

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

Temperature distribution in a fin cooled electronic component
(Carnegie Mellon)

Last updated September 2008 – Allen Umali

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.

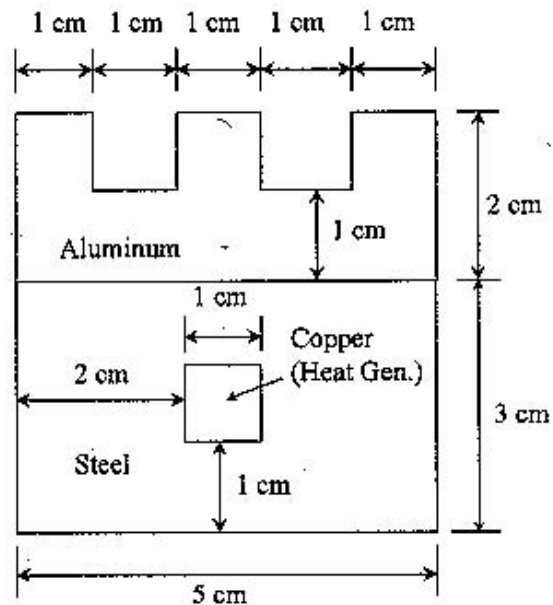
Boundary conditions: There is convection along all the boundaries except the bottom, which is insulated. The Film Coefficient is 50 W/m² K and the Bulk Temperature is 20°C.

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:**



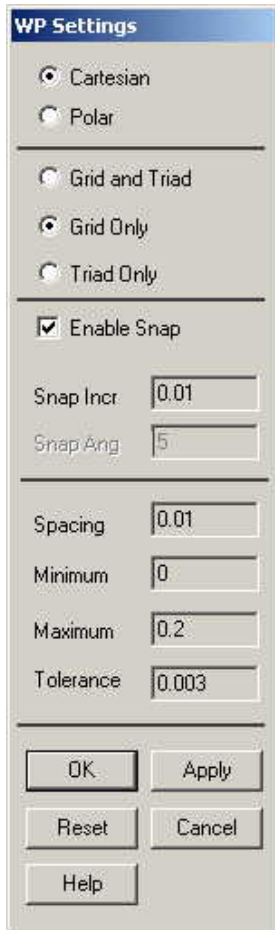
IMPORTANT: Convert all dimensions and forces into SI units.

STARTING ANSYS

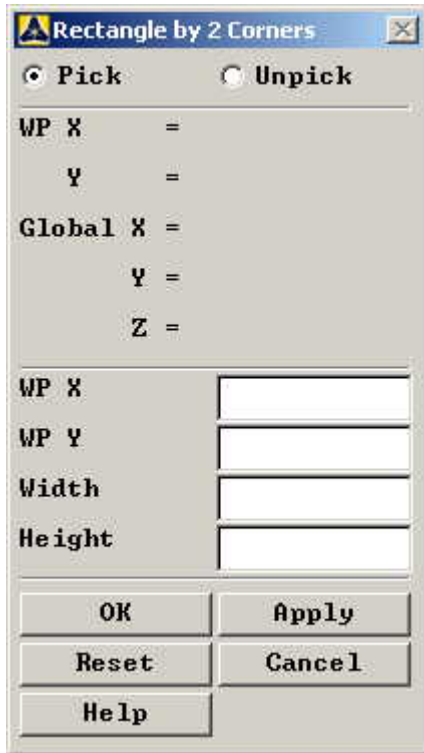
Click on ANSYS 11.0 in the programs menu.

MODELING THE STRUCTURE

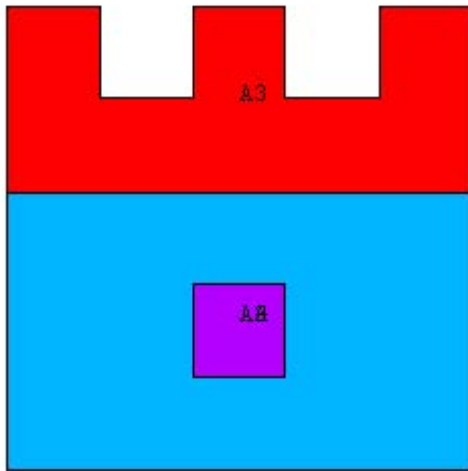
- Go to the ANSYS Utility Menu.
- Click **Workplane>WP Settings**.
- The following window comes up



- 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**.
- The following window comes up:

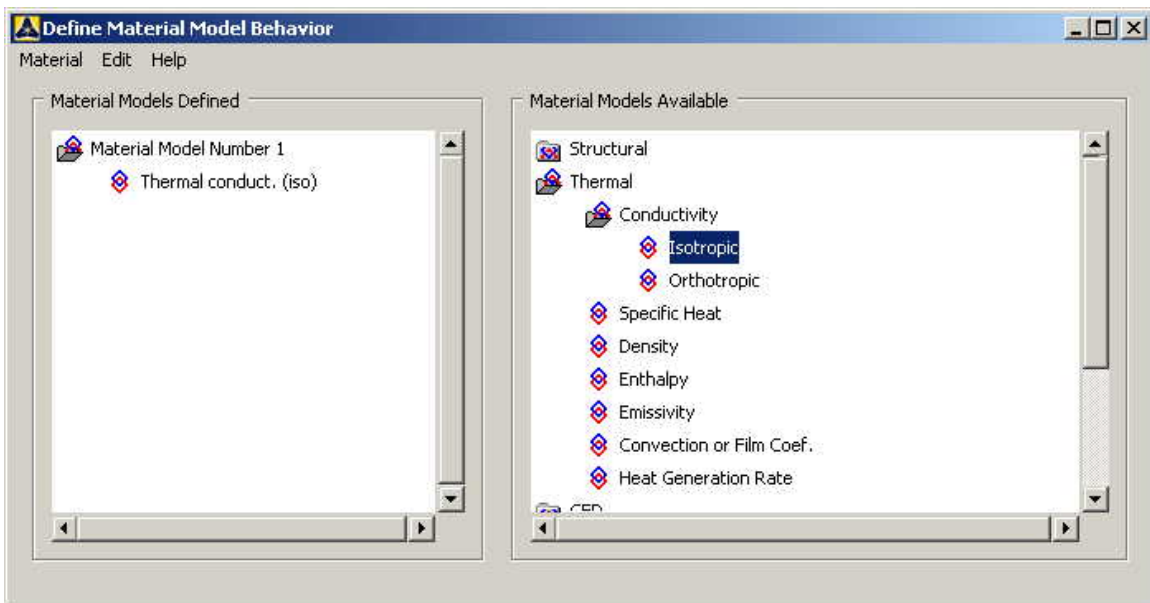


- Now we will pick the end points of the rectangles.
- First make the steel rectangle of dimensions 5cm X 3 cm, i.e. 5 units by 3 units on the grid.
- Next make the copper square of dimensions 1cm X 1cm.
- Next make the aluminum part by making a rectangle of dimensions 5cm X 2cm and then
- creating two smaller rectangles, which can then be subtracted from the main part to make the fins.
- From Preprocessor, choose **Modeling>Operate>Boolean>Overlap>Areas**. Choose the Steel area and then the Copper area, then click OK.
- From Preprocessor, choose **Modeling>Operate>Boolean>Glue>Areas**. Choose the Steel area and then the Aluminum area, and then click OK. 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 workplane then go to **Utility Menu>Plot Controls>Pan Zoom Rotate** and zoom out to see the entire workplane.
- The model should look like the one below.

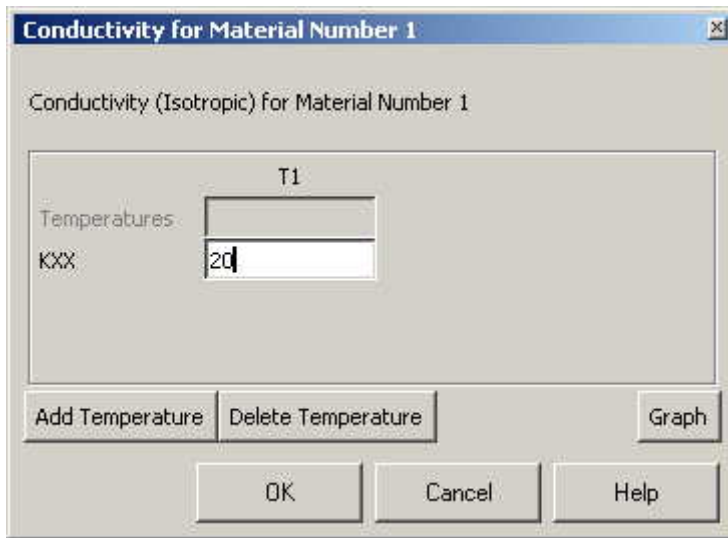


MATERIAL PROPERTIES

- 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**.



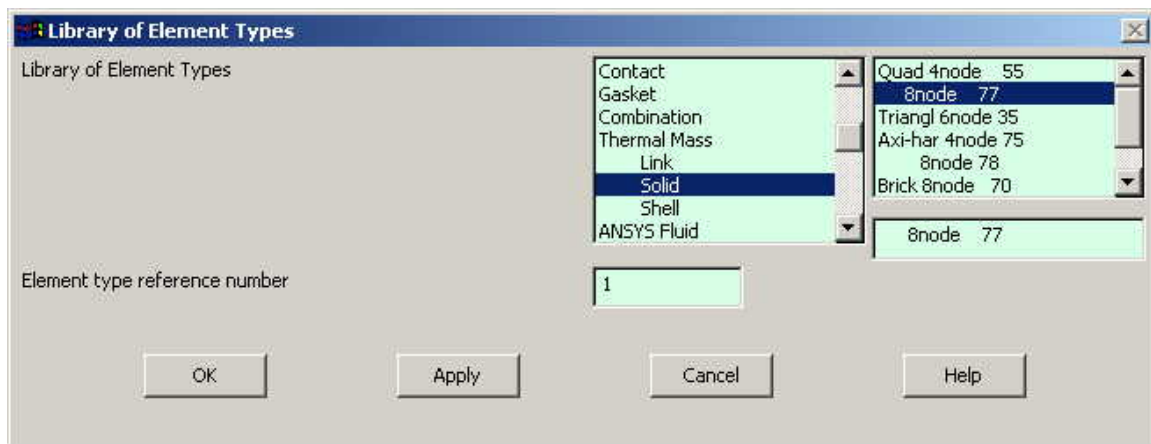
- Enter 1 for the Material Property Number and click OK. The following window comes up.



- 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

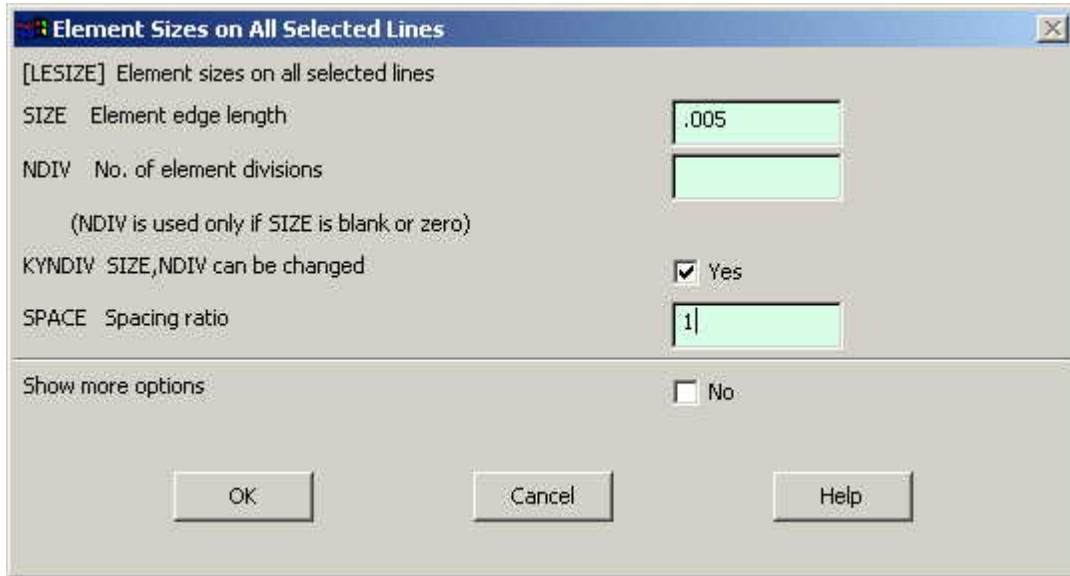
SELECTING ELEMENT TYPE: Click **Preprocessor>Element Type>Add/Edit/Delete...** In the 'Element Types' window that opens click on Add... The following window opens.



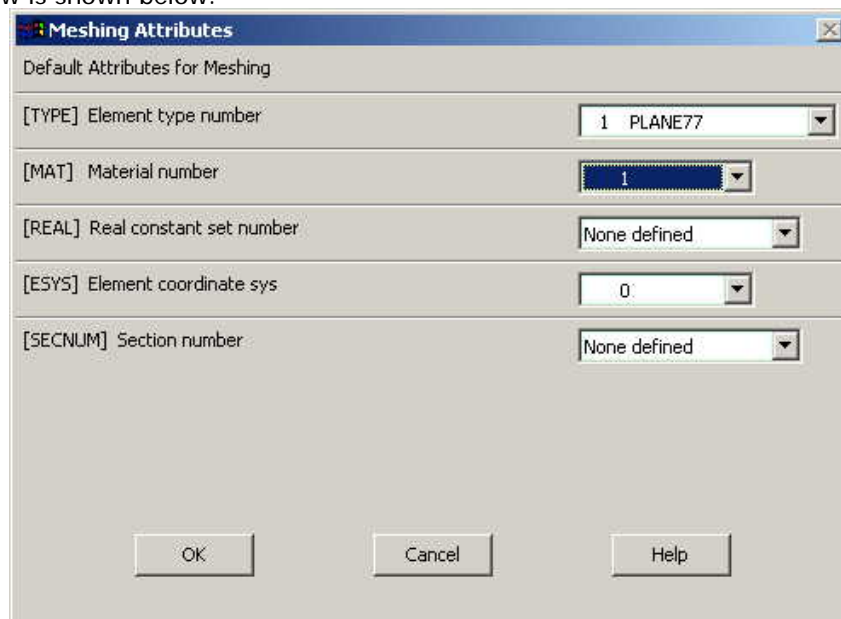
- Type **1** in the Element type reference number.
- Click on **Thermal Mass Solid** and select **Quad 8node 77**. Click OK. Close the 'Element types' window.
- So now we have selected Element type 1 to be a thermal solid 8node element. The component will now be modeled with thermal solid 8node elements.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

MESHING

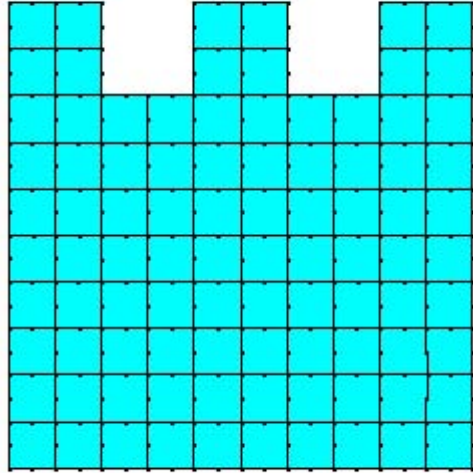
- DIVIDING THE TOWER INTO ELEMENTS:
- Go to **Preprocessor>Meshing>Size Controls>Manual Size>Lines>All Lines**. In the menu that comes up type 0.005 in the field for 'Element edge length'.



- 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.

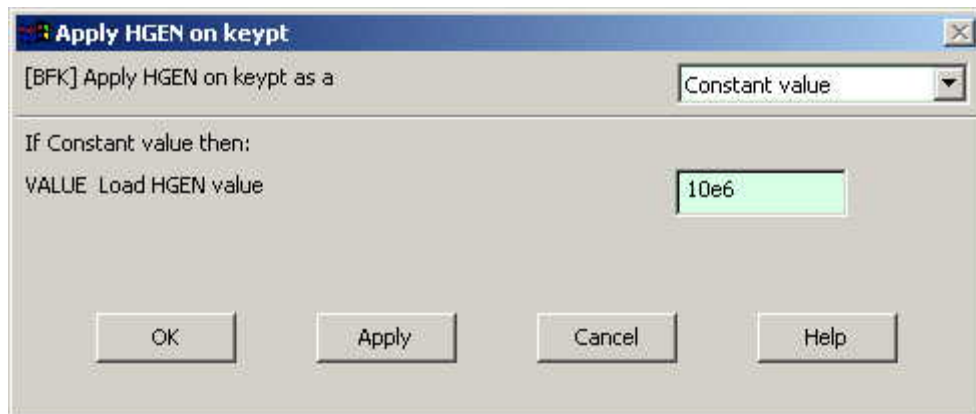


- Now go to **Preprocessor>Meshing>Mesh>Areas>Free**. Pick the steel area and click OK.
- Repeat the same process for the aluminum and copper areas. Make sure you use the correct material number (2 and 3 respectively) for both the areas. Also since the steel and the copper areas overlap make sure you 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.

Apply CONV on lines

[SFL] Apply Film Coef on lines Constant value

If Constant value then:
VALI Film coefficient

[SFL] Apply Bulk Temp on lines Constant value

If Constant value then:
VAL2I Bulk temperature

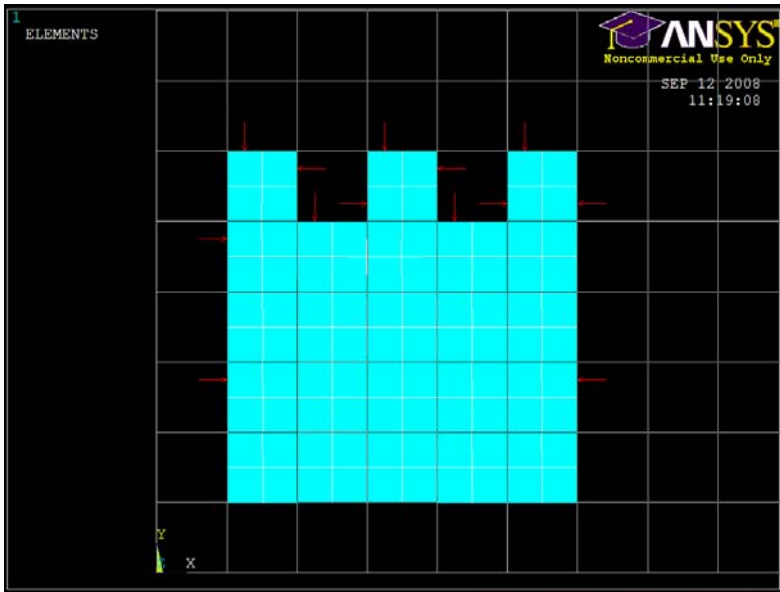
If Constant value then:
Optional CONV values at end J of line
(leave blank for uniform CONV)

VALJ Film coefficient

VAL2J Bulk temperature

OK Apply Cancel Help

- 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:

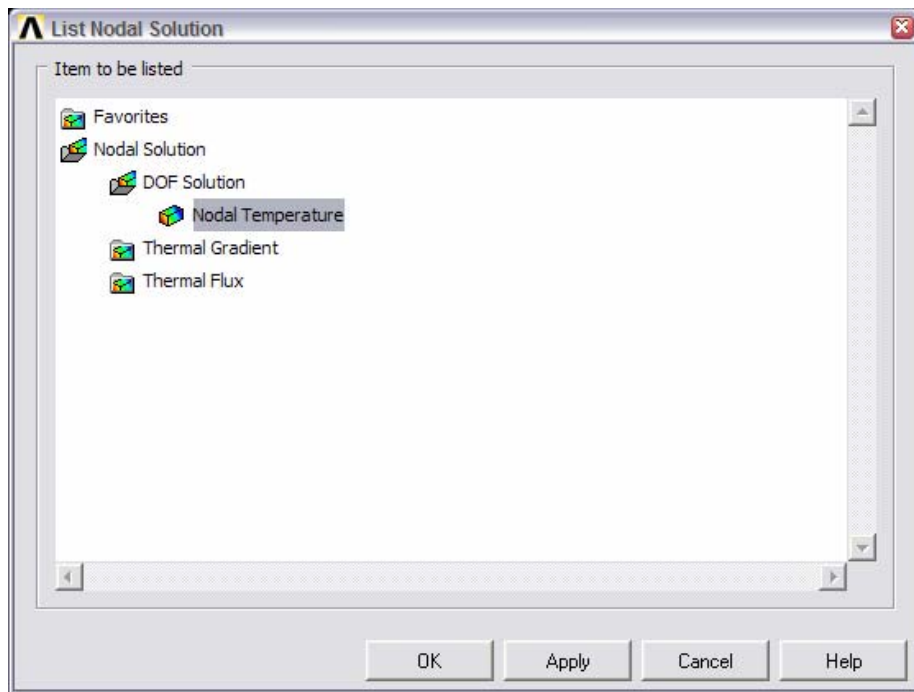


SOLUTION

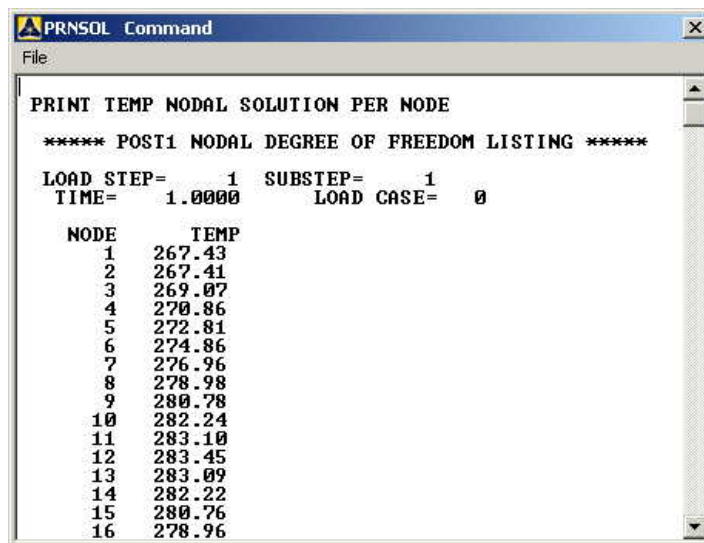
- Go to ANSYS **Main Menu>Solution>Analysis Type>New Analysis**.
- Select **Steady State** and click on OK.
- Go to **Solution>Solve>Current LS**.
- An error window may appear. Click OK on that window and ignore it.
- Wait for ANSYS to solve the problem.
- Click on OK and close the 'Information' window.

POST-PROCESSING

- Listing the results.
- Go to ANSYS Main Menu **General Postprocessing>List Results>Nodal Solution**. The following window will come up.



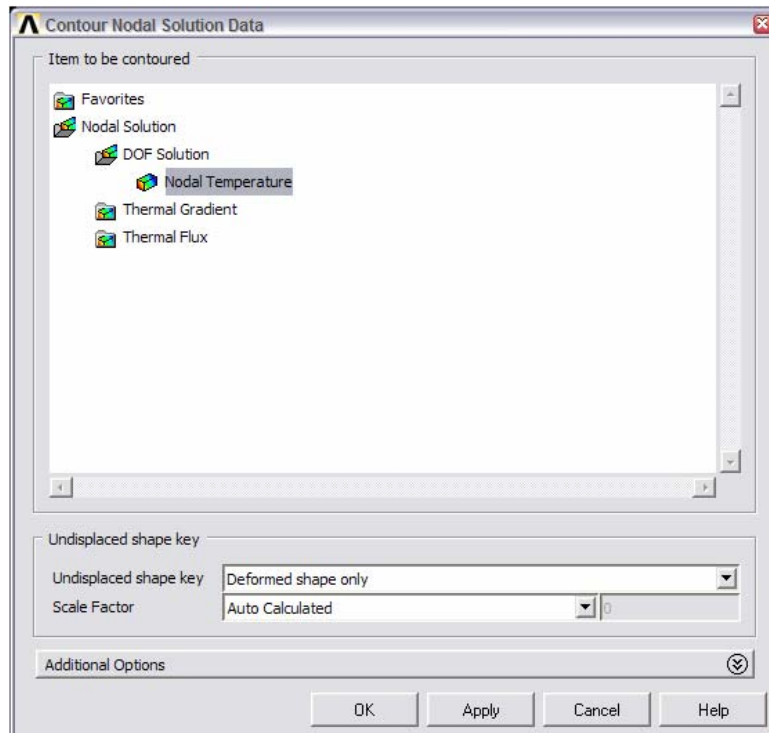
- Under **Nodal Solution**, Select **DOF solution** and click on **Nodal Temperature**.
- Click on OK. The nodal displacements will be listed as follows.



- 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:

