Turbulent Flow Through a Shell-and-Tube Heat Exchanger Cross Section

Introduction

This example describes a part of a shell-and-tube heat exchanger (see Figure 1), where hot water enters from above. The cooling medium, commonly water, flows through the pipes and enters from the side. The tubes are assumed to be made of stainless steel.
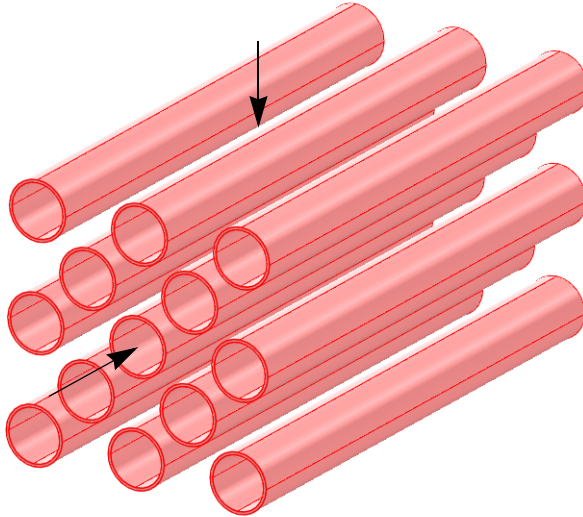

Figure 1: A tube bundle from a shell-and-tube heat exchanger. The arrows indicate the flow directions.

Assuming that the cooling water is in abundant supply, the flow through the pipes has a constant temperature. Under that assumption, you can model this heat exchanger by a 2D model as shown in Figure 2, and the corresponding 2D domain appears in Figure 3. Note that the pipe interiors are not part of the domain because the model assumes the temperature to be constant there.
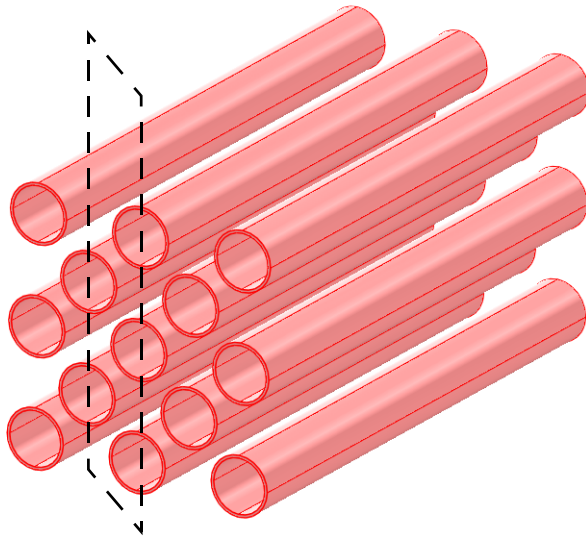
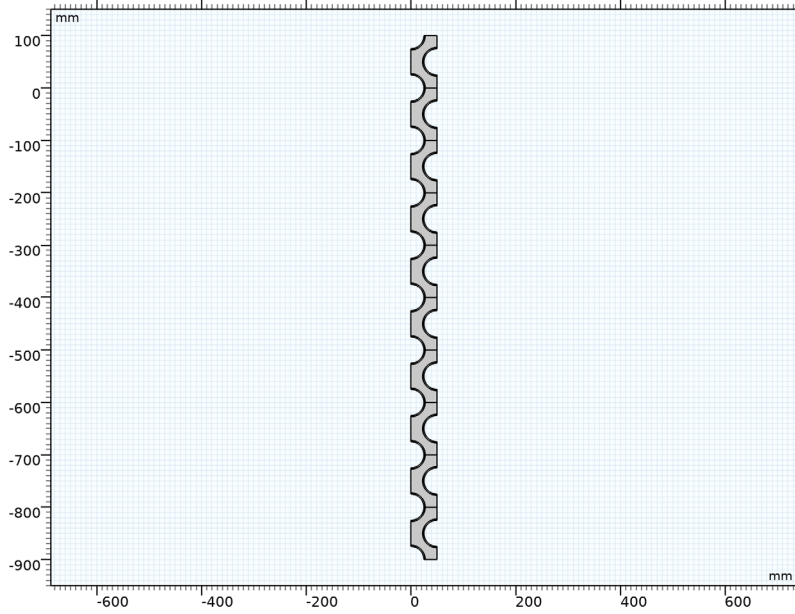Figure 2: The dashed line marks the model region, which is shown in Figure 3.

Figure 3: The modeled 2D region.

An important characteristic of the flow is the Reynolds number, Re, defined as

$$\mathrm{Re} = \frac{UL}{\eta}$$

where U is a velocity scale, L is a length scale, and η is a kinematic viscosity. If the Reynolds number is low, no turbulence model is needed. If, on the other hand, the Reynolds number is high, then the flow is dominated by convection, and a turbulence model is necessary.

In this case, a suitable velocity scale is the mean inlet velocity, which is 1.5 m/s, and L is set to the pipe radius. Then, using standard values for water for the density and viscosity, the equation gives an approximate Reynolds number of 75,000, which is high enough to warrant the use of a turbulence model (see Ref. 1 for more information on flow regimes for different Reynolds numbers).

The following example demonstrates how to model a coupled fluid-thermal interaction problem with COMSOL Multiphysics, using the Nonisothermal Flow multiphysics coupling.

Model Definition

SOLID AND FLUID HEAT TRANSFER — INCLUDING THE FLUID DYNAMICS

The governing equations in this model are (Ref. 2–Ref. 4)

- The Reynolds-Averaged Navier-Stokes (RANS) equations and a k-ε turbulence model.
- Heat transport equations in the water and the solid (steel) tube walls.

The Nonisothermal Flow multiphysics coupling sets up both sets of equations together with the applicable couplings, making it easy to model the fluid-thermal interaction.

The model uses temperature-dependent properties for water and steel from the built-in material library. It incorporates the influence of the turbulent fluctuations on the temperature field by using the Kays-Crawford model for the turbulent Prandtl number (for details, see the section *Turbulent Nonisothermal Flow Theory* in the *Heat Transfer Module User's Guide*). Furthermore, to account for the effect of mixing due to eddies, it is necessary to correct the fluid's thermal conductivity. The turbulence results in an effective thermal conductivity, k_{eff}, according to the equation

$$k_{\mathrm{eff}} = k + \kappa_\mathrm{T} \qquad \kappa_\mathrm{T} = \frac{C_p \mu_\mathrm{T}}{\mathrm{Pr}_\mathrm{T}}$$

where k is the fluid's physical thermal conductivity and κ_T is the turbulent thermal conductivity. Furthermore, μ_T denotes the turbulent dynamic viscosity, C_p is the heat capacity, and Pr_T refers to the turbulent Prandtl number. Be careful not to confuse k in the meaning of thermal conductivity with k in the meaning of turbulent kinetic energy.

The boundary conditions for the problem are:

- k-ε equations in the fluid domain:
  - Specified initial velocity
  - Symmetry at the region borders
  - Wall function at the pipe/water interfaces
  - Fixed outlet pressure
- Heat transport equations:
  - Fixed temperature at the inlet
  - Convection-dominated transport at the outlet
  - Symmetry (thermal insulation) at the region borders
  - Thermal wall function at the pipe/water interfaces
  - Fixed temperature at the inner pipe surfaces

Results for the Flow/Heat Model

Figure 4 shows the velocity in the inlet of several segments in the middle of the computational domain. The flow field is periodic in the y direction with the constant pressure drop of about 1310 Pa. (Figure 5).
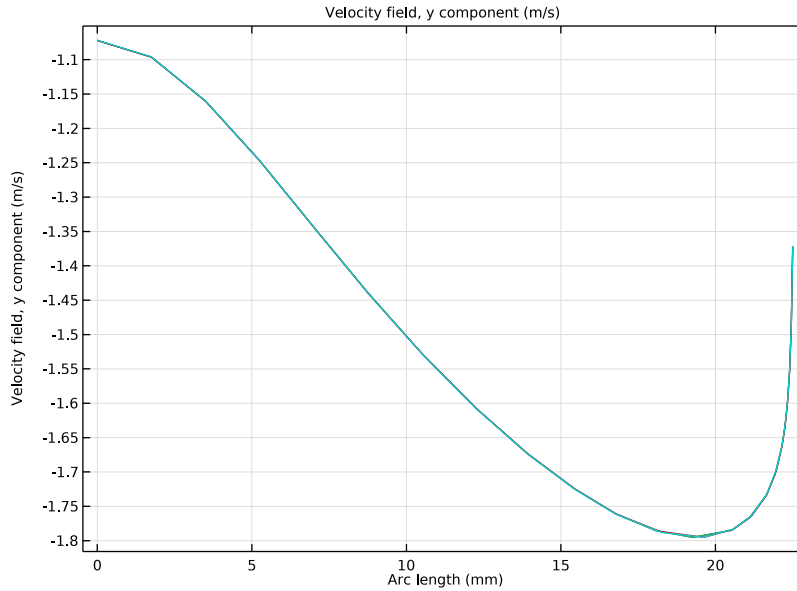
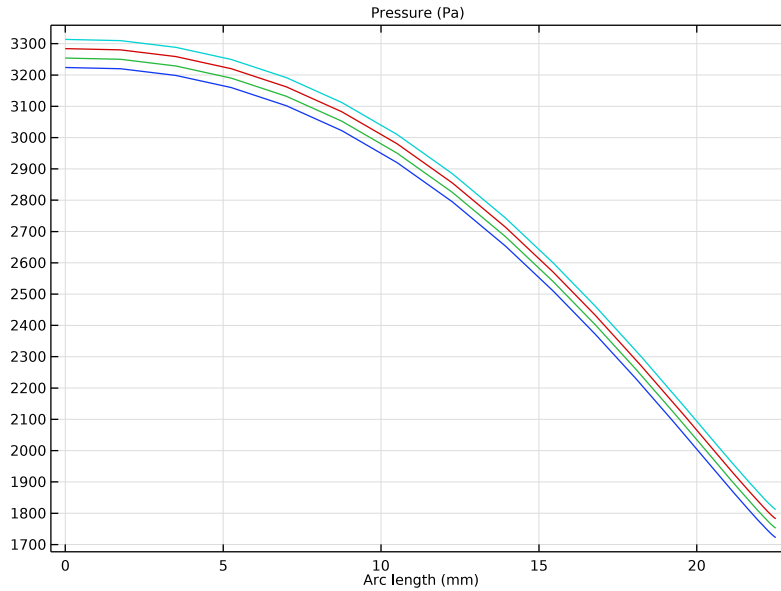

Figure 4: Velocity at y = 400, 500, 600, and 700 mm.

Figure 5: Pressure at y = 400, 500, 600, and 700 mm.

Figure 6 depicts the temperature distribution and velocity streamlines in the segment. The heat transfer is strongly influenced by the details of the velocity field. Observe the low-temperature zones behind the pipes created by the recirculation zones.

Figure 7 shows the temperature profiles at *y* = 400, 500, 600, and 700 mm.
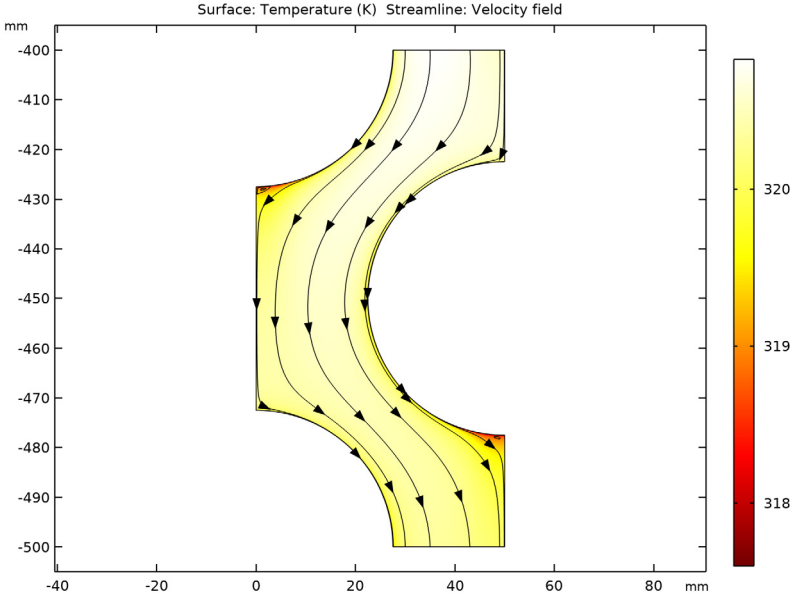
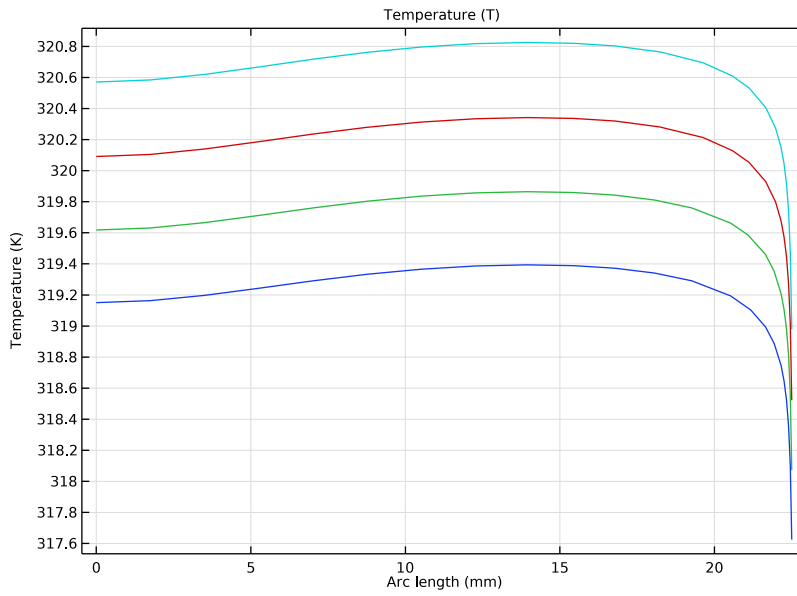*Figure 6: Temperature distribution in the segment.*

Figure 7: Temperature distribution at $y = 400, 500, 600,$ *and* 700 *mm.*

Notes About the COMSOL Implementation

You start by performing an initial calculation for the single segment of the computational domain (Figure 8). Proceed by creating a linear array of the segments. The appropriate physical interface settings are automatically extended to all elements of the array.
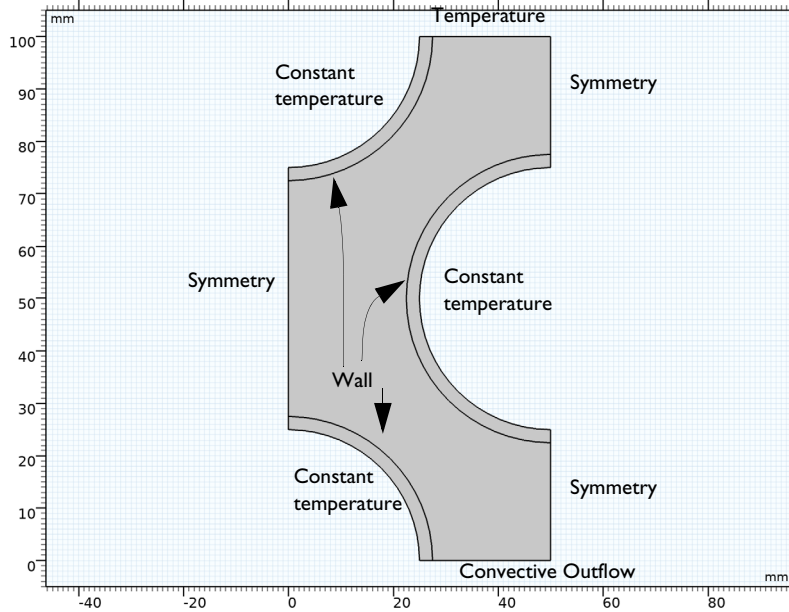

Figure 8: Modeled segment of the heat exchanger with boundary conditions.

References

1. J.R. Welty, C.E. Wicks, and R.E. Wilson, *Fundamentals of Momentum, Heat and Mass Transfer,* 3rd ed, John Wiley & Sons, 1984.

2. B.E. Launder and D.B. Spalding, "The Numerical Computation of Turbulent Flows," *Computer Methods in Applied Mechanics*, vol. 3, pp. 269–289, 1974.

3. D.C. Wilcox, *Turbulence Modeling for CFD,* 2nd ed, DCW Industries, 2000.

4. H.K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics*, Prentice Hall, 1995.

Application Library path: Heat_Transfer_Module/Heat_Exchangers/
turbulent_heat_exchanger

Modeling Instructions

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

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

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

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

3 Click **Add**.

4 Click **Study**.

In this model, you consider the flow to be nearly incompressible and independent of
temperature variations. Under these assumptions, the **Stationary, One-Way NITF** study
performs best.

5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics>Stationary, One-
Way NITF**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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
T_in	323[K]	323 K	Inflow temperature
T_pipe	278[K]	278 K	Pipe temperature
v_in	-1.5[m/s]	-1.5 m/s	Inflow velocity

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**.

Now draw the segment of the heat exchanger that forms the model geometry for the first simulation.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click ▭ **Rectangle**.

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

3 In the **Width** text field, type 50.

4 In the **Height** text field, type 100.

5 Click ▦ **Build Selected**.

Circle 1 (c1)

1 In the **Geometry** toolbar, click ⊙ **Circle**.

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

3 In the **Radius** text field, type 27.5.

4 Click ▦ **Build Selected**.

Circle 2 (c2)

1 In the **Geometry** toolbar, click ⊙ **Circle**.

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

3 In the **Radius** text field, type 25.

4 Click ▦ **Build Selected**.

Difference 1 (dif1)

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

2 Select the object **c1** only.

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

4 Find the **Objects to subtract** subsection. Select the ▭ **Activate Selection** toggle button.

5 Select the object **c2** only.

6 Click ▦ **Build Selected**.

Copy 1 (copy1)

1 In the **Geometry** toolbar, click ⟳ **Transforms** and choose **Copy**.

2 Select the object **dif1** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **x** text field, type 0 50.

5 In the **y** text field, type 100 50.

6 Click ⊞ **Build Selected**.

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**, click ⊞ **Build Selected**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **uni1**, select Domains 1, 2, 5, 7, 8, and 10 only.

It might be easier to select the domains by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)

5 Click ⬚ **Build Selected**.

You should now see the following geometry.
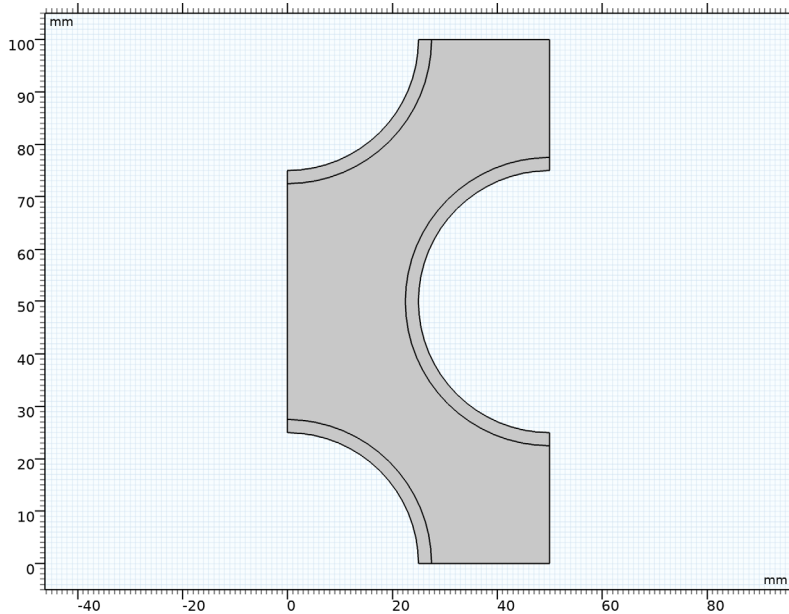
A D D   M A T E R I A L

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>Steel AISI 4340**.

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.

7 In the **Home** toolbar, click ⬚ **Add Material** to close the **Add Material** window.

M A T E R I A L S

Steel AISI 4340 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click
Steel AISI 4340 (mat1).

2 Select Domains 1, 3, and 4 only.
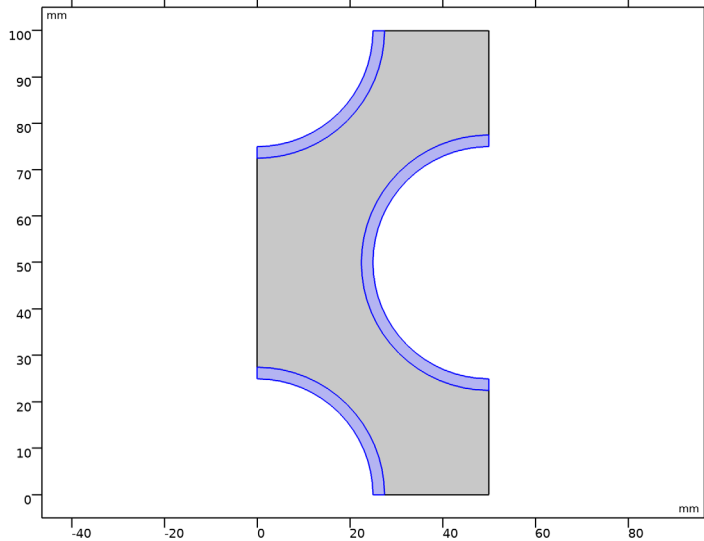
Water, liquid (mat2)

I In the **Model Builder** window, click **Water, liquid (mat2)**.

2 Select Domain 2 only.

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

4 Click **Create Selection**.

5 In the **Create Selection** dialog box, type Water in the **Selection name** text field.

6 Click **OK**.

HEAT TRANSFER IN FLUIDS (HT)

Set the reference temperature to an estimated value of T_in where the flow operates.

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

2 In the **Settings** window for **Heat Transfer in Fluids**, locate the **Physical Model** section.

3 In the T_{ref} text field, type T_in.

Solid I

I In the **Physics** toolbar, click **Domains** and choose **Solid**.

2 Select Domains 1, 3, and 4 only.

TURBULENT FLOW, K-ε (SPF)

The density variations are small enough to consider the fluid as incompressible. Change the compressibility option accordingly in the physics interface. The reference temperature is set in the Heat Transfer interface.

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 **Incompressible flow**.

4 Locate the **Domain Selection** section. From the **Selection** list, choose **Water**.
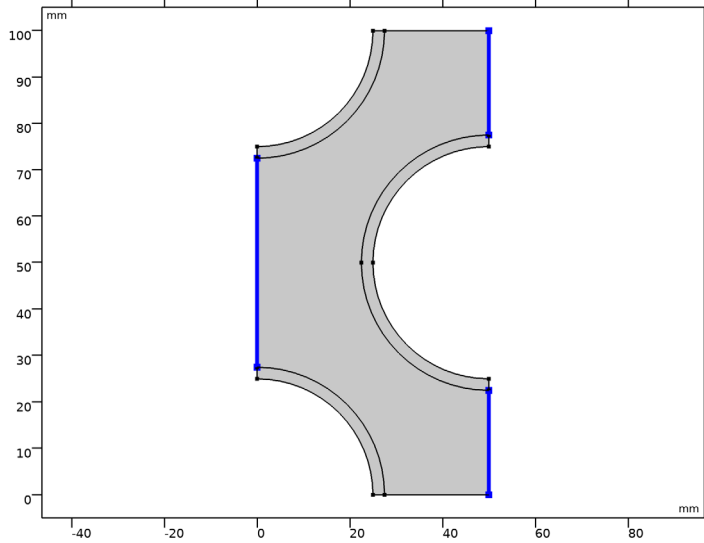
Inlet 1

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

2 Select Boundary 7 only.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 Click the **Velocity field** button.

5 Specify the $\mathbf{u}_0$ vector as

0	x
v_in	y

Symmetry 1

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

2 Select Boundaries 2, 8, and 11 only.

Outlet 1

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

2 Select Boundary 6 only.

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 Select Boundary 7 only.

3 In the **Settings** window for **Inflow**, locate the **Upstream Properties** section.

4 In the T_{ustr} text field, type T_in.

Outflow 1

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

2 Select Boundary 6 only.

Temperature 1

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

2 Select Boundaries 12, 15, 18, and 19 only.

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

**4** In the T_0 text field, type T_pipe.

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 **Fine**.

Size

Right-click **Component 1 (comp1)>Mesh 1** and choose **Edit Physics-Induced Sequence**.

Size 2

1 In the **Settings** window for **Size**, locate the **Element Size** section.

2 Click the **Custom** button.

3 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

4 In the associated text field, type 0.5.

5 Click ▦ **Build All**.
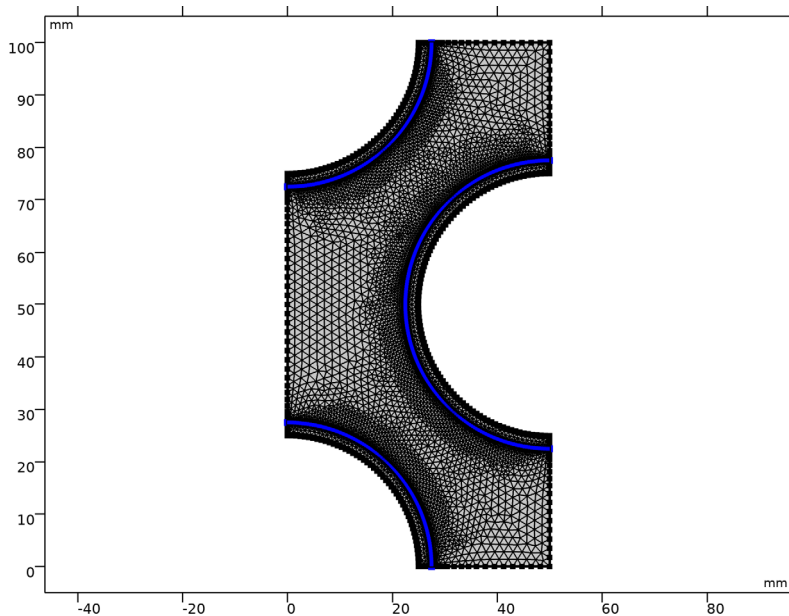
STUDY 1

Solve the problem for a single heat exchanger segment.

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

Velocity (spf)

The default plots show the velocity, the wall resolution, and the temperature distribution in the segment.

GEOMETRY 1

Continue with creating the array of the elements.

Array 1 (arr1)

1 In the **Geometry** toolbar, click ⟳ **Transforms** and choose **Array**.

2 Select the object **del1** only.

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

4 From the **Array type** list, choose **Linear**.

5 In the **Size** text field, type 10.

6 Locate the **Displacement** section. In the **y** text field, type -100.

7 Click ▦ **Build All Objects**.

8 Click the ⊕ **Zoom Extents** button in the **Graphics** toolbar.

You should now see the following geometry:
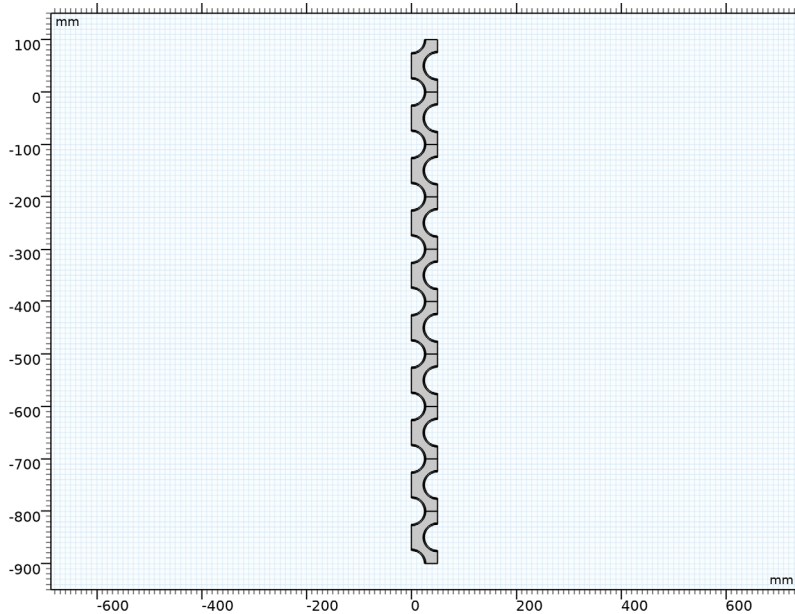
COMSOL Multiphysics automatically extends the material properties, meshes, and physical interface settings to all segments of the exchanger.

STUDY 1

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

RESULTS

Default Plot Groups
Default plots are generated for the velocity magnitude, pressure, boundary wall resolution, and temperature.

Proceed as follows to plot the *y*-velocity component for a few segments in the middle of the computational domain (Figure 4).

Velocity along Arc Length
1 In the **Home** toolbar, click 🗗 **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type Velocity along Arc Length in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type Velocity field, y component (m/s).

Line Graph 1

1 In the **Velocity along Arc Length** toolbar, click 〰 **Line Graph**.

2 Click the ⟷ **Zoom Extents** button in the **Graphics** toolbar.

3 Select Boundary 44 only.

4 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Turbulent Flow, k-ε>Velocity and pressure>Velocity field - m/s>v - Velocity field, y component**.

5 In the **Velocity along Arc Length** toolbar, click 🔲 **Plot**.

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

3 Click 🖌 **Clear Selection**.

4 Select Boundary 45 only.

5 In the **Velocity along Arc Length** toolbar, click 🔲 **Plot**.

Line Graph 3 and Line Graph 4

Create two more line graphs by repeating the steps for **Line Graph 2**, except select boundaries 46 and 47, respectively.

Velocity along Arc Length

From Figure 4 it is evident that the flow field is periodic in the *y* direction. The pressure drop on the segment is about 1,320 Pa. To plot the pressure distribution (Figure 5), proceed as follows.

Pressure along Arc Length

1 In the **Model Builder** window, right-click **Velocity along Arc Length** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type Pressure along Arc Length in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type Pressure (Pa).

Line Graph 1

1 In the **Model Builder** window, expand the **Pressure along Arc Length** node, then click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Turbulent Flow, k-ε>Velocity and pressure>p - Pressure - Pa**.

Line Graph 2

1 In the **Model Builder** window, click **Line Graph 2**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type p-1320.

Line Graph 3

1 In the **Model Builder** window, click **Line Graph 3**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type p-1320*2.

Line Graph 4

1 In the **Model Builder** window, click **Line Graph 4**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type p-1320*3.

4 In the **Pressure along Arc Length** toolbar, click ▣ **Plot**.

To reproduce , add the velocity streamlines to the temperature plot.

Study 1/Solution 1 (3) (sol1)

In the **Results** toolbar, click ▦ **More Datasets** and choose **Solution**.

Selection

1 In the **Results** toolbar, click ⚲ **Attributes** and choose **Selection**.

2 Click the ⤢ **Zoom Extents** button in the **Graphics** toolbar.

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

4 From the **Geometric entity level** list, choose **Domain**.

5 Select Domain 14 only.

Temperature (ht)

1 In the **Model Builder** window, click **Temperature (ht)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 1/Solution 1 (3) (sol1)**.

4 In the **Temperature (ht)** toolbar, click ▣ **Plot**.

5 Click the ⤢ **Zoom Extents** button in the **Graphics** toolbar.

Streamline 1

1 In the **Temperature (ht)** toolbar, click 〰 **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 From the **Positioning** list, choose **Starting-point controlled**.

4 From the **Entry method** list, choose **Coordinates**.

5 In the **x** text field, type `0.2 1.9 27.6 30 35 43 49 48 47`.

6 In the **y** text field, type `-429 -428 -500 -500 -500 -500 -500 -478 -479.3`.

7 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

8 Select the **Number of arrows** check box.

9 In the associated text field, type 30.

10 In the **Temperature (ht)** toolbar, click ▣ **Plot**.

Finally, plot the temperature on boundaries 45–48 (Figure 7).

Temperature along Arc Length

1 In the **Model Builder** window, right-click **Velocity along Arc Length** and choose **Duplicate**.

2 In the **Settings** window for **ID Plot Group**, type `Temperature along Arc Length` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Temperature (T)`.

Line Graph 1

1 In the **Model Builder** window, expand the **Temperature along Arc Length** node, then click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Heat Transfer in Fluids>Temperature>T - Temperature - K**.

Line Graph 2–Line Graph 4

Repeat the steps in the previous section for **Line Graph 2** through **Line Graph 4**.

To create a 3D plot of the temperature through the steel pipes and the velocity streamlines in the water, follow the instructions below.

Mirror 2D 1

1 In the **Results** toolbar, click ▦ **More Datasets** and choose **Mirror 2D**.

2 In the **Settings** window for **Mirror 2D**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 1/Solution 1 (3) (sol1)**.

4 Locate the **Axis Data** section. In row **Point 1**, set **x** to 50.

5 In row **Point 2**, set **x** to 50.

Extrusion 2D 1

1 In the **Results** toolbar, click ▦ **More Datasets** and choose **Extrusion 2D**.

2 In the **Settings** window for **Extrusion 2D**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 2D 1**.

4 Locate the **Extrusion** section. In the **z maximum** text field, type 100.

Study 1/Solution 1 (4) (sol1)

In the **Results** toolbar, click ▦ **More Datasets** and choose **Solution**.

Selection

1 In the **Results** toolbar, click ▦ **Attributes** and choose **Selection**.

2 Click the ⊹ **Zoom Extents** button in the **Graphics** toolbar.

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

4 From the **Geometric entity level** list, choose **Domain**.

5 Select Domains 15 and 35 only.

Extrusion 2D 1, Mirror 2D 1

In the **Model Builder** window, under **Results>Datasets**, Ctrl-click to select **Mirror 2D 1** and **Extrusion 2D 1**.

Mirror 2D 2

1 Right-click and choose **Duplicate**.

2 In the **Settings** window for **Mirror 2D**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 1/Solution 1 (4) (sol1)**.

Extrusion 2D 2

1 In the **Model Builder** window, click **Extrusion 2D 2**.

2 In the **Settings** window for **Extrusion 2D**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 2D 2**.

Velocity and Temperature 3D

1 In the **Results** toolbar, click ▥ **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type Velocity and Temperature 3D in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

1 Right-click **Velocity and Temperature 3D** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Extrusion 2D 2**.

4 Locate the **Expression** section. In the **Expression** text field, type T.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

Streamline 1

1 In the **Model Builder** window, right-click **Velocity and Temperature 3D** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 From the **Positioning** list, choose **Magnitude controlled**.

4 In the **Min distance** text field, type 0.1.

5 In the **Max distance** text field, type 0.3.

6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.

7 Select the **Radius scale factor** check box.

8 In the **Tube radius expression** text field, type 0.7.

9 Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

10 From the **Arrow distribution** list, choose **Equal inverse time**.

Color Expression 1

1 Right-click **Streamline 1** and choose **Color Expression**.

2 In the **Velocity and Temperature 3D** toolbar, click **Plot**.