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 general categories of:

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

The details of these outlined steps are detailed 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: **EBANKOD**
4. Click on *OK* to accept changes.

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: **Entry and Exit**
3. Select the *Calculation Methods* tab from the dialog and select the method types as shown below:
   Morgenstern-Price - Half Sine
4. Select the *Convergence* tab from the dialog and change the values for the Convergence Options as noted below:
   Number of Slices: **50**
   Check the box for Minimum Slide Surface Depth and enter the value: **10 ft**
5. Press ***OK*** to close the dialog.

c. Enter Geometry (Geometry)

Model geometry is defined as a set of regions. This model will be divided into seven regions, which are named R1 through R7. Each region will have one of the materials specified as its material properties. The shapes that define each material region will now be created. The user may enter geometry by i) drawing on the CAD ii) cut and paste data or they may iii) import geometry from existing model. Each option is presented below.

- **CAD Drawing**

Please refer to the 2D Basic Slope example, section C for methodology.

- **Cut and Paste**

Alternatively, the regions can be created by cutting and pasting data from the tables below. Follow these steps:

1. Open the *Regions* dialog by selecting *Geometry > Regions*... from the menu,
2. Click the *New* button in the *Regions* dialog 6 times to create regions **R2 thru R7**
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 *Region: 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 to 7** to create regions **R2 through R7** using the data below,
9. Click *OK* on the *Region Properties* dialog and on the *Regions* dialog to accept the changes.

Region: R1

x (ft)	y (ft)
467	203
475	203
500	203
503	203
515	202
527	203
552	202
592	198
520	175
422	147
333	119
466	193

Region: R2

x (ft)	y (ft)
120	87
139	90
184	97
208	100
230	102
279	108
322	113
333	119
422	147
322	99
230	86
211.6	86

Region: R3

x (ft)	y (ft)
0	40
158	53
230	61
278	73
322	88
373	99
322	99
230	86
211.6	86
120	87
50	78
0	76

Region: R4

x (ft)	y (ft)
0	20
158	39
230	51
322	80
469	95
574	162
700	194
700	205
574	175
469	119
373	99
322	88
278	73
230	61
158	53
0	40

Region: R5

x (ft)	y (ft)
372	155.3
457	201
467	203
466	193
333	119
322	113
279	108

Region: R6

x (ft)	y (ft)
322	99
422	147
520	175
592	198
700	220
700	205
574	175
469	119
373	99

Region: R7

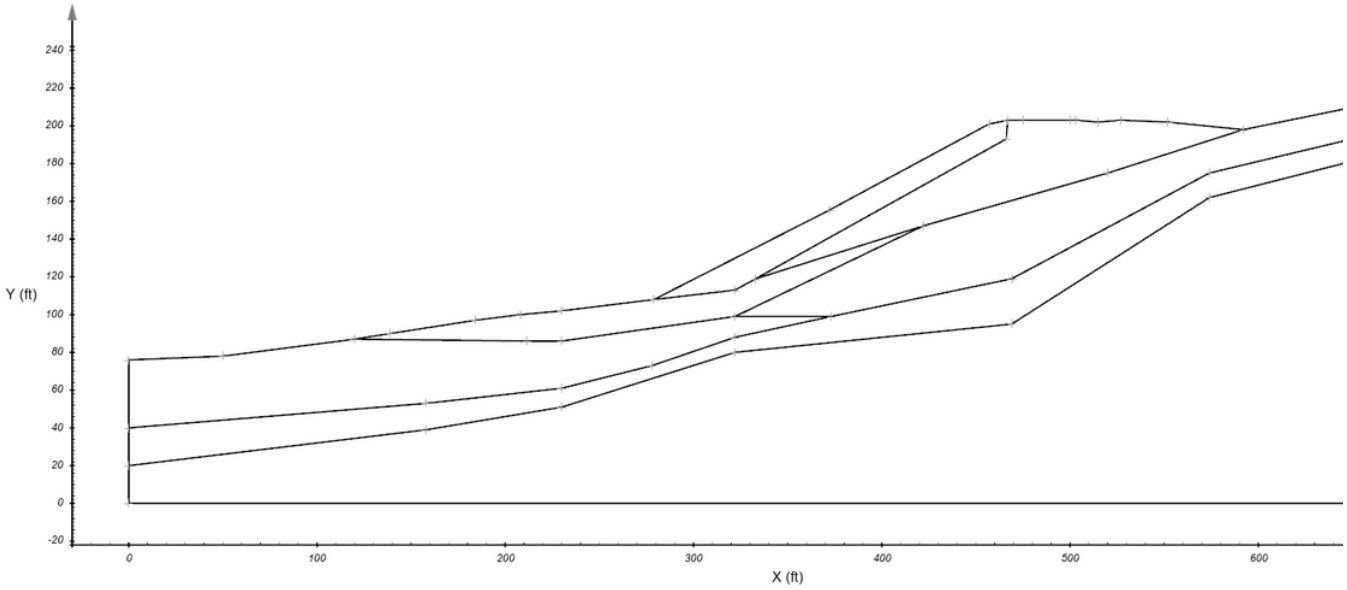
x (ft)	y (ft)
0	20
0	0
700	0
700	194
574	162
469	95
322	80
230	51
158	39

- **Import Geometry from Existing Model**

  Also, the regions can be created by importing them from existing models. In this tutorial, the geometry will be imported from the complete tutorial model which is included in the distribution models. Follow these steps to import geometry from existing model.

  1. Open the *Import Geometry* dialog by selecting *Geometry* > *Import* > *From existing Model* ... from the menu,
  2. Select Slopes_3D from the projects list,
  3. Select DFI_Case_Base_2D in the models list,
  4. Click the *Import* button to import geometry,
  5. Click Yes to Import Geometry pop-up message.

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 materials that will be used in the model. This section will provide instructions on creating the materials. In this case we assume that the user has measured the *Shear Strength* parameters, which can be found in the table below.

Tabs	Parameters	Material						
		Embankment Fill	**Colluvial 1**	**Silt and Clay**	**Sand and gravel**	**Rockfill**	**Colluvial 2**	**Glacial Till**
New Material	Method	Mohr Coulomb	Mohr Coulomb	Mohr Coulomb	Mohr Coulomb	Mohr Coulomb	Mohr Coulomb	Mohr Coulomb
Shear Strength	Cohesion (psf)	0	0	0	0	0	0	0
	Friction Angle, phi (deg)	32	30	25	36	45	34	38
	Unit Weight (lbf/ft^3)	125	125	120	130	135	125	135

1. Open the *Materials Manager* dialog by selecting *Materials* > *Manager* from the menu,
2. Click the *New* button to create a material,
3. Enter **Embankment Fill** for the material name in the dialog that appears
4. Choose **Mohr Coulomb** for the *Shear Strength Type* 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. Click the *OK* button to close the *Shear Strength* dialog,
9. **Repeat** steps 2 - 8 to create the remaining materials using the information provided in the table above,
10. Click *Ok* to close the *Materials Manager* Dialog.

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

1. The next step is to define which materials are applied to which regions.

1. Select *Geometry* > *Regions* ...,
2. For each region the appropriate material type must be selected from the combo box. The material assignments will be as follows:

Region	Material
R1	**Embankment Fill**
R2	**Colluvial 1**
R3	**Silt and Clay**
R4	**Sand and Gravel**
R5	**Rockfill**
R6	**Colluvial 2**
R7	**Glacial Till**

3. Click *OK* once the material assignments have been made.

e. Specify Search Method Geometry

The Entry and Exit method of searching for the critical slip surface has already been selected in a previous step. Now the user must specify the geometry that defines the search method. This is accomplished through the following steps:

1. Open the *Entry and Exit* dialog through the *Slips* > *Entry and Exit* ... menu option,
2. Enter the ***X* and *Increment*** values for the entry range and exit range as specified in the table below (note that the Y coordinates are calculated automatically),
3. Click *OK* to close the dialog.

Entry Range				Exit Range		
Right Side				Left Side		
	Left Point	Right Point			Left Point	Right Point
X	310	603		X	0	335
Increments	10			Increments	10	
Radius Increments:					10	

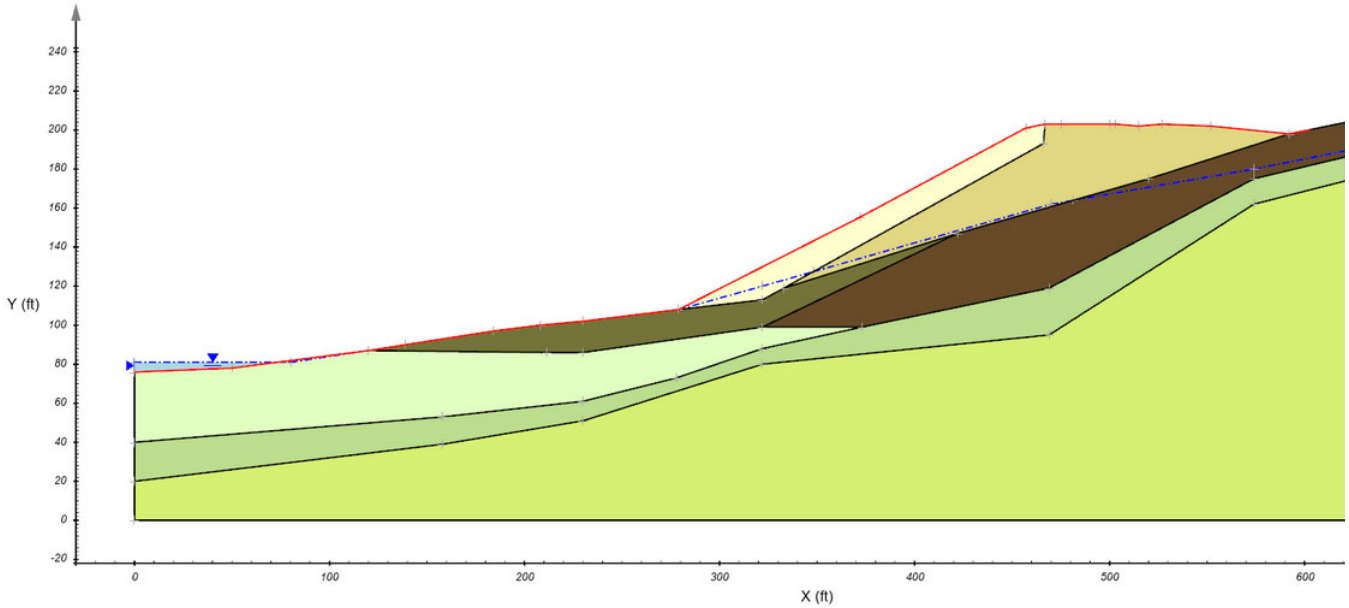
f. Specify Pore-Water (Pore-Water > Settings)

A water table or a piezometric line must be specified as Pore-Water for this model. In this model a piezometric line and water table will be used. In order to specify that a water table and piezometric line will be entered, the user needs to follow these steps:

1. Select *Pore-Water > Settings* ... from the menu,
2. Select the *PWP* tab, and choose **Water Surfaces** as the Pore-Water Pressure Method from the drop down menu. This will allow a water table to be entered,
3. Use the *Water Table...* button to access the *Water Table* dialog,
4. Copy *X* and *Y* coordinates as provided in the table below and click *Paste Points* button,
5. Under the Apply to Regions section ensure the all boxes except for R4 are checked,
6. Check the Show Water Table Line box on the top right hand corner of the dialog,
7. Check the Show outside of model box on the top right hand corner of the dialog,
8. Press *OK* to close the *Water Table* dialog,
9. Press *OK* to close the *Pore Water Pressure* dialog.

X (ft)	Y (ft)
0	81
73.333	81
80	81
120	87
139	90
184	97
208	100
230	102
279	108
322	120
348.119	127.412
470	162
481.415	163.976
574	180
700	205

Now your screen will look like the diagram below.

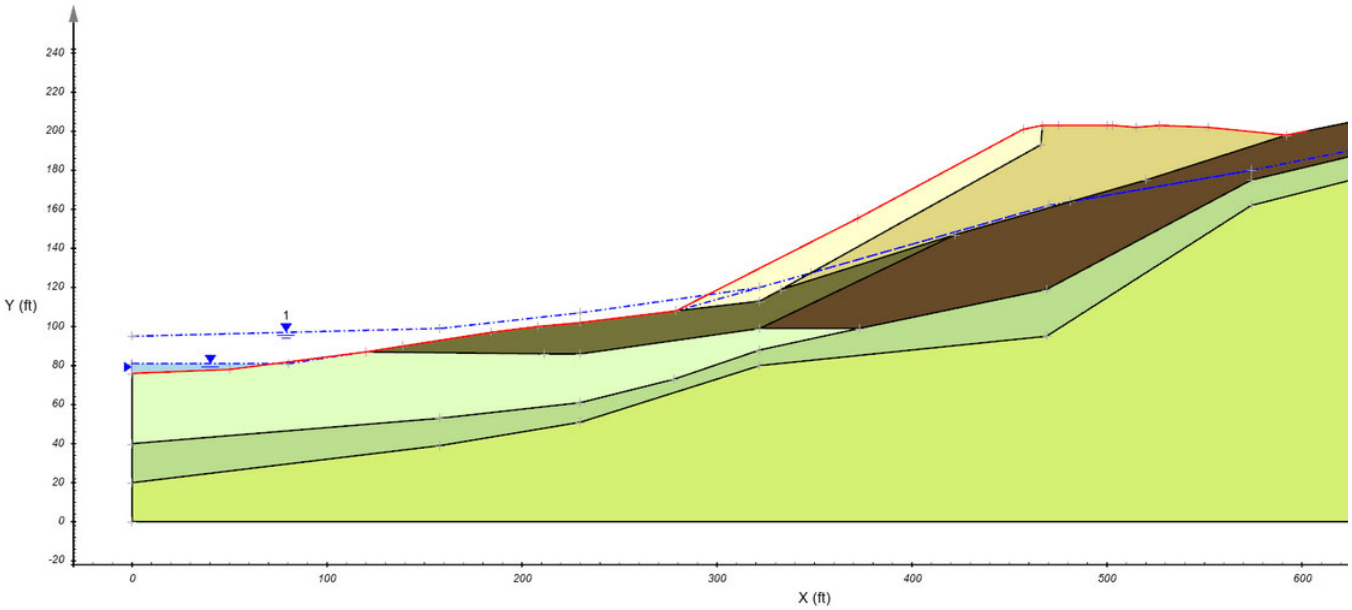


The user must then proceed to enter the piezometric line coordinates:

1. Select *Pore-Water > Piezometric Line* ...,
2. Select Piezometric Line 1 on the left hand side of the dialog box, and copy the *X* and *Y* coordinates as provided in the table below using CTRL-C and click the *Paste Points* button on the dialog to enter the coordinates for the piezometric line,
3. Under the Apply to Regions section ensure the check box for R4 only is checked.
4. Click the box for Show Piezometric Line 1 on the top right hand corner of the dialog.
5. Press *OK* to close the dialog.

X (ft)	Y (ft)
0	95
158	99
230	107
322	120
470	162
574	180
700	205

Now your screen will look like the diagram below.



Now that there is a water table noted in the model, we must revisit the material properties to add in the Unit Weight for the different materials above the water table (WT).

1. Open the *Material Properties* dialog by selecting Materials >*Manager* from the menu,
2. Select the Embankment Fill material, and clicking the *Properties* button to open the *Material Properties* dialog
3. At the bottom of the dialog, under Unit Weight, Check the box Unit Weight Above WT so that you can enter in the value of 120 in the box to the right.
4. Click *OK* to accept the changes.
5. Compete steps 2 - 4 for all of the materials using the properties given in the table below:

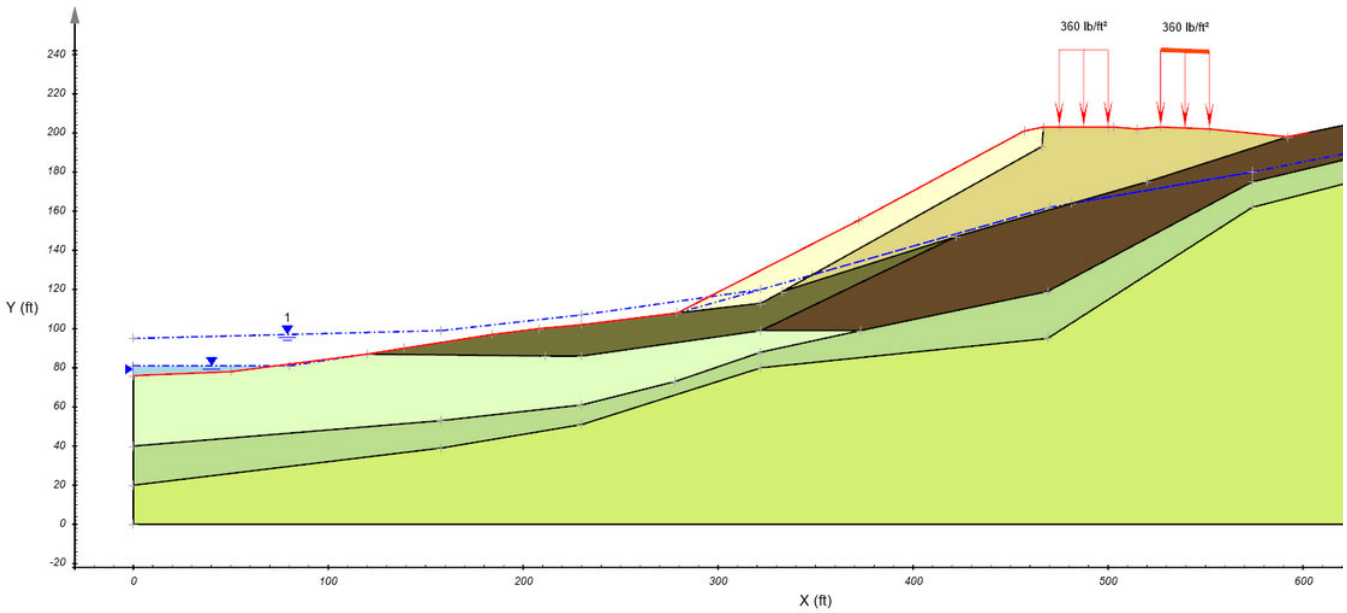
Tabs	Parameters	Material						
		Embankment Fill	**Colluvial 1**	**Silt and Clay**	**Sand and gravel**	**Rockfill**	**Colluvial 2**	**Glacial Till**
Shear Strength	Unit Weight above WT (lbf/ft^3)	120	120	117	125	130	120	130

g. Specify Loading Conditions (Loading)

There are 2 distributed loads used for the current model. The following steps are required in order to apply these distributed loads to the current model.

1. Ensure you have Region 1 chosen.
2. Open the *Distributed Load* dialog by selecting *Loading* > *Distributed Load* from the menu,
3. Click the *New* button to create a New Distributed Load 1 object,
4. Under Orientation, choose **Vertical**.
5. Under Type, choose **Constant**.
6. Enter a value of **360 lb/ft^2** for the Magnitude,
7. In the Acting Points Section, click the *Select* button and select the segment so that the Start Point equals **475 ft** for X and 203 for Y; the End Point will be automatically calculated based on the segment
8. **Ensure the box Show Distributed Load 1** is checked in order to see the load on the model.
9. Click the *New* button to create a New Distributed Load 2 object,
10. Under Orientation, choose **Vertical**.
11. Under Type, choose **Constant**.
12. Enter a value of **360 lb/ft2** for the Magnitude,
13. In the Acting Points Section, click the *Select* button and select the segment so that the Start Point equals **527 ft** for X and 203 for Y; the End Point will be automatically calculated based on the segment
14. **Check the box** Show Distributed Load 2 in order to see the load on the model.
15. Click *OK* to close the dialog.

Now you screen will look like the diagram below.



h. 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

j. 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 switch between the results of the different methods selected, click on the drop down menu (as shown below) at the top of the screen and select the method you would like to view.

To view the results in more detail proceed to Results and Discussion.

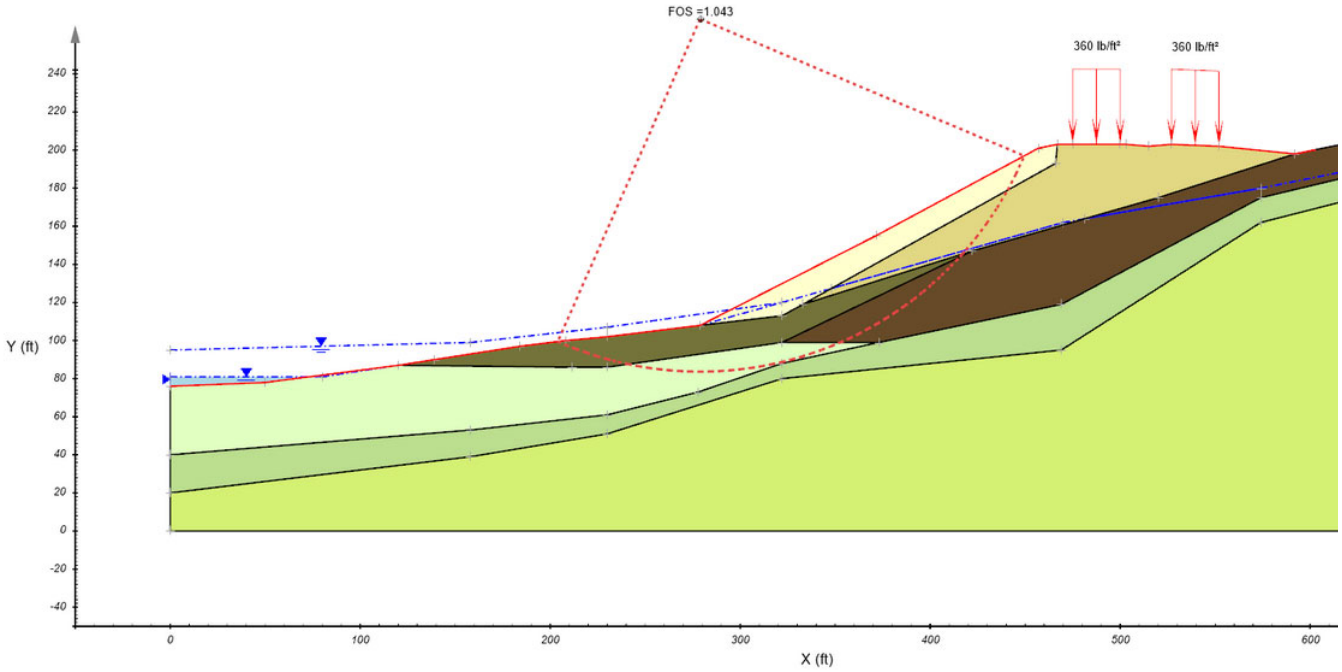
NOTE:

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

The critical slip surface for the numerical model is displayed when the model Results are first opened. The critical slip surface for M-P method is shown below.



The X axis may be different than shown. In order to change your axis, go to *View* > *World Coordinate System* and change it from Automatic to Manual. Enter in -50 for X and -50 for Y and the axis will chang