



SVDYNAMIC™

2D Slope Stability Modeling Software

Tutorial Manual

ED-1B

Date: November 24, 2004

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

Information contained within this Tutorial Manual is copyrighted and all rights are reserved by SoilVision Systems Ltd. The SVDYNAMIC 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

SVDYNAMIC™ is a trademark of SoilVision Systems Ltd.

SVSOLID™ is a trademark of SoilVision Systems Ltd.

SVFLUX™ is a trademark of SoilVision Systems Ltd.

Copyright © 2004
by
SoilVision Systems Ltd.
Saskatoon, Saskatchewan, Canada
ALL RIGHTS RESERVED
Printed in Canada

1	TUTORIAL EXAMPLE PROBLEM.....	4
2	SVSOLID PROBLEM SETUP.....	6
2.1	ADDING A PROJECT.....	6
2.2	ADDING A PROBLEM.....	7
2.3	OPENING THE PROBLEM.....	10
2.4	DEFINING THE PROBLEM.....	10
2.4.1	Specify Settings.....	10
2.4.2	Setting the Workspace.....	11
2.4.3	Define Material Properties.....	13
2.4.4	Adding Regions.....	16
2.4.5	Defining Region Geometry Shapes.....	16
2.4.6	Specify Boundary Conditions.....	18
2.5	SPECIFYING THE WATER TABLE.....	20
2.6	SPECIFY PLOTS.....	21
2.7	SPECIFY OUTPUT FILES.....	21
2.8	ANALYZE.....	23
3	SVDYNAMIC PROBLEM SETUP.....	24
3.1	ADDING A PROJECT AND PROBLEM.....	24
3.2	DEFINING THE PROBLEM.....	24
3.2.1	Importing Geometry from SVSOLID.....	24
3.2.2	Load Input Data Files.....	26
3.3	ADDING THE GROUND SURFACE.....	27
3.4	SEARCH BOUNDARY SPECIFICATION.....	28
3.5	ANALYZE THE SLOPE.....	29
3.6	RESULTS.....	29
4	REFERENCES.....	30
5	INDEX.....	31

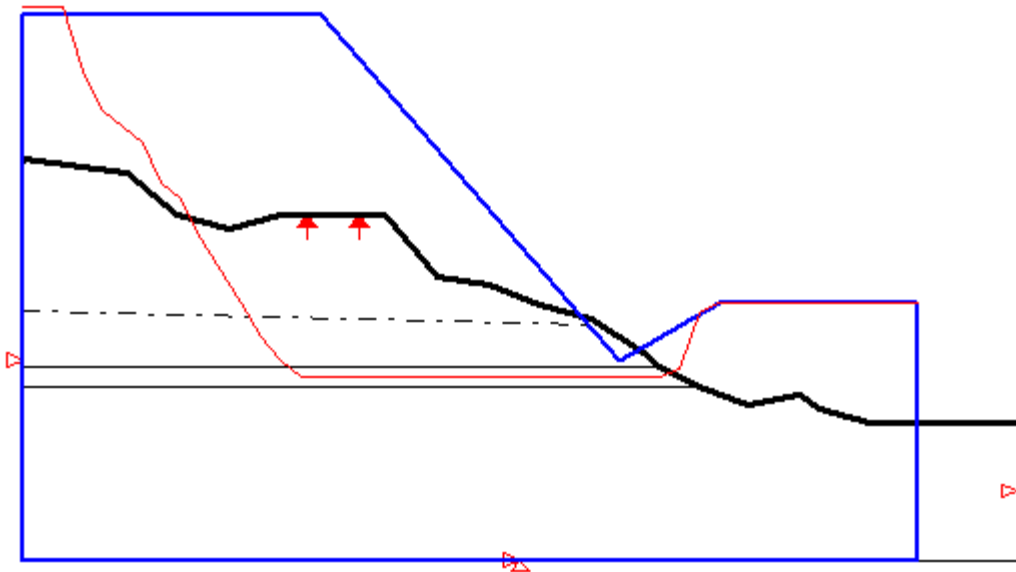
1 TUTORIAL EXAMPLE PROBLEM

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. A water table is present as well as a proposed roadbed to which a load is applied. The purpose of this problem is to determine the slip surface for the present conditions and the resulting factor of safety for the slip surface. The problem dimensions and soil properties are provided below.

SVDYNAMIC -
SVSOLID -

ProjectID: Tutorial ProblemID: Tutorial1
ProjectID: Tutorial ProblemID: Tutorial_SVDynamic

Proposed Roadway on Slope With Weak Seam



Region Geometry

Ground

X	Y
0	0
100	0
100	20
85	20
80	22

Seam

X	Y
0	25
68	25
64	28
0	28

Water Table

X	Y
0	36
58	34

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

Soil Properties

Soil 1: Till

Young's Modulus, $E = 10000$ kPa
Poisson's Ratio, $\nu = 0.4$
Initial Void Ratio, $e_o = 0.3$
Vertical Body Load, $\gamma_y = -21$ kN/m³
Cohesion, $c = 25$ kPa
Friction Angle, $\phi = 30^\circ$

Soil 2: Weak Clay

Young's Modulus, $E = 3000$ kPa
Poisson's Ratio, $\nu = 0.35$
Initial Void Ratio, $e_o = 0.3$
Vertical Body Load, $\gamma_y = -18$ kN/m³
Cohesion, $c = 0$ kPa
Friction Angle, $\phi = 10^\circ$

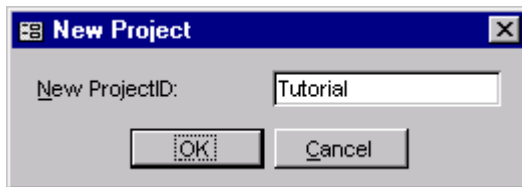
2 SVSOLID PROBLEM SETUP

2.1 ADDING A PROJECT

The first step in defining a problem is to decide the project under which the problem is going to be organized. If the project is not yet included you must add the project before proceeding with the problem. In this case, the problem is placed under a project called Tutorial.

In order to add the project follow these steps:

1. Select **Model > Projects/Problems...** from the menu to open the Projects / Problems form.
2. Click **New Project...** in the lower left of the form.
3. The Project Properties form is opened along with a prompt asking for a new ProjectID.



4. Type "Tutorial" as the new ProjectID and press OK.

The Project Properties form is where you information specific to each project is stored. This will include the Project ID, Project Name, Location, Start Date, End Date, Project Notes, client information, contractor and project engineer information.



Tip!

The Project ID is the only required information needed to define a project. The rest of the fields are optional.

The form is opened ready to accept information.

Project Properties

Problems In Project:

Project ID: Tutorial

Project Name: SVSolid Tutorial Manual Problems

Project Location: Saskatoon, SK Canada

Project Start Date: Jan 01/2002

Project End Date: Aug 30/2003

Project Notes: The tutorial example problems are stored under this project.

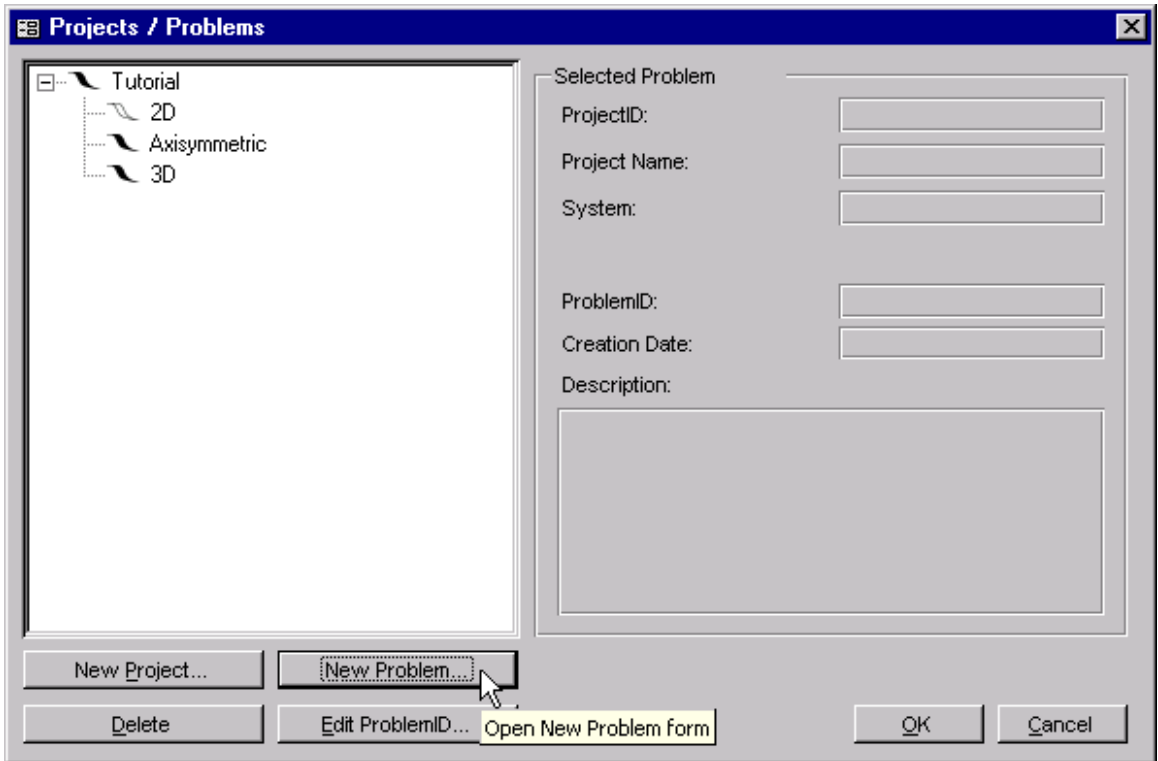
Edit ProjectID... OK

It should be noted that once the project is defined it would be identified by the ProjectID throughout the rest of the program. Also, SVSOLID does not allow you to specify two projects with the same ProjectID.

5. Fill out the form with the desired **information**.
6. To exit this form and return to Projects / Problems click **OK**. The project information is automatically saved upon entry.

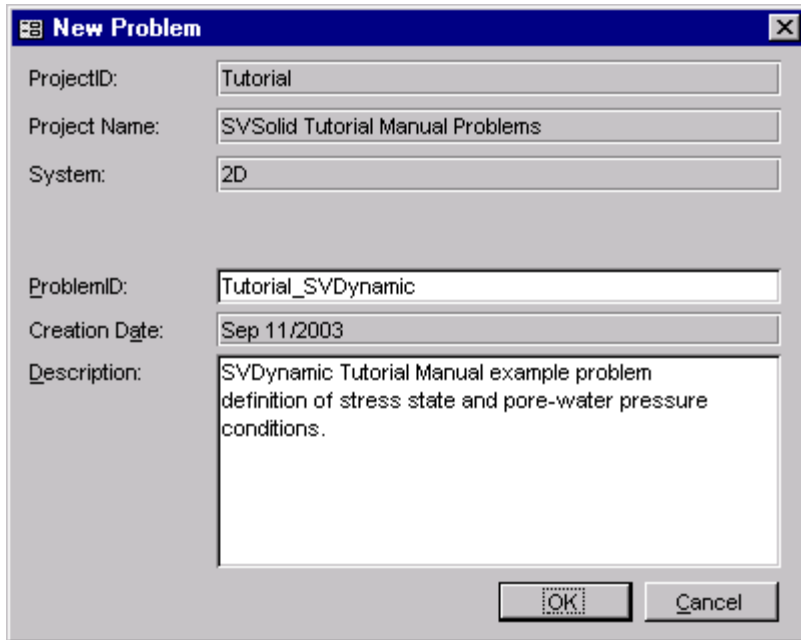
2.2 ADDING A PROBLEM

Once a project has been created any number of problems may be stored in it.



When the Projects/Problems form is opened there will be a list of the projects that have been defined. In this case there is only the Tutorial project. To add a problem:

1. Click on the plus sign and expand the project **Tutorial**.
2. This example will be modeled in two-dimensions so we must expand **2D** by clicking the plus sign beside "2D"
3. Click the **New Problem** button. The New Problem form will open.



New Problem

ProjectID: Tutorial

Project Name: SVSolid Tutorial Manual Problems

System: 2D

ProblemID: Tutorial_SVDynamic

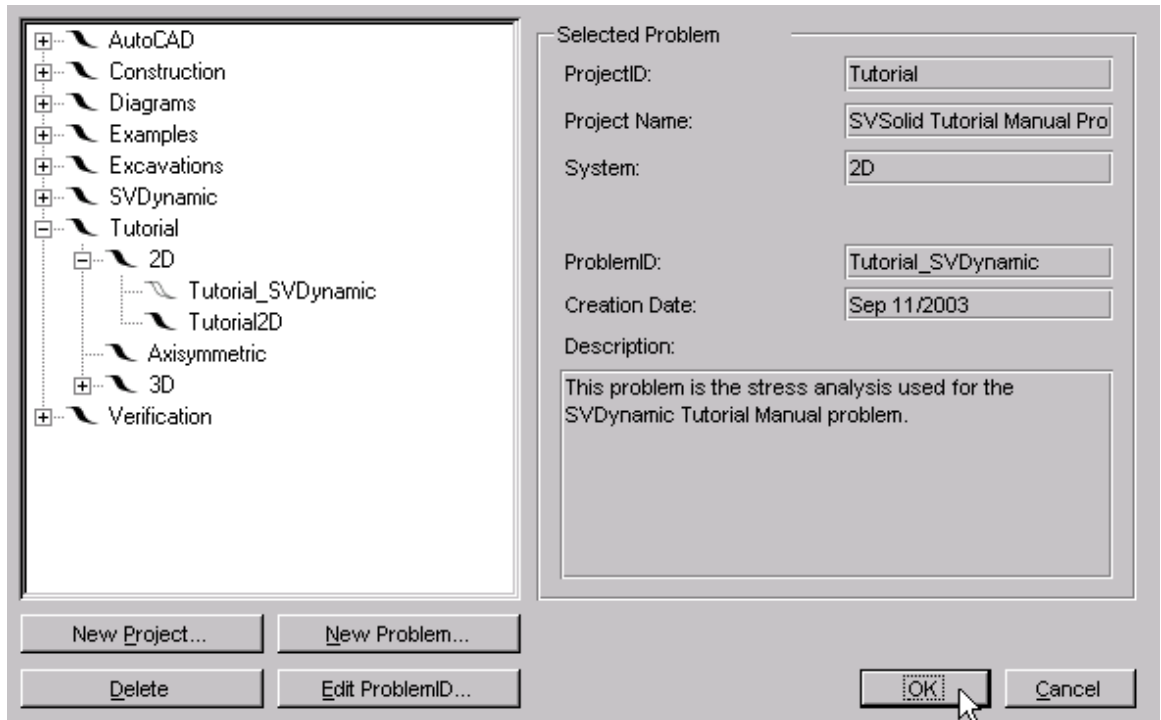
Creation Date: Sep 11/2003

Description: SVDynamic Tutorial Manual example problem definition of stress state and pore-water pressure conditions.

OK Cancel

4. Enter the **ProblemID: Tutorial_SVDynamic**. The description is optional.
5. Click the **OK** button to save the problem and close the New Problem form.

2.3 OPENING THE PROBLEM



To open the problem:

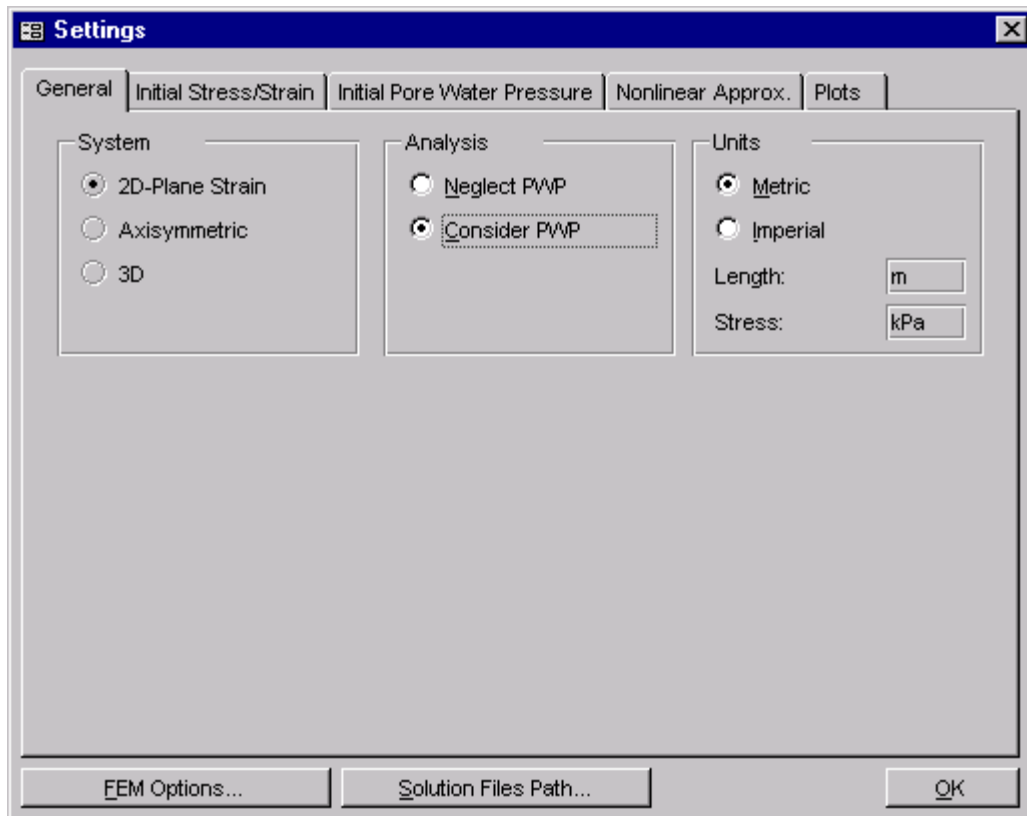
1. **Navigate** back to the problem via the Tutorial - 2D item.
2. Select **Tutorial_SVDynamic**.
3. The problem may be opened by clicking the **OK** button or by double clicking on the ProblemID in the tree. The problem will now be opened in the workspace.

2.4 DEFINING THE PROBLEM

The following section provides instructions on how to begin defining the problem in the workspace.

2.4.1 Specify Settings

The first step in defining the problem is to specify the settings that will be used for the problem. To open the Settings form select **Model > Settings** in the workspace menu.



The Settings form will contain information about the current problem System, Analysis, Units, Initial Conditions Settings, and more. The data for the problem is in metric so the units will remain metric. Options must be selected here in order to specify a water table as the initial pore-water pressure conditions. Drawing of the water table is explained later in this tutorial.

1. Select **Consider PWP** as the Analysis option.
2. Move to the **Initial Pore Water Pressure** tab.
3. Select **Draw Water Table** as the Initial PWP Option.
4. Click **OK** to close the form.

2.4.2 Setting the Workspace

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

1. Access the World Coordinate System form by either of two ways. Click on the button:



or select **View > WCS**. The button is located in the view toolbar to the left of the

drawing space.

World Coordinate System

World Coordinate System

Bottom Left

X: Y:

Upper Right

X: Y:

CAD Window

Bottom Left

X: Y:

Upper Right

X: Y:

Region Geometry Extents

Minimum

X: Y:

Maximum

X: Y:

Surface Grid Extents

Minimum

X: Y:

Maximum

X: Y:

Global Coordinate Offset

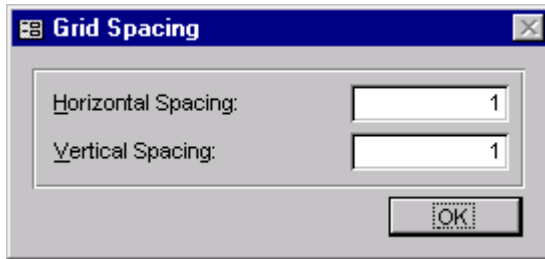
X: Y:

*All coordinate-related data will be adjusted by this offset on an import or paste operation. Adjusting this offset once data is entered will result in a problem-wide adjustment.

2. Enter the **world coordinate system** coordinates as shown above.
3. Also set the **CAD Window** (drawing space) coordinates.
4. Click **OK** to close the form.

The workspace grid spacing needs to be set to aid in defining region shapes.

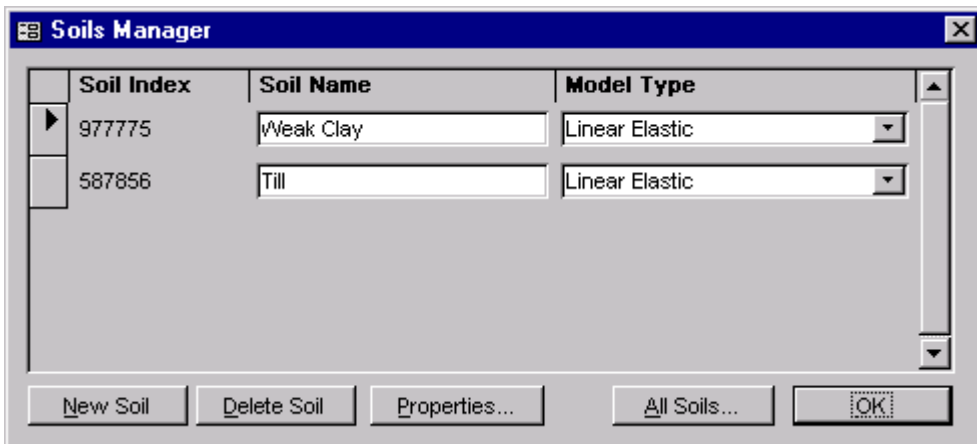
5. Select **View > Grid Spacing** from the menu.



The Grid Spacing dialog box has a title bar with a close button. It contains two input fields: 'Horizontal Spacing' and 'Vertical Spacing', both with the value '1' entered. An 'OK' button is located at the bottom right.

6. Enter 1 for both the horizontal and vertical spacing.
7. Click **OK** to close the form.


2.4.3 Define Material Properties



The Soils Manager dialog box features a table with three columns: Soil Index, Soil Name, and Model Type. It includes buttons for 'New Soil', 'Delete Soil', 'Properties...', 'All Soils...', and 'OK'.

Soil Index	Soil Name	Model Type
977775	Weak Clay	Linear Elastic
587856	Till	Linear Elastic

The next step in defining the problem is to enter the material properties for the 2 soils that will be used in the model. A till is defined for the slope and the seam will consist of a clay soil. This section will provide instructions on creating the clay soil. Repeat the process to add the other soil.

1. Open the Soils form by selecting **Model > Soils** from the menu or click the soils button,  in the Tools toolbar.
2. Click the **New Soil** button to create a soil in the database. A unique Soil Index is generated that is used to reference the soil in other SVSOLID forms.
3. Select the new soil and click Properties to open the **Soil Properties** form.

Soil Properties: Linear Elastic

Soil Index: 977775

Description | PWP | Parameters | Body Load | SVDynamic | Reference

Soil Name: Weak Clay

USCS Texture: Silty clay

Soil Description:

Geologic Description:

Notes: Tutorial weak clay.

OK

4. Enter the information above into the appropriate fields on the **Description** tab
5. Move to the **Parameters** tab.

Soil Properties: Linear Elastic

Soil Index: 977775

Description PWP Parameters Body Load SVDynamic Reference

Elastic Parameters

Young's Modulus Option

Young's Modulus: 3,000 kPa

Poisson's Ratio: 0.350

Bulk Modulus Option

Bulk Modulus: 3,333 kPa

Shear Modulus: 1,111 kPa

Initial Void Ratio, eo: 0.30

OK

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/m³.
11. Enter the **Y-Axis Body Load** as -18 kN/m³.
12. Move to the **SVDynamic** tab.
13. Enter the **Cohesion** as 0 kPa.
14. Enter the **Friction Angle** as 10 °.
15. **Repeat** these steps to create the till soil.




Tip!

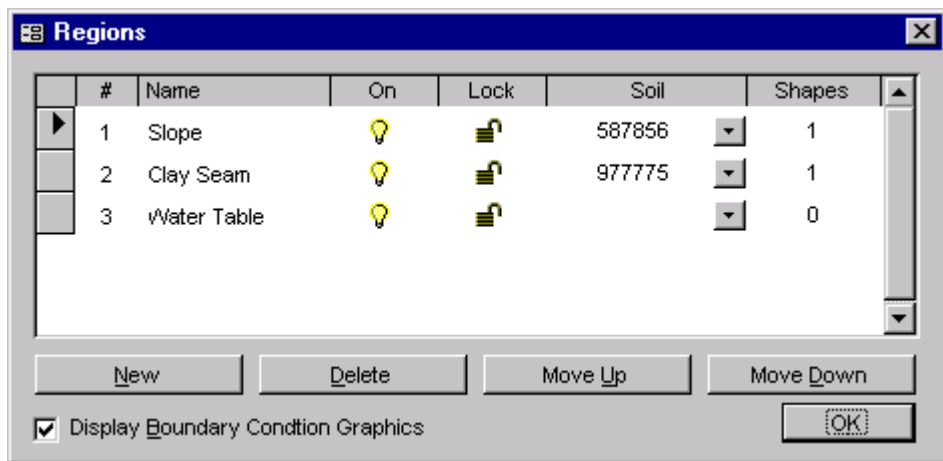
The negative sign for the body load indicates that the vertical body load will act down.

2.4.4 Adding Regions

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

This problem will be divided into three regions, which are named Slope, Clay Seam, and Water Table. The first 2 regions will have one of the soils just defined specified as its soil 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 form by clicking the **Regions** button,  at the top of the workspace.




2. Change the first region name from Region 1 to **Slope**. Highlight the name and type new text.
3. Select the Soil 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** soil for the Clay Seam region.
7. Click **New** to add the third region.
8. Name it **Water Table**.
9. Select the box **Display Boundary Condition Graphics**. This will display graphical representations for the boundary conditions when they are defined later in this tutorial.
10. Click **OK** to close the form.

2.4.5 Defining Region Geometry Shapes

The shapes that define each soil 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.


- **Define the Ground**

To draw the slope the instructions below explain the use of the command line to create the core shape.

1. Ensure that “**Slope**” is current in the region selector.
2. Click on the **Draw Polygon Region Shape** button, .
3. The command line will be set to **Start Point** and the cursor focus will be in the command line.
4. Type 0,0 and press the **Enter** key on the keyboard.
5. Type **100,0** and press Enter.
6. Refer to the **geometry table** at the beginning of this tutorial at enter the remaining points.
7. Type **f** and press Enter to complete the region shape.


- **Define the Clay Seam**

In the instructions for drawing the slope shape the command line was used. To draw the seam the instructions below explain the use of the mouse to create the core shape.

1. Ensure the “**Clay Seam**” region is current in the region selector.
2. Click on the **Draw Polygon Region Shape** button, .
3. The cursor will now be changed to cross hairs.
4. 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 just above the command line.
5. 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).
6. To select the point as part of the shape **left click** on the point.
7. 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).
8. Refer to the **geometry table** at the beginning of this tutorial and add the remaining points.
9. 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).



Tip!

Select a shape with the mouse and press the Delete button,  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 form.



Tip!

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.

2.4.6 Specify Boundary Conditions

Now that all of the regions and the problem geometry 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**. The boundary conditions form will open.

Boundary Conditions

Region: Slope Select Shape Index: 355397754

Segment Point

	Point		Boundary Condition		
	X	Y	Direction	X	Y
	41.6	41	Condition: Continue	Continue	Continue
			Expression:		
▶	36.4	50	Condition: Continue	Load Expression	
			Expression:	-1000	
	31.2	50	Condition: Continue	Continue	Continue
			Expression:		
	26	50	Condition: Continue	Free	
			Expression:		
	20.8	48	Condition: Continue	Continue	Continue
			Expression:		
	15.6	50	Condition: Continue	Continue	Continue
			Expression:		

Segment Length: 5.2 m

Note: boundary conditions defined at a point remain in effect until re-defined at a subsequent point.

OK

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. From the **X Boundary Condition** drop down select a **Continue** boundary condition.
8. From the **Y Boundary Condition** drop down select a **Free** boundary condition.
9. Select the **point** (100,20) from the list.
10. From the **X Boundary Condition** drop down select a **Free** boundary condition.
11. Select the **point** (36.4,50) from the list.
12. From the **Y Boundary Condition** drop down select a **Load Expression** boundary condition. This will cause the Load Expression box to be enabled.
13. In the Expression box enter a **load of -1000**.

14. Select the **point** (26,50) from the list.
15. From the **Y Boundary Condition** drop down select a **Free** boundary condition.
16. For the last point, select Fixed as the X Boundary Condition.
17. Click the **OK** button to close the form.

**T i p !**


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

2.5 SPECIFYING THE WATER TABLE

A water table will be drawn across the problem 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 problem.

1. Click on the Draw Initial Water Table button, .
2. The Water Table form 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 problem.



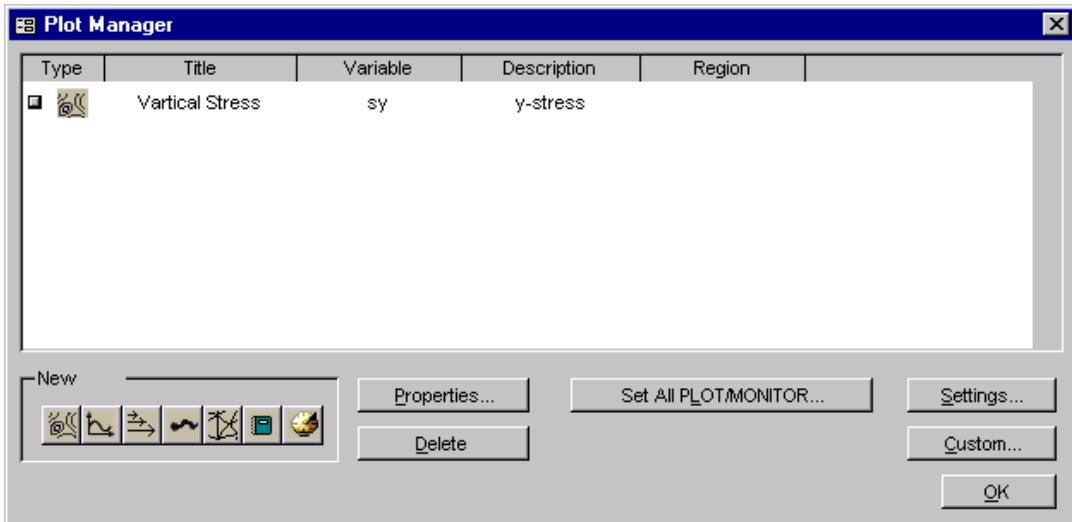
Tip!

The water table points could also be entered in the command line.

2.6 SPECIFY PLOTS

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.

- Open the Plot Manager form by selecting **Model > Plot Manager** from the menu.



- The toolbar at the bottom left of the form contains a button for each plot type. Click on the Contour button to begin adding the first contour plot. The Plot Properties form will open.



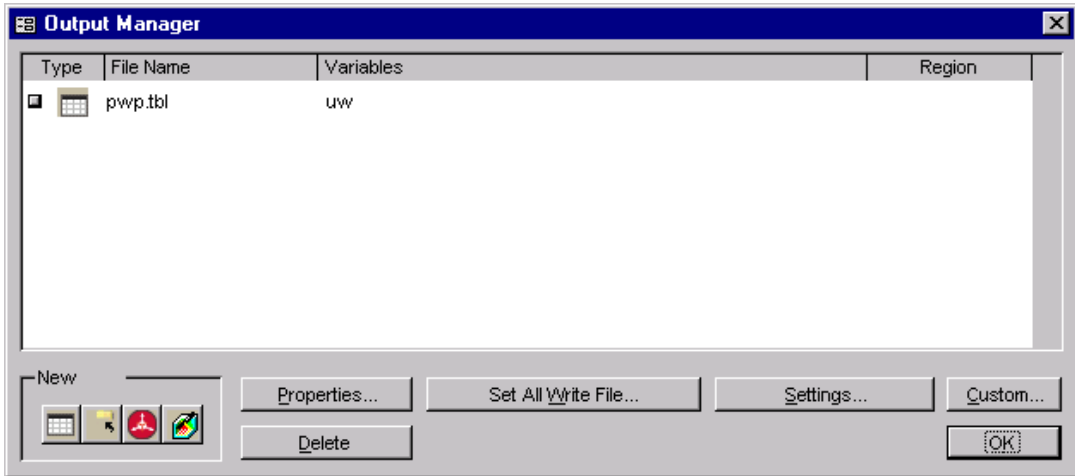
- Enter the title Vertical Stress.
- Select sy as the variable to plot from the drop-down.
- Select the **PLOT** output option.
- Click **OK** to close the form and add the plot to the list.
- Click **OK** to close the Plot Manager and return to the workspace.

2.7 SPECIFY OUTPUT FILES

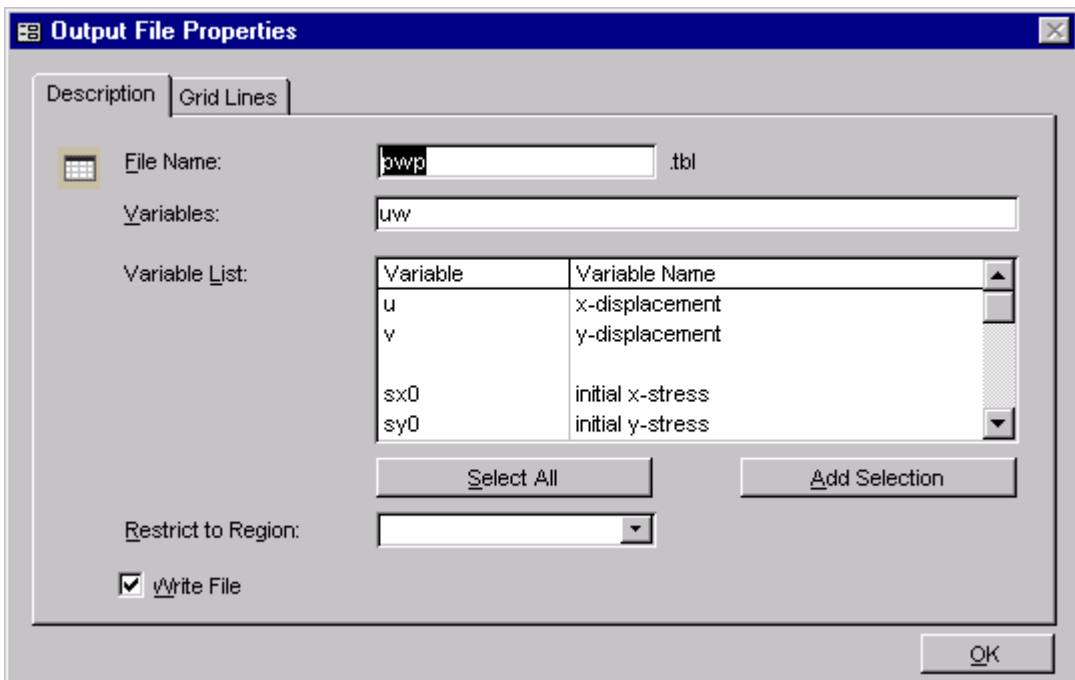
There are 4 output file types that can be specified to export the results of the model. A pore-water pressure table file is required for the SVDYNAMIC analysis. A stress table file will be generated automatically as long as the required soil properties are provided. The file will be called:

Tutorial_Tutorial_SVDynamic_SVDynamicData.tbl.

1. Open the Output Manager form by selecting **Model > Output Manager** from the menu.



2. The toolbar at the bottom left of the form contains a button for each output file type. Click on the Table button to begin adding the output file. The Output File Properties form will open.



3. Enter the title **pwp**.
4. Select all the variable **uw** in the variable list.
5. Press the **Add Selection** button.
6. Check the **Write File** box.
7. Click **OK** to close the form and add the output file to the list.
8. Click **OK** to close the Output Manager and return to the workspace.

2.8 ANALYZE

The next step is to analyze the problem. Click the Analyze button located in the toolbar. This action will write the descriptor file and open the SVSOLID solver. The solver will automatically begin solving the problem.

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

3 SVDYNAMIC PROBLEM SETUP

After the model has been created and solved in SVSOLID, the problem data can be imported into SVDYNAMIC and the slope stability setup can be done.

3.1 ADDING A PROJECT AND PROBLEM

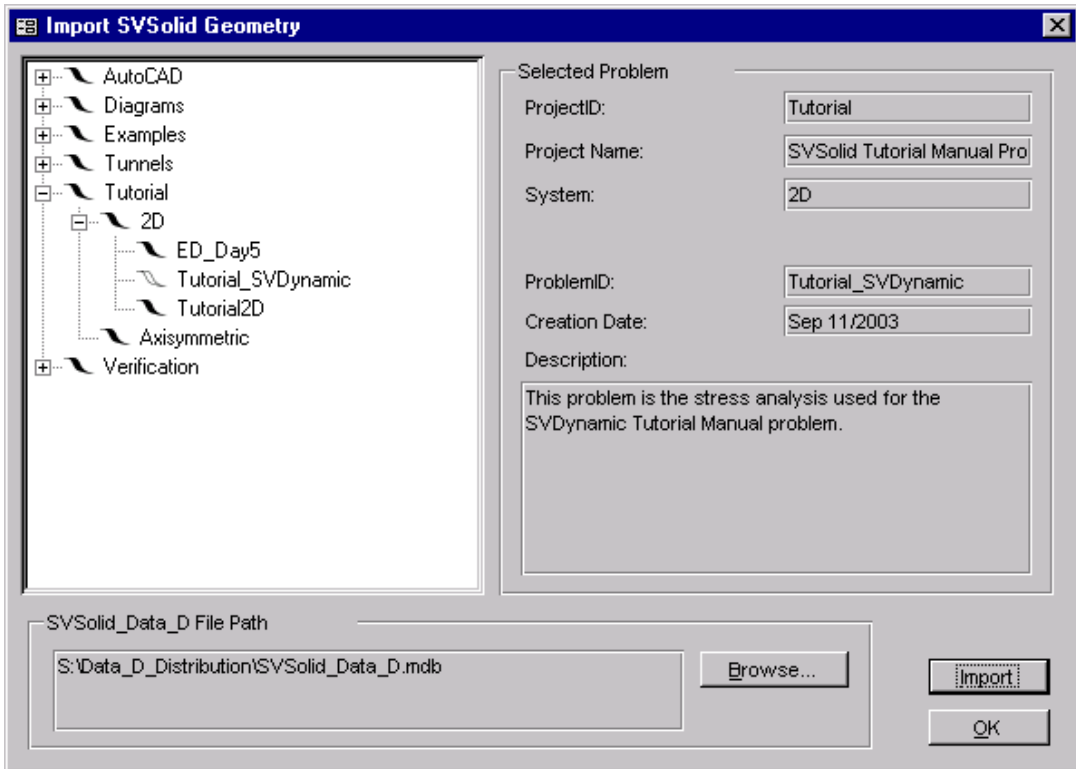
Projects and problems in SVDYNAMIC are created the same way as in SVSOLID. Refer to the SVSOLID portion of this Tutorial Manual and create the ProjectID: **Tutorial** and the ProblemID: **Tutorial1**.

3.2 DEFINING THE PROBLEM

The following section provides instructions on how to set up the problem in SVDYNAMIC.

3.2.1 Importing Geometry from SVSOLID

Geometry for the SVDYNAMIC problem is imported from a SVSOLID database of the correct format. The geometry must be imported before any other modeling operations are available. The import includes regions, region shapes, world coordinate system settings, and water tables. The units for the SVDYNAMIC model are set to the SVSOLID model units during import.



To import the geometry from the SVSOLID problem previously created:

1. Select **Model > Load SVSolid Geometry** to open the above Import SVSolid Geometry form.
2. Click **Browse**.
3. Specify the path to the SVSolid_Data_D.mdb file containing the problem that the geometry will be imported from.
4. Select the **ProblemID: Tutorial_SVDynamic** from the tree.
5. Press **Import**.

3.2.2 Load Input Data Files

Specify a Stress Data .tbl file:

The stress data table file generated by SVSolid has the filename format:
ProjectID_ProblemID_SVDynamicData.tbl.

File Path

S:\Data_D\My_Problems\Tutorial\2D\Tutorial_SVDynamic\Tutorial_Tutorial_SVDynamic_SVDynamicData.tbl

Browse...
Clear

Specify a Pore Water Pressure .tbl file:

File Path

S:\Data_D\My_Problems\Tutorial\2D\Tutorial_SVDynamic\pwp.tbl

Browse...
Clear

Load Data OK

1. Select **Model > Load Input Data Files** to open the following form.
2. Click **Browse** to specify the path to the **Tutorial_Tutorial_SVDynamic_SVDynamicData.tbl** file corresponding to the SVSOLID geometry that was imported. The file includes stress in the X direction, s_x , stress in the Y direction, s_y , shear stress, s_{xy} , cohesion, c , and the friction angle, ϕ .
3. Click **Browse** to specify the path to the **pwp.tbl** file that was generated by SVSOLID
4. Press the **Load Data** button to load the data from the specified table files into the model. Once the data is loaded correctly the problem grid will be displayed in the workspace along with the default search boundary.
5. Click **OK** to close the form.

**Tip!**

A pore-water pressure file can be generated in SVSOLID or SVFLUX.

3.3 ADDING THE GROUND SURFACE


The Ground Surface is a necessary component of the SVDYNAMIC analysis and should correspond to the top of the problem 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

The screenshot shows the 'Ground Surface' dialog box. It features a table with the following data:


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

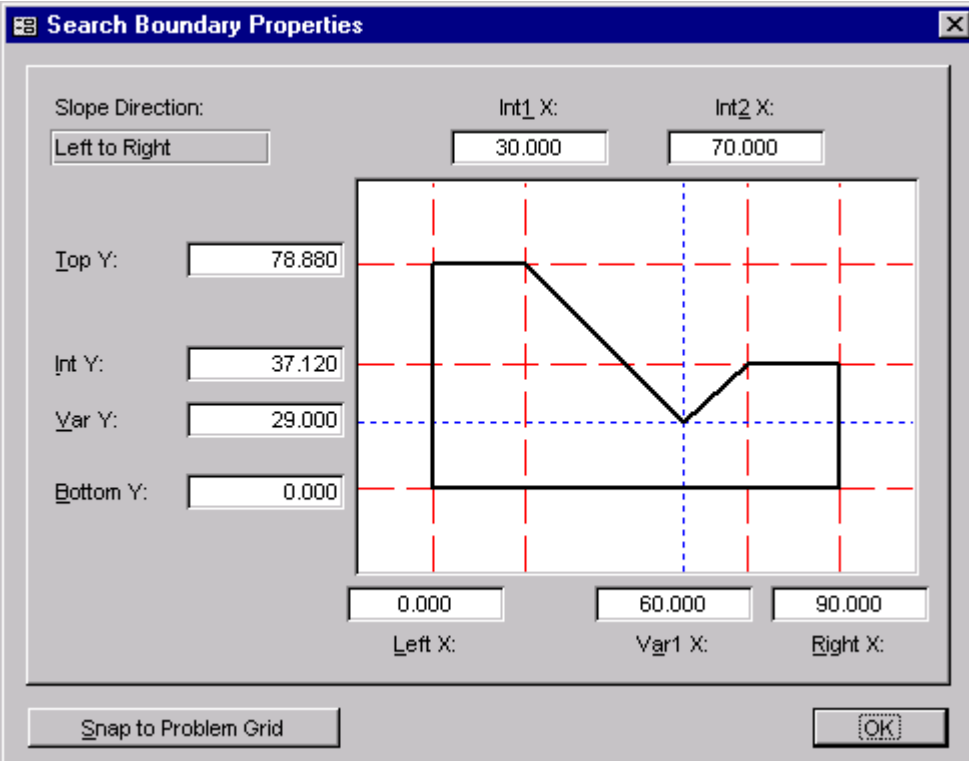
The dialog also includes buttons for 'Draw', 'Delete', 'Paste Data...', 'Insert Point', and 'OK'. A note in the 'Edit' section states: 'Delete Points: Select Points, Press DEL key'.

1. Select **Model > Ground Surface** from the menu or  in the tool bar.
2. The above form is displayed without any data.
3. Click **Draw** and the command line will be available.
4. Enter the first coordinate point for the ground surface.
5. Press the Enter key.
6. Enter the remaining points in the same manner.
7. When the desired points have been entered, type “f”
8. Press the Enter key to complete the ground surface.
9. Open the Ground Surface form again to confirm that the points have been entered correctly. If any points are wrong type the correct data directly into the form.

3.4 SEARCH BOUNDARY SPECIFICATION

The search boundary defines the area of the problem 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 problem grid.

1. Open the search boundary form by selecting **Model > Search Boundary** from the menu, by pressing this button , or by double-clicking the search boundary object in the workspace.



Search Boundary Properties

Slope Direction:

Int1 X: Int2 X:

Top Y:

Int Y:

Var Y:

Bottom Y:

Left X: Var1 X: Right X:

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

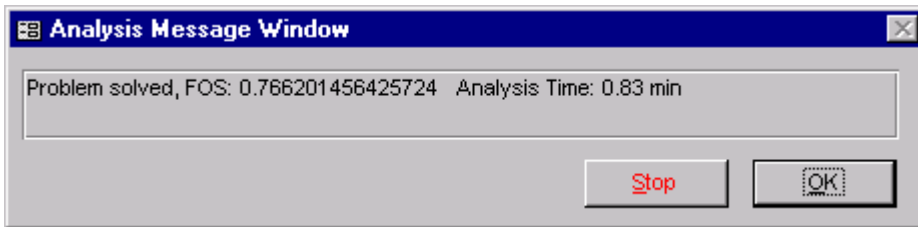


See the SVDYNAMIC User's Manual for all the requirements of the search boundary.

Tip!

3.5 ANALYZE THE SLOPE

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




3.6 RESULTS

After the SVDYNAMIC solver has analyzed a problem it will display the final 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 **0.77**.

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. Press the button:  or select **Model > Plots** to open the Distance Graph form.

4 REFERENCES

- D. G. Fredlund and H. Rahardjo, 1993. Soil Mechanics for Unsaturated Soils. John Wiley & Sons, New York
- Ha T. V. Pham, 2002. Slope Stability Analysis Using Dynamic Programming Method With A Finite Element Stress Analysis. University of Saskatchewan, Canada
- 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.

5 INDEX

Analysis, 29, 30
Body Load, 5, 15
Boundaries, 18
Boundary Conditions, 18
Cohesion, 5, 15
Distance Graph, 29
Fill, 7
Geometry, 4, 16, 24, 25
Grid, 12, 28
Ground Surface, 27
Output, 21, 22, 23
Output Files, 21
Plots, 21, 29
Problem, 4, 6, 7, 8, 9, 10, 24, 28
ProblemID, 4, 9, 10, 23, 24, 25
Problems, 6, 7, 8
Project, 6, 24
ProjectID, 4, 6, 7, 23, 24
Projects, 6, 7, 8, 24
Properties, 5, 6, 13, 17, 21, 22
PWP, 11
Region, 4, 16, 17
Regions, 16
Search Boundary, 28
Selector, 16
Shape, 17
Shapes, 16
Slip Surface, 29
Soil, 5, 13, 16, 30
Soils, 13, 30
Stress, 21, 30
Surface, 27, 29
View, 11, 12
Water Table, 4, 11, 16, 20
Workspace, 11

This page is intentionally left blank.