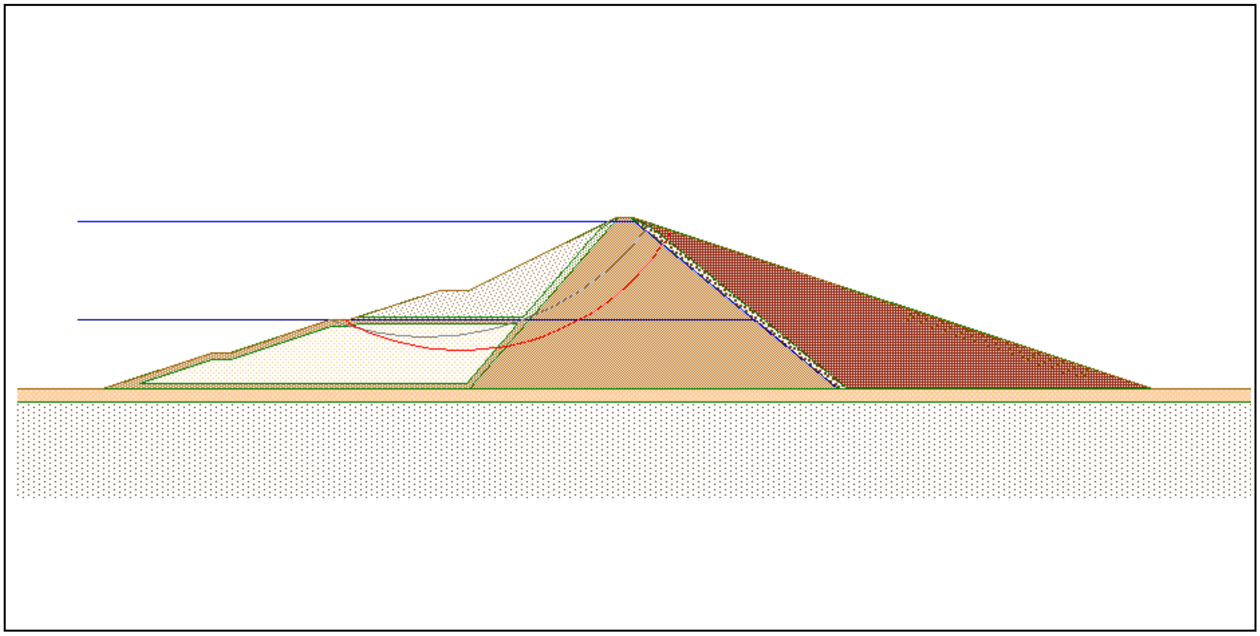
v. 9.2

GMS 9.2 Tutorial

UTEXAS – Dam Profile Analysis

Use UTEXAS to find the critical failure surface for a complex earth dam



Objectives

Import a CAD file and convert it to Feature Objects. Use the Feature Objects to create a UTEXAS conceptual model. Finally, run UTEXAS and view the results.

Prerequisite Tutorials

- None

Required Components

- GIS
- Map
- UTEXAS

Time

- 30-60 minutes

1 Contents

2 Introduction

This tutorial illustrates how to perform a UTEXAS slope stability analysis of a complex earth dam using GMS. This tutorial is similar to tutorial number nine 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. The figure shows a complex earth dam with many zones and materials. Two piezometric lines are shown: one at the reservoir-full level and one at about the half-full level. The dam will be analyzed for stability under rapid drawdown conditions using a multi-stage approach. Since the geometry is complex, we will import the geometry from an existing CAD file instead of creating it from scratch in GMS. The process of building the geometry from scratch is described in the other UTEXAS tutorials. This also provides an opportunity to learn how to work with CAD data since many cross-sections are available in CAD files.

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

2.1 Outline

Here's what we will do in this tutorial:

1. Import a CAD file defining the dam cross section.
2. Convert the CAD data to *Feature Objects*.
3. Create a UTEXAS conceptual model. The UTEXAS model will be a multi-stage model.
4. Run UTEXAS.
5. View the results.

2.2 Required Modules/Interfaces

You will need the following components enabled to complete this tutorial:

- Map, GIS
- UTEXAS

You can see if these components are enabled by selecting the *Help* | *Register* command.

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 organized into a set of layers or groups called *coverages*. Normally, we would select the set of coverages to use based on the types of features we want to include in our simulation and GMS would create an empty coverage for each of the types selected and we would create features inside of each coverage. In this case, however, we will be importing the GIS features from a CAD file and the coverages are created as part of the import process. So we will not select any of the pre-defined coverages

2. Change the *Conceptual model name* to **Multi-Stage Dam.**

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

4. Uncheck the *Profile lines* option and click *OK*.

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

5 Save the GMS Project File

Before continuing, we will save what we have done so far to a GMS project file:

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

2. Navigate to the **Tutorials/UTEXAS/dam profile** folder.

3. Enter a name for the project file (ex. "**profile-utexas.gpr**") and select the *Save* button.

You may wish to select the *Save*  button occasionally to save your work as you continue with the tutorial.

6 Import the CAD File

The cross section of the dam we are examining in this tutorial is complex. Fortunately we have a CAD file of the cross section. Instead of creating the geometry in GMS, we'll simply import the CAD file and convert the components of the CAD drawing to GIS features in our conceptual model.

1. Select the *Open File* button.
2. Change the *Files of type* to **All Files (*.*)**.
3. Locate and open the directory entitled **Tutorials\UTEXAS\dam_profile**.
4. Open the file named **dam_profile.dxf**.

GMS displays the CAD file. It should appear as shown in Figure 1 below:

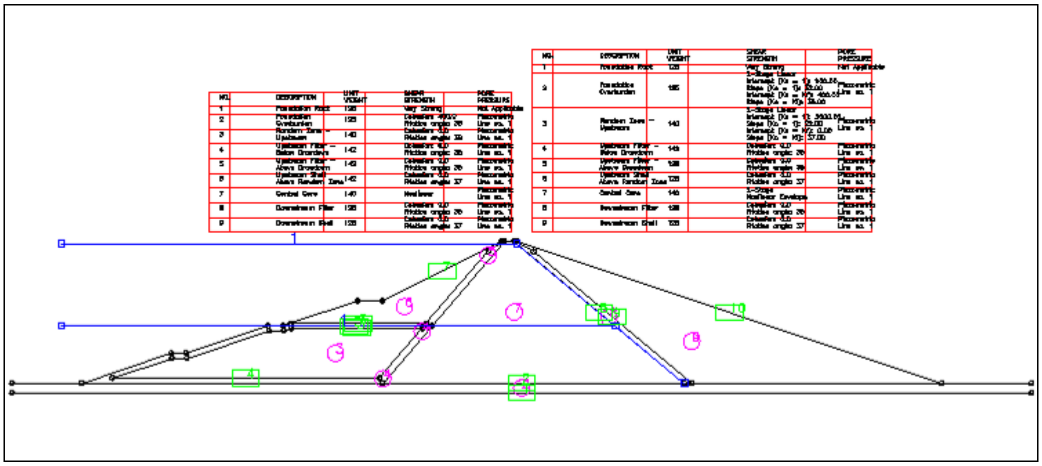


Figure 1. GMS after importing the CAD file.

7 CAD → Feature Objects

We need to convert the CAD data to *feature objects* so that we can manipulate them in GMS.

We'll create three coverages; one for the profile lines, one for the piezometric line for stage 1, and one for the piezometric line for stage 2.

1. In the *Project Explorer*, right-click on the *CAD* folder and select the *CAD To | Feature Objects* command from the pop-up menu.

There is more information in the CAD file than we need for our model. We'll convert only the CAD layers dealing with the profile lines and the piezometric line.

2. In the *CAD → Feature Objects* dialog, uncheck all the layers except for the *Profile_Lines* layer. Also change the *Coverage name* to **Profile lines**. The dialog should look as shown in Figure 2 below.

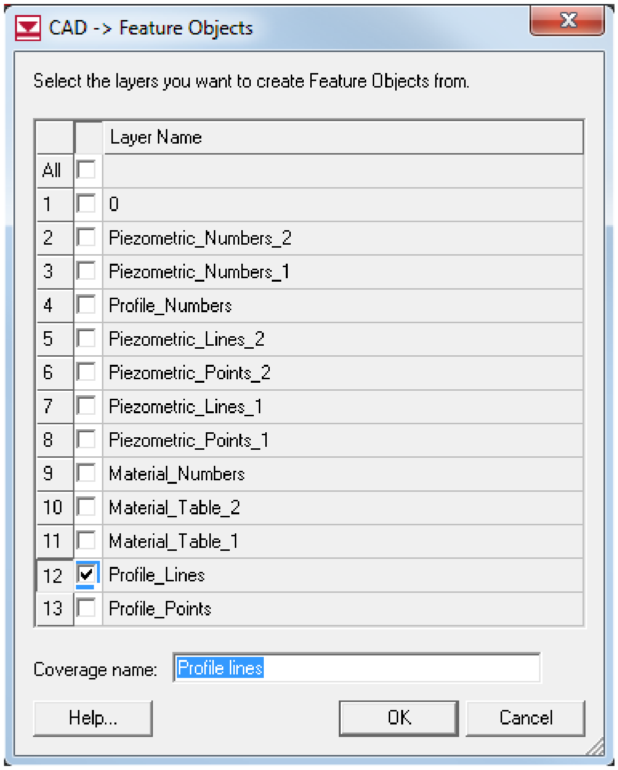


Figure 2. CAD -> Feature Objects dialog.

3. When you've made the changes as shown, click *OK*.

4. Now repeat steps 1 – 3 above to create another coverage, but this time convert only the *Piezometric_Lines_1* CAD layer. Name the coverage **Piezometric line 1**.

5. Repeat steps 1 – 3 above again to create another coverage, but this time convert only the *Piezometric_Lines_2* CAD layer. Name the coverage **Piezometric line 2**.

We're now done with the CAD data, so we'll delete it.

6. In the *Project Explorer*, select the *CAD* folder and hit the *Delete* key.

The *Graphics Window* should now look similar to Figure 3 below.

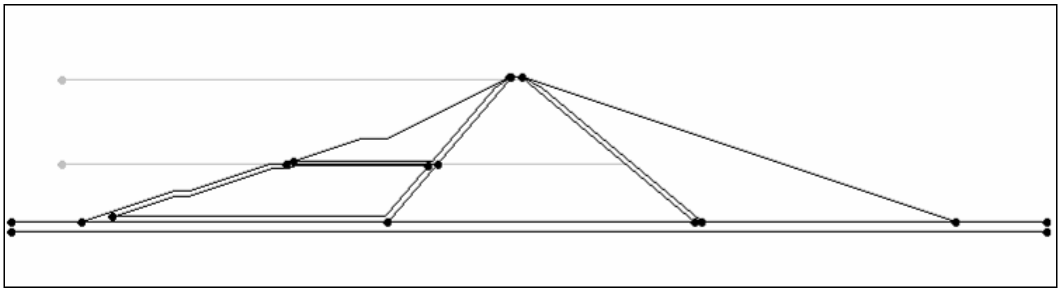


Figure 3. Feature objects in the graphics window.

8 Move coverages to the Conceptual Model

The CAD to feature objects conversion process created three new coverages, but they are generic coverages and are not yet associated with our conceptual model. We now need to organize the new coverages so that they are part of the conceptual model.

1. In the *Project Explorer,* drag the *Profile lines* coverage and drop it below the conceptual model.
2. Click *Yes* if GMS warns you that moving the coverage can change the attribute table.
3. Also drag the *Piezometric line* 1 coverage to be beneath the conceptual model. Again click *Yes* at the prompt.
4. Finally, drag the *Piezometric line* 2 coverage to be beneath the conceptual model. Again click *Yes* at the prompt.

The *Project Explorer* should now look like Figure 5 below. The order of the coverages is not important, but it is important that they are underneath the *Multi-Stage Dam* conceptual model.

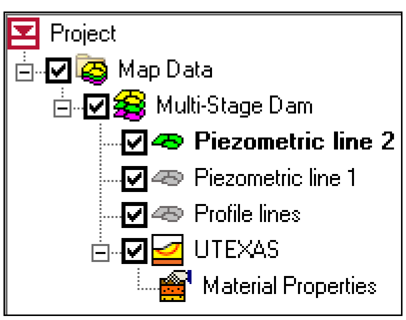


Figure 5. The Project Explorer.

9 Complete the Dam Geometry

We need to make a few modifications to the new feature objects before we can begin to assign attributes.

9.1 Clean the Coverage

Before we begin manipulating the features, we will run the *Clean* command. This command does several things to help clean up any errors in the geometry, including snapping nodes together that are close but don't quite touch, removing overlapping arcs, etc.

1. In the *Project Explorer*, select the *Profile lines* coverage to make it the active coverage.

2. Select the *Feature Objects* | *Clean* menu command.

3. In the *Clean Options* dialog, accept the default settings and click *OK*.

We don't need to clean the piezometric line coverages since they just contain a single arc.

9.2 Add a Bottom Arc

At this point the dam appears to sit on a very thin layer of material. Obviously there is some type of material below that but it is not represented in our model. With the profile line approach, the model is correct as is and the bottom profile line represents all the material below the dam. However, GMS doesn't use profile lines, it uses soil zones, so we need to add another zone below the model.

1. Select the *Create Arcs* tool.

2. Create a somewhat horizontal arc somewhere below the dam, as shown in Figure 66 below. Don't worry about making the line perfectly horizontal or the exact coordinates of the endpoints.

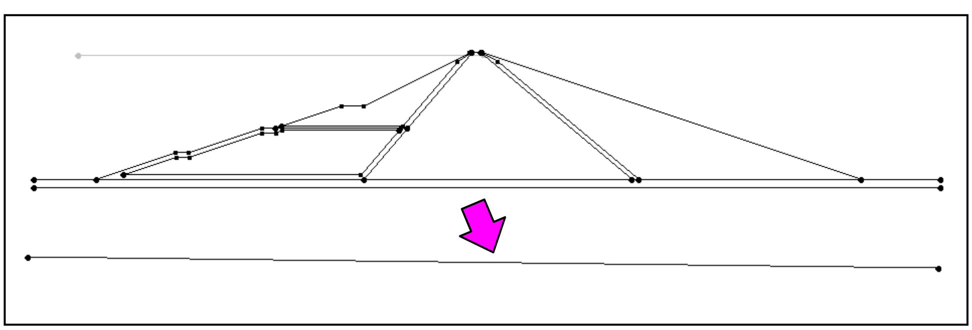


Figure 6. Drawing the bottom arc.

3. Switch to the *Select Node* tool.

4. Select the node on the left end of the new bottom arc and in the *Edit Window*, change its XY coordinates to **(400, 0)**.

5. Select the node on the right end of the new bottom arc and in the *Edit Window*, change its XY coordinates to **(2450, 0)**.

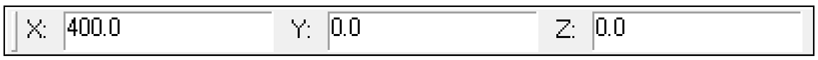


Figure 7. The Edit window.

9.3 Close the Polygons

Since GMS uses polygons to define material zones, we need to close the zones below the dam by adding arcs on the left and right sides.

1. Switch to the *Create Arcs* tool.

2. Create four arcs, two on the left and two on the right of the model by clicking on the ends of the existing arcs to connect the nodes, as shown in Figure 8 below.

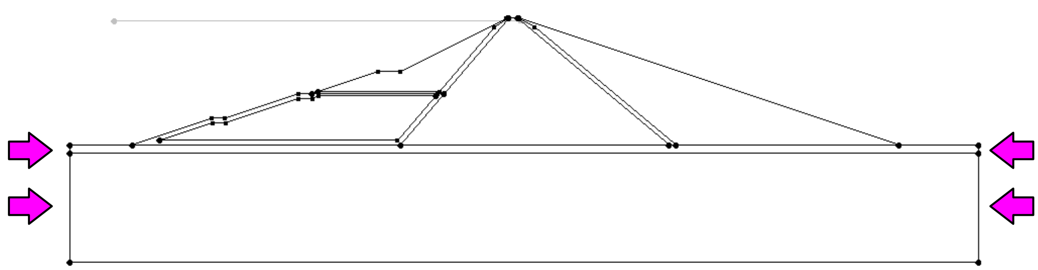


Figure 8. Closing the bottom zone polygons.

9.4 Build Polygons

In GMS, you must always explicitly build polygons before a polygon object exists. It is not enough to simply have a closed loop. We'll do that now.

1. Select the *Build Polygons* icon.

Notice that the polygonal zones have been filled with color.

10 Piezometric Lines

Thus far we've been working in the *Profile lines* coverage. We need to set up the other coverages to work as piezometric lines. When dealing with an upstream water surface in UTEXAS, the water must be represented as a distributed load in order to ensure that the total stresses will be properly calculated. This can be done either by explicitly defining surface loads on the arcs corresponding to the submerged ground surface or by marking the "distributed load" option on the associated piezometric line. With the second option, UTEXAS automatically calculates the distributed loads and applies them to the proper profile lines. We will use the second option.

1. In the *Project Explorer*, double-click on the *Piezometric line 1* coverage.
2. In the *Coverage Setup* dialog, in the list of *Sources/Sinks/BCs*, turn on the following properties:
   - *Piezometric Line*
   - *Distributed Load*
3. Click *OK* to exit the dialog.
4. Make sure the *Select Arc* tool is the current tool.
5. Double-click on the arc that is the piezometric line – the only arc in the coverage.
6. In the *Properties* dialog, note that the *Type* is automatically set to **piezometric line**.
7. Turn on *Dist. Load Stage 1*. Note: you may need to scroll to the right or expand the *Properties* dialog in order to see this option.
8. Click *OK* to exit the dialog.
9. Repeat steps 1 – 8 above but on the *Piezometric line 2* coverage and turning on *Dist. Load Stage 2* instead of *Dist. Load Stage 1*.

That's all we need to do with the piezometric lines. We'll go back to the *Profile lines* coverage.

10. In the *Project Explorer*, click on the *Profile lines* coverage to make it active.

11 Material Properties – UTEXAS Stage 1

Now we need to specify the UTEXAS material properties. Since we are performing a two-stage analysis, we need to enter two complete sets of material properties, one for each stage. We'll enter the stage 1 properties now and the stage 2 properties later.

1. In the Project Explorer, double-click on *Material Properties* under *UTEXAS*.

This brings up the GMS *Material Properties* dialog.

2. Double click on **material_1** in the spreadsheet.
3. Rename it "**Foundation Rock**".

The *Foundation Rock* material is considered very strong and there are just a few properties we have to specify.

4. Make sure the *Show stage 1* option is **on**, and the *Show stage 2* option is **off** at the bottom of the *Materials* dialog.
5. In the row for the *Foundation Rock* material, change the *Unit Weight Stage 1* to **125**.
6. In the same row, set the *Shear Strength Method Stage 1* to **Very Strong material**.

To create the next material:

7. Type "**Foundation Overburden**" in the blank row at the bottom of the spreadsheet and hit the *Return* key.

The rest of the materials will use the conventional method of defining shear strength.

8. In the row for the *Foundation Overburden* material change the *Unit Weight Stage 1* to **125**.
9. In the same row, change the *Shear Strength Method Stage 1* to **Conventional**.
10. Change the *Cohesion Stage 1* to **400**.
11. Change the *Angle of Internal Friction Phi Stage 1* to **38**.
12. Change the *Pore Water Pressure Method* to **Piezometric Line**.
13. Click on the button in the *Piezometric Line Coverage Stage 1* column. This brings up the *Select Coverage* dialog.
14. Select the coverage named "*Piezometric line 1*" under the *Multi-Stage Dam conceptual model.*
15. Click *OK* to exit the *Select Coverage* dialog

Now we'll do the rest of the materials.

16. Repeat the steps above to assign the appropriate properties to the remaining materials as given in the following table.

Material Properties – UTEXAS Stage 1						
Id	Name	Unit Weight	Shear Strength		
			Method	Cohesion c	Friction angle
1	Foundation Rock	125	Very Strong	NA	NA
2	Foundation Overburden	125	Conventional	400	38
3	Random Zone – Upstream	140	Conventional	0	36
4	Upstream Filter – Below Drawdown	142	Conventional	0	35
5	Upstream Filter – Above Drawdown	142	Conventional	0	35
6	Upstream Shell Above Random Zone	142	Conventional	0	37
7	Central Core	140	Nonlinear Mohr-Coulomb Envelope*	*	*
8	Downstream Filter	128	Conventional	0	35
9	Downstream Shell	128	Conventional	0	37
*Described in the following steps					

17. For all the materials (except for Foundation Rock) set the *Pore Water Pressure Method Stage 1* to **Piezometric Line.**

18. Make sure the *Piezometric Line Coverage Stage 1* column for all the materials is set to the "*Piezometric line 1*". (Since there's more than one piezometric line coverage GMS couldn't make this assignment automatically. You can save time by using the *All* row at the top of the spreadsheet. Click the button in this row and select the coverage from the dialog.)

11.1 Central Core

The *Central Core* material uses a nonlinear curve to determine its shear strength. To define it:

1. In row 7, in the *Nonlinear Mohr Coulomb Envelope Stage 1* column, click on the button.

2. In the *XY Series Editor*, enter the values in the table below. The dialog should appear as shown in Figure 9.

Normal Stress	Shear Stress
0.0	0.0
1100.0	1100.0
1250.0	1150.0
3250.0	2200.0
20000.0	13500.0

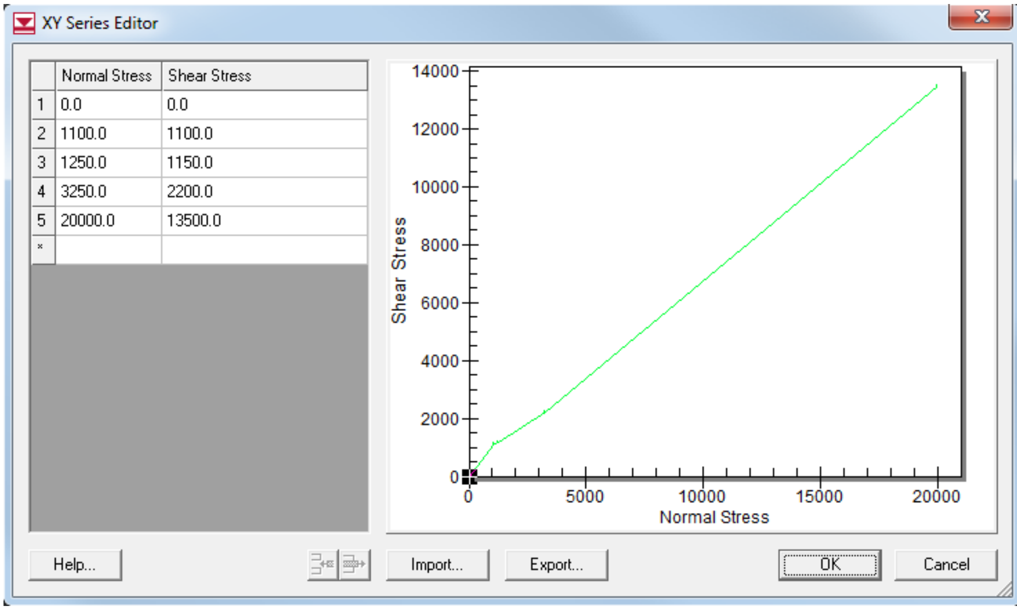


Figure 9. XY Series Editor.

3. When finished, click *OK* to exit the *XY Series Editor*.

12 Material Properties – UTEXAS Stage 2

Now we'll enter all the material properties for stage 2.

1. Turn **on** the *Show stage 2* checkbox.
2. Turn **off** the *Show stage 1* checkbox.
3. Apply what you learned in the previous section on how to assign material properties to assign the stage two properties given in the following table. If you are viewing an electronic form of this document, you can cut and paste some of these numbers (unit wt, cohesion, friction angle).

Material Properties – UTEXAS Stage 2					
Id	Name	Unit Weight	Shear Strength		
			Method	Cohesion c	Friction angle
1	Foundation Rock	125	Very Strong	NA	NA
2	Foundation Overburden	125	2-stage Linear*	NA	NA
3	Random Zone – Upstream	140	2-stage Linear*	NA	NA
4	Upstream Filter – Below Drawdown	142	Conventional	0	35
5	Upstream Filter – Above Drawdown	128	Conventional	0	35
6	Upstream Shell Above Random Zone	128	Conventional	0	37
7	Central Core	140	2-stage Nonlinear*	*	*
8	Downstream Filter	128	Conventional	0	35
9	Downstream Shell	128	Conventional	0	37
*Described in the following steps					

4. For all the materials (except Foundation Rock) set the *Pore Water Pressure Method Stage 2* to **Piezometric Line.**

5. For all the materials, set the *Piezometric Line Coverage Stage 2* to **Piezometric line 2** coverage. (Since there's more than one piezometric line coverage GMS couldn't make this assignment automatically. You can save time by using the *All* row at the top of the spreadsheet. Click the button in this row and select the coverage from the dialog.)

12.1 *Foundation Overburden* and *Random Zone – Upstream* Materials

For the *Foundation Overburden* and *Random Zone – Upstream* materials, the shear strength method is *2-stage Linear*.

1. Enter the values in the table below for the *2-stage Linear* properties for the two materials. If you are viewing an electronic form of this document, you can cut and paste these numbers.

Stage 2 Material Properties, 2-stage Linear					
Id	Name	2-stage Linear Intercept	2-stage Linear Slope	2-stage Linear Stress Cohesion	2-stage Linear Stress Angle
2	Foundation Overburden	900	32	400	38
3	Random Zone – Upstream	3000	22	0	37

12.2 Central Core

The *Central Core* material uses the *2-stage Nonlinear* option for the second stage. To define it, we have to enter two XY series curves in GMS. The UTEXAS manual defines these curves as follows:

"One envelope is the same as the effective stress envelope and corresponds to an anisotropic consolidation stress ratio, $K_c = K_f$; the other envelope is the envelope of τ_{ff} versus $\underline{\sigma}_{fc}$ corresponding to isotropic consolidation ($K_c = 1$). Each of the two nonlinear strength envelopes is defined in terms of points on the envelope, connected by straight lines. … Points on each envelope share common values of effective normal stress. Accordingly, whenever there is a break in either of the two envelopes a point must be defined on both envelopes."

("UTEXAS4, A Computer Program For Slope Stability Calculations", Stephen G. Wright)

This is illustrated by the following figure from the UTEXAS manual.

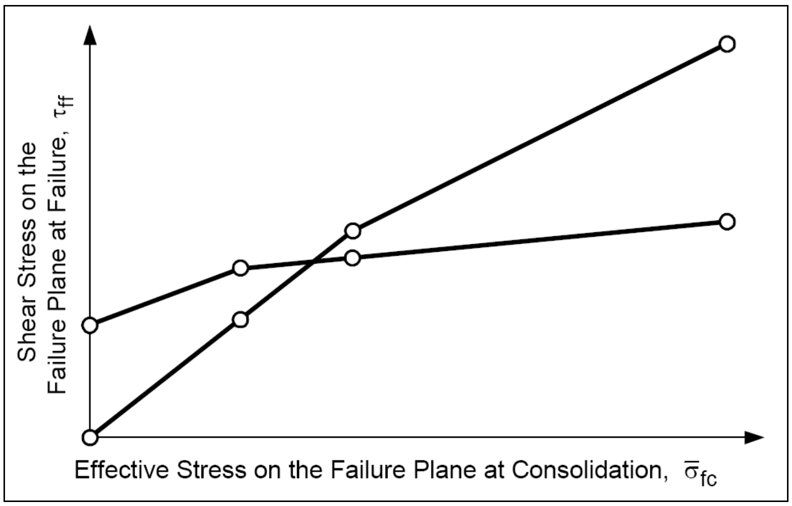


Figure 10. Figure 7.5 from the UTEXAS manual, "UTEXAS4, A Computer Program For Slope Stability Calculations", Stephen G. Wright

2-stage Nonlinear Envelope 1

1. In row 7, in the *2-stage Nonlinear Envelope 1* column, click on the button.
2. In the *XY Series Editor*, enter the values in the table below.

Normal Stress	Shear Stress
-10000.0	0.0
0.0	0.0
1100.0	700.0
1250.0	800.0
3250.0	1350.0
20000.0	5800.0

3. When finished, click *OK* to exit the *XY Series Editor*.

2-stage Nonlinear Envelope 2

1. In row 7, in the *2-stage Nonlinear Envelope 2* column, click on the button.
2. In the *XY Series Editor*, enter the values in the table below.

Normal Stress	Shear Stress
-10000.0	0.0
0.0	0.0
1100.0	1100.0
1250.0	1150.0
3250.0	2200.0
20000.0	13500.0

3. When finished, click *OK* to exit the *XY Series Editor*.

And with that, we're finally done entering in all the material properties.

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

13 Assign Materials to Polygons

Now that the materials are all defined, we can assign the appropriate material to each polygon.

1. In the *Project Explorer*, select the *Profile lines* coverage to make it the active coverage.
2. Select the *Select Polygons* tool.
3. Double-click on the central core of the dam which is zone 7 in Figure 4 below.

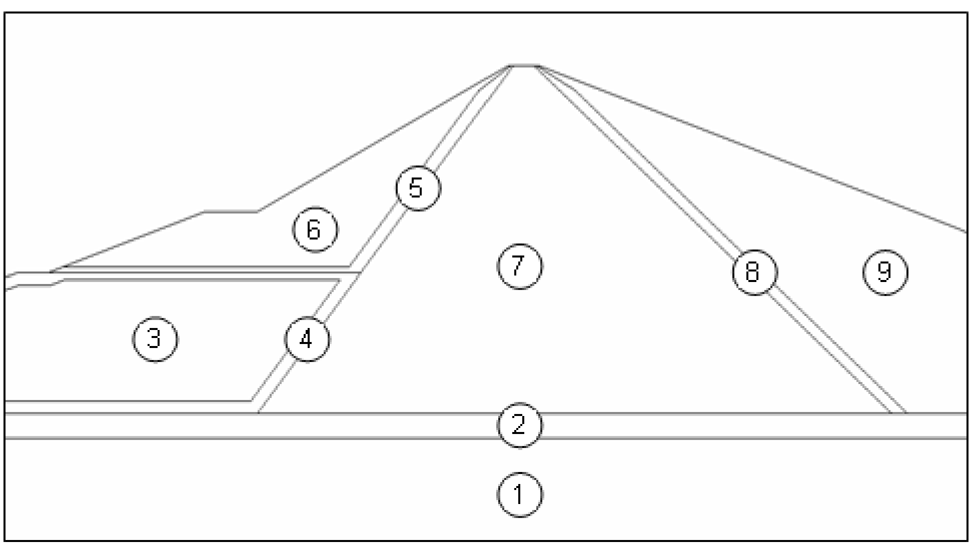


Figure 41. Polygons and their material IDs.

4. In the *Properties* dialog, change the *Material* to **Central Core** and click *OK*.

5. Repeat steps 2 – 3 assigning the appropriate materials to all the polygons. Use the table below as your guide. The numbers in the table correspond to the numbers in the above figure.

	Name
1	Foundation Rock
2	Foundation Overburden
3	Random Zone – Upstream
4	Upstream Filter – Below Drawdown
5	Upstream Filter – Above Drawdown
6	Upstream Shell Above Random Zone
7	Central Core
8	Downstream Filter
9	Downstream Shell

14 UTEXAS Analysis Options

The only thing left to do before we save and run UTEXAS is to set the UTEXAS analysis options. We will assume a circular failure surface and perform an automatic search for the critical factor of safety. We will use Spencer's method and perform a multi-stage, sudden drawdown analysis.

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 *Analysis Options* to match those shown in Figure 12 and 13 below.

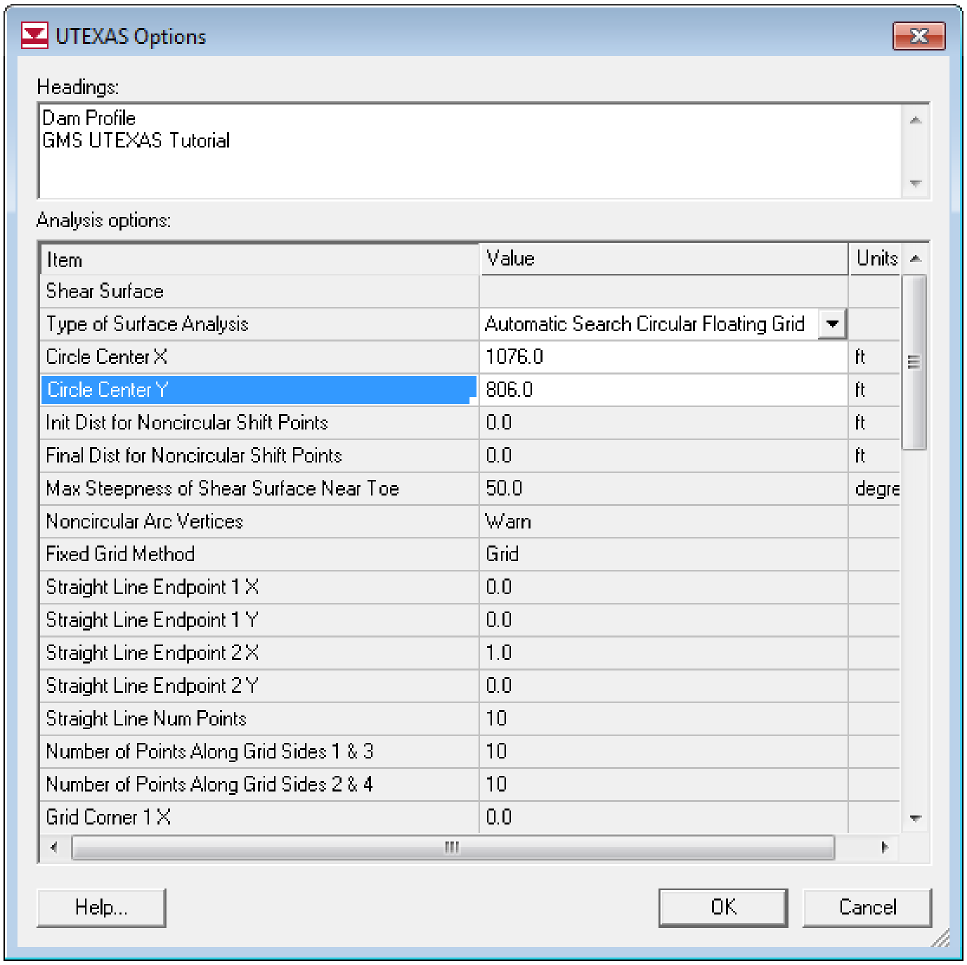


Figure 12. UTEXAS Analysis Options – Part I.

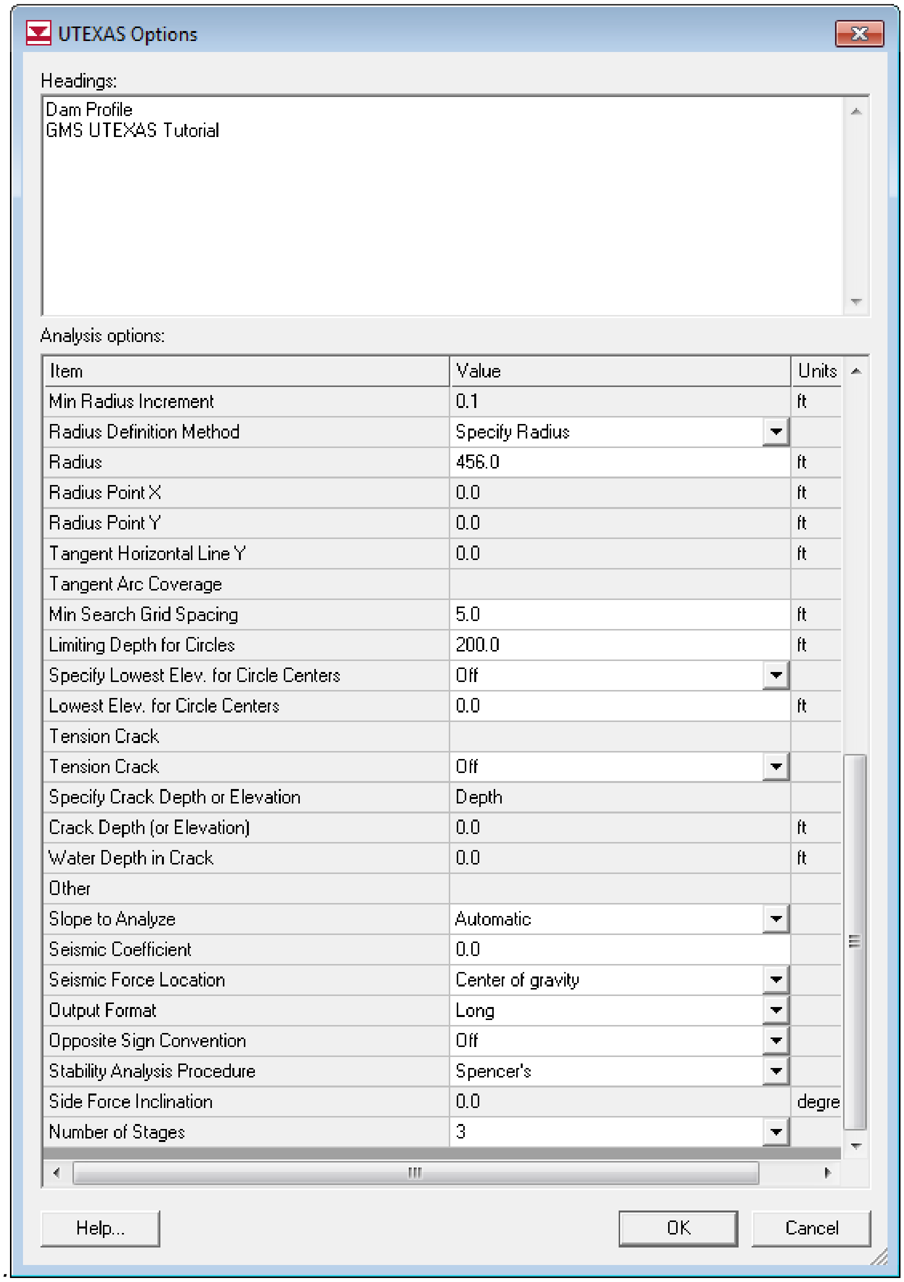


Figure 13. UTEXAS Analysis Options – Part II.

3. When you're finished, click *OK* to exit the dialog.

At this point you should see the starting circle displayed since its radius is no longer 0.0.

15 Export the Model

We're ready to save the model.

1. In the *Project Explorer*, right-click on the **UTEXAS** model and select the *Export* command from the pop-up menu.

2. In the *Export UTEXAS* dialog, locate and open the directory entitled **Tutorials\UTEXAS\dam_profile** (you should already be there).

3. Change the *File name* to **dam_profile** and click *Save*.

16 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 **dam profile.utx** file you just saved (in the **Tutorials\UTEXAS\dam_profile**) folder and open it.

5. When UTEXAS4 finishes, 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.

17 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 **dam_profile.OUT**.

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

18 Conclusion

This concludes the tutorial. You may wish to continue to experiment with the model. For example, you could change stage 1 to use a SEEP2D model solution instead of the *Piezometric line 1* coverage and compare the results.

Here are some of the key concepts in this tutorial:

- GMS can read in the profile model geometry from a CAD file.
- You may have to add a bottom arc to your profile line model so that the bottom material zone is represented in GMS.
- For materials that are defined using the *2-stage Nonlinear* method, the envelope is defined using the *XY Series Editor*.

