Journal of Complex Flow, Vol. 6 No. 2 (2024) 1-9

FAZ Publishing
JCF

Journal of Complex Flow

Journal homepage: www.fazpublishing.com/jcf

e-ISSN: 2672-7374

Double Straight Tube Counter-Flow Heat Exchanger for Optimum Heat Transfer by Numerical Simulation

Wan Mahafiz Rosni[1, *], Azlan Adam[1], Muhammad Hazzaruddin Jumaidi[2], Elron Freassier Gornica[2], Wan Muhammad Irfan Wan Mokhtar[2], Imran Basri Shamih[2], Nurul Humaira Ainaa Setia Jaya[2]

[1] Department of Mechanical Engineering,
Polytechnic Kuching, 93050 Kuching Sarawak, MALAYSIA
[2] Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia,
Parit Raja, Batu Pahat, 86400, MALAYSIA

*Corresponding Author

Received 17 August 2024;
Accepted 25 Nov 2024;
Available online 15 Dec 2024

Abstract: The study provides a complete computational fluid dynamics (CFD) analysis of a twin straight tube counter-flow heat exchanger. The study sought to evaluate the temperature, velocity, and pressure distributions within the heat exchanger under a variety of input circumstances for hot and cold fluids. The key goals were to predict heat transfer and flow behavior with advanced CFD techniques such defined boundary conditions, geometric modelling, and producing a computational grid. The study technique used CAD software to build the heat exchanger shape and implemented the k-ω turbulence model in CFD simulations. The simulation results were thoroughly tested against experimental data to demonstrate the CFD model's reliability and accuracy. The average errors were 0.52% for the hot fluid outlet temperature, 8.15% for the cold fluid outlet temperature, and 9.73% for the overall heat transfer rate, with the hot fluid outlet temperature being notably well-matched. The maximum and minimum heat transfer rate obtained from the simulation is 1521.80 W and 1390.88 W respectively. The maximum hot and cold fluid temperature are 62.17°C and 18.79°C respectively. This extensive CFD analysis of the twin straight tube counter-flow heat exchanger gives vital insights into its performance and thermal properties, helping to optimize and enhance this critical industrial equipment. The successful validation of numerical predictions versus actual data emphasizes the need of combining computational and experimental approaches for designing and analyzing heat exchangers. The combination of these techniques can lead to a better understanding of the complicated fluid flow and heat transfer processes within the heat exchanger, allowing for more efficient and dependable designs to be built.

Keywords: CFD Simulation, double straight tube counter-flow, heat exchanger, k-ω turbulence

1. Introduction

Heat exchangers are indispensable in a wide range of industrial processes, facilitating the transfer of thermal energy between fluids while maintaining their separation. Among the various heat exchanger designs, the double straight tube counter-flow configuration is particularly notable for its versatility and efficiency. These devices operate based on fundamental thermodynamic principles, adhering to the second law by transferring heat from regions of higher temperature to lower temperature without mixing the fluids. In this design, convection and conduction are the primary mechanisms of heat transfer, with radiation playing a minimal role [1].

The double straight tube counter-flow heat exchanger is characterized by the flow of hot and cold fluids in opposite directions within straight, concentric tubes. This counter-flow arrangement maximizes the temperature gradient between the fluids along the length of the exchanger, thereby enhancing the overall heat transfer efficiency. The straight tube design not only simplifies the manufacturing and maintenance processes but also supports high-pressure applications and scenarios requiring significant temperature differentials [2]. This makes double straight tube counter-flow heat exchangers highly suitable for a variety of industrial applications, including chemical processing plants, steam power generation, and household appliances such as refrigerators and car radiators.

*Corresponding author: mahafez@poliku.edu.my

In recent years, the role of computational fluid dynamics (CFD) in the design and analysis of heat exchangers has become increasingly important. Software tools like FLUENT, CFX, and others enable engineers to simulate fluid flow, heat transfer, and other critical parameters. These simulations are crucial for optimizing the performance and efficiency of heat exchangers. However, the accuracy of CFD simulations is heavily dependent on the validation of these models through experimental testing, especially due to the complexities associated with turbulent flow modelling [3].

Heat exchangers play a pivotal role in diverse industrial processes, facilitating the transfer of thermal energy between fluids while maintaining their separation. Among the array of heat exchanger designs, the double-pipe counter-flow configuration stands out for its versatility and efficiency [4]. Heat exchangers operate based on the fundamental principles of thermodynamics. According to the second law of thermodynamics, the energy will be transformed in a form of heat from higher to lower temperature region without mixing the fluids [5]. While radiation plays a minimal role, convection and conduction are the predominant heat transfer mechanisms employed within these devices [6].

Double-pipe counter-flow heat exchangers, characterized by the flow of hot and cold fluids in opposite directions within concentric tubes, find extensive usage across a spectrum of industries. Their construction allows for an efficient rate of heat transfer in high-pressure applications and scenarios requiring a wide range of temperature differentials. The applications of heat exchanger at the industry include chemical processing plants, heating, ventilation, and air-conditioning systems (HVAC) and vehicles automotive applications such as radiator, intercoolers, and oil coolers [7].

In computational fluid dynamics (CFD), software tools such as FLUENT and CFX have become indispensable for the design and simulation analysis of heat exchangers. These software tools enable engineers to predict the fluid flow, heat transfer rate, and other critical parameters, aiding in optimizing heat exchanger performance and efficiency [8]. However, the accuracy of CFD simulations necessitates validation through experimental testing, particularly due to the complexities of turbulent flow modelling.

The primary objective of the study outlined in the referenced article is to validate CFD simulations of double-pipe counter-flow heat exchangers through comparison with experimental data by using K-Omega standard viscous simulation model. This involves generating lab models using CAD software, conducting simulation fluid flow analysis using CFD Ansys fluent software, and comparing numerical results with measured temperatures of hot and cold fluids at the outlets and inlets of the double pipe heat exchanger.

2. Literature Review and Related Theory

Heat exchangers play a critical role in various industrial processes by facilitating the transfer of thermal energy between fluids without allowing them to mix. The double straight tube counter-flow heat exchanger, a prominent design among various heat exchanger configurations, is renowned for its versatility and efficiency. This review explores the theoretical foundations, design principles, applications, recent advancements in the computational modelling of double straight tube counter-flow heat exchangers, and the specific equations used in CFD simulations, particularly those performed using ANSYS software [9].

The operation of double straight tube counter-flow heat exchangers is grounded in fundamental thermodynamic principles. According to the second law of thermodynamics, heat flows from regions of higher temperature to regions of lower temperature. In these exchangers, convection and conduction are the primary mechanisms facilitating this energy transfer, while radiation plays a minimal role due to the enclosed nature of the system [10].

Double straight tube counter-flow heat exchangers are designed with hot and cold fluids flowing in opposite directions within concentric straight tubes. This counter-flow arrangement maximizes the temperature gradient along the length of the exchanger, thereby enhancing the overall heat transfer efficiency [11]. Compared to parallel flow designs, counter-flow heat exchangers achieve higher heat transfer rates and can handle greater temperature differentials, making them suitable for high-pressure applications.

The straight tube design simplifies manufacturing and maintenance while offering robustness under high-pressure conditions. This makes such heat exchangers particularly advantageous for applications in chemical processing plants, steam power generation, and household appliances such as refrigerators and car radiators [12]. In recent years, Computational Fluid Dynamics (CFD) has become an indispensable tool in the design and analysis of heat exchangers. CFD software like ANSYS enables engineers to simulate fluid flow and heat transfer processes, providing insights into performance optimization that are otherwise challenging to obtain through experimental means alone [13].

CFD simulations help predict temperature distributions, pressure drops, and heat transfer coefficients within heat exchangers. However, the accuracy of these simulation hinges on their validation against experimental data. This is especially critical when modelling turbulent flows, which introduce significant complexity [14]. These equations account for the effects of turbulence on the flow field, enabling more accurate predictions of heat and momentum transfer in turbulent regimes.

The reliability of CFD models is contingent upon rigorous validation through experimental testing. Validating these models ensures that simulations accurately reflect real-world performance, thus enhancing the credibility and utility of CFD in the design process. The k-ω turbulence model is frequently employed in these simulations to address the challenges associated with turbulent flow modelling [15].

Experimental validation typically involves creating lab-scale models using CAD software like SOLIDWORKS, followed by detailed flow analysis using ANSYS CFD tools. By comparing the numerical results with measured outlet temperatures of hot and cold fluids, researchers can refine their models to ensure precise and dependable outcomes [16]. Double straight tube counter-flow heat exchangers are integral to various industrial applications due to their efficient heat transfer capabilities and robust performance. Their applications span from chemical processing and steam power generation to everyday household devices, highlighting their versatility and importance [17].

Recent advancements in CFD and experimental techniques have significantly contributed to the optimization of these heat exchangers. Enhanced simulation capabilities allow for more accurate predictions of performance, while experimental validations ensure these predictions hold under practical condition [18]. The theory underlying heat exchangers revolves around the principles governing the exchange of heat between two fluids with disparate temperatures. These devices are engineered to enable this heat transfer by employing a barrier, typically a solid wall, between the fluids. The fundamental aim of a heat exchanger is to optimize heat transfer efficiency while mitigating energy dissipation [19].

Different kinds of heat exchangers exist, including shell and tube, fin-tube, plate, to compact heat exchangers. These variations differ in their configuration and how fluids flow within them, encompassing arrangements like parallel flow, counter flow, and cross flow (IQS). In this project, the model of the heat exchanger that was used was a double-pipe counter flow heat exchanger.

Furthermore, heat exchangers have several thermodynamics governing heat exchangers that involve principles such as energy conservation, the first and second laws of thermodynamics, and entropy considerations [20]. Three main heat transfer processes occur convective heat transfer from the fluid to the inner tube wall, conductive heat transfer through the tube wall, and convective heat transfer from the outer tube wall to the surrounding fluid. Below are the thermodynamic concepts related to heat exchangers [21].

Second Law of Thermodynamics: This law introduces the concept of entropy, stating that in any energy transfer or transformation, the total entropy of a closed system will always increase over time until it reaches equilibrium. In the context of double pipe heat exchangers, the second law guides the direction of heat transfer and the efficiency of the process. In these systems, heat naturally flows from the hotter fluid in the inner pipe to the cooler fluid in the outer pipe, and the efficiency of heat exchange depends on minimizing entropy generation. This is achieved by optimizing the flow rates and temperatures of the fluids to enhance heat transfer while reducing thermal resistance and energy dissipation [22].

In understanding the importance of the heat exchanger applications in turbomachinery, using an accurate method is very crucial for their computational fluid dynamics (CFD) simulation. Choosing the right turbulence modelling method is important for this project since it has a big influence on how accurate the outcomes are. The student has decided to use the k-omega turbulence model for this project, which is a commonly used method in the turbomachinery and aerospace industries. A variety of choices and sub-models of the k-omega model may be customized to replicate certain flow phenomena found in heat exchanger simulations. Several sub-models/options of k–ω: compressibility effects, transitional flows and shear-flow corrections.

By carefully selecting and implementing the appropriate k-omega turbulence model options, the project team aims to ensure that the CFD simulations of the heat exchangers accurately reflect the real-world behavior and performance, serving as a valuable tool for the design, development, and optimization of these critical components in the turbomachinery industry [23].

3. Methodology and Simulation Setup

3.1 Governing equation

The governing equation of SIMPLE used for the simulation. Starting with the initial guess for p^*, u^*, v^*, ϕ^*, the following discretized momentum equation will be solved.

$$a_{i,j}\, u^*_{i,j} = \sum a_{nb}\, u^*_{nb} + \left(p^*_{i-1,j} - p^*_{i,j-1}\right) A_{i,j} + b_{i,j} \tag{1}$$

$$a_{i,j}\, v^*_{i,j} = \sum a_{nb}\, v^*_{nb} + \left(p^*_{i-1,j} - p^*_{i,j-1}\right) A_{i,j} + b_{i,j} \tag{2}$$

Solve pressure correction equation (u^*, v^*);

$$a_{i,j}\, p'_{i,j} = a_{i-1,j}\, p'_{i-1,j} + a_{i+1,j}\, p'_{i+1,j} + a_{i,j-1}\, p'_{i,j-1} + a_{i,j+1}\, p'_{i,j+1} + b'_{i,j} \tag{3}$$

Correct pressure and velocities (p');

$$p_{i,j} = p^*_{i,j} + p'_{i,j} \tag{4}$$

$$u_{i,j} = u^*_{i,j} + d_{i,j}(p'_{i-1,j} - p'_{i,j}) \tag{5}$$

$$v_{i,j} = v^*_{i,j} + d_{i,j}(p'_{i,j-1} - p'_{i,j}) \tag{6}$$

Solve all other discretized transport equations;

$$a_{i,j}\, \phi_{i,j} = a_{i-1,j}\, \phi_{i-1,j} + a_{i+1,j}\, \phi_{i+1,j} + a_{i,j-1}\, \phi_{i,j-1} + a_{i,j+1}\, \phi_{i,j+1} + b_{\phi i,j} \tag{7}$$

A two-transport-equation model solves for k and ω, depending on the particular dissipation rate (ε/k) [24]. This is the default k-ω model. Shows greater performance for wall-bounded and low Reynolds number flows. Demonstrates the ability to forecast change. Options include transitional, free shear, and compressible flows. Excellent performance for wall-bounded boundary layers, free shear, and low Reynolds number flow. Suitable for complicated boundary layer flow with an unfavourable pressure gradient and separation. Can be utilized for transitional flows (although tends to forecast early transitions). Separation is often expected to be extreme and early.

$$\mu_T = f(\frac{\rho k}{\omega}) \tag{8}$$

$$\mu_T = \alpha * \rho \frac{k}{\omega} \tag{9}$$

$$\rho \frac{Dk}{Dt} = \tau_{ij} \frac{\partial \overline{u_i}}{\partial x_j} - \rho \beta^* f_{\beta^*} k \omega + \frac{\partial}{\partial x_j}\left[(\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial k}{\partial x_j}\right] \tag{10}$$

$$\rho \frac{D\omega}{Dt} = \alpha \frac{\omega}{k} \tau_{ij} \frac{\partial \overline{u_i}}{\partial x_j} - \rho \beta f_\beta \omega^2 + \frac{\partial}{\partial x_j}\left[(\mu + \frac{\mu_t}{\sigma_\omega}) \frac{\partial \omega}{\partial x_j}\right] \tag{11}$$

Specific dissipation rate,

$$\omega \approx \frac{\varepsilon}{k} \propto \frac{1}{\tau} \tag{12}$$

The rate of heat transfer (Q) given as,

$$\dot{Q} = \dot{m}_h C_{ph}(T_{hi} - T_{ho}) = \dot{m}_c C_{pc}(T_{co} - T_{ci}) = A_t U_m \Delta T_{In} \tag{13}$$

The logarithmic mean temperature difference (LMTD) between the hot and cold fluids,

$$\Delta T_{In} = (\Delta T_O - \Delta T_L)\{\ln(\Delta T_O) - \ln \Delta(T_L)\}^{-1} \tag{14}$$

Effectiveness (ε) is defined as,

$$\varepsilon = \dot{Q}\dot{Q}_{max}^{-1} \tag{15}$$

The maximum possible heat transfer rate,

$$\dot{Q}_{max} = \left(\dot{m}C_p\right)_{min}(T_{hi} - T_{ci}) \tag{16}$$

Here, $\dot{m}_c$ and $\dot{m}_h$ are the mass flow rates of cold and hot fluids respectively; C_{pc} and $C_{ph}$are the specific heats of cold and hot fluids respectively; $(\dot{m}C_p)_{min}$ is the smaller of $\dot{m}_h C_{ph}$ and $\dot{m}_c C_{pc}$ for the hot and cold fluids; T_{hi} and $T_{ho}$are the inlet temperatures of hot and cold fluids respectively; T_{ho} and T_{co} are the outlet temperatures of hot and cold fluids respectively; $\Delta T_0 = T_{hi} - T_{co}$ and $\Delta T_L = T_{ho} -$; U_m is the overall heat transfer coefficient; $A_t = \pi L(D_i + d_o)$, is the total heat transfer area; d_i and d_o are the inner and outer diameters of the inner tube respectively; D_i is the inner diameter of the outer tube; $D_e = (D_i^2 - d_o^2)d_o^{-1}$, is the equivalent diameter; $D_h = D_i - d_0$, is the hydraulic diameter; $u_{mh} = 4\dot{m}_h(\pi\rho_h d_i^2)^{-1}$ and $u_{mc} = 4\dot{m}_c\{\pi\rho_c(D_i^2 - d_o^2)\}^{-1}$ are the average velocity of the hot and cold fluids respectively.

For heat exchangers, the total heat transfer is the key metric that describes the overall amount of thermal energy transferred between the fluid streams over a period of time. This total heat transfer depends on factors like the temperature difference, heat transfer area, and duration of operation. Accurately determining and maximizing the total heat transfer is critical for ensuring the heat exchanger meets its design objectives, such as reaching a target outlet temperature or delivering a required thermal load. Analyzing and optimizing total heat transfer is therefore a crucial part of heat exchanger engineering and performance assessment.

Referring to the heat transfer rate at the equation, the hot side will have this equation,

$$Q = WC_p(T_{in} - T_{out}) \tag{17}$$

The cold side will have this equation,

$$Q = WC_p(T_{out} - T_{in}) \tag{18}$$

3.2 Simulation Setup

Computational fluid dynamics (CFD) is concerned with the numerical solution of partial differential equations regulating the transfer of mass, momentum, and energy in fluids. The continuity equation is obtained by using the law of mass conservation. Momentum equations are obtained for velocity (u_j) in directions x_j (j = 1, 2, 3) using Newton's second rule of motion and Stoke's Stress Laws. The energy equation for transport temperature (T) or enthalpy (h) is obtained using the first law of thermodynamics and Fourier's law of heat conduction ($q_{i,cond} = -k\frac{\partial T}{\partial x_i}$). The continuity, momentum, and energy equations for moving fluids are [24].

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_j)}{\partial x_j} = 0 \tag{19}$$

$$\boldsymbol{\frac{\partial(\rho u_j)}{\partial t} + \frac{\partial(\rho u_j u_i)}{\partial x_j} = -\frac{\partial \rho}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu_{eff}\frac{\partial u_i}{\partial x_j}\right) + \rho B_i + S_{u_i}} \tag{20}$$

$$\boldsymbol{\frac{\partial(\rho h)}{\partial t} + \frac{\partial(\rho u_j h)}{\partial x_j} = \frac{\partial}{\partial x_j}\left\{\kappa_{eff}\frac{\partial T}{\partial x_j}\right\} + Q^m} \tag{21}$$

In this equation, p represents pressure, $h = C_p(T - T_{ref})$ represents enthalpy, B_i represents body force in x directions (i = 1, 2, 3), and Su_i represents viscous terms in addition to those indicated by $\frac{\partial}{\partial x_j}(\mu_{eff}\frac{\partial u_i}{\partial x_j})$; Q_m represents the volumetric rate of heat generation, μ_{eff} the effective viscosity, and k_{eff} the effective thermal conductivity. In laminar flows, μ_{eff} and k_{eff} are fluid properties, whereas in turbulent flows, they are flow properties. Turbulent fluxes are often seen in practical applications. The unstable laminar flow equations are turned into time-averaged equations by assuming fast and unpredictable variations in the mean value. Additional terminology originating from this process include Reynold's stresses, turbulent heat flux, and so on. A turbulent model's function is to represent these fluxes in terms of the flow's mean characteristics. A turbulent flow is computationally comparable to a laminar flow with a complex effective viscosity. The popular Launder and Spalding "two-equation models" of turbulence use, as one of the equations, the equation for the kinetic energy (k) of the fluctuating motion, which read

$$\boldsymbol{\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right) + G - \rho\varepsilon} \tag{22}$$

In this equation, Γ_k represents the diffusion coefficient for k, G is the rate of turbulence energy production, and ε is the kinematic rate of dissipation. The quantity $(G - \rho\varepsilon)$ represents the equation's net source term. A similar differential equation regulates the variable ε. SOLIDWORKS may be used to model the heat exchanger, while ANSYS (FLUENT) is used for flow analysis.

ANSYS (FLUENT) has turbulent models such as the k-ε standard model, k-ε RNG model, and k-ω standard model. Due to the absence of generally applicable turbulence models, CFD simulations must be compared to test data, and design validation through testing is inevitable.

3.3 Method and Boundary Condition

The process below illustrates the process of simulating a double straight tube counter-flow heat exchanger, starting from defining the problem and its objectives and continuing until obtaining the result. This flow will help ensure that the workflow runs smoothly. ANSYS software is a powerful tool for performing CFD simulations, allowing for detailed analysis and optimization of heat exchangers. The use of ANSYS in this study involves several steps:

1. CAD Model Creation: The geometry of the heat exchanger is created using CAD software like SOLIDWORKS, which is then imported into ANSYS for simulation.
2. Mesh Generation: ANSYS Meshing is used to create a high-quality computational grid that accurately captures the geometry and flow features.
3. Setting up the Simulation: Boundary conditions, material properties, and the k-ω turbulence model are defined within ANSYS Fluent or CFX.
4. Running the Simulation: The simulation is executed, and results are obtained for temperature distributions, pressure drops, and heat transfer coefficients.
5. Validation: The CFD results are validated against experimental data to ensure accuracy and reliability[24].

The previous work investigated double-pipe (straight) heat exchangers, testing for specified entrance temperatures and measuring the exit temperatures of both cold and hot fluids[24]. Their computer models closely matched the test results. Their mathematical model consists of a heat balance equation for both hot and cold fluids, which includes factors like mass flow rates, intake temperatures, and unknown output temperatures. The rate of heat transfer ($\dot{Q}$) was calculated using the product of (U_m), (A_t), and (ΔT_{ln}). They used an iterative technique to generate and solve two nonlinear equations that determined the output temperatures of the cold and hot fluids. They ran two types of tests: The heat exchanger (L) measured 750mm in length and had 1mm-thick inner and outer tubes. The inner diameters were 10 and 20 mm, respectively. The hot fluid's mass flow rate ($\dot{m}_h$) ranged from 0.0380 to 0.0405kg/sec, while the cold fluid's ($\dot{m}_c$) ranged from 0.0333 to 0.0527kg/sec. The cold fluid had an intake temperature of 10.6°C, while the hot fluid's inlet temperature ranged from 70.2 to 72.7°C. These settings will serve as the simulation's boundary conditions. The SIMPLE and Upwind Difference Schemes are utilized in the simulation. Table 1 shows the thermal parameters of the fluid (water), which are required for analysis using ANSYS (FLUENT).

Table 1: Thermal characteristics of water stated in numerical simulations of a double-pipe heat exchanger (straight)

Temperature (°C)	ρ (kg/m³)	C_p (J/kg. K)	μ (cP)
10.60	999.40	4088.70	12.458
70.40	977.12	4066.90	3.917
70.20	977.20	4066.90	3.928
70.20	977.20	4067.00	3.928
70.50	977.66	4067.00	3.911
71.10	976.71	4067.00	3.880
71.00	976.77	4067.00	3.885
71.40	976.53	4067.10	3.864
71.20	976.65	4067.10	3.875
72.00	976.18	4067.20	3.873
72.70	975.76	4067.40	3.798

Figure 1 and Figure 2 show the actual work of counter flow heat exchanger include the cold inlet, cold outlet, hot inlet and hot outlet.

Figure 1: Schematic diagram of double-pipe counter flow heat exchanger.

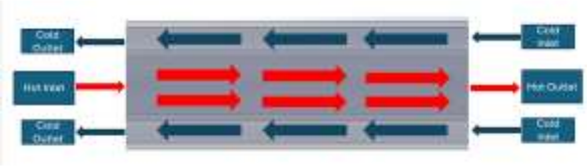


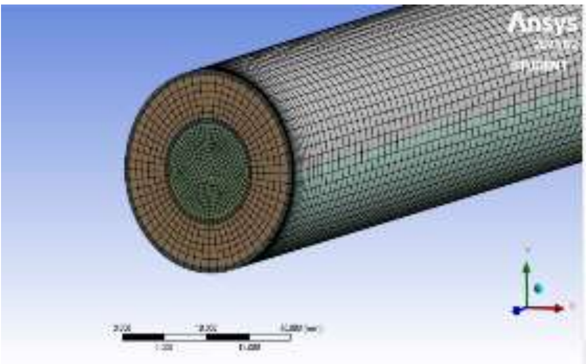
Figure 2: Schematic diagram of internal flow from the section view of double-pipe counter flow heat exchanger.

Figure 3 shows the double-pipe heat exchanger (straight tube) model generated using SOLIDWORKS. The geometry of the model is included in the figure below.

Figure 3: Design and dimension of double-pipe heat exchanger (straight tube)

The meshing of the three-dimensional model, performed using the ANSYS FLUENT R2 software package, is depicted in Figure 3**.** The steel pipe's parameters are density (ρ) = 8030 kg/m³, specific heat (c_p) = 502.48 J/kg·K, and thermal conductivity (k) = 16.27 W/m·K. The mass flow rates at the inlets of the double-pipe heat exchanger's inner and annulus tubes are characterized as velocity, while the outputs are defined as pressure. The tube walls are treated with a non-slip coating. Table 1 shows the thermal characteristics of both cold and hot water. To allow the counter-flow configuration, the annulus tube's inlet and outlet conditions are reversed from those of the inner tube.

The meshing technique employs a cut-cell approach, with rectangular cells for the tube wall and fluid domain. The 3D model consists of 367,086 components and 376,830 nodes. To optimize quality, set goal skewness to 0.9, smoothing to high, mesh metric to skewness, and min, max, average, and standard deviation to $2.133e^{-2}$, 0.87598, 0.3524, and 0.25992, respectively. ANSYS FLUENT R2 formulates the governing flow equations using a finite volume model. The momentum, turbulent dissipation rate, and temperature components are modelled via a second-order upwind technique. The SIMPLE algorithm is used for the pressure-velocity coupling system. At ensure convergence, the relative residual is set at 10^{-6} for all variables. Based on the previous comparison of the turbulent model used $k-\omega$ standard model shows the nearest result as the test data [24]. Thus, the $k-\omega$ standard turbulent model will be used for this simulation [25].



.

Figure 4: Meshing of the model.

The computational grid or mesh in Figure 4 is crucial for accurately capturing the flow and thermal characteristics. The grid was refined near the walls where gradients are expected to be high, ensuring accurate boundary layer resolution. The exact details of the mesh parameters (e.g., number of cells, grid independence studies) but a converged solution of residual scales is presented, indicating a well-resolved computational domain.

4. Results and Discussion

The validation of this study is done using k-ω standard turbulence models with the experimental study model by [21]. The validation of the experiment and simulation test for the outlet hot and cold temperatures and total heat transfer rate are shown in Figure 5, Figure 6 and Figure 7, respectively.

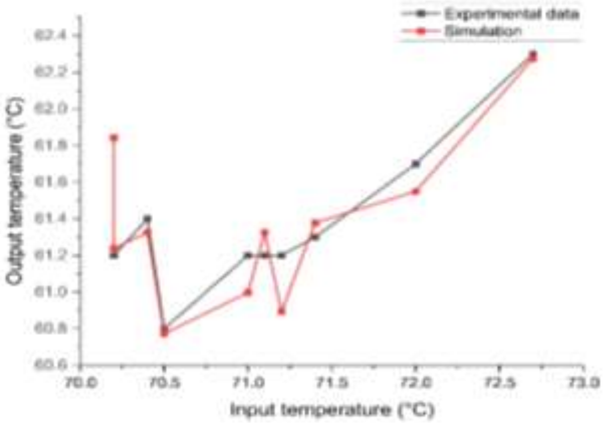


Figure 5: The graph validation of the output temperature hot fluid between experimental data and simulation.

Figure 5 shows the graph of the output hot fluid temperature of k-ω standard turbulence models and experimental test models with different input temperatures along the straight double-pipe heat exchanger. The result of output hot fluid temperature can be determined by computing the value of temperature from the solution setup in ANSYS software by using surface integral. Based on the graph above, the trend shows that the output hot fluid temperature from the simulation is almost like the experiment test result. It also shows that the input temperature is directly proportional to the output hot fluid temperature. As the input temperature increases, the output hot fluid temperature also increases.

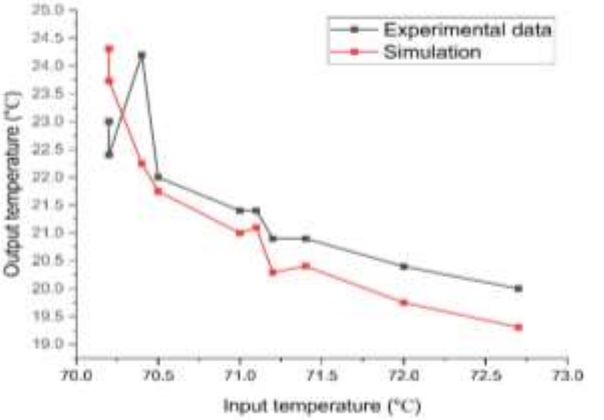


Figure 6: The graph validation of the output temperature cold fluid between experimental data and simulation.

Figure 6 shows the graph of the output cold fluid temperature of k-ω standard turbulence models and experimental test models with different input temperatures along the straight double-pipe heat exchanger. The result of output cold fluid temperature can be determined by computing the value of temperature from the solution setup in ANSYS software by using surface integral. Based on the graph above, the trend shows that the output hot fluid temperature is inversely proportional to the input temperature. Hence, the higher the input temperature, the lower the output temperature.

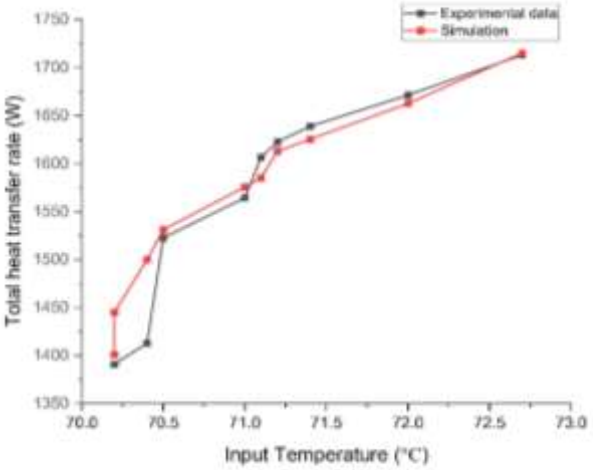


Figure 7: The graph validation of the total heat transfer rate between experimental data and simulation.

Figure 7 shows the graph of the total heat transfer rate against input temperature along the straight double-pipe heat exchanger. The result of total heat transfer rate can be determined by computing the value of temperature from the solution setup in ANSYS software by using surface integral. Based on the graph above, the trend shows that the total heat transfer rate is directly proportional to the input temperature. Hence, the higher the input temperature value, the higher the value of total heat transfer rate.

Table 2: The percentage error for hot fluid outlet temperature.

No.	Hot outlet temperature from Experiment test (℃)	Hot outlet temperature from simulation (℃)	Error percentage (%)
1	61.20	61.92	1.18
2	61.20	61.57	0.60
3	61.40	61.62	0.36
4	60.80	61.30	0.82
5	61.20	61.43	0.38
6	61.20	61.62	0.69
7	61.20	61.37	0.28
8	61.30	61.65	0.57
9	61.70	61.75	0.08
10	62.30	62.17	0.21
Average	61.35	61.64	0.52

Table 3: The percentage error for cold fluid outlet temperature.

No.	Cold outlet temperature from experiment test (℃)	Cold outlet temperature from simulation (℃)	Error percentage (%)
1	23.00	21.39	7.00
2	22.40	21.09	5.85
3	24.20	20.32	16.03
4	22.00	20.06	8.81
5	21.40	19.67	8.08
6	21.40	19.72	7.85
7	20.90	19.30	7.66
8	20.90	19.36	7.37
9	20.40	19.02	6.76
10	20.00	18.79	6.05
Average	21.66	19.87	8.15

Table 4: The percentage error for total heat transfer rate.

No.	Total heat transfer rate from experimental data (W)	Total heat transfer rate from simulation (W)	Error percentage (%)
1	1390.88	1266.35	8.95
2	1390.91	1302.29	6.37
3	1412.84	1346.78	4.68
4	1522.76	1372.13	9.89
5	1564.37	1408.56	9.96
6	1606.51	1416.17	11.85
7	1622.77	1438.58	11.35
8	1639.00	1448.89	11.60
9	1671.50	1479.00	11.52
10	1713.18	1521.80	11.17
Average	1553.47	1400.86	9.73

Figure 8: The contour of the temperature from the simulation.

Figure 8 shows the results of the contour temperature distribution along the double pipe heat exchanger. Based on the result obtained from the ANSYS fluent simulation, it shows that the temperature of the inner pipe is higher than the outer pipe. It is because the hot fluid flows in the inner pipe. The maximum temperature of the hot fluid is 343 K. The low temperature is located at the outer pipe. The cold fluid flows along the outer pipe. The heat transfer occurs between the boundary of the inner and outer pipe. The yellow color of the contour shows the temperature is in the medium level since the heat exchange in the boundary between the two pipes.

Figure 9: The contour of the velocity from the simulation.

Figure 9 shows the results of the contour velocity magnitude along the double straight pipe. From the result, it shows that the inner pipe experiences a high velocity compared to the outer pipe. It is because the viscosity of the hot fluid decreases. The low viscosity will contribute to less flow resistance that allows the hot fluid flow in high speed compared to cold fluid. The maximum velocity of the fluid flow is 0.717m/s.

Figure 10: The contour of the pressure from the simulation.

Figure 10 shows the results contour of pressure along the double straight pipe. From the result, it shows that the inner pipe experiences a high pressure compared to the outer pipe. It is because the hot fluid has low density due to thermal expansion. The maximum pressure of the fluid flow is 484 pascals.

5. Conclusion

In conclusion, the study focused on investigating the temperature, velocity, and pressure distributions within the heat exchanger under varying inlet conditions for the hot and cold fluids. The CFD simulations were conducted using a well-designed computational model, with the geometry created using CAD software and the computational grid refined near the walls to accurately capture the boundary layer effects. The k-ω standard turbulence model was employed to simulate the flow and heat transfer processes.

The simulation results were extensively validated against experimental data from previous studies, demonstrating the reliability and accuracy of the CFD model. The average error percentages were 0.52% for the hot fluid outlet temperature, 8.15% for the cold fluid outlet temperature, and 9.73% for the total heat transfer rate, with the hot fluid outlet temperature showing particularly strong agreement with the experimental data. The maximum and minimum heat transfer rate obtained from the simulation is 1521.80 W and 1390.88 W respectively. The maximum hot fluid and cold fluid are 62.17°C and 18.79°C respectively. Overall, the detailed information provided in the attached documents, including the boundary conditions, computational grid, and evidence of convergence and accuracy, indicates a well-designed and executed CFD study that offers valuable insights into the performance of the double-tube counter-flow heat exchanger.

Therefore, here are some recommendations that can be made to increase the performance of the double pipe counter-flow heat exchanger. The dimensions of the pipe length must be longer, and the diameter pipe must be larger so the total surface area will increase. The increase of surface area will help to increase the overall heat transfer rate within the heat exchanger. Last and not least, to perform the simulation process, make sure used a high specification device in order to run the simulation or otherwise, it will be disrupted.

Acknowledgement

The authors would like to express their gratitude to the Department of Mechanical Engineering, Polytechnic Kuching for the financial support through study leave scheme and the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia for supporting the data and technical advice to complete this research.

References

[1] K. Bartecki and A. M. Aliyu, “An Approximate Transfer Function Model for a Double-Pipe Counter-Flow Heat Exchanger,” *Energies 2021, Vol. 14, Page 4174*, vol. 14, no. 14, p. 4174, Jul. 2021, doi: 10.3390/EN14144174.

[2] R. Laskowski, “The black box model of a double-tube counter-flow heat exchanger,” *Heat and Mass Transfer/Waerme- und Stoffuebertragung*, vol. 51, no. 8, pp. 1111–1119, Aug. 2015, doi: 10.1007/S00231-014-1482-2/TABLES/3.

[3] N. Padoin, T. Matiazzo, H. G. Riella, and C. Soares, “A perspective on the past, the present, and the future of computational fluid dynamics (CFD) in flow chemistry,” *Journal of Flow Chemistry 2024 14:1*, vol. 14, no. 1, pp. 239–256, Feb. 2024, doi: 10.1007/S41981-024-00313-4.

[4] R. Cabello, A. E. Plesu-Popescu, J. Bonet-Ruiz, D. C. Cantarell, and J. Llorens, “Validation of CFD Models for Double-Pipe Heat Exchangers with Empirical Correlations,” *Chem Eng Trans*, vol. 88, pp. 1243–1248, 2021, doi: 10.3303/CET2188207.

[5] R. J. Brogan, “HEAT EXCHANGERS,” *A-to-Z Guide to Thermodynamics, Heat and Mass Transfer, and Fluids Engineering*, 2011, doi: 10.1615/ATOZ.H.HEAT_EXCHANGERS.

[6] “TEMA designations for shell-and-tube heat exchangers

[7] “Double pipe Heat exchanger Construction, Working, advantages and disadvantage | by BANTI SINGH | Medium.” Accessed: Jun. 26, 2024.

[8] P. S. A. Banu, D. N. S. R. Lohith, M. P. Kalyan, V. D. Sai, and B. Hemanth Sai, “Simulation of Double Pipe Heat Exchanger and Validation with CFD Analysis,” *AIP Conf Proc*, vol. 2648, no. 1, Nov. 2022, doi: 10.1063/5.0116411/2831245.

[9] B. I. Master, K. S. Chunangad, A. J. Boxma, D. Kral, and P. Stehlík, “Most frequently used heat exchangers from pioneering research to worldwide applications,” *Heat Transfer Engineering*, vol. 27, no. 6, pp. 4–11, Jun. 2006, doi: 10.1080/01457630600671960.

[10] K. Bartecki, “An approximate transfer function model for a double-pipe counter-flow heat exchanger,” *Energies (Basel)*, vol. 14, no. 14, Jul. 2021, doi: 10.3390/en14144174.

[11] “Double Pipe Heat Exchanger Experiment | PDF.” Accessed: Jun. 26, 2024. [Online]. Available: https://www.slideshare.net/slideshow/double-pipe-heat-exchanger-experiment/262716833%236

[12] M. K. Muhamad Azim, A. Arifutzzaman, R. Saidur, M. U. Khandaker, and D. A. Bradley, “Recent progress in emerging hybrid nanomaterials towards the energy storage and heat transfer applications: A review,” *J Mol Liq*, vol. 360, Aug. 2022, doi: 10.1016/j.molliq.2022.119443.

[13] “Review On Heat Exchanger Tube-To- Tubesheet Joining.” Accessed: Jun. 26, 2024. [Online]. Available: https://www.researchgate.net/publication/347967455_Review_On_Heat_Exchanger_Tube-To-_Tubesheet_Joining

[14] C. D. Widiawaty, A. I. Siswantara, Budiarso, G. G. R. Gunadi, H. Pujowidodo, and M. H. G. Syafei, “A CFD simulation and experimental study: Predicting heat transfer performance using SST k-? turbulence model,” *IOP Conf Ser Mater Sci Eng*, vol. 909, no. 1, Dec. 2020, doi: 10.1088/1757-899X/909/1/012004.

[15] S. Gilani, H. Montazeri, and B. Blocken, “CFD simulation of stratified indoor environment in displacement ventilation: Validation and sensitivity analysis,” *Build Environ*, vol. 95, pp. 299–313, Jan. 2016, doi: 10.1016/J.BUILDENV.2015.09.010.

[16] “Cfd simulations and validation through test data of a double pipe counter flow heat exchanger.” Accessed: Jun. 26, 2024. [Online]. Available: https://www.researchgate.net/publication/317805887_Cfd_simulations_and_validation_through_test_data_of_a_double_pipe_counter_flow_heat_exchanger

[17] A. Heydari *et al.*, “An investigation of multi-parameters effects on the performance of liquid-to-liquid heat exchangers in rack level cooling,” *InterSociety Conference on Thermal and Thermomechanical Phenomena in Electronic Systems, ITHERM*, vol. 2023-May, 2023, doi: 10.1109/ITHERM55368.2023.10177525.

[18] M. M. Aslam Bhutta, N. Hayat, M. H. Bashir, A. R. Khan, K. N. Ahmad, and S. Khan, “CFD applications in various heat exchangers design: A review,” *Appl Therm Eng*, vol. 32, no. 1, pp. 1–12, 2012, doi: 10.1016/J.APPLTHERMALENG.2011.09.001.

[19] M. S. Kim, “Environmental Thermal Engineering Lecture Note #4”.

[20] “Working Principles of Heat Exchanger.” Accessed: Jun. 26, 2024. [Online]. Available: https://unacademy.com/content/gate/study-material/mechanical-engineering/heat-exchanger/

[21] C. Patrascioiu and S. Rădulescu, “Modeling and Simulation of the Double Tube Heat Exchangers . Case Studies,” 2012.

[22] “What is the second law of thermodynamics? | Live Science.” Accessed: Jun. 26, 2024. [Online]. Available: https://www.livescience.com/50941-second-law-thermodynamics.html

[23] R. N. Pinto, A. Afzal, L. V. D’Souza, Z. Ansari, and A. D. Mohammed Samee, “Computational Fluid Dynamics in Turbomachinery: A Review of State of the Art,” *Archives of Computational Methods in Engineering*, vol. 24, no. 3, pp. 467–479, Jul. 2017, doi: 10.1007/S11831-016-9175-2.

[24] P. Dilleswara Rao, B. Nageswara Rao Professor, and B. Nageswara Rao, “Article ID: IJMET_08_05_089 Cite this Article: Peddi Dilleswara Rao and B. Nageswara Rao. CFD Simulations and Validation Through Test Data of a Double Pipe Counter Flow Heat Exchanger,” *International Journal of Mechanical Engineering and Technology*, vol. 8, no. 5, pp. 818–831, 2017, [Online]. Available: http://iaeme.com/Home/journal/IJMET818editor@iaeme.comhttp://iaeme.com/Home/issue/IJMET?Volume=8&Issue=5http://iaeme.com/Home/issue/IJMET?Volume=8&Issue=5http://iaeme.com/Home/journal/IJMET819

[25] D. M. Ingram, D. M. Causon, and C. G. Mingham, “Developments in Cartesian cut cell methods,” *Math Comput Simul*, vol. vol. 61, no. 3–6, pp. 561–572, Jan. 2003, doi: 10.1016/S0378-4754(02)00107-6