# Example 6. Modelling of Heat Transfer in a Heat Exchanger

# 6.1. Introduction and Objectives

The heat transfer between fluids is one of the most frequently used processes in engineering. In the previous example we considered convection by mixing of two water streams with different temperature in a T-junction. In engineering practice, the heat transfer is usually accomplished by means of a heat exchanger. Common applications of heat exchangers include boilers, coolers, and condensers. The most common design of a heat exchanger has two fluid streams of different temperature - one of them flowing through metal tubes and the second flowing around the tubes. The heat spontaneously flows from a hotter to a colder fluid and is transferred from one fluid to the other through the tubes wall by conduction. In this example we will consider a single-phase shell-and-tube type heat exchanger that consist of a set of tubes closed in a shell. The main objectives of this exercise are as follows:

- create 3D geometry from 2D drawings located in different sketches, Add Frozen option,
- generate the mesh with inflation,
- illustrate the setup of the simulation with heat transfer between two different streams of fluid via solid wall,
- visualization of results contours of fluid temperature, mean value of temperature at the inlets and outflows.

# **6.2. Problem Description**

Details of the analysed flow problem are shown in Figure E6-1.



Figure E6-1. Problem description.

## 6.3. Geometry

Geometry in this example will be drawn using primitives and figures that are extruded from 2D sketches. As you can observe from the problem description we have here the shell that contains four tubes inside. Note that the real shell-and-tube heat exchangers have more tubes inside. The limitation to four tubes in this example results from the limitation of the student version of the software, where simulations can be performed for meshes with a maximum number of cells of 512 000 (user is able to generate bigger mesh in *Ansys Meshing* however it cannot be implemented into the *Ansys Fluent*). The user should be aware that meshes generated for real heat exchanger designs can have up to tens of millions of computational cells.

Comparing to the previous examples, a new feature of this geometry is that three layers of the flow domain (*cold liquid*, *solid body* and *hot liquid*) touch each other but they are not connected in one body. In contrary, they still are separated bodies. To draw this geometry, we have to use three different sketches that are placed in the same plane. Open *Ansys DesignModeler* and go to the *Sketching* tab. Make sure that *Auto Constraints* options are enabled and setup a proper view by clicking *Look At Face/Plane/Sketch*.

# Sketch1

(Fig. E6-2b).

Draw to circles starting in the origin of the coordinate system. Then measure the diameter of these circles and set their values as 0.12 m and. 0.029 m, respectively. In the next step you have to draw three circles with diameter of 0.029 that are located symmetrically around the

centre. In this purpose first select from the *Draw Toolbox* the *Polygon* with n = 3 and draw it starting from the origin and ending at any positive point of the *X* axis. Then, measure the distance from the centre to the node of polygon and set its value as 0.032 m. Nodes of the polygon are centres of the circles, therefore now

draw three circles starting from the nodes and measure their diameter setting the value of 0.029 m (Fig. E6-2a). If during dimensioning the red (unnecessary) dimension will appear, close the *Warning* dialog box, click with RMB and select *Cancel*. In the last step select from the sketch (by clicking) lines that form the polygon (they will be highlighted by yellow) and remove them pressing *Delete* from the keyboard. The first sketch is ready





Figure E6-2. View of the Sketch1.

# Sketch2

The second part of the geometry, i.e. the circles that will form the solid body, has to be drawn in other sketch. Therefore, a new sketch has to be created first by selecting from the top

ribbon the *New Sketch* icon (Fig. E6-3). The view of Sketch1 will disturb you, therefore go to the *Modeling* tab, select the *Sketch1* with RMB and choose *Hide Sketch* option. Then select *Sketch2* again to make it active and come back to the *Sketching* tab.

File Create Concept Tools Units View Help	_
🛛 🖉 🔚 📳 📫 🗍 Đ Undo 📿 Redo 🗍 Select	: *& %*
<b>Ⅲ - Ⅲ - / - / - / - / - / - /</b> F XYPlane - 木 Sketch1 - 契	🤣 Generate
Sketching Toolboxes	Sketch Brap
Draw	
Modify	
Dimensions	*

Figure E6-3. Creating of the *New Sketch*.

Using the same options of drawing the polygon and circles that were used during preparing of *Sketch1*, draw and measure the geometry that it is presented in Fig. E6-4a, setting values of the diameters as 29 mm and 25 mm for the big and small circles, respectively.

## Sketch3

The last part of the geometry which is placed in the same plane also has to be drawn in other sketch. Create therefore *Sketch3* and draw four circles with diameter of 25 mm, that present the streams of hot liquid (Fig. E6-4b).





When three sketches are ready you can go to the *Modeling* tab in order to create a 3D geometry. To do this use the *Extrude* tool three times, separately for each sketch. In the case of *Sketch1* set up the *Operation* as *Add Material*, the *Extent Type* as *Fixed* and enter the *FD1*, *Depth* (>0) value as 400 mm. Generate the operation.

In the case of *Sketch2* and *Sketch3* set up the same *Extent Type* and value of depth, however select the *Add Frozen* option from the *Operation* dropdown list (Fig. E6-5). By using this option, the extruded geometry will not connect with the second one even if they stay in direct contact. Therefore, for such part of the geometry it is possible to define a separate boundary

conditions, different materials, etc. Please note that 3D geometries which are not connected into one body (do not represent the same stream of the fluid) are marked with different colours (Fig. E6-6). Also, the default setup for *Add Frozen* geometries is that they are semi-transparent (Fig. E6-6a). This setup and also other options connected with body displaying can be changed (Fig. E6-6b) in main menu *View*.

D	etails View		4
Ξ	Details of Extrude2		
	Extrude	Extrude2	
	Geometry	Sketch2	
	Operation	Add Frozen	•
	Direction Vector	None (Normal)	
1	Direction	Normal	
	Extent Type	Fixed	
1	FD1, Depth (>0)	400 mm	
1	As Thin/Surface?	No	
1	Merge Topology?	Yes	
Ξ	Geometry Selection: 1		
	Sketch	Sketch2	

Figure E6-5. Details of the *Extrude* option of *Sketch2*.



Figure E6-6. Geometry generated using Extrude option with enabled Add Material and Add Frozen Operation.

In the last step of preparing the geometry you have to add two sections of the pipeline that represents both inlet and outlet of the cold stream. To do this use *Primitives*  $\rightarrow$  *Cylinder* twice, settings the values of variables available in the *Details View* windows presented in Fig. E6-7. Remember that cylinders are created in different plane comparing to the sketches. Please note that in the *Details View* window, except the *Base Plane* and type of the *Operation*, you can change also two types of values: below the *Origin Definition*, that are used to change the position of the geometry in the work space, and below the *Axis Definition*, that are used to set up the proper dimension of the geometry. In this example we will use both this options. Although the *Problem Description* says the distance from the axis of the horizontal pipe to the end of the vertical pipe is equal 0.09 m, we cannot create the cylinder placed directly at the axis with length of 0.09 m. It is because the new cylinders cannot intersect the prepared earlier

geometries of hot streams and solid body. Therefore, a combination of *Origin Definition* and *Axis Definition* is necessary. Here we draw two cylinders with length of 0.04 m that are moved along *Z* axis by a distance 0.05 m and -0.09 for *Cylinder1* and *Cylinder2*, respectively.

Details View 🛛		Details View	
Details of Cylinder1		Details of Cylinder2	
Cylinder	Cylinder1	Cylinder	Cylinder2
Base Plane	ZXPlane	Base Plane	ZXPlane
Operation	Add Material	Operation	Add Material
Origin Definition	Coordinates	Origin Definition	Coordinates
FD3, Origin X Coordinate	100 mm	FD3, Origin X Coordinate	300 mm
FD4, Origin Y Coordinate	0 mm	FD4, Origin Y Coordinate	0 mm
FD5, Origin Z Coordinate	50 mm	FD5, Origin Z Coordinate	-90 mm
Axis Definition	Components	Axis Definition	Components
FD6, Axis X Component	0 mm	FD6, Axis X Component	0 mm
FD7, Axis Y Component	0 mm	FD7, Axis Y Component	0 mm
FD8, Axis Z Component	40 mm	FD8, Axis Z Component	40 mm
FD10, Radius (>0)	20 mm	FD10, Radius (>0)	20 mm
As Thin/Surface?	No	As Thin/Surface?	No

Figure E6-7. Details of *Cylinder1* and *Cylinder2*.

Note, that we used one of the possible way to draw the geometry of the flow domain. In fact, the same geometry can be drawn much faster only using the *Primitives* and a combination of operations such as *Add Material*, *Add Frozen*, *Cut Material* or *Slice Material*.

In the *Tree Outline* you can find now *9 Parts, 9 Bodies* (Fig. E6-8a). With LMB and pressing the *Ctrl* button select from the tree four bodies that represents tubes (they will be highlighted with yellow), click on them with RMB and select *Form New Part* option to connect them together (Fig. E6-8b). Then rename *New Part* as *Tubes* again by clicking on it with RMB and choosing *Rename Bodies* option. Do the same with bodies that represent hot stream and rename another new part as *Hot*. The remaining body rename as *Cold*. Now in the tree you can find *3 Parts, 9 Bodies* (Fig. E6-8c). You can close *Ansys DesignModeler*.



Figure E6-8. Steps in forming a New Part.

### 6.4. Mesh

Open the Ansys Meshing, rename the geometry as Heat\_Exchanger, and set the Material as a Fluid for Cold and Hot bodies, and as a Solid for Tubes bodies. Next, choose the Mesh from the Outline Tree. Ensure that in the Details of "Mesh" the Physics Preference is set as CFD and Solver Preference as Fluent, and confirm by clicking Generate Mesh.

Since the geometry contains three different parts – *Cold*, *Tubes* and *Hot*, we have to enter the proper mesh settings partially, step by step for each part.

Mesh settings for the Cold part of the geometry

In the *Outline Tree* click with RMB on the *Tubes* and select *Hide Body* option. Perform the same operation for *Hot* part. Now in the main window you can see only *Cold* part of the flow domain. Select *Mesh* from the *Outline Tree*. On the top ribbon new tabs will appear. Select the *Mesh* tab and choose the *Inflation* operation (Fig. E6-9). In the *Details of "Inflation" – Inflation* dialog box select as a *Geometry* the whole body (Fig. E6-10a), and as *Boundary* four cylindrical surfaces (Fig. E6-10b). Now from the *Mesh* tab select the *Sizing* operation. As the *Geometry* select two edges – circles that represent border of the shell (Fig. E6-10c). In the *Details of "Edge Sizing" – Sizing* dialog box select the *Definition Type* as *Number of Divisions* and enter the *Number of Divisions* as 120 (Fig. E6-10d). Again select the *Sizing* operation from the *Mesh* tab and select eight edges that represent outer border of the tubes as the *Geometry* (Fig. E6-10e). Set the *Definition Type* as *Number of Divisions* entering 60 divisions (Fig. E6-10f).



Figure E6-9. Inflation operation on the top ribbon.



Figure E6-10. Settings of the mesh for *Cold* part of the flow domain.

Mesh settings for the *Hot* part of the geometry

In the *Outline Tree* click with RMB on the *Hot* and select *Show Body* option. Click again on the *Cold* and select *Hide Body* option. Now in the main window you can see only *Hot* part of the flow domain. Select the *Inflation* operation. In the *Details of "Inflation" – Inflation* dialog box select as a *Geometry* four tubes and as *Boundary* four cylindrical surfaces (Fig. E6-11a). Now select the *Sizing* operation. As the *Geometry* select eight edges that represent inner border of the tubes (Fig. E6-11b). Enter the *Number of Divisions* as 32.



Figure E6-11. Settings of the mesh for *Hot* part of the flow domain.

Mesh settings for the *Tubes* part of the geometry

In the *Outline Tree* click with RMB on the *Tubes* and select *Show Body* option. Click again on the *Hot* and select *Hide Body* option. Now in the main window you can see only *Tubes* part of the flow domain. Select the *Sizing* operation. As the *Geometry* select 16 edges that represent both outer and inner border of the tubes. Enter the *Number of Divisions* as 30 (Fig. E6-12a). Now from the *Mesh* tab select the *Face Meshing* operation. As the *Geometry* select from the *Tubes* 8 annular faces and enter the *Internal Number of Divisions* as 3 (Fig. E6-12b).





Figure E6-12. Settings of the mesh for *Tubes* part of the flow domain.

When the above settings are entered you can show all bodies in the main window and click *Generate* button in order to generate the mesh. As a result, you should obtain the mesh with two layers of regular cells that are located around the tubes wall (Fig. E6-13). Please note that the cell size inside the tubes is still quite big. Nevertheless, due to the limitation described earlier we are not able to increase the total number of cell.

Figure E6-13. The resultant mesh.



In the last step create seven *Named Selections – Hot\_inlet* and *Hot\_outlet* (both of them contain four surfaces), *Cold\_inlet* and *Cold\_outlet*, *Wall*, *Tubes\_outside* and *Tubes\_inside* (both of them contain four surfaces). They are presented in Fig. E6-14. Note that in this flow problem the necessary *Named Selections* are inlets and outlets only. Nevertheless the remaining *Named Selections* are needed in the case when we want to generate the results directly on the surfaces they represent.



Figure E6-14. Named Selections.

### 6.5. Simulations

Open the Ansys Fluent with enabled Double Precision and Display Mesh After Reading options and set the Solver Processes as 1. Run down the Outline View Tree to edit subsequent parameters and variables:

# A. Setup – General

Examine the following settings:  $Mesh \rightarrow Check$ ,  $Mesh \rightarrow Report Quality$ ,  $Solver Type \rightarrow Pressure-Based$ ,  $Solver Velocity Formulation \rightarrow Absolute$ ,  $Time \rightarrow Steady$ .

## B. Setup – Models

From the *Models Task Page* select the *Energy* equation and the *Viscous*  $\rightarrow$  *k-epsilon* (2 *eqn*) model, since the water flow in shell and tubes of the heat exchanger is turbulent. Use the default *Standard* from the *k-epsilon Model* group and choose *Enhanced Wall Treatment* from the *Near-Wall Treatment*. Do not change the values of *Model Constants*. Click *OK* to accept the model settings and close the *Viscous Model* dialog box.

## C. Setup – Materials

Open the *Materials Task Page* and add *water-liquid* (*h2o*<*l*>) to the to the *Fluid Materials* list. In this example, however, we will use as material not only a fluid, but also solid that

represents the tubes. The default solid material is aluminum, while the tubes in this example are made of copper (see problem description). Therefore, we must add also the copper to the *Solid Materials* list. It can be done analogically as in the case of fluid material adding – when the *Solid* filed in the *Materials Task Page* is highlighted click on the *Create/Edit*... to open new dialog box (Fig. E6-15a). Make sure that the *Material Type* is set up as *solid* (Fig. E6-15b) and then from the *Fluent Database* select copper (Fig. E6-15c). Click *Copy* and







**Figure E6-15.** Steps of adding the solid material.

### D. Setup – Cell Zone Conditions

Open the *Cell Zone Conditions Task Page* and select the *water-liquid* as a process fluid that flows in the *cold* and *hot* zones. Then open the *tubes* zone and here set the material as *copper*. Confirm your choice by clicking *Apply*.

# E. Setup – Boundary Conditions

Open the Boundary Conditions Task Page and set the following boundary conditions:

- 1. cold\_inlet: Type  $\rightarrow$  velocity-inlet, Momentum tab Velocity Magnitude  $[m/s] \rightarrow 0.5$ , Thermal tab - Temperature  $[K] \rightarrow 283$ ,
- 2. *cold\_outlet*: *Type*  $\rightarrow$  *outflow*,
- 3. hot\_inlet: Type  $\rightarrow$  velocity-inlet, Momentum tab Velocity Magnitude  $[m/s] \rightarrow 0.1$ , Thermal tab - Temperature  $[K] \rightarrow 353$ ,
- 4. *cold\_outlet*: *Type*  $\rightarrow$  *outflow*.

# F. Solution – Methods

Open the Solution Methods Task Page and ensure that the options within this page are set up as: Scheme  $\rightarrow$  Coupled, Flux Type  $\rightarrow$  Auto Select, Gradient  $\rightarrow$  Least Squares Cell Based, Pressure  $\rightarrow$  Second Order, Momentum, Turbulent Kinetic Energy, k, Turbulent Dissipation Rate,  $\varepsilon$ , and Energy  $\rightarrow$  Second Order Upwind, and other options are disabled.

## G. Solution – Monitors

Open the *Residual Monitors* dialog box, make sure that all equations are monitored and their convergence is checked. Then set all *Absolute Criteria* as 1e-7.

#### H. Solution – Initialization

Open the *Solution Initialization Task Page* and initialize the calculation selecting the *Hybrid Initialization*.

## I. Solution – Run Calculation

Open the *Run Calculation Task Page* and examine your setup selecting *Check Case*... You should see the recommendation to consider *realizable k-epsilon turbulence model*, nevertheless do not apply this model and close the dialog box. You can start the calculation by entering 1000 iterations. During the calculations in the main window you will observe the *Scaled Residuals*.

# 6.6. Results

The contours of the fluid static temperature are presented in two mid-planes of the heat exchanger, located along the *ZX* (Figs. E6-16a and b) and *ZY* planes (Figs. E6-16c). The mid-planes had to be created before data reading. Figs. E6-16a and E6-16c show the full range of static temperature, while in Fig. E6-16b the temperature range has been limited in order to better visualize the temperature change inside the tubes. If we need to display a narrowed range of the analysed property, it can be done by disabling the *Auto Range* option in the *Contours* dialog box (Fig. E6-17). Then the fields with the minimum and maximum value of the analysed property become active, so we can easily define them.



Figure E6-16. Contours of fluid velocity magnitude and temperature in the mid-plane and at the outflow.

Contour Name	
contour-1	
Options	Contours of
✓ Filled	Temperature *
✓ Node Values	Static Temperature 💌
Boundary Values	Min [K] Max [K]
Contour Lines	330 353.0645
Auto Range	Surfaces Filter Text
<ul> <li>Clip to Range</li> <li>Draw Profiles</li> <li>Draw Mesh</li> </ul>	Inlet     cold_inlet     hot_inlet
Coloring	Outlet     cold_outlet     hot outlet
<ul> <li>Smooth</li> </ul>	Plane-surface     nlane-55
	Display State
Colormap Options	None

Figure E6-17. Contours dialog box with enabled Auto Range option.

The mean value of the static temperature at the inlets and outlets were read in the same way as in the *Example 5*, using the *Reports* tab and *Surface Integrals* option. Obtained values of the static temperature are presented in Fig. E6-18.

KANNEL LVID	Tield Maniable	
Report Type	Field variable	
Area-Weighted Average	Temperature	•
Custom Vectors	Static Temperature	*
Vectors of		
•	Surfaces Filter Text	F.) [💙] [🐳]
Custom Vectors	🕞 Inlet	
custom vectors	cold_inlet	
	hot_inlet	
	Interface	
	<ul> <li>Outlet</li> </ul>	
Save Output Parameter	cold_outlet	
	hot_outlet	
	Plane-surface	
	(*) Wall	
	Area-Weighted Average [K]	
	0	
Compute M		
	rite Close Help	
Compute		
(compute) [W		
Console		1
Console Area-Weighted Average		
Console Area-Weighted Average Static Temperature	a [K]	
Console Area-Weighted Average Static Temperature	e [K]	
Console Area-Weighted Average Static Temperature cold_inlet	e [K] 283	
Console Area-Weighted Average Static Temperature cold_inlet cold_outlet	E [K] 283 287.86658	
Console Area-Weighted Average Static Temperature cold_inlet cold_outlet hot_inlet	[K] 283 287.86658 353 297.2609	
Console Area-Weighted Average Static Temperature cold_inlet cold_outlet hot_inlet hot_outlet	E [K] 283 287.86658 353 337.36028	

Figure E6-18. Average value of temperature at the outflow.