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:

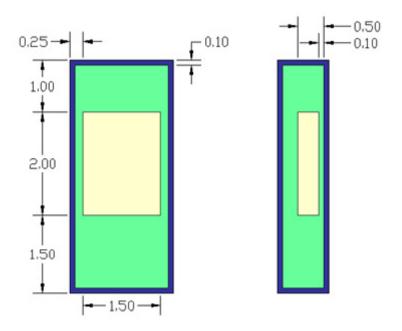
- We assume that our phone is a rectangular solid, with filleted corners as they appear in the image.
- All units are S.I.
- Boundary Conditions:
 - 2) All faces except that of the battery have convective boundary layers.
 - 3) The battery generates heat at a rate of 50 W/m^2/s.
 - 5) Heat is uniformly generated in the bock at a rate of 20 W/m^2.
- Material Properties: (Steel)
 - $h = 50 W/(m^2K)$
 - 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.

A Interactive 6.1	
Product selection	ANSYS/University Low
📕 Enable ANSYS Parallel Pe	rformance 🛛 🗖 Use ANSYS Drop Test Module
Working directory	C:\DOCUMENTS AND SETTIN
Graphics device name	win32
Initial jobname	file
── MEMORY REQUESTED (megab ✓ Use Default Memory Mod	
for Total Workspace	64
for Database	32
Read START.ANS file at star	t-up? Yes ▼
Parameters to be defined (-par1 val1 -par2 val2)	
Language Selection	[english]
Execute a customized ANSYS	5 executable
Run Close	Reset Cancel About

Click RUN •

Modeling the Structure:

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

WP Settings			
	Cartesian		
O Polar			
Grid and	l Triad		
🔘 Grid Onlj	y		
🔿 Triad Or	ıly		
🔽 Enable Snap			
Snap Incr	0.00254		
Snap Ang 5			
Spacing	0.0127		
Minimum 0			
Maximum 0.2			
Tolerance	0.0001		
ОК			
Reset	Cancel		
Help			

- 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:

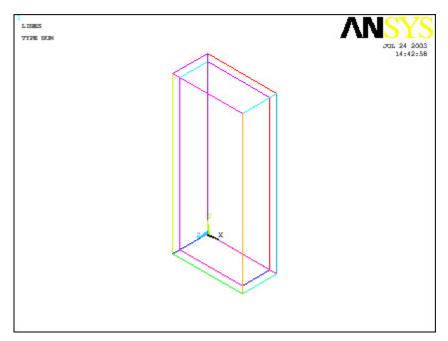
🛽 Block by 2 Corners & Z 🛛 🔛		
• Pick	🔿 Unpick	
WPX =		
¥ =		
Global X =		
¥ =		
Z =		
WP X	0	
WP Y	0	
Width	0.04572	
Height	0.10922	
Depth	0.02032	
ок	Apply	
Reset	Cancel	
Help		

- 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:

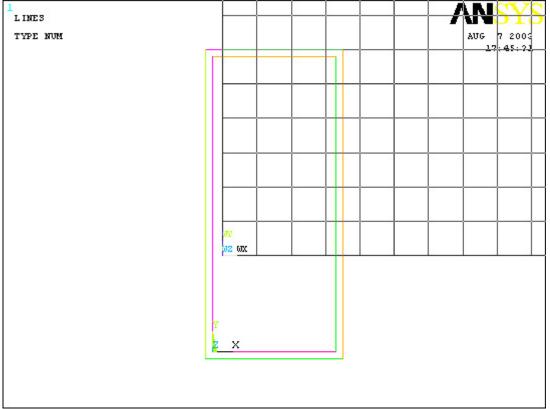
🔼 Block by 2 Corners & Z 🛛 🔀		
• Pick	🔿 Unpick	
WPX =		
¥ =		
Global X =		
¥ =		
Z =		
WP X	0	
WP Y	0	
Width	0.0508	
Height	0.1143	
Depth	0.0254	
ок	Apply	
Reset	Cancel	
Help		

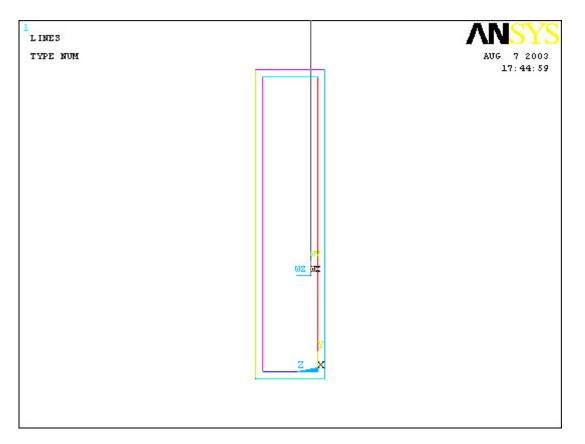
Note that these dimensions are simply the previous ones, minus 0.00508 which is .1x2 inches.

- 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.

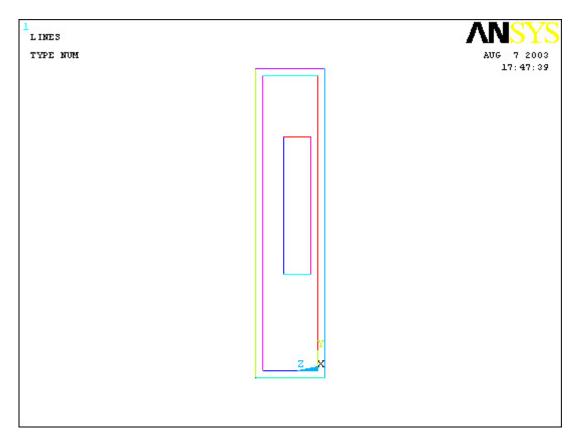




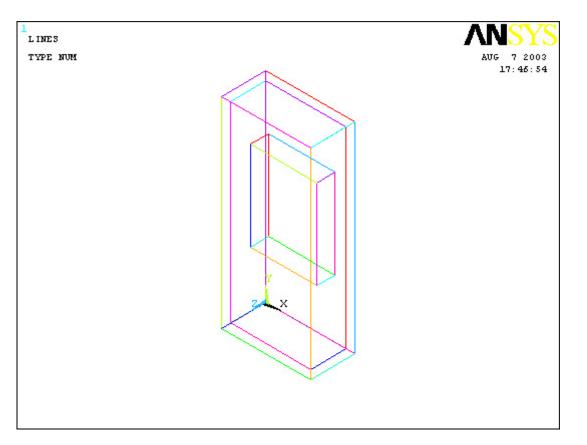
• Now create the volume for the battery:

📥 Block by 2 C	orners & Z 🛛 🔯
• Pick	C Unpick
WPX =	
¥ =	
Global X =	
¥ =	
Z =	
WP X	0
WP Y	0
Width	0.0381
Height	0.0508
Depth	0.01016
OK	Apply
Reset	Cancel
Help	

• 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:

🔺 Define Material Model Behavior		
Material Edit Help		interest date
Material Models Defined	Material Models Available	1
Material Model Number 1	Structural Thermal Conductivity Storapic Social Conductivity Social Conductivity	
•		_

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

Conductivity for Material Number 1 🛛 🛛 🔯			
Conductivity (Iso	tropic) for Materia	l Number 1	
	T1	_	
Temperatures			
KXX	84.8	-	
	,		
	1	1	
Add Temperatur	e 🛛 Delete Temper	ature	Graph
	ОК	Cancel	Help

- 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:

				_
Library of Element Types	Combination Thermal Mass Link Solid Shell ANSYS Fluid FLOTRAN CFD Magnetic Vector	<	Triangl 6node 35 Axi-har 4node 75 8node 78 Brick 8node 70 20node 90 Tet 10node 87 Tet 10node 87	
Element type reference number	plyCancel		Help	

- 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:

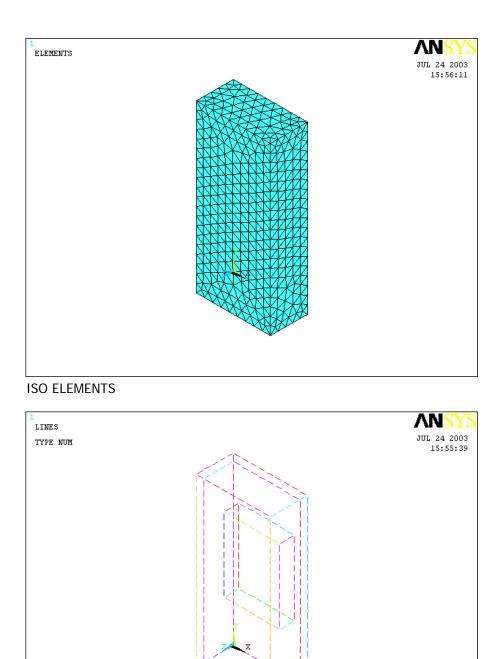
- This section is responsible for telling ANSYS how to divide the block such that it has enough nodes, or points, to analyze to make an accurate enough analysis.
- Go to Preprocessor>Meshing>Size Controls>Manual Size>Global>Size. In the menu that comes up type 0.006 in the field for "Element edge length".

🗖 Global Element Sizes		
[ESIZE] Global element sizes and divisions (applies only		
to "unsized" lines)		
SIZE Element edge length	0.006	
NDIV No. of element divisions -	0	
- (used only if element edge length, SIZE, is blank or zero)		
ок	Cancel	Help

- 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:

Meshing Attributes	$\overline{\mathbf{X}}$
Default Attributes for Meshing	
[TYPE] Element type number	1 SOLID87
[MAT] Material number	
[REAL] Real constant set number	None defined
[ESYS] Element coordinate sys	0 💌
[SECNUM] Section number	None defined
OK Cancel	Help

- 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 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:

Apply CONV on areas	
[SFA] Apply Film Coef on areas	Constant value
If Constant value then:	
VALI Film coefficient	50
[SFA] Apply Bulk Temp on areas	Constant value
If Constant value then:	
VAL2I Bulk temperature	294
LKEY Load key, usually face no.	1
(required only for shell elements)	,
OK Apply Cancel	Help

• 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 m³) = 139235.843 W/m³
- Enter **139235.843** W/m³ as the heat generation value in the pop-up window that appears:

Apply HGEN on volume	
[BFV] Apply HGEN on volume as a	Constant value
If Constant value then:	
VALUE Load HGEN value	139235.843
OK Apply Cancel	Help

• 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:

(PRNSOL) List Nodal Solution		
Dem, Comp. Item to be laited	DOF solution Flux & gradent Norlinear items	Tengerature TDHP
		Temperature TEMP
	cercel Cercel	Help

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

RNSOL Command	X
Eile	
PRINT TEMP NODAL SOLUTION PER NODE	^
***** POST1 NODAL DEGREE OF FREEDOM LISTING *****	
LOAD STEP= 1 SUBSTEP= 1 TIME= 1.0000 LOAD CASE= 0	
NODE TEMP	
1 298.93 2 298.88	
2 298.88 3 298.95 4 298.96 5 298.98 6 298.99 7 299.00 8 299.00 9 299.01	
4 298.96	
5 298.98	
6 298.99	
7 299.00	
8 299.00 9 299.01	
10 299.01	
11 299.00	
12 299.00	
13 298.99	
14 298.98	
15 298.97	~
16 298.96	•

• 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...

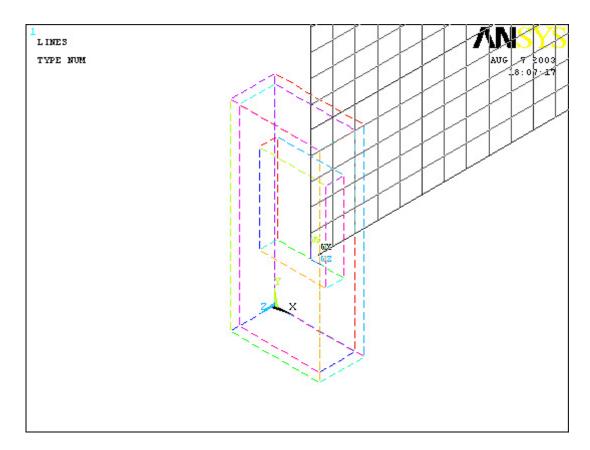
- First go to Utility Menu>PlotCtrls>Style>Hidden Line Options
- The following window appears. Choose Q Slice Z Buffer and Working plane as shown below

Hidden-Line Options	
[/TYPE] [/SHADE] Hidden-Line Options	
WN Window number	Window 1
[/TYPE] Type of Plot	Q-Slice Z-buffer
[/CPLANE] Cutting plane is	Working plane
(for section and capped displays only)	
[/SHADE] Type of shading	Gouraud
[/GRAPHICS] Used to control the way a model is displayed	
Graphic display method is	PowerGraphics 💌
[/REPLOT] Replot upon OK/Apply?	Replot
OK Apply	Cancel Help

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

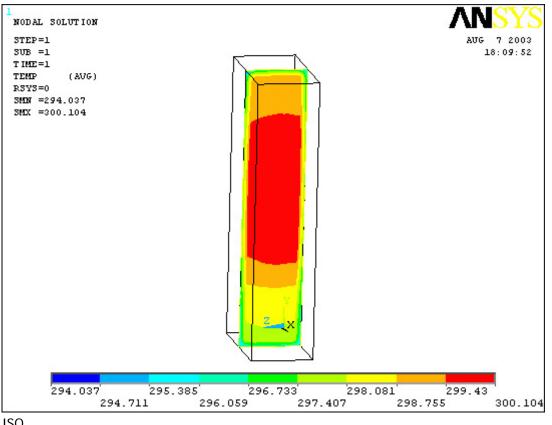
■Then use Workplane>Offset WP by increments and use offset by angles (change increments to 90 degrees) Offset in the +Y direction 90 degrees, then by 3 snap increments in the +Z direction. Should look something like this:



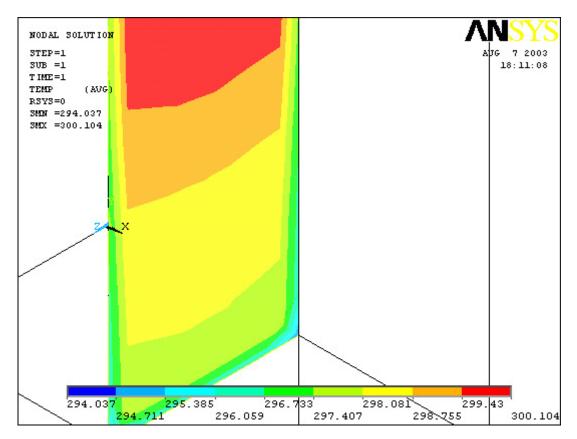
Choose General Postprocessing>Plot Results>Contour Plot>Nodal Solution. The following window will come up:

[PLNSCL] Contour Nodal Solution Data	
Item, Comp Rem to be contoured	DOF solution Flux & gradent Contact
	Temperature TE
KUND Items to be plotted	
	(F Def shape only
	C Def + undeformed
	C Def + undef edge
/EFACET] Interpolation Nodes	
	Generally
	Corner + midside

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



ISO



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.

