Created in COMSOL Multiphysics 6.1

Convective Cooling of a Potcore Inductor

This model is licensed under the COMSOL Software License Agreement 6.1.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

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 potcore inductor that is cooled by convective cooling.

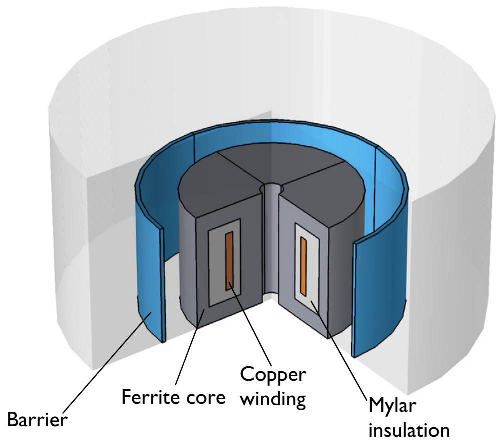


Figure 1: 3D view of the model geometry.

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.

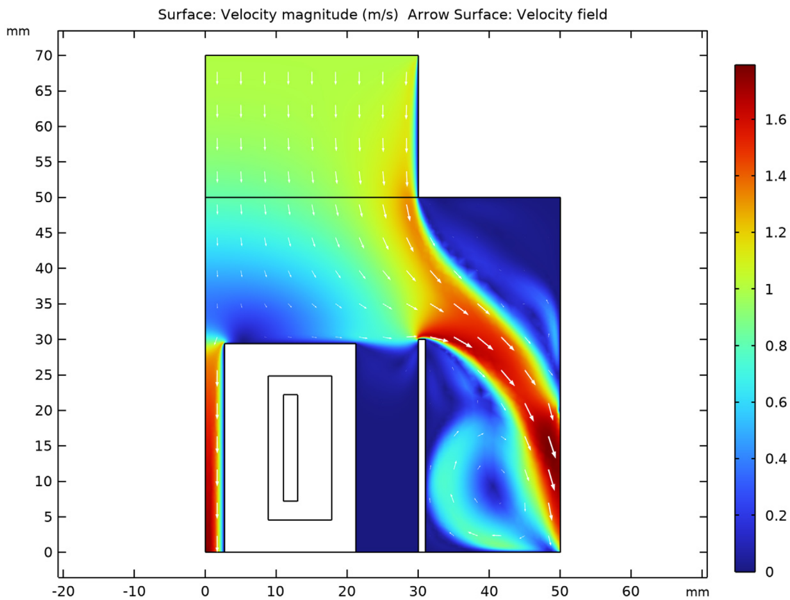


Figure 2: Magnitude and arrow plot of the velocity field.

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.

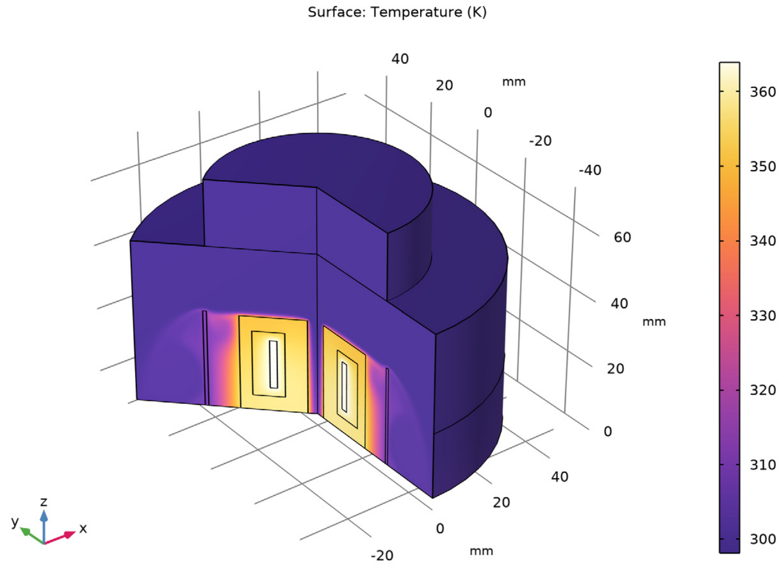


Figure 3: Temperature distribution.

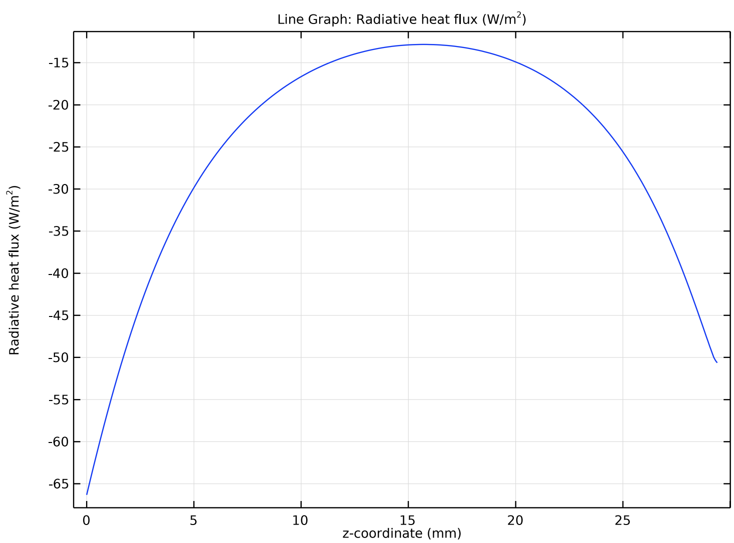


Figure 4: Cross-sectional plot of the net radiative flux.

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 application, 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/m³	Heat source in the core
Q_copper	8.657e5[W/m^3]	8.657E5 W/m³	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.

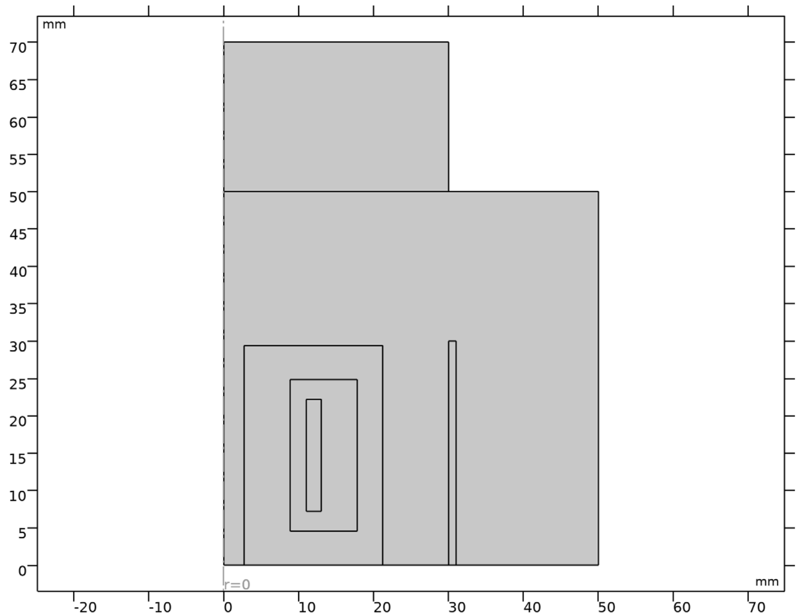
3 In the **r** text field, type 50.

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 box, 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 **Home** 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 **Add to Component** 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 ; kii = k_iso, kij = 0	5	W/(m·K)	Basic
Density	rho	4800	kg/m³	Basic
Heat capacity at constant pressure	Cp	750	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 ; kii = k_iso, kij = 0	0.2	W/(m·K)	Basic
Density	rho	1393	kg/m³	Basic
Heat capacity at constant pressure	Cp	1000	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 ; kii = k_iso, kij = 0	6.1	W/(m·K)	Basic
Density	rho	2648	kg/m³	Basic
Heat capacity at constant pressure	Cp	759	J/(kg·K)	Basic

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-in>Copper**.

3 Click **Add to Component** 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 (ampr1)**.

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 (ampr1)**.

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)

In the **Model Builder** window, under **Component 1 (comp1)** click **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`.

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 (ampr1)**.

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** check box for **Laminar Flow (spf)**.

5 In the tree, select **Heat Transfer>Radiation>Heat Transfer with Surface-to-Surface Radiation**.

6 Click **Add to Component** 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 **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Finer**.

4 Click **Build All**.

STUDY 1

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

RESULTS

Temperature, 3D (ht)

The default 3D plot of the temperature distribution is obtained by first generating a revolution of the 2D axisymmetric dataset (Figure 3).

Isothermal Contours (ht)

This default plot shows the isothermal contours in the inductor.

1 In the **Model Builder** window, click **Isothermal Contours (ht)**.

Arrow Surface 1

1 In the **Isothermal Contours (ht)** toolbar, click **Arrow Surface**.
2 In the **Settings** window for **Arrow Surface**, locate the **Coloring and Style** section.
3 From the **Color** list, choose **Gray**.
4 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**.
5 In the **Isothermal Contours (ht)** toolbar, click **Plot**.

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, under **Results** 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 **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D 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`.

Temperature and velocity 3D

1 In the **Model Builder** window, right-click **Velocity, 3D (spf)** and choose **Duplicate**.

2 In the **Model Builder** window, click **Velocity, 3D (spf) 1**.

3 In the **Settings** window for **3D Plot Group**, type `Temperature and velocity 3D` in the **Label** text field.

4 Locate the **Color Legend** section. Select the **Show units** check box.

Surface 2

1 Right-click **Temperature and velocity 3D** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Thermal>HeatCameraLight** in the tree.

5 Click **OK**.

Surface

1 In the **Model Builder** window, click **Surface**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Revolution 2D 2**.

4 In the **Temperature and velocity 3D** toolbar, click **Plot**.