SVDYNAMIC™

2D Slope Stability Modeling Software

Tutorial Manual

Written by:
Murray Fredlund, Ph.D.
Gilson Gitirana, Ph.D.
Robert Thode, B.Sc.G.E.

Edited by:
Murray Fredlund, Ph.D., PEng

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

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

1 SVDynamic Tutorial Manual

SVDYNAMIC™

2D Slope Stability Modeling Software

Tutorial Manual

Written by:
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 models.

Trademarks

Windows™ is a registered trademark of Microsoft Corporation.
SoilVision® is a registered trademark of SoilVision Systems Ltd.
SVOFFICE ™ is a trademark of SoilVision Systems, Ltd.
CHEMFLUX ™ is a trademark of SoilVision Systems Ltd.
SVFLUX ™ is a trademark of SoilVision Systems Ltd.
SVHEAT ™ is a trademark of SoilVision Systems Ltd.
SVAIRFLOW ™ is a trademark of SoilVision Systems Ltd.
SVSOLID ™ is a trademark of SoilVision Systems Ltd.
SVDYNAMIC ™ is a trademark of SoilVision Systems Ltd.
ACUMESH ™ is a trademark of SoilVision Systems Ltd.
FlexPDE® is a registered trademark of PDE Solutions Inc.

1.1 Introduction

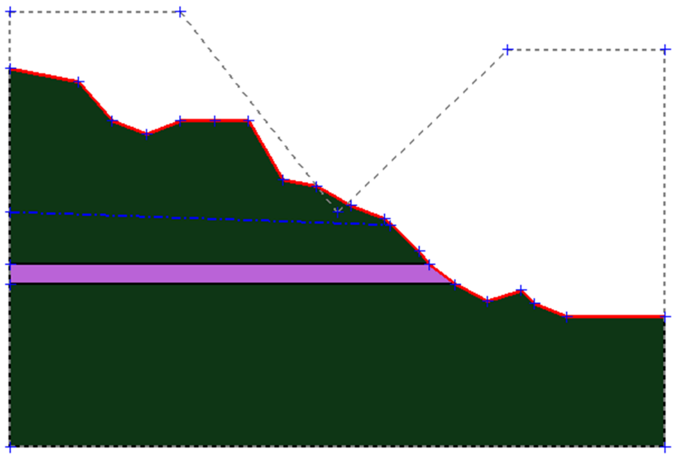
The Tutorial Manual serves a special role in guiding the first time users of the SVDYNAMIC software through a typical example problem. The example is "typical" in the sense that it is not too rigorous on one hand and not too simple on the other hand.

The Tutorial Manual serves as a guide by: i) assisting the user with the input of data necessary to solve the boundary value problem, ii) explaining the relevance of the solution from an engineering standpoint, and iii) assisting with the visualization of the computer output. An attempt has been made to ascertain and respond to questions most likely to be asked by first time users of SVDYNAMIC.

1.2 Gitirana (2005)

The following example will introduce the operation and features of SVDYNAMIC and will provide instructions on how to setup a model of a simple till slope with a weak clay seam. This model was set up and solved in the Ph.D. thesis of Gitirana (2005). A water table is present as well as a proposed roadbed to which a load is applied. The purpose of this model is to determine the slip surface for the present conditions and the resulting factor of safety for the slip surface. The model dimensions and material properties are provided below.

SVSOLID / SVDYNAMIC
Project: Slopes
Model: Tutorial_SVDYNAMIC



Region Geometry

The following geometry and material properties represent the building blocks for the tutorial model.

Ground **Seam** **WaterTable**

X	Y		X	Y		X	Y
0	0		0	25		0	36
100	0		68	25		58	34
100	20		64	28			

85	20		0	28			
80	22						
78	24						
72.8	22.4						
68	25						
64	28						
62.4	30						
57.2	35						
52	37						
46.8	40						
41.6	41						
36.4	50						
31.2	50						
26	50						
20.8	48						
15.6	50						
10.4	56						
0	58						

Material Properties

Material 1: Till

Young's Modulus, E = 10000 kPa
Poisson's Ratio, ν = 0.4
Initial Void Ratio, e_o = 0.3
Vertical Body Load, γ_y = -21 kN/m^3
Cohesion, c = 25 kPa
Friction Angle, ϕ = 30^o

Material 2: Weak Clay

Young's Modulus, E = 3000 kPa
Poisson's Ratio, ν = 0.35
Initial Void Ratio, e_o = 0.3
Vertical Body Load, γ_y = -18 kN/m^3
Cohesion, c = 0 kPa
Friction Angle, ϕ = 10^o

1.2.1 SVSOLID Model Setup

The first step in creation of the SVDYNAMIC model is the creation of the model in SVSOLID in order to determine the stress state. 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. Enter geometry
c. Specify initial conditions
d. Specify boundary conditions
e. Apply material properties
f. Specify model output
g. Run model
h. Visualize results

a. Create Model

The following steps are required to create the model:

1. Open the SVOFFICE Manager dialog.
2. Create a new project called *UserTutorial* by pressing the *New* button next to the list of projects. Select the *UserTutorial* project if it already exists.
3. Create a new model called *Tutorial_SVDynamicUser* 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.
4. Select *2D* for System,
5. Select *Steady-State* for Type,
6. Select *Metric* for Units,
7. Select *Consider PWP* as the Analysis option.
8. Click *OK* to close the dialog.

Before entering any model geometry it is best to set the World Coordinate System to ensure that the model will fit in the drawing space.

1. Access the *Format Axis* dialog.
2. Enter the *World Coordinates System* coordinates shown below into the dialog.

   x-minimum = -5
   y-minimum = -5
   x-maximum = 105
   y-maximum = 105

3. Click *OK* to close the dialog.

b. Enter Geometry

This model will be divided into three regions, which are named Slope, Clay Seam, and Water Table. The first 2 regions will have one of the materials just defined specified as its material properties. The third region will be used just to draw the water table. To add the necessary regions follow these steps:

1. Open the regions dialog by selecting *Model > Geometry > Regions* from the menu.
2. Change the first region name from Region 1 to Slope. Highlight the name and type new text.
3. Select the *Material Index* from the drop-down corresponding to the till.
4. Press the *New* button to add a second region.
5. Change the name of the second region to Clay Seam.
6. Select the clay material for the Clay Seam region.
7. Click *New* to add the third region.
8. Name it Water Table.
9. Click *OK* to close the dialog.

The shapes that define each material region will now be created. Note that when drawing geometry shapes the region that is current in the region selector is the region to which the geometry will be added. The Region Selector is at the top of the workspace.

- **Define the Ground**
  To draw the slope the instructions below explain the use of the Region Properties dialog to create the core shape.

1. Ensure that *"Slope"* is current in the region selector.
2. Select *Model > Geometry > Region Properties* from the menu.
3. The Shape 1 tab will open.
4. Type *0* into the *X* column and 0 into the *Y* column.
5. Type *100* into the *X* column and 0 into the *Y* column.
6. Refer to the geometry table at the beginning of this tutorial at enter the remaining points.

- **Define the Clay Seam**
  To draw the seam the instructions below explain the use of the mouse to create the core shape.

7. Ensure the *"Clay Seam"* region is current in the region selector.
8. Select *Draw > Model Geometry > Polygon Region* from the menu.
9. The cursor will now be changed to cross hairs.
10. Move the cursor near (0,25) in the drawing space. You can view the coordinates of the current position the mouse is at in the status bar.
11. When the cursor is near the point, right click. This will cause the cursor to snap to the point (The SNAP and GRID options in the status bar must both be bold).
12. To select the *point* as part of the shape left click on the *point*.
13. Now move the cursor near (68,25). Right click to snap the cursor to the exact point and then

left click on the *point*. A line is now drawn from (0,25) to (68,25).

14. Refer to the geometry table at the beginning of this tutorial and add the remaining points.
15. To add the last point, move the cursor near the point (0,28) and right click snapping the cursor to the point. Double-click on the *point* to finish the shape. A line is now drawn from (64,28) to (0,28) and the shape is automatically finished by SVSOLID by drawing a line from (0,28) back to the start point, (0,25).

NOTE:

Select a shape with the mouse and select *Edit > Delete* from the menu if a mistake was made entering the coordinate points for a shape. This will remove the entire shape from the region. To edit the shape use the Region Properties dialog.

NOTE:

At times it may be tricky to snap to a grid point that is near a line defined for a region. Turn the object snap off by clicking on *"OSNAP"* in the status bar to alleviate this problem.

c. Specify Initial Conditions

Options must be selected here in order to specify a water table as the initial pore-water pressure conditions.

1. Select *Model > Initial Conditions > Settings* from the menu. Select the *Initial Pore Water Pressure* tab.
2. Select *Draw Water Table* as the Initial PWP Option.
3. Click *OK* to close the dialog.

A water table will be drawn across the model to indicate the initial pore-water pressure conditions. The pore-water pressure equals 0 at the water table elevation and is hydrostatic in the remainder of the model.

1. Select *Draw > Initial Water Table* from the menu.
2. The Water Table dialog will open. Click the *Draw* button.
3. With the mouse click on the *point (0,36)*.
4. To finish the Water Table double-click on the *point (58,34)*. A dashed-dot line is drawn across the model.
5. Click the *OK* button to close the dialog. The water table should now be displayed in the workspace.

d. Specify Boundary Conditions

Now that all of the model geometry and initial conditions have been successfully defined, the next step is to specify the boundary conditions. A load expression will be defined on the roadway location on the slope region with the sides being fixed in the *X*-direction. At the base the region will be fixed in both the *X* and *Y* directions. The Clay Seam region is internal to the Slope region and will not need to be altered. The steps for specifying the boundary conditions are thus:

1. Select the *"Slope"* region in the drawing space.
2. From the menu select *Model* > *Boundaries* > *Boundary Conditions*. The boundary conditions dialog will open.
3. Select the *point (0,0)* from the list on the Segment tab.
4. From the *X* Boundary Condition drop-down select a *Fixed* boundary condition.
5. From the *Y* Boundary Condition drop-down select a *Fixed* boundary condition.
6. Select the *point (100,0)* from the list.
7. Repeat Steps 4 through 6 for the remaining boundary conditions shown in the chart below.
8. Click the *OK* button to close the dialog.

NOTE:

The Free *Y* boundary condition for the point (100,0) becomes the boundary condition for the following line segments that have a Continue boundary condition until a new boundary condition is specified. By specifying a Load Expression condition at point (36.4,50) the Continue is turned off.

Boundary Condition Summary

X	Y	X Boundary Condition	Y Boundary Condition
0	0	Fixed	Fixed
100	0	Continue	Free
100	20	Free	Continue
85	20	Continue	Continue
80	22	Continue	Continue
78	24	Continue	Continue
72.8	22.4	Continue	Continue
68	25	Continue	Continue
64	28	Continue	Continue
62.4	30	Continue	Continue
57.2	35	Continue	Continue
52	37	Continue	Continue
46.8	40	Continue	Continue
41.6	41	Continue	Continue
36.4	50	Continue	Load Expression = -1000
31.2	50	Continue	Continue
26	50	Continue	Free
20.8	48	Continue	Continue
15.6	50	Continue	Continue
10.4	56	Continue	Continue
0	58	Fixed	Continue

e. Apply Material Properties

The next step in defining the model is to enter the material properties for the 2 materials that will be used in the model. A till is defined for the slope and the seam will consist of a clay material. This section will provide instructions on creating the clay material. Repeat the process to add the other material. For the majority of the slope stability analysis only a linear elastic material model is required.

1. Open the Materials dialog by selecting *Model > Material > Manager* from the menu.
2. Click the *New Material* button to create a material.
3. Select the new material and click *Properties* to open the Material Properties dialog.
4. Enter the information above into the appropriate fields on the Description tab.
5. Move to the Parameters tab.
6. Refer to the data provided at the beginning of this tutorial. Enter the Young's Modulus value of 3000 kPa.
7. Enter the Poisson's Ratio value of 0.35.
8. Enter the Initial Void Ratio value of 0.3.
9. Move to the Body Load tab.

10. Enter the *X*-Axis Body Load as 0 kN/m^3.
11. Enter the *Y*-Axis Body Load as –18 kN/m^3.
12. Move to the Shear Strength tab.
13. Enter the *Cohesion as* 0 kPa.
14. Enter the *Friction Angle as 10^o.*
15. Repeat these steps to create the till material.

NOTE:
The negative sign for the body load indicates that the vertical body load will act down.

Once the materials have been defined, they must be assigned to regions. Follow the instructions below to assign the materials to the regions.

1. Open the Regions dialog by selecting Geometry > Regions from the menu.
2. Click on the Material drop-down for the Slope region and select the Till material.
3. Click on the Material drop-down for the Clay Seam region and select the Weak Clay material.
4. Click the OK button to close the dialog.

f. Specify Model Output

Two levels of output may be specified: i) output (graphs, contour plots, fluxes, etc.) which are displayed during model solution, and ii) output which is written to a standard finite element file for viewing with AcuMesh software. Output is specified in the following two dialogs in the software:

i) Plot Manager:	Output displayed during model solution.
ii) Output Manager:	Standard finite element files written out for visualization in AcuMesh or for initial condition input to other finite element packages.

PLOTMANAGER

There are many plot types that can be specified to visualize the results of the model. A vertical stress contour plot will be generated for this example.

1. Open the Plot Manager dialog by selecting *Model > Plot Manager* from the menu.
2. The toolbar at the bottom left corner of the dialog contains a button for each plot type. Click on the *Contour* button to begin adding the first contour plot. The Plot Properties dialog will open.
3. Enter the title *Vertical Stress*.
4. Select s_y as the variable to plot from the drop-down.
5. Select the *Output Options* tab and select the *PLOT* output option.
6. Click *OK* to close the dialog and add the plot to the list.
7. Click *OK* to close the Plot Manager and return to the workspace.

Alternatively, the user may press the *Add Default Plots* button and typical plots will be added to the plot list.

OUTPUTFILES

A pore-water pressure table file is required for the SVDYNAMIC analysis, as well as, a stress table file containing various stress variables.

1. Open the Output Manager dialog by selecting *Model* > *Reporting* > *Output Manager* from the menu.
2. The toolbar at the bottom left corner of the dialog contains a button for each output file type. Click on the *Table* button to begin adding the output file. The Output File Properties dialog will open.
3. Enter the title pwp.
4. Select all the *variable* u_w in the variable list.
5. Press the *Single Right Arrow* button.
6. Click *OK* to close the dialog and add the output file to the list.
7. Click on the *SVDynamic* button to begin adding another output file. The Output File Properties dialog will open and a default title, variables, and grid lines will be set.
8. Click *OK* to close the dialog and add the output file to the list.
9. Click *OK* to close the Output Manager and return to the workspace.

g. Run Model

Once the solver has completed analyzing the model it will generate the stress state .TBL that is called *Tutorial_SVDynamic_SVDynamicData.tbl*. The default format for this file is *ModelName_SVDynamicData.tbl*. The specified pwp.tbl file will also be created.

h. Visualize Results

The stress conditions for the current model may be examined by selecting the *View* > *ACUMESH* menu option.

1.2.2 SVDYNAMIC Model Setup

Additional slope stability information must be entered in order to analyze the model using the SVDYNAMIC solver. SVDYNAMIC information may be entered by switching to the SVDYNAMIC software through the process toolbar SVDYNAMIC button.

1.2.2.1 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. Specify ground surface
b. Specify search boundary

c. Run model

d. Visualize results

a. Specify Ground Surface

The Ground Surface is a necessary component of the SVDYNAMIC analysis and should correspond to the top of the model geometry. The following list contains the ground surface points:

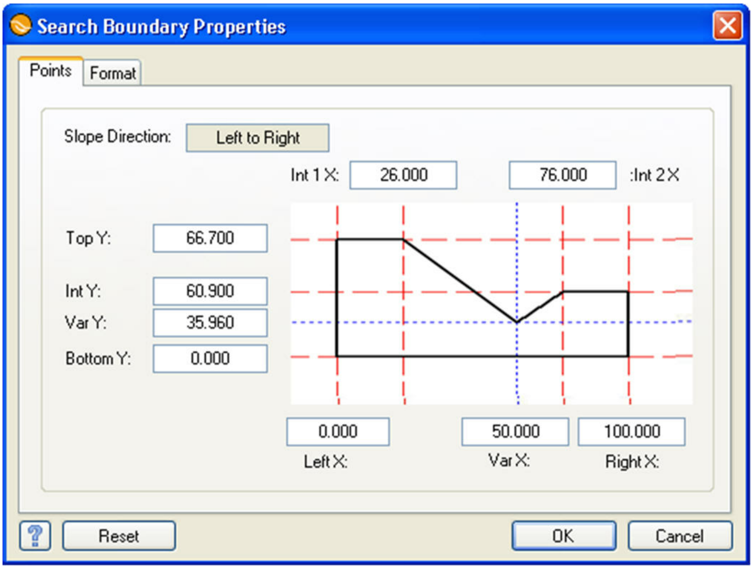
X	Y
100	20
85	20
80	22
78	24
72.8	22.4
68	25
64	28
62.4	30
57.2	35
52	37
46.8	40
41.6	41
36.4	50
31.2	50
26	50
20.8	48
15.6	50
10.4	56
0	58

1. Select *Model* > *Ground Surface* from the menu.
2. The *Ground Surface* dialog is displayed without any data.
3. Enter the above coordinate points for the ground surface.
4. Press the *OK* key to complete the ground surface.

b. Specify Search Boundary

The search boundary defines the area of the model that will be examined to determine a potential slip surface. When the input data is loaded a default search boundary is generated to roughly fit the model grid.

1. Open the search boundary dialog by selecting *Model* > *Initial Conditions* > *Search Boundary* from the menu.



2. Enter the search boundary coordinates as shown in the above screenshot.
3. Press the *Snap to Model Grid* button. The search boundary coordinates must fall on model gridlines.
4. Click *OK* to close the dialog.

NOTE:
See the SVDYNAMIC User's Manual for all the requirements of the search boundary.

c. Run Model

The model setup is now complete. From the menu select *Solve* > *Analyze* or press the *Analyze* button in the toolbar. SVDYNAMIC will begin solving the model and will open the *Analysis Message Window*.

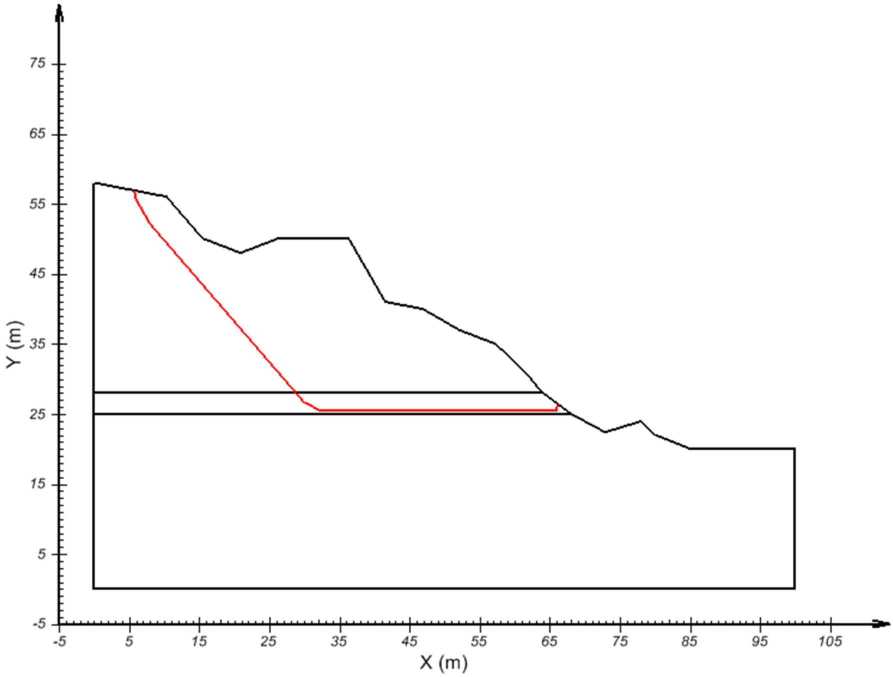
d. Visualize Results

Once the analysis is completed, the SVDYNAMIC software will display a factor of safety in an alert box. The critical slip surface can be examined in the ACUMESH visualization software and is represented by a red line. The user may switch to the Acumesh software by selecting the Acumesh icon on the process toolbar.

1.2.2.2 Results & Discussion

After the SVDYNAMIC solver has analyzed a model it will display the critical factor of safety and analysis time in the *Message Window*. The resulting *Slip Surface* will also be drawn in the workspace and the factor of safety for the slip surface will be shown in the upper right portion of the workspace.

The factor of safety for the slip surface of this slope under the defined conditions is 1.085.



Once an analysis has been completed and the slip surface generated distance graphs can be plotted. A distance graph is an input or output variable vs. distance along the slip surface. Select *Model > Plots* to open the *Distance Graph* dialog.

1.3 References

Fredlund, D.G., and Gitrana Jr, G.F.N. 2003. Analysis Of Transient Embankment Stability Using The Dynamic Programming Method. Proceedings of the 56th Canadian Geotechnical Conference, Winnipeg, Canada. Sept.29 to Oct 1, Vol. 1, pp. 807 to 814.

Fredlund, D.G. and Rahardjo, H. 1993. Soil Mechanics for Unsaturated Soils. John Wiley & Sons, New York

Pham, Ha T. V. 2002. Slope Stability Analysis Using Dynamic Programming Method With A Finite Element Stress Analysis. University of Saskatchewan, Canada