GMS 7.0 TUTORIALS

UTEXAS – Rapid Drawdown

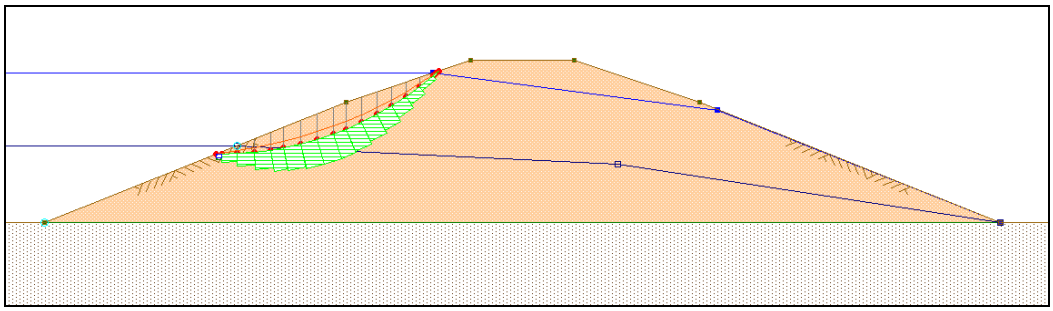

Figure 1. Embankment Subjected to Rapid Drawdown.

1 Introduction

This tutorial illustrates how to build a UTEXAS model in GMS that incorporates a multi-stage analysis. The problem is illustrated in Figure 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. 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.

1.1 Contents

1.2 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.

1.3 Required Modules/Interfaces

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

- Map
- UTEXAS

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

2 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.

3 Set the Units

We will start by setting the units we are using. GMS will display the units we select next to the input fields to remind us what they are.

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.

4 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. 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.

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

5 Create the Embankment

First, we will create the geometry defining the embankment using a UTEXAS conceptual model.

5.1 Create the Conceptual Model

To create the conceptual model:

1. Right-click in the *Project Explorer* and select the *New* | *Conceptual Model* menu command.
2. In the *Conceptual Model Properties* dialog, change the *Name* to **Embankment**.
3. Change the *Type* to **SEEP2D/UTEXAS**.
4. Turn **off** the *SEEP2D* option.
5. Select the *OK* button to exit the dialog.

5.2 Create a New Coverage

The embankment profile will reside in its own coverage. We'll create that coverage now.

1. In the *Project Explorer,* right-click on the *Embankment* conceptual model you just created and select the *New Coverage* command from the pop-up menu.
2. In the *Coverage Setup* dialog, change the *Coverage Name* to **Profile** and click *OK* (we don't need to turn on any properties for the coverage).

5.3 Create the Points

Now we'll create the embankment profile. The XY locations of the key points of the profile have already been determined. We just need to enter them.

1. In the *Project Explorer*, right-click on the **Profile** coverage and select the *Attribute Table* command from the pop-up menu.
2. In the dialog, change the *Feature type* to **Points**.
3. Make sure the *Show point coordinates* option is turned on.
4. 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

5. Verify that the dialog looks like the figure below and click *OK*.

6. Now select the *Frame* macro .

You should now see the points on the screen.

5.4 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.

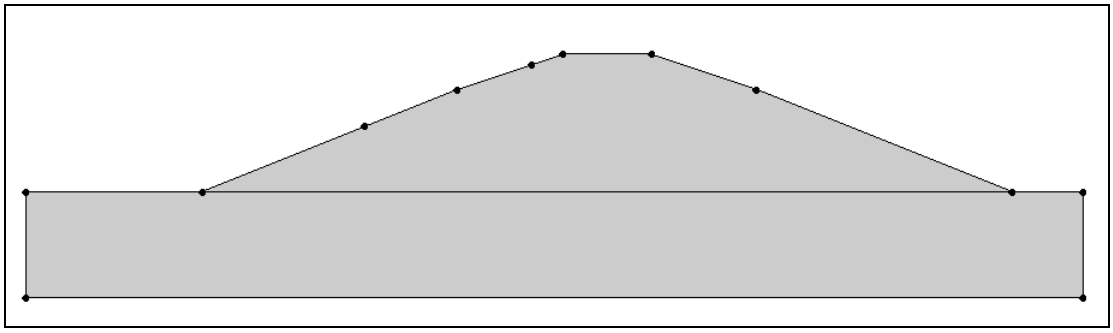

Figure 2. Arcs Connecting Points.

4. Select the *Feature Objects* | *Build Polygons* menu command.

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

6 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.

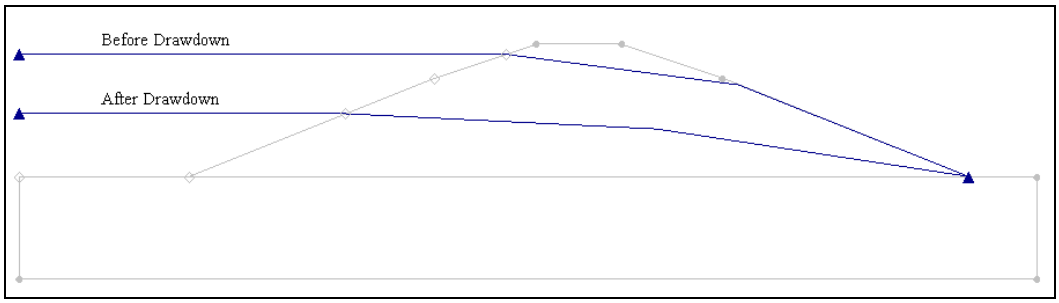

Figure 3. Piezometric Lines Before and After Drawdown.

6.1 Create a New Coverage

1. In the *Project Explorer,* right-click on the *Embankment* conceptual model and select the *New Coverage* command from the pop-up menu.

2. Change the options in the *Coverage Setup* dialog to be as shown in the following figure.

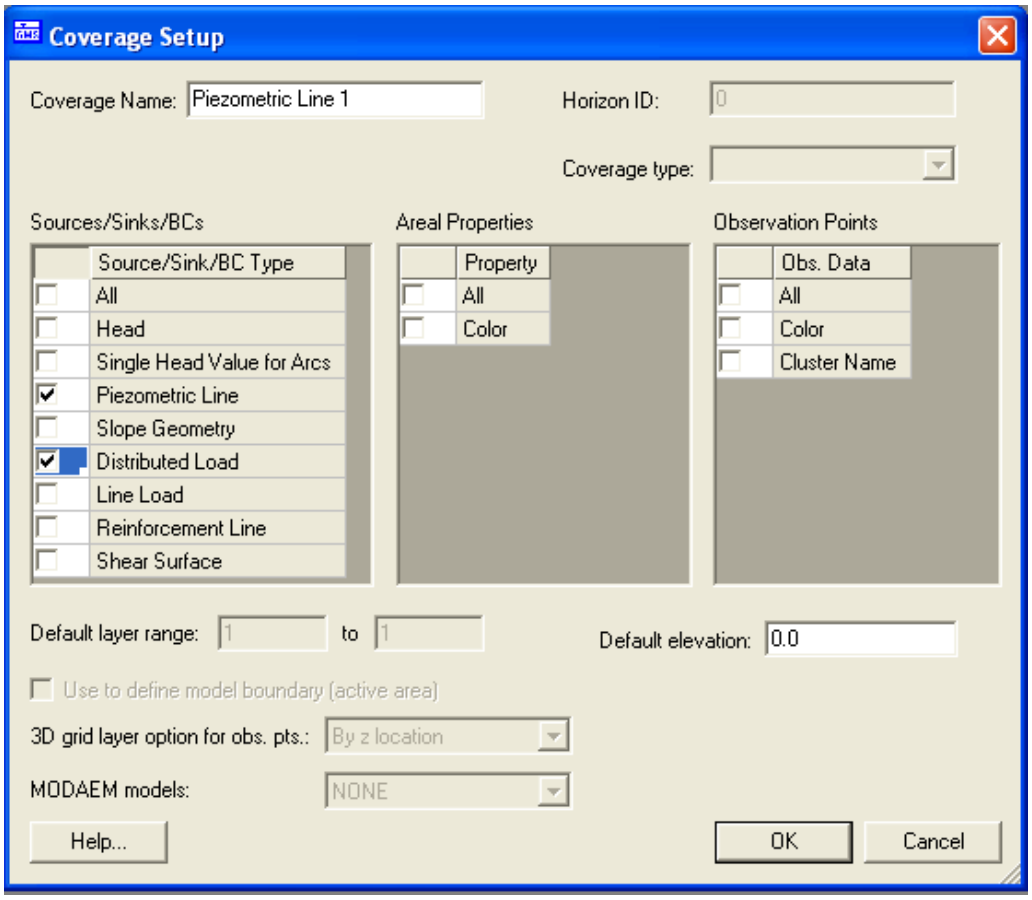

Figure 4. Coverage Setup dialog.

3. When done, click *OK* to exit the dialog.

6.2 Create the Points

1. In the *Project Explorer*, right-click on the **Piezometric Line1** coverage and select the *Attribute Table* command from the pop-up menu.

2. In the dialog, change the *Feature type* to **Points**.

3. Make sure the *Show point coordinates* option is turned on.

4. 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

6.3 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. Using the *Create Arc* tool connect the points to form the "Before Drawdown" arc shown in Figure 3 above. If you can't see all four nodes, turn off the *Profile* coverage in the *Project Explorer*.

2. Using the *Select Node* tool, select the two middle nodes.

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

4. Select the *Select Arc* tool.

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

6. Change the *Type* to **piezometric line**.

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

8. Click *OK* to exit the dialog.

9. Click anywhere off the arc to unselect it.

Note that GMS has changed the arc color and node symbols to indicate the arc type.

6.4 Stage 2

Now we'll do the same thing to create the piezometric line for stage 2.

1. Repeat the above steps to create the piezometric line for stage 2. Here are the steps in brief:

   - Create a new coverage named **Piezometric Line 2** with *Piezometric Line* and *Distributed Load* properties enabled.
   - Create points a the following XY locations:

X	y
-100	657
92.5	657
276	648
460	620

   - Connect the points to form arcs.
   - Convert the middle nodes to vertices so there is only one arc.
   - Set the arc *Type* to **Piezometric line** and 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* coverage if it was off.

7 Material Properties

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

1. Select the *Edit* | *Materials* menu command.
2. Click on **material_1** and change the name to "**Bedrock**".
3. Change the C*olor/Pattern* to **Teal**, or some other color you like.
4. Create a new material by entering "**Embankment**" in the *Name* column of the blank row at the bottom of the spreadsheet.
5. Change the C*olor/Pattern* to **Yellow**, or some other color you like.

7.1 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 1** coverage.

7.2 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 2** coverage.

5. Click *OK* to exit the dialog.

7.3 Assign Materials to Polygons

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

1. In the *Project Explorer,* select the *Profile* coverage.

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*.

8 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.

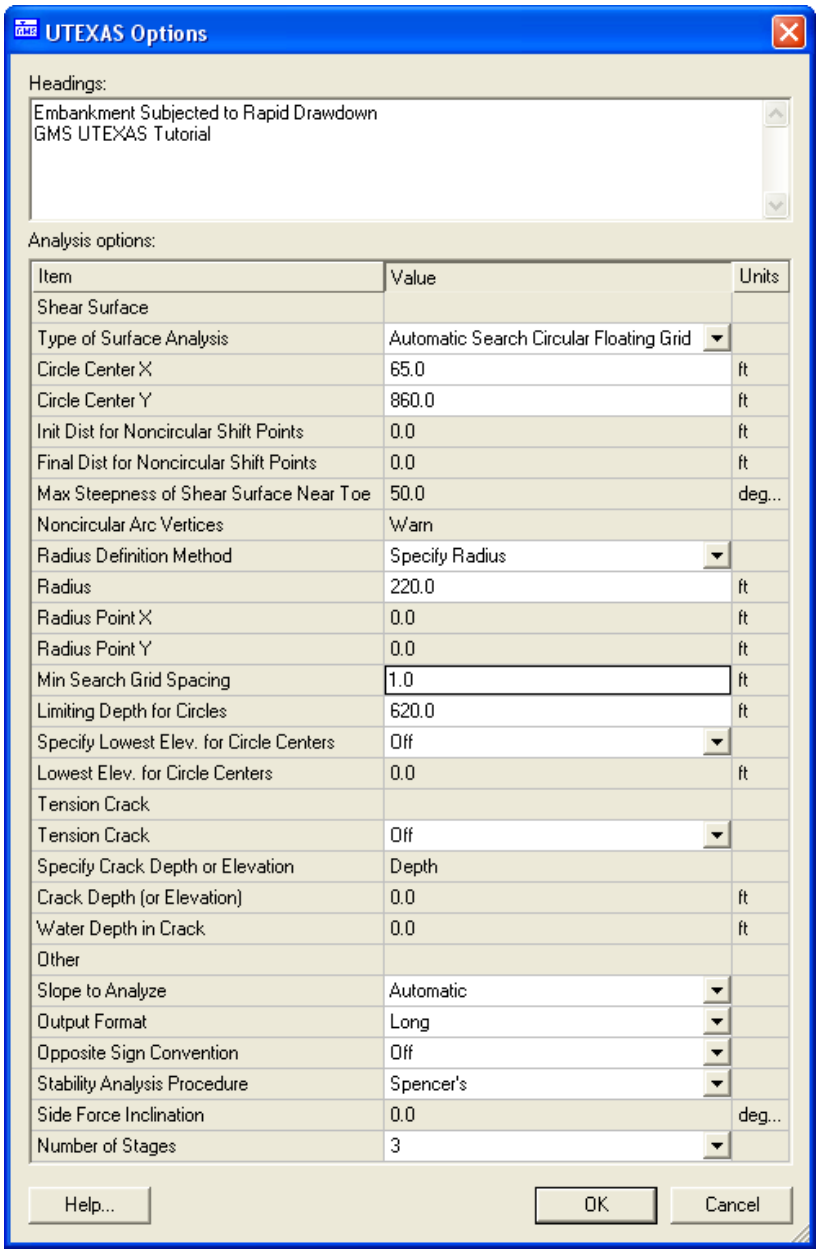

Figure 5. UTEXAS Options

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

At this point you should see the starting circle displayed.

9 Save the GMS file

We will save the GMS project file before continuing.

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

10 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*.

11 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. When UTEXAS4 finishes, look at the things mentioned in the *Errors, Warnings* window, then close the window.

12 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.

13 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.