

SVSOLIDTM

2D / 3D Stress Deformation Modeling Software

Tutorial Manual

ED-4C

Date: February 4, 2005

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 SVSOLID 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

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.

ACUMESH[™] is a trademark of SoilVision Systems Ltd.

FlexPDE[®] is a registered trademark of PDE Solutions Inc.

Copyright © 2005
by
SoilVision Systems Ltd.
Saskatoon, Saskatchewan, Canada
ALL RIGHTS RESERVED
Printed in Canada

1	A Two Dimensional Example Problem.....	7
1.1	Adding a Project.....	8
1.2	Adding a Problem	10
1.3	Opening the Problem	11
1.4	Defining the Problem.....	12
1.4.1	Specify Settings.....	12
1.4.2	Setting the Workspace	13
1.4.3	Define Material Properties.....	15
1.4.4	Adding Regions	17
1.4.5	Defining Region Geometry Shapes.....	18
1.4.6	Specify Boundary Conditions.....	20
1.5	Specifying the Water Table.....	22
1.6	Specify Plots.....	23
1.7	Specify Output Files.....	24
1.8	Analyze.....	25
1.9	Results.....	25
1.9.1	Solution Mesh.....	26
1.9.2	Vertical Stress Contours.....	27
1.9.3	Displacement Vectors.....	28
1.9.4	ACUMESH Mohr Circle Diagram	28
2	A Three Dimensional Example Problem	30
2.1	Adding a Project.....	31
2.2	Adding a Problem	33
2.3	Opening the Problem	34
2.4	Defining the Problem.....	35
2.4.1	Setting the Workspace	35
2.4.2	Define Material Properties.....	37
2.4.3	Define 3D Surfaces	39
2.4.4	Adding Regions	43
2.4.5	Defining Region Geometry Shapes.....	44
2.4.6	Specifying a Soil by Region and Layer	46
2.4.7	Specify Boundary Conditions.....	47
2.5	Specify Plots.....	49
2.6	Specify Output Files.....	51
2.7	Analyze.....	52
2.8	Results.....	52
2.8.1	Solution Mesh.....	52
2.8.2	Stress Contours	53

2.8.3	Displacement Vectors.....	54
3	2D: Edge Drop of a Flexible Impervious Cover.....	55
3.1	Problem Overview	55
3.2	Initial SVFLUX Seepage Analysis Setup	56
3.2.1	Adding an SVFLUX Project.....	56
3.2.2	Adding the Initial SVFLUX Problem.....	58
3.2.3	Opening the Initial SVFLUX Problem.....	59
3.3	Defining the Initial SVFLUX Problem.....	60
3.3.1	Setting the Workspace	60
3.3.2	Define Material Properties.....	62
3.3.3	Adding a Region.....	63
3.3.4	Defining Region Geometry Shapes.....	64
3.3.5	Specify Boundary Conditions.....	65
3.4	Specify Initial SVFLUX Analysis Plots.....	67
3.5	Specify Initial SVFLUX Analysis Output Files	68
3.6	Analyze the Initial SVFLUX Problem	69
3.7	Initial SVFLUX Analysis Results.....	69
3.8	SVFLUX Transient Seepage Analysis Setup.....	71
3.8.1	Adding the SVFLUX Transient Problem.....	71
3.8.2	Opening the Transient SVFLUX Problem.....	72
3.9	Defining the Transient SVFLUX Problem.....	72
3.9.1	Specify Settings.....	72
3.9.2	Define Material Properties.....	73
3.9.3	Importing Geometry.....	76
3.9.4	Assign the Region Soil.....	76
3.9.5	Specify Boundary Conditions.....	77
3.10	Specify SVFLUX Transient Analysis Plots.....	79
3.11	Specify SVFLUX Transient Analysis Output Files.....	81
3.12	Analyze the Transient SVFLUX Problem.....	82
3.13	SVFLUX Transient Analysis Results.....	82
3.13.1	Pressure Contours.....	83
3.13.2	Pore-water Pressure Elevation Profiles.....	84
3.14	SVSOLID Stress/Deformation Analysis	86
3.14.1	Adding an SVSOLID Project.....	86
3.14.2	Adding an SVSOLID Problem.....	88
3.14.3	Opening the SVSOLID Problem.....	89
3.15	Defining the SVSOLID Problem.....	89
3.15.1	Specify Settings.....	89

3.15.2	Setting the Stages.....	92
3.15.3	Define Material Properties.....	92
3.15.4	Importing Geometry From SVFLUX.....	94
3.15.5	Assign the Region Soil.....	94
3.15.6	Specify Boundary Conditions.....	94
3.16	Specify SVSOLID Plots.....	96
3.17	Analyze the SVSOLID Problem.....	97
3.18	SVSOLID Results.....	97
3.18.1	Solution Mesh.....	98
3.18.2	Vertical Stress Contours.....	99
3.18.3	Suction Contours.....	100
3.18.4	Displacement Vectors.....	101
3.19	SVSOLID Summary File Results.....	101
3.19.1	Vertical Displacement Contours.....	102
3.19.2	Vertical Displacement at Ground Surface.....	103
3.19.3	Vertical Displacement Below Cover Edge.....	104
3.20	Problem Conclusion.....	104
4	3D: Edge Drop of a Flexible Impervious Cover.....	105
4.1	Problem Overview.....	105
4.2	Initial SVFLUX Seepage Analysis Setup.....	106
4.2.1	Adding an SVFLUX Project.....	106
4.3	Adding the Initial SVFLUX Problem.....	108
4.3.1	Opening the Initial SVFLUX Problem.....	109
4.4	Defining the Initial SVFLUX Problem.....	109
4.4.1	Setting the Workspace.....	109
4.4.2	Define Material Properties.....	111
4.4.3	Define 3D Surfaces.....	112
4.4.4	Adding Regions.....	114
4.4.5	Defining Region Geometry Shapes.....	115
4.4.6	Specifying a Soil by Region and Layer.....	116
4.4.7	Specify Boundary Conditions.....	117
4.5	Specify Initial SVFLUX Analysis Plots.....	120
4.6	Specify Initial SVFLUX Analysis Output Files.....	121
4.7	Analyze the Initial SVFLUX Problem.....	122
4.8	Initial SVFLUX Analysis Results.....	122
4.9	SVFLUX Transient Seepage Analysis Setup.....	124
4.9.1	Adding the SVFLUX Transient Problem.....	124
4.9.2	Opening the Transient SVFLUX Problem.....	124

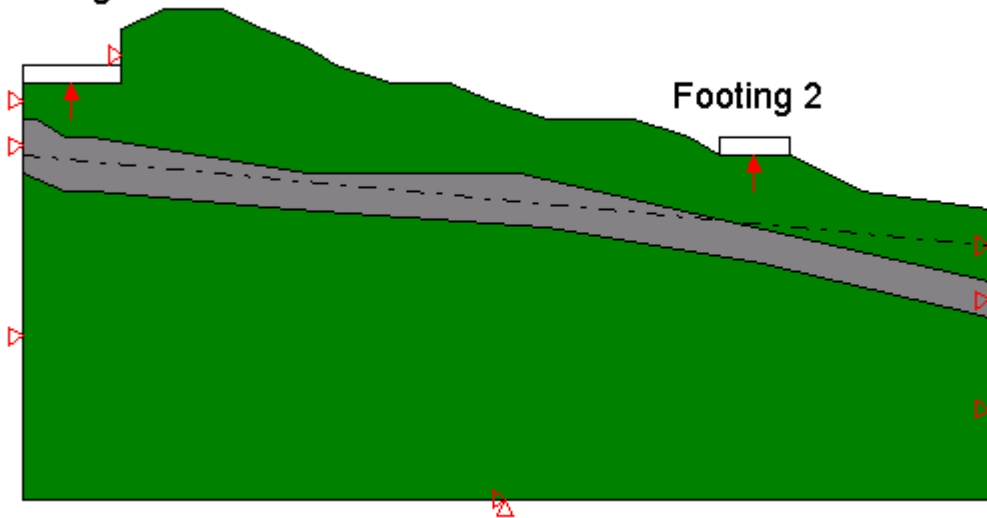
4.10	Defining the Transient SVFLUX Problem	125
4.10.1	Specify Settings	125
4.10.2	Define Material Properties	126
4.10.3	Importing Geometry	129
4.10.4	Specifying a Soil by Region and Layer	129
4.10.5	Specify Boundary Conditions	130
4.11	Specify SVFLUX Transient Analysis Plots	133
4.12	Specify SVFLUX Transient Analysis Output Files	136
4.13	Analyze the Transient SVFLUX Problem	137
4.14	SVFLUX Transient Analysis Results	137
4.14.1	Pressure Contours	138
4.14.2	Pore-water Pressure Elevation Profiles	139
4.15	SVSOLID Stress/Deformation Analysis	141
4.15.1	Adding an SVSOLID Project	141
4.15.2	Adding an SVSOLID Problem	143
4.15.3	Opening the SVSOLID Problem	144
4.16	Defining the SVSOLID Problem	144
4.16.1	Specify Settings	144
4.16.2	Setting the Stages and Error Limit	147
4.16.3	Define Material Properties	147
4.16.4	Importing Geometry From SVFLUX	149
4.16.5	Specifying a Soil by Region and Layer	149
4.16.6	Specify Boundary Conditions	150
4.17	Specify SVSOLID Plots	151
4.18	Specify SVSOLID Analysis Output Files	152
4.19	Analyze the SVSOLID Problem	153
4.20	SVSOLID Results	154
4.20.1	Vertical Stress Contours	154
4.20.2	Suction Contours	155
4.20.3	Displacement Vectors	156
4.21	SVSOLID Summary File Results	156
4.21.1	Vertical Displacement Contours	157
4.21.2	Vertical Displacement at Ground Surface	158
4.21.3	Vertical Displacement Below Cover Edge	159
4.22	Contour Displacements in AcuMesh	159
4.23	Problem Conclusion	161
5	References	162
6	Index	163

1 A TWO DIMENSIONAL EXAMPLE PROBLEM

The following example will introduce some of the features included in SVSOLID and will set up a model of a simple slope with 2 foundations. A water table is present. The purpose of this problem is to determine the stress condition in the slope and the magnitude of displacement under each footing. The problem dimensions and soil properties are provided below.

ProjectID: Tutorial ProblemID: Tutorial2D

Footing 1



Footing 2

Region Geometry

Ground

X	Y
5	41
5	38
5	20
73	20
73	30
73	32
73	36
64	37
59	39
54	39

Seam

X	Y
5	38
8	37
10	37
24	36
42	35
57	33
73	30
73	32
40	38
25	38

Water Table

X	Y
5	39
73	34

52	40
48	41
42	41
38	42
35	43
31	43
27	44
25	45
22	46
19	47
15	47
12	46
12	43
5	43

10	40
8	40
6	41
5	41

Soil Properties

Soil 1: Till

Young's Modulus, $E = 50000$ kPa

Poisson's Ratio, $\nu = 0.4$

Initial Void Ratio, $e_o = 1$

Vertical Body Load, $\gamma_v = -21$ kN/m³

Soil 2: Clay

Young's Modulus, $E = 3000$ kPa

Poisson's Ratio, $\nu = 0.4$

Initial Void Ratio, $e_o = 1$

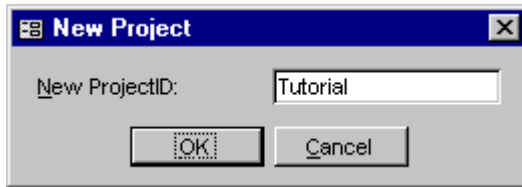
Vertical Body Load, $\gamma_v = -18.5$ kN/m³

1.1 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 this 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.



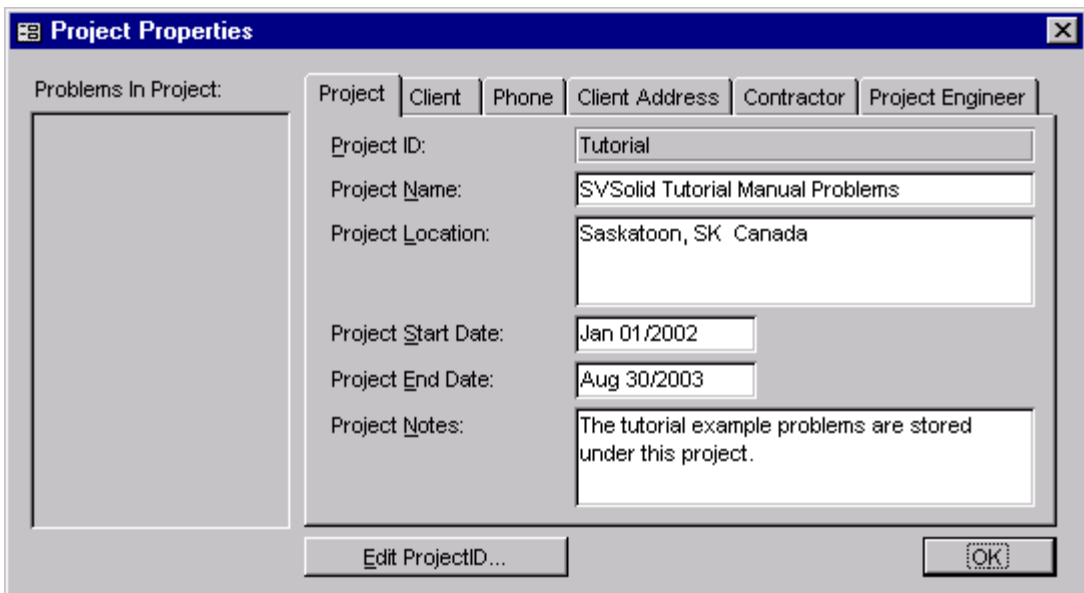
4. Type "Tutorial" as the new ProjectID and press OK.

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.

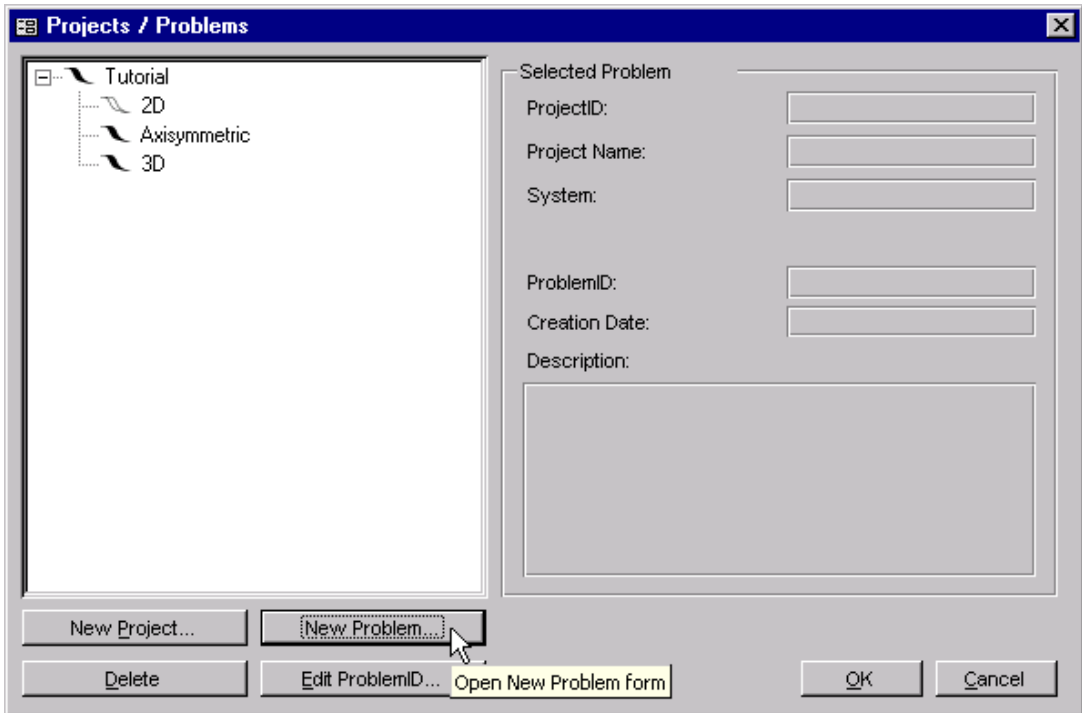


It should be noted that once the project is defined it will be identified by the ProjectID throughout the rest of the program. Also, SVSOLID does not allow you to specify two projects with the same ProjectID.

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.

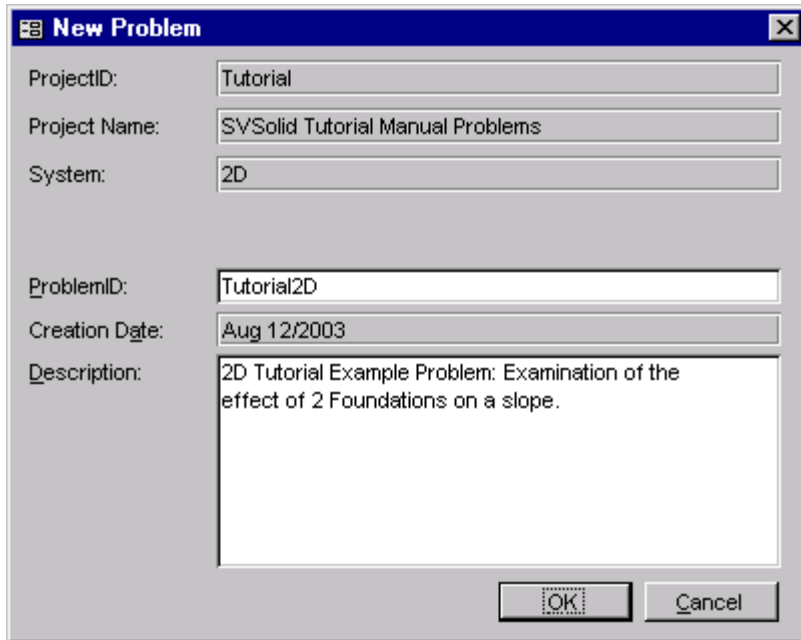
1.2 ADDING A PROBLEM

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: Tutorial2D

Creation Date: Aug 12/2003

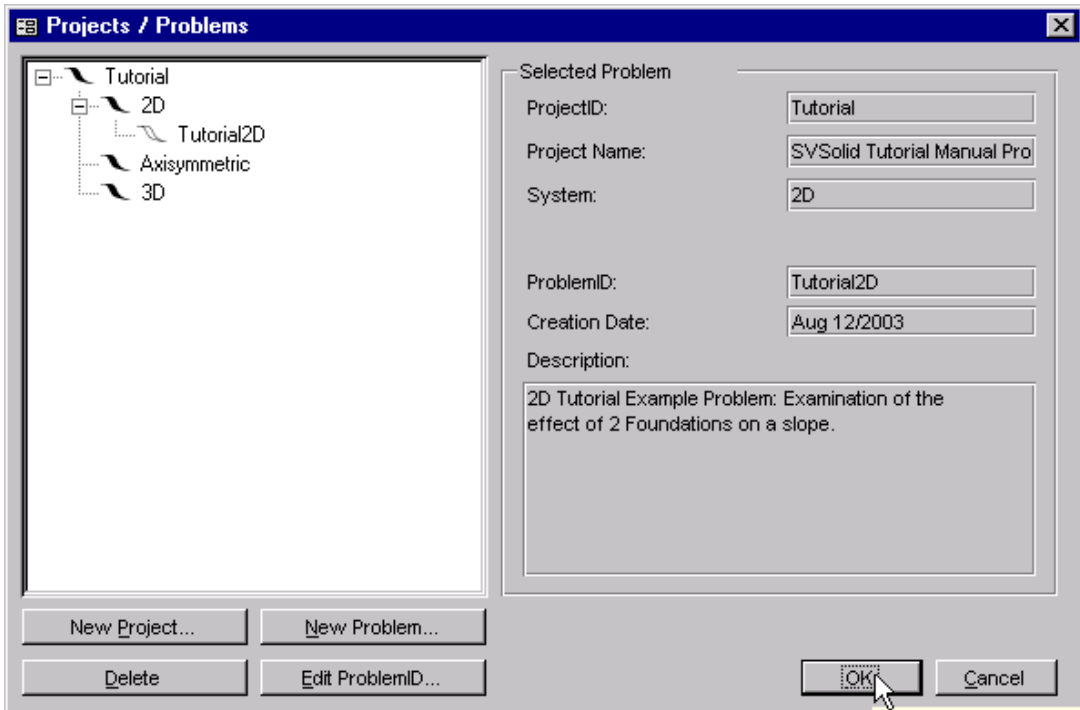
Description: 2D Tutorial Example Problem: Examination of the effect of 2 Foundations on a slope.

OK Cancel

4. Enter the **ProblemID: Tutorial2D**. The description is optional.
5. Click the **OK button** to save the problem and close the New Problem form.
6. The new problem will automatically be opened in the workspace.

1.3 OPENING THE PROBLEM

If the problem was just added it will already be open in the workspace. When returning to the problem follow these steps to open it in the workspace:



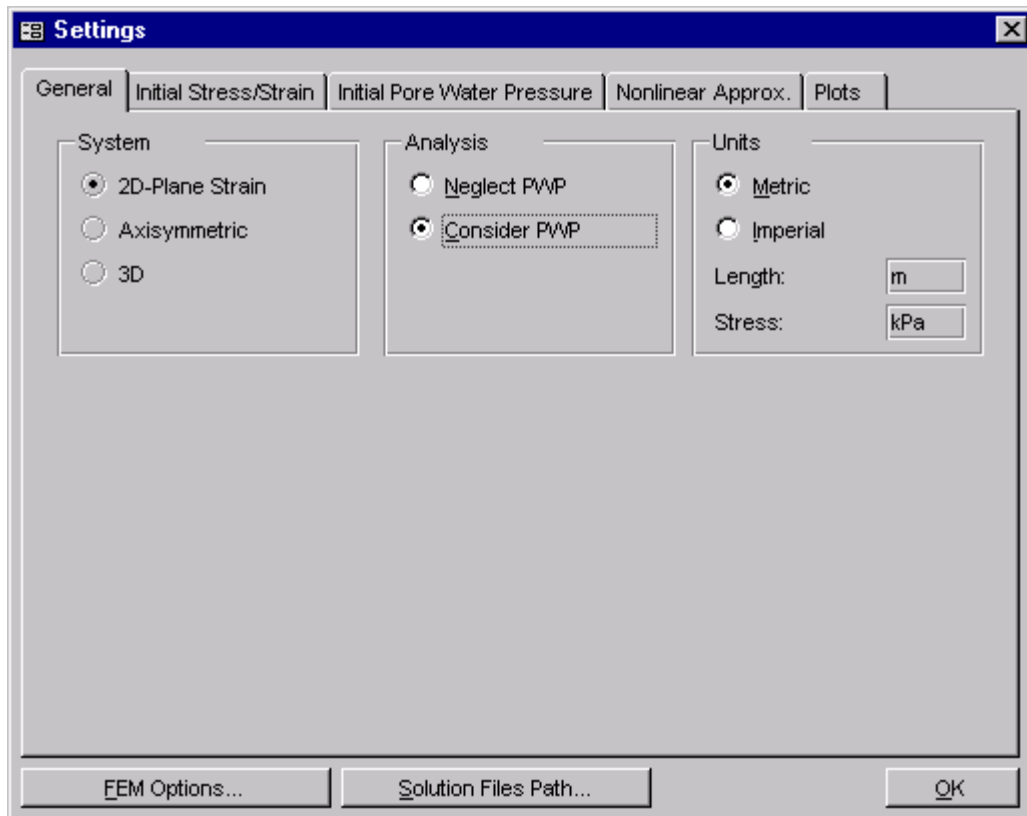
7. **Navigate** back to the problem via Tutorial, 2D.
8. Select **Tutorial2D**.
9. The problem may be opened by clicking the **OK** button or by double clicking on the ProblemID.

1.4 DEFINING THE PROBLEM

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

1.4.1 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.

1.4.2 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

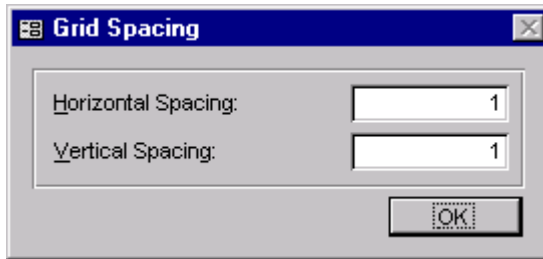
X: Y:

*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. The geometry of the problem has coordinates of a precision of 1m. In order to effectively draw geometry with this precision using the mouse the grid spacing must be set to a maximum of 1.

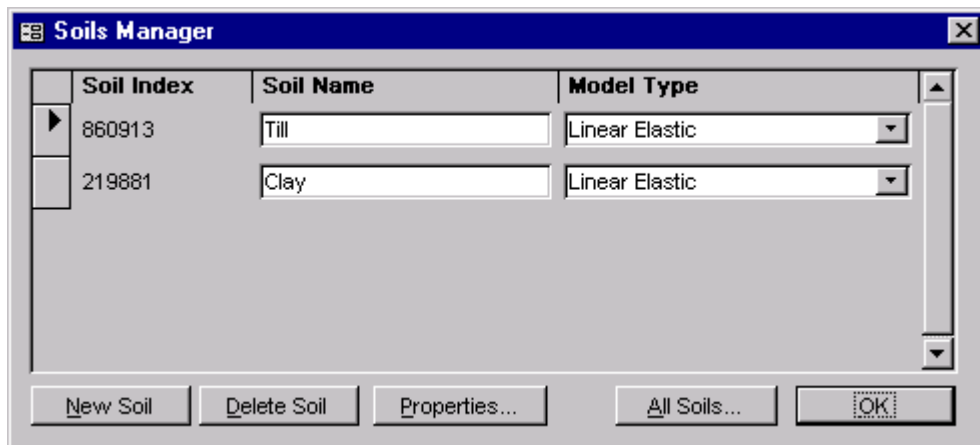
5. Select **View > Grid Spacing** from the menu.



The 'Grid Spacing' dialog box has a title bar with a blue background and a close button. It contains two input fields: 'Horizontal Spacing' and 'Vertical Spacing', both with the value '1' entered. Below the input fields is an 'OK' button.

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


1.4.3 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
860913	Till	Linear Elastic
219881	Clay	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 ground 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: 219881

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

Soil Name: Clay

USCS Texture: Silty clay

Soil Description:

Geologic Description:

Notes: Clay soil for 2D tutorial problem.

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: 219881

Description PWP Parameters Body Load SVDynamic Reference

Elastic Parameters

Young's Modulus Option

Young's Modulus: 3,000 kPa

Poisson's Ratio: 0.400

Bulk Modulus Option

Bulk Modulus: 5,000 kPa

Shear Modulus: 1,071 kPa

Initial Void Ratio, eo: 1.00

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 0.4.
8. Enter the **Initial Void Ratio** value of 1.
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 -18.5 kN/m^3 .
12. **Repeat** these steps to create the till soil.

**Tip!**

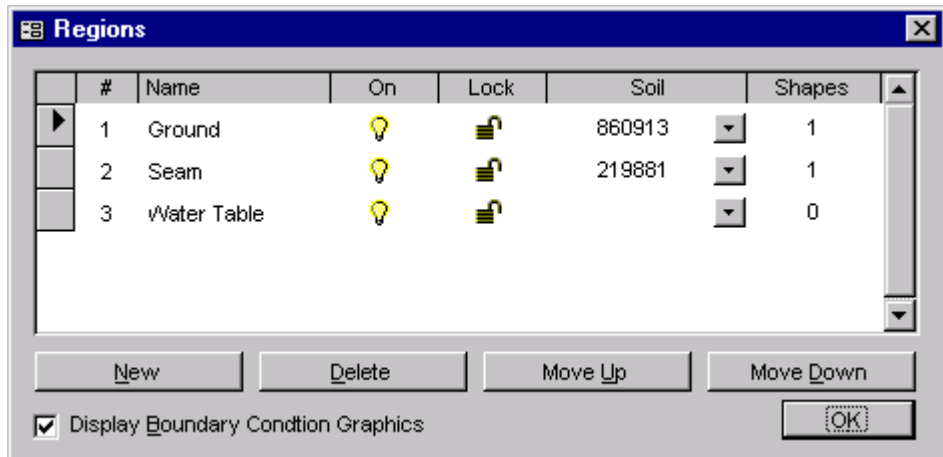
The negative sign for the body load indicates that the vertical body load will act down.

1.4.4 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 Ground, 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 **Ground**. 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 **Seam**.
6. Select the **clay** soil for the 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.

1.4.5 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 ground the instructions below explain the use of the command line to create the core shape.

1. Ensure that "**Ground**" 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.

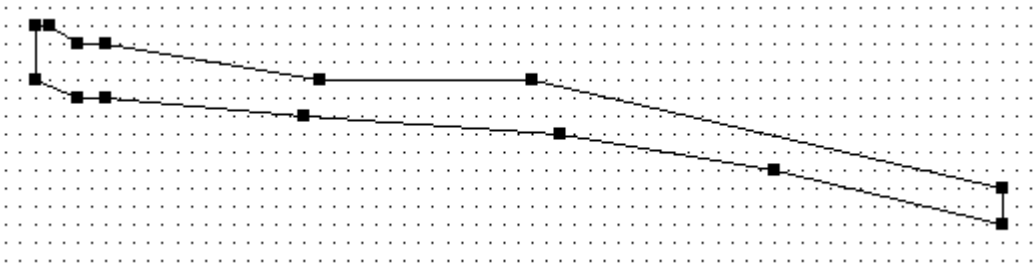
4. Type 5,41 and press the **Enter** key on the keyboard.
5. Type **5,38** and press Enter.
6. Refer to the **geometry table** at the beginning of this tutorial at enter the remaining points.
7. Type **f** and press Enter to complete the region shape.

- **Define the Seam**


In the instructions for drawing the ground 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 **“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 (5,38) 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 (8,37). **Right click** to snap the cursor to the exact point and then left click on the point. A line is now drawn from (5,38) to (8,37).
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 (5,41) and right click snapping the cursor to the point. **Double click** on the point to finish the shape. A line is now drawn from (6,41) to (5,41) and the shape is automatically finished by SVSOLID by drawing a line from (5,41) back to the start point, (5,38).

If the seam geometry been entered correctly the shape should look like the following:



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.




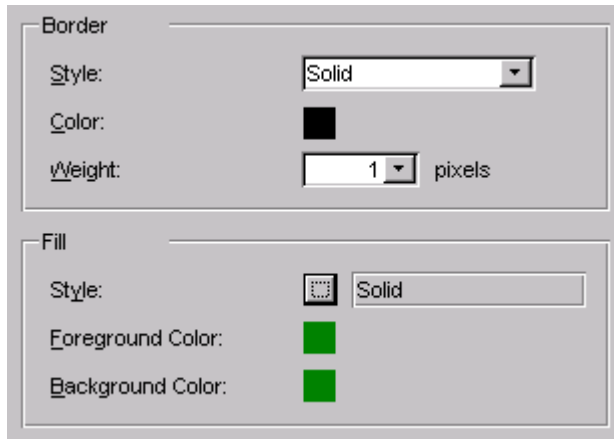
Tip!

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.

- **Formatting a Shape**

To set the Ground region shape to a solid green:

1. Select the **Ground** shape in the drawing space.
2. Press the **Object Properties** button,  to open the Region Properties form:



The image shows a dialog box titled "Region Properties" with two sections: "Border" and "Fill".

Border section:

- Style: Solid (dropdown menu)
- Color: Black (color swatch)
- Weight: 1 (dropdown menu) pixels

Fill section:

- Style: Solid (dropdown menu)
- Foreground Color: Green (color swatch)
- Background Color: Green (color swatch)

3. Click the **Fill Style** button. The Fill form will open.
4. Select the **solid** fill style and press **OK**.
5. Click the **Foreground** color box on the Region Properties form. The Color Palette will appear.
6. Select a **green** color and press **OK**.
7. Close the Region Properties form by pressing **OK**.

Repeat these steps to give the seam a solid gray fill.

After all the region geometry has been entered it will as appear like the diagram at the beginning of this tutorial.

1.4.6 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 each of the footing locations on the ground 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 Seam region is internal to the Ground region and will not need to be altered. The steps for specifying the boundary conditions are thus:

1. Select the "Ground" region in the drawing space.
2. From the menu select **Model > Boundaries**. The boundary conditions form will open.

Boundary Conditions [X]

Region: Ground Select Shape Index: 585874

Segment Point

	Point		Boundary Condition		
	X	Y	Direction	X	Y
	22	46	Condition:	Continue	Continue
			Expression:		
	19	47	Condition:	Continue	Continue
			Expression:		
	15	47	Condition:	Continue	Continue
			Expression:		
	12	46	Condition:	Fixed	Continue
			Expression:		
	12	43	Condition:	Free	Load Expression
			Expression:		-80
	5	43	Condition:	Fixed	Free

Segment Length: 3 m

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

OK

3. Select the **point** (5,41) 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 **Free** boundary condition.
6. Select the **point** (5,20) from the list.
7. From the **Y Boundary Condition** drop down select a **Fixed** boundary condition.
8. Select the **point** (73,20) from the list.
9. From the **Y Boundary Condition** drop down select a **Free** boundary condition.
10. Select the **point** (73,36) from the list.
11. From the **X Boundary Condition** drop down select a **Free** boundary condition.
12. Select the **point** (59,39) from the list.
13. From the **Y Boundary Condition** drop down select a **Load Expression** boundary condition.
This will cause the Load Expression box to be enabled.
14. In the Expression box enter a **load of -100**.
15. Select the next **point** (54,39) from the list.
16. From the **Y Boundary Condition** drop down select a **Free** boundary condition.
17. Select the **point** (12,46) from the list.
18. From the **X Boundary Condition** drop down select a **Fixed** boundary condition.
19. Select the **point** (12,43) from the list.
20. From the **X Boundary Condition** drop down select a **Free** boundary condition.
21. From the **Y Boundary Condition** drop down select a **Load Expression** boundary condition.
This will cause the Load Expression box to be enabled.
22. In the Expression box enter a **load of -80**.
23. For the last point, select Fixed as the X Boundary Condition and Free as the Y Boundary Condition.
24. Click the **OK** button to close the form.




The Fixed X boundary condition for the point (5,41) 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 Free condition at point (73,36) the Continue is turned off.

Boundary Condition Summary

X	Y	X Boundary Condition	Y Boundary Condition
5	41	Fixed	Free
5	38	Continue	Continue
5	20	Continue	Fixed
73	20	Continue	Free
73	30	Continue	Continue
73	32	Continue	Continue
73	36	Free	Continue
64	37	Continue	Continue
59	39	Continue	Load Expression = -100
54	39	Continue	Free
52	40	Continue	Continue
48	41	Continue	Continue
42	41	Continue	Continue
38	42	Continue	Continue
35	43	Continue	Continue
31	43	Continue	Continue
27	44	Continue	Continue
25	45	Continue	Continue
22	46	Continue	Continue
19	47	Continue	Continue
15	47	Continue	Continue
12	46	Fixed	Continue
12	43	Free	Load Expression = -80
5	43	Fixed	Free

1.5 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. With the mouse click on the point (5,39).
3. To finish the Water Table double-click on the point (73,34). A dashed-dot line is drawn across the problem.

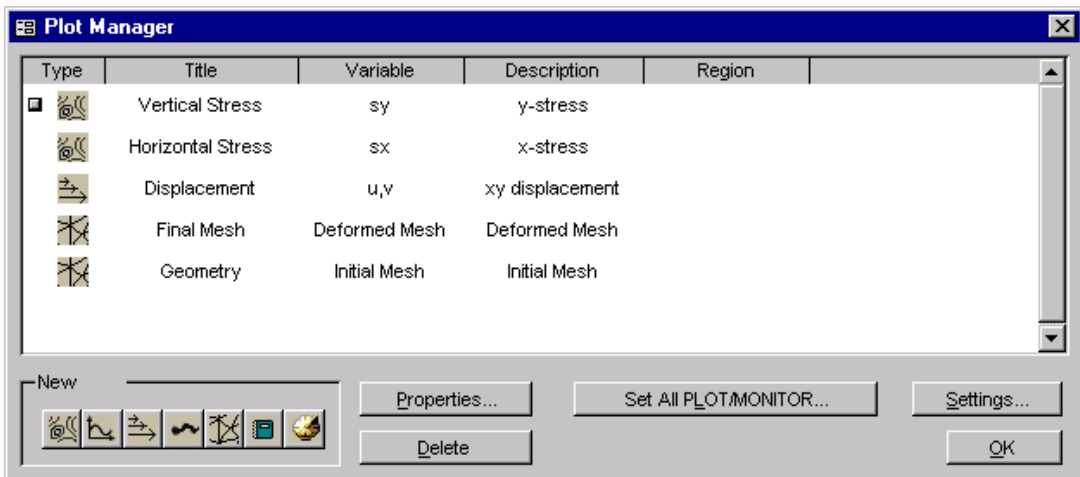
**Tip!**

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

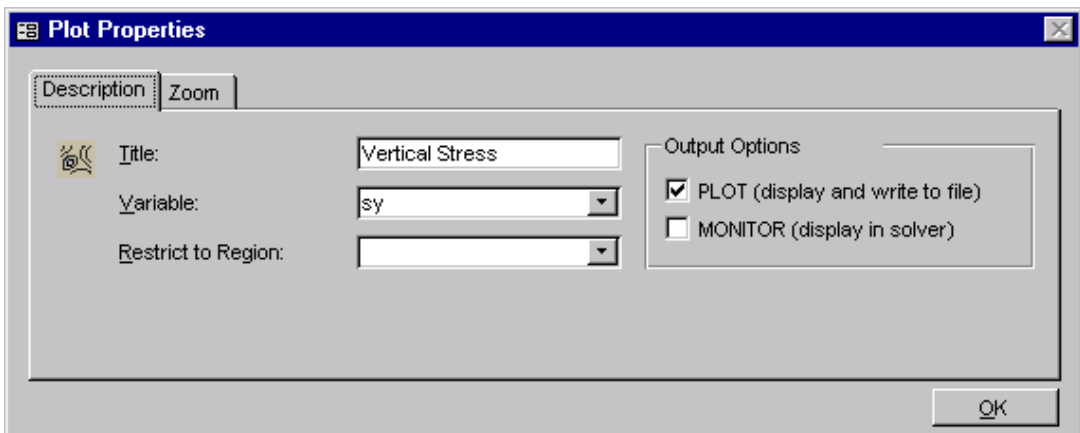
1.6 SPECIFY PLOTS

There are many plot types that can be specified to visualize the results of the model. A few will be generated for this tutorial example problem including a plot of the solution mesh, stress contours, and displacement vectors.

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



2. 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.

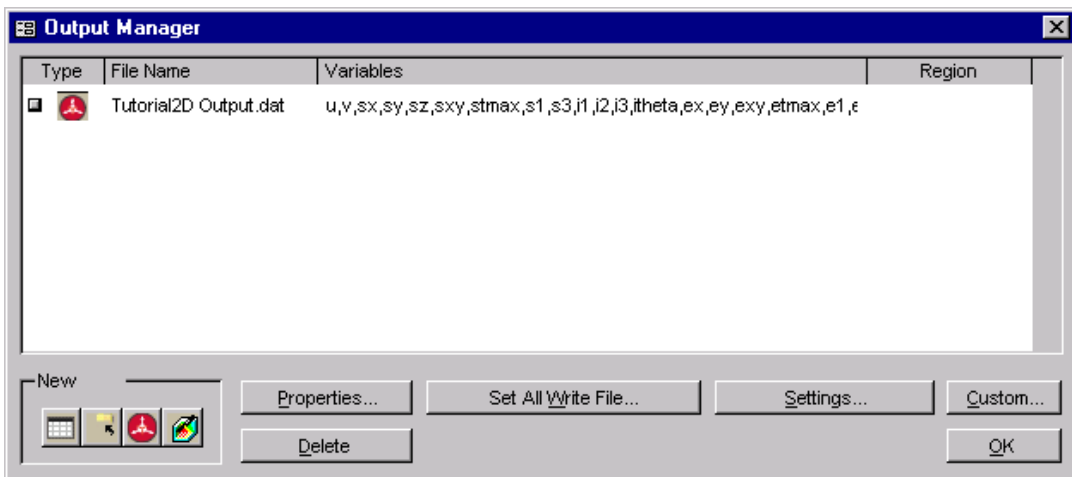


3. Enter the title Vertical Stress.
4. Select sy as the variable to plot from the drop-down.
5. Select the **PLOT** output option.
6. Click OK to close the form and add the plot to the list.
7. Repeat these steps 2 – 6 to create the plots shown above.
8. Click **Settings** to open the Problem Plot Settings form.
9. Enter a **Magnification Factor of 50**. This will allow the areas of deformation to be examined easier.
10. Click **OK** to close the Plot Manager and return to the workspace.

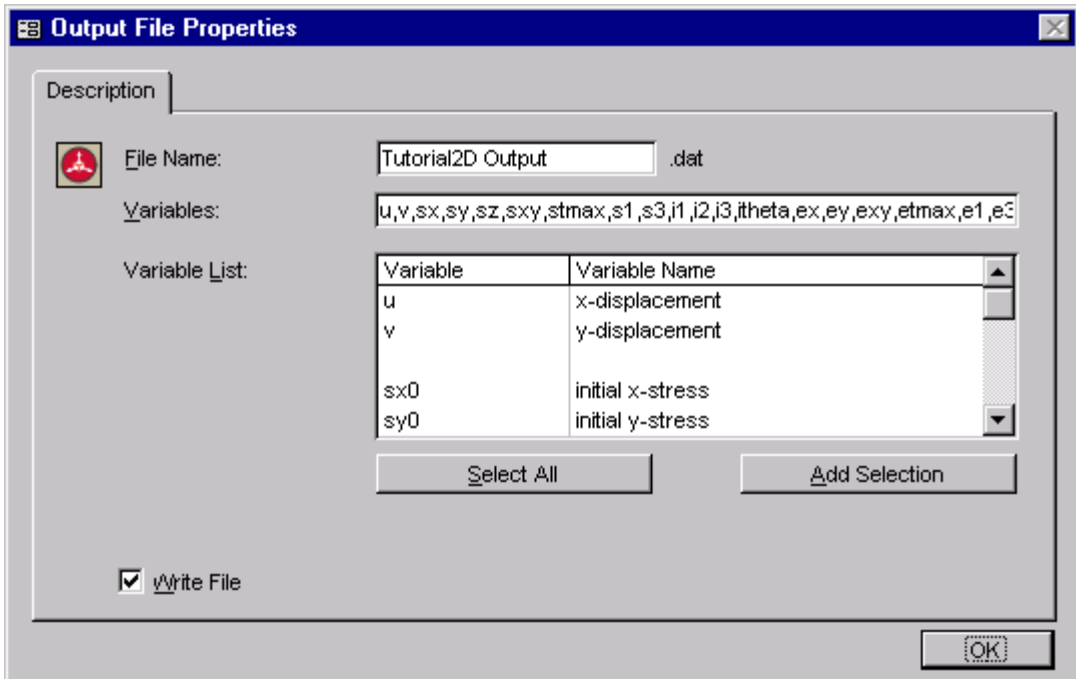
1.7 SPECIFY OUTPUT FILES

There are 4 output file types that can be specified to export the results of the model. One will be generated for this tutorial example problem: a plot to transfer the results to AcuMesh.

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 AcuMesh button to begin adding the output file. The Output File Properties form will open.



3. Enter the title Tutorial2D Output.
4. Select all the variables 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.

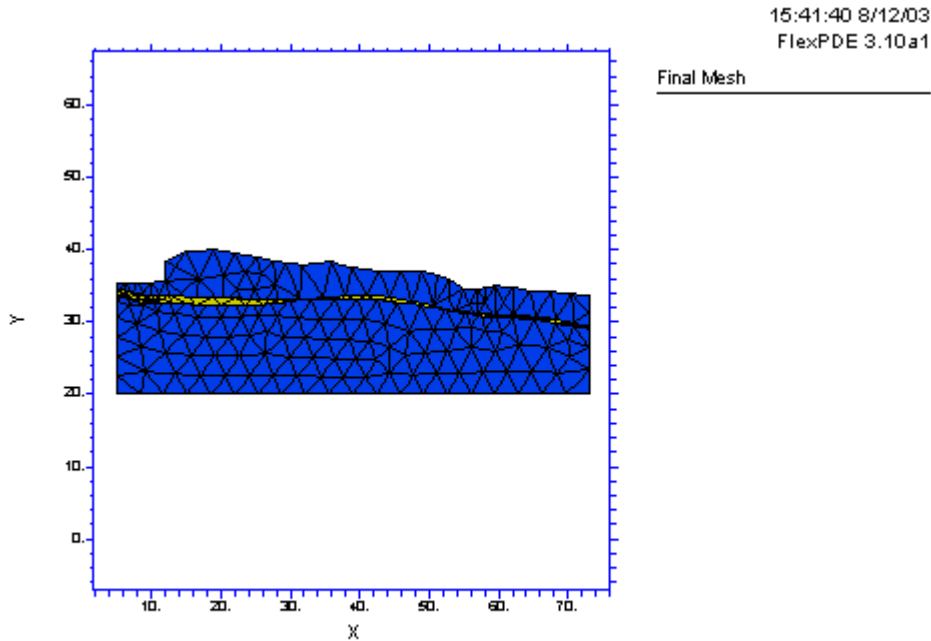
1.8 ANALYZE

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

1.9 RESULTS

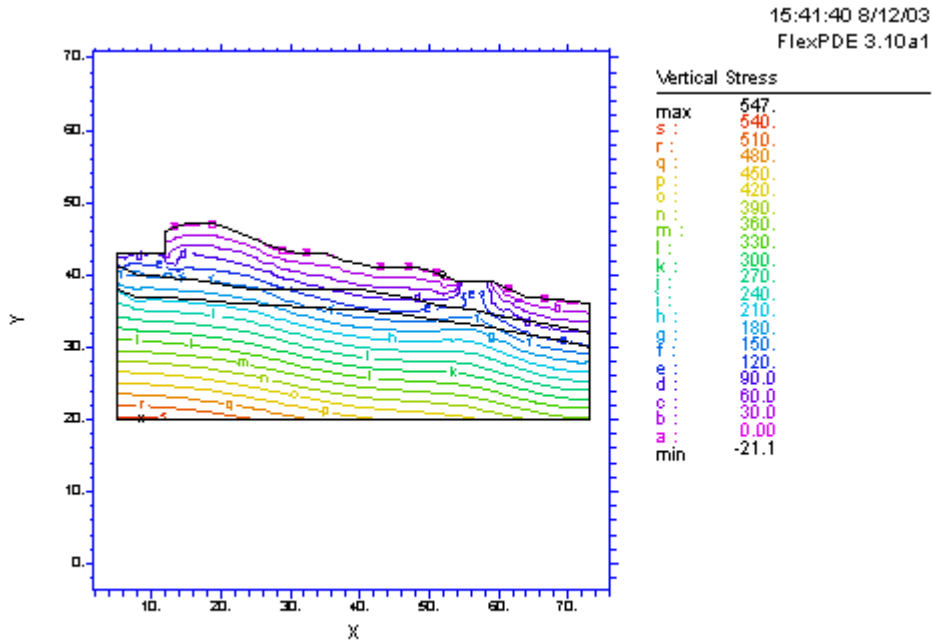
After the problem has finished solving, the results will be displayed in the form of thumbnail plots within the SVSOLID solver. Right-click the mouse and select Maximize to enlarge any of the thumbnail plots. This section will give a brief analysis for each plot that was generated.

1.9.1 Solution Mesh



The Mesh plot displays the finite-element mesh generated by the solver. The mesh is automatically refined in critical areas such as directly beneath the footings where there is a greater influence of the loads. The displacements in this plot are magnified by 50 times. Note that the greatest displacement occurs directly beneath the footings. Also the displacement in the clay seam is much greater than the surrounding till due to large difference in Young's Modulus values.

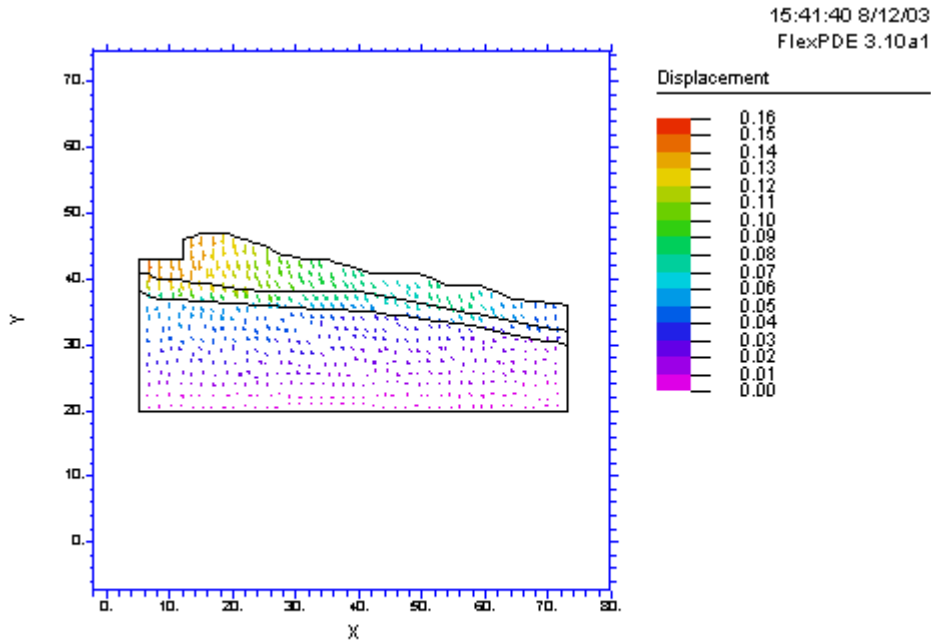
1.9.2 Vertical Stress Contours



Tutorial_Tutorial2D: Grid#2 p2 Nodes=769 Cells=352 RMS Err= 1.4e-4
Integral= 358655.1

A stress bulb is generated beneath each footing due to each footing load. The body load of both materials generates the overall stress state.

1.9.3 Displacement Vectors

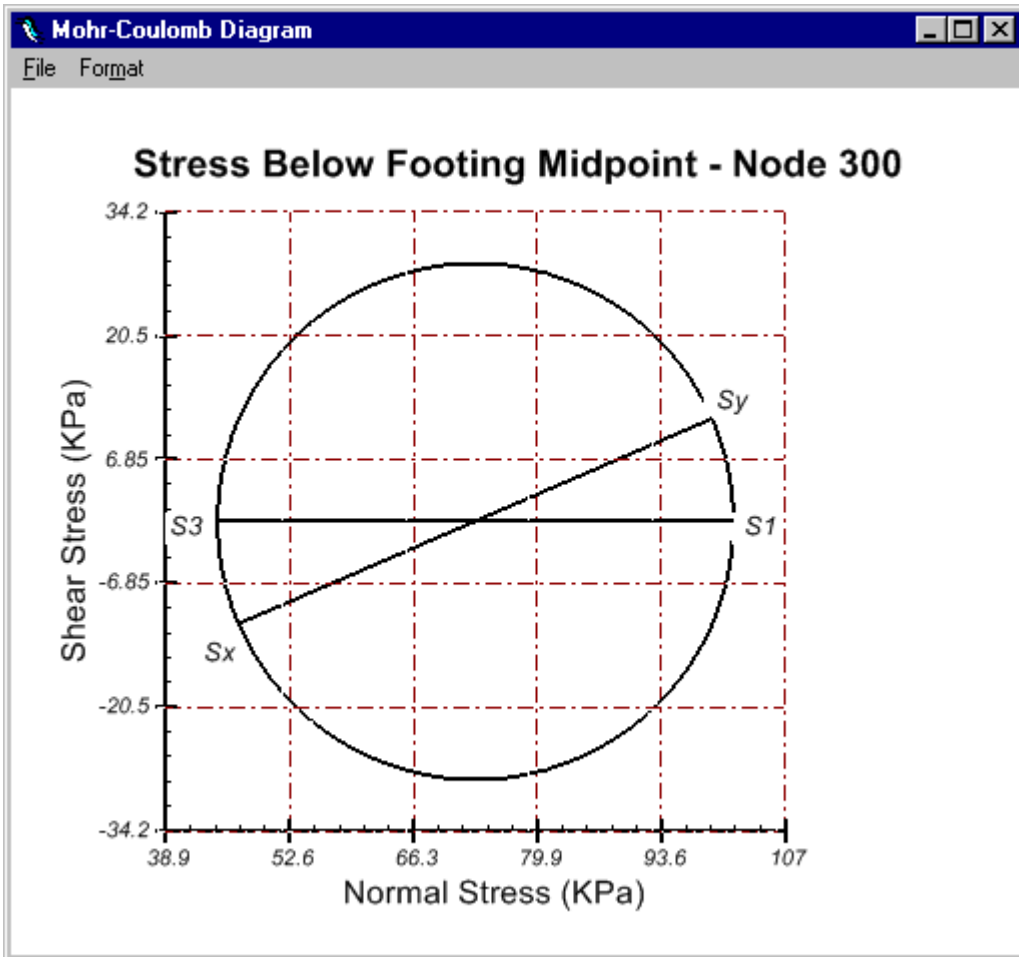


Tutorial_Tutorial12D: Grid#2 p2 Nodes=769 Cells=352 RMS Err= 1.4e-4

Displacement Vectors show both the direction and the magnitude of the displacement at specific points in the problem. The lower Young's Modulus in the clay seam result it greater displacements here than in the surrounding till. The maximum displacement of 0.16m occurs beneath Footing 1 where the load is greater and the distance to the clay seam is less.

1.9.4 ACUMESH Mohr Circle Diagram

Use ACUMESH, the SoilVision Systems Ltd. 2D visualization module for improved graphics and greater range of plotting options. Create plots of principle stress crosses or view Mohr Circles as below.

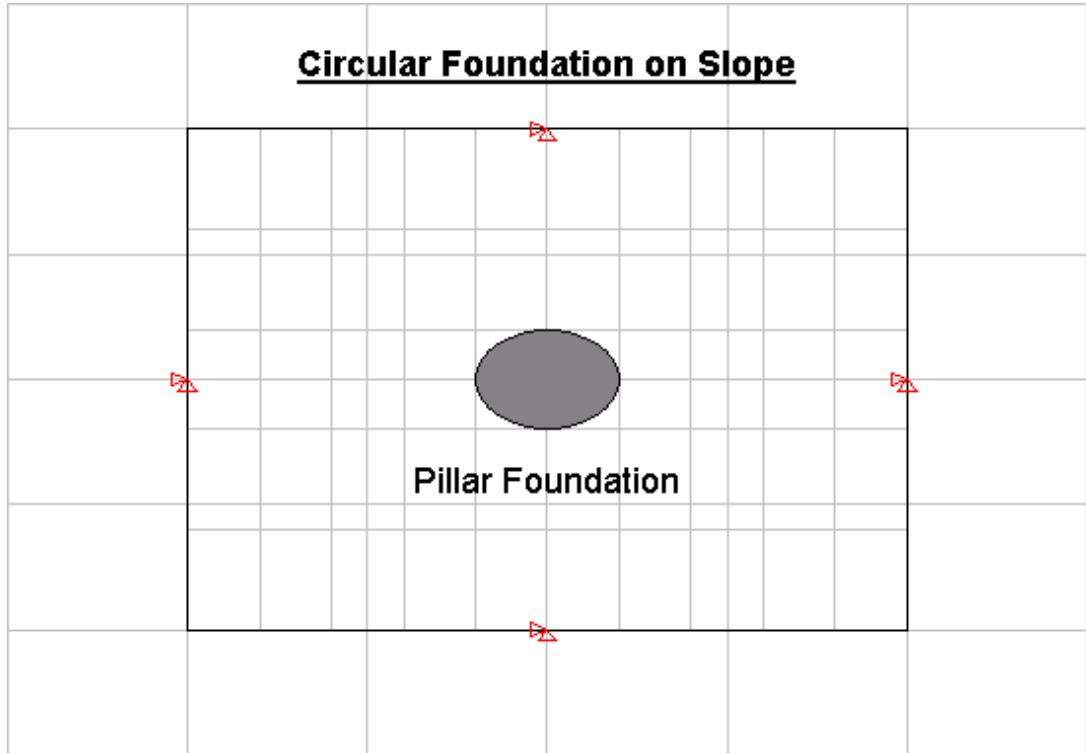


2 A THREE DIMENSIONAL EXAMPLE PROBLEM

The following example will introduce you to the three dimensional model in SVSOLID. The model considers the stress and displacement in the soil of a slope. A circular pillar foundation has been placed in mid-slope. The problem is modeled using 2 regions, 3 surfaces, and 2 soils. The problem data and soil properties are provided below. This problem will be set up to run with the Student Version.

ProjectID: Tutorial

ProblemID: Tutorial3D



Region Geometry

Slope

X	Y
0	0
20	0
20	20
0	20

Pillar

	X	Y
Center:	10	10
Radius:	2	

Soil Properties

Soil 1: Till

Young's Modulus, $E = 10,000$ kPa

Poisson's Ratio, $\nu = 0.4$

Initial Void Ratio, $e_0 = 1$

Vertical Body Load, $\gamma_y = -21$ kN/m³

Soil 2: Concrete

Young's Modulus, $E = 29,580,000$ kPa

Poisson's Ratio, $\nu = 0.2$

Initial Void Ratio, $e_0 = 0$

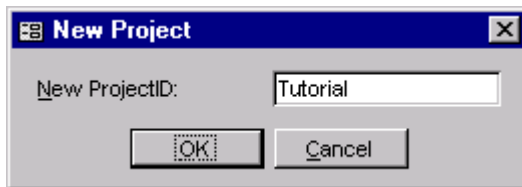
Vertical Body Load, $\gamma_y = -23.5$ kN/m³

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 this 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 | Client | Phone | Client Address | Contractor | Project Engineer

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

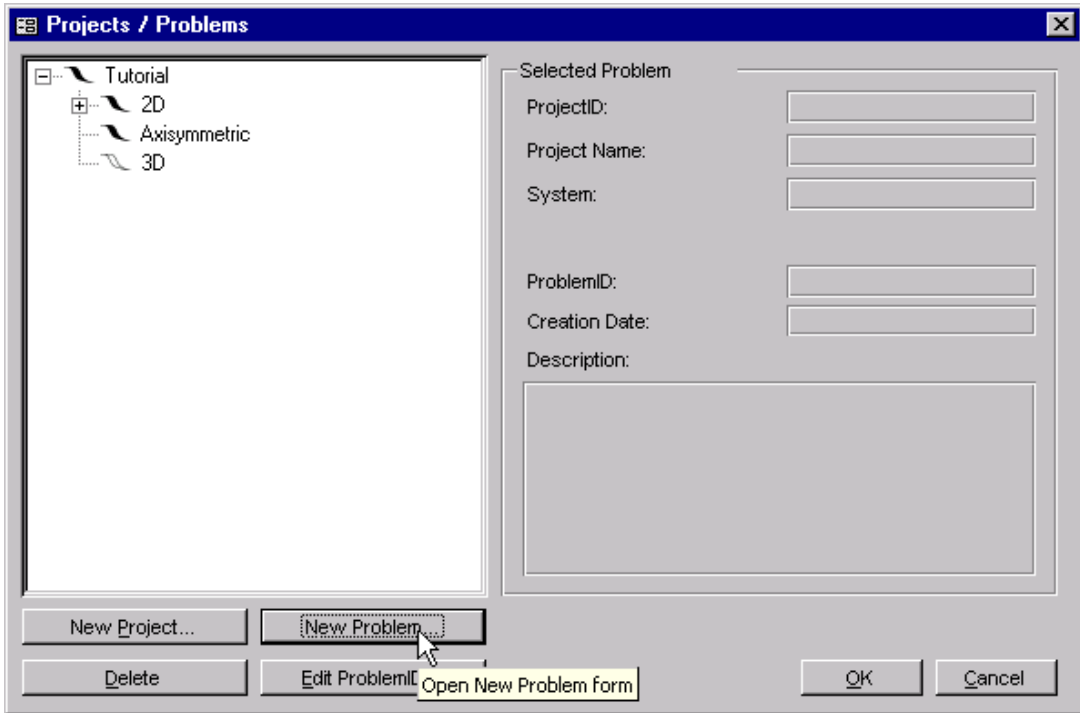
Project Notes: The tutorial example problems are stored under this project.

Edit ProjectID... OK

It should be noted that once the project is defined it will 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.

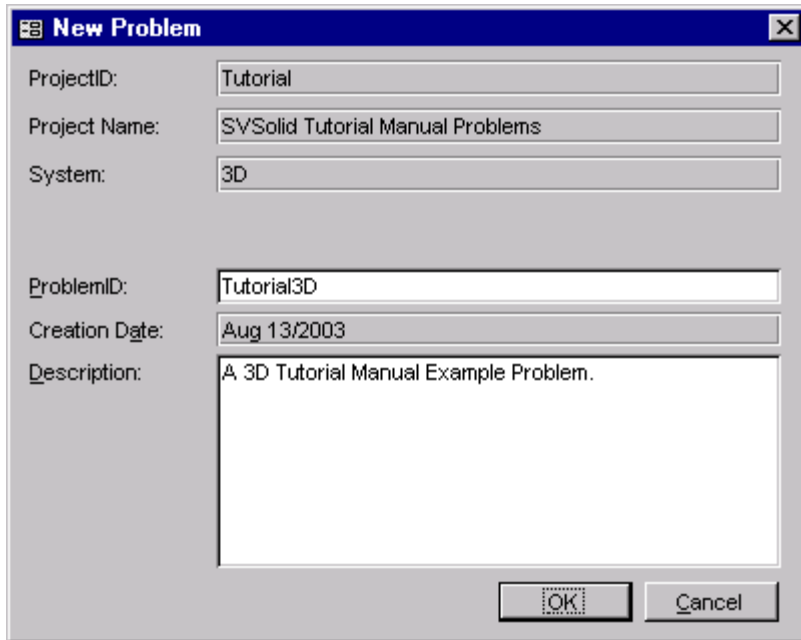
2.2 ADDING A PROBLEM



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 three-dimensions, expand **3D** by clicking the plus sign beside "3D"
3. Click the **New Problem** button. The New Problem form will open.



New Problem

ProjectID: Tutorial

Project Name: SVSolid Tutorial Manual Problems

System: 3D

ProblemID: Tutorial3D

Creation Date: Aug 13/2003

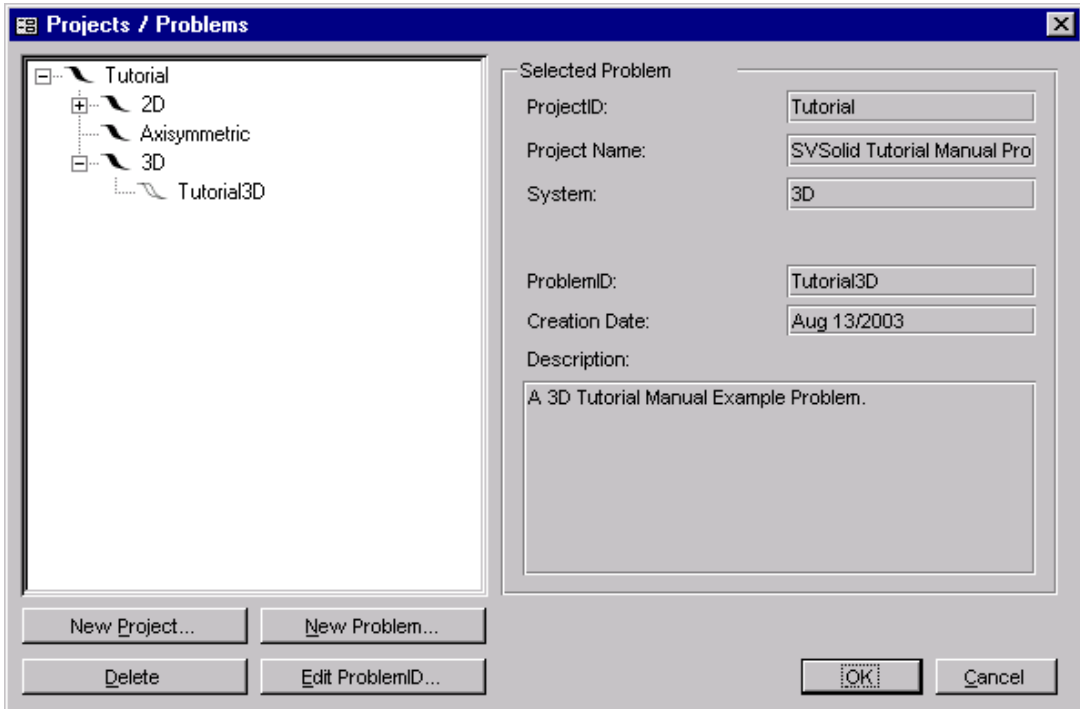
Description: A 3D Tutorial Manual Example Problem.

OK Cancel

4. Enter the **ProblemID: Tutorial3D**. The description is optional.
5. Click the **OK** button to save the problem and close the New Problem form.
6. The new problem will automatically be opened in the workspace.

2.3 OPENING THE PROBLEM

If the problem was just added it will already be open in the workspace. When returning to the problem follow these steps to open it in the workspace:




1. **Navigate** back to the problem via Tutorial, 3D.
2. Select **Tutorial3D**.
3. The problem may be opened by clicking the **OK** button or by double clicking on the ProblemID.

2.4 DEFINING THE PROBLEM

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

2.4.1 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

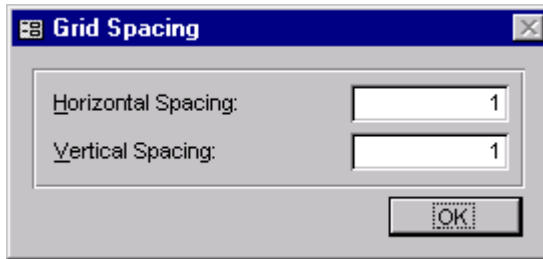
X: Y:

*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. The geometry data for this problem has coordinates of a precision of 1m. In order to effectively draw geometry with this precision using the mouse the grid spacing must be set to a maximum of 1.

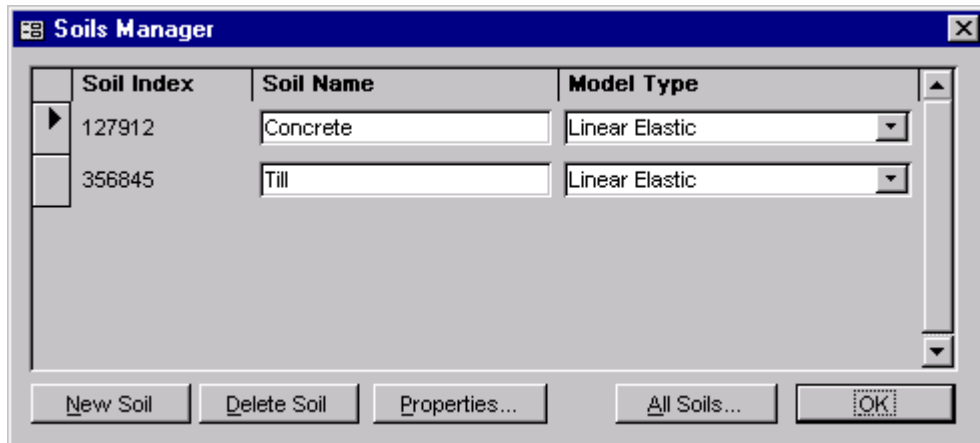
5. Select **View > Grid Spacing** from the menu.



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.


2.4.2 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
127912	Concrete	Linear Elastic
356845	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 pillar foundation will consist of concrete. This section will provide instructions on creating the till soil. Repeat the process to add the other soil as described at the beginning of this tutorial.

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: 356845

Description | PWP | Parameters | Body Load | Reference

Soil Name: Till

USCS Texture:

Soil Description: Insitu Till

Geologic Description:

Notes:

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: 356845

Description PWP Parameters Body Load Reference

Elastic Parameters

Young's Modulus Option

Young's Modulus: 10,000 kPa

Poisson's Ratio: 0.400

Bulk Modulus Option

Bulk Modulus: 16,667 kPa

Shear Modulus: 3,571 kPa

Initial Void Ratio, eo: 1.00

OK

6. Refer to the data provided at the beginning of this tutorial. Enter the **Young's Modulus** value of 10000 kPa.
7. Enter the **Poisson's Ratio** value of 0.4.
8. Enter the **Initial Void Ratio** value of 1.
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 0 kN/m^3 .
12. Enter the **Z-Axis Body Load** as -21 kN/m^3 .
13. **Repeat** these steps to create the concrete soil.

**Tip!**


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

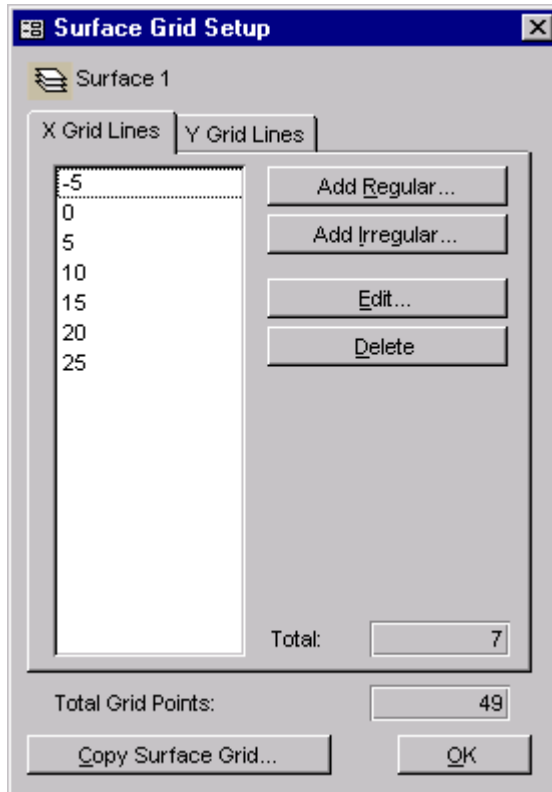
2.4.3 Define 3D Surfaces

This problem consists of three surfaces of different dimensions and grid densities. By default every problem initially has 2 surfaces.

- **Define Surface 1**


This surface is already present so the next step is to define the grid lines.

1. Select Surface 1 in the **Surface Selector**.
2. Click the **Surface Grid Setup** button, .




3. There will be default grid lines of 0 and 10 present. Click the **Add Regular** button to open the Add Regular X Gridlines form.
4. Enter -5 for **Start**, 5 for **Increment Value**, and 25 for **End**.
5. Click **Add** to add the gridlines and close the form.
6. Move to the **Y Grid Lines** tab and repeat steps 3 – 5 for the Y gridlines

Now that the grid has been set up, elevations must be specified for all the grid points:


1. Select Surface 1 in the Surface Selector.
2. Click the Elevations button,  to open the Elevations form.
3. Enter 0 in the Set Nulls To field.
4. Click the button to the left of the Set Nulls To field and all the missing elevations will be set to 0.

- **Define Surface 2**

This surface is already present. The grid extents are smaller than Surface 1 and the grid is denser. The Surface 2 grid also has different densities in the X and Y directions.


1. Select Surface 2 in the **Surface Selector**.
2. Click the **Surface Grid Setup** button, .
3. There will be default grid lines of 0 and 10 present. Click the **Add Regular** button to open the Add Regular X Gridlines form.
4. Enter 0 for **Start**, 2 for **Increment Value**, and 20 for **End**.
5. Click **Add** to add the gridlines and close the form.
6. Move to the **Y Grid Lines** tab.
7. There will be default grid lines of 0 and 10 present. Click the **Add Regular** button to open the Add Regular Y Gridlines form.
8. Enter 0 for **Start**, 4 for **Increment Value**, and 20 for **End**.
9. Click **Add** to add the gridlines and close the form.

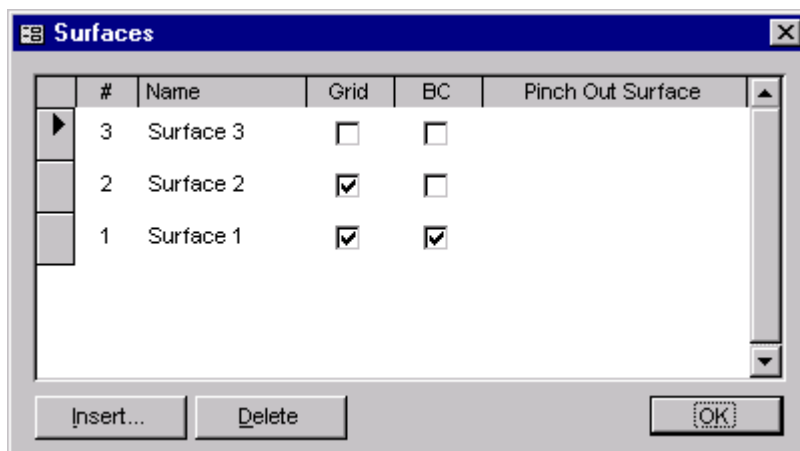
Now that the grid has been set up, elevations must be specified for all the grid points:

10. Select Surface 2 in the Surface Selector.
11. Click the Elevations button,  to open the Elevations form.
12. Enter 4 in the Set Nulls To field.
13. Click the button to the left of the Set Nulls To field and all the missing elevations will be set to 4m.

- **Define Surface 3**

Follow these steps to add the third surface to the problem.

1. To open the surface form you may click on the **Surface** command button,  located at the top of the workspace or select **View > Surfaces** from the menu.



2. Click **Insert** to open the Insert Surfaces form:

Insert Surfaces

Number of New Surfaces:

Place New Surfaces

At The Top

Below Surface:

Insert New Surfaces

With Default Grid

Copy Grid From An Existing Surface

Surfaces with Grids:

Elevations


Exclude

Include

Offset Elevations By: m

3. Enter 1 as the **Number of New Surfaces**.
4. Select to place the new surface **At The Top**.
5. Select **Copy Grid From An Existing Surface**.
6. Select **Surface 2** from the drop-down.
7. Choose to **Exclude** the elevations.
8. Press **OK** to add the surface.

Now that Surface 3 has been added and its grid has already been set up the next step is to provide the elevations for it. The slope will be generated using the 3D Plane Interpolation method:

1. Select Surface 3 in the **Surface Selector**.
2. Click the **Elevations** button,  to open the Elevations form.

Elevations

Surface 3

	X	Y	Z
	0	0	6
	0	4	6
	0	8	6
	0	12	6
	0	16	6
	0	20	6
	2	0	6.4
	2	4	6.4
	2	8	6.4
	2	12	6.4
	2	16	6.4
	2	20	6.4
	4	0	6.8
	4	4	6.8
	4	8	6.8
	4	12	6.8
	4	16	6.8

Total Grid Points:

Pinch Out Surface At:

General

Find Missing Elevations

Delete Elevations

Paste Surface Data...

Import XYZ Data...

Surface Grid Integrity

Add Missing Grid Points

Check for Duplicate Points

Adjust Overlap...

Crop Elevations...

Set Elevations For Point

X-Slope

Y-Slope

Bilinear Interpolation

Snap Above

Snap Below

Set Elevations For Surface

3D Plane Interpolation

Set Nulls To: m

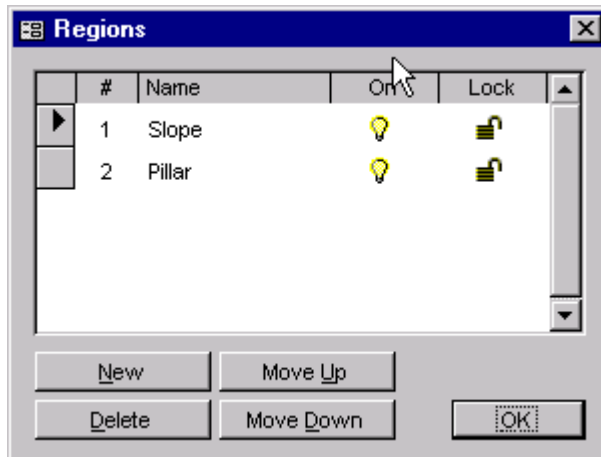
3. Select point (0,0).
4. Enter a Z elevation of **6**.
5. Select point (0,20).
6. Enter a Z elevation of **6**.
7. Select point (20,20).
8. Enter a Z elevation of **10**.
9. Press the **3D Plane Interpolation** button
10. Press **OK** to close the form.

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 2 regions, which are named Slope and Pillar. Each region will have one of the soils just defined specified as its soil properties. 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. Press the **New** button to add a second region.
4. Change the name of the second region to **Pillar**.
5. Click **OK** to close the form.

2.4.5 Defining Region Geometry Shapes


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 beginning of this tutorial for the geometry points for each region.

- **Define the Slope region**

1. Ensure the "**Slope**" 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 (0,0) 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 (20,0). Right click to snap the cursor to the exact point and then left click on the point. A line is now drawn from (0,0) to (20,0).
8. Now move the cursor near (20,20). Right click to snap the cursor to the exact point and then left click on the point.


- For the **last** point (0,20), right click to snap the cursor to the point. Double click on the point to finish the shape. A line is now drawn from (20,20) to (0,20) and the shape is automatically finished by SVSOLID by drawing a line from (0,20) back to the start point, (0,0).



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.

- Define the Pillar**

In the instructions for drawing the Slope shape the mouse was used. To draw the pillar the instructions below explain the use of the command line to create the Pillar shape.


- Ensure that “Pillar” is current in the region selector.
- Click on the Draw Polygon Region Shape button, .
- The command line will be set to Circle Center and the cursor focus will be in the command line.
- Type 10,10 and press the Enter key on the keyboard. The command line will be set to Radius.
- Type 2 and press Enter to complete the circular region shape.



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.

- Formatting a Shape**

To set the Pillar region shape to a solid gray:

- Select the **Pillar** shape in the drawing space.
- Press the **Object Properties** button,  to open the Region Properties form:

Region:

Points

	X	Y	Node Spacing
Center:	10	10	
Radius:	2		

Display Properties

Border

Style:

Color:

Weight: pixels

Fill

Style:

Foreground Color:

Background Color:

Mesh Spacing:


Sticky Points - Global Setting

3. Click the **Fill Style** button. The Fill form will open.
4. Select the **solid** fill style and press **OK**.
5. Click the **Foreground** color box on the Region Properties form. The Color Palette will appear.
6. Select a **gray** color and press **OK**.
7. Close the Region Properties form by pressing **OK**.


After all the region geometry has been entered it will as appear like the diagram at the beginning of this tutorial.

2.4.6 Specifying a Soil by Region and Layer

Each region will cut through all the layers in a problem creating a separate “block” on each layer. Each block can be assigned a soil or be left as **void**. A void area is essentially air space. In this problem all “blocks” will be assigned a soil.

1. Select “**Slope**” in the Region Selector.
2. Press the **Region Soils** button,  at the top of the workspace to open the Region Soils form.

	Bound By	Layer	Soil
▶	Surface 3 Surface 2	2	356845
	Surface 2 Surface 1	1	356845

3. Select the **Till** soil from the drop-down for **Layer 2**. (Note that the Soil Indexes will be different than depicted here)
4. Select the **Till** soil from the drop-down for **Layer 1**.
5. Close the form using the **OK** button.
6. Select "**Pillar**" in the Region Selector.
7. Press the **Region Soils** button,  at the top of the workspace to open the Region Soils form.
8. Select the **Concrete** soil from the drop-down for **Layer 2**.
9. Select the **Till** soil from the drop-down for **Layer 1**.
10. Close the form using the OK button.

2.4.7 Specify Boundary Conditions

Now that all of the regions, surfaces, and the soils have been successfully defined, the next step is to specify the boundary conditions on the region shapes. The vertical boundaries of the slope will be fixed as will the base. A load of 500 kPa will be applied to the top of the pillar. The steps for specifying the boundary conditions are thus:

- **Slope Region**

1. Select the "Slope" region in the drawing space.
2. Select Surface 1 in the surface selector.
3. From the menu select **Model > Boundaries**. The boundary conditions form will open and display the boundary conditions for Surface 1. These boundary conditions will extend from Surface 1 to Surface 2 over Layer 1.

Boundary Conditions

Location

Region: Slope

Surface: Surface 1

Surface Boundary Condition

Dir:	X	Y	Z
Cond:	Fixed	Fixed	Fixed
Expr:			

Segment Boundary Conditions

	Point		Boundary Condition			
	X	Y	Dir:	X	Y	Z
▶	0	0	Cond:	Fixed	Fixed	Continue
			Expr:			
	20	0	Cond:	Continue	Continue	Continue
			Expr:			
	20	20	Cond:	Continue	Continue	Continue
			Expr:			
	0	20	Cond:	Continue	Continue	Continue
			Expr:			

Segment Length: 20 m

Notes... Copy To Surface... OK

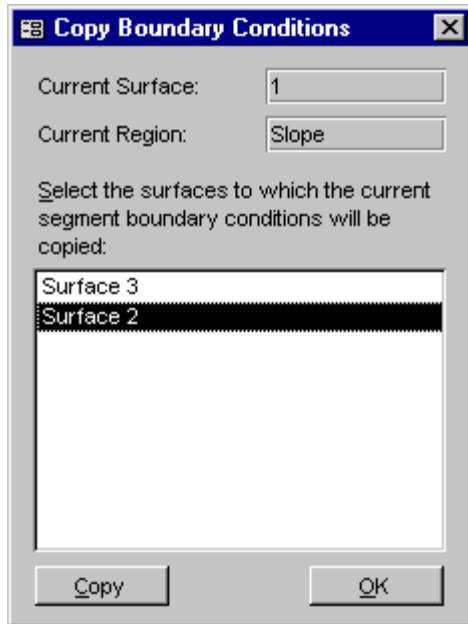
4. Consider the **Surface Boundary Condition** section at the top of the form.
5. From the **X Boundary Condition** drop down select a **Fixed** boundary condition.
6. From the **Y Boundary Condition** drop down select a **Fixed** boundary condition.
7. From the **Z Boundary Condition** drop down select a **Fixed** boundary condition.
8. Consider the **Segment Boundary Condition** section at the bottom of the form.
9. From the **X Boundary Condition** drop down select a **Fixed** boundary condition.
10. From the **Y Boundary Condition** drop down select a **Fixed** boundary condition.



Tip! The Fixed boundary condition for the point (0,0) becomes the boundary condition for the following line segments that have a Continue boundary condition until a new boundary condition is specified.

The boundary conditions for the slope region are to be the same over Layer 2 as Layer 1; therefore the Surface 1 segment boundary conditions can be copied to Surface 2:

1. In the Boundary Conditions form ensure **Surface 1** is current in the drop-down.
2. Press the **Copy To Surface** button to open the form:



3. Select **Surface 2** from the list.
4. Press **Copy**.
5. Click **OK** to close the form.
6. Close the Boundary Conditions form.

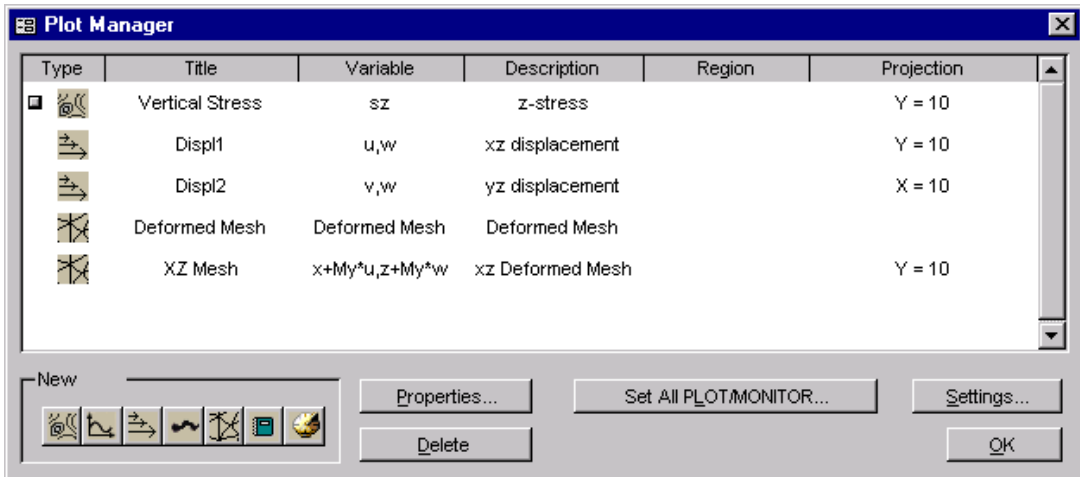
- **Pillar Region**

1. Select the "Pillar" region in the drawing space.
2. Select Surface 3 in the surface selector.
3. From the menu select **Model > Boundaries**. The boundary conditions form will open and display the boundary conditions for Surface 3.
4. Consider the **Surface Boundary Condition** section at the top of the form.
5. From the **Z Boundary Condition** drop down select a **Load Expression** boundary condition.
6. Enter a value of **-500** in the expression field.
7. Click **OK** to close the form.

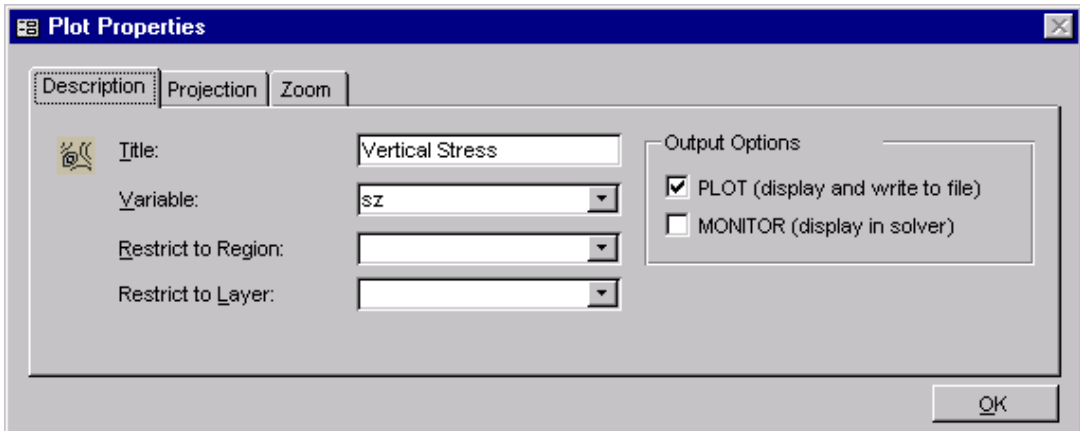
2.5 SPECIFY PLOTS

There are many plot types that can be specified to visualize the results of the model. A few will be generated for this tutorial example problem including vertical stress contours, deformed mesh, and displacement vectors.

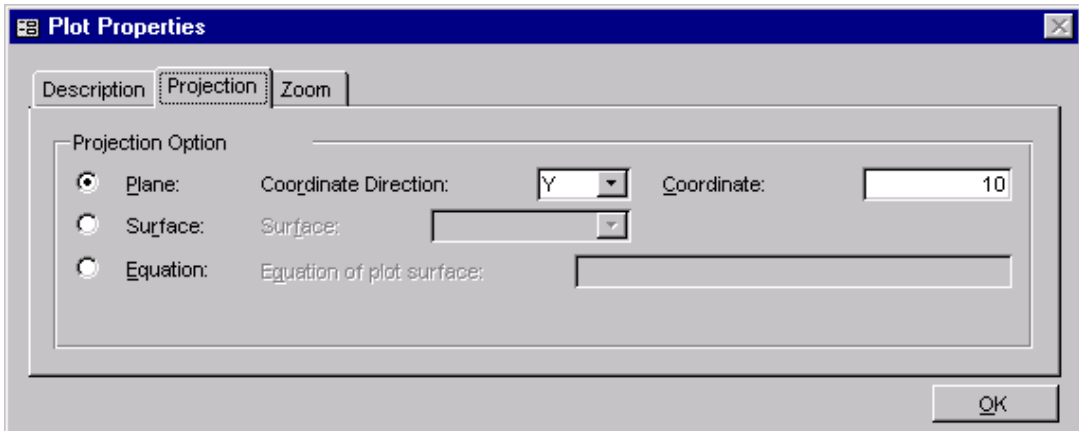
1. Open the Plot Manager form by selecting **Model > Plot Manager** from the menu.



2. 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.



3. Enter the title **Vertical Stress**.
4. Select **sz** as the variable to plot from the drop-down.
5. Move to the **Projection** tab.

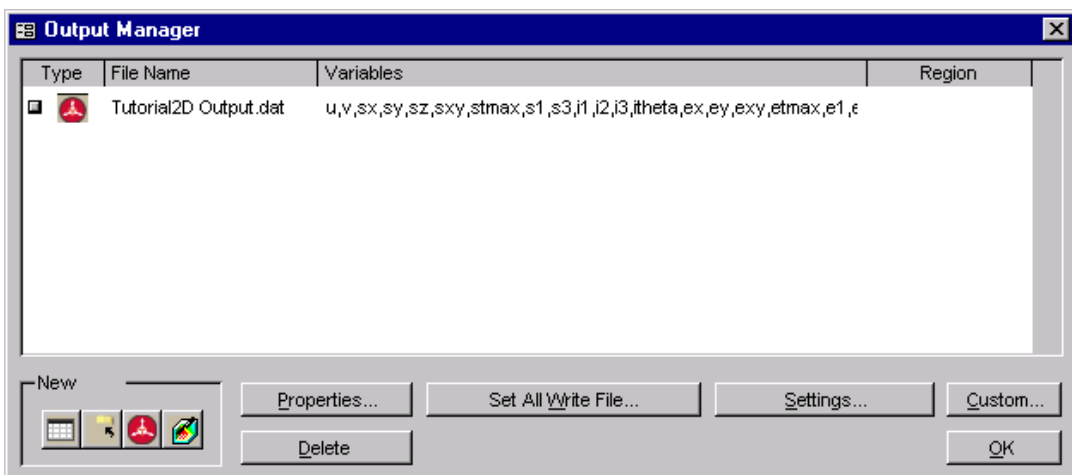


6. Select **Plane** Projection option.
7. Select **Y** from the Coordinate Direction drop-down.
8. Enter **10** in the Coordinate field. This will generate a 2D slice at $Y = 10\text{m}$ on which the stress contours will be plotted.
9. Select the **PLOT** output option.
10. Click **OK** to close the form and add the plot to the list.
11. Repeat these steps 2 – 10 to create the plots shown above.
12. Click **OK** to close the Plot Manager and return to the workspace.

2.6 SPECIFY OUTPUT FILES

There are 4 output file types that can be specified to export the results of the model. One will be generated for this tutorial example problem: a plot to output the results to AcuMesh.

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 AcuMesh button to begin adding the output file. The Output File Properties form will open.
3. Enter the title AcuMeshOut.
4. Select all the variables 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.

2.7 ANALYZE

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

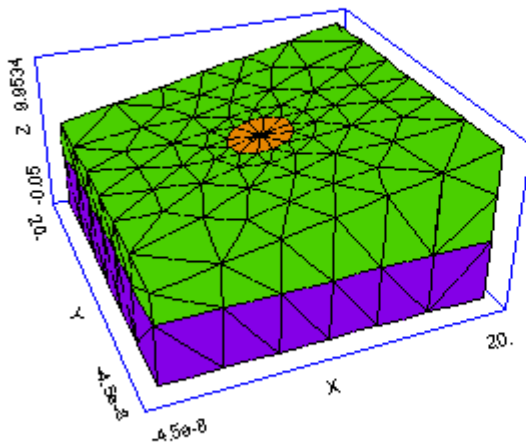
2.8 RESULTS

After the problem has finished solving, the results will be displayed in the form of thumbnail plots within the SVSOLID 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.

2.8.1 Solution Mesh

11:43:25 8/14/03
FlexPDE 3.10a1

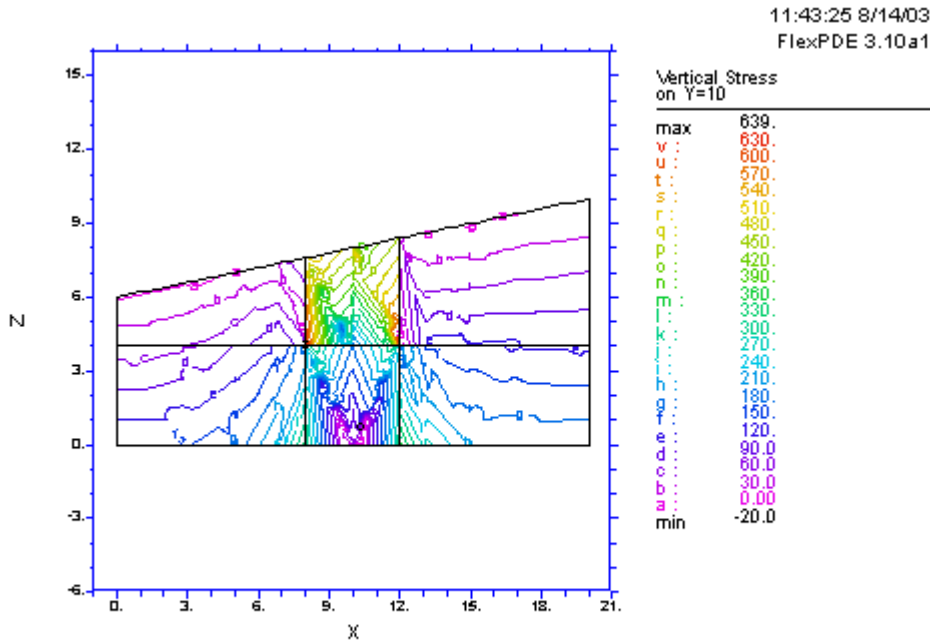
Deformed Mesh
(-24.4,-73.2, 30.)



Tutorial_Tutorial3D: Grid#1 p2 Nodes=1525 Cells=835 RMS Err= 3.2e-4

The Mesh plot displays the finite-element mesh generated by the solver. The mesh is automatically refined in critical areas. Right-click on the plot and select Rotate to enable the rotate window.

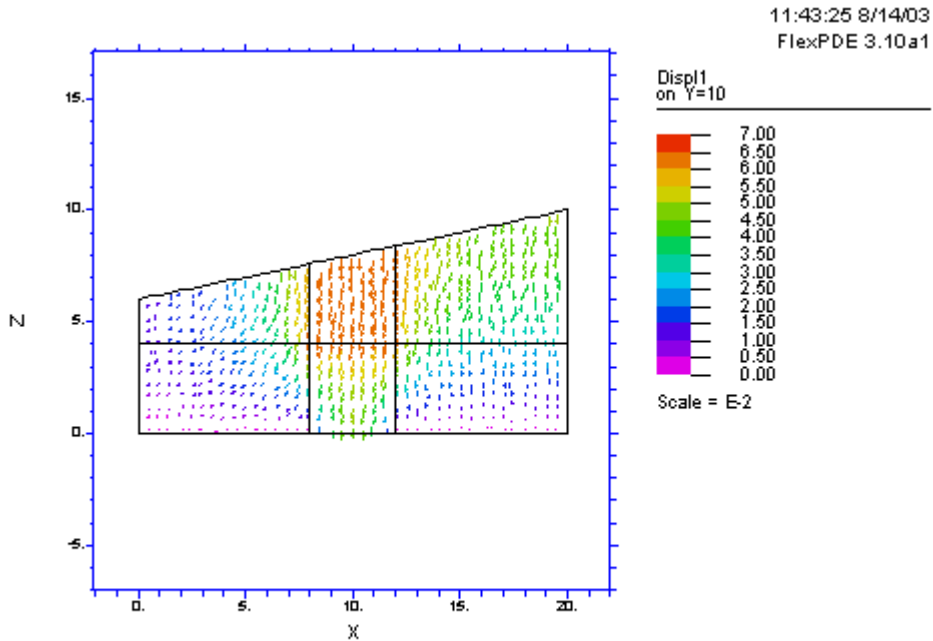
2.8.2 Stress Contours



Tutorial_Tutorial3D: Grid#1 p2 Nodes=1525 Cells=835 RMS Err= 3.2e-4
Integral= 22843.00

The stress state generated by the load on the pillar can be examined. Notice the effects of both vertical load and skin friction.

2.8.3 Displacement Vectors



Tutorial_Tutorial3D: Grid#1 p2 Nodes=1525 Cells=835 RMS Err= 3.2e-4

Displacement Vectors show both the direction and the magnitude of the displacement at specific points in the problem. The pillar displaces 0.07m due to the 500 kPa load.

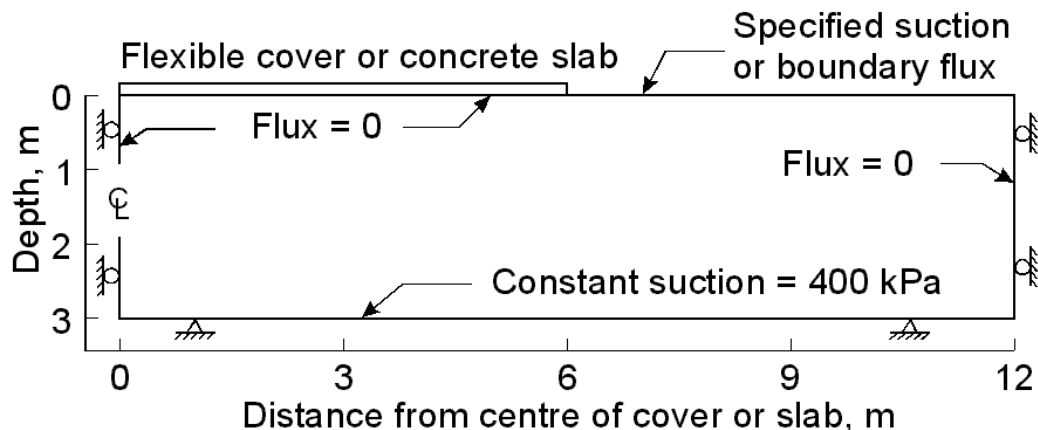
3 2D: EDGE DROP OF A FLEXIBLE IMPERVIOUS COVER

3.1 PROBLEM OVERVIEW

The following example will provide a step by step guide to modeling slab movement using a manual iteration technique involving the SVFLUX and SVSOLID software packages. There are two main scenarios to consider; the **Edge Drop** of a slab caused by shrinking of the soil due to evaporation (increase in soil suction) and **Edge Lift** of a slab caused by swelling due to infiltration (decrease in soil suction). This tutorial problem will consider the **Edge Drop** scenario.

- **Geometry and Boundary Conditions**

A 12-m wide flexible impervious cover in 2D is considered. Since the cover is symmetrical the portion to the right of the centerline will be modeled. A soil region 3m deep and 12m wide is used.



- **Soil Properties**

Seepage Soil Properties	Values
Coefficient of permeability at saturation, k_{sat}	1×10^{-8} m/s
Volumetric water content at saturation, θ_s	0.45
Parameters for SWCC (Fredlund & Xing, 1994) and permeability function (Leong and Rahardjo, 1997)	$a = 300$ kPa $n = 1.5$ $m = 1$ $hr = 3000$ kPa Sat. Suction = 0.1 kPa $\rho = 1$
Stress Soil Properties	Values

Total Unit Weight, γ_t	17.2 kN/m ²
Initial Void Ratio, e_o	1
Swelling Index, C_s	0.15
Swelling Index, C_m	0.13
Poisson's Ratio, μ_s	0.4
Coefficient of earth pressure at rest, K_o	0.33

• Solution Outline

This manual iteration method involves a number of steps to arrive at the final displacements. These example problems are included in the SVFLUX and SVSOLID databases for reference under the ProjectID of Tutorial. The ProblemID is indicated in the parentheses.

1. Initial SVFLUX seepage analysis (ED_Initial)
2. SVFLUX transient seepage analysis (ED_Transient)
3. SVSOLID stress/deformation analysis (ED_Day5)

3.2 INITIAL SVFLUX SEEPAGE ANALYSIS SETUP

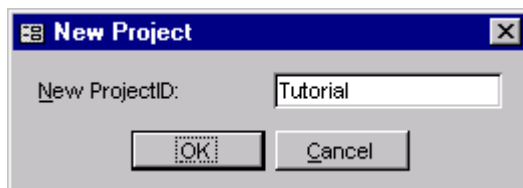
The purpose of the initial SVFLUX seepage analysis is to get an initial head profile to use as initial conditions for the SVFLUX transient seepage and to get an initial pore-water pressure profile to use as initial conditions for the SVSOLID stress/deformation analyses.

3.2.1 Adding an SVFLUX Project

The first step in defining this 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.

Follow these steps in order to add the project:

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 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.



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

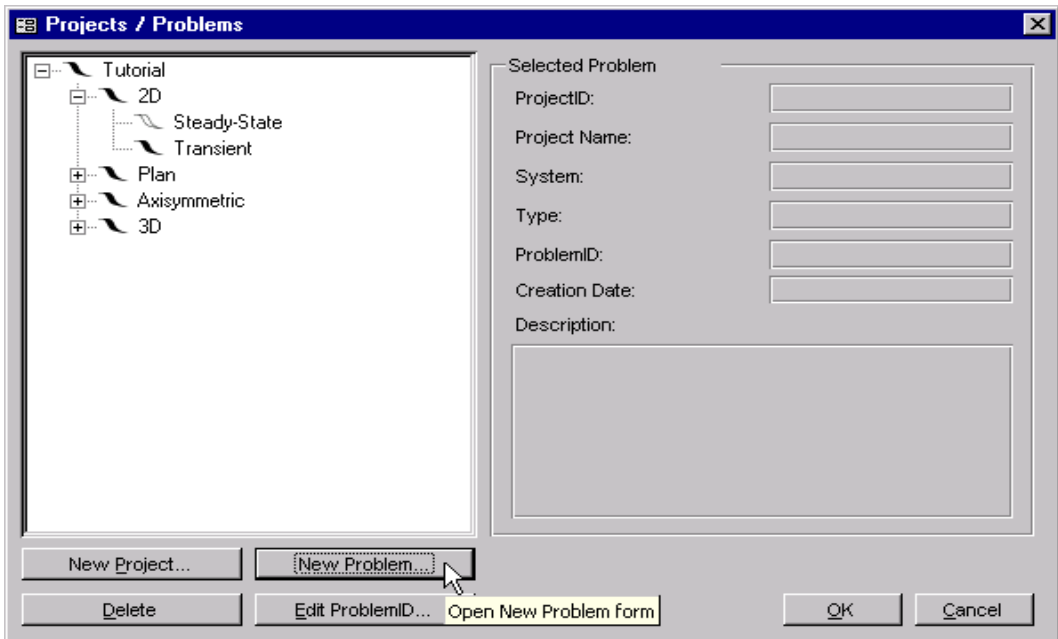
The form is opened ready to accept information.

Project	Client	Phone	Client Address	Contractor	Project Engineer
Project ID:	Tutorial				
Project Name:	SVFlux tutorials				
Project Location:	Saskatoon, SK Canada				
Project Start Date:	Jan 01/2003				
Project End Date:	Jan 02/2003				
Project Notes:	The tutorial example problems are stored under this project.				

It should be noted that once the project is defined it would be identified by the ProjectID throughout the rest of the program. Also, SVFLUX 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.

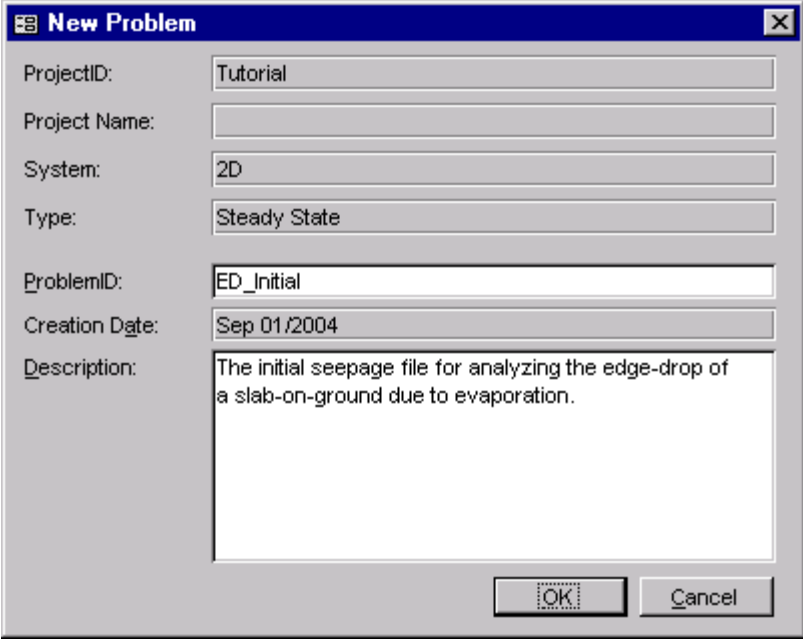
3.2.2 Adding the Initial SVFLUX Problem



Once a project has been created any number of problems can be created and 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 and is steady-state so we must expand **2D** by clicking the plus sign beside "2D"
3. Select **Steady-State**.
4. Click the **New Problem** button. The New Problem form will open.



New Problem

ProjectID: Tutorial

Project Name:

System: 2D

Type: Steady State

ProblemID: ED_Initial

Creation Date: Sep 01/2004

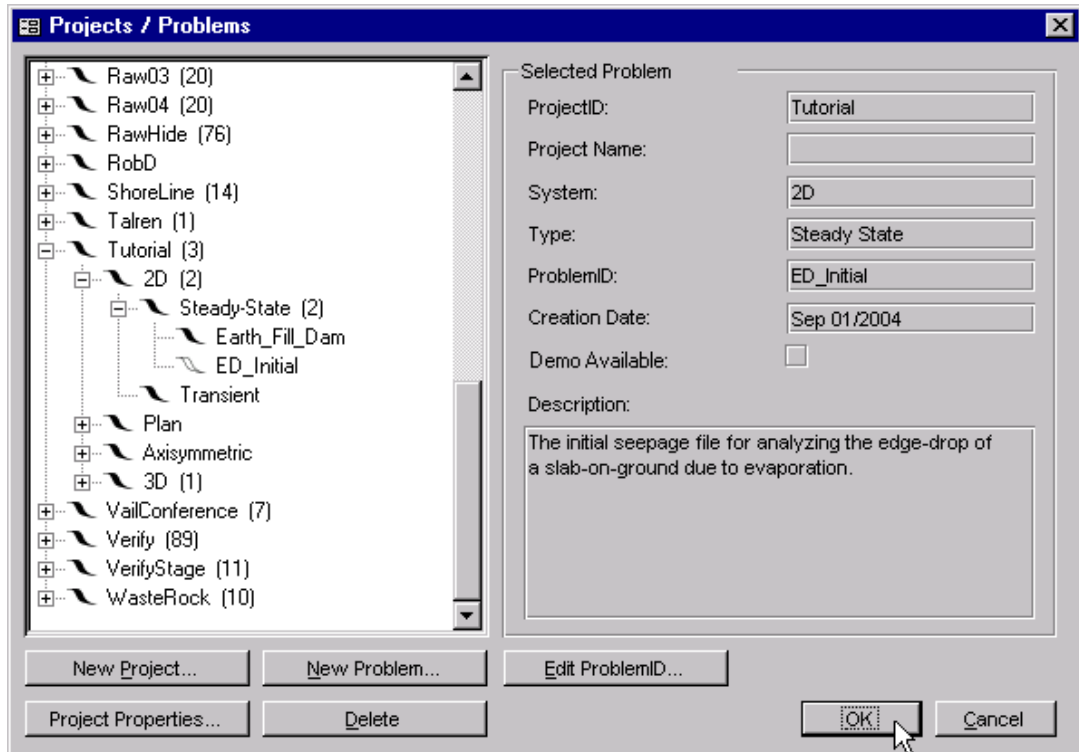
Description: The initial seepage file for analyzing the edge-drop of a slab-on-ground due to evaporation.

OK Cancel

5. Enter the **ProblemID: ED_Initial**. The description is optional.
6. Click the **OK button** to save the problem and close the New Problem form.
7. The new problem will automatically be opened in the workspace.

3.2.3 Opening the Initial SVFLUX Problem

If the problem was just added, it will already be open in the workspace. When returning to the problem, follow these steps to open it in the workspace:




1. Select Model > Projects/Problems from the menu.
2. **Navigate** back to the problem via Tutorial, 2D, Steady-State.
3. Select **ED_Initial**.
4. The problem may be opened by clicking the **OK** button or by double clicking on the ProblemID in the tree control.

3.3 DEFINING THE INITIAL SVFLUX PROBLEM

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

3.3.1 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:

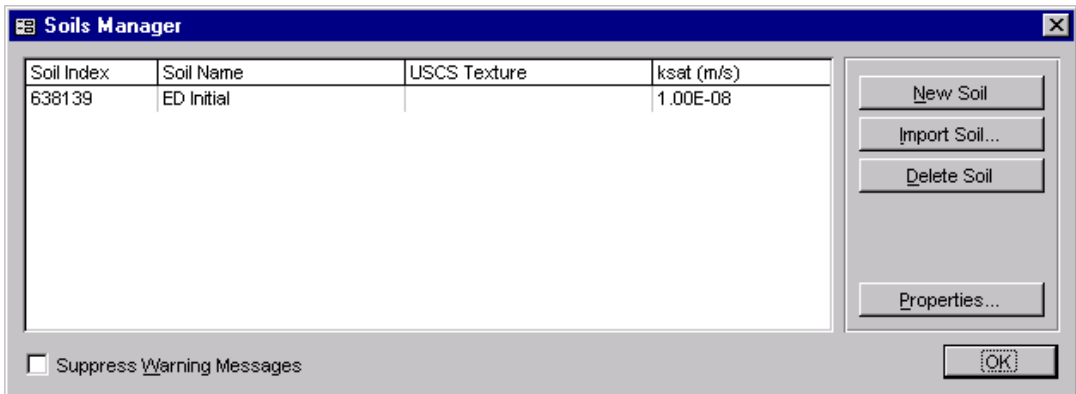
Global Coordinate Offset

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.

3.3.2 Define Material Properties



The next step in defining the problem is to enter the material properties for the soil that will be used in the model.

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 SoilIndex is generated that is used to reference the soil in other SVFLUX forms.
3. Double-click on the new soil to open the **Soil Properties** form.

The screenshot shows the 'Soil Properties' dialog box with the 'Description' tab selected. The fields are filled as follows:

- Soil Index: 638139
- Soil Name: ED Initial
- USCS Texture: (empty dropdown)
- Soil Description: (empty text box)
- Geologic Description: (empty text box)
- Notes: This soil will define the region for the edge-drop initial conditions analysis

Buttons at the bottom include 'Stage Parameters...' and 'OK'.

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


The screenshot shows the 'Saturated Hydraulic Conductivity' dialog box. The 'ksat' field is set to 1.000E-08 m/s. There is an unchecked 'Staged' checkbox.

6. Refer to the data provided at the beginning of this tutorial. Enter the k_{sat} value of 1.000E-08 m/s

3.3.3 Adding a Region

A region in SVFLUX 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 SVFLUX CAD workspace. A region will have a set of geometric shapes that define its soil boundaries. Also, other modeling objects including features, flux sections, water tables, text, and line art are defined on any given region.

This problem will consist of only one region, which is named Ground. Follow these steps to add the region:

1. Open the regions form by clicking the Regions button,  at the top of the workspace.
2. One region is present by default. Change the region name from Region 1 to **Ground**. Highlight the name and type new text.
3. Select the **SoilIndex** from the drop-down.

4. Select the box Display Boundary Condition Graphics. This will display graphical representations for the boundary conditions when they are defined later in this tutorial.
5. Click **OK** to close the form.


3.3.4 Defining Region Geometry Shapes

The shape that defines the 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.

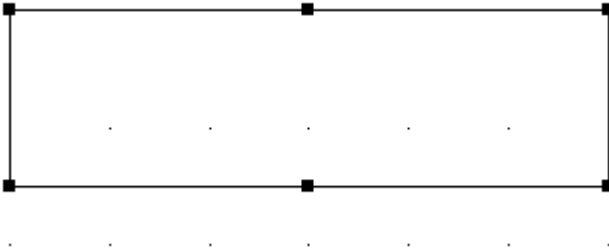
Region Geometry


X	Y
0	-3
6	-3
12	-3
12	0
6	0
0	0

The instructions below explain the use of the command line to create the ground shape. Alternately, the ground shape could be drawn with the mouse or the data pasted into the Region Properties form.

1. Ensure that “Ground” 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,-3** and press the Enter key on the keyboard.
5. Type **6,-3** and press Enter.
6. Type **12,-3** and press Enter.
7. Type **12,0** and press Enter.
8. Type **6,0** and press Enter.
9. Type **0,0** and press Enter.
10. Type **f** and press **Enter** to complete the region shape.

If the geometry has been entered correctly the shape should look like the following:



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.

3.3.5 Specify Boundary Conditions

Now that the region and the problem geometry have been successfully defined, the next step is to specify the boundary conditions. A suction of 20 kPa is required at the ground surface while a suction of 400 kPa exists at the bottom of the soil region. The suction values must be converted to head values for SVFLUX entry. The steps for specifying the boundary conditions are thus:

1. Select the “Ground” region in the drawing space.
2. From the menu select **Model > Boundaries**. The boundary conditions form will open. By default the first boundary segment will be given a Zero Flux condition.

Boundaries

Region: Ground Select Shape Index: 718539290

X	Y	Boundary Condition	Expression or Data	Units
0	-3	Head Expression	-43.787	m
6	-3	Continue		
12	-3	Zero Flux		
12	0	Head Expression	-2.039	m
6	0	Continue		
0	0	Zero Flux		

Update Selected Segment Segment Length: 6 m

1. Select Boundary Condition: Head Expression m

2. Provide: a) Expression: -2.039 Build Equation...
 - or -
 b) Flux Data Index: Expr Reference...
 c) Evaporation Index: (optional)

3. Update

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

OK

3. Select the **point** (0,-3) from the list.
4. From the **Boundary Condition** drop down select a **Head Expression** boundary condition. This will cause the Expression box to be enabled.
5. In the Expression box enter a **head of -43.787**.
6. Click the **Update** button to save the boundary condition to the list.
7. Select the **point** (12,-3) from the list.
8. From the **Boundary Condition** drop down select a **Zero Flux** boundary condition.
9. Click the **Update** button to save the boundary condition to the list.
10. Select the **point** (12,0) from the list.
11. From the **Boundary Condition** drop down select a **Head Expression** boundary condition. This will cause the Expression box to be enabled.
12. In the Expression box enter a **head of -2.039**.
13. Click the **Update** button to save the boundary condition to the list.
14. Select the **point** (0,0) from the list.
15. From the **Boundary Condition** drop down select a **Zero Flux** boundary condition.
16. Click the **Update** button to save the boundary condition to the list.



Tip!

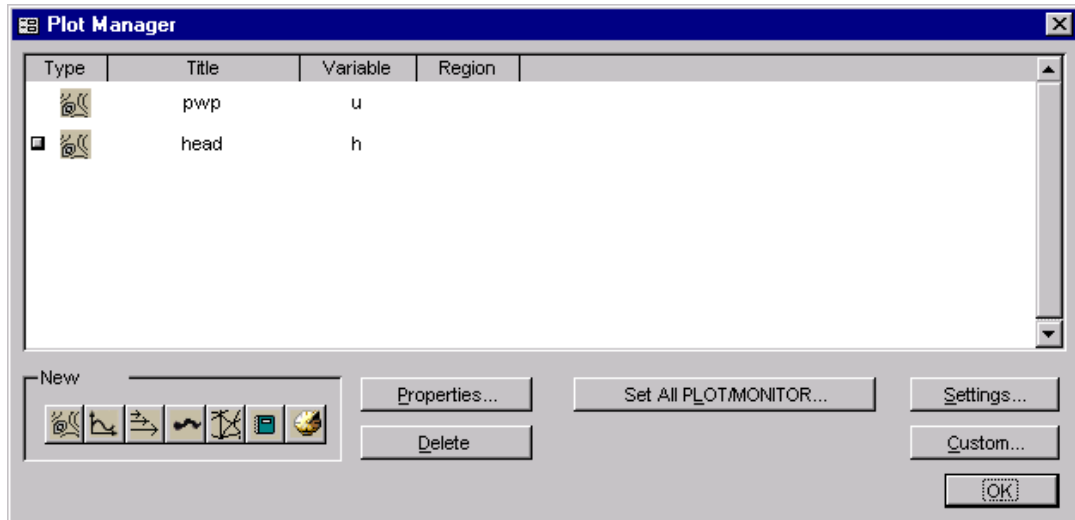
The Continue boundary condition indicates that the previously defined boundary condition will apply to the current boundary segment.

17. Click **OK** to return to the workspace.

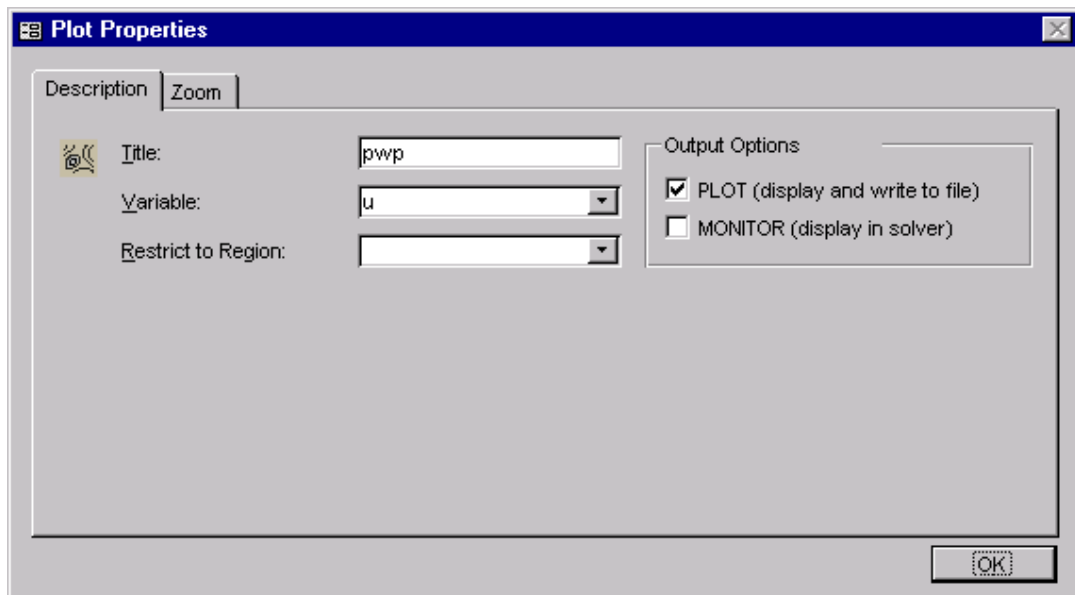
3.4 SPECIFY INITIAL SVFLUX ANALYSIS PLOTS

There are many plot types that can be specified to visualize the results of the model. A contour plot of pore-water pressure and a contour plot of head will be generated for this example.

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



2. 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.

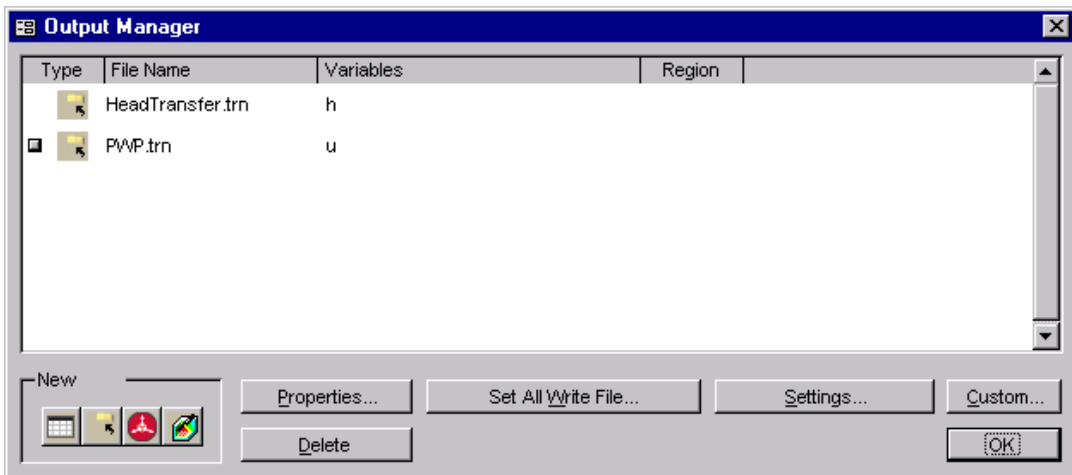


3. Enter the title pwp.
4. Select u as the variable to plot from the drop-down.
5. Select the **PLOT** output option.
6. Click **OK** to close the form and add the plot to the list.
7. Repeat these steps 2 – 6 to create the head plot.
8. Click **OK** to close the Plot Manager and return to the workspace.

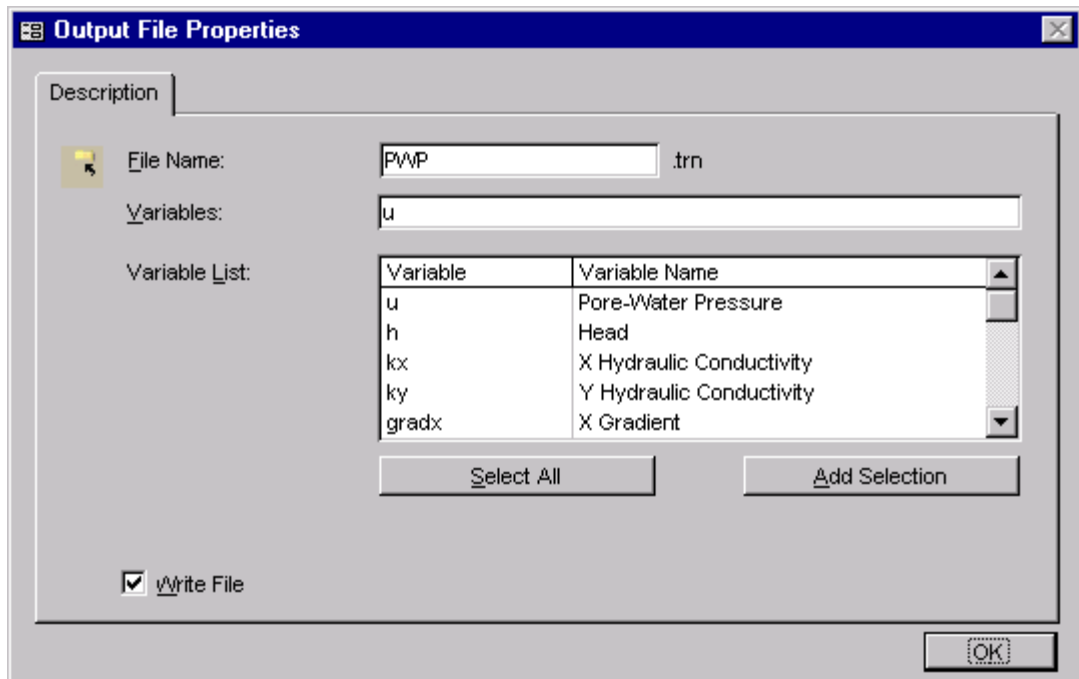
3.5 SPECIFY INITIAL SVFLUX ANALYSIS OUTPUT FILES

There are 4 output file types that can be specified to export the results of the model. Two transfer files will be generated for this tutorial example problem: a transfer file of pore-water pressures, and a transfer file of heads. Note that the file HeadTransfer.trn is already present. It is generated by default for every problem.

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 **Transfer** button to begin adding the output file. The Output File Properties form will open.



3. Enter the title PWP.
4. Select the variable u 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.

3.6 ANALYZE THE INITIAL SVFLUX PROBLEM

The next step is to analyze the problem. Click the Analyze button located on the left of the workspace. This action will write the descriptor file and open the SVFLUX solver. The solver will automatically begin solving the problem and the Run Log form will open in SVFLUX.

When the solver has finished running press the **Read File** button on the Run Log form to record the run data.

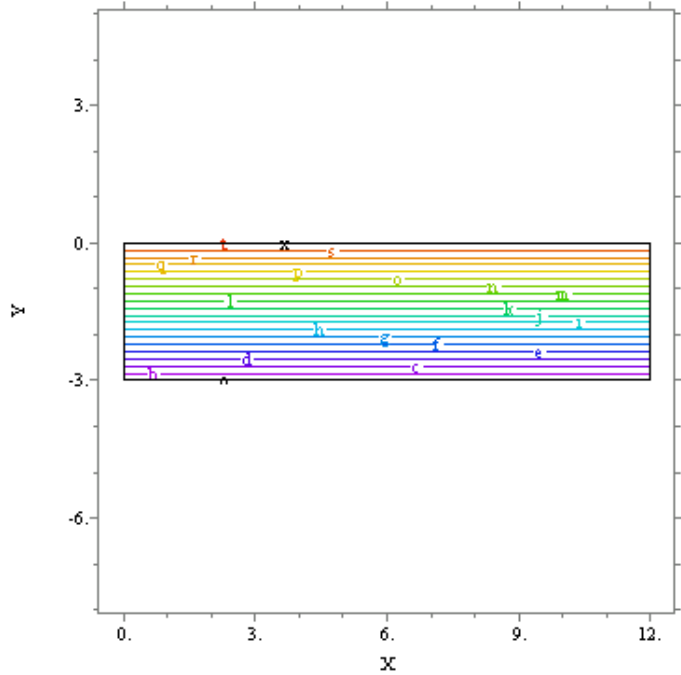
3.7 INITIAL SVFLUX ANALYSIS RESULTS

After the problem has finished solving, the results will be displayed in the form of thumbnail plots within the SVFLUX solver. Right-click the mouse and select Maximize to enlarge any of the thumbnail plots.

The output files requested (PWP.trn and HeadTransfer.trn) will be located in the solution file directory for the problem.

T

16:06:52 9/1/04
FlexPDE 4.0.7b



pwp	
max	-20.0
t	-20.0
s	-40.0
r	-60.0
q	-80.0
p	-100.0
o	-120.0
n	-140.0
m	-160.0
l	-180.0
k	-200.0
j	-220.0
i	-240.0
h	-260.0
g	-280.0
f	-300.0
e	-320.0
d	-340.0
c	-360.0
b	-380.0
a	-400.0
min	-400.0

Tutorial_ED_Initial: Grid#1 p2 Nodes=293 Cells=128 RMS Err= 6.4e-9
Stage 1 Integral= -7559.902

The contour plot of pore-water pressure above indicates -20 kPa at the ground surface and a decrease with depth to -400 kPa at the bottom.

3.8 SVFLUX TRANSIENT SEEPAGE ANALYSIS SETUP

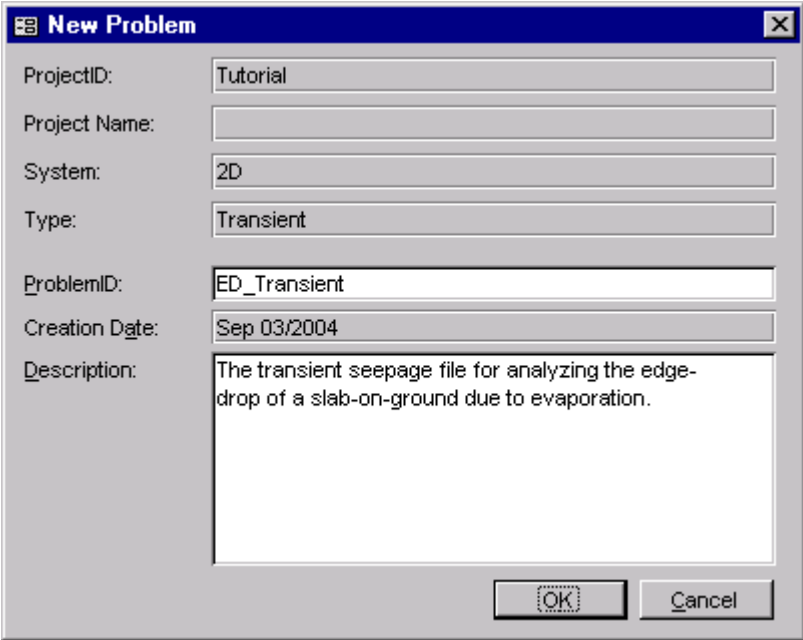
The SVFLUX transient seepage analysis will use the initial head transfer file as initial conditions. This step will output a pore-water pressure transfer (.trn) file for the SVSOLID stress/deformation analysis.

3.8.1 Adding the SVFLUX Transient Problem

A project was created previously (Tutorial) and the SVFLUX transient problem can be entered beneath this project.

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

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



The screenshot shows a 'New Problem' dialog box with the following fields and values:

- ProjectID: Tutorial
- Project Name: (empty)
- System: 2D
- Type: Transient
- ProblemID: ED_Transient
- Creation Date: Sep 03/2004
- Description: The transient seepage file for analyzing the edge-drop of a slab-on-ground due to evaporation.

Buttons: OK, Cancel

5. Enter the **ProblemID: ED_Transient**. The description is optional.
6. Click the **OK button** to save the problem and close the New Problem form.
7. The new problem will automatically be opened in the workspace.

3.8.2 Opening the Transient SVFLUX Problem

If the problem was just added it will already be open in the workspace. When returning to the problem refer to the steps in the section [Opening the Initial SVFLUX Problem](#) to open it in the workspace.

3.9 DEFINING THE TRANSIENT SVFLUX PROBLEM

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

3.9.1 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 screenshot shows the 'Settings' dialog box with the 'General' tab selected. The 'System' section has radio buttons for '2D' (selected), 'Plan', 'Axisymmetric', and '3D'. The 'Type' section has radio buttons for 'Steady-State' and 'Transient' (selected). The 'Units' section has radio buttons for 'Metric' (selected) and 'Imperial'. Below these are input fields for 'Time' (set to 'day'), 'Length' (set to 'm'), 'Force' (set to 'kN'), 'Pressure' (set to 'kPa'), and 'Conductivity' (set to 'm/day'). A 'Solution Files Path...' button is at the bottom left, and an 'OK' button is at the bottom right. A note at the bottom states: 'The above choices are read only. Problem choices are set when a new problem is created.'

The Settings form will contain information about the current problem System, Type, Units, and Transient Settings.

1. Select **Metric** Units.
2. The problem is defined in terms of days. Select **day** from the drop-down control.
3. Move to the Time tab.
4. Set the **Start Time as 0**, the **Time Increment as 1**, and the **End Time as 5** days.
5. Move to the Initial Conditions tab.

File Path

O:\Data_J\My_Problems\Tutorial\2D\SteadyState\ED_Initial\HeadTransfer.trn

Browse...

Clear

Re-link

6. Click the **Browse** button and specify the path to the **HeadTransfer.trn** file generated by the ED_Initial problem.
7. Click OK to close the Settings form.

3.9.2 Define Material Properties

Soil Index	Soil Name	USCS Texture	ksat (m/day)
292531	ED Transient		8.64E-04

New Soil

Import Soil...


Delete Soil

Properties...

Suppress Warning Messages

OK

The next step in defining the problem is to enter the material properties for the soil that will be used in the model.

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 SVFLUX forms.
3. Double-click on the new soil to open the **Soil Properties** form.

Soil Properties

Description | Hydraulic Conductivity | SWCC | Volume-Mass Parameters | Vapour Diffusion | Sink/Source

Soil Index: 292531

Soil Name: ED Transient

USCS Texture: [Dropdown]

Soil Description: [Text Box]

Geologic Description: [Text Box]

Notes: This soil has the properties required for the transient analysis of the edge-drop tutorial example.

OK

4. Enter the information above into the appropriate fields on the **Description** tab. (The suggested name is ED Transient, but both the name and notes are optional)
5. Move to the **SWCC** tab.
6. Refer to soil properties table in the [problem overview](#) section of this manual. Enter the **Saturated VWC** value of 0.45.
7. Select the **Fredlund and Xing Fit** as the SWCC option.
8. Press the button to the right of the Fredlund & Xing Fit option to open the form below.

Fredlund and Xing Fit

af: kPa

nf:

mf:

hr: kPa

Fredlund SWCC Fit:

Fredlund Error: R²

Fredlund Residual WC: (volumetric water content)

Fredlund AEV: kPa

Saturated Conditions

Saturation Suction: kPa

Saturated WWC:

Coefficient of Volume Change, mw: 1/kPa

Messages:

Apply Fit Graph SWCC... Graph Storage... Properties...

Iteration: Log Suction

9. Enter the fit parameters from the soil properties table.
10. Click OK to close the form.
11. Move to the **Hydraulic Conductivity** tab.
12. Enter the **Saturated Hydraulic Conductivity** value of 6.64E-04 m/day.
13. Select the **Leong and Rahardjo** unsaturated hydraulic conductivity option.
14. Press the button to the right of the Leong and Rahardjo option to open the **Leong and Rahardjo Estimation** form.
15. Enter the Leong p value of 1.

Leong and Rahardjo Estimation

Leong p:

Leong Predicted:

Leong Error: Exponential R²

k minimum: m/day

Apply k minimum:

16. Click OK to close the form.
17. Click OK to close the Soil Properties form.

3.9.3 Importing Geometry

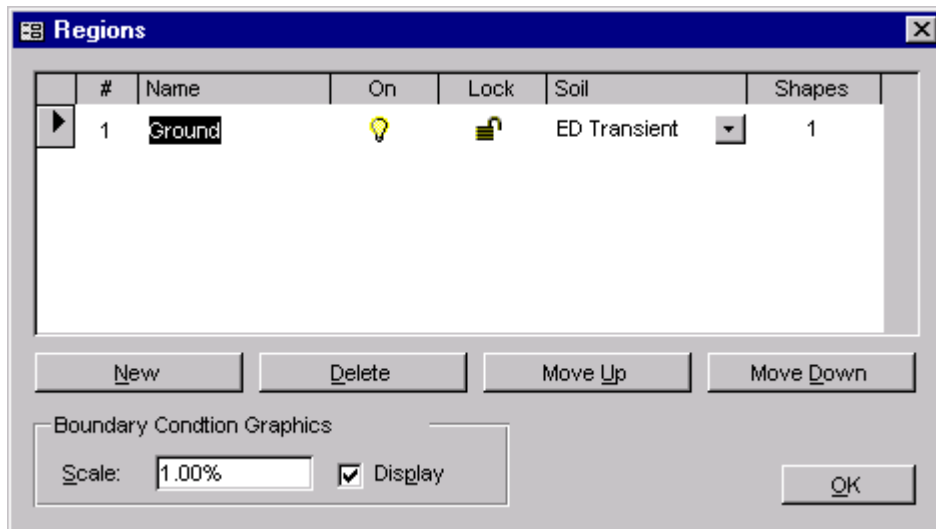
Since the Ground region and its geometry were defined previously for the Initial SVFLUX Analysis, the **Import SVFLUX Geometry** feature can be used to save time for this analysis.

1. Select **Model > Import SVFLUX Geometry** from the menu.
2. Press the **Browse** button and specify the path to the current **SVFlux_Data_J.mdb** database. The tree view control will be filled with all the problems in the database.
3. Navigate to the **ED_Initial** problem. (Tutorial – 2D – Steady-State – ED_Initial)
4. Click the **Import** button.
5. Say Yes to the warning message.
6. The Ground region, region shape, and world coordinate system settings will be imported.
7. Click **OK** to close the import form.

3.9.4 Assign the Region Soil

The soil that was previously defined will need to be assigned to the Ground region that was just imported.

1. Select **Model > Regions** from the menu to open the Regions form.



2. Select the ED_Transient soil from the Soil drop-down.
3. Click OK to close the Regions form.

3.9.5 Specify Boundary Conditions

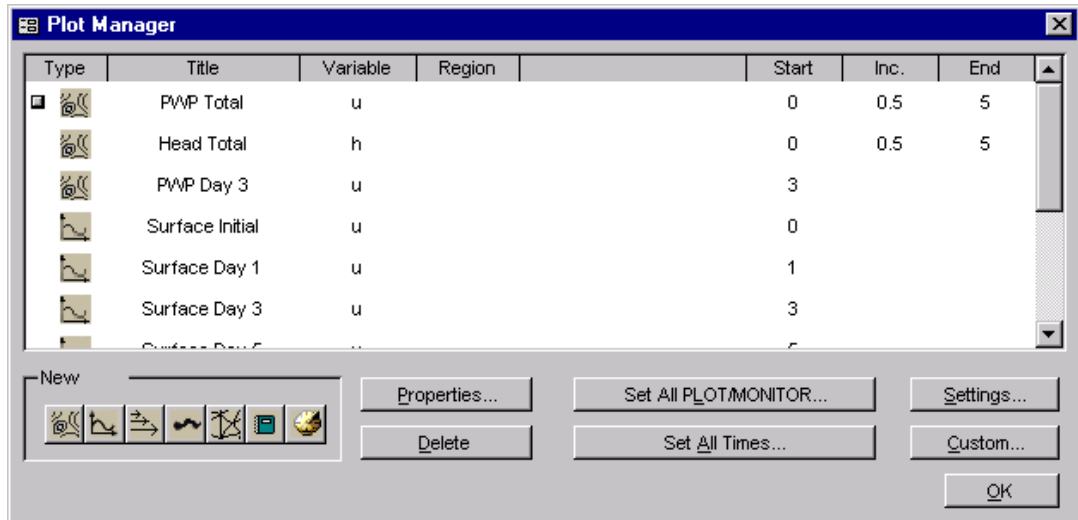
Now that the region and the problem geometry have been successfully imported, the next step is to specify the boundary conditions. A zero flux condition is required at the ground surface beneath the cover while a suction of 400 kPa exists at the bottom of the soil region. The suction values must be converted to head values for SVFLUX entry. An evaporation rate of 10 mm/day is represented by the normal flux condition over the uncovered ground surface. The steps for specifying the boundary conditions are thus:

1. Select the "Ground" region in the drawing space.
2. From the menu select **Model > Boundaries**. The boundary conditions form will open. By default the first boundary segment will be given a Zero Flux condition.

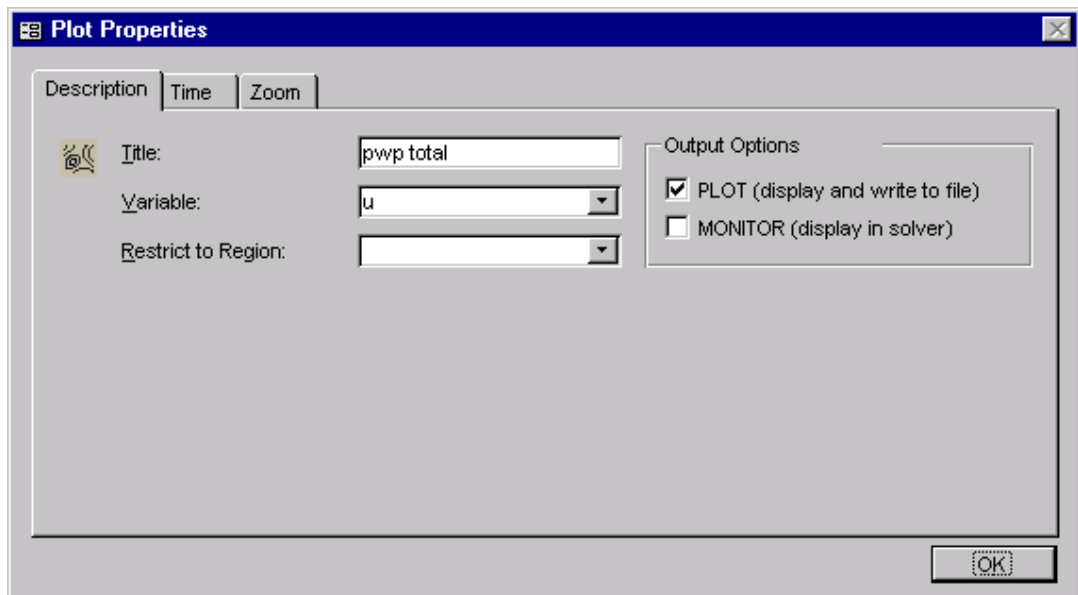
3.10 SPECIFY SVFLUX TRANSIENT ANALYSIS PLOTS

There are many plot types that can be specified to visualize the results of the model. A few contour and elevation plots will be generated for this tutorial example problem.

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



2. 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.



3. Enter the title pwp total.
4. Select **u** as the variable to plot from the drop-down.
5. Select the **PLOT** output option.
6. Move to the **Time** tab
7. Enter a Start Time of 0, a Time Increment of 0.5, and an End Time of 5 days.
8. Click OK to close the form and add the plot to the list.
9. Repeat these steps 2 – 8 to create the suggested contour plots listed below. (Note that the plots are not required for problem solution, but are useful for visualization)

10. Click on the **Elevation** button to begin adding the first elevation plot. The Plot Properties form will open.
11. Enter the title Surface Initial.
12. Select **u** as the variable to plot from the drop-down.
13. Select the **PLOT** output option.
14. Move to the **Time** tab
15. Enter a **Start** Time of 0.

16. Move to the **Range** tab.
17. Enter the values – **X1: 0, Y1: 0, X2: 12, Y2: 0.**

18. Click OK to close the form and add the plot to the list.
19. Repeat these steps 10 – 18 to create the suggested elevation plots listed below. (Note that the plots are not required for problem solution, but are useful for visualization)
20. Click **OK** to close the Plot Manager and return to the workspace.

Suggested Plots

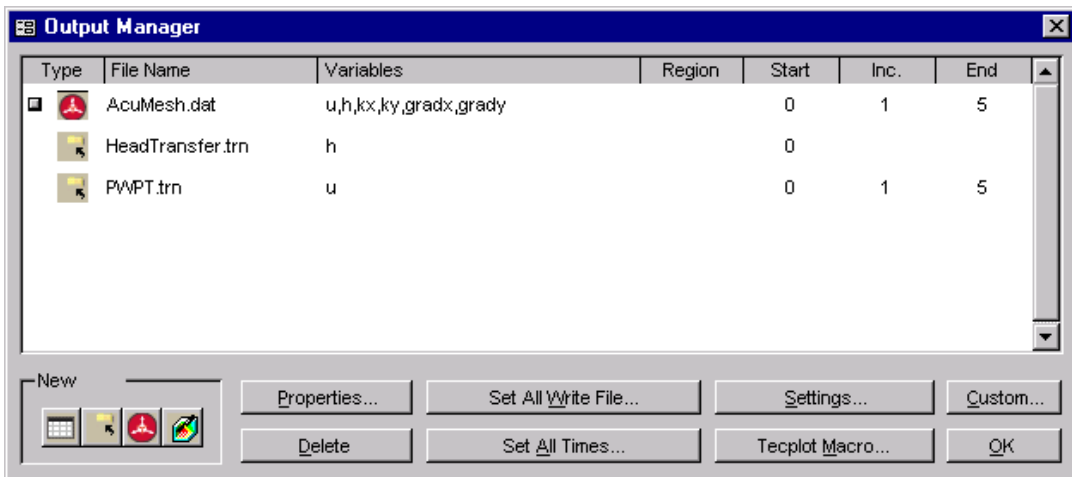
Plot Type	Title	Variable	Time			Range
			Start	Inc	End	
Contour	PWP Total	u	0	0.5	5	
Contour	Head Total	h	0	0.5	5	
Contour	PWP Day 3	u	3			

Elevation	Surface Initial	u	0	(0,0) to (12,0)
Elevation	Surface Day 1	u	1	(0,0) to (12,0)
Elevation	Surface Day 3	u	3	(0,0) to (12,0)
Elevation	Surface Day 5	u	5	(0,0) to (12,0)
Elevation	Depth Initial	u	0	(6,0) to (6,-3)
Elevation	Depth Day 1	u	1	(6,0) to (6,-3)
Elevation	Depth Day 3	u	3	(6,0) to (6,-3)
Elevation	Depth Day 5	u	5	(6,0) to (6,-3)

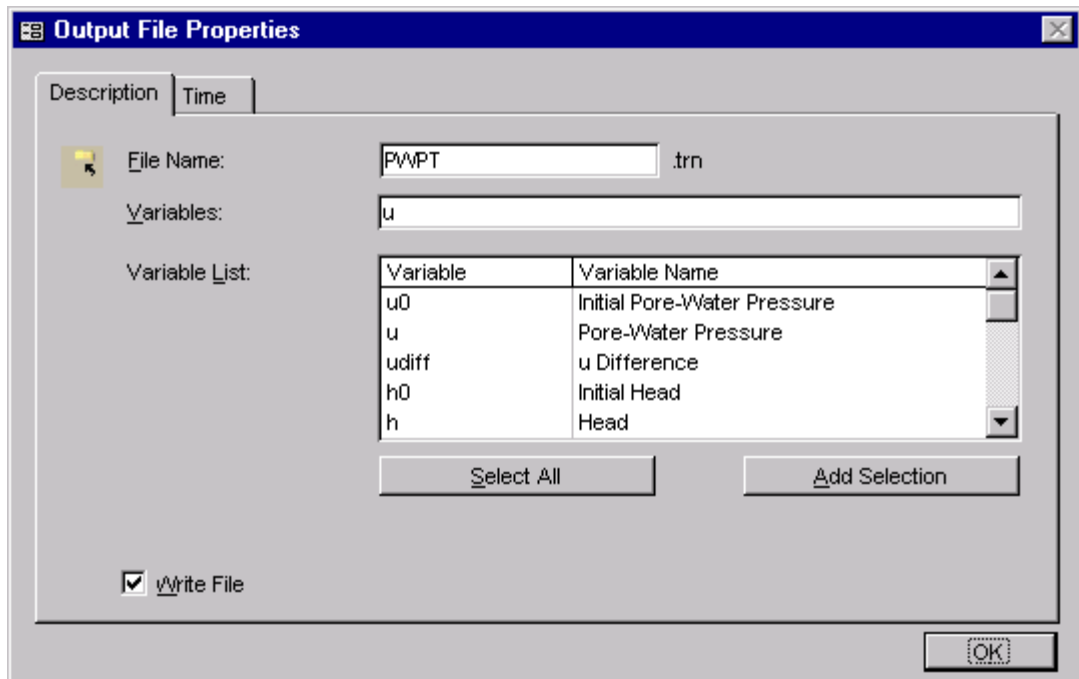
3.11 SPECIFY SVFLUX TRANSIENT ANALYSIS OUTPUT FILES

There are 4 output file types that can be specified to export the results of the model. Two will be generated for this tutorial example problem: a transfer file of pore-water pressure for use in the SVSOLID Analysis, and a plot to transfer the results to AcuMesh for advanced visualization. Note that the file HeadTransfer.trn is already present. It is generated by default for every problem.

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 **Transfer** button to begin adding an output file. The Output File Properties form will open.



3. Enter the title PWPT.
4. Select the variable **u** 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. Repeat steps 2 – 7 to add the AcuMesh output file.
9. Click **OK** to close the Output Manager and return to the workspace.

3.12 ANALYZE THE TRANSIENT SVFLUX PROBLEM

The next step is to analyze the problem. Click the Analyze button located on the left of the workspace. This action will write the descriptor file and open the SVFLUX solver. The solver will automatically begin solving the problem and the Run Log form will open in SVFLUX.

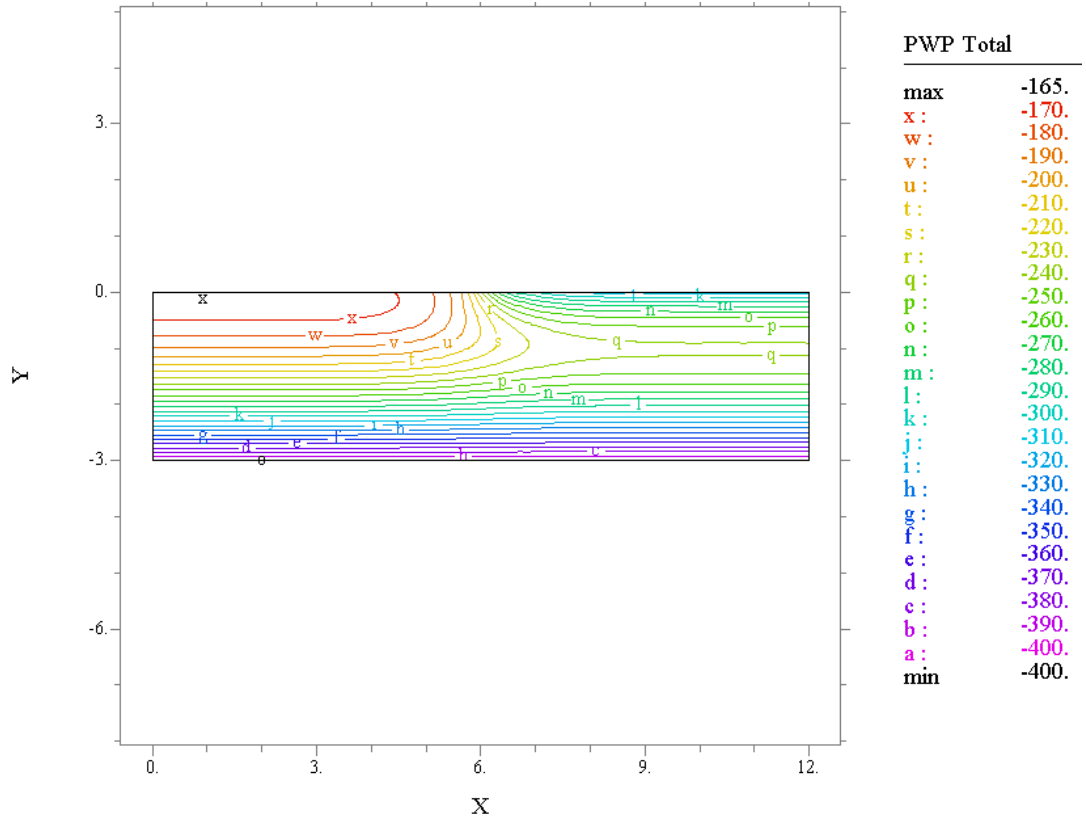
When the solver has finished running press the **Read File** button on the Run Log form to record the run data.

3.13 SVFLUX TRANSIENT ANALYSIS RESULTS

After the problem has finished solving, the results will be displayed in the form of thumbnail plots within the SVFLUX solver. Right-click the mouse and select Maximize to enlarge any of the thumbnail plots. This section will give a brief analysis for a few plots that were generated.

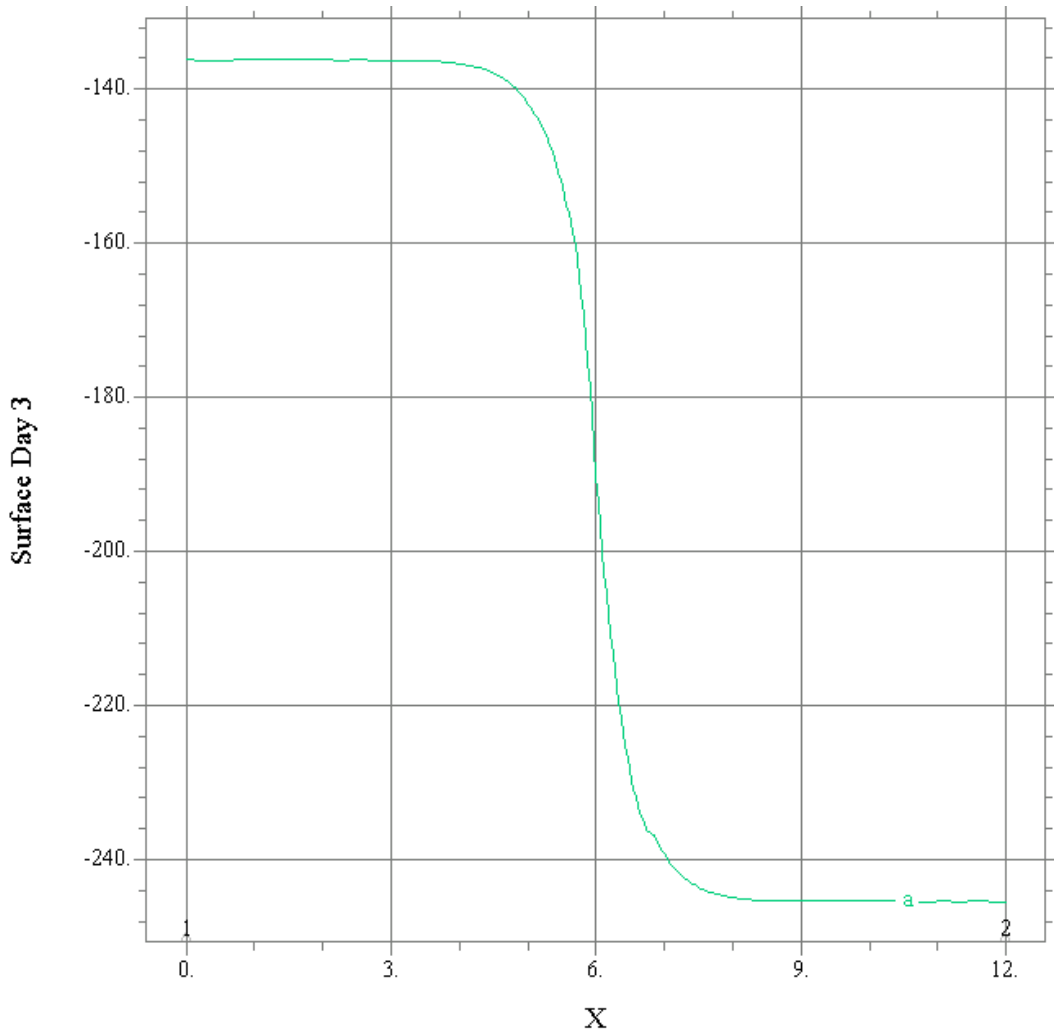
The output files requested (PWPT.trn, HeadTransfer.trn, and AcuMesh.dat) will be located in the solution file directory for the problem.

3.13.1 Pressure Contours

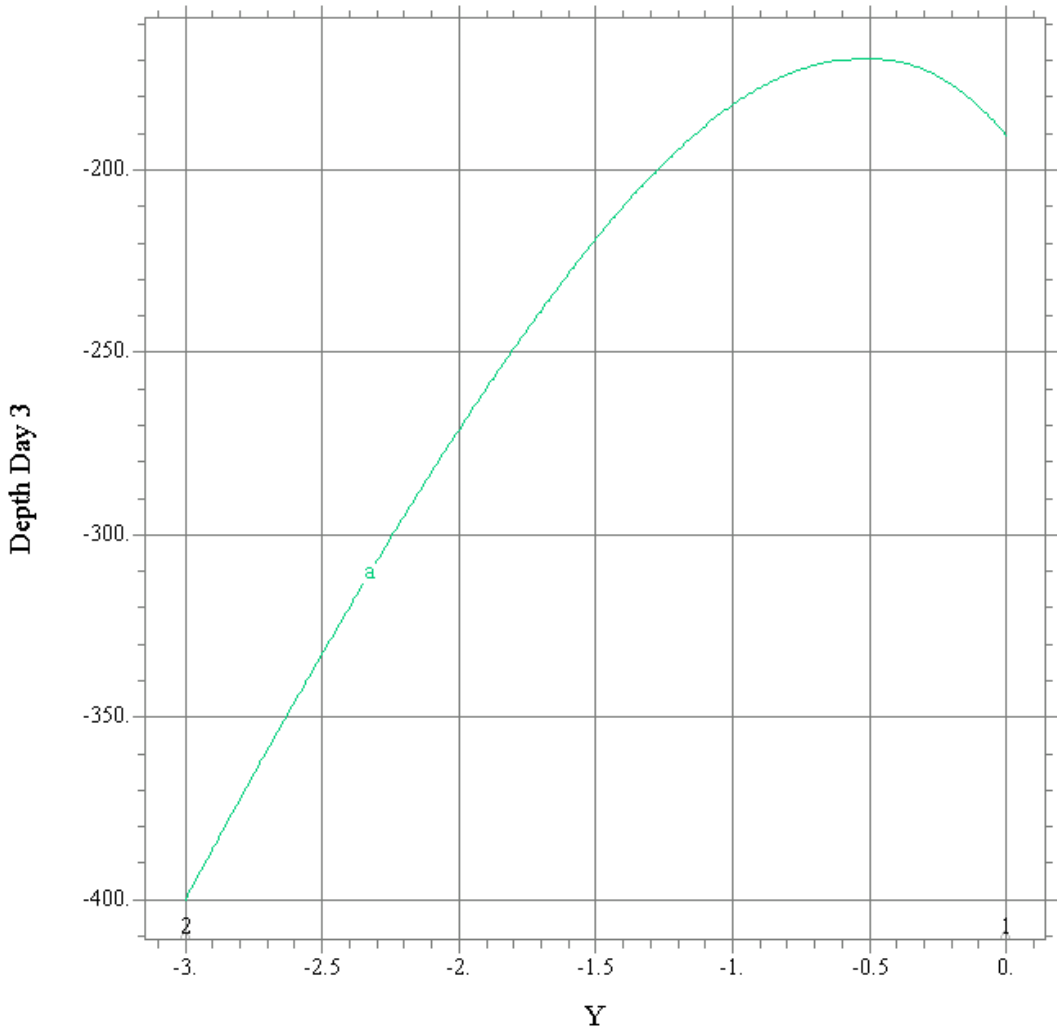


The pore-water pressure contour plot above indicates increased suction near the un-covered ground surface due to the evaporation in that area.

3.13.2 Pore-water Pressure Elevation Profiles



The plot above shows the pore-water pressure along the ground surface after 3 days of evaporation. There is a high suction gradient within 1 meter of the outside edge of the cover and the suction change is uniform elsewhere. In the plot below the pore-water pressure profile below the cover edge after 3 days of evaporation is shown. The majority of the suction change occurs near the ground surface. When compared to the plots for 1 and 5 days of evaporation it can be seen that the suction change advances deeper with time.



3.14 SVSOLID STRESS/DEFORMATION ANALYSIS

Now that the seepage component of the problem has been completed, a stress analysis must be defined using SVSOLID. This analysis will use the initial pore-water pressure transfer file from the Initial SVFLUX Analysis and the final pore-water pressure transfer file from the SVFLUX Transient Analysis. This stress analysis will be run for 25 stages and the displacements calculated and output at each stage. These incremental displacements will be summed to obtain the total movements using a summary file.



Tip!

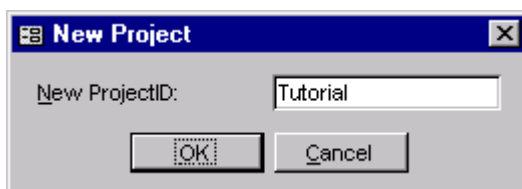
Note that the operation of the SVSOLID software is similar to SVFLUX. Many of the following steps will be the same.

3.14.1 Adding an SVSOLID 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.

Follow these steps in order to add the project:

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 ProjectID 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

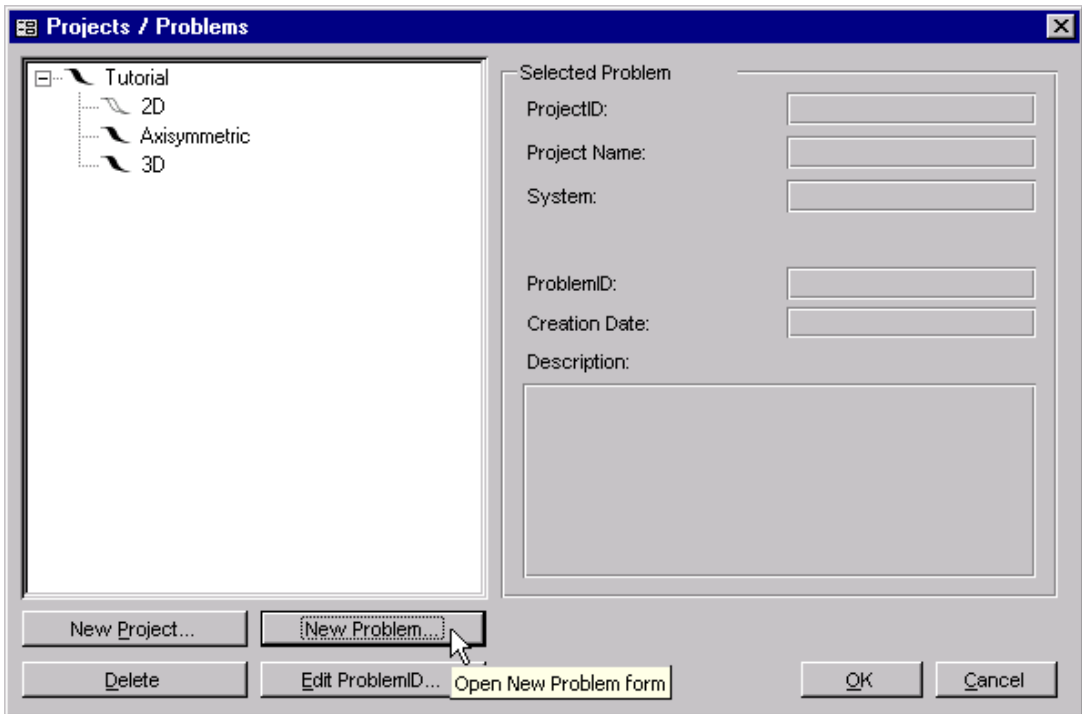
Project Notes: The tutorial example problems are stored under this project.

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.

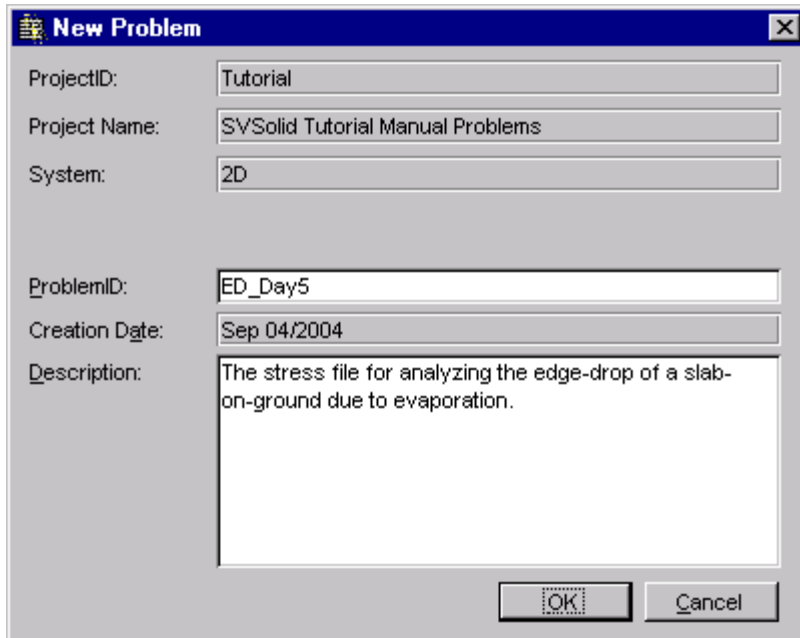
3.14.2 Adding an SVSOLID Problem



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.



The screenshot shows a 'New Problem' dialog box with the following fields and values:

- ProjectID: Tutorial
- Project Name: SVSolid Tutorial Manual Problems
- System: 2D
- ProblemID: ED_Day5
- Creation Date: Sep 04/2004
- Description: The stress file for analyzing the edge-drop of a slab-ground due to evaporation.

4. Enter the **ProblemID: ED_Day5**. The description is optional.
5. Click the **OK button** to save the problem and close the New Problem form.
6. The new problem will automatically be opened in the workspace.

3.14.3 Opening the SVSOLID Problem

If the problem was just added it will already be open in the workspace. When returning to the problem follow these steps to open it in the workspace:

1. Select Model > Projects/Problems from the menu.
2. **Navigate** back to the problem via Tutorial, 2D.
3. Select **ED_Day5**.
4. The problem may be opened by clicking the **OK** button or by double clicking on the ProblemID.

3.15 DEFINING THE SVSOLID PROBLEM

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

3.15.1 Specify Settings

The first step in defining the problem is to specify the settings that will be used for the problem.

System <input checked="" type="radio"/> 2D-Plane Strain <input type="radio"/> Axisymmetric <input type="radio"/> 3D	Analysis <input type="radio"/> Neglect PWP <input checked="" type="radio"/> Consider PWP	Units <input checked="" type="radio"/> Metric <input type="radio"/> Imperial Length: <input type="text" value="m"/> Stress: <input type="text" value="kPa"/>
---	---	---

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. The initial stress conditions, initial pore-water pressure conditions, and final pore-water pressure conditions

1. To open the Settings form select **Model > Settings** in the menu.
2. Select **Consider PWP** as the Analysis option.
3. Move to the **Initial Stress/Strain** tab.
4. Select **K_o -Loading** as the Initial Stress Option. (A coefficient of earth pressure at rest, K_o value will be entered later on the Soil Properties form)

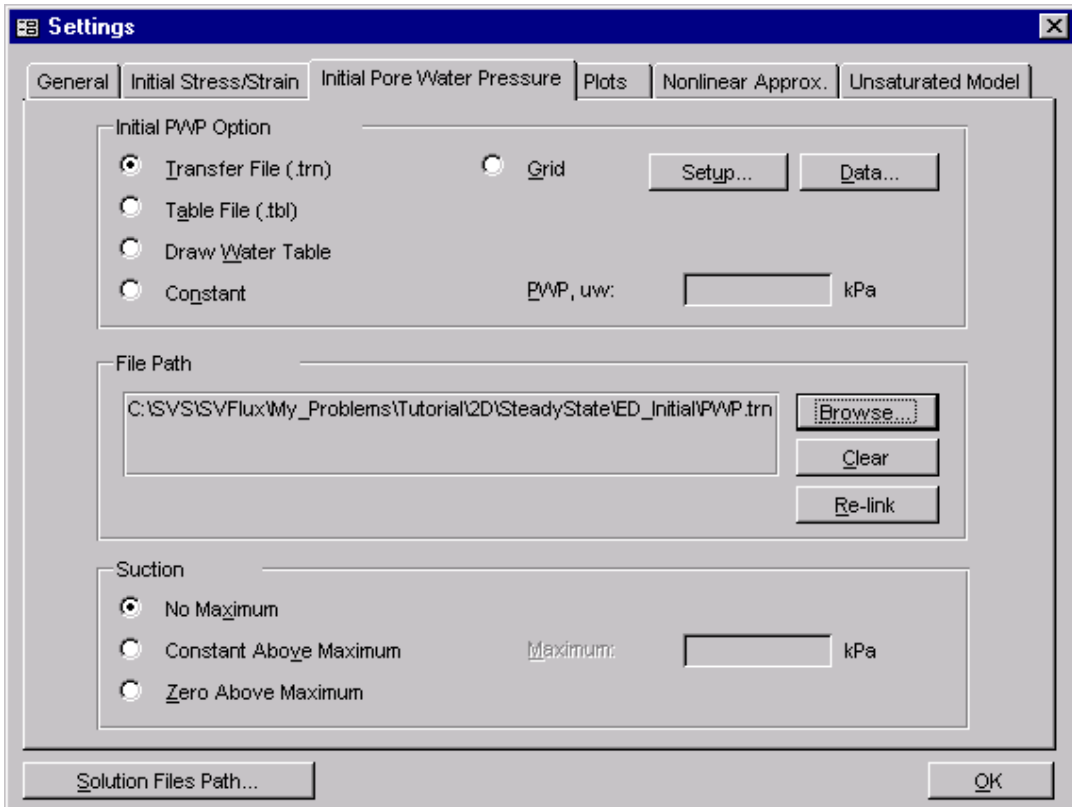
Initial Stress Option <input type="radio"/> None <input type="radio"/> Transfer File (.trn) <input checked="" type="radio"/> K_o -Loading Specify K_o values for each soil on the Soil Properties form.
--



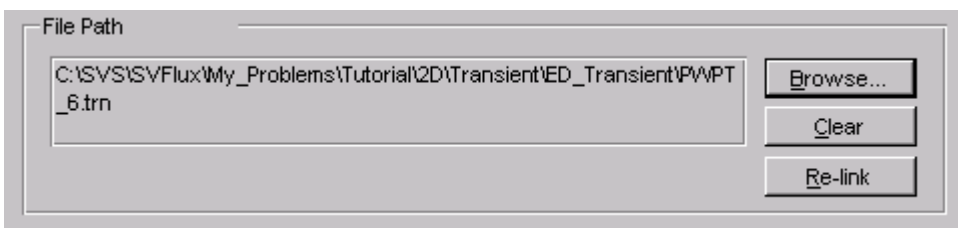
Tip!

The K_o -Loading option means that initial vertical stress will be a function soil unit weight and elevation and the horizontal initial stresses will be equal to the initial vertical stress multiplied by the K_o value. To use this option the ground surface will be flat at an elevation of 0. A K_o of 0.33 for a drying path will be used as suggested by Lytton (1997)

5. Move to the **Initial Pore Water Pressure** tab.
6. Select **Transfer File (.TRN)** as the Initial PWP Option.
7. Press the **Browse** button.
8. Then specify the path to the **PWP.trn** file output by the Initial SVFLUX Analysis.



9. Move to the **Unsaturation Model** tab.
10. Select **Transfer File (.TRN)** as the Final PWP Option.
11. Press the **Browse** button.
12. Then specify the path to the **PWPT_6.trn** file output by the SVFLUX Transient Analysis. (The file corresponds to the pore-water pressure after 5 days of evaporation. The file counter considers time 0 as **_1**, therefore **_6** corresponds to day 5)



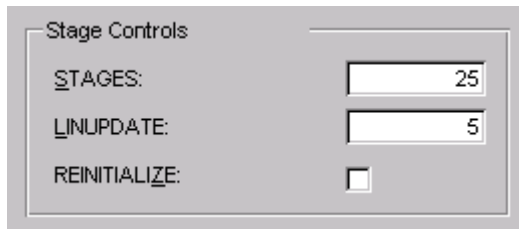
13. Click **OK** to close the form.

3.15.2 Setting the Stages

The number of stages will control the incremental change in suction. The total change in suction is difference between the initial and final suctions from the specified transfer files. The stages will be set to 25 for this analysis, therefore, the incremental displacements will be calculated for each suction increment at each stage.

To set the stages to 25 follow these steps:

1. Access the FEM Options form by selecting **Model > FEM Options** from the menu.



Stage Controls

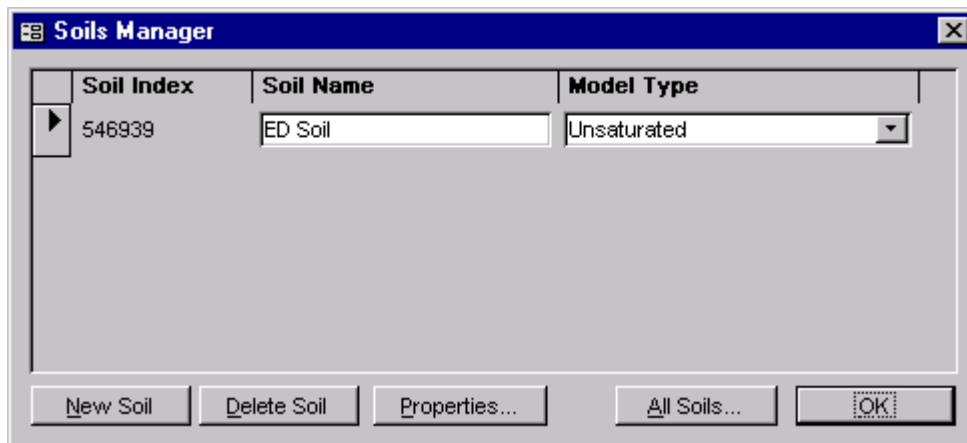
STAGES:

LINUPDATE:

REINITIALIZE:

2. Enter the **25** in the STAGES field.
3. Click Yes for the message.
4. Click **OK** to close the form.


3.15.3 Define Material Properties



Soil Index	Soil Name	Model Type
546939	ED Soil	Unsaturated

New Soil Delete Soil Properties... All Soils... OK

The next step in defining the problem is to enter the material properties for the soil that will be used in the model. Refer to the [Problem Overview](#) section of this tutorial for the relevant soil properties.

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. Enter a **Soil Name** of ED Soil.

4. Select **Unsaturated** as the Model Type
5. Select the new soil and click Properties to open the **Soil Properties** form.

Soil Properties: Unsaturated

Soil Index: 546939

Description Parameters Body Load Shear Strength Reference

Elastic Parameters

Poisson's Ratio, ν : 0.400 Staged

Coefficient of earth pressure at rest, K_o : 0.330

Relate ν and K_o by [$\nu=K_o/(1+K_o)$]:

Void Ratio Parameters

Initial Void Ratio, v_{ro} : 1.00

Swelling Index, C_s : 0.15 Staged

Volume Change Index with respect to Matric Suction, C_m : 0.13 Staged

Graph Void Ratio...

Stage Parameters... OK

6. Move to the **Parameters** tab.
7. Enter the **Poisson's Ratio** value of 0.4.
8. Enter the **K_o** value of 0.33.



Tip!

Note that the Poisson's Ratio and K_o are not related for this analysis. Be sure to leave the Relate ν and K_o by [$\nu=K_o/(1+K_o)$] checkbox unchecked. The K_o is used for determining initial conditions while the Poisson's Ratio is used in the general stress solution.

9. Enter the **Initial Void Ratio** value of 1.
10. Enter the **C_s** value of 0.15.
11. Enter the **C_m** value of 0.13.
12. Move to the **Body Load** tab.
13. Enter the **Y-Axis Body Load** as -17.2 kN/m^3 .
14. Click **OK** to close the Soil Properties form.
15. Click **OK** to close the Soils Manager form.



For the Unsaturated soil model the void ratio is a function of the net mean stress as well as the matric suction. Use the Graph Void Ratio button to view graphs of Void Ratio vs. Net Mean Stress at a given matric suction and of Void Ratio vs. Matric Suction for a given net mean stress.

3.15.4 Importing Geometry From SVFLUX

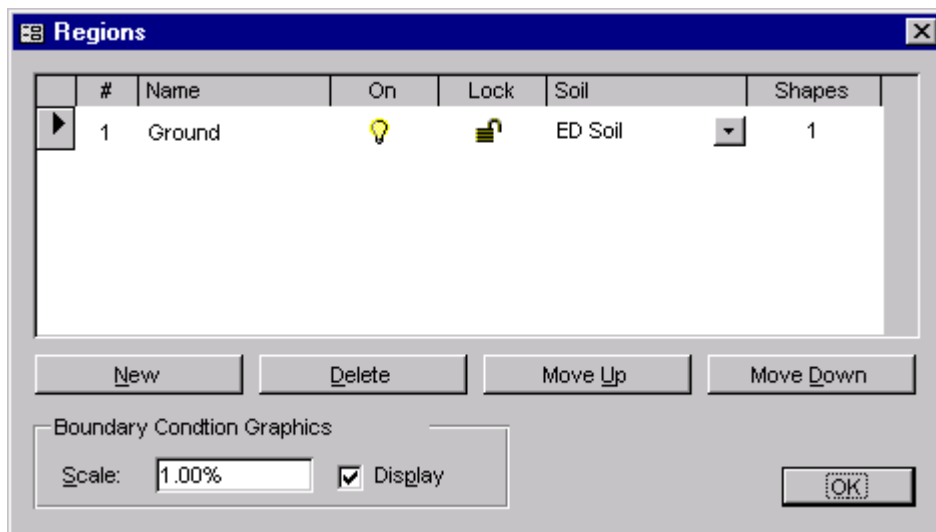
Since the Ground region and its geometry were defined previously for the Initial SVFLUX Analysis, the **Import SVFLUX Geometry** feature can be used to save time for this analysis by importing the geometry from the SVFLUX software.

Follow the steps in the section [Defining the Transient SVFLUX Problem – Importing Geometry](#).

3.15.5 Assign the Region Soil

The soil that was previously defined will need to be assigned to the Ground region that was just imported.

1. Select **Model > Regions** from the menu to open the Regions form.



2. Select the **ED Soil** soil from the Soil drop-down.
3. Click OK to close the Regions form.

3.15.6 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. The base of the problem will be fixed in both the X and Y directions. The left and right boundaries will be fixed in the X-direction, but will be free to move in the Y-direction. The ground surface is free to move in both directions. The steps for specifying the boundary conditions are thus:

1. Select the “Ground” region in the drawing space.

- From the menu select **Model > Boundaries**. The boundary conditions form will open.

	Point		Boundary Condition	
	X	Y	Direction	
	0	-3	Condition: Fixed	Y: Fixed
	6	-3	Condition: Continue	Y: Continue
	12	-3	Condition: Continue	Y: Free
	12	0	Condition: Free	Y: Continue
	6	0	Condition: Continue	Y: Continue
	0	0	Condition: Fixed	Y: Continue

Segment Length: 3 m

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

- Select the point **(0,-3)** from the list on the Segment tab.
- From the **X Boundary Condition** drop down select a **Fixed** boundary condition.
- From the **Y Boundary Condition** drop down select a **Fixed** boundary condition.
- Select the point **(12,-3)** from the list.
- From the **Y Boundary Condition** drop down select a **Free** boundary condition.
- Select the point **(12,0)** from the list.
- From the **X Boundary Condition** drop down select a **Free** boundary condition.
- Select the last point **(0,0)** from the list.
- From the **X Boundary Condition** drop down select a **Fixed** boundary condition.
- Click the **OK** button to close the form.



Tip!

The Fixed X boundary condition for the point (0,-3) 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 Free condition at point (12,0) the Continue is turned off and the Free condition established.

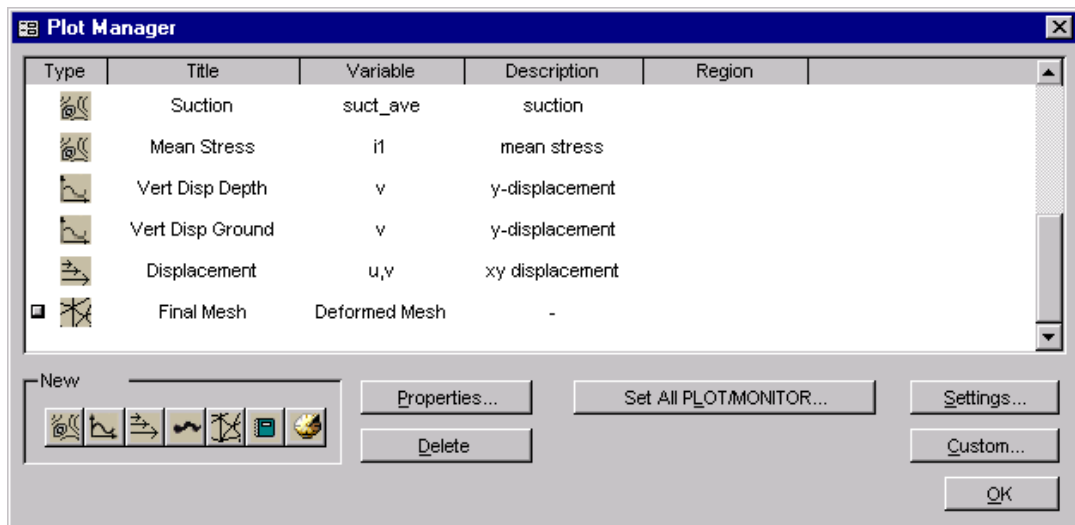
Boundary Condition Summary

X	Y	X Boundary Condition	Y Boundary Condition
0	-3	Fixed	Fixed
6	-3	Continue	Continue
12	-3	Continue	Free

12	0	Free	Continue
6	0	Continue	Continue
0	0	Fixed	Continue

3.16 SPECIFY SVSOLID PLOTS

There are many plot types that can be specified to visualize the results of the model. A few will be generated for this tutorial example problem including a plot of the solution mesh, stress contours, and displacement vectors. Plots are not required for problem solution, but no results can be seen without them. Define at least 1 plot from the Suggested Plots table below.



Refer to the section [Specify SVFLUX Transient Analysis Plots](#) earlier in this tutorial for instructions on adding plots.

Suggested Plots

Plot Type	Title	Variable	Range
Contour	Vertical Stress	<i>sy</i>	
Contour	Horizontal Stress	<i>sx</i>	
Contour	Pore-water Pressure	<i>uw</i>	
Contour	Suction	<i>suct_ave</i>	
Contour	Mean Stress	<i>i1</i>	
Contour	Void Ratio	<i>vr</i>	
Contour	Young's Modulus	<i>E</i>	
Contour	H Modulus	<i>Hms</i>	
Contour	Vertical Displacement	<i>v</i>	

Vector	Displacement	u, v	
Mesh	Final Mesh	Deformed Mesh	
Elevation	Vert Disp Depth	v	(6,0) to (6,-3)
Elevation	Vert Disp Ground	v	(0,0) to (12,0)

3.17 ANALYZE THE SVSOLID PROBLEM

The next step is to analyze the problem. Click the Analyze button located at the left in the workspace. This action will write the descriptor files and open the SVSOLID solver. The solver will automatically begin solving the problem and the Run Log form will open in SVSOLID. There are 3 .pde files that will be created:

1. Tutorial_ED_Day5_BATCH.pde
2. Tutorial_ED_Day5.pde
3. Tutorial_ED_Day5_Summary.pde

The Batch file will call the other 2 files in sequence and they will be solved automatically. When the solver has finished running:

1. Press OK for the **Batch Done** message.
2. Click the **Read File** button on the Run Log in SVSOLID form to record the run data.

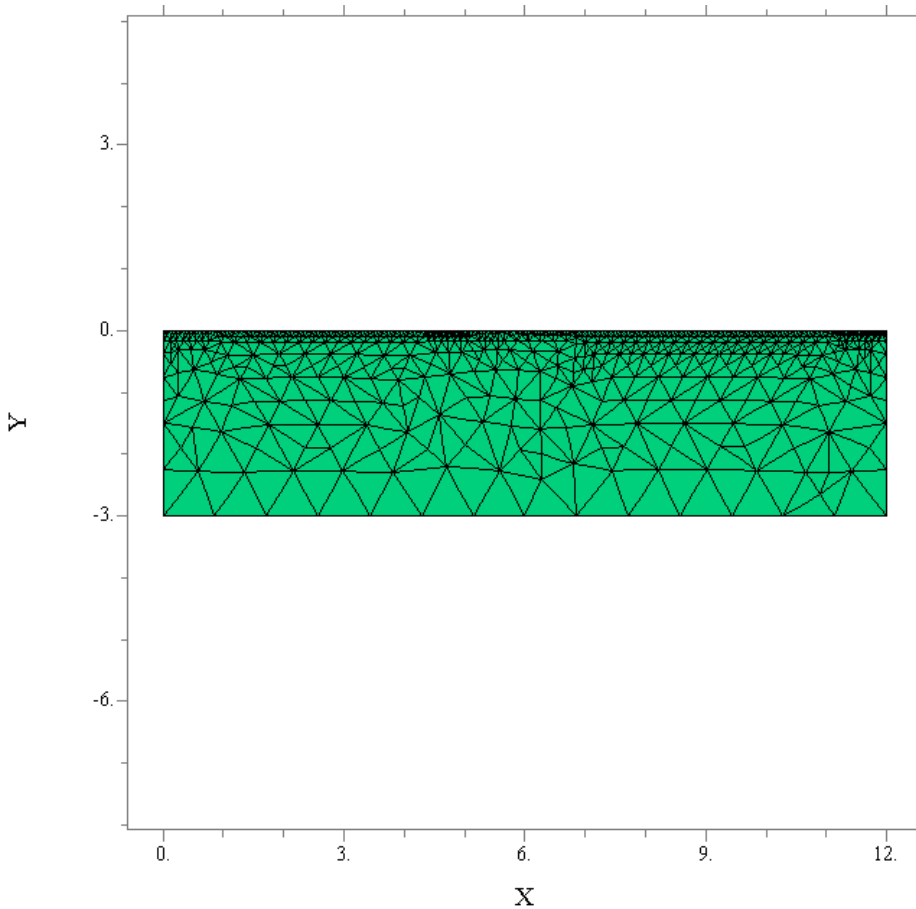


Tip! While the problem is running, the results will be displayed in the form of thumbnail plots within the SVSOLID solver. Right-click the mouse and select Maximize to enlarge any of the thumbnail plots.

3.18 SVSOLID RESULTS

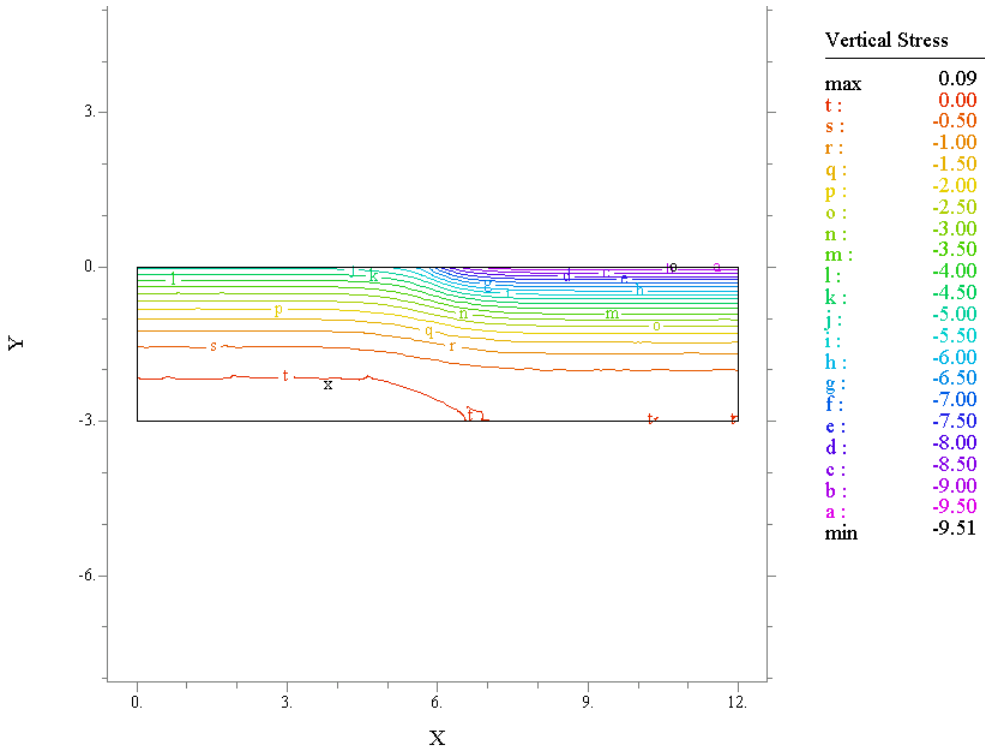
To view the SVSOLID results once the problem has finished select **Analyze > View Results** from the SVSOLID menu. The results will be displayed in the form of thumbnail plots within the SVSOLID solver. Right-click the mouse and select Maximize to enlarge any of the thumbnail plots. This section will give a brief analysis for some plots that were generated.

3.18.1 Solution Mesh



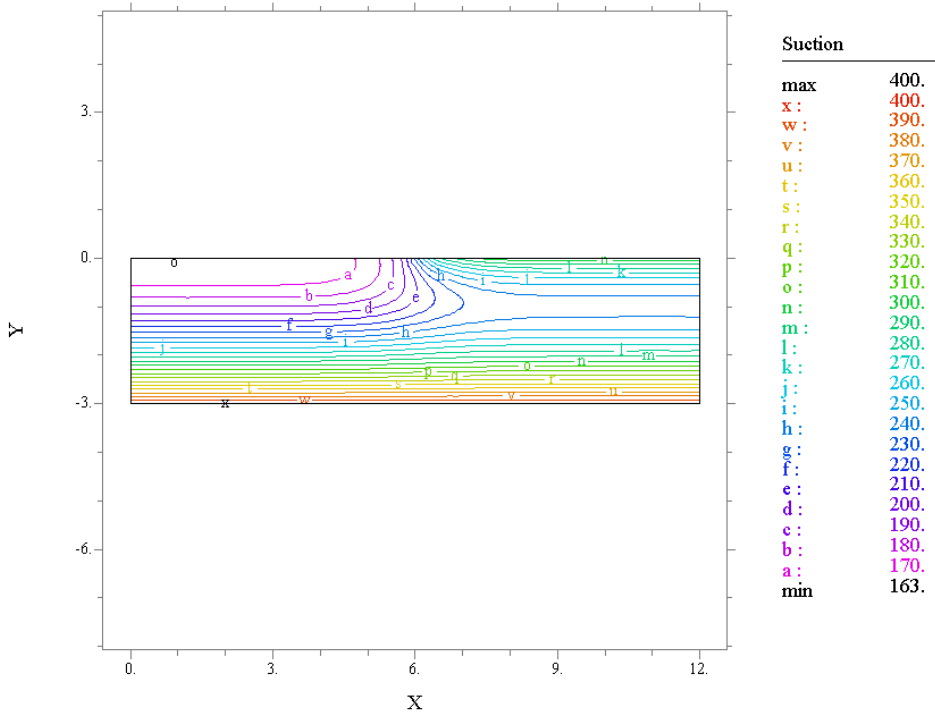
The Mesh plot displays the finite-element mesh generated by the solver. The mesh is automatically refined in critical areas such as near the ground surface where there is a greater void ratio change. The displacements in this plot are magnified by 50 times.

3.18.2 Vertical Stress Contours



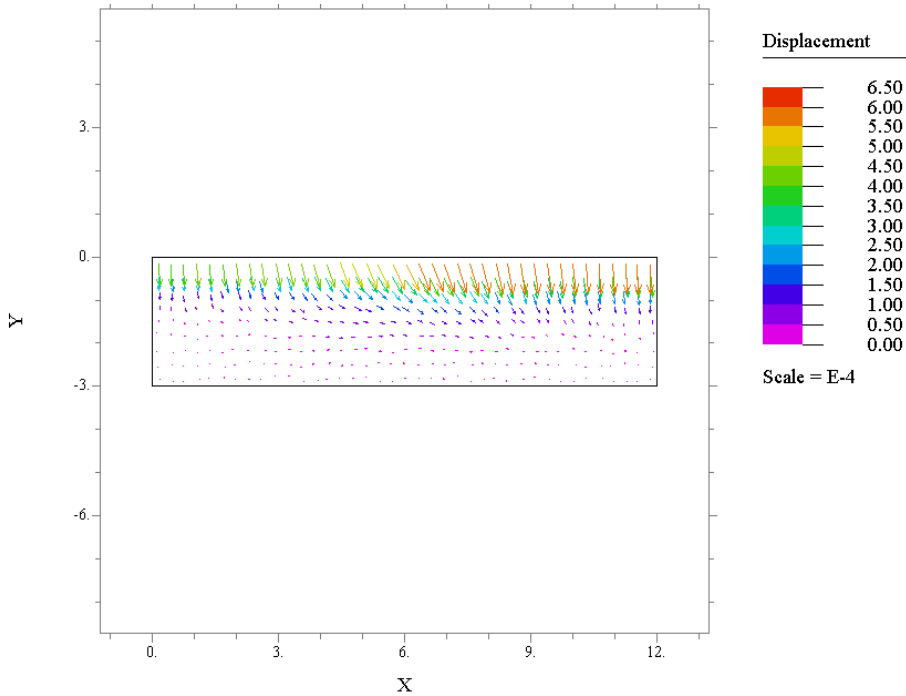
A negative or tensile vertical stress develops most significantly near the uncovered ground surface.

3.18.3 Suction Contours



The contour plot of suction indicates the development of higher suction at the uncovered boundary due to evaporation.

3.18.4 Displacement Vectors

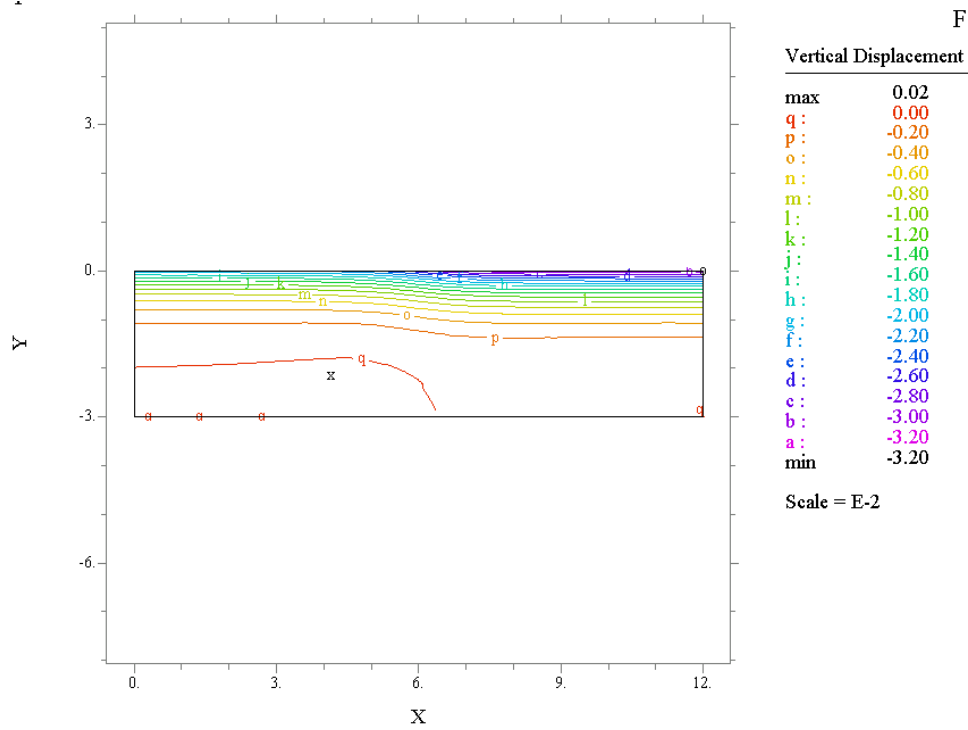


Displacement Vectors show both the direction and the magnitude of the displacement at specific points in the problem. Settlement is greatest under the un-covered ground surface.

3.19 SVSOLID SUMMARY FILE RESULTS

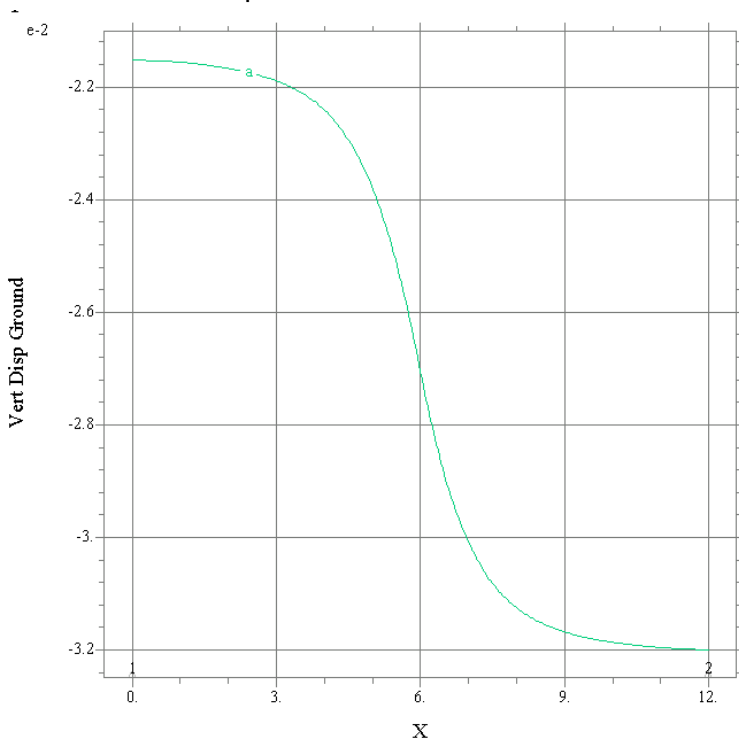
The SVSOLID Summary file will generate the same plots as the regular SVSOLID analysis that apply to the displacement variables. Only in this case the plots will be the summation of the results from each stage.

3.19.1 Vertical Displacement Contours



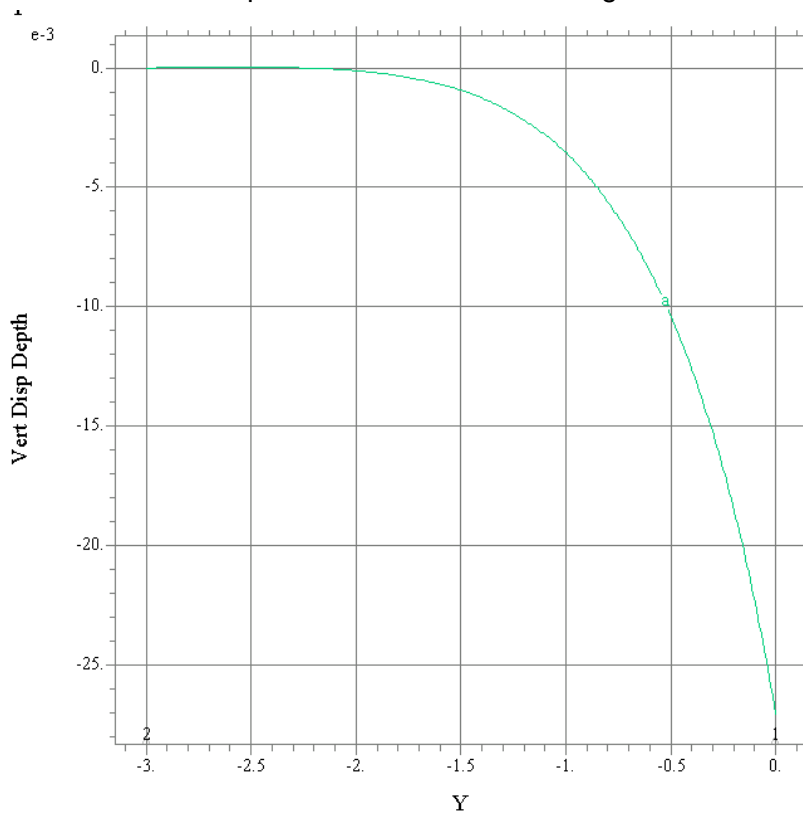
The vertical displacement contours plot shows a maximum settlement of 32mm at the top right corner of the soil region.

3.19.2 Vertical Displacement at Ground Surface



The above plot shows vertical displacement along the length of the ground surface. Differential settlements took place at the edge of the cover. About 10mm of settlement has occurred after 5 days of evaporation.

3.19.3 Vertical Displacement Below Cover Edge



The above plot shows vertical displacement below the edge of the cover. Most of the settlement took place near the ground surface where the change in matric suction is largest and where the soil has a low elastic modulus.

3.20 PROBLEM CONCLUSION

The 2D **Edge Drop** of a Flexible Impervious Cover tutorial problem is now complete.

4 3D: EDGE DROP OF A FLEXIBLE IMPERVIOUS COVER

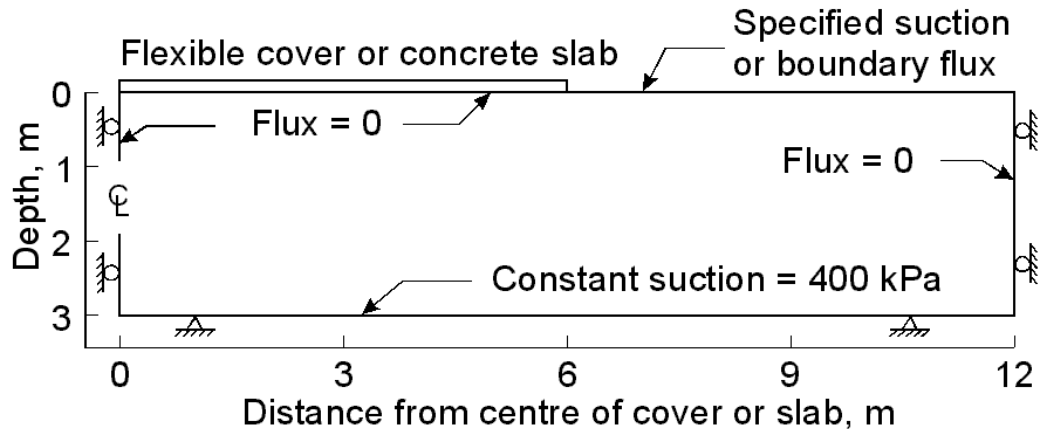
4.1 PROBLEM OVERVIEW

The following example will provide a step by step guide to modeling slab movement using a manual iteration technique involving the SVFLUX and SVSOLID software packages. There are two main scenarios to consider; the **Edge Drop** of a slab caused by shrinking of the soil due to evaporation (increase in soil suction) and **Edge Lift** of a slab caused by swelling due to infiltration (decrease in soil suction). This tutorial problem will consider the **Edge Drop** scenario.

This problem is the problem, Slab Movement – A Two-Dimensional Example Problem presented earlier in the manual extended into three dimensions.

- **Geometry and Boundary Conditions**

A 12-m long flexible impervious cover is considered. Since the cover is symmetrical the portion to the right of the centerline will be modeled. A soil region 3m deep and 12m long is used. The problem is 1m wide in the X-direction. The following view is a two-dimensional slice through the problem.



- **Soil Properties**

Seepage Soil Properties	Values
Coefficient of permeability at saturation, k_{sat}	1×10^{-8} m/s
Volumetric water content at saturation, θ_s	0.45
Parameters for SWCC (Fredlund & King, 1994) and permeability function (Leong and Rahardjo, 1997)	$a = 300$ kPa $n = 1.5$ $m = 1$ $hr = 3000$ kPa Sat. Suction = 0.1 kPa $\rho = 1$
Stress Soil Properties	Values
Total Unit Weight, γ_t	17.2 kN/m ²
Initial Void Ratio, e_o	1
Swelling Index, C_s	0.15
Swelling Index, C_m	0.13
Poisson's Ratio, μ_s	0.4
Coefficient of earth pressure at rest, K_o	0.33

- **Solution Outline**

This manual iteration method involves a number of steps to arrive at the final displacements. These example problems are included in the SVFLUX and SVSOLID databases for reference under the ProjectID of Tutorial. The ProblemID is indicated in the parentheses.

1. Initial SVFLUX seepage analysis (ED3D_Initial)
2. SVFLUX transient seepage analysis (ED3D_Transient)
3. SVSOLID stress/deformation analysis (ED3D_Day5)

4.2 INITIAL SVFLUX SEEPAGE ANALYSIS SETUP

The purpose of the initial SVFLUX seepage analysis is to get an initial head profile to use as initial conditions for the SVFLUX transient seepage and to get an initial pore-water pressure profile to use as initial conditions for the SVSOLID stress/deformation analyses.

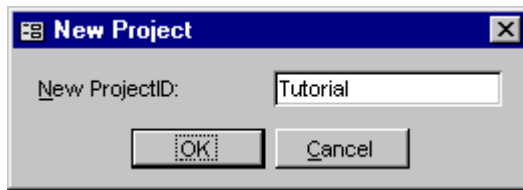
4.2.1 Adding an SVFLUX Project

The first step in defining this 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.

Follow these steps in order to add the project:

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.



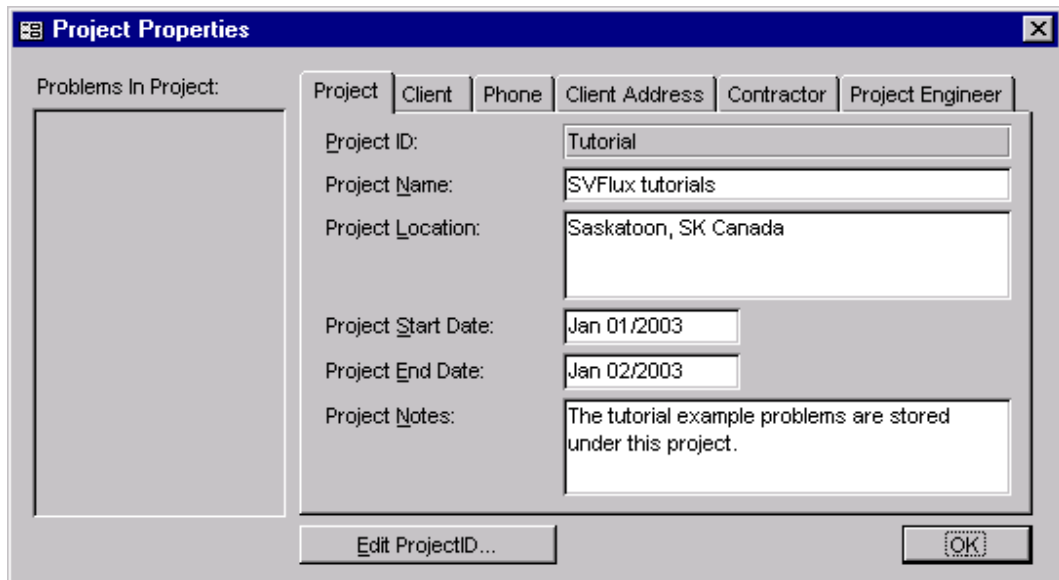
4. Type "Tutorial" as the new ProjectID and press OK.

The Project Properties form is where 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.



The ProjectID 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.

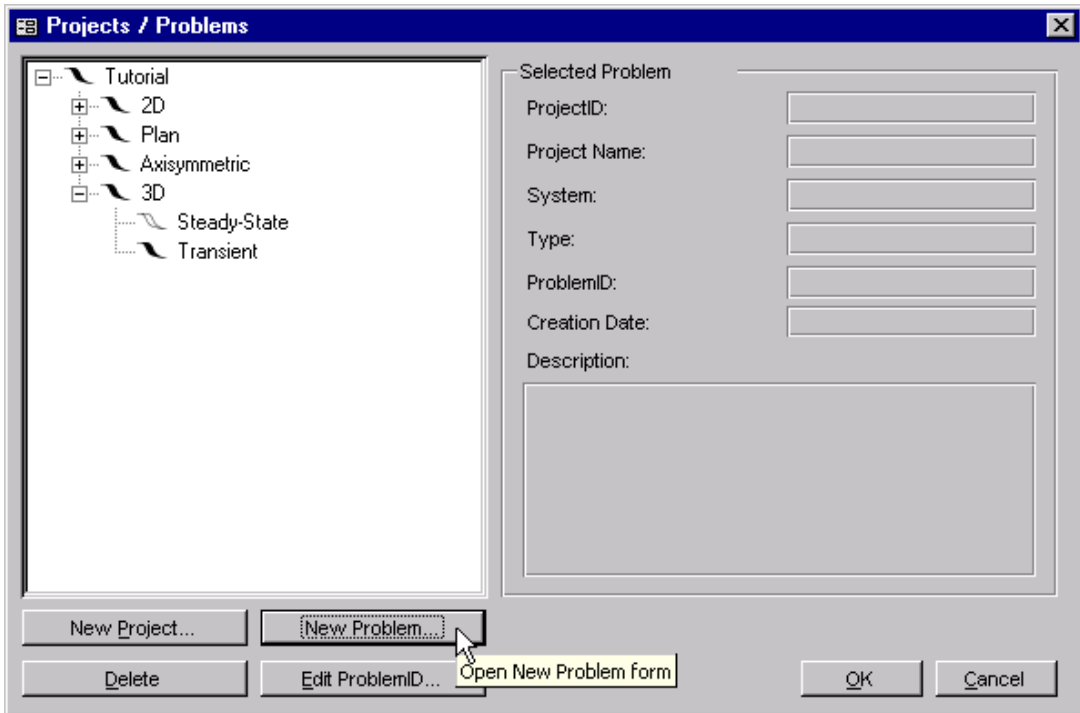


It should be noted that once the project is defined it would be identified by the ProjectID throughout the rest of the program. Also, SVFLUX does not allow you to specify two projects with the same ProjectID.

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.

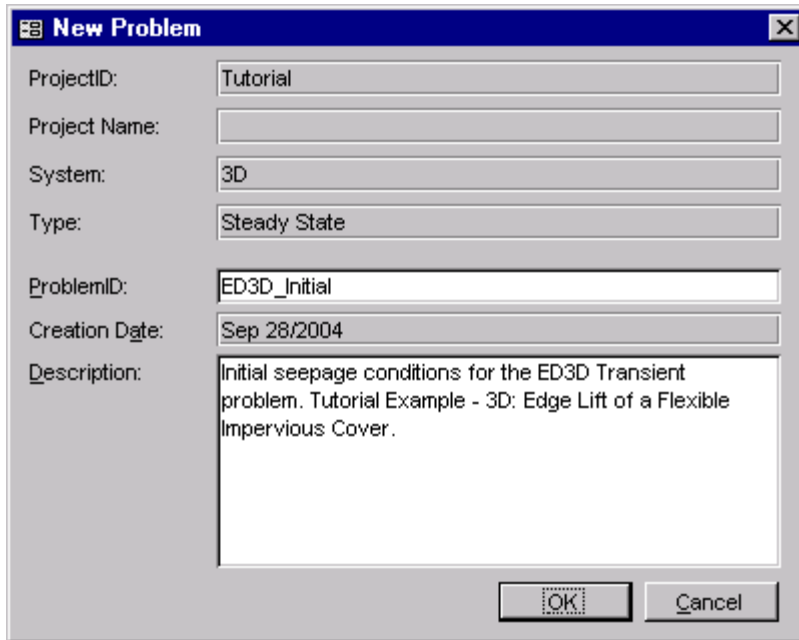
4.3 ADDING THE INITIAL SVFLUX PROBLEM

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 three-dimensions and is steady-state so we must expand **3D** by clicking the plus sign beside “3D”
3. Select **Steady-State**.
4. Click the **New Problem** button. The New Problem form will open.



New Problem

ProjectID: Tutorial

Project Name:

System: 3D

Type: Steady State

ProblemID: ED3D_Initial

Creation Date: Sep 28/2004

Description: Initial seepage conditions for the ED3D Transient problem. Tutorial Example - 3D: Edge Lift of a Flexible Impervious Cover.

OK Cancel

5. Enter the **ProblemID: ED3D_Initial**. The description is optional.
6. Click the **OK** button to save the problem and close the New Problem form.
7. The new problem will automatically be opened in the workspace.

4.3.1 Opening the Initial SVFLUX Problem

If the problem was just added it will already be open in the workspace. When returning to the problem follow these steps to open it in the workspace:


1. Select Model > Projects/Problems from the menu.
2. **Navigate** back to the problem via Tutorial, 3D, Steady-State.
3. Select **ED3D_Initial**.
4. The problem may be opened by clicking the **OK** button or by double clicking on the ProblemID in the tree control.

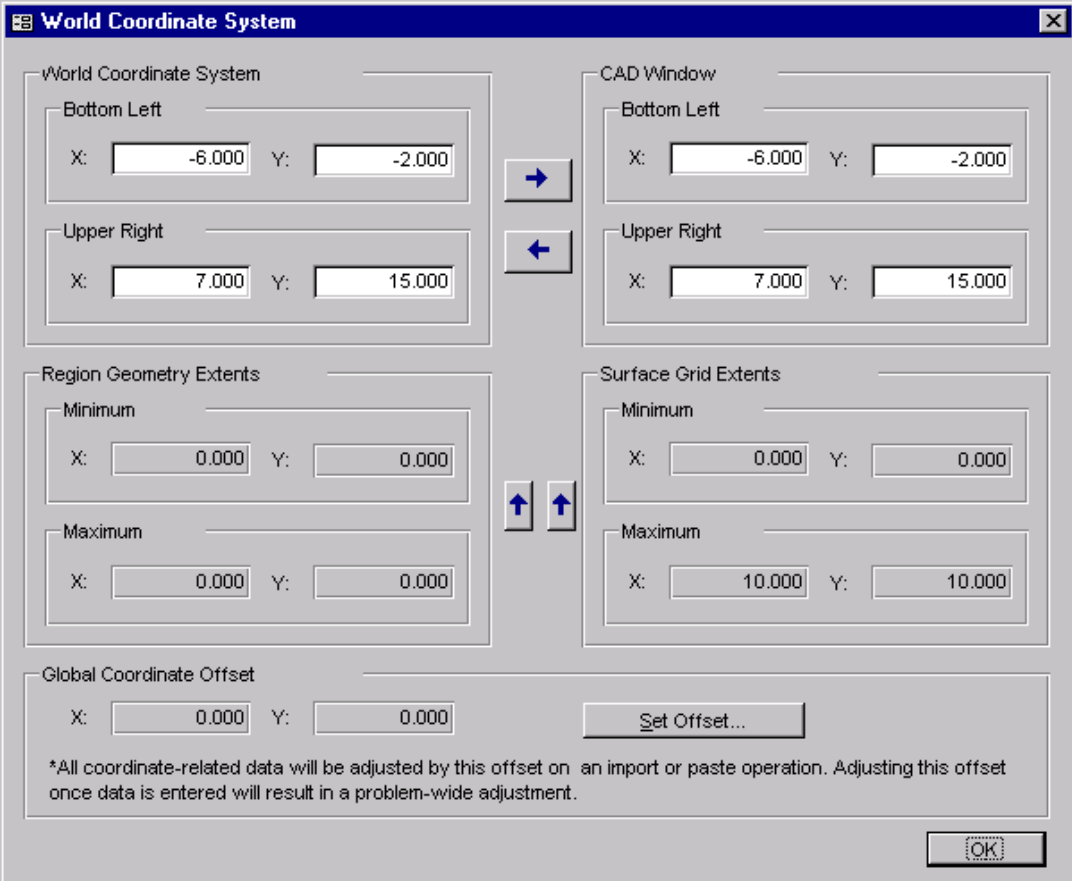
4.4 DEFINING THE INITIAL SVFLUX PROBLEM

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

4.4.1 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.

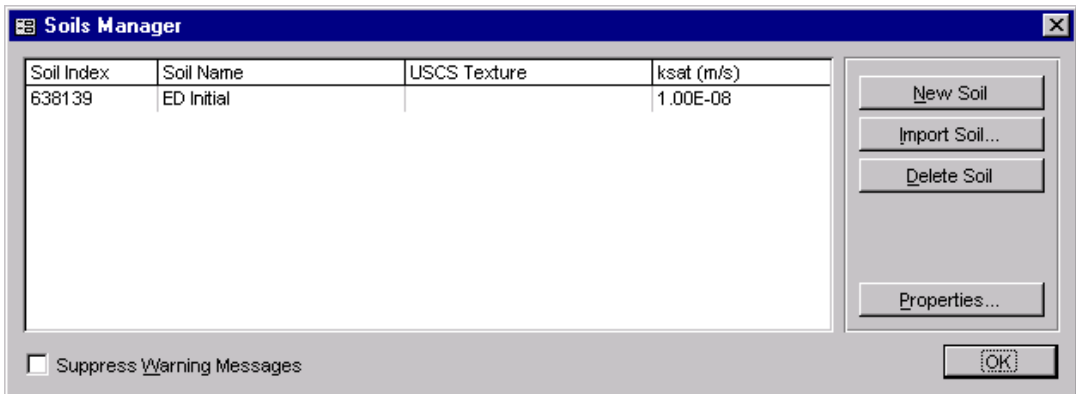


The image shows a software dialog box titled "World Coordinate System". It is divided into several sections for defining coordinate systems and extents. The "World Coordinate System" section has "Bottom Left" (X: -6.000, Y: -2.000) and "Upper Right" (X: 7.000, Y: 15.000) fields. The "CAD Window" section has identical "Bottom Left" (X: -6.000, Y: -2.000) and "Upper Right" (X: 7.000, Y: 15.000) fields. The "Region Geometry Extents" section has "Minimum" (X: 0.000, Y: 0.000) and "Maximum" (X: 0.000, Y: 0.000) fields. The "Surface Grid Extents" section has "Minimum" (X: 0.000, Y: 0.000) and "Maximum" (X: 10.000, Y: 10.000) fields. A "Global Coordinate Offset" section at the bottom has X: 0.000, Y: 0.000, and a "Set Offset..." button. A note at the bottom states: "*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." An "OK" button is in the bottom right corner.


Section	Field	X Value	Y Value
World Coordinate System	Bottom Left	-6.000	-2.000
	Upper Right	7.000	15.000
CAD Window	Bottom Left	-6.000	-2.000
	Upper Right	7.000	15.000
Region Geometry Extents	Minimum	0.000	0.000
	Maximum	0.000	0.000
Surface Grid Extents	Minimum	0.000	0.000
	Maximum	10.000	10.000
Global Coordinate Offset	X	0.000	0.000
Global Coordinate Offset	Y	0.000	0.000

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.

4.4.2 Define Material Properties



The next step in defining the problem is to enter the material properties for the soil that will be used in the model.

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 SoilIndex is generated that is used to reference the soil in other SVFLUX forms.
3. Double-click on the new soil to open the **Soil Properties** form.

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


6. Refer to the data provided at the beginning of this tutorial. Enter the k_{sat} value of 1.000E-08 m/s

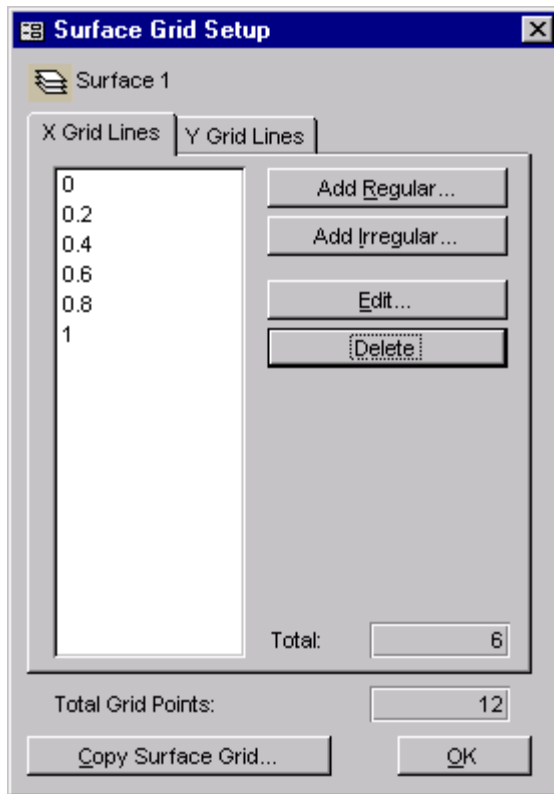
4.4.3 Define 3D Surfaces

This problem consists of two surfaces of different dimensions and grid densities. By default every problem initially has 2 surfaces.

- **Define Surface 1**


This surface is already present so the next step is to define the grid lines.

1. Select Surface 1 in the **Surface Selector**.
2. Click the **Surface Grid Setup** button, .



3. There will be default grid lines of 0 and 10 present. Click the **Add Regular** button to open the Add Regular X Gridlines form.
4. Enter 0 for **Start**, 0.2 for **Increment Value**, and 1 for **End**.
5. Click **Add** to add the gridlines and close the form.
6. Select 10 from the list.
7. Press **Delete**.
8. Move to the **Y Grid Lines** tab
9. There will be default grid lines of 0 and 10 present. Click the **Add Regular** button to open the Add Regular Y Gridlines form.
10. Enter 0 for **Start**, 0.5 for **Increment Value**, and 12 for **End**.
11. Click **Add** to add the gridlines and close the form.
12. Click **OK** to close the Surface Grid Setup form.


Now that the grid has been set up, elevations must be specified for all the grid points:

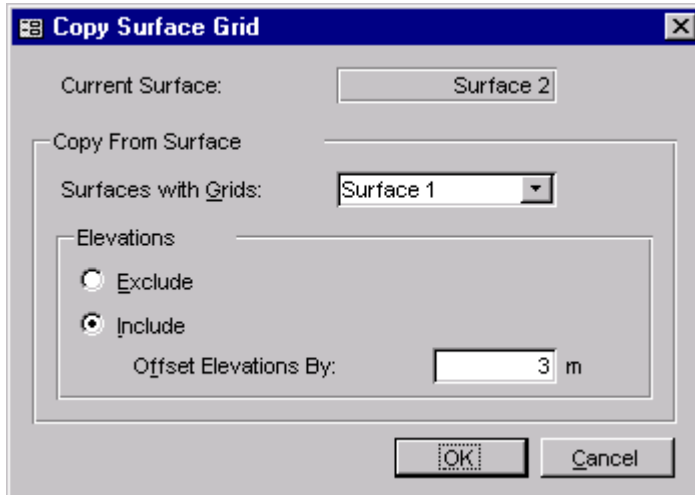
13. Select Surface 1 in the Surface Selector.
14. Click the **Elevations** button,  to open the Elevations form.
15. Enter -3 in the **Set Nulls To** field.
16. Click the button to the left of the Set Nulls To field and all the missing elevations will be set to -3.

17. Click OK to close the Elevations form.

- **Define Surface 2**

This surface is already present so the next step is to define the grid lines.

1. Select Surface 2 in the **Surface Selector**.
2. Click the **Surface Grid Setup** button, .
3. Click **Copy Surface Grid** to open the Surface Grid Setup form.



4. Select **Surface 1** as the Copy From Surface.
5. Check **Include** Elevations.
6. Enter **3 m** in the Offset Elevations By field.
7. Click **OK** to copy the surface and close the form.



Tip!

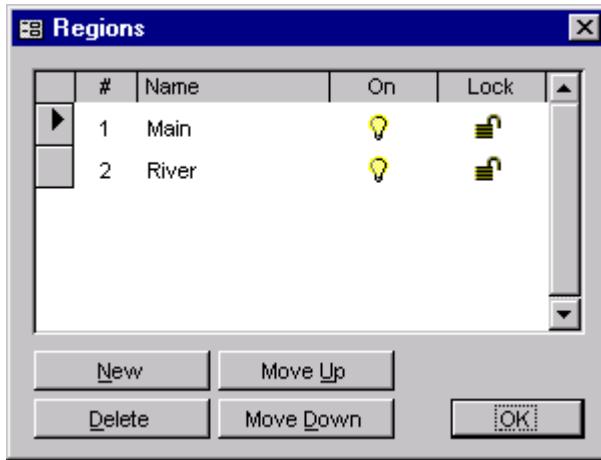
To view one of the surface grids just created, press the Surfaces button to the left of the Surface Selector to open the Surfaces form. Then check the Grid box for the desired surface.

4.4.4 Adding Regions

A region in SVFLUX 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 SVFLUX CAD workspace. A region will have a set of geometric shapes that define its soil boundaries. Also, other modeling objects including features, flux sections, water tables, text, and line art are defined on any given region.

This problem will be divided into 2 regions, which are named Slab and Ground. Each region will have the soil previously defined specified as its soil properties. 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 **Ground**. Highlight the name and type new text.
3. Press the **New** button to add a second region.
4. Change the name of the second region to **River**.
5. Click **OK** to close the form.

4.4.5 Defining Region Geometry Shapes

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.

Region Geometry

Ground

Slab

X	Y		X	Y
0	6		0	0
0	12		0	6
1	12		1	6
1	6		1	0

- **Define the Ground Region**

The instructions below explain the use of the command line to create the ground shape. Alternately, the ground shape could be drawn with the mouse or the data pasted into the Region Properties form.

1. Ensure that “Ground” 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 **0,6** and press the Enter key on the keyboard.

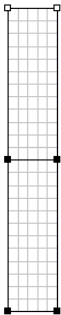
5. Type **0,12** and press Enter.
6. Type **1,12** and press Enter.
7. Type **1,6** and press Enter.
8. Type **f** and press **Enter** to complete the region shape.


- **Define the Slab Region**

In the instructions for drawing the Main shape the command line was used. These instructions will explain how to enter the points directly into the Region Properties form.

1. Ensure that “Slab” is current in the region selector.
2. Click on the **Region Properties** button, .
3. Press the **Paste Data** button to open the Paste Geometry Data form.
4. Enter the region **points** as shown above.
5. Check the Generate Closing Point box.
6. Click **OK** to add the points (Respond Yes to the message) and return to the Region Properties form.
7. Click **OK** to close the Region Properties form.


If the geometry been entered correctly the shapes should look like the following:

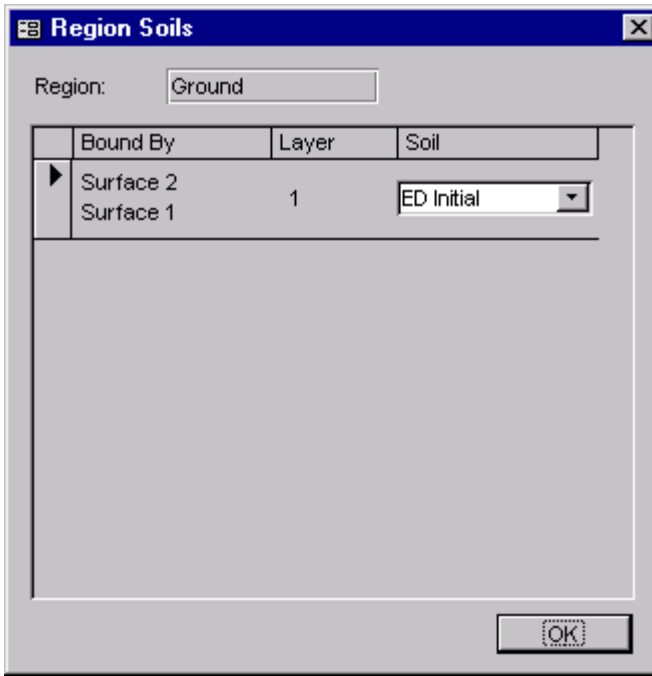


Select a shape with the mouse and press the Delete button,  if a mistake was made entering the coordinate points for a shape. This will remove the entire shape from the region. To edit the shape use the Region Properties form.

4.4.6 Specifying a Soil by Region and Layer

Each region will cut through all the layers in a problem creating a separate “block” on each layer. Each block can be assigned a soil or be left as **void**. A void area is essentially air space. In this problem all “blocks” will be assigned a soil.

1. Select “**Ground**” in the Region Selector.
2. Press the **Region Soils** button,  at the top of the workspace to open the Region Soils form.




Region Soils

Region:

	Bound By	Layer	Soil
▶	Surface 2	1	ED Initial
	Surface 1		

OK

3. Select the **ED Initial** soil from the drop-down for **Layer 1**.
4. Close the form using the **OK** button.
5. Select "**Slab**" in the Region Selector.
6. Press the **Region Soils** button,  at the top of the workspace to open the Region Soils form.
7. Select the **ED Initial** soil from the drop-down for **Layer 1**.
8. Close the form using the OK button.

4.4.7 Specify Boundary Conditions

Now that all of the regions, surfaces, and the soils have been successfully defined, the next step is to specify the boundary conditions on the region shapes. A suction of 20 kPa is required at the ground surface while a suction of 400 kPa exists at the bottom of the soil region. The suction values must be converted to head values for SVFLUX entry.

Ground Region Boundary Conditions

X	Y	Boundary Condition
0	6	Zero Flux
0	12	Continue
1	12	Continue
1	6	No BC
Surface 1		Head Expression = -43.787
Surface 2		Head Expression = -2.039

Slab Region Boundary Conditions

X	Y	Boundary Condition
0	0	Zero Flux
0	6	No BC
1	6	Zero Flux
1	0	Continue
Surface 1		Head Expression = -43.787
Surface 2		Head Expression = -2.039

The steps for specifying the boundary conditions are thus:

1. Select the “Ground” region in the region selector.
2. Select “Surface 1” in the surface selector.
3. From the menu select **Model > Boundaries**. The boundary conditions form will open and display the boundary conditions for the ground region on Surface 1. These boundary conditions will extend from Surface 1 to Surface 2 over Layer 1. By default the first boundary segment will be given a No BC condition.

Boundaries

Location

Region: Surface:

Segment Boundary Conditions | **Surface Boundary Conditions**

X	Y	Boundary Condition	Expression or Data	Units
0	6	Zero Flux		
0	12	Continue		
1	12	Continue		
1	6	No BC		

Update Selected Segment Segment Length: m

1. Select Boundary Condition:

2. Provide: a) Expression:

- or -

b) Flux Data Index:

c) Evaporation Index: (optional)

4. Select the **point** (0,6) from the list.
5. From the **Boundary Condition** drop down select a **Zero Flux** boundary condition.
6. Click the **Update** button to save the boundary condition to the list.
7. Select the **point** (1,6) from the list.
8. From the **Boundary Condition** drop down select a **No BC** boundary condition.
9. Click the **Update** button to save the boundary condition to the list.
10. Move the **Surface Boundary Conditions** tab.
11. From the **Boundary Condition** drop down select a **Head Expression** boundary condition. This will cause the Head Expression box to be enabled.
12. In the Head Expression box enter a **head of -43.787**.
13. Select **Surface 2** from the drop down box.
14. Move the **Surface Boundary Conditions** tab.
15. From the **Boundary Condition** drop down select a **Head Expression** boundary condition. This will cause the Head Expression box to be enabled.
16. In the Head Expression box enter a **head of -2.039**.
17. Click the **OK** button to close the Boundaries form.
18. **Repeat** these steps to add the boundary conditions for the Slab region indicated in the above table.

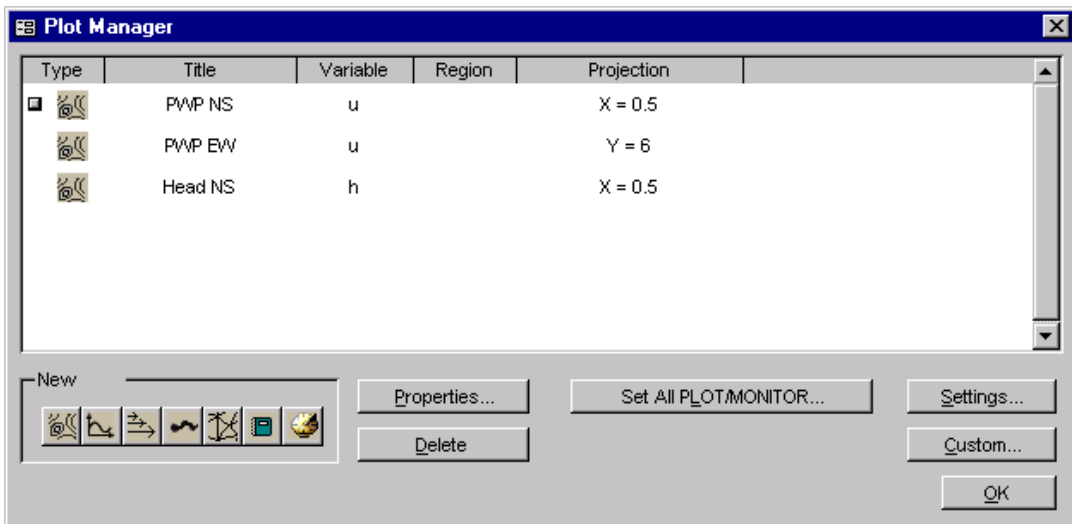


The Continue boundary condition indicates that the previously defined boundary condition will apply to the current boundary segment.

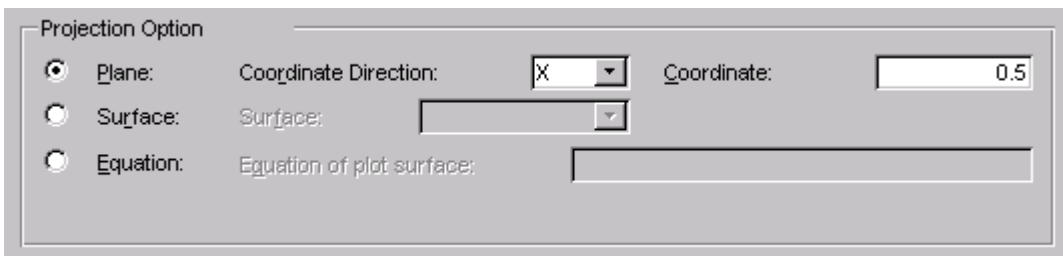
4.5 SPECIFY INITIAL SVFLUX ANALYSIS PLOTS

There are many plot types that can be specified to visualize the results of the model. 2 contour plots of pore-water pressure and a contour plot of head will be generated for this example.

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



2. 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.



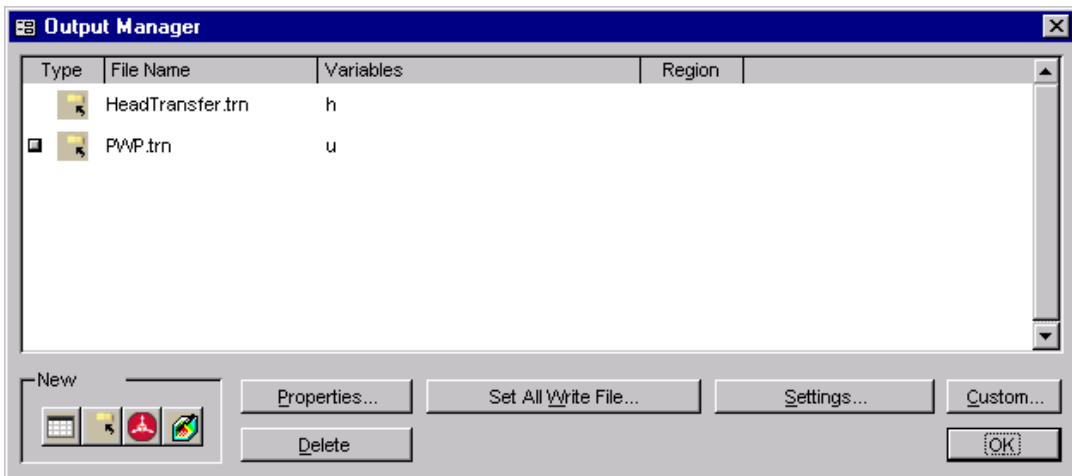
3. Enter the title **PWP NS**.
4. Select **u** as the variable to plot from the drop-down.
5. Select the **PLOT** output option.
6. Move to the **Projection** tab.
7. Select **Plane** Projection option.
8. Select **X** from the Coordinate Direction drop-down.

9. Enter **0.5** in the Coordinate field. This will generate a 2D slice at $X = 0.5\text{m}$ on which the pore-water pressure contours will be plotted.
10. Click **OK** to close the Plot Properties form and add the plot to the list.
11. Repeat these steps to create the plots shown in the screenshot above.
12. Click **OK** to close the Plot Manager and return to the workspace.

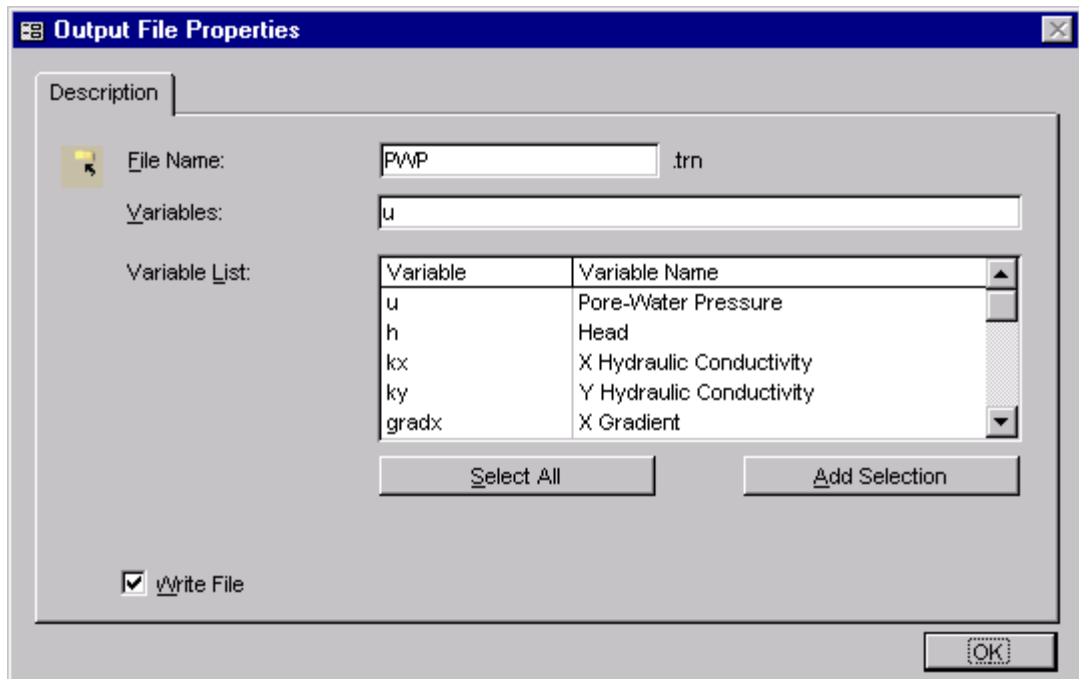
4.6 SPECIFY INITIAL SVFLUX ANALYSIS OUTPUT FILES

There are 4 output file types that can be specified to export the results of the model. Two transfer files will be generated for this tutorial example problem: a transfer file of pore-water pressure, and a transfer file of heads. Note that the file HeadTransfer.trn is already present. It is generated by default for every problem.

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 **Transfer** button to begin adding the output file. The Output File Properties form will open.



3. Enter the title PWP.
4. Select the variable u 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.

4.7 ANALYZE THE INITIAL SVFLUX PROBLEM

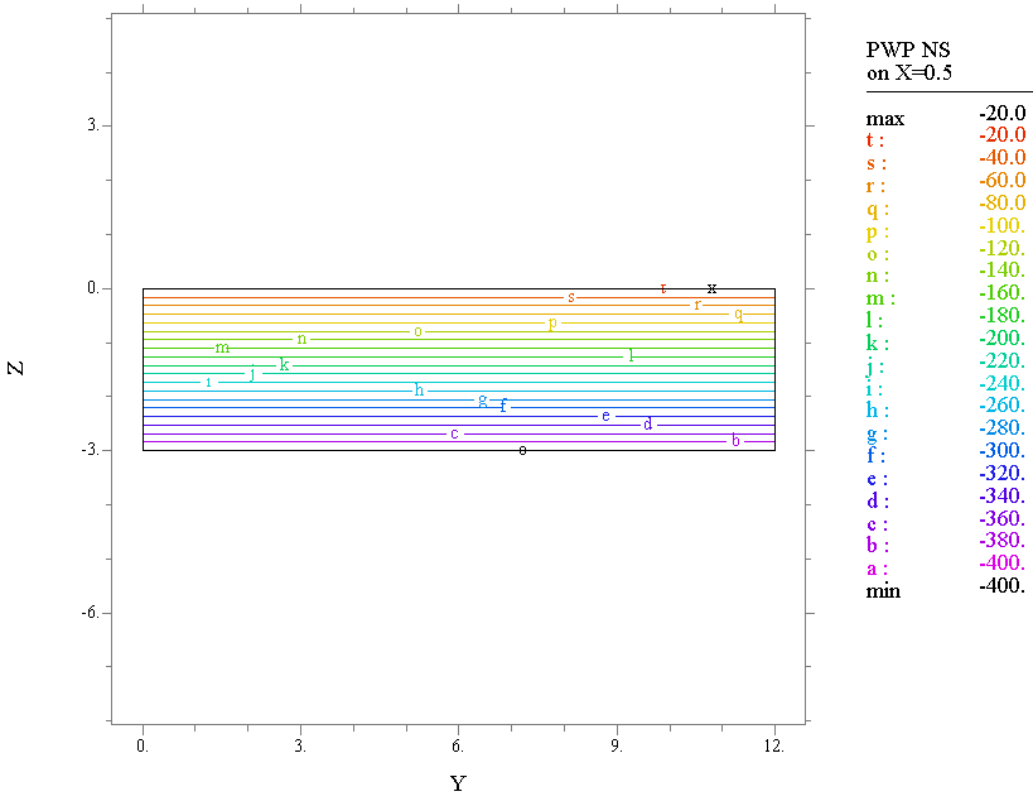
The next step is to analyze the problem. Click the Analyze button located on the left of the workspace. This action will write the descriptor file and open the SVFLUX solver. The solver will automatically begin solving the problem and the Run Log form will open in SVFLUX.

When the solver has finished running press the **Read File** button on the Run Log form to record the run data.

4.8 INITIAL SVFLUX ANALYSIS RESULTS

After the problem has finished solving, the results will be displayed in the form of thumbnail plots within the SVFLUX solver. Right-click the mouse and select Maximize to enlarge any of the thumbnail plots.

The output files requested (PWP.trn and HeadTransfer.trn) will be located in the solution file directory for the problem.



The contour plot of pore-water pressure above indicates -20 kPa at the ground surface and a decrease with depth to -400 kPa at the bottom.

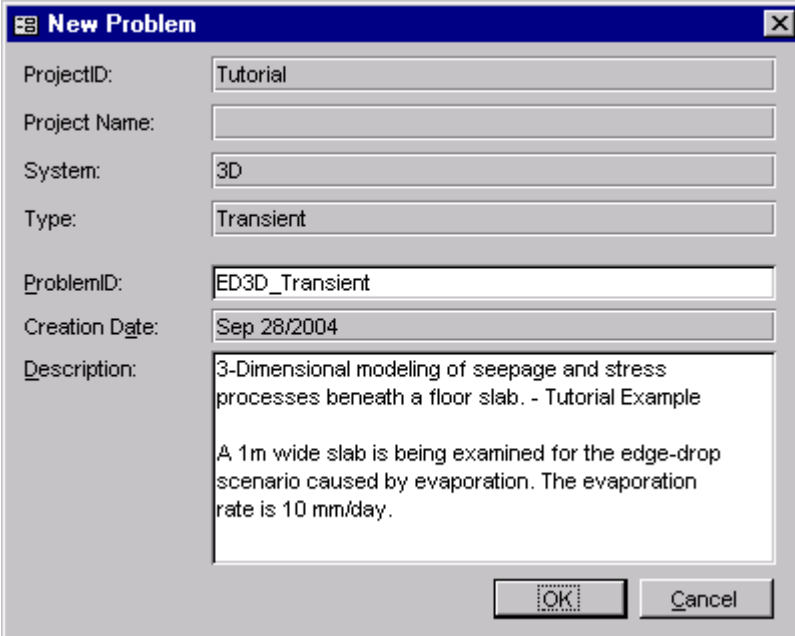
4.9 SVFLUX TRANSIENT SEEPAGE ANALYSIS SETUP

The SVFLUX transient seepage analysis will use the initial head transfer file as initial conditions. This step will output a pore-water pressure transfer (.trn) file for the SVSOLID stress/deformation analysis.

4.9.1 Adding the SVFLUX Transient Problem

A project was created previously (Tutorial) and the SVFLUX transient problem can be stored under it. When the **Projects/Problems** form is opened there will be a list of the projects that have been defined. To add the problem:

1. Click on the plus sign and expand the project **Tutorial**.
2. Expand **3D** by clicking the plus sign beside “3D”
3. Select **Transient**.
4. Click the **New Problem** button. The New Problem form will open.



New Problem

ProjectID: Tutorial

Project Name:

System: 3D

Type: Transient

ProblemID: ED3D_Transient

Creation Date: Sep 28/2004

Description:
3-Dimensional modeling of seepage and stress processes beneath a floor slab. - Tutorial Example

A 1m wide slab is being examined for the edge-drop scenario caused by evaporation. The evaporation rate is 10 mm/day.

OK Cancel

5. Enter the **ProblemID: ED3D_Transient**. The description is optional.
6. Click the **OK button** to save the problem and close the New Problem form.
7. The new problem will automatically be opened in the workspace.

4.9.2 Opening the Transient SVFLUX Problem

If the problem was just added it will already be open in the workspace. When returning to the problem refer to the steps in the section [Opening the Initial SVFLUX Problem](#) to open it in the workspace.

4.10 DEFINING THE TRANSIENT SVFLUX PROBLEM

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

4.10.1 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 screenshot shows the 'Settings' dialog box with the following configuration:

- System:** 2D, Plan, Axisymmetric, 3D
- Type:** Steady-State, Transient
- Units:** Metric, Imperial
- Time:** day (dropdown)
- Length:** m (dropdown)
- Force:** kN (dropdown)
- Pressure:** kPa (dropdown)
- Conductivity:** m/day (dropdown)

The above choices are read only. Problem choices are set when a new problem is created.

The Settings form will contain information about the current problem System, Type, Units, and Transient Settings.

1. Select **Metric** Units.
2. The problem is defined in terms of days. Select **day** from the drop-down control.
3. Move to the Time tab.
4. Set the **Start Time as 0**, the **Time Increment as 1**, and the **End Time as 5** days.
5. Move to the Initial Conditions tab.

File Path

O:\Data_J\My_Problems\Tutorial\3D\SteadyState\ED3D_Day5\HeadTransfer.trn

Browse...

Clear

Re-link

6. Click the **Browse** button and specify the path to the **HeadTransfer.trn** file generated by the ED3D_Initial problem.
7. Click OK to close the Settings form.

4.10.2 Define Material Properties

Soils Manager

Soil Index	Soil Name	USCS Texture	ksat (m/day)
292531	ED Transient		8.64E-04

New Soil

Import Soil...


Delete Soil

Properties...

Suppress Warning Messages

OK

The next step in defining the problem is to enter the material properties for the soil that will be used in the model.

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 SVFLUX forms.
3. Double-click on the new soil to open the **Soil Properties** form.

The screenshot shows a software window titled "Soil Properties" with a blue header bar. Below the header is a tabbed interface with the following tabs: "Description", "Hydraulic Conductivity", "SWCC", "Volume-Mass Parameters", "Vapour Diffusion", and "Sink/Source". The "Description" tab is selected. The main area contains the following fields:

- Soil Index: 292531
- Soil Name: ED Transient
- USCS Texture: (empty dropdown menu)
- Soil Description: (empty text box)
- Geologic Description: (empty text box)
- Notes: This soil has the properties required for the transient analysis of the edge-drop tutorial example.

An "OK" button is located at the bottom right of the dialog box.

4. Enter the information above into the appropriate fields on the **Description** tab. (The suggested name is ED Transient, but both the name and notes are optional)
5. Move to the **SWCC** tab.
6. Refer to soil properties table in the [problem overview](#) section of this manual. Enter the **Saturated VWC** value of 0.45.
7. Select the **Fredlund and Xing Fit** as the SWCC option.
8. Press the button to the right of the Fredlund & Xing Fit option to open the form below.

Fredlund and Xing Fit

af: 300 kPa

nf: 1.5

mf: 1

hr: 3000 kPa

Fredlund SWCC Fit:

Fredlund Error: R²

Fredlund Residual WC: (volumetric water content)

Fredlund AEV: kPa

Saturated Conditions

Saturation Suction: 0.1 kPa

Saturated WWC: 0.000

Coefficient of Volume Change, mw: 0 1/kPa

Messages: Unable to find Fredlund and Xing fit parameters
A saturated volumetric water content must first be calculated.

Apply Fit Graph SWCC... Graph Storage... Properties...

Iteration: Log Suction

9. Enter the fit parameters from the soil properties table.
10. Click OK to close the form.
11. Move to the **Hydraulic Conductivity** tab.
12. Enter the **Saturated Hydraulic Conductivity** value of 6.64E-04 m/day.
13. Select the **Leong and Rahardjo** unsaturated hydraulic conductivity option.
14. Press the button to the right of the Leong and Rahardjo option to open the **Leong and Rahardjo Estimation** form.
15. Enter the Leong p value of 1.

Leong and Rahardjo Estimation

Leong p:

Leong Predicted:

Leong Error: Exponential R²

k minimum: m/day

Apply k minimum:

16. Click OK to close the form.
17. Click OK to close the Soil Properties form.


4.10.3 Importing Geometry

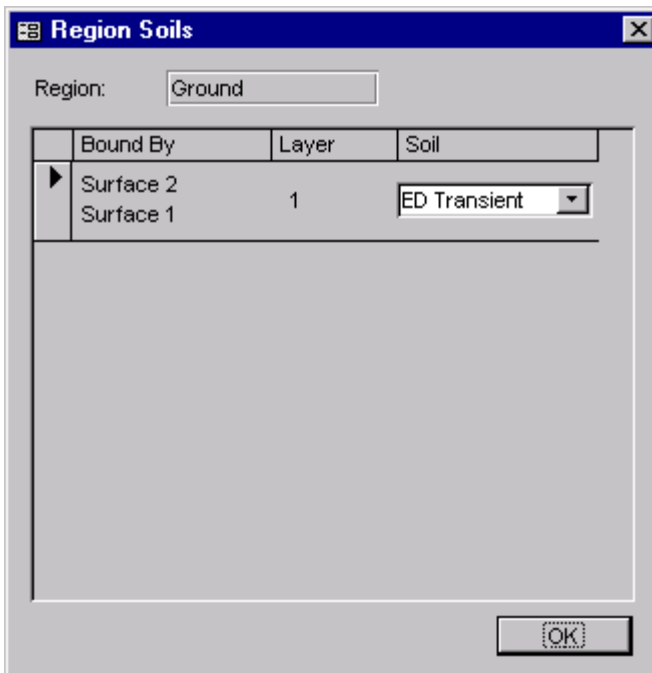
Since the regions and geometry for the Ground and Slab were defined previously for the Initial SVFLUX Analysis, the **Import SVFLUX Geometry** feature can be used to save time for this analysis.

1. Select Model > Import SVFLUX Geometry from the menu.
2. Press the **Browse** button and specify the path to the current **SVFlux_Data_J.mdb** database. The tree view control will be filled with all the problems in the database.
3. Navigate to the **ED3D_Initial** problem. (Tutorial – 2D – Steady-State – ED3D_initial)
4. Click the **Import** button.
5. Say Yes to the warning message.
6. The surfaces, surface grids, regions, region shapes, and world coordinate system settings will be imported.
7. Click **OK** to close the import form.

4.10.4 Specifying a Soil by Region and Layer

The soil that was previously defined will now be assigned to each region-layer block.

1. Select “**Ground**” in the Region Selector.
2. Press the **Region Soils** button,  at the top of the workspace to open the Region Soils form.




Region Soils

Region:

	Bound By	Layer	Soil
▶	Surface 2	1	ED Transient
	Surface 1		

OK

3. Select the **ED Transient** soil from the drop-down for **Layer 1**.
4. Close the form using the **OK** button.
5. Select "**Slab**" in the Region Selector.
6. Press the **Region Soils** button,  at the top of the workspace to open the Region Soils form.
7. Select the **ED Transient** soil from the drop-down for **Layer 1**.
8. Close the form using the OK button.

4.10.5 Specify Boundary Conditions

Now that the region and the problem geometry have been successfully imported, the next step is to specify the boundary conditions. A zero flux condition is required at the ground surface beneath the cover while a suction of 400 kPa exists at the bottom of the soil region. The suction values must be converted to head values for SVFLUX entry. An evaporation rate of 10 mm/day is represented by the normal flux condition over the uncovered ground surface.

Ground Region Boundary Conditions

X	Y	Boundary Condition
0	6	Zero Flux
0	12	Continue
1	12	Continue
1	6	No BC
Surface 1		Head Expression = -43.787
Surface 2		Normal Flux = -0.01

Slab Region Boundary Conditions

X	Y	Boundary Condition
0	0	Zero Flux
0	6	No BC
1	6	Zero Flux
1	0	Continue
Surface 1		Head Expression = -43.787
Surface 2		Zero Flux

The steps for specifying the boundary conditions are thus:

1. Select the “Ground” region in the region selector.
2. Select “Surface 1” in the surface selector.
3. From the menu select **Model > Boundaries**. The boundary conditions form will open and display the boundary conditions for the ground region on Surface 1. These boundary conditions will extend from Surface 1 to Surface 2 over Layer 1. By default the first boundary segment will be given a **No BC** condition.

Location

Region: Ground Surface: Surface 1

Segment Boundary Conditions | **Surface Boundary Conditions**

X	Y	Boundary Condition	Expression or Data	Units
0	6	Zero Flux		
0	12	Continue		
1	12	Continue		
1	6	No BC		

Update Selected Segment Segment Length: 6 m

1. Select Boundary Condition: Zero Flux

2. Provide: a) Expression: Build Equation...
- or -
b) Flux Data Index: Expr Reference...
c) Evaporation Index: (optional)

4. Select the **point** (0,6) from the list.
5. From the **Boundary Condition** drop down select a **Zero Flux** boundary condition.
6. Click the **Update** button to save the boundary condition to the list.
7. Select the **point** (1,6) from the list.
8. From the **Boundary Condition** drop down select a **No BC** boundary condition.
9. Click the **Update** button to save the boundary condition to the list.
10. Move the **Surface Boundary Conditions** tab.
11. From the **Boundary Condition** drop down select a **Head Expression** boundary condition. This will cause the Expression box to be enabled.
12. In the Expression box enter a **head of -43.787**.
13. Select **Surface 2** from the drop down box.
14. Move the **Surface Boundary Conditions** tab.
15. From the **Boundary Condition** drop down select a **Normal Flux** boundary condition. This will cause the Expression box to be enabled.
16. In the Expression box enter a **flux of -0.01**.
17. Click the **OK** button to close the Boundaries form.
18. **Repeat** these steps to add the boundary conditions for the Slab region indicated in the above table. (Be sure to enter a Zero Flux condition for Surface 2)

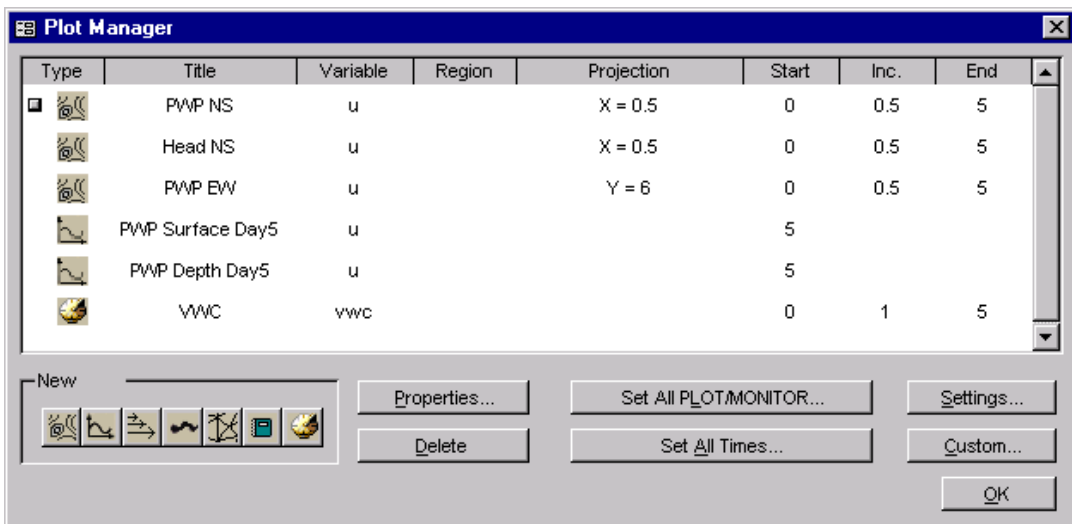


The Continue boundary condition indicates that the previously defined boundary condition will apply to the current boundary segment.

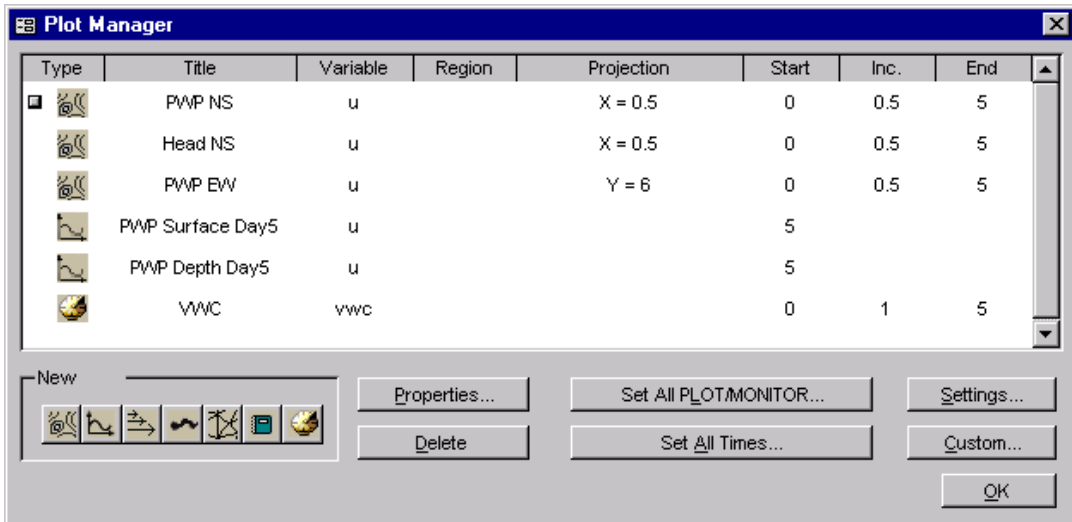
4.11 SPECIFY SVFLUX TRANSIENT ANALYSIS PLOTS

There are many plot types that can be specified to visualize the results of the model. A few contour and elevation plots, as well as a history plot will be generated for this tutorial example problem.

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



2. 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.



3. Enter the title PWP NS.
4. Select **u** as the variable to plot from the drop-down.
5. Select the **PLOT** output option.
6. Move to the **Time** tab
7. Enter a Start Time of 0, a Time Increment of 0.5, and an End Time of 5 days.
8. Click OK to close the form and add the plot to the list.
9. Repeat steps 2 – 8 to create the suggested contour plots listed below. (Note that the plots are not required for problem solution, but are useful for visualization)
10. Click on the **Elevation** button to begin adding the first elevation plot. The Plot Properties form will open.
11. Enter the title PWP Surface Day5.
12. Select **u** as the variable to plot from the drop-down.
13. Select the **PLOT** output option.
14. Move to the **Time** tab
15. Enter a **Start** Time of 5.

Plot Time (day)

Start:

Increment:

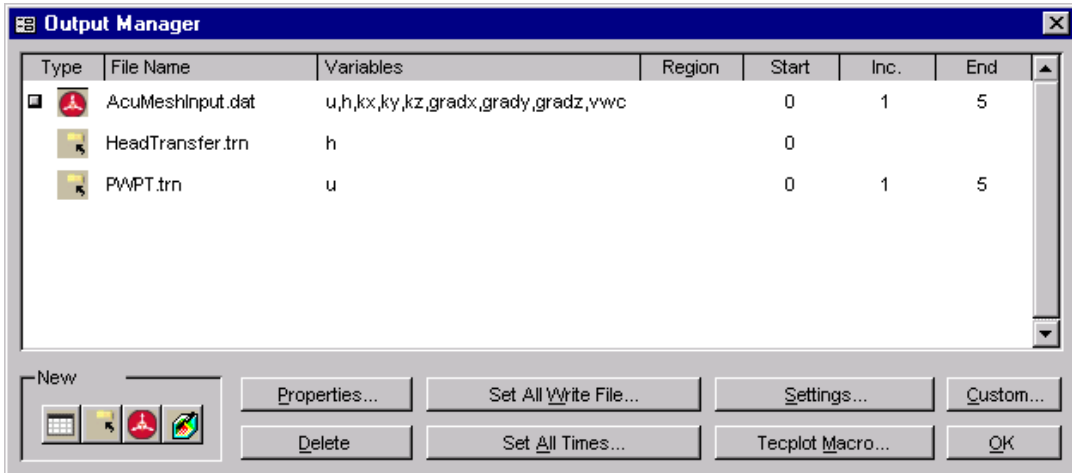
End:

16. Move to the **Range** tab.
17. Enter the values – **X1: 0.5, Y1: 0, Z1: 0, X2: 0.5, Y2: 12, Z2: 0.**

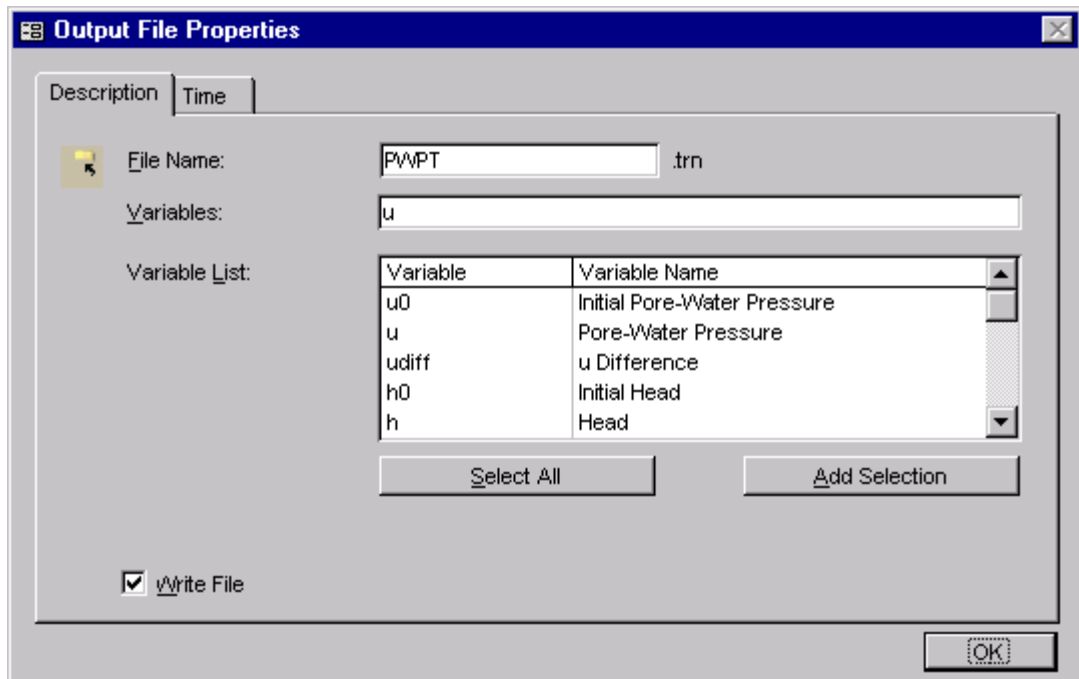
4.12 SPECIFY SVFLUX TRANSIENT ANALYSIS OUTPUT FILES

There are 4 output file types that can be specified to export the results of the model. Two will be generated for this tutorial example problem: a transfer file of pore-water pressure for use in the SVSOLID Analysis, and a plot to transfer the results to AcuMesh for advanced visualization. Note that the file HeadTransfer.trn is already present. It is generated by default for every problem.

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 **Transfer** button to begin adding an output file. The Output File Properties form will open.



3. Enter the title PWPT.
4. Select the variable **u** 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. Repeat steps 2 – 7 to add the AcuMesh output file.
9. Click **OK** to close the Output Manager and return to the workspace.

4.13 ANALYZE THE TRANSIENT SVFLUX PROBLEM

The next step is to analyze the problem. Click the Analyze button located on the left of the workspace. This action will write the descriptor file and open the SVFLUX solver. The solver will automatically begin solving the problem and the Run Log form will open in SVFLUX.

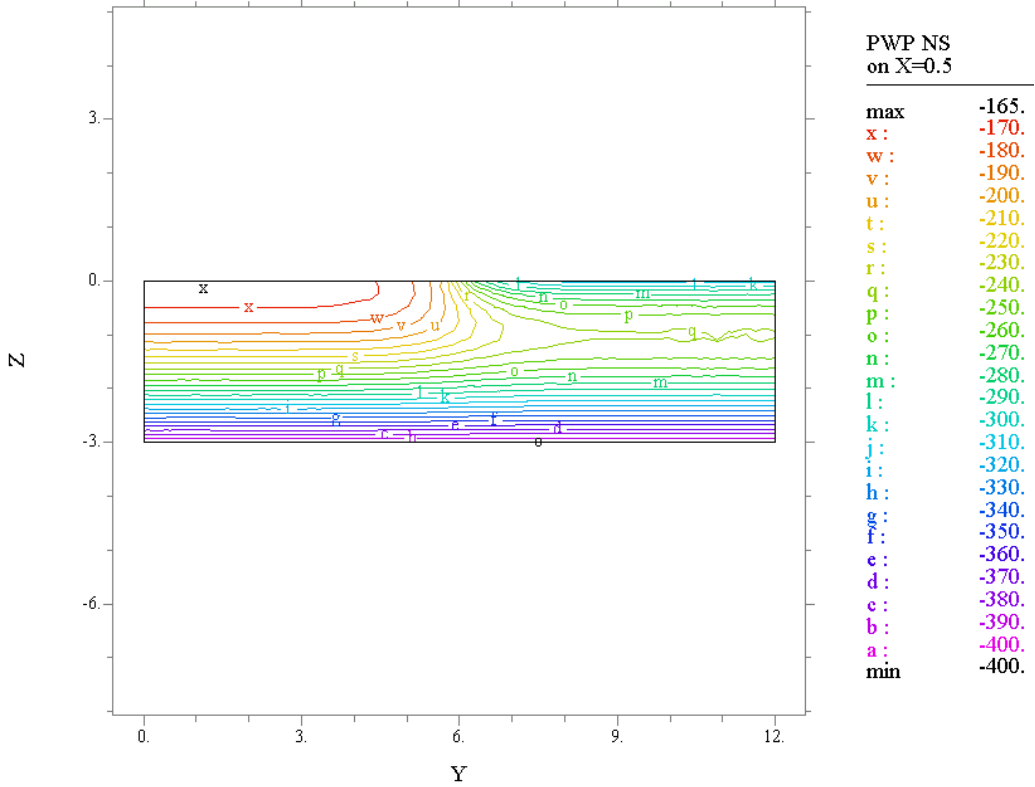
When the solver has finished running press the **Read File** button on the Run Log form to record the run data.

4.14 SVFLUX TRANSIENT ANALYSIS RESULTS

After the problem has finished solving, the results will be displayed in the form of thumbnail plots within the SVFLUX solver. Right-click the mouse and select Maximize to enlarge any of the thumbnail plots. This section will give a brief analysis for a few plots that were generated.

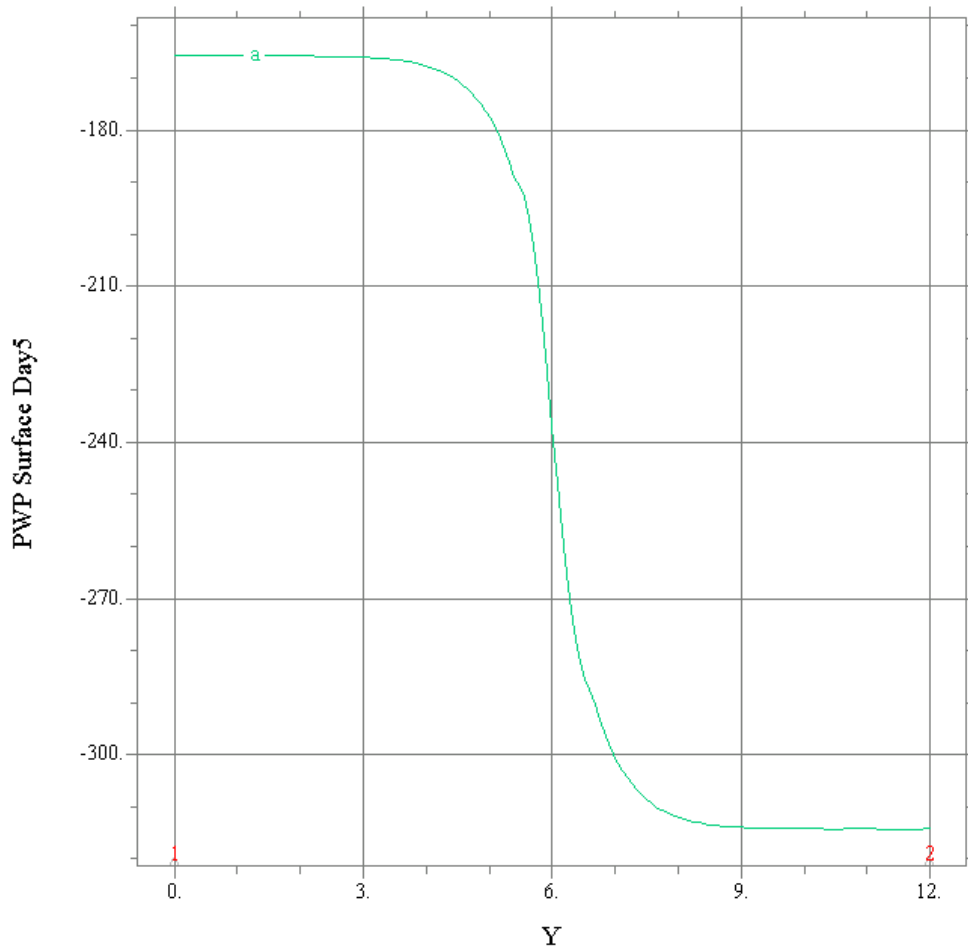
The output files requested (PWPT.trn, HeadTransfer.trn, and AcuMeshInput.dat) will be located in the solution file directory for the problem.

4.14.1 Pressure Contours

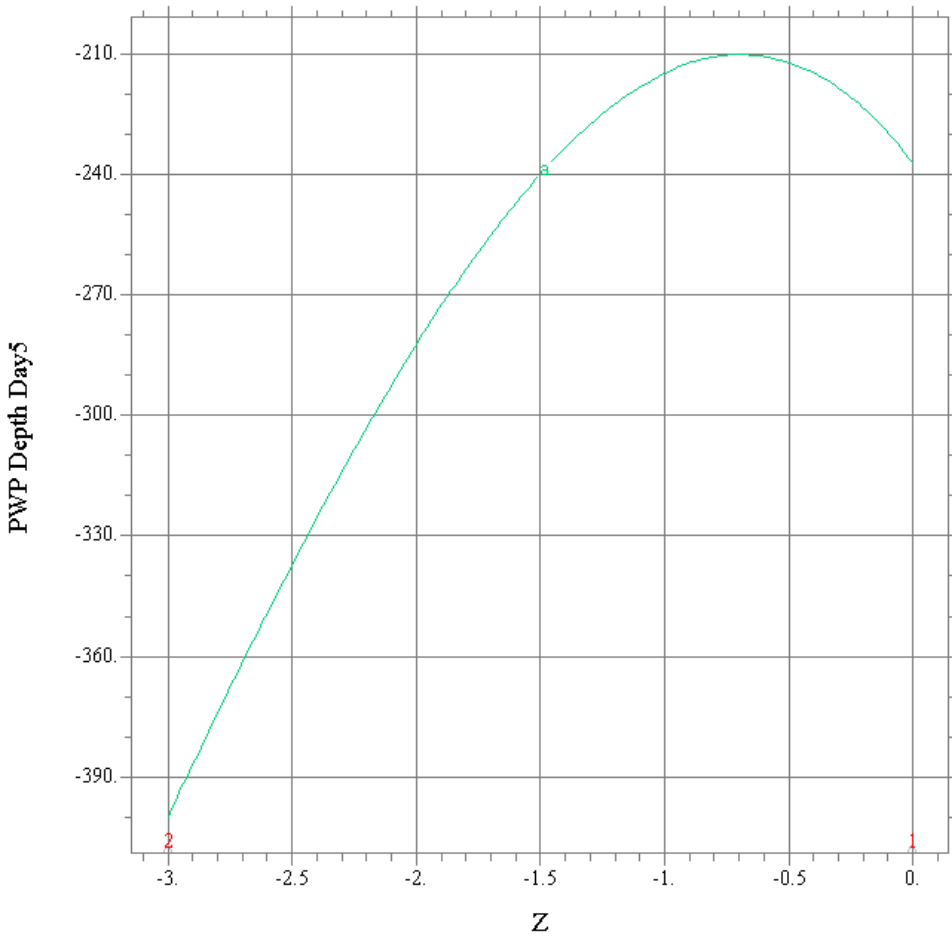


The pore-water pressure contour plot above indicates increased suction near the uncovered ground surface due to the evaporation in that area.

4.14.2 Pore-water Pressure Elevation Profiles



The plot above shows the pore-water pressure along the ground surface after 5 days of evaporation. There is a high suction gradient within 1 meter of the outside edge of the cover and the suction change is uniform elsewhere. In the plot below the pore-water pressure profile below the cover edge after 5 days of evaporation is shown. The majority of the suction change occurs near the ground surface.



4.15 SVSOLID STRESS/DEFORMATION ANALYSIS

Now that the seepage component of the problem has been completed, a stress analysis must be defined using SVSOLID. This analysis will use the initial pore-water pressure transfer file from the Initial SVFLUX Analysis and the final pore-water pressure transfer file from the SVFLUX Transient Analysis. This stress analysis will be run for 25 stages and the displacements calculated and output at each stage. These incremental displacements will be summed to obtain the total movements using a summary file.



Tip!

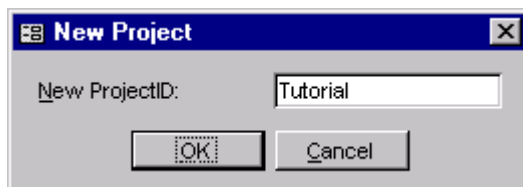
Note that the operation of the SVSOLID software is similar to SVFLUX. Many of the following steps will be the same.

4.15.1 Adding an SVSOLID 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.

Follow these steps in order to add the project:

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

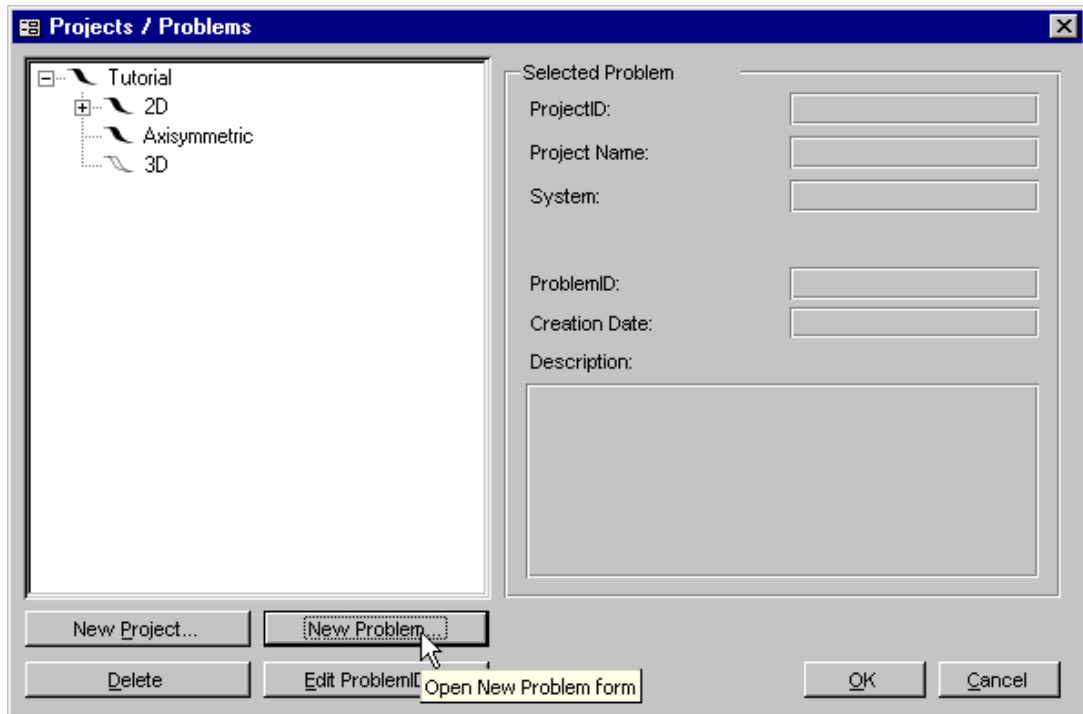
Project Notes: The tutorial example problems are stored under this project.

Edit ProjectID... OK

It should be noted that once the project is defined it will be identified by the ProjectID throughout the rest of the program. Also, SVSOLID does not allow the specification of 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.

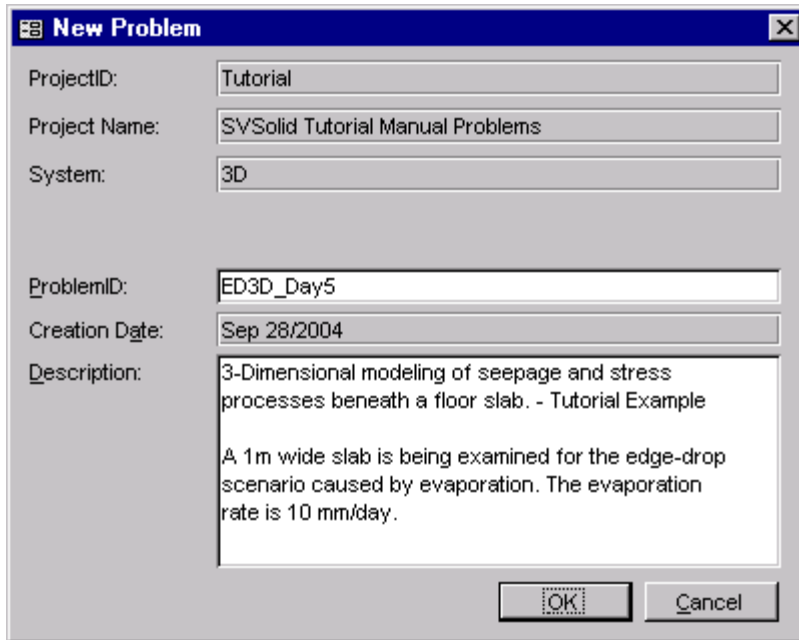
4.15.2 Adding an SVSOLID Problem



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 three-dimensions so we must expand **3D** by clicking the plus sign beside "3D"
3. Click the **New Problem** button. The New Problem form will open.



New Problem

ProjectID: Tutorial

Project Name: SVSolid Tutorial Manual Problems

System: 3D

ProblemID: ED3D_Day5

Creation Date: Sep 28/2004

Description: 3-Dimensional modeling of seepage and stress processes beneath a floor slab. - Tutorial Example
A 1m wide slab is being examined for the edge-drop scenario caused by evaporation. The evaporation rate is 10 mm/day.

OK Cancel

4. Enter the **ProblemID: ED3D_Day5**. The description is optional.
5. Click the **OK button** to save the problem and close the New Problem form.
6. The new problem will automatically be opened in the workspace.

4.15.3 Opening the SVSOLID Problem

If the problem was just added it will already be open in the workspace. When returning to the problem follow these steps to open it in the workspace:

1. Select Model > Projects/Problems from the menu.
2. **Navigate** back to the problem via Tutorial, 3D.
3. Select **ED3D_Day5**.
4. The problem may be opened by clicking the **OK** button or by double clicking on the ProblemID.

4.16 DEFINING THE SVSOLID PROBLEM

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

4.16.1 Specify Settings

The first step in defining the problem is to specify the settings that will be used for the problem.

System <input checked="" type="radio"/> 2D-Plane Strain <input type="radio"/> Axisymmetric <input type="radio"/> 3D	Analysis <input type="radio"/> Neglect PWP <input checked="" type="radio"/> Consider PWP	Units <input checked="" type="radio"/> Metric <input type="radio"/> Imperial Length: <input type="text" value="m"/> Stress: <input type="text" value="kPa"/>
---	---	---

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. The initial stress conditions, initial pore-water pressure conditions, and final pore-water pressure conditions

1. To open the Settings form select **Model > Settings** in the menu.
2. Select **Consider PWP** as the Analysis option.
3. Move to the **Initial Stress/Strain** tab.
4. Select **Ko-Loading** as the Initial Stress Option. (A coefficient of earth pressure at rest, Ko value will be entered later on the Soil Properties form)

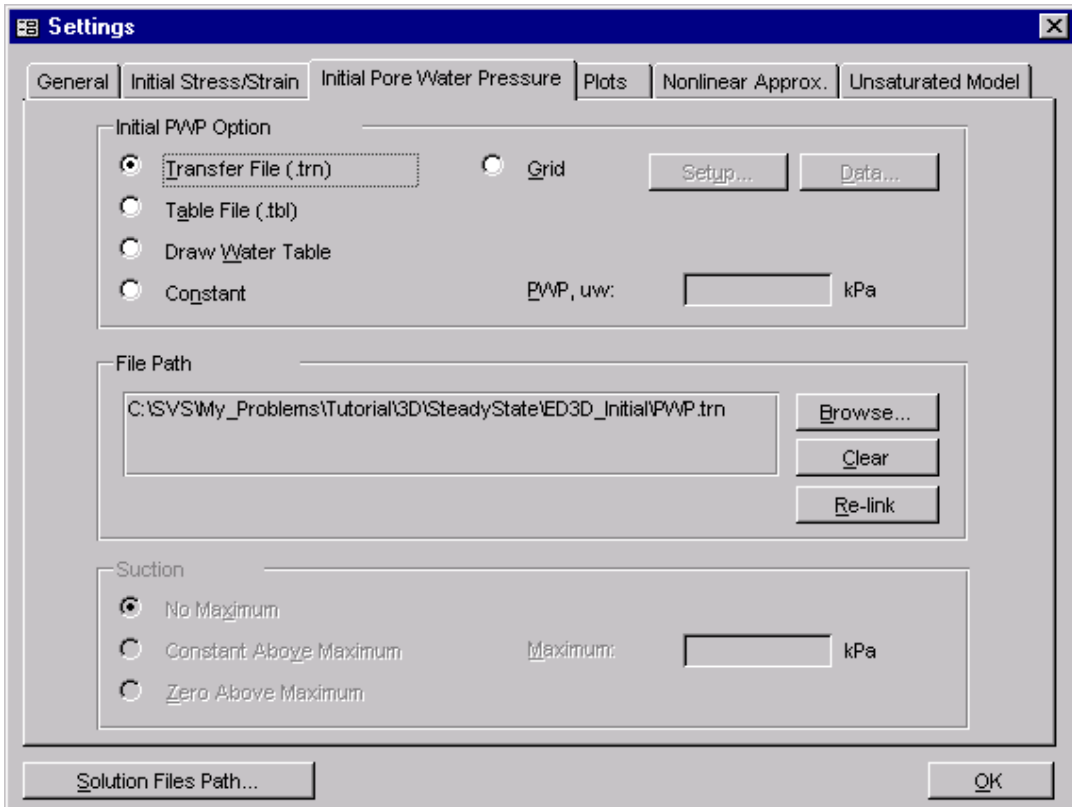
Initial Stress Option <input type="radio"/> None <input type="radio"/> Transfer File (.trn) <input checked="" type="radio"/> Ko-Loading Specify Ko values for each soil on the Soil Properties form.



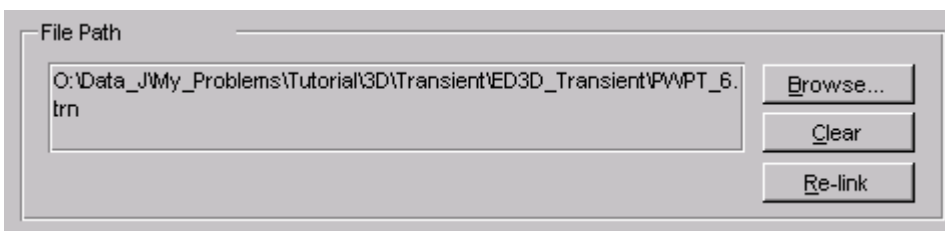
Tip!

The Ko-Loading option means that initial vertical stress will be a function soil unit weight and elevation and the horizontal initial stresses will be equal to the initial vertical stress multiplied by the Ko value. To use this option the ground surface will be flat at an elevation of 0. A Ko of 0.33 for a drying path will be used as suggested by Lytton (1997)

5. Move to the **Initial Pore Water Pressure** tab.
6. Select **Transfer File (.TRN)** as the Initial PWP Option.
7. Press the **Browse** button.
8. Then specify the path to the **PWP.trn** file output by the Initial SVFLUX Analysis.



9. Move to the **Unsaturation Model** tab.
10. Select **Transfer File (.TRN)** as the Final PWP Option.
11. Press the **Browse** button.
12. Then specify the path to the **PWPT_6.trn** file output by the SVFLUX Transient Analysis. (The file corresponds to the pore-water pressure after 5 days of evaporation. The file counter considers time 0 as **_1**, therefore **_6** corresponds to day 5)



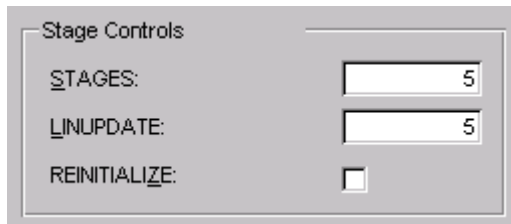
13. Click **OK** to close the form.

4.16.2 Setting the Stages and Error Limit

The number of stages will control the incremental change in suction. The total change in suction is difference between the initial and final suctions from the specified transfer files. The stages will be set to 5 for this analysis, therefore, the incremental displacements will be calculated for each suction increment at each stage. The error limit for the solver will be set to 0.01 to speed up the solution in this case.

To set the stages and error limit follow these steps:

1. Access the FEM Options form by selecting **Model > FEM Options** from the menu.

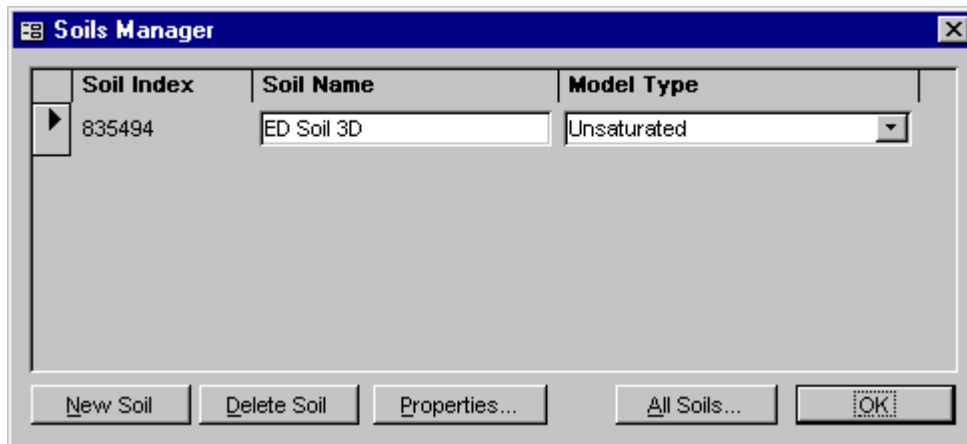


Stage Controls

STAGES:	<input type="text" value="5"/>
LINUPDATE:	<input type="text" value="5"/>
REINITIALIZE:	<input type="checkbox"/>

2. Enter the **5** in the STAGES field.
3. Click Yes for the message.
4. Enter **0.01** in the ERRLLIM field.
5. Click **OK** to close the form.

4.16.3 Define Material Properties



Soils Manager

Soil Index	Soil Name	Model Type
835494	ED Soil 3D	Unsaturated

New Soil Delete Soil Properties... All Soils... OK

The next step in defining the problem is to enter the material properties for the soil that will be used in the model. Refer to the [Problem Overview](#) section of this tutorial for the relevant soil properties.

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. Enter a Soil Name of **ED Soil 3D**.
4. Select **Unsaturated** as the Model Type
5. Select the new soil and click Properties to open the **Soil Properties** form.

Soil Properties: Unsaturated

Soil Index: 546939

Description Parameters Body Load Shear Strength Reference

Elastic Parameters

Poisson's Ratio, ν : 0.400 Staged

Coefficient of earth pressure at rest, K_0 : 0.330

Relate ν and K_0 by [$\nu=K_0/(1+K_0)$]:

Void Ratio Parameters

Initial Void Ratio, v_{ro} : 1.00

Swelling Index, C_s : 0.15 Staged

Volume Change Index with respect to Matric Suction, C_m : 0.13 Staged

Graph Void Ratio...

Stage Parameters... OK

6. Move to the **Parameters** tab.
7. Enter the **Poisson's Ratio** value of 0.4.
8. Enter the **K_0** value of 0.33.



Note that the Poisson's Ratio and K_0 are not related for this analysis. Be sure to leave the Relate ν and K_0 by [$\nu=K_0/(1+K_0)$] checkbox unchecked. The K_0 is used for determining initial conditions while the Poisson's Ratio is used in the general stress solution.

9. Enter the **Initial Void Ratio** value of 1.
10. Enter the **C_s** value of 0.15.
11. Enter the **C_m** value of 0.13.
12. Move to the **Body Load** tab.
13. Enter the **Z-Axis Body Load** as -17.2 kN/m^3 .
14. Click **OK** to close the Soil Properties form.
15. Click **OK** to close the Soils Manager form.



For the Unsaturated soil model the void ratio is a function of the net mean stress as well as the matric suction. Use the Graph Void Ratio button to view graphs of Void Ratio vs. Net Mean Stress at a given matric suction and of Void Ratio vs. Matric Suction for a given net mean stress.


4.16.4 Importing Geometry From SVFLUX

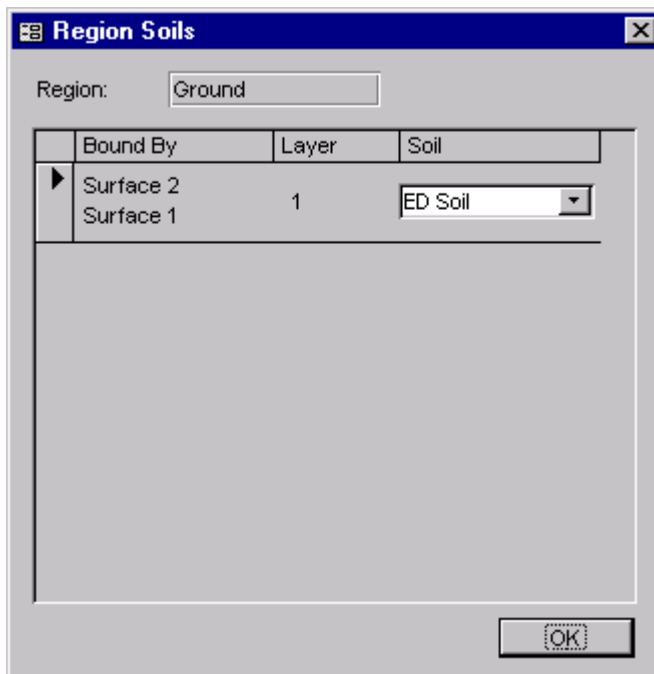
Since the Ground and Slab regions and their geometry were defined previously for the Initial SVFLUX Analysis, the **Import SVFLUX Geometry** feature can be used to save time for this analysis by importing the geometry from the SVFLUX software.

Follow the steps in the section [Defining the Transient SVFLUX Problem – Importing Geometry](#).

4.16.5 Specifying a Soil by Region and Layer

The soil that was previously defined will now be assigned to each region-layer block.


1. Select “**Ground**” in the Region Selector.
2. Press the **Region Soils** button,  at the top of the workspace to open the Region Soils form.



Region Soils dialog box showing the configuration for the Ground region. The Region is set to "Ground". The table below shows the configuration for the layers:

	Bound By	Layer	Soil
▶	Surface 2	1	ED Soil
	Surface 1		

An OK button is located at the bottom right of the dialog box.

3. Select the **ED Soil** soil from the drop-down for **Layer 1**.
4. Close the form using the **OK** button.
5. Select “**Slab**” in the Region Selector.
6. Press the **Region Soils** button,  at the top of the workspace to open the Region Soils form.

7. Select the **ED Soil** soil from the drop-down for **Layer 1**.
8. Close the form using the OK button.

4.16.6 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. The base of the problem will be fixed in all directions. The left and right vertical boundaries will be fixed in the X-direction, but will be free to move in the Y-direction. The top and bottom vertical boundaries will be fixed in the Y-direction, but will be free to move in the X-direction. The ground surface is free to move in all directions. The steps for specifying the boundary conditions are thus:

1. Select the “Ground” region in the region selector and “Surface 1” in the surface selector.
2. From the menu select **Model > Boundaries**. The boundary conditions form will open.

Boundary Conditions

Location

Region: Ground

Surface: Surface 1

Surface Boundary Condition

Dir:	X	Y	Z
Cond:	Fixed	Fixed	Fixed
Expr:			

Segment Boundary Conditions

	Point		Boundary Condition			
	X	Y	Dir:	X	Y	Z
	0	6	Cond:	Fixed	Free	Free
			Expr:			
	0	12	Cond:	Free	Fixed	Continue
			Expr:			
	1	12	Cond:	Fixed	Free	Continue
			Expr:			
▶	1	6	Cond:	Free	Continue	Continue
			Expr:			

Segment Length: 1 m

Copy To Surface... Notes... Expr Reference... OK

3. In the **Surface Boundary Condition** section, Select **Fixed** for the X, Y, and Z directions.
4. Select the point **(0,6)** from the list in the Segment Boundary Conditions section.
5. From the **X Boundary Condition** drop down select a **Fixed** boundary condition.
6. From the **Y Boundary Condition** drop down select a **Free** boundary condition.
7. Select the point **(0,12)** from the list and set X as Free and Y as Fixed.
8. Select the point **(1,12)** from the list and set X as Fixed and Y as Free.
9. Select the last point **(0,12)** from the list and set X as Free.

10. From the **X Boundary Condition** drop down select a **Fixed** boundary condition.
11. Click the **OK** button to close the form.
12. **Repeat** the above steps for the **Slab** region. (Refer to the table below)

**Tip!**

The Continue boundary condition means that the current segment takes on the boundary condition of the previous segment.

Boundary Condition Summary

Ground Region

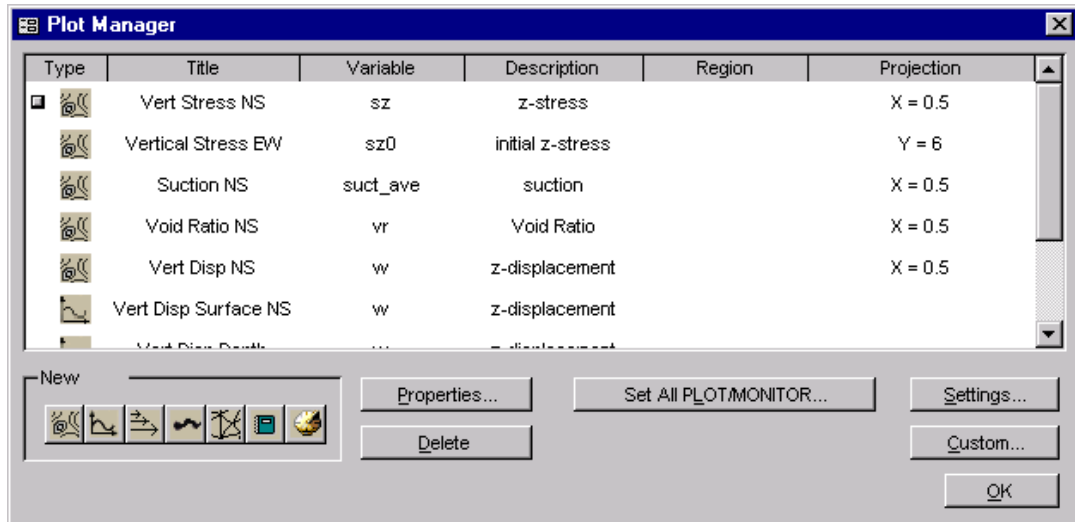
X	Y	X Boundary Condition	Y Boundary Condition	Z Boundary Condition
0	6	Fixed	Free	Free
0	12	Free	Fixed	Continue
1	12	Fixed	Free	Continue
1	6	Free	Continue	Continue
Surface 1		Fixed	Fixed	Fixed
Surface 2		None	None	None

Slab Region

X	Y	X Boundary Condition	Y Boundary Condition	Z Boundary Condition
0	0	Fixed	Free	Free
0	6	Free	Continue	Continue
1	6	Fixed	Continue	Continue
1	0	Free	Fixed	Continue
Surface 1		Fixed	Fixed	Fixed
Surface 2		None	None	None

4.17 SPECIFY SVSOLID PLOTS

There are many plot types that can be specified to visualize the results of the model. A few will be generated for this tutorial example problem including a plot of the solution mesh, stress contours, and displacement vectors. Plots are not required for problem solution, but no results can be seen without them. Define at least 1 plot from the Suggested Plots table below.



Refer to the section [Specify SVFLUX Transient Analysis Plots](#) earlier in this tutorial for instructions on adding plots.

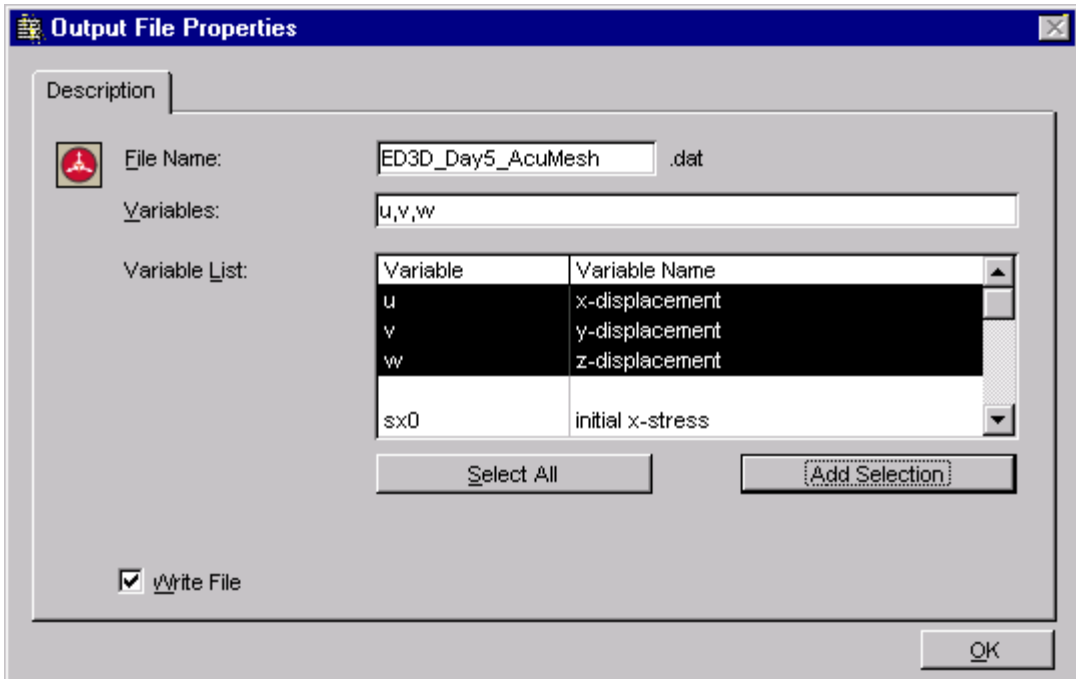
Suggested Plots

Plot Type	Title	Variable	Projection	Range
Contour	Vert Stress NS	sz	X=0.5	
Contour	Vertical Stress EW	sz	Y=6	
Contour	Suction NS	suct_ave	X=0.5	
Contour	Void Ratio NS	vr	X=0.5	
Contour	Vert Disp NS	w	X=0.5	
Elevation	Vert Disp Surface NS	w		(0.5,0,0) to (0.5,12,0)
Elevation	Vert Disp Depth	w		(0.5,6,0) to (0.5,6,-3)
Elevation	Void Ratio Depth	vr		(0.5,6,0) to (0.5,6,-3)
Vector	Disp NS	v,w	X=0.5	
Mesh	Final Mesh	YZ Deformed Mesh	X=0.5	

4.18 SPECIFY SVSOLID ANALYSIS OUTPUT FILES

There are 4 output file types that can be specified to export the results of the model. A plot to transfer the results to AcuMesh for advanced visualization of displacements will be created in this example.

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 **AcuMesh** button to begin adding an output file. The Output File Properties form will open.



3. Enter the title ED3D_Day5_AcuMesh.
4. Select the displacement variable **u**, **v**, and **w** 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.



Tip!

Add other variables of interest to the AcuMesh file if desired.

4.19 ANALYZE THE SVSOLID PROBLEM

The next step is to analyze the problem. Click the Analyze button located at the left in the workspace. This action will write the descriptor files and open the SVSOLID solver. The solver will automatically begin solving the problem and the Run Log form will open in SVSOLID. There are 3 .pde files that will be created:

1. Tutorial_ED3D_Day5_BATCH.pde
2. Tutorial_ED3D_Day5.pde
3. Tutorial_ED3D_Day5_Summary.pde

The Batch file will call the other 2 files in sequence and they will be solved automatically. When the solver has finished running:

1. Press OK for the **Batch Done** message.

2. Click the **Read File** button on the Run Log in SVSOLID form to record the run data.



Note that the solution time for this example is approximately 15 minutes on a Pentium 4 computer with 1GB of RAM. Solution times will vary depending on the computer used.

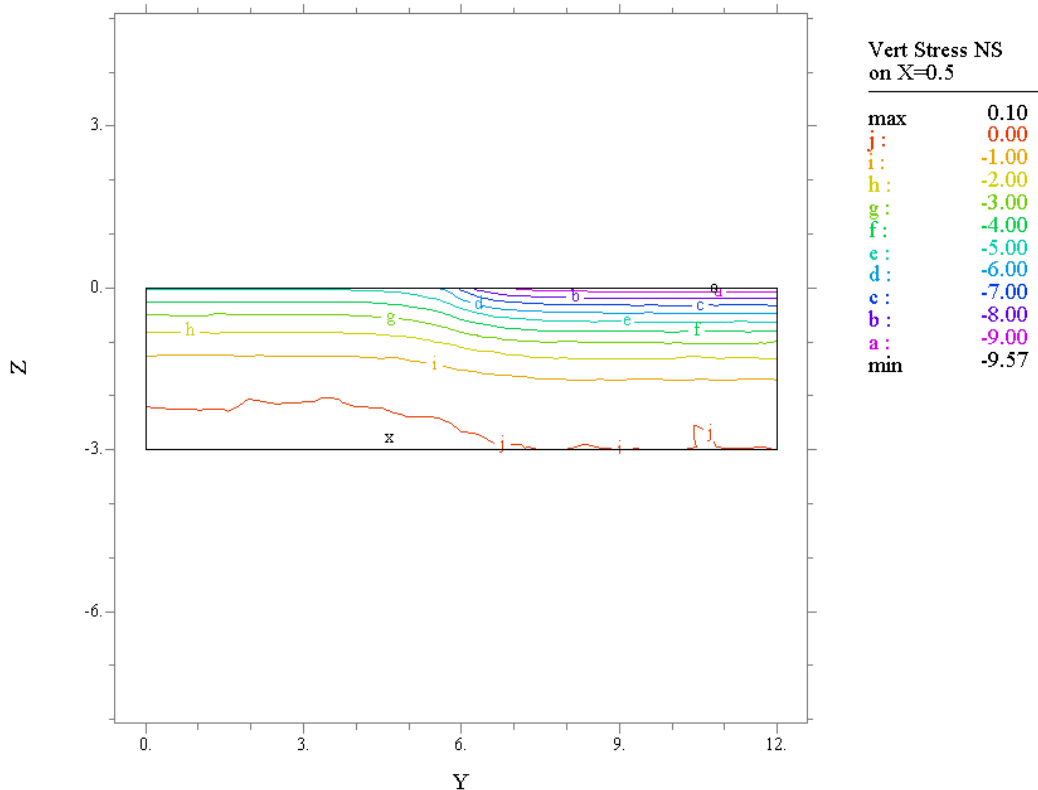
4.20 SVSOLID RESULTS

To view the SVSOLID results once the problem has finished select **Analyze > View Results** from the SVSOLID menu. The results will be displayed in the form of thumbnail plots within the SVSOLID solver. Right-click the mouse and select Maximize to enlarge any of the thumbnail plots. This section will give a brief analysis for some plots that were generated.



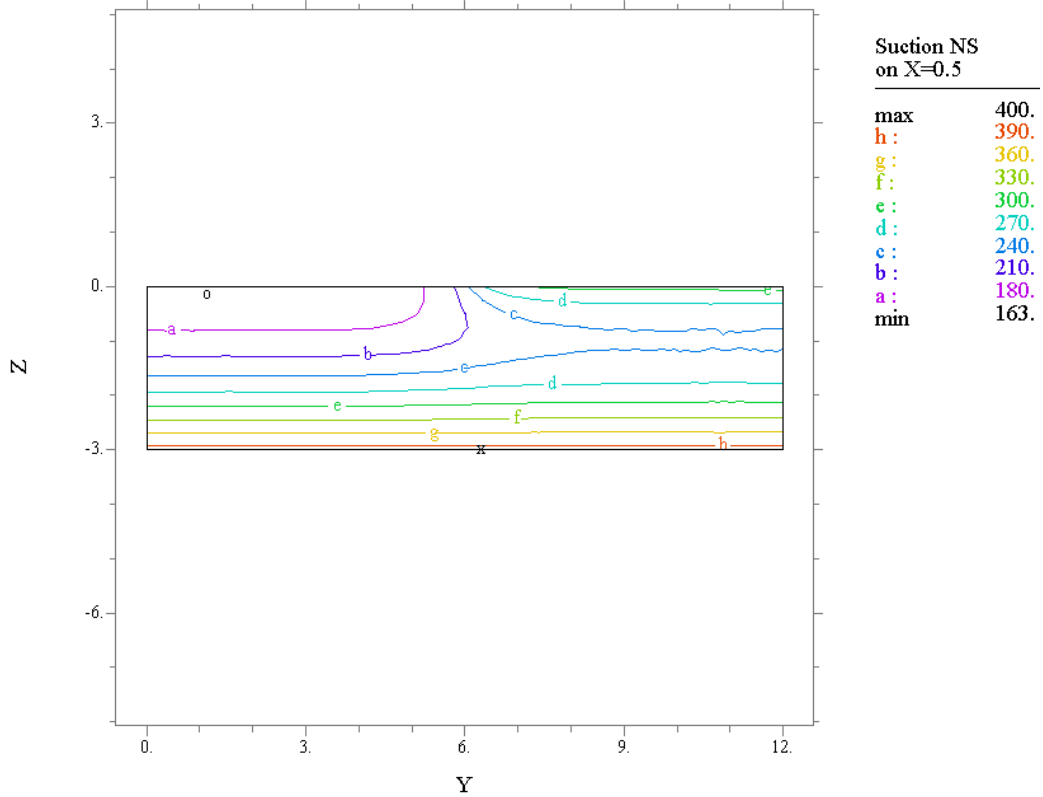
While the problem is running, the results will be displayed in the form of thumbnail plots within the SVSOLID solver. Right-click the mouse and select Maximize to enlarge any of the thumbnail plots.

4.20.1 Vertical Stress Contours



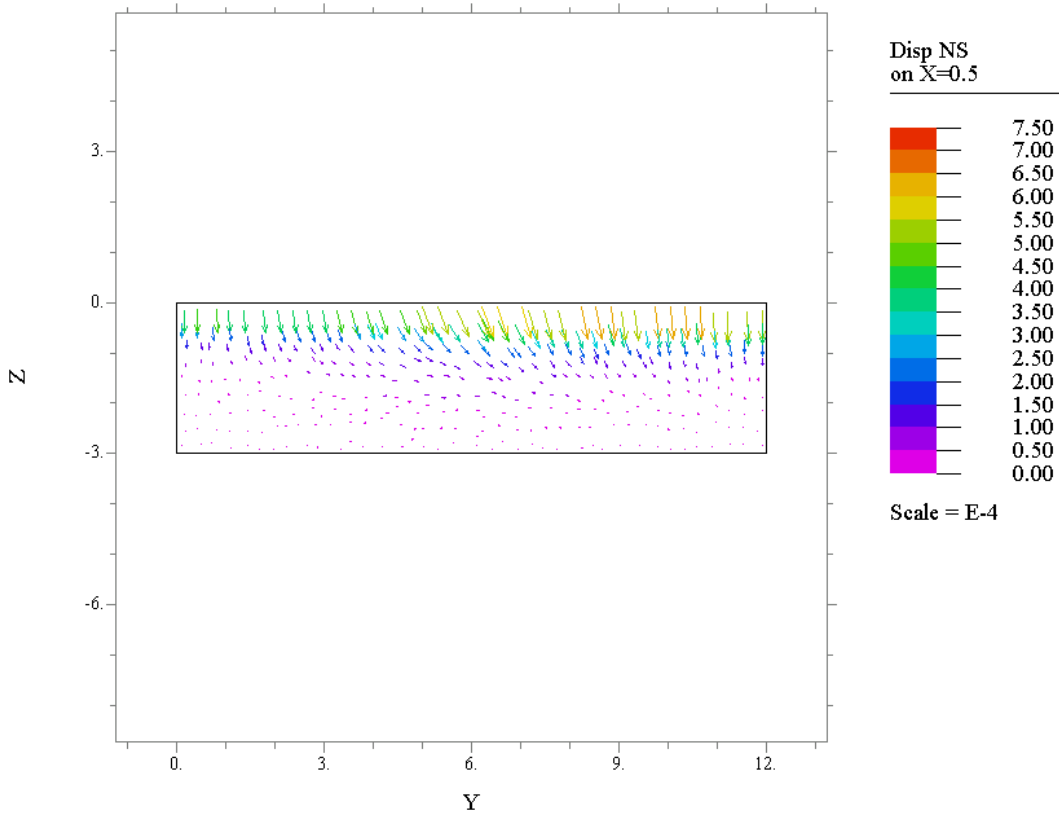
A negative or tensile vertical stress develops most significantly near the uncovered ground surface.

4.20.2 Suction Contours



The contour plot of suction indicates the development of higher suction at the uncovered boundary due to evaporation.

4.20.3 Displacement Vectors

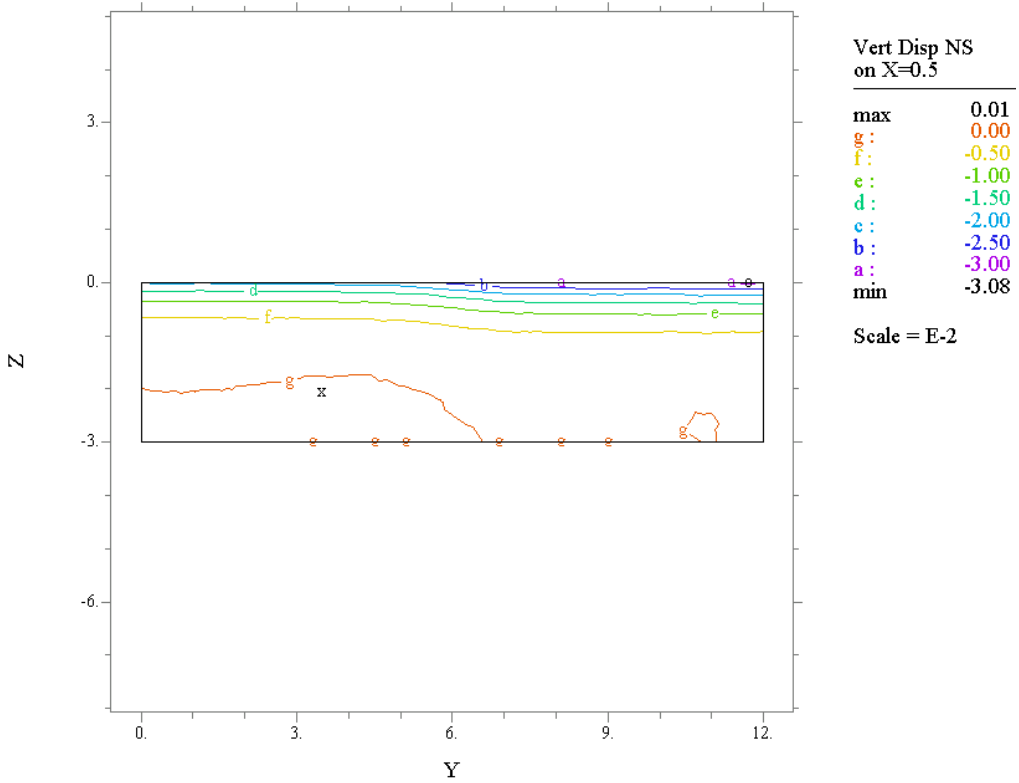


Displacement Vectors show both the direction and the magnitude of the displacement at specific points in the problem. Settlement is greatest under the uncovered ground surface.

4.21 SVSOLID SUMMARY FILE RESULTS

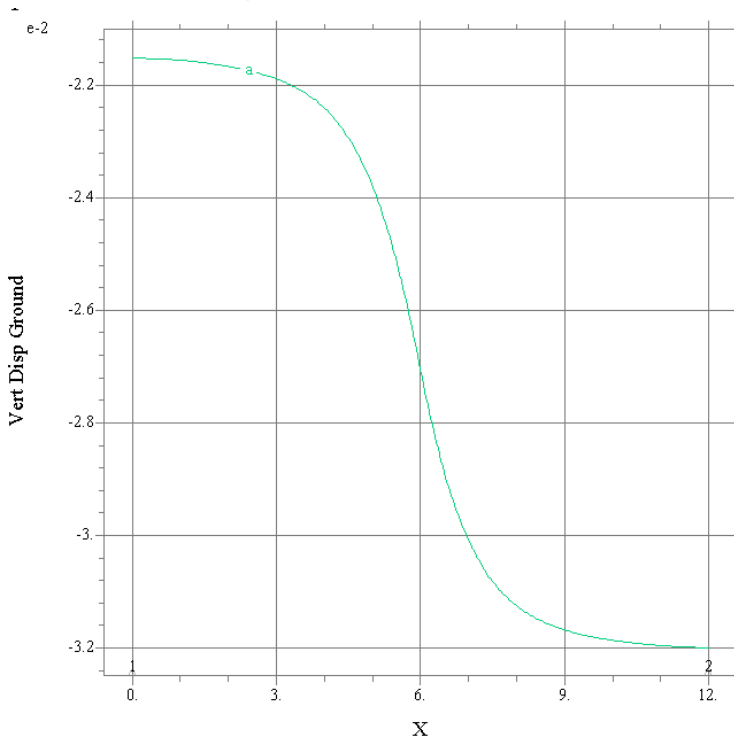
The SVSOLID Summary file will generate the same plots as the regular SVSOLID analysis that apply to the displacement variables. Only in this case the plots will be the summation of the results from each stage.

4.21.1 Vertical Displacement Contours



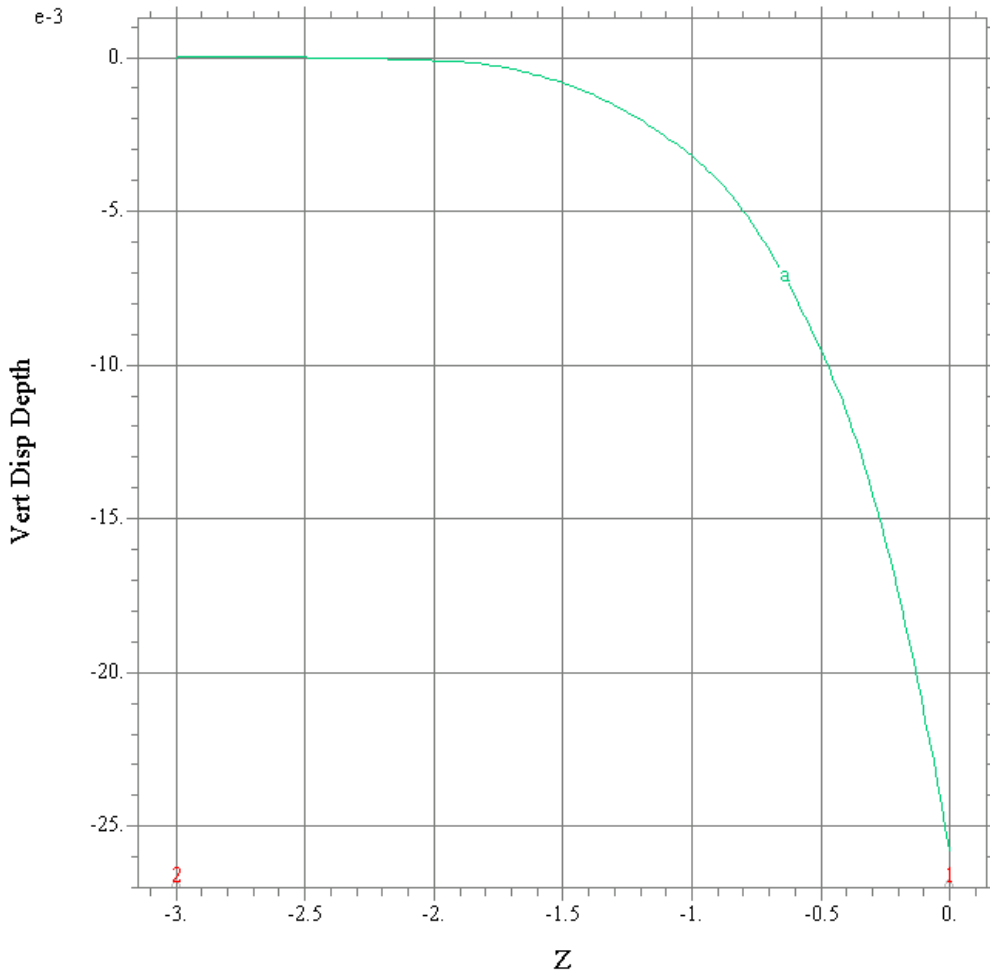
The vertical displacement contours plot shows a maximum settlement of 31mm at the top right corner of the soil region.

4.21.2 Vertical Displacement at Ground Surface



The above plot shows vertical displacement along the length of the ground surface. Differential settlements took place at the edge of the cover. About 10mm of settlement has occurred after 5 days of evaporation.

4.21.3 Vertical Displacement Below Cover Edge

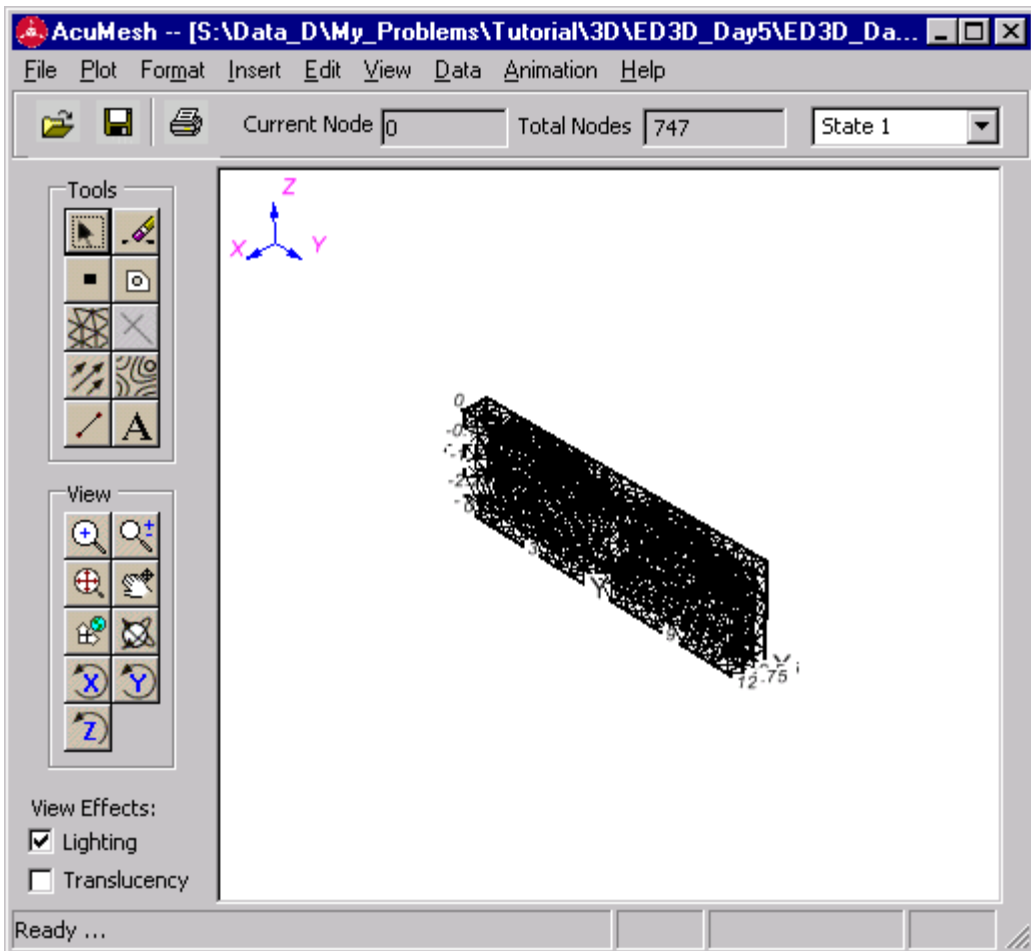


The above plot shows vertical displacement below the edge of the cover. Most of the settlement took place near the ground surface where the change in matric suction is largest and where the soil has a low elastic modulus.

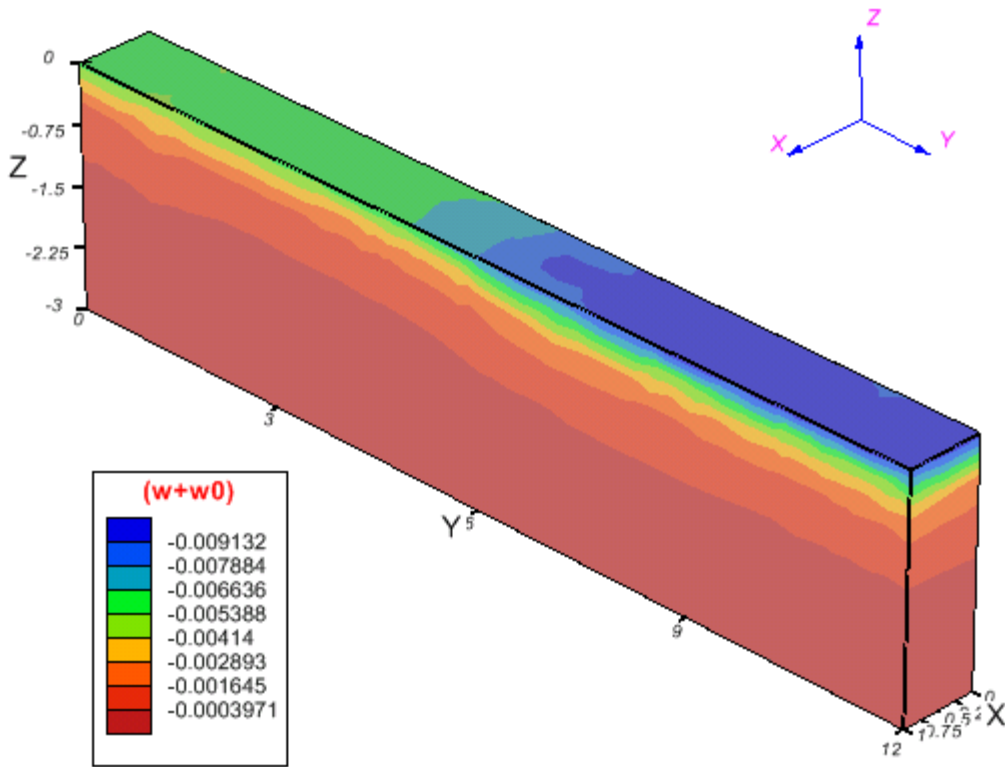
4.22 CONTOUR DISPLACEMENTS IN ACUMESH

The AcuMesh visualization software can be used to view advanced plots in 3D. To view contours of vertical displacement in 3-dimensions follow these steps:

1. Select **Solve > Send to AcuMesh** from the SVSOLID menu.
2. The AcuMesh software will open and display the 3D mesh for the problem.



3. Select **Plot > Mesh** to open the Format Mesh form.
4. Uncheck the box for **Region 1**.
5. Click OK to close the Format Mesh form.
6. Select **Plot > Contours** to open the Format Contours form.
7. Check the **Region 1** box
8. Choose **(w+w0)** from the Variable drop-down.
9. Click OK to close the Format Contours form.



4.23 PROBLEM CONCLUSION

The 3D **Edge Drop** of a Flexible Impervious Cover tutorial problem is now complete.

5 REFERENCES

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

FlexPDE 4.x Reference Manual, 2004. PDE Solutions Inc. Antioch, CA 94509

6 INDEX

- Analysis, 4, 5, 6, 13, 56, 67, 68, 69, 71, 76, 79, 81, 82, 86, 90, 91, 94, 96, 106, 120, 121, 122, 124, 129, 133, 136, 137, 141, 145, 146, 149, 152
- Body Load, 8, 17, 31, 39, 93, 148
- Boundaries, 20, 47, 49, 65, 77, 95, 118, 119, 131, 132, 150
- Boundary Conditions, 3, 4, 5, 6, 20, 47, 48, 49, 55, 65, 77, 94, 105, 117, 119, 130, 132, 150
- Color, 20, 46
- Deformation, 1, 4, 6, 86, 141
- Elevations, 40, 41, 42, 113, 114
- Fill, 9, 20, 32, 46, 57, 87, 107, 142
- Fredlund and Xing, 74, 127
- Geometry, 3, 4, 5, 6, 7, 18, 30, 44, 55, 64, 76, 94, 105, 115, 116, 129, 149
- Grid, 14, 36, 40, 41, 42, 112, 113, 114
- Ground Surface, 5, 6, 103, 158
- Head Expression, 66, 78, 119, 132
- History, 135
- Hydraulic Conductivity, 63, 75, 112, 128
- ksat, 55, 63, 106, 112
- Leong and Rahardjo, 55, 75, 106, 128
- Mesh, 3, 5, 26, 52, 53, 97, 98, 160
- No BC, 118, 119, 131, 132
- Output, 3, 4, 5, 6, 24, 25, 51, 52, 68, 69, 81, 82, 121, 122, 136, 137, 152, 153
- Output Files, 3, 4, 5, 6, 24, 51, 68, 81, 121, 136, 152
- Plots, 3, 4, 5, 6, 23, 49, 67, 79, 80, 96, 120, 133, 135, 151, 152
- Problem, 3, 4, 5, 6, 7, 10, 11, 12, 24, 30, 33, 34, 35, 55, 58, 59, 60, 69, 71, 72, 82, 88, 89, 92, 94, 97, 104, 105, 108, 109, 122, 124, 125, 137, 143, 144, 147, 149, 153, 161
- ProblemID, 7, 11, 12, 30, 34, 35, 56, 59, 60, 71, 89, 106, 109, 124, 144
- Problems, 8, 9, 10, 31, 32, 33, 56, 57, 58, 60, 71, 86, 87, 88, 89, 106, 107, 108, 109, 124, 141, 142, 143, 144
- Project, 3, 4, 5, 6, 8, 9, 31, 56, 86, 106, 107, 141
- ProjectID, 7, 8, 9, 30, 31, 32, 56, 57, 86, 87, 106, 107, 141, 142
- Projects, 8, 9, 10, 31, 32, 33, 56, 57, 58, 60, 71, 86, 87, 88, 89, 106, 107, 108, 109, 124, 141, 142, 143, 144
- Properties, 3, 4, 5, 6, 8, 9, 15, 19, 20, 23, 24, 31, 37, 45, 46, 50, 52, 55, 56, 62, 64, 65, 67, 68, 73, 76, 79, 80, 81, 86, 90, 92, 93, 106, 107, 111, 115, 116, 120, 121, 126, 129, 133, 134, 135, 136, 141, 145, 147, 148, 152
- PWP, 13, 69, 80, 90, 91, 120, 122, 134, 145, 146
- Region, 3, 4, 5, 6, 7, 18, 19, 20, 30, 44, 45, 46, 47, 49, 63, 64, 65, 76, 94, 115, 116, 117, 129, 130, 149, 160
- Regions, 3, 5, 17, 18, 43, 44, 63, 76, 77, 94, 114
- Run Log, 69, 82, 97, 122, 137, 153, 154
- Seepage, 4, 5, 55, 56, 71, 106, 124
- Selector, 18, 40, 41, 42, 44, 46, 47, 64, 112, 113, 114, 115, 116, 117, 129, 130, 149
- Shape, 18, 19, 44, 45, 64, 115
- Shapes, 3, 4, 5, 18, 44, 64, 115
- Soil, 3, 4, 5, 6, 8, 15, 18, 31, 37, 46, 47, 55, 62, 73, 76, 77, 90, 92, 93, 94, 106, 111, 116, 126, 129, 145, 148, 149, 150, 162
- Soils, 15, 37, 46, 47, 62, 73, 92, 93, 111, 116, 117, 126, 129, 130, 147, 148, 149, 162
- Solution, 3, 5, 26, 52, 56, 98, 106, 154
- Steady-State, 58, 60, 76, 108, 109, 129
- Strain, 90, 145
- Stress, 1, 3, 4, 5, 6, 24, 27, 50, 53, 55, 86, 90, 94, 96, 99, 106, 141, 145, 149, 154
- Suction, 5, 6, 55, 94, 96, 100, 106, 149, 155
- Surface, 40, 41, 42, 47, 48, 49, 80, 81, 112, 113, 114, 118, 119, 131, 132, 134, 150
- Surfaces, 3, 5, 39, 41, 42, 112, 114
- SVFLUX, 2, 4, 5, 6, 55, 56, 57, 58, 59, 60, 62, 63, 65, 67, 68, 69, 71, 72, 73, 76, 77, 79, 81, 82, 86, 90, 91, 94, 96, 105, 106, 107, 108, 109, 111, 114, 117, 120, 121, 122, 124, 125, 126, 129, 130, 133, 136, 137, 141, 145, 146, 149, 152

SWCC, 55, 74, 106, 127

Time, 73, 80, 125, 134

Transient, 4, 5, 6, 56, 71, 72, 74, 77, 79, 81,
82, 86, 91, 94, 96, 106, 124, 125, 127,
130, 133, 136, 137, 141, 146, 149, 152

View, 13, 14, 35, 36, 41, 60, 97, 110, 154

Water Table, 3, 7, 13, 18, 22

Workspace, 3, 4, 5, 13, 35, 60, 109

Zero Flux, 65, 66, 77, 78, 119, 132