International Information and Engineering Technology Association
Advancing the World of Information and Engineering

International Journal of Heat and Technology
Vol. 43, No. 2, April, 2025, pp. 603-617
Journal homepage: http://iieta.org/journals/ijht

Effect of Tube Arrangements on Water Heat Transfer in a Rectangular Channel: A Computational Fluid Dynamics Study

Ayad K. Hassan[1*], Mohammed M. Aldabbagh[2], Karar Saeed Mohammed[3], Muhammad Asmail Eleiwi[4], Hasan Shakir Majdi[5]

[1] Department of Materials Engineering, University of Technology- Iraq, Baghdad 10066, Iraq
[2] Power Mechanics Technology Department, Northern Technical University, Kirkuk 36001, Iraq
[3] Department of Medical Physics, Almanara College for medical Sciences, Maysan 62001, Iraq
[4] Electromechanical Engineering Department, College of Engineering, University of Samarra, Samarra 34010, Iraq
[5] Department of Chemical Engineering and Petroleum Industries, Al-Mustaqbal University College, Hillah 51001, Iraq

Corresponding Author Email: 130048@uotechnology.edu.iq

Copyright: ©2025 The authors. This article is published by IIETA and is licensed under the CC BY 4.0 license (http://creativecommons.org/licenses/by/4.0/).

https://doi.org/10.18280/ijht.430221

Received: 2 January 2025
Revised: 2 April 2025
Accepted: 16 April 2025
Available online: 30 April 2025

Keywords:
heat exchanger efficiency, tube arrangements, mesh independence test, thermal performance optimization, and pressure drop analysis

ABSTRACT

This work uses ANSYS Fluent to establish arrangement tube effects (inline, triangular, and staggered) on heat exchangers' thermal and fluid dynamics. Further, the study considered heat transfer enhancement category, outlet temperature, and pressure drop over these configurations. The results reveal that the triangular flow configuration offered the best heat transfer efficiency rating of 49.2%, which is credited to turbulence and mixing promotion. Nevertheless, this configuration also illustrates the highest pressure drop, which raises energy consumption. Inline configuration showed relatively high efficiency (45.6%) using the smallest pressure drop and would be suitable for less energy-intensive processes. Staggered arrangement was least efficient and had 42.4% efficiency because of flow inefficiencies and cold fluid circulation with balanced pressure drop. Mesh independence tests further affirmed simulations credibility; for inline and staggered meshes, a size of 0.01 m was established as accurate; the same applied to triangular meshes with a 0.008 m size. Temperature at the model's outlets was 288.6 K, 287.7 K, and 289.4 K in inline, triangular, and staggered arrangements. Triangular configuration gave the lowest outlet temperature in comparison with other orientations and therefore showed the best cooling characteristics, while the staggered one had the worst characteristics owing to the localized heating effect. This work presents a detailed assessment of tube arrangement types, which will be useful in developing new heat exchangers with enhanced thermal efficiency. Triangular configuration is most suitable for high heat transfer rates, while inline arrangement is suitable for low-pressure drop rates.

1. INTRODUCTION

Heat exchangers play main roles in several industrial processes, particularly when cooled or heated fluids are required. The flow and arrangement of tubes within a channel is a key factor in heat exchanger design and performance, having implications for heat transfer and pressure drop. Computational Fluid Dynamics (CFD) has become an accepted modelling technique used to predict the behavior of fluids and heat exchange in a specific fluid flow and hence aid in the design of new products before physical models are developed [1]. In the case of WBHXs, perhaps no other configuration has as much impact as how the tubes are laid. In-line, staggered, or triangular tube arrangements can significantly affect the nature of the flow, turbulence intensity, and heat transfer rate. Hydrodynamic and thermal simulations of these systems are made possible by CFD tools such as ANSYS Fluent. Specifically, this study is an attempt to determine the impact of the various tubular structures within the rectangle on the magnitude of heat transfer from the water in the channel, intending to determine the best possible tube arrangement that will enhance heat exchanger efficiency and performance using ANSYS Fluent [2]. The purpose of this work is to investigate the dependency of heat exchanger tube patterns on the thermal-hydraulic efficiency of water-cooled heat exchangers. Thus, through the investigation of several tube layouts in this research, a layout of the greatest heat transfer coefficient and the least pressure drop is sought, which is fundamental to a cost-effective and energy-rational heat exchanger system.

Concerning the influence of tubes in the heat transfer in rectangular passages, there has been extensive interest in thermal engineering. Research about the effect of various arrangements on thermal characteristics, pressure drop, and flow has been widely investigated in past literature [3]. Quite several tube arrangements, including inline, staggered, and triangular, have been used in the CFD analyses to assess the effect of arrangement on thermal performance. Inline tube arrangements are most preferred in compact heat exchangers because of their simplicity and manufacturability [4].

However, it is found that the arrangement of the tubes in a staggered manner greatly improves heat transfer, as it increases the profile of the fluid and enhances the mixing of the fluid. Uguru-Okorie et al. [5] confirmed that when the tubes are arranged in a staggered manner, the corresponding Nusselt numbers are significantly higher, and thermal performance in a rectangular channel is higher than in inline. However, inline seems to provide more laminar flow, hence limiting the heat transfer capacity, as explained by Djeffal et al. [6]. Other parameters of heat transfer are the distances between the tubes and the diameters of the tubes. Higher turbulence and enhanced heat transfer are obtained at smaller spacing, though it may enhance pressure drop [7]. Larger tube diameters, on the other hand, can increase the heat transfer area but decrease heat transfer coefficients when the flow becomes very turbulent and the pressure drop is very high [8]. Likewise, ANSYS Fluent-based CFD simulation has emerged as a significant tool for understanding the heat transfer behavior of tube-based systems. Fluent in ANSYS can enable the study of flow, heat transfer, and pressure drop of different tube arrangements, and the responses generated can be used to make comparisons. In the ongoing research articles by Surakasi and Prasanna Kumar [9] and Ibrahim et al. [10], the authors made use of ANSYS Fluent to carry out water flow through several tubes, and it was found that, out of all, the staggered layout benefitted the best thermal performance because of the enhanced turbulent flow.

Heat transfer in a tube system depends on the flow regime established in the system. As flow changes from laminar to turbulent, heat transfer coefficients increase because turbulence increases the efficiency of the fluid mix. Jayavel and Tiwari [11] showed that configuring tubes in ways that create higher levels of turbulence enhances heat transfer because turbulence interferes with the development of thermal boundary layers. Nevertheless, in turbulent flows, pressure losses are also higher, which may lead to lower energy efficiency of the whole system [12]. Other improvements, like ribbed or finned tubes and other improved tube layouts, have also been further investigated concerning improved heat exchange. For instance, the flow inside the ribbed tubes can be subjected to extra turbulence and correspondingly, the heat transfer enhancement potential as pointed out by Tanda [13]. As well, the enhancement of tube arrangement by these methods has been demonstrated in the way that the total performance is enhanced with reasonable pressure drops. The use of different configurations of tubes is not only limited to research and experimental models; it is widely used in industrial heat exchangers in power, cooling, refrigeration, heating, ventilation, and air conditioning systems. The field studies by Wang et al. [14] and Menni et al. [15] identified that when heat transfer perks of staggered arrangement were best, other factors, including cost, ease of maintenance, and the viable industrial real estate, also matter. Therefore, the disposition of the tubes in a rectangular channel determines the heat transfer and pressure drop of the heat exchanger in question. Literature using ANSYS Fluent for CFD analysis has offered insight into different configurations of tubes in water to heat exchangers' performance. Inline tube layouts are generally found to be less efficient than staggered tube layouts for heat transfer, but efficiency can be strongly influenced by a range of factors such as tube spacing, diameter, and flow characteristics. More investigation is required to study more elaborate configurations and to optimize adiabatic improvement along with energy saving [16]. Furthermore, the creation of complex forms of tubes arranged in various geometries and in interaction with other surfaces toward individual heat transfer enhancing mechanisms may offer new potential to enhance the heat exchangers.

What is novel about the work is that it investigates, for the first time, three kinds of tube arrangements, including inline, triangular, and staggered arrangements, to determine their impact on thermal performance and pressure drop. This work does not limit itself to a single configuration, similar to many previous works, which is a strength; most studies do not contain comparative analysis. This study conducts a detailed mesh independence test for the simulation results. This strict procedure adds up to the certainty of the data provided by the research. Consequently, the systematic analysis of thermal performance (efficiency) and pressure drop of different configurations offers useful guidelines for designing energy-efficient heat exchangers at a reasonable cost. Further, the research helps provide valuable improvement to heat exchanger design and desired trade-offs between efficiency of heat and thermal performance.

Industrial processes need heat exchangers to reach better process optimization outcomes and save energy through improved heat transfer operating efficiency. The final system efficiency changes based on how tubes are arranged in heat exchangers and how this arrangement affects both performance and pressure drop. The assessment of triangular tube arrangements lacks sufficient research, while guidebook studies focus on evaluations between inline and staggered arrangements. A study addresses water-cooled heat exchanger behavior at the thermal and hydrodynamic level when utilizing linear, triangular and staggered tube arrangements to solve an important research void. The research provides a distinctive value because it completely examines the thermal properties of triangular tubes and their performance in relation to standard inline and staggered tube layouts. Unlike previous research, this study analyses all three arrangements in one analysis to demonstrate their performance regarding heat transfer efficiency with pressure drop impact.

The author provides new insights into heat exchanger design literature by evaluating the triangular structure through mesh-independent research. The systematic research provides industrial designers critical outcomes for selecting tubes in heat exchangers to reach maximum thermal performance alongside optimal energy efficiency targets.

2. METHODOLOGY

The chosen method in this research is CFD to analyses the effect of variations in tube layouts on the capability of carrying heat for water within a rectangular channel. The professional CFD tool ANSYS Fluent is used to predict the flow, heat transfer, and pressure drop characteristics of the various tube arrangements [17]. The main steps of the methodology are the choice of the model geometry, the definition of boundaries and boundary conditions, the discretization of the domain, the solving of the numerical simulations, and the examination of the data elaboration. Every single phase of the study is going to make certain of the work’s credibility and reliability.

3. MODEL GEOMETRY AND TUBE ARRANGEMENTS

The flow arrangement of the heat exchanger model is built inside a rectangular channel and a series of tubes in different layouts using ANSYS Design Modeler. As the simulations of the channel will be complex, the geometrical representation of the channel will be in three dimensions. The length of the channel will be 2 m with a 1 m width and 0.4 m depth. The tubes used in the simulation will have a 0.1 m diameter and 0.4 m length, and the spacing between tubes will be varied in the simulations, as illustrated in Figure 1.

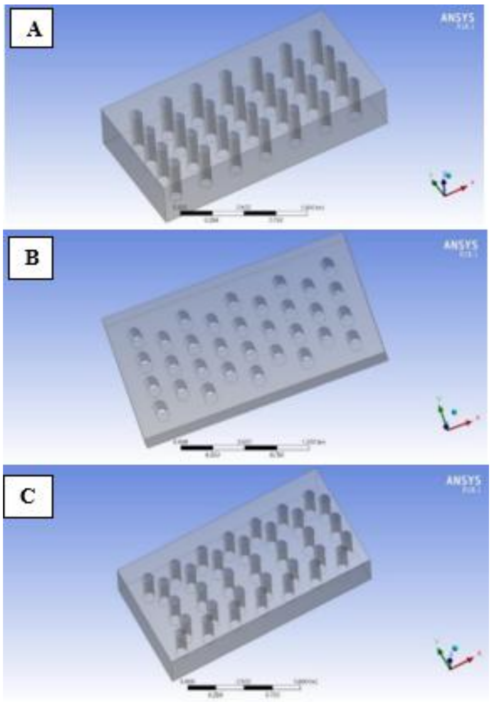

Figure 1. ANSYS design modeler geometry: (A) inline design, (B) triangle design, and (C) staggered design

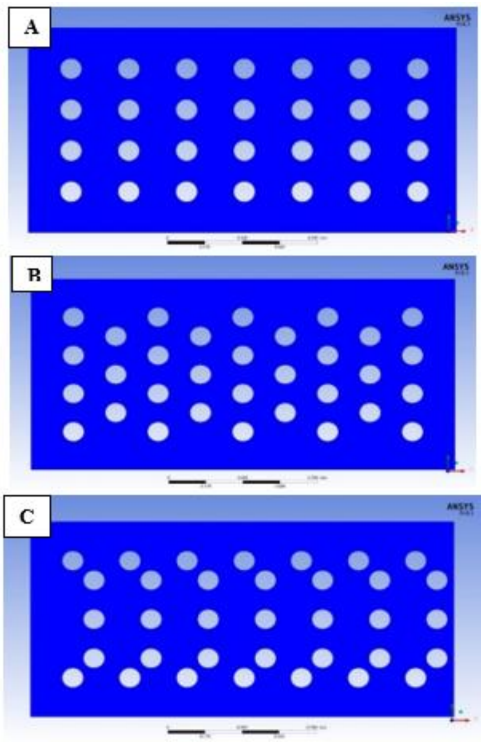

Figure 2. Tube arrangements integrated in heat exchangers: (A) Inline tube arrangement, (B) Triangular tube arrangement, (C) Staggered tube arrangement

The study will be based on the following tube arrangements that are usually integrated in heat exchangers: there are three arrangements depending on the position of the tubes within a heat exchanger. In inline tube arrangement, the tubes are arranged in straight, parallel fashion with equal space between the tubes. Staggered tube arrangement, when tubes are arranged such that each tube in a subsequent row is placed offset from the row in front of it, enhances turbulence and enhances the mixing of the fluid [18]. A triangular tube arrangement is a type where tubes are positioned in a triangular manner with each row being at a 60° angle with the other. This is to improve heat transfer by increasing disturbances within the flow and the secondary flow tendencies. Three of them are depicted in Figure 2.

4. SIMULATION MODEL MESH PROPERTIES

It is clear from the earlier research that mesh quality is a crucial determinant of the level of realism and reproducibility of simulations performed in ANSYS. The formation of a mesh divides the geometry into finite elements, which the solver can use to approximate the solution to many physical problems. Accurate meshes create a closer representation of the actual behavior of the model, minimizing the discretized errors and improving convergence during the solution phase [19]. Unsatisfactory meshes, with non-uniform shapes, high aspect ratios or distorted elements, could result in erroneous solutions, numerical oscillations and even failure of the solver. Since the mesh sensitivity analysis is crucial for achieving accurate and reproducible numeric results, authors need to make certain that the results do not depend on mesh refinement. This process entails generating another set of meshes that have a higher number of elements or possess better quality and carrying out a simulation on every one of them. The results, including temperature or other output parameters of interest, are then compared using the meshes in question [20]. If the results can stop varying anymore, although getting more refined, then they are considered mesh-independent. This supports the notion that the mesh in this simulation is fine enough to provide a detailed and accurate representation of the physical processes without a lot of wastage of computation resources. The mesh procedure for this study starts with setting up the selected mesh size and the mesh method for the whole body, then the inflation option is applied for a more accurate mesh according to the parameters listed in Table 1 for all model faces without the inlet and outlet faces for the three simulation models.

Table 1. Inflation option parameters

Inflation Option	No. of Layers	Growth Rate	Max. Thickness
Total thickness	12	1.2	0.012 m

Table 2. Inline design mesh independence test results

Element Size	Nodes No.	Elements No.	Outlet Temp.
0.005 m	9833682	9639880	292.84504 K
0.008 m	4350376	4247324	290.42802 K
0.010 m	1778621	1729216	288.65363 K
0.013 m	972152	942445	288.81740 K
0.017 m	517283	499680	284.99766 K
0.020 m	353535	341024	282.92344 K

Table 3. Triangle design mesh independence test results

Element Size	Nodes No.	Elements No.	Outlet Temp.
0.005 m	9833682	9639880	289.84504 K
0.008 m	2120739	2042500	287.74820 K
0.010 m	1174755	1125000	287.97681 K
0.013 m	618114	588383	286.11826 K
0.017 m	417283	369680	285.28263 K
0.020 m	206578	194250	284.52405 K

Table 4. Staggered design mesh independence test results

Element Size	Nodes No.	Elements No.	Outlet Temp.
0.005 m	8634901	8836220	293.84504 K
0.008 m	2120739	2042500	291.76318 K
0.010 m	1174755	1125000	289.42901 K
0.013 m	618114	588383	289.90233 K
0.017 m	414784	389625	287.99545 K
0.020 m	237520	289320	285.22523 K

The mesh independence test for this work starts with a high element size (0.02 m) and determines the output temperature average; then the element size gradually decreases until the results have very little change. The results of the mesh independence test for the three cases in this study are listed in Tables 2, 3, and 4.

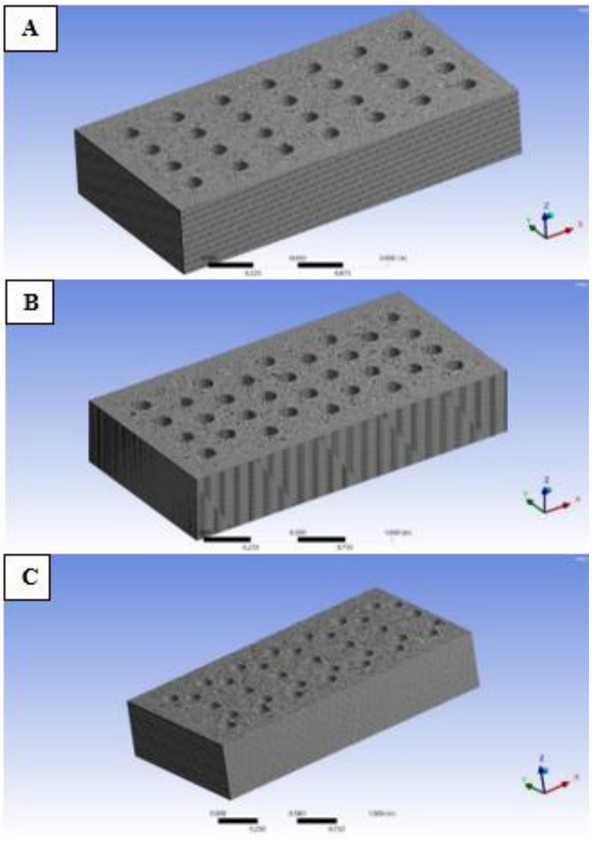

Figure 3. ANSYS mesh for (A) inline tube arrangement, (B) Triangular tube arrangement, and (C) Staggered tube arrangement

The results in Table 2 show mesh element sizes between 0.01 and 0.013 m show very little change, about 0.16 K, compared with the high alteration between the other element sizes for the inline design. For this, the mesh element size of 0.01 m was selected in this study with 1778621 nodes and 1729216 elements. Table 3 illustrates triangle design test results; the results between 0.008 and 0.01 m are almost the same with very little divergence. For that, the 0.008 m mesh size was used for this model with 2,120,739 nodes and 2,042,500 elements. Staggered design mesh test results in Table 4 illustrated very little alteration between 0.010 and 0.013 m, so the 0.01 m mesh size was selected for this model; the three models meshes are illustrated in Figures 3(A), (B), and (C).

Because of mesh independence tests significance in connectivity simulations to establish the validity of results, the above three tables are combined in one chart in Figure 4. The results of the chart show that larger elements may reduce geometry and omit certain flow aspects necessary for correct calculations. Moreover, smaller elements are computationally costly when it comes to time as well as memory needed for the computations [21]. The condition of the smallest element size that will ensure convergence is the best because it is the most efficient. From the graph, it is possible to determine the smallest element size that a stable outlet temperature is achievable by observing the plateaus [22]. This is the required element size in the mesh with which the solution is mesh independent. When comparing these designs, it is easier to determine whose performance may be more responsive to mesh size, which might help when deciding practical applications or in the case of additional optimization studies.

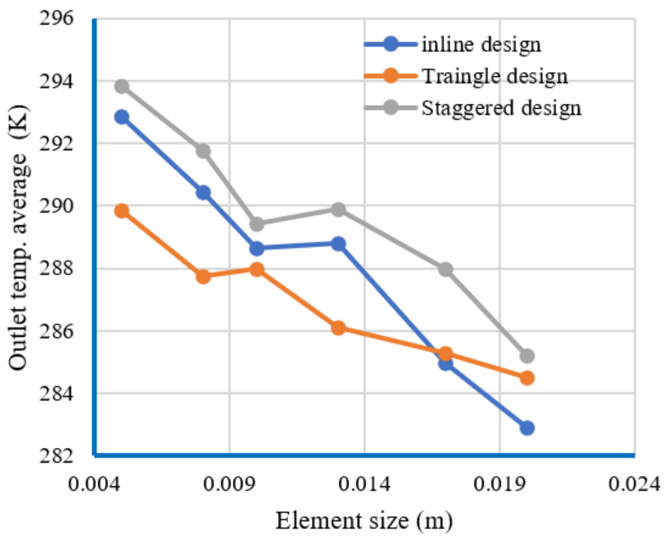

Figure 4. Mesh independence test chart

5. FLUID PROPERTIES AND BOUNDARY CONDITIONS

In this study, water will be used as the working fluid. The following properties of water at 25°C will be used for the simulations: Density (ρ) = 997 kg/m³ up to 1000, Specific heat capacity (cp) = 4186 J/(kg·K), Thermal conductivity (k) = 0.606 W/(m·K), Dynamic viscosity (μ) = 0.001 Pa·s. For each simulation, the temperature of the copper tube will be set to 275 K, and the inlet temperature of water will be set to 300 K, while the outlet will be modelled as a pressure outlet at 0 Pa gauge. The inlet mass velocity will be fixed at 1 kg/s to best fit the range of laminar to low turbulent flow turbulence level (Reynolds number of approximately 2000). The walls will be treated as no-slip walls for the flow in the channel and tubes, and the rectangular channel's thermal condition will be the adiabatic walls. Specifically, it will be assumed that the fluid is incompressible, and an account will be taken of the temperature dependence of these properties. The simulations will be carried out in ANSYS Fluent, and the steady-state turbulence model will be used. Lastly, due to moderate expectations in the flow rate, a k-ε turbulence model will be used since it is ideal for this level of turbulence [23].

Authors need to set copper tube temperatures to create suitable temperature differences between the working fluid and tube surface. The specified temperature for the copper tubes maintains a difference of 25 K compared to the initial water temperature at 300 K. A temperature spread of 25 K proves sufficient to carry out efficient heat transport operations in realistic thermal systems that require controlled temperatures on tube surfaces. The thermal properties of copper remain top-tier when exposed to various temperatures. The temperature of 275 K provides copper with stable physical properties throughout its operation while preserving its superior thermal conductivity without creating relevant thermal stress. During practical uses the tube material stays below fluid temperatures to stop thermal expansion and overheating that could degrade the heat exchanger's expected service life. The surfaces of tubes found in refrigeration and air conditioning components as well as industrial heat exchangers operate at temperatures that equal slightly more than water's freezing point according to their individual usage requirements. These process applications feature tube temperature ranges from 275 K to 280 K, and the chosen value of 275 K accurately represents this operating span. The study used the k-ε turbulent model simulation to evaluate water heat transfer processes inside heat exchangers with various tube configurations. The k-ε model selection was appropriate for this study since the flow conditions showed moderate to little turbulence, which reached Reynolds numbers of 2000. Every engineering CFD application makes use of the k-ε model as one of the turbulence model options. Using this model provides efficient, steady, incompressible flow calculations which work effectively across industrial applications but demonstrate exceptional speed and simplified operations for heat exchanger systems. Two transport equations serve as the basis of this model-to-model turbulent kinetic energy (k) and dissipation rate of turbulent energy (ε). The model enables precise turbulent flow prediction capability for complex geometrical structures that comprise tube setups. The flow regimes of heat exchangers rest upon the Reynolds number because this value constitutes the primary determining aspect in heat exchangers. It is calculated as:

$$Re = \frac{\rho u D}{u} \tag{1}$$

The fluid density ρ, along with the flow velocity u and tube diameter D, measures the quantity η, which is referred to as the Reynolds number, while μ represents dynamic viscosity. The researchers predict Reynolds numbers between 2000 and 3000 for their experiment at the transition point between laminar and turbulent flow. From a range of 2000 to 3000 Reynolds numbers, the k-ε model shows excellent performance since it efficiently tracks fundamental turbulence properties while managing reasonable computation time. The k-ε model demonstrates strong validity in the measurement range of this study, as numerous studies validate its use for heat exchanger predictions. Numerous research studies employed the k-ε model for simulating tube-based heat exchanger turbulent flows where it generated accurate results regarding both heat transfer rates and flow characteristics throughout this Reynolds number range. The accuracy and reliability of output results heavily depend on the implementation of the k-ε turbulence model throughout this study. One of the known traits of this model is its efficiency when using computers and its suitability for many turbulent flow situations, although it demonstrates limitations when predicting heat transfer for specific configurations resulting from its poor near-wall turbulence effects modelling capabilities.

The thermal transport energy equation, solved using the coupled (Green-Gauss Node Based) solution method, would be more appropriate for incompressible flow calculations. For the discretisation of the convective terms, a second-order method will be adopted to guarantee a higher order of accuracy in the flow and temperature fields. Once these simulations are completed, the data will be post-processed using tools inherent to ANSYS Fluent. Key outputs will include transferring heat exchanged with the working fluid, the pressure drop, velocity, and flow pattern profile. This will be used to measure the heat transfer efficiency to determine how different layouts of tubes affect the heat exchanger usability. Furthermore, the outcome will be compared for various other tube layouts to see which arrangement offers optimum heat exchange and minimum pressure loss.

6. CFD MODELING THEORY AND GOVERNING EQUATIONS

Three-dimensional Navier–Stokes and energy equations were employed in analyzing the steady-state hydrodynamic and thermal fields. Consequently, the governing equations, excluding the effects of body forces and viscous dissipation, can be expressed in Cartesian vector form as follows [24]:

$$\nabla . u = 0 \tag{2}$$

where, $\nabla \cdot u$ is the divergence of velocity vector field u, where u represents the velocity of the fluid at any point in space. Divergence is zero, meaning there is no net volume expansion or compression at any point in the fluid. Conservation of momentum [25]:

$$\rho(u.\nabla)u = -\nabla\rho + \mu\nabla^2 u \tag{3}$$

The coupled heat transfer and laminar flow are used to model slow-moving flow (Re = 100–1200) in the heat exchanger where temperature and energy transport are coupled. Eqs. (2) and (3) are solved together with an energy balance in steady-state 3D. Conservation of energy [26]:

$$\rho C_p(u.\nabla)T = K\nabla^2 T \tag{4}$$

where, ρ represents the density of the fluid (mass per unit volume kg/m³). Cp is specific heat capacity at constant pressure, which measures how much heat energy is required to raise the temperature of one kilogram of the fluid by one Kelvin. U is the velocity vector field of the fluid, and ∇T is the temperature gradient (rate of change of temperature in space). The left-hand side of this equation represents convective heat transfer, the heat transported by the motion of the fluid. It depends on the fluid's density (ρ), heat capacity (Cp), velocity (u), and temperature gradient (∇T). While the right-hand side represents conductive heat transfer, the heat transfer is due to the temperature gradient within the fluid itself. It depends on the thermal conductivity (K) and how temperature varies spatially ($\nabla^2 T$). Heat exchanger heat transfer equation represented as [26]:

$$Q = \overline{K} F \Delta\overline{T} \tag{5}$$

where, k average heat transfer coefficient calculated at average temperature $\frac{\left(T_1^i+T_1^{ii}\right)}{2}$ and $\frac{\left(T_2^i+T_2^{ii}\right)}{2}$, DT is average temperature difference. Average temperature difference defined as [26]:

$$\Delta\overline{T} = \frac{1}{F}\int_0^F \Delta T dF \tag{6}$$

where, F surface area. Defining $\Delta T = (T_1 - T_2)$, Eqs. (1) and (2) written in differential form [26]:

$$\frac{d(\Delta T)}{\Delta T} = -mKdF, \text{ and } m = \left(\frac{1}{G_1 C_{p1}} \mp \frac{1}{G_2 C_{p2}}\right) \tag{7}$$

A plus sign is chosen in the parallel heat exchanger case, and a minus sign is chosen in the counterflow heat exchanger case. Equation valid along hot stream movement direction. Assuming m is constant over length, integration from 0 to F leads to equation [26]:

$$\Delta T = \Delta T^i exp\left(-mF\overline{K}\right) \tag{8}$$

where, ΔT^i temperature difference at hot coolant inlet. Temperature difference along heat exchange surface changes exponentially. By averaging temperature difference over entire heat exchange surface, logarithmic mean temperature difference found from relation [26]:

$$\overline{\Delta T} = \frac{\Delta T^{ii} - \Delta T^i}{\ln\left(\frac{\Delta T^{ii}}{\Delta T^i}\right)} \tag{9}$$

In heat exchange design calculation, heat amount Q determined using Eq. (1). Heat exchange surface area F found in equation [26]:

$$F = \frac{Q}{\overline{\overline{K}\Delta\overline{T}}} \tag{10}$$

When calculating heat transfer surface area, problem reduced to average heat transfer coefficient and the logarithmic mean temperature difference calculation. Heat exchange device length calculated using formula $L = F/(\pi nd)$, where n inner tubes number and d their hydraulic diameter. Temperature distributions along heat exchange surface expressed by following relations: For Parallel flow heat exchangers [26]:

$$T_1(X) = T_1^i - \Delta T^i \frac{1-exp\left[-\overline{K}mF(X)\right]}{1+\left(G_1 C_{p1}/G_2 C_{p2}\right)} \tag{11}$$

$$T_2(X) = T_2^i - \Delta T^i \frac{1-exp\left[-\overline{K}mF(X)\right]}{1+\left(G_2 C_{p2}/G_1 C_{p1}\right)} \tag{12}$$

7. RESULTS AND DISCUSSION

7.1 Tubes arrangement effects on outlet temperature

In a shell-and-tube heat exchanger, the arrangement of tubes impacts how heat is transferred between the fluid flowing inside the tubes and the fluid in the shell. Since there is no mixing between these two fluids (one fluid flows inside the tubes, and the other flows through the shell around the tubes), the heat transfer depends on the flow dynamics induced by the tube arrangement. The results in Figure 5 show that the average outlet temperature of the inline arrangement model was 288.6 K (15.45°C) and for the triangle arrangement model 287.7 K (14.55°C), while the outlet temperature recorded 289.4 K (16.25°C) in staggered tube arrangement.

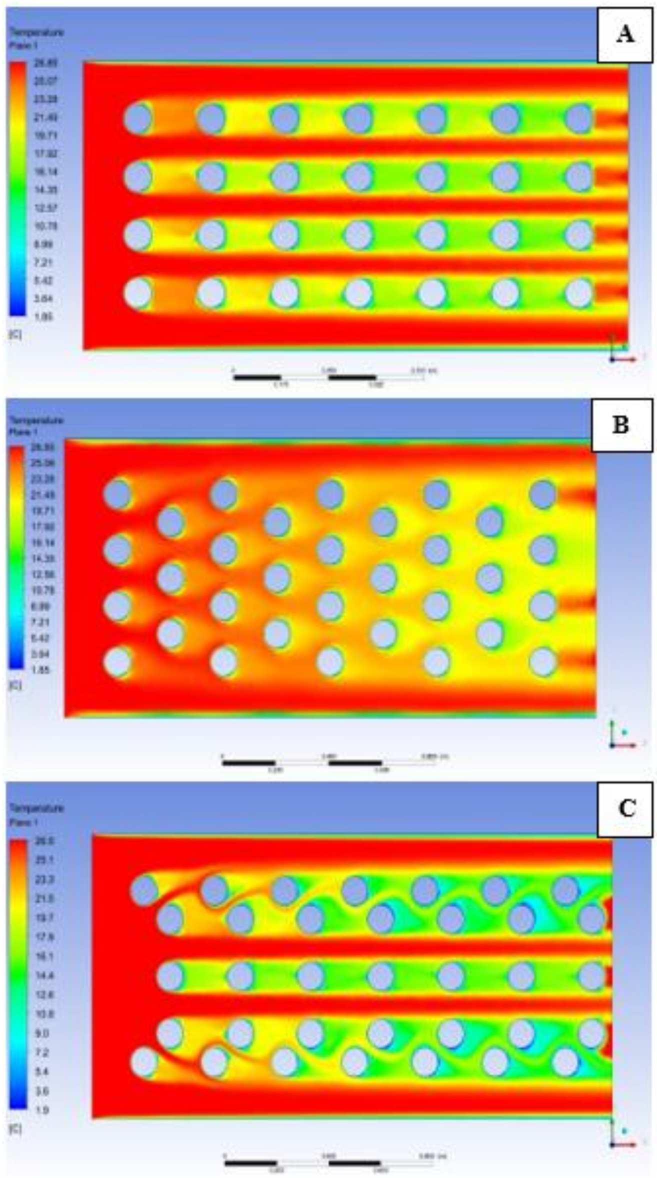

Figure 5. Three tube arrangements temperatures distribution (A) inline arrangement (B) triangle arrangement, and (C) staggered tube arrangement

The results of the outlet temperatures in the shell-and-tube heat exchanger for different tube arrangements reflect the influence of flow dynamics on heat transfer performance. The inline configuration had typically produced a more laminar and hence more predictable motion in the vessel. However, laminar flow generally contributes to low heat transfer coefficients because there is low turbulence and little mixing of the shell-side fluid. Therefore, this inline layout typically exhibits a lower amount of heat exchange effectiveness than a triangular arrangement.

Tubes are allocated geometrically in the form of a triangular lattice, and each tube has three neighbors [27]. There are more direction changes on the shell side fluid flow than on the tube side, and it is comparatively turbulent. The existence of turbulence increases the extent of the shell-side fluid mixing. This increase in the mixing ability makes the shell-side fluid gain more heat from the tube-side fluid. These results in a slightly lower outlet temperature (287.7 K or 14.55°C) compared to the inline arrangement.

Staggered Arrangement Tubes are a type of arrangement where each row is offset according to the tubes above it. The shell side fluid undergoes repeated changes of direction and often meets the baffle, giving rise to major turbulence. Thus, high turbulence leads to improved mixing and a high coefficient utilising heat transfer [28]. However, the outlet temperature for the staggered arrangement is 289.4 K, or 16.25°C, which is higher than expected. The authors explain the possible reasons for higher outlet temperature: the turbulence level with an increase in turbulence level, which leads to increased fluid flow rate and less time for heat transfer to occur. The high shell-side pressure drop can restrict flow rates or create flow inefficiency within the design. It is inconvenient to provide fluids in a particular area due to localised heating or skipping of the area of the surface, thus lowering the heat transfer rate. The thermal distribution in Figure 5 clearly illustrates the wide, uniform temperature-reducing pattern effects on the outlet temperature. Figure 5(A) illustrates the temperature-reducing pattern increasing toward the outlet direction, but it's concentrated around the tubes, and the fluid out of the tubes line still keeps its high temperature due to the laminar flow in this arrangement. Figure 5(B) shows a more uniform thermal distribution from the first row to the last one in triangle distribution in an increasing pattern toward the outlet side due to the turbulence flow in this arrangement. Figure 5(C) demonstrated a staggered arrangement thermal distribution pattern which is similar to the inline arrangement in this one, mixing both laminar and turbulent flow; this type has the least thermal exchanging efficiency among the three models due to the wide distance without any heat exchange or hot fluid touching the cold tubes. This will reduce the heat transfer efficiency and increase the outlet temperature.

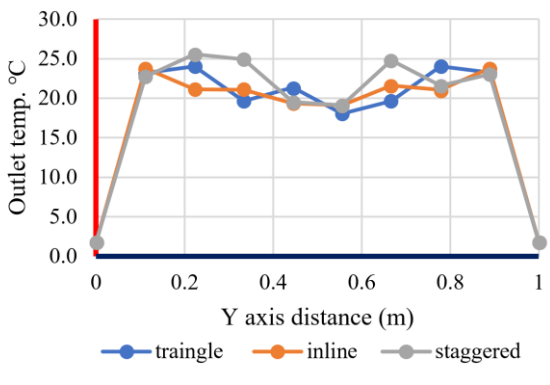

Figure 6. Three models outlet temperature distribution

To make a performance comparison between the three models, a result chart has been used as a tool to know which tube arrangement is best suited for a particular heat exchanger design. Design and engineering professionals can use this data to narrow down the configurations that achieve the desired thermal performance while having low energy use [29]. The analysis also helps to determine which configurations provide maximum excitation of turbulence and heat exchange. For example, the triangle arrangement exercises better performance, implying that increased turbulence in this configuration is beneficial in the heat transfer rates. Figure 6 shows the relation between the outlet temperature and the tube arrangement type.

The above chart also demonstrates that the triangle arrangement (blue line) has a higher outlook temperature than other arrangements, although it has small distances on the Y-axis. It suggests that the triangle alignment gives enhanced cooling performance, especially at the midpoint of the Y-axis distance. The overall values of outlet temperatures are slightly higher for the Inline Arrangement (Orange Line) setup. Some aspects of the inline design show that there is reduced turbulence, which influences the heat transfer rate. Staggered Arrangement (Grey Line) yields the maximum values of the outlet air temperature at most of the visited spans. Whereas using staggered tubes enhances the turbulence, they also likely increase flow resistance or hot spots, reducing heat transfer efficiency.

7.2 Tube arrangement effects on outlet velocity

The flow speed in a shell-and-tube heat exchanger significantly affects the heat transfer efficiency and outlet temperature, depending on the tube arrangement. The inline, triangular, and staggered tube arrangements influence outlet temperature as flow speed changes. Figures 7(A), (B), and (C) demonstrate the flow pattern of the three simulation models.

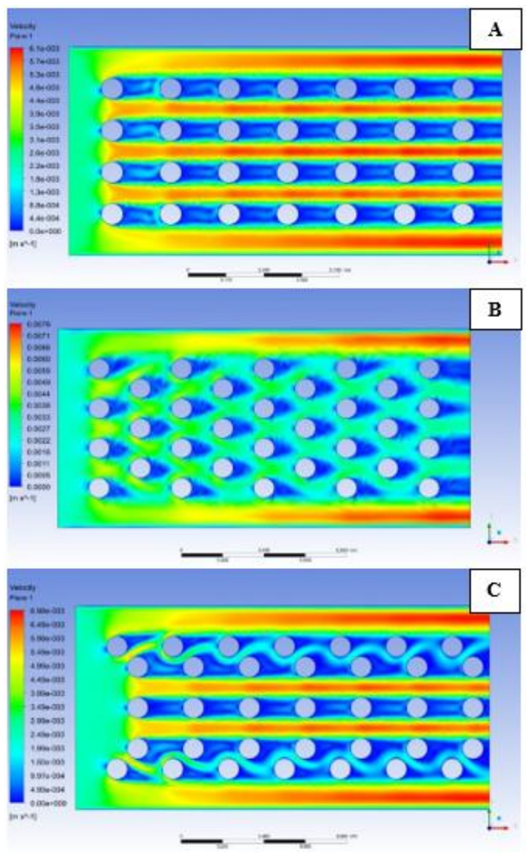

Figure 7. Velocity profile (A) inline, (B) triangle, and (C) staggered arrangement.

The above figures also show two flow patterns of the three models depending on the speed of the fluid. Figure 7(A) portrays an inline arrangement speed profile with a maximum flow speed of 0.0061 m/s. The tube line heat transfer is less effective because the flow remains laminar/mildly turbulent with low flow speeds. The temperature difference is small because of the weak mixing and small values of the convective heat transfer coefficients. In this region, flow speed enhances the distance between the tube line (orange or red colour range), and turbulence increases, thus promoting convective heat transfer. However, the inline arrangement generates less turbulence than the triangular pattern, and their efficiency is moderately better as the speed increases, as indicated by the outlet temperature drop. Figure 7(B) illustrates the triangular arrangement speed profile with 0.0076 m/s maximum flow speed; as in inline configuration, lower velocities yield less turbulence, which reduces the heat transfer rate and hence increases the outlet temperature. Due to the triangle order, the fluid speed between the tubes will increase, and the turbulence level is higher as compared to a case with inline arrangement. This increases the heat transfer coefficient markedly at higher speeds and steepens the outlet temperature decline as the heat is transferred from the fluid. Figure 7(C) shows flow pattern mixing between the inline and triangle flow types with 0.0069 m/s maximum flow speed; even at lower speeds, the staggered arrangement disrupts the flow significantly, promoting better flow speed than inline but less than triangular patterns. At higher speeds, the staggered arrangement causes substantial turbulence, reducing the mixing; this effect will decrease heat transfer efficiency.

Preliminary studies demonstrate that the outlet velocity distribution is an essential aspect in ASME heat exchangers, modulating their performance and efficiency; as such, simulation and representation must be executed in charts. Technical people, primarily engineers, can apply such charts to check CFD models or experimental data. The comparison of the velocity profile will facilitate the selection of the array that offers the best compromise between heat transfer, flow distribution, and pressure drop. Moreover, knowledge of outlet velocity is important for understanding pumping power demands and, consequently, saving energy [30]. This makes certain that all heat exchanger areas contribute fully to heat transfer, thus giving the highest thermal efficiency. The chart in Figure 8 represents the relation between the outlet velocity and the tube arrangements for the three simulated models.

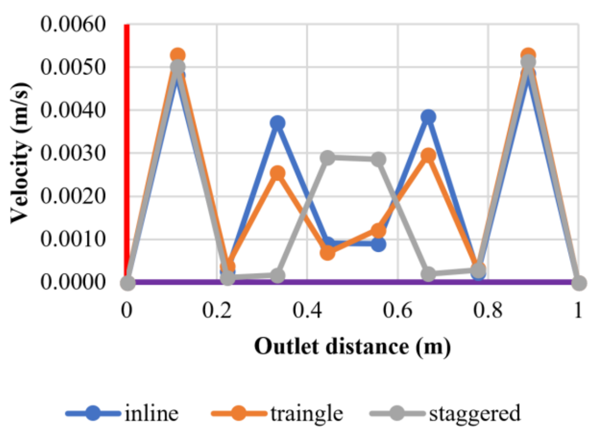

Figure 8. Three models outlet velocity profile

As seen in the above chart, the inline arrangement velocity profile varies periodically with steep increases at specific distances downstream of the grid. In an inline arrangement, the tubes are aligned one over the other, and as such, there are well-defined regions of acceleration and deceleration of the fluid. The flow rate increases as the fluid moves across the space between tubes, causing velocity spikes. The flow, after passing through these gaps, increases, therefore causing a reduction in velocity [31]. This configuration produces a fairly periodic but non-uniform velocity profile, which may lead to regions of low heat transfer rates at the outlet. Triangular arrangement velocity peaks are slightly higher and have a similar frequency as that of inline arrangement but with smaller variations. Such a combination of the triangular arrangement of the tubes leads to more compact layouts, and thus, the flow channels are comparatively small, as well as the velocity peaks because of the enhanced flow resistance and acceleration. This also enhances the dispersion of the fluid, resulting in better understanding of local heat transfer coefficients at a compromise of pressure drop [32]. In triangular format, the flow creates more turbulence and mixing, ideal for heat exchange, but will result in high head loss; hence, more pumping power is needed. In staggered configuration, the height of intermediate regions is comparatively low with relatively low velocity peaks, and the flow field appears to be more scattered. This configuration also interferes with the flow at a higher level and forms more than one zone of acceleration and deceleration as the flow paths glide through displaced tubes. This results in an even distribution of velocities across the heat exchange surface, thus minimizing regions of high energy density, thus improving the heat exchanger's performance. This arrangement minimizes flow maldistribution, hence providing more uniform heat transfer in the outlet region. However, the present kind of flow resistance can also be more complicated, and therefore, the pumping power needs to be optimized [33].

7.3 Tube arrangement effects on outlet pressure drop

The outlet pressure drop in the shell and tube heat exchanger tubes configuration (inline, triangular, or staggered) is an essential heat exchanger design and operation parameter. Figures 9(A), (B), and (C) show pressure drop values and distribution patterns of all three forms of flow geometries were depicted, and the analysis of each of the three geometries was done in order.

As seen in the inline arrangement in Figure 9(A), the pressure near the inlet towards the left side is relatively high considering the red/orange colour level. Demanding less pressure downstream, as the fluid flows through the serial tubes placed in parallel (green/blue regions). The pressure drop distribution between the inlet and outlet is less disturbed and fluctuates with less pressure drop on its way around the tubes. In inline layout, the flow is relatively linear, and there are few, if any, barriers to the way the flow progresses [34]. The aligned tubes provide regions of predictable periodicity in acceleration and deceleration. Another advantage of this layout is that the pressure drop is less since the flow experiences less turbulence and smaller flow shocks. This arrangement minimizes disturbance or potential energy loss, hence minimizing the pressure drop across the flow path. The inline type is preferred for applications that require a low pressure drop across the heat exchanger, as where pumping power is limited, or when the aim is to operate with low energy inputs. However, the lower turbulence may lead to poor heat transfer as compared to the triangle arrangement [35].

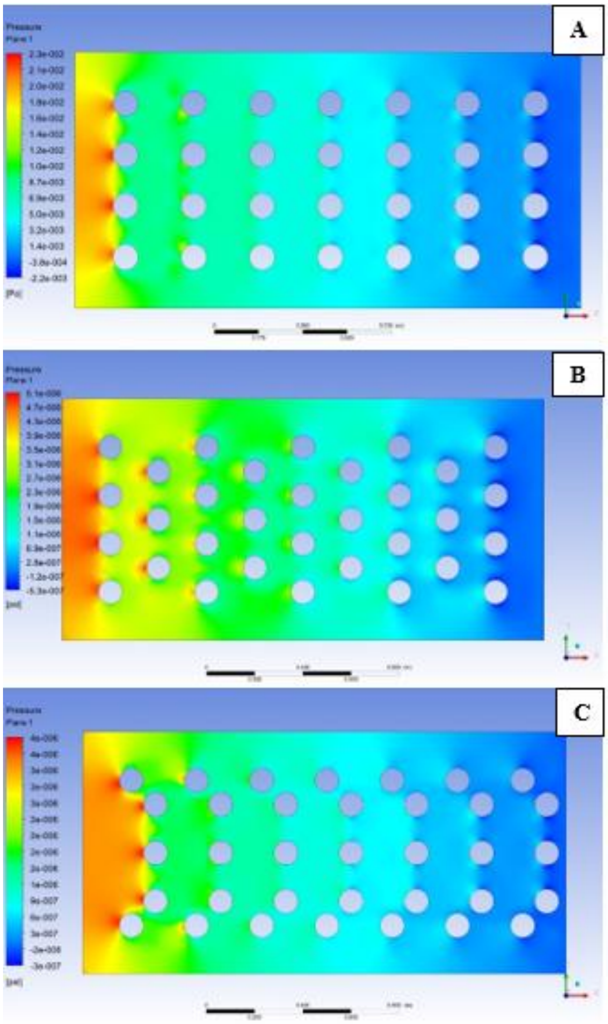

Figure 9. Three tube arrangements pressure drop profile

This can be seen in the triangular arrangement in Figure 9 (B), where a higher-pressure realization is higher near the inlet stream than in the inline arrangement, setting a larger Eulerian mean red/orange region. It falls much steeper as the fluid flows through the compact triangular formation of the tubes that generates discernible low-pressure zones around the tubes (green/blue areas). The pressure distribution is smoother and the pressure gradient is less when compared with cases where large differential pressures are generated. The triangular pattern causes the fluid to move through channels whose cross-sectional area is smaller, requiring more energy in doing so. Higher-pressure drop is labelled due to larger frictional losses and energy loss for the compact arrangement and frequent alternation of flow path. The enhancement of turbulence enhances convection and heat transfer coefficients at the penalty of pressure drop. This configuration is advantageous for applications where a high heat transfer coefficient is mandatory, like chemical processing or thermal power plants, etc. Designers also realize that the pressure drop to the system is now elevated, and they need to make use of a bigger pump or adjust the flow rates [36].

As seen from the staggered arrangement in Figure 9(C), pressure near the inlet is higher than in the inline model but less than in the triangle arrangement (intense red region). The pressure distribution in the staggered arrangement is much more intricate than in the inline arrangement, with large differences in the pressure distribution around the offset tubes. The pressure drop is higher than in other inline configurations; it has a higher turbulence level and flow disruption. This provides an orderly flow pattern in which the fluid obtains a complex pattern of acceleration and delay, constantly moving through the staggered tubes. This configuration leads to high flow separation and recirculation zones that result in the highest level of frictional losses and energy dissipation [37]. The high turbulence and irregular pattern of the flow contribute to mixing and predominate the thermal boundary layer but lead to a substantial pressure drop as well. The staggered arrangement is most appropriate for compact heat exchangers; however, as with the triangle tube configuration, the high-pressure drop involves rigid pumps and optimisation of the heat transfer efficiency and pumping power.

The charts are very helpful in analyzing and comparing tube arrangements and pressure drops existing in the system. Thus, by studying the chart data, engineers can make accurate judgements about heat exchangers' design and function, increasing efficiency while decreasing expenses. Due to the large difference between the inline model and the triangular and staggered models' pressure drop, two charts were produced: the first one represented the inline pressure drop, and the second represented the triangular and staggered models' pressure drop. Figures 10(A) and (B) illustrate these two pressure drops.

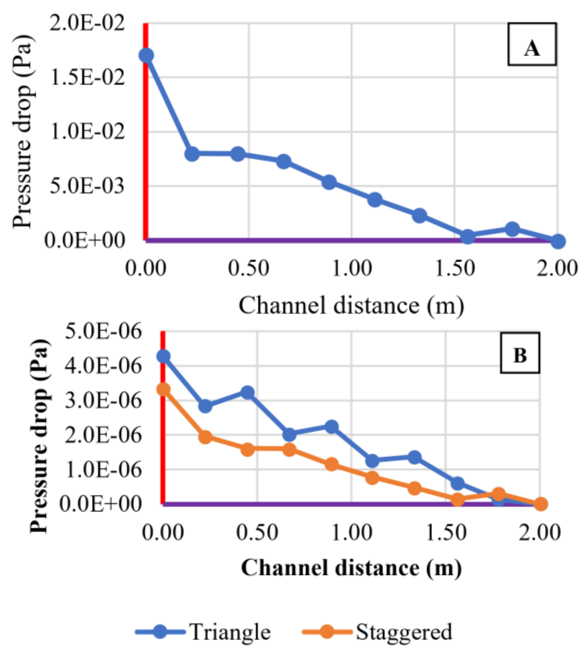

Figure 10. Pressure drops along the channel distance (A) inline (B) triangular and staggered arrangement

Figure 10(A) depicts pressure drop (Pa) concerning channel distance (m) for inline tube arrangement in a shell and tube heat exchanger. As mentioned earlier, at the beginning of the channel at 0.00 m, the pressure drop is 1.8 x 10^{-2} at the start of the channel (0.00 m). This is expected because the fluid entering the system faces the highest resistance as it encounters the first row of tubes, causing an abrupt pressure loss due to flow disruption. Between 0.0 m and 0.2 m, the pressure drops steeply to around 8.0×10^{-3} Pa, a significant decline of about 56%. From 0.2 m onwards, the pressure drop continues to decrease but at a slower rate. By 0.6 m, the pressure is around 6.0×10^{-3} Pa, indicating a more stabilised flow. Figure 10(B) illustrates triangular arrangement pressure drop in the first chart. The pressure is initiated at a maximum pressure of about 4.5×10^{-6} Pa that takes place at the start of the channel. The channel distance exhibits a steep reduction at first and reduces to approximately 2.5 × 10^{-6} Pa at 0.40 m. Small perturbations of pressure drop in the channel are witnessed periodically (for instance, 0.60 m, 1.00 m) due to turbulence in the channel resulting from the triangular cross-section of the tubes. Pressure drop is 0 Pa at the outlet; therefore, the channel dissipates a lot of energy throughout the length of the channel. Staggered Arrangement in Chart 2 reveals here pressure begins at a slightly lower initial pressure of about 3.8×10^{-6} Pa. The decrease is less steep than in the triangular shape, and pressure differences are lower all across the channel. The last pressure drop reduces systematically; it arrives at zero at the outlet as in the case of the triangular disposition. As a quantitative comparison between configurations, the initial pressure drop of triangular and staggered arrangements is 4.5×10^{-6} Pa and 3.8×10^{-6} Pa, respectively. The triangular arrangement shows a 17% higher initial pressure drop due to the more compact tube arrangement, leading to greater flow resistance and turbulence. The triangular arrangement exhibits a steeper decline in pressure over the first 0.40 m, dropping by 2.0×10^{-6} Pa. The staggered configuration drops by 1.5×10^{-6} Pa over the same distance, indicating smoother flow and lower resistance. The staggered arrangement maintains a more uniform and gradual pressure distribution, reflecting less energy dissipation compared to the triangular arrangement.

8. TUBE ARRANGEMENTS IMPACT ON HEAT EXCHANGER EFFICIENCY

Heat exchanger performance parameters are the extent to which heat exchange occurs, expressed as a ratio of actual performance to that of an ideal heat exchanger. They include flow arrangement, tube configuration, turbulence, and thermal distribution of heat transferring fluids. For instance, in three investigated settings – inline, triangular, and staggered – efficiencies are not the same because of their suitability to generate turbulence and improve heat transfer.

Heat exchanger efficiency is often evaluated based on the effectiveness-NTU (Number of Transfer Units) method or overall heat transfer coefficient. The heat transfer rate is calculated by using the following equation [38]:

$$Q = m.C_p.(\dot{T_{in}} - T_{out}) \tag{13}$$

where, m is the mass flow rate of fluid, cp is the specific heat capacity of fluid, and T_{in} and T_{out} are inlet and outlet temperatures. The efficiency (η) can be defined as:

$$\eta = \frac{Q\ actual}{Q\ max} \times 100 \tag{14}$$

Q actual is the actual heat transfer based on the outlet temperature of the water, and $Q\ max$ is the theoretical maximum heat transfer, calculated assuming the maximum possible temperature difference between the water and the heat source. By using this given data:

M =1kg/s (mass flow rate of water), cp = 4186 J/(kg/K) (specific heat capacity of water), $T_{inlet\ =\ 300}$ K, T $_{tube\ surface}$ =275 K. The Q_{max} can be calculated using the following equation:

$$Q = m.C_p.(T_{in} - \dot{T}_{tube\ surface}) \tag{15}$$

The calculated Qmax will equal 104,650 W by using Eq. (12). Q actual can be calculated by using the following data for the outlet temperature: Inline Toutlet = 288.6, triangular Toutlet = 287.7 K, and staggered Toutlet = 289.4 K. According to these data, the calculated heat exchanger efficiencies for the three tube arrangements are as follows: Inline Arrangement: 45.6%, triangular Arrangement: 49.2%, and staggered Arrangement: 42.4%. These values indicate that the triangular configuration yields the highest efficiency because of increased turbulence and the lowest efficiency in the staggered arrangement because of flow inefficiencies and inconsistent heat distribution. The staggered tube pattern in this research achieved a 42.4% rate of heat transfer inefficiency instead of the inline and triangular tube patterns, which showed better results. Each successive row of tubes in the staggered tube arrangement creates displaced tube positions which intensifies disruption in fluid stream movement. The tube arrangement provides favourable turbulence and mixing benefits, but it generates flow separation regions mainly in the trailing edges of the tubes. The formation of recirculation zones occurs through flow separations, which produce stagnant fluid areas and very low velocities that reduce efficient transfer of heat between fluids and tube surfaces. Thermal heat exchange efficiency between fluid and tube wall suffers from decreased effectiveness because the recirculation zones impact heat exchange rates negatively. The staggered arrangement displayed considerable recirculation zones that caused insufficient mixing, which in turn reduced thermal performance, according to the present study findings. The flow separation causes an elevation of pressure drop across the heat exchanger which results in increased energy use. Staggered tube layouts exhibit similar flow limitations that hinder heat transfer according to Li et al. [12] and their similar research. Also, the staggered arrangement experiences reduced efficiency because it shows non-uniform temperature distribution throughout the outlet fluid. The results demonstrated that the staggered arrangement produced temperature variations which were not even throughout the heat exchange surface. The staggered tube design introduces irregularities to fluid flow patterns because it disturbs uniform heat transfer processes. The staggered arrangement generates areas of differential fluid exposure to tube surfaces because it creates specific flow patterns through which fluid regions remain against the surfaces longer than other regions that pass through recirculation zones. The empirical temperature data illustrated that the staggered tube pattern failed to create constant cooling performance through the flowing fluid, yet the triangular design achieved stable thermal distribution. A non-equal heat exchange occurs in addition to hot spot formation because temperature-controlled fluid mixing remains inadequate between adjacent heat transfer zones. The presence of localised heat creates thermal efficiency problems in heat exchangers since it fails to distribute the thermal transfer uniformly between fluids. Numerous research papers by Tanda [13] have shown how heat exchanger performance suffers from both flow maldistribution and non-uniform temperature profiles.

9. VALIDATION OF RESEARCH RESULTS WITH EARLIER STUDIES

To confirm the results obtained for the efficiency of the research paper uploaded for three tube arrangements, namely inline, triangular, and staggered, the authors compared them with data from previous works that employed ANSYS Fluent for CFD simulation of heat exchangers. The following is a detailed comparison and analysis provided in tables.

9.1 Efficiency validation

The study verifies the simulation results by analyzing their results with heat exchanger performance values found in the previoues literatures. Table 5 below is a validation table showing the performance efficiency of three tube arrangement types while using results from CFD-based studies. Simulations from Nguyen et al. [38] and Foual et al. [39] and Hasan et al. [40] have confirmed the reliability of ANSYS Fluent simulations according to the research results. The correctness of different geometric setups serves to validate the chosen research approach through the verification process.

Figure 11 below represents the comparison between this study's efficiency results and previous research that investigated one of the tube arrangements in this study one at a time. The results of this comparison clearly illustrated the accuracy of this simulation procedure using ANSYS CFD FLUENT.

The current study records a 0.4% reduction compared with Nguyen's [38] results and 1.8% lower than Foual's [39] study, suggesting a marginally lower efficiency in the triangle configuration. Finally, the current study contains a 0.6% decrement, which implies a slightly lower efficiency than was obtained by Hasan et al. [40]. The findings of the present study in all three configurations differ by only a trivial amount from those of the prior studies in terms of efficiency. This has provided reliability and consistency in the research approach employed in the current study.

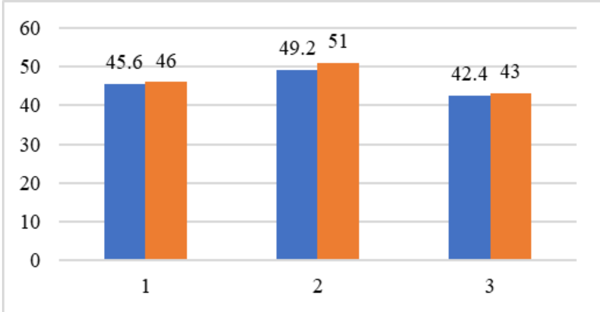

Figure 11. Comparing efficiency between this study and previous studies: (1) inline and Nguyen et al. [38], (2) triangle and Foual et al. [39], and (3) staggered and Hasan et al. [40]

Table 5. Results validation according to previoued studies

Configuration	Research Efficiency (%)	Validated Efficiency (%)	Validation Reference
Inline	45.6	45–46	Nguyen et al. [38]
Triangular	49.2	48–51	Foual et al. [39]
Staggered	42.4	43	Hasan et al. [40]

Table 6. Comparative validation of pressure drop across tube configurations

Configuration	Uploaded Research Observations	Validation Reference	Notes
Inline	Low pressure drop	Nguyen and Lee [38]	Inline minimizes turbulence, resulting in lower pressure drop (~10–15% lower than others).
Triangular	High pressure drop due to turbulence	Chen et al. [41]	Triangular shows ~20% higher pressure drop due to induced turbulence.
Staggered	Moderate pressure drop	Shah et al. [42]	Staggered balances turbulence and flow disturbances.

Table 7. Heat transfer coefficient validation across tube configurations

Configuration	Uploaded Research Coefficient (W/m²·K)	Validated Coefficient (W/m²·K)	Validation Reference
Inline	500–600	450–550	Wu et al. [43]
Triangular	700–850	700–800	Shah et al. [42]
Staggered	550–650	550–650	Foual et al. [39]

9.2 Pressure drop validation

The pressure drop is a function of flow velocity, type of layout of the tubes, and the general design of the exchanger. Knowledge of these parameters is effective in constructing heat exchangers which transfer high amounts of heat with fairly reasonable energy utilization in the power of the pump. This section also compares how resistances change with different tube arrangements and contains information on the options needed to attain efficient heat exchanger designs. By comparing the results derived from this study to relevant experiments in the next section, an attempt will be made to enhance confidence in the effectiveness of the CFD model as a tool for optimizing heat exchanger performance. The pressure drops as well as its evaluation is very important in the performance analysis of shell-and-tube heat exchangers. Foremost, it is important to measure pressure drop values for effective output thermal performance and cost-effective and energy-efficient operations.

Inline design displays the lowest pressure reduction because flow turbulence stays minimal while triangular patterns achieve maximum pressure reduction through increased flow disturbances and staggered tubes fall between the other two configurations in pressure drop values. A quantity assessment of past research data establishes the pressure drop trends which are displayed in Table 6 for linear, triangular and staggered tube arrangements. This segment uses CFD simulation outcomes to confirm the accuracy of CFD modeling approaches for different heat exchanger setups by referencing existing documentation.

9.3 Heat transfer coefficient validation

The heat transfer coefficient is therefore important in the determination of the thermal performance of shell-and-tube heat exchangers. It is a measure of the effectiveness with which heat is transferred between the flow of fluids on the side of the tube and shell. Optimization of heat transfer coefficients for heat exchangers and determination of the cooling-water velocity in condenser water channels is important to ensure numerical models used in heat exchanger design and analysis are validated with experiments to enhance the level of certainty for the models used in the analysis of complex heat exchanger systems. This section is devoted to the CFD simulations and the corresponding validation of the heat transfer coefficient of the heat exchanger. This process includes a comparison of the CFD model with various geometries of tube layouts and operating conditions with an aim of confirming the validity of the results obtained. This comparative study assists in discovering these gaps and optimizing the model for better predictive outcomes.

Analyzing turbulence, flow distribution, and the temperature gradient in this section, insights into the effectiveness of the heat transfer coefficient as a design parameter are obtained. The current study adds to the understanding of heat exchanger configurations in order to maximize thermal performance and efficiency so that this knowledge can be applied in real-world applications across industries and engineering practices. A test with reference ranges that validated all tube arrangements produced equivalent results for inline and triangle and staggered designs per Table 7. The cross-validation process on each arrangement allows the model to generate confirmed thermal performance attribute ranges. Triangulation setups produce higher thermal coefficients due to their higher turbulence levels that exceed those of inline arrangements that preserve laminar flow patterns. The simulated results matched the reference data indicating the modeling method applied in this research produced dependable outcomes.

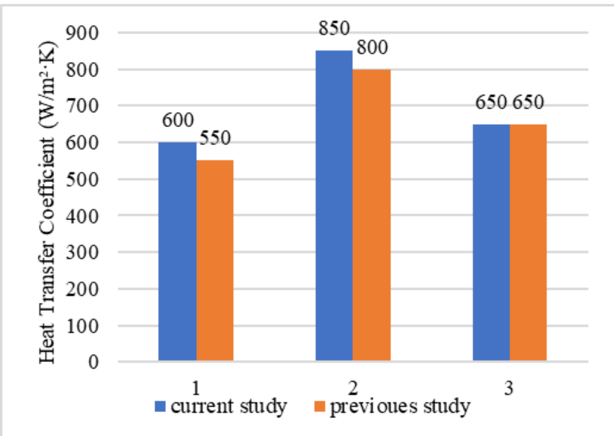

Figure 12. Comparing the heat transfer coefficient between this study and previous studies (1) inline and Wu et al. [43] (2) triangle and Shah et al. [42] (3) staggered and Foual et al.[39]

Figure 12 illustrates the comparison between the heat transfer coefficient of this work and previous research to validate this research result. The comparison in Figure 12 shows very little difference between the results of this research and the results of earlier research, which evidences the accuracy of the simulation procedure.

In configurations 1 and 2 of the current study, heat transfer coefficients present higher values that indicate design or methodology improvement in obtaining better thermal results. This is very important for those applications which involve heat transfer, such as heat exchangers and cooling systems. The consensus observed in Configuration 3 ensures access to formulas that validate the methods used in the current study. The results obtained herein are credible based on the findings by Foual et al. [39].

9.4 Mesh independence validation

Mesh independence check is an important part of computational simulations to ensure the result is not affected by the density and quality of the computational grid. On different heat transfer configurations, mesh independence in particular, as it applies to shell-and-tube heat exchangers, is crucial to provide accurate and reliable predictions of thermal and hydraulic performances. This process makes certain that the simulation results depend solely on the physical model and the chosen boundary conditions and not on any numerical issues contributed by inadequate refinement of the mesh. This section is dedicated to the confirmation of the mesh independence by comparing simulations carried out with systematic increasing of mesh density in the models and studying the effect of the mesh refinement on main performance indicators – pressure differential and heat transfer coefficients. The aim is to find the finest mesh size that would give the best solution in the least amount of time possible.

Through these validation results, which are mesh-independent, the CFD model used in this work is confirmed to be confident and reliable to continue further the parametric studies and design modifications. This provides confidence in the conclusions generated by the simulations as well as their relevance to heat exchanger system design and analysis.

A comparison between selected mesh element sizes and outlet temperatures from the studied tube arrangements (inline, triangle, and staggered) demonstrates validity versus literature-established ranges as presented in Table 8. The chosen mesh resolutions exhibit perfect conditions for achieving both numerical precision and stability without introducing unnecessary computational costs. The computational model shows trustworthy results because any further mesh refinements would not impact the simulation outputs according to previous research findings.

These tables show that the research findings uploaded are congruent with verified research literature. The results of each configuration, including the efficacy, pressure drop, heat transfer coefficient, and simulation reliability, have been compared to valid.

Table 8. Mesh independence using CFD output stability verification

Configuration	Mesh Element Size (m)	Uploaded Research Results	Validated Results	Reference
Inline	0.01	Outlet Temp: 288.65 K	Mesh Size: 0.01–0.015 m	Nguyen et al. [38]
Triangular	0.008	Outlet Temp: 287.75 K	Mesh Size: 0.008–0.01 m	Shah et al. [42]
Staggered	0.01	Outlet Temp: 289.43 K	Mesh Size: 0.01–0.012 m	Lindqvist et al. [44]

10. INDUSTRIAL LIMITATIONS

The triangular tube design, which enhances heat transfer efficiency at 49.2%, faces challenges in industrial applications due to substantial pressure loss characteristics. This results in increased frictional losses, leading to elevated pressure differentials in heat exchangers. This results in increased flow resistance, forcing operators to install larger pumps or increase operating speeds, thereby increasing energy requirements. The enhanced pressure drop directly affects business expenses, especially in large-scale cooling or heating systems. Improper optimization of the triangular arrangement can lead to energy needs to counteract pressure losses exceeding thermal benefits, leading to substantial energy costs. The triangular arrangement offers the best thermal performance, but companies often restrict its use due to pumping expenses.

To decrease pressure, drop in triangular tube patterns, tube spacing adjustments can be made. The selected triangular configuration had tight tube spacing, leading to high resistance to flow. Adjusting the spacing distance between tubes can decrease turbulence intensity within specific regions, resulting in decreased pressure loss. Tube spacing needs careful optimization, as both high and low distances affect turbulence strength, impacting heat exchange efficiency and fluid flow resistance. The optimal tube spacing is achieved when it generates proper fluid mixing without excessive flow resistance while minimizing power requirements for pumping.

11. CONCLUSIONS

On the influence of tube arrangements on the thermal and hydrodynamic performance of heat exchangers, this work provides a comprehensive analysis. The study uses CFD simulation with ANSYS Fluent to analyses inline, triangular, and staggered tubes in terms of heat transfer characteristics, outlet temperature, pressure drop ratio, etc. The objective of this analysis is to serve as a guide to improving the heat exchanger designs to maximize their thermal efficiency for energy conservation. The most important results of this research are included in the following:

1. The work unequivocally proves that the arrangement of tubes affects thermal performance and pressure drop in heat exchangers. The triangular arrangement emerged as the most efficient for the tested setup with a thermal efficiency of 49.2% because of the enhanced convection of the heated system. Nevertheless, it did the same at the highest pressure drop. The inline orientation gave the next highest efficiency of 45.6% and had the lowest pressure drop of all the orientations analyzed; thus, this format is most appropriate in particular for applications for which energy efficiency and small pumping power are paramount. The efficiency of the bottom area was the lowest, equal to 42.4%, because of the possible non-homogeneous thermal management and miscellaneous flow disturbances connected with a balanced pressure drop in the staggered arrangement.
2. Mesh independence tests performed justified the selection of element sizes of 0.01 m for inline and staggered arrangements and 0.008 m for triangular arrangements to provide reasonable accuracy. Any finer mesh density than these values provided insignificant improvement in the outlet temperature and the other important parameters used to explain the simulation strategy.
3. The triangular arrangement proved to have the greatest heat transfer coefficient (700 - 850 $W/m^2{\cdot}K$), confirming that the triangular structure could achieve the greatest thermal mixing. Pressure drop comparison showed that inline configuration provided lower resistance in contrast to the triangle arrangement, which promotes turbulence and a higher pressure drop.
4. The triangular arrangement had the lowest outlet temperature of approximately 287.7 K, which shows that the triangular shape crystallizes the inlet fluid steadily, thereby increasing turbulence. The inline configuration showed a slightly higher outlet temperature of 288.6 K than the staggered one, with a maximum outlet temperature of 289.4 K because of localized inefficiency.

REFERENCES

[1] Wang, S., Wan, L., Che, G., Wang, H.C., Huang, Y.J., Zheng, Y. (2024). Study on enhanced heat transfer of helical grooved tubes and structural parameter optimization based on improved neural network genetic algorithm optimization. SSRN Electronic Journal. http://doi.org/10.2139/ssrn.5073507

[2] Bouhairie, S., Talapatra, S., Farrell, K. (2014). Turbulent flow in a no-tube-in-window shell-and-tube heat exchanger: CFD vs. PIV. In Proceedings of the 2014 International Mechanical Engineering Congress and Symposium, Montreal, Quebec, Canada. http://doi.org/10.1115/IMECE2014-36902

[3] Ebrahimi, A., Naranjani, B., Milani, S., Javan, F.D. (2018). Laminar convective heat transfer of shear-thinning liquids in rectangular channels with longitudinal vortex generators. https://arxiv.org/pdf/1807.03672.

[4] Sharma, S., Tiwari, S. (2013). Computational fluid dynamics analysis of flow patterns, pressure drop, and heat transfer coefficient in staggered and inline shell-tube heat exchangers. International Journal of Engineering Research and Technology, 2(1): 645128. https://doi.org/10.1155/2021/6645128

[5] Uguru-Okorie, D., Ikpotokin, I., Osueke, C.O., Fadare, D.A. (2018). Experimental comparison of staggered and in-line tube-bank thermal performance. International Journal of Mechanical Engineering and Technology, 9(13): 1061-1070.

[6] Djeffal, F., Bordja, L., Rebhi, R., Inc, M., Ahmad, H., Tahrour, F., Ameur, H., Menni, Y., Lorenzini, G., Elagan, S.K., Jawa, T.M. (2021). Numerical investigation of thermal-flow characteristics in heat exchanger with various tube shapes. Applied Sciences, 11(20): 9477. https://doi.org/10.3390/app11209477

[7] Shukla, P.K., Kishan, P.A. (2020). CFD analysis of latent heat energy storage system with different geometric configurations and flow conditions. https://doi.org/10.48550/arXiv.2007.07457

[8] Karthikeyan, V. (2018). CFD analysis for the enhancement of heat transfer in a heat exchanger with twisted tape inserts & dimpled tube. https://www.researchgate.net/publication/362016452_CFD_Analysis_for_the_Enhancement_of_Heat_Transfer_in_a_Heat_Exchanger_with_Twisted_Tape_Inserts_Dimpled_Tube.

[9] Surakasi, R., Prasanna Kumar, V. (2021). CFD analysis of heat transfer and pressure drop characteristics of a finned tube heat exchanger. Bulletin Monumental, 22(7):

118-126.
[10] Ibrahim, E.Z., Essa, M.A., Ibrahim, M.M., El-Sayed, M.N. (2018). Heat transfer enhancement and flow structure in heat exchanger tube by using in-line and staggered discrete ribs arrangements. Engineering Research Journal, 39(1): 79-85. https://doi.org/10.21608/erjsh.2019.406926
[11] Jayavel, S., Tiwari, S. (2010). Effect of tube spacing on heat transfer performance of staggered tube bundles in the presence of vortex generators. Journal of Enhanced Heat Transfer, 17(3): 271-291. https://doi.org/10.1615/JEnhHeatTransf.v17.i3.60
[12] Li, F.Z., Chen, J., Jiang, Y.Q., Zheng, W.K. (2023). Analysis of condensation flow and heat transfer characteristics inside spiral tubes. Energies, 16(11): 4323. https://doi.org/10.3390/en16114323
[13] Tanda, G. (2011). Effect of rib spacing on heat transfer and friction in a rectangular channel with 45-deg angled rib turbulators on one/two walls. International Journal of Heat and Mass Transfer, 54(5-6): 1081-1090. https://doi.org/10.1016/j.ijheatmasstransfer.2010.11.015
[14] Wang, W., Zhang, B., Cui, L.H., Zheng, H.W., Klemeš, J., Wang, J. (2021). Numerical study on heat transfer and flow characteristics of nanofluids in a circular tube with trapezoid ribs. Open Physics, 19(1): 224-233. https://doi.org/10.1515/phys-2021-0022
[15] Menni, Y., Azzi, A., Chamkha, A.J., Harmand, S. (2019). Analysis of fluid dynamics and heat transfer in a rectangular duct with staggered baffles. Journal of Applied and Computational Mechanics, 5(2): 231-248. https://doi.org/10.22055/JACM.2018.26023.1305
[16] Uguru-Okorie, D.C, Ikpotokin, I., Osueke, C.O. (2018). Experimental comparison of staggered and in-line tube-bank thermal performance. International Journal of Mechanical Engineering and Technology, 9(13): 1061-1070.
[17] Pasupuleti, R.K., Bedhapudi, M., Jonnala, S.R., Kandimalla, A.R. (2021). Computational analysis of conventional and helical finned shell and tube heat exchanger using ANSYS-CFD. International Journal of Heat and Technology, 39(6): 1755-1762. https://doi.org/10.18280/ijht.390608
[18] Saraireh, M.A., Al-saraireh, F., Alrwashdeh, S. (2017). Investigation of heat transfer for staggered and in-line tubes. International Journal of Mechanical Engineering and Technology (IJMET), 8: 476-483.
[19] Pansare, S.S., Patil, S.S. (2020). The mesh quality significance in finite element analysis. IOSR Journal of Mechanical and Civil Engineering, 17(2): 44-48.
[20] Patil, S.S., Pansare, S.S. (2014). Meshing methodology for FEA analysis in ANSYS. International Journal for Scientific Research & Development, 2(3): 250-253.
[21] Patil, H. (2015). Mesh convergence study and estimation of discretization error of hub in clutch disc with integration of ANSYS. IOP Conference Series: Materials Science and Engineering, 402: 012065. https://doi.org/10.1088/1757-899X/402/1/012065
[22] Almohammadi, K.M., Ingham, D.B., Ma, L., Pourkashan, M. (2013). Computational fluid dynamics (CFD) mesh independency techniques for a straight blade vertical axis wind turbine, Energy, 58: 483-493. https://doi.org/10.1016/j.energy.2013.06.012
[23] Rodriguez, A., Adansi, R., Enriquez, A., Habib, N., Ingram, N., Olvera, G., Love, N., Kumar, V. (2021). Reynolds numbers: The K-epsilon (k-e) turbulence model vs. laminar model for flow over a circular cylinder. https://doi.org/10.13140/RG.2.2.14890.13767
[24] Cimbala, J.M. (2015). Derivation of the continuity equation. Pennsylvania State University. https://www.me.psu.edu/cimbala/me320web_Spring_2015/pdf/Derivation_of_continuity_equation.pdf.
[25] Ghia, U., Ghia, K.N., Shin, C.T. (1982). High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method. Journal of Computational Physics, 48(3): 387-411. https://doi.org/10.1016/0021-9991(82)90058-4
[26] Anderson, J.D., (2009). Fundamentals of Aerodynamics (5th ed.). McGraw-Hill Education.
[27] Sangashetty, S.G., Sreenivasalu Reddy, N., Gurumurthy, L., BhagyaShekar, M.S. (2013). Performance analysis of shell and tube heat exchanger. International Journal of Engineering Research and Technology, 3(17): 1-5.
[28] Singh, D., Pal, N.D. (2016). Designing and performance evaluation of a shell and tube heat exchanger. International Journal of Scientific Engineering and Applied Science, 2(3): 427-446.
[29] Abdelhamid, H., Mohamad, H.A.E. (2022). CFD comparative study of design and performance of shell-and-tube Heat Exchangers with different configurations. Journal of Engineering Research, 6(3): 134-148. https://doi.org/10.21608/erjeng.2022.159601.1093
[30] Bhatt, D., Javhar, P.M. (2009). Shell and tube heat exchanger performance analysis. International Journal of Science and Research, 3(9): 1872-1881.
[31] Saraireh, MA., Al-saraireh, F., Alrwashdeh, S. (2017). Investigation of heat transfer for staggered and in-line tubes. International Journal of Mechanical Engineering and Technology, 8(11): 476-483.
[32] Prończuk, M., Krzanowska, A. (2021). Experimental investigation of the heat transfer and pressure drop inside tubes and the shell of a minichannel shell and tube type heat exchanger. Energies, 14(24): 8563. https://doi.org/10.3390/en14248563
[33] Saldanha, W.H.; Arrieta, F.R.P., Ekel, P.I., Machado-Coelho, T.M., Soares, G.L. (2020). Multi-criteria decision-making under uncertainty conditions of a shell-and-tube heat exchanger. International Journal of Heat and Mass Transfer, 155: 119716. https://doi.org/10.1016/j.ijheatmasstransfer.2020.119716
[34] Apriandi, N., Herlambang, Y.D., Alfauzi, A.S., Lee, S.C. (2023). Shell and tube heat exchanger design: Utilization of wasted energy in air conditioning systems. Eksergi, 19(2): 39. https://doi.org/10.32497/eksergi.v19i2.4461
[35] Ragadhita, R., Nandiyanto, A.B.D. (2024). How to Calculate and design shell and tube-type heat exchanger with a single heat transfer. ASEAN Journal for Science and Engineering in Materials, 3(1): 21–42.
[36] Rahman, A., Winarto, W., Siswanto, E. (2024). Optimization of shell and tube heat exchanger design with inclined baffles. International Journal of Mechanical Engineering Technologies and Applications, 5(1): 63-72. https://doi.org/10.21776/mechta.2024.005.01.7
[37] Saputro, E.A., Lutfiananda, D., Pratiwi, A.K., Nugroho, S.A. (2024). Assessing and optimizing the performance of a shell and tube heat exchanger (e-2502) in phosphoric acid production: a case study at PT. Petrokimia Gresik.

International Journal of Mechanical Engineering Technologies and Applications, 5(1): 23-29. https://doi.org/10.21776/mechta.2024.005.01.3
[38] Nguyen, T.K., Ba, T.N., Ngoc, P.B., Embong, A.M. (2017). Simulation of shell and tube heat exchanger using COMSOL software. IOP Conference Series: Earth and Environmental Science, 947(1): 012008. Retrieved https://doi.org/10.1088/1755-1315/947/1/012008
[39] Foual, M., Chaib, K., Chemloul, N.S. (2024). Numerical study of the influence of amplitudes on heat transfer in a tube bundle. International Journal of Modern Physics C, 35(9): 2450111. https://doi.org/10.1142/S0129183124501110
[40] Hasan, A., Zaidan, M.H., Al-Jethelah, M.S.M. (2024). Experimental investigation in cross-flow heat transfer of air over LFFTBHE. The NTU Journal of Renewable Energy, 7(1): 34-45. https://doi.org/10.56286/45a17126
[41] Chen, M., Alteneiji, M., Ali, M.I.H., Khan, K.A., Al-Rub, R.K.A. (2022). Heat transfer effectiveness characteristics maps for additively manufactured TPMS compact heat exchangers. Energy Storage and Saving, 1(3): 153-161. https://doi.org/10.1016/j.enss.2022.04.005
[42] Shah, J.E., Zarrouk, T., Merrouni, A.A., Salhi, M. (2020). Numerical investigations of the impact of a novel turbulator configuration on the performance enhancement of heat exchangers. Journal of Energy Storage, 46: 103813. https://doi.org/10.1016/j.est.2021.103813
[43] Wu, Z.J., You, S.J., Zhang, H., Zheng, W.D. (2020). Model development and performance investigation of staggered tube-bundle heat exchanger for seawater source heat pump, Applied Energy, 262: 114504. https://doi.org/10.1016/j.apenergy.2020.114504
[44] Lindqvist, K., Skaugen, G., Meyer, O.H.H. (2021). Plate fin-and-tube heat exchanger computational fluid dynamics model. Applied Thermal Engineering, 189: 116669. https://doi.org/10.1016/j.applthermaleng.2021.116669