Getting Started with Mechanical: Transient Thermal Solution – Power Resistor

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

Conventions Used in this Guide

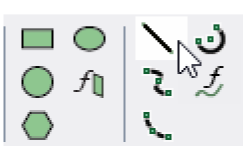
Please take a moment to review how instructions and other useful information are presented in this documentation.

- 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 set up and solve a *Transient Thermal* analysis of a *Mechanical* design in the *Ansys Electronics Desktop* application. You will learn how to use datasets to define time-varying parameters. Specifically, you will use datasets in piecewise constant (pwc) functions to vary heat generation, convection film coeffieient, and the rate at which field results are saved.

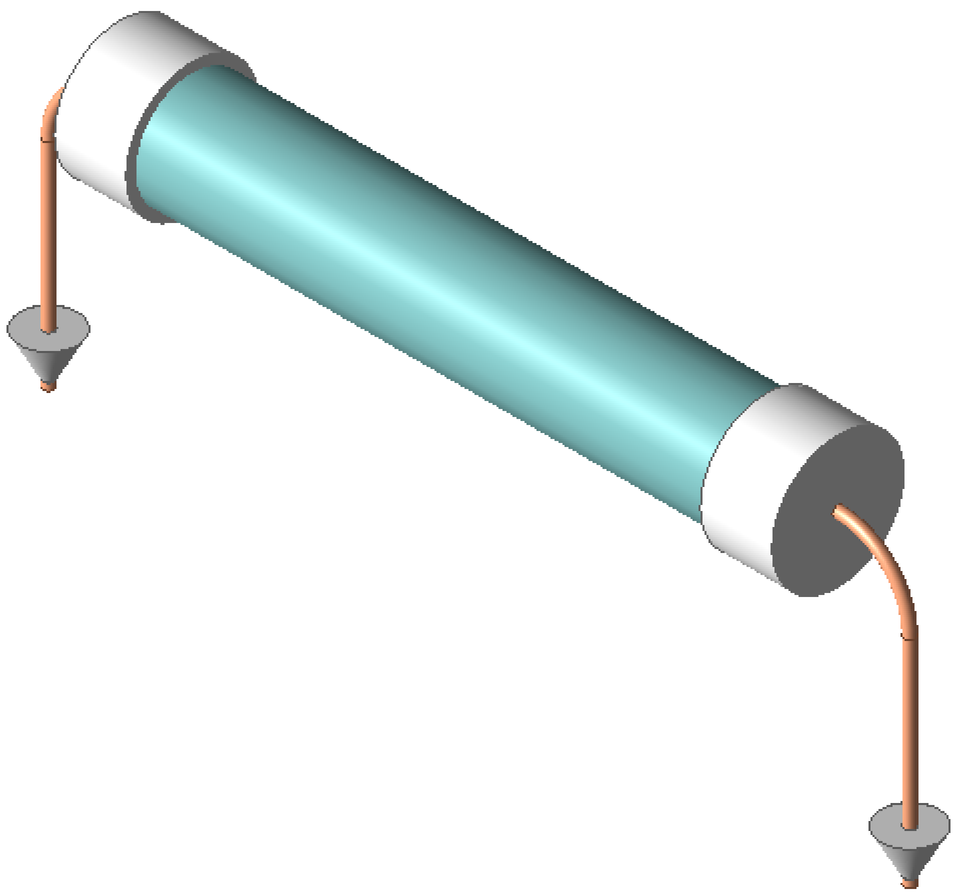
The focus is on transient thermal analysis and not solid modeling. Therefore, an Ansys Electronics Desktop project file has been provided as the starting point for the exercise.

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

- Open the source project and save it to working folder
- Set 3D UI options (this guide is based on the current view orientation scheme introduced in v2024 R1, not the legacy orientations)
- Modify the material properties (to reduce the transient event duration and solution time, you will decrease the specific heat of all materials by an order of magnitude)
- Assign convection boundaries and define a dataset to vary one of the boundaries with time
- Assign a heat generation excitation, including a dataset to vary it over time
- Define the ambient temperature
- Draw two points and assign each of them as a thermal monitor point
- Assign initial temperatures to selected objects
- Set up the solution, including a dataset to control the rate at which field results are saved
- Validate and solve the transient thermal analysis, monitoring the thermal results as the solution progresses
- Create and animate a temperature overlay
- Create a fields summary of the heat flow rate of the heat generation excitation and convection boundaries

The model consists of a power resistor that generates 4 watts of heat in low-power mode (from 0 to 4 seconds) and 7.5 watts in high-power mode (from 4 to 40 seconds). It is a slightly modified version of the model used in the *Structural Solution – Power Resistor* getting started guide. The resistor body is ceramic, the end caps nickel, the leads are copper, and a solder termination is included where the leads are connected to a printed circuit board (PCB), which is not included in the model. Heat generation occurs in the resistor body, and heat is dissipated via convection, which is assigned to all exposed faces except for two of them. This convection has a time-varying film coefficient. In low-power mode, the coefficient represents the slow-speed operation of the circuit's cooling fan. During high-power operation, the film coefficient is increased to represent high-speed operation of the cooling fan. The convection film coefficient increases by a lesser factor than the heat generation. Therefore, the resistor temperature will increase during high-power operation, though not as much as if the convection boundary parameters were to remain constant.

The horizontal face of each solder joint, which is the PCB contact face, has a separate boundary condition from the time-varying one. In the *Structural Solution – Power Resistor* getting started guide, it was assumed that 0.5 watts of heat would conduct from the power resistor to the circuit board at each solder joint. This effect was achieved using an assigned heat flux boundary, and the setup resulted in the solder face being approximately 20° C above ambient temperature. For this tranisent solution, we will assign a roughly equivalent convection boundary to produce the same result. That is, based on the surface area of the solder-to-PCB contact faces, 0.5 W of assumed heat flow per joint, and Delta T = 20° C, we will calculate the convection film coefficient that produces these results. This method will be used as an alternative to assigning a time-varying heat flux boundary at the solder-to-PCB contact faces.



2 - Set Up the Project

In this chapter, you will perform the following tasks:

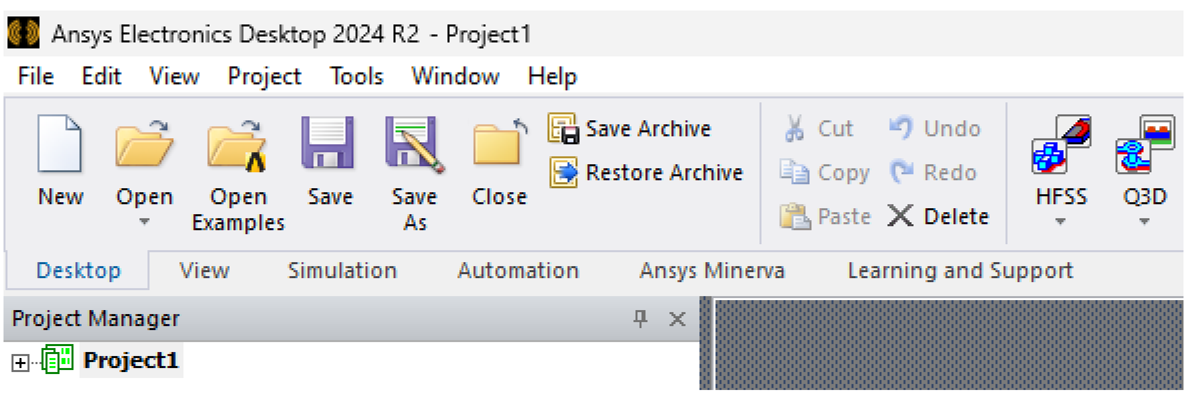
- Launch the Ansys Electronics Desktop application and close any open or new project
- Open the source project file and save it to a different working folder
- Set 3D UI options

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 ensure that no project is open, as follows:

1. If EDT is not already running, double-click the **Ansys Electronics Desktop** shortcut on your desktop (or the same shortcut on your Start Menu) to launch it.

   The application launches with a new project created automatically:

   Then, do the following:

   a. Right-click the new project name (typically **Project*x***) at the top of the Project Manager and choose **Delete Project Permanently from Disk** from the shortcut menu that appears.
   b. Click **OK** when prompted to confirm the deletion.

2. Alternatively, if EDT was already running, save and close any open projects before proceeding to the next topic.

Open the Source Project

The source project file is installed with the Ansys Electromagnetics Suite and is located in the **...\Help\Mechanical** subfolder. After opening this project, you will save it to a working folder to avoid attempting to save changes to the source project. Since the source file is located under the Program Files branch of the system drive, you likely have restricted write access. Additionally, it is best to leave the source project unaltered .

1. On the **Desktop** ribbon tab, click **Open Examples**.

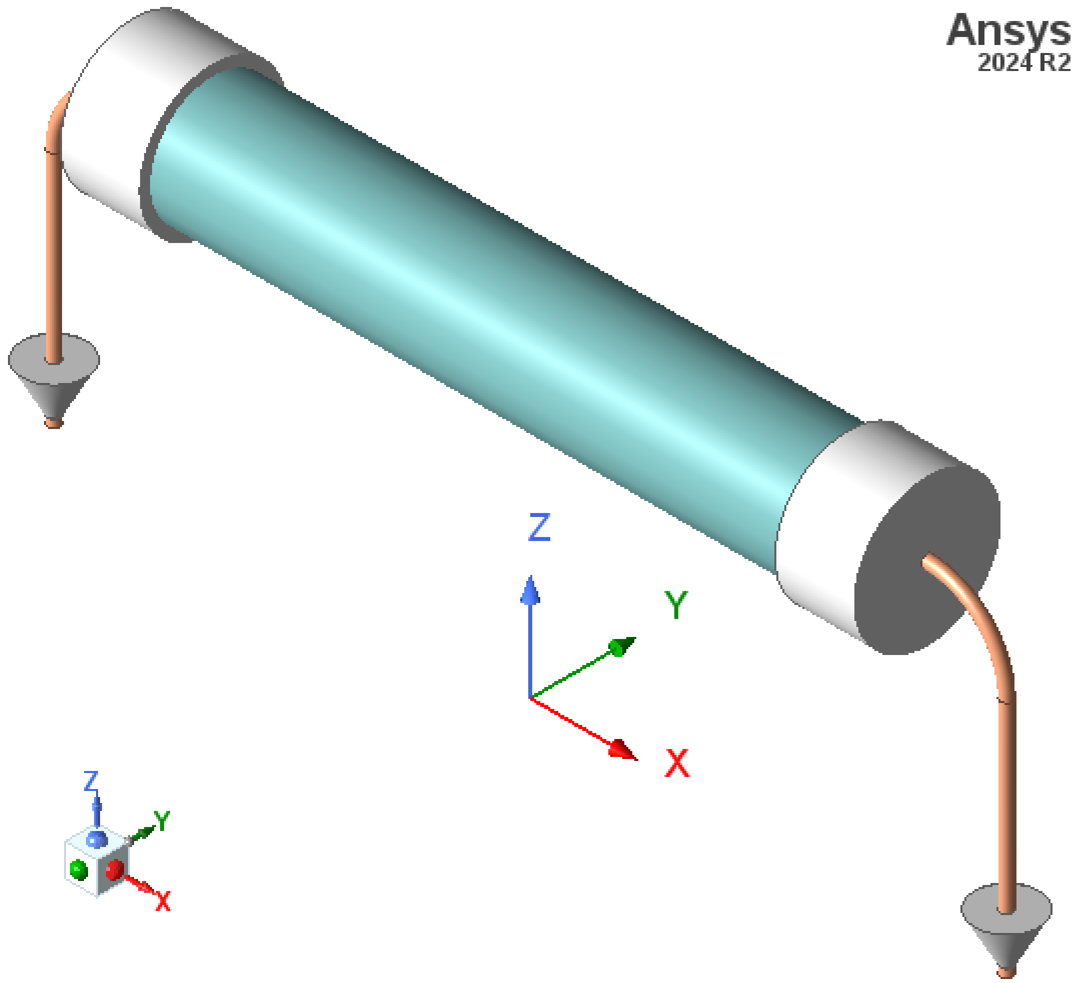
   The *Open* dialog box appears with the *Example* folder selected.

2. Click **Up One Level** to navigate to the parent folder of *Examples*.

   You will now be looking at the *Win64* or *Linux64* folder contents, depending on your platform.

3. Locate the **Help** subfolder and double-click it.
4. In the *Help* folder, locate the **Mechanical** subfolder and double-click it.
5. In the *Mechanical* folder, select **TransThermal_PwrResistor.aedt** and click **Open**.

   The model appearance should be as follows:



6. On the **Desktop** ribbon tab, click  **Save As**.
7. Navigate to the working folder of your choice and click **Save** to keep the original source project filename.

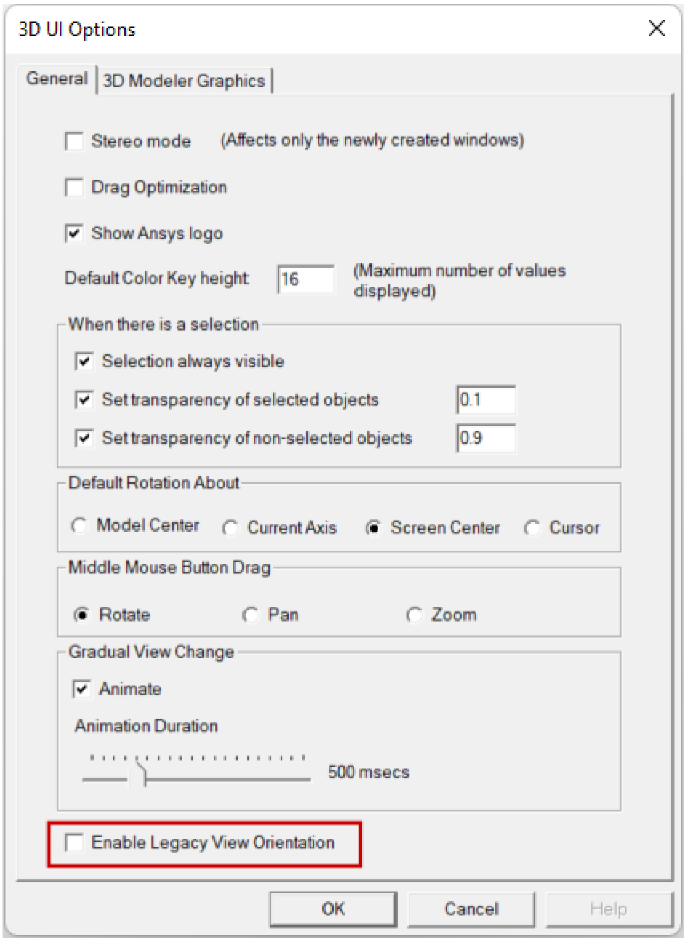
Set 3D UI Options

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:


3D UI Options
General
3D Modeler Graphics
Stereo mode
(Affects only the newly created windows)
Drag Optimization
Show Ansys logo
Default Color Key height
16
(Maximum number of values displayed)
When there is a selection
Selection always visible
Set transparency of selected objects
0.1
Set transparency of non-selected objects
0.9
Default Rotation About
Model Center
Current Axis
Screen Center
Cursor
Middle Mouse Button Drag
Rotate
Pan
Zoom
Gradual View Change
Animate
Animation Duration
500 msecs
Enable Legacy View Orientation
OK
Cancel
Help

3. Click **OK**.

3 - Modify Materials

The units and solution type have already been defined in the source design. Materials have also been specified along with the desired color choices. However, we will revise the material properties for this exercise.

Thermal changes can take a fairly long time to occur due to the mass and specific heat of materials. These properties affect how much energy is needed to change the object's temperature. For any given thermal power input, the time it takes to produce a given temperature change increases in proportion to the product of specific heat and mass. To accelerate the achievement of steady-state results, we will decrease the specific heat of each material in this model by one order of magnitude. That is, each specific heat value will be divided by 10. The remaining properties will remain unchanged. As such, the time points in the transient solution of this model will be approximately one-tenth of the actual time that would be required for the same results (when based on the true material specific heat values).

> **Note:** Even though you will be running a transient thermal solution, the duration of the transient event will be sufficient to achieve steady-state results.

Modify the *Specific Heat* property of each material as follows:

1. Under *Model > Solids > Al2_O3_ceramic* in the History Tree, select **Body**.
2. In the docked *Properties* window, choose **Edit** from the **Material** drop-down menu.

   The *Select Definition* dialog box appears with the current material assignment selected. Perform the following steps:

   a. Click **Clone Material(s)**.
   b. The *View / Edit Material* dialog box appears, and it displays a new material with the same properties as the cloned one except for the name, which has " - Copy" appended to it. Change the material name and specific heat value as follows:
      - **Material Name** = **Al2_O3_ceramic_LSH** (the "_LSH" suffix is for *"Low Specific Heat"*)
      - **Specific Heat** = **85 J/kg-C** (the original value divided by 10)
   c. Click **OK** to apply this material to the resistor body and to close the *Select Definition* dialog box.
3. Under *copper* in the History Tree, select **Wire_1** and **Wire_2**.

   Repeat steps 2 through 2c but this time, specify the following properties:

   - **Material Name** = **copper_LSH**
   - **Specific Heat** = **38.5 J/kg-C**
4. Click **OK**.
5. Under *nickel, DC* in the History Tree, select **EndCap_1** and **EndCap_2**.

Repeat steps 2 through 2c but this time, specify the following properties:

- **Material Name** = **nickel, DC_LSH**
- **Specific Heat** = **44.4 J/kg-C**

6. Click **OK**.
7. Under *solder* in the History Tree, select **Solder_1** and **Solder_2**.

   Repeat steps 2 through 2c but this time, specify the following properties:

   - **Material Name** = **solder_LSH**
   - **Specific Heat** = **16.7 J/kg-C**
8. Click in the Modeler window background area to clear the selection.

4 - Assign Convection Boundaries

You will assign two different convection boundaries to this model. The first one uses a calculated and constant film coefficient that is intended to approximate conduction of heat from the solder joints to the traces of a printed circuit board (PCB) that is not included in the model.

First Convection Boundary

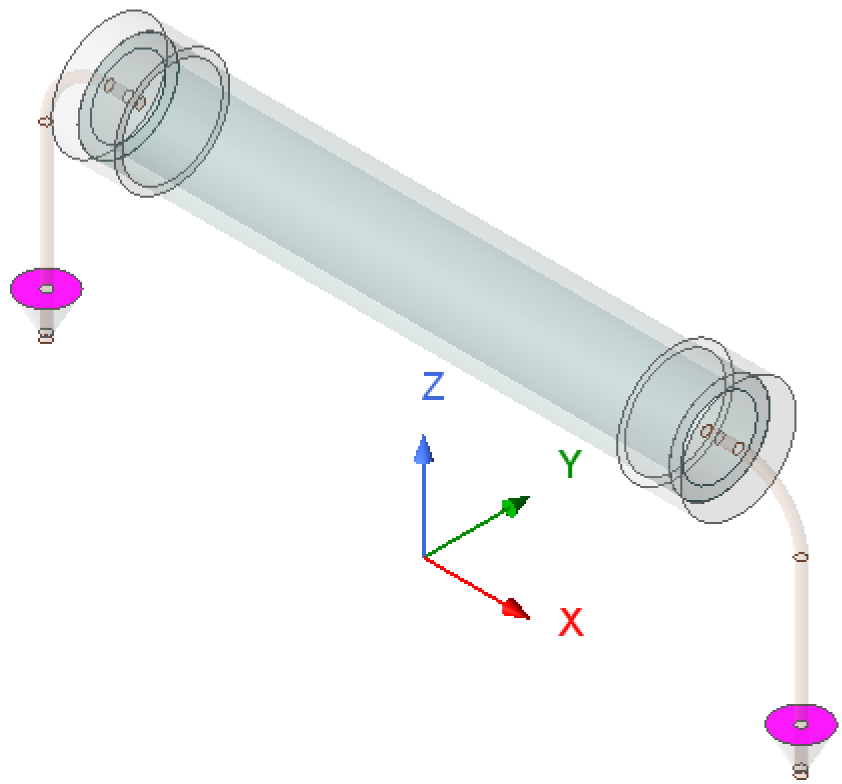
As mentioned in the *Introduction* to this guide, a nearly identical power resistor is used in the *Structural Solution – Power Resistor* getting started guide. A negative heat flux boundary was assigned to each solder-to-PCB contact face to remove a specified amount of heat from the model. The specified power was -0.5 W per face, and the resulting delta T, relative to ambient temperature was approximately 20° C. That assumption was based on a total heat generation of 7.5 W in the resistor body.

For our transient thermal solution, we will calculate an equivalent convection boundary. The heat generation in this solution varies. A convection boundary at the solder-to-PCB contact faces will allow the temperature of these faces, and the heat conducted through them, to vary as the heat generation varies. The derivation of the film coefficient follows:

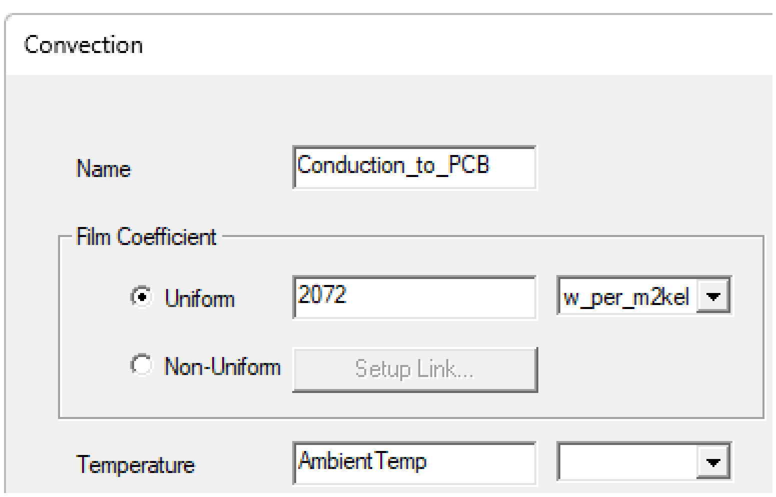
- Assumed thermal power conducted to the PCB at each joint = 0.5 W
- Assumed solder face temperature = 40° C
- Ambient temperature = 20° C
- Delta-T at convection faces = 40° C - 20° C = 20° C or 20° K
- Surface area of PCB contact face of each solder joint = 12.0637 mm^2 = 1.20637 x 10^{-5} m^2

Therefore, the **Film Coefficient** = 0.5 W / (20° K * 1.20637 x 10^{-5} m^2) = **2072 W / (m^2**. Assign this boundary as follows:

1. Click in the Modeler window to make sure that it is active and then press **F** to switch to the *Face* selection mode.
2. Select the top, horizontal face of each solder joint, as shown in the following image. Click the first face and Ctrl+click the second one:



3. Right-click in the Modeler window and choose **Assign Boundary** > **Convection** from the shortcut menu.
4. In the *Convection* dialog box that appears, specify the following settings:
    - *Name:* **Conduction_to_PCB**
    - *Film Coefficient:* **Uniform** **2072** **w_per_m2kel**
    - *Tenperature:* **AmbientTemp** (units selection is unsupported; the units are specified in the variable definition)



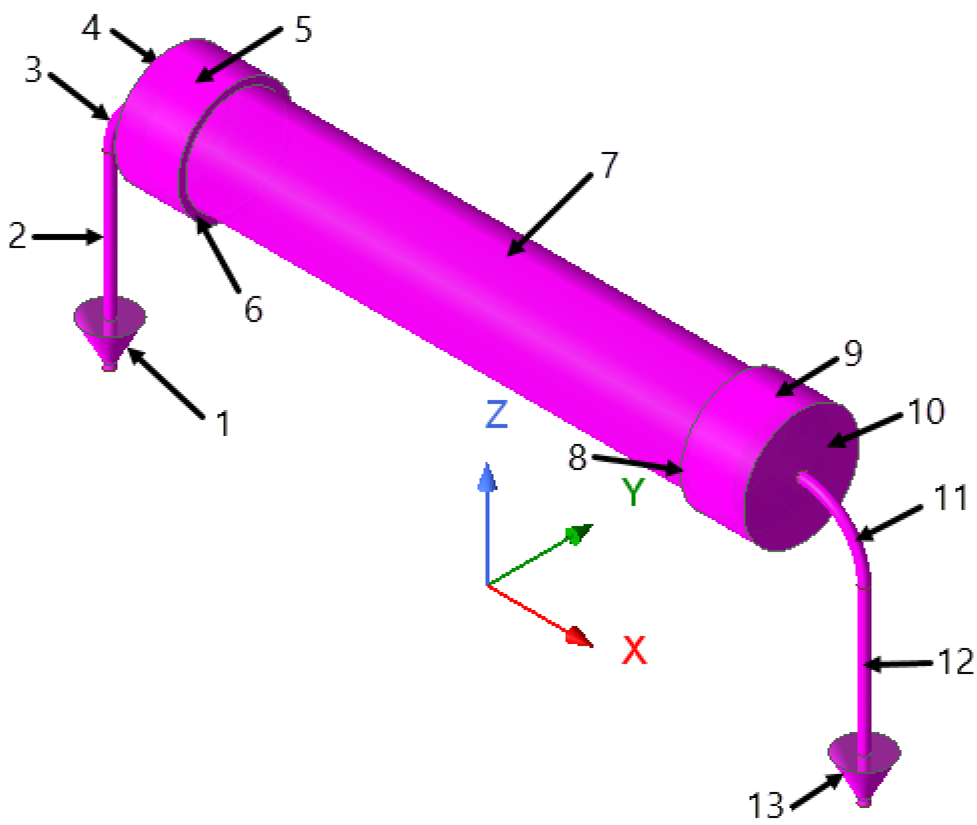
> **Note:**
>
> In the next topic, you will verify that the *AmbientTemp* variable is set at 20° C.

5. Click **OK**.

Second Convection Boundary

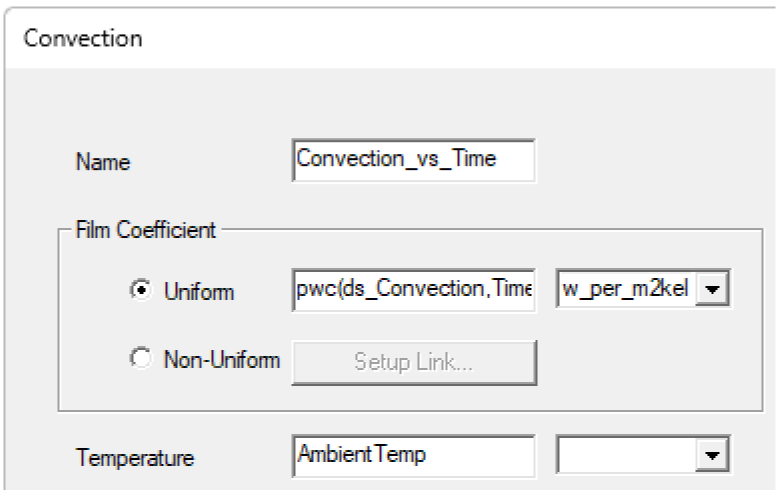
The second convection boundary will be assigned to all exposed faces of the power resistor, excluding the two faces where the first convection boundary has been assigned. The second boundary will use a design dataset referenced within a piecewise constant function. The dataset will specify two different film coefficients—one representing the low-speed mode of the circuit's cooling fan, and the other the high-speed mode. The assumed film coefficients are 25 and 40 W/(m^2·K), respectively, for the low- and high-speed fan modes. The circuit will switch from low to high power at 4 seconds and maintain high power throughout the remainder of the transient solution.

1. Select the thirteen faces indicated in the following image:



> **Note:**
>
> For faces hidden from view, you can either rotate the model viewpoint or Ctrl+click a face in front of the one you wish to select and then press **B** to change the selection to the face **B**ehind the initial selection.

2. Right-click **Boundaries** in the Project Manager and choose **Assign** > **Convection** from the shortcut menu.
3. In the *Convection* dialog box that appears, specify the following settings:
   - *Name:* **Convection_vs_Time**
   - *Film Coefficient:* **Uniform** **pwc(ds_Convection,Time)** **w_per_m2kel**
   - *Tenperature:* **AmbientTemp**

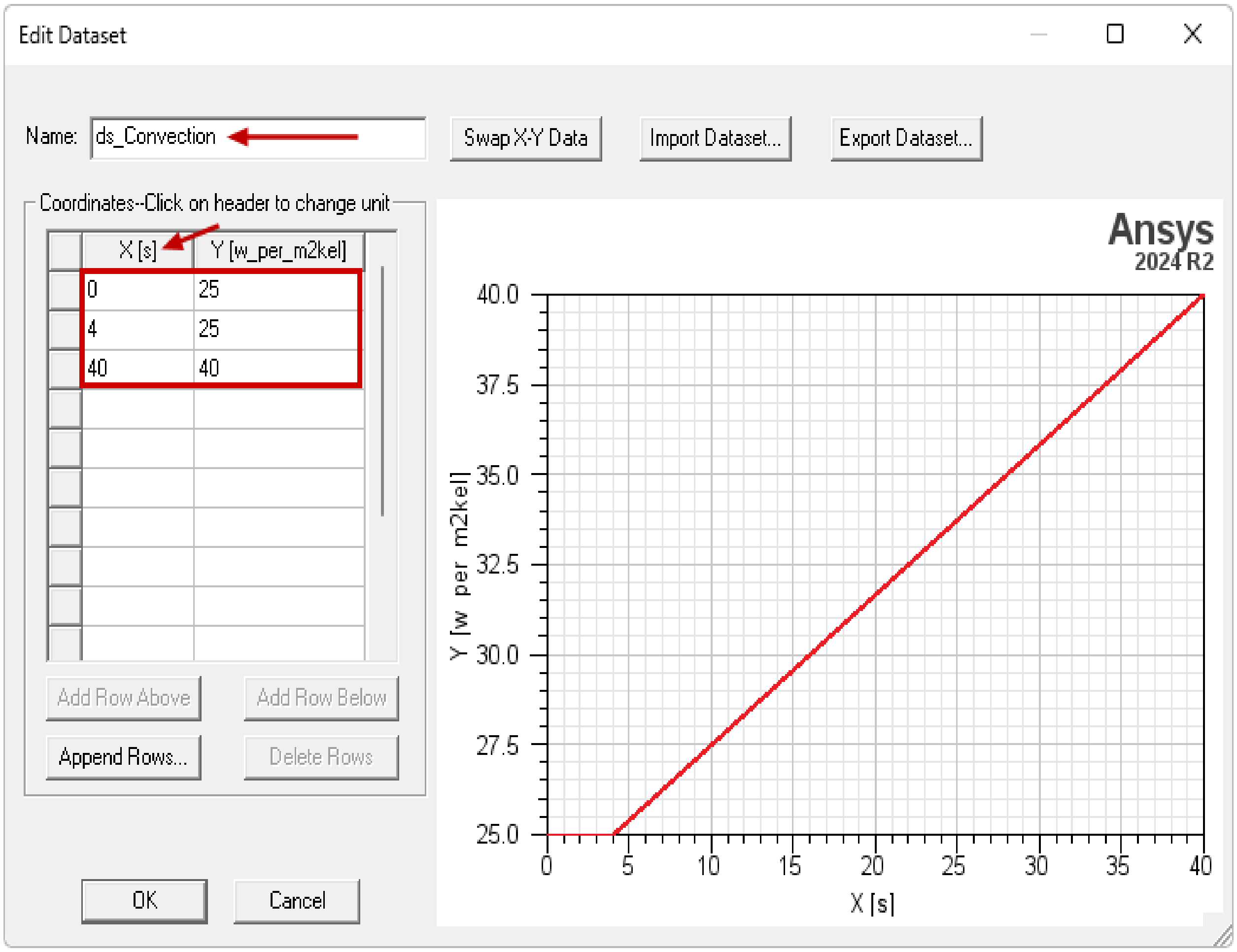


> **Note:**
>
> - "**pwc** indicates that the data will be interpreted using the *piecewise constant* function. When a value changes at a given time, the parameter being controlled steps to the new value **at the previous timepoint** and remains constant throughout the interval from the previous timepoint to the current one. Whereas, for a piecewise linear (pwl) function, the value would change linearly (as a ramp) from the value at the previous timepoint to the value at the current timepoint. This will become clearer when we define the dataset.
> - **ds_Convection** is the name of the dataset, and **Time** indicates that the values will be defined as a function of time. That is, the convection film coefficient will be time-varying in this case.
> - When a dataset is referenced in the uniform film coefficient, units selection becomes disabled. Additionally, after completing the convection asignment and dataset definition, if you revisit the *Convection* dialog box to review the boundary properties, the film coefficient units will be blank. The reason is that the units are specified within the dataset definition.

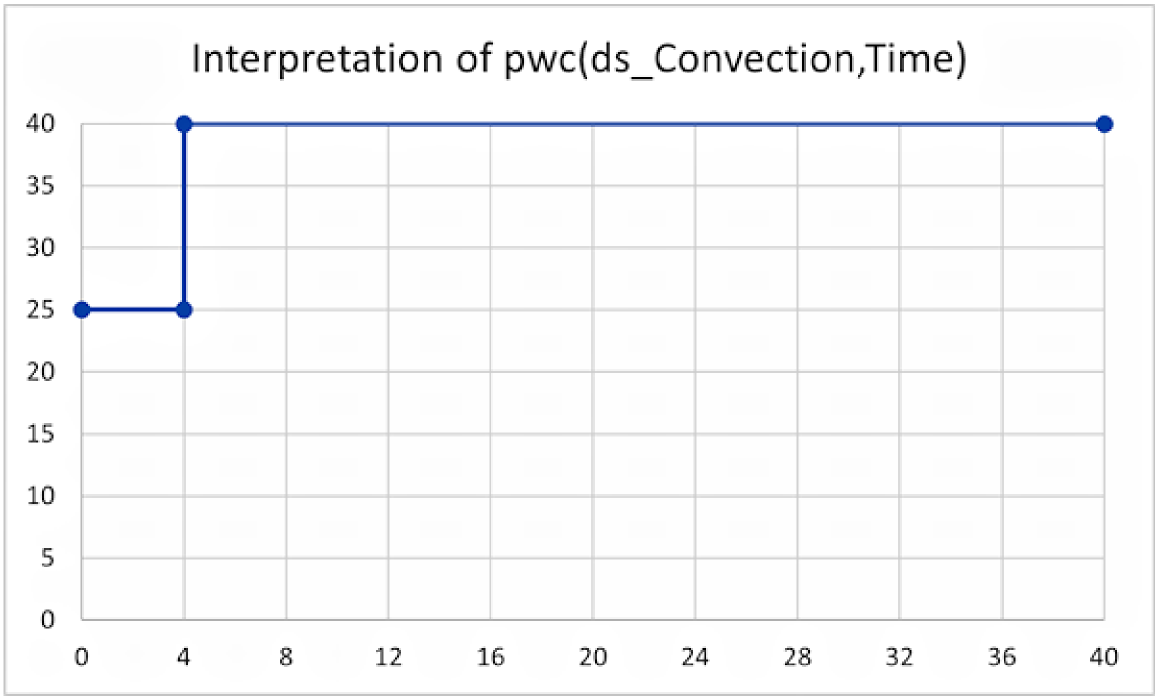
4. Click **OK**.

   The *Add Dataset* dialog box appears. Note that the dataset *Name* and the units of the X and Y columns have already been defined. We will change the time units from nanoseconds (the default) to seconds, and we will keep the default convection film coefficient units (w_per_m2kel).

5. Click the **X column heading** and choose **s** (seconds) from the drop-down menu.
6. Define the datapoints as shown in the following image:

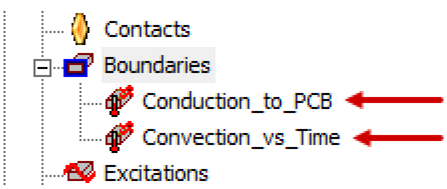


Within the *Add Dataset* dialog box, the data is always previewed according to the piece-wise linear function (pwl). However, when the datset is referenced within a piecewise constant function (pwc) it will be interpreted differently. Specifically, for this example, the resultant curve of convection film coefficient versus time will be as follows:

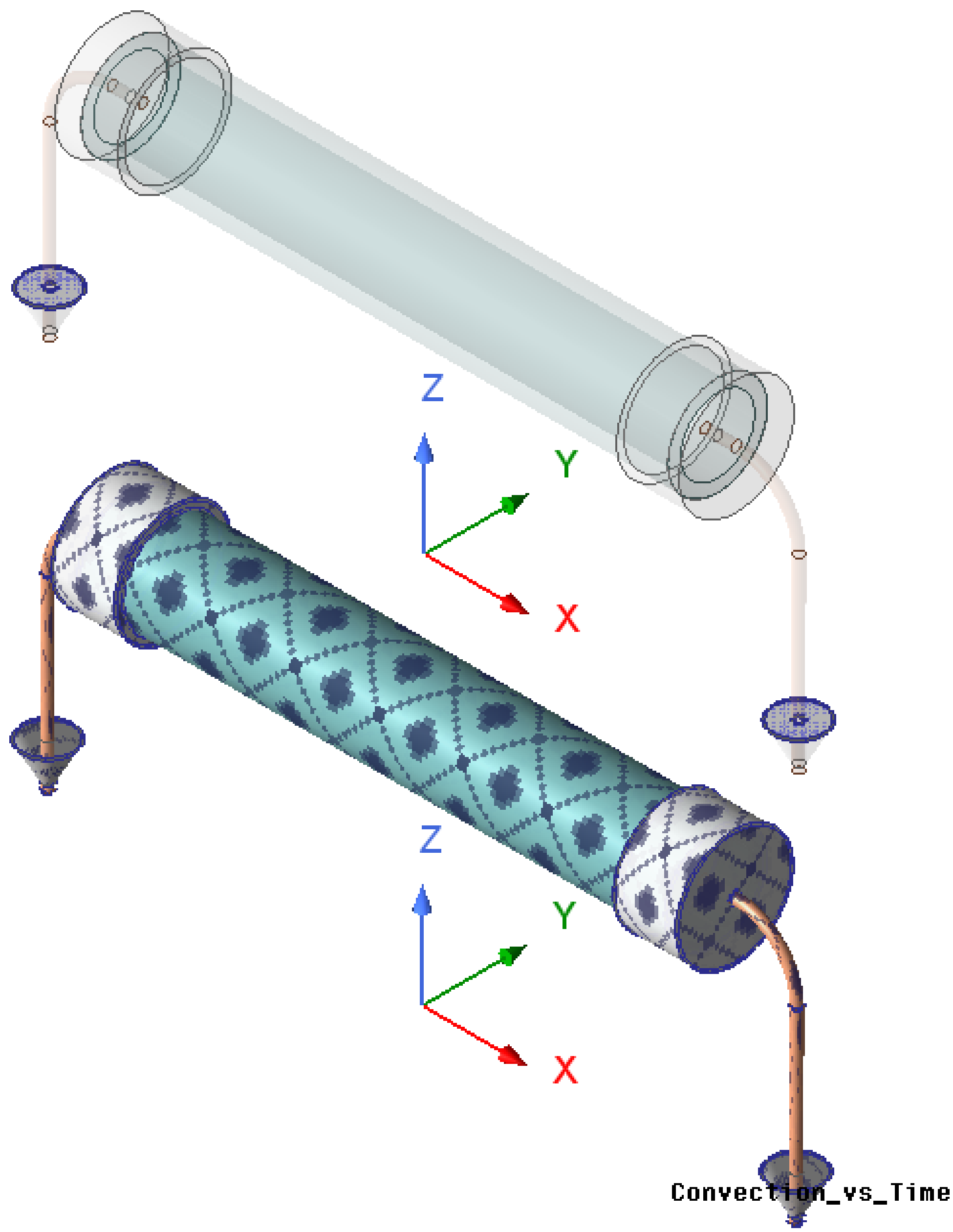


7. Click **OK**.

   The two convection boundaries that have been assigned are listed under *Boundaries* in the Project Manager:

8. You can select either one to see a boundary visualization overlaid on the model geometry:



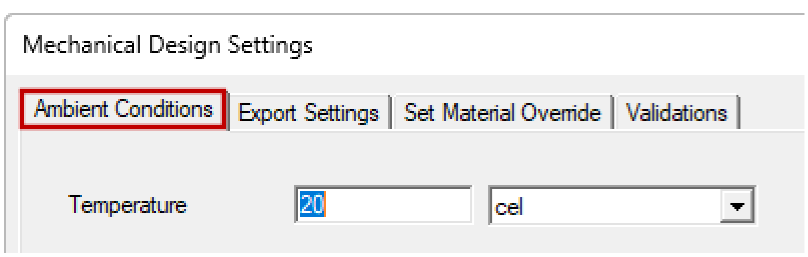
9. Click in the Modeler window background to clear any selection or boundary visualization before proceeding to the next topic.

> **Note:**
>
> Convection boundaries are automatically excluded from portions of assignment faces that are in default bonded contact with other solid objects. You need not 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.
>
> For this design, no thermal contact areas are assigned, and all objects are in default bonded contact. Therefore, the portion of the body's OD face that is inserted into the end caps, and the portion of each wire that passes through the solder will automatically be excluded from participating in convective heat flow.

Verify Ambient Temperature

For a clean installation of the Ansys Electromagnetics Suite, the default ambient temperature in Mechanical – Thermal designs is 20° C. Verify that this default temperature has not been changed, and set it to 20° C for this design if it has been changed.

1. Using the menu bar, click **Mechanical** > **Design Settings**.
2. In the **Ambient Conditions** tab of the *Mechanical Design Settings* dialog box that appears, ensure that the settings are as follows:
   - **Temperature** = **20**
   - **cel** (°C) is selected in the units drop-down menu.

3. Click **OK**.

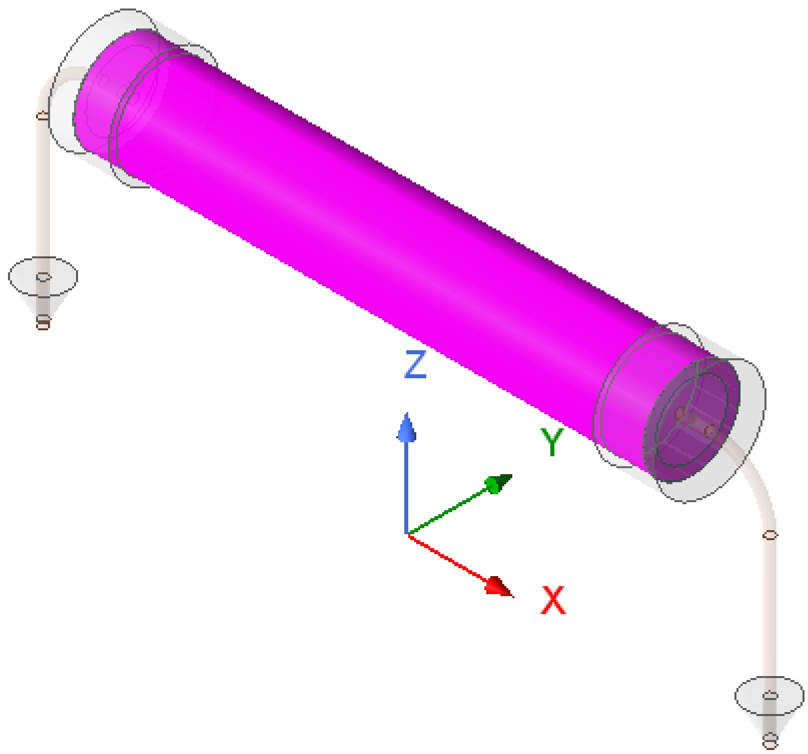
   Any boundary with *AmbientTemp* specified as the *Temperature* setting will be based on this global 20° C value.

5 - Assign Heat Generation

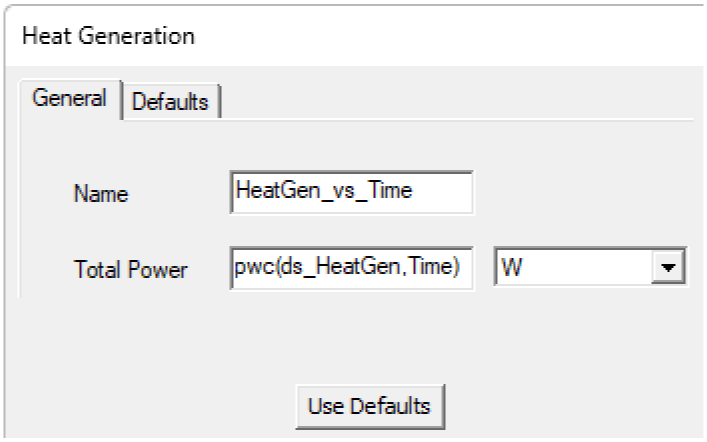
Heat input for this model comes from the transistor. Apply a *Heat Generation* excitation to the transistor as follows:

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 **Body** from the History Tree (under *Model* > *Solids* > *Al2_O3_ceramic_LSH*).

3. Right-click **Excitations** in the Project Manager and choose **Assign** > **Heat Generation** from the shortcut menu.
4. In the *Heat Generation* dialog box that appears, specify the following settings:
   - *Name:* **HeatGen_vs_Time**
   - *Total Power:* **pwc(ds_HeatGen,Time)** **W**

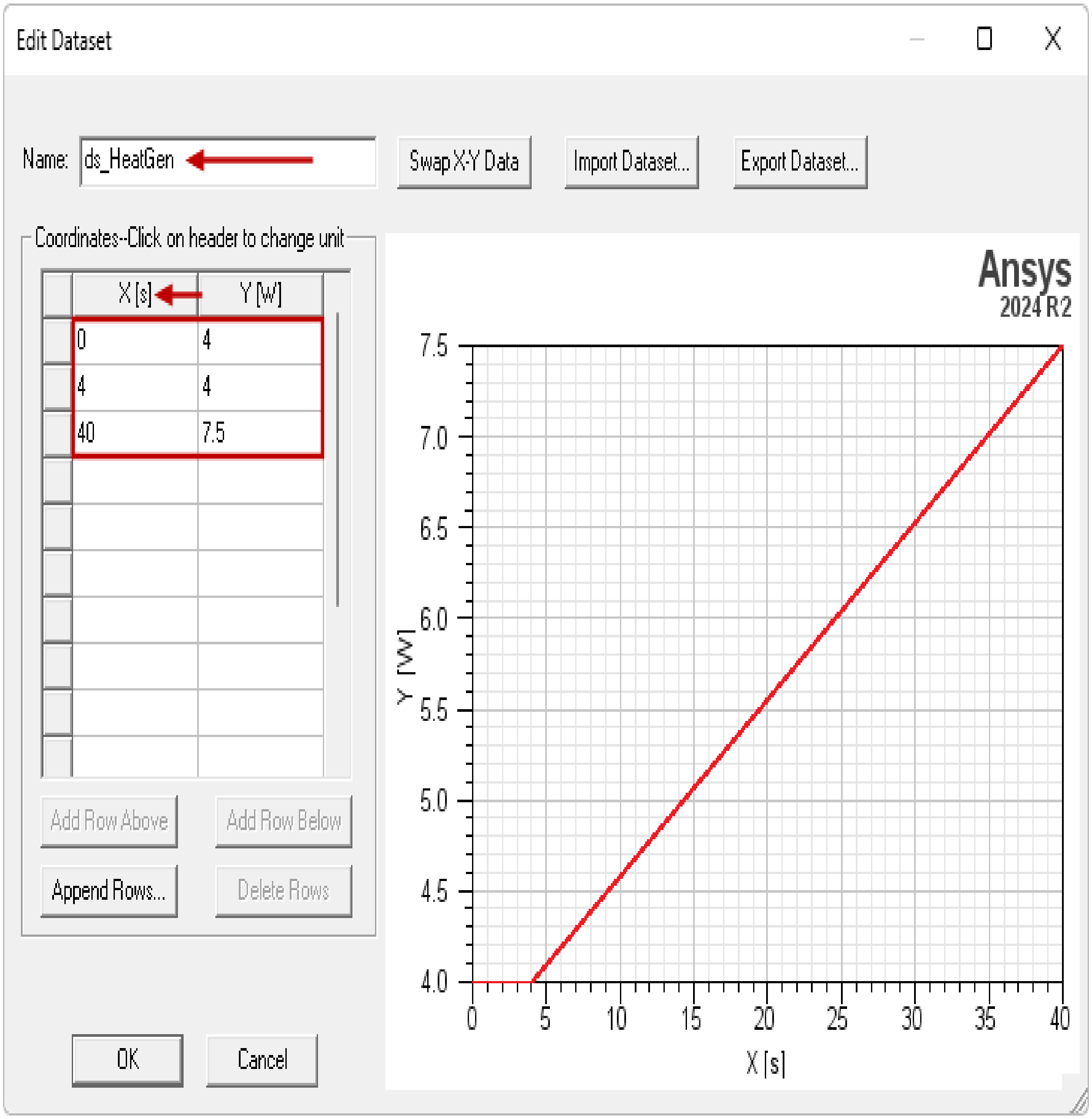


> **Note:**
>
> - *Total Power* refers to the heat dissipated by the device, not the electrical power the device conducts or outputs to the circuit. The power specified is generated per each selected assignment object.
> - When a dataset is referenced in the *Total Power* text box, units selection becomes disabled. Additionally, after completing the heat generation assignment and dataset definition, if you revisit the *Heat Generation* dialog box to review the excitation properties, the total power units will be blank. The reason is that the units are specified within the dataset definition.

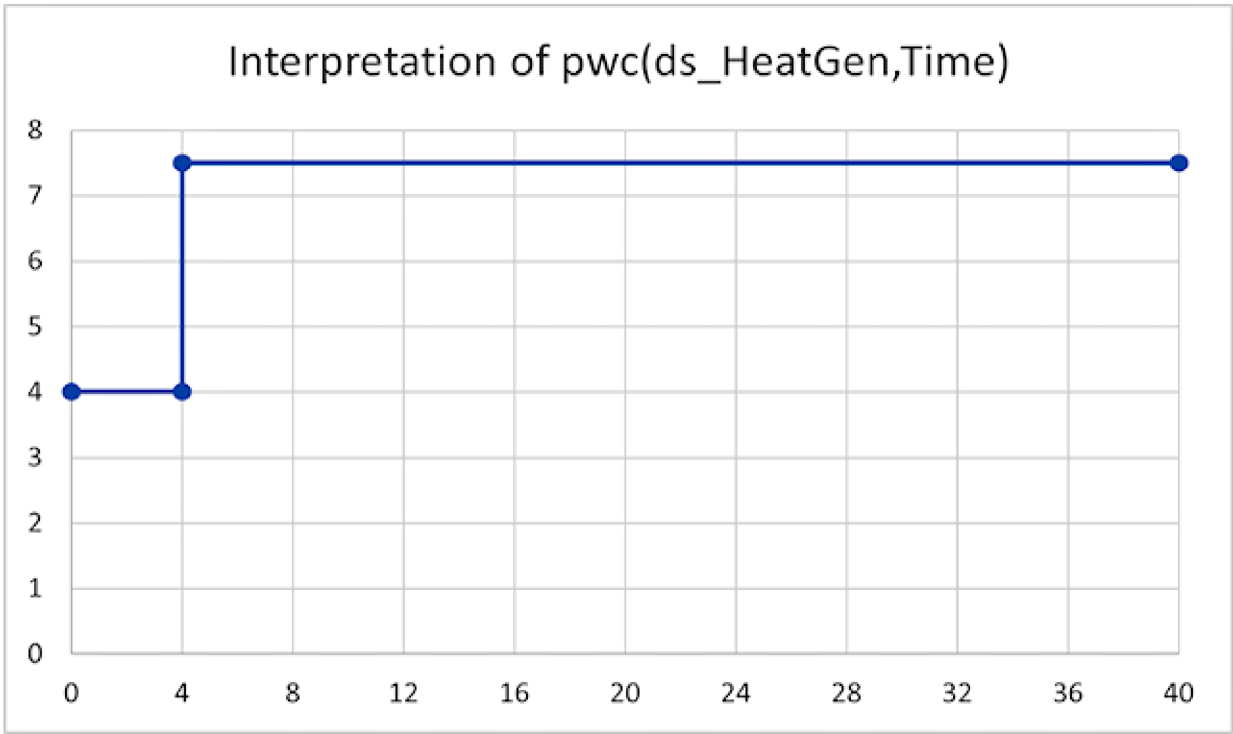
5. Click **OK**.

   The *Add Dataset* dialog box appears. Note that the dataset *Name* and the units of the X and Y columns have already been defined. We will change the time units from nanoseconds (the default) to seconds, and we will keep the default power units (W).

6. Click the **X column heading** and choose **s** (seconds) from the drop-down menu.
7. Define the datapoints as shown in the following image:

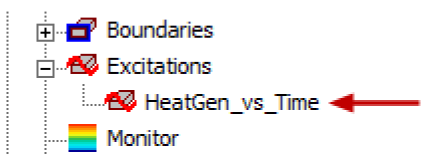


Within the *Add Dataset* dialog box, the data is always previewed according to the piece-wise linear function (pwl). However, when the datset is referenced within a piecewise constant function (pwc) it will be interpreted differently. Specifically, for this example, the resultant curve of convection film coefficient versus time will be as follows:

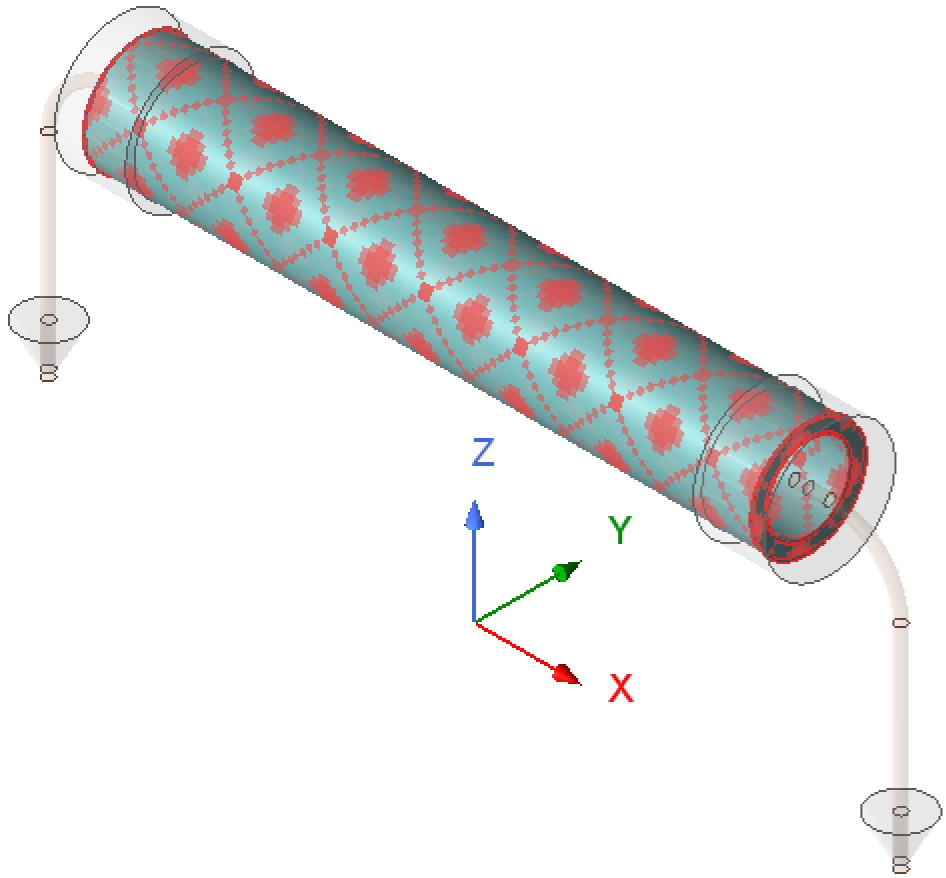


8. Click **OK**.

   HeatGen_vs_Time appears under *Excitations* in the Project Manager:

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



9. Clear the selection.

6 - Draw and Assign Thermal Monitor Points

In transient thermal solutions, you can define thermal monitor points at which the solver will output temperature versus time results. Unlike saved fields, temperatures at monitor points are reported for every timestep and for every substep the solver runs during the full transient anslysis duration. Since the data is written to your hard drive only for specified coordinates, the memory requirement and solution time can be significantly lower than what is needed to save fields results, which are global and recorded for all nodes or elements. Often, the monitor points provide all the necessary output, and you can choose either to not save fields or to only save them for several time points of interest.

You can assign monitor points to objects. However, the resultant point is at the object's centroid, which may be outside of the object's body (for example, if the body is hollow, C-shaped, or L-shaped). In such cases, you can draw a point inside the body's volume and assign a thermal monitor to the point. You will do so now to provide thermal monitors at the centroid of the hole where the lead wire is inserted into an end cap and at the center of a solder-to-PCB contact face. The resistor body, which is the heat source, will be the hottest object. No point is needed because the thermal monitor automatically includes a trace of the global maximum temperature. The two points you draw are both at the centerline of a lead wire but are good indicators of the minimal end cap temperature and the temperature where the solder contacts the PCB.

1. On the **Draw** ribbon tab, click **Point**. Then, complete the following steps:
    a. Press **Tab** to jump to the **X** coordinate entry text box in the status bar at the bottom of the Ansys Electronics Desktop window.

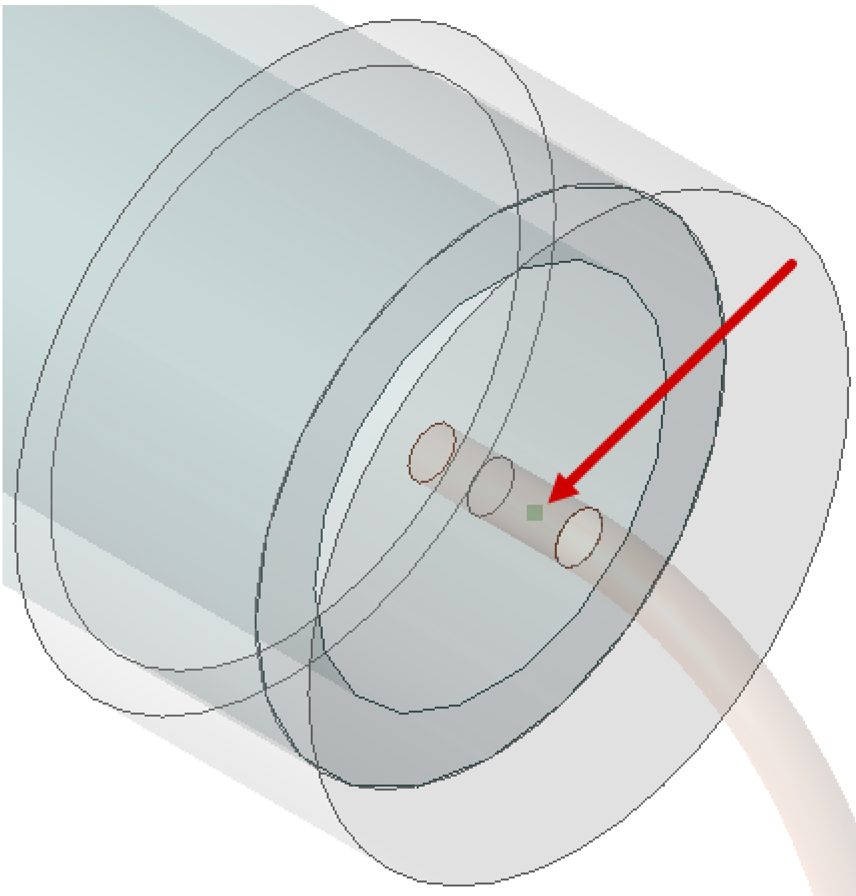
    > **Important:**
    >
    > Be careful not to touch, move, or bump the mouse while using the keyboard to navigate and fill the coordinate entry text boxes. If you do, the coordinate entry method will revert to the pointing mode, and any typed coordinates will be replaced by the cursor location.

    b. Type **24.25** in the **X** text box.
    c. Press **Tab**.
    d. Type **0** in the **Y** text box.
    e. Press **Tab**.
    f. Type **20** in the **Z** text box.

    g. Press **Enter**.
2. Under *Points* in the History Tree, select **Point1**. Then:

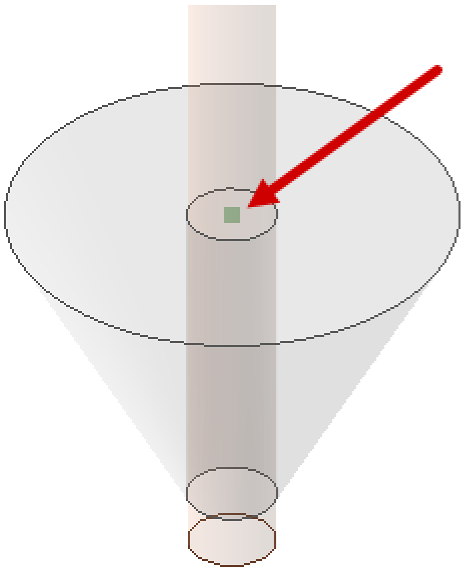
3. Verify that the point is located at the centerline of the hole into which the right wire is inserted into the right end cap and at the mid-length of the hole:



4. With *Point1* still selected, right-click **Monitor** in the Project Manager, and choose **Assign** > **Point** from the shortcut menu. Then, in the *Monitor Setup* dialog box that appears:
   a. Change the **Name** to **EndCap**.
   b. Click **OK**.
5. On the **Draw** ribbon tab, click **Point**. Then, complete the following steps:
   a. Press **Tab**.
   b. Type **30** in the **X** text box.
   c. Press **Tab**.
   d. Type **0** in the **Y** text box.
   e. Press **Tab**.
   f. Type **3.5** in the **Z** text box.

   g. Press **Enter**.

6. Under *Points* in the History Tree, select **Point2**. Then:
7. Verify that the point is located at the centerline of the right wire and in the plane of the top, horizontal face of the solder:



8. With *Point2* still selected, right-click **Monitor** in the Project Manager, and choose **Assign** > **Point**. Then, in the *Monitor Setup* dialog box that appears:
   a. Change the **Name** to **Solder**.
   b. Click **OK**.
9. Clear the selection.

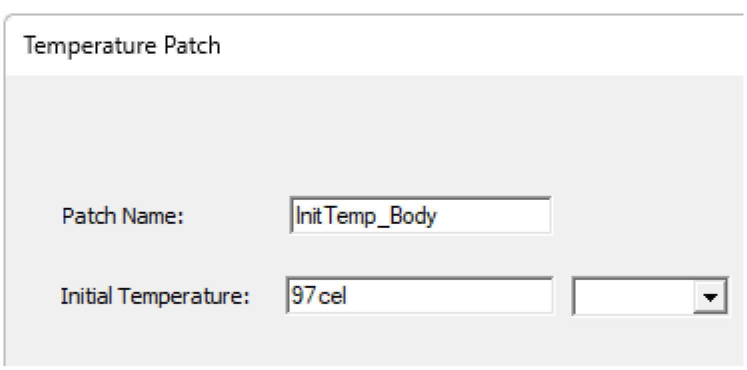
7 - Assign Initial Temperatures

To shorten the transient event duration and solution time, you will specify initial temperatures for all objects. These temperatures will be slightly less than the expected steady-state temperatures that occur for the low-power operating mode. However, once the power is increased, it will take longer to achieve steady-state results for the high-power mode.

There is a global initial temperature parameter in the solution setup that you will use in a later procedure for the lead wires and solder joints (four objects). For the remaining three objects (the body and end caps), you will define an object-based intital temperature assignment, which will override the global parameter.

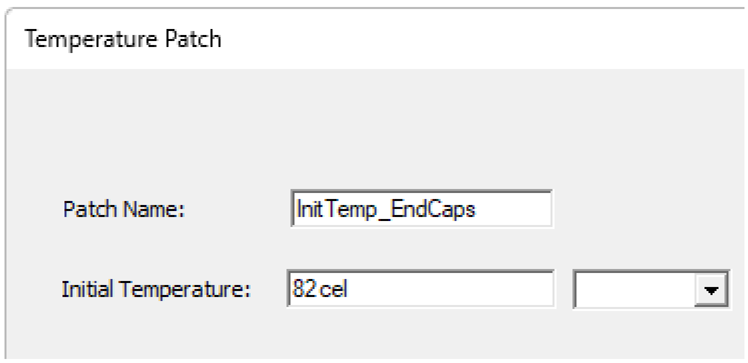
1. In the **Object** selection mode, select the resistor **Body**.
2. In the Project Manager, right-click **Initial Temperature** and choose **Assign** from the shortcut menu.
3. In the *Temperature Patch* dialog box that appears, specify the following properties:
    - **Patch Name** = **InitTemp_Body**
    - **Initial Temperature** = **97cel**

> **Tip:**
>
> As an alternative to choosing the units from the drop-down menu, you can type them into the text box. You must use the exact abbreviations that are supported, which are listed in the units drop-down menu. The input will be accepted whether or not a space is used between the number and the units abbreviation. If you later revisit the dialog box, the units will be shown as a drop-down menu selection, rather than remaining appended to the number in the text box.



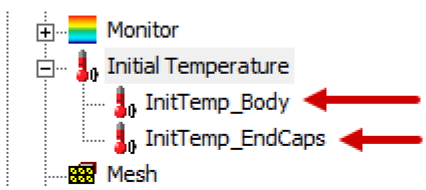
4. Click **OK**.
5. Select objects **EndCap_1** and **EndCap_2**.

6. Right-click in the Modeler window and choose **Assign Initial Temperature** from the shortcut menu.
7. In the *Temperature Patch* dialog box that appears, specify the following properties:
    - **Patch Name** = **InitTemp_EndCaps**
    - **Initial Temperature** = **82cel**

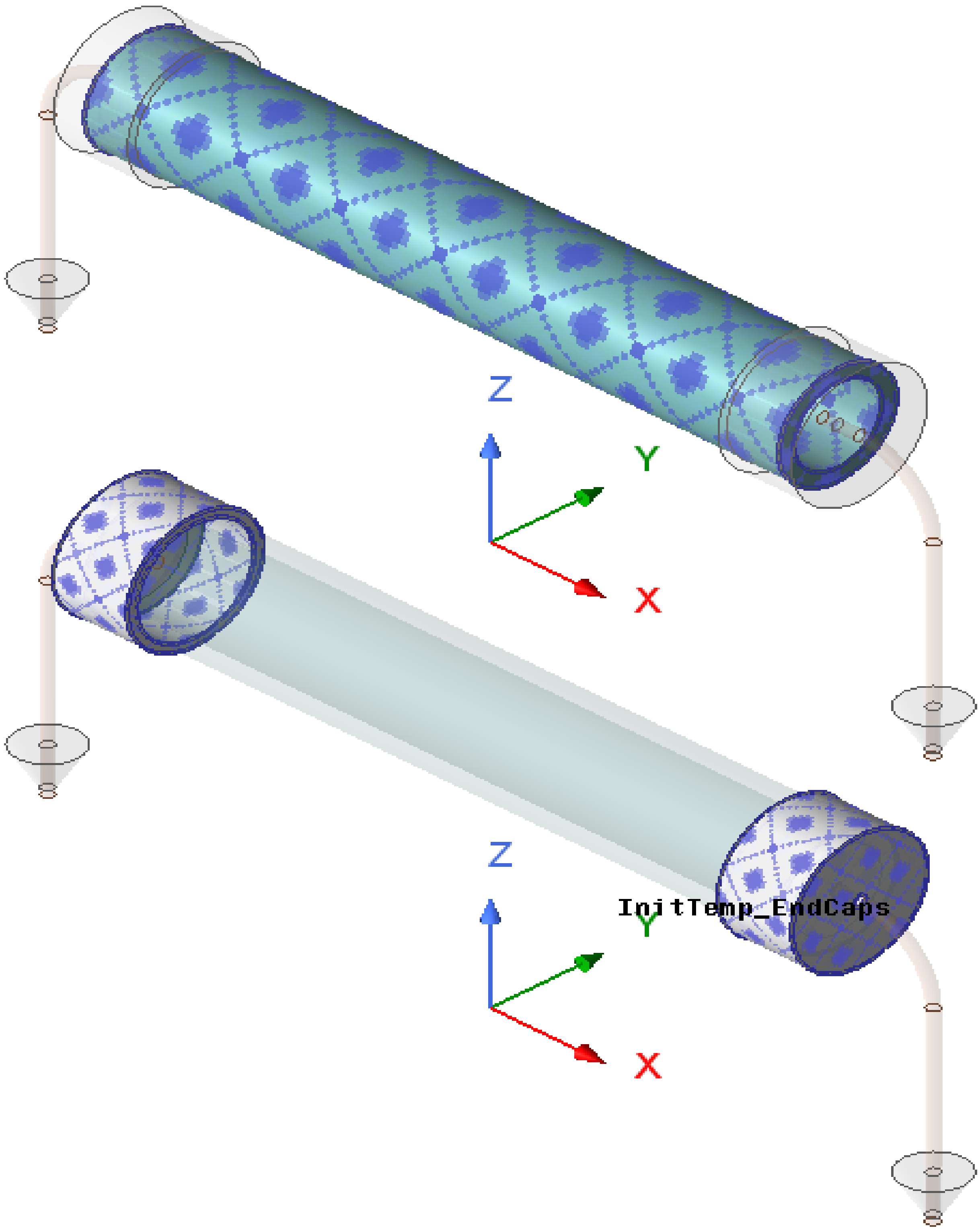


8. Click **OK**.

   The assigned initial temperatures are listed in the Project Manager.



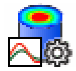
You can select each of them to see a visualization overlaid on the model geometery:



9. Clear the selection.

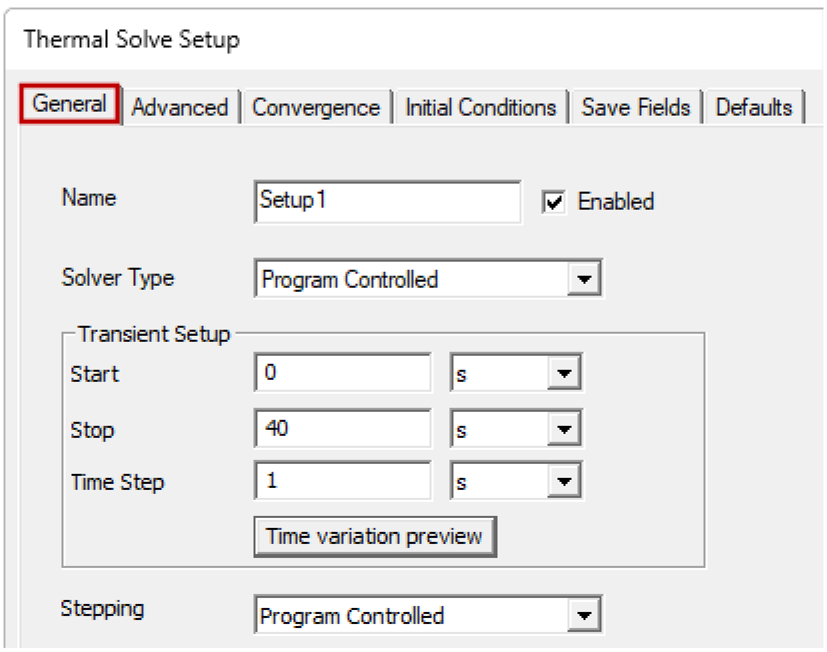
8 - Set Up, Validate, and Analyze Model

Create an analysis setup.p, validate the design, and run the analysis, as follows:

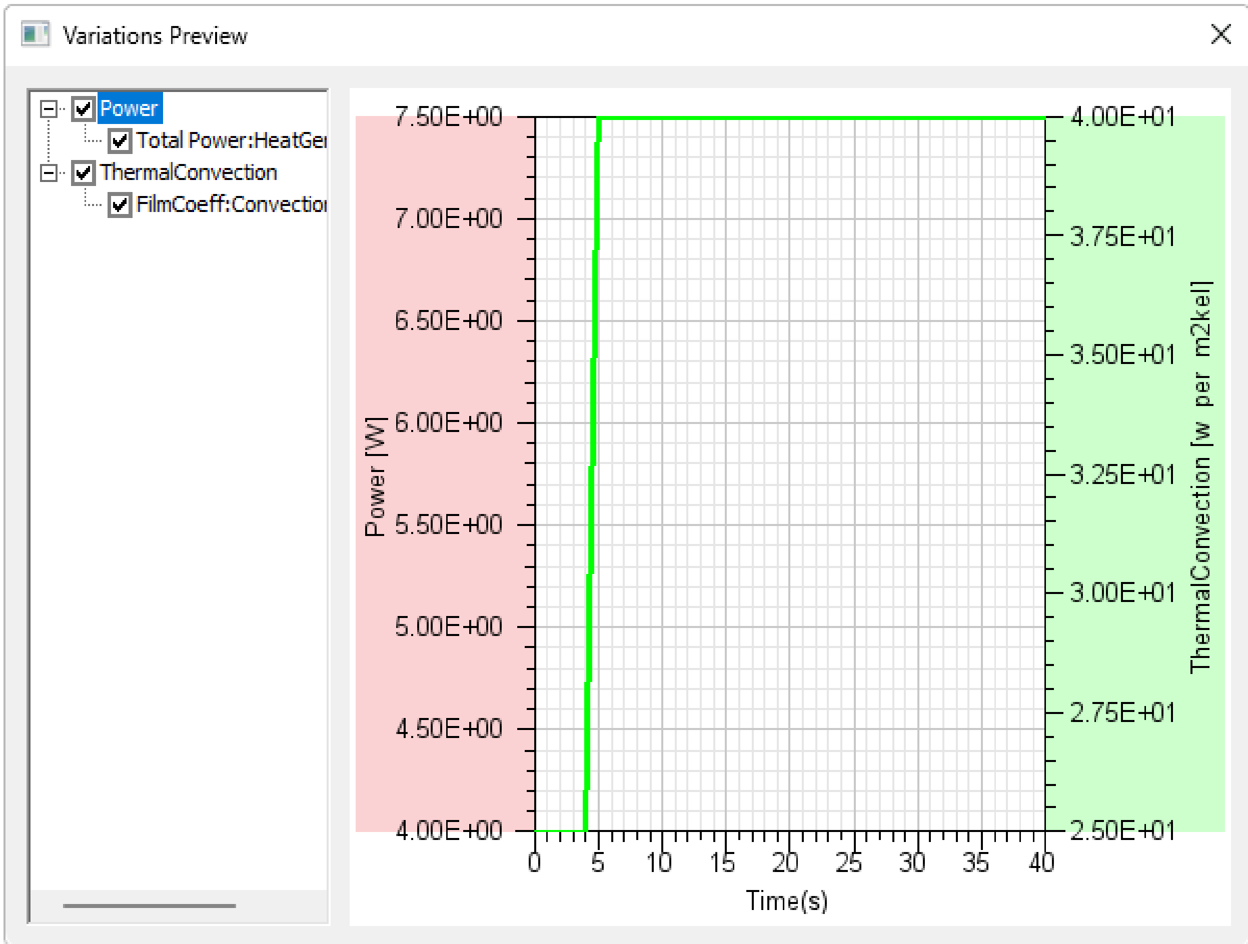
1. On the **Simulation** ribbon tab, click  **Setup**.

   The*General* tab of the *Thermal Solve Setup* dialog box appears.

2. Specify the following parameters, all in units of seconds (s):
   - *Start:* **0 s**
   - *Stop:* **40 s**
   - *Step:* **1 s**



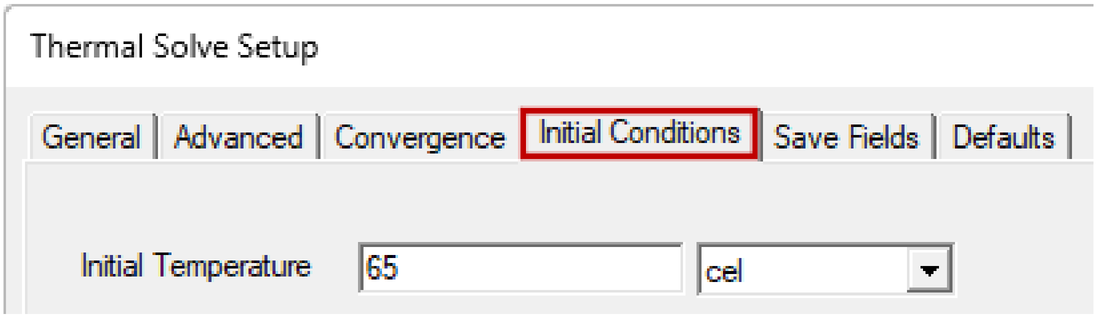
3. Click **Time variation preview** to see a plot of all time-varying boundaries and excitations:



> **Note:**
>
> - The Y axis on the left (red background) shows the values for the heat generation power. The Y axis on the right (green background) shows the convection film coefficient values. The green film coefficient trace exactly coincides with the red heat generation power trace and obscures it. Clear the **ThermalConvection** checkbox to see the red trace.
> - Each trace is based on a single value at each timestep (at each second in this case). Up to 4 seconds, the low-power convection and heat generation values are constant. Although the values immediately jump to the high-power mode at 4 seconds (as defined in the datasets), the next requested timestep after 4s is 5s, which reflects the increased values. Therefore, the variations preview shows a ramp with the values changing over a one second period, rather than a step change at 4s.

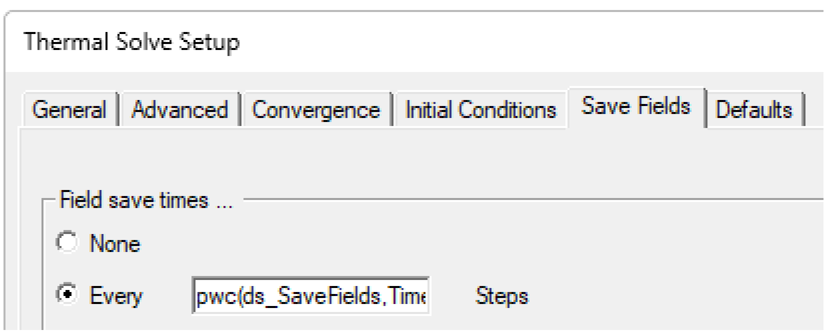
4. Close the *Variations Preview* window.

5. Sselect the **Intial Conditions** tab, type **65** in the **Initial Temperature** text box, and choose **cel** from the units drop-down menu:



> **Note:**
>
> This parameter sets the intial temperature globally, which applies to anthing that does not have an object-based initial temperature assignemnt. In this case, 65° C will be the starting temperature for the wires and solder, since the body and end caps have overriding temperature assignments.

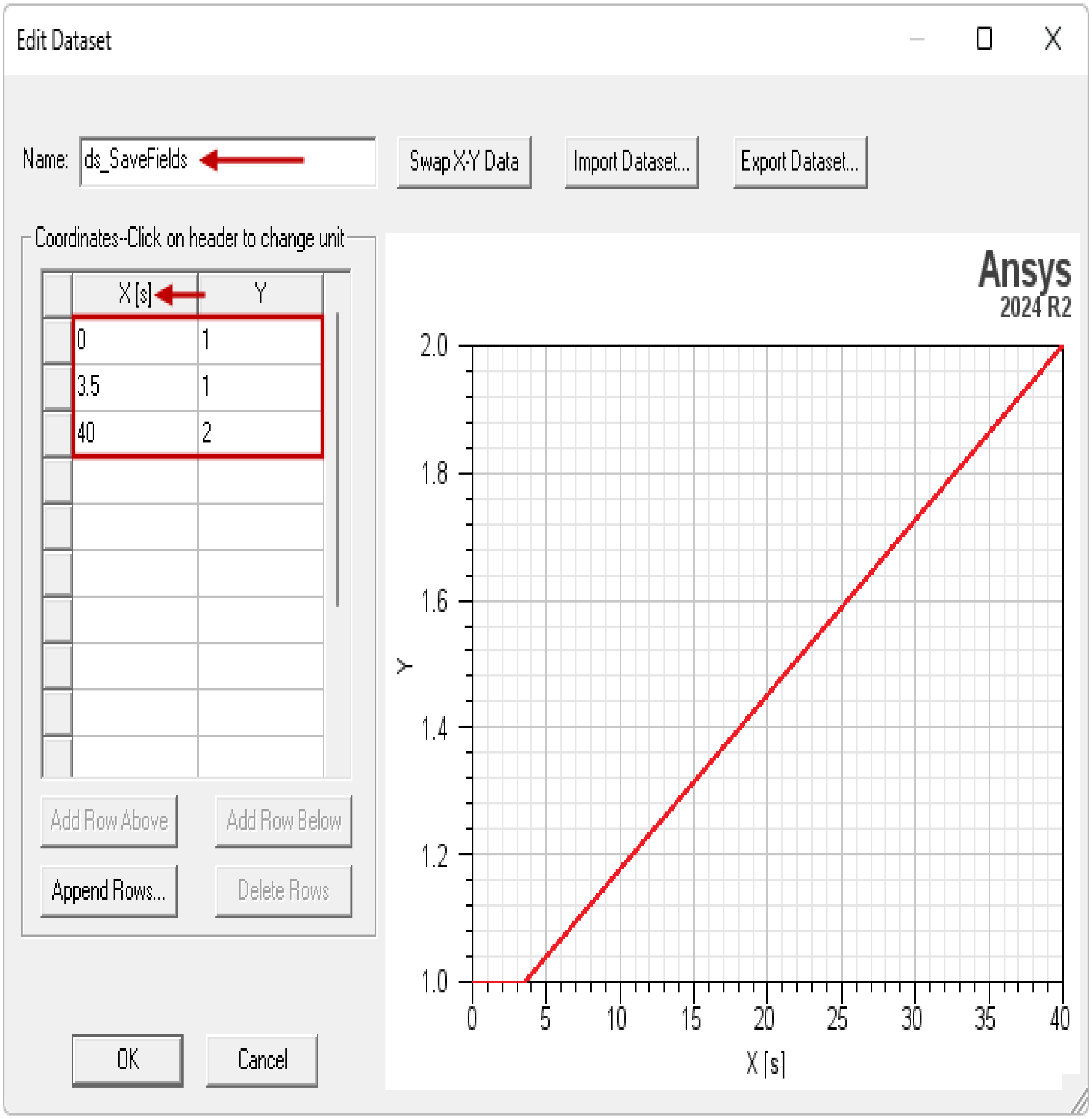
6. Select the **Save Fields** tab and then do the following:
   - Select the **Every** option.
   - In the **Steps** text box, type **pwc(ds_SaveFields,Time)**.



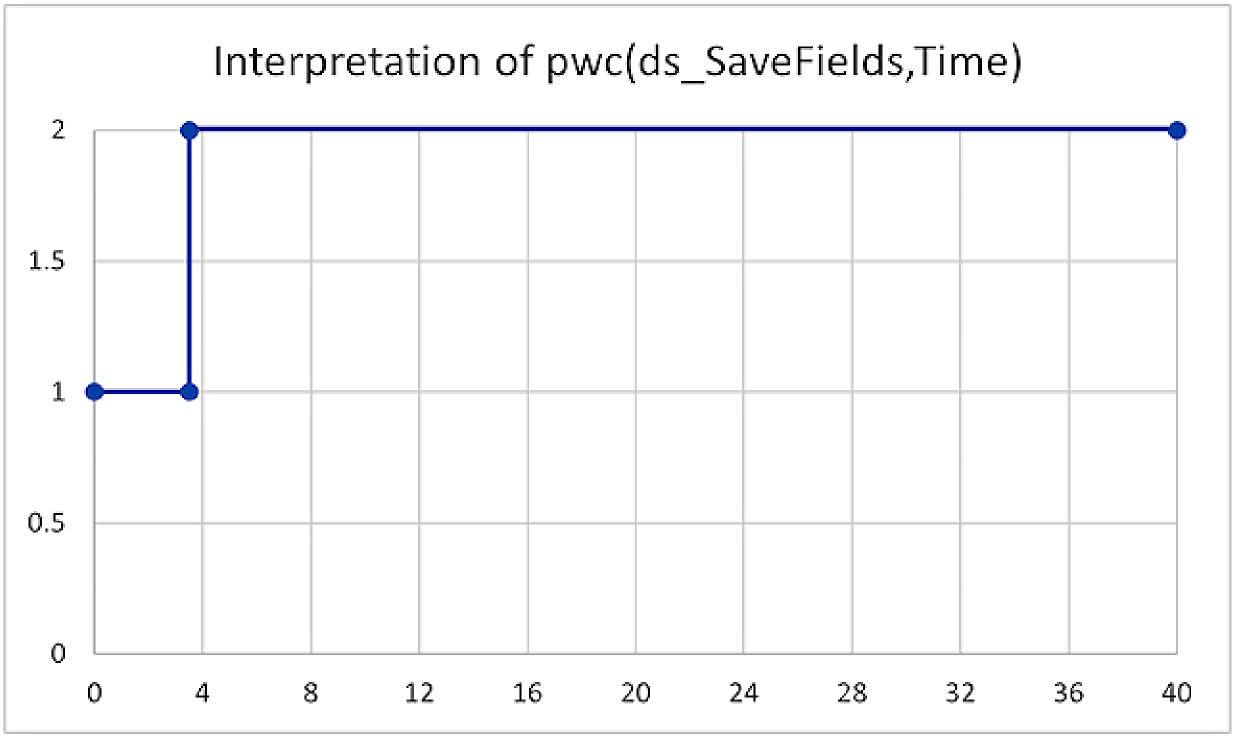
7. Click **OK** to complete the transient solution setup.

   The *Edit Dataset* dialog box appears. Note that the dataset *Name* and the units of the X have already been defined. We will change the time units from nanoseconds (the default) to seconds. The Y column values are dimensionless, and no unit will be selected.

8. Click the **X column heading** and choose **s** (seconds) from the drop-down menu.
9. Define the datapoints as shown in the following image:



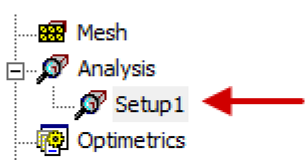
Within the *Add Dataset* dialog box, the data is always previewed according to the piece-wise linear function (pwl). However, when the datset is referenced within a piecewise constant function (pwc) it will be interpreted differently. Specifically, for this example, the resultant curve of the field-saving interval versus time will be as follows:



The fields will be saved for every timestep from 0 through 4 seconds. Thereafter, fields will be saved for every other timestep.

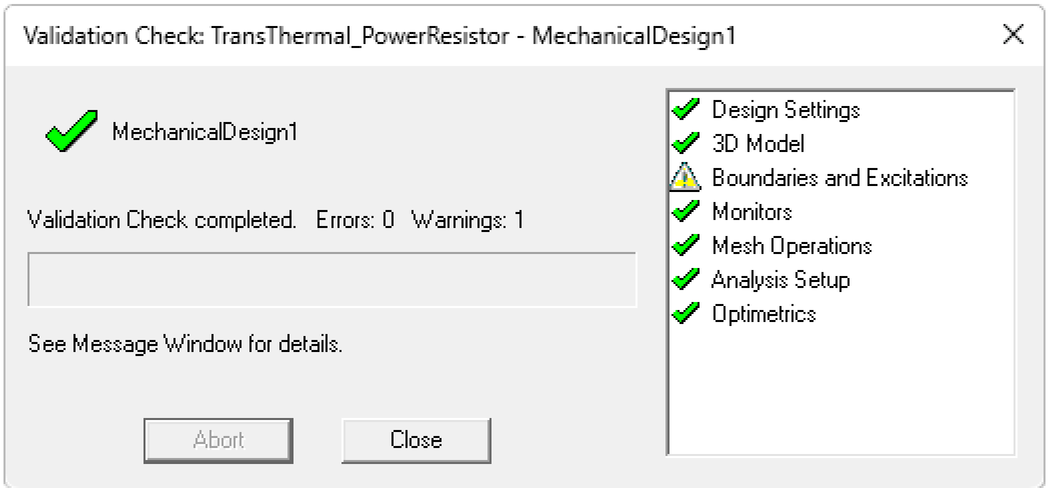
10. Click **OK**.

    Setup1 has been added under *Analysis* in the Project Manager:

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

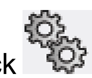
    The *Validation Check* window appears, and there should be no errors or warnings:



Note:

There is a warning associated with *Boundaries and Excitations*. If you look in the Messages window, you will see a warning about the initial temperature assignments at the body and end caps overlapping. Each end cap shares a cylindrical face with a portion of the body OD, and these objects are at two different initial temperatures. This warning is inconsequential and can be ignored.

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

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

    The solution should take about thirty minutes to complete on a current, high-end computer workstation. Procede immediately to the next topic while the solution is running.

9 - Evaluate Results

In this final section, you will observe the thermal monitor results as the solution progresses, generate a mesh overlay, and overlay the temperature results for the last timestep. Additionally, you will animate the temperature results versus time and create a summary of the heat flow rates.

The following three topics are covered in this section:

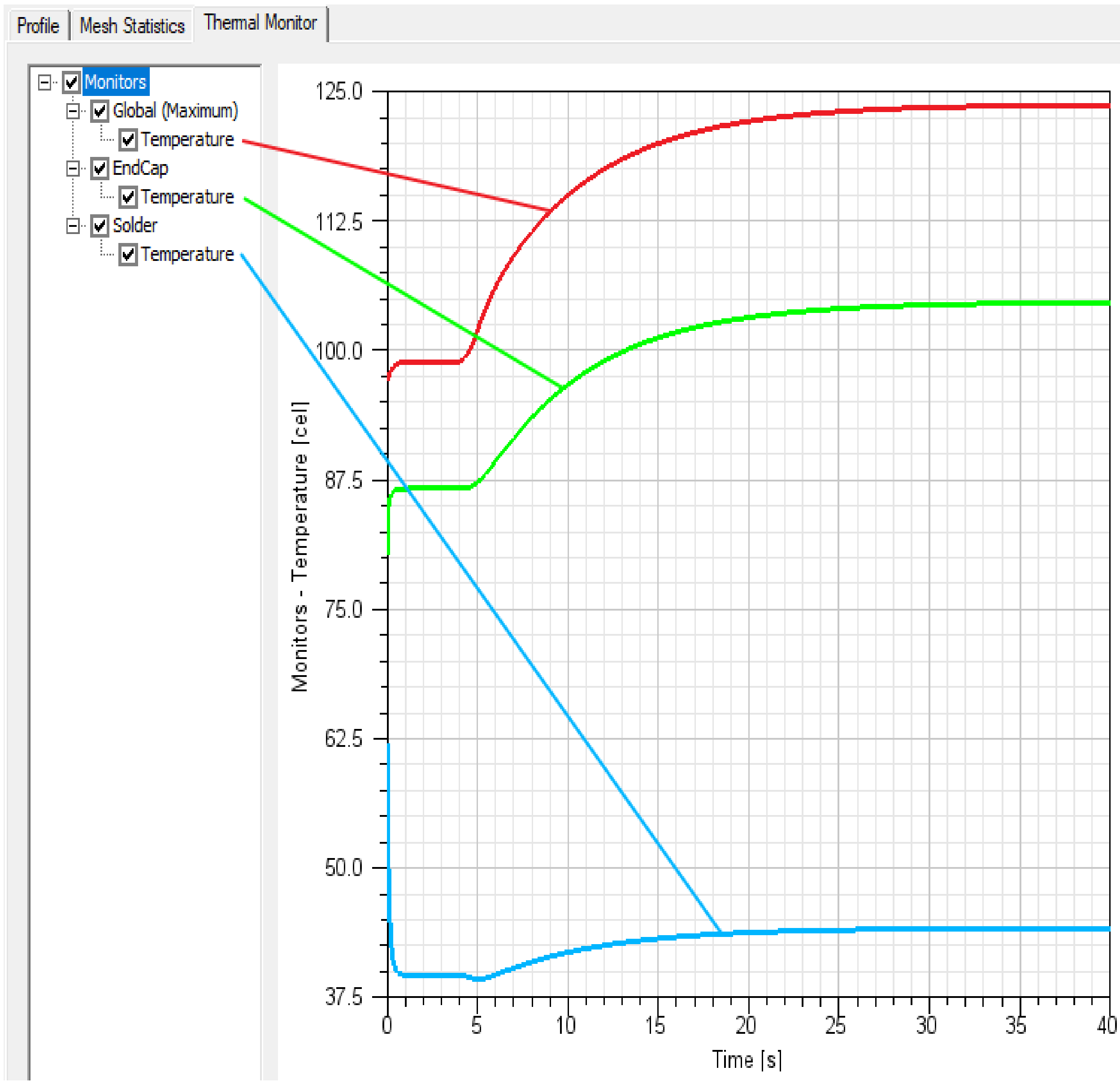
- **Thermal Monitor:** Observe temperature versus time results (global maximum and at monitor points)
- **Mesh Overlay:** Overlay the generated elements on the geometry in the Modeler window
- **Temperature Overlay**: Overlay the temperature results at the last timestep and animate them over the full transient event

Thermal Monitor

While the solution is still progressing, you can observe the temperature versus time results in the *Thermal Monitor*, as follows:

1. Under *Analysis* in the Project Manager, right-click **Setup1** and choose **Thermal Monitor** from the shortcut menu.

   The *Solutions* window appears with the *Thermal Monitor* tab selected. The traces develop gradually as each timestep is solved. The following figure is the appearance of the plot at the conclusion of the solution. Annotation lines have been added to match each monitor point to its trace:


Profile
Mesh Statistics
Thermal Monitor
Monitors
Global (Maximum)
Temperature
EndCap
Temperature
Solder
Temperature
Monitors - Temperature [cel]
125.0
112.5
100.0
87.5
75.0
62.5
50.0
37.5
0
5
10
15
20
25
30
35
40
Time [s]

Observations:

- The global maximum temperature corresponds to the resistor body, which is the source of heat generation. The maximum temperature is expected to occur near the middle of the body's length. The combination of convection at the end caps and wires, and conduction of heat through the wires and solder, should make the ends of the body cooler than its center. The temperature overlay created in the next topic will verify the expected behavior.
- For the low-power operating mode, steady-state temperature results are achieved very quickly due to the well-chosen initial temperature assignments, as intended.
- High-power operation occurs from 4 through 40 seconds (36 seconds duration). The rate of temperature change is maximal at the begining of this interval, and the slope decreases until the traces level off. By the end of the transient event, steady-state temperatures have been achieved at all objects. More precisely, at the resolution of this plot, no further temperature increase is observed beyone time ≈ 33 seconds (or about 29 seconds after the power increase occurred).

  Had the specific heat property of the materials not been modified, steady-state results would have been achieved about 290 seconds (10 * 29 s) after the beginning of the high-power mode. A longer-duration transient analysis, using unmodified materials, would verify that expectation, albeit with a significantly longer solution time. In addition to modifying the analysis setup, the time points in each design dataset would have to be revised. Additionally, *Save Fields* could be set to *None* to lessen the input/output time and storage requirements.
- Immediately after the switch to high-power mode occurs (at 4 seconds), the temperature at the solder-to-PCB junction (blue trace) decreases momentarily before starting to increase. This phenomenon is easy to explain. The convection film coefficent increases at the same time as the heat generation in the body increases. The enhanced convection at the conical face of the solder immediately starts cooling it off before the increased heat generation has sufficiently raised the body and end cap temperatures and conducted down through the wires to the solder joints. So there is a delay before solder heating begins.

  A similar effect likely occurs at the end cap-to-wire intersection area, but to a lesser extent and for a shorter duration, making it insignificant. The effect is not apparent at the the end cap monitor point (green trace) due to the limited resoltuion of this plot.

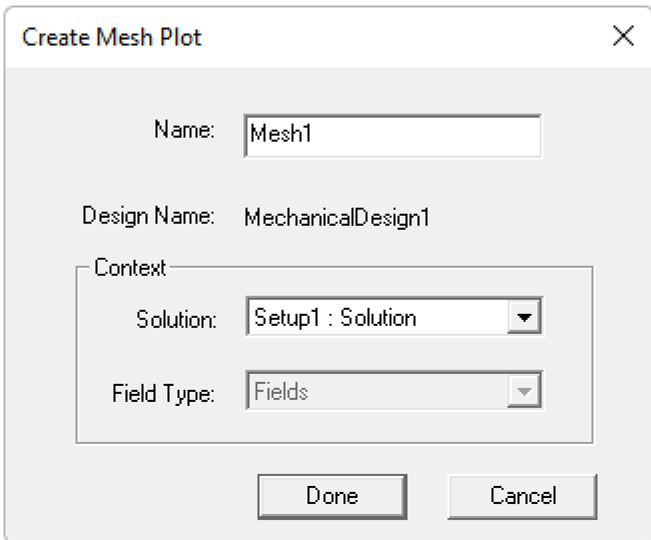
2. Close the *Thermal Monitor* window.

Mesh Overlay

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

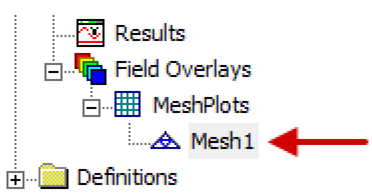
1. In *Object* selection mode, click in the Modeler window and press **Ctrl+A** to select all objects.
2. Right-click **Field Overlays** in the Project manager and choose **Plot Mesh** from the short-cut menu.

   The *Create Mesh Plot* dialog box appears:

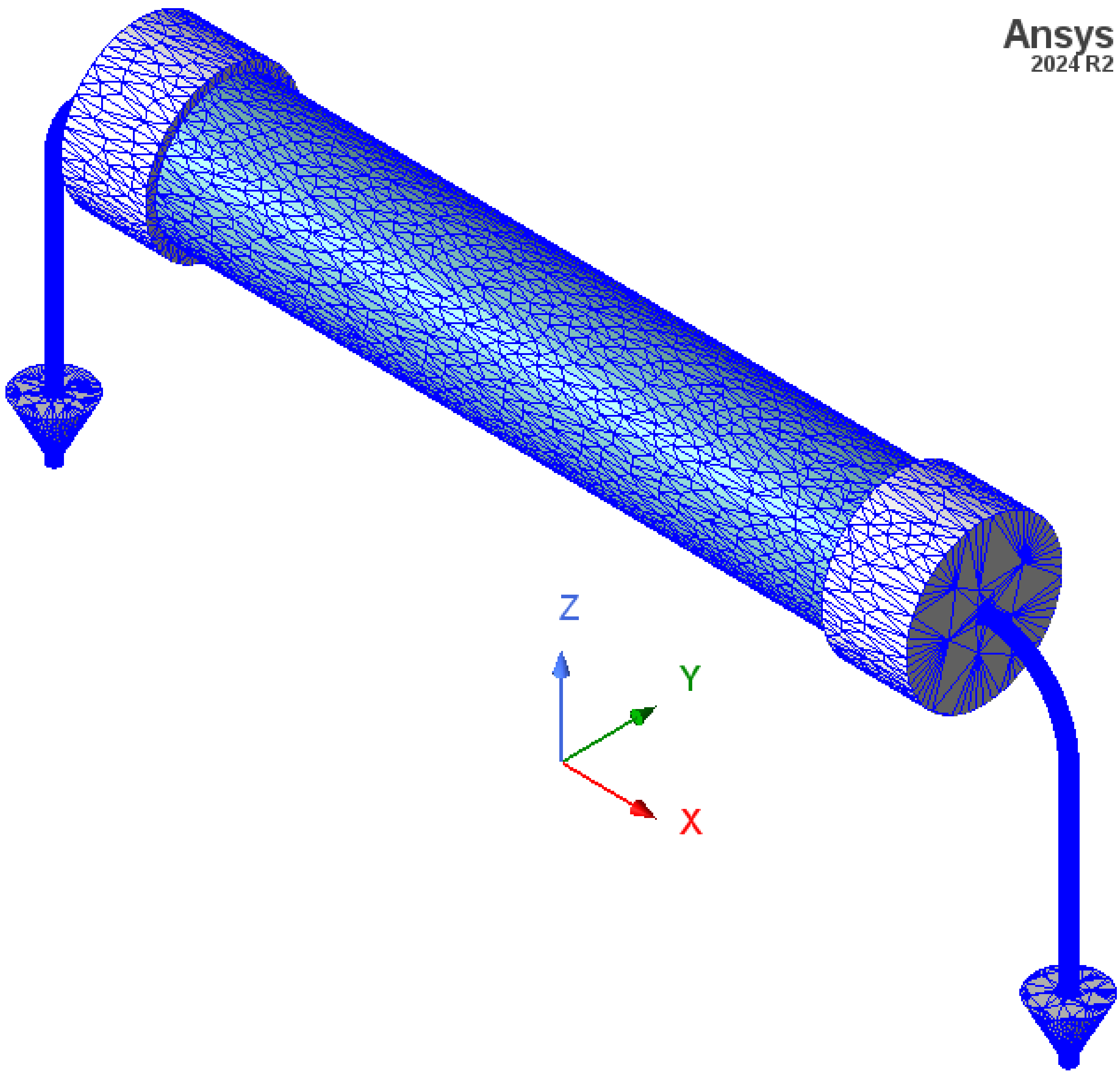


3. Click **Done** to accept the default plot setup and generate the mesh overlay.

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



   Your model should now resemble the following image:



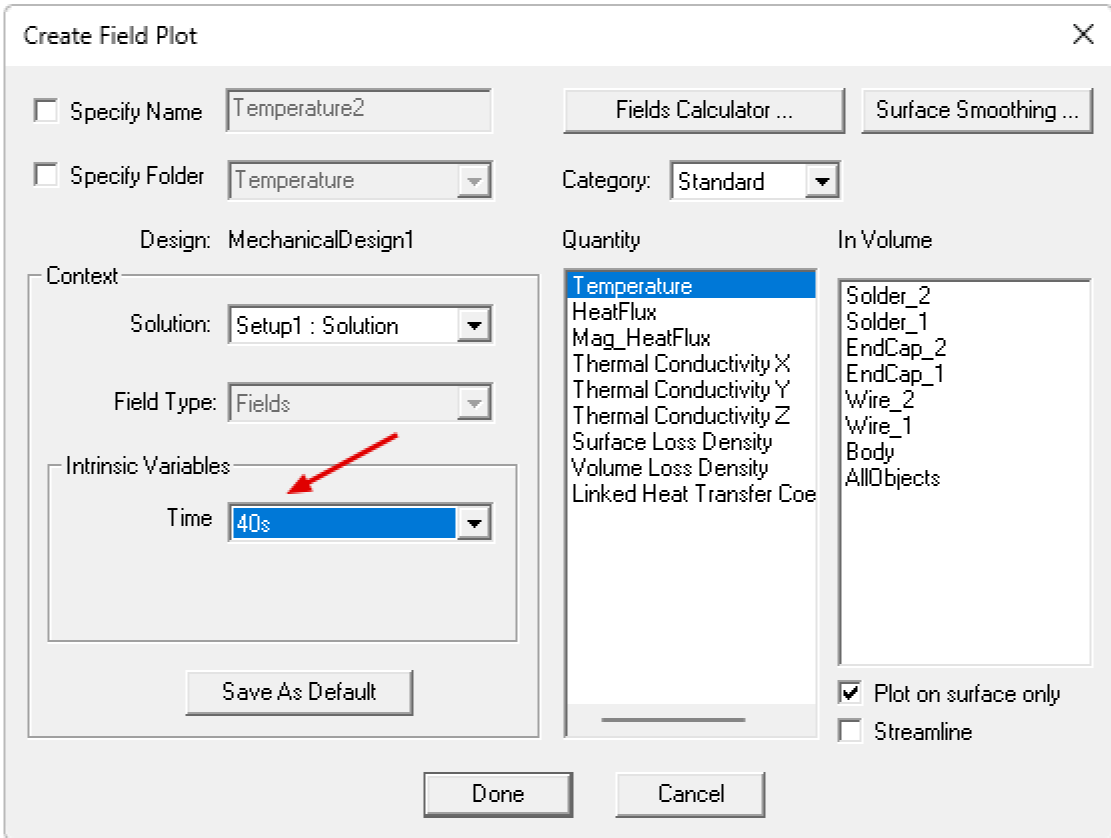
After reviewing the mesh, hide it so that it won't obscure the temperature overlay you will create next.

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

Temperature Overlay

Create an overlay of the temperature results if the last timestep:

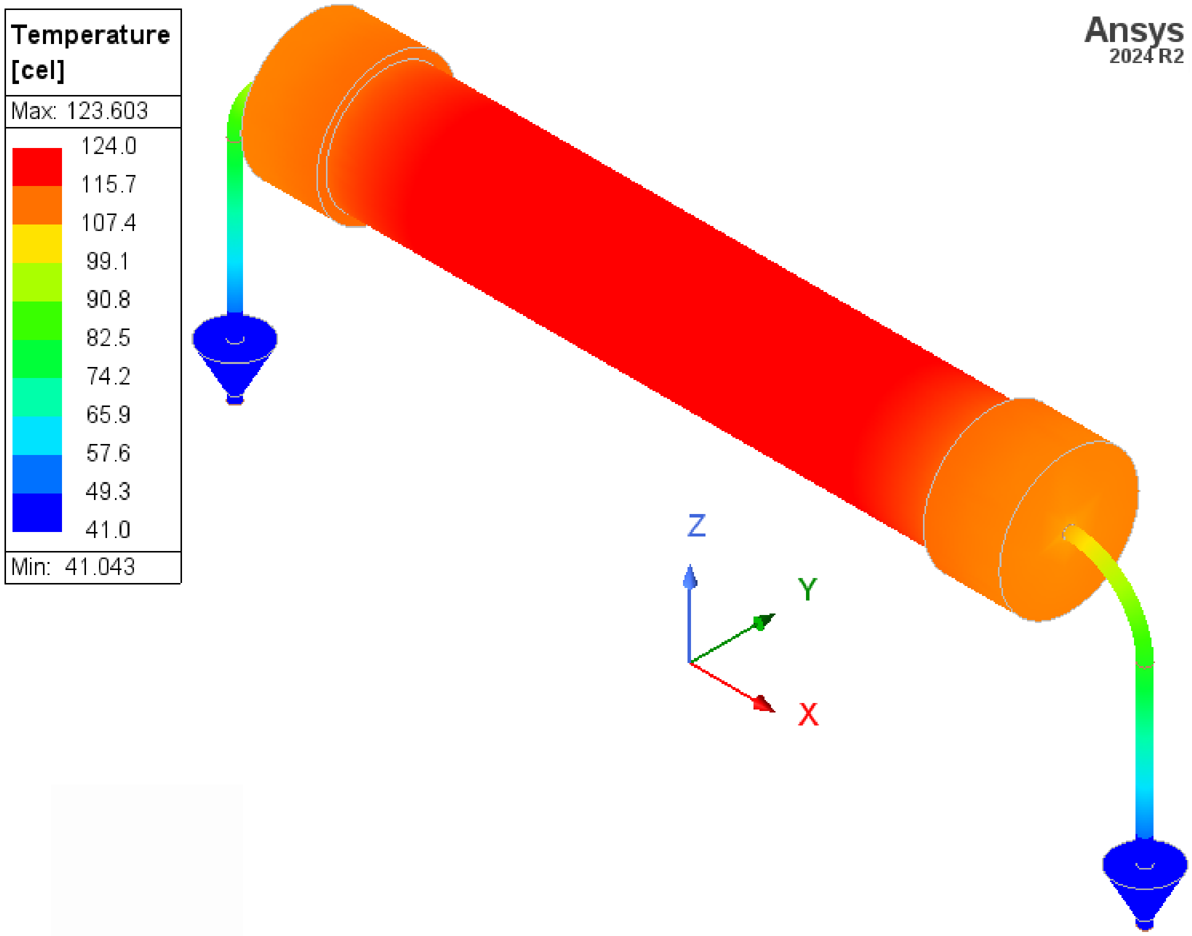
1. Click in the Modeler window to ensure that it is the active window. Then, press **Ctrl+A** to select all objects.
2. Right-click **Field Overlays** and choose **Plot Fields** > **Temperature**. Then, in the *Create Field Plot* dialog box that appears, do the following:
   a. Under *Intrinsic Variables*, choose **40s** from the **Time** drop-down menu.
   b. Ensure that the remaining settings match the following image:



   c. Click **Done**.

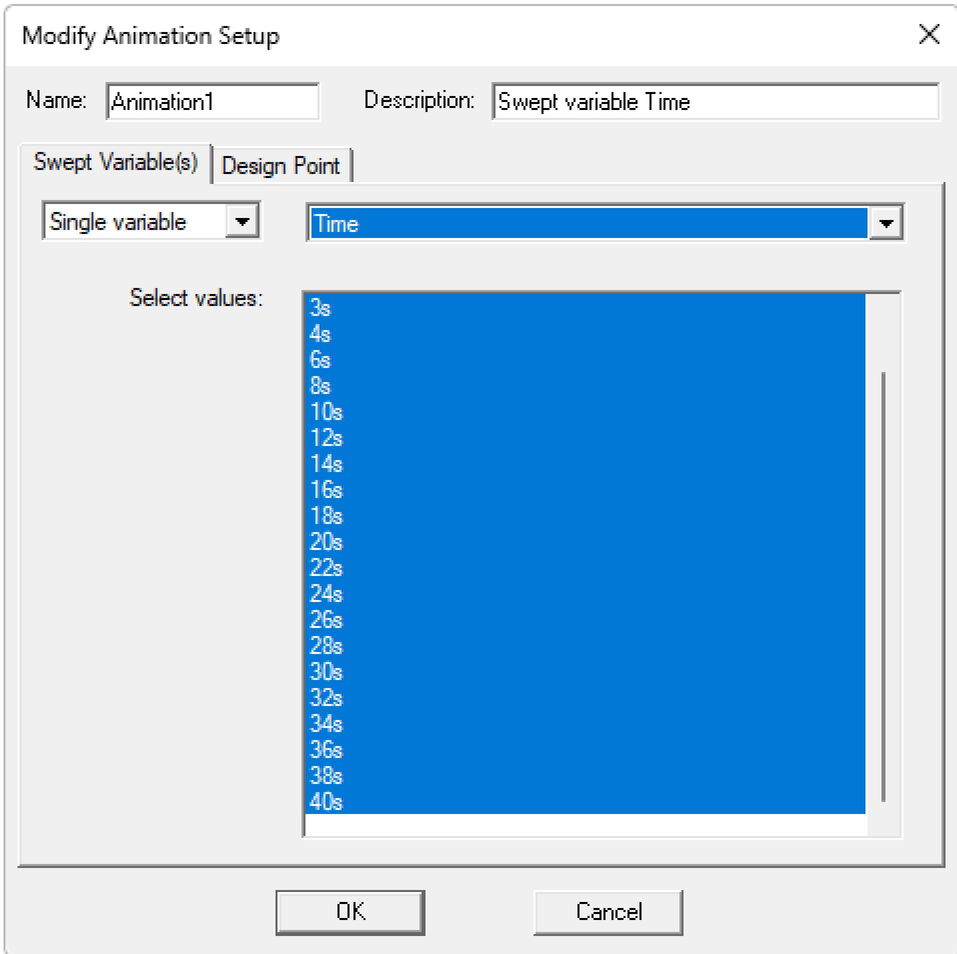
      The color contour overlay and temperature legend appear in the Modeler window.
3. Press **F6** to suppress shading of the CAD surfaces. Doing so improves the appearance of the overlay contours on curved faces.

   Your temperature field overlay should now resemble the following image:



Optionally, continue to the final few steps to animate the temperature overlay.

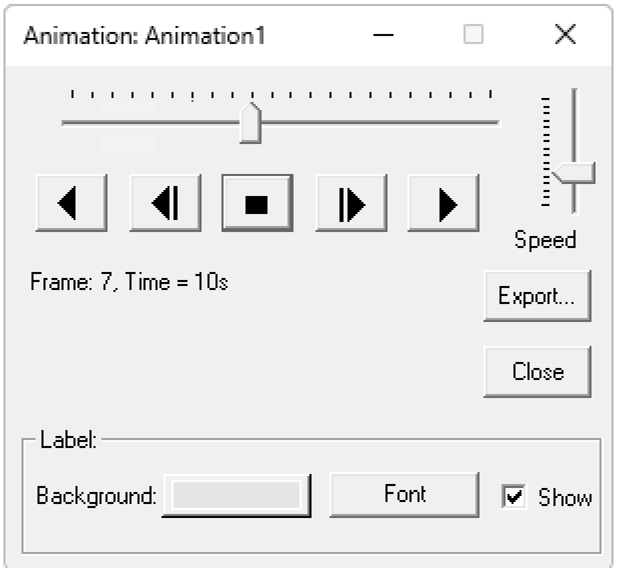
4. Under *Field Overlays* > *Temperature* in the Project Manager, right-click **Temperature1** and choose **Animate** from the shortcut menu.
5. In the *Create Animation Setup* dialog box that appears, ensure that all timesteps are selected:



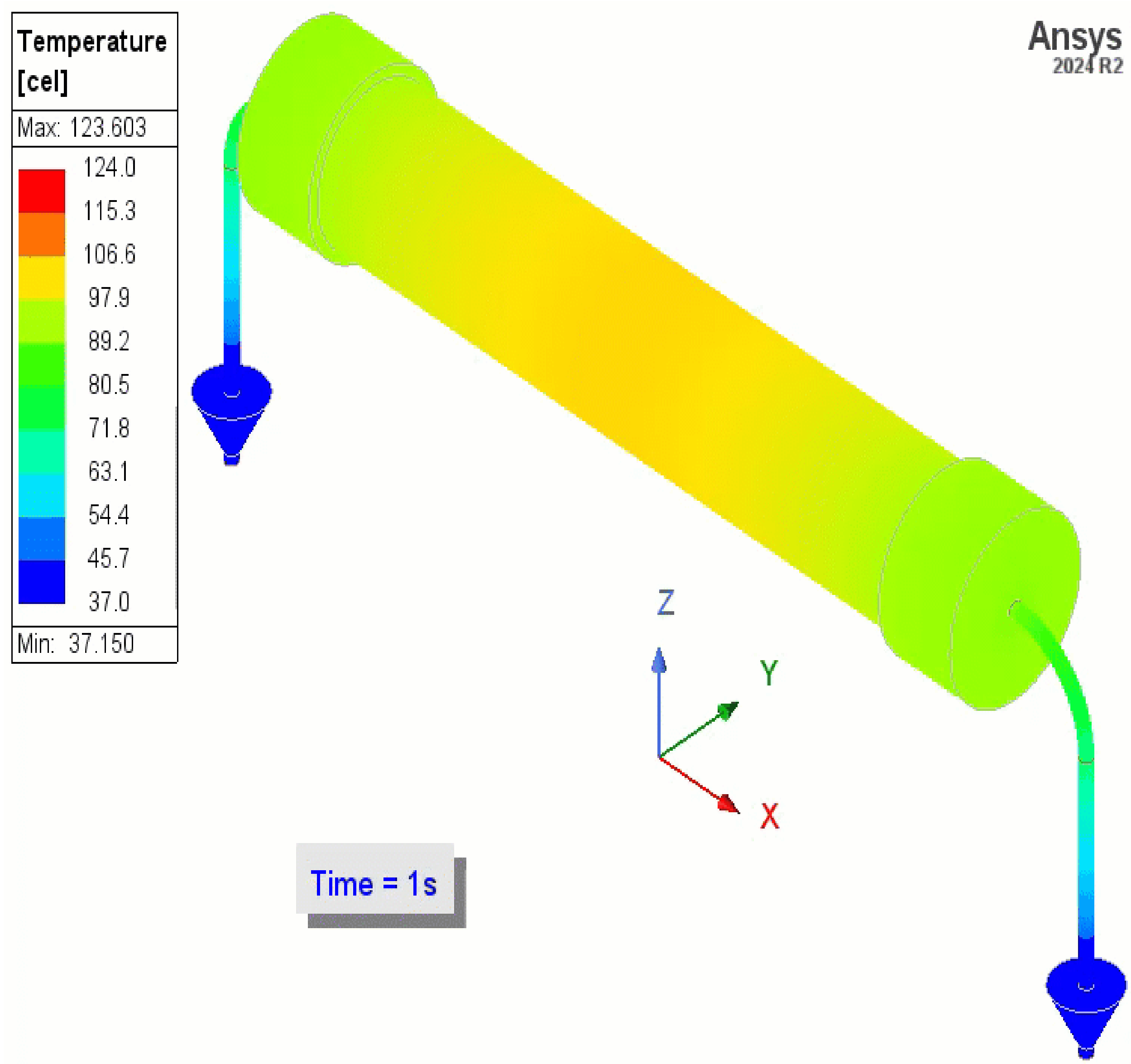
6. Click **OK**.

   It will take a minute or two for the frames to be generated for all timesteps before the animation begins to play.

7. Drag the *Time* annotation to position it as desired. You can also adjust the play speed, start, stop, or reverse the animation using the controls in the *Animation* dialog box:



Your animation should resemble the following one:



8. Click **Close** when you are done reviewing the animation.

Congratulations on completing this transient thermal getting started guide. You can save and close your project now.