IOP Conf. Series: Materials Science and Engineering **420** (2018) 012036 doi:10.1088/1757-899X/420/1/012036

Numerical simulation of the effect of baffle spacing to the effectiveness of a shell and tube heat exchanger

M. A. Zebua[1] and H. Ambarita[1,2*]

[1]Department of Mechanical Engineering, Faculty of Engineering, Universitas Sumatera Utara, Jl. Almamater Kampus USU Medan 20155, Indonesia
[2]Sustainable Energy and Bioenergy Centre of Excellent, Universitas Sumatera Utara, Jl. Almamater Kampus USU Medan 20155, Indonesia

Email: himsar@usu.ac.id

Abstract. Heat Exchanger is an important equipment in engineering application. In order to improve the efficiency of a heat exchanger many modifications can be promoted such as baffles. The objective of the present study is to explore the effect of baffle spacing to the effectiveness of a heat exchanger. A U-tube type of shell and tube heat exchanger has been analysed numerically using a commercial code of computational fluid dynamic. The baffle spacing is varied from 40 mm, 50 mm and 60 mm. The temperature distribution, path line and vector velocity are plotted and discussed. The temperature effectiveness and the heat transfer rate of the heat exchanger is then examined. The results show that the smaller baffle spacing results in higher thermal effectiveness.

1. Introduction

A heat exchanger is a device to facilitate heat transfer between two (or more) streams of fluid with different temperature. Heat Exchangers are used in many industrial applications such as power plant, chemical industries, food industries, electronics, environmental, waste heat recovery, manufacture industry, air conditioning industry, and many more applications [1, 2]. Heat exchangers can be segregated into several types. One of most used type of heat exchanger is the shell and tube heat exchanger. This is because its simple construction and the maintenance cost is relatively low. In addition, the ratio of heat transfer area to its total volume is relatively high [1].

The shell and tube heat exchanger, even though it has been many used in service with, but still becoming a popular topic in research in order to get the optimum performance. As a note, in a common shell and tube heat exchanger the streams consist of fluid flow in shell side and fluid flow in tube side. In general, many researches focus the theme on the fluid flow in shell side of the heat exchanger due to complex flow characteristics and lower heat transfer rate. Since the direction of the fluid in and out and main flow is a crosswise, there are many complex flow circulations presence in the shell side. Furthermore, installing baffle make the flow even more complex [3, 4]. In this area, many researchers have published their investigations in literature. The research topics include the effect of baffle spacing on the heat transfer rate and pressure drop [5], optimization of baffle spacing and proposed appropriate correlations [6 – 8], effect of baffle configuration on heat transfer rate and pressure drop [9, 10], effect of baffle clearance on the flow direction in the shell side [11], effect of baffle inclination on the flow characteristics [12, 13], etc.

The above literatures show that baffle has strong effect on the fluid flow characteristics and heat transfer rate of the fluid in the shell side. Based on this fact, the authors focus on the

IOP Conf. Series: Materials Science and Engineering **420** (2018) 012036 doi:10.1088/1757-899X/420/1/012036

baffle spacing on the performance of a shell and tube heat exchanger by using a commercial code computational fluid dynamic (CFD). TheCFD has been applied on the heat exchanger research in the last two decades. Recently, the use of CFD in engineering applications include heat exchanger application is increasing [4]. Sunden et al. [3] clearly mentioned that CFD has been successfully replace the typical prototype investigation in the heat exchanger design process. Thus, the using of CFD in this research is a suitable approach to solve the problem. The objective of this study is to explore the effect of the baffle scaping on the effectiveness of a shell and tube heat exchanger.

2. Methods

In this research, a small-scale shell and tube heat exchanger has been investigated. The heat exchanger consists of shell, a bundle of tube and baffle. The heat exchanger and its dimension are shown in figure 1 and table 1, respectively.

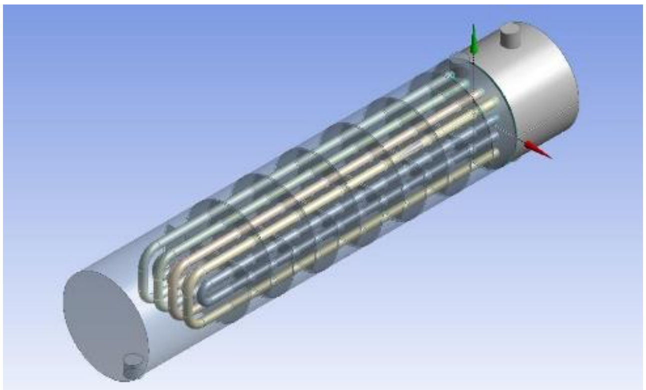

Figure 1. Model of the heat exchanger

Table 1. Dimension of the shell and tube heat exchanger

Parameter	Dimension
Diameter of the shell (D_s)	156 mm
Diameter of the outer tube (d_o)	13 mm
Diameter of the inner tube (d_i)	11.6 mm
Number of tube (N_t)	8
Length of the Heat exchanger (L)	700 mm
Tube pitch (P_t)	27 mm
Baffle cut	25%
Flow rate of the shell fluid	10 LPM
Flow rate of the tube fluid	1.8 LPM

The working fluids in this study are water and oil. The water acts as a cold fluid that flows in shell side. On the other hand, the oil acts as hot fluid that flows in the tube side. Thermal properties of the working fluids as a function of its temperature and drawn from the database provided in the CFD code. The model of the present heat exchanger is developed using ANSYS Design Modeler. In the numerical experiment, the baffle spacing is varied. It is 40 mm, 50 mm and 60 mm, respectively. The baffle cut is fixed at 25% and it is single segment and uniform for all varied spacing.

The computational domain in here is a three-dimensional problem. The governing equations have been developed for three-dimensional case. Several assumptions are made such as steady state flow and laminar low. Based of the assumptions the developed governing equations are mass conservation, momentum equation and energy equations. Those equations are given below.

$$\nabla \cdot \left(\rho \vec{V}\right) = 0 \tag{1}$$

$$\nabla \cdot \left(\rho u \vec{V}\right) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \tag{2}$$

$$\nabla \cdot \left(\rho v \vec{V}\right) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho g \tag{3}$$

$$\nabla \cdot \left(\rho w \vec{V}\right) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \tag{4}$$

$$\nabla \cdot \left(\rho e \vec{V}\right) = -p \nabla \cdot \vec{V} + \nabla \cdot \left(k \nabla T\right) + q + \Phi \tag{5}$$

In the energy equation of equation (5), Φ is a dissipation function that can be calculated using the below equation.

$$\Phi = \pi \left[2\left\{ \left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial w}{\partial z}\right)^2 \right\} + \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)^2 + \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)^2 \right] + \lambda \left(\nabla \cdot \vec{V}\right)^2 \tag{6}$$

The computational domain is divided into 4,500,000 element using Ansys Meshing. The boundary conditions are imposed on the computational domain. The temperature of hot and cold fluid entering the domain is 60°C and 27°C, respectively. The heat exchange occurs only between hot fluid and cold fluid. The heat loss to the surrounding is neglected. The flow field, pressure field and temperature field are solved iteratively is SIMPLE algorithm. All of the governing equations are discretised using first order up-wind scheme.

3. Results and Discussions

In this numerical study, the main focus is to explore the effect of the baffle spacing to the performance of the heat exchanger. The heat transfer rate and effectiveness are estimated by using temperature data resulted from the CFD simulations. Figure 2, figure 3, and figure 4 show temperature distribution for baffle spacing of 40 mm, 50 mm, and 60 mm respectively. In the figure, two type of views are shown, they are isometric view and two-dimensional view of middle plane of the heat exchanger. The variation of temperature of both fluid flows can be seen clearly. In the figure, temperature change in the tube and in the shell side can be seen. In the cold side, fluid in the shell, increase from the inlet to the outlet. On the other hand, in the hot fluid within the tube, temperature decreased as the flow close to the exit. Effect of the baffle spacing to the temperature distribution in the computational domain, can be examined by comparing figure 2 to figure 4. At low baffle spacing (shown in Figure 2), the temperature change of the hot fluid (within the tube) is more rapid than at higher baffle spacing (shown in figure 3 and figure 4). This is because at low baffle spacing heat transfer rate is better than at high baffle spacing. The velocity gradient of the fluid in the shell side is higher than at higher baffle spacing. The similar trend of temperature change is captured in the cold fluid within the shell side. This fact suggests that decreasing baffle spacing will increase the temperature change in both fluid flow.

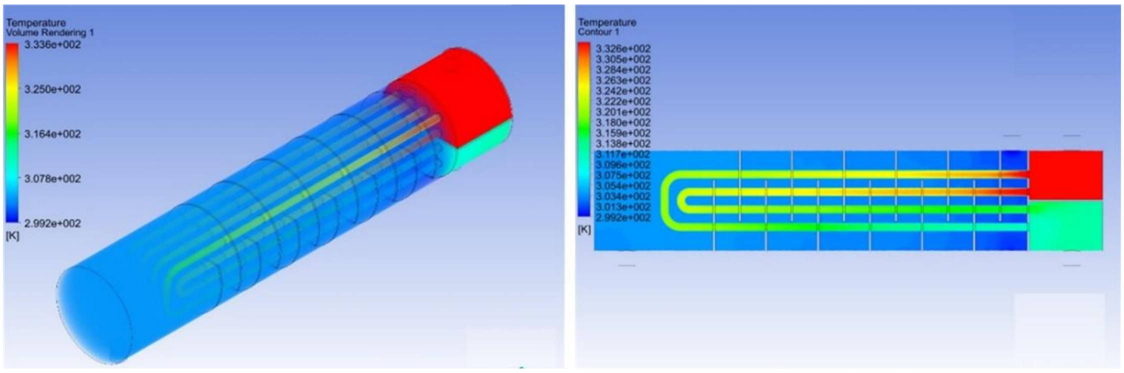

Figure 2. Temperature distributions at baffle spacing at 40 mm

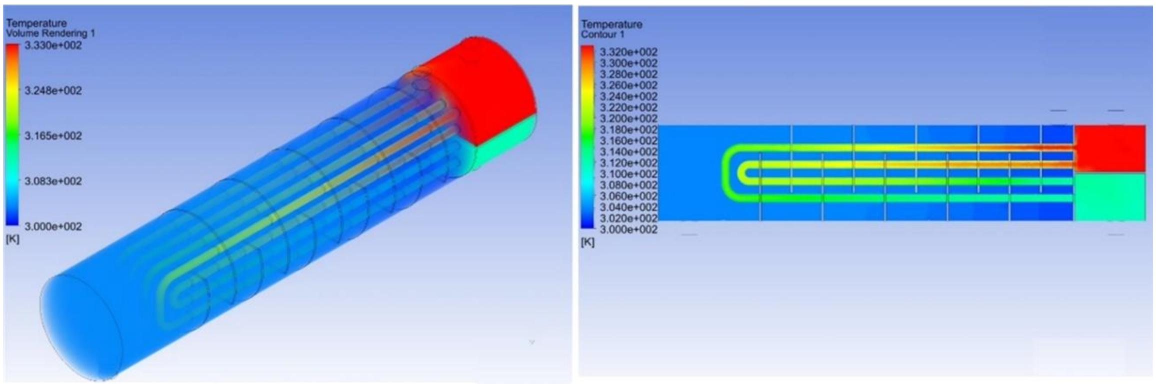

Figure 3. Temperature distributions at baffle spacing at 50 mm

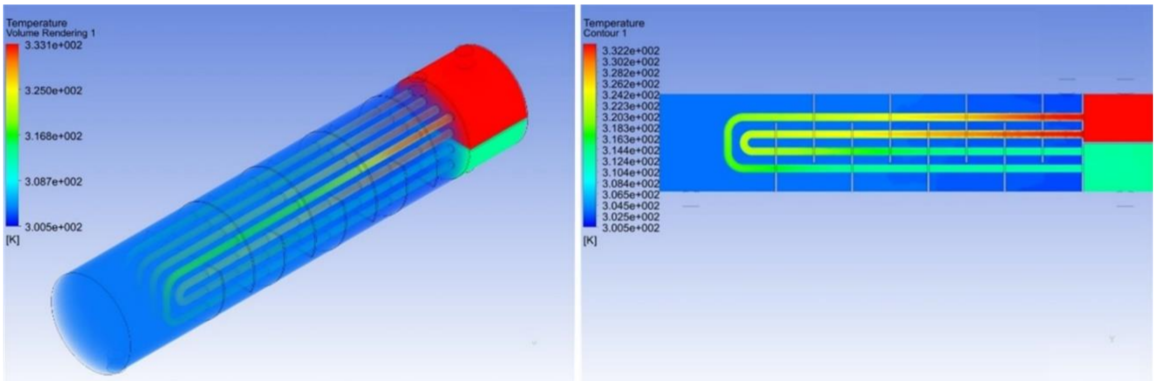

Figure 4. Temperature distributions at baffle spacing at 60 mm

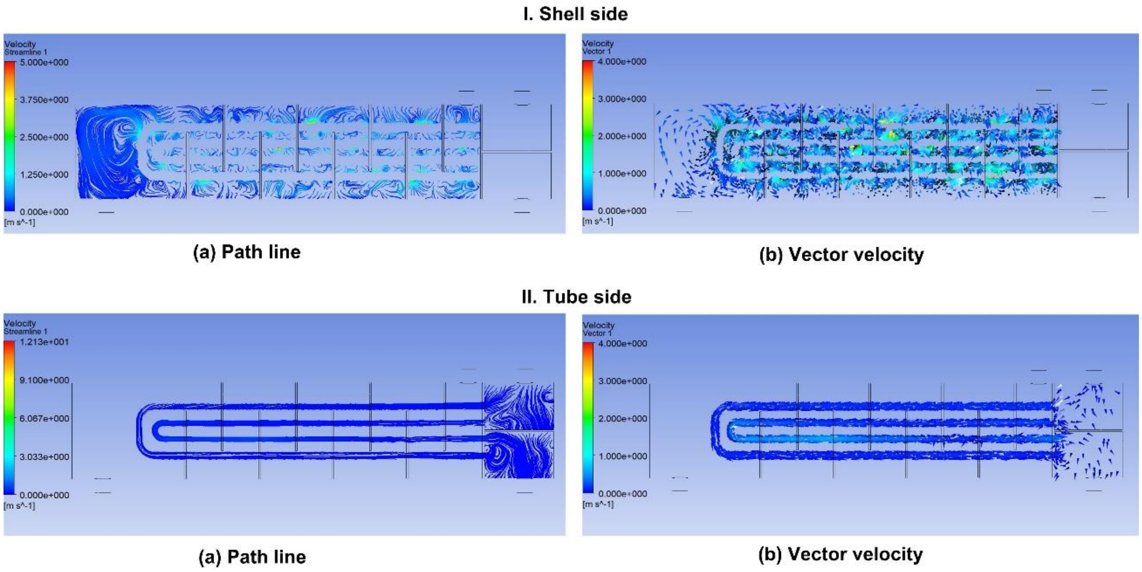

Figure 5. Path line and vector velocity at baffle spacing 60 mm

Figure 5 shows path line and vector velocity in the shell side and tube side of the heat exchanger. In the figure, the path line and vector velocity divided into shell side and tube side. It can be seen that the baffle affects the flow significantly. In addition, this is only for shell side of the heat exchanger. On the other hand, the baffle do not affect the flow within the tube.

In order to make a clear examination on the effect of the baffle spacing to the effectiveness of the heat exchanger, the calculations have been made. The temperature effectiveness of the heat exchanger at different baffle spacing are shown in table 2. The effectiveness is calculated using all the temperature in the inlet and outlet of each fluid. The temperature inlets are given, while the temperature outlets are drawn from the numerical simulation. In the simulation, heat transfer rate between hot fluid flow and cold fluid flow is calculated. The results at different baffle spacing are also shown in table 2. The table shows that increasing baffle spacing will decrease heat transfer rate in the heat exchanger. This is due to lower heat transfer rate at higher baffle spacing. As expected, increasing baffle spacing will decreasing temperature effectiveness of the heat exchanger.

Table 2. Heat transfer rate and effectiveness of the heat exchanger

Baffle spacing	Heat transfer rate (W)	Effectiveness (%)
40 mm	283.35	65.18
50 mm	309.46	62.9
60 mm	349.53	57.85

4. Conclusions

Computational fluid dynamics has been used to examine the effect of the baffle spacing on the temperature effectiveness of a shell and tube heat exchanger. A shell and tube heat exchanger with diameter of 156 mm and length 700 mm has been analysed. Commercial code Ansys FLUENT is used to solve the problem.The temperature distribution, path line and vector velocity are plotted and discussed. The temperature effectiveness and the heat transfer rate of the heat exchanger is then examined. The results show that the smaller baffle spacing results in higher thermal effectiveness.

5. References

[1] Kakac S and Hongtan L 2002 *Heat Exchangers: Selection, Rating, and Thermal Design* 2nd ed. Florida: CRC Press LLC
[2] Incropera F P et al 2011 *Fundamentals of Heat and Mass Transfer* 7th ed. New York: John Wiley & Sons, Inc.
[3] Ozden E and Tari I 2010 *Energy Conversion and Management***21** 1004-1014.
[4] Pal E, Kumar I, Joshi J B and Maheshwari N K 2016 *Chemical Engineering Science* 143 314 – 340
[5] Li H and Volker K 1998 *International Journal of Heat Mass Transfer***41** 1303-1311.
[6] Saffar-Avval M and Damangir E 1995 *International Journal of Heat and Mass Tranfer***38** 2501-2506.
[7] Soltan B K, Saffar-Avval M and Damangir E 2004 *Applied Thermal Engineering***24** 2801-2810.
[8] Eryener D 2006 *Energy Conversion and Management***47** 1478-1489.
[9] Wang X, Zheng N, Liu Z and Liu W 2018 *International Journal of Heat and Mass Transfer***124** 247-259.
[10] Ambekar S A, Sivakumar R, Anantharaman N and Vivekenandan M 2016 *Applied Thermal Engineering***108** 999 – 1007
[11] Leoni G B, Klein T S and Medronho R A 2017 *Applied Thermal Engineering***112** 479 – 506
[12] Promvonge P, Jedsadaratanachai W and Kwankaomeng 2010 *Applied Thermal Engineering***30** 1292-1303.
[13] Promvonge P, Sripattanapipat S, Tamma S, Kwankaomeng S and Thianpong C 2010 *International Communications in Heat and Mass Transfer***37** 170-177