SOILVISION 10 Help Manual - 12/16/2019

2D Partition Model

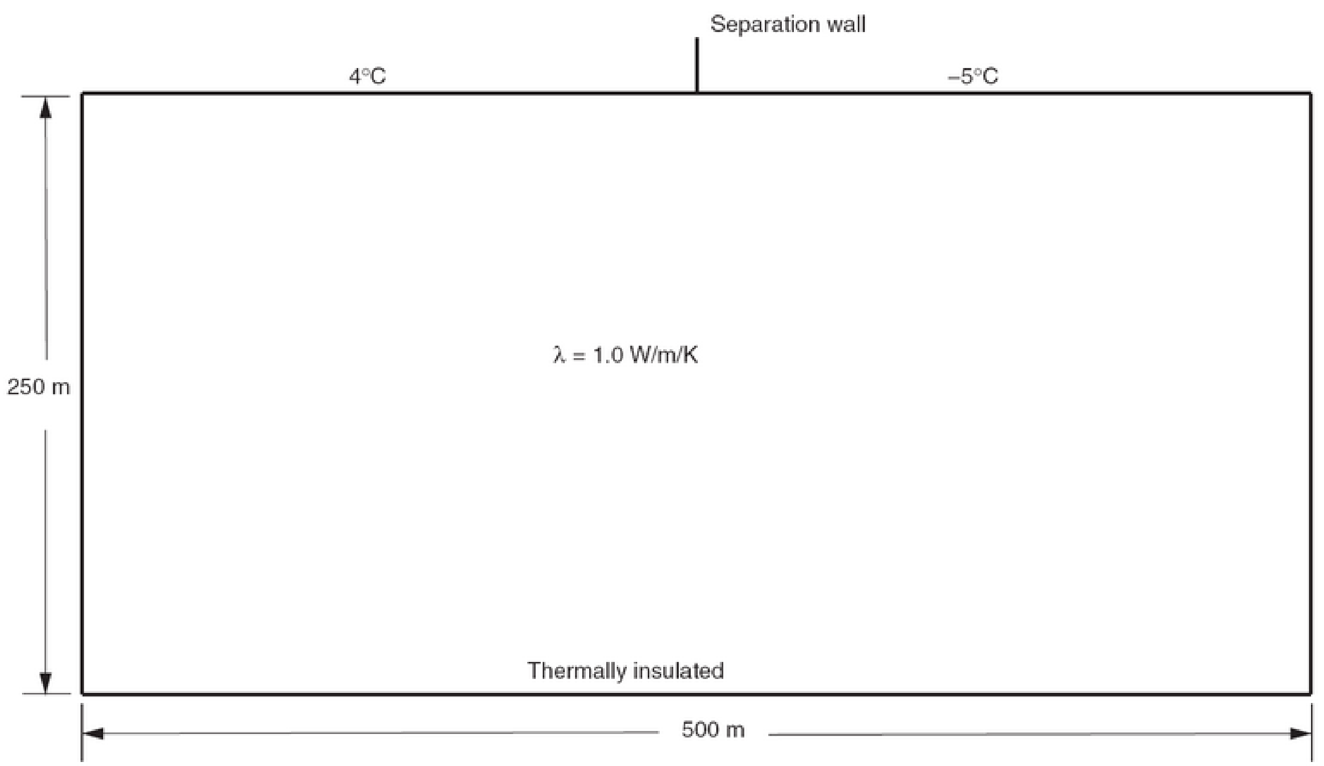
The following example demonstrates how to setup and analyze a two dimensional area that models the steady-state conditions between two adjacent areas with surface temperatures of 4 °C and -5 °C. The results from SVHEAT SVENVIRO will be compared to the results of an analytical solution published by Harlan and Nixon (1978).

Project:	USMEP_Textbook
Model:	HarlanNixon1978
System:	2D
Type:	Steady-State

Minimum license required to complete this tutorial: 2D SVENVIRO

Model Geometry and Description

The model geometry is composed of a single rectangular area with width 500 m and depth 250 m. The model is setup to represent two adjacent semi-infinite areas. The division between the left and right sides is the midpoint of the geometry width (250 m). The left side surface temperature is set to 4 °C and the right side surface temperature is set to -5 °C. The left side represents the inner portion of a simulated heated building and the right side represents the outdoors.



SOILVISION 10 Help Manual - 12/16/2019

Model Setup

The following steps will be required to set up this model:

a. Create model
b. Enter geometry
c. Apply material properties

d. Specify boundary conditions
e. Specify model output
f. Analyze model
g. Results

> **NOTE:**
> Any values on the dialogs there are not specifically mentioned in the steps below are assumed to be the default values currently present.

a. Create Model

The following steps are required to create the model:

1. Open the SOILVISION Manager dialog.
2. In LEARNING MODE, select the SVHEAT module icon and click New Model. The model is automatically stored in **MyProject** project.
3. Select the following entries:
   Module: **SVHEAT SVENVIRO**
   System: **2D**
   Type: **Steady-State**
   Units: **Metric**
   Time Units: **Seconds (s)**
   Model Name: **PARTITION**
4. Click the *OK* button to save the model and close the *New Model* dialog,
5. The new model will be automatically added to the models list and the new model will be opened.

b. Enter Geometry (Geometry)

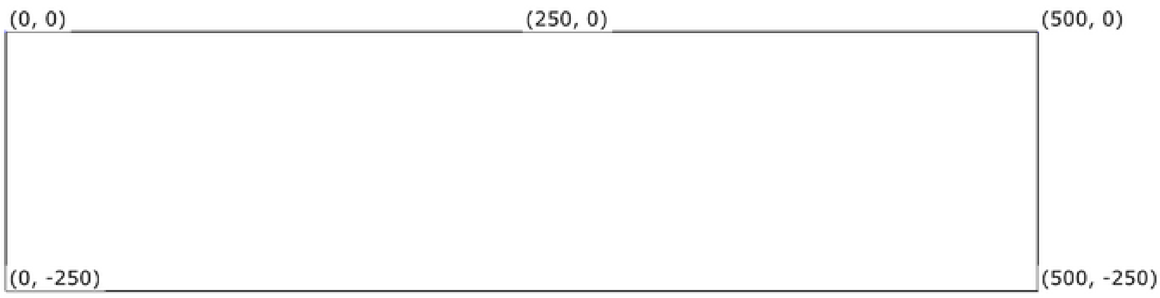
Model geometry is defined as a set of regions and can be either drawn by the user or defined as a set of coordinates. This model consists of a single region. The user may enter geometry by i) drawing on the CAD, ii) using the dynamic input method or they may or they may iii) cut and paste data. Each option is presented below.

- **CAD Drawing**
  1. Select *View > World Coordinate Systems*,
  2. Select **Manual** in the *World Coordinate System* dialog,
  3. Enter the coordinates as shown in the table below,

	Minimum	Maximum
X	0	500
Y	-250	0

  4. Click OK to close dialog,
  5. Select *Geometry > Draw Region Polygon* to draw regions as show below (perform the drawing in counter-clockwise order),
  6. Double click to complete the region drawn,

  Draw the Ground Region according to the following figure:



- **Dynamic Input**

  Alternatively, the regions can be created by using the dynamic input method. Follow these steps:
  1. Ensure that Dynamic Input is turned ON in the task bar,
  2. Select *Geometry > Draw Region Polygon*, the user will see coordinate values that change as the mouse is moved,
  3. Enter **0** as the X coordinate for the first point,
  4. Press the Tab key on your keyboard to move to the Y coordinate,
  5. Enter **0** as the Y coordinate for the first point,
  6. Press the Enter key on your keyboard to finish point 1,
  7. **Repeat** the steps 3-6 to enter all the data points in the Ground Region table below,
  8. Use Shift + Enter after the last point to create region,

- **Cut and Paste**

  Alternatively, the regions can be created by cutting and pasting data from the tables below. Follow these steps:
  1. Open the *Regions* dialog by selecting *Geometry > Regions...* from the menu,

2. Change the region name from *R1* to ***Ground.*** To do this, highlight the name and type the text,

The shapes that define each material region will now be created. The steps to create the *Ground* region are as follows:

1. Click on the ***Ground*** region item in the region list box and press the ***Properties...*** button,
2. Click on the ***New Polygon...*** button to open the *New Region Polygon* dialog,
3. Copy and paste the region coordinates from the table below into the dialog using the ***Paste Points*** button,
4. Press *OK* to close the dialog.

Region: Ground

X (m)	Y (m)
0	0
0	-250
500	-250
500	0
250	0

c. Apply Material Properties (Materials > Manager)

The next step in defining the model is to enter the material properties. In this case we assume that the user has measured the *Conductivity, Volumetric Heat Capacity, SFCC,* and *VWC* for the material. The properties are found in the table below. Define a material called *Soil* as follows:

Tabs	Parameters	Material
		Soil
Conductivity	Thermal Conductivity	1
Volumetric Heat Capacity	Heat Capacity	1950000
SFCC	From (Tef) °C	-0.01
	To (Tep) °C	-0.5
	SFCC Method	None
VWC	SatVWC (Porosity)	0.35
	VWC	0.35

1. Open the *Materials Manager* dialog by selecting *Materials > Manager ...* from the menu,
2. Click the ***New...*** button to create a material,
3. Enter ***Soil*** for the material name,
4. Click *OK* and the *Material Properties* dialog will appear.

NOTE:
When a new material is created, the display color of the material can be specified using the Fill Color box in the Material Properties dialog. Any region that has a material assigned to it will display the corresponding material fill color.

Specify *Conductivity:*

5. On the *Conductivity* tab, select **Constant** from the *Thermal Conductivity Option* drop-down,
6. Check the option "**Same value for unfrozen or frozen material**"
7. Enter **1** in the *Unfrozen Material* box,

Specify *Volumetric Heat Capacity:*

8. Move to the *Volumetric Heat Capacity* tab,
9. Check the option "**Frozen VHC Equals Unfrozen VHC**"
10. From the *Heat Capacity Option* select the **Constant** option and enter the **1950000** value.

Specify *SFCC:*

11. Move to the *SFCC* tab,
12. Enter the **Phase Change Temperatures** found in the table above,
13. Set the *SFCC Method* to **None** using the drop down.

Specify *VWC:*

14. Move to the *VWC* tab,
15. Enter the **Volumetric Water Content** values found in the table above,
16. Click *OK* to close the *Material Properties* and *Materials Manager dialogs.*

The material will need to be applied to the model region by following these steps:

1. Open the *Regions* dialog by selecting *Geometry > Regions ...* from the menu,
2. For the ***Ground*** region, select **Soil** from the Material drop down list,
3. Click *OK* to close the regions dialog.

d. Specify Boundary Conditions (Boundaries Conditions)

Now that the model geometry has been defined, the next step is to specify the boundary conditions. A temperature of 4 °C will be applied to the ground surface on the left side of the model and a temperature of -5 °C will be applied to the ground surface on the right side of the model. By default, a No BC boundary condition is applied to the remainder of the model.

The steps for specifying the boundary conditions are as follows:

1. Switch to Line Segments Selection mode by selecting *View > Selection Mode > Line Segments* from the menu,
2. Select the ground surface of the left side of the model by left-clicking the mouse on the line segment,
3. Right-click the mouse and select *Temperature > Constant* from the pop-up menu,
4. In the *Constant* box enter a temperature of **4** °C,
5. Click OK to close the dialog,
6. Select the ground surface of the right side of the model by left-clicking the mouse on the line segment,
7. Right-click the mouse and select *Temperature > Constant* from the pop-up menu,
8. In the *Constant* box enter a temperature of **-5** °C,
9. Click *OK* to close the dialog.

e. Specify Model Output (Results > Graph Manager)

In this model the plots of interest are the temperature throughout the model. For demonstration purposes the temperature along a cross-section of the model will also be plotted. This section covers how the user may output these plots.

1. Open the *Graph Manager* dialog by selecting *Results > Graph Manager...* from the menu,
2. On the *Range* tab select the **Add New Range Graph** button located on the lower left of the dialog,
3. On the Description tab, Enter a Title of **Temp along Y=-125**,
4. Select the **Temperature** from the *variable* drop down list,
5. Move to the *Range* tab and enter the following coordinates:

   X1: **0** Y1: **-125**

   X2: **500** Y2: **-125**
6. Select the *Output Options* tab,
7. Under *Solver Options* select **Display**,
8. Check the **Write .txt File** check box so that the plot is viewable in the Results module,
9. Click *OK* to close the *Graph Properties - Plot* dialog and *Graph Manager* dialog.

The most basic plots have now been defined. As the user becomes familiar with the software additional plots may be created and customized.

f. Analyze model (Solve > Analyze)

The next step is to analyze the model. Select *Solve > Analyze* in the menu. This action will write the solver file and open the **FlexPDE** solver. The solver will automatically begin solving the model.

For more information on FlexPDE click this link: FlexPDE Solver

g. Results (Solve > Results)

The visual results for the current model may be examined by selecting the *Solve > Results* menu option or click on Results icon .

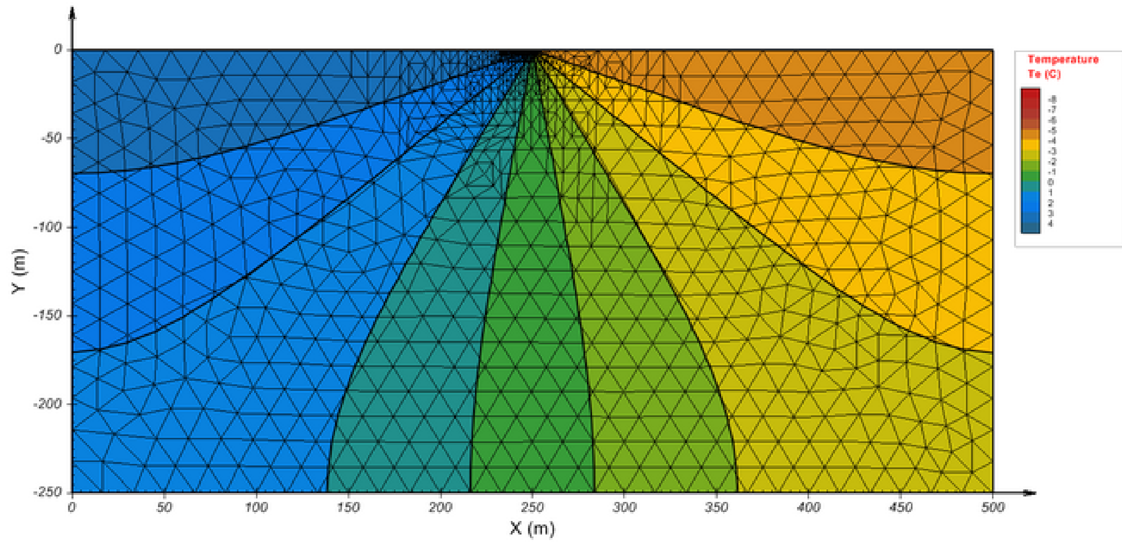
The model results will be displayed. To view the results in more detail proceed to Results and Discussion.

> **NOTE:**
> To transfer from viewing results to the SVHEAT SVENVIRO design module click on the SVHEAT icon found on the left vertical tool bar.

SOILVISION 10 Help Manual - 12/16/2019

Results and Discussion

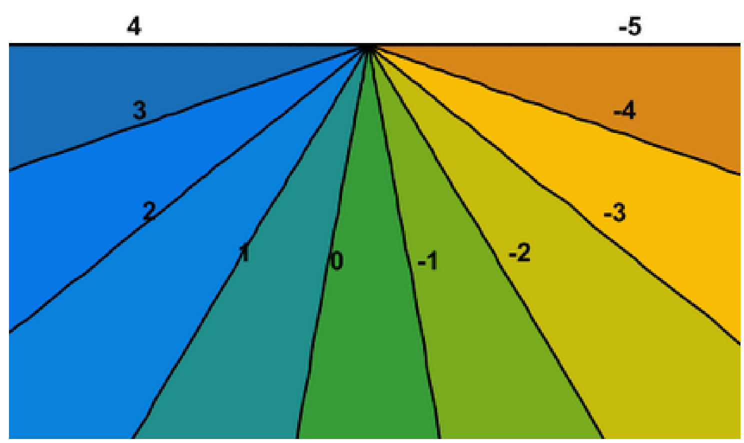
The default plot that appears in Results is a contour plot of the Temperature variable. The finite element mesh used to solve the model is also displayed by default. The effect of automatic mesh refinement can be seen at the midpoint on the ground surface where the temperature value changes from 4 °C to -5 °C.



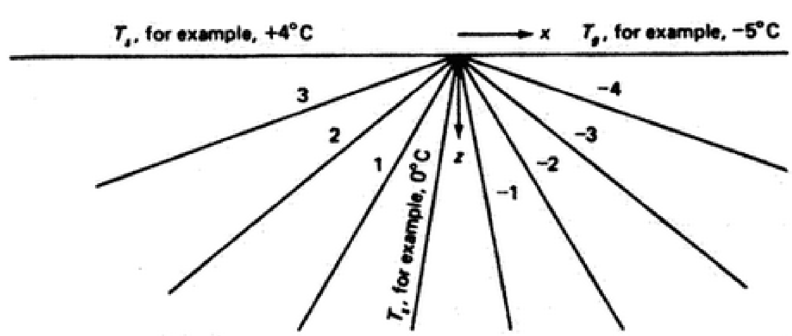
Analytical Solution

Figure 1 and Figure 2 below show the temperature results as produced by SVHEAT SVENVIRO and the analytical solution, respectively. The solutions are in agreement with respect to the location of the freezing front as well as the remaining temperature contours in both the frozen and thawed portions of the material.

1. Select *Plot* > *Contours*,
2. Under the *General* tab in the *Contour Display* section select:
   Show Region Contours
   Show Contour Labels
3. Click *OK* to close the dialog



SVHEAT SVENVIRO Temperature contours



Analytical temperature contours (Harlan and Nixon 1978)