

SOILVISION 10 Help Manual - 12/16/2019

Model Setup



The following steps will be required in order to set up the model described in the preceding section. The steps fall under the following general categories:

- a. Create model
- b. Specify analysis settings
- c. Enter geometry
- d. Apply material properties
- e. Specify search method geometry
- f. Specify Pore-Water
- g. Analyze model
- h. Results

The details of these outlined steps are given in the following sections.

NOTE:

Any values on the dialogs that are not specifically mentioned in the steps below are assumed to be the default values currently present.

a. Create Model

The following steps are required to create the model:

1. Open the SOILVISION **Manager**  dialog,
2. In **LEARNING MODE**, select the SVSLOPE module icon  and click New Model. The model is automatically stored in **MyProject** project.
3. Select the following:
 - Module: **SVSLOPE**
 - System: **2D**
 - Units: **Imperial**
 - Slip Direction: **Right to Left**
 - Model Name: **DRAWDOWN**
4. Click on **OK**.

b. Specify Analysis Settings (Model > Settings)

In SVSLOPE the *Settings* dialog is used to specify the method for determining the critical slip surface and the details of the applicable search techniques to be used in the analysis. For this model the settings will be entered as follows:

1. Select **Model > Settings**  from the menu,
2. Select the **Slip Surface** tab,
 - Slip Direction: **Right to Left**
 - Slip Shape: **Circular**
 - Search Method: **Grid and Tangent**
3. Select the **Calculation Methods** tab from the dialog and select the method types as shown below:
 - GLE (Fredlund)**
4. Check the **Apply Rapid Drawdown Analysis** check box,
5. Select the **Advanced** tab from the dialog and ensure that only the boxes for Use Steffensen's Iteration Method, and Set shear strength as zero when base in tension boxes are selected. Ensure Check Trial Slip Surfaces having $m\text{-}\alpha < 0.2$ is not selected.
6. Press **OK** to close the dialog.

∴ Enter Geometry (Geometry)

Model geometry is defined as a set of **regions**. Geometry can be either drawn by the user or defined as a set of coordinates. Model Geometry can also be imported from sources such as **.DXF files** or from **existing models**. In this example the geometry will be created by cutting and pasting the geometry into the model.

• Cut and Paste

The shapes that define each region will now be created. Refer to the data in the tables below for the geometry points for the four regions.

Define Region R1

1. Open the Regions dialog by selecting **Geometry > Regions**  ... from the menu,
2. Click the **New** button **3 times** to create the second, third and fourth regions,
3. Select the region **R1** and click the **Properties...** button to open the **Region Properties** dialog,
4. Click the **New Polygon...** button to open the **New Region Polygon** dialog,
5. Copy the region coordinate data for R1 provided below and click the **Paste** button on the **New Region Polygon** dialog to paste the region data into the data grid,
6. Click **OK** to close the dialog and create the new region,
7. Click the **right arrow** at the top right of the Region Properties dialog to move to the second region **R2**,
8. **Repeat** the Steps 4 - 7 to complete regions **R2**, **R3**, and **R4**,
9. Click **OK** on the **Region Properties** dialog and on the **Regions** dialog to accept the region changes.

Region: R1

X (ft)	Y (ft)
1005	550
870	380

550	380
700	450
770	450

Region: R2

X (ft)	Y (ft)
870	380
770	250
0	250
265	320
490	380
550	380

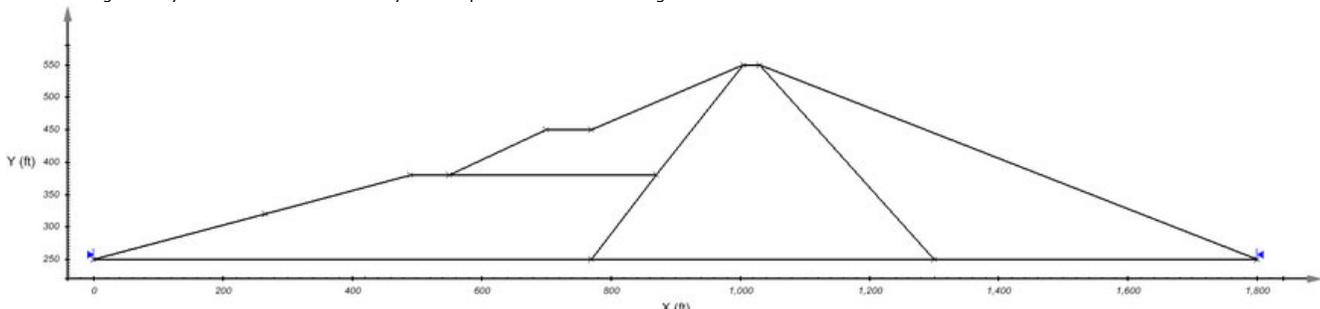
Region: R3

X (ft)	Y (ft)
1005	550
1030	550
1300	250
770	250
870	380

Region: R4

X (ft)	Y (ft)
1030	550
1800	250
1300	250

If all model geometry has been entered correctly the shape will look like the diagram below.



d. Apply Material Properties (Materials)

The next step in defining the model is to enter the [material properties](#) for the two materials that will be used in the model. It is assumed that the user has measured the [Shear Strength](#) parameters for the two materials. These can be found in the table below.

Tabs	Parameters	Material	
		Silty Clay Core	Compacted Rockfill
New Material	Strength Type	Mohr Coulomb	Mohr Coulomb
Shear Strength	Cohesion (psf)	0	0
	Friction Angle, phi (deg)	36	37
	Apply Rapid Drawdown	checked	Unchecked
	Cohesion, cT (psf)	2000	
	Friction Angle, phiT (deg)	18	
	Unit Weight (lb/ft ³)	140	142

1. Open the *Materials Manager* dialog by selecting *Materials > Manager* from the menu,
2. Click the **New** button to create a material,
3. Enter **Silty Clay Core** for the material name in the dialog that appears
4. Choose **Mohr Coulomb** for the *Method* of this material,
5. Press **OK** to close the dialog. The *Material Properties* dialog will open automatically,

NOTE:

When a new material is created, you can specify the display color of the material using the Fill Color box on the Material Properties menu. Any region that has a material assigned to it will display that material's fill color.

6. Move to the *Shear Strength* tab,
7. Enter the *Shear Strength* parameters found in the table above,
8. Check the **Apply Rapid Drawdown** checkbox,
9. Enter the *Cohesion, cT* and the *Friction Angle, phiT* value found in the table
10. Press **OK** button to close the *Material Properties* dialog,
11. **Repeat** steps 2 - 10 to create the **compacted Rockfill** material using the information provided in the table below.

Once all material properties have been entered, we must apply the materials to the corresponding regions.

1. Open the *Regions* dialog by selecting *Geometry > Regions* ... from the menu,
2. Assign the **compacted Rockfill** material to region **R1** using the drop down,
3. Assign the **Silty Clay Core** material to region **R2** using the drop down,
4. Assign the **Silty Clay Core** material to region **R3** using the drop down,
5. Assign the **compacted Rockfill** material to region **R4** using the drop down,
6. Press the OK button to accept the changes and close the dialog.

e. Specify Search Method Geometry (Slips)

The *Grid and Tangent* method of searching for the critical slip surface has already been selected in Step b. Now the user must specify the exact geometry of these objects. This is accomplished through the following steps:

GRID

1. Select *Slips > Grid and Tangent* ...,
2. Select the *Grid* tab,
3. Enter the values for the grid as specified in the **table below**,
4. Move to entering the tangent values.

TANGENT

1. Select the *Tangent* tab,
2. Enter the values for the tangent as specified in the **table below**,
3. Click *OK* to close the dialog.

Grid

X (ft)	Y (ft)
500	1000
500	725
900	725

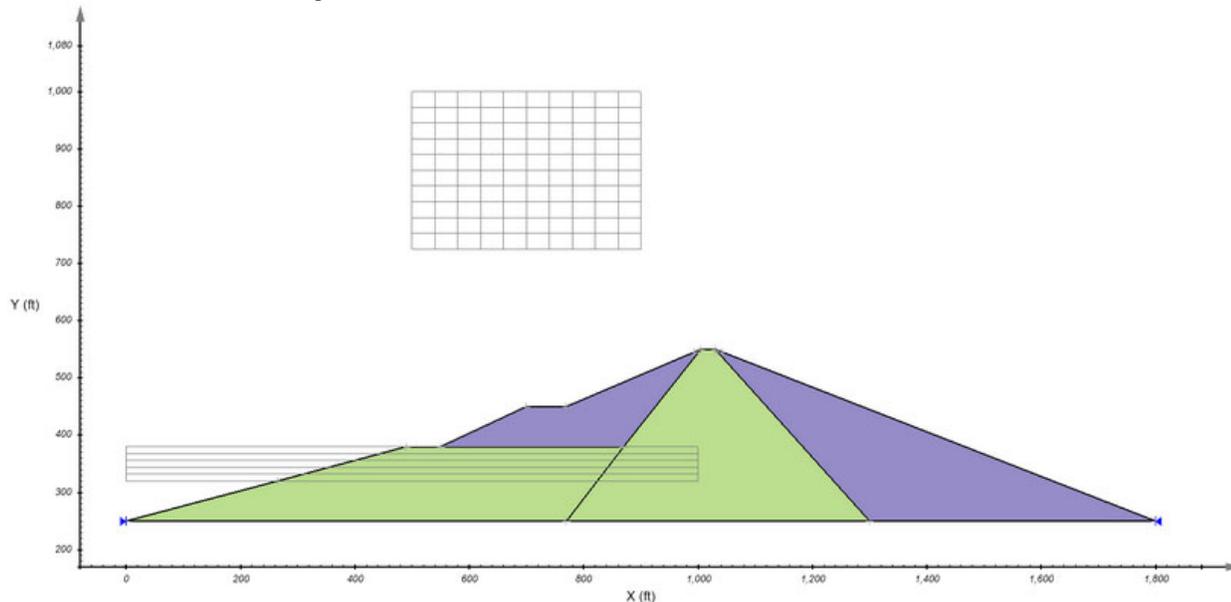
X increments	10
Y increments	10

Tangent

X (ft)	Y (ft)
0	380
0	320
1000	320
1000	380

Radius Increments	5
-------------------	---

Your screen will now look like the image below.



f. Specify Pore-Water (Pore-Water > Settings)

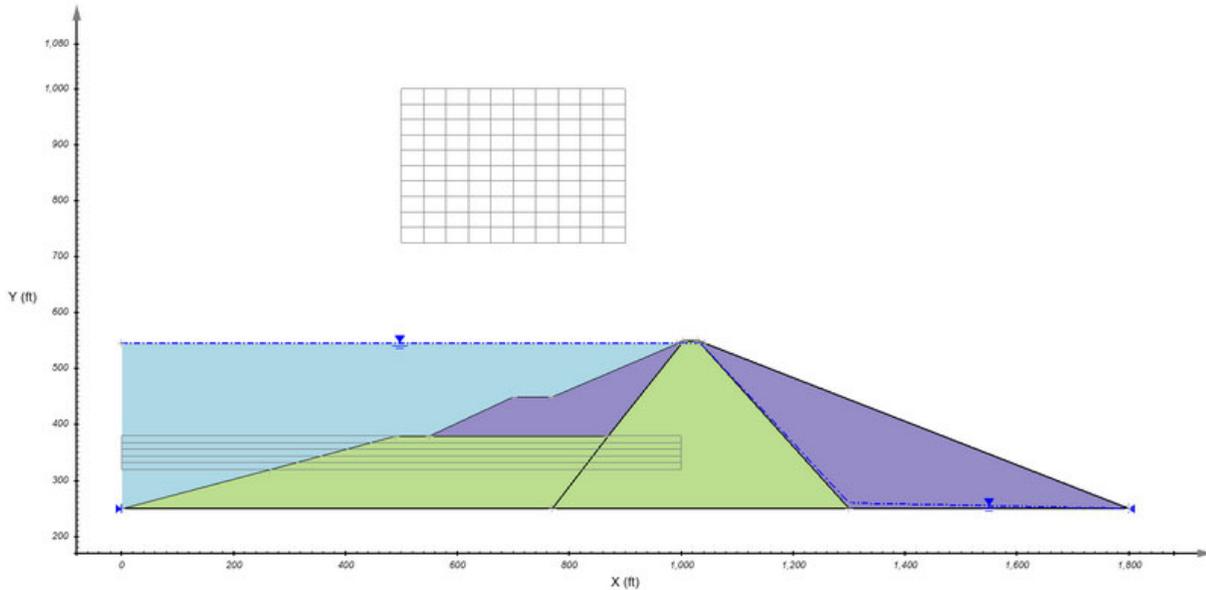
The user must then proceed to enter the initial water table coordinates:

1. Select *Pore-Water > Water Table* ...,
2. Under the *Points* tab enter the *X* and *Y* coordinates as provided for the initial water table in the **table below**,
3. In the *Apply To Regions* Section, confirm that boxes for all regions are checked,
4. Press *OK* to close the dialog,

Initial Water Table:

X (ft)	Y (ft)
0	545
1035	545
1300	260
1800	250

Your screen will now look like the image below.



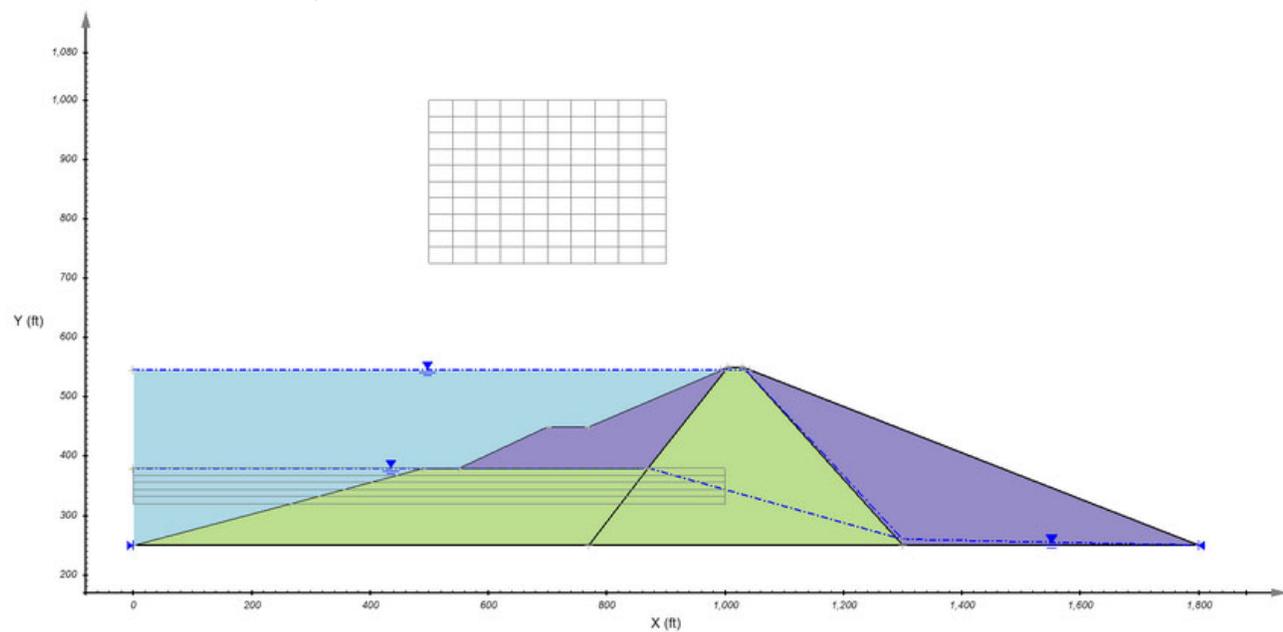
The user must then proceed to enter the final water table coordinates:

5. Select *Pore Water > Final Water Table* ...,
6. Under the *Points* tab enter the *X* and *Y* coordinates as provided for the final water table in the **table below**,
7. In the *Apply to Regions* section, confirm that boxes for all regions are checked,
8. Press *OK* to close the dialog.

Final Water Table:

X (ft)	Y (ft)
0	380
870	380
1300	260
1805	250

Your screen will now look like the image below.



g. Analyze model (Solve > Analyze)

The next step is to analyze the model.

1. Select *Solve > Analyze*  from the menu. The **SVSLOPE Solver** dialog will pop-up and automatically solve.
2. Select the **Results** button to view results

h. Results (Solve > Results)

The visual results for the current model may be examined by selecting the *Solve > Results* menu option or click on Results icon .

The model results will be displayed. To view the results in more detail proceed to [Results and Discussion](#).

NOTE :

To transfer from viewing results to the SVSLOPE design module click on the SVSLOPE icon  found on the left vertical tool bar.

SOILVISION 10 Help Manual - 12/16/2019

Results and Discussion



After the model has completed solving the user may view the results by pressing the Results icon on the process toolbar. The sliding mass is displayed in the CAD for the selected calculation method. To switch between the results of the different calculation methods, click on the drop down menu at the top of the screen and select the method you would like to view. The analysis results in a factor of safety of 1.501 for *GLE (Fredlund)* method. The critical slip surface for *GLE (Fredlund)* method is shown in the following screenshot.

