

Heat Exchanger Efficiency

COSMOSFloWorks can be used to study the fluid flow and heat transfer for a wide variety of engineering equipment. In this example we use COSMOSFloWorks to determine the efficiency of a counterflow heat exchanger and to observe the temperature and flow patterns inside of it. With COSMOSFloWorks 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 efficiency is its “effectiveness” in transferring a given amount of heat from one fluid at a high temperature to another fluid at a lower temperature. The effectiveness can be determined if the temperature at all flow openings are known. In COSMOSFloWorks the temperature at the fluid inlets are specified and the temperature at the outlets can be easily determined. Heat exchanger effectiveness 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 were to undergo a temperature change equal to the maximum temperature difference present in the exchanger, which is the difference in the inlet temperature of the hot and cold fluids respectively $(T_{hot}^{inlet} - T_{cold}^{inlet})$. Thus, for a counterflow heat exchanger

the effectiveness is defined as follows: $\varepsilon = \frac{T_{hot}^{inlet} - T_{hot}^{outlet}}{T_{hot}^{inlet} - T_{cold}^{inlet}}$ - if hot liquid capacity rate is

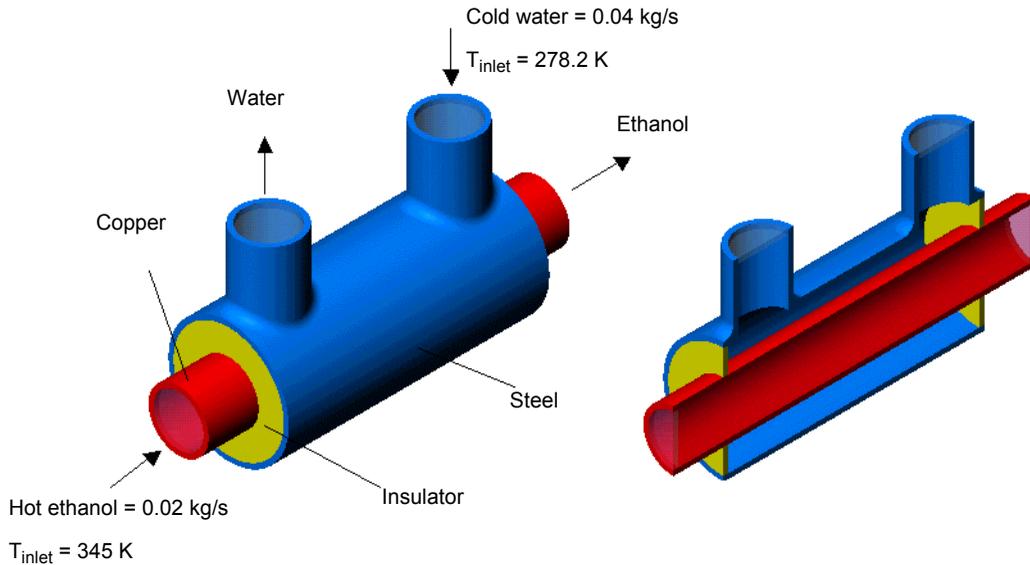
less than cold liquid capacity rate $\varepsilon = \frac{T_{cold}^{outlet} - T_{cold}^{inlet}}{T_{hot}^{inlet} - T_{cold}^{inlet}}$ - if hot liquid capacity rate is more

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

The goal of the project is to calculate the effectiveness of the counterflow heat exchanger.

Open the Model

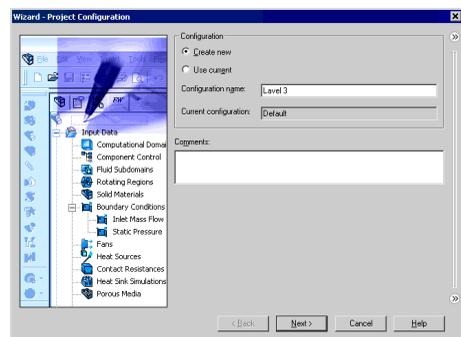
Click **File, Open**. In the **Open** dialog box, browse to the **Heat Exchanger.SLDASM** assembly located in the **Tutorial 3 - Heat Exchanger Efficiency** folder and click **Open** (or double-click the assembly). Alternatively, you can drag and drop the **heat exchanger.SLDASM** file to an empty area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



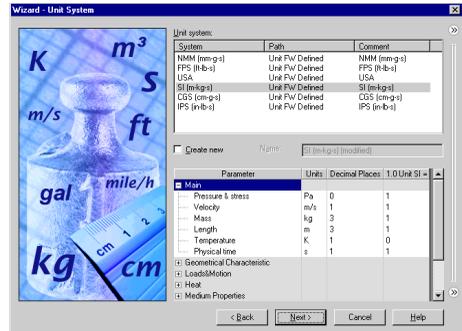
Creating a Project

- 1 Click **FloWorks, Project, Wizard**.
- 2 Click **Create new**. In the **Configuration name** box type **Level 3**. The 'Level3' name was chosen because this problem will be calculated using **Result Resolution** level 3.

Click **Next**.



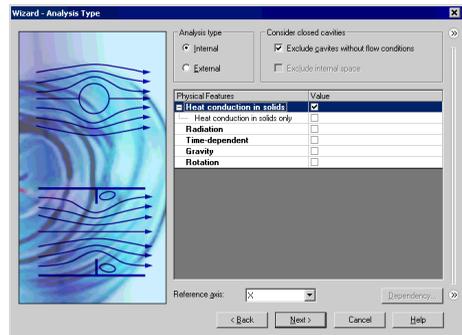
- In the **Units** dialog box select the desired system of units for both input and output (results). For this project we will use the International System **SI** by default.



Click **Next**.

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

COSMOSFloWorks always automatically considers heat conduction within the fluid and between the walls and the fluid (i.e., convection).

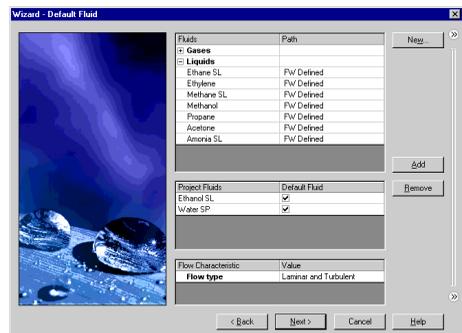


*By default, COSMOSFloWorks will not consider heat conduction in solids. Selecting the **Heat conduction in solids** option enables the combination of convection and conduction heat transfer, known as conjugate heat transfer. In this project we will analyze heat transfer between the fluids through the model walls, as well as inside the solids.*

Click **Next**.

- Since both water and ethanol are used in this project, expand the **Liquids** folder and add **Water SP** item and then **Ethanol SL**.

***SL** (Saturation Line) means that temperature dependencies of a liquid are taken along the saturation line.*



Click **Next**.

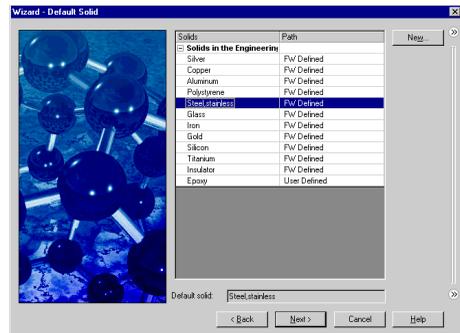
- Since we intend to calculate heat conduction in solids, the **Default Solid** dialog box appears. In this dialog you specify the default solid material applied to all solid components. To specify a different material to a particular assembly component you need to create a **Solid Material** condition for this component.

Chapter 6 Heat Exchanger Efficiency

If the material you wish to analyze is not under the **Solids** table you can click **New** and define a new substance in the **Engineering Database**. The heat exchanger in this project is constructed of steel, copper and insulators.

Click **Steel, stainless** to make it default material.

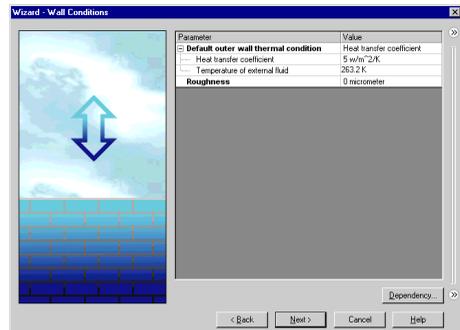
Click **Next**.



If a component has been previously assigned a solid material by the SolidWorks' Materials Editor, you can import this material into COSMOSFloWorks and apply it to the component by using the **Insert Material from Model** option accessible under **Floworks, 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 flux from the outer model walls to an external fluid (not modeled) by specifying the reference fluid temperature and the heat transfer coefficient value.



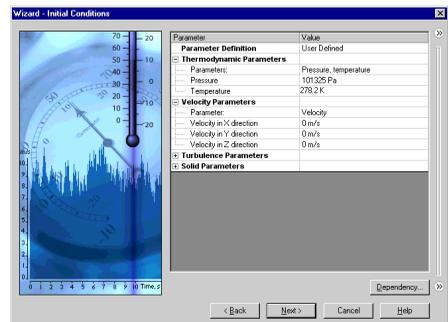
Set the **Heat transfer coefficient** value to $5 \text{ W/m}^2/\text{K}$ and the **Temperature of external fluid** to 263.2 K.

In this project we also will not concern rough walls.

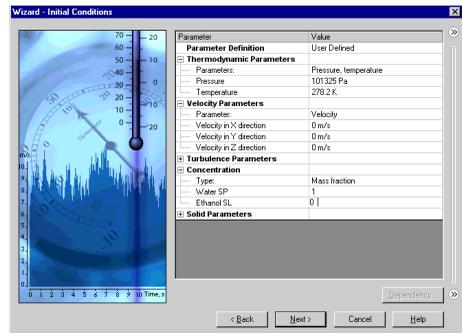
Click **Next**.

- 8 In the **Initial and Ambient Conditions** dialog box specify initial values of the flow parameters. Since the inlet water temperature is equal to 278.2 K, we will specify it as an initial condition to be consistent with the specified boundary condition.

Double-click the **Value** cell of the **Temperature** item and type 278.2.



- 9 Since we specified two liquids the **Concentration** group appear. COSMOSFloWorks allows you to specify relative fractions of the project fluids either by **Mass fraction** or by **Volume fraction**. COSMOSFloWorks uses the specified concentrations as initial conditions in the computational domain and as a default inlet boundary condition in **Boundary Conditions** and **Fans**.



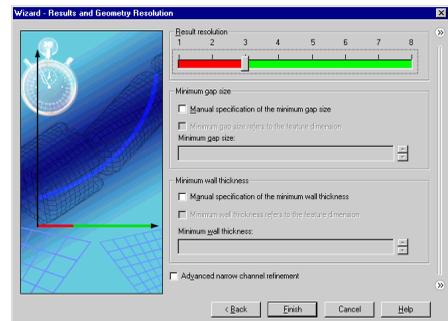
In this project two different fluids (water and ethanol) do not mix with one another, flowing in two different channels separated by a wall. The corresponding boundary conditions governing this occupation, as well as these fluids' inlet temperatures, will be specified later. In addition to these boundary conditions it is necessary to specify initial conditions, which are rather arbitrary since in this example we search a steady-state solution. For a steady-state solution it is not obligatory to specify the exact initial conditions (water in the water's region and ethanol in the ethanol tube) since the water initially specified in the ethanol flow passage will be replaced later, i.e. during the calculation, by the ethanol entering this region from the region's inlet boundary. But it is better for searching the solution to avoid extremes and specify ethanol as a flow initial condition in its region. In order to specify the fluids' initial different concentrations (0 or 1) in their flow passages, in the **Concentration** group, we will specify the water's concentration equaled to 1 (and the ethanol concentration equaled to 0) as the fluid initial condition applied by default to all the fluid regions, including the ethanol region. To specify initial flow conditions (pressure, temperature, concentration, etc.) for selected volumes that differ from the default conditions we will use the **Initial Condition** dialog box.

Double-click the corresponding **Concentration** value and type 0 for **Ethanol**, and 1 for **Water**. This implies that water initially exists in the entire **Computational Domain**.

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

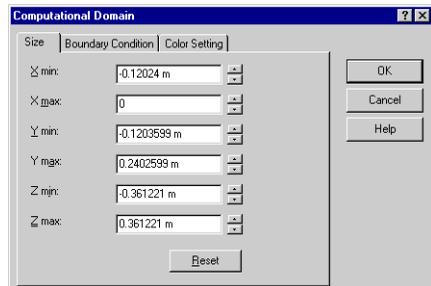
Click **Finish**.

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

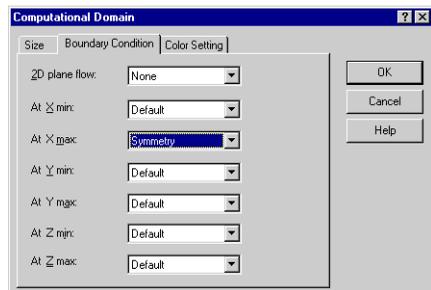
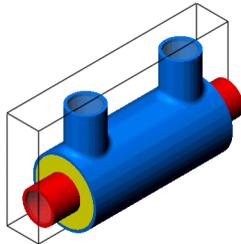


Symmetry Condition

- 1 Click **FloWorks, Computational Domain**.
- 2 In the **X max** box type 0.
- 3 Click the **Boundary Condition** tab.



- 4 In the **At X max** list select **Symmetry**. Click **OK**.



 *Creating an initial condition allows you to specify initial flow conditions in specific fluid regions that differ from the default initial conditions specified in the Wizard or General Settings.*

Specifying Initial Conditions – Fluid Initial Conditions

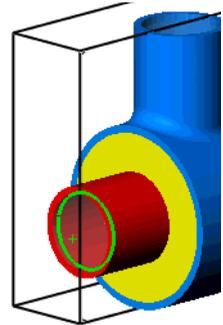
We will specify the ethanol concentration of 1 (and, correspondingly, the water concentration of 0) in this selected region. In addition, let us specify the ethanol's inlet temperature of 345 K as its thermal initial condition in this region. This initial thermal condition differing from the default one is not obligatory since this problem is too simple, so the same steady-state solution is quickly obtained from any initial solid and fluid temperatures. Naturally, in more complex and intricate tasks well-selected (i.e., closer to the solution) initial conditions can substantially increase the solution convergence rate, thus reducing the required CPU time.

- 1 Click **FloWorks, Insert, Initial Condition**.

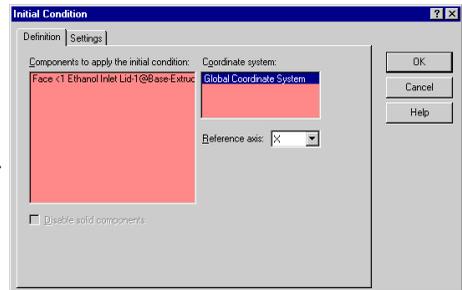
 To specify the initial condition within a fluid region we must specify this condition on the one of the faces lying on the region's boundary - i.e. on the boundary between solid and fluid substances. The initial condition specified on the region's boundary will be applied to the entire fluid region.

2 Select the **Ethanol Inlet Lid** inner face (in contact with the fluid).

 If you want to specify an initial condition to a fluid region within a closed internal fluid volume, instead of creating this volume as a separate component (and disabling it), you can select any surface bounding this fluid volume to be automatically considered by the program as the volume to apply the fluid initial conditions.



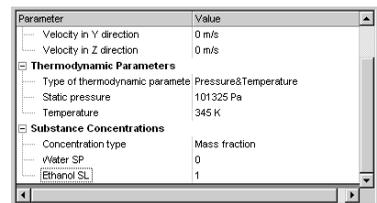
3 Accept the default **Coordinate system** and the **Reference axis**. Click the **Setting** tab. On the **Settings** tab COSMOSFloWorks allows you to specify initial flow parameters, initial thermodynamic parameters, initial turbulence parameters (if the **Show advanced parameters** check box is selected), and initial concentrations.



These settings are applied to the fluid region inside the **Tube** component.

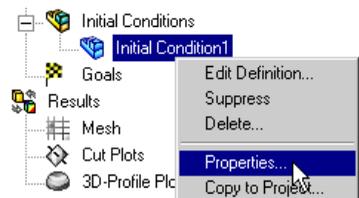
4 Double-click the **Value** cell of **Temperature** and type 345.

5 Expand the **Substance Concentrations** item and specify concentration as follows:
Ethanol - 1, Water - 0.



6 Click **OK**. The new **Initial Condition1** item appears in the analysis tree.

7 To easily identify the specified condition you can give a more descriptive name for the **Initial Condition1** item. Right-click the **Initial Condition1** item and select **Properties**. In the **Name** box type **Hot Ethanol** and click **OK**.



 You can also click-pause-click an item to rename it directly in the COSMOSFloWorks analysis tree.



The specified initial condition means that initially hot (345 K) ethanol exists within the tube. Although the specification of this initial condition was optional it will result in a dramatic decrease in the total computation time. The next step is to specify boundary conditions to define the water and ethanol flows passing through the model.

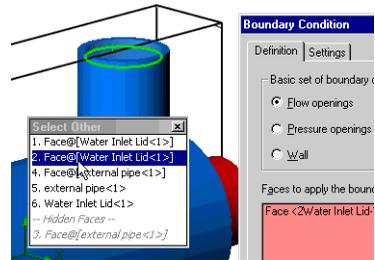
Specifying Boundary Conditions

- 1 Right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**. The **Boundary Condition** dialog box appears.

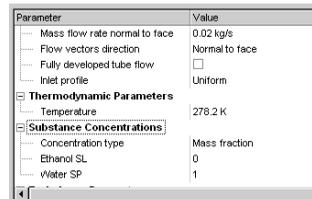
- 2 Select the **Water Inlet Lid** inner face (in contact with the fluid).

The selected face appears in the **Faces to apply boundary condition** list.

- 3 Accept the default **Inlet Mass Flow** condition and the default **Coordinate system** and **Reference axis**. Click the **Settings** tab.



- 4 Double-click the **Value** cell of the **Mass flow rate normal to face** item and set it equal to 0.02 kg/s. Since the symmetry plane halves the opening we need to specify a half of the actual mass flow rate.



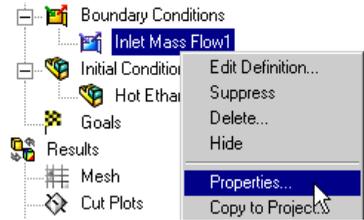
- 5 Expand the **Thermodynamic Parameters** item. The default temperature value is equal to the value specified as initial temperature in the **Wizard**. We accept this value.
- 6 Expand **Substance Concentrations** item to check that the concentration of the inlet fluid is the default concentration that you specified in the **Wizard**. The zero concentration for Ethanol implies that water is the only liquid entering the model through this opening.
- 7 Click **OK**. The new **Inlet Mass Flow1** item appears in the analysis tree.

This boundary condition specifies that water enters the steel jacket at a mass flow rate of 0.04 kg/s and temperature of 278.2 K.

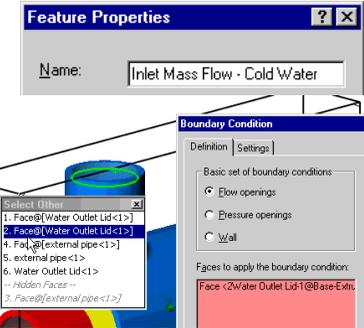
- Rename the **Inlet Mass Flow1** item to Inlet Mass Flow - Cold Water.

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

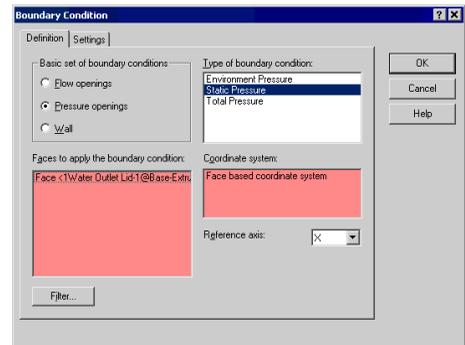
- In the COSMOSFloWorks analysis tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.



- Select the **Water Outlet Lid** inner face (in contact with the fluid). The selected face appears in the **Faces to apply boundary condition** list.



- Click **Pressure openings** and in the **Type of boundary condition** list, select the **Static Pressure** item. Click the **Settings** tab.



- Accept the default value for static pressure (101325 Pa), temperature (278.2 K) and all other parameters.

- Click **OK**. The new **Static Pressure1** item appears in the COSMOSFloWorks analysis tree.

Parameter	Value
Thermodynamic Parameters	
Static pressure	101325 Pa
Temperature	278.2 K
Substance Concentrations	
Turbulence Parameters	
Boundary Layer Parameters	

- Rename the **Static Pressure1** item to Outlet Static Pressure - Water.

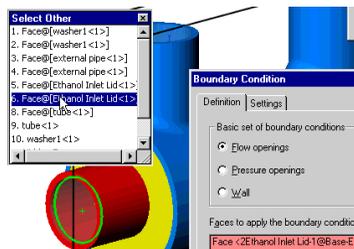


Next we will specify the boundary conditions for the ethanol flow.

- In the COSMOSFloWorks analysis tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**.

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

The selected face appears in the **Faces to apply boundary condition** list.



- 3 Accept the default **Inlet Mass Flow** condition and the default **Coordinate system** and **Reference axis**. Click the **Settings** tab.

- 4 Double-click the **Value** cell of the **Mass flow rate normal to face** item and set it equal to 0.01 (type the value, the units will appear automatically). Since the symmetry plane halves the opening we need to specify a half of the actual mass flow rate.

- 5 Expand the **Thermodynamic Parameters** item. Double-click the **Value** cell of the **Temperature** item and type 345.

Parameter	Value
Flow Parameters	
Mass flow rate normal to face	0.01 kg/s
Flow vectors direction	Normal to face
Fully developed tube flow	<input type="checkbox"/>
Inlet profile	Uniform
Thermodynamic Parameters	
Temperature	345 K
Substance Concentrations	
Concentration type	Mass fraction
Ethanol SL	1
Water SP	0

- 6 Expand the **Substance Concentrations** item and specify concentration as follows: Ethanol - 1, Water - 0.

- 7 Click **OK**. The new **Inlet Mass Flow1** item appears in the analysis tree.

This boundary condition specifies that ethanol enters the copper tube at a mass flow rate of the 0.02 kg/s and a temperature of 345 K.

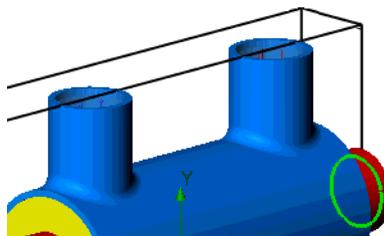
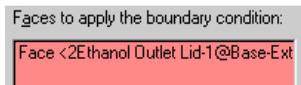
- 8 Rename the **Inlet Mass Flow1** item to `Inlet Mass Flow - Hot Ethanol`.

Next specify the ethanol outlet **Static Pressure** condition.

- 9 In the COSMOSFloWorks analysis tree, right-click the **Boundary Conditions** icon and select **Insert Boundary Condition**. The **Boundary Condition** dialog box appears.

- 10 Select the **Ethanol Outlet Lid** inner face (in contact with the fluid).

The selected face appears in the **Faces to apply boundary condition** list.



- 11 Click **Pressure openings** and in the **Type of boundary condition** list, select the **Static Pressure** item. Click the **Settings** tab.

- 12 Set **Temperature** to 345 K.

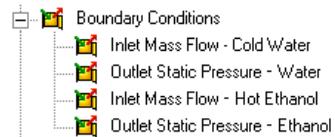
- 13 Specify concentration as follows:

Ethanol – 1,
Water – 0.

Click **OK**.

Parameter	Value
Thermodynamic Parameters	
Static pressure	101325 Pa
Temperature	345 K
Substance Concentrations	
Concentration type	Mass fraction
Ethanol SL	1
Water SP	0

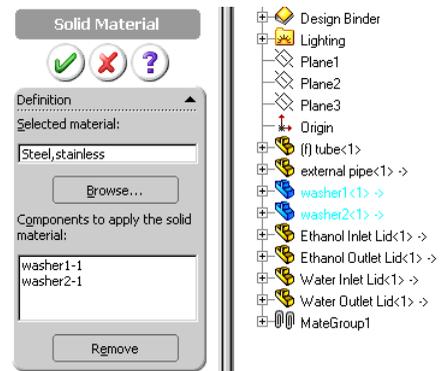
- 14 Rename the new item **Static Pressure1** to Outlet Static Pressure - Ethanol.



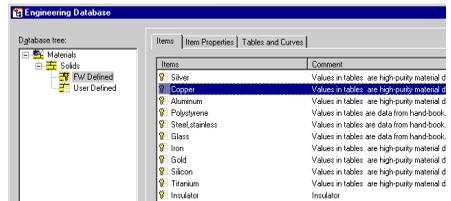
For analysis of heat transfer in solids you must specify the solid materials in the model and the initial solid temperature. To do this, define material conditions for the solid components.

Specifying Solid Materials

- 1 In the COSMOSFloWorks analysis tree, right-click the **Solid Materials** icon and select **Insert Solid Material**. The **Solid Material** dialog box appears.
- 2 In the FeatureManager select the **washer1** and **washer2** components. They appear in the **Components to apply the solid material** list.
- 3 Click **Browse** to select the solid from the **Engineering database**.

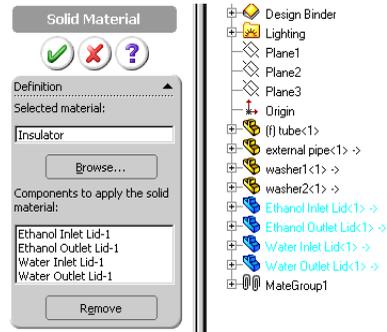


- 4 Select the **Copper** item under the **Solids, FW Defined**, item.
- 5 Click **OK** to return to the **Solid Material** dialog box. Click **OK**.
- 6 Rename the **Copper Solid Material 1** item to **Copper**.



Notice that the auxiliary lids on the openings are solid. Since the material for the lids is the (default) 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 analysis by specifying the lids as insulators.

- 7 Right-click the **Solid Materials** icon and select **Insert Solid Material**.
- 8 In the FeatureManager, select all lid features. As you select the lids, their names appear in the **Components to apply the solid material** list.
- 9 Click **Browse** and in the **Engineering database** select the **Insulator** solid.
- 10 Click **OK**. Now all of the auxiliary lids are defined as insulators.



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

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

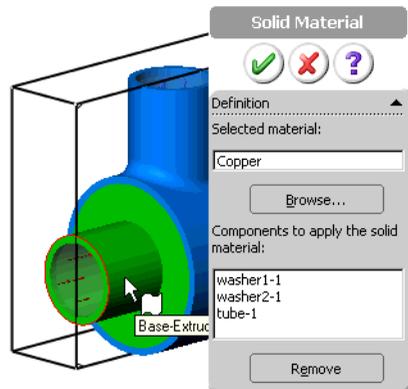
Next specify a material for the inner pipe.

- 12 Right-click the **Copper** icon and select **Edit Definition**.
- 13 In the graphics area select one of the tube faces. In addition to using the FeatureManager, you can select a component by selecting a component face, edge or point.
- 14 In the **Selected material** list keep the **Copper** item.



- 15 Click **OK**.

For the last **external pipe** component, the steel is already applied as the default setting.



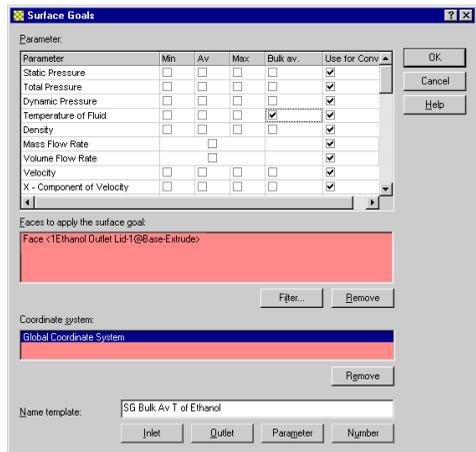
The heat exchanger effectiveness can be easily calculated, but first we must determine the liquid with the minimum capacity rate ($C = \dot{m}c$). In this example the water mass flow rate is 0.04 kg/s and the ethanol mass flow rate is 0.02 kg/s. The specific heat of water is about double that of ethanol. Thus, the ethanol capacity rate is less than the water capacity rate. Therefore, from page 6-2 the heat exchanger effectiveness is calculated as follows:

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

where T_{hot}^{inlet} - is the temperature of the ethanol at the inlet, T_{hot}^{outlet} - is the temperature of the ethanol at the outlet, and T_{cold}^{inlet} - is the temperature of the water at the inlet. Since we already know the ethanol temperature at the inlet (345 K) and the water temperature at the inlet (278.2 K), our goal is to determine the ethanol temperature at the outlet. The easiest and fastest way to find this parameter is to specify the corresponding engineering **Goal**.

Specifying a Surface Goal

- 1 In the COSMOSFloWorks analysis tree, right-click the **Goals** icon and select **Insert Surface Goals**.
- 2 In the COSMOSFloWorks analysis tree, select the **Outlet Static Pressure - Ethanol** item. The face that belongs to this boundary condition is automatically selected.
- 3 In the **Parameter** table select the **Bulk Av.** check box in the **Temperature of Fluid** row.
- 4 Accept selected **Use for Conv** check box to use this goal for convergence control.
- 5 In the **Name template** type SG Bulk Av T of Ethanol.
- 6 Click **OK**.



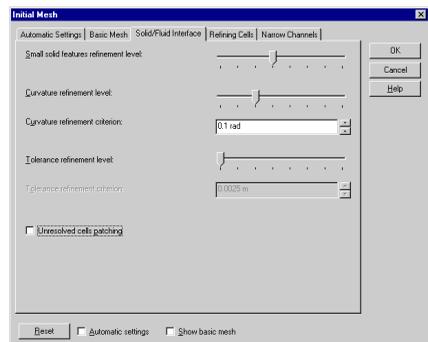
Click **FloWorks, Project, Rebuild**.

Adjusting Mesh Settings

- 1 Click **FloWorks, Initial Mesh** and switch off the automatic settings of initial mesh by clearing the **Automatic settings** check box.
- 2 On the **Solid/Fluid Interface** tab set the **Curvature refinement level** to 2 and the **Curvature refinement criterion** to 0.1 rad.

This will invoke additional cell refinement at the tube's interface, providing a more accurate heat exchange between the hot ethanol and the water.

- 3 Click **OK**.



Running the Calculation

- 1 Click **FloWorks, 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 COSMOSFloWorks analysis tree you can use COSMOSFloWorks Toolbars to get fast and easy access to the most frequently used COSMOSFloWorks features. Toolbars are very convenient for displaying results.

Click **View, Toolbars, COSMOSFloWorks Results, Main**. The **COSMOSFloWorks Results Main** toolbar appears.



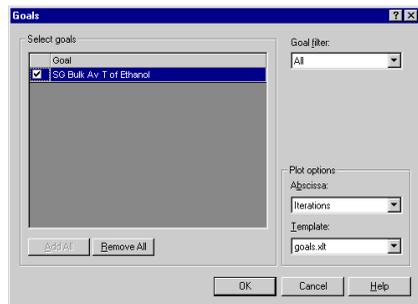
Click **View, Toolbars, COSMOSFloWorks Results Insert**. The **COSMOSFloWorks Results Insert** toolbar appears.



Click **View, Toolbars, COSMOSFloWorks Results Display**. The **COSMOSFloWorks Results Display** toolbar appears.



- 1 Click **Generate Goals Plot**  on the **Main** toolbar. The **Goals** dialog box appears.
- 2 Select the **SG Bulk Av T of Ethanol** item.
- 3 Click **OK**. The **goals1** Excel workbook is created.



You can view the temperature of the ethanol leaving 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
SG Bulk Average T of Ethano	[K]	339.9732883	339.994	339.962	340.108	100	Yes

Now you can calculate the exchanger effectiveness from the formula mentioned above:

$$\varepsilon = \frac{345 - 340}{345 - 278.2} = 0.07$$

Let us now look inside the exchanger to investigate the mechanisms for the heat transfer in the liquids and solids. First we will create a cut plot to see a temperature distribution.

Creating a Cut Plot

1 Click **Insert Cut Plot**  on the **Insert** toolbar. The **Cut Plot** dialog box appears.

2 In the FeatureManager select **Plane3**.

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

4 Click **View Settings** in order to specify the temperature as the parameter for the contour plot. By default the static **Pressure** is specified.

5 On the **Contours** tab, in the **Parameter** box, select **Temperature**.

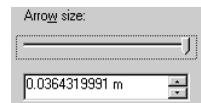
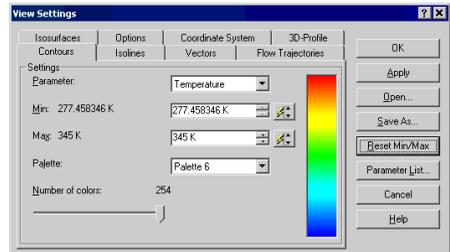
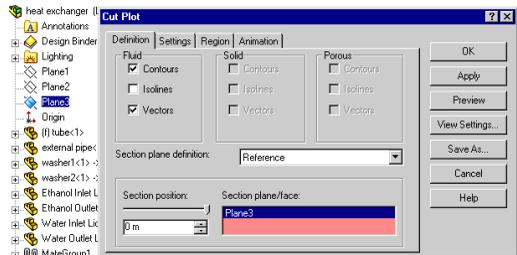
6 Using the slider set the **Number of colors** to maximum.

7 In the **View Settings** dialog box, click the **Vectors** tab and using the slider set the **Arrow size** to the maximum.

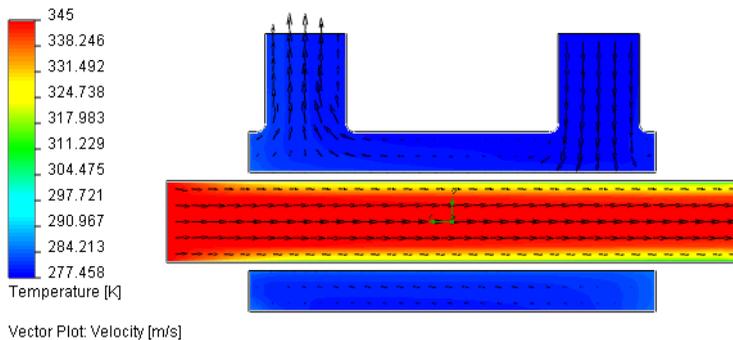
8 Click **OK** to save the changes and return to the **Cut Plot** dialog box.

9 Click **OK**. The cut plot is created but the model overlaps it.

10 Click the **Right** view on the **Standard Views** toolbar.



11 Click **Display Model Geometry**  on the **Display** toolbar to hide the model.



Let us now display the flow development inside the exchanger.

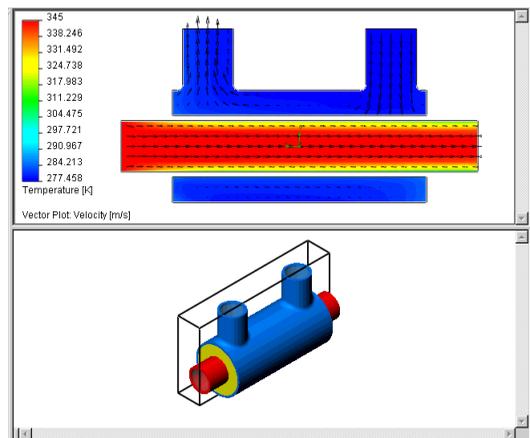
COSMOSFloWorks allows you to display results in all four possible panes of the SolidWorks graphics area. Moreover, for each pane you can specify different View Settings.

Drag the splitter bar down to split about halfway.

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.



On the **Display** toolbar, click **Display Model Geometry** , then on the **View** toolbar click

Hidden Lines Visible  to show the face outlines.

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

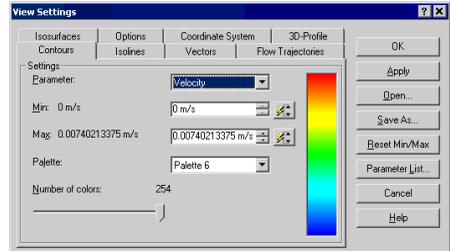
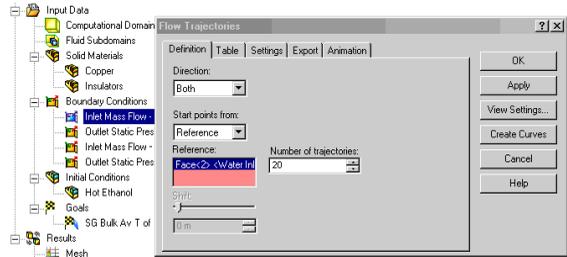
Displaying Flow Trajectories

1 Click **Insert Flow Trajectories**  on the **Insert** toolbar. The **Flow Trajectories** dialog box appears.

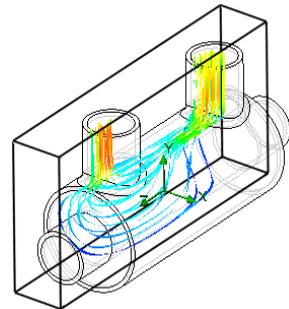
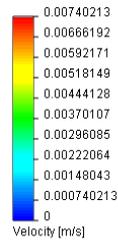
- In the COSMOSFloWorks analysis tree select the **Inlet Mass Flow – Cold Water** item.

This selects the inner face of the **Water Inlet Lid** whose points will be used as trajectory start points.

- Click **View Settings**.
- On the **Contours** tab, in the **Parameter** box, select **Velocity**.
- Click **OK** to save changes and return to the **Flow Trajectories** dialog box.
- Click **OK**. Trajectories are created and displayed.



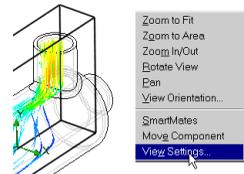
By default the trajectories are colored in accordance with the distribution of the parameter specified in the **Contours** tab of the View Settings dialog box. This is controlled by the **Use from contours** option on the **Flow Trajectories** tab of the **View Settings** dialog box. Since you specified velocity for the contour plot, the trajectory color corresponds to the velocity value.



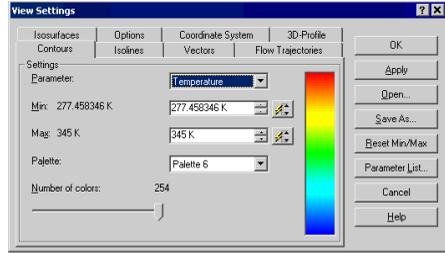
Notice that in the top pane the temperature contours are still displayed. The different view settings for each pane allow you to display contour plots for different physical parameters simultaneously.

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

- Right-click in the graphics area and select **View Settings**.

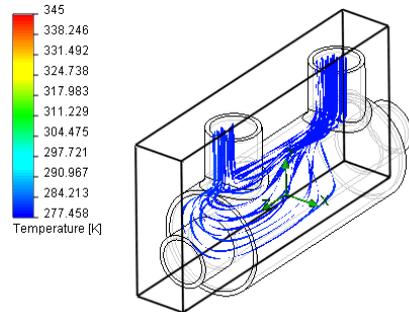


- 2 On the **Contours** tab, in the **Parameter** box, select **Temperature**.
- 3 Click **OK**. Immediately the trajectories are updated.



The water temperature range is less than the default overall (**Global**) range (277 – 345), 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.

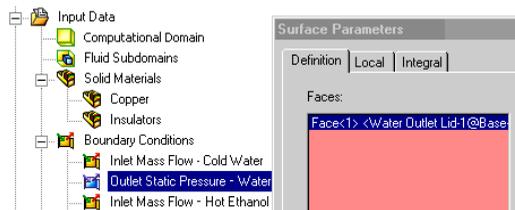


The water minimum temperature value is near 277 K. The maximum water temperature value can be determined by using the **Surface Parameters** command at the water outlet.

 **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 **Computation of Surface Parameters**  on the **Insert** toolbar. The **Surface Parameters** dialog box appears.
- 2 Click the **Outlet Static pressure - Water** item to select the face of the **water outlet lid**.
- 3 Click **Evaluate**.
- 4 After the parameters are calculated click the **Local** tab.



You can see that the average water temperature at the outlet is about 280 K. This value will be used as the maximum value.

5 Click **Cancel** to close the dialog.

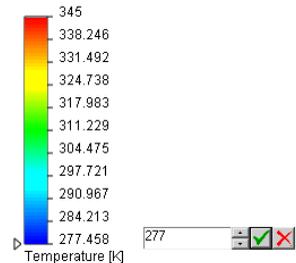
Now you can specify the display range for the temperature.

Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	101.325	101.325	101.325	101.325
Temperature (fluid) [K]	278.2	281.962	279.51	279.58
Density [kg/m ³]	999.755	999.925	999.866	999.863
Total Pressure [Pa]	101.325	101.325	101.325	101.325
Dynamic Pressure [Pa]	5.5727...	0.0266567	0.0142575	0.0191308
Velocity [m/s]	0.0003...	0.007302...	0.004958...	0.0059882
Temperature (solid) [K]	278.127	280.007	278.906	279.053
Water SP Mass Fraction []	1	1	1	1
Water SP Volume Fraction []	1	1	1	1
Ethanol SL Mass Fraction []	0	0	0	0
Ethanol SL Volume Fraction []	0	0	0	0

Specifying the Parameter Display Range

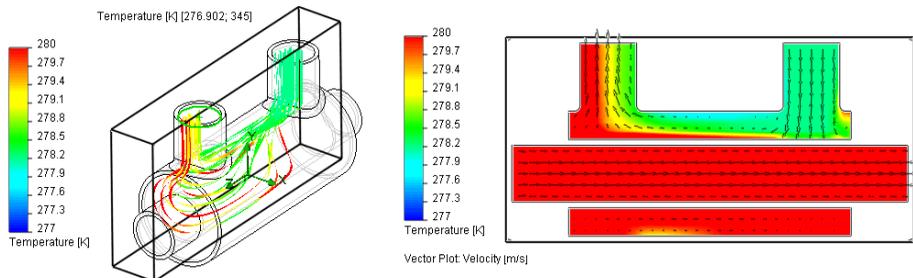
You can adjust minimum and maximum palette's values in the color bar, no need to open **View Settings** dialog box.

- 1 Click minimum value at the color bar palette.
- 2 Type **277** in the box and click . The trajectory colors are updated immediately.
- 3 Click maximum value and change it to **280**.



If you specify the range, you may want to display the global (calculated over the **Computational Domain**) minimum and maximum values of the current contour plot parameter.

- 4 Click **Display Global Min Max**  on the **Display** toolbar. The temperature global minimum and maximum values appear at the top of the active (bottom) pane.



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

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

