Shell-and-Tube Heat Exchanger

Introduction

Shell-and-tube heat exchangers are commonly used in oil refineries and other large-scale chemical processes. 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

Figure 1 shows the shell-and-tube heat exchanger geometry.

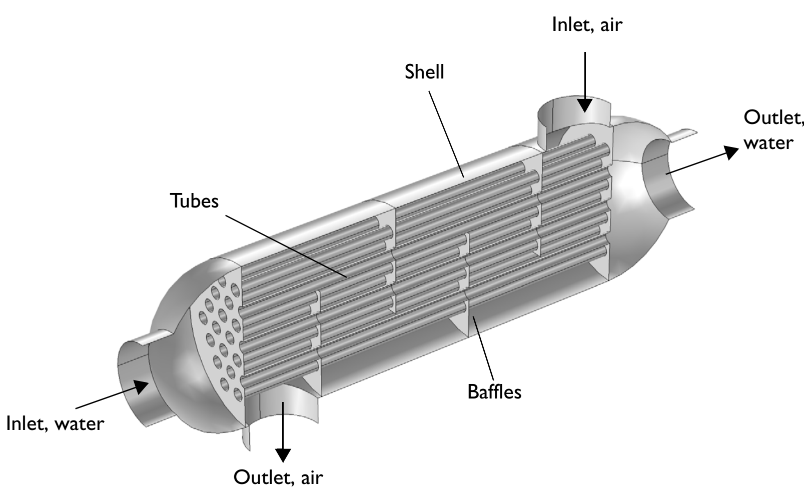

Figure 1: Geometry of the shell-and-tube heat exchanger.

The heat exchanger is made of structural steel. The participating fluids are water flowing through the tube side and air flowing through the shell side. 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 user 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.

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 δw 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.

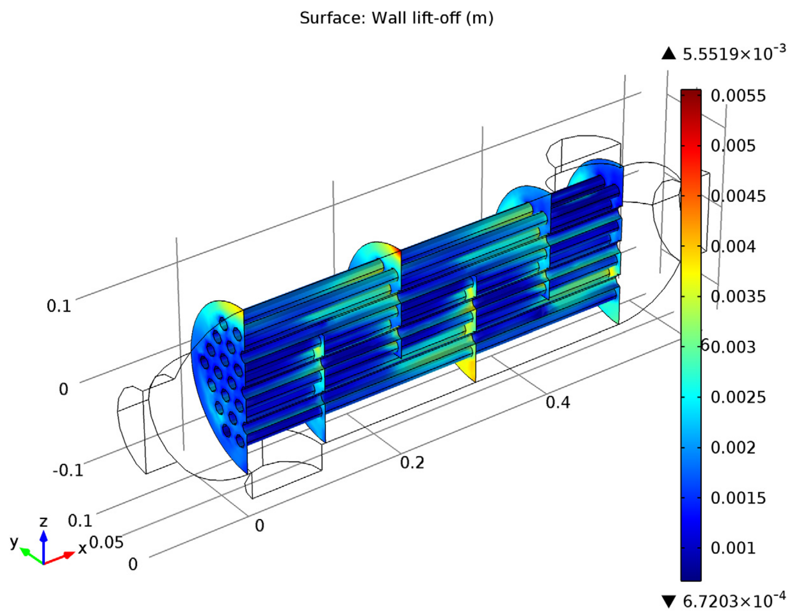

Figure 2: Wall lift-off for the tubes.

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.

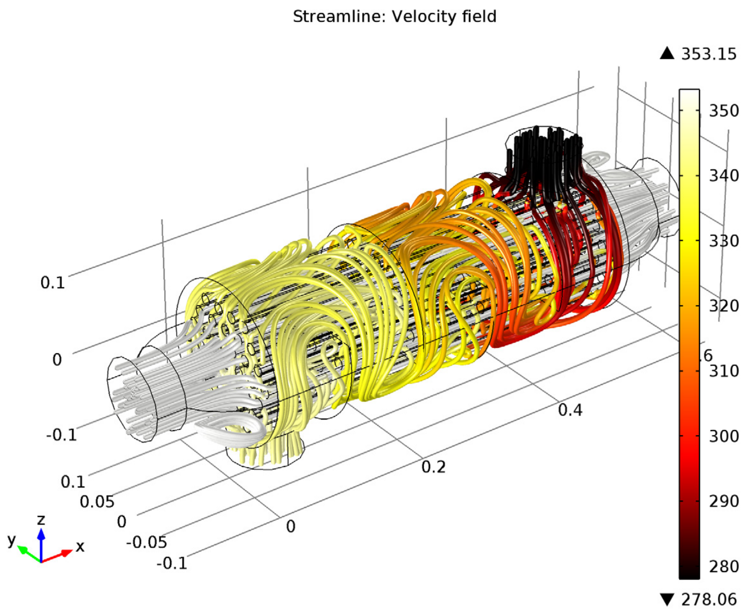

Figure 3: Velocity streamlines.

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

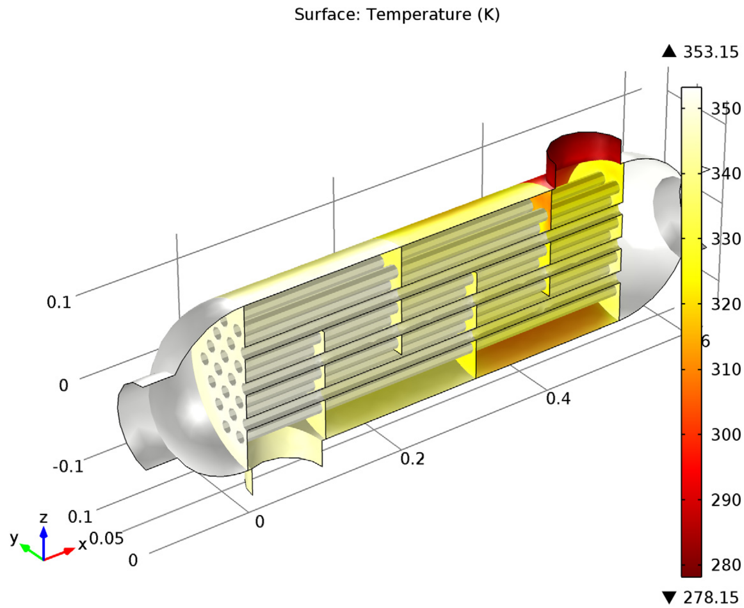

Figure 4: Temperature on the heat exchanger walls.

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

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

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 391.8 $\mathrm{W/(m^2 \cdot K)}$.

The pressure drop is 35 Pa on the tube side and 12.4 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 20 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`

Modeling Instructions

The first instructions create parameters and selections needed to set up the model. To skip this part, load the file `shell_and_tube_heat_exchanger_geom.mph` and continue the instructions from the Definitions section.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **Fluid Flow>Non-Isothermal Flow>Turbulent Flow>Turbulent Flow, k-ε (nitf)**.

4 Click **Next**.

5 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

6 Click **Finish**.

GLOBAL DEFINITIONS

At first the parameters for the inlet velocities and temperatures are defined.

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Description
`u_water`	`0.1[m/s]`	`Inlet velocity water`
`u_air`	`1[m/s]`	`Inlet velocity air`
`T_water`	`80[degC]`	`Inlet temperature water`
`T_air`	`5[degC]`	`Inlet temperature air`

GEOMETRY 1

The geometry is imported as an MPHBIN-file, which contains some faces that have no physical relevance. The Virtual Operations features provide appropriate tools to remove them before defining the physics.

Import 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Import**.

2 In the **Import** settings window, locate the **Import** section.

3 Click the **Browse** button.

4 Browse to the model's Model Library folder and double-click the file `shell_and_tube_heat_exchanger_geom.mphbin`.

5 Click the **Import** button.

6 Click the **Wireframe Rendering** button on the Graphics toolbar.

Ignore Faces 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Virtual Operations>Ignore Faces**.

2 On the object **fin**, select Boundaries 15, 21, 107, 111, 113, 188, 196, 202, 285, 289, 291, 366, 374, and 380 only.

For more convenience, you can select the boundaries by clicking on the **Paste Selection** button and pasting the face numbers.

3 Click the **Build All** button.

It is useful to define selections that represent domains or boundaries having the same physical properties. The first selections are for the two domains with the different fluids.

DEFINITIONS

Explicit 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Explicit**.
2 Select Domain 1 only.
3 Right-click **Model 1>Definitions>Explicit 1** and choose **Rename**.
4 Go to the **Rename Explicit** dialog box and type `Water domain` in the **New name** edit field.
5 Click **OK**.

Explicit 2a

1 Right-click **Definitions** and choose **Selections>Explicit**.
2 Select Domain 2 only.
3 Right-click **Model 1>Definitions>Explicit 2a** and choose **Rename**.
4 Go to the **Rename Explicit** dialog box and type `Air domain` in the **New name** edit field.
5 Click **OK**.

These selections are used as a starting point for creating the selections for the boundaries.

Water domain 1

1 In the **Model Builder** window, under **Model 1>Definitions** right-click **Water domain** and choose **Duplicate**.
2 In the **Explicit** settings window, locate the **Output Entities** section.
3 From the **Output entities** list, choose **Adjacent boundaries**.
4 Right-click **Model 1>Definitions>Water domain 1** and choose **Rename**.
5 Go to the **Rename Explicit** dialog box and type `Water domain exterior boundaries` in the **New name** edit field.
6 Click **OK**.

Adjacent 1

1 Right-click **Definitions** and choose **Selections>Adjacent**.

2 In the **Adjacent** settings window, locate the **Input Entities** section.

3 Under **Input selections**, click **Add**.

4 Go to the **Add** dialog box.

5 In the **Input selections** list, select **Air domain**.

6 Click the **OK** button.

7 Right-click **Model 1>Definitions>Adjacent 1** and choose **Rename**.

8 Go to the **Rename Adjacent** dialog box and type `Air domain exterior boundaries` in the **New name** edit field.

9 Click **OK**.

It is now easy to set up a selection for the baffles.

Air domain exterior boundaries 1

1 Right-click **Model 1>Definitions>Adjacent 1** and choose **Duplicate**.

2 In the **Adjacent** settings window, locate the **Output Entities** section.

3 Clear the **Exterior boundaries** check box.

4 Select the **Interior boundaries** check box.

5 Right-click **Model 1>Definitions>Air domain exterior boundaries 1** and choose **Rename**.

6 Go to the **Rename Adjacent** dialog box and type `Baffles` in the **New name** edit field.

7 Click **OK**.

For the symmetry boundaries follow these steps:

Explicit 4

1 Right-click **Definitions** and choose **Selections>Explicit**.

2 In the **Explicit** settings window, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 2 only.

5 In the **Explicit** settings window, locate the **Input Entities** section.

6 Select the **Group by continuous tangent** check box.

7 Right-click **Model 1>Definitions>Explicit 4** and choose **Rename**.

8 Go to the **Rename Explicit** dialog box and type `Symmetry` in the **New name** edit field.

9 Click **OK**.

The next selections are the inlet and outlet boundaries for water and air.

Explicit 5

1 Right-click **Definitions** and choose **Selections>Explicit**.
2 In the **Explicit** settings window, locate the **Input Entities** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 1 only.
5 Right-click **Model 1>Definitions>Explicit 5** and choose **Rename**.
6 Go to the **Rename Explicit** dialog box and type `Inlet water` in the **New name** edit field.
7 Click **OK**.

Explicit 6

1 Right-click **Definitions** and choose **Selections>Explicit**.
2 In the **Explicit** settings window, locate the **Input Entities** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 339 only.
5 Right-click **Model 1>Definitions>Explicit 6** and choose **Rename**.
6 Go to the **Rename Explicit** dialog box and type `Outlet water` in the **New name** edit field.
7 Click **OK**.

Explicit 7

1 Right-click **Definitions** and choose **Selections>Explicit**.
2 In the **Explicit** settings window, locate the **Input Entities** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 332 only.
5 Right-click **Model 1>Definitions>Explicit 7** and choose **Rename**.
6 Go to the **Rename Explicit** dialog box and type `Inlet air` in the **New name** edit field.
7 Click **OK**.

Explicit 8

1 Right-click **Definitions** and choose **Selections>Explicit**.
2 In the **Explicit** settings window, locate the **Input Entities** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 89 only.
5 Right-click **Model 1>Definitions>Explicit 8** and choose **Rename**.

6 Go to the **Rename Explicit** dialog box and type `Outlet air` in the **New name** edit field.

7 Click **OK**.

Boolean operations of the already defined selections create the next ones.

Difference 1

1 Right-click **Definitions** and choose **Selections>Difference**.

2 In the **Difference** settings window, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 Go to the **Add** dialog box.

6 In the **Selections to add** list, select **Water domain exterior boundaries**.

7 Click the **OK** button.

8 In the **Difference** settings window, locate the **Input Entities** section.

9 Under **Selections to subtract**, click **Add**.

10 Go to the **Add** dialog box.

11 In the **Selections to subtract** list, choose **Symmetry**, **Inlet water**, and **Outlet water**.

12 Click the **OK** button.

13 Right-click **Model 1>Definitions>Difference 1** and choose **Rename**.

14 Go to the **Rename Difference** dialog box and type `Water domain walls` in the **New name** edit field.

15 Click **OK**.

Difference 2

1 Right-click **Definitions** and choose **Selections>Difference**.

2 In the **Difference** settings window, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 Go to the **Add** dialog box.

6 In the **Selections to add** list, choose **Air domain exterior boundaries** and **Baffles**.

7 Click the **OK** button.

8 In the **Difference** settings window, locate the **Input Entities** section.

9 Under **Selections to subtract**, click **Add**.

10 Go to the **Add** dialog box.

11 In the **Selections to subtract** list, choose **Symmetry**, **Inlet air**, and **Outlet air**.

12 Click the **OK** button.

13 Right-click **Model 1>Definitions>Difference 2** and choose **Rename**.

14 Go to the **Rename Difference** dialog box and type `Air domain walls` in the **New name** edit field.

15 Click **OK**.

Union 1

1 Right-click **Definitions** and choose **Selections>Union**.

2 In the **Union** settings window, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 Go to the **Add** dialog box.

6 In the **Selections to add** list, choose **Water domain walls** and **Air domain walls**.

7 Click the **OK** button.

8 Right-click **Model 1>Definitions>Union 1** and choose **Rename**.

9 Go to the **Rename Union** dialog box and type `Walls` in the **New name** edit field.

10 Click **OK**.

Intersection 1

1 Right-click **Definitions** and choose **Selections>Intersection**.

2 In the **Intersection** settings window, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to intersect**, click **Add**.

5 Go to the **Add** dialog box.

6 In the **Selections to intersect** list, choose **Water domain walls** and **Air domain walls**.

7 Click the **OK** button.

8 Right-click **Model 1>Definitions>Intersection 1** and choose **Rename**.

9 Go to the **Rename Intersection** dialog box and type `Water-air walls` in the **New name** edit field.

10 Click **OK**.

The preprocessing part ends here. If you loaded the file `shell_and_tube_heat_exchanger_geom.mph`, continue the step-by-step instructions from here.

MATERIALS

Material Browser

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** settings window, In the tree, select **Built-In>Air**.

3 In the **Material_browser** window, click **Add Material to Model**.

Material Browser

1 In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** settings window, In the tree, select **Built-In>Water, liquid**.

3 Click **Add Material to Model**.

Water, liquid

1 In the **Model Builder** window, under **Model 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.

Material Browser

1 In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** settings window, In the tree, select **Built-In>Structural steel**.

3 Click **Add Material to Model**.

Structural steel

1 In the **Model Builder** window, under **Model 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. Start with the boundary conditions for the flow equations. 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 In the **Model Builder** window, under **Model 1** right-click **Non-Isothermal Flow** and choose the boundary condition **Turbulent Flow, k-ε>Inlet**.
2 In the **Inlet** settings window, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Inlet water**.
4 Locate the **Velocity** section. In the U_0 edit field, type `u_water`.
5 Right-click **Model 1>Non-Isothermal Flow>Inlet 1** and choose **Rename**.
6 Go to the **Rename Inlet** dialog box and type `Inlet water` in the **New name** edit field.
7 Click **OK**.

Outlet 1

1 Right-click **Non-Isothermal Flow** and choose the boundary condition **Turbulent Flow, k-ε>Outlet**.
2 In the **Outlet** settings window, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Outlet water**.
4 Locate the **Boundary Condition** section. From the **Boundary condition** list, choose **Normal stress**.
5 Right-click **Model 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 Right-click **Non-Isothermal Flow** and choose the boundary condition **Turbulent Flow, k-ε>Inlet**.
2 In the **Inlet** settings window, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Inlet air**.
4 Locate the **Velocity** section. In the U_0 edit field, type `u_air`.
5 Right-click **Model 1>Non-Isothermal Flow>Inlet 2** and choose **Rename**.
6 Go to the **Rename Inlet** dialog box and type `Inlet air` in the **New name** edit field.
7 Click **OK**.

Outlet 2

1 Right-click **Non-Isothermal Flow** and choose the boundary condition **Turbulent Flow, k-ε>Outlet**.
2 In the **Outlet** settings window, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Outlet air**.
4 Locate the **Boundary Condition** section. From the **Boundary condition** list, choose **Normal stress**.
5 Right-click **Model 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 Right-click **Non-Isothermal Flow** and choose the boundary condition **Turbulent Flow, k-ε>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 Right-click **Non-Isothermal Flow** and choose the boundary condition **Turbulent Flow, k-ε>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 Right-click **Non-Isothermal Flow** and choose the boundary condition **Heat Transfer>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 **Model 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 Right-click **Non-Isothermal Flow** and choose the boundary condition **Heat Transfer>Outflow**.

2 In the **Outflow** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outlet water**.

4 Right-click **Model 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 Right-click **Non-Isothermal Flow** and choose the boundary condition **Heat Transfer>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 **Model 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 Right-click **Non-Isothermal Flow** and choose the boundary condition **Heat Transfer>Outflow**.

2 In the **Outflow** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outlet air**.

4 Right-click **Model 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 Right-click **Non-Isothermal Flow** and choose the boundary condition **Heat Transfer>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 Right-click **Non-Isothermal Flow** and choose the boundary condition **Heat Transfer>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]`.

You have now defined the physics. For evaluating the equivalent heat transfer coefficient according to Equation 1 directly after solving the model, you need to define the following model coupling operators. These operators can also be defined and evaluated after computing the solution, in which case you need to choose **Update Solution** (after right-clicking on the **Study** node) to make them available without running the model again.

DEFINITIONS

Average 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Model Couplings>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 **Inlet water**.
5 Right-click **Model 1>Definitions>Average 1** and choose **Rename**.
6 Go to the **Rename Average** dialog box and type `Average 1: Inlet water` in the **New name** edit field.
7 Click **OK**.

Average 2

1 Right-click **Definitions** and choose **Model Couplings>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 **Inlet air**.
5 Right-click **Model 1>Definitions>Average 2** and choose **Rename**.
6 Go to the **Rename Average** dialog box and type `Average 2: Inlet air` in the **New name** edit field.
7 Click **OK**.

Integration 1

1 Right-click **Definitions** and choose **Model Couplings>Integration**.
2 In the **Integration** 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 **Model 1>Definitions>Integration 1** and choose **Rename**.
6 Go to the **Rename Integration** dialog box and type `Integration 1: 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 estimate of the heat exchanger performance a coarse mesh is sufficient. The solution is obtained very quickly and provides qualitatively good results. Reliable quantitative results require a good resolution, especially of the wall regions.

MESH 1

1 In the **Mesh** settings window, locate the **Mesh Settings** section.
2 From the **Element size** list, choose **Extremely coarse**.

STUDY 1

1 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

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` and `nitf.delta_w_u/d`, respectively. The critical regions in terms of the wall resolution are in the tubes.

RESULTS

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 Click the **Plot** button.

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 In the **Model Builder** window, under **Results** right-click **Data Sets** and choose **More Data Sets>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 In the **Model Builder** window, right-click **Results** 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 model coupling operators defined previously.

Derived Values

1 In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Global Evaluation**.

2 In the **Global Evaluation** settings window, locate the **Expression** section.

3 In the **Expression** edit field, type `intop1(nitf.ntflux)/(intop1(1)*(aveop2(T)-aveop1(T)))`.

4 Click the **Evaluate** button.

5 Right-click **Results>Derived Values>Global Evaluation 1** and choose **Rename**.

6 Go to the **Rename Global Evaluation** dialog box and type `Heat Transfer Coefficient` in the **New name** edit field.

7 Click **OK**.

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

8 Right-click **Derived Values** and choose **Average>Surface Average**.

9 In the **Surface Average** settings window, locate the **Selection** section.

10 From the **Selection** list, choose **Inlet water**.

11 Locate the **Expression** section. In the **Expression** edit field, type `p`.

12 Click the **Evaluate** button.

13 Right-click **Results>Derived Values>Surface Average 1** and choose **Rename**.

14 Go to the **Rename Surface Average** dialog box and type `Inlet Pressure, Water` in the **New name** edit field.

15 Click **OK**.

16 Right-click **Derived Values** and choose **Average>Surface Average**.

17 In the **Surface Average** settings window, locate the **Selection** section.

18 From the **Selection** list, choose **Inlet air**.

19 Locate the **Expression** section. In the **Expression** edit field, type `p`.

20 Click the **Evaluate** button.

21 Right-click **Results>Derived Values>Surface Average 2** and choose **Rename**.

22 Go to the **Rename Surface Average** dialog box and type `Inlet Pressure, Air` in the **New name** edit field.

23 Click **OK**.

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