Thermal 5: 3D Heat Conduction within a Solid(Carnegie Mellon University)

Introduction:

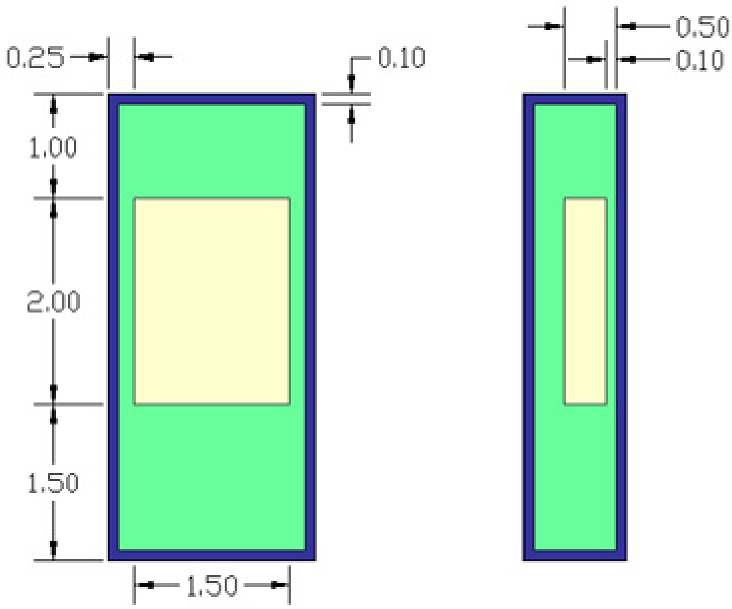
In this example you will learn to build and assess #3D geometries in heat transfer by modeling an object subjected to requirements and specific boundary conditions. Using ANSYS will allow you to output the temperature distribution in an extremely simple and accurate way.

Problem Description:

- We assume that our phone is a rectangular solid, with filleted corners as they appear in the image.
- All units are S.I.
- **Boundary Conditions:**
  - 2) All faces except that of the battery have convective boundary layers.
  - 3) The battery generates heat at a rate of 50 W/m^2/s.
  - 5) **Heat is uniformly generated** in the bock at a rate of 20 W/m^2.
- **Material Properties: (Steel)**
  - h = 50 W/(m^2*K)
  - k(innards) = 10 W/m-K
  - k(lithium) = 84.8 W/m-K
  - k(plastic) = 0.18 W/m-K
- **Objective**: To determine the nodal temperature distribution and create contour plot.
- **Dimensions**
  1. **Cellphone: 0.1143m** long x **0.0254m** thick x **0.0508m** wide (4.5 inch x 1 inch x 2 inch)
  2. **Battery:** Length: **50.8 millimeters** Thickness: **10.16 millimeters** Width **38.1 millimeters** (2 inch x 0.4 inch x 1.5 inch)

Note that the actual dimensions of the battery are: Length: **53 millimeters** Width **37 millimeters** Thickness: **10 millimeters** (2.087 inch x 1.457 inch x 0.3937)
We will use approximations because the heat transfer will still display the same general distribution

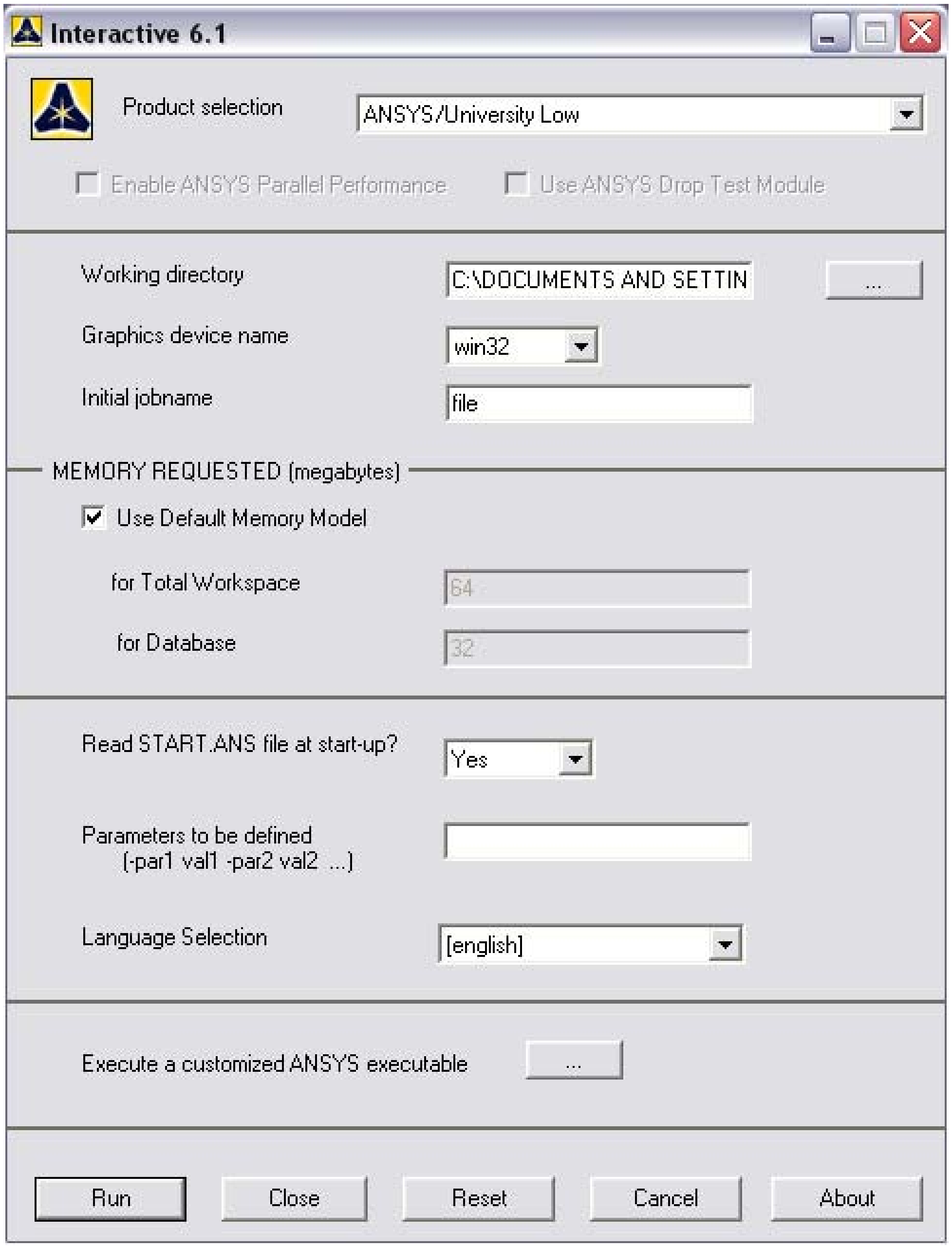
Note: For any necessary conversions, this site is useful: http://www.convert-me.com/en/convert/length



The dimensions of the drawing are in English because the specs of the phone given on the web are in English (making the CAD drawing easier to build in English)
REMEMBER TO CODE ANSYS WITH **SI, not English**
Note: .1 inch = 2.54 mm
Also, R0.50in = 0.0127m

Starting ANSYS:

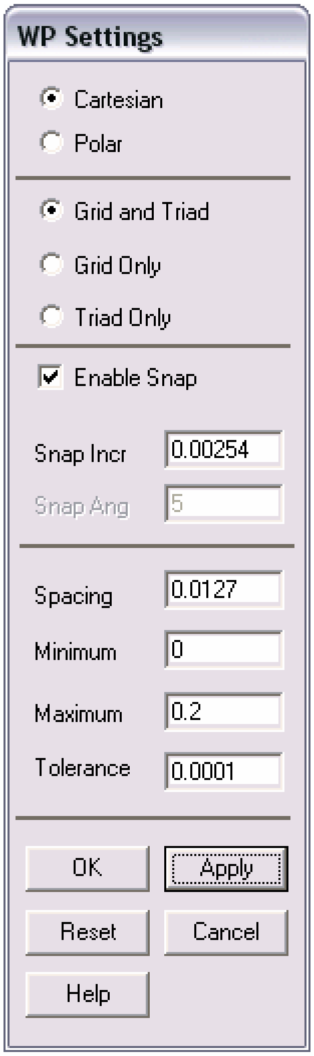
- Click on **ANSYS 6.1** in the programs menu.
- Select **Interactive**.
- The following menu comes up. Enter the working directory. All your files will be stored in this directory. Also under **Use Default Memory Model** make sure the values **64** for **Total Workspace**, and **32** for **Database** are entered. To change these values unclick **Use Default Memory Model.**



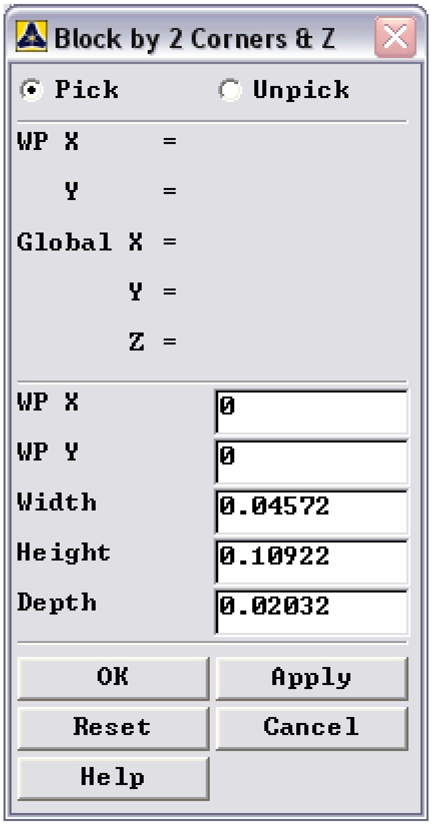
- Click **RUN**

Modeling the Structure:

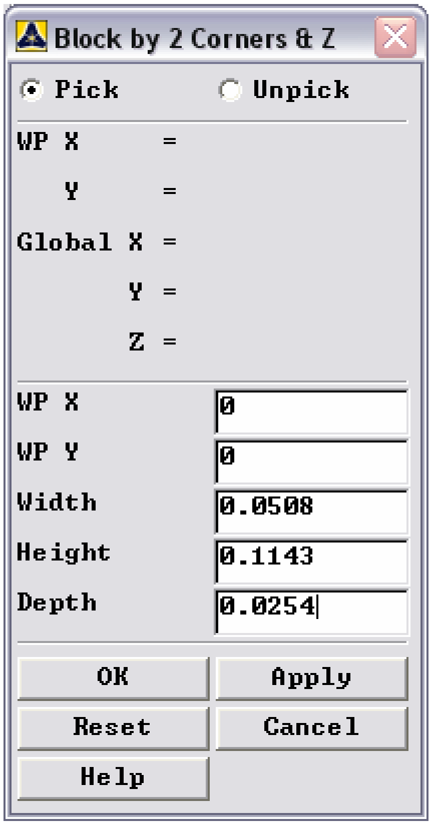
- Go to the ANSYS Utility Menu (the top bar)
- Click **Workplane>WP Settings...**
- The following widow comes up:



- Check the **Cartesian** and **Grid and Triad Only** buttons
- The first step is to create the inner volume to represent the space in the phone that is occupied by the microchips and transistors.
- Enter the values shown in the figure above and then click **OK**. Note that we are using a spacing increment of .1 inches or 2.54 millimeters. This will help in a modeling step.
- Go to the ANSYS Main Menu (on the left hand side of the screen) and click **Preprocessor>Modeling>Create>Volumes>Blocks>By 2 Corners and Z**
- The following window comes up:

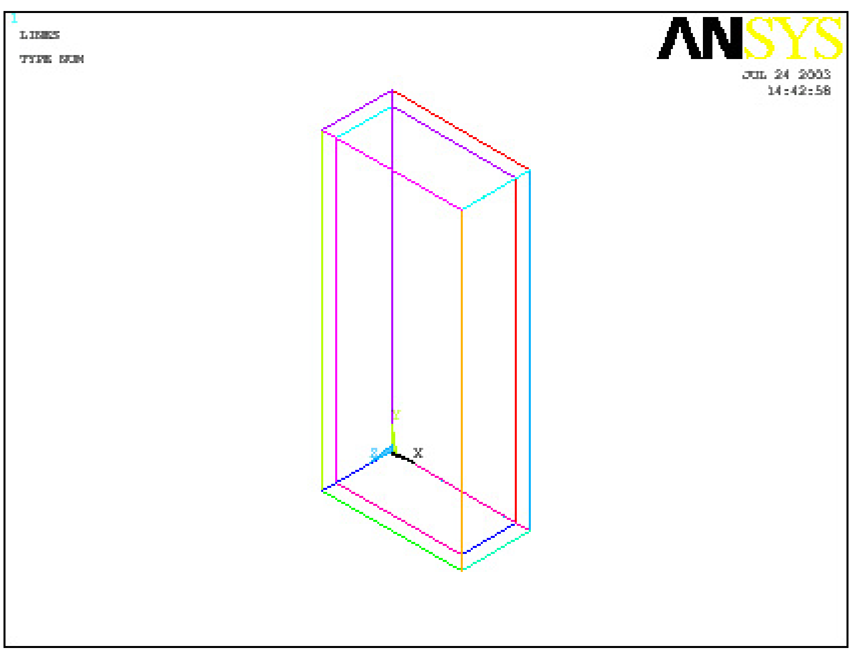


- Enter the values as shown and click **OK**.
- Now change the view to isometric mode, using **Menu>Plot Controls>Pan Zoom Rotate** and by clicking the ISO button. The plot should have zoomed to the new part.
- Now you have created the external phone. If at any time you cannot see the complete Workplace then go to **Utility Menu>Plot Controls>Pan Zoom Rotate** and zoom out to see the entire Workplace. If you want to see the grid itself, go to **Utility Menu>Workplane>Display Working Plane**
- The next step is to create the outer volume.
- Go to **Utility Menu>Workplane>Offset WP by increments** and click these buttons to offset the workplane by -.1 inches in each direction. This will enable you to create the inner volume easily.
- Next use **Preprocessor>Modeling>Create>Volumes>Blocks>By 2 Corners and Z** again and this time enter the following:

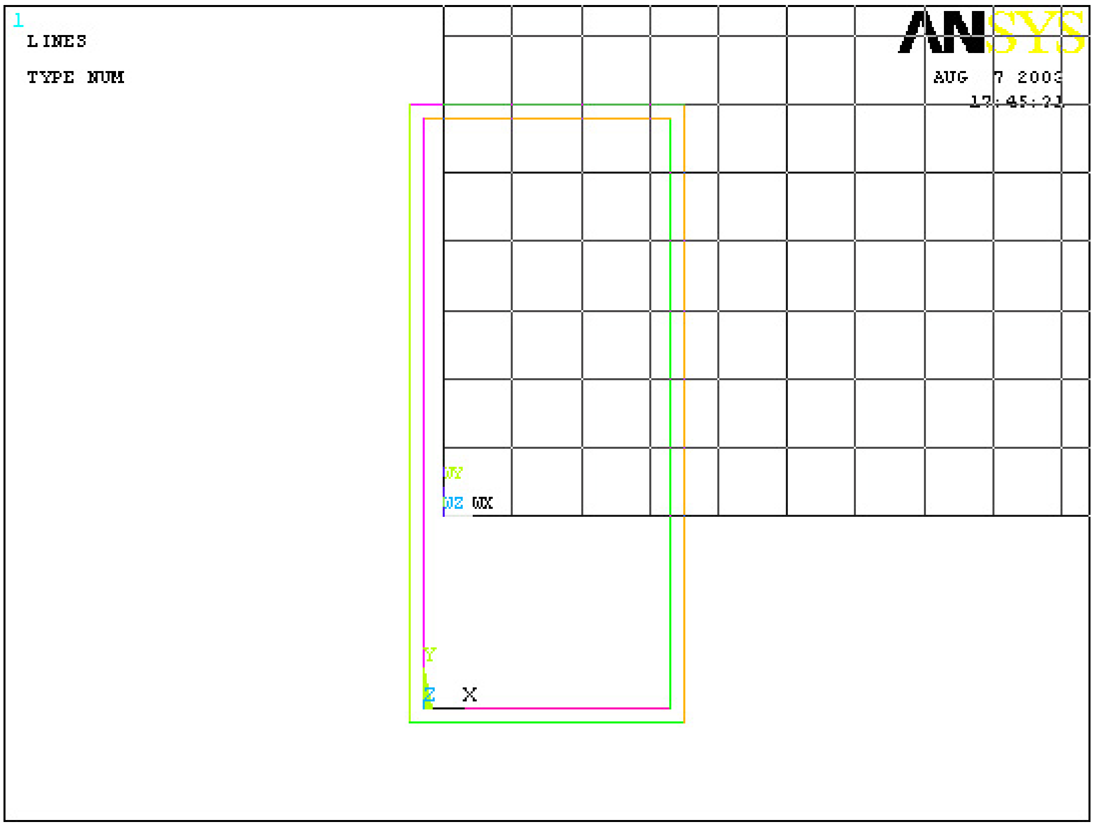


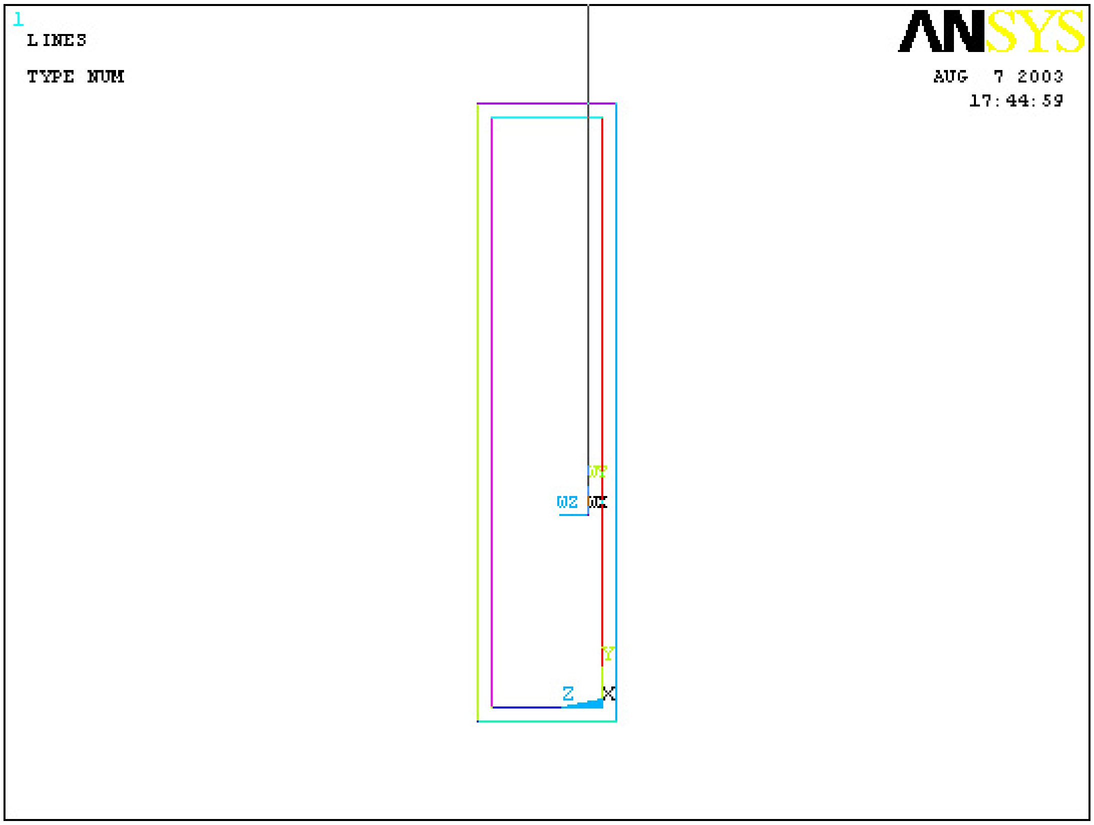
Note that these dimensions are simply the previous ones, minus 0.00508 which is .1x2 inches.

- If you messed something up, remember not to select **Pick All** when deleting anything now, since you don't want to destroy the model of the inner volume.
- The model should look like this now if you plot lines (**Utility Menu>Plot>Lines**)and dynamically rotate the solid(**Utility Menu>PlotCntrls>Pan Zoom Rotate**): (note, you have a black background)

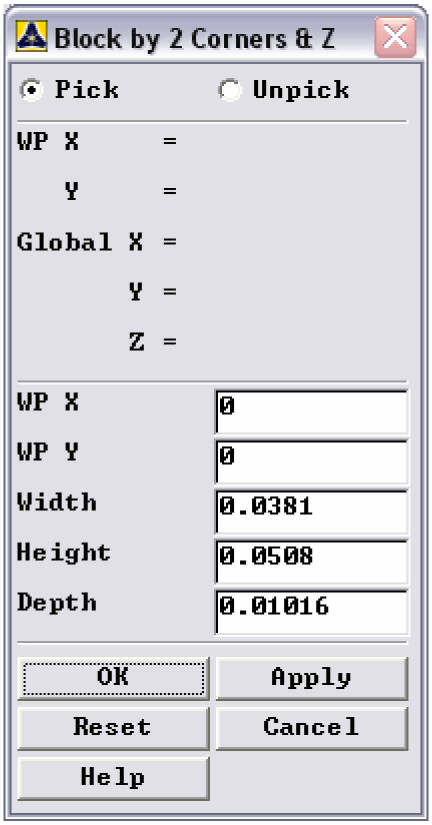


- The next series of steps involves creating the volume for the battery.
- First, use Offset WP by increments and increment the Workplane by **two positive increments** in the **Z** direction. Next, change the snap and grid increments of the workplane settings to **0.0381 m** (1.5 inches, the distance between the bottom of the phone and the beginning of the battery). Use Offset WP and offset in the **Y** direction by **one positive increment**. Next, change the increments once more to **0.00635 m** (0.25 inches). Displace the WP in the **X** direction **one positive increment**.
- The workplane will appear here: Note that The Pan Zoom Rotate settings here are "Front" instead of ISO and only the lines are plotted: Basically, the image should look like the CAD drawing from the top of the tutorial, and the bottom corner is where the battery will be referenced from.

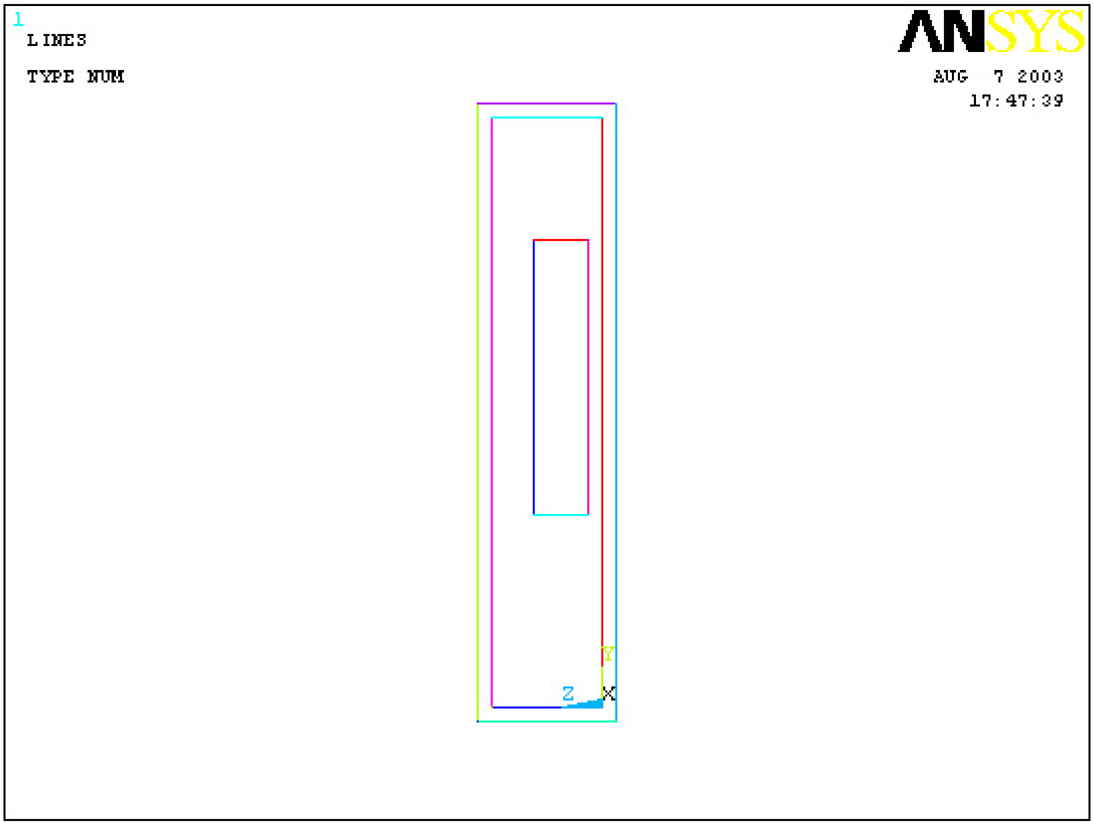




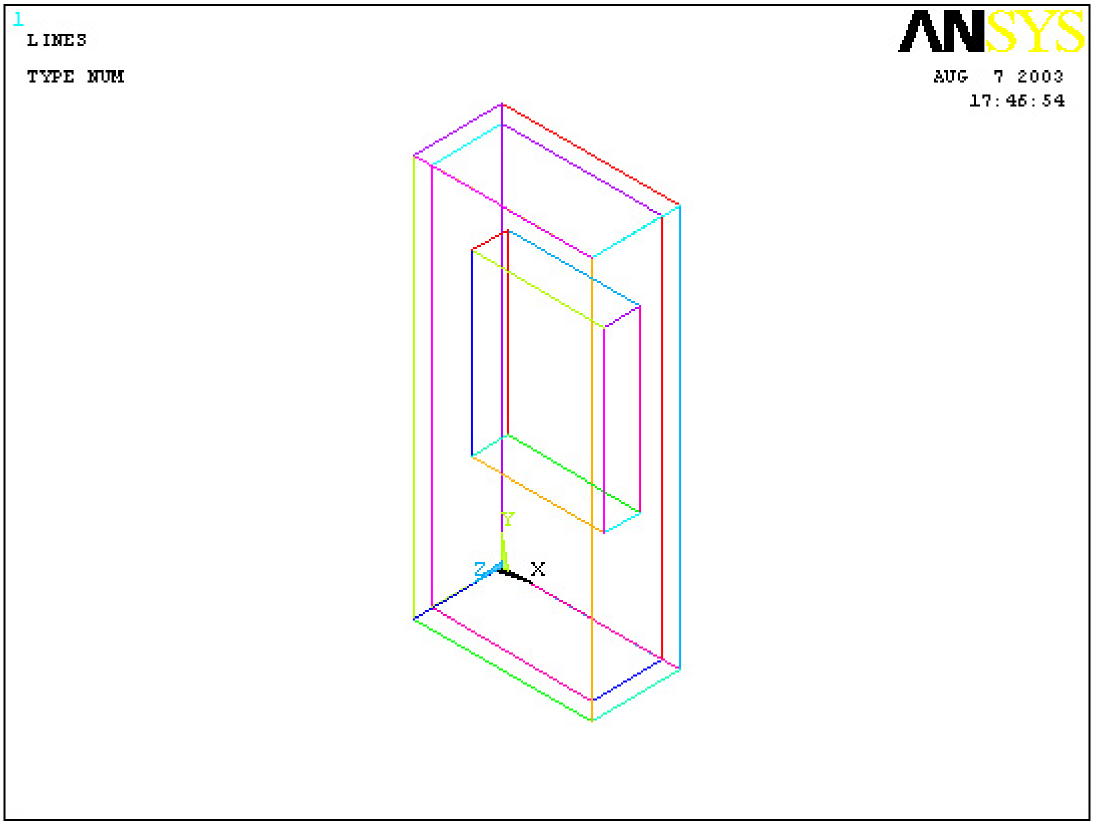
- Now create the volume for the battery:



- The completed model will look like this: (note that I did not replot the volumes because we wouldn't be able to see any of the inner volumes!



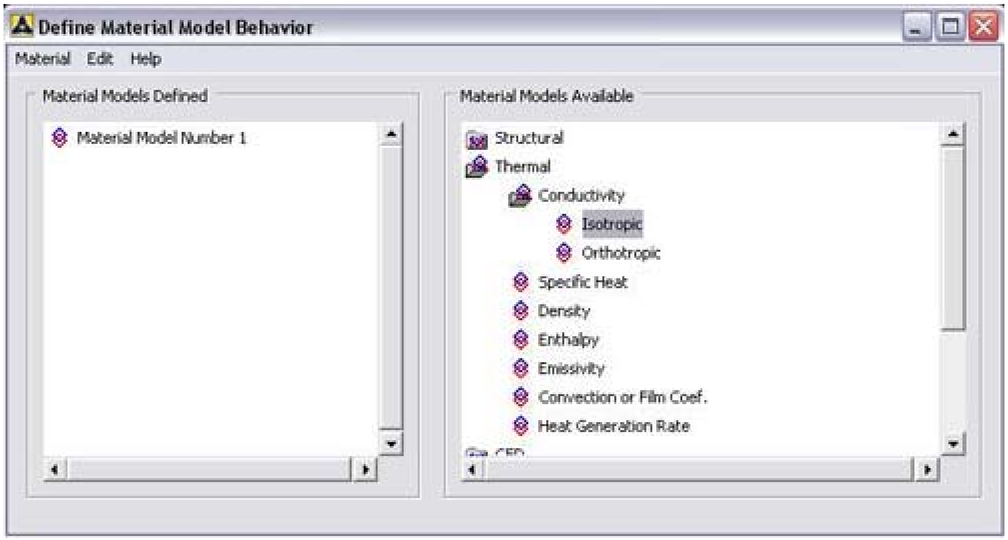
And in ISO mode



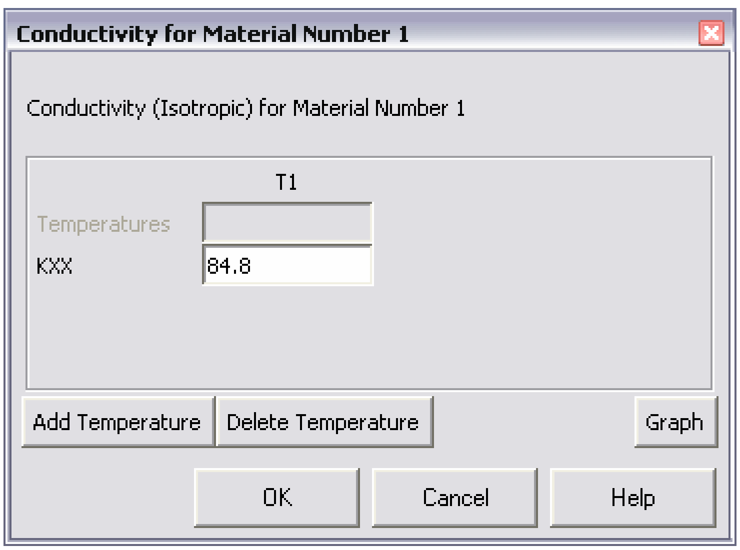
- One final step that needs to be executed involves explaining that the volumes overlap each other, so that when meshing, the volumes are separate from each other. This is accomplished by choosing **Preprocessor>Modeling>Operate>Overlap>Volumes**
- First select the outside layer and the inside layer (not the battery!) and hit Apply. Then select the inside volume and the battery and hit OK. This should resolve all the volumes. You can test this but plotting lines and then trying the step again, the volumes should each be selected separately… if so, then hit cancel and move forward.

Material Properties:

- Now that we have built the model, material properties need to be defined such that ANSYS understands how heat travels through this **composite** solid.
- Go to the ANSYS Main Menu
- Click **Preprocessor>Material Props>Material Models**.
- The pop-up window will now look like this:



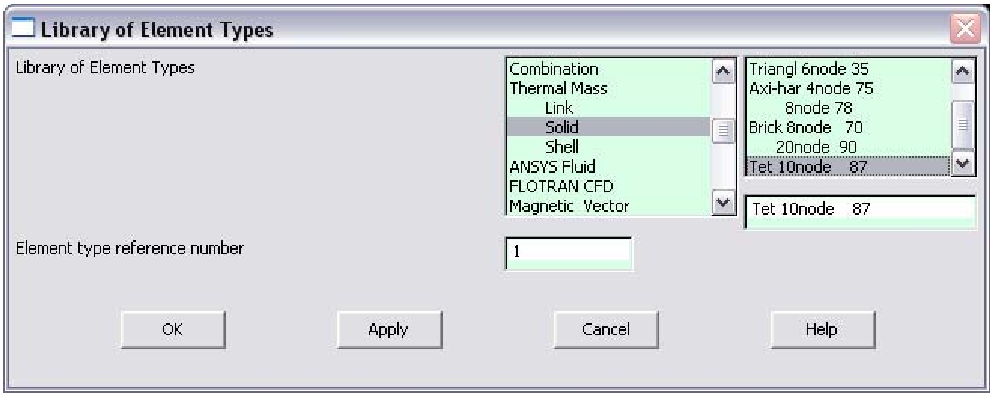
- In the window that comes up choose **Thermal>Conductivity>Isotropic**. (Double click Isotropic). The following window comes up:



- Fill in **84.8** for Thermal conductivity. This defines the conductivity of Lithium and correlates material 1 with it. Click **OK**.
- Choose **Define Material Model Behavior>Material>New Model** and define another conductivity for the new model, that of **the innards** (10 W/m K) and then repeat to define the Plastic case of the phone (**0.18** W/m K)
- Now exit the "Define Material Model Behavior" Window

Element Properties:

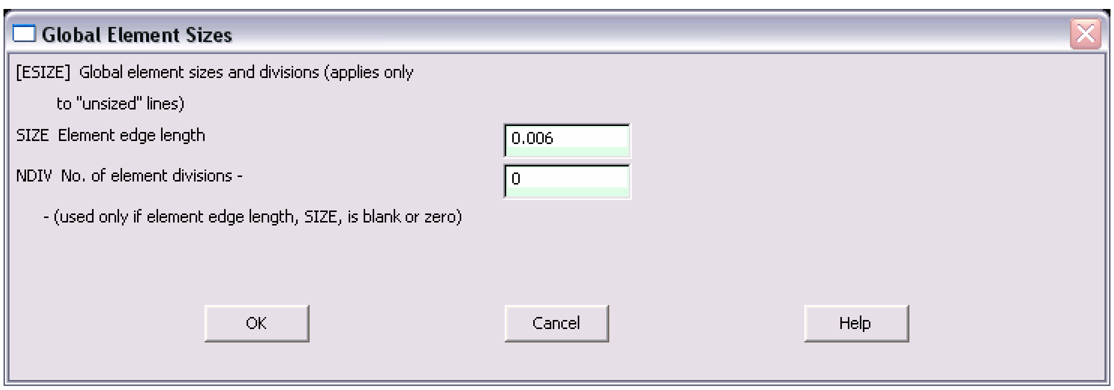
- Now that we've defined **what** material ANSYS will be analyzing, we have to define **how** ANSYS should analyze our block.
- Click **Preprocessor>Element Type>Add/Edit/Delete...** In the **'Element Types'** window that opens click on **Add...** The following window opens:



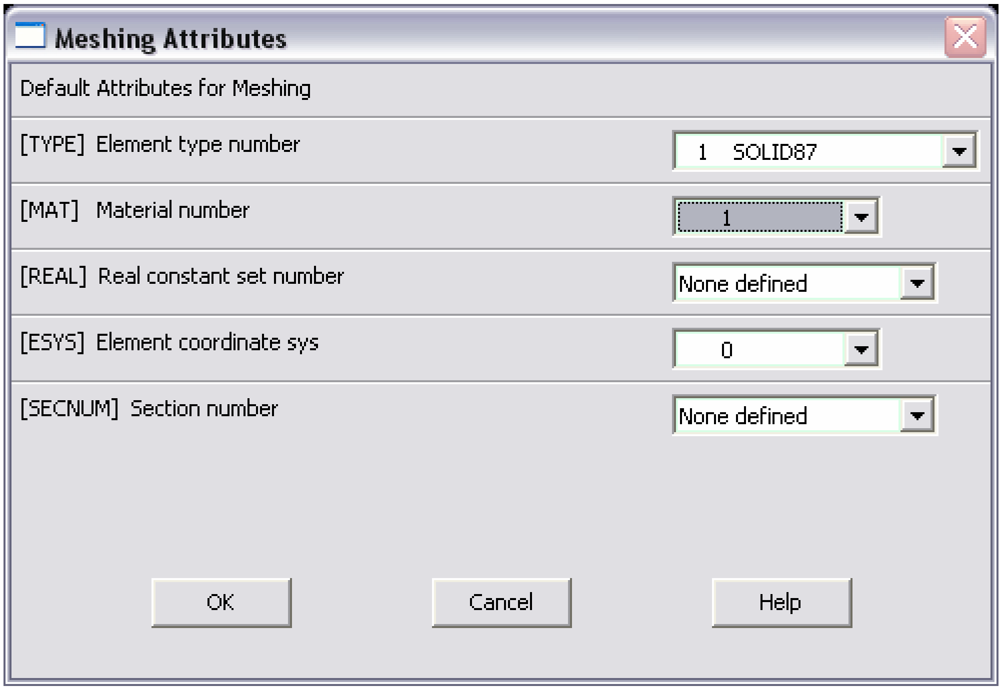
- Type **1** in the **Element Type reference number**.
- Click on **Thermal Mass Solid** and select **Tet 10node 87**. Click **OK**. Close the **'Element types'** window.
- Now we have selected **Element Type 1** to be a **Thermal Solid 10node Element**. This finishes the section defining how the part is to be analyzed.

Meshing:

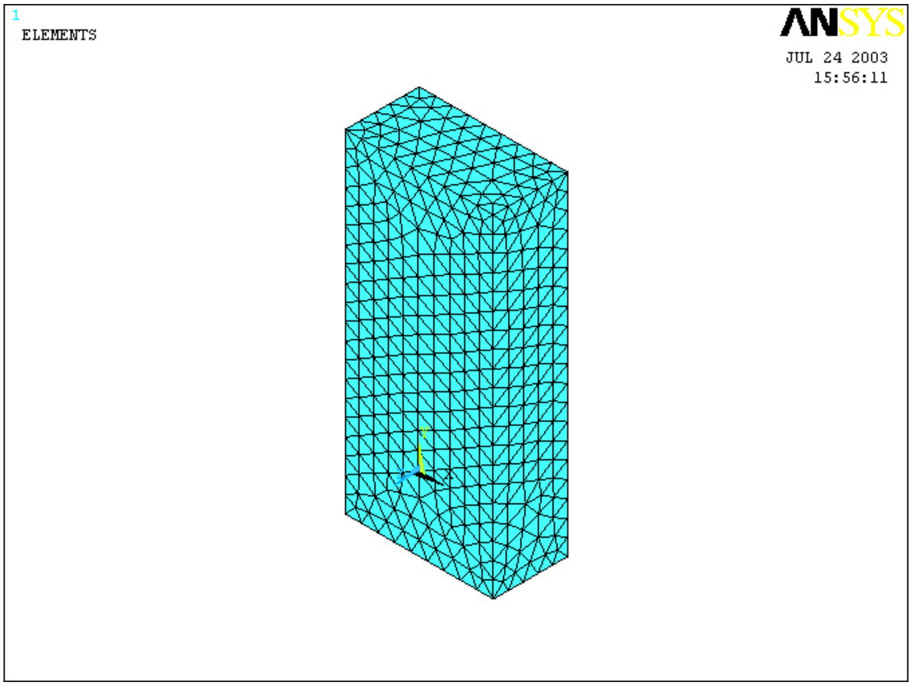
- This section is responsible for telling ANSYS how to divide the block such that it has enough nodes, or points, to analyze to make an accurate enough analysis.
- Go to **Preprocessor>Meshing>Size Controls>Manual Size>Global>Size**. In the menu that comes up type **0.006** in the field for "**Element edge length**".



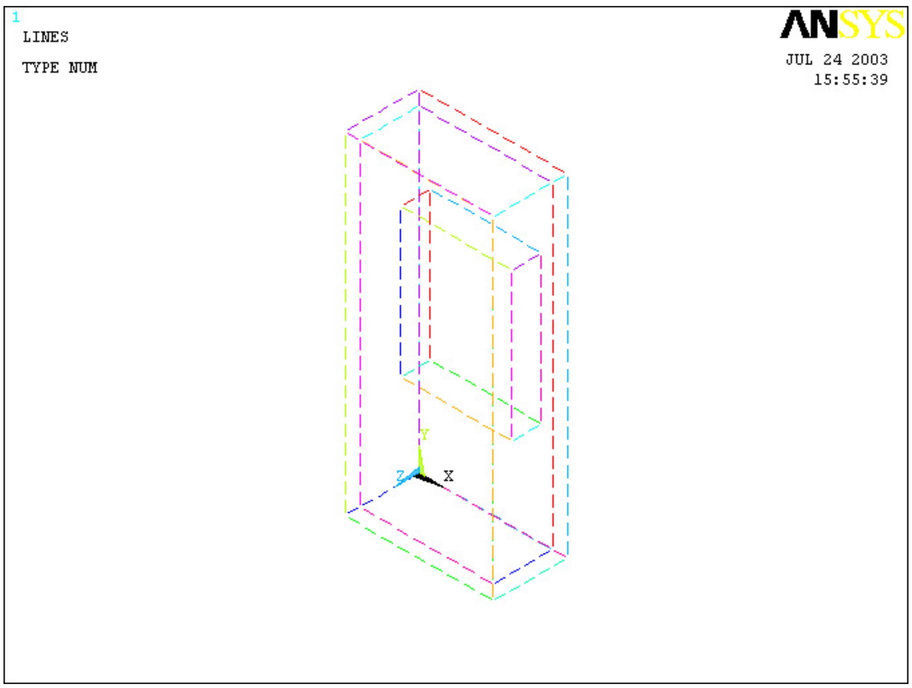
- Click on **OK**. Now when you mesh the figure ANSYS will automatically create square meshes that have an edge length of **0.006m** along the lines you selected.
- Now go to **Preprocessor>Meshing>Mesh Attributes>Default Attributes**. The window is shown below:



- Here you finally put together material model and material type. Select the appropriate material to mesh (first **1**, lithium, as defined in the Material Properties section) and pick what element type to use (selected by the **Element Type Number**). Once this has been verified, Click **OK** and proceed to **Preprocessor>Meshing>Mesh>Volumes>Free**
- A popup window will appear on the left hand side of the screen. This window allows you to select the area to be meshed.
- Click anywhere within the lithium battery you created to select the volume and then click **OK** in the pop-up window.
- Return to **Default Attributes** and this time, select Material Number 2, to model the innards of the phone.
- Finally, choose material 3 and mesh the plastic casing.
- The model should now look like this:



ISO ELEMENTS



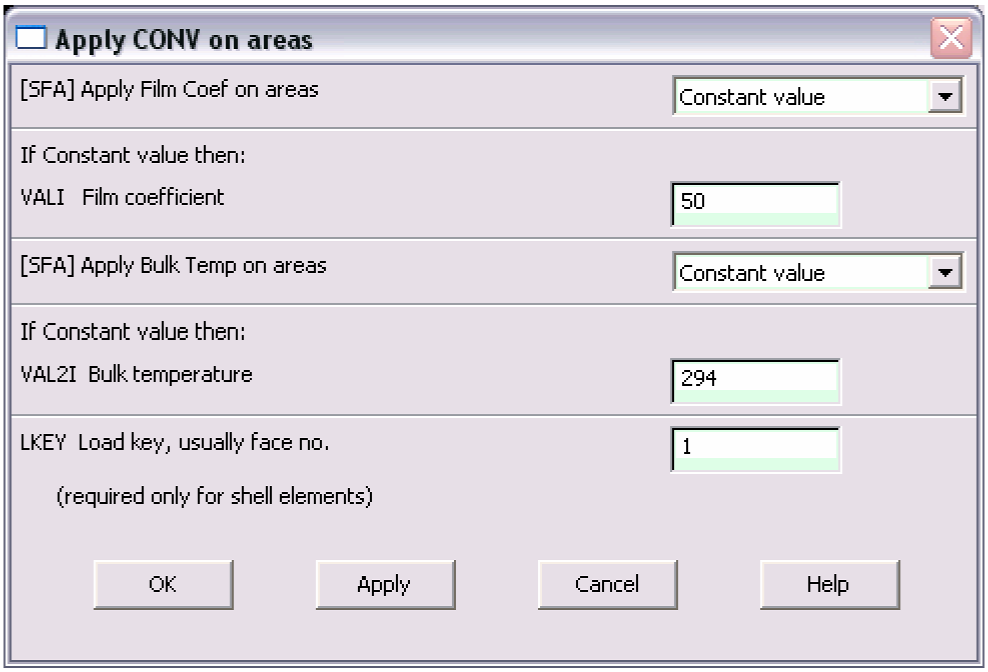
ISO LINES

Boundary Conditions and Constraints:

- Now that we have modeled the phone and defined how ANSYS is to analyze it we will apply the appropriate Boundary Conditions.
- Go to **Preprocessor>Loads>Define Loads>Apply>Thermal** (from here one can apply any of the loads, or Boundary Conditions, offered by ANSYS.)

Apply Convection (Case)

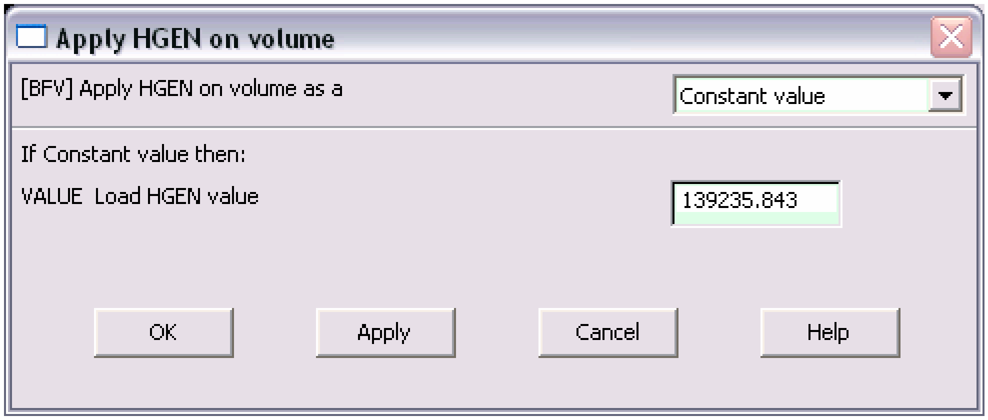
- First we'll apply the Convection Boundary layer around the model. For this click **Convection>On Areas** within the Thermal Load category.
- A dialog window will appear on the left hand side of the screen. This window allows you to select the areas you wish the load to be applied.
- Select the outside areas of the phone and click **OK.** The following window will appear:



- Fill in the **h** value in the **Film Coefficient** blank and the Air temperature in the **Bulk Temperature** blank. Click **OK** when finished.

Apply Heat Generation

- The next step is to add the constraint of heat generation.
- **Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Areas**. (Heat Generat is just short for Heat Generation). You select Areas again because you have to apply this condition uniformly across the block.
- Click anywhere within the area to select it and then click **OK**.
- The voltage of the battery is rated as 3.7V and the internal resistance is on the order of 200 milli ohm. Therefore, the total power is I^2R = 2.738 W which break down to (V = 0.0000196644768 m^3) = 139235.843 W/m^3
- Enter **139235.843** W/m^3 as the heat generation value in the pop-up window that appears:



- Now we have applied all the necessary boundary conditions so we move on to the Solution.

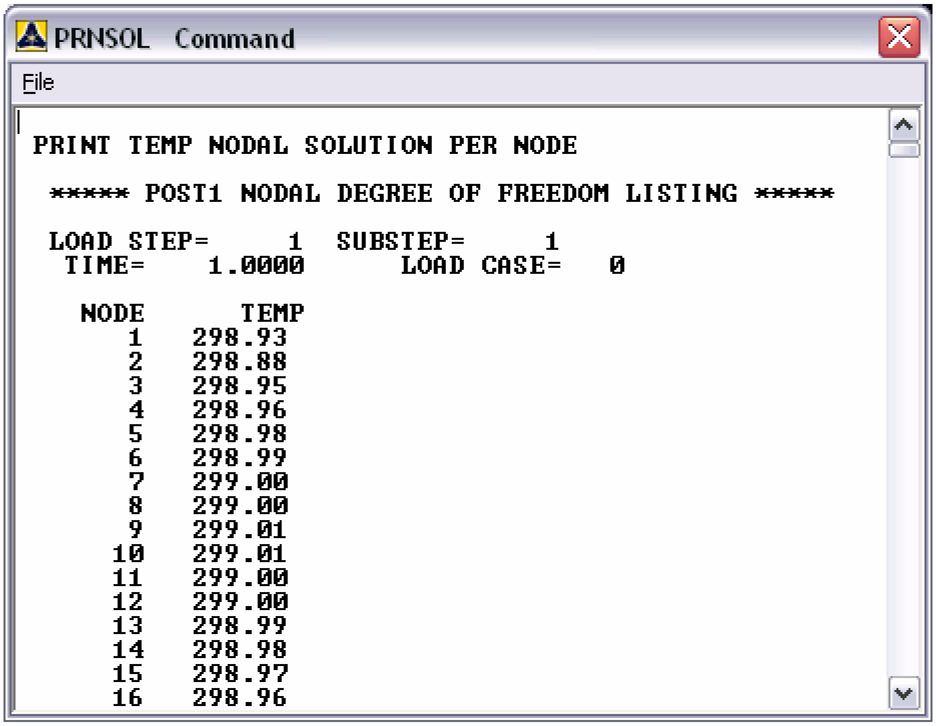
Solution:

- 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:

- This section is designed so that one can list the results of their analysis as a nodal solution
- Go to the ANSYS Main Menu. Click **General Postprocessing>List Results>Nodal Solution**. The following window will come up:

- Select **DOF solution** and **Temperature**. Click on **OK**. The nodal temperatures will be listed as follows:

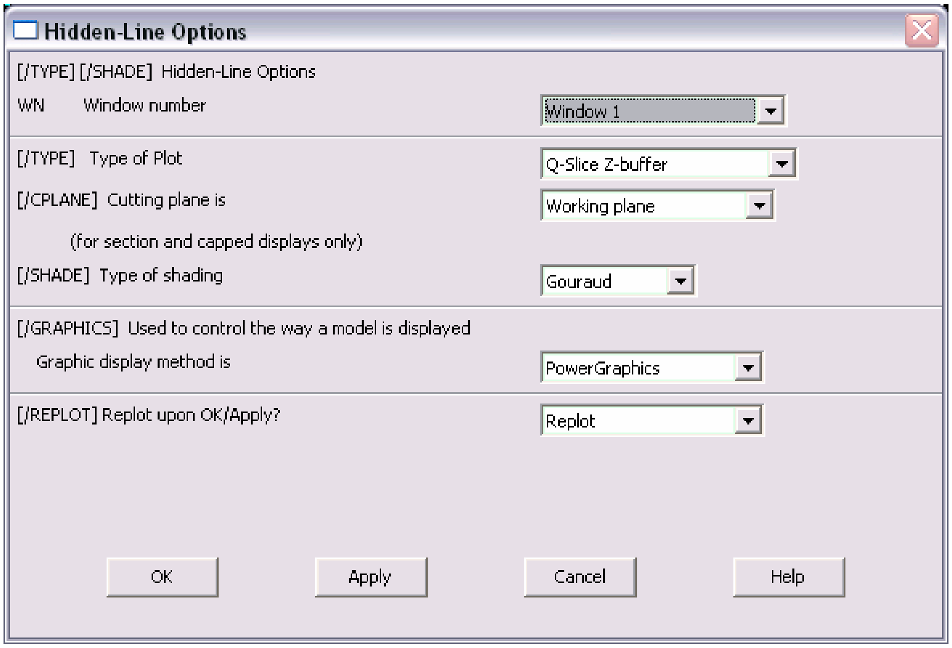


- Within this window one can numerically find the maximum and minimum value of the temperature within the block.

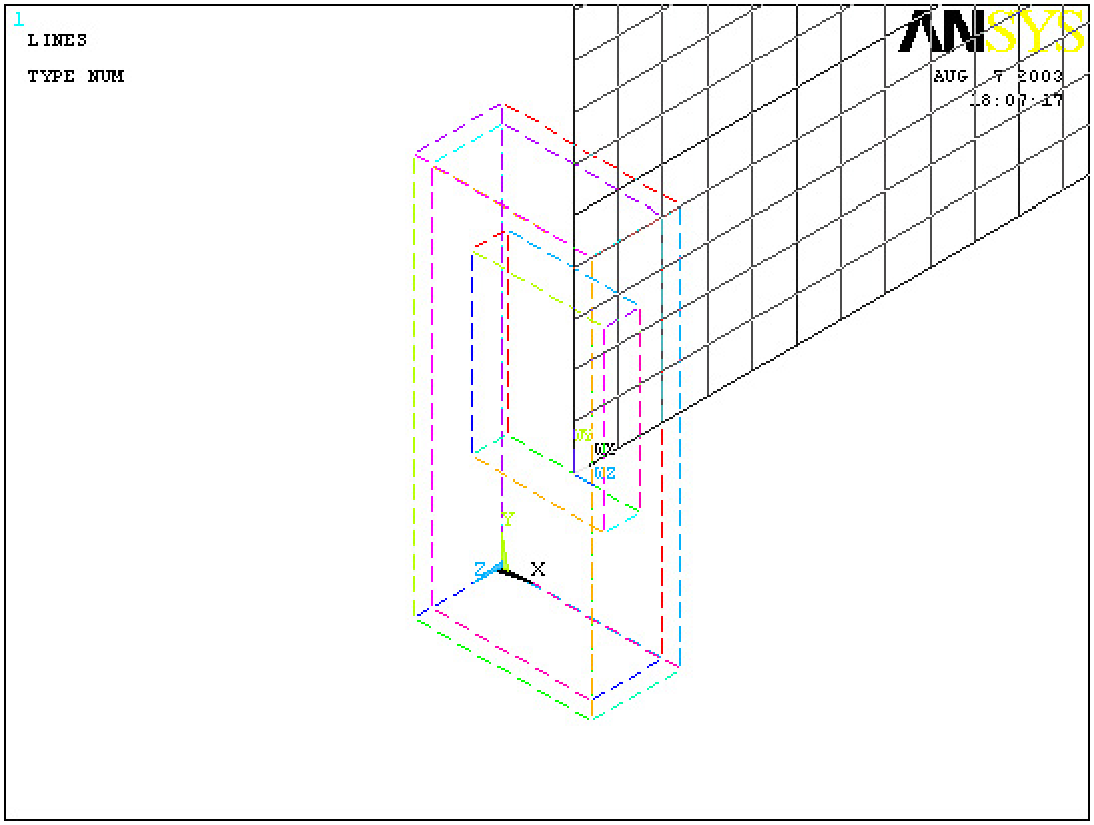
Modification / Plotting the Results:

The last section displayed the numerical results, but most analyses will require a plot of the temperatures on the block in addition to the numerical results. This is how you go about doing that...

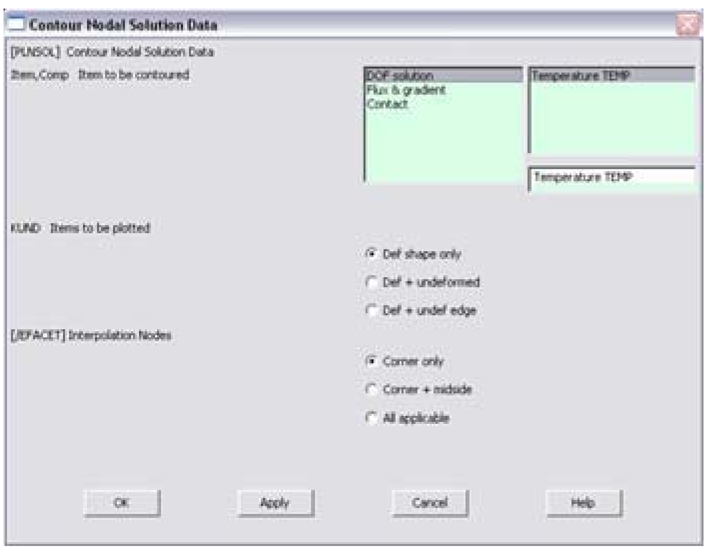
- First go to **Utility Menu>PlotCtrls>Style>Hidden Line Options**
- The following window appears. Choose **Q Slice Z Buffer** and **Working plane** as shown below



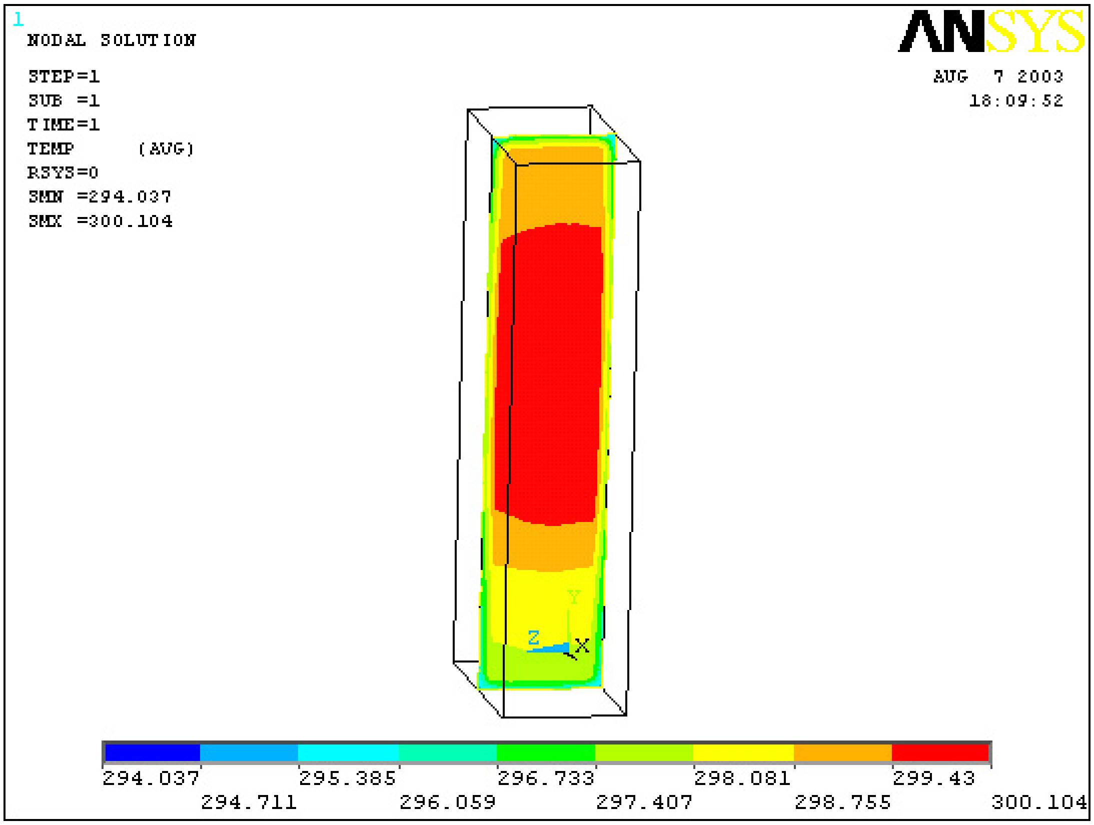
- Now there will be a cross section shown of the temperature distribution in the direction of the workplane. Try rotating the workplane so that this slice is shown (you may want to replot lines):
- To see the workplane, return to **Utility Menu>Workplane>WP Settings...** and choose **Grid and Triad**
- Then use **Workplane>Offset WP by increments** and use offset by angles (change increments to 90 degrees) Offset in the +Y direction 90 degrees, then by 3 snap increments in the +Z direction. Should look something like this:



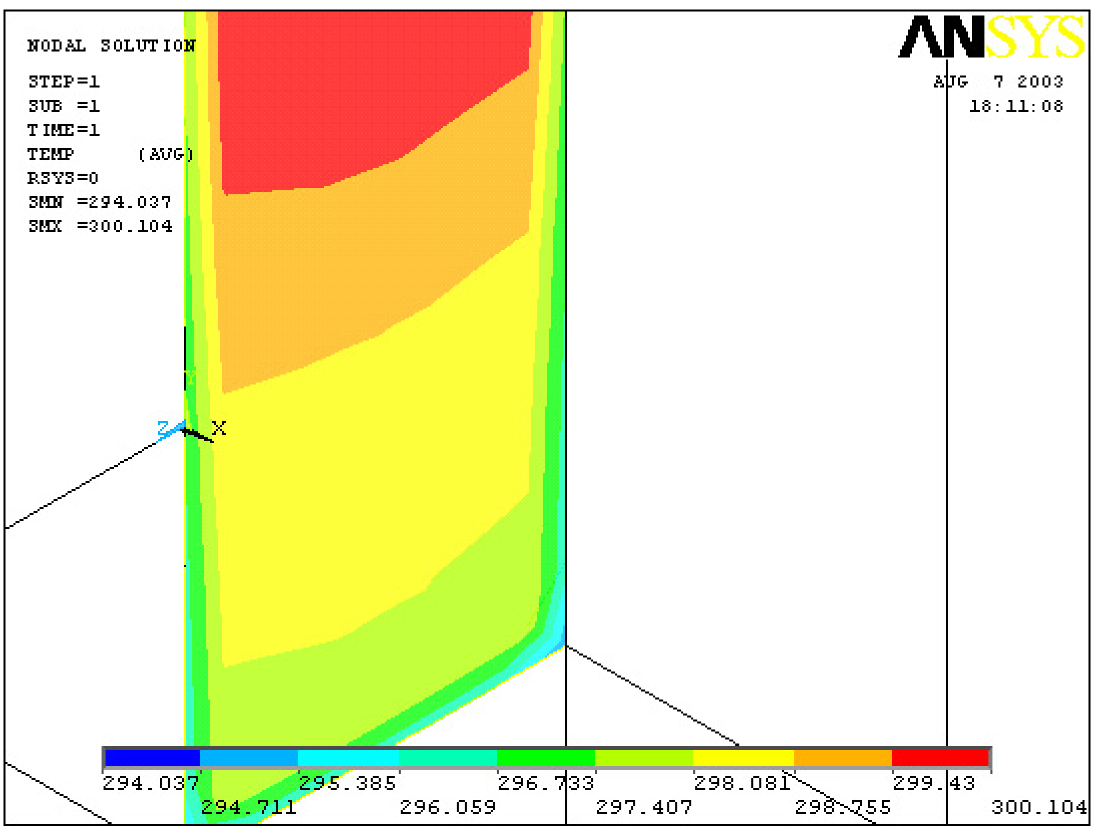
Choose **General Postprocessing>Plot Results>Contour Plot>Nodal Solution**. The following window will come up:



Check the entries and hit OK. The result should be something like below!


1
NODAL SOLUTION
STEP=1
SUB =1
TIME=1
TEMP (AVG)
RSYS=0
SMN =294.037
SMX =300.104
ANSYS
AUG 7 2003
18:09:52
Y
Z
X
294.037
294.711
295.385
296.059
296.733
297.407
298.081
298.755
299.43
300.104

ISO


NODAL SOLUTION
STEP=1
SUB =1
TIME=1
TEMP (AVG)
RSYS=0
SMN =294.037
SMX =300.104
ANSYS
AUG 7 2003
18:11:08
Z
X
294.037
294.711
295.385
296.059
296.733
297.407
298.081
298.755
299.43
300.104

ISO zoomed in on bottom

Now offset the WP back to the original angle (rotate in the –Y direction by 90 degrees) and then choose **Utility Menu>Workplane>Offset WP to>Global Origin.** Change the WP settings to 0.02261 and offset the WP once more by 1 snap increment in the +Z direction. If you replot the contour, you can see that the temperature of the phone near the ear will be on the order of 296 K which is 73.13 degrees Fahrenheit. Some people might find that uncomfortable. This of course, is a crude model of the phone as most of us with phones have experienced more discomfort. Note that the phone is also warmer in the middle area, directly across from the battery…which is sensible and also more likely to be noticed in general.