 Numerical Simulation The Fluid Flow of a Triple Concentric Tube Heat Exchanger with Computational Fluid Dynamic (CFD)

Terang U H S Ginting¹, Tulus B Sitorus¹, Farida Ariani¹ and T E Julianto¹

¹Department of Mechanical Engineering, Faculty of Engineering, Universitas Sumatera Utara, Medan, Indonesia
Email: terang1@usu.ac.id

Abstract. Heat Exchanger (HE) is an important equipment in engineering application. It is widely used in many industries, such as automotive industries, power plant and food industries etc. This study aims focuses on Triple Concentric Tube Heat exchanger which it developed from double tube HE to know outlet temperature each fluid with Computational fluids dynamic method and then experimental study is used to validation for this study. It used for cooling water with the same fluid, in this research variation are inlet hot fluid temperature 50 °C, 55 °C, 60 °C and mass flowrate 0.0249kg/s, 0.033kg/s and 0.041kg/s counter flow fluid.

1. Introduction
Heat Exchanger (HE) is a device that support thermal energy exchange between two or more fluids with temperature differences. HE is used in many applications such as power generation, chemical and food industries, electronics, environment, waste heat reuse, manufacturing, air conditioning, and more [1,2]. One of the heat exchangers that has been is developed the three-layer concentric tube heat exchanger. This tool is an extension a centralized two-tube heat exchanger, which is widely used in the chemical industry, especially as a key component of the production system for producing products in the case of cooling or heating fluids. But not least used as a component that maintains stable operating conditions, powerful industrial tools function by controlling the heat generated due to the friction that occurs during the production process as the function of the HE can handle heat exchange with the fluid media. In this study, HE is used to cooling the hot water fluid by circulating the same cold fluid to the temperature in accordance with the experimentally performed. To conduct this research, it is necessary to design and build the appropriate test equipment for better results. The design of this these three-layer concentric tube heat exchanger, like other HE, requires technical considerations and production cost considerations. Technical considerations involve the size and material used, which directly affects the performance of the HE. Meanwhile, cost considerations make the HE comparable to the quality demonstrated through the effectiveness of the tool. Of course, in order to produce good GER, it requires materials with good thermal conductivity, and this is also in line with the increase in cost required.

A method had been developed to analyze various fluid phenomena known as Computational Fluid Dynamic (CFD). With the numerical approach, the method used in this CFD solves the problem of cost increases and technical complexity involved in developing various studies. CFD is a very useful
tool for understanding fluid dynamics in heat exchangers with the aim of developing a design. Unlike analytical methods, experiments, and computations on lower dimensions, multidimensional CFD modeling enables designers to simulate and display complex fluid dynamics by solving core physical laws for mass, momentum, and energy in 3D geometry.

2. Methods
This research was conducted using a small HE model by changing the release of hot fluid entering the HE. Some geometrical parameters are shown in Table 1. The working fluid selected is water as cold fluid for the flow on the side of the shell and oil as the hot fluid for flow on the tube side. The properties of the fluid are then adjusted according to the specified conditions by setting the properties of the fluid that are already available in the Fluent database. The geometry model is optimized by changing the inlet fluid flow to 1.5LPM, 2.0LPM, and 2.5 LPM.

2.1. Geometry Modelling
The HE Geometry is drawn using Ansys Design Modeler. Figure 1 shows the APK geometry model of three central tube layers drawn with Ansys Design Modeler according to the HE geometry parameters used.

![Fig. 1 Model of HE geometry](image)

| Tube | Specifications | (mm)   |
|------|---------------|--------|
|      | Tube length   | 2570   |
| 1    | Di-1          | 0.0127 |
|      | Do-1          | 0.0143 |
|      | Tube length   | 2220   |
| 2    | Di-2          | 0.0254 |
|      | Do-2          | 0.02728|
|      | Tube length   | 1740   |
| 3    | Di-3          | 0.04   |
|      | Do-3          | 0.0425 |
2.2. Governing equation

The regulator equation used for the flow is adapted to the simulation conditions. Because of this case the flow is considered stable, then the relationship with time is eliminated, and the equations are as follows:

- **Mass conservation:**
  \[ \nabla \cdot (\rho \vec{V}) = 0 \]  \( (1) \)

- **Momentum – x:**
  \[ \nabla \cdot (\rho \rho \vec{V} \cdot \vec{i}) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho g \]  \( (2) \)

- **Momentum – y:**
  \[ \nabla \cdot (\rho \rho \vec{V} \cdot \vec{j}) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} \]  \( (3) \)

- **Momentum – z:**
  \[ \nabla \cdot (\rho \rho \vec{V} \cdot \vec{k}) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \]  \( (4) \)

- **Energy:**
  \[ \nabla \cdot (\rho e \vec{V}) = -p \nabla \cdot \vec{V} + \nabla \cdot (k \nabla T) + q + \Phi \]  \( (5) \)

In equation \((5)\), \(\Phi\) is a dissipation function that can be calculated from:

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

2.3. Meshing

Meshing is performed for each volume in the system using Ansys Meshing. In this study, the mesh settings were performed automatically with the default settings in the software. The number of nodes generated is 270173 and the number of elements is 1058097.

2.4. Boundary condition

The mass flow rate and initial fluid temperature were initialized to the inlet of the HE with variations of 1.5 LPM, 2.0LPM, 2.5LPM for the HE while the cold fluid release was 1.5 LPM. Hot fluid temperatures in tube 2 differ from 500°C 550°C and 600°C and 250°C cold fluids in tubes 1 and 3. There is no slipping on all surface areas and zero heat flux on the shell wall as it is considered to be completely insulated.

2.5. Other modelling choices

In this study the SIMPLE algorithm is used to combine pressure and velocity fields as the flow used will be laminar. To solve the momentum and energy equations, an upward-pointing option is used.

3. Results and Discussions

In this experiment, observe the effect of the variation of the debit and the temperature of the fluid entering the HE. The rate and effectiveness of heat transfer are calculated using temperature data generated from numerical simulations. Figure 2-4 shows the results as temperature distribution in the HE with 3D views and a slice in the middle of one of the tubes to show the phenomenon of temperature changes that occurred during the process. Table 3 shows the results of the heat transfer rate calculation and the effectiveness of the tool. The calculation was performed using the Log Mean Temperature Difference (LMTD) method and heat transfer on each side of the Kern method tube was used [1]. Then, the simulation display for a temperature variation of 600°C with different emission.

Figure 2 below shows the periodic changes along tubes 1 & 3 in cold fluids and tube 2 for hot fluids. As the temperature decreases in the hot fluid side, the temperature of the cold fluid increases. The left corner shows the temperature index with the appropriate color. The temperature acquisition can be seen in Table 2.

There are no different from the phenomenon of temperature changes in the 1.5LPM hot fluid, similar changes occur in the 2.0LPM hot fluid debit. If it is adjusted to the color index on the left side
of the picture, the hot fluid entering the side of tube 2 experiences a decrease in temperature and vice versa the cold fluid in the foreground 1 & 3 has a slight increase in temperature.

**Figure 2.** Distribution of 1.5LPM inlet hot fluid for temperature of 600°C
(a) isometric view and (b) cut view

The difference between these two variations is seen from the distance the point at which the hot fluid begins to change color. Figure 2 (b) and Figure 3 (b) show the release of 2.0 LPM. The change in color of the hot fluid requires longer distances than the debit of 1.5 LPM, as well as the changes occurring in the cold fluid side.

**Figure 3.** Temperature distribution of inlet hot fluid 2.0 LPM at temperature of 600°C.
(a) isometric view and (b) cut view

**Figure 4.** Temperature distribution of inlet hot fluid 2.5 LPM for temperature of 600°C
(a) isometric view and (b) view at the cutting
Figures 1, 2, and 3 above, shows that each variation represents a different character. This is reflected in the color change that occurs. Table 2 shows the color changes as well as the temperature, and it can be said that the setting of the incoming hot fluid flow affects the HE performance. In this case, the increase in the flow of the hot inlet resulted in a distribution far from the starting point, but in terms of its effectiveness decreased (Table 4).

### Table 2. Data of experimental temperature recovery

| DEBIT | Tube 2 ($T_{h_{in}}$) | Tube 2 ($T_{h_{out}}$) | Tube 1 ($T_{c_{1_{in}}}$) | Tube 1 ($T_{c_{1_{out}}}$) | Tube 3 ($T_{c_{2_{in}}}$) | Tube 3 ($T_{c_{2_{out}}}$) |
|-------|----------------------|-----------------------|--------------------------|--------------------------|--------------------------|--------------------------|
| 1.5 LPM | 61.2                 | 41.5                  | 25.05                    | 37.63                    | 25.28                    | 37.01                    |
| 2.0 LPM | 60.17                | 43.1                  | 25.29                    | 37.67                    | 25.31                    | 34.02                    |
| 2.5 LPM | 61.11                | 44.57                 | 24.98                    | 38.85                    | 24.93                    | 34.38                    |

### Table 3. Data of temperature acquisition simulation

| DEBIT | Tube 2 ($T_{h_{in}}$) | Tube 2 ($T_{h_{out}}$) | Tube 1 ($T_{c_{1_{in}}}$) | Tube 1 ($T_{c_{1_{out}}}$) | Tube 3 ($T_{c_{2_{in}}}$) | Tube 3 ($T_{c_{2_{out}}}$) |
|-------|----------------------|-----------------------|--------------------------|--------------------------|--------------------------|--------------------------|
| 1.5 LPM | 60                   | 40.59                 | 25                       | 38.44                    | 25                       | 36.77                    |
| 2.0 LPM | 60                   | 42.17                 | 25                       | 38.71                    | 25                       | 36.24                    |
| 2.5 LPM | 60                   | 43.96                 | 25                       | 39.06                    | 25                       | 36.04                    |

### Table 4. Data of the numerical analysis of variations in hot fluid debit for temperatures of 600°C to heat transfer rate and effectiveness.

| DEBIT  | Heat transfer rate (W) | Effectiveness % |
|--------|------------------------|------------------|
| 1.5LPM | 721.14                 | 55.45            |
| 2.0LPM | 799.27                 | 50.94            |
| 2.5LPM | 539.49                 | 45.82            |

### Table 5. The comparison of numerical simulation results with experiments

| DEBIT | Total heat transfer rate % difference | Effectiveness % difference |
|-------|--------------------------------------|---------------------------|
| 1.5 LPM | 19.35                                | 1.44                       |
| 2.0 LPM | 37.85                                | 4.06                       |
| 2.5 LPM | 37.36                                | 0.17                       |

### 4. Conclusion

This study was conducted to determine the effect of changes in the incoming hot fluid flow on the Heat Exchanger performance with opposite flow sides. Three-layer concentric tube of HE was drawn using Ansys Design Modeler, and analyzed with a numerical approach (CFD). This research used FLUENT software to simulate the performance of three-layer concentric tube of HE. The debit fluid differs at 1.5LPM, 2.0LPM and 2.5LPM. The results showed that the effect hot fluid debit directly affects the debit temperature of both fluids, both sides of tubes 1 & 3 for cold fluid and tube 2 for hot fluid. By using this temperature, then the effectiveness and overall heat transfer rate are calculated. In
line with the increase of inlet hot fluid, the effectiveness of the device decreases temporarily, vice versa.

References
[1] Kakac S and Hongtan L 2002 Heat Exchangers: Selection, Rating, and Thermal Design (Florida: CRC Press)
[2] Incropera F P et al 2011 Fundamentals of Heat and Mass Transfer 7th ed (New York: John Wiley & Sons, Inc)
[3] Holman J P 2010 Heat Transfer Tenth Edition (New York: McGraw-Hill Companies, Inc)
[4] Frank Kreith 1877 Principles of Heat Transfer Fourth Edition (New York: Harper and Row)
[5] Cengel Yunus A 2003 Heat Transfer: A Practical Approach Second Edition (New York: McGraw-Hill Companies, Inc)
[6] Ambarita H 2017 Buku mata kuliah CFD (Computational Fluid Dynamic) pp 1-8
[7] ANSYS Fluent Version 18 1 Fluent 18 1 user’s guide ANSYS Inc
[8] Radulescu S, Negoita I L and Onutu I 2012 Heat transfer coefficient solver for a triple concentric tube heat exchanger in transition regime REV.CHIM p 63
[9] M, Rikanto, 2018 Rancang Bangun APK Tiga Lapis Tabung Konsentr is (Medan: Universitas Sumatera Utara)
[10] Munson, Bruce R, et al 2009 Fluid Mechanics Sixth Edition (New York: John Wiley & Sons, Inc)
[11] Hundry G F, A R Trott and T C Welch Refrigeration and Air Conditioning Fourth Edition (B H)