

© 2026 Synopsys, Inc and/or ANSYS, Inc. All rights reserved.
Unauthorized use, distribution, or duplication is prohibited.

Getting Started with Icepak®: RF Amplifier

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 2026 R1
January 2026

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

Copyright and Trademark Information

© 1986-2026 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, Ansys Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXlm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

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

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for Ansys proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table of Contents

Table of Contents **Contents-1**
1 - Introduction **1-1**
Set 3D UI Options 1-2
2 - Create a Project and Build the Model **2-1**
Set Beta Option 2-1
Create a Project 2-1
Build the Model 2-1
Resize the Cabinet 2-1
2-2
Create the Opening 2-2
2-3
Create the Housing Plates 2-3
Create the Wall 2-4
Create the PCB 2-5
Create the PCB Material 2-5
Create the PCB Geometry and Assign the Material 2-6
Create the Devices 2-6
Create a Device 2-6
Create Duplicates of the Device 2-7
Assign the Source Boundary Condition 2-7
Create the Heatsink 2-8
Create the Fan 2-8
View the Design List 2-9
3 - Generate and Display Coarse Mesh **3-1**
Generate a Coarse Mesh 3-1
Display the Coarse Mesh 3-1
4 - Assign Mesh Regions **4-1**
Create the RF Amplifier Mesh Region 4-1

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create a Non-Model Box 4-1
Assign a Mesh Region to the Non-Model Box 4-1
Assign a Mesh Region to the Fan Component 4-1
5 - Generate and Display Refined Mesh 5-1
Generate and Display Refined Mesh 5-1
Display the Refined Mesh 5-1
6 - Create Monitor Points 6-1
Create a Thermal Monitor 6-1
Create a Flow Monitor 6-1
7 - Define the Simulation Settings and Run the Analysis 7-1
Define the Design Settings 7-1
Add a Solution Setup 7-1
Run the Simulation 7-2
8 - Post-process the Results 8-1
Create Object Field Overlays 8-1
Plot Temperature on the Heat Sink 8-1
Plot Temperature on the Devices 8-2
Create Plane Cut Field Overlays 8-3
Create a Plane 8-3
Plot Speed 8-4
Create a Fields Summary Report 8-5
9 - Summary 9-1

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 - Introduction

This document is intended as supplementary material to Icepak for beginners and advanced users. It demonstrates how to model an RF amplifier using Ansys Icepak as well as many features and functions essential to any Ansys Icepak project.

In this tutorial you will learn how to:

- Create a new project
- Create a model using openings, fans, sources, blocks, heat sinks, and walls
- Use non-conformal meshing
- Set up a simulation with various physical conditions and parameters, including turbulence and natural convection
- Calculate a solution
- Post-process your results by using power and temperature limits, object faces, plane cuts, isosurfaces and variation plots

This chapter contains the following topic:

- "Sample Project - The RF Amplifier" below

Sample Project - The RF Amplifier

RF amplifiers are typically sealed enclosures that are placed within larger systems. They present a challenge from the thermal management perspective because no direct exchange of air exists between the interior of the amplifier and the ambient. The common method of cooling such subsystems is to mount a large heat sink on the amplifier housing that cools all the devices within the enclosure. A simplified version of an RF amplifier will serve as the model for this tutorial. There will be free convection inside the amplifier and forced convection in the external domain.

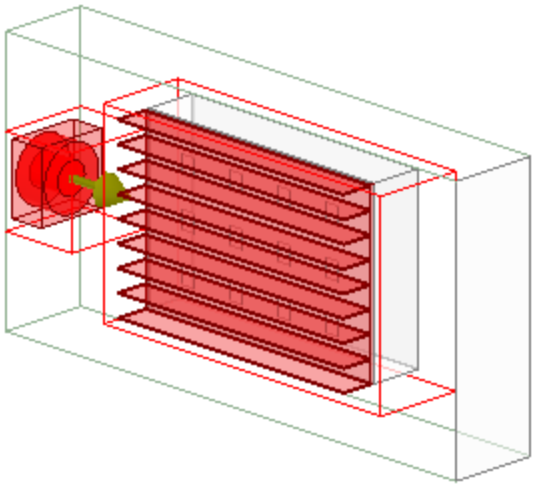


Figure 1-1: RF Amplifier

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

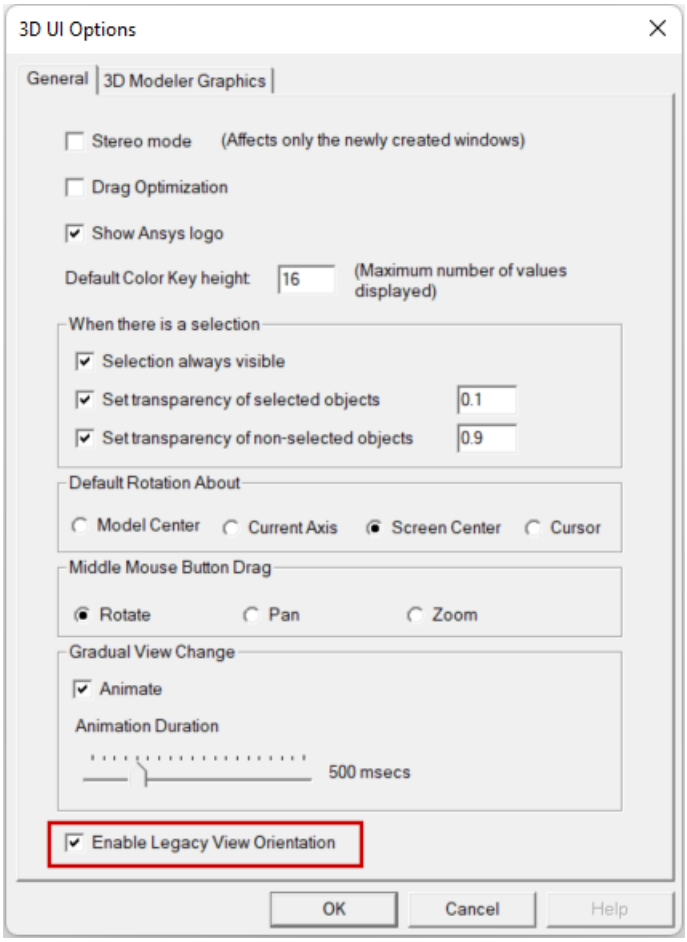
Set 3D UI Options

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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 Beta Option

> **Note:** To use **Slack** settings, enable the Icepak **Virtual Mesh Regions** beta options in the General Options.

1. From the **Tools** menu, select **Options** > **General Options**.
2. In the **Options** dialog box, expand **General** > **Desktop Configuration**.
3. Click **Beta Options**.
4. Enable **Icepak Virtual Mesh Regions** and click **OK**.
5. Click **OK** to close the **Options** dialog box.

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 "RF_Amplifier."
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**: 0.1 meter
    - **-X Padding Type**: Absolute Position
    - **-X Padding Data**: 0 meter
    - **+Y Padding Type**: Absolute Position
    - **+Y Padding Data**: 0.6 meter

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **-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**: -0.05 meter

Create the Opening

Create opening sheet geometry for the opening. The geometry serves as the assignment for the opening thermal boundary condition and, later this in this guide, a flow monitor point.

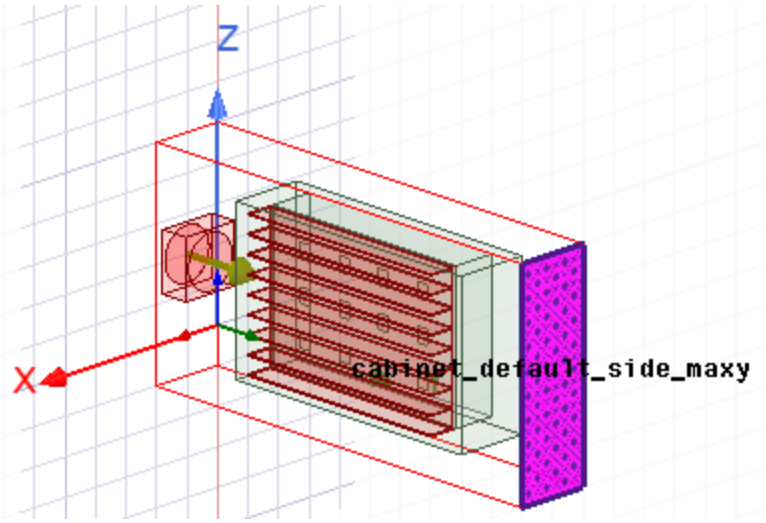
1. From the **Draw** menu, select **Rectangle**.
2. Press **F4** to enter dialog entry mode.

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

3. In the **CreateRectangle** dialog box **Command** tab, define the following size parameters and units:
   - **Position**: 0 ,0.6 ,-0.05 meter
   - **Axis**: Y
   - **Length**: 0.1 meter
   - **Width**: 0.3 meter
4. On the **Attribute** tab, enter **cabinet_default_side_maxy** for the **Name**.
5. Click **OK**.
6. In the History tree, expand **Model** > **Sheets** > **Unassigned**.
7. Right-click **cabinet_default_side_maxy**and select **Assign Thermal** > **Opening** > **Free**.
8. In the **Opening Thermal Model** dialog box, enter **cabinet_default_side_maxy** as the **Name**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Click **OK** to accept the default thermal and flow specifications.



Create the Housing Plates

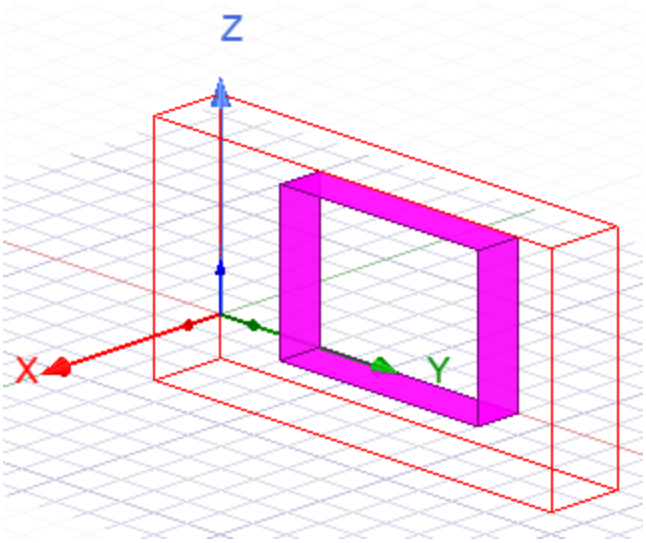
1. From the **Draw** menu, select **Rectangle**.
2. In the **CreateRectangle** dialog box **Command** tab, define the following size parameters and units:
   - **Position**: 0 ,0.45 ,0 meter
   - **Axis**: Y
   - **Length**: 0.2 meter
   - **Width**: 0.06 meter
3. On the **Attribute** tab, enter **Housing_maxy** for the **Name**.
4. Click **OK**.
5. Repeat steps 2 through 5 to create the remaining three housing plates based on the specifications in the following table.

> **Note:** The order of the **Length** and **Width** rows changes based on the **Axis** selection. Ensure that you enter the width and length values into the correct cells when editing the size properties.

Name	Position	Axis	Width	Length
Housing_maxz	0 ,0.15 ,0.2 meter	Z	0.06 meter	0.3 meter
Housing_miny	0 ,0.15 ,0 meter	Y	0.2 meter	0.06 meter
Housing_minz	0 ,0.15 ,0 meter	Z	0.06 meter	0.3 meter

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. In the History tree, right-click **Housing_maxy** and select **Assign Thermal** > **Plate** > **Conducting**.
7. In the **Conducting Plate Thermal Model** dialog box, define the following parameters:
   - **Name**: Housing_maxy
   - **Thermal Specification**: Thickness
   - **Thickness**: 0.001 meter
   - **Solid Material**: Polystyrene-rigid-R12
   - **Shell Conduction**: Enabled
8. Click **OK**.
9. Repeat steps 7 and 8 for **Housing_maxz**, **Housing_miny**, and **Housing_minz**.



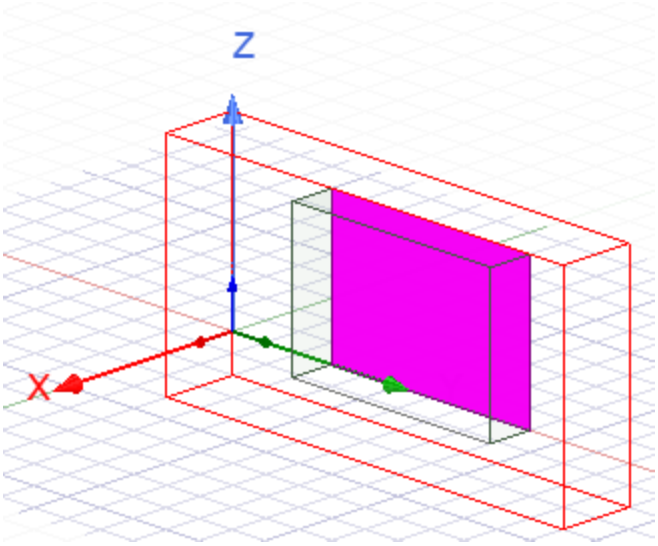
Create the Wall

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

1. From the **Draw** menu, select **Rectangle**.
2. In the **CreateRectangle** dialog box **Command** tab, define the following size parameters and units:
   - **Position**: 0 ,0.15 ,0 meter
   - **Axis**: X
   - **Width**: 0.3 meter
   - **Length**: 0.2 meter
3. On the **Attribute** tab, enter **Xmin** for the **Name**.
4. Click **OK**.
5. In the History tree, expand **Model** > **Sheets** > **Unassigned**.
6. Right-click **Xmin** and select **Assign Thermal** > **Wall** > **Stationary**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. In the **Stationary Wall Thermal Model** dialog box, define the following parameters:
   - **Name**: Xmin
   - **Wall Thickness**: 0.001 meter
   - **Solid Material**: Polystyrene-rigid-R12
   - **External Condition**: Heat Transfer Coefficient
   - **Heat Transfer Coefficient**: 5 w_per_m2kel
8. Click **OK**.



Create the PCB

Model the PCB using box geometry and a user-defined material with custom properties.

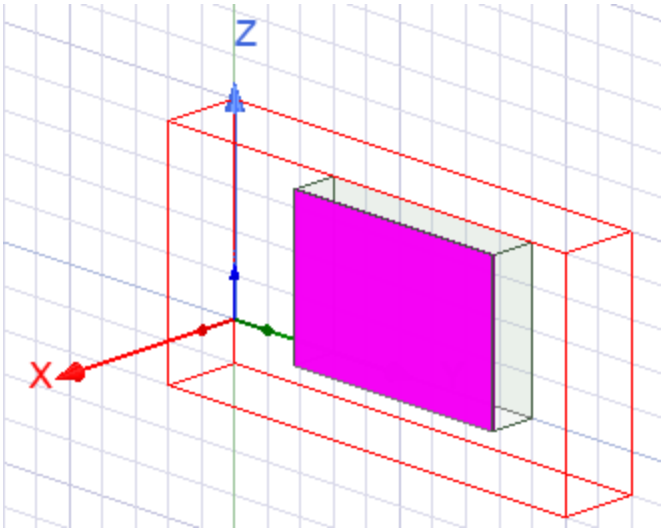
Create the PCB Material

1. From the **Tools** menu, select **Edit Libraries** > **Materials**.
2. In the **Edit Libraries** dialog box, Click **Add Material**.
3. In the **View/Edit Materials** dialog box, define the following properties:
   - **Name**: pcb_1_substrate
   - **Thermal Conductivity Type**: Anisotropic
   - **T(1,1) Value**: 0.361276 W/m-C
   - **T(2,2) Value**: 9.30516 W/m-C
   - **T(3,3) Value**: 9.30516 W/m-C
   - **Mass Density Value**: 1427.669375 kg/m^3
   - **Specific Heat Value**: 1277.8579921875 J/kg-C
4. Click **OK** to close the **View/Edit Materials** dialog box.
5. Click **OK** to close the **Edit Libraries** dialog box.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create the PCB Geometry and Assign the Material

1. From the **Draw** menu, select **Box**.
2. In the **CreateBox** dialog box, define the following size parameters and units:
   - **Position**: 0.0584 ,0.15 ,0 meter
   - **XSize**: 0.0016 meter
   - **YSize**: 0.3 meter
   - **ZSize**: 0.2 meter
3. On the **Attribute** tab, define the following parameters:
   - **Name**: pcb_1_substrate
   - **Material**: pcb_1_substrate
4. Click **OK**.



Create the Devices

There are twelve devices on the bottom side of the PCB. Model the devices as rectangle geometry and assign source thermal boundary conditions to define thermal properties.

Create a Device

1. From the **Draw** menu, select **Rectangle**.
2. In the **CreateRectangle** dialog box **Command** tab, define the following size parameters and units:
   - **Position**: 0.0584 ,0.194 ,0.035 meter
   - **Axis**: X
   - **Width**: 0.016 meter
   - **Length**: 0.02 meter

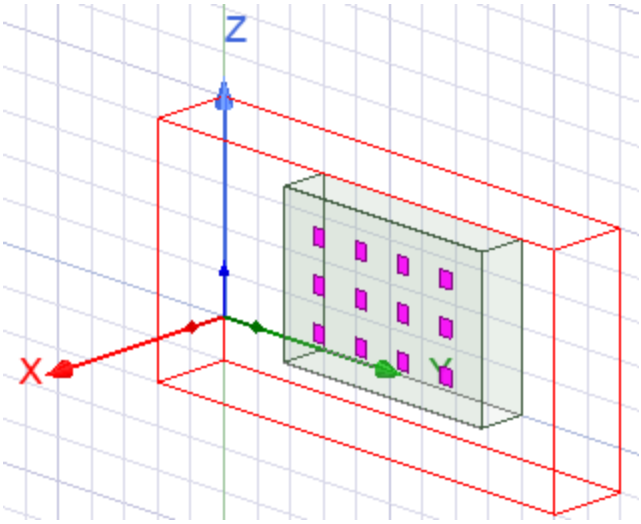
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. On the **Attribute** tab, enter **device** for the **Name**.
4. Click **OK**.

Create Duplicates of the Device

Using the first device as a template, create two duplicates in the Z direction. Then create three duplicates of those devices in the Y direction to create all twelve.

1. In the History tree, expand **Model** > **Sheets** > **Unassigned**.
2. In the History tree, right-click **device** and select **Edit** > **Duplicate** > **Along Line**.
3. In the **DuplicateAlongLine** dialog box, enter **3** for **Total Number**.
4. Enter **0 ,0 ,0.055** for the **Vector** to copy the device in the Z direction.
5. Change the **Unit** to **meter**.
6. Click **OK**.
7. In the History tree, expand **Model** > **Sheets** > **Unassigned**.
8. In the History tree, use the **Ctrl** key to select **device**, **device_1**, and **device_2**.
9. Right-click and select **Edit** > **Duplicate** > **Along Line**.
10. In the **DuplicateAlongLine** dialog box, enter **4** for **Total Number**.
11. Enter **0 ,0.064 ,0** for the **Vector** to copy the devices in the Y direction.
12. Change the **Unit** to **meter**.
13. Click **OK**.



Assign the Source Boundary Condition

Each device has a total power of 7 W. When assigning a boundary condition to multiple objects, the thermal properties are applied to all twelve devices.

1. In the History tree under **Model** > **Sheets**, right-click **Unassigned** and select **Select All**.
2. Right-click select **Assign Thermal** > **Source**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **Source Thermal Model** dialog box, define the following parameters:
    - **Name**: Devices
    - **Thermal Condition**: Total Power
    - **Total Power**: 7 W
4. Click **OK**.

Create the Heatsink

Create an extruded fin heat sink with the flow in the Y direction to remove heat from the PCB. Use the **Move** feature to position the heatsink.

1. In the **Project Manager**, right-click **3D Components** and select **Create** > **Heatsink**.
2. On the **Heatsink Component Geometry** tab, define the following parameters:
    - **Plane**: YZ
    - **Overall height**: 0.04 meter
    - **Base Length**: 0.3 meter
    - **Base Width**: 0.2 meter
    - **Base Height**: 0.004 meter
    - **Fin Type**: Extruded
    - **Flow Direction**: Y
    - **Fin Count**: 9
    - **Fin Thickness**: 0.002 meter
    - **Y Offset**: 0 meter
    - **Z Offset**: 0 meter
3. On the **Properties** tab, review the **Solid Material** and **Surface Material** selections for the fins and base.
4. Click **OK**.
5. In the History tree under **Heatsink1**, right-click **Heatsink1_1** and select **Edit** > **Arrange** > **Move**.
6. In the **Move** dialog box, enter **0.06 ,0.3 ,0.1** for the **Move Vector**.
7. Change the **Unit** to **meter**.
8. Click **OK**.

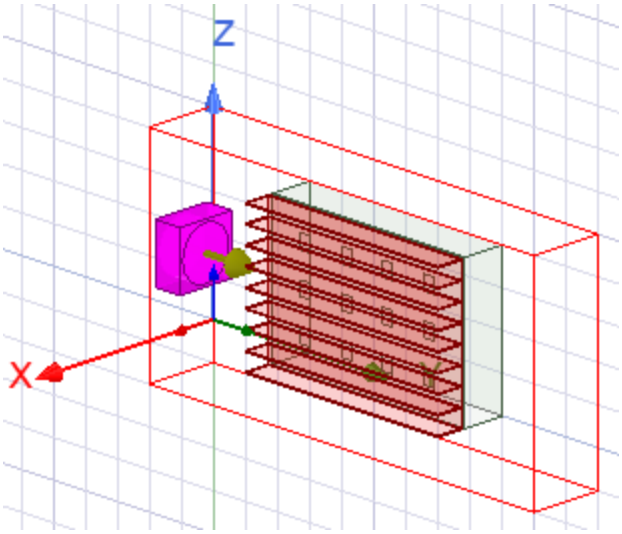
Create the Fan

For this model, add an existing fan from the Component Libraries.

> **Note:** Ensure the **Component Libraries** window is displayed by selecting **Component Libraries** from the **View** menu.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Component Libraries** window, expand **Icepak Components** > **Fan** > **Delta**.
2. Double-click **Delta_FFB0812_24EHE** to add the fan component to the model.
3. In the **Project Manager** under **3D Components**, right click **delta_FFB0812_24EHE** and select **Edit Definition**.
4. In the **Fan Component** dialog box **Geometry** tab, change the **Cross-section** to **ZX**.
5. On the **Properties** tab, review the thermal and flow properties.
6. Click **OK**.
7. In the History tree, expand **delta_FFB0812_24EHE**.
8. Right-click **delta_FFB0812_24EHE1** and select **Edit** > **Arrange** > **Move**.
9. In the **Move** dialog box, enter **0.05 ,0 ,0.1** for the **Move Vector Value**.
10. Change the **Unit** to **meter**.
11. Click **OK**. Review the final geometry.

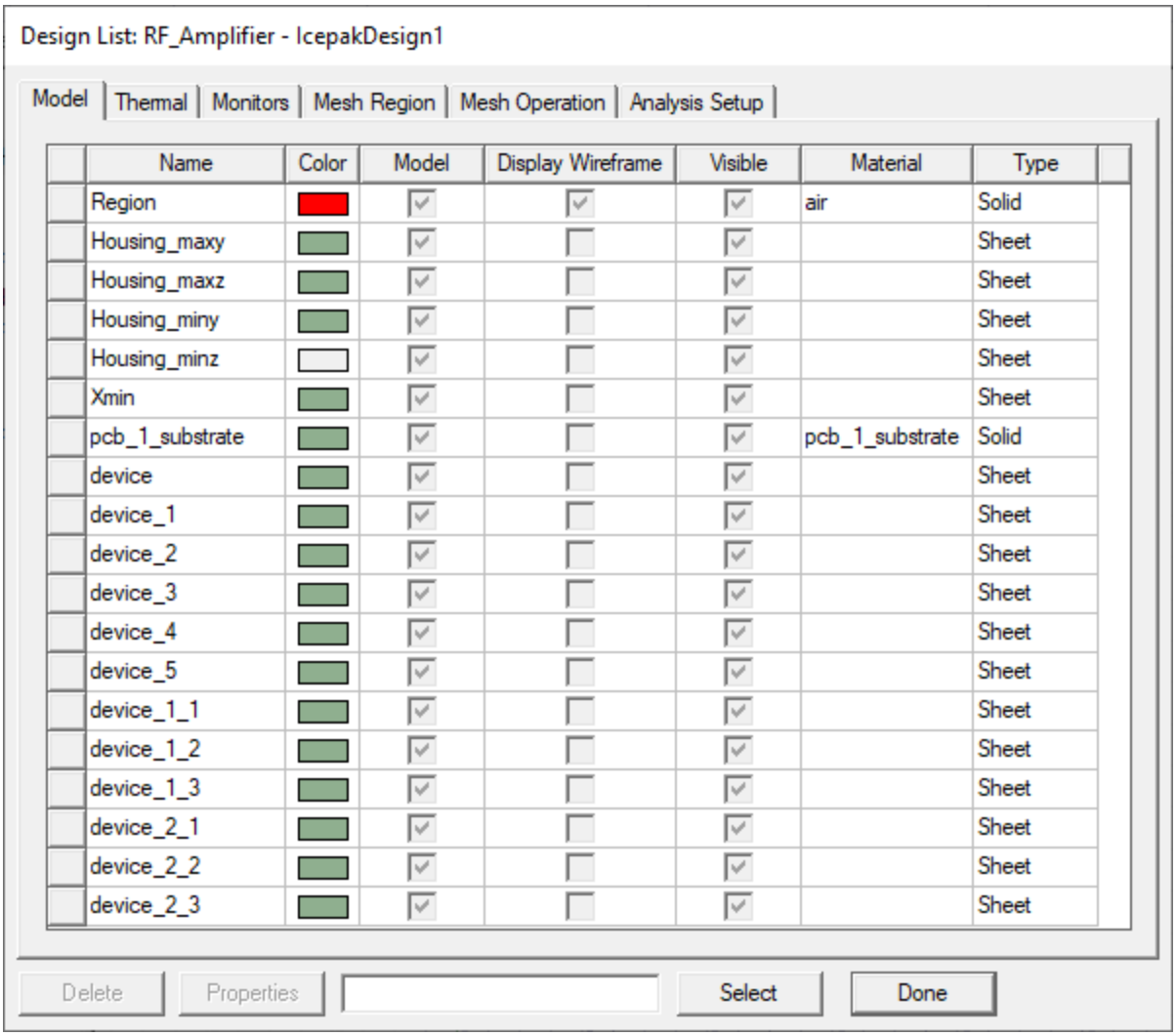


View the Design List

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.

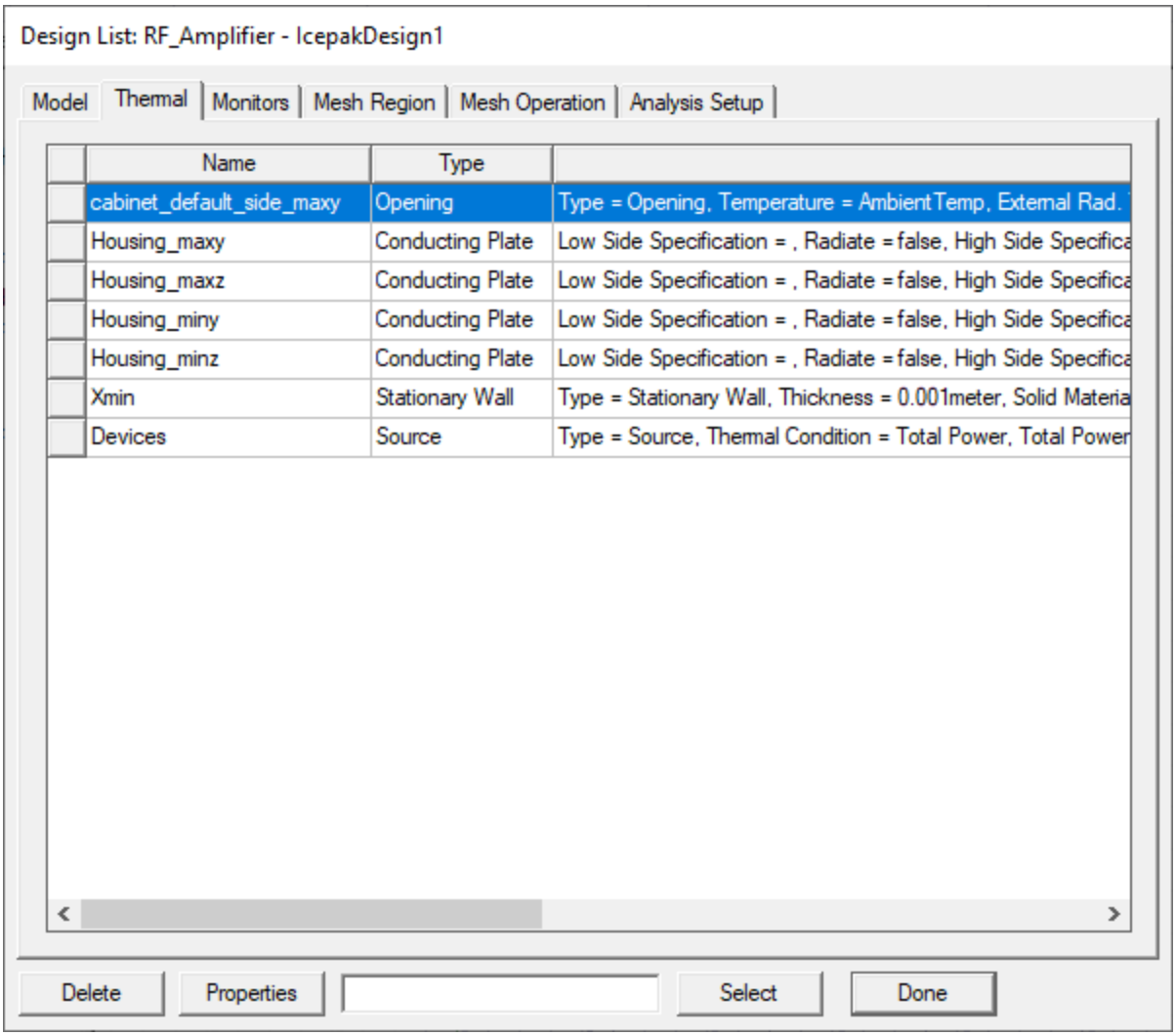
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	Color	Model	Display Wireframe	Visible	Material	Type
Region		☑	☑	☑	air	Solid
Housing_maxy		☑	☐	☑		Sheet
Housing_maxz		☑	☐	☑		Sheet
Housing_miny		☑	☐	☑		Sheet
Housing_minz		☑	☐	☑		Sheet
Xmin		☑	☐	☑		Sheet
pcb_1_substrate		☑	☐	☑	pcb_1_substrate	Solid
device		☑	☐	☑		Sheet
device_1		☑	☐	☑		Sheet
device_2		☑	☐	☑		Sheet
device_3		☑	☐	☑		Sheet
device_4		☑	☐	☑		Sheet
device_5		☑	☐	☑		Sheet
device_1_1		☑	☐	☑		Sheet
device_1_2		☑	☐	☑		Sheet
device_1_3		☑	☐	☑		Sheet
device_2_1		☑	☐	☑		Sheet
device_2_2		☑	☐	☑		Sheet
device_2_3		☑	☐	☑		Sheet

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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

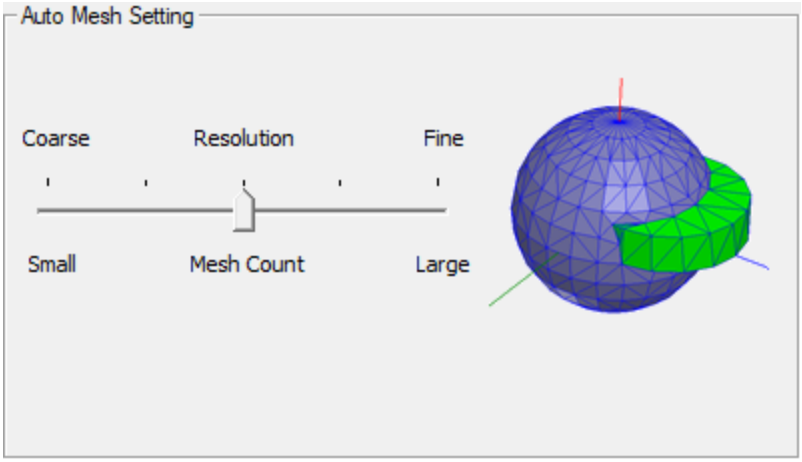
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 - Generate and Display Coarse 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.



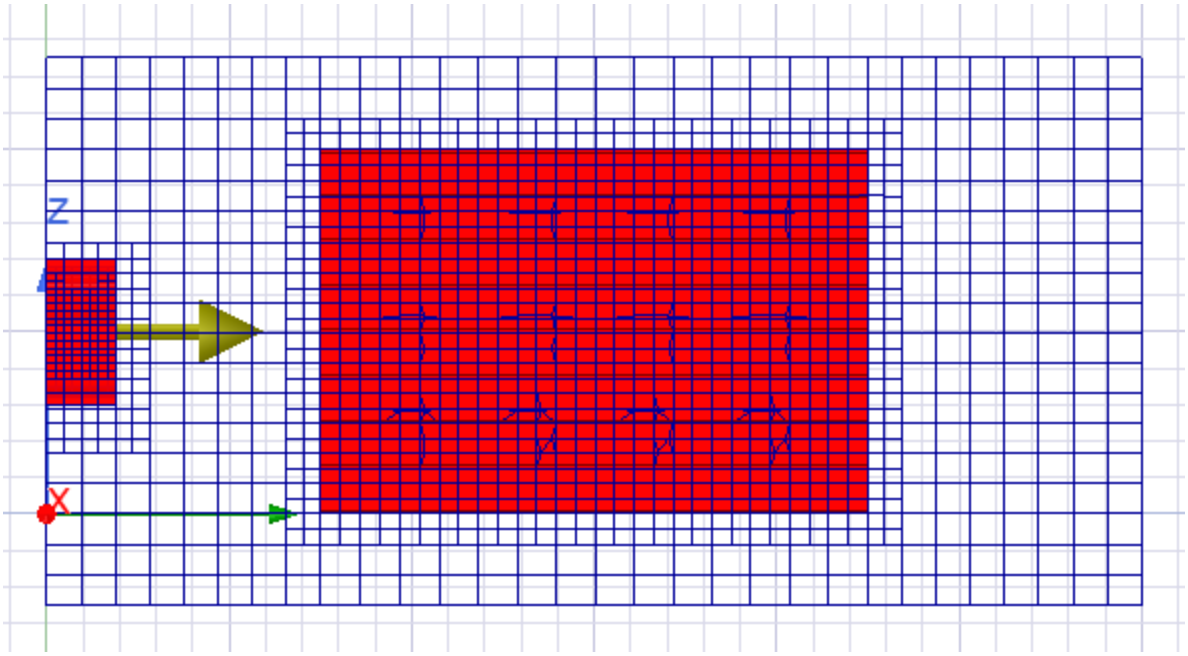
4. Click **OK**.
5. In the **Project Manager**, right-click **Mesh** and select **Generate Mesh**.

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. On the **Model** ribbon, select **Front** from the **Orient** drop-down list.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click **Close**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4 - 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 wall, enclosure plates, 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. In the **CreateBox** dialog box, define the following size parameters and units:
   - **Position**: 0 ,0.13 ,-0.01 meter
   - **XSize**: 0.1 meter
   - **YSize**: 0.37 meter
   - **ZSize**: 0.22 meter
3. On the **Attribute** tab, define the following parameters:
   - **Name**: amplifier_meshregion
   - **Model**: Disabled
4. Click **OK**.

Assign a Mesh Region to the Non-Model Box

1. In the History tree, expand **Model** > **Solids** > **Non-Model**.
2. Right-click **amplifier_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 **Amplifier** 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 **delta_FFB0812_24EHE1**.
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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **+X**: 0.01 mm
- **-X**: 0.01 mm
- **+Y**: 0.05 mm
- **-Y**: 0 mm
- **+Z**: 0.01 mm
- **-Z**: 0.01 mm

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**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5 - Generate and Display Refined Mesh

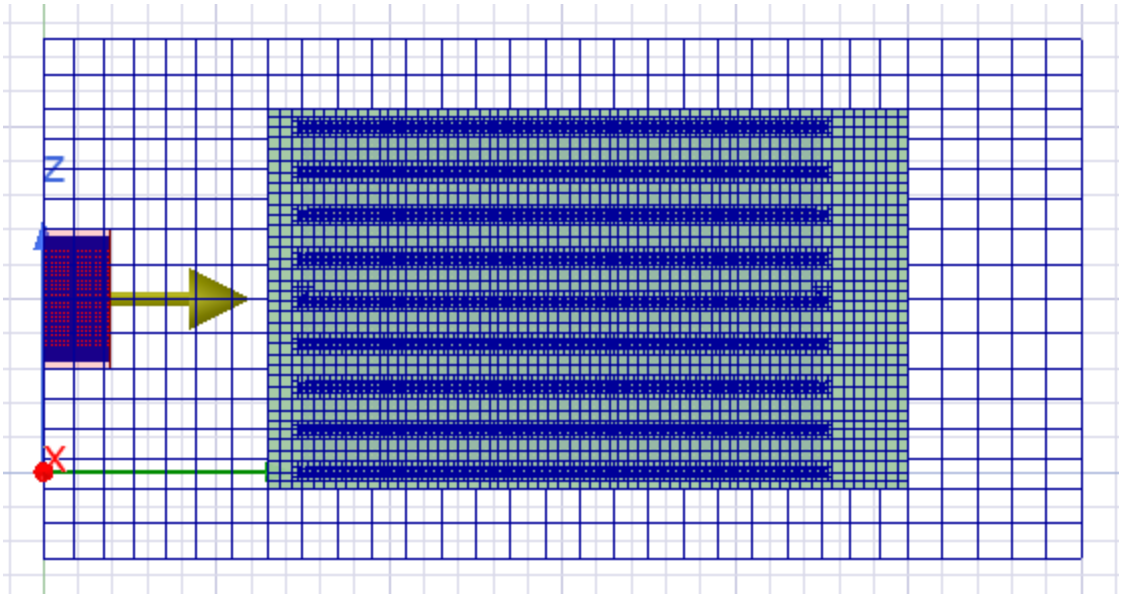
After creating the mesh regions, generate and display the mesh.

Generate and Display Refined 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 **Model** ribbon, select **Front** from the **Orient** drop-down list.

Display the Refined 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** > **delta_FFB0812_24EHE** > **delta_FFB0812_24EHE1** > **Solids** > **AL-Extruded**.
4. Use the **Ctrl** key to deselect **delta_FFB0812_24EHE_Case**.



5. Click **Close**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6 - Create Monitor Points

Before running the simulation, create thermal and flow monitors for device_1_1 and cabinet_default_side_maxy.

Create a Thermal Monitor

1. In the History tree, expand **Model** > **Sheets** > **Source**.
2. Right-click **device_1_1** and select **Assign Monitor** > **Point**.
3. In the **Monitor Setup** dialog box, enter **Device_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 **cabinet_default_side_maxy** and select **Select Assignment**.
2. From the **Icepak** menu, select **Monitor** > **Assign** > **Point**.
3. In the **Monitor Setup** dialog box, enter **Cabinet_opening_speed** for the **Name**.
4. Expand **Flow** and select **Speed**.
5. Click **OK**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7 - Define the Simulation Settings and Run the Analysis

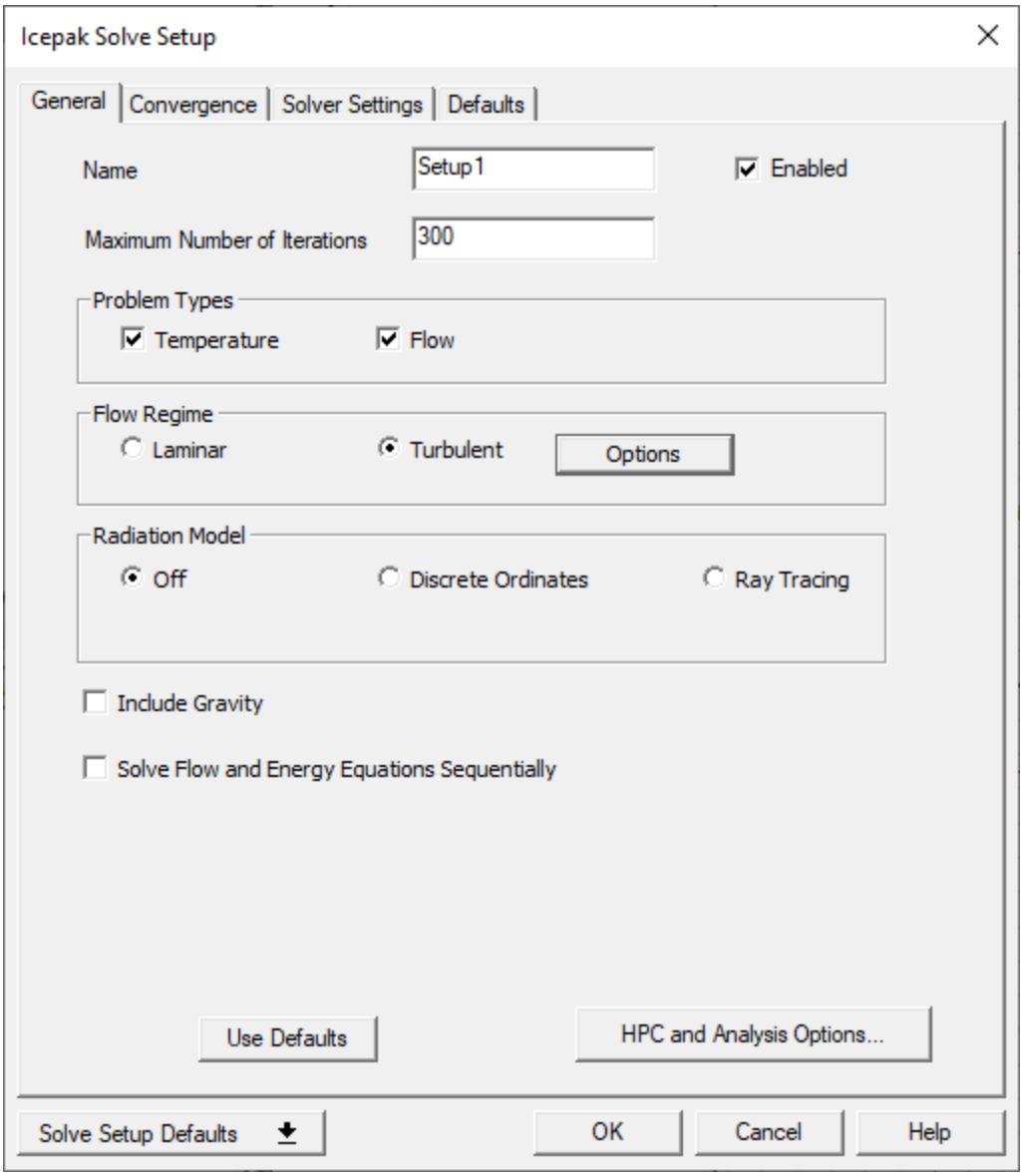
Define the Design Settings

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 **300**.
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.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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

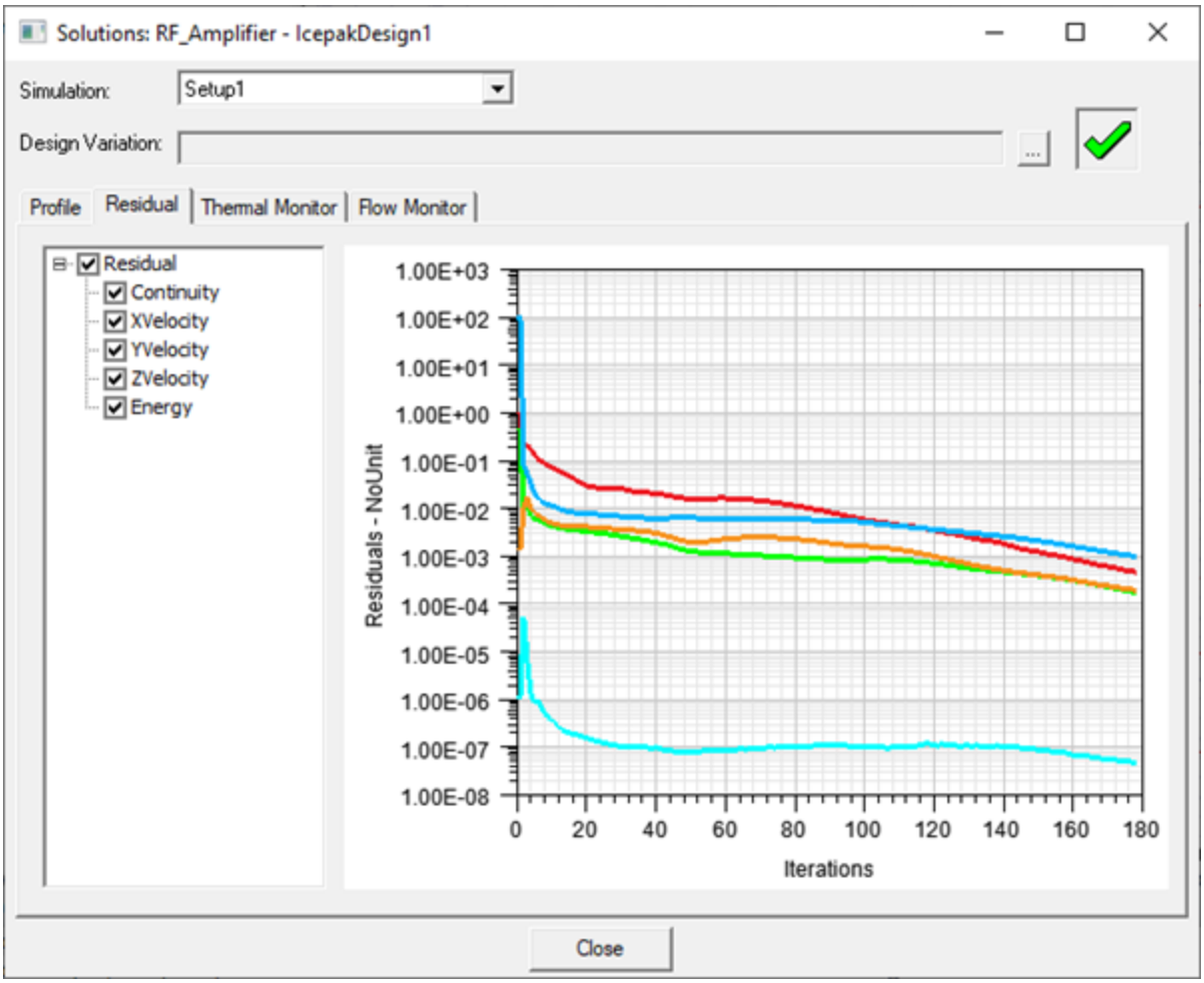
Run the Simulation

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

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

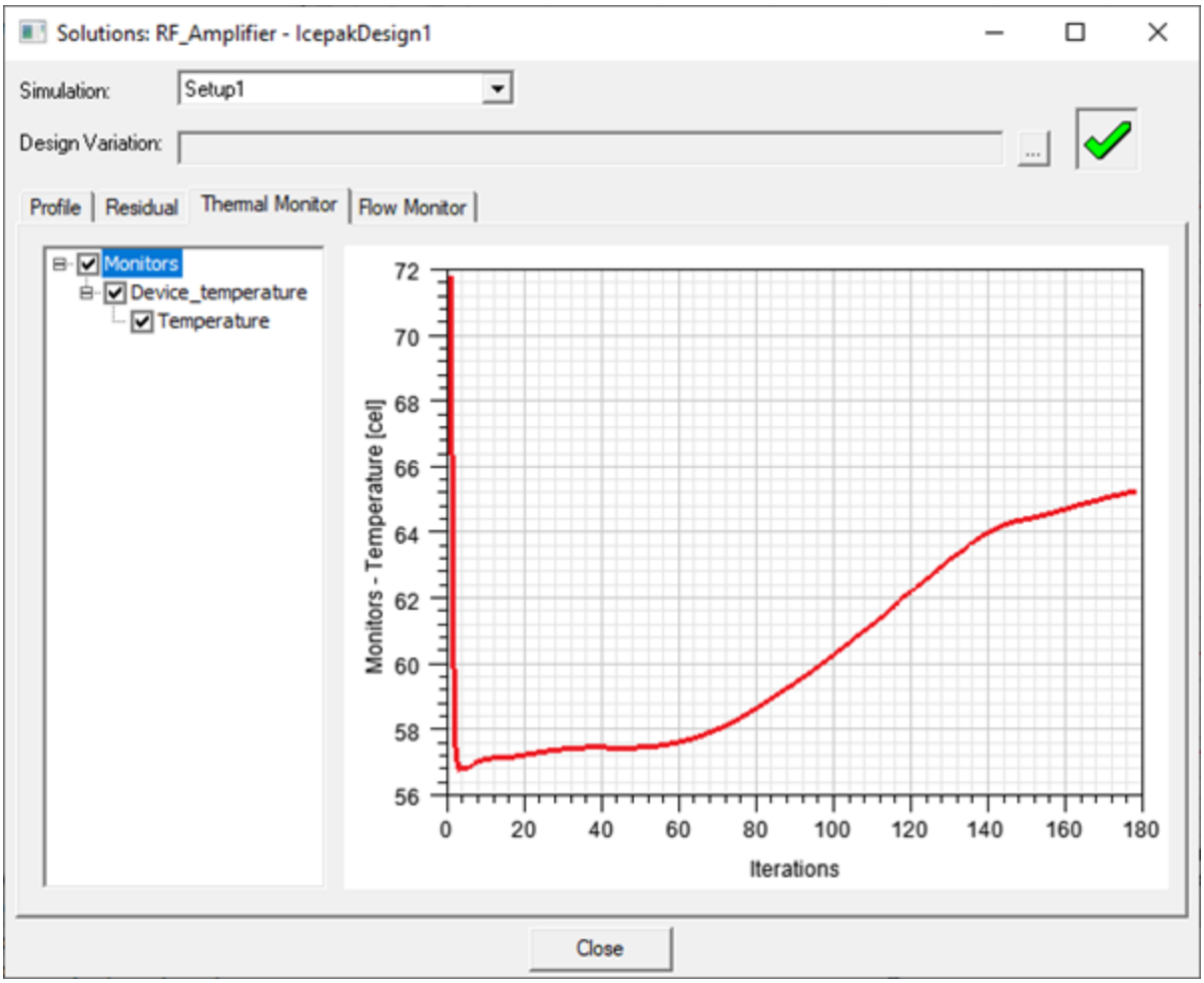
Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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



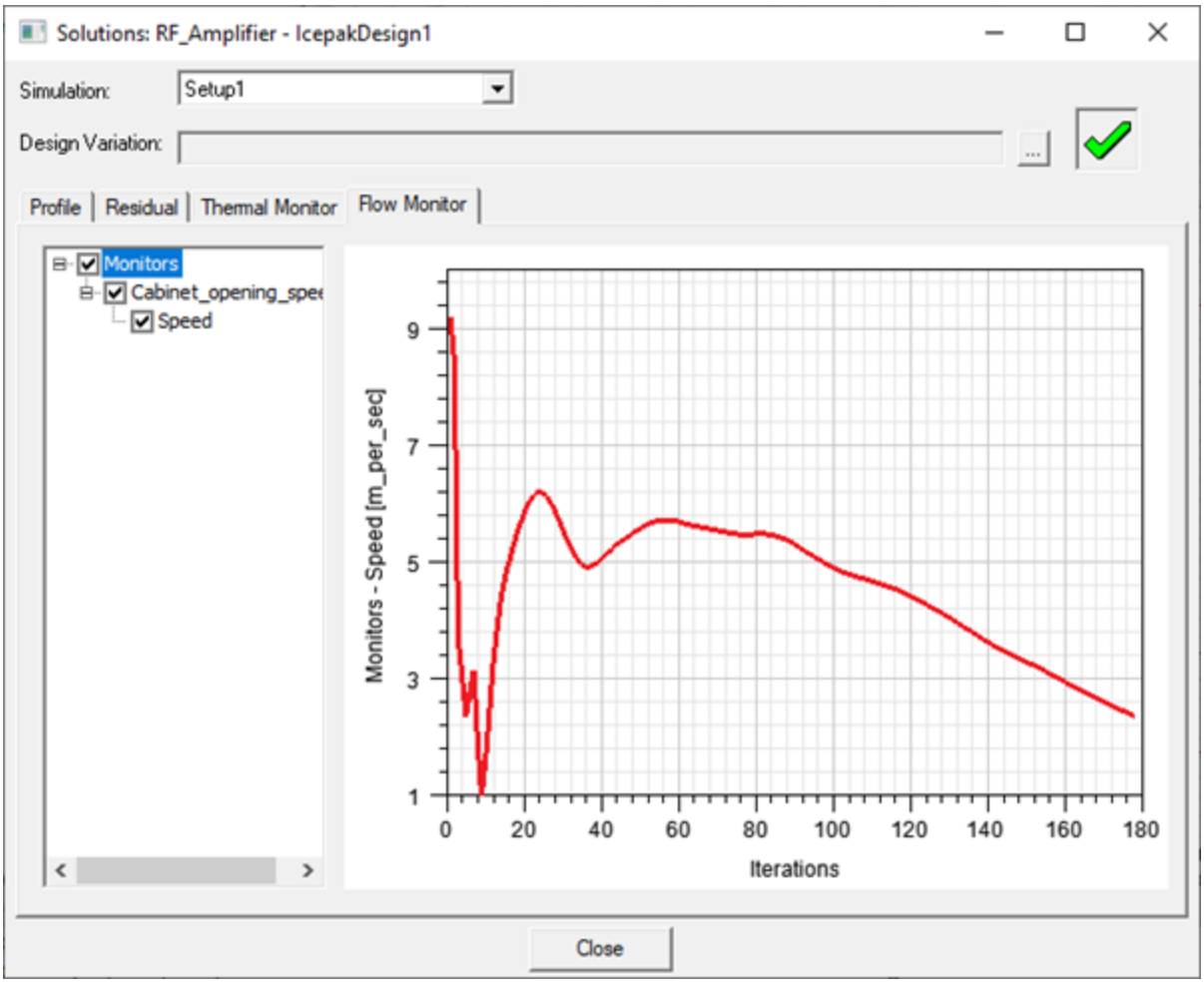
4. Click the **Thermal Monitor** tab to review the temperature monitor on the device.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Click the **Flow Monitor** tab to review the speed monitor on the cabinet opening.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. Click **Close**

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8 - Post-process the Results

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

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

Create Object Field Overlays

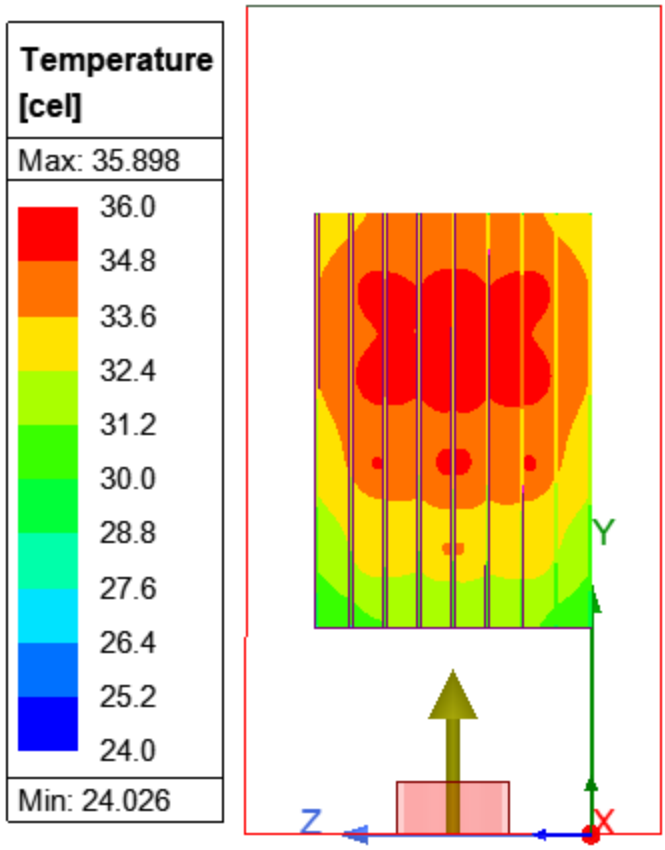
Plot Temperature on the Heat Sink

1. In the History tree, expand **Heatsink1**.
2. Right-click **Heatsink1_1** and select **Select All**.
3. Right-click in the **3D Modeler** window and select **Plot Fields** > **Temperature** > **Temperature**.
4. In the **Create Field Plot** dialog box, enable **Specify Name** and enter **Heatsink**.
5. Retain the **Temperature** selection for **Quantity**.
6. Enable **Plot on surface only**.
7. Click **OK**.

> **Note:** The field overlay appears in the **Project Manager** under **Field Overlays** > **Temperature**. You can hide the overlay by right-clicking on it and selecting

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

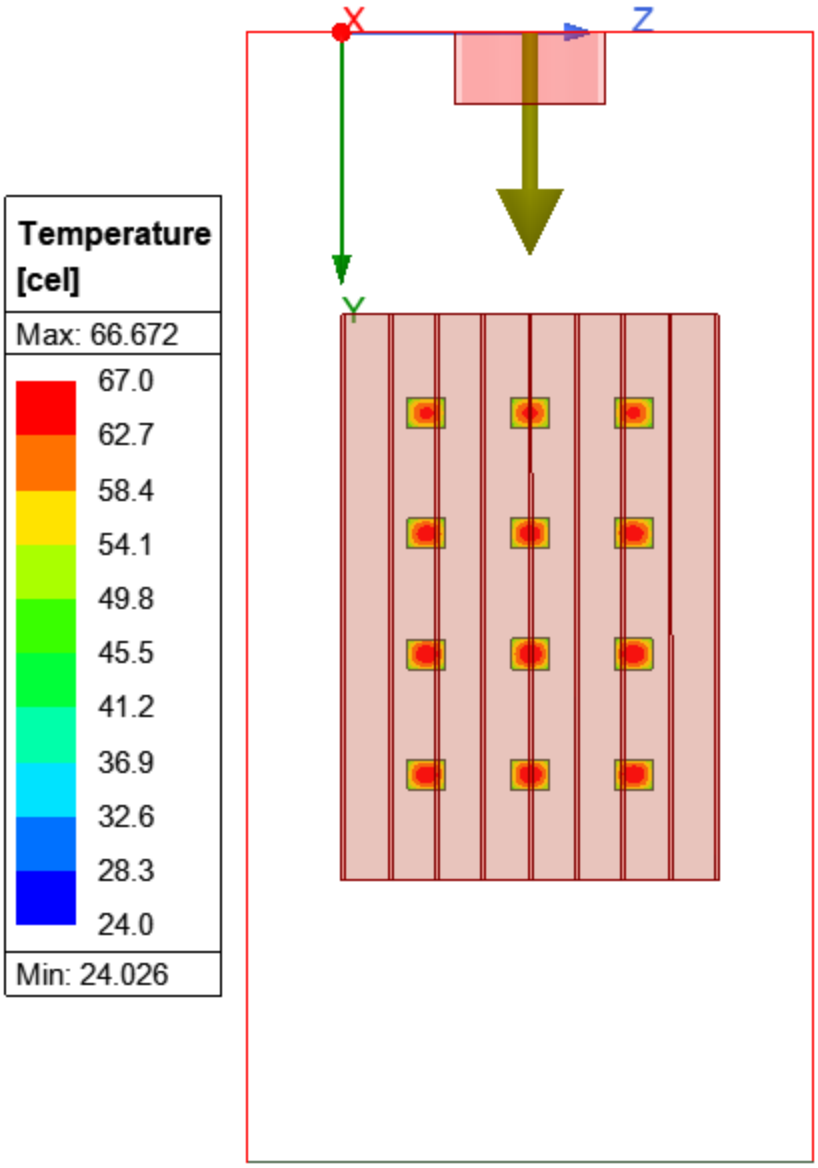
Plot visibility.



Plot Temperature on the Devices

1. In the History tree, expand **Model** > **Sheets**.
2. Right-click **Source** and select **Select All**.
3. Right-click in the **3D Modeler** window and select **Plot Fields** > **Temperature** > **Temperature**.
4. In the **Create Field Plot** dialog box, enable **Specify Name** and enter **Devices_temperature**.
5. Retain the **Temperature** selection for **Quantity** and click **Done**.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create Plane Cut Field Overlays

Create a Plane

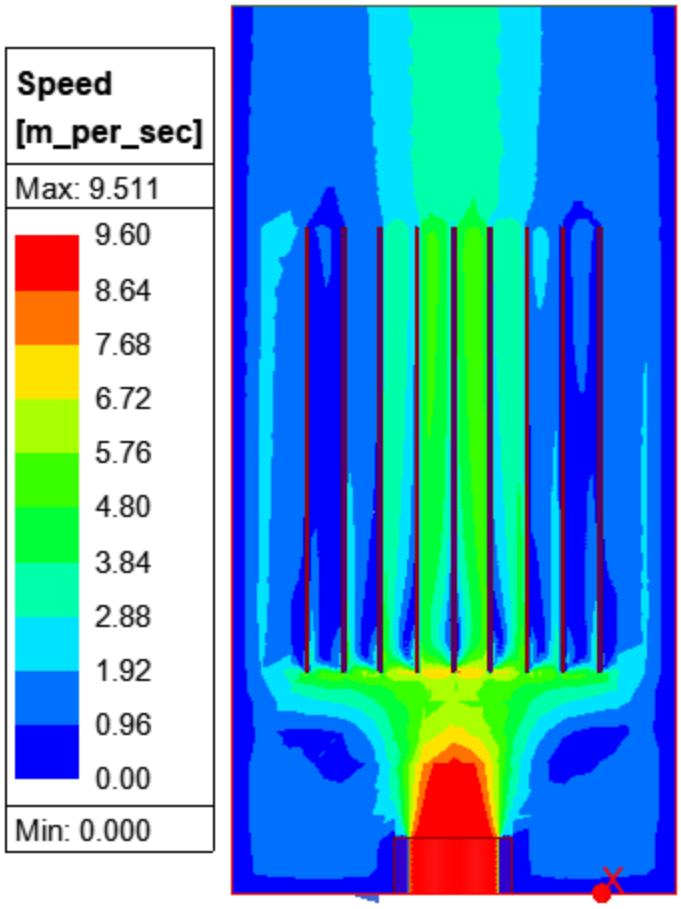
1. From the **Draw** menu, select **Plane**.
2. In the **3D Modeler** window, click twice to draw a plane.
3. In the History tree, expand **Planes** and select **Plane1**.
4. In the **Properties** window, edit the following properties:
    - **Name**: cut-plane

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Root point**: 0.068 ,0.31 ,0.136 meter
- **Normal**: 1 ,0 ,0 meter

Plot Speed

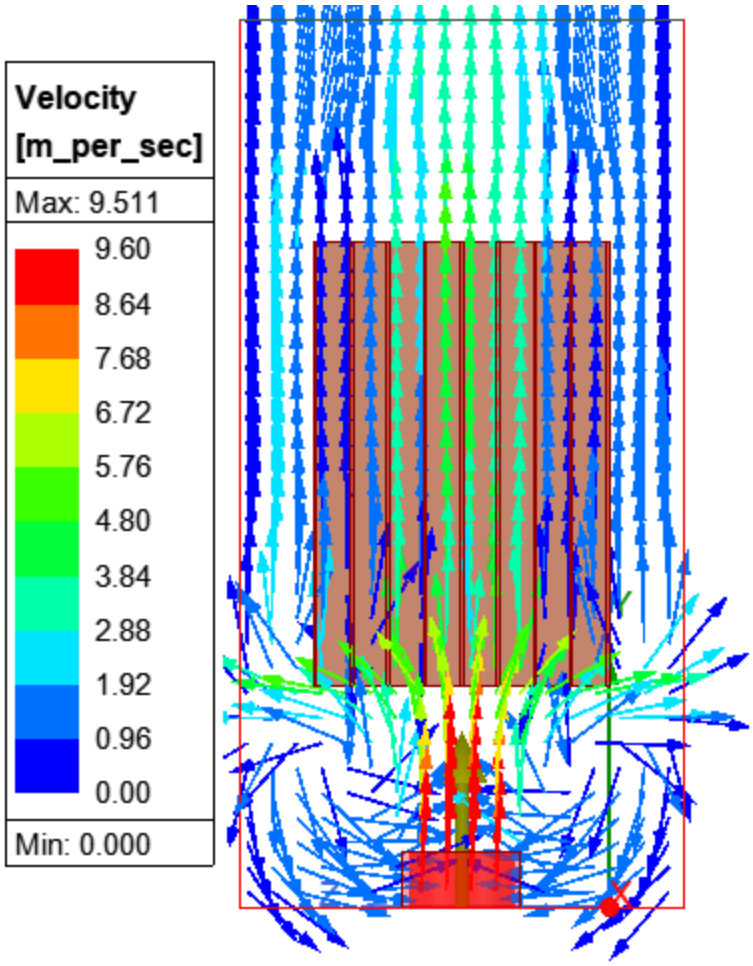
1. In the History tree, select **cut-plane**.
2. Right-click in the **3D Modeler** window and select **Plot Fields** > **Velocity** > **Speed**.
3. In the **Create Field Plot** dialog box, enable **Specify Name** and enter **cut-speed**.
4. Retain the **Speed** selection for **Quantity** and click **OK**.



5. In the **Project Manager**, expand **Field Overlays** > **Speed**.
6. Right-click **cut-velocity** and select **Modify Plot**.
7. In the **Modify Plot** dialog box, select **Velocity Vectors** for the **Quantity** and click

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Done.overlay_cut-velocity



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.

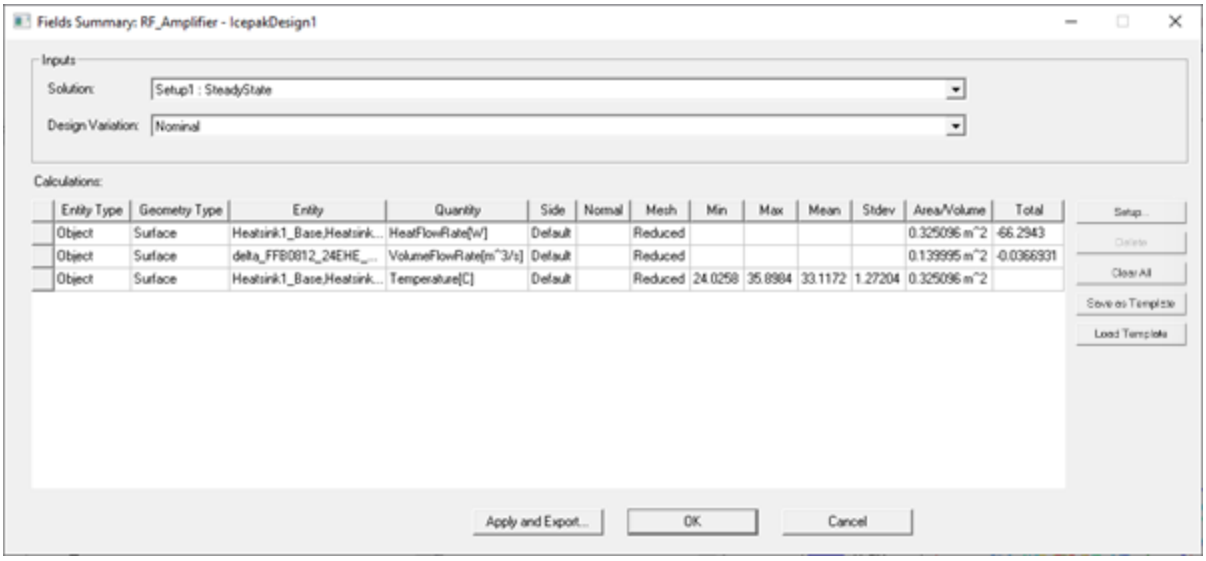
1. From the **Icepak** menu, select **Fields** > **Create Fields Summary**.
2. In the **Setup Calculation** dialog box, select **Object** for **Entity Type**.
3. In the **Entity** list, select all of the **Heatsink** objects.
4. In the **Quantity** list, select **HeatFlowRate**.
5. From the **Add** drop-down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

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

Entity	Quantity
delta_FFB0812_24EHE_Case, delta_FFB0812_24EHE_Passage, delta_FFB0812_24EHE_Surface	VolumeFlowRate
All Heatsink objects	Temperature

Note: When creating the entry for temperature on all heatsink objects, enable **Include all surface meshes** before adding the calculation.



Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9 - Summary

In this tutorial, you have learned about the basic usage of model using openings, fans, sources, blocks, heat sinks, and walls to create an RF amplifier. You have also learned how to use the Ansys Electronics Desktop Component Libraries to add a fan to a design. Furthermore, this tutorial has introduced you to evaluating mesh quality as well as non-conformal meshing to reduce cell count and computational cost.

Ansys Electromagnetics Suite 2026 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.