Semarak Journal of Thermal-Fluid Engineering

Journal homepage:
https://semarakilmu.my/index.php/sjotfe/index
ISSN: 3030-6639

Comparative Study of Reynolds Number Effects on Shell and Tube Heat Exchanger with Single Segmental Baffles Using CFD

Alicia Lai Jia Qi[1], Ishkrizat Taib[1,*]

[1] Department of Mechanical Engineering, Faculty of Mechanical and Manufacturing Engineering, Universiti of Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor, Malaysia

ARTICLE INFO

Article history:
Received 26 April 2026
Received in revised form 5 May 2026
Accepted 22 May 2026
Available online 30 June 2026

ABSTRACT

Shell and tube heat exchangers are among the most widely used heat transfer equipment in industrial applications due to their simple design, mechanical reliability, ease of maintenance, and ability to operate under high-temperature and high-pressure conditions. The incorporation of segmental baffles within the shell side enhances fluid mixing, promotes cross-flow over the tube bundle, and improves heat transfer performance. However, the increased turbulence generated by baffles may also result in higher pressure losses, which can increase pumping power requirements and operating costs. Therefore, understanding the influence of Reynolds number on both thermal and hydraulic performance is important for optimizing shell and tube heat exchanger operation. In this study, Computational Fluid Dynamics (CFD) simulations were performed to investigate the effect of Reynolds number on the performance of a shell and tube heat exchanger equipped with segmental baffles. The heat exchanger geometry consisted of a cylindrical shell with three straight tubes and segmental baffles positioned along the shell side. The Standard k-ε turbulence model was employed for all simulation cases due to its suitability for turbulent internal flow analysis. The simulation results showed that increasing Reynolds number significantly affected the hydraulic performance of the heat exchanger. The pressure drop increased from 60.44 Pa at Reynolds number 5,041 to 301.29 Pa at Reynolds number 16,803, indicating a substantial increase in flow resistance as the inlet velocity increased. In contrast, the outlet temperature exhibited only a small variation, with the hot fluid outlet temperature approaching the inlet temperature at higher Reynolds numbers. Temperature and pressure distribution analyses revealed that stronger flow mixing occurred at higher Reynolds numbers due to enhanced turbulence generated by the segmental baffles. The pressure distribution along the tube length also increased more rapidly for higher Reynolds number cases. Overall, the study demonstrates that Reynolds number has a greater influence on pressure drop than on outlet temperature for the investigated heat exchanger configuration. The findings contribute to a better understanding of the thermal-hydraulic behaviour of shell and tube heat exchangers and may assist engineers in selecting appropriate operating conditions for improved energy efficiency and system performance.

Keywords:
Shell and tube heat exchanger; Reynolds number; pressure drop; temperature distribution; segmental baffle; CFD

* *Corresponding author.*
E-mail address: iszat@uthm.edu.my

https://doi.org/10.37934/sjotfe.9.1.3648a

1. Introduction

Heat exchangers are thermal devices used to transfer heat between fluids at different temperatures and are widely applied in power generation, petrochemical processing, HVAC, refrigeration, energy recovery, and process industries [1,2]. Among the different types of heat exchangers, shell and tube heat exchangers are commonly used because they have a simple structure, strong mechanical durability, and the ability to operate under demanding temperature and pressure conditions [3]. A typical shell and tube heat exchanger consists of a tube bundle enclosed inside a cylindrical shell, where one fluid flows through the tubes and another fluid flows across the shell side [1,3]. The thermal and hydraulic performance of this system depends on several operating and design parameters, including inlet velocity, temperature difference, tube geometry, baffle arrangement, baffle spacing, allowable pressure drop, and fluid properties [4]. Baffles are important components because they guide shell-side flow across the tube bundle, improve fluid mixing, increase turbulence, and enhance heat transfer between the hot and cold streams [5,6]. However, the use of segmental baffles may also increase pressure drop, produce recirculation regions, create dead zones, increase fouling tendency, and contribute to flow-induced vibration [5,6]. Therefore, the main problem in shell and tube heat exchanger design is to improve heat transfer performance while maintaining an acceptable pressure drop and avoiding excessive pumping power requirement [1,6].

Previous studies have shown that Reynolds number, inlet velocity, baffle arrangement, baffle cut, and turbulence behaviour strongly affect heat transfer and pressure loss in heat exchanger systems [7,8]. Reynolds number is an important non-dimensional parameter because it describes the relationship between inertial force and viscous force in internal flow, and it is commonly used to classify laminar, transition, and turbulent flow regimes [9,10]. In shell and tube heat exchangers, increasing Reynolds number can improve fluid mixing and convective heat transfer, but it can also increase pressure drop due to higher wall friction and stronger turbulence [7]. The increasing Reynolds number in a shell-and-tube heat exchanger with segmental baffles has been reported to increase both Nusselt number and pressure drop, showing that heat transfer improvement is usually accompanied by higher hydraulic loss [7]. In addition, Reynolds number, baffle distance, and baffle angle have been shown to affect turbulent flow and heat transfer behaviour in baffled internal flow [8]. Based on these findings, this study compares three inlet velocities of 0.1 m/s, 0.2 m/s, and 0.3 m/s to investigate the effect of Reynolds number on outlet temperature, pressure drop, and flow distribution in a shell and tube heat exchanger with segmental baffles. The Standard k-ε turbulence model is selected and kept constant because it is commonly used for turbulent internal flow and baffled heat exchanger analysis [3,9].

The study is expected to show how increasing inlet velocity influences the thermal and hydraulic behaviour of the heat exchanger. Higher inlet velocity increases Reynolds number and flow intensity, which can improve mixing between fluid layers and enhance convective heat transfer [2,7]. Stronger mixing is useful because it reduces thermal resistance near the wall and allows better heat exchange between the hot fluid inside the tube and the cold fluid on the shell side [2]. However, higher velocity also increases frictional resistance, turbulence intensity, and flow disturbance around the baffles, which leads to higher pressure drop [3,10]. Pressure drop is an important design parameter because it represents hydraulic loss and is related to the pumping power required to maintain flow through the heat exchanger [3]. Therefore, outlet temperature and pressure drop are selected as the main output parameters in this study. Temperature contours are used to observe heat distribution, pressure contours are used to identify pressure variation and hydraulic loss, and velocity streamlines are used to analyse flow movement, recirculation, and baffle effects. A Grid Independence Test is also applied to ensure that the numerical results are not significantly affected by mesh size and that the selected mesh provides reliable results with reasonable computational cost [11,12].

This study is related to heat exchanger performance analysis because velocity and pressure determine the balance between thermal improvement and hydraulic resistance. A higher flow velocity may improve heat transfer by increasing Reynolds number and fluid mixing, but it may also increase pressure drop and energy consumption [1,6]. Therefore, heat exchanger performance should not be evaluated only based on outlet temperature, but also by considering the pressure loss produced by stronger flow and baffle-induced turbulence. In this study, Computational Fluid Dynamics is used to predict the temperature, pressure, and velocity fields inside the shell and tube heat exchanger. This approach allows detailed observation of internal flow behaviour that may be difficult to obtain experimentally. Therefore, the objectives of this study are to calculate the Reynolds number for each inlet velocity, compare the effect of inlet velocity on outlet temperature and pressure drop, observe temperature, pressure, and streamline distributions, and verify numerical reliability using Grid Independence Test. The findings are expected to provide a clearer understanding of how inlet velocity affects the thermal and hydraulic performance of a shell and tube heat exchanger with segmental baffles.

2. Methodology

This study was carried out using Computational Fluid Dynamics (CFD) in ANSYS Fluent to evaluate the thermal and hydraulic performance of a shell and tube heat exchanger with segmental baffles. CFD was selected because it allows detailed prediction of velocity, pressure, temperature distribution, and streamline behaviour inside complex internal flow geometries [13]. Similar CFD-based methods have been used in previous shell and tube heat exchanger studies to analyse pressure distribution, temperature drop, velocity streamlines, and turbulence model effects [2,6,14]. The methodology consists of geometry preparation, boundary condition setup, mesh generation, Grid Independence Test, Reynolds number calculation, and result evaluation.

2.1 Geometry of Shell and Tube Heat Exchanger

The geometry used in this study is a shell and tube heat exchanger consisting of three straight tubes enclosed inside a cylindrical shell. Segmental baffles are placed inside the shell to guide shell-side flow across the tube bundle and improve fluid mixing. Baffles are important in shell and tube heat exchangers because they influence flow distribution, heat transfer performance, and pressure drop. Previous research numerically investigated shell-and-tube heat exchangers with inclined folded helical baffles and showed that baffle design plays an important role in optimizing heat transfer and hydraulic performance [15]. Another present study also applies CFD analysis to evaluate the thermal and hydraulic behaviour of a shell and tube heat exchanger with segmental baffles [16]. The geometry of the shell and tube heat exchanger with segmental baffles is shown in Figure 1 created by using ANSYS Design Modeler.

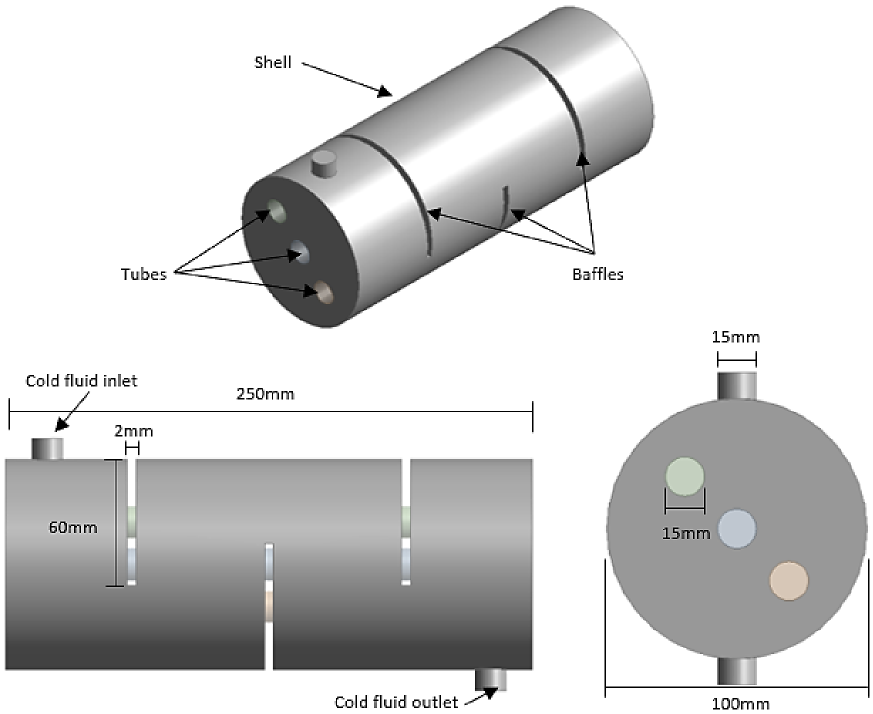


Fig. 1. Geometry of shell and tube heat exchanger with segmental baffles

2.2 Meshing Model

After the geometry was prepared, the model was meshed in ANSYS Meshing before being transferred to ANSYS Fluent for simulation. Mesh generation is important because the computational domain must be divided into smaller elements to solve the governing equations. The meshing after 3D modelling stated that proper mesh quality is required to obtain reliable simulation results [3]. In this study, refinement was applied near the tube walls, baffle regions, inlet, outlet, and contact areas because these regions are expected to experience higher velocity, pressure, and temperature gradients. The generated mesh of the shell and tube heat exchanger is shown in Figure 2. The mesh type of the tubes and shell is used tetrahedrons. The Inflation layers were applied near the wall regions to improve the resolution of the boundary layer around the tube and baffle surfaces.

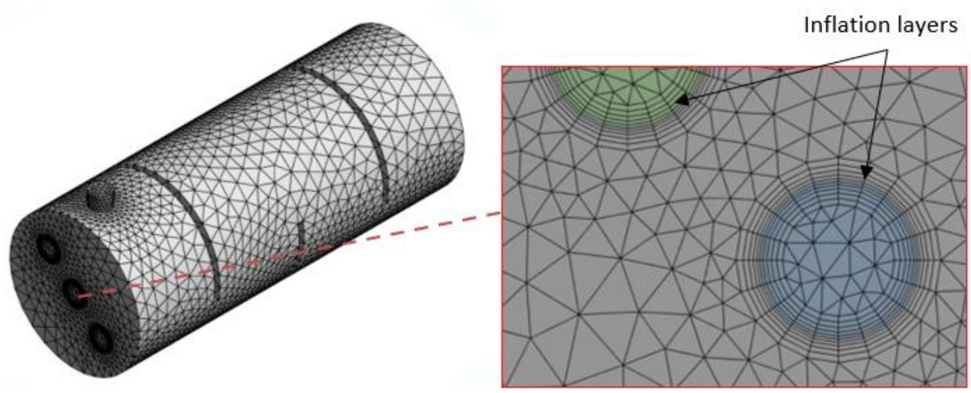


Fig. 2. Computational mesh of shell and tube heat exchanger

2.3 Boundary Conditions

The boundary conditions were applied to define the flow and thermal behaviour of the heat exchanger as shown in Figure 3. The boundary condition located at hot inlet, hot outlet, cold inlet, cold outlet, shell wall and tube wall. Water (H_2O) is selected as the working fluid for both hot and cold streams because it is commonly used in heat exchanger analysis and has suitable thermal properties for heat transfer evaluation [4,17]. The hot inlet temperature was set to 350 K, while the cold inlet temperature was set to 300 K. Similar boundary condition approaches have been used in shell and tube heat exchanger simulations, where inlet temperature, flow rate or velocity, and outlet pressure are specified to evaluate thermal and hydraulic performance [3,18]. Three hot inlet velocities of 0.3 m/s, 0.6 m/s, and 1.0 m/s were applied to study the effect of Reynolds number on outlet temperature and pressure drop. Inlet velocity was selected as the main operating variable because previous studies showed that fluid velocity and Reynolds number influence heat transfer rate and pressure drop in heat exchanger systems [2,7]. The outlet boundary condition was defined as pressure outlet with 0 Pa gauge pressure. Pressure drop was calculated from the difference between inlet pressure and outlet pressure. The Standard k-ε turbulence model was selected because it is widely used for turbulent internal flow analysis and has been applied in shell and tube heat exchanger simulations [3,9].

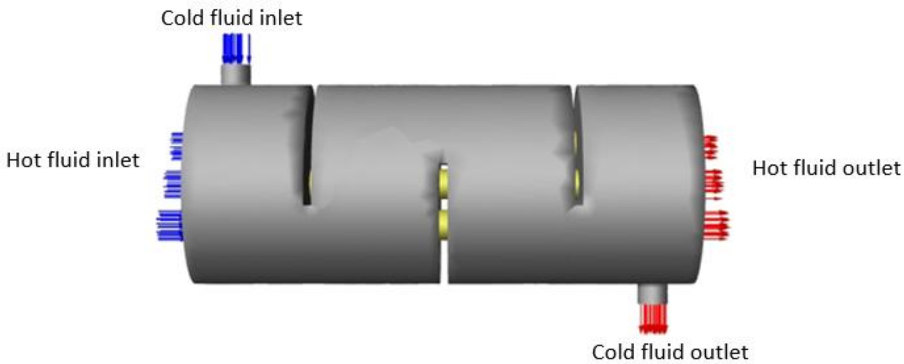


Fig. 3. Boundary conditions setup for the shell and tube heat exchanger

2.4 Grid Independence Test

A Grid Independence Test was conducted to ensure that the results were not significantly affected by mesh size. Different mesh densities were generated and tested under the same boundary conditions. The outlet temperatures and pressure drop were selected as comparison parameters because they represent the thermal and hydraulic performance of the heat exchanger. The grid independence checking is important to reduce discretization error [11]. The final mesh was selected when further mesh refinement produced only negligible changes in the output parameters. This approach helps maintain result accuracy while reducing unnecessary computational cost [11,12]. In this study, the percentage error is calculated by using Eq. (1) to prove that the mesh is independent when the percentage error is less than 5%.

$$Percentage\ error = \left|\frac{Present\ value - Previous\ value}{Present\ value}\right| \times 100 \tag{1}$$

2.5 Governing Equations

The fluid flow and heat transfer in the heat exchanger were solved using the conservation equations of mass, momentum, and energy. The continuity, momentum, energy, and turbulence equations are listed in their mathematical modelling section for shell and tube heat exchanger analysis [3]. In this study, the governing equations were used to determine velocity, pressure, and temperature fields inside the heat exchanger.

Continuity equation written as Eq. (2):

$$\nabla \cdot \vec{V} = 0 \tag{2}$$

where $\vec{V}$ is the velocity vector.

Momentum equation written as Eq. (3):

$$\rho(\vec{V} \cdot \nabla)\vec{V} = -\nabla p + \mu \nabla^2 \vec{V} \tag{3}$$

where ρ is fluid density, p is pressure, and μ is dynamic viscosity.

Energy equation written as Eq. (4):

$$\rho C_p(\vec{V} \cdot \nabla T) = k \nabla^2 T \tag{4}$$

where C_p is specific heat capacity, T is temperature, and k is thermal conductivity.

2.6 Reynolds Number Calculation

The Reynolds number was calculated to classify the flow regime for each inlet velocity. Reynolds number is important because it relates velocity, tube diameter, density, and viscosity to flow behaviour. Present research studied the influence of Reynolds number on heat exchange performance and reported that Reynolds number affects heat transfer behaviour and Nusselt number, which supports the selection of inlet velocity as the main variable [19]. The Reynolds number equation is written as Eq. (5). The calculated Reynolds number values for the three inlet velocities are summarized in Table 1. All the cases have the turbulence flow regime.

$$Re = \frac{\rho V D}{\mu} \tag{5}$$

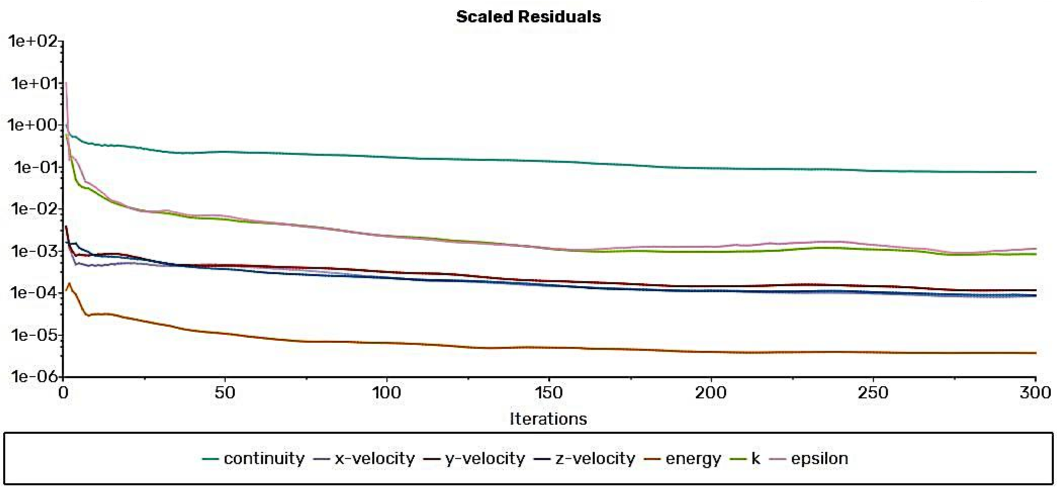
where Re is Reynolds number, ρ is fluid density, V is inlet velocity, D is tube diameter, and μ is dynamic viscosity. Assuming the hot fluid is water at room temperature, ρ is 998 kg/m³, μ = 0.001 Pa·s, D = 0.015 m.

Table 1
Reynolds number for different inlet velocities

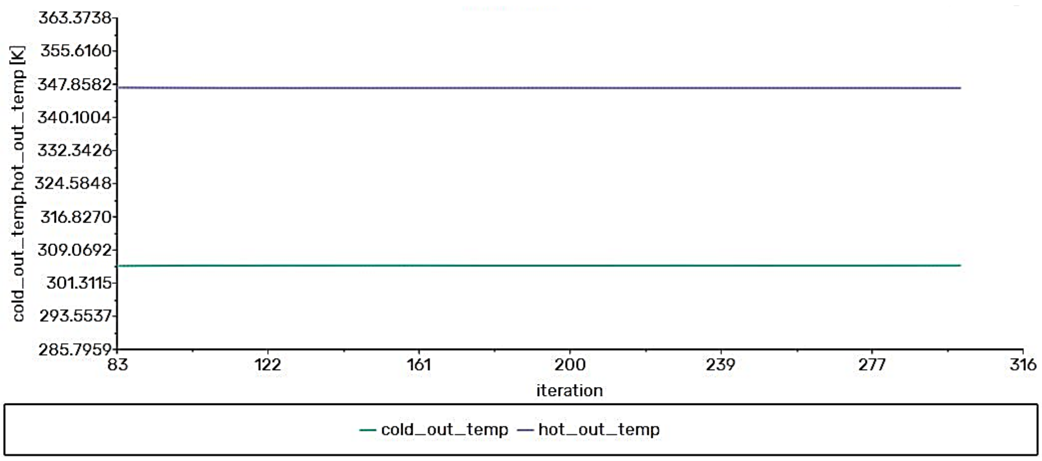
Inlet velocity (ms^{-1})	Reynolds number
0.3	5041
0.6	10082
1.0	16803

2.7 Solution Setup and Convergence

After applying the governing equations, boundary conditions, and turbulence models, the simulation was solved under steady-state conditions. The energy equation was activated because heat transfer between the hot and cold fluids was included [20]. Hybrid initialization was used before running the calculation and the iteration is set to 300. The convergence must be achieved to obtain a sound solution, and that temperature and pressure drop results were obtained after convergence [21]. In this study, convergence was evaluated and monitored output parameters, which is hot outlet temperature. The residual plot is shown as Figure 4(a). and the hot outlet temperature monitor plot as shown as Figure 4(b).



(a)



(b)

Fig. 4. Convergence monitoring plots (a) Residual (b) Outlet temperature

3. Results

3.1 Grid Independence Test

Three mesh densities were tested, namely coarse, medium, and fine mesh. The hot outlet temperature, cold outlet temperature, and pressure drop were selected as comparison parameters because they represent the main thermal and hydraulic performance of the shell and tube heat exchanger. The Grid Independence Test results are presented in Table 2.

Table 2
Grid independence test results

Mesh type	Number of nodes	Number of elements	Pressure drop (Pa)	Percentage Difference (%)
Coarse	122525	375677	301.31	-
Medium	128159	402680	301.29	0.006
Fine	156331	545717	301.33	0.013

Based on Table 2, the hot outlet temperature changed only slightly from 348.92 K for the coarse mesh to 348.93 K for the medium and fine meshes. The cold outlet temperature also showed a very small change, decreasing from 306.45 K to 306.43 K as the number of elements increased. In addition, the pressure drop decreased only slightly from 301.31 Pa for the coarse mesh to 301.29 Pa for the medium and fine meshes. All the differences are less than 5% and these small differences indicate that further mesh refinement did not significantly affect the simulation results.

Line 1 was created along the flow direction of the heat exchanger to obtain variable profiles from inlet to outlet. This line was used to observe the variation of temperature, pressure, and velocity along the flow path. Plane 1 was created as a sectional cutting plane through the heat exchanger to visualize internal contour distributions. This plane was used to display temperature and pressure contours inside the shell and tube heat exchanger. The location of line 1 and plane 1 are shown in Figure 5. The Grid Independence Test graph is shown in Figure 6. The graph shows that the results became almost constant when the number of elements increased from 375677 to 545717 and the Z is represented the length of the line 1 which is also the length of the tube. This indicates that the medium mesh already produced a mesh-independent solution. Therefore, the medium mesh was selected for the remaining simulations because it provides reliable results while reducing computational time compared with the fine mesh. This selection is also consistent with the purpose of Grid Independence Test, where the final mesh should give stable results without unnecessary computational cost.

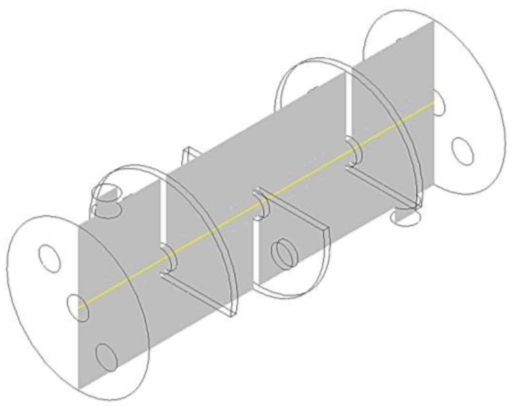


Fig. 5. Location of line 1 and plane 1

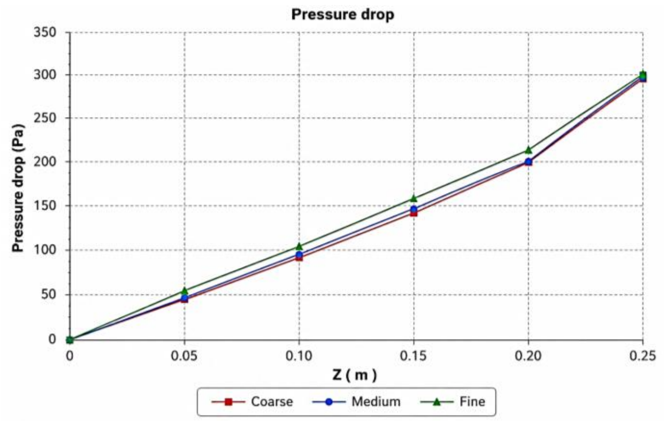


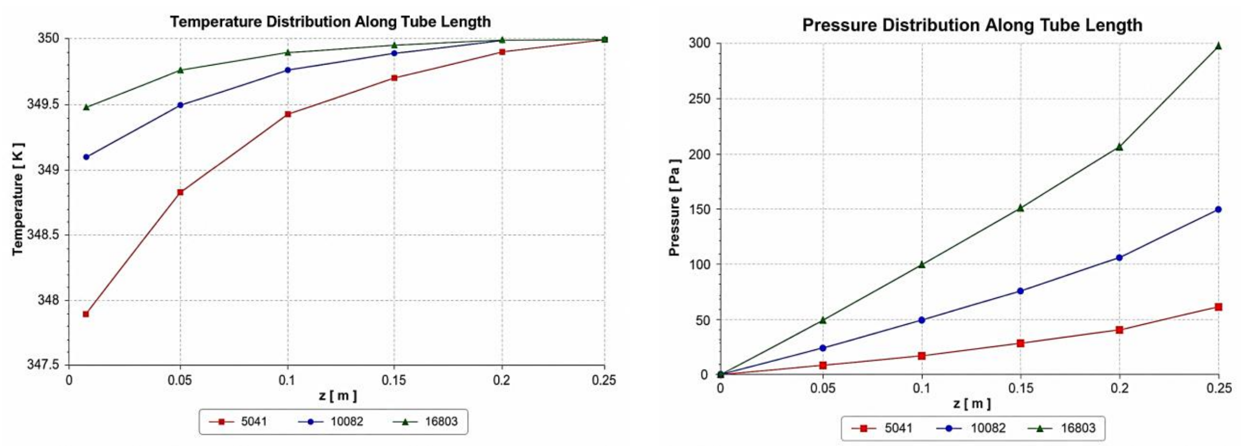
Fig. 6. Pressure drop of every mesh case for grid independence test

3.2 Reynolds Number Comparison

The Reynolds number comparison was conducted using three inlet velocities of 0.3 m/s, 0.6 m/s, and 1.0 m/s. The results are shown in Table 3. The pressure drop increased as the Reynolds number increased. This trend agrees with Singh and Sehgal [7], who reported that increasing Reynolds number increased pressure drop in a shell-and-tube heat exchanger with segmental baffles. The relationship between Reynolds number and temperature as well as Reynolds number and pressure drop are shown in Figure 7. As Reynolds number increases, both hot and cold outlet temperature also increase. The pressure drop increased from 60.44 Pa to 301.29 Pa when the velocity increased from 0.3 m/s to 1.0 m/s. This happens because higher velocity increases flow momentum, frictional effect, and turbulence intensity. Therefore, higher Reynolds number requires a greater pressure difference to move the fluid through the heat exchanger.

Table 3
Reynolds number comparison results

Inlet velocity (ms^{-1})	Reynolds number	Hot outlet temperature (K)	Cold outlet temperature (K)	Pressure drop (Pa)
0.3	5041	346.98	305.44	60.44
0.6	10082	348.40	305.77	153.11
1.0	16803	348.93	306.44	301.29



(a) (b)

Fig. 7. Relationship between Reynolds number (a)Temperature (b) Pressure drop

3.3 Temperature Distribution

The temperature contour comparison is shown in Figure 8. The contour was used to observe the temperature distribution inside the heat exchanger for the three inlet velocity cases. Temperature contour plots are commonly used in heat exchanger CFD studies to visualize thermal behaviour and outlet temperature response. Based on Table 3 the hot outlet temperature increased from 346.98 K at 0.3 m/s to 348.93 K at 1.0 m/s. The cold outlet temperature increased from 305.44 K to 306.44 K. The increase in cold outlet temperature indicates that heat transfer occurred between the hot and cold fluids. However, the short tube length of 250 mm and limited residence time of the fluid inside the heat exchanger.

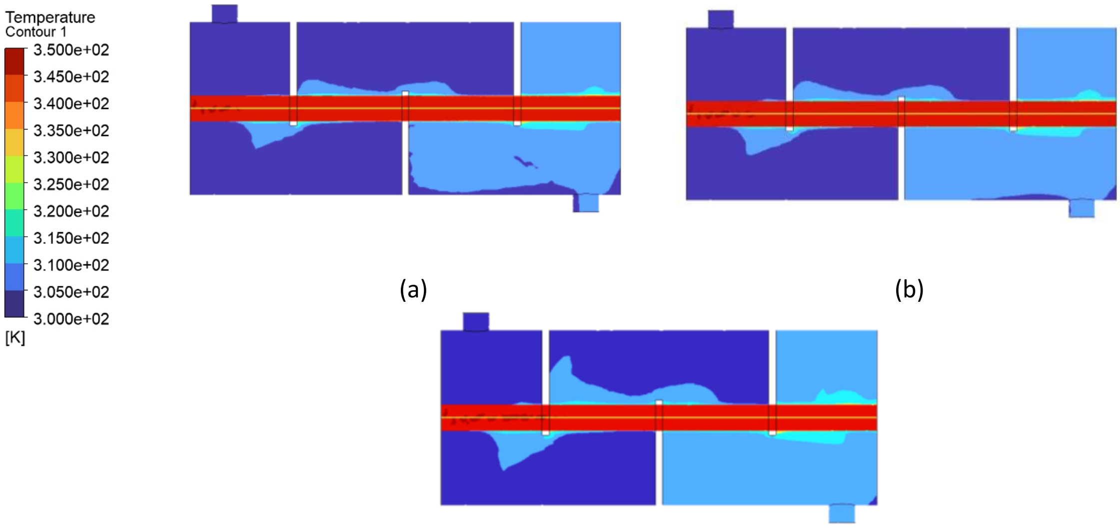


Fig. 8. Temperature contour comparison for inlet velocities of (a) 0.3 m/s (b) 0.6 m/s (c) 1.0 m/s

3.4 Pressure Distribution

The pressure contour comparison is shown in Figure 9. The contour was used to observe the pressure distribution and hydraulic resistance inside the heat exchanger. Pressure contour plots are useful for identifying hydraulic loss and pressure variation across the shell and tube heat exchanger. The pressure drop increased significantly as inlet velocity increased. The highest-pressure drop was obtained at 1.0 m/s, while the lowest pressure drop was obtained at 0.3 m/s. This shows that pressure drop is more sensitive to velocity compared with outlet temperature. Segmental baffles also contribute to pressure loss because they disturb the flow path and force the fluid to move across the tube bundle.

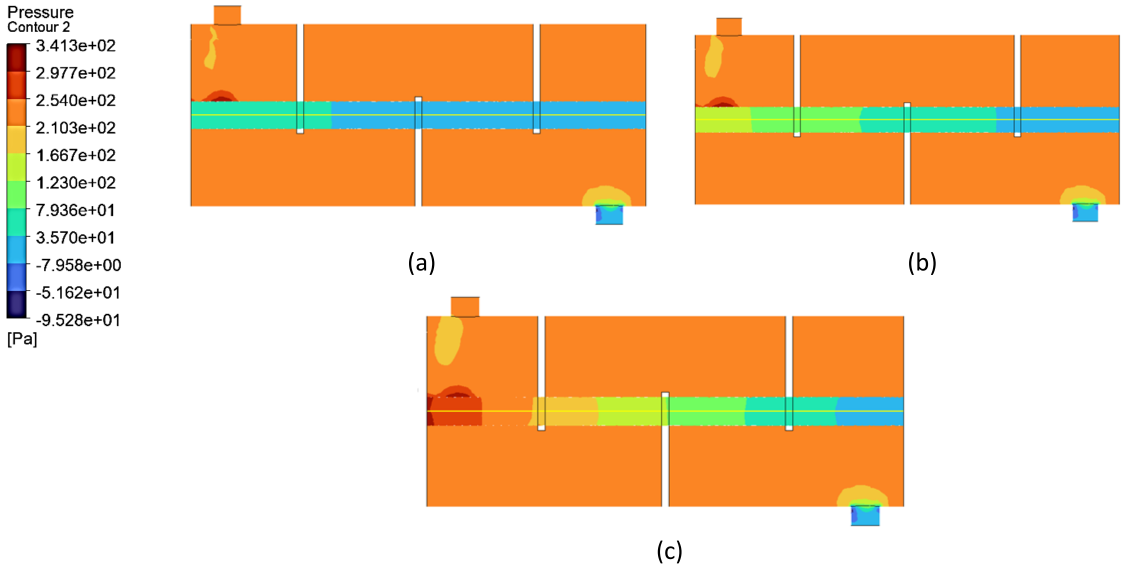


Fig. 9. Pressure contour comparison for inlet velocities of (a) 0.3 m/s (b) 0.6 m/s (c) 1.0 m/s

3.5 Velocity Streamline Behaviour

The velocity streamline comparison is shown in Figure 10. The streamline plot was used to observe the flow path and flow disturbance caused by the segmental baffles. Velocity streamlines are commonly used to observe flow movement, recirculation, and disturbance inside baffled heat exchangers. The streamline results show that the fluid flow was disturbed by the segmental baffles. This disturbance improves mixing but also increases flow resistance. The 1.0 m/s case showed stronger flow movement compared with the 0.3 m/s and 0.6 m/s cases. This explains why the 1.0 m/s case produced the highest-pressure drop.

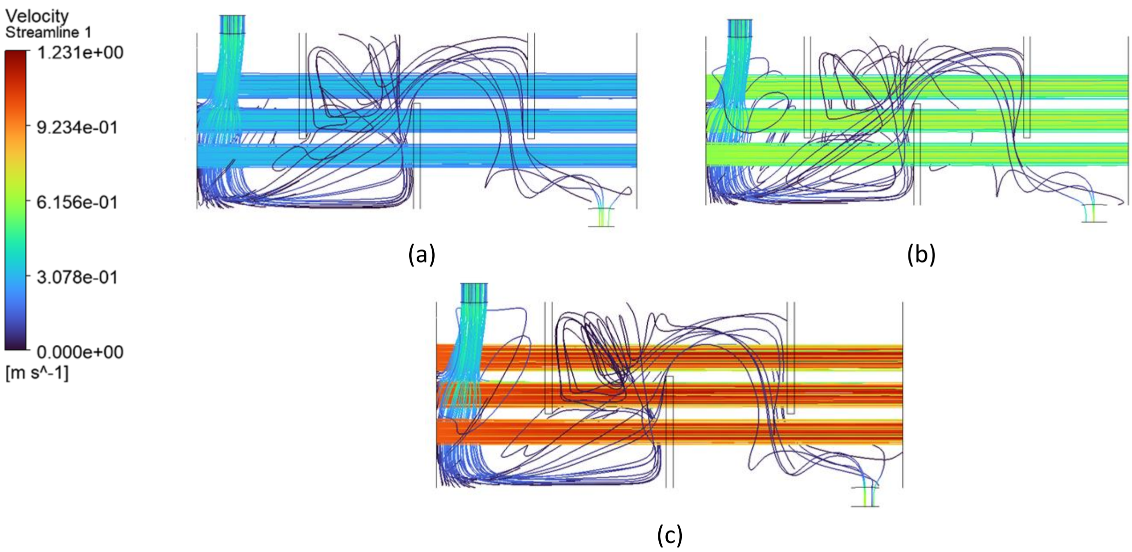


Fig. 10. Velocity streamline comparison for inlet velocities of (a) 0.3 m/s (b) 0.6 m/s (c) 1.0 m/s

3.6 Discussion

Overall, the results show that increasing Reynolds number increases pressure drop significantly, while outlet temperature changes only slightly. The 0.3 m/s case produced the lowest pressure drop but also the lowest cold outlet temperature. The 1.0 m/s case produced the highest cold outlet temperature but also the highest-pressure drop. Therefore, the 0.6 m/s case can be considered the most balanced operating condition because it provides moderate thermal improvement with lower pressure drop than the 1.0 m/s case. This supports the need to evaluate both thermal performance and hydraulic resistance when designing shell and tube heat exchangers with segmental baffles.

4. Conclusions

This study investigated the effect of Reynolds number on the performance of a shell and tube heat exchanger with segmental baffles. The Grid Independence Test confirmed that the medium mesh with 402,683 elements produced results close to the fine mesh. Therefore, the medium mesh was selected for the final analysis. Overall, 0.6 m/s is considered the most balanced operating condition because it provides moderate heat transfer improvement with lower pressure drop than 1.0 m/s.

Three inlet velocities of 0.3 m/s, 0.6 m/s, and 1.0 m/s were compared, corresponding to Reynolds numbers of 5,041, 10,082, and 16,803. All cases were in the turbulent flow region, which satisfies the objective of evaluating turbulent flow behaviour. The results show that increasing inlet velocity slightly improved the thermal response, where the cold outlet temperature increased from 305.44 K to 306.44 K. However, pressure drop increased significantly from 60.44 Pa to 301.29 Pa. This shows that pressure drop is more sensitive to Reynolds number than outlet temperature.

The temperature contours confirmed heat transfer between the hot and cold fluids, while the pressure contours showed greater hydraulic loss at higher velocity. The streamline results showed that the segmental baffles disturbed the flow and improved mixing but also increased flow resistance. Therefore, this study successfully achieved its objectives by calculating the Reynolds number for each inlet velocity, comparing outlet temperature and pressure drop, observing temperature, pressure, and streamline distributions, and verifying the numerical reliability through Grid Independence Test.

References

[1] Adebayo, Jacob Kolawole, Ademola Adebukola Dare, and Moses Omolayo Petinrin. "Performance characteristics of shell and tube heat exchanger using sectoral baffle." *Journal of Engineering and Applied Science* 72, no. 1 (2025): 13. https://doi.org/10.1186/s44147-024-00555-y

[2] Khan, Abdullah, Imran Shah, Waheed Gul, Tariq Amin Khan, Yasir Ali, and Syed Athar Masood. "Numerical and experimental analysis of shell and tube heat exchanger with round and hexagonal tubes." *Energies* 16, no. 2 (2023): 880. https://doi.org/10.3390/en16020880

[3] Saffarian, Mohammad Reza, Farivar Fazelpour, and Mehrzad Sham. "Numerical study of shell and tube heat exchanger with different cross-section tubes and combined tubes." *International Journal of Energy and Environmental Engineering* 10, no. 1 (2019): 33-46. https://doi.org/10.1007/s40095-019-0297-9

[4] Zamri, Muhammad Haziq Mohd, and Nurhafizzah Hassan. "Comparison investigation of thermal performance of shell and tube heat exchanger with helical baffles." *Research Progress in Mechanical and Manufacturing Engineering* 6, no. 1 (2025): 393-402.

[5] Ahire, Dinesh. "CFD analysis of Shell & Tube Type Heat Exchanger with Baffles." (2017).

[6] Waleed, Muhammad, Syed Murawat Abbas Naqvi, Hammad Mustafa, Mohammed K. Al Mesfer, Mohd Danish, Kashif Irshad, and Hasan Shahzad. "Numerical analysis of shell and tube heat exchanger with combination of different baffles." *Case Studies in Thermal Engineering* 65 (2025): 105658. https://doi.org/10.1016/j.csite.2024.105658

[7] Singh, Amarjit, and Satbir S. Sehgal. "Thermohydraulic analysis of shell-and-tube heat exchanger with segmental baffles." *International Scholarly Research Notices* 2013, no. 1 (2013): 548676. https://doi.org/10.1155/2013/548676

[8] Turgut, Oguz, and Erkan Kizilirmak. "Effects of reynolds number, baffle angle, and baffle distance on 3-d turbulent flow and heat transfer in a circular pipe." *Thermal Science* 19, no. 5 (2015): 1633. https://doi.org/10.2298/TSCI121011045T

[9] Launder, Brian Edward, and Dudley Brian Spalding. "The numerical computation of turbulent flows." In *Numerical Prediction of Flow, Heat Transfer, Turbulence and Combustion*, pp. 96-116. Pergamon, 1983. https://doi.org/10.1016/B978-0-08-030937-8.50016-7
[10] Cieśliński, Janusz T., Jacek Barański, Paweł Dąbrowski, Maciej Fabrykiewicz, Kamil Stasiak, and Krzysztof Tesch. "Heat transfer and pressure drop in a shell-and-tube heat exchanger with segmental baffles." *Energies* 19, no. 7 (2026): 1760. https://doi.org/10.3390/en19071760
[11] Versteeg, Henk Kaarle. *An introduction to computational fluid dynamics the finite volume method, 2/E*. Pearson Education India, 2007.
[12] Patankar, Suhas. *Numerical heat transfer and fluid flow*. CRC press, 2018. https://doi.org/10.1201/9781482234213
[13] Singh, Amarjit, and Satbir S. Sehgal. "Thermohydraulic analysis of shell-and-tube heat exchanger with segmental baffles." *International Scholarly Research Notices* 2013, no. 1 (2013): 548676. https://doi.org/10.1155/2013/548676
[14] Abdelhamid, Heba Roushdy Mohamed, Ayman Ibrahim Bakry, and Hagar Alm-Eldin Bastawissi. "CFD comparative study of design and performance of shell-and-tube heat exchangers with different configurations." *Journal of Engineering Research* 6, no. 4 (2026). https://doi.org/10.21608/jer.2026.502597
[15] Cao, Kan, Jiaxing Liang, Huihui Xu, Wannian Ma, Shaofei Huang, Lei Li, and Zezi Tan. "Numerical investigation and optimization of shell-and-tube heat exchangers with inclined folded helical baffles." *Case Studies in Thermal Engineering* (2025): 107418. https://doi.org/10.1016/j.csite.2025.107418
[16] Mohanty, Shuvam, and Rajesh Arora. "CFD analysis of a shell and tube heat exchanger with single segmental baffles." *International Journal of Automotive and Mechanical Engineering* 17, no. 2 (2020): 7890-7901. https://doi.org/10.15282/ijame.17.2.2020.08.0589
[17] Bergman, Theodore L. *Fundamentals of heat and mass transfer*. John Wiley & Sons, 2011.
[18] Najm, Alyaa Qasim, Saadoon Fahad Dakhil, and Ahmed Qasim Mohammed. "Numerical analysis of shell and tube heat exchanger with different baffle configurations performance." *International Journal of Heat & Technology* 42, no. 4 (2024). https://doi.org/10.18280/ijht.420423
[19] Han, Tianding, Qifei Li, Lin Shang, Xiangyu Chen, Feng Zhou, and Wangxu Li. "Study on the influence of reynolds number on heat exchange performance and nusselt number of spray coil heat exchanger." *Processes* 13, no. 2 (2025): 588. https://doi.org/10.3390/pr13020588
[20] Cengel, Yunus A., and Afshin Jahanshahi Ghajar. "Heat and mass transfer: fundamentals and applications." *(No Title)* (2015).
[21] Ambekar, Aniket Shrikant, R. Sivakumar, N. Anantharaman, and M. Vivekenandan. "CFD simulation study of shell and tube heat exchangers with different baffle segment configurations." *Applied Thermal Engineering* 108 (2016): 999-1007. https://doi.org/10.1016/j.applthermaleng.2016.08.013