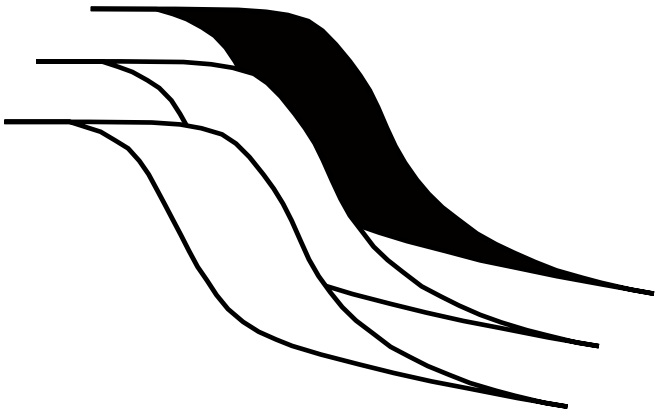


SVFLUX™

Tutorial Manual

Saturated/unsaturated finite element 2D/3D seepage modeling

ED-3A

Written for use with SVFLUX version 3.x

October 20, 2003

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 registered trademark of SoilVision Systems Ltd.
Tecplot® is a registered trademark of Amtec Engineering Inc.
FlexPDE® is a registered trademark of PDE Solutions Inc.

Copyright © 1996-2003
by
SoilVision Systems Ltd.
Saskatoon, Saskatchewan, Canada
ALL RIGHTS RESERVED
Printed in Canada

1.0 A Two Dimensional Example Problem 5
1.1 Adding a Project 5
1.2 Adding a Problem 6
1.3 Opening the Problem 8
1.4 Defining the Problem 9
1.4.1 Specify Settings 9
1.4.2 Setting the Workspace 9
1.4.3 Define Material Properties 11
1.4.4 Adding Regions 13
1.4.5 Defining Region Geometry Shapes 13
- Define the Dam 14
- Define the Core 14
- Define the Filter 15
- Formatting a Shape 15

1.4.6 Specify Boundary Conditions 16
- Dam 17
- Filter 17

1.5 Specifying a Flux Section 18
1.6 Specify Plots 20
1.7 Verify the Problem 21
1.8 Analyze 22
1.9 Results 22
1.9.1 Solution Mesh 23
1.9.2 Pressure Contours 23
1.9.3 Head Contours 25
1.9.4 Flow Vectors 26
1.9.5 Flux Report 26
2.0 A Three Dimensional Example Problem 27
2.1 Adding a Project 28
2.2 Adding a Problem 29
2.3 Opening the Problem 30
2.4 Defining the Problem 31
2.4.1 Specify Settings 31
2.4.2 Setting the Workspace 31
2.4.3 Define Material Properties 33
2.4.4 Define 3D Surfaces 35
- Define Surface 1 36
- Define Surface 2 36
- Define Surface 3 37
- A Pinch Out Surface 38

2.4.5 Adding Regions 39
2.4.6 Defining Region Geometry Shapes 39
- Define the Main region 39
- Define the River 40
- Formatting a Shape 41

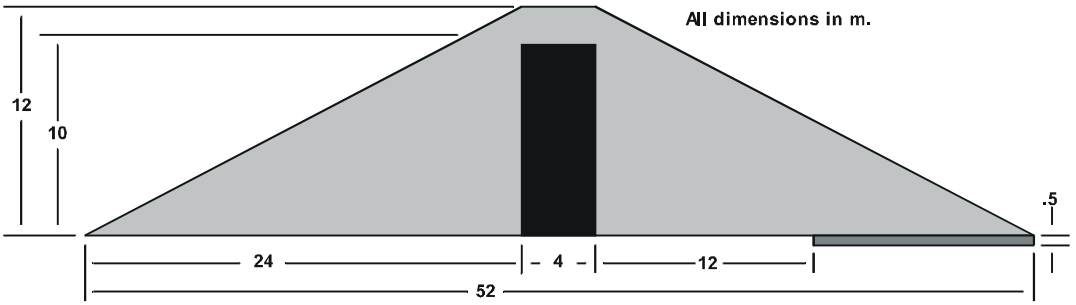
2.4.7 Specifying a Soil by Region and Layer 42
2.4.8 Specify Boundary Conditions 42

2.5 Adding Features 44
2.6 Specify Plots 45
2.7 Verify the Problem 46
2.8 Analyze 47
2.9 Results 47
2.9.1 Solution Mesh 48
2.9.2 Pressure Contours 48
2.9.3 Head Contours 50
2.9.4 Flow Vectors 51
3.0 References 52
4.0 3D Example Problem Data 54
5.0 Index 58

1.0 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

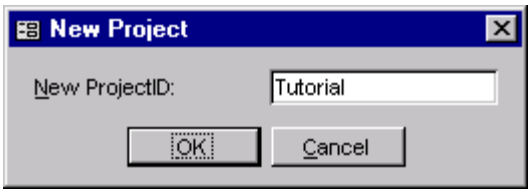
Sand: ks=1E-04

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



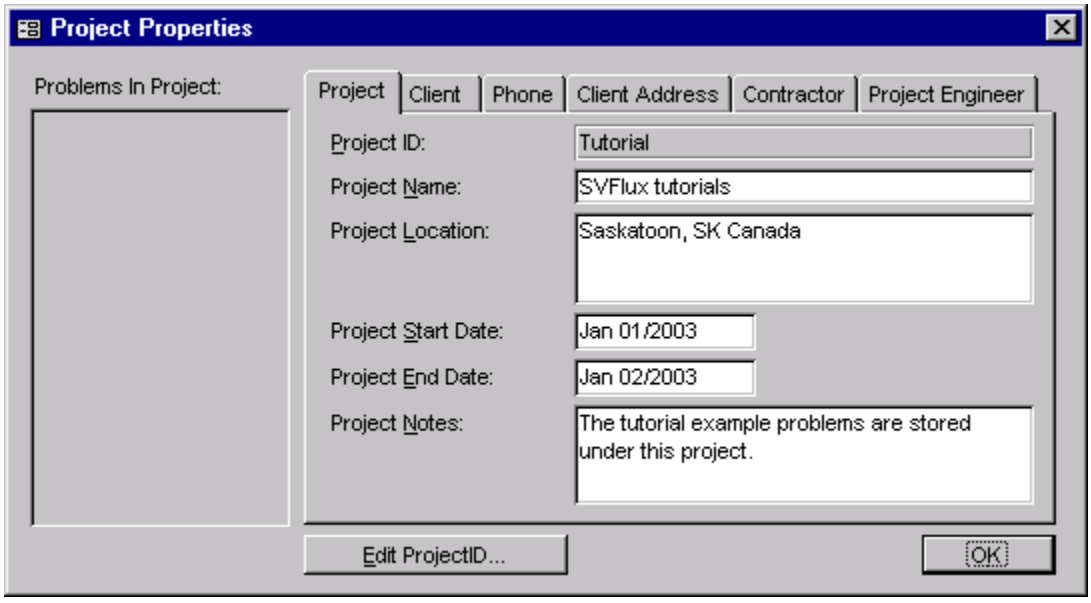
4. Type “Tutorial” as the new ProjectID and press OK.

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.



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.

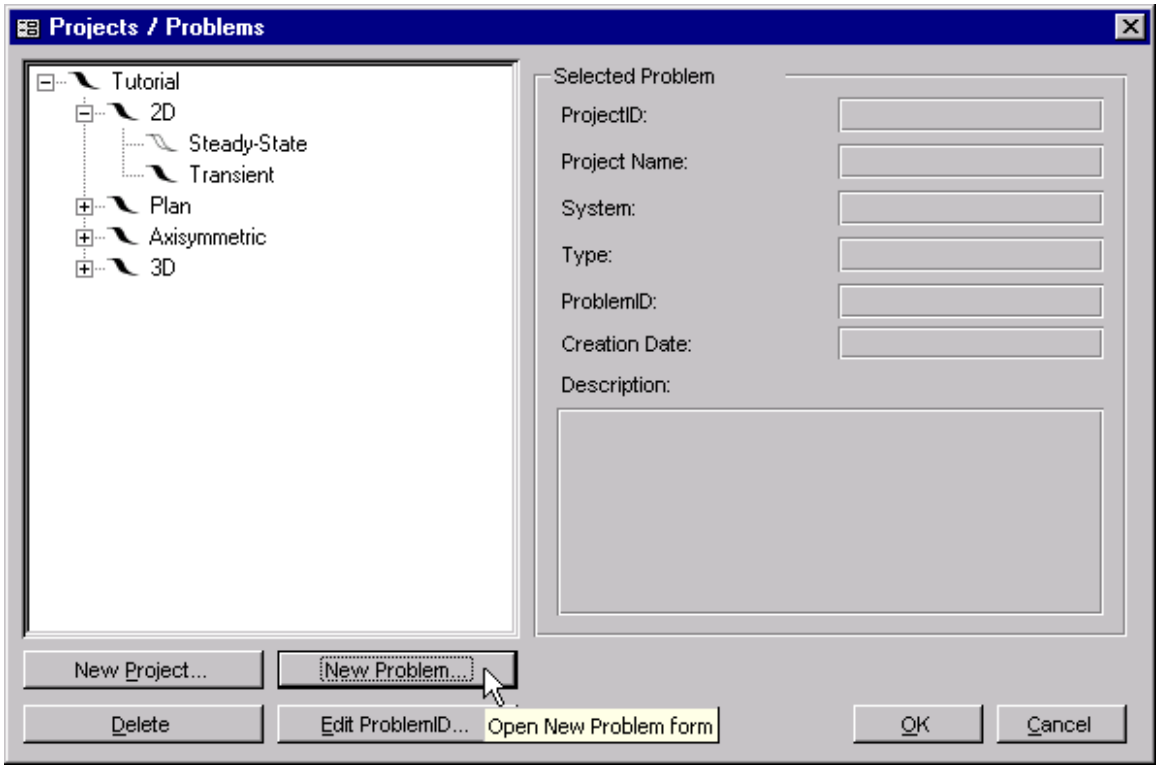


It should be noted that once the project is defined it will be identified by the ProjectID throughout the rest of the program. Also, SVFLUX does not allow you to specify two projects with the same ProjectID.

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.

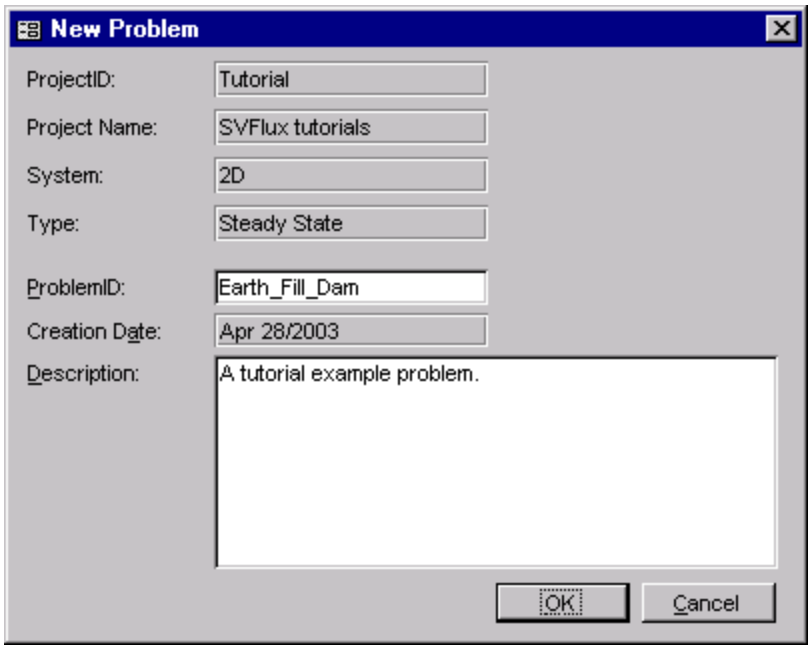
1.2 Adding a Problem

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:

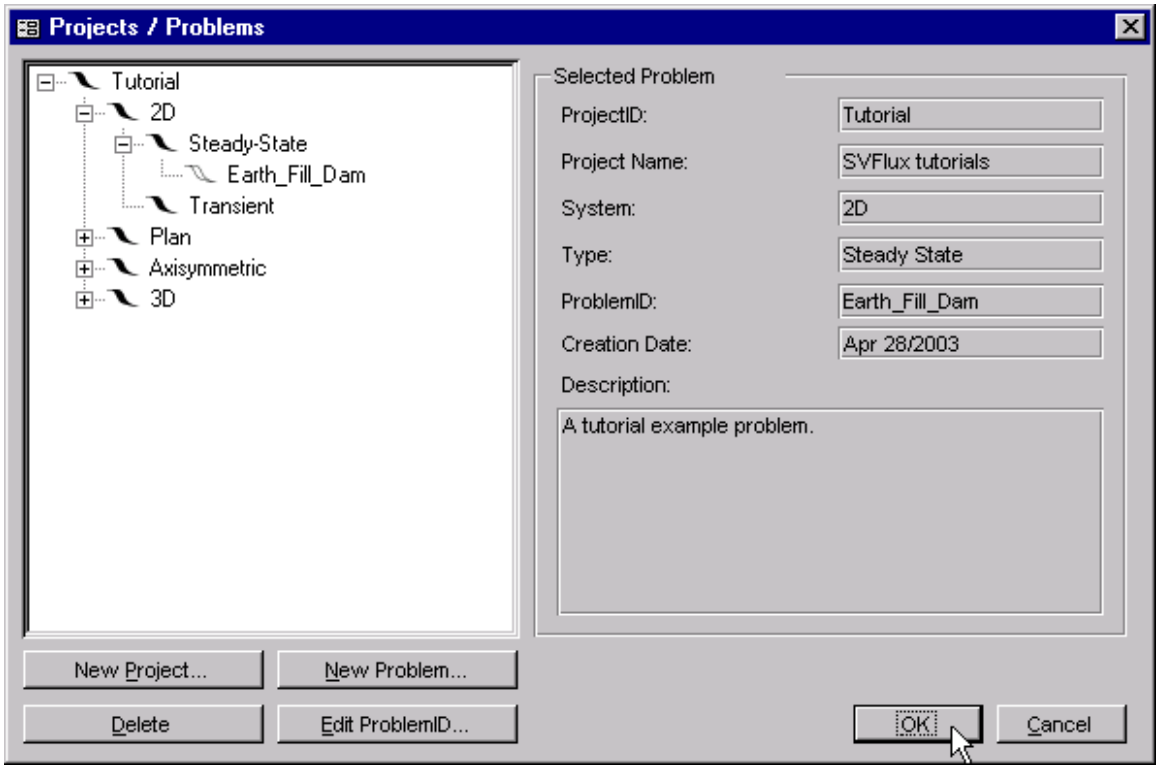
1. Click on the plus sign and expand the project **Tutorial**.
2. This example will be modeled in two-dimensions and is steady-state so we must expand **2D** by clicking the plus sign beside "2D"
3. Select **Steady-State**.
4. Click the **New Problem** button. The New Problem form will open.



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.

1.3 Opening the Problem

To open the problem:



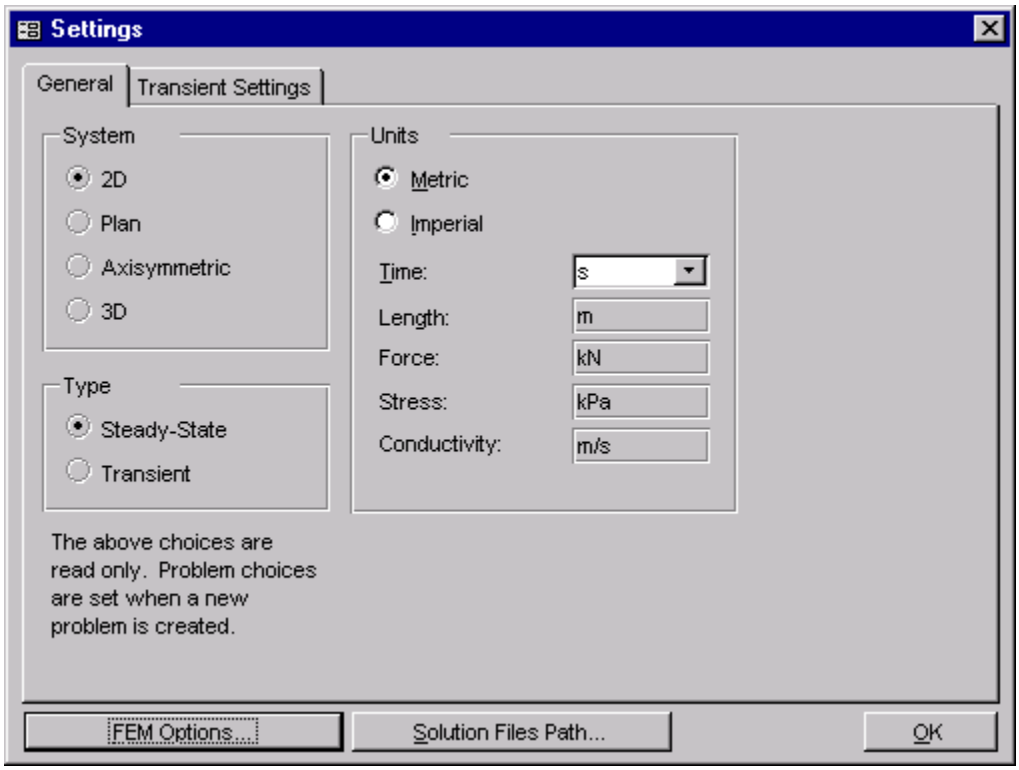
1. **Navigate** back to the problem via Tutorial, 2D, Steady-State.
2. Select **Earth_Fill_Dam**.
3. The problem may opened by clicking the **OK** button or by double clicking on the ProblemID. The problem will now be opened in the workspace.

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.

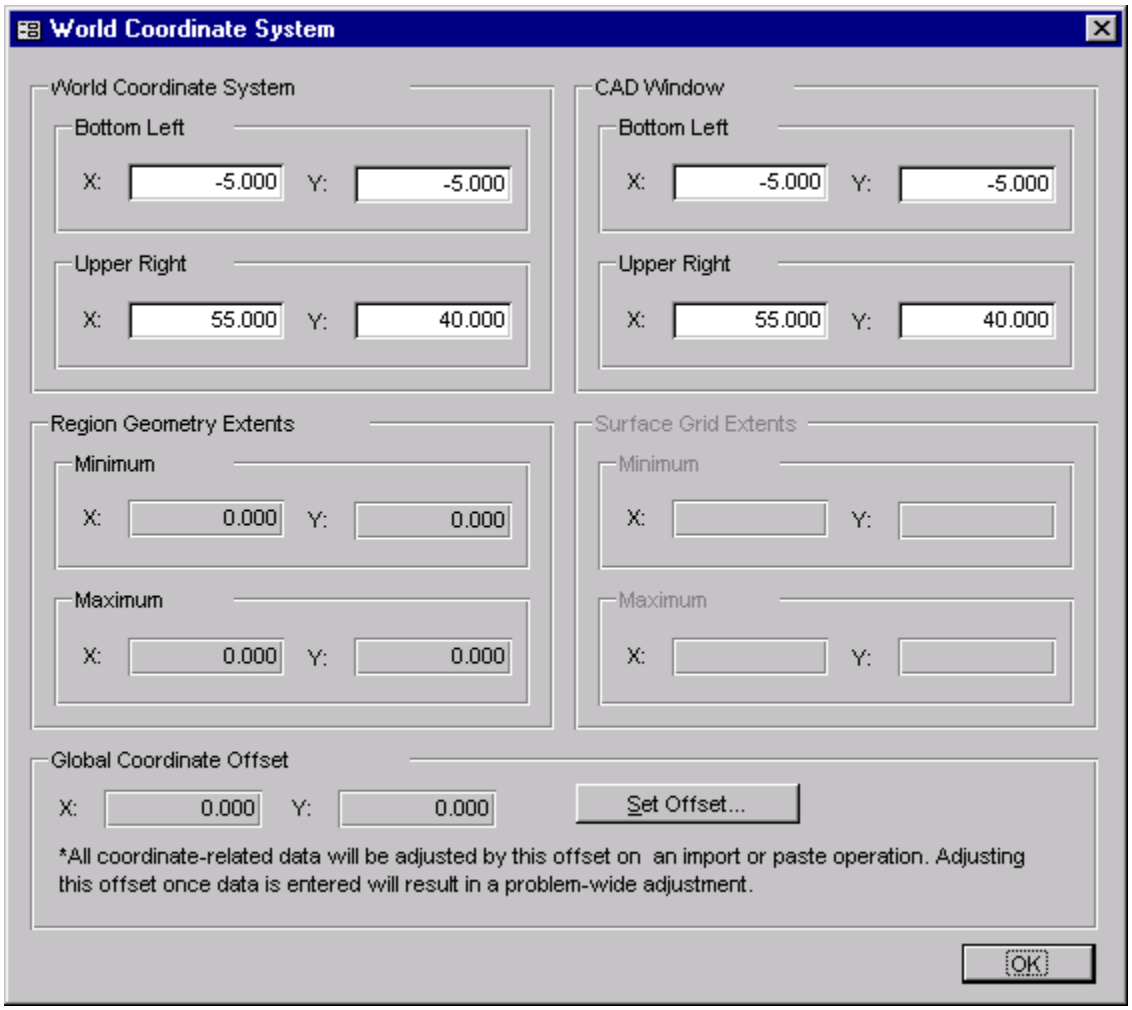


The Settings form will contain information about the current problem System, Type, Units, and Transient Settings. The hydraulic conductivity data for the soils contained in the problem are reported as m/s so the time units will remain s, for seconds.

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.

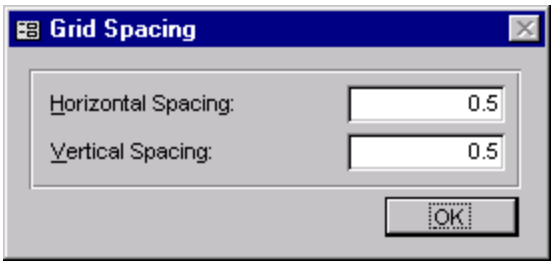
1. Access the World Coordinate System form by either of two ways. Click on the button:  or select **View > WCS**. The button is located in the view toolbar to the left of the drawing space.



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.

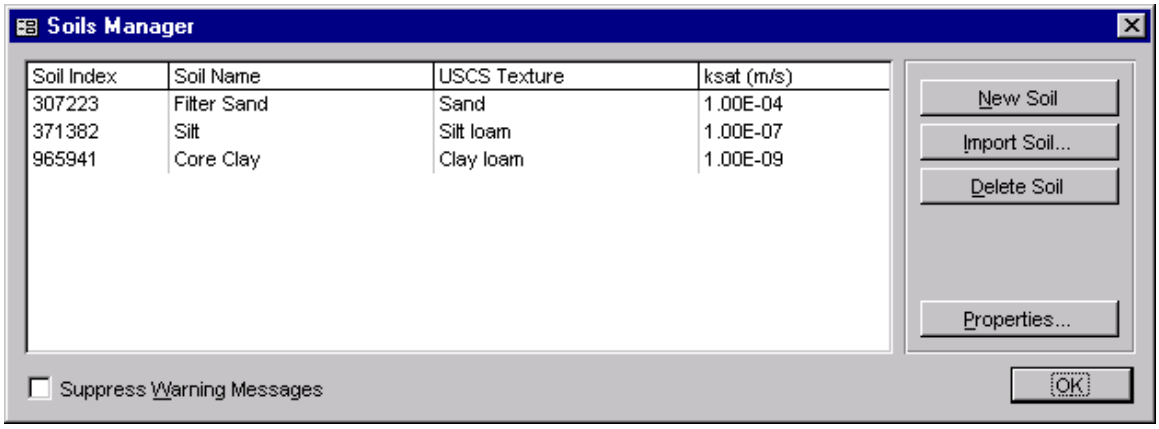
The workspace grid spacing needs to be set to aid in defining region shapes. The filter portion of the problem has coordinates of a precision of 0.5m. In order to effectively draw geometry with this precision using the mouse the grid spacing must be set to a maximum of 0.5.

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



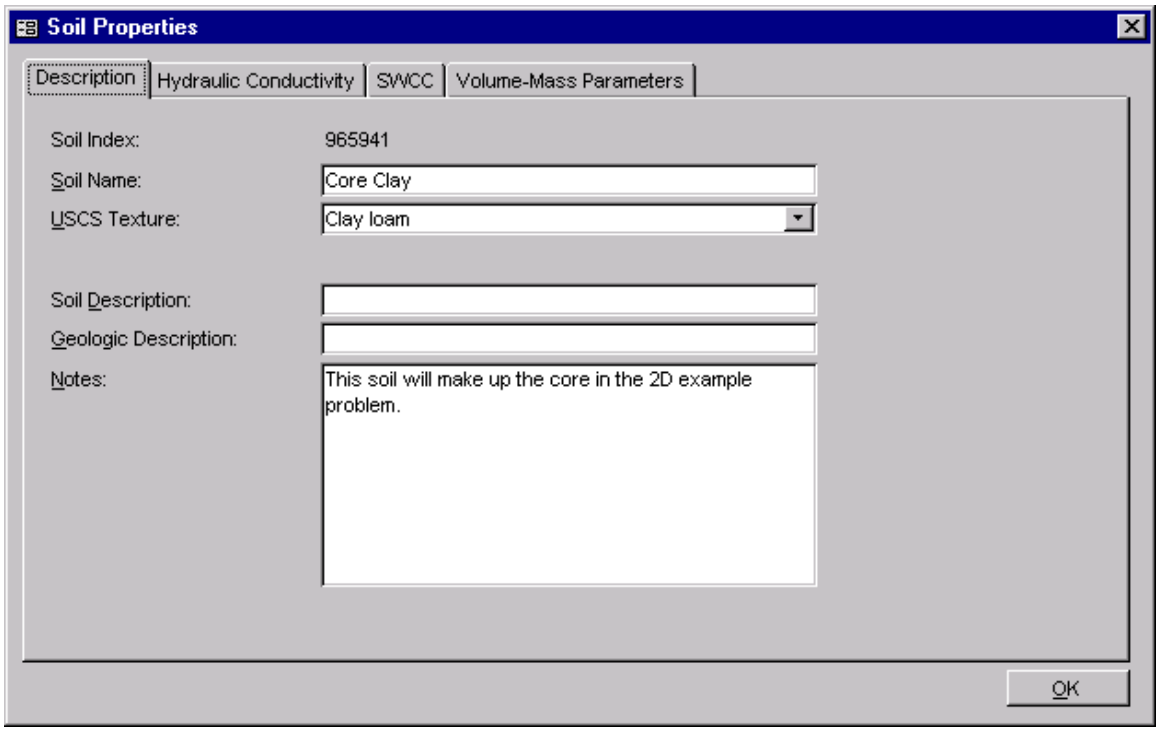
6. Enter **0.5** for both the horizontal and vertical spacing.
7. Click **OK** to close the form.

1.4.3 Define Material Properties

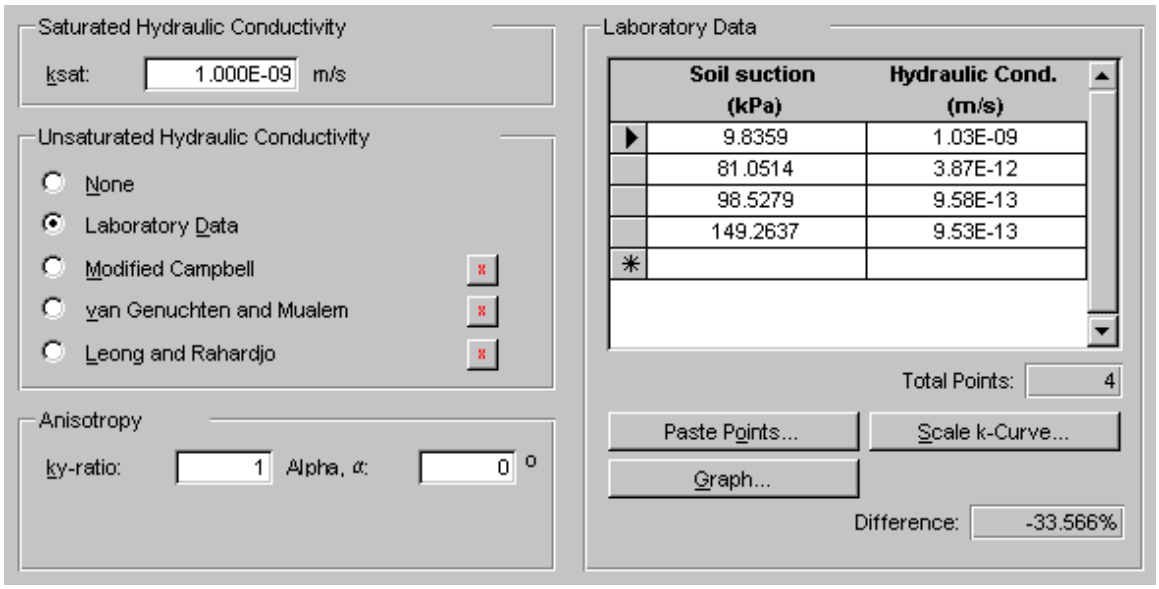


The next step in defining the problem is to enter the material properties for the three soils that will be used in the model. A clay is defined for the core, the silt will make up the dam, and the filter will consist of a sand. This section will provide instructions on creating the clay soil. Repeat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



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-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!

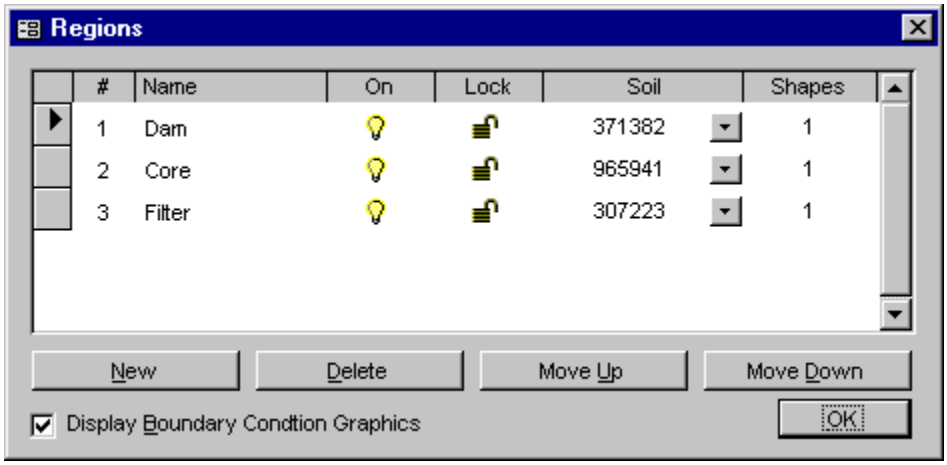
To view the hydraulic conductivity curve of the laboratory data press the Graph button.

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.



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

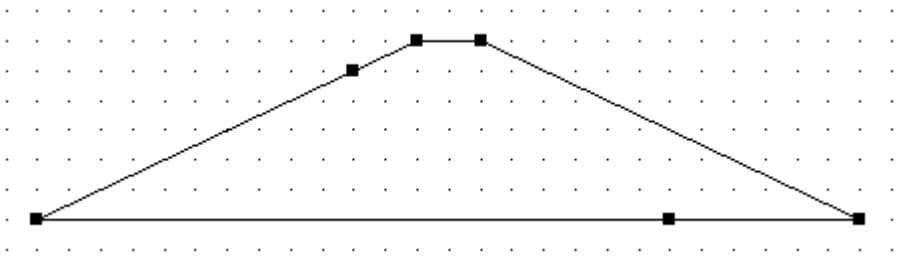
Filter

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:





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 24,10 and press Enter.
8. 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.

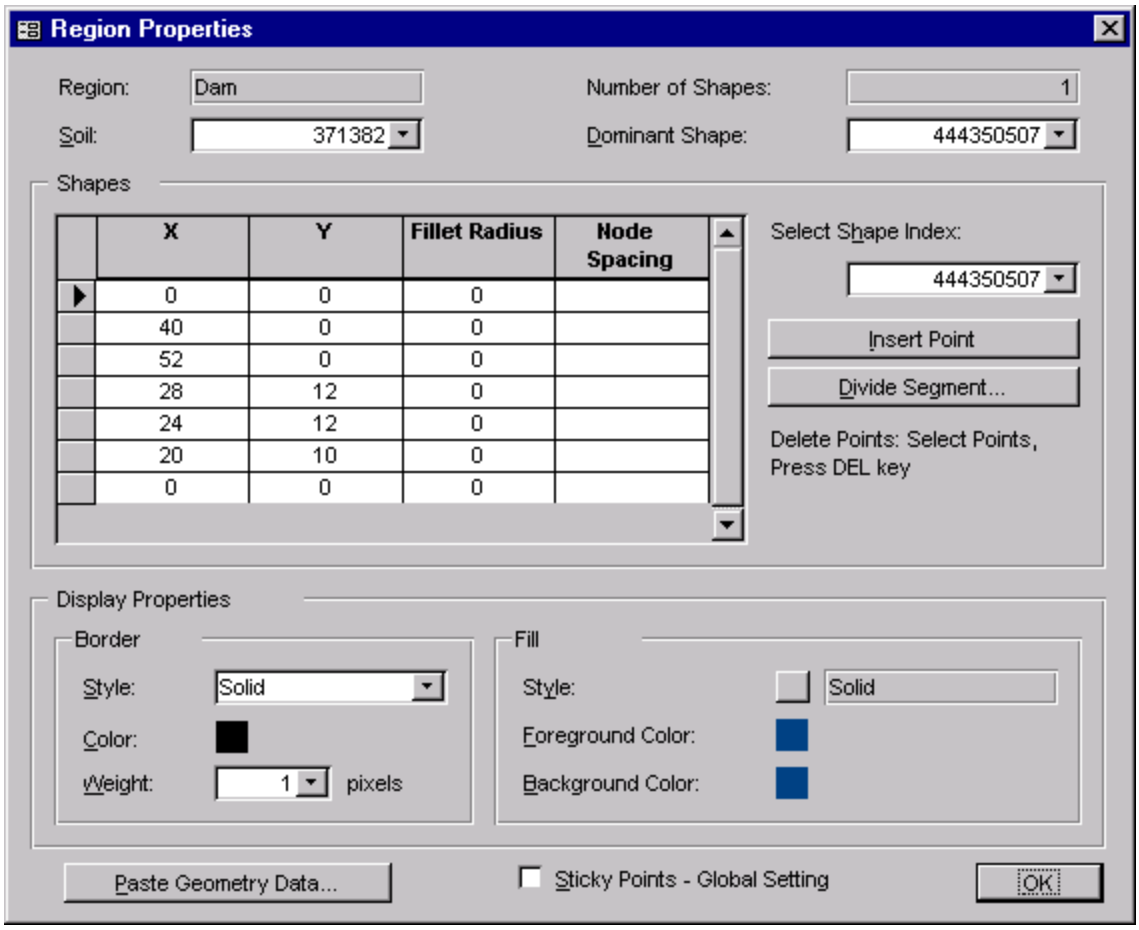


At times it may be tricky to snap to a grid point that is near a line defined for a region. Turn the object snap off by clicking on “OSNAP” in the status bar to alleviate this problem.

- **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:



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:



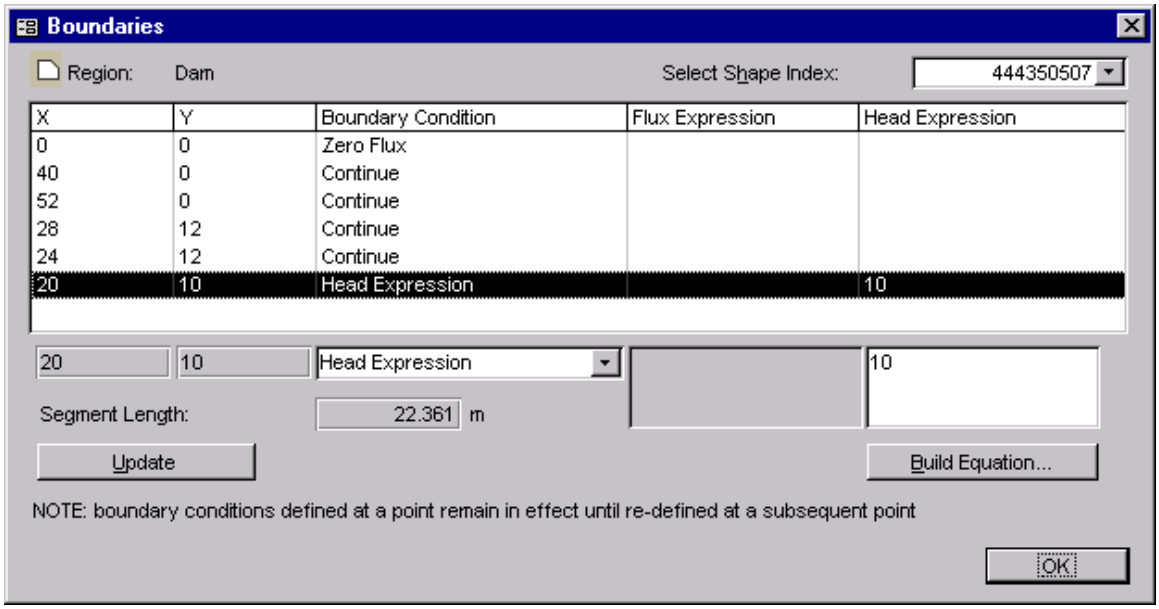
1.4.6 Specify Boundary Conditions

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.

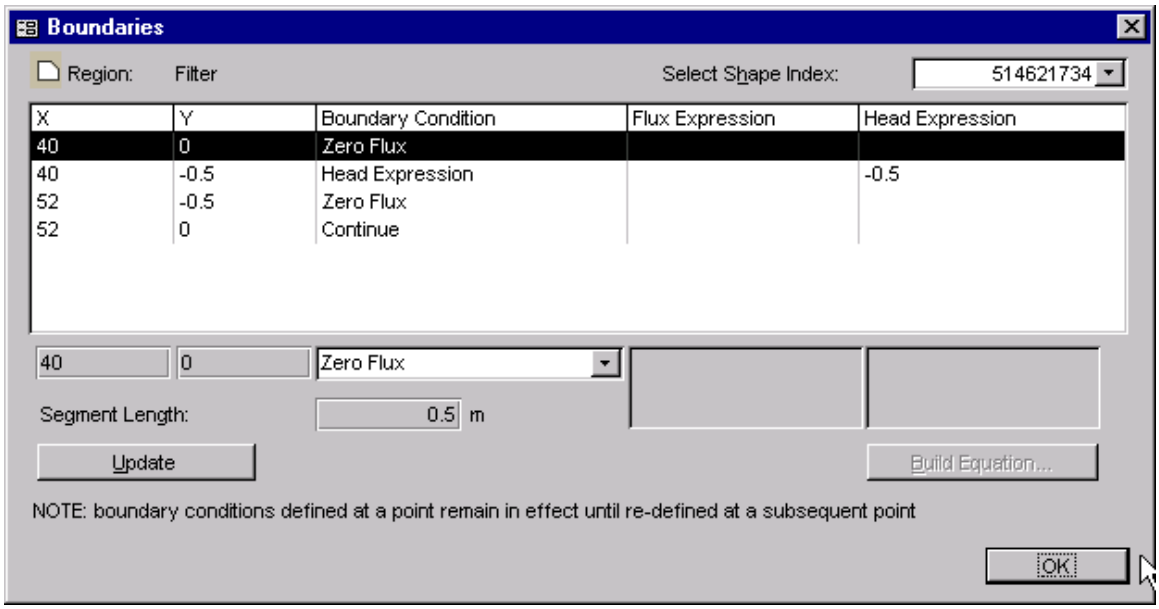


3. Select the **point** (20,10) from the list.
4. From the **Boundary Condition** drop down select a **Head Expression** boundary condition. This will cause the Head Expression box to be enabled.
5. In the Head Expression box enter a **head of 10**.
6. Click the **Update** button to save the boundary condition to the list.



The Zero Flux 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. In this case the line segments from (0,0) to (52,0), (52,0) to (28,12), (28,12) to (24,12), and (24,12) to (20,10) are all given a zero flux boundary condition. By specifying a head of 10 at point (20,10) the boundary condition for the line segment extending from (20,10) to (0,0) is set to a head equal to 10m.

- **Filter**

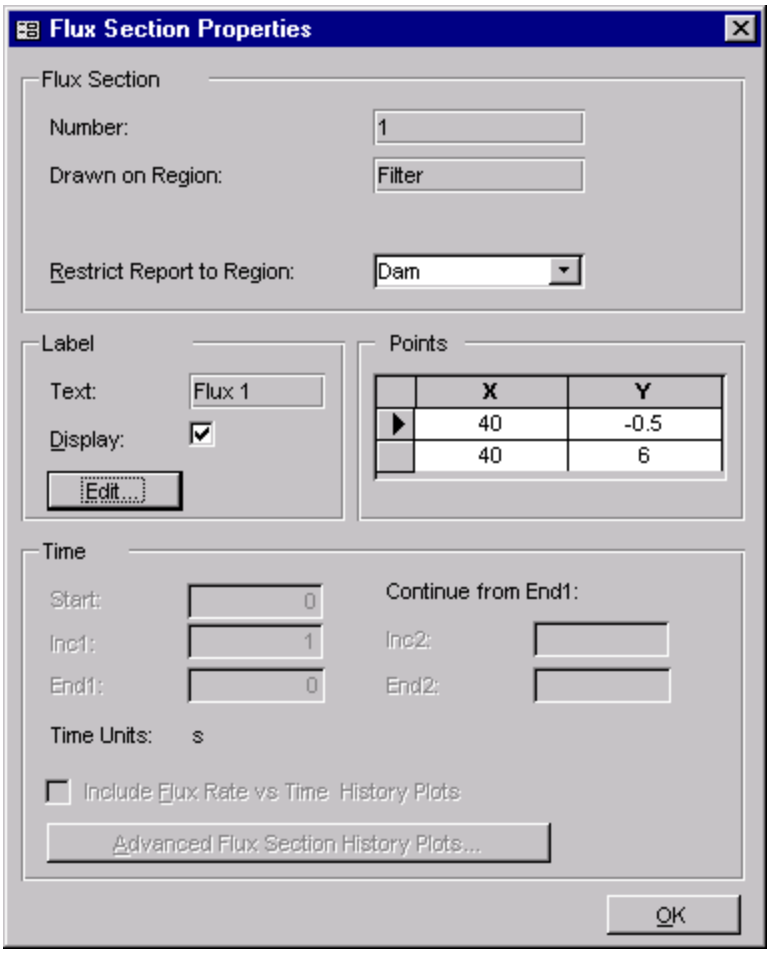


1. Select the Filter region from the drawing space.
2. Press the **Boundaries** button,  to open the Boundaries form.
3. Select the point (40,-0.5) from the list.
4. In the Boundary Condition drop down select a **Head Expression** boundary condition.
5. Enter the value **-0.5**.
6. Click the **Update** button to save the change.
7. Select the point (52,-0.5) from the list.
8. In the Boundary Condition drop down select a **Zero Flux** boundary condition.
9. Click the **Update** button to save the change.
10. Click **OK** to return to the workspace.

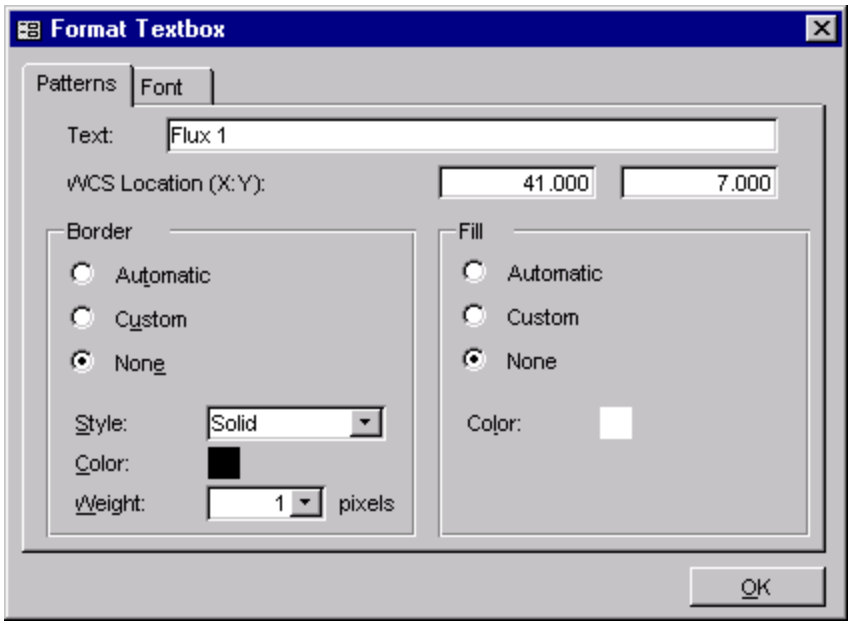
1.5 Specifying a Flux Section

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 (28,-05).
3. To finish the Flux Section click on the point (28,12). 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.



7. 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.
8. 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.



9. Enter 41 and 7 in the WCS Location fields.
10. Press OK to close the form.
11. Close the Flux Section Properties form.
12. Close the Flux Section List form.

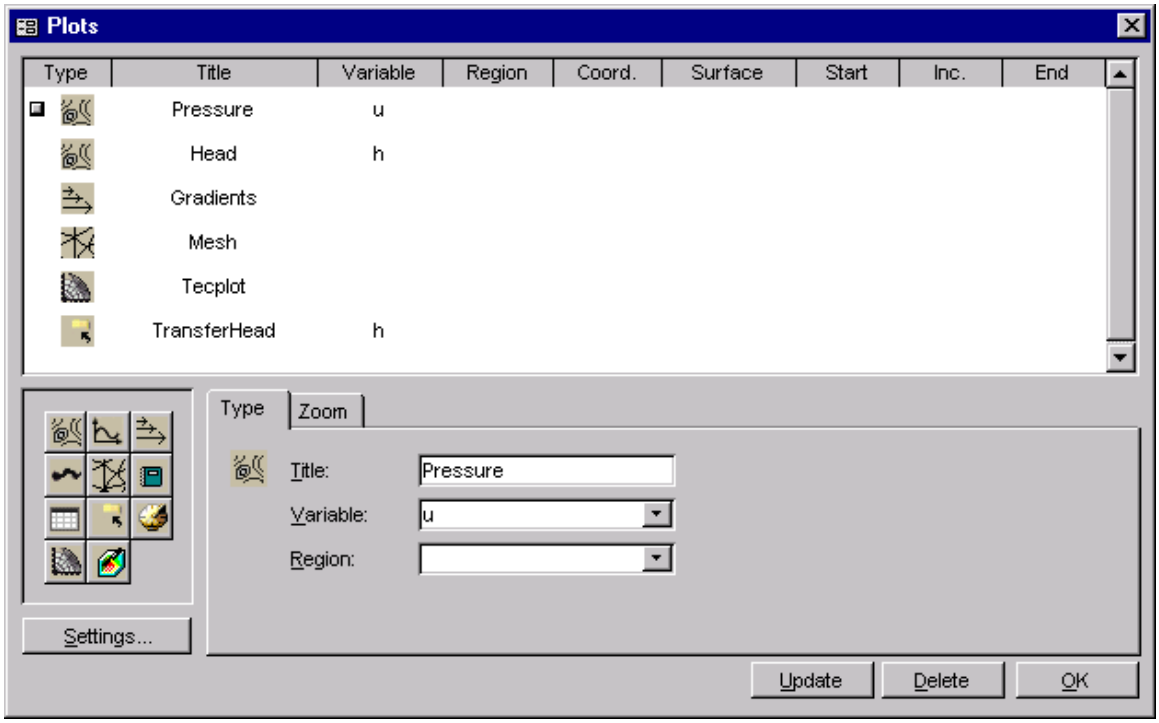


Flux Section labels can be formatted in the same manner as regular textboxes.

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, gradient vectors, and a plot to transfer the results to Tecplot.

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

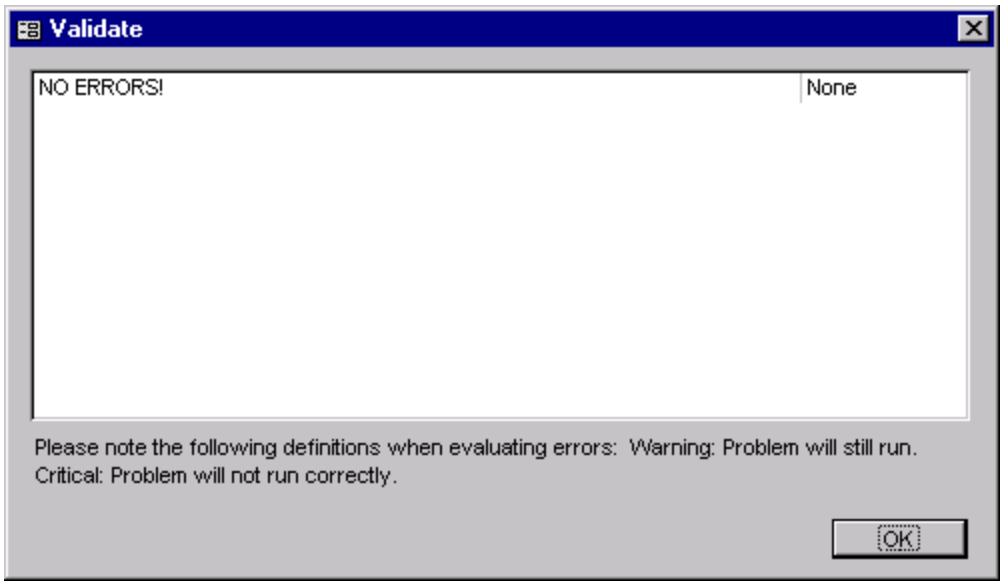


2. The toolbar at the bottom left of the form contains a button for each plot type. Click on the Contour button to begin adding the first contour plot.
3. Enter the title Pressure.
4. Select u as the variable to plot from the drop-down.
5. Click **Update** to add the plot to the list.
6. Repeat these steps 2 – 5 to create the plots shown above. Note that the Mesh and Tecplot plots do not require entry of a variable. The Tecplot plot will generate a file containing all the mesh and variable data calculated by the solver.
7. Click **OK** to return to the workspace.

1.7 Verify the Problem

The validation form will provide a summary of checks and errors that are present in the model before the problem is analyzed. To validate the problem, click the check mark as shown below.





In the case shown here the problem is showing that there are no errors and it is ready to be analyzed.

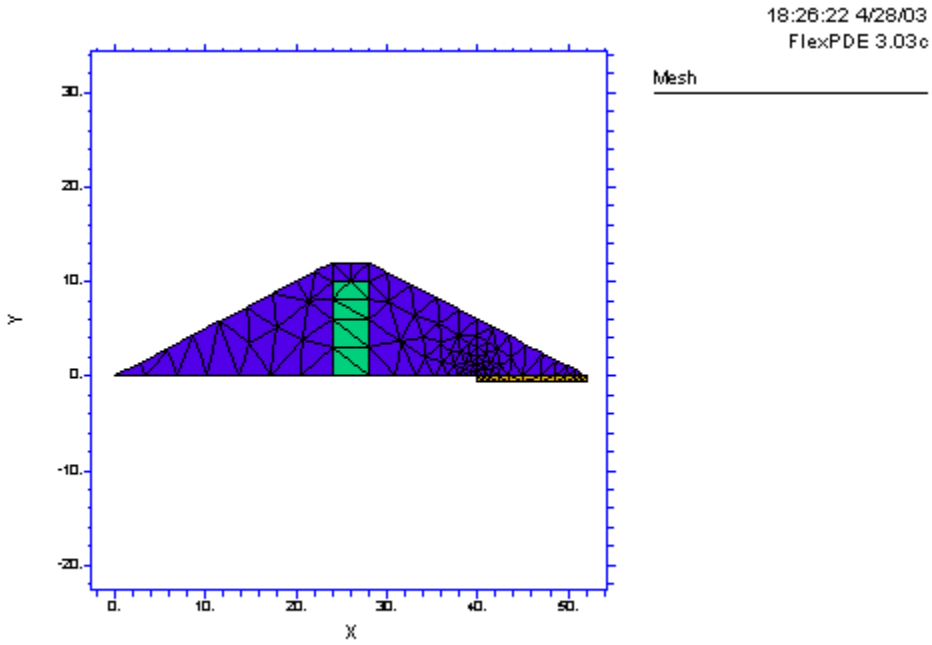
1.8 Analyze

Since no errors were encountered during validation, the next step is to analyze the problem. Click the Analyze button located beside the Validate check mark. 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.

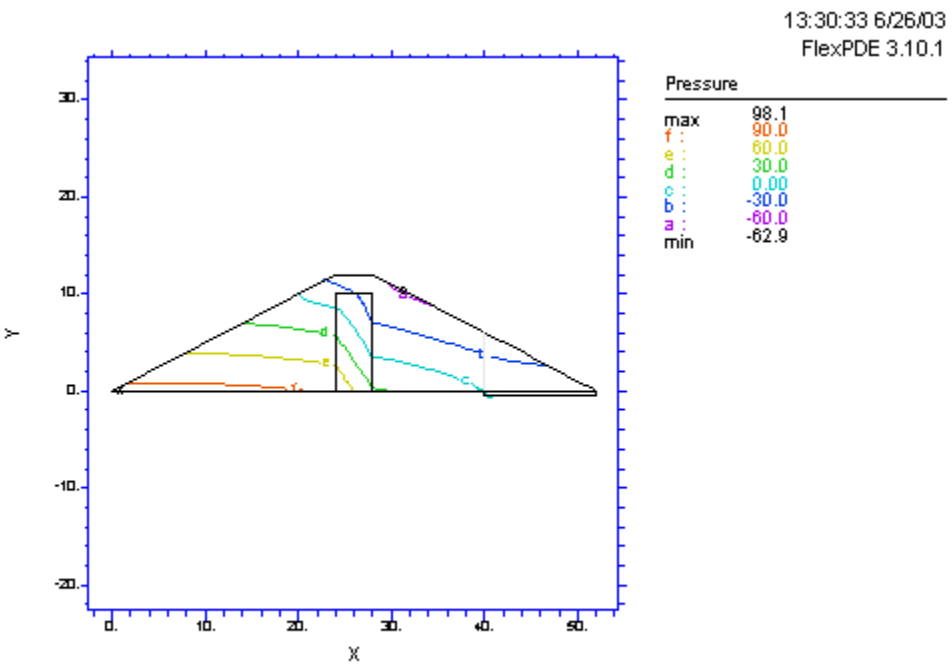
1.9.1 Solution Mesh



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.

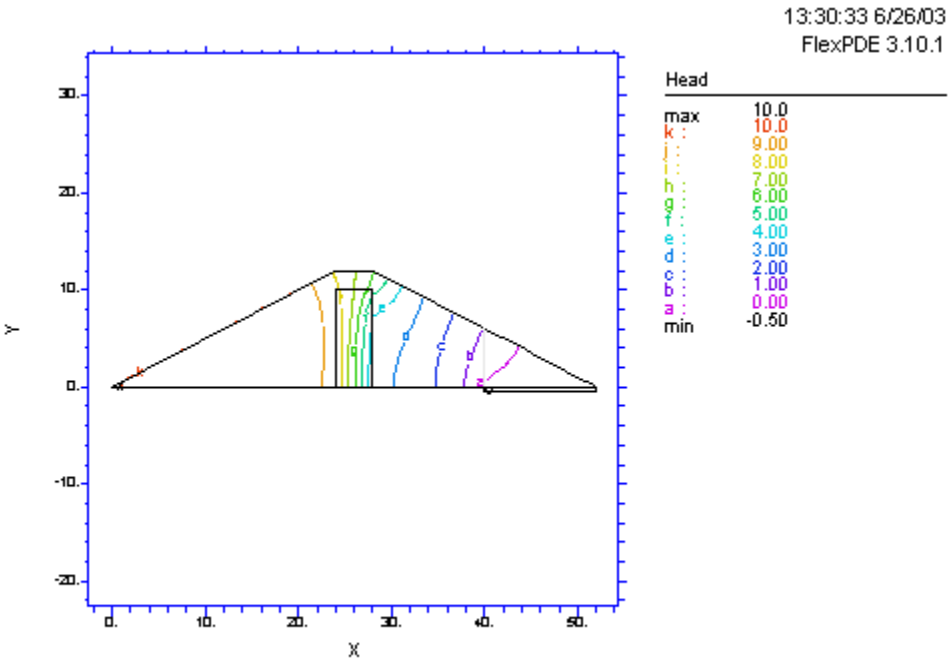
1.9.2 Pressure Contours



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.

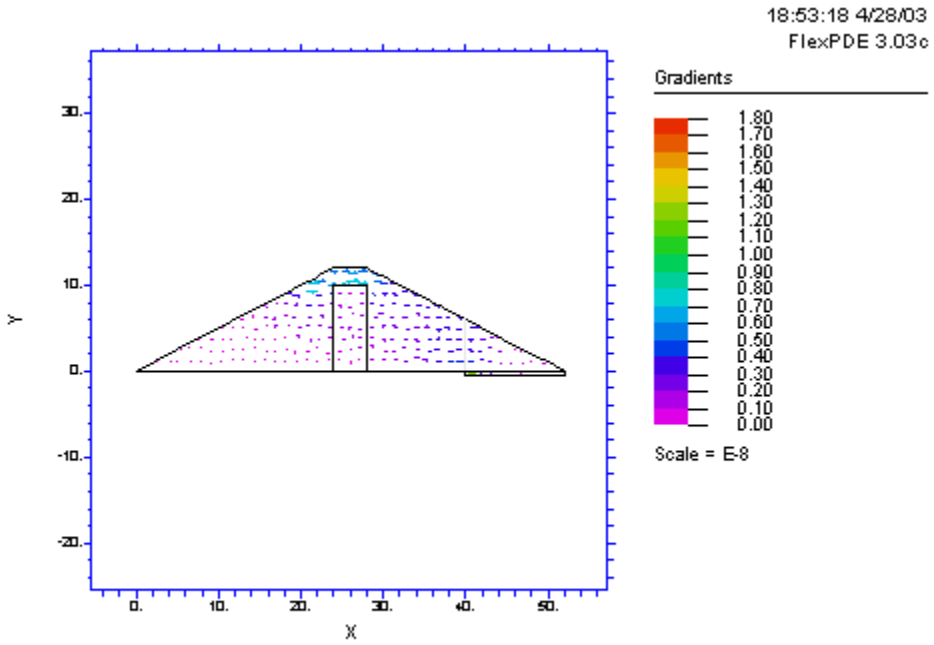
1.9.3 Head Contours



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



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 (m^3/s)= 2.372461e-8
Y Component of Flow in (m^3/s)= 0.00
Normal Flow in (m^3/s)= 2.372461e-8

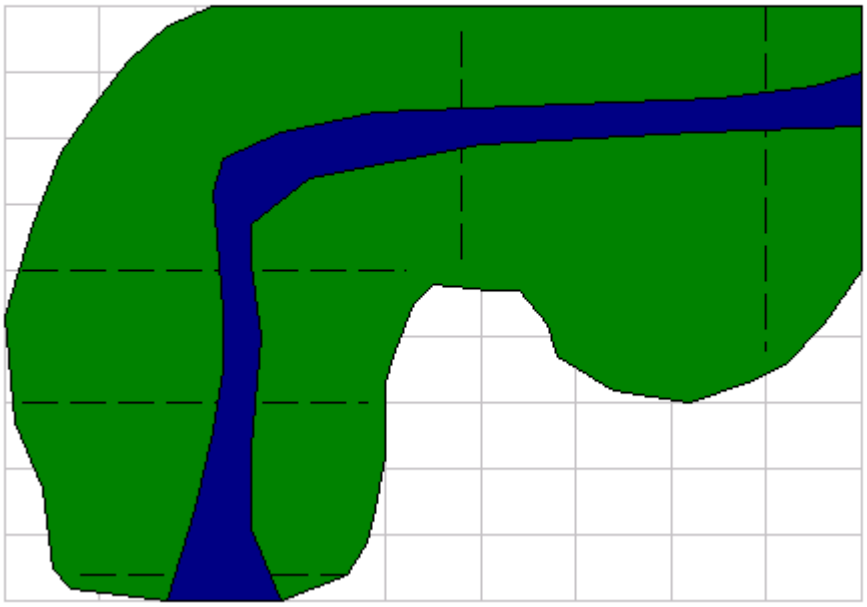
The Flux through the problem is displayed in the form of report showing a breakdown of the X, Y, and Normal components of flow through the problem.

2.0 A Three Dimensional Example Problem

The following example will introduce you to the three dimensional model in SVFLUX. The model considers the flow and pressure conditions in a river valley. The problem is modeled using 2 regions, 3 surfaces, and 2 soils. The problem data and soil properties are provided below.

ProjectID: Tutorial ProblemID: Outflow

OutFlow with Pinch Out



Sand: k_{sat} = 1e-02 kPa

The soil-water characteristic curve is fit by the Fredlund and Xing method.

SWCC curve laboratory data:

Suction (kPa) Water Content

0.01	0.35
0.1	0.33
1	0.3
100	0.22
1000	0.1

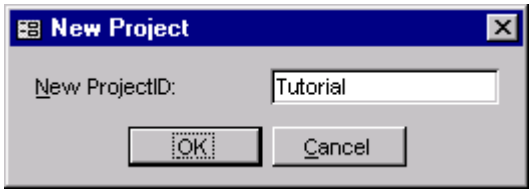
Till: k_{sat} = 1e-02 kPa

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.



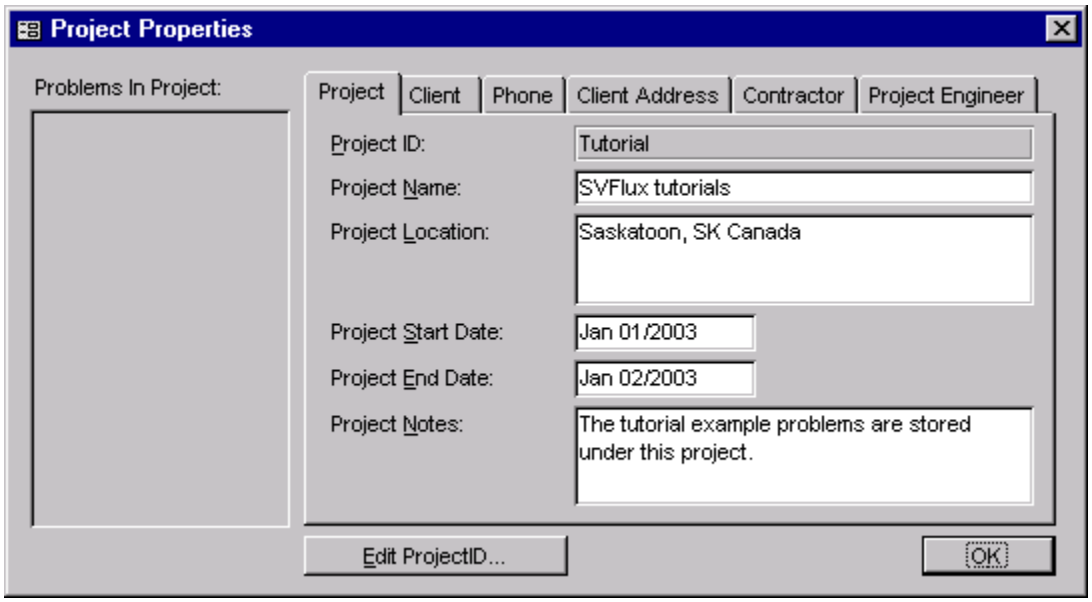
4. Type "Tutorial" as the new ProjectID and press OK.

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.



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.



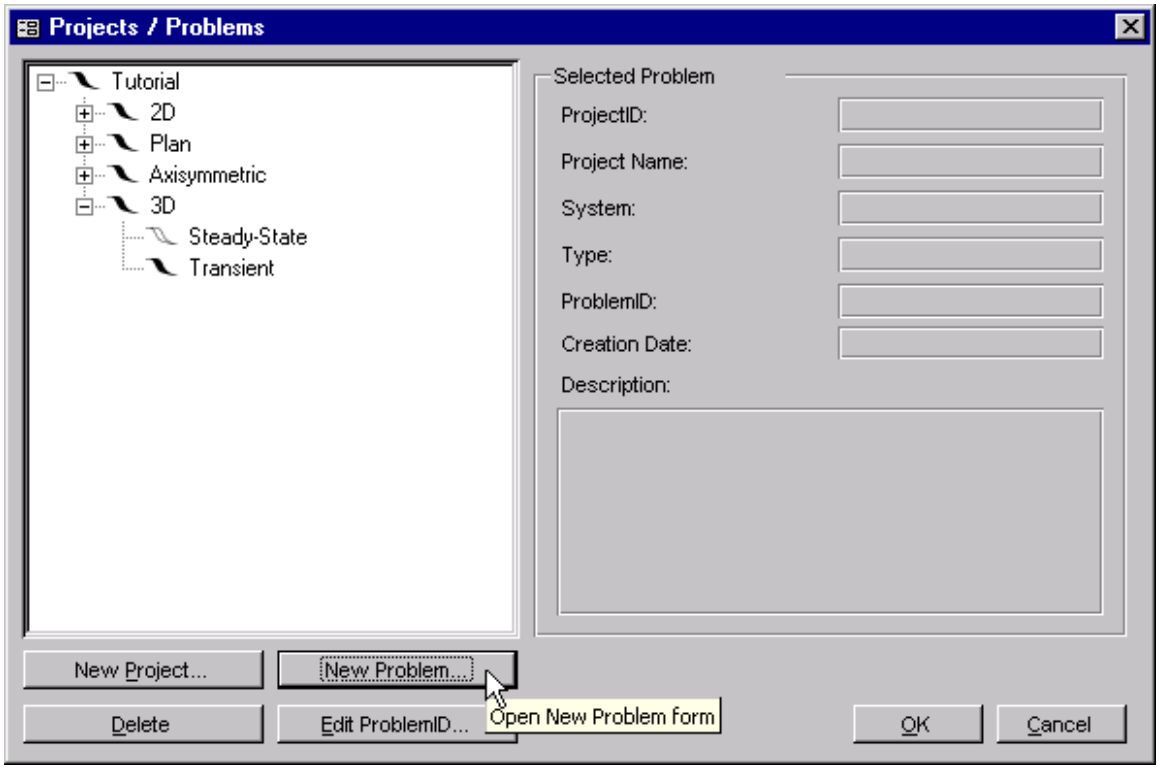
It should be noted that once the project is defined it will be identified by the ProjectID throughout the rest of the program. Also, SVFLUX does not allow you to specify two projects with the same ProjectID.

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.

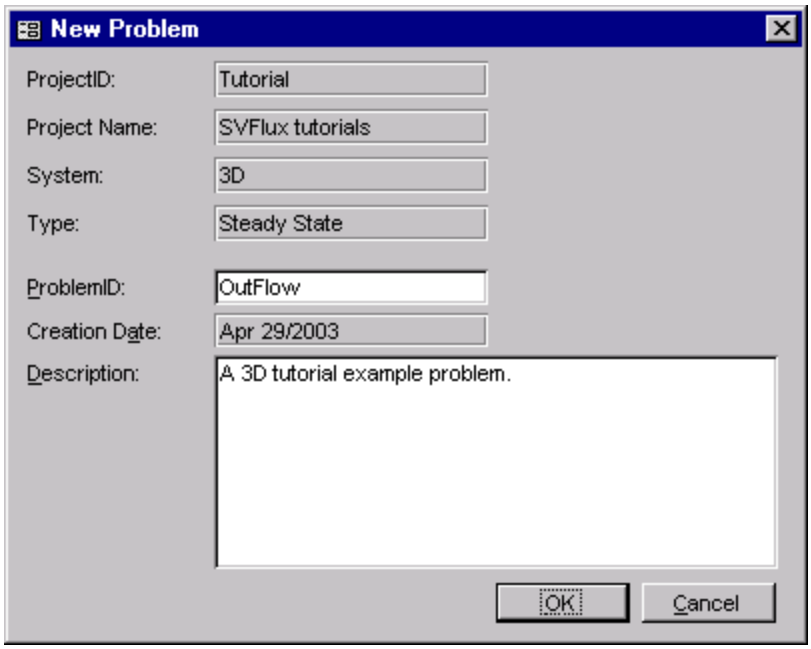
2.2 Adding a Problem

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:

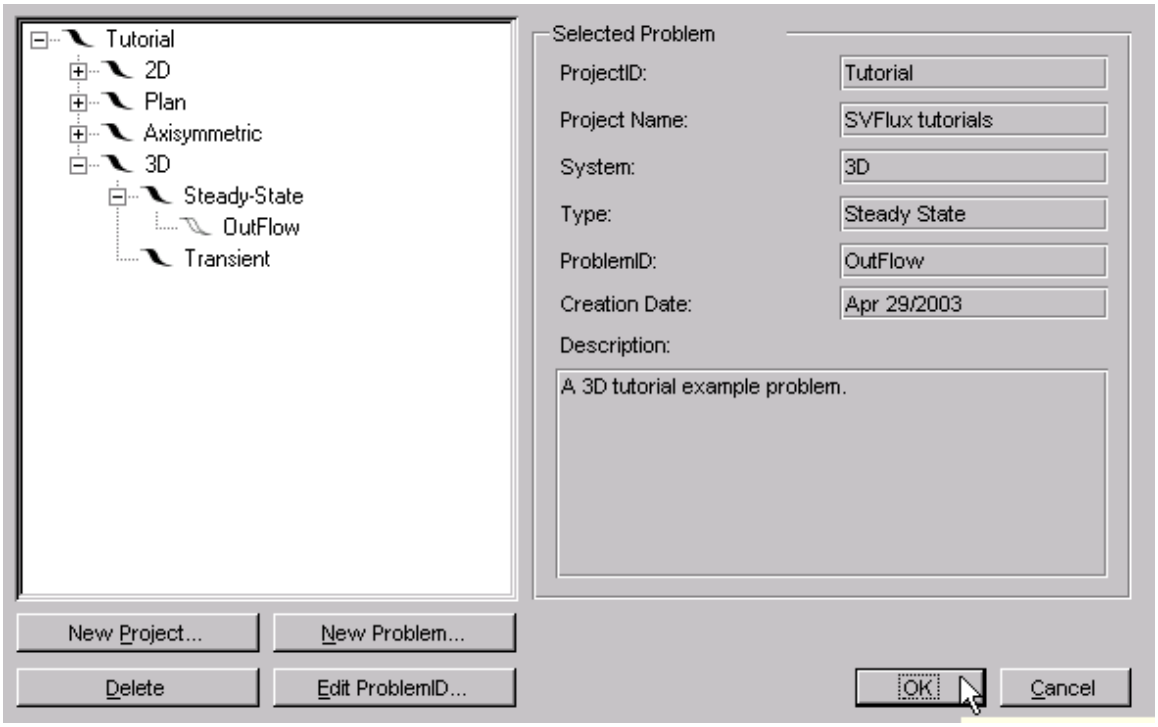
1. Click on the plus sign and expand the project **Tutorial**.
2. This example will be modeled in three-dimensions and is steady-state so we must expand **3D** by clicking the plus sign beside "3D"
3. Select **Steady-State**.
4. Click the **New Problem** button. The New Problem form will open.



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

2.3 Opening the Problem

To open the problem:



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

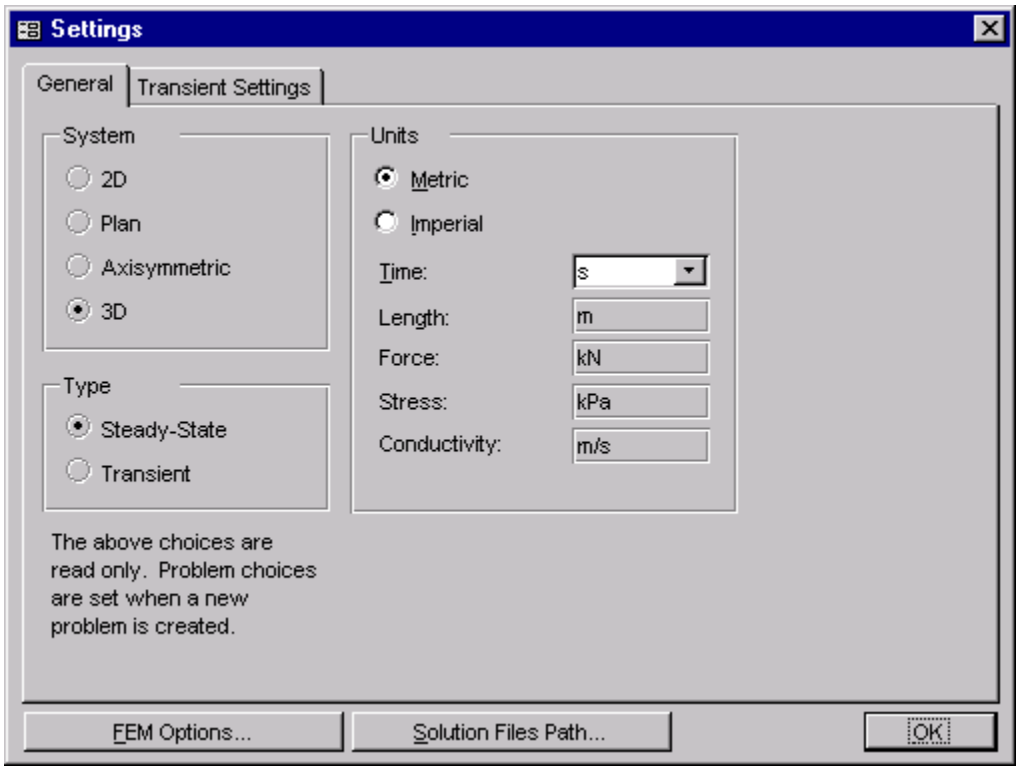
2. Select **OutFlow**.
3. The problem may opened by clicking the **OK** button or by double clicking on the ProblemID. The problem will now be opened in the workspace.

2.4 Defining the Problem

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

2.4.1 Specify Settings

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

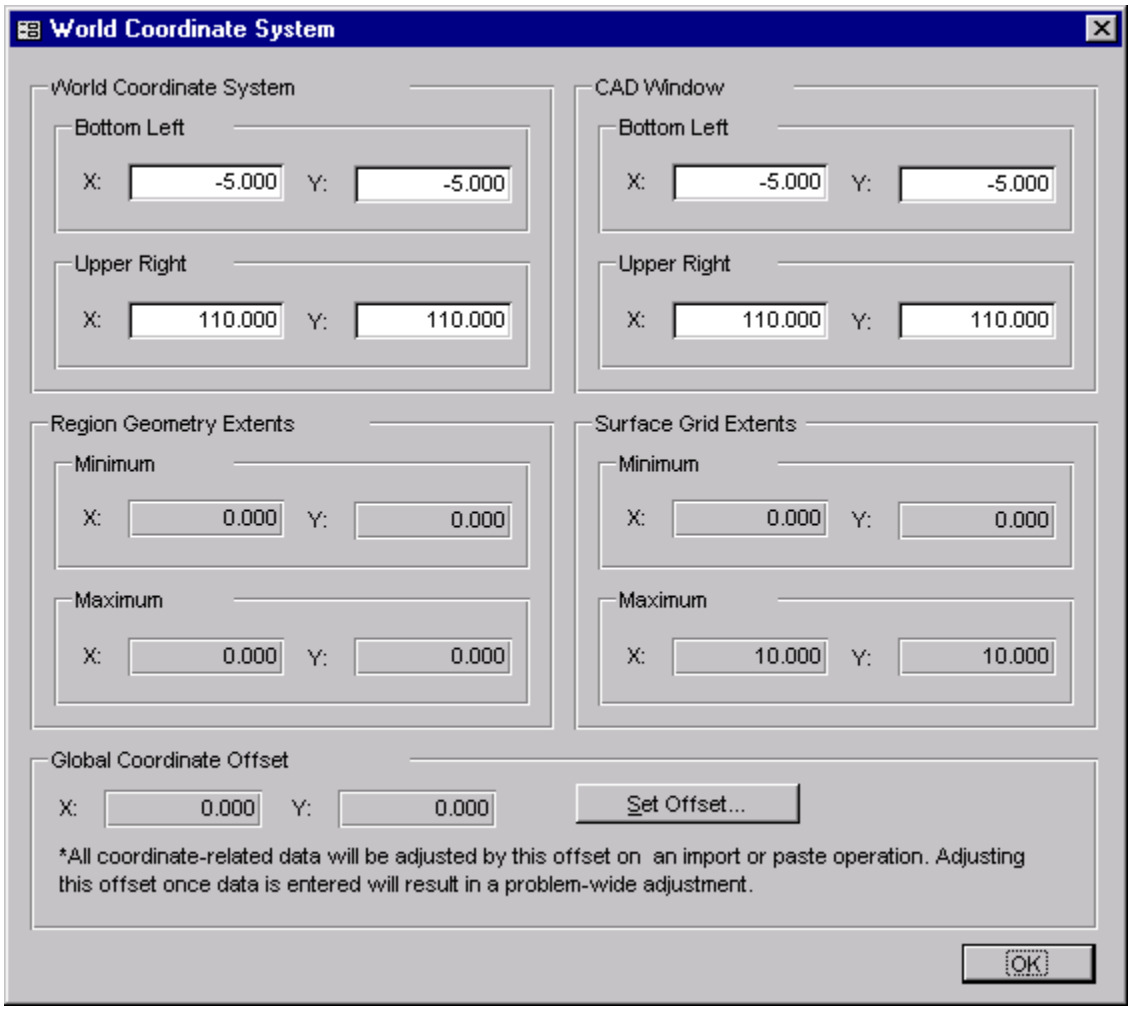


The Settings form will contain information about the current problem System, Type, Units, and Transient Settings. The hydraulic conductivity data for the soils contained in the problem are reported as m/s so the time units will remain s, for seconds.

2.4.2 Setting the Workspace

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

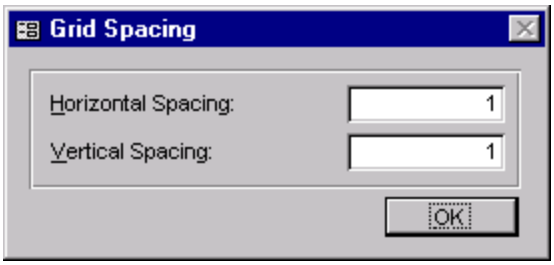
1. Access the World Coordinate System form by either of two ways. Click on the button:  or select **View > WCS**. The button is located in the view toolbar to the left of the drawing space.



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.

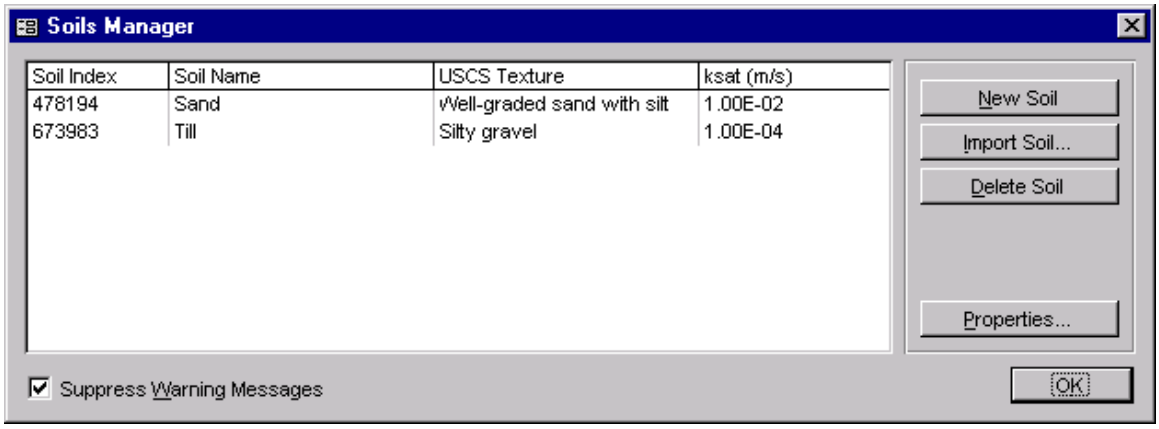
The workspace grid spacing needs to be set to aid in defining region shapes. The geometry data for this problem has coordinates of a precision of 1m. In order to effectively draw geometry with this precision using the mouse the grid spacing must be set to a maximum of 1.

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



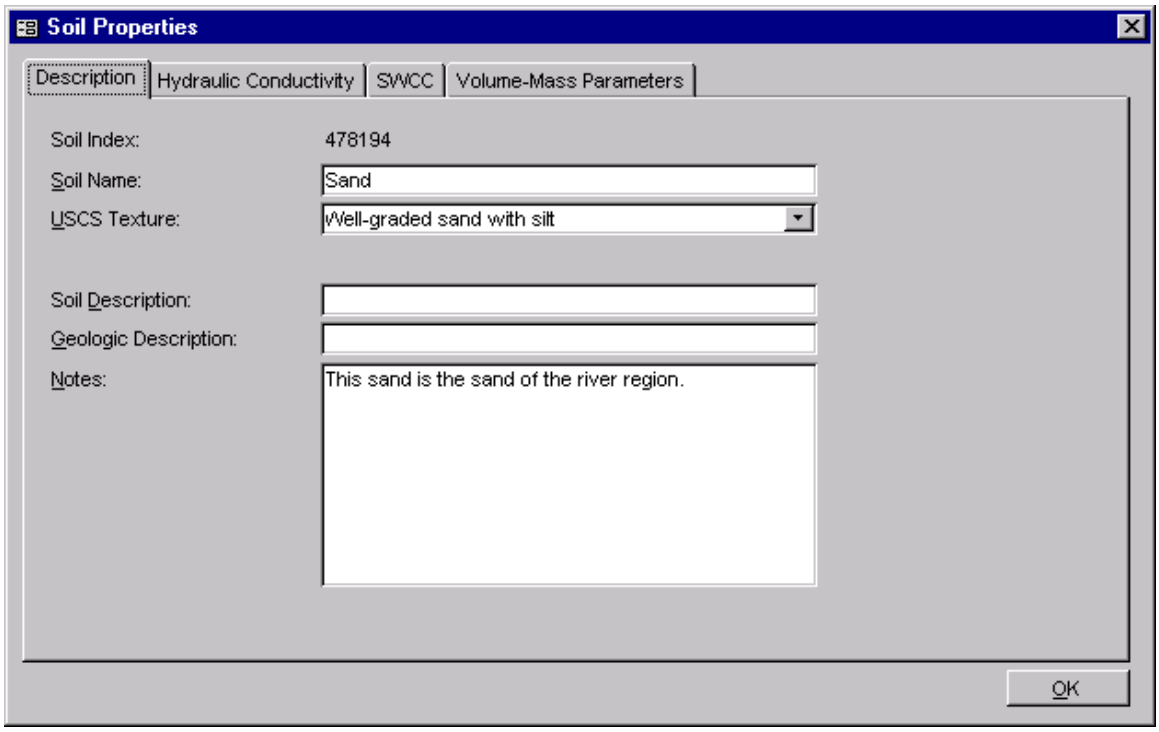
6. Enter **1** for both the horizontal and vertical spacing.
7. Click **OK** to close the form.

2.4.3 Define Material Properties

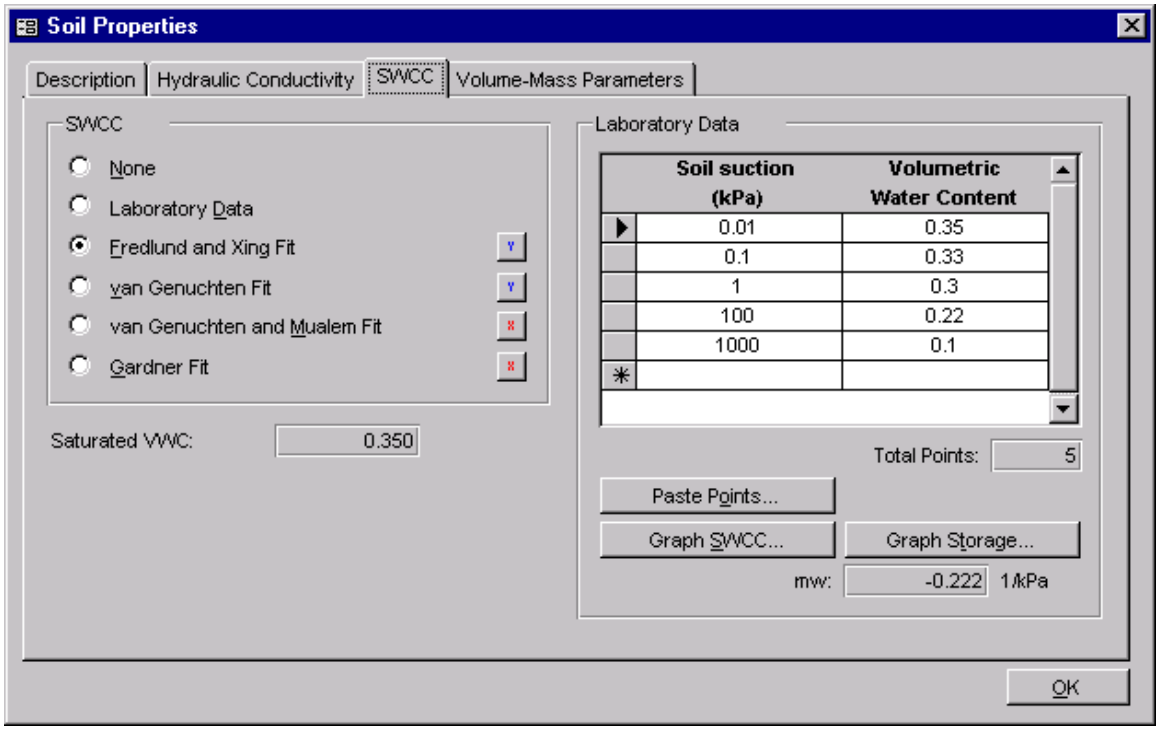


The next step in defining the problem is to enter the material properties for the 2 soils that will be used in the model. A till is defined for the main region and the river will consist of a sand.

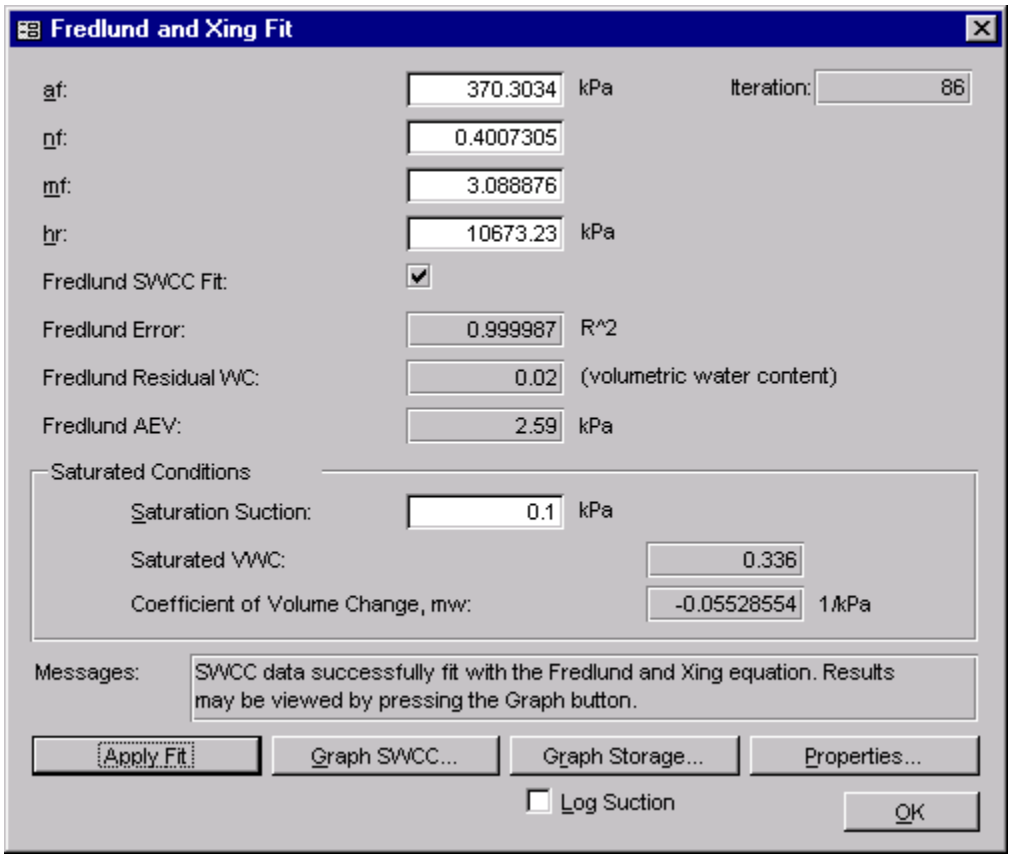
1. Open the Soils form by selecting **Model > Soils** from the menu or click the soils button,  in the Tools toolbar.
2. Click the **New Soil** button to create a soil in the database. A unique Soil Index is generated that is used to reference the soil in other SVFLUX forms.
3. Double-click on the new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



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-02 m/s.
7. Select **None** from the Unsaturated Hydraulic Conductivity option group.
8. Move to the **SWCC** tab.



9. Select **Fredlund and Xing Fit** from the SWCC option group.
10. Enter the laboratory data **points** as provided.
11. Press the button to the right of the Fredlund and Xing option to open the **Fredlund and Xing Fit** form.



12. Press Apply Fit to execute the fit. The fit data will display in the form.



To view the fit curve and the laboratory data press the Graph SWCC or Graph Storage buttons.

13. Close the Fredlund and Xing Fit form from the OK button. The fit button for the Fredlund and Xing fit on the Soil Properties form will display a blue Y instead of a red X.
14. Click the **New Soil** button to create a new soil. Double-click on the new soil to open it in the **Soil Properties** form.
15. Enter **Till** as the Soil Name and select **Silty Gravel** as the USCS Texture on the **Description** tab.
16. Move to the **Hydraulic Conductivity** tab.
17. Enter the k_{sat} value of 1.000E-04 m/s.
18. Press **OK** on the Soils Manager form to close both forms.

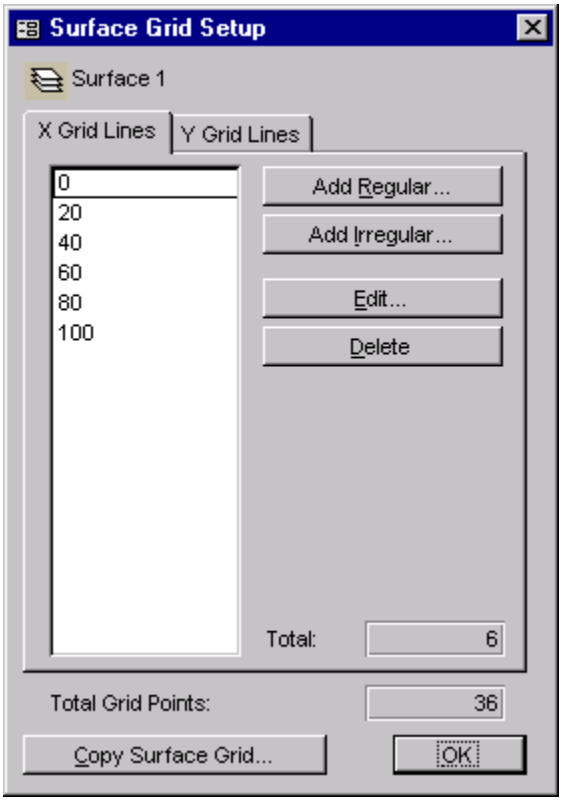
2.4.4 Define 3D Surfaces

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 0 for **Start**, 20 for **Increment Value**, and 100 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 and repeat steps 3 – 7 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.

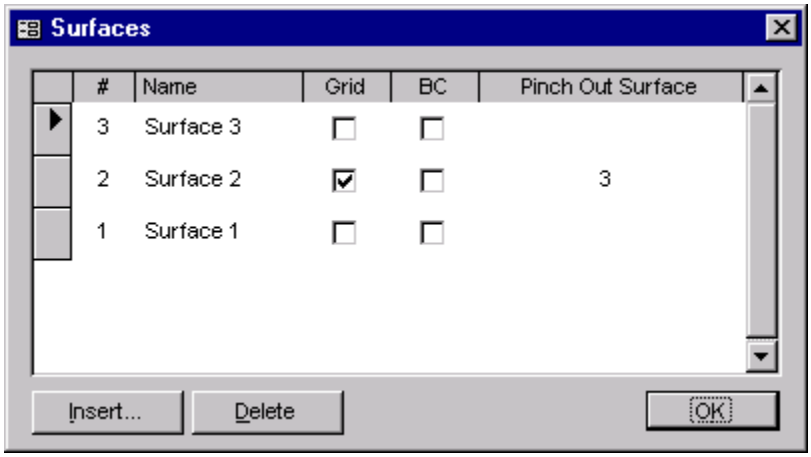
- **Define Surface 2**

This surface is already present. Follow the steps for defining Surface 1 except provide 10 for the Increment Value and 90 for the End. Also set all the Surface 2 elevations to 6m.

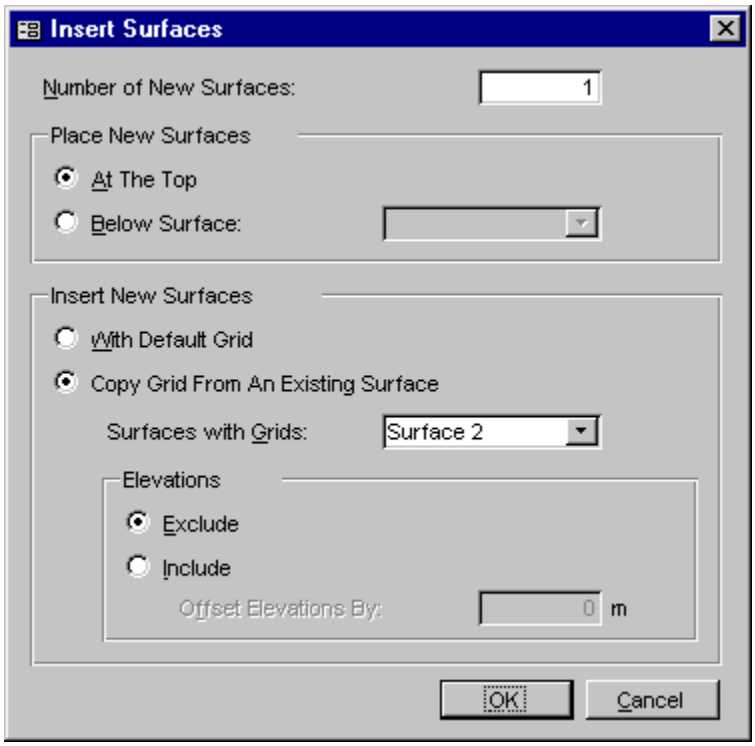
- **Define Surface 3**

  Follow these steps to add the third surface to the problem. Since Surface 2 and Surface 3 have the same grid lines the Surface 2 grid will be copied during insertion of the new surface:

  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:

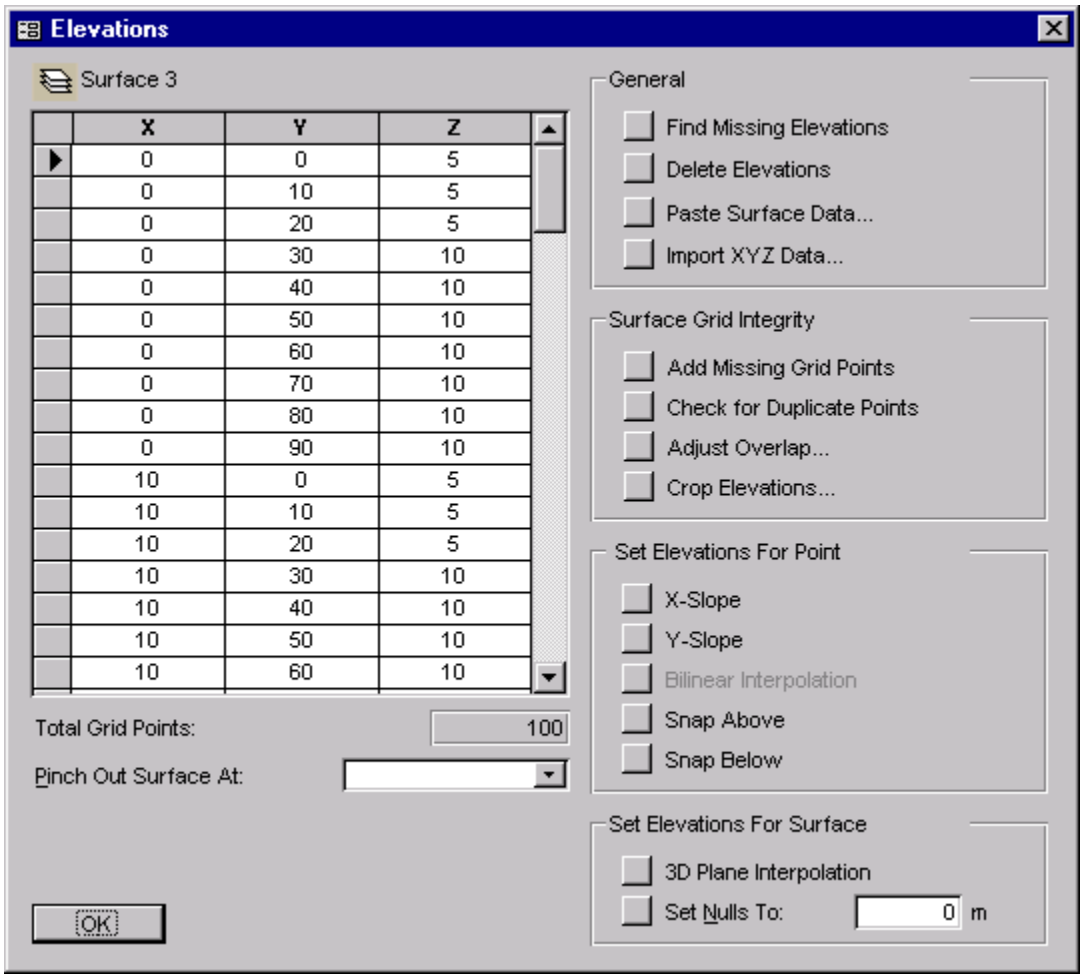


  3. Enter 1 as the **Number of New Surfaces**.
  4. Select to place the new surface **At The Top**.

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:

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



3. Enter the **elevations** as provided in Appendix A.
4. Press **OK** to close the form.



If you have the Tutorial example data .XLS file then use the Paste Surface Data option to quickly add the data for Surface 3.

- **A Pinch Out Surface**

Since some of the elevations defined for Surface 3 are lower than the constant Z = 6m defined for Surface 2 a pinch out surface specification will be used as surfaces cannot overlap.

1. Select **Surface 2** in the Surface Selector.
2. Click the **Elevations** button,  to open the Elevations form.
3. Select **Surface 3** from the Pinch Out Surface At drop-down.
4. Close the form.

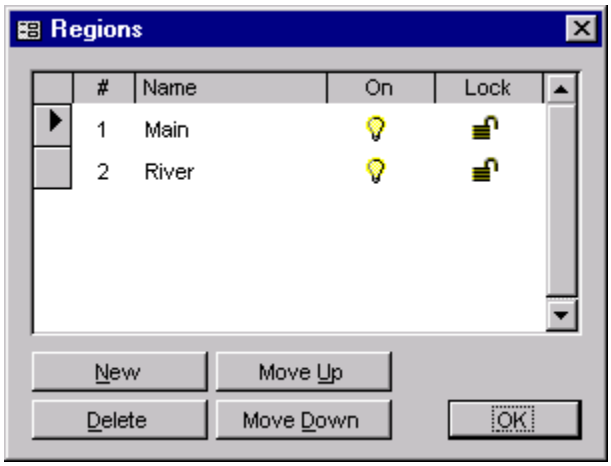
Now during analysis of the problem, everywhere that Surface 2 and Surface 3 overlap Surface 2 will assume the elevation of Surface 3.

2.4.5 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 are defined on any given region.

This problem will be divided into 2 regions, which are named Main and River. 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 **Main**. 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.

2.4.6 Defining Region Geometry Shapes

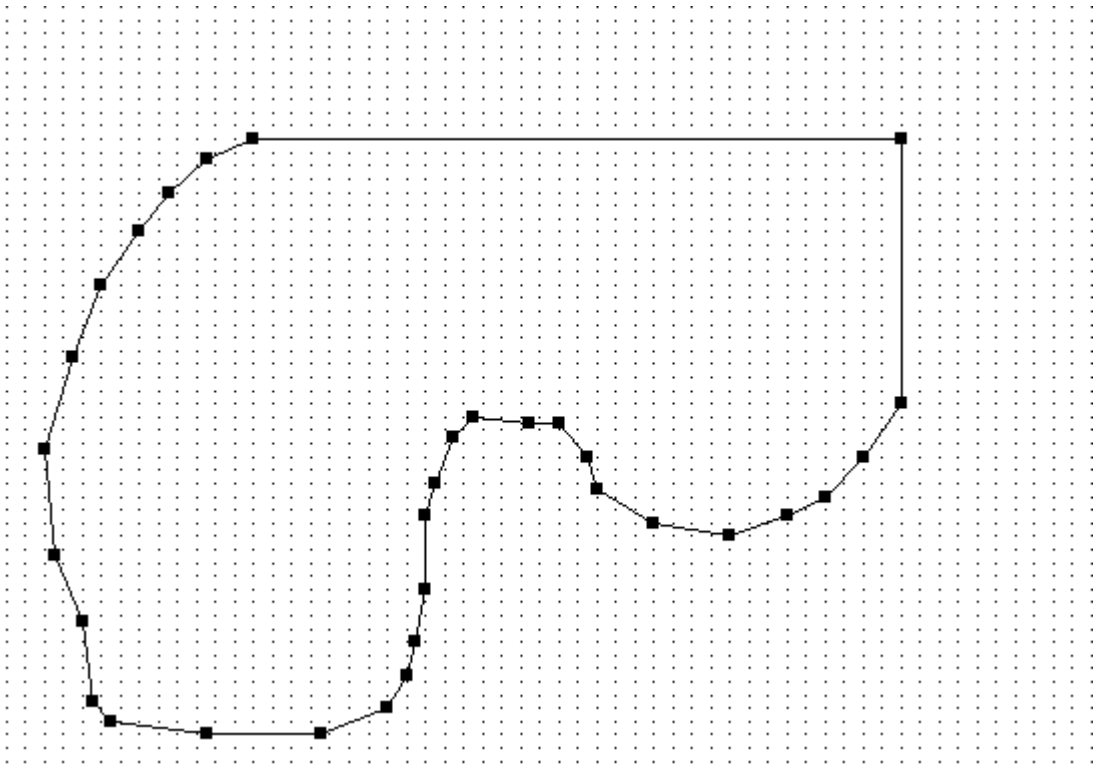
The shapes that define each region will now be created. Note that when drawing geometry shapes the **region that is current in the region selector is the region the geometry will be added to**. The Region Selector is at the top of the workspace. Refer to Appendix A for the geometry points for each region.

- **Define the Main region**

1. Ensure the "Main" 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 (0,43) 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 (1,27). Right click to snap the cursor to the exact point and then left click on the point. A line is now drawn from (0,43) to (1,27).
8. Repeat this process for the remaining points.
9. For the last point (3,57), right click to snap the cursor to the point. Double click on the point to finish the shape. A line is now drawn from (6,68) to (3,57) and the shape is automatically finished by SVFLUX by drawing a line from (3,57) back to the start point, (0,43).

If the Main geometry 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.

- **Define the River**

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

1. Ensure that “River” 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 0,43 and press the Enter key on the keyboard.
5. Enter the remaining coordinates.
6. Type f and press Enter to complete the region shape.

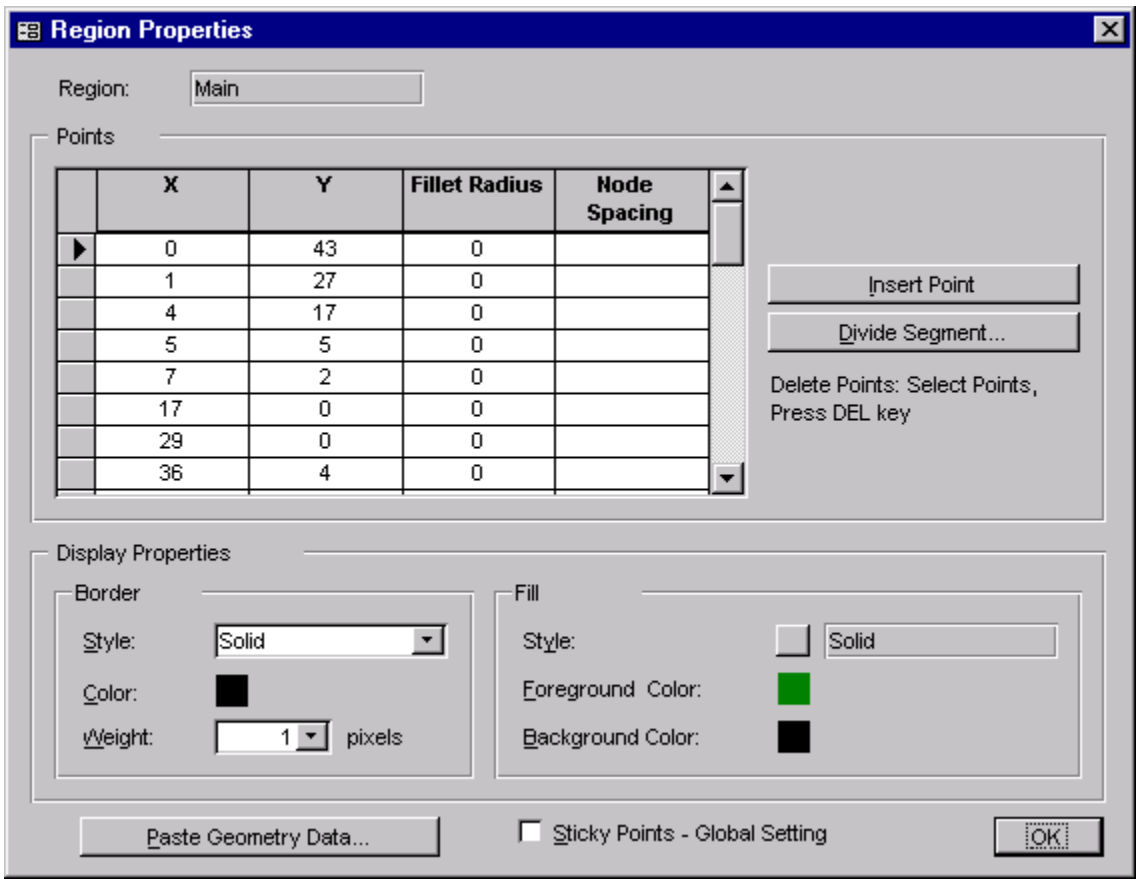


At times it may be tricky to snap to a grid point that is near a line defined for a region. Turn the object snap off by clicking on "OSNAP" in the status bar to alleviate this problem.

- **Formatting a Shape**

To set the Main region shape to a solid green:

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



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**.

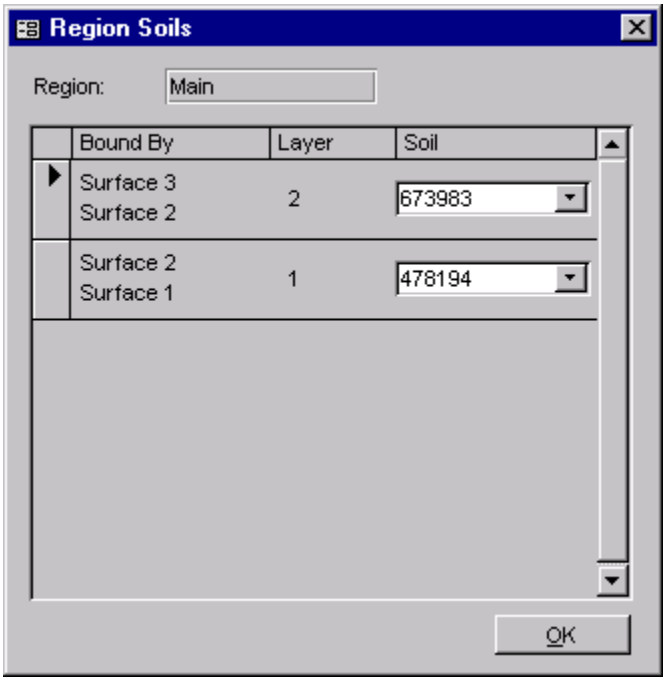
Repeat these steps to give the River a solid blue fill.

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

2.4.7 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 "**Main**" in the Region Selector.
2. Press the **Region Soils** button,  at the top of the workspace to open the Region Soils form.



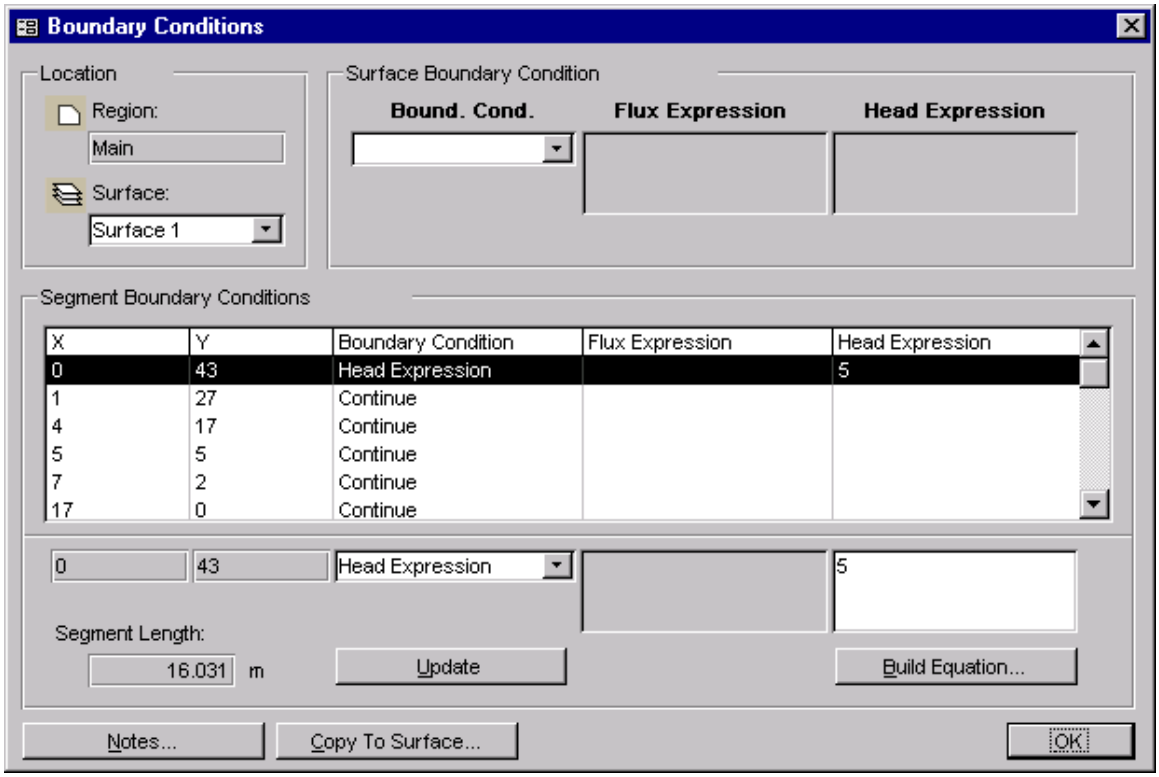
3. Select the **Till** soil from the drop-down for **Layer 2**. (Note that the Soil Indexes will be different than depicted here)
4. Select the **Sand** soil from the drop-down for **Layer 1**.
5. Close the form using the **OK** button.
6. Select "**River**" in the Region Selector.
7. Press the **Region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

2.4.8 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 Main

region segments with the Zero Flux condition being applied to the remainder. The River will be set to a Zero Flux condition by default and will not need to be altered. The steps for specifying the boundary conditions are thus:

1. Select the "Main" 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.



3. Select the **point** (0,43) 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 5**.
6. Click the **Update** button to save the boundary condition to the list.
7. Select the **point** (38,9) 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** (90,50) from the list.
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 10**.
13. Click the **Update** button to save the boundary condition to the list.
14. Select the **point** (90,90) 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.
17. Select the **point** (22,90) from the list.
18. From the **Boundary Condition** drop down select a **Head Expression** boundary condition. This will cause the Head Expression box to be enabled.
19. In the Head Expression box enter a **head of 5**.

20. Click the **Update** button to save the boundary condition to the list.

Main Region Boundary Condition Summary

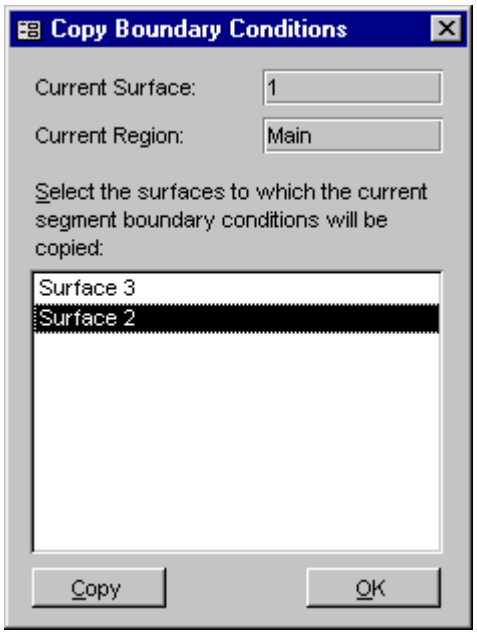
X	Y	Boundary Condition	Expression
0	43	Head Expression	5
38	9	Zero Flux	
90	50	Head Expression	5
90	90	Zero Flux	
22	90	Head Expression	5



The Head Expression boundary condition for the point (0,43) 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 (0,43) to (36,4 are all given a Head Expression boundary condition.

The boundary conditions for the main 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.

2.5 Adding Features

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

Feature	X	Y

1	78	90
	78	33
2	0	43
	43	45
3	2	30
	38	30
4	5	5
	36	4
5	51	90
	51	47

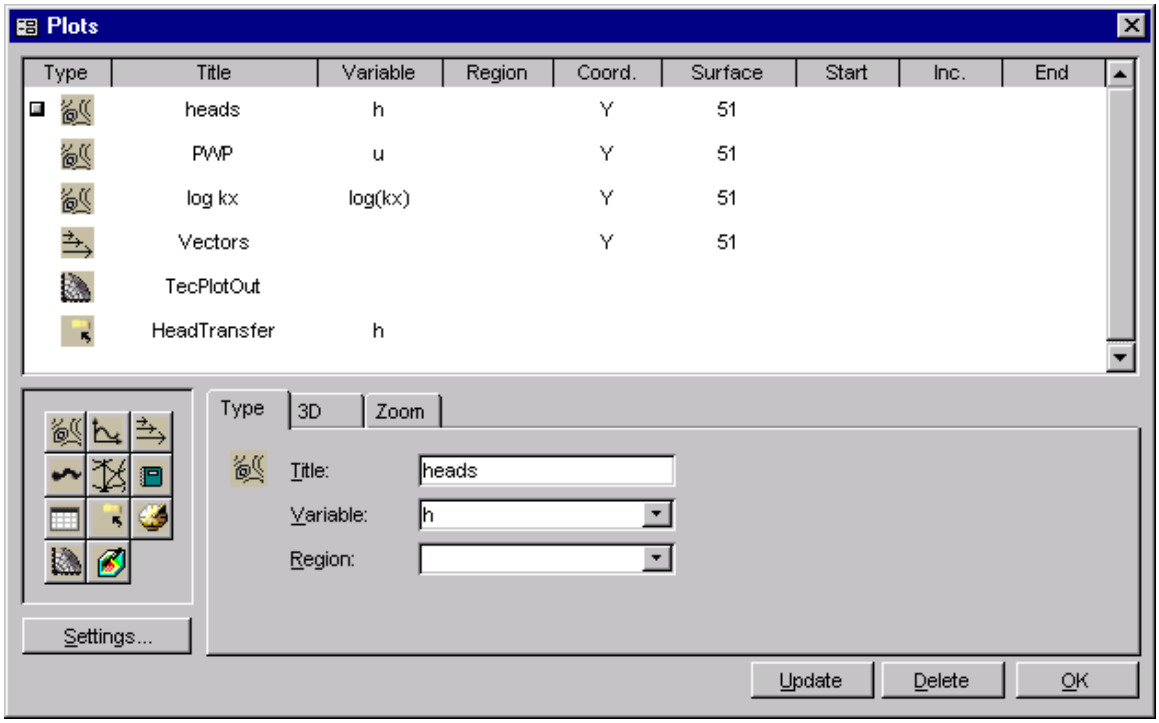
To add the features:

1. Click on the **Feature** button, .
2. With the mouse **click** on the point (78,90). (Use the right button on the mouse to snap to grid points before left clicking to add the point. The SNAP and GRID options must be on in the workspace.)
3. **Double-click** on the point (78,33) to finish the Feature 1.
4. **Repeat** steps 1 –3 to add the other 4 features.

2.6 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, gradient vectors, and a plot to transfer the results to Tecplot. A plot of the solution mesh is generated by default.

1. Open the Plots form by selecting **Model > Plots** 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.
3. Enter the title heads.
4. Select h as the variable to plot from the drop-down.
5. Move to the 3D tab.
6. Select Plane 3D option.
7. Select Y from the Coordinate Direction drop-down.
8. Enter 58 in the Coordinate field. This will generate a 2D slice at Y = 51m on which the heads will be plotted.
9. Click **Update** to add the plot to the list.
10. Repeat these steps 2 – 5 to create the plots shown above. Note that the Tecplot plot does not require entry of a variable and the Tecplot and Transfer plots do not require a 3D specification. The Tecplot plot will generate a file containing all the mesh and variable data calculated by the solver for import into the Tecplot visualization software.
11. Click **OK** to return to the workspace.

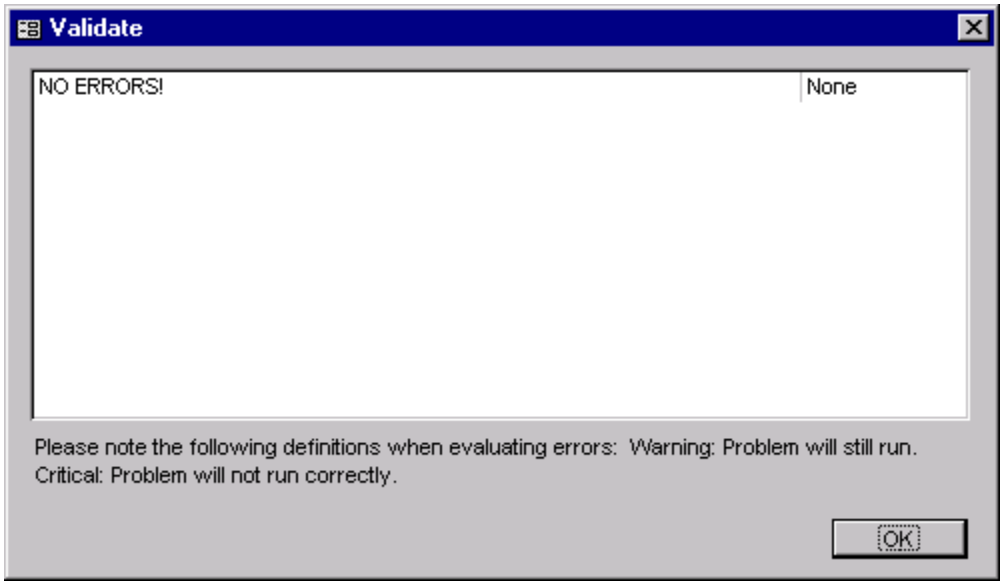


Feature 5 has been drawn at Y = 51m and it helps improve the plot resolution for plots on the Y = 51m plane.

2.7 Verify the Problem

The validation form will provide a summary of checks and errors that are present in the model before the problem is analyzed. To validate the problem, click the check mark as shown below.





In the case shown here the problem is showing that there are no errors and it is ready to be analyzed.

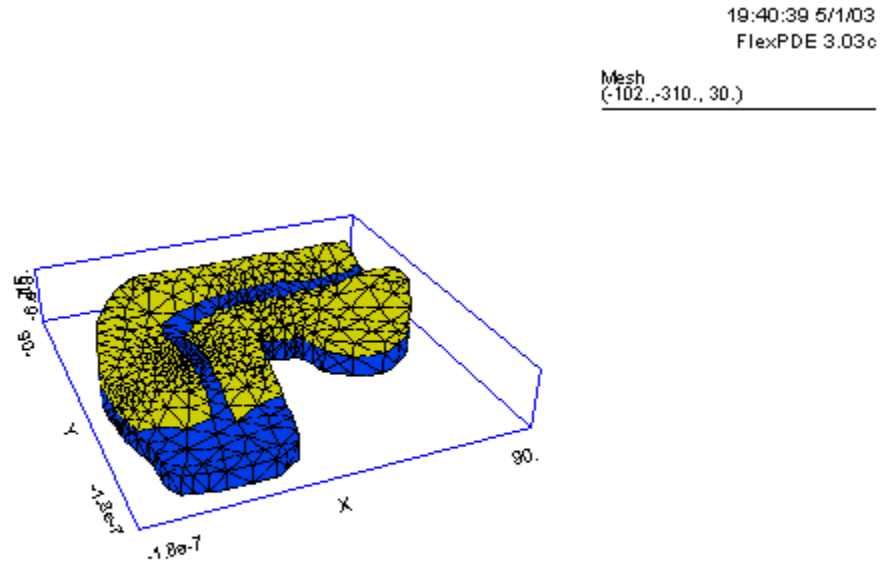
2.8 Analyze

Since no errors were encountered during validation, the next step is to analyze the problem. Click the Analyze button located beside the Validate check mark. This action will write the descriptor file and open the SVFLUX solver. The solver will automatically begin solving the problem.

2.9 Results

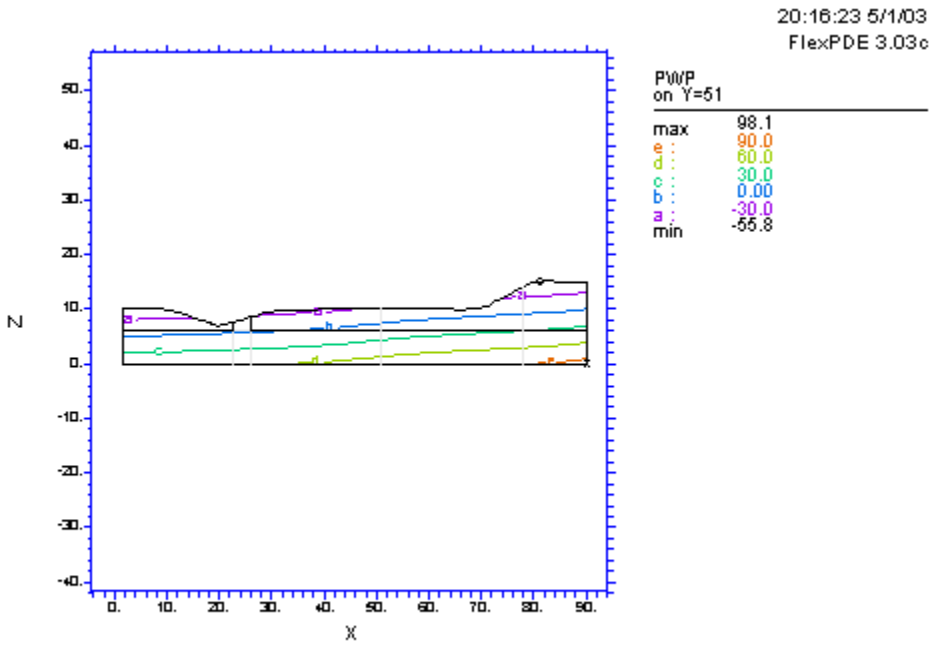
After the problem has finished solving, the results will be displayed in the form of thumbnail plots within the SVFLUX solver. Right-click the mouse and select Maximize to enlarge any of the thumbnail plots. This section will give a brief analysis for each plot that was generated.

2.9.1 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.

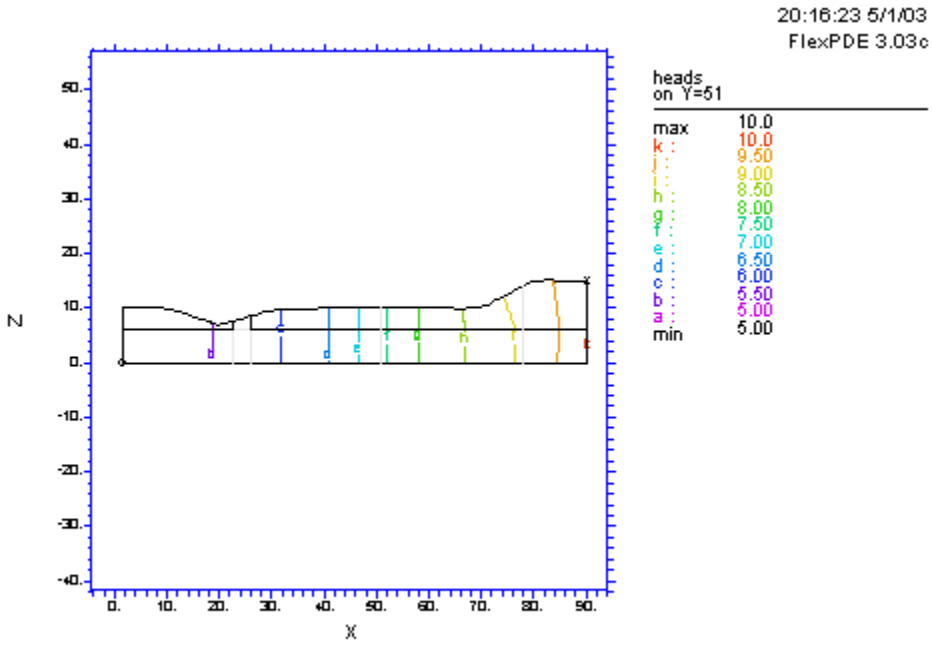
2.9.2 Pressure Contours



Examples_OutFlow: Grid#1 p2 Nodes=14040 Cells=8470 RMS Err= 3.5e-5
Stage 2 Integral= 17602.90

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 right to the left in the plot.

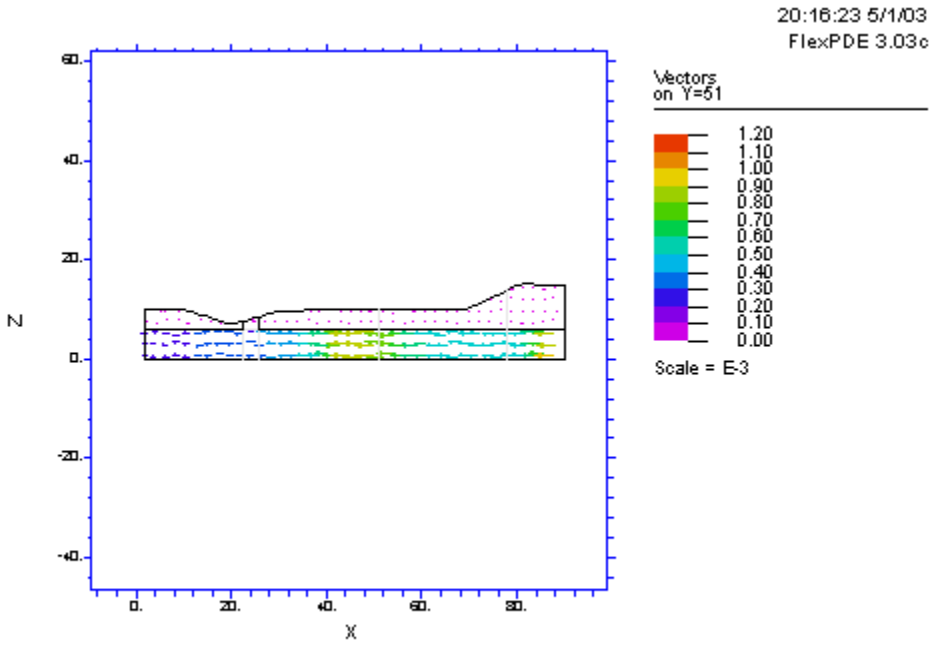
2.9.3 Head Contours



Examples_OutFlow: Grid#1 p2 Nodes=14040 Cells=8470 RMS Err= 3.5e-5
Stage 2 Integral= 6813.580

As expected, the head is 10m at the right of the plot where this condition was specified as a boundary condition. The head decreases to 5m on the left edge of the plot where the boundary condition was set to 5m.

2.9.4 Flow Vectors



Flow Vectors show both the direction and the magnitude of the flow at specific points in the problem. The higher conductivity of the sand causes the majority of the flow through Layer 1 and Layer 2 – River Region where the sand was specified as the soil resulting in increased gradients in this area. Vectors illustrate that flow is from right to left in this view.

3.0 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

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

Tecplot User's Manual, version 8.0, 1999. Amtec Engineering Inc. Bellevue, WA 98009-3633

Appendix A - Data

4.0 3D Example Problem Data

Surface Data for Tutorial – OutFlow:

Surface 1

X	Y	Z
0	0	0
0	20	0
0	40	0
0	60	0
0	80	0
0	100	0
20	0	0
20	20	0
20	40	0
20	60	0
20	80	0
20	100	0
40	0	0
40	20	0
40	40	0
40	60	0
40	80	0
40	100	0
60	0	0
60	20	0
60	40	0
60	60	0
60	80	0
60	100	0
80	0	0
80	20	0
80	40	0
80	60	0
80	80	0
80	100	0
100	0	0
100	20	0
100	40	0
100	60	0
100	80	0
100	100	0

Surface 2

X	Y	Z
0	0	6
0	10	6
0	20	6
0	30	6
0	40	6
0	50	6
0	60	6
0	70	6
0	80	6
0	90	6
10	0	6
10	10	6
10	20	6
10	30	6
10	40	6
10	50	6
10	60	6
10	70	6
10	80	6
10	90	6
20	0	6
20	10	6
20	20	6
20	30	6
20	40	6
20	50	6
20	60	6
20	70	6
20	80	6
20	90	6
30	0	6
30	10	6
30	20	6
30	30	6
30	40	6
30	50	6
30	60	6
30	70	6
30	80	6
30	90	6
40	0	6

Surface 3

X	Y	Z
0	0	5
0	10	5
0	20	5
0	30	10
0	40	10
0	50	10
0	60	10
0	70	10
0	80	10
0	90	10
10	0	5
10	10	5
10	20	5
10	30	10
10	40	10
10	50	10
10	60	10
10	70	10
10	80	10
10	90	10
20	0	5
20	10	5
20	20	5
20	30	7
20	40	7
20	50	7
20	60	6
20	70	6
20	80	8
20	90	10
30	0	5
30	10	5
30	20	5
30	30	10
30	40	10
30	50	10
30	60	6
30	70	6
30	80	10
30	90	10
40	0	5

40	10	6	40	10	5
40	20	6	40	20	5
40	30	6	40	30	10
40	40	6	40	40	10
40	50	6	40	50	10
40	60	6	40	60	10
40	70	6	40	70	6
40	80	6	40	80	10
40	90	6	40	90	10
50	0	6	50	0	10
50	10	6	50	10	10
50	20	6	50	20	10
50	30	6	50	30	10
50	40	6	50	40	10
50	50	6	50	50	10
50	60	6	50	60	10
50	70	6	50	70	6
50	80	6	50	80	10
50	90	6	50	90	10
60	0	6	60	0	10
60	10	6	60	10	10
60	20	6	60	20	10
60	30	6	60	30	10
60	40	6	60	40	10
60	50	6	60	50	10
60	60	6	60	60	10
60	70	6	60	70	6
60	80	6	60	80	10
60	90	6	60	90	10
70	0	6	70	0	10
70	10	6	70	10	10
70	20	6	70	20	10
70	30	6	70	30	10
70	40	6	70	40	10
70	50	6	70	50	10
70	60	6	70	60	10
70	70	6	70	70	6
70	80	6	70	80	7
70	90	6	70	90	10
80	0	6	80	0	10
80	10	6	80	10	10
80	20	6	80	20	10
80	30	6	80	30	10
80	40	6	80	40	10
80	50	6	80	50	15
80	60	6	80	60	15
80	70	6	80	70	6

80	80	6	80	80	6
80	90	6	80	90	10
90	0	6	90	0	10
90	10	6	90	10	10
90	20	6	90	20	10
90	30	6	90	30	10
90	40	6	90	40	10
90	50	6	90	50	15
90	60	6	90	60	15
90	70	6	90	70	6
90	80	6	90	80	6
90	90	6	90	90	10

Region shape data for Tutorial - OutFlow:

Main Region

X	Y
0	43
1	27
4	17
5	5
7	2
17	0
29	0
36	4
38	9
39	14
40	22
40	33
41	38
43	45
45	48
51	47
54	47
57	42
58	37
64	32
72	30
78	33
82	36
86	42
90	50
90	90
22	90
17	87
13	82
10	76
6	68

River Region

X	Y
17	0
20	14
22	26
23	35
23	44
22	62
23	67
29	71
39	74
54	75
75	76
85	78
90	80
90	72
73	71
50	69
32	64
26	57
26	51
27	40
26	23
26	11
29	0
17	0

3	57
0	43

5.0 Index

Boundaries, 17, 18, 43
Boundary Conditions, 16, 42, 44
Color, 16, 41
Elevations, 36, 38, 39
Features, 44
Fill, 5, 6, 8, 9, 16, 27, 28, 41
Flux Section, 18, 20
Fredlund and Xing, 27, 35
Geometry, 13, 39
Grid, 10, 32, 36, 38
Head Expression, 17, 18, 43, 44
Hydraulic Conductivity, 12, 34, 35
ksat, 12, 27, 34, 35
Mesh, 21, 23, 48
Pinch Out, 38, 39
Plots, 20, 45
Problem, 5, 6, 7, 8, 9, 21, 27, 29, 30, 31, 46, 54
ProblemID, 5, 8, 9, 27, 30, 31
Problems, 5, 6, 7, 28, 29, 52
Project, 5, 6, 28
ProjectID, 5, 6, 27, 28
Projects, 5, 6, 7, 28, 29
Properties, 5, 6, 11, 14, 15, 16, 18, 20, 28, 33, 35, 40, 41
Region, 13, 14, 15, 16, 19, 39, 40, 41, 42, 44, 51, 56
Regions, 13, 39
Seepage, 52
Selector, 13, 36, 38, 39, 42
Shape, 14, 15, 40, 41
Shapes, 13, 39
Soil, 11, 13, 33, 35, 42, 52
Soils, 11, 33, 35, 42, 52
Solution, 23, 48, 52
Steady-State, 7, 9, 29, 30
Suction, 5, 27
Surface, 36, 37, 38, 39, 43, 44, 54
Surfaces, 35, 37
SWCC, 12, 27, 34, 35
Tecplot, 2, 20, 21, 45, 46, 52
Transient, 9, 31
View, 10, 32, 37
Workspace, 9, 31
Zero Flux, 17, 18, 43, 44