

Tutorial to simulate a thermoelectric module with heatsink in ANSYS

Few details can be found in the pictures attached.

All the material properties can be found in Dr. Lee's book and on the web.

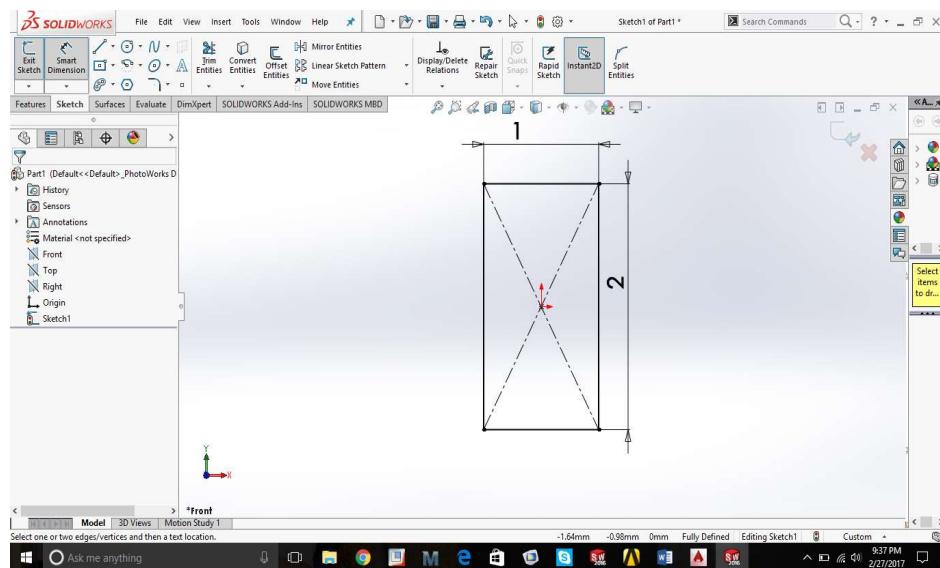
Don't blindly follow the tutorial. You need to explore more options and shortcuts.

The material properties change with the material that is being used. (Ex: Aluminum Nitride, Skutterudite, etc.)

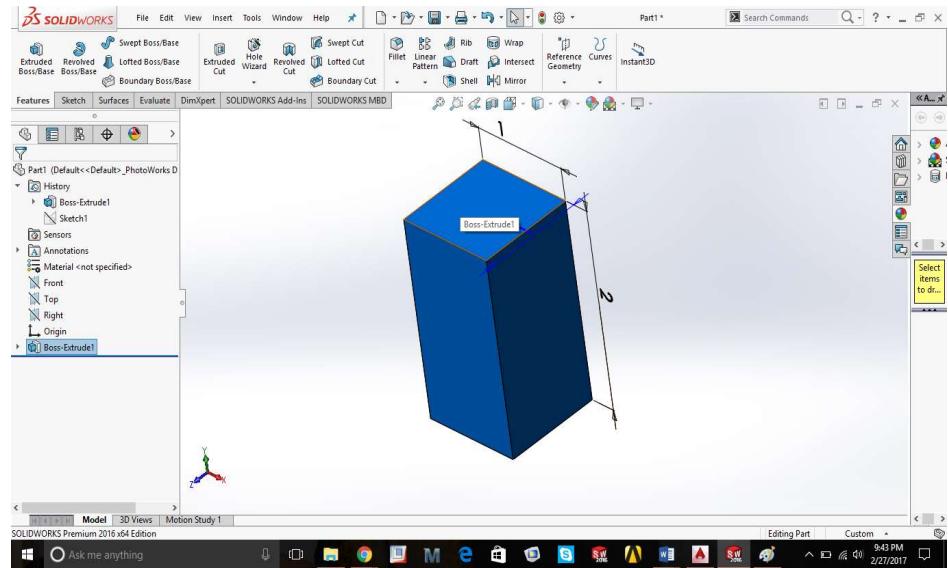
1 Modeling the thermoelectric generator module

1.1 Modeling p leg and n leg

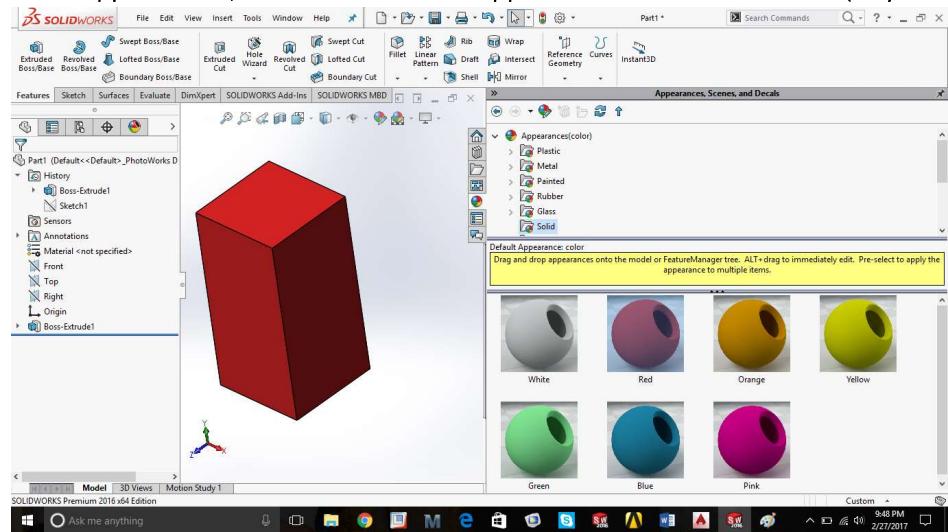
- Open part modeling in SolidWorks
- Select 'front plane' from the 'tree' panel
- Go to sketch tab> sketch> center rectangle
- Draw a rectangle with origin as the center
- Select 'smart dimensions' and dimension each side of the rectangle (2mm*1mm)



- Exit the sketch
- Select Features > Extrude (1mm) > Select rectangle> Enter thickness> Click O.K. (Green tick mark)



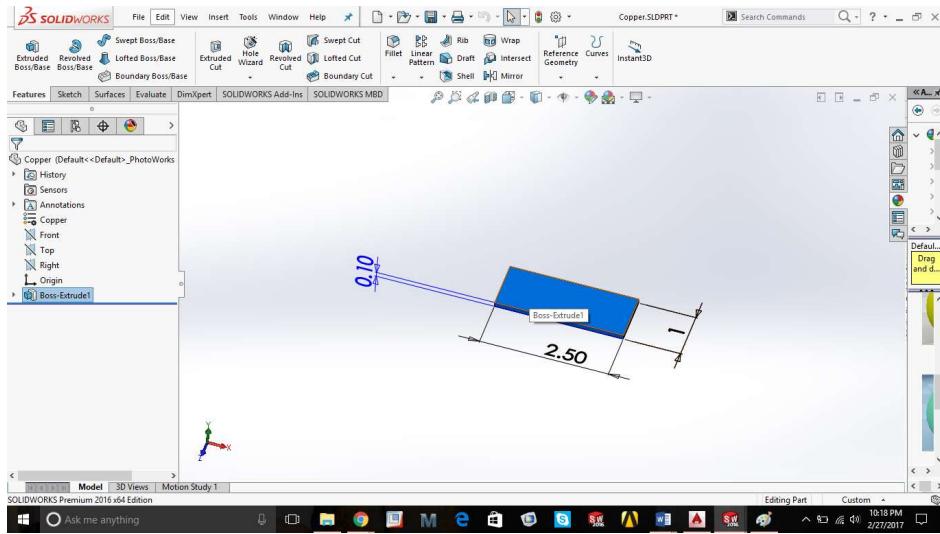
- Go to appearances, scenes and decals > appearances> solids> red color (Any color)



- Click save as and name the file “pleg”
- Follow the same steps again, give a different color and save the file as “nleg”

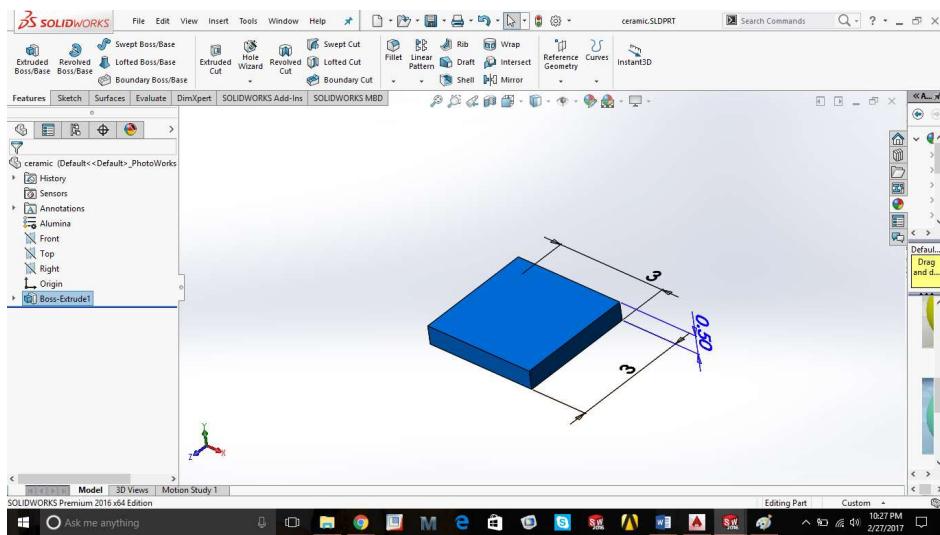
1.2 Modeling the copper (Electric conductor)

- Follow the same steps and create a copper model with specified dimensions (2.5mm*1mm*0.1mm)
- Save the file as “Copper”



1.3 Modeling the ceramic (Electrical insulator)

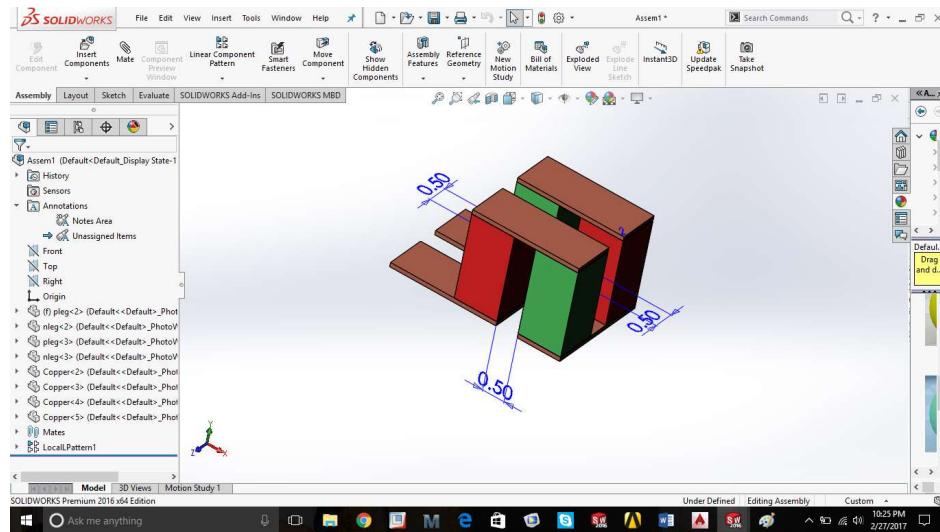
- Follow the same steps and create a ceramic model with specified dimensions (3mm*3mm*0.5mm)
- Save the file as “Ceramic”



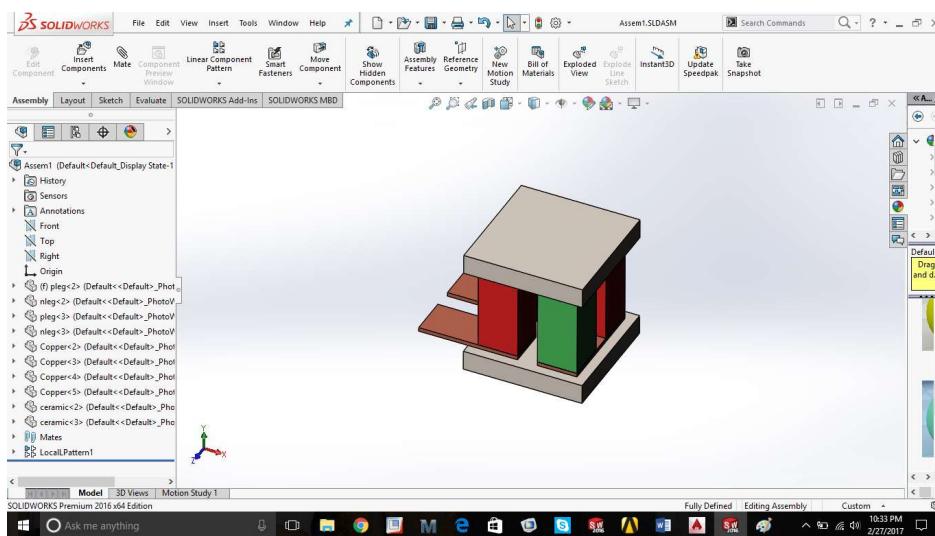
1.4 Assembling the parts

- Open assembly mode In SolidWorks
- Go to Insert components > Browse > Select pleg > Place it anywhere in the graphics window
- Go to Insert components > Browse > Select nleg > Place it beside the pleg
- Go to Mate > select faces facing each other > enter distance (0.5mm) > Click O.K.
- Select top faces of both legs, click O.K.

- Select front faces of both legs, click O.K.
- Insert p leg and repeat the steps to form the figure as shown
- Now go to Insert components > Browse > Copper
- Follow the same steps and assemble copper onto the legs by mating the faces



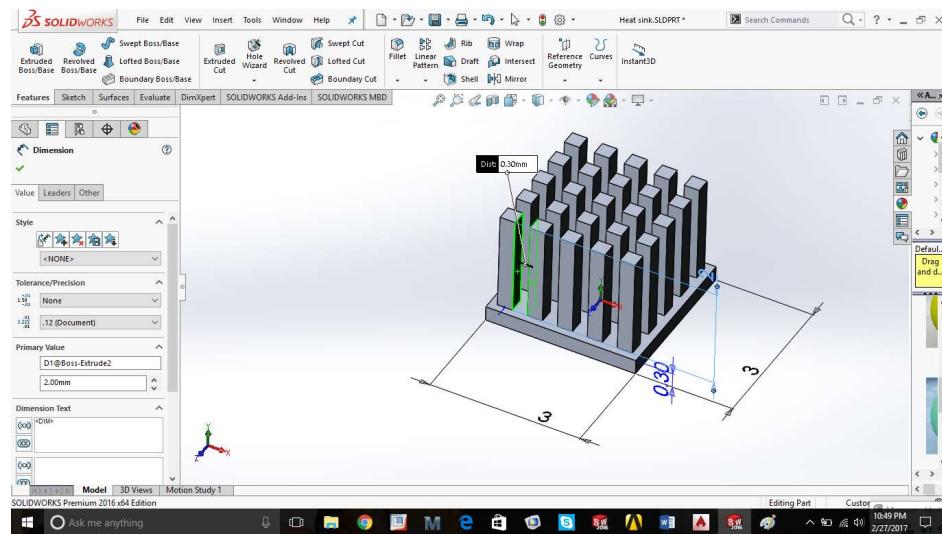
- Insert the ceramic and assemble it symmetrically over the copper parts on either side of the legs as shown



1.5 Model a heatsink as shown in the figure (it has to be modeled as a single part)

- Base area: 3mm*3mm
- Base thickness: 0.3mm
- Fin height: 2mm

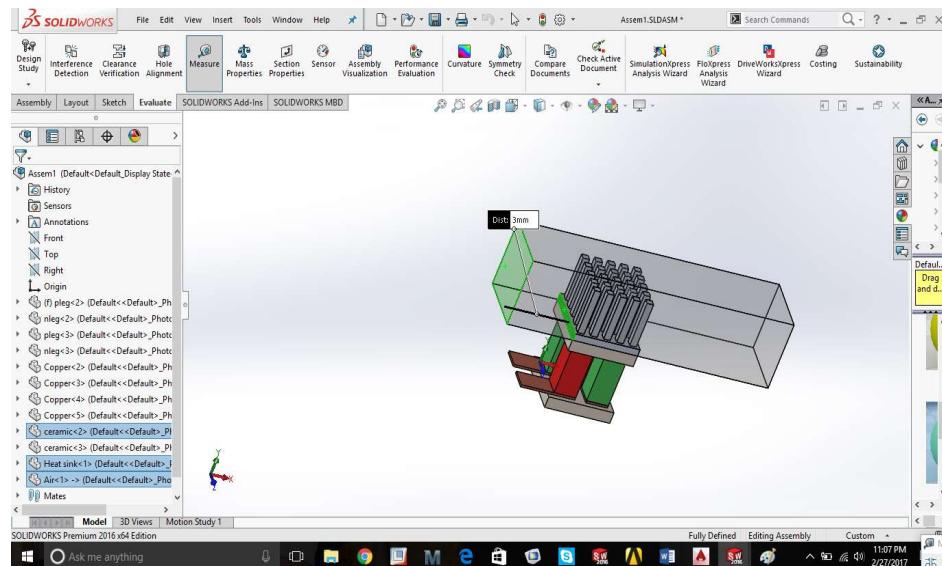
- Fin thickness: 0.3mm
- Fin spacing: 0.3mm



- Now assemble the heatsink over the thermoelectric generator module

1.6 Modeling an air duct for fluid flow and assembly

- Draw a rectangle and extrude it to form a cuboid (12mm*3mm*3mm)
- Assemble it onto the heatsink using the “mate” options
- Select the air duct body go to assembly > edit component > Insert > Molds > Cavity > Select the heatsink body > Click O.K.

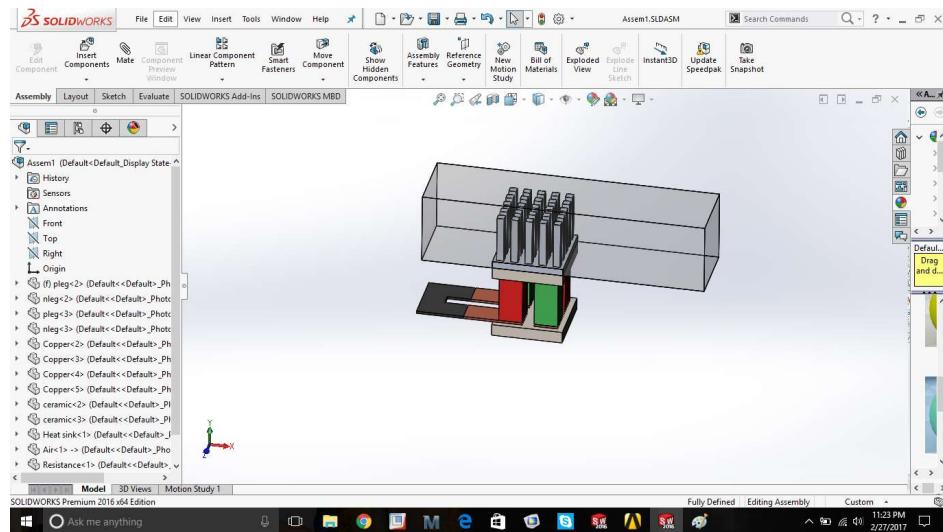


Creating the cavity is done in order to subtract the heatsink from the air duct. The final setup looks like the figure above

1.7 Modeling the external load resistance

In order to simulate a TEG, an external load resistance needs to be provided to the module that will now be designed.

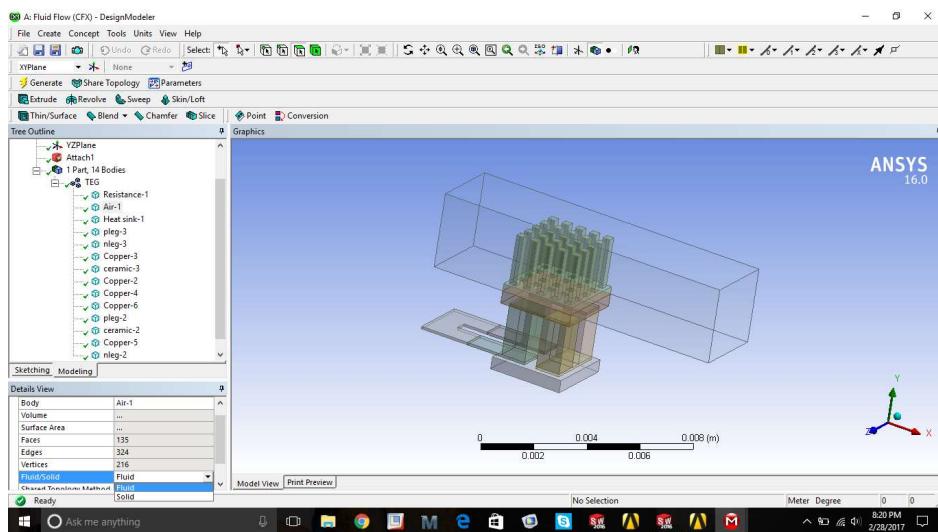
- Draw a U shaped extrusion with dimension (2mm*2.5mm*0.1mm) and assemble it into the module to connect the copper electrical conductors as shown



2 ANSYS Simulation (Fluid Flow CFX)

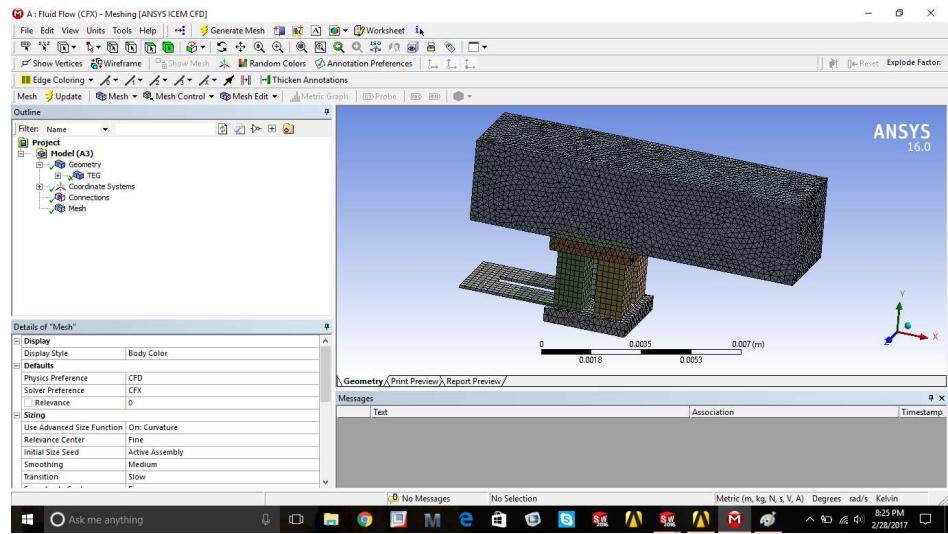
2.1 Setting up the geometry

- Open ANSYS Workbench > Fluid Flow (CFX) > Geometry (Right click) > Import geometry > Browse > Browse and select the assembly file you had previously saved
- Open geometry in design modeler > click generate and wait for the geometry to be imported and generated into the design modeler
- You will see 14 parts, 14 bodies in the tree > Select all the parts > right click and select form a new part > rename your part (TEG)
- Select the air domain in the geometry > detail view > details of the body > fluid/solid > select fluid
- Close the design modeler



2.2 Building the mesh for the geometry

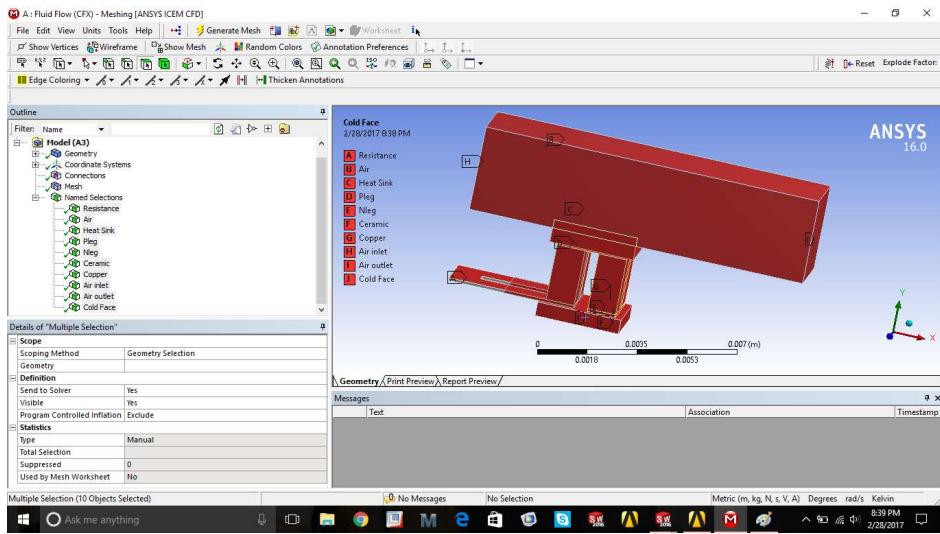
- Open mesh
- Go to mesh > relevance center > fine > update mesh (if your geometry is big or if you have more number of legs in the module, select coarse. Few versions of the software do not allow too many nodes while meshing)



2.3 Creating Named Selections

- Filter your selection to body mode (Ctrl+B) > select resistance part > right click on the body > Create named selection > name it resistance
- Follow the same steps and create named selection for the following bodies:
 - Air body
 - Heatsink body
 - pleg bodies
 - nleg bodies
 - ceramic bodies
 - copper bodies
- Filter your selection to face mode (Ctrl+F) > select air inlet face > right click on the face > Create named selection > name it air inlet
- Repeat the steps to create named selection for the following faces
 - Air outlet
 - Cold side

If there are heat sinks on both the sides, there is not “cold side” face to be created. You instead create an air inlet and air outlet on the cold side as well.



- Close the ‘mesh’ window
- In project schematic, right click on mesh and click update

2.4 Setting up the model for simulation

- Go to materials option in the left side outline tree
- Right click materials > insert > material
- Name the material pleg
- In the details window that opens, basic settings > thermodynamic state > select solid
- In the material properties tab, enter the following properties:
 - Molar mass = 800 g/mol
 - Density = 7.7 g/cm³
 - Specific heat capacity = 544J/kg*K
 - Transport properties: thermal conductivity = 2.6W/m*K
- Follow the same steps to create the material for nleg and aluminum oxide (alumina). Change the properties to the desired number as per the requirement

The values can be found in the following link:

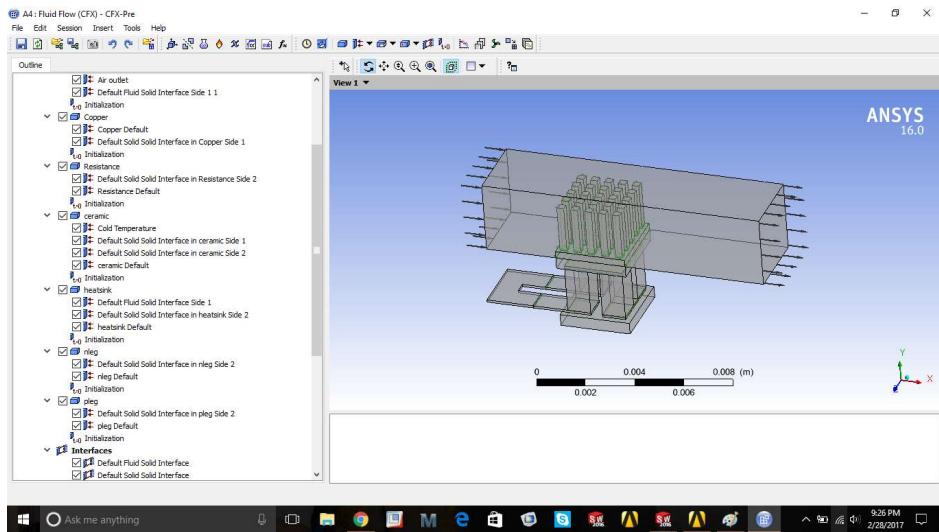
<http://www.customthermoelectric.com/MaterialProperties.htm>

- Open the setup window
- Go to insert > domain > name the domain air > click O.K.
- In location > select air
- Domain type > Fluid domain
- Material > Air ideal gas
- Go to Fluid models tab > Heat transfer > option > thermal energy (Turbulence > K-epsilon, scalable)
- Go to Initialization tab > Check domain initialization box > click O.K.
- Go to insert > domain > name the domain plegs > click O.K.
- In location > select pleg
- Domain type > Solid domain

- Material > pleg
- Go to Solid models tab > Heat transfer > option > thermal energy (Go to Initialization tab >
- Check domain initialization box > click O.K.
- Repeat the same steps to create a domain each for nleg, heatsink and, copper and resistance

Domain	Domain type	Domain location	Material
Air	Fluid	Air	Air ideal gas
Copper	Solid	Copper	Copper
Ceramic	Solid	Ceramic	Aluminum Oxide (Custom material)
Resistance	Solid	Resistance	Copper
Heatsink	Solid	Heatsink	Aluminum
Pleg	Solid	Pleg	Bismuth Telluride (Custom material)
nleg	Solid	nleg	Bismuth Telluride (Custom material)

- In the outline tree, in the interfaces drop down, double click default fluid/solid interface > additional interface models tab > check the heat transfer box, repeat the step for default solid/solid interface
- Now, right click air domain > insert boundary > name it “air inlet”
- Boundary type: inlet > location: air inlet
- Boundary details tab > mass and momentum > option > normal speed: depending on input speed of air (1m/s)
- Heat transfer > static temperature: Given input temperature (770 K) > Click O.K.
- Now, right click air domain > insert boundary > name it “air outlet”
- Boundary type: outlet > location: air outlet
- Boundary details tab > mass and momentum > option > average static pressure > relative pressure 0 Pa > Click O.K.
- Now, right click ceramic domain > insert boundary > name it “cold temperature”
- Boundary type: wall > location: cold face
- Boundary details tab > heat transfer > option > Temperature > Fixed temperature > Given cold side temperature (300 K) > Click O.K.

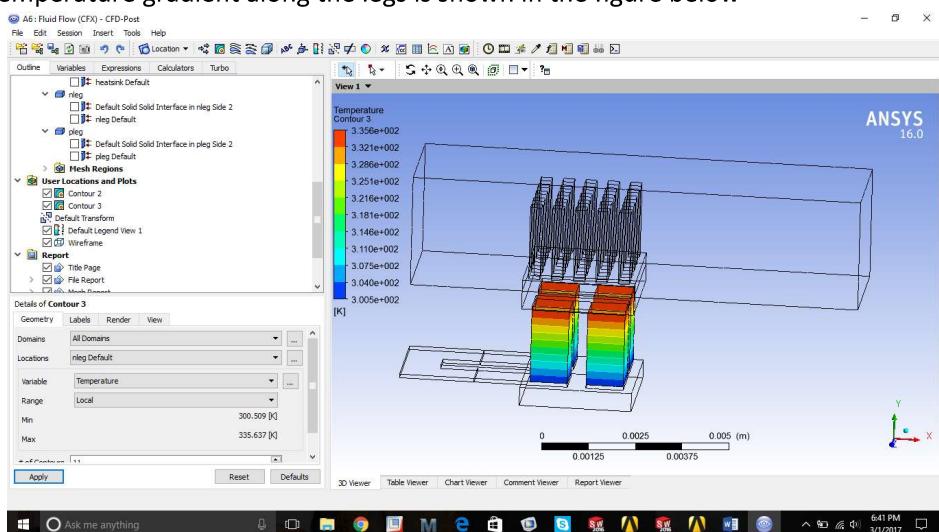


- Close the setup window
- Double click solution in project schematic
- Click start run

Wait until the solution completes normally.

2.5 Checking the results

- Open results in the project schematic
- Go to insert > location > plane > O.K.
- In the details of pane 1 tab, definition > method : XY plane > Z = 0.25mm > apply
- Go to insert > contour > O.K.
- In the details of contour tab, variable: temperature > location: pleg > range: local > apply
- Go to insert > contour > O.K.
- In the details of contour tab, variable: temperature > location: nleg > range: local > apply
- The temperature gradient along the legs is shown in the figure below

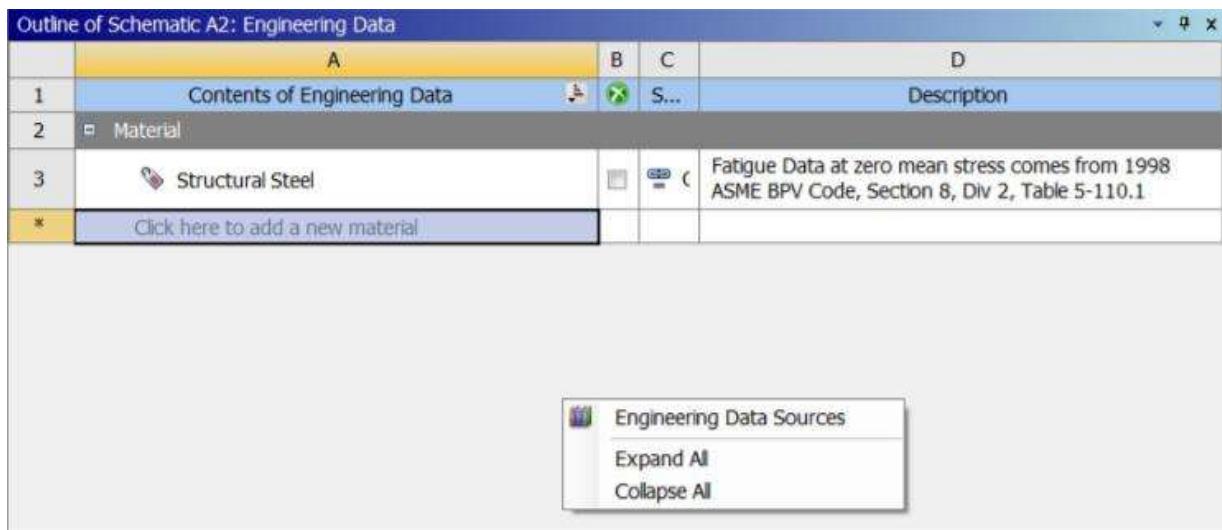


3 ANSYS Simulation (Thermal-electric effect)

In the toolbox menu on the left portion of the window, double click Thermal-Electric. A project will now appear in the project schematic window of Workbench.

3.1 Specifying the Materials and Properties

- Double-click on Engineering Data to open the material data. You will see Structural Steel as the default material in the Outline of Schematic A2: We are going to enter the following materials in the Engineering Data: Copper Alloy, p-type semiconductor, and n-type semiconductor, Ceramic (Aluminum Oxide)



- In the Data Source of Outline Filter, click on General Materials. In the Outline of General Materials pane, right-click on Copper Alloy and select Add to Engineering Data or click on the '+' icon. A 'book' symbol will appear once the material has been added.

Engineering Data Sources				
	A	B	C	D
1	Data Source		Loc...	Description
2	Favorites			Quick access list and default items
3	General Materials			General use material samples for use in various analyses.

Outline of General Materials					
	A	B	C	D	E
1	Contents of General Materials		Add	S...	Description
					ASME BPV CODE, SECTION 8, DIV 2, TABLE 5-110.1
4	Air		+	=	General properties for air.
5	Aluminum Alloy		+	=	General aluminum alloy. Fatigue properties come from MIL-HDBK-5H, page 3-277.
6	Concrete		+	=	
7	Copper Alloy		+	=	

- In the Outline Filter, click on Engineering Data, you will see that Copper Alloy in the Outline Schematic A2 is newly added material to the Structural Steel.
 - Now we want to add two more materials (p-type and n-type semiconductors). Click on the empty box below the Structural Steel and name it as **p-type**. In the Toolbox pane, double-click Isotropic Thermal Conductivity to include this property to the p-type.
In the Properties of Outline Row 5: p-type: The following values are entered as:
Isotropic Thermal Conductivity: 1.46 W m-1 K -1 (According to the given values)
Isotropic Resistivity: 1.64e-5 Ω m (According to the given values)
Isotropic Seebeck Coefficient: 187e-6 V K-1 (According to the given values)
 - Create **n-type** by duplicating the p-type. Right-click on p-type in the Outline of Schematic A2 and select Duplicate. A duplicate of the p-type material will appear below named p-type 2. Rename this material to n-type. The value of the Isotropic Seebeck Coefficient is now changed to the negative as -187e-6.
In the Properties of Outline Row 5: n-type: make sure the final values to be as:
Isotropic Thermal Conductivity: 1.46 W m-1 K -1 (According to the given values)
Isotropic Resistivity: 1.64e-5 Ω m (According to the given values)
Isotropic Seebeck Coefficient: -187e-6 V K-1 (According to the given values)
 - Create **ceramic** material by repeating the process as in creating the p type with the following properties
Isotropic Thermal Conductivity: 27 W m-1 K -1 (According to the given values)
 - Create **Resistance** material by repeating the process as in creating the p type with the following properties
Isotropic Resistivity: 1.822*E-7 Ω*m

$$R_{Load} = R_{int} \text{ (only for maximum load condition)}$$

$$\rho_{Load} * (L_{Load}/A_{Load}) = \rho_{int} * (L_{int}/A_{int}) \text{ (int = legs)}$$

$$\rho_{int} = \rho_{pleg} + \rho_{pneg} + \rho_{nleg} + \rho_{nleg} = 4 * \rho_{pleg} \text{ (If } \rho_{pleg} = \rho_{nleg} \text{) (Resistance in series)}$$

$$L_{int} = 2 \text{ mm (According to the Solid works Model)}$$

$$A_{int} = 1\text{mm}*1\text{mm (According to the Solid works Model)}$$

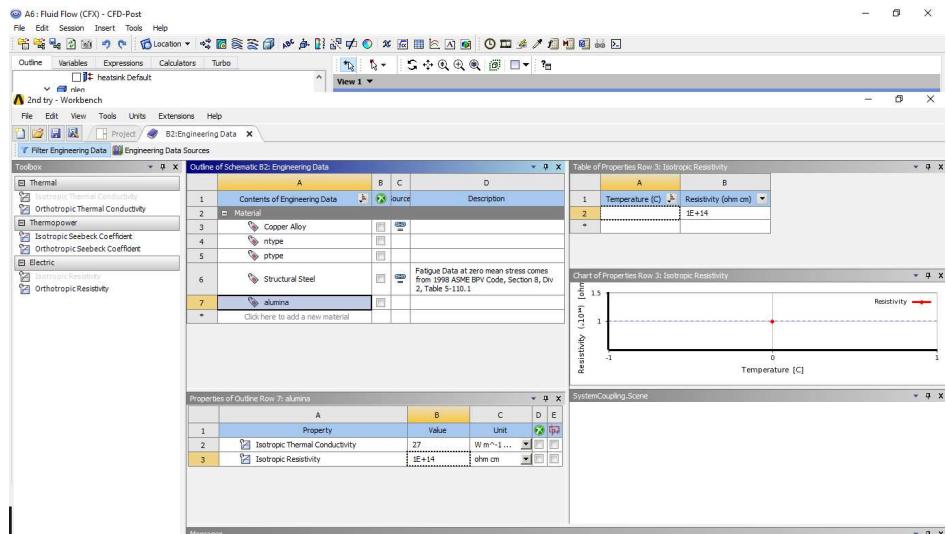
$$L_{Load} = 4.5\text{mm (According to the Solid works Model)}$$

$$A_{Load} = 0.1\text{mm}*1\text{mm (According to the Solid works Model)}$$

$$\rho_{Load} = \rho_{int} * (L_{int}/A_{int}) * (A_{Load}/L_{Load})$$

$$\rho_{Load} = \rho_{pleg} * 4 * (2\text{mm}/1\text{mm}^2) * ((0.1\text{mm}^2)/4.5\text{mm})$$

$$\rho_{Load} = 4.5E-6 \Omega*m \text{ (for } \rho_{pleg} = 1.27e-5 \Omega m \text{)}$$
- Click on the ‘Return to Project’ icon in the menu bar to return to the Project.
 - Save the project.



3.2 Creating the Geometry

- Right click on the geometry > import geometry > Import the original Solidworks Assembled file.

3.3 Setting up the model

- In the Workbench, double-click on Model to launch the solver. This may take several seconds up to a minute.
- In the Outline pane, expand Geometry.
Right click on Air > Suppress Body
Right click on Heat sink > Suppress Body
 - Specify the material for each body by clicking on it and changing the Assignment under the Material section in the Details of " " pane. Click on p-leg and change the Assignment in the Details of "pleg" to p-type. Repeat the step with the following bodies.

Body	Material Assignment
Pleg (2 pleg bodies)	pype
Nleg (2 nleg bodies)	ntype
Copper(All the 5 copper bodies)	copper
Ceramic (Both the ceramic Bodies)	Alumina
Resistance	Resistance

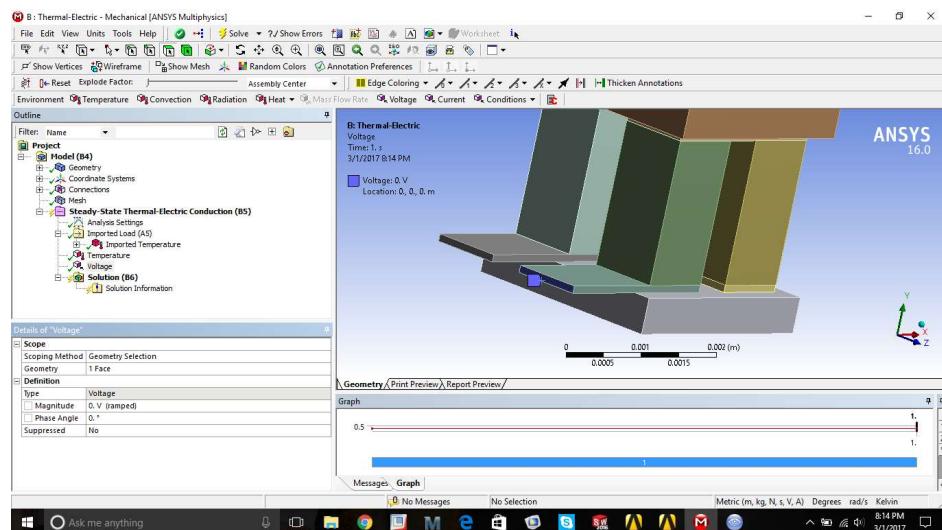
- In the Outline pane right-click on Mesh and select Update. This may take several seconds.
With mesh options
Mesh Relevance Center> medium
- Close the model window

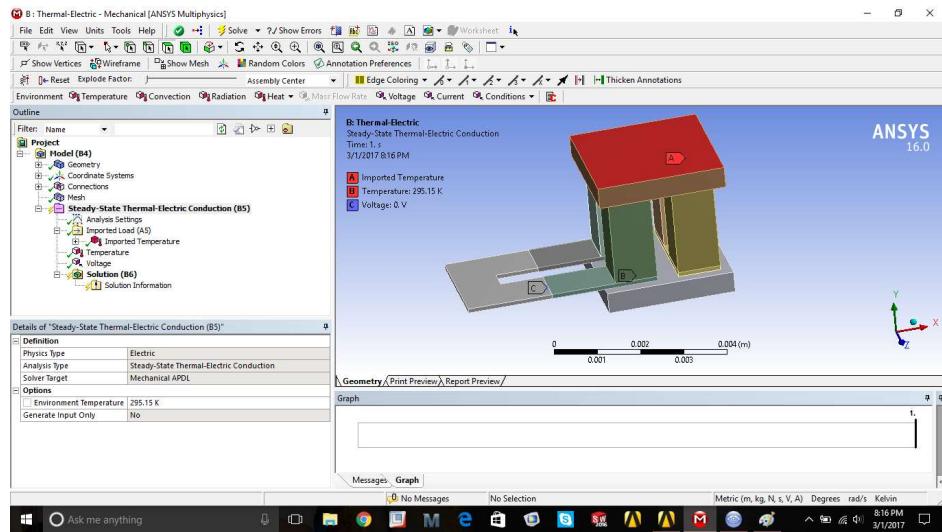
Now, link the **solution** of the CFX model to the thermal-electric **setup**

- In order to do this, click and drag the solution bar and drop it onto the setup bar in thermal-electric

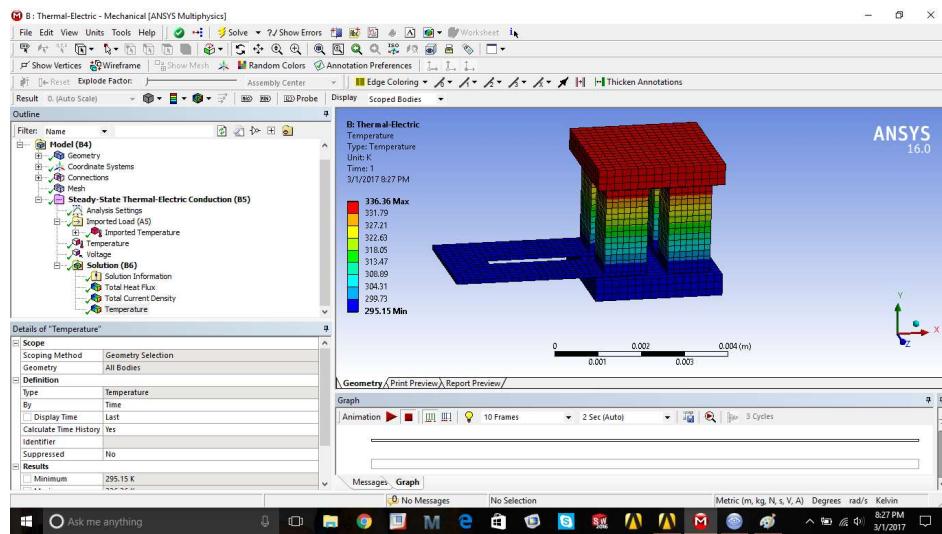
3.4 Input conditions- Setup

- Double click on the setup bar (if prompted to update, say YES)
- In the Outline pane, expand Stead-State Thermal-Electric Conduction (B5), right click on imported load, select Insert and click on Temperature. (you will see a new imported temperature in the outline pane)
- Click on imported temperature > details of imported temperature > geometry > select the hot side surface (ceramic) > apply
- Go to transfer definition > CFD surface > select default-default solid interface in ceramic side 1
- Right click imported temperature, click import load
- Right click on Stead-State Thermal-Electric Conduction (B5), insert temperature
- In the details of temperature window, go to geometry > select cold side surface (other ceramic) > apply
- Go to definition > magnitude = 300 K
- Right click on Stead-State Thermal-Electric Conduction (B5), insert voltage
- Click on the resistance body then right click on the body and select hide body
- Select the face of the cross sectional area of copper on the pleg side as shown in the figure
- Input the value of input voltage to be 0V





- Right click on solution > insert > thermal > temperature
- Right click on solution > insert > Probe > Reaction probe (For hot side)
- Right click on solution > insert > Probe > Reaction probe (For cold side)
- Right click on solution > solve
- Once the solver has completed its tasks, click on any of the solutions (Temperature, Total Current Density, etc.) to display the results.



- Close the solver and save the project

The power output is the sum of magnitudes obtained from the reaction probes.

For a TEC, the input conditions change. In place of the external load resistance, there will be a 0V (ground) on one side of the copper while there is a current input on the other side. A cooling power is obtained from the input current. This input current depends on the given conditions.

The power output is then maximized with respect to the load resistance in TEG and with respect to the input current in a TEC.