Essay 5

Tutorial for a Three-Dimensional Heat Conduction Problem Using ANSYS

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.

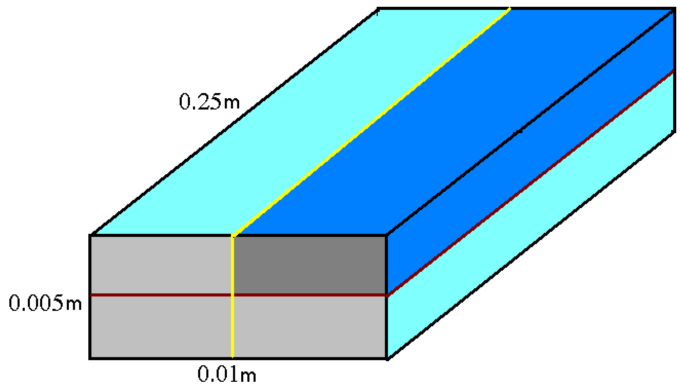

Fig. 5. 1 Geometry of the selected three-dimensional solid for the heat conduction analysis

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 $22℃$. The convective heat transfer coefficients on the two vertical sides are equal and have values of $25\, W/m^2 \cdot ℃$. On the vertical face at the far end of the solid, the heat transfer coefficient is $31\, W/m^2 \cdot ℃$. The top and bottom faces of the solid have a heat transfer coefficient of $19\, W/m^2 \cdot ℃$.

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.

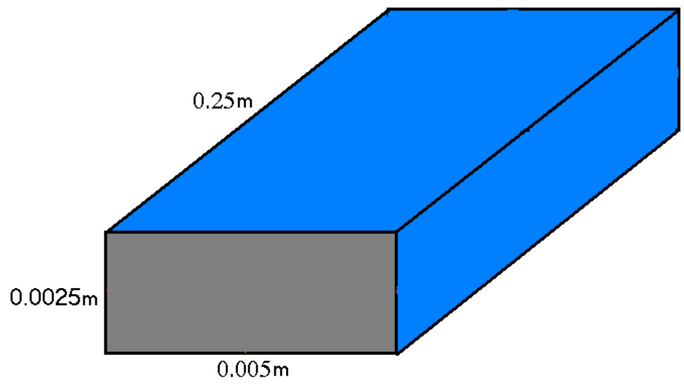

Fig. 5. 2 Upper-right-hand quadrant of the solid of Fig. 5.1

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, $q = constant$, remains as before. On the right-hand face, $h = 25\ W/m^2 \cdot$ °C, and on the end face, $h = 31\ W/m^2 \cdot$ °C. Also, as before, the value of h on the top face is 19 $W/m^2 \cdot$ °C.

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

(a) A color-contour diagram showing the temperature distributions on the top, right-hand side, and far-end faces
(b) 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 Pre-Processor

Specifying problem physics:

Preferences

… check "Thermal" > click on "OK"

Specifying element type:

Preprocessor > Element Type > Add/Edit/Delete

… click on "Add …"

… select “Thermal Mass – Solid” > Select “Brick 8node 70” > click on “OK” > click on “Close”

(Brick 8node 70 is for three-dimensional heat conduction. For two-dimensional analysis Quad 4node 55 may be used.)

Setting material properties:

> Material Props > Material Models

… select “Thermal” > select “Conductivity” > select “Isotropic” > enter k value > click on “OK” > select “Exit” from “Material” menu

Creating the geometry:

> Modeling > Create > Volumes > Block > By Dimensions

… enter coordinates of two opposite corners > click on “OK”

Specifying element sizing:

> Meshing > Size Ctrls > Manual Size > Global > Size

… enter element edge length > click on “OK”

(This meshing choice gives cubic elements.)

Creating the mesh:

> Mesh > Volumes > Mapped > 4 to 6 Sided

… pick the volume > click on “OK”

Applying the boundary conditions:

Two paths:

> Preprocessor > Loads > Define Loads

> Solution > Define Loads

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:

On the front face

> Apply > Thermal > Heat Flux > On Areas

… pick the front face > click on “OK”

… enter the heat flux value, $1000\ W/m^2$ > click on “OK”

Applying convection BCs:

On the far-end face

> Apply > Thermal > Convection > On Areas

… pick the far-end face > click on “OK”

… enter heat transfer coefficient in the “Film Coefficient” text box

> enter the ambient temperature in the “Bulk Temperature” text box

> click on “OK”

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

5.3 ANSYS Solver

Solution > Solve > Current LS (Load Step)

… review the status report presented by the software and then click on “OK”

(This starts the solution)

5.4 ANSYS Post-Processor

Displaying color-contour diagrams of temperature

General Postproc > Plot Results > Contour Plot > Nodal Solu

… select “DOF Solution” > select “Nodal Temperature” > click on “OK”

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 “PlotCtrls” from the main menu and then from the drop-down menu, select “Hard Copy” and select “To File …” from the menu.

… Specify file format (use JPG to get a more compressed file) > check “Reverse Video” to have an image with bright background > enter the file name > click on “OK”

(The colors that appear on a color-contour diagram are keyed to a color stripe displayed at the bottom 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. To change the range of the temperatures that are displayed at respective ends of the stripe, the following steps can be performed.)

PlotCtrls > Style > Contours > Uniform Contours

… Specify the number of intervals (or the "contour value incr" at the bottom of the window) > Specify the minimum and maximum values of the desired temperature range > make sure the "user specified" is checked > click on "OK"

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

> Path Operations > Define Path > By Nodes

… select the two end points of the path (it is easiest to select these points by clicking near them on the graphical window. We may have to rotate and/or move the object in order to be able to see the axis displayed.) > click on "OK"

… specify the "path name" and "number of divisions" (the higher the number of divisions, the smoother the graph will be.) > click on "OK"

Mapping the temperature distribution on the path

> select "Map onto Path" under the "Path Operations"

… enter the variable name > select "DOF Solution" and "Temperature TEMP" > click on "OK"

Displaying the graph

> Plot Path Items > On Graph

… select the variable created in the previous step > click on "OK"