Model

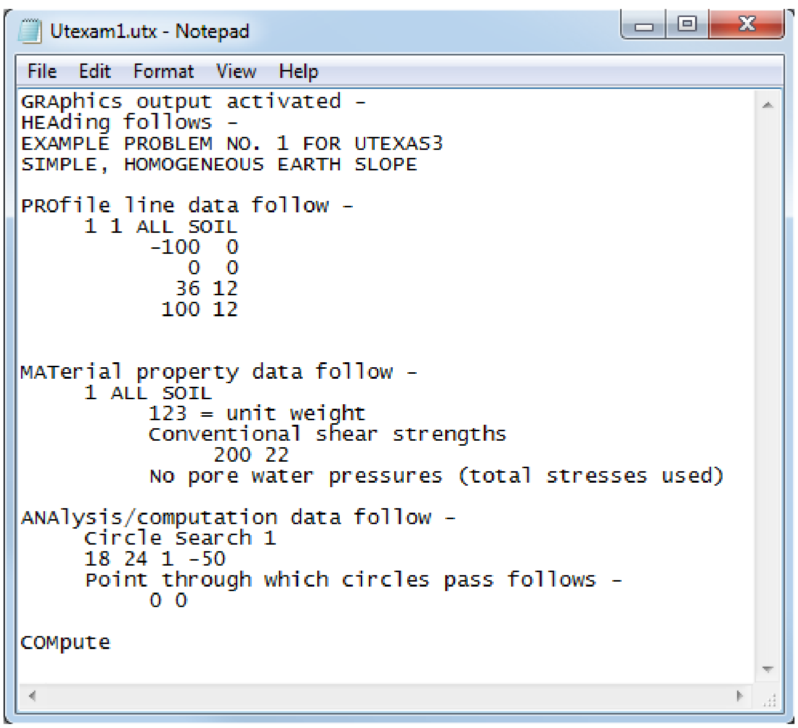
The user will start by importing an existing model into GMS. This model is one of the standard UTEXAS4 example problems. Later in the tutorial, the user will create this same model from scratch.

## 4.1 View the File

First take a look at the file to be imported. It is a plain text file.

1. Launch a text editor such as Notepad.
2. Locate and open the directory entitled *Tutorials\UTEXAS\embankment*.
3. Select the file named "Utexam1.utx."
4. Click **Open**.

The user should now see the file as shown in Figure 3 below:




```
GRAphics output activated -
HEAding follows -
EXAMPLE PROBLEM NO. 1 FOR UTEXAS3
SIMPLE, HOMOGENEOUS EARTH SLOPE

PROfile line data follow -
     1 1 ALL SOIL
          -100  0
             0  0
            36 12
           100 12


MATerial property data follow -
     1 ALL SOIL
          123 = unit weight
          Conventional shear strengths
               200 22
          No pore water pressures (total stresses used)

ANAlysis/computation data follow -
     Circle Search 1
     18 24 1 -50
     Point through which circles pass follows -
          0 0

COMpute
```

*Figure 3 Utexam1.utx file*

This file defines a simple embankment model. The file serves to both define the geometry and provide instructions to UTEXAS4 on how to analyze the problem.

Notice the four main sections of this file: 1) headings, 2) profile lines, 3) material properties, and 4) analysis options. Further notice that there is just one profile line that defines the embankment. The line does not form a closed polygonal area—it is simply a polyline.

Later in the tutorial, the user will create a model from scratch in GMS and export it. The exported file will look similar to the one in Figure 3.

## 4.2 Open the File

Now import this file into GMS. GMS will read and interpret the file and display the embankment.

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. Later in this tutorial, the user will use the coverage options to set up a conceptual model. But in this case, the user is simply importing an existing file so the user can ignore most of the options for now.

3. Select *Open an existing project* and click on the **OK** button
4. Locate and open the directory entitled *Tutorials\UTEXAS\embankment*
5. At the bottom of the *Open* dialog, change the *Files of type* to "All Files (*.*)."
6. Select the file named "Utexam1.utx."
7. Click **Open.**

The user should now see the embankment model as shown in Figure 4 below.

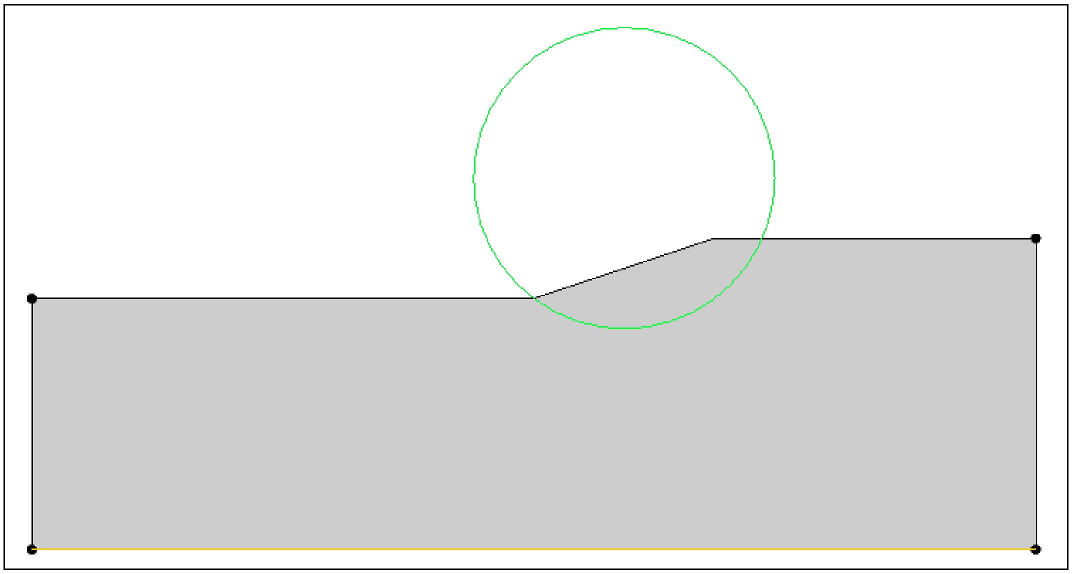


*Figure 4. Embankment in Utexam1.utx file.*

Notice that the filled area represents the soil mass and the slope that are being analyzed. The circle is the starting circle that is used as an initial guess when finding the critical failure surface.

Notice what GMS put in the Project Explorer, as shown in Figure 5 below. GMS created a conceptual model called "Utexam1.utx", a "UTEXAS" model, and a coverage named "Profile lines". These items define the UTEXAS simulation and will be described in more detail below.

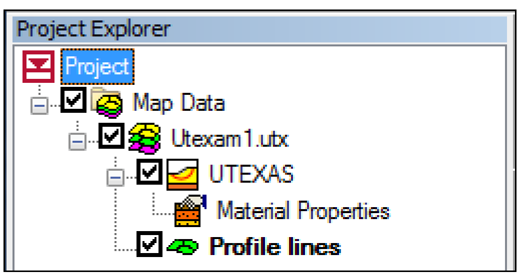



*Figure 5 Project Explorer showing the UTEXAS model*

Hence, if the user has existing UTEXAS input files, the user can import them into GMS to view and edit.

# 5 Save the GMS Project File

Before continuing, save the changes to a GMS project file:

1. Select the **Save** button. This brings up the *Save As* dialog.

2. Enter a name for the project file (e.g., “embank-utexas.gpr”).

3. Select the **Save** button.

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

# 6 Create the Embankment Graphically

Now the user can create the same model from scratch using the GMS interface.

## 6.1 Create the Conceptual Model

The first step is to create a conceptual model. A conceptual model consists of one or more layers called “coverages” containing GIS feature objects (points, lines, and polygons) defining the model geometry. This process can be done automatically in GMS 2D mode.

1. Select the *New* button to delete the current project and start from scratch. If asked to save the changes, select **No**.

2. Change the *Conceptual model name* to “Embankment.”

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

At this point, the user would normally select a set of coverages corresponding to the features/processes that the user wishes to include in the model. However, this model is so simple the user needs only two coverages.

4. Select the following coverage options:

    - *Profile lines*
    - *Starting circle*

5. Click the **OK** button.

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

## 6.2 Turn on Snapping

Before drawing the embankment, the user will turn on the *snap* option so the points clicked out will snap to the nearest integer coordinate.

1. Select the **Display Options** button to open the *Display Options* dialog.

2. Select the **Drawing Grid** option in the list at the upper left.

3. Change the *Spacing* to "2.0" (all of the coordinates are a multiple of 2).
4. Turn on the *Snap* option.
5. Click **OK** to exit the dialog.

## 6.3 Set the Window Bounds

The embankment coordinates are as shown in the figure below.

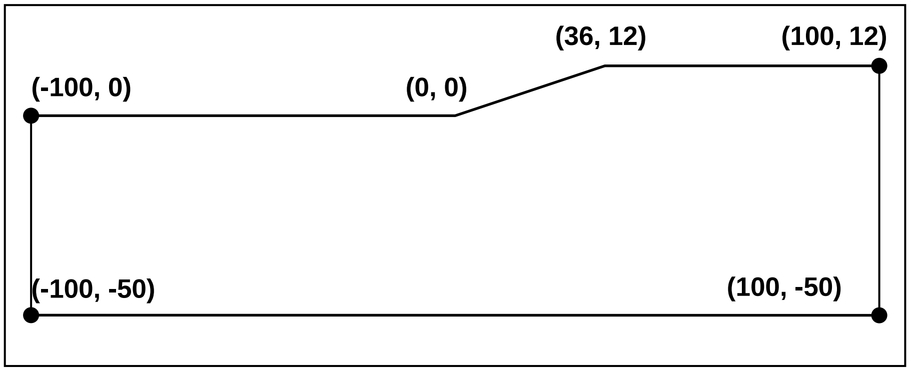



*Figure 6. Embankment coordinates.*

Before the user can draw the embankment, it is necessary to set the view so that it includes this space.

1. Select the *Display* | *View* | **Window Bounds** menu command.
2. Select the third option, *X and Y range to be specified (alters aspect ratio)*.
3. Set *X at left* to "-110.0."
4. Set *Y at top* to "20.0."
5. Set *X at right* to "110.0."
6. Set *Y at bottom* to "-110.0."
7. The coordinates should match those shown in the figure below.
8. Click **OK**.

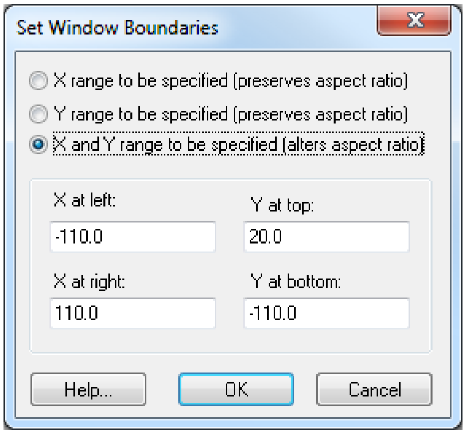



*Figure 7 Setting the Window Boundaries*

## 6.4 Create Arcs and Polygons

Now the user will create the arcs and build a polygon.

1. Click on the "Profile lines" coverage to make it active.
2. Select the **Create Arcs** tool.
3. Use the tracking coordinates in the bottom of the *Graphics Window* to position the mouse in the right place for each click (see the figure below).

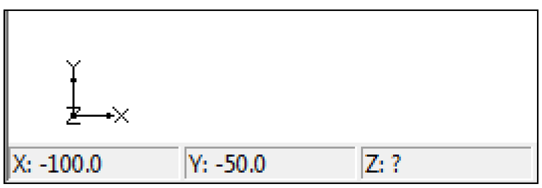



*Figure 8. Tracking coordinates.*

4. Click out the two arcs shown in the figure below. Double-click to end each arc. If the user accidentally clicks somewhere and is still in the process of drawing the arc, the user can hit the backspace key, which will delete the point last clicked. If the user finishes the arc and some points are in the wrong place, use the **Select Node** tool or the **Select Vertex** tool to move the points to the correct place.

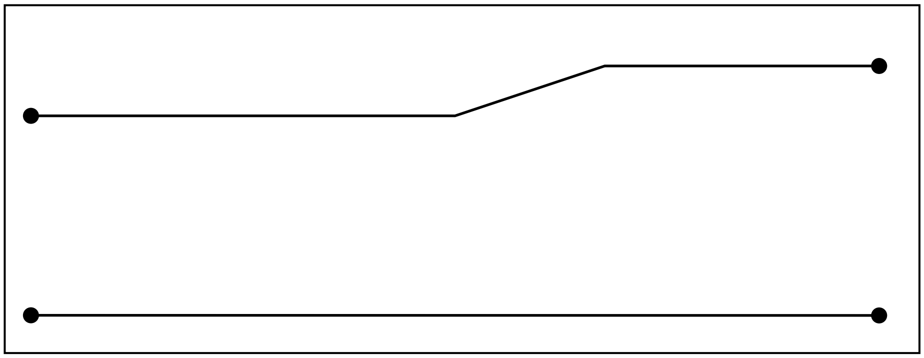


*Figure 9 Drawing the top and bottom arcs*

5. Using the *Create Arcs* tool, draw the vertical side arcs as shown in the figure below.

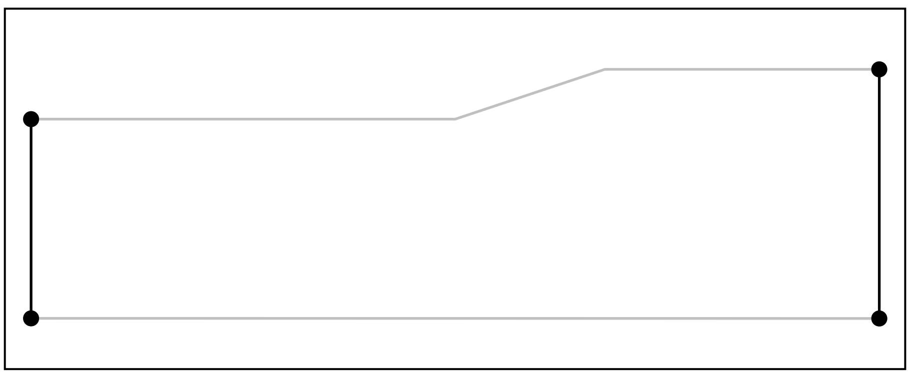


*Figure 10 Drawing the side arcs*

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

# 7 Create the Embankment by Entering Text Coordinates

Another way to define the model is to enter the coordinates directly. If the user already knows the coordinates of the profile lines, it may be easier to just type in those coordinates instead of clicking out the profile lines with the mouse. Or, if the user already has the coordinates entered in a file, the user can copy and paste them in to GMS. The user will do that now.

First delete the arcs that were just created.

1. Switch to the **Select Arcs** tool.

2. Select the *Edit* | **Select All** menu command to select all the arcs or drag a box around the arcs.

3. Hit the *Delete* key to delete all the arcs.

4. In the Project Explorer, right-click on the “Profile lines”  coverage.

5. Select the **Attribute Table** command from the pop-up menu to open the *Attribute Table* dialog.

6. Change the *Feature type* to “Points.”

7. Make sure the *Show coordinates* option is turned on.

8. Enter the X and Y coordinates show in the table below. If you are viewing this tutorial electronically, copy and paste these values into the GMS spreadsheet. Don’t worry about the z values. They can be left at zero since they are ignored by UTEXAS.

| X | Y |
|---|---|
| -100 | 0 |
| 0 | 0 |
| 36 | 12 |
| 100 | 12 |
| -100 | -50 |
| 100 | -50 |

9. Verify that the dialog looks like the figure below.

10. Click **OK**.

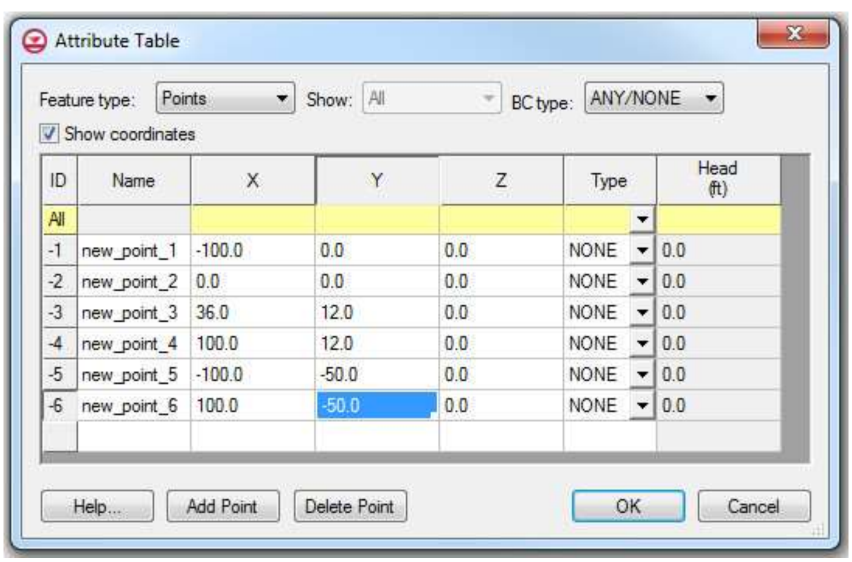



*Figure 11 Typing in the point coordinates*

The user should now see the points on the screen.

11. Select the **Create Arcs** tool.

12. 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 whenever you want to stop creating arcs.

13. Click on the points to connect them with arcs to create the same embankment created before.

14. Select the **Build Polygons** icon in the map toolbar.

At this point, this tutorial has shown that there is more than one way to create a UTEXAS model in GMS. In the "UTEXAS – Dam Profile Analysis" tutorial, the user will learn yet another way, which is to import a CAD file.

# 8 Material Properties

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

1. In the Project Explorer, double-click on "Material Properties" under "UTEXAS". This will open the *Materials* dialog.

2. Change the material properties to the following:

| Unit Weight Stage 1 | Shear Strength Method Stage 1 | Cohesion Stage 1 | Angle of Internal Friction Phi Stage 1 |
|---|---|---|---|
| 123 | Conventional | 200 | 22 |

3. Leave all the other settings at the defaults.

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

For most slope stability problems, the objective is to find the failure surface with the minimum factor of safety. The user will be using a circular failure surface, and the user will let UTEXAS perform an automatic search for the critical failure surface. To begin this process, the user must supply the size and position of the initial circle (i.e., the "starting circle"). This can be accomplished either by entering some numerical parameters defining the circle location in the *Analysis Options* dialog (see the following section) or by graphically entering a circle. This tutorial will use the graphical option.

When the user set up the conceptual model, he or she created a starting circle coverage as a placeholder for the starting circle. The location of the starting circle is defined by creating a single arc segment corresponding to the radius of the circle. The first point on the arc corresponds to the circle center and the ending point corresponds to a point on the perimeter of the circle. To create the circle:

1. Click on the "Starting circle" coverage to make it active.

2. Select the *Create Arc* tool .

3. Create the arc by clicking on a starting point somewhere above the slope and double-clicking on a point near the toe of the slope as shown in Figure 12.

Once the circle is created, it can be edited by dragging the nodes at the ends of the arc.

4. Select the **Select Point/Node** tool.

5. Click on the node at the center of the circle (the first end point) and drag it to a new location. Note how the circle changes.

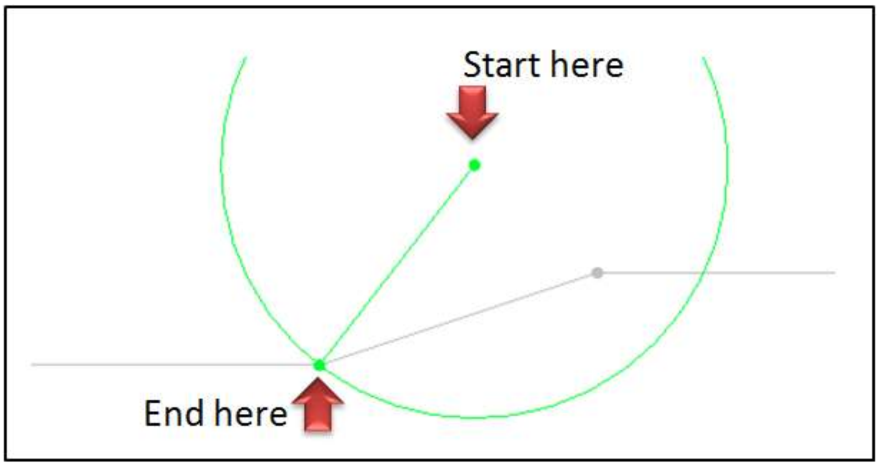



*Figure 12 Creating the starting circle*

# 9 Analysis Options

The only thing left to do before saving and running the model is to set the UTEXAS analysis options. The user will assume a circular failure surface and will let UTEXAS search for the critical circle. The user will provide the initial or starting circle by defining the circle center and a point through which the circle passes.

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:

   - "Embankment Model"

   - "GMS UTEXAS Tutorial"

3. Change the *Type of Surface Analysis* to "Automatic Search Circular Floating Grid."

4. Change *Circle Center X* to “18.0.”

5. Change *Circle Center Y* to “24.0.”

6. Make sure all the other entries match those in the table below.

| Category | Value |
|---|---|
| Radius Definition Method | Point on Circle |
| Radius Point X | 0.0 |
| Radius Point Y | 0.0 |
| Min. Search Grid Spacing | 1.0 |
| Limiting Depth for Circles | -50.0 |
| Specify Lowest Elev. for Circle Centers | Off |
| Lowest Elev. for Circle Centers | 0.0 |
| Tension Crack | Off |
| Slope to Analyze | Automatic |
| Seismic Coefficient | 0.0 |
| Seismic Force Location | Center of gravity |
| Output Format | Long |
| Opposite Sign Convention | Off |
| Stability Analysis Procedure | Spencer’s |
| Number of Stages | 1 |

7. The options should match those shown in the dialog below. Don’t bother changing the coordinates of the circle. The center point coordinates and the coordinates of the point on the circle at the toe may not correspond exactly to the values shown but that is OK since the starting circle is just a starting location for the automated search.

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

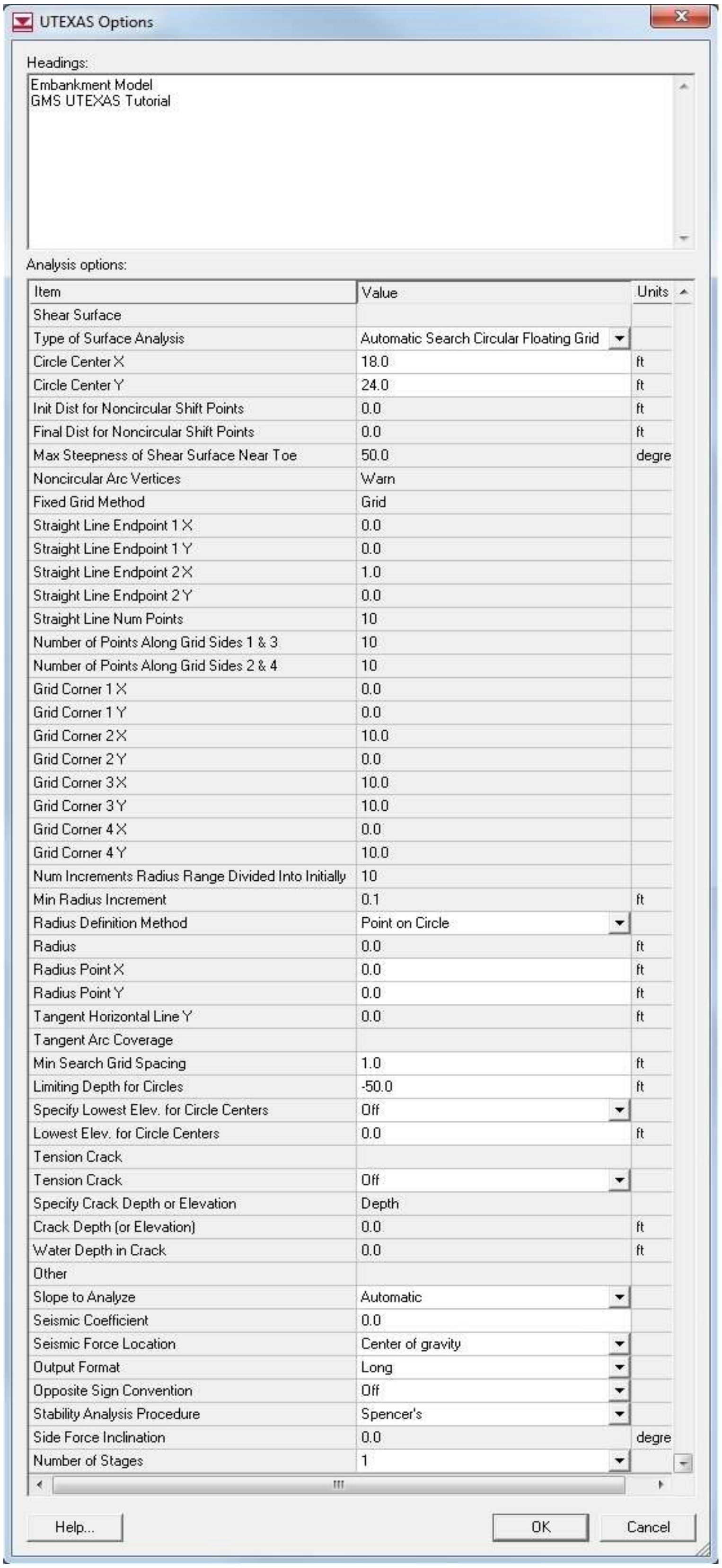



*Figure 13 UTEXAS Options*

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

Note that while GMS supports many of the analysis options that are available in UTEXAS, some of the options are not supported.

## 10 Export the Model

Now it is possible to export the UTEXAS input file prior to running 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\embankment*.
4. Change the *File name* to "embank-utexas."
5. Click **Save**.

The user has now created a UTEXAS input file called "embank-utexas.utx." The user may want to open this file in a text editor and examine its contents. The user could also compare it to the "Utexam1.utx" file that was imported earlier.

## 11 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** button.
4. Change the *Files of type* to "All Files (*.*)."
5. Locate the "embank-utexas.utx" file that was just saved in the *Tutorials\UTEXAS\embankment* 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 two windows.

# 12 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 "embank-utexas.OUT."
4. Click **Open**.

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

# 13 Display Options

Now take a look at the UTEXAS display options in GMS.

1. Select the **Display Options** button to open the *Display Options* dialog.
2. Make sure the *Map Data* item is selected in the list in the upper left of the dialog.
3. Select the *UTEXAS* tab. The dialog should appear as shown in the figure below.

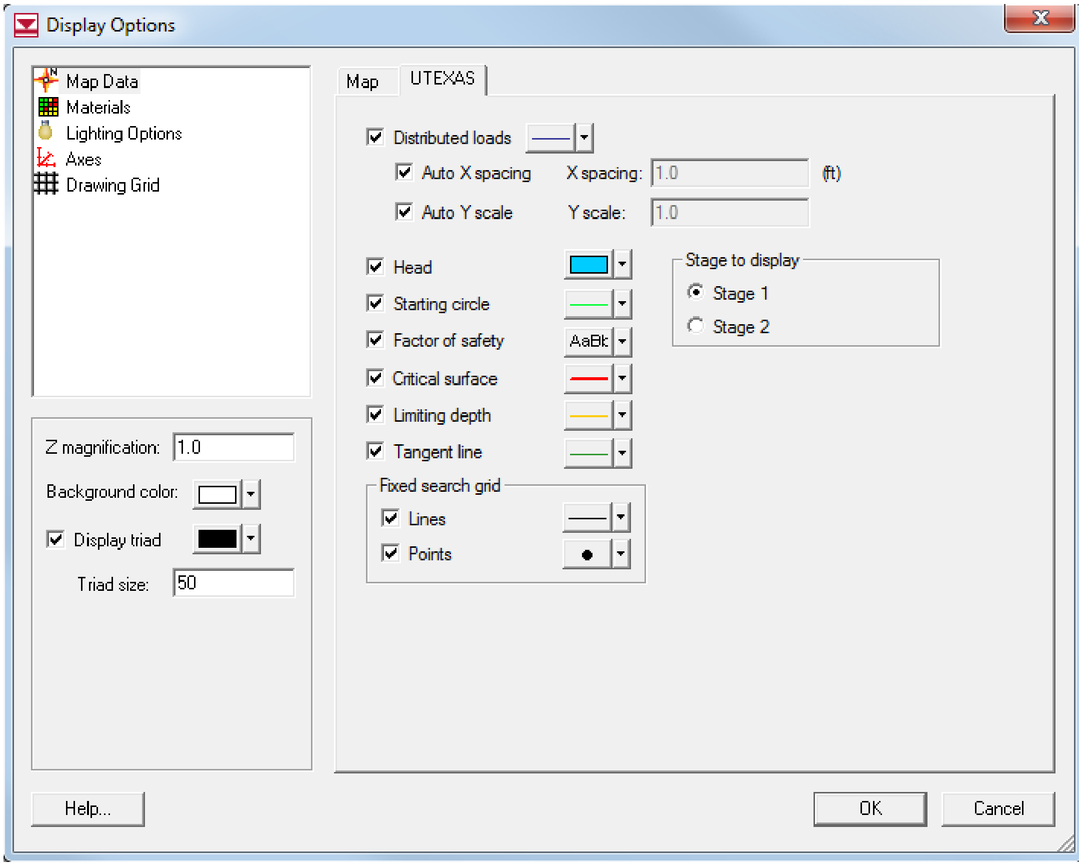



*Figure 14 UTEXAS Display Options*

4. Try changing the display options and see how it affects the display.

## 14 Conclusion

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

- UTEXAS uses the profile line approach to define the model geometry, but GMS uses arcs and polygons. GMS automatically extracts the profile lines from the arcs and polygons when exporting the UTEXAS input file.
- GMS can read UTEXAS input files created outside of GMS.
- The user can create a UTEXAS model in GMS by drawing the arcs or by entering in the coordinates of the profile lines and connecting the points with arcs.