# ME345: Modeling and Simulation 3D Heat Conduction within a Solid-Cell Phone

Stevens Institute of Technology Professor Frank Fisher Updated by Lin Dong, Feb 21<sup>st</sup>, 2012 Software used in the tutorial: ANSYS 13.0

#### **INTRODUCTION**

In this tutorial 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.
- All units are S.I.
- Objective: To determine the nodal temperature distribution and create contour plot.
- Boundary Conditions:
  - 1) All faces except that of the battery have convective boundary layers.
  - 2) The battery generates heat at a rate of 50 W/m<sup>2</sup>/s. (h = 50 W/(m<sup>2</sup>\*K))
  - 3) Heat is uniformly generated in the bock at a rate of 20 W/m<sup>2</sup>.
- Material Properties: (Steel)
  - k(innards) = 10 W/m-K
  - k(lithium) = 84.8 W/m-K
  - k(plastic) = 0.18 W/m-K
- 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 mm Thickness: 10.16 mm Width 38.1mm (2 inch x 0.4 inch x 1.5 inch)



Note that the actual dimensions of the battery are: Length: **53 mm** Width **37mm** Thickness: **10 mm** (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. 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. Note: .1 inch** = **2.54 mm** and **R0.50in** = **0.0127m** 

# STARTING ANSYS

- Click on **ANSYS 13.0** in the programs menu.
- Click FILE>Change Working Directory to the place you want to save (like username\\storage01)
- Click FILE>Change Jobname as Cellphone

# **MODELING THE STRUCTURE**

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

WP Setting	s	
<ul> <li>Cartesian</li> <li>Polar</li> </ul>		
<ul> <li>Grid and</li> <li>Grid Onl</li> <li>Triad Or</li> </ul>	l Triad y nly	
Enable 9	Gnap	
Snap Incr Snap Ang	0.00254 5	
Spacing	0.0127	
Minimum	0	
Maximum	0.2	
Tolerance	0.0001	
ОК	Apply	
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) and click
Preprocessor>Modeling>Create>Volumes>Blocks>By 2 Corners & Z
The following window comes up

A Block by 2 C	orners & Z 🛛 🔀
• Pick	O Unpick
WPX =	
Y =	
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.

The next step is to create the inner volume.

•Go to Utility Menu>Workplane>Offset WP by increments and offset the workplane by -0.1 inches in each direction. This will enable you to create the inner volume easily. (as shown in the figure below)



• Click X- button, Y- button, Z- button separately, you will see the Global X,Y Z are all -0.00254 (-0.1in=-0.00254m)

Next use Preprocessor>Modeling>Create>Volumes>Blocks>By 2 Corners & Z again and this time enter the following

A Block by 2 C	orners & Z 🛛 🔀		70
• Pick	🔿 Unpick	TYPE NUM Noncommercial Use	only
WPX =		FEB 22 20	12
¥ =			
Global X =			$\square$
¥ =			$\square$
Z =			$\square$
	<b>a</b>		$\square$
WP V	0		
Width			
Hojaht	0.4442		
Denth	0.1143		
	0.0254		$\geq$
ок	Apply		$\neg$
Reset	Cancel		$\searrow$
Help			

•The model should look like the above figure now if you plot lines (Utility Menu>Plot>Lines) and dynamically rotate the solid (Utility Menu>PlotCntrls>Pan Zoom Rotate)

•The next series of steps involves creating the volume for the battery.

1) First, use Offset WP by increments and increment the Workplane by two positive increments in the Z

direction.



• make sure the Snaps X, Y, Z Offsets is empty and click +Z button twice and click OK 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).

WP Settings	
<ul> <li>Cartesian</li> <li>Cartesian</li> </ul>	•make sure the Spacing is empty and enter the Snap
	11107 0.0301
<ul> <li>Grid and Triad</li> </ul>	
C Grid Only	
C Triad Only	
🔽 Enable Snap	
Snap Incr 0.0381	
Snap Ang 5	
Spacing	
Minimum 0	
Maximum 0.2	
Tolerance 0.0001	Offset WP
	X- +X
OK Apply	Y- +Y
Beset Capcel	Z- +Z
	Snaps
	X, Y, Z Offsets

3) Use Offset WP and offset in the **Y** direction by **one positive increment**.

• make sure the Snaps X, Y, Z Offsets is empty and click +Y button and click OK



4) Change the snap and grid increments of the workplane settings once more to **0.00635 m** (0.25inches)

Incr 0.00635

WP Settings		
<ul> <li>Cartesian</li> <li>Polar</li> </ul>		
Grid and Triad     Grid Only     Triad Only		
Snap Incr 0.00635 Snap Ang 5		
Spacing 🤇		
Minimum	0	
Maximum	0.2	
Tolerance	0.0001	
ОК	Apply	
Reset	Cancel	

- 5) Displace the WP in the X direction by **one positive increment**.
  - make sure the Snaps X, Y, Z Offsets is empty and click +X button and click OK



• make sure the Spacing is empty and enter the Snap

•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.



(Front View)

(Right View)

Now create the volume for the battery. Go to **Preprocessor>Modeling>Create>Volumes>Blocks>By 2 Corners & Z** again and gives the values as below

\Lambda Block by 2 Corners & Z 🛛 🔀		
• Pick	C Unpick	
WPX ÷	=	
Y :	=	
Global X :	=	
¥ :	=	
Z ·	=	
WP X	0	
WP Y	0	
Width	0.0381	
Height	0.0508	
Depth	0.01016	
ок	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).



(Front View)

(Right View)

•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 and it can be accomplished by choosing Preprocessor>Modeling>Operate>Booleans>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

∧ Conductivity for Material Number 1	
Conductivity (Isotropic) for Material Number 1	
T1 Temperatures KXX 84.8	
Add Temperature Delete Temperature	Graph
OK Cancel	Help

∧ Define Material ID	×
Define Material ID 🛛	
OK 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	
Library of Element Types	Gasket  Cohesive Combination Thermal Mass Link Solid Shell ANSYS Fluid Triangl 6node 35 Axi-har 4node 75 8node 70 20node 90 Tet 10node 87 Tet 10node 87
Element type reference number OK Apply	1 Cancel 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		2
[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. as it is shown below

Meshing Attributes	
Default Attributes for Meshing	
[TYPE] Element type number	1 SOLID87
[MAT] Material number	1
[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 (material 1 is lithium, as defined in the Material Properties section) and pick what element type to use (selected by the **Element Type Number**).

• Click **OK** and proceed to **Preprocessor>Meshing>Mesh>Volumes>Free** to select the battery area (anywhere within the lithium battery you created) to be meshed.

•Return to **Default Attributes** and this time, select **Material Number 2**, to model and **mesh** the **innards of the phone**.

•Finally, choose Material 3 and mesh the plastic casing.

•The model should now look like this:





#### **BOUNDARY CONDITIONS**

•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

### **APPLY CONVECTION**

•First we'll apply the Convection Boundary layer around the model Click Convection>On Areas within the Thermal Load category.

•Select the outside areas of the phone (6 sides) and click OK. (You may use PlotCtrls>Pan Zoon Rotate to help to choose the areas.) And the following window will appear

Apply CONV on areas	$\mathbf{X}$
[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 Cancel	Help

•Fill in the h value of 50 in the Film Coefficient blank and the Air temperature of 294 (21°C) in the Bulk Temperature blank. Click OK to close this window.

# **APPLY HEAT GENERATION**

•The next step is to add the constraint of heat generation.

Go to Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Areas.

•Click the inner area to as it is shown blow and click OK. (You may use PlotCtrls>Pan Zoon Rotate to help to choose the areas.)



•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 U^2/R which break down to (Volume =0.0000196644768 m^3) = 139235.843 W/m3 •Enter 139235.843 W/m3 as the heat generation value in the pop-up window that appears

▲ Apply HGEN on areas	
[BFA] Apply HGEN on areas as a	Constant value
If Constant value then:	
VALUE Load HGEN value	139235.843
OK Cancel	Help

# **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

▲ List Nodal Solution				X
┌─ Item to be listed ────				
<ul> <li>☑ Favorites</li> <li>☑ Nodal Solution</li> <li>☑ DOF Solution</li> <li>☑ Nodal Temper</li> <li>☑ Thermal Gradient</li> <li>☑ Thermal Flux</li> </ul>	ature			*
<u>.</u>				
	0K	Apply	Cancel	Help

•Select DOF Solution and Temperature. Click OK. The nodal temperatures will be listed as follows

RNSOL Command	X
Ele	
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	
2827 296.22	
2828 296.22	
2829 296.29	
2830 296.32	
2831 296.34	
2832 296.34	
2833 296.34	
2834 296.32	
2835 296.29	
2836 296.73	
2837 296.31	~

•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
[/HBC] Hidden BCs are	Displayed
[/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)

•Then use Workplane>Offset WP by increments Offset in the +Y direction 90 degrees, then by 3 snap increments in the +Z direction as it is shown below





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

∧ Contour Nodal Solution Data					
_ Item to be contoured —					
<ul> <li>☑ Favorites</li> <li>☑ Favorites</li> <li>☑ DOF Solution</li> <li>☑ DOF Solution</li> <li>☑ Nodal Temp</li> <li>☑ Thermal Gradien</li> <li>☑ Thermal Flux</li> </ul>	erature t				
-					<u>_</u>
Undisplaced shape key -					
Undisplaced shape key	Deformed shape only				•
Scale Factor	Auto Calculated		•0		
Additional Options					۲
		ОК	Apply	Cancel	Help

The plot results will be shown up.

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(spacing is empty)
Offset 1 snap increment in the +Z direction



Replot the contour; you can see that the temperature of the phone near the ear will be on the order of 296K 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.

