SVSLOPE™

2D Slope Stability Modeling Software

Tutorial Manual

ED-1A
Date: March 7, 2007

Written by:
Murray Fredlund, Ph.D.
Tiequn Feng, Ph.D.
Robert Thode, B.Sc.G.E.

Edited by:
Murray Fredlund, Ph.D.

SoilVision Systems Ltd.
Saskatoon, Saskatchewan, Canada

Software License

The software described in this manual is furnished under a license agreement. The software may be used or copied only in accordance with the terms of the agreement.

Software Support

Support for the software is furnished under the terms of a support agreement.

Copyright

Disclaimer of Warranty

SoilVision Systems Ltd. reserves the right to make periodic modifications of this product without obligation to notify any person of such revision. SoilVision does not guarantee, warrant, or make any representation regarding the use of, or the results of, the programs in terms of correctness, accuracy, reliability, currentness, or otherwise; the user is expected to make the final evaluation in the context of his (her) own problems.

Trademarks

1 Introduction

The Tutorial Manual serves a special role in guiding the first time users of the SVSLOPE software through a typical example problem. The example is "typical" in the sense that it is not too rigorous on one hand and not too simple on the other hand.

In particular this tutorial manual is designed to guide users through the range of reasonable models which may be encountered in typical slope stability modeling. The following examples represent the most typical models encountered in the traditional slope stability modeling practice and therefore include:

1. Basic slope (grid and radius), and
2. Weak layer example.

2 Basic Slope

The following example will introduce some of the features included in SVSLOPE and will set up a model using limited equilibrium method of slices and the Grid and Radius search method for circular slip surfaces. The purpose of this model is to determine the factor of safety of a simple model. The model dimensions and material properties are in the next section.

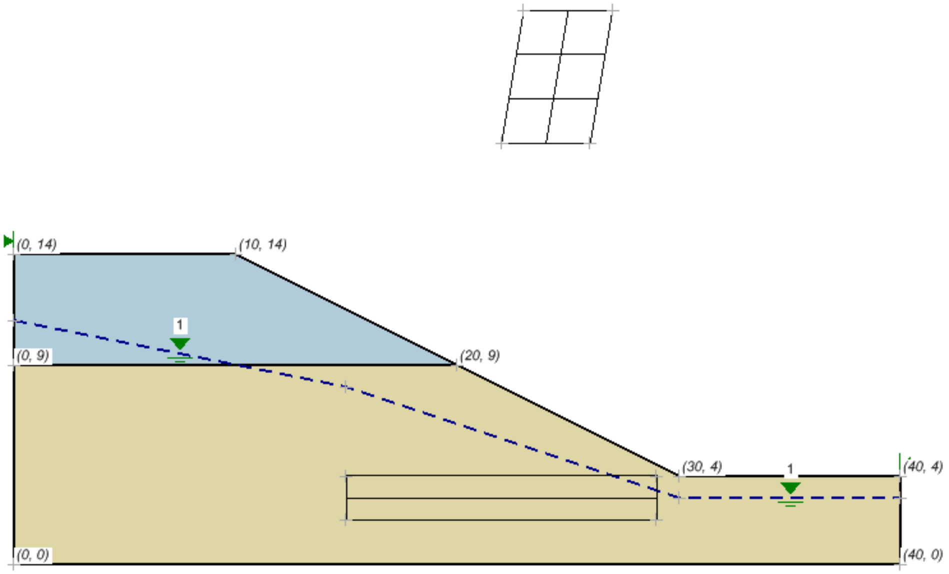
This example consists of a simple two layers slope with a water table. The problem is analyzed using the Bishop Simplified method as well as the Morgenstern-Price method. The purpose of this example is to illustrate the calculation of the factor of safety for a simple slope example.

This original model can be found under:

Project: Slopes_Group_2
Model: VW_9

Minimum authorization required to complete this tutorial: STUDENT

2.1 Model Description and Geometry



Region: R1

x (m)	y (m)
0	9
0	14
10	14
20	9

Region: R2

x (m)	y (m)
0	0
0	9
20	9
30	4
40	4
40	0

Material Properties

Material	c (kN/m^2)	ϕ (degrees)	γ (kN/m^3)
Upper Soil	5.0	20.0	15.00
Lower Soil	10.0	25.0	18.00

Grid and Tangent

Grid

x (m)	y (m)
23	25
22	19
26	19

Tangent

x (m)	y (m)
15	4
15	2
29	2
29	4

Piezometric Line

x (m)	y (m)
0	11
15	8
30	3
40	3

2.2 Model Setup

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

a. Create model

b. Specify analysis settings
c. Enter geometry
d. Specify search method
e. Specify initial conditions
f. Apply material properties
g. Run model
h. Visualize results

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

a. Create Model

Since FULL authorization is required for this tutorial, the user must perform the following steps to ensure full authorization is activated:

1. Plug in the USB security key,
2. Go to the *File > Authorization* dialog on the SVOFFICE Manager,
3. Software should display full authorization. If not, it means that the security codes provided by SoilVision Systems at the time of purchase have not yet been entered. Please see the the Authorization section of the SVOFFICE User's Manual for instructions on entering these codes.

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,
2. Select "ALL" under the Applications combo box and "ALL" for the Model Origin combo box,
3. Create a new project called "UserTutorial" by pressing the *New* button next to the list of projects,
4. Create a new model called "Basic Slope" by pressing the *New* button next to the list of models. Use the settings below when creating this new model:

   Application: SVSLOPE

   Model Name: Basic Slope

   Units: Metric

   Slope Direction: Left to Right

6. Click on the *World Coordinate System* tab,
7. Enter the World Coordinates System coordinates shown below into the dialog (leave Global Offsets as zero),

   x - maximum: 40

   y - maximum: 25

   x - minimum: -5

y - minimum: -2

8. Click on *OK*.

The new model will be automatically added under the recently created UserTutorial project.

SVSLOPE now opens to show a grid and the Options dialog (*View* > *Options)* pops up. Click *OK* to accept the default horizontal and vertical grid spacing of 1.0.

b. Specify Analysis Settings

In SVSlope the Analysis Settings provide the information for what model output will be available in ACUMESH. These settings will be specified as follows:

1. Select *Model* > *Settings* from the menu,
2. Move to the *Slip Surface* tab and ensure that the following items are selected:

Slope Direction:	*Left to Right*,
Slip Shape:	*Circular*
Search Method:	*Grid and Tangent*

3. Select the *Calculation Methods* tab from the dialog and select the method type as shown below:

   Bishop

4. Press *OK* to close the dialog.

c. Enter Geometry

Model geometry is defined as a series of layers and can be either drawn by the user or defined as a set of coordinates. Model Geometry can be imported from either.DXF files or from existing models.

This model will be divided into two regions, which are named R1 and R2. Each region will have one of the materials specified as its material properties. The shapes that define each material region will now be created. Note that when drawing a geometric shape, information will be added to the region that is current in the Region Selector. The Region Selector is at the top of the workspace.

- **Define R1 Region**

1. Select *Draw* > *Model Geometry* > *Polygon Region* from the menu,
2. The cursor will now be changed to a cross hair,
3. Move the cursor near (0,9) in the drawing space. You can view the coordinates of the current position of the mouse in the status bar,
4. To select the point as part of the shape left click on the point,
5. Now move the cursor near (0,14) and left-click the mouse. A line is now drawn from (0,9) to (0,14),
6. In the same manner then enter the following points:

(10,14)

(20,9)

7. Move the cursor near the point (20,9). Double click on the point to finish the shape. A line is now drawn from (0,9) to (20,9) and the shape is automatically finished by SVSLOPE by drawing a line from (20,9) back to the start point, (0,9).

Repeat this process to define the R2 region according to the information provided at the start of this tutorial.

Region: R2

x (m)	y (m)
0	0
0	9
20	9
30	4
40	4
40	0

NOTE:

If an error is made when entering the region geometry the user may recover from the error and start again by one of the following methods:

a. Press the escape (esc) key.
b. Select a region shape and press the delete key.
c. Use the Undo function on the Edit menu.

If all model geometry has been entered correctly the shape should look like the diagram at the beginning of this tutorial.

d. Specify Search Method

The Grid and Tangent method of searching for the critical slip surface has already been selected in the previous step. Now the user must draw the graphical representation of the grid and tangent objects on the screen. This is accomplished through the following steps:

GRID

1. Select *Model > Slip Surface > Grid and Tangent*,
2. Select the *Grid* tab,
3. Enter the values for the grid as specified at the start of this tutorial (the grid values may also be drawn on the CAD window),
4. Move to entering the tangent values.

TANGENT

1. Select the *Tangent* tab,
2. Enter the values for the tangent as specified at the start of this tutorial (the grid values may also be drawn on the CAD window),

3. Close the dialog.

The grid and tangent graphics should now be displayed on the CAD window.

e. Specify Initial Conditions

A water table or a piezometric line must be specified as an initial condition for this model. In this model a piezometric line will be used. In order to specify that a piezometric line will be entered the user needs to following these steps:

1. Select *Model* > *Initial Conditions* > *Settings*...,
2. Select "Water Surfaces" as the Pore-Water Pressure Method,
3. Press *OK* to close the dialog.

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

1. Select *Model* > *Initial Conditions* > *Piezometric Line*,
2. Under the *Points* tab, click on the *New Line* button,
3. Enter in the *X*(m) and *Y*(m) co-ordinates as provided at the start of this tutorial,
4. Under the Apply to Regions section put a check mark in the R1, R2 and R3 boxes to apply the line to both regions,
5. Press *OK* to close the dialog.

f. Apply Material Properties

The next step in defining the model is to enter the material properties for the two materials that will be used in the model. R1 region will have the Upper Soil applied to it and R2 will have the Lower Soil applied to it. This section will provide instructions on creating the Upper Soil. Repeat the process to add the second material.

1. Open the *Materials* dialog by selecting *Model* > *Materials* > *Manager* from the menu,
2. Click the *New* button to create a material,
3. Enter "Upper Soil" for the material name in the dialog that appears and choose Mohr Coulomb for the Shear Strength type of this material,
4. 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.

5. Move to the *Shear Strength* tab,
6. Enter the "Unit Weight" value of 15.000 kN/m^3,
7. Enter the "Cohesion", c: value of 5.000 Pa,
8. Enter the "Friction Angle", phi value of 20.000 deg,

9. Click the *OK* button to close the *Shear Strength* dialog.
10. Repeat these steps to create the Lower Soil material using the information provided at the beginning of the tutorial.

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

1. Open the *Region Properties* dialog by selecting *Model > Geometry > Region Properties* from the menu,
2. Select the *R1* region and assign the *Upper Soil* material to this region,
3. Select the R2 region and assign the *Lower Soil* material to this region,
4. Press the *OK* button to accept the changes and close the dialog.

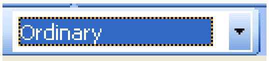
g. Run Model

The next step is to analyze the model.

1. Select *Solve > Analyze* from the menu. A pop-up dialog will appear,
2. Click on the *green arrow* button on the bottom of the dialog to start the solver. This action will finish the calculations and save the results,
3. Click on the *Close* button to close the dialog.

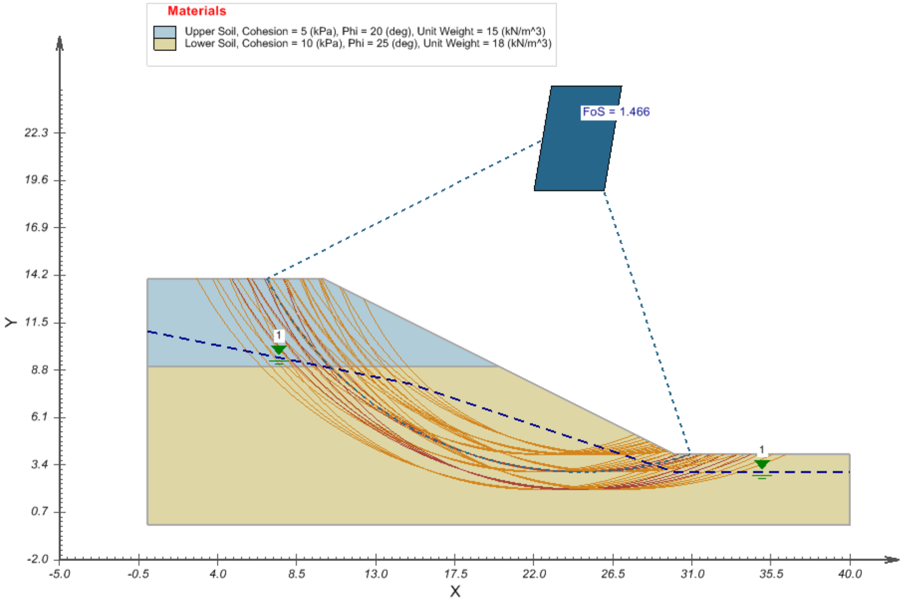
h. Visualize Results

After the model has been run, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click on *Yes*. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH as appears in the diagrams following. To switch back and forth between your original geometry and ACUMESH click on the SVSLOPE or ACUMESH icon which appears below the toolbars on the top left hand side of the screen. 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.



2.3 Results & Discussion

If the model has been appropriately entered into the software the approximate following results should be shown for the Bishop method. The user may display results from different methods by clicking the combo box on the display which lists the different analysis methods (Bishop, Spencer, etc.). It should be noted that it is typically recommended that the search grid of centers be somewhat larger in order to ensure that a critical center is not missed.



The correct results for this example are:

Method	SVSLOPE	
	Moment	Force
Bishop Simplified	1.466	
Spencer	1.469	1.469
M-P	1.468	1.468
GLE	1.468	1.468

3 Weak Layer Example

This is a more complex example involving a weak layer, pore-water pressures and surcharges. The ACADS verification program received a wide range of answers for this model and fully expected this during the program. The soil parameters, external loadings and piezometric surface are shown in the following diagram. Tension cracks are ignored in this example. The model requirement is that the noncircular slip surface and the corresponding factor of safety are required.

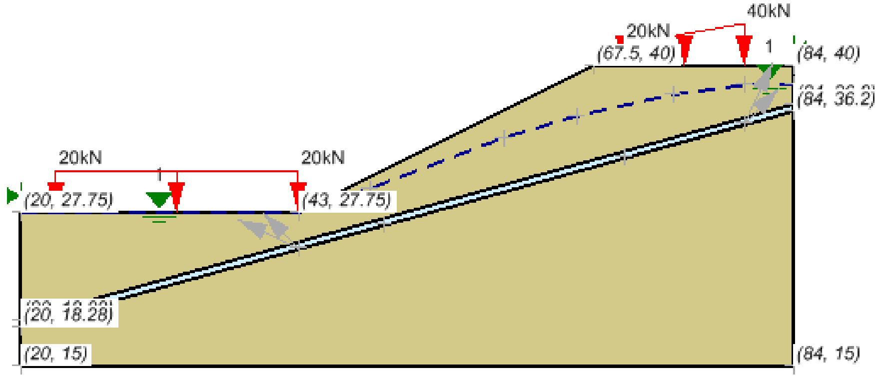
This original model can be found under:

Project: Slopes_Group_1
Model: VS_9
Minimum authorization required to complete this tutorial: FULL

(The STUDENT version of this software will not allow this model to be analyzed. Only purchased and authorized versions of SVSlope will allow entry and solving of this tutorial).

3.1 Model Description and Geometry

A block search for the critical noncircular failure surface is carried out by defining two line searches to block search squares within the weak layer. A number of different random surfaces were generated by the search and the results compared well with the actual results.



Region: R1

x (m)	y (m)
20	27
20	18
84	36
84	40
67	40
43	27

Region: R2

x (m)	y (m)
20	18

20	18
84	36
84	36

Region: R3

x (m)	y (m)
20	18
20	15
84	15
84	36

Material Properties

Material	c (kN/m^2)	ϕ (degrees)	γ (kN/m^3)
Soil #1	28.5	20.0	18.84
Soil #2	0.0	10.0	18.84

Piezometric Line

Pt. #	Xc (m)	Yc (m)
1	20.0	27.75
2	43.0	27.75
3	49.0	29.8
4	60.0	34.0
5	66.0	35.8
6	74.0	37.6
7	80.0	38.4
8	84.0	38.4

Loading

X (m)	Y (m)	Normal Stress (kN/m^2)
23.00	27.75	20.00
43.00	27.75	20.00
70.00	40.00	20.00
80.00	40.00	40.00

Block Search Parameters

Left Block

43	24.807
43	24.807
50	26.769

X increments: 10
Y increments: 1
Start Angle: 135 degrees

End Angle: 155 degrees
Left Increments: 2

Right Block

70	32.376
70	32.376
80	35.179

X increments: 10
Y increments: 1
Start Angle: 45 degrees
End Angle: 65 degrees
Left Increments: 2

3.2 Model Setup

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

a. Create model
b. Specify analysis settings
c. Enter geometry
d. Specify search method
e. Specify initial conditions
f. Specify loading conditions
g. Apply material properties
h. Run model
i. Visualize results

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

a. Create Model

Since FULL authorization is required for this tutorial, the user must perform the following steps to ensure full authorization is activated:

1. Plug in the USB security key,
2. Go to the *File > Authorization* dialog on the SVOFFICE Manager,
3. Software should display full authorization. If not, it means that the security codes provided by SoilVision Systems at the time of purchase have not yet been entered. Please see the the Authorization section of the SVOFFICE User's Manual for instructions on entering these codes.

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,
2. Select "ALL" for the Application, Model Origin, and Category combo boxes,
3. Create a new project called UserTutorial by pressing the *New* button next to the

list of projects,

4. Create a new model called "Weak Layer Example" by pressing the *New* button next to the list of models. Use the settings below when creating this new model:

   Application: SVSLOPE
   Model Name: Weak Layer Example
   Type: Steady-State
   Units: Metric
   Slope Direction: Left to Right

6. Click on the *World Coordinate System* tab,
7. Enter the World Coordinates System coordinates shown below into the dialog (leave Global Offsets as zero),

   x - maximum: 90
   y - maximum: 60
   x - minimum: 15
   y - minimum: 10

8. Click on *OK*.

The new model will be automatically added under the UserTutorial project.

SVSLOPE now opens to show a grid and the Options dialog (*View* > *Options)* pops up. Click *OK* to accept the default horizontal and vertical grid spacing of 1.0.

b. Specify Analysis Settings

In SVSlope the Analysis Settings provide the information for what model output will be available in ACUMESH. These settings will be specified as follows:

1. Select *Model* > *Settings* from the menu,
2. Move to the *Slip Surface* tab and ensure that the following items are selected:

   Slope Direction: *Left to Right*
   Slip Shape: *Non-Circular*
   Search Method: *Block*

3. Select the *Calculation Methods* tab from the dialog and select the method types as shown below:

   Spencer
   GLE

4. Press *OK* to close the dialog.

c. Enter Geometry

Model geometry is defined as a series of layers and can be either drawn by the user or defined as a set of coordinates. Model Geometry can be imported from either.DXF files or from existing models.

This model will be divided into three regions, which are named R1, R2 and R3. Each region will have one of the materials specified as its material properties. To add the necessary regions follow these steps:

1. Open the *Regions* dialog by selecting *Model > Geometry > Regions* from the menu,
2. Press the *New* button to add a second region and name it R2,
3. Click *OK* to close the dialog.

The shapes that define each material region will now be created. Note that when drawing a geometric shape, information will be added to the region that is current in the Region Selector. The Region Selector is at the top of the workspace.

- **Define R1 Region**

1. Ensure the R1 region is current in the region selector. The region selector appears underneath the menus at the top of the screen,
2. Select *Draw > Model Geometry > Polygon Region* from the menu,
3. The cursor will now be changed to cross hairs,
4. Move the cursor near (20,27) in the drawing space. You can view the coordinates of the current position the mouse is at in the status bar just above the command line,
5. When the cursor is near the point left-click. This will cause the cursor to snap to the point (The SNAP and GRID options in the status bar must both be on),
6. Now move the cursor near (20,18) and left-click. A line is now drawn from (20,27) to (20,18),
7. In the same manner then enter the following points:
   (84,36)
   (84,40)
   (67,40)
   (43,27)
8. Move the cursor near the point (43,27) and double-click on the point to finish the shape.

Repeat this process to define the R2 and R3 regions according to the information provided at the start of this tutorial.

Region: R2

x (m)	y (m)
20	18
20	18
84	36
84	36

Region: R3

x (m)	y (m)
20	18
20	15
84	15
84	36

NOTE:

If an error is made when entering the region geometry the user may recover from the error and start again by one of the following methods:

a. Press the escape (esc) key
b. Select a "Region Shape" and press the *delete* key
c. Use the Undo function on the Edit menu

If all model geometry has been entered correctly the shape should look like the diagram at the beginning of this tutorial.

d. Specify Search Method

This particular model makes use of a block search methodology. The block search parameters may be entered through the following steps:

1. Open the *block search* dialog through the *Model > Slip Surface > Block Search...* menu option,
2. Enter the block search data as specified in the previous section,
3. Click *OK* to close the dialog.

e. Specify Initial Conditions

Initial conditions are generally associated with transient model runs. Their purpose is to provide a reasonable starting point for the solver. In a steady-state model, initial conditions can be used to "precondition" the solver to allow faster convergence. Generally speaking, the user will enter information either for a water table or a piezometric line. In this model a piezometric line will be used. In order to specify that a piezometric line will be entered the user needs to following these steps:

1. Select *Model > Initial Conditions > Settings...*,
2. Select "Water Surfaces" as the Pore-Water Pressure Method,
3. Press *OK* to close the dialog.

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

1. Select *Model > Initial Conditions > Piezometric Line*,
2. Under the *Points* tab, click on the *New Line* button,

3. Enter in the *X*(m) and *Y*(m) co-ordinates as provided at the start of this tutorial,
4. Under the Apply to Regions section put a check mark in the R1, R2 and R3 boxes to apply the line to both regions,
5. Press *OK* to close the dialog.

f. Specify Loading Conditions

Two distributed loads are applied in this numerical model. The instructions for applying these distributed loads are as follows:

1. Select *Draw > Loading > Distributed Load*, then
2. Enter the data as provided in the previous section,
3. Click OK to close the dialog. You will need to do this for each load separately.

g. Apply Material Properties

The next step in defining the model is to enter the material properties for the two materials that will be used in the model. R1 region will have the Upper Soil applied to it and R2 will have the Lower Soil applied to it. This section will provide instructions on creating the Upper Soil. Repeat the process to add the second material.

1. Open the *Materials* dialog by selecting *Model > Materials > Manager* from the menu,
2. Click the *New* button to create a material,
3. Enter "Upper Soil" for the material name in the dialog that appears and choose Mohr Coulomb for the Shear Strength type of this material,
4. 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.

5. Move to the *Shear Strength* tab,
6. Enter the Unit Weight value of 15.000 kN/m^3,
7. Enter the Cohesion, c: value of 5.000 Pa,
8. Enter the Friction Angle, phi value of 20.000 degrees,
9. Click the *OK* button to close the *Shear Strength* dialog,
10. Repeat these steps to create the Lower Soil material using the information provided at the beginning of the tutorial.

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

1. Open the *Region Properties* dialog by selecting *Model > Geometry > Region Properties* from the menu,

2. Select the R1 region and assign the Upper Soil material to this region,
3. Select the R2 region and assign the Lower Soil material to this region,
4. Press the *OK* button to accept the changes and close the dialog.

h. Run Model

The next step is to analyze the model.

1. Select *Solve* > *Analyze* from the menu. A pop-up dialog will appear,
2. Click on the *green arrow* button on the bottom of the dialog to start the solver. This action will finish the calculations and save the results,
3. Click on the *Close* button to close the dialog.

i. Visualize Results

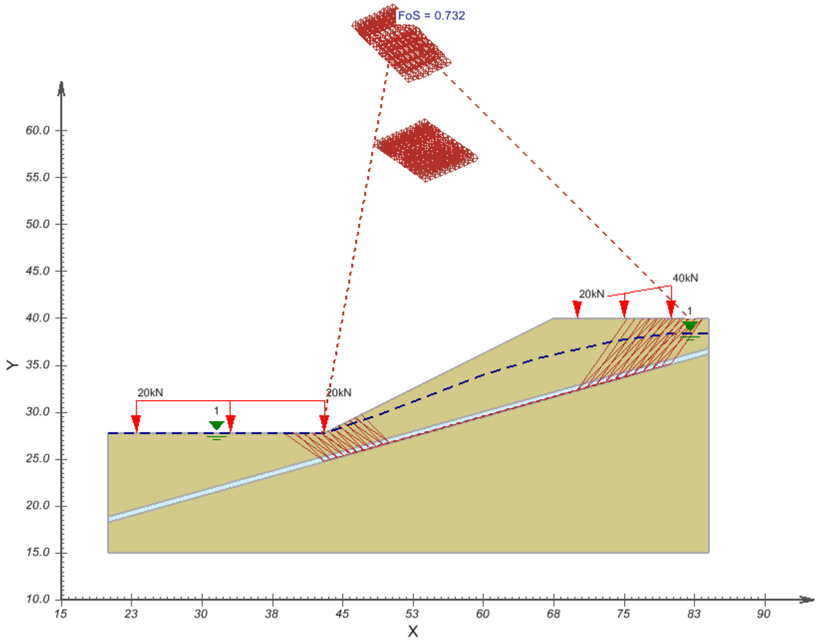
After the model has been run, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click on *Yes*. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH as appears in the diagrams following. To switch back and forth between your original geometry and ACUMESH click on the SVSLOPE or ACUMESH icon which appears below the toolbars on the top left hand side of the screen. 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.

3.3 Results & Discussion

After the model is completed the user may view the results in the ACUMESH software by pressing the open ACUMESH icon on the process toolbar. The results will contain all trial slip surfaces as well as the most critical slip surface results. In order to identify the most critical slip surface the user may perform the following steps:

1. Select "Slip Surfaces" from the menu item, and
2. Click the *Show Trial Slip Surfaces* button, this will cause all the trial slip surfaces to not be displayed.

The user may also plot the slices used in the analysis of the critical slip surfaces through the slips show slices menu option. The information on any particular slice may be displayed through the slips slice information dialog. A slice information dialog will appear and the user may click on a new particular slice on the slope to display the details of that slice. The analysis results in a factor of safety of 0.732.


FoS = 0.732
40kN
20kN
20kN
20kN
1
1
Y
60.0
55.0
50.0
45.0
40.0
35.0
30.0
25.0
20.0
15.0
10.0
15
23
30
38
45
53
60
68
75
83
90
X

This page is left blank intentionally.