2024 R2
Take A Leap Of Certainty™

Getting Started with Mechanical: Steady-State Thermal Solution – Heat Sink

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
https://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2024 R2
July 2024

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001:2015 companies.

Conventions Used in this Guide

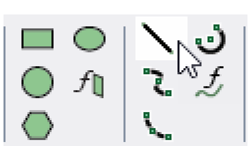
Please take a moment to review how instructions and other useful information are presented in this documentation.

- Procedures are presented as numbered lists. A single bullet indicates that the procedure has only one step.
- Bold type is used for the following:
    - Keyboard entries that should be typed in their entirety exactly as shown. For example, “**copy file1**” means you must type the word **copy**, then type a space, and then type **file1**.
    - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by greater than signs (>). For example, “click **HFSS** > **Excitations** > **Assign** > **Wave Port**.”
    - Labeled keys on the computer keyboard. For example, “Press **Enter**” means to press the key labeled **Enter**.
- Italic type is used for the following:
    - Emphasis.
    - The titles of publications.
    - Keyboard entries when a name or a variable must be typed in place of the words in italics. For example, “**copy** *filename*” means you must type the word **copy**, then type a space, and then type the name of the file.
- The plus sign (+) is used between keyboard keys to indicate that you should press the keys at the same time. For example, “Press Shift+F1” means to press the **Shift** key and, while holding it down, press the **F1** key also. You should always depress the modifier key or keys first (for example, Shift, Ctrl, Alt, or Ctrl+Shift), continue to hold it/them down, and then press the last key in the instruction.

Accessing Commands: *Ribbons, menu bars,* and *shortcut menus* are three methods that can be used to see what commands are available in the application.

- The *Ribbon* occupies the rectangular area at the top of the application window and contains multiple tabs. Each tab has relevant commands that are organized, grouped, and labeled. An example of a typical user interaction is as follows:

  "Click **Draw** > **Line**"

This instruction means that you should click the **Line** command on the **Draw** ribbon tab. An image of the command icon, or a partial view of the ribbon, is often included with the instruction.

- The *menu bar* (located above the ribbon) is a group of the main commands of an application arranged by category such File, Edit, View, Project, etc. An example of a typical user interaction is as follows:

  "On the **File** menu, click the **Open Examples** command" means you can click the **File** menu and then click **Open Examples** to launch the dialog box.

- Another alternative is to use the *shortcut menu* that appears when you click the right-mouse button. An example of a typical user interaction is as follows:

  “Right-click and select **Assign Excitation** > **Wave Port**” means when you click the right-mouse button with an object face selected, you can execute the excitation commands from the shortcut menu (and the corresponding sub-menus).

Getting Help: Ansys Technical Support

For information about Ansys Technical Support, go to the Ansys corporate Support website, http://www.ansys.com/Support. You can also contact your Ansys account manager in order to obtain this information.

All Ansys software files are ASCII text and can be sent conveniently by e-mail. When reporting difficulties, it is extremely helpful to include very specific information about what steps were taken or what stages the simulation reached, including software files as applicable. This allows more rapid and effective debugging.

Help Menu

To access help from the Help menu, click **Help** and select from the menu:

- **[product name] Help** - opens the contents of the help. This help includes the help for the product and its *Getting Started Guides*.
- **[product name] Scripting Help** - opens the contents of the *Scripting Guide*.
- **[product name] Getting Started Guides** - opens a topic that contains links to Getting Started Guides in the help system.

Context-Sensitive Help

To access help from the user interface, press **F1**. The help specific to the active product (design type) opens.

You can press **F1** while the cursor is pointing at a menu command or while a particular dialog box or dialog box tab is open. In this case, the help page associated with the command or open dialog box is displayed automatically.

Table of Contents

1 - Introduction

In this *Getting Started* guide, you will learn how to determine temperature and heat flux distributions using the *Ansys Electronics Desktop* application. Specifically, the guide provides an example of setting up, solving, and reviewing the results of a *Steady-State Thermal* solution in a *Mechanical* design. The focus is on thermal analysis and not solid modeling. Therefore, a CAD solid model has been provided as the starting point for the exercise.

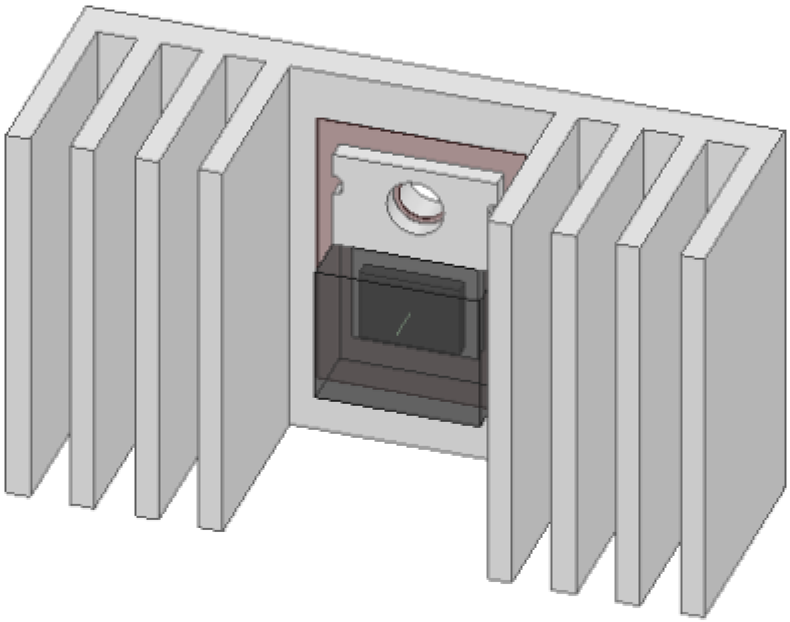
By following the procedures in this guide, you will learn how to perform the following tasks:

- Set general options
- Insert a mechanical design and choose the thermal solution type
- Set 3D UI options and choose the model units
- Import a CAD solid model
- Group the transistor components (body, die, and tab)
- Specify the materials
- Imprint the insulator on the heat sink
- Assign contact to the front and back insulator faces (to account for thermal resistance)
- Assign a convection boundary
- Define the ambient temperature
- Assign a heat generation excitation
- Assign length-based mesh refinement
- Draw a series of polyline segments that will be used as a basis of plotting temperature results
- Set up, validate, and solve the thermal analysis
- Create a mesh overlay
- Create a plot of the temperature versus the distance along the previously drawn polyline
- Create a temperature magnitude overlay
- Create a heat flux vector overlay
- Create a fields summary of the heat flow rate of the convection boundaries

The CAD solid model you will import contains a transistor assembly (consisting of the die, mounting tab, and body) along with a heat sink and a thin rectangular insulator. The transistor is a standard JEDEC TO-220 package, but the leads have been omitted because heat dissipation via the leads is not being considered for this analysis, which is a conservative assumption.

You will assign contact to the front and back faces of the insulator and specify an appropriate thermal resistance. You will also assign a heat generation excitation to the transistor die, and a convection boundary to all exposed faces of the heat sink.

For the purpose of this exercise, we will assume that the back of the heat sink is not exposed to ambient air but is mounted to an enclosure or circuit board. We will assume that there is no heat dissipation from this face, which is another conservative assumption. Therefore, the convection load will be assigned to all faces of the heat sink except for the back face. The portion of the front-middle face that is covered by the insulator will be isolated from the rest of the face by imprinting and will also be omitted from the convection boundary assignment.



2 - Set Up the Project

In this chapter, you will perform the following tasks:

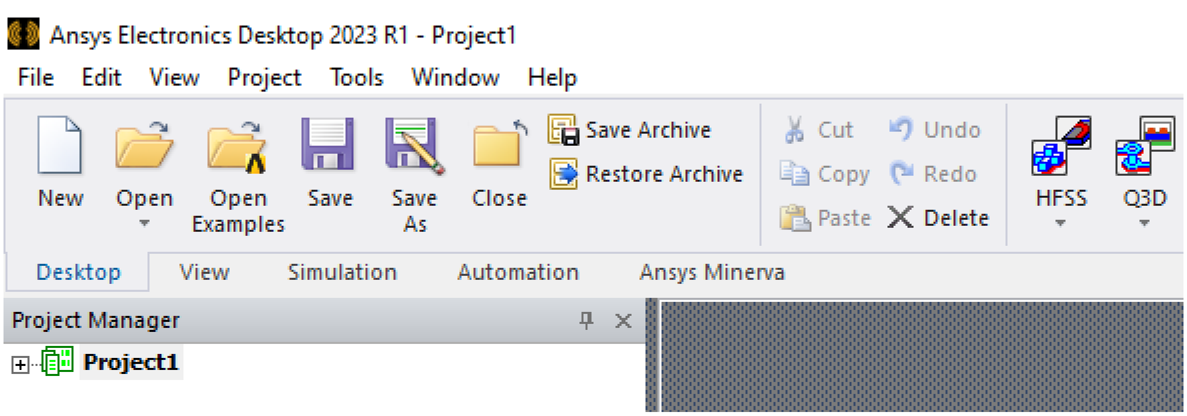
- Launch the Ansys Electronics Desktop application
- Set General Options
- Insert a Mechanical design
- Choose the solution type (Thermal)
- Set 3D UI options
- Choose the model units (in)
- Import the CAD solid model
- Group the transistor objects
- Save the project to a working folder

Launch Ansys Electronics Desktop

For convenience, a shortcut to the Ansys Electronics Desktop (EDT) application is placed on your desktop during program installation. Optionally, you may want to pin the shortcut to your Windows Start Menu too. Before proceeding to the next topic, launch EDT and add a blank project, as follows:

1. Double-click the EDT **Ansys Electronics Desktop** shortcut on your desktop (or the same shortcut on your Start Menu).

   The Ansys Electronics Desktop application opens:



> **Note:**
>
> When you launch the application, a new, blank project is created automatically. However, if you were previously using the application and closed the project, you will have to create a new project by completing the following optional step.

2. If no project is listed in the Project Manager, click **New** on the **Desktop** ribbon tab.

   Project*x* appears at the top of the Project Manager.

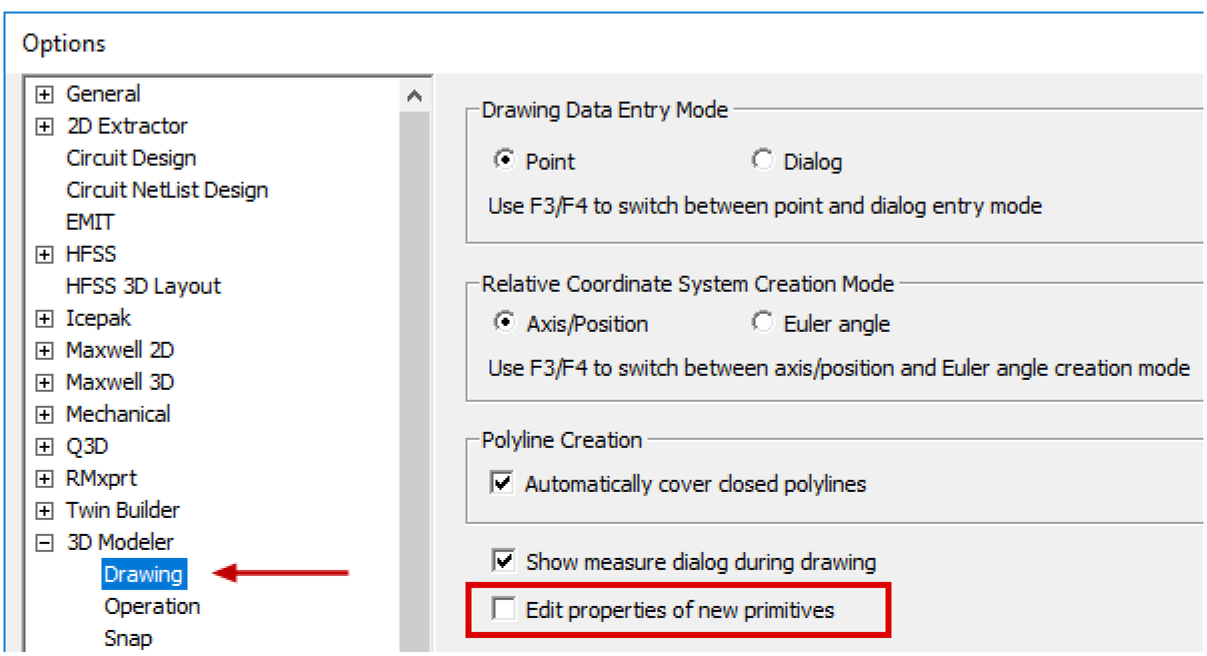
Set General Options

Before importing geometry and setting up the thermal analysis, there is one drawing option that you will ensure is disabled.

1. On the **Desktop** ribbon tab, click **General Options**.

   The *Options* dialog box appears.

2. In the options tree, expand the **3D Modeler** branch and select **Drawing**.
3. Ensure that the **Edit properties of new primitives** option is cleared:



> **Note:**
>
> Disabling this option prevents a *Properties* dialog box from opening automatically when you draw a new object. In this exercise, you will draw a multi-segment polyline, after which you will edit the object attributes in the docked *Properties* window, not the *Properties* dialog box.

4. Click **OK**.

Insert Mechanical Design

Insert a Mechanical design and choose the Thermal solution type as follows:

1. On the **Desktop** ribbon tab, click **Mechanical**.

   The mechanical design is listed beneath *Projectx* in the Project Manager:

   > **Note:**
   >
   > The default solution type for mechanical designs is *Steady-State Thermal*. However, you can change the default choice to the type you use most frequently. In the next step, you will determine the current *Solution Type* and change it if necessary.

2. If the design is not listed as **MechanicalDesign*x* (Steady-State Thermal)** in the Project Manager use the menu bar and click **Mechanical** > **Solution Type**. Then:
   a. Select **Steady-State Thermal** in the *Solution Type* dialog box that appears.
   b. Click **OK**.

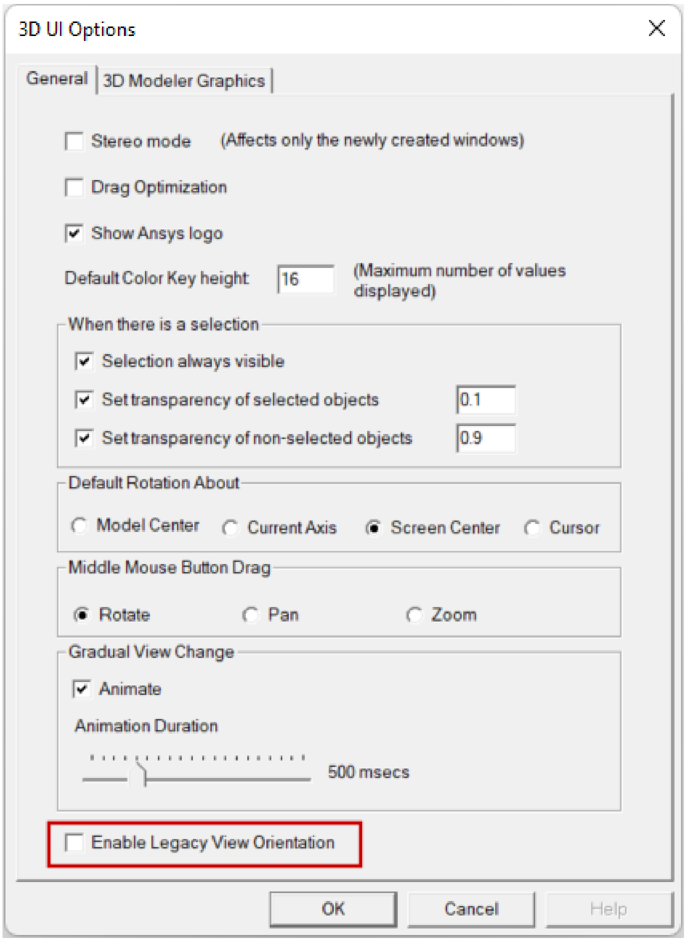
Set 3D UI Options and Units

Ensure that the legacy view orientation scheme is not being used, since the instructions and images in this guide are based on the new view orientation scheme introduced in release 2024 R1.

1. From the menu bar, click **View** > **Options**.

   The *3D UI Options* dialog box appears.

2. Ensure that **Enable Legacy View Orientation** is cleared:



3. Click **OK**.

The heat sink CAD model was originally constructed using English inch (in) length units. Set the Ansys Electronics Desktop model units to match this type. In a later procedure, you will add a series of polyline segments to the model to be used as the basis for plotting temperature results. The instructions for creating the polyline are based on coordinates in inches.

4. On the Draw ribbon tab, click **Units**. (This command has no icon. It is located immediately to the left of the *Grid* settings.)

   The *Set Model Units* dialog box appears.

5. Choose **in** (inch) from the **Select units** drop-down menu.
6. Click **OK**.

Import CAD Model and Group Parts

1. Using the menu bar, click **Modeler** > **Import**. Then, in the *Import File* dialog box that appears:

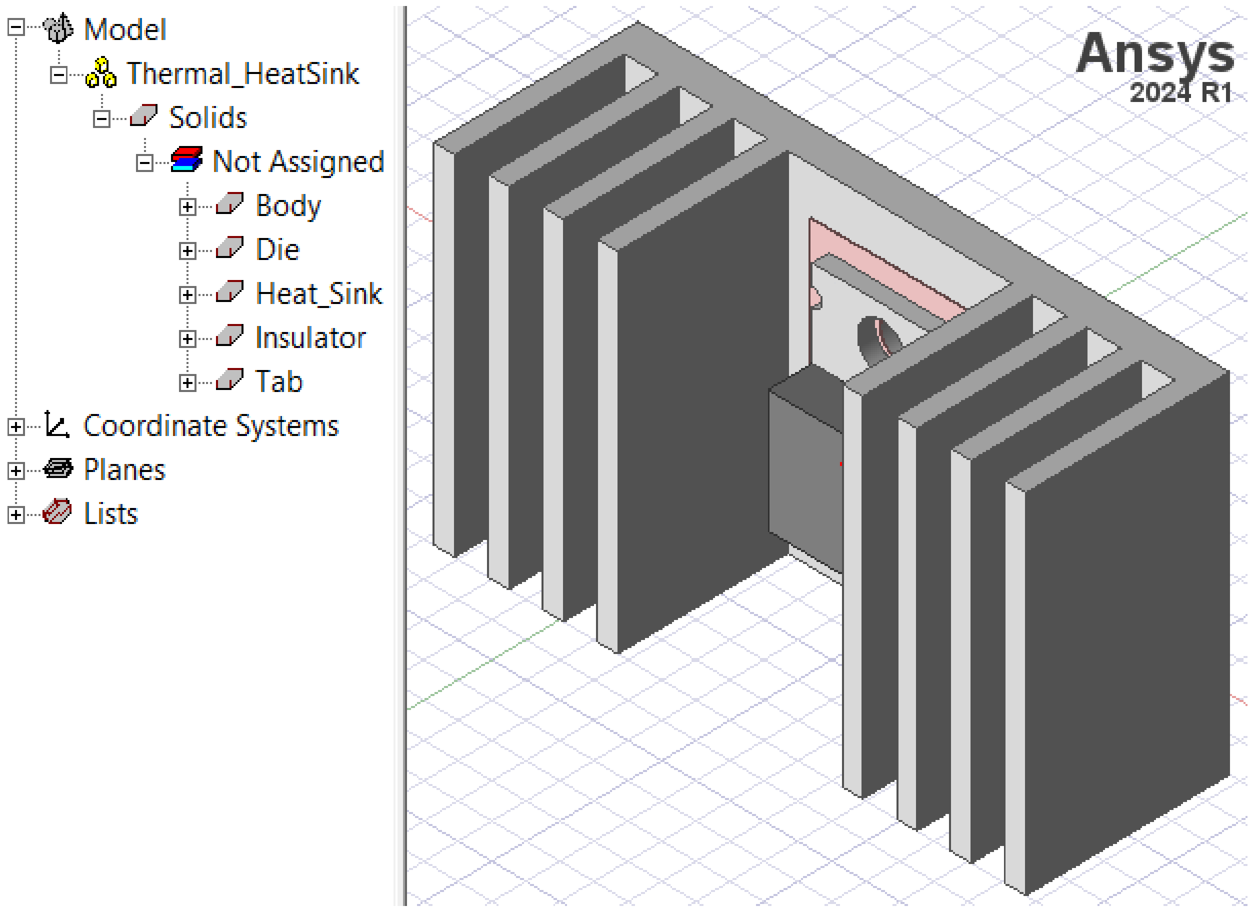
a. Navigate to the Ansys Electromagnetics Suite installation folder (typically C:\Program Files\AnsysEM\v*xxx*\<*platform*>, where *xxx* is the version number, such as v241 for version 2024 R2, and <*platform*> is either Win64 or Linux64.

b. From this folder, drill down to the subfolder **\Help\Mechanical**.

c. Select the file, **Thermal_HeatSink.x_t**, and click **Open**.

   The geometry appears in the Modeler window.

2. Click in the Modeler window's background area to clear the geometry selection. You should see the following:



Since you will not be creating any solid geometry during this exercise, hide the grid, ruler, and coordinate system axes, as follows. The *View Orientation Gadget* will remain visible as a global directions indicator.

3. On the **Draw** ribbon tab, click **Grid** to toggle off the grid visibility.
4. On the **Draw** ribbon tab, click **Ruler** to toggle off the ruler visibility.
5. From the menu bar, click **View** > **Coordinate System** > **Hide**.

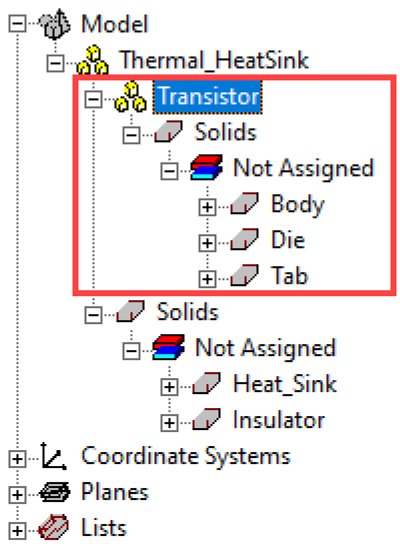
The transistor consists of three separate bodies. For convenience of selecting the transistor objects and for better model organization, group the transistor objects, as follows:

6. Under *Model* > *Solids* > *Thermal_HeatSink* > *Not Assigned* in the History Tree, select **Body**, **Die**, and **Tab**.
7. Right-click one of the selected items and choose **Group** > **Create** from the shortcut menu.

   Group1 is created, listed in the History Tree, and its properties appear in the docked *Properties* window.

8. In the *Value* column of the docked *Properties* window, change the group **Name** to **Transistor** and press **Enter**.

   The appearance of the History Tree should now be as follows:

Finally, save the project to a suitable working folder and specify the file name, as follows:

9. On the **Desktop** ribbon tab, click  **Save As**. Then, in the *Save As* dialog box that appears, do the following:
   a. Navigate to a working folder of your choice. (Do not attempt to write to the program installation path.)

      > **Note:**
      >
      > Optionally, you can click the **Create New Folder** icon () within the *Save As* dialog box to create a new working folder in a suitable location.

   b. In the **File name** text box, type **Thermal_HeatSink**.
   c. Click **Save**.

3 - Define Materials

The imported CAD solid bodies have no material assignment. Assign materials to the five objects as follow:

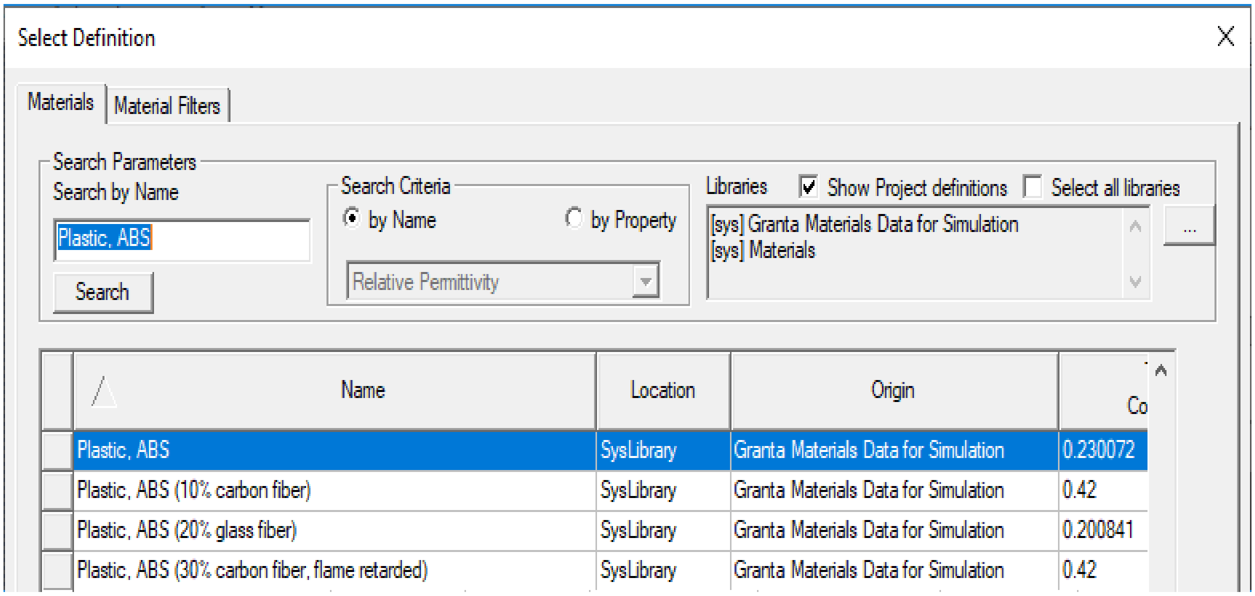
1. Under *Model* > *Thermal_HeatSink* > *Transistor* > *Solids* > *Not Assigned* in the History Tree, select **Body**.
2. In the docked *Properties* window, choose **Edit** from the **Material** drop-down menu.

   Do the following in the *Select Definition* dialog box that appears:

   a. Click the **ellipsis** button (...) near the upper-right corner of the dialog box to display the list of material libraries to include.
   b. Ensure that the following two libraries are selected:
      - **[sys] Granta Materials Data for Simulation**
      - **[sys] Materials**

      If you have additional libraries already selected, you can leave them selected too.
   c. Click the **X** in the upper-right corner of the libraries list to close it.
   d. Type **plastic, a** in the **Search by Name** text box.
   e. In the pop-up list of matching materials, locate and double-click **Plastic, ABS [sys:Granta Materials Data for Simulation]**.

      The pop-up list is dismissed, and the material, **Plastic, ABS**, is located and selected in the full *Materials* list:

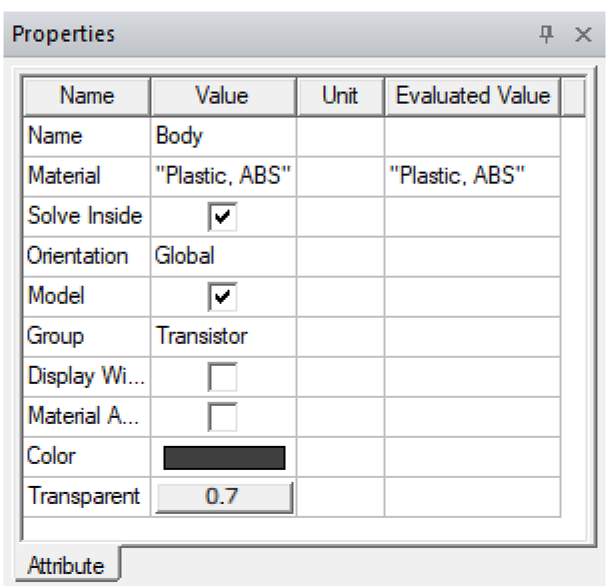


f. Click **OK** to apply this material to the transistor body and to close the *Select Definition* dialog box.

3. Keep the original component color, but set the **Transparent** value to **0.7**.

   Body transparency enables the internal *Die* object to be seen.

   The *Attribute* tab of the docked *Properties* window should now match the following image:



4. Under *Model* > *Thermal_HeatSink* > *Transistor* > *Solids* > *Not Assigned* in the History Tree, select **Die**.

5. In the docked *Properties* window, choose **Edit** from the **Material** drop-down menu.

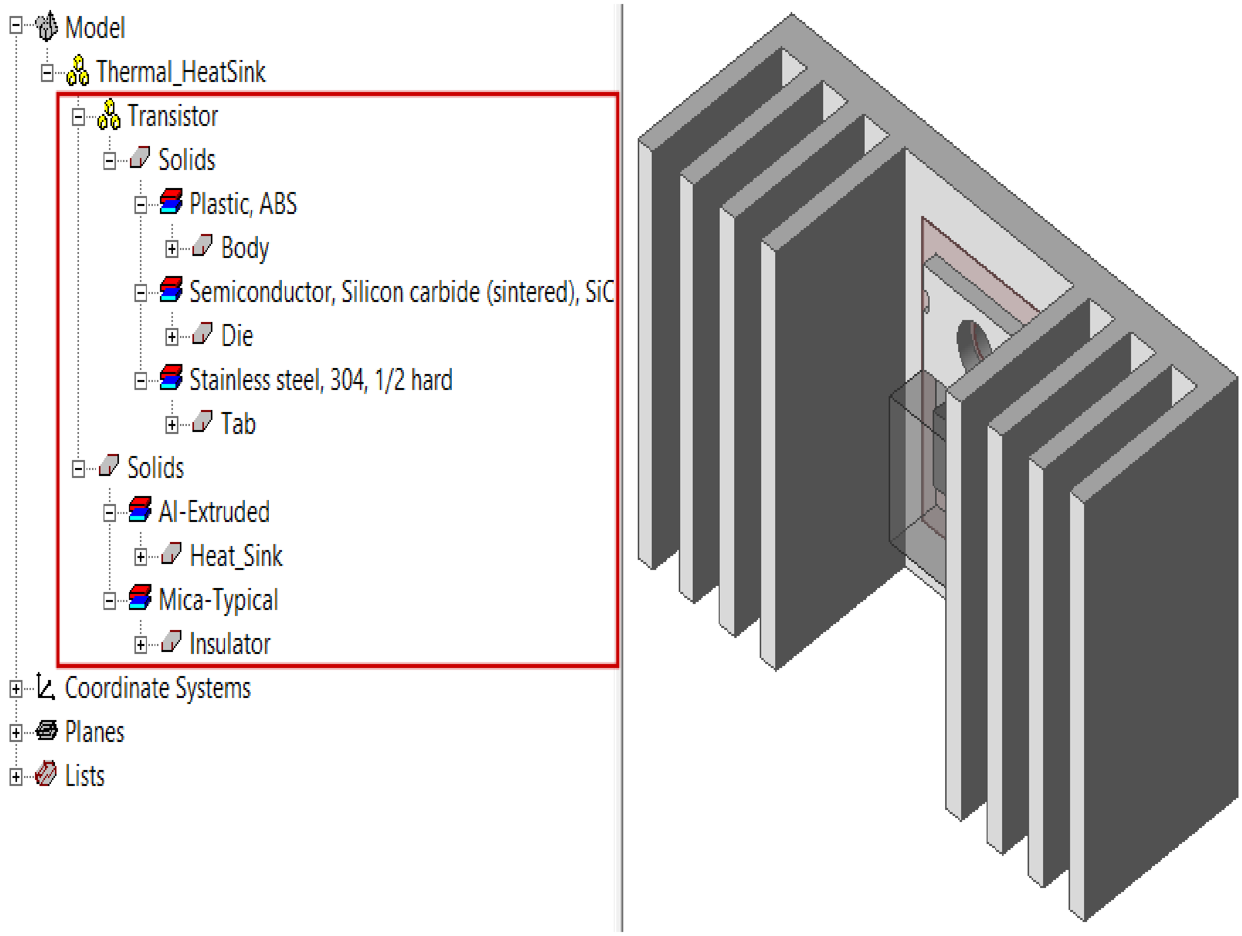
   Do the following in the *Select Definition* dialog box that appears:

   a. Type **semicon** in the **Search by Name** text box.
   b. In the pop-up list of matching materials, scroll down as needed and double-click the material, **Semiconductor, Silicon carbide (sintered), SiC sys:Granta Materials Data for Simulation** to select it and to dismiss the pop-up list.
   c. Click **OK**.
6. Under *Model* > *Thermal_HeatSink* > *Transistor* > *Solids* > *Not Assigned* in the History Tree, select **Tab**.
7. In the docked *Properties* window, choose **Edit** from the **Material** drop-down menu.
   a. Type **stain** in the **Search by Name** text box.
   b. In the pop-up list of matching materials, scroll down as needed and double-click the material, **Stainless steel, 304, 1/2 hard [sys:Granta Materials Data for Simulation]** to select it.
   c. Click **OK**.
8. Under *Model* > *Thermal_HeatSink* > *Solids* > *Not Assigned* in the History Tree, select **Heat_Sink**.
9. In the docked *Properties* window, choose **Al-Extruded** from the **Material** drop-down menu.

   This material is accessible without opening the *Select Definition* dialog box because it is the default material for thermal solutions.

10. Under *Model* > *Thermal_HeatSink* > *Solids* > *Not Assigned* in the History Tree, select **Insulator**.
11. In the docked *Properties* window, choose **Edit** from the **Material** drop-down menu.
    a. Type **mica** in the **Search by Name** text box.
    b. In the pop-up list of matching materials, locate and double-click **Mica-Typical [sys:Materials]** to select it.
    c. Click **OK**.
12. Set the **Transparent** value to **0.7**.
13. Clear the selection.

    Your History Tree should now look like the following image:



14. **Save** your project.

Even though Ansys Electronics Desktop automatically saves your projects after a pre-defined number of edits, it's a good idea to save your progress often.

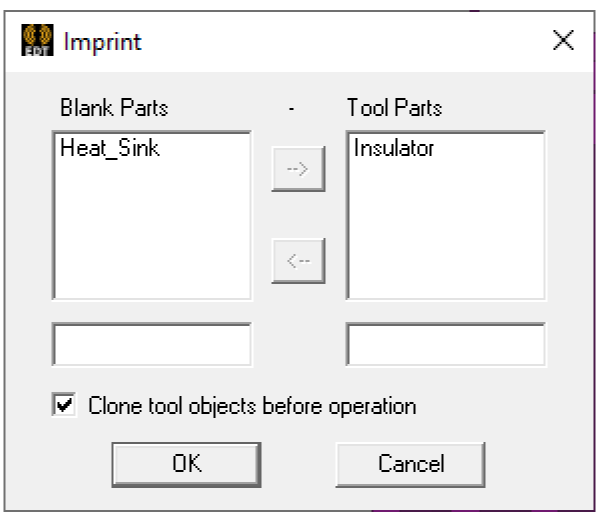
4 - Imprint Insulator on Heat Sink

In a later procedure, you will apply a convection boundary to the exposed faces of the heat sink. Additionally, one of those convection faces is partially covered by the insulator, and a contact will be defined at this insulator-to-heat sink interface to specify thermal resistance.

It is invalid to have convective heat loss from a face that is not exposed to ambient air, such as any portion of an object's face that is in direct contact with another object. Convection boundaries in Mechanical–Thermal solutions are automatically ignored for any portion of a convection face that is in contact with another body. However, this capability is currently limited to default bonded contact between adjacent parts. If you manually assign contact for the purpose of imposing thermal resistance, the contact face will **not** be automatically filtered out of the convection boundary. Therefore, you must imprint the insulator on the heat sink to split the exposed portion of the middle-front heat sink face from the contact portion of the face. Then, when later assigning the convection boundary, you will exclude the heat sink face under the insulator from the selection set.

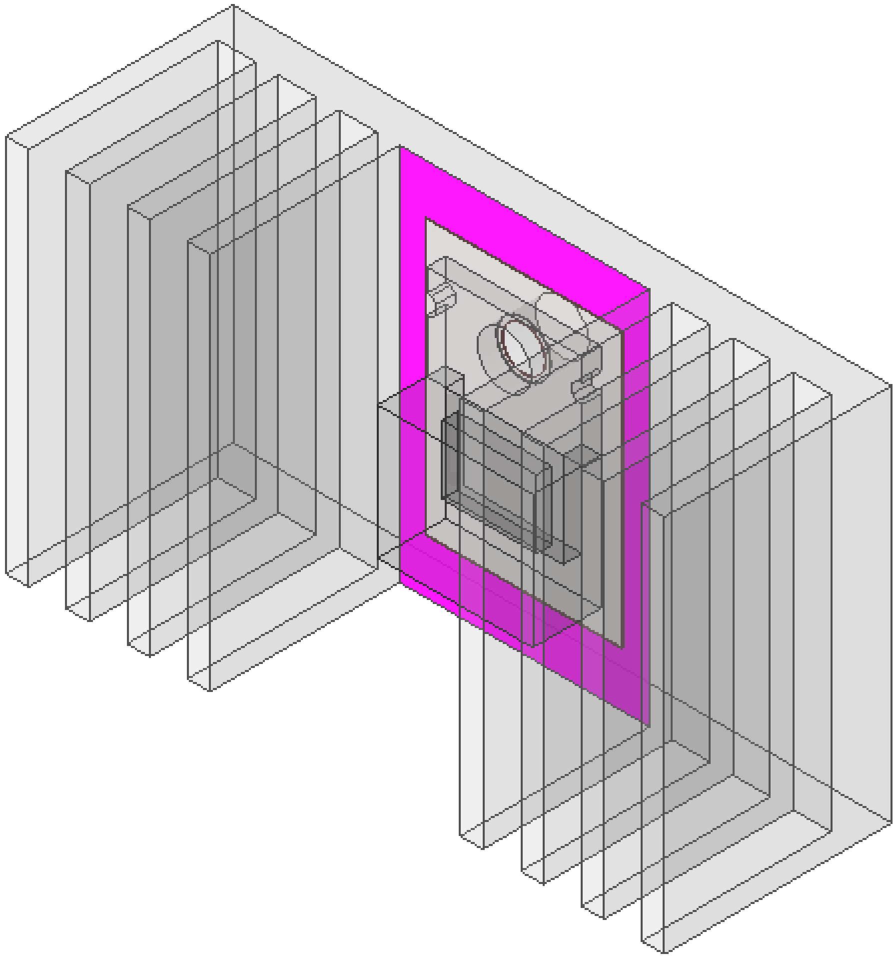
1. In **Object** selection mode, click the **Heat Sink** to select it.
2. **Ctrl+click** the **Insulator** to select this object also.
3. On the **Draw** ribbon tab, click **Imprint**.
4. In the *Imprint* dialog box that appears, ensure that *HeatSink* is the *Blank* part and *Insulator* the *Tool* part.
5. Select **Clone tool objects before operation**.

   The dialog box should look like the following image:

6. Click **OK**.
7. Press **F** to switch to the *Face* selection mode.
8. Click the exposed portion of the heat sink face to which the transistor and insulator are mounted.

The highlighting should confirm that the area underneath the insulator is no longer a part of the selected heat sink face, as shown below:



9. Clear the selection.

5 - Assign Contact

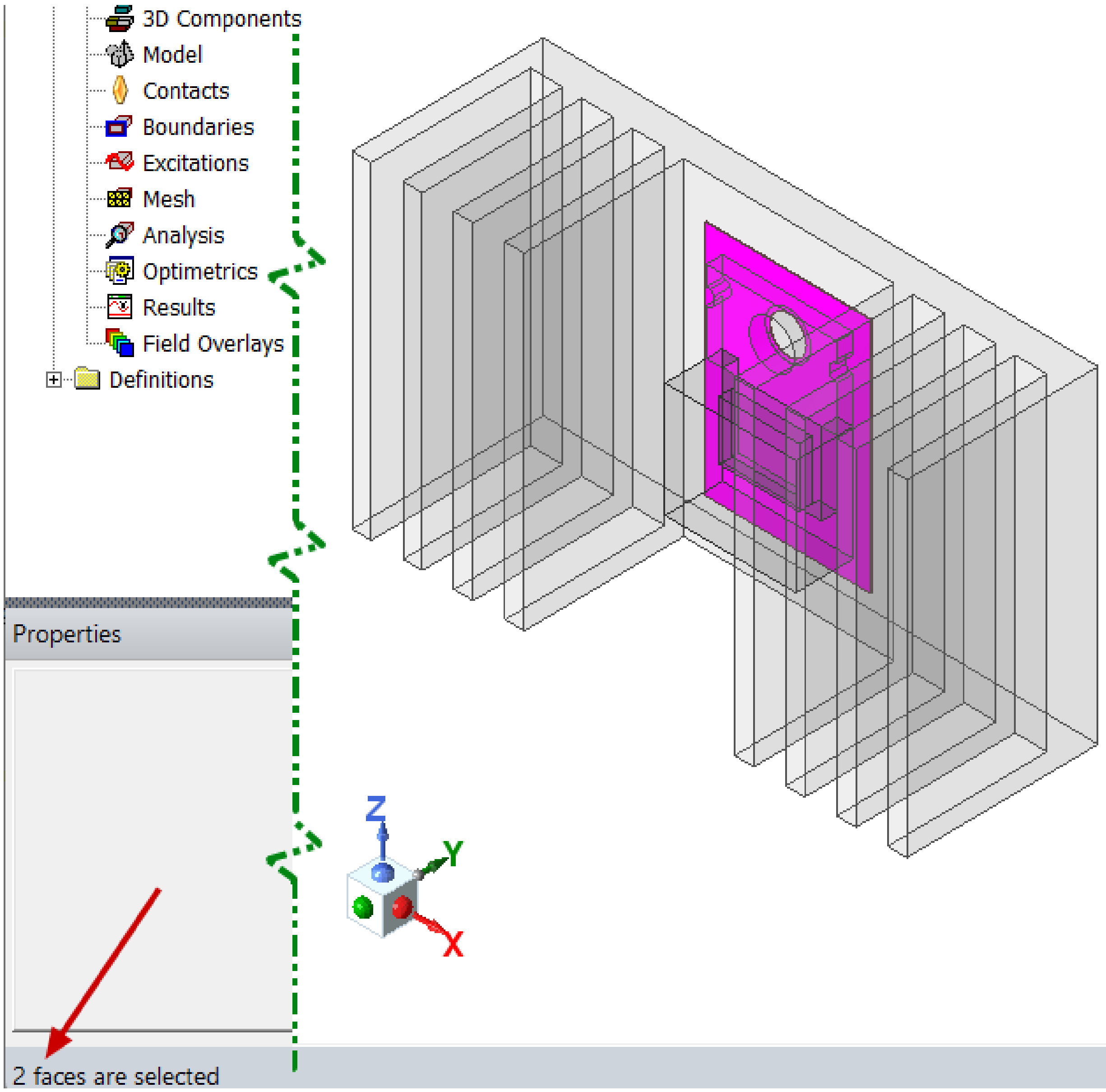
It is possible to account for the effects of the electrical insulator and contact resistance between it and the heat sink on one side and the semiconductor on the other. To do so, you would simply assign contact and specify a conductivity or resistance value that represents the total effect of the insulator material conductivity and thickness plus the thermal resistance on both sides of the insulator. However, for this exercise, the insulator is included in the model as a thin solid object. In this way, you will be able to plot the temperature along a line running through the semiconductor, insulator, and heat sink, and you will see the temperature gradient through the insulator thickness. You could also overlay the temperature results on the insulator alone to see the different contour colors on the front and back faces.

In Mechanical–Thermal designs, you select a single face to define contact, not the pair of faces that are touching. The target face is determined automatically by the solver.

We will define a thermal impedance, which is an area-based resistance. The value will be 0.15 $°C \cdot inch^2/W$ at both the front and back faces. Define the contact as follows:

1. In *Face* selection mode, click the exposed front face of the insulator about midway between it's outer perimeter and the edge of the transistor. Then, Press **B** to change the selection to the back face of the insulator.
2. **Ctrl+click** the exposed front face of the insulator again to select it too.

   The status message in the lower left corner of the application window should show that "*2 faces are selected*":

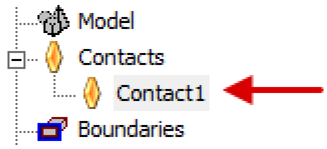


3. In the Project Manager, right-click **Contacts** and choose **Assign** > **Contacts** from the shortcut menu.

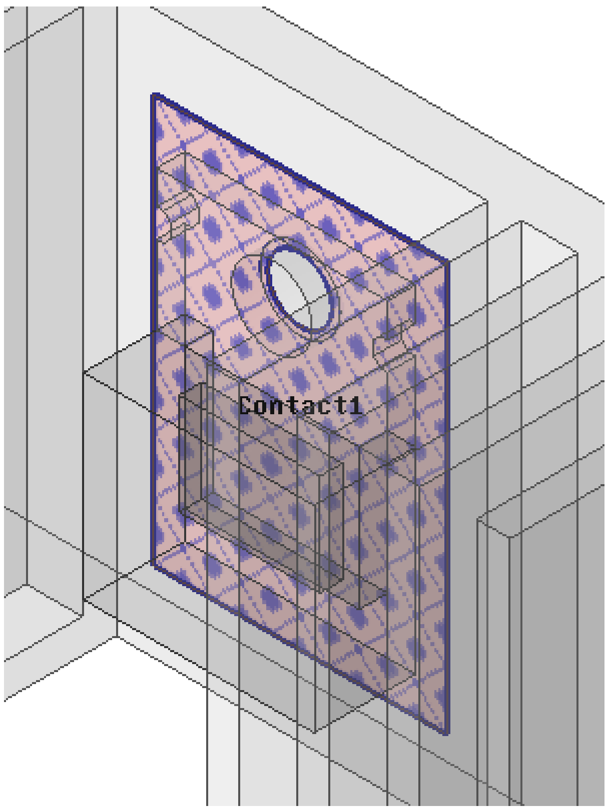
   Then, in the *Contact* dialog box appears, do the following:

   a. From the **Resistance Type** drop-down menu, select **Thermal Impedance**.
   b. In the **Impedance** text box, type **0.15** and select the units **cel_in2_per_W** from the adjacent drop-down menu.
   c. Click **OK**.

Contact1 appears in the Project Manager:



4. In the Project Manager, select **Contact1**, to visualize the assignment on the model, as shown below:



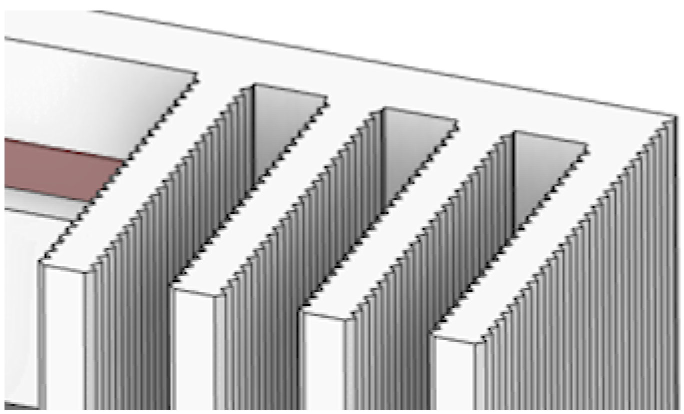
5. Clear the selection.

6 - Assign Convection Boundaries

A convection boundary transfers heat between object faces and the ambient environment based on a heat transfer film coefficient and ambient temperature that you specify. You do not actually include an air region to represent the ambient environment, it is accounted for mathematically. The film coefficient for this exercise is consistent with a low to moderate amount of forced convection. That is, the film coefficient represents air flow that does not occur solely due to buoyancy and natural convection effects but is assisted by a fan.

Heat is removed from this model by two separate convection boundaries, each assigned to various faces of the heat sink that are exposed to ambient air. Two heat sink faces will be excluded—the back (-X) face, which is assumed to be mounted to an enclosure or circuit board, and the portion of the heat sink's transistor mounting face that is in contact with the insulator. The remaining faces are grouped as follows:

- **Left and right faces of each fin:** In the actual heat sink that the model represents, these faces are V-grooved to increase the surface area by approximately 41% relative to smooth faces:

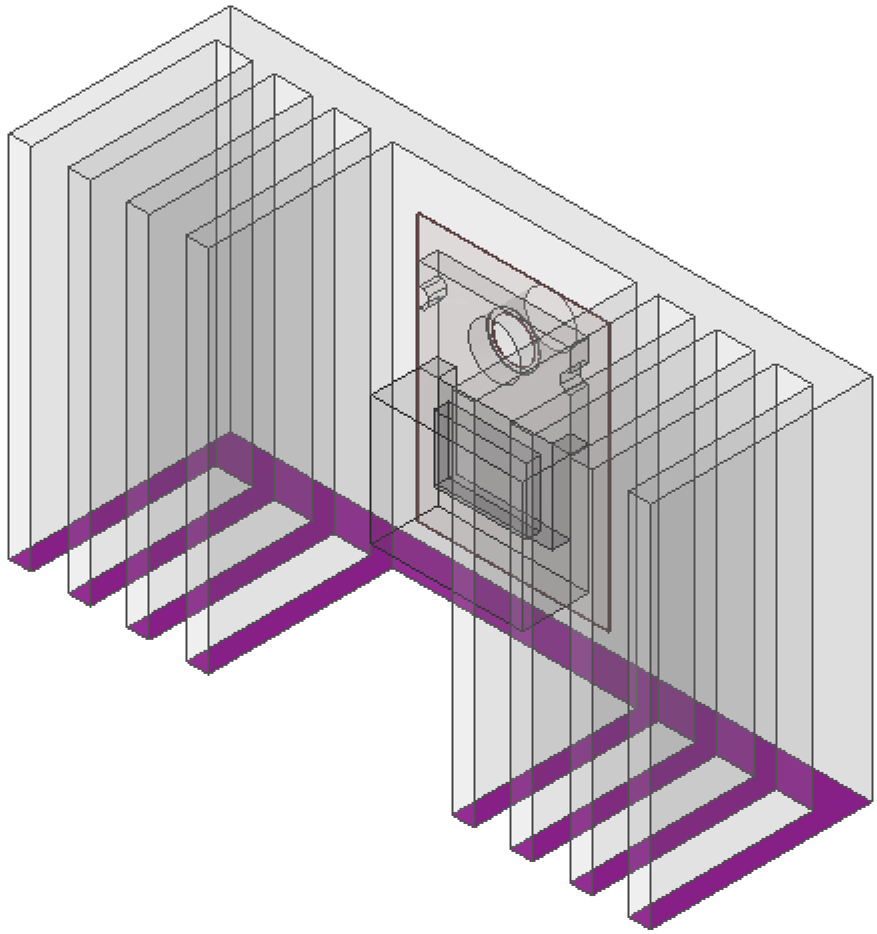
  However, including the grooves in the CAD model unnecessarily complicates the geometry, increasing the element count and solution time. There is a much more efficient way to account for the grooved surfaces. All faces of the imported model are smooth. When assigning convection to the left and right faces of the fins, simply increase the film coefficient by 41%, to **14.1 W/(m^2·°C)**, achieving the same increase in heat flow as the grooved surface area would achieve at the actual film coefficient of 10 W/(m^2·°C). The modeled thickness of the fins is the average thickness of the actual grooved fins.

- **All other faces:** These faces are smooth, and the film coefficient is assumed to be **10 W/(m^2·°C)**.

> **Note:**
>
> Convection boundaries are automatically excluded from portions of assignment faces that are in default bonded contact with other solid objects. It is no longer necessary to use imprinting operations to split such faces to isolate contact regions from convection boundaries. However, for user-defined contact faces (those with thermal resistance), portions of larger faces that have thermal contact with other objects are *not* automatically excluded from convection boundaries. In an earlier step, you imprinted the insulator onto the heat sink to split the mounting face into two faces. For this exercise, you will include only the portion of the transistor mounting face that is outside of the insulator contact area in the convection boundary assignment. You will not include the portion of the face underneath the insulator.

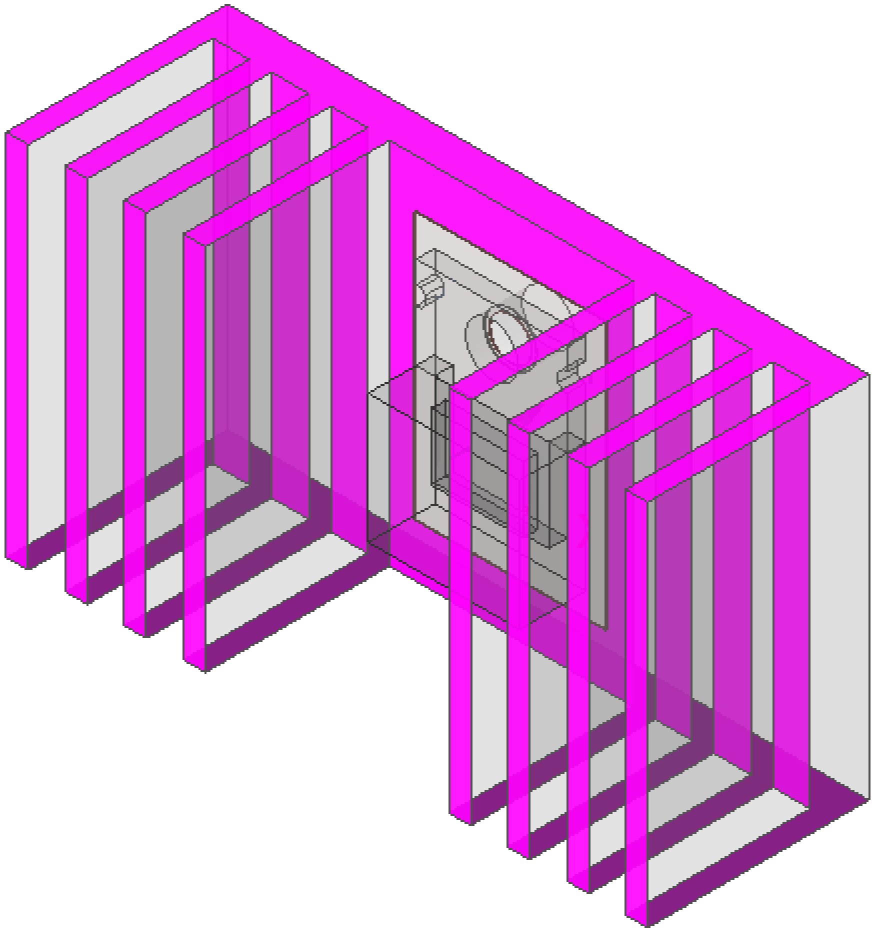
For the purpose of this exercise, you will ignore the small amount of convection from the exposed faces of the transistor and insulator. Ignoring heat dissipation at these faces and at the back of the heat sink is conservative. It is better to slightly overestimate the transistor temperature than to underestimate it.

1. With the model view at the default *Isometric* orientation, select all of the smooth faces of the heat sink as follows:
   a. Click inside the Modeler window to ensure that it is the active window and press **F** to switch to the *Face* selection mode.
   b. Click the rightmost face, just slightly above its bottom edge. Then press **B** to select the *Next **B**ehind* face. The bottom of the heat sink should now be selected:



c. **Ctrl+click** while also selecting sixteen (16) additional heat sink faces.

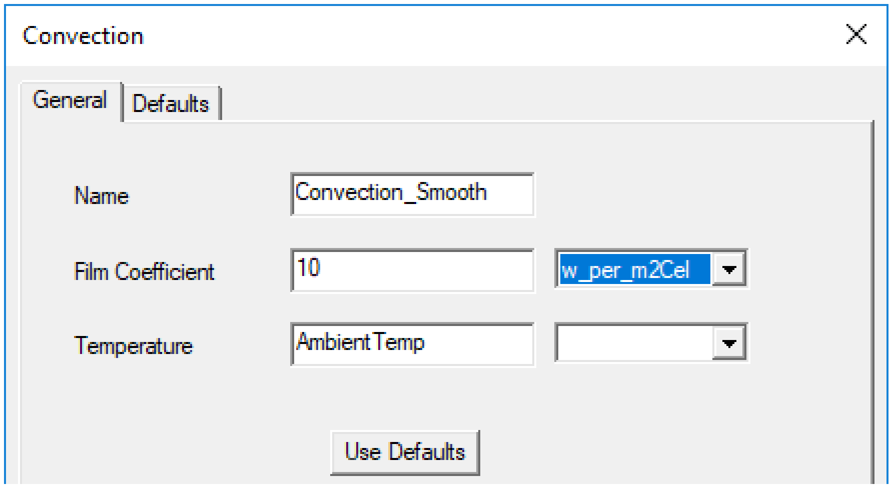
   A total of seventeen (17) faces should now be selected, as shown in the following image:



2. Right-click **Boundaries** in the Project Manager and choose **Assign** > **Convection** from the shortcut menu.

   Then, in the *Convection* dialog box that appears, make the following changes:

   a. Change the default **Name** to **Convection_Smooth**.
   b. Under *Film Coefficient* type **10** in the **Uniform** text box.
   c. Choose **w_per_m2cel** [W/(m^2°C)] from the adjacent units drop-down menu.

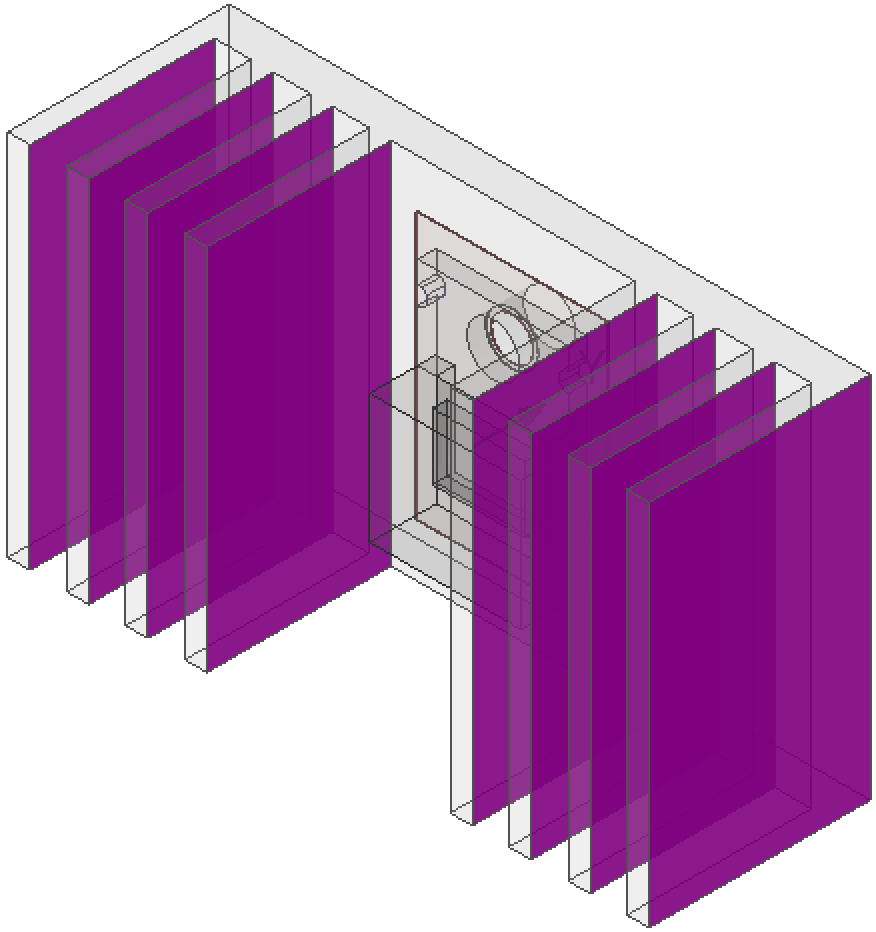


> **Note:**
>
> You can define the *Temperature* for a *Convection* boundary directly within this dialog box, enabling multiple convection boundaries based on different temperatures. Or, you can use the *AmbientTemp* design variable. For this exercise, you will use the design variable method, which is the default. In the next topic, you will learn how to modify the *AmbientTemp* design variable.

d. Click **OK** to apply the convection load to the selected faces.

3. Select the grooved faces of the heat sink as follows:

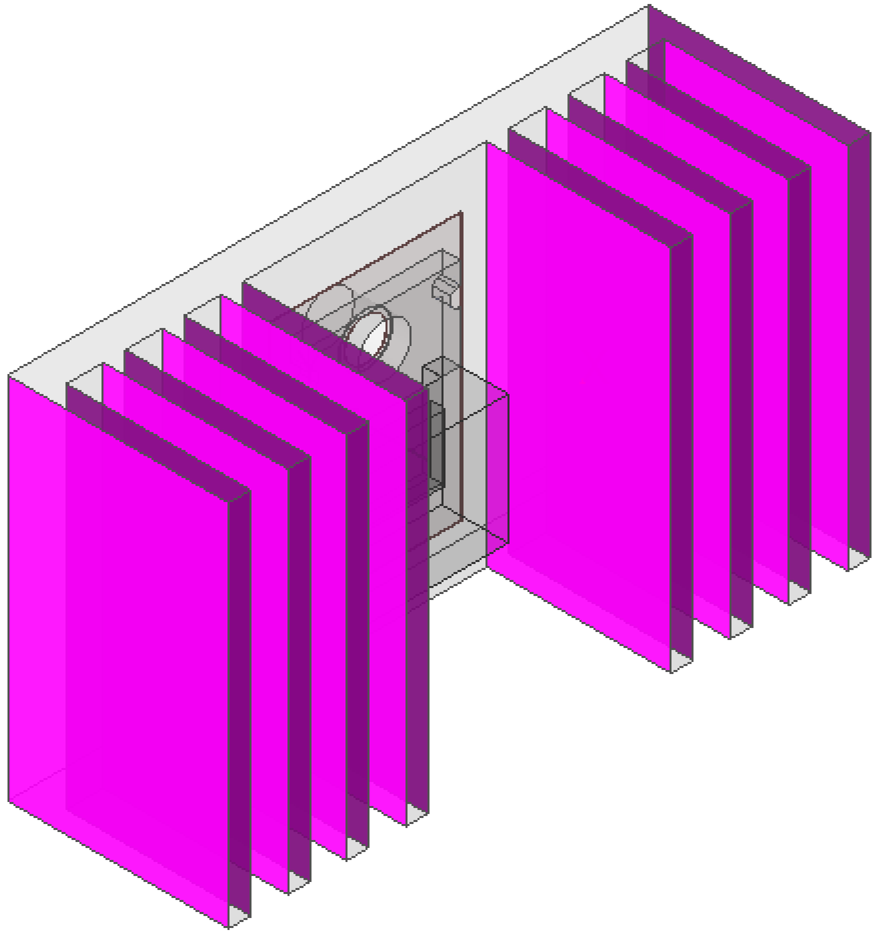
a. Select the eight (8) **right-side faces** of the heat sink fins:



b. **Alt + double-click** just inside the upper-left corner of the Modeler window's drawing canvas. This action changes the orientation to an alternative isometric view, which facilitates selection of the left faces of each fin.

c. **Ctrl+click** to additionally select the eight (8) left-side faces of the heat sink fins.

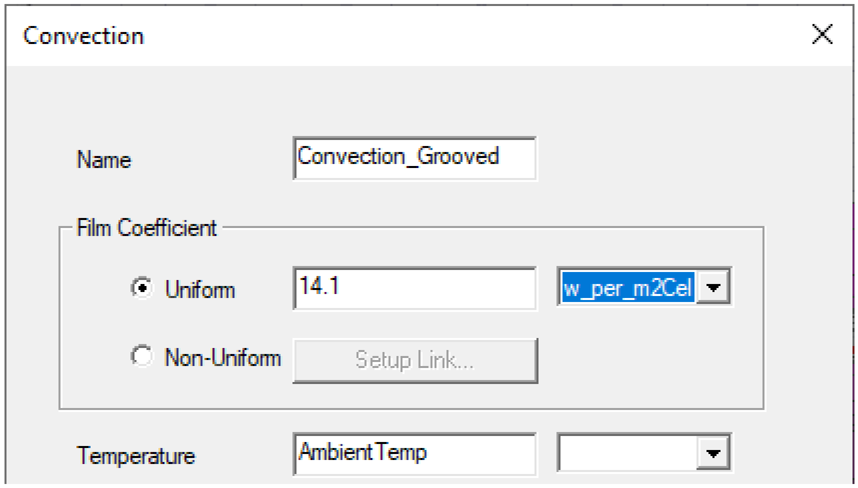
A total of sixteen (16) faces should now be selected, and the model should resemble the following image:



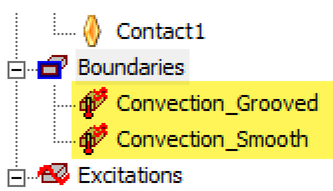
4. Right-click **Boundaries** in the Project Manager and choose **Assign** > **Convection**.

   Then, in the *Convection* dialog box that appears, make the following changes:

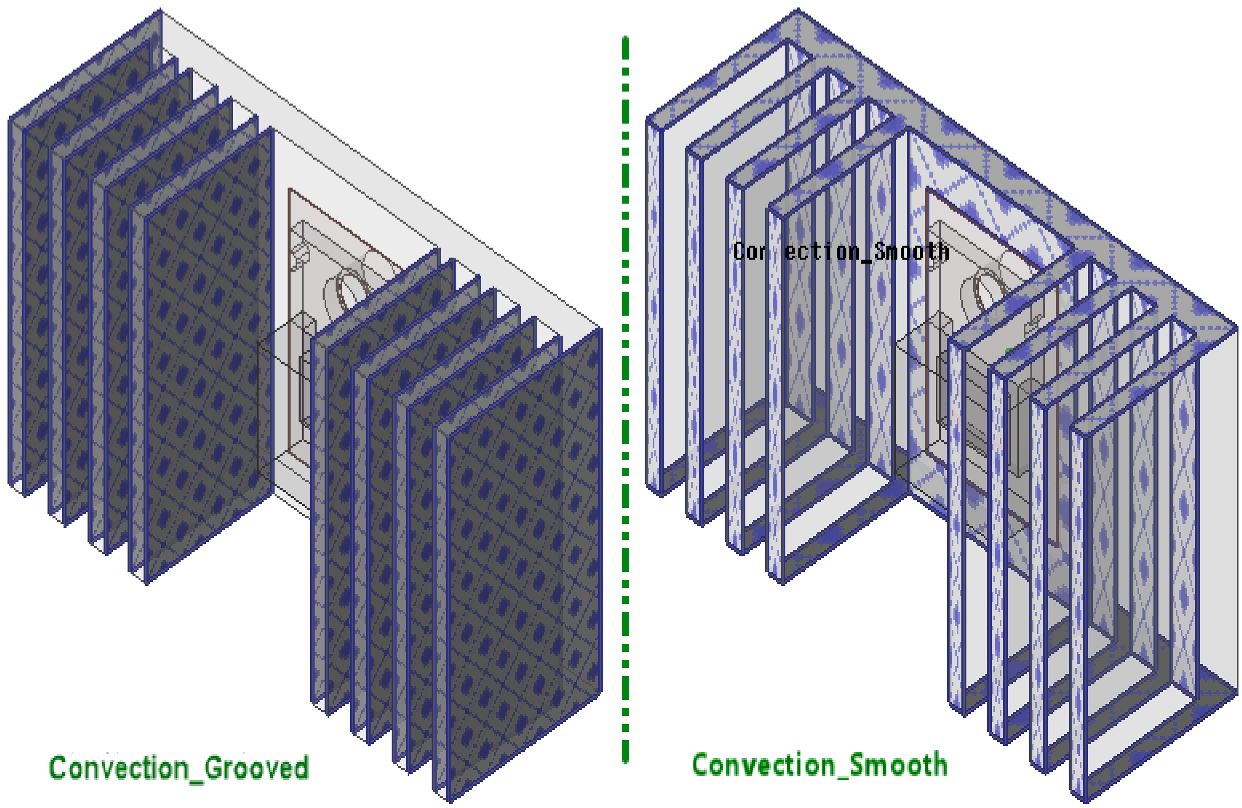
   a. Change the default **Name** to **Convection_Grooved**.
   b. Under *Film Coefficient* type **14.1** in the **Uniform** text box.
   c. Choose **w_per_m2cel** from the adjacent units drop-down menu.



5. Click **OK** to apply the convection load to the selected faces.
6. On the **Draw** ribbon tab, click **Orient** to restore the default *Isometric* view of the model. (You do not have to access the *Orient* drop-down menu because *Isometric* is the default view.)
7. The two convection boundaries are listed in the Project Manager:



Select each boundary in turn to verify the face assignments:

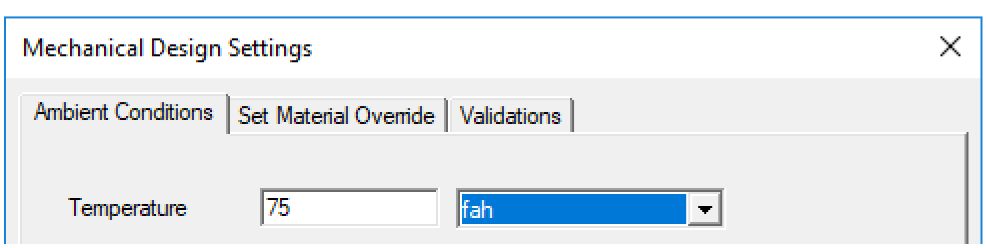


8. Clear the selection and **Save** your project.

Set Ambient Temperature

Define a global ambient temperature of 75° F for this model, as follows:

1. Using the menu bar, click **Mechanical** > **Design Settings**.
2. In the **Ambient Conditions** tab of the *Mechanical Design Settings* dialog box that appears, make the following changes:
   a. Type **75** in the **Temperature** text box.
   b. Select **fah** (°F) from the **Temperature** units drop-down menu.



3. Click **OK**.

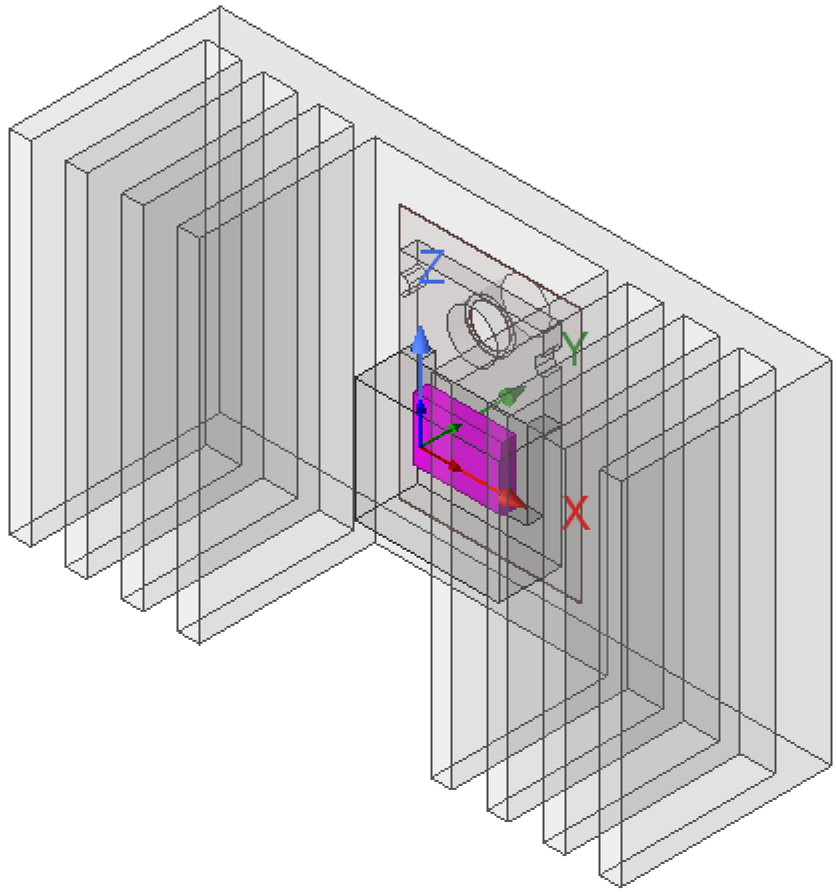
   Any boundary with *AmbientTemp* specified as the *Temperature* setting will be based on this global 75° F value.

7 - Assign Heat Generation

Heat input for this model comes from the transistor. Apply a *Heat Generation* excitation to the transistor as follows:

1. On the **Draw** ribbon tab, choose **Object** from the **Select** drop-down menu (or, with the Modeler window active, press **O**) to switch to the *Object* selection mode.
2. Click the visible portion of the transistor *Body* with the cursor positioned over the enclosed **Die**. Then, press **B** to select the **Die**.

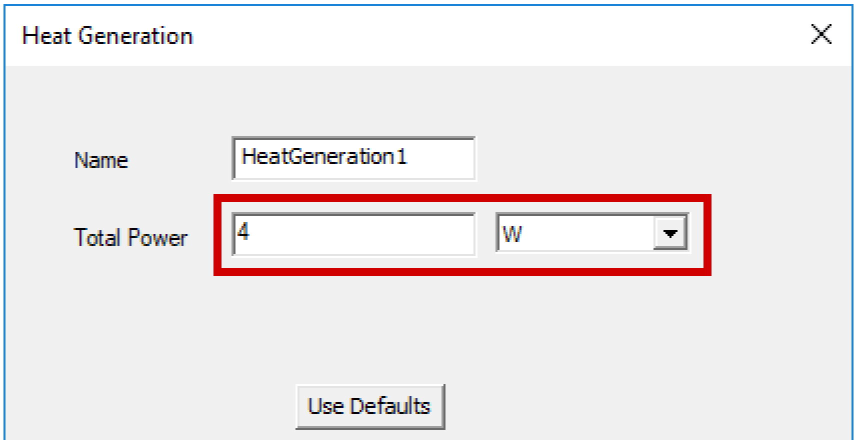
   Alternative, select **Die** from the History Tree (under *Transistor* > *Solids* > *Semiconductor*…).



3. Right-click **Excitations** in the Project Manager and choose **Heat Generation** from the shortcut menu.

   The *Heat Generation* dialog box appears.

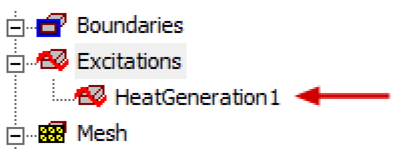
4. Specify a **Total Power** of **4 W**.



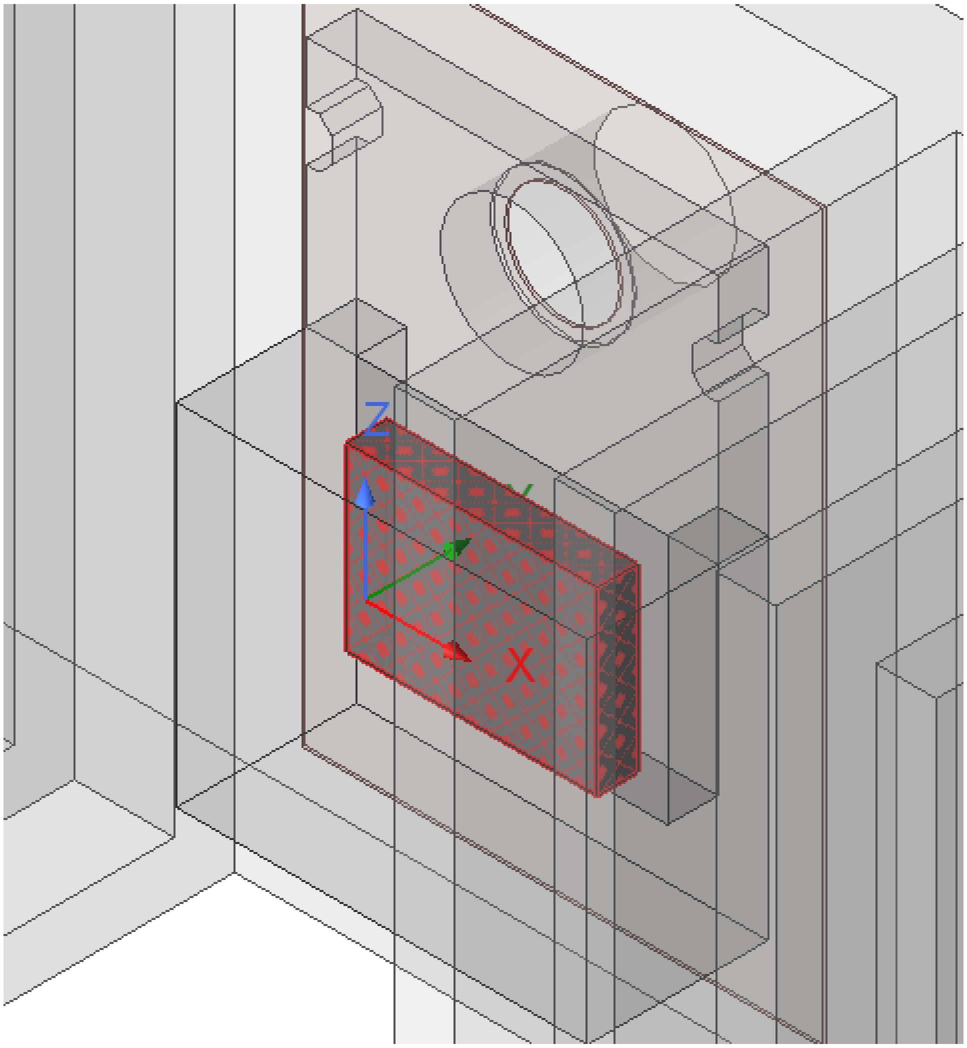
> **Note:**
>
> *Total Power* refers to the heat dissipated by the device, not the electrical power the device conducts or outputs to the circuit.

5. Click **OK**.

   HeatGeneration1 appears under *Excitations* in the Project Manager:



While this excitation is selected, you can visualize it on the model:

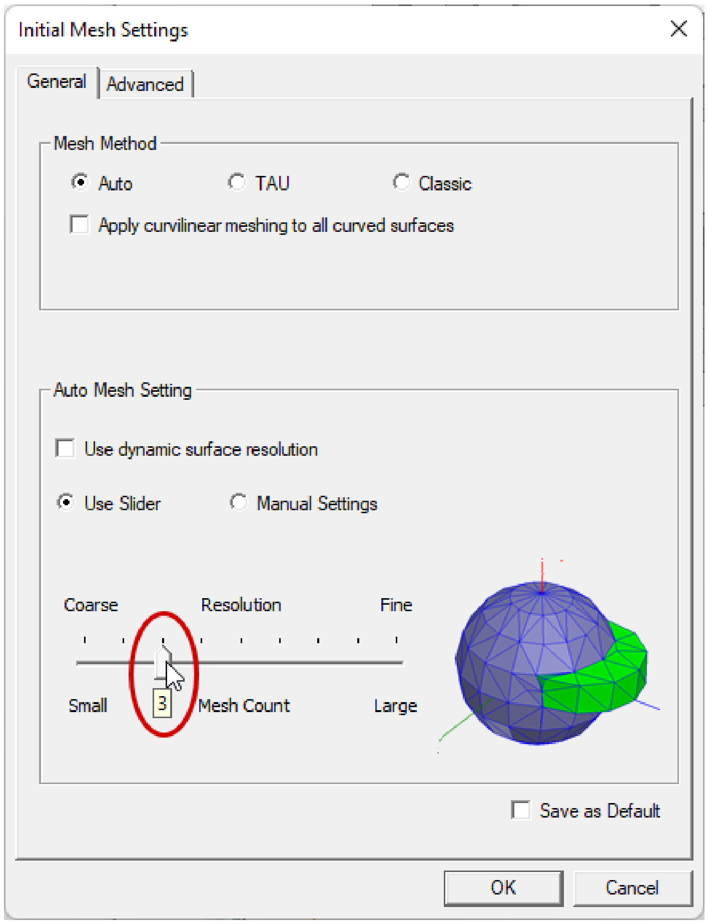


6. Clear the selection.

8 - Mesh Settings

Next, you will adjust the initial mesh settings and assign a length based mesh operation to the transistor and insulator objects.

1. In the Project Manager, right-click **Mesh** and choose **Initial Mesh Settings** from the shortcut menu. Then, in the dialog box that appears, do the following:
   a. Click and drag the slider two ticks to the left (to position 3)



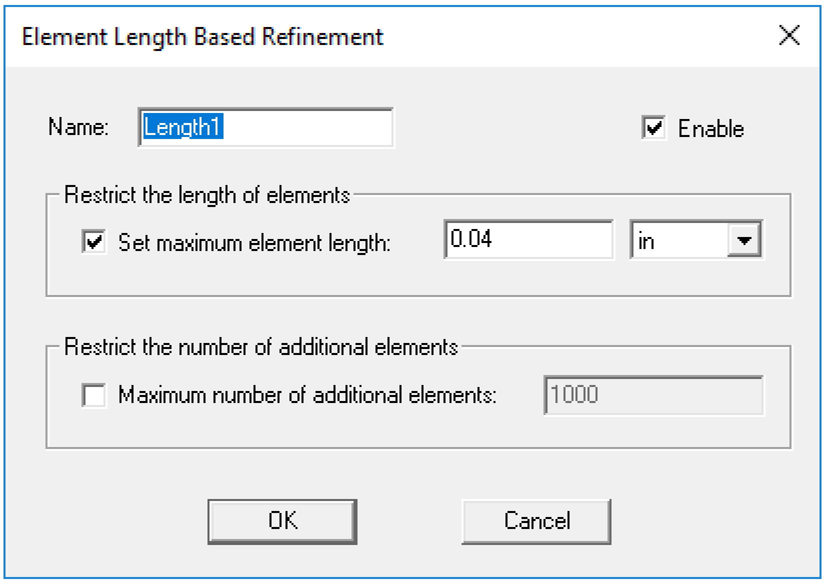
> **Note:**
>
> Mesh refinement is automatically performed on boundary faces. All of the heat sink, excluding the back face and the insulator contact face, will have a convection boundary assigned. Making this global mesh setting coarser prevents the heat sink from being unnecessarily finely meshed.

   b. Click **OK**.

2. Under *Model* > *Thermal_HeatSink* in the History Tree, right-click **Transistor** and choose **Select All**.
3. Under*Model* > *Thermal_HeatSink* > *Solids* > *Mica-Typical* in the History Tree, Ctrl+click **Insulator** to select it too.
4. Right-click **Mesh** in the Project Manager and choose **Assign Mesh Operation** > **Inside Selection** > **Length Based** from the shortcut menu.

   The *Element Length Based Refinement* dialog box appears.

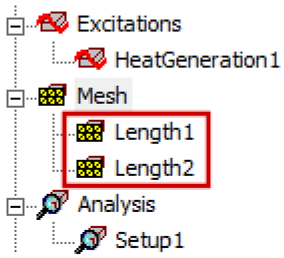
   a. Ensure that **Set maximum element length** is selected and specify **0.04 in** in the adjacent text box and units drop-down menu.

   b. Click **OK**.
5. Press **F** to switch to the *Face* selection mode.
6. Click the front face of the heat sink, between the insulator and top edge. Then, press **B** to change the selection to the back face of the heat sink.
7. Right-click in the Modeler window and choose **Assign Mesh Operation** > **On Selection** > **Length Based**. Then:
   a. Specify the same settings as shown in the preceding image (step 4.a).
   b. Click OK.

> **Note:**
>
> The back face of the heat sink will *not* have a convection boundary assigned to it and therefore will not be refined automatically. This mesh operation ensures a more uniform mesh throughout the heat sink.

Length1 and *Length2* are listed appears under *Mesh* in the Project Manager:



After running the analysis, you will create a mesh overlay to verify that there are at least two elements through the thickness of the fins throughout the heat sink. Heat flux results, in particular, benefit from a somewhat refined mesh.

9 - Draw Polyline

You can use a single or multi-segment polyline to specify a path for plotting thermal results. The selected result is plotted versus the distance along the simple or complex polyline. In this exercise, you will draw a three-segment polyline traversing the following points:

- Transistor body's front face (centered over the die)
- In the +Y direction to the midplane of the heat sink's back thickness
- In the +X and +Z directions to the top-right edge of the heat sink
- In the -Y direction to the top-right-front corner of the fin

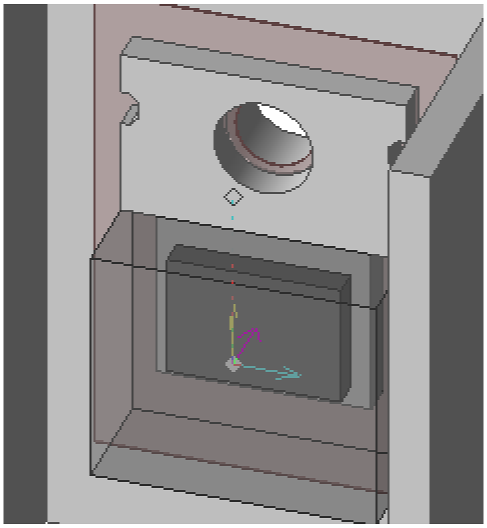
These line segments specify a path through the hottest area of the model (the transistor), through the insulator, and continuing through the heat sink to the coolest area (furthest from the transistor).

1. On the **Draw** ribbon tab, click **Orient** > **Trimetric** for a better view of the front of the transistor.
2. On the **Draw** ribbon tab, click **Draw line**.
3. Press **Tab** to jump to the **X** coordinate text box and specify **0**.

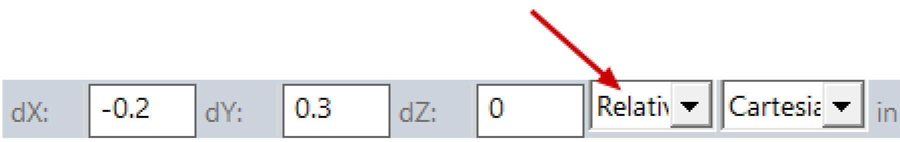
> **Note:**
>
> Be careful not to bump your mouse while entering coordinates in these text boxes. If you do, the cursor location will replace the numerically specified values, and you will have to enter them again.
>
> Additionally, do not switch the active application window (such as from AEDT, to your web browser, and back to AEDT). Doing so will reset the base point used as the reference for relative coordinates, which you will specify in later steps.

4. Press **Tab** again and specify **-0.283** in the **Y** text box.
5. Press **Tab** again, specify **-0.27** in the **Z** text box, and press **Enter** to complete the first segment.

   A dot and miniaxes should appear on the front face of the transistor body centered over the die:



6. Move the cursor slightly, and the coordinate entry method for subsequent points should automatically change to the **Relative** mode, as indicated in the status bar:



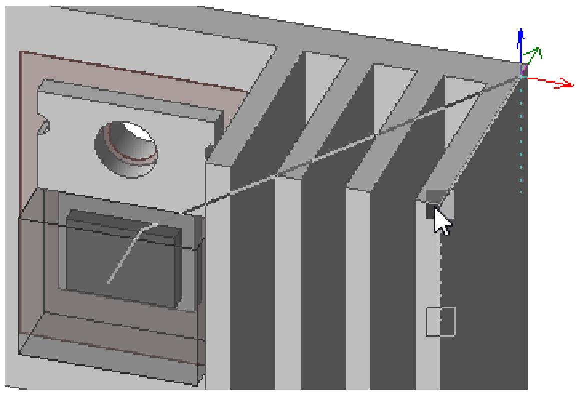
If the **Absolute** mode is still selected, manually select **Relative** from the drop-down menu.

7. In the same manner as for the first point, **Tab** through the coordinate entry text boxes and specify the following relative coordinates for the first segment's endpoint. Then press **Enter**:
   - **dX** = **0**
   - **dY** = **0.233**
   - **dZ** = **0**

   A segment's endpoint is also the start point for the next segment.

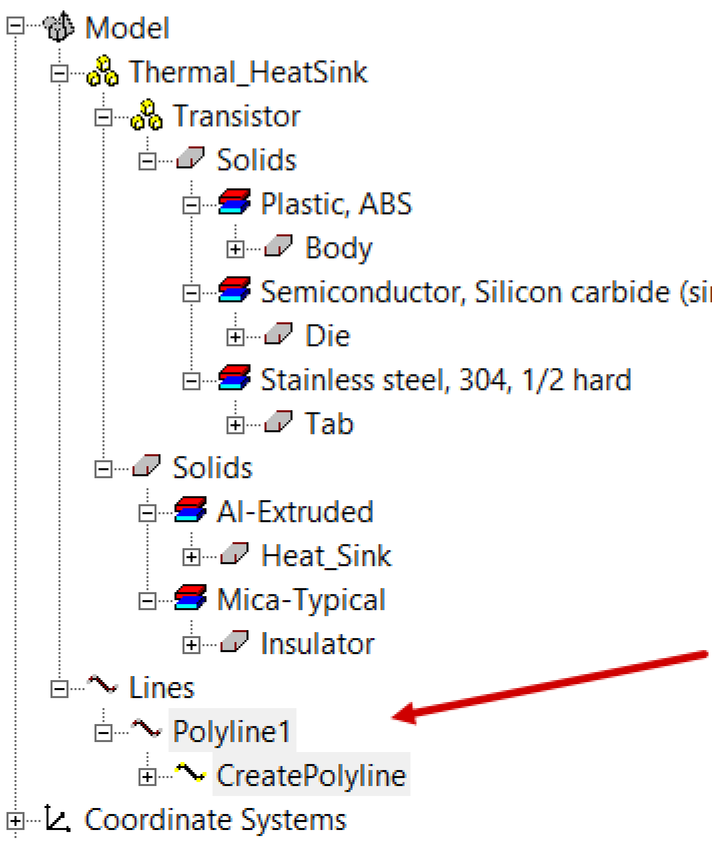
8. Enter the following coordinates for the second segment's endpoint and press **Enter**:
   - **dX** = **0.87**
   - **dY** = **0**
   - **dZ** = **0.57**

9. For the final segment's endpoint, graphically snap to the top-right-front corner of the heat sink:



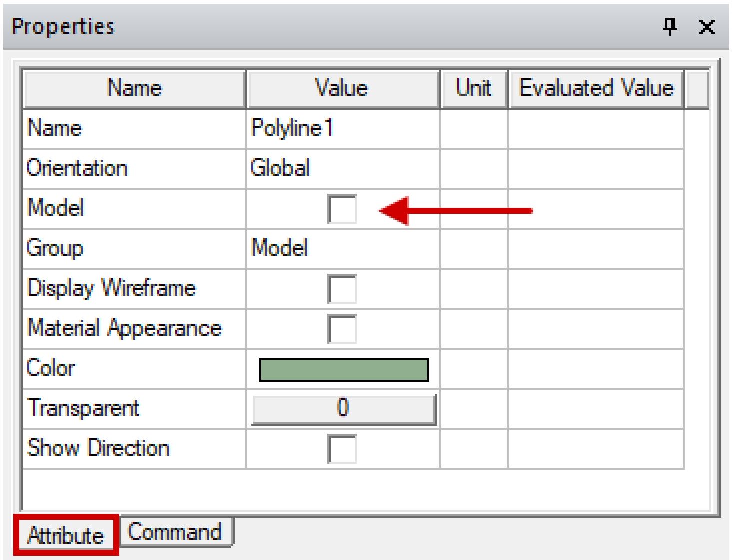
10. Right-click in the Modeler window and choose **Done** to terminate the *Line* command.

    Polyline1 is added under *Model* > *Lines* in the History Tree.



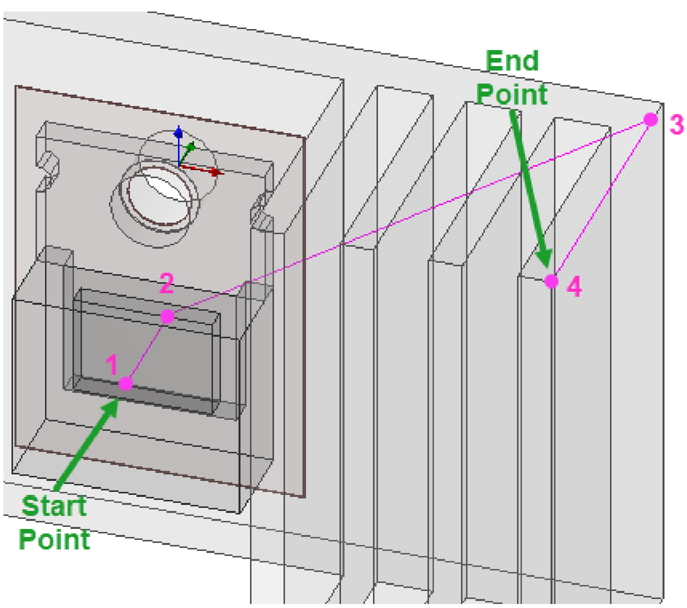
    Additionally, *Polyline1* and its associated *CreatePolyline* operation are currently selected, and the command properties and object attributes are displayed in the docked *Properties* window.

11. In the **Attribute** tab of the docked *Properties* window, clear the **Model** option:



The polyline is used only for post-processing and does not participate in the analysis.

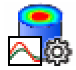
With Polyline1 still selected, your model should resemble the following image (point markers were added for clarity):



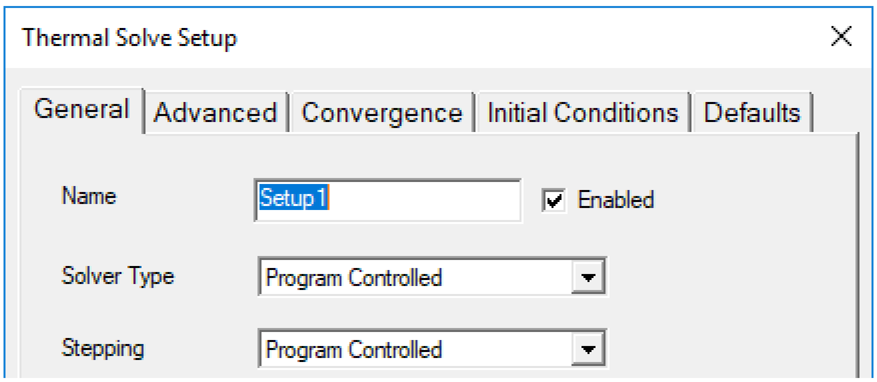
12. Clear the selection and **Save** your project.

10 - Set Up, Validate, and Analyze Model

Create an analysis setup using default settings, validate the design, and run the analysis, as follows:

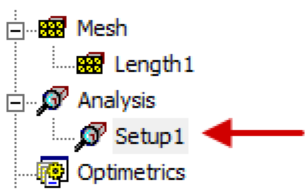
1. On the **Simulation** ribbon tab, click  **Setup**.

   The *Thermal Solve Setup* dialog box appears.

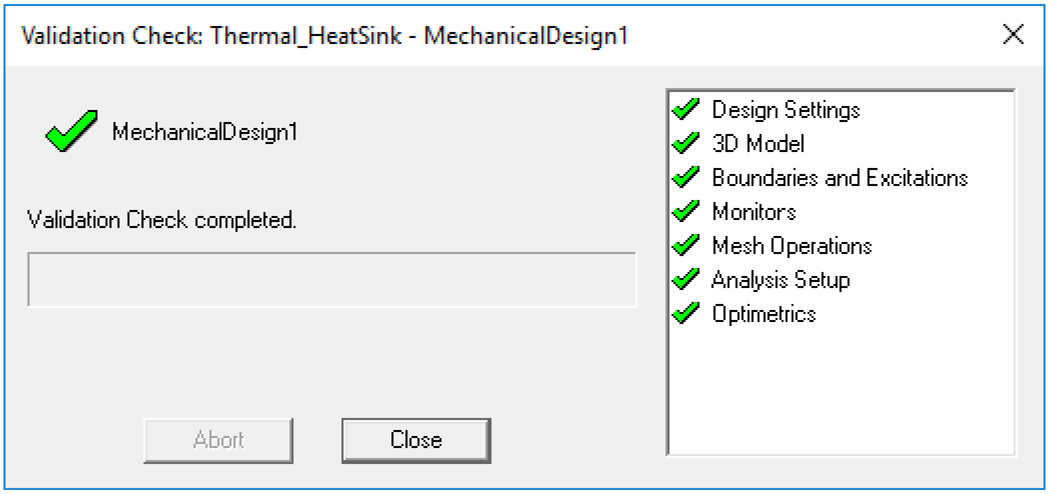
2. Click **OK** to accept the default setup.

   Setup1 appears under *Analysis* in the Project Manager:

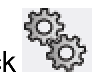
   

3. On the **Simulation** ribbon tab, click  **Validate**.

   The *Validation Check* window appears, and there should be no errors or warnings:



4. Click **Close** to dismiss the *Validation Check* window.

5. On the **Simulation** ribbon tab, click  **Analyze All**.

   The solution should take about two to three minutes to complete on a current, high-end computer workstation.

11 - Evaluate Results

In this final section, you will generate a mesh overlay, a 2D plot of the temperatures along the previously defined polyline, and overlays of the temperature magnitude and heat flux vectors.

The following three topics are covered in this section:

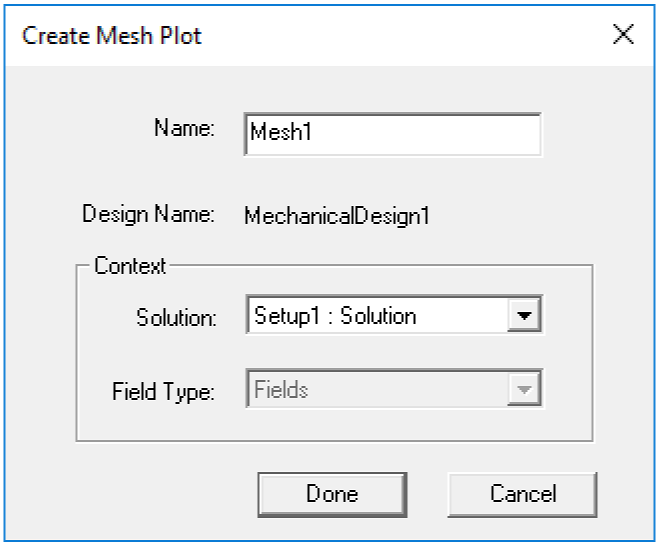
- **Mesh Overlay:** Overlay the generated elements on the geometry in the Modeler window.
- **Temperature vs. Distance Plot:** Create a 2D plot of the temperature versus distance along *Polyline1*.
- **Temperature Overlay**: Overlay the temperature results on the geometry in the Modeler window.
- **Heat Flux Vector Overlay:** Overlay the heat flux vector results on the geometry in the Modeler window.
- **Fields Summary:** Create a summary of the Heat Flow Rate results at the convection boundaries.

Mesh Overlay

Overlay a mesh plot on all five solid parts of the model, as follows:

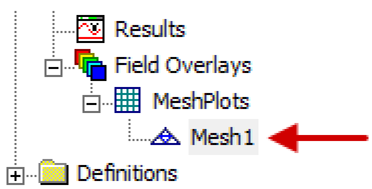
1. In *Object* selection mode, click in the Modeler window and press **Ctrl+A** to select all objects.
2. Under *Model* > *Lines* in the History Tree, **Ctrl+click** on **Polyline1** to deselect it while leaving all solid objects selected.
3. Right-click **Field Overlays** in the Project manager and choose **Plot Mesh** from the shortcut menu.

   The *Create Mesh Plot* dialog box appears:

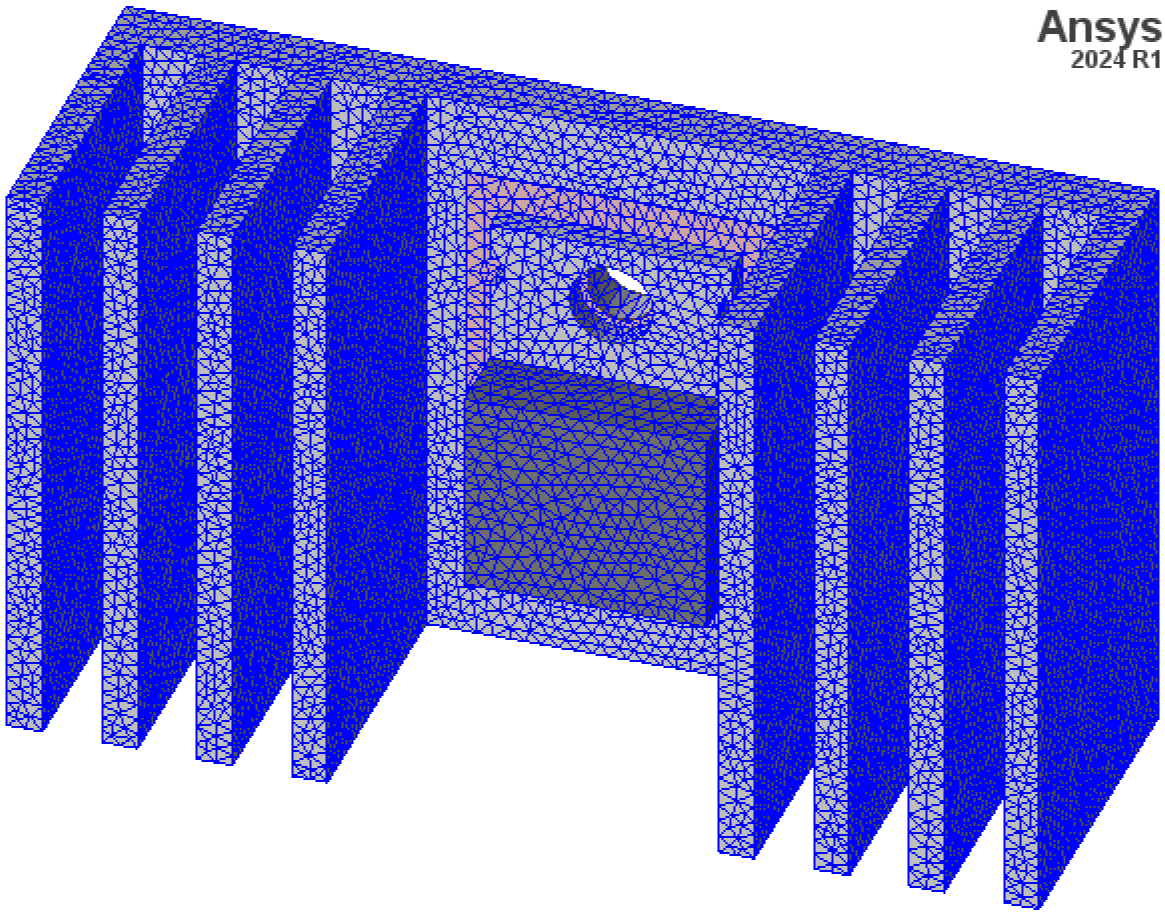


4. Click **Done** to accept the default plot setup and generate the mesh overlay.

   Mesh1 appears under *Field Overlays* > *MeshPlots* in the Project Manager:



   Your model should now resemble the following image:

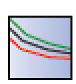
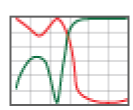


After reviewing the mesh, hide it so that it won't obscure temperature and heat flux overlays you will create later in this exercise.

5. Under *Field Overlays > MeshPlots* in the Project Manager, right-click **Mesh1** and clear the **Plot Visibility** option.

Temperature vs. Distance Plot

Plot the temperatures along *Polyline1* (from the top of the transistor, through the contact resistance disk and heat sink base, to the top of an outermost fin).

1. On the **Results** ribbon tab, click  **Fields Report** >  **2D**. Then, in the *Report* dialog box that appears, do the following:
   a. Select **Polyline1** from the **Geometry** drop-down menu.
   b. In the **Points** text box, type **9116**.

Note:

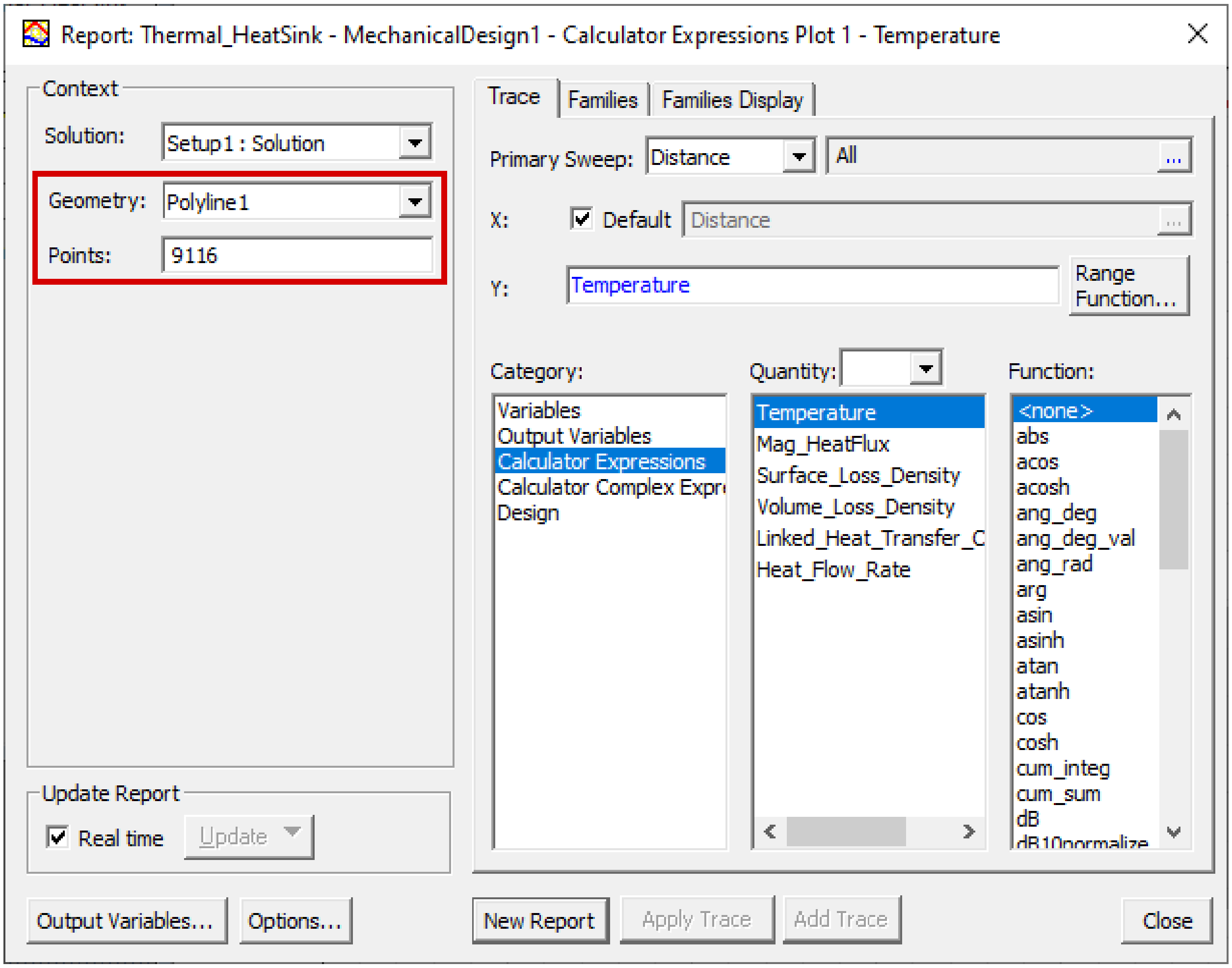
Polyline1 has a total length of 1.823 inches. Because the insulator is very thin (0.003") and, for the purpose of capturing temperatures very close to contact faces, you are specifying a data point resolution of 0.0002" (1.823 / 0.0002 + 1 = 9116 points).

Temperatures captured exactly on contact faces are ambiguous because there are actually two coplanar faces with differing temperatures. Therefore, to capture the abrupt temperature change across a contact area, we will look at the results at +/-0.0002 inches from the distances along the polyline that are associated with the contact faces. One contact face is at Distance = 0.180", and the other is at Distance = 0.183". Therefore, we will place markers at the following Distance values (all in inches):

- 0.1798
- 0.1802
- 0.1828
- 0.1832

Additionally, we will mark data points where the transistor body and die meet and where the die and mounting tab meet. Thermal resistance is not being considered at these object interfaces. Therefore, there will be no temperature discontinuities at these faces.

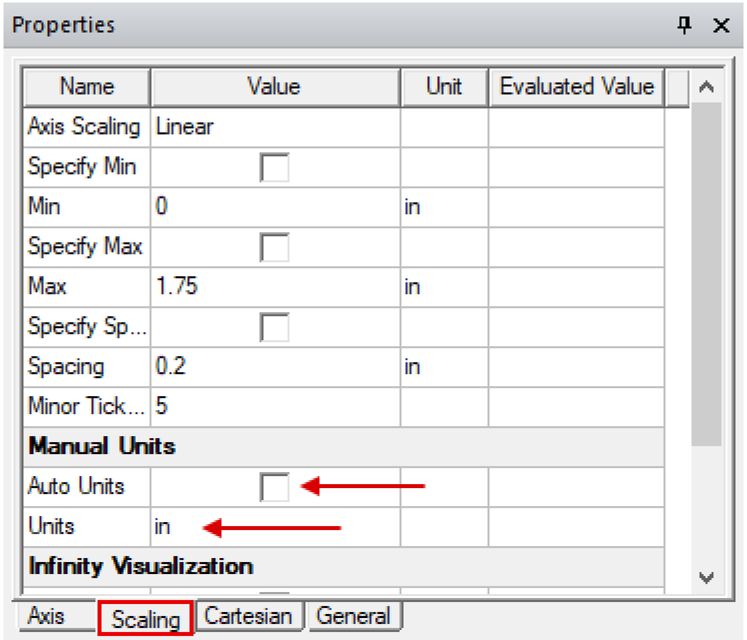
c. Ensure that the remaining settings are as shown in the following image:



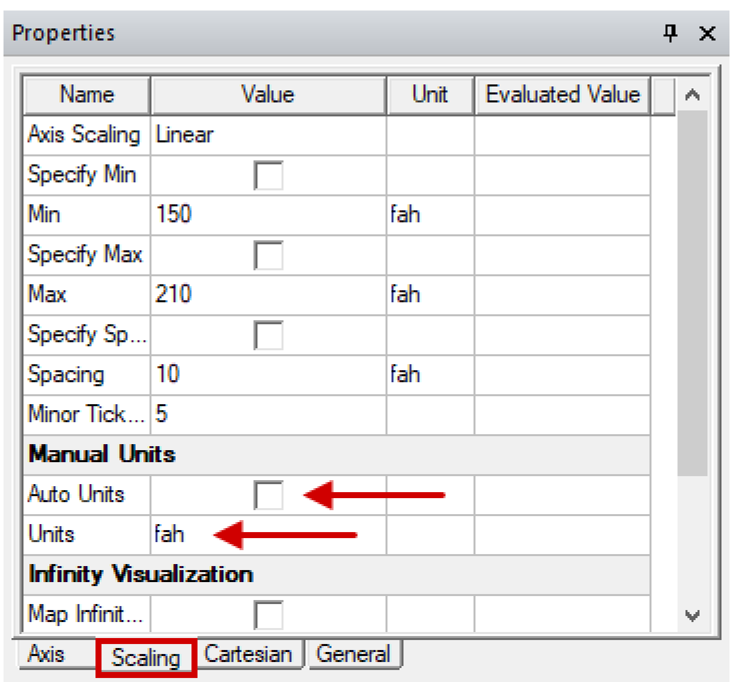
d. Click **New Report** and then click **Close**.

Calculator Expressions Plot 1 appears in a new window.

2. In the *plot window*, click the X axis label, **Distance [mm]**, to access the associated properties.
3. In the **Scaling** tab of the docked *Properties* window, make the following changes:
   a. Clear the **Auto Units** option.
   b. Choose **in** (inches) as the **Units** value.

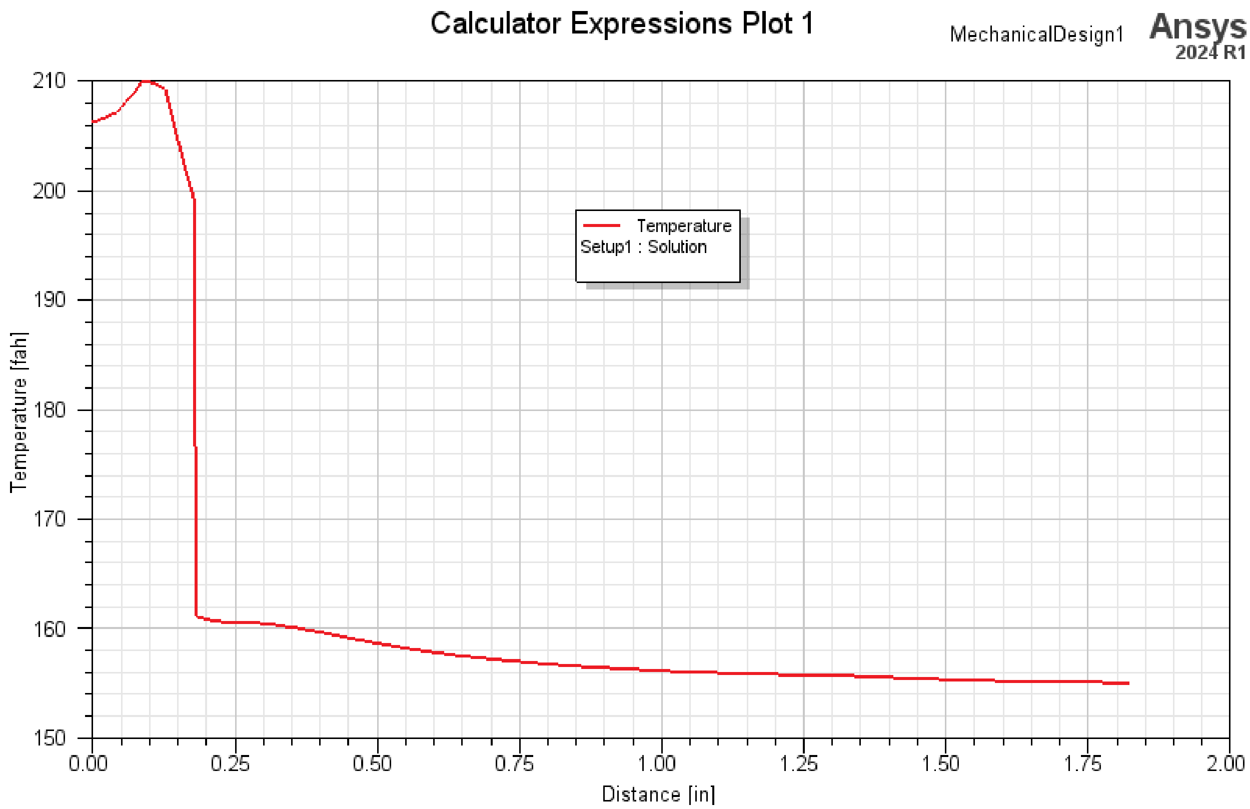


4. In the *plot window*, click the Y axis label, **Temperature [cel]**, to access the associated properties.
5. In the **Scaling** tab of the docked *Properties* window, make the following changes:
    a. Clear the **Auto Units** option.
    b. Choose **fah** (°F) as the **Units** value.



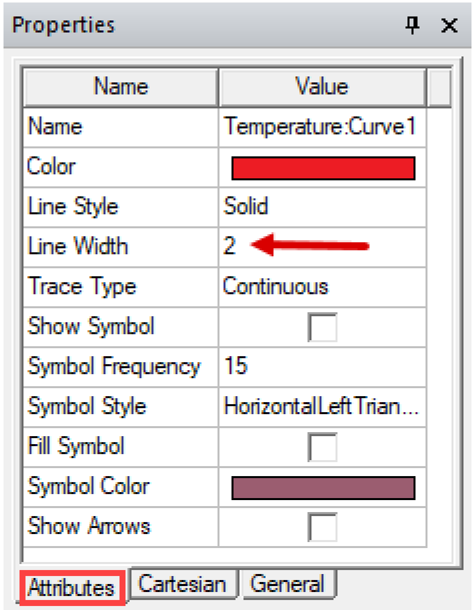
6. Click in the background area of the plot to clear the selection.

Your plot should now look like the following image:



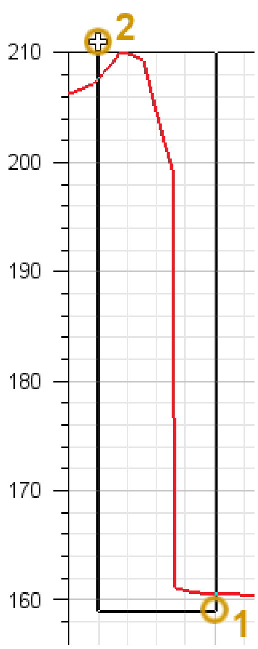
The steepest temperature gradient occurs as a result of thermal resistance at the insulator contact faces and limited thermal conductivity through the thickness of the mica insulator. Also, the temperature continues to trend downward as the distance from the transistor (the source of the heat) increases. These results are expected.

7. Click on the red trace to select it. Then, in the **Attributes** tab of the docked *Properties* window, do the following:
   a. For the **Line Width** *Value*, type **2**.
   b. Press **Enter**.



In the following steps, you will zoom into the area of interest of the plot and add several markers. A thinner trace facilitates more precise placement of the markers.

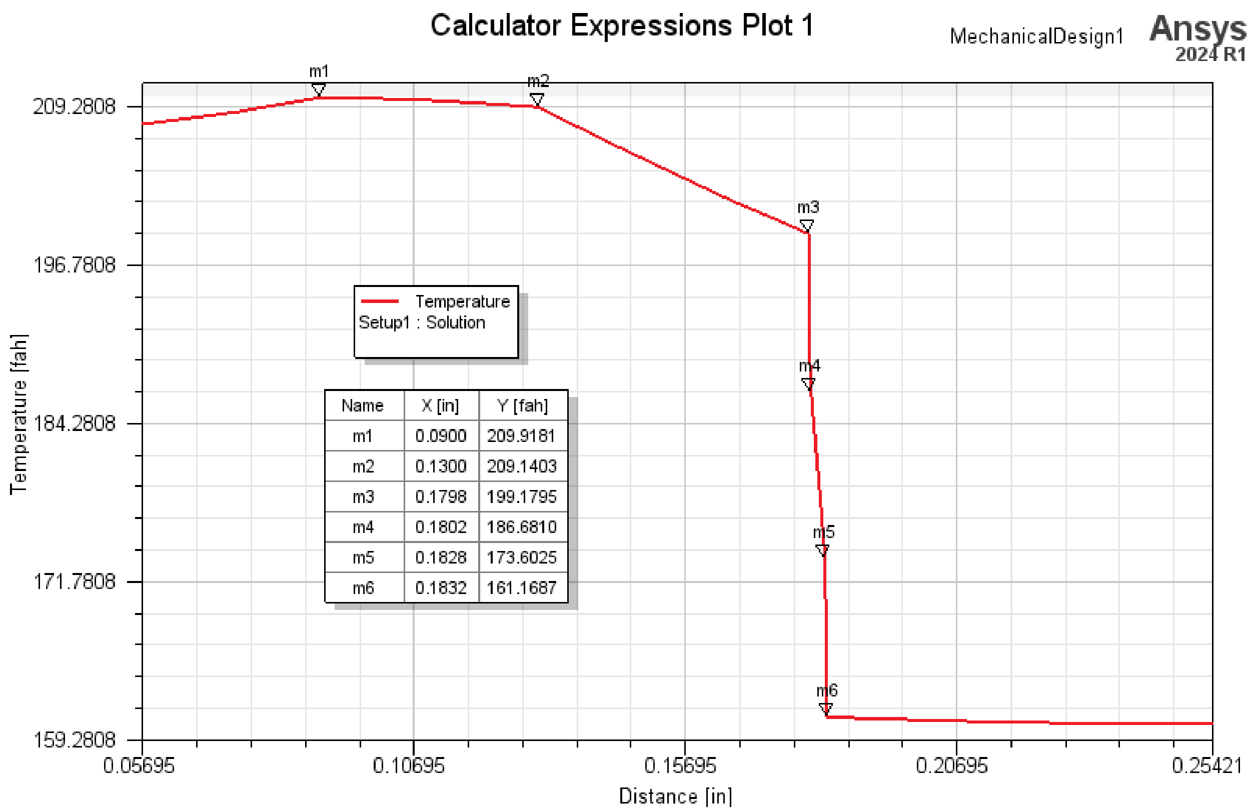
8. With the 2D Plot the active window, on the **View** ribbon tab, click **Zoom Area**. Then:
   a. Using the following image as a guide, click at point "**1**" and drag to point "**2**" to define a zoom area tightly enclosing the steepest portion of the curve:



   b. After releasing the mouse button, press **Esc** to terminate the *Zoom Area* mode.

9. Right-click in the plot window and choose **Marker** > **Add Marker** from the shortcut menu. Then:
   a. Using the plot image below as a guide, click to place markers at the six points shown (*m1* through *m6*).

      Do not be concerned if an X location differs slightly from the value shown in the image. You can adjust the exact distances for the markers as needed using the instructions in the note below the image.
   b. Press **Esc** to terminate the *Add Marker* command.



Name	X [in]	Y [fah]
m1	0.0900	209.9181
m2	0.1300	209.1403
m3	0.1798	199.1795
m4	0.1802	186.6810
m5	0.1828	173.6025
m6	0.1832	161.1687

> **Note:**
>
> If you accidentally place a marker at a different X distance than intended, you can drag the marker to a new location along the curve, after terminating the *Add Marker* command. Additionally, you can select a marker in the table and edit its **Distance** value in the **Marker** tab of the docked *Properties* window.
>
> Because the temperature curve is so steep between markers **m3** and **m6** (almost vertical), a very slight change in the X value (even in the fifth decimal place) significantly affects the Y value. The docked *Properties* window shows the distance with greater precision than the default marker table precision of 4.

The transistor temperature is significantly hotter than it would be if you were to assume perfect contact (no thermal resistance) and exclude the insulator. You can see the importance of considering these details. The combined effects of the two contact faces and the insulator raise the transistor's temperature about 37.1 °F (or 20.6 °C). The observations listed below detail the significance of the six markers you added to the plot.

Observations:

- Markers **m1** and **m2** correspond to the front and back faces of the die. Prior to **m1**, there is a slight temperature increase as the die is approached. This gradient is due to the small amount of die heat dissipated by conduction through the plastic body to the heat sink. The temperature is fairly flat through the die thickness, which is not surprising since it is the heat source, and it has a high thermal conductivity relative to the adjacent objects. The side of the die facing towards the heat sink is slightly cooler than the front side.
- Between markers **m2** and **m3**, the temperatures correspond to the stainless steel tab. Stainless steel is a poorer heat conductor than aluminum, and the cross-section through which heat is flowing is relatively small. Therefore, the temperature gradient here is steeper than at the portion of the plot associated with the heat sink (X > 0.183").
- The contact face between the transistor and insulator occurs at X = 0.180". The temperature differential between the nearest data points on either side of this contact face (**m3** and **m4**) is approximately 12.5 °F due to thermal contact resistance.
- Between markers **m4** and **m5**, the temperatures correspond to the mica insulator. Due to the material's low thermal conductivity, the temperature gradient is steep through this very thin part, though not as steep as the gradients at the contact faces.
- The contact face between the insulator and heat sink occurs at X = 0.183". The temperature differential between the nearest data points on either side of this contact face (**m5** and **m6**) is approximately 12.4 °F due to thermal contact resistance. This result closely matches the temperature differential between markers *m3* and *m4*, which is expected (since the thermal resistance and total heat flow rate are the same), though the areas differ between the two contact faces. However, the effective contact area on the back side is about the same because in-plane heat flow in the insulator should be minimal. That is, the path of least resistance is normal to the insulator's contact faces, through the 0.003 inch thickness.

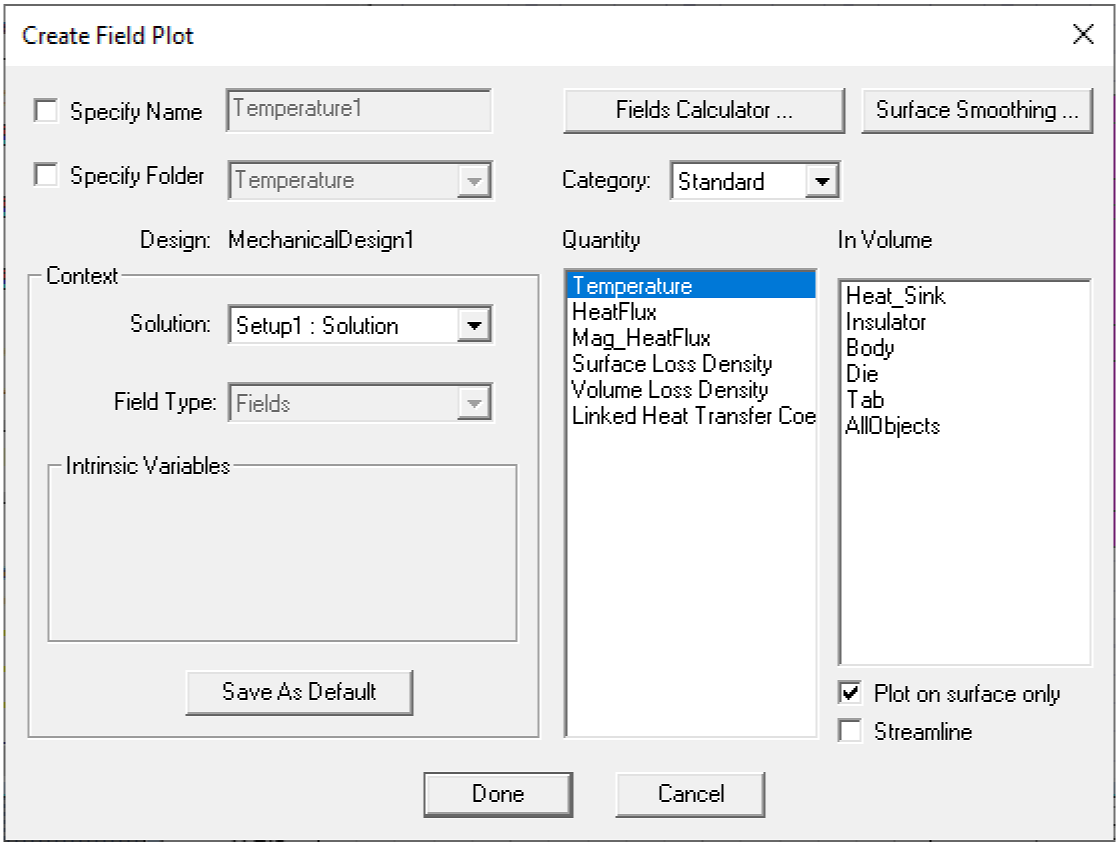
Temperature Overlay

Create an overlay of the temperature results:

1. From the menu bar, click **Window** > **1 Thermal_Heatsink - MechanicalDesign1 - Modeler** to make the Modeler window the active window and to bring it to the foreground.
2. Under *Field Overlays* > *MeshPlots* in the Project Manager, right-click **Mesh1** and choose **Select Assignment** from the shortcut menu.

This action is a convenient way to reselect all solid objects and exclude the polyline, since this selection set was the basis of the mesh overlay.

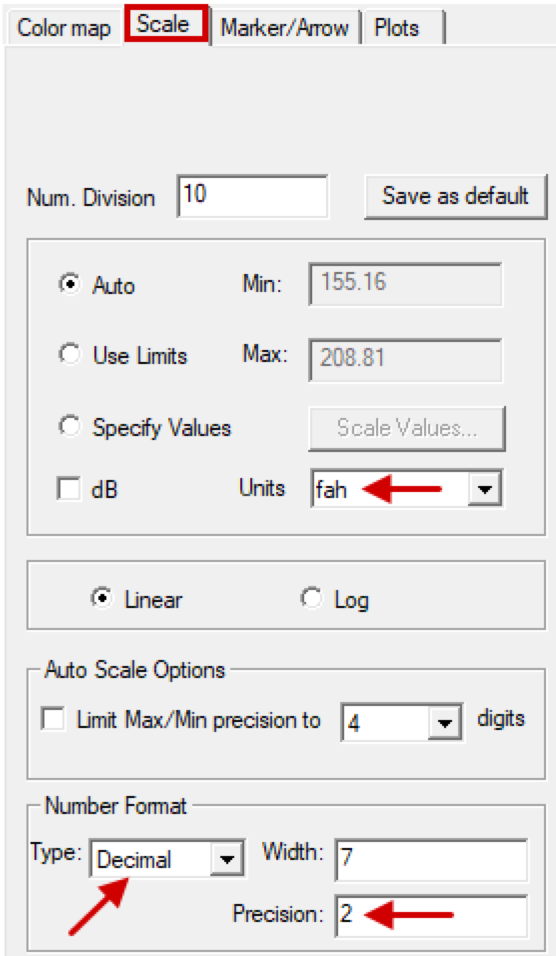
3. Right-click **Field Overlays** and choose **Plot Fields** > **Temperature**. Then, in the *Create Field Plot* dialog box that appears, do the following:
   a. Verify that the settings match the following image:



   b. Click **Done**.

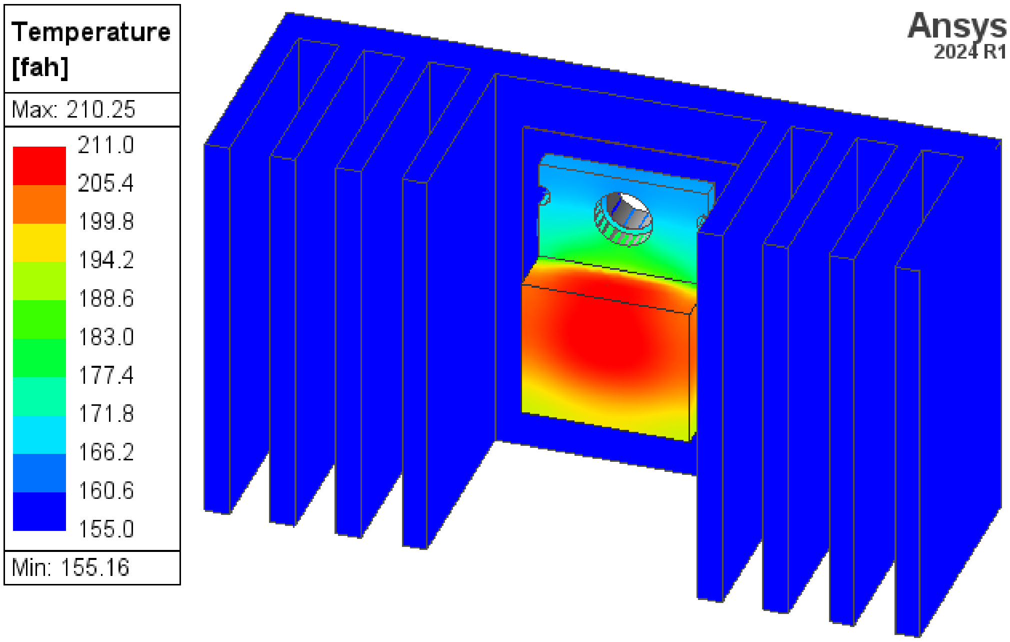
      The color contour overlay and temperature legend appear in the Modeler window.

4. Double-click inside the plot legend to access the plot settings. Then, make the following changes in the *Temperature* dialog box that appears.
   a. In the **Scale** tab, choose **fah** (°F) from the **Units** drop-down menu.
   b. In the *Number Format* section, choose **Decimal** from the **Type** drop-down menu.
   c. Change the **Precision** value to **2**.



d. Click **Apply** and then click **Close**.

Your temperature overlay should now resemble the following image:



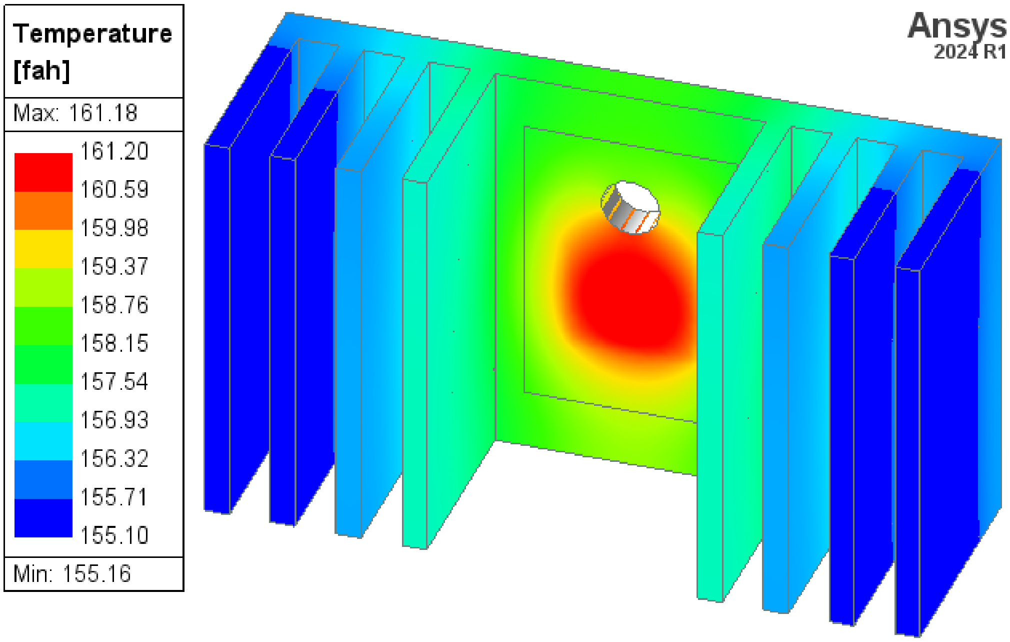
Because of the large temperature gradient across the contact faces and mica insulator the heat sink and insulator are all dark blue. The transistor is overlaid with the remaining colors of the spectrum. To see a full spectrum of color on each of these objects, complete the following optional procedure:

5. Select the **HeatSink** object.
6. On the **Draw** ribbon tab, click **Show only selected objects in active view**.

   The transistor and insulator are still visible because they are specified as the selection for the temperature overlay.

7. Once again, select the **HeatSink** object.
8. Under **Field Overlays** > **Temperature** in the Project Manager, right-click **Temperature1** and choose **Reassign**.

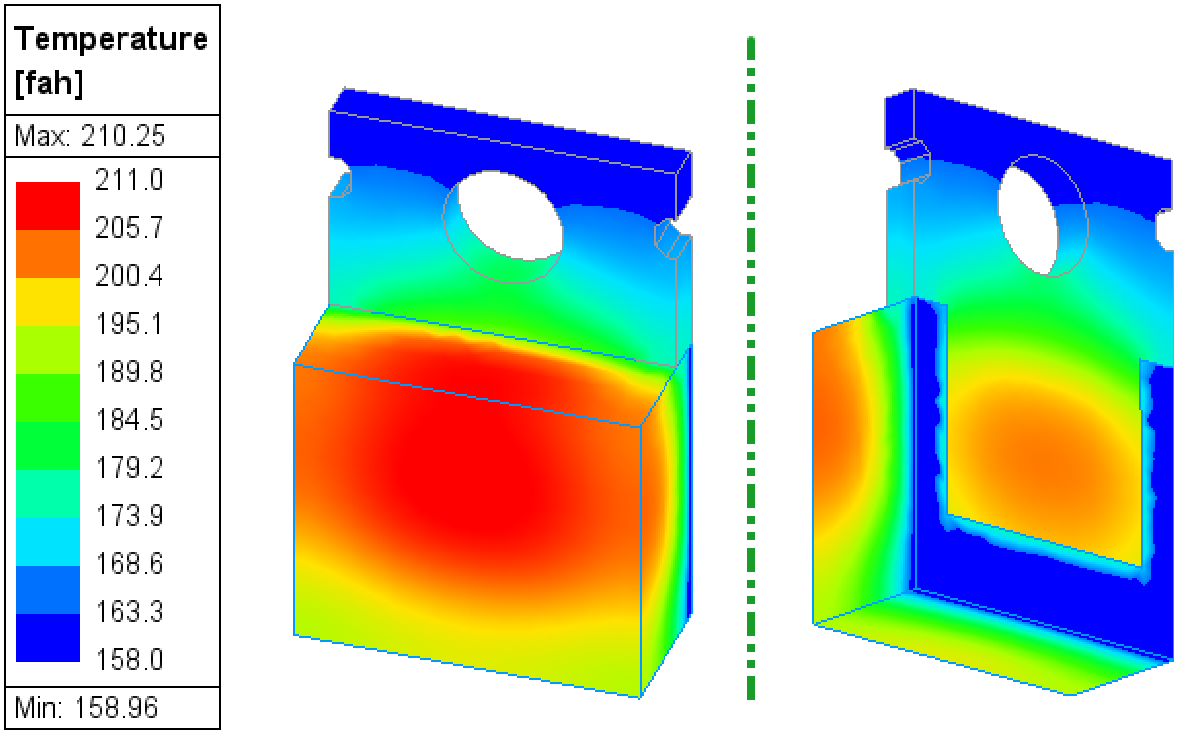
   The overlay is updated and the legend adjusted to show only the range of temperatures within the heat sink, as shown in the following image:



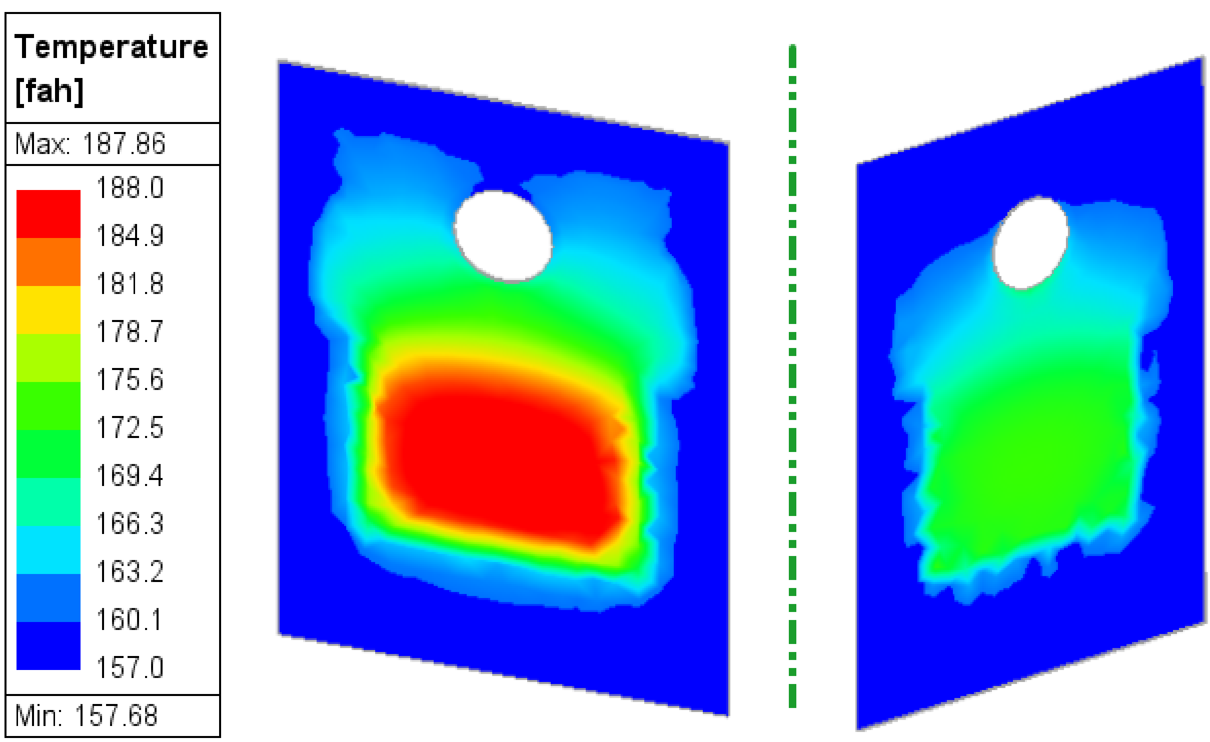
9. Using the same method as described in steps 5 through 8, change the object visibility and temperature overlay again, but this time select only the **Transistor**. To select the three objects comprising the transistor in a single operation: Under *Model* > *Thermal_HeatSink* in the History Tree, right-click **Transistor** and choose **Select All**.

   Also, optionally look at the temperature distribution in the Insulator alone.

   The revised overlay showing transistor contours should look like the following image (rotated to see the back face):



The revised overlay showing insulator contours should look like the following images (views of front and back shown):



10. On the **Draw** ribbon tab, click **Show all objects in active view** to restore the visibility of all model objects and **Orient** to restore the default *Isometric* view.
11. Optionally, you can select all solid objects and reassign them to the **Temperature1** overlay, to restore color contours to all objects.

Note:

Rather than reassigning the temperature overlay to different objects and selectively hiding objects, you could alternatively create a Fields Summary. You can set up the summary to list the minimum, maximum, and mean temperature results individually for each object in the model. The preceding method was chosen because of the visualization of the range of temperatures as a full color spectrum that it provides for each component.

In the final procedure in this guide, you will create a fields summary of heat flow rates for the assigned heat generation excitation and convection boundaries.

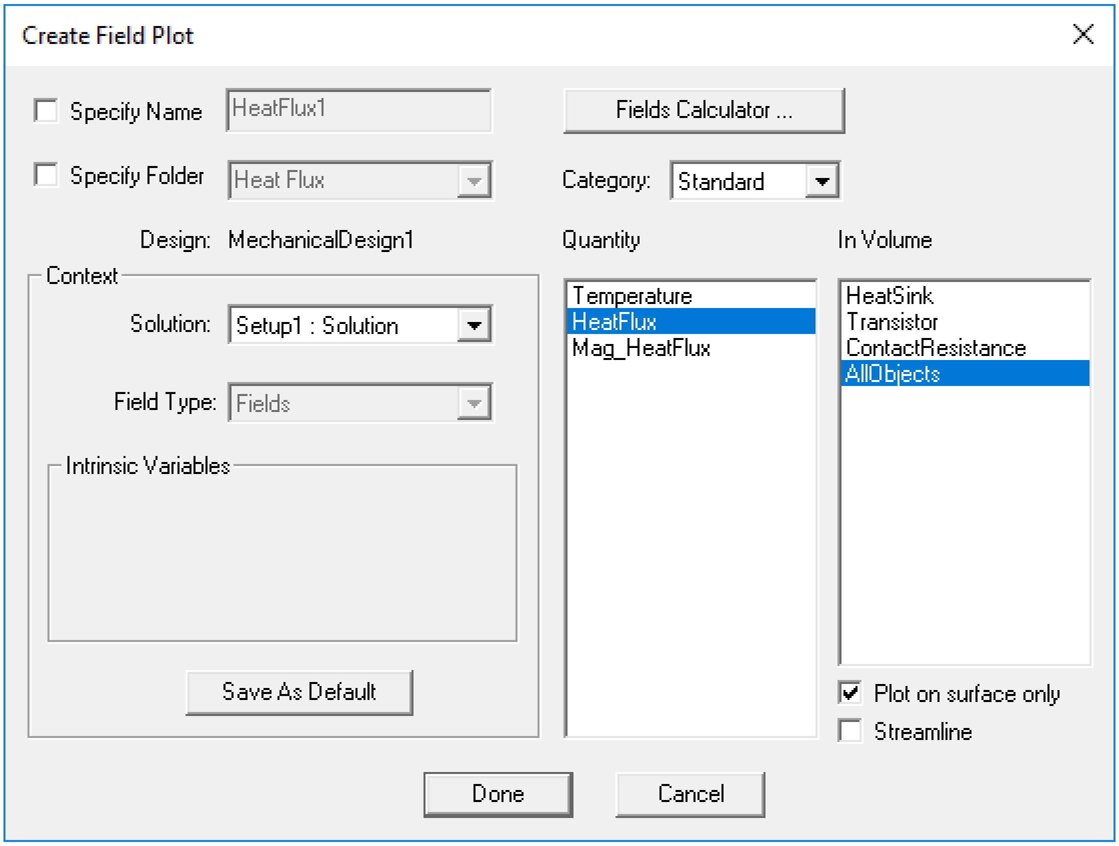
Heat Flux Vector Overlay

Finally, create a heat flux vector overlay. But, before doing so, turn off the visibility of the temperature overlay.

1. Under *Field Overlays* > *Temperature* in the Project Manager, right-click **Temperature1** and clear the **Plot Visibility** option in the shortcut menu.
2. Under *Field Overlays* > *MeshPlots* in the Project Manager, right-click **Mesh1** and choose **Select Assignment**.
3. Right-click **Field Overlays** in the Project manager and choose **Plot Fields** > **Heat Flux** > **HeatFlux** from the shortcut menu.

   In the *Create Field Plot* dialog box that appears:

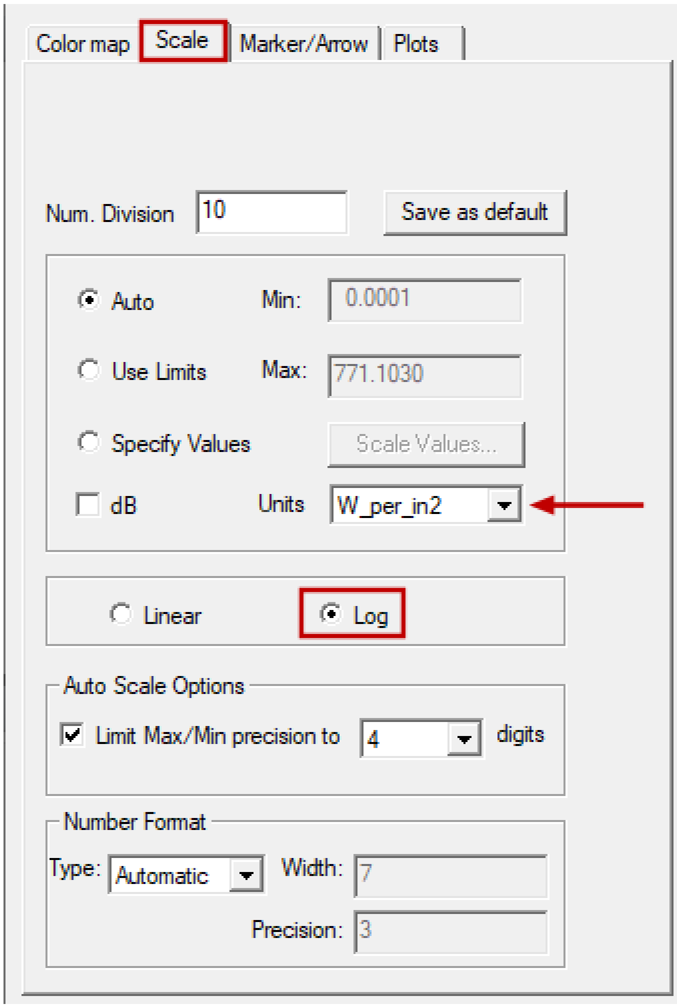
a. Ensure that the settings match the following figure:



b. Click **Done** to accept the settings and generate the vector overlay.

A heat flux vector overlay and legend appear in the Modeler window.

4. Double-click inside the legend to access the plot settings.
5. Under the **Scale** tab of the *Heat Flux* dialog box that appears, make the following changes:
   a. Choose **W_per_in2** (W/in^2) from the **Units** drop-down menu.
   b. Select the **Log** option.

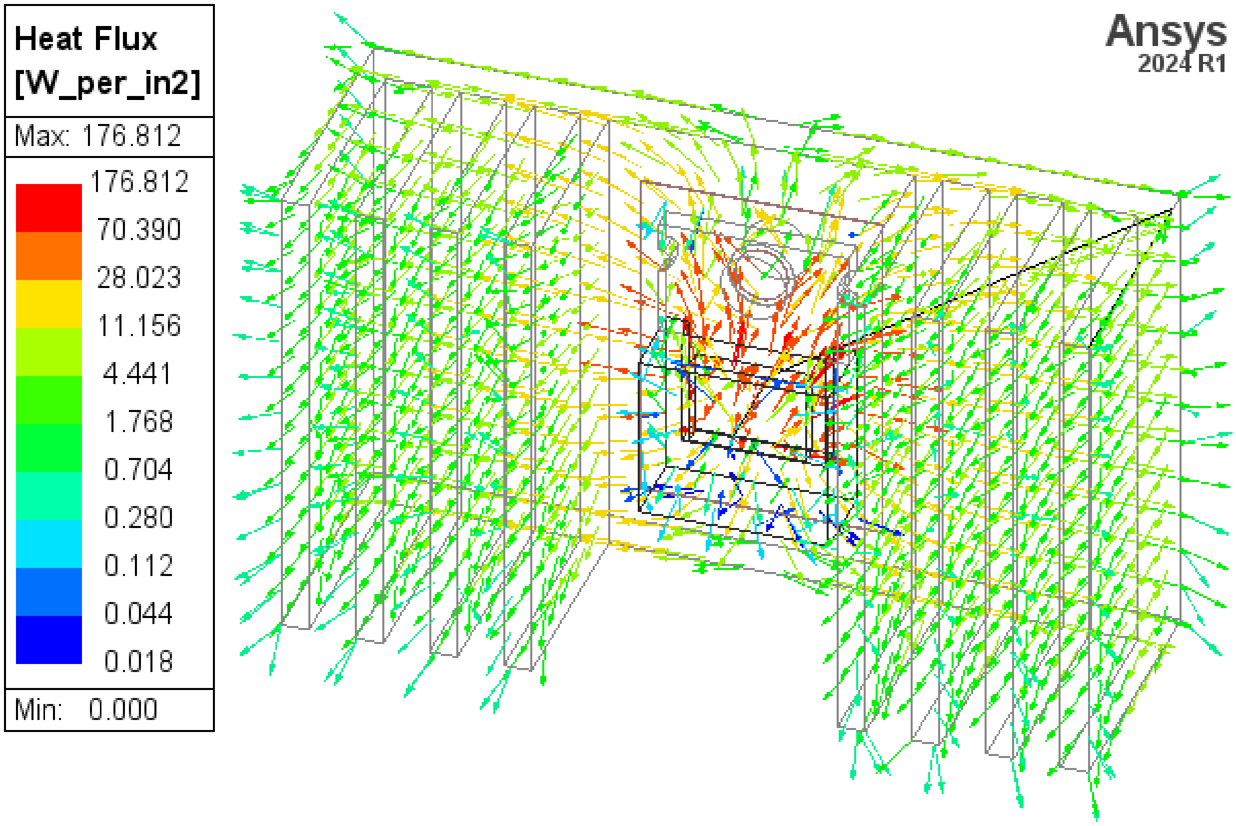


The logarithmic scale is better suited for the wide range of heat flux values than the default linear scale.

c. Click **Apply** and then click **Close**.

6. Press **F6** to display the model as a wireframe, suppressing face shading for a cleaner vector display.

   Your model overlay should now resemble the following image:



7. Optionally, press **F7** to restore the shaded model view.

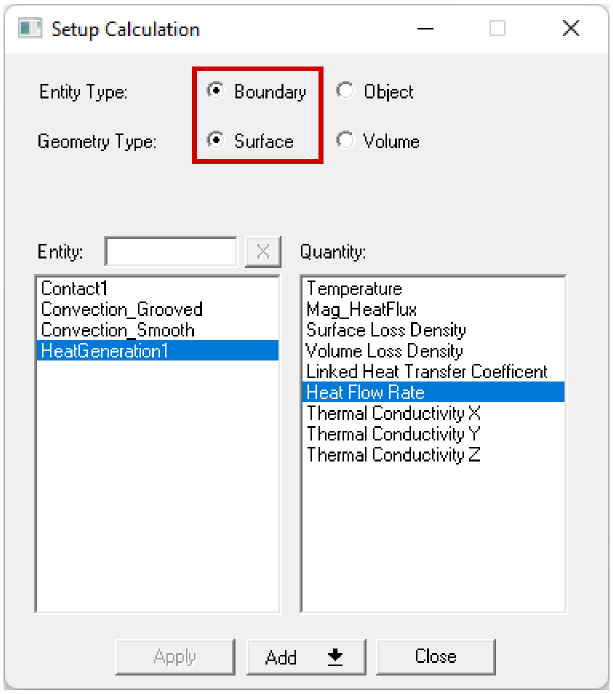
Create Fields Summary

Finally, create a fields summary to report the total heat flow rates for the heat generation excitation and convection boundaries. You will report the individual totals for the smooth and grooved surface convection boundaries as well as the combined total for both convection boundaries.

1. Right-click **Field Overlays** in the Project manager and choose **Create Fields Summary** from the shortcut menu.

   Two dialog boxes appear: *Fields Summary* and *Setup Calculation*.

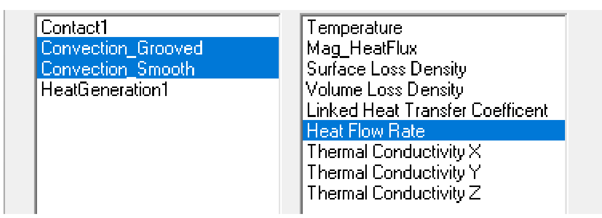
2. In the *Setup Calculation* dialog box, do the following:
   a. Ensure that the **Entity Type** and **Geometry Type** selections are **Boundary** and **Surface**, respectively.
   b. In the **Entity** list, select **Heat Generation**.
   c. In the **Quantity** list, select **Heat Flow Rate**.



d. Click **Add** and select **Add as Single Calculation** from the drop-down menu.

A row showing the total heat generation appears in the *Calculations* table of the *Fields Summary* dialog box.

e. In the **Quantity** list, deselct **Heat Flow Rate** and select **Convection_Grooved** and **Convection_Smooth**.

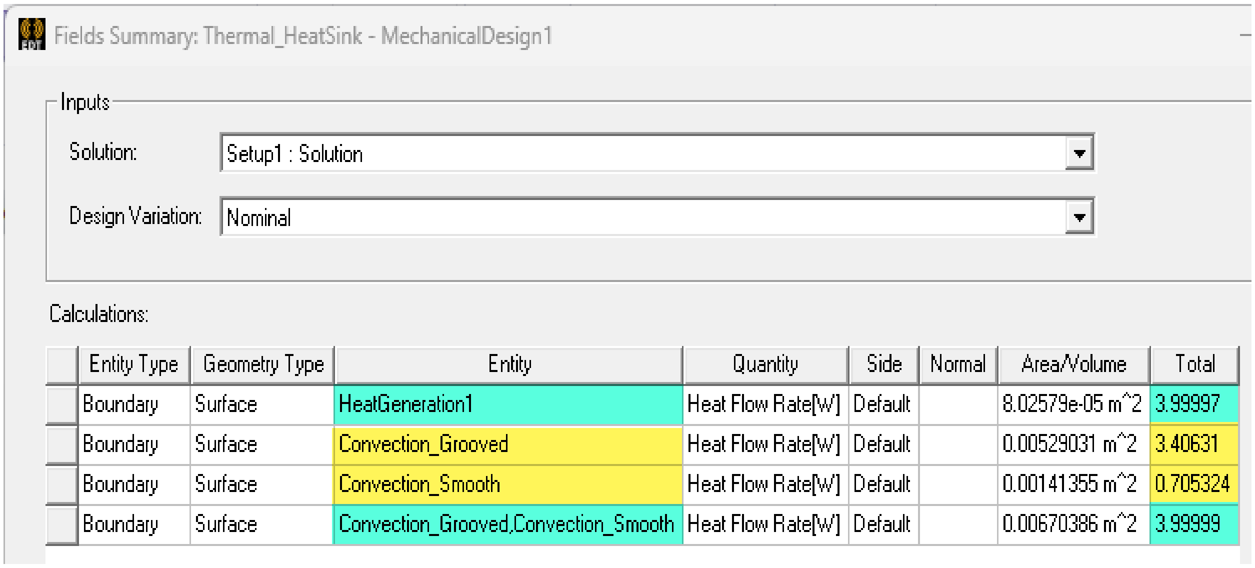


f. Click **Add** and select **Add as Multiple Calculation** from the drop-down menu.

Two rows appear in the *Calculations* table of the *Fields Summary* dialog box, one for each convection boundary.

g. Click **Add** again but, this time, select **Add as Single Calculation**.

A fourth row appears in the *Calculations* table showing the total heat flow rate for the two convection boundaries combined:



	Entity Type	Geometry Type	Entity	Quantity	Side	Normal	Area/Volume	Total
	Boundary	Surface	HeatGeneration1	Heat Flow Rate[W]	Default		8.02579e-05 m^2	3.99997
	Boundary	Surface	Convection_Grooved	Heat Flow Rate[W]	Default		0.00529031 m^2	3.40631
	Boundary	Surface	Convection_Smooth	Heat Flow Rate[W]	Default		0.00141355 m^2	0.705324
	Boundary	Surface	Convection_Grooved,Convection_Smooth	Heat Flow Rate[W]	Default		0.00670386 m^2	3.99999

Observations:

- The total heat generation (row 1 with blue-green highlighting) is virtually identical to the 4W heat generation excitation assigned to the transistor die.
- Manually sum the two separate total power values reported for the grooved and smooth convection boundaries (rows 2 and 3 with yellow highlighting). The sum is 4.1116W, which is approximately 2.8% (0.1116W) in excess of the total assigned heat generation of 4W. The cause of this slight power imbalance is reporting redundancy that occurs at edges shared by two adjacent convection boundaries. There are many such edges in this model.
- However, when the combined effect of both boundaries is reported as a single calculation in the fields summary, the redundancy does not occur (each shared edge's heat flow is only counted once). Therefore, the total heat flow reported in row 4 (with blue-green highlighting) matches the heat generation reported in row 1 to within an infinitesimal error margin.
- For the following observations, each individual convection boundary's total heat flow will be reduced by half of the excessive flow reported (that is, half of 0.1116W). The adjusted values, totaling 4W, are 3.3505W and 0.6495W, respectively, for the grooved and smooth faces:
    - The grooved convection faces represent 78.9% of the total convection surface area (100% * 0.00529031m^2 / 0.00670386m^2) and about 83.8% of the total convective heat flow rates of the two boundaries combined (100% * 3.3505W / 4W). By comparison, the smooth faces represent 21.1% of the total convection surface area and produce about 16.2% of the total convective heat flow. Does this result match expectations? To answer that question, we can estimate what the relative convective heat flow rates would be based on the surface areas and film coefficients of each boundary.
    - If all faces were assumed to be at the same temperature, the grooved faces (78.9% of the total area), with a film coefficient of 14.1 W/(m^2·°C), would account for 84.1% of the total convective heat flow. The smooth faces (21.1% of the total area), with a film coefficient of 10 W/(m^2·°C), would account for 15.9% of the total convective heat flow. The actual results are skewed slightly from these values, with the grooved faces accounting for slightly less of the total heat flow, and the

> smooth faces slightly more, as compared to the equal-temperature assumption. The explanation of the actual results is that the heat sink face closest to the transistor, and therefore the hottest face, is smooth. Additionally, the smooth faces at the gaps between each pair of fins are closer to the heat source than most of the grooved surface area and are therefore hotter. Higher temperatures drive higher convective heat flow rates for any film coefficient, accounting for the sligtly increased relative contribution of the smooth faces.

3. In the *Fields Summary* dialog box, click **OK** to close both dialog boxes.
4. **Save** your project.

Congratulations, you have completed the *Thermal Solution – Heat Sink* getting started guide.