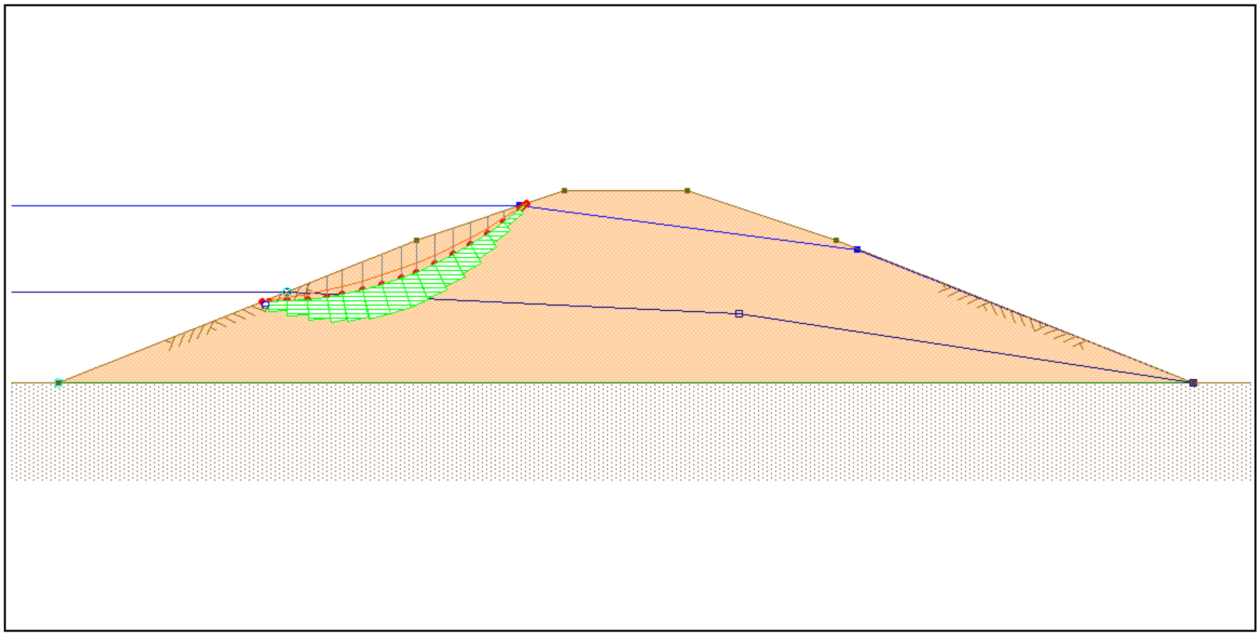
v. 8.0

GMS 8.0 Tutorial

UTEXAS – Rapid Drawdown

Run a multi-stage analysis with UTEXAS simulating rapid drawdown of a reservoir



Objectives

Illustrate how to build a UTEXAS model in GMS that incorporates a multi-stage analysis.

Prerequisite Tutorials

- None

Required Components

- GIS
- Map
- UTEXAS

Time

- 30-60 minutes

1 Contents

1	**Contents**	**2**
2	**Introduction**	**2**
2.1	Outline	2
3	**Program Mode**	**3**
4	**Getting Started**	**3**
5	**Set the Units**	**4**
6	**Save the GMS Project File**	**4**
7	**Create the Embankment**	**4**
7.1	Create the Points	5
7.2	Create the Arcs	5
7.3	Creating the Polygons	6
8	**Create Piezometric Lines and Distributed Loads**	**6**
8.1	Create the Points	6
8.2	Create the Arc	7
8.3	Stage 2	8
9	**Material Properties**	**8**
9.1	Stage 1	8
9.2	Stage 2	9
9.3	Assign Materials to Polygons	9
10	**Analysis Options**	**10**
11	**Save the GMS file**	**12**
12	**Export the Model**	**12**
13	**Run UTEXAS**	**12**
14	**Read the Solution**	**13**
15	**Conclusion**	**13**

2 Introduction

This tutorial illustrates how to build a UTEXAS model in GMS that incorporates a multi-stage analysis. The problem is illustrated on page 1. A dam is being analyzed for its stability when subjected to rapid drawdown. The figure shows two piezometric lines, one before the drawdown and one after. This tutorial is similar to tutorial number six in the UTEXAS tutorial manual (“UTEXPREP4 Preprocessor For UTEXAS4 Slope Stability Software” by Stephen G. Wright, Shinoak Software, Austin Texas, 2003.).

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 beginning this tutorial. You may also wish to consult the UTEXAS user guide for a more detailed explanation of multi-stage slope stability analyses.

2.1 Outline

This is what you will do:

1. Create the model profile.

2. Create piezometric lines defining the pore water pressures.

3. Use the piezometric lines for the load associated with the water.

4. Adjust the analysis options.

5. Save the model, run UTEXAS4 to get a solution, 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 **Rapid Drawdown Model**.

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

4. Select the following coverages:

Profile lines
Piezometric line
Piezometric line –Stage 2

5. Select the *OK* button.

You should see a new conceptual model object appear in the *Project Explorer*. The conceptual model should contain the three coverages that you specified in the previous steps.

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* menu 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. Locate and open the directory entitled **tutfiles\UTEXAS\rapid_drawdown**
3. Enter a name for the project file (ex. "**embank-utexas.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 Embankment

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 Points

The XY locations of the key points of the profile have already been determined. We just need to enter them.

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.
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 show in the table below. If you are viewing this tutorial electronically, you can copy and paste these values into the GMS spreadsheet.

x	y
-100	620
0	620
92.5	657
145	678
187	692
205	698
255	698
315	678
460	620
500	620
500	560
-100	560

6. Verify that the dialog matches the figure above and click *OK*.
7. Now select the *Frame* macro to center the view on the new points.

You should now see the points on the screen.

7.2 Create the Arcs

Now we'll connect the points to form arcs.

1. Select the *Create Arc* 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 2 below as a guide, click on the existing points to create arcs between the points around the perimeter of the model.

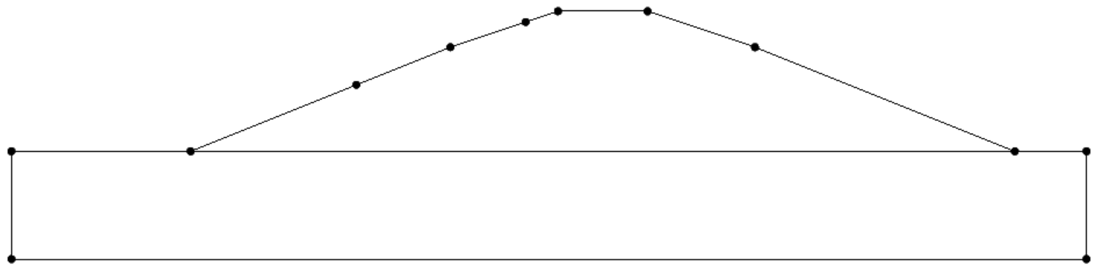
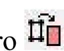


Figure 2. Arcs connecting points.

7.3 Creating 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:

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

These lines define the slope geometry. The lower polygon defines the foundation, while the upper polygon defines the soil comprising the embankment.

8 Create Piezometric Lines and Distributed Loads

In this model we'll use two piezometric lines to define the pore water pressures for both stages. We'll also use the piezometric lines to define the distributed loads.

We'll create each piezometric line in a separate coverage. This is important because, as you'll see later, the material properties refer to the piezometric lines by indicating the coverage they reside in. Thus you cannot have two piezometric lines in a single coverage or GMS won't know which one goes with which material.

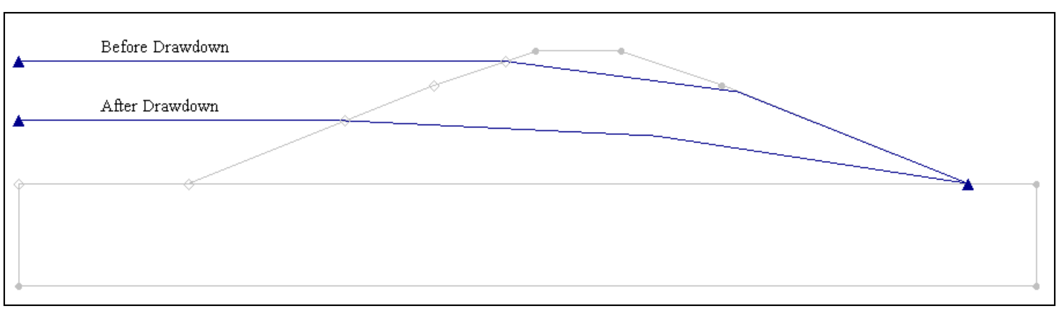


Figure 3. Piezometric lines before and after drawdown.

8.1 Create the Points

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.

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 show in the table below and click *OK*. If you are viewing this tutorial electronically, you can copy and paste these values into the GMS spreadsheet.

x	y
-100	692
187	692
324	674
460	620

8.2 Create the Arc

Now we'll connect the points to form arcs. Then we'll merge the arcs together so that there is just one arc.

1. Turn off the *Profile lines* coverage by unselecting its toggle so that we can more easily see the piezometric line points.

2. Click on the *Piezometric line* coverage to activate it.

3. Using the *Create Arc* tool connect the points to form the "Before Drawdown" arc shown in Figure 3 above.

When we start with a set of four points and connect them to form arcs, we end up with three separate arcs. However, the UTEXAS input requires that we use a single arc to define the piezometric line. To convert the three arcs to a single arc, we 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 or holding down the shift key).

5. Right-click on one of the selected nodes and choose *Node->Vertex*.

Finally, we will turn on the distributed load option for the profile line so that we can use it to define distributed surface loads automatically (without having to explicitly define the loads).

6. Select the *Select Arc* tool .

7. Double-click the newly created arc to bring up its *Properties* dialog.

8. Turn **on** *Dist. Load Stage 1* (you may have to scroll to the right).

9. Click *OK* to exit the dialog.

10. Click anywhere off the arc to unselect it.

8.3 Stage 2

Now we'll follow a similar process to create the piezometric line for stage 2. Repeat the above steps to create the piezometric line for stage 2. Here are the steps in brief:

- In the *Project Explorer,* click on the *Piezometric line stage 2* coverage to activate it.
- Create points at the following XY locations:

x	y
-100	657
92.5	657
276	648
460	620

- Connect the four points to form arcs.
- Convert the middle nodes to vertices so there is only one arc.
- Turn **on** *Dist. Load Stage 2* (not stage 1).

At this point you should have created the two piezometric lines shown in Figure 3 (above) and can turn **on** the *Profile lines* coverage if it was off.

9 Material Properties

The next step is to define the properties associated with the soil material. We have to define separate properties for each stage.

11. Select the Materials macro (or select the *Edit* | *Materials* menu command).
12. Select the *UTEXAS* tab.
13. Click on **material_1** and change the name to "**Bedrock**".
14. Change the C*olor/Pattern* to **Teal**, or some other color you like.
15. Create a new material by entering "**Embankment**" in the *Name* column of the blank row at the bottom of the spreadsheet.
16. Change the C*olor/Pattern* to **Yellow**, or some other color you like.

9.1 Stage 1

To enter the properties for stage 1:

1. Make sure the *UTEXAS* tab is selected.

2. At the bottom of the dialog make sure *Show Stage 1* is **on** and *Show Stage 2* is **off**.

3. Change the material properties for the **Bedrock** (material #1) to the following:

Unit Weight Stage 1	Shear Strength Method Stage 1
160	Very Strong material

4. Change the material properties for the **Embankment** to the following:

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
135	Conventional	0	45	Piezometric Line

5. For *Piezometric Line Coverage Stage 1*, select the  button and select the **Piezometric line** coverage.

9.2 Stage 2

To enter the properties for stage 2:

1. Turn **on** *Show Stage 2* and turn **off** *Show Stage 1*.

2. Fill in the attributes for the **Bedrock** material during stage 2 as follows:

Unit Weight Stage 2	Shear Strength Method Stage 2
160	Very Strong material

3. Fill in the attributes for the **Embankment** during stage 2 as follows:

Unit Weight Stage 2	Shear Strength Method Stage 2	2-stage Linear Intercept	2-stage Linear Slope	2-stage Linear Stress Cohesion	2-stage Linear Stress Angle	Pore Water Pressure Method Stage 2
135	2-stage Linear	64	24	0	45	Piezometric Line

4. For *Piezometric Line Coverage Stage 2*, select the  button and select the **Piezometric line stage 2** coverage.

5. Click *OK* to exit the dialog.

9.3 Assign Materials to Polygons

Now that we have created the materials we will assign the appropriate material to each polygon.

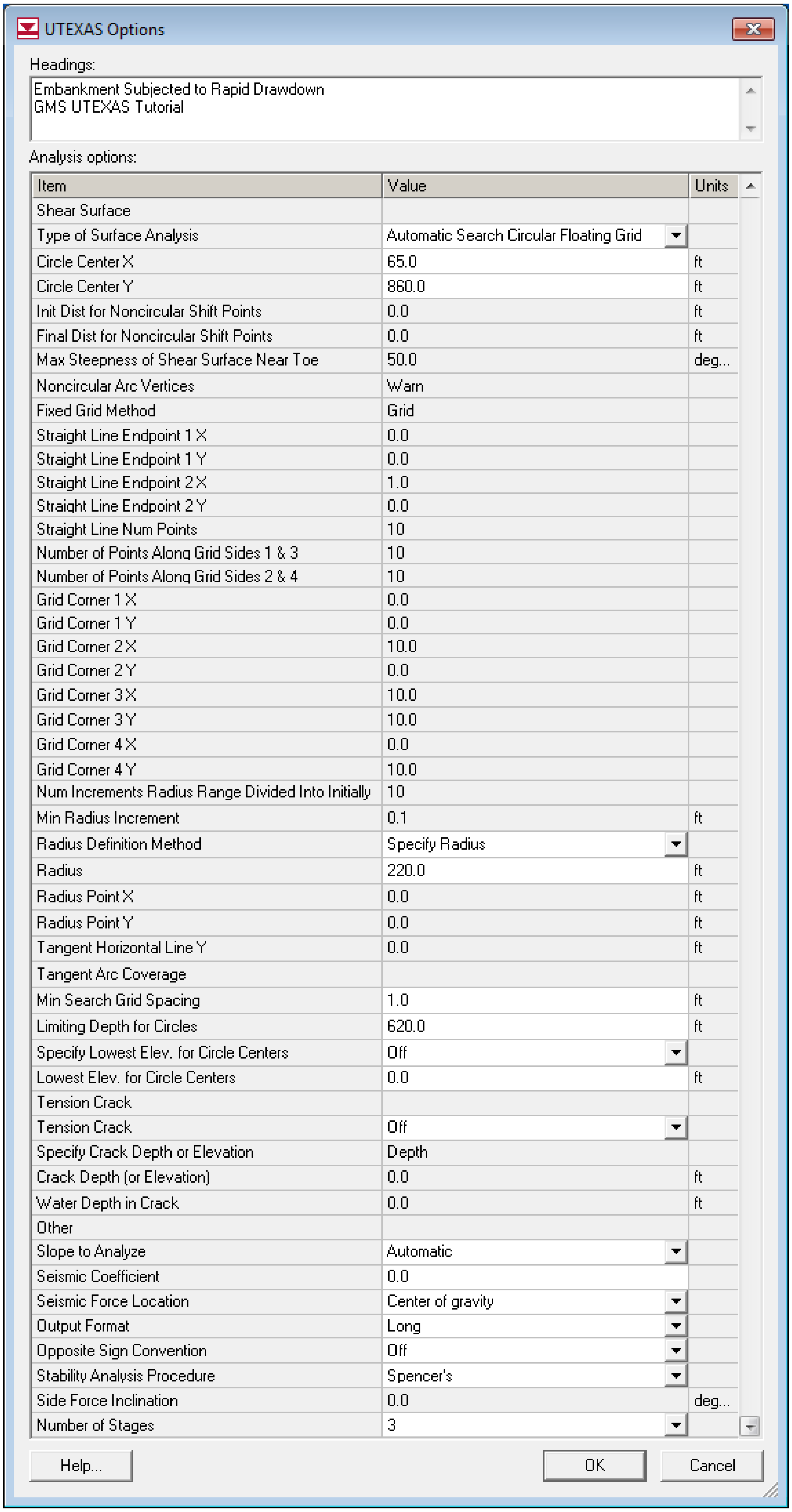
1. Select the *Profile lines* coverage to make it active.

2. Select the *Select Polygons* tool.
3. Double-click on the Embankment polygon (the upper polygon) to bring up its properties.
4. In the *Properties* dialog, change the *Material* to **Embankment** and click *OK*.
5. Double-click the Bedrock polygon (the lower polygon) to bring up its properties.
6. In the *Properties* dialog, make sure the *Material* is set **Bedrock** and click *OK*.

10 Analysis Options

The only thing left to do before we save and run the model is to set the UTEXAS analysis options. We will perform an automated search using a circular failure surface and Spencer's Method using a multi-stage approach. We will define the location of the starting circle by entering the coordinates of the center of the circle and the circle radius.

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 options to match those shown in the dialog below.



Item	Value	Units
Shear Surface		
Type of Surface Analysis	Automatic Search Circular Floating Grid	
Circle Center X	65.0	ft
Circle Center Y	860.0	ft
Init Dist for Noncircular Shift Points	0.0	ft
Final Dist for Noncircular Shift Points	0.0	ft
Max Steepness of Shear Surface Near Toe	50.0	deg...
Noncircular Arc Vertices	Warn	
Fixed Grid Method	Grid	
Straight Line Endpoint 1 X	0.0	
Straight Line Endpoint 1 Y	0.0	
Straight Line Endpoint 2 X	1.0	
Straight Line Endpoint 2 Y	0.0	
Straight Line Num Points	10	
Number of Points Along Grid Sides 1 & 3	10	
Number of Points Along Grid Sides 2 & 4	10	
Grid Corner 1 X	0.0	
Grid Corner 1 Y	0.0	
Grid Corner 2 X	10.0	
Grid Corner 2 Y	0.0	
Grid Corner 3 X	10.0	
Grid Corner 3 Y	10.0	
Grid Corner 4 X	0.0	
Grid Corner 4 Y	10.0	
Num Increments Radius Range Divided Into Initially	10	
Min Radius Increment	0.1	ft
Radius Definition Method	Specify Radius	
Radius	220.0	ft
Radius Point X	0.0	ft
Radius Point Y	0.0	ft
Tangent Horizontal Line Y	0.0	ft
Tangent Arc Coverage		
Min Search Grid Spacing	1.0	ft
Limiting Depth for Circles	620.0	ft
Specify Lowest Elev. for Circle Centers	Off	
Lowest Elev. for Circle Centers	0.0	ft
Tension Crack		
Tension Crack	Off	
Specify Crack Depth or Elevation	Depth	
Crack Depth (or Elevation)	0.0	ft
Water Depth in Crack	0.0	ft
Other		
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	
Side Force Inclination	0.0	deg...
Number of Stages	3	

Figure 5. *UTEXAS Options.*

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

At this point you should see the starting circle displayed.

11 Save the GMS file

We will save the GMS project file before continuing.

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

12 Export the Model

We're ready to export the model for use in UTEXAS.

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 **tutfiles\UTEXAS\rapid_drawdown** (you should already be there).

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

13 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 **Rapid.utx** file you just saved (in the **tutfiles\UTEXAS\rapid_drawdown**) 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.

14 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 **Rapid.OUT**.

You 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:

- GMS can be used to set up a multistage UTEXAS analysis.

- You can only have one piezometric line per coverage.

