



Model created in COMSOL Multiphysics 6.4

Convective Cooling of a Pot-Core Inductor

Introduction

The inductor is a common component in a variety of different electrical devices. Its usage ranges from power transformation to measurement systems. In small devices with many components, such as in laptop computers, heat generation can be a problem and has to be accounted for in the design. This application describes the heat transfer in a pot-core inductor that is cooled by convective cooling.

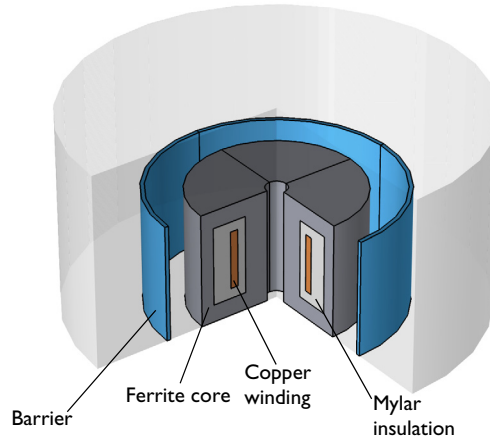


Figure 1: 3D view of the model geometry.

A varying current in the copper induces a magnetic field that is strengthened by the ferrite core. Heat is generated in the core and the winding due to resistive heating.

This tutorial does not include the resistive heating due to induced currents, but instead assumes that a specific amount of heat is generated uniformly in the core and in the copper.

The component is cooled by air that enters from the top of the geometry and exits through the center and the lower part of the outer boundary.

Results

Figure 2 shows the velocity distribution together with an arrow plot of the same field. The arrow plot reveals that the airflow between the barrier and the ferrite core is very close to zero. Note also the recirculation zone at the bottom right.

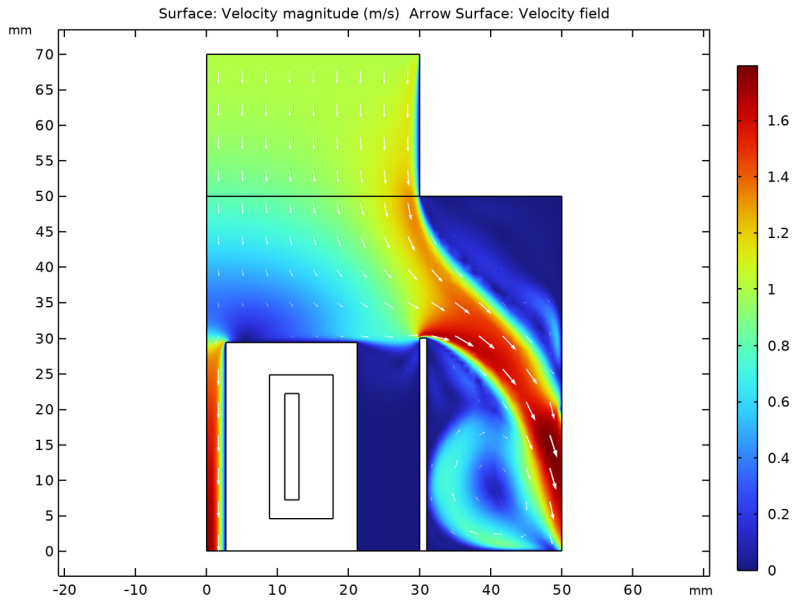


Figure 2: Magnitude and arrow plot of the velocity field.

The temperature distribution is shown in Figure 3. The temperature reaches a maximum in the copper winding where most of the heat is generated. It is clear that the airflow has a cooling effect on the temperature although this effect is not optimal.

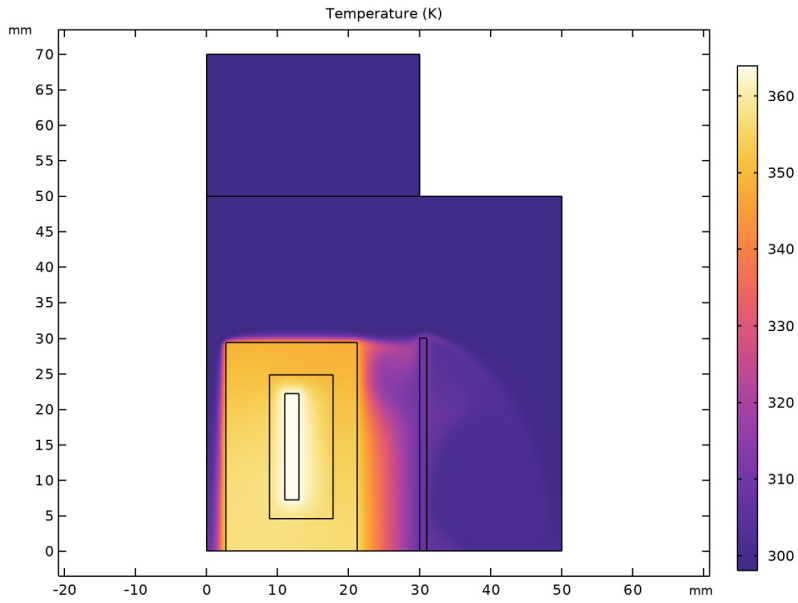


Figure 3: Temperature distribution.

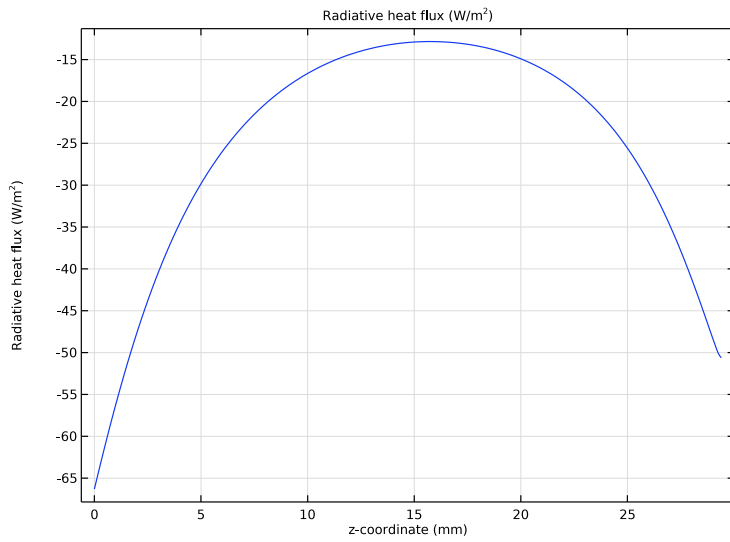


Figure 4: Cross-sectional plot of the net radiative flux.

In the overall heat balance, radiation is responsible for about 10% of the total heat loss at steady state. The plot in [Figure 4](#) shows a cross-sectional plot of the net radiative flux along the inner, vertical, boundary of the central hole (see [Figure 1](#)). Note that away from the open ends, the emitted and reflected radiation is almost balanced by the incident energy, so even if the temperature and radiation levels are high, the net flux is small in this region. The main part of the radiative losses instead originates from the outside of the inductor.

Notes About the COMSOL Implementation


To set up the model, use the Conjugate Heat Transfer predefined multiphysics coupling of the Heat Transfer Module. To provide cooling for the component, air enters the domain at the top of the geometry at the speed of 1 m/s. To include the airflow, the model uses the Weakly Compressible Navier–Stokes equation. The viscosity and density of air and hence the airflow depend on the temperature; on the other hand, the temperature distribution depends on the flow around the component. This means that this multiphysics model has to be solved simultaneously. In this axisymmetric model, some of the surfaces are exposed to heat radiation from other surfaces, which means that surface-to-surface radiation must be accounted for.

Application Library path: Heat_Transfer_Module/
Power_Electronics_and_Electronic_Cooling/potcore_inductor


Modeling Instructions



From the **File** menu, choose **New**.

NEW

In the **New** window, click  **Model Wizard**.

MODEL WIZARD

- 1 In the **Model Wizard** window, click  **2D Axisymmetric**.
- 2 In the **Select Physics** tree, select **Heat Transfer > Conjugate Heat Transfer > Laminar Flow**.
- 3 Click **Add**.
- 4 In the **Select Physics** tree, select **Heat Transfer > Radiation > Surface-to-Surface Radiation (rad)**.
- 5 Click **Add**.

- 6 Click  **Study**.
- 7 In the **Select Study** tree, select **General Studies > Stationary**.
- 8 Click  **Done**.

GLOBAL DEFINITIONS

Parameters 1



- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
Q_core	$7.64e4 [W/m^3]$	76400 W/m ³	Heat source in the core
Q_copper	$8.657e5 [W/m^3]$	8.657E5 W/m ³	Heat source in the copper



GEOMETRY 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
- 2 In the **Settings** window for **Geometry**, locate the **Units** section.
- 3 From the **Length unit** list, choose **mm**.



Rectangle 1 (r1)

- 1 In the **Geometry** toolbar, click  **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 50.
- 4 In the **Height** text field, type 50.
- 5 Click  **Build Selected**.



Rectangle 2 (r2)

- 1 In the **Geometry** toolbar, click  **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 30.
- 4 In the **Height** text field, type 20.
- 5 Locate the **Position** section. In the **z** text field, type 50.
- 6 Click  **Build Selected**.



Rectangle 3 (r3)

- 1 In the **Geometry** toolbar, click  **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 18.5.
- 4 In the **Height** text field, type 29.4.
- 5 Locate the **Position** section. In the **r** text field, type 2.7.
- 6 Click  **Build Selected**.



Rectangle 4 (r4)

- 1 In the **Geometry** toolbar, click  **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 8.95.
- 4 In the **Height** text field, type 20.3.
- 5 Locate the **Position** section. In the **r** text field, type 8.85.
- 6 In the **z** text field, type 4.55.
- 7 Click  **Build Selected**.

Rectangle 5 (r5)



- 1 In the **Geometry** toolbar, click  **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 2.
- 4 In the **Height** text field, type 15.
- 5 Locate the **Position** section. In the **r** text field, type 11.
- 6 In the **z** text field, type 7.2.
- 7 Click  **Build Selected**.

Rectangle 6 (r6)

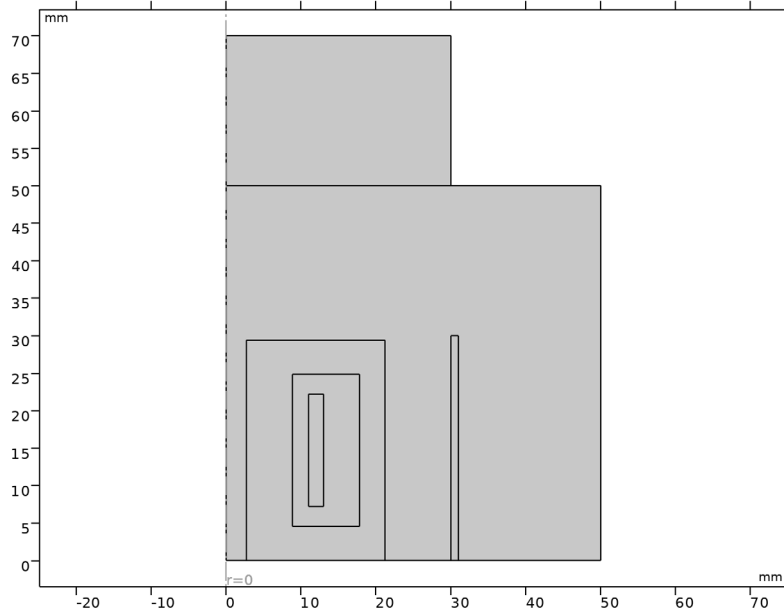
- 1 In the **Geometry** toolbar, click  **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Height** text field, type 30.
- 4 Locate the **Position** section. In the **r** text field, type 30.
- 5 Click  **Build Selected**.

Point 1 (pt1)

- 1 In the **Geometry** toolbar, click  **Point**.


- 2 In the **Settings** window for **Point**, locate the **Point** section.
- 3 In the **r** text field, type 50.
- 4 In the **z** text field, type 20.
- 5 In the **Geometry** toolbar, click  **Build All**.
- 6 Click the  **Zoom Extents** button in the **Graphics** toolbar.

This completes the geometry modeling stage. The geometry should now look like that in the figure below.




DEFINITIONS

Ambient Properties 1 (ampr1)

- 1 In the **Physics** toolbar, click  **Shared Properties** and choose **Ambient Properties**.
- 2 In the **Settings** window for **Ambient Properties**, locate the **Ambient Conditions** section.
- 3 In the T_{amb} text field, type 25 [degC].

LAMINAR FLOW (SPF)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.
- 2 Select Domains 1 and 2 only.
- 3 In the **Settings** window for **Laminar Flow**, locate the **Domain Selection** section.

- 4 Click  **Create Selection**.
- 5 In the **Create Selection** dialog, type Air in the **Selection name** text field.
- 6 Click **OK**.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)


Fluid 1

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Heat Transfer in Solids and Fluids (ht)** click **Fluid 1**.
- 2 In the **Settings** window for **Fluid**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Air**.

MATERIALS

Now proceed to setting up the material properties.

ADD MATERIAL


- 1 In the **Materials** toolbar, click  **Add Material** to open the **Add Material** window.
- 2 Go to the **Add Material** window.
- 3 In the tree, select **Built-in > Air**.
- 4 Click the **Add to Component** button in the window toolbar.

MATERIALS

Air (mat1)

By default, the first material you add apply on all domains. Keep this setting and add other materials that override the material properties for selected domains.


Ferrite

- 1 In the **Materials** toolbar, click  **Blank Material**.
- 2 In the **Settings** window for **Material**, type Ferrite in the **Label** text field.
- 3 Select Domain 3 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_{iso} ; $k_{ij} = k_{iso}$, $k_{ij} = 0$	5 [W/ (m* K)]	W/(m·K)	Basic
Density	ρ	4800 [kg/ m ³]	kg/m ³	Basic
Heat capacity at constant pressure	C_p	750 [J/ (kg*K)]	J/(kg·K)	Basic

Mylar

1 In the **Materials** toolbar, click  **Blank Material**.


2 In the **Settings** window for **Material**, type Mylar in the **Label** text field.

3 Select Domain 4 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_{iso} ; $k_{ij} = k_{iso}$, $k_{ij} = 0$	0.2 [W/ (m* K)]	W/(m·K)	Basic
Density	ρ	1393 [kg/ m ³]	kg/m ³	Basic
Heat capacity at constant pressure	C_p	1000 [J/ (kg*K)]	J/(kg·K)	Basic

Quartz

1 In the **Materials** toolbar, click  **Blank Material**.

2 In the **Settings** window for **Material**, type Quartz in the **Label** text field.


3 Select Domain 6 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_{iso} ; $k_{ij} = k_{iso}$, $k_{ij} = 0$	6.1 [W/ (m* K)]	W/(m·K)	Basic

Property	Variable	Value	Unit	Property group
Density	rho	2648 [kg / m ³]	kg/m ³	Basic
Heat capacity at constant pressure	Cp	759 [J / (kg * K)]	J/(kg·K)	Basic

ADD MATERIAL


- 1 Go to the **Add Material** window.
- 2 In the tree, select **Built-in > Copper**.
- 3 Click the **Add to Component** button in the window toolbar.
- 4 In the **Materials** toolbar, click  **Add Material** to close the **Add Material** window.

MATERIALS

Copper (mat5)


Select Domain 5 only.

Ferrite (Boundary)

- 1 In the **Materials** toolbar, click  **Blank Material**.
- 2 In the **Settings** window for **Material**, type Ferrite (Boundary) in the **Label** text field.
- 3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 6, 8, and 17 only.
- 5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Surface emissivity	epsilon_rad	0.2	1	Basic

Quartz (Boundary)


- 1 In the **Materials** toolbar, click  **Blank Material**.
- 2 In the **Settings** window for **Material**, type Quartz (Boundary) in the **Label** text field.
- 3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 19, 21, and 24 only.

5 Locate the **Material Contents** section. In the table, enter the following settings:


Property	Variable	Value	Unit	Property group
Surface emissivity	epsilon_rad	0.8	1	Basic

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)


Temperature 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Temperature**.
- 2 Select Boundary 22 only.
- 3 In the **Settings** window for **Temperature**, locate the **Temperature** section.
- 4 From the T_0 list, choose **Ambient temperature (amp1)**.


Inflow 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Inflow**.
- 2 Select Boundary 5 only.
- 3 In the **Settings** window for **Inflow**, locate the **Upstream Properties** section.
- 4 From the T_{ustr} list, choose **Ambient temperature (amp1)**.


Outflow 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Outflow**.
- 2 Select Boundaries 2 and 26 only.

Heat Source 1


- 1 In the **Physics** toolbar, click  **Domains** and choose **Heat Source**.
- 2 Select Domain 5 only.
- 3 In the **Settings** window for **Heat Source**, locate the **Heat Source** section.
- 4 In the Q_0 text field, type Q_copper.

Heat Source 2


- 1 In the **Physics** toolbar, click  **Domains** and choose **Heat Source**.
- 2 Select Domain 3 only.
- 3 In the **Settings** window for **Heat Source**, locate the **Heat Source** section.
- 4 In the Q_0 text field, type Q_core.

LAMINAR FLOW (SPF)

Inlet 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Inlet**.
- 2 Select Boundary 5 only.
- 3 In the **Settings** window for **Inlet**, locate the **Velocity** section.
- 4 In the U_0 text field, type 1 [m/s].

Outlet 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Outlet**.
- 2 Select Boundaries 2 and 26 only.



SURFACE-TO-SURFACE RADIATION (RAD)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Surface-to-Surface Radiation (rad)**.
- 2 Select Boundaries 6, 8, 17, 19, 21, and 24 only.

Diffuse Surface 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)** > **Surface-to-Surface Radiation (rad)** click **Diffuse Surface 1**.
- 2 In the **Settings** window for **Diffuse Surface**, locate the **Ambient** section.
- 3 From the T_{amb} list, choose **Ambient temperature (amp1)**.

ADD MULTIPHYSICS


- 1 In the **Physics** toolbar, click  **Add Multiphysics** to open the **Add Multiphysics** window.
- 2 Go to the **Add Multiphysics** window.
- 3 In the tree, select **No Predefined Multiphysics Available for the Selected Physics Interfaces**.
- 4 Find the **Select the physics interfaces you want to couple** subsection. In the table, clear the **Couple** checkbox for **Laminar Flow (spf)**.
- 5 In the tree, select **Heat Transfer > Radiation > Heat Transfer with Surface-to-Surface Radiation**.
- 6 Click the **Add to Component** button in the window toolbar.
- 7 In the **Physics** toolbar, click  **Add Multiphysics** to close the **Add Multiphysics** window.

When the **Heat Transfer with Surface-to-Surface Radiation** is added the default domain opacity is set to **From heat transfer interface** which means that the solid domains are opaque by default while the fluid domains are transparent by default. You can override


these default settings by adding one or multiple **Opacity** node(s) under the **Surface-to-Surface Radiation** interface.

MESH 1

Use a finer physics-controlled mesh to improve the fluid flow resolution.

- 1 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.
- 2 From the **Element size** list, choose **Finer**.
- 3 Click  **Build All**.

STUDY 1

In the **Study** toolbar, click  **Compute**.

RESULTS

Temperature (ht)



The first default plot shows the temperature distribution in the simulation domain (Figure 3).

Velocity (spf)

This default plot displays the velocity magnitude in a 2D slice of the axisymmetric geometry. Reproduce the plot in Figure 2 with the following steps.


- 1 In the **Model Builder** window, click **Velocity (spf)**.

Arrow Surface 1



- 1 In the **Velocity (spf)** toolbar, click  **Arrow Surface**.
- 2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1) > Laminar Flow > Velocity and pressure > u,w - Velocity field**.
- 3 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.
- 4 In the **Velocity (spf)** toolbar, click  **Plot**.

Create a cross-sectional plot of the net radiative flux as in Figure 4 with the following steps:

Radiative Heat Flux

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Radiative Heat Flux in the **Label** text field.

Line Graph 1


- 1 In the **Radiative Heat Flux** toolbar, click  **Line Graph**.
- 2 Select Boundary 6 only.
- 3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1) > Surface-to-Surface Radiation > Radiative heat flux > rad.rflux - Radiative heat flux - W/m²**.
- 4 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1) > Geometry > Coordinate > z - z-coordinate**.
- 5 In the **Radiative Heat Flux** toolbar, click  **Plot**.

Finally, display the temperature and velocity together in 3D as in the model thumbnail picture by using two partial revolutions of the 2D axisymmetric dataset.



Revolution 2D

- 1 In the **Model Builder** window, expand the **Results > Datasets** node, then click **Revolution 2D**.
- 2 In the **Settings** window for **Revolution 2D**, click to expand the **Revolution Layers** section.
- 3 In the **Start angle** text field, type 30.
- 4 In the **Revolution angle** text field, type 120.

Revolution 2D 2

- 1 In the **Results** toolbar, click  **More Datasets** and choose **Revolution 2D**.
- 2 In the **Settings** window for **Revolution 2D**, locate the **Revolution Layers** section.
- 3 In the **Start angle** text field, type -75.
- 4 In the **Revolution angle** text field, type 105.

RESULT TEMPLATES

- 1 In the **Results** toolbar, click  **Result Templates** to open the **Result Templates** window.
- 2 Go to the **Result Templates** window.
- 3 In the tree, select **Study 1/Solution 1 (sol1) > Heat Transfer in Solids and Fluids > Temperature (ht)**.
- 4 Click the **Add Result Template** button in the window toolbar.
- 5 In the **Results** toolbar, click  **Result Templates** to close the **Result Templates** window.

RESULTS


Volume 2

- 1 In the **Model Builder** window, expand the **Temperature (ht) 1** node.
- 2 Right-click **Results > Temperature (ht) 1 > Volume 1** and choose **Duplicate**.

Temperature and velocity 3D

- 1 In the **Settings** window for **3D Plot Group**, type **Temperature and velocity 3D** in the **Label** text field.
- 2 Locate the **Color Legend** section. Select the **Show units** checkbox.

Volume 1

- 1 In the **Model Builder** window, click **Volume 1**.
- 2 In the **Settings** window for **Volume**, locate the **Expression** section.
- 3 In the **Expression** text field, type **spf.U**.
- 4 Locate the **Data** section. From the **Dataset** list, choose **Revolution 2D 2**.
- 5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Rainbow**.
- 6 In the **Temperature and velocity 3D** toolbar, click  **Plot**.