**ANSYS EXERCISE – ANSYS 5.6**
**Temperature Distribution in a Plate**

John R. Baker; phone: 270-534-3114; email: jbaker@engr.uky.edu

---

*This exercise is intended only as an educational tool to assist those who wish to learn how to use ANSYS. It is not intended to be used as a guide for determining suitable modeling methods for any application. The author assumes no responsibility for the use of any of the information in this tutorial. There has been no formal quality control process applied to this tutorial, so there is certainly no guarantee that there are not mistakes on the following pages. The author would appreciate feedback at the email address above if mistakes are discovered in this tutorial.*

---

In this exercise, you will solve the 2-D heat conduction problem below, using ANSYS. The problem is adapted from the textbook, *Introduction to Heat Transfer*, by Frank Incropera and David P. Dewitt. You will solve for the temperature distribution within the rectangular plate, based on the specified temperatures on the plate edges, and the plate dimensions. Step-by-step instructions are provided beginning on the following page.

---

Note: Thermal conductivity of the plate, KXX=401 W/(m-K)
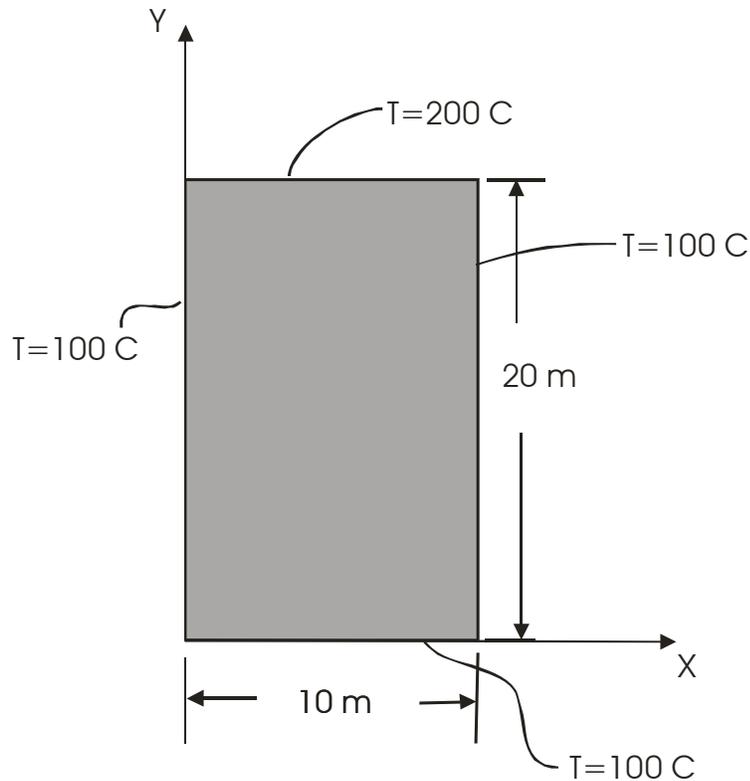
The steps that will be followed, after launching ANSYS, are:

**Preprocessing:**
1. Change Jobname.
2. Define element type. (Plane55 element, which is a 2-D, 4-node element for thermal analysis.)
3. Define material properties. (Thermal Conductivity -- only property required for this analysis.)
4. Create the rectangular area.
5. Specify mesh density controls. (We will specify numbers of element divisions along lines.)
6. Mesh the area to create nodes and elements.

**Solution:**
7. Specify temperatures along lines that define the edges of the plate.
8. Solve.

**Postprocessing:**
9. Plot the temperature distribution.
10. Select nodes along the plate center (x=5 in.).
11. List locations of these center nodes by node number.
12. List the temperatures at each of these nodes.

**Re-analysis**
13. Modify Mesh / Re-analyze. (Primarily repeating earlier steps.).

**Exit**
14. Exit the ANSYS program, saving all data.
_____


**Launching ANSYS:**

You should be able to simply click on the ANSYS icon on the Windows Desktop.  The ANSYS Launcher menu should appear.  It is shown at the top of the next page.  The only input you will likely need on this menu is specification of your "Working Directory".  Specify any directory (or, "Folder") as your working directory.  To browse to find the desired working directory, click on the button with the three dots to the far right on the Launcher Menu on the line that says "Working Directory".  Once the working directory is specified, click on "Run" at the bottom of the Launcher Menu.

The ANSYS menus should open up.  You will see a Main Menu, illustrated on the following page, and a large black graphics window.  You are now ready to begin creating the model and performing the analysis.
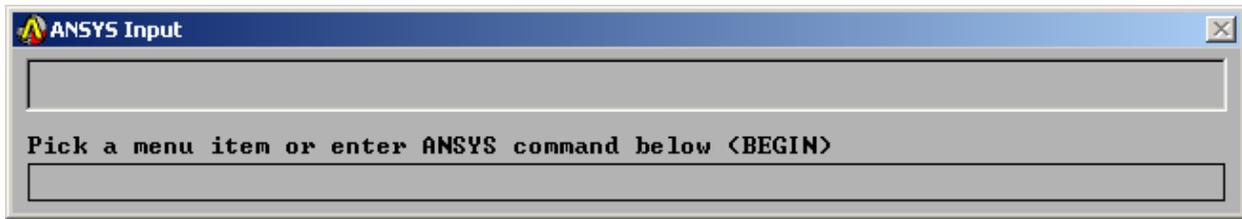

ANSYS Launcher Menu:
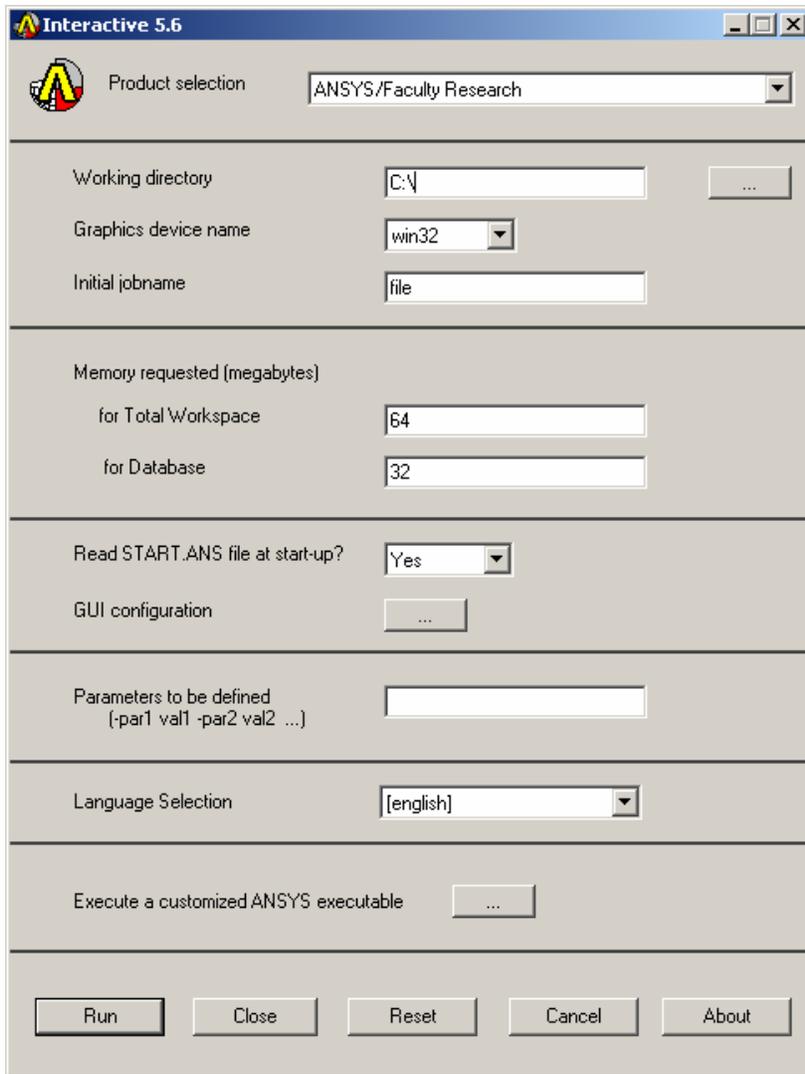
ANSYS Main Menu:
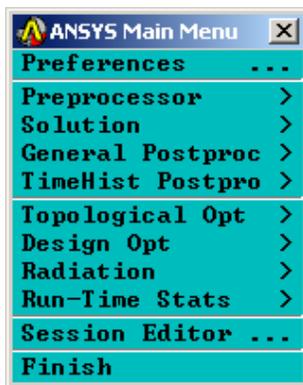


**Note:** The menu picks needed to perform all required tasks are specified in italics in the step-by-step instructions below. It is sometimes more convenient to enter certain commands directly at the command line. The command line is seen on the screen as:

```
ANSYS Input                                                    ⊠

Pick a menu item or enter ANSYS command below (BEGIN)
```

The method of direct command line entry, however, is not covered in this exercise. Primarily, in this exercise, the analysis will be performed using menu picks from the ANSYS Graphical User Interface. Often, as an alternative, an input file, known as a "batch file", is created, which is simply an ASCII text file containing a string of ANSYS commands in the appropriate order. ANSYS can read in this file as if it were a program, and perform the analysis in "batch mode", without ever opening up the Graphical User Interface. The batch file option is not covered in this exercise.

SUGGESTION: As you work through this exercise, on the ANSYS Toolbar (upper right in the ANSYS Graphical User Interface) click on "SAVE_DB" often! This Toolbar appears as:



```
ANSYS Toolbar                                                  ⊠
SAVE_DB
RESUM_DB
QUIT
POWRGRPH
```
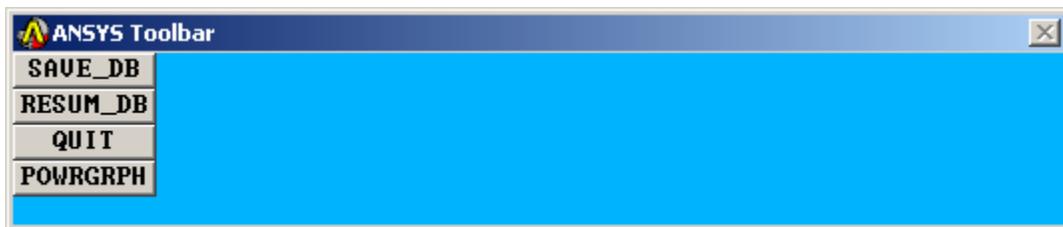
At any point, if you want to resume from the previous time the model was saved, simply click on "RESUM_DB" on this same Toolbar. Any information entered since the last save will be lost, but this is a nice feature in the event that you make an input mistake, and are unsure of how to correct it.

There are a number of ways to model a system and perform an analysis in ANSYS. The steps shown on the following pages present only one method.

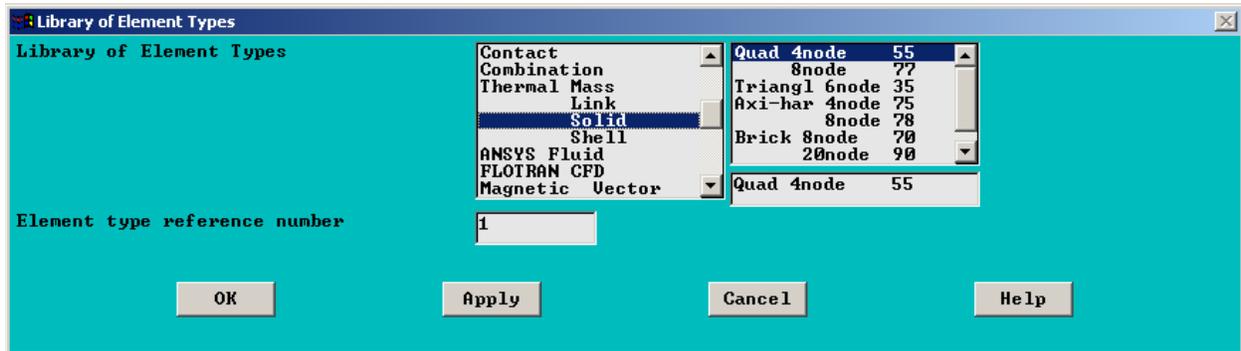**Preprocessing:**

**1. Change jobname.** On the Utility Menu across the very top of the screen, select:

*File -> Change Jobname*

Enter "platetmp", and click on "OK".

**2. Define element type:** *Preprocessor -> Element Type -> Add/Edit/Delete*

Click on "Add". The "Library of Element Types" menu appears, as shown. Scroll down to highlight "Thermal Solid", and "Quad 4node 55" as shown. Click on "OK", then "Close". Note that in ANSYS, this element is sometimes referred to as "Plane55", because it is a 2-D planar element, and it is element type 55 in the ANSYS element library.



**3. Define Material Properties**: *Preprocessor -> Material Properties -> -Constant- Isotropic*
"OK" for material set number 1, then enter 401 for KXX (thermal conductivity), then click on "OK". KXX is the only material property needed for this analysis.

**4. Create a rectangular area:**

*Preprocessor -> -Modeling- Create -> -Areas- Rectangle -> By Dimensions*

Fill in the fields as shown, then click "OK".
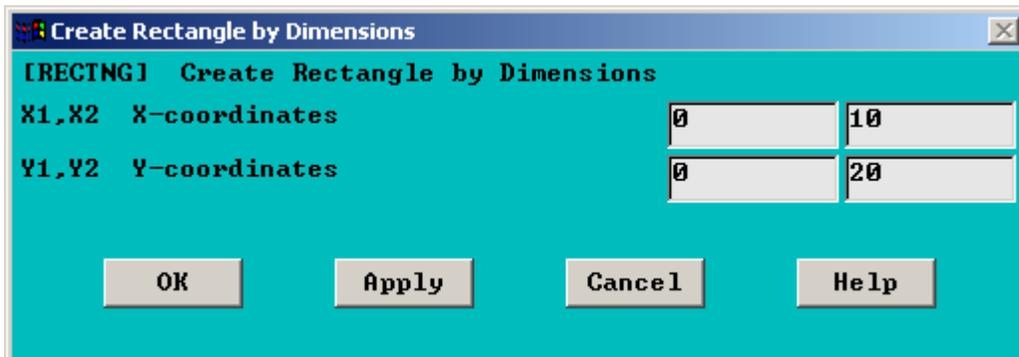


**5. Specify mesh density controls.**

We will specify numbers of element divisions along lines.  Choose:

*Preprocessor* -> -Meshing- *Size Controls* -> -Lines- *Picked Lines*

The picking menu (below left) appears.  On the graphics window, click on the bottom horizontal line (this is one of the 10 meter lines), to highlight it.  Then, click "OK" in the picking menu.  Then, the "Element Size" menu (below right) appears.  Enter "6" for "NDIV", as shown, then click "OK".
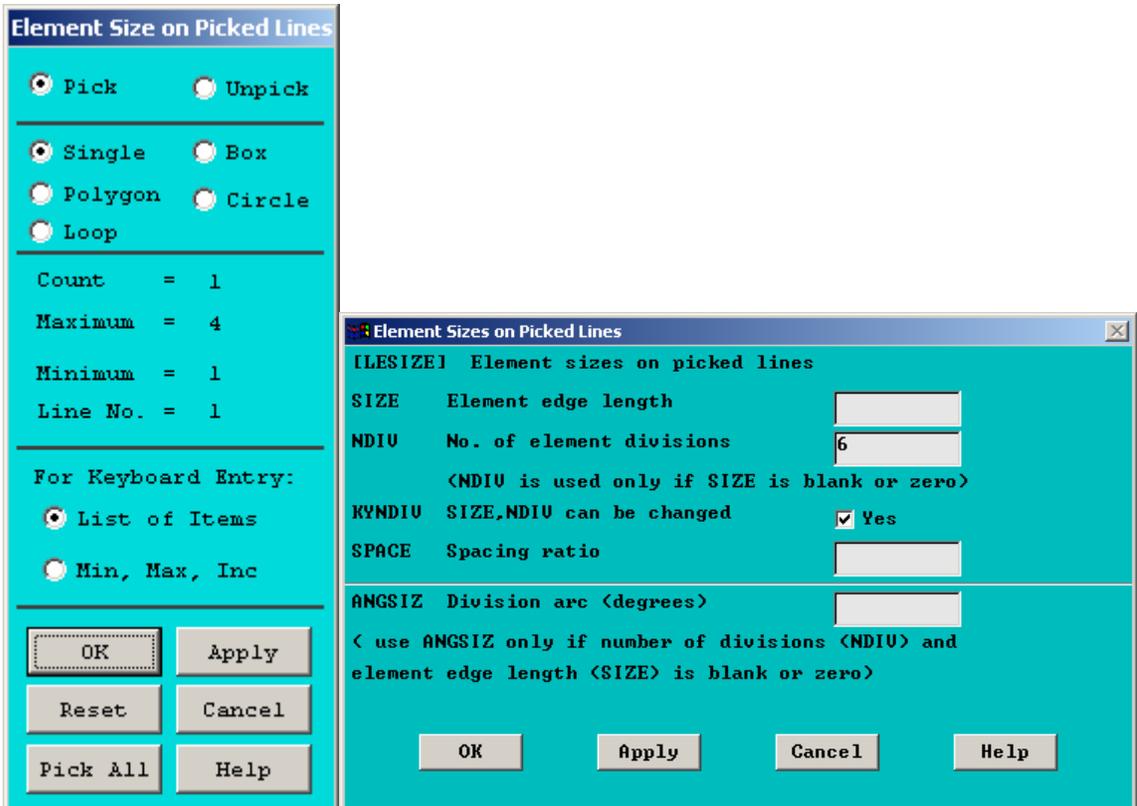


Now, repeat the above process to specify 12 divisions along either of the vertical lines.  It is not necessary to specify a value for all four lines, you just need a value for one of the horizontal lines and one of the vertical lines.  However, if you do specify a value for all four lines, make sure you use the same number of divisions for both horizontal lines, and the same number of divisions for both vertical lines.  Of course, as specified above, the vertical and horizontal lines do not need to have the same value, it is only a problem if you specify different values for parallel lines on opposite ends of the plate.

**6. Mesh the rectangle to create nodes and elements.**

*Preprocessor -> -Meshing- Mesh -> Areas -> Mapped -> 3 or 4 Sided*

A picking menu appears. Select "Pick All".  The rectangle will be meshed.  You will see a number of small rectangles drawn on the larger rectangular area.  Each small rectangle is a "finite element". There are four nodes associated with each individual element.

**Solution:**

**7. Apply temperatures around the edges:**

*Solution -> -Loads- Apply -> -Thermal- Temperature -> On Lines*

A picking menu appears.  Highlight the two vertical lines (the 20 meter lines), which have a temperature of 100 C, then click on "OK" in the picking menu.  The box below appears.  Enter "100" for temperature.  For this problem, there is a bit of a dilemma.  We do not know the temperature at the corners at the "tops" of these two lines where they intersect the 200 C line.  So, we might choose "No" for "Apply TEMP to endpoints?".  By default, this question has the answer "Yes".  Simply click on the "Yes" box to toggle the answer to "No", as shown below.   Then click on "OK".
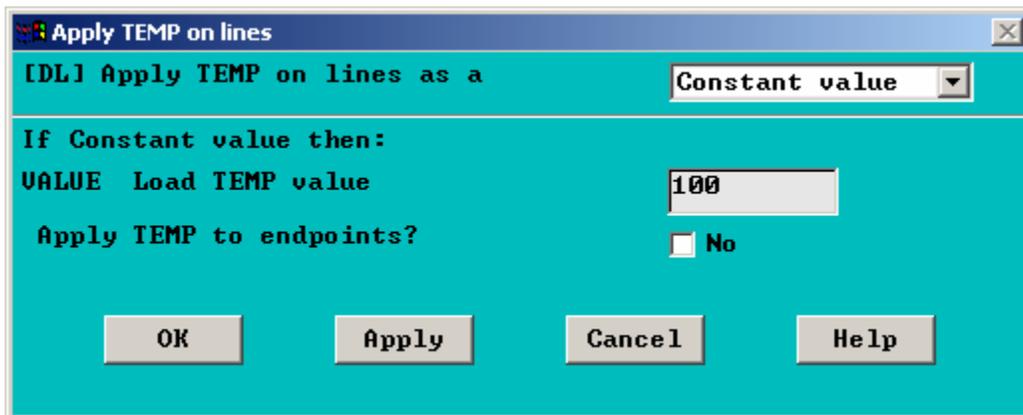


Repeat the above process to apply the 100 C temperature to the bottom horizontal line, but in this case, choose "Yes" for "Apply TEMP to endpoints?".   Repeat the process once more, to apply the 200 C temperature to the top horizontal line, but in this case, choose "No" for  "Apply TEMP to endpoints?".

Now, to address the fact that two corners do not have a specified temperature, as an approximation, we will set the temperature at these to corners to 150 C.

*Solution -> -Loads- Apply -> -Thermal- Temperature -> On Keypoints*

A picking menu appears. Note that there are four "keypoints" in the model, one at each corner of the large rectangular area. Click on the upper two corners, at the intersections of the 100 C and 200 C lines. When these corner "keypoints" are highlighted, choose "OK" in the picking menu, and the following box appears:
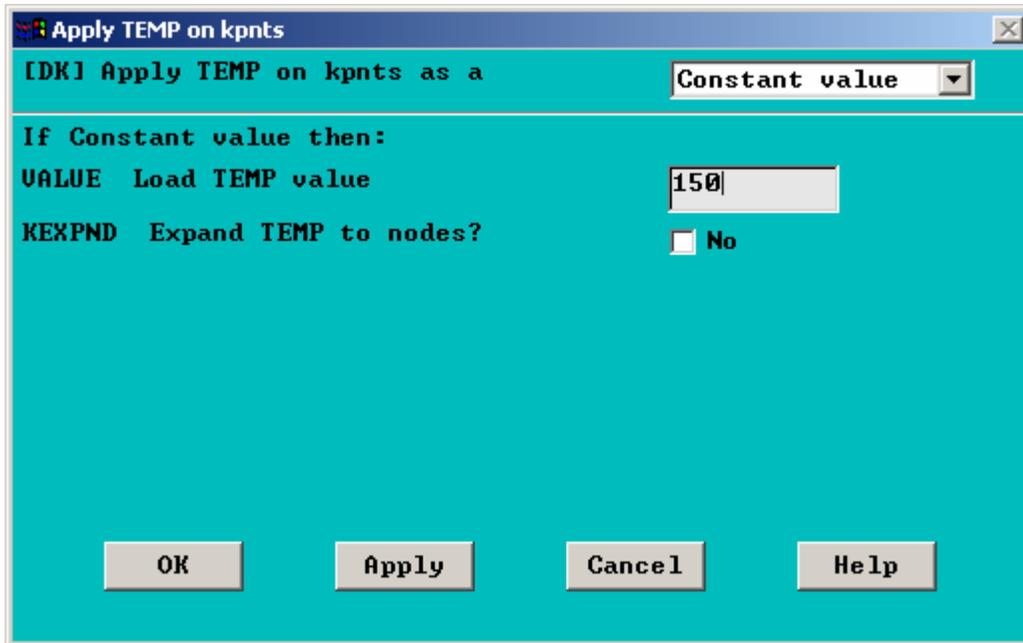


Enter "150" for VALUE, then click "OK".

Note: In some cases, when you want to repeat the same process two or more times, you can click "Apply" in the box, instead of "OK". That issues the command, but leaves the box open. Clicking "OK" issues the command, but leaves the box open. Sometimes, though, using the "Apply" option can cause problems, if you are not careful. You might click on "Apply" to issue the command, then immediately click on "OK" to close the box, and this issues the command twice. Sometimes, that is not a problem, and sometimes it is. So, in this exercise, you are just asked to click on "OK" in each case, then repeat the entire procedure. For the relatively simple modeling effort undertaken in this exercise, this approach does not add much work, and is probably less likely to result in errors.

**8. Solve the problem:** *Solution -> -Solve- Current LS*

Click "OK" in the "Solve Current Load Step" Box. Ignore any warnings you see about "DOF Constraint TEMP from line......". Possibly, there is a small bug in ANSYS related to the applied temperatures at endpoints of lines which cause these warnings (this endpoint situation was discussed in step 7). However, these warnings to not indicate that there is a problem with this particular analysis. In general, an ANSYS user should pay close attention to warnings, but ignore these. Soon after clicking "OK", you should see a note in a yellow box saying "Solution is done!" You may close this yellow box.

**Postprocessing:**

## 9. Plot the temperature distribution:

*General Postproc -> Plot Results ->* -Contour Plot- *Nodal Solu*

The box below appears. Highlight "DOF solution" and "Temperature TEMP", as shown, and click "OK". In the graphics window, a plot, as shown on the following page, should appear.
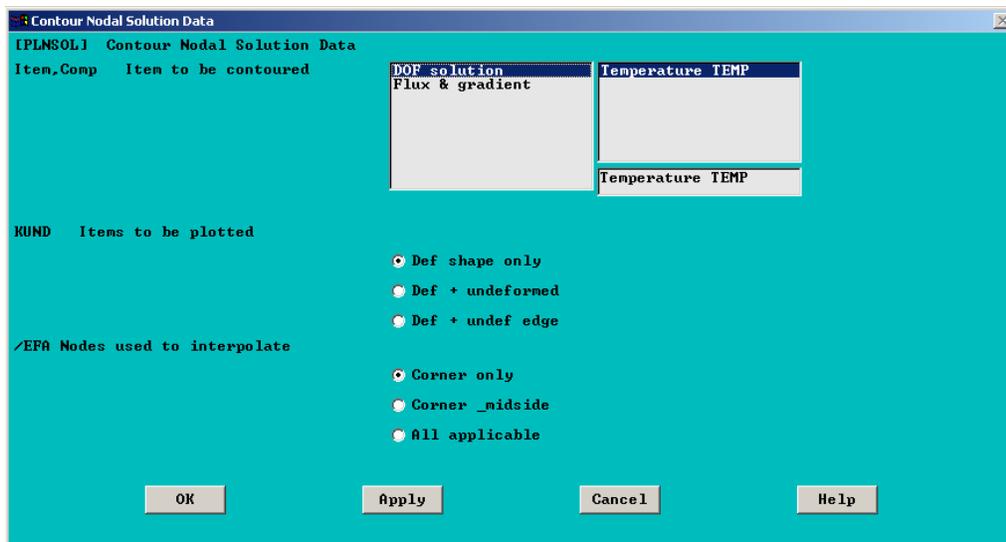
**Temperature Distribution Color Contour Plot:**   Note that the temperature values corresponding to the colors are shown in the legend to the right.
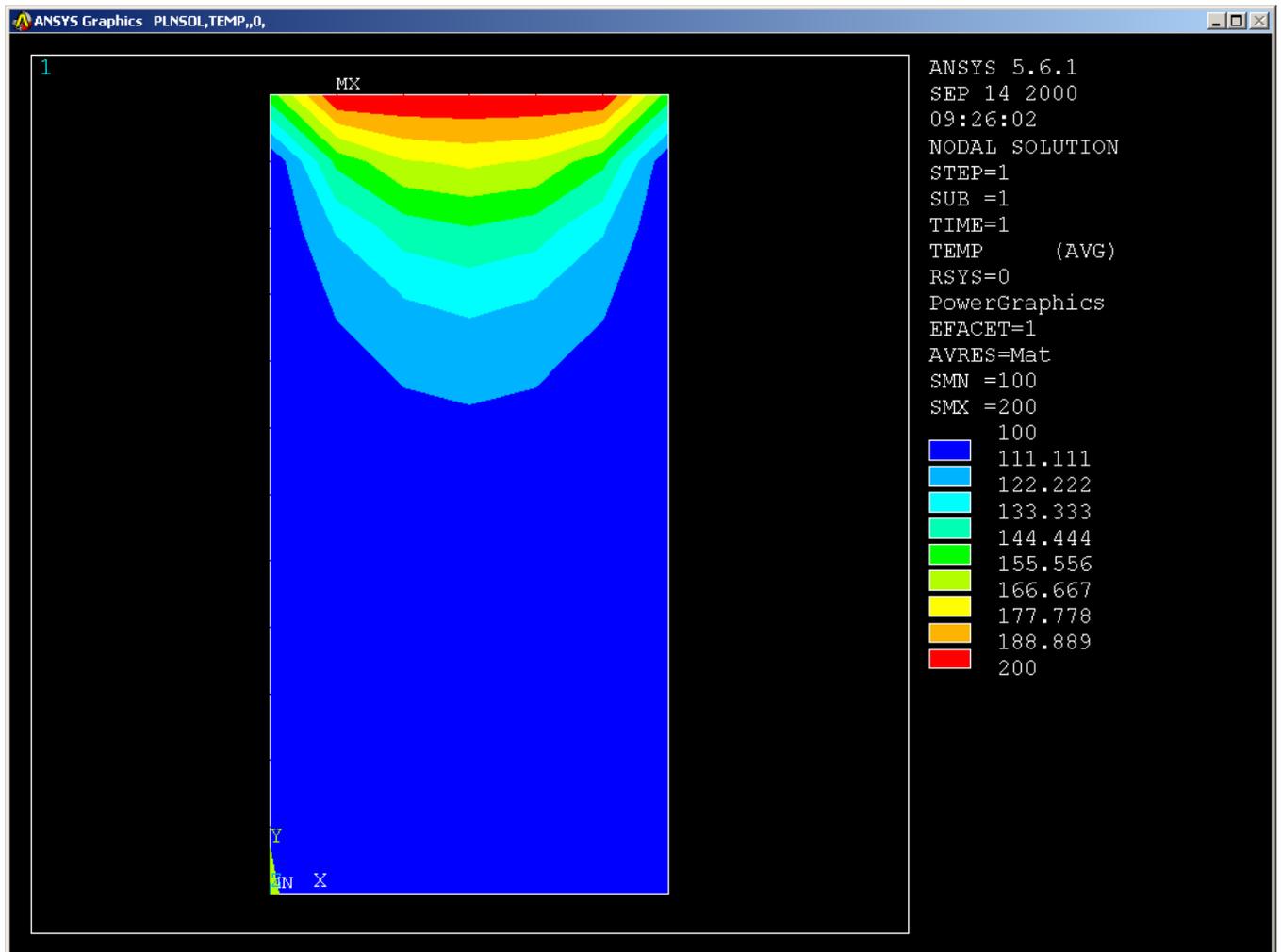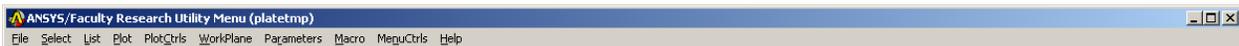
**10. Select nodes along the plate center (x=5 meters).**

For comparison with the analytical solution, you will need a listing of specific temperatures at specific locations in the plate. ANSYS has calculated a temperature at each "node". Because of our method of creating the model by automatic "meshing" of the rectangle, at this time, we do not know specific node numbers at specific locations. But, we can get a listing of node numbers, including the locations of each node, and also a listing of temperatures by node numbers. To keep the amount of information to a workable level, it is probably best to include in these lists only a subset of nodes. To get such a list, we can first select only the nodes at x=5 meters. This is a case where it is probably easiest to just use the direct command line entry option, rather than operate through the menus. On the command line, type "nsel,s,loc,x,5", as shown below, and hit enter:
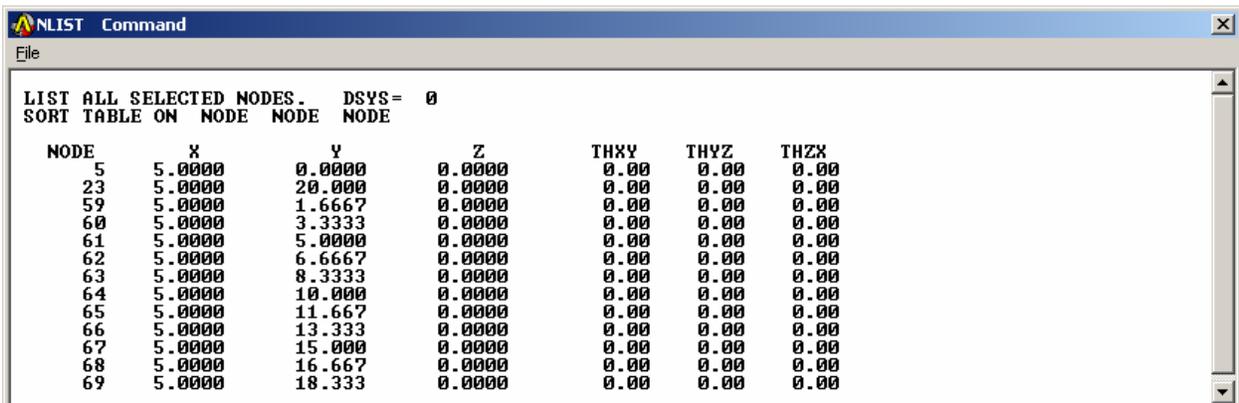
```
ANSYS Input                                                    ×

Pick a menu item or enter ANSYS Command below (POST1)
nsel,s,loc,x,5
```

**11. List the locations of the selected nodes.**

On the top "Utility Menu":

```
ANSYS/Faculty Research Utility Menu (platetmp)
File  Select  List  Plot  PlotCtrls  WorkPlane  Parameters  Macro  MenuCtrls  Help
```

Choose *List -> Nodes*. In the green box that appears, just click on "OK" at the bottom. A listing box appears:

```
NLIST  Command                                                 ×
File

LIST ALL SELECTED NODES.    DSYS=  0
SORT TABLE ON  NODE  NODE  NODE

  NODE      X          Y          Z         THXY    THYZ    THZX
     5    5.0000     0.0000     0.0000      0.00    0.00    0.00
    23    5.0000    20.000      0.0000      0.00    0.00    0.00
    59    5.0000     1.6667     0.0000      0.00    0.00    0.00
    60    5.0000     3.3333     0.0000      0.00    0.00    0.00
    61    5.0000     5.0000     0.0000      0.00    0.00    0.00
    62    5.0000     6.6667     0.0000      0.00    0.00    0.00
    63    5.0000     8.3333     0.0000      0.00    0.00    0.00
    64    5.0000    10.000      0.0000      0.00    0.00    0.00
    65    5.0000    11.667      0.0000      0.00    0.00    0.00
    66    5.0000    13.333      0.0000      0.00    0.00    0.00
    67    5.0000    15.000      0.0000      0.00    0.00    0.00
    68    5.0000    16.667      0.0000      0.00    0.00    0.00
    69    5.0000    18.333      0.0000      0.00    0.00    0.00
```

You can get a hard copy of the information in this box by clicking, in this box (see above): *File -> Print.* It should also be possible to save this information to a file using

the option, *File -> Save As*.  However, this "Save As" option does not appear to be working properly in the current ANSYS version.

**12. List the temperatures at each of these nodes**.

*General Postproc -> List Results -> Nodal Solution*

In the box that appears, highlight "DOF Solution" and "Temperature TEMP", as shown, then click "OK".
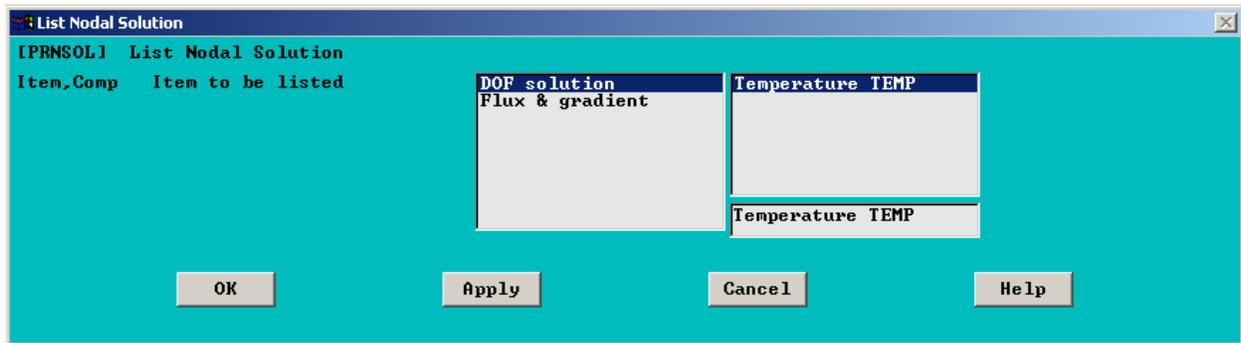


A listing, as shown below, should appear.  The locations of the same nodes have already been listed, in Step 11 above, so the results for these nodes can be checked with the analytical solution.
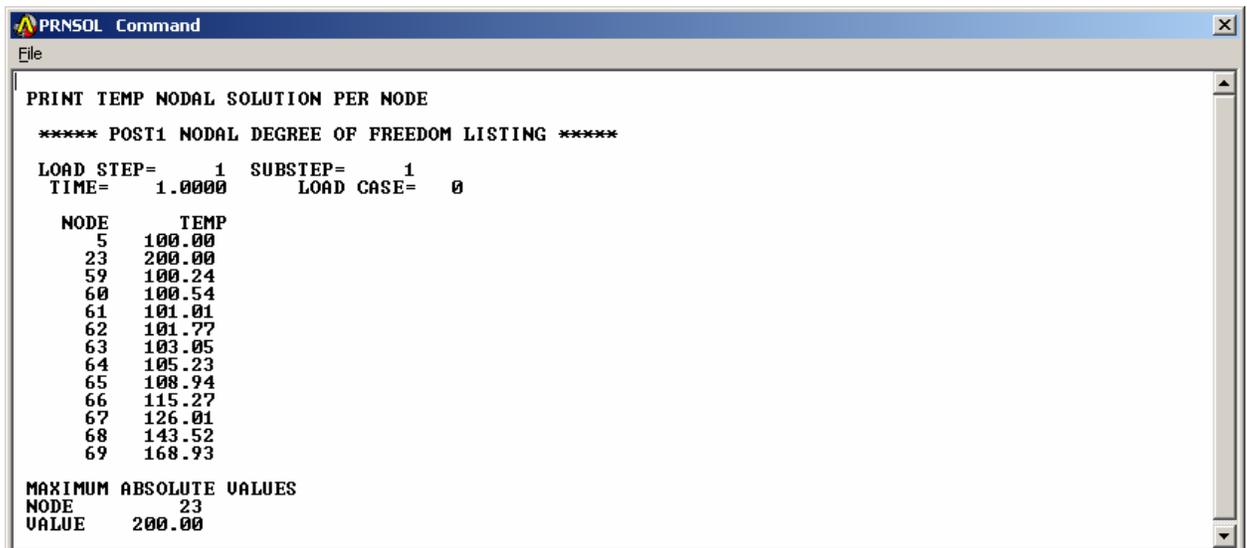
Re-select all nodes in the model for additional plotting, or listing, as desired. To do this, simply type, at the command line: "allsel" and hit enter:

```
ANSYS Input                                                    ⊠

Pick a menu item or enter ANSYS Command below (POST1)
allsel
```

Subsequent lists and plots will include all nodes. Steps 10, 11, and 12 could be repeated to get listings of temperatures of nodes at other locations.

## 13. Modify Mesh / Re-analyze

The general idea in finite element modeling is that the finer the mesh (i.e. the more nodes and elements used), the more accurate the solution. There is a cost associated with using a large model, however. The solution time, the computer memory requirements, and the computer hard drive space needed increase as the mesh density is increased. For this particular analysis, however, unless an extremely fine mesh is used, the solution will probably be completed very quickly, and the available memory and hard drive space should be sufficient. This same problem can be re-solved, with a finer mesh, without starting back at the beginning of these instructions. To do this, follow the steps below:

13a) Clear the existing nodes and elements:

*Preprocessor -> -Meshing- Clear -> Areas*

A picking menu appears. Select "Pick All". At this point, the model in the graphics window will probably disappear. However, the rectangular area still exists. You have only deleted the nodes and elements. To see the rectangular area in the graphics window, you may need to click on "Plot" on the top "Utility Menu",

```
ANSYS/Faculty Research Utility Menu (platetmp)                            _ |□| ✗
File  Select  List  Plot  PlotCtrls  WorkPlane  Parameters  Macro  MenuCtrls  Help
```

Then, choose "Areas". The rectangle should appear.

13b) Clear the previously specified mesh controls:

*Preprocessor -> -Meshing- Size Controls -> -Lines- Clr Size*

A picking menu appears. Select "Pick All".

13c) Specify new mesh controls. Repeat step 5 from the initial analysis. Except, in this case, use larger values for number of element divisions (NDIV). In step 5 previously, the number of element divisions chosen along the bottom horizontal line was 6, and the number of element divisions chosen along a vertical line was 12. This time, increase

these numbers.  However, if you want to compare your results with the analytical solution along the x=5 meter location, you will need to make sure you use an even number of divisions along the horizontal line.   That way, when you select nodes at x=5, there will be nodes along that center line.

13d) Remesh the model.  Repeat step 6 from the initial analysis.

13e) Re-solve.  Repeat step 8 from the initial analysis.  You do not need to repeat step 7 from the initial analysis.  Specified temperatures were applied to lines, instead of directly to nodes.  So, even though we re-meshed, the temperature boundary conditions still exist, because we never modified the lines that define the large rectangle.

13d) Post-process to obtain results for the finer mesh. Repeat steps 9-12 from the previous analysis.  For a finer mesh, you should see smoother curves on the temperature contour plot, and the calculated temperatures should be in better agreement with the analytical solution.


**14. Exit ANSYS, Saving All Data.**  On the ANSYS Toolbar, shown below, choose:

*Quit ->Save Everything -> OK*



To recall the model and solution at a later date, assuming you have deleted no files, simply re-launch ANSYS, specify the same working directory as before, re-issue the same jobname as used in Step 1 of these instructions, and then click on "RESUME_DB" on the ANSYS Toolbar shown above.

To see the resumed model in the graphics window, you may then need to click on "Plot" on the top "Utility Menu",



Then, choose either "Elements", "Nodes", or "Areas", depending on which entities you wish to plot.