A CFD simulation and experimental study: predicting heat transfer performance using SST k-ω turbulence model

C D Widiawaty\textsuperscript{1,2,a}, A I Siswantara\textsuperscript{2b}, Budiarsa\textsuperscript{2}, G G R Gunadi\textsuperscript{1,2}, H Pujowidodo\textsuperscript{1,3}, M H G Syafei\textsuperscript{2}

\textsuperscript{1}Department of Mechanical Engineering, Politeknik Negeri Jakarta, Depok 16424 Indonesia
\textsuperscript{2}Department of Mechanical Engineering, Universitas Indonesia, Depok 16424, Indonesia
\textsuperscript{3}Center for Thermodynamics, Engine, and Propulsion, BPPT, Serpong 15314, Indonesia

*Email: candra.damis.widiawati@mesin.pnj.ac.id

Abstract. The heat exchanger is the most common equipment in the industry. Nowadays, CFD simulation has been used widely to predict the performance of heat transfer. The accuracy of the CFD depends on several factors such as simulation parameters, models, and quality of the grid. The heat exchanger in industrial application has big heat transfer capacity, since it consists lot of tube and big shell size. In this study, the fluid heat transfer on the tube is represented by uniform wall tube temperature. The objective of this work is to evaluate the performance of the SST kω turbulent and by applying heat transfer simulation of fluid on the tube side represented by uniform wall tube temperature. This present work compared CFD simulation and Experimental study. While the working fluids which are used in the experimental study are cooking oil in the shell and exhaust gas in the tubes. The constant of SST kω turbulent are cmu: 0.09, c1: 1.44, c2: 1.92, and sigma epsilon 1.3. The CFD simulation result shows in 50 s simulation that the temperature of cooking oil is from 48 °C to 54 °C. This tendency has good agreement with the experimental study.

1. Introduction

Recently, in the industrial field shell and tube heat exchanger has been utilized extensively since it has a good ability in high temperature and pressure, good resistance in corroded fluid, easy in maintenance and manufacturing process [1]. The general procedure in designing heat exchanger based on A. Jones [2] (cited in[3]) started by collecting fluid parameters such as temperature and mass flow rate, determining the heat load at the tube and shell sides, evaluating effective corrected temperature difference, determining heat exchanger configuration such as shell and tube type, tube dimension, baffle distance and tube bundle. Also, the flow direction of cold and hot fluids influence on the performance of heat transfer. Both heat exchanger configuration and fluid flow direction impact the fluid dynamic and heat transfer which finally will have an impact on heat exchanger performance.

In its development, the design and reverse engineering of the heat exchanger can be done conventionally based on the basic calculation of heat transfer. For example, using Kern method, Delaware method, Effectiveness - Number of transfer unit method (ε-NTU Method), and Logarithmic mean temperature difference method (LMTD Method). Kern method has been used in reverse engineering of cooler at PLTA Jatiluhur [4], by changing the tube diameter and tube number. The existing equipment has 16 mm diameter and 88 tubes, the new design has 15 mm diameter and 124 tubes. The Kern Method used to predict the heat transfer and the pressure drop. LMTD is a simple method in designing a heat exchanger, it can be used if the only inlet and outlet fluid temperature is
determined [5]. R. Rajesh et.al [5] used LMTD method for designing perforated heat exchanger at utilization of exhaust engine.

Advances in computer technology and modelling mathematics make it easier for engineers to design and reverse engineer using computational programs such as HTRI, Hysis, and Computational Fluid Dynamic. Recently, computational velocity and numerical techniques have been improved significantly. Furthermore, Computational Fluid Dynamics (CFD) was introduced by scientists gradually since it is meaningful to assess the operation of numerous heat exchangers in terms of heat and mass transfer [6]. Practically, CFD methods show uncertainties/errors which are similar to the experimental studies. Apart from the user errors, CFD includes technical errors which are parametric, numerical discretization (e.g. spatial mesh size and temporal resolution) and modelling. In the mathematical model, overgeneralization of flow phenomena placed the most substantial modeling errors. Numerical discretization errors ranked second due to inadequate mesh resolution. While the parametric errors due to less comprehensive information of the geometry and properties of fluid[7]. The study which had been done by Candra et. al showed that grid quality influences the accuracy of fluid temperature on shell and tube heat exchanger[8]. Wangetal [9](cited in [6]) utilized commercial CFD FLUENT to inspect overall heat transfer rate and the pressure drop of the shell with helical baffles. The authors acquired that the CFD prediction tends to the experimental studies. Ozden and Tari [10] proposed the use of CFD simulation for predicting the heat and mass transfer behavior the shell side of a small heat exchanger.

Nowadays, lots of researchers evaluate the impact of turbulent model on the accuracy of CFD simulation results. There are various turbulent models on CFD program such as k-epsilon standard, k-ω epsilon, RNG and RSM [11]. Leutwyler and Dalton stated that compared to the Spalart-Allmaras model, the standard k-w model, the RNG k-e model and the standard k-w model, it is showed that the SST k-ω is the most appropriate model to evaluating the pressure distribution on the valve disk [7]. Additionally, an experiment on the turbulence model was conducted to prove that the standard k-ε model performs excellent outcomes for the velocity profile together with heat transfer in shell and tube heat exchanger [6]. Another CFD model simulates the thermo-hydraulic performance by using the RNG k-ε turbulence model employing the standard wall functions. The simulation studies showed that the pressure drop of the shell side is likely to the experimental results. The deviation is less than 15% between those simulation and experimental results [12]. Conversely, minor deviances were obtained with the predictions of the SST k-ω model. Moreover, in case of the cross-flow over tube bundles the standard k-ε and the SST k-ω model expects reasonably accurate within band +/- 5% [6].

Previous research showed that turbulent model impact on the accuracy of simulation results. Moreover, it is proven that the SST k-ω model gives a good result with +/- 5% error [6]. In this study, the prediction of heat transfer is represented by the temperature of cooking oil in the outlet and SST k-ω turbulent model is used in this simulation. The heat transfer of exhaust gas on tube is simplified by applying uniform tube wall temperature.

This simplification used to reduce the time and cost of the simulation process, especially in the large-dimensional STHE design. Hence, the novelty of this research is the accuracy test of the SST k-ω model and the use of wall temperature on the tube to the simulation results. The accuracy of the simulation result is known by comparing the cooking oil temperature on the outlet with the experimental result. Statistical approach is used to determine the uncertainty of measurements. Statistical method of repeated measurement for one measurement parameter is used in this study.

2. Experimental setup
The main component of the experiment setup consists of the heat exchanger 450 mm in length x 250 mm in width x 250 mm in thickness. A heat source is from the exhaust gas of the engine. The exhaust gas flows along the tube with 0.0011 kg/s. A pump flows the oil along with the shell with 0.015 kg/s in mass flow rate. K-type thermocouples made of Chromel/Alumel are used to measure temperature. There are four k-type thermocouples to measure the temperature, i.e. inlet and outlet of exhaust gas and inlet and outlet of cooking oil. Heat exchanger surface is isolated to reduce heat losses to the
surrounding.

**Figure 1.** Schematic diagram of heat exchanger (1) oil pump (2) feed cooking oil storage (3) k-type thermocouple (4) k-type thermocouple (5) Laptop (6) DAQ (7) Oil inlet to He (8) k-type thermocouple (9) k-type thermocouple (10) Oil receiver (11) digital anemometer (12) engine

3. **CFD Simulation**

3.1. **Governing Equation**

This study analyzes heat transfer in 3-dimension flow, transient and turbulence. Governing equations as followed:

Governing transport equation[13]

Continuity:

\[ \frac{\partial \rho}{\partial t} + \text{div}(\rho u) = 0 \]  \hspace{1cm} (1)

Momentum:

x – momentum:

\[ \frac{\partial (\rho u)}{\partial t} + \text{div}(\rho u u) = -\frac{\partial p}{\partial x} + \text{div}(\mu \text{ grad } u) + S_{Mx} \]  \hspace{1cm} (2)

y – momentum:

\[ \frac{\partial (\rho v)}{\partial t} + \text{div}(\rho v u) = -\frac{\partial p}{\partial y} + \text{div}(\mu \text{ grad } v) + S_{My} \]  \hspace{1cm} (3)

z – momentum:

\[ \frac{\partial (\rho w)}{\partial t} + \text{div}(\rho w w) = -\frac{\partial p}{\partial z} + \text{div}(\mu \text{ grad } w) + S_{Mz} \]  \hspace{1cm} (4)

Energy:

\[ \rho \frac{\partial E}{\partial t} = \text{div}(\rho u) + \left[ \frac{\partial (u u)}{\partial x} + \frac{\partial (v v)}{\partial y} + \frac{\partial (w w)}{\partial z} \right] + \text{div}(k \text{ grad } T) + S_{E} \] \hspace{1cm} (5)

General equations of turbulence SST k - \omega [6]:

\[ \frac{\partial k}{\partial t} + \frac{\partial (u_k)}{\partial x} = \frac{P_k}{\rho} - \beta^* \omega_k + \frac{1}{\rho} \frac{\partial}{\partial x} \left( \mu + \sigma_k \mu \right) \frac{\partial k}{\partial x} \] \hspace{1cm} (6)

\[ \frac{\partial (u_k)}{\partial t} + \frac{\partial (u_k u_k)}{\partial x} = \frac{P_k}{\rho} - \beta^* \omega_k + \frac{1}{\rho} \frac{\partial}{\partial x} \left( \mu + \sigma_k \mu \right) \frac{\partial k}{\partial x} \] \hspace{1cm} (7)

\[ F_1 = \tanh(a_1 q) \] \hspace{1cm} (8)

\[ a_1 = \min \left[ \max \left( \frac{\sqrt{\kappa}}{0.09y_0}, \frac{500}{y_0} \right), \frac{4\rho_m v^2 k}{CD_{m0}v^2} \right] \] \hspace{1cm} (9)
\[ v_l = \frac{a_1 k}{\text{max}(a_1 \omega_S F_2)} \]
\[ S = \sqrt{\left( \frac{\partial(u)}{\partial x_j} \right)^2 + \left( \frac{\partial(u)}{\partial x_i} \right)^2} \]
\[ F_2 = \tanh(a_1^4) \]
\[ a_2 = \text{max} \left( 2 \sqrt{\frac{k}{0.09 \omega_y}} \right) \]
\[ C_D k_\omega = \text{max} \left( 2 \rho \frac{1}{\sigma_w^2} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}, 10^{-10} \right) \]

3.2. Boundary Condition

This study analyses the accuracy of the result of SST k - \( \omega \) model and simplifying the process conditions of transferring tube flows into a constant temperature wall tube in the system (Fig 2). The outer wall is adiabatic. This simulation uses a 3D model. Cartesian grid is used with a total grid number of 2,000,000. Cooking oil with a temperature of 38 °C flows from the bottom of the nozzle towards the shell and exits on the top of the nozzle. Flue gas flows along the tube. The shell and tube flow is a cross-flow type. Grid and Boundary condition simulation can be seen in Figure 2 and simulation parameters can be seen in Table 1.

![Figure 2](image)

**Figure 2.** (a) grid (b) boundary condition

| Description                | Value       | Note                                               |
|----------------------------|-------------|---------------------------------------------------|
| Model                      | Transient, Incompressible, Subsonic                |
| Time Set                   | 0.0001 s    |                                                   |
| Turbulence                 | SST-k\( \omega \)                                 |
| Turbulence Model Coefficient| \( c_\mu = 0.09 \); c1 = 1.44; c2 = 1.92; \( \sigma_e \) = 1.3 |
| Fluid                      | Cooking oil                                        |
| Density                    | 880 kg/m³   |                                                   |
| Inlet Velocity             | 0.0456 m/s  |                                                   |
| Outlet                     | Pressure                                            |
| Temperature Inlet          | 38 °C       |                                                   |
| Tube temperature           | 65 °C       |                                                   |
4. Result and Discussion

The main components of the experiment are the engine generator, heat exchanger, and a pump. The experiments are conducted until the fluctuation of gas and cooking oil temperature are quite stable. The temperature range of flue gas inlet temperature is 181°C - 189°C and cooking oil outlet temperature is 46°C - 47°C. The cooking oil temperature at the outlet are analysed by statistical approach to obtain the uncertainty of measurements [14]. Statistical calculation for cooking oil temperature at the outlet includes mean temperature 46.57 °C, standard deviation 0.306, and Standard error of the mean 0.0385. Based on level of confident 95%, then the uncertainty of temperature is 0.077, so T=(46.57 °C ± 0.0770).

Simulation results are verified and validated. Cooking oil flows on the shell side. Inlet cooking oil flows from the bottom of the nozzle, flows through the baffles, and then comes out on the upper side of the nozzle. The simulation flow phenomenon is under the real condition of the system. The streamline of the cooking oil can be seen in Figure 3.

![Figure 3. Stream line of cooking oil](image)

Cooking oil comes from the inlet to the path between the tube bundles. It appears that the baffles lead the flow of cooking oil. At the inlet side, the cooking oil flows upwards then downward furthermore continues until the outlet. Figure 4 shows the tube bundle configuration and a cutting section.

![Figure 4. Tube bundle configuration](image)

The velocity distribution in the heat exchanger affects the heat transfer process significantly. Therefore, the velocity distribution is verified. The simulation results of the velocity distribution can be seen in Figure 5 and Figure 6.
Figure 6 shows the velocity distribution contour of the cross-section of the nozzle. It is shown that the largest velocity gradient is in the middle of the nozzle while the lowest is in the wall. The simulation shows that the velocity ranges from 0.005 to 0.08 m/s as shown in Figure 7. This phenomenon is following the concept of the boundary layer. It can have been seen that the viscous is dominant in nozzle. The fluid consists of layers, so there is a shift in each layer. Viscosity causes the fluid layer closest to the wall to rub against the wall, resulting in shear stress that causes a velocity gradient [14].

The velocity of cooking oil on the shell is on average of 0.01 m/s which is smaller than the velocity on the inlet, this is caused by the area of the gap in the shell is greater than the area of the inlet of the nozzle. The difference in the cross-sectional area causes a change in velocity with a range of 0.001 to 0.0811 m/s. This phenomenon is in accordance with the theory of continuity. The velocity of cooking oil in the inlet and outlet nozzle is greater than in the tube bundles. To the heat exchanger geometry, the average cross-sectional area in the tube bundle is greater than the cross-sectional area at the inlet and outlet nozzle. The nozzle cross-section is 2025.08 mm² in area and the bundle is 15241.9 mm² in the average cross-sectional area of the tube.
Tube bundle configuration and mass rate parameters cause cooking oil flowing on the space between the tube bundle and the shell (Figure 7). This is indicated by the velocity that tends to be higher in this particular area. After 30 seconds of simulation, there is no significant velocity contour change. This shows that the flow began steadily. Therefore, it can be continued with temperature distribution analysis. Analysis of temperature distribution can be seen in Figure 8.

![Figure 8. Cooking oil temperature distribution in shell](image-url)
Figure 8. shows the temperature distribution in sections B, C, D, E, and F for 50 seconds. In general, changes in temperature distribution in each section after 40 seconds are not significant. In section B, there is a gradient temperature; further away from the inlet cooking oil temperature is increasing. However, high-temperature cooking oil fills in the tube gap compared to the area between the tube bundle and the shell. The temperature in section B reaches 60°C. This phenomenon occurs because the tube is a source of heat. Consequently, the fluid around the tube will transfer the heat between fluid molecules faster than the fluid that is far from the tube. Also based on the simulation of the velocity distribution [Fig 8], it is seen that the fluid velocity between the tube gaps is less than the fluid between the tube bundle gap and the shell. The slow-motion of fluid velocity causes the contact time of the fluid molecule with the heat source to be longer.

Cooking oil temperature distribution in sections C, D, E, and F tends to be uniformly distributed on the gap between the tubes. However, there is still a temperature gradient between the fluid in the tube gap and the fluid in the gap between the tube bundle and the shell. The temperature of the Cooking oil in the gap between the tube bundle and the shell is in the range of 45 – 48°C. While the Cooking oil temperature of in the tube gap is about 60 – 63°C. Distribution of temperature along the shell causes temperature gradient in the nozzle outlet [Section G]. Qualitatively the simulation results of temperature distribution are in accordance with the literature [6]. The temperature gradient in the nozzle is 48 – 54°C. The calculation method of control volume is discretization, then each cell has its temperature. Furthermore, the average cooking oil temperature at the outlet is 51°C.

The average temperature based on simulation is validated with an experimental study which is \( T = (46.57°C \pm 0.0770) \) atau 46.49 - 46.65°C. While the average fluid temperature distribution [Fig 8] is 51°C. Thus the use of SST k-\(\omega\) turbulent models and wall temperatures of 65°C at 50 second gives quite high accuracy results in the analysis of the velocity distribution, but there is a different between simulation to experimental \(\pm 9.7\)\% for the temperature distribution.

5. Conclusion
This study analyzes the accuracy of the selection of SST k-\(\omega\) turbulent models and wall temperatures in systems with heat source temperatures that fluctuate from 178 - 189°C. The 3D model is used in this simulation. The simulation of velocity distribution is in accordance with the boundary layer concept. Based on the statistical approach, then the temperature of the experimental result is \( T = (46.57°C \pm 0.0770) \) and the simulation average temperature is 51°C. Thus the simulation parameters create a temperature difference of \(\pm 9.7\)%.

Acknowledgement
The author would like to thank “KEMENTERIAN RISET, TEKNOLOGI, DAN PENDIDIKAN TINGGI INDONESIA” for funding this research through “PDUPT NKB-2834/UN2.RST/HKP.05.00/2020 and PT. CCIT for a licence of CFDSof®.

References
[1] Shah R K and Sekulic D P 2003 Fundamentals of heat exchanger design John Wiley & Sons.
[2] Jones A E, 2002 Thermal design of the shell-and-tube: exchanger sizing takes a trial and error approach. Use these guidelines to center your aim.(Cover Story) Chem. Eng. 109 60–66.
[3] Nitsche M and Gbadamosi R O 2015 Heat exchanger design guide: a practical guide for planning, selecting and designing of shell and tube exchangers Butterworth-Heinemann.
[4] Gaos Y S and Widiawati C D, 2017 REVERSE ENGINEERING OIL COOLER DOUBLTUBE PLTA JATILUHUR AME (Aplikasi Mek. dan Energi) J. Ilm. Tek. Mesin 3 1 41–44
[5] Rajesh R, Senthilkumar P and Mohanraj K 2018 Design of heat exchanger for exhaust heat recovery of a single cylinder compression ignition engine J. Eng. Sci. Technol. 13 2153–2165
[6] Pal E, Kumar I, Joshi J B and Maheshwari N K, 2016 CFD simulations of shell-side flow in shell-and-tube type heat exchanger with and without baffles Chem. Eng. Sci. 143 314–340
[7] Duan Y, Jackson C, Eaton M D and Bluck M J 2019 An assessment of eddy viscosity models on
predicting performance parameters of valves *Nucl. Eng. Des.* **342** 60–77

[8] Widiawaty C D, Gunadi G G R and Syuriadi A, 2017 PEMODELAN DAN ANALISIS KINERJA SHELL AND TUBE HEAT EXCHANGER DENGAN METODE CFD *J. Poli-Teknologi* **16** 3

[9] Wang Q, Chen Q, Chen G and Zeng M 2009 Numerical investigation on combined multiple shell-pass shell-and-tube heat exchanger with continuous helical baffles *Int. J. Heat Mass Transf.* **525** 1214–1222

[10] Ozden E and Tari I 2010 Shell side CFD analysis of a small shell-and-tube heat exchanger *Energy Convers. Manag.* **51** 1004–1014

[11] Davidson L 2015 Fluid mechanics, turbulent flow and turbulence modeling

[12] Lei Y, Li Y, Jing S, Song C Lyu Y and Wang F 2017 Design and performance analysis of the novel shell-and-tube heat exchangers with louver baffles *Appl. Therm. Eng.* **125** 870–879.

[13] Versteeg H K and Malalasekera W 2007 *An introduction to computational fluid dynamics: the finite volume method* Pearson education

[14] Harinaldi D I and Eng M 2005 Prinsip-prinsip statistik untuk teknik dan sains *Jakarta: Erlangga*