

## Heat Exchanger Efficiency

---

Flow Simulation can be used to study the fluid flow and heat transfer for a wide variety of engineering equipment. In this example we use Flow Simulation to determine the efficiency of a counterflow heat exchanger and to observe the temperature and flow patterns inside of it. With Flow Simulation the determination of heat exchanger efficiency is straightforward and by investigating the flow and temperature patterns, the design engineer can gain insight into the physical processes involved thus giving guidance for improvements to the design.

A convenient measure of heat exchanger performance is its “efficiency” in transferring a given amount of heat from one fluid at higher temperature to another fluid at lower temperature. The efficiency can be determined if the temperatures at all flow openings are known. In Flow Simulation the temperatures at the fluid inlets are specified and the temperatures at the outlets can be easily determined. Heat exchanger efficiency is defined as follows:

$$\varepsilon = \frac{\text{actual heat transfer}}{\text{maximum possible heat transfer}}$$

The actual heat transfer can be calculated as either the energy lost by the hot fluid or the energy gained by the cold fluid. The maximum possible heat transfer is attained if one of the fluids was to undergo a temperature change equal to the maximum temperature difference present in the exchanger, which is the difference in the inlet temperatures of the hot and cold fluids, respectively:  $(T_{hot}^{inlet} - T_{cold}^{inlet})$ . Thus, the efficiency of a counterflow

heat exchanger is defined as follows:  $\varepsilon = \frac{T_{hot}^{inlet} - T_{hot}^{outlet}}{T_{hot}^{inlet} - T_{cold}^{inlet}}$  - if hot fluid capacity rate is less

than cold fluid capacity rate or  $\varepsilon = \frac{T_{cold}^{outlet} - T_{cold}^{inlet}}{T_{hot}^{inlet} - T_{cold}^{inlet}}$  - if hot fluid capacity rate is more than

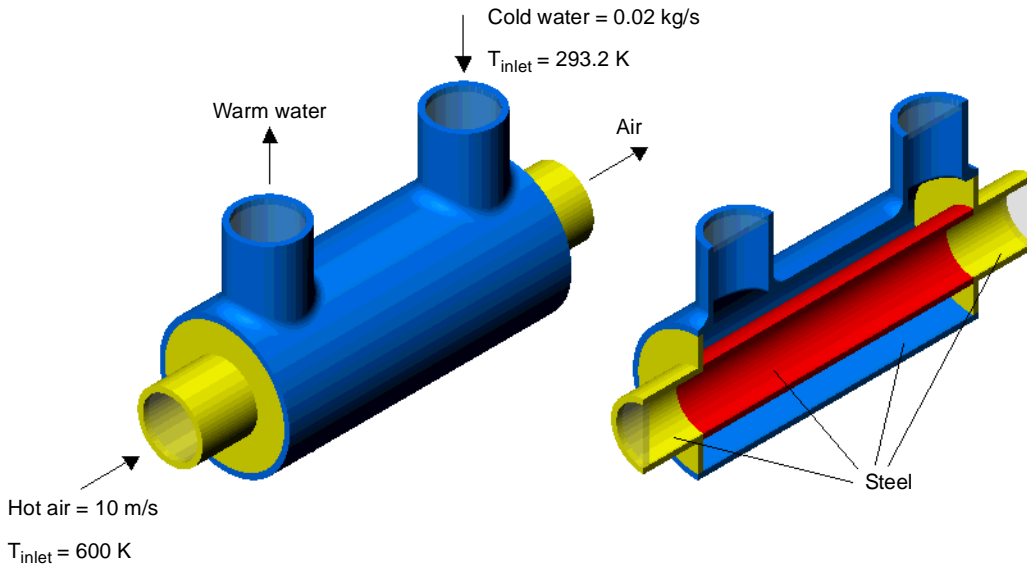
cold fluid capacity rate, where the capacity rate is the product of the mass flow and the specific heat capacity:  $C = \dot{m}C$  (Ref.2)

## Chapter 6 Heat Exchanger Efficiency

The goal of the project is to calculate the efficiency of the counterflow heat exchanger. Also, we will determine the average temperature of the heat exchanger central tube's wall. The obtained wall temperature value can be further used for structural and fatigue analysis.

### Open the Model

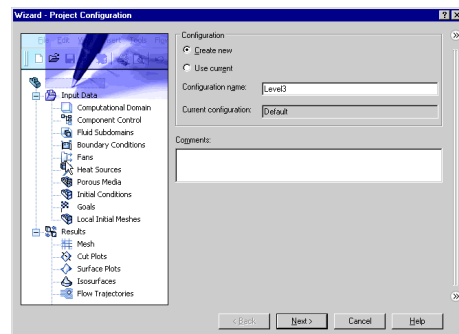
Click **File, Open**. In the **Open** dialog box, browse to the **Heat Exchanger.SLDASM** assembly located in the **Tutorial 3 - Heat Exchanger** folder and click **Open** (or double-click the assembly). Alternatively, you can drag and drop the **Heat Exchanger.SLDASM** file to an empty area of SolidWorks window.



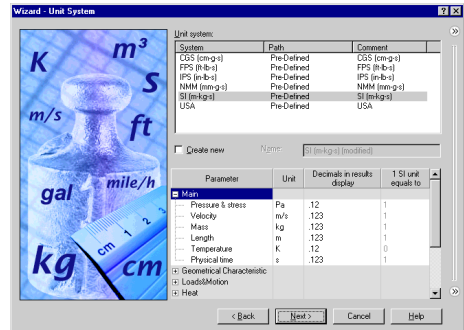
### Creating a Project

- 1 Click **Flow Simulation, Project, Wizard**.
- 2 Select **Create new**. In the **Configuration name** box type **Level 3**. The 'Level 3' name was chosen because this problem will be calculated using **Result Resolution** level 3.

Click **Next**.



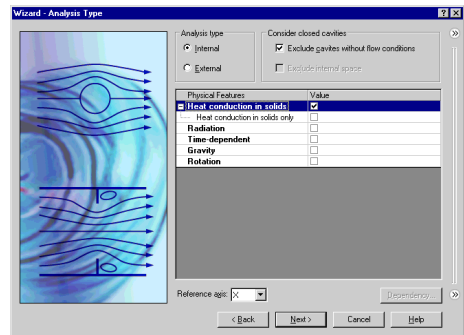
- In the **Units** dialog box select the desired system of units for both input and output (results). For this project we will use the International System **SI** by default.



Click **Next**.

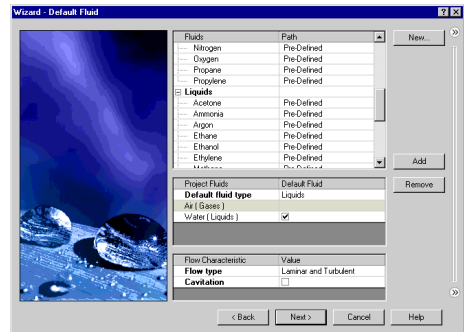
- In the **Analysis Type** dialog box among **Physical features** select **Heat conduction in solids**.

By default, Flow Simulation will consider heat conduction not in solids, but only within the fluid and between the walls and the fluid (i.e., convection). Selecting the **Heat conduction in solids** option enables the combination of convection and conduction heat transfer, known as conjugate heat transfer. In this project we will analyze heat transfer between the fluids through the model walls, as well as inside the solids.



Click **Next**.

- Since two fluids (water and air) are used in this project, expand the **Liquids** folder and add **Water** and then expand the **Gases** folder and add **Air** to the **Project Fluids** list. Check that the **Default fluid type** is **Liquids**.



Click **Next**.

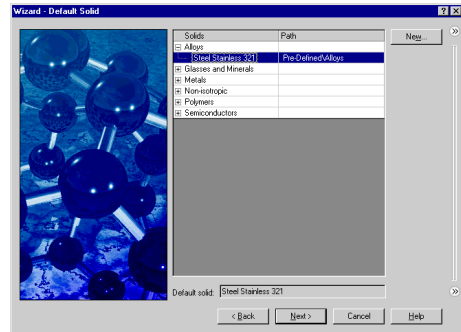
- Since we have selected the **Heat conduction in solids** option at the **Analysis Type** step of the Wizard, the **Default Solid** dialog box appears. In this dialog you specify the default solid material applied to all solid components. To assign a different material to a particular assembly component you need to create a **Solid Material** condition for this component.


## Chapter 6 Heat Exchanger Efficiency

If the solid material you wish to specify as the default is not available in the **Solids** table, you can click **New** and define a new substance in the **Engineering Database**. The tube and its cooler in this project are made of stainless steel.


Expand the **Alloys** folder and click **Steel Stainless 321** to make it the default solid material.

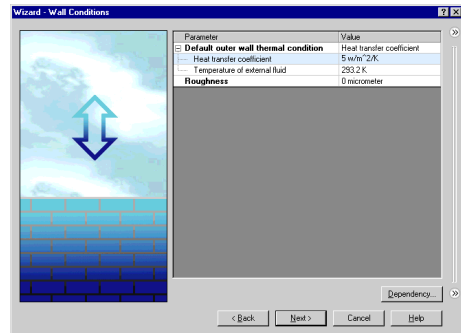
Click **Next**.



 If a component has been previously assigned a solid material by the SolidWorks' Materials Editor, you can import this material into Flow Simulation and apply this solid material to the component in the Flow Simulation project by using the **Insert Material from Model** option accessible under **Flow Simulation, Tools**.

- 7 In the **Wall Condition** dialog box, select **Heat transfer coefficient** as **Default outer wall thermal condition**.

 This condition allows you to define the heat transfer from the outer model walls to an external fluid (not modeled) by specifying the reference fluid temperature and the heat transfer coefficient value.

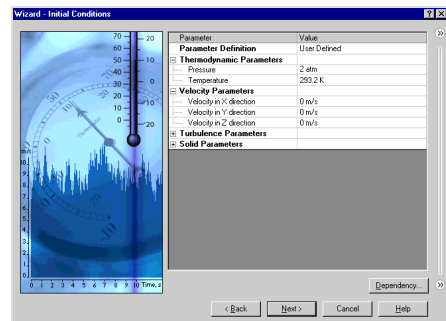


Set the **Heat transfer coefficient** value to  $5 \text{ W/m}^2/\text{K}$ .

In this project we do not consider walls roughness.

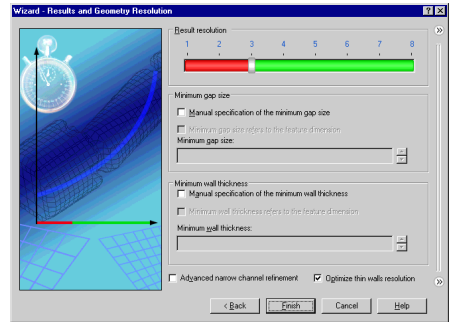
Click **Next**.

- 8 In the **Initial Conditions** dialog box under **Thermodynamics parameters** enter  $2 \text{ atm}$  in the **Value** cell for the **Pressure** parameter. Flow Simulation automatically converts the entered value to the selected system of units.



Click **Next** accepting the default values of other parameters for initial conditions.



- In the **Results and Geometry Resolution** dialog box we accept the default result resolution level 3 and the default minimum gap size and minimum wall thickness.




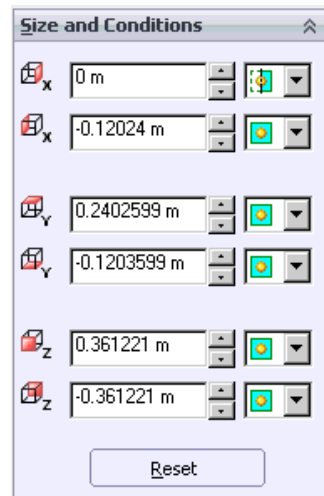
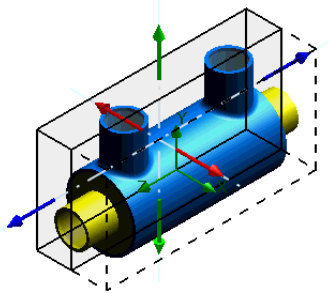
Click **Finish**.


After finishing the **Wizard** you will complete the project definition by using the Flow Simulation Analysis tree. First of all you can take advantage of the symmetry of the heat exchanger to reduce the CPU time and memory required for the calculation. Since this model is symmetric, it is possible to “cut” the model in half and use a symmetry boundary condition at the plane of symmetry. This procedure is not required, but is recommended for efficient analyses.

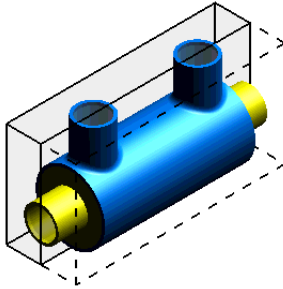
## Symmetry Condition

- In the Flow Simulation Analysis tree, expand the **Input Data** item.
- Right-click the **Computational Domain** icon and select **Edit Definition**.
- Under **Size and Conditions** select the **Symmetry**  condition at the **X max** boundary and type 0 in the **X max**  edit-box

 *To resize the domain manually, select the **Computational Domain** item in the Flow Simulation analysis tree, and in the graphics area click and drag the arrow handles at the sides of the computational domain frame to the desired positions, then adjust the exact coordinates in the appearing callouts..*




- 4 Click **OK** .

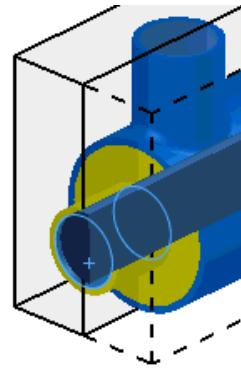


## Specifying a Fluid Subdomain

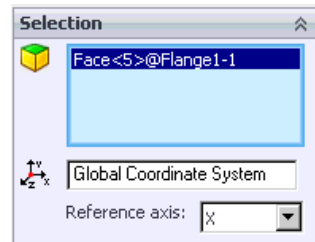
Since we have selected **Liquids** as the **Default fluid type** and **Water** as the **Default fluid** in the Wizard, we need to specify another fluid type and select another fluid (air) for the fluid region inside the tube through which the hot air flows. We can do this by creating a **Fluid Subdomain**. When defining a **Fluid Subdomain** parameters we will specify **Gas** as the fluid type for the selected region, **Air** as the fluid and the initial temperature of 600 K and flow velocity of 10 m/s as the initial conditions in the selected fluid region.

- 1 Click **Flow Simulation, Insert, Fluid Subdomain**.
- 2 Select the **Flange 1** inner face (in contact with the fluid). Immediately the fluid subdomain you are going to create is displayed in the graphics area as a body of blue color.

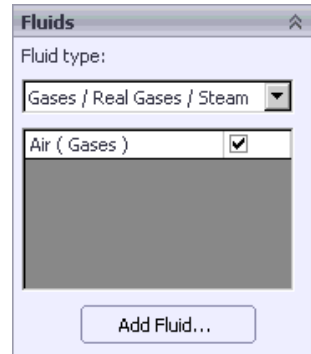
 *To specify the fluid subdomain within a fluid region we must specify this condition on the one of the faces lying on the region's boundary - i.e. on the boundary between solid and fluid substances. The fluid subdomain specified on the region's boundary will be applied to the entire fluid region. You may check if the region to apply a fluid subdomain is selected properly by looking at the fluid subdomain visualization in the graphics area.*



- 3 Accept the default **Coordinate System**  and the **Reference axis**.

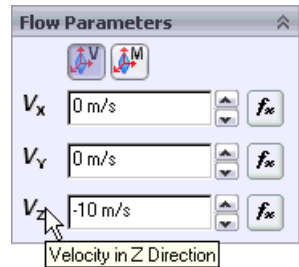


- 4 In the **Fluid type** list select **Gases / Real Gases / Steam**. Because **Air** was defined in the Wizard as one of the **Project fluids** and you have selected the appropriate fluid type, it appears as the fluid assigned to the fluid subdomain.



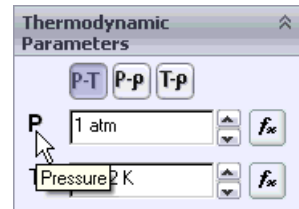
- In the **Fluids** group box, *Flow Simulation* allows you to specify the fluid type and/or fluids to be assigned for the fluid subdomain as well as flow characteristics, depending on the selected fluid type.

- 5 Under **Flow Parameters** in the **Velocity in Z Direction**  $V_z$  box enter -10.



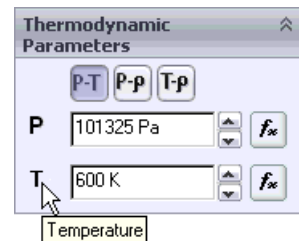
- Flow Simulation* allows you to specify initial flow parameters, initial thermodynamic parameters, and initial turbulence parameters (after a face to apply the Fluid Subdomain is selected). These settings are applied to the specified fluid subdomain.

- 6 Under **Thermodynamic parameters** in the **Static Pressure** **P** box enter 1 atm. *Flow Simulation* automatically converts the entered value to the selected system of units.



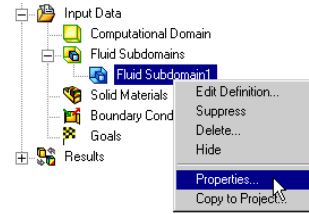
- 7 Under **Thermodynamic parameters** in the **Temperature** **T** box enter 600.

These initial conditions are not necessary and the parameters of the hot air inlet flow are defined by the boundary condition, but we specify them to improve calculation convergence.



- 8 Click **OK** . The new **Fluid Subdomain 1** item appears in the Analysis tree.

- To easily identify the specified condition you can give a more descriptive name for the **Fluid Subdomain 1** item. Right-click the **Fluid Subdomain 1** item and select **Properties**. In the **Name** box type **Hot Air** and click **OK**.



You can also click-pause-click an item to rename it directly in the Flow Simulation Analysis tree.



## Specifying Boundary Conditions

- Right-click the **Boundary Conditions** icon in the Flow Simulation Analysis tree and select **Insert Boundary Condition**. The **Boundary Condition** dialog box appears.

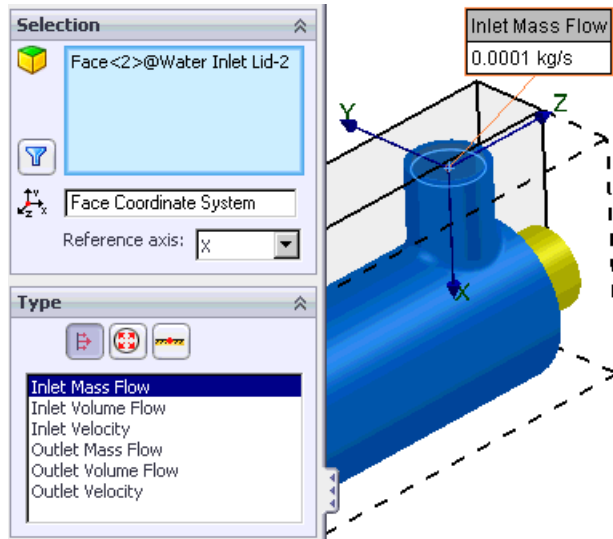
- Select the **Water Inlet Lid** inner face (in contact with the fluid).

The selected face appears in the **Faces to Apply the**

**Boundary Condition** list.

- Accept the default **Inlet Mass Flow** condition and the default **Coordinate**

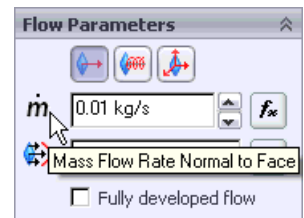
**System** and **Reference axis**.



If you specify a new boundary condition, a callout showing the name of the condition and the default values of the condition parameters appears in the graphics area. You can double-click the callout to open the quick-edit dialog.

- Click the **Mass Flow Rate Normal to Face** box and set its value equal to  $0.01 \text{ kg/s}$ . Since the symmetry plane halves the opening, we need to specify a half of the actual mass flow rate.

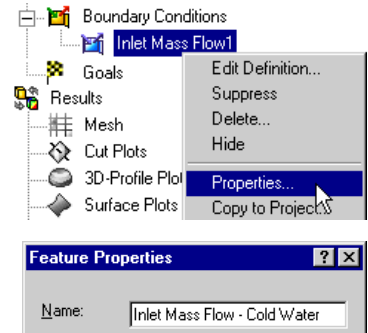
- Click **OK**. The new **Inlet Mass Flow 1** item appears in the Analysis tree.






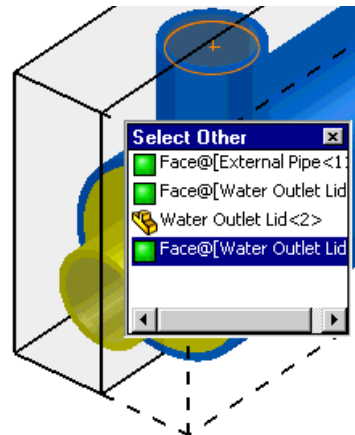
This boundary condition specifies that water enters the steel jacket of the heat exchanger at a mass flow rate of 0.02 kg/s and temperature of 293.2 K.


- 6 Rename the **Inlet Mass Flow 1** item to Inlet Mass Flow - Cold Water.

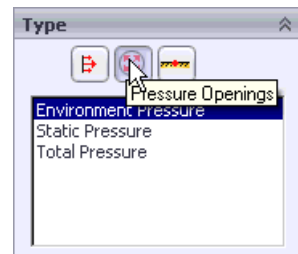


Next, specify the water outlet **Environment Pressure** condition.

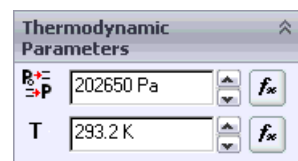
- 7 In the Flow Simulation Analysis tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.
- 8 Select the **Water Outlet Lid** inner face (in contact with the fluid). The selected face appears in the **Faces to Apply the Boundary Condition**  list.



- 9 Click **Pressure Openings**  and in the **Type of Boundary Condition** list select the **Environment Pressure** item.

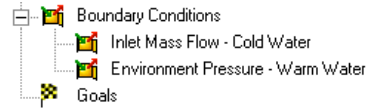


- 10 Accept the value of **Environment Pressure**  $P_0 = P$  (202650 Pa), taken from the value specified at the **Initial Conditions** step of the **Wizard**, and the default values of **Temperature**  $T$  (293.2 K) and all other parameters.



11 Click **OK** . The new **Environment Pressure 1** item appears in the Flow Simulation Analysis tree.



12 Rename the **Environment Pressure 1** item to Environment Pressure - Warm Water.



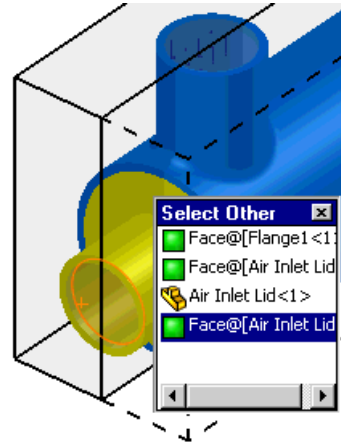
Next we will specify the boundary conditions for the hot air flow.


13 In the Flow Simulation Analysis tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.

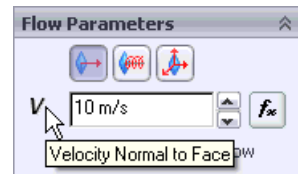
14 Select the **Air Inlet Lid** inner face (in contact with the fluid).

The selected face appears in the **Faces to Apply the Boundary Condition**  list. Accept the default **Coordinate System**  and **Reference axis**.

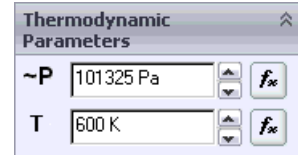
15 Under **Type** select the **Inlet Velocity** condition.




16 Click the **Velocity Normal to Face**  box and set its value equal to 10 (type the value, the units will appear automatically).



17 Expand the **Thermodynamic Parameters** item. The default temperature value is equal to the value specified as the initial temperature of air in the **Fluid Subdomain** dialog box. We accept this value.



18 Click **OK** . The new **Inlet Velocity 1** item appears in the Analysis tree.

This boundary condition specifies that air enters the tube at the velocity of 10 m/s and temperature of 600 K.

19 Rename the **Inlet Velocity 1** item to Inlet Velocity - Hot Air.

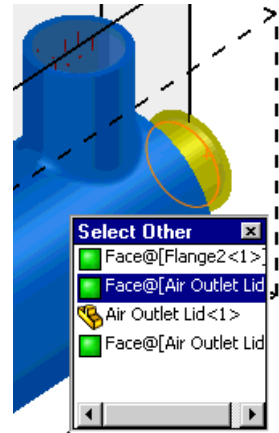
Next specify the air outlet **Environment Pressure** condition.


**20** In the Flow Simulation Analysis tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**. The **Boundary Condition** dialog box appears.

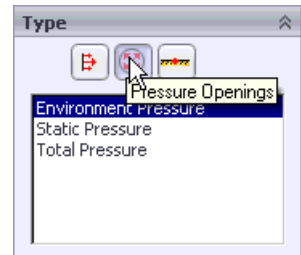
**21** Select the **Air Outlet Lid** inner face (in contact with the fluid).


The selected face appears in the **Faces to Apply the**

**Boundary Condition**  list.

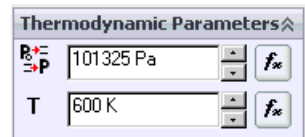


**22** Click **Pressure Openings**  and in the **Type of Boundary Condition** list select the **Environment Pressure** item.

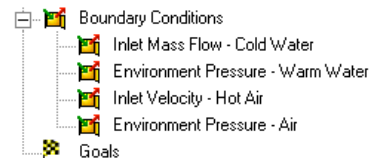


**23** Check the values of **Environment Pressure**  (101325 Pa) and **Temperature** **T** (600 K). If they are different, correct them. Accept the default values of other parameters.

Click **OK** .



**24** Rename the new item **Environment Pressure 1** to **Environment Pressure - Air**.

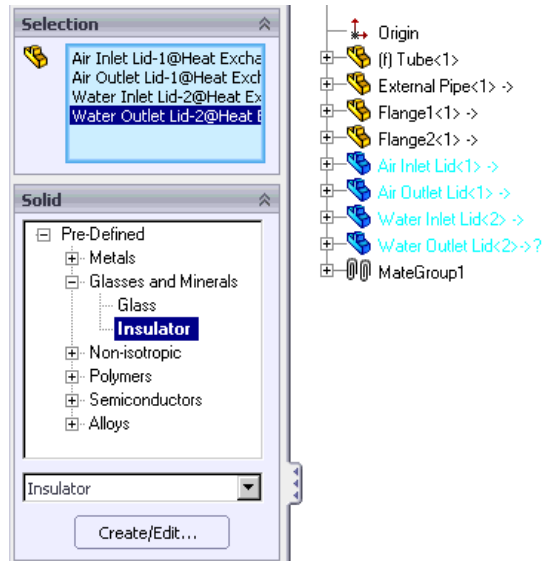



This project involving analysis of heat conduction in solids. Therefore, you must specify the solid materials for the model's components and the initial solid temperature.

## Specifying Solid Materials

Notice that the auxiliary lids on the openings are solid. Since the material for the lids is the default stainless steel, they will have an influence on the heat transfer. You cannot suppress or disable them in the **Component Control** dialog box, because boundary conditions must be specified on solid surfaces in contact with the fluid region. However, you can exclude the lids from the heat conduction analysis by specifying the lids as insulators.


- 1 Right-click the **Solid Materials** icon and select **Insert Solid Material**.
- 2 In the flyout FeatureManager design tree, select all the lid components. As you select the lids, their names appear in the **Components to Apply the Solid Material** list.
- 3 In the **Solid** group box expand the list of **Pre-Defined** materials and select the **Insulator** solid in the **Glasses & Minerals** folder.
- 4 Click **OK** . Now all auxiliary lids are defined as insulators.

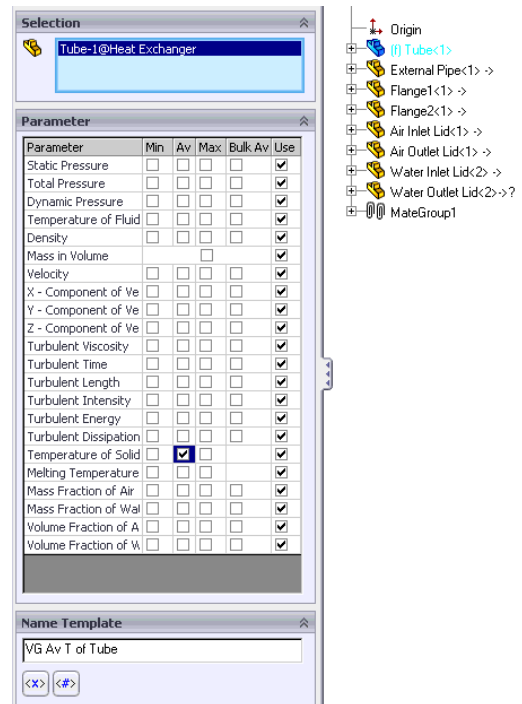


 *The thermal conductivity of the **Insulator** substance is zero. Hence there is no heat transferred through an insulator.*

- 5 Rename the **Insulator Solid Material 1** item to **Insulators**.

## Specifying a Volume Goal

- 1 In the Flow Simulation Analysis tree, right-click the **Goals** icon and select **Insert Volume Goals**.
- 2 In the Flyout FeatureManager Design tree select the **Tube** part.
- 3 In the **Parameter** table select the **Av** check box in the **Temperature of Solid** row. Accept the selected **Use for Conv.** check box to use this goal for convergence control.
- 4 In the **Name template** type VG Av T of Tube.
- 5 Click **OK** .



## Running the Calculation

- 1 Click **Flow Simulation, Solve, Run**. The **Run** dialog box appears.
- 2 Click **Run**.

After the calculation finishes you can obtain the temperature of interest by creating the corresponding **Goal Plot**.

## Viewing the Goals

In addition to using the Flow Simulation Analysis tree you can use Flow Simulation Toolbars and SolidWorks CommandManager to get fast and easy access to the most frequently used Flow Simulation features. Toolbars and SolidWorks CommandManager are very convenient for displaying results.

Click **View, Toolbars, Flow Simulation Results**. The **Flow Simulation Results** toolbar appears.



Click **View, Toolbars, Flow Simulation Results Features**. The **Flow Simulation Results Features** toolbar appears.





Click **View, Toolbars, Flow Simulation Display**. The **Flow Simulation Display** toolbar appears.



The SolidWorks CommandManager is a dynamically-updated, context-sensitive toolbar, which allows you to save space for the graphics area and access all toolbar buttons from one location. The tabs below the CommandManager is used to select a specific group of commands and features to make their toolbar buttons available in the CommandManager. To get access to the Flow Simulation commands and features, click the **Flow Simulation** tab of the CommandManager.



If you wish, you may hide the Flow Simulation toolbars to save the space for the graphics area, since all necessary commands are available in the CommandManager. To hide a toolbar, click its name again in the **View, Toolbars** menu.

- 1 Click **Generate goal plot**  on the **Results Main** toolbar or CommandManager. The **Goal Plot** dialog box appears.
- 2 Select the goals of the project (actually, in our case there is only one goal) .
- 3 Click **OK**  . The **Goals1** Excel workbook is created.



You can view the average temperature of the tube on the **Summary** sheet.

### **heat exchanger.SLDASM [level 3]**

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress (%)	Use In Convergence
VG Av T of Tube	[K]	343.2771368	342.7244032	341.792912	343.2771368	100	Yes

Iterations: 40

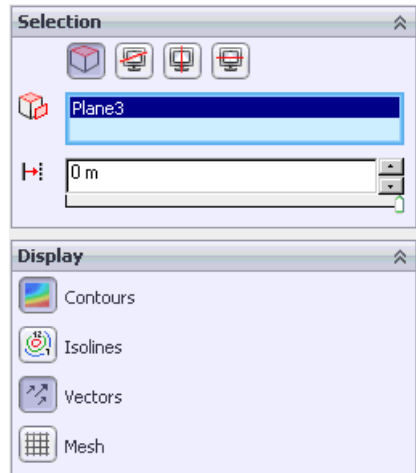
Analysis interval: 20


## Creating a Cut Plot

1 Click **Cut Plot**  on the **Flow Simulation Results Features** toolbar. The **Cut Plot** dialog box appears.



2 In the flyout FeatureManager design tree select **Plane3**.

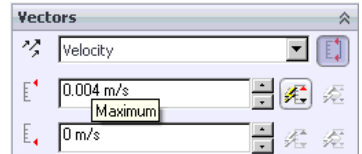
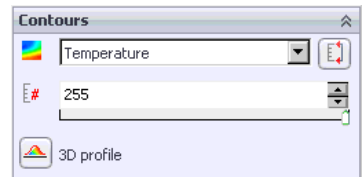
3 In the **Cut Plot** dialog, in addition to displaying **Contours** , select **Vectors** .




4 Under **Contours** specify the parameter which values to show at the contour plot. In the **Parameter**  box, select **Temperature**.


5 Using the slider set the **Number of Levels** to maximum.

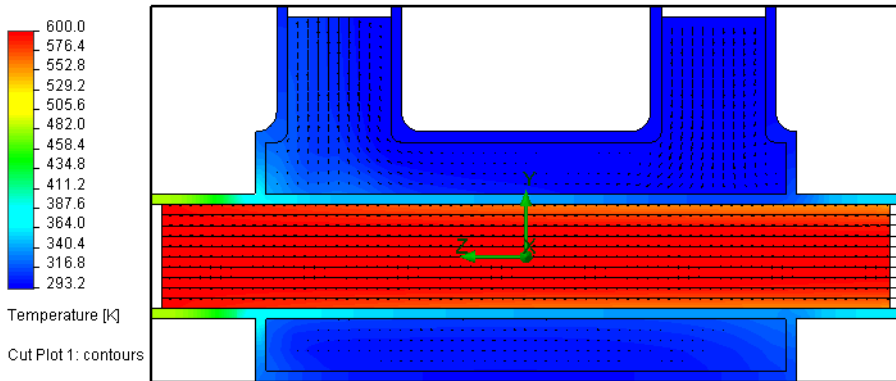
6 Under **Vectors** click the **Adjust Minimum and Maximum**  and change the **Maximum**  velocity to 0.004 m/s.



7 Click **OK** . The cut plot is created but the model overlaps it. Click the **Right** view on the **Standard Views** toolbar.



8 Click **Geometry**  on the **Flow Simulation Display** toolbar to hide the model.

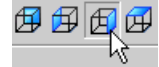


Let us now display the flow development inside the exchanger.

Flow Simulation allows you to display results in all four possible panes of the SolidWorks graphics area. Moreover, for each pane you can specify different View Settings.

9 Click **Window, Viewport, Two View - Horizontal**.

10 To restore the view orientation in the top pane, click **Right** view on the **Standard Views** toolbar.




11 Click the bottom pane and select the **Isometric** view on the **Standard Views** toolbar.



The gray contour around the pane border indicates that the view is active.

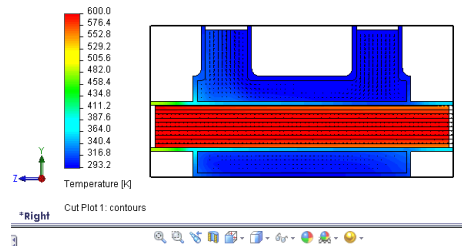
12 On the **Flow Simulation Display** toolbar,

click **Geometry** , then on the **View** toolbar click **Hidden Lines**

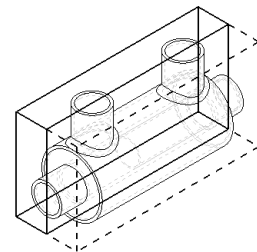
**Visible**  to show the face outlines.

Click the top pane and set the same display mode for it by clicking **Hidden**

**Lines Visible**  again.




To see how the water flows inside the exchanger we will display the **Flow Trajectories**.

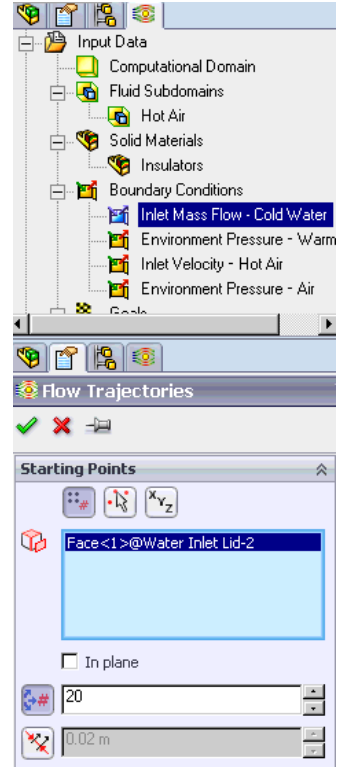






Click the bottom pane to make it the active pane.

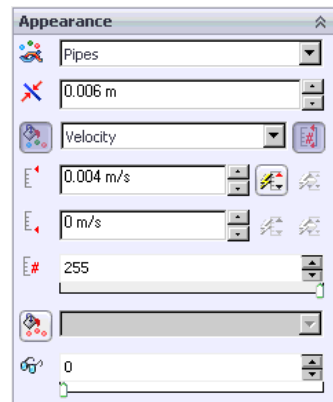


## Displaying Flow Trajectories



- 1 Click **Flow Trajectories**  on the **Flow Simulation Results Features** toolbar. The **Flow Trajectories** dialog appears.
- 2 Click the Flow Simulation Analysis tree tab and select the **Inlet Mass Flow – Cold Water** item.  
This selects the inner face of the **Water Inlet Lid** to place the trajectories start points on it.

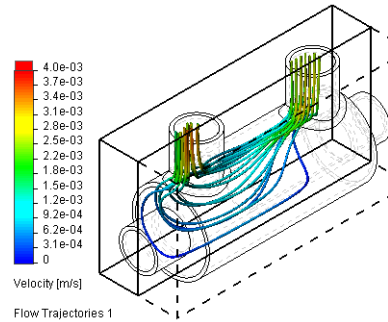



- 3 Under **Appearance**, in the **Color by Parameter**  list, select **Velocity**.
- 4 Click the **Adjust Minimum/Maximum and Number of Levels**  and set **Maximum**  velocity to 0.004 m/s.
- 5 Click **OK** . Trajectories are created and displayed.




By default the trajectories are colored in accordance with the distribution of the parameter specified in the

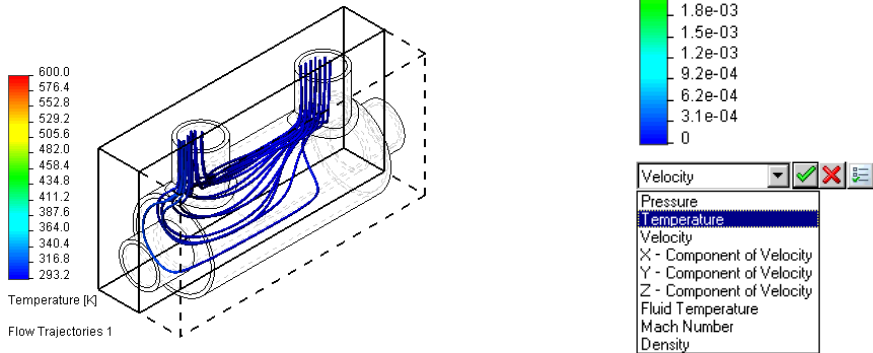
**Color by Parameter**  list. Since you specified velocity, the trajectory color corresponds to the velocity value. To define a fixed color for flow trajectories click **Color**  and select a desired color.



 Notice that in the top pane the temperature contours are still displayed.

Since we are more interested in the temperature distribution let us color the trajectories with the values of temperature.


- 1 In the velocity palette bar click the caption with the name of the current visualization parameter and select **Temperature** in a dropdown list.
- 2 Click . Immediately the trajectories are updated.




The water temperature range is less than the default overall (**Global**) range (293 – 600), so all of the trajectories are the same blue color. To get more information about the temperature distribution in water you can manually specify the range of interest.

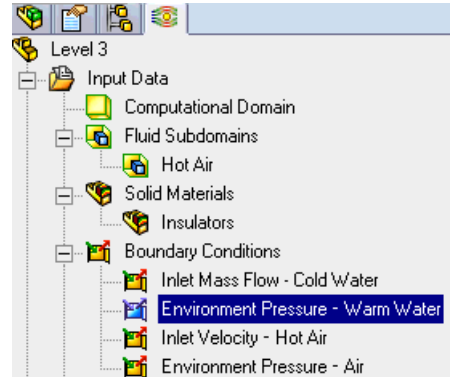
Let us display temperatures in the range of **inlet-outlet** water temperature.

The water minimum temperature value is close to 293 K. Let us obtain the values of air and water temperatures at outlets using Surface Parameters. You will need these values to calculate the heat exchanger efficiency and determine the appropriate temperature range for flow trajectories visualization.

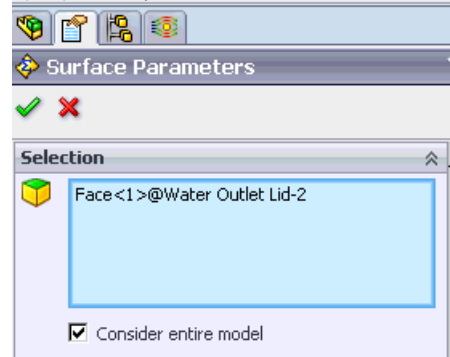
 **Surface Parameters** allows you to display parameter values (minimum, maximum, average and integral) calculated over the specified surface. All parameters are divided into two categories: *Local and Integral*. For local parameters (pressure, temperature, velocity etc.) the maximum, minimum and average values are evaluated.

# Computation of Surface Parameters

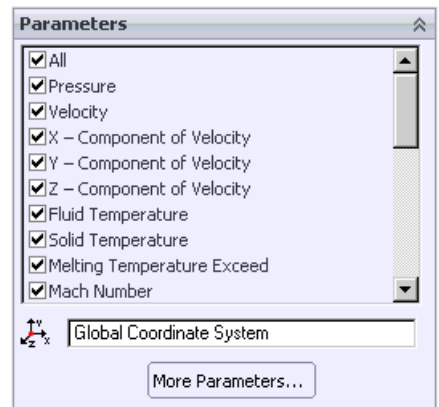
- 1 Click **Surface Parameters**  on the **Flow Simulation Results Features** toolbar. The **Surface Parameters** dialog appears.
- 2 Click the Flow Simulation Analysis tree tab and select the **Environment Pressure - Warm Water** item to select the inner face of the **Water Outlet Lid**.



- 3 Select **Consider entire model** to take into account the **Symmetry** condition to see the values of parameters as if the entire model, not a half of it, was calculated. This is especially convenient for such parameters as mass and volume flow.



- 4 Under **Parameters**, select **All**.
- 5 Click **Show**. The calculated parameters values are displayed on the pane at the bottom of the screen. Local parameters are displayed at the left side of the bottom pane, while integral parameters are displayed at the right side.



- 6 Take a look at the local parameters.


Local Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	202650	202650	202650	202650
Density [kg/m <sup>3</sup> ]	990.994	997.357	995.695	995.534
Velocity [m/s]	0.000273491	0.00374912	0.00246499	0.00300914
X - Component of Velocity [m/s]	-0.000169079	0.000170163	-2.83344e-005	-4.91866e-005
Y - Component of Velocity [m/s]	0.000231458	0.00374815	0.00245914	0.00300585
Z - Component of Velocity [m/s]	-0.000177123	0.00014593	1.36949e-005	4.30211e-005
Mach Number [-]	0	0	0	0
Surface Heat Flux [W/m <sup>2</sup> ]	-0	-0	0	0
Fluid Temperature [K]	294.008	314.304	299.778	300.357
Solid Temperature [K]	296.155	305.591	300.171	300.765
Melting Temperature Exceed [K]	-1386.99	-1377.56	-1383.09	-1382.39

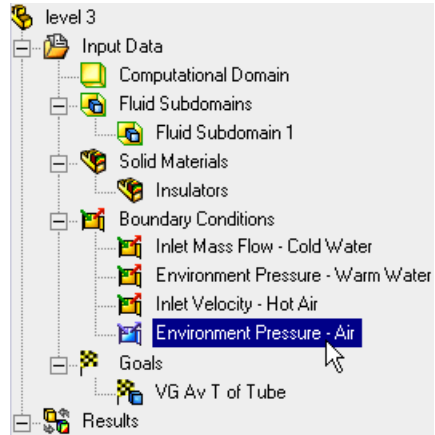
## Chapter 6 Heat Exchanger Efficiency

You can see that the average water temperature at the outlet is about 300 K.

Now let us determine the temperature of air at the outlet.

7 Click the **Environment Pressure - Air** item to select the inner face of the **Air Outlet Lid**.

8 At the bottom pane, click **Refresh** .



9 Look at the local parameters at the left side of the bottom pane.


You can see that the average air temperature at the outlet is about 585 K.

Local Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	101325	101325	101325	101325
Density [kg/m <sup>3</sup> ]	0.588229	0.667233	0.604387	0.602515
Velocity [m/s]	6.33569	10.1413	9.72103	9.7692
X - Component of Velocity [m/s]	-0.170256	0.164865	-0.0389458	-0.0381724
Y - Component of Velocity [m/s]	-0.164565	0.169146	0.00122779	0.00110737
Z - Component of Velocity [m/s]	-10.1412	-8.33412	-9.7206	-9.76881
Mach Number [ ]	0.0181421	0.0208413	0.0202105	0.0202842
Surface Heat Flux [W/m <sup>2</sup> ]	-0	-0	0	0
Fluid Temperature [K]	528.939	599.981	584.611	586.315
Solid Temperature [K]	357.211	361.034	359.134	359.125
Melting Temperature Exceed [K]	-1325.94	-1322.12	-1324.02	-1324.02

10 The values of integral parameters are displayed at the right side of the bottom pane. You can see that the mass flow rate of air is 0.046 kg/s.

This value is calculated with the **Consider entire model** option selected, i.e. taking into account the **Symmetry** condition.

Integral Parameter	Value	X-component	Y-component	Z-component
Heat Transfer Rate [W]	0			
Mass Flow Rate [kg/s]	-0.0460101			
Volume Flow Rate [m <sup>3</sup> /s]	-0.076427			
Surface Area [m <sup>2</sup> ]	0.0106466	-0.000216528	-9.24096e-006	0.0100263
Total Enthalpy Rate [W]	-27508.6			
Uniformity Index [ ]	1.95368			
CAD Fluid Area [m <sup>2</sup> ]	0.00787803			

11 Click **OK**  to close the dialog box.

## Calculating the Heat Exchanger Efficiency

---

The heat exchanger efficiency can be easily calculated, but first we must determine the fluid with the minimum capacity rate ( $C = \dot{m}c$ ). In this example the water mass flow rate is 0.02 kg/s and the air mass flow rate is 0.046 kg/s. The specific heat of water at the temperature of 300 K is about five times greater than that of air at the temperature of 585 K. Thus, the air capacity rate is less than the water capacity rate. Therefore, according to *Ref.2*, the heat exchanger efficiency is calculated as follows:

$$\varepsilon = \frac{T_{hot}^{inlet} - T_{hot}^{outlet}}{T_{hot}^{inlet} - T_{cold}^{inlet}},$$


where  $T_{hot}^{inlet}$  is the temperature of the air at the inlet,  $T_{hot}^{outlet}$  is the temperature of the air at the outlet and  $T_{cold}^{inlet}$  is the temperature of the water at the inlet.

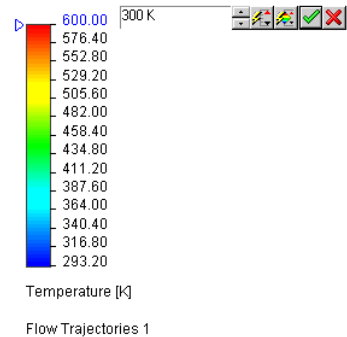
We already know the air temperature at the inlet (600 K) and the water temperature at the inlet (293.2 K), so using the obtained values of water and air temperatures at outlets, we can calculate the heat exchanger efficiency:

$$\varepsilon = \frac{T_{hot}^{inlet} - T_{hot}^{outlet}}{T_{hot}^{inlet} - T_{cold}^{inlet}} = \frac{600 - 584}{600 - 293.2} = 0.052$$

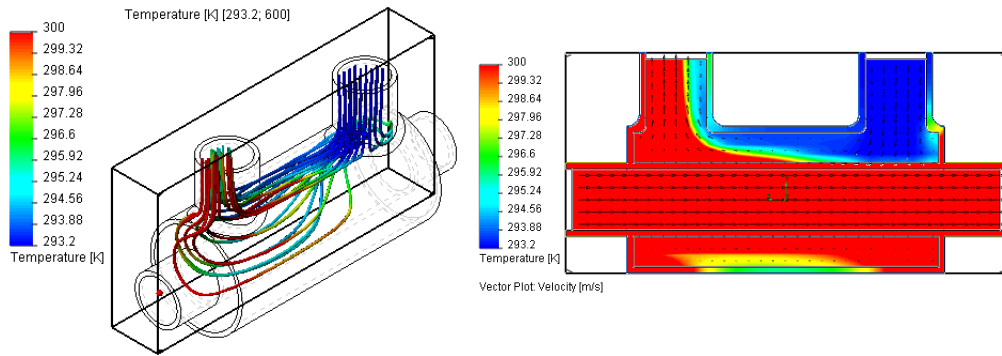
## Specifying the Parameter Display Range

---

- 1 In the temperature palette bar click the maximum value and type 300 K in an edit box
- 2 Click . Immediately the trajectories are updated.



## Chapter 6 Heat Exchanger Efficiency



As you can see, Flow Simulation is a powerful tool for heat-exchanger design calculations.

Ref. 2 J.P. Holman. "Heat Transfer" Eighth edition.