COMSOL MULTIPHYSICS®

Cooling Flange

SOLVED WITH COMSOL MULTIPHYSICS 3.5a

© COPYRIGHT 2008. All right reserved. No part of this documentation may be photocopied or reproduced in any form without prior written consent from COMSOL AB. COMSOL, COMSOL Multiphysics, COMSOL Reaction Engineering Lab, and FEMLAB are registered trademarks of COMSOL AB. Other product or brand names are trademarks or registered trademarks of their respective holders.

COMSOL

Cooling Flange

Introduction

In the chemical industry, processes often cool reaction fluids using glass flanges. In most cases the coolant is the surrounding air. An obvious design parameter for this type of device is the cooling power, and the surface temperatures might also be of interest. Heat transfer in this type of device is dominated by convection to and from the surfaces, although the conduction within the glass flange can also influence performance. A convenient method to analyze convection cooling is to use a heat transfer coefficient, h. This coefficient describes the influence of the fluid-flow field and the convective fluxes. Thus it is not necessary to model the flow field, which greatly simplifies simulations.

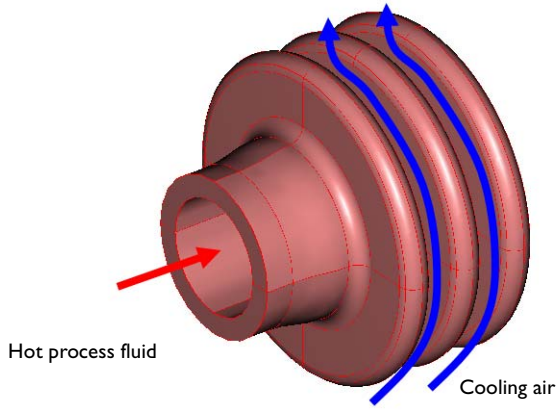


Figure 1: Operating principle of the cooling flange.

Semi-empirical expressions for computing the heat transfer coefficient for different cases are available in the literature. For this model, the author obtained the heat transfer coefficient for the outer surface by using semi-empirical data available for natural convection around a cylinder. The heat transfer coefficient for the surface that faces the tube is valid for forced convection in a tube. The model uses the General Heat Transfer application mode.

Model Definition

Figure 2 presents a drawing of the modeled geometry.

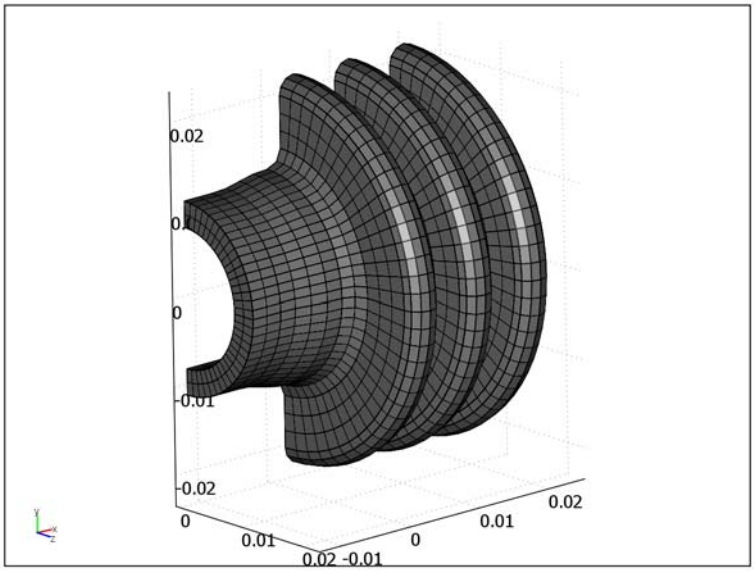


Figure 2: Drawing of the cooling flange.

The pipe connecting the flange has an inner diameter of 16 mm and a wall thickness of 3 mm. In the flange section, the pipe is 4 mm thick. The flanges are 4 mm thick and 10 mm in height.

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·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

$$-\mathbf{n} \cdot (-k\nabla T) = q_0 + h(T_{\text{inf}} - T)$$

where $\mathbf{n}$ is the normal vector of the boundary, h is the heat transfer coefficient (W/(m^2·K)), and T_{inf} is the temperature of the surrounding medium (K). For this simulation, set T_{inf} 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/(m^2·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 (0.06 W/(m·K)), L is the characteristic length that in this case is the outer diameter of the flange (44 mm), and $f(\theta)$ is an empirical coefficient tabulated in Table 1. (Figure 3 illustrates the definition of the angle θ). Finally, Gr is the Grashof number defined as

$$\mathrm{Gr} = \frac{\beta g \Delta T L^3}{\mu^2}$$

where β is the thermal expansion coefficient (1/K), which equals $1/T_\infty$ for an ideal gas, g is the gravitational acceleration (9.81 m/s^2), and μ is the kinematic viscosity ($18{\cdot}10^{-6}$ Pa·s).

TABLE 1: EMPIRICAL TRANSFER COEFFICIENTS

INCIDENT ANGLE [DEG.]	$f(\theta)$
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

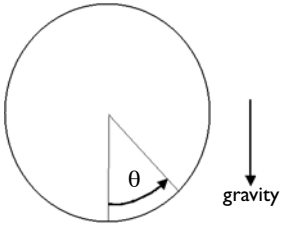


Figure 3: Definition of the angle θ.

Table 1 summarizes the material properties of the flange material.

TABLE 1: MATERIAL PROPERTIES

MATERIAL	k [W/(m·K)]	ρ [kg/m^3]	C_p [J/(kg·K)]
Silica glass	1.38	2203	703

Results and Discussion

Figure 5 shows the surface temperature of the flange at steady state.

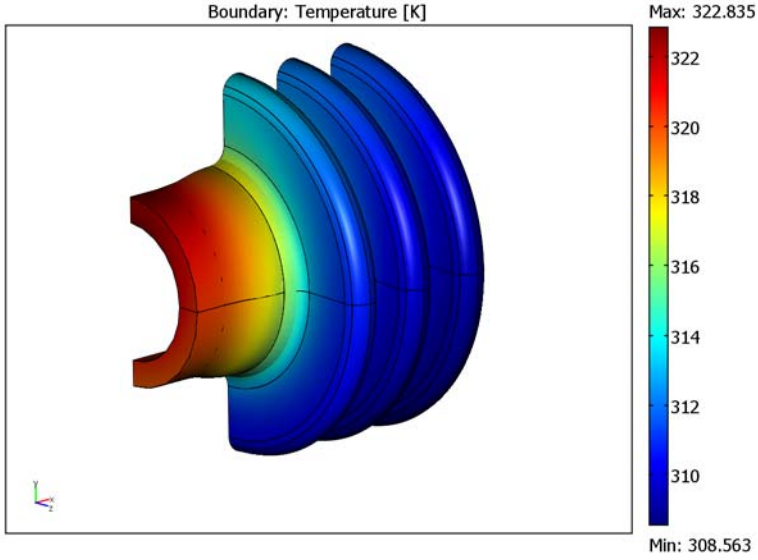


Figure 4: Stationary surface temperature of the flange.

The temperature at the flange shoulders is approximately 14 K lower than that at the tube surface. The temperature difference between the process fluid at 363 K and the inner surface of the pipe is approximately 40 K, while that between the outer flange surface and the air stream is approximately 10 K. These values indicate that the heat transfer from the flange outer surfaces is efficient. It also indicates that the heat transfer from the fluid to the flange is a limiting factor. To improve the flange's performance, it is a good idea to increase the tube diameter.

Figure 5 shows the heat transfer coefficient for the flange and pipe walls.

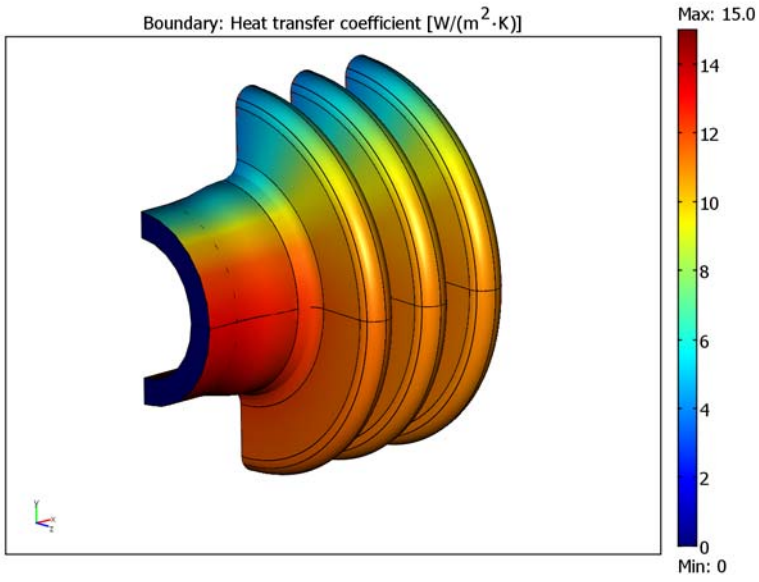


Figure 5: Heat transfer film coefficient, h, for the flange.

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

Calculate the flange's total cooling power by integrating the heat flux on the outer surfaces. The entire flange, that is, taking both symmetry halves into account, has a cooling power of approximately 1.2 W.

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).

Model Library path: `Heat_Transfer_Module/Process_and_Manufacturing/cooling_flange`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

Open the **Model Navigator** and go to the **New** page. In the **Space dimension** list select **2D**, then click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu select **Constants**. Define the following names, expressions, and (optionally) descriptions; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
D	44[mm]	Outer flange diameter
k	0.06[W/(m*K)]	Thermal conductivity
Tair	298[K]	Cooling air temperature
Tinner	363[K]	Process fluid temperature
Hh	15[W/(m^2*K)]	Heat transfer coefficient
visc	18e-6[m^2/s]	Kinematic viscosity
beta	1/Tair	Thermal expansion coefficient
grav	9.81[m/s^2]	Gravitational acceleration

2 From the **Options** menu select **Functions**. In the dialog box that appears, click **New** to open the **New Function** dialog box.

3 In the **Function name** edit field, type `graph`. Click **Interpolation** and select **Table** in the **Use data from** list. Click **OK**.

4 Select **Linear** in the **Interpolation method** list in the **Functions** dialog box that appears. Enter the following values in the columns for **x** and **f(x)**; when done, click **OK**.

x	f(x)
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 From the **Options** menu select **Axes/Grid Settings**.

6 Clear the **Axis equal** check box, then enter the properties in the following table:

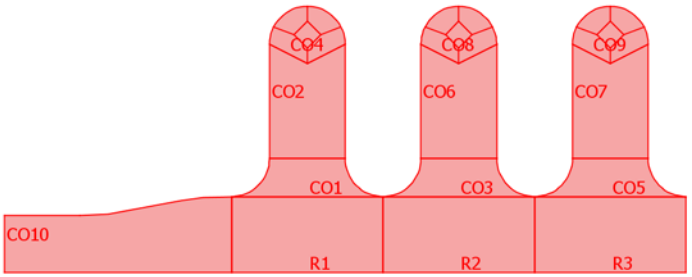
PROPERTY	VALUE
x min	-1.4
x max	2.6
y min	0.2
y max	2.8

7 Go to the **Grid** page. Clear the **Auto** check box, then enter the properties in the following table; when done, click **OK**.

PROPERTY	VALUE
x spacing	0.1
y spacing	0.1

GEOMETRY MODELING

Start by creating this 2D geometry:



Create the rectangle R1 with these steps:

1. Shift-click the **Rectangle/Square** button on the Draw toolbar.
2. In the dialog box that appears, enter the following settings; when done, click **OK**.

OBJECT DIMENSIONS	EXPRESSION
Width	0.8
Height	0.4
Base	Corner
x-position	0
y-position	0.8

Create the composite object CO1 with these steps:

1. Click the **2nd Degree Bézier Curve** button on the Draw toolbar.
2. Click on the coordinates (0, 1.2), (0.2, 1.2), and (0.2, 1.4).
3. Click the **Line** button on the Draw toolbar, then click on the coordinate (0.6, 1.4).
4. Click the **2nd Degree Bézier Curve** on the Draw toolbar.
5. Draw the Bézier curve by clicking on the coordinates (0.6, 1.4), (0.6, 1.2), and (0.8, 1.2).
6. Click the right mouse button to create the geometry object.

Create the composite object CO2 with these steps:

1. Click the **Line** button on the Draw toolbar, then click on the coordinates (0.2, 1.4), (0.2, 2.0), (0.4, 1.9), (0.6, 2.0), and (0.6, 1.4).
2. Click the right mouse button to create the geometry object.

Create the composite object CO4 with these steps:

1. Click the **2nd Degree Bézier Curve** button on the Draw toolbar.
2. Click on the coordinates (0.2, 2.0), (0.2, 2.2), (0.4, 2.2), (0.6, 2.2), and (0.6, 2.0).
3. Click the **Line** button on the Draw toolbar, then click on the coordinates (0.4, 1.9) and (0.2, 2.0). Click the right mouse button.

   This creates a composite geometry object, CO3. To create CO4, you still must add a square and some lines to the object.
4. Shift-click the **Rectangle/Square (Centered)** button on the Draw toolbar.
5. In the dialog box that appears, enter the following settings; when done, click **OK**.

PROPERTY	VALUE
Width	0.1
α	45
Base	Center
x-position	0.4
y-position	2.05

6. Click the **Line** button on the Draw toolbar. Click on the left corner of the small rectangle, then click on the coordinate (0.2, 2.1). Click the right mouse button.
7. Click the **Line** button on the Draw toolbar, click on the right corner of the small rectangle, then click on the coordinate (0.6, 2.1). Click the right mouse button.
8. Click the **Line** button on the Draw toolbar, click on the upper corner of the small rectangle, then click on the coordinate (0.4, 2.2). Click the right mouse button.
9. Click the **Line** button on the Draw toolbar, click on the lower corner of the small rectangle, then click on the coordinate (0.4, 1.9). Click the right mouse button.
10. To create the composite object, select CO3, SQ1, B1, B2, B3, and B4, then click the **Coerce to Solid** button on the Draw toolbar.

Create the other two flanges with these steps:

1. Select R1, CO1, CO2, and CO4, then click the **Array** button on the Draw toolbar.
2. In the dialog box that appears, enter the following properties; when done, click **OK**.

PROPERTY	VALUE
Displacement, x	0.8
Displacement, y	0

PROPERTY	VALUE
Array size, x	3
Array size, y	1

Create the composite object CO10 with these steps:

1. Click the **Line** button on the Draw toolbar.
2. Click on the lower left corner of rectangle R1, then click, in order, on the coordinates (–1.2, 0.8), (–1.2, 1.1), and (–0.8, 1.1).
3. Click the **3rd Degree Bézier Curve** button on the Draw toolbar.
4. Click, in order, on the coordinates (–0.4, 1.1), (–0.4, 1.2), and (0, 1.2).
5. Click the right mouse button to create the geometry object CO10.

Conclude the geometry modeling by scaling the geometry with these steps:

1. Select all geometry objects by pressing Ctrl+A.
2. Click the **Scale** button on the Draw toolbar.
3. In the dialog box that appears, enter the following scaling properties; when done, click **OK**.

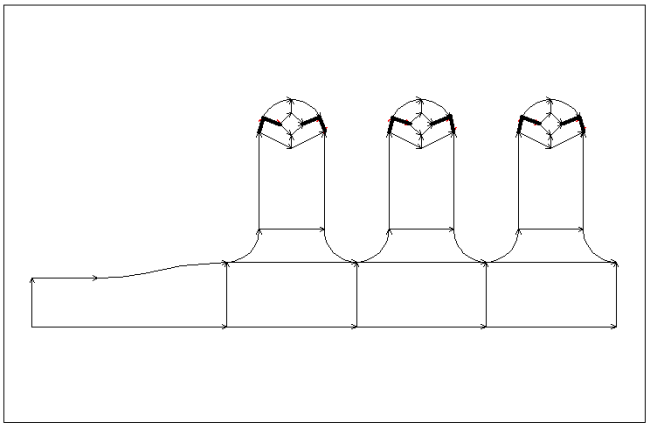
PROPERTY	VALUE
Scale factor, x	0.01
Scale factor, y	0.01
Scale base point, x	0
Scale base point, y	0

4. Double-click the **EQUAL** button on the status bar at the bottom of the user interface.
5. Click the **Zoom Extents** button on the Main toolbar.

MESH GENERATION

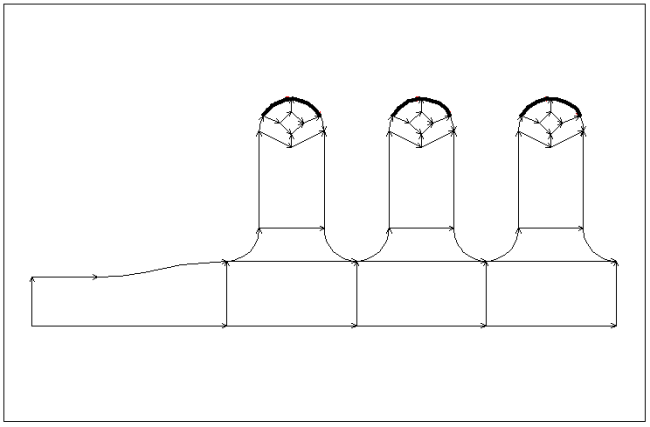
1. From the **Mesh** menu select **Mapped Mesh Parameters**.

2 On the **Boundary** page, select the boundaries marked in the following figure:



3 Select the **Constrained edge element distribution** check box (keep the default value for the **Number of edge elements** at 1).

4 Select the boundaries marked in the following figure:



5 Select the **Constrained edge element distribution** check box, then in the **Number of edge elements** edit field type 2.

6 Click **Remesh**, then click **OK**.

7 From the **Mesh** menu select **Revolve Mesh**.

8 In the dialog box that appears, in the α**2** edit field type 180.

9 Click the **Angle from x-axis** button, and in the θ edit field type 0.

10 Click **OK**.

PHYSICS SETTINGS

1 From the **Multiphysics** menu select **Model Navigator**.

2 In the **Multiphysics** menu on the right side of the dialog box select **Geom2 (3D)**.

3 From the list of application modes on the left side of the dialog box select **Heat Transfer Module>General Heat Transfer>Steady-state analysis**.

4 Click **Add**, then click **OK**.

Subdomain Settings

1 From the **Physics** menu, select **Subdomain Settings**.

2 Select all subdomains.

Next, select the material properties from the materials library:

3 Press the **Load** button. In the dialog box that appears, in the **Materials** list select **Library I>Silica glass**. Click **OK**.

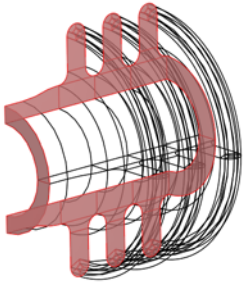
4 Click the **Init** tab, then in the **Temperature** edit field type 323.

5 Click **OK**.

Boundary Settings

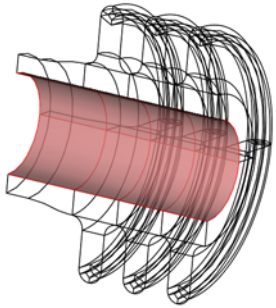
1 From the **Physics** menu select **Boundary Settings**.

2 Select the boundaries at the geometry's ends and the symmetry boundaries, as shown in the following figure:



3 In the **Boundary condition** list select **Insulation/Symmetry**.

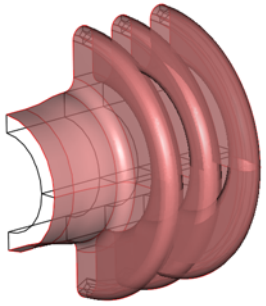
4 Select the boundaries that face the inner channel, as shown in the following figure:



5 In the **Boundary condition** list select **Heat flux**, then enter the following properties:

PROPERTY	VALUE
h	Hh
T_{inf}	Tinner

6 Select Boundary 3, then select the **Select by group** check box. All boundaries on the outside of the geometry should now be selected, as in the following figure:



7 In the **Boundary condition** list select **Heat flux**, then enter the following properties:

PROPERTY	VALUE
h	Hc
T_{inf}	Tair

8 Click **OK**.

9 From the **Options** menu select **Expressions>Boundary Expressions**. Keep the selection of all outward-facing boundary segments from Step 6.

10 Enter the following expressions; when done, click **OK**.

EXPRESSION	VALUE
angle	atan(y/z)*180/pi+90
Gr	grav*beta*(T-Tair)*D^3/visc^2
Hc	k*graph(angle)*Gr^0.25/D

COMPUTING THE SOLUTION

1. Click the **Solver Parameters** button on the Main toolbar.
2. In the **Linear system solver** list select **GMRES**.
3. In the **Preconditioner** list select **Algebraic multigrid**.
4. Click **OK**.
5. Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

To reproduce the plot in Figure 4, displaying the boundary temperature distribution, follow these instructions:

1. Click the **Plot Parameters** button on the Main toolbar.
2. Clear the **Slice** check box and select the **Boundary** check box, then click **Apply**.
3. Double-click the **GRID** button on the status bar at the bottom of the user interface.

To reproduce the plot in Figure 5, continue with these steps:

1. Still in the **Plot Parameters** dialog box, click the **Boundary** tab.
2. In the **Predefined quantities** list select **Heat transfer coefficient**, then click **OK**.

To integrate the heat flux over the outer surface area of the flange, do as follows:

1. From the **Postprocessing** menu open the **Boundary Integration** dialog box.
2. In order to select all outer boundaries, open the **Boundary Settings** dialog box, then select Boundary 3 and select the **Select by group** check box. Click **Cancel** to close the **Boundary Settings** dialog box.
3. Return to the **Boundary Integration** dialog box. From the **Predefined quantities** list select **Normal total heat flux**, then click **OK**.

The integrated value appears in the message log at the bottom of the user interface.