SciEn

Proceedings of 8th International Conference on Mechanical Engineering and Renewable Energy 2025
10-12 December, 2025, Chattogram, BANGLADESH

SciEn Conference Series: Engineering Vol. 4, 2026, pp 99-104 https://doi.org/10.38032/scse.2026.4.47

Performance of Fin Enhanced Compact Heat Exchanger Using Experimental Analysis

Umme Habiba, Bodius Salam, Swadhin Ghosh, Md. Tuhin Mia*

Department of Mechanical Engineering, Chittagong University of Engineering and Technology, Chattogram-4349, Bangladesh

ABSTRACT

The compact heat exchanger is popular in industrial and renewable energy systems, as they are highly efficient as a heat exchanger, and their construction is lightweight and compact. This paper experimentally analyses the thermal effectiveness of a fin-based compact heat exchanger at different hot- and cold-water flow rates (4-7 LPM). The Nusselt number, overall heat transfer coefficient, and compactness factor were determined using temperature data, and a CFD model run on ANSYS 2025 R1 using the k-ε turbulence model was created to validate the results. The findings indicate that the higher the flow rate of the cold side, the better the convective heat transfer, and the resultant maximum increase in the overall heat transfer coefficient was 11.95%. The experiments were in accordance with CFD predictions, and the deviations ranged between 0.9 percent and 11 percent, which proves that the model was reliable. The design is compact, as confirmed by the calculated compactness factor of 231.9 m^2/m^3, which was compact enough to operate in liquid-to-liquid mode. These results have shown that cold-side flow optimization can enhance thermal performance significantly and give a framework on how to design more efficient compact heat exchangers.

Keywords: Compact heat exchanger, Nusselt number, Thermal performance.

1. Introduction

Compact heat exchangers are small in size and can be used in situations where size and weight are of concern, including in automotive, aerospace, and HVAC systems. They are high-heat transfer devices that are compact; hence, their extensive research on enhancing performance and energy recovery. Other past studies have suggested fin geometry changes, shell-and-tube design, and computational fluid dynamics (CFD) modelling to improve thermal performance. Corrugated fin design measured by ANSYS Workbench with 3D laminar flow modelling demonstrated a better Nusselt number and thermal mixing, which resulted in 7.05-10 per cent more heat transfer and a decreasing pressure drop by 5.0-6.2 per cent [1]. An experimental and numerical study was carried out by Bari and Hossain [2] on a pancake-shaped shell-and-U-tube heat exchanger using the SST k-ω model, where the effectiveness was found to be 3% higher, and power output was found to be 7% higher. Abeykoon [3] did CFD-based optimisation, changing baffle cut, number of tubes, and baffle number, and got less than 2 per cent difference between the theoretical and numerical performances. A diffusion-bonded square-channel compact heat exchanger was experimentally studied by Sarmiento et al. [4], and it was discovered that the thermal conductance rose linearly with Reynolds number up to 7500, which proves the distribution of flow to be uniform. Pasupuleti et al. [5] compared the conventional and helical shapes of a fin using the SST k-model and found that helical fins were more efficient and effective in terms of thermal mixing but reduced pressure drop. Gholami et al. [6] examined oval tube exchangers using corrugated fins and identified a maximum improvement in thermal performance of 15 per cent, a pressure drop reduction of 19 per cent, and a 20 per cent rise in the Nusselt number. Studying designs of SECT, Saleh-Abadi et al. [7] observed the enhancement of the thermal effectiveness up to 75.2 with increased pressure losses. Kundu [8] examined longitudinal fins with trapezoidal and rectangular geometries, in which trapezoidal fins were found to be less pressure-losing than rectangular ones, and the latter had more heat transfer when having the same dimensions. Compact heat exchanger design and performance analysis have been further developed in recent studies. An extensive survey was able to summarise new methods to improve the shell-and-tube exchangers and determine the necessity of better experimental validation in changing operating conditions [9]. In experimental and computer-based geometry comparisons, modified shell-and-tube designs were found to be more efficient in energy at varying baffle intervals [10]. The other recent review also pointed to the performance trends of internal-finned exchangers, which pointed to the absence of liquid-liquid experimental studies [11]. Most recently, numerical analyses of twisted-angle fin designs were undertaken because they exhibit better thermal-hydraulic results, but this design has not been experimentally verified [12].

Whereas these publications play a big role in comprehending the operation of compact heat exchangers, the majority of publications are restricted to numerical analysis or gas-phase operation, substantiated by a lack of experimental data on the liquid-to-liquid operation. In addition, the joint effect of the differences in the flow rate on the long side and the cold side of longitudinal fin compact heat exchangers, as far as their convective performance is concerned, is not extensively studied. These research gaps limit the creation of experimentally tested models of flow-dependent performance prediction.

In a bid to overcome these drawbacks, the current study carries out experimental research on a longitudinal fin compact heat exchanger to operate in liquid-to-liquid mode at different flow rates with CFD simulations as a supportive tool. The aim of the work is to show that controlled variations in the flow rate on both the hot and cold sides have been experimentally investigated in terms of their effects on heat transfer and convective performance, which is a scarce addition to the experimental knowledge and optimisation of compact heat exchangers in industries.

2. Procedure of Experiment

SolidWorks was used to model the precise dimensions and layout of a compact heat exchanger, defining its shape and configuration. SolidWorks could be used to properly model flow channels, plate or tube configurations, and overall compactness.

Fig.1: Compact heat exchanger model.

Figure 1 represents a compact heat exchanger model designed in SolidWorks 2018. The model geometry measurements were provided as the tube's has a length of 30 cm, and the inside diameter of 2.0 cm. Longitudinal rectangular fin number were 10; tube external diameter: 2.2 cm; shell outside diameter: 10.2 cm; and shell internal diameter: 10 cm. The dimensions of the fins were 30 cm for length, 0.2 cm for thickness, and 3.5 cm for height. The heat exchanger's overall length, including the head and flange, was 41.0 cm. The fixed head length was 5.0 cm. After a thorough analysis, stainless steel 304 was chosen as it combines corrosion resistance, mechanical strength, thermal stability, manufacturability, and low cost, making it excellent for compact heat exchangers. Also the material used was selected due to its widely recognized versatility and widespread use. Longitudinal rectangular fins were positioned on the tubes using acetylene welding.

Following the completion of the apparatus, an experiment was conducted. After the heat exchanger was installed, it was inspected for any potential leaks to ensure proper sealing and system integrity. The apparatus was ready to take experimental values once leaks were fixed. Data collection was delayed by five minutes in order to reach the steady state position of the setup. After reaching steady-state conditions, experimental data were collected, following the approach previously adopted by Keya et al. [13]. During the experiment, the compact heat exchanger was affixed firmly in position using a stand made of wood. Two rotameters were used to measure the water flow rates.The accuracies and uncertainties of the measuring devices were considered with all of them being calibrated. A case in point is a thermocouple employed in measuring temperature with an uncertainty of ±.05 ºC, but flow meters have an accuracy of one percent of the value. During the experiments, the conditions of control, such as the inlet temperatures and flow rates, had been kept constant so as to obtain repeatability and reliability of the results.. For precise temperature measurements of the hot and cold water as fluid, fasteners were utilized to secure the thermometers to the apparatus. Two pumps that were connected to the heat exchanger via a ¾-inch pipe and a variety of pipe fittings were installed to provide hot water to the heat exchanger. As a supply, two water containers were utilized in the experimental configuration of the compact heat exchanger. An adjacent arrangement of the initial pair of containers designates one for the storage of cold water and the other for hot water. The experimental arrangement is displayed in Figure 2.

Fig.2: Setup of experiment

The mass flow rates of hot and cold water were varied, and the corresponding inlet and outlet temperatures were recorded. In order to record temperature data for both the hot and cold fluid, the hot fluid flow on the tube side was initially kept at 4 LPM while the cold fluid flow outside the tube was varied from 4 LPM to 7 LPM in increments of 1 LPM. Similarly, the experiments were repeated with the cold fluid flow outside the tube being altered in the same way, and the hot fluid flow on the tube side was kept at 6 LPM. Another set of experiments recorded the corresponding inlet and outlet temperatures of both fluids while varying the hot fluid flow outside the tube from 4 LPM to 7 LPM (1 LPM increments) and maintaining the cold fluid flow on the tube side at 4 LPM. Only the most representative results was presented in the research paper due to the vast amount of data collected from these studies.

3.Simulation

The SolidWorks 2018 was used to create the compact heat exchanger's 3D model, which was subsequently loaded into ANSYS 2025 R1 for simulation. The geometry was correctly created and meshed after import, and then the required boundary requirements and material properties were applied. After that, the simulation was run to examine the intended thermal performance in light of the study's goals.

3.1 CFD Analysis

A thorough analysis of the imported geometry's mesh quality was conducted to guarantee the precision and dependability of the simulation results. A reasonably fine mesh that was appropriate for capturing intricate flow and thermal characteristics was shown by the 52,379 nodes and 199679 elements that were produced. With an average skewness of 0.35919 and a standard deviation of 0.21781, the mesh was found to be within the permitted range of CFD simulations, guaranteeing both numerical stability and satisfactory mesh quality. The majority of the elements retained a balanced shape without undue stretching or distortion, as evidenced by the average aspect ratio of 2.3837 with a standard deviation of 1.2049. The mesh quality metrics indicated that the mesh produced was properly organized and suitable for precise simulation outcomes in ANSYS 2025 R1. Mesh of the compact heat exchanger is shown in Figure 3.

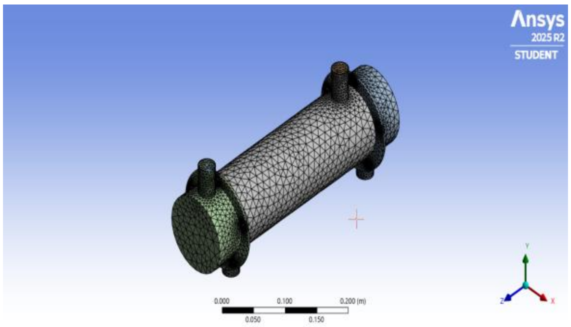

Fig.3: Mesh

The mass flow rate was adjusted in the boundary conditions between 0.067 kg/s to 0.12 kg/s and it was corresponding to the experimental test range. The hot water inlet temperature was adjusted to 322 K and cold water inlet adjusted to 292 K at varying flow rates as per the experimental values.
The convergence plot shows in figure 4 that all residuals including continuity, velocity components, energy, k, and ε- steadily decrease with iterations, reaching values below
10-3 to 10-5. This indicates that the numerical solution has achieved stable convergence, ensuring the flow and thermal fields are well-resolved in the simulation.

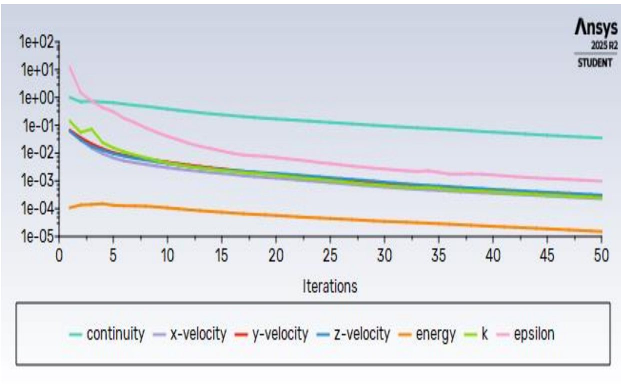

Fig.4: Convergence curve

3.2 Solution

The equation for energy was activated. For the viscous regime, the conventional two-equation turbulence model, k–ε, was used. The material qualities were classified as steel for the solid region and water (liquid) for the fluid domain. In compliance with the experimental setup and the model's specifications, boundary conditions were established. Then, for each test point, CFD simulations were run to calculate the corresponding hot- and cold-side outlet temperatures while maintaining the same mass flow rates and hot- and cold-side inlet temperatures as the experimental cases.

4. Theory and Calculation

A compact heat exchanger was the subject of a study in which temperature data were recorded under various settings by varying the mass flow rate outside the tube. The following formulas were used to determine the rate of heat transfer, effectiveness, and logarithmic mean temperature difference (LMTD) using the experimentally measured and numerically predicted temperatures.
Heat gained by cold fluid and heat loss by hot water :

$$Q_c = m_c \times c_{p,c} \times (T_{c,o} - T_{c,i}) \qquad (1)$$

Log Mean Temperature Difference(LMTD):

$$\text{LMTD} = \frac{(T_{h,i}-T_{c,o})-(T_{h,o}-T_{c,i})}{ln((T_{h,i}-T_{c,o})/(T_{h,o}-T_{c,i}))} \qquad (2)$$

Overall heat transfer coefficient :

$$U = \frac{Q_c}{LMTD \times Heat\ transfer\ area, A} \qquad (3)$$

Effectiveness percentage :
Tube side Reynolds number :

$$Re_i = \frac{\rho \times V_i \times D}{\mu} \qquad (4)$$

Tube side Nusselt number :

$$Nu_i = 3.66 + \frac{0.0668 \times D\sqrt{L} \times Re_i \times Pr}{1+0.04[(D\sqrt{L}) \times Re_i \times Pr]^{2/3}} \qquad (5)$$

Factor of compactness:

$$\beta = \frac{Surface\ area\ of\ heat\ transmission}{Volume} \qquad (6)$$

$$= \frac{0.746791}{0.00322} \text{m}^2/\text{m}^3$$

$$= 231.91 \text{ m}^2/\text{m}^3$$

Equations are from [14]. For gas-to-gas type compact heat exchangers, the factor of compactness is typically 500 m^2/m^3, 700 m^2/m^3, or higher; whereas for liquid-to-liquid type compact heat exchangers, the value is generally 200 m^2/m^3 or above and can exceed 400 m^2/m^3 [15,16]. Since the experiment was done on to liquid-to-liquid type, it is a compact heat exchanger for this study only but it is not a compact heat exchanger for a gas-to-gas type.

5. Result and Discussion

A compact heat exchanger construction is displayed in Figure 5, which was plotted in the ANSYS 2025 R1 temperature contour, which clearly revealed a temperature gradient through its length. Effective heat transmission from the hotter inlet sections at both ends (yellow/red zones) into the cooler core tube section (blue/cyan) was indicated by the temperature range of roughly 292 K (blue/cold) to 346 K (red/hot). Although slight temperature variations in the middle may indicate non-uniform flow as well as localised inefficiencies, the distribution indicated that heat was being extracted effectively as the fluid passed through the exchanger. With discernible heat exchange along its length, the system was operating as planned overall.

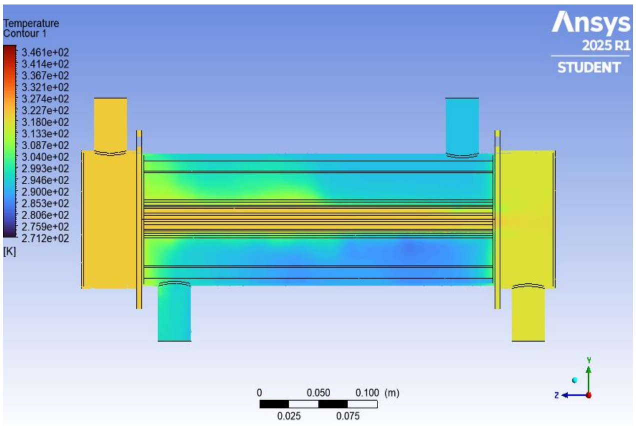

Fig.5: Temperature contour.

Figure 6 indicates that the heat transfer coefficient has a positive relationship with mass flow rate due to increasing turbulence and thinning of the thermal boundary layer. With the increasing rate of mass flow, a rise in flow velocity occurs, resulting in a greater Reynolds number that facilitates turbulent flow. Turbulence enhances diffusion of the fluid close to the wall, disruption of steady laminar sublayer, and decreases thermal resistance. The outcome of this process is a thinner boundary layer, which means that heat is moved more effectively between the wall and the fluid. This, in turn, results in an increased temperature gradient at the wall with a consequent increase in the convective heat transfer coefficient. The highest values are observed in the curve of cold water at 6 LPM, which is more turbulent and disturbs the boundary layer more than the curves of 4 LPM with the hot and cold water, which have similar values because of the similar conditions in the flow. Generally, the plot indicates that the greater the flow rate, the greater is the convective heat transfer since the flow rate fosters turbulence and decreases the thickness of the boundary layer.

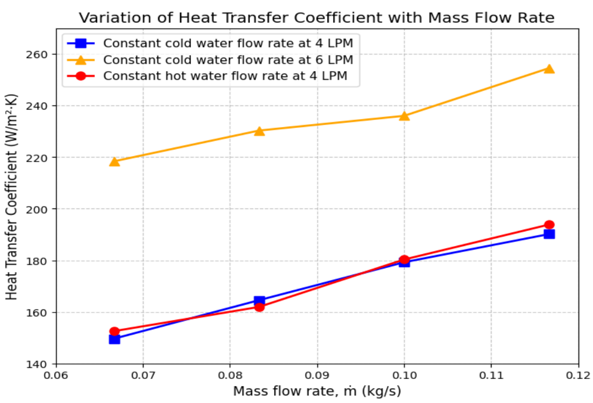

Fig. 6: Overall heat transfer coefficient vs mass flow rate.

Figure 7 displays that the Nusselt number (Nu) rises with the Reynolds number (Re) by portraying the increasing pre-eminence of convective heat transfer, and the flow alters its laminar flow regime into a turbulent flow regime. Physically, the Nusselt number is the ratio between conductive and convective heat transfer, and therefore, an increase in the Nusselt number means increased energy transfer by fluid movement. With Re, the velocity of the fluid increases, producing more inertial forces, which destabilise the flow approaching the wall and create more turbulence. Due to this turbulence, there is the generation of eddies and fluctuations, which relentlessly combine hot and cold layers of fluid that making the fluid in contact with the wall, which promotes more efficient heat exchange. The impact of turbulence also increases the momentum and thermal diffusion to be increased, making the temperature set of the wall and bulk fluid smaller. This increase in the curve implies that heat transfer increases at a higher rate than the inertia of the flow, which is in agreement with empirical correlations.

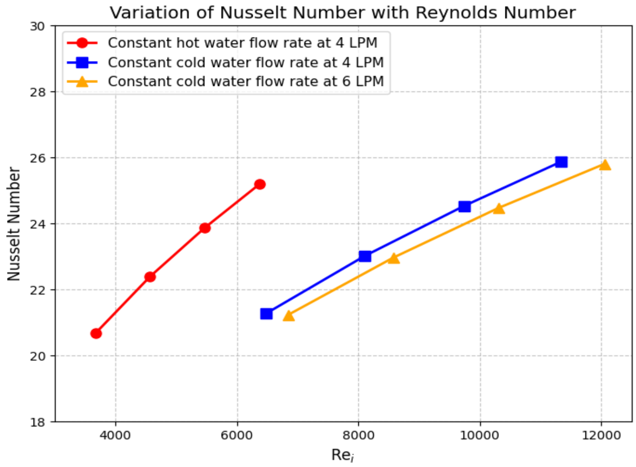

Fig.7: Nusselt number vs Reynolds number.

The findings show a more significant influence of the cold-side flow rate since the external fluid has direct contact with the fin surfaces, which creates a greater shear and more intense eddies that amplify turbulence. The rectangular fins enhance the secondary flow and local vortex, which enhances the thinning of the boundary layer and enhances convective mixing. This geometrical increase justifies the increase in Nusselt number with Reynolds number at the cold side. The trends obtained are consistent with classical correlations like the Dittus-Boelter in that the turbulence induced by the fin is an efficient way of increasing the heat transfer with a low price in terms of pressure. Furthermore, the numerical forecast revealed the percentage of deviation ranging between 1 and 11, which demonstrates that the numerical model of CFD reflected the process of physical behaviour and justified the experimentally obtained increase in thermal efficiency.

5.1 Validation of CFD

The results of the experimental and CFD are compared in the following table 1, 2 and 3 with an excellent agreement, and it proved that the given model predicts rather well under different flow conditions. The numerical predictions are regarded as valid even though the maximum difference between the experimental and CFD results was between 10 and 11%. This is because such differences are within an acceptable range for turbulent thermal-fluid simulations that involve experimental uncertainties in temperature and flow rate measurements. The observed deviation is primarily due to practical measurement errors rather than model inaccuracy because the CFD model replicated the same boundary conditions, material properties, and turbulence characteristics as the experiment.

Table 1: Comparison between experimental and CFD results for constant cold water flow rate at 4 LPM.

Overall heat transfer coefficient	Overall heat transfer coefficient in numerical	Percentage error
149.5902	165.9829	10.95839
164.5245	170.5569	3.66653
179.2477	186.6053	4.104708
190.1819	202.3873	6.417729

Table 2: Comparison between experimental and CFD results for constant cold water flow rate at 6 LPM.

Overall heat transfer coefficient	Overall heat transfer coefficient in numerical	Percentage error
218.3868	198.5769	9.071041
230.3136	224.3364	2.595245
236.0287	238.0948	0.875361
254.5517	274.2769	7.748967

Table 3: Comparison between experimental and CFD results for constant Hot water flow rate at 4 LPM.

20

Overall heat transfer coefficient	Overall heat transfer coefficient in numerical	Percentage error
152.5193	170.7545	11.95603
161.9154	168.0665	3.798961
180.3077	174.4497	3.248932
193.8765	181.7441	6.257819

6. Conclusion

This study concluded that the heat exchanger system's thermal performance and prediction accuracy were greatly improved by raising the cold water flow rate. Experimental data and numerical simulations agreed quite well, particularly at higher cold-side flow rates. An example is the increase in the overall heat transfer coefficient by a maximum of 11.95 percent, achieved by increasing the cold-side flow rate velocity from 4 to 7 LPM. A good agreement was found between the experimental and CFD results, with deviations ranging from 0.88 to 10.96. These observations suggest that maximum cold-side flow can significantly increase convective heat transfer, and such information can be effectively used to provide practical suggestions on designing more efficient compact heat exchangers in the industry. The Nusselt-Reynolds number connections and temperature contour plots provided additional evidence that improved turbulence brought on by higher flow promoted convective heat transfer. In order to create more dependable and effective heat exchanger designs, these results emphasized the significance of optimizing flow conditions, especially on the cold side. These findings also strengthened the models for real-world engineering applications.

References

[1] A. Gholami, M. A. Wahid, H. A. Mohammed, A. Saat, M. Y. M. Fairus, M. M. Sies, N. Kamaruzaman, Md. M. Rahman, and Z. Abdul-Malek, 2016, October. Investigation on heat transfer enhancement in a corrugated fin-and-tube compact heat exchanger. *Jurnal of Teknologi (Sciences & Engineering)*, vol. 78, no. 10-2, pp. 7–13.

[2] S. Bari and S. N. Hossain, 2015. Design and optimization of compact heat exchangers to be retrofitted into a vehicle for heat recovery from a diesel engine. *Procedia Engineering*, vol. 105, pp. 472–479.

[3] C. Abeykoon, 2020, January. Compact heat exchangers – Design and optimization with CFD. *International Journal of Heat and Mass Transfer*, vol. 146, pp. 1–15.

[4] A. P. C. Sarmiento, V. H. T. Soares, G. G. Carqueja, J. V. C. Batista, F. H. Milanese, and M. B. H. Mantelli, 2020, July. Thermal performance of diffusion-bonded compact heat exchangers. *International Journal of Thermal Sciences*, vol. 153, pp. 106384.

[5] R. K. Pasupuleti, M. Bedhapudi, S. R. Jonnala, and A. R. Kandimalla, 2021, December.Computational analysis of conventional and helical finned shell and tube heat exchanger using ANSYS-CFD," *International Journal of Heat and Technology*, vol. 39, no. 6, pp. 1755–1762.

[6] A. Gholami, M. A. Wahid, and H. A. Mohammed, 2017, March. Thermal–hydraulic performance of fin-and-oval tube compact heat exchangers with innovative design of corrugated fin patterns. *International Journal of Heat and Mass Transfer*, vol. 106, pp. 573–592.

[7] M. Saleh-Abadi, M. Rostami, and A. Farajollahi, 2023, September. Successive expansion and contraction of tubes (SECTs) in a novel design of shell-and-tube heat exchanger: a comparison between basic, finned and non-finned designs. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, vol. 45, no. 9.

[8] B. Kundu, 2015, March. Beneficial design of unbaffled shell-and-tube heat exchangers for attachment of longitudinal fins with trapezoidal profile. *Case Studies in Thermal Engineering*, vol. 5, pp. 104–112.

[9] S. A. Marzouk, M. M. Abou Al-Sood, E. M. S. El-Said, and M. K. El-Fakharany, 2023. A comprehensive review of methods of heat transfer enhancement in shell and tube heat exchangers. *Journal of Thermal Analysis and Calorimetry*, vol. 148, pp. 7539–7578.

[10] M. A. Kartal and A. Feyzioğlu, 2024, June. Experimental comparison and numerical heat analysis of newly designed shell-and-tube heat exchanger with designed geometry at different baffle intervals: status

of energy efficiency. *International Journal of Low-Carbon Technologies*, vol. 19, no. 1, pp. 1323-1336.

[11] F. Ahmad, S. Sachindra, and S. S. Kumar, 2024, April. Internal finned heat exchangers: Thermal and hydraulic performance review. *Int. Journal Heat Technol*., vol. 42, no. 2, pp. 583–592.

[12] W. Haw, C. S. Oon, and E. V. Lau, 2025. Analysis of hydrodynamics and thermal–hydraulic performance of twisted angle fins within an annular conduit. *Journal of Thermal Analysis and Calorimetry*, vol. 148, pp. 7539–7578.

[13] K. A. Keya, K. A. Rahman, A. A. Arzu, and A. A. Punam, 2019, December. Theoretical and Experimental Analysis of Helical Double Tube in Tube Helical Coil Heat Exchanger. in *Proc. Int. Conf. on Mechanical Engineering and Renewable Energy (ICMERE2019)*, Chattogram, Bangladesh.

[14] F. P. Incropera, D. P. DeWitt, T. L. Bergman, and A. S. Lavine, 2017. *Fundamentals of Heat and Mass Transfer*, 8th ed. Hoboken, NJ, USA: John Wiley & Sons.

[15] W. M. Kays and A. L. London, *Compact Heat Exchangers*, 1984. 3rd ed. New York, NY, USA: McGraw-Hill.

[16] B. Zohuri, 2015. *Application of Compact Heat Exchangers for Combined Cycle Driven Efficiency in Next Generation Nuclear Power Plants: A Novel Approach*. Cham, Switzerland: Springer.