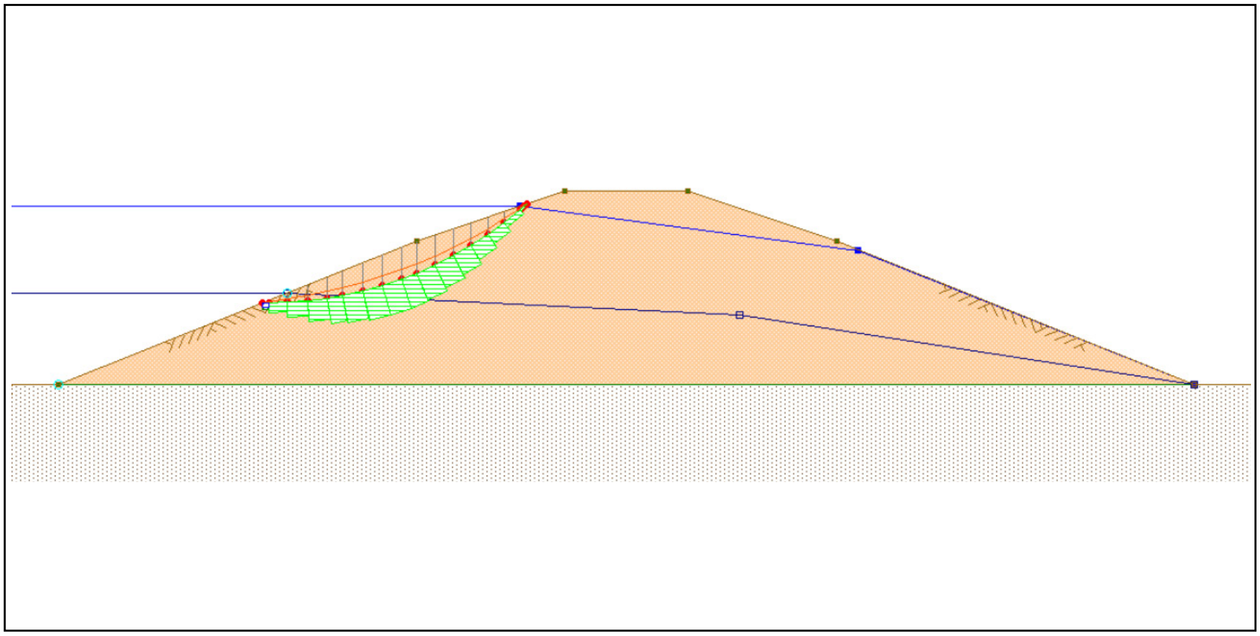
v. 10.1

GMS 10.1 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

- UTEXAS – Natural Slope

Required Components

- GIS Module
- Map Module
- UTEXAS

Time

- 15–30 minutes

1 Introduction

This tutorial illustrates how to build a UTEXAS model in GMS that incorporates a multi-stage analysis. 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.[1]

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

1.1 Outline

Here are the steps to this tutorial:

1. Create the model profile.
2. Create piezometric lines defining the pore water pressures.

---

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

3. Use the piezometric lines for the load associated with the water.
4. Adjust the analysis options.
5. Save the model.
6. Run UTEXAS4 to get a solution.
7. 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. Then 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 "Rapid Drawdown Model."
4. Turn off the *SEEP2D* option in the *Numerical models* section.
5. Select the following coverages:
   - *Profile lines*
   - *Piezometric line*
   - *Piezometric line – Stage 2*
6. Select the **OK** button.

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

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, select "ft" for the *Length* units.
3. If necessary, select "lb" for the *Force* units.
4. Select the **OK** button.

5 Save the GMS Project File

Before continuing, save the project as a GMS project file:

1. Select the *File* | **Save As** command.
2. Locate and open the directory entitled *Tutorials\UTEXAS\rapid_drawdown*.
3. Enter a name for the project file (e.g., "embank-utexas.gpr").
4. Select the **Save** button.

Click on the **Save** macro frequently to save all changes.

6 Create the Embankment

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

6.1 Create the Points

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

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 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 show 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
-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

7. Click **OK**.
8. Now select the **Frame** macro to center the view on the new points.

The user should now see the points on the screen.

6.2 Create the Arcs

Now the user will connect the points to form arcs.

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

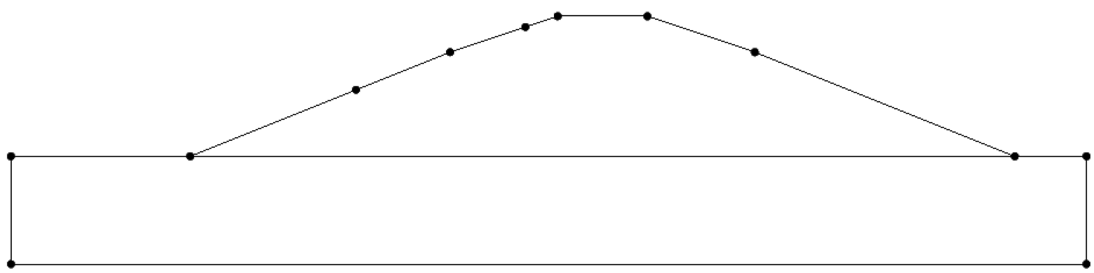


Figure 1 Arcs connecting points

6.3 Creating the Polygons

Now that the arcs are created, the user can use the arcs to build polygons representing the regions enclosed by the arcs. Later in this tutorial, the user 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.

7 Create Piezometric Lines and Distributed Loads

In this model, the user will use two piezometric lines to define the pore water pressures for both stages. The user will also use the piezometric lines to define the distributed loads.

The user will create each piezometric line in a separate coverage. This is important because, as the user will see later, the material properties refer to the piezometric lines by indicating the coverage they reside in. Thus, it is not possible to have two piezometric lines in a single coverage because GMS won't know which one goes with which material.

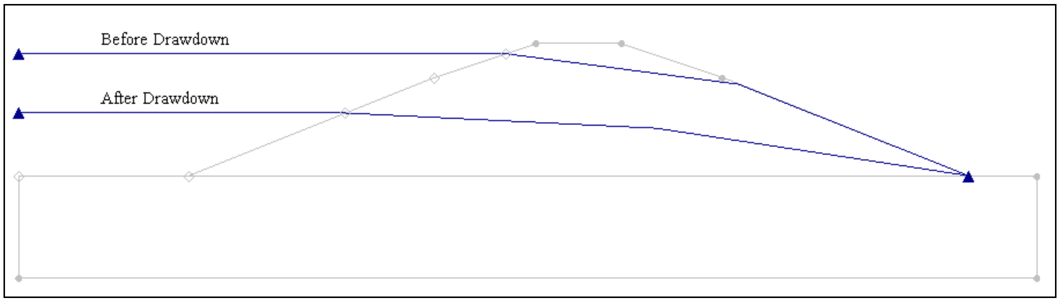


Figure 2 Piezometric lines before and after drawdown

7.1 Create the Points

1. Click on the "Piezometric line" coverage to make it active.
2. Right-click on the "Piezometric line" coverage.
3. Select the **Attribute Table** command from the pop-up menu to open the *Attribute Table* dialog.
4. In the dialog, change the *Feature type* to "Points."
5. Make sure the *Show coordinates* option is turned on.
6. Enter the X and Y coordinates show 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
-100	692
187	692
324	674
460	620

7. Click **OK**.

7.2 Create the Arc

Now the user will connect the points to form arcs. Then the user will merge the arcs together so that there is just one arc.

1. Turn off the "Profile lines" coverage by unselecting its toggle so that it is easier to 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 2 above.

When the user starts with a set of four points and connects them to form arcs, the user will end up with three separate arcs. However, the UTEXAS input requires that the user only uses a single arc to define the piezometric line. To convert the three arcs to a single arc, the user 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, the user will turn on the distributed load option for the profile line so that the user 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 *Attribute Table* dialog.

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

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

10. Click anywhere away from the arc to unselect it.

7.3 Stage 2

Now the user will 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:

1. In the Project Explorer, click on the "Piezometric line stage 2" coverage to activate it.

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

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

4. Create points at the following XY locations:

X	Y
-100	657
92.5	657
276	648
460	620

5. Connect the four points to form arcs.

6. Convert the two middle nodes to vertices so there is only one arc.

7. Turn on *Dist. Load Stage 2* (not stage 1).

At this point, the user should have created the two piezometric lines shown in Figure 2 (above) and can turn on the "Profile lines" coverage if it was off.

8 Material Properties

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

1. Select the **Materials** macro (or select the *Edit* | **Materials** menu command) to open the *Materials* dialog.
2. Select the *UTEXAS* tab.
3. Click on "material_1" and change the name to "Bedrock."
4. Change the C*olor/Pattern* to Teal.
5. Create a new material by entering "Embankment" in the *Name* column of the blank row at the bottom of the spreadsheet.
6. Change the C*olor/Pattern* to Yellow.

8.1 Stage 1

Do the following 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. (The user may have to expand the item in the dialog.)

8.2 Stage 2

Do the following 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. (The user may have to expand the item in the dialog.)

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

8.3 Assign Materials to Polygons

Now that the user has created the materials, the user 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).

4. In the *Attribute Table* dialog, change the *Material* to "Embankment."

5. Click **OK**.

6. Double-click the Bedrock polygon (the lower polygon).

7. In the *Attribute Table* dialog, make sure the *Material* is set "Bedrock."

8. Click **OK**.

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 perform an automated search using a circular failure surface and Spencer's Method using a multi-stage approach. The user 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.
2. Select the **Analysis Options** command from the pop-up menu.
3. Add the following in the *Headings* box:
   - "Embankment Subjected to Rapid Drawdown"
   - "GMS UTEXAS Tutorial"
4. Change *Circle Center X* to "65.0."
5. Change *Circle Center Y* to "860.0."
6. Change the *Radius* to "220.0."
7. Change the *Limiting Depth for Circles* to "620.0."
8. Change the *Number of Stages* to "3."
9. The options should match those shown in the dialog below.

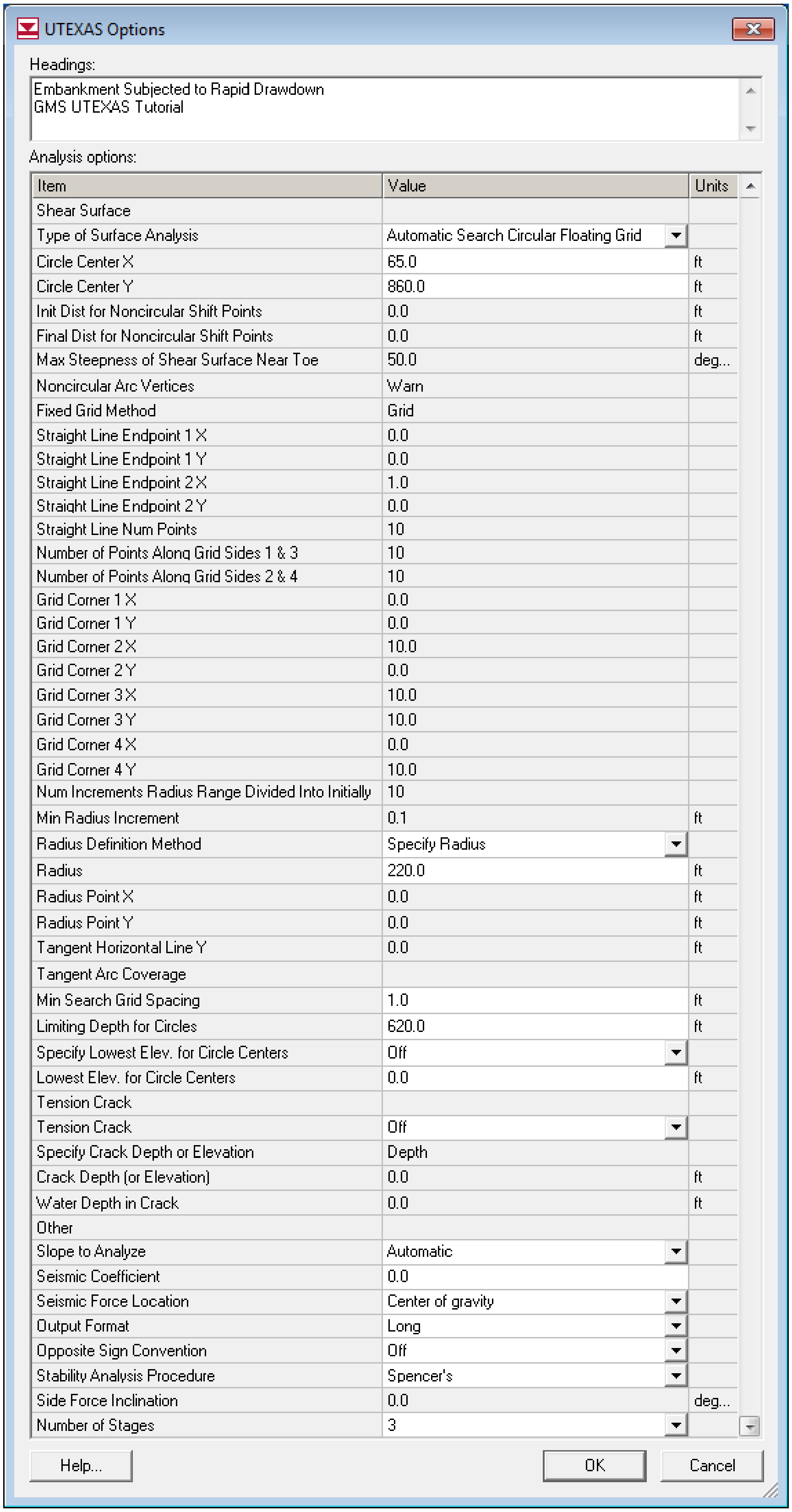


Figure 3 UTEXAS Options

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

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

10 Save the GMS file

Save the GMS project file before continuing.

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

11 Export the Model

Now it is possible to export the model for use in 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\rapid_drawdown*.
4. Change the *File name* to "Rapid."
5. Click **Save**.

12 Run UTEXAS

Now that the user has saved the UTEXAS input file, it's 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 "Rapid.utx" file that was just saved (in the *Tutorials\UTEXAS\rapid_drawdown* folder).
6. Click **Open**.
7. Press **Save** in the *Open file for graphics output* dialog box. This will save a TexGraf4 output file.

8. Look at the things mentioned in the *Errors, Warnings* window, then close the window.

9. Close the UTEXAS window as well.

13 Read the Solution

Now the user needs 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 "Rapid.OUT."

4. Click **Open**.

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

14 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.

- The user can only have one piezometric line per coverage.

