



SVFLUX[™]

2D / 3D Seepage Modeling Software

Tutorial Manual

ED-5C

Date: May 27, 2005

Written by:

Jason Stianson, B.Sc.C.E.

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 SVFLUX 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

SVFLUX™ is a trademark of SoilVision Systems Ltd.

FlexPDE® is a registered trademark of PDE Solutions Inc.

AcuMesh™ is a trademark of SoilVision Systems Ltd.

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

1	A TWO DIMENSIONAL EXAMPLE PROBLEM	5
1.1	ADDING A PROJECT	5
1.2	ADDING A PROBLEM.....	7
1.3	OPENING THE PROBLEM.....	9
1.4	DEFINING THE PROBLEM.....	9
1.4.1	Specify Settings	9
1.4.2	Setting the Workspace	10
1.4.3	Define Material Properties	12
1.4.4	Adding Regions.....	14
1.4.5	Defining Region Geometry Shapes	15
1.4.6	Specify Boundary Conditions	18
1.5	SPECIFYING A FLUX SECTION.....	20
1.6	SPECIFY PLOTS.....	21
1.7	SPECIFY OUTPUT FILES	22
1.8	ANALYZE	24
1.9	RESULTS	24
1.9.1	Solution Mesh	24
1.9.2	Pressure Contours	25
1.9.3	Head Contours.....	26
1.9.4	Flow Vectors	27
1.9.5	Flux Report.....	27
2	2D STOCHASTIC EXAMPLE.....	28
2.1	CREATE A NEW PROBLEM	28
2.2	SET STAGES	29
2.3	MODIFY SOIL PROPERTIES	29
2.4	OUTPUT.....	30
2.5	PROBLEM SOLUTION	30
3	A THREE DIMENSIONAL EXAMPLE PROBLEM	32
3.1	ADDING A PROJECT	33
3.2	ADDING A PROBLEM.....	35
3.3	OPENING THE PROBLEM.....	37
3.4	DEFINING THE PROBLEM.....	37
3.4.1	Specify Settings	37
3.4.2	Setting the Workspace	38
3.4.3	Define Material Properties	40
3.4.4	Define 3D Surfaces	41
3.4.5	Adding Regions.....	44
3.4.6	Defining Region Geometry Shapes	44

3.4.7 Specifying a Soil by Region and Layer47

3.4.8 Specify Boundary Conditions48

3.5 ADDING FEATURES50

3.6 SPECIFY PLOTS50

3.7 SPECIFY OUTPUT FILES52

3.8 ANALYZE53

3.9 RESULTS53

 3.9.1 Solution Mesh54

 3.9.2 Pressure Contours54

 3.9.3 Head Contours55

 3.9.4 Flow Vectors56

4 REFERENCES57

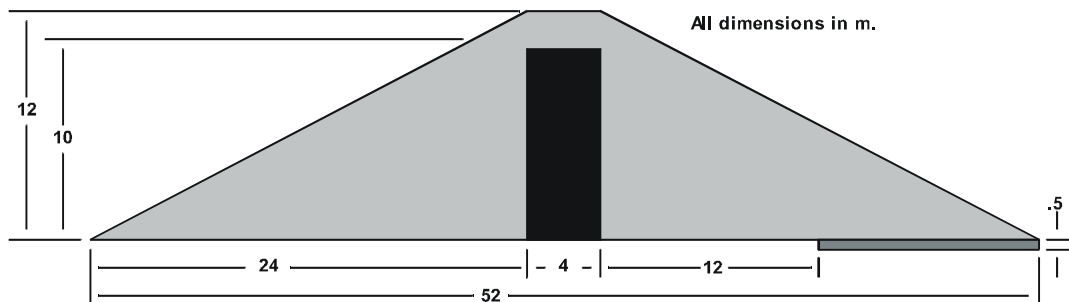
5 3D EXAMPLE PROBLEM DATA58

6 INDEX60

1 A TWO DIMENSIONAL EXAMPLE PROBLEM

The following example will introduce some of the features included in SVFLUX and will set up a model of a simple earth fill dam. The purpose of this problem is to determine the effects a clay core and filter will have on the final position of the phreatic surface and to determine the total flux that is passing through the dam. The problem dimensions and soil properties are provided below.

ProjectID: Tutorial ProblemID: Earth_Fill_Dam



Dimensions created in SVFLUX.

Silt:

Suction (kPa)	Hydraulic Conductivity (m/s)
37.9	1.13E-08
61.76	1.77E-09
82.8	3.72E-10
100.76	8.87E-11
130.84	2.18E-11
169.07	3.92E-12
200	1.00E-12

ks=1.0E-07

Clay:

Suction (kPa)	Hydraulic Conductivity (m/s)
9.8359	1.03E-09
81.0514	3.87E-12
98.5279	9.58E-13
149.2637	9.53E-13

ks=1.0E-09

Sand: ks=1E-04

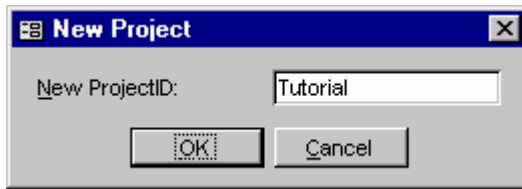
1.1 ADDING A PROJECT

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.

- The Project Properties form is opened along with a prompt asking for a new ProjectID.



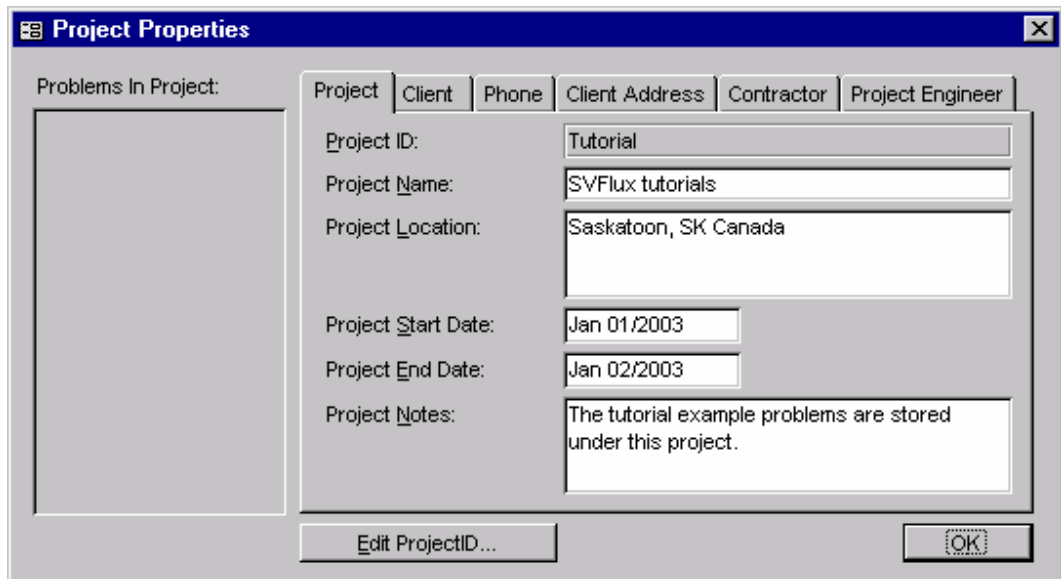
- Type "Tutorial" as the new ProjectID and press OK.

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.



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.

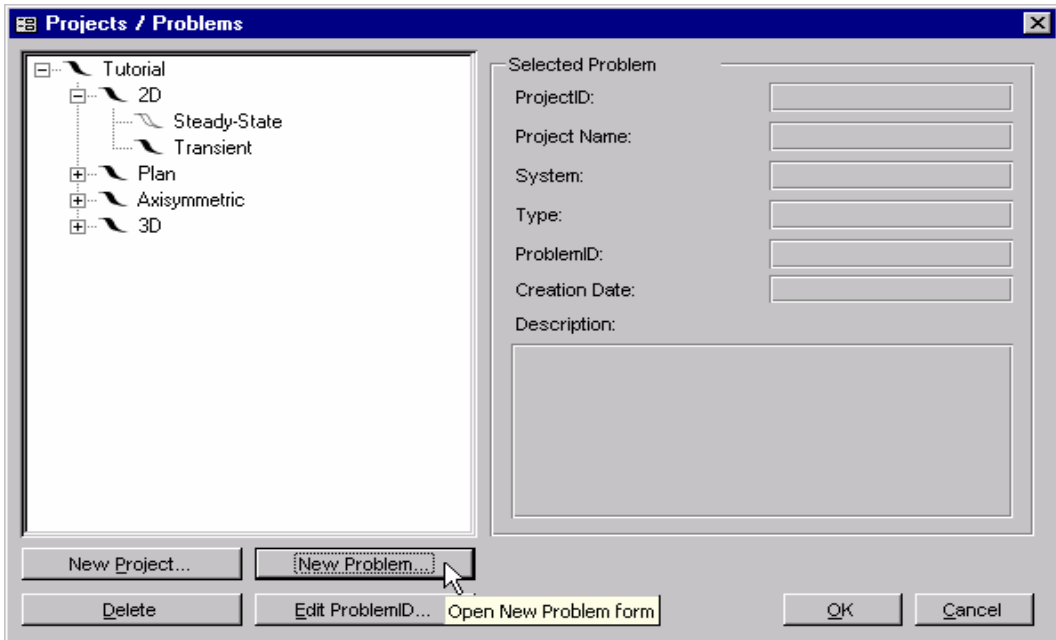


It should be noted that once the project is defined it would be identified by the ProjectID throughout the rest of the program. Also, SVFLUX does not allow you to specify two projects with the same ProjectID.

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

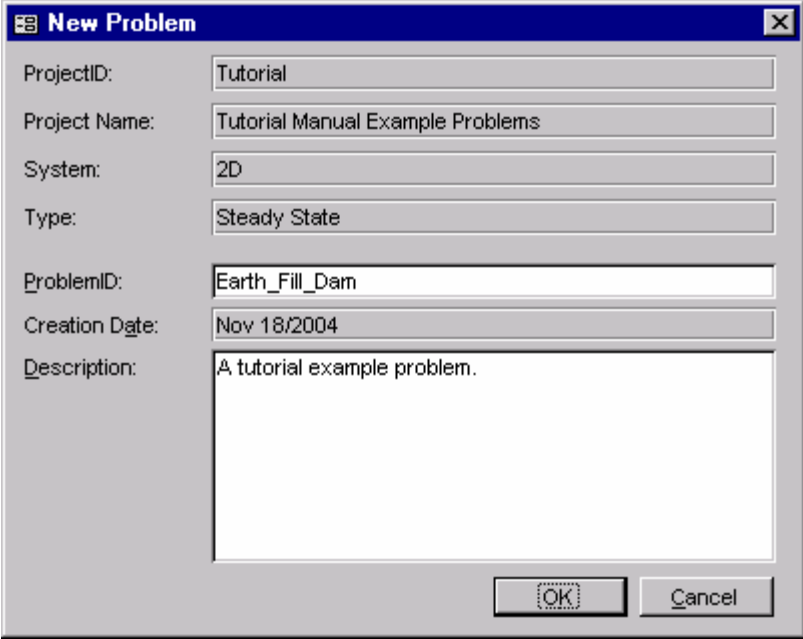
1.2 ADDING A PROBLEM

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 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: Tutorial Manual Example Problems

System: 2D

Type: Steady State

ProblemID: Earth_Fill_Dam

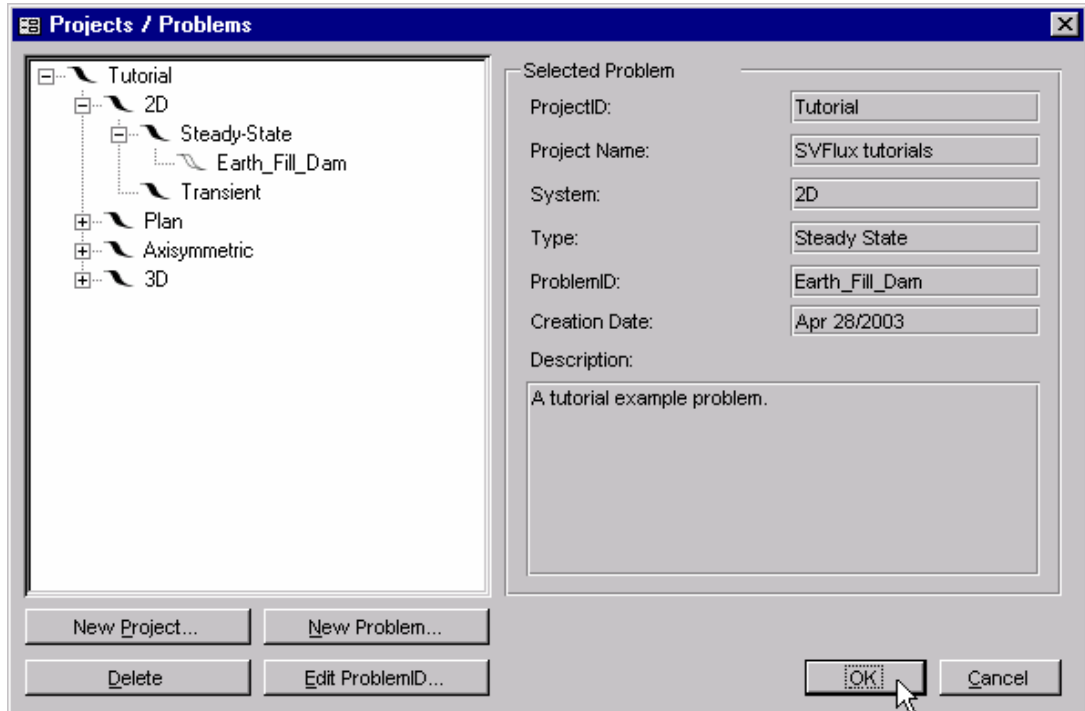
Creation Date: Nov 18/2004

Description: A tutorial example problem.

OK Cancel

5. Enter the **ProblemID: Earth_Fill_Dam**. 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.

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:

1. **Navigate** back to the problem via Tutorial, 2D, Steady-State.
2. Select **Earth_Fill_Dam**.
3. 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.

Settings

General | Time | Initial Conditions | Transient Formulation

System

2D

Plan

Axisymmetric

3D

Type

Steady-State

Transient

Units

Metric

Imperial

Time:

Length:

Force:

Pressure:


Conductivity:

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. The hydraulic conductivity data for the soils contained in the problem are reported as m/s so the time units will remain s, for seconds.

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

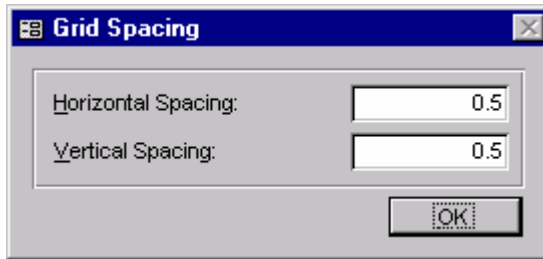
X: Y:

*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 filter portion of the problem has coordinates of a precision of 0.5m. In order to effectively draw geometry with this precision using the mouse the grid spacing must be set to a maximum of 0.5.

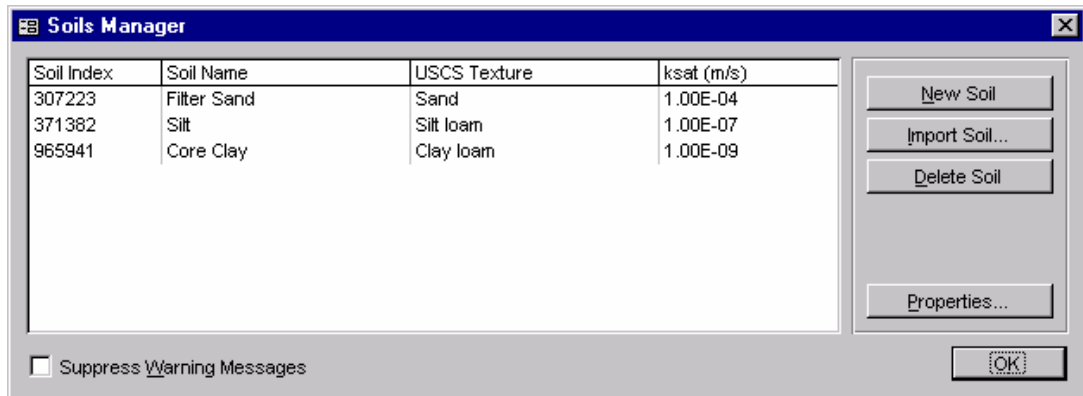
5. Select **View > Grid Spacing** from the menu.



The Grid Spacing dialog box has a title bar with a close button. It contains two input fields: 'Horizontal Spacing' with a value of 0.5 and 'Vertical Spacing' with a value of 0.5. An 'OK' button is located at the bottom right.

6. Enter **0.5** 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 the following data:

Soil Index	Soil Name	USCS Texture	ksat (m/s)
307223	Filter Sand	Sand	1.00E-04
371382	Silt	Silt loam	1.00E-07
965941	Core Clay	Clay loam	1.00E-09

On the right side, there are buttons for 'New Soil', 'Import Soil...', 'Delete Soil', and 'Properties...'. At the bottom left, there is a checkbox labeled 'Suppress Warning Messages' which is currently unchecked. An 'OK' button is at the bottom right.

The next step in defining the problem is to enter the material properties for the three soils that will be used in the model. A clay is defined for the core, the silt will make up the dam, and the filter will consist of a sand. This section will provide instructions on creating the clay soil. Repeat the process to add the other 2 soils.

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

Soil Index: 965941

Soil Name: Core Clay

USCS Texture: Clay loam

Soil Description:
 Geologic Description:
 Notes: This soil will make up the core in the 2D example problem.

OK

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

Saturated Hydraulic Conductivity

ksat: 1.000E-09 m/s

Unsaturated Hydraulic Conductivity

None
 Laboratory Data
 Modified Campbell
 van Genuchten and Mualem
 Leong and Rahardjo

Anisotropy

ky-ratio: 1 Alpha, α : 0

Laboratory Data

	Soil suction (kPa)	Hydraulic Cond. (m/s)
▶	9.8359	1.03E-09
	81.0514	3.87E-12
	98.5279	9.58E-13
	149.2637	9.53E-13
*		

Total Points: 4

Paste Points... Scale k-Curve...
 Graph...
 Difference: -33.566%

6. Refer to the data provided at the beginning of this tutorial. Enter the k_{sat} value of 1.000E-09 m/s
7. Select **Laboratory Data** from the Unsaturated Hydraulic Conductivity option group.
8. Enter the laboratory data **points** as provided.
9. Move to the **SWCC** tab and ensure the **None** option is selected.
10. **Repeat** these steps to create the silt and sand soils.




Tip!

To view the hydraulic conductivity curve of the laboratory data press the Graph button.

1.4.4 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 three regions, which are named Dam, Core, and Filter. 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.

#	Name	On	Lock	Soil	Shapes
1	Dam	☛	🔒	371382	1
2	Core	☛	🔒	965941	1
3	Filter	☛	🔒	307223	1

Display Boundary Condition Graphics

2. Change the first region name from Region 1 to Dam. Highlight the name and type new text.
3. Select the Soil Index from the drop-down corresponding to the silt.
4. Press the New button to add a second region.
5. Change the name of the second region to Core.
6. Select the clay soil for the Core region.
7. Click New to add the third soil.
8. Name it Filter and select the sand as the soil.
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.

Region Geometry

Dam

X	Y
0	0
40	0
52	0
28	12
24	12
20	10
0	0


Core

X	Y
24	0
24	10
28	10
28	0
24	0

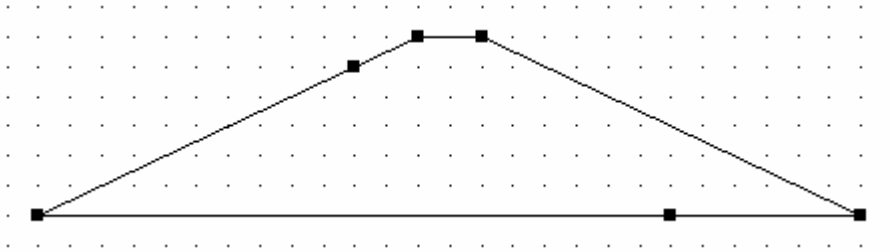
Filter


X	Y
40	0
40	-0.5
52	-0.5
52	0
40	0

- **Define the Dam**

1. Ensure the "Dam" 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 (52,0). 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 (52,0).
8. Move the cursor near the point (28,12). Right click to snap the cursor to the point. Left click on the point and a line is now drawn from (52,0) to (28,12).
9. Move the cursor near the point (24,12) and right click snapping the cursor to the point. Left click on the point to draw a line from (28,12) to (24,12).
10. Move the cursor near the point (20,10) and right click snapping the cursor to the point. Double click on the point to finish the shape. A line is now drawn from (24,12) to (20,10) and the shape is automatically finished by SVFLUX by drawing a line from (20,10) back to the start point, (0,0).


If the dam 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.

- **Define the Core**

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

1. Ensure that “Core” 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 24,0 and press the Enter key on the keyboard.
5. Type 28,0 and press Enter.
6. Type 28,10 and press Enter.
7. Type 40,0 and press Enter.
8. Type 24,10 and press Enter.
9. Type f and press Enter to complete the region shape.

- **Define the Filter**

Draw the filter using either the mouse or command line referring to the coordinate points in the above table.

**Tip!**

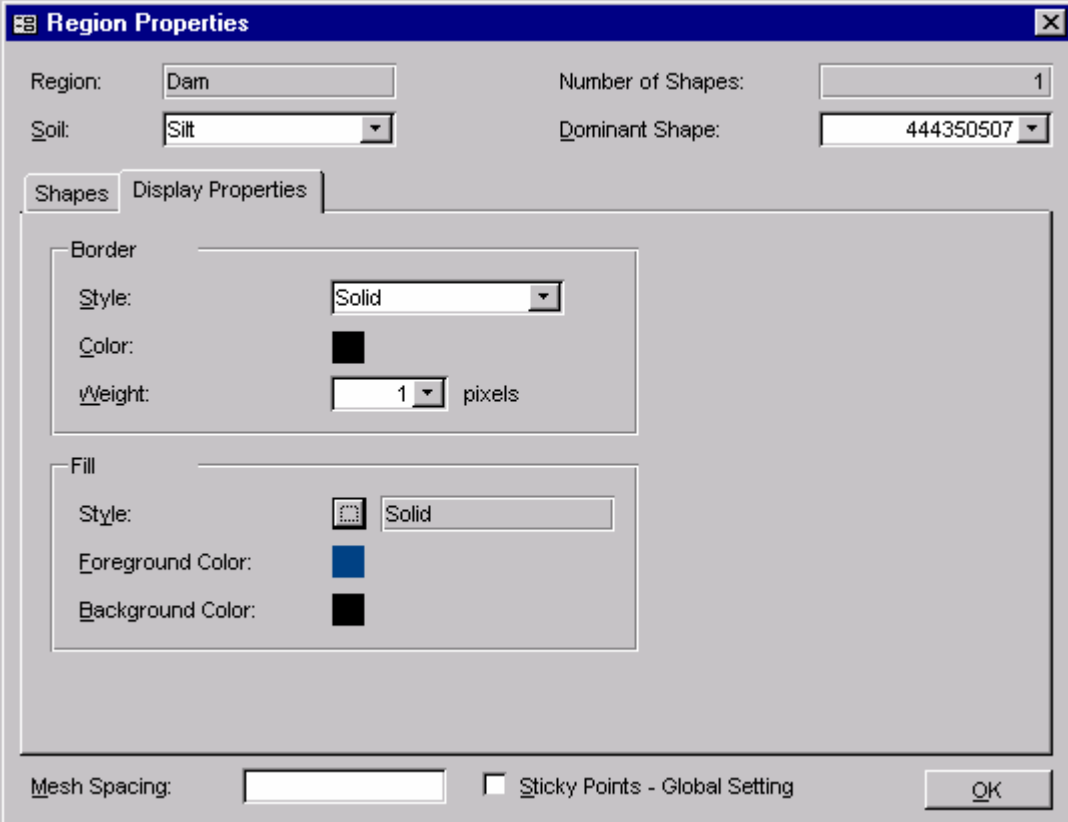
At times it may be tricky to snap to a grid point that is near a line defined for a region. Turn the object snap off by clicking on “OSNAP” in the status bar to alleviate this problem.

- **Formatting a Shape**

To set the Dam region shape to a solid blue:

1. Select the **Dam** shape in the drawing space.

2. Press the **Object Properties** button,  to open the Region Properties form:



Region Properties

Region: Number of Shapes:

Soil: Dominant Shape:

Shapes | **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 **blue** color and press **OK**.
7. Close the Region Properties form by pressing **OK**.

Repeat these steps to give the core a solid black fill and the filter a solid grey fill.

After all the region geometry has been entered it will as appear like this:



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 head of 10 m will be defined on the upstream face of the Dam region with the Zero Flux condition being applied to the remainder. The Core will be set to a Zero Flux condition by default and will not need to be altered. The Filter will have a head of -0.5m at its base. The steps for specifying the boundary conditions are thus:

- **Dam**
 1. Select the “Dam” 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: Dam
Select Shape Index: 444350507

X	Y	Boundary Condition	Expression or Data	Units
0	0	Zero Flux		
40	0	Continue		
52	0	Continue		
28	12	Continue		
24	12	Continue		
20	10	Head Expression	10	m

Update Selected Segment Segment Length: 22.361 m

1. Select Boundary Condition: Head Expression m

2. Provide: a) Expression: 10 Build Equation...

- or -

 b) Flux Data Index: Expr Reference...

 c) Evaporation Index: (optional)


3. Update OK

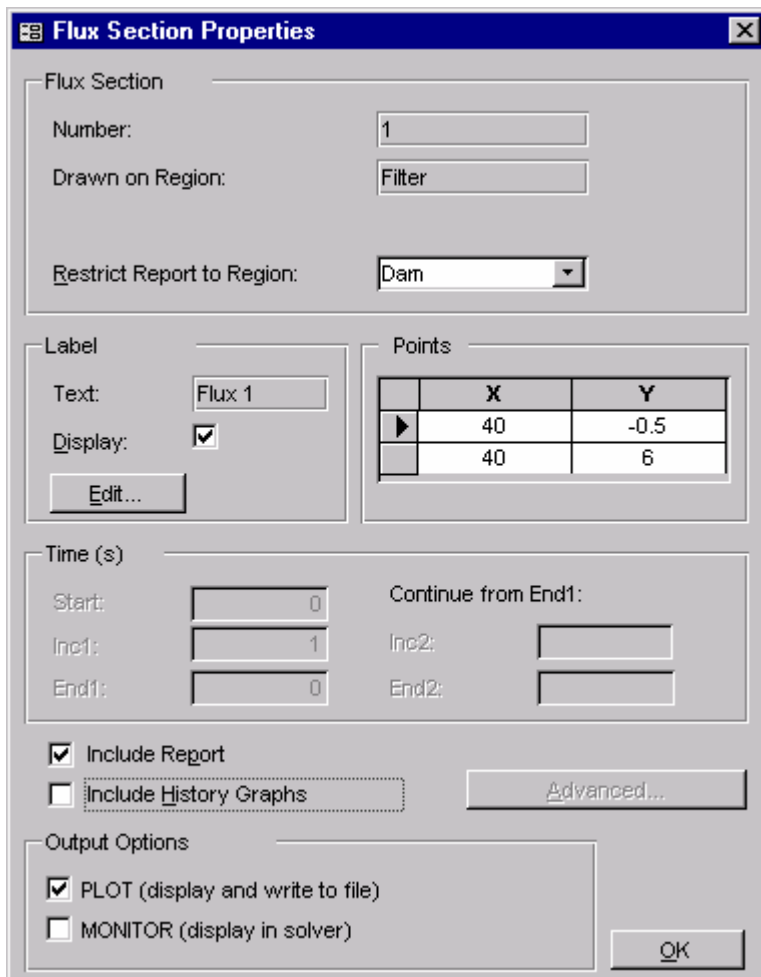
NOTE: boundary conditions defined at a point remain in effect until re-defined at a subsequent point

3. Select the **point** (20,10) from the list.

1.5 SPECIFYING A FLUX SECTION

Flux sections are used to report the rate of flow across a portion of the problem for a steady state analysis and the rate and volume of flow moving across a portion of the problem in a transient analysis.

1. Click on the **Flux Section** button, .
2. With the mouse click on the **point** (40,-05).
3. To finish the Flux Section click on the **point** (40,6). A blue line with an arrow on the end should be drawn across the dam.
4. Select **Model > Flux Sections** from the menu. The Flux Section List form will open.
5. Select **Flux 1** from the list.
6. Click **Properties** to open the Flux Section Properties form.



	X	Y
▶	40	-0.5
	40	6

- The flux section has been drawn such that it overlaps both the Dam and Filter regions. The flux section report is only desired for the Dam region in this tutorial. Select Dam from the **Restrict Report to Region** drop-down.
- Notice that the flux section label is partially on the region boundary in the workspace. To move the label location, first click the Edit button to open the Format Textbox form.

The screenshot shows the 'Format Textbox' dialog box with the following settings:

- Text: Flux 1
- WCS Location (X:Y): 41.000, 7.000
- Border: Automatic, Custom, None
- Style: Solid
- Color: Black
- Weight: 1 pixels
- Fill: Automatic, Custom, None
- Color: White

- Enter **41** and **7** in the WCS Location fields.
- Press **OK** to close the form.
- Close the Flux Section Properties form.
- Close the Flux Section List form.

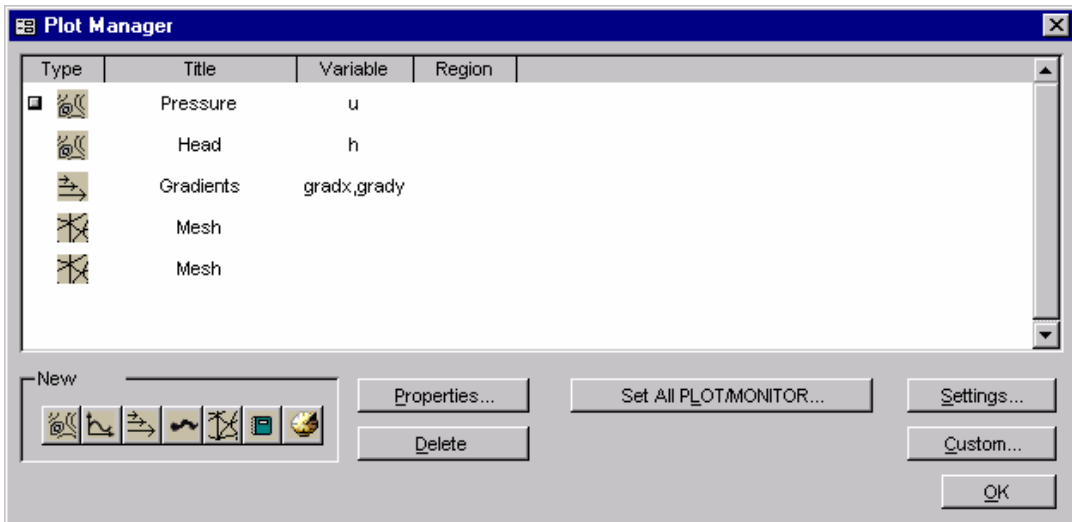
**Tip!**

Flux Section labels can be formatted in the same manner as regular textboxes.

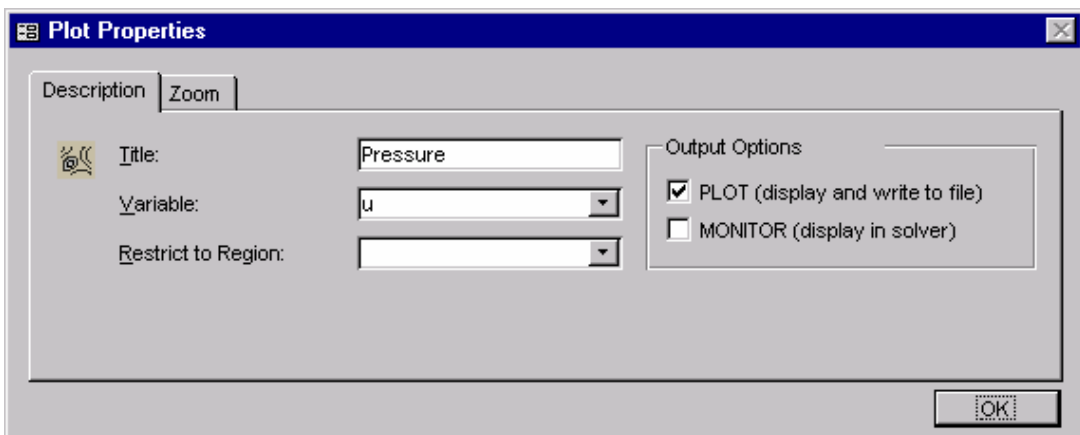
1.6 SPECIFY PLOTS

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, pressure contours, head contours, and gradient vectors.

- 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.

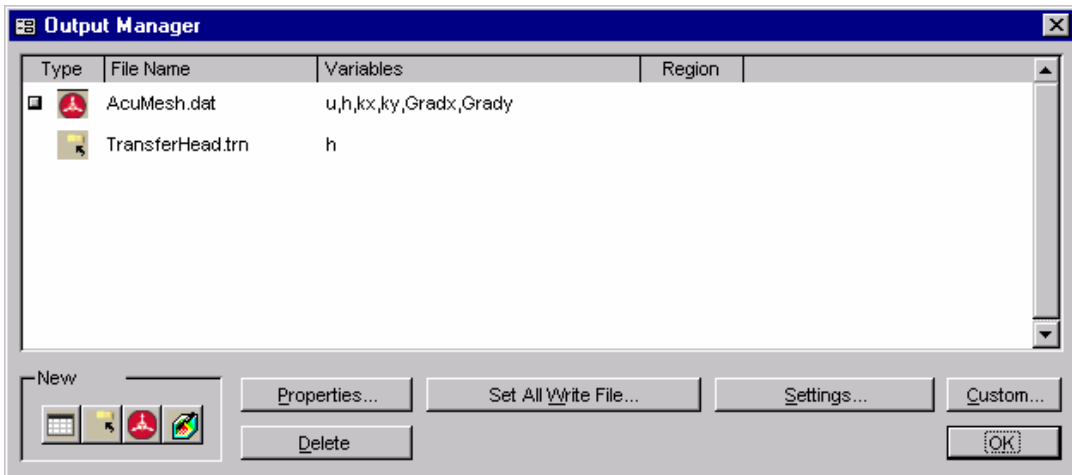


3. Enter the title Pressure.
4. Select u as the variable to plot from the drop-down.
5. Select the **PLOT** output option.
6. Click OK to close the form and add the plot to the list.
7. Repeat these steps 2 – 6 to create the plots shown above.
8. Click **OK** to close the Plot Manager and return to the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

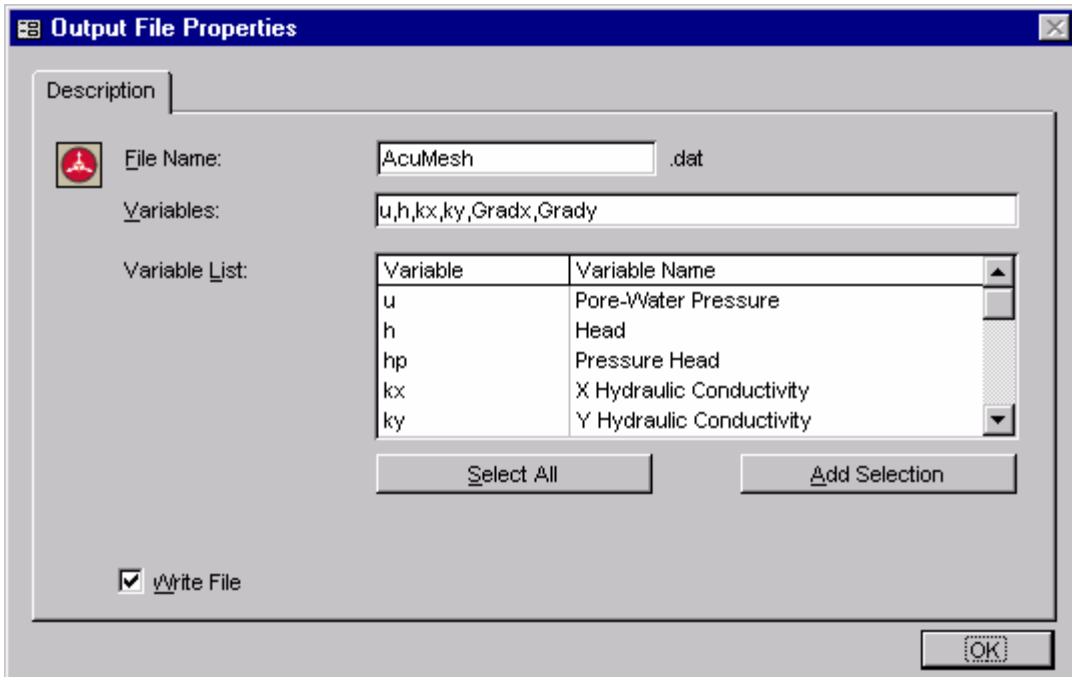
1.7 SPECIFY OUTPUT FILES

There are 4 output file types that can be specified to export the results of the model. Two will be generated for this tutorial example problem: a transfer file of heads, and a plot to transfer the results to AcuMesh. 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 AcuMesh button to begin adding the output file. The Output File Properties form will open.



3. Enter the title AcuMesh.
4. Select the variables u, h, kx, ky, gradx, and grady 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.

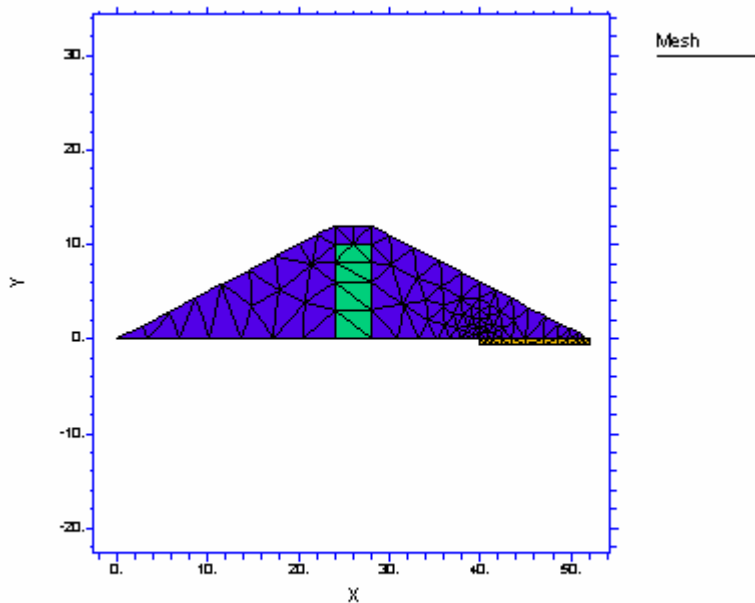
1.8 ANALYZE

The next step is to analyze the problem. Click the Analyze button located 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.

1.9 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 each plot that was generated.

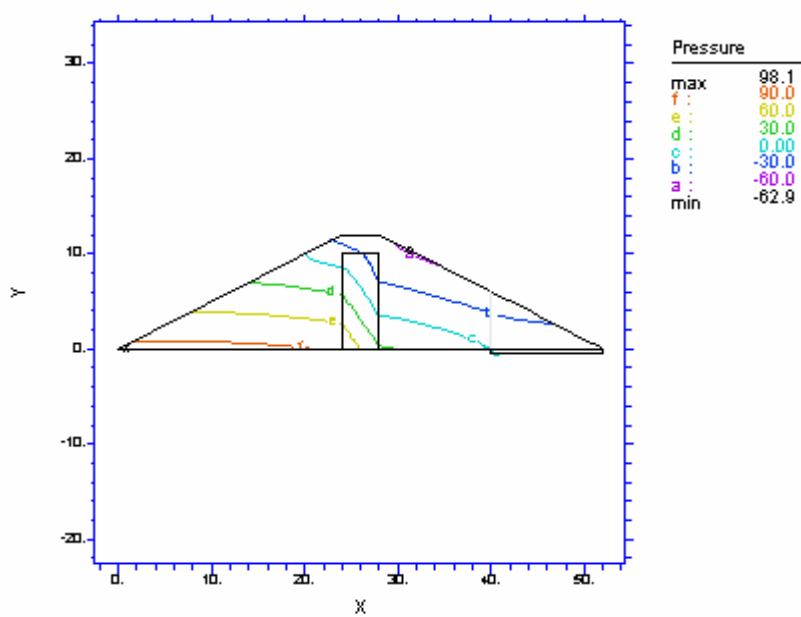
1.9.1 Solution Mesh



Examples_Earth_Fill_Dam: Grid#1 p2 Nodes=387 Cells=166 RMS Err= 9.9e-4
Stage 2

The Mesh plot displays the finite-element mesh generated by the solver. The mesh is automatically refined in critical areas such as the Dam – Filter contact where there is a significant change in hydraulic conductivity.

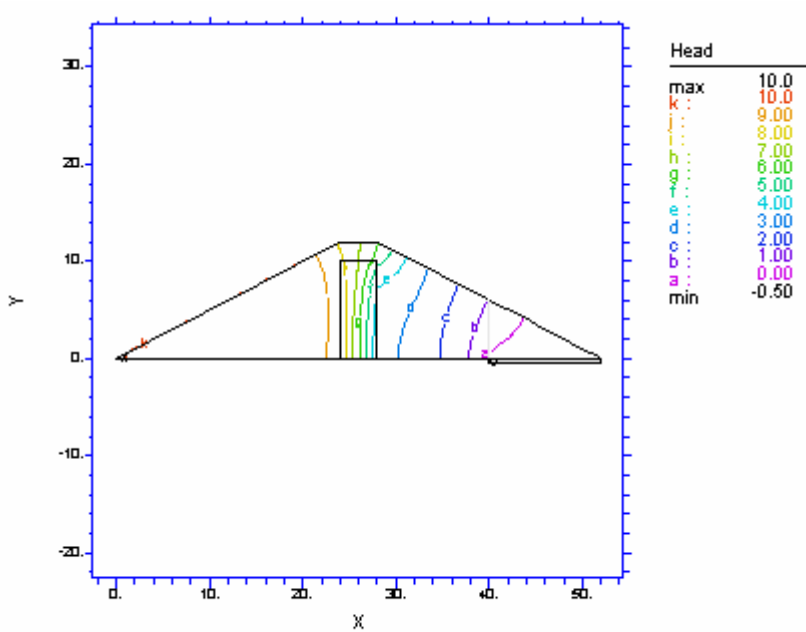
1.9.2 Pressure Contours



Examples_Earth_Fill_Dam: Grid#1 p2 Nodes=387 Cells=166 RMS Err= 8.9e-4
 Stage 2 Integral= 5129.242

The most important contour in the above plot is the one that corresponds to zero pressure. This contour represents the phreatic surface. All soil that lies below this line is saturated and all soil that lies above this line is considered to be unsaturated. The above design would be acceptable as the water table exits the dam at the beginning of the filter. If the water table had extended to the toe of the dam, there would be concern that the toe of the dam would become unstable due to piping failure.

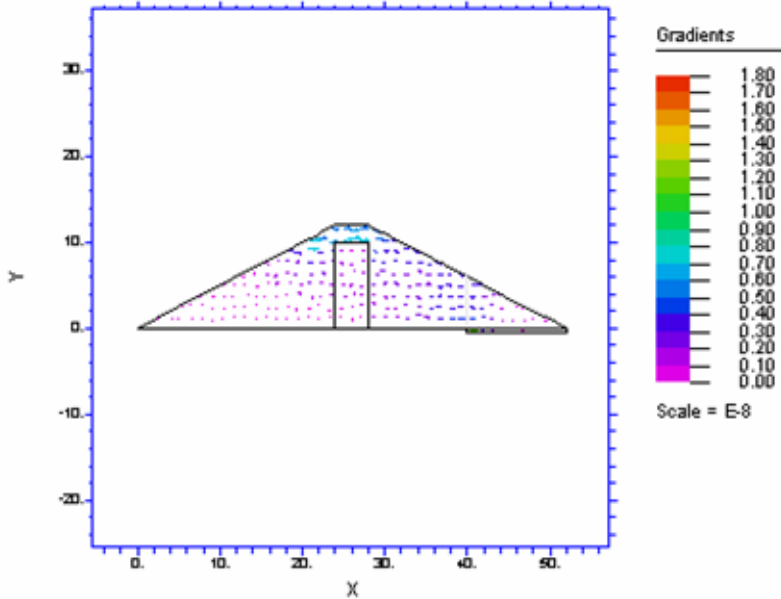
1.9.3 Head Contours



Examples_Earth_Fill_Dam: Grid#1 p2 Nodes=387 Cells=166 RMS Err= 8.9e-4
 Stage 2 Integral= 1961.518

As expected, most of the head is dissipated in the core of the dam. This is illustrated in by how close the contours are in the core. The maximum head in the problem occurs on the upstream face of the dam and is equal to ten. This is expected, as this was the boundary condition set on the upstream face of the dam. The lowest head occurs at the filter and is equal to -0.5m .

1.9.4 Flow Vectors



Examples_Earth_Fill_Dam: Grid#1 p2 Nodes=387 Cells=166 RMS Err= 0.0011 Stage 2

Flow Vectors show both the direction and the magnitude of the flow at specific points in the problem. The low conductivity of the core causes the majority of the flow to go up and over the core causing increased gradients in this area. The other area of interest is at the filter. Vectors illustrate that flow is exiting the dam in this region.

1.9.5 Flux Report

Flux Section Report: Flux 1 and Dam

X Component of Flow in (m³/s)= 2.372461e-8
 Y Component of Flow in (m³/s)= 0.00
 Normal Flow in (m³/s)= 2.372461e-8

The Flux through the problem is displayed in the form of report showing a breakdown of the X, Y, and Normal components of flow through the problem.

2 2D STOCHASTIC EXAMPLE

A single run of a numerical model is often not sufficient. SVFLUX implements the ability to incorporate statistical uncertainty into the input of soil properties. This tutorial section guides the user through the setup and execution of a problem implementing the stochastic or statistical features available in SVFLUX.

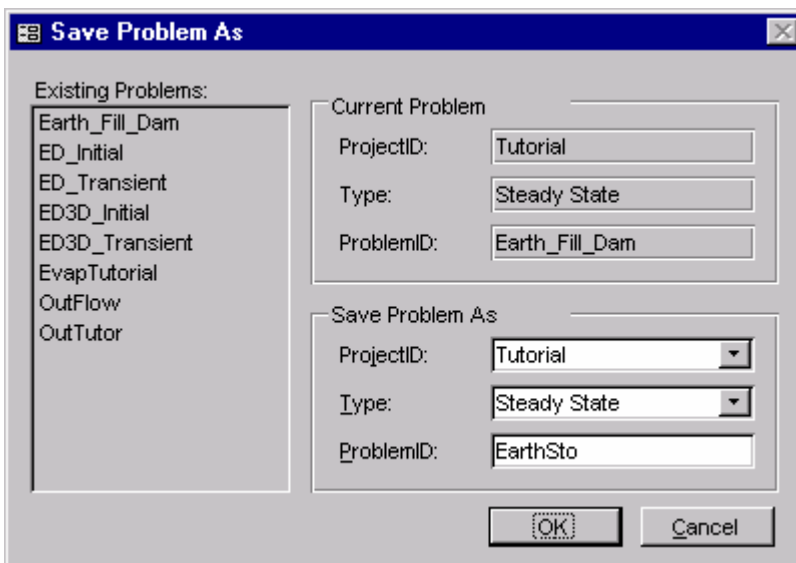
The earth dam example presented in Section 1 is extended to incorporate a variation in the slope of the unsaturated portion of the hydraulic conductivity curve. The current example makes use of the Modified Campbell method of estimating the unsaturated hydraulic conductivity. The slope of the unsaturated portion is controlled by a “p” parameter. Instead of specifying a single value for this parameter a Monte Carlo analysis will be used to generate 50 runs of the same problem varying the “p” parameter of the Modified Campbell estimation method.

Why do we want to do this? In this example we want to determine the influence of the slope of the unsaturated hydraulic conductivity curve on the difference in flow through the core or over the top of core. The unsaturated hydraulic conductivity curve is often not measured and it is useful to determine the sensitivity of flow through various portions of the dam on this curve.

The final problem is presented in the distribution database under the VerifyStage > 2D > Steady State > EarthFillDamSto problem.

2.1 CREATE A NEW PROBLEM

The first step in creating the stochastic problem is to save a copy of the example problem created in section 1. This is accomplished through the following steps:



1. Open the **Tutorial > Earth_Fill_Dam** problem,
2. Select **Model > Save Problem As...**

3. Type the name “**EarthSto**” and click OK.

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

2.2 SET STAGES

The second step is to describe the number of stages/runs that will be used in the analysis. Stages are controlled in SVFLUX by opening the **Model > FEM Options** form. The STAGES field should be set to **50** under the Stage Settings area of the form. Click Yes for the warning message and then close the FEM Options form.

2.3 MODIFY SOIL PROPERTIES

The next step that must occur is that the Modified Campbell “p” parameter will be staged. In this example the parameter will be set to have a mean value of 5 and a standard deviation of 2. This staging is accomplished through the following process:

1. Select **Model > Soils Manager...** to open the list of current soils,
2. Select the “Silt” and click the “Properties...” button,
3. Click the “**Stage Parameters...**” button located in the lower left of the form,
4. Select the “**Mcampbell p**” parameter and click the “Include” button,
5. Select the “**Mcampbell p**” parameter again,
6. Click the “Stage Values...” button,
7. Click the “Stage Distribution...” button,
8. Select “**Monte Carlo Normal**” as the distribution method,
9. Enter a mean of 5 and a standard deviation of 2 and click the **Add** button,

The image shows a software dialog box titled "Stage Distribution". On the left, under "Distribution Method", the "Monte Carlo Normal" option is selected with a radio button. Other options include "3 Stage Normal", "5 Stage Normal", "Monte Carlo Log Normal", "Triangular", "Uniform", "Exponential", and "Poisson". On the right, under "Settings", there are input fields for "Problem Stages" (set to 50), "Mean" (set to 5), "Standard Deviation" (set to 2), "Minimum", "Mode", and "Maximum". At the bottom right, there are "Add" and "Cancel" buttons.

10. A list of 50 generated values should now be displayed in the list box.
11. Press OK on all remaining forms to close them.

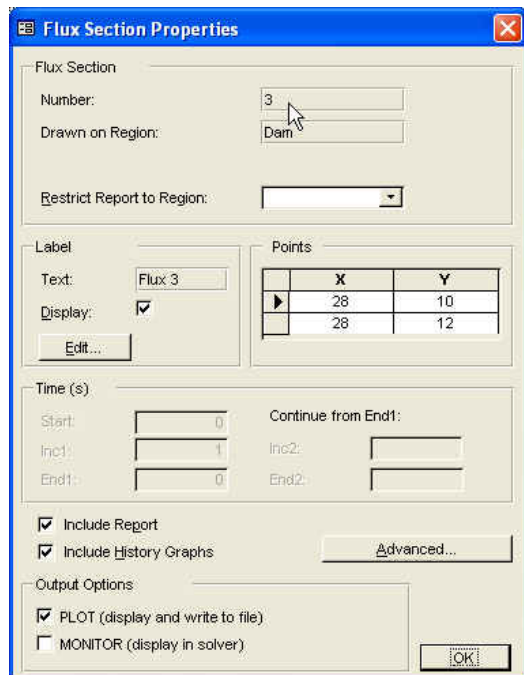
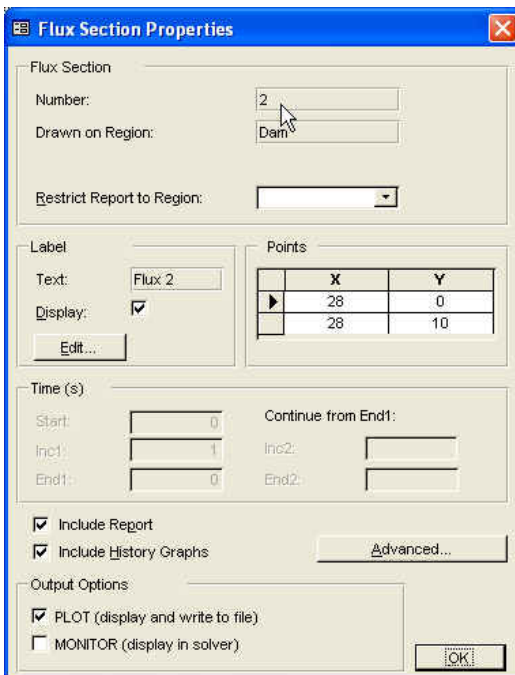
The parameter has now been staged.

2.4 OUTPUT

Since we are trying to determine the amount of flow through different portions of the dam we will place flux sections at the right hand side of the dam core in addition to the flux section already present near the filter.

The flux sections should have the following coordinates: Flux 2 (28,0) to (28,10), Flux 3 (28,10) to (28,12).

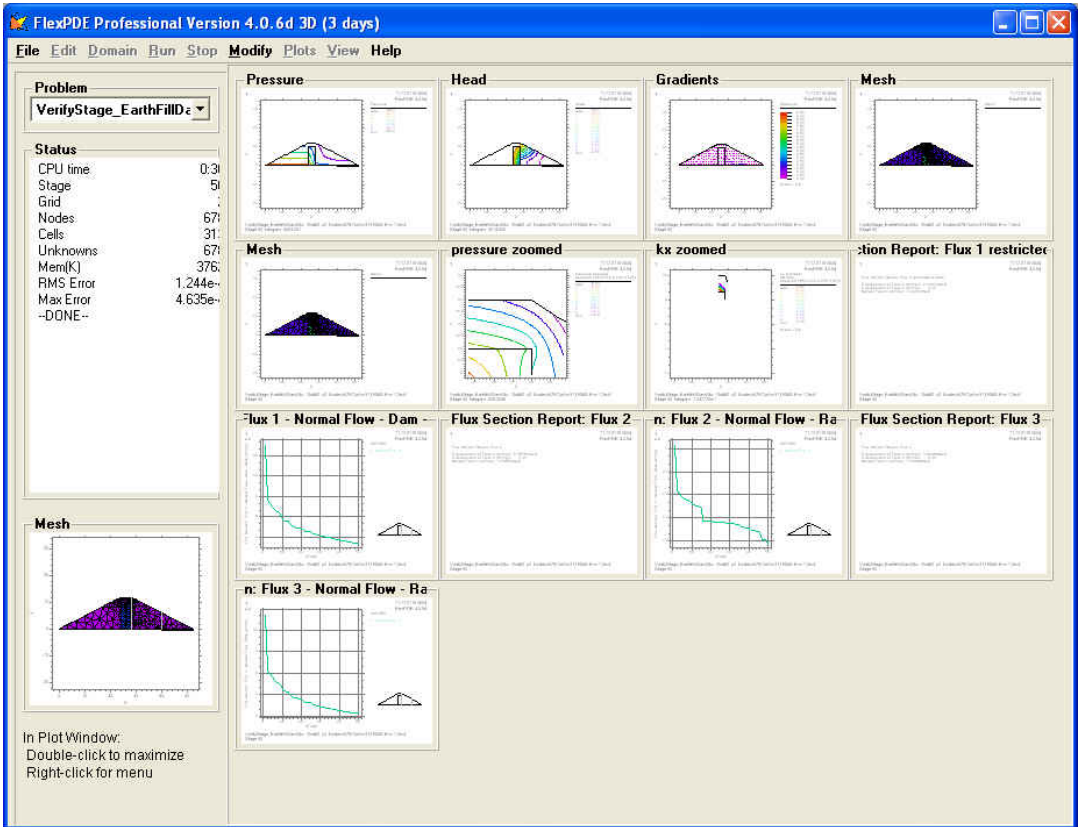
The flux sections may be drawn graphically or entered through the **Model > Flux Sections...** menu option. Refer to [Specifying a Flux Section](#) in this manual for instructions on adding Flux Sections.



2.5 PROBLEM SOLUTION

Clicking the Analyze button will initiate problem solution. Attention should be directed to the stage display in the finite element solver. Once a particular stage is complete the next stage will automatically be initiated. The various flows through differing portions of the earth dam may be seen in the output of the flux section history plots for flux sections 2 and 3.

From these history plots it can be seen that the model is quite sensitive to the slope of the unsaturated hydraulic conductivity curve.



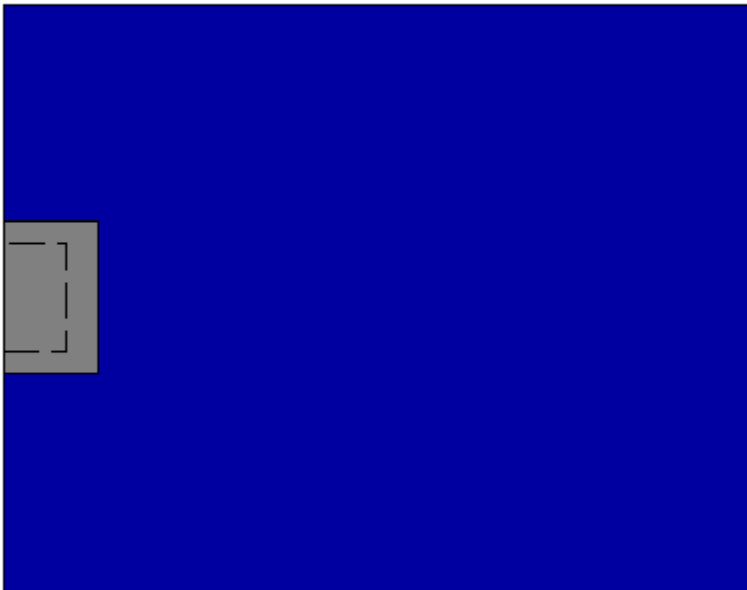
3 A THREE DIMENSIONAL EXAMPLE PROBLEM

The following example will introduce you to the three dimensional SVFLUX modeling environment. This example is used to investigate the flow and pressure conditions existing within a slope due to a holding pond at the crest. The example is modeled using 2 regions, 2 surfaces, and 1 soil. The problem data and soil properties are provided below.

ProjectID: Tutorial

ProblemID: Reservoir3D

Reservoir3D



Tutorial Soil - Reservoir3D:

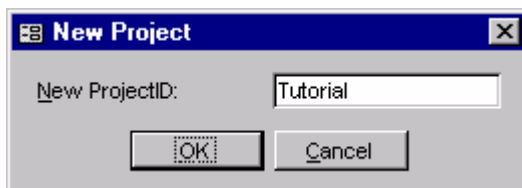
$k_{\text{sat}} = 2.512\text{E-}06 \text{ kPa}$

3.1 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. If a project named Tutorial already exists skip to section [3.2 Adding a Problem](#).
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 your 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: SVFlux Tutorials

Project Location: Saskatoon, SK, Canada

Project Start Date: May 20/2005

Project End Date: May 20/2005

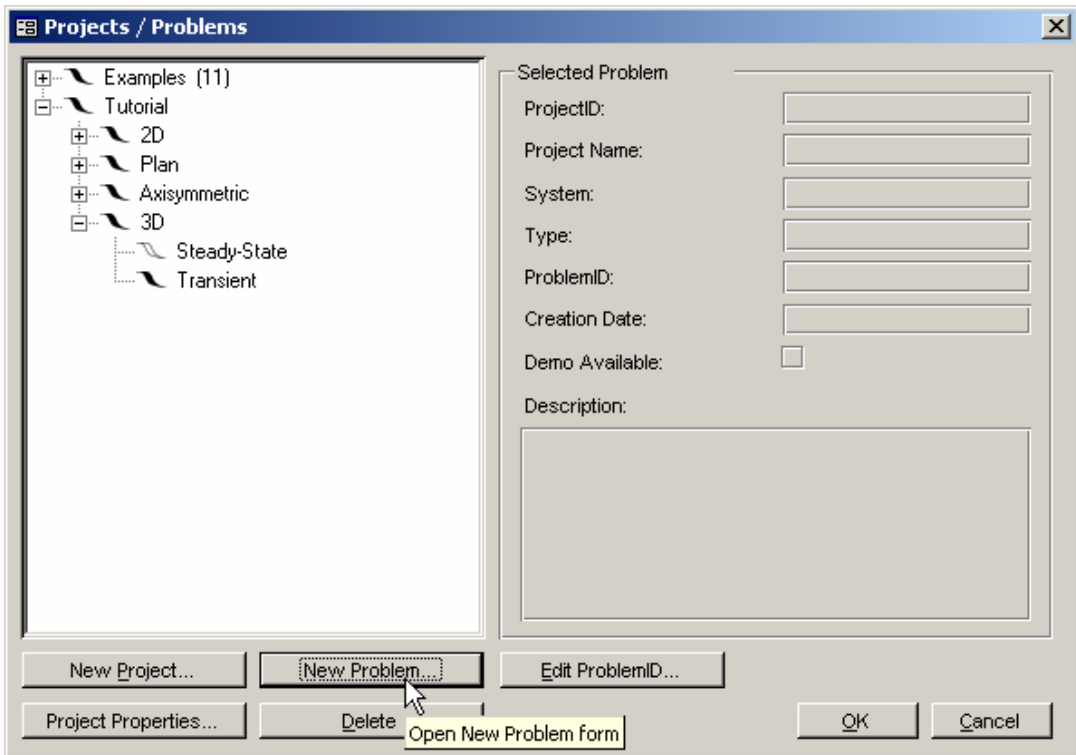
Project Notes: The tutorial example problems are stored under this project.

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, SVFLUX does not allow you to specify two projects with the same ProjectID.

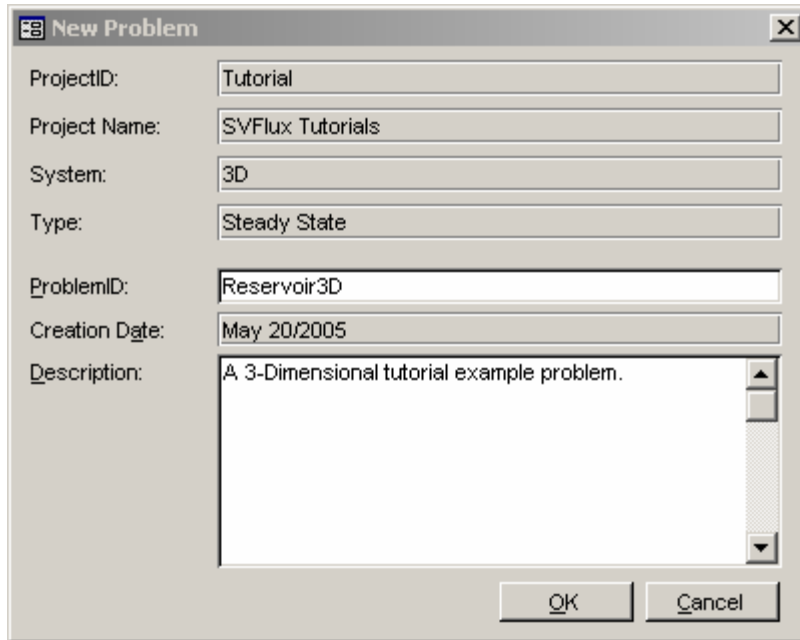
5. Fill out the form with the desired **information**.
6. To exit this form and return to Projects / Problems click **OK**. The project information is automatically saved upon entry.

3.2 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. To add a problem to the Tutorial project follow these steps:

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.



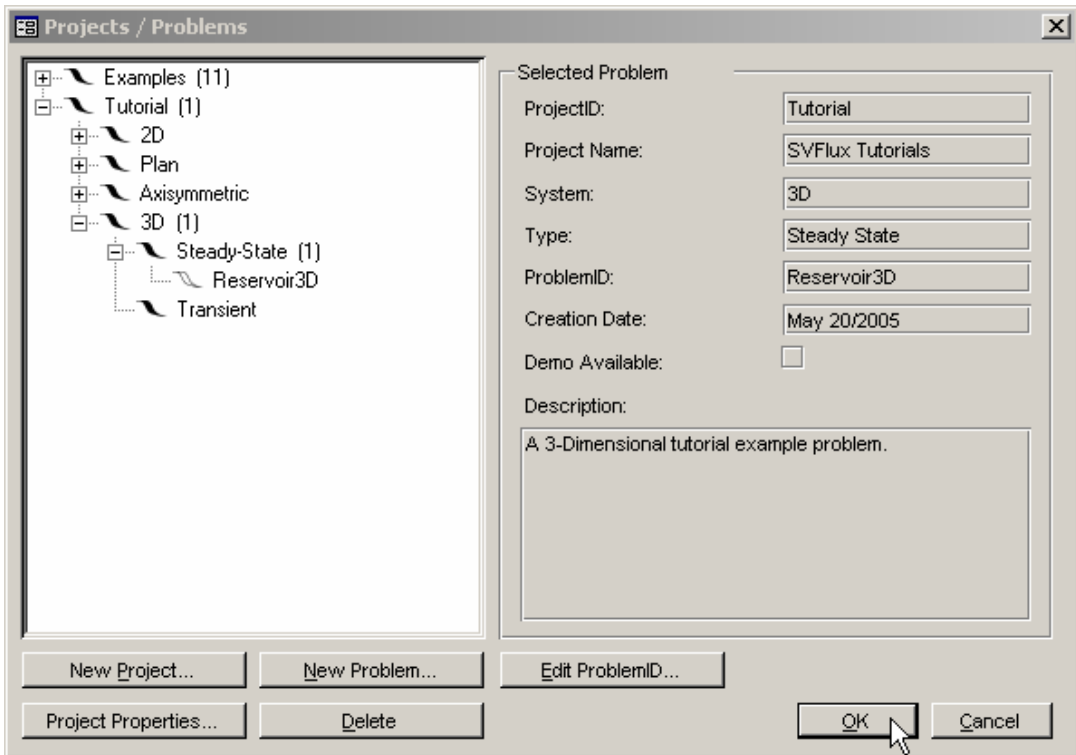
The image shows a 'New Problem' dialog box with the following fields and values:

- ProjectID: Tutorial
- Project Name: SVFlux Tutorials
- System: 3D
- Type: Steady State
- ProblemID: Reservoir3D
- Creation Date: May 20/2005
- Description: A 3-Dimensional tutorial example problem.

Buttons: OK, Cancel

5. Enter the **ProblemID: Reservoir3D**. 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.

3.3 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, Steady-State.
2. Select **Reservoir3D**.
3. The problem may be opened by clicking the **OK** button or by double clicking on the ProblemID.

3.4 DEFINING THE PROBLEM

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


3.4.1 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, Type, Units, and Transient Settings. The hydraulic conductivity data for the soils contained in the problem are reported as m/s so the time units will remain s, for seconds.

3.4.2 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 Settings...**. The button is located in the view toolbar to the left of the drawing space.

The screenshot shows the 'World Coordinate System' dialog box with the following settings:

- World Coordinate System:**
 - Bottom Left: X: -5.000, Y: -5.000
 - Upper Right: X: 30.000, Y: 30.000
- CAD Window:**
 - Bottom Left: X: -5.000, Y: -5.000
 - Upper Right: X: 30.000, Y: 30.000
- Region Geometry Extents:**
 - Minimum: X: 0.000, Y: 0.000
 - Maximum: X: 0.000, Y: 0.000
- Surface Grid Extents:**
 - Minimum: X: 0.000, Y: 0.000
 - Maximum: X: 10.000, Y: 10.000

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.

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.

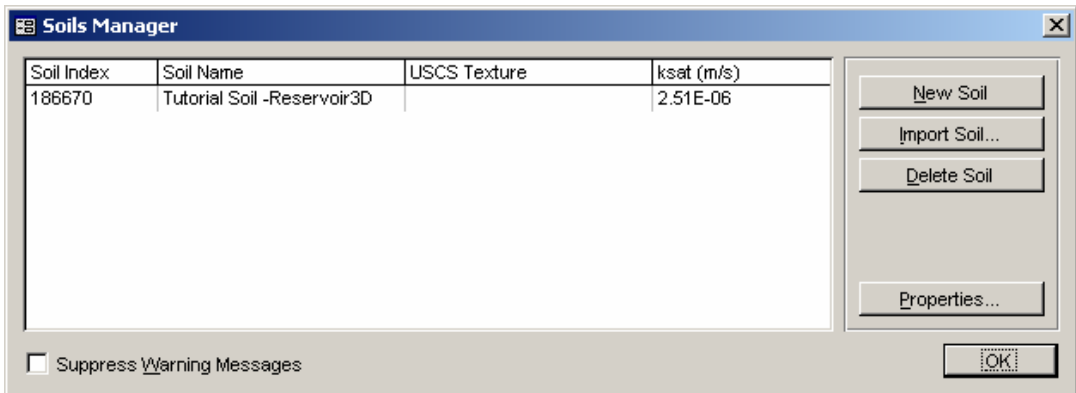
5. Select **View > Grid Spacing** from the menu.

The screenshot shows the 'Grid Spacing' dialog box with the following settings:


- Horizontal Spacing: 1
- Vertical Spacing: 1

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

3.4.3 Define Material Properties



The next step in defining the problem is to enter the material property for the soil that will be used in the model. It will be defined for both the slope and reservoir regions.

1. Open the Soils Manager form by selecting **Model > Soils Manager...** from the menu or click the soils button,  in the Tool 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 and the Soil Properties form will open automatically.
3. Double-clicking on a soil in the list will also open the **Soil Properties** form.

Soil Properties

Description Hydraulic Conductivity SWCC Volume-Mass Parameters Sink/Source

Soil Index: 186670

Soil Name: Tutorial Soil -Reservoir3D

USCS Texture: [dropdown]

Soil Description: [text box]

Geologic Description: [text box]

Notes: This is the soil for the entire region.

Stage Parameters... OK


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 $2.512E-06$ m/s.
7. Select **None** from the Unsaturated Hydraulic Conductivity option group.
8. Press **OK** on the Soils Manager form to close both forms.

3.4.4 Define 3D Surfaces

This problem consists of two surfaces and each will be defined by a different method. By default every problem initially has 2 surfaces.


- **Define Surface 1**

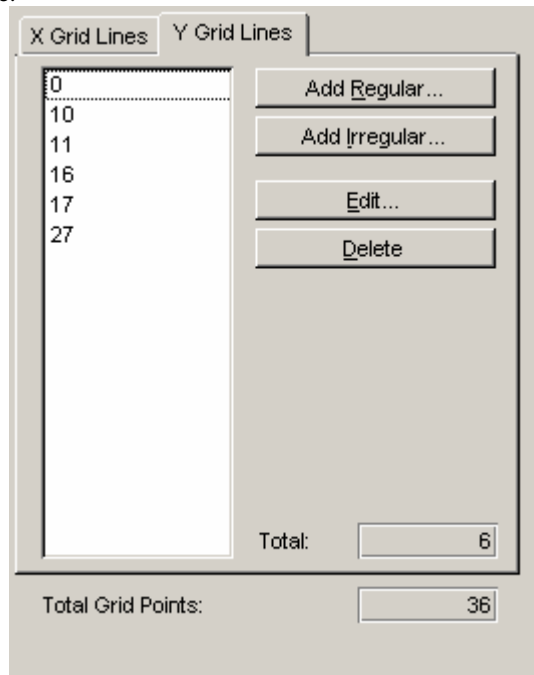
This surface will be defined by providing a constant elevation.

1. Select Surface 1 in the **Surface Selector** located at the top of the workspace.
2. Click the **Surface Properties** button,  adjacent to the **Surface Selector**.
3. For the Surface Definition option, select Expression (Relative)
4. Open the **2d: Expressions** tab
5. Enter a Surface Expression of 0.
6. Click **OK** to close the form.

- **Define Surface 2**

This surface will be defined by providing a grid of (X,Y) points and corresponding elevations.

1. Select Surface 2 in the **Surface Selector**.
2. Click the **Surface Grid Setup** button, .
3. For Surface Definition, select **Elevation Data**.
4. Open the **2a: Grid** tab. There will be default grid lines of 0 and 10 present. Click the **Add Irregular** button to open the Add Irregular X Gridlines form.
5. Enter 0, 2, 3, 14, 21, and 24.
6. Click **Add** to add the gridlines and close the form.
7. Select 10 from the list.
8. Press **Delete**.



Grid Line
0
10
11
16
17
27

Total: 6

Total Grid Points: 36

9. Move to the **Y Grid Lines** tab and click the **Add Irregular** button to open the Add Irregular Y Gridlines form.
10. Enter 0, 10, 11, 16, 17, and 27.
11. Click **Add** to add the gridlines and close the form.



You can import XYZ data, paste surface data, or paste surface grid data in this menu for faster data entry.

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

1. Click the **2b: Elevations** tab to view the elevations for surface 2.

	X	Y	Z
	0	0	11
	0	10	11
	0	11	10
	0	16	10
	0	17	11
	0	27	11
	2	0	11
	2	10	11
	2	11	10
	2	16	10
	2	17	11
	2	27	11
	3	0	11
	3	10	11
	3	11	11
	3	16	11
	3	17	11


Min: 0.000 0.000 1.00

Max: 24.000 24.000 11.00

Expression Reference... OK

2. Enter the **elevations** as provided in [3D Example Problem Data](#).
3. Press **OK** to close the form.

The grid for surface 2 can be made viewable in the workspace.

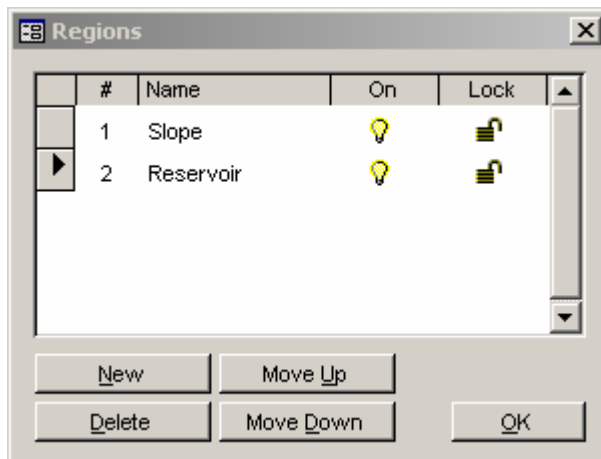
1. Click the **Open Surfaces Form** button, .
2. Click the Grid check box for surface 2. A check mark should appear in the box.
3. Click the Grid check box for Surface 1. The check mark should disappear.
4. Press **OK** to close the form.

3.4.5 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 can be defined on any given region.

This problem will be divided into 2 regions, which are named Slope and Reservoir. 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 R1 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 **Reservoir**.
5. Click **OK** to close the form.

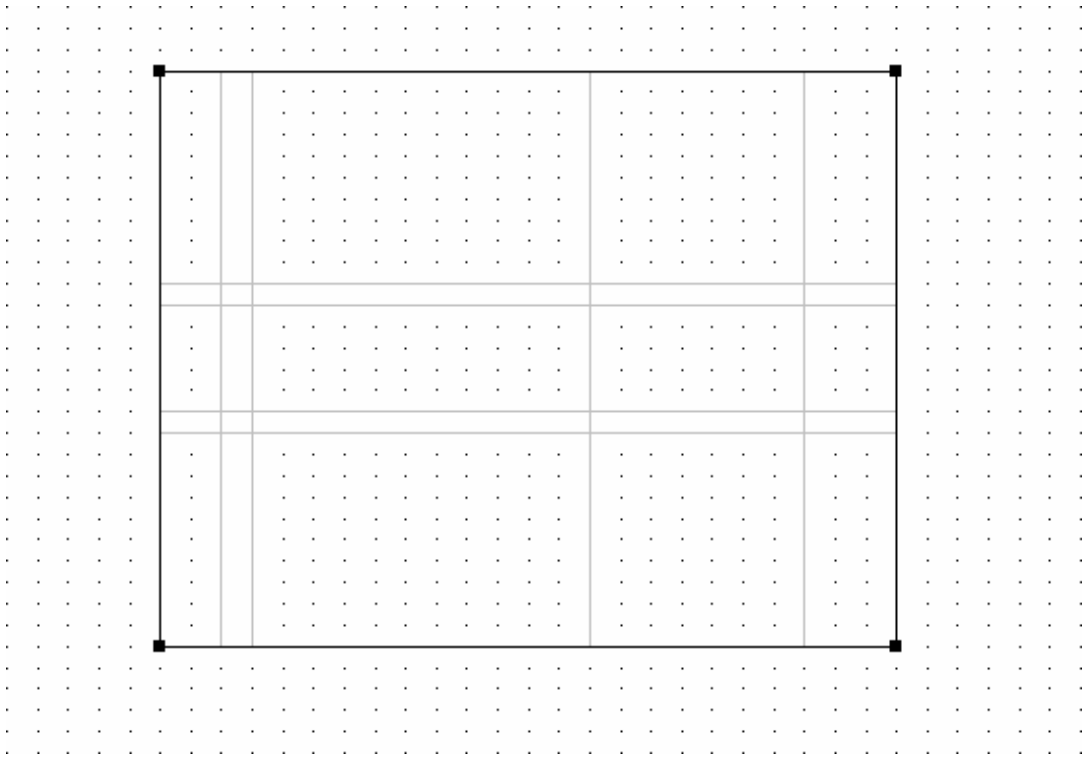
3.4.6 Defining Region Geometry Shapes

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 [3D Example Problem Data](#) section 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 command line will be set to Start Point and the cursor focus will be in the command line.
4. Type 0,0 and press the Enter key on the keyboard.
5. Enter the remaining coordinates (14,0), (24,0), (24,27), (14,27) and (0,27).
6. Type f and press Enter to complete the region shape.

If the Main geometry has 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.

- **Define the Reservoir**

In the instructions for defining the Slope shape the command line was used. To draw the reservoir the instructions below explain the use of the drawing tool to create the Reservoir shape.

1. Ensure that "Reservoir" is current in the region selector.
2. Click on the Draw Polygon Region Shape button, .
3. Move the cursor near (0,10) 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.
4. When the cursor is near the point, click the left mouse button. The cursor should automatically snap to the point (0,10) as long as the cursor is close to the point and the SNAP and GRID options in the status bar are both bold.


5. Now move the cursor near (3,10). Left click on the point. A line is now drawn from (0,10) to (3,10).
6. Repeat this process for the remaining points. If a mistake is made simply press the delete key on the keyboard and start over.
7. For the last point (0,17), double click on the point to finish the shape. A line is now drawn from (3,17) to (0,17) and the shape is automatically finished by SVFLUX by drawing a line from (0,17) back to the start point, (0,0).

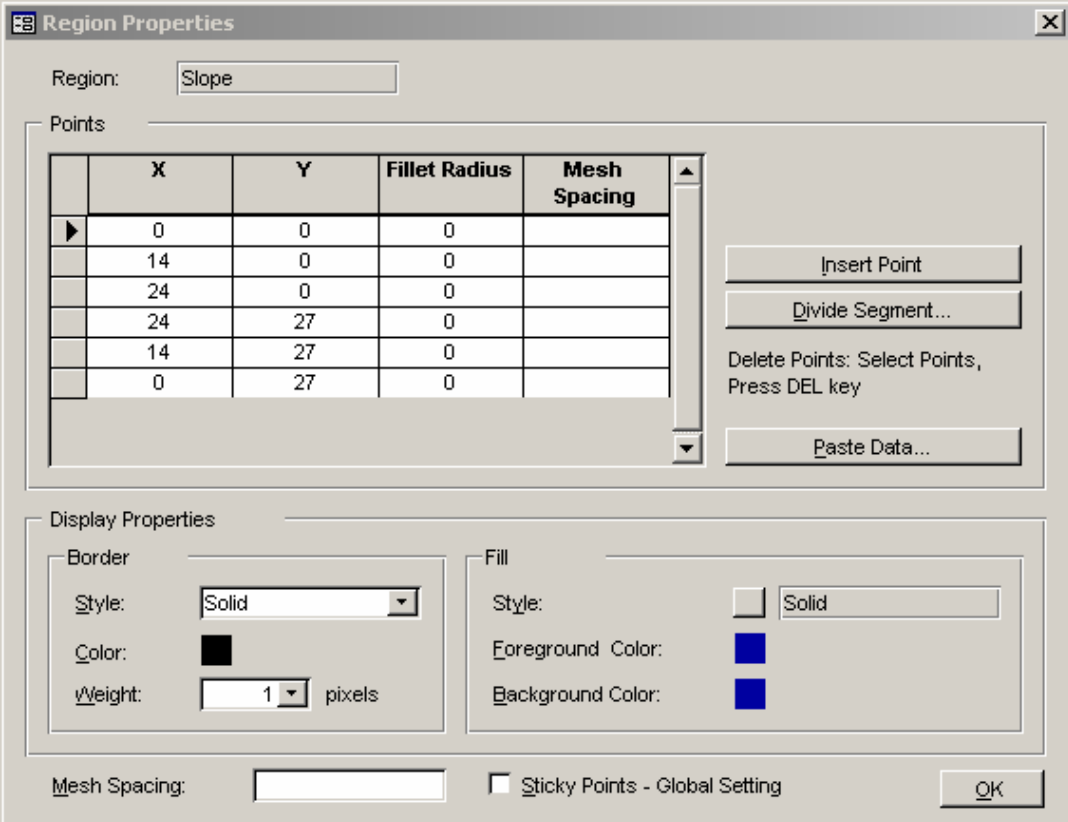


At times it may be tricky to snap to a grid point that is near a line defined for a region. Turn the object snap off by clicking on "OSNAP" in the status bar to alleviate this problem.

• Formatting a Shape

To set the Slope region shape to a solid blue:

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



Region:

Points

	X	Y	Fillet Radius	Mesh Spacing
▶	0	0	0	
	14	0	0	
	24	0	0	
	24	27	0	
	14	27	0	
	0	27	0	

Insert Point
Divide Segment...
Delete Points: Select Points, Press DEL key
Paste Data...

Display Properties

Border

Style:

Color:

Weight: pixels

Fill

Style: Solid

Foreground Color:

Background Color:

Mesh Spacing:

Sticky Points - Global Setting

OK

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 **blue** color and press **OK**.
7. Close the Region Properties form by pressing **OK**.

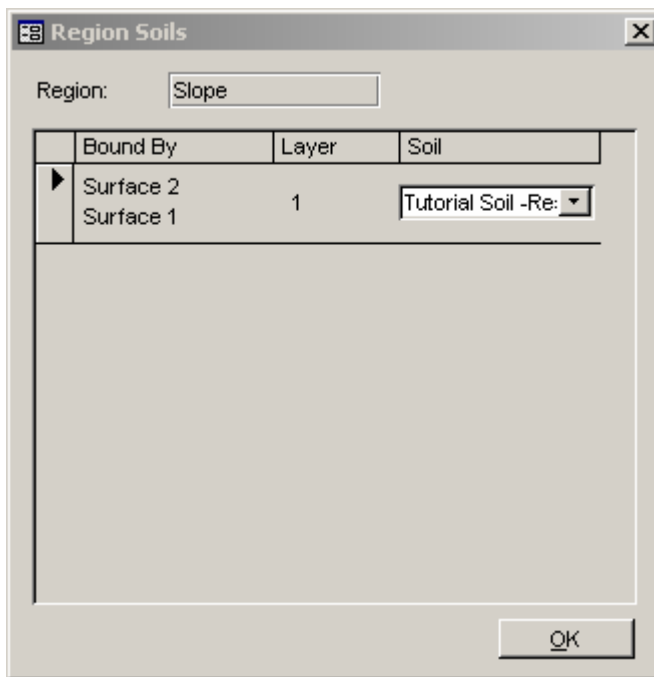
Repeat these steps to give the Reservoir a solid gray fill.

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


3.4.7 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 2 Surface 1	1	Tutorial Soil -Re: ▼

3. Select the **Tutorial Soil –Reservoir3D** soil from the drop-down for **Layer 1**.
4. Select “**Reservoir**” in the Region Selector.
5. Press the **Region Soils** button,  at the top of the workspace to open the Region Soils form.
6. Select the **Tutorial Soil –Reservoir3D** soil from the drop-down for **Layer 1**.
7. Close the form using the **OK** button.

3.4.8 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 number of heads will be defined on the Slope region segments with the Zero Flux condition being applied to the remainder. The Reservoir will be set to have a head of 10.5m as a Surface 2 boundary condition. The steps for specifying the boundary conditions include:

1. Select the "Slope" region in the drawing space.
2. 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. By default the first boundary segment will be given a Zero Flux condition.

Boundaries

Location

Region: Slope Surface: Surface 1

Segment Boundary Conditions | Surface Boundary Conditions

X	Y	Boundary Condition	Expression or Data	Units
0	0	Zero Flux		
14	0	Head Expression	7	m
24	0	Continue		
24	27	Continue		
14	27	Zero Flux		
0	27	Continue		

Update Selected Segment Segment Length: 14 m

1. Select Boundary Condition: Zero Flux

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

3. Update

Copy To Surface... Notes... OK

3. Select the **point** (14,0) from the list.
4. From the **Boundary Condition** drop down select a **Head Expression** boundary condition. This will cause the Head Expression box to be enabled.
5. In the Head Expression box enter a **head of 7**.
6. Click the **Update** button to save the boundary condition to the list.
7. Select the **point** (14,27) 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. Close the form using the **OK** button.

Slope Region Boundary Condition Summary

X	Y	Boundary Condition	Expression
0	0	Zero Flux	
14	0	Head Expression	7
24	0	Continue	
24	27	Continue	
14	27	Zero Flux	
0	27	Continue	



Tip!

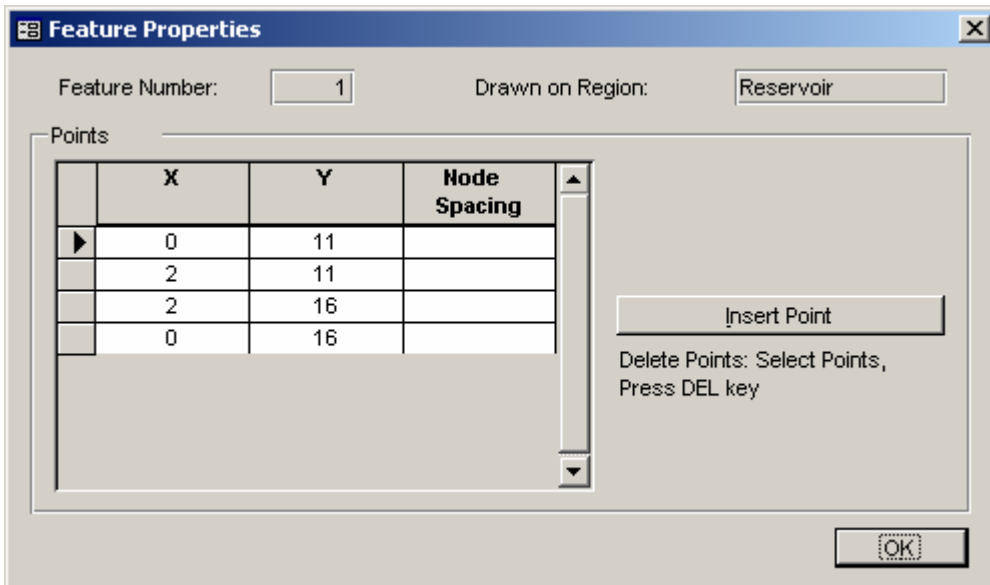
The Head Expression boundary condition for the point (14,0) becomes the boundary condition for the following line segments that have a Continue boundary condition until a new boundary condition is specified. In this case the line segments from (14,0) to (14,27) are all given a Head Expression boundary condition.

Now, to set the Reservoir's Surface 2 Boundary Condition to 10.5:


1. Select the “Reservoir” region in the drawing space.
2. From the menu select **Model > Boundaries**.
3. Select **Surface 2** from the surface drop-down.
4. Click the **Surface Boundary Condition** tab.
5. Select the **Head Expression** boundary condition and enter a value of 10.5 in the expression box.
6. Close the form using the **OK** button.

3.5 ADDING FEATURES

Features are used in this problem to control mesh density for increased resolution at selected slices.



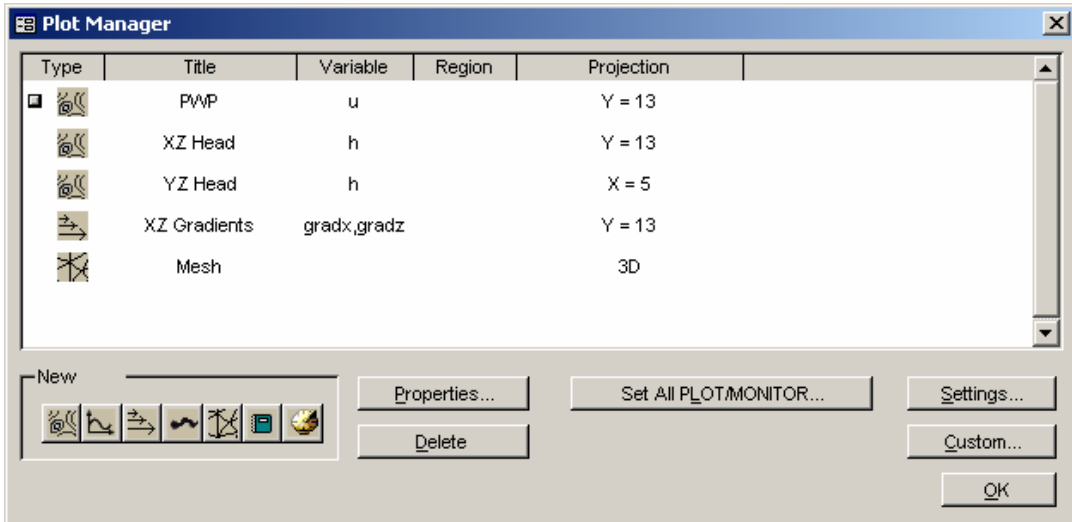
To add the features:

1. Select the **Reservoir** region from the region selector.
2. Click on the **Feature** button, .
3. With the mouse **click** on the point (0,11). (The SNAP and GRID options must be on in the workspace.)
4. **Click** on the point (2,11).
5. **Click** on the point (2,16).
6. **Double-click** on the point (0,16) to finish the Feature.

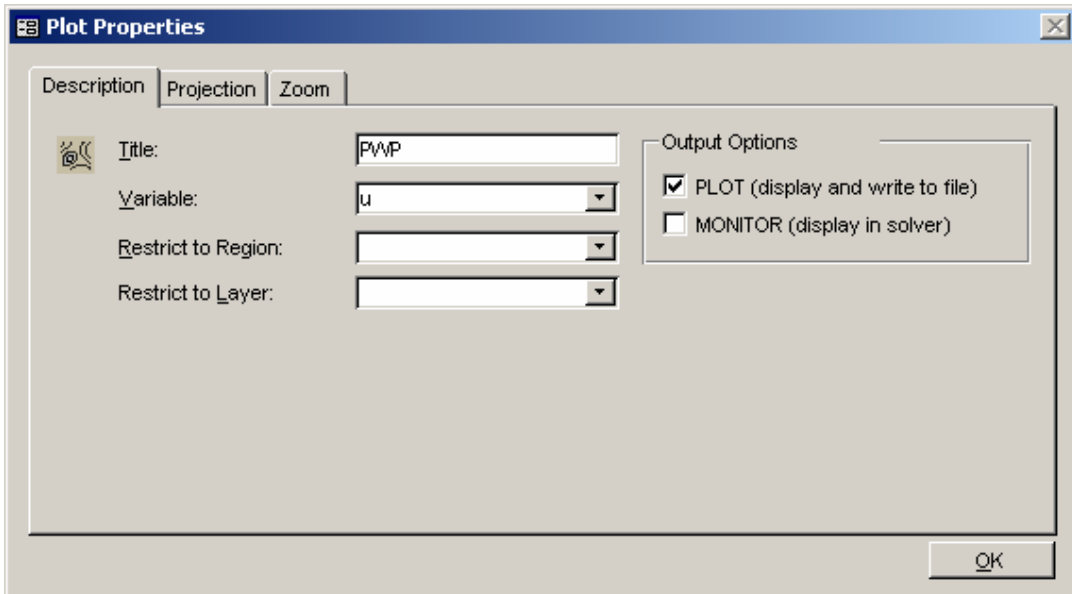
3.6 SPECIFY PLOTS

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 pressure contours, head contours, solution mesh, and gradient 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.



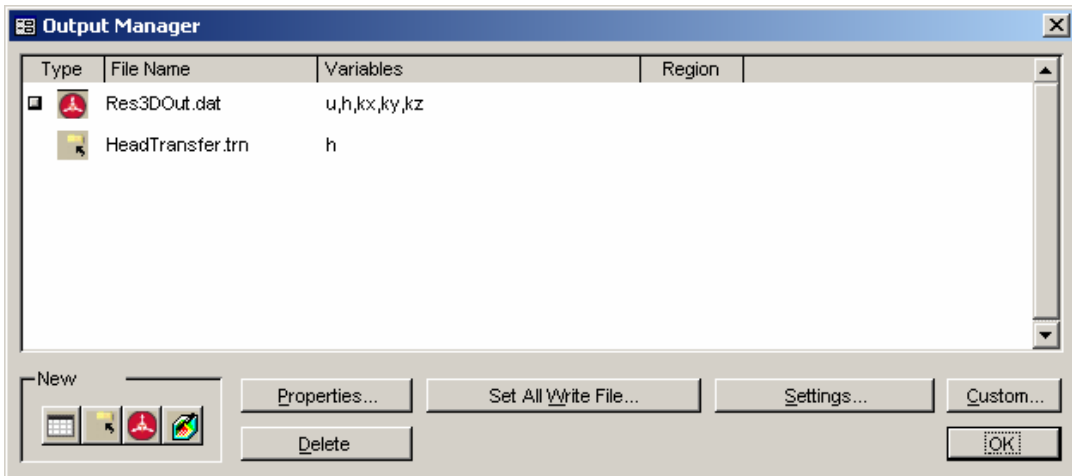
3. Enter the title **PWP**.
4. Select **u** as the variable to plot from the drop-down.
5. Select the **PLOT** output option.
6. Move to the **Projection** tab.
7. Select **Plane** Projection option.
8. Select **Y** from the Coordinate Direction drop-down.

9. Enter **13** in the Coordinate field. This will generate a 2D slice at $Y = 13\text{m}$ on which the pore water pressures will be plotted.
10. Click **OK** to close the form and add the plot to the list.
11. Repeat these steps 2 – 9 to create the plots shown above. Note that the Vector and Mesh plots do not require entry of a variable.
12. Click **OK** to close the Plot Manager and return to the workspace.

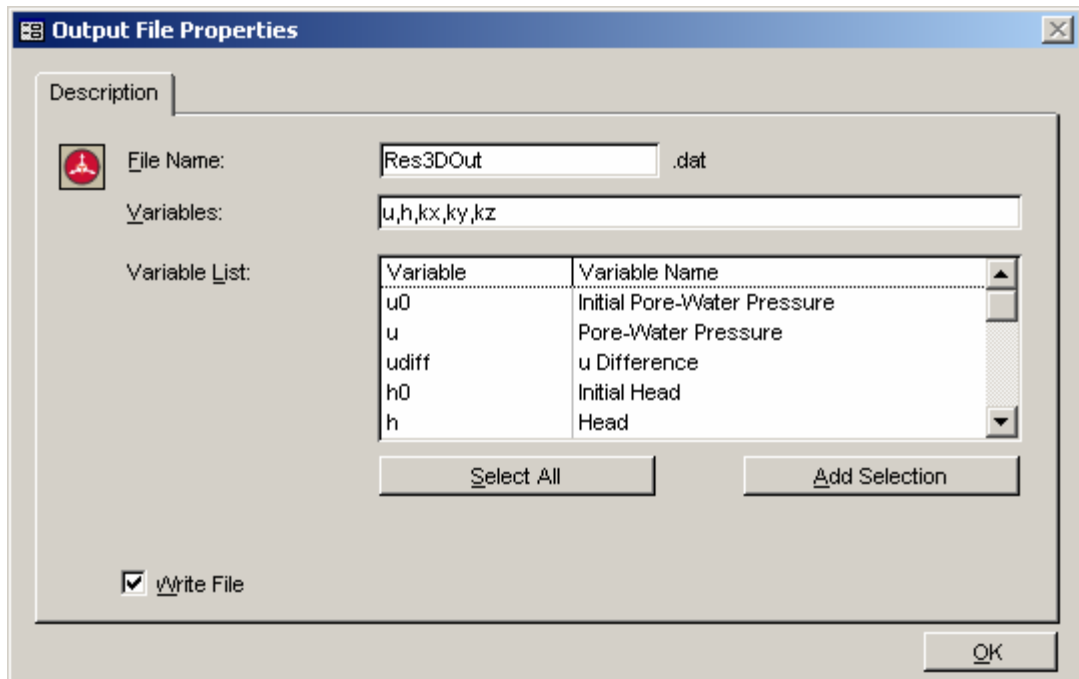
3.7 SPECIFY OUTPUT FILES

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 heads, and a plot to transfer the results to AcuMesh. 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 AcuMesh button to begin adding the output file. The Output File Properties form will open.



3. Enter the title Res3DOut.
4. Select the variables u, h, kx, ky, and kz in the variable list.
5. Press the **Add Selection** button.
6. Make sure the **Write File** box contains a check mark.
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.

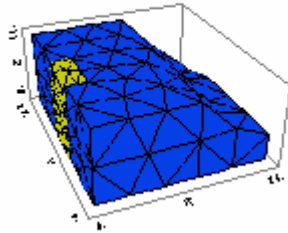
3.8 ANALYZE

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.

3.9 RESULTS

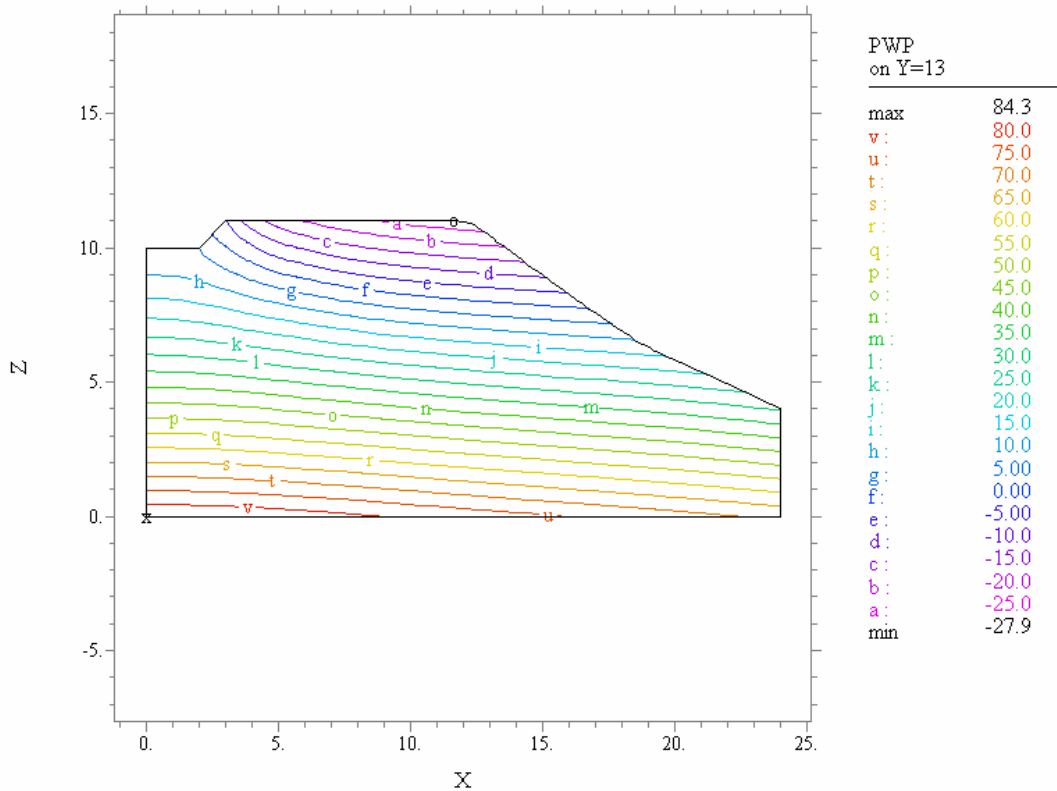
After the problem has finished solving, the results will be displayed in the form of thumbnail plots within the SVFLUX solver. Right-click the mouse and select Maximize to enlarge any of the thumbnail plots. This section will give a brief analysis for each plot that was generated.

3.9.1 Solution Mesh



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.

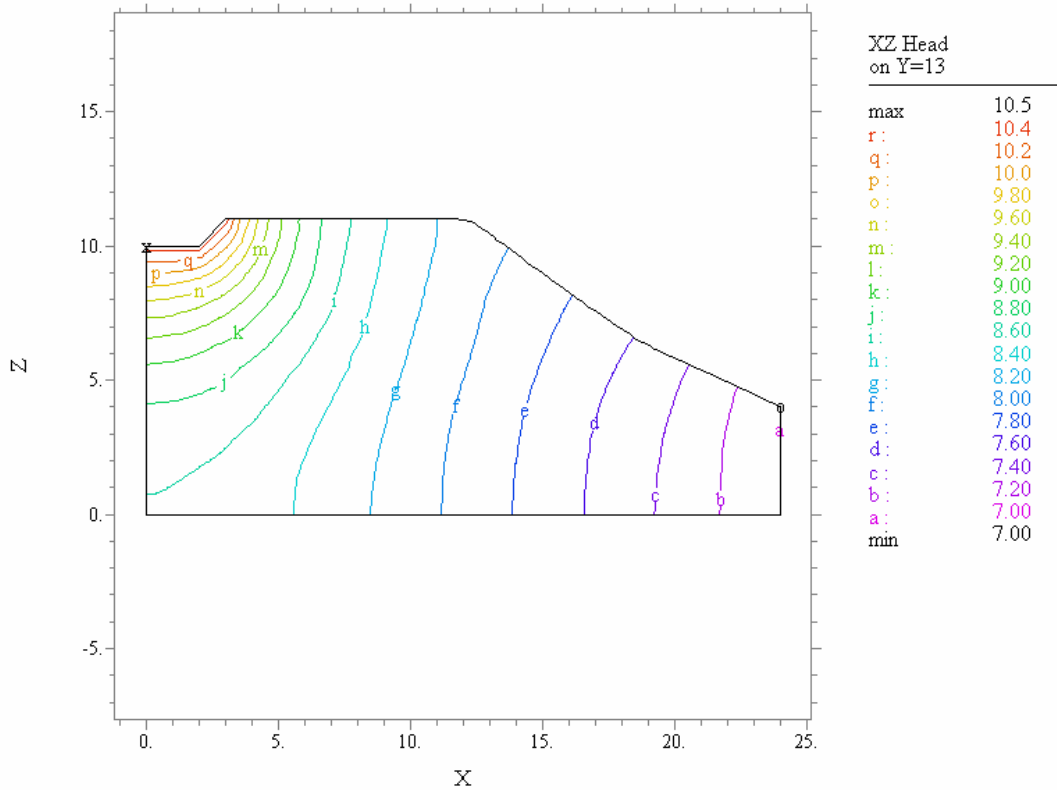
3.9.2 Pressure Contours



Tutorial_Reservoir3D: Grid#4 p2 Nodes=1605 Cells=900 RMS Err= 1.9e-4
 Stage 1 Integral= 7220.713

The most important contour in the above plot is the one that corresponds to zero pressure. This contour represents the phreatic surface. All soil that lies below this line is saturated and all soil that lies above this line is considered to be unsaturated. The plot indicates a gradual decrease in water pressure from the left to the right in the plot.

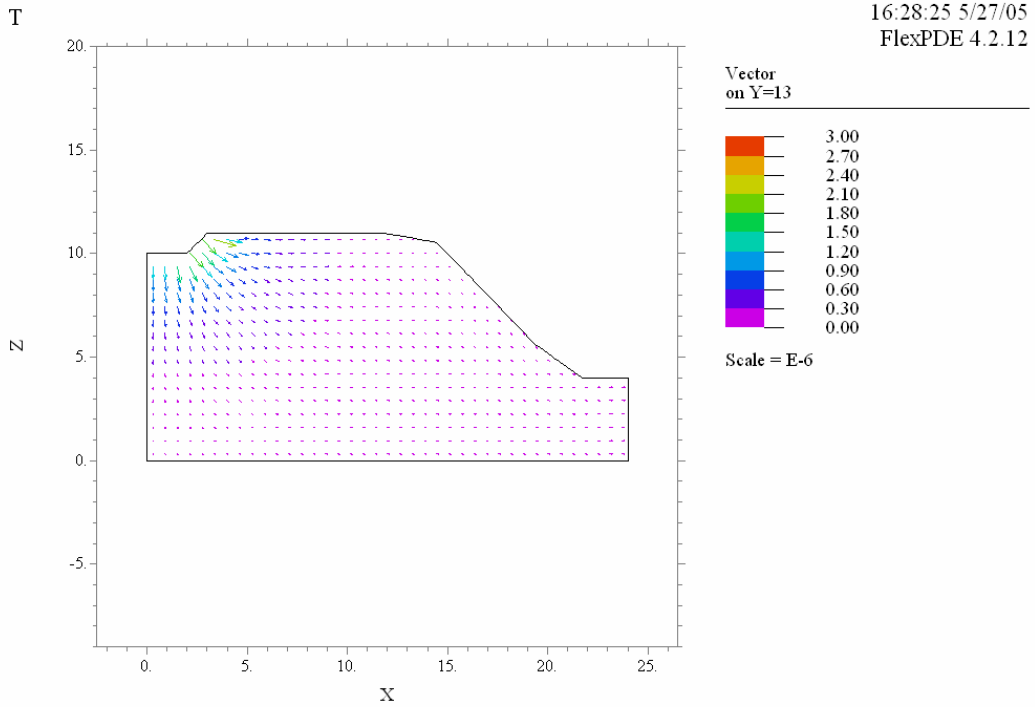
3.9.3 Head Contours



Tutorial Reservoir3D: Grid#4 p2 Nodes=1605 Cells=900 RMS Err= 1.9e-4
 Stage 1 Integral= 1771.237

As expected, the head is 10.5m at the left of the plot where this condition was specified as a boundary condition. The head decreases to 7m on the right edge of the plot where the boundary condition was set to 7m.

3.9.4 Flow Vectors



Tutorial_Reservoir3D: Grid#5 p2 Nodes=5754 Cells=3539 RMS Err= 7.3e-5
Stage 1

Flow Vectors show both the direction and the magnitude of the flow at specific points in the problem. Vectors illustrate that flow is from left to right in this view.

4 REFERENCES

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

FlexPDE 3.x Reference Manual, 1999. PDE Solutions Inc. Antioch, CA 94509

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

Jacob Bear, 1972. *Dynamics of Fluids in Porous Media*. Dover Publications Inc., New York

Jason S. Pentland, 2000. *Use of a General Partial Differential Equation Solver for Solution of Heat and Mass Transfer Problems in Soils*. University of Saskatchewan, Canada

Nguyen Thi Minh Thieu, 1999. *Solution of Saturated/Unsaturated Seepage Problems Using a General Partial Differential Equation Solver*. University of Saskatchewan, Canada

R. Allan Freeze and John A. Cherry, 1979. *Groundwater*. Prentice –Hall, Inc., Englewood Cliffs, New Jersey

5 3D EXAMPLE PROBLEM DATA

Surface Data for Tutorial – Reservoir3D:

Surface 2

X	Y	Z
0	0	11
0	10	11
0	11	10
0	16	10
0	17	11
0	27	11
2	0	11
2	10	11
2	11	10
2	16	10
2	17	11
2	27	11
3	0	11
3	10	11
3	11	11
3	16	11
3	17	11
3	27	11
14	0	11
14	10	11
14	11	11
14	16	11
14	17	11
14	27	11
21	0	4
21	10	4
21	11	4
21	16	4
21	17	4
21	27	4
24	0	4
24	10	4

24	11	4
24	16	4
24	17	4
24	27	4

Region shape data for Tutorial – Reservoir3D:

Slope Region

X	Y
0	0
14	0
24	0
24	27
14	27
0	27

Reservoir Region

X	Y
0	10
3	10
3	17
0	17

6 INDEX

- Boundaries, 18, 19, 48, 50
- Boundary Conditions, 18, 48
- Color, 17, 47
- Conductivity, 5
- Elevations, 43
- Features, 50
- Fill, 5, 6, 8, 9, 17, 28, 34, 46
- Flux Section, 20, 21, 30
- Geometry, 15, 44
- Grid, 11, 39, 42
- Head Expression, 19, 48, 49
- Heat, 57
- Hydraulic Conductivity, 13, 14, 41
- ksat, 14, 32, 41
- Mass, 57
- Mesh, 24, 52, 54
- Modified Campbell, 28, 29
- Output, 22, 23, 24, 52, 53
- Output Files, 22, 52
- Plots, 21, 50
- Problem, 5, 7, 8, 9, 28, 30, 32, 35, 36, 37, 58
- ProblemID, 5, 8, 9, 32, 36, 37
- Problems, 5, 6, 7, 33, 34, 35, 57
- Project, 5, 6, 33
- ProjectID, 5, 6, 32, 33, 34
- Projects, 5, 6, 7, 33, 34, 35
- Properties, 6, 12, 16, 17, 20, 21, 22, 23, 29, 33, 40, 45, 46, 47, 51, 52
- Region, 14, 15, 16, 17, 21, 44, 45, 46, 47, 49, 59
- Regions, 14, 44
- Save**, 28
- Seepage, 1, 57
- Selector, 15, 41, 42, 44, 47
- Shape, 15, 16, 17, 44, 45, 46
- Shapes, 15, 44
- Soil, 12, 14, 40, 47, 57
- Soils, 12, 29, 40, 41, 47, 57
- Solution, 24, 54, 57
- Stage, 29
- Steady-State, 7, 9, 35, 37
- Suction, 5
- Surface, 41, 42, 48, 58
- Surfaces, 41
- SVFLUX, 2, 5, 6, 12, 14, 15, 24, 32, 34, 40, 44, 46, 53
- SWCC, 14
- Transient, 10, 38
- View, 10, 11, 38, 39
- Workspace, 10, 38
- Zero Flux, 18, 19, 48, 49