FEMLAB® 3

Multiphysics Modeling

Cooling Flange solved with FEMLAB 3

© COPYRIGHT 2004 by COMSOL AB. All right reserved.
No part of this documentation may be photocopied or reproduced in any form without prior written consent from COMSOL AB.
FEMLAB is a registered trademark of COMSOL AB. Other product or brand names are trademarks or registered trademarks of their respective holders.

COMSOL

Cooling Flange

In the chemical process industry, reaction fluids are often cooled using cooling flanges made of glass. The coolant is in most cases the surrounding air. An obvious design consideration for this type of device is the cooling power. The temperature of the surfaces may 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 may also influence on the performance. A convenient method to treat convection cooling is to use heat transfer coefficients, h. The coefficient describes the influence of the fluid flow field and the convective fluxes. Thus, the flow field is not necessary to model. This greatly simplifies the simulations.

Figure 0-1: Operation principle of the cooling flange.

Semi-empirical expressions for computing the heat transfer coefficient for different cases are available in the literature. In this model, the heat transfer coefficient for outer surface is obtained by using the 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 is built in the General Heat Transfer application mode.

Model Definitions

A schematic drawing of the modeled geometry is presented in Figure 3-13.

Figure 0-2: Schematic 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 also 4mm thick, and are 10 mm in height.

During operation the hot process fluid heats the inside of the tube. The heat is conducted within the flange and then transferred to the surrounding air. As the air is heated, buoyancy effects cause a convective flow to occur.

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 [$Wm^{-1}K^{-1}$], and T is the temperature [K]. On the external boundaries of the flange, which face the air and process fluid, the following boundary condition is set:

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

where $\mathbf{n}$ is the normal vector of the boundary, h is the heat transfer coefficient [$Wm^{-2}K^{-1}$], and T_{inf} is the temperature of the surrounding media [K].

The value for the heat transfer coefficient, h, on the process fluid side is approximated with a constant value of 15 $Wm^{-2}K^{-1}$, since the fluid velocity is close to constant and the temperature of the fluid is assumed to decrease only slightly.

The h-expression on the air side is more elaborate. We assume that the free convection process around the flange is similar to that around a cylinder. The heat transfer coefficient for a cylinder can be found in the literature, [1].

The heat transfer coefficient is defined by the following expression:

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

where k is the thermal conductivity of air, 0.06 $Wm^{-1}K^{-1}$, L is the characteristic length that in this case is the outer diameter of the flange, 44 mm. $f(\theta)$ is an empirical coefficient that is tabulated in Table 3-2. The definition of the angle θ is shown in Figure 3-14. Finally, Gr is the Grashof number, defined by:

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

where β is the thermal expansion coefficient [K^{-1}], which is $1/T_\infty$ for an ideal gas, g is the gravitational acceleration, 9.82 ms^{-2}, and μ is the kinematic viscosity, 18×10^{-6} Nsm^{-2}.

TABLE 0-1: 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 0-3: Definition of the angle θ.

The material properties of the flange material are summarized in Table 3-3.

TABLE 0-2: MATERIAL PROPERTIES

MATERIAL	k [$Wm^{-1}K^{-1}$]	ρ [kgm^{-3}]	C_p [$Jkg^{-1}K^{-1}$]
Silica glass	1.38	2203	703

Results and Discussion

Figure 3-16 shows the surface temperature of the flange at steady-state.

Figure 0-4: Stationary surface temperature of the flange.

The temperature at the shoulders of the flanges is approximately 12 K lower than the temperature at the tube surface. The temperature difference between the process fluid, which is 363 K, and the inner surface of the pipe is approximately 40 K, while that between the flange outer surface and the air stream is approximately 15 K. This indicates 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 performance of the device, it is a good idea to increase the diameter of the tube.

Figure 3-16 shows the heat transfer coefficient for the flange and pipe walls.

Figure 0-5: Heat transfer film coefficient, h, for the flange.

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

The total cooling power of the flange is calculated by integrating the heat flux on the outer surfaces. The whole flange, that is, taking both symmetry halves into account, has a cooling power of approximately 1.6 W.

Reference

[1] B. Sundén, *Kompendium i värmeöverföring*, Section 10-3, Department of Heat and Power Engineering, Lunds Institute of Technology, 2003.

Model Library path: `Heat_Transfer_Module/Process_and_Manufacturing/cooling_flange`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 On the **New** page, set **Space dimension** to **2D**, and click **OK.**

OPTIONS AND SETTINGS

1 Define the following constant expressions in **Constants** under the **Options** menu:

NAME	EXPRESSION
D	4.4e-2
K	0.06
Tinf	298
Tinner	363
Hh	15
visc	18e-6
beta	1/Tinf
grav	9.81

2 Click **OK**.

3 Select **Interpolation Functions** from the **Options** menu.

4 In the dialog that appears, click **New** and enter `graph` in the **Function name** edit field.

5 Select **Linear** from the **Interpolation function** list.

6 Click **OK**.

7 In the **Interpolation Functions** dialog box, enter the following values in the columns for **x** and **f(x)**:

x	f(x)
0	0.48
90	0.46
100	0.45
110	0.435
120	0.42
130	0.38

x	f(x)
140	0.35
150	0.28
160	0.22
180	0.15

8 Click **OK**.

9 Select **Axes/Grid Settings** from the **Options** menu.

10 Clear the **Axis equal** check box and enter the properties as shown in the table below:

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

11 On the **Grid** page, clear the **Auto** check box and enter the properties as shown in the table below:

PROPERTY	VALUE
x spacing	0.1
y spacing	0.1

Click **OK**.

GEOMETRY MODELING

We start by creating the 2D geometry shown below:

Create rectangle R1 by doing the following steps:

1 Press the **Shift** key and click the **Rectangle/Square** button in the Draw toolbar.

2 In the dialog box that appears, enter the rectangle properties given below.

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

3 Click **OK**.

Create the composite object CO1 by doing the following steps:

1 Click **2nd Degree Bezier Curve** in the Draw toolbar.

2 Click on the coordinates `(0,1.2)`, `(0.2,1.2)`, and `(0.2,1.4)`.

3 Click **Line** in the Draw toolbar and then click on the coordinate `(0.6,1.4)`.

4 Click **2nd Degree Bezier Curve** in the Draw toolbar.

5 Draw the Bezier 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 by doing the following steps:

1 Click **Line** in the Draw toolbar and 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 by doing the following steps:

1 Click **2nd Degree Bezier Curve** in 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 **Line** in the Draw toolbar and then click on the coordinates `(0.4,1.9)` and `(0.2,2.0)`.

4 Click the right mouse button. This creates a geometry object. To create CO4, we still need to add a square and some lines to the object.

5 Press the **Shift** key and click the **Rectangle/Square (Centered)** button in the Draw toolbar.

6 In the dialog box that appears, enter the rectangle properties given below.

OBJECT DIMENSIONS	EXPRESSION
Width	`0.1`
α	45
Base	`Center`
x-position	`0.4`
y-position	`2.05`

7 Click **OK**.

8 Click **Line** in the Draw toolbar and then click on the left corner of the small rectangle and then on the coordinate `(0.2,2.1)`.

9 Click **Line** in the Draw toolbar and then click on the right corner of the small rectangle and then on the coordinate `(0.6,2.1)`.

10 Click **Line** in the Draw toolbar and then click on the upper corner of the small rectangle and then on the coordinate `(0.4,2.2)`.

11 Click **Line** in the Draw toolbar and then click on the lower corner of the small rectangle and then on the coordinate `(0.4,1.9)`.

12 To create the composite object, select CO3, SQ1, B1, B2, B3 and B4 and then click **Coerce to Solid** in the Draw toolbar.

Create the other two flanges by doing the following steps:

1 Select R1, CO1, CO2, and CO4 and then click **Array** in the Draw toolbar.

2 In the dialog that appears, enter the following properties:

PROPERTY	VALUE
displacement - x	0.8
displacement - y	0
array size - x	3
array size - y	1

3 Click **OK**.

Create object CO10 by doing the following steps:

1 Click **Line** in the Draw toolbar.

2 Click on the lower left corner of rectangle R1 and then click on the coordinates `(-1.2,0.8)`, `(-1.2,1.1)`, and `(-0.8,1.1)`.

3 Click **3rd Degree Bezier Curve** in the Draw toolbar.

4 Click 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 according to the following steps:

1 Select all geometry objects with **Ctrl-A**.

2 Click **Scale** in the Draw toolbar.

3 In the dialog that appears, enter the following scaling properties:

PROPERTY	VALUE
Scale factor - x	1e-2
Scale factor - y	1e-2
Scale base point - x	0
Scale base point - y	0

4 Click **OK**.

5 Click the **Zoom Extents** button.

MESH

1 Select **Map Mesh** from the **Mesh** menu.

2 On the **Boundary** tab, select the boundaries that are marked in the figure below:

3 Select the **Constrained edge element distribution** check box and keep the default value for the **Number of edge elements**, which is 1.

4 Select the boundaries that are marked in the figure below:

5 Select the **Constrained edge element distribution** check box and enter 2 in the **Number of edge elements** edit field.

6 Click **Remesh** and then **OK**.

7 Select **Revolve Mesh** from the **Mesh** menu.

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

9 Select **Angle from x-axis** and enter 0 in the θ edit field.

10 Click **OK**.

PHYSICS SETTINGS

1 Select **Model Navigator** from the **Multiphysics** menu.

2 In the **Multiphysics** area, select **Geom2 (3D)**.

3 Select **Heat Transfer Module>General Heat Transfer>Steady-state analysis**.

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

Subdomain Settings

1 Open **Subdomain Settings** from the **Physics** menu.

2 Select all subdomains.

3 Select the material properties from the material library by pressing **Load**. In the dialog that appears, select **Silica Glass** from the **Materials** list. Click **OK**.

4 On the **Init** tab, enter 323 in the **Temperature** edit field.

5 Click **OK**.

Boundary Settings

1 Open **Boundary Settings** from the **Physics** menu.

1 Select all outer boundaries of the geometry with **Ctrl-A**.

2 Select the **Heat Flux** boundary condition and enter the following properties:

PROPERTY	VALUE
h	Hc
T_{inf}	Tinf

3 Click **Apply**.

4 Select the boundaries on the ends of the geometry and the symmetry boundaries, according to the figure below:

5 Select the **Thermal insulation** boundary condition for these boundaries.

6 Click **Apply**.

7 Select the boundaries that face the inner channel, according to the figure below:

8 Select the **Heat flux** boundary condition and enter the following properties:

PROPERTY	VALUE
h	`Hh`
T_{inf}	`Tinner`

9 Click **OK**.

10 From the **Options** menu, select **Expressions** and then **Boundary Expressions**.

11 In the dialog that appears, select all the outer boundaries of the geometry that face the outside, according to the figure below. One way to select all outer boundaries is to also open the **Boundary Settings** dialog box and then select the **Select by group** check box in that dialog box. When the boundaries are selected, return to the **Boundary Expressions** dialog box.

12 Enter the following expressions for the selected boundaries:

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

13 Click **OK**.

SOLVING THE MODEL

1. Open the **Solver Parameters** dialog box from the **Solve** menu.
2. Select the **Stationary nonlinear** solver from the **Solver** list.
3. Select **GMRES** from the **Linear system solver** list.
4. Select **General** from the **Solution form** list.
5. Click **OK**.
6. Click **Solve** in the main toolbar.

POSTPROCESSING AND VISUALIZATION

1. To generate Figure 3-15 open the **Plot Parameters** dialog box from the **Postprocessing** menu.
2. On the **General** page clear the **Slice** check box and select the **Boundary** check box.
3. On the **Boundary** page select **Temperature (htgh)** from the **Predefined quantities** list. Click **Apply**.
4. Figure 3-16 is generated by changing the plotted boundary variable to **Heat transfer coefficient**, by selecting it from the **Predefined quantities** list of the **Boundary** page, and clicking **OK**.
5. To integrate the heat flux over the outer surface area of the flange open the **Boundary Integration** dialog box from the **Postprocessing** menu.
6. One way to select all outer boundaries is to also open the **Boundary Settings** dialog box and then select the **Select by group** check box.
7. Select boundary 3 form the list. This automatically selects all outer boundaries.
8. Go back to the **Boundary Integration** dialog box, and select **Normal total heat flux** from the **Predefined quantities** list. Click **OK**.

The integrated value is displayed in the message log in the lower part of the graphical interface.