

2D Slope Stability Modeling Software

Tutorial Manual

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

Information contained within this Tutorial Manual is copyrighted and all rights are reserved by SoilVision Systems Ltd. The SVSLOPE software is a proprietary product and trade secret of SoilVision Systems. The Tutorial Manual may be reproduced or copied in whole or in part by the software licensee for use with running the software. The Tutorial Manual may not be reproduced or copied in any form or by any means for the purpose of selling the copies.

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

WindowsTM is a registered trademark of Microsoft Corporation. SoilVision® is a registered trademark of SoilVision Systems Ltd. SVFLUX TM is a trademark of SoilVision Systems Ltd. CHEMFLUX TM is a trademark of SoilVision Systems Ltd. SVSOLID TM is a trademark of SoilVision Systems Ltd. SVHEAT TM is a trademark of SoilVision Systems Ltd. SVSLOPE ® is a registered trademark of SoilVision Systems Ltd. ACUMESH TM is a trademark of SoilVision Systems Ltd. FlexPDE® is a registered trademark of PDE Solutions Inc.

> Copyright © 2011 by SoilVision Systems Ltd. Saskatoon, Saskatchewan, Canada ALL RIGHTS RESERVED Printed in Canada

3

1 Introduction		4
2 Basic Slope		5
2.1 Model Setup		7
2.2 Results and Disc	ussions	11
3 Weak Layer Example		12
3.1 Model Setup		14
3.2 Results and Disc	ussions	18
4 Geomembrane Exam	nple	20
4.1 Model Setup		22
4.2 Results and Disc	ussions	27
5 Dynamic Programmi	ng Example	28
5.1 Model Setup		30
5.2 Results and Disc	ussions	38
6 Kulhawy Method		40
6.1 Model Setup		40
6.2 Results and Disc	ussions	42
7 3D Multi Planar Exam	nple	44
7.1 Model Setup		46
7.2 Results and Disc	ussions	51
8 3D Grid and Tangent	Submergence Example	53
8.1 Model Setup		55
8.2 Results and Disc	ussions	61

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,
- 2. Weak Layer Example,
- 3. Geomembrane Example, and
- 4. Dynamic Programming 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

Model Description and Geometry





Region Geometries

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 (kPa)	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

2.1 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 geometry
- e. Specify pore water pressure
- f. Apply material properties
- g. Run model
- h. Visualize results

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

a. Create Model

The following steps are required to create the model:

- 1. Open the SVOFFICE Manager dialog,
- 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:SVSLOPEModel Name:Basic SlopeUnits:Metric

Slope Direction: Left to Right

- 5. Click on the World Coordinate System tab,
- 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
- 7. 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 (Model > 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 Simplified, Spencer, M-P, and GLE

4. Press *OK* to close the dialog.

c. Enter Geometry (Model > 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)
- 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.

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 Geometry

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 Pore Water Pressure (Model > Pore Water Pressure)

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 > Pore Water Pressure > 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 > Pore Water Pressure > 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,
- Under the Apply to Regions section put a check mark in the R1 and R2 boxes to apply the line to both regions,
- 5. Press *OK* to close the dialog.

f. Apply Material Properties (Model > 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. 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.

- 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 kPa,
- 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.

- 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 (Solve > Analyze)

The next step is to analyze the model.

- Select Solve > Analyze from the menu. A pop-up dialog will appear and the solver will start,
- 2. Press the *OK* button to close the dialog.

h. Visualize Results (Window > AcuMesh)

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.



2.2 Results and Discussions

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:

M. d. J		
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: STUDENT

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	27.75
20	18.88
84	36.8
84	40
67.5	40
43	27.75

Region: R2

x (m)	y (m)
20	18.88
20	18.28
84	36.2

|--|

Region: R3

x (m)	y (m)
20	18.28
20	15
84	15
84	36.2

Material Properties

Material	c (kPa)	$\phi(\text{degrees})$	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

Type: Trapezoid

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 Right Increments: 2

3.1 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 geometry
- e. Specify pore water pressure
- f. Specify loading conditions
- g. Apply material properties
- h. Run model
- i. Visualize results

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

a. Create Model

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,
- 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:SVSLOPEModel Name:Weak Layer ExampleUnits:MetricSlope Direction:Right to Left

- 5. Click on the World Coordinate System tab,
- 6. 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
- 7. 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 (Model > 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:	Right to Left
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. For GLE method, press the *Lambda...* button,
- 5. Enter a Start Value of -1.25, an Interval of 0.25, and a Number of 11,
- 6. Press the Generate button,
- 7. Press OK to close the dialogs.

c. Enter Geometry (Model > 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:

- 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 (20,27.75) 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),
- Now move the cursor near (20,18.88) and left-click. A line is now drawn from (20,27.75) to (20,18.88),
- 7. In the same manner then enter the following points:
 - (84,36.8) (84,40) (67.5,40) (43,27.75) Move the c
- 8. Move the cursor near the point (43,27.75) 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.

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 Geometry

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

- Open the *block search* dialog through the *Model > Slip Surface > Block Search...* menu option,
- 2. Enter the right block search data and then left block search data as specified in

the start of this tutorial,

3. Click *OK* to close the dialog.

e. Specify Pore Water Pressure (Model > Pore Water Pressure)

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 > Pore Water Pressure > 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 > Pore Water Pressure > 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 all 3 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 start of this tutorial,
- 3. Click *OK* to close the dialog. You will need to do this for each load separately.

g. Apply Material Properties (Model > 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. 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.

- 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,

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 18.84 kN/m^3,
- 7. Enter the Cohesion, c: value of 28.5 kPa,
- 8. Enter the Friction Angle, phi value of 20.0 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.

- 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. Select the R3 region and assign the Upper Soil material to this region,
- 5. Press the *OK* button to accept the changes and close the dialog.

h. Run Model (Solve > Analyze)

The next step is to analyze the model.

- Select Solve > Analyze from the menu. A pop-up dialog will appear and the solver will start,
- 2. Press the OK button to close the dialog.

i. Visualize Results (Window > AcuMesh)

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.

rdinary 🗾

3.2 Results and Discussions

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 Slips, 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 0.741 for the Spencer method and 0.708 for the GLE method.



4 Geomembrane Example

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 effects of reinforcements. The model dimensions and material properties are in the next section.

This original model can be found under: Project: Slopes_Group_2

Model: VW_6_Fabric

Minimum authorization required to complete this tutorial: STUDENT

Model Description and Geometry



Region Geometries

Region: R1

x (m)	y (m)
0	5
0	15
9	15
19	5

Region: R2

x (m)	y (m)
0	3

0	5
19	5
29	5
29	3

Material Properties

Sandy Clay

Shear Strength	Unit Weight	Cohesion, c:	Friction Angle, phi:
Туре	kN/m^3	kPa	deg
Mohr Coulomb	18	10	30

Silty Clay

,,			
Shear Strength	Unit Weight	Cohesion, c:	Friction Angle, phi:
Туре	kN/m^3	kPa	deg
Mohr Coulomb	18	10	25

Grid and Tangent

Grid - Points			Tangent - Poin	ts	
	Х	Y		Х	Y
Upper Left	18	17	Upper Left	12	4
Lower Left	18	17	Lower Left	12	4
Lower Right	18	17	Lower Right	20	4
			Upper Right	20	4

X increments Y increments Increments

Y increments

Loading

Line Load #1

Orientation:	Vertical			
	Start Point		End Point	
M agnitude	10	kN		kN
X-coord:	8	m		m
Y-Coord:	15	m		m

Analysis Settings

Pore-Water Pressure		
Pore Fluid Unit Weight	9.807	kN/m^3

LE Convergence Options

Number of		Max. no. of
Slices	Tolerance	iterations
30	0.0010	50

4.1 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 geometry
- e. Specify pore water pressure
- f. Specify loading conditions
- g. Apply material properties
- h. Add Supports
- i. Run model
- j. Visualize results

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

a. Create Model

The following steps are required to create the model:

- 1. Open the SVOFFICE Manager dialog,
- Select "ALL" under the Applications combo box and "ALL" for the Model Origin combo box,
- Create a new project called UserTutorial by pressing the New button next to the list of projects,
- Create a new model called VW_6_Tutorial by pressing the *New* button next to the list of models. The new model will be automatically added under the recently created UserTutorial project. Use the settings below when creating this new model,
- 5. Select the following:

Application:SVSLOPESystem:2DUnits:MetricSlope 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 minimum: -2
- y minimum: 2
- x maximum: 30
- y maximum: 18
- 3. Click OK to close the dialog,
- 4. Click *OK* to close the Options dialog, which will pop-up.

b. Specify Analysis Settings (Model > Settings)

In SVSlope the Analysis Settings provide the information for what type of analysis will be performed. These settings will be specified as follows:

- 1. Select *Model* > *Settings* from the menu,
- 2. Select the Slip Surface tab,

Slope Direction:Left to RightSlip Shape:CircularCreate tension crack for reverse curvature: OnSearch Method:Grid and Tangent

- 3. Select the *Calculation Methods* tab from the dialog and select the method types as shown below:
 - Ordinary / Felenius Bishop Simplified Janbu Simplified
 - Morgenstern-Price
 - GLE (Fredlund)
- 4. For GLE method, press the Lambda... button,
- 5. Enter a Start Value of -1.25, an Interval of 0.25, and a Number of 11,
- 6. Press the Generate button,
- 7. Press OK to close the dialogs.

c. Enter Geometry (Model > 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. The shapes that define each material region can be created by the following steps.

Define R1 Region

- 1. Select Draw > Model Geometry > Region Polygon from the menu,
- 2. Move the cursor near (0,5) 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,

3. Continue drawing the following points in order, and

x (m)	y (m)
0	5
0	15
9	15
19	5

4. Double click on the final point to finish the shape.

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

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 Geometry

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 Pore Water Pressure (Model > Pore Water Pressure)

There are no initial conditions associated with this tutorial.

f. Specify Loading Conditions

There is a single line-load used for the current model. The following steps are required in order to apply this line load to the current model.

- Open the Line Load dialog by selecting Model > Loading > Line Load from the menu,
- 2. Click the New button to create a new line load object,
- 3. Enter a value of 10 kN for the magnitude,
- 4. Enter a value of 8 m for the X coordinate,
- 5. Make sure that the load has a "Vertical" orientation,
- 6. Click OK to close the dialog.

g. Apply Material Properties (Model > 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. *R1* region will have the *Sandy Clay* applied to it and *R2* will have the *Silty Clay* applied to it. This section will provide instructions on creating the Sandy Clay. Repeat the process to add the second material.

- Open the Materials dialog by selecting Model > Materials > Manager from the menu,
- 2. Click the New button to create a material,
- 3. Enter "Sandy Clay" 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 18 kN/m^3,
- 7. Enter the Cohesion, c: value of 10 kPa,
- 8. Enter the Friction Angle, phi value of 30 degrees,
- 9. Click the OK button to close the Shear Strength dialog, and
- 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.

 Open the Region Properties dialog by selecting Model > Geometry > Regions 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. Add Supports (Model > Support)

The next step is to analyze the model.

- Open the Support Type Manager dialog by selecting Solve > Support > Type Manager from the menu,
- 2. Press the New button to open the New Support Property dialog,
- 3. Select Geotextile as the Support Type and enter a name,
- 4. Click OK,
- 5. Select Passive as the Force Application,
- 6. Set 0 kPa for Adhesion,
- 7. Click OK, to close the dialog,
- 8. Open the Support Geometry dialog by selecting Solve > Support > Geometry from the menu,
- 9. Click New to create a new support entry,
- 10. Leave the Orientation as None and enter the coordinates (18,6) and (6,6),
- 11. Click New to create a second support entry,
- 12. Leave the Orientation as None and enter the coordinates (14,10) and (2,10),
- 13. Click *OK*, to close the dialog.

i. Run Model (Solve > Analyze)

The next step is to analyze the model.

- 14. Select Solve > Analyze from the menu. A pop-up dialog will appear,
- 15. 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,

16. Click on the *Close* button to close the dialog.

j. Visualize Results (Window > AcuMesh)

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 & 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.



4.2 Results and Discussions

The results of the calculation factor of safety may be seen below. At the end of calculation the factor of safety is a result approximately 1.525. The support force distribution in shown along each support in the screenshot below from 0 to 50.



5 Dynamic Programming Example

This example will introduce the user to modeling in SVSLOPE using the SAFE-DP stressbased limit equilibrium calculation method and the Dynamic Programming search method for non-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 three-layer slope with a thin weak layer. A combination of SVSolid and SVSlope are used to solve this model. The purpose of this example is to illustrate the calculation of the factor of safety for a the slope.

This original model can be found under:

Project:	Slopes_STUDENT
Model:	SAFE_52_EDU

Minimum authorization required to complete this tutorial: PROFESSIONAL



Model Description and Geometry

Region Geometries

Region:Upper Layer

x (m)	y (m)
0	27
0	13
80	13
80	15
44	15
20	27

Region: Weak Layer x (m) y (m)

0	13
0	12
80	12
80	13

Region: Lower Layer

x (m)	y (m)	
0	12	
0	0	
80	0	
80	12	

Material Properties

It should be noted that the Mohr-Coulomb soil properties are required for the SVSlope portion of the analysis only. The SVSolid portion of the analysis is a linear elastic analysis and the Mohr-Coulomb properties are not required for the stress portion of the analysis. Current research into the Dynamic Programming method has shown that the difference in the computed FOS between whether an elasto-plastic strength model is used or a linear elastic strength model is used in the base finite element analysis makes for negligible difference if the FOS is greater than 1.0. Therefore a linear elastic stress analysis is more than adequate for most situations.

Material	SVS olid Type	SVSlope Type	c (kPa)	¢(degrees)	γ (kN/m³)	E (kPa)	v
Upper Soil	Linear Elastic	Mohr	10.0	30.0	15.00	15000	0.33
		Coulomb					
Weak	Linear Elastic	Mohr	0.0	10.0	18.00	2000	0.45
Layer		Coulomb					
Lower Soil	Linear Elastic	Bedrock	-	-	20.00	100000	0.35

Search Boundary

Point	Value (m)
Тор Ү	29.967
Int Y	17.151
VarY	16.132
Bottom Y	7.981
Left X	10.256
Int 1 X	18.462
Var X	34.872
Int 2 X	41.026
Right X	53.333

Dynamic Programming Grid Lines

х	40
у	160

5.1 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:

SVSolid Steps:

- a. Create model
- b. Enter geometry
- c. Specify boundary conditions
- d. Apply SVSolid material properties
- e. Specify model output
- f. Run SVSolid model

SVSlope Steps:

- g. Combine SVSlope with SVSolid
- h. Specify analysis settings
- i. Specify dynamic programming grid
- j. Specify search boundary coordinates
- k. Apply SVSlope material properties
- I. Run SVSlope model
- m. Visualize results

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

a. Create Model

Since FULL authorization is required for this tutorial, 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, and
- 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,

- 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 "DP Example" by pressing the *New* button next to the list of models. Use the settings below when creating this new model:

Application:SVSOLIDModel Name:DP ExampleSystem:2DUnits:Metric

- 5. Click on the World Coordinate System tab,
- Enter the World Coordinates System coordinates shown below into the dialog (leave Global Offsets as zero),
 - x maximum: 90
 - y maximum: 35
 - *x* minimum: -5
 - y minimum: -5
- 7. 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. Enter Geometry (Model > 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 Upper Layer, Weak Layer, and Lower Layer. Each region will have one of the materials listed above 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 Upper Layer 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 (0,27) 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 (0,13) and left-click the mouse. A line is now drawn from (0,27) to (0,13),

6. In the same manner then enter the following points:

(80,13) (80,15) (44,15)

7. Move the cursor near the point (20,27). Double click on the point to finish the shape. A line is now drawn from (44,15) to (20,27) and the shape is automatically finished by SVSLOPE by drawing a line from (20,27) back to the start point, (0,27).

Repeat this process to define the Weak Layer and Lower Layer regions according to the information provided at the start of this tutorial.

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.

• Specify Region Names

- 1. Select Model > Geometry > Regions from the menu,
- 2. The Regions dialog will be opened,
- 3. Select the name "R1" in the list,
- 4. Enter the name "Upper Layer",
- 5. Select the name "R2",
- 6. Enter the name "Weak Layer",
- 7. Select the name "R3",
- 8. Enter the name "Lower Layer",
- 9. Press OK to close the dialog.

c. Specify Boundary Conditions (Model > Boundaries)

Boundary conditions must be applied to region points. Once a boundary condition is applied to a boundary point the starting point is defined for that particular boundary condition. The boundary condition will then extend over subsequent line segments around the edge of the region in the direction in which the region shape was originally entered. Boundary conditions remain in effect around a shape until re-defined. The user may not define two different boundary conditions over the same line segment.

More information on boundary conditions can be found in *Menu System > Model Menu > Boundary Conditions > 2D Boundary Conditions* in your User's Manual.

Now that all of the regions and the model geometry have been successfully defined, the next step is to specify the boundary conditions. The sides of the model will be fixed in the x direction to prevent lateral movement and the model will be fixed in both directions at its base. The steps for specifying the boundary conditions are as follows:

Upper Layer

- 1. Select the "Upper Layer" region in the region selector,
- From the menu select Model > Boundaries > Boundary Conditions. The boundary conditions dialog will open. By default the first boundary segment will be given a Free condition in both the x and y directions,
- 3. Select the point (0,27) from the list on the Segment Boundary Conditions tab,
- 4. From the X Boundary Condition drop-down select a Fixed boundary condition,
- 5. Select the point (0,13) from the list,
- 6. In the X Boundary Condition drop-down select a Free boundary condition,
- 7. Click OK to save the input Boundary Conditions and return to the workspace,

NOTE:

The Free Y boundary condition for the point (0,27) becomes the boundary condition for the following line segments that have a Y Continue boundary condition and the Free X boundary condition for the point (0,13) becomes the boundary condition for the following line segments that have a X Continue boundary condition, until a new boundary condition is specified.

• Weak Layer

- 8. Select the "Weak Layer" region in the region selector,
- From the menu select Model > Boundaries > Boundary Conditions to open the Boundaries dialog,
- 10. Select the point (0,13) from the list,
- 11. From the X Boundary Condition drop-down select a Fixed boundary condition,
- 12. Select the point (0,12) from the list,
- 13. From the X Boundary Condition drop-down select a Fixed boundary condition,
- 14. Select the point (80,12) from the list,
- 15. From the X Boundary Condition drop-down select a Fixed boundary condition,
- 16. Select the point (80,13) from the list,
- 17. From the X Boundary Condition drop-down select a Fixed boundary condition,
- 18. Click OK to save the input Boundary Conditions and return to the workspace,

Lower Layer

- 19. Select the "Lower Layer" region in the region selector,
- 20. From the menu select *Model* > *Boundaries* > *Boundary Conditions* to open the *Boundaries* dialog,
- 21. Select the point (0,12) from the list,
- 22. From the X Boundary Condition drop-down select a Fixed boundary condition,
- 23. Select the point (0,0) from the list,
- 24. From the X Boundary Condition drop-down select a Fixed boundary condition,

- 25. From the Y Boundary Condition drop-down select a Fixed boundary condition,
- 26. Select the point (80,0) from the list,
- 27. From the X Boundary Condition drop-down select a Fixed boundary condition,
- 28. From the Y Boundary Condition drop-down select a Free boundary condition,
- 29. Select the point (80,12) from the list,
- 30. From the X Boundary Condition drop-down select a Free boundary condition,
- 31. From the Y Boundary Condition drop-down select a Free boundary condition,
- 32. Click *OK* to save the input Boundary Conditions and return to the workspace.

d. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the materials that will be used in the model. The material names in this tutorial match the region names. This section will provide instructions on creating the materials and entering the SVSolid material parameters. The SVSlope material parameter specification will be described below.

The steps below are instructions for the Upper Layer material. Repeat the process to add the other materials.

- Open the Materials dialog by selecting Model > Materials > Manager from the menu,
- 2. Click the New button to create a material,
- 3. Enter "Upper Layer" for the material name in the dialog that appears and choose Linear Elastic for the Data 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. The Parameters tab will be current,
- 6. Enter the "Young's Modulus" value of 15000 kPa,
- 7. Enter the "Poisson's Ratio" value of 0.33,
- 8. Enter the "Unit Weight" value of 15 kN/m^3,
- 9. Check the "Apply Vertical Body Load" box,
- 10. Click the OK button to close the dialog,
- 11. Repeat these steps to create the Weak Layer and Lower Layer materials 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.

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

- Select the Upper Layer region and assign the Upper Layer material to this region,
- 3. Select the Weak Layer region and assign the *Weak Layer* material to this region,
- 4. Select the Lower Layer region and assign the *Lower Layer* material to this region,
- 5. Press the *OK* button to accept the changes and close the dialog.

e. Specify Model Output

PLOT MANAGER (Model > Reporting > Plot Manager)

The plot manager dialog is used to specify information to display in the solver. There are many plot types that can be specified to visualize the results of the model. The defaults will be generated for this tutorial example model.

- 1. Open the *Plot Manager* dialog by selecting *Model* > *Reporting* > *Plot Manager* from the menu,
- The default plots for the model are automatically generated and displayed in the plot manager,
- 3. Use the Properties button to view more details on any of the plots listed,
- 4. Click OK to close the *Plot Manager* and return to the workspace.

OUTPUT MANAGER (Model > Reporting > Output Manager)

The output manager dialog is used to specify information to export to other software, including SVSlope and the AcuMesh visualization software.

Since a combined SVSolid/SVSlope model is being created an output file of the finiteelement stress results must be specified.

- 1. Open the *Output Manager* dialog by selecting *Model* > *Reporting* > *Output Manager* from the menu,
- The default AcuMesh output file for the model is automatically generated and displayed,
- 3. Click the New SVSlope button to add the required output file,
- The Output Properties dialog will be open and the default title of DP Example SlopeStabilityData and variables sx,sy,sxy will be displayed,
- 5. Click OK to close the *Plot Properties* dialog,
- 6. Click OK to close the *Plot Manager* and return to the workspace.

f. Run Model (Solve > Analyze)

The next step is to analyze the SVSolid component of the model. Select *Solve > Analyze* from the menu. This action will write the descriptor file and open the FlexPDE solver. The solver will automatically begin solving the model. After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the FlexPDE solver. Right-click the mouse and select "Maximize" to enlarge any of the thumbnail plots. This section will give a brief analysis for each plot that was generated.

These reports are intended to provide the user with low-quality graphs which give a rough indication of the results. Creating professional-quality visualizations of the results can be accomplished with ACUMESH software.

g. Combine SVSlope with SVSolid (File > Add Coupling)

Modeling of SVSolid and SVSlope can be done independently or in "Combination" by specifying SVSolid and SVSlope components in the same model file. This methodology makes it easy to use the finite-element stress results when using the dynamic programming search method.

- 1. Select File > Add Coupling,
- 2. The Add Coupling dialog will be displayed,
- 3. Check the SVSlope box,
- Note that this process creates a new model file with the combined components in the same Project,
- 5. Click OK to close the dialog..

h. Specify Analysis Settings (Model > Settings)

In SVSlope the Analysis Settings provide the information for what model output will be available in ACUMESH. These settings are set by default when the combination was performed:

- 1. Select *Window > SVSlope* to switch to the SVSlope environment,
- 2. Select *Model* > *Settings* from the menu,
- 3. 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: Dynamic Programming

4. Select the *Calculation Methods* tab to see the method type as shown below is selected:

SAFE-DP

5. Press *OK* to close the dialog.

i. Specify Dynamic Programming Grid (Model > Slip Surface > Dynamic Programming > Grid Points)

The stress field generated by SVSolid is used as the initial conditions for SVSlope. In a combined SVSolid/SVSlope model the stress input file is already specified on the Initial Conditions dialog. The dynamic programming grid lines will be adjusted for this tutorial:

- 1. Select Model > Slip Surface > Dynamic Programming > Grid Points...,
- 2. Enter 40 for the *X* grid lines,
- 3. Enter 160 for the Y grid lines,
- 4. Press *OK* to close the dialog.

j. Specify Search Boundary Coordinates (Model > Search Boundary)

The Dynamic Programming method of searching for the critical slip surface has already been selected. Now the search boundary must be defined. This is accomplished through the following steps:

- 1. Select the Model > Search Boundary...,
- 2. The *Search Boundary* dialog will open with the default search boundary coordinates encompassing most of the model,
- 3. Refer to the list of search boundary coordinates at the beginning of this tutorial and enter the coordinates in the appropriate boxes,
- 4. Close the dialog by clicking *OK*.

The adjusted search boundary graphics are now be displayed on the CAD window.

k. Apply SVSlope Material Properties (Model > Materials)

Previously, the materials for the model were defined in SVSolid and assigned to regions. Now the SVSlope properties need to adjusted for those same materials. This section will provide instructions on adjusting the Upper Layer material. Repeat the process to adjust the other materials. Refer to the beginning of this tutorial for the list of material properties. Note that the Bedrock material type does not require entry of any values for SVSlope.

- Open the Materials dialog by selecting Model > Materials > Manager from the menu,
- 2. Select the Upper Layer material in the list and click the *Change Type* button,
- 3. On the Change Type dialog select "Mohr Coulomb" as the *New Data Type*,
- 4. Press *OK* to close the dialog. The *Material Properties* dialog will open automatically,
- 5. Move to the Shear Strength tab,
- The "Unit Weight" value of 15 kN/m³ will already be present as it was defined in SVSolid,
- 7. Enter the "Cohesion", c: value of 10 kPa,
- 8. Enter the "Friction Angle", phi value of 30 deg,
- 9. Click the OK button to close the Material Properties dialog,
- 10. Repeat these steps to adjust the Weak Layer and Lower Layer material using the information provided at the beginning of the tutorial,
- 11. Click the OK button to close the Material Manager dialog.

I. Run SVSlope Model (Solve > Analyze)

The next step is to analyze the model.

- Select Solve > Analyze from the menu. A pop-up dialog will appear and the solver will start,
- 2. Press the OK button to close the dialog.

m. Visualize Results (Window > AcuMesh)

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.

5.2 Results and Discussions

If the model has been appropriately entered into the software the following results should be shown for the SAFE method.



The correct results for this example are:

Mathad			
Method	SVSLOPE		
	Moment	Force	
SAFE		1.428	

This page is left blank intentionally.

6 Kulhawy Method

The dynamic programming example presented in Section 5 is extended to incorporate the Kulhawy stress-based slope stability analysis method.

This original model can be found under:

Project:	Slopes_STUDENT
Model:	

Minimum authorization required to complete this tutorial: PROFESSIONAL

Model Description and Geometry



Entry and Exit

Entry Range			Exit Range		
	Left Side			Right Side	
	Х	Y		Х	Y
Left Point	9	27	Left Point	40	17
Right Point	23	25.5	Right Point	55	15
Increments		12	Increments		12

6.1 Model Setup

The following steps will be required to set up this model:

- a. Create model
- b. Specify analysis settings
- c. Specify search method geometry

- d. Run SVSolid model
- e. Run SVSlope model
- f. Visualize results

a. Create Model

Since FULL authorization is required for this tutorial, 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,
- 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.

In order to create the Kulhawy model, save a copy of the dynammic programming example model created in Section 5. This is accomplished through the following steps:

- 1. Select the "UserTutorial" project and Open the "DP Example" model,
- 2. Select File > Save As,
- 3. Type the name User_Kulhawy and click *OK*.

Now a new model has been created and loaded into the workspace that will be modified to include Kulhawy analysis.

b. Specify Analysis Settings (Model > Settings)

In SVSlope the Analysis Settings provide the information for what type of analysis will be performed. These settings will be specified as follows:

- 1. Select *Model* > *Settings* from the menu,
- Select the Slip Surface tab, Slope Direction: Left to Right
 Slip Shape: Composite Circular
 Create tension crack for reverse curvature: On
 Search Method: Entry and Exit
- 3. Click *OK* to close the dialog asking you to remove SVSolid coupling.
- 4. Select the *Calculation Methods* tab from the dialog and select the method types as shown below:
 - Spencer Morgenstern-Price GLE (Fredlund) Kulhawy
- 5. Press *OK* to close the dialog.

c. Specify Search Method Geometry

The Entry and Exit 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 entry range and exit range on the screen. This is accomplished through the following steps:

- 1. Open the *Entry* and *Exit* dialog through the *Model* > *Slip Surface* > *Entry* and *Exit*... menu option,
- 2. Enter the values for the entry range and exit range as specified at the start of this tutorial (the range values may also be drawn on the CAD window),
- 3. Click *OK* to close the dialog.

d. Run SVSolid Model (Solve > Analyze)

The next step is to analyze the SVSolid component of the model. Select Window > SVSolid from the menu. Then Select Solve > Analyze from the menu. This action will write the descriptor file and open the FlexPDE solver for the SVSolid component of the model. The solver will automatically begin solving the model. After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the FlexPDE solver. Right-click the mouse and select "Maximize" to enlarge any of the thumbnail plots.

These reports are intended to provide the user with low-quality graphs which give a rough indication of the results. Creating professional-quality visualizations of the results can be accomplished with ACUMESH software.

e. Run SVSlope Model (Solve > Analyze)

The next step is to analyze the SVSlope component of the model.

- 1. Select Win*dow* > *SVSlope* from the SVOffice menu.
- Select Solve > Analyze from the menu. A pop-up dialog will appear and the solver will start,
- 3. Press the OK button to close the dialog.

f. Visualize Results (Window > AcuMesh)

After the model has been run, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click the *Acumesh...* button. 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.

6.2 Results and Discussions

If the model has been appropriately entered into the software the following results should be shown. 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 Slips, 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 user may display results from different methods by clicking the combo box on the display which lists the different analysis methods (Spencer, Morgentern-Price, etc.). The analysis results in a factor of safety of 1.393 for the Kulhawy method.



7 3D Multi Planar Example

The following example will introduce you to the three-dimensional SVSLOPE modeling environment. This example is used to investigate the use of a wedge slip surface method in determining the critical slip surface. A simple geometry is utilized in this example which is extruded from a 2D cross-section.

This original model can be found under:

Project:	Slopes_3D
Model:	Multi_Planar_Wedges

Minimum authorization required: STANDARD

Model Description and Geometry

A simple 120m by 180m area is created. A non-level plane is added to model the ground surface. A triangular pile is then added to the flat ground surface.



3D Planar Geometry Model

2D Region Data:

Region 1: R1

Х	Y
120	5
0	5
0	10
20	12
80	18
120	22

Region 2: R2

X	Y
120	22
80	18
20	12
80	58
120	30

2D Water Table Data:

Х	Y
0	10
20	12
80	18
120	22

Material Properties

Material	Shear Strength Type	Cohesion (kPa)	Friction Angle (deg)	Unit Weig ht (kN/m ³)	Water Surfaces	Ru Coefficie nt
Fill	Mohr Coulomb	0	35	18	Off	0.4
ClayFoundati on	Mohr Coulomb	50	20	20	On	-
Disc	Mohr Coulomb	0	12	0.001	On	-

Wedge Sliding Surface Data

	x	Y	z	Dip	Dip Direction	Discontinuity material
Wedge 1	0	90	10	7	0	Disc
Wedge 2	60	90	12	32	0	None
Wedge 3	0	90	-35	45	87	None
Wedge 4	0	90	-35	45	-87	None

46 of 62

7.1 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 following general categories:

- a. Create model
- b. Enter geometry
- c. Specify pore water pressure
- d. Apply material properties
- e. Extrude 2D model to 3D
- f. Specify analysis settings
- g. Specify search method geometry
- h. Run model
- i. Visualize results

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

a. Create Model

Since FULL authorization is required for this tutorial, 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, and
- 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,
- Select "ALL" under the Applications combo box and "ALL" for the Model Origin combo box,
- Create a new project called "UserTutorial" by pressing the New... button next to the list of projects,
- 4. Create a new model called "UserMultiWedge2D" by pressing the *New...* button next to the list of models. Note that initially the model is constructed as a 2D model to be later extruded to a 3D model. Use the settings below when creating this new model:

Application:	SVSLOPE
Model Name:	UserMultiWedge2D
System:	2D
Units:	Metric

Slope Direction: Right to Left

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

xmin = 0 ymin = 0

xmax = 120 ymax = 70

7. 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. Enter Geometry (Model > Geometry)

A region in SVSLOPE is the basic building block for a model. A region represents both a physical portion of material being modeled and a visualization area in the SVSLOPE CAD workspace. A region will have a set of geometric shapes that define its material boundaries. Also, other modeling objects including features, flux sections, water tables, text, and line art can be defined on any given region.

We start by specifying the 2D geometry that will be later extruded to a 3D geometry. The 2D geometry forms a cross-section of the final 3D geometry. Two regions will be utilized:

- Open the *Regions* dialog by selecting *Model* > *Geometry* > *Regions...* from the menu,
- 2. Click New to create the second region,
- 3. Click OK to close the dialog.

The shapes that define each region will now be created. Note that when drawing geometry shapes, the region that is current in the Region Selector is the region the geometry will be added to. The Region Selector is at the top of the workspace. Refer to the 3D Multi Planar Example Model Data section for the geometry points for each region.

• Define Region R1

- 1. Select *R1* from the Region Selector,
- 2. Select Draw > Model Geometry > Polygon Region from the menu,
- 3. The cursor will now be changed to a cross hair,
- 4. Move the cursor near (120,5) in the drawing space,
- 5. To select the point as part of the shape left click on the point,
- Now move the cursor near (0,5) and left-click the mouse. A line is now drawn from (120,5) to (0,5),
- 7. In the same manner then enter the following points:

(0,10) (20,12) (80,18)

8. Move the cursor near the point (120,22). Double click on the point to finish the shape. The shape is automatically finished by SVSLOPE by drawing a line from

(120,22) back to the start point (120,5).

Repeat this process to define the region R2.

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.

c. Specify Pore Water Pressure (Model > Pore Water Pressure)

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 water table will be used. In order to specify that a water table will be entered the must perform the following steps:

- 1. Select Model > Pore Water Pressure > Settings...,
- 2. Select "Water Surfaces" as the Pore-Water Pressure Method,
- Check the "Allow application of RU coefficients with water surfaces or discrete points" checkbox,
- 4. Press *OK* to close the dialog.

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

- 1. Select Model > Pore Water Pressure > Water Table...,
- Under the *Points* tab enter the *X* and *Y* coordinates as provided in 3D Planar Example Example Model Data,
- 3. Under the Apply to Regions section check the *R1* and *R2* boxes to apply the water table to both regions,
- 4. Press *OK* to close the dialog.

d. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the three materials that will be used in the model. The bottom region represents the foundation and will be assigned a clay material. The top region represents a pile placed above the foundation and will be assigned a fill material. An additional material is created to later define a wedge corresponding to a discontinuity. This section will provide instructions on creating the *Fill* material. Repeat the process to add the other two materials.

 Open the Materials Manager dialog by selecting Model > Materials > Manager... from the menu,

- 2. Click the *New...* button to create a material,
- 3. Enter "Fill" 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,
- 5. Move to the Shear Strength tab,
- 6. Enter the Unit Weight value of 18 kN/m^3,
- 7. Enter the Cohesion, c value of 0 kPa,
- 8. Enter the Friction Angle, phi value of 35 degrees,
- 9. Move to the Water Parameters tab,
- 10. Select "Off" for the Water Surfaces,
- 11. Enter a Ru Coefficient value of 0.4,
- 12. Click the OK button to close the Mohr Coulomb dialog,
- 13. Repeat these steps to create the *Clay Foundation* and *Disc* materials using the information provided at the beginning of the tutorial,
- 14. Press the OK button on the Materials Manager dialog to accept the changes and close the dialog.

Once all three 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 using the arrows at the top right of the dialog.
- 3. Under the "Region Settings" group select the Clay Foundation material from the combo box to assign this material to R1,
- 4. Select the *R2* region and assign the *Fill* material to this region,
- 5. Press the OK button to accept the changes and close the dialog.

e. Extrude 2D Model to 3D (File > Save As)

All of the previous steps may be transferred to a 3D version of this model. A new model is created with 3D geometry by extruding the 2D cross-section from the current model. This is accomplished through the following steps:

- 1. First, save the current model by clicking *File > Save* from the menu,
- 2. Next, to begin the extrusion process select *File > SaveAs...* from the menu,
- 3. Select the General tab,

System:

3D New File Name: User Multi Planar 3D

- 4. Select the *Spatial* tab,
- 5. Enter the following model extrusion parameters,

Model Width: 180 m

- 6. Press OK to close the dialog,
- 7. Press *OK* to accept the reset of some items.

The 3D geometry is now complete.

f. Specify Analysis Settings (Model > Settings)

The Analysis Settings provide the information for what type of analysis will be performed. These settings will be specified by the following steps:

- 1. Select *Model* > *Settings...* from the menu,
- 2. Select the 3D Slip Surface tab,
 - Search Method: Fully Specified Wedges
- 3. Select the *Calculation Methods* tab from the dialog and select the method types as shown below:
 - Bishop Simplified
 - Spencer
 - Morgenstern-Price
 - GLE (Fredlund)
- 4. For GLE method, press the *Lambda...* button,
- 5. Enter a Start Value of -0.5, an Interval of 0.25, and a Number of 8,
- 6. Press the Generate button,
- 7. Press *OK* to close the dialogs.

g. Specify Search Method Geometry (Model > Slip Surface)

This model makes use of a fully-specified search methodology. The wedge shape is used as the slip surface geometry. Four wedges will be specified through the following steps:

- Open the Wedges Sliding Surface dialog through the Model > Slip Surface > Fully Specified > Wedges... menu option,
- 2. Enter the data for the 4 wedges as specified in the start of this tutorial,
- 3. Click OK to close the dialog.

h. Run Model (Solve > Analyze)

The next step is to analyze the model.

4. Select Solve > Analyze from the menu. A pop-up dialog will appear and the solver will start.

i. Visualize Results (Window > AcuMesh)

After the model has completed solving, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click on the *Acumesh...* button. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH. 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.

7.2 Results and Discussions

After the model has completed solving the user may view the results in the ACUMESH software by pressing the ACUMESH 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. In order to view the sliding mass area more clearly the user may edit the *Sliding Mass Display Settings* dialog:

- 1. Select Slips > Sliding Mass Display ... from the menu,
- 2. Uncheck the *Sliding Mass with Explosion* checkbox.

The analysis results in a factor of safety of 1.172 for the Spencer method.



The user may also plot the column information for a particular column chosen either in plan view or from a vertical cross-section. The column information settings are set in the *Column Information* dialog. To access this dialog click the *Slips* > *Column Information...* menu item. Once the *Column Information* dialog is closed by clicking *OK*, the user may select a particular column by clicking on it in the CAD. The details of the selected column will appear in the CAD.



8 3D Grid and Tangent Submergence Example

The following example is used to illustrate the use of a grid and tangent search method in determining the critical slip surface of a submerged slope. The example is modeled using four regions, five surfaces, and four materials. A simple geometry is utilized in this example which is extruded from a 2D cross-section.

This original model can be found under:

Project:	Slopes_3D
Model:	Grid_Tangent_Toe_Submergence

Minimum authorization required: STANDARD

Model Description and Geometry

A simple 500ft by 200ft area is created. A non-level plane is added to model the underlying bedrock layer. The slope shape lying above bedrock is composed of 3 surfaces. The top surface intersects the water table at a height 150.07ft.



3D Grid and Submergence Tangent Model

2D Region Data:

Region 1: R1

Х	Y
500	0
0	0
0	107
193	107
196	103
219	103
225	107
500	107

Region 2: R2

X	Y
500	107
225	107
440	250
445	256
500	256

Region 3: R3

X	Y
440	250
225	107
219	103
196	103
193	107
0	107
0	112
100	112
189	112
425	250

Region 4: R4

Х	Y
445	256
440	250
425	250
189	112
100	112
420	256

2D Water Table Data:

X	Y
0	150.07
183.37	150.07
246.76	146.76
267.15	134.64
451.8	137.39

Material Properties

Materia I	Shear Strength Type	Cohesion (psf)	Friction Angle (deg)	Unit Weigh t (ft/lb ³)	Water Surfaces	Ru Coefficie nt
RockFill	Mohr Coulomb	0	35	70.6	Off	0.4
Core	Mohr Coulomb	100	29	70.6	On	-
Fill	Mohr Coulomb	0	28	70.6	On	_
R1	Mohr Coulomb	10000	35	100	On	-

Grid and Tangent

Grid - Points			Tangent - Po	oints	
	X	Y		X	Y
Upper Left	160.11	276.88	Upper Left	0	128.4
Lower Left	160.11	195.46	Lower Left	0	108.4
Lower Right	234.45	195.46	Lower Right	500	108.4
			Upper Right	500	128.4
X increments		5	Increments		2
Y increments		5			

Aspect Ratio: 1.0

8.1 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 following general categories:

- a. Create model
- b. Specify analysis settings
- c. Enter geometry
- d. Specify pore water pressure
- e. Apply material properties
- f. Specify search method geometry
- g. Extrude 2D model to 3D
- h. Run model

i. Visualize results

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

a. Create Model

Since FULL authorization is required for this tutorial, 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, and
- 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,
- Select "ALL" under the Applications combo box and "ALL" for the Model Origin combo box,
- Create a new project called "UserTutorial" by pressing the New... button next to the list of projects,
- 4. Create a new model called "User Submergence 2D" by pressing the New... button next to the list of models. Note that initially the model is constructed as a 2D model to be later extruded to a 3D model. Use the settings below when creating this new model:

Application:	SVSLOPE	
Model Name:	User Submergence 2D	
System:	2D	
Units:	Imperial	
Slope Direction: Right to Left		

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

xmin = 0 ymin = 0

xmax = 550 ymax = 300

7. 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. Change the default horizontal and vertical grid spacing to 1.0 ft and click *OK*.

b. Specify Analysis Settings (Model > Settings)

In SVSlope the Analysis Settings provide the information for what type of analysis will be performed. These settings will be specified as follows:

- 1. Select *Model* > *Settings* from the menu,
- 2. Select the Slip Surface tab,

Slope Direction:Right to LeftSlip Shape:CircularCreate tension crack for reverse curvature: OnSearch Method:Grid and Tangent

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

Bishop Simplified

Janbu Simplified

4. Press *OK* to close the dialog.

c. Enter Geometry (Model > Geometry)

A region in SVSLOPE is the basic building block for a model. A region represents both a physical portion of material being modeled and a visualization area in the SVSLOPE CAD workspace. A region will have a set of geometric shapes that define its material boundaries. Also, other modeling objects including features, flux sections, water tables, text, and line art can be defined on any given region.

We start by specifying the 2D geometry that will be later extruded to a 3D geometry. The 2D geometry forms a cross-section of the final 3D geometry. Four regions will be utilized:

- Open the *Regions* dialog by selecting *Model* > *Geometry* > *Regions...* from the menu,
- 2. Click New to create each of the four regions,
- 3. Click *OK* to close the dialog.

The shapes that define each region will now be created. Note that when drawing geometry shapes, the region that is current in the Region Selector is the region the geometry will be added to. The Region Selector is at the top of the workspace. Refer to the 3D Submergence Example Model Data section for the geometry points for the four regions.

Define Region R1

- 1. Select *R1* from the Region Selector,
- 2. Select Draw > Model Geometry > Polygon Region from the menu,
- 3. The cursor will now be changed to a cross hair,
- 4. Move the cursor near (500,50) in the drawing space,
- 5. To select the point as part of the shape left click on the point,
- Now move the cursor near (0,50) and left-click the mouse. A line is now drawn from (500,50) to (0,50),
- 7. In the same manner then enter the following points:

- (0,107) (193,107) (196,103) (219,103) (225,107)
- Move the cursor near the point (500,107). Double click on the point to finish the shape. A line is now drawn from (225,107) to (500,107). The shape is automatically finished by SVSLOPE by drawing a line from (500,107) back to the start point (500,50).

Repeat this process to define the regions R2, R3 and R4.

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.

d. Specify Pore Water Pressure (Model > Pore Water Pressure)

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 water table will be used. In order to specify that a water table will be entered the user must perform the following steps:

- 1. Select Model > Pore Water Pressure > Settings...,
- 2. Select "Water Surfaces" as the Pore-Water Pressure Method,
- Check the "Allow application of RU coefficients with water surfaces or discrete points" checkbox,
- 4. Press *OK* to close the dialog.

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

- 1. Select Model > Pore Water Pressure > Water Table...,
- Under the *Points* tab enter the *X* and *Y* coordinates as provided in 3D Submergence Example Model Data,
- 3. Under the Apply to Regions section ensure the *R1*, *R2*, *R3* and *R4* boxes are checked to apply the water table to all four regions,
- 4. Press *OK* to close the dialog.

e. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the four materials that will be used in the model. The *Extents* region cuts through all the surfaces in a model, creating a separate "block" on each layer. Each block can be assigned a material or be left as void. A void area is essentially air space. In this model all blocks will be assigned a

material. There are five surfaces resulting in four layers. Each layer will contain a different material. This section will provide instructions on creating the R1 material. Repeat the process to add the other three materials.

- Open the Materials dialog by selecting Model > Materials > Manager... from the menu,
- 2. Click the New... button to create a material,
- 3. Enter "R1" 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,
- 5. Move to the Shear Strength tab,
- 6. Enter the Unit Weight value of 100 lb/ft^3,
- 7. Enter the Cohesion, c value of 10000 psf,
- 8. Enter the Friction Angle, phi value of 35 degrees,
- 9. Move to the Water Parameters tab,
- 10. Select "On" for the Water Surfaces,
- 11. Click the OK button to close the Mohr Coulomb dialog,
- 12. Repeat these steps to create the *Fill, Core,* and *RockFill* materials using the information provided at the beginning of the tutorial,
- 13. Press the *OK* button on the *Materials Manager* dialog to accept the changes and close the dialog.

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

- Open the Region Properties dialog by selecting Model > Geometry > Region Properties... from the menu,
- 2. Select the *R1* region using the arrows at the top right of the dialog.
- 3. Under the "Region Settings" group select the *R1* material from the combo box to assign this material to region *R1*,
- 4. Select the R2 region and assign the Fill material to this region,
- 5. Select the R3 region and assign the Core material to this region,
- 6. Select the R4 region and assign the RockFill material to this region,
- 7. Press the *OK* button to accept the changes and close the dialog.

f. Specify Search Method Geometry (Model > Slip Surface)

The Grid and Tangent method of searching for the critical slip surface has already been selected in Step b. 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,
- Accept the warning message stating that the corner of the grid is below the top of the slope.

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

g. Extrude 2D Model to 3D (File > Save As)

All of the previous steps may be transferred to a 3D version of this model. A new model is created with 3D geometry by extruding the 2D cross-section from the current model. This is accomplished through the following steps:

- 1. First, save the current model by clicking File > Save from the menu,
- 2. Next, to begin the extrusion process select File > SaveAs... from the menu,
- 2. Select the General tab,

System: 3D

New File Name: User Submergence 3D

- 3. Select the *Spatial* tab,
- 4. Enter the following model extrusion parameters,

Model Width: 200 ft

Model Center: 100 ft

- 5. Press OK to close the dialog,
- 6. Press *OK* to accept the reset of some items.

The Y-coordinates for the search method geometry need to be updated in the 3D model:

- 9. Select Model > Slip Surface > Grid and Tangent...,
- 10. Enter the following model extrusion parameters,

Min Value: 0 Max Value: 0 No. of Points: 1

The water table surface also needs to be applied in the new 3D geometry:

1. Select Model > Initial Conditions > Settings...,

- 2. Move to the Apply tab,
- 3. Press the *Select All* button,
- 4. Press *OK* to close the dialog.

The 3D model is now complete and ready to be analyzed.

h. Run Model (Solve > Analyze)

The next step is to analyze the model.

1. Select *Solve* > *Analyze* from the menu. A pop-up dialog will appear and the solver will start.

i. Visualize Results (Window > AcuMesh)

After the model has completed solving, an ACUMESH notification will appear asking if you want to view the results in ACUMESH. Click on the *Acumesh...* button. The SVSLOPE screen will then change to reflect the results as visualized by ACUMESH. 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.

8.2 Results and Discussions

After the model has completed solving the user may view the results in the ACUMESH software by pressing the ACUMESH 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. In order to view the sliding mass area more clearly the user may edit the *Sliding Mass Display Settings* dialog:

- 1. Select Slips > Sliding Mass Display... from the menu,
- 2. Uncheck the Sliding Mass with Explosion checkbox.

The analysis results in a factor of safety of 1.032 for Bishop's method.



The user may also plot the column information for a particular column chosen either in plan view or from a vertical cross-section. The column information settings are set in the *Column Information* dialog. To access this dialog click the *Slips* > *Column Information...* menu item. Once the *Column Information* dialog is closed by clicking *OK*, the user may select a particular column by clicking on it in the CAD. The details of the selected column will appear in the CAD.

