Ansys
2025 R2
POWERING INNOVATION THAT DRIVES HUMAN ADVANCEMENT

Getting Started with Icepak®: Cold Plate Model

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

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

Conventions Used in this Guide

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

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

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

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

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

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

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

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

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

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

Getting Help: Ansys Technical Support

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

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

Help Menu

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

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

Context-Sensitive Help

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

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

Table of Contents

1 - Introduction

This document is intended as supplementary material to Icepak for beginners and advanced users. It demonstrates how to model a cold plate.

In this tutorial you will learn how to:

- Use a mesh region and mesh operation to mesh complicated model setups
- Use multiple fluids in a single model.
- Account for external natural convection and internal forced convection.
- Create mesh regions to reduce the overall mesh count.
- Specify per-object meshing parameters.

This chapter contains the following topic:

- "Sample Project - The Cold Plate" below

Sample Project - The Cold Plate

The model consists of a cold-plate, where the cold-plate fluid is transporting a significant fraction of the heat from two plates mounted on either side of it. The natural convection in the external air is also instrumental to heat transfer in this case. The objective of this exercise is to illustrate the use of two different fluids

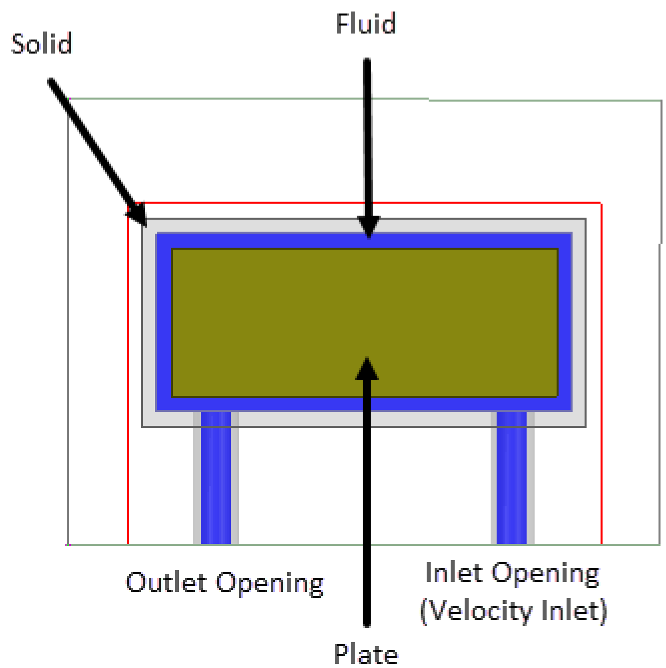


Figure 1-1: Cold Plate

2 - Open the Project and Review the Model

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

Launch the Ansys Electronics Desktop

A shortcut of the Ansys Electronics Desktop application appears on your desktop once the application is installed.

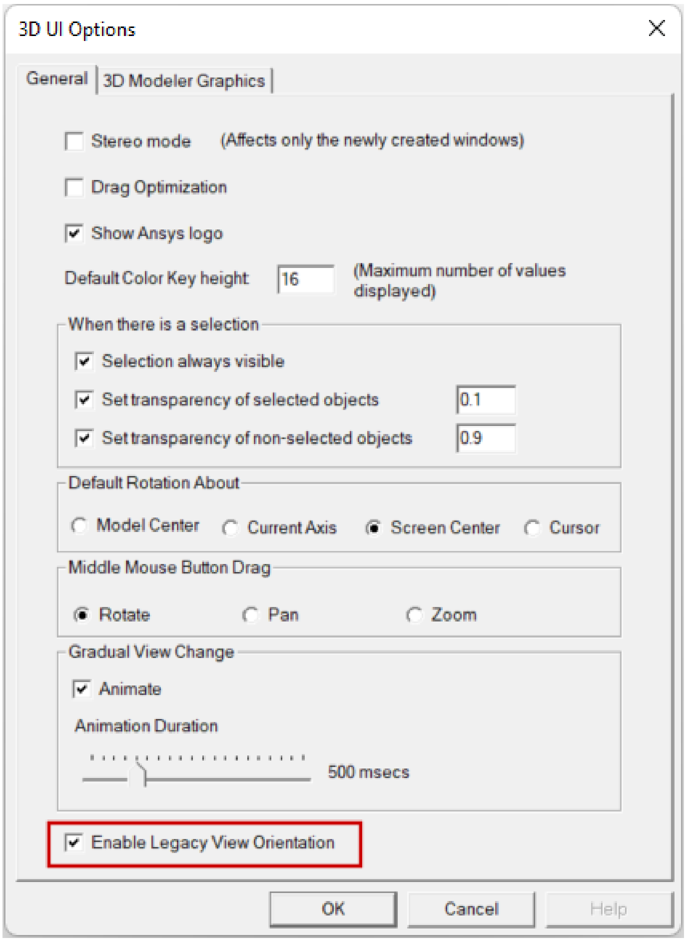
Set 3D UI Options

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**.

Open the Project

1. On the **Desktop** ribbon tab, click  **Open Examples**.
2. Double-click the **Icepak** folder.
3. Select the file **Cold_Plate.aedt** and click **Open**.
4. The model is displayed in the **3D Modeler** window.

   Note: You can hide the grid by selecting **View** > **Grid Settings** and then selecting **Hide** in the **Grid Spacing** dialog box. Also, from the **View** > **Coordinate Systems** menu, you can hide the large coordinate triad and display a smaller coordinate triad in the bottom of the **3D Modeler** window.

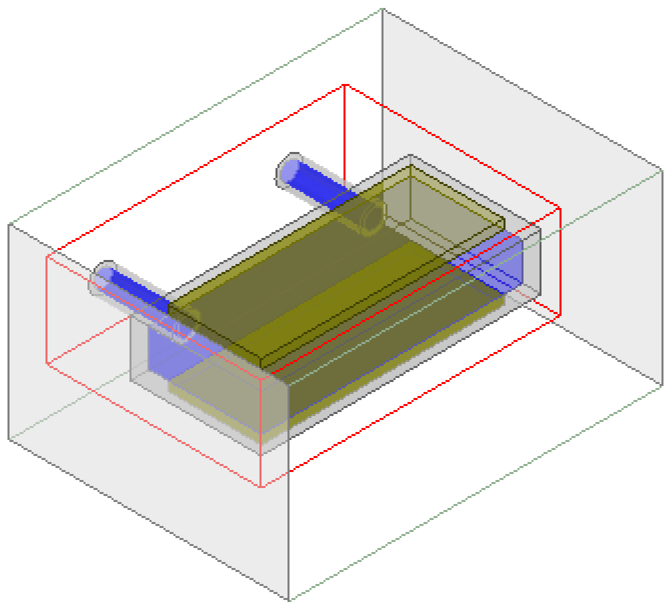
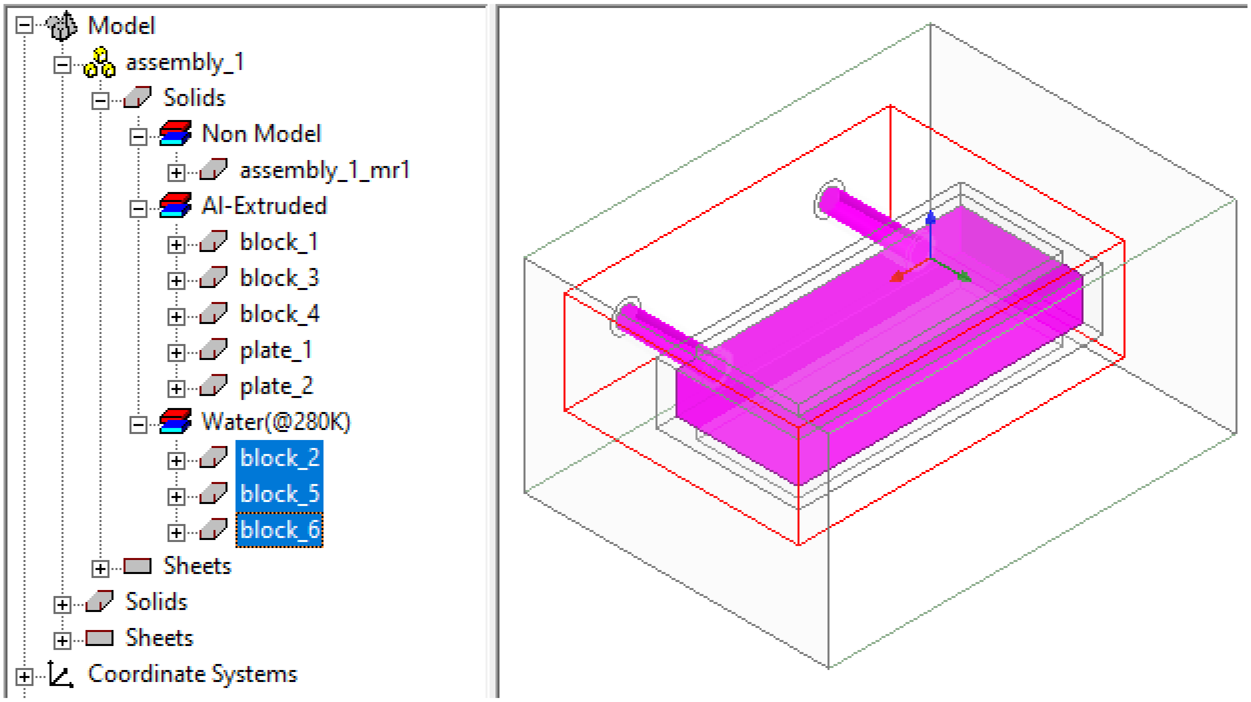


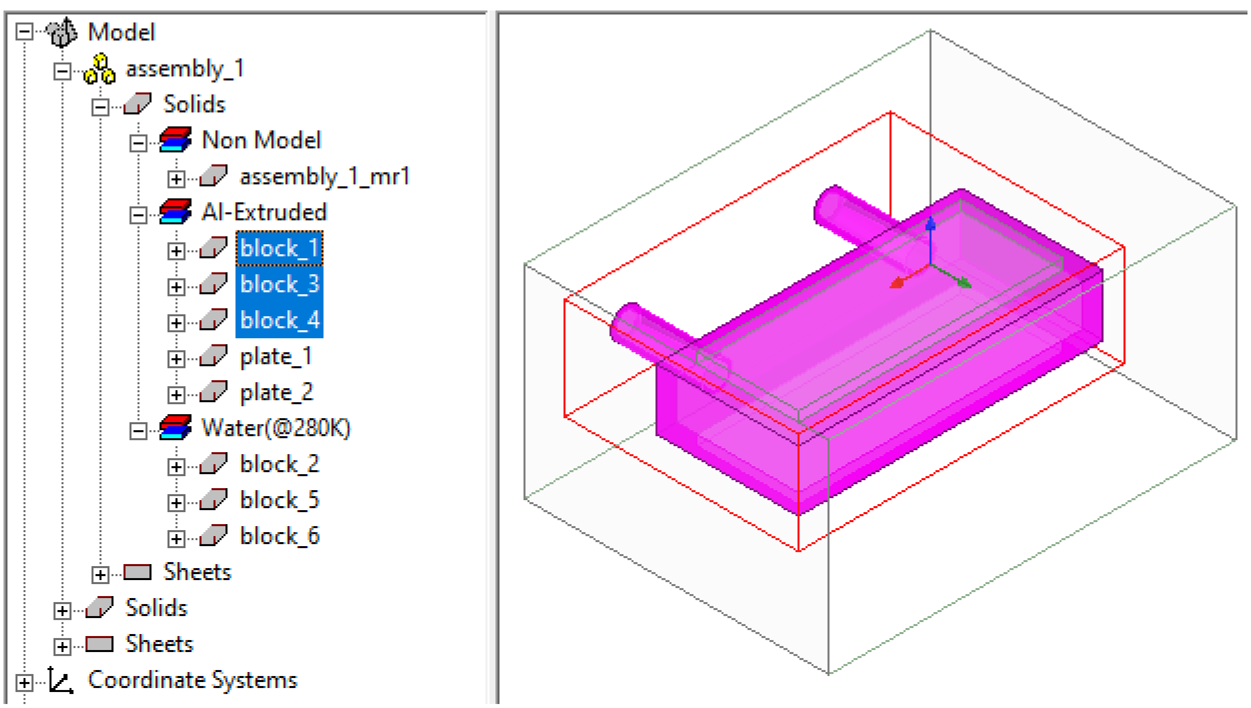
Figure 2-1: Cold Plate model in the 3D Modeler window

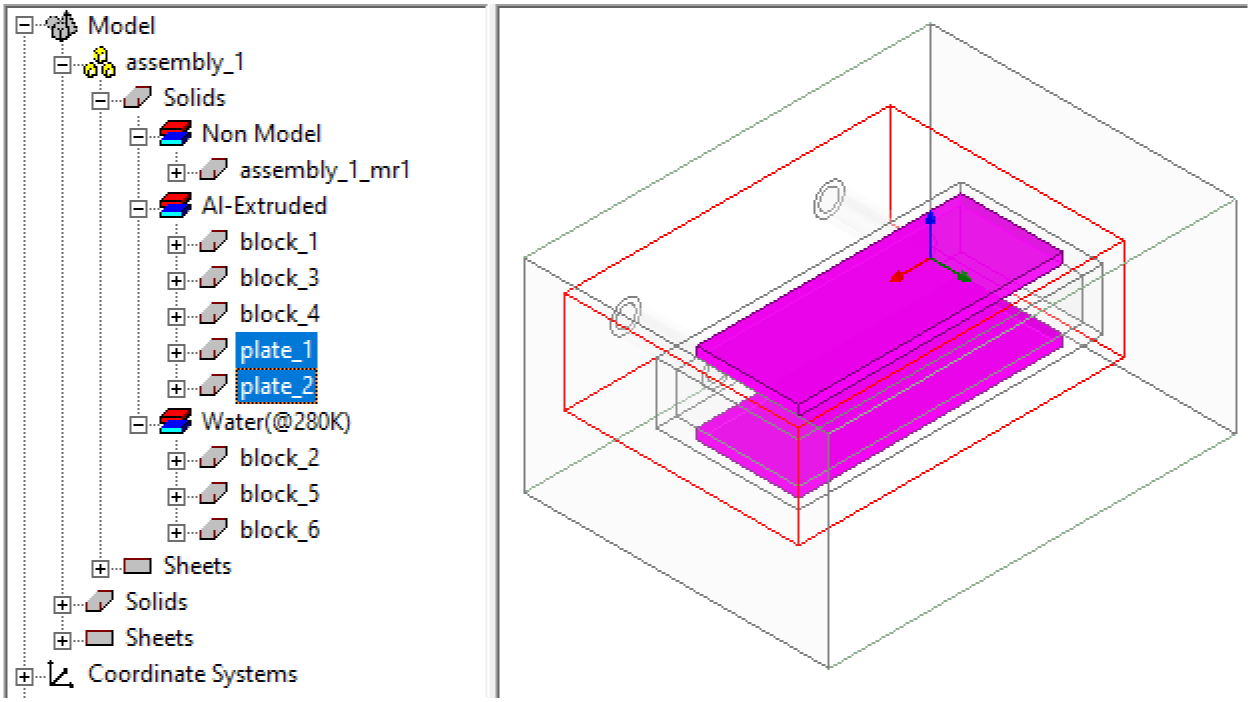
5. From the **File** menu, select **Save As**, and save the project in the desired working directory.

Review the Model

The model includes two heated plates, cooled by water circulating inside the cold-plate cavity, as well as by air driven by natural convection externally. Mesh regions will be employed to reduce the overall mesh count in the domain.


Model
assembly_1
Solids
Non Model
assembly_1_mr1
Al-Extruded
block_1
block_3
block_4
plate_1
plate_2
Water(@280K)
block_2
block_5
block_6
Sheets
Solids
Sheets
Coordinate Systems


Model
assembly_1
Solids
Non Model
assembly_1_mr1
Al-Extruded
block_1
block_3
block_4
plate_1
plate_2
Water(@280K)
block_2
block_5
block_6
Sheets
Solids
Sheets
Coordinate Systems


Model
assembly_1
Solids
Non Model
assembly_1_mr1
Al-Extruded
block_1
block_3
block_4
plate_1
plate_2
Water(@280K)
block_2
block_5
block_6
Sheets
Solids
Sheets
Coordinate Systems

3 - Assign Boundary Conditions

Assign Openings

Assign Cabinet Openings

The openings at the cabinet boundary define external air natural convection.

1. Press **F** to enter face selection mode.
2. In the **3D Modeler** window, select the +X face of the Region.
3. Right-click and select **Assign Thermal** > **Opening** > **Free**.
4. In the **Opening Thermal Model** dialog box, retain the default settings and click **OK**.
5. In the **3D Modeler** window, select the -X face of the Region.
6. Right-click and select **Assign Thermal** > **Opening** > **Free**.
7. In the **Opening Thermal Model** dialog box, retain the default settings and click **OK**.

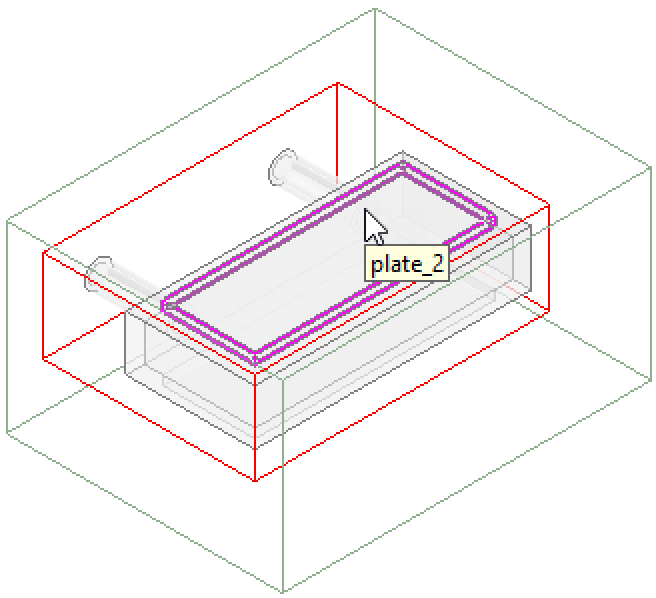
Assign Liquid Inflow and Outflow Openings

The circular openings on the -Y side of the assembly define the inflow and outflow of the liquid cold plate.

1. In the History tree, expand **Model** > **assembly_1** > **Sheets**.
2. Right-click **opening_1** and select **Assign Thermal** > **Opening** > **Free**.
3. In the **Opening Thermal Model** dialog box, enter Outlet for the **Name**, retain the default thermal settings, and click **OK**.
4. In the History tree, right-click **opening_2** and select **Assign Thermal** > **Opening** > **Free**.
5. In the **Opening Thermal Model** dialog box, define the following parameters:
   - **Name**: Inlet
   - **Flow Specification Inlet Type**: Velocity
   - **Y Velocity**: 0.2 m_per_sec
6. Click **OK**.

Assign Plates

1. Press **O** to enter object selection mode.
2. Hover the cursor above plate_2 as shown below and then click to select the geometry.



3. Right-click and select **Assign Thermal** > **Block**.
4. In the **Block Thermal Model** dialog box, enter **200 W** for the **Total Power**.
5. Click **OK**.
6. In the History tree, expand **Model** > **assembly_1** > **Solids** > **Al-Extruded**.
7. Right-click **plate_1** and select **Assign Thermal** > **Block**.
8. In the **Block Thermal Model** dialog box, enter **200 W** for the **Total Power**.
9. Click **OK**.

4 - Assign a Mesh Region and Operation

Assign a Mesh Region to the Non-Model Box

1. In the History tree, expand **Model** > **Assembly_1 Solids** > **Non-Model**.
2. Right-click **assembly_1_mr1** 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 **assembly** for the **Name**.
5. On the **Mesh Region Advanced** tab, select **User Specified**.
6. Define the **Maximum Element Size**:
    - **X**: 0.02 meter
    - **Y**: 0.015 meter
    - **Z**: 0.01 meter
7. Disable **Multi-level meshing**.

Assign a Mesh Operation to the Cold Plate

The mesh needs to be refined for the inner prismatic fluid block (block_2). Assign a mesh operation and use local mesh parameters to refine the mesh.

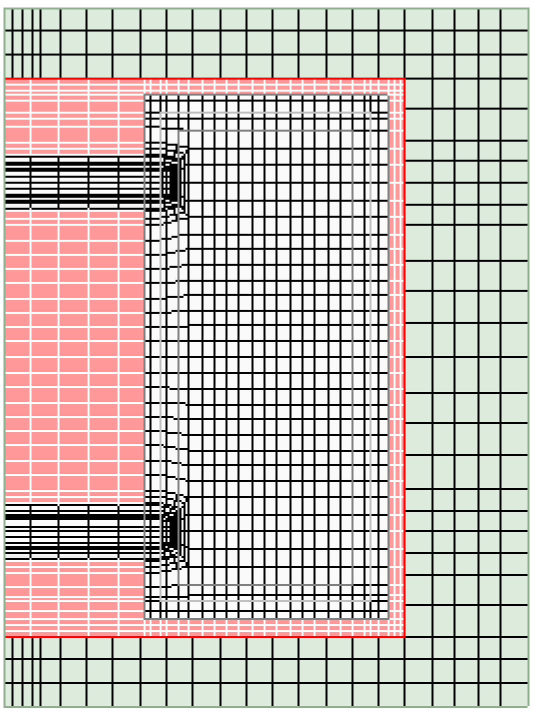
> **Note:** The X, Y, and Z counts are also known as element counts, the number of divisions into which an edge is subdivided. In other words, an element count is the number of elements that lie along the edge.

1. In the History tree, expand **Model** > **assembly_1** > **Solids** > **Water(@280K)**.
2. Right-click **block_2** and select **Assign Mesh Operation**.
3. In the **Mesh Operation** dialog box, select **Local mesh parameters**.
4. On the **Local Mesh Parameter** tab, define the following parameters:
    - **X count**: 30
    - **Y count**: 16
    - **Z count**: 10
5. Click **OK**.

5 - Generate and Display the Mesh

After creating the mesh region and operation, generate and display 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 **Cut plane**.
3. On the **Model** ribbon, select **Top** from the **Orient** drop-down list.



6 - Create Monitor Points

Before running the simulation, create thermal and flow monitors for the outlet opening, a hot plate, and the cold plate.

Create Thermal Monitors

1. In the History tree, expand **Model** > **assembly_1** > **Solids** > **Al-Extruded**.
2. Right-click **plate_2** and select **Assign Monitor** > **Point**.
3. In the **Monitor Setup** dialog box, enter a **Name** of **Hot Plate Temp**.
4. Expand **Thermal** and select **Temperature**.
5. Click **OK**.
6. In the History tree, expand **Model** > **Water(280K)**.
7. Right-click **block_2** and select **Assign Monitor** > **Point**.
8. In the **Monitor Setup** dialog box, enter a **Name** of **Cold Plate Temp**.
9. Expand **Thermal** and select **Temperature**.
10. Click **OK**.

Create a Flow Monitor

1. In the History tree, expand **Model** > **assembly_1** > **Sheets** > **Opening**.
2. Right-click **opening_1** and select **Assign Monitor** > **Point**.
3. In the **Monitor Setup** dialog box, enter **Outlet Speed** for the **Name**.
4. Expand **Flow** and select **Speed**.
5. Click **OK**.

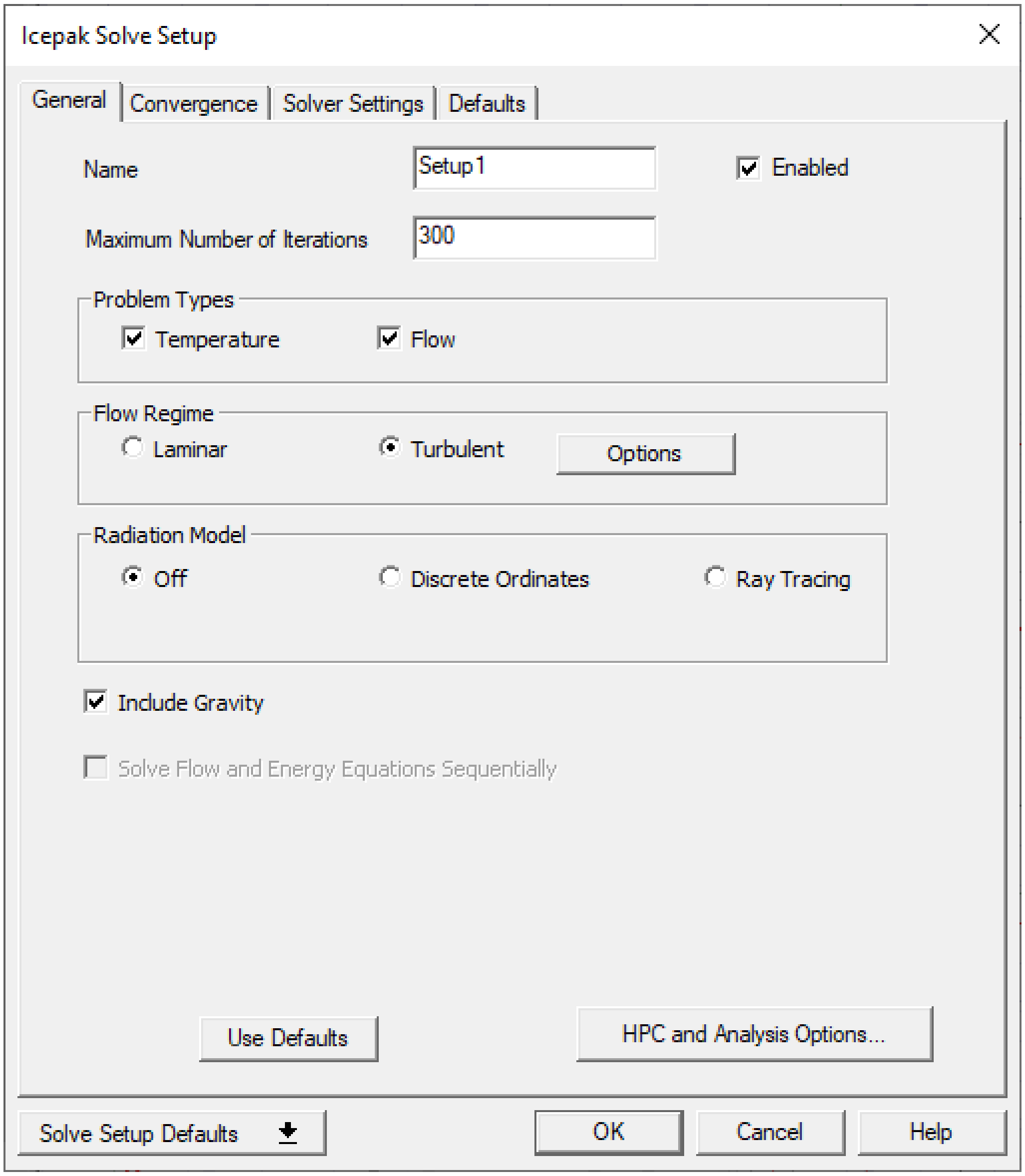
7 - Define the Simulation Settings and Run the Analysis

Define the Design Settings

1. From the **Icepak** menu, select **Design Settings**.
2. On the **Icepak Design Settings** dialog box **Gravity** tab, select **Global::X** 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, select **Two Equation** selection and click **OK**.
5. Under **Radiation Model**, retain the **Off** selection to ignore heat transfer due to radiation.
6. Select **Include Gravity**.



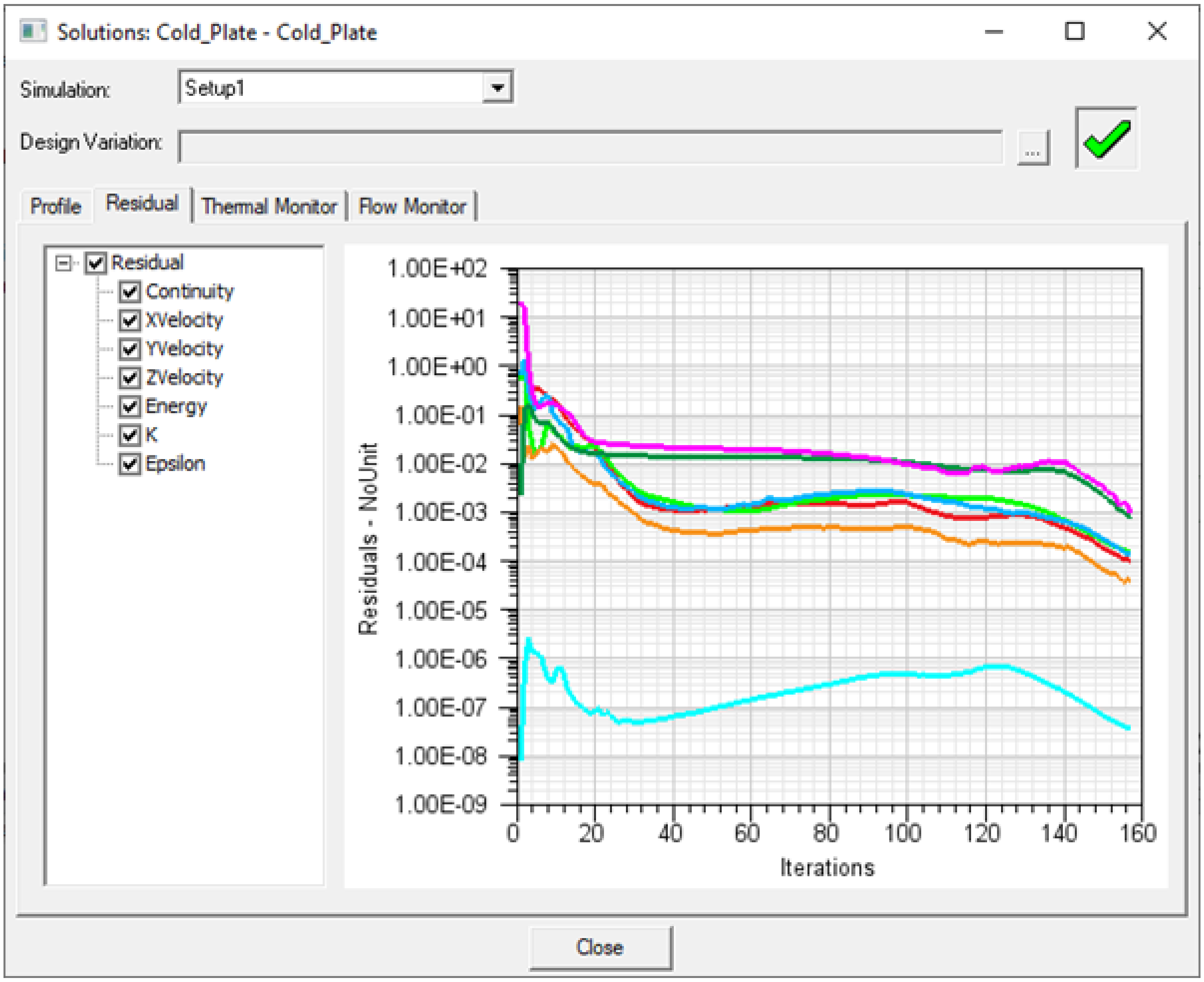
7. Click **OK**
8. From the **File** menu, click **Save**.

Run the Simulation

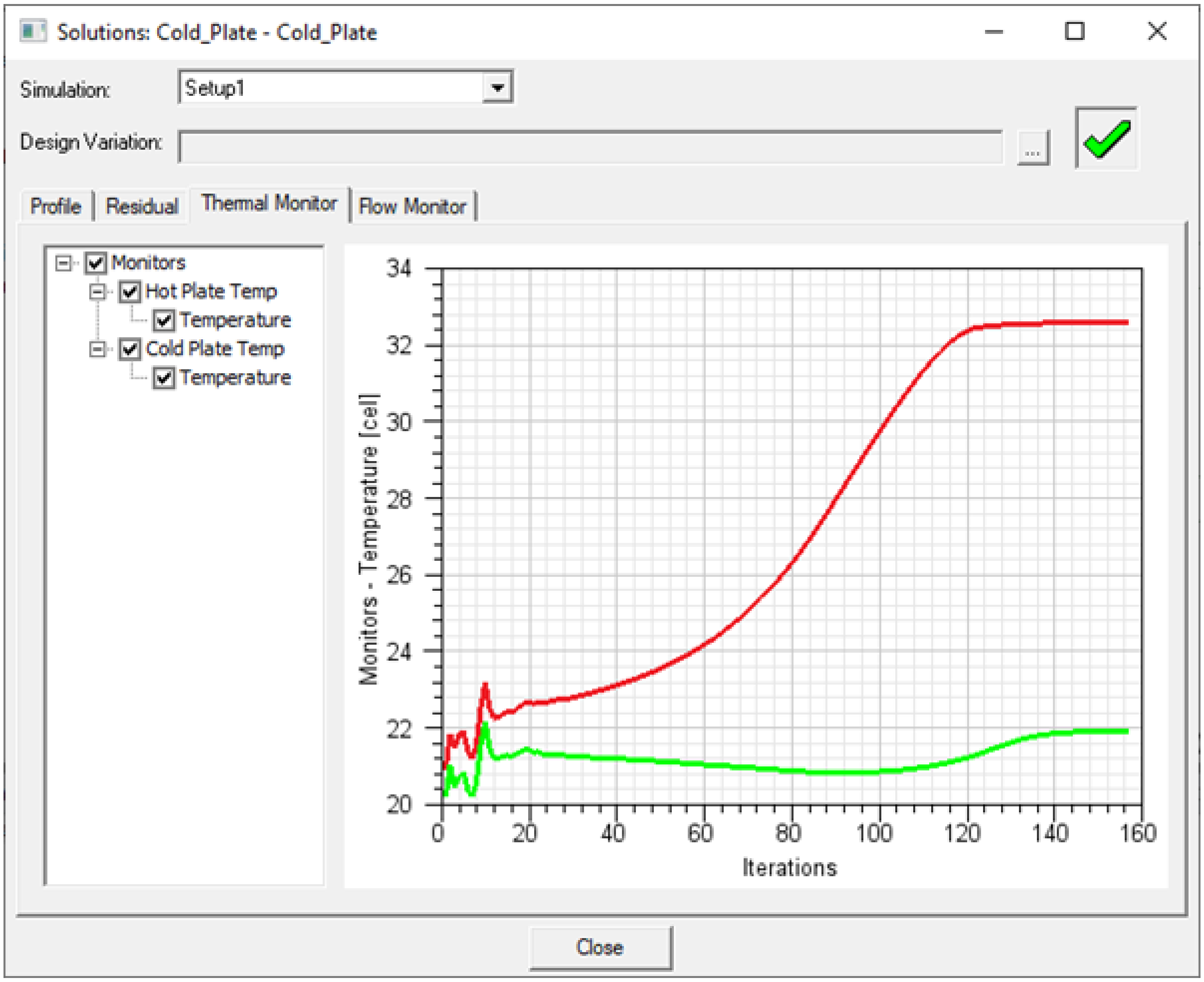
1. In the **Project Manager**, expand **Analysis**.
2. Right-click **Setup1** and select **Analyze**.

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

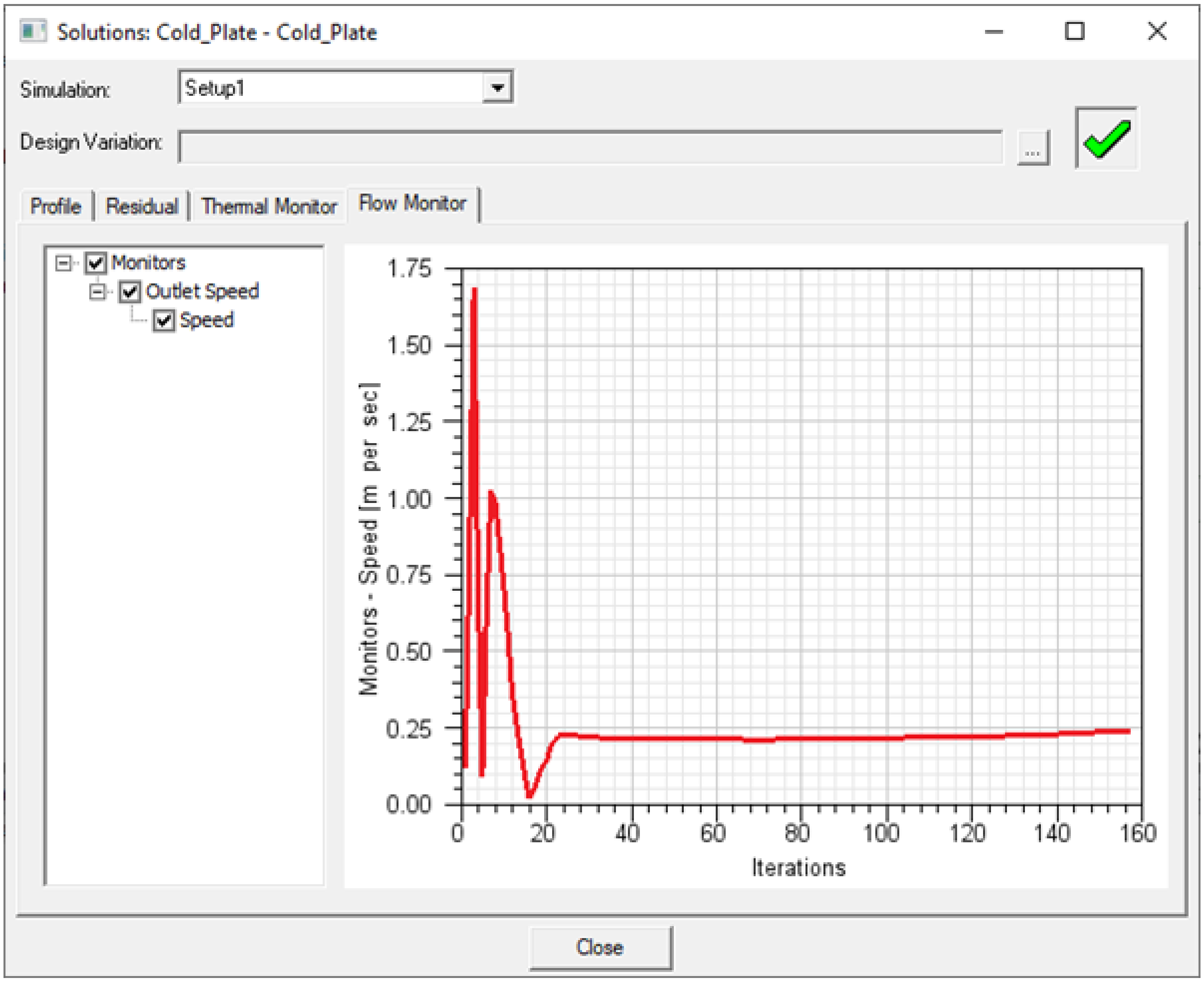
3. Right-click **Setup1** and select **Residuals** to monitor the simulation residuals.



4. Click the **Thermal Monitor** tab to review the temperature monitor on the hot and cold plates.



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



6. Click **Close**

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

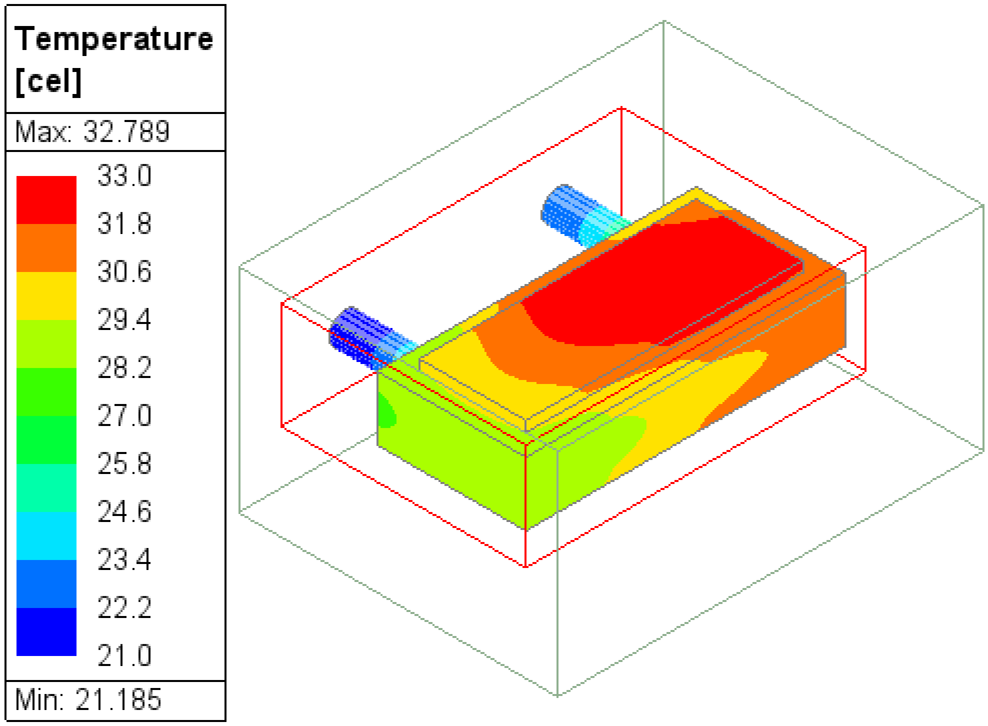
Create Object Field Overlays

Plot Temperature on the Heat Sink

1. In the History tree, expand **Model** > **assembly_1** > **Solids**.
2. Right-click **Al-Extruded** 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 **assembly**.
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

Plot visibility.



Create Plane Cut Field Overlays

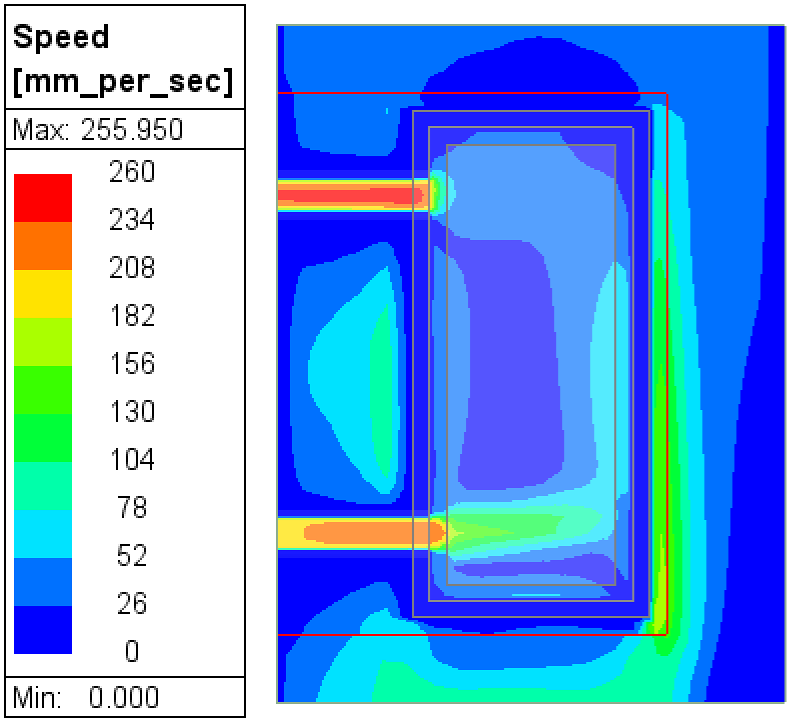
Create a Plane

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**: 0.2 ,0.15 ,0.1 meter
    - **Normal**: 0 ,0 ,1 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**.



9 - Summary

In this tutorial, you modeled a cold-plate that included two heat plates cooled by water circulating inside the cold-plate cavity as well as air driven by natural convection externally. This exercise has also demonstrated how to model multiple fluids in a single model, account for external natural convection and internal forced convection, create a mesh region to reduce the overall mesh count, and specify per-object local meshing parameters.