

Essay 7

Tutorial for a Three-Dimensional, Transient Heat Conduction Problem Using ANSYS

7.1 Introduction

The problem in this essay is similar to the problem in essay 5. However, instead of having a solid block and steady state, we have a block with a water-filled channel through the center of the symmetry quadrant and a transient analysis. There are several additional steps that are necessary to solve a transient problem. The steps covered in the essay 5 will be documented, but not to the degree of detail in essay 5.

We have symmetry on the lower face, left face, and rear face. The dimensions of the symmetry block are:

X: 6 in
Y: 9 in
Z: 15 in

The water cooling channel is centered at (3, 4.5, Z) with a radius of .25 in.

The boundary conditions for the non-symmetric faces are 1.05 Btu/hr ft² on the vertical faces (front face and right face), and 0.85 Btu/hr ft² on the top face. The ambient air is at 40 °F. The material of the rectangular block is steel with conductivity of 30.5 Btu/hr ft °F, specific heat of 0.15 Btu/lbm °F, and density of 410 lbm/ft³. The water in the channel has conductivity of 0.37 Btu/hr ft °F, specific heat of 1.0 Btu/lbm °F, and density of 62 lbm/ft³. The initial condition for the entire system is 400 °F.

7.2 ANSYS Pre-Processor

Select thermal from the preferences option:

Preferences-> Thermal

Expand the Preprocessor.

Add element types. Since this is a 3-D problem we need to use both 2-D elements and 3-D elements. Our 2-D element is a quad 4node 55 and our 3-D element is a brick 8node 70.

Element Type->Add/Edit/Delete->Select element thermal solid ->quad 4node 55, click apply.

Select element brick 8node 70, click OK.

Add materials. Select Material Props->Material Models.

Specify conductivity (iso), specific heat, and density for material 1

(steel) under thermal properties. To add the second material (water), go to Material (on menu bar)-> New Model. Specify conductivity (iso), specific heat, and density for material 2.

Close the Material Models window.

Create the geometry.

We'll start with the rectangular area.

Select Modeling->Create->Areas->Rectangle->By Dimensions.

Enter the dimensions of the rectangle (0,0), (6/12,9/12).

Click OK.

Now we'll create a circle **over** the rectangle.

Select Modeling->Create->Areas->Circle->Solid Circle.

Enter the center point of the circle (6/12/2, 9/12/2) and the radius of the circle (0.25/12). Click OK.

Since we have overlapping areas, we'll have to remove the circle from the rectangle and create a new circle in the resulting space.

To remove the circle from the rectangle we'll use the subtract command. Go to Modeling->Operate->Booleans->Subtract->Areas. The first window asks for the area **from** which the smaller area is to be removed. In our case, this is the rectangle. Select the rectangle and click OK. Now a new window appears asking what area we want to **remove** from the area. In our case this is the circle. Select the circle and click OK. You should end up with a rectangle with a hole in the middle of it. This is our solid steel region. We'll now add the water region to the hole.

Modeling->Create->Areas->Circle->Solid Circle.

Enter the center point of the circle (6/12/2, 9/12/2) and the radius of the circle (0.25/12).

Click OK.

An alternative to subtracting and recreating the circular area is to use the overlap command. Once both areas are created go to Modeling->Operate->Booleans->Overlap->Areas. Select the areas to overlap and click OK. In this case since the only areas we have overlap, you could choose pick all.

Now we have two separate areas. For heat to transfer between each of them they need to be in thermal contact. In ANSYS this means we need to glue these areas together. Modeling->Operate->Booleans->Glue->Areas->Pick all. The two areas are now glued.

Next we need to mesh the areas. After we have meshed the 2-D areas, we will extrude them in the Z direction to complete our 3-D model. Before we can mesh our areas we need to specify the material used in each area. By default material 1 is assigned to all areas. In our case the center water channel is actually material 2. To change this we go to Meshing->Mesh Attributes->Picked Areas and select the circle. A window opens up with several options. In the first pull-down window (Material Number) select 2. Click OK.

Now define an element size using Meshing->Size Cntrls->Manual Size->Global->Size. Be careful to select an appropriate size as 3-D models can be resource intensive (you might run out of nodes pretty easily). Now we can mesh the areas. Select Meshing->Mesh->Areas->Free. Note: We cannot use mapped mesh for areas that are not three or four sided. We have curves which means we are confined to using free meshes or other more elaborate techniques that will be discussed later. Select

pick all to mesh all areas.

To create a volume out of our 2-D mesh we need to extrude it in the Z direction. Go to Modeling->Operate->Extrude->Elem Ext Opts. Most things in this window will be left as default. However, the second to last text box specifies how many layers we will extrude. The number you put here will depend on how many nodes you used on your face. In our license of ANSYS we can have up to 256,000 nodes. To see the maximum value you can put in No. Elem divs, divide 256,000 by the number of nodes on your face. Round down to the nearest integer and subtract one. This is the maximum number you can use. For example, if you have 25,600 nodes on your face you end up with 9 ($256,000/25,600 - 1$). Using this number of nodes will be an intensive calculation, and you probably don't need to use all the available nodes. Use your best judgment, but keep your maximum in mind. Finally, check the **ACLEAR** box at the bottom. This deletes the 2-D mesh once the 3-D mesh is established. Click OK.

To actually extrude the mesh select Modeling->Operate->Extrude->Areas->By XYZ Offset. Pick the areas to extrude (all). Enter the extrusion offsets (0,0,15/12). Click OK. ANSYS is now creating a volume mesh of our model.

We are finished with the Preprocessor.

7.3 ANSYS Solver

Loads are best defined in the Solver as certain transient-specific options are not available in the Loads menu of the Preprocessor. First we need to tell ANSYS what type of simulation this is. Select Solution->Analysis Type->New Analysis. Choose Transient and click OK. A new window appears. Accept the defaults by clicking OK.

Now define loads. Select Define Loads->Apply->Thermal->Convection->On Areas. Choose the vertical faces (Front and Right). Enter the film coefficients for these faces and the ambient temperature of 40 °F. Select Define Loads->Apply->Thermal->Convection->On Areas and choose the top face. Enter the boundary condition here.

Input the initial conditions under Solution->Define Loads->Apply->Initial Condit'n->Define. Choose pick all and enter the initial temperature of the model (400 °F). Click OK.

Next, set up the solution output controls. This is under Solution->Load Step Opts->Output Cntrls->Solu Printout. This selects what is displayed as the solution runs. Select Every substep and click OK. Select Solution->Load Step Opts->Output Cntrls->DB/Results File. Select Every substep and click OK.

Finally we specify the time step options. Go to Solution->Load Step Opts->Time/Frequenc->Time – Time Step. [Time] is the length of the entire simulation. In this example it is two hours. Since our convection coefficients have time units of hours enter two in this box. [DELTIM] is the delta

time of the first time step. Make this small. Remember the units are hours, so 1 minute is 1/60. The ramped v. stepped boundary conditions is an important choice. Stepped applies boundary conditions as specified starting at $t = 0$. Ramped starts with no boundary conditions and ramps up to the specified values over time. Please choose stepped. Select automatic time stepping->on. Specify a minimum time step and a maximum time step. Remember these are in hours. When you are finished click OK.

Select Solution->Solve->Current LS. A summary of the analysis appears as well as a confirmation window. Click OK to begin solving. Depending on your number of nodes it may take a while.

7.4 ANSYS Post-Processor

Transient problems have several sets of results. For each time step a set of results are generated. Results can be read into the Postprocessor from any given time step. To read results into ANSYS select General Postproc->Read Results->By Pick. This opens a window of all available time steps and the time stamp associated with them. Once results are read in, post processing can take place as done previously. Contour plots, path plots, vector plots, and other information can be gleaned from each time step. If results are needed for a specific time and that time does not correspond to one of the time steps do not fear. Select General Postproc->Read Results->By Time/Freq. You can enter anytime in the Time text box. ANSYS will interpolate results for each node by looking at the surrounding time steps.

7.5 ANSYS Time History Post-Processor

Finally, if you want to see how the solution changes at a particular node or element over time use the TimeHist Postpro. You have to add locations by clicking the small green + sign in the upper left corner. You can select various types of outputs including nodal temperature, heat flux, gradients, etc. After choosing the type, you need to select the location. You can graph how this property at this location changes over time by clicking the small graph icon on the top. You can select more than one location and graph several locations over time. In addition, more than one of the aforementioned problems can be plotted on the same graph, either at one selected location or at different selected locations. For example, one could graph both heat flux verse time and temperature verse time at the same location.