RESEARCH ARTICLE **ISSN 2738-9898 (Print) ISSN 2738-9901 (Online)**

Effect of Baffles on Heat Transfer and Pressure Drop of Shell and Tube Heat Exchanger

Sanjeev Maharjan* and Shaurav Suwal

Department of Mechanical and Aerospace Engineering, Pulchowk Campus
Institute of Engineering, Tribhuvan University, Lalitpur, Nepal
** Author to whom correspondence should be addressed; Email: mechsanjeev@ioe.edu.np*

Received : 01 July 2025; Received in revised form : 25 November 2025; Accepted : 05 December 2025; Published: 29 January 2026

Abstract

This research emphasizes on the effect of baffles and baffle cut for a simple shell and tube type heat exchanger. Heat transfer between separate medium can result in heat loss to the environment and inefficiency in heat transfer. So, in order to minimize losses and improve heat transfer properties, corrective action is needed. This is where baffle spacing and baffle cut plays its role and improves the heat transfer phenomena. The process in analyzing consists of modelling, meshing and simulating the geometry of shell and tube heat exchanger using computational Fluid Dynamics (CFD) in ANSYS. The geometry consists of seven tubes within the shell where water is considered as the working fluid and only shell side flow characteristics is observed. The highest heat transfer was achieved for the shell and tube heat exchanger with 25% baffle cut and 8 number of baffles.

Keywords

Baffle, Heat Transfer, shell, tube, heat exchanger

1. Introduction

Heat exchangers are devices that transfer heat from one medium to another. These media may consist of gases, liquids, or a mixture of both. The media may be separated by solid walls to avoid mixing, or they may be in direct contact. Heat exchangers transfer heat from systems that don't need it to other systems where it can be used more effectively, increasing a system's energy efficiency. To facilitate the heat transfer in a heat exchanger, the fluid typically flows rapidly by forced convection. The fluid's pressure drops as a result of the rapid flow. The efficiency of a heat exchanger refers to how well it transfers heat in relation to the resulting pressure drop. The latest heat exchanger technology minimizes pressure drop while maximizing heat transfer and also meets other design goals such as resistance to high fluid pressures, resistance to contamination and corrosion, and the ability to be cleaned and repaired [1].

Shell and tube heat exchangers are the most popular and widely used type of heat exchanger. In these heat exchangers one fluid flows through inside the tubes and the other fluid flows through the shell over the tubes. It is possible to arrange tubes inside the shell to permit cross-flow, counter-flow, parallel flow, or both. Large pressures can be handled by this type of exchanger because of its tubular construction. Flow diverters are frequently used in shell and tube heat exchangers to increase the fluids ability to transfer heat by causing the shell-side fluid to flow more turbulently.

Common flow diverters include rod, disk, and donut baffles, helical baffles, segmental baffles, and orifice baffles etc. Baffles enhance the heat transfer by directing the flow through shell in a desired pattern which increases the turbulence and reduces the stagnant pockets in the heat exchanger. Baffles are often used as a central element in the design of shell and tube heat exchangers. In shell and tube heat exchangers, baffles are crucial for directing flow and generating turbulence. As a result, the heat exchanger operates more efficiently [2].

Computational fluid dynamics (CFD) uses applied mathematics, physics, and computer software to visualize the flow of gases and liquids and their impact on the objects they pass through. The Navier-Stokes equations serve as the foundation for computational fluid dynamics. The relationship between a moving fluid's temperature, pressure, density, and velocity is expressed by these equations. CFD is useful for studying fluid flow, temperature, heat transfer etc. In order to find numerical solutions for pressure distributions and temperature gradients, CFD splits the system up into small cells and applies governing equations to each of these individual elements. Before adding actual physics to the model, the software can also build a virtual prototype of a system or gadget. It also offers data and images that forecast how well this design will work.

The research aims in analyzing the performance of a shell and tube heat exchanger with varying number of baffles and baffle cut using computational Fluid Dynamics. Baffle cut is the height of segment that is cut in each baffle to allow the shell side fluid to flow across the baffle; it is commonly expressed as a percentage of the shell inside diameter that is not covered by the baffle. When designing shell and tube heat exchangers, the standard recommended value of 20% to 25% cut of shell diameter is applied. The larger the cut, the lower the pressure drop; however, this has a negative effect on heat transfer because it creates a poorly distributed flow with large eddies, which leads to dead spaces or stagnant areas behind the shell's baffles. This lowers the heat transfer coefficient. With a smaller cut the pressure drop is higher and the heat transfer coefficient is also higher [2]. The baffle cut orientation can differ from horizontal cut and vertical cut and varies between 15% and 45% of the shell inside diameter. This research is be based on studying the pressure, temperature and flow characteristics inside the shell using horizontal baffle with 8 number of baffles and baffle cut 25%, 35% and 45%.

2. Literature Review

Various researches have been carried out to minimize the loss and improve the heat transfer at minimum cost. Ender Ozden et al. [3] investigated how shell diameter, baffle spacing, baffle cut, and pressure drop are related to the coefficient of heat transfer. They conducted the simulation for a heat exchanger with a single pass single shell and tube. They discovered that more cross flow and heat transfer area were attained with a reduction in baffle spacing. The area behind the baffles was less effective in the heat transfer process when there was greater spacing between them because after striking the baffle, the flow direction changes and a recirculation zone develops. However, if there is less space between the baffles, the fluid strikes the back face of the previous baffles again, increasing the effective heat transfer area. They so noticed that more heat transfer was occurred with a greater number of baffles.

Avinash et al. [4] used numerical analysis to examine how baffle spacing, baffle cut, and shell diameter affect the heat transfer coefficient and pressure drop. They contrasted their findings with those obtained using the Bell-Delaware approach. They conducted a series of simulations with a different number of baffles and turbulent flow for two baffle cut values. By adjusting the flow rate, they looked at the impact of the baffle spacing to shell diameter ratio on the heat exchanger's performance. It was found that the results depend on the choice of turbulence model, and the optimal model among those taken into consideration was chosen by contrasting the Bell-Delaware method results with the CFD results for the heat transfer coefficient, outlet temperature, and pressure drop.

Arjun et al. [5] conducted research to predict the performance of a shell and tube heat exchanger. The heat exchanger's performance was evaluated using the CFD package FLUENT and compared to existing experimental values. An attempt was also made to calculate the performance of the heat exchanger using helix baffles rather than regular Segmental Baffles, and the results were compared. The results of the numerical experimentation revealed that the performance of the heat exchanger is improved when using a helical baffle rather than a segmental baffle. Noor et al. [6] used CFD to analyze a simple shell and tube heat exchanger, running a series of simulations for different mass flow rates by changing the baffle orientation and baffle cut values. The shell outlet temperature, shell side pressure drop, and heat transfer coefficient values were calculated using the CFD results. Mesh dependency was tested to see how mesh density affected the outlet characteristics. The effect of baffle orientation was observed by comparing the results of horizontal and vertical baffle orientation. For lower baffle cut values, the horizontal baffle produced a greater pressure drop than the vertical baffles. Heat transfer coefficients was also higher in horizontal orientation than in vertical orientation. Chetan et al. [7] calculated the effect of baffle cut on heat transfer coefficient and pressure drop keeping the baffle spacing constant. They discovered that the pressure drop was less for the 30% baffle cut and the heat transfer coefficient was nearly the same for the 30% and 25% baffle cuts.

Peng et al. [8] evaluated the influence of different baffles on heat transfer coefficient while maintaining a constant pressure drop. They discovered that a shell and tube heat exchanger with a segmental baffle has a poorer heat transfer coefficient than a shell and tube heat exchanger with helical baffles for the same pressure drop. They also discovered a relationship between the Nusselt number and the Reynolds number, as well as the friction factor and the Reynolds number. Kiran et al. [9] examined the influence of baffle spacing on heat transfer, pressure drop, and shell side outlet temperature in a shell and tube heat exchanger with baffles. They hypothesized that baffle spacing has little effect on exit temperature, however mass flow rate has a big impact. They also discovered that the pressure loss varies significantly with mass flow rate and baffle spacing. Prasanna et al. [10] investigated the hydrodynamic and heat transfer effects of varying baffle cut and spacing on shell and tube heat exchangers. They discovered that a 25% baffle cut yields marginally better performance. Heat transfer increases when baffle spacing decreases.

3. Methodology

The research is aimed to predict the performance of a shell and tube heat exchanger with varying numbers of baffles and baffle cuts using Computational Fluid Dynamics (CFD) as shown in Figure 1. In this study, three different baffle cuts are implemented in the shell, creating distinct flow paths across the tube bundle. The geometry modeling is executed using ANSYS software, and the models are compared by altering the baffle cut configurations within the shell. Subsequently, each geometric model undergoes CFD analysis, and results are obtained through meshing the geometry with appropriate parameters. This research involves three steps: geometry development, mesh generation and finally simulation and visualization.

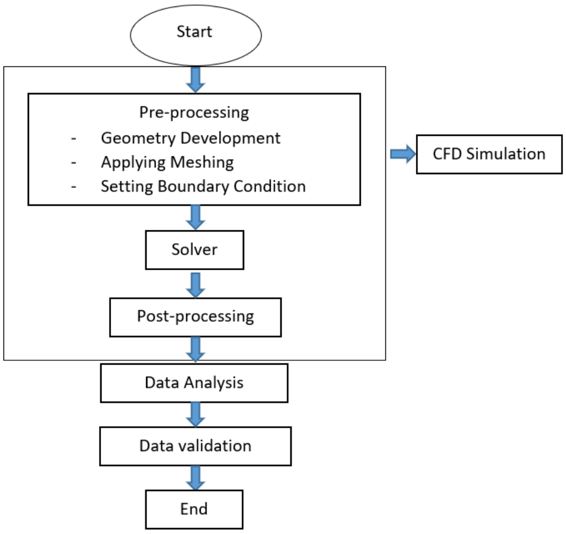

Figure 1: Design methodology

3.1 Pre-Processing

The preprocessing stage encompasses tasks such as creating the geometry, preparing the geometry for Computational Fluid Dynamics (CFD) simulation, and dividing the domain into small volumes or cells through a process known as meshing or grid generation.

3.2 Geometry Development

This entails designing the geometry of the shell and tube heat exchanger with three different numbers of segmental baffles and baffle cuts. The dimensions of various parts of the heat exchanger are detailed in the Table 1.

Table 1: Design Parameters of Heat Exchanger

Particulars	Value
Shell diameter(D_s)	100mm
Shell inlet & outlet tube diameter(d_s)	20mm
Tube Diameter (d)	20 mm
Tube bundle geometry and pitch	Triangular, 25 mm
Number of tubes (N_t)	7
Heat exchanger length (L)	450 mm
Shell side inlet temperature (T)	363 K
Tube side inlet temperature (t)	300 K
Baffle cut (B_c)	25%, 35%, 45%
Number of baffle (N_b)	8
Baffle spacing	Equispaced

The shell and tube heat exchanger with a baffle cut of 25%, and the configuration of 8 baffles is illustrated in Figure 2 (a). The baffles are uniformly spaced throughout the length of the shell and tube heat exchanger, with cold water flowing through the tubes and hot water circulating inside the shell over the tubes.

3.3 Mesh Generation

Meshing of the geometry is accomplished using the ANSYS meshing tool. The shell volume and surfaces of the model are meshed utilizing tetrahedral elements. For 8 numbers of baffles and 25 % baffle cut, the number of mesh elements are 885468. For 35% baffle cut the elements are 887338 and for 45% baffle cut, the elements are 890960 respectively. Mesh geometry are shown in Figure 2 (b) and (c). Mesh convergence simulation is provided in Figure 2 (d). Simulation convergence is necessary to obtain the parameters of shell and heat tube exchanger. Continuity, Velocity in X, Y and Z directions, energy, k and epsilon must converge in a certain region and remain flat or steady after iterations. The energy value can be set 10^{-4} for convergence criteria for general CFD simulation work. It depends on level of complexity of mesh geometry and shapes.

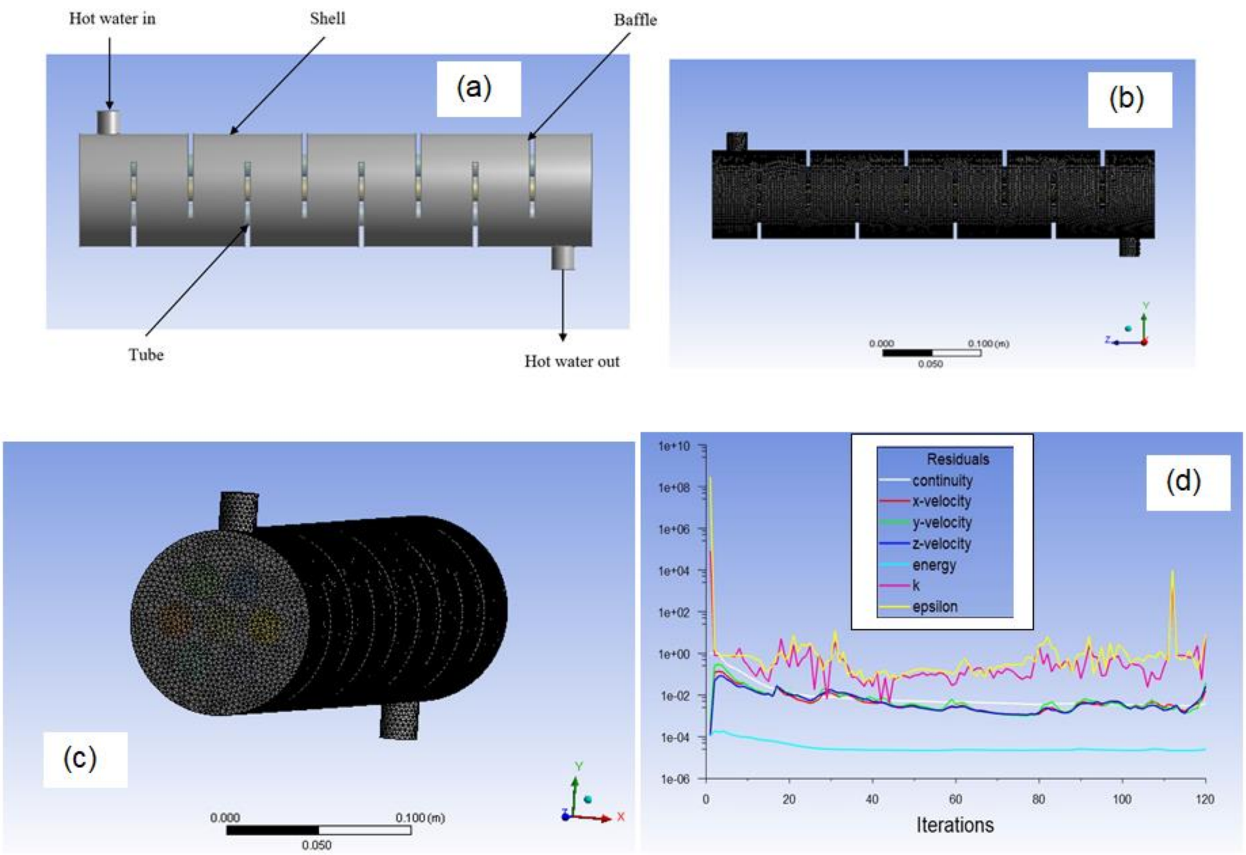

Figure 2: (a) Geometry of shell and tube heat exchanger with 8 number of baffles (b) and (c) Meshing (d) convergence simulation

3.4 Boundary Conditions

For various zones, different boundary conditions are used. The outlets and inlets are pressure outlets and velocity inlets respectively. Both hot and cold fluids input velocities 0.4 m/s and 1.2 m/s respectively are maintained constant. The default outlet pressure, or atmospheric pressure, is maintained. The inlet temperature of the hot fluid is 363 K, while the cold fluid is maintained at 300 K. In accordance, the other wall conditions are defined. The ambient air temperature is maintained at 300 K.

3.5 Solving or Processing

In this step, the discretized equations are now being iteratively solved by the CFD simulation software using the CFD solver. The set of solution methods involved in solving are power law of energy, coupled scheme, least square cell based gradient, linear pressure, second order upwind momentum, second order upwind turbulent kinetic energy, and second order turbulent dissipation rate.

3.6 Solution Initialization

Initialization of the solution is done using the standard method. The simulation is set for 60 iterations with the boundary condition and solution initialize condition mentioned above.

3.7 Post-Processing

After the completion of the solution, the simulation's outcomes is qualitatively and quantitatively analyzed and visualized using plots, reports, monitors, 2D/3D images, and animations. This involves the analysis and visualization of flow, temperature and pressure inside the shell with different segmental baffle cut. The cross-sectional velocity, streamline, temperature, and pressure contours are studied with the help of simulation in ANSYS for baffle cut plane. A plane at the cross section is used to create the contours, and the features are represented in a color map based on their magnitude. Following assumptions are considered during the analysis a) the flow is incompressible and the fluid's density remains constant b) leakage from the space between the baffle and tube is not considered c) heat transferred to the baffles are not considered in the analysis d) water serves as the working fluid, and its properties are considered to be constant and e) the impact of the header is not considered.

3.8 Governing Equations

The governing equations in Computational Fluid Dynamics (CFD) are rooted in the conservation laws of a fluid's physical properties. These laws encompass the conservation of mass, momentum, and energy, and they form the foundation for understanding and simulating the behavior of fluids in various scenarios. The equation for the conservation of mass is

$$\frac{D\rho}{Dt} + \rho(\nabla.\vec{v}) = 0 \tag{1}$$

where ρ is the density, $\vec{v}$ the velocity and ∇ the gradient operator and

$$\vec{\nabla} = \vec{i}\frac{\partial}{\partial x} + \vec{j}\frac{\partial}{\partial y} + \vec{k}\frac{\partial}{\partial z} \tag{2}$$

If the density is constant, the flow is assumed to be incompressible, and the continuity equation simplifies to

$$\frac{D\rho}{Dt} = 0 \rightarrow \nabla.\overrightarrow{v} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{dz} = 0 \tag{3}$$

The conservation of momentum, often referred to as the Navier-Stokes Equation, is given by

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla.(\rho\vec{v}\vec{v}) = -\nabla p + \nabla.\bar{\bar{\tau}} + \rho\vec{g} \quad (4)$$

where p is static pressure, $\bar{\bar{\tau}}$ is viscous stress tensor and $\rho\vec{g}$ is the gravitational force per unit volume. Viscous stress tensor $\bar{\bar{\tau}}$ can be specified as follow in accordance with Stoke's Hypothesis is

$$\tau_{ij} = \mu\frac{\partial v_i}{dx_j} + \frac{\partial v_j}{\partial x_i} - \frac{2}{3}(\nabla.\vec{v})\delta_{ij} \quad (5)$$

If the fluid is assumed to be incompressible with constant viscosity coefficient μ is assumed constant the Navier-Stokes equation simplifies to

$$\rho\frac{D\vec{v}}{Dt} = -\nabla p + \mu\nabla^2\vec{v} + \rho\vec{g} \quad (6)$$

The first law of thermodynamics, conservation of energy, states that an increase in heat and work added to a system results in an increase in energy within the system

$$dE_t = dQ + dW \quad (7)$$

where dQ is the heat added to the system, dW is the work done on the system and dE_t is the increment in the total energy of the system. The basic equation for heat transfer across a surface is given by:

$$Q = U \times A \times \Delta T_m \quad (8)$$

Where Q is heat transferred per unit time, U is overall heat transfer coefficient, A is heat transfer area and ΔT_m is log mean temperature difference.
The log mean temperature difference ΔT_m is

$$\Delta T_m = \frac{(T_1 - t_2) - (T_2 - t_1)}{ln\frac{(T_1 - t_2)}{(T_2 - t_1)}} \quad (9)$$

Where T_1 is inlet shell side fluid temperature, T_2 is outlet shell side fluid temperature, t_1 is inlet tube side temperature and t_2 is outlet tube side temperature.

4. Results and Discussion

After CFD modeling and boundary condition applied, a steady temperature is maintained for the flow through the tubes and hot fluid flows inside the shell. Water serves as the working fluid. The model used is a shell with a diameter (D) of 100 mm and an inlet and outlet with a circular cross section diameter (ds) of 20 mm. The shell's length is estimated to be 450 mm. 7 tubes of 16 mm in diameter (d) are placed. Constant fluid properties are assumed throughout the analysis.

Simulation involves a shell-and-tube configuration with eight baffles and three distinct baffle cut percentages (25%, 35%, and 45%). Using the simulation tool ANSYS, the velocity, streamline, temperature, and pressure contours at cross-sections are thoroughly examined. These characteristics are visually represented, depicting their magnitudes through a color map.

4.1 Velocity Contour

The velocity profile is analyzed to comprehend to comprehend the distribution of flow across the cross-section at various positions within the heat exchanger. Velocity plots across the cross-sections, corresponding to different baffle cuts (25%, 35%, and 45%) with 8 baffles, are presented in Figure 3.

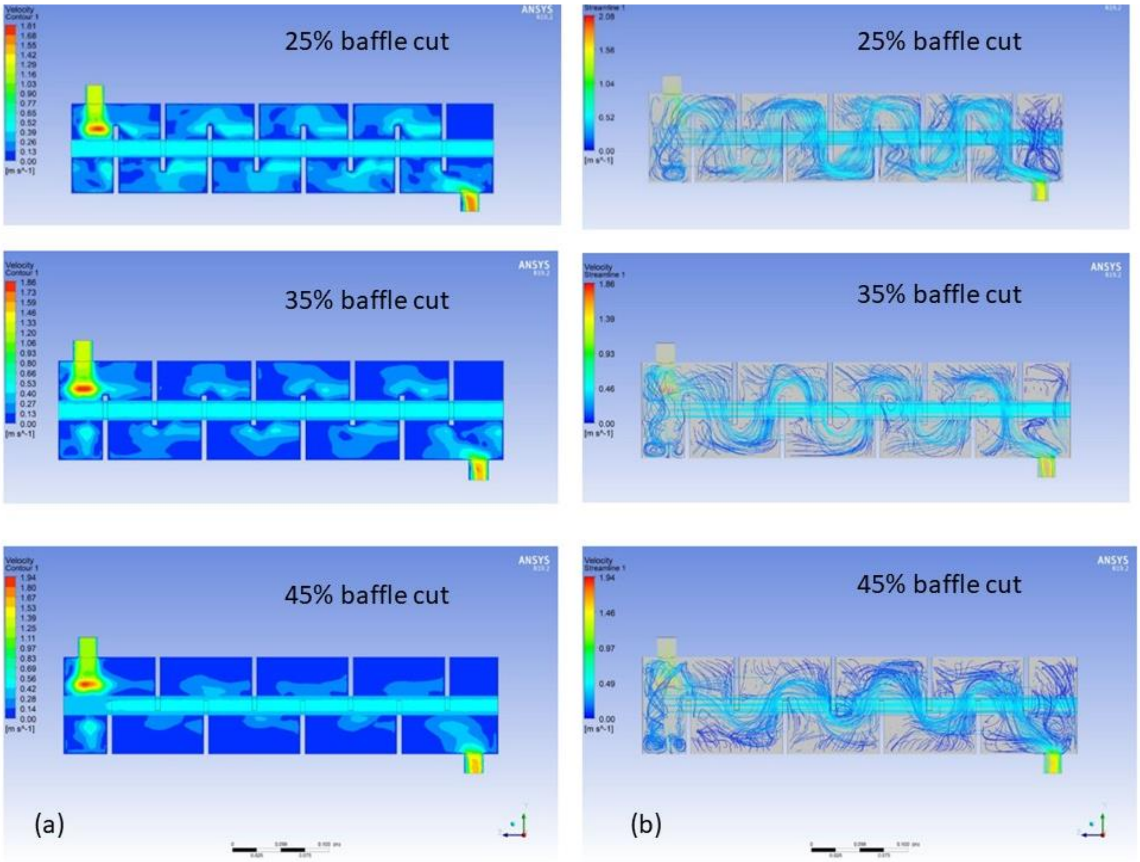

Figure 3: (a) velocity and (b) streamline for 25 %, 35 % and 45 % baffle cuts

Distinct flow patterns emerge with varying baffle cuts. The velocity contour associated with a 25% baffle cut reveals the creation of a recirculation zone at the backside of the baffle as the hot fluid enters the shell and traverses the baffles. In this region, the flow is notably reduced. Consequently, the available area for shell-side fluid flow is constrained, fostering more efficient heat transfer in this specific configuration. The velocity contour for the 35% baffle cut is depicted in second row of figure 3. In this configuration, the flow area is expanded compared to the shell and tube with a 25% baffle cut. However, this expansion results in dead zones forming between the two adjacent baffles, contributing to lower heat transfer efficiency. The velocity contour for the 45% baffle cut is illustrated in last row of the Figure 3. In this, the flow area is greater compared to the baffle cuts of 25% and 35%. However, it is observed that there is minimal cross-flow, resulting in comparatively lower heat transfer efficiency compared to the previously discussed cases.

4.2 Temperature Contour

The temperature contour plots across the cross-section, considering eight baffles are presented in figure 4. The initial temperature of the hot water entering the shell and tube heat exchanger is set at 363K, and variations along the length of the heat exchanger are observed.

The temperature contour plots for a 25% baffle cut are shown in Figure 4. As the hot water traverses the shell and baffles, a recirculation zone forms at the back side of the baffle, leading to reduced flow and subsequently lower temperatures in this region. Consequently, a more uniform temperature distribution is achieved with a 25% baffle cut. The outlet temperature on the shell side is measured at 354.34K, indicating higher heat transfer efficiency compared to shell and tube heat exchangers with 35% and 45% baffle cuts.

The temperature contour plots across the cross-section for a baffle cut of 35% are depicted in Figure 4. The presence of a recirculation zone at the back side of the baffle, accompanied by reduced flow, results in heightened heat transfer occurring predominantly at the edges. Consequently, the temperature of the hot fluid experiences more pronounced reductions at the edges in this configuration. The temperature contour plots across the cross-section for a baffle cut of 45% are presented in figure 4. In this configuration, the expanded area for the flow of shell-side fluid, in comparison to baffle cuts of 25% and 35%, results in reduced cross-flow. Consequently, the temperature in this scenario experiences a lesser decrease compared to the aforementioned cases, leading to lower heat transfer efficiency. The outlet temperature on the shell side is measured at 356.23K.

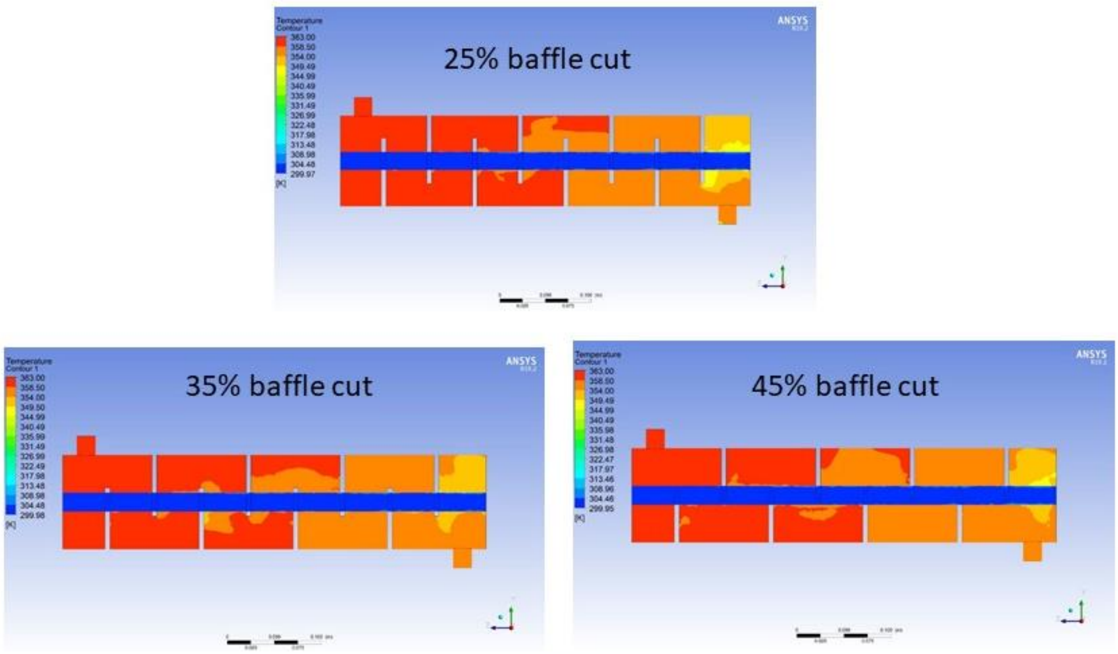

Figure 4: Temperature contour at 25 % , 35 % and 45 % baffle cuts

4.3 Pressure Contour

The pressure contour for a baffle cut of 25% is illustrated in figure 5. With a 25% baffle cut, the area available for fluid flow is restricted, leading to a higher pressure drop inside the shell.The pressure contours for a baffle cut of 35% are depicted in Figure 5.

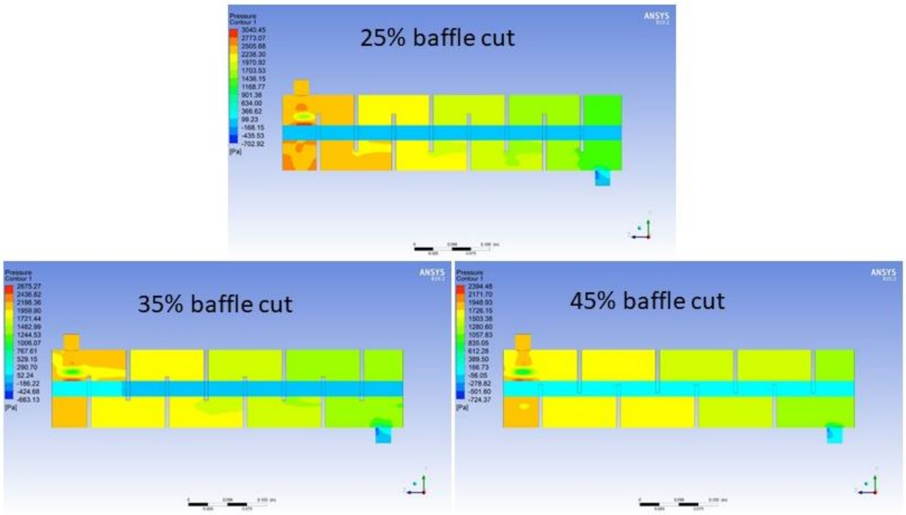

Figure 5: Pressure contour for 25 %, 35 % and 45 % baffle cuts

It is found that an increase in baffle cut corresponds to a decrease in pressure drop inside the shell. Consequently, with a 35% baffle cut, the pressure drop is reduced compared to a 25% baffle cut.The pressure contours for a baffle cut of 45% is also shown in Figure 5. With a 45% baffle cut, the increased area for fluid passage results in a reduced pressure drop inside the shell compared to both the 25% and 35% baffle cut configurations.

4.4 Validation of Results

For the validation and comparative analysis of the current CFD model, the research study conducted by Mishra [11] has been selected, which investigated shell and tube heat exchangers with varying numbers of baffles. The validation process involves assessing total heat transfer and pressure drop for configuration with 8 no of baffles. The comparative analysis reveals a consistent trend in both heat transfer and pressure drop, indicating an increase with the number of baffles for a constant baffle cut value. Notably, the maximum heat transfer is observed in the shell and tube heat exchanger with 8 baffles. Specifically, current research indicates a 6% to 8 % enhancement in heat transfer for the configuration with 8 baffles compared to 4 baffles. This aligns with existing literature findings, further validating the accuracy and reliability of our simulated results for heat transfer and pressure drop as has been seen in Figure 6.

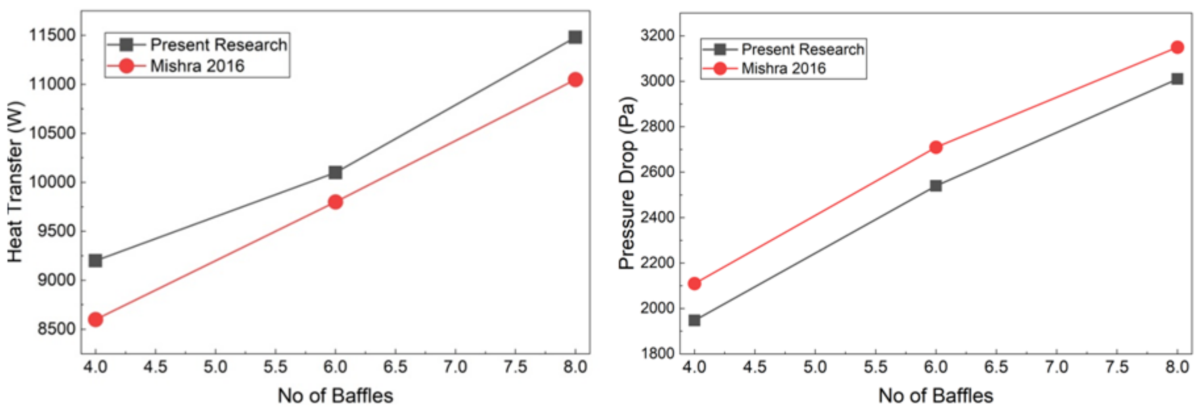

Figure 6 : Validation of CFD result with existing research result

5. Conclusions

In this study, CFD is used to analyze a small shell and tube heat exchanger. For shell and tube heat exchangers with different numbers of baffle cuts, a number of CFD simulations are run. Computational modeling proves to be an efficient technique for studying this type of thermal element. By adjusting the number of baffle cut values, models of shell and tube heat exchangers are designed and simulations are performed to examine their impact on heat transfer and pressure drop. The study observes the influence on the heat exchanger by comparing results obtained with different numbers of baffle cuts. The modification in baffle arrangement induces more turbulence in the shell-side flow, improving the heat exchanger's performance with a suitable baffle cut orientation. Furthermore, increasing the baffle cut is found to decrease the pressure drop inside the shell. Notably, a 25% baffle cut results in higher pressure drop compared to other percentages, decreasing as the baffle spacing and cut increase. Higher pressure drop necessitates more pumping power, reducing the overall system efficiency. Finally, 25% baffle cut results in efficient heat transfer.

Conflicts of interest statement

The authors declare that there are no conflicts of interest related to this work.

Data availability statement

The data that support the findings of this study are available from the corresponding author upon reasonable request.

References

1. Shukla, A. S., K. K. Bhabor, and D. B. Jani. "Investigation on Shell and Tube Heat Exchanger by Using CFD." *International Journal of Advanced Research in Science, Communication and Technology*, vol. 2, no. 5, 2022.
2. Akbar, A. A. A., and M. Reza. "Shell and Tube Heat Exchanger Optimization Using New Baffle and Tube Configuration." *Applied Thermal Engineering*, vol. 157, no. 1, 2019.
3. Ozden, E., and I. Tari. "Shell Side CFD Analysis of a Small Shell and Tube Heat Exchanger." *Energy Conversion and Management*, vol. 51, no. 5, 2010, pp. 1004–1014.
4. Jadhav, A. D., and T. A. Koli. "CFD Analysis of Shell and Tube Heat Exchanger to Study the Effect of Baffle Cut on the Pressure Drop." *Research in Aeronautical and Mechanical Engineering*, vol. 2, no. 7, 2014, pp. 1–7.
5. Sunil, A. *Design of Shell and Tube Heat Exchanger Using Computational Fluid Dynamics Tools*. 2014.
6. Afsar, N., and I. M. Ilias. "CFD Analysis of Shell and Tube Heat Exchanger with Different Baffle Orientation and Baffle Cut." *International Conference on Mechanical Engineering*, vol. 1980, no. 1, 2018.
7. Patil, C. N., and N. S. Bhalkikar. "CFD Analysis of Shell and Tube Heat Exchanger to Study the Effect of Baffle Cut on the Pressure Drop and Heat Transfer Coefficient." *International Journal for Scientific Research and Development*, vol. 2, 2014, pp. 649–654.
8. Peng, B., et al. "An Experimental Study of Shell-and-Tube Heat Exchangers with Continuous Helical Baffles." *Journal of Heat Transfer*, vol. 129, no. 10, 2007, pp. 1425–1431.

9. Kiran, K., A. M. Mulla, Manoj, and C. J. Umesh. “Investigation of Baffle Spacing Effect on Shell Side Heat Transfer Characteristics in Shell and Tube Heat Exchanger Using Computational Fluid Dynamics.” *Thermal Engineering*, vol. 73, 2014, pp. 26022–26026.
10. Prasanna, J., et al. “A Numerical Analysis of Hydrodynamic and Heat Transfer Effects of Shell and Tube Heat Exchanger for Different Baffle Space and Cut.” *Mechanica Confab Conference*, vol. 2, 2013.
11. Mishra, A. *CFD Analysis of Shell and Tube Heat Exchanger with Segmental Baffles*. National Institute of Technology, 2016.