Created in COMSOL Multiphysics 5.4

# 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 application, 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 tutorial 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.

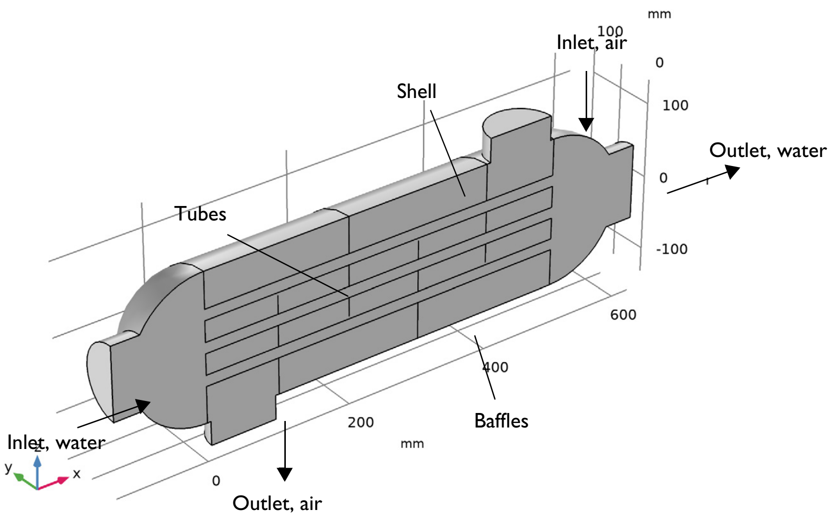


*Figure 1: Geometry of the shell-and-tube heat exchanger.*

## 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.

The heat exchanger is made of structural steel. In this example, two fluids pass through the heat exchanger. The first fluid, in this case water, flows 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 it, 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 Nonisothermal Flow predefined multiphysics coupling configured 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 **Thin Layer** boundary condition is applied. This boundary condition simulates heat transfer in thin shell structures. Here, the shell is supposed made of steel and with a 5 mm thickness.

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 $\delta_w^+$ 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 $\delta_w$ has to be small compared to the dimension of the geometry. On interior walls you have $\delta_w$ 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.

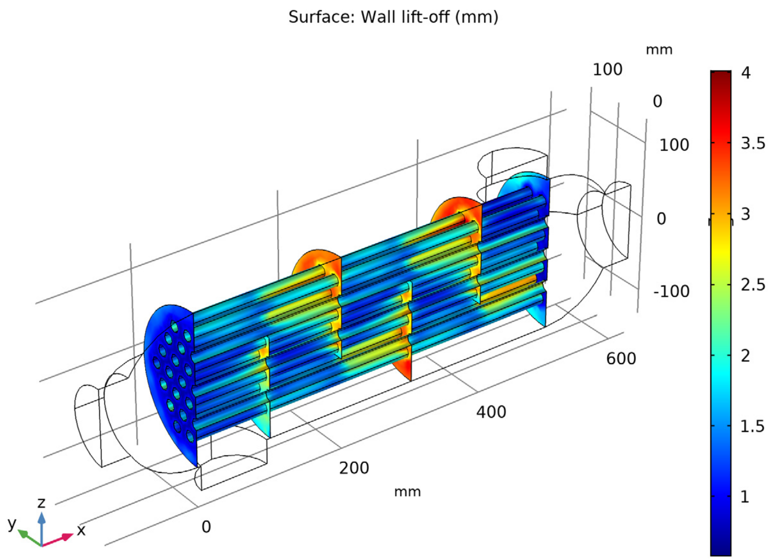


*Figure 2: Wall lift-off for the tubes.*

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.

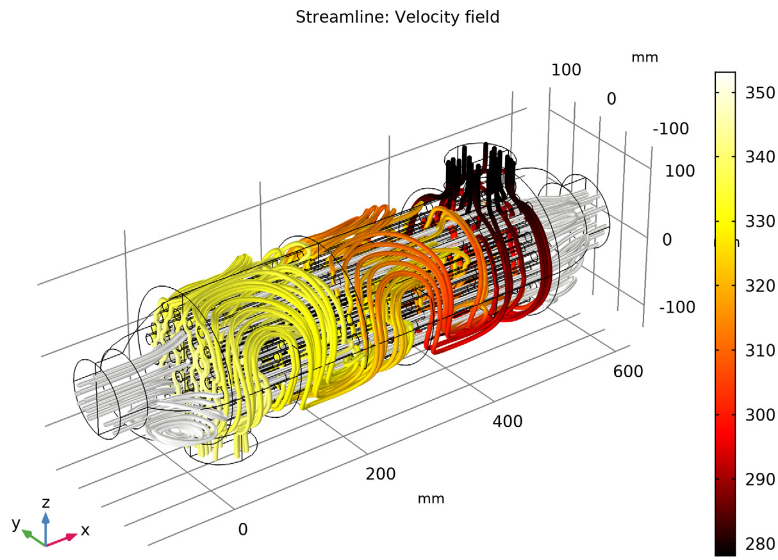


*Figure 3: Velocity streamlines.*

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

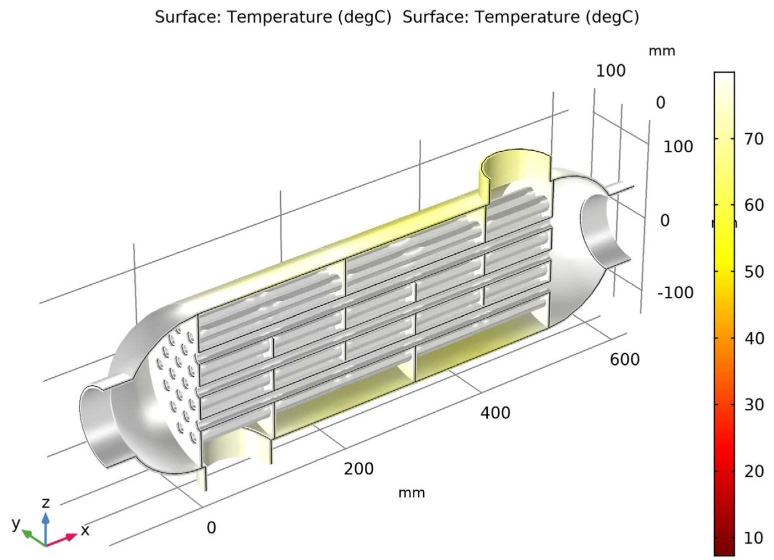


*Figure 4: Temperature at the heat exchanger boundaries.*

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 $h_{\mathrm{eq}}$ is 5.6 W/(m$^2$·K).

The pressure drop is about 37 Pa on the tube side and 13 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 Multiphysics 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 application requires about 5 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 take 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 concurrent to countercurrent heat exchanger, you only need to redefine the selections.

---

**Application 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*

From the **File** menu, choose **New**.

### NEW

In the **New** window, click **Model Wizard**.

### MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Fluid Flow>Nonisothermal Flow>Turbulent Flow>Turbulent Flow, k-**ε.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

**6** Click **Done**.

### GEOMETRY 1

The geometry sequence for the model is available in a file. If you want to create it from scratch yourself, you can follow the instructions in the Geometry Modeling Instructions section. Otherwise, insert the geometry sequence as follows:

**1** In the **Geometry** toolbar, click **Insert Sequence**.

**2** Browse to the model's Application Libraries folder and double-click the file `shell_and_tube_heat_exchanger_geom_sequence.mph`.

**3** In the **Geometry** toolbar, click **Build All**.

**4** Click the **Zoom Extents** button in the **Graphics** toolbar.

You should now see the geometry shown in Figure 1.

**5** Click the **Wireframe Rendering** button in the **Graphics** toolbar.

### GLOBAL DEFINITIONS

**1** In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

**3** In the table, enter the following settings:

| Name | Expression | Value | Description |
|---|---|---|---|
| u_water | 0.1[m/s] | 0.1 m/s | Inlet velocity, water |
| u_air | 1[m/s] | 1 m/s | Inlet velocity, air |
| T_water | 80[degC] | 353.15 K | Inlet temperature, water |
| T_air | 5[degC] | 278.15 K | Inlet temperature, air |

### ADD MATERIAL

**1** 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>Air**.

**4** Click **Add to Component** in the window toolbar.

**5** In the tree, select **Built-In>Water, liquid**.

**6** Click **Add to Component** in the window toolbar.

### MATERIALS

*Air (mat1)*

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Air Domain**.

*Water, liquid (mat2)*

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Water, liquid (mat2)**.

2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Water Domain**.

### TURBULENT FLOW, K-ε (SPF)

Since the density variation is not small, the flow cannot be regarded as incompressible. Therefore set the flow to be compressible.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, k-ε (spf)**.

2 In the **Settings** window for **Turbulent Flow, k-ε**, locate the **Physical Model** section.

3 From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.

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 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 In the **Settings** window for **Inlet**, type `Inlet 1: Water` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Inlet Water**.

4 Locate the **Turbulence Conditions** section. In the $L_T$ text 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 $U_0$ text field, type `u_water`.

*Outlet 1*

1 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 In the **Settings** window for **Outlet**, type `Outlet 1: Water` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Outlet Water**.

4 Locate the **Pressure Conditions** section. Select the **Normal flow** check box.

*Inlet 2*

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 In the **Settings** window for **Inlet**, type `Inlet 2: Air` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Inlet Air**.

4 Locate the **Turbulence Conditions** section. In the $L_T$ text 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 $U_0$ text field, type `u_air`.

*Outlet 2*

1 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 In the **Settings** window for **Outlet**, type `Outlet 2: Air` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Outlet Air**.

4 Locate the **Pressure Conditions** section. Select the **Normal flow** check box.

*Symmetry 1*

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry**.

*Interior Wall 1*

1 In the **Physics** toolbar, click **Boundaries** and choose **Interior Wall**.

2 In the **Settings** window for **Interior Wall**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Interior 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.

## HEAT TRANSFER IN FLUIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Fluids (ht)**.

*Inflow 1*

1 In the **Physics** toolbar, click **Boundaries** and choose **Inflow**.

2 In the **Settings** window for **Inflow**, type `Inflow 1: Water` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Inlet Water**.

4 Locate the **Upstream Properties** section. In the $T_{ustr}$ text field, type `T_water`.

*Outflow 1*

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

2 In the **Settings** window for **Outflow**, type `Outflow 1: Water` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Outlet Water**.

*Inflow 2*

1 In the **Physics** toolbar, click **Boundaries** and choose **Inflow**.

2 In the **Settings** window for **Inflow**, type `Inflow 2: Air` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Inlet Air**.

4 Locate the **Upstream Properties** section. In the $T_{ustr}$ text field, type `T_air`.

*Outflow 2*

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

2 In the **Settings** window for **Outflow**, type `Outflow 2: Air` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Outlet Air**.

*Symmetry 1*

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry**.

*Thin Layer 1*

1 In the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.

2 In the **Settings** window for **Thin Layer**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Walls**.

4 Locate the **Layer Model** section. From the **Layer type** list, choose **Thermally thin approximation**.

The **Thin Layer** feature requires a layered material that defines the layer properties (in particular its thickness and material). Use the **Add single layer material** button to define it.

5 Locate the **Layer Selection** section. Click **Add Single Layer Material**.

6 In the **Model Builder** window, click **Thin Layer 1**.

7 In the **Settings** window for **Thin Layer**, locate the **Layer Selection** section.

8 Click **Go to Material**.

#### MATERIALS

*Single Layer Material 1 (slmat1)*

1. In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Single Layer Material 1 (slmat1)**.
2. In the **Settings** window for **Single Layer Material**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Walls**.
4. Locate the **Layer Definition** section. In the **Thickness** text field, type `5[mm]`.
5. Click **Add Material from Library**.

#### ADD MATERIAL TO SINGLE LAYER MATERIAL 1 (SLMAT1)

1. Go to the **Add Material to Single Layer Material 1 (slmat1)** window.
2. In the tree, select **Built-In>Structural steel**.
3. Click **OK**.

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 (COMP1)

*Average 1 (aveop1)*

1. In the **Definitions** toolbar, click **Component Couplings** and choose **Average**.
2. In the **Settings** window for **Average**, type `Average Operator on Water-Air Interface` in the **Label** text field.
3. Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4. From the **Selection** list, choose **Water-Air Interface**.

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 default 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 the $x$-direction. This results in an anisotropic mesh that is suitable for the minor changes of the flow field and temperature field in the $x$-direction.

### MESH 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2. In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.
3. From the **Element size** list, choose **Extremely coarse**.

#### *Free Tetrahedral 1*

1. Right-click **Component 1 (comp1)>Mesh 1** and choose **Edit Physics-Induced Sequence**.
2. In the **Settings** window for **Free Tetrahedral**, click to expand the **Scale Geometry** section.
3. In the **x-direction scale** text field, type `0.5`.

#### *Boundary Layer Properties 1*

1. In the **Model Builder** window, expand the **Component 1 (comp1)>Mesh 1>Boundary Layers 1** node, then click **Boundary Layer Properties 1**.
2. In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Layer Properties** section.
3. In the **Number of boundary layers** text field, type `3`.
4. Click **Build All**.

### STUDY 1

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

### RESULTS

#### *Velocity (spf)*

A slice plot of the velocity field, a contour pressure plot, 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 `spf.d_w_plus` should be close to `11` and the wall lift-off `spf.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 `spf.d_w_plus_u/d` or `spf.delta_w_u/d`, respectively. The critical regions in terms of the wall resolution are in the tubes.

*Wall Resolution*

Delete the **Upside Wall Lift-Off** and **Downside Wall Lift-Off** from the **Wall Resolution (spf)** plot group.

1 In the **Model Builder** window, expand the **Results>Wall Resolution (spf)** node, then click **Wall Resolution**.
2 In the **Settings** window for **Surface**, locate the **Data** section.
3 From the **Data set** list, choose **Interior Walls**.
4 Locate the **Expression** section. In the **Expression** text field, type `spf.delta_w_u`.
5 In the **Wall Resolution (spf)** toolbar, click **Plot**.

Figure 4 shows the temperature distribution on all wall boundaries. The default temperature plot uses the **Surface 1** data set created automatically and contains exterior walls only. It is easy to change it by using the selection created at the beginning.

*All Walls*

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

*Temperature (ht)*

1 In the **Model Builder** window, under **Results** click **Temperature (ht)**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Data set** list, choose **All Walls**.

*Surface 1*

1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface 1**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 From the **Unit** list, choose **degC**.

*Surface 2*

1 In the **Model Builder** window, under **Results>Temperature (ht)** click **Surface 2**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 From the **Unit** list, choose **degC**.

*Surface 1*

1 In the **Model Builder** window, under **Results>Temperature (ht)** click **Surface 1**.
2 In the **Temperature (ht)** toolbar, click **Plot**.

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:

*Mirror 3D 1*

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

*3D Plot Group 6*

1. In the **Results** toolbar, click **3D Plot Group**.
2. In the **Settings** window for **3D Plot Group**, type `Velocity, Streamlines` in the **Label** text field.
3. Locate the **Data** section. From the **Data set** list, choose **Mirror 3D 1**.

*Streamline 1*

1. In the **Velocity, Streamlines** toolbar, click **Streamline**.
2. In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
3. In the **Points** text field, type `100`.
4. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.

*Color Expression 1*

1. In the **Velocity, Streamlines** toolbar, click **Color Expression**.
2. In the **Settings** window for **Color Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1> Heat Transfer in Fluids>Temperature>T - Temperature - K**.
3. Locate the **Coloring and Style** section. From the **Color table** list, choose **Thermal**.

*Velocity, Streamlines*

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

*Global Evaluation 1*

1. In the **Results** toolbar, click **Global Evaluation**.
2. In the **Settings** window for **Global Evaluation**, type `Heat Transfer Coefficient` in the **Label** text field.

3 Locate the **Expressions** section. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| `aveop1(abs(nitf1.qwf_u))/(T_water-T_air)` | `W/(m^2*K)` | |

The variable `nitf1.qwf_u` efficiently computes the heat flux through the wall separating water and air domains, based on the wall function definition.

4 Click **Evaluate**.

To evaluate the pressure drop, the average inlet pressures are computed.

*Surface Average 1*

1 In the **Results** toolbar, click **More Derived Values** and choose **Average>Surface Average**.

2 In the **Settings** window for **Surface Average**, type `Inlet Pressure, Water` in the **Label** text field.

3 Locate the **Selection** section. From the **Selection** list, choose **Inlet Water**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Turbulent Flow, k-ε>Velocity and pressure>p - Pressure**.

5 Click **Evaluate**.

*Surface Average 2*

1 In the **Results** toolbar, click **More Derived Values** and choose **Average>Surface Average**.

2 In the **Settings** window for **Surface Average**, type `Inlet Pressure, Air` in the **Label** text field.

3 Locate the **Selection** section. From the **Selection** list, choose **Inlet Air**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Turbulent Flow, k-ε>Velocity and pressure>p - Pressure**.

5 Click **Evaluate**.

### TABLE

1 Go to the **Table** window.

The tube side pressure drop is 36 Pa and the shell side pressure drop is 13 Pa.

## *Geometry Modeling Instructions*

If you wish to create the geometry yourself, follow these steps.

### GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

*Cylinder 1 (cyl1)*

1 In the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `100`.

4 In the **Height** text field, type `500`.

5 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

*Cylinder 2 (cyl2)*

1 In the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `50`.

4 In the **Height** text field, type `750`.

5 Locate the **Position** section. In the **x** text field, type `-125`.

6 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

*Sphere 1 (sph1)*

1 In the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for **Sphere**, locate the **Size** section.

3 In the **Radius** text field, type `100`.

*Sphere 2 (sph2)*

1 In the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for **Sphere**, locate the **Size** section.

3 In the **Radius** text field, type `100`.

4 Locate the **Position** section. In the **x** text field, type `500`.

*Cylinder 3 (cyl3)*

1 In the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `45`.

4 In the **Height** text field, type `132.5`.

**5** Locate the **Position** section. In the **x** text field, type `50`.

**6** In the **z** text field, type `-132.5`.

*Cylinder 4 (cyl4)*

**1** Right-click **Cylinder 3 (cyl3)** and choose **Duplicate**.

**2** In the **Settings** window for **Cylinder**, locate the **Position** section.

**3** In the **x** text field, type `450`.

**4** In the **z** text field, type `0`.

*Union 1 (uni1)*

**1** In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

**2** Click in the **Graphics** window and then press Ctrl+A to select all objects.

**3** In the **Settings** window for **Union**, locate the **Union** section.

**4** Clear the **Keep interior boundaries** check box.

*Block 1 (blk1)*

**1** In the **Geometry** toolbar, click **Block**.

**2** In the **Settings** window for **Block**, locate the **Size and Shape** section.

**3** In the **Width** text field, type `500`.

**4** In the **Depth** text field, type `300`.

**5** In the **Height** text field, type `300`.

**6** Locate the **Position** section. In the **y** text field, type `-150`.

**7** In the **z** text field, type `-150`.

**8** Click to expand the **Layers** section. Find the **Layer position** subsection. Select the **Right** check box.

**9** Clear the **Bottom** check box.

**10** In the table, enter the following settings:

| Layer name | Thickness (mm) |
|---|---|
| Layer 1 | 100 |
| Layer 2 | 100 |
| Layer 3 | 100 |
| Layer 4 | 100 |

*Block 2 (blk2)*

**1** In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `500`.

4 In the **Depth** text field, type `300`.

5 In the **Height** text field, type `65`.

6 Locate the **Position** section. In the **y** text field, type `-150`.

7 In the **z** text field, type `-32.5`.

### *Partition Objects 1 (par1)*

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

2 Select the object **uni1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 Find the **Tool objects** subsection. Select the **Active** toggle button.

5 Select the objects **blk1** and **blk2** only.

### *Work Plane 1 (wp1)*

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **yz-plane**.

### *Plane Geometry*

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1>**
**Work Plane 1 (wp1)** click **Plane Geometry**.

### *Work Plane 1 (wp1)>Circle 1 (c1)*

1 In the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `7.5`.

4 Locate the **Position** section. In the **xw** text field, type `-75`.

5 In the **yw** text field, type `-43.5`.

### *Work Plane 1 (wp1)>Array 1 (arr1)*

1 In the **Work Plane** toolbar, click **Transforms** and choose **Array**.

2 Select the object **c1** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **xw size** text field, type `7`.

5 In the **yw size** text field, type `3`.

6 Locate the **Displacement** section. In the **xw** text field, type `25`.

7 In the **yw** text field, type `43.5`.

*Work Plane 1 (wp1)>Circle 2 (c2)*

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>Plane Geometry** right-click **Circle 1 (c1)** and choose **Duplicate**.

2 In the **Settings** window for **Circle**, locate the **Position** section.

3 In the **xw** text field, type `-62.5`.

4 In the **yw** text field, type `-65.25`.

*Work Plane 1 (wp1)>Array 2 (arr2)*

1 In the **Work Plane** toolbar, click **Transforms** and choose **Array**.

2 Select the object **c2** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **xw size** text field, type `6`.

5 In the **yw size** text field, type `4`.

6 Locate the **Displacement** section. In the **xw** text field, type `25`.

7 In the **yw** text field, type `43.5`.

8 Click **Build Selected**.

*Work Plane 1 (wp1)>Delete Entities 1 (del1)*

1 In the **Work Plane** toolbar, click **Delete**.

2 Select the objects **arr1(1,1)**, **arr1(1,3)**, **arr1(7,1)**, **arr1(7,3)**, **arr2(1,1)**, **arr2(1,4)**, **arr2(6,1)**, and **arr2(6,4)** only.

*Work Plane 1 (wp1)*

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

*Extrude 1 (ext1)*

1 In the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

| Distances (mm) |
|---|
| 500 |

*Block 3 (blk3)*

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `1[m]`.

4 In the **Depth** text field, type `150`.

5 In the **Height** text field, type `400`.

6 Locate the **Position** section. In the **x** text field, type `-200`.

7 In the **y** text field, type `-150`.

8 In the **z** text field, type `-200`.

*Difference 1 (dif1)*

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the objects **ext1** and **par1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **blk3** only.

*Ignore Faces 1 (igf1)*

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Faces**.

2 Click the **Go to Default View** button in the **Graphics** toolbar.

3 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

4 On the object **fin**, select Boundaries 10, 17, 18, 25, 26, 32, 35, 38, 41, 52, 55, 58, 67, 70, 73, 76, 87, 90, 93, 102, 105, 112, 125, 132, 133, 138, 140, 141, 147, 150, 153, 156, 167, 170, 173, 182, 185, 188, 191, 202, 205, 208, 217, 220, 227, 234, 237, 244, 245, 252, 253, 259, 262, 265, 268, 279, 282, 285, 294, 297, 300, 303, 314, 317, 320, 329, 332, 339, 349, 356, 357, 362, 364, 365, 371, 374, 377, 380, 391, 394, 397, 406, 409, 412, 415, 426, 429, 432, 441, 444, 451, 458, 461, 468, 469, 476, 477, 483, 486, 489, 492, 503, 506, 509, 518, 521, 524, 527, 538, 541, 544, 553, 556, 563, 576, 578, 580, 582–587, and 589–598 only.

*Explicit Selection 1 (sel1)*

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Water Domain` in the **Label** text field.

3 On the object **igf1**, select Domain 1 only.

*Explicit Selection 2 (sel2)*

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Air Domain` in the **Label** text field.

3 On the object **igf1**, select Domain 2 only.

*Adjacent Selection 1 (adjsel1)*

1 In the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.

2 In the **Settings** window for **Adjacent Selection**, type `Water Domain, Exterior Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Add**.

4 In the **Add** dialog box, select **Water Domain** in the **Input selections** list.

5 Click **OK**.

*Adjacent Selection 2 (adjsel2)*

1 In the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.

2 In the **Settings** window for **Adjacent Selection**, type `Air Domain, Exterior Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Add**.

4 In the **Add** dialog box, select **Air Domain** in the **Input selections** list.

5 Click **OK**.

*Adjacent Selection 3 (adjsel3)*

1 In the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.

2 In the **Settings** window for **Adjacent Selection**, type `Baffles` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Add**.

4 In the **Add** dialog box, select **Air Domain** in the **Input selections** list.

5 Click **OK**.

6 In the **Settings** window for **Adjacent Selection**, locate the **Output Entities** section.

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

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

*Box Selection 1 (boxsel1)*

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, type `Symmetry` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **y maximum** text field, type `0`.

**5** Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

*Explicit Selection 3 (sel3)*

**1** In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

**2** In the **Settings** window for **Explicit Selection**, type `Inlet Water` in the **Label** text field.

**3** Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

**4** On the object **igf1**, select Boundary 1 only.

*Explicit Selection 4 (sel4)*

**1** In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

**2** In the **Settings** window for **Explicit Selection**, type `Outlet Water` in the **Label** text field.

**3** Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

**4** On the object **igf1**, select Boundary 340 only.

*Explicit Selection 5 (sel5)*

**1** In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

**2** In the **Settings** window for **Explicit Selection**, type `Inlet Air` in the **Label** text field.

**3** Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

**4** On the object **igf1**, select Boundary 332 only.

*Explicit Selection 6 (sel6)*

**1** In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

**2** In the **Settings** window for **Explicit Selection**, type `Outlet Air` in the **Label** text field.

**3** Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

**4** On the object **igf1**, select Boundary 89 only.

*Difference Selection 1 (difsel1)*

**1** In the **Geometry** toolbar, click **Selections** and choose **Difference Selection**.

**2** In the **Settings** window for **Difference Selection**, type `Water Domain, Walls` in the **Label** text field.

**3** Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

**4** Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, select **Water Domain, Exterior Boundaries** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click **Add**.

9 In the **Add** dialog box, in the **Selections to subtract** list, choose **Symmetry**, **Inlet Water**, and **Outlet Water**.

10 Click **OK**.

*Difference Selection 2 (difsel2)*

1 In the **Geometry** toolbar, click **Selections** and choose **Difference Selection**.

2 In the **Settings** window for **Difference Selection**, locate the **Geometric Entity Level** section.

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

4 In the **Label** text field, type `Air Domain, Walls`.

5 Locate the **Input Entities** section. Click **Add**.

6 In the **Add** dialog box, in the **Selections to add** list, choose **Air Domain, Exterior Boundaries** and **Baffles**.

7 Click **OK**.

8 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

9 Click **Add**.

10 In the **Add** dialog box, in the **Selections to subtract** list, choose **Symmetry**, **Inlet Air**, and **Outlet Air**.

11 Click **OK**.

*Union Selection 1 (unisel1)*

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

2 In the **Settings** window for **Union Selection**, locate the **Geometric Entity Level** section.

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

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Water Domain, Walls** and **Air Domain, Walls**.

6 Click **OK**.

7 In the **Settings** window for **Union Selection**, type `Walls` in the **Label** text field.

*Intersection Selection 1 (intsel1)*

1 In the **Geometry** toolbar, click **Selections** and choose **Intersection Selection**.

2 In the **Settings** window for **Intersection Selection**, type `Water-Air Interface` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to intersect** list, choose **Water Domain, Walls** and **Air Domain, Walls**.

6 Click **OK**.

*Union Selection 2 (unisel2)*

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

2 In the **Settings** window for **Union Selection**, locate the **Geometric Entity Level** section.

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

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Baffles** and **Water-Air Interface**.

6 Click **OK**.

7 In the **Settings** window for **Union Selection**, type `Interior Walls` in the **Label** text field.