Ansys

2025 R2

POWERING INNOVATION THAT DRIVES HUMAN ADVANCEMENT

Getting Started with Icepak®: Optimization of Fan Location

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 R2
July 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 demonstrates and optimization features with the help of a small system-level model.

In this tutorial you will learn how to:

- Use network blocks as one way of modeling packages.
- Specify a contact resistance using side specifications of a block object.
- Define a variable as a parameter and solve the parametric trials to optimize your model for maximum performance.
- Specify fan curves and dynamically update them.
- Use local coordinate systems.
- Generate a fields summary report for multiple parametric solutions.

This chapter contains the following topic:

- "Sample Project - The System-Level Model" below

Sample Project - The System-Level Model

The system-level model consists of a series of IC chips on a PCB. A fan is used for forced convection cooling of the power dissipating devices. A bonded fin extruded heat sink with eight 0.008 m thick fins is attached to the IC chips. The fan flow rate is defined by a nonlinear fan curve. The system also consists of a perforated thin grille. A study is carried out for the optimum location of the fan by using the parameterization feature in Icepak.

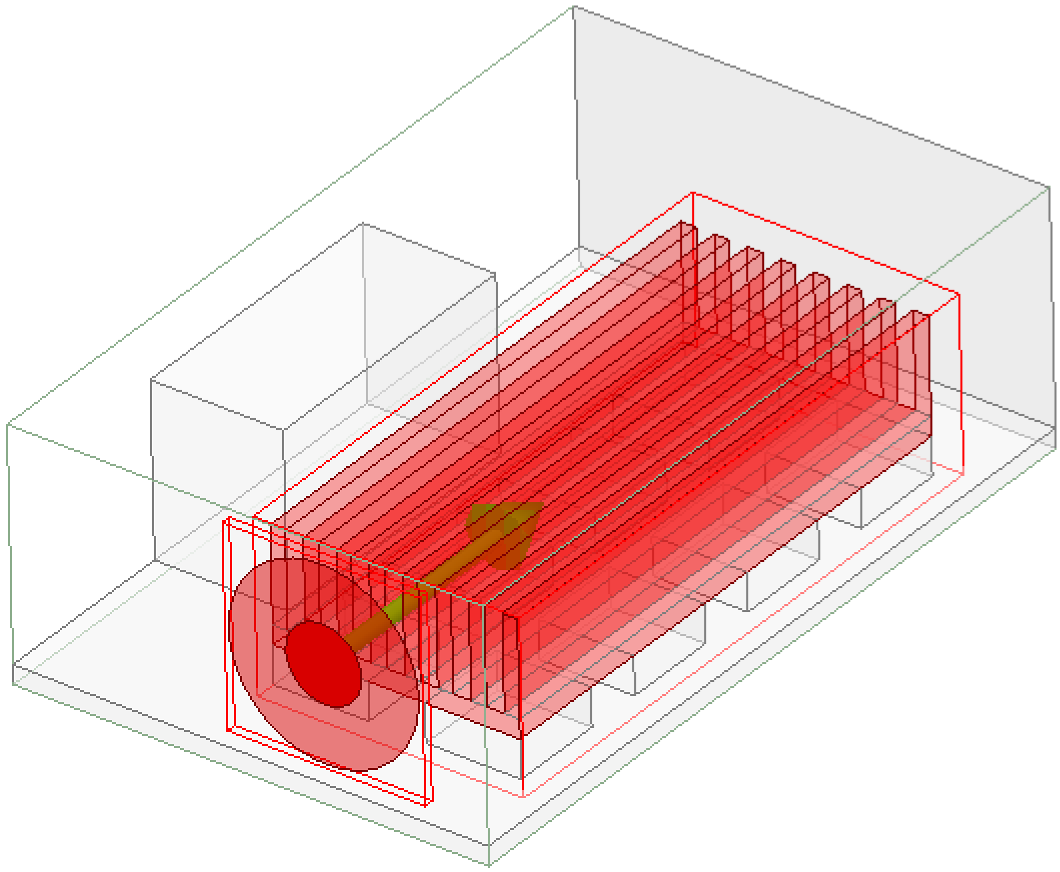


Figure 1-1: System-Level Model

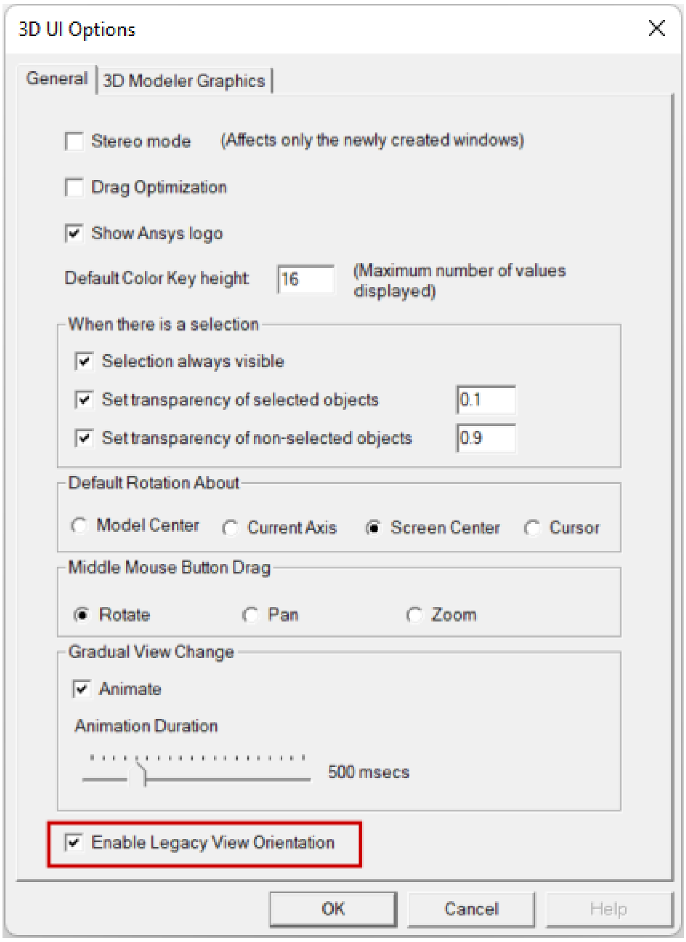
Set 3D UI Options

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:


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**.

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.

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 "Parameterize_Fan."
5. From the **File** menu, select **Save**.
6. Save the project in your working directory.

Define Options

The model in this project primarily uses meters as the length unit. Also, Icepak provides two methods of presenting the user interface when creating native 3D components and boundary conditions: a wizard or a tabbed interface. The instructions in this guide are written for the tabbed interface. In the options, define the default length unit and set the component and boundary condition interface option.

1. From the **Modeler** menu, select **Units**.
2. In the **Set Modeler Units and Max Extent** dialog box, select meter from the **Select units** drop-down and click **OK**.
3. From the **Tools** menu, select **Options** > **General Options**.
4. Expand **Icepak** and click **Thermal**.
5. Disable **Use Wizards for data input when creating new boundaries**.
6. Click **OK**.

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**: 0.4 meter
    - **-X Padding Type**: Absolute Position

- **-X Padding Data**: 0 meter
- **+Y Padding Type**: Absolute Position
- **+Y Padding Data**: 0.13 meter
- **-Y Padding Type**: Absolute Position
- **-Y Padding Data**: 0 meter
- **+Z Padding Type**: Absolute Position
- **+Z Padding Data**:0.25 meter
- **-Z Padding Type**: Absolute Position
- **-Z Padding Data**: 0 meter

Define a Variable and Create the Fan

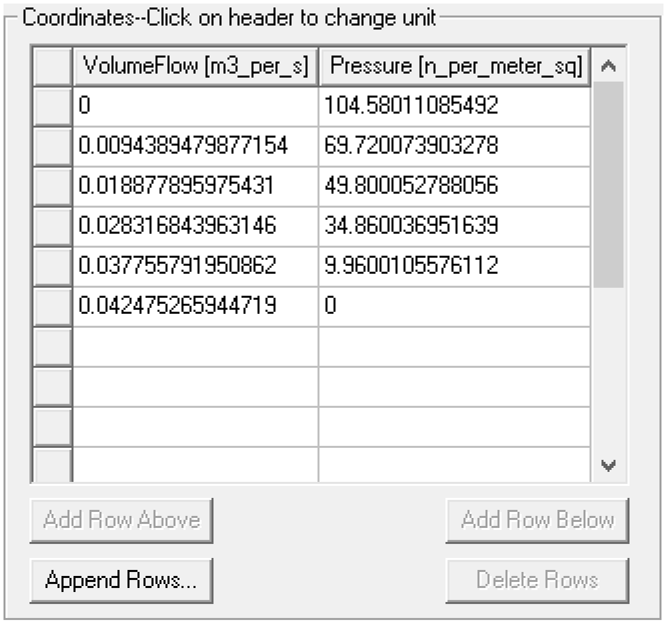
First define a variable to be used for the location of the fan and then create the fan component.

Define a Variable

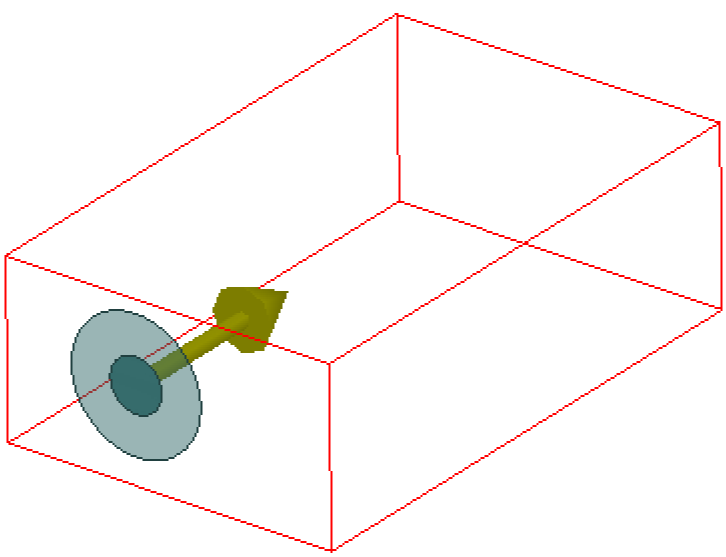
1. From the **Project** menu, select **Project Variables**.
2. In the **Properties** dialog box, click **Add**.
3. In the Add Property dialog box, define the following parameters:
   - **Name**: $zc
   - **Unit Type**: Length
   - **Units**: meter
   - **Value**: 0.1
4. Click **OK** to close the **Add Property** dialog box.
5. Click **OK** to close the **Properties** dialog box.

Create the Fan Component

1. In the **Project Manager**, right-click **3D Components** and select **Create** > **Fan**.
2. In the **Fan Component** dialog box **Geometry** tab, define the following parameters:
   - **Cross-section**: **YZ**
   - **Radius**: 0.05 meter
   - **Hub Radius**: 0.02 meter
3. On the **Properties** tab, retain the **Flow Type** selection of **Curve** and click **Edit Curve**.
4. In the **Edit Dataset** dialog box, enter the values displayed in the following image.



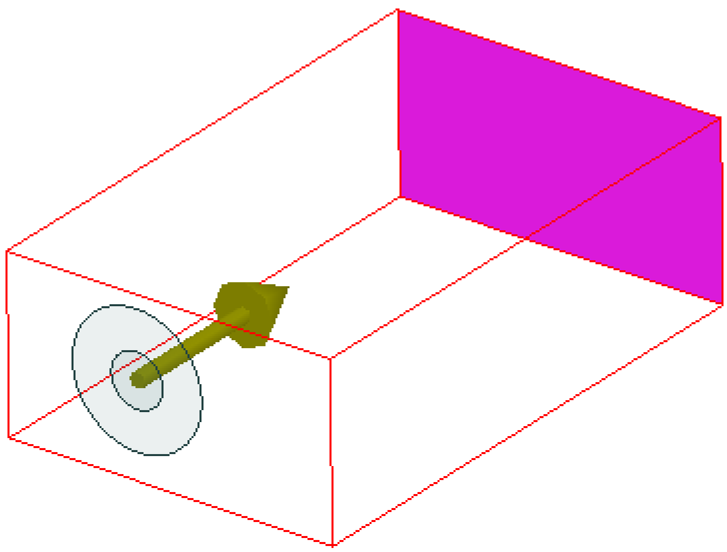
5. Click **OK** to close the **Edit Dataset** dialog box.
6. Under **Swirl**, select **RPM** and enter 4000 for **RPM** and 2000 for **Operating RPM**.
7. Click **OK**.
8. In the History tree, expand **Model** > **Fan1**.
9. Right-click **Fan1_1** and select **Edit** > **Arrange** > **Move**.
10. Press **F4** to display the **Move** dialog box.
11. In the **Move** dialog box, enter **0 ,0.07 ,0.1** for the **Move Vector Value**.
12. Replace **0.1** with the variable **$zc**.
13. Click **OK**.



> **Note:** To change the color of the fan geometry, expand **Model** > **Fan1** > **Fan1_1** > **Sheets** > **Unassigned** and select both **Fan1_Hub** and **Fan1_Passage**. In the **Properties** window, click the current color in the **Value** cell and select the new color in the in the **Color** window.

Create the Grille

1. Press **F** to enter face selection mode.
2. In the **3D Modeler** window, select the cabinet face opposite the fan.



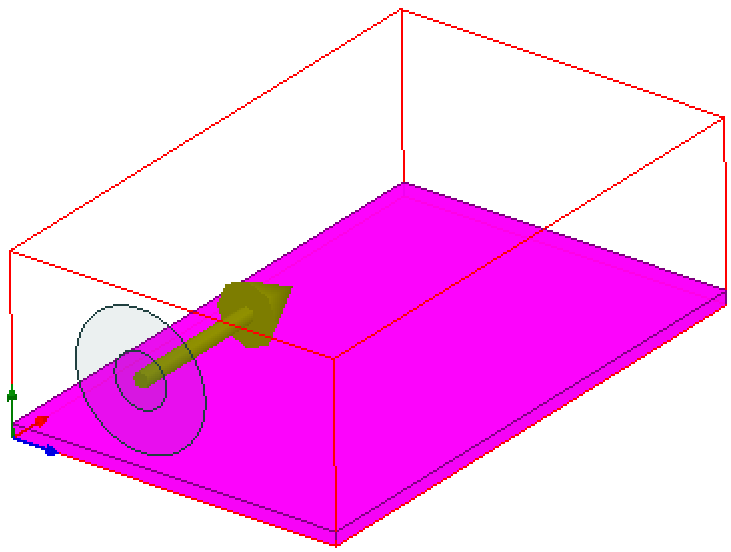
3. Right-click and select **Assign Thermal** > **Grille**.
4. In the **Grille Thermal Model** dialog box under **Flow Specification**, enter a **Free Area Ratio** of 0.5.
5. Click **OK.**

Create the Wall

Create a wall on the Ymax face of the amplifier housing to cover the Xmin side of the enclosure.

1. From the **Draw** menu, select **Box**.
2. In the **CreateBox** dialog box, define the following size parameters and units:
    - **Position**: 0 ,0 ,0 meter
    - **XSize**: 0.4 meter
    - **YSize**: 0.01 meter
    - **ZSize**: 0.25 meter

3. On the **Attribute** tab, define the following parameters:
    - **Name**: wall_1
    - **Material**: FR-4
4. Click **OK**.



5. In the History tree, expand **Model** > **Solids** > **FR-4**.
6. Right-click **wall_1** and select **Select** > **All Faces**.
7. Right-click in the **3D Modeler** window and select **Assign Thermal** > **Wall** > **Stationary**.
8. In the **Stationary Wall Thermal Model** dialog box, define the following parameters:
    - **Name**: wall_1
    - **Wall Thickness**: 0.001 meter
    - **Solid Material**: FR-4
    - **External Condition**: Heat Flux
    - **Heat Transfer Coefficient**: 20 irrad_W_per_m2
9. Click **OK**.

Create Solid Blocks

Create block geometry and duplicate it to create four solid blocks that dissipate 5 W each and have a contact resistance of 0.005 C/W.

Create Block Geometry

1. From the **Draw** menu, select **Box**.
2. In the **CreateBox** dialog box, define the following size parameters and units:
    - **Position**: 0.05 ,0.01 ,0.1 meter
    - **XSize**: 0.05 meter

- **YSize**: 0.02 meter
- **ZSize**: 0.05 meter

3. On the **Attribute** tab, define the following parameters:
   - **Name**: block_1
   - **Material**: Al_Extruded
   - **Color**: Select a unique color
4. Click **OK**.

Create Duplicates of the Block

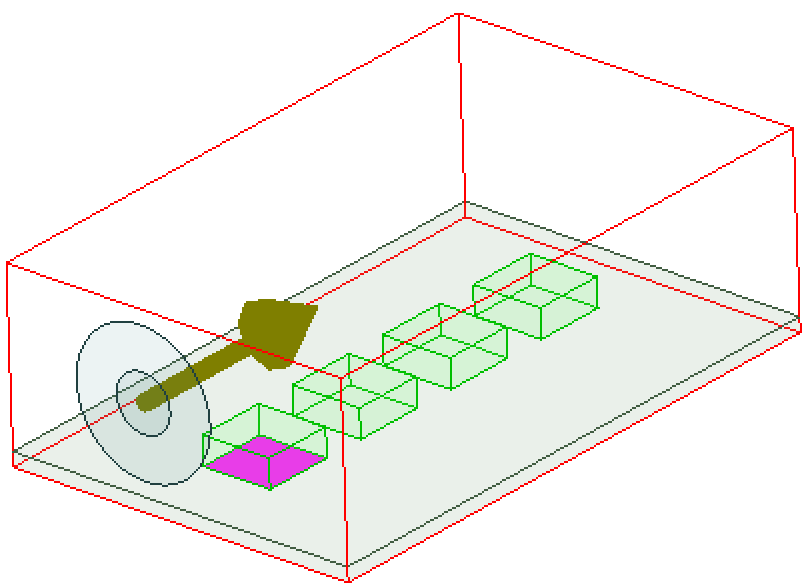
1. In the History tree, expand **Model** > **Solids** > **Al-Extruded**.
2. Right-click **block_1** and select **Edit** > **Duplicate** > **Along Line**.
3. Enter **0.08 ,0 ,0** for the **Vector** to copy the devices in the X direction.
4. In the **DuplicateAlongLine** dialog box, enter **4** for **Total Number**.
5. Click **OK**.

Assign Block Boundary Conditions

1. In the History tree, right-click **block_1** and select **Assign Thermal** > **Block**.
2. In the **Block Thermal Model** dialog box, define a **Total Power** of **5 W**.
3. Click **OK**.
4. Repeat steps 1 through 3 for **block1_1**, **block,1_2**, and **block1_3**.

Assign Conducting Plate Boundary Conditions

1. In the **3D Modeler** window, select the min Y face of block_1.



2. Right-click and select **Assign Thermal** > **Plate** > **Conducting**.
3. In the **Conducting Plate Thermal Model** dialog box, define the following parameters:

- **Thermal Specification**: Thermal Resistance
- **Thermal Resistance**: 0.005 cel_per_w

4. Repeat steps 1 through 4 for the min Y face of **block1_1**, **block,1_2**, and **block1_3**.

Create Network Blocks

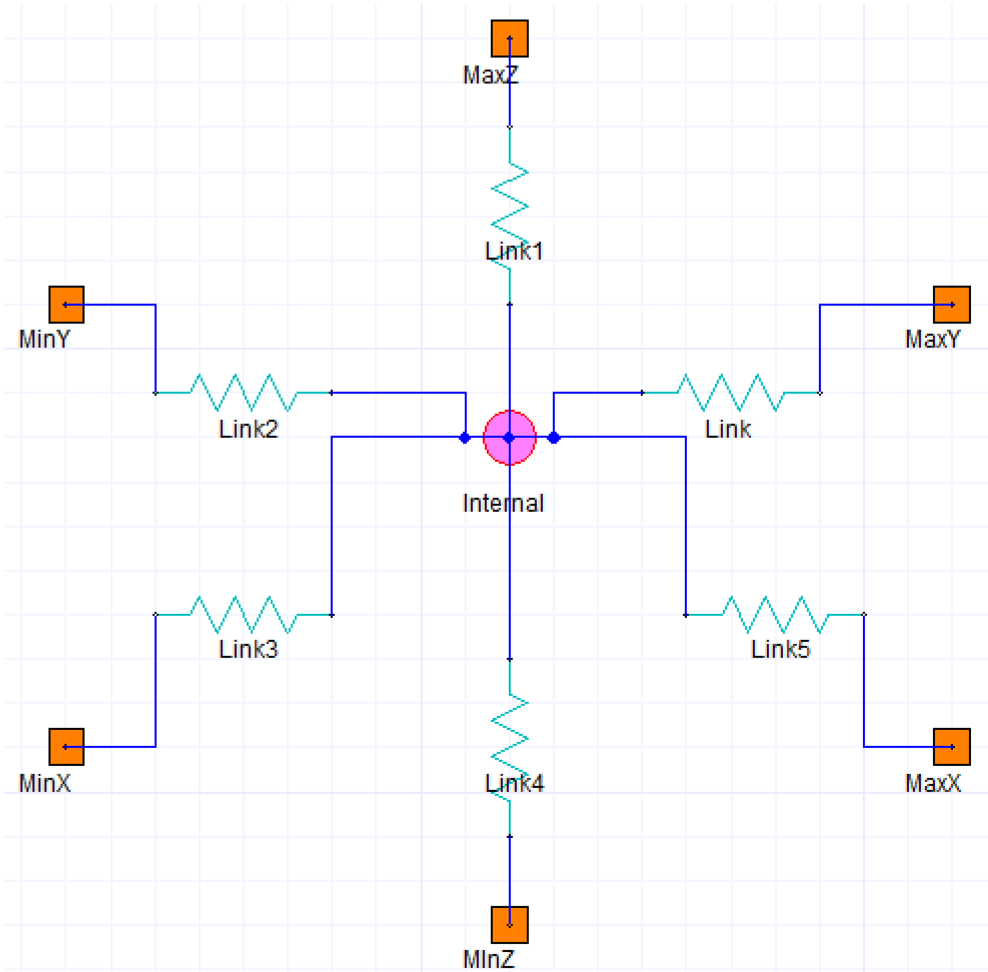
Create block geometry and duplicate it to create four network blocks to model integrated circuit chips.

Create Block Geometry

1. From the **Draw** menu, select **Box**.
2. In the **CreateBox** dialog box, define the following size parameters and units:
    - **Position**: 0.05 ,0.01 ,0.18 meter
    - **XSize**: 0.05 meter
    - **YSize**: 0.02 meter
    - **ZSize**: 0.05 meter
3. On the **Attribute** tab, define the following parameters:
    - **Name**: block_2
    - **Material**: Al_Extruded
    - **Solve Inside**: Disabled
    - **Color**: Select a unique color
4. Click **OK**.

Assign Network Boundary Condition

1. Right-click **block_2** and select **Select** > **All Faces**.
2. Right-click in the **3D Modeler** window and select **Assign Thermal** > **Network**.
3. In the **Network Thermal Model** dialog box, create a network with the following:
    - **Face Nodes**: 6
    - **Internal Nodes**: 1

- **Resistance Links**: 6
- 

4. For the internal node, define the following parameters:
   - **Power**: 10 W
   - **Mass**:0.001 kg
   - **Specific Heat**: 1000 J_per_Kelkg
5. For the resistance links, define a **Thermal Resistance** of **5 cel_per_w**.
6. Click **OK**.

Create Duplicates of the Block and Network

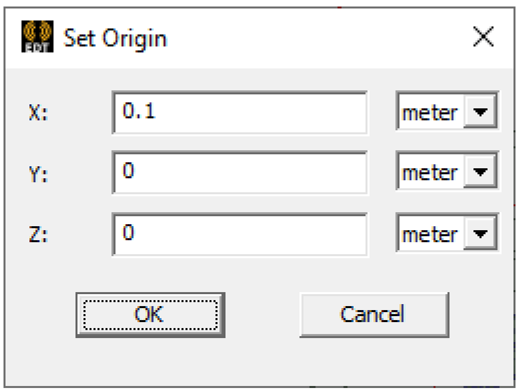
1. In the History tree, expand **Model** > **Solids** > **Al-Extruded**.
2. Right-click **block_2** and select **Edit** > **Duplicate** > **Along Line**.
3. Enter **0.08 ,0 ,0** for the **Vector** to copy the devices in the X direction.
4. In the **DuplicateAlongLine** dialog box, enter **4** for **Total Number**.
5. Change the **Unit** to **meter**.
6. Click **OK**. The duplicate geometry is created and boundary conditions assigned.

Create a Hollow Block

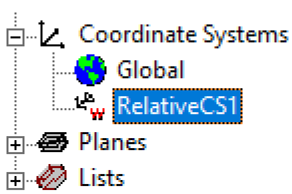
To cut out a section of the cabinet from the computational domain, create a hollow block. This represents a region that does not directly affect heat transfer via solid conduction but that does, however, alter the flow patterns surrounding this region. To demonstrate the creation of a local coordinate system, create a local coordinate system and use it to position the hollow block.

Create a Local Coordinate System

1. From the **Modeler** menu, select **Coordinate System** > **Create** > **Relative CS**> **Offset**.
2. Press **F4** to display the **Set Origin** dialog box.
3. Enter **0.1 meter** for the **X** offset.



4. Click **OK**.
5. In the History tree, note that the **RelativeCS1** coordinate system is now the working coordinate system, denoted by the letter W.



Create a Hollow Block

With the local coordinate system set as the working coordinate system, create a hollow block.

1. From the **Draw** menu, select **Box**.

> **Note:** If the **CreateBox** dialog box is displayed, press **F3** to enter point drawing mode.

2. In the **CreateBox** dialog box, define the following size parameters and units:
    - **Position**: 0 ,0.01 ,0 meter
    - **XSize**: 0.15 meter
    - **YSize**: 0.09 meter
    - **ZSize**: 0.07 meter
3. On the **Attribute** tab, define the following parameters:
    - **Name**: block_3
    - **Material**: Al_Extruded
    - **Solve Inside**: Disabled
    - **Transparent**: 1
4. Click **OK**.

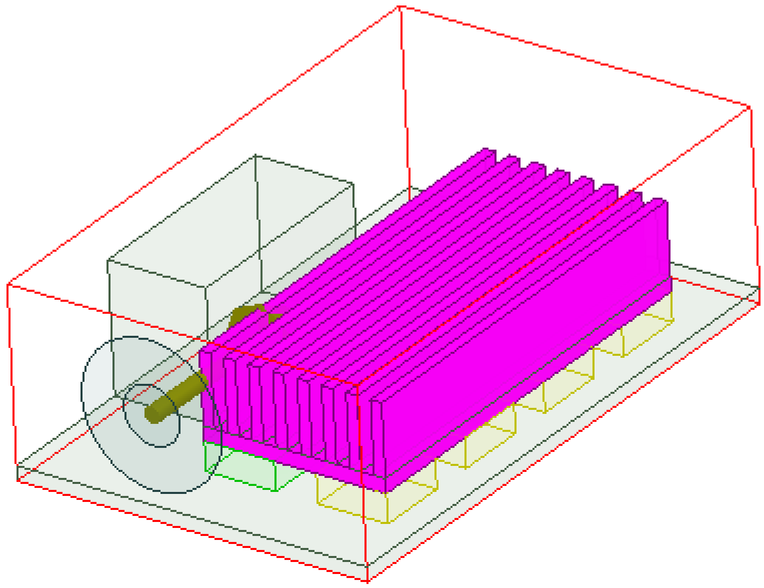
Create a Heat Sink

Create a detailed heat sink and bond plates. The heat sink base acts as a spreader for all the IC chips.

Create the Heat Sink

1. In the History tree, click **Global** to set the global coordinate system as the working coordinate system.
2. In the **Project Manager**, right-click **3D Components** and select **Create** > **Heatsink**.
3. In the **Heatsink Component** dialog box **Geometry** tab, define the following parameters:
    - **Plane**: ZX
    - **Overall height**: 0.06 meter
    - **Base Length**: 0.13 meter
    - **Base Width**: 0.29 meter
    - **Base Height**: 0.01 meter
    - **Fin Type**: Extruded
    - **Fin Flow Direction**: X
    - **Fin Count**: 8
    - **Fin Thickness**: 0.008 meter
4. In the **Heatsink Component** dialog box **Properties** tab, define the **Heatsink Base Solid Material** as **Cu-Pure**.
5. Click **OK**.
6. In the History tree, expand **Heatsink1**.
7. Right-click **Heatsink1_1** and select **Edit** > **Arrange** > **Move**.
8. In the **Move** dialog box, enter a **Move Vector** of **0.195 ,0.03 ,0.165 meter**.

9. Click **OK**.



Create the Pin Bond Plates

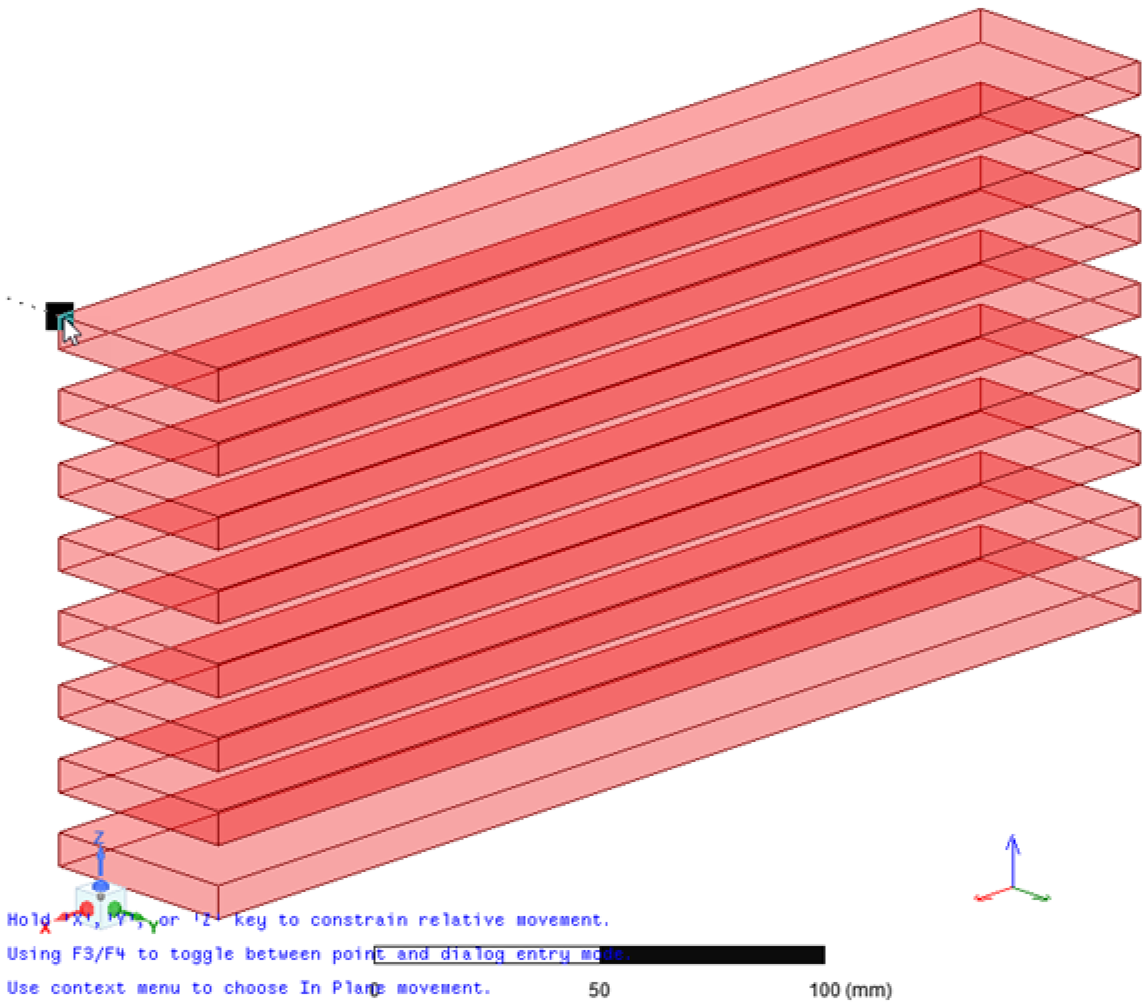
Draw rectangle geometry for each pin bond plate. The plates are the same dimensions as the bottom pin face.

1. In the History tree, expand **Model** > **Heatsink1** > **Heatsink1_1** > **Solids** > **Al-Extruded**.
2. Use the **Shift** key to select all Heatsink1 pin geometry simultaneously.
3. In the **3D Modeler** window, right-click and select **View** > **Show Only Selection**.
4. On the **View** ribbon, select **Dimetric** from the **Orient** drop-down.
5. Click **Fit All** to zoom in on the heatsink pin geometry.
6. From the **Draw** menu, select **Rectangle**.

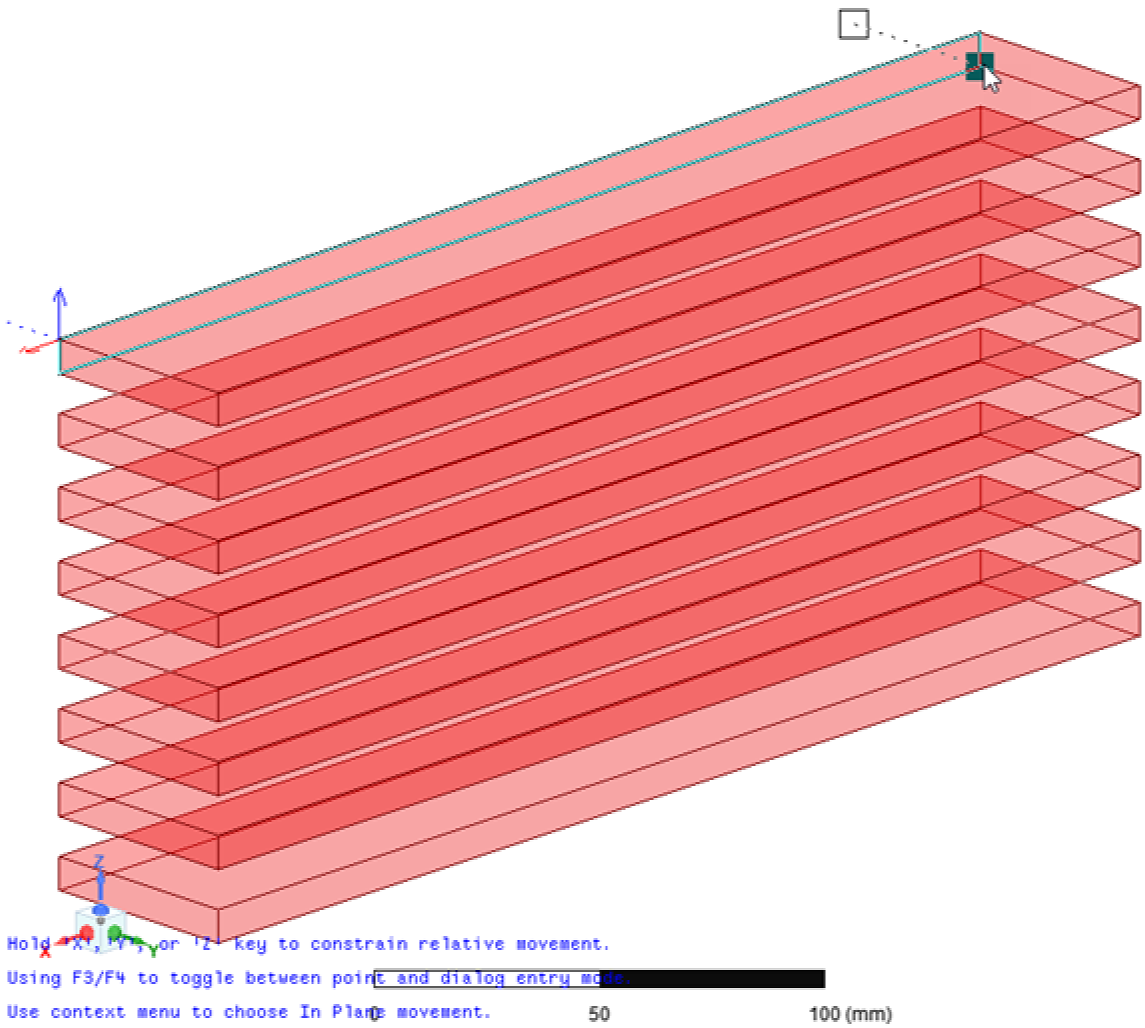
> **Note:** If the **CreateBox** dialog box is displayed, press **F4**.

7. Right-click in the **3D Modeler** window and set the following options:
   - **Movement Mode**: 3D
   - **Grid Plane**: XZ

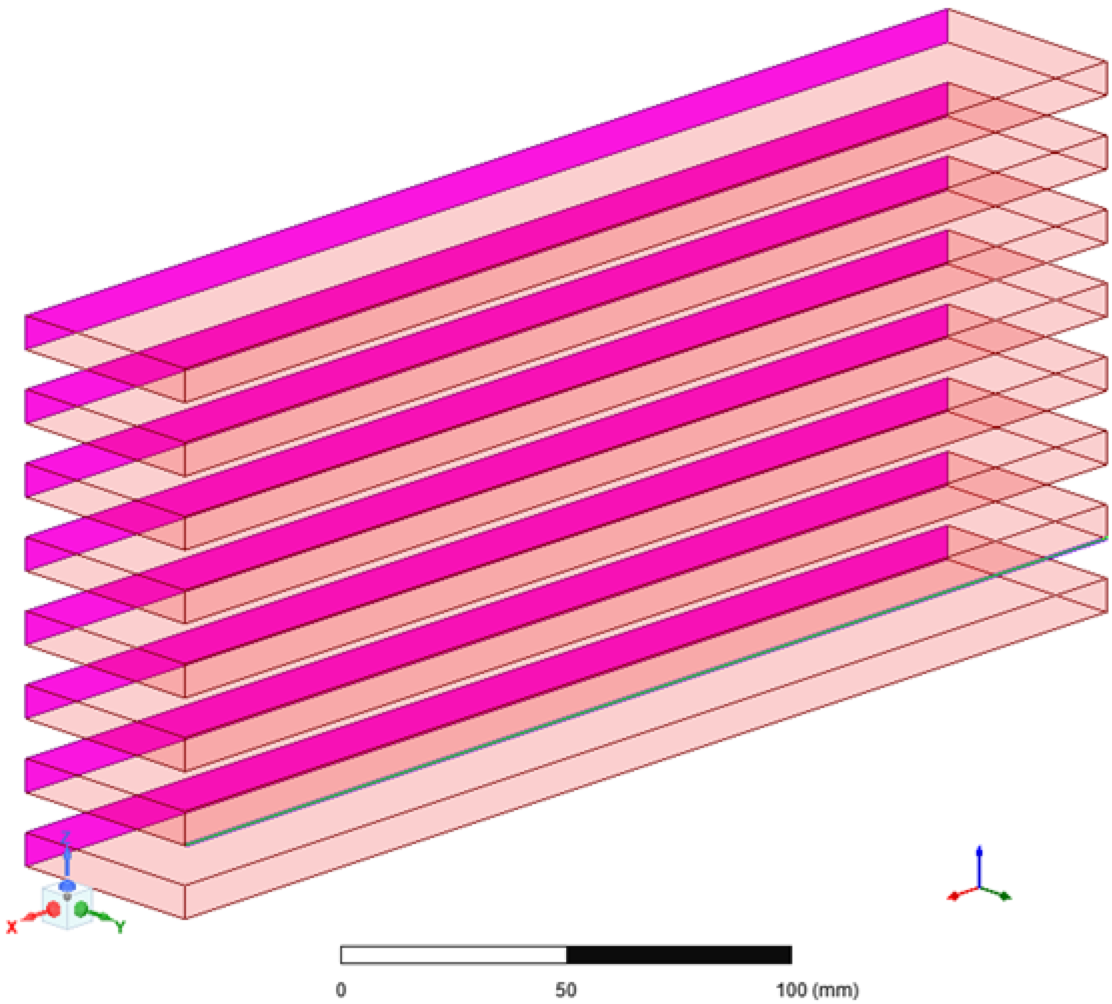
8. Click on the upper left corner of the top pin as displayed in the following image.



9. Then click on the lower right corner of the same pin as displayed in the following image.



10. Repeat steps 6 through 9 for the remaining seven pins.

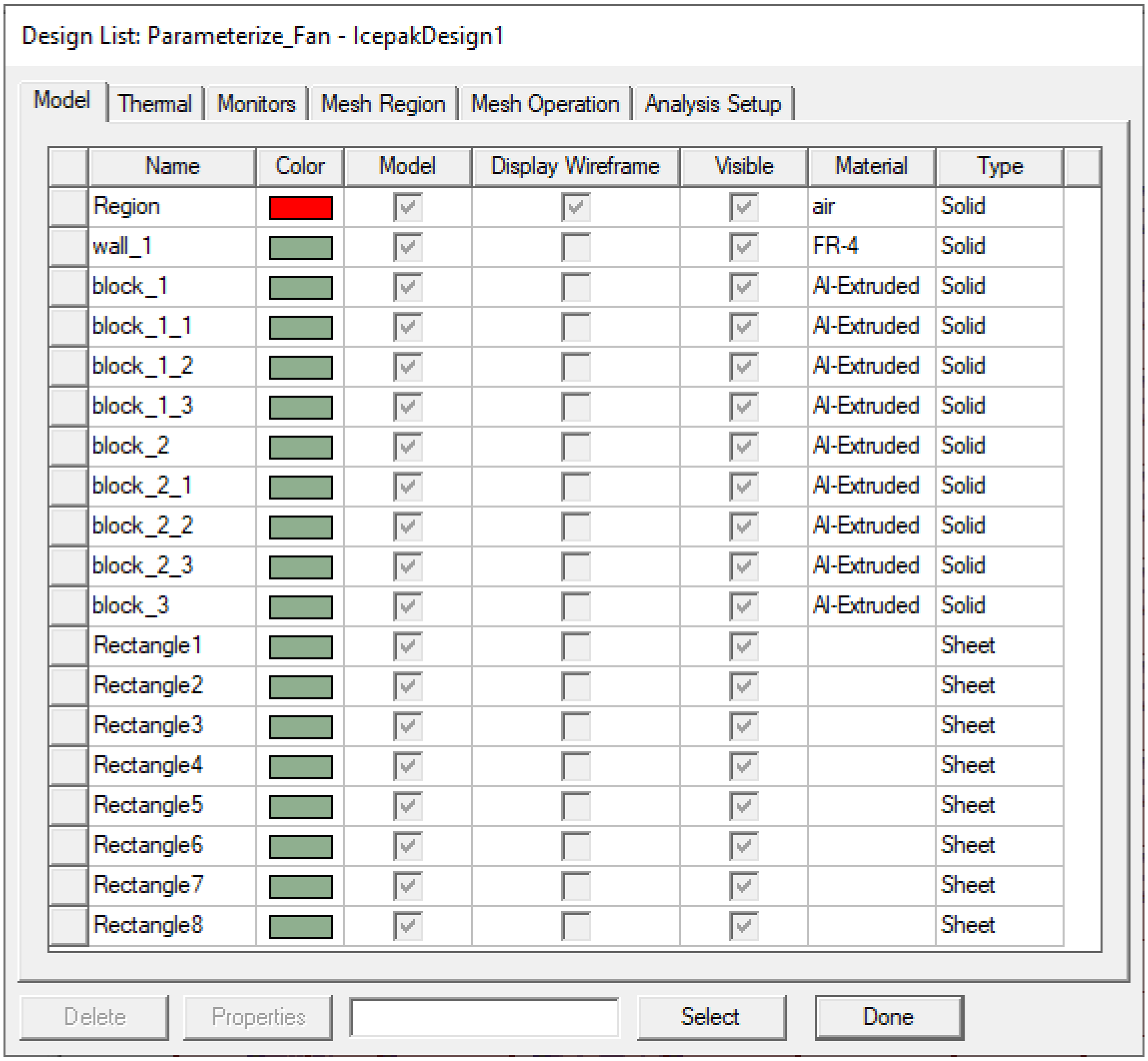


11. In the History tree, expand **Model** > **Sheets** > **Unassigned**.
12. Right-click **Unassigned** and select **Select All**.
13. In the **3D Modeler** window, right-click and select **Assign Thermal** > **Plate** > **Conducting**.
14. In the **Conducting Plate Thermal Model** dialog box, define the following parameters:
    - **Thickness**: 0.0002 meter
    - **Shell Conduction**: Enabled
    - **Side specification**: **Low side** and **High side** enabled
15. Click **OK**.
16. In the **3D Modeler** window, right-click and select **View** > **Show All**.
17. On the **View** ribbon, click **Fit All**.

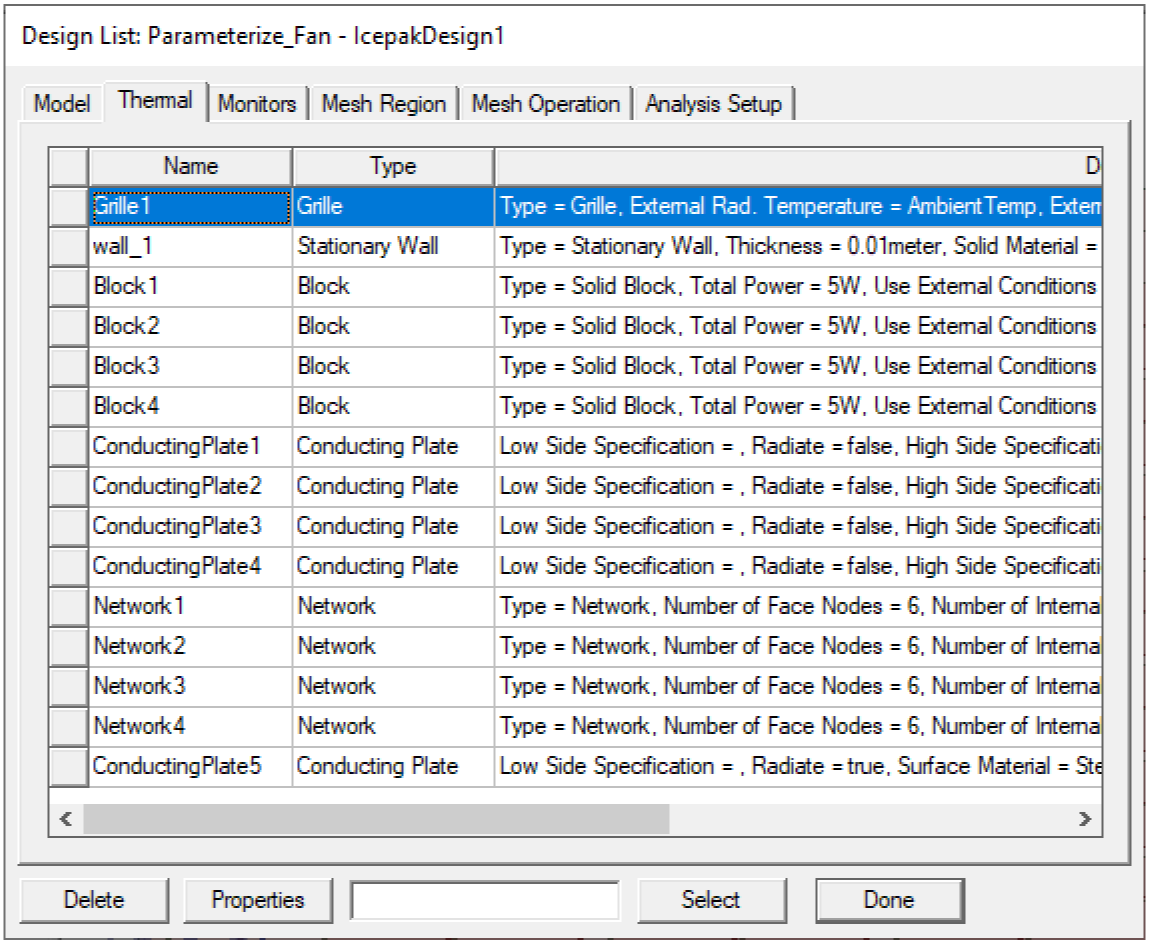
View the Design List

Display the Design List to review the model geometry and thermal boundary condition assignments.

1. From the **Icepak** menu, select **List**.
2. On the **Model** tab, review the model geometry.



3. Click the **Thermal** tab and review the boundary condition assignments.



3 - Assign Mesh Regions

To generate a finer mesh in the fan and enclosure, create two mesh regions. The first mesh region consists of the RF amplifier; the second consists only of the fan.

Create the RF Amplifier Mesh Region

The RF amplifier consists of the blocks and heatsink. First create a non-model box to encompass all of the components.

Create a Non-Model Box

Non-model geometry do not have thermal or material assignments and can be used to assign mesh regions to custom shapes.

1. From the **Draw** menu, select **Box**.
2. Press **F4** to display the **CreateBox** dialog box.
3. In the **CreateBox** dialog box, define the following size parameters and units:
   - **Position**: 0.045 ,0.005 ,0.095 meter
   - **XSize**: 0.31 meter
   - **YSize**: 0.09 meter
   - **ZSize**: 0.14 meter
4. On the **Attribute** tab, define the following parameters:
   - **Name**: blocks_heatsink_meshregion
   - **Model**: Disabled
   - **Transparent**: 1
5. Click **OK**.

Assign a Mesh Region to the Non-Model Box

1. In the History tree, expand **Model** > **Solids** > **Non-Model**.
2. Right-click **blocks_heatsink_meshregion** and select **Assign Mesh Region**.
3. In the **SubRegion** dialog box, click **OK** to close the dialog box without adding padding.
4. In the **Mesh Region** dialog box, enter **Blocks_Heatsink** for the **Name**.
5. Under **Auto Mesh Setting**, move the slider bar to the **Fine/Large** position.
6. Click **OK**.

Assign a Mesh Region to the Fan Component

1. In the **Project Manager** under **3D Components**, select **Fan1_1**.
2. In the **3D Modeler** window, right-click and select **Assign Mesh Region**.
3. On the **SubRegion** dialog box, select **Pad individual directions** for **Padding Data**.

4. For each **Direction**, select **Absolute Offset** from the **Padding type** drop-down and enter the following values.
    - **+X**: 0.005 meter
    - **-X**: 0 meter
    - **+Y**: 0.002 meter
    - **-Y**: 0.002 meter
    - **+Z**: 0.002 meter
    - **-Z**: 0.002 meter
5. In the **Mesh Region** dialog box, enter **Fan** for the **Name**.
6. Under **Auto Mesh Setting**, move the slider bar to the **Fine/Large** position.
7. Click **OK**.

4 - Generate and Display Mesh

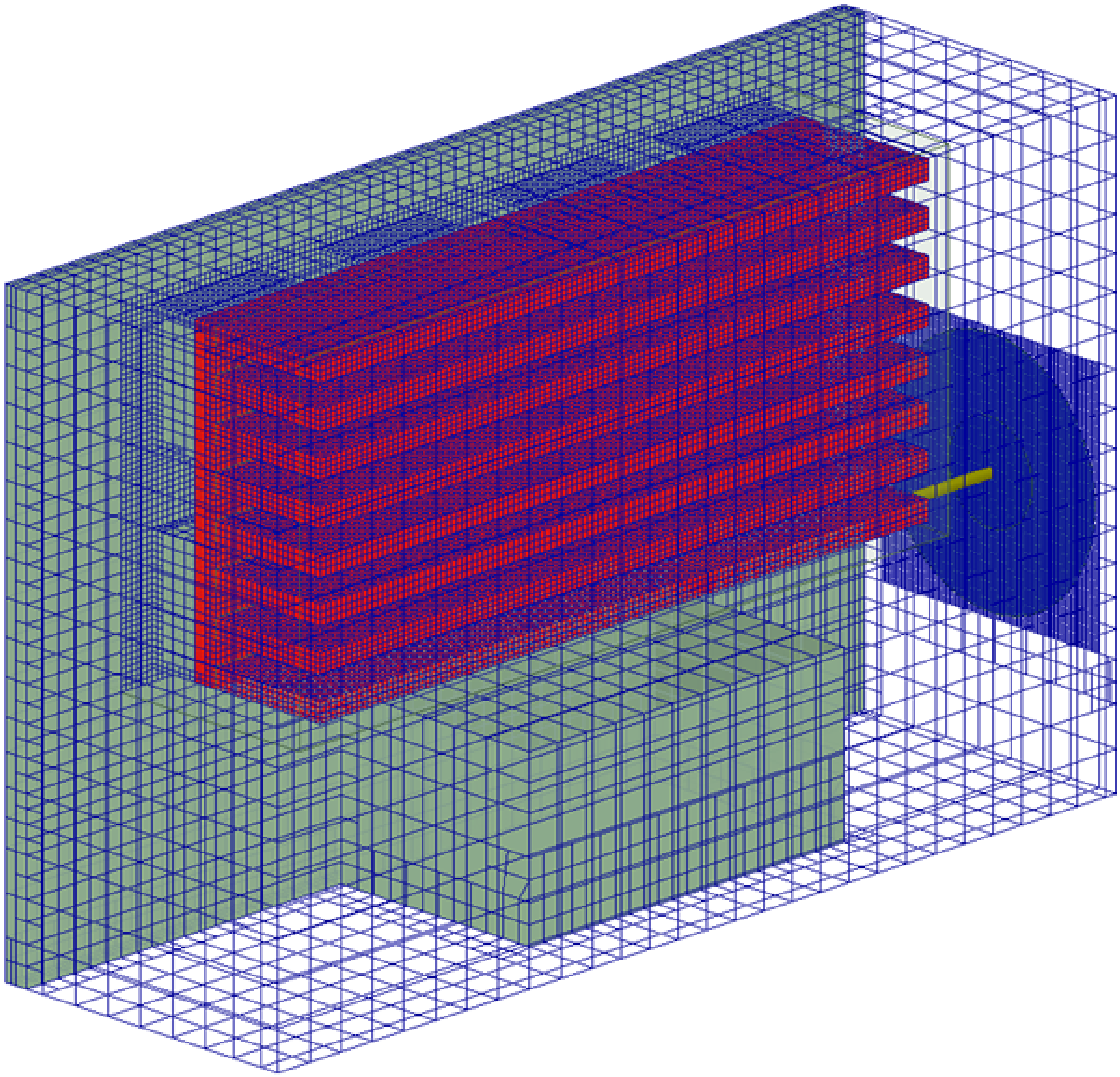
After creating the mesh regions, generate and display the mesh. Also, check the mesh quality by examining the face alignment.

Generate the Mesh

1. In the **Project Manager**, right-click **Mesh** and select **Generate Mesh**. The **Mesh Visualization** dialog box automatically appears after the meshing process completes.
2. In the **Mesh Visualization** dialog box under **Mesh display on**, enable **Show** and select **Geometry/Boundary selection**.
3. In the History tree, right-click **Model** and select **Select All**.
4. On the **View** ribbon, select **Front** from the **Orient** drop-down list.

Display the Mesh

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. Expand **Model** > **Solids** > **Non Model**.
4. Use the **Ctrl** key to deselect **blocks_heatsink_meshregion**.

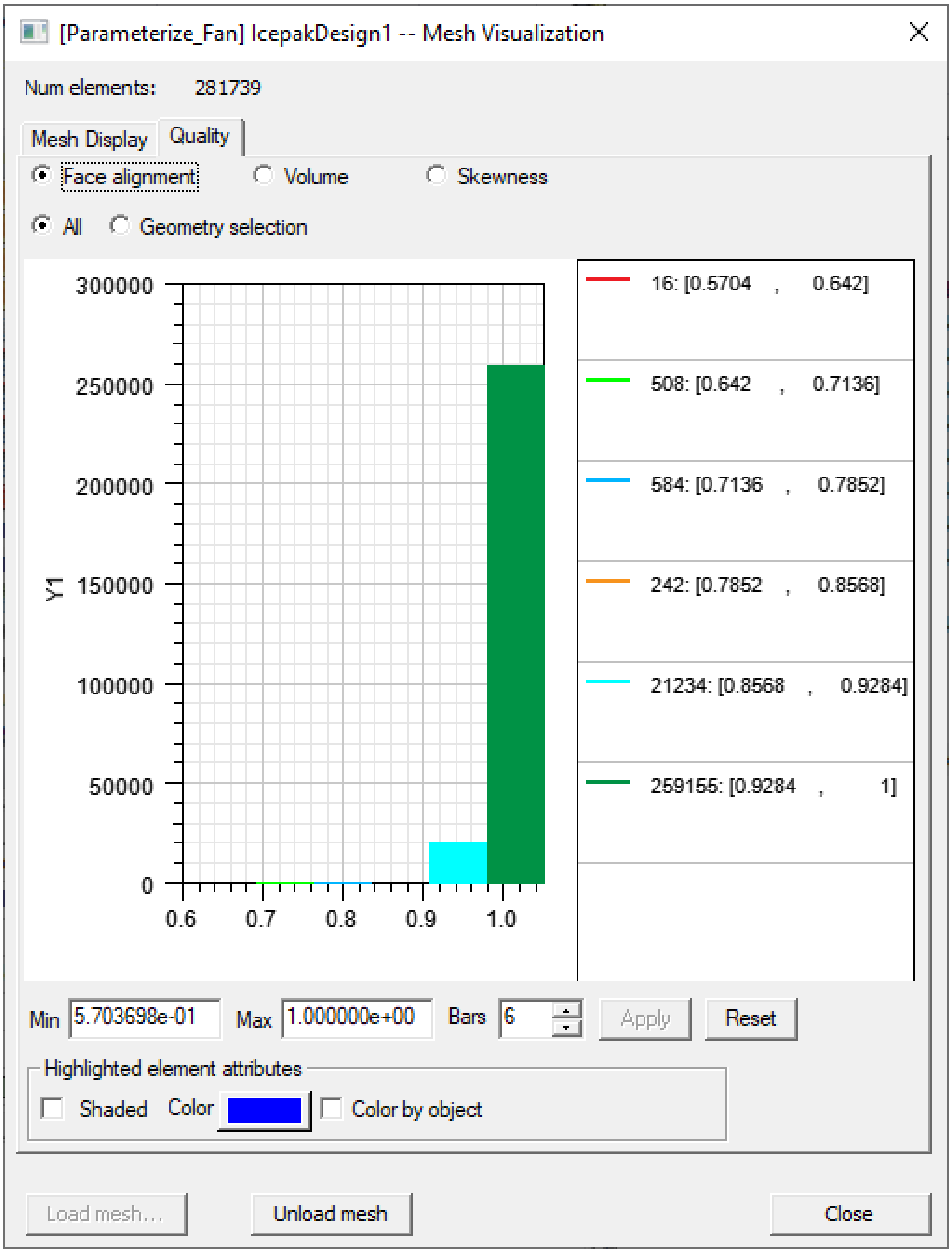


5. Examine the mesh by taking plane cuts in all directions under the **Mesh Display** tab.

Check Mesh Quality

1. In the **Mesh Visualization** dialog box, click the **Quality** tab and examine face alignment.

> **Note:** Due to differences among different machines, the numbers may not be exactly the same as those in the following image.



> **Note:** This graph displays the cell count versus face alignment. Adjacent mesh faces that are not aligned can result in long, narrow elements. A value of 1 indicates perfect alignment. Values less than 0.15 indicate a severely distorted mesh.

2. Click **Close**.

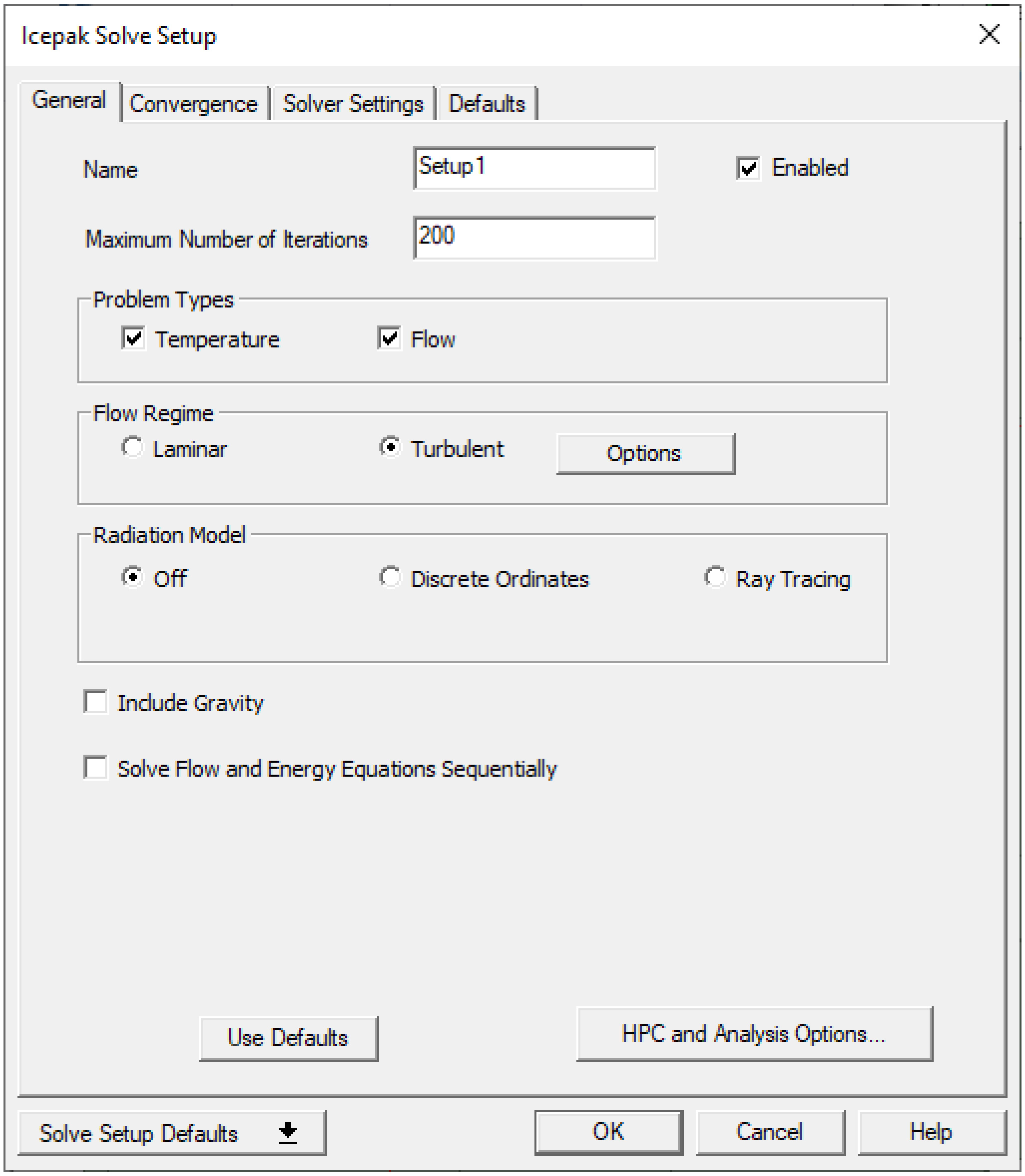
5 - Define the Simulation Settings

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**.

6 - Create Monitor Points

Before running the simulation, create thermal and flow monitors for block_1 and Grille_1.

Create a Thermal Monitor

1. In the History tree, expand **Model** > **Solids** > **Al-Extruded**.
2. Right-click **block_1** and select **Assign Monitor** > **Point**.
3. In the **Monitor Setup** dialog box, enter **Block_temperature** for the **Name**.
4. Expand **Thermal** and select **Temperature**.
5. Click **OK**.

Create a Flow Monitor

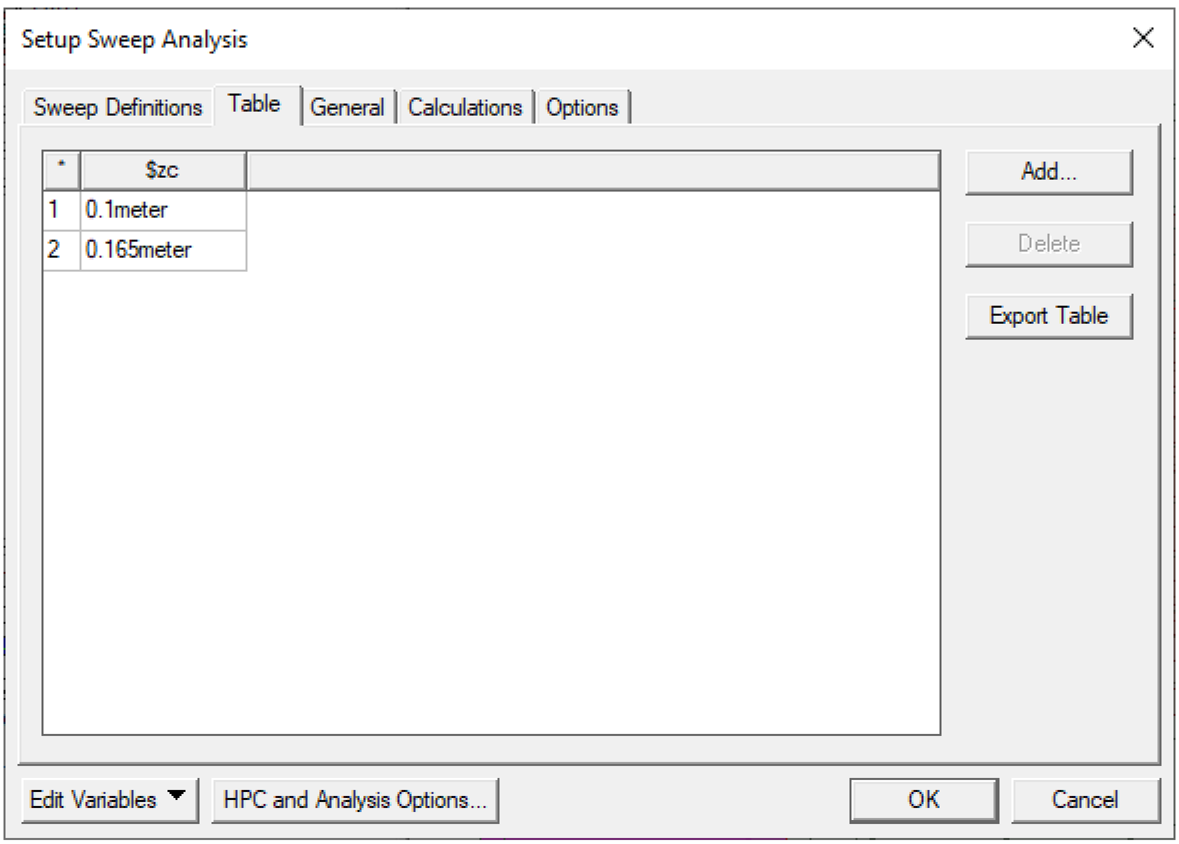
1. In the **Project Manager** under **Thermal**, right-click **Grille_1** and select **Select Assignment**.
2. From the **Icepak** menu, select **Monitor** > **Assign** > **Point**.
3. In the **Monitor Setup** dialog box, enter **Grille_speed** for the **Name**.
4. Expand **Flow** and select **Speed**.
5. Click **OK**.

7 - Set Up and Run the Parametric Trials

Set Up the Parametric Trials

Define the parametric trials using the $zc variable assigned to the location of the fan component.

1. In the **Project Manager**, right-click **Optimetrics** and select **Add** > **Parametric**.
2. In the **Setup Sweep Analysis** dialog box, click **Add**.
3. In the **Add/Edit Sweep** dialog box, select **Single value**.
4. Enter **0.1 meter** for the **Value** and click **Add**.
5. Enter **0.165 meter** for the **Value** and click **Add**.
6. Click **OK** to close the **Add/Edit Sweep** dialog box.
7. Click the **Table** tab to display the two sweeps to be performed in the trials.



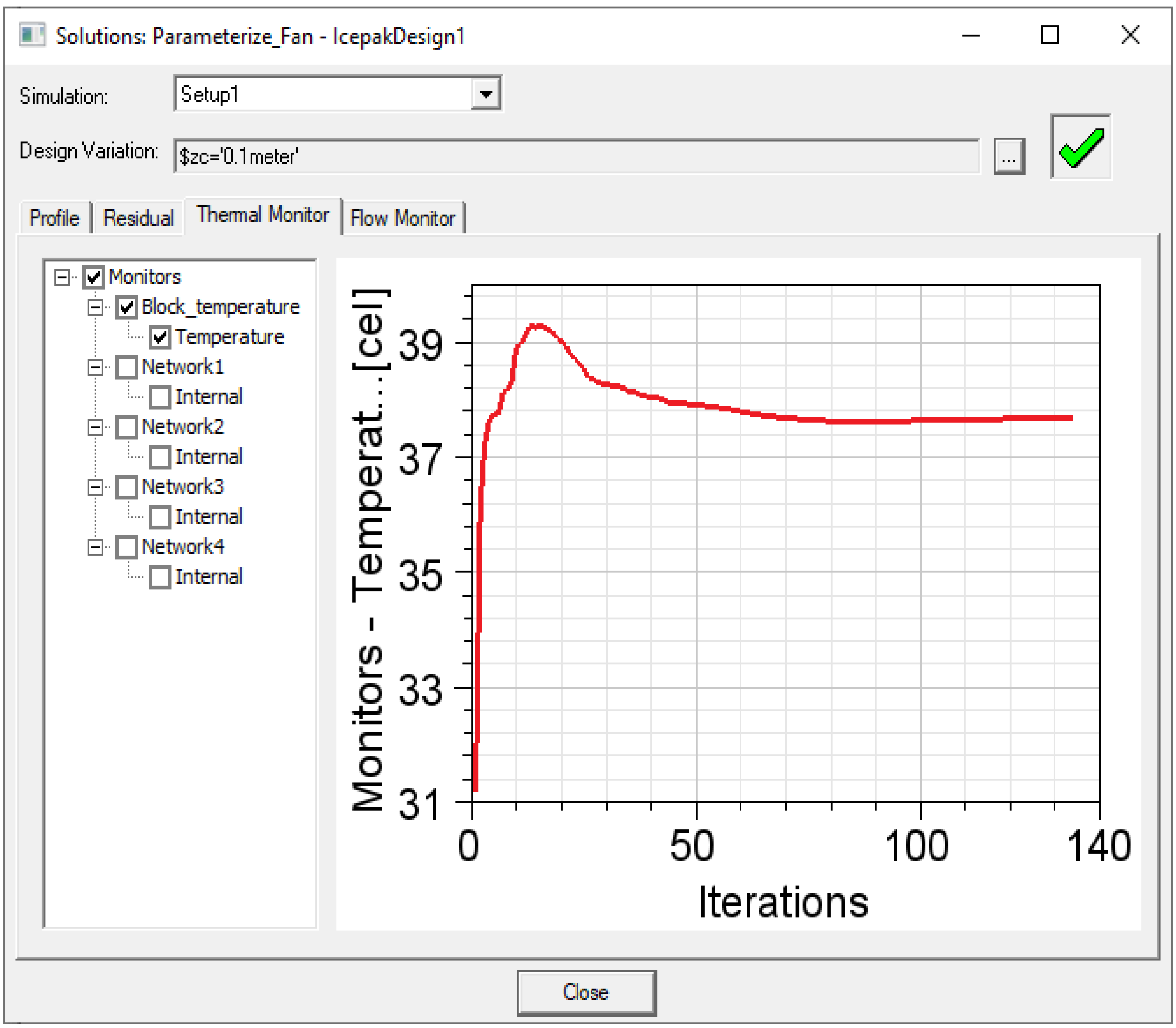
8. Click the **Options** tab and enable **Save Fields and Mesh**. This will allow you to switch between variation data when displaying post-processing objects.
9. Click **OK**.

Run the Parametric Trials

1. In the **Project Manager**, expand **Optimetrics**.
2. Right-click **ParametricSetup1** and select **Analyze**.
3. In the **Project Manager**, expand **Analysis**.
4. Right-click **Setup1** and select **Residual** to monitor the residuals as the trials run.

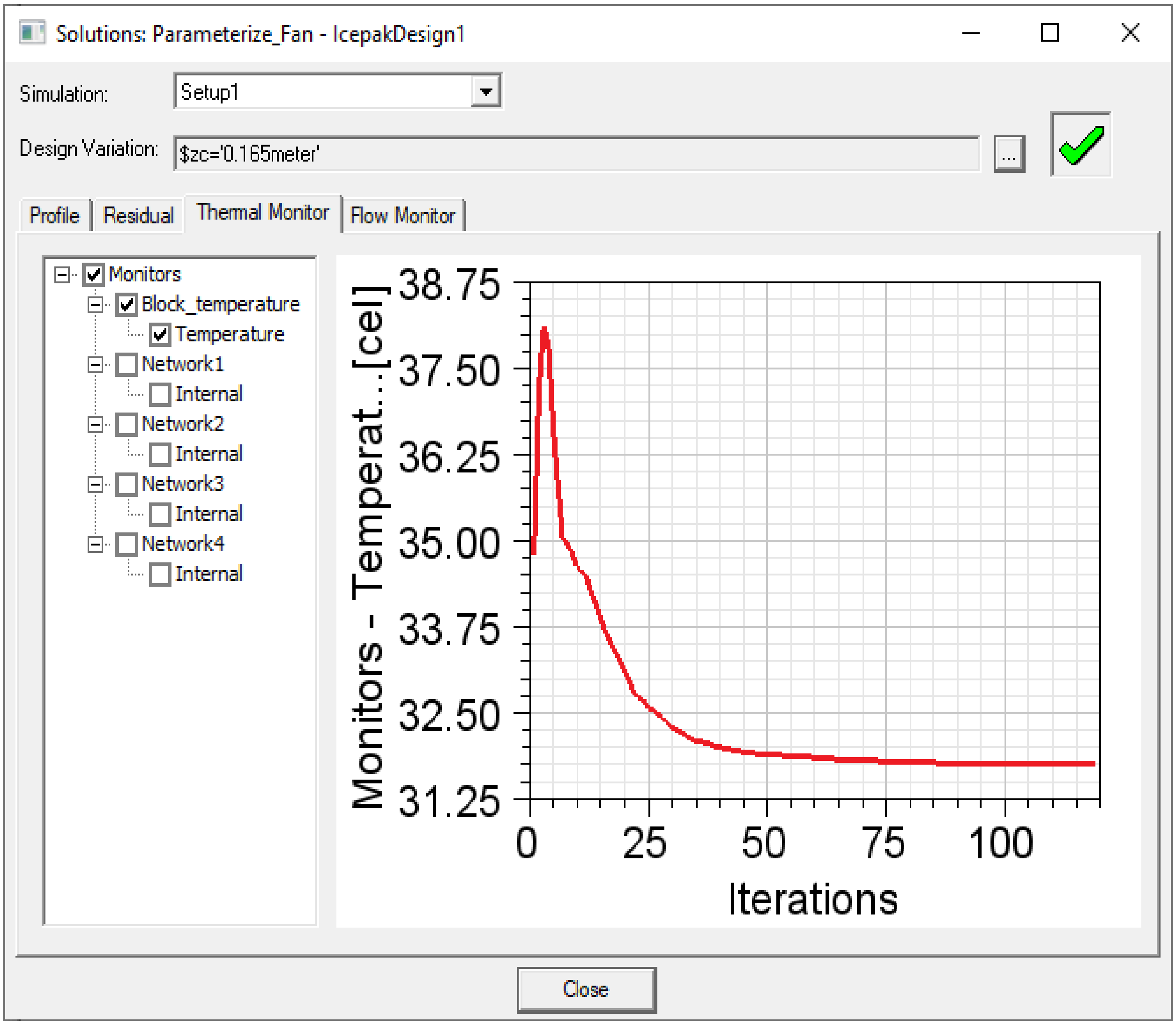
> **Note:** When the simulation is complete, a message is displayed in the **Messages** window stating "Parametric Analysis is done."

5. Click the **Thermal Monitor** and **Flow Monitor** tabs for the 0.1 meter design variation.



6. Click the **[...]** button and select the 0.165 meter variation in the **Set Design Variation** dialog box.
7. Click the **Residual**, **Thermal Monitor**, and **Flow Monitor** tabs for the 0.165 meter design variation.

> **Note:** Use the navigation arrows on the image below to view each tab.



> **Note:** On the **Thermal Monitor** tab, note the more efficient cooling by the fan in the second variation's location.

8 - Post-process the Results

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

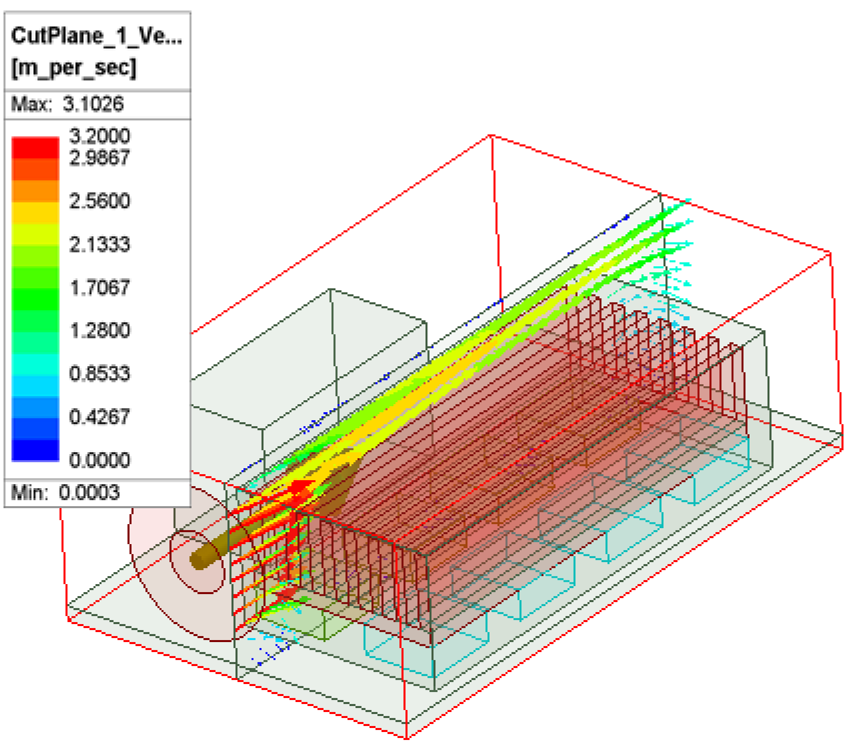
- Plane cut views
- Object face views
- Fields summary report

Create a Plane Cut

Create a plane cut plotting velocity using the Cut Plane toolkit. The current design variation is set to the orignal fan offset location ($zc = 0.1 meter).

Create a Plane Cut

1. From the **Icepak** menu, select **Toolkit** > **Reporting** > **Cut_Plane**.
2. Under **Plane Location**, select **Z Plane through Center**.
3. Select **Show Vectors**.
4. Click **Create**.
5. Click **Exit**.



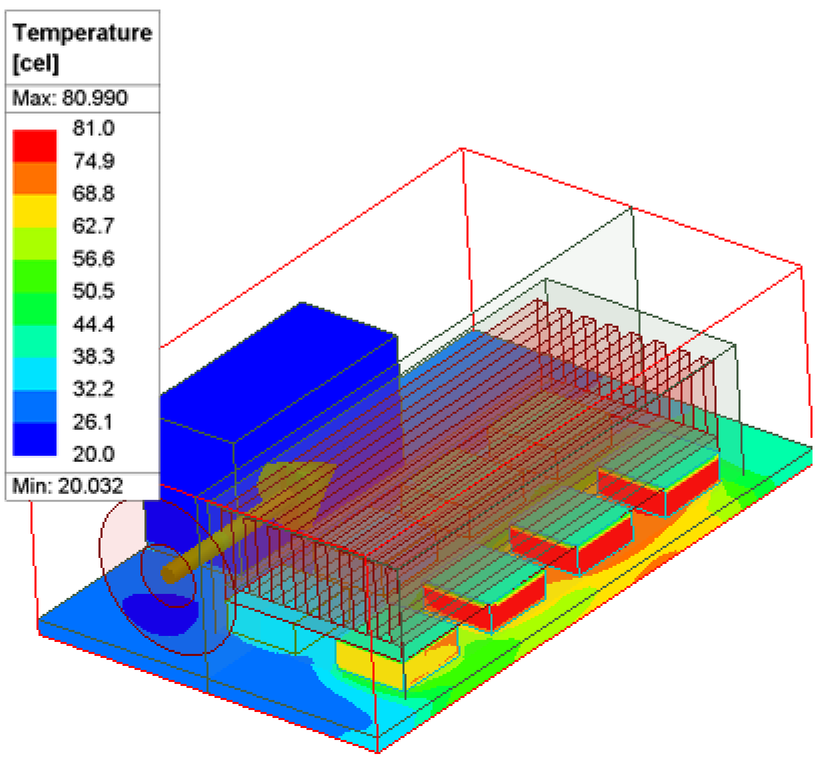
6. In the **Project Manager**, expand **Field Overlays** > **CutPlane_1_Vector**.
7. Right-click **Velocity_Vectors1** and select **Plot Visibility** to hide the plane cut.

Create Object Field Overlays

Create a temperature plot on the wall. The current design variation is set to the original fan offset location ($zc = 0.1 meter).

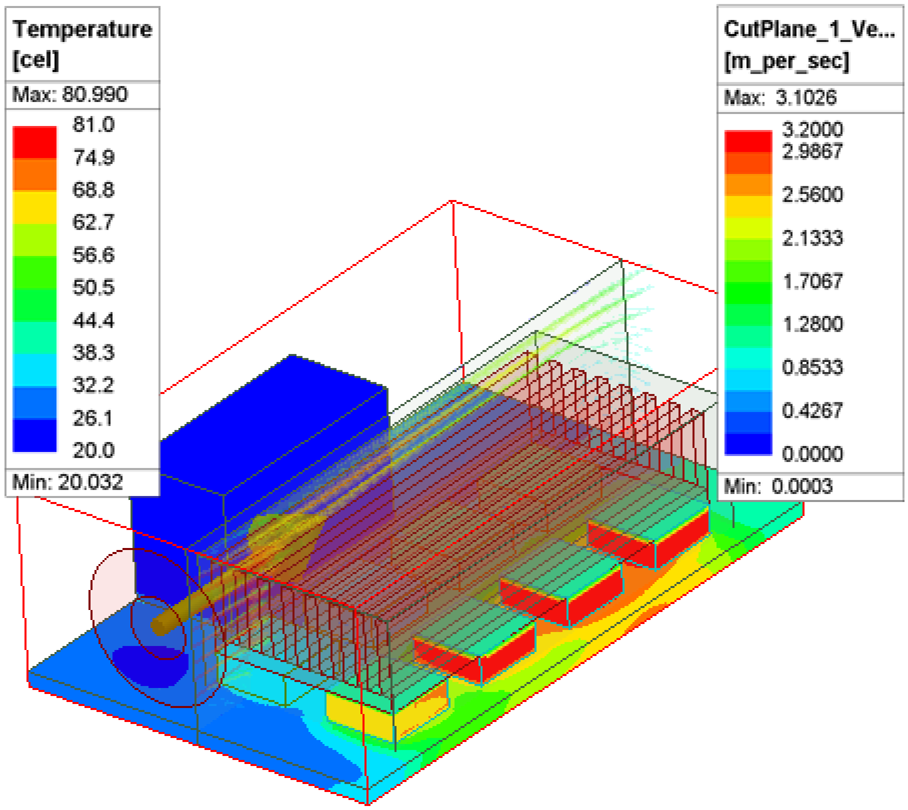
Plot Temperature on the Wall and Blocks

1. In the History tree, expand **Model** > **Solids** and **FR-4**.
2. Right-click **Al-Extruded** and select **Select All**.
3. Press **Ctrl** and select **wall_1** under **FR-4**.
4. Right-click in the **3D Modeler** window and select **Plot Fields** > **Temperature** > **Temperature**.
5. In the **Create Field Plot** dialog box, retain the **Temperature** selection for **Quantity**.
6. Enable **Plot on surface only**.
7. Click **Done**.



8. In the **Project Manager**, expand **Field Overlays** > **CutPlane_1_Vector**.
9. Right-click **Velocity_Vectors1** and select **Plot Visibility** to display both post-processing

objects simultaneously.

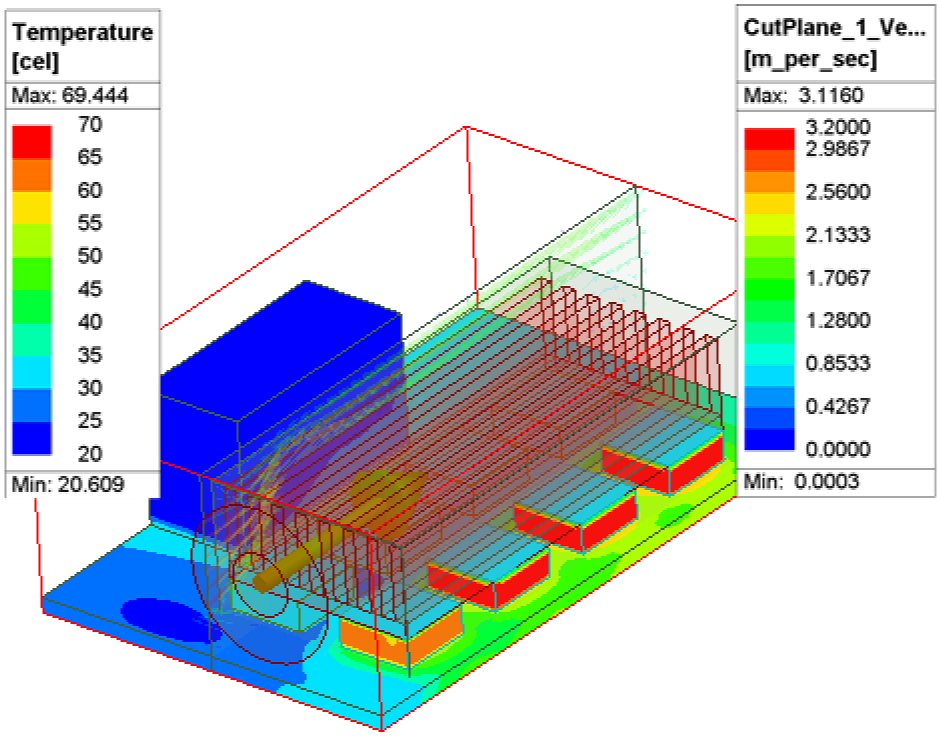


Switch the Design Variation

After you've reviewed the plane cut and field plot results for the initial design variation, change the design variation to review the results at the alternative fan location.

1. In the **Project Manager**, expand **Optimetrics**.
2. Right-click **ParametricSetup1** and select **View Analysis Result**.
3. In the **Post Analysis Display** dialog box, select the **0.165meter** variation and click **Apply**.

4. Click **Close**.



Create a Fields Summary Report

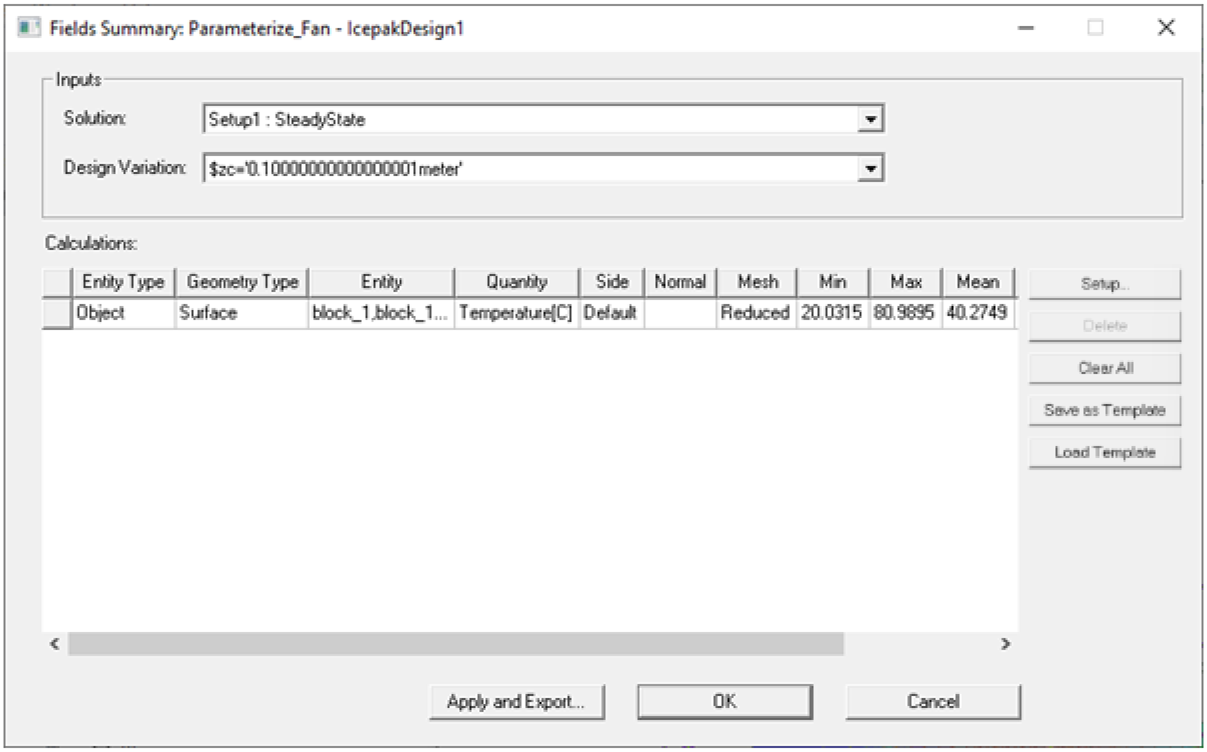
Create a fields summary report of object-specific solution data. Fields summary reports can provide physical information from the solution about specific model boundary conditions or objects.

1. From the **Icepak** menu, select **Fields** > **Create Fields Summary**.
2. In the **Fields Summary** dialog box, for **Design Variation**, select **$zc=0.1meter**.
3. In the **Setup Calculation** dialog box, select **Object** for **Entity Type**.
4. In the **Entity** list, select all of the **Block** objects.

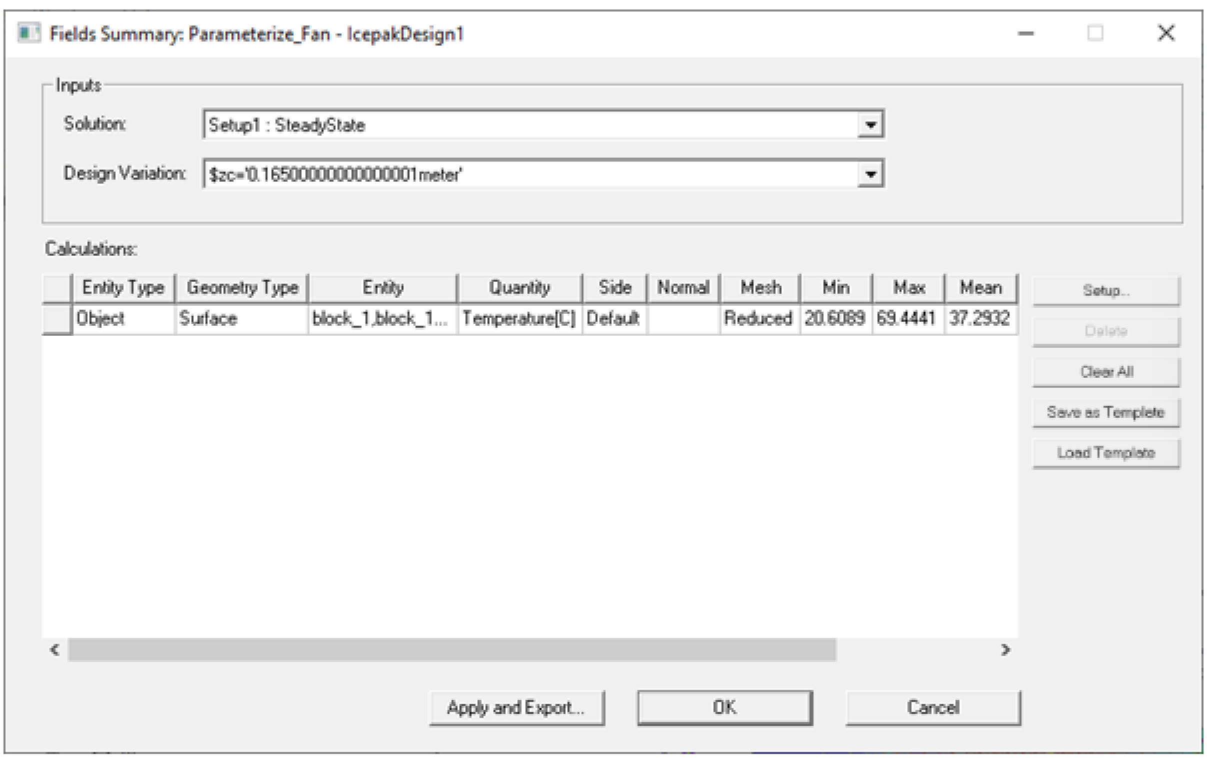
> **Note:** You can select multiple entities at once. When an entity is selected, select it again to clear the selection.

5. In the **Quantity** list, select **Temperature**.

6. From the **Add** drop-down list, select **Add As Single Calculation**.



7. In the **Fields Summary** dialog box, for **Design Variation**, select **$zc=0.165**.



> **Note:** Note the lower maximum and mean temperaures in the $zc=0.165meter variation.

9 - Summary

In this tutorial, you learned how to set up and solve multiple trials to optimize a parameter, specify a dynamically updating fan curve, create a new local coordinate system, and use separate mesh regions to reduce mesh counts. The use of network blocks to model packages has been demonstrated as well as how to specify contact resistance using side specifications of a block object. You also learned how to generate a summary report for multiple solutions.