Stevens Institute of Technology
ME345 Modeling and Simulation – Prof. Frank Fisher

Temperature distribution in a fin cooled electronic component
**(Carnegie Mellon)**
Last updated October 2015 – Ryan Oldja

Introduction: In this example you will learn to model a cooling fin for electronics. This involves heat generation, conduction and convection.

Physical Problem: All electronic components generate heat during the course of their operation. To ensure optimal working of the component, the generated heat needs to be removed and thus the electronic component be cooled. This is done by attaching fins to the device which helps in rapid heat removal to the surroundings.

Problem Description:

- For the sake of simplicity we assume that the electronic circuit is made of copper with thermal conductivity of 386 W/m K. Also it generates heat at the rate of 10e6 W.
- The enclosing container is made of steel with thermal conductivity of 20 W/m K.
- The fins are made of aluminum with thermal conductivity of 180 W/m K.
- **Units**: Use **S.I. units** ONLY
- **Geometry**: See figure.
- **Boundary conditions**: There is convection along all the boundaries except the bottom, which is insulated. The Film Coefficient is 50 W/m^{2}K and the Bulk Temperature is 20°C.
- **Objective**:
  - To determine the nodal temperature distribution. To determine the
  - maximum value of temperature in the component.

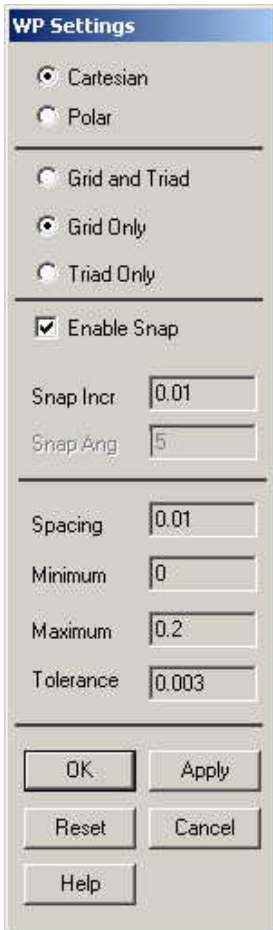- You are required to hand in print outs for the above.

- **Figure**:
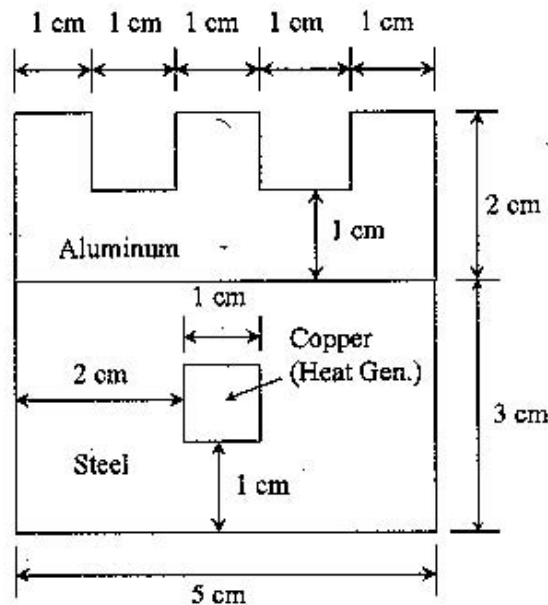
IMPORTANT: Convert all dimensions and forces into SI units.

STARTING ANSYS

Click on ANSYS 11.0 in the programs menu.

MODELING THE STRUCTURE

■ Go to the ANSYS Utility Menu.
■ Click **Workplane>WP Settings**.
■ The following window comes up

■ Check the Cartesian and Grid Only buttons.
■ Enter the values shown in the figure below.
■ If the Cartesian Grid does not appear, click on the Utility Menu **Work Plane>Display Working Plane**
■ Go to the ANSYS Main Menu **Preprocessor>Modeling>Create>Areas>Rectangle>2 Corners**.

- Now we will pick the end points of the rectangles.
- First make the steel rectangle of dimensions .05m X .03m (i.e. 5 units by 3 units on the grid).
  - o To do this, select the work plane origin as the bottom-left corner of the rectangle, then move the cursor 5 grid spaces to the right and 3 grid spaces up and click to place the top-right corner of the rectangle.
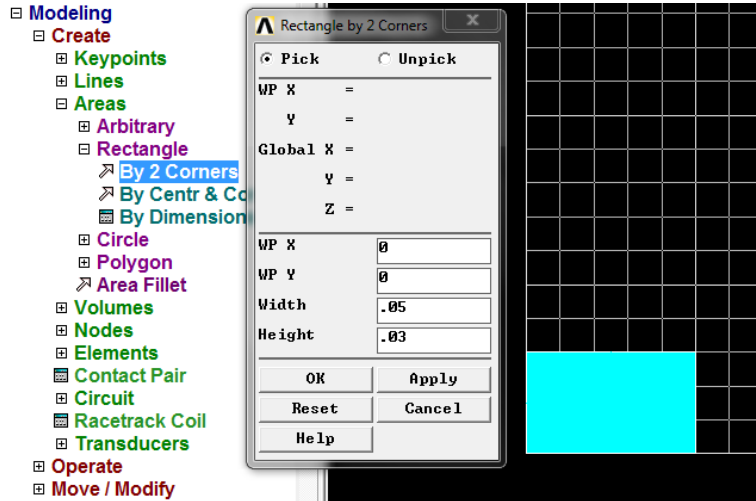

- Next make the copper square of dimensions .01m X .01m.
- Next make the aluminum part by making a rectangle of dimensions .05m X .02m.
- Then create two .01m X .01m squares on top of the aluminum area (these areas will be subtracted from the aluminum in order to create the fins).
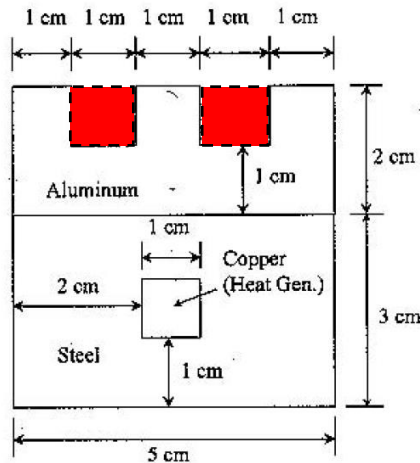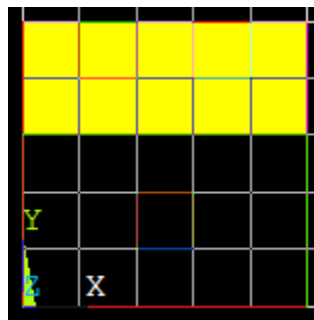
▪ Your model should now look like this:
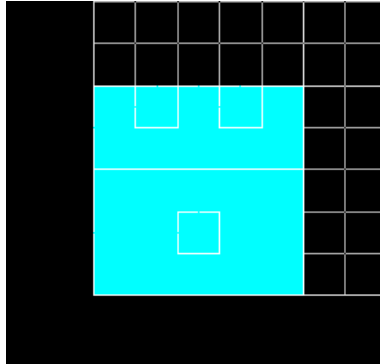
▪ Now, select **Utility Menu>Plot>Lines**
▪ To subtract the two square areas to make the fins:
  o **Modeling>Operate>Boolean>Subtract>Areas**. Notice the prompt at the bottom-left of the screen, which says to 'select the area from which to subtract'
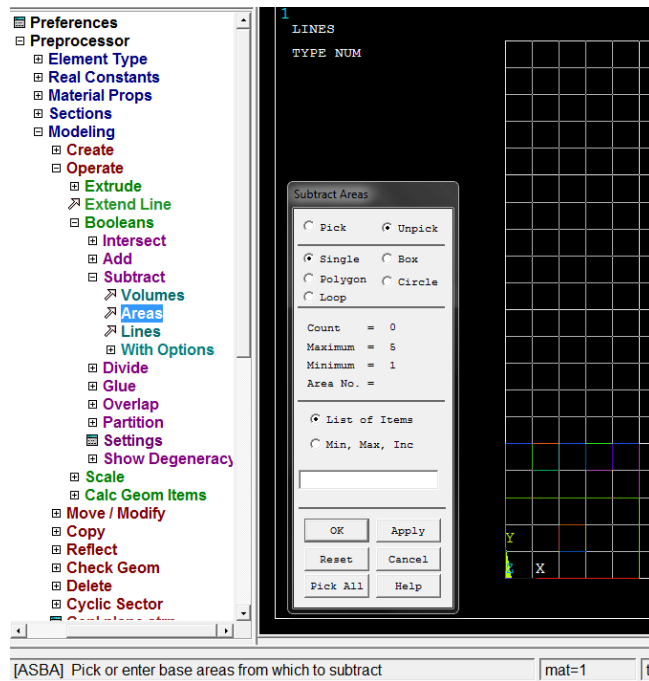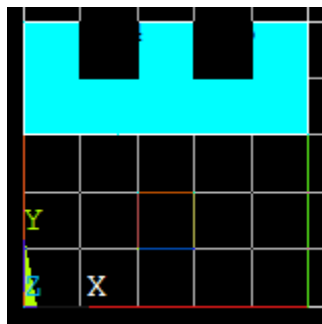

  o Select the entire .05m X .02m aluminum area. This is the base area. Click 'OK'

- Now, select the two .01m X .01m squares. These are the areas that will be subtracted from the base area. Click 'OK'
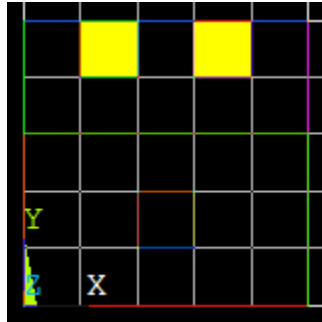

- The fins are now created.

- From Preprocessor, choose **Modeling>Operate>Boolean>Overlap>Areas**
  - Select the Steel area. A window may open warning you that there are two areas in this location (steel and copper). Use 'Prev' and 'Next' to cycle through areas until the correct area is highlighted. Click 'OK' in the warning window.
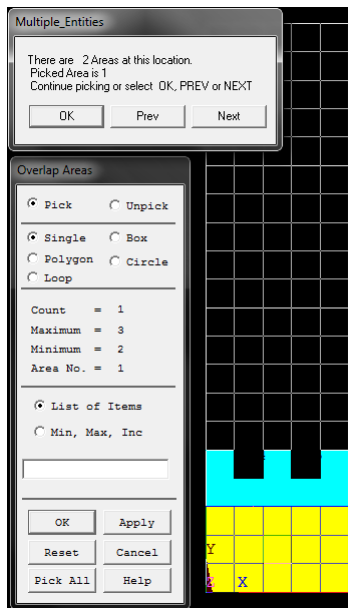
- o Select the Copper area (once selected, the copper square will become black again; ensure 'Count = 2'). Click 'OK'
- Again, go to **Utility Menu>Plot>Lines**
- From Preprocessor, choose **Modeling>Operate>Boolean>Glue>Areas**.
  - o Choose the Steel area and then the Aluminum area. Ensure that the copper area is not selected (should remain black). Click 'OK'.
  - o The reason why we don't glue the copper and the steel is that they overlap. Picture a copper plate resting on the steel area. The steel and aluminum are connected more intimately, and must be glued together.
- If you cannot see the complete work plane then go to **Utility Menu>Plot Controls>Pan Zoom Rotate** and zoom out to see the entire work plane.
- Use **Utility Menu>Plot Controls>Numbering>Area Numbers>On** to display the individual model areas. The model should look like the one below.
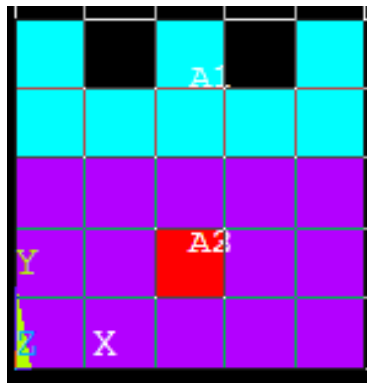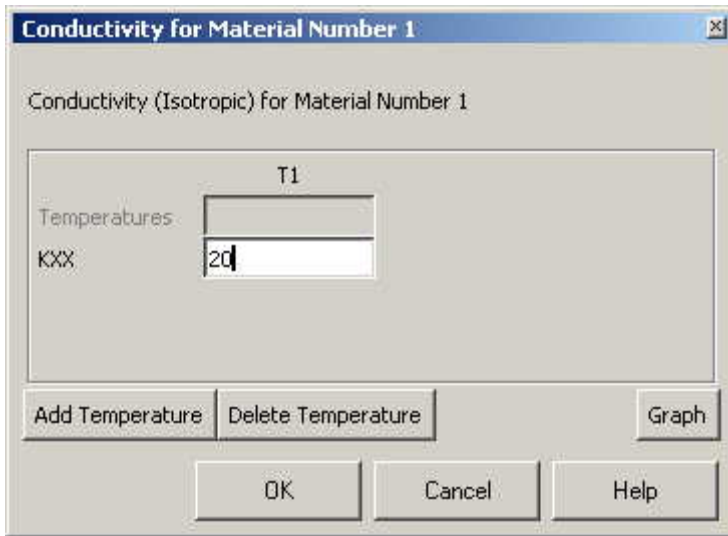

MATERIAL PROPERTIES

- We need to define material properties separately for steel, aluminum, and copper.
- Go to the ANSYS Main Menu.
- Click **Preprocessor>Material Props>Material Models**. In the window that comes up
- Choose **Thermal>Conductivity>Isotropic**.
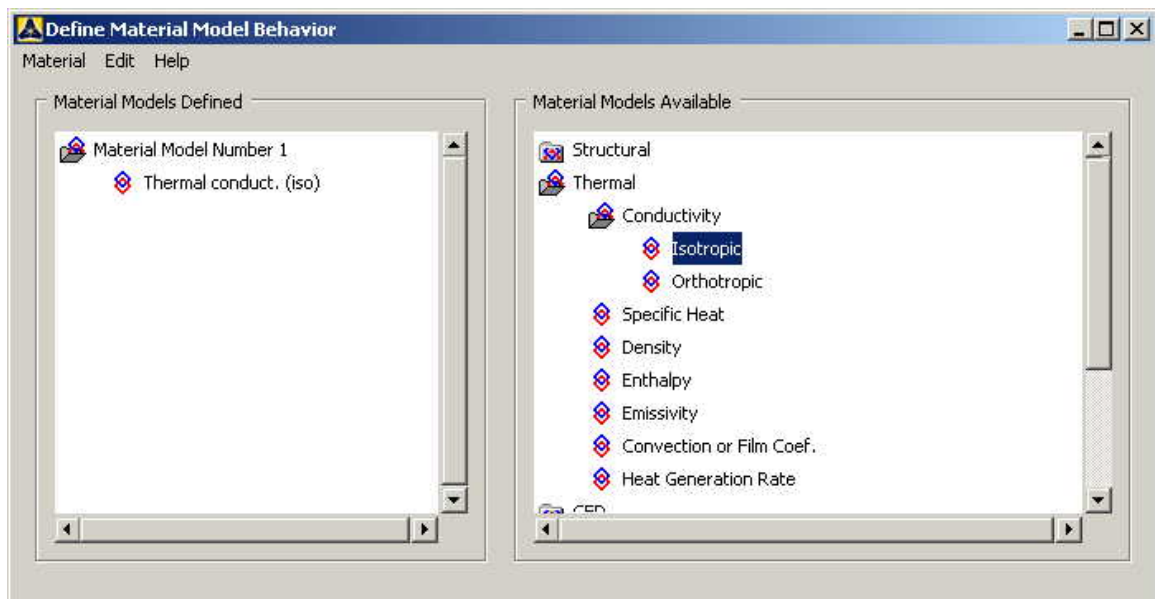
- Enter 1 for the Material Property Number and click OK. The following window comes up.

- Fill in **20** for Thermal conductivity. Click OK.
- Now the material 1 has the properties defined in the above table. This represents the material properties for **steel**. Repeat the above steps to create material properties for **aluminum** (k=180, Material number 2), and **copper** (k=386, Material number 3). Do this by selecting **Material>New Model** in the "Define Material Model Behavior" window.

ELEMENT PROPERTIES

SELECTING ELEMENT TYPE:Click **Preprocessor>Element Type>Add/Edit/Delete...** In the 'Element Types' window that opens click on Add... The following window opens.
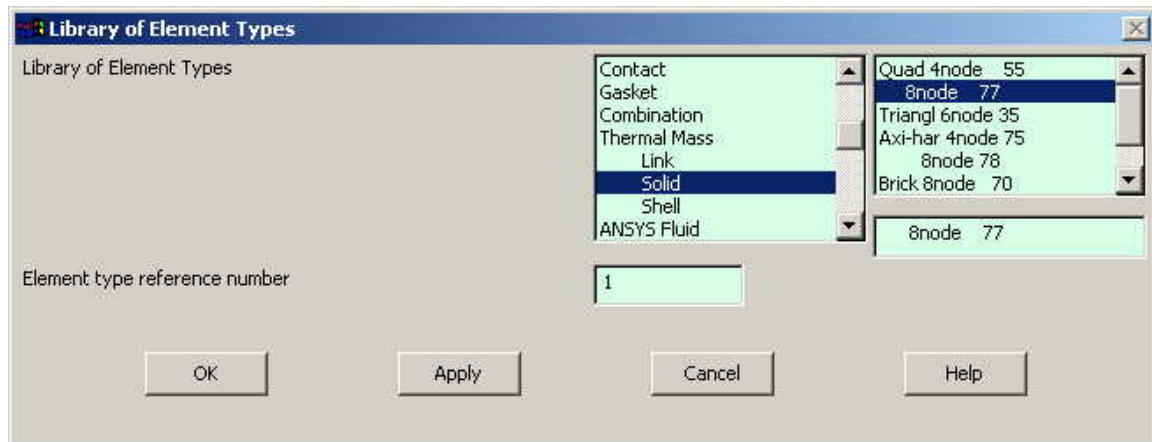

- Type **1** in the Element type reference number.
- Click on **Thermal Mass Solid** and select **Quad 8node 77**. Click OK. Close the 'Element types' window.
- So now we have selected Element type 1 to be a thermal solid 8node element. The component will now be modeled with thermal solid 8node elements. This finishes the selection of element type.

MESHING

- DIVIDING THE TOWER INTO ELEMENTS:
- Go to **Preprocessor>Meshing>Size Controls>Manual Size>Lines>All Lines**. In the menu that comes up type 0.005 in the field for 'Element edge length'.
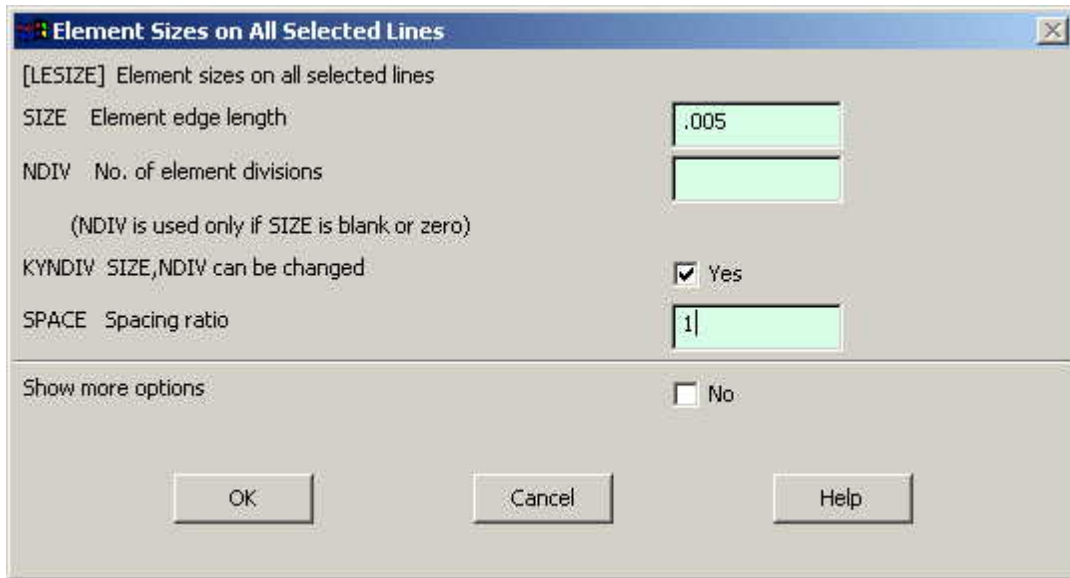
- Click on OK. Now when you mesh the figure ANSYS will automatically create meshes that have an edge length of **0.005m** along the lines you selected.
- First we will mesh the steel area. Go to **Preprocessor>Meshing>Mesh Attributes>Default Attributes**. Make sure the window indicates "Material Ref.#1". The window is shown below.
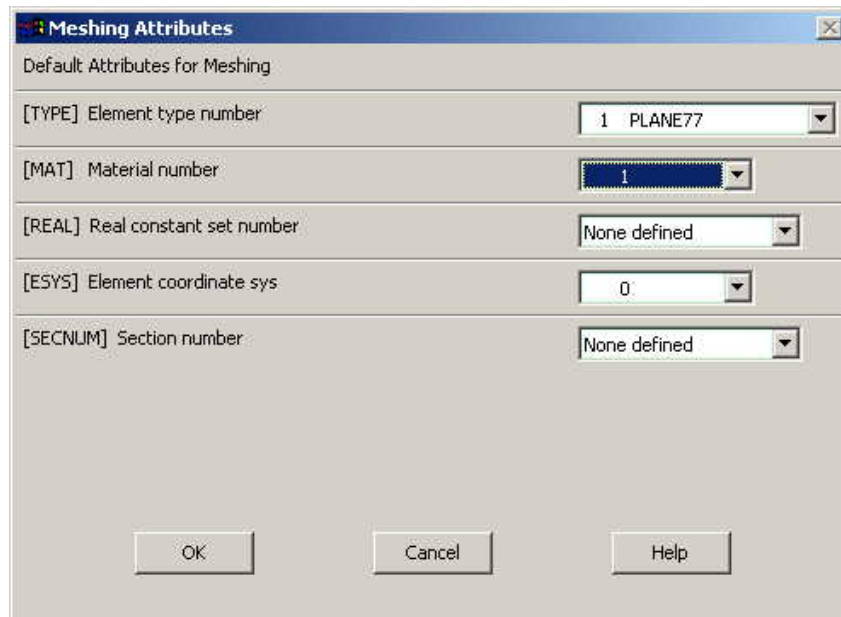
- Click **Utility Menu>Plot>Lines.**
- Now go to **Preprocessor>Meshing>Mesh>Areas>Free**. Pick the steel area and click OK.
- Repeat the same process for the aluminum and copper areas. Make sure you use the correct material number (2 and 3 respectively) for both the areas. Also since the steel and the copper areas overlap make sure you pick the right area. If you choose the wrong area, use **Preprocessor>Meshing>Clear** to undo the previous mesh and then repeat the previous steps. The meshed area should look like this:
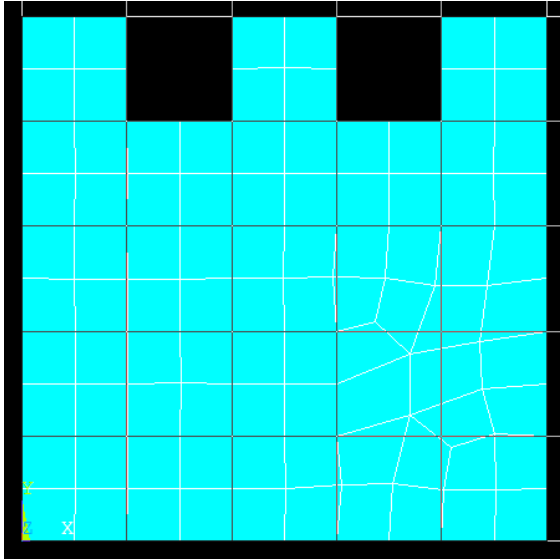

BOUNDARY CONDITIONS AND CONSTRAINTS

- Go to **Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generate>On Keypoints**.
- Select the corners of the copper square. Click OK. The following window comes up.
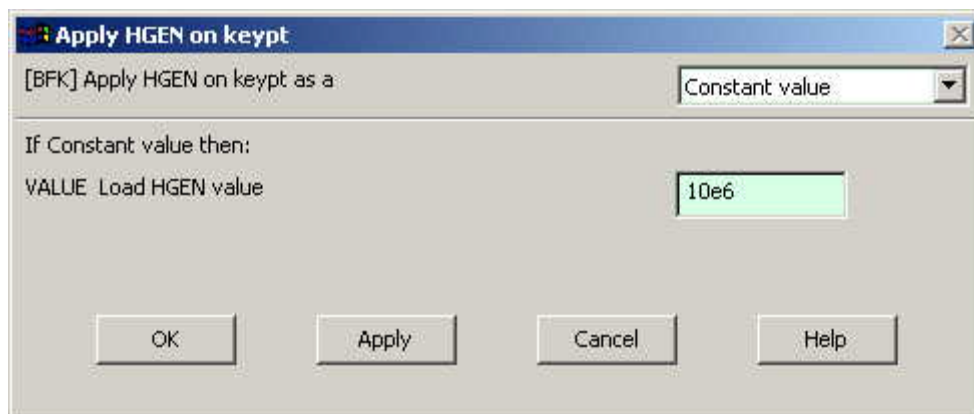
- Enter **10e6** for the HGEN value and click OK.
- Go to **Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>On Lines**.

■ Pick all the lines on the outside of the object except the bottom one where the object is considered insulated. Click OK. The following window comes up.
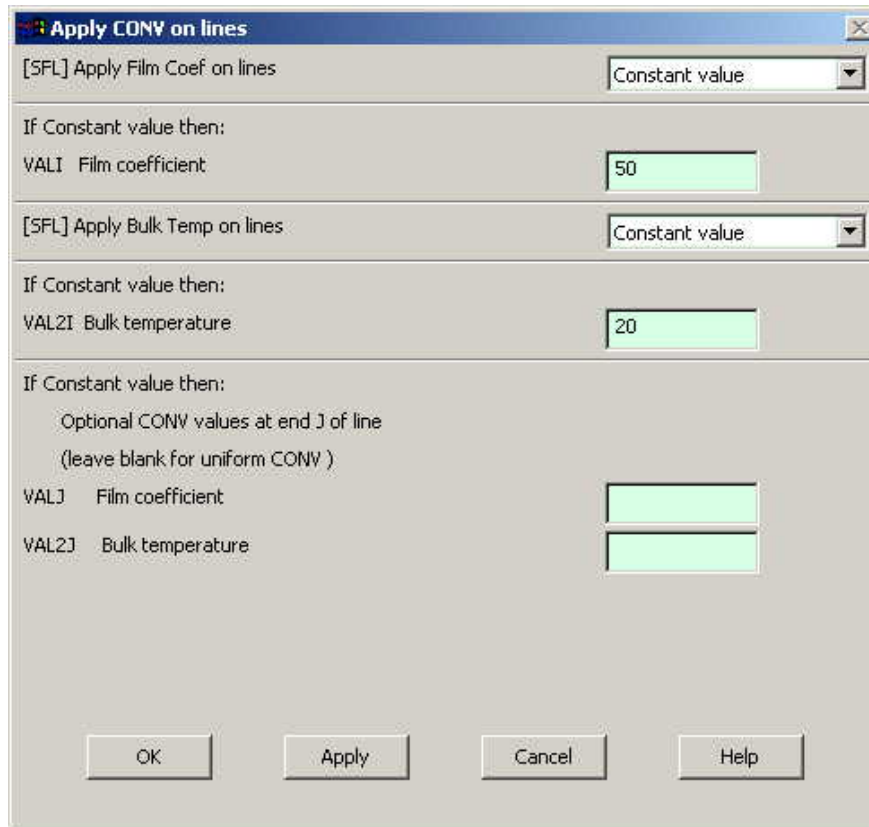
■ Enter **50** for "Film Coefficient" and **20** for "Bulk Temperature" and click OK.
■ Now the Modeling of the problem is done.
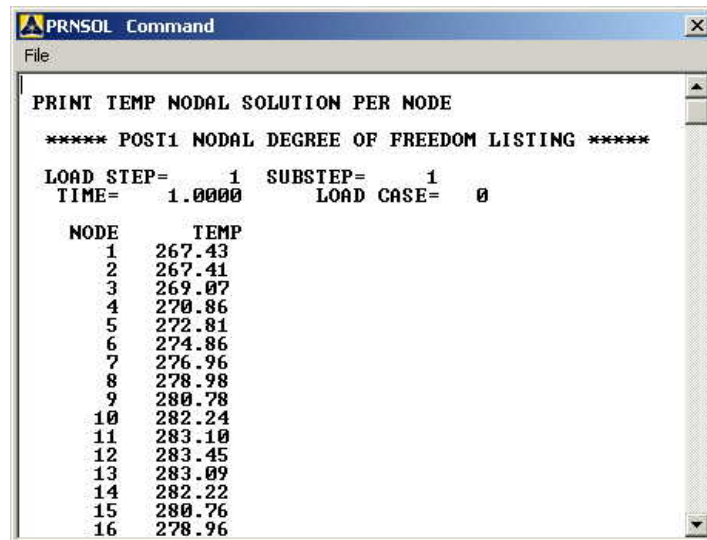■ The model should look as follows:
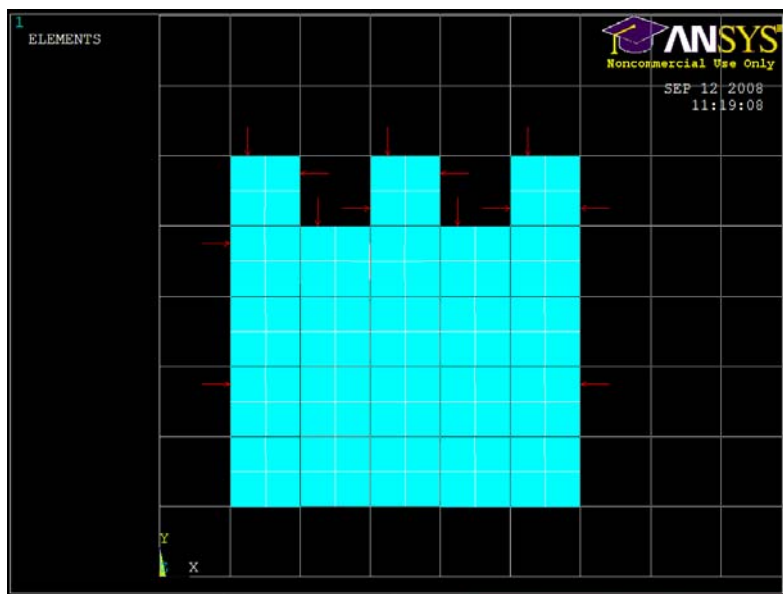
SOLUTION

■ Go to ANSYS **Main Menu>Solution>Analysis Type>New Analysis**.
■ Select **Steady State** and click on OK.
■ Go to **Solution>Solve>Current LS**.
■ An error window may appear. Click OK on that window and ignore it.
■ Wait for ANSYS to solve the problem.
■ Click on OK and close the 'Information' window.

POST-PROCESSING

■ Listing the results.
■ Go to ANSYS Main Menu **General Postprocessing>List Results>Nodal Solution**. The following window will come up.
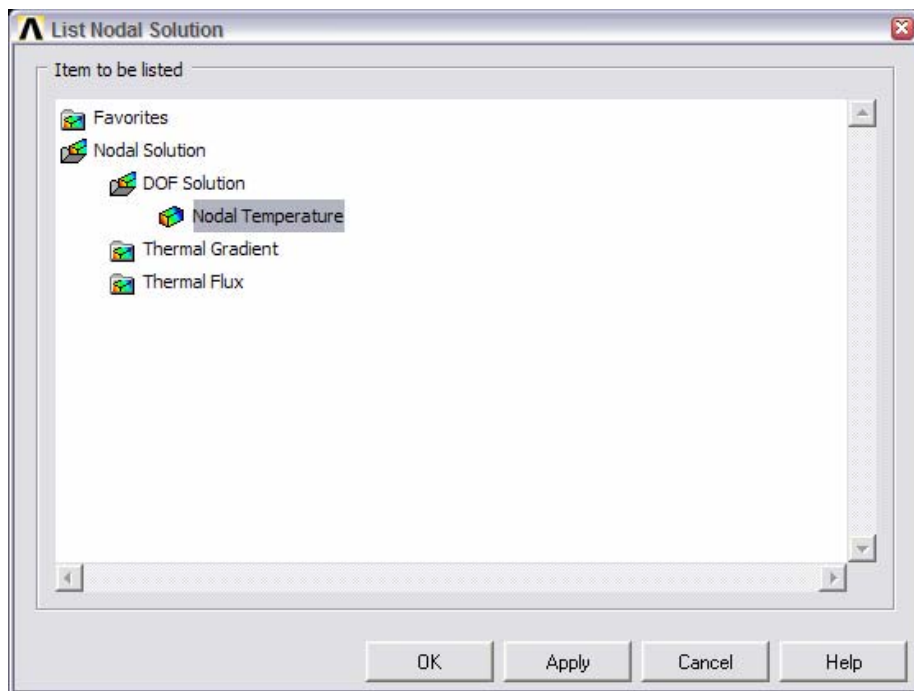
■ Under **Nodal Solution**, Select **DOF solution** and click on **Nodal Temperature**.
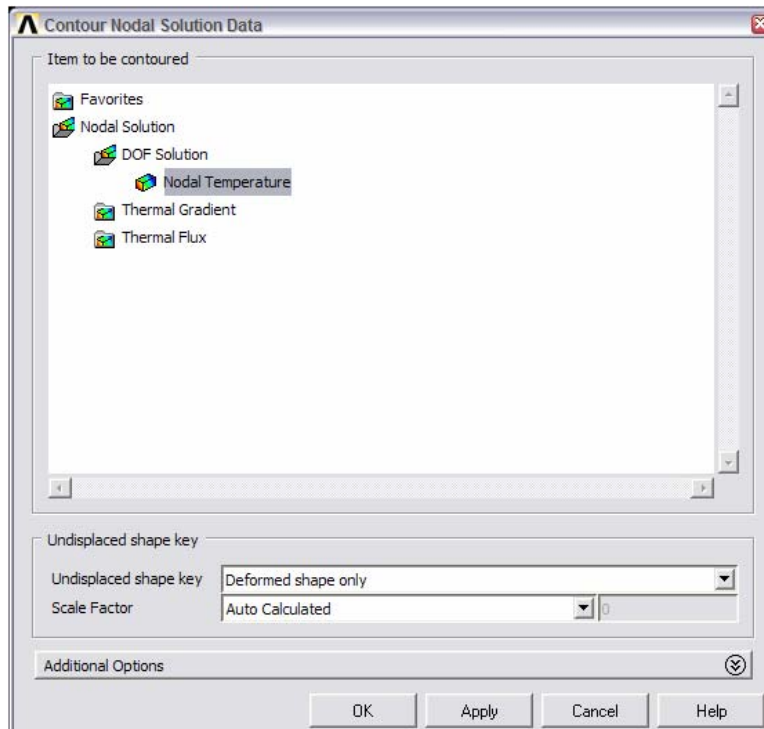■ Click on OK. The nodal displacements will be listed as follows.

■ You will find the maximum value of temperature at the end of the above table.

MODIFICATION

■ You can also plot the displacements and stress.
■ Go to **General Postprocessing>Plot Results>Contour Plot>Nodal Solution**. The following window will come up:

■ Under Nodal Solution, select **DOF solution** and **Nodal Temperature** to be plotted and click OK. The output will be like this: