Ansys 2025 R1

POWERING INNOVATION THAT DRIVES HUMAN ADVANCEMENT

Getting Started with Icepak®: Finned 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 2025 R1
January 2025

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

This document is intended as supplementary material to Icepak for beginners and advanced users. It how to model a finned heat sink using Ansys Icepak as well as many features and functions essential to any Ansys Icepak project. For the sake of brevity, many of the later tutorials do not cover basic steps or explain the steps in detail as those tutorials assume you have completed this tutorial beforehand.

In this tutorial you will learn how to:

- Create a new project
- Create a model using geometry and thermal boundary conditions
- Generate a mesh for your model
- Set up a simulation with various physical conditions and parameters, including turbulence
- Calculate a solution
- Post-process your results by plotting field overlays of contours and vectors on plane cuts and object geometry

This chapter contains the following topic:

- "Sample Project - The Finned Heat Sink" below

Sample Project - The Finned Heat Sink

The cabinet contains an array of five high-power devices, a backing plate, ten fins, three fans, and a free opening, as shown in the figure below. The fins and backing plate are constructed of extruded aluminum. Each fan has a total volume flow rate of 18 cfm and each source dissipates power at the rate of 33 W. According to the design objective, the base of the devices should not exceed 65°C when air sweeps the fins at an ambient temperature of 20°C.

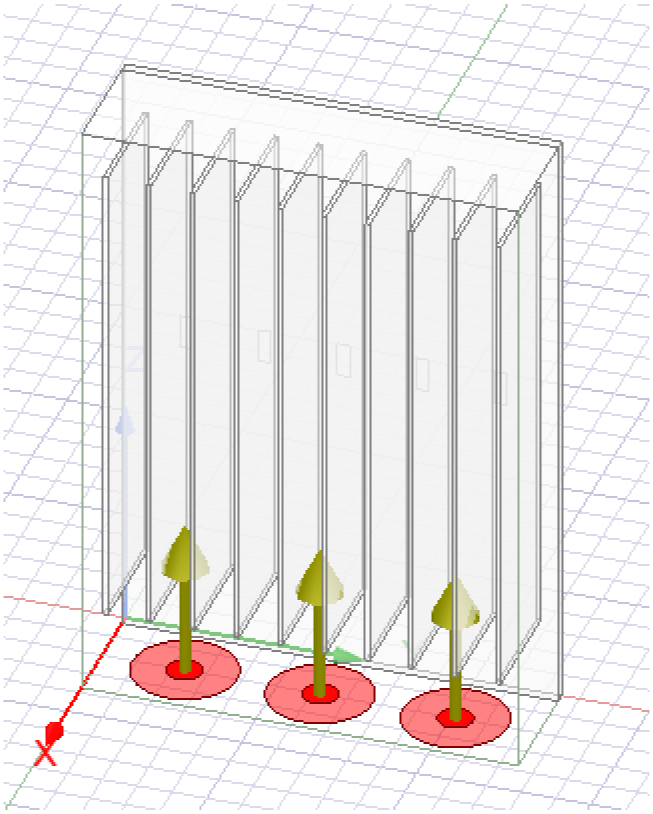


Figure 1-1: Finned Heat Sink

2 - Create a Project and Build the Model

After launching the Ansys Electronics Desktop, create a project, insert an Icepak design, and build the model.

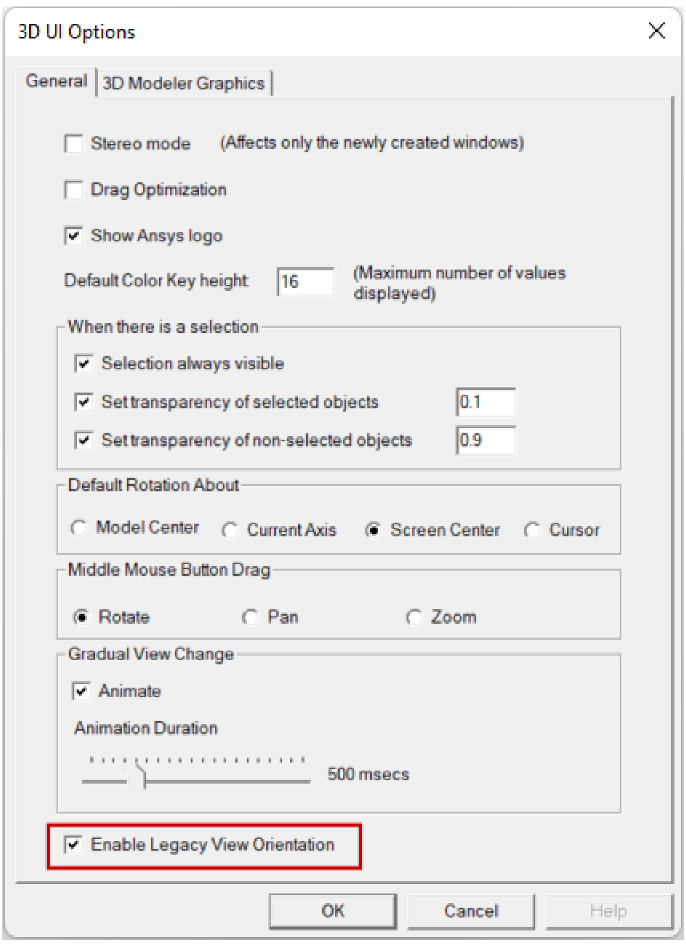
Set 3D UI Options

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

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

   The *3D UI Options* dialog box appears.

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



3. Click **OK**.

Create a Project

1. On the **Desktop** ribbon, click **New**.
2. From the **Project** menu, select **Insert Icepak Design**.
3. In the **Project Manager**, right-click on the project name and select **Rename**.
4. Rename the project "Finned_Heat_Sink."
5. From the **File** menu, select **Save**.
6. Save the project in your working directory.

Build the Model

Resize the Cabinet

1. In the History tree, expand **Model** > **Solids** > **air** > **Region** and select **CreateRegion**.
2. In the **Properties** window, define the following parameters:
    - **+X Padding Type**: Absolute Position
    - **+X Padding Data**: 0.075 meter
    - **-X Padding Type**: Absolute Position
    - **-X Padding Data**: 0 meter
    - **+Y Padding Type**: Absolute Position
    - **+Y Padding Data**: 0.25 meter
    - **-Y Padding Type**: Absolute Position
    - **-Y Padding Data**: 0 meter
    - **+Z Padding Type**: Absolute Position
    - **+Z Padding Data**:0.356 meter
    - **-Z Padding Type**: Absolute Position
    - **-Z Padding Data**: 0 meter

Create the Backing Plate

The backing plate is 0.006 m thick and divides the cabinet into two regions: the device side (where the high-power devices are contained in a housing) and the fin side (where the fins dissipate heat generated by the devices). The backing plate is represented in the model by a solid prism block.

> **Note:** Blocks and conducting thick plates allow six-sided control for meshing and thermal specifications. Conducting thin plates, however, have no physical thickness and therefore allow for only two-sided control.

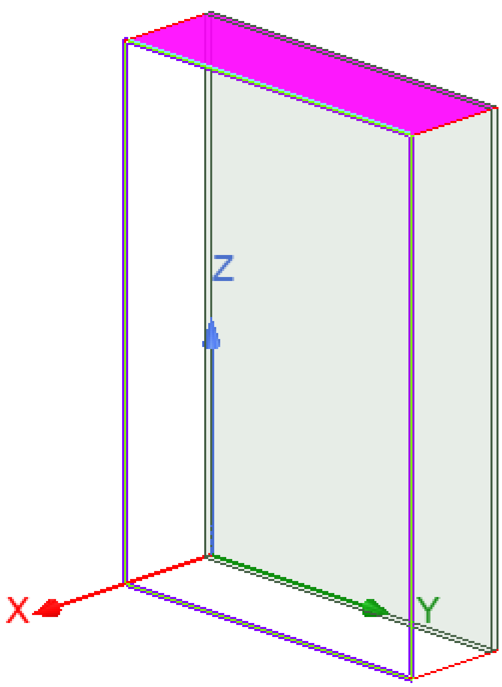
1. From the **Draw** menu, select **Box**.
2. Press **F4** to enter dialog entry mode.

> **Note:** You can toggle between drawing methods by pressing F3 (point mode) or F4 (dialog entry mode).

3. In the **CreateBox** dialog box **Command** tab, define the following size parameters and units:
   - **Position**: 0, 0, 0 meter
   - **XSize**: 0.006 meter
   - **YSize**: 0.25 meter
   - **ZSize**: 0.356 meter
4. On the **Attribute** tab, enter **backing_plate** for the **Name**.
5. Click **OK**.

Create the Free Opening

1. Press **F** to enter face selection mode.
2. In the **3D Modeler** window, select the top face of the Region (cabinet).



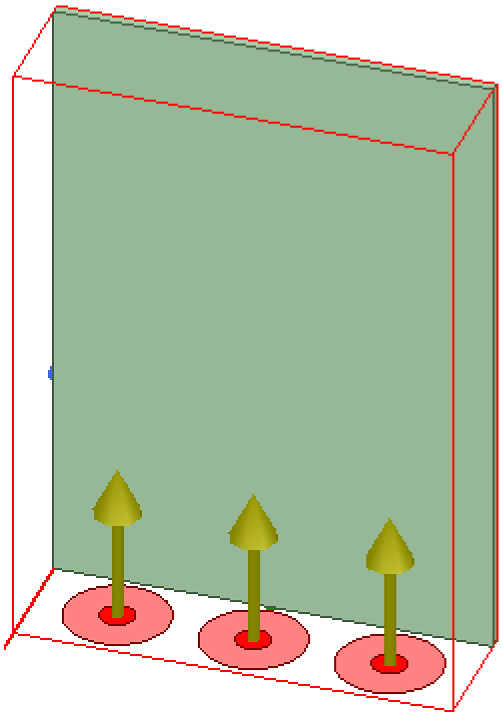
3. Right-click and select **Assign Thermal** > **Opening** > **Free**.
4. In the **Opening Thermal Model** dialog box, enter **FreeOpening** as the **Name**.
5. Click **OK** to accept the default thermal and flow specifications.

Create the Fans

Each fan is physically identical to the others, except with respect to its location on the cabinet wall. To create the set of three fans, you will build a single fan as a template and then create two copies, each with a specified offset in the Y direction.

1. In the **Project Manager**, right-click on **3D Components** and select **Create** > **Fan**.
2. In the **Fan Component** dialog box **Geometry** tab, define the following parameters:
   - **Radius**: 0.03 meter
   - **Hub Radius**: 0.01 meter
3. On the **Properties** tab, define the following parameters:
   - **Flow Type**: Fixed Volumetric - 18 cfm
4. Click **OK**.
5. In the History tree, expand **Fan1**.
6. Right-click **Fan1_1** and select **Edit** > **Arrange** > **Move**.
7. In the **Move** dialog box, enter **0.04, 0.0475, 0** for the **Move Vector Value**.
8. Change the **Unit** to **meter**.
9. Click **OK**.
10. In the History tree, right-click **Fan1_1** and select **Edit** > **Duplicate** > **Along Line**.
11. In the **DuplicateAlongLine** dialog box, enter **3** for the **Total Number**.
12. Enter **0 ,0.0775 ,0** for the **Vector**.
13. Change the **Unit** to **meter**.

14. Click **OK**.

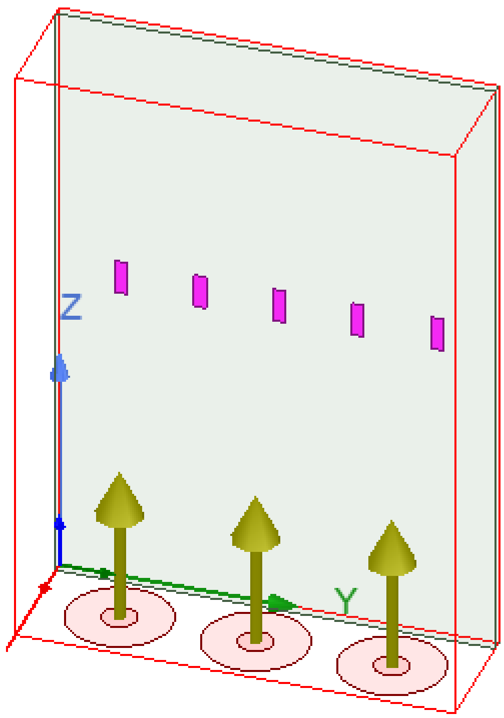


Create the High Power Devices

Like the fans, each device is physically identical to the others, except with respect to its location. To create the set of five devices, you will build a single rectangular planar source as a template and then create four copies, each with a specified offset in the Y direction.

1. From the **Draw** menu, select **Rectangle**.
2. In the **CreateRectangle** window **Command** tab, define the following size parameters and units:
   - **Position**: 0, 0.0315, 0.1805 meter
   - **Axis**: X
   - **XSize**: 0.007 meter
   - **ZSize**: 0.02 meter
3. On the **Attribute** tab, enter **HighPowerDevice1** as the **Name**.
4. In the History tree, right-click **HighPowerDevice1** and select **Assign Thermal** > **Source**.
5. In the **Source Thermal Model** dialog box, enter **33 W** for the **Total Power**.
6. Click **OK**.
7. In the History tree, right-click **HighPowerDevice1** and select **Edit** > **Duplicate** > **Along Line**.
8. In the **DuplicateAlongLine** dialog box, enter **5** for **Total Number**.
9. Enter **0, 0.045, 0** for the **Vector**.

10. Change the **Unit** to **meter**.
11. Click **OK**.

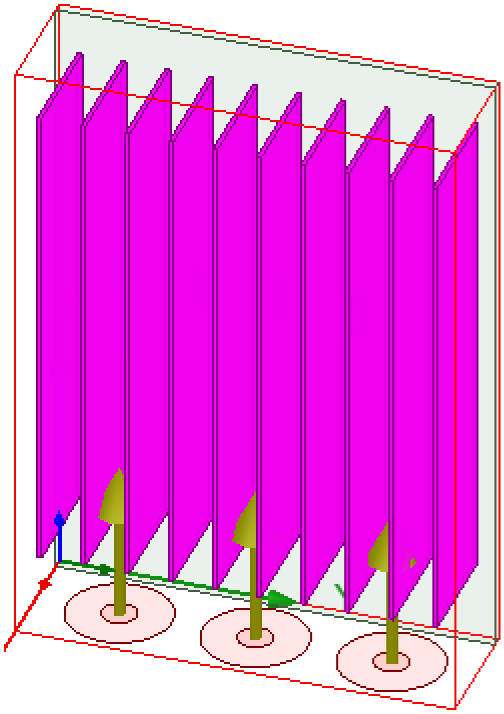


Create the Fins

Like the fans and devices, each fin is physically identical to the others, except with respect to its location in the cabinet. To create the array of ten fins, create a rectangular plate as a template. Then create nine copies, each with a specified offset in the Y direction.

1. From the **Draw** menu, select **Box**.
2. In the **CreateBox** dialog box, define the following size parameters and units:
   - **Position**: 0.006 ,0.0125 ,0.05 meter
   - **XSize**: 0.069 meter
   - **YSize**: 0.0025 meter
   - **ZSize**: 0.281 meter
3. Click **OK**.
4. In the History tree, right-click **Box1** and select **Edit** > **Duplicate** > **Along Line**.
5. In the **DuplicateAlongLine** dialog box, enter **10** for **Total Number**.
6. Enter **0, 0.025, 0** for the **Vector**.
7. Change the **Unit** to **meter**.

8. Click **OK**.



Review Model Components

The Design List displays information about the model components.

1. From the **Icepak** menu, select **List**.
2. Review the component information in the **Design List** dialog box.

> **Note:** Select one or more components in the Design List to select them in the History tree and **3D Modeler** window.

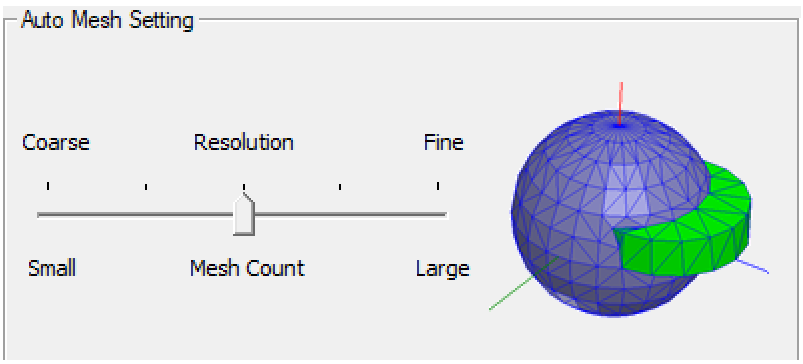
3. From the **File** menu, select **Save**.

3 - Generate and Display Mesh

After building the model, generate and display the mesh.

Generate a Coarse Mesh

1. In the **Project Manager**, right-click **Mesh** and select **Edit Global Region**.
2. On the **Advanced** tab, disable the **User specified** check box.
3. On the **General** tab under **Auto Mesh Setting**, ensure the slider bar is in the middle between **Coarse/Small** and **Fine/Large** position.

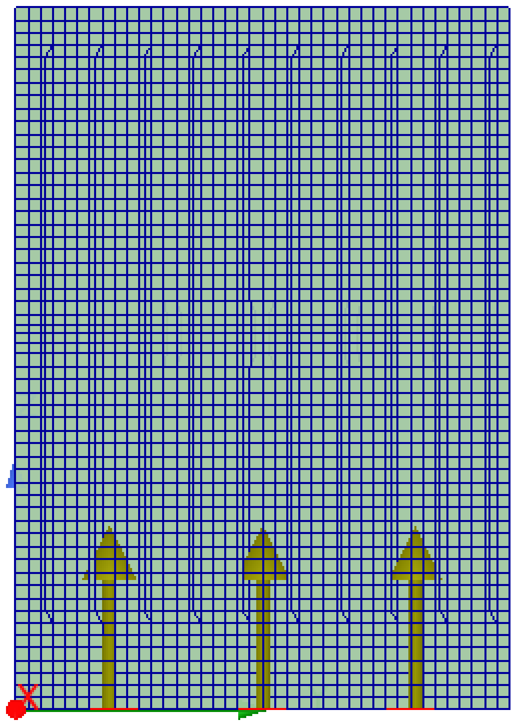


4. Click **OK**.
5. In the **Project Manager**, right-click **Mesh** and select **Generate Mesh**.

Display the Coarse Mesh

The **Mesh Visualization** dialog box automatically appears after the meshing process completes.

1. In the **Mesh Visualization** dialog box under **Mesh display on**, enable **Show** and select **Geometry/Boundary selection**.
2. In the History tree, right-click **Model** and select **Select All**.



3. Click **Close**.

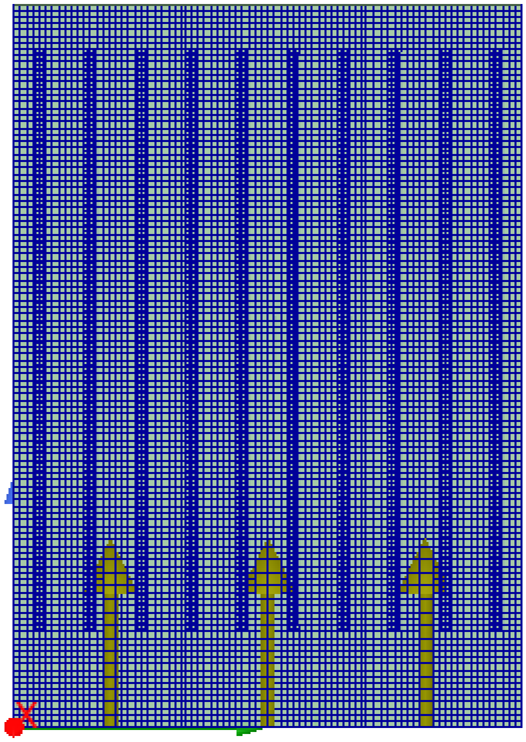
Generate a Fine Mesh

1. In the **Project Manager**, right-click **Mesh** and select **Edit Global Region**.
2. On the **General** tab under **Auto Mesh Setting**, move the slider bar to the **Fine/Large** position.
3. Click **OK**.
4. In the **Project Manager**, right-click **Mesh** and select **Generate Mesh**.

Display the Fine Mesh

The **Mesh Visualization** dialog box automatically appears after the meshing process completes.

1. In the **Mesh Visualization** dialog box under **Mesh display on**, enable **Show** and select **Geometry/Boundary selection**.
2. In the History tree, right-click **Model** and select **Select All**.



3. Click **Close**.

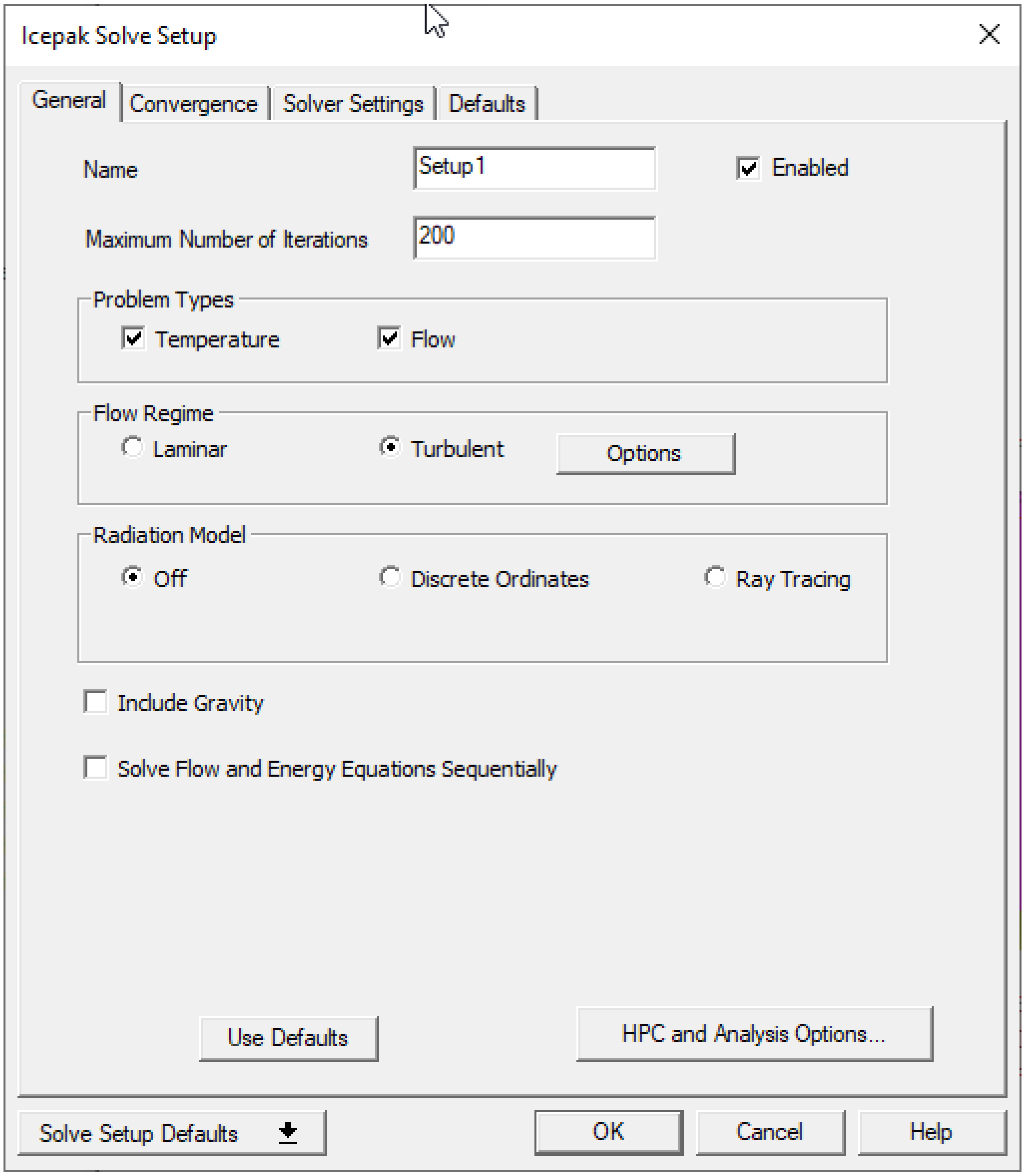
4 - Define the Simulation Settings and Run the Analysis

Define the Design Settings

1. From the **Icepak** menu, select **Design Settings**.
2. On the **Icepak Design Settings** dialog box **Gravity** tab, select **Global::Y** and **Negative** to define the force of gravity in the negative Y direction based on the Global coordinate system.
3. Click **OK**.

Add a Solution Setup

1. In the **Project Manager**, right-click **Analysis** and select **Add Solution Setup**.
2. On the **Icepak Solve Setup** dialog box **General** tab, change the **Maximum Number of Iterations** to **200**.
3. Under **Problem Types**, retain the selections to solve for both **Temperature** and **Flow**.
4. Under **Flow Regime**, select **Turbulent** and click **Options**. In the **Turbulent Flow Model** dialog box, retain the **Zero Equation** selection and click **OK**.
5. Under **Radiation Model**, retain the **Off** selection to ignore heat transfer due to radiation.

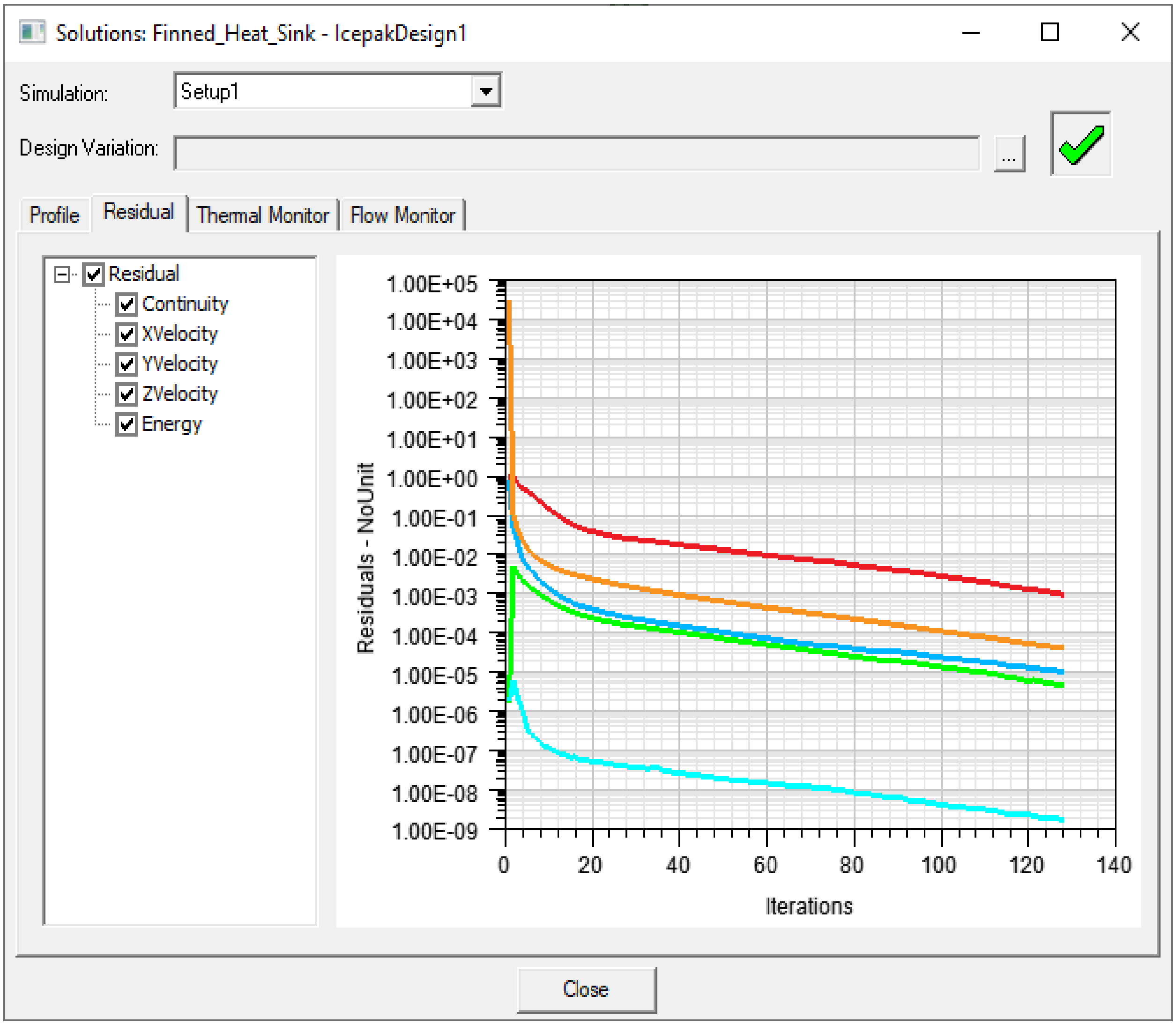


6. Click **OK**
7. From the **File** menu, click **Save**.

Run the Simulation

1. In the **Project Manager**, expand **Analysis**.
2. Right-click **Setup1** and select **Analyze**.

3. Right-click **Setup1** and select **Residuals** to monitor the simulation residuals.



Note: When the simulation is complete, a message is displayed in the **Messages** window stating "Normal completion of simulation on server."

5 - Post-process the Results

The Ansys Electronics Desktop provides a number of ways to view and examine the solution results, including:

- Plane cut views
- Object face views
- Fields summary report

> **Note:** The objective of this exercise is to determine whether the air flow and heat transfer associated with the heat sink (fans and fins) are sufficient to maintain device temperatures below 65°C. You can accomplish this by creating different plane cuts and monitoring the velocity vector and temperature on it. Plane-cut views allow you to observe the variation in a solution variable across the surface of a plane.

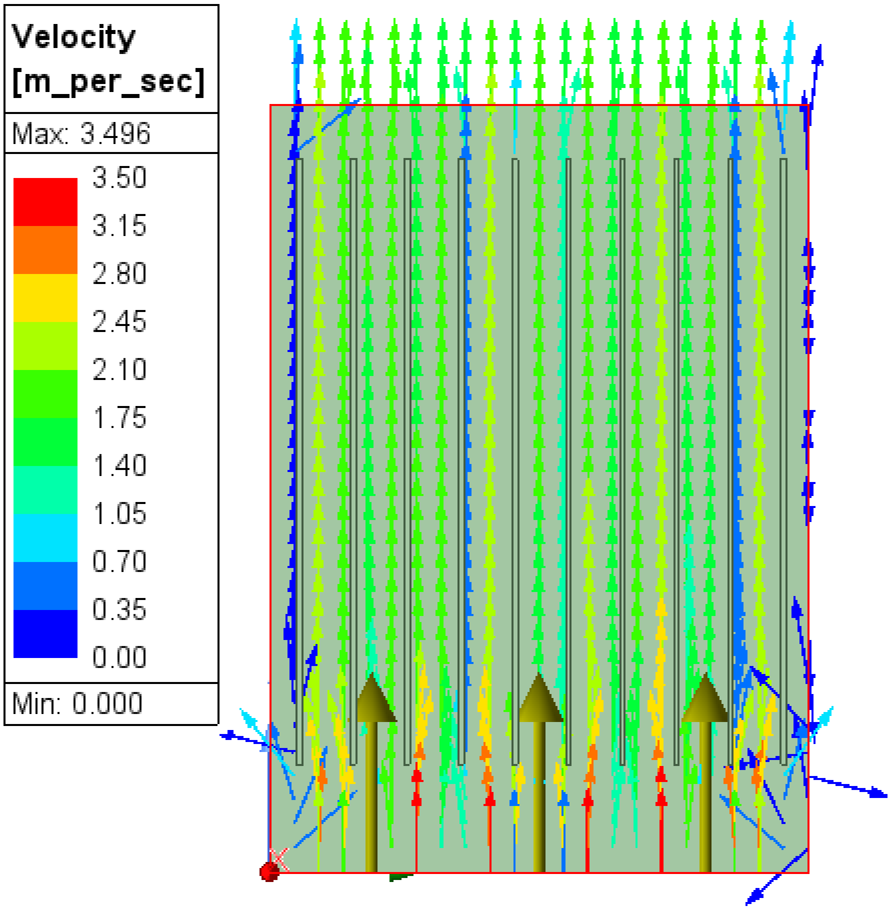
Create Plane Cut Field Overlays

Create a Plane

1. From the **Draw** menu, select **Plane**.
2. In the **3D Modeler** window, click twice to draw a plane.
3. In the History tree, expand **Planes** and select **Plane1**.
4. In the **Properties** window, edit the following properties:
   - **Name**: cut-plane
   - **Root point**: 0.0375 ,0.125 ,0.178 meter
   - **Normal**: 1 ,0 ,0 meter

Plot Speed

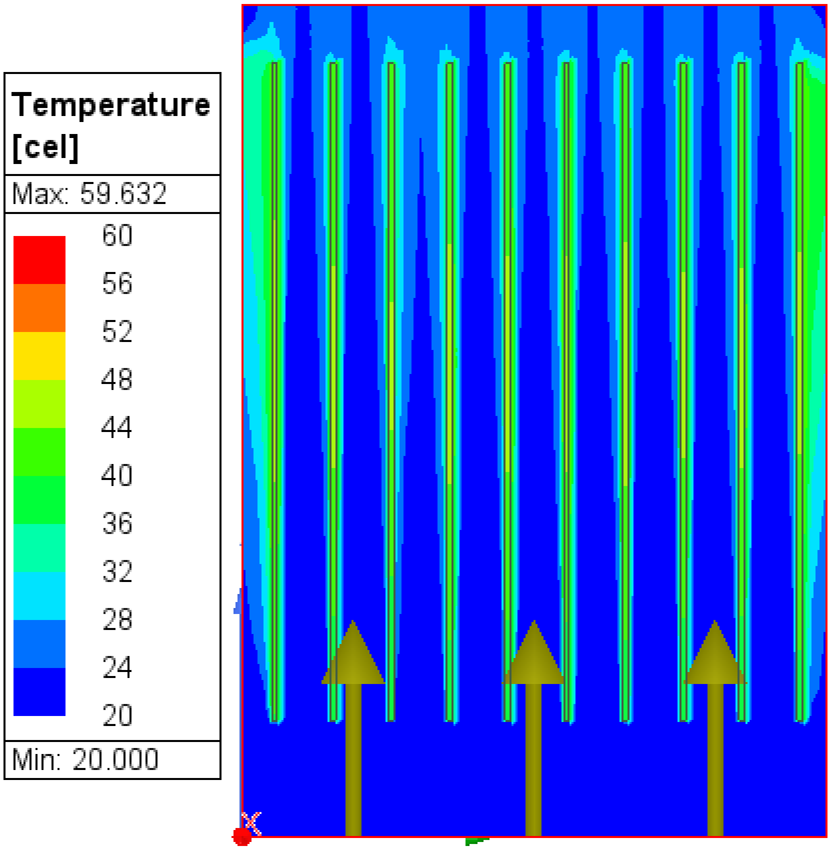
1. In the History tree, select **cut-plane**.
2. Right-click in the **3D Modeler** window and select **Plot Fields** > **Velocity** > **Velocity Vectors**.
3. In the **Create Field Plot** dialog box, enable **Specify Name** and enter **cut-velocity**.
4. Retain the **Velocity Vectors** selection for **Quantity** and click **OK**.



5. In the **Project Manager**, expand **Field Overlays** > **Velocity**.
6. Right-click **cut-velocity** and select **Plot Visibility** to hide the field plot.

Plot Temperature

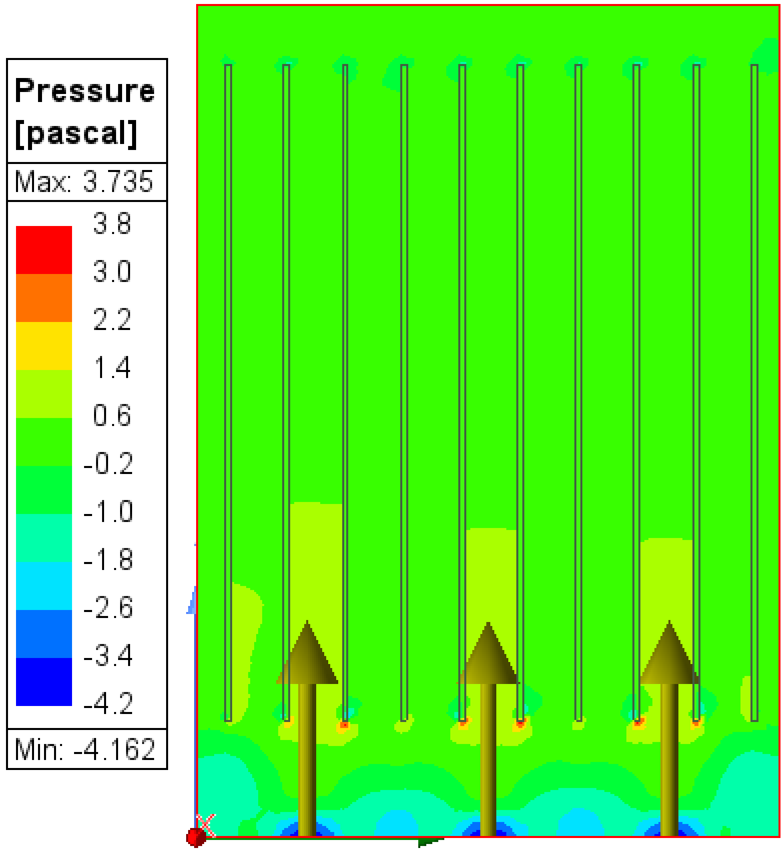
1. In the History tree, select **cut-plane**.
2. Right-click in the **3D Modeler** window and select **Plot Fields** > **Temperature** > **Temperature**.
3. In the **Create Field Plot** dialog box, enable **Specify Name** and enter **cut-temperature**.
4. Retain the **Temperature** selection for **Quantity** and click **OK**.



5. In the **Project Manager**, expand **Field Overlays** > **Velocity**.
6. Right-click **cut-velocity** and select **Plot Visibility** to hide the field plot.

Plot Pressure

1. In the History tree, select **cut-plane**.
2. Right-click in the **3D Modeler** window and select **Plot Fields** > **Pressure** > **Pressure**.
3. In the **Create Field Plot** dialog box, enable **Specify Name** and enter **cut-pressure**.
4. Retain the **Pressure** selection for **Quantity** and click **OK**.



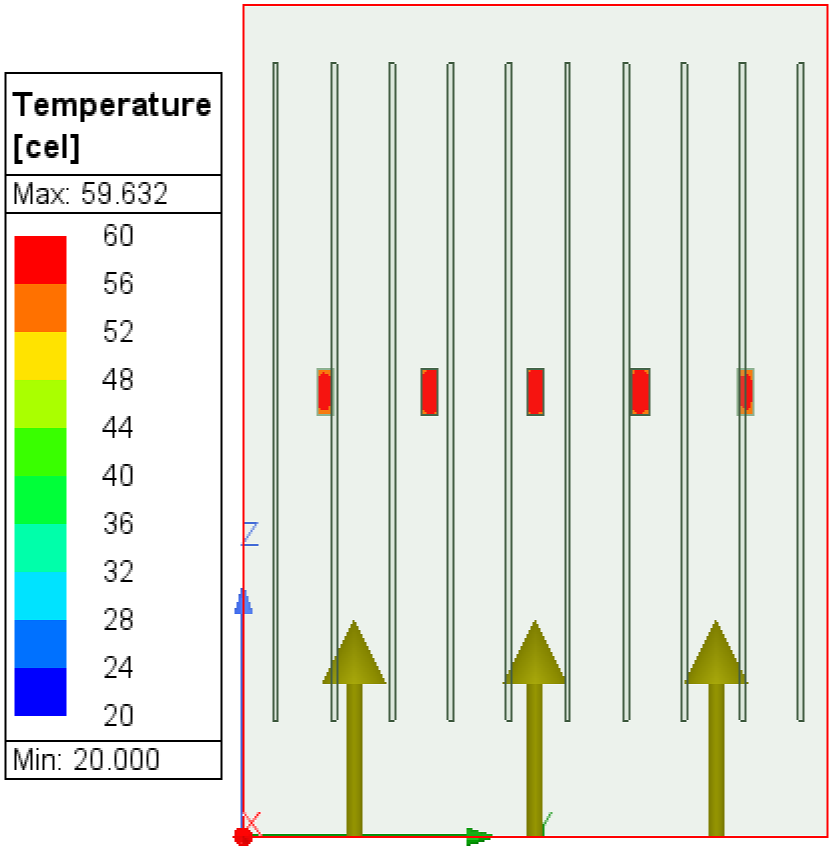
5. In the **Project Manager**, expand **Field Overlays** > **Pressure**.
6. Right-click **cut-pressure** and select **Plot Visibility** to hide the field plot.

Create Object Field Overlays

Plot Temperature on High Power Devices

1. In the History tree, expand **Sources** and select all five high power devices.
2. Right-click in the **3D Modeler** window and select **Plot Fields** > **Temperature** > **Temperature**.
3. In the **Create Field Plot** dialog box, enable **Specify Name** and enter **highpowerdevice-temperature**.

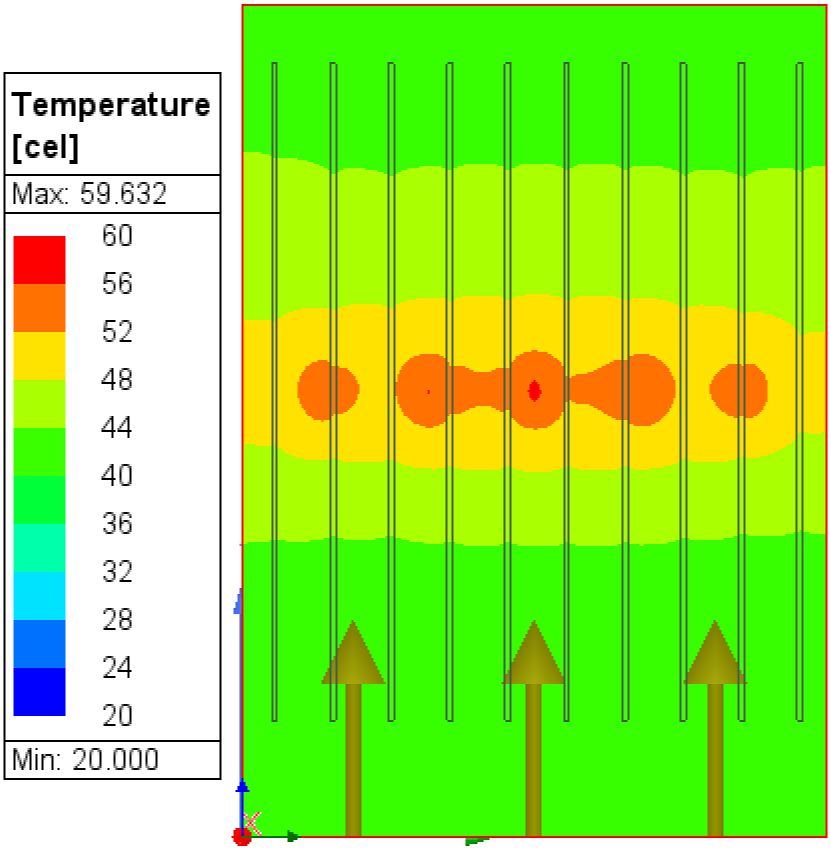
4. Retain the **Temperature** selection for **Quantity** and click **OK**.



Plot Temperature on the Backing Plate

1. In the History tree, expand **Al-Extruded** and select **backing_plate**.
2. Right-click in the **3D Modeler** window and select **Plot Fields** > **Temperature** > **Temperature**.
3. In the **Create Field Plot** dialog box, enable **Specify Name** and enter **backingplate-temperature**.
4. Retain the **Temperature** selection for **Quantity**.

5. Enable **Plot on surface only** and click **OK**.



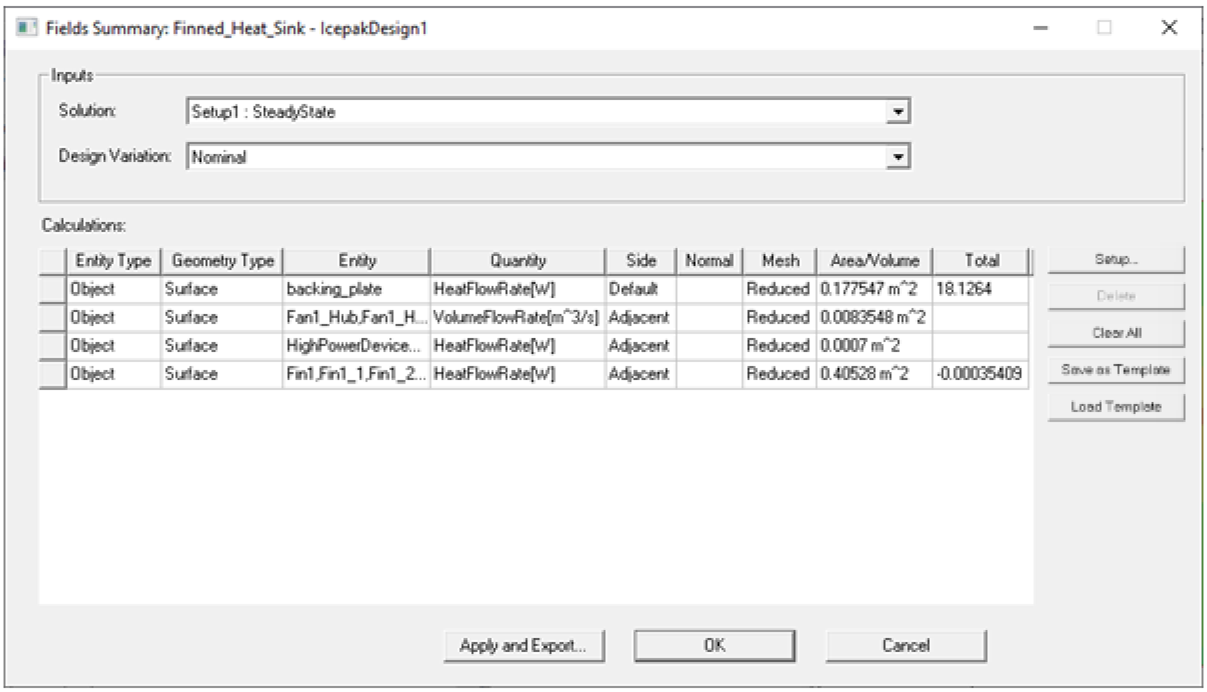
Create a Fields Summary Report

Create a fields summary report of object-specific solution data. Fields summary reports can provide physical information from the solution about specific model boundary conditions or objects.

1. From the **Icepak** menu, select **Fields** > **Create Fields Summary**.
2. In the **Setup Calculation** dialog box, select **Object** for **Entity Type**.
3. In the **Entity** list, select **backing_plate**.
4. In the **Quantity** list, select **HeatFlowRate**.
5. From the **Add** drop-down list, select **Add As Single Calculation**.
6. Repeat steps 3 through 5 for the following entities and quantities.

> **Note:** You can select multiple entities at once. When an entity is selected, select it again to clear the selection.

Entity	Quantity
Fan1_Hub, Fan1_Hub2, Fan1_Hub3, Fan1_Passage, Fan1_Passage2, Fan1_Passage3	VolumeFlowRate
HighPowerDevice1, HighPowerDevice1_1, HighPowerDevice1_2, HighPowerDevice1_3, HighPowerDevice1_4	HeatFlowRate
Fin1, Fin1_1, Fin1_2, Fin1_3, Fin1_4, Fin1_5, Fin1_6, Fin1_7, Fin1_8, Fin1_9	HeatFlowRate



6 - Summary

In this tutorial, you have determined the ability of the specified heat sink to maintain source temperatures below 65°C. Post-processing results show that the maximum source temperature is about 60°C, indicating that the heat sink provides adequate cooling for the sources. In addition, you have learned the basic workflow of an Icepak project, including model building, mesh generation, problem setup, solution calculation, and post-processing as well essential features and functions that you will likely use in later tutorials or your own projects.