

Temperature Field in a Cooling Flange

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

Chemical reaction fluids can be cooled using glass flanges. The reaction fluid is passed through the flange and the air surrounding the flange then serves as the coolant. Engineers looking to optimize the cooling performance of such flanges can look to simulation for help.



Figure 1: Operating principle of the cooling flange.

First off, the physics to analyze in this case is heat transfer, involving both convection and conduction. Heat transfer occurs via convection with both interior and exterior surfaces of the flange, and via conduction through the glass. The tube is heated from the inside by the process fluid, conduction then diffuses the heat in the flange, which in turn heats the air, causing a convective flow due to buoyancy effects. The cooling performance of the flange therefore depends on both convection and conduction.

To simplify the simulation convection cooling is analyzed using the heat transfer coefficient, h. Since this describes the fluid flow influence and the convective fluxes, the flow field does not have to be computed.

This particular example uses external research data (Ref. 1) for the outer surface heat transfer coefficient, which was originally obtained through semi-empirical data for natural

convection around a cylinder. For the tube-facing surface a coefficient for forced convection in a tube is used.

Model Definition

Figure 2 presents the modeled geometry.



Figure 2: Drawing of the cooling flange.

The glass flange consists of a 4 mm thick pipe and 4 mm thick and 10 mm tall flanges, with a connecting pipe that has a 3 mm thick wall and an inner diameter of 16 mm. Note that in this tutorial, a parametric study on the pipe inner diameter is performed while the other dimensions remain fixed.

During operation, the hot process fluid heats the inside of the tube. The flange conducts the heat and transfers it to the surrounding air. As the air is heated, buoyancy effects cause a convective flow.

The heat transfer within the flange is described by the stationary heat equation

$$\nabla \cdot (-k\nabla T) = 0$$

where *k* is the thermal conductivity $(W/(m \cdot K))$, and *T* is the temperature (K). On the flange's exterior boundaries, which face the air and process fluid, the applicable boundary condition is

$$-\mathbf{n} \cdot (-k\nabla T) = h(T_{\text{ext}} - T)$$

where **n** is the normal vector of the boundary, h, is the heat transfer coefficient (W/ (m²·K)), and T_{inf} is the temperature of the surrounding medium (K). For this simulation, set T_{ext} to 298 K for the cooling air and to 363 K for the process fluid.

You can approximate the value for the heat transfer coefficient, h, on the process fluid side with a constant value of 15 W/(m²·K) because the fluid's velocity is close to constant and the model assumes that its temperature decreases only slightly.

The *h* expression on the air side is more elaborate. Assume that the free-convection process around the flange is similar to that around a cylinder. The heat transfer coefficient for a cylinder is available in the literature (Ref. 1), and you can use the expression

$$h = \frac{k}{L}f(\theta)\mathrm{Gr}^{1/4}$$

where *k* is the thermal conductivity of air (26.2 mW/(m·K) at 298 K); *L* is the typical length, which in this case is the outer diameter of the flange (44 mm); and $f(\theta)$ is an empirical coefficient tabulated in Table 1 as a function of the incidence angle θ , which is shown in Figure 3. Finally, **Gr** is the Grashof number defined as

$$Gr = \frac{g\alpha_p \Delta TL^3}{v^2}$$

~

where α_p is the coefficient of thermal expansion (1/K), which equals that for an ideal gas, g is the gravitational acceleration (9.81 m/s²), and v is the kinematic viscosity (18·10⁻⁶ m²/s). For the flange material, use silica glass.

TABLE I: EMPIRICAL TRANSFER COEFFICIENT VS. INCIDENCE ANGLE.

θ (DEG)	$f(\theta)$
0	0.48
90	0.46
100	0.45
110	0.435
120	0.42
130	0.38

TABLE I: EMPIRICAL TRANSFER COEFFICIENT VS. INCIDENCE ANGLE.

θ (DEG)	$f(\theta)$
140	0.35
150	0.28
160	0.22
180	0.15



Figure 3: Definition of the incidence angle θ .

Results and Discussion



Figure 4 shows the flange surface temperature at steady state.

Figure 4: Stationary surface temperature of the flange.

As you can see in the surface temperature plot above to the left, the tube surface temperature is about 13 K higher than the flange shoulders. From the results, we can learn that the heat transfer from the outer surfaces of the flange is pretty efficient; there is a temperature difference of roughly 19 K between the outer flange surface and the air stream.

Figure 5 shows the heat transfer coefficient for the flange's outer walls.



Surface: Heat transfer film coefficient (W/(m²*K))

Figure 5: Heat transfer film coefficient, h, for the flange's outer walls.

As you can see, the coefficient decreases significantly along the vertical position of the flange's outer boundary.

Calculating the flange's total cooling power by integrating the normal total heat flux over the outer surfaces gives a value of 0.51 W which correspond to 1.02 W for the full geometry.

The surface temperature plot shown on Figure 4 further gives light to an inefficiency in the current flange design. When it enters the flange the process fluid is 363 K, while the inner surface of the pipe is only 32 K lower. Increasing the tube diameter would improve the heat transfer here, which is performed in a second stage of the modeling by varying the pipe's diameter, but keeping all other factors constant. By plotting in Figure 6 the global cooling power as a function of the inner pipe radius, you can now analyze how altering the radius impacts the performance of the cooling flange.



Figure 6: Total cooling power versus inner pipe radius.

Reference

1. B. Sundén, Kompendium i värmeöverföring [Notes on Heat Transfer], Sec. 10-3, Dept. of Heat and Power Engineering, Lund Inst. of Technology, 2003 (in Swedish).

Application Library path: Heat_Transfer_Module/Thermal_Processing/ cooling_flange

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click 🔗 Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 间 3D.
- 2 In the Select Physics tree, select Heat Transfer>Heat Transfer in Solids (ht).
- 3 Click Add.
- 4 Click \bigcirc Study.
- 5 In the Select Study tree, select General Studies>Stationary.
- 6 Click M Done.

GLOBAL DEFINITIONS

Parameters 1

Define parameters for relevant temperatures, material properties, and geometric dimensions. The parameters related to the geometry dimensions make it easier to perform parametric studies where you let some of these dimensions vary.

I In the Model Builder window, under Global Definitions click Parameters I.

2 In the Settings window for Parameters, locate the Parameters section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
k	26.2[mW/(m*K)]	0.0262 W/(m·K)	Thermal conductivity, air
T_inner	363[K]	363 K	Temperature, process fluid
Hh	15[W/(m^2*K)]	15 W/(m ² ·K)	Heat transfer coefficient
nu	18e-6[m^2/s]	1.8E-5 m ² /s	Kinematic viscosity
r_inner	8[mm]	0.008 m	Inner pipe radius
11	12[mm]	0.012 m	Pipe length excluding flanges
t1	3[mm]	0.003 m	Pipe thickness
t2	4[mm]	0.004 m	Pipe thickness, flange section
hf	10[mm]	0.01 m	Flange height
wf	4[mm]	0.004 m	Flange width
D	2*(r_inner+t2+hf)	0.044 m	Outer flange diameter

Next, create an interpolation function defined by the data in Table 1.

Interpolation 1 (int1)

I In the Home toolbar, click f(X) Functions and choose Global>Interpolation.

2 In the Settings window for Interpolation, locate the Definition section.

3 In the Function name text field, type f.

4 In the table, enter the following settings:

t	f(t)
0	0.48
90	0.46
100	0.45
110	0.435
120	0.42
130	0.38
140	0.35
150	0.28
160	0.22
180	0.15

5 Locate the **Units** section. In the **Argument** table, enter the following settings:

Argument	Unit
t	deg

6 In the **Function** table, enter the following settings:

Function	Unit
f	1





GEOMETRY I

In a first geometry modeling stage, create a 2D geometry by following the steps below.

Work Plane I (wp1)

- I In the Geometry toolbar, click 📥 Work Plane.
- 2 In the Settings window for Work Plane, click 📥 Show Work Plane.

Work Plane I (wpI)>Plane Geometry

In the Model Builder window, click Plane Geometry.

Work Plane I (wp1)>Rectangle I (r1)

- I In the Work Plane toolbar, click 📃 Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type 2*wf.
- 4 In the **Height** text field, type t2.
- 5 Locate the **Position** section. In the **yw** text field, type r_inner.
- 6 In the Work Plane toolbar, click 🟢 Build All.
- 7 Click the 🕂 Zoom Extents button in the Graphics toolbar.

Work Plane I (wpl)>Quadratic Bézier I (qbl)

- I In the Work Plane toolbar, click 🚧 More Primitives and choose Quadratic Bézier.
- 2 In the Settings window for Quadratic Bézier, locate the Control Points section.
- 3 In row I, set yw to r_inner+t2.
- 4 In row 2, set xw to wf/2, and yw to r_inner+t2.
- 5 In row 3, set xw to wf/2, and yw to r_inner+t2+wf/2.

Work Plane I (wp1)>Line Segment I (ls1)

- I In the Work Plane toolbar, click 😕 More Primitives and choose Line Segment.
- 2 In the Settings window for Line Segment, locate the Starting Point section.
- **3** From the **Specify** list, choose **Coordinates**.
- 4 Locate the Endpoint section. From the Specify list, choose Coordinates.
- 5 Locate the Starting Point section. In the xw text field, type wf/2.
- 6 Locate the Endpoint section. In the xw text field, type 3*wf/2.
- 7 Locate the Starting Point section. In the yw text field, type r inner+t2+wf/2.
- 8 Locate the **Endpoint** section. In the **yw** text field, type r_inner+t2+wf/2.

Work Plane I (wpI)>Quadratic Bézier 2 (qb2)

- I In the Work Plane toolbar, click 🚧 More Primitives and choose Quadratic Bézier.
- 2 In the Settings window for Quadratic Bézier, locate the Control Points section.
- 3 In row I, set xw to 3*wf/2, and yw to r_inner+t2+wf/2.
- 4 In row 2, set xw to 3*wf/2, and yw to r_inner+t2.
- 5 In row 3, set xw to 2*wf, and yw to r_inner+t2.

Work Plane 1 (wp1)>Line Segment 2 (ls2)

- I In the Work Plane toolbar, click 😕 More Primitives and choose Line Segment.
- 2 In the Settings window for Line Segment, locate the Starting Point section.
- 3 From the Specify list, choose Coordinates.
- 4 Locate the Endpoint section. From the Specify list, choose Coordinates.
- **5** Locate the **Starting Point** section. In the **yw** text field, type r_inner+t2.
- 6 Locate the Endpoint section. In the xw text field, type 2*wf.
- 7 In the **yw** text field, type r_inner+t2.
- 8 Drag and drop below Quadratic Bézier 2 (qb2).

Work Plane I (wp1)>Convert to Solid I (csol1)

- I In the Work Plane toolbar, click া Conversions and choose Convert to Solid.
- 2 Select the objects Is1, Is2, qb1, and qb2 only.
- 3 In the Work Plane toolbar, click 📗 Build All.
- **4** Click the **Com Extents** button in the **Graphics** toolbar.

Work Plane 1 (wp1)>Rectangle 2 (r2)

- I In the Work Plane toolbar, click Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type wf.
- 4 In the **Height** text field, type hf-wf.
- 5 Locate the Position section. In the xw text field, type wf/2.
- 6 In the yw text field, type r_inner+t2+wf/2.
- 7 In the Work Plane toolbar, click 🟢 Build All.
- 8 Click the **Zoom Extents** button in the **Graphics** toolbar.

Work Plane I (wpI)>Circle I (cI)

- I In the Work Plane toolbar, click 🕑 Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type wf/2.
- 4 In the Sector angle text field, type 180.
- 5 Locate the Position section. In the xw text field, type wf.
- 6 In the yw text field, type r_inner+t2+hf-wf/2.
- 7 In the Work Plane toolbar, click 🟢 Build All.
- 8 Click the 🕂 Zoom Extents button in the Graphics toolbar.

Work Plane I (wp1)>Array I (arr1)

- I In the Work Plane toolbar, click 💭 Transforms and choose Array.
- 2 Click in the Graphics window and then press Ctrl+A to select all objects.
- 3 In the Settings window for Array, locate the Size section.
- 4 From the Array type list, choose Linear.
- **5** In the **Size** text field, type **3**.
- 6 Locate the Displacement section. In the xw text field, type 2*wf.
- 7 In the Work Plane toolbar, click 📗 Build All.

8 Click the + Zoom Extents button in the Graphics toolbar.

Work Plane I (wp1)>Polygon I (pol1)

- I In the Work Plane toolbar, click / Polygon.
- 2 In the Settings window for Polygon, locate the Object Type section.
- **3** From the **Type** list, choose **Open curve**.
- **4** Locate the **Coordinates** section. From the **Data source** list, choose **Vectors**.
- **5** In the **xw** text field, type 0 0 -11 -11 -11 -11 -2*11/3.
- 6 In the yw text field, type r_inner+t2 r_inner r_inner r_inner r_inner+t1 r_inner+t1 r_inner+t1.

Work Plane I (wp1)>Cubic Bézier I (cb1)

- I In the Work Plane toolbar, click 🚧 More Primitives and choose Cubic Bézier.
- 2 In the Settings window for Cubic Bézier, locate the Control Points section.
- 3 In row I, set xw to -2*11/3, and yw to r_inner+t2.
- 4 In row 2, set xw to -11/3, and yw to r_inner+t2.
- 5 In row 3, set xw to -11/3, and yw to r_inner+t2.
- 6 In row 4, set yw to r_inner+t2.

Work Plane I (wp1)>Convert to Solid 2 (csol2)

- I In the Work Plane toolbar, click 🕅 Conversions and choose Convert to Solid.
- 2 Select the objects **cb1** and **pol1** only.
- 3 In the Work Plane toolbar, click 📗 Build All.
- **4** Click the **Comextents** button in the **Graphics** toolbar.

5 In the Model Builder window, right-click Geometry 1 and choose Build All. You obtain the following 2D geometry.



Next, revolve the embedded 2D geometry to create the 3D model geometry.

Revolve I (rev1)

- I In the Geometry toolbar, click \leftarrow Revolve.
- 2 In the Settings window for Revolve, locate the Revolution Angles section.
- **3** Click the **Angles** button.
- 4 In the End angle text field, type 180.
- 5 Locate the Revolution Axis section. Find the Direction of revolution axis subsection. In the xw text field, type 1.
- **6** In the **yw** text field, type 0.
- 7 In the Geometry toolbar, click 🟢 Build All.

8 Click the $\sqrt[1]{}$ Go to Default View button in the Graphics toolbar.



DEFINITIONS

Variables I

- I In the Home toolbar, click a= Variables and choose Local Variables.
- 2 In the Settings window for Variables, locate the Geometric Entity Selection section.
- 3 From the Geometric entity level list, choose Boundary.

4 Select Boundaries 3, 7, 9, 10, 13, 19, 23, 25, 29, 32, 35, 38–42, 45, 51, 55, 57, 61, 64, 67, 70–74, 77, 83, 87, 89, 93, 96, 99, and 102–106 only.

To do this, first copy the list of boundaries from this text, then use the **Paste Selection** button in the **Geometric Entity Selection** section, to paste these numbers.



For use when specifying the boundary condition for the flange's outer surface, create a selection.

- 5 Click here a Create Selection.
- 6 In the **Create Selection** dialog box, type **Outer Boundaries** in the **Selection name** text field.
- 7 Click OK.
- 8 In the Settings window for Variables, locate the Variables section.
- **9** In the table, enter the following settings:

Name	Expression	Unit	Description
alphap	1/ampr1.T_amb		Coefficient of thermal expansion
Gr	g_const*alphap*(T- ampr1.T_amb)*D^3/nu^2		Grashof number

Name	Expression	Unit	Description
theta	atan(y/z)+90[deg]	rad	Incidence angle
Нс	k*f(theta)*Gr^0.25/D		Heat transfer film coefficient

ADD MATERIAL

- I 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>Silica glass**.
- 4 Click Add to Component in the window toolbar.
- 5 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

DEFINITIONS

Ambient Properties 1 (ampr1)

- I In the Physics toolbar, click \equiv 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 298[K].

HEAT TRANSFER IN SOLIDS (HT)

The next steps set the temperature shape function to **Quadratic serendipity** in order to reduce computational costs without significant deterioration in accuracy.

- I In the Model Builder window, under Component I (compl) click Heat Transfer in Solids (ht).
- **2** In the **Settings** window for **Heat Transfer in Solids**, click to expand the **Discretization** section.
- 3 From the Temperature list, choose Quadratic serendipity.

Initial Values 1

- I In the Model Builder window, under Component I (comp1)>Heat Transfer in Solids (ht) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** In the *T* text field, type **323.15**[K].

Symmetry I

- I In the Physics toolbar, click 🔚 Boundaries and choose Symmetry.
- 2 Click the 🕀 Wireframe Rendering button in the Graphics toolbar.

3 Select Boundaries 2, 8, 12, 15, 21, 22, 24, 27, 33, 34, 44, 47, 53, 54, 56, 59, 65, 66, 76, 79, 85, 86, 88, 91, 97, and 98 only.





I In the Physics toolbar, click 🔚 Boundaries and choose Heat Flux.

2 Select Boundaries 4, 6, 16, 18, 48, 50, 80, and 82 only.



- 3 In the Settings window for Heat Flux, locate the Heat Flux section.
- 4 From the Flux type list, choose Convective heat flux.
- **5** In the *h* text field, type Hh.
- **6** In the T_{ext} text field, type T_inner.
- 7 Click the Wireframe Rendering button in the Graphics toolbar to go back to the original view.

Heat Flux 2

- I In the Physics toolbar, click 🔚 Boundaries and choose Heat Flux.
- 2 In the Settings window for Heat Flux, locate the Boundary Selection section.
- 3 From the Selection list, choose Outer Boundaries.
- 4 Locate the Heat Flux section. From the Flux type list, choose Convective heat flux.
- **5** In the *h* text field, type Hc.
- **6** From the $T_{\rm ext}$ list, choose **Ambient temperature (amprl)**.

MESH I

Mapped I

- I In the Mesh toolbar, click \triangle Boundary and choose Mapped.
- 2 Click the \$\frac{\timesy}{\timesy}\$ Go to XY View button in the Graphics toolbar. Do this twice to see the boundary mesh.
- **3** Select Boundaries 8, 21, 22, 33, 53, 54, 65, 85, 86, and 97 only.



Size I

- I In the Mesh toolbar, click Size Attribute and choose Extra Fine.
- 2 In the Settings window for Size, click 🖷 Build Selected.

Free Triangular 1

- I In the Mesh toolbar, click \triangle Boundary and choose Free Triangular.
- 2 Select Boundaries 34, 66, and 98 only.



3 In the Settings window for Free Triangular, click 📗 Build Selected.

4 Click the $\sqrt[1]{}$ **Go to Default View** button in the **Graphics** toolbar.

Swept I

In the Mesh toolbar, click As Swept.

Distribution I

- I Right-click Swept I and choose Distribution.
- 2 In the Settings window for Distribution, locate the Distribution section.
- **3** In the **Number of elements** text field, type **30**.

4 Click 📗 Build All.



STUDY I

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

RESULTS

Temperature (ht)

The first default plot group shows the temperature field on the surface. To get a more intuitive view with gravity along the vertical, rotate the geometry in the **Graphics** window. You can preserve a view for a plot by creating a **View** feature node as follows:

- I Click the 🐱 Show More Options button in the Model Builder toolbar.
- 2 In the Show More Options dialog box, in the tree, select the check box for the node Results>Views.
- 3 Click OK.

View 3D 3

- I In the Model Builder window, under Results right-click Views and choose View 3D.
- 2 Use the Graphics toolbox to get a satisfying view.
- 3 In the Settings window for View 3D, locate the View section.

4 Select the **Lock camera** check box.

Next, apply the view to the temperature plot.

Temperature (ht)

- I In the Model Builder window, under Results click Temperature (ht).
- 2 In the Settings window for 3D Plot Group, locate the Plot Settings section.
- 3 From the View list, choose View 3D 3.
- 4 In the Temperature (ht) toolbar, click 💿 Plot.

Compare with Figure 4.

Delete the second plot group to make a new surface plot of the heat transfer film coefficient.

Isothermal Contours (ht)

- I In the Model Builder window, under Results right-click Isothermal Contours (ht) and choose Delete.
- 2 Click Yes to confirm.

Heat Transfer Film Coefficient

- I In the Home toolbar, click 🚛 Add Plot Group and choose 3D Plot Group.
- 2 In the Settings window for 3D Plot Group, type Heat Transfer Film Coefficient in the Label text field.
- 3 Locate the Plot Settings section. From the View list, choose View 3D 3.

Surface 1

- I In the Heat Transfer Film Coefficient toolbar, click 🔲 Surface.
- In the Settings window for Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (comp1)>Definitions> Variables>Hc Heat transfer film coefficient W/(m²·K).
- **3** In the **Heat Transfer Film Coefficient** toolbar, click **I** Plot.

Compare this plot with that in Figure 5.

Outgoing Heat Flux

- I In the Results toolbar, click ^{8,85}_{e-12} More Derived Values and choose Integration> Surface Integration.
- **2** In the **Settings** window for **Surface Integration**, type **Outgoing Heat Flux** in the **Label** text field.
- 3 Locate the Selection section. From the Selection list, choose Outer Boundaries.

- 4 Click Replace Expression in the upper-right corner of the Expressions section. From the menu, choose Component I (compl)>Heat Transfer in Solids>Boundary fluxes>ht.ntflux -Normal total heat flux - W/m².
- 5 Click **=** Evaluate.

TABLE

I Go to the Table window.

The integrated value, approximately 0.51 W, appears in the **Table** tab below the **Graphics** window. Taking both flange halves into account, the total cooling power of the flange is thus roughly 1 W.

Finally, extend the model by performing a parametric sweep over the inner pipe radius. Before adding a separate study for this purpose, define a variable for the total cooling power.

DEFINITIONS

Integration, Outer

- I In the Definitions toolbar, click 🖉 Nonlocal Couplings and choose Integration.
- 2 In the Settings window for Integration, type Integration, Outer in the Label text field.
- 3 In the **Operator name** text field, type intop_outer.
- **4** Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
- 5 From the Selection list, choose Outer Boundaries.

Variables 2

- I In the **Definitions** toolbar, click $\partial =$ **Local Variables**.
- 2 In the Settings window for Variables, locate the Variables section.
- **3** In the table, enter the following settings:

Name	Expression	Unit	Description
P_cooling	2*intop_outer(ht.q0)	W	Cooling power

ADD STUDY

- I In the Home toolbar, click $\stackrel{\sim}{\sim}$ Add Study to open the Add Study window.
- 2 Go to the Add Study window.
- 3 Find the Studies subsection. In the Select Study tree, select General Studies>Stationary.
- 4 Click Add Study in the window toolbar.

5 In the Home toolbar, click \sim Add Study to close the Add Study window.

STUDY 2

Parametric Sweep

- I In the Study toolbar, click **Parametric Sweep**.
- 2 In the Settings window for Parametric Sweep, locate the Study Settings section.
- 3 Click + Add.
- **4** In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
r_inner (Inner pipe radius)	range(6,1,10)	mm

This gives a sweep centered around the original radius value.

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

RESULTS

Temperature (ht) 1

You get a new surface plot of the temperature for the parametric solution. From the **Parameter value** list, you can choose the radius value for which to display the result.

I In the Settings window for 3D Plot Group, locate the Plot Settings section.

2 From the View list, choose View 3D 3.



Surface temperature for an inner radius of 10 mm.

Finally, replace the slice plot in the fourth plot group by a graph of the total cooling power versus tube radius.

Isothermal Contours (ht)

- I Right-click Isothermal Contours (ht) and choose Delete.
- 2 Click Yes to confirm.

Cooling Power

- I In the Home toolbar, click 📠 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Cooling Power in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Study 2/ Parametric Solutions 1 (sol3).

Global I

- I In the Cooling Power toolbar, click (Global.
- In the Settings window for Global, click Add Expression in the upper-right corner of the y-Axis Data section. From the menu, choose Component I (compl)>Definitions> Variables>P_cooling Cooling power W.

3 Click to expand the Legends section. Clear the Show legends check box.

Cooling Power

- I In the Model Builder window, click Cooling Power.
- 2 In the Settings window for ID Plot Group, locate the Plot Settings section.
- 3 Select the x-axis label check box. In the associated text field, type Inner pipe radius (m).
- **4** In the **Cooling Power** toolbar, click **I Plot**.

Compare the result with the graph in Figure 6.