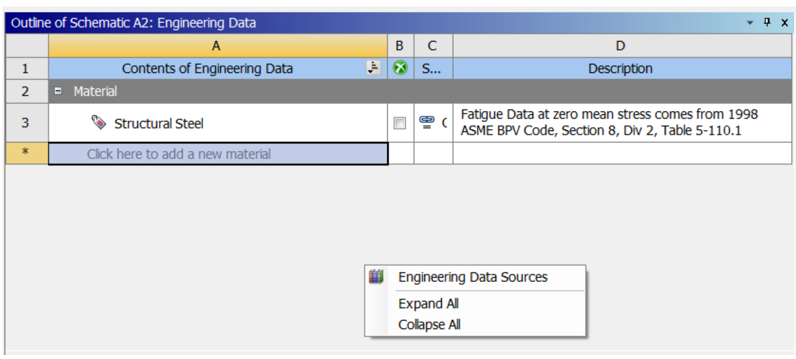
ANSYS Thermoelectric Generator (TEG) Tutorial

Preparing the ANSYS Workbench

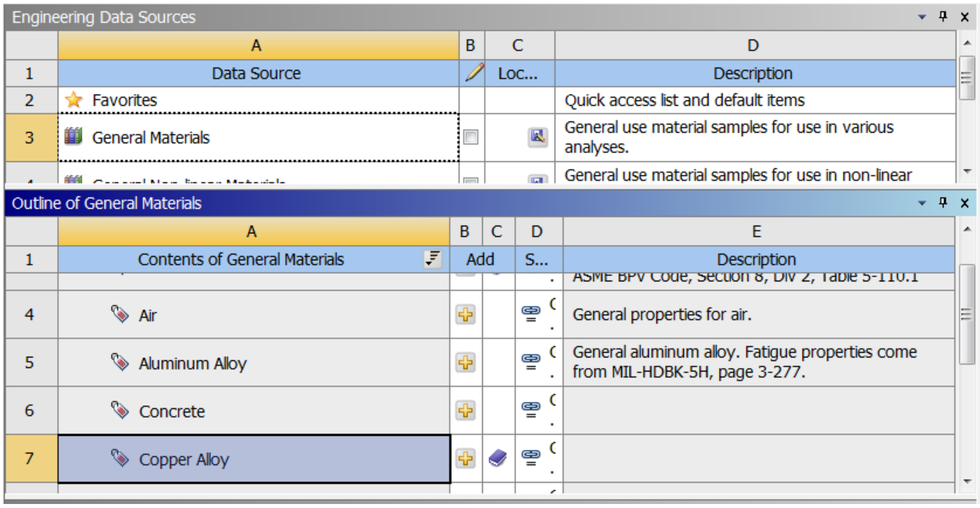
1) Go **Start → Menu → All Programs →Simulation → ANSYS 12.1 → Workbench**
2) In the toolbox menu on the left portion of the window, double click **Thermal-Electric**. A project will now appear in the project schematic window of Workbench.
3) Right-click **Thermal-Electric** at the top of the Project Schematic pane and select **Rename** as **TEG Ex 3.1 Tutorial**.
4) Save the project as **Thermoelectric-Generator-Workbench**.

Specifying the Materials and Properties

1) 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 three materials (Copper Alloy, p-type semiconductor, and n-type semiconductor) in the **Engineering Data**.



2) 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 th  icon. A symbol  will appear once the material has been added.



3) 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.

4) Now we want to add two more materials (p-type and n-type semiconductors). Double-click on **Structural Steel** and rename it as **p-type**. In the **Toolbox** pane, double-click **Isotropic Thermal Conductivity** to include this property to the p-type.

5) 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}
Isotropic Resistivity: **1.64e-5** Ω m
Isotropic Seebeck Coefficient: **187e-6** V K^{-1}

6) 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}
   Isotropic Resistivity: **1.64e-5** Ω m
   Isotropic Seebeck Coefficient: **–187e-6** V K^{-1}

7) Click on the Return to Project icon in the menu bar to return to the Project.

8) **Save** the project.

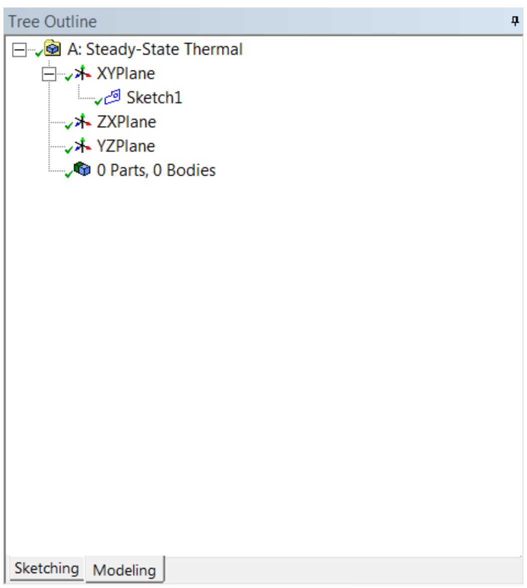
Creating the Geometry

1) In the project **A** under the **Project Schematic**, double-click on **Geometry** to launch the Design Modeler.

2) Select **Millimeter** as the desired length unit and click **OK**.

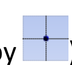
3) In the **Tree Outline** pane, right-click on the **XY Plane** and select **Look at**. Add a new sketch by clicking on the  icon in the menu bar.

4) **Sketch1** will appear below the XY Plane. Click on the **Sketch1** and select the **Sketching** tab at the bottom of the **Tree Outline** pane.



5) Click on the **Settings** tab and **Grid** and check the boxes of both **Show in 2D** and **Snap** options as

6) Click on the **Draw** tab and select **Rectangle by 3 points**.

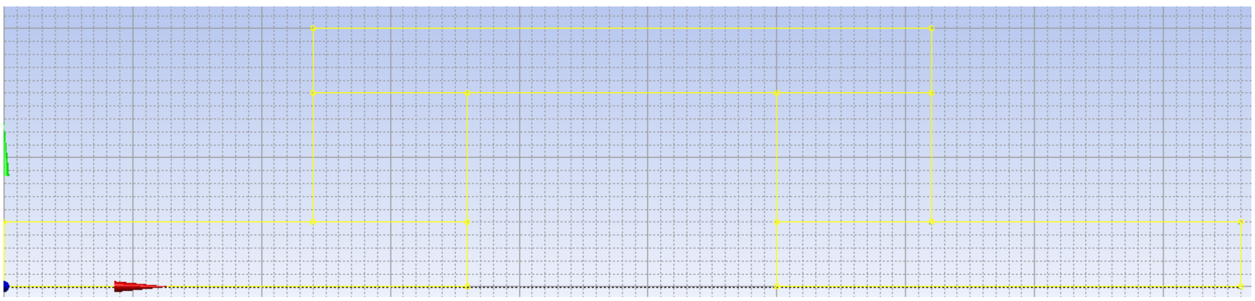
7) Once the **Rectangle by 3 points** is selected, click on the origin (indicated by )of the **Graphics** pane to add the first point of the rectangle. Place the second point of the rectangle at **(X=36,Y=0)**. Place the third and final point of the rectangle at **(36, 5)**. This is the sketch for the base of the p-type element.
*Note: The coordinate position is displayed in millimeter at the bottom right of the **Graphics**.*

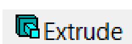
8) With the Rectangle by 3 points still selected place another first point at **(24, 5)**. Place the next point at **(36, 5)**. Place the final point at **(36, 15)**. This is the sketch for the p-element.

9) Place another first point at **(24,15)**, the second point at **(24, 20)** and the final point at **(72, 20)**. This is the sketch for the top plate.

10) Place another first point at **(72, 15)**, the second point at **(60, 15)** and the final point at **(60, 5)**. This is the sketch for the n-element.

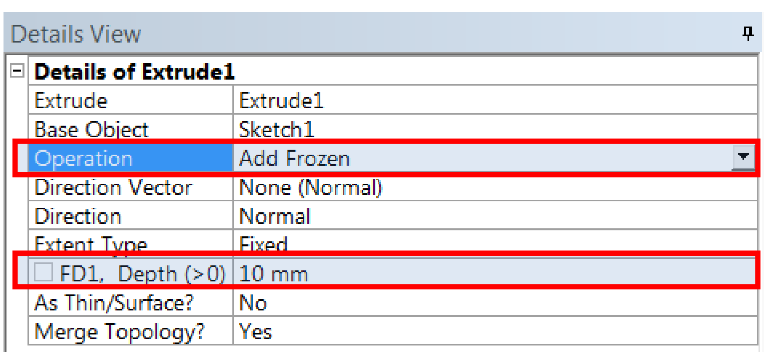
11) Place another first point at **(60, 5)**, the second point at **(96, 5)** and the final point at **(96, 0)**. This is the sketch for the base of the n-element. Upon completion all rectangles will form the shape below.



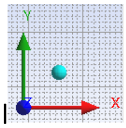
12) In the Tree Outline, click on the **Modeling** tab and select **Sketch1** and extrude it by clicking the 

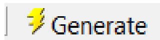
icon in the menu bar or by going to **Create → Extrude** on the menu bar.

13) **Extrude1** will appear in the **Tree Outline**. Click on it. In the **Details View** pane, change the **Operation** from Add Material to **Add Frozen**. Change the **Depth** to 10 mm.



*Note: The '***Add Frozen***' option keeps the various elements from merging. The '***Add Material***' option will perform merging to one element.*

14) To see an isometric view, click on the ball  of the global coordiante axes.

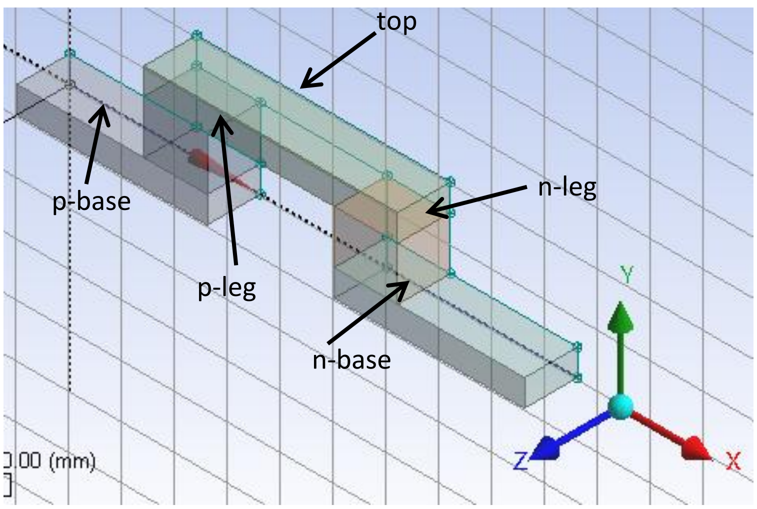
15) Click on the 

icon in the menu bar to generate the extrusion or right-click on **Extrude1** and click **Generate**.

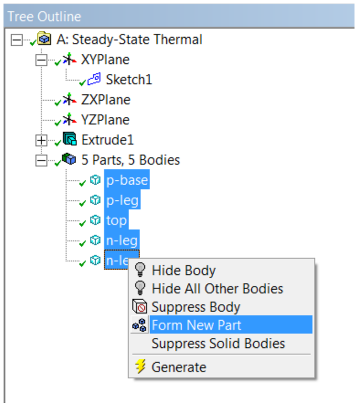
Note: The  *symbol will change to a*  *symbol and, indicating that the command is successful.*

16) In the **Tree Outline**, there will now be **5 Parts** and **5 Bodies** as a result of the extrusion.

17) Under the **5 Parts, 5 Bodies** in the **Tree Outline**, select each body and rename it according to the figure shown below. Rename each part by right-clicking on it in the Tree Outline and selecting **Rename**. The currently selected part will become highlighted in the **Graphics** pane.



18) Ensure that each part is a **Solid** (by default) in the Details View pane under the **Fluid/Solid** option. Select 5 bodies in the Tree Outline by clicking on them while holding down the **Ctrl** key. Right-click and select **Form a new part**. Right-click and rename the part to **TEG**.

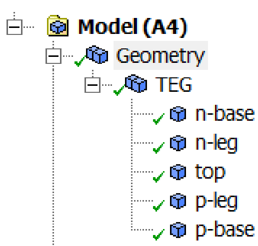


19) **Close** the Design Modeler. **Save** the project from the Workbench.

Setting Up the Model

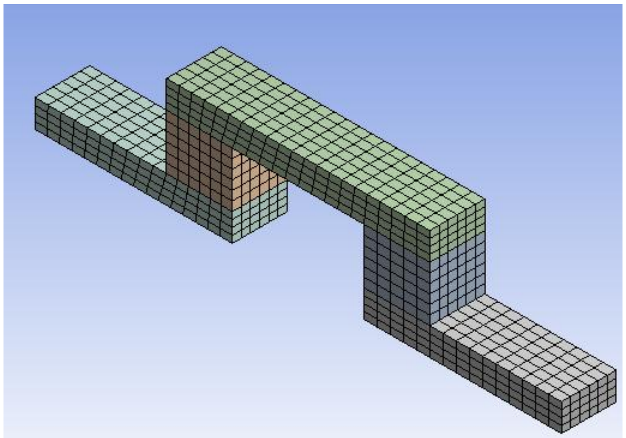
1) In the Workbench, double-click on **Model** to launch the solver. This may take several seconds up to a minute.

2) In the Outline pane, expand **Geometry → TEG**.

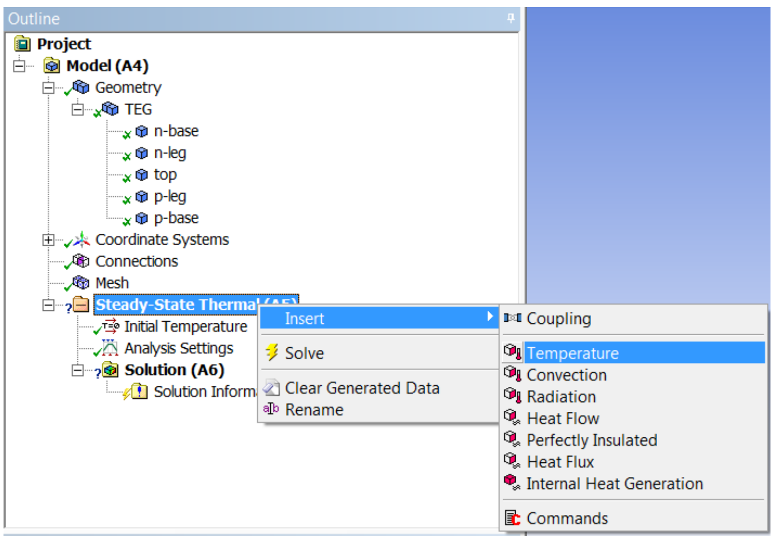


Specify the material for each body by clicking on it and changing the **Assignment** under the **Material** section in the **Details of " "** pane.
Click on **n-base** and change the **Assignment** in the **Details of "n-base"** to **Copper Alloy**.
Click on **n-leg** and change the **Assignment** in the **Details of "n-leg"** to **n-type**.
Click on **top** and change the **Assignment** in the **Details of "top"** to **Copper Alloy**.
Click on **p-leg** and change the **Assignment** in the **Details of "p-leg"** to **p-type**.
Click on **p-base** and change the **Assignment** in the **Details of "p-base"** to **Copper Alloy**.

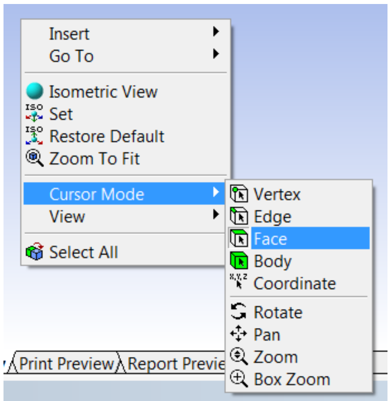
3) In the Outline pane right-click on **Mesh** and select **Update**. This may take several seconds.



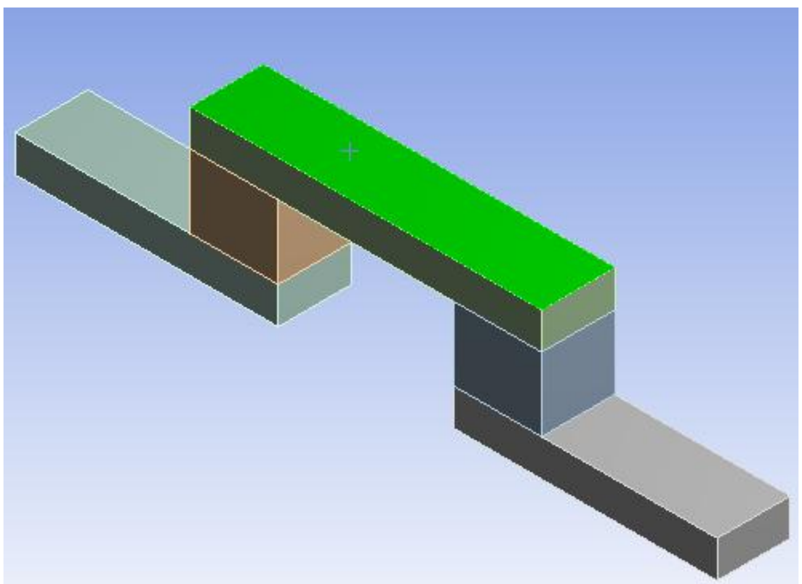
4) In the **Outline** pane, right-click on **Stead-State Thermal-Electric Conduction (A5)**, select **Insert** and click on **Temperature**.



Click on the newly added **Temperature** under the Steady-State Thermal-Electric Conduction (A5) to ensure that it is selected. Right-click anywhere in the graphics pane on the right, select **Cursor Mode** and click **Face**. This cursor mode allows you to select surfaces.



Click on the **top surface of the top body** to select it. Highlighted surfaces are indicated in green. In the **Details of "Temperature"** pane, click **Apply** in the **Geometry** option. The applied surface will change the color which confirms the application.



Change the **Magnitude** in the Details pane to **452** °C (ramped).

Definition	
Type	Temperature
Magnitude	452. °C (ramped)
Suppressed	No

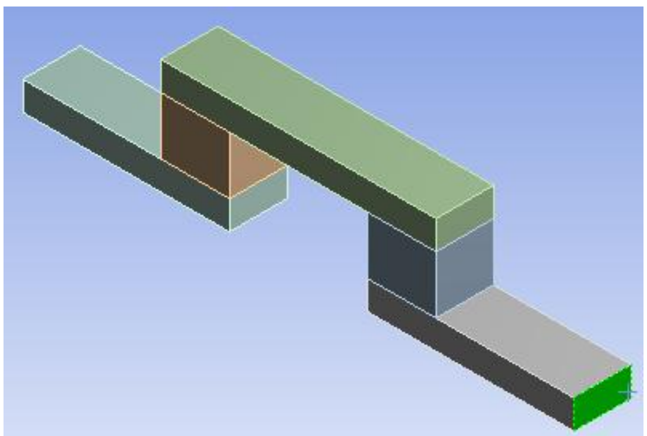
Right-click on **Temperature** in the Outline pane and rename it to **Hot Junction**. This is the hot junction boundary condition.

5) In the **Outline** pane, right-click on **Stead-State Thermal-Electric Conduction (A5)**, select **Insert** and click on **Temperature** again.

*Note: Right-click anywhere in the **Graphics** pane and select **Cursor Mode** and click **Rotate**. Switch the cursor mode back to **Face,** click on the bottom surface of n-base and p-base while holding down **Ctrl**.*

In the **Details of "Temperature"** pane, click **Apply** in the **Geometry** option. The applied surface will change the color. Change the **Magnitude** in the Details pane to **22** °C (ramped). Right-click on **Temperature** in the **Outline** pane and rename it to **Cold Junction**. This is the cold junction boundary condition.

6) In the Outline pane, right-click on **Stead-State Thermal-Electric Conduction (A5)**, select **Insert** and click on **Voltage** .In the Graphics pane, change the view back to **isometric** by clicking the small ball in the global coordinate axes. Select the outer face of the base of the n-leg (parallel to the y-z plane).



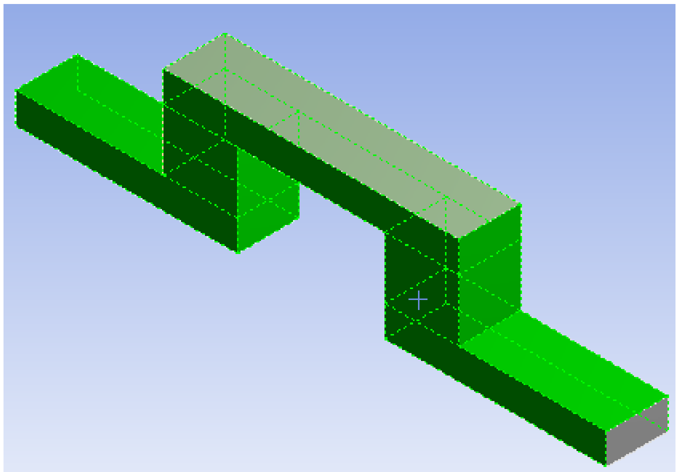
In the **Details of "Voltage"** pane, click **Apply** in the **Geometry** option. The applied surface will change the color. Ensure the **Magnitude** in the Details pane is **0** V (ramped). Right-click on **Voltage** in the Outline pane and rename it to **Low Potential**. This is the lower electric potential boundary condition.

7) In the Outline pane, right-click on **Stead-State Thermal-Electric Conduction (A5)**, select **Insert** and click on **Voltage** again.

Note: Right-click anywhere in the Graphics pane and select **Cursor Mode** and click **Rotate** and rotate the thermocouple until the other-side outer face of the base of the p-leg (parallel to the y-z plane) can be seen. And select the outer face.

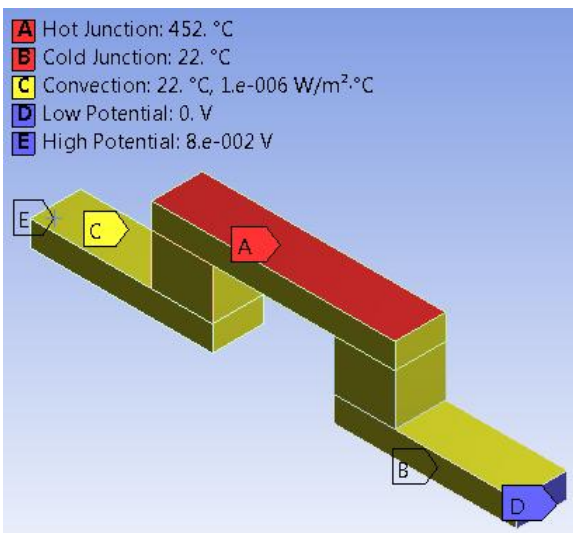
In the Details of "Voltage" pane, click **Apply** in the **Geometry** option. The applied surface will change the color. Change the **Magnitude** in the Details pane is **0.08** V (ramped). Right-click on **Voltage** in the Outline pane and rename it to **High Potential**. This is the higher electric potential boundary condition.

8) In the Outline pane, right-click on **Steady-State Thermal-Electric Conduction (A5)**, select **Insert** and click on **Convection**. In the Graphics pane, select all faces excluding those which have been assigned boundary conditions previously. Select multiple faces by holding down the **Ctrl** key.



In the Details of “Convection” pane, click **Apply** in the **Geometry** option. The applied surface will change the color. The **Geometry** option should read **21 Faces** upon application. Change the **Magnitude** in the Details pane is **1e-6** W/m^2·°C (ramped) for negligible convection. Right-click on **Convection** in the Outline pane and rename it to **Insulation**.

9) In the Outline pane, click on **Steady-State Thermal-Electric Conduction (A5)** to view the boundary conditions selected as shown below.



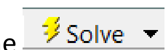
10) In the Outline pane, right-click on **Solution (A6)**, select **Insert →Thermal → Temperature**. Make sure that **All Bodies** is default under Geometry in the Details of “Temperature” pane. And leaves

others as they are.

11) In the Outline pane, right-click on **Solution (A6)**, select **Electric → Total Current Density**. Make sure that **All Bodies** is default under Geometry in the Details of "Total Current Density" pane. And leaves others as they are. These render the desired results for display.

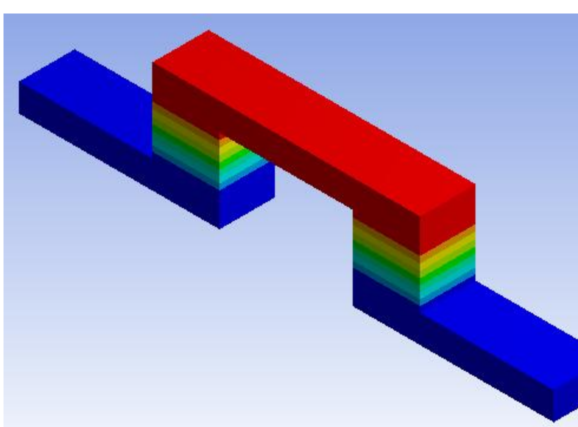
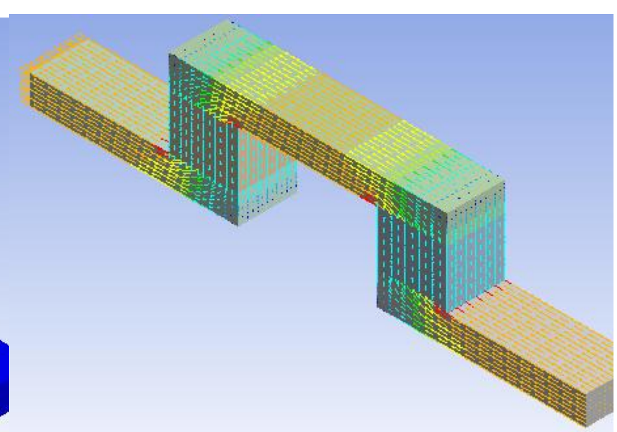
12) In the Outline pane, right-click on **Solution (A6)**, select **Probe → Current Reaction**. In the Details of "Current reaction" pane, select **Low Potential** in the **Boundary Condition** option. Right-click on **Reaction Probe** in the Outline pane and rename it to **Current**.

13) Repeat the insertion process for **Probe → Heat Reaction.** In the Details of "Heat Reaction" pane, select **Hot Junction** in the **Boundary Condition** option. Right-click on **Reaction Probe** in the Outline pane and rename it to **Hot Junction Heat Absorbed**.

14) In the Outline pane, right-click **Solution (A6)** and click **Solve** or click on the 

icon. The solver will take up to several minutes to complete.

15) Once the solver has completed its tasks, click on any of the solutions (Temperature, Total Current Density, etc.) to display the results. For the Total Current Density, you can display vectors (to indicate the direction of current flow) instead of contours by clicking on the  icon below the menu bar. Zoom in closer by **scrolling up on the mouse wheel** to observe the direction of current. The results of the probes (Current and Hot Junction Heat Absorbed), when selected, are displayed in Tabular Data pane at the bottom right of the window. The computed current and heat absorbed will be 28.942 A and 21.682 W, which are in good agreement with those results in Example 3.1 in the textbook.

16) **Close** the Mechanical Solver and **Save** the project.