Getting Started with Icepak: Heat Pipe 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 2024 R2
July 2022

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 includes instructions to model simple heat pipes and an active heat sink using Ansys Icepak.

This chapter contains the following topic:

- "Sample Project - The Heat Pipe" below
- Open the Project

Sample Project - The Heat Pipe

Heat-pipes are used to transport heat from a heat source area, where there is limited space for heat dissipation, to a place where it can be dissipated more easily. The objective of this exercise is not to model the detailed physics inside a heat pipe. Instead, you will model a heat pipe by using a series of cylindrical solid blocks that connect the heat source to an air-cooled heat sink. These blocks will have an anisotropic conductivity with a very large conductivity in the pipe axis direction along which the heat is carried away. The model will be constructed using the default metric unit system. You will also make use of nested non-conformal meshing using mesh regions to reduce the cell count in the model.

This tutorial demonstrates how to model simple heat pipes and an active heat sink using ANSYS Icepak. In this tutorial, you will learn how to:

- Create anisotropic solid materials.
- Assign materials to simulate a simplified heat-pipe in a system.
- Assign nested non-conformal mesh regions.
- Add a solution setup and run the simulation.
- Create temperature and velocity vector field plots.

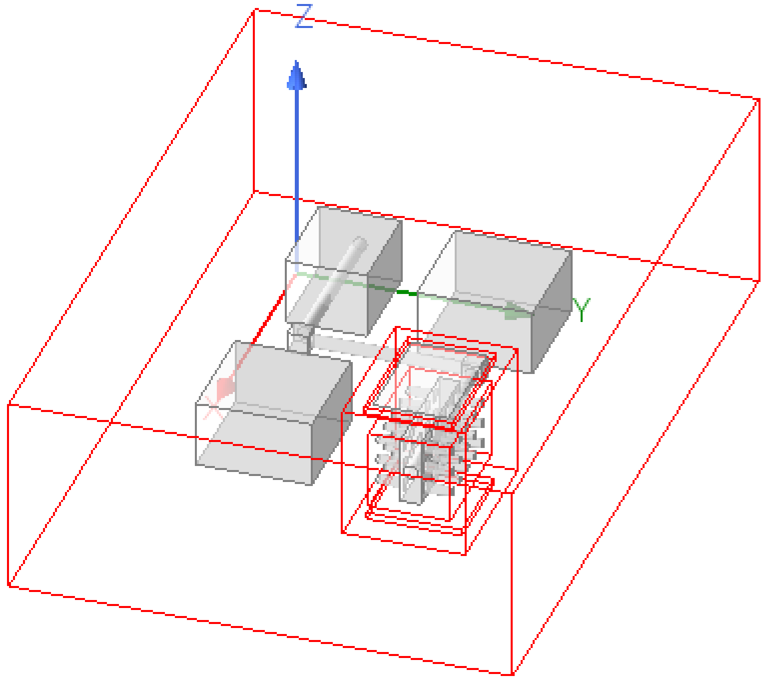


Figure 1-1: Heat Pipe

Open the Project

This chapter contains the following topics:

- Launch the Ansys Electronics Desktop

Launch the Ansys Electronics Desktop

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

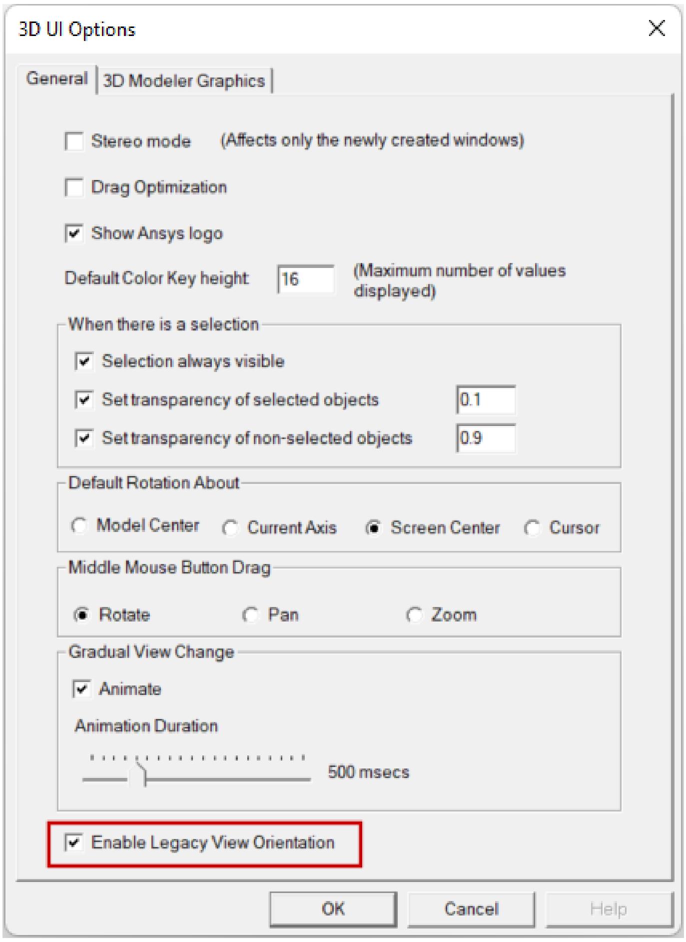
Set 3D UI Options

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.

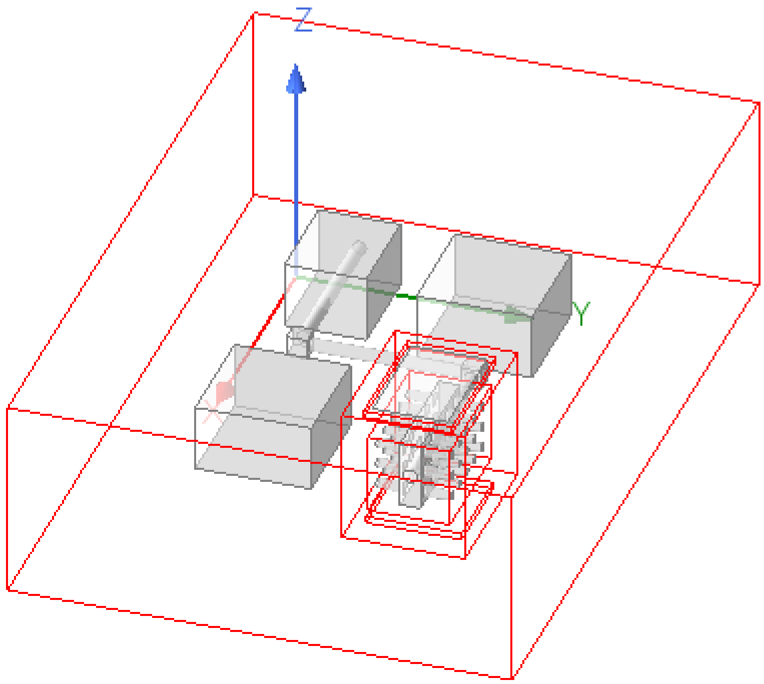
2. Ensure that **Enable Legacy View Orientation** is enabled:



3. Click **OK**.

Open the Project

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



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

2 - Create and Assign Materials

Create anisotropic materials to have very high conductivity in the pipe heat removal directions but normal conductivity in the other directions.

Create Anisotropic Materials

1. From the **Tools** menu, select **Edit Libraries** > **Materials**.
2. In the **Edit Libraries** dialog box, click **Add Material**.
3. In the **View/Edit Material** dialog box, define the following material properties:
    - **Thermal Conductivity**: Anisotropic
    - **T(1,1)**:20000*1.0
    - **T(2,2)**: 20000*0.005
    - **T(3,3)**: 20000*0.005
    - **Mass Density**: 1 kg/m^3
    - **Specific Heat**: 1 J/kg-C
4. Click **OK**.
5. Repeat steps 1 through 4 using the following conductivity tensor values:
    - **T(1,1)**: 20000*0.005
    - **T(2,2)**: 20000*1.0
    - **T(3,3)**: 20000*0.005
6. Repeat steps 1 through 4 using the following conductivity tensor values:
    - **T(1,1)**: 20000*1.0
    - **T(2,2)**: 20000*1.0
    - **T(3,3)**: 20000*0.005

Assign the Materials

1. In the History tree, expand **Model** > **Solids** > **Al-Extruded**.
2. Press and hold the **Ctrtl** key and select **pipe1** and **pipe2**.
3. Right-click and select **Assign Material**.
4. In the **Select Definition** dialog box, select **Material1**.
5. Click **OK**.
6. In the History tree, select **pipe3**.
7. Right-click and select **Assign Material**.
8. In the **Select Definition** dialog box, select **Material2**.
9. Click **OK**.

10. In the History tree, select **Joint1** and **Joint2**.
11. Right-click and select **Assign Material**.
12. In the **Select Definition** dialog box, select **Material3**.
13. Click **OK**.

3 - Assign Boundary Conditions

Assign a Block

1. In the History tree, expand **Model** > **Solids**.
2. Right-click *block_1* and select **Assign Thermal** > **Block**.
3. In the **Block Thermal Model** dialog box, enter a **Total Power** of **25 W**.
4. Click **OK**.

Assign a Grille

1. In the History tree, expand **HS_vent_fan_asy** > **Vent_asy** > **Sheets** > **Unassigned**.
2. Right-click *vent_1* and select **Assign Thermal** > **Grille**.
3. In the **Grille Thermal Model** dialog box, enter a **Free Area Ratio** of **0.8**.
4. Click **OK**.

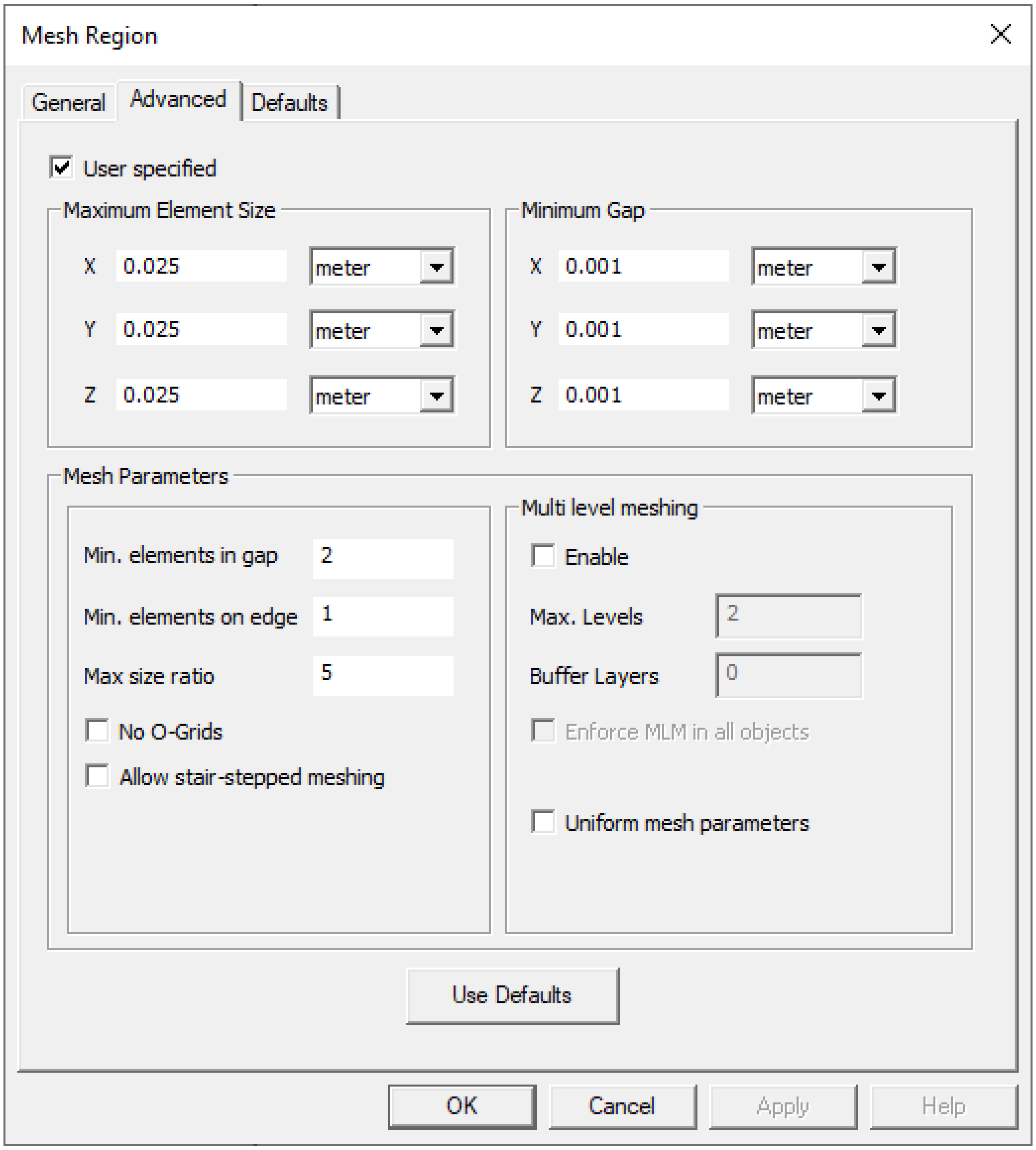
4 - Assign Mesh Regions

In this exercise, our goal is to reduce the overall cell count to a reasonable level while retaining a good cell resolution within the model, especially where the velocity and temperature gradients are higher. This model contains mesh regions assigned to non-model box geometry created for the purpose of creating refined mesh. Assign a mesh region to mesh regions to each box surrounding the fan, heat sink, and vent. Then, assign an outer mesh region to the box encompassing the other mesh regions.

> **Note:** You can also assign mesh regions directly to model geometry.

Assign a Mesh Region to the Fan

1. In the History tree, expand **Model** > **HS_vent_fan_asy** > **Fan_asy** > **Solids** > **Non Model**.
2. Right-click *Fan_asy_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 **General** tab, enter a **Name**.
5. On the **Advanced** tab, enable **User specified** and define the **Maximum Element Size** values:
    - **X**: 0.025 meter
    - **Y**: 0.025 meter
    - **Z**: 0.025 meter
6. Define the **Minimum Gap** values:
    - **X**: 0.001 meter
    - **Y**: 0.001 meter
    - **Z**: 0.001 meter
7. Define the following **Mesh Parameters**:
    - Min. elements in gap: 2
    - Min. elements on edge: 1
    - Max size ratio: 5
8. Under **Multi-level meshing**, deselect **Enable** to disable multi-level meshing.



9. Click **OK**.

Assign a Mesh Region to the Heat Sink

1. In the History tree, expand **Model** > **HS_vent_fan_asy** > **Heatsink_asy** > **Solids** > **Non Model**.
2. Right-click *Heatsink_asy_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 **General** tab, enter a **Name**.
5. On the **Advanced** tab, enable **User specified** and define the **Maximum Element Size** values:

- **X**: 0.025 meter
- **Y**: 0.025 meter
- **Z**: 0.025 meter

6. Define the **Minimum Gap** values:
   - **X**: 0.001 meter
   - **Y**: 0.001 meter
   - **Z**: 0.001 meter
7. Define the following **Mesh Parameters**:
   - Min. elements in gap: 2
   - Min. elements on edge: 1
   - Max size ratio: 5
8. Under **Multi-level meshing**, deselect **Enable** to disable multi-level meshing.
9. Click **OK**.

Assign a Mesh Region to the Vent

1. In the History tree, expand **Model** > **HS_vent_fan_asy** > **Vent_asy** > **Solids** > **Non Model**.
2. Right-click *Vent_asy_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 **General** tab, enter a **Name**.
5. On the **Advanced** tab, enable **User specified** and define the **Maximum Element Size** values:
   - **X**: 0.025 meter
   - **Y**: 0.025 meter
   - **Z**: 0.025 meter
6. Define the **Minimum Gap** values:
   - **X**: 0.001 meter
   - **Y**: 0.001 meter
   - **Z**: 0.001 meter
7. Define the following **Mesh Parameters**:
   - Min. elements in gap: 2
   - Min. elements on edge: 1
   - Max size ratio: 5
8. Under **Multi-level meshing**, deselect **Enable** to disable multi-level meshing.
9. Click **OK**.

Assign an Outer Mesh Region

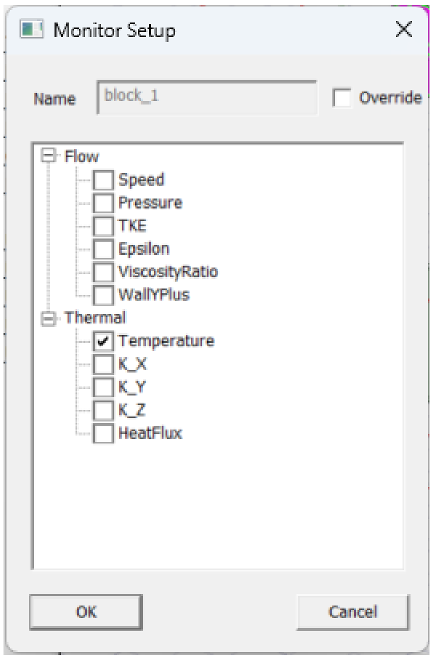
1. In the History tree, expand **Model** > **HS_vent_fan_asy** > **Solids** > **Non Model**.
2. Right-click *HS_vent_fan_asy_mr1* and select **Assign Mesh Region**.
3. In the **SubRegion** dialog box, click **OK** to close the dialog box without adding padding.
4. In the **Mesh Region** dialog box **General** tab, enter a **Name**.
5. On the **Advanced** tab, enable **User specified** and define the **Maximum Element Size** values:
    - **X**: 0.025 meter
    - **Y**: 0.025 meter
    - **Z**: 0.025 meter
6. Define the **Minimum Gap** values:
    - **X**: 0.001 meter
    - **Y**: 0.001 meter
    - **Z**: 0.001 meter
7. Define the following **Mesh Parameters**:
    - Min. elements in gap: 2
    - Min. elements on edge: 1
    - Max size ratio: 5
8. Under **Multi-level meshing**, deselect **Enable** to disable multi-level meshing.
9. Click **OK**.

5 - Assign Monitors

Create a Thermal Monitor

Create a thermal monitor to measure the temperature of block_1.

1. In the History tree, right-click on the *block_1* object and select **Assign Monitor** > **Point**. The **Monitor Setup** dialog box appears.

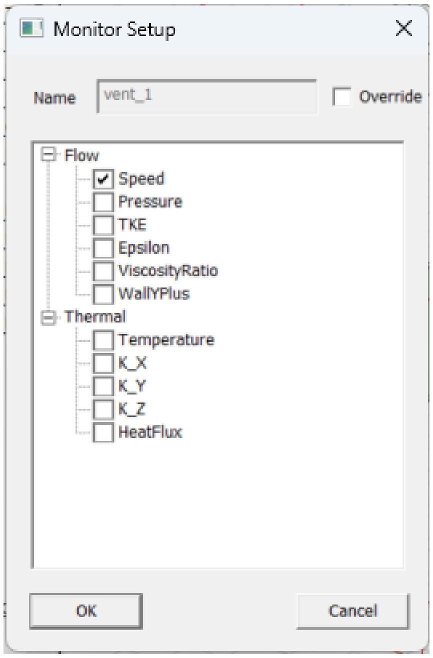


2. In the **Monitor Setup** dialog box, enter a **Name**.
3. Under **Thermal**, select the **Temperature** check box.
4. Click **OK**.

Create a Flow Monitor

Create a flow monitor to measure the speed of the flow through the vent.

1. In the History tree, right-click on the *vent_1*object and select **Assign Monitor** > **Point**.

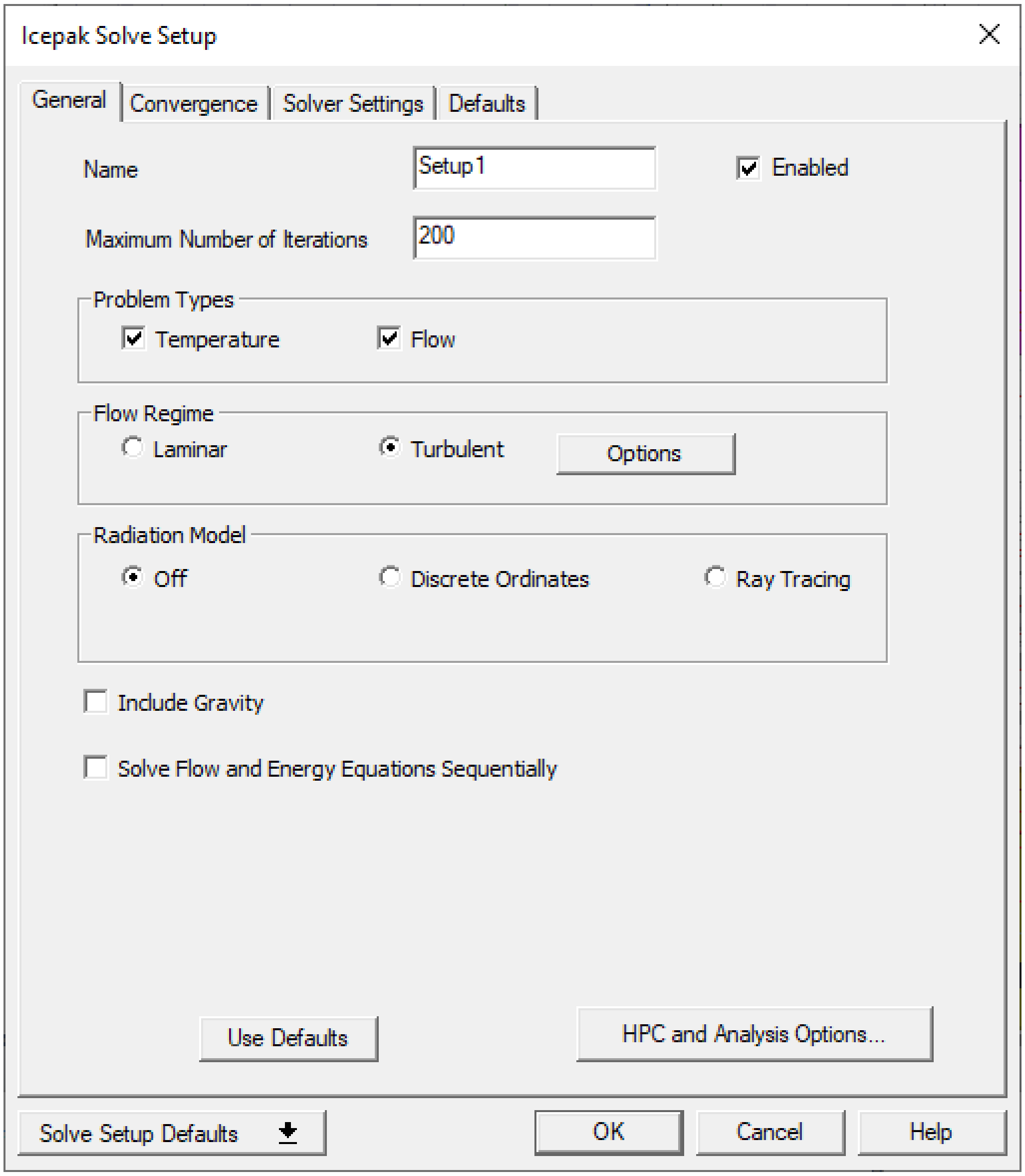


2. In the **Monitor Setup** dialog box, enter a **Name**.
3. Under **Flow**, select the **Speed** check box.
4. Click **OK**.

6 - Add a Solution Setup

Prior to generating a mesh, you must create a solution setup, in which you specify general and solution settings.

1. In the **Project Manager**, right-click on **Analysis** and select **Add Solution Setup**.



2. On the **General** tab, enter a **Maximum Number of Iterations** of 200.
3. In the **Icepak Solve Setup Dialog** under **Flow Regime**, select **Turbulent** and click **Options**.

4. In the **Turbulent Flow Model** dialog box, retain the default selection of **Zero Equation** and click **OK**.
5. On the **Solver Settings** tab, enter -0.1 as the **Z Velocity** initialization and retain the unit **m_per_sec**.
6. Click **OK** to save the settings. The solution setup is added under **Analysis** in the **Project Manager**.
7. From the **File** menu, click **Save**.

7 - Run the Icepak Simulation

1. In the **Project Manager** under **Analysis**, right-click **Setup1** and select **Analyze**.
2. Right-click on **Setup1** again and select **Residual** to open the **Solutions** dialog box, where you can view the solution residuals as they are updated with each iteration.

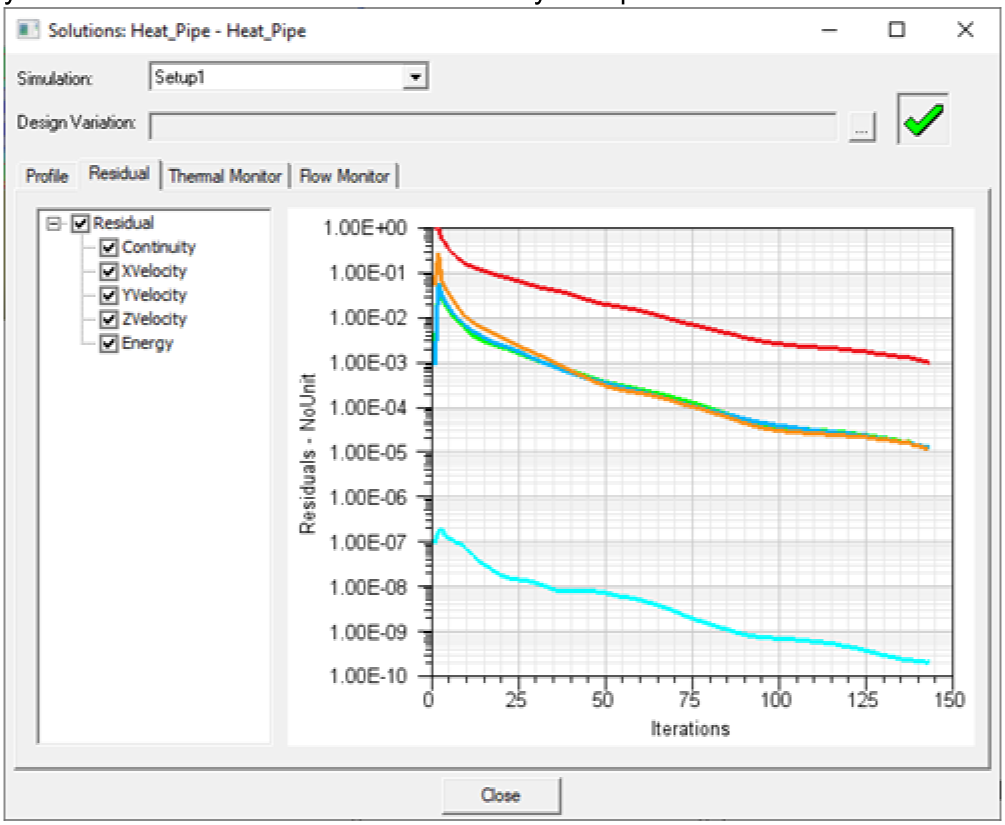


Figure 7-1: Solutions Dialog Box - Residual tab

3. Click the **Thermal Monitor** tab and review the thermal monitor data.

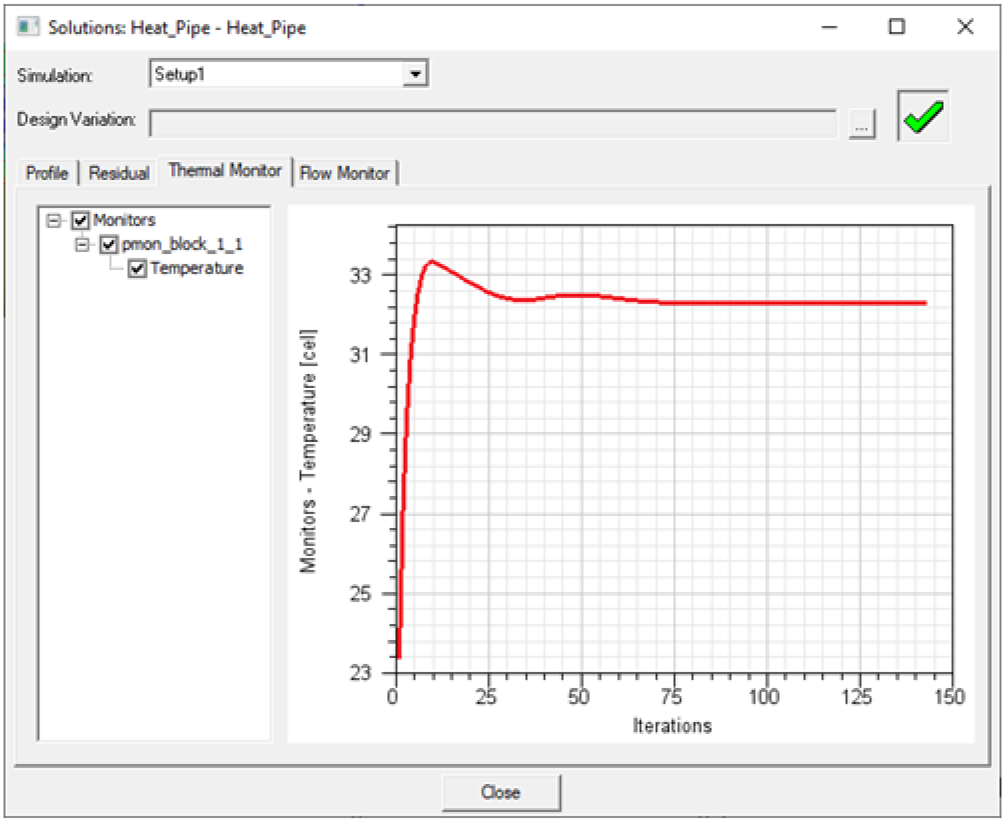


Figure 7-2: Solutions Dialog Box - Thermal Monitor tab

4. Click the **Flow Monitor** tab and review the flow monitor data.

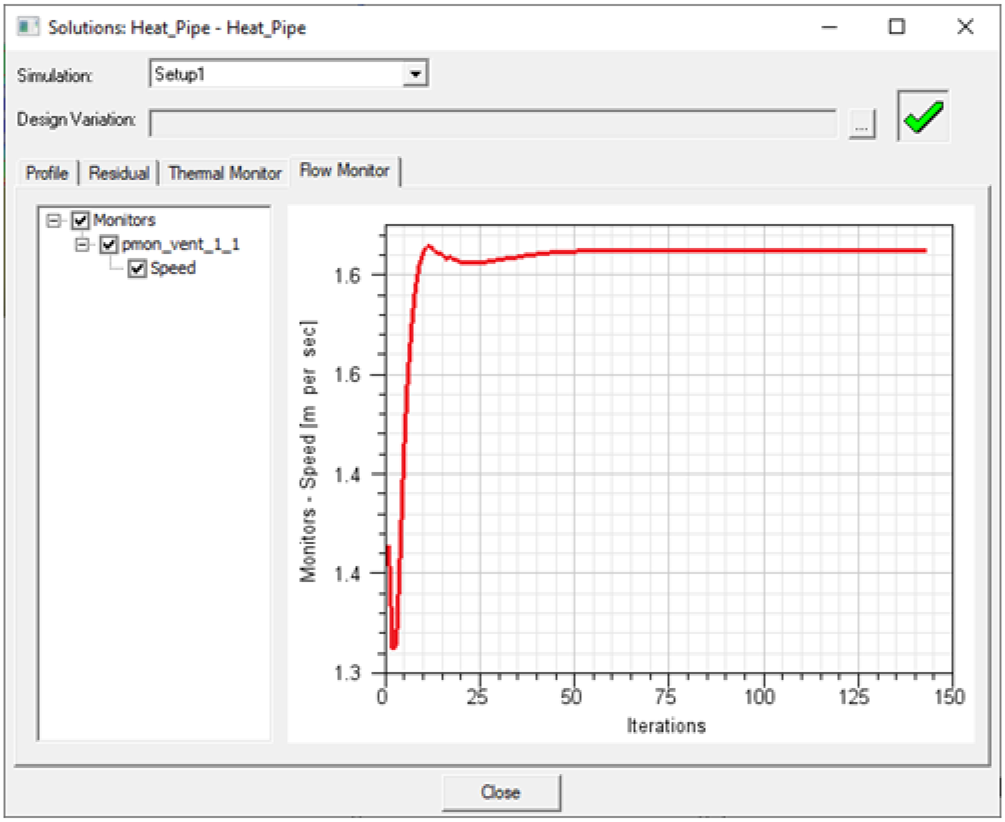


Figure 7-3: Solutions Dialog Box - Flow Monitor tab

5. Click **Close**.

8 - Post-process the Icepak Simulation

Create a Temperature Field Plot

1. In the History tree, right-click *Model* and select **Select All**.
2. Press the **Ctrl** key and click on each of the following pieces of geometry to deselect them:
    - *HS_vent_fan_asy > Fan_asy > fan_1_1 > Sheets > Unassigned > fan_1_Hub*
    - *HS_vent_fan_asy > Fan_asy > fan_1_1 > Sheets > Unassigned > fan_1_Passage*
    - *HS_vent_fan_asy > Fan_asy> Solids > Non Model > Fan_asy_mr1*
    - *HS_vent_fan_asy > Heatsink_asy> Solids > Non Model > Heatsink_asy_mr1*
    - *HS_vent_fan_asy > Vent_asy> Solids > Non Model > Vent_asy_mr1*
    - *HS_vent_fan_asy > Vent_asy> Sheets > Grille > vent_1*
    - *HS_vent_fan_asy > Solids > Non Model > HS_fan_vent_asy_mr1*
    - *Solids > air > Region*
3. Right-click in the **3D Modeler** window and select **Plot Fields** > **Temperature** > **Temperature**.
4. In the **Create Field Plot** dialog box, retain the default selections under **Quantity** and **In Volume**.
5. Select the **Plot on surface only** check box.
6. Click **Done**.

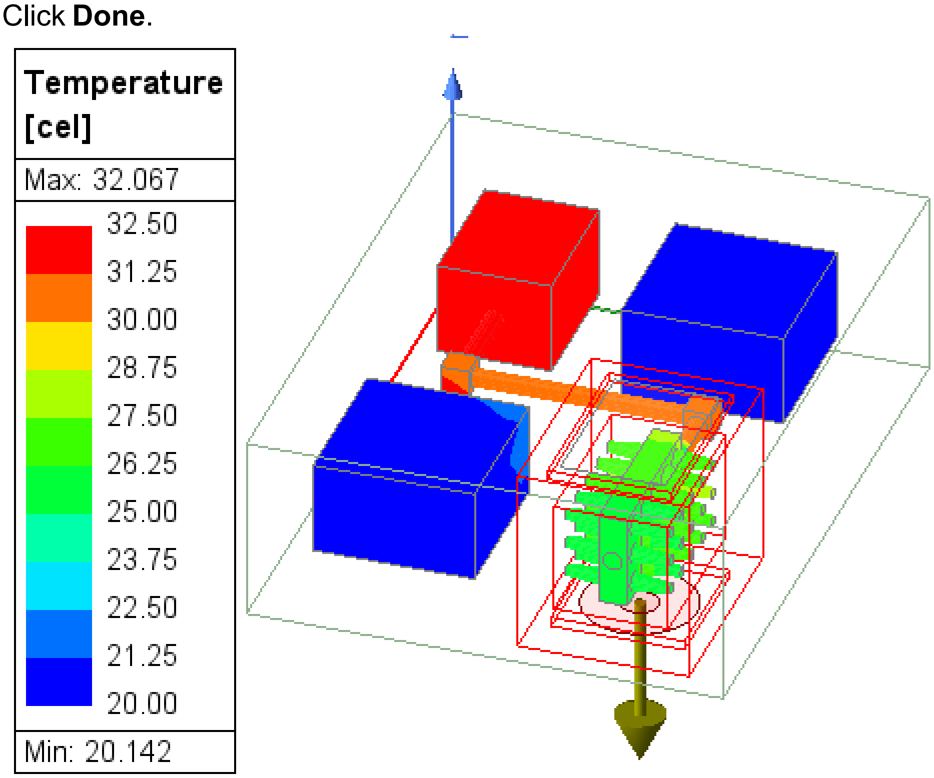


Figure 8-1: Temperature Field Plot

Create a Velocity Vector Plot

1. In the **Project Manager**, expand **Field Overlays** > **Temperature**.
2. Right-click **Temperature1** and select **Plot Visibility** to hide the temperature field plot.
3. In the history tree, expand **Planes** and select the *cut_1_plane*.
4. In the **3D Modeler** window, right-click and select **Plot Fields** > **Velocity** > **Velocity Vectors**.
5. In the **Create Field Plot** dialog box, retain the default selection of **Velocity** under **Quantity** and click **Done**. Velocity vectors are displayed on the plane.

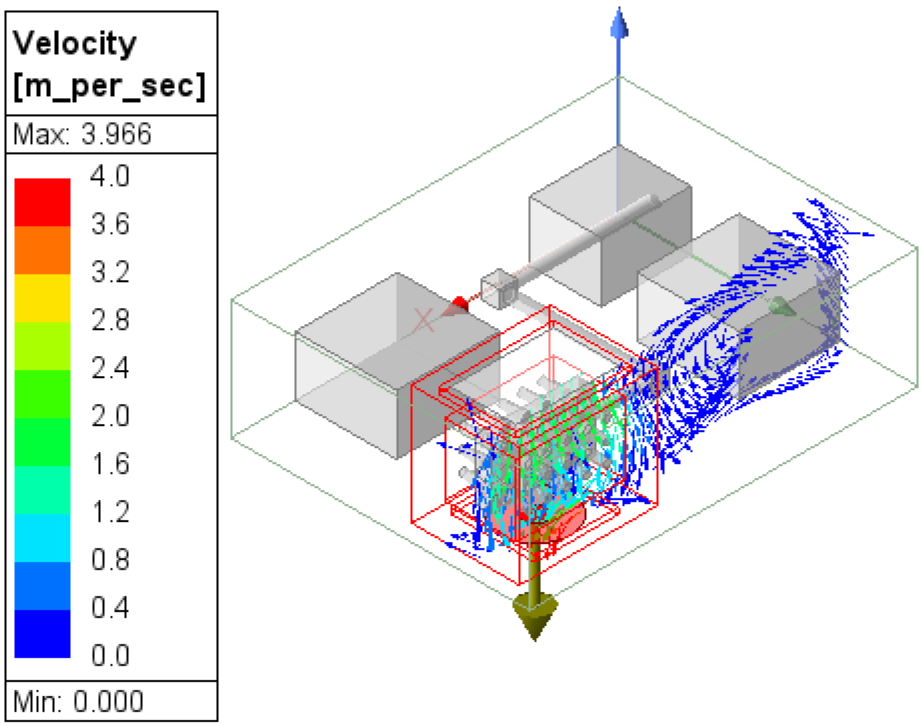


Figure 8-2: Velocity vectors on a plane

9 - Additional Exercise

For more experience with this heat pipe model, refer to Getting Started with Icepak - Transient Simulation. This guides you through the following tasks:

- Selecting the Transient solution type
- Reviewing a dataset
- Assigning a piecewise linear transient power to a block boundary condition
- Reviewing transient solve setup options
- Monitoring a transient simulation
- Post-processing transient simulation results

- Selecting the Transient solution type
- Reviewing a dataset
- Assigning a piecewise linear transient power to a block boundary condition
- Reviewing transient solve setup options
- Monitoring a transient simulation
- Post-processing transient simulation results