

Essay 5

Tutorial for a Three-Dimensional Heat Conduction Problem Using ANSYS Workbench

5.1 Introduction

The problem selected to illustrate the use of ANSYS software for a three-dimensional steady-state heat conduction problem is exhibited in Fig. 5.1.

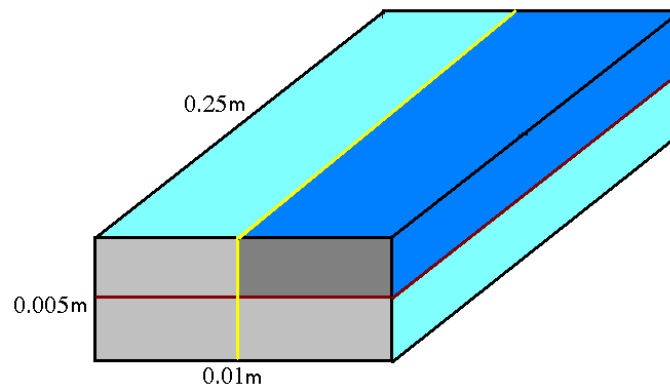


Fig. 5.1 Geometry of the selected three-dimensional solid for the heat conduction analysis

The front face of the figure, shown in gray tones, is heated uniformly by a paste-on heating element. The heat flux q is uniform over that face. All of the other five faces of the solid exchange heat by convection with the surrounding air environment. The temperature of the air is 22°C . The convective heat transfer coefficients on the two vertical sides are equal and have values of $25 \text{ W/m}^2 \cdot ^{\circ}\text{C}$. On the vertical face at the far end of the solid, the heat transfer coefficient is $31 \text{ W/m}^2 \cdot ^{\circ}\text{C}$. The top and bottom faces of the solid have a heat transfer coefficient of $19 \text{ W/m}^2 \cdot ^{\circ}\text{C}$.

The fact that the top and bottom faces have the same heat transfer coefficient means that the temperature of the solid is symmetric about a horizontal plane that bisects the height of the solid. That plane is indicated by the red lines in the figure. Similarly, the fact that the two sides have identical heat transfer coefficients creates a symmetry plane identified by the yellow lines. These planes subdivide the solid into four quadrants such that the temperature solution in each quadrant is identical. Therefore, it is only necessary to solve the heat conduction problem in one of the quadrants. The upper-right-hand quadrant will be selected for this purpose and is displayed in Fig. 5.2.

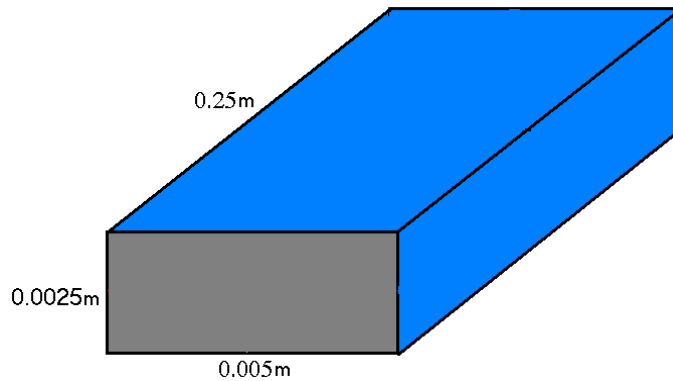


Fig. 5. 2 Upper-right-hand quadrant of the solid of Fig. 5.1

For this newly defined geometry, it is appropriate to restate the boundary conditions. Of particular note are the bottom and left-hand faces of this new geometry. Both of these faces are symmetry planes. Since the temperatures above and below a symmetry plane are equal, there can be no heat transfer across the bottom face of the solid of Fig. 5.2. A similar conclusion follows for the left-hand face. As was stated in Essay 4, a no-heat-transfer surface of a solid is the default boundary condition for the numerical solution.

The boundary conditions for the front face, $q = \text{constant}$, remains as before. On the right-hand face, $h = 25 \text{ W/m}^2 \cdot ^\circ\text{C}$, and on the end face, $h = 31 \text{ W/m}^2 \cdot ^\circ\text{C}$. Also, as before, the value of h on the top face is $19 \text{ W/m}^2 \cdot ^\circ\text{C}$.

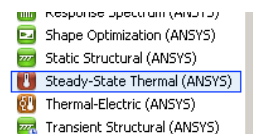
The desired results to be extracted from the ANSYS solution are:

- A color-contour diagram showing the temperature distributions on the top, right-hand side, and far-end faces
- The temperature distribution along the length of the solid at the symmetry point which is created by the intersection of the two symmetry planes

5.2 ANSYS Workbench

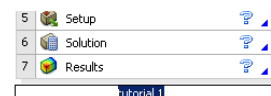
Specifying problem physics:

Select “Steady-State Thermal” from the toolbox to the left and drag it to the project schematic window (inside the green dashed box that has appeared)



Renaming the project:

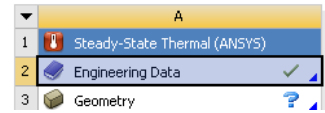
Double-click on the project name, shown at the very bottom of the project schematic, and type “tutorial 1”.



5.3 Engineering Data

Editing engineering data:

Double-click on “engineering Data” or right-click on it, and select “Edit ...”.

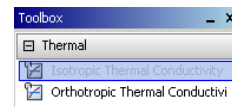


Specifying material properties:

Double click on “Click here to add a new material” and type the new materials name.



From the Toolbox to the left, select “Isotropic Thermal Conductivity” and drag it to the properties list at the bottom of the window.



Enter the conductivity value.

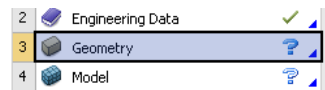
1	Property	Value	Unit
2	Isotropic Thermal Conductivity	174	W m ⁻¹ C ⁻¹

Click on “Return to Project” from the toolbar.

5.4 ANSYS DesignModeler

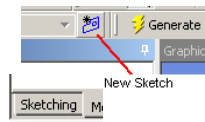
Entering DesignModeler:

Edit the “Geometry” in project schematic by either double-clicking on it, or right-clicking and selecting “Edit...”.



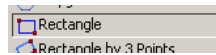
Creating the two-dimensional sketch:

Select “XY Plane” from the tree outline to the left, and click on the

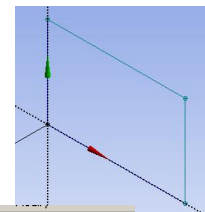


“New Sketch” icon in the toolbar.

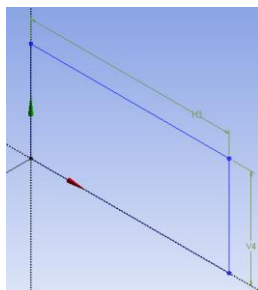
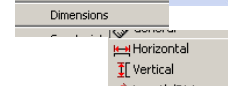
Select the Sketching tab.



Draw a rectangle using the “Rectangle” tool.



Select the “Dimensions” tab, and use “Horizontal” and “Vertical” tools to assign the correct dimensions to the rectangle.



Dimensions: 2	
H3	0.005 m
V4	0.0025 m

Creating the three-dimensional block:

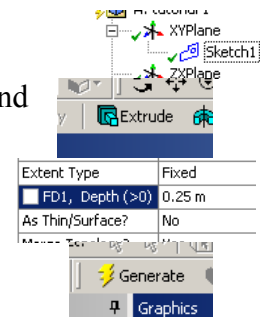
From the “Tree Outline” in the “modeling” tab, select the sketch, and

click on “Extrude” to add an extrusion to the geometry.

In the appeared “Details View” enter the “Depth” value.

Click on “Generate” icon from the toolbar.

Close the DesignModeler window.



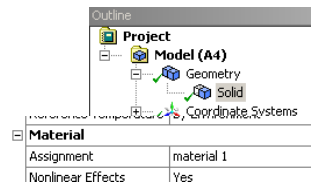
5.5 ANSYS Mechanical

Entering the ANSYS Mechanical:

From the project schematic, double-click on “Model”.

Selecting the material:

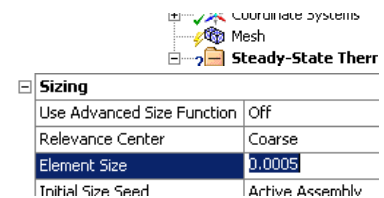
From the “Outline” tree view, select the created solid, and select the desired material from “Details of “Solid”” window.



Specifying element size:

Select “Mesh” from the “Outline” tree view.

Enter the desired element size in the “Details of “Mesh”” window.



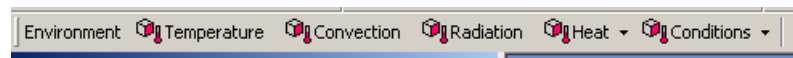
Generating the mesh:

From the toolbar select “Generate Mesh”.



Applying boundary conditions:

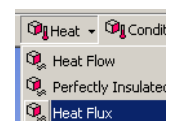
Select the analysis from the Outline tree view, and the toolbar will change to display all the available boundary conditions.



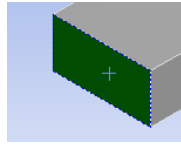
On the symmetry surfaces, there is no need to apply boundary conditions because, by default, the no-heat transfer condition is automatically applied.

Applying heat flux BC:

Select “Heat” then “Heat Flux” from the toolbar.



Select the front face > click on “Apply” > enter the heat flux value.



Scope	
Scoping Method	Geometry Selection
Geometry	Apply
Definition	

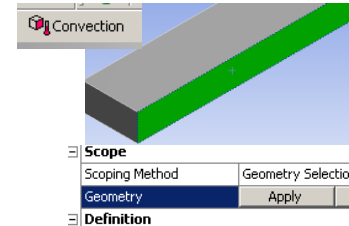
Definition	
Type	Heat Flux
Magnitude	1000
Suppressed	No

Applying convection BCs:

From the toolbar, select “Convection”.

Select the right-hand face and then click on “Apply”.

Enter the heat transfer coefficient as “Film Coefficient”.



Geometry	1 Face
Definition	
Type	Convection
Film Coefficient	25
Ambient Temperature	22. °C (ramped)
Suppressed	No

On the top and far-end faces, repeat the steps that were outlined for the right-hand face.

5.6 ANSYS Solver

Select Solve from the toolbar.



(This starts the solution)

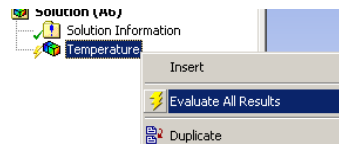
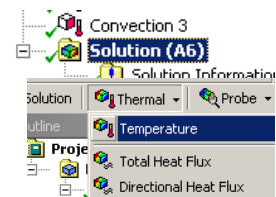
5.4 ANSYS Post-Processor

Displaying color-contour diagrams of temperature

Select the solution.

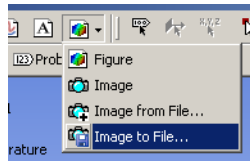
From the toolbar, select “Thermal” > Select “Temperature”.

Right-click on the newly-created “Temperature” item in the outline view, and select “Evaluate All Results”.



Rotate the geometry using the graphic tools on the right-hand side panel to reveal the appropriate faces. (top, far-end, and right-hand side respectively, one at a time)

In order to save the resulting picture as an image file, select the image icon, and then “Image to File”, then type in the file name, and select the file type.



(The colors that appear on a color-contour diagram are keyed to a color stripe displayed at the left-hand side of the diagram. The temperature range exhibited are the lowest and highest temperatures obtained by the solution. Usually, there are nine color blocks in the stripe.)

Displaying the temperature distribution along the symmetry axis of the solid

In order to plot the temperature variation along the symmetry axis of the solid, we will create a “path” on the axis, and then “map” temperature values onto the path, and then “plot” the resulting map.

Defining the path

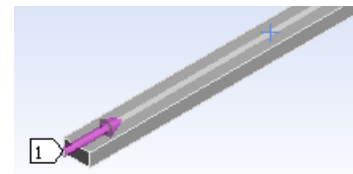
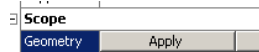
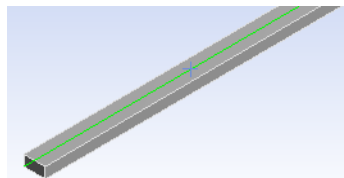
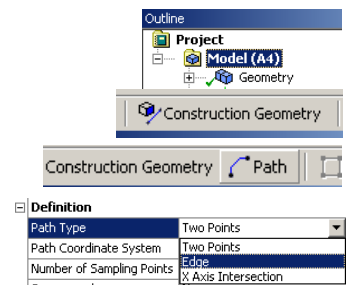
Select the Model.

Select “Construction Geometry” from the toolbar.

Select “Path” from the now-changed toolbar.

Change the path type from “Two points” to “Edge”.

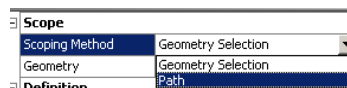
Select the symmetry axis, and click on “apply”.



Mapping the temperature distribution on the path

Select the solution then, very similar to contour plot, select “Thermal” and then “Temperature”.

Change the “scoping method” from “Geometry Selection” to “Path”.



Select the created path as “Path”.

Scoping Method	Path
Path	
Geometry	Path
Definition	

Displaying the graph

Right-click on the newly-created object under the solution, and select “Evaluate All Results”.

