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:

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

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)

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 doubleclick 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.

3 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.

- 4 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

The second se	Unit sustem				
	Conten	Dath	L Comm		- i
	System	Path	Lomm	hent	_ 1
K	Cois (chi-g-s)	Pie-Deline	0 0051	(cm-g-s)	
	FPS (RIDIS)	Pre-Define	d FPS (R-ID-S)	
Constanting of	NMM (mm = c)	Pre-Define	d 175 (ii	n-io-sj (om a c)	
	NMM (mm-g-s)	Pre-Define	d NMM	(mm-g-s)	
	51 (m-kg-s)	Pre-Define	-mile b	Kg-sj	
male	USM	FIE-Deline	d 054		
m/s 64					
No. 20 March 19					
	Create new	lame:	SI (m-ko-s) (modified)		_
	-				
			Decision de la constitu-	A Cloub	G1
mila /h	Parameter	Unit	Decimats in results	T ST unit	
and mile/n			disbiah	equars to	1
yai se	🗏 Man				
	Pressure & stress	Pa	.12	1	
	- Velocity	m/s	.123	1	
A CARDE A CARD	- Mass	kg	.123	1	
	- Length	m	123	1	
3. 186.	Temperature	ĸ	12	li i	_
	Districted time		122	1 i	
KA \cm	Privacarome	÷	.160		
	+ Geometrical Characteristi	c			
77.3 8 8 4	E Loads&Motion				
					- »
					_
	< Back	Nas	Carcel	Heb	1
	· Book				
Wizard - Analysis Type					7 X
Wizard - Analysis Type					? ×
Wizard - Analysis Type	Analysis lyne	Consider d	osed cavilies		? ×
Wizard - Analysis Type	Analysis type	Consider d	osed cavities		? ×
Wizard - Analysis Type	Analysis type finternal	Consider d	osed cavities ide gavites without flor	w conditions	? × »
Wizard - Analyzis Type	Analysis type © Internal	Consider d	osed cavities ide gavites without flor	w conditions	? ×
Wizard - Analysis Type	Analysis type Internal External	Consider d F Excli	osed cavities ade gavites without flor ade internal space	w conditions	? ×
Wizard - Analysis Type	Analysis type	Consider d F Excli	osed cavities ide gavites without flor ide internal space	w conditions	? ×
Wizard - Analyzis Type	Analysis type © Internal © External	Consider d F Excl Excl	osed cavities ide <u>c</u> avites without flor ide internal space	w conditions	> ×
Wizard - Analysis Type	Analysis type Internal External Physical Features	Consider d F Excli	osed cavities ade gavites without flor ade internal space Value	w conditions	> ×
Wizard - Analyzis Type	Analysis type © Internal © External Physical Features ■ Heat conduction in	Consider d Excl Excl Excl solids	osed cavities ade gavites without flor ade internel space Value	w conditions	? ×
Wizard - Analysis Type	Analysis type Internal External Physical Features Interst conduction in Heat conduction in	Consider of Excline Excline solids	osed cavilies ade gavites without flor ade internel space Value	w canditions	? ×
Wizard - Analysis Type	Analysis type	Consider d Excl Excl Excl solids olids only	osed cavities ude gavites without flor de internal space Value	w conditions	? ×
Wizard - Analysis Type	Analysis type C Internal C External Physical Features Heat conduction in a Radiation Time-dependent	Consider d Excli Excl Solids olids only	osed cavilies ade gavites without flor ide internel ispase Value	w conditions	? ×
Wizerd - Analysis Type	Analysis type C Internal C External Physical Features I Heat conduction in Radiation Time-dependent Gravite	Consider d Excl Excl Excl Solids olids only	osed cavities ade gavites without flor de internal space Value	w conditions	? X »
Wized - Analysis Type	Analysis type C Internal C External Physical Features Heat conduction in a Radiation Time-dependent Gravity Participe	Consider d F Excl Excl Excl solids olids only	ased cavities ade gavites without flow ide internel space	w conditions	? X »
Wizard - Analysis Type	Analysis type C [nternal C External Physical Features Iteat conduction in Radiation Time-dependent Gravity Rotation	Consider of F Excl Excl solids olids only	osed cavities ade gavites vibiout flor de internal space Value	w conditions	? X >
Vicad · Analysis Type	Analysis type C Internal C External Physical Features Heat conduction in a Radiation Time-dependent Gravity Rotation	Consider of V Excli Excline solids olids only	osed covities de gavkes without flor de internel space Value - - -	w conditions	3 X
Wood-Analysis Type	Analysis type C [pternd] C External Physical Features Heat conduction in a Radiation Time dependent Gravity Rotation	-Consider d ↓ Excli ↓ Excli Excl solids only	osed cevities de gavites without floot de internal space Value 	w conditions	3 X
Vicad · Analysis Type	Analysis type C Internal C External Physical Features Heat conduction in Radiation Time-dependent Gravity Rotation	Consider of Excl Excl solids olids only	osed cavities ade gavites without flood de internal space Value	w conditions	2 X
Vicad · Analysis Type	Analysis type C Internal C External Physical Factures Heat conduction in Heat conduction in Time-dependent Gravity Rotation	- Consider d - Exclu- -	osed cavities ade gavites without floot de internal space	w conditions	3 X
Wood-Andym Type	Analysis type C Internal C External Physical Features I Incel conduction in Heat conduction Time dependent Gravity Rotation	Consider of F Excli E Excli E Excli E Excli	osed cavities ade gavites without flood de internal space Value	w conditions	3 X
Vicad · Analysis Type	Analysis type C joternal C Esternal Physical Features Hitsk consider in an Heat conduction in Heat conduction in Time dependent Gravity Rotation	Consider of Excl Excl solids olids only	osed cevilies de gavies vilhout flor de internel space Value	w conditions	? ×
Wood's Analytis Type	Analysis type C Internal C Esternal Physical Features Heat conductions in Radiation Time dependent Gravity Rotation	Consider of	osed cevities ade gevites without flow ade internal apase Value I I I I I I I I I I I I I I I I I I I	w conditions	? ×
Woad Analytin Type	Analysis type C Internal C External Physical Features I Inde Conductions in Red C	Consider of Excl Excl solids olids only	osed cavilies de gavies véhout flor ide/interné space Value	w canditions	? ×
Vized · Analytin Type	Analysis type Control of Internal External History Control Control History Control Control History Control Control History Control History Control	Consider of	osed cevilies de gavies without flor ade internel apase Vake	w conditions	? ×
Wood's Analytis Type	Analysis type C Internal C External Prymod Fachate Head conduction in a Head conduction in Head conduction in Head conduction Time dependent Giardty Rotation	Consider of Excl Excl solids olids only	osed covities de gavies vithout flor de interné space Value	w canditions	? ×
Waad-Andyn Tye	Analysis type C Internal C External Physical Features Heat conductions in Read conductions in Readown Read Conductions in Readown	Consider of Exclusion solids olids only	osed covites de gavies vithout flor de internal space Value	w conditions	? ×
Vozel - Andyth Type	Analysis type C Internal C External Internal Constraints on Internal Constraints on Internal Constraints on Internal Constraints on Time dependent Gravity Rotation	Consider of	osed covites de gavies vithout flot de internal space Value 	w conditions	? ×
Wood's Analytis Type	Analysis type C Internal C External Prymod Feature Head conduction in Head conduction in Head conduction Head conduction Reduction Constry Rotation Beference, and X Y	Consider of F Excl Exclusion olids only	orad covites de gevites vithout flou de interné apore Value O O O O O O O O O O O O O O O O O O O	w conditions	? ×
Woad-Andritt Type	Analysis type C jotenal C getenal Physical Features I Inde conduction in Heat con	Consider of F Excl Exclusion Solids only	oosd coviles de gevies without flou de internal apoee.	w conditions	? ×
Wood-Analysis Type	Analysis type C Internal C External Proyect Features Head conduction on Head conduction on Head conduction on Head conduction on Caserby Rotation Reference age X	Consider of	ored cavilies de gavies without flor de internal apoes Value	Dependence	× × ?

? X

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.

5 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.

Wizard - Default Fluid					? ×
	Fluids	Path	.	New	1 »
A CONTRACTOR OF	Nitrogen	Pre-Defined	- 1		_
	- Oxvgen	Pre-Defined			
	Propane	Pre-Defined			
	- Propylene	Pre-Defined			
	E Liquids		1		
	- Acetone	Pre-Defined			
	- Amnoria	Pre-Defined			
and the second	- Aroon	Pre-Defined			
	Ethane	Pre-Defined			
	Ethanol	Pre-Defined			
	Ethylene	Pre-Defined	- I I		1
	E mini	0.0.6	≞ _	2004	
	Project Fluids	Default Fluid		Remove	1
	Default fluid type	Liquids			_
100 M	Air (Gases)				
	Water (Liquids)	•			
	Flow Characteristic	Value			
	Flow type	Laminar and Turbulent			
A CONTRACTOR OF	Cavitation				
					>>>
	< Back	Next> Cancel		Help	J

Click Next.

6 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.

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 $W/m^2/K$.

In this project we do not consider walls roughness.

Click Next.

 8 In the Initial Conditions dialog box under Thermodynamics parameters enter
 2 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.

Wizard - Initial Conditions		?	×
70 - 20	Parameter	Value	»
60-1-20	Parameter Definition	User Defined	
50 - 10	Thermodunamic Parameters		
	- Pressure	2 atm	
	Temperature	293.2 K	
a contraction of the second	Velocity Parameters		
13 1 20 - 210	 Velocity in X direction 	0 m/s	
10-	 Velocity in Y direction 	0 m/s	
0	 Velocity in Z direction 	0 m/s	
	Turbulence Parameters		
689	Solid Parameters		
Polyter the distance state			
A REAL ASSAULT AND A REAL ASSAULT			
Z. Carlo and a second			
6. 6.			
5.			
4			
01			ő
0 1 2 3 4 3 6 7 8 9 10 10 M.S		Lependency	<i>"</i>
	< Back Ne	st> Cancel <u>H</u> elp	

Flow Simulation 2011 Tutorial

9 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

- 1 In the Flow Simulation Analysis tree, expand the Input Data item.
- 2 Right-click the Computational Domain icon and select Edit Definition.
- 3 Under Size and Conditions select the Symmetry ignormalized condition at the X max boundary and type 0 in the X max ⊕_x 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.
- Accept the default Coordinate System A and the Reference axis.



Sele	ction 🕆
7	Face<5>@Flange1-1
J.× z→x	Global Coordinate System
	Reference axis: X

- 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** In the new Fluid Subdomain 1 item appears in the Analysis tree.

Fluids 🔗
Fluid type:
Gases / Real Gases / Steam 💌
Air (Gases)
Add Fluid







9 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

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



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.

- 4 Click the Mass Flow Rate Normal to Face *m* box and set its value equal to 0.01 kg/s. Since the symmetry plane halves the opening, we need to specify a half of the actual mass flow rate.
- 5 Click OK item appearsin 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 Ist.



- 9 Click Pressure Openings and in the Type of Boundary Condition list select the Environment Pressure item.
- 10 Accept the value of Environment Pressure (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.



Select Other

Face@[External Pipe<1] Face@[Water Outlet Lid

Ther Para	modynamic meters	*
R+≕ ≕P	202650 Pa	▲ f *
т	293.2 K	🚔 f 🗶

- **11** Click **OK 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.



ar

Ther Para	rmodynamic meters	*
~P	101325 Pa	▲ f ∗
т	600 K	▲ f ≈

- **16** Click the **Velocity Normal to Face V** 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.





23 Check the values of **Environment Pressure** $\stackrel{\text{B}+}{\rightarrow}$ (101325)

22 Click Pressure Openings 2 and in the Type of Boundary Condition list select the Environment

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



Pressure item.

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



Boundary Conditions
 Inlet Mass Flow - Cold Water
 Inf Inlet Mass Flow - Cold Water
 Inf Environment Pressure - Warm Water
 Inlet Velocity - Hot Air
 Inf Environment Pressure - Air
 Soals

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.
- 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 🗹 .

arameter						~		🔖 Fla 🗞 Air	ange2<1> -> Inlet Lid<1> ->
Parameter	Min	A٧	Max	Bulk Av	Use			😽 Aii	Outlet Lid<1>
itatic Pressure					•		<u></u>	🖲 w	ater Inlet Lid<2:
otal Pressure					✓		⊨ _•	Śŵ	ater Outlet Lidz
ynamic Pressure					✓		L L	າດີມີ	atel o anet Elat
emperature of Fluid					✓			1. (h. 1819	ateditoup1
ensity					✓				
1ass in Volume					✓				
elocity					✓				
- Component of Ve					✓				
- Component of Ve					✓				
- Component of Ve					✓				
urbulent Viscosity					✓				
urbulent Time					•		a		
urbulent Length					✓		3		
urbulent Intensity					✓				
urbulent Energy					✓				
urbulent Dissipation					•				
emperature of Solid		\checkmark			•				
Alleng Temperature					•				
lass Fraction of Air					✓				
lass Fraction of Wal					✓				
olume Fraction of A					✓				
olume Fraction of V.					✓				
1 -									

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 Solution**. The **Goals1** Excel workbook is created.

Goa	ls 🕆
*	VG Av T of Tube
₽ ↓→×	Iterations 💌

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 Feature Manager design tree select Plane3.
- 3 In the Cut Plot dialog, in addition to

displaying Contours	🧾, select Vectors
23	

Sele	ction 🛛 🕆
	🗊 🗐 🖶 🖶
B	Plane3
H	0 m
	0
Disp	lay 🌣
	Contours
٢	Isolines
2	Vectors
	Mesh

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

box, select **Temperature**. 2

- 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.



Vect	ors	*
17,	Velocity	-
E	0.004 m/s Maximum	. . 2
E,	0 m/s	• 建烧

7 Click **OK V** . 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** (1997), 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 in 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 ^[1] 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 issue list. Since you specified velocity, the trajectory color corresponds to the velocity value. To define a fixed color for flow

trajectories click **Color** (2) and select a desired color.

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



4.0e-03 3.7e-03

3.4e-03

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 أي on the Flow Simulation Results Features toolbar. The Surface Parameters dialog appears.
- 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^3]	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^2]	-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

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** 47.



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.

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

Local Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	101325	101325	101325	101325
Density [kg/m^3]	0.588229	0.667233	0.604387	0.602515
Velocity [m/s]	8.33569	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^2]	-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

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^3/s]	-0.076427			
5urface Area [m^2]	0.0106466	-0.000216528	-9.24096e-006	0.0100263
Total Enthalpy Rate [W]	-27508.6			
Jniformity Index []	1.95368			
CAD Fluid Area [m^2]	0.00787803			

Consider entire model option selected, i.e. taking into account the **Symmetry** condition.

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.





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

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