Model created in COMSOL Multiphysics 6.4

Cross-Flow Heat Exchanger

Introduction

This application simulates the fluid flow and heat transfer in a micro heat exchanger of cross-flow type made of stainless steel. Heat exchangers of this type are found in lab-on-chip devices in biotechnology and microreactors, for example for micro fuel cells. The application takes into account heat transferred through both convection and conduction. The geometry and material properties are taken from Ref. 1.

Model Definition

Figure 1 shows the heat exchanger's geometry. Notice that the fluid channels have a square cross section rather than the circular cross section more commonly used in micro heat exchangers. A cross-flow heat exchanger can typically consist of about 20 unit cells. However, because the unit cells are identical except for edge effects in the outer cells, you can restrict the model to a single unit cell.

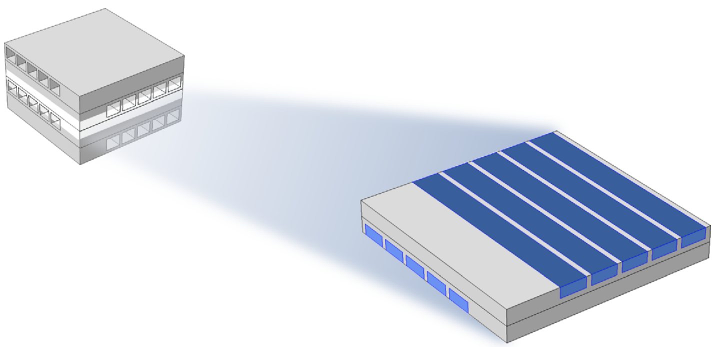


Figure 1: Depiction of the modeled part of the micro-heat exchanger.

Because heat is transferred by convection and conduction, the model uses a Conjugate Heat Transfer interface in the laminar flow regime.

The boundary conditions are insulating for all outer surfaces except for the inlet and outlet boundaries. At the inlets for both cold and hot streams, the temperatures are constant and a laminar inflow profile with an average velocity of 50 mm/s is defined.

At the outlet, the heat transport is dominated by convection, which makes the outflow boundary condition suitable. For the flow field, the model applies the outlet boundary condition with a constant pressure.

As shown in Figure 1, you can take advantage of the model's symmetries to model only half of the channel height. Therefore, the symmetry boundary condition applies to the channels.

Results and Discussion

Figure 2 shows the temperature at the channel walls as well as temperature isosurfaces in the device, which clearly reveal the influence of the convective term.

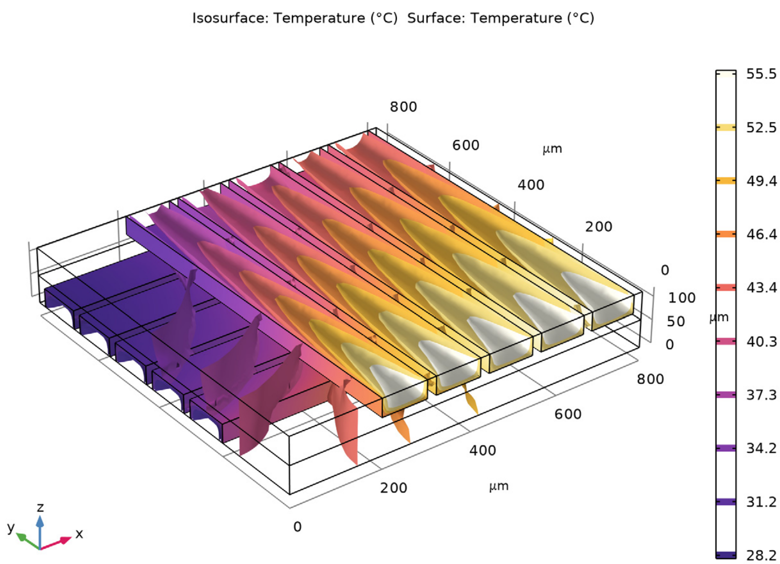


Figure 2: Channel wall temperature and isotherms through the cell geometry.

As can be seen in Figure 3, the temperature differs significantly between the different outlets in both hot and cold streams. This implies that the hot stream is not cooled uniformly.

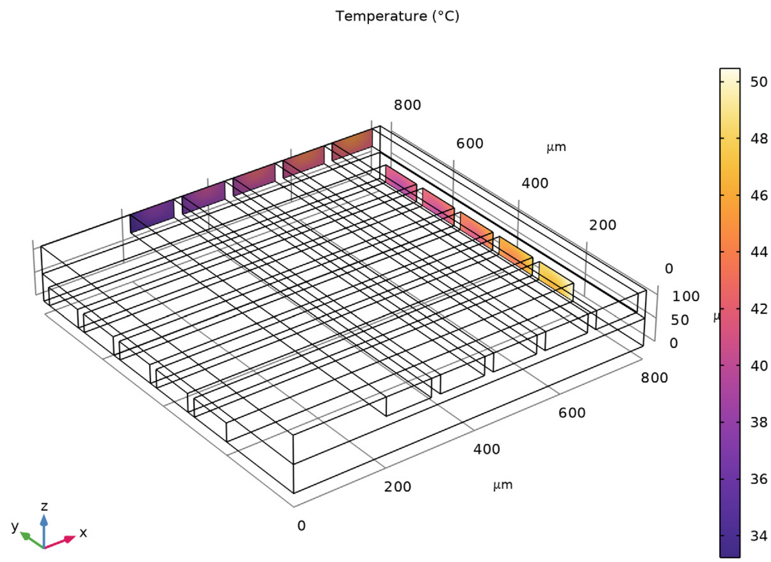


Figure 3: Temperature field at the outlet boundaries.

The flow field in the channels is a typical laminar velocity profile; see Figure 4.

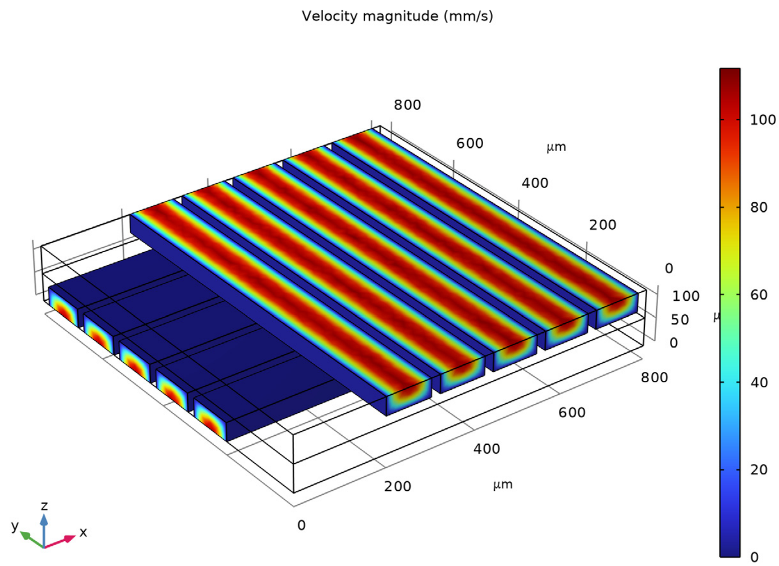


Figure 4: Velocity profile in the channels.

There are several quantities that describe the characteristics and effectiveness of a heat exchanger. The mixing-cup temperature of the fluid leaving the heat exchanger is calculated according to (1.4 in Ref. 2).

$$\langle T\rangle = \frac{\int_{\mathrm{outlet}} \rho C_p T u ds}{\int_{\mathrm{outlet}} \rho C_p u ds} \tag{1}$$

COMSOL Multiphysics provides built-in variables to easily calculate $\langle T\rangle$. At the upper channels, the outlet mixing-cup temperature is about 40.5°C. At the lower channels, a higher value of 43°C is found for the outlet mixing-cup temperature. The maximum pressure drop in the heat exchanger is about 92 Pa.

The overall heat transfer coefficient is another interesting quantity. It is a measure of the performance of a heat exchanger design defined as

$$h_{\mathrm{eq}} = \frac{P}{A(T_{\mathrm{hot}} - T_{\mathrm{cold}})} \tag{2}$$

where P is the total exchanged power and A is the surface area through which P flows. In this model, the value of h_{eq} is about 3150 W/(m^2·K).

References

1. W. Ehrfeld, V. Hessel, and H. Löwe, *Microreactors*, John Wiley & Sons, 2000.

2. P.K. Nag, *Heat and Mass Transfer*, 2nd ed., Tata McGraw Hill, 2007.

Application Library path: `Heat_Transfer_Module/Heat_Exchangers/crossflow_heat_exchanger`

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 **Heat Transfer** > **Conjugate Heat Transfer** > **Laminar Flow**.

3 Click **Add**.

4 Click **Study**.

In this model, the flow will be considered nearly incompressible and independent of temperature variations. Under these assumptions, the **Stationary** 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_cold	300[K]	300 K	Temperature, cold stream
T_hot	330[K]	330 K	Temperature, hot stream
u_avg	50[mm/s]	0.05 m/s	Average inlet velocity

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 **µm**.

First, create the cross section of one unit cell and extrude it.

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 800.

4 In the **Depth** text field, type 800.

5 In the **Height** text field, type 60.

6 Click **Build Selected**.

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 800.

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

5 In the **Height** text field, type 40.

6 Locate the **Position** section. In the **y** text field, type 200.

7 Click **Build Selected**.

Array 1 (arr1)

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

2 Select the object **blk2** 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 `5`.

6 Locate the **Displacement** section. In the **y** text field, type `120`.

7 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

8 In the **New Cumulative Selection** dialog, type `Channels` in the **Name** text field.

9 Click **OK**.

Rotate 1 (rot1)

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

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

3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 In the **Angle** text field, type `180`.

5 Locate the **Point on Axis of Rotation** section. In the **z** text field, type `60`.

6 Locate the **Rotation** section. From the **Axis type** list, choose **Cartesian**.

7 In the **x** text field, type `1`.

8 In the **y** text field, type `1`.

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

Keep the existing unit cell by the following step.

10 Locate the **Input** section. Select the **Keep input objects** checkbox.

11 Click **Build All Objects**.

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

Define several selections that will help throughout the model setup.

DEFINITIONS

Upper Inlets

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Upper Inlets` in the **Label** text field.

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

4 Select Boundaries 41, 48, 55, 62, and 69 only.

Lower Inlets

1 In the **Definitions** toolbar, click **Explicit**.

2. In the **Settings** window for **Explicit**, type `Lower Inlets` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 8, 14, 20, 26, and 32 only.

Upper Outlets

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Upper Outlets` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 44, 51, 58, 65, and 72 only.

Lower Outlets

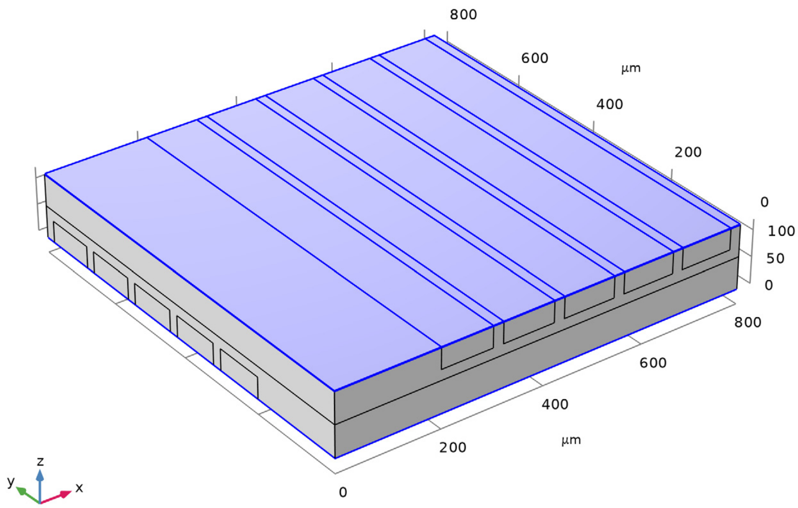
1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Lower Outlets` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 77–81 only.

Symmetry

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Symmetry` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select the **Group by continuous tangent** checkbox.

Select one of the uppermost and lowermost boundaries, which now automatically will select all uppermost and lowermost boundaries thanks to the continuous tangency.



The next selections are needed to evaluate the equivalent heat transfer coefficient.

Outlets

1 In the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `Outlets` in the **Label** text field.

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

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

5 In the **Add** dialog, in the **Selections to add** list, choose **Upper Outlets** and **Lower Outlets**.

6 Click **OK**.

MATERIALS

Define the material properties.

Stainless Steel

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Stainless Steel` in the **Label** text field.

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

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`15[W/(m*K)]`	W/(m·K)	Basic
Density	rho	`7800[kg/m^3]`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`420[J/(kg*K)]`	J/(kg·K)	Basic

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

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

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

5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Water, liquid (mat2)

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

2 From the **Selection** list, choose **Channels**.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

Set the reference temperature to an estimated value of `(T_cold+T_hot)/2` where the flow operates.

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

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

3 In the T_{ref} text field, type `(T_cold+T_hot)/2`.

Fluid 1

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

2 In the **Settings** window for **Fluid**, locate the **Domain Selection** section.

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

Inflow 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inflow**.
2 In the **Settings** window for **Inflow**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Upper Inlets**.
4 Locate the **Upstream Properties** section. In the T_{ustr} text field, type `T_hot`.

Inflow 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Inflow**.
2 In the **Settings** window for **Inflow**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Lower Inlets**.
4 Locate the **Upstream Properties** section. In the T_{ustr} text field, type `T_cold`.

Outflow 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outflow**.
2 In the **Settings** window for **Outflow**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Upper Outlets**.

Outflow 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Outflow**.
2 In the **Settings** window for **Outflow**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Lower Outlets**.

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**.

So far, the boundary conditions for heat transfer have been specified. Continue with the setup of the flow equation.

LAMINAR FLOW (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 **Laminar Flow (spf)**.
2 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.
3 From the **Compressibility** list, choose **Incompressible flow**.

4 Locate the **Domain Selection** section. From the **Selection** list, choose **Channels**.

Because of the different inlet temperatures, the densities for the hot and cold stream vary and produce different velocities when the laminar inflow boundary condition is used. In order to have the same velocity profile on each inlet, define the laminar inflow boundary condition for the hot and cold inlet boundaries separately.

Inlet 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2. In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Upper Inlets**.
4. Locate the **Boundary Condition** section. From the list, choose **Fully developed flow**.
5. Locate the **Fully Developed Flow** section. In the U_{av} text field, type `u_avg`.

Inlet 2

1. In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2. In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Lower Inlets**.
4. Locate the **Boundary Condition** section. From the list, choose **Fully developed flow**.
5. Locate the **Fully Developed Flow** section. In the U_{av} text field, type `u_avg`.

Outlet 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
2. In the **Settings** window for **Outlet**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Upper Outlets**.

Outlet 2

1. In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
2. In the **Settings** window for **Outlet**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Lower Outlets**.

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**.

Next, create a selection for the channel walls. This will be used later for postprocessing.

Wall 1

1. In the **Model Builder** window, click **Wall 1**.
2. In the **Settings** window for **Wall**, locate the **Boundary Selection** section.
3. Click **Create Selection**.
4. In the **Create Selection** dialog, type `Channel Walls` in the **Selection name** text field.
5. Click **OK**.

After solving the model, the equivalent heat transfer coefficient is evaluated according to Equation 2. To do so, define the following nonlocal coupling.

MULTIPHYSICS

Nonisothermal Flow 1 (nitf1)

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Multiphysics** click **Nonisothermal Flow 1 (nitf1)**.
2. In the **Settings** window for **Nonisothermal Flow**, locate the **Material Properties** section.
3. Select the **Boussinesq approximation** checkbox.

DEFINITIONS

Average on Upper Channel Walls

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.
2. In the **Settings** window for **Average**, type `Average on Upper Channel Walls` in the **Label** text field.
3. Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 40, 42, 45, 47, 49, 52, 54, 56, 59, 61, 63, 66, 68, 70, and 73 only.

   To select more easily these boundaries, use the **Paste** button and insert the list of numbers above in the **Paste Selection** dialog.

STUDY 1

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

RESULTS

Change the unit of the temperature results to degrees Celsius.

Preferred Units 1

1. In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

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

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **General** > **Temperature (K)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Temperature	K	°C

8 Click **Apply**.

Temperature (ht)

COMSOL Multiphysics automatically creates four default plots: a volume plot for the temperature field, a multislice plot for the velocity field, a surface plot for the pressure field, and a plot showing both the temperature of solids and walls, and the velocity in the fluid. Additional plots, such as the isothermal contours are available in the **Result Templates**. The latter will be used to create the plot shown in Figure 2.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Heat Transfer in Solids and Fluids** > **Isothermal Contours (ht)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Isothermal Contours (ht)** node.

2 Right-click **Isothermal Contours (ht)** and choose **Surface**.

3 In the **Settings** window for **Surface**, click to expand the **Inherit Style** section.

4 From the **Plot** list, choose **Isosurface 1**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

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

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

4 In the **Isothermal Contours (ht)** toolbar, click **Plot**.

To visualize the velocity field as in Figure 4, follow the steps below:

Multislice 1

1 In the **Model Builder** window, expand the **Velocity (spf)** node.

2 Right-click **Multislice 1** and choose **Delete**.

Surface 1

1 In the **Velocity (spf)** toolbar, click **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Laminar Flow** > **Velocity and pressure** > **spf.U - Velocity magnitude - m/s**.

3 Locate the **Expression** section. From the **Unit** list, choose **mm/s**.

4 In the **Velocity (spf)** toolbar, click **Plot**.

Next, show the temperature on the outlet boundaries only, as in Figure 3.

Outlet Temperature

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

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

Surface 1

1 In the **Outlet Temperature** toolbar, click **Surface**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **HeatCameraLight**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

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

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

4 In the **Outlet Temperature** toolbar, click **Plot**.

Mixing-Cup Temperatures

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Mixing-Cup Temperatures` in the **Label** text field.

3 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Heat Transfer in Solids and Fluids** > **Temperature** > **Weighted average temperature** > **ht.ofl1.Tave - Weighted average temperature - K**.

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

Expression	Unit	Description
`ht.ofl1.Tave`	`°C`	`Weighted average temperature (Upper Outlets)`
`ht.ofl2.Tave`	`°C`	`Weighted average temperature (Lower Outlets)`

5 Click **Evaluate**.

TABLE 1

1 Go to the **Table 1** window.

The mixing-cup temperature at upper outlets is about 40.5°C and at the lower outlets about 43°C.

RESULTS

To calculate the maximum pressure drop proceed as follows:

Maximum Pressure Drop

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

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

3 From the **Selection** list, choose **All boundaries**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Laminar Flow** > **Velocity and pressure** > **p - Pressure - Pa**.

5 In the **Label** text field, type `Maximum Pressure Drop`.

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

Expression	Unit	Description
`p`	`Pa`	`Maximum pressure drop`

7 Click **Evaluate**.

TABLE 2

1 Go to the **Table 2** window.

The maximum pressure is about 92 Pa. The minimum pressure is defined by the outlet boundary conditions and is zero. Thus, the maximum pressure drop is also 92 Pa.

Now, evaluate the equivalent heat transfer coefficient as defined in Equation 2. You can use the integration operators defined previously in **Component 1** > **Definitions**.

RESULTS

Heat Transfer Coefficient

1 In the **Results** toolbar, click 8.5 **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`aveop1(ht.ntflux)/(T_hot-T_cold)`	`W/(m^2*K)`	`Heat transfer coefficient`

4 In the **Label** text field, type `Heat Transfer Coefficient`.

5 Click **Evaluate**.

TABLE 3

1 Go to the **Table 3** window.

The equivalent heat transfer coefficient is about 3143 W/(m^2·K).