SVHEAT™

2D / 3D Geothermal Modeling Software

Tutorial Manual

ED-4B
Date: November 24, 2004

Written by:
Robert Thode, B.Sc.G.E.
Jason Stianson, B.Sc.C.E.

Edited by:
Murray Fredlund, Ph.D.

SoilVision Systems Ltd.
Saskatoon, Saskatchewan, Canada

Software License
The software described in this manual is furnished under a license agreement. The software may be used or copied only in accordance with the terms of the agreement.

Software Support
Support for the software is furnished under the terms of a support agreement.

Copyright

Disclaimer of Warranty
SoilVision Systems Ltd. reserves the right to make periodic modifications of this product without obligation to notify any person of such revision. SoilVision does not guarantee, warrant, or make any representation regarding the use of, or the results of, the programs in terms of correctness, accuracy, reliability, currentness, or otherwise; the user is expected to make the final evaluation in the context of his (her) own problems.

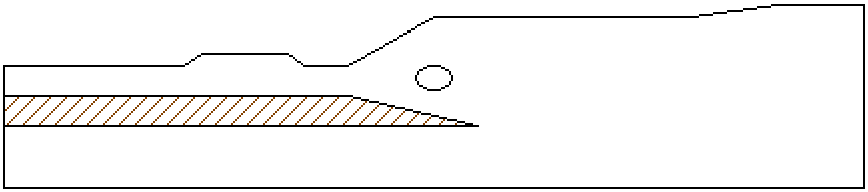
Trademarks
SVHEAT ™ is a trademark of SoilVision Systems Ltd.
SVFLUX ™ is a trademark of SoilVision Systems Ltd.
FlexPDE® is a registered trademark of PDE Solutions Inc.

1 A TWO DIMENSIONAL EXAMPLE PROBLEM

The following example will introduce some of the features included in SVHEAT and will set up a model of a simple buried pipeline. The purpose of this problem is to determine the effects of the heated pipeline on the frozen ground and the nearby roadway foundation. The problem is modeled using 2 regions and 2 soils. The problem data and soil properties are provided below.

ProjectID: Tutorial ProblemID: Tutorial2D

Heated Pipline near Roadway



Region Geometry

Slope Region

Shape 1 - polygon

X	Y
0	0
100	0
100	30
90	30
80	28
50	28
40	20
35	20
33	22
23	22
21	20
0	20
0	15
0	10

Shape 2 - circle

	X	Y
Center:	50	18

Radius:	2

Seam Region

X	Y
0	10
55	10
40	15
0	15

Soil Properties

Soil 1: Thermal Conductivity curve laboratory data:

Temperature (^{o}C)	Conductivity (J/hr-m- ^{o}C)
-10	1.58E+05
-1	1.57E+05
-0.1	1.56E+05
0	1.43E+05
0.1	1.28E+05
1	1.29E+05
10	1.30E+05

Soil 2: Thermal Conductivity curve laboratory data:

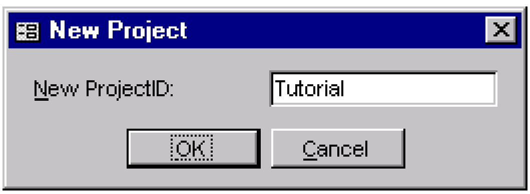
Temperature (^{o}C)	Conductivity (J/hr-m- ^{o}C)
-10	2.00E+05
-1	1.90E+05
-0.1	1.80E+05
0	1.50E+05
0.1	1.30E+05
1	1.25E+05
10	1.20E+05

1.1 ADDING A PROJECT

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

In order to add this project follow these steps:

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



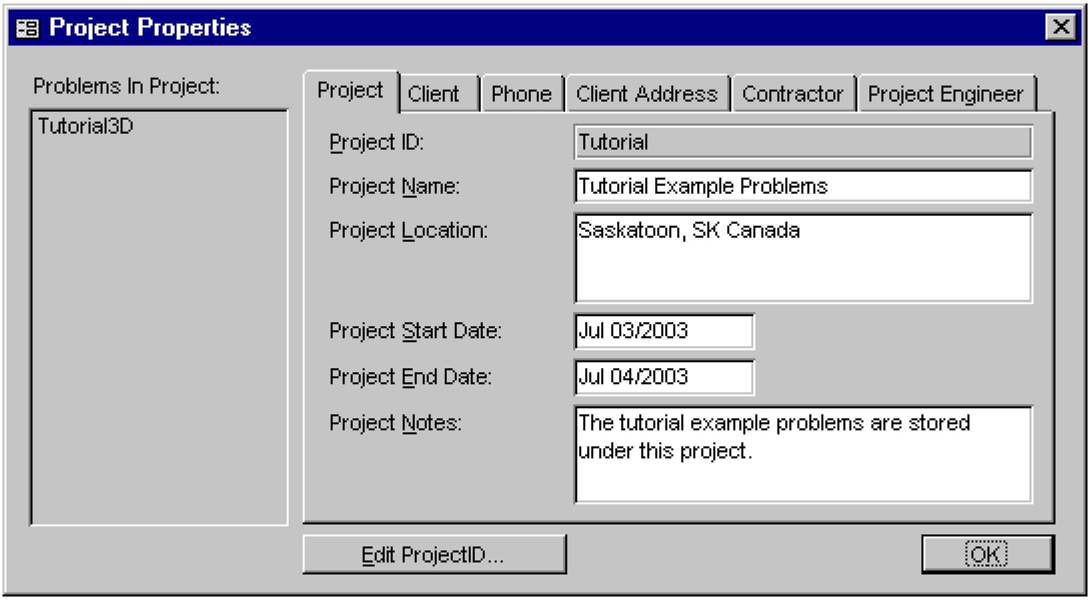
4. Type "Tutorial" as the new ProjectID and press OK.

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

Tip!

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

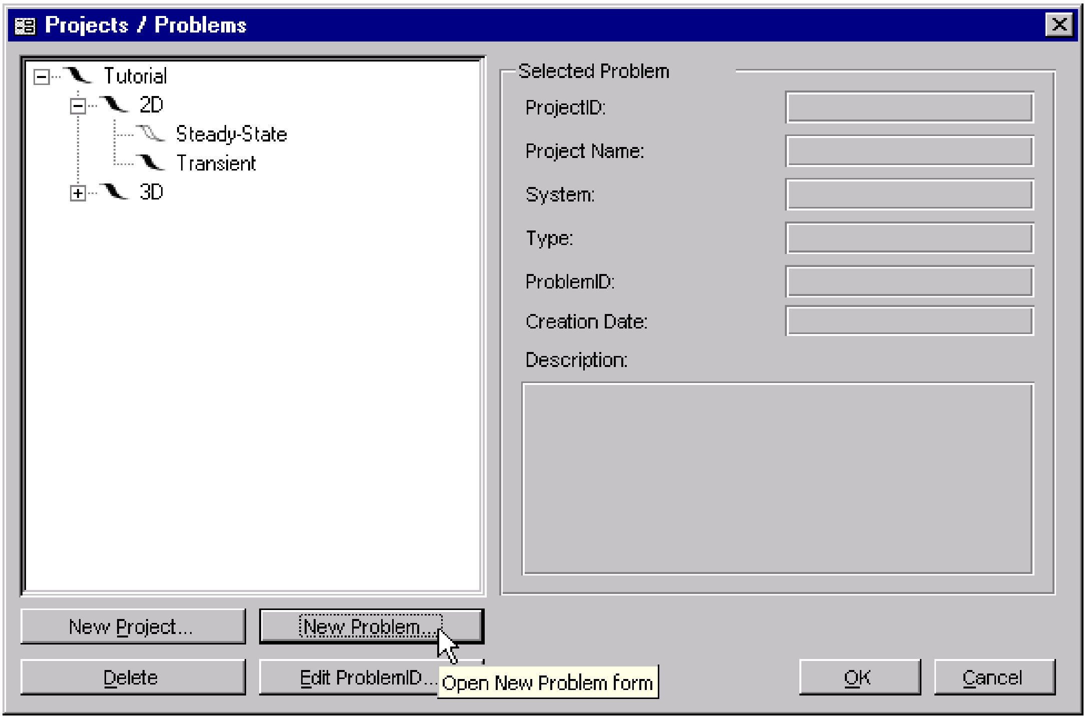
The form is opened ready to accept information.



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

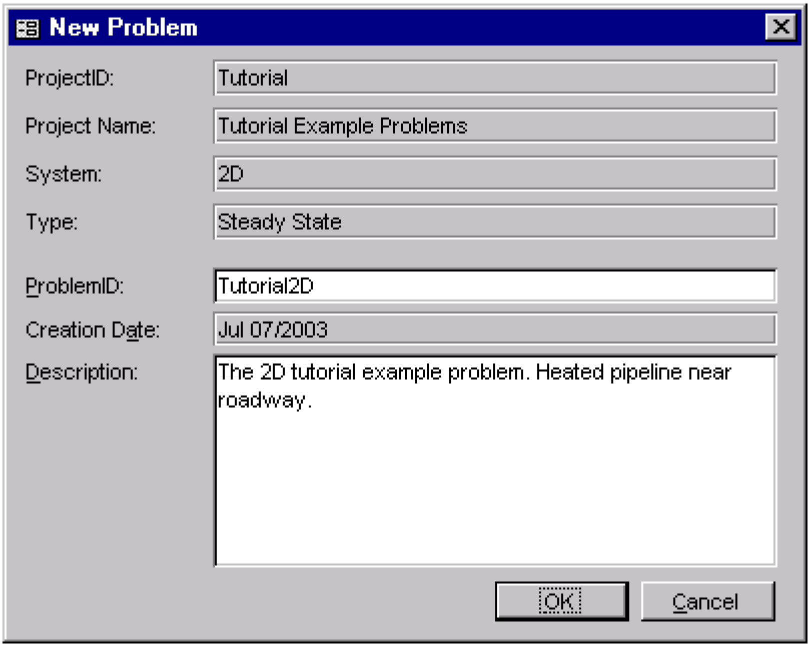
5. Fill out the form with the desired **information**.
6. To exit this form and return to Projects / Problems click **OK**. The project information is automatically saved upon entry.

1.2 ADDING A PROBLEM



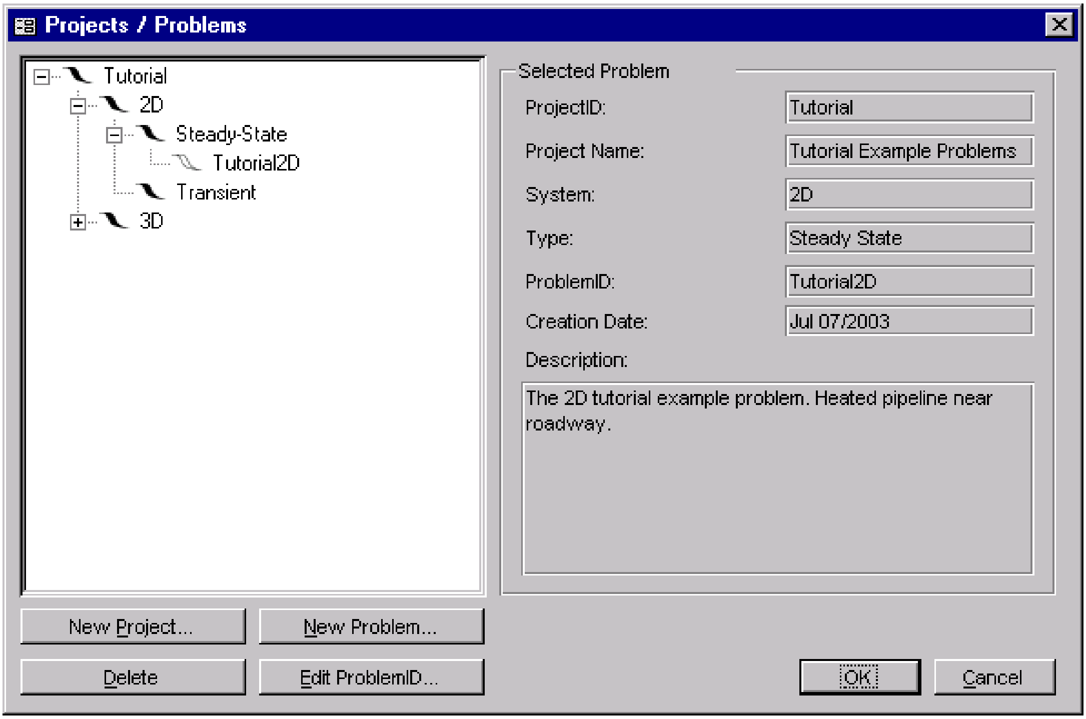
Once a project has been created any number of problems may be stored in it. When the Projects/Problems form is opened there will be a list of the projects that have been defined. In this case there is only the Tutorial project. To add a problem:

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: Tutorial2D**. 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:

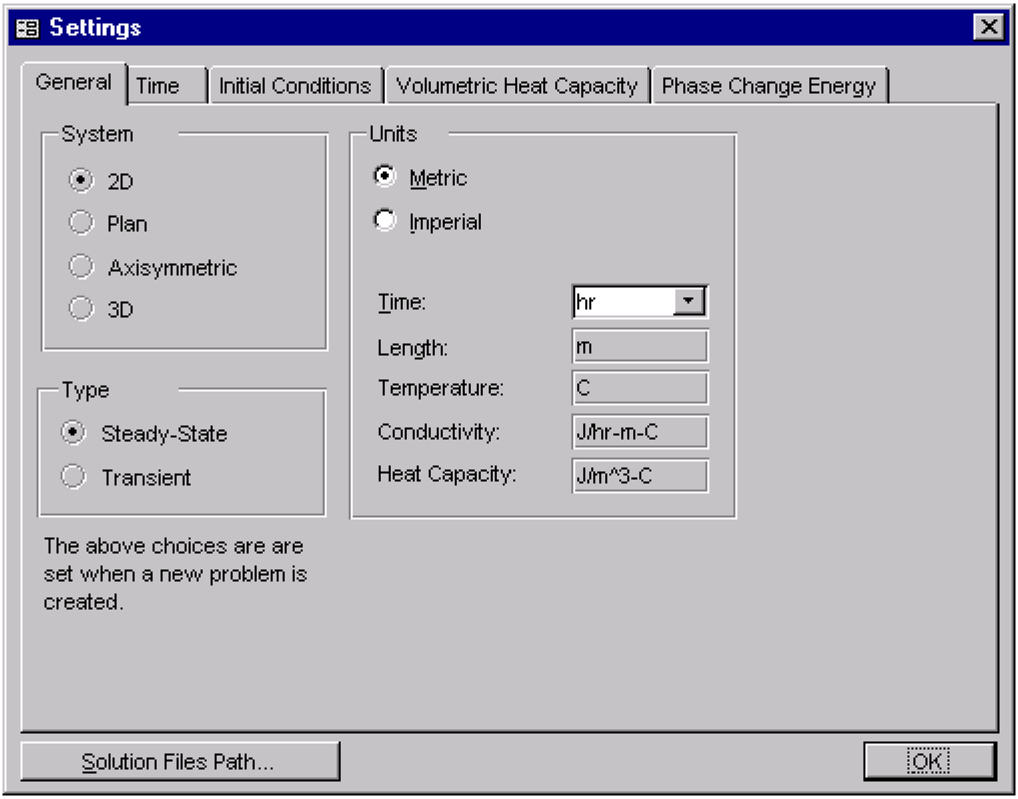
1. **Navigate** back to the problem via Tutorial, 2D, Steady-State.
2. Select **Tutorial2D.**
3. The problem may opened by clicking the **OK** button or by double clicking on the ProblemID.

1.4 DEFINING THE PROBLEM

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

1.4.1 Specify Settings

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



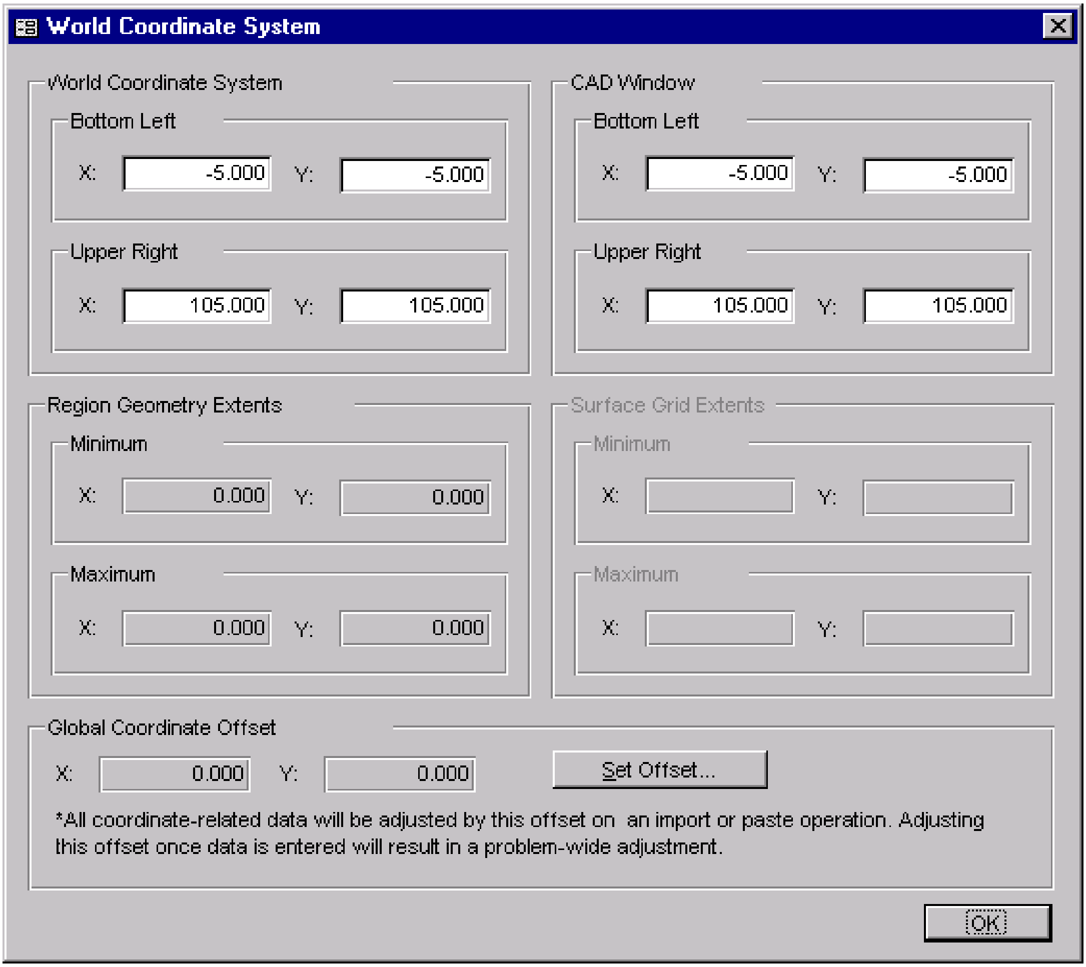
The Settings form will contain information about the current problem System, Type, Units, and Transient Settings. The thermal conductivity data for the soils contained in the problem are reported as J/hr-m-^{o}C.

Select **hr** from the **Time** drop-down to set the problem time units to hours.

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.

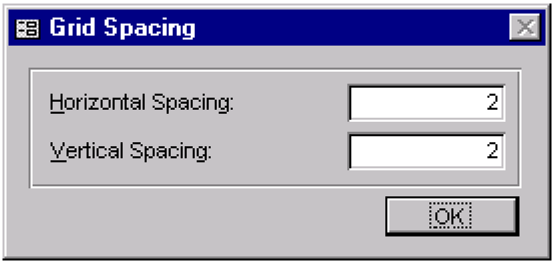
1. Access the World Coordinate System form by either of two ways. Click on the button:  or select **View > WCS**. The button is located in the view toolbar to the left of the drawing space.



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.

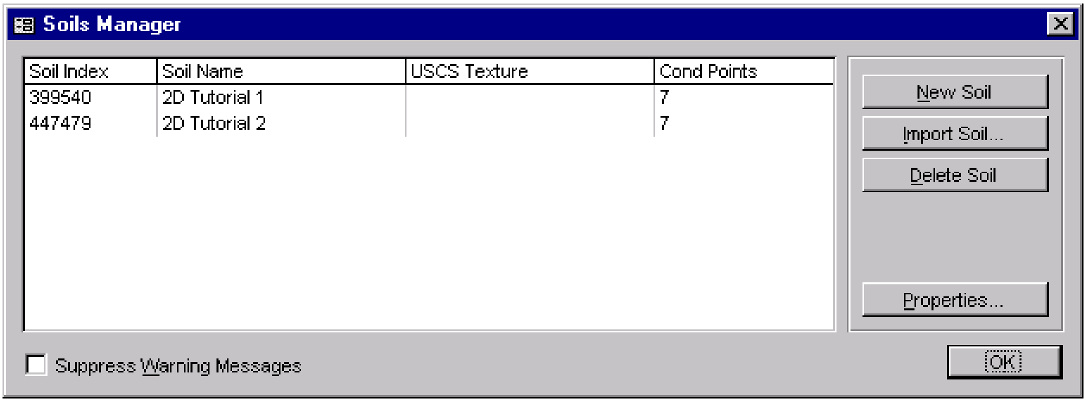
The workspace grid spacing needs to be set to aid in defining region shapes. The filter portion of the problem has coordinates of a precision of 0.5m. In order to effectively draw geometry with this precision using the mouse the grid spacing must be set to a maximum of 0.5.

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

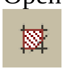


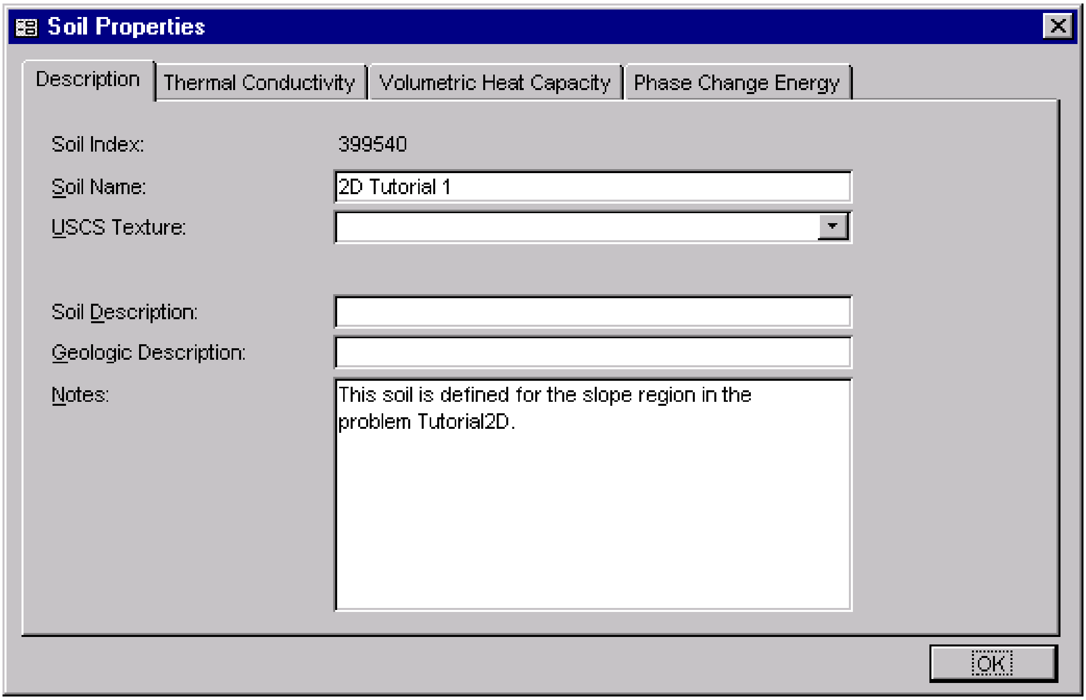
6. Enter **2** for both the horizontal and vertical spacing.
7. Click **OK** to close the form.

1.4.3 Define Material Properties

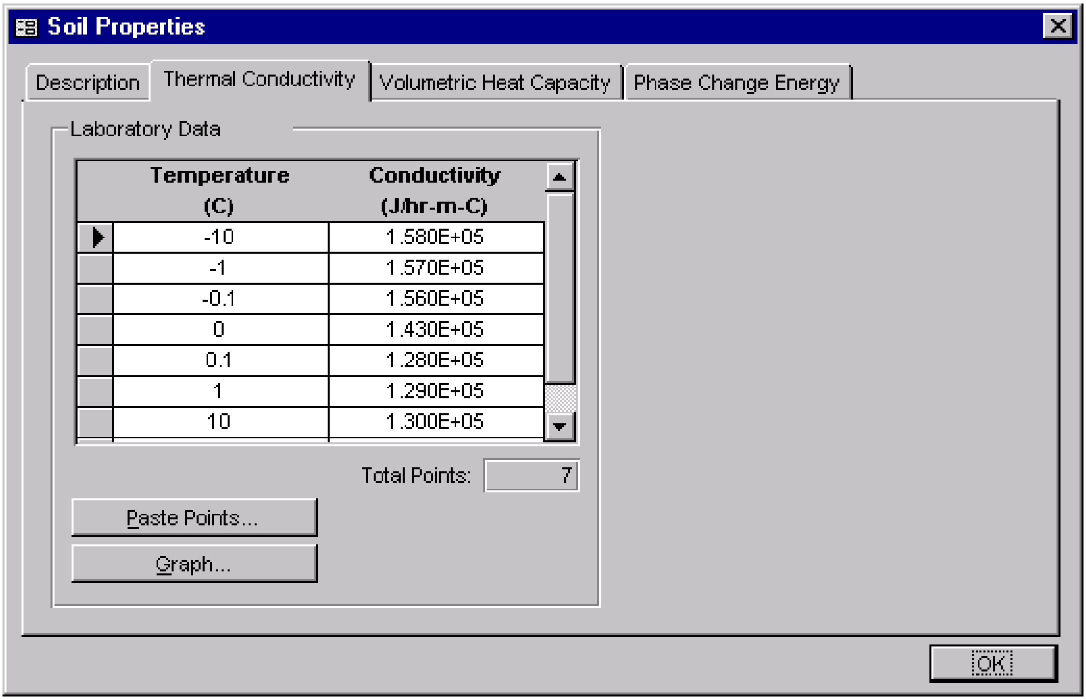


The next step in defining the problem is to enter the material properties for the 2 soils that will be used in the model. A soil called **2D Tutorial 1** is defined for the major soil region and the soil called **2D Tutorial 2** is defined for the seam. This section will provide instructions on creating the first soil. Repeat the process to add the other soil.

1. Open the Soils form by selecting **Model > Soils** from the menu or click the soils button,  in the Tools toolbar.
2. Click the **New Soil** button to create a soil in the database. A unique Soil Index is generated that is used to reference the soil in other SVHEAT 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 **Thermal Conductivity** tab.



6. Refer to the data provided at the beginning of this tutorial. Enter the laboratory data **points** as provided.
7. **Repeat** these steps to create the other soil.

Tip!

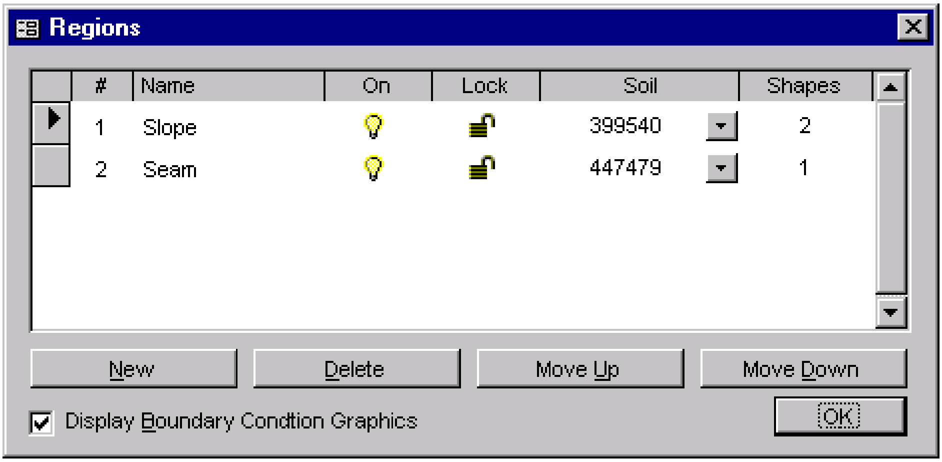
To view the thermal conductivity curve of the laboratory data press the Graph button.

1.4.4 Adding Regions

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

This problem will be divided into 2 regions, which are named Slope and Seam. Each region will have one of the soils just defined specified as its soil properties. To add the necessary regions follow these steps:

1. Open the regions form by clicking the Regions button, at the top of the workspace.



2. Change the first region name from Region 1 to **Slope**. Highlight the name and type new text.
3. Select the **Soil Index** from the drop-down corresponding to the soil: **2D Tutorial 1**.
4. Press the **New** button to add a second region.
5. Change the name of the second region to **Seam**.
6. Select the **Soil Index** from the drop-down corresponding to the soil: **2D Tutorial 2**.
7. Select the box Display Boundary Condition Graphics. This will display graphical representations for the boundary conditions when they are defined later in this tutorial.
8. Click **OK** to close the form.

1.4.5 Defining Region Geometry Shapes

The shapes that define each soil region will now be created. The Slope region will consist of 2 shapes: the dominant shape defining the slope and a circular interior shape representing the pipe. Note that when drawing geometry shapes the **region that is current in the region selector is the region the geometry will be added to**. The Region Selector is at the top of the workspace.

- **Define the Slope Region – Shape 1**

1. Ensure the "**Slope**" region is current in the region selector.
2. Click on the **Draw Polygon Region Shape** button, .
3. The cursor will now be changed to cross hairs.
4. Move the cursor near (0,0) in the drawing space. You can view the coordinates of the current position the mouse is at in the status bar just above the command line.
5. When the cursor is near the point, right click. This will cause the cursor to snap to the point (The SNAP and GRID options in the status bar must both be bold).
6. To select the point as part of the shape left click on the point.
7. Now move the cursor near (100,0). 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 (100,0).
8. **Repeat** this process for the remaining points as provided at the beginning of this tutorial.

9. For the **last point** (0,10), right click to snap the cursor to the point. Double click on the point to finish the shape. A line is now drawn from (0,15) to 0,10) and the shape is automatically finished by SVHEAT by drawing a line from (0,10) back to the start point, (0,0).

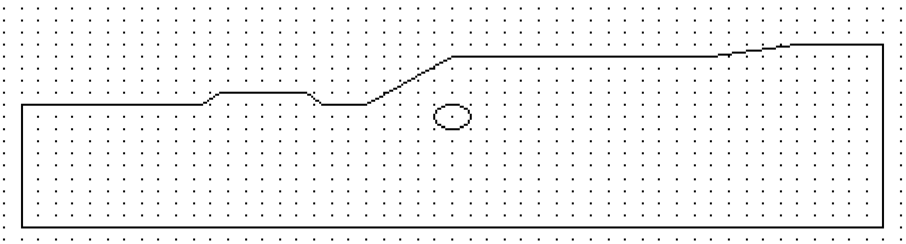
Tip!

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.

- **Define the Slope Region – Shape 2**

1. Ensure the "**Slope**" region is current in the region selector.
2. Click on the **Draw Circle Region Shape** button, .
3. The cursor will now be changed to cross hairs.
4. Move the cursor near (50,18) 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 the **circle center** left click on the point.
7. Draw the cursor out to a **radius** of 2.
8. Left-click to finish the circle.
9. To ensure that the radius is set to 2 open the double-click on the circle shape to open the Region Properties form.
10. Enter a radius of 2 if necessary.
11. Click OK to close the Region Properties form.

If the slope 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 Seam Region**

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

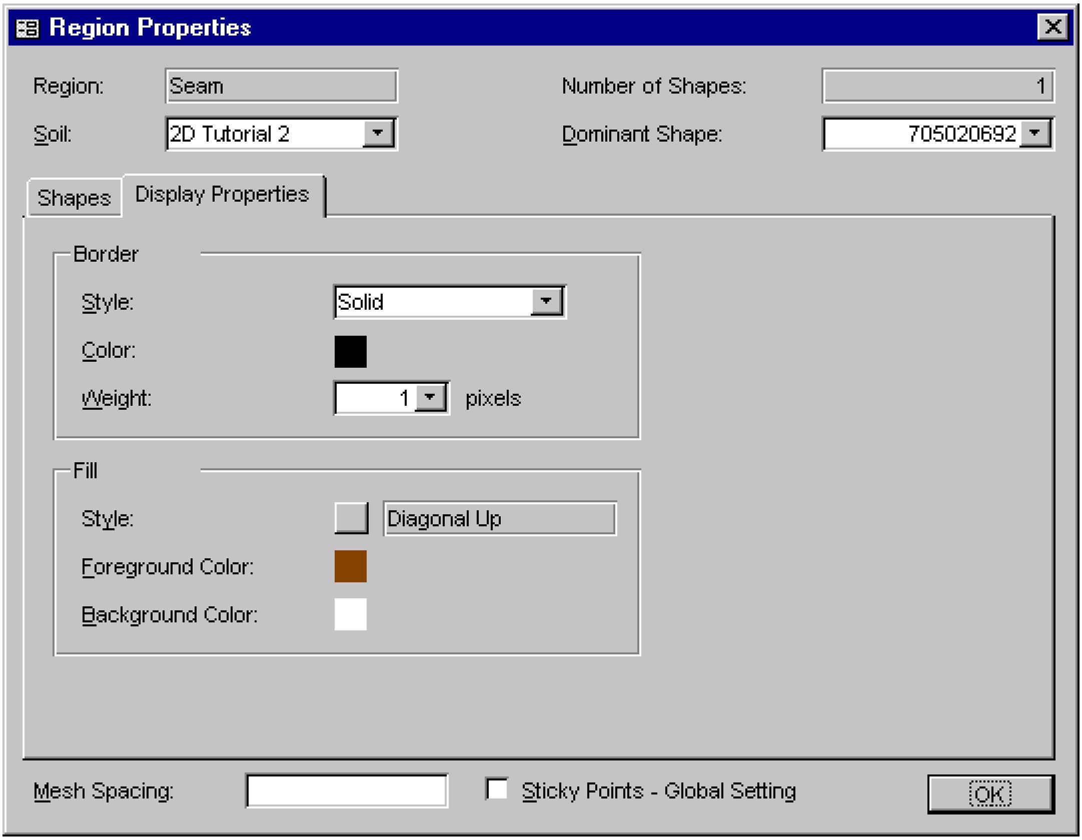
1. Ensure that "**Seam**" is current in the region selector.

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 0,10 and press the Enter key on the keyboard.
5. Type 55,10 and press Enter.
6. Type 40,15 and press Enter.
7. Type 0,15 and press Enter.
8. Type **f** and press **Enter** to complete the region shape.

- **Formatting a Shape**

  To set the Seam region shape fill to a diagonal brown:

1. Select the **Seam** 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 **Diagonal UP** fill style and press **OK**.
5. Click the **Foreground** color box on the Region Properties form. The Color Palette will appear.
6. Select a **brown** color and press **OK**.

7. Click the **Background** color box on the Region Properties form. The Color Palette will appear.
8. Select white and press **OK**.
9. Close the Region Properties form by pressing **OK**.

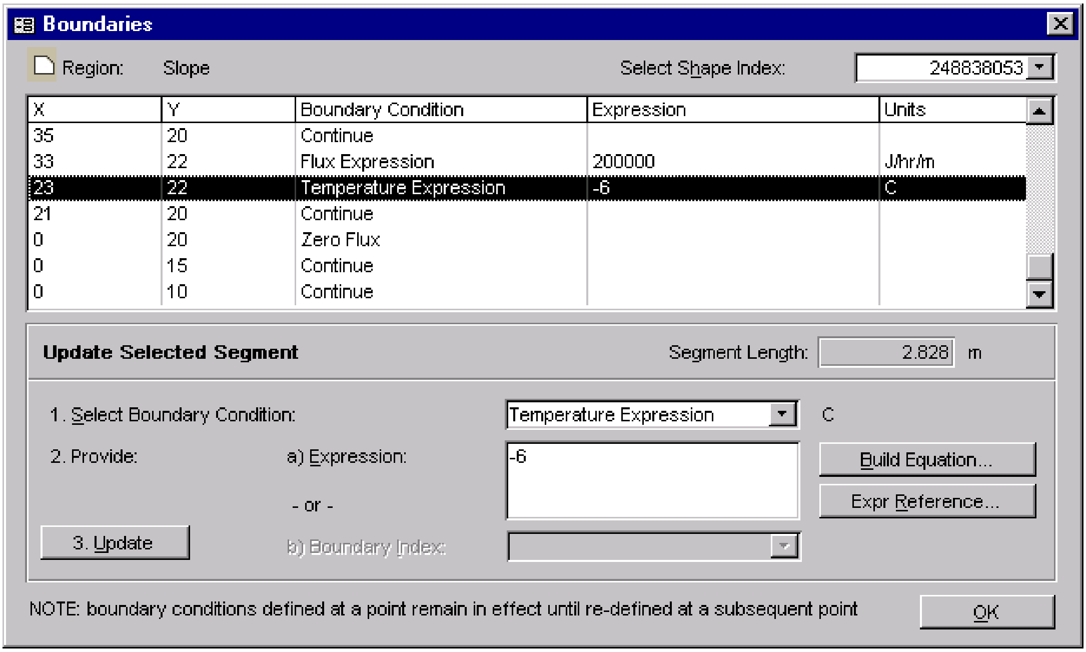
After all the region geometry has been entered it will as appear as in the diagram at the beginning of this tutorial.

1.4.6 Specify Boundary Conditions

Now that all of the regions and the problem geometry have been successfully defined, the next step is to specify the boundary conditions. An approximate geothermal gradient of 1^{o}C/30m will be simulated by setting the temperature at the surface to -6 ^{o}C and the base of the problem to -5 ^{o}C. The temperature of the pipe is 9 ^{o}C and the heat generated by the warming of the roadway is set as 200 kJ/hr. The steps for specifying the boundary conditions are thus:

- **Slope – Dominant Shape**
-

1. Select the “Slope” dominant region in the drawing space.
2. From the menu select **Model > Boundaries**. The boundary conditions form will open.



3. Select the **point** (0,0) from the list.
4. From the **Boundary Condition** drop down select a **Temperature Expression** boundary condition. This will cause the Temperature Expression box to be enabled.
5. In the Temperature Expression box enter a **temperature of -5**.
6. Click the **Update** button to save the boundary condition to the list.

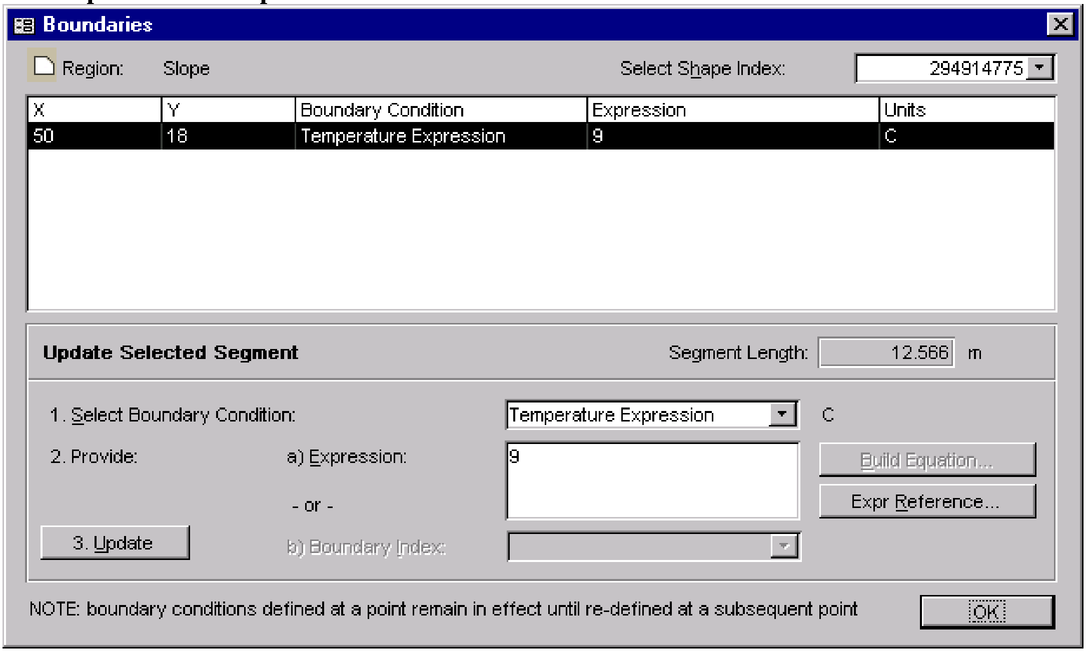
7. Select the **point** (100,0) from the list.
8. From the **Boundary Condition** drop down select a **Zero Flux** boundary condition.
9. Click the **Update** button to save the boundary condition to the list.
10. Repeat for the remaining points referring to this table:

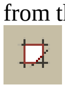
X	Y	Boundary Condition
0	0	Temperature Expression = -5
100	0	Zero Flux
100	30	Temperature Expression = -6
90	30	Continue
80	28	Continue
50	28	Continue
40	20	Continue
35	20	Continue
33	22	Flux Expression = 200000
23	22	Temperature Expression = -6
21	20	Continue
0	20	Zero Flux
0	15	Continue
0	10	Continue

Tip!

The Temperature Expression boundary condition for the point (100,30) becomes the boundary condition for the following line segments that have a Continue boundary condition until a new boundary condition is specified. By specifying a Zero Flux condition at point (33,22) the Continue boundary condition is stopped.

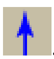
- **Slope – Circle Shape**

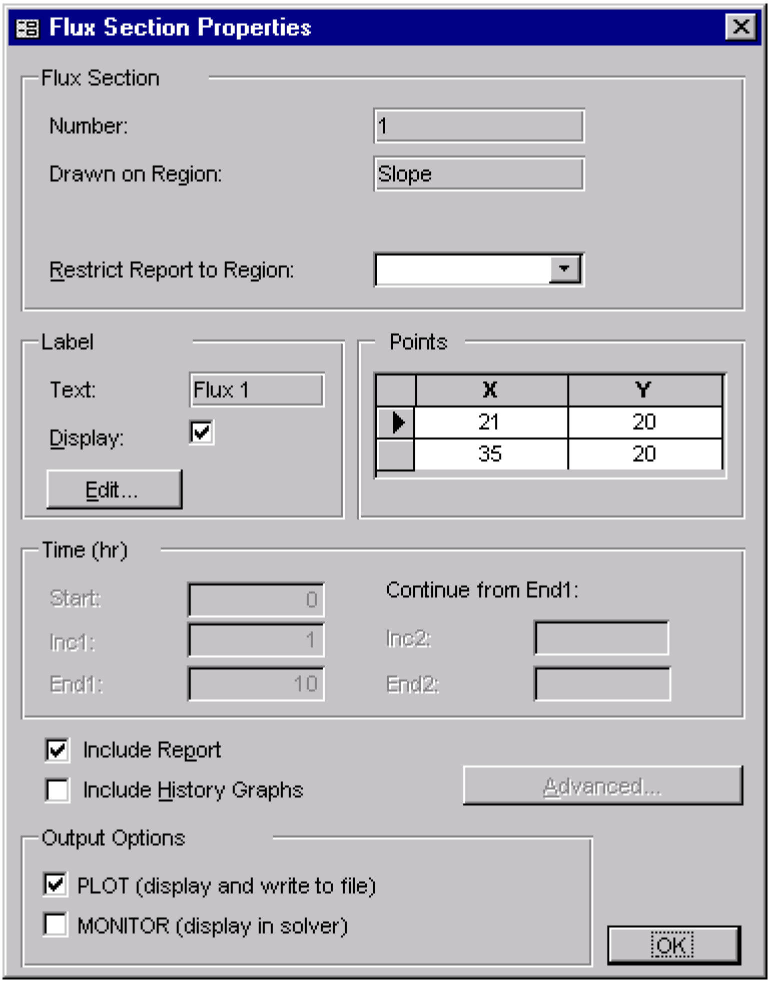


1. Select the Slope circle region from the drawing space.
2. Press the **Boundaries** button,  to open the Boundaries form.
3. In the Boundary Condition drop down select a **Temperature Expression** boundary condition.
4. Enter the value **9**.
5. Click the **Update** button to save the change.
6. Click **OK** to return to the workspace.

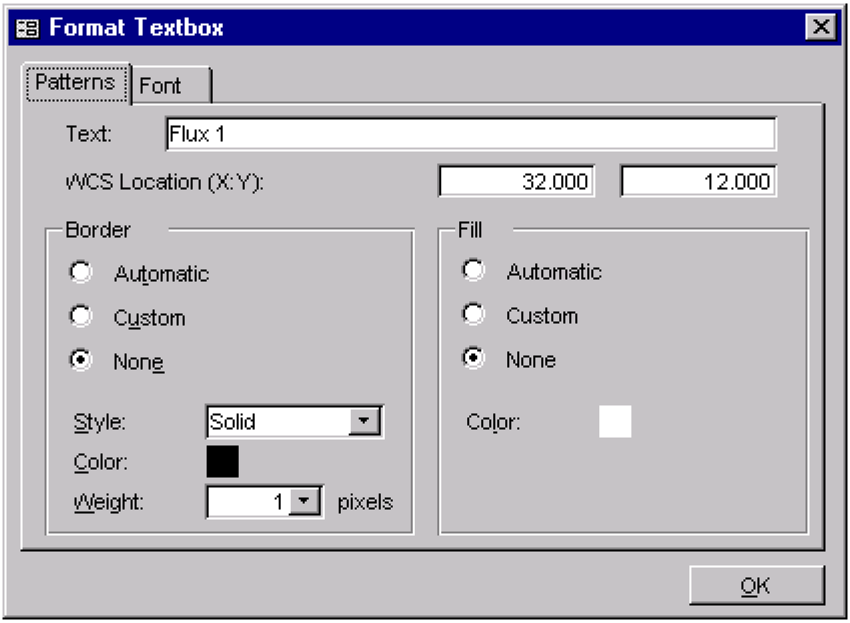
1.5 SPECIFYING A FLUX SECTION

Flux sections are used to report the rate of heat flow across a portion of the problem for a steady state analysis and the rate and total heat 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** (21,20).
3. To finish the Flux Section click on the **point** (35,20). 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. 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.



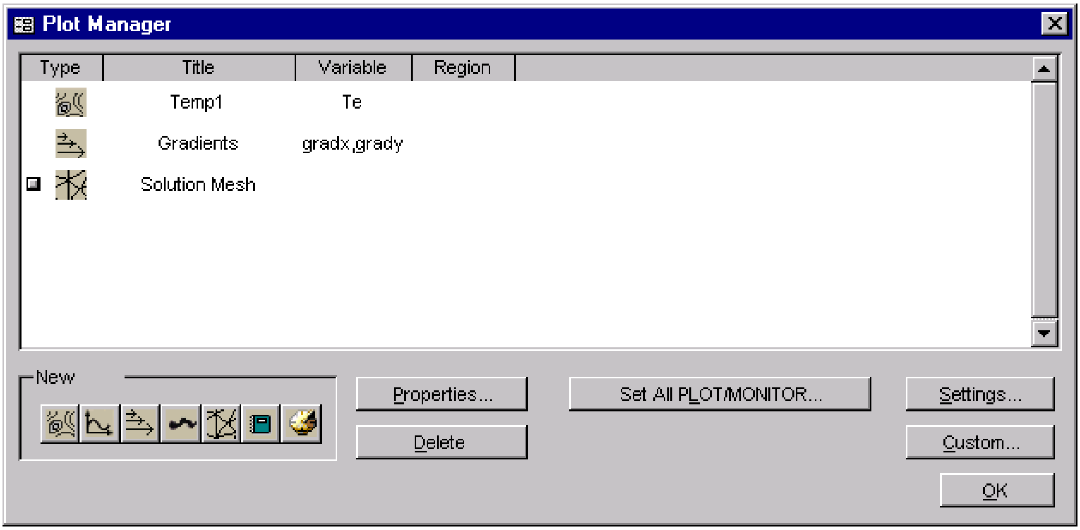
8. Enter 32 and 12 in the **WCS Location** fields.
9. Press **OK** to close the form.
10. Close the Flux Section Properties form.
11. Close the Flux Section List form.

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

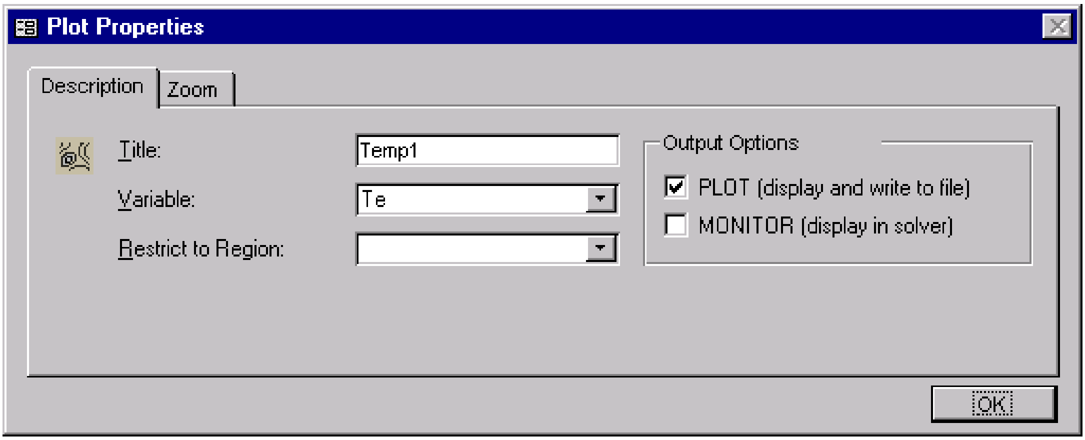
1.6 SPECIFY PLOTS

There are many plot types that can be specified to visualize the results of the model. Three will be generated for this tutorial example problem: temperature contours, thermal gradient vectors, and the solution mesh.

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.

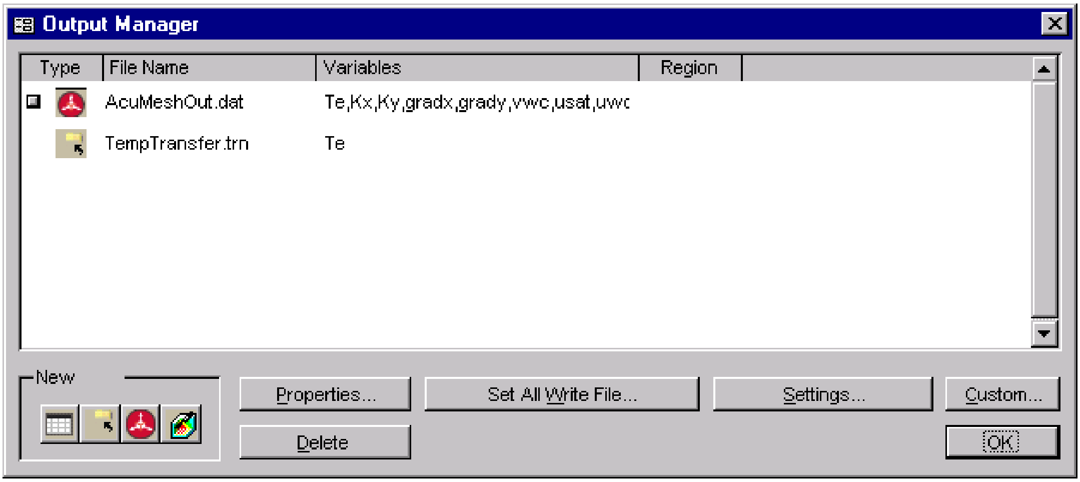


3. Enter the title Temp1.
4. Select Te as the variable to plot from the drop-down.
5. Select the **PLOT** output option.
6. Click OK to close the form and add the plot to the list.
7. Repeat these steps 2 – 6 to create the plots shown above. Note that the Mesh plots do not require entry of a variable.
8. Click **OK** to close the Plot Manager and return to the workspace.

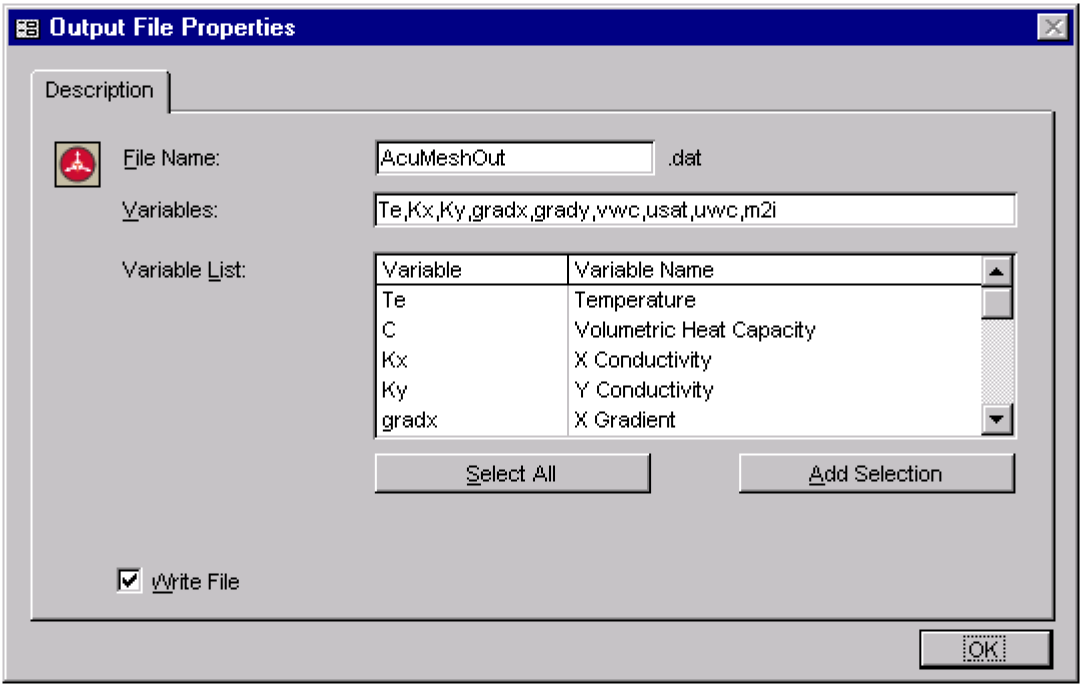
1.7 SPECIFY OUTPUT FILES

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 temperatures, and a plot to transfer the results to AcuMesh. Note that the file TempTransfer.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 AcuMeshOut.
4. Select all the variables in the variable list.
5. Press the **Add Selection** button.
6. Check the **Write File** box.
7. Click OK to close the form and add the output file to the list.
8. Click **OK** to close the Output Manager and return to the workspace.

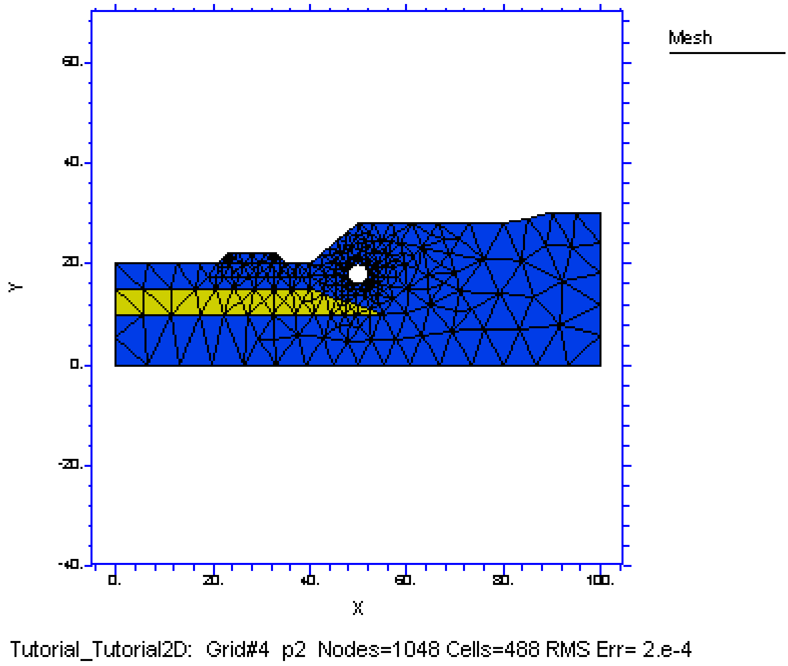
1.8 ANALYZE

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 SVHEAT solver. The solver will automatically begin solving the problem.

1.9 RESULTS

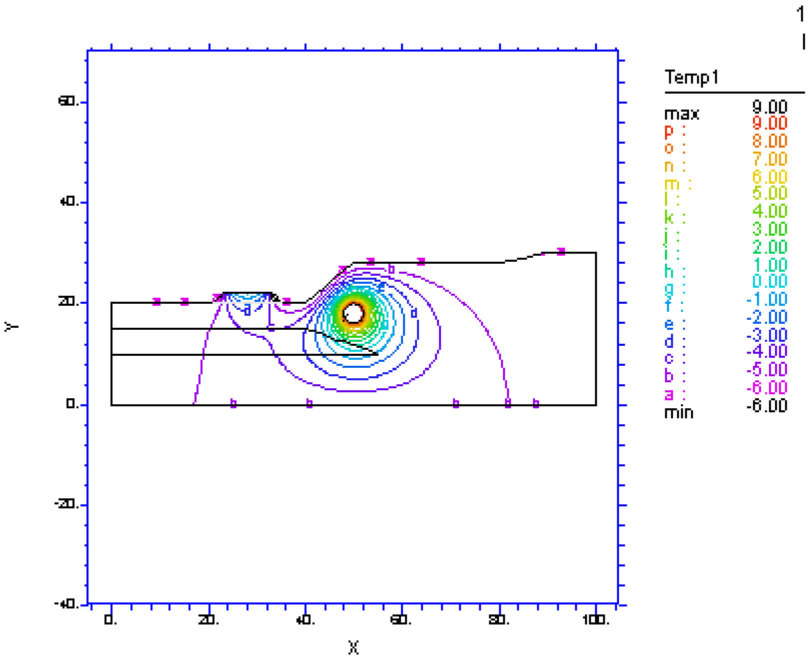
After the problem has finished solving, the results will be displayed in the form of thumbnail plots within the SVHEAT solver. Right-click the mouse and select Maximize to enlarge any of the thumbnail plots. This section will give a brief analysis for each plot that was generated.

1.9.1 Solution Mesh



The Mesh plot displays the finite-element mesh generated by the solver. The mesh is automatically refined in critical areas such as around the pipe contact where there is a significant change in temperature.

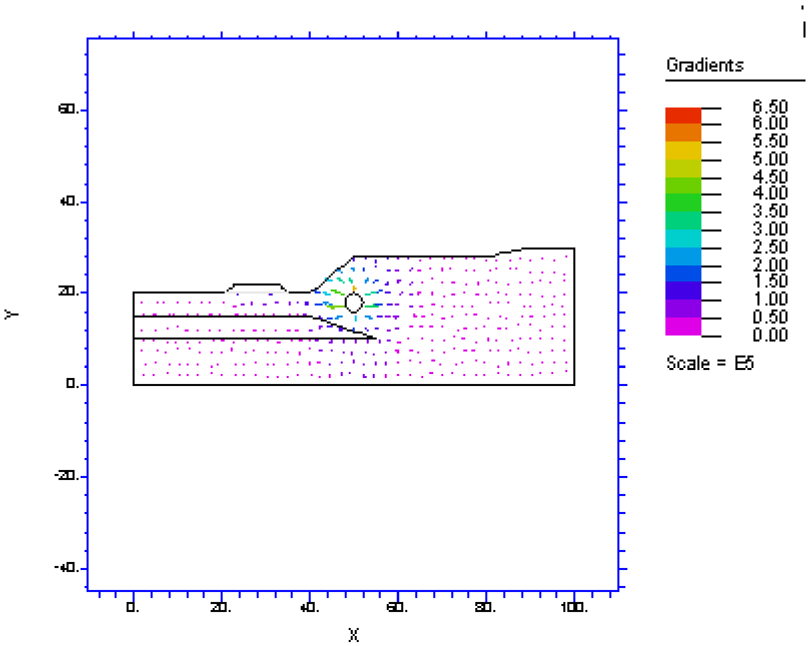
1.9.2 Temperature Contours



Tutorial_Tutorial2D: Grid#4 p2 Nodes=1048 Cells=488 RMS Err= 2.e-4
Integral= -10560.20

The temperature contours indicate that the temperature of the pipe does not significantly influence the base of roadway. Instead the heat flux due to the warming of the roadway surface has an influence of a few degrees.

1.9.3 Gradient Vectors



Tutorial_Tutorial2D: Grid#4 p2 Nodes=1048 Cells=488 RMS Err= 2.e-4

Gradient Vectors show both the direction and the magnitude of the heat flow at specific points in the problem. Vectors illustrate that heat flow is away from the pipeline.

1.9.4 Flux Section Report

Flux Section Report: Flux 1

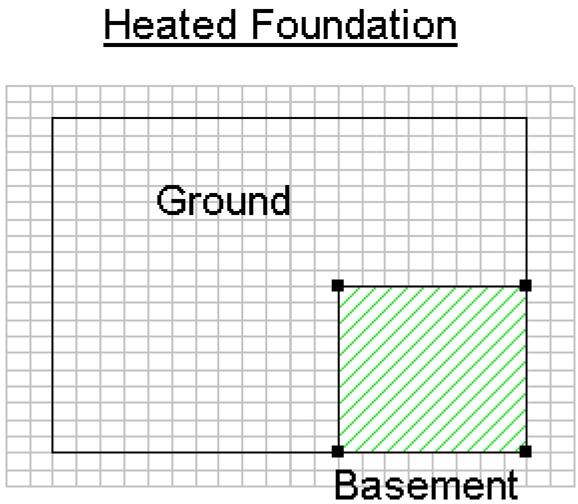
X Component of Flow in (J/hr)= 0.00
Y Component of Flow in (J/hr)= -466233.7
Normal Flow in (J/hr)= -466233.7

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

2 A THREE DIMENSIONAL EXAMPLE PROBLEM

The following example will introduce you to the three dimensional model in SVHEAT. The model will be used to investigate the steady-state condition of a soil resulting from a heated foundation in winter conditions. The tutorial is a detailed set of instructions guiding the user through the creation of the 3D heat transfer problem. The problem is modeled using 2 regions, 3 surfaces, and 1 soils. The problem data and soil properties are provided below.

ProjectID: Tutorial ProblemID: Tutorial3D



Soil: Thermal Conductivity curve laboratory data:

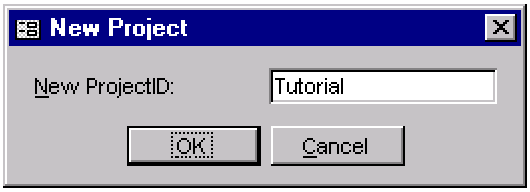
Temperature (^oC)	Conductivity (J/s-m- ^oC)
-2	2000
-1	1999
0	1000
1	1001
2	1002

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.

Follow these steps in order to add this project:

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

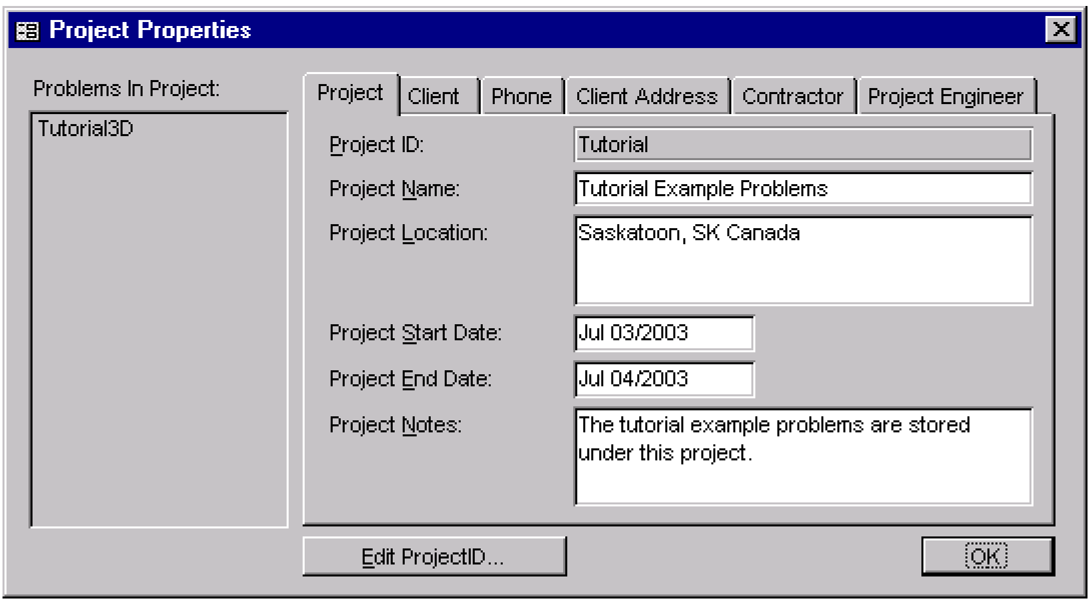


4. Type “Tutorial” as the new ProjectID and press OK.

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

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

The form is opened ready to accept information.

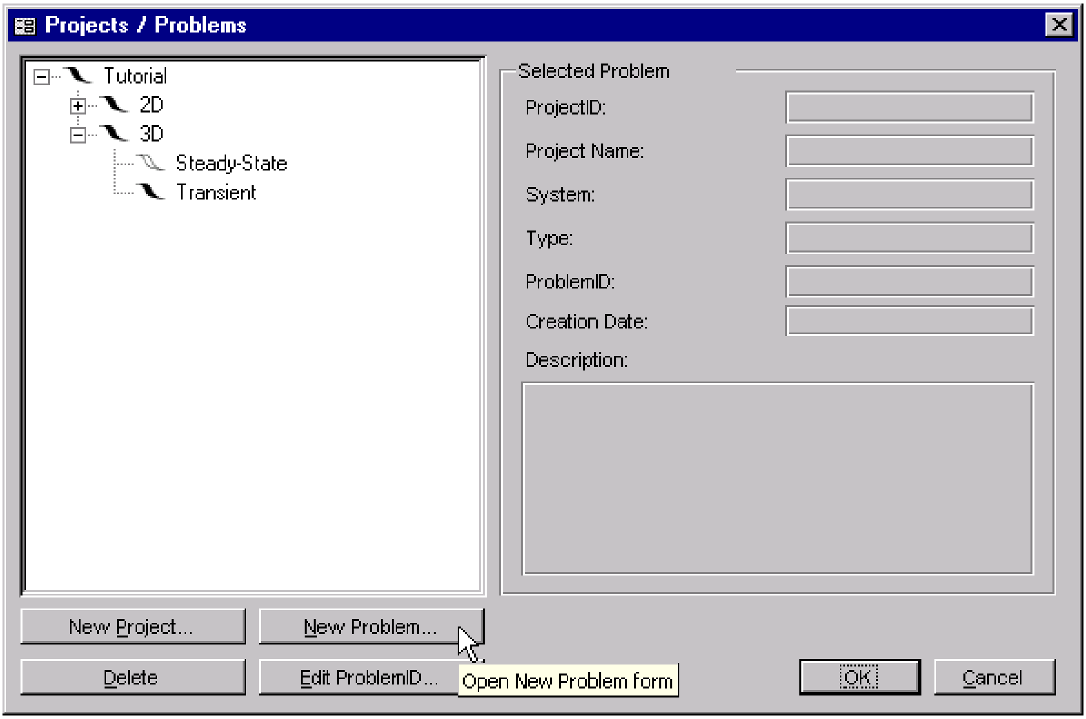


It should be noted that once the project is defined it would be identified by the ProjectID throughout the rest of the program. Also, SVHEAT does not allow you to specify two projects with the same ProjectID.

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.

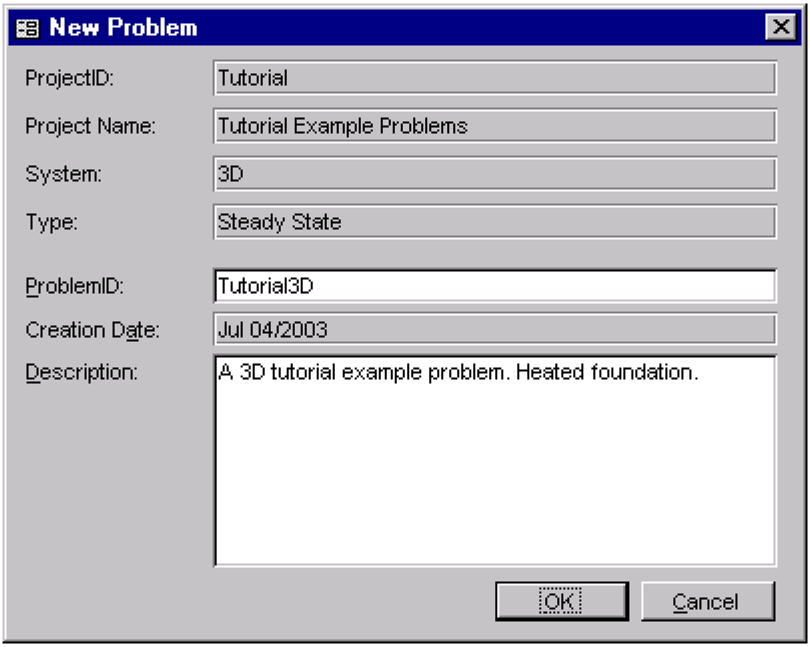
2.2 ADDING A PROBLEM

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:

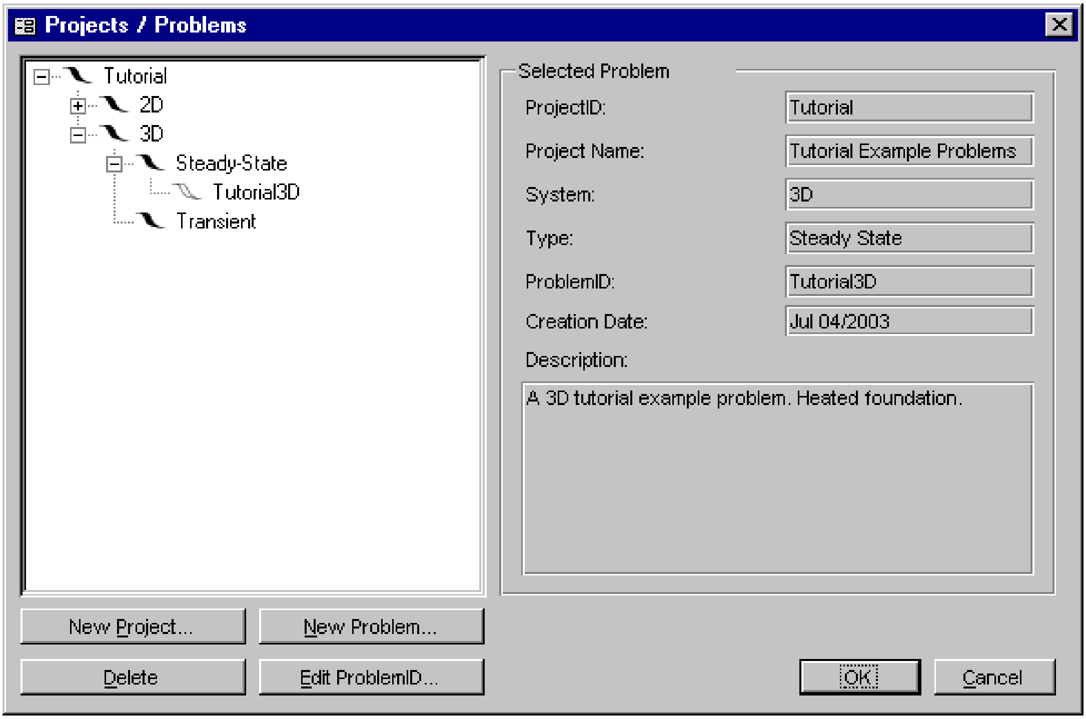
1. Click on the plus sign and expand the project **Tutorial**.
2. This example will be modeled in three-dimensions and is steady-state so we must expand **3D** by clicking the plus sign beside "3D"
3. Select **Steady-State**.
4. Click the **New Problem** button. The New Problem form will open.



5. Enter the **ProblemID: Tutorial3D**. 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.

2.3 OPENING THE PROBLEM

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



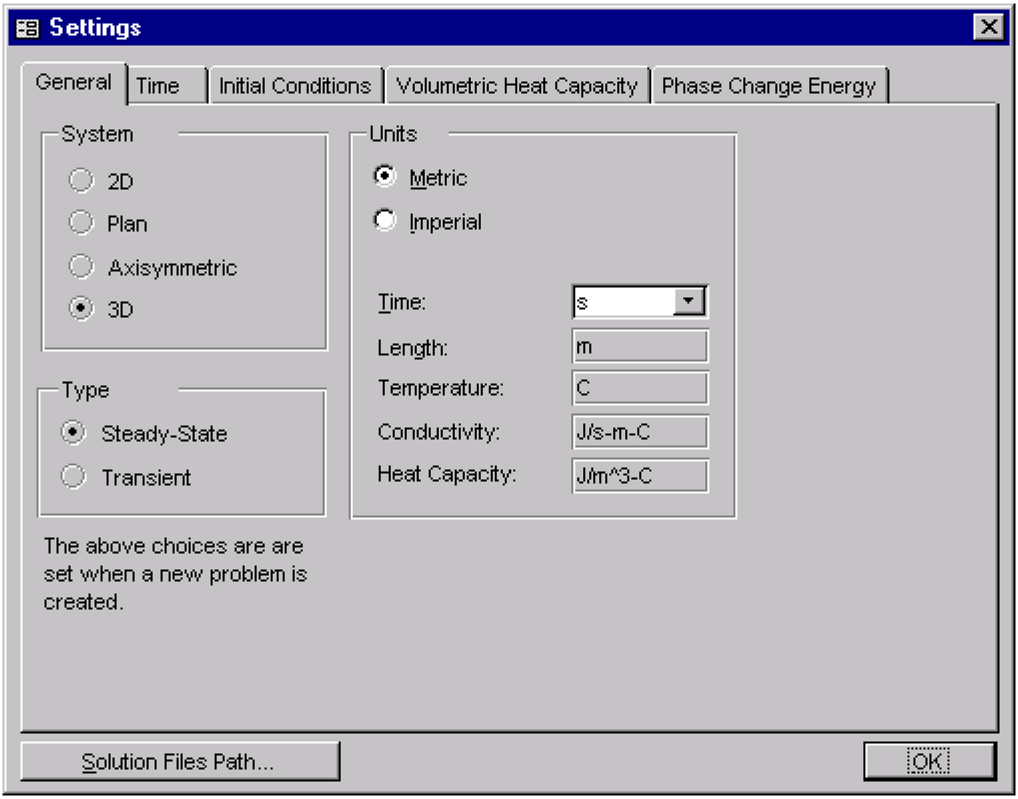
1. **Navigate** back to the problem via Tutorial, 3D, Steady-State.
2. Select **Tutorial3D**.
3. The problem may opened by clicking the **OK** button or by double clicking on the ProblemID.

2.4 DEFINING THE PROBLEM

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

2.4.1 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.

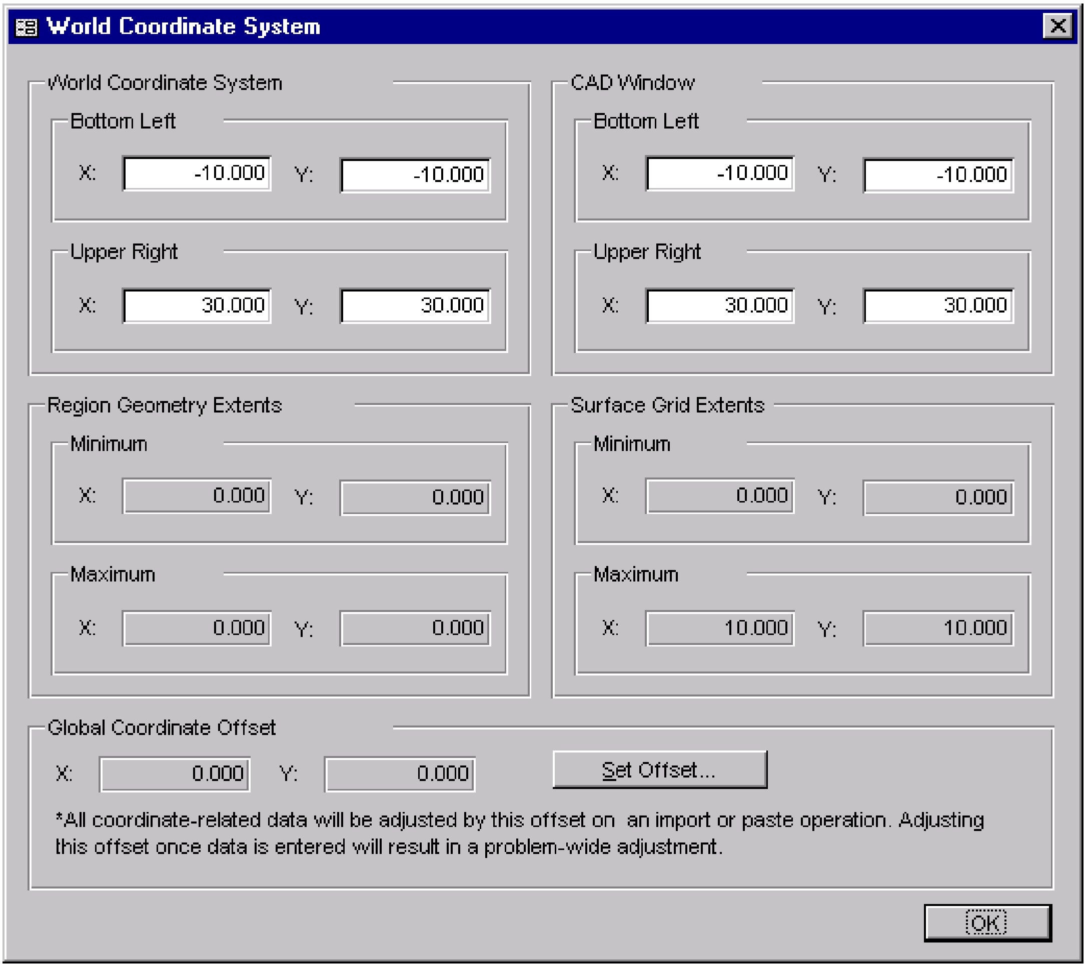


The Settings form will contain information about the current problem System, Type, Units, and Transient Settings. The thermal conductivity data for the soils contained in the problem are reported as J/s-m-^{o}C so the time units will remain s, for seconds.

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.

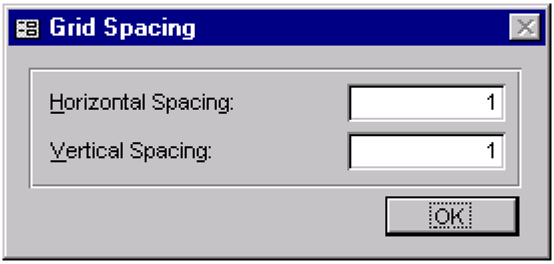
1. Access the World Coordinate System form by either of two ways. Click on the button:  or select **View > WCS**. The button is located in the view toolbar to the left of the drawing space.



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.

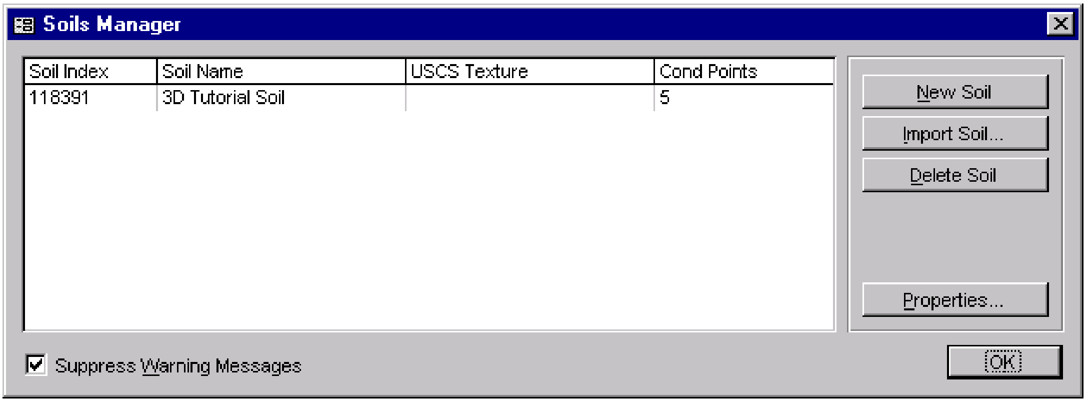
The workspace grid spacing needs to be set to aid in defining region shapes. The geometry data for this problem has coordinates of a precision of 1m. In order to effectively draw geometry with this precision using the mouse the grid spacing must be set to a maximum of 1.

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

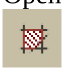


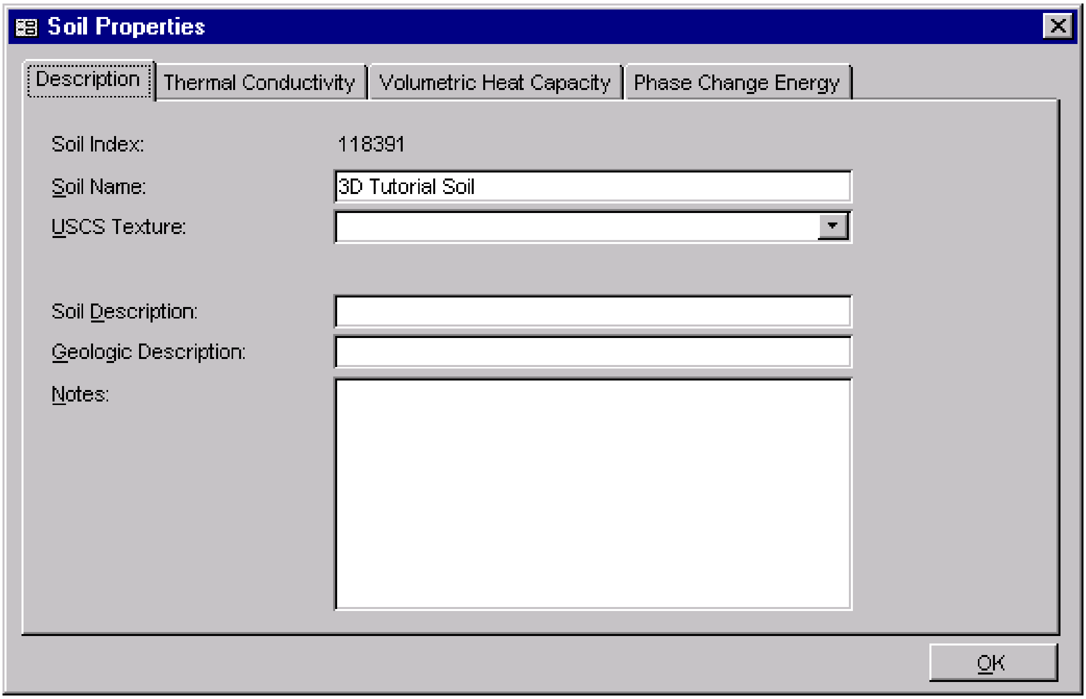
6. Enter **1** for both the horizontal and vertical spacing.
7. Click **OK** to close the form.

2.4.3 Define Material Properties

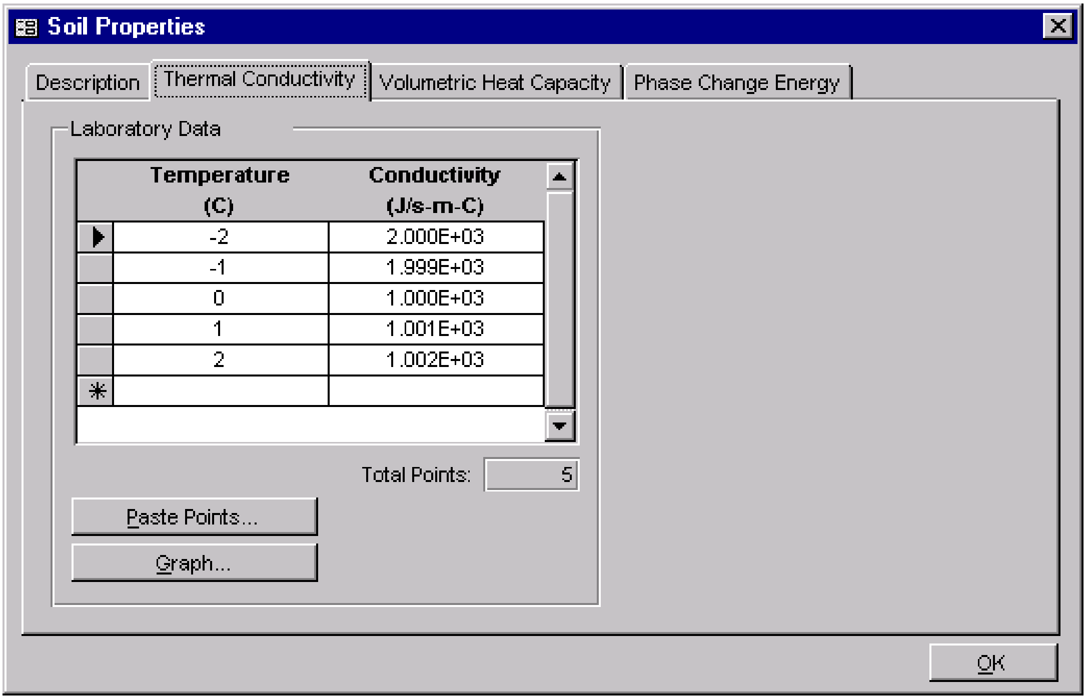


The next step in defining the problem is to enter the material properties for the soil that will be used in the model.

1. Open the Soils form by selecting **Model > Soils** from the menu or click the soils button,  in the Tools toolbar.
2. Click the **New Soil** button to create a soil in the database. A unique Soil Index is generated that is used to reference the soil in other SVHEAT 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 **Thermal Conductivity** tab.



6. Refer to the data provided at the beginning of this tutorial. Enter the laboratory data **points** as provided.

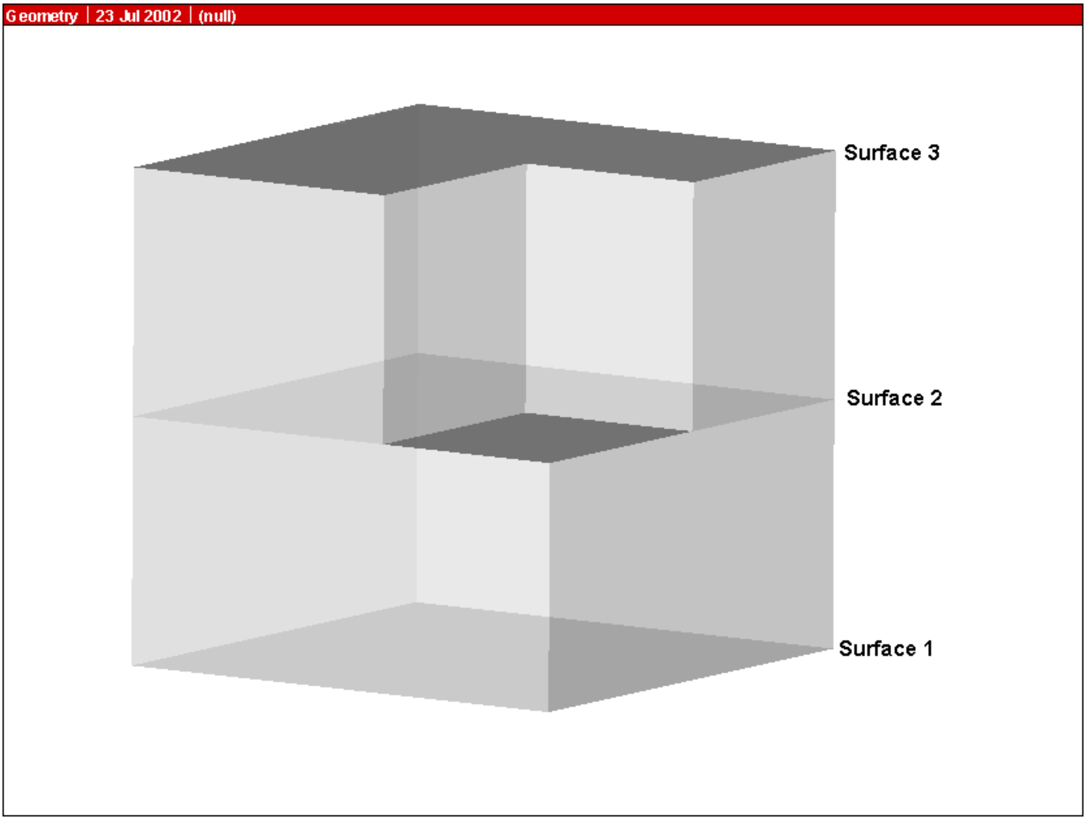
Tip!

To view the laboratory data press the Graph button.

7. Press **OK** on the Soils Manager form to close both forms.

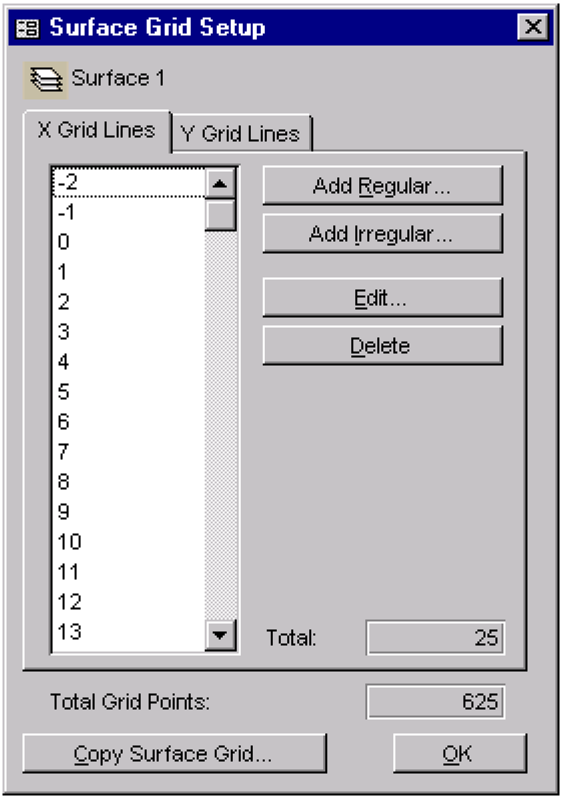
2.4.4 Define 3D Surfaces

This problem consists of three surfaces. Although it is not required the surface grids have the same 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 -2 for **Start**, 1 for **Increment Value**, and 22 for **End**.
5. Click **Add** to add the gridlines and close the form.
6. Move to the **Y Grid Lines** tab and repeat steps 3 – 5 for the Y gridlines

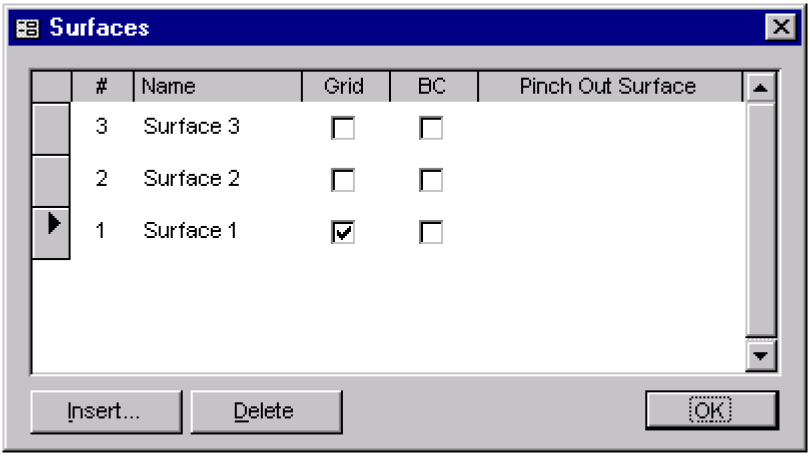
Now that the grid has been set up, elevations must be specified for all the grid points:

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

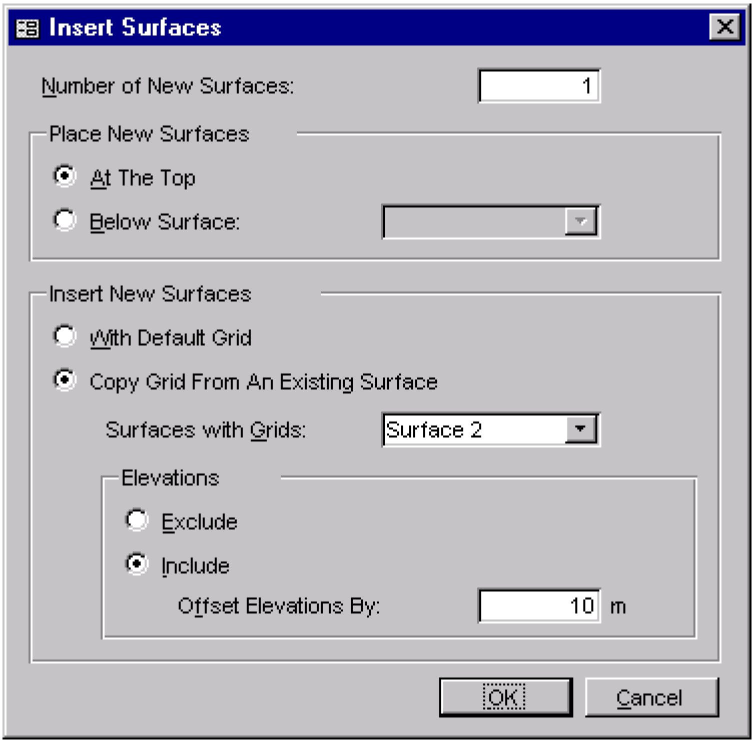
- **Define Surface 2**
  This surface is already present. Follow the steps for defining Surface 1 except set all the Surface 2 elevations to 10m.

- **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:

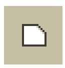


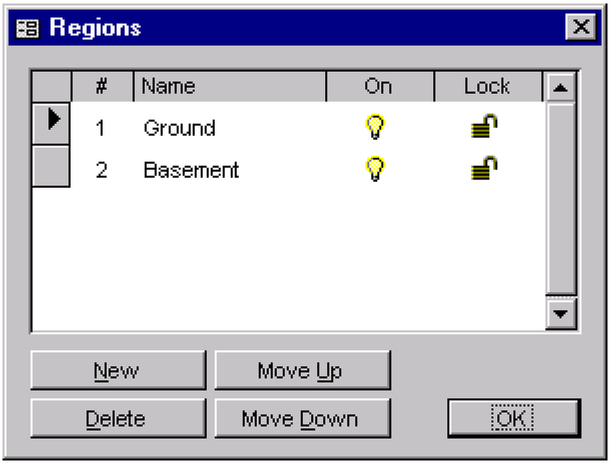
3. Enter 1 as the **Number of New Surfaces**.
4. Select to place the new surface **At The Top**.
5. Select **Copy Grid From An Existing Surface**.
6. Select **Surface 2** from the drop-down.
7. Choose to **Include** the elevations.
8. Enter **10m** as the offset from Surface 2.
9. Press **OK** to add the surface.

2.4.5 Adding Regions

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

This problem will be divided into 2 regions, which are named Ground and Basement. 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 **Ground**. Highlight the name and type new text.
3. Press the **New** button to add a second region.
4. Change the name of the second region to **Basement**.
5. Click **OK** to close the form.

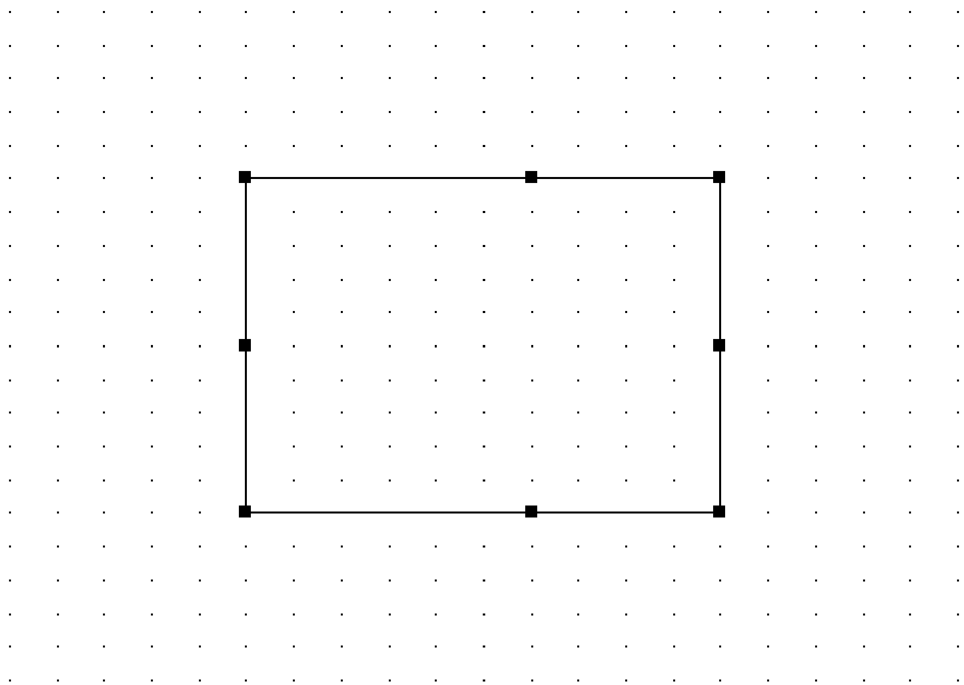
2.4.6 Defining Region Geometry Shapes

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 "Ground" 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 (12,0). 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 (12,0).
8. Repeat this process for the remaining points.
9. For the last point (0,10), right click to snap the cursor to the point. Double click on the point to finish the shape. A line is now drawn from (0,20) to 0,10) and the shape is automatically finished by SVHEAT by drawing a line from (0,10) back to the start point, (0,0).

If the Main 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 Basement**
  In the instructions for drawing the Ground shape the mouse was used. To draw the basement the instructions below explain the use of the command line to create the Basement shape.

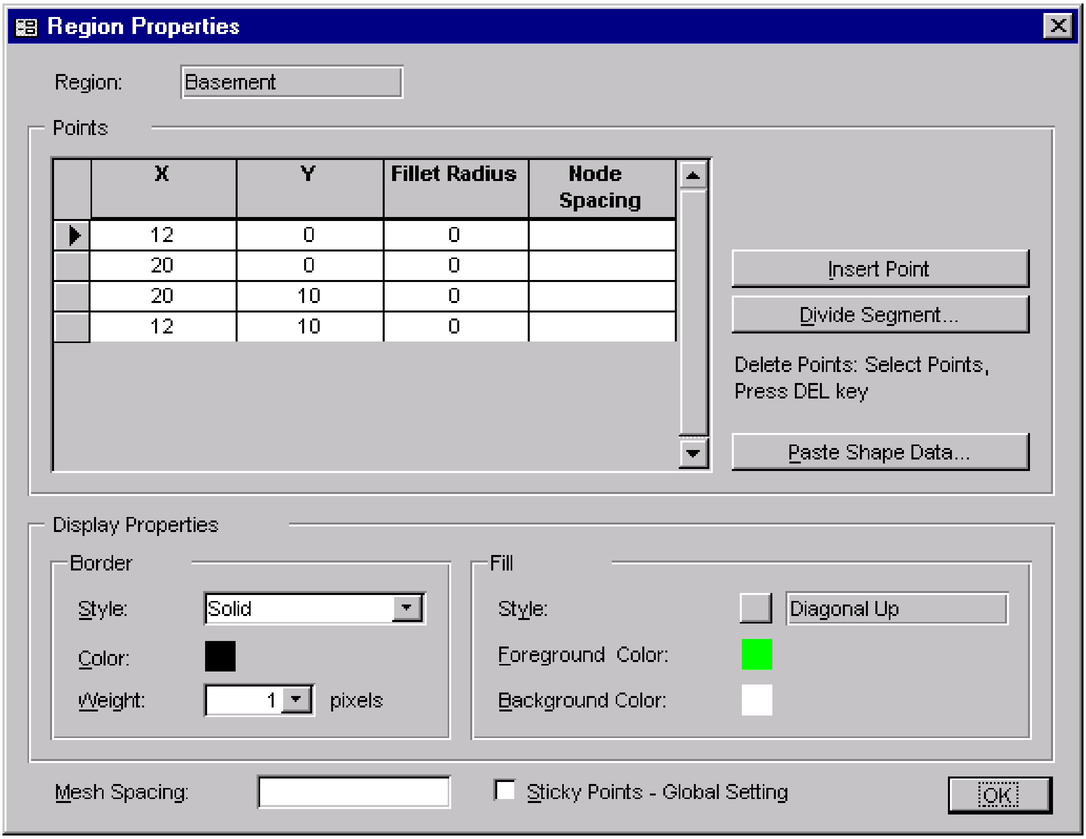
  1. Ensure that "Basement" is current in the region selector.
  2. Click on the Draw Polygon Region Shape button, .
  3. The command line will be set to Start Point and the cursor focus will be in the command line.
  4. Type 12,0 and press the Enter key on the keyboard.
  5. Enter the remaining coordinates.
  6. Type f and press Enter to complete the region shape.

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 Basement region shape to diagonal green lines:

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



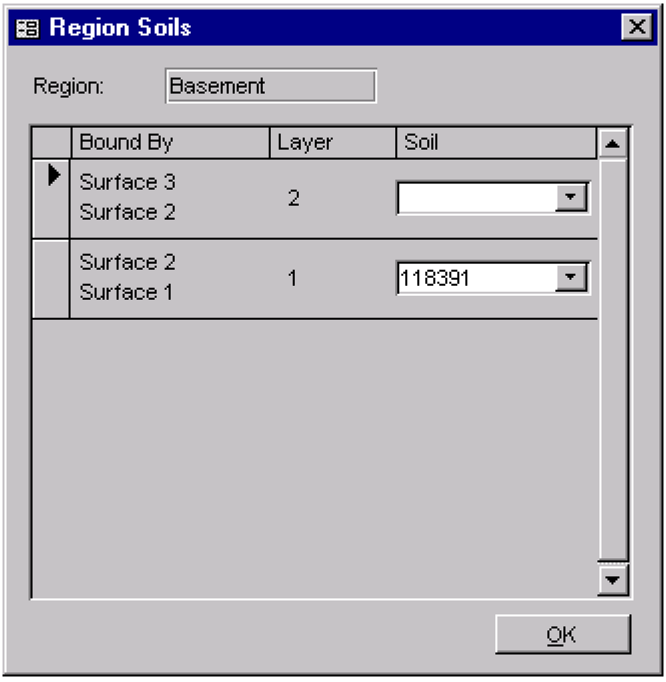
3. Click the **Fill Style** button. The Fill form will open.
4. Select the **Diagonal Up** fill style and press **OK**.
5. Click the **Foreground** color box on the Region Properties form. The Color Palette will appear.
6. Select a **green** color and press **OK**.
7. Click the **Background** color box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

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

2.4.7 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.

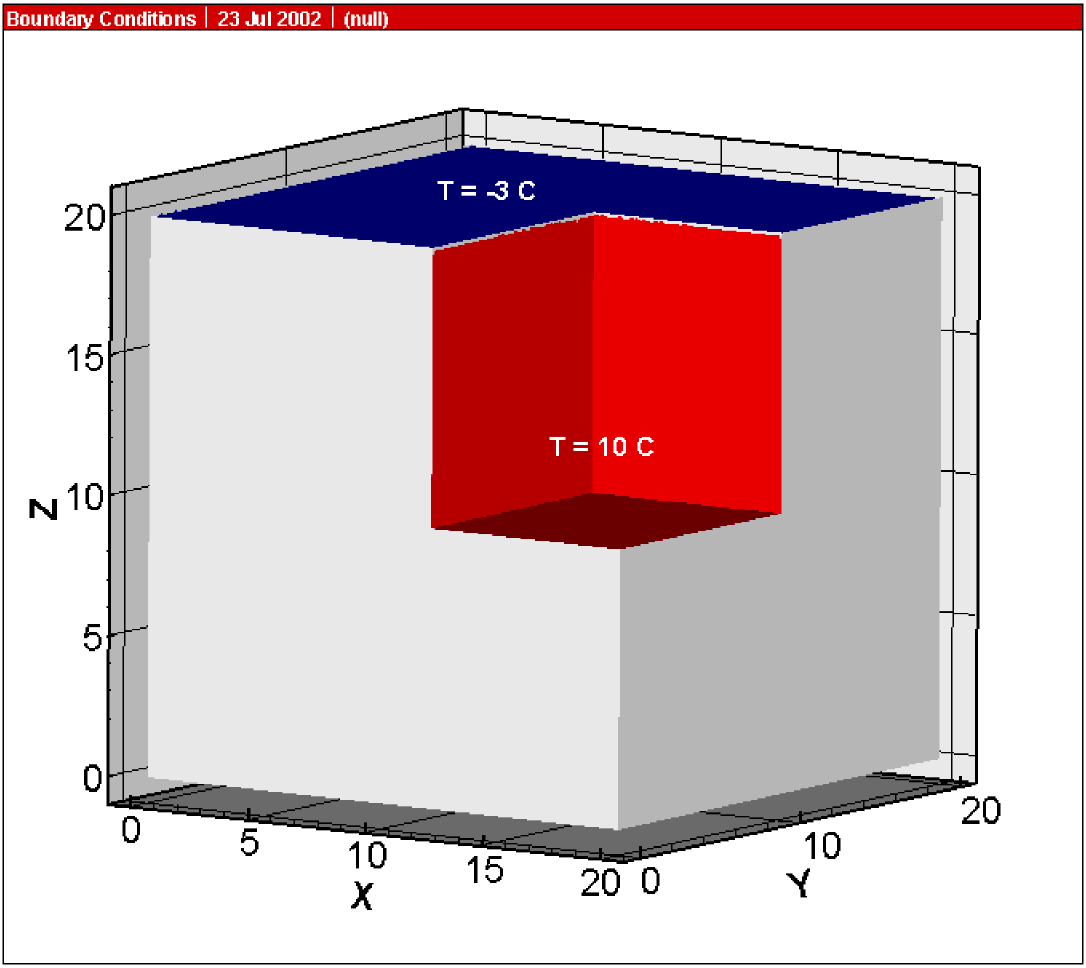
1. Select "**Ground**" in the Region Selector.
2. Press the **Region Soils** button, at the top of the workspace to open the Region Soils form.
3. Select the soil from the drop-down for **Layer 2**. (Note that the Soil Indexes will be different than depicted here)
4. Select the soil from the drop-down 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. Leave the drop-down blank for **Layer 2**.
9. Select the 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 temperature of –3°C will be defined on the Ground region to simulate an outdoor temperature. The basement will be set to a temperature of 10 °C.

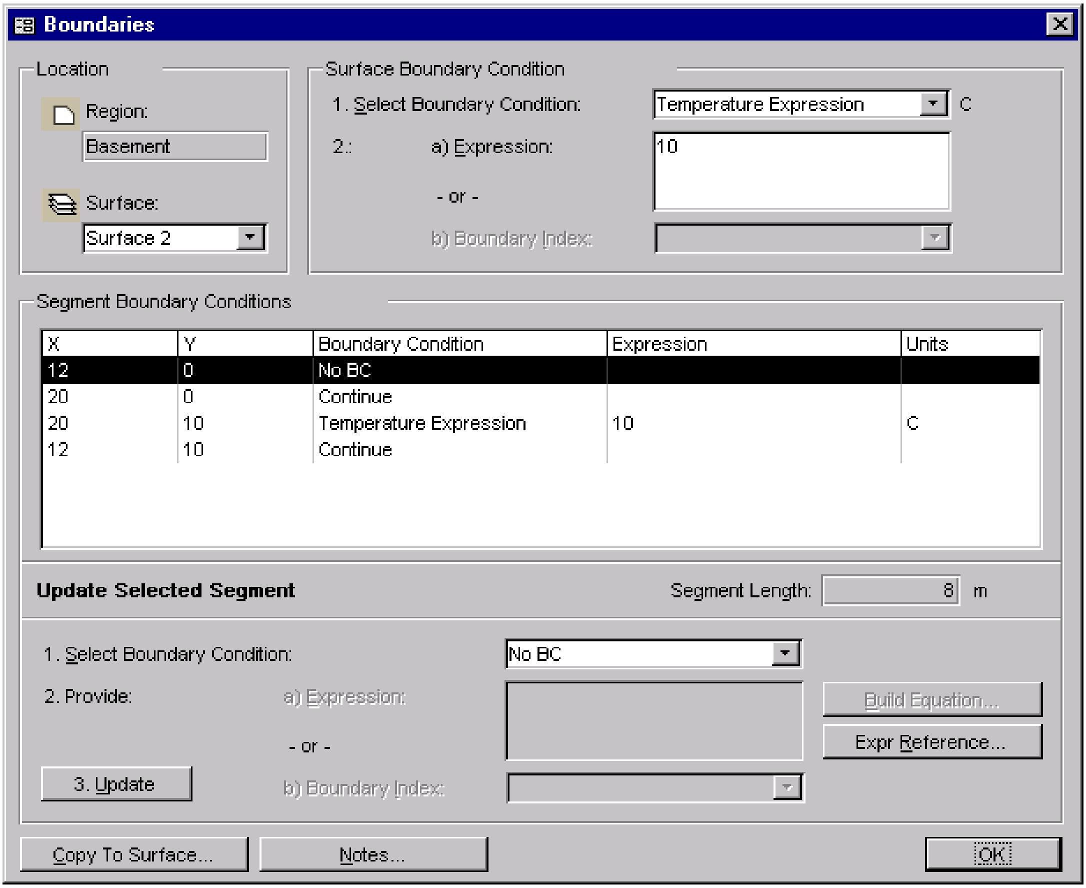


The steps for specifying the boundary conditions are thus:

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

4. Select a **Temperature Expression** boundary condition from the surface boundary condition drop-down box.
5. Enter **–3** in the Temperature Expression field.

6. Select the “Basement” region in the drawing space.
7. From the menu select **Model > Boundaries.** The boundary conditions form will open and display the boundary conditions for Surface 1.
8. Select **Surface 2** from the drop-down box.



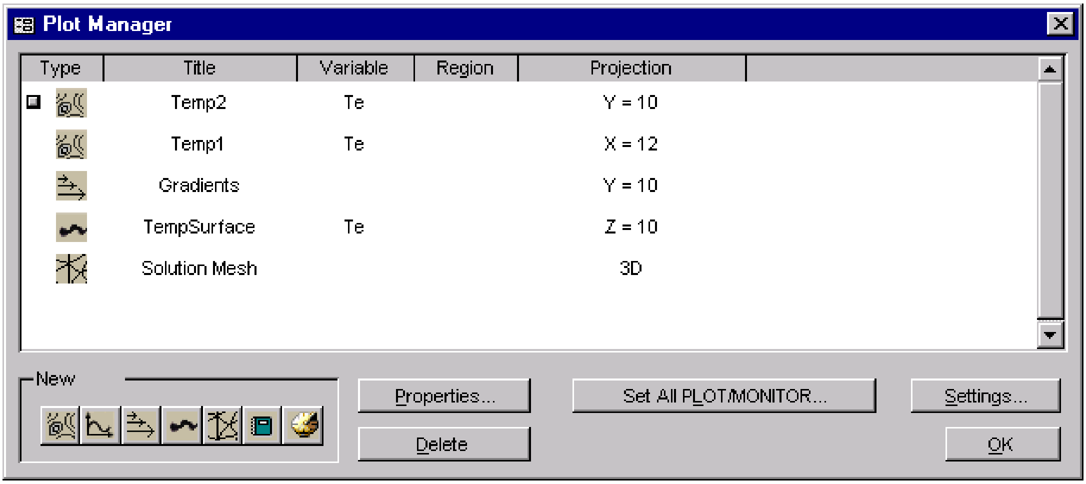
9. Select the **point** (12,0) from the list.
10. From the **Boundary Condition** drop down select a **No BC** boundary condition.
11. Select the **point** (20,10) from the list.
12. From the **Boundary Condition** drop down select a **Temperature Expression** boundary condition. This will cause the Expression box to be enabled.
13. In the Expression box enter a **temperature of 10.**
14. Click the **Update** button to save the boundary condition to the list.

15. Select a **Temperature Expression** boundary condition from the surface boundary condition drop-down box.
16. Enter **10** in the Expression field.
17. Press OK to close the form.

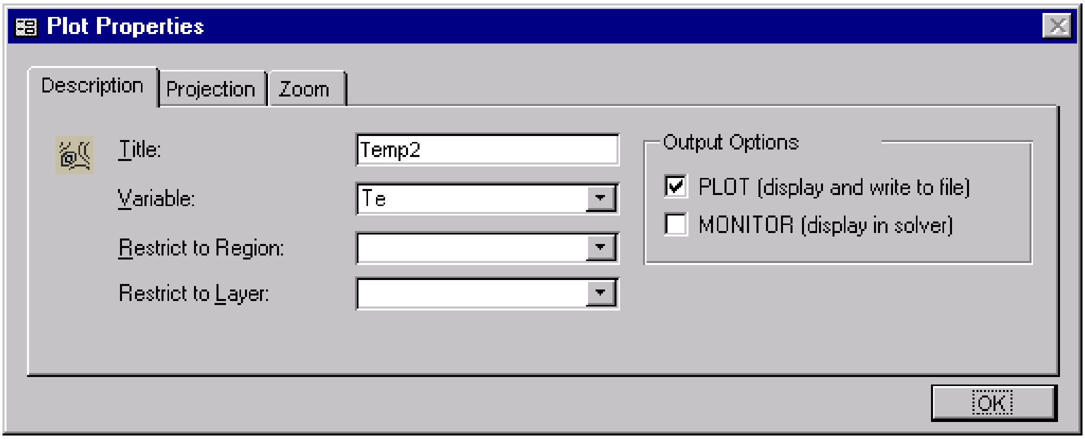
2.5 SPECIFY PLOTS

There are many plot types that can be specified to visualize the results of the model. A few will be generated for this tutorial example problem including a plot of the temperature contours on both an X and Y plane, gradient vectors, temperature surface plot, and the solution mesh.

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.

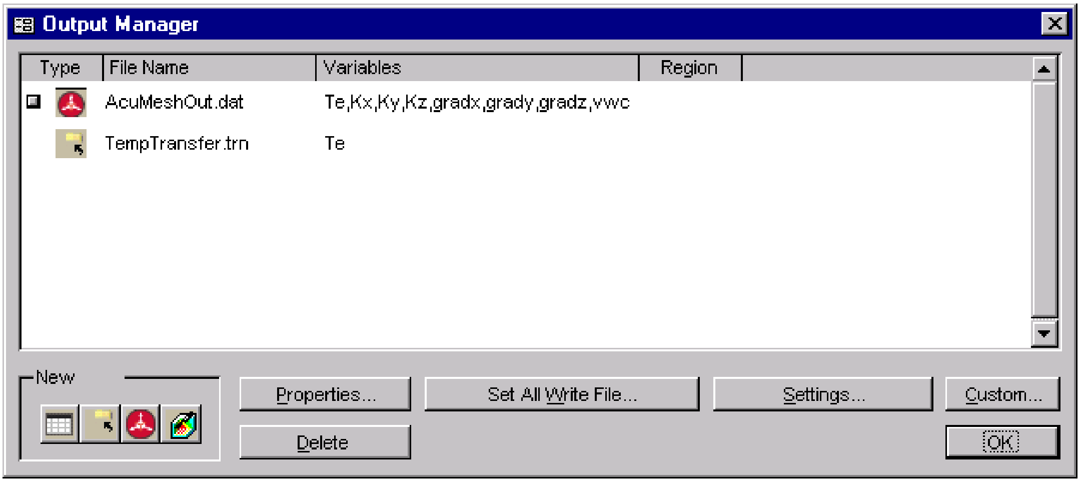


3. Enter the title **Temp2**.
4. Select **Te** as the variable to plot from the drop-down.
5. Move to the **Projection** tab.
6. Select **Plane** Projection option.
7. Select **X** from the Coordinate Direction drop-down.
8. Enter **12** in the Coordinate field. This will generate a 2D slice at X = 12m on which the temperature contours will be plotted.
9. Select the **PLOT** output option.
10. Click OK to close the form and add the plot to the list.
11. Repeat these steps 2 – 9 to create the plots shown above. Note that the Mesh plots do not require entry of a variable.
12. Click **OK** to close the Plot Manager and return to the workspace.

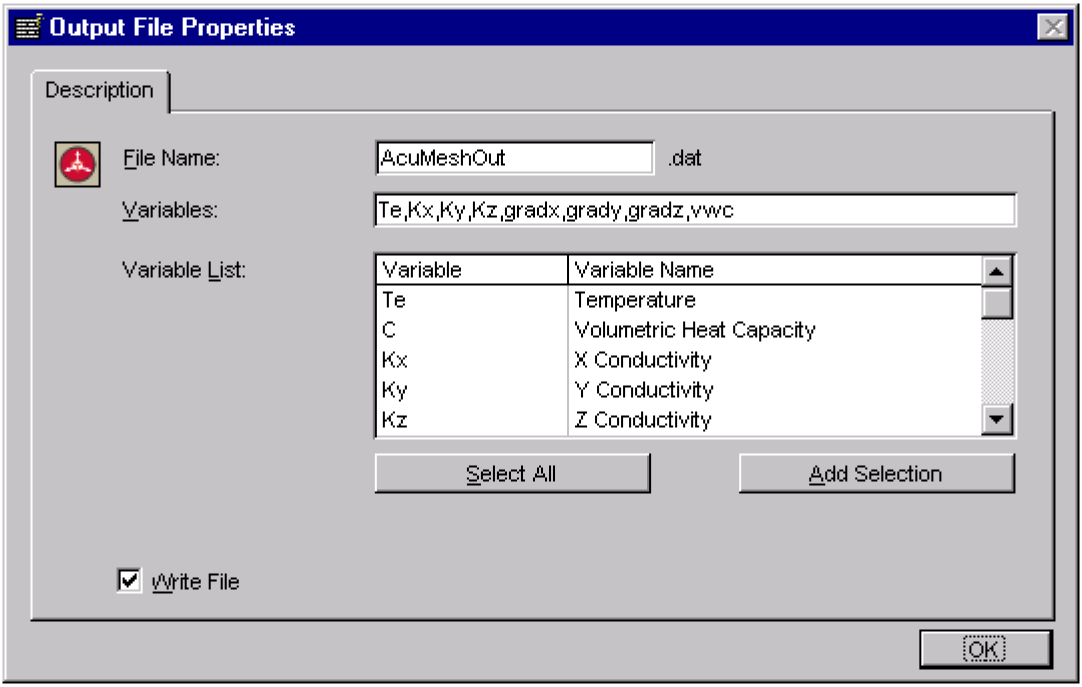
2.6 SPECIFY OUTPUT FILES

There are 4 output file types that can be specified to export the results of the model. Two will be generated for this tutorial example problem: a transfer file of temperatures, and a plot to transfer the results to AcuMesh. Note that the file TempTransfer.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 AcuMeshOut.
4. Select all the variables in the variable list.
5. Press the **Add Selection** button.

6. Check the **Write File** box.
7. Click OK to close the form and add the output file to the list.
8. Click **OK** to close the Output Manager and return to the workspace.

2.7 ANALYZE

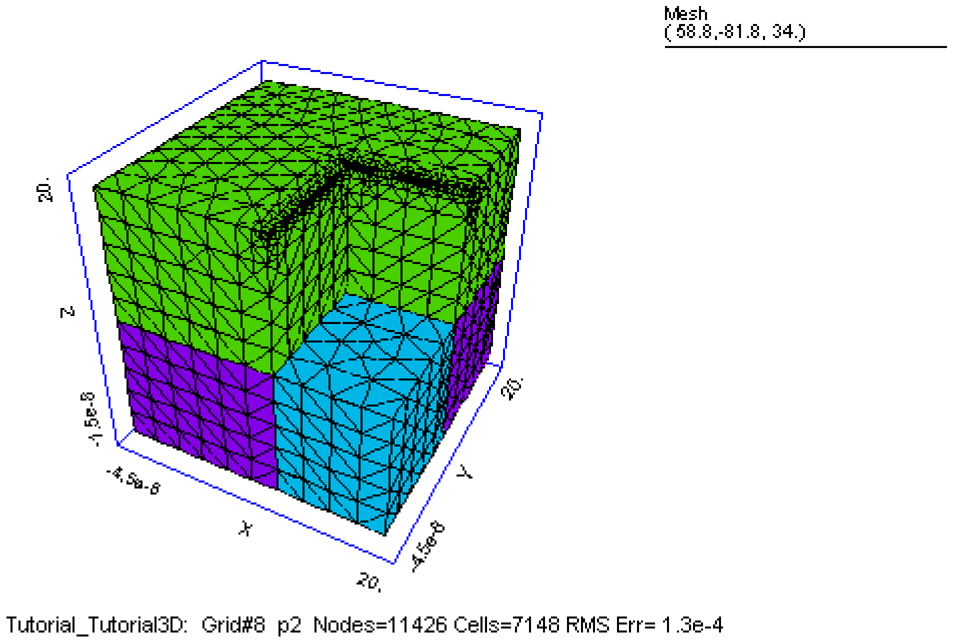
The next step is to analyze the problem. Click the Analyze button located on the left of the workspace. This action will write the descriptor file and open the SVHEAT solver. The solver will automatically begin solving the problem.

When the Regrid Limit message appears click No and the solver will begin generating the plots.

2.8 RESULTS

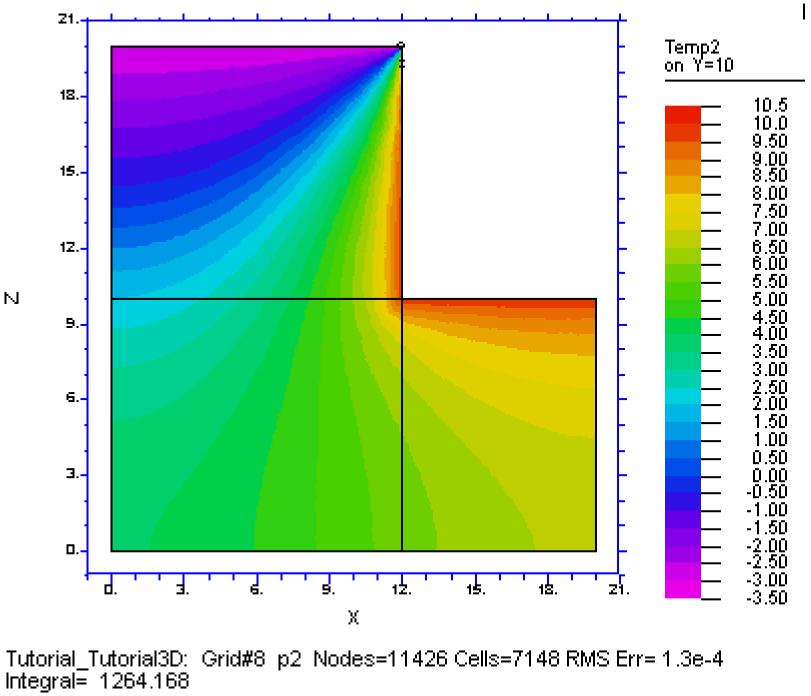
After the problem has finished solving, the results will be displayed in the form of thumbnail plots within the SVHEAT solver. Right-click the mouse and select Maximize to enlarge any of the thumbnail plots. This section will give a brief analysis for each plot that was generated.

2.8.1 Solution Mesh



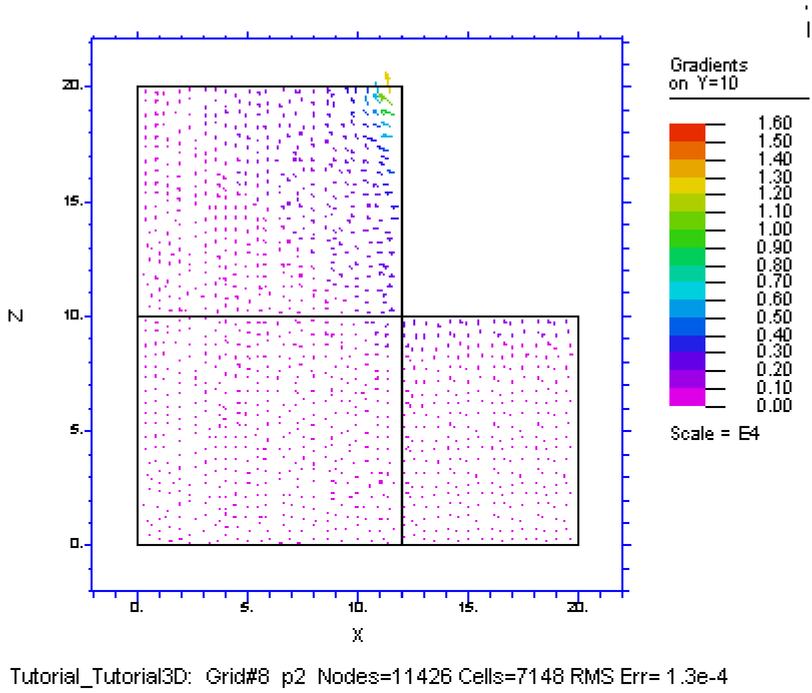
The Mesh plot displays the finite-element mesh generated by the solver. The mesh is automatically refined in critical areas. Right-click on the plot and select Rotate to enable the rotate window.

2.8.2 Temperature Contours



The influence of the basement heat in the surrounding soil is represented here.

2.8.3 Heat Flow Vectors



Gradient Vectors show both the direction and the magnitude of the heat flow at specific points in the problem. Vectors illustrate that flow is away from the basement as expected.

3 REFERENCES

Couttes, R.J. and J.-M. Konrad, (1994). Finite element modeling of transient non-linear heat flow using the node state method. Ground Freezing 94. Balkema, Rotterdam, Netherlands, pp. 39-47.

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

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

Harlan, R.L. and J.F. Nixon, (1978). Ground Thermal Regime. In Geotechnical Engineering for Cold Regions, eds. O.B. Andersland and D.M. Anderson, pp. 103-163.

Pentland, J.S. (2000). Use of a General Partial Differential Equation Solver for Solution of Heat and Mass Transfer Problems in Soils, University of Saskatchewan, Saskatchewan, DC.

Appendix A - Data

4 3D EXAMPLE PROBLEM DATA

Region Shape Data for Tutorial – Tutorial3D:

Ground Region

X	Y
0	0
12	0
20	0
20	10
20	20
12	20
0	20
0	10

Basement Region

X	Y
12	0
20	0
20	10
12	10

5 INDEX

This page is intentionally left blank.