

Shell-and-Tube Heat Exchanger

Introduction

Shell and tube heat exchangers are one of the most widely used type of heat exchanger in the processing industries (65% of the market according to [Ref. 1](#)) and are commonly found in oil refineries, nuclear power plants, and other large-scale chemical processes. Additionally, they can be found in many engines and are used to cool hydraulic fluid and oil. In this model, two separated fluids at different temperatures flow through the heat exchanger: one through the tubes (tube side) and the other through the shell around the tubes (shell side). Several design parameters and operating conditions influence the optimal performance of a shell-and-tube heat exchanger.

The main purpose of this model is to show the basic principles for setting up a heat exchanger model. It can also serve as a starting point for more sophisticated applications, such as parameter studies or adding additional effects like corrosion, thermal stress, and vibration.

Model Definition

The concept used to design a shell and tube heat exchanger is examined by exploring the working model of a straight, cross-flow, one pass shell and tube heat exchanger.

The geometry of such a model is shown in [Figure 1](#).

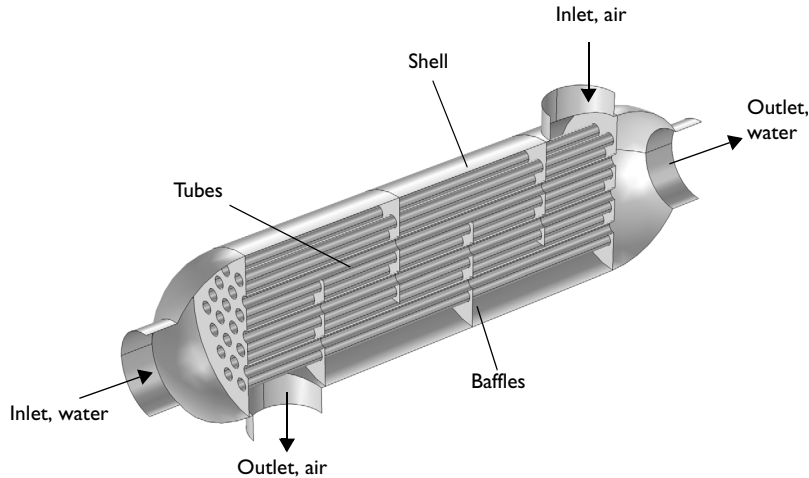


Figure 1: Geometry of the shell-and-tube heat exchanger.

The heat exchanger is made of structural steel. In this example, two fluids are passed through the heat exchanger. The first fluid, in this case water, is passed through the tubes, while the second fluid, air, circulates within the shell of the heat exchanger but outside of the tubes. Both of these fluids have different starting temperatures when entering the heat exchanger, however after circulating within the shell and tubes, the fluids are brought closer to an equilibrium temperature. The baffles introduce some cross-flow to the air and such increasing the area of heat exchange. Another advantage is that baffles reduce vibration due to the fluid motion.

This model uses the Non-Isothermal Flow physics interface together with the k - ϵ turbulence model. It takes advantage of symmetries to model only one half of the heat exchanger, thereby reducing model size and computational costs.

BOUNDARY CONDITIONS

All heat exchanger walls including the baffles are modeled as shells in 3D. This requires special boundary conditions for the flow and heat transport equations.

The interior wall boundary condition for the flow separates the fluids from each other and is also used to describe the baffles. On both sides, it applies the wall functions needed for simulating walls with the k - ϵ turbulence model.

To account for the in-plane heat flux in the shell, the highly conductive layer boundary condition is applied:

$$-\mathbf{n} \cdot (-k\nabla T) = -\nabla_t \cdot (-d_s k_s \nabla_t T)$$

Here, ∇_t is the tangential derivative, d_s is the layer thickness, and k_s is the thermal conductivity of the shell.

Water enters the tube side with a velocity of 0.1 m/s and a temperature of 80 °C. Air enters the shell side with a velocity of 1 m/s and a temperature of 5 °C. At both inlets, the recommended values for the turbulence length scale is $0.07 L$, where L is the channel radius (see [Inlet Values for the Turbulence Length Scale and Turbulent Intensity](#)). For the water inlet, this radius is equal to 5 cm and for the air inlet, it is 4.5 cm.

Beside the symmetry plane, all remaining exterior boundaries are thermally insulated walls.

Results and Discussion

An important criterion for estimating the accuracy of a turbulence model is the wall resolution. Hence, COMSOL Multiphysics creates a plot of the wall resolution by default. The value for δ_w^+ has to be 11.06, which corresponds to the distance from the wall where the logarithmic layer meets the viscous sublayer. Furthermore, the wall lift-off δ_w has to be small compared to the dimension of the geometry. On interior walls you have δ_w for the upside and downside of the wall. To visualize the upside and downside directions, use an arrow surface plot with the components unx, uny, and unz for the up direction and dnx, dny, and dnz for the down direction.

Figure 2 shows the upside wall lift-off. This is the wall lift-off inside the tubes where the probably most critical area in terms of mesh resolution is located. It is about 10% of the tube radius, which is sufficient.

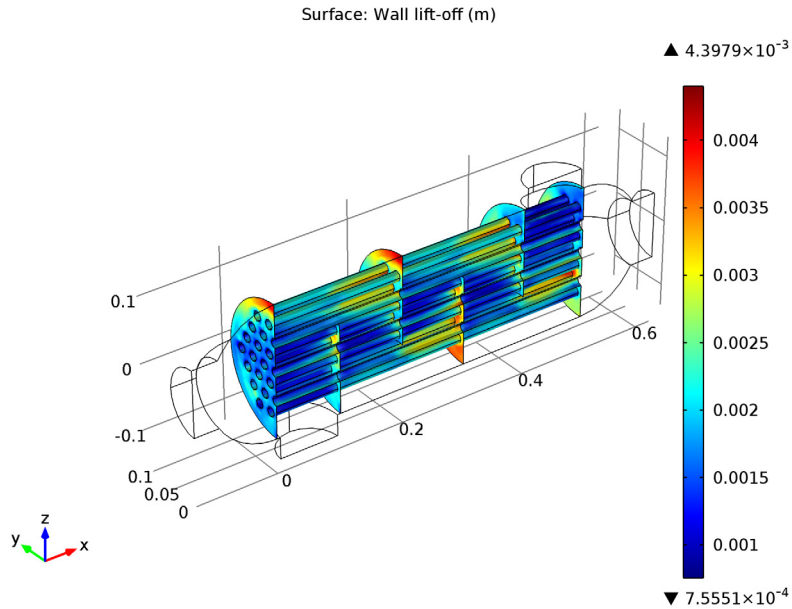


Figure 2: Wall lift-off for the tubes.

The tube side velocity shows a uniform distribution in the tubes. Before water enters the tubes, recirculation zones are present. The streamline colors represent the

temperature and you can see that the temperatures at both outlets are close to each other.

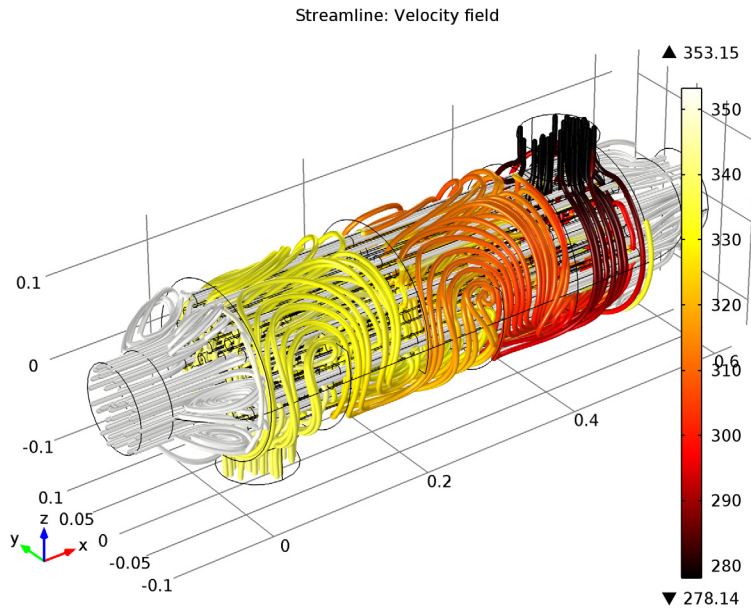


Figure 3: Velocity streamlines.

Figure 4 shows the temperature distribution on the heat exchanger walls.

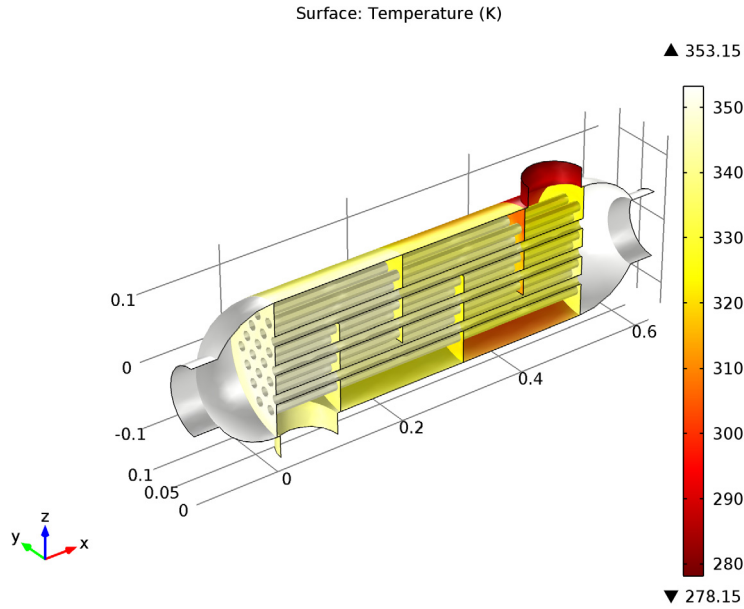


Figure 4: Temperature on the heat exchanger walls.

There are several quantities that describe the characteristics and effectiveness of a heat exchanger. One is the equivalent heat transfer coefficient given by

$$h_{\text{eq}} = \frac{P}{A(T_{\text{hot}} - T_{\text{cold}})} \quad (1)$$

where P is the total exchanged power and A is the surface area through which P flows. In this model the value of P is $5.3 \text{ W}/(\text{m}^2 \cdot \text{K})$.

The pressure drop is about 33 Pa on the tube side and 12 Pa on the shell side.

Notes About the COMSOL Implementation

Solve the model using a physics controlled mesh. For flow applications this means that COMSOL automatically generates a mesh sequence where the mesh size depends on whether the flow is laminar or turbulent and where a boundary layer mesh is applied to all no-slip walls. Even if the coarsest mesh size is used, the mesh is still fine enough to resolve the flow pattern and thus the temperature distribution well. Nevertheless,

this model requires about 8 GB RAM. Alternatively, you can set up a coarser mesh manually, but keep in mind that this can lead to lower accuracy.

The first part of the modeling process is the preprocessing. This includes defining parameters, preparing the geometry, and defining relevant selections. You can skip this part by loading the file `shell_and_tube_heat_exchanger_geom.mph`. However, we recommend that you have a look at these steps at least once. Especially when developing models intended for optimization and sophisticated analyses, these steps can significantly simplify and accelerate the modeling process.

Defining parameters beforehand enables setting up a parametric study immediately, also for multiple parameter sets. In addition, this provides a fast overview of the operating conditions. In the [Modeling Instructions](#), several selections are also created. Once defined, they are available in every step of the modeling process. If you want to change from cocurrent to countercurrent heat exchanger, you only need to redefine the selections.

Model Library path: Heat_Transfer_Module/Heat_Exchangers/
shell_and_tube_heat_exchanger

Reference

1. H. S. Lee, *Thermal Design*, John Wiley & Sons, 2010.

Modeling Instructions

The file `shell_and_tube_heat_exchanger_geom.mph` contains a parameterized geometry and prepared selections for the model. Start by loading this file.

- 1 From the **File** menu, choose **Open**.
- 2 Browse the model's Model Library folder and double-click the file `shell_and_tube_heat_exchanger_geom.mph`.

MATERIALS

On the **Home** toolbar, click **Add Material**.

ADD MATERIAL

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

- 2 In the tree, select **Built-In>Air**.
- 3 In the **Add material** window, click **Add to Component**.

MATERIALS

On the **Home** toolbar, click **More Windows** and choose **Add Material**.

ADD MATERIAL

- 1 Go to the **Add Material** window.
- 2 In the tree, select **Built-In>Water, liquid**.
- 3 In the **Add material** window, click **Add to Component**.

MATERIALS

Water, liquid

- 1 In the **Model Builder** window, under **Component 1>Materials** click **Water, liquid**.
- 2 In the **Material** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Selection** list, choose **Water domain**.

The heat exchanger itself is made of structural steel. Apply this material to the walls of the heat exchanger.

- 4 On the **Home** toolbar, click **More Windows** and choose **Add Material**.

ADD MATERIAL

- 1 Go to the **Add Material** window.
- 2 In the tree, select **Built-In>Structural steel**.
- 3 In the **Add material** window, click **Add to Component**.

MATERIALS

Structural steel

- 1 In the **Model Builder** window, under **Component 1>Materials** click **Structural steel**.
- 2 In the **Material** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 From the **Selection** list, choose **Walls**.

NON-ISOTHERMAL FLOW

The next step is to set the boundary conditions. At first the boundary conditions for the flow equations are applied. These are the inlet and outlet boundary conditions,

symmetry as well as the interior walls which separate the air from the water domain. The default wall boundary condition then applies to the outer boundaries.

Inlet 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
- 2 In the **Inlet** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Inlet water**.
- 4 In the L_T edit field, type $0.07*5$ [cm].
This is an estimation for the turbulence length scale as explained in the [Model Definition](#) section.
- 5 Locate the **Velocity** section. In the U_0 edit field, type u_water .
- 6 Right-click **Component 1>Non-Isothermal Flow>Inlet 1** and choose **Rename**.
- 7 Go to the **Rename Inlet** dialog box and type **Inlet water** in the **New name** edit field.
- 8 Click **OK**.

Outlet 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
- 2 In the **Outlet** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Outlet water**.
- 4 Locate the **Pressure Conditions** section. Select the **Normal flow** check box.
- 5 Right-click **Component 1>Non-Isothermal Flow>Outlet 1** and choose **Rename**.
- 6 Go to the **Rename Outlet** dialog box and type **Outlet water** in the **New name** edit field.
- 7 Click **OK**.

Inlet 2

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
- 2 In the **Inlet** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Inlet air**.
- 4 In the L_T edit field, type $0.07*4.5$ [cm].
This is an estimation for the turbulence length scale as explained in the [Model Definition](#) section.
- 5 Locate the **Velocity** section. In the U_0 edit field, type u_air .
- 6 Right-click **Component 1>Non-Isothermal Flow>Inlet 2** and choose **Rename**.
- 7 Go to the **Rename Inlet** dialog box and type **Inlet air** in the **New name** edit field.

8 Click **OK**.

Outlet 2

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
- 2 In the **Outlet** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Outlet air**.
- 4 Locate the **Pressure Conditions** section. Select the **Normal flow** check box.
- 5 Right-click **Component 1>Non-Isothermal Flow>Outlet 2** and choose **Rename**.
- 6 Go to the **Rename Outlet** dialog box and type `Outlet air` in the **New name** edit field.
- 7 Click **OK**.

Symmetry, Flow 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry, Flow**.
- 2 In the **Symmetry, Flow** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Symmetry**.

Interior Wall 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Interior Wall**.
- 2 In the **Interior Wall** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Walls**.

The boundary conditions that set up the heat transfer equation are the temperatures at the inlets and the outflow at the outlets, the symmetry and for all walls the highly conductive layer feature accounts for the heat conduction through the shell.

Temperature 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.
- 2 In the **Temperature** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Inlet water**.
- 4 Locate the **Temperature** section. In the T_0 edit field, type `T_water`.
- 5 Right-click **Component 1>Non-Isothermal Flow>Temperature 1** and choose **Rename**.
- 6 Go to the **Rename Temperature** dialog box and type `Temperature water` in the **New name** edit field.
- 7 Click **OK**.

Outflow 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Outflow**.

- 2 In the **Outflow** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Outlet water**.
- 4 Right-click **Component 1>Non-Isothermal Flow>Outflow 1** and choose **Rename**.
- 5 Go to the **Rename Outflow** dialog box and type `Outflow water` in the **New name** edit field.
- 6 Click **OK**.

Temperature 2

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.
- 2 In the **Temperature** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Inlet air**.
- 4 Locate the **Temperature** section. In the T_0 edit field, type `T_air`.
- 5 Right-click **Component 1>Non-Isothermal Flow>Temperature 2** and choose **Rename**.
- 6 Go to the **Rename Temperature** dialog box and type `Temperature air` in the **New name** edit field.
- 7 Click **OK**.

Outflow 2

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Outflow**.
- 2 In the **Outflow** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Outlet air**.
- 4 Right-click **Component 1>Non-Isothermal Flow>Outflow 2** and choose **Rename**.
- 5 Go to the **Rename Outflow** dialog box and type `Outflow air` in the **New name** edit field.
- 6 Click **OK**.

Symmetry, Heat 2

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry, Heat**.
- 2 In the **Symmetry, Heat** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Symmetry**.

Highly Conductive Layer 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Highly Conductive Layer**.
- 2 In the **Highly Conductive Layer** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Walls**.
- 4 Locate the **Model Inputs** section. In the d_s edit field, type `0.005[m]`.

The physics is now defined. For evaluating the equivalent heat transfer coefficient according to [Equation 1](#) directly after solving the model, you need to define the following component coupling operator. It can also be defined and evaluated after computing the simulation, but in this case you need to choose **Update Solution** (after right-clicking on the **Study 1** node) to make this operator available without running the model again.

DEFINITIONS

Average 1

- 1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.
- 2 In the **Average** settings window, locate the **Source Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 From the **Selection** list, choose **Water-air walls**.
- 5 Right-click **Component 1>Definitions>Average 1** and choose **Rename**.
- 6 Go to the **Rename Average** dialog box and type **Average operator on water-air walls** in the **New name** edit field.
- 7 Click **OK**.

The heat exchanger properties and operating conditions are well defined and the model is ready to solve. For a first estimation of the heat exchanger performance a coarse mesh is satisfying. The solution is obtained very fast and provides qualitatively good results. Reliable quantitative results require a good resolution especially of the wall regions.

The **physics-controlled mesh** is a good starting point and can be customized to reduce computational costs. Here you reduce the number of boundary layers and scale the geometry for the meshing sequence in x -direction. This results in an anisotropic mesh that is suitable for the minor changes of the flow field and temperature field in x -direction.

MESH 1

- 1 In the **Mesh** settings window, locate the **Mesh Settings** section.
- 2 From the **Element size** list, choose **Extremely coarse**.
- 3 Right-click **Component 1>Mesh 1** and choose **Edit Physics-Induced Sequence**.

Free Tetrahedral 1

- 1 In the **Model Builder** window, under **Component 1>Mesh 1** click **Free Tetrahedral 1**.
- 2 In the **Free Tetrahedral** settings window, click to expand the **Scale geometry** section.

- 3 Locate the **Scale Geometry** section. In the **x-direction scale** edit field, type 0.5.

Boundary Layer Properties 1

- 1 In the **Model Builder** window, expand the **Component 1>Mesh 1>Boundary Layers 1** node, then click **Boundary Layer Properties 1**.
- 2 In the **Boundary Layer Properties** settings window, locate the **Boundary Layer Properties** section.
- 3 In the **Number of boundary layers** edit field, type 3.
- 4 Click the **Build All** button.

STUDY 1

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

RESULTS

Velocity (nitf)

A slice plot of the velocity field, a surface plot of the wall resolution and a surface plot of the temperature are generated by default. You can either customize these plots or create new plot groups to visualize the results.

The wall resolution indicates the accuracy of the flow close to the walls where the wall functions are applied. The variable `nitf.d_w_plus` should be 11.06 and the wall-lift off `nitf.delta_w` needs to be significantly smaller than the dimension of the geometry. On interior boundaries, these variables are available for the upside and downside of the wall indicated by `nitf.d_w_plus_u/d` or `nitf.delta_w_u/d`, respectively. The critical regions in terms of the wall resolution are in the tubes.

Wall Resolution (nitf)

- 1 In the **Model Builder** window, expand the **Results>Wall Resolution (nitf)** node, then click **Surface 1**.
- 2 In the **Surface** settings window, locate the **Expression** section.
- 3 In the **Expression** edit field, type `nitf.delta_w_u`.
- 4 On the **3D plot group** toolbar, click **Plot**.

Figure 4 shows the temperature distribution on all wall boundaries. The default temperature plot uses the **Surface** data set created automatically and contains exterior walls only. It is easy to change it by using the selection created at the beginning.

Data Sets

- 1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Surface 1**.
- 2 In the **Surface** settings window, locate the **Selection** section.
- 3 From the **Selection** list, choose **Walls**.

Temperature (nitf)

- 1 In the **Model Builder** window, expand the **Results>Temperature (nitf)** node, then click **Surface 1**.
- 2 In the **Surface** settings window, locate the **Expression** section.
- 3 From the **Unit** list, choose **degC**.

In [Figure 3](#), the streamlines are plotted for the full 3D geometry. Even if only one half of the heat exchanger is modeled the solution can be mirrored to obtain a full 3D view of the results. To do so, follow the steps below:

Data Sets

- 1 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 3D**.
- 2 In the **Mirror 3D** settings window, locate the **Plane Data** section.
- 3 From the **Plane** list, choose **zx-planes**.

3D Plot Group 4

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
- 2 In the **3D Plot Group** settings window, locate the **Data** section.
- 3 From the **Data set** list, choose **Mirror 3D 1**.
- 4 Right-click **Results>3D Plot Group 4** and choose **Streamline**.
- 5 In the **Streamline** settings window, locate the **Streamline Positioning** section.
- 6 In the **Points** edit field, type 100.
- 7 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.
- 8 Right-click **Results>3D Plot Group 4>Streamline 1** and choose **Color Expression**.
- 9 In the **Color Expression** settings window, locate the **Coloring and Style** section.
- 10 From the **Color table** list, choose **Thermal**.
- 11 In the **Model Builder** window, right-click **3D Plot Group 4** and choose **Rename**.
- 12 Go to the **Rename 3D Plot Group** dialog box and type **Velocity, streamlines** in the **New name** edit field.
- 13 Click **OK**.

Evaluate the equivalent heat transfer coefficient by using the component coupling operators defined previously.

Derived Values

- 1 On the **Results** toolbar, click **Global Evaluation**.
- 2 In the **Global Evaluation** settings window, locate the **Expression** section.
- 3 In the **Expression** edit field, type $\text{aveop1}(\text{nitf.qwf_u}) / (\text{T_water} - \text{T_air})$.
The variable `nitf.qwf_u` efficiently computes the heat flux through the wall separating water and air domains, based on the wall function definition.
- 4 Click the **Evaluate** button.

TABLE

- 1 Right-click **Results>Derived Values>Global Evaluation 1** and choose **Rename**.
- 2 Go to the **Rename Global Evaluation** dialog box and type Heat Transfer Coefficient in the **New name** edit field.
- 3 Click **OK**.

To evaluate the pressure drop the average inlet pressures are evaluated.

RESULTS

Derived Values

- 1 On the **Results** toolbar, click **More Derived Values** and choose **Average>Surface Average**.
- 2 In the **Surface Average** settings window, locate the **Selection** section.
- 3 From the **Selection** list, choose **Inlet water**.
- 4 Locate the **Expression** section. In the **Expression** edit field, type `p`.
- 5 Click the **Evaluate** button.

TABLE

- 1 Right-click **Results>Derived Values>Surface Average 1** and choose **Rename**.
- 2 Go to the **Rename Surface Average** dialog box and type Inlet Pressure, Water in the **New name** edit field.
- 3 Click **OK**.

RESULTS

Derived Values

- 1 On the **Results** toolbar, click **More Derived Values** and choose **Average>Surface Average**.
- 2 In the **Surface Average** settings window, locate the **Selection** section.
- 3 From the **Selection** list, choose **Inlet air**.
- 4 Locate the **Expression** section. In the **Expression** edit field, type p .
- 5 Click the **Evaluate** button.

TABLE

- 1 Right-click **Results>Derived Values>Surface Average 2** and choose **Rename**.
- 2 Go to the **Rename Surface Average** dialog box and type **Inlet Pressure, Air** in the **New name** edit field.
- 3 Click **OK**.

The tube side pressure drop is 35 Pa and the shell side pressure drop is 12 Pa.